



# HiCAD

## HiCAD 2016

News Overview

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ISD Software und Systeme GmbH  
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## Basics

### Service Pack 2 2016 (V 2102)

#### Important note on updates!

When you run the tool **Computer Parameter Configurator** (HiCAD exe\ParKonfigComp.exe) – either during installation (Extended Settings > User Configuration tab) or later – various system files in the HiCAD sys directory will be replaced (without any further query!) with the files supplied in the HiCAD templates directory which have been designed especially for a working with the selected module and the activated checkboxes (e.g. DSTV or Management + BIM). These files include, for example

- HiCAD masks for the part attributes (\*.HDX),
- HELiOS masks for projects, documents and articles (\*.MSK),
- Various configuration files (\*.HDB) and
- Template files for annotations (\*.FTD).

**You should therefore always make backups of all files that you have customized before using the ParKonfigComp.exe tool!**

**If you have any questions, please contact your nearest ISD branch office.**

#### STB\_PARAMETER.DAT -> Configuration Editor

In previous versions various settings for Steel Engineering could be specified in the STB\_PARAMETER.DAT, which is now no longer available. Its setting options can now be found in the Configuration Editor at **Steel Engineering**.

Therefore, the corresponding settings have been removed from the dialogue window that is open via  > Settings.

#### Insert Parts/Variants - Improved preview

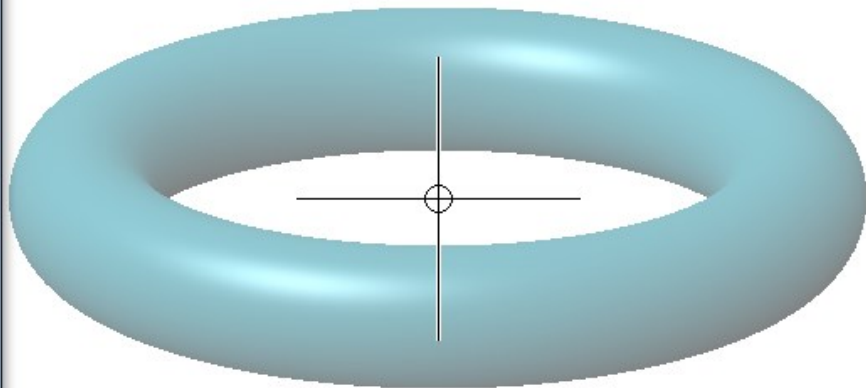
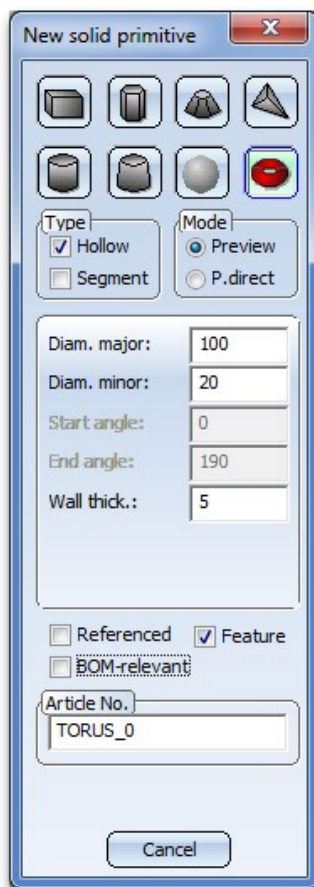
When you insert parts and variants, a shaded representation of them will be displayed before their placement in the drawing. This facilitates the placing in the drawing.

The representation type for fitting point determination on the part will not be influenced by this - this parameter can still be set in the Configuration Editor at **System settings > Visualization > Views > Insertion view**.

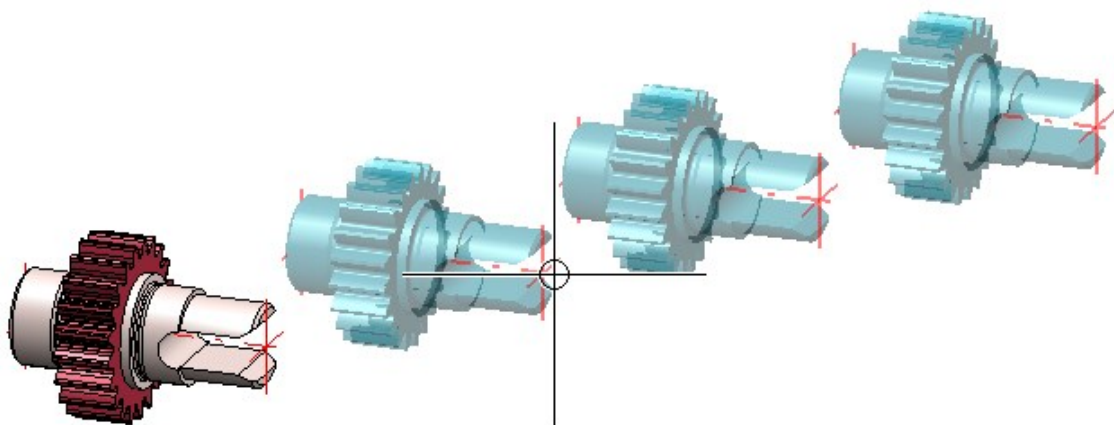
The changes concern various functions throughout all industries, e.g.

- all functions at Drawing > Insert Part,
- the pasting from the Clipboard,
- the creation of base sheets,
- the attaching of flanges via sketches,
- the moving and rotating of objects,
- the insertion of Standard Parts,

and much more.



Example: Insertion of a solid primitive



Example: Fitting point specification in drawing for "Clone + Move"

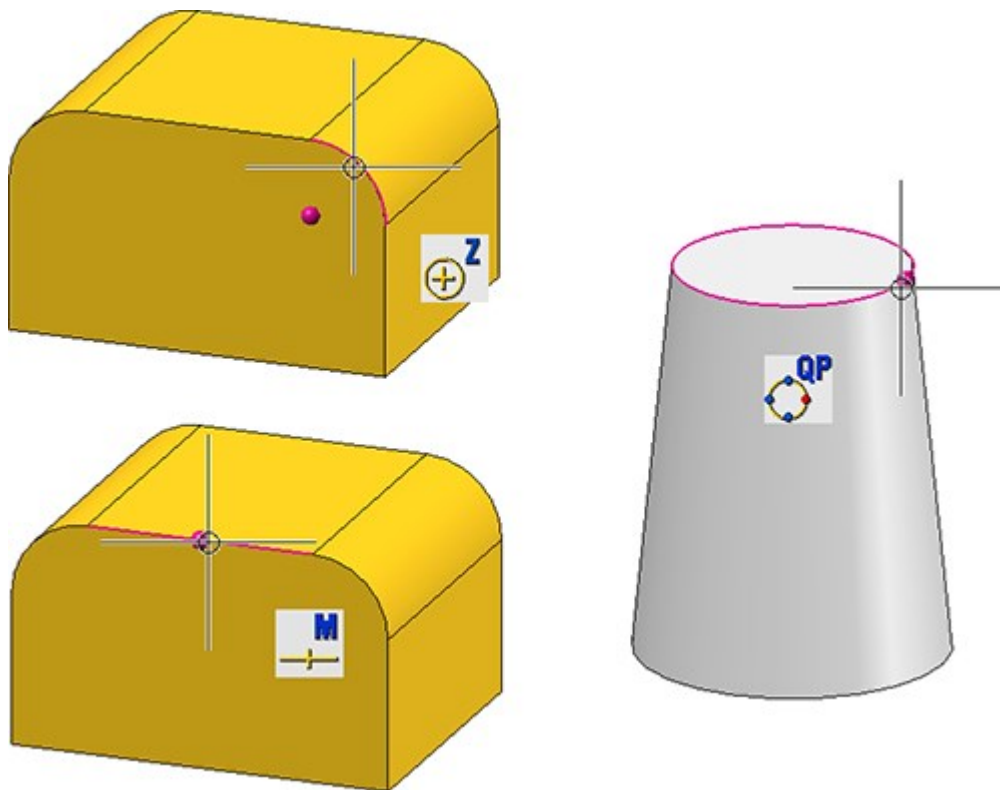
Furthermore, the function dialogue has been changed. When HiCAD prompts you to specify a fitting point in the drawing, you have now in many situations the option to right-click and open a context menu with further functions. For instance, you can use the **Origin** function to apply the origin of the active coordinate system.

Fitting point in drawing (RMB = Further options)



## Point selection with the Autopilot

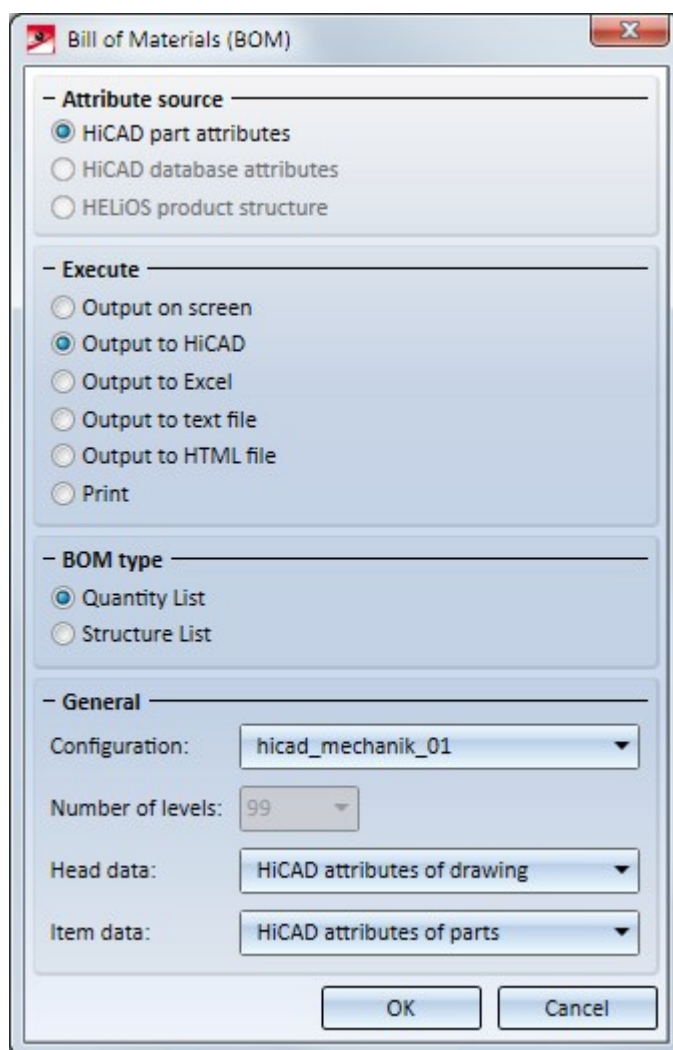
- The HiCAD Autopilot can now also snap quad points.
- When specifying points with the HiCAD Autopilot, the found points and the original geometry relevant for point specification are now highlighted in a different colour (Special colour, Marking 1), e.g. the circle for quad points, the arc/circle for centres, or the line for mid points.



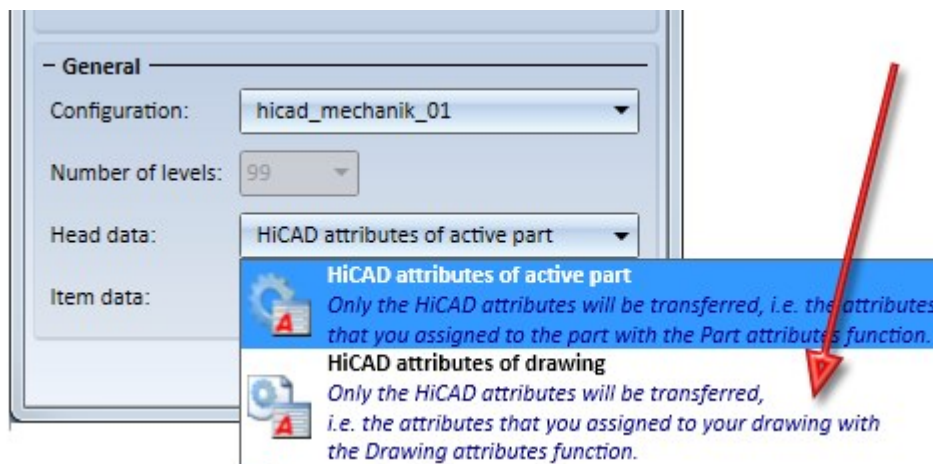
## BOMs

The dialogue windows for the functions **Bill of Materials, for entire drawing** and **Bill of Materials, for active assembly** have been revised. In the selection boxes for Head data and the Item data, the name of the corresponding HB file will no longer be shown, but an explanatory text that is linked to the corresponding HDB file.

*Bill of Materials (BOM) dialogue window*



When creating BOMs for the active assembly, you have now the option to transfer the attributes of the drawing as head data.

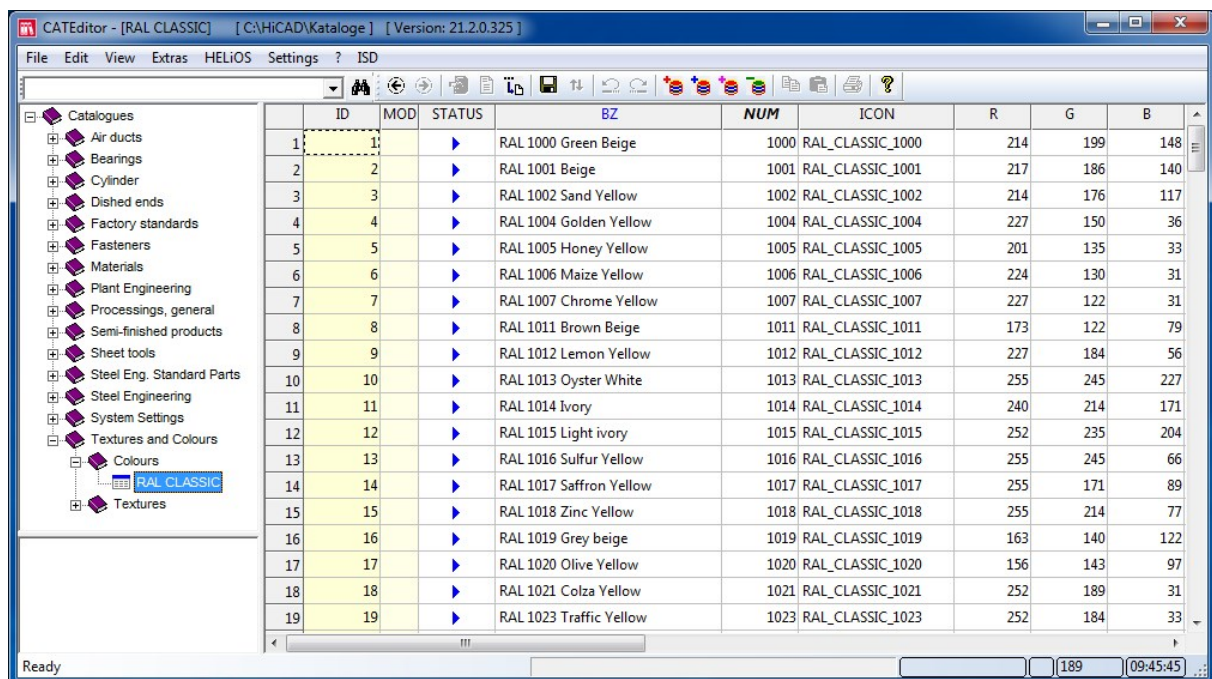


If you have previously used your own control files, e.g. individual HDB files, for the BOM output, you can still use them if you want. In this case, please read the information given in the BOM Output with Individual Control Files topic.

## RAL colours

RAL colours were previously defined in the system file RALRGB.DAT. As of SP2, this file is no longer available; instead, you will find the RAL colours in the Catalogue Editor at **Textures and Colours > Colours > RAL CLASSIC**.

*RAL CLASSIC colour table*



ID	MOD	STATUS	BZ	NUM	ICON	R	G	B
1	1	▶	RAL 1000 Green Beige	1000	RAL_CLASSIC_1000	214	199	148
2	2	▶	RAL 1001 Beige	1001	RAL_CLASSIC_1001	217	186	140
3	3	▶	RAL 1002 Sand Yellow	1002	RAL_CLASSIC_1002	214	176	117
4	4	▶	RAL 1004 Golden Yellow	1004	RAL_CLASSIC_1004	227	150	36
5	5	▶	RAL 1005 Honey Yellow	1005	RAL_CLASSIC_1005	201	135	33
6	6	▶	RAL 1006 Maize Yellow	1006	RAL_CLASSIC_1006	224	130	31
7	7	▶	RAL 1007 Chrome Yellow	1007	RAL_CLASSIC_1007	227	122	31
8	8	▶	RAL 1011 Brown Beige	1011	RAL_CLASSIC_1011	173	122	79
9	9	▶	RAL 1012 Lemon Yellow	1012	RAL_CLASSIC_1012	227	184	56
10	10	▶	RAL 1013 Oyster White	1013	RAL_CLASSIC_1013	255	245	227
11	11	▶	RAL 1014 Ivory	1014	RAL_CLASSIC_1014	240	214	171
12	12	▶	RAL 1015 Light ivory	1015	RAL_CLASSIC_1015	252	235	204
13	13	▶	RAL 1016 Sulfur Yellow	1016	RAL_CLASSIC_1016	255	245	66
14	14	▶	RAL 1017 Saffron Yellow	1017	RAL_CLASSIC_1017	255	171	89
15	15	▶	RAL 1018 Zinc Yellow	1018	RAL_CLASSIC_1018	255	214	77
16	16	▶	RAL 1019 Grey beige	1019	RAL_CLASSIC_1019	163	140	122
17	17	▶	RAL 1020 Olive Yellow	1020	RAL_CLASSIC_1020	156	143	97
18	18	▶	RAL 1021 Colza Yellow	1021	RAL_CLASSIC_1021	252	189	31
19	19	▶	RAL 1023 Traffic Yellow	1023	RAL_CLASSIC_1023	252	184	33

## Renamed colour configuration file

In Version 2014 the predefined Marking colours have been changed. The previous name of the corresponding configuration file was `hicad_2014.hcf`. This file has been renamed to **hicad.hcf**.



## Load/Save

When loading and saving files, access to network directories is now established much faster. This applies in particular to slow networks with a large number of files.

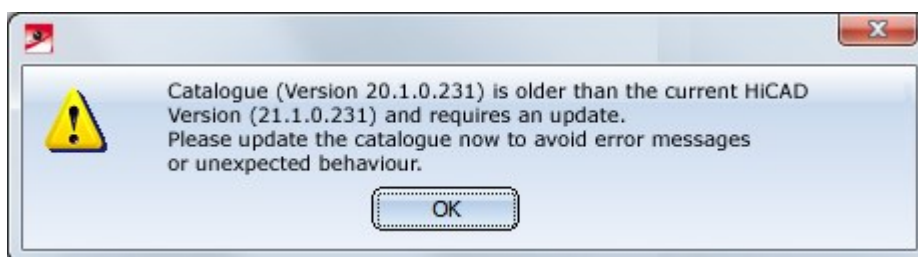
## Service Pack 1 2016 (V 2101)

## Start HiCAD

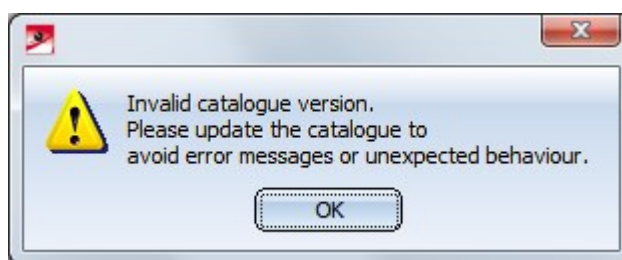
### Catalogue directory check

The version of the utilized catalogue directory will be checked upon HiCAD start. If the catalogue directories are old there is a risk that HiCAD will search for catalogue entries which do not yet exist in the old catalogues. One example of this are the Textures that were integrated in the catalogue some time ago. If these are missing in the catalogue, you will have no access to these textures in HiCAD.

If you start HiCAD with an old catalogue directory (e.g. HiCAD 2100.1 with Catalogue 2100.0 or HiCAD 2100.0 with Catalogue 2002.2), a corresponding message will be displayed:



If the version of the catalogue cannot be detected, the following message will be displayed:



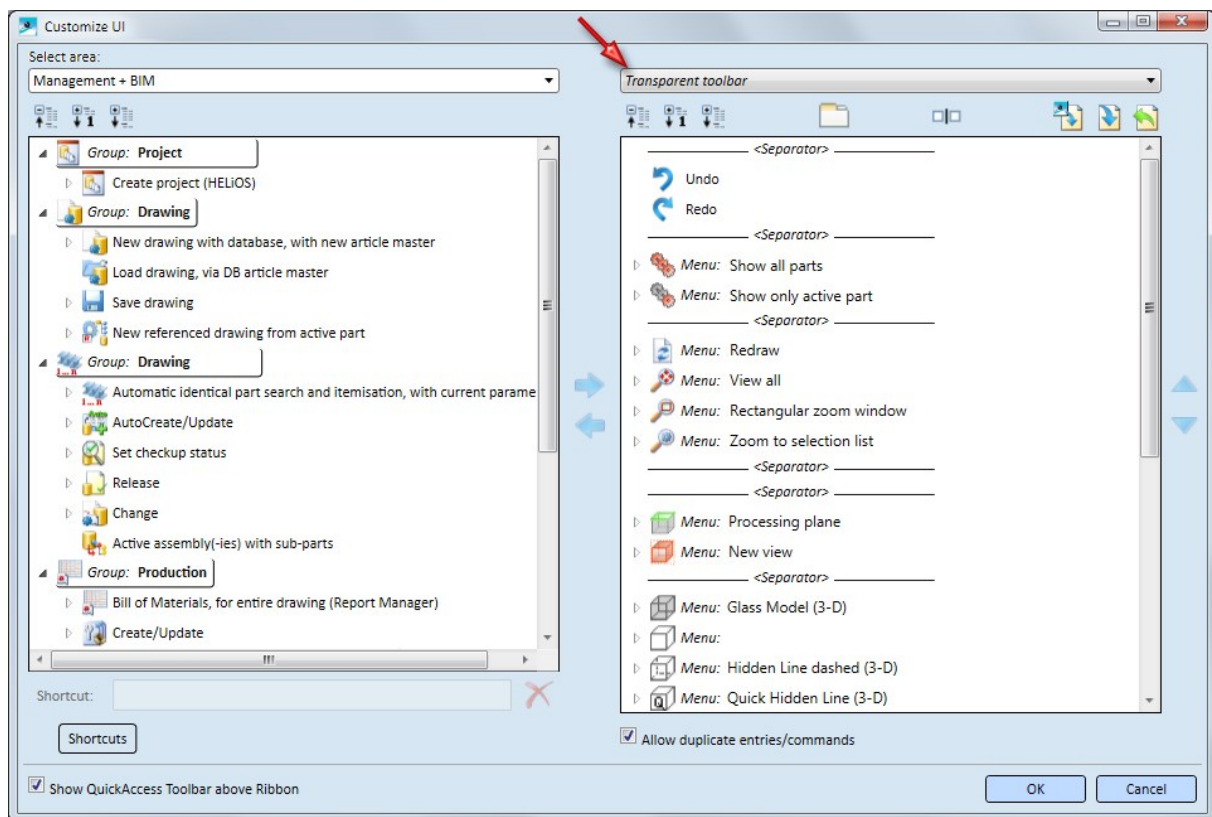
## User interface

### Zoom functions

The Zoom functions have been removed from the Ribbon. You can still access the functions via the transparent toolbar and the shortcuts.

### Customize transparent toolbar

You can now use the **UserInterfaceEditor.exe** tool to customize the transparent toolbar.



Furthermore, you have now the option add functions to your own Ribbon tabs that were previously only available in the transparent toolbar or in the QuickAccess toolbar. Examples of such functions are:

- **Show all parts** and **Show only active part**
- **Redraw** (incl. **Reshade** and **Recalculate active view** from the sub-menu)
- **Zoom** functions
- **3-D processing plane** (incl. sub-menu)
- **Find (Part filter)**

## Load/Save

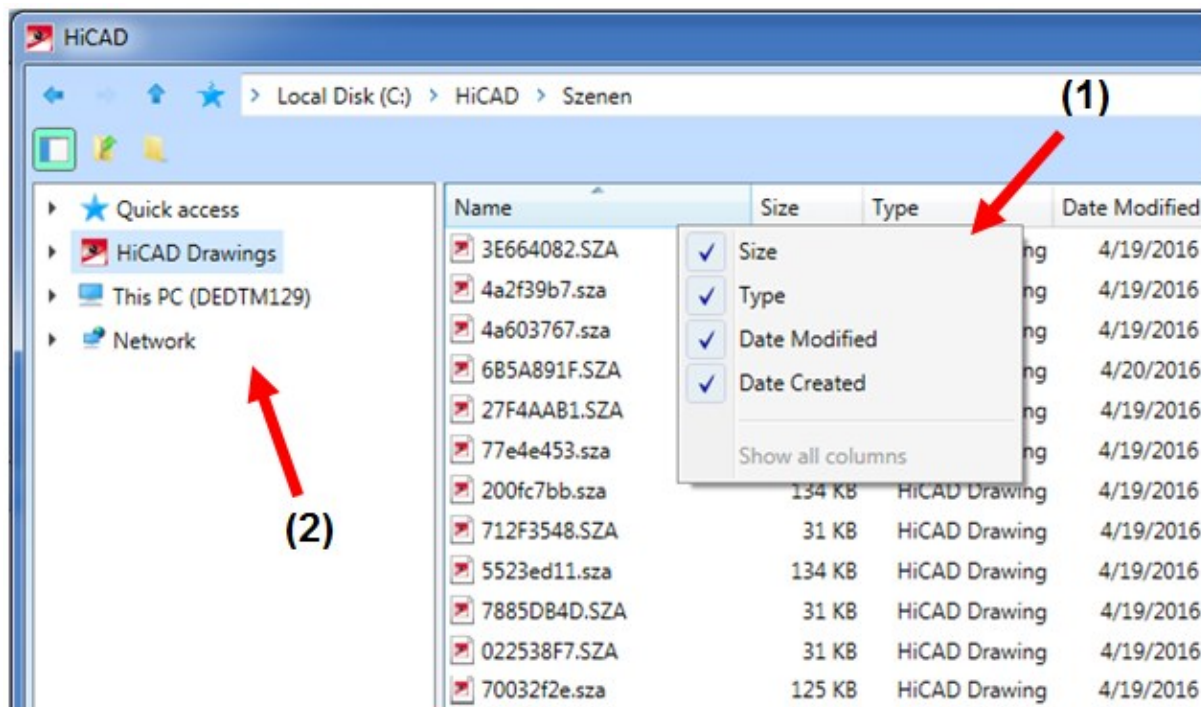
The dialogue for the loading and saving of files has been revised.

## Hide/Show columns

The columns of the file list can now be hidden and redisplayed. Right-click on a column header and select the desired column from the context menu. (1)

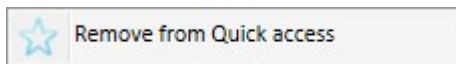
## Structure tree

The structure tree in the navigation area has been slightly modified. (2)



## Others


- To remove a folder stored for Quick access, first activate the Quick access option in the navigation area. Right-click the object you want to remove and choose **Remove from Quick access**.

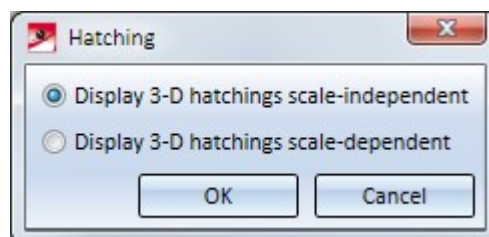


- Click the arrow symbol ▼ next to the file name to open a selection box with a History / History (Explorer) list. Beneath **History**, all files that were last opened in/saved to HiCAD are shown; beneath **History (Explorer)**, the files that were all files that were last opened in/saved to the Windows file Explorer are shown.

## Other changes

### Scale-dependency of 3-D hatchings

The new **Hatching, Scale-dependency**  function for hatchings allows you to specify for 3-D hatchings whether they should be scale-dependent or scale-independent. You can find the function on the **Drawing** tab, at **Properties > Attr.** > ... as well as in the context menu of the drawing.

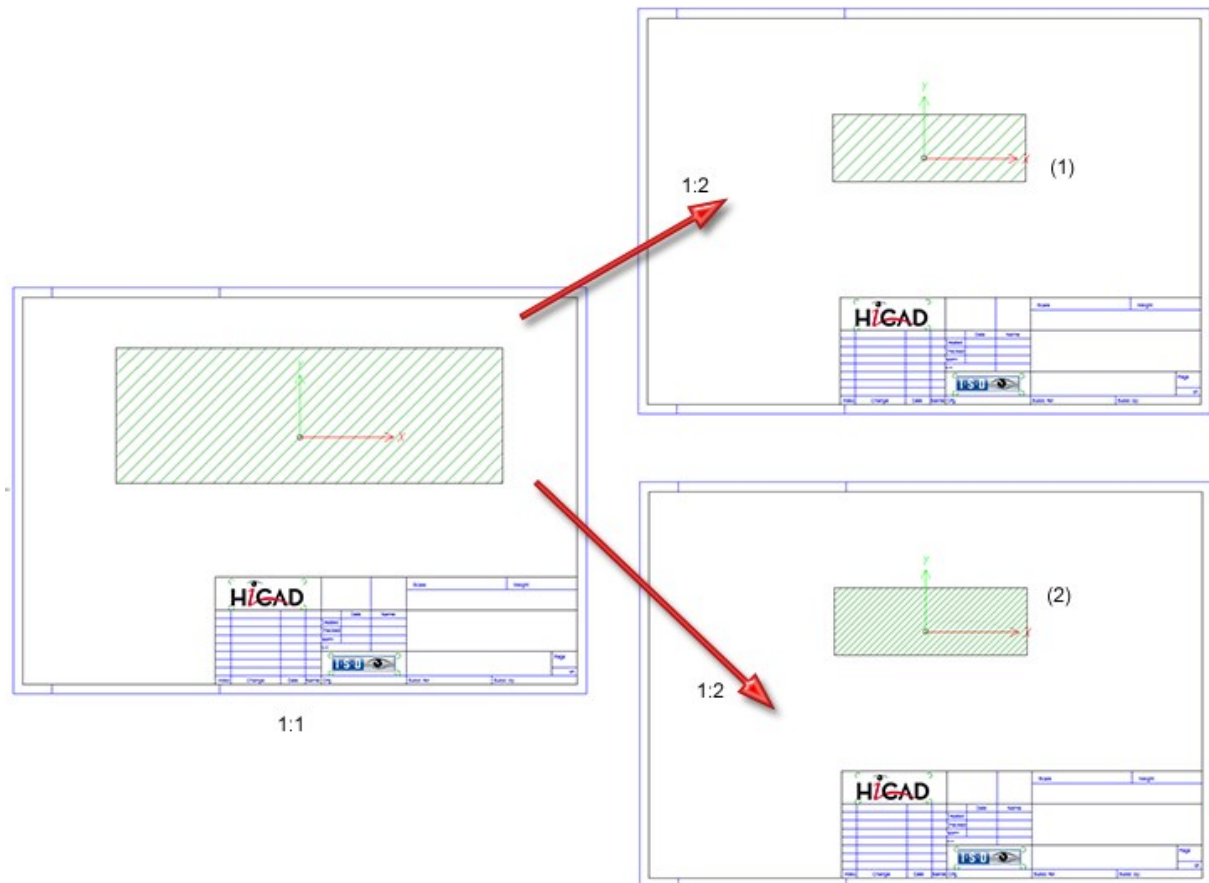


The default setting is **Scale-independent**. With this setting, the spacing between the hatching lines will always be the same, no matter which scale was selected.

If you want the hatching line spacing to be scaled together with the object in case of scale changes, choose the setting **Scale-dependent**.

The default setting shown in the dialogue window can be changed in the Configuration Editor, at **System settings > Visualisation > Scale dependent 3-D hatching**.

*Example*



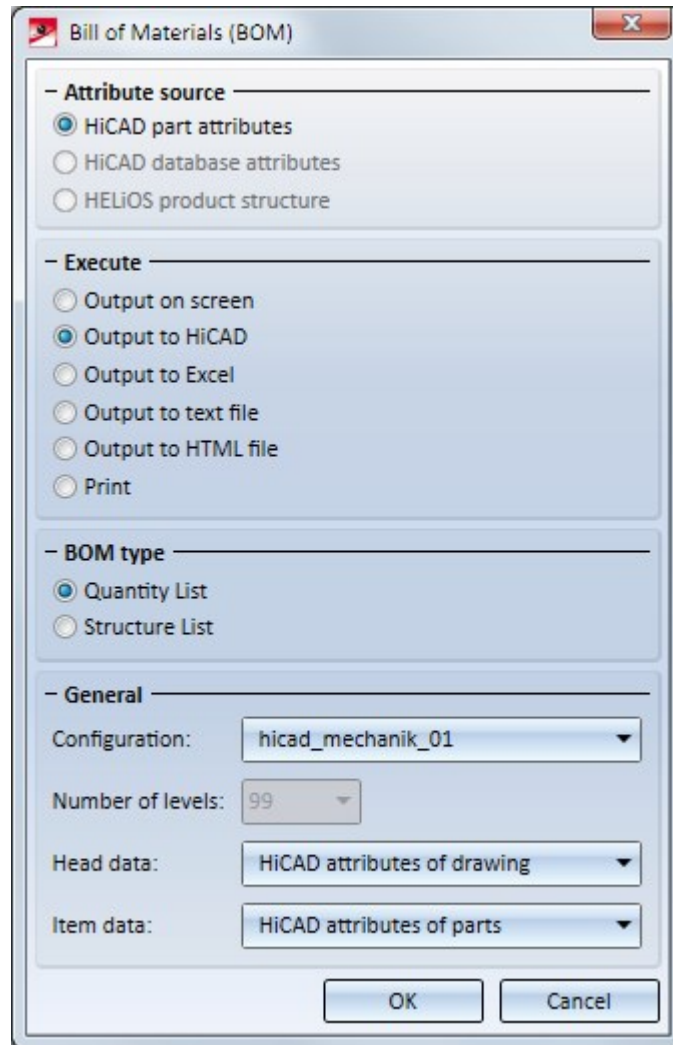
### Revised dialogue windows for BOMs and weld seam test protocols

The dialogue windows for the functions


- BOM for entire drawing,
- BOM for active assembly,
- BOM Settings,
- Weld seam test protocols for entire drawing, and
- Weld seam test protocol for active assembly

have been modified.


BOM dialogue window



## Delete symbol at cursor

For various Delete functions a  symbol was shown at the cursor in previous versions. This is no longer the case.

## Insert part - Now also for .PAA archives

The functions **Insert part, via Explorer**  and **Insert part, via HELIOS document master/article master** now also allow the insertion of parts from .PAA archives, which are, for instance, used in Plant Engineering.

If a .PAA archive contains different representations, these will be offered for selection before insertion takes place.

The insertion from a .PAA file via HELIOS takes place for pipe parts via the Combined search and the link **AllgTyp-Geometrie** (GeneralType-Geometry), for all other parts via the link **Bauteil-Teilegeometrie** (Part-Part geometry).



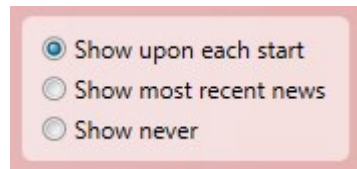
## Major Release 2016 (V 2100)

### Start HiCAD

#### The News window

In order that you can take full advantage of the numerous new features and enhancements offered by HiCAD 2016, check out the videos explaining the new and changed HiCAD functionalities. From now on we will provide you with such videos with each Major Release and Service Pack to keep you constantly informed in the best possible way.

You can configure the display of the News window at the bottom left. We recommend using the **Show most recent news** option to ensure that you will never miss any news about HiCAD. You can call the News window again at any time and change the display option by clicking on the “?” symbol at the top right and selecting **Show news**.




#### Please note:

To suppress the 'News' window on the Client computers in the case of a use on a network (i.e. to achieve a behaviour as if the user had chosen **Never show** him-/herself), the Administrator must copy the file HiCADNews.Settings.XML to the

**%APPDATA%\ISD Software und Systeme\HiCAD 2016\GUI\News**

directory of the Client computers. In line 4 of the XML file, the **Value** needs to be set to 2:

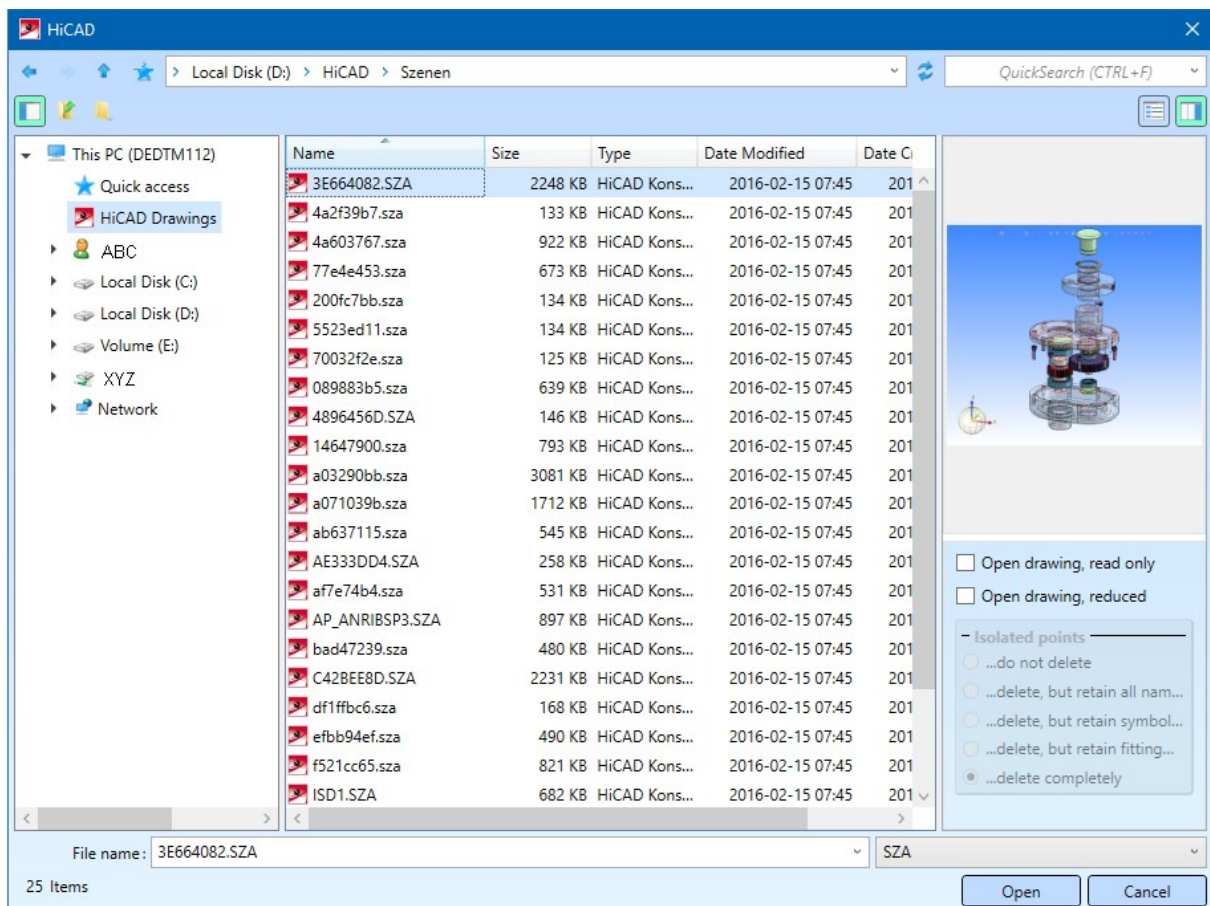
```
1  <?xml version="1.0" encoding="utf-16" standalone="no"?>
2  <!--created from XMLFileDepot-->
3  <Settings>
4    <Property Name="ShowOnStartup" Type="ISDDepot.XMLFileDepotEnumType" Value="2" />
5  </Settings>
```



## Load/Save

### Redesigned dialogue windows

The HiCAD dialogues for the loading and saving of files have been adapted to Microsoft Windows 10 standards. For example, frequently required folders can now be stored beneath a **Quick access** item in the structure tree of the browser.



The Open dialogue window

### Progress bar during loading and saving

The progress bar that is shown during the loading of drawings has been improved and is now displayed continuously. This also applies to the loading of large 3-D parts (.KRA).



If referenced parts need to be updated when loading drawings, the progress bar will be stopped. After successful updating, it will be continued.

## HELiOS terminology and dialogue window

### Terminological changes

As of HELiOS Version 2016 (2100.0), HELiOS "Parts" will be called **Articles**, the former "Part master" will be renamed to **Article master** and the former "Part attributes" to **Article attributes**. These changes also required some

adjustments of function names and dialogue texts in HiCAD, e.g. when opening, or saving drawings via the HELiOS database.

Please also note that parts in HiCAD will continue to be called **Parts**. Only through their management in HELiOS will they become **Articles**.

Some other terms have been changed as well. You can inform yourself about these changes in the "What's New" topic (News) of the HELiOS Online Help.

### Integration of the new HELiOS masks

The current data input and search masks of HELiOS have been integrated into HiCAD. As a result, the HiCAD-HELiOS interface will benefit from the significantly greater functional range of the new masks and result lists.

## Coordinate systems

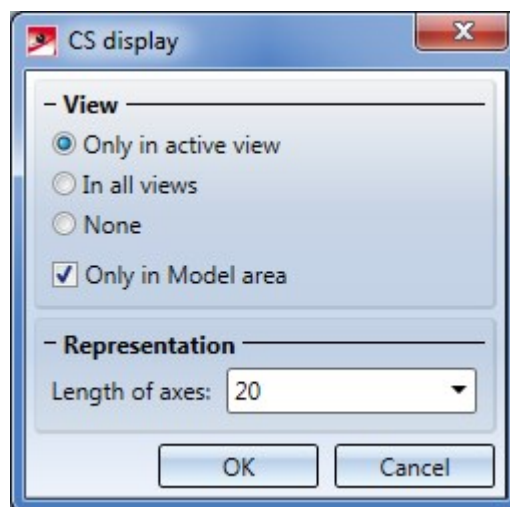
### Coordinate system display

The previous functions for the display and representation of coordinate systems have been combined into the new



CS display

function.



Specify via the radio buttons whether the coordinate system is to be displayed in all views or only in the active view. Specify whether this setting is to apply only to the Model area or to the Model and sheet area by activating or deactivating the **Only in Model area** checkbox. If the checkbox is activated, no coordinate systems will be displayed in the Sheet area (irrespective of the chosen radio button).

The **Length of axes** value determines the size of the coordinate system display. The value must be greater than 0.

The coordinate system display will be saved when you close HiCAD. This means that if the display is active when HiCAD is closed, it will still be active when HiCAD is started the next time.




#### Please note:

- The **CS display** function does not refer to this coordinate system that can be hidden and shown via F3, but to the coordinate system that is displayed by default in the origin (0,0,0) of each view.
- The representation as space cube is no longer available.

## Fitting coordinate systems


The behaviour of fitting coordinate systems has changed.

- A part can only have one fitting coordinate system. If you have defined several fitting coordinate systems in a row, the last defined fitting coordinate system will be used and saved to the base data. This rule also applies if several entries for fitting coordinate systems exist in the feature log of the corresponding part.  
This behaviour also applies for legacy data from older HiCAD Versions (before 2002.2)!
- If a fitting coordinate system has been defined for a part several times, and the **Delete Fitting CS...**  function is then applied to this part, the previously defined fitting coordinate systems will still be shown in the feature log. After a recalculation, the last but one fitting coordinate system will apply.
- If the base data of a part (from HiCAD versions before 2100.0) contains several fitting coordinate systems, the last defined fitting coordinate system will apply here, too.

## ICN

### New symbol for standard part groups

The symbol for 3-D standard part groups in the ICN part structure has been changed.

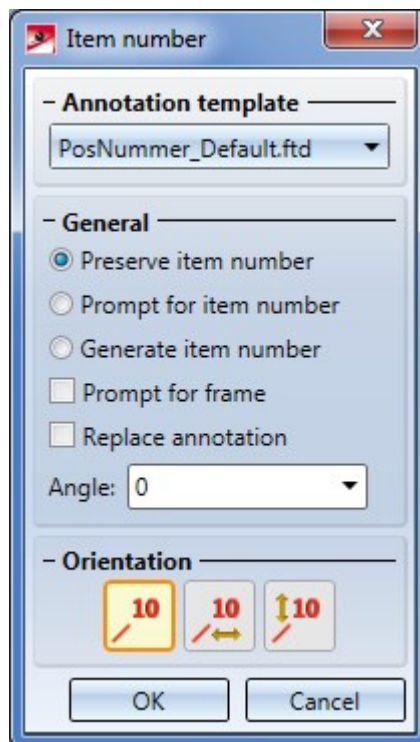
Standard part groups are now identified by the  symbol.


## 2-D


### Service Pack 1 2016 (V 2101)

#### Itemise part

The dialogue window of the **Itemize part** function has been redesigned.



The functions that could previously be found in the lower area of the **Item number** dialogue window can now be found in the pull-down menu  of the **Item.** function in the **Text** function group, or in the context menu of an item number.

When you choose the options **Prompt for item number** and **Generate item number**, a warning icon  will be displayed. If you move the cursor over the icon, the following warning message will be shown:

*This function assigns item numbers without checking for possible conflicts; i.e. conflicts with other items will not be recognized, so that different item numbers may be assigned to identical parts!*

## 3-D

### Service Pack 2 2016 (V 2102)

#### Simulations

##### Polyline motor with reference point

Without an explicit reference point specification, Polyline motors will always start at the start or end point of the polyline, and will calculate transformations in such a way as if the part to be moved was located in this position. If the part to be moved is located far away from this start point, the part will perform an unexpected movement.

Similar problems occur when the part is to start right in the middle of an edge. In this situation, it is located far away from the start point and end point of the edge, thus causing unexpected movements.

Such problems can be avoided by specifying a suitable reference point on the part. This reference point will be projected onto the polyline. The Polyline motor will then calculate the transformations relative to this point on the polyline, and no longer relative to the formerly chosen start point.

[Click here to view an example.](#)

#### Standard Parts / Standard Processings

##### Interaction of CADCLICK with HiCAD

The ISD Group cooperates with the company KiM GmbH (<http://www.cadclick.com/>). Parts in supplier catalogues, e.g. HILTI dowels, can now directly be inserted in HiCAD via the CAD click solutions.

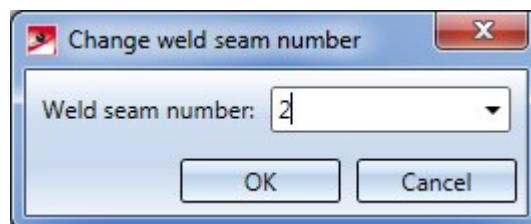
Extensive information can be found in the Help for **Interfaces**.

##### Bore patterns based on sketches

The shape of bore patterns is stored in the **Factory standards** catalogue. In addition to the previously used 2-D HCM variants (.DCF) you can now also use planar sketches saved as 3-D parts (.KRA).

##### Change weld seam numbers

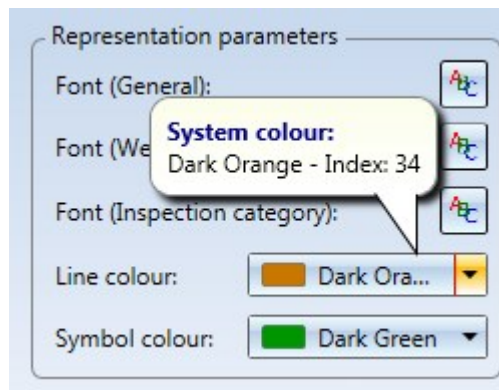
The Change weld seam number dialogue window has had a redesign.



### Weld symbols - Improved display for colour selection

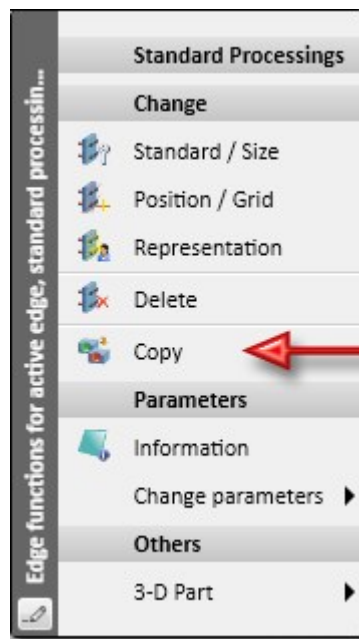
When inserting or editing weld symbols, longer colour names were simply cut off in the dialogue window. As of SP2 the following procedure applies:

- The colour name in the selection field will be abbreviated.
- When you move the cursor over the colour selection field, the complete colour name will be shown in a tooltip.




### Copy Standard Processings

The Copy Standard Processing function is now also available in the context menu for Standard Processings.



### Boltings/Rivetings

Boltings and Rivetings can now also be inserted repeatedly. This means that the **New bolting/riveting**  function will not be ended automatically after insertion, but remains active, allowing you to insert further boltings or rivetings with the same settings (processing plane, bolting assignment, clamping length etc.). What can be modified, though, is the grid used for insertion.






## Sketches

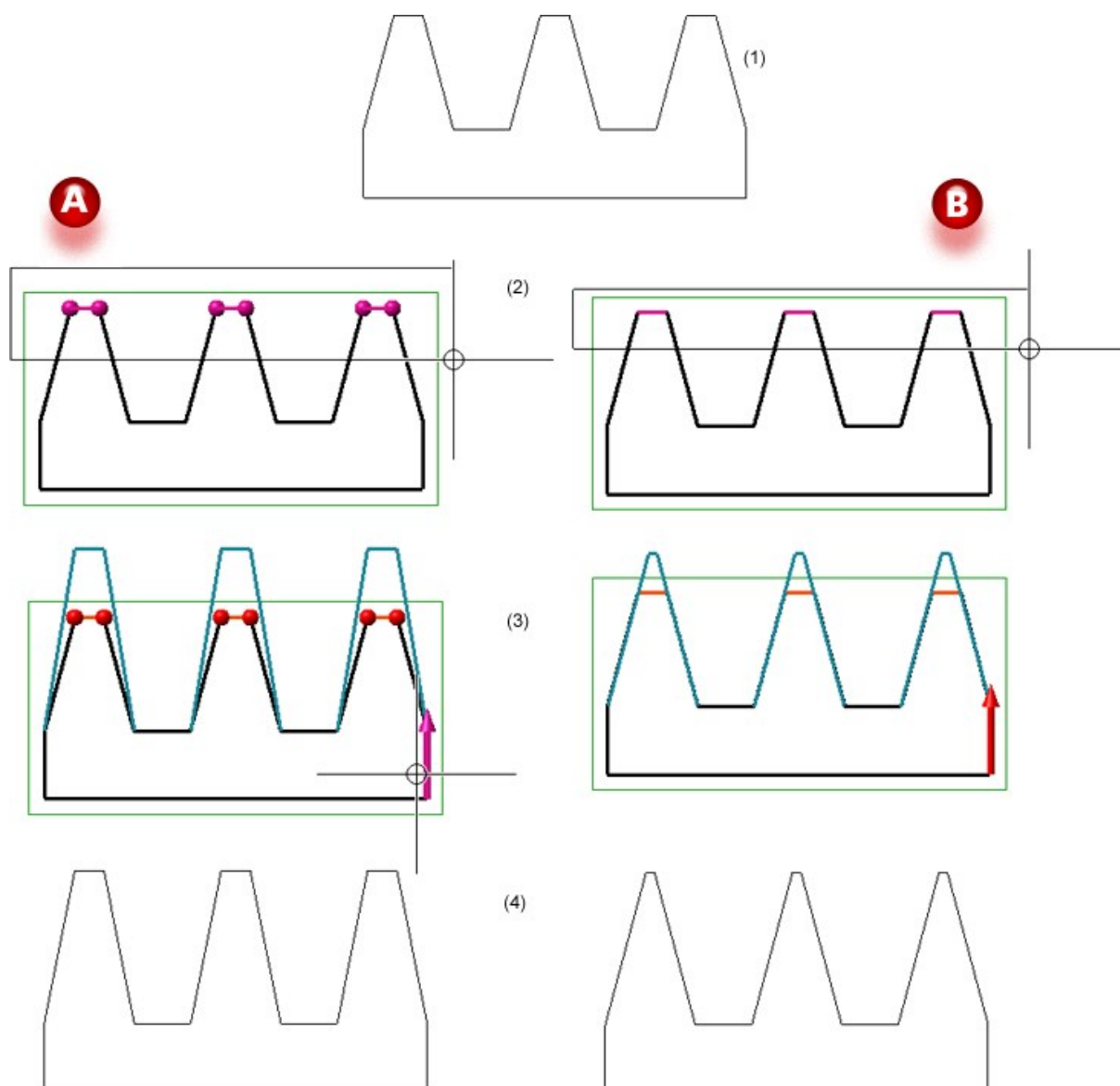
### Move points

The two functions

- Move points and
- Move points, in rectangle

at **Sketch > Transform** have been combined into one new function, **MovePts** . The old **Move point** function (previously to be found at Sketch > Process > Trim  > Polyline) has also been withdrawn.





You use the new **MovePts**  function to move one or several points and lines of a sketch. Basically, the procedure is very similar to that of the Move, Preserve connected lines and adjust length function for sketch elements. The difference here is that not only isolated points but also individual start or end points of the sketch elements can be moved, and that the preserving of connected lines does not take place here via length adjustments, but is based on the logic of the HCM dragger. In the process, HiCAD will try to preserve the HCM constraints where possible.



In Case A the original sketch (1) has been processed with the **MovePts** function; in Case B, the **Move, Preserve connected lines and adjust length** option has been chosen. In both cases the elements have been chosen via a rectangle (2), and the same direction (3) and the same distance has been chosen. (4) shows the results.



### Transform / clone sketches

When transforming or cloning sketches, the **Distance** input fields will be locked as long as no displacement direction has been chosen. This concerns the following functions:

- Move sketch elements 
- Move + Rotate sketch elements 
- Clone + Move sketch elements 
- Clone, Move + Rotate sketch elements 

### Scale sketch elements

The effect of the middle mouse button has also changed. This concerns the following functions:

- Scale sketch elements 
- Clone + Scale sketch elements 

When you middle-click on the drawing while the dialogue window is open, the following will happen:

- If you have made sufficient value inputs (at least specify length and fixed point), the function will be executed. The dialogue window will remain open afterwards. (same as with the **Apply** button).
- If the value inputs are insufficient, HiCAD will jump to the next required selection. For instance, if you have chosen the line to be scaled and then press the middle mouse button, HiCAD will jump to the selection of the fixed point. If you press the middle mouse button again after the fixed point selection, the function will be executed - even without a direction selection.

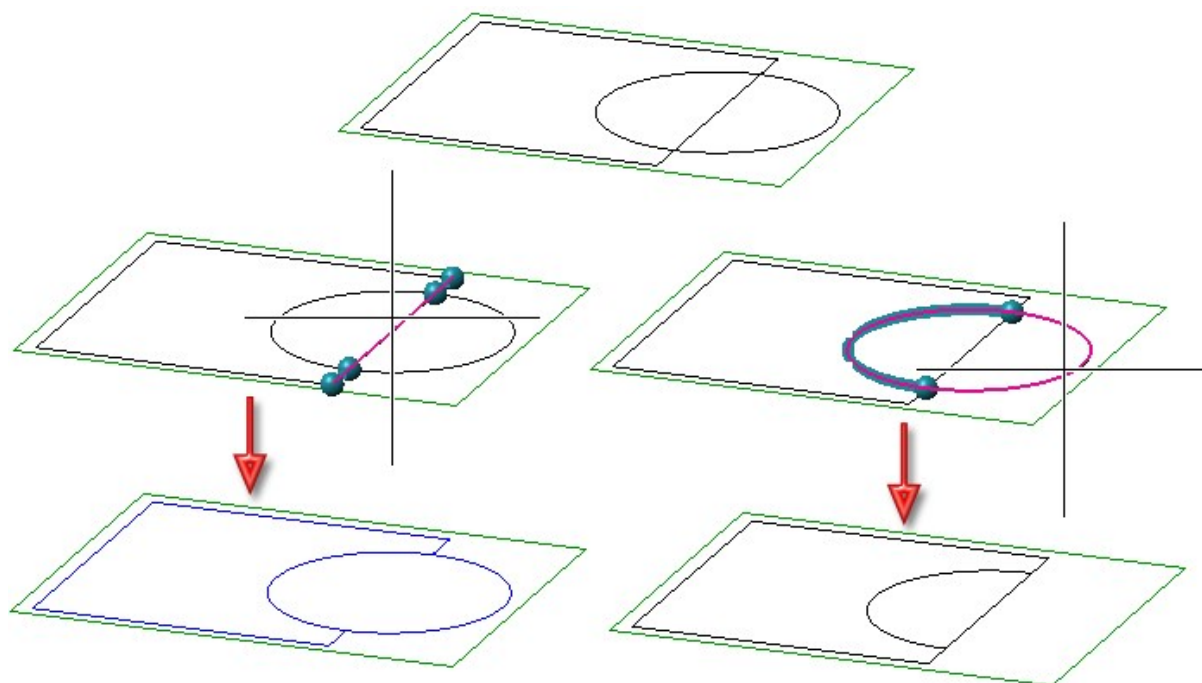
### Trim sketch elements

The dialogue window for the function Sketch > Process > Trim...  has been modernized.

When you move the cursor over the sketch in your drawing, HiCAD will check, depending on the current cursor position, whether the sketch element has any intersection points with other sketch elements. If this is the case, the portion of the element that would be deleted will be highlighted in a different colour (magenta, Special colour Marking 1). The portions of the element that would be preserved as well as their end points will also be highlighted (Special colour Preview). The end points will be represented as small spheres.


Click to perform the deletion.

Example (Sketch)




The handling of HCM constraints, too, has been improved.

## Divide sketches

The Divide line, via number of segments  function has been enhanced:

- The **Convert to lines** checkbox now only affects curves. As a result, HCM constraints on straight lines will be preserved.
- In the context menu, the functions **All lines** and **All curves** have been renamed to **Connected lines (only straight lines)** and **Connected lines (only curves)**, respectively.

## Convert 2-D splines to sketches

Sketch > New > Convert 2-D part to sketch  function now also considers and correctly converts 2-D splines that are, for example, created during 2-D DXF import.

## Assemblies

### Form assemblies

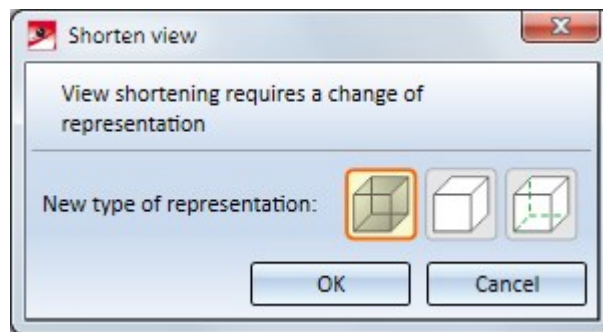
When forming assemblies from already existing parts, HiCAD will now automatically display the **Part attributes** dialogue window after the creation of the assembly. In this way, the attributes of the assembly can be determined immediately.

Right-click a part to choose the **Form assembly**  function in the context menu.

## Views

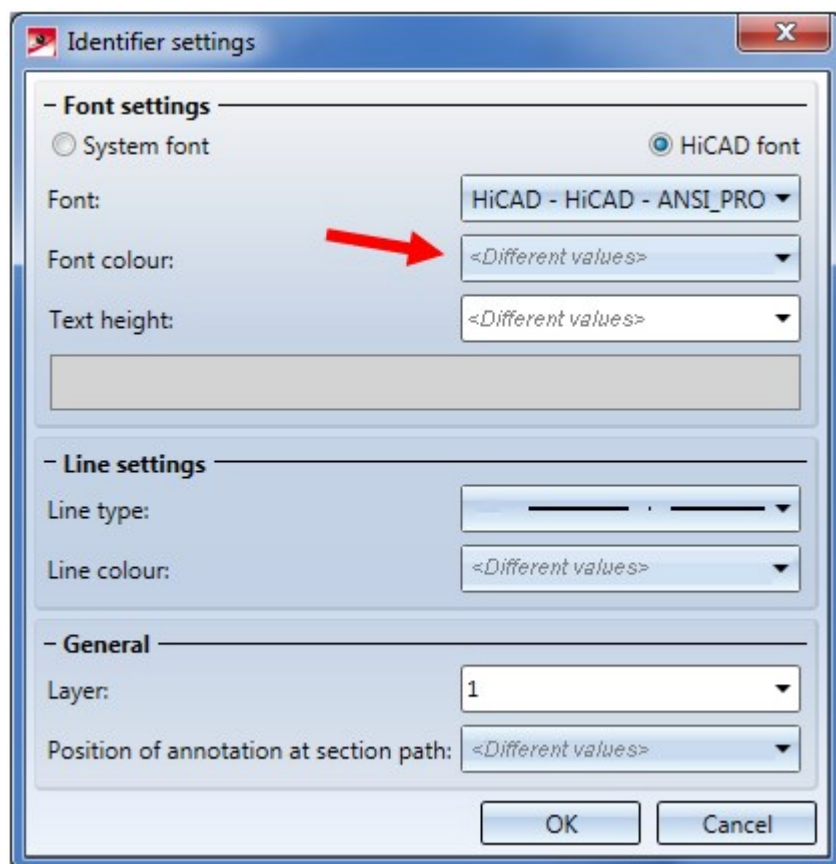
### Shortened views

If a view shortening is not possible, e.g. because of a currently active, shaded OpenGL view, an appropriate message will be displayed. You have then the option to choose how to proceed further. The corresponding dialogue window has been adjusted in SP2.



### Change sectional view identifiers

If you choose the Change sectional view identifier, All  function in a model drawing that contains views with different identifier settings, this will be indicated in the corresponding input fields by the text *<Different values>*.



If you enter a value into one of those fields and confirm with **OK**, the entered value will be applied to all identifiers. If you enter no value and confirm with **OK**, the different values for the sectional view identifiers will be preserved.


## Overlapped edges in HiddenLine views

The representation of edges in HiddenLine views has been improved. For example, previous issues with centre lines, crosshairs and symmetry lines have been resolved, so that part edges are no longer overlapped by crosshairs etc.

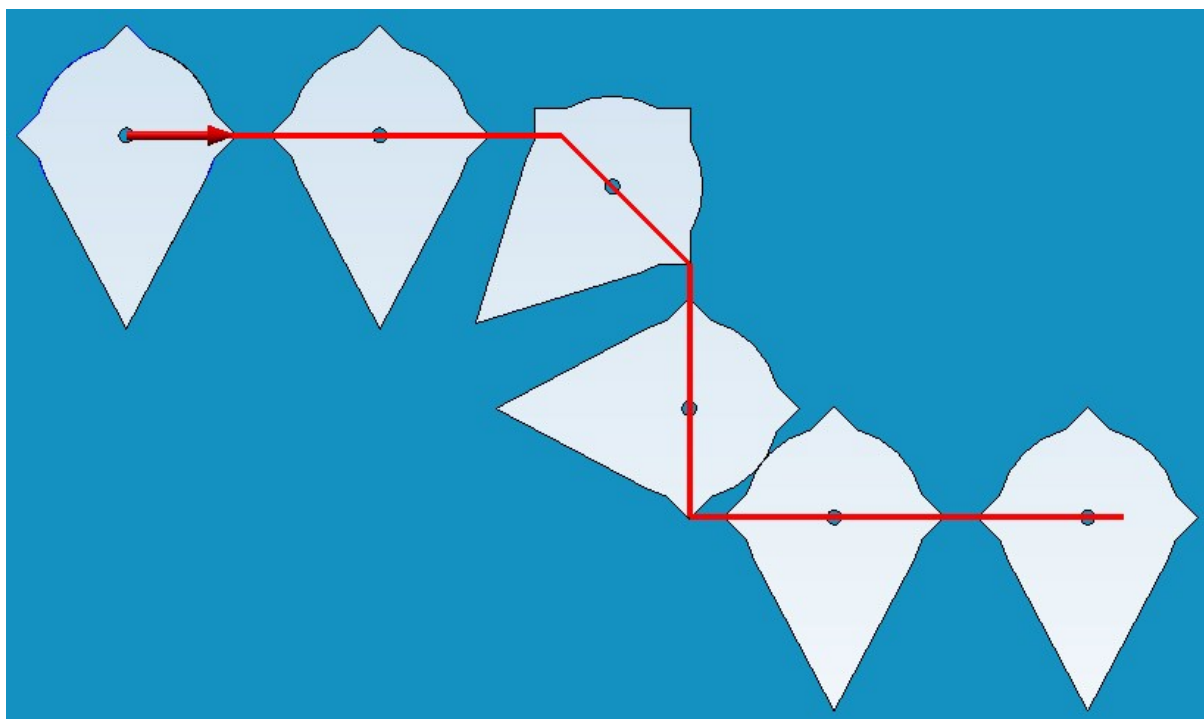
## Process parts

### Clone processings

- The 2 new cloning pattern types "Polyline pattern" and "Translatory and rotatory pattern" have been added to the

**Clone processings**  function. This allows you to define clonings along polylines and composite edges, or combined rotatory and translatory clonings.

*Example of the cloning of a subtraction along a polyline*



- Besides the definition of omissions in a table you have now also the option to select them intuitively by identifying the clones to be omitted in the sketch.

### Clean up intersections

The calculation time for the cleaning up of intersections could be reduced significantly. In test examples, time savings between 30% and 50% could be achieved.

Please note that the number of selected surface has a significant influence on the calculation time. This means that, for instance, in thin-walled 3-D parts (e.g. hollow cylinders) the accuracy will not be decreased if only one surface (inside or outside) will be selected.

## Chamfer

Since Version 2014, HiCAD has used a new algorithm for Boolean operations offering significantly enhanced stability - most notably for legacy data conversion or foreign data imports. This algorithm is now also applied for additions and subtractions or the chamfering of parts.

## Select cylinder/cone axis as direction

You can select a lateral surface of a cylinder or cone to select the cylinder/cone axis. The direction will be determined by the current cursor position.

This concerns the following functions:



3-D-Standard > Process > Wall ▾ > Envelope



3-D-Standard > Process > Clone processings...



Sketch > Transform > Move...



Sketch > Transform > Rotate



Sketch > Transform > Mirror...



Sketch > Clone > Move...



Sketch > Clone > Rotate



Sketch > Clone > Mirror...

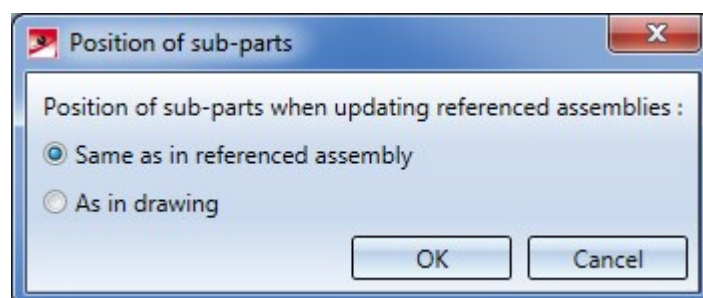


Sketch > Draw > Freehand ▾ > Transition curve

## Referencing

### Settings Update, Position - Dialogue

The dialogue window of the Settings Update, Position function in the **Referencing** menu has had a redesign.



## Processing of locked parts

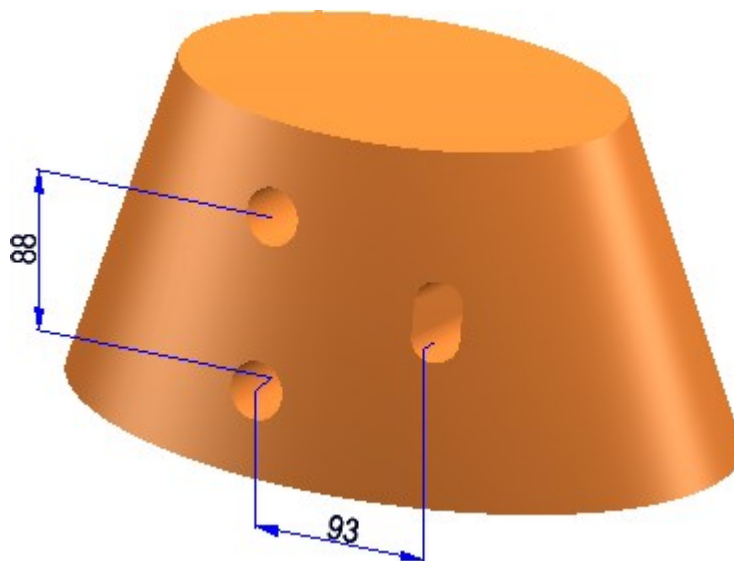
Normally, the processing of locked parts (locked referenced parts or parts with "released" status) will be prevented by HELiOS. In (very rare) cases in which a function will not prevent the processing of locked parts, a warning message will be displayed, telling you that you have processed referenced parts for which you have no write permissions due to the lock. In such cases HiCAD will attempt to revert your changes automatically. This is, however, only possible if the KRA file exists. In any case an appropriate message will be displayed, telling you whether the reverting of the changes was successful or not.

## Service Pack 1 2016 (V 2101)

### Bores

#### Selection of centre of bores through cylinders

Centres of bores through cylinders or in arbitrarily curved surfaces can now be snapped with the Autopilot.



### Referencing

#### Fitting CS of referenced parts

A so-called **Fitting coordinate system** (short: Fitting CS) is automatically assigned to 3-D parts when they are referenced, and entered into the feature log. Fitting coordinate systems that may already exist at the moment of referencing will be deleted.





Proceed as follows:

- If a part is referenced directly upon its creation (which is, for example, possible for Solid primitives), its Fitting CS will be the current local coordinate system. If no local coordinate system is defined, the Fitting CS will be determined by the points (0,0,0), (1,0,0) and (0,1,0).
- If a part is inserted, and referenced directly after its insertion, its coordinate system will be the World coordinate system.
- If a part is inserted, and further parts are added to the drawing, and for one of these parts a local coordinate system drawing is created, this local coordinate system will become the Fitting coordinate system when the first part is referenced. This rule also applies if the referencing of the first part is removed and the part is then re-referenced.

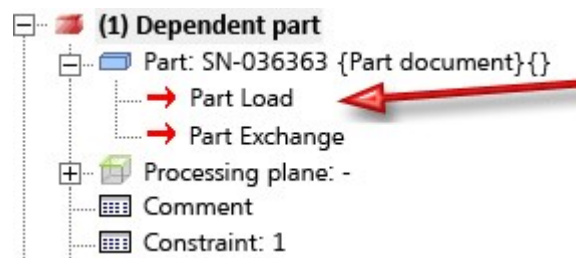
### Handling of article masters for referenced parts and assemblies

If a main assembly is referenced and has an article master which is different from the article master of the drawing, no automatic corrections will be applied to the article master assignment and/or the HELiOS links of the main assembly.

If the main part has an article master which is different from the article master of the drawing, no document master links will be checked or corrected (this also applies to the Management + BIM functions). If the main part is also referenced, no article master links will be checked or corrected either (this also applies to the Management + BIM functions).

### Dependent parts - Load part

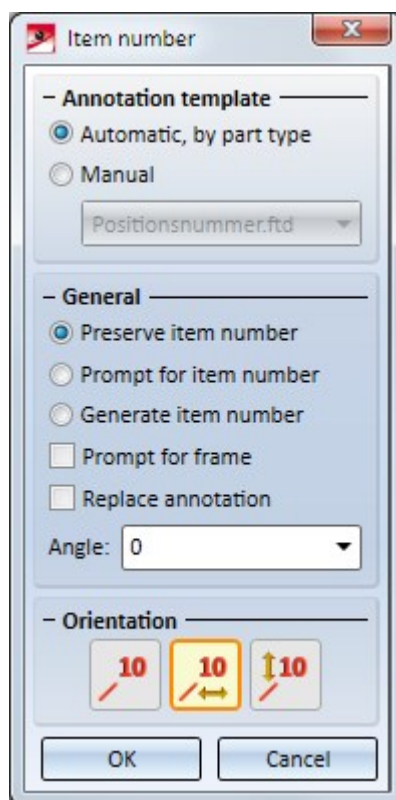
In previous versions it was not possible to activate the feature function "Load" for dependent parts based on a referenced parts (or variant parts managed by HELiOS) if the referenced part (or the variant part) was a read-only part. As of HiCAD 2016 SP1 it is also possible to load locked/read-only base parts.




## 3-D Dimensioning + Text

### Itemize part

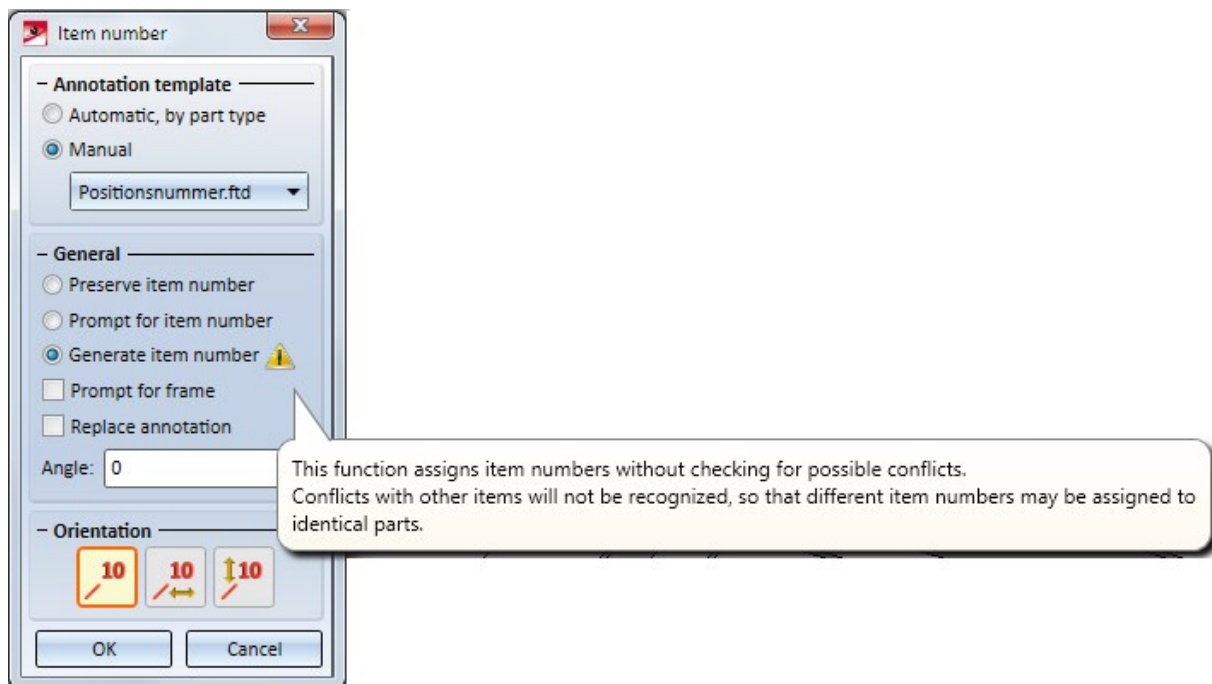
The **Item number** dialogue window has been optimized.






Please note that no check for conflicts will take place when you have activated the options **Prompt for item number** and **Generate item number**. Possible conflicts with other items will not be detected, and identical parts may have different item numbers as a result.

Therefore, a warning symbol  will be shown when you have chosen one of these options. Click on the symbol to read the warning message:

### Warning message



The functions that were previously available beneath **Edit and Delete** are no longer available, with the exception of the following functions:

-  **Increment range of numbers**
-  **Align horizontally**
-  **Align vertically**

Click  beneath **Item...** to open the pull-down menu containing these functions.

## Process parts

### Exchange 3-D part



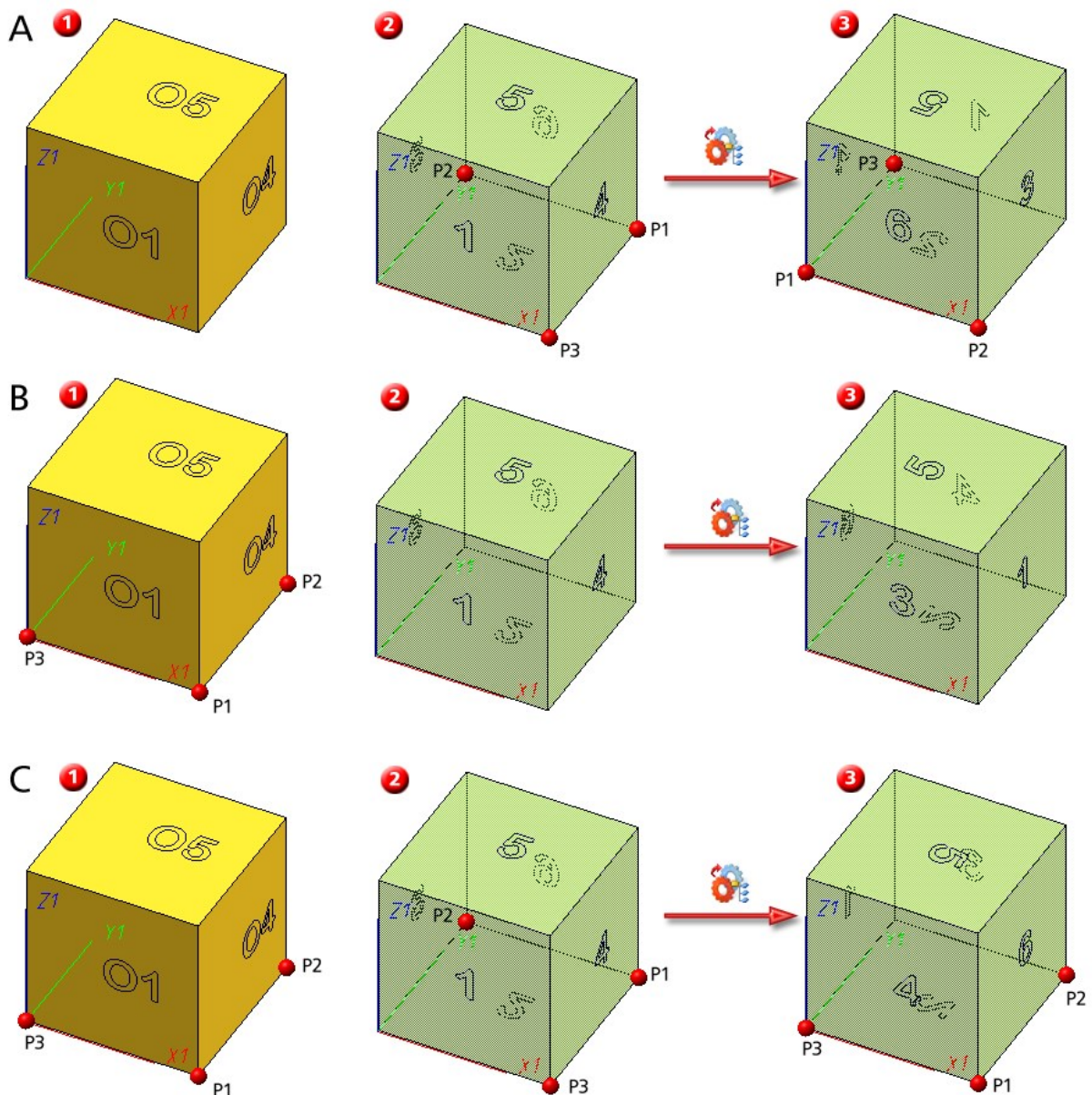
When using the **Drawing > Insert Part > Exchange** function in previous HiCAD versions, the Parts CS was used for the original parts and the replacement part. As of HiCAD 2016 SP1, an existing Fitting CS will take priority over the Part CS.

If for one of the selected parts or both parts a Fitting CS has been defined, not the Part CS, but the Fitting CS will be used. This applies to the insertion of the replacement part (use of Fitting CS of replacement part) as well as to the placing in the drawing (use of Fitting CS of original part).

The image below shows 3 different cases:



- A (1) Original part has no Fitting CS.  
(2) Replacement part has a Fitting CS.  
(3) Result after exchange.
- B (1) Original part has a Fitting CS.  
(2) Replacement part has no Fitting CS.  
(3) Result after exchange.
- C (1) Original part has a Fitting CS.  
(2) Replacement part has a Fitting CS.  
(3) Result after exchange.

In all cases, the origin (right-click) has been selected as the fitting point.





## Aligning of Steel Engineering plates

The functions beneath Alignment of derived drawings in the context menu for parts:

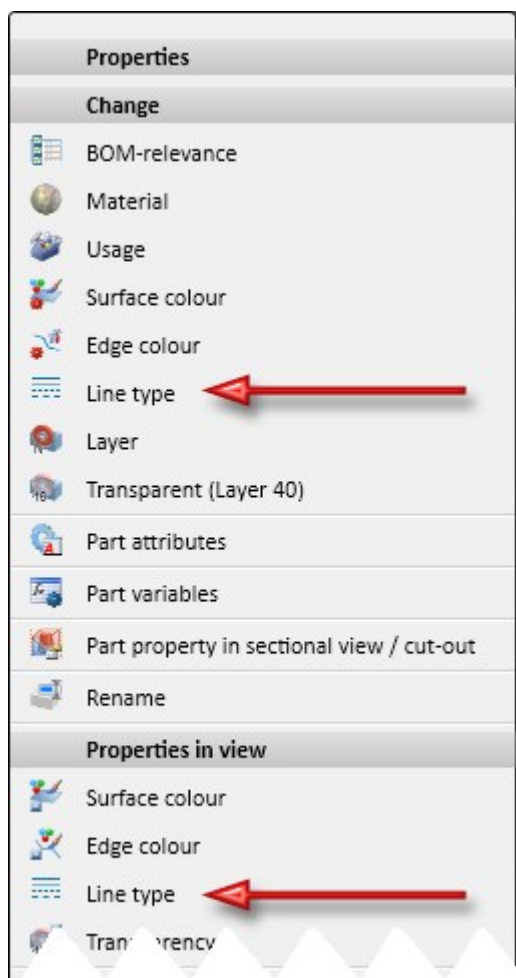
- Active view as front view  and
- Active view as top view 

can now also be applied to Steel Engineering plates. This allows you to specify the alignment of plates in Workshop drawings.

## Line types of 3-D parts

- For all 3-D parts to which an edge colour can be assigned, the context menu now contains the **Line type**  function. This function can also be found at **3-D Standard > Tools > Attr.**  **> Properties....**
- The line type can be assigned to all views, or only to the currently active view.

*Context menu for parts*



- The function can also be applied to multiple selections.

## Views

### Settings/configuration for views

The "Automatic ViewAll" parameter in the Configuration Editor has been renamed to **Automatic ViewAll after projection change**.

The dialogue text have also been modified.

Use the parameter **Automatic ViewAll after projection change** to specify when the ViewAll representation is to be applied automatically:

- **Never**  
ViewAll will not be activated automatically. An exception are the functions **4 standard views** and **6 standard views**. Here, ViewAll will always be activated automatically.
- **After calling a standard projection**  
ViewAll will automatically be activated when you call a standard projection. This applies to:
  - Front, side and top view, and bottom view, rear view,
  - Axonometries,
  - Isometries and
  - Active XY-plane in screen plane
- **Also after step-by-step rotation**  
ViewAll will also be automatically activated after a step-by-step rotation. This is the default setting.



## Create/process sketches

### Creation of spirals

The creation of spirals has been optimized in such a way that, while space requirements remain unchanged, a considerably higher precision can be achieved. This applies only to spirals with constant radius and constant lead.



### Transform/Clone dialogues

The dialogue windows for Sketch > Transform and Sketch > Clone have changed as follows:

- The term **Number** has been replaced with the term **Clones**. This concerns all Sketch > Clone functions. This means that the actual number of clones must be entered in the dialogue now, the original is no longer included.
- The term **Length** has been replaced with the term **Distance**. This concerns the following functions:
  - Sketch > Transform > Move
  - Sketch > Transform > Rotate >  Move + Rotate
  - Sketch > Clone > Move
  - Sketch > Clone > Rotate >  Move + Rotate.
- The text **Distribute evenly across length** has been changed to **Total distance**, the text **Distribute evenly across angle** has been changed to **Total angle**. This concerns the following functions:
  - Sketch > Clone > Move
  - Sketch > Clone > Rotate

Furthermore, the **Sketch > Clone > Rotate** function now offers the additional selection option **Evenly, 360°** that enables you to distribute original and clones with even distances along a full circle.



- When you select a direction, the text *Displacement direction* in the input field has been replaced with the text *Direction* angezeigt. This concerns the following functions:
  - Sketch > Transform > Move
  - Sketch > Transform > Rotate >  Move + Rotate
  - Sketch > Transform > Scale
  - Sketch > Clone > Move
  - Sketch > Clone > Rotate >  Move + Rotate.
  - Sketch > Clone > Scale

### Transform - New option

The Transform functions for sketches now allow you to specify the handling of connected lines during line transformations.



#### Scale lines, individually


If this option is active, only the actually selected lines (highlighted red) will be scaled. Connected lines will not be adjusted.



#### Preserve connected lines and adjust length

If this option is active, HiCAD will attempt to preserve the position of the connected lines if possible. In the process, the length of the lines to be scaled may be adjusted by means of new intersections. For distances the angle between connected lines will be preserved.


### Close polyline

The "Close c-edge, with line" function has been renamed to **Close polyline** . Also, the dialogue window of the function has had a redesign.

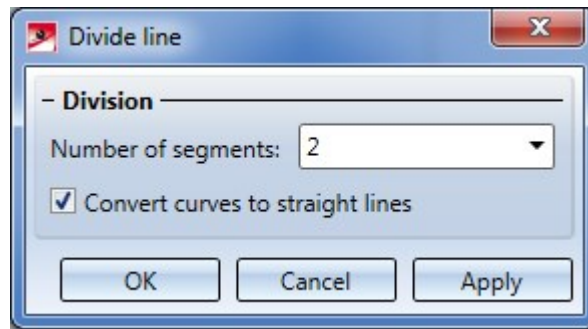
- Move the cursor near an edge of the polyline you want to close. If a closing of the polyline is possible, a preview image will be displayed.
- Apply the connection with a left-click. The function remains active, i.e. you can close further polylines if desired. Press the middle mouse button to exit the function.
- Right-click to open a context menu.
- Polylines of non-active sketches can also be closed with this function. The created new connecting lines always belong to the processed sketch.
- You can use UNDO to revoke your last action.

### Divide line



The **Divide, Number**  function has been enhanced. ist erweitert worden. If you want to convert the segments to lines, activate the new **Convert to lines** checkbox:





The context menu of the function offers the following new functions:



#### All lines

Divides all lines of a polyline. Curves will not be considered.





#### All curves

Divides all curves of a polyline. Lines will not be considered.

The following functions are no longer available:

- Divide polyline, All lines
- Divide c-edge, All curves and
- Divide c-edge, All edges.

## Change purpose


The purpose of planar sketches can now be changed. To do this, select **Sketch > Tools > Sort GE**  **>** .


**Change purpose.**

## Weld seams

### Settings for Insert new weld seam / Insert weld symbol functions

The last specified settings for the

- Insert new weld seam  and

- Insert weld symbol 

functions will be the new default settings (also after a restart of HiCAD).

## Variants

### Variables names in variants

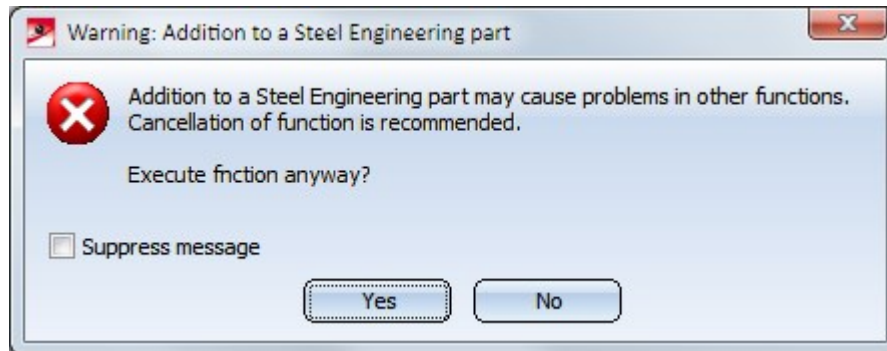
The maximum length of variables names has been increased from 4 to 255 possible characters.

## Other new features and enhancements

### Adding of 3-D parts to Steel Engineering parts

The adding of 3-D parts to a Steel Engineering parts may cause problems with subsequently called functions, e.g. mitre cuts, Steel Engineering connections, NC/NCX export, deriving of drawings, etc. Attributes (e.g. weight, length, etc.) may also be calculated incorrectly. In such cases, HiCAD will display the following warning message

after selecting 3-D-Standard > Process > Add  function :



You can then decide whether you want to execute the function or not.

If you do no longer want this message to be displayed, activate the **Suppress message** checkbox. In this case, the adding will either be accepted or refused, depending on your last choice in the message window.

Please remember that the checkbox status applies only to the current HiCAD session!

## Major Release 2016 (V 2100)


### Performance

#### HiddenLine representation

The HiddenLine calculation could be speeded up significantly - in various test drawings, speed increases between 20% and 80% could be achieved.






### Changed '3-D Standard' Ribbon

#### Clone processings/feature

Use the new **Clone processings**  function to copy existing features onto other surfaces. If the same feature steps are required in several places of a part, this function can help you to save a lot of time. You can find the function at **3-D Standard > Process** and in the Feature context menu, where it is called **Clone feature**.

Extensive information on this function can be found in the Feature Technology Help.

#### Add/Subtract

- The Subtract, via rotation  function can now be found in the pull-down-menu  of the **Subt.**  function.
- The Add, via rotation  function can now be found in the pull-down menu of the **Add**  function.

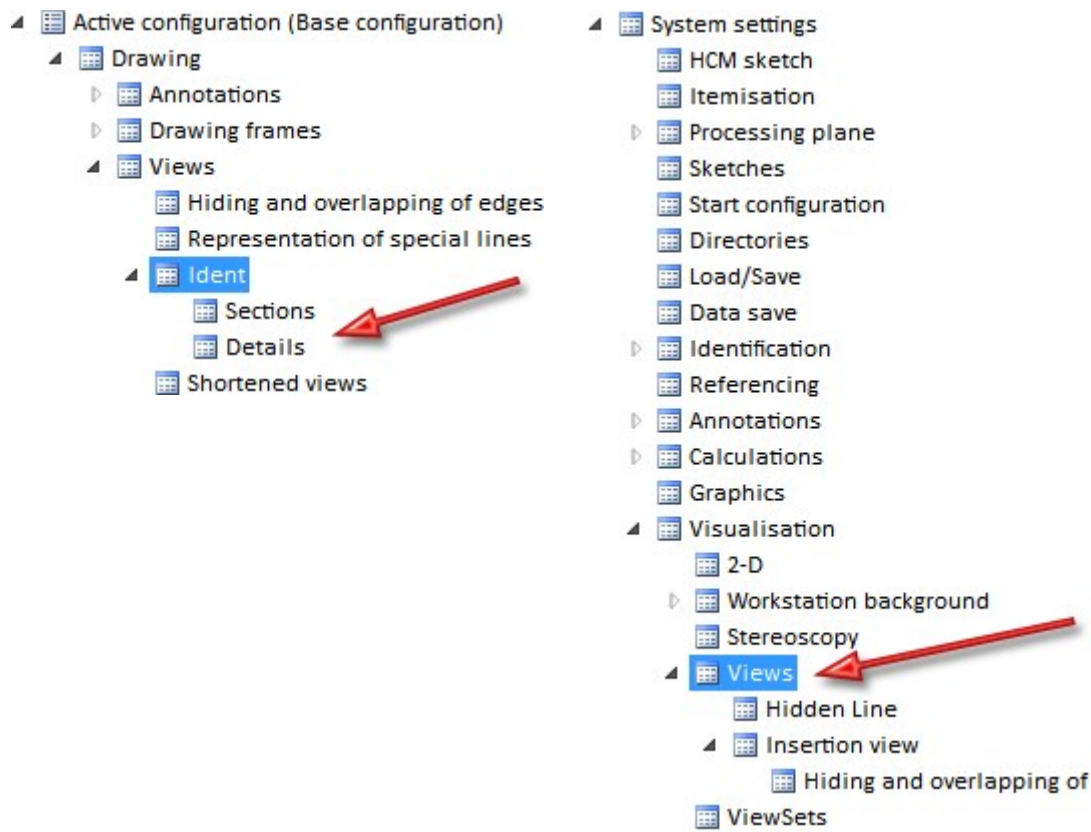
### Views - Settings and editing

#### Settings for views (ANSGEN.DAT)

The settings for views, which were previously specified in the system file ANSGEN.DAT have be transferred to the Configuration Editor. You can now find these setting options at

- Drawing > Views
- System settings > Visualisation
- System settings > Visualisation > Views
- System settings > Visualisation > View group

Here, you can now separately specify the settings for sectional and detail views, and also specify the settings for the representation (colour, line type and layer) of the frames for view groups for drawing derivations.



Please also read the News for the Configuration Editor.

### Sorting and numbering of sheet areas

Sheet areas can now be sorted and re-numbered.

The display order of the Sheet areas can be changed conveniently via Drag & Drop. You can move individual Sheet areas as well as several Sheet areas in one step.

Individual Sheet areas, but also multiple selections of Sheet areas can be re-numbered. The first number is freely selectable. The other numbers will be assigned consecutively, in the display order of the Sheet areas.

Also new is the Sort, ascending function that allows you to sort the the Sheet areas by Sheet numbers in ascending order, e.g. after changing of Sheet numbers or after Drag & Drop actions.





The display order of the Sheet areas will be saved together with the drawing.

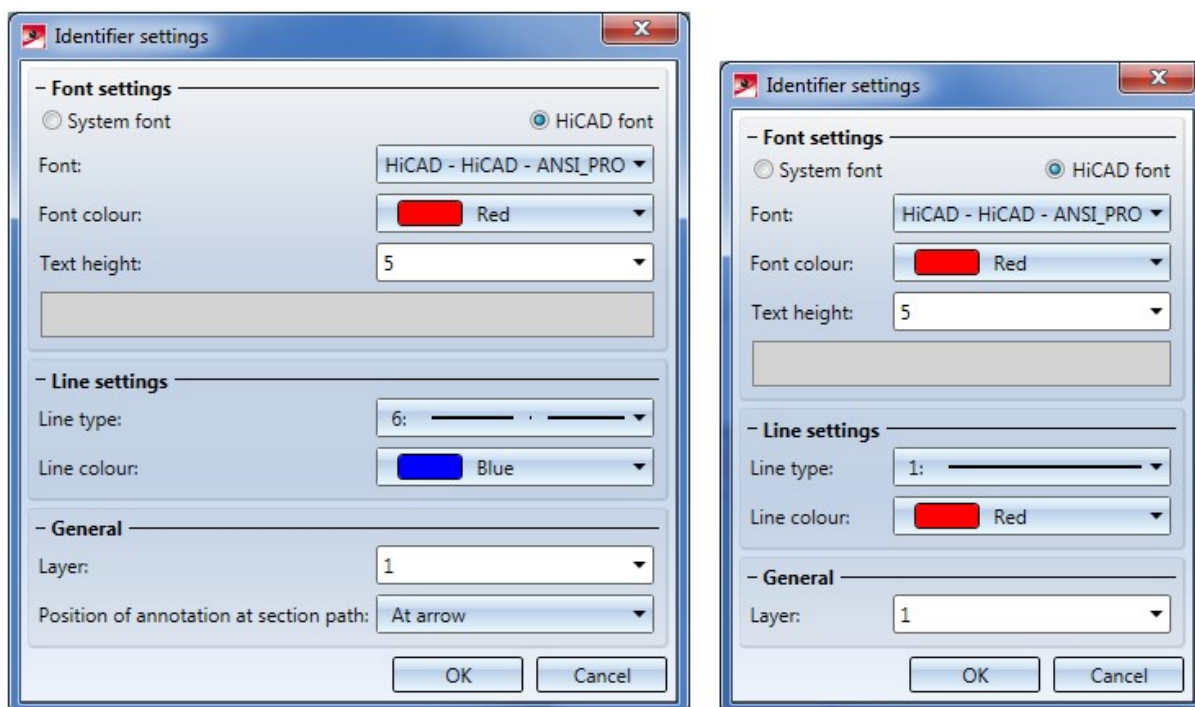
## Enhanced options for sectional/detail views

For the definition of sectional and detail views you can now also specify the text and line parameters of the view identifiers.

For this purpose, the following options are available in the dialogue window of each function:



-  Change text and line parameters (sectional view) and
-  Change text and line parameters (detail view)




The default setting of these parameters can be specified in the Configuration Editor, at **Drawing > Views > Ident.**



Text and line parameters for sectional view (left) and detail views (right)


To change **all** identifiers in a sectional view or detail view, use the functions



-  Change sectional view identifier, All, and
-  Change detail view identifier, All.

You can find these function in the pull-down menus  **Views > Edit > Section**  or **Detail** .


## Structure and Properties

### Delete main part identifier

In the context menu of 3-D parts you will find the new **Delete main part identifier**  in the sub-menu of **Part/Assembly structure**.

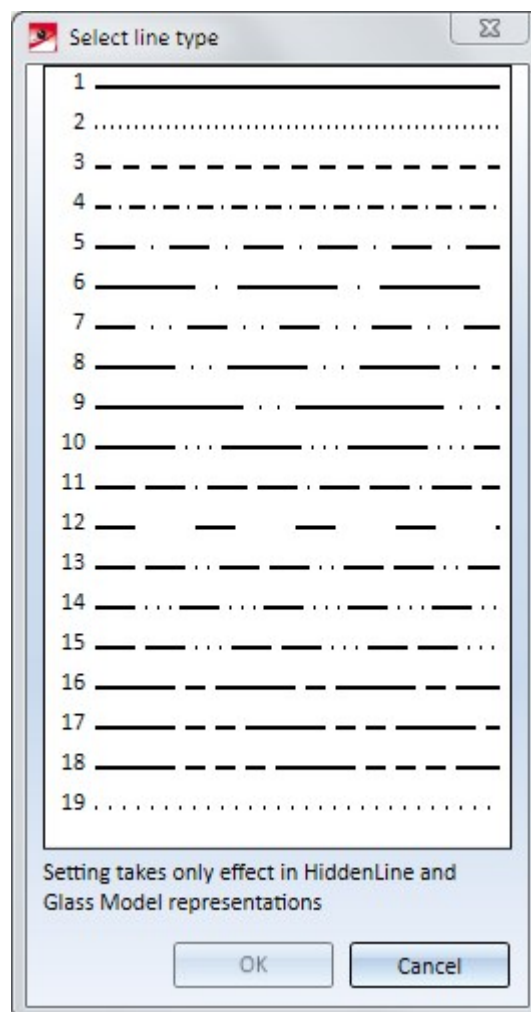
Use this function to delete the identifier of an assembly main part. . The part will become a "normal" part and the identifier  will be removed. This function makes sense in welded Steel Engineering assemblies, where you want to remove the identifier of an assembly main part that was created in the welded assembly.

### Line type of 3-D solids

Use the Line type  function to assign a line type to 3-D parts of the type "Solid". To call the function, right-click the 3-D part and select **Properties > Line type**.

In this way, you can, for instance, display the contour edges of neighbouring parts by dash-dotted lines, as is quite common according to certain standards.

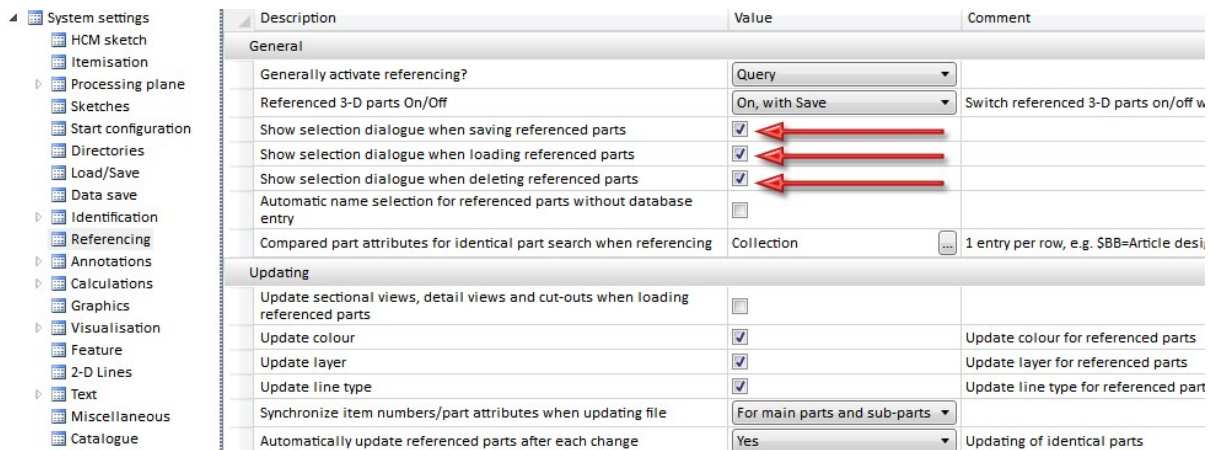
Select line type





## Please note:

- The setting takes only effect in HiddenLine and Glass model representations.
- The selected line type also applies to highlighted edges.
- In connection with this function, the settings for referenced parts in the Configuration Editor have been revised. There you can specify at **System settings > Referencing**, separately for colour, line type and layer, whether these parameters are to be updated when loading referenced parts.



## 3-D dimensioning

### Dimensioning symbols

When placing dimensioning symbols with the functions available at 3-D Dimensioning + Text > Edit > Symbols... and during editing of dimension figures the additional

### Envelope condition

symbol according to DIN EN ISO 8015 (Envelope conditions for fits) is now available.

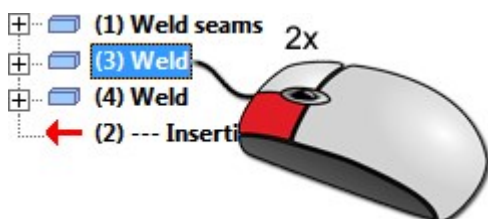
## Standard Parts and Boltings

### Insert weld seams

The function of the **Apply** button has been changed. When you now click this button, the weld seam will be created with the specified parameters. After creation of the tag the **Weld seam** function will remain active, and the selection of parts to be welded will be requested repeatedly.

### Process weld seams

- The dialogue for weld seam creation can now also be opened by double-clicking the **Weld** feature.



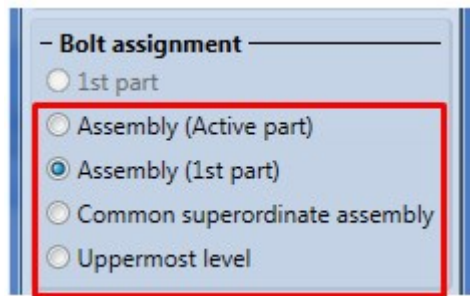


## Spring connectors

When inserting spring connectors you can now also select the edge of the bore instead of the insertion point.

## Bolting settings - New 'Bolt assignment' options

The settings dialogue for boltings now offers a number of new Bolt assignment options:



The new functions enable you to bolt parts together that are located in locked assemblies.

## Derive/edit sketches

### Purposes of planar sketches

HiCAD automatically assigns a purpose to planar sketches upon their creation. One distinguishes between

- Purpose = Part
- Purpose = Create/Edit

Sketches with the purpose **Create/Edit** are, in principle, "auxiliary elements" from which other parts, processings and sectional views, detail views or cut-outs. Sketches with this purpose are visualized with broader lines in the active view. They are also visible in shaded models.

Sketches with the purpose **Part** are regarded as "normal" parts that are used as elements of the drawing. They distinguish themselves from sketches with the purpose **Create/Edit** as follows:

- In cut-outs, sectional views and detail views the sketch will be cut.
- The sketches have no special representation and are not shown in shaded models (without edges).
- The sketch can be edited with the functions available at 3-D-Standard > Tools > Crossh., i.e. it may contain centre lines, crosshairs etc.

When a new sketch is created, HiCAD automatically determines its purpose. For example, sketches that are created in connection with View functions, during deriving of new 3-D parts from a sketch, during processing of 3-D parts with sketches etc. will obtain the purpose **Create/Edit**.

In contrast, if a sketch is created with the functions of the **Sketch** tab, the purpose specified in the Configuration Editor will be assigned. The default setting is **Create/Edit**. This default setting can be changed in the Configuration Editor, at **System settings > Sketches**.

System settings	Description	Value	Comment
HCM sketch	Purpose for new sketches	Always Create / Edit	
Itemisation	Purpose for new sketches in Metal Engineering grid	Always Create / Edit	Special case ME grid
Processing plane	Sketch representation for cut-out/sectional/detail view	Only in active view	
Sketches	Sketch representation for creation of new bodies	In all views	
Start configuration	Sketch representation for processing of bodies with sketch	In all views	
Directories			
Load/Save			

## Purpose for new sketches / new sketches in Metal Engineering grid


This setting determines which purpose will be assigned when a sketch is explicitly created with the functions of the **Sketch** tab (default: **Always Create/Edit**). The alternatives are **Always part** and **Always query**. If the setting **Always query** is used, you will be asked to specify the purpose whenever you create a new sketch. For sketches which have been created via the Metal Engineering plugin the setting **Always query** is not available.

## Sketch representation for...

This setting concerns sketches that are created during definition of sectional views, detail views and cut-outs. These sketches can either be shown in all views (default) or only in the active view.

## Subsequent changing of the purpose

To change the purpose of a sketch, right-click the sketch either in the drawing or in the ICN and select **Further >**

Change purpose .

## Fillet/chamfer sketches

You can now also use the functions Sketch > Process > Fillet  and Sketch > Process > Chamfer  to process corners of non-active sketches or 3-D sketches. Please note however that HCM constraints that cannot be kept to will be deleted.



An exception is the selection of corners via selection rectangle. Here, only elements of the active sketch will be considered.

## Behaviour of the middle mouse button

The behaviour of the middle mouse button has changed with regard to various functions. If you middle-click in the drawing while the dialogue window is displayed, the following will happen:

- If enough inputs have been made, the function will be executed. The dialogue window will remain open (corresponds to the **Apply** option).
- If not enough inputs have been made, the function will be ended.

This behaviour applies to the following Sketch functions:



Offset with end caps



Fillet



Chamfer

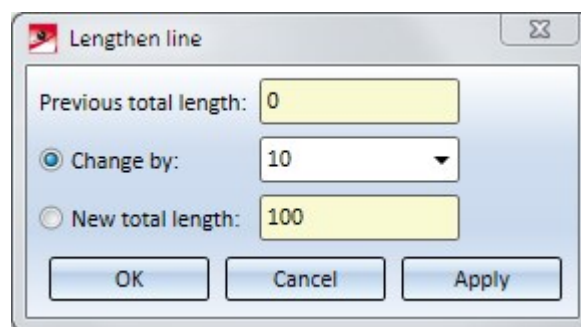


Divide, Via number of segments

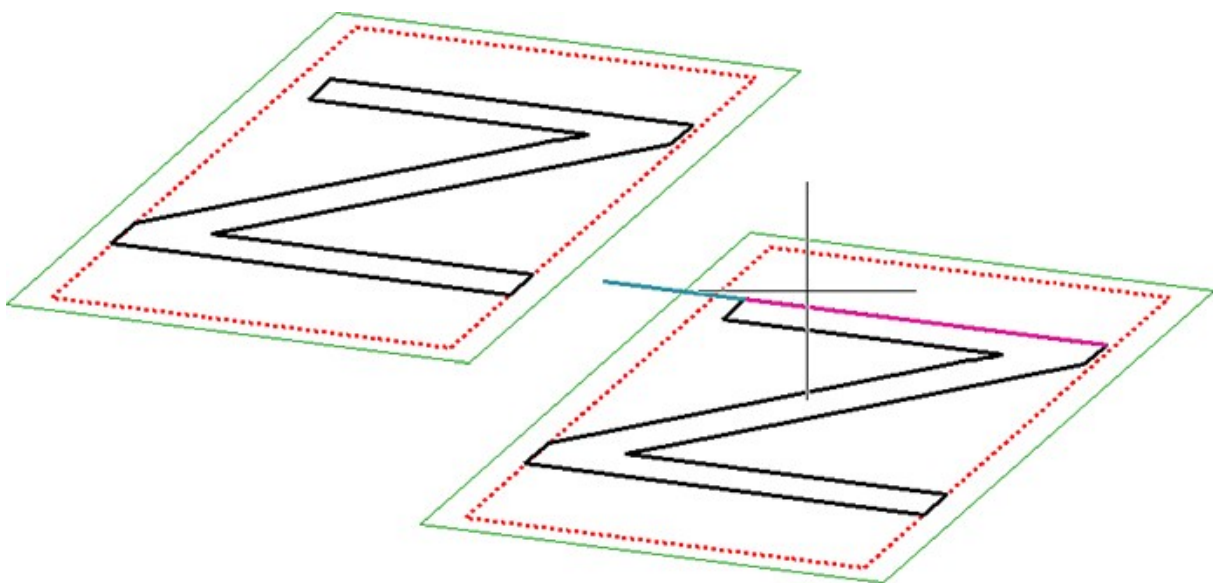
### Lengthen line, by value



The Lengthen line, by value function has had a complete redesign. Graphical elements, i.e. straight lines and arcs of a sketch, can now not only be lengthened/shortened by a particular value, but also by specifying a new total length.



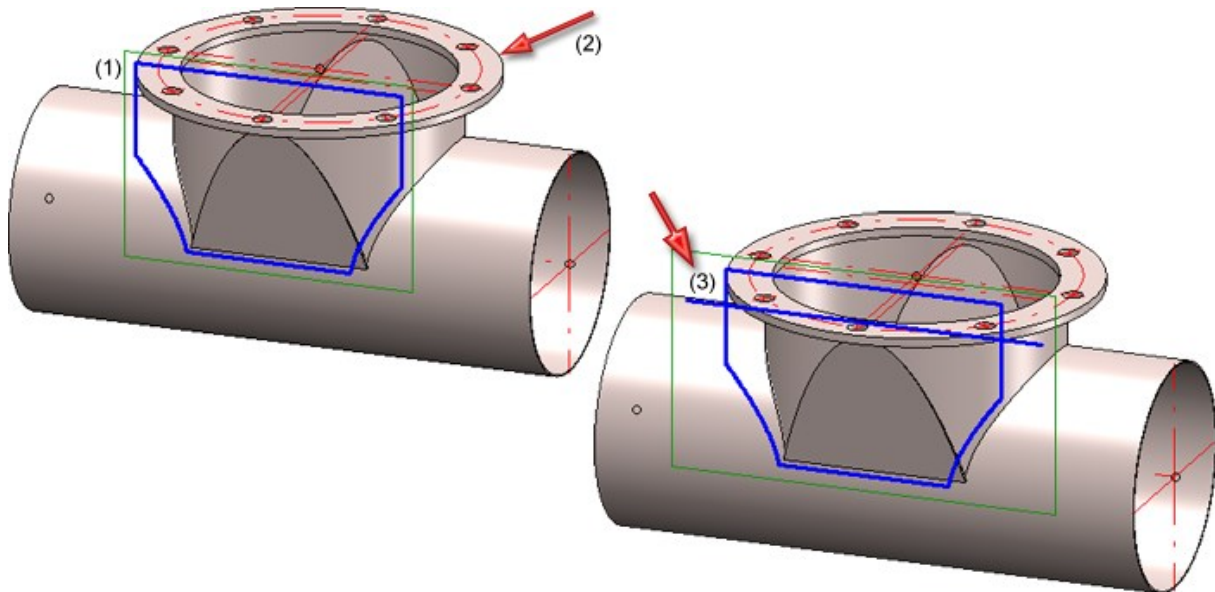
When you move the cursor over the lines of a sketch while the dialogue window is open, a dynamic preview of the lengthening or shortening will be displayed.



Also new is that HCM constraints will be considered by this function.

## Sketches - Projection with automatic clean up of overlaps

- Conic section edges (circles, ellipses, hyperbolas, parabolas) the planes of which contain the projection direction, often produce overlaps in the projection. HiCAD automatically recognizes and cleans up such situations in vertical and directed projections, by replacing such edges with straight line segments. The same applies to NURBS curves in such planes.



(1) Active sketch; (2) Selected conic section; (3) Projected straight line segment

## Transform/Clone sketches

The Transform and Clone functions for graphical elements of planar sketches have been redesigned.

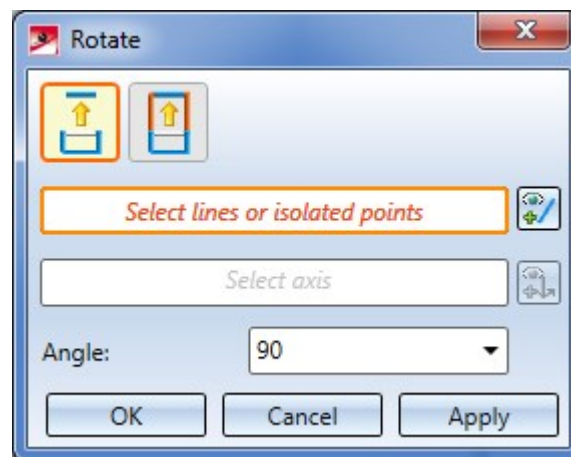


For example, the Move, Rotate, Mirror and Scale functions for graphical elements have been combined into one central function that can be used for planar sketches and 3-D sketches. The same applies to the Clone functions.

The old functions in the pull-down menus are not available any more, since the new functions cover almost all special cases. Exceptions are the functions Similarity, Affinity and Move+Rotate, which are no longer there.

The Undo/Redo functions for sketch transformations have been withdrawn as well. To revoke transformations of sketch elements, you use the general Undo function now. To apply the last performed transformation to other sketch elements, use the **Apply** button of the function dialogues.


Example: New **Rotate** dialogue window



In all functions, the selected elements, the displacement directions, the rotation axis etc. will be visualized dynamically, and a graphical preview of the transformation will be displayed.


## Simplify parts


### Delete superordinate parts

Sometimes, a part structure contains levels and dummy parts that are bothersome during simplification. Use the new **Delete superordinate parts**  function to solve this problem. You can find the function on the **Simplify** tab.



This function deletes all "superordinate parts" in the part structure. This means that all parts that are only there for the sake of structuring will be removed and the remaining parts will be shifted to the uppermost level of the part structure. Main assemblies, assemblies and dummy parts will be deleted, their sub-parts will be preserved. Assemblies and dummy parts without sub-parts will also be deleted.



### Merge all solid

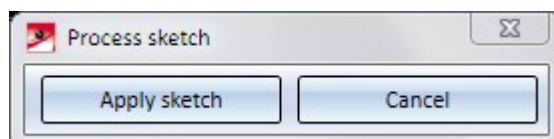
Use the new Merge all solids  function to merge all parts of the type "Solid" that belong to a part list (multiple selection) to one new solid by means of a Boolean operation. Proceed as follows:

- Define the part list, e.g. using the context menu functions in the ICN or the part filters.
- Call the **Merge all solids**  function.
- Specify whether the original solids are to be retained or deleted after merging

## Envelope with sketch processing

When using the Envelope  function in connection with the Translation  option you can now, for each of the three defined directions, process the sketch on which the corresponding extruded envelope is based, without having to exit the function. For this to happen, you first need to click the **Preview** button.

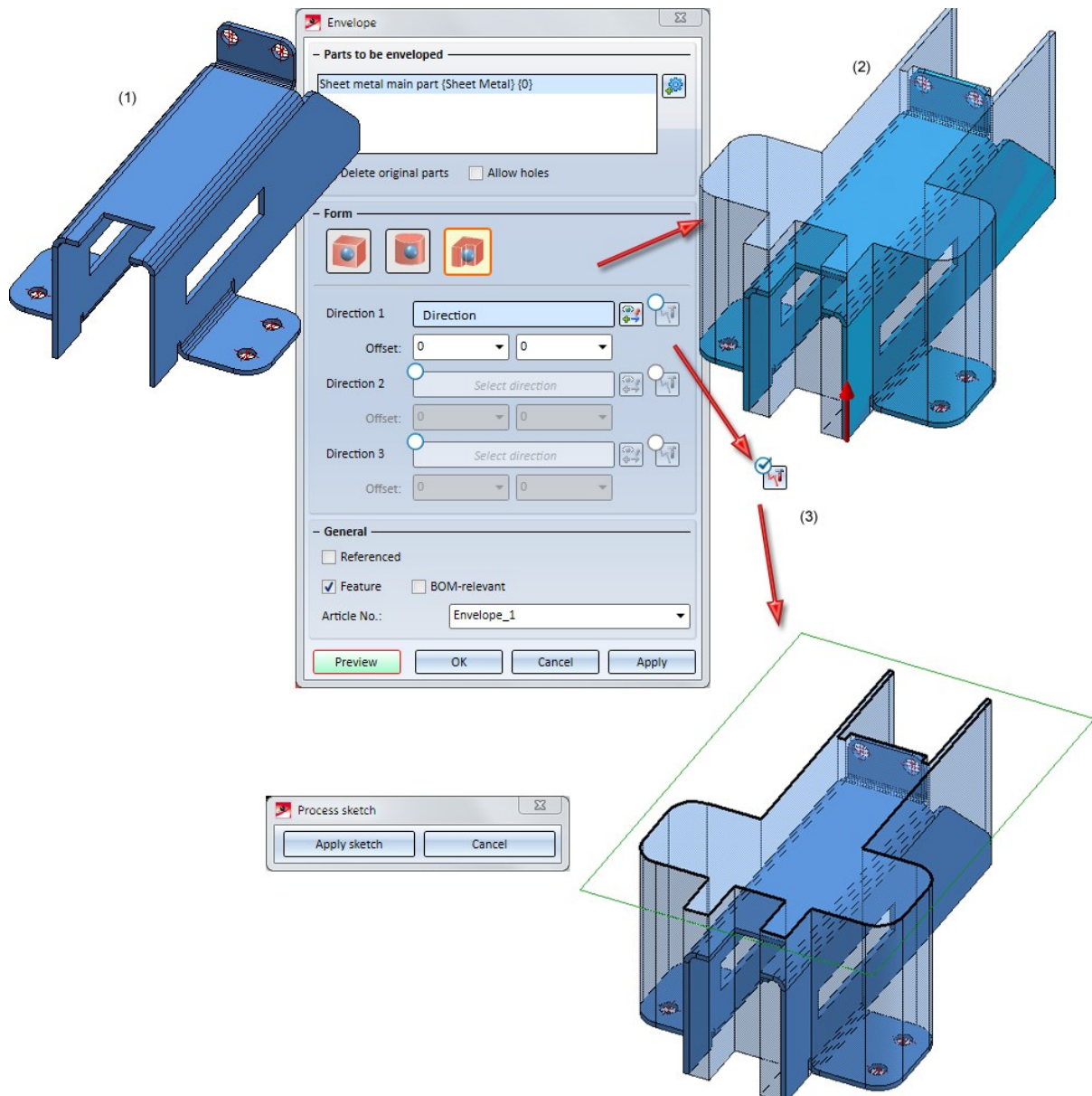
Click the **Manual/Automatic**  radio button for the corresponding direction, and then, click the **Process sketch**  symbol. The corresponding sketch and the following dialogue window will be displayed:



You can now process the sketch with the Sketch functions. After completion of the processing, confirm with **Apply sketch**. The envelopoe will then be adjusted accordingly.





Example - (1) Original part; (2) Extruded envelope; (3) Sketch to be processed



## Surface and Edge functions

### Copy / Take surface - not for assemblies

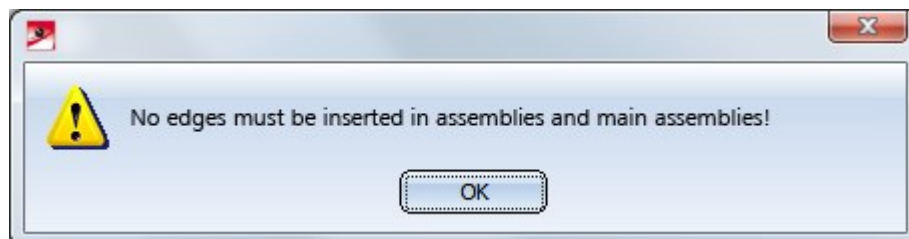
For the functions Copy surface  and Take surface  assemblies are not allowed as target.





## Insert edges - not for assemblies

Free edges cannot be inserted into assemblies and main assemblies.



## Catalogue Editor

### Service Pack 2 2016 (V 2102)

#### New system attribute: Order number

In the **System Settings** catalogue a new attribute is now available in the **System attributes** table:

**Order number** (Attribute name \$ON)

This attribute will, for instance, obtain a value when you directly insert parts (e.g. of the company HILTI) via electronic part catalogues which have been realized through solutions by CADClick.

#### "Filler plate" usage

At **Factory standards > Usage > Civil Engineering > Steel Engineering** you can now find the new **Filler plate** usage in the **Plate type** table.

	ID	MOD	STATUS	BZ	CONFIGKEY
1	5		▶	Filler plate	FILLER-PLATE
2	4		▶	Base plate	BASE-PLATE
3	2		▶	Top plate	TOP-PLATE
4	1		▶	Stiffener	STIFFENER
5	3		▶	Reinforcement plate	REINFORCEMENT-PLATE

This usage will be automatically assigned to filler plates upon insertion of the corresponding Steel Engineering connections.

#### Composite panels as semi-finished products

Composite panels are now available as Sheet Metal semi-finished products at **Factory standards > Sheets > Composite panels**. These composite panels consist of two 0,5 mm aluminium cover sheets with a mineral core. In HiCAD the plate is represented simplified as a one-layer sheet. Since the weight calculation is done based on a homogeneous material, an own material is assigned to the pane.

#### New column for multiple tables

At **Extras > Catalogue** you will now find the new **New column for several tables** function, that allows the adding of a new column to multiple (or all) tables of a catalogue in one step.

To do this, activate the desired catalogue and call the function.

In the displayed dialogue window, specify the properties of the new column on the right. On the left, all tables belonging to the catalogue are listed. Click to select the tables to which to want to assign a new column. Multiple selections in one step are possible by simultaneously holding down the CTRL or SHIFT key. As soon as you close the window with **Apply**, all selected tables will be processed.

## RAL colours

The RAL colours in HiCAD were previously defined in the system file RALRGB.DAT. As of SP2, the RAL colours are defined in the RAL CLASSIC table. You can find this table in the Catalogue Editor at **Textures and Colours > Colours**.

## CUSTOM columns

For fasteners the further CUSTOM columns CUSTOM6 - CUSTOM9 are now available, allowing you to assign customer-specific data to fasteners. For instance, this provides access to HELiOS attributes which are not of the type STRING.

Column	Attribute	Data type
CUSTOM6	free	Integer
CUSTOM7	free	Integer
CUSTOM8	free	Floating point
CUSTOM9	free	Floating point

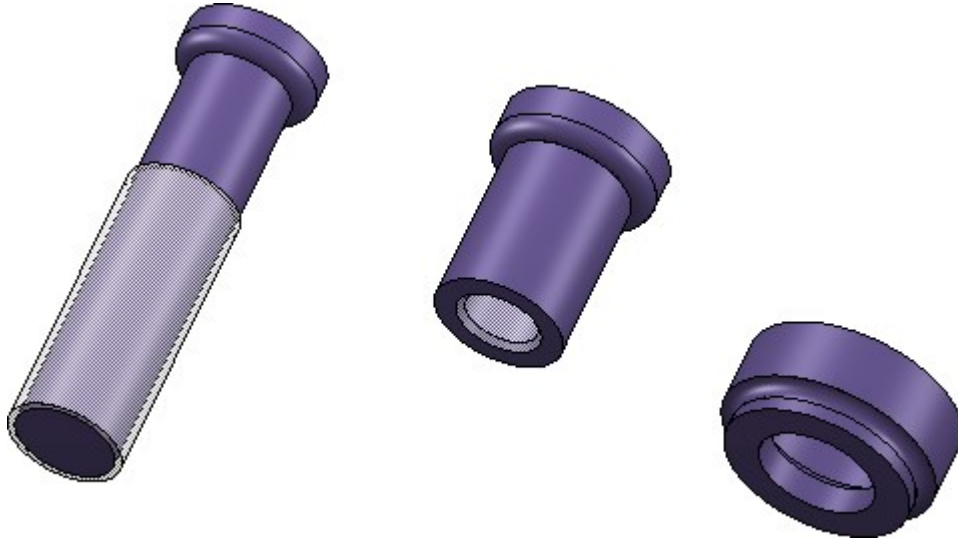
The columns have the initial value 0.

## Service Pack 1 2016 (V 2101)

### New studs and washers acc. to DIN EN ISO 13918

The catalogues for fasteners of the type Studs and Washers acc. to DIN EN ISO 13918 have been expanded.

- In the catalogue **Fasteners > Washers > Ceramic rings** you can now find
  - Ceramic rings of the type PF, RF and UF.
- In the catalogue **Fasteners > Studs > Welded studs** you can now find the following new studs:
  - Welded studs of the type ID, IS and IT
  - Welded studs of the type UD, US and UT
  - Threaded welded studs of the type RD, PD, PS and PT



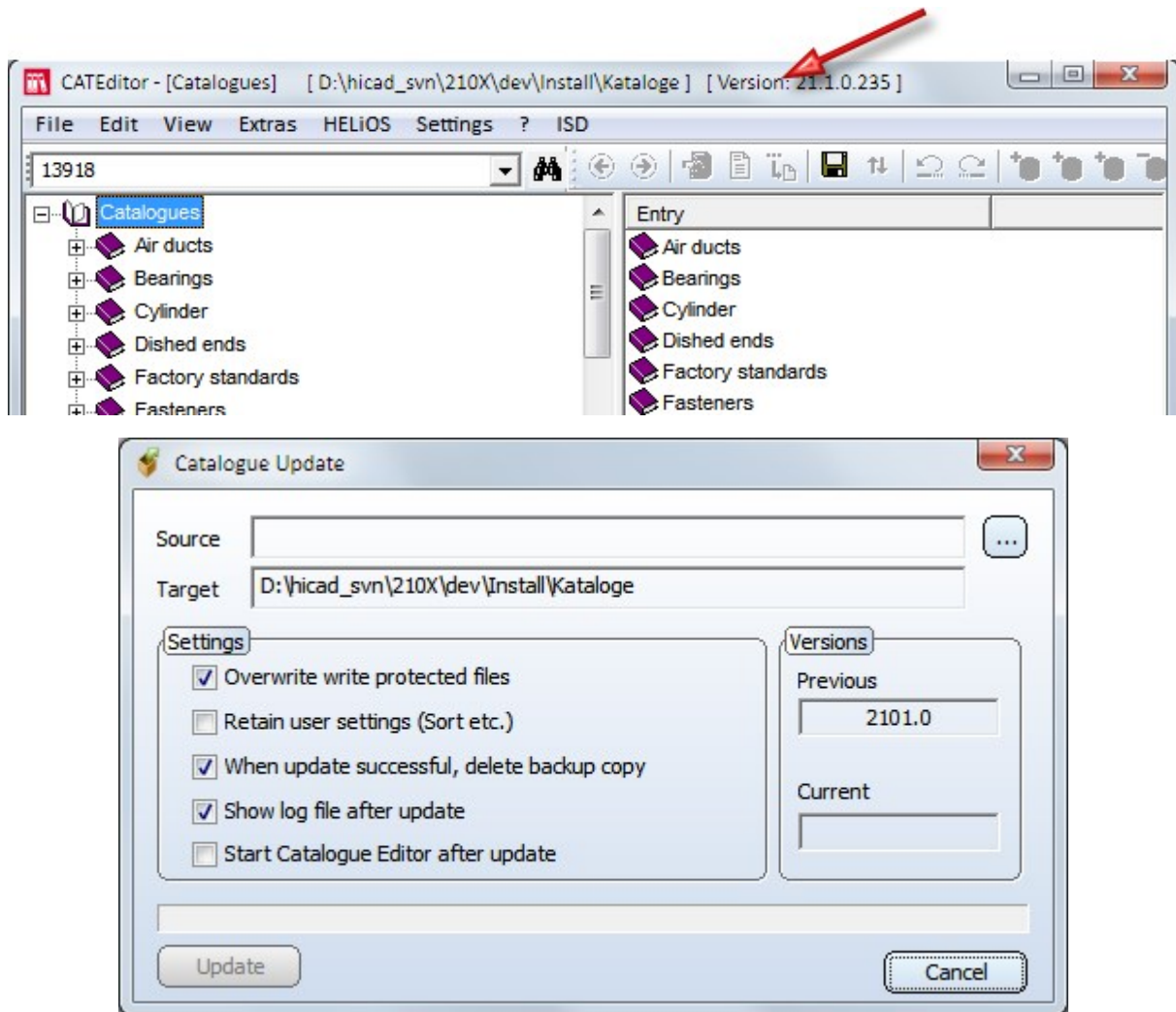
## Pipes acc. to DIN 10305

New in the **Semi-finished products > Beams + Profiles > Steel pipes** catalogue are pipes acc. to DIN 10305:

- 10305-1 Seamless cold-drawn pipes
- 10305-2 Welded cold-drawn pipes
- 10305-3 Welded custom rolled pipes
- 10305-4 Seamless cold-drawn pipes for hydraulics and pneumatics
- 10305-5 Welded custom rolled pipes with quadratic and square cross-section

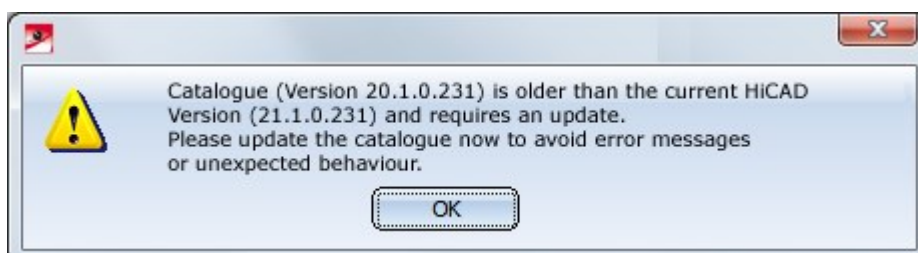
## Checking of catalogue directory upon HiCAD start

It will now be checked to which HiCAD version the catalogue directory belongs. The version is shown in the caption of the dialogue window of the Catalogue Editor. When performing an update with the CatalogueUpdate.exe tool, the corresponding HiCAD version will be shown in the dialogue window of the tool.

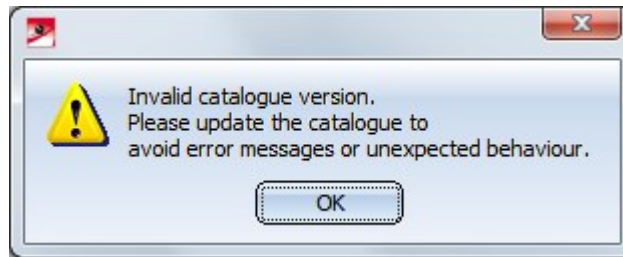


The version of the utilized catalogue directory will be checked upon HiCAD start. If the catalogue directories are old there is a risk that HiCAD will search for catalogue entries which do not yet exist in the old catalogues. One example of this are the Textures that were integrated in the catalogue some time ago. If these are missing in the catalogue, you will have no access to these textures in HiCAD.

If you start HiCAD with an old catalogue directory (e.g. HiCAD 2100.1 with Catalogue 2100.0 or HiCAD 2100.0 with Catalogue 2002.2), a corresponding message will be displayed:



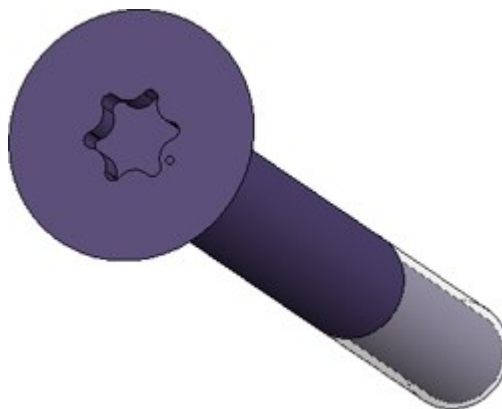
If the version of the catalogue cannot be detected, the following message will be displayed:



## Major Release 2016 (V 2100)

### New bolts + screws

- New in the catalogue **Fasteners > Bolts+Screws > Countersunk head bolts** are hexagonal socket countersunk head bolts acc. to **DIN EN ISO 14582**.



- The following bolts acc. to DIN EN 14399 are available:
  - Countersunk head bolts acc. to **DIN EN 14399-7**
  - Fitting screws acc. to **DIN EN 14399-8**
  - Hexagonal head screw acc. to **DIN EN 14399-10**
  - Pan head screws acc. to **DIN EN 14399-10 button-head**

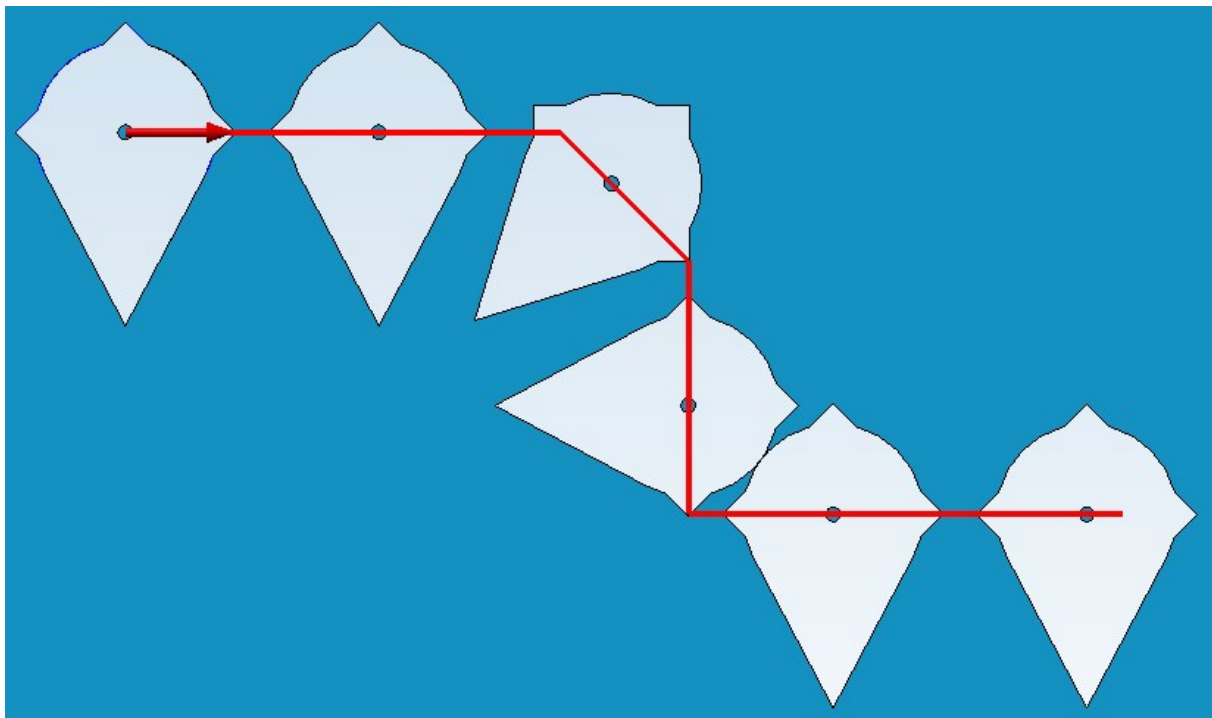


## Feature Technology

Service Pack 2 2016 (V 2102)

### Clone processing function - 2 new patterns

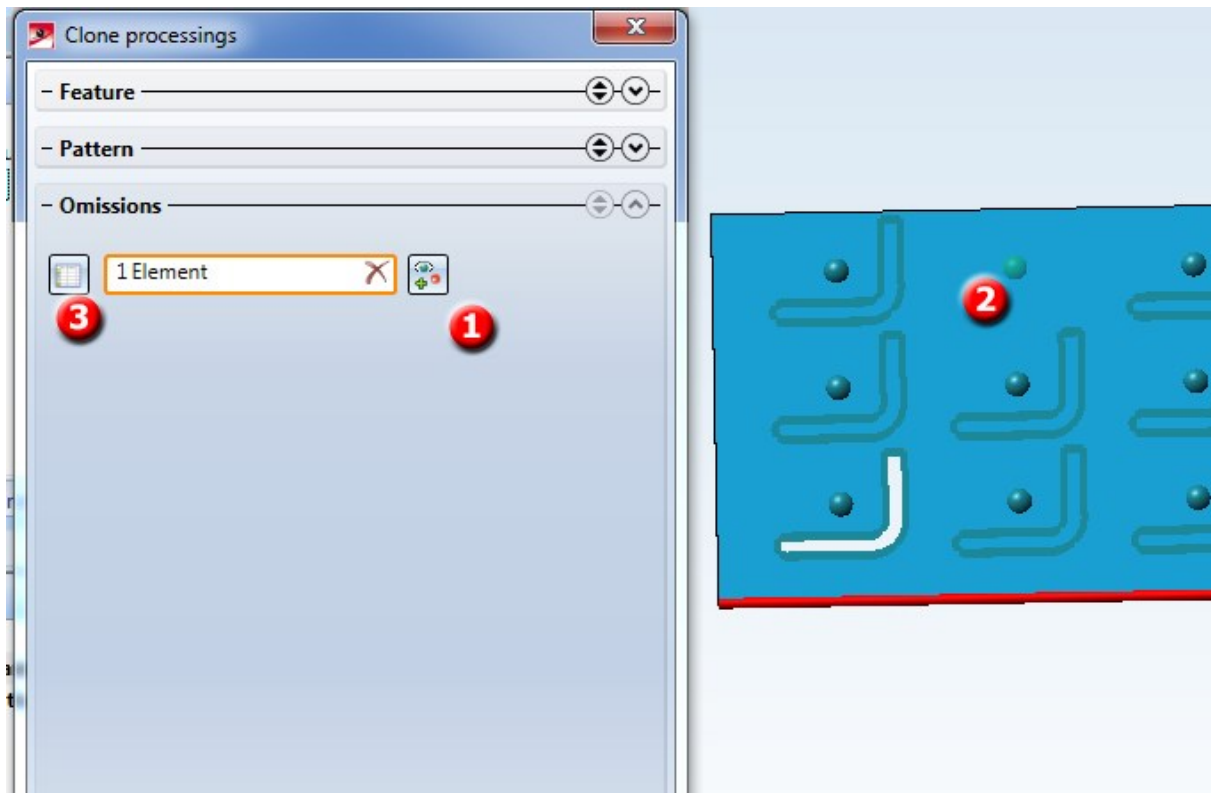
The 2 new cloning pattern types "Polyline pattern" and "Translatory and rotatory pattern" have been added to the **Clone processings** dialogue window. This allows you to define clonings along polylines and composite edges, or combined rotatory and translatory clonings.



Example of a subtraction along a c-edge

### Easier definition of omissions for cloned features

In the **Omissions** area you can specify individual clonings that are to be omitted. To do this, click on the **Select omission** button (1) and then identify individual omissions in the drawing (2).



Alternatively you can click **Show table** (3) button show the omissions in tabular form. One column per cloning pattern will be shown, with one row representing one omission. Click on the "Plus" button to add further omission; right-click a row and select **Delete row** to remove it again.

## Message when loading files from a newer HiCAD version

As of Version 2102 HiCAD will check upon opening of files whether these contain an incompatible feature log. Such situations may occur if the file was created with a newer HiCAD version.

In such cases a message will be displayed, warning you that if you save the file with your HiCAD version, the feature log or part thereof will be lost.

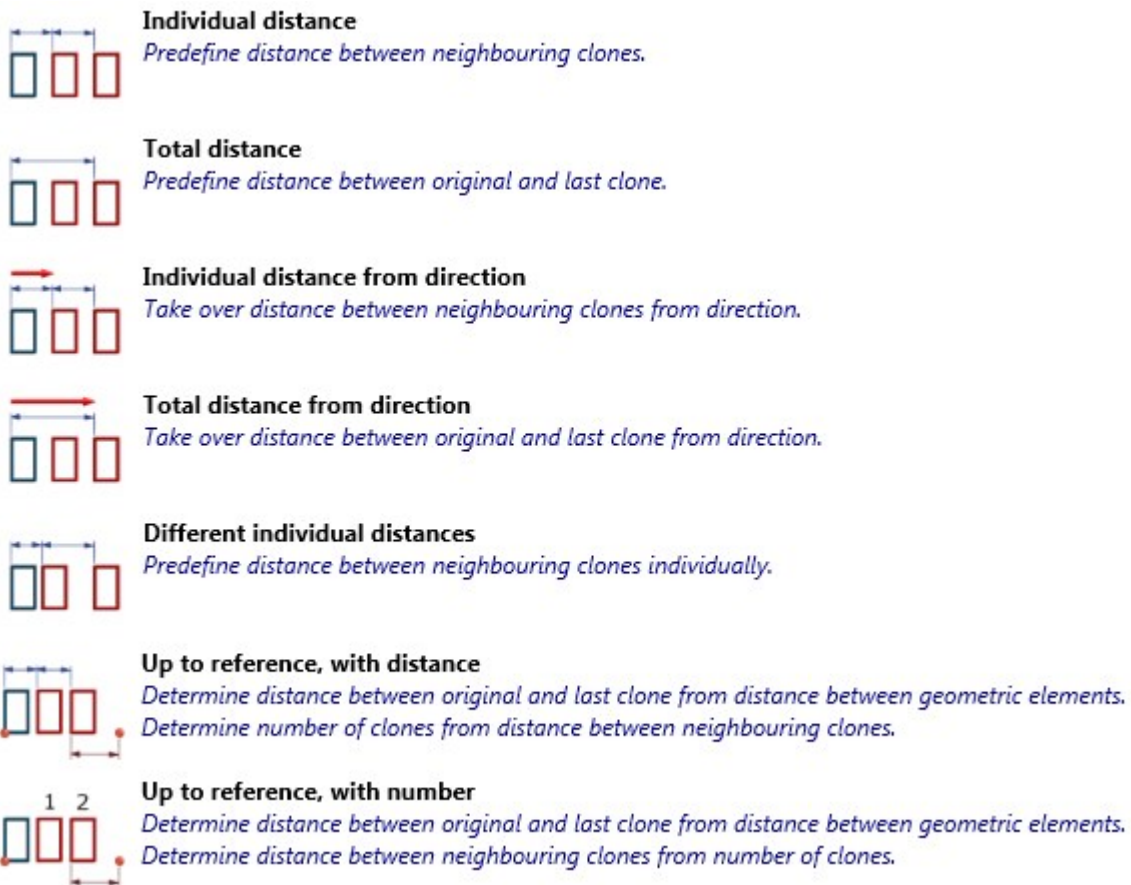
## Service Pack 1 2016 (V 2101)

### Move multiple features

It is now possible to move several features in one step: Mark the features by clicking them while holding down the CTRL or SHIFT key and move them with Drag & Drop.

### Additional 'Clone' options

The Clone processings function has been expanded: For linear patterns new options are available that enable a using of reference lengths for distance specifications. The dialogue window has been enhanced as well, now offering a small sketch and a short description of the distance and angle options:



You have now the option to define omissions in patterns, and to pick lengths and angles directly from the drawing. Furthermore, you can now also clone other features: Appearance, Move edges, Crosshairs, Wrap, Fitting CS.

## Recalculation of API variants after deletion of parts

Features created via API variants, such as stairs and railings, can now also be updated or modified after deletion of parts that were created by these variants. However, these deleted parts will be re-created if they are required according to the specified parameters or the geometrical situation.

## Lists as new part type for formulae

You can now use Lists as a new data type. These lists contain one or several values, formulae, or further lists. They can be used, for instance, as lists of omissions when working with the Clone processings functions.

## Feature-compatible deletion of crosshairs

Automatically inserted (e.g. after insertion of a bore) and manually deleted crosshairs will now remain deleted after feature recalculations and will not be restored automatically any more.

## Longer variables names for variants

The previous limitation of variables names to not more than 4 characters does no longer exist.

Variables names can now consist of up to 255 characters.

## Major Release 2016 (V 2100)

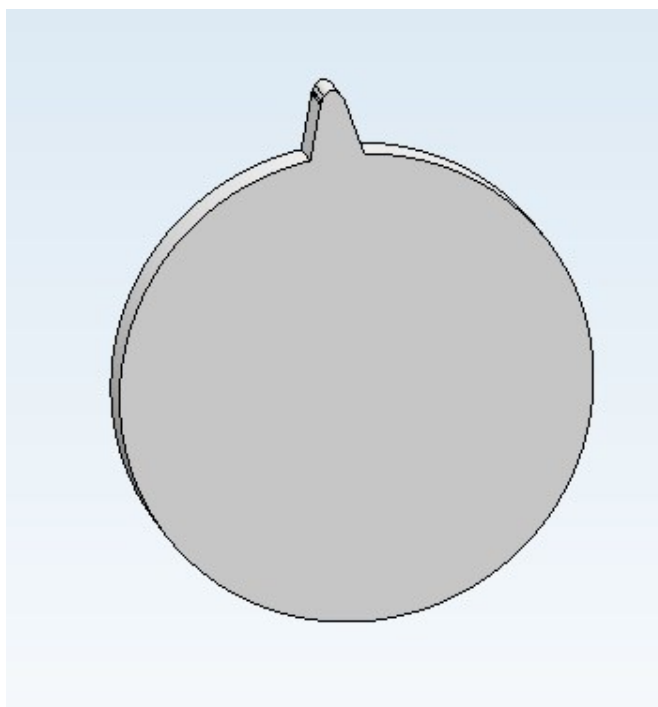
### Clone processings



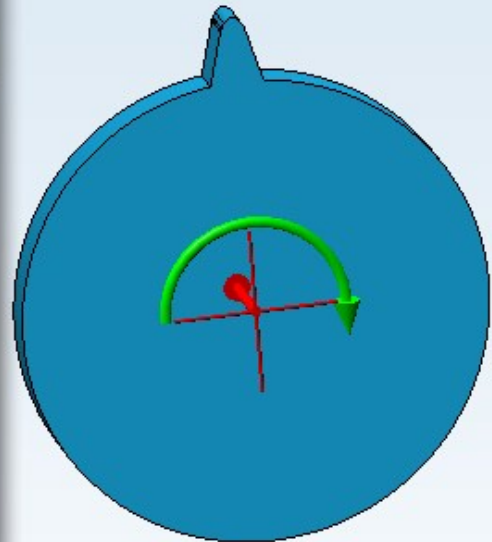
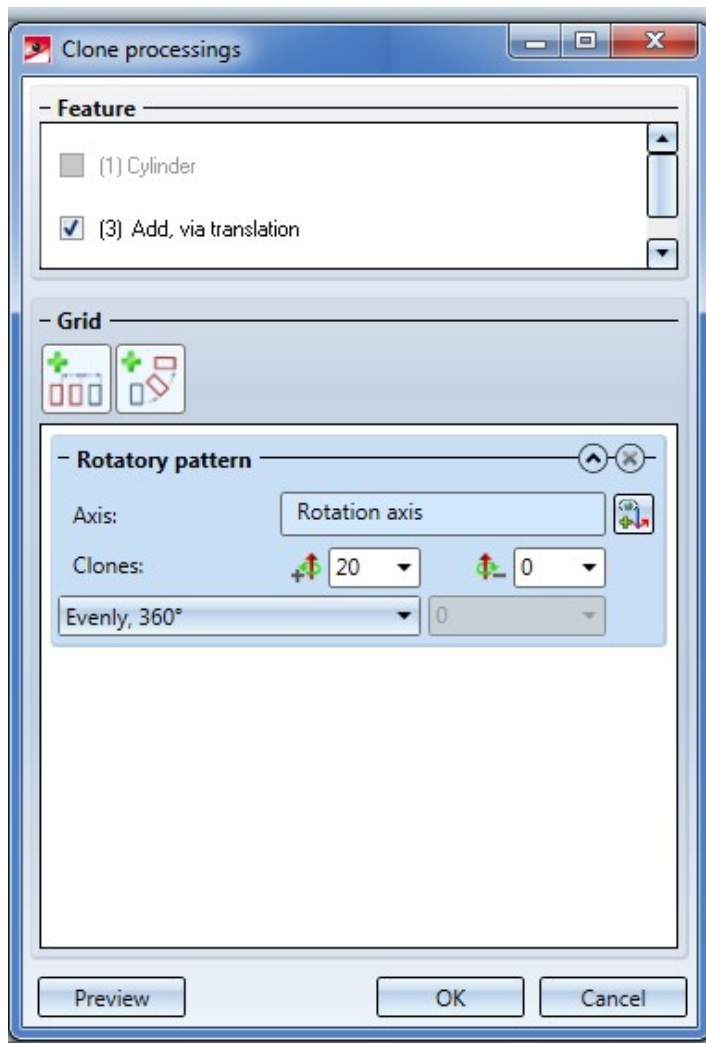
The new **Clone processings** function allows you to apply one or several feature steps in the feature log repeatedly. Both linear and rotatory pattern can be used for this.

Example: Creation of a gearwheel.

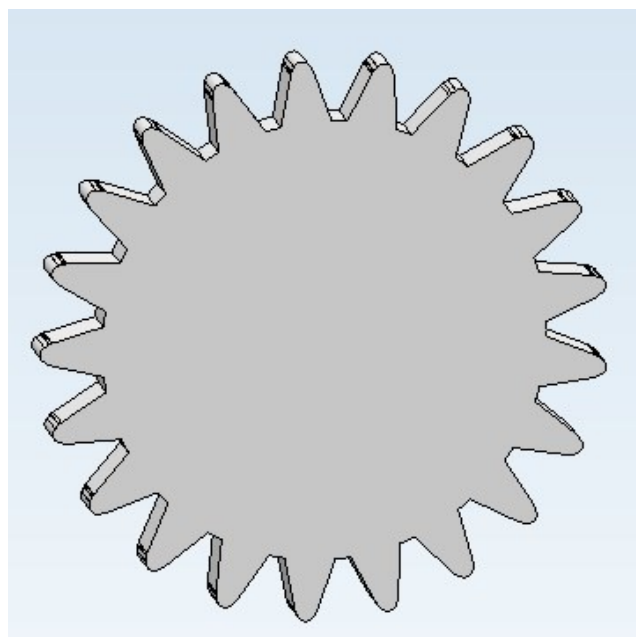
Create a disc-shaped plate as a Cylinder and extrude a tooth by means of a translatory movement.



When you call the function, the **Clone processings** dialogue window will be displayed. Select the centre axis of the cylinder as the rotation axis. The clonings are to be distributed evenly about  $360^\circ$  (full circle), and 20 clones (copies) are to be created:



Result:



## Configuration Management

### Service Pack 2 2016 (V 2102)

#### Settings of STB\_PARAMETER.DAT moved to Configuration Editor

The Steel Engineering settings that were previously specified in the system file STB\_PARAMETER.DAT have been moved to the Configuration Editor. There, you can find the settings at **Steel Engineering** and **Compatibility**.

Therefore, the corresponding settings have also been removed from the Steel Eng. Settings dialogue window.

#### Settings of SSTINI.DAT and SSTINI3D.DAT moved to Configuration Editor

The settings of the files SSTINI.DAT and SSTINI3D.DAT are no longer relevant for HiCAD and are therefore no longer available.

#### Rearranged Management + BIM settings in the Configuration Editor

The settings for Management + BIM have been rearranged. You can still find them at **PDM > Management + BIM**.

#### DSTV-NC parameters

The parameters at **Steel Engineering > DSTV-NC**

- Configuration of drawing number and
- Configuration of order number

have been moved to the export dialogue.

#### IFC attribute mapping

During the data transformation between IFC and HiCAD the attribute values from the original data will be transferred to permissible attribute values of the target system. The rules according to which the Attribute mapping configuration is to be carried out can be defined for the import and the export of IFC files at **Interfaces > IFC**.

#### New Management + BIM settings

##### NCW format for part data

You have now the option to automatically create NCW data for part and to manage and link them in HELiOS.

A prerequisite for the creation of this file format is that the parameter **Create NCW/NCX data** at **PDM > Management +BIM > External production drawing** has been set to **Yes**. The default setting is **No**, i.e. no NCW data are normally created.

##### Automatic creation of DWG files of production drawings during the workflow

You have now the option to automatically create DWG files for production drawings, either upon

- Checkup and Release, or upon
- creation and updating

of the drawings. For this purpose, the settings at **PDM > Management + BIM > External production drawings** have been expanded accordingly.

### Clean up project

The new Clean up function (in the Project function group) cleans up the active project by detecting and deleting invalid article masters.

Using this function only makes sense if the parameter **Clean up project when saving** at **PDM > Management + BIM** has been set to **No**. The default setting is **Yes**, i.e. whenever you save your drawing, invalid article masters will be removed automatically.

### Hide boltings during drawing derivation

Boltings in production drawings can be hidden via appropriate settings in the Configuration Editor. You can specify for each usage whether bolts are to be shown or hidden. The corresponding settings can be found at **Automatic drawing derivation > Production drawing > Usage-dependent > name > Views** (with *name* being the name of the respective usage).

## Service Pack 1 2016 (V 2101)

### Settings of PARAMASS.DAT file moved to Configuration Editor

The settings of the system file PARAMASS.DAT for parametric dimensions and the representation of HCM symbols have been moved to the Configuration Editor. They can now be found at

- **System settings > Dimensioning, 3-D > HCM dimensions**
- **System settings > Dimensioning, 3-D > Parametric dimension**

The changes in the Configuration Editor will only take effect after a restart of HiCAD.

### Handling of article masters when exchanging semi-finished products

The handling of manually assigned article masters when exchanging semi-finished materials can be defined in the Configuration Editor at **System settings > HELiOS - Handling of article master**.

The collection of the **Taking over of semi-finished product attributes** parameter will be evaluated in the process.



#### Please note:

This parameter will not be considered for the exchanging of Sheet Metal parts via feature.

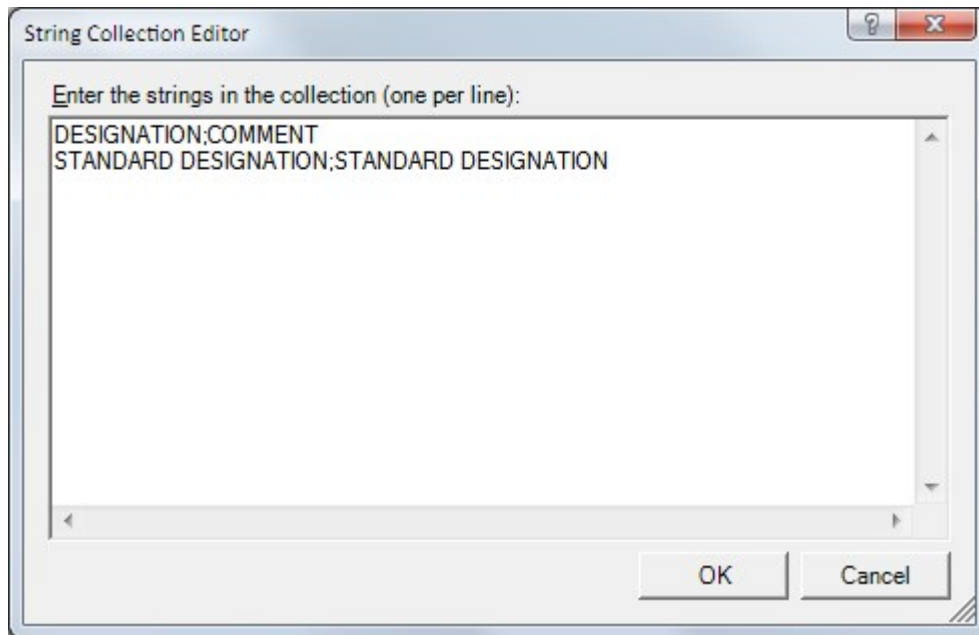
### Taking over of semi-finished product attributes

At **Taking over of semi-finished product attributes** (System settings > HELiOS > ...) you can specify whether particular semi-finished product attributes are to be assigned to the manually assigned article attributes during



exchanging of semi-finished products. Each assignment is defined in one row:

Semi-finished product attribute;Article attribute.



## Configuration of views

The parameter **Automatic ViewAll** (System settings > Visualisation) has been renamed to **Automatic ViewAll after projection change**. The dialogue texts have also been changed.

## Scale-dependency of hatchings.

You can specify in HiCAD whether . The default setting is **Scale-independent**. With this setting, the spacing between the hatching lines will always be the same, no matter which scale was selected.

The default setting shown in the dialogue window can be changed in the Configuration Editor, at **System settings > Visualisation > Scale dependent 3-D hatching**.

## PLMXML interface

The XML-based accompanying format PLMXML is a supplement to the JT interface. It allows the defining and storing of product structures and process data of complex assemblies in a readable form. Import and export options are handled in the same way as in the JT interface.

Which export functions are to be activated or deactivated upon function call can be specified in the Configuration Editor, at **Interfaces > JT/PLMXML**.

## Major Release 2016 (V 2100)

### Settings of ANSGEN.DAT file moved to Configuration Editor





The settings of the file ANSGEN.DAT that contains setting options for views can now be found in the Configuration Editor. There, the settings for views can now be found at:

- Drawing > Views
- System settings > Visualisation
- System settings > Processing plane
- System settings > Load/Save
- System settings > Sketches
- Compatibility
- Metal Engineering

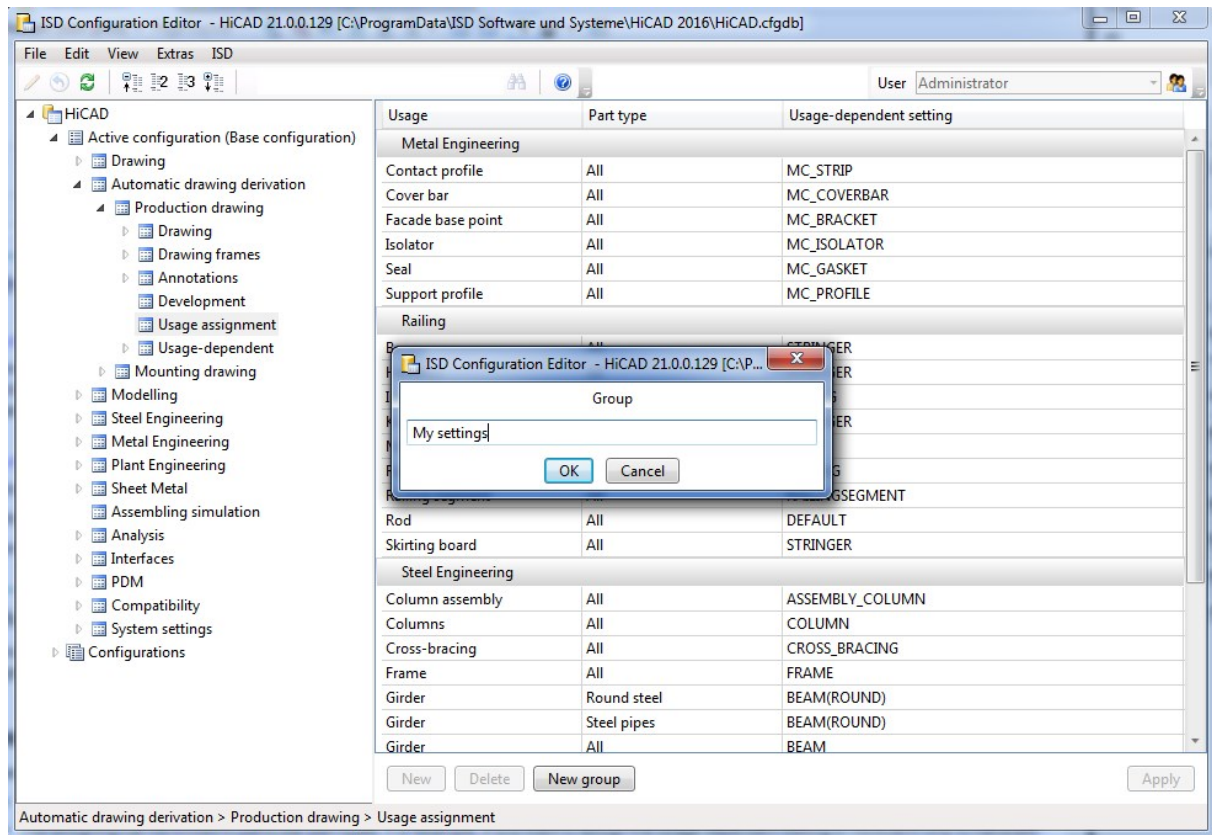
New is the possibility to specify the settings separately for sectional and detail views, and that the representation (colour, line type and layer) of view group frames can now be specified for derived drawings.

### Expanded toolbar

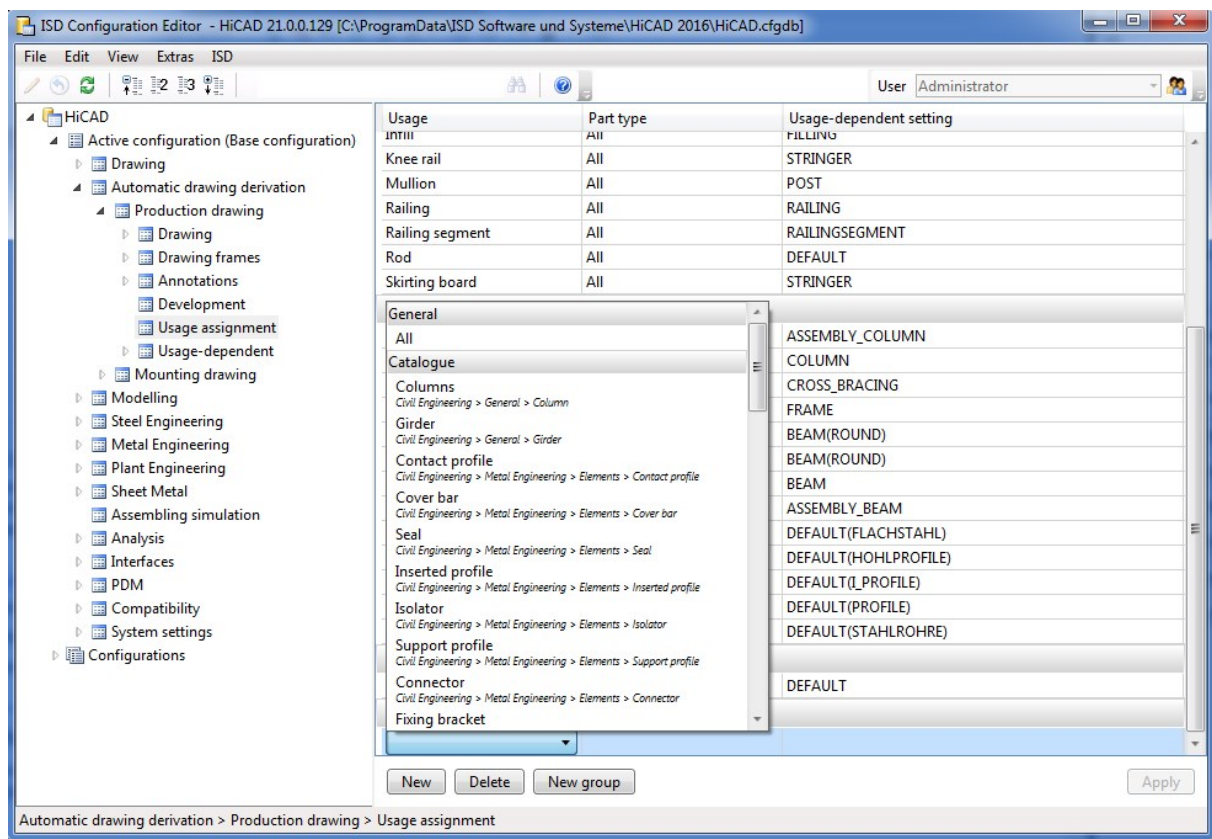
The following functions have been added to the toolbar of the Configuration Editor:

	<b>Collapse all</b>	This function collapses the structure display of the Configuration Editor and shows only the 1st level.
	<b>Expand 2 levels</b>	Shows you the first 2 levels of the configuration structure.
	<b>Expand 3 levels</b>	Shows you the first 3 levels of the configuration structure.
	<b>Expand all</b>	Expands the complete structure display of the Configuration Editor.

## Add new data record entry



In some areas you have the options to add new data records, or new groups with data records. To do this, click the **New group** button, enter a name and click **OK**. The group will then be added and an empty data record entry will be shown. When you now click on the input fields, you can compose the data record from the values in the displayed selection boxes.



Confirm your entries with **Apply**.

Use the **New** and **Apply** buttons to add data record also to already existing groups.

To remove a data record, mark it and click the **Delete** button. Click **Apply** to confirm the deletion.

## Derived drawings

You now distinguish between Production drawing and Mounting drawing. As a result, the tree structure has changed here.

- The selection of the usage now takes no longer place via a key, but via plain text.
- The assigning can be limited to particular part types. For example, if you want to assign the usage-dependent setting STRINGER only to handrails of the type "Flat steel", select the same-named entry beneath Part type.
- The usages can be neatly arranged in groups. Predefined are the groups Railing, Metal Engineering, Steel Engineering and Others.
- The creation of new assignments, and also the definition of new usage settings can take place directly on the list of assignments.

## General view shortenings for derived drawings

At **Automatic drawing derivation > Production drawing > Usage-dependent > ... > Views > Shortened view** you will find the new **General view shortenings** section. The settings will now be loaded from the Configuration Editor not only for the views of beams and profiles, but also for other views, e.g. sheet views etc.

### Setting: Standard part annotation

In HiCAD 2015, standard parts were annotated if you explicitly activated the annotation of sub-parts or (as of SP1) the annotation of boltings. As of HiCAD 2016 the annotation of standard parts can be switched on or off, independent of other annotation types.

### View groups - Frames, alignment, visualisation

A new listbox for frames and view groups offers the following selection options:

- Do not insert
- Always insert
- Insert, if there are multiple view groups

The default setting is **Always insert**. This presetting can be changed in the usage-dependent templates in the Configuration Editor at: **Automatic drawing derivation > Production drawing > Usage-dependent > ... > View groups > Frames around view groups**.

If a drawing frame contains only one view group, it will be centred in the drawing frame. This behaviour, too, can be changed in the Configuration Editor, at **Automatic drawing derivation > Production drawing > Drawing > Centre individual view group**. If you deactivate the checkbox, individual view groups will be placed at the top left of the drawing frame instead.

Colour, layer and line type of the frames can be specified in the Configuration Editor at **System settings > Visualisation > View group**.

When creating drawing derivations, you can distinguish between "Mounting" and "Workshop" for semi-finished products.

The relevant parameters can be found in the Configuration Editor, at **PDM > Management+BIM > Individual part type**.

For production drawings you have now the option to create DXF and PDF files either

- during Checkup or Release, or
- during creation and updating

of the drawings. For this purpose, the Configuration Editor now offers additional setting options at **PDM > VManagement + BIM**.

### Automatic creation of DXF-/PDF files during a workflow

For production drawings you have now the option to create DXF and PDF files either

- during Checkup or Release, or
- during creation and updating

of the drawings. For this purpose, the Configuration Editor now offers additional setting options at **PDM > Management + BIM**.

## Line type for referenced parts

The presettings for part edge representation during loading of referenced parts in sectional views, detail views and cut-outs have been changed. Colour, layer and line type can now be set at **System settings > Referencing - Updating**.

## Hatching for rise triangles in Plant Engineering isometries

The settings of the vertical and the horizontal hatching angle are now calculated automatically for the sake of an optimal visual representation. The previous input fields for manual settings have been removed. However, if you prefer to use other hatching angles, you can still do this by specifying the desired angles in the Configuration Editor, at **Plant Engineering > Isometry and Pipe spool drawing - Dimensioning**.

In Metal Engineering, a purpose will be automatically assigned to sketches. One distinguishes between **Purpose = Part** and **Purpose = Create /Edit**.

In the Configuration Editor, at **System settings > Sketches > Purpose for new sketches in ME grid** (for a working with the Metal Construction plugin) you can specify which of the two purposes is to be assigned when a new sketch is created.

## Parametrics (HCM)



## Service Pack 2 2016 (V 2102)

### Easier assigning of HCM constraints in Sketches

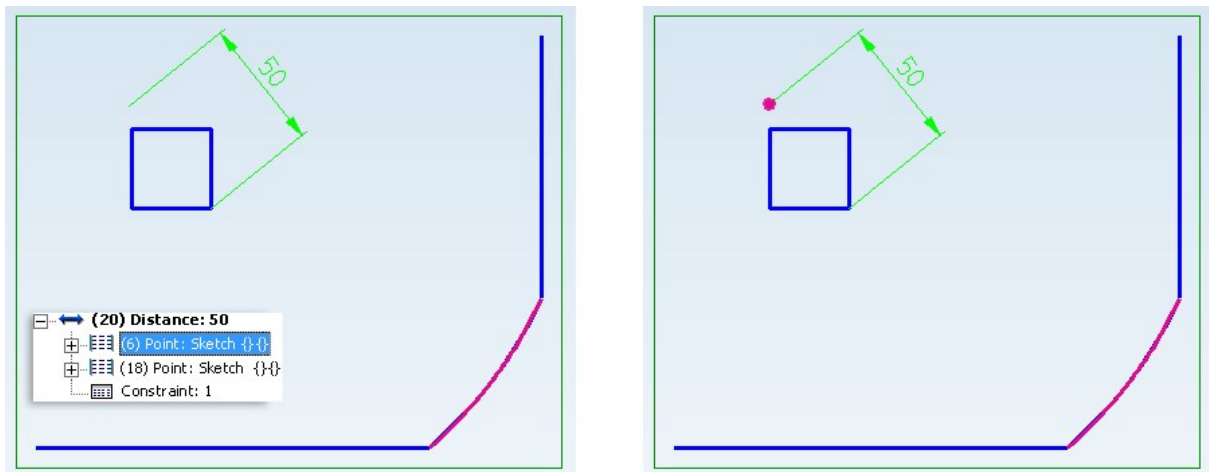
The assigning of HCM constraints has been made easier due to the following changes:

- When you set constraints of the type **Symmetrical** the symmetry axis will be queried only once. After this, you can then directly choose geometry pairs which obtain the symmetry constraints.
- The same applies to the dimensional constraint **Double distance**.
- When you define positional and dimensional constraints, only geometries can be identified that actually make sense for the chosen constraint. Example: A positional constraint of the type **Mid-point** can only be created between two points or between two straight lines. If the first chosen geometry is a straight line, the second chosen geometry cannot be a point.

### Marking of reference geometries for particular point options

When you choose a constraint in the HCM window of the ICN that uses the point option **Centre**, the reference geometry of this point will also be marked. In this way it becomes clearer to which element the dimensional constraint refers.

For instance, in versions before HiCAD 2016 SP2 it was not obvious that the upper end of the distance constraint refers to the centre point of the circular arc at the bottom right in the image below. From SP2 onwards, this point will also be marked.



Left: Before HiCAD 2016 SP2; Right: As of HiCAD 2016 SP2

## Service Pack 1 2016 (V 2101)

### Improved dragging

Dragging in 3-D sketches has been improved. Besides individual lines and points, you can now also drag several lines or complete polylines. Also, you can automatically create HCM constraints while dragging. Furthermore, dragging is now much more convenient during working with freeform curves.

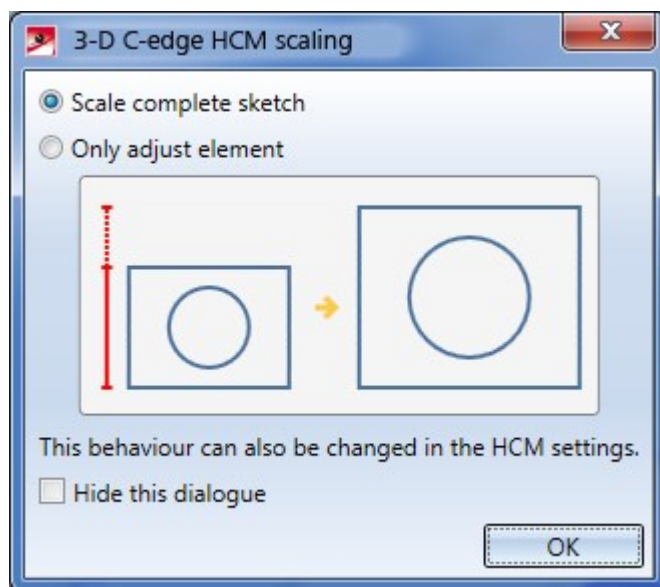
## Revised dialogue windows

For the Sketch HCM, the Automatic positional constraint and Automatic dimensional constraint dialogue windows as well as their Settings have been revised.

For the 3-D Part HCM the Settings dialogue window has been revised.

The dialogue windows for automatic dimensional constraints, automatic positional constraints and the relevant settings have been revised. They have been adjusted to the HiCAD layout and have now a clearer structure.

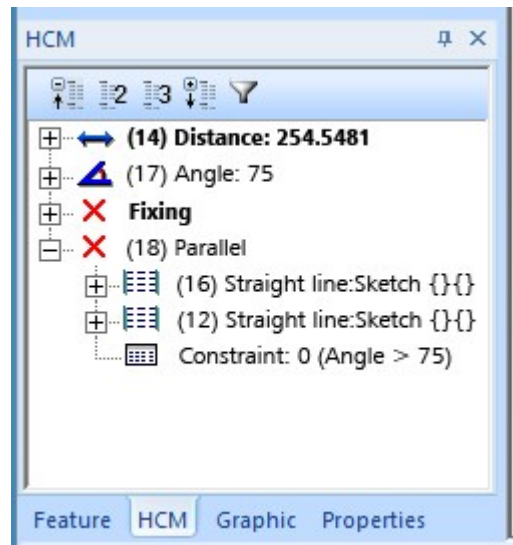
In addition, the 3-D C-edge HCM scaling dialogue window has had a redesign and now displays a small example sketch illustrating the effects of the selected option.



## Major Release 2016 (V 2100)

### "Conditional" HCM constraints

HCM constraints can now, similar to Feature steps, activated or deactivated under certain conditions. For instance, you can determine that a positional constraint of the type "Parallel" is only to be applied if an "Angle" variable is greater than 75:



Further information on this function can be found in the The HCM Model in the ICN topic.

### Support of further point options in the Assemblies HCM

In the Assemblies HCM you can now also use virtual points, created through the use of particular point options, for the assigning of dimensional and positional constraints. These point options are:

- Mid-point of line/edge (M),
- Mid-point between 2 points (M2),
- Surface centroid (SP) and
- Body centroid (SK).

A prerequisite for this is that the points created by means of a point option are located on the same part. It is not possible to create a distance constraint that points to the mid point between two points on two different parts.

## Service Pack 2 2016 (V 2102)

In the Report Manager configuration file HICAD\_STAHLBAU\_01.RMS the column **DSTV Part type** has been removed.

When creating a BOM with the **Bill of Materials, for active assembly** function, you have now the option to transfer attributes of the model drawing as head data.

You have now the option to display assembly images in axonometric representation in shipping lists.

For this purpose three new parameters have been added to the he options on the "Settings" table sheet: Assembly image height, Assembly image width and Assembly image representation. The Steel Engineering templates that are supplied with HiCAD are set to the default values for these parameters.

[illegible]

### Example of a shipping list with AXO representations

## Transfer drawing attributes to BOMs for active assembly

When you call the **BOM, for active assembly** function, you have now the option to transfer attributes of the drawing as head data.

## Service Pack 1 2016 (V 2101)

### Revised dialogue windows

The dialogue windows for BOM creation and for the settings for BOM creation have been revised. Their design has been adjusted to the other dialogue windows in HiCAD and have a clearer structure.

## Major Release 2016 (V 2100)

### Excel BOMs for Plant Engineering and P+ID

With the release of Version 2016, new, conveniently adjustable BOM templates have been introduced to the Excel-based system. These templates are:

- **ENG\_LD1\_COMPONENTS1** and **ENG\_LD1\_SYMBOLS**) for P+ID, and
- **anlagenbau\_ohne\_db\_szn.rms**, **anlagenbau\_ohne\_db\_rl.rms** and **anlagenbau\_mit\_db\_geslen.rms** for Plant Engineering.

A detailed description of these BOM templates and notes on their adjustment can be found in the topic Adjust Templates - Plant Engineering and P+ID.

### Bar lists - Residual lengths

The possibilities for the creation of bar lists in Excel BOMs have been enhanced. For instance, it is now possible to specify residual lengths for semi-finished products. As a result, the corresponding semi-finished products will only be used in the bar lists in a limited way.

Furthermore a new setting option is now available, which causes a preferred use of residual lengths, even if there will be a larger amount of waste.

## Variant Editor

Service Pack 2 2016 (V. 2102)

### Auxiliary functions for data transfer

The command line call of the Variant Editor **VariantenEditor.exe** supports two calling parameters, thus offering two auxiliary functions for data transfer.

- Calling parameter **autovaa**

A variant is described by various attributes, some of which are identical or almost identical for all subtypes. Examples of this are the file name or the order note. In practice, customers often manage a list of parts that are used in their company. If these customers now wish to use HiCAD variants, the question arises as to which of the many variants actually match the corresponding entries of the customer list. Here, the calling parameter **autovaa** can help you find the best matching variant for each entry. For this purpose the attributes of the variant will be used.

- Calling parameter **autoderive**


This calling parameter provides the Variant Editor with an automatism that allows an automatic derivation of existing variant files.


## Sheet Metal

Service Pack 2 2016 (V 2102)

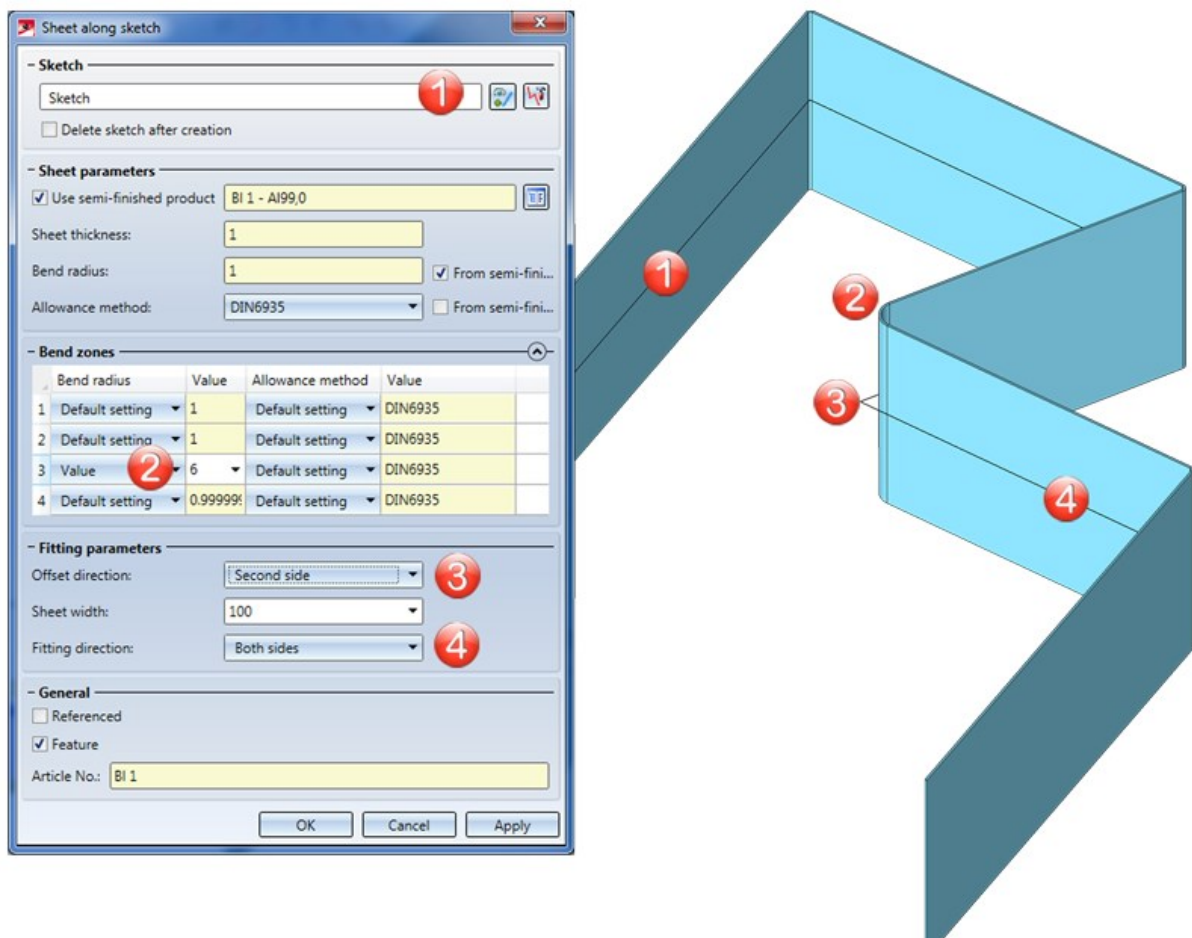
### Sheet along sketch

The revised sheet creation function **New sheet along sketch** allows a processing of sketches while the dialogue window is open.

As soon as you have identified the sketch a preview of the new sheet will be displayed. To change the sketch, click on the  **Process sketch** icon. The **Sketch** tab and the **Process sketch** dialogue will then be displayed. Change the sketch and confirm with **Apply sketch**. The **Sheet along sketch** dialogue will then be displayed again.

You can expand the **Bend zones** area by clicking the  icon, and enter an individual radius and allowance for each bend zone.

The **Offset direction** and the **Fitting direction** each offer 3 options than can be selected (complete with visualization) from a pull-down menu.



- (1) Sketch
- (2) Bend radius changed
- (3) Sheet thickness
- (4) Fitting direction



## Sheet Metal cassettes

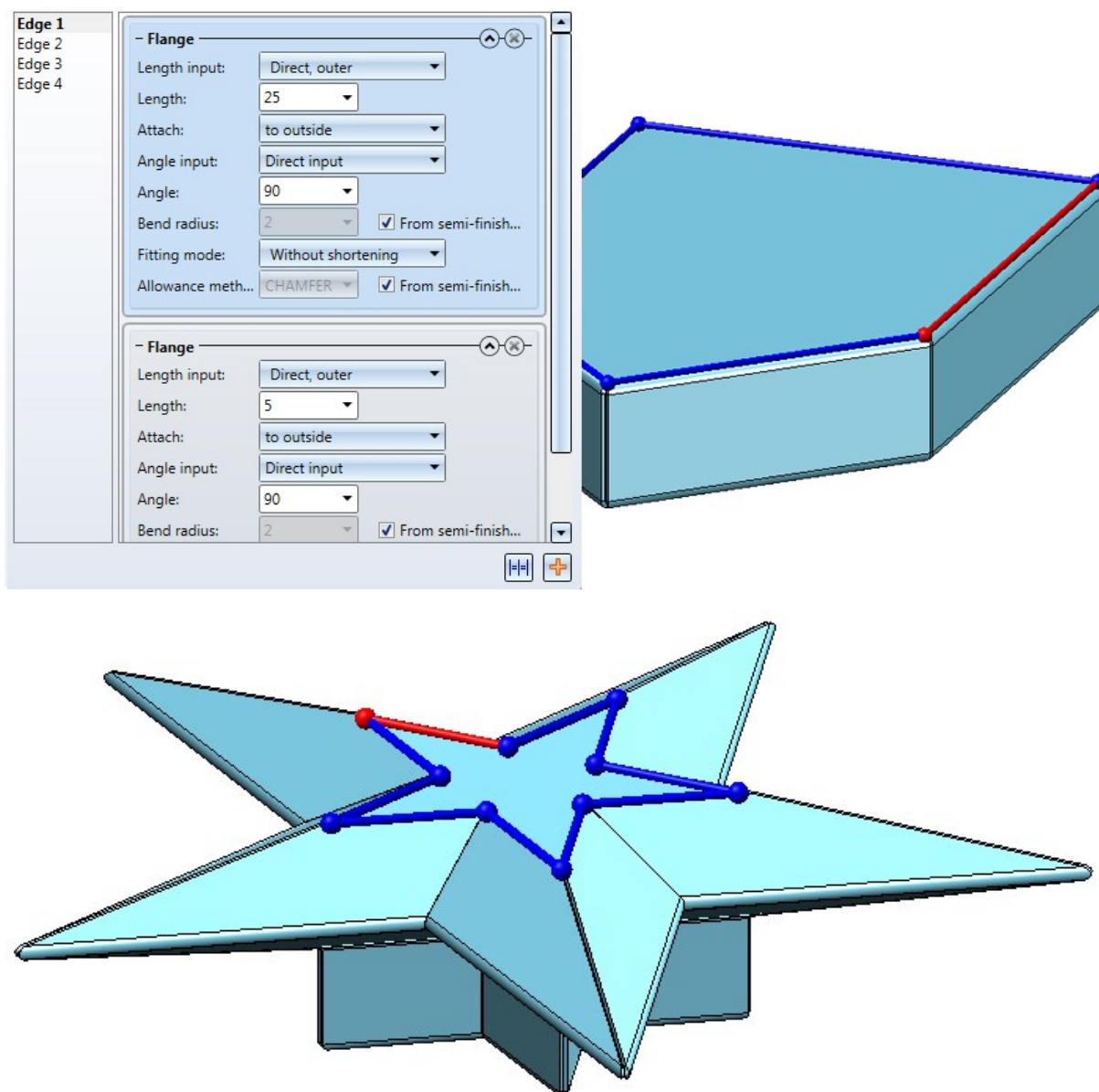
HiCAD now offers the new **Sheet Metal cassette** Design Variant for the construction of high-level functionality facades with a striking appearance. This Design Variant allows you to create rectangular or polygonal cassettes using Sketch technology.

The dialogue window consists of two tabs:

On the **Base sheet** tab you determine the corner points of the contour. You can either enter the coordinates, or identify the points in the drawing, or select a sketch. Use the options beneath **Close corner** to to mitre attached flanges (or sheets and bend zones, respectively) in one step. In the process, the bend zone will either be trimmed, adjusted or, if you choose **Mitre (indiv.) with lengthening** ignored.

On the **Attach flange** you can then define the attached flanges. You can assign a different flange to each edge of the base sheet. You can use most parameters of the Sheet Metal function **Attach flanges** here.

Click Preview to check and, if required, modify the result. Later changes can be applied via the feature log.



## Bore patterns, Moulding tools, Punching tools

Libraries for Bore patterns and Punching tools can be found in the **Factory standards** catalogue, and a library for Moulding tools can be found in the **Sheet tools** catalogue of the Catalogue Editor. An expanding of these libraries was previously only possible with 2-D parts that could be parameterized with the 2-D HCM (file format: .DCF). Now, it is also possible to store planar sketches that can be parameterized with the Sketch HCM (file format: .KRA).

## Agrafes

The **Bore patterns** library now also contains **Agrafes**. These can be used, e.g., for the anchoring of plates in facade constructions; they are not visible from the outside.

Parameters in the catalogue:

$l1 = 20$

$l2 = 20$

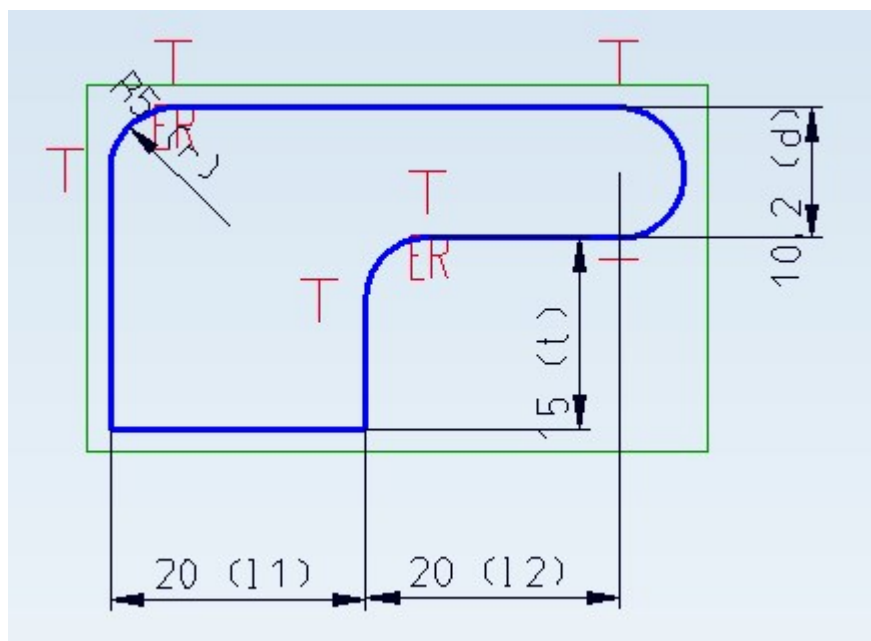
$d = 10,2$

$r = 5$

$t = 15$

Insertion point: Centre of right semi-circle

The image below shows an agrafe for the left side. The right side can be selected via the Orientation.



## Export section contours as DXF file

Three new functions for the export of section contours are available. These functions export the contours of the blanks of Sheet Metal parts and Steel Engineering plates as DXF files. A prerequisite for this is that they are:

- **BOM-relevant** and
- have an **Item number**.

Blanks in the drawing will be ignored. HiCAD will always output a blank that is created temporarily for the export. The current pre-settings in the drawing will be used for blank generation. The alignment is only unambiguous when it has been specified with the **Processing direction** function.

The 3 available functions support you with the selection of the sheets or plates:

**All sheets:** The developments of all sheet main parts in the drawing will be exported.

**Sheets from list:** All marked sheets and plates (part selection list) will be exported as a DXF file. The "quantity" of the parts per item number will be taken from the drawing - irrespective of the selected and displayed parts.

**Active sheet:** Only the active sheet main part will be exported.

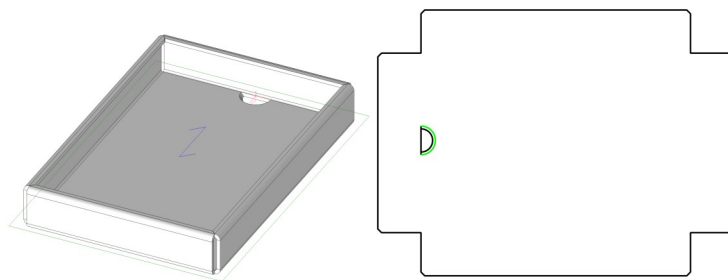
Proceed as follows:

1. After calling the function, select a directory for the DXF files.

A dialogue window with a list of the DXF files to be exported will be displayed

2. Select **Close** to end the export.




The files will then be located in the previously selected directory.



## Export blank as DXF file

The behaviour during DXF export of Sheet Metal part and Steel Engineering plate blanks has changed.

This affects the following functions:

- Sheet Metal > Sheet development > Extras  > DXF: Complete - All sheets
- Sheet Metal > Sheet development > Extras  > DXF: Complete - Sheets from list
- Sheet Metal > Sheet development > Extras  > DXF: Complete - Active sheet

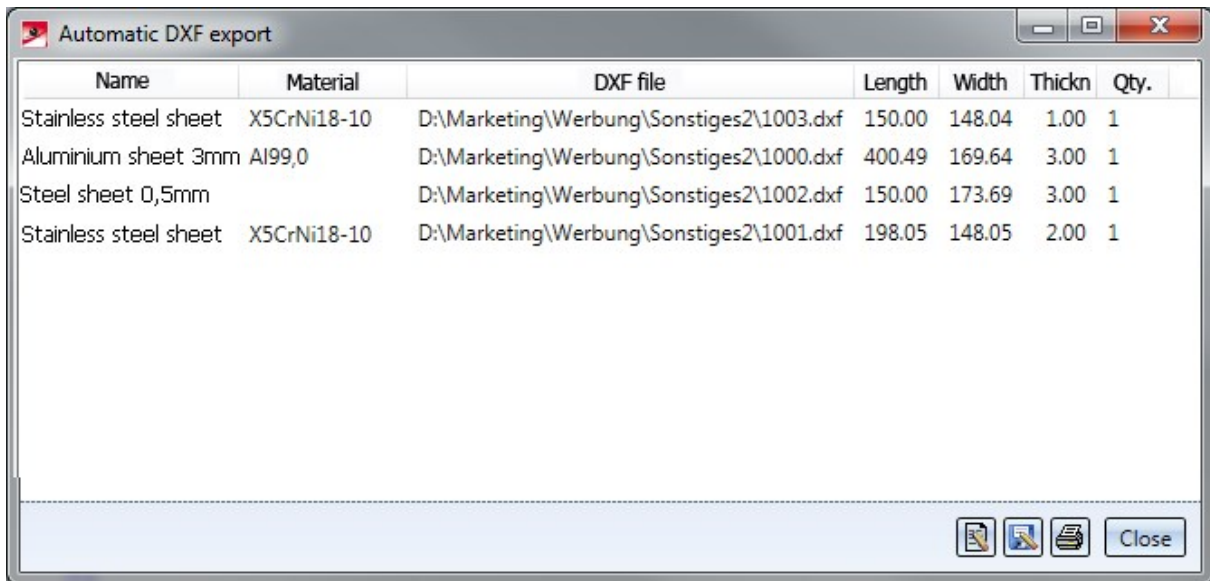
Use these functions to export the blanks of Sheet Metal parts and Steel Engineering plates as DXF files. A prerequisite for this is that they are:

- **BOM-relevant** and
- have an **Item number**.

As of SP2, blanks in the drawing will be ignored. HiCAD will always output a blank that is created temporarily for the export. The current pre-settings in the drawing will be used for blank generation. The alignment is only unambiguous when it has been specified with the **Processing direction** function.

## Output of CSV files for DXF export

When outputting a temporarily created blank for the export, a list of DXF files is shown. This list can now be saved as a CSV file.



Name	Material	DXF file	Length	Width	Thickn	Qty.
Stainless steel sheet	X5CrNi18-10	D:\Marketing\Werbung\Sonstiges2\1003.dxf	150.00	148.04	1.00	1
Aluminium sheet 3mm	Al99,0	D:\Marketing\Werbung\Sonstiges2\1000.dxf	400.49	169.64	3.00	1
Steel sheet 0,5mm		D:\Marketing\Werbung\Sonstiges2\1002.dxf	150.00	173.69	3.00	1
Stainless steel sheet	X5CrNi18-10	D:\Marketing\Werbung\Sonstiges2\1001.dxf	198.05	148.05	2.00	1

Click the  icon to save the list as NAME.TXT or NAME.CSV to the selected directory.

In the file ABWPAR.DAT you can specify the output mode (Standard or Extended) (Default or Expanded) and the output format (TXT file or CSV file). The settings will only take effect after a restart of HiCAD.

## Increased performance for cleaning up of intersections

The calculation time required for the cleaning up of intersections could be reduced significantly. In test examples, time savings from 30% to 50% could be observed.

## Bend angle-dependent parameters in blanks

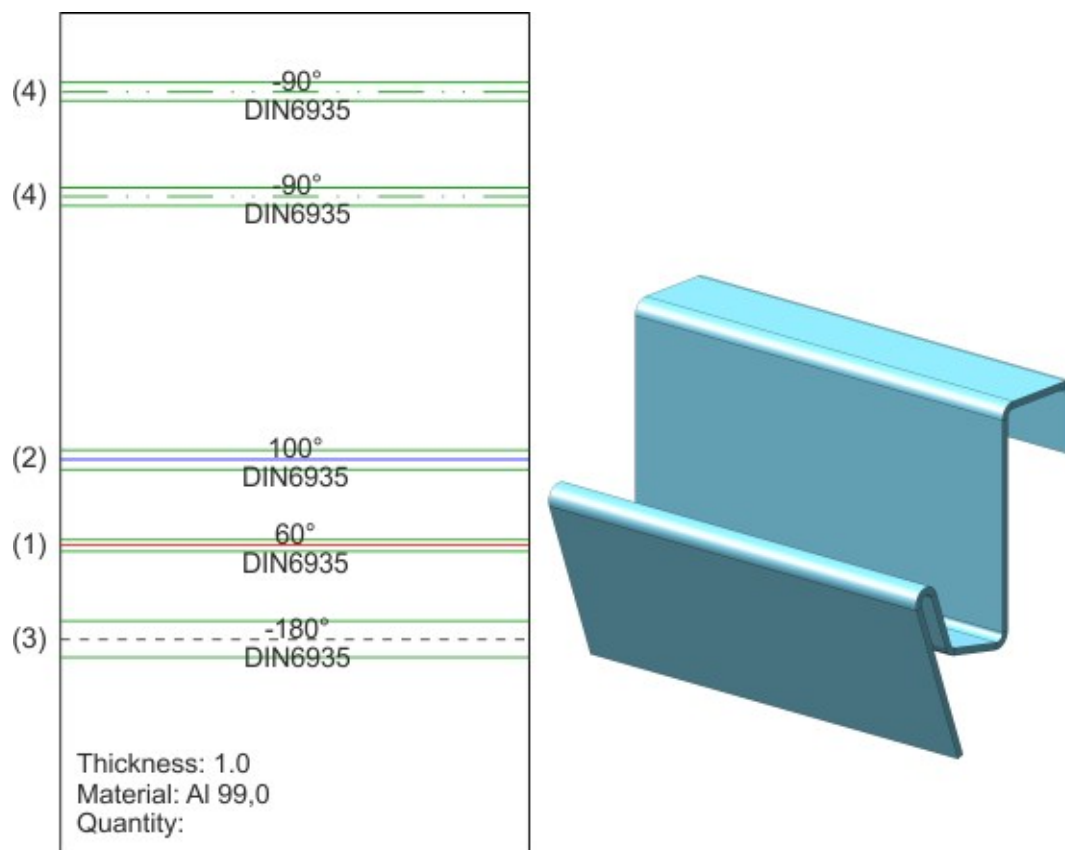
You have now the option to individually specify bend lined in blanks with the settings of the file ABWCOL\_BEND\_ANGLE.DAT. Whether this file will be evaluated depends on the corresponding setting in the file ABWCOL.DAT.

Set the following parameter in the file ABWCOL.DAT to **1** and restart HiCAD:

```
Biegelinienfarbe soll in Abhängigkeit vom Biegewinkel gesetzt werden; Datei "sys/abwcol_bend_angle.dat"
(1: Eigenschaften aus Datei lesen; 0: nicht aus Datei)
1
```

Then, change the settings in the file ABWCOL\_BEND\_ANGLE.DAT. Simply save the file to apply the changes after updating of the blank.

Positive and negative bend angles are distinguished the file ABWCOL\_BEND\_ANGLE.DAT. Only the data within the BEGIN-END block may be changed.



- (1) Bend angle  $0^\circ$ -  $95^\circ$ , Line code (Layer, Colour, Weight, Type) 2411
- (2) Bend angle  $95^\circ$ -  $360^\circ$ , Line code (Layer, Colour, Weight, Type) 2311
- (3) Bend angle  $-360^\circ$ -  $-95^\circ$ , Line code (Layer, Colour, Weight, Type) 2013
- (4) Bend angle  $-95^\circ$ -  $0^\circ$ , Line code (Layer, Colour, Weight, Type) 2117

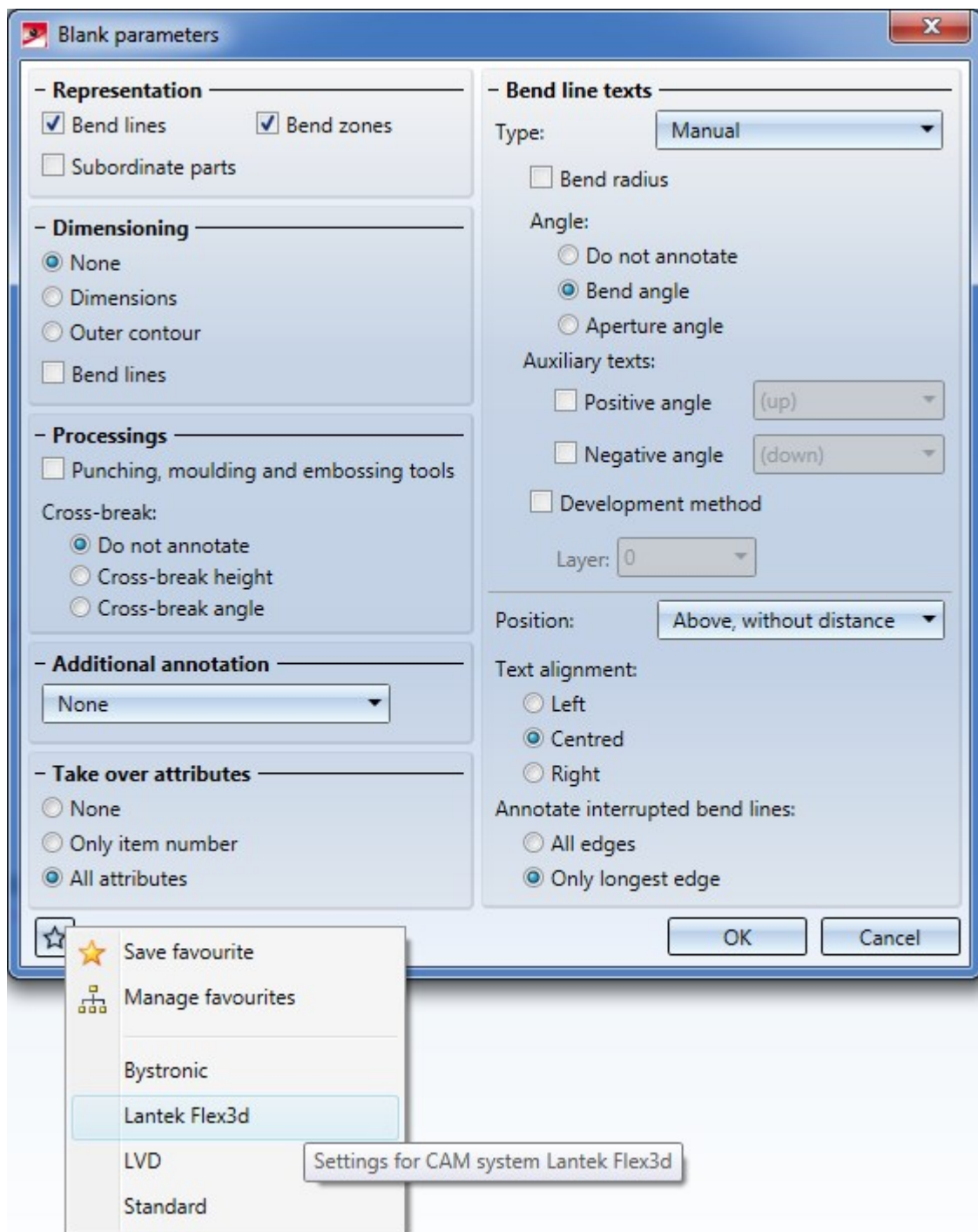
## Sheet developments for Lantek Flex3d as Favourites


Lantek Flex3d is a software solution for the automation of sheet metal cutting machine programming.

The interface between HiCAD and Lantek Flex3d is created via the export of the blank as DXF file.

To ensure a smooth data exchange, the blank needs to be composed according to Lantek-specific requirements.

1. Call the **Parameters, Default setting**  to view the blank parameters.



2. Click the  icon to open the Favourites. Load the parameters for Lantek Flex3d.
3. Create the blank.

4. Now you can export the blank with the **Export complete blank as DXF file**   function.

The format of the abwpar.dat and the abwcol.dat files has changed due to changes of the files in connection with **Bend angle-dependent bend line parameters**.



## Service Pack 1 2016 (V 2101)


### Coating

Use the **Coating** function to define the coating description, the coating colour (RAL colour, system colour) and the coating type (outer side, inner side, edge) of a Sheet Metal main part.

The coating description and the coating type are taken over into the part attributes mask and are then also available in BOMs.

After selecting the outer sheet side you will be shown a preview of the coating, provided that you have not activated any options requiring further value inputs. If you have activated the **Apply immediately** checkbox, the coating parameters will be applied immediately if the value inputs were correct, enabling you to select the outer sheet side of a new Sheet Metal main part. Changes can then be made via the feature log.

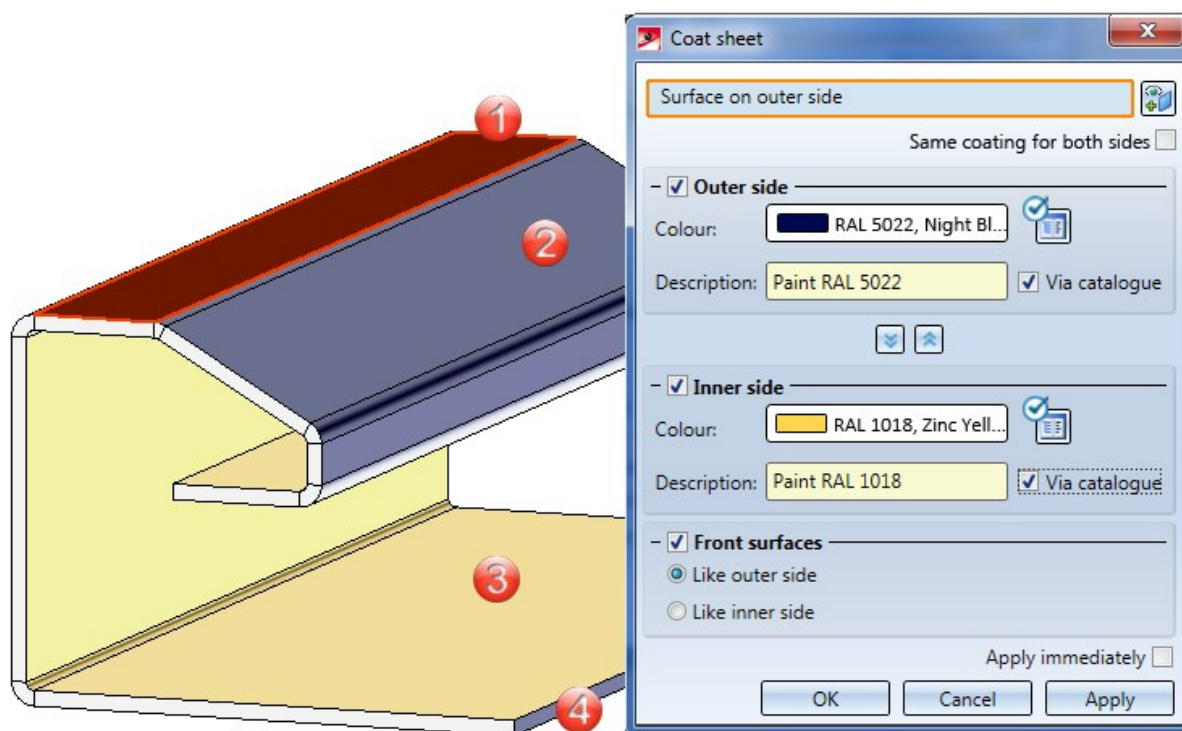
If you want to change a surface after its identification, click the  icon and identify the new surface.

If you are working in the shaded mode, the coating will be shown in the selected colour. You can select a RAL colour from the catalogue or a system colour as the coating colour. Activate and click the  icon to display a selection list with the RAL colours. Otherwise, you can pick a system colour from the **Colour** selection box.

If you have selected a RAL colour, you can also take the description from the catalogue .

Use the   icons to copy, for instance, the parameters of the outer side to the inner side and vice versa.

You can display the front surfaces in the colour of the outer side or inner side, respectively. This setting will be taken over into the part attributes mask.



(1) Surface on the outer side

(2) Outer side



- (3) Inner side
- (4) Front surface like outer side

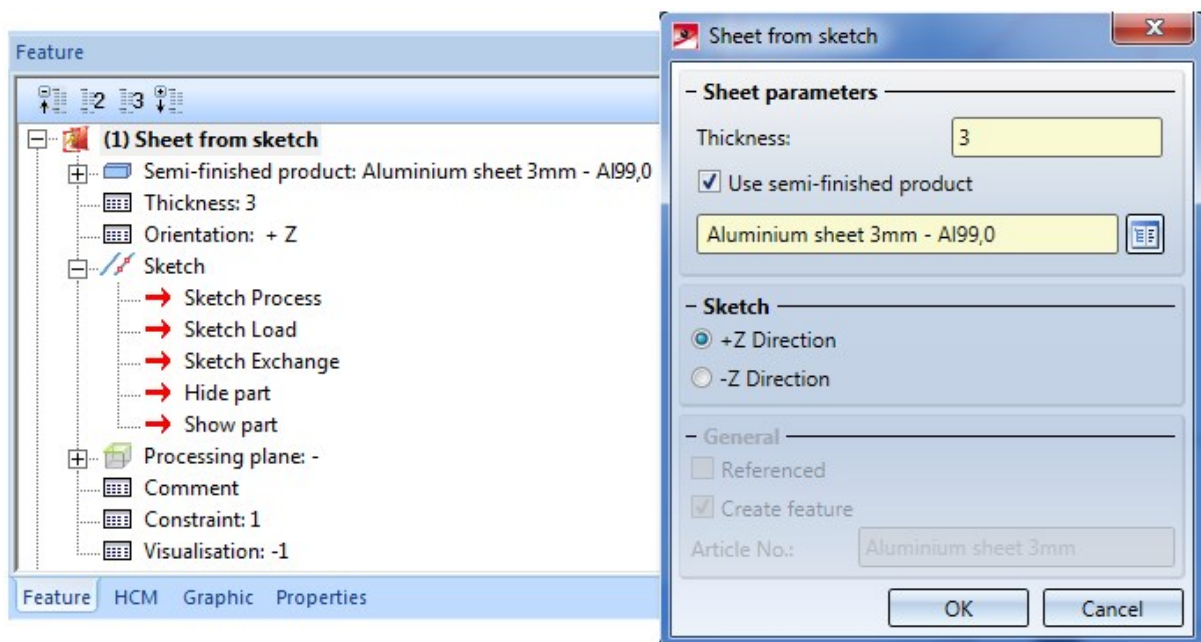
## Change semi-finished product subsequently


The functions

- New sheet from sketch
- New sheet along sketch

now offer the option to activate, deactivate or change the semi-finished product by double-clicking the feature.

*Example*



Double-clicking the feature **Sheet from sketch** opens the same-named dialogue window. You can then activate or deactivate the **Use-semi-finished product** checkbox, or click  to and a different semi-finished product. Confirm with **OK**.

## Taking over of semi-finished product attributes to manually created article master

In the Configuration Editor you can specify the semi-finished product attributes that you wish to take over into the a manually created article master. It is important that you created the article master yourself (manually) using the **Create new article master + assign** function instead of applying a semi-finished products article master through selection of a semi-finished product during base sheet creation.

Also, each catalogue table column to be taken over needs to be linked to a HELiOS attribute. The table will then be transferred to HELiOS.

## Change article number during sheet creation

When creating sheets with the functions

- New sheet from solid
- New sheet from surface
- Connecting sheet

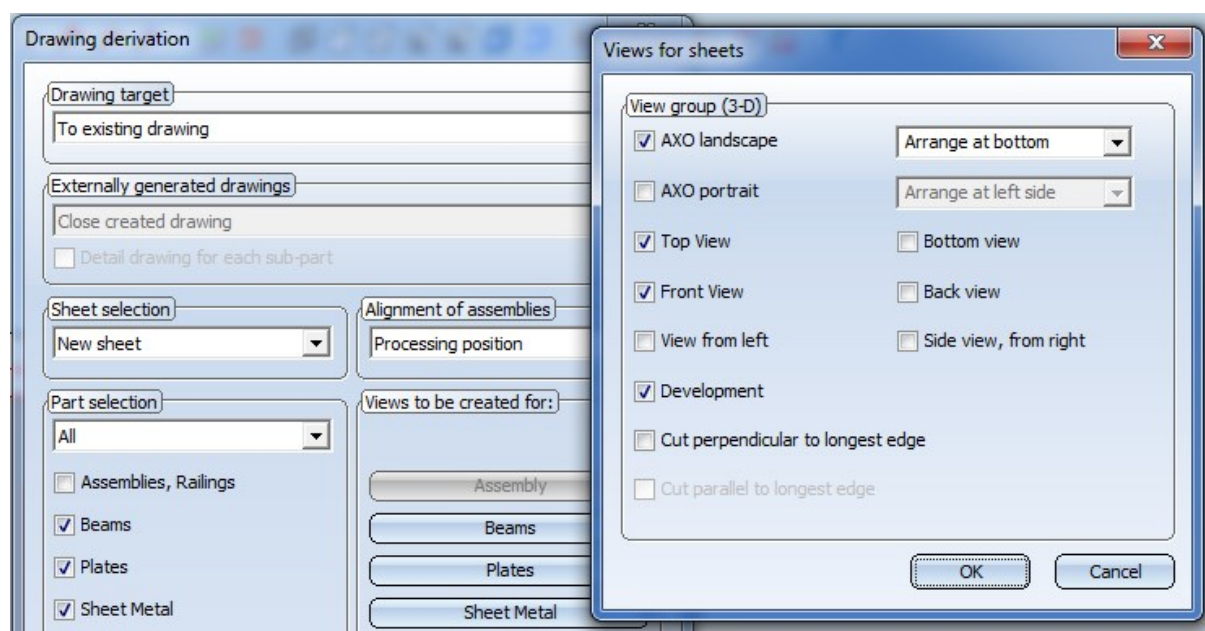
you have now the option to change the article number, if no semi-finished product has been selected. If the **Use-semi-finished product** checkbox has been activated, the article number of the sheet will be generated automatically.

Double-clicking the feature opens the dialogue window for sheet creation. Here, too, the article number can only be changed if the **Use-semi-finished product** checkbox has not been activated.

## Sheet Metal parts in workshop drawings

You can now select the following sectional views for Sheet Metal parts in the **Views for sheets** dialogue window that you open by clicking the **Sheet Metal** button in the **Drawing derivation** dialogue window (Drawing > Itemisation/Detailing > Derive):

- Cut perpendicular to longest edge  
This cut will only be created if a bend zone exists.
- Cut parallel to longest edge  
This cut will only be created if a bend zone running perpendicular to the section path exists.



## Fold flange

The user guidance texts of the **Fold flange** function have been revised:

- When you select a fold line for the **Select, without sketch** the text **Straight line** will appear in the input field.
- While the **Select, without sketch** function is active, the following user guidance texts will be shown in the info bar at the bottom left of the HiCAD screen:
  - Select 1st point, straight edge/line or surface for fold line (RMB = Further options)
  - Select end point of line (RMB = Further options)

## Major Release 2016 (V 2100)

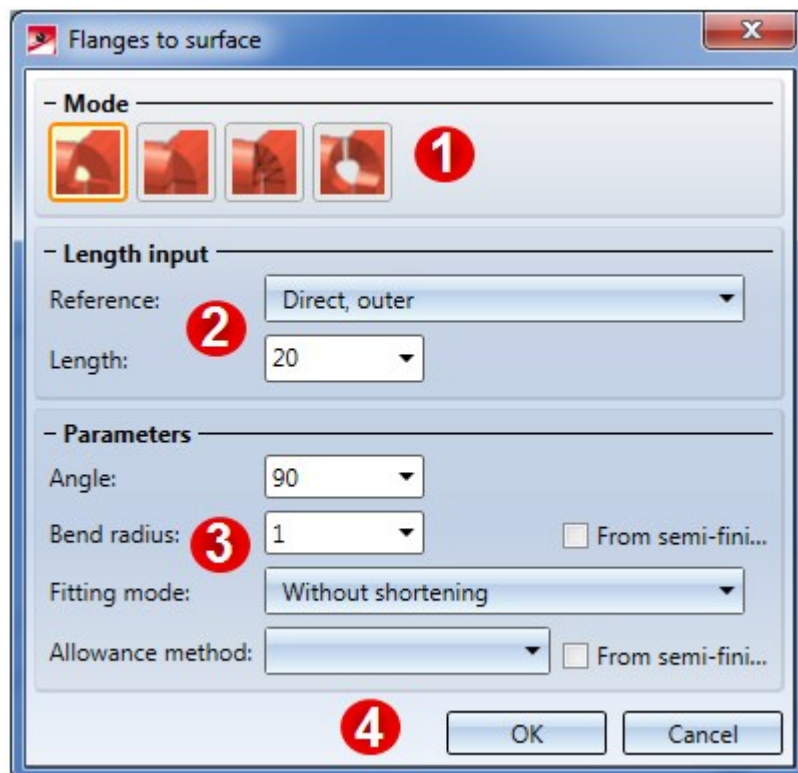
### Discontinuation of HiCAD Unfold

HiCAD 2016 (Version 2100.0) will be the first version that does not contain HiCAD Unfold.

### Attach flanges to surface

For the attaching of a flange to a surface, the following 4 options are available:

- Flange to surface, with free bend zone
- Flanges to surface, bend zone closed
- Flanges to surface, with drainage area
- Flanges to surface, with circular cut-out



Lengths and parameters are specified in the same way for all 4 variants.

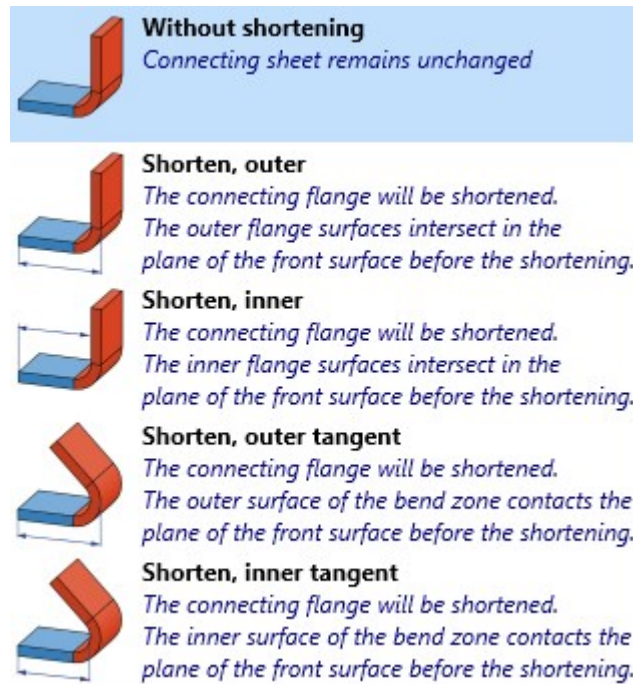
You can enter a value for the desired length. If you right-click the **Length** input field, a context menu with further options opens. You can then, for example, choose **Pick distance** to take the length directly from the drawing.

There are several options to define the reference of the length with the help of a graphic.

Likewise, the bend angle can either be entered in the **Angle** input field or taken from the drawing by right-clicking the **Angle** input field and selecting **Pick angle**.

The **Bend radius** is located at the inner side of the bend zone. If you right-click the Bend radius input field you can directly pick the radius from the drawing. For semi-finished products, the bend radius will always be taken from the Catalogue Editor.


The **Fitting mode** refers to the processing of the connecting sheet. Here, too, the selection is facilitated by a graphical representation:

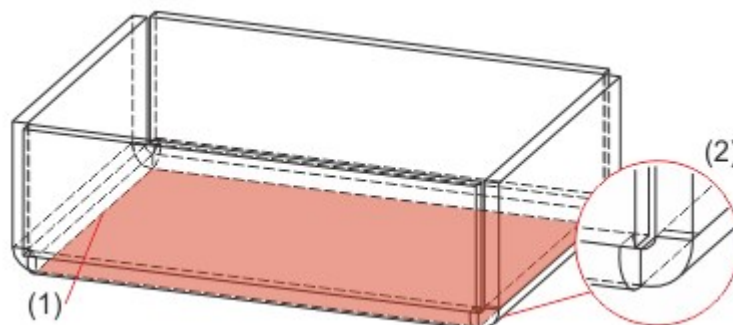


To obtain the blank (development) of the modelled Sheet Metal part, HiCAD offers you developments according to different calculation methods (allowance methods) that take the material-dependent length change of the work-piece during bending processes into account. For this, HiCAD also offers the common empirical, practice-oriented allowance methods.

The calculation methods are represented by prepared, exemplary system files. In practice, however, each user will store his own method(s) in this way. Before attaching the flange you can select the required system file.

## Attach flanges to surface, bend zone free


Use the **Bend zone free**  function to attach flanges around a surface (base sheet). The selection of the top surface determines the direction of the new flanges.




(1) Top surface

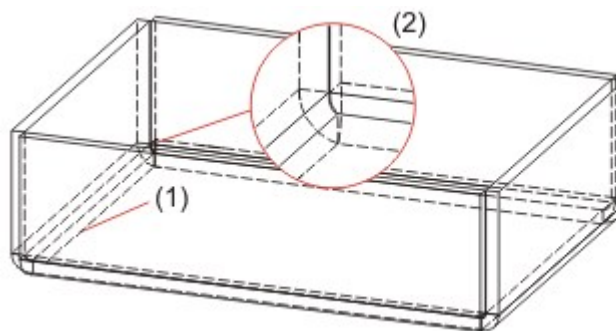
(2) Flange with free bend zone

### Attach flanges to surface, bend zone closed

Use the **Bend zone closed**  function to attach flanges with closed bend zones around a surface (base sheet). Sheets and bend zones will be lengthened up to the common intersection point of the inner edge. The selection of the top surface determines the direction of the new flanges.


### Attach flanges to surface, with drainage area

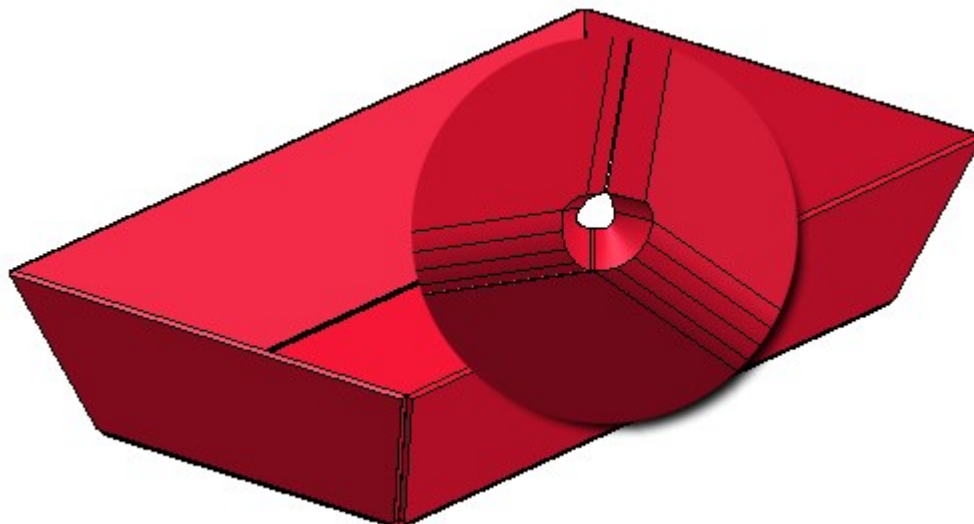
Use the **Drainage area**  function to attach flanges with drainage area around a surface (base sheet). The selection of the top surface determines the direction of the new flanges.



- (1) Top surface
- (2) Flanges with drainage area

### Attach flanges to surface, with circular cut-out







Use the **Circular**  function to attach flanges with a circular cut-out around a surface (base sheet). The selection of the top surface determines the direction of the new flanges.





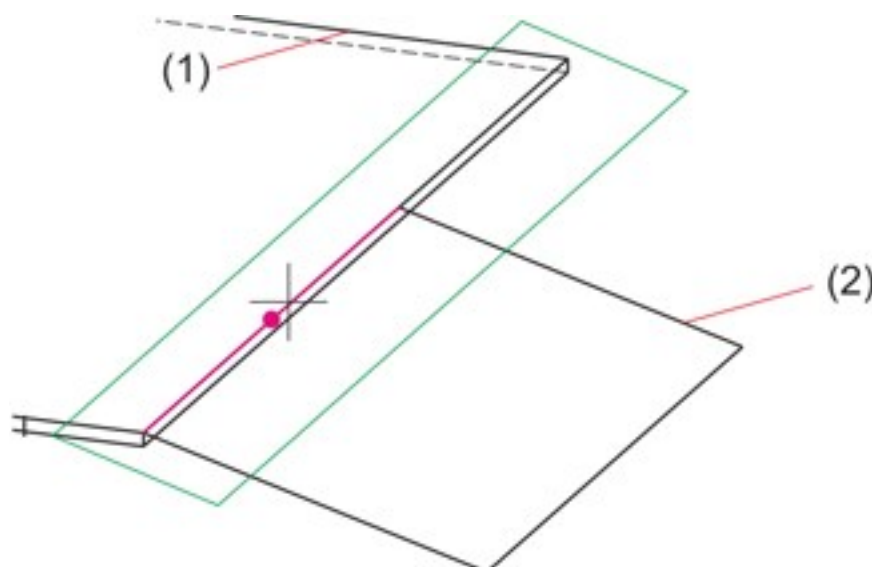
## Several blanks for one sheet

Thanks to the enhanced blank functionality you can now derive several blanks with different parameter settings from a Sheet Metal part. The parameter settings are saved in the blank. The structure of the corresponding function has changed as follows:

Functions	Description	
	<b>Adjust, Active sheet</b>	Use this function to update all blanks of the active Sheet Metal part. If the blanks have different blank parameters, these will be retained, as the blank parameters are saved in the blank.
Pull-down menu  Update		
	<b>Adjust, All sheets</b>	Use this function to update all blanks in the drawing, e.g. after changes to the Sheet Metal part or the parameters. If the blanks have different blank parameters, these will be retained, as the blank parameters are saved in the blank.
	<b>Parameters, Individual</b>	Use this function to change the blank parameters of a selected blank.
	<b>Parameters, Default setting</b>	Use this function to change the blank parameters of new developments or blanks that were created with the <b>Develop sheet</b> function. In the dialogue window, activate the checkboxes and options for the parameters that are to be shown in the blank.
	<b>Parameters, All</b>	Use this function to change the blank parameters of all developments or blanks in the drawing. In the dialogue window, activate the checkboxes and options for the parameters that are to be shown in the blank.

## Attach flange, with sketch

The operation of the **With sketch** option in the **Attach flange** dialogue window has been simplified: After selecting the outer connecting edge you can only activate sketch edges, even if they are covered by a flange edge.

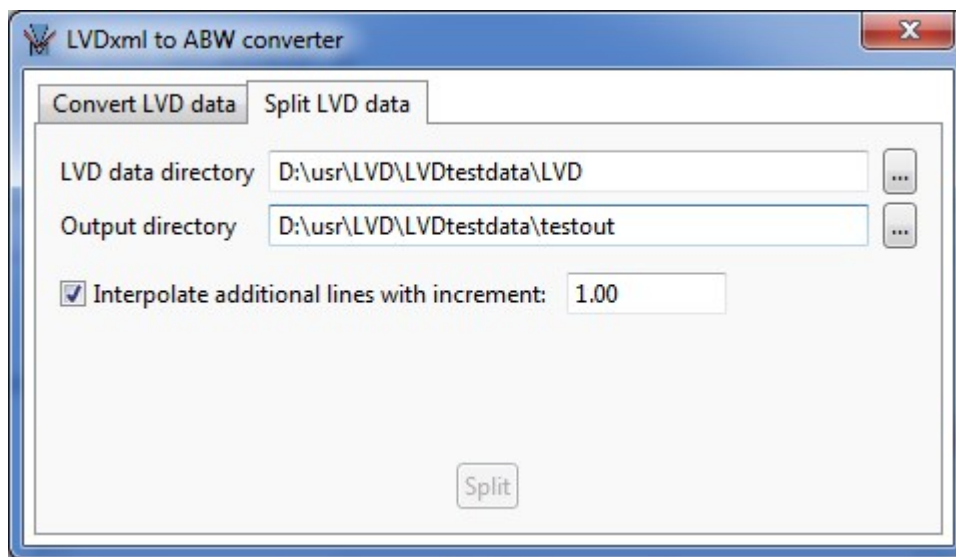


(1) Flange

(2) Sketch

## LVDxml to ABW converter

The options of the **Split LVD data** tab have been expanded:



Activate the **Interpolate additional lines with increment:** checkbox to calculate additional intermediate values for the data in the ABW file.

Example:

The values for bend angles in the LVD data are 10°, 20°, etc.

If the increment is 2, the values for 12°, 14°, 16°, ... will be entered into the ABW file.

### BOM-relevance for new sheets

The part attribute **BOM-relevant** can be set for all new sheets in the SYS/ABWPAR.DAT file.

# Sheets BOM-relevant (yes=1 , no=0)

1

This setting will now also be evaluated by the functions **Sheet from solid** and **Connecting sheet** functions.



## Steel Engineering

Service Pack 2 2016 (V 2102)

### Important notes

#### Note on updates

When you run the tool **Computer Parameter Configurator** (HiCAD exe\ParKonfigComp.exe) – either during the installation/update (Extended Settings > User Configuration tab) or later – various system files in the HiCAD sys directory will be replaced, without any further query, with the files supplied in the HiCAD templates directory which have been designed especially for a working with the selected module and the activated checkboxes (e.g. ...DSTV or Management + BIM). These files include, for example

- HiCAD masks for the part attributes (\*.HDX),
- HELIOS masks for projects, documents and articles (\*.MSK),
- Various configuration files (\*.HDB) and
- Template files for annotations (\*.FTD).

**You should therefore always make backups of all files that you have customized before using the ParKonfigComp.exe tool!**

**If you have any questions, please contact your nearest ISD branch office.**

#### Changed attribute masks when using the default template for Steel/Metal Engineering

If you have selected the ISD default template for Steel/Metal Engineering for the parameter configuration (s.a.), the Part attributes masks in the HiCAD sys directory will be replaced accordingly. These masks have been slightly modified in SP2. For example, these masks now contain the fields Article No. and Standard instead of the fields "Part name" and "Article number". Also, the position of some fields has changed.

## Assembly

**Part attributes**

Article No.	Assembly IPE 300	BOM-relevant	<input checked="" type="checkbox"/>
Standard		Total quantity	1
Drawing No.		Qty. per part	1
Item No.	1	Item text	
Length	2425.0	Width	150.0
Weight	110.70	Height	320.00
	<input type="checkbox"/> Fixed		
Execution class	EXC2 (EN 1090)		
Additional tolerance	Class 1		
Usage	..		
Coating			
Coating, internal		Coating, external	
Designation 1			
Designation 2	1x Assembly IPE 300		
Comment			
System notes			
Part type	Assembly		

Apply changes Cancel

Beam

**Part attributes**

Article No.	IPe 300	BOM-relevant	<input checked="" type="checkbox"/>
Standard	DIN 1025-5	Total quantity	1
Drawing No.		Qty. per part	1
Item No.	102	Item text	
Length	2200.0	Width	
Angle bottom/left - XZ	0.0	height	
Angle bottom/left - YZ	0.0	Angle bottom/right - XZ	0.0
Surface	2.55	Angle bottom/right - YZ	0.0
Weight	92.84	<input type="checkbox"/> Fixed	
Substance/Material	S235JR		
Usage	..		
Coating	▼		
Coating, internal		Coating, external	
Designation 1			
Designation 2	1x IPE 300-2200.000 S235JR		
Comment			
System notes			
Part type	I - Beams		

Apply changes Cancel

## Steel Engineering plate

**Part attributes**

Article No.	BI 25	BOM-relevant	<input checked="" type="checkbox"/>
Standard		Total quantity	1
Drawing No.		Qty. per part	1
Item No.	103	Item text	
Length	320.0	Width	150.0
Surface	0.10	height	25.00
Weight	9.42	<input type="checkbox"/> Fixed	
Substance/Material	S235JR		
Usage	..		
Coating	▼		
Coating, internal		Coating, external	
Designation 1			
Designation 2	1x BI 25x150.000x320.000 S235JR		
Comment			
System notes			
Part type	Plates		

Apply changes Cancel

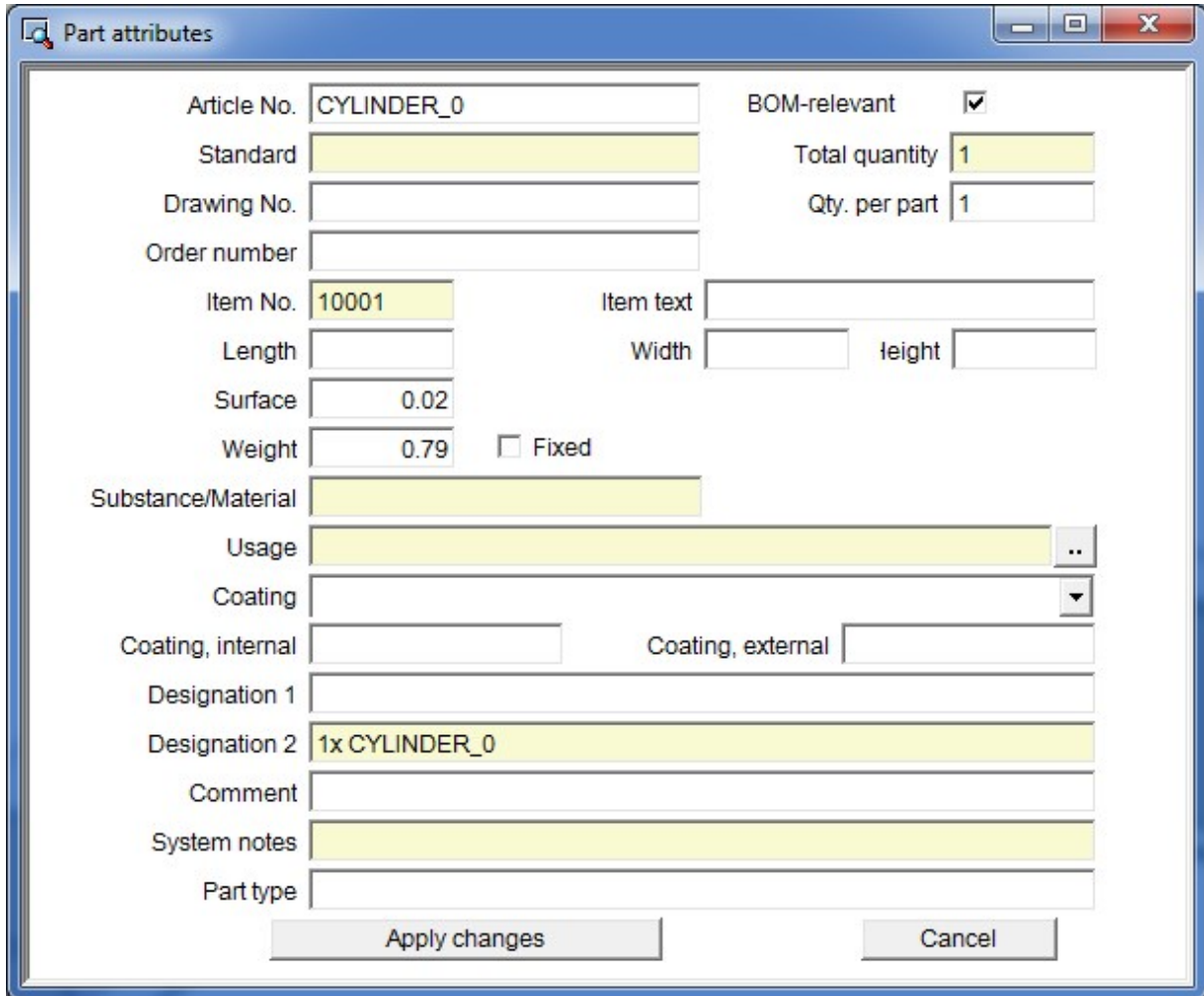
Sheet Metal part

**Part attributes**

Article No.	Sheet flange		BOM-relevant	<input type="checkbox"/>
Standard			Total quantity	
Drawing No.			Qty. per part	1
Order number				
Item No.		Item text		
Length		Width		Height
Surface				
Weight		<input type="checkbox"/> Fixed		
Substance/Material				
Usage				
Coating				
Coating, internal		Coating, external		
Designation 1				
Designation 2				
Comment				
System notes				
Part type				

Apply changes      Cancel

## General 3-D part



The 'Part attributes' dialog box contains the following fields and controls:

- Article No.: CYLINDER\_0
- Standard: (empty)
- Drawing No.: (empty)
- Order number: (empty)
- Item No.: 10001
- Item text: (empty)
- Length: (empty)
- Width: (empty)
- Height: (empty)
- Surface: 0.02
- Weight: 0.79
- ☐ Fixed
- BOM-relevant: ☒
- Total quantity: 1
- Qty. per part: 1
- Substance/Material: (empty)
- Usage: (empty)
- Coating: (empty)
- Coating, internal: (empty)
- Coating, external: (empty)
- Designation 1: (empty)
- Designation 2: 1x CYLINDER\_0
- Comment: (empty)
- System notes: (empty)
- Part type: (empty)
- Buttons: Apply changes, Cancel

## Options / Settings

## STB\_PARAMETER.DAT / Configuration Management

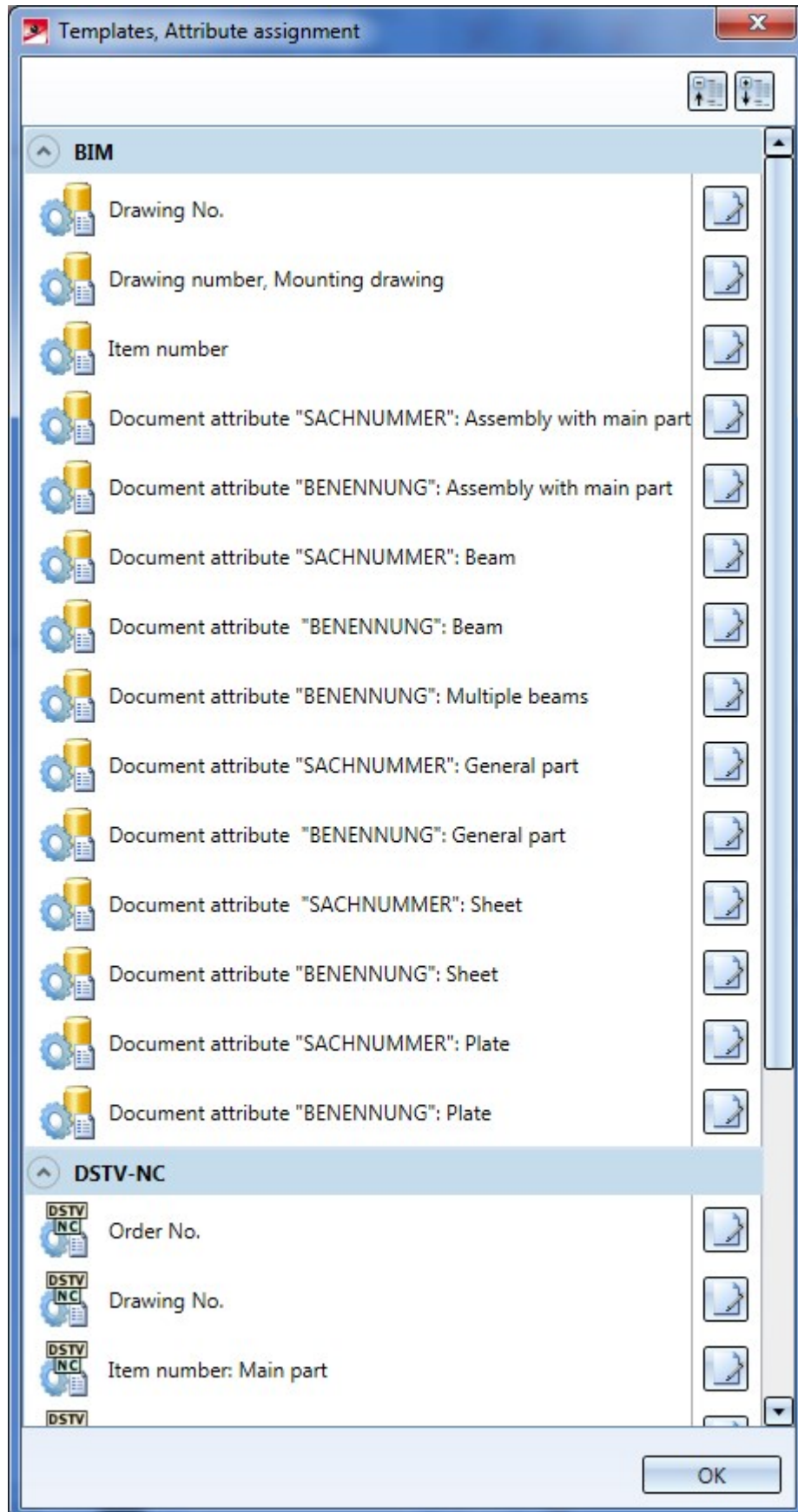
Previously, various Steel Engineering settings could be specified in the system file STB\_PARAMETER.DAT which is now no longer available. Its setting options have been moved to the Configuration Editor at **Steel Engineering**.

## Templates, Attribute assignment

Use the new Templates, Attribute assignment  function to adjust the templates for attribute assignment

- of the Document and Drawing Management in Civil Engineering (Management + BIM) and
- of the DSTV-NC export (see also "Interfaces - What's New?")

Templates, Attribute assignment *dialogue window*







Standard templates for BIM	Settings for
BIM_PDM_Draw- ingnumberGeneration.ftd	Generation of individual drawing numbers The number will be assigned to the HELiOS attribute <b>DRAWINGNUMBER_TEXT</b> .
BIM_PDM_ItemnumberGeneration.ftd	Generation of individual item numbers The number will be assigned to the HELiOS attribute <b>COMPONENT_ITEMNR_TEXT</b> .
BIM_PDM_Mount- ingDrawingnumberGeneration.ftd	Generation of individual drawing numbers for mounting drawings The number will be assigned to the HELiOS attribute <b>DRAWINGNUMBER_TEXT</b> .
BIM_PDM_WSD_Assembly_ ArticleNumber.ftd	Configuration of the HELiOS document attribute <b>SACHNUMMER</b> for detail drawings of assemblies(assemblies with a main part)
BIM_PDM_WSD_Assembly_Desig- nation.ftd	Configuration of the HELiOS document attribute <b>BENENNUNG</b> for detail drawings of assemblies(assemblies with a main part)
BIM_PDM_WSD_Beam_ArticleNum- ber.ftd	Configuration of the HELiOS document attribute <b>SACHNUMMER</b> for detail drawings of beams
BIM_PDM_WSD_Beam_Desig- nation.ftd	Configuration of the HELiOS document attribute <b>BENENNUNG</b> for detail drawings of beams
BIM_PDM_WSD_Multi_Designation.ftd	Configuration of the HELiOS document attribute <b>BENENNUNG</b> for detail drawings of multi-part beams
BIM_PDM_WSD_Part_ArticleNum- ber.ftd	Configuration of the HELiOS document attribute <b>SACHNUMMER</b> for detail drawings of general parts
BIM_PDM_WSD_Part_Designation.ftd	Configuration of the HELiOS document attribute <b>BENENNUNG</b> for detail drawings of general parts
BIM_PDM_WSD_SheetMetal_ ArticleNumber.ftd	Configuration of the HELiOS document attribute <b>SACHNUMMER</b> for detail drawings of Sheet Metal parts
BIM_PDM_WSD_SheetMetal_Desig- nation.ftd	Configuration of the HELiOS document attribute <b>BENENNUNG</b> for detail drawings of Sheet Metal parts
BIM_PDM_WSD_SteelPlate_ ArticleNumber.ftd	Configuration of the HELiOS document attribute <b>SACHNUMMER</b> for detail drawings of Steel Engineering plates
BIM_PDM_WSD_SteelPlate_Desig- nation.ftd	Configuration of the HELiOS document attribute <b>BENENNUNG</b> for detail drawings of Steel Engineering plates
Standard templates for DSTV-NC	Settings for the composition of
DSTV_NC_ORDERNUMBERTEXT .FTD	Order number
DSTV_NC_DRAWINGNUMBERTEXT .FTD	Drawing number
DSTV_NC_H_ITEMNUMBERTEXT .FTD	Item number: Main part
DSTV_NC_H_PARTNUMBERTEXT .FTD	Part number: Main part
DSTV_NC_ITEMNUMBERTEXT .FTD	Item number: Loose parts
DSTV_NC_PARTNUMBERTEXT .FTD	Part number: Loose parts
DSTV_NC_W_ITEMNUMBERTEXT .FTD	Item number: Attached part
DSTV_NC_W_PARTNUMBERTEXT .FTD	Part number: Attached part

DSTV_NC_SIGNATURETEXT .FTD	Lettering
DSTV_NC_FILENAME .FTD	File name - NC



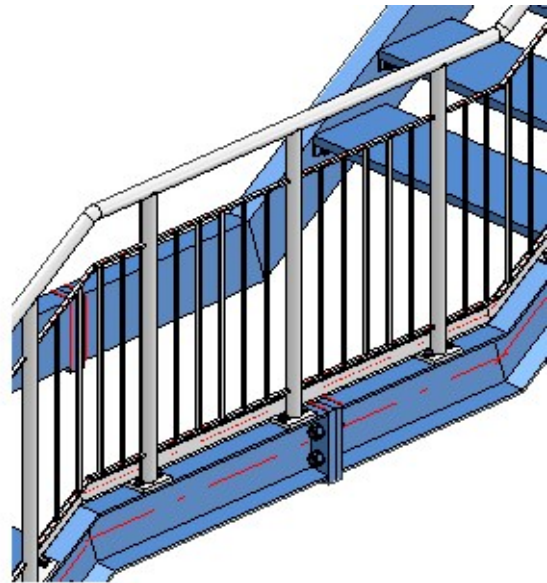
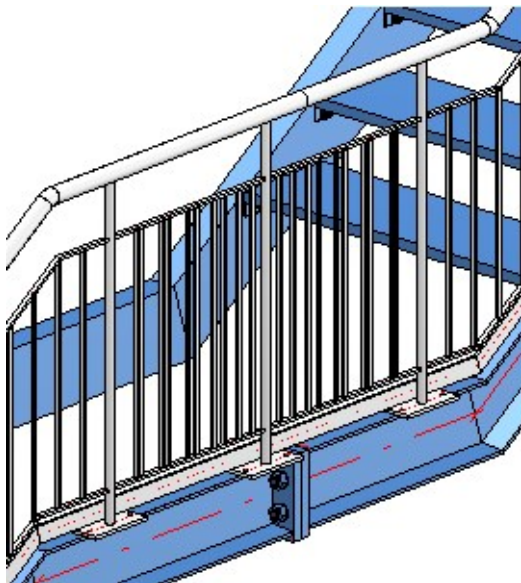
**Please note:**

- The BIM templates are used when the parameter **PDM > Management + BIM > Production drawings > Create HELIOS attributes from FTD file** in the Configuration Editor has been set to **Yes** (Default). For the generation of the drawing number and the item number, specify the number of digits for the parameter **Generate n-digit drawing number** in the Configuration Editor at **PDM > Management + BIM > Production drawings** (or **PDM > Management + BIM**, respectively).
- The DSTV templates will be considered if the corresponding parameters in the Configuration Editor at **Steel Engineering > DSTV-NC** have been set to **Configuration**, and in the **DSTV-NC interface** the option **From templates file** has been selected for letterings and the drawing number, part number, item number and order number.

## Railing Configurator

### Railings on beams in succession aligned in the same direction

If there are several beams in succession that are aligned in the same direction, the railing elements of these beams will be combined into one segment assembly. As a result, continuous hand rails and knee rails will be formed on these beams.



Left: Before HiCAD 2016 SP2; HiCAD 2016 SP2 or higher

## Rectangular plates and Stiffeners

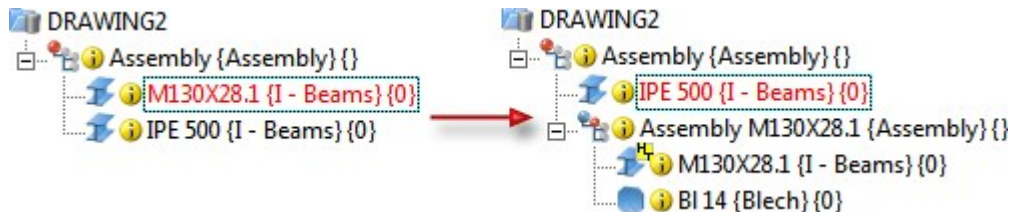
### Automatic calculation of stiffener widths

If you selected the automatic calculation, the calculated width will be displayed in the (then greyed out) input field. Previously the value of the last manual width adjustment was shown here.

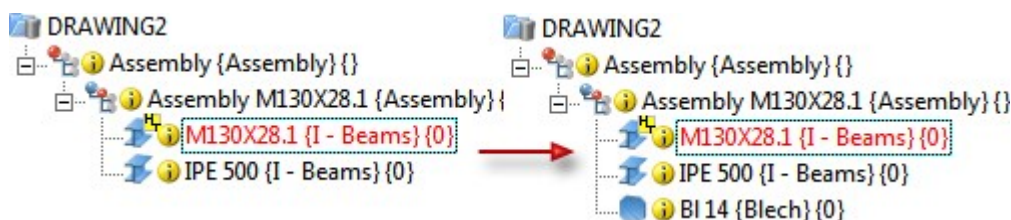
## Forming of assemblies when inserting rectangular plates and stiffeners

When you insert rectangular plates or stiffeners, HiCAD will now proceed as follows for the creation of welded assemblies:

If the reference beam is a main part that does not belong to an assembly, an assembly will be formed, with the beam as assembly main part and the plate as sub-part.




If the reference beam belongs to an assembly, the plate will also be assigned to this assembly as a sub-part.



## Connections

### Front plate connection to flange (2330)

If you use the **All equal**  button on the **Stiffeners** tab to apply the settings to all stiffeners in the Front plate connection (2330), please note that the placing of the stiffeners will be automatically mirrored in the process. For instance, if you have placed the 1st stiffener on the **Front side**, the 2nd stiffener will be placed on the **Back side** when you call the **All equal** function.

### Filler plates with Usage

For the

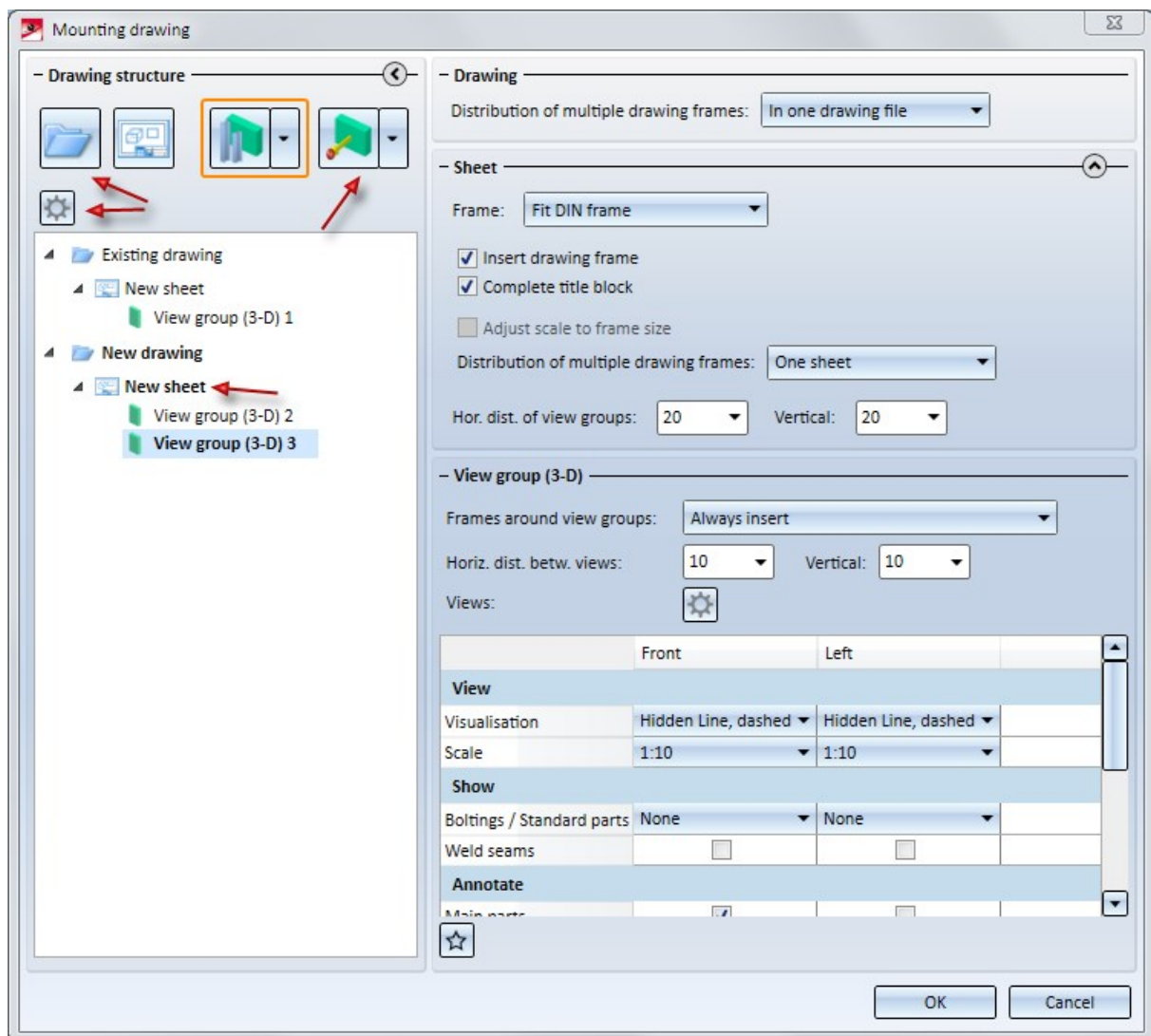
- Front plate connection to web/flange (2320),
- Front plate connection (double-sided) to web (2322),
- Front plate connection to web (2330),
- Purlin joint, 2 plates with mitre cut acc. to DAST IH (2201) and
- End plate (2102)

the Usage **Filler plate** is now automatically assigned to filler plates. In this way you can, e.g. also create usage-dependent configurations for filler plates, which will then be considered for drawing derivations.

## Mounting drawings

### Enhanced dialogue and output to external drawings

The dialogue window for the creation or modification of mounting drawings has been revised and enhanced. For example, the bounding boxes for view groups can now be changed subsequently, and mounting drawings can now also be output to external drawings.



## ■ Output to external drawings

As of SP2, mounting drawings can also be output to external drawings. Please note that mounting drawings in external drawing files can only be updated after modifications in the original model drawing if you use document and drawing management in civil engineering (**Management + BIM**). In this case you will use the corresponding function in the Management+BIM module for the creation of the mounting drawings. In this way, the mounting drawing will be linked to the original model drawing.

## ■ Mounting drawing: Processing of bounding box

The dimensions of the transparent bounding box for the active view group can be changed via the functions of the



menu. The following functions are available:



### Change box dimensions, via facet + point

Here you select a surface of the box and move it to the desired target point. The moving can also be performed dynamically with the cursor.



## Change box dimensions, via facet + value

Here you select a surface of a box and determine the value by which the box is to be moved. The surface will be moved by the value that you enter in the input field.

Also new is the option to re-determine the box, namely, via the following functions:



### Re-determine box via 2 beams



Re-determine box, via plane



Re-determine box, via grid edge


The deleting of the box is now also possible - use the



## Delete box

function for this.

- Extended settings

Clicking the  button opens the dialogue window shown below. Here you can determine the size of the boxes by specifying an excess length for different direction, i.e. enlarge the box in this way. Also, you can determine here how external drawings should be handled.

Create new view groups in...

☒ Active sheet

☐ New sheet

☐ New drawing

Offsets for new boxes

Front / Back: 200

Left / Right: 200

Top / Bottom: 200

Handling of external drawings

☒ Close and back to original drawing

☐ Leave open and back to original drawing

☐ Leave open and show as active drawing



## Retain visibility

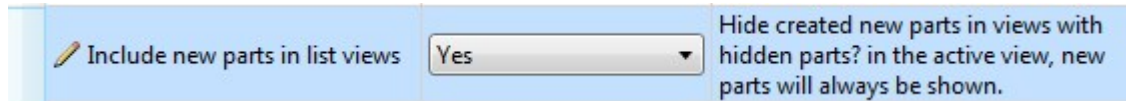
Via the parameter **Include new parts in list views of mounting drawings** you can specify whether created new parts are to be shown or hidden in the views of the mounting drawing upon its updating.

### No

If a view of the mounting drawing contains hidden parts, the created new parts will also be hidden in these views.

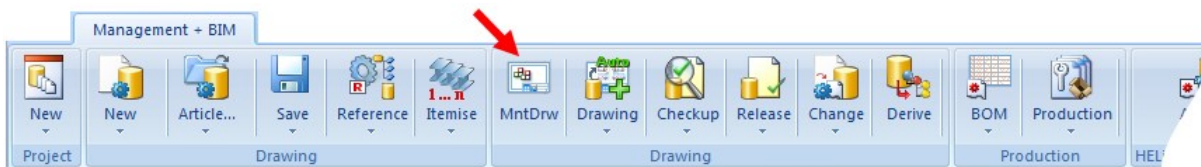
### Yes

Created new parts will always be shown in the views of the mounting drawing. This is the default setting. .



## Management of mounting drawings via Document and Drawing Management in Civil Engineering (Management + BIM)



Like production drawings, mounting drawings can now be centrally managed with the functions of the **Management + BIM** module.



## Production drawings

### Alignment of beams and profiles

The Alignment of derived view functions in the context menu for parts (right-click)

- Active view as front view  and
- Active view as top view 

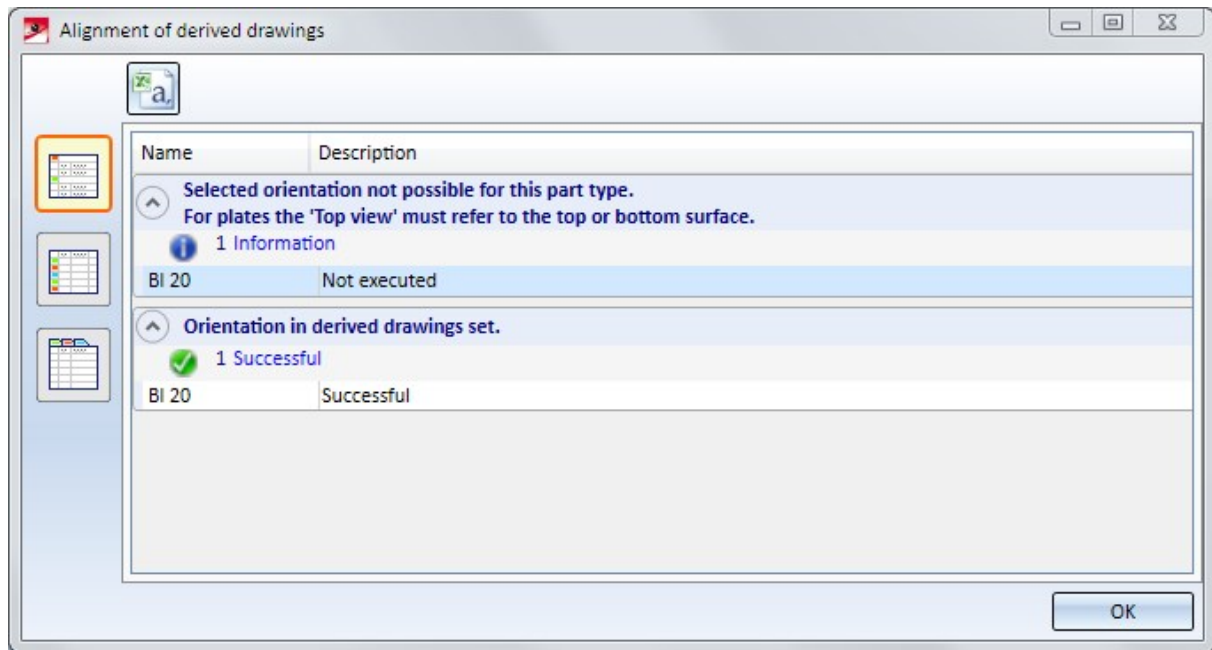
can now also be applied to beams and profiles. In this way you can specify the alignment of views in workshop drawings.

When you specify the alignment, HiCAD will check whether the new alignment of the beam/profile is actually allowed. Please note the following:

- Front views of beams show either the front side or the rear side of the web.
- Front views of round profiles must run horizontally, while allowing a free rotation about their axes. This enables a rotation of the view-defining coordinate system for trimmed round beams in such a way that the cutting angle can be dimensioned in the workshop drawing.

The result of the check will then be shown in a dialogue window. Here it will be listed for which of the chosen beams/profiles the new alignment was successful, and to which no new alignment could be applied.



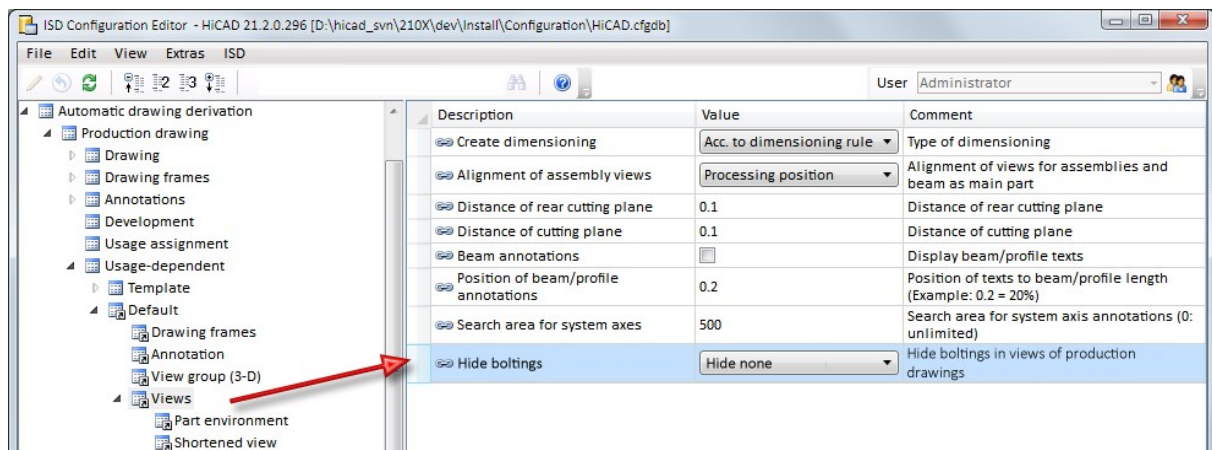


## Hide boltings

For each usage you can specify whether the boltings in the production drawing are to be shown or hidden. You can find the relevant settings in the Configuration Editor at

Automatic drawing derivation > Production drawing > Usage-dependent > *name* > Views

with *name* being the name of the respective usage.



The following options are possible:

- Hide none** No bolts are hidden. This is the default setting and corresponds to the behaviour before HiCAD 2016 SP1.
- Hide all** All bolts are hidden.
- Hide only "site bolts"** Only bolts with the "Site assembly" or "On-site construction" property are hidden.

**Please note:**

- These settings do not apply to bolts which are located in a structure assembly (e.g. boltings that have been inserted as loose parts). These bolts will not be shown in the workshop drawing.
- The settings specified in the Configuration Editor when generating the production drawing will not be saved with the model drawing. If a production drawing is updated, the current settings in the Configuration Editor will be used, and not the settings that were active when the drawing was generated.

**Sectional views for plate-like attached parts**

When generating the production drawing, a sectional view will now not only be created and dimensioned for attached parts of the type "Plate", but also for other, plate-like attached parts, such as flat steels.

**Main parts without assembly**

Main parts that do not belong to an assembly, e.g. parts on the highest level of a part structure, were previously treated as main part of an assembly when a production drawing was created. This means: If the **Main parts as assembly** checkbox beneath **View groups to be created for:** in the **Drawing derivation** dialogue window was activated, the settings for assemblies were used for these parts. If the **Main parts, individually** checkbox was activated, the settings for the corresponding part type were used for these parts. If both checkboxes were activated, two drawings (view groups) were created - one for the assembly and one for the individual part.

As of SP2 these parts will only be considered for the production drawing if the **Main parts, individually** checkbox beneath **View groups to be created for:** in the **Drawing derivation** dialogue window was activated. In this way it is assured that only one view group will be created for such a part.

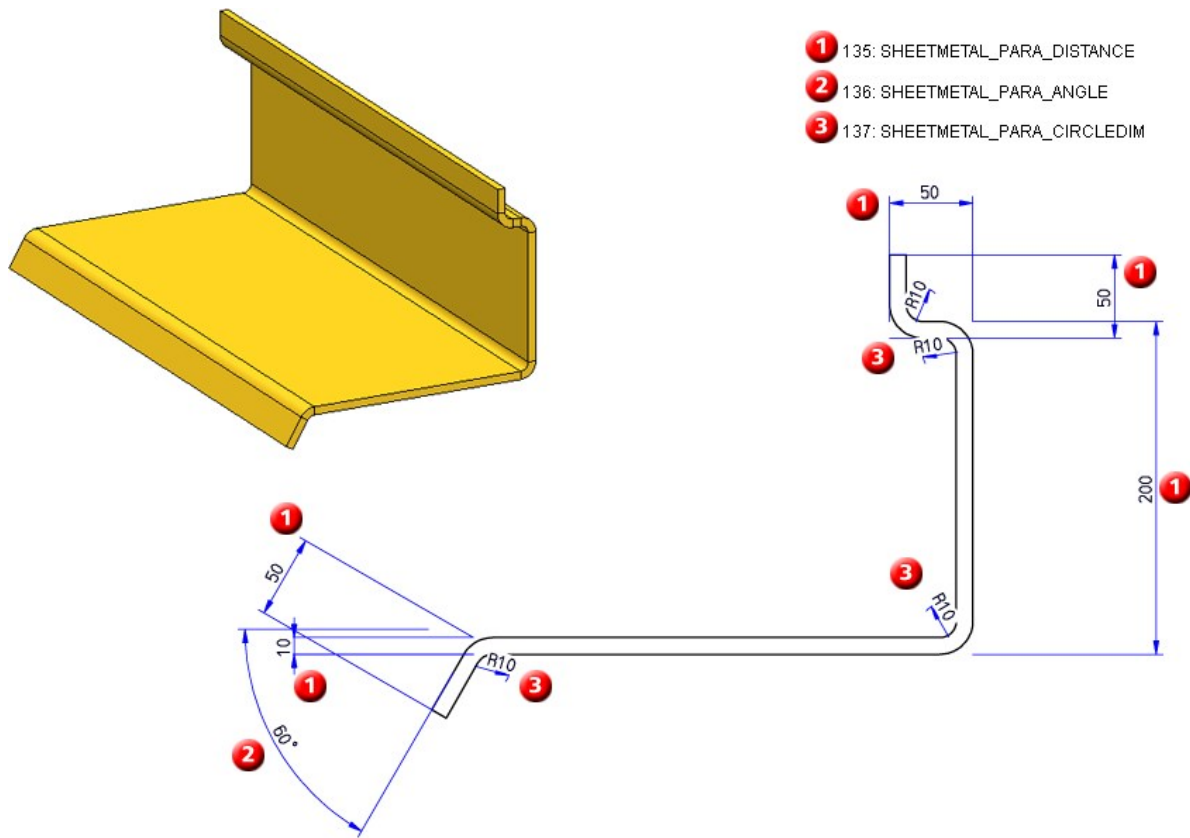
**New / expanded dimensioning rules****Parametric dimensions for Sheet Metal parts**

You can now also use parametric Sheet Metal dimensions for the automatic dimensioning of Sheet Metal parts in workshop drawings. The following rule are available for this purpose:

Rule		Internal designation
132	Sheet length	SHEETMETAL_LENGTH
133	Sheet width	SHEETMETAL_WIDTH
134	Sheet height	SHEETMETAL_HEIGHT
135	Parametric linear dimensions of Sheet Metal parts	SHEETMETAL_PARA_DISTANCE
136	Parametric angular dimensions of Sheet Metal parts	SHEETMETAL_PARA_ANGLE
137	Parametric circle dimensions of Sheet Metal parts	SHEETMETAL_PARA_CIRCLEDIM



Please note that for Rule 136 angles with 90, 180, 270 and 360 degrees will never be shown.



Example- 3-D sheet and detail of production drawing (front view) with dimensioning according to Rules 135, 136 and 137

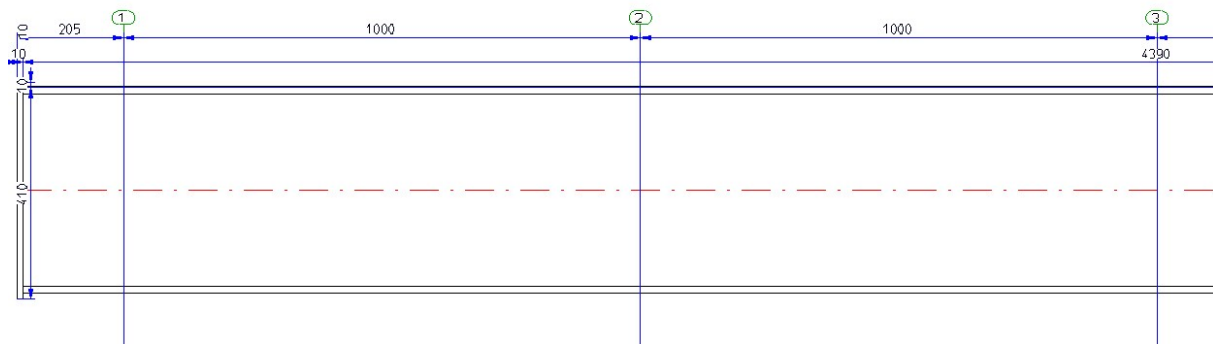
## Expansion of rules for sub-systems

The following rules for sub-systems:

- 71: Sub-system in upper flange area,
- 72: Sub-system in lower flange area,
- 73: Sub-system in web area,
- 74: Sub-system in upper flange and lower flange area and
- 75: Sub-system in upper flange, lower flange and web area

offer further selection options for reference selection:

- **System axes**
- **Beam with attached parts**  
Beam start or end without front-mounted attached parts
- **Beam without attached parts**  
Beam start or end with front-mounted attached parts



### Attached plates/sheets on relevant beam sub-parts

Sheets and plates in welded assemblies (welded frameworks) do not always refer to the main part of the assembly, but may also refer to the neighbouring beams (defined by contact surfaces). Such beams need not necessarily be main parts.

Based on the Rules 57 - 61, similar rules are now available for attached plates and sheets on relevant beam sub-parts.

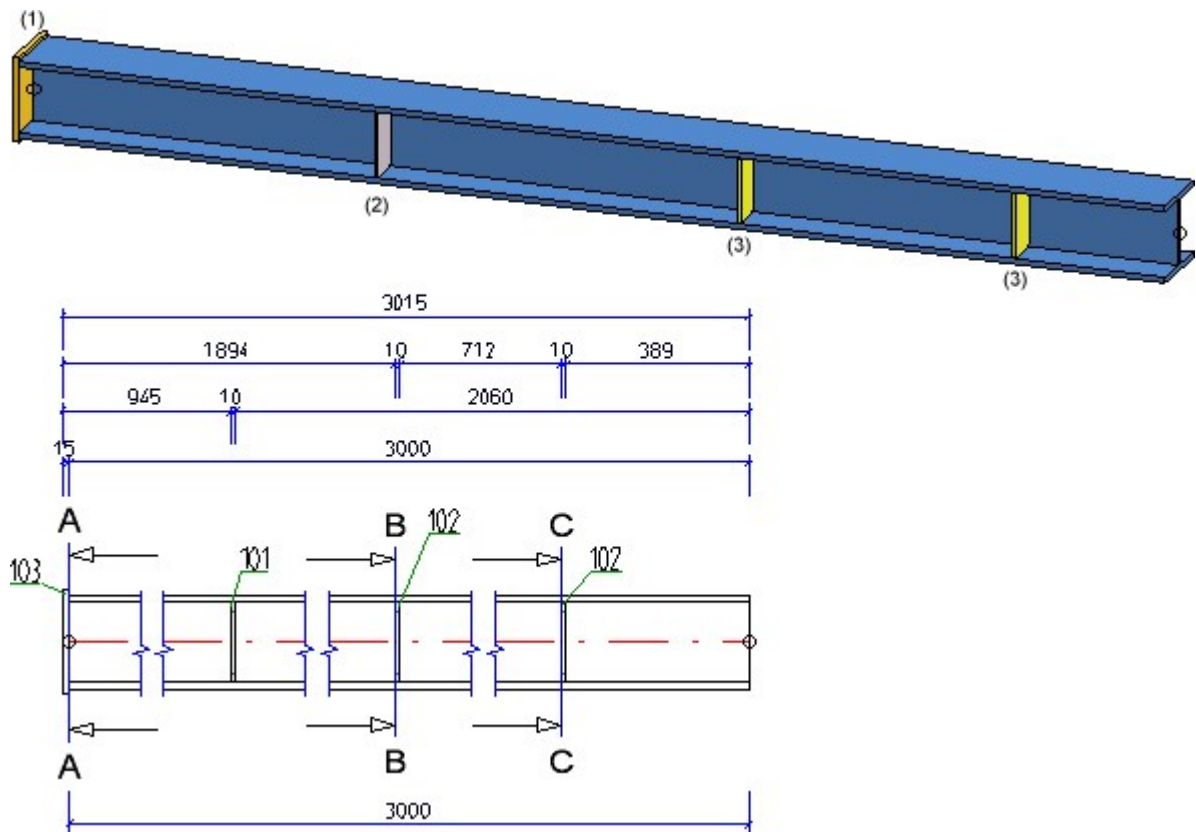
105	Attached plates/sheets on relevant beam sub-parts; one dimension chain for each plate/sheet	ATTACHING_SHEETS_SEPERATELY_RELVPROFS
106	Attached plates/sheets on upper beam sides on relevant beam sub-parts; via bores or outer contour	ATTACHING_SHEET_UPPERSIDE_RELVPROFS
107	Attached plates/sheets on lower beam sides on relevant beam sub-parts; via bores or outer contour	ATTACHING_SHEET_LOWERSIDE_RELVPROFS
108	Attached plates/sheets on web surface on relevant beam sub-parts, before web, via bores or outer contour	ATTACHING_SHEET_BEFORE_WEB_RELVPROFS
109	Attached plates/sheets on web surface on relevant beam sub-parts, after web, via bores or outer contour	ATTACHING_SHEET_BEHIND_WEB_RELVPROFS

### Separate chain dimensions for different usages

You have the option to create separate chain dimensions per different usage for chain dimensions describing the fitting situation of parts. To change this behaviour, open the **STW\_DimSettings.XML** (conventional dimensioning settings) file in the HiCAD sys directory and set the parameter **SEPERATEDSUBPARTPOSITION** to **1**:

**</PARAM><PARAM Name="SEPERATEDSUBPARTPOSITION" Typ="INT" Value="1">**

The image below shows a beam with 4 plates for different usages (1=Front plate, 2=Reinforcement plate, (3) Stiffener) and the front view of the production drawing.



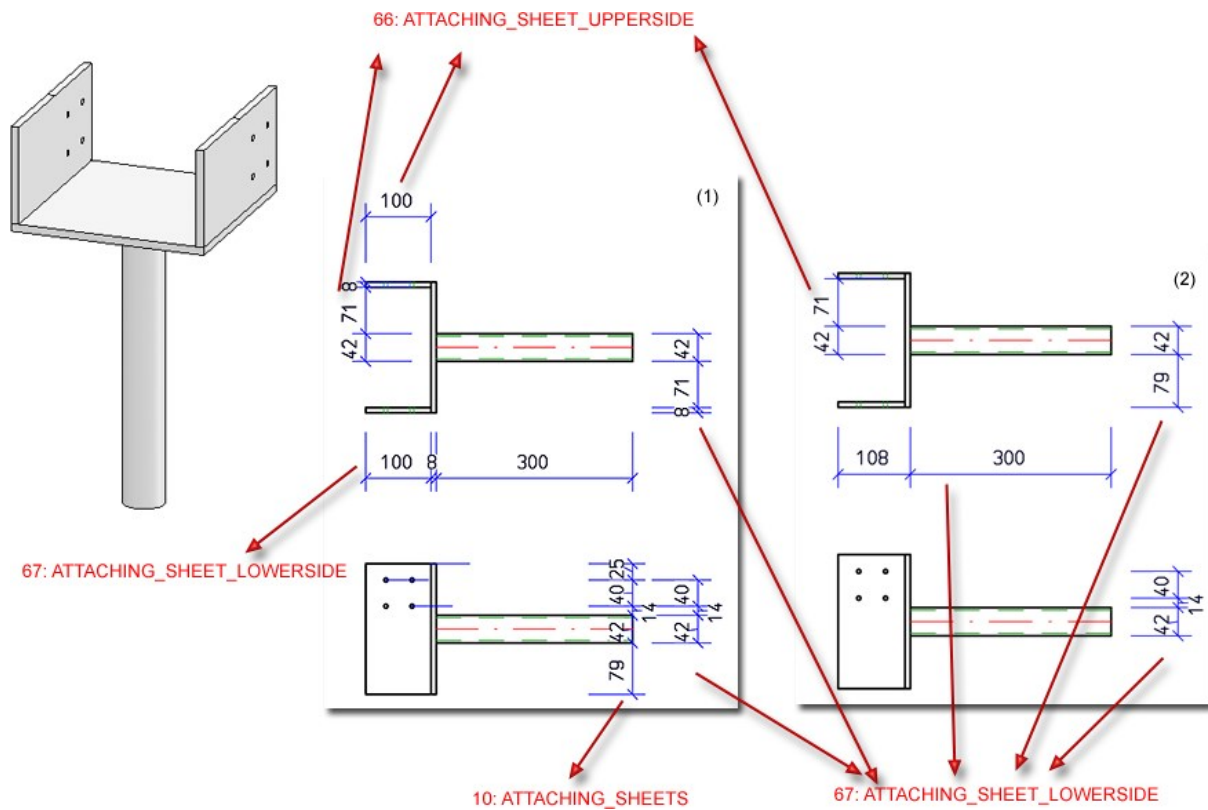
## Attached parts - only dimension first contact point

When dimensioning attached sheet/plate parts or plates and beams that border on dimensionally relevant beams, both contact points will always be dimensioned. If only the first contact point in the chain is to be dimensioned, open the **STW\_DimSettings.XML** (conventional dimensioning settings) file in the HiCAD sys directory and set the parameter

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

to 1.

This setting affects the Dimensioning rules 8, 10, 66, 67, 68, 69, 70, 94,96, 110, 113, 114, 115, 116, 117.



Example - (1) without changing STW\_DimSettings.XML ; (2) with change of STW\_DimSettings.XML

### Shipping list with axonometric representation of assemblies

When generating BOMs using the **HiCAD\_Stahlbau.RMS** template you can display an axonometric representation of the assembly as an image in the Shipping list in Excel format.

*Example of a shipping list with assembly images*

More information can be found in the News for the Report Manager.

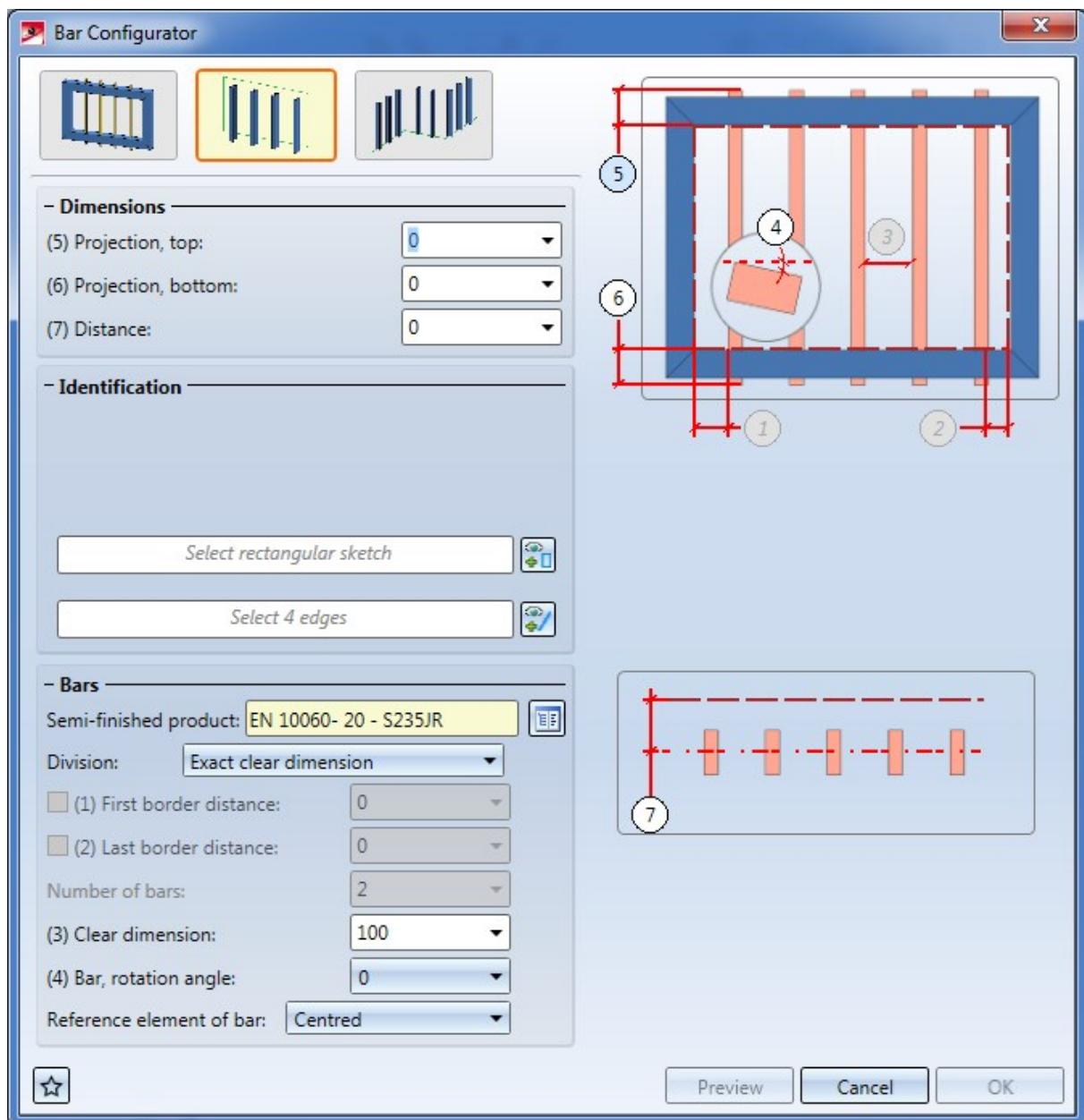
## Service Pack 1 2016 (V 2101)

## General Design Variants

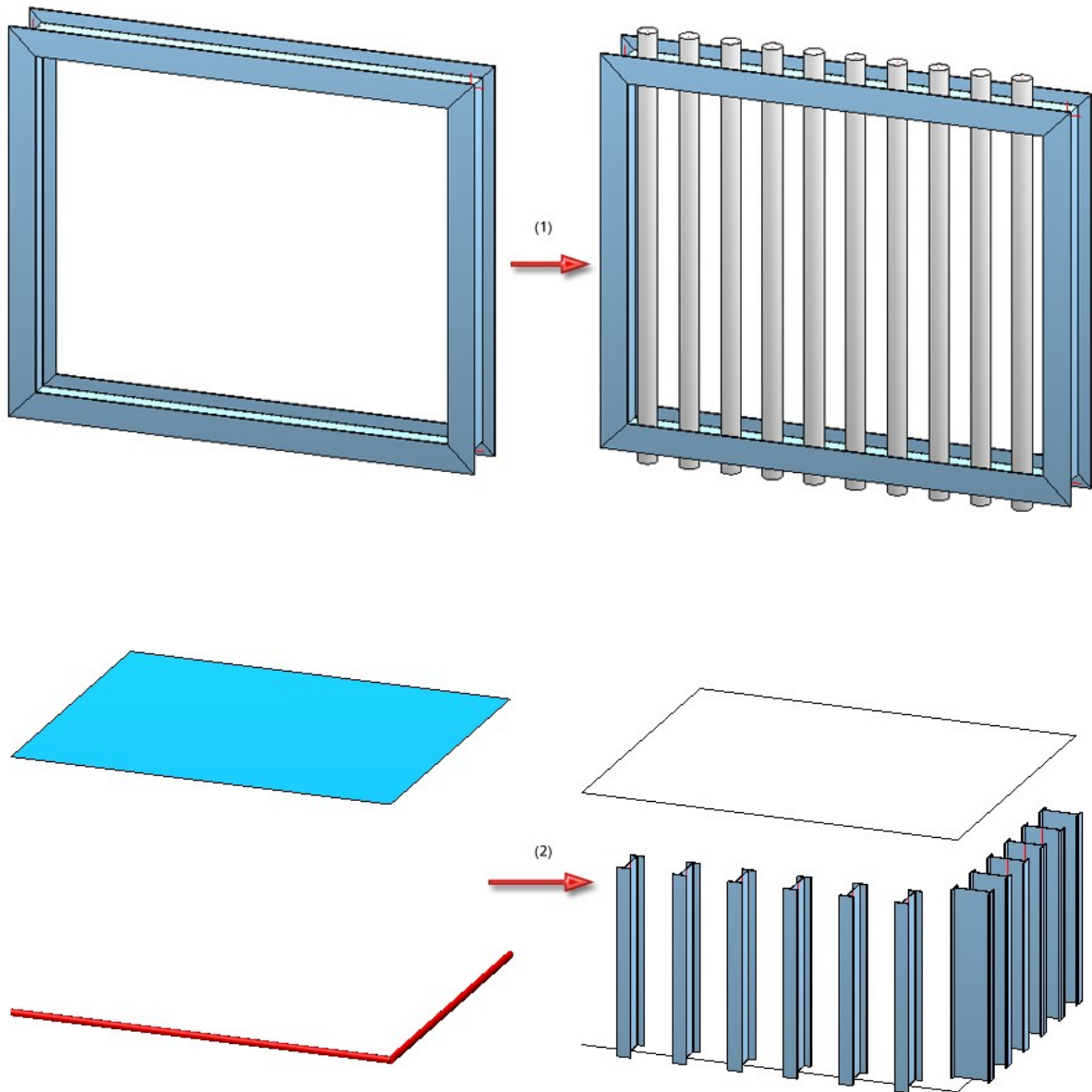
## Bar Configurator

Use the **Bar Configurator** dialogue window to fill the area between 4 boundary beams, or the area between 4 edges of a sketch with bars. The placing of bars along composite edges is also possible.





Example - (1) Bars between beams; (2) Bars along composite edge

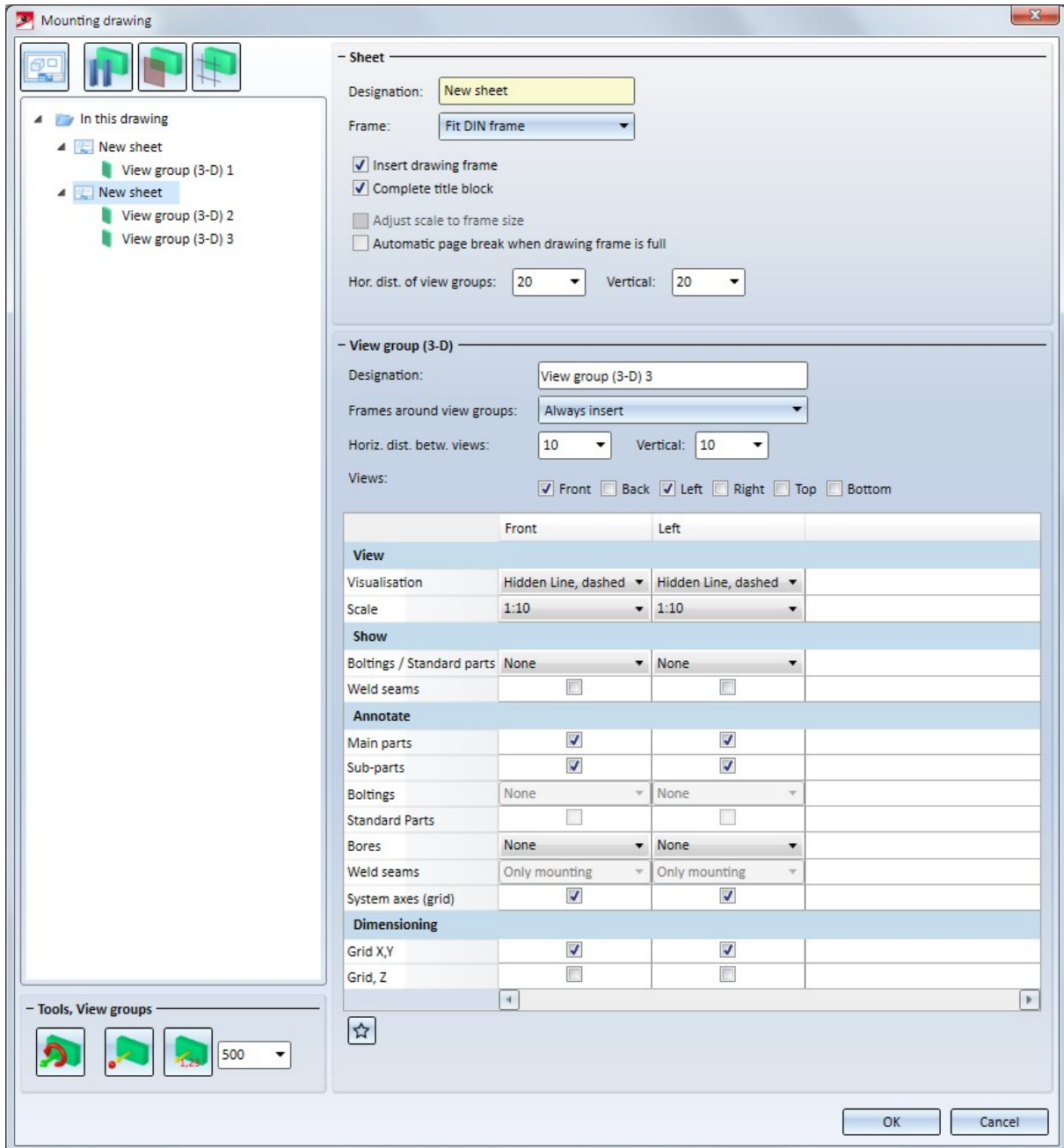


## Mounting drawings

### Redesigned dialogue window

The dialogue window for the creation and processing of mounting drawings has had a redesign:

*New dialogue window*



**Mounting drawing**

**- Sheet -**

Designation:

Frame:

☒ Insert drawing frame

☒ Complete title block

☐ Adjust scale to frame size

☐ Automatic page break when drawing frame is full

Hor. dist. of view groups:  Vertical:

**- View group (3-D) -**

Designation:




Frames around view groups:

Horiz. dist. betw. views:  Vertical:

Views: ☒ Front ☐ Back ☒ Left ☐ Right ☐ Top ☐ Bottom

	Front	Left	
<b>View</b>			
Visualisation	<input type="text" value="Hidden Line, dashed"/>	<input type="text" value="Hidden Line, dashed"/>	
Scale	<input type="text" value="1:10"/>	<input type="text" value="1:10"/>	
<b>Show</b>			
Boltings / Standard parts	<input type="text" value="None"/>	<input type="text" value="None"/>	
Weld seams	<input type="checkbox"/>	<input type="checkbox"/>	
<b>Annotate</b>			
Main parts	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
Sub-parts	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
Boltings	<input type="text" value="None"/>	<input type="text" value="None"/>	
Standard Parts	<input type="checkbox"/>	<input type="checkbox"/>	
Bores	<input type="text" value="None"/>	<input type="text" value="None"/>	
Weld seams	<input type="text" value="Only mounting"/>	<input type="text" value="Only mounting"/>	
System axes (grid)	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
<b>Dimensioning</b>			
Grid X,Y	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
Grid, Z	<input type="checkbox"/>	<input type="checkbox"/>	

**- Tools, View groups**

OK Cancel

## New options

The following new option for the specification of bounding boxes for detail views are available:



### New view group by selection of 2 beams

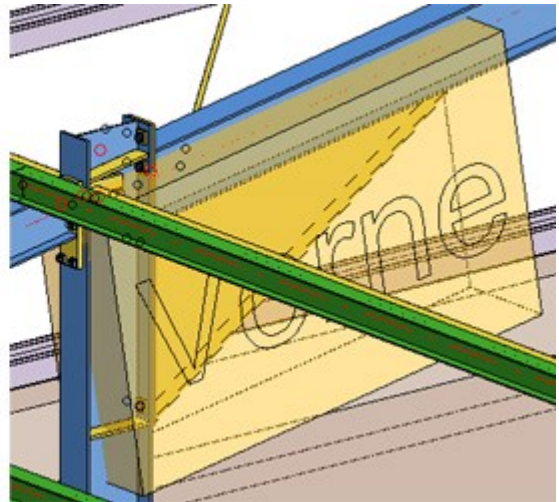
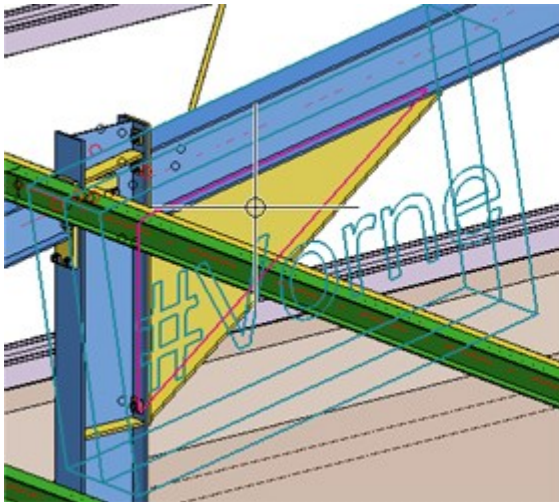
Here the bounding box is determined by the selection of 2 beam ends. HiCAD automatically calculates the smallest possible cuboid that completely covers the two beams, as visualized it as a transparent box in the drawing.

View group 1: Beam (1) and (2); View group 2: Beam (3) and (4)



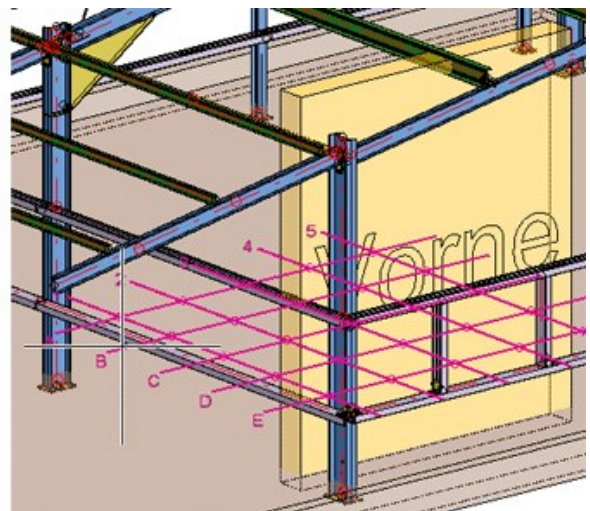
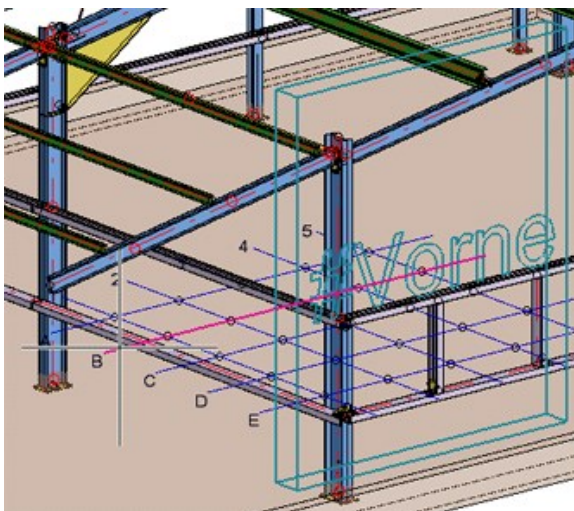
### New view group by plane selection

The bounding box is determined by the selection or definition of a processing plane. For the selection or definition of the processing plane you can use the same options as for the Processing plane function. The bounding box is created by extrusion of the processing surface to the front and to the back.



### New view group by grid edge selection

The bounding box is created by the selection of a Steel Engineering grid edge. The bounding box is created by extrusion of the edge running perpendicular to the grid edge to the front and to the back.



The transparent box that visualizes the bounding box can be changed via the functions beneath **Tools, View groups**. The bounding box can also be rotated.

Also new is the option to auto-create grid annotation tags for multi-level grids and add chain dimensions.



## Staircase and Railing Configurator

## Railing Configurator - Lateral post connections

The lateral post connection has been optimized. You can now also choose between 2 bores or 4 bores in fixing plates.

[illegible]

### Railing Configurator - Insertion of corner posts

When inserting corner posts, you can now specify on the **Corner post - Substructure** tab how the post is to be mounted to the substructure.

– Corner post (for 'Post distribution' you need to select 'Create corner post') —

Variants: Corner post connection, lateral

\* See Parameter 8 for 'Post'

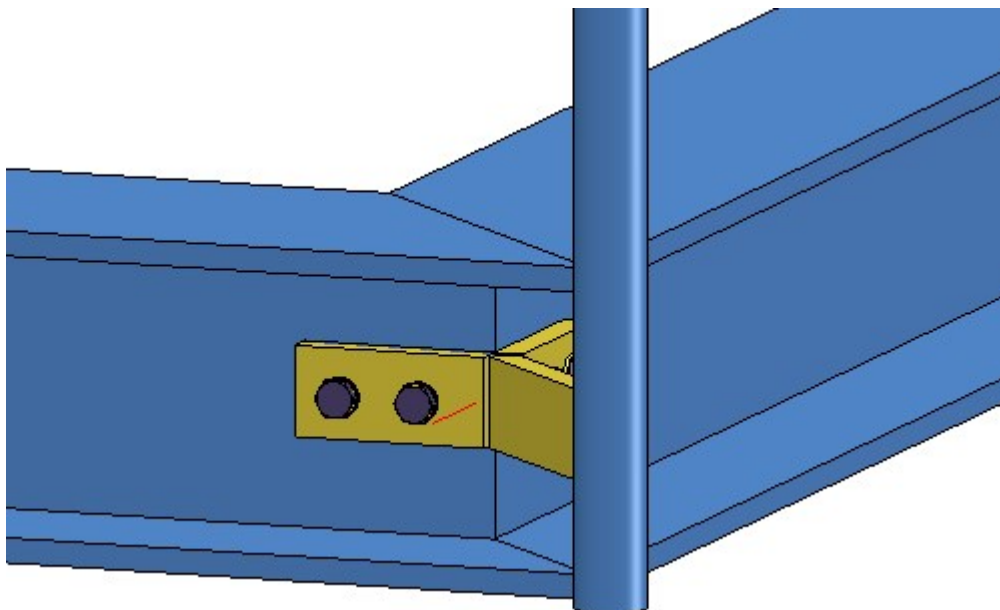
Distance plate: FI 50x15 - S235JR

Base plate: ☒ BI 14 - S235JR

(3) 50 (2) 100  
 (4) 75  
 (9) 75

Boltings: ☒ DIN EN 14399-3-M12-8.8

(7) Excess length of post: 20



## Railing Configurator - Corner rods

For vertical rods you have the additional options to place one rod in the corner. To do this, activate the **Corner rod** checkbox on the **Infill** tab.

*Input values for vertical rods with booms*

**- Infill, corner**

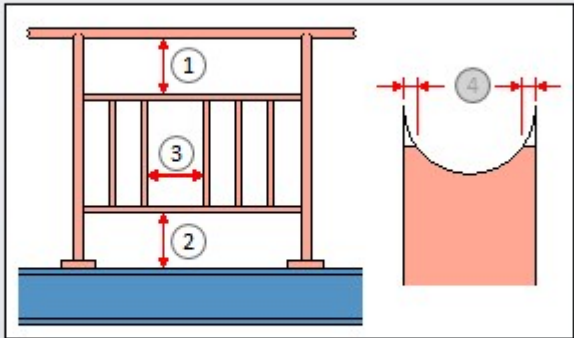
Variants: **Vertical rods with booms**

(1) Top: 150  
(2) Bottom: 100  
(3) Distance: 100  
(4) Width: 1

☐ Trim to post

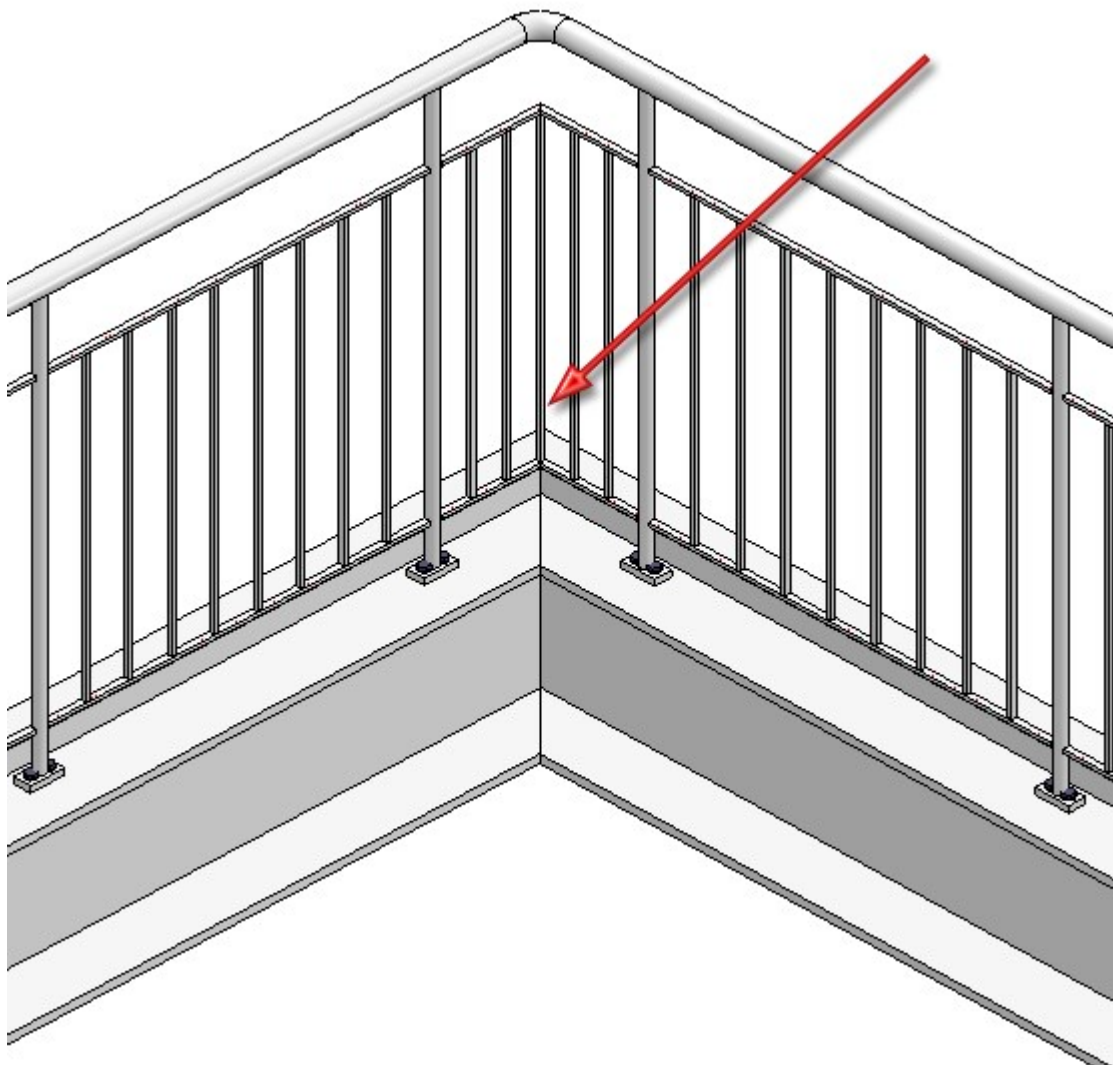
Booms: FI 15x8 - S235JR  
Rods: VKT 10 - S235JR

☒ Corner rod







*Railing with corner rod*



## Change/Update stairs and railings

The functions

- Update connection  and
- Change connection 

can also be applied to stairs and railings that were created with the Staircase/Railing Generator if you have subsequently deleted parts that were created by the corresponding variant. However, these deleted parts will be re-created upon updating if these are required due to the parameters and the geometrical situation.

## Boltings

### New bolt types

Steel Engineering boltings  now also include the new bolt types

- DIN EN 14399-7,
- DIN EN 14399-8 and
- DIN EN 14399-10.

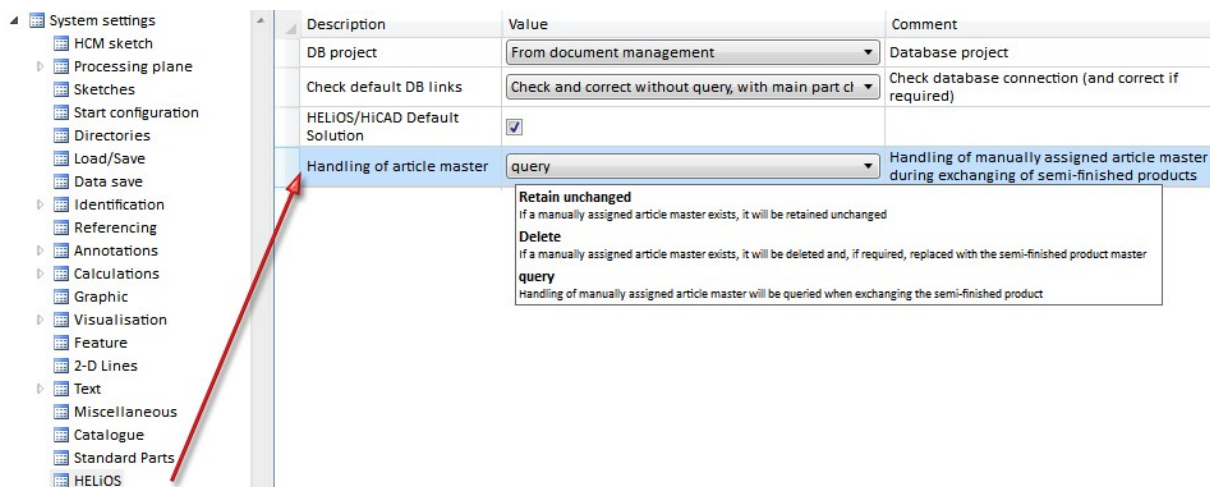
## Process beams

### Selection of beams

Previously, assembly points could be snapped during selection of beams and profile, causing the confusing error message *Part is no Steel Engineering part*. From HiCAD 2016 SP1 onwards, only actual beam and profile-related data will allow the selection of beams.

### Handling of article masters during exchanging of semi-finished products

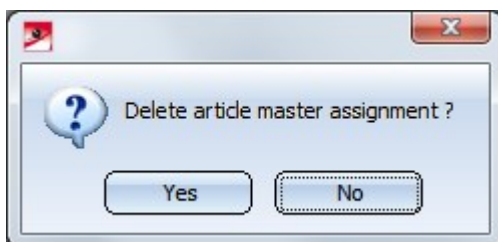
The handling of manually assigned article masters during exchanging of semi-finished products can be specified in the **Configuration Editor** at **System settings > HELIOS**.



The following setting options for the parameter **Handling of article master** are available:

#### ■ Query

If the semi-finished product to be exchanged has a manually assigned article master, a query will be issued:



If you choose **Yes**, the article master of the semi-finished product to be exchanged will be deleted and, if required, replaced with the article master of the semi-finished product to be exchanged. If you choose **No**, the article master of the semi-finished product to be exchanged will be retained unchanged. This is the default setting.

## ■ Delete

An article master that is manually assigned to the semi-finished product to be exchanged will be deleted without any further query and, if required, replaced with the article master of the semi-finished product to be exchanged.

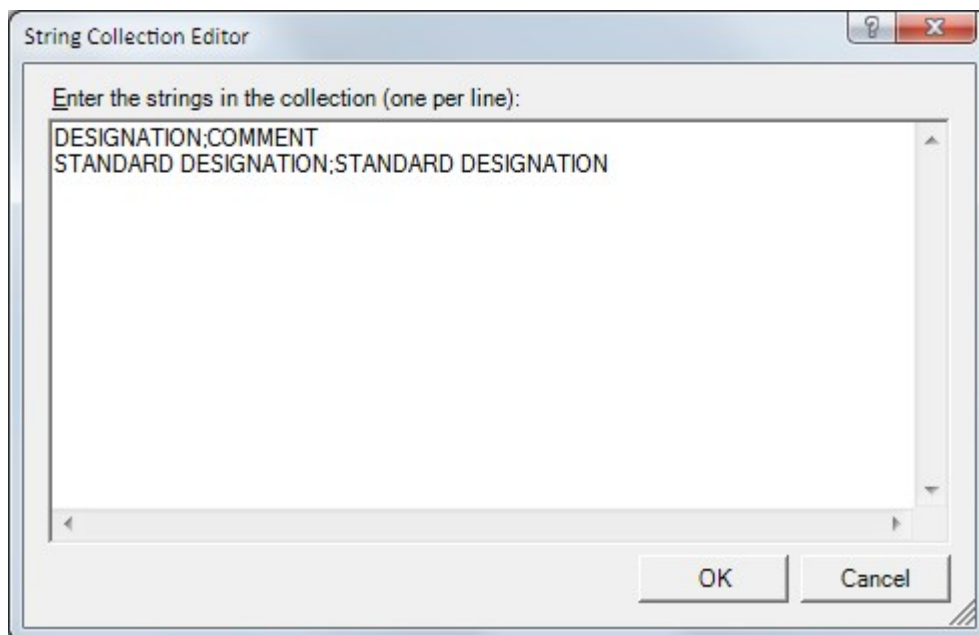
## ■ Retain unchanged

An article master that is manually assigned to the semi-finished product to be exchanged will be retained unchanged in the exchange process. No query will be issued.

At **Taking over of semi-finished product attributes** you can additionally specify which semi-finished attributes will be assigned to the manually assigned article attributes. The assignment must be defined as follows:

### **Semi-finished product attribute;Article attribute**

Each assignment must be specified in a separate row in the collection, e.g. as follows:



The String Collection Editor is initially empty by default.

## Connections

### Assigning of boltings

The assigning of bolting has been changed for the following connections:

- Beam to web with 2 Plates + Stiffener (1211),
- Front Plate Connection to Web/Flange (2320)
- Front Plate Connection to Flange (2330)

### 1211

The bolting is inserted as a **Bolting** group. Next to **Assignment** on the **Boltings** tab you can specify whether this Bolt- ing is to be assigned to the assembly of the 1st beam or whether it is to be inserted as a "Loose part". If you choose the "Loose part" option, the **Bolting** group will be inserted as a separate assembly on the same level as the assembly of the 1st beam.

### 2320

The bolting is inserted as a **Bolting** group. Next to **Assignment** on the **Boltings** tab you can choose whether this

Bolting group and the reinforcement plates are to be assigned to the assembly of the 1st beam or are to be inserted as **Loose parts**. If you activate the **Assembly (1st beam)** option, the **Bolting** group and the reinforcement plates are assigned to the assembly of the 1st beam. If you activate the **Loose part** option, the **Bolting** group will be inserted as a separate assembly on the same level as the assembly of the 1st beam. The reinforcement plates will be assigned to a structure assembly called **Loose parts** that also contains filler plates. This structure assembly is also located on the same level as the 1st beam.

### 2330

The bolting is inserted as a **Bolting** group. Next to **Assignment** on the **Boltings** tab you can specify to which assembly the boltings and the reinforcement plates are to be assigned. If the **Assembly 1st beam** option is active, the boltings and the reinforcement plates will be assigned to the assembly of the 1st beam. If you activate the **Loose part** option, the **Bolting** group will be inserted as a separate assembly on the same level as the assembly of the 1st beam. The reinforcement plates will be assigned to a structure assembly called **Loose parts**. If you have also inserted filler plates, reinforcement plates and filler plates will be located in this assembly.

### Purlin joint, 2 plates with mitre cut acc. to DAST-IH (2201)

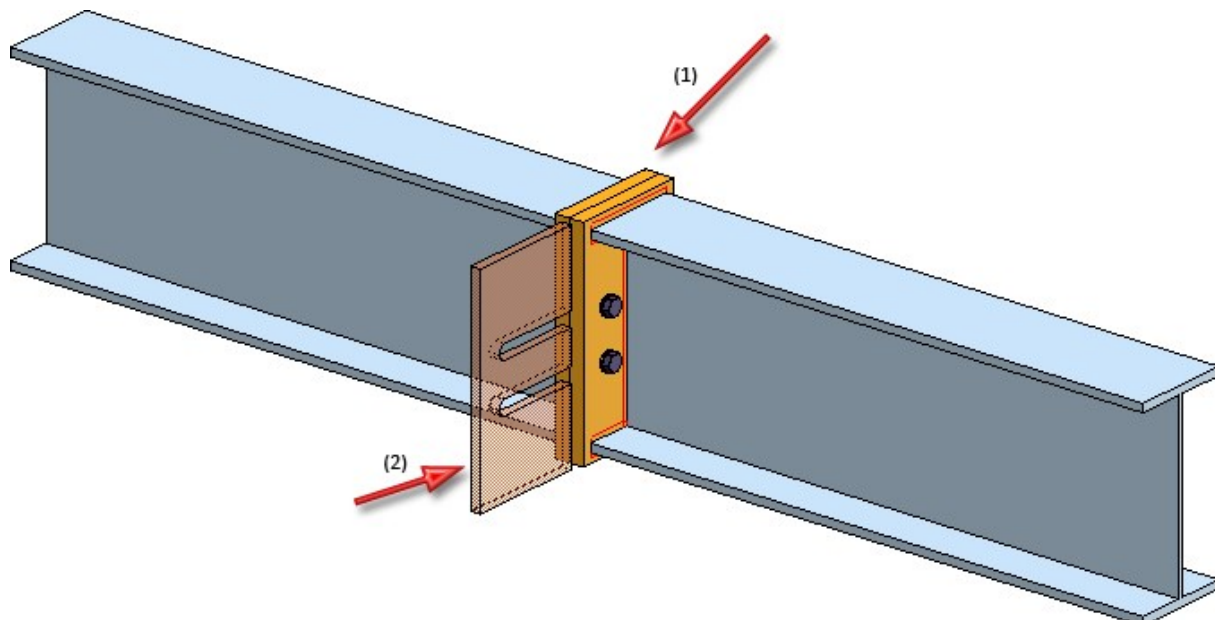
Use this new function to connect two beams by means of two rigid front plates that are bolted together. These can either be flush or with excess length to the top, the bottom or the sides. The bolting can be aligned symmetrical or unsymmetrical. If required, the beams can be automatically shortened, lengthened or mitre cut. Furthermore, weld seams or filler plates can be directly inserted.

The connection can be made either to the web or to the flange. Allowed beam types are I-beams, U-beams, L-beams, T-beams, Z-profiles, pipes and hollow profiles.

The connection can either be user-defined or configured via predefined DAST tables integrated in HiCAD.

This connection replaces the previous Purlin joint 1201.

*Example 1 - Front plate connection (1) with non-pre-planned filler plate (2)*



### End plate (2102)

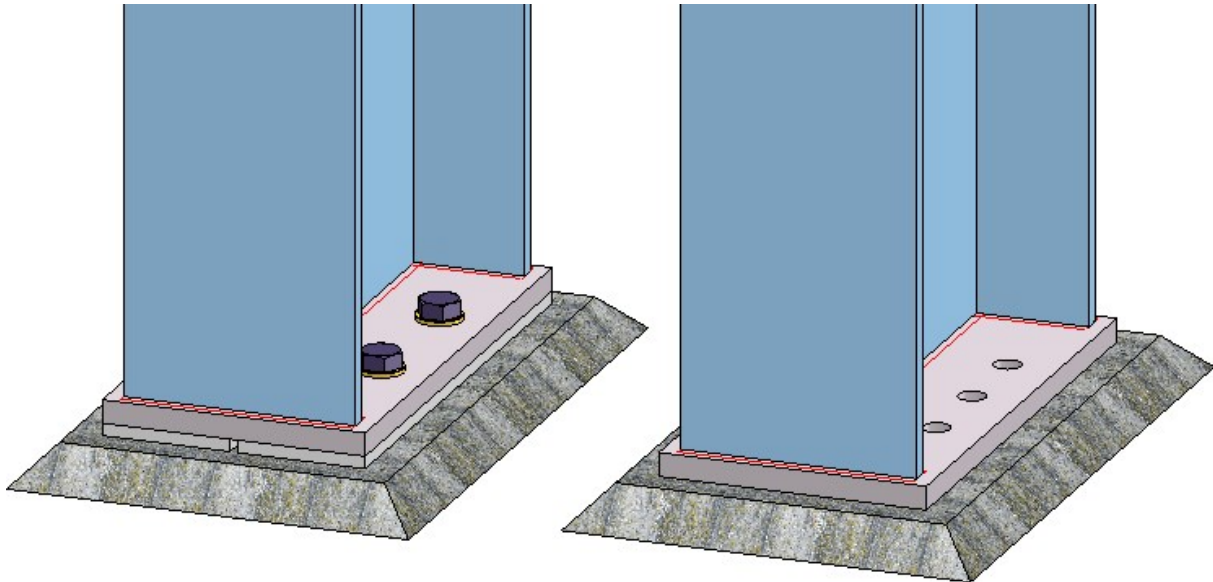
Use this function to connect an end plate to a beam. The plate can be inserted with a bolting, pre-drilled, with or without filler plates, and with or without base. If desired, weld seams can be directly inserted.

Possible beam types are I-beams, U-beams, L-beams, T-beams, Z-profiles, pipes and hollow profiles.

Allowed semi-finished products for the plate are simple Plates, Flat steels, FLUTZ profiles, A3 Steel bars by Frank-stahl or C3 Stainless steel bars.

The Design Variant 2102 replaces the previous Design Variant End plate 1102.

*Example - End plate*



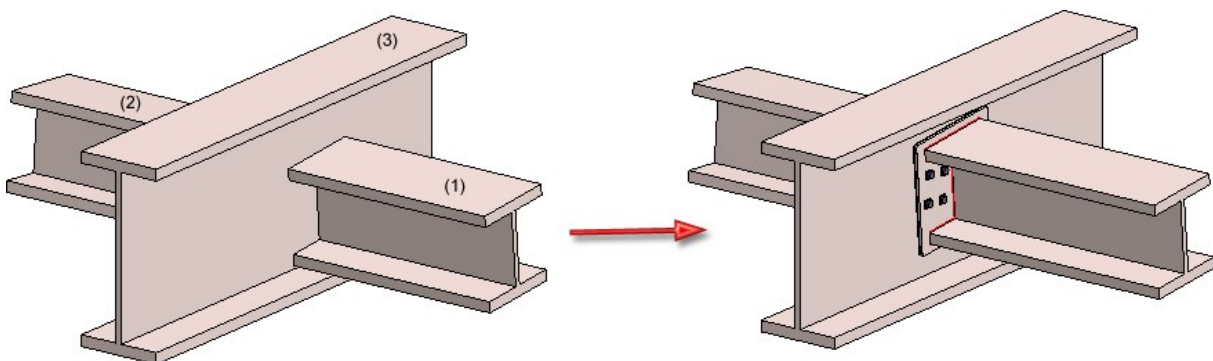
## Front plate connection (double-sided) to web (2322)

Use this function to connect two beams via front plate connections to a third beam. The connection can be made to the web or to the flange - with or without filler plates.

Depending on the fitting situation, the connected beam will be automatically notched appropriately. Furthermore, weld seams can also be directly inserted and annotated.

The connection can be a user-defined connection or configured via predefined DAST tables supplied with HiCAD.

Supported beam and profile types are I-beams, U-beams, T-beams, Z-profiles, pipes and hollow profiles.



(1) 1st beam to be connected, (2) 2nd beam to be connected, (3) Target beam to which the connection is to be made

## Workshop drawing

### Sheet Metal

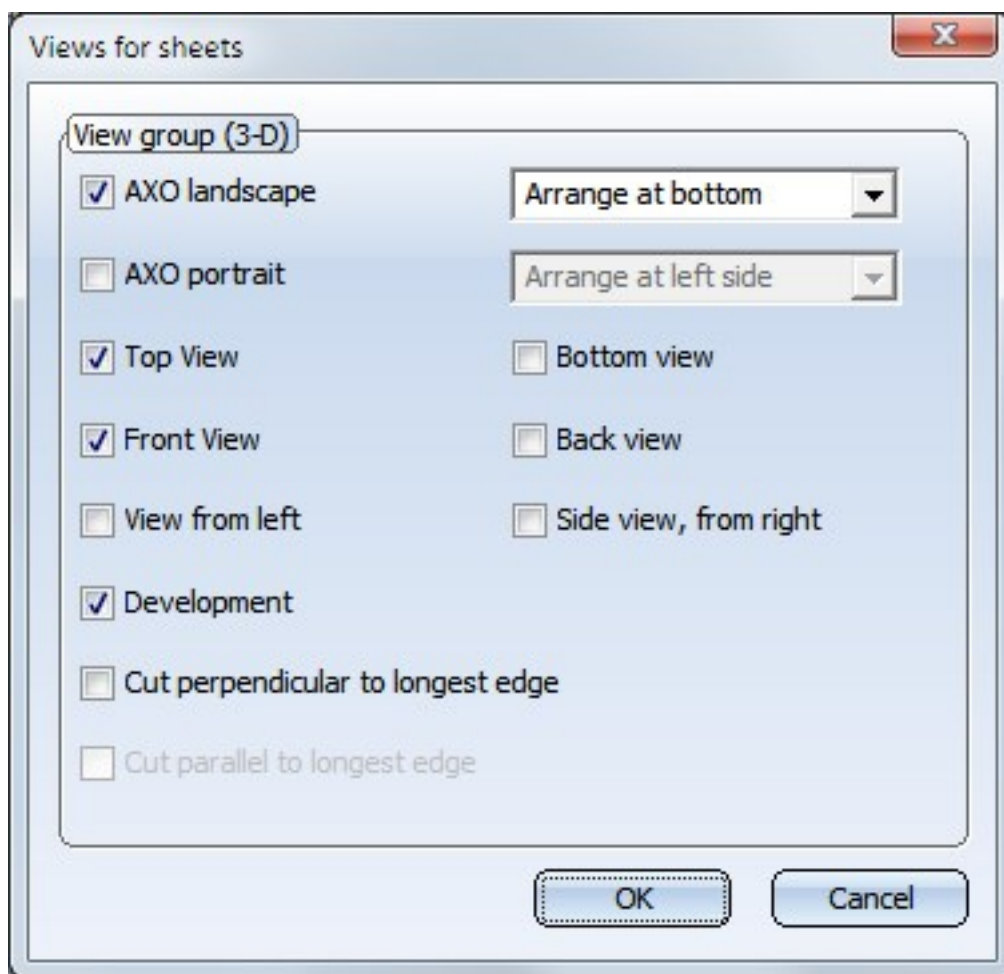
The following sectional views can now be selected for Sheet Metal parts in the **Views for sheets** dialogue window:

- Cut perpendicular to longest edge

This sectional view is only created if a bend zone exists.

- Cut parallel to longest edge

This sectional view is only created if a bend zone running perpendicular to the section path exists.



### Hide boltings

Bolts in workshop drawings can be hidden if desired. This is currently only possible via the Windows\_Registry. To do this, select

**HKEY\_CURRENT\_USER\Software\ISD Software und Systeme\HiCAD 2016**

and create a new key:





Key	Name	WorkshopDrawing	
DWORD value	Name	ModeScrewInvisibility	
	Type	DWORD	
	Value	0	No bolts are hidden. This is the default setting and corresponds to the behaviour of HiCAD before Version 2016 SP1.
		1	All bolt are hidden.
		2	Only bolts with the property "Site assembly" or "Site production" are hidden.
		<0 or >2	The default value 0 is used.



These settings do not apply to bolts and screws that are located in a structure assembly (e.g. to Boltings inserted as "Loose parts"). These bolts and screws are not shown in the workshop drawing.

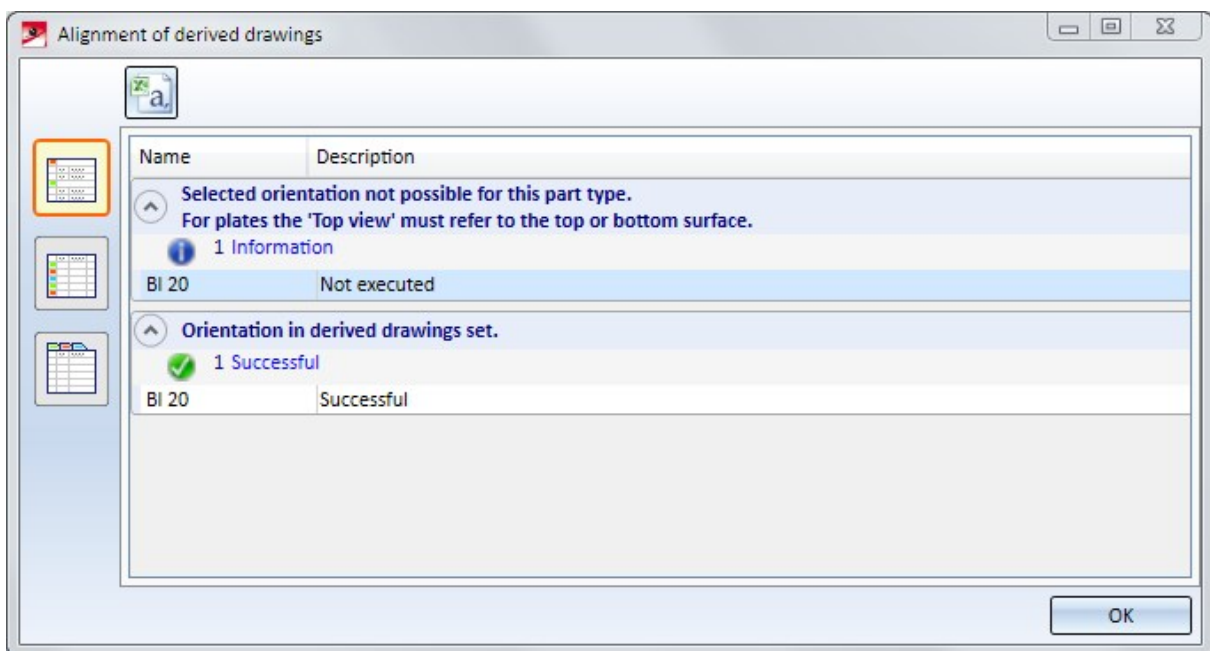
## Aligning of Steel Engineering plates

The functions beneath Alignment of derived drawings in the context menu for parts:

- Active view as front view  and
- Active view as top view 

can now also be applied to Steel Engineering plates. This allows you to specify the alignment of plates in Workshop drawings. HiCAD will check whether the new alignment of the plate is actually allowed. The top view must be perpendicular to the upper or lower plate side!

The result of the check will then be shown in a dialogue window. Here it will be listed for which of the chosen plates the new alignment was successful, and to which no new alignment could be applied.





## New/changed dimensioning rules

### Separate dimensioning of beam sub-parts which are not vertical to main part axis

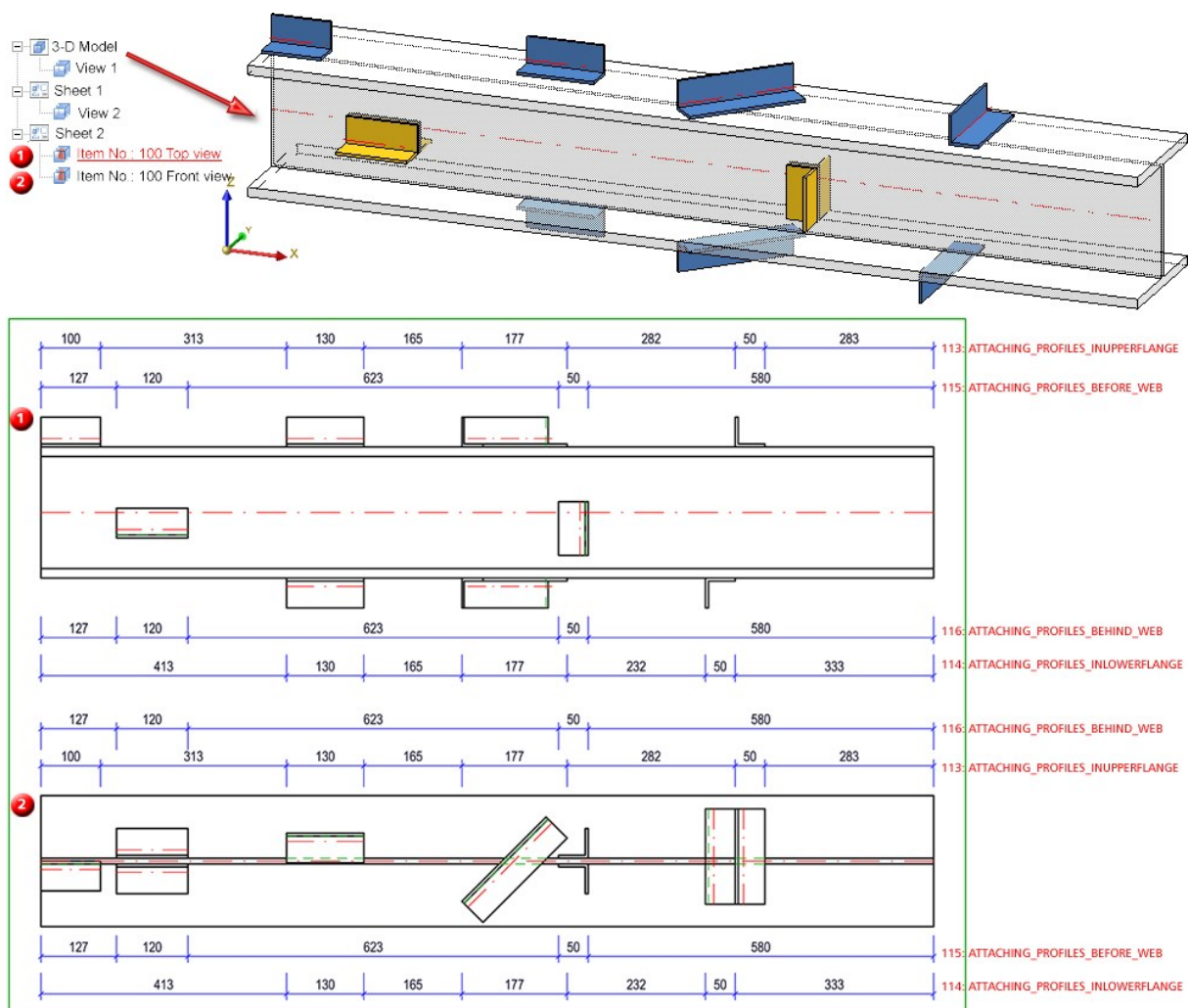
The following new rule allow a dimensioning of beam sub-parts that are not vertical to the main part axis.

Rule		Internal designation
113	Beam sub-parts, not vertical to main part axis	ATTACHING_PROFILES_INUPPERFLANGE
114	Beam sub-parts on lower flange, not vertical to main part axis	ATTACHING_PROFILES_INLOWERFLANGE
115	Beam sub-parts in front of web, not vertical to main part axis	ATTACHING_PROFILES_BEFORE_WEB
116	Beam sub-parts behind web, not vertical to main part axis	ATTACHING_PROFILES_BEHIND_WEB
117	Beam sub-parts, not vertical to main part axis	ATTACHING_PROFILES

If you select the direction **Parallel to beam axis** you can, depending on the respective position, create a chain of dimensions for the beam sub-parts.

If you select the direction **Vertical to beam axis**, a separate chain of dimensions will be created - vertical to the main part axis - for each beam sub-part.

Example 1 - 113 -116 parallel to beam axis



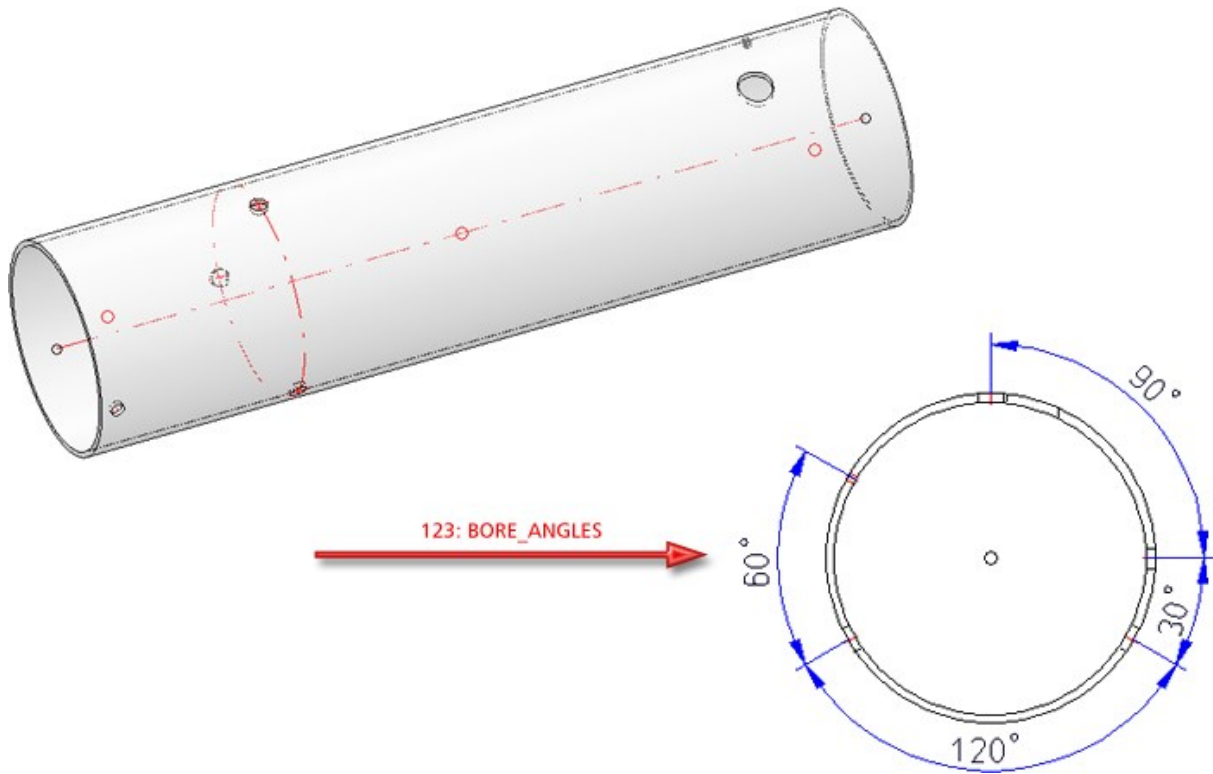
## Dimensioning of bores with intersecting axes

Use the new dimensioning rule

### 123: Angle between bore axes (BORE\_ANGLES)

to dimension bores with intersecting axes - this rule is of particular importance for machine-controlled drilling. This rule dimensions bores in beams and profiles with circular cross-sections (round steels, steel pipes), but only if the bore axes are running in the direction of the beam/profile axis.

*Example - Side view, from left*



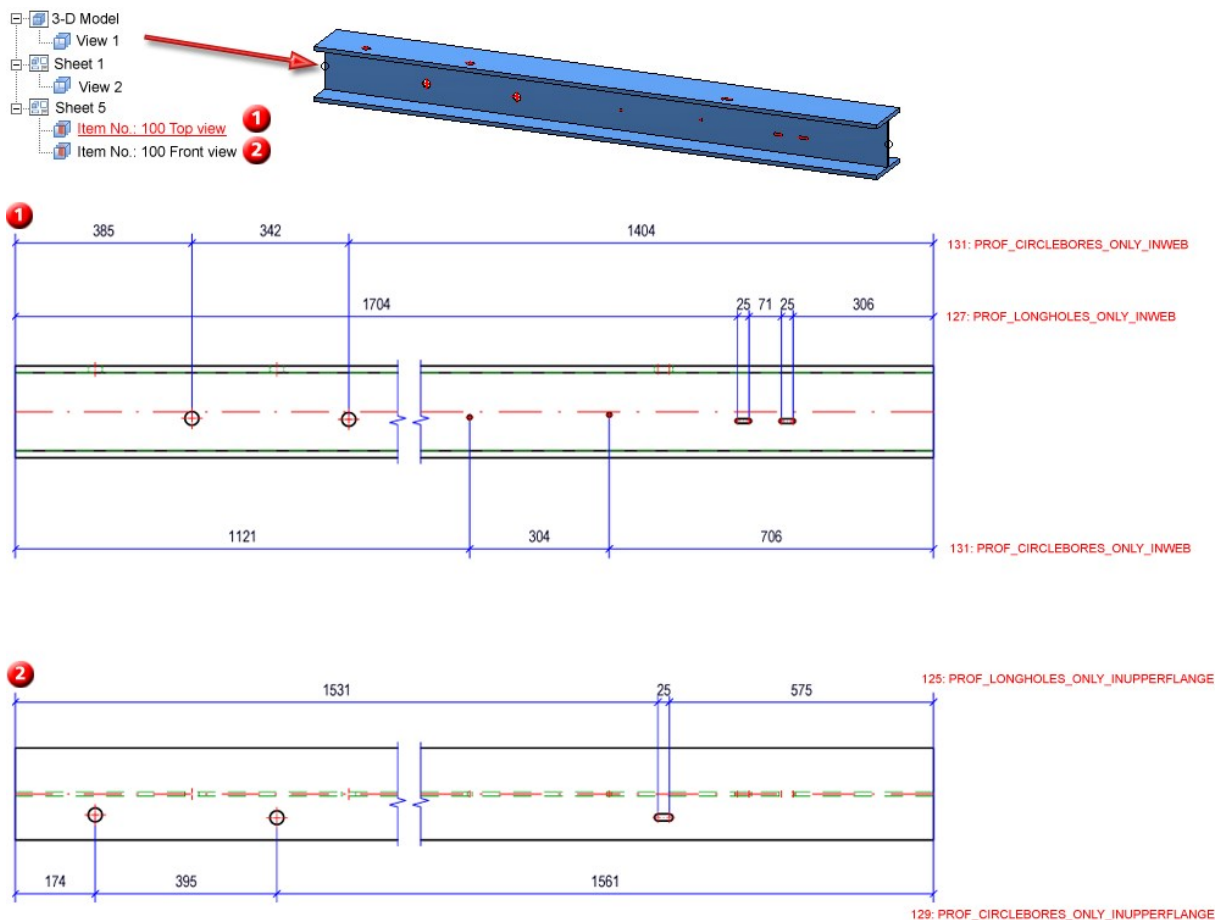
### Dimensioning rules for various bore types

When creating production drawings, different dimensioning rules are now available for different bore types:

Rule		Internal designation
124	Beam processings of the type Slot	PROF_LONGHOLES_ONLY
125	Beam processings of the type Slot in upper flange	PROF_LONGHOLES_ONLY_INUPPERFLANGE
126	Beam processings of the type Slot in lower flange	PROF_LONGHOLES_ONLY_INLOWERFLANGE
127	Beam processings of the type Slot in web	PROF_LONGHOLES_ONLY_INWEB
128	Beam processings of the type Circular bore	PROF_CIRCLEBORES_ONLY
129	Beam processings of the type Circular bore in upper flange	PROF_CIRCLEBORES_ONLY_INUPPERFLANGE
130	Beam processings of the type Circular bore in lower flange	PROF_CIRCLEBORES_ONLY_INLOWERFLANGE
131	Beam processings of the type Circular bore in web	PROF_CIRCLEBORES_ONLY_INWEB

Only bores with bore axis in viewing direction will be considered. For circular bores, a separate chain of dimensions will be created for each bore diameter;

*Example*



## Dimensioning rules for stairs

For staircases you can now use the following dimensioning rules:

	Regel		Interne Bezeichnung
Total dimensions, axially parallel	2	Assembly length	ASSEMBLYLENGTH
	19	Assembly height	ASSEMBLY_HEIGHT
	20	Assembly width	ASSEMBLY_WIDTH
Transport dimensions	90	Diagonal total dimensions of an assembly	TOTAL_SIZE_ASSEMBLY
Angles between stringers	111	Angle between beam main part and connected sub-part beams	ANGLE_CONNECTED_PROFILES(for the main part)
	112	Angle between relevant beams and connected sub-part beams	ANGLE_CONNECTED_PROFILES_RELPROFS (for the relevant sub-parts)
Sub-parts	94	All attached parts of relevant beam sub-parts, via outer contour	ATTACHING_PARTS_RELVPROFS
	96	Attached Sheet Metal parts with bores on relevant beam sub-parts	ATTACHING_SHEETS_RELPROFS
	110	Beam sub-parts on relevant beam sub-parts	ATTACHING_PROFILES_VERT_RELVPROFS (for the relevant sub-parts)
Bores	98	Bores in web of relevant beam sub-parts	PROFBORES_INWEB_RELVPROFS
	99	Bores in upper flange of relevant beam sub-parts	PROFBORES_INUPPERFLANGE_RELVPROFS
	100	Bores in lower flanges of relevant beam sub-parts	PROFBORES_INLOWERFLANGE_RELVPROFS
	101	Bores in flanges of relevant beam sub-parts	PROFBORES_INFLANGES_RELVPROFS

## Other new features and enhancements

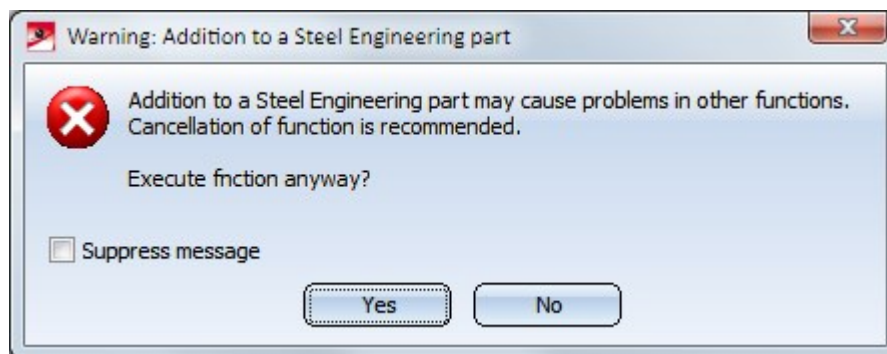
### Adding of 3-D parts to Steel Engineering parts

The adding of 3-D parts to a Steel Engineering parts may cause problems with subsequently called functions, e.g. mitre cuts, Steel Engineering connections, NC/NCX export, deriving of drawings, etc. Attributes (e.g. weight, length, etc.) may also be calculated incorrectly.

In such cases, HiCAD will display the following warning message after selecting 3-D-Standard > Process > Add



function :



You can then decide whether you want to execute the function or not.

If you do no longer want this message to be displayed, activate the **Suppress message** checkbox. In this case, the adding will either be accepted or refused, depending on your last choice in the message window.

Please remember that the checkbox status applies only to the current HiCAD session!

## Major Release 2016 (V 2100)

### Please note:

When creating boltings via Design Variants as accessory parts (Staircase/Railing Configurator) or loose parts (Connections 2320, 2322, 2330, 1305, 1306, 1211) in a non-BOM-relevant assembly, no DSTV BOM can be written, because the superordinate part is missing. This can be avoided for the mentioned connections by activating the Assignment: Assembly option on the Boltings tab of the corresponding dialogue window. Bolting structures created by the Staircase and Railing Configurator must currently be adjusted manually.

## Mounting planning - Basic functionality



### Create mounting drawing

You can now create, modify and expand Steel Engineering Mounting drawings in only a few steps.

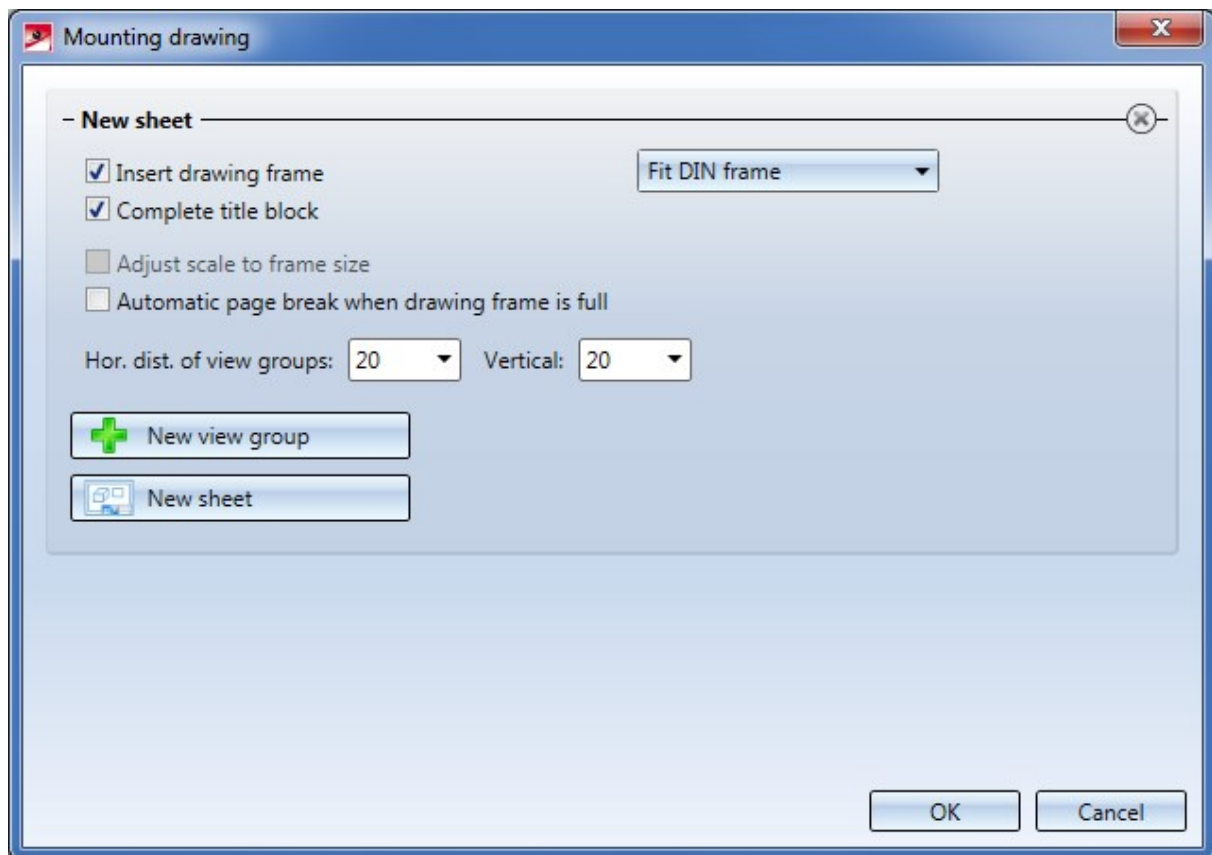
These mounting drawings have the following properties:

- The drawings show a section through the drawing. The drawing views are cuboid detail views, the size of these cuboids can be individually adjusted in the dialogue.
- The essential parts can be automatically annotated.
- Grids can also be automatically annotated.
- The drawings contain one or several views which are arranged in view groups.
- The drawings are saved in the drawing, i.e. HiCAD will create corresponding Sheet areas

For the creation, changing and updating of mounting drawings the functions

- Mounting drawing  and
- Update mounting drawing 

are now available at **Drawing > Itemisation/Detailing**.

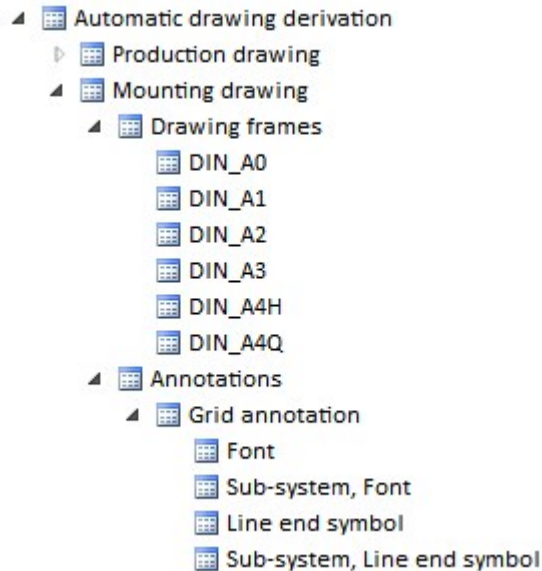
*Mounting drawing dialogue window*

Please note that the mounting planning functionality supplied with the Major Release HiCAD 2016 (V. 2100.0) is only a basic functionality, which will be continually enhanced in future versions.



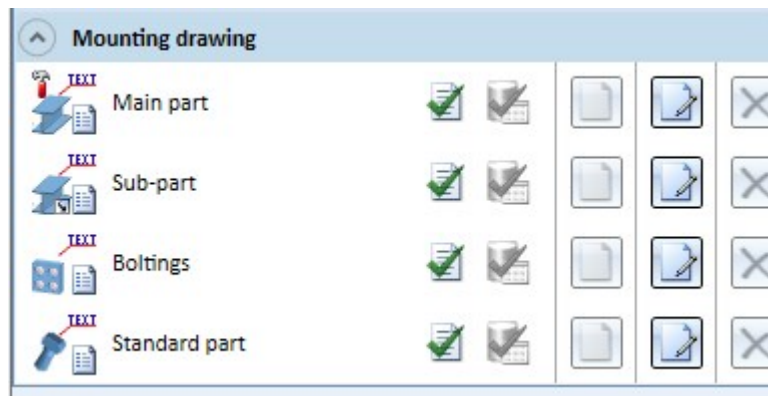
## Settings for mounting drawings in the Configuration Editor

As with production drawings, you can specify various settings for mounting drawings in the Configuration Editor. You can find these settings at **er Automatic drawing derivation > Mounting drawings**:



## Additional settings in the 'Part annotation templates'\* function

The Part annotation templates function has been expanded for mounting drawings. There, you can now specify the representation of annotations for main parts, sub-parts, standard parts and boltings:



Standard template files for mounting drawings	Annotation settings for
POSNUMMER_MTZ_HAUPTTEIL	Main parts in assembly drawings
POSNUMMER_MTZ_NEBENTEIL	Sub-parts in mounting drawings
POSNUMMER_MTZ_NORMTEIL	Standard parts in mounting drawings
POSNUMMER_MTZ_VERSCHRAUBUNG	Boltings in mounting drawings

## Connections

### Front plate connection - New functions

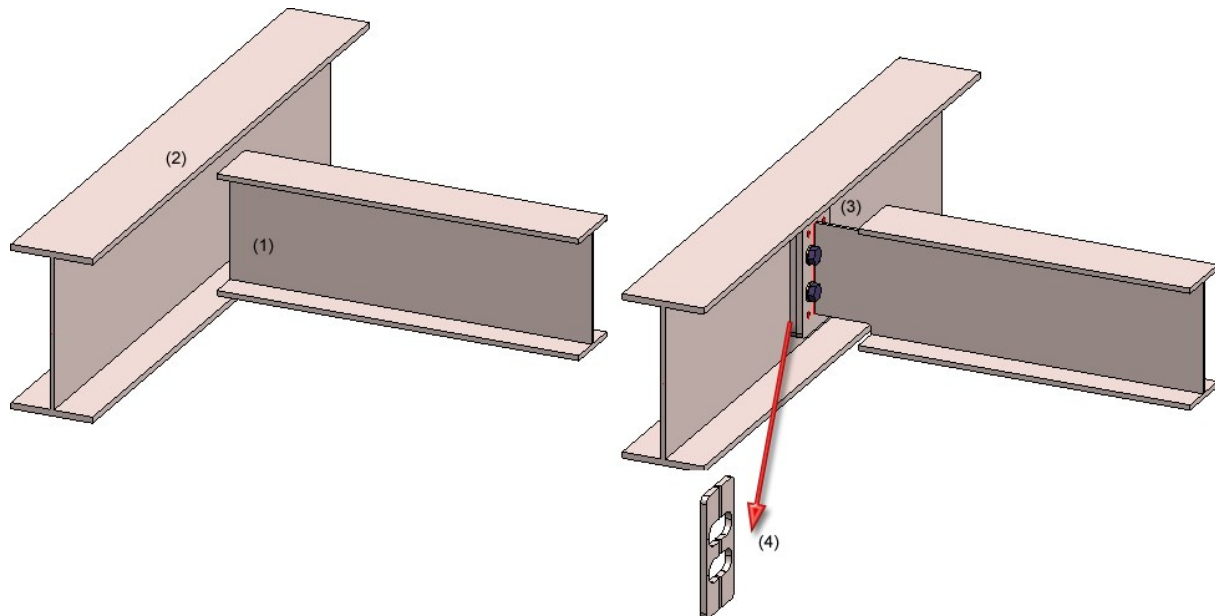
New front plate connections are available, replacing the previous connections 1320 and 1322 and providing significantly more flexibility during insertion.

#### Front plate connection to web/flange (2320)

Use this function to connect 2 beams via a bolted front plate connection. The connection can be made to the web or to the flange - with or without filler plates. Depending on the fitting situation, the connected beam will be automatically notched appropriately. Furthermore, weld seams can also be directly inserted and annotated.

The connection can be a user-defined connection or configured via predefined DAST tables supplied with HiCAD.

Supported beam and profile types are I-beams, U-beams, T-beams, Z-profiles, pipes and hollow profiles.



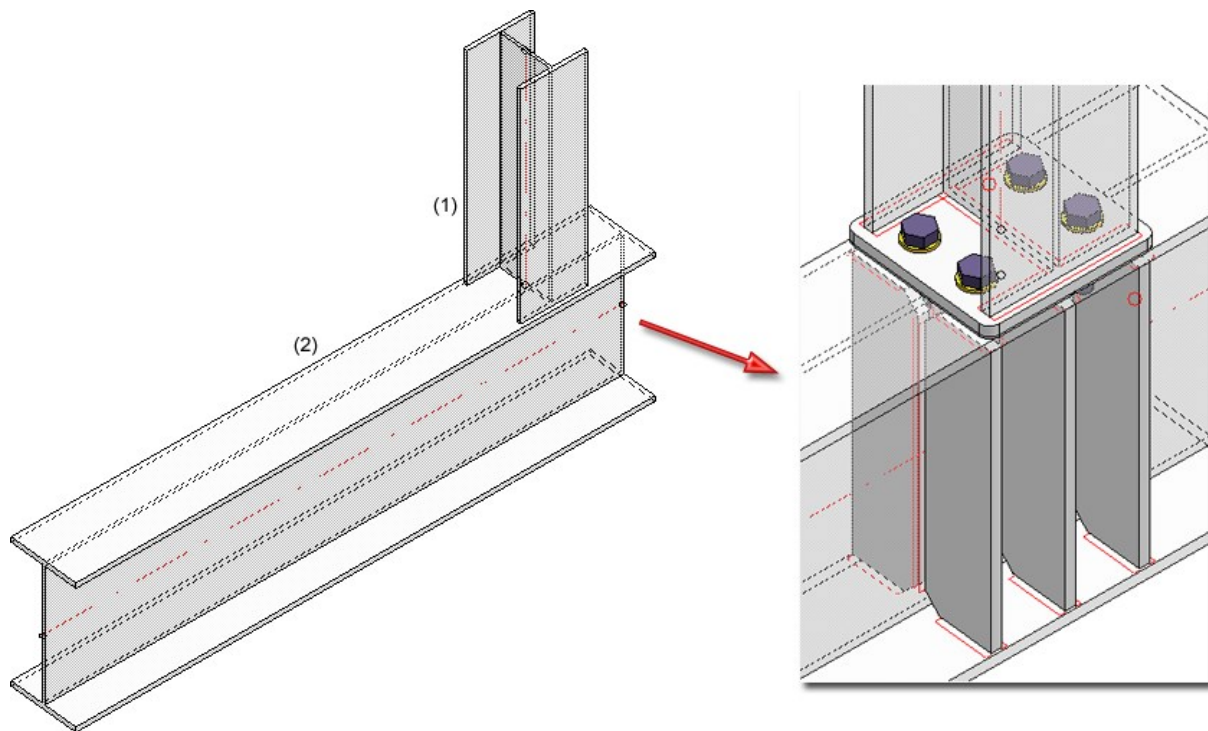
(1) Beam to be connected, (2) Target beam to which the connection is to be made, (3) Created connection, (4) With filler plates

#### Front plate connection to flange (2330)

Use this function to connect two beams via a bolted front plate connection. The connection can be made to the web or to the flange - with or without filler plates. In contrast to the connection 2320 the connected beam will not be automatically notched. Furthermore, you have the option to insert stiffeners between the flanges of the beam to which the connection is to be made.

The connection can be a user-defined connection or configured via predefined DAST tables supplied with HiCAD.

Supported beam and profile types are I-beams, U-beams, T-beams, Z-profiles, pipes and hollow profiles.



(1) (1) Beam to be connected, (2) Target beam to which the connection is to be made, (3) Created connection with stiffeners

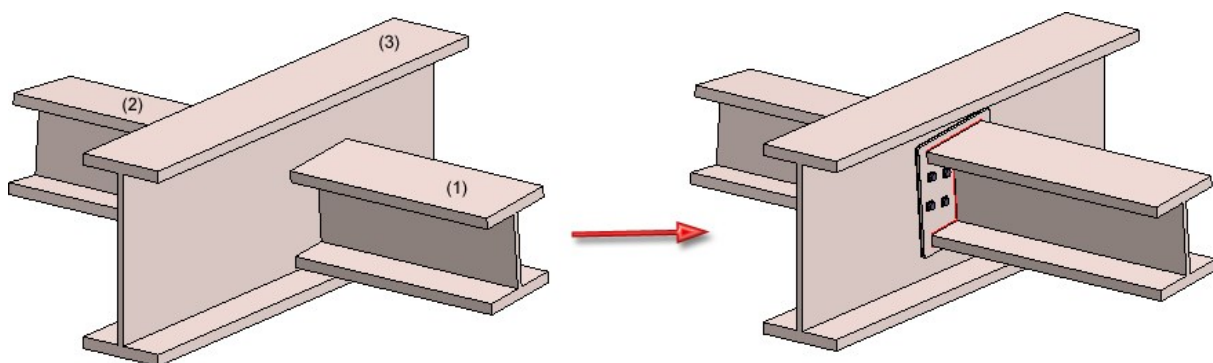
## Front plate connection to web, double-sided (2322)

Use this function to connect two beams via front plate connections to a third beam. The connection can be made to the web or to the flange - with or without filler plates.

Depending on the fitting situation, the connected beam will be automatically notched appropriately. Furthermore, weld seams can also be directly inserted and annotated.

The connection can be a user-defined connection or configured via predefined DAST tables supplied with HiCAD.

Supported beam and profile types are I-beams, U-beams, T-beams, Z-profiles, pipes and hollow profiles.



(1) 1st beam to be connected, (2) 2nd beam to be connected, (3) Target beam to which the connection is to be made

## Beam to web, with 2 plates and stiffener (1211)

This function now offers the option to define whether the bolting is to be assigned to the assembly of the 1st beam or if it is to be inserted as a "loose part". The **Bolting** tab has been expanded for this purpose.

**- Boltings**

DIN 7990-M12-4.6 / M12 (Ø13)

☒ Fit  
☐ Invert

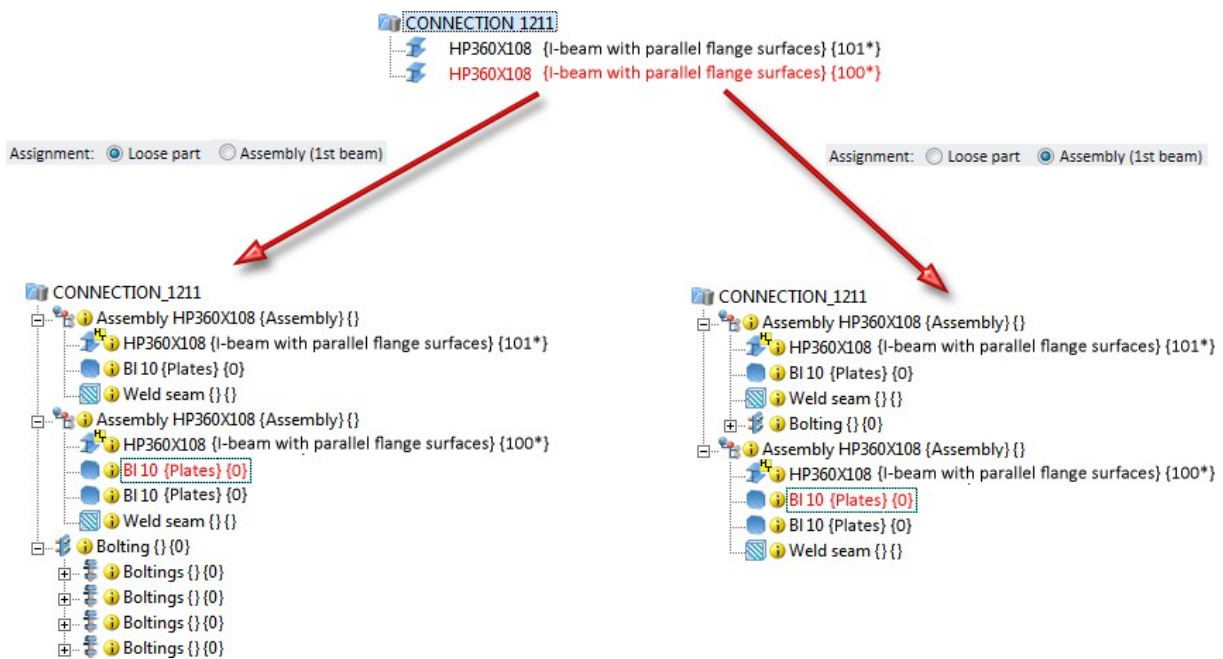
Assignment: ☒ Loose part ☐ Assembly (1st beam)

**- Usage (Assembly)**

1st beam: Girder (beam) assembly

2nd beam: Girder (beam) assembly

### Bolting assignment



## Angle connection, one sided (1305) and double-sided (1306)

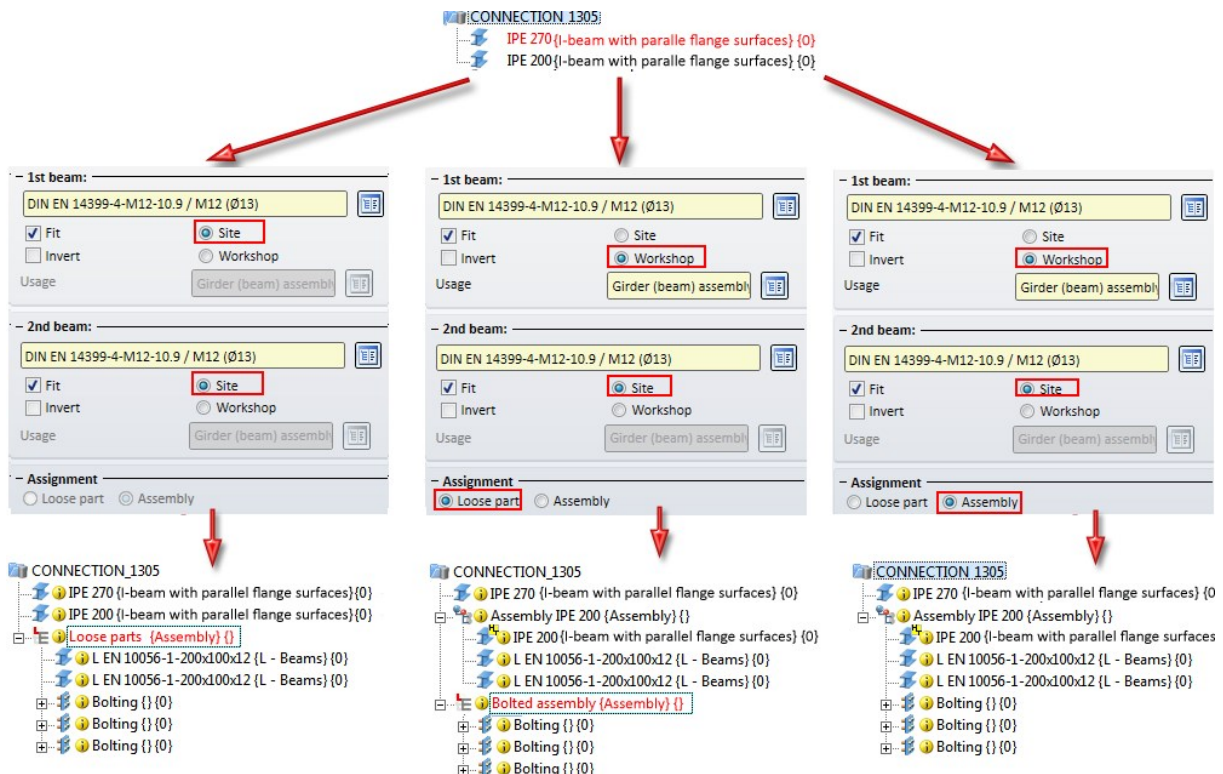
The assigning of boltings has changed here. The **Bolting** tab has been expanded for this purpose.

The screenshot shows the configuration panel for an angle connection. It is divided into three main sections:

- 1st beam:** Contains a dropdown menu for the beam type (DIN EN 14399-4-M12-10.9 / M12 (Ø13)), a 'Fit' checkbox (checked), an 'Invert' checkbox (unchecked), and a 'Usage' dropdown (Girder (beam) assembl). The 'Site' radio button is selected, and a red arrow points to it.
- 2nd beam:** Similar to the 1st beam section, with the 'Site' radio button selected and a red arrow pointing to it.
- Assignment:** Contains two radio buttons: 'Loose part' (selected) and 'Assembly' (unchecked). A red arrow points to the 'Loose part' button.

- If you have selected the production type **Workshop** for one of the beams, you can specify beneath **Assignment** by activating the corresponding radio button whether the boltings are to be assigned to the assembly of this beam, or if they are to be inserted as a structure assembly called "Loose parts".
- If you have selected the production type **Site**, all parts of the connection will be combined into one structure assembly called "Loose parts".

### Example - Assignment of boltings

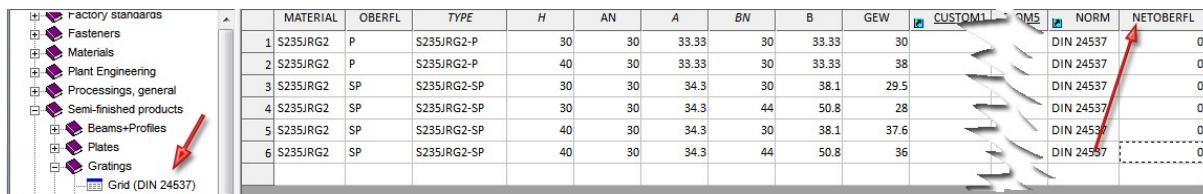




## Gratings

### Calculation of net weight and net surface area

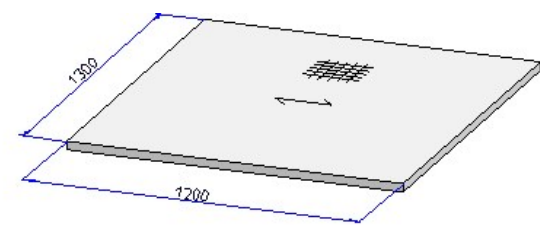
Previously, gratings were handled like plates with regard to the calculation of their weights and surface areas. It is now possible to specify the surface area per square metres of installation area for **Gratings according to DIN 24537**. For this purpose, the **NETOBERFL** column is available in the corresponding table of the Catalogue Editor.



	MATERIAL	OBERFL	TYPE	H	AN	A	BN	B	GEW	CUSTOM1	NORM	NETOBERFL
1	S235JRG2	P	S235JRG2-P	30	30	33.33	30	33.33	30		DIN 24537	0
2	S235JRG2	P	S235JRG2-P	40	30	33.33	30	33.33	38		DIN 24537	0
3	S235JRG2	SP	S235JRG2-SP	30	30	34.3	30	38.1	29.5		DIN 24537	0
4	S235JRG2	SP	S235JRG2-SP	30	30	34.3	44	50.8	28		DIN 24537	0
5	S235JRG2	SP	S235JRG2-SP	40	30	34.3	30	38.1	37.6		DIN 24537	0
6	S235JRG2	SP	S235JRG2-SP	40	30	34.3	44	50.8	36		DIN 24537	0

The default setting is the value 0. For the calculation of the net weight and the net surface area, enter the desired value here and save the table.

When inserting gratings, the value specified in the table will be considered for the calculation of the net surface area. In addition, subtractions and chamfers are also taken into account when weights and surface areas are calculated.

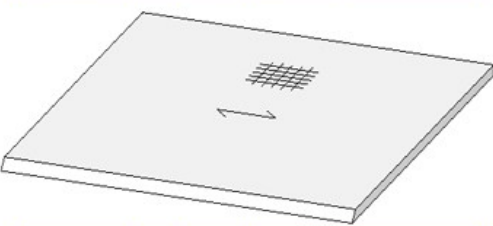


(1) **NETOBERFL**

								0.0		46.8
H	100	1	Gitterrost DIN 24537-30-30-P	1300.0	1200.0	S235JR		0.0	46.8	46.8
DSTV-Teiletyp	Pos.	Menge	Bezeichnung	B[mm]	L[mm]	Material	Benennung	Fl[m²]	Gew[kg]	Ges.gew.

(2) **NETOBERFL**

								8.6		46.8
H	100	1	Gitterrost DIN 24537-30-30-P	1300.0	1200.0	S235JR		8.6	46.8	46.8
DSTV-Teiletyp	Pos.	Menge	Bezeichnung	B[mm]	L[mm]	Material	Benennung	Fl[m²]	Gew[kg]	Ges.gew.



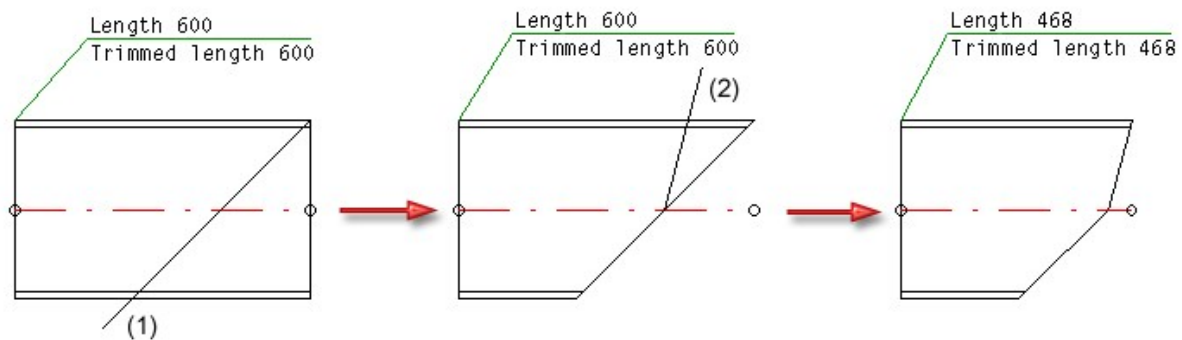
								8.4		46.1
H	-100	1	Gitterrost DIN 24537-30-30-P	1280.0	1200.0	S235JR		8.4	46.1	46.1
DSTV-Teiletyp	Pos.	Menge	Bezeichnung	B[mm]	L[mm]	Material	Benennung	Fl[m²]	Gew[kg]	Ges.gew.

(1) Grating insertion with value 0, (2) Grating insertion with value 5.5, (3) Grating with chamfer

## Cut beams

### New part attribute: Trimmed length

When beams are trimmed, the value assigned to the part attribute **Length** will be the total length of the trimmed beam. If two trimmings are applied to a beam, the calculated length will be the smaller of the two individual total lengths after applying the trimming. The beam length and the trimmed length will then be identical.

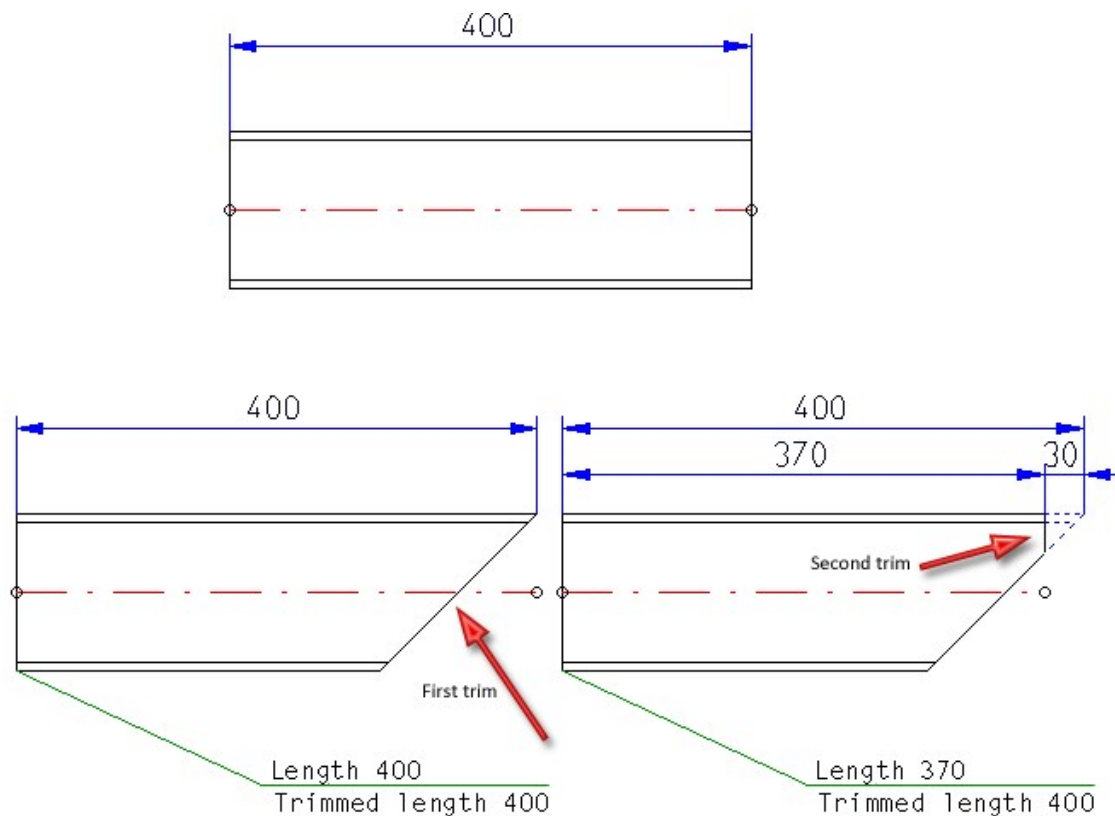


Line for 1st trimming; (2) Line for 2nd trimming

Exceptions are cases where one of the trimmings is performed at a 90° angle. In such cases, the length resulting from the 90° trimming will not be considered for the calculation of the minimum (the reason for this are processing steps with a saw). Use the part attribute

### Trimmed length (§24)

for a correct calculation of the trimmed length. This attribute can, for instance, be used for annotations, in part attribute masks or in BOMs.





## BOMs in Excel

### Use of residual lengths for bar optimisation

For bar optimisations ("bars"=bar-shaped symbolic representations of beams) you have now the option to shorten a particular number of bars by specifying the number of residual lengths. This can be done with waste minimisation or with a preference of residual pieces.



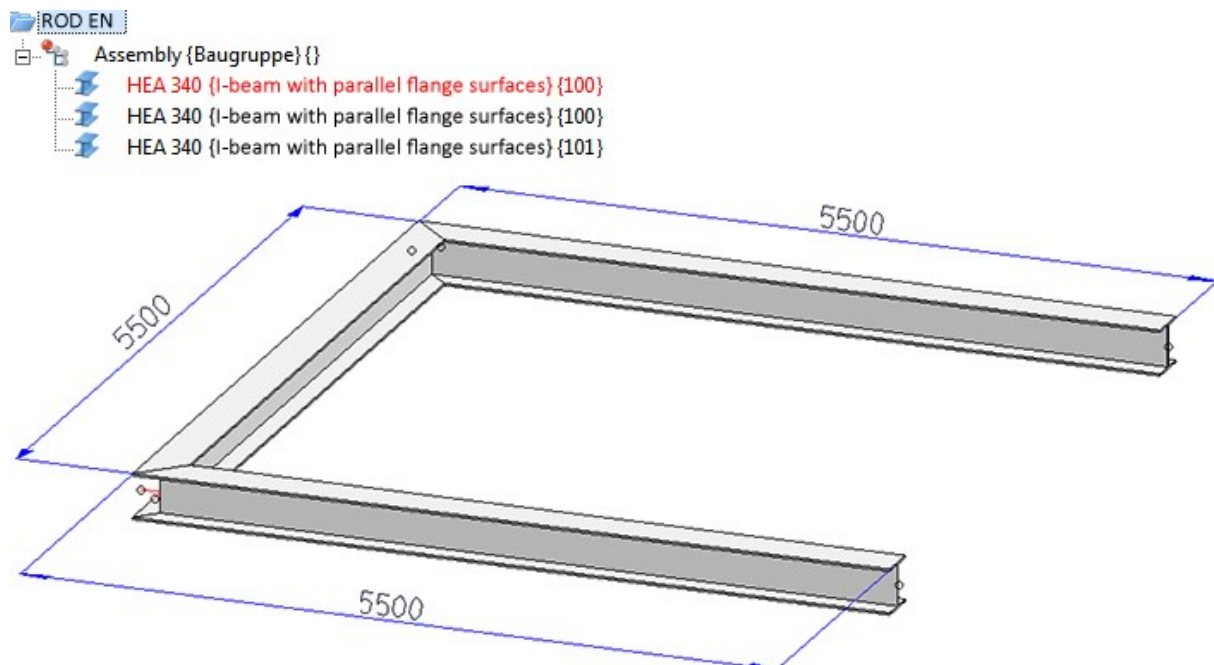
The specification of the bars and the number of residual pieces must be entered manually into the Excel template before creating the BOM.

For this purpose, the **Bar settings** sheet has been added in the **HiCAD\_Stahlbau.xlsx** template for the output of BOMs to Excel.

- New is the **Qty.** column. Here you can enter the number of residual pieces. During the placement, a semi-finished products/lengths combination with quantity (=Residual pieces > 0) will be used, together with an unlimited number of bars.
- At the bottom of the **Bar settings** sheet you will find the **Prefer residual lengths** switch. The default setting is **false**. In this case the bars will be used in such a way that the amount of waste is as small as possible. If the switch is set to **true**, the bars with a quantity of > 0 will be preferably used. The amount of waste, however, can be greater here.

### An example:

The drawing shown below consists of three beams of the type HEA 340 and a length of 5500 each.



Profiles with a length of 6000 and 6500 are to be used. When using the length of 6500, however, not more than one residual piece is to be created. For this to happen, the default settings on the **Bar settings** sheet in the file **HiCAD\_Stahlbau.xlsx** must be changed as follows:

	A	B	C	D	E	F	G	H	I
1	Beam	Material	Length	Cutting width	Cutting allowance	Distance, Start	Distance, End	Use	Qty.
2			6000	4,5	0	10	10	1	
3	HEA 340	S235JR	6000	4,5	0	10	10	1	
4	HEA 340	S235JR	6500	4,5	0	10	10	1	1

Prefer residual lengths

true

The amount of waste is to be as small as possible; therefore, the **Prefer residual lengths** switch must remain on **false**.

If you generate the Excel file with this setting, three bars with a length of 6000 mm will be used. The amount of waste will be 1500 mm here.

Bar list						
Drawing No.				Customer		
Order No.				Created by		
Order text				Created		
Designation						
HEA 340, S235JR			3 x 6000 mm		Waste: 1500 mm (8.33 %)	
Item	Qty.	Length (mm)	Cut (Web)	Cut (Flange)	Designation	
2x						
				100 (5500)		476
100	2	5500		45°		
				101 (5500)		476
101	1	5500	45°	45°		

In contrast, if you set the **Prefer residual lengths** switch to **true**, two 6000 mm bars and one 6500 mm bar (because of Qty. = 1). Here, the amount of waste will be 2 x 1000 mm.

	A	B	C	D	E	F	G	H	I
1	Beam	Material	Length	Cutting width	Cutting allowance	Distance, Start	Distance, End	Use	Qty.
2			6000	4,5	0	10	10	1	
3	HEA 340	S235JR	6000	4,5	0	10	10	1	
4	HEA 340	S235JR	6500	4,5	0	10	10	1	1

Prefer residual lengths

true

Bar list						
Drawing No.				Customer		
Order No.				Created by		
Order text				Created		
Designation						
HEA 340, S235JR			2 x 6000 mm		Waste: 1000 mm (8.33 %)	
Item	Qty.	Length (mm)	Cut (Web)	Cut (Flange)	Designation	
				100 (5500)		476
100	1	5500		45°		
				101 (5500)		476
101	1	5500	45°	45°		
HEA 340, S235JR			1 x 6500 mm		Waste: 1000 mm (15.38 %)	
Item	Qty.	Length (mm)	Cut (Web)	Cut (Flange)	Designation	
				100 (5500)		976
100	1	5500		45°		

If the quantity of the 6500 mm bars was 2, one 6000 mm bar and two 6500 mm bars would be used etc.

Further information on the output of BOMs can be found here.

## Staircase configurator

### Stringers tab changed

The **Stringers** tab of the Staircase Configurator has been changed. Here, you now have different dialogues for stringers made of Sheet Metal U and Sheet Metal C.

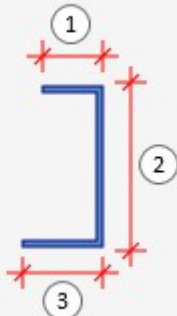
**Sheet Metal**

BI 3 - S235JR

(1) Width, top: 50

(2) Height: 250

(3) Width, bottom: 50



**Sheet Metal**

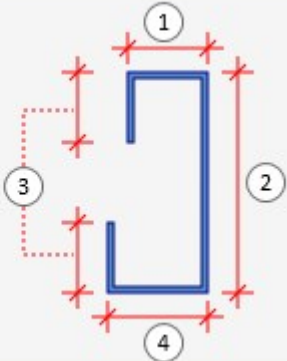
Bulb plate-T-8mm - S235JR

(1) Width, top: 100

(2) Height: 300

(3) Leg length: 50

(4) Width, bottom: 100



## Railing Configurator

### Selectable bore grid in Post - Substructure tab

The Post - Substructure tab has been expanded.

**Post**

Variants: Post connection, top

Bore grid:

(1) 20 (4) 80  
(2) 40  
(3) 75

Base plate: BI 14 - S235JR

Boltings: ☒ DIN EN 14399-3-M12-8.8

---

**Stiffeners**

☒ None ☐ One-sided ☐ Double-sided

BI 10 - S235JR, Width=Automatic

---

**Corner post (for 'Post distribution' you need to select 'Create corner post')**

Variants: Corner post connection, lateral

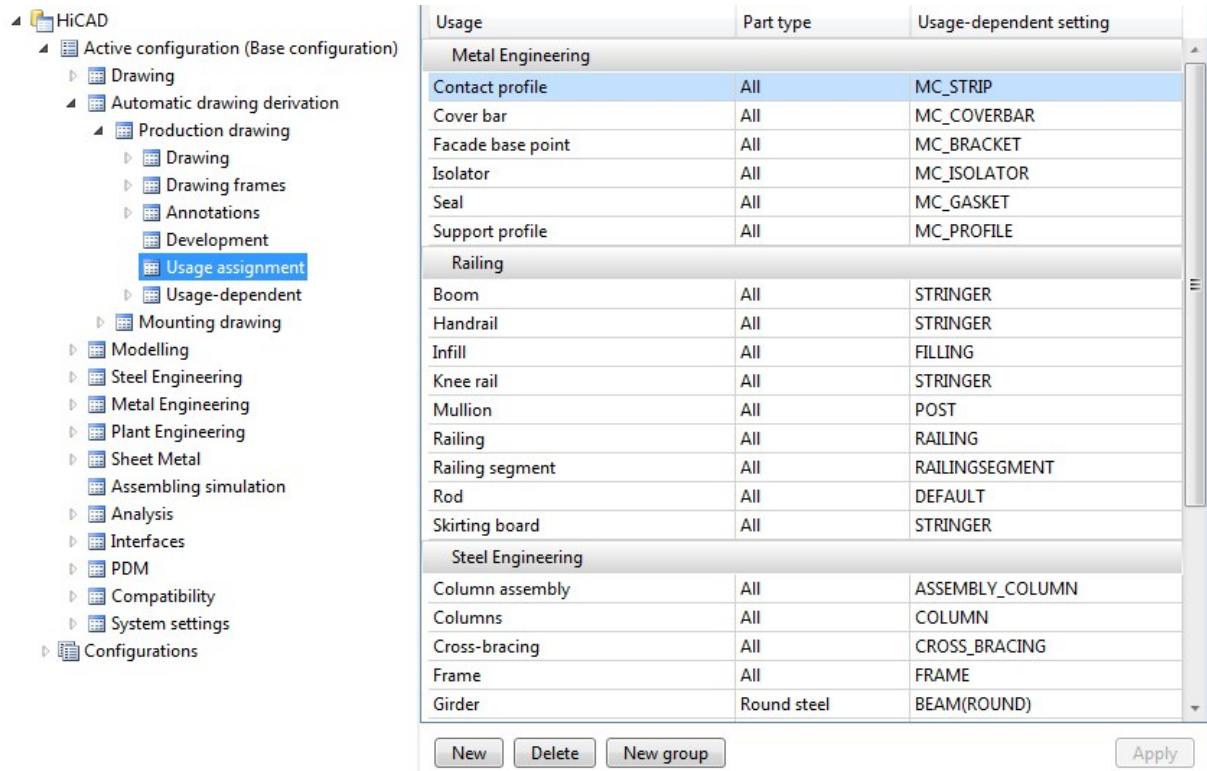
If you select the **Post connection, top** variant, you can choose a bore grid. You have now the following options:

- No bores,
- 2 bores
- 4 bores

## Derived drawings

### Usage-dependent settings

In the Configuration Editor, the assigning of usage-dependent settings has been simplified and enhanced.



Usage	Part type	Usage-dependent setting
<b>Metal Engineering</b>		
Contact profile	All	MC_STRIP
Cover bar	All	MC_COVERBAR
Facade base point	All	MC_BRACKET
Isolator	All	MC_ISOLATOR
Seal	All	MC_GASKET
Support profile	All	MC_PROFILE
<b>Railing</b>		
Boom	All	STRINGER
Handrail	All	STRINGER
Infill	All	FILLING
Knee rail	All	STRINGER
Mullion	All	POST
Railing	All	RAILING
Railing segment	All	RAILINGSEGMENT
Rod	All	DEFAULT
Skirting board	All	STRINGER
<b>Steel Engineering</b>		
Column assembly	All	ASSEMBLY_COLUMN
Columns	All	COLUMN
Cross-bracing	All	CROSS_BRACING
Frame	All	FRAME
Girder	Round steel	BEAM(ROUND)

Buttons: New, Delete, New group, Apply

- The selection of the usage row takes no longer place via a key name, but via plain text.
- The assigning can be limited to particular part types. For example, if you want to assign the usage-dependent setting STRINGER only to handrails of the type "Flat steel", select the same-named entry beneath Part type.
- The usages can be neatly arranged in groups. Predefined are the groups Railing, Metal Engineering, Steel Engineering and Others.
- The creation of new assignments, and also the definition of new usage settings can take place directly on the list of assignments.

### Shortened view

Previously, only views of beams and profiles could be shortened. This is now also possible for other views, e.g. views of Sheet Metal parts etc. For this purpose, the **Settings for views** dialogue window has been modified. There you can specify beneath **Shortened view** which views are to be shortened.



**Shortened view**

☒ Show views of beams as shortened views

Shortening factor, percental

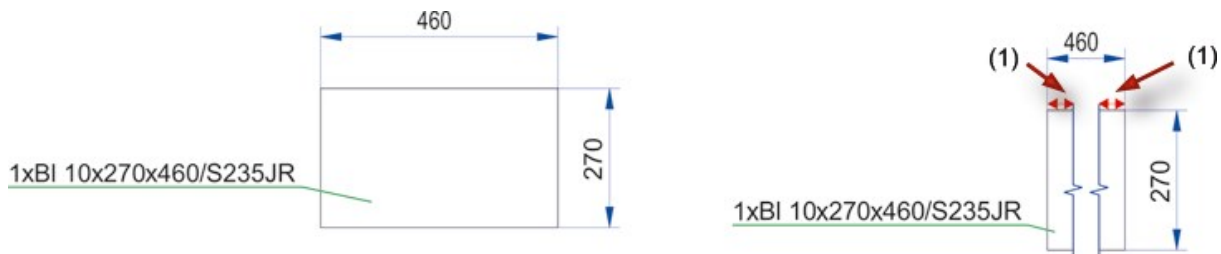
Minimum shortening length

☐ Show other views as shortened views

Minimum width of shortening area

Distance to relevant geometry

If you also want to shorten other views, e.g. views of the Sheet Metal parts, activate the **Show other views as shortened views** checkbox, enter the minimum width of the area to be removed by the shortening, and the distance of the shortening to the relevant geometry.



(1) Distance of shortening to relevant geometry

If the settings for drawing derivation are loaded from a configuration, the settings specified in the **Configuration Editor** at **Automatic drawing derivation > Production drawing > Usage-dependent > name > Views > Shortened view** will be evaluated. *name* denotes the name of the corresponding usage.

- HiCAD
  - Active configuration (Base configuration)
    - Drawing
      - Automatic drawing derivation
        - ProductionDrawing
          - Drawing
            - Drawing frames
              - Annotations
                - Development
                  - Usage assignment
                    - Usage-dependent
                      - Template
                        - Default
                          - Drawing frames
                            - Annotation
                              - View group (3-D)
                                - Views
                                  - Part environment
                                    - Shortened view**

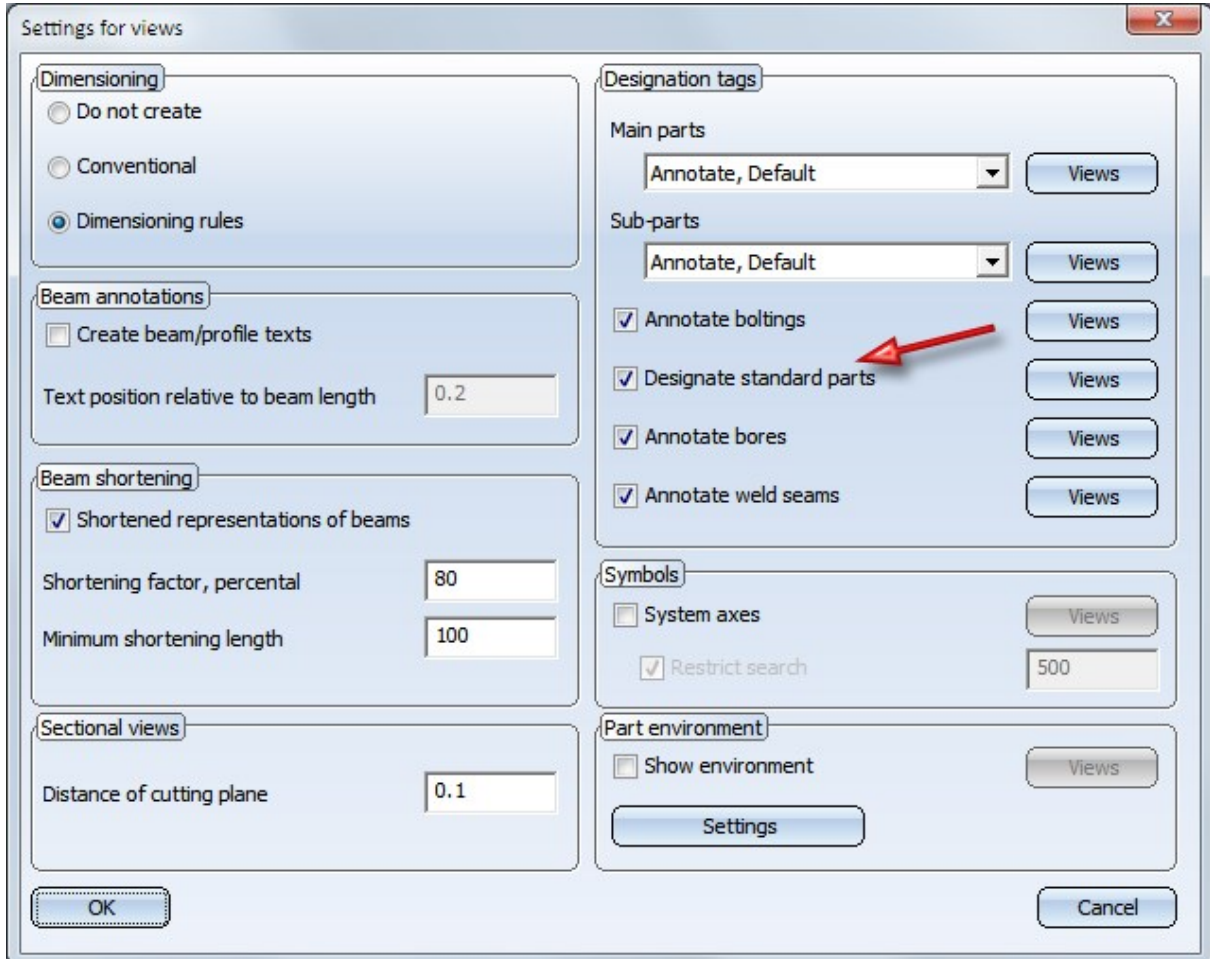
| Description                       | Value                               | Comment   |
|-----------------------------------|-------------------------------------|---|
| <b>General view shortenings</b>   |                                     |   |
| View shortenings, general         | <input type="checkbox"/>            | Shorten views                                     |
| Minimum width of a shortened area | 20                                  | Minimum width of a shortened area                 |
| Distance to relevant geometry     | 5                                   | Distance to relevant geometry                     |
| <b>View shortenings for beams</b> |                                     |   |
| Shortened view                    | <input checked="" type="checkbox"/> | Shorten views (only for beams/profiles)           |
| Minimum shortening length         | 100                                 | Minimum shortening length (reference: expression) |
| Shortening factor, percental      | 80                                  | >= 5; <= 90                                       |

## Sectional views for flat steels

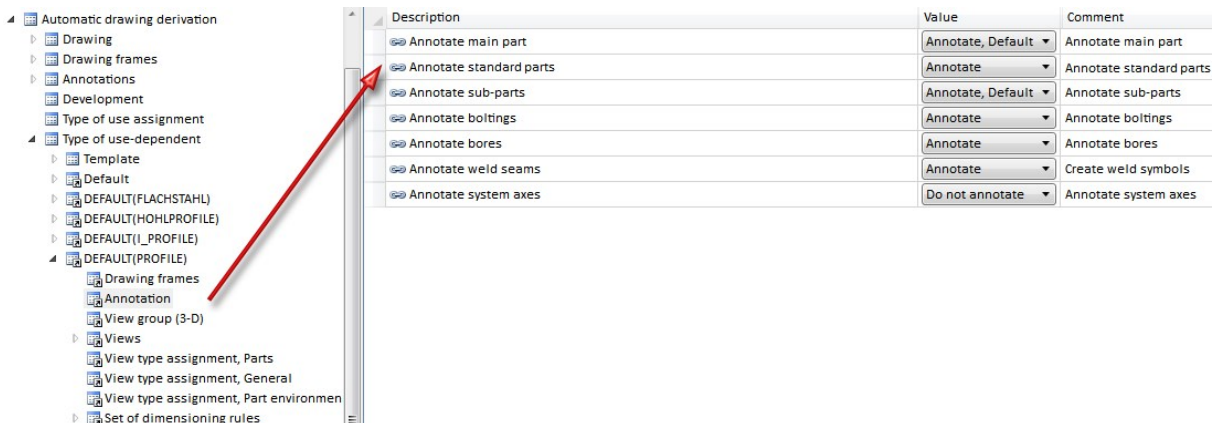
You can now also create sectional views for standard beams of the category **Flat steel** for derived drawings, if this was correspondingly specified at View groups.

## Setting: Standard part annotation

In HiCAD 2015, standard parts were annotated if you explicitly activated the annotation of sub-parts or (as of SP1) the annotation of boltings. As of HiCAD 2016 the annotation of standard parts can be switched on or off, independent of other annotation types.



If the settings for drawing derivation are loaded from a configuration, the settings specified in the **Configuration Editor** at **Automatic drawing derivation > Production drawing > Usage-dependent > name > Views > Shortened view** will be evaluated. *name* denotes the name of the corresponding usage.



| Description             | Value             | Comment                 |
|-------------------------|-------------------|-------------------------|
| Annotate main part      | Annotate, Default | Annotate main part      |
| Annotate standard parts | Annotate          | Annotate standard parts |
| Annotate sub-parts      | Annotate, Default | Annotate sub-parts      |
| Annotate boltings       | Annotate          | Annotate boltings       |
| Annotate bores          | Annotate          | Annotate bores          |
| Annotate weld seams     | Annotate          | Create weld symbols     |
| Annotate system axes    | Do not annotate   | Annotate system axes    |



## New dimensioning rules for trussed rafters

The following new dimensioning rules are available for workshop drawings:

- **90: Diagonal total dimensions of an assembly**

Use this rule to create diagonal individual dimensions (shipped dimensions) between the outer corners of an assembly.

| 90: Diagonal total dimensions of an assembly |  |
|--|--|
| ID:  | 90: Diagonal total dimensions of an assembly |
| Type of dimension:                           | Individual dimensions                        |
| Position of dimension chain:                 | Bottom                                       |
| Dimension chain reference:                   | Start/End point of complete assembly         |
| Direction of dimension chain:                | Direction determined by 2 points             |

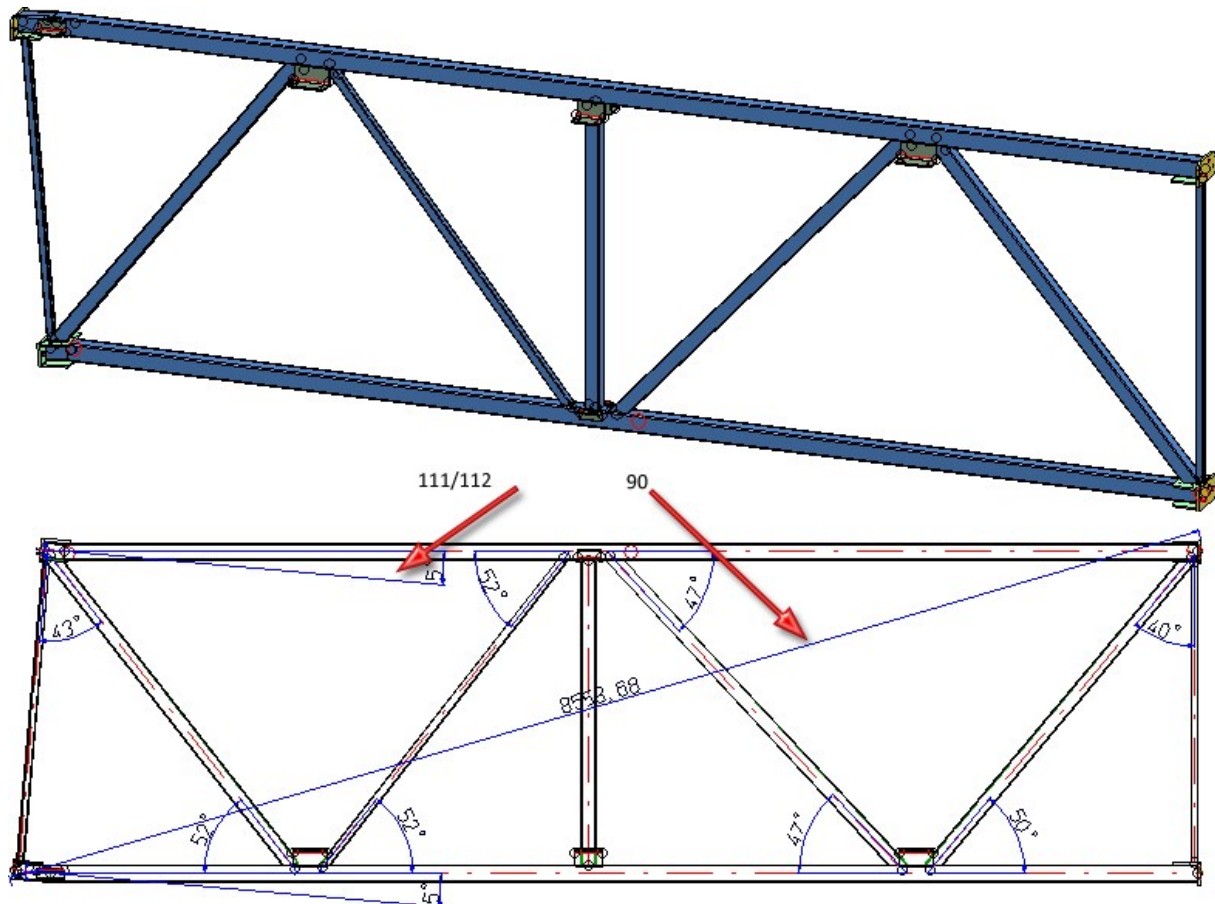
- **111: Angle between beam main part and connected sub-part beams /**

- **112: Angles between relevant beams and connected sub-part beams**

Here, angles between directly connected beams the angles to each other of which are not 90° or 180° are dimensioned. Angles which are not displayable in the corresponding view will not be created either. **Important:** The dimensioned angle is not necessarily located in the screen plane, but can be located anywhere in space. Rule 111 dimensions only angles between the beam main part and the beam sub-parts connected to it. Rule 112 dimensions all relevant beams and the beam sub-parts connected to them.

- **80: Connected sub-part beams on relevant beams**

Here, all beams which are directly connected to another beam are dimensioned in a chain. Clear distances between the beams will also be considered. The already existing Rule **77: Connected beam sub-parts** only considers the beam sub-parts which are not connected to the beam main part. The new Rule 80, in contrast, also considers all relevant beams and the beam sub-parts connected to them.



### Plate-like profiles in sectional views

Previously, sectional views were only created for Steel Engineering plates. This is now also possible for plate-like profiles. Therefore, the dimensioning rules for plates now also apply to plate-like profiles in sectional views.

### View groups - Frames, alignment, visualisation



A new listbox for frames and view groups offers the following selection options:

- Do not insert
- Always insert
- Insert, if there are multiple view groups

The default setting is **Always insert**. This presetting can be changed in the usage-dependent templates in the Configuration Editor at: **Automatic drawing derivation > Production drawing > Usage-dependent > ... > View groups > Frames around view groups**.

If a drawing frame contains only one view group, it will be centred in the drawing frame. This behaviour, too, can be changed in the Configuration Editor, at **Automatic drawing derivation > Production drawing > Drawing > Centre individual view group**. If you deactivate the checkbox, individual view groups will be placed at the top left of the drawing frame instead.

Colour, layer and line type of the frames can be specified in the Configuration Editor at **System settings > Visualisation > View group**.

| Description        | Value  | Comment                                 |
|--------------------|--|---|
| Colour of frame    | 11:  Dark Green ▼ | Colour of lines for frames              |
| Line type of frame | 1:  ▼             | Type of frame line around view groups   |
| Layer of frame     | 1  | Layer of frame lines around view groups |



If the centring leads to collisions with locked areas (e.g. the title block), the view group will be placed at the top left.

## Steel Engineering - Management + BIM

Service Pack 2 2016 (V. 2102)

### Important note on updates!

When you run the tool **Computer Parameter Configurator** (HiCAD exe\ParKonfigComp.exe) – either during installation or later – and activate the **Management + BIM** checkbox there, various system files in the HiCAD sys directory will be replaced (without any further query!) with the files supplied in the HiCAD templates directory which have been designed especially for a working with the Management + BIM module. These files include, for example

- HiCAD masks for the part attributes (\*.HDX),
- HELiOS masks for projects, documents and articles (\*.MSK),
- Various configuration files (\*.HDB) and
- Template files for annotations (\*.FTD).

**You should therefore always make backups of all files that you have customized before using the ParKonfigComp.exe tool!**


**If you have any questions, please contact your nearest ISD branch office.**

### New dialogue masks

The dialogue masks for Document and Drawing Management in Civil Engineering have been redesigned. For example, the new dialogue masks for documents and articles now contain the attribute CONSTRUCTION\_SECTION for the identification of construction units (documents and articles) and the attribute STAHLBAUAUTO for a subsequent setting of the Steel Engineering Drawing Management identifier (article).

*The new HELiOS masks for project, article and document data input*

**Enter project**


|   |  |   |
|---|--|---|
| <b>Consignment/Project</b><br>Proj. No./Cons. No. <input type="text" value="BIM_TEST"/><br>Assignment: Project-independent<br>Consignment: <input type="text"/><br>Batch: <input type="text"/><br>Project type: <input type="text"/><br>Designation: <input type="text"/> | <b>Additional attributes for execution classes</b><br>Tolerances: <input type="text"/> EXC cl.: <input type="text"/><br>Assessment gr.: <input type="text"/> Weld prep.: <input type="text"/><br>Weld suppl. 1: <input type="text"/> Shielding ga: <input type="text"/><br>Weld proc. 1: <input type="text"/> Undesignated weld seams: <input type="text"/><br>Weld suppl. 2: <input type="text"/> Grind: <input type="text"/><br>Weld proc. 2: <input type="text"/> Comment: <input type="text"/> | <br><b>Structural engineer</b><br>Name: <input type="text"/><br>Street: <input type="text"/><br>ZIP code: <input type="text"/><br>Customer N: <input type="text"/><br>E-mail: <input type="text"/> |
| <b>Material/Surfaces</b><br>Surf. Steel: <input type="text"/><br>Surf. Alu.: <input type="text"/><br>Surf. Stainl. st.: <input type="text"/><br><br>Mat. Steel: <input type="text"/><br>Mat. Alu.: <input type="text"/><br>Mat. Stainl. st.: <input type="text"/>         | <b>Customer</b><br>Name: <input type="text"/> Customer N: <input type="text"/><br>Street: <input type="text"/> E-mail: <input type="text"/><br>ZIP code: <input type="text"/><br><br><b>Constructor:</b><br>Name: <input type="text"/> Customer N: <input type="text"/><br>Street: <input type="text"/> E-mail: <input type="text"/><br>ZIP code: <input type="text"/>   | <b>Architect</b><br>Name: <input type="text"/><br>Street: <input type="text"/><br>ZIP code: <input type="text"/><br><br>Customer N: <input type="text"/><br>E-mail: <input type="text"/>  |

☒ Activate project

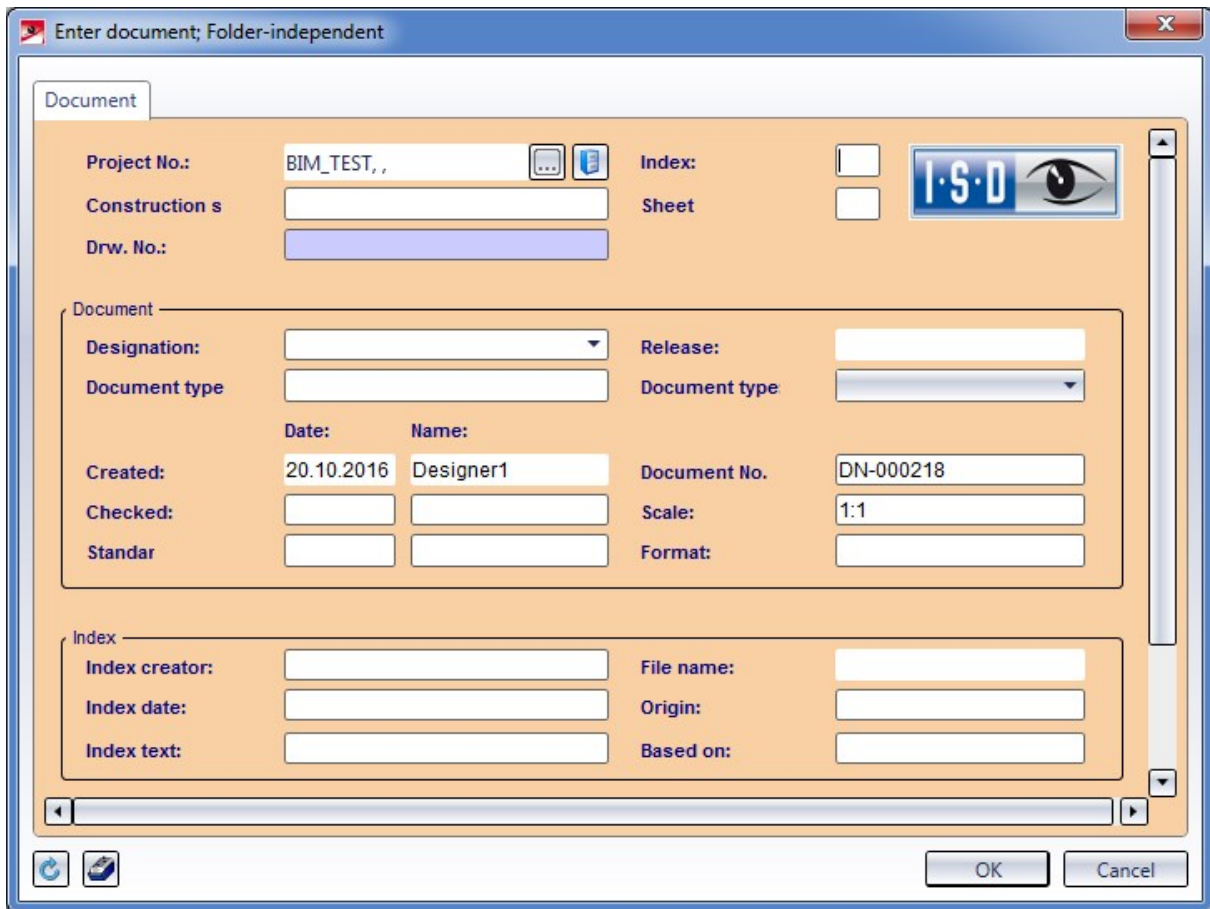
OK Cancel

**Enter article; Folder-independent**

Article

|   |   |   |
|---|---|---|
| Project No.: <input type="text" value="BIM_TEST,,"/><br>Construction s: <input type="text"/><br>Item No.: <input type="text"/>  | Project-related: <input checked="" type="checkbox"/><br>Index: <input type="text"/> |  |
| <b>Part</b><br>Standard designation: <input type="text"/> Release: <input type="text"/><br>Designation 1: <input type="text"/> Part type: <input type="text" value="Assembly"/><br>Designation 2: <input type="text"/> Drawing/Manuf.: <input type="text"/>   |   |   |
| <b>Part info</b><br>Material: <input type="text"/> Order quantity: <input type="text"/><br>Weight: <input type="text"/> [kg] Resourcing: <input type="text"/><br>Dimensions: <input type="text"/> Order note: <input type="text"/><br>Comment: <input type="text"/><br>Article number: <input type="text" value="SN-025647"/> |   |   |
| <b>Index</b><br>Index creator: <input type="text"/> Created: 20.10.2016 Designer1<br>Index date: <input type="text"/> Origin: <input type="text"/><br>Index text: <input type="text"/> Based on: <input type="text"/>   |   |   |

OK Cancel



**Enter document; Folder-independent**

**Document**

**Project No.:** BIM\_TEST,, **Index:** ☐

**Construction s**  **Sheet** ☐

**Drw. No.:**

**Document**

**Designation:**  **Release:**

**Document type**  **Document type**

**Date:** 20.10.2016 **Name:** Designer1 **Document No.** DN-000218

**Checked:**  **Scale:** 1:1

**Standar**  **Format:**

**Index**

**Index creator:**  **File name:**

**Index date:**  **Origin:**

**Index text:**  **Based on:**

When specifying the parameter configuration accordingly, the HELIOS dialogue masks will be replaced with the new masks.

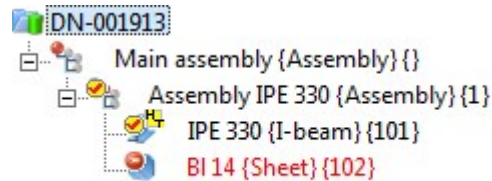


## Modified Ribbon

The Workflow functions at **Management + BIM > HELiOS PDM functions for...** have been removed. The functions are still available on the **HELiOS PDM** tab and in the context menu of the drawing beneath **HELiOS**.

## Indication of Workflow status in ICN

Parts from Released 🟡 drawings and parts from drawings in the Checkup 🟡 status are now indicated as such in the ICN.



## Management of mounting drawings

Besides production drawings you can now also create, update and manage mounting drawings with the Management + BIM functions. For this purpose you use the Dazu steht Ihnen unter die Funktion

### Mounting drawing

function on the **Management + BIM > Drawing**. The function works in the same way as the same-named function at **Drawing > Itemisation/Detailing**, but directly assigns the corresponding HELiOS links to the drawings.

In contrast to the function at **Drawing > Itemisation/Detailing**, no separate function for the updating of the mounting drawing is available here. As you would do for production drawings, use the Create / Update drawing instead.

## New arrangement of settings in Configuration Editor

The Management + BIM settings in the Configuration Editor have been rearranged. They can still be found at ... **PDM > Management + BIM**.

|  | Description                           | Value                                   | Comment  |
|--|---------------------------------------|---|--|
| <div> <div>HiCAD</div> <div> <div>Active configuration (Base configuration)</div> <div> <div>Drawing</div> <div>Automatic drawing derivation</div> <div>Modelling</div> <div>Steel Engineering</div> <div>Metal Engineering</div> <div>Roof/Wall/Facade</div> <div>Plant Engineering</div> <div>Sheet Metal</div> <div>Assembling simulation</div> <div>Analysis</div> <div>Interfaces</div> <div>PDM</div> <div> <div>Management + BIM</div> <div>Production drawings</div> <div>External production drawings</div> <div>Individual part type</div> <div>Revision clouds</div> <div>HiCAD-HELiOS interface</div> <div>Compatibility</div> </div> </div> </div> </div> | Workflow status Checkup               | Checkup                                 | Name of workflow status for checking of parts or documents   |
|  | Workflow status In progress           | In progress                             | Name of workflow status for processing of parts or documents   |
|  | Generate n-digit item number          | No                                      | Generate n-digit item numbers, preceded by left-justified zeroes, when generating production drawings (HELiOS part attribute COMPONENT_ITEM NUMBER)? |
|  | Use item numbers of locked parts      | No                                      | Consider item numbers of locked parts for identical part search?   |
|  | Indices for 3-D models                | No                                      | Also derive model drawing when performing drawing derivation?  |
|  | Manage general 3-D parts              | No                                      | Consider general 3-D parts for drawing management  |
|  | Condition for 'Drawing is up to date' | Main parts and sub-parts, if detail drw | Status 'Drawing is up to date' depends on derived drawing(s).  |
|  | Only allow 2-level structures         | No                                      | Check before data transfer to HELiOS structure   |
|  | Itemisation of locked parts           | Only manually assigned part masters     | Allow itemisation of locked parts?   |
|  | Type of part referencing              | Only one-way                            | One-way referencing (Steel Engineering workshop drawing mode) or Normal referencing  |
|  | Create part document                  | No                                      | Also create the corresponding part documents when auto-creating referenced parts for the production drawing?   |
|  | Clean up project when saving          | Yes                                     | Remove invalid article masters when saving?  |
|  | With BIM                              | <input type="checkbox"/>                | Management via HELiOS active?  |
|  | Create STEP data                      | No                                      | For which parts do you want to generate STEP data?   |




Furthermore, you can now find the switch for the activation of the document and drawing management in civil engineering with BIM here. In older versions these settings could be found in the system file STB\_PARAMETER.DAT. This checkbox will be activated automatically when you perform the parameter configuration.

### Transfer article attributes of semi-finished products to document attributes of detail drawings

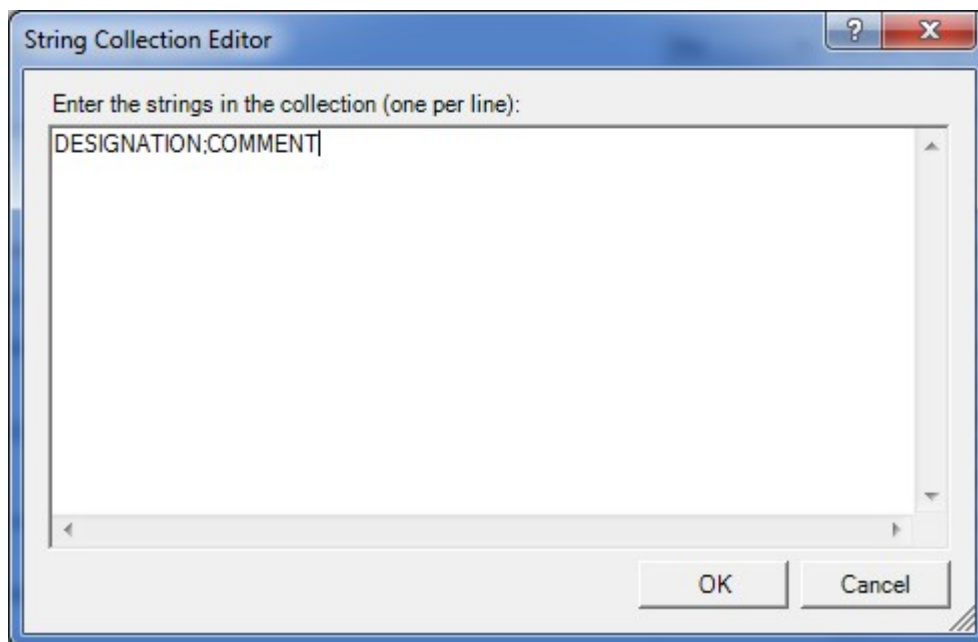
When generating detail drawings with the **Management + BIM** functions you have the option to transfer article attributes of the semi-finished products to the document attributes of the detail drawings. Such document attributes can, for instance, be used in the HELIOS result lists as filters or search criteria.

The assigning takes place in the Configuration Editor at **...PDM > Management + BIM > Production drawings > Transfer article attributes**.

To transfer article attributes of the semi-finished product to the document attributes of the detail drawing, click on the  icon and define the desired assignments. Each assignment is defined in one row and is composed like this:

#### Article attribute;Document attribute

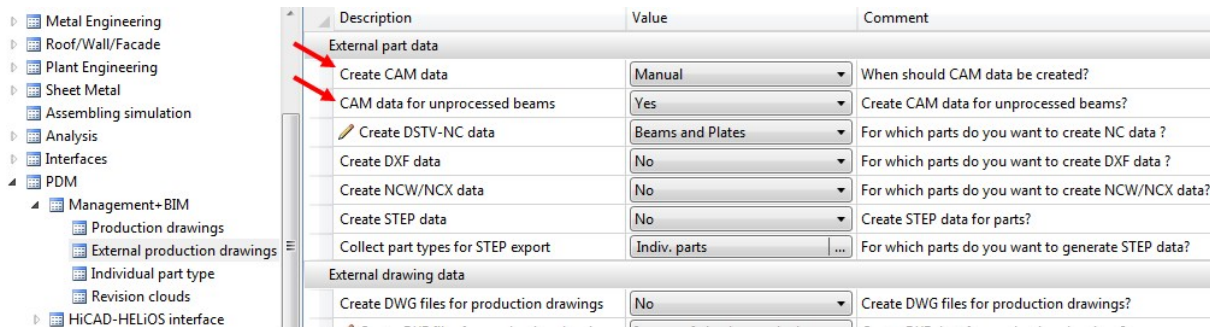
If, for instance, you want to assign the Article attribute BENENNUNG (Designation) to the Document attribute BEMERKUNG (Comment), the assignment must look like this:



## Creation of CAM data

You have the option to create the CAM data for the parts in the production drawings of the current model drawing, manage them in HELiOS and link them accordingly. This is possible for the formats DSTV-NC, NCW, DXF and STEP.

The creation of these CAM data can now take place manually or automatically. This can be specified in the Configuration Editor at ...PDM > Management + BIM > External production drawings > Create CAM data.

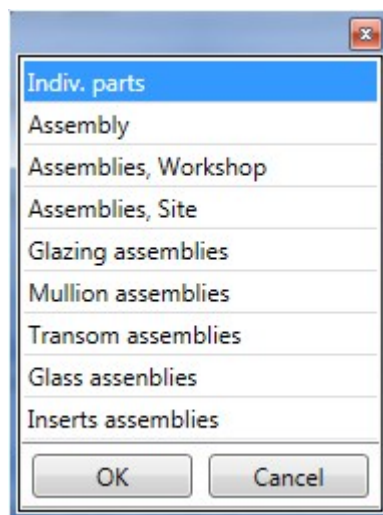


The default setting is **Manual**, i.e. you need to execute the **Production** function for the creation of the CAM data.

Which CAM data will be created also depends on the settings at ...PDM > Management+BIM > External production drawings. Only those CAM data will be created that you explicitly selected in the settings. For instance, if all settings for the **Create ... data** parameters have been set to **No**, no CAM data will be created. This applies both to the automatic and to the manual creation, i.e. even the **Production** function will then generate no CAM data.

Via the switch **CAM data for unprocessed beams** you can specify whether "unprocessed" beams are to be considered for the CAM data creation or not. In the drawing you distinguish between processed and unprocessed beams, depending on the corresponding feature log entries of a beam. For instance, a lengthening, trimming or moving of points in a rectangle will not make a beam a "processed" beam. All other processings, e.g. bores, subtractions, notches etc. will make the beam a "processed" beam.

Also new is the possibility to select the part types to be considered for STEP export.



## NCW format for part data

You can now automatically create NCW data for part, manage them in HELiOS and link them.

To enable the creation of this file format, the settings in the Configuration Editor at ...**PDM > Management + BIM > External production drawings** must have been specified accordingly. The ISD default setting is **No**, i.e. no NCW data will be created.

Please note in this context that the **Export...** menu has been expanded.

## Automatic creation of DWG files of production drawings during the Workflow

DWG files of production drawings can now be automatically created

- upon Checkup and Release or
- upon creation and updating

of drawings. For this purpose, the settings in the Configuration Editor at **PDM > Management + BIM > External production drawings** have been expanded accordingly.

Please note in this context that the **Export...** menu has also been expanded.

## Derive assembly



When you use the Derive, Active assembly(-ies) with sub-parts function you can now immediately place the copy at the right position in the drawing. After function call a preview of the copy will be displayed at the cursor. Specify a fitting point in the preview, and then specify the position of this point in the drawing. Instead of a point option you can also choose to place the fitting point in the origin (0, 0, 0) of the part coordinate system.

## Part attributes - Locking of item number in BIM drawings

If a drawing is active that has been created and processed with the **Management + BIM** functions, the **Item number** field in the **Part attributes** window will be locked.

## Manual cleaning up of projects



New in the **Projects** menu is the Clean-up function.

This function cleans up the active project by searching for invalid article masters and removing them. The active construction must be itemized and saved before using the function. Otherwise, a corresponding message will prompt you to do so.

Using this function only makes sense if the parameter **Clean up project when saving** at **PDM > Management + BIM** has been set to **No**. The default setting is **Yes**, i.e. whenever you save your drawing, invalid article masters will be removed automatically.



## Checkup and Release of production drawings

Previously, production drawings could only obtain the Workflow status **Checkup** or **Released** if the associated model drawing was loaded. Now, the Release and the Checkup functions can also be applied if a production drawing is active.

If your answer to the query *Release drawing including all parts?* or *Set drawing including all parts to "Checkup"?*, respectively, is **Yes**, the drawings and all parts contained in it will be set to the status **Checkup** or **Released**, respectively. When you load the associated model drawing, these parts will then be locked against processing there.

## Assigning of the attributes SACHNUMMER (Article number) and BENENNUNG (Designation) via FTD templates

If you want to create the HELiOS attributes SACHNUMMER (Article number) and BENENNUNG (Designation) from template files, open the Configuration Editor, choose **PDM > Management + BIM > Production drawings** and set the parameter **Create HELiOS attributes from FTD file** to **Yes**. These templates could previously be edited with the Steel Engineering function **Edit templates for part annotation**.


As of HiCAD 2016 SP2 the new **Templates, Attribute assignment**  function will be available for this. You can find the function at **Steel Engineering > Further functions > Settings**  > ....

| Template file                            | Settings for the  | Pre-setting  |
|--|---|--|
| BIM_PDM_WSD_Assembly_ArticleNumber.ftd   | Configuration of the HELiOS document attribute <b>SACHNUMMER</b> (Article number) for detail drawings of assemblies (assemblies with one main part) | {Total quantity (Part attribute)} x {Article number (Part attribute)}  |
| BIM_PDM_WSD_Assembly_Designation.ftd     | Configuration of the HELiOS document attribute <b>BENENNUNG</b> (Designation) for detail drawings of assemblies (assemblies with one main part)     | %TS(TEXTE_STB401) {Item number (Part attribute)}<br>%TS(TEXTE_STB401) stands for the content of the text key STB_401 which is the text <b>Item No.:</b>            |
| BIM_PDM_WSD_Beam_ArticleNumber.ftd       | Configuration of the HELiOS document attribute <b>SACHNUMMER</b> (Article number) for detail drawings of beams                                      | {Total quantity (Part attribute)} {Article number (Part attribute)} x {Länge (Part attribute)}   |
| BIM_PDM_WSD_Beam_Designation.ftd         | Configuration of the HELiOS document attribute <b>BENENNUNG</b> (Designation) for detail drawings of beams  | %TS(TEXTE_STB401) {Item number (Part attribute)}   |
| BIM_PDM_WSD_Multi_Designation.ftd        | Configuration of the HELiOS document attribute <b>BENENNUNG</b> (Designation) for detail drawings of multi-part beams                               | %TS(TEXTE_STB401)  |
| BIM_PDM_WSD_Part_ArticleNumber.ftd       | Configuration of the HELiOS document attribute <b>SACHNUMMER</b> (Article number) for detail drawings of general parts                              | {Total quantity (Part attribute)} x {Article number (Part attribute)}  |
| BIM_PDM_WSD_Part_Designation.ftd         | Configuration of the HELiOS document attribute <b>BENENNUNG</b> (Designation) for detail drawings of general parts                                  | %TS(TEXTE_STB401) {Item number (Part attribute)}   |
| BIM_PDM_WSD_SheetMetal_ArticleNumber.ftd | Configuration of the HELiOS document attribute <b>SACHNUMMER</b> (Article number) for detail drawings of Sheet Metal parts                          | {Total quantity (Part attribute)} x {Article number (Part attribute)} x {Length of 2-D development (Part attribute)} x {Width of 2-D development (Part attribute)} |
| BIM_PDM_WSD_SheetMetal_Designation.ftd   | Configuration of the HELiOS document attribute <b>BENENNUNG</b> (Designation) for detail drawings of Sheet Metal parts                              | %TS(TEXTE_STB401) {Item number (Part attribute)}   |
| BIM_PDM_WSD_SteelPlate_ArticleNumber.ftd | Configuration of the HELiOS document attribute <b>SACHNUMMER</b> (Article number) for detail drawings of Steel Engineering plates                   | {Total quantity (Part attribute)} x {Article number (Part attribute)} x {Length (Part attribute)} x {Width (Part attribute)}                                       |
| BIM_PDM_WSD_SteelPlate_Designation.ftd   | Configuration of the HELiOS document attribute <b>BENENNUNG</b> (Designation) for detail drawings of Steel Engineering plates                       | %TS(TEXTE_STB401) {Item number (Part attribute)}   |

## Drawing function group

### Auto-create / update drawings

At **Management + BIM > Drawing > ...** you can find the new AutoCreate/Update  function.

In contrast to the Create/Update Drawing - Manual Settings  function, the settings for drawing derivation (i.e. for drawing sheets, views, view groups etc.) are not be specified in the dialogue, but are loaded from the corresponding settings files. This means that the **Drawing derivation** window will not be displayed here. In this way, drawing creation can be speeded up.

By default, the following settings files are supplied with HiCAD:

- BIM\_WSD\_Default\_IndividualPart.DAT for individual parts
- BIM\_WSD\_Default\_Assembly.DAT for assemblies
- BIM\_WSD\_Default\_Bolted\_Assembly.DAT for bolted assemblies
- BIM\_WSD\_Default\_Welded\_Assembly.DAT for welded assemblies

The files are located in the HiCAD sys directory.


Besides the option to use the predefined settings files supplied with HiCAD, you can also create, depending on the respective usage/part type, your own settings files.

### Set / withdraw 'Checkup' status of assemblies

As of HiCAD 2016 SP1 you have the option to assign the Workflow status **Checkup** to the drawings of individual or several assemblies. This also includes the sub-parts of the assemblies. Likewise, you can withdraw the assigned status if desired. For this to happen, you can use the functions beneath Checkup shown below:



### Derive assemblies

Use the new Derive, active assembly(-ies) with sub-parts  function to derive (copy) a new assembly from the active assembly (also structure assemblies) including all its sub-parts. A prerequisite for this is that the assembly has been released.

When the copy of the assembly is created, please note the following:

- Article masters and item numbers will be deleted during the derivation.
- The linked documents, i.e. the drawings, will not be copied.

## Settings in the Configuration Editor

### Production drawing for unprocessed beams

Depending on the corresponding feature entries, you distinguish between **processed** and **unprocessed** beams in your model drawing. For example, a lengthening, trimming, or moving points in a rectangle will not make the beam a "processed" beam. All other functions, e.g. as bores, subtractions, notches etc. on a beam will make the beam a "processed" beam.

The following settings are possible:

- **Yes**  
Unprocessed beams will also be considered for derived drawings. This is the default setting.
- **No (cuts are no processings)**  
The HELIOS link **No drawing required** will be assigned to unprocessed beams when the model drawing is saved. This means that no drawings will be created for these beams. Cuts are not interpreted as processings, i.e. they will not be considered for derived drawings.
- **No (cuts are processings)**  
The HELIOS link **No drawing required** will be assigned to unprocessed beams when the model drawing is saved. This means that no drawings will be created for these beams. Cuts are interpreted as processings, i.e. they will be considered for derived drawings.



## Major Release 2016 (V 2100)

### Licensing

#### **BIM functionalities for HiCAD industry suites and modules**

The BIM functionality will be available for the following product combinations:

- HiCAD Steel Engineering Package + HELiOS Engineer+ (as of HiCAD 2015, SP2)
- HiCAD Steel Engineering + HELiOS Engineer+ (as of HiCAD 2016)
- HiCAD Steel Engineering + HiCAD Stairs + Railings + HELiOS Engineer + (as of HiCAD 2016)
- HiCAD Metal Engineering + HELiOS Engineer+ (as of HiCAD 2016)
- HiCAD Sheet Metal professional + HELiOS Engineer+ (as of HiCAD 2016)
- HiCAD Sheet Metal suite + HELiOS Engineer+ (as of HiCAD 2016)
- HiCAD Sheet Metal suite premium + HELiOS Engineer+ (as of HiCAD 2015, SP2)
- HiCAD Steel Engineering suite + HELiOS Engineer+ (as of HiCAD 2015, SP2)
- HiCAD Steel Engineering suite premium + HELiOS Engineer+
- HiCAD Metal Engineering suite + HELiOS Engineer + (as of HiCAD 2015, SP2)
- HiCAD Metal Engineering suite premium + HELiOS Engineer+ (as of HiCAD 2015, SP2)
- HiCAD Mechanical Engineering suite premium + HELiOS Engineer+ (as of HiCAD 2015, SP2)
- HiCAD Student + HELiOS Engineer+ (as of HiCAD 2016)
- HiCAD Trainee suite + HELiOS Engineer+ (as of HiCAD 2016)

### Configuration

#### **HicadBIMConfig.EXE replaces HeliosBIMConfig.EXE**

Previously, the tool HeliosBIMConfig.EXE was used to perform updates for the database and the database links. This tool has been replaced with the tool

#### **HicadBIMConfig.EXE.**



Due to the enhanced release functionality, customers who have already used the Management+BIM module before HiCAD 2016, must execute the **HicadBIMConfig.exe** tool in the HiCAD EXE directory once before they start working with HiCAD 2016. In this way it is ensured that links without relevance will be created, and that the release relevance works properly when working with the new functions. Customers who use the Management+BIM module of HiCAD 2016 for the first time do not need to do this, as the relevant database links will then be automatically correct.

#### **Individual part type - Mounting or Workshop**

You often distinguish between production processes where parts are welded in a workshop and production processes where parts are sent to a construction site after processing for mounting tasks.

Such "mounting parts" can be beams, profiles, plates, sheets, and general parts, and are called "loose parts" in HiCAD. Such "loose parts" are parts that do not belong to a BOM-relevant assembly in which an assembly main part has been defined (welding assembly).

Use the settings at **Management + BIM > Individual part type** in the Configuration Editor to achieve that these parts will be automatically recognized by HiCAD during detail drawing creation and marked correspondingly in the title block of the drawing.

| Description  | Value           | Comment   |
|--|-----------------|---|
| Individual part type Main part of assembly   | Individual part | Initializes the HELiOS attribute COMPONENT_WSDTYPE)               |
| Individual part type Sub-part of assembly  | Individual part | Initializes the HELiOS attribute COMPONENT_WSDTYPE)               |
|  Individual part type Loose part            | Mounting part   | Initializes the HELiOS attribute COMPONENT_WSDTYPE)               |
|  Distinguish loose parts and assembly parts | Yes             | Distinguish loose parts and assembly parts in production drawing? |

|   |      |             |         |          |                     |   |                |          |
|---|------|-------------|---------|----------|---------------------|---|----------------|----------|
| 1   | 104  | IPE 400     | 2150 01 |          |                     | [-.....]  | [-.....]       | 140.1 kg |
| Qty.  | Item | Designation | Length  | Width    | Strength            | Angle View  | Angle Top View | Weight   |
| Project/Project No.                         |      | LOT         |         | Material |                     | <br>THE WORLD OF CAD AND PDM SOLUTIONS |                |          |
| w5  |      |             |         | S235JR   |                     |   |                |          |
| Content                                     |      |             |         | Check    |                     |   |                |          |
| Mounting part104                            |      |             |         | Drawn    | 29.01.2016          |   |                |          |
| 1x IPE 400x2159                             |      |             |         | Date     |                     | Index   | Modification   | Date     |
| Used in the following assemblies:           |      |             |         | Qty.     | TYPE                |   | Index          |          |
| Contained in the following drawing numbers: |      |             |         | 1        | Beam detail drawing |   |                |          |
|   |      |             |         |          | Drawing No.         |   |                |          |
|   |      |             |         |          | w5-                 |   |                |          |
| Scale: 1:10                                 |      |             |         |          |                     |   |                |          |

## Manage projects

### Copy projects

Projects can now be derived from existing BIM projects at the push of a button, with all articles and documents complete with all links. Even documents which do not belong to the project, such as standard parts, will be considered.

To do this, use the new **Copy project**  function (**Management+BIM > Project > New...**).

## Production document creation

### Automatic creation of DXF-/PDF files during a workflow

For production drawings you have now the option to create DXF and PDF files either

- during Checkup or Release, or
- during creation and updating


of the drawings. For this purpose, the Configuration Editor now offers additional setting options at **PDM > Management + BIM**:

|  |                                |  |
|--|--------------------------------|--|
| Create DSTV-NC data                      | No                             | For which parts do you want to create NC data ?    |
| Create DXF data                          | No                             | For which parts do you want to create DXF data ?   |
| Create STEP data                         | No                             | For which parts do you want to generate STEP data? |
| Create DXF files for production drawings | No                             | Create DXF data for production drawings?           |
| Create PDF files for production drawings | No                             | Create PDF files for production drawings?          |
| Postprocessor for PDF creation           | No                             | Postprocessor for PDF creation                     |
|  | In case of checkup and release |  |
|  | Upon creation and update       |  |

## Combined functions

The functions

- Production
- Production documents - Update, All and
- Production documents - Update, via HELiOS

have been combined into one, central **Production**  function. This function finds - depending on the current settings in the Configuration Editor - all parts of the current drawing for which no production documents exist yet, i.e. parts with the links

- Without DSTV-NC data
- DSTV-NCdata not up to date
- Without DXF data
- DXF data not up to date
- Without STEP data
- STEP data not up to date

and displays them in the search mask of the HELiOS Desktop. There you can select the corresponding parts.

## Automated processes for detail drawing creation

Previously, the **Drawing derivation** function needed to be performed repeatedly when generating detail drawings into external drawings in order to ensure that all drawings are actually created.

### Example:

In the Configuration Editor, the **Main parts and sub-parts, if detail drws created for them** option has been selected for the parameter **Condition for 'Drawing is up to date'**. Previously, the drawing derivation needed to be created for assemblies, and then also for the main parts and sub-parts! Now, HiCAD automatically recognizes objects that do not have a drawing yet, and lists them without you having to exit the **Drawing derivation** function.

Please see also Example 1, Step 6.

## Metal Engineering

### Service Pack 2 2016 (V 2102)

#### Improved behaviour in the event of LogiKal crashes

The behaviour of HiCAD and LogiKal in the event of LogiKal crashes has been improved: While a restart of HiCAD and LogiKal was formerly required in such cases, the interface will now be restored automatically and will be available again immediately.

### Service Pack 1 2016 (V 2101)

#### AutoSearch for LogiKal installations removed

The automatic search for LogiKal installations that previously took place when no path to a LogiKal installation does no longer take place for performance reasons.

This may require an explicit entering of the correct path to the LogiKal installation in the settings.

### Major Release 2016 (V 2100)



#### New menu structure, new functions

The arrangement of functions on the Metal Engineering tab has changed. A logical restructuring has taken place - for example, functions that manage parts in HiCAD and functions for LogiKal items have been arranged into one group.

All functions for the creation or the import of parts in HiCAD can now be found in the **New** function group, functions for the processing of metal and glazing constructions can be found in the **Process** function group. These groups replace the previous function groups **Metal Engineering, 3-D, LogiKal, 3-D.** and **Process, LogiKal 3D**.

**Update** functions for parts, as well as the **Import/ Export** between HiCAD and LogiKal can be found in the same-named function groups on the Ribbon tab.



New is the  **Transfer HiCAD processings to LogiKal** function. With this function you can not only transfer changes made in LogiKal to HiCAD, but, conversely, also transfer processings made in HiCAD (circular bores, slots, rectangular holes, blind holes, threads) to the corresponding LogiKal item via .

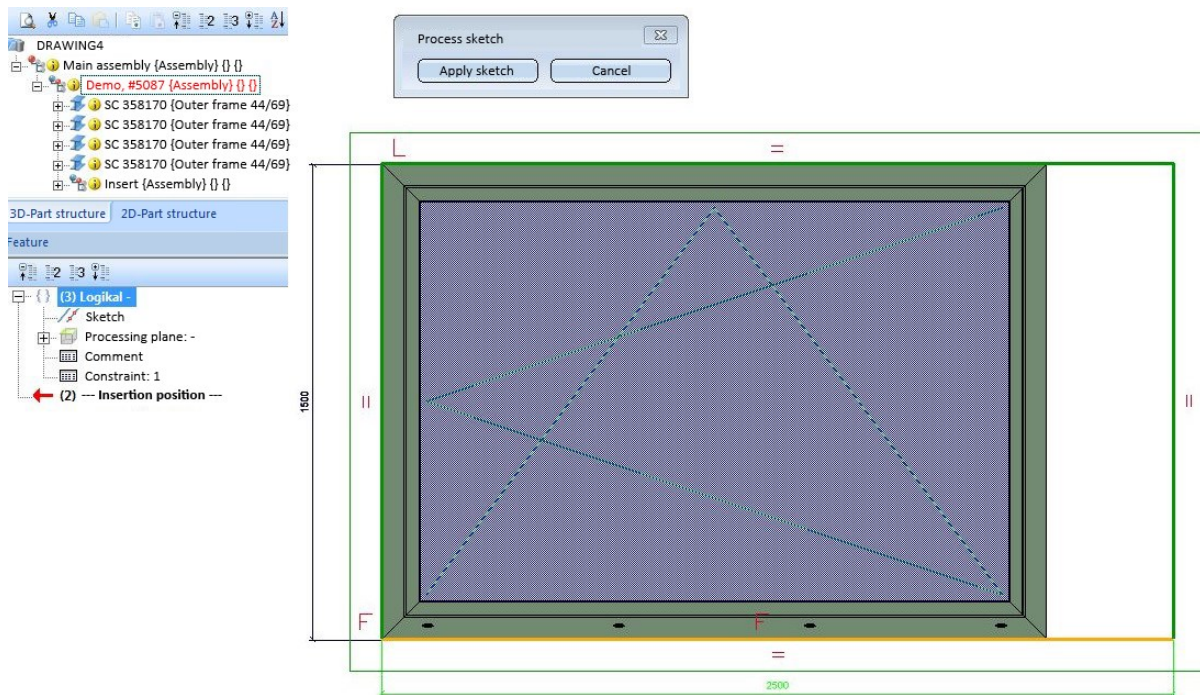


The **Report** function in the **Others** function group allows you to call the LogiKal report for the active part (or a multiple selection of parts).

## Change item from sketch subsequently



When you choose **Metal Engineering > New > Sketch** to create a glass element or a facade in a sketch, a feature log entry will be created for the sketch of the LogiKal item. Via this feature entry you can modify the sketch subsequently, e.g. you can change its dimensions, which will also change the dimensions of the corresponding LogiKal item with only a few mouse clicks.



## Purpose for new sketches in the Metal Engineering grid

HiCAD automatically assigns a purpose to planar sketches upon their creation. One distinguishes between

- Purpose = Part
- Purpose = Create/Edit

Sketches with the purpose **Create/Edit** are, in principle, "auxiliary elements" from which other parts, processings and sectional views, detail views or cut-outs. Sketches with this purpose are visualized with broader lines in the active view. They are also visible in shaded models.

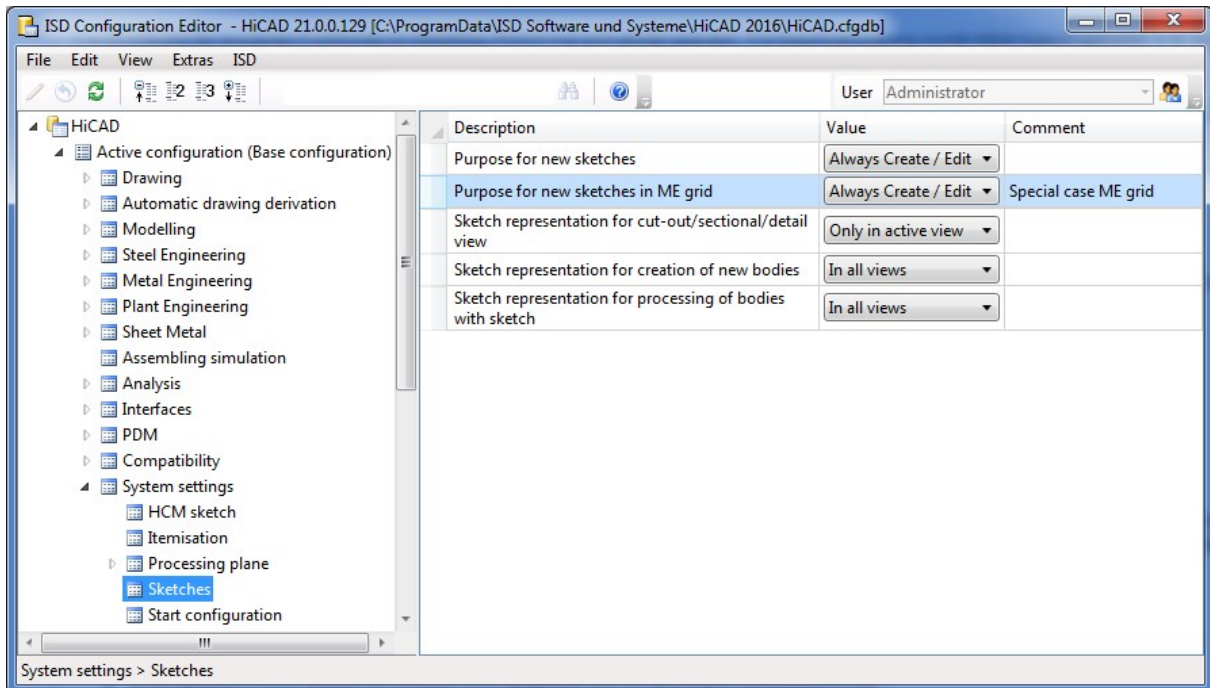
Sketches with the purpose **Part** are regarded as "normal" parts that are used as elements of the drawing. They distinguish themselves from sketches with the purpose **Create/Edit** as follows:

- In cut-outs, sectional views and detail views the sketch will be cut.
- The sketches have no special representation and are not shown in shaded models (without edges).
- The sketch may contain technical tools such as centre lines, crosshairs etc.

In the Configuration Editor, at **System settings > Sketches > Purpose for new sketches in ME grid** (for a working with the Metal Construction plugin) you can specify which of the two purposes is to be assigned when a new sketch is created.



The default setting is **Always Create/Edit**.



## Plant Engineering

Service Pack 2 2016 (V 2102)

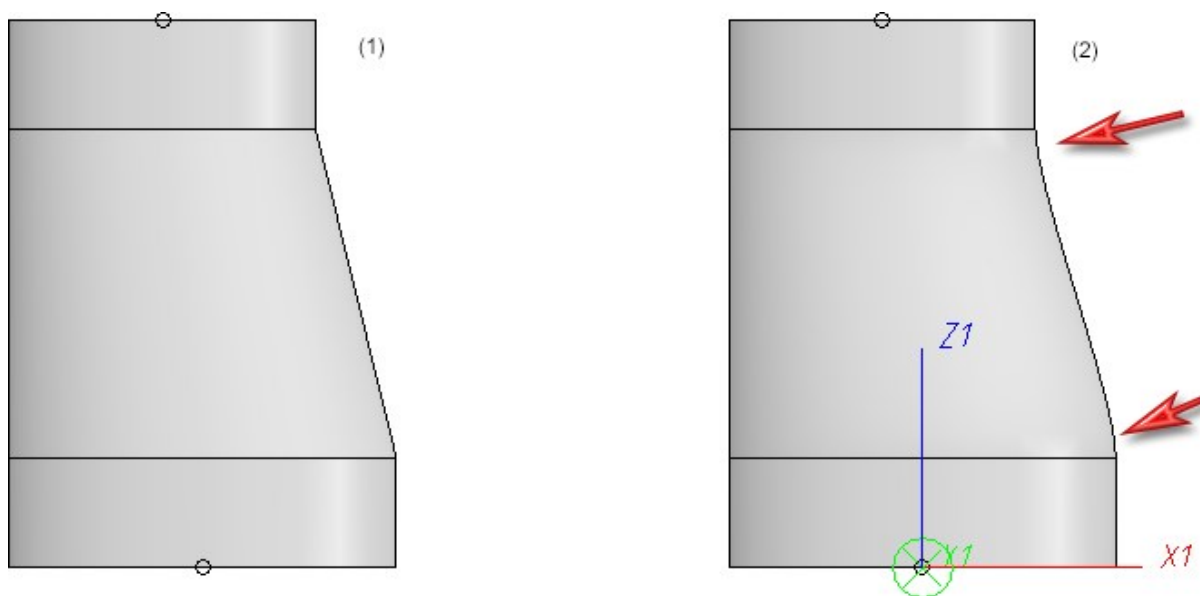
New/changed parts

### Further macro variants replaced with feature variants

The following macro variants have been replaced with feature variants:

- ASME\_REDUEX.vaa
- EN10253-2-A\_REDUCER\_EXC.vaa
- EN10253-2-B\_REDUCER\_EXC.vaa
- EN10253-4-A\_REDUCER\_EXC.vaa
- EN10253-4-A\_REDUCER\_EXC\_STR.vaa
- EN10253-4-B\_REDUCER\_EXC.vaa
- EN10253-4-B\_REDUCER\_EXC\_STR.vaa

The new variants also support the insertion option Create hollow body, with all available details. If the option is active, the transitions will be slightly filleted:



Create hollow body with all available details - (1) inactive, (2) active

### New ASME parts

Many new ASME parts are available - mostly ASME fittings with flange connections and ASME flanges.

The following additional Standard Part feature variants were created:



- ASME\_2004\_PIPE.vaa
- ASME\_2012\_CAP.VAA
- ASME\_2012\_CROSS.VAA
- ASME\_2012\_CROSS\_RED.VAA
- ASME\_2012\_ELBOWLR\_180.VAA
- ASME\_2012\_ELBOWLR\_45.VAA
- ASME\_2012\_ELBOWLR\_90.VAA
- ASME\_2012\_ELBOWLR\_RED.VAA
- ASME\_2012\_ELBOWSR\_180.VAA
- ASME\_2012\_ELBOWSR\_90.VAA
- ASME\_2012\_REDUCER.VAA
- ASME\_2012\_REDUCER\_EX.VAA
- ASME\_2012\_TEE.VAA
- ASME\_2012\_TEE\_RED.VAA
- ASME\_2013\_BRANCH45\_FLANGED150.VAA
- ASME\_2013\_BRANCH45\_FLANGED300.VAA
- ASME\_2013\_CROSS\_FLANGED150.VAA
- ASME\_2013\_CROSS\_FLANGED300.VAA
- ASME\_2013\_ELBOW45\_FLANGED150.VAA
- ASME\_2013\_ELBOW45\_FLANGED300.VAA
- ASME\_2013\_ELBOWLR\_FLANGED150.VAA
- ASME\_2013\_ELBOWLR\_FLANGED300.VAA
- ASME\_2013\_ELBOWSR\_FLANGED150.VAA
- ASME\_2013\_ELBOWSR\_FLANGED300.VAA
- ASME\_2013\_LWNFLANGE150.vaa
- ASME\_2013\_LWNFLANGE1500.vaa
- ASME\_2013\_LWNFLANGE2500.vaa
- ASME\_2013\_LWNFLANGE300.vaa
- ASME\_2013\_LWNFLANGE400.vaa
- ASME\_2013\_LWNFLANGE600.vaa
- ASME\_2013\_LWNFLANGE900.vaa
- ASME\_2013\_REDUCER\_EX\_FLANGED150.VAA
- ASME\_2013\_REDUCER\_EX\_FLANGED300.VAA
- ASME\_2013\_REDUCER\_FLANGED150.VAA
- ASME\_2013\_REDUCER\_FLANGED300.VAA
- ASME\_2013\_TEE\_FLANGED150.VAA
- ASME\_2013\_TEE\_FLANGED300.VAA
- ASME\_2013\_Y-PIPE\_FLANGED150.VAA
- ASME\_2013\_Y-PIPE\_FLANGED300.VAA
- ASME\_BLFLANGE150.VAA
- ASME\_BLFLANGE1500.VAA
- ASME\_BLFLANGE2500.VAA
- ASME\_BLFLANGE300.VAA
- ASME\_BLFLANGE400.VAA
- ASME\_BLFLANGE600.VAA
- ASME\_BLFLANGE900.VAA
- ASME\_LAPPEDFLG150.VAA
- ASME\_LAPPEDFLG1500.VAA
- ASME\_LAPPEDFLG2500.VAA
- ASME\_LAPPEDFLG300.VAA
- ASME\_LAPPEDFLG400.VAA
- ASME\_LAPPEDFLG600.VAA
- ASME\_LAPPEDFLG900.VAA
- ASME\_SOFLANGE150.VAA
- ASME\_SOFLANGE1500.VAA
- ASME\_SOFLANGE300.VAA
- ASME\_SOFLANGE400.VAA
- ASME\_SOFLANGE600.VAA
- ASME\_SOFLANGE900.VAA
- ASME\_WNFLANGE150.vaa
- ASME\_WNFLANGE1500.vaa
- ASME\_WNFLANGE2500.vaa
- ASME\_WNFLANGE300.vaa
- ASME\_WNFLANGE400.vaa
- ASME\_WNFLANGE600.vaa
- ASME\_WNFLANGE900.vaa

In case of an update these parts may require to be entered into the database (or the HiCAD catalogue, respectively) first. For this, corresponding archive file lists are available via DBPlantDataImport and PartDataAutoSync:

- ASME\_B16-5\_Blindflanges.lst
- ASME\_B16-5\_Lapped\_Flanges.lst

- ASME\_B16-5\_Slip\_On\_Flanges.lst
- ASME\_B16-5\_Weld\_Neck\_Flanges.lst
- ASME\_B16-5\_Flanged\_Fittings.lst

## ERMETO parts

In HiCAD 2016 SP2 the ERMETO parts have been revised, to allow a displaying of the parts as separate parts both in the drawing and in BOMs.

The revised ERMETO variants can be found arranged by series in the files

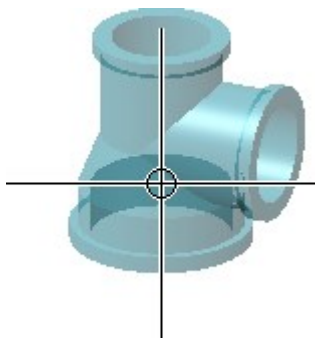
- EO\_LL.lst
- EO\_L.lst
- EO\_S.lst

The previous ERMETO\_O variants are no longer supplied.

## Operation

### Preview for the insertion of pipe parts and components

When you insert pipe parts and components a shaded, semi-transparent preview of the part will be displayed (as is the case when inserting other 3-D parts).



## Changes / enhancements in HELiOS

### Revised HELiOS masks for classification of article masters in Plant Engineering

The HELiOS masks for classification of article masters in Plant Engineering have been revised and now contain a 3-D image and a sectional view of the chosen part type. The advantage of the new mask consists in the provision of additional, technical relevant attributes on the component masks (e.g. for Pumps) and a reliable assignment of geometric parameters.

Classification mask - Branch, round

**Find article**

**Search conditions**

Article Classification **Branch, round**

Connection 1

Nominal diameter / NPS: 50.000000 mm  inch

Outer diameter (1):  Wall thickness (  )

Connection type:

Connection 2

Nominal diameter / NPS:  mm  inch

Outer diameter (3):  Wall thickness (  )

Connection type:

Connection 3

Nominal diameter / NPS:  mm  inch

Outer diameter (5):  Wall thickness (  )

Connection type:

Pipe part properties

Angle (7):

Schedule:

Pressure:

Fitting







Preferred type:

Accessory set:

P-ID symbols

**Search result**

Standard

| Rt  | Number   | In | Designation        | Design | Standard de  | Part type      | Release statu | Creation dat | User          | Nomi      | NPS ( | Outer | Conn  | Nomi      | f |
|---|----------|----|--------------------|--------|--------------|----------------|---------------|--------------|---------------|-----------|-------|-------|-------|-----------|---|
|  | TN-06289 |    | +GF+ T-piece 45° S |        | EN 10242 S16 | Semi-fin. proc | In Progress   | 29.08.2005   | Administrator |           |       |       | 32000 |           |   |
|  | TN-06290 |    | +GF+ T-piece 45° S |        | EN 10242 S16 | Semi-fin. proc | In Progress   | 29.08.2005   | Administrator | 10 3/8"   |       | 17.2  | 32000 | 10 3/8"   |   |
|  | TN-06291 |    | +GF+ T-piece 45° S |        | EN 10242 S16 | Semi-fin. proc | In Progress   | 29.08.2005   | Administrator | 15 1/2"   |       | 21.3  | 32000 | 15 1/2"   |   |
|  | TN-06292 |    | +GF+ T-piece 45° S |        | EN 10242 S16 | Semi-fin. proc | In Progress   | 29.08.2005   | Administrator | 20 3/4"   |       | 26.9  | 32000 | 20 3/4"   |   |
|  | TN-06293 |    | +GF+ T-piece 45° S |        | EN 10242 S16 | Semi-fin. proc | In Progress   | 29.08.2005   | Administrator | 25 1"     |       | 33.7  | 32000 | 25 1"     |   |
|  | TN-06294 |    | +GF+ T-piece 45° S |        | EN 10242 S16 | Semi-fin. proc | In Progress   | 29.08.2005   | Administrator | 32 1 1/4" |       | 42.4  | 32000 | 32 1 1/4" |   |

Number of data records: 69

Find OK Cancel

Classification mask - Vessel

Find article

Search conditions

Article Classification Vessel

General vessel parameters

Type designation

Design

Dimensions & Content

Vessel diameter (1)  mm

Cyl. vessel length (2)  mm

Useable volume  m³/h

Insertion properties

Preferred type:

P-ID symbols

Search result

Standard

| R# | Number   | In | Designation      | Design | Standard de | Part type      | Release statu | Creation dat | User          | Prefer | P-ID | Type | Const | Volun | Out |
|----|----------|----|------------------|--------|-------------|----------------|---------------|--------------|---------------|--------|------|------|-------|-------|-----|
| 1  | TN-01497 |    | Example vessel 1 |        | (BEH1)      | Semi-fin. proc | In Progress   | 23.06.2000   | Administrator |        |      |      |       |       |     |
| 2  | TN-01498 |    | Example vessel 2 |        | (BEH2)      | Semi-fin. proc | In Progress   | 23.06.2000   | Administrator |        |      |      |       |       |     |

Number of data records: 2

Find OK Cancel

## New HELiOS attributes

In conjunction with the revised HELiOS masks, 5 new attributes for plant Engineering are now available:

| Attribute name | Benennung  |
|----------------|--|
| FOERDERHOEHE   | Delivery height (of a pump)                      |
| FOERDERMENGE   | Delivery volume (of a pump)                      |
| NPSH           | NPSH (Net Positive Suction Head) value of a pump |
| P_NENN         | Nominal performance (of a pump)                  |
| TEMPERATUR     | Temperature (for valves)                         |

To add the new attributes to your HELiOS database, use the DBPlantDataImport.exe tool. There you can find the **Update HELiOS for Plant Engineering** function.

## Part Tools

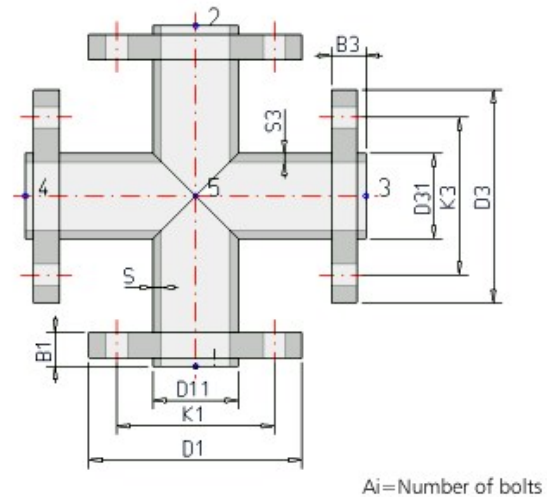
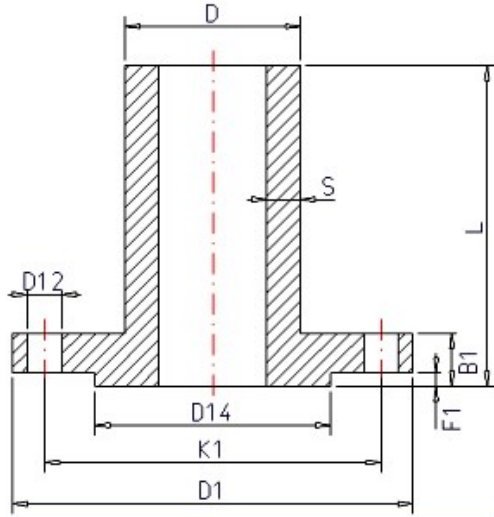
### Bolted flange connections for own parts or variants

You can use the **Flange connection, bolted** functions of the Plant Engineering module to place boltings on flange connections. To ensure that the boltings match the flanges, the functions will evaluate the variables of the part variant to which the flange belongs. This also needs to be considered if construct your own parts/variants with flange connections, because otherwise, problems will occur during their processing with the flange connection functions.

The following table shows the lists of variables. The second place of the variable's designation must match the connection number. The list here refers to the example of the first connection of a part.

|            |  |
|------------|--|
| <b>B1</b>  | Thickness of flange plate ( $B1 = C1 + F1$ ) |
| <b>C1</b>  | Thickness of flange plate (without collar)   |
| <b>F1</b>  | Thickness of collar                          |
| <b>A1</b>  | Number of bolts                              |
| <b>K1</b>  | Diameter of bore circle                      |
| <b>D12</b> | Bore diameter                                |
| <b>L</b>   | Part length (not connection-specific)        |

Hence, the variables of the second connection will have the designations B2, C2, F2, A2, D22.



Additional, part-specific rules apply during evaluation:

1. If the variables C1 and F1 are defined in a loose flange, the grip length of the bolting will be determined via the value  $C1+F1$  instead of using B1.
2. For blank flanges the variable L instead of B1 will be used for grip length determination provided that L has been defined.

The value of the variables NI (capital "N", capital "i", NO "1"! ) is used as an additional attribute during creation or querying of an article master in HELiOS. It represents the nominal diameter in Inches.

## Using connection type attributes

### Connection type ID with priority information

In practice it can happen that two parts coincide which both specify additional parts to be inserted via their connection type (often Fasteners). In such cases, the last but one digit of the connection will be interpreted as a priority indicator. If one part is connected to another part, on the connecting points of which fasteners are already located, the prioritization will be evaluated. If the part to be connected has a higher priority, the fasteners that already exist on the connection will be removed and replaced with the part to be connected.

If the connection type of the part to be connected has a lower priority, its fasteners will not be inserted.

In cases of equal priority new fasteners will only be inserted if no fasteners exist on the connection yet.

The priority information will also be evaluated if a part does not supply any further elements. Even in such cases, existing fasteners will be removed if the part to be connected has a connection type with a higher priority than that of the existing connection. This means that a part can remove all fasteners because it acts as a fastener itself.

### Connection type ID - List of part standards

Instead of specifying a standard in the supplement, it is also possible to specify a list of standards separated by commas, e.g.

**41002 1 5971010 Standard\_A,Standard\_B,Standard\_C**

During part search in the catalogue or in the HELiOS database the standards will be combined by means of an OR condition. This means that not several parts are specified here, but the range of allowed parts will be extended here.





#### Please note:

In the part standards list, whitespaces will be interpreted as allowed characters of a standard designation. Therefore, no additional whitespaces must be entered to the right and to the left of the separating commas. Since the standards are separated by commas, they must not contain any commas themselves.

## Pipeline Tools

### Merge pipelines



Use the new Merge pipelines  function to merge several pipelines into one pipeline. You can find the function at **Plant Engineering > Plant Engineering Tools > Change  > ....**

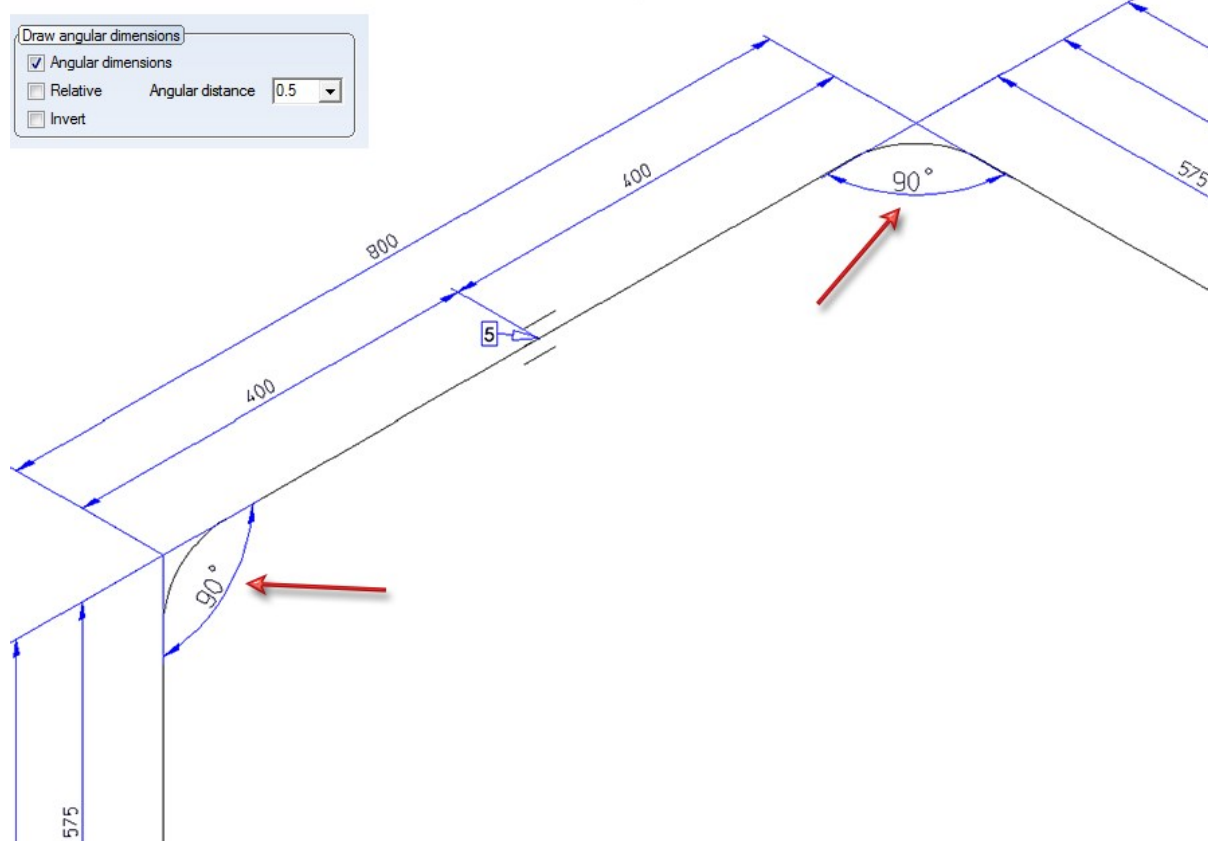
## Isometry/Pipe Spool Drawing

### Dimensioning of bent pipes

Owing to the 2-D projection of isometric drawings it can sometimes be difficult to see which angle is located between two straight pipe sections of a bent pipe. Therefore, a dimensioning of the respective angles between those pipe pairs makes sense. Previously, this could only be done manually, using the common angle dimensioning functions. To facilitate the processing of pipes on bending machines, HiCAD 2016 SP2 now also supports an automatic generation of angular dimensions between all straight pipe segments of a bent pipe. To facilitate pipe

processing on bending machines, HiCAD 2016 SP2 offers the the automatic generation of angular dimensions between all straight pipe segments of a bent pipe. The **Automatic dimensioning** tab of the **Isometry: Settings** dialogue now offers the new **Draw angular dimensions** area with various setting options for this purpose. To activate the automatic angular dimensioning, tick the **Angular dimensions** checkbox.

The following image shows two automatically generated angular dimensionings.



Use the other settings beneath **Draw angular dimensions** to determine the distance and the orientation of the drawn angles.

### Subdivision of isometry - Visualisation of already existing subdivisions



If you call the Specify division function in an isometry that has already been subdivided into several drawings, the corresponding division points will be highlighted (Marking colour 3) as soon as the cursor is close enough at the pipeline to be processed.

### Placing of text objects

Text objects are additional, individual annotations in the isometry. They enable you to place property information or comments directly next to parts. Previously, parts such as straight pipes could obtain additional annotations within the isometry by selecting the part and typing the desired text for the text tag into a dialogue box that also offered some font parameter selections. This tag was always placed centrally beside the pipe.

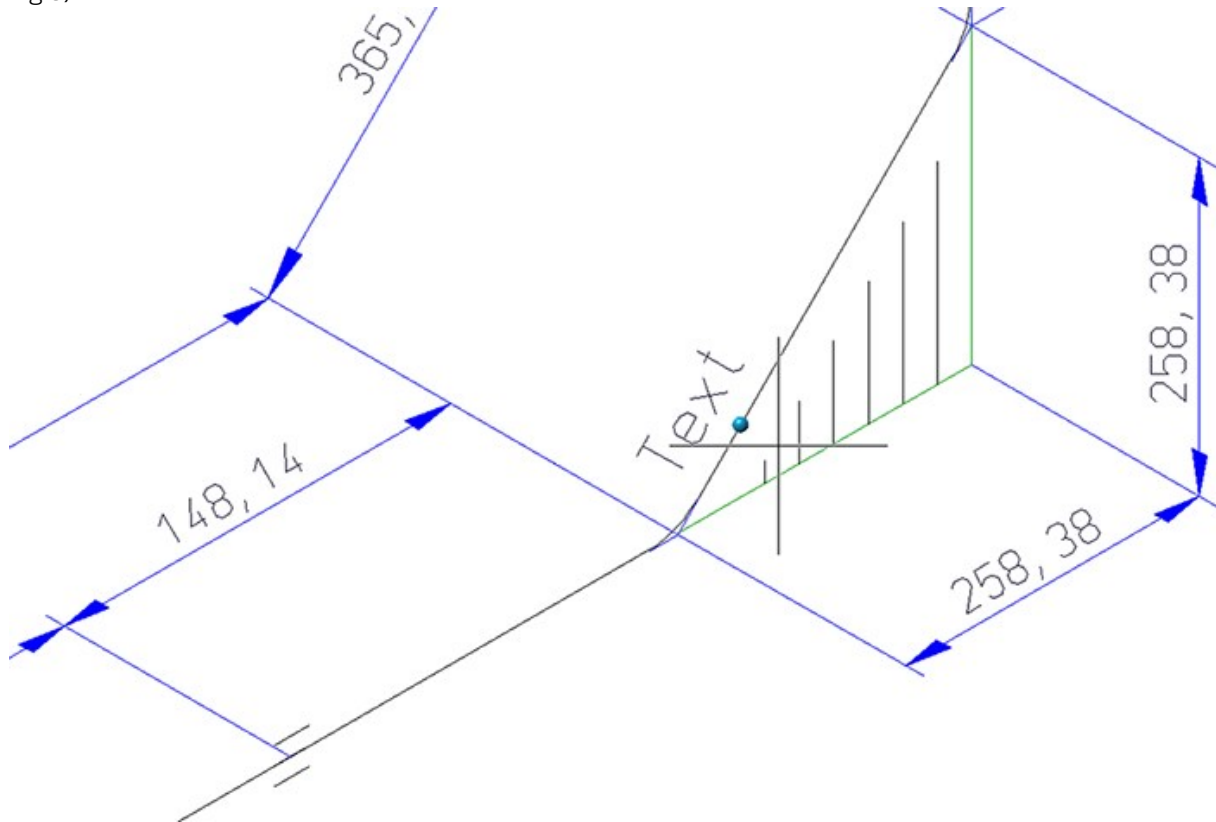


As of SP2 the Create assigned text object function offers the following, additional options for text placement:



- **Selection acc. to part** (Default setting after function call)

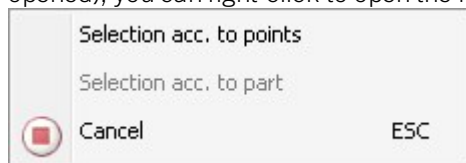
Here the text object can be placed at an arbitrary point beside the selected part. Move the cursor over the part. The location where the text object is to be created will be marked with a coloured sphere (Special colour Marking 3).



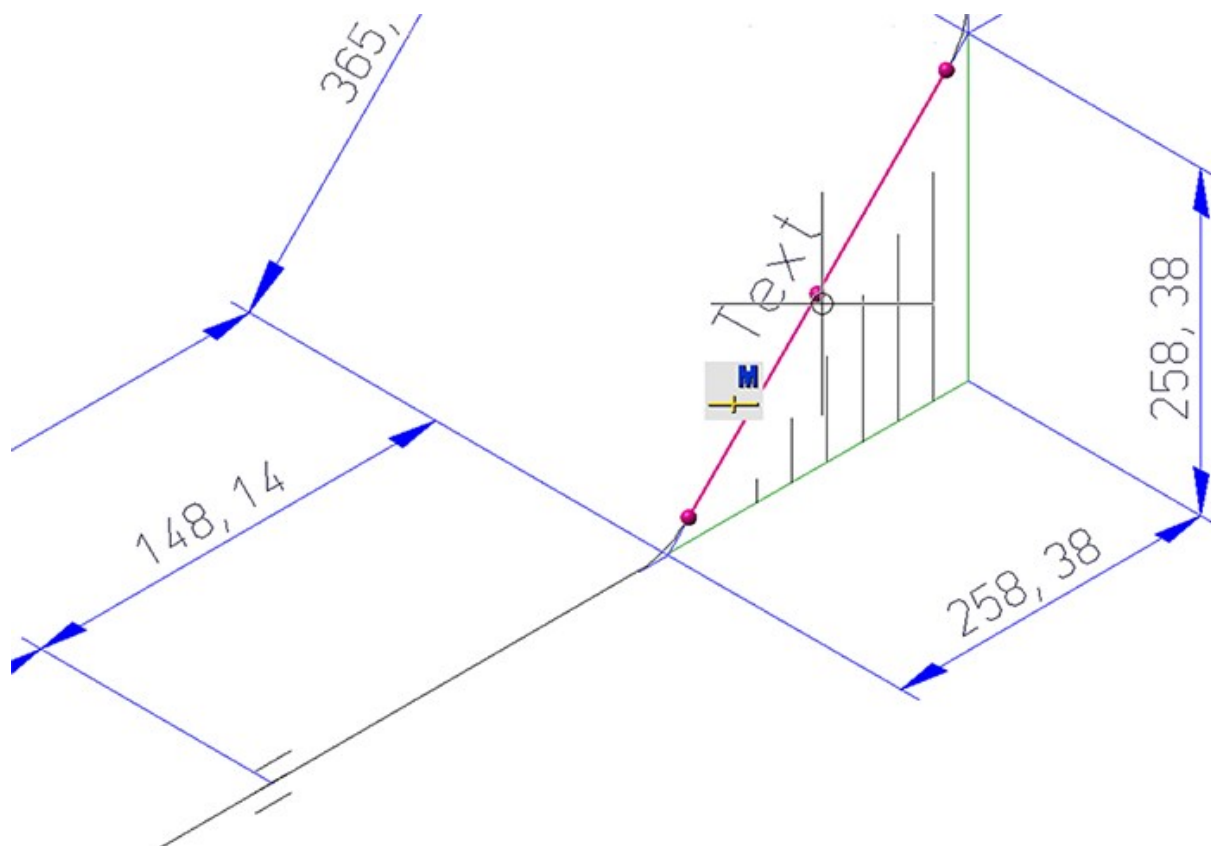
Click to apply a marked point. The **Item number** dialogue window (see 2.) will then be displayed.

- **Selection acc. to points**

As long as you are asked to place the text tag (i.e. as long as the **Item number** dialogue window has not been opened), you can right-click to open the following context menu.



There you can find the **Selection acc. to points** function. This function allows you to specify the position of the text object with the help of the HiCAD point options. A possible example would be the placing of the text object next to the centre of a pipeline. For this to happen, you need to move the cursor near the desired pipe; the HiCAD Autopilot will then automatically suggest point option **M**. Confirm with a mouse click. After calling the function, the part that is nearest to the current cursor position will be highlighted (Special colour Marking 1).



You can now specify any point using the Autopilot or the Point options. This point needs not necessarily be located on the part. After specifying the point, the **Item number** dialogue window will be displayed. If you want to go back to the **Selection acc. to part** mode before specifying the point, right-click and choose the function in the context menu.

## Isometry settings

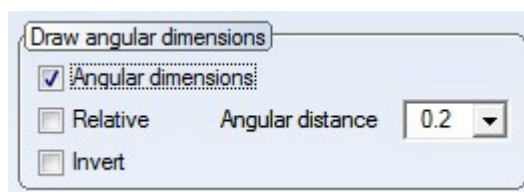
### Texts/Lines

The distance of a text object without leader line (reference line) can now also be preset. The **Texts/Lines** tab of the **Isometry: Settings** dialogue window now offers an additional option for this purpose:



### Automatic dimensioning

To facilitate pipe processing on bending machines, HiCAD 2016 SP2 supports the automatic generation of angular dimensions between all straight pipe segments of a bent pipe. The **Automatic dimensioning** tab of the **Isometry: Settings** dialogue now offers the new **Draw angular dimensions** area with various setting options for this purpose:



## Service Pack 1 2016 (V 2101)

### Working with HELiOS

#### **Redesigned SML masks**

In the article data search masks, the tabs for the Plant Engineering part classification via HELiOS have had a redesign.

Example - Branch, round

**Find article**

Search conditions

Article Classification **Branch, round**

Connection 1

Nominal diameter: 50.000000 mm  inch

Outer diameter:

Connection type:

Connection 2

Nominal diameter:  mm  inch

Outer diameter:

Connection type:

Connection 3

Nominal diameter: 50.000000 mm  inch

Outer diameter:

Connection type:

Pipe part properties

Angle:  $\geq 89.999000$

Wall thickness:

Schedule:

Pressure:

Fitting





Preferred type:

Accessory set:

P+ID symbols

**Search result**

Standard

| Re  | Number   | Index: | Designation                           | Designation | Standard designation |
|---|----------|--------|---------------------------------------|-------------|----------------------|
|  | TN-06749 |        | +GF+ T-Stück S 130, red. 2/3 EN 10242 |             | EN 10242 S130R23     |
|  | TN-06750 |        | +GF+ T-Stück S 130, red. 2/3 EN 10242 |             | EN 10242 S130R23     |
|  | TN-06751 |        | +GF+ T-Stück S 130, red. 2/3 EN 10242 |             | EN 10242 S130R23     |
|  | TN-06752 |        | +GF+ T-Stück S 130, red. 2/3 EN 10242 |             | EN 10242 S130R23     |

Number of data records: 4

Find OK Cancel

## Insertion of parts and components

### Free placing of loose flanges

In previous versions, loose flanges could only be placed through an automatism during automatic placing of parts on guidelines. From HiCAD 2016 SP1 onwards you can also add loose flanges to a pipeline manually.

Please note the following:

- The **Connection type** attribute of the HELIOS article search mask is now editable. The default value of the search field or Default-Wert für das Suchfeld **Connection type** is 20000 and originated from the file **sml\_flansch.msk** in a standard installation. As loose flanges have the Connection type 20100, they will only be found if the search criteria will be adjusted accordingly.
- The manual insertion will only work if the guideline mode has been deactivated in the Plant Engineering settings.
- As a loose flange can only be usefully connected at the end of a pipe part, you need to select, the **Connection 1 to target connection** option via the context menu (right-click) during insertion.

**Find article**

Search conditions

Article Classification Flange, round

Connection 1

Nominal diameter: 50.000000 mm inch

Connection type: 20100

Connection 2

Connection type:

Pipe part properties

Pressure:

Fitting

Preferred type:

Seal:

Accessory set:

P+D symbols:

Search result

| R# | Number    | Index | Designation              | Desig | Standard d   | Part type | Release sta | Creation d |
|----|-----------|-------|--------------------------|-------|--------------|-----------|-------------|------------|
|    | SN-035526 |       | Plate flange for welding |       | EN 1092-1/01 |           | In Progress | 19.04.2016 |
|    | SN-035552 |       | Plate flange for welding |       | EN 1092-1/01 |           | In Progress | 19.04.2016 |
|    | SN-035582 |       | Plate flange for welding |       | EN 1092-1/01 |           | In Progress | 19.04.2016 |

Number of data records: 8

Find OK Cancel

### Insertion of hollow body nozzles into solid body vessels

When inserting a nozzle into a vessel, it was necessary in previous versions to select the type of representation, i.e. a hollow body or a solid body, for the vessel in the Plant Engineering Settings. Now you have the option to select the

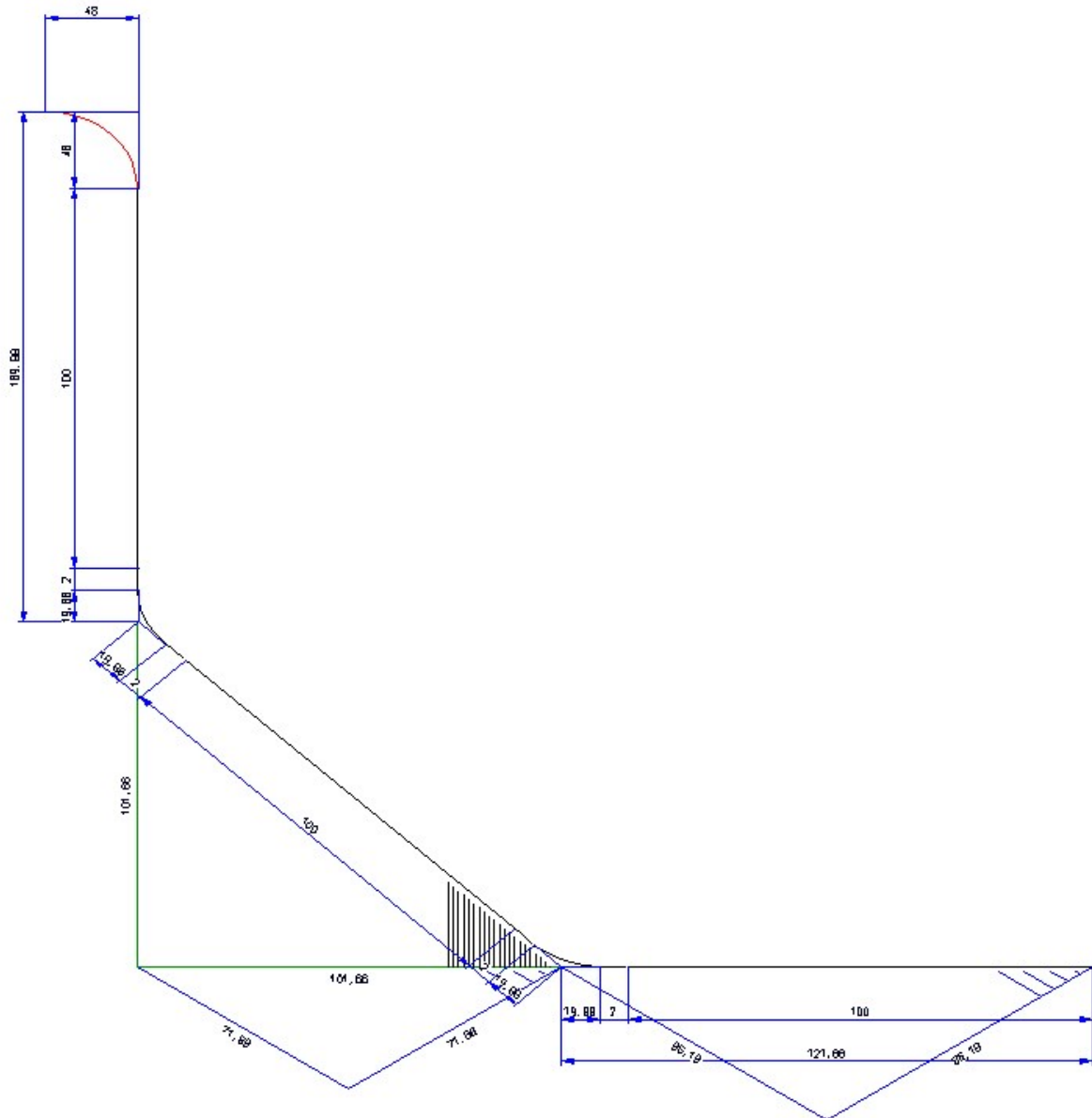
desired type of representation for the nozzle.

## Pipeline isometry

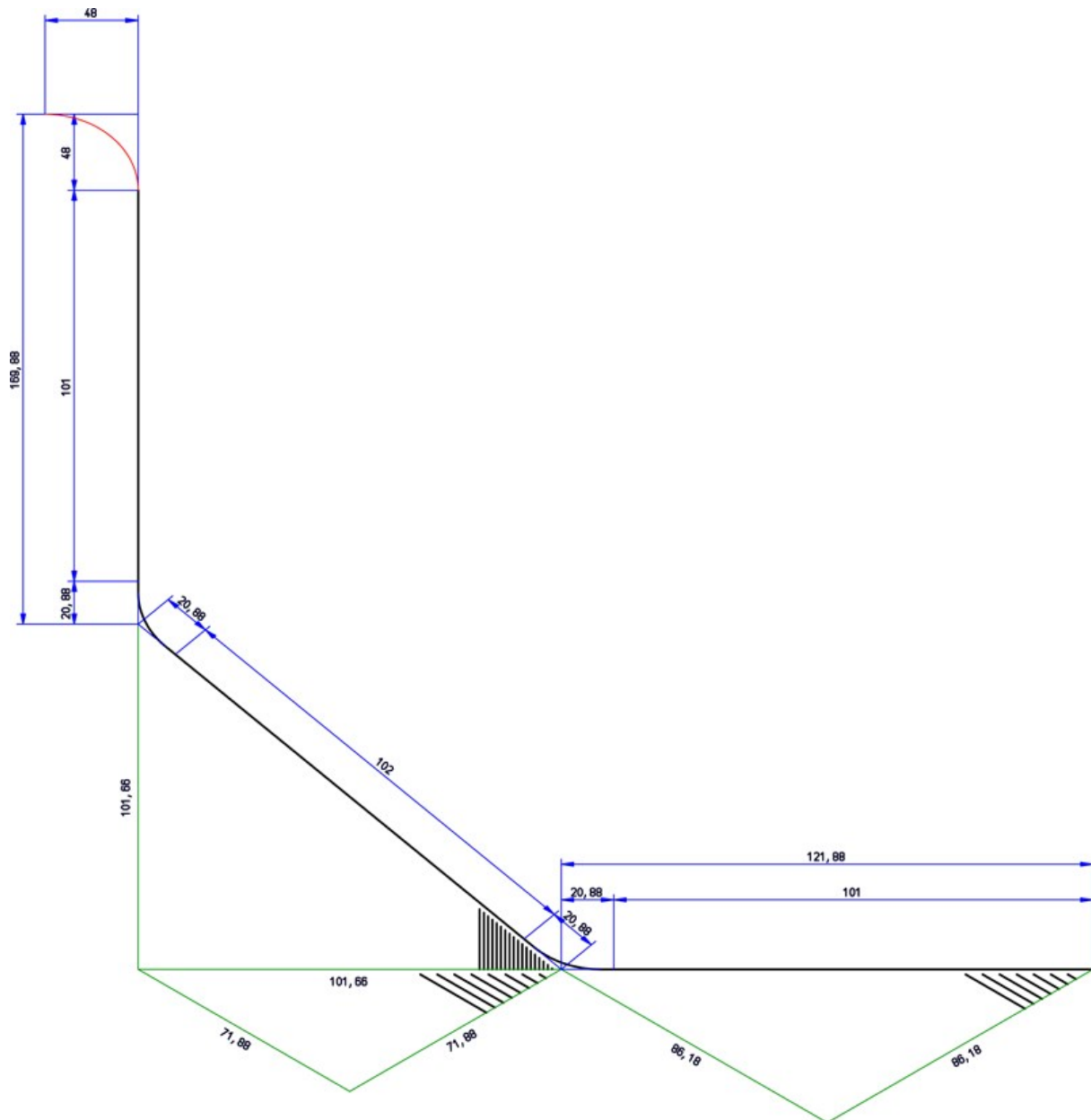
### Improved readability

When generating pipeline isometries, HiCAD now avoids collisions of dimensionings and rise triangles if possible, thus further improving the readability of the isometries.

*Example - Before HiCAD 2016 SP1*




Example - as of HiCAD 2016 SP1



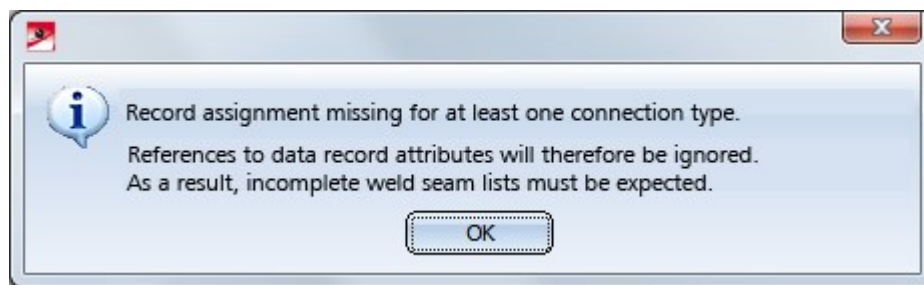
## Message issued if data record assignment for connection types is missing

If a HELIOS data record has not been assigned to all connection types when you call the **AutoGenerate drawing**


(Iso)  function, the references to data record attributes will be ignored. This may in particular lead to incomplete weld seam lists.

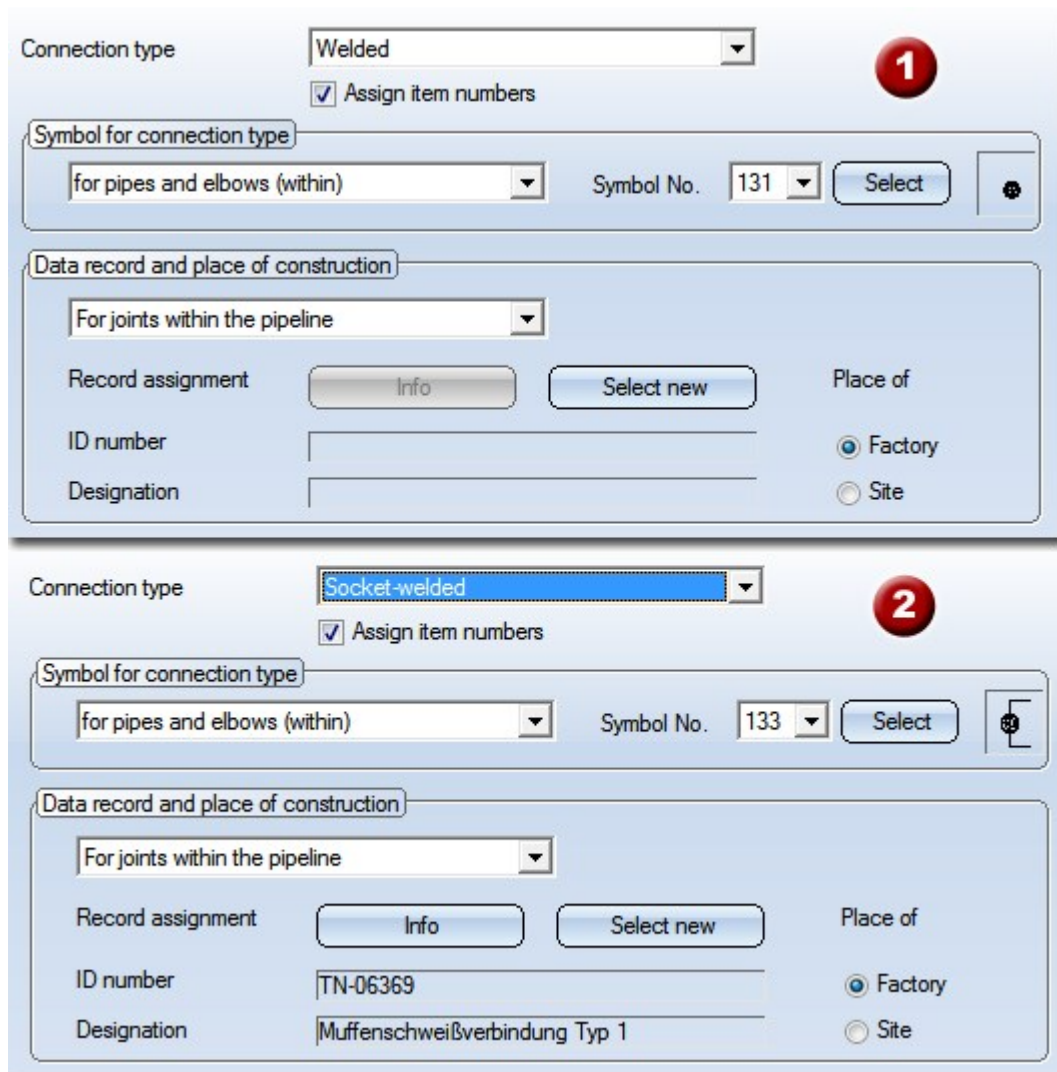
In such cases, a corresponding message will be displayed:





Click **OK** to continue with the generation of the isometry.

Define missing assignments on the **Connections** tab of the **Isometry: Settings**  dialogue window.



**Connection type** Welded 1

☒ Assign item numbers

**Symbol for connection type**

for pipes and elbows (within) Symbol No. 131 Select

**Data record and place of construction**

For joints within the pipeline

Record assignment Info Select new

ID number

Designation

Place of ☒ Factory  
☐ Site

---

**Connection type** Socket-welded 2

☒ Assign item numbers

**Symbol for connection type**

for pipes and elbows (within) Symbol No. 133 Select

**Data record and place of construction**

For joints within the pipeline

Record assignment Info Select new

ID number TN-06369


Designation Muffenschweißverbindung Typ 1

Place of ☒ Factory  
☐ Site

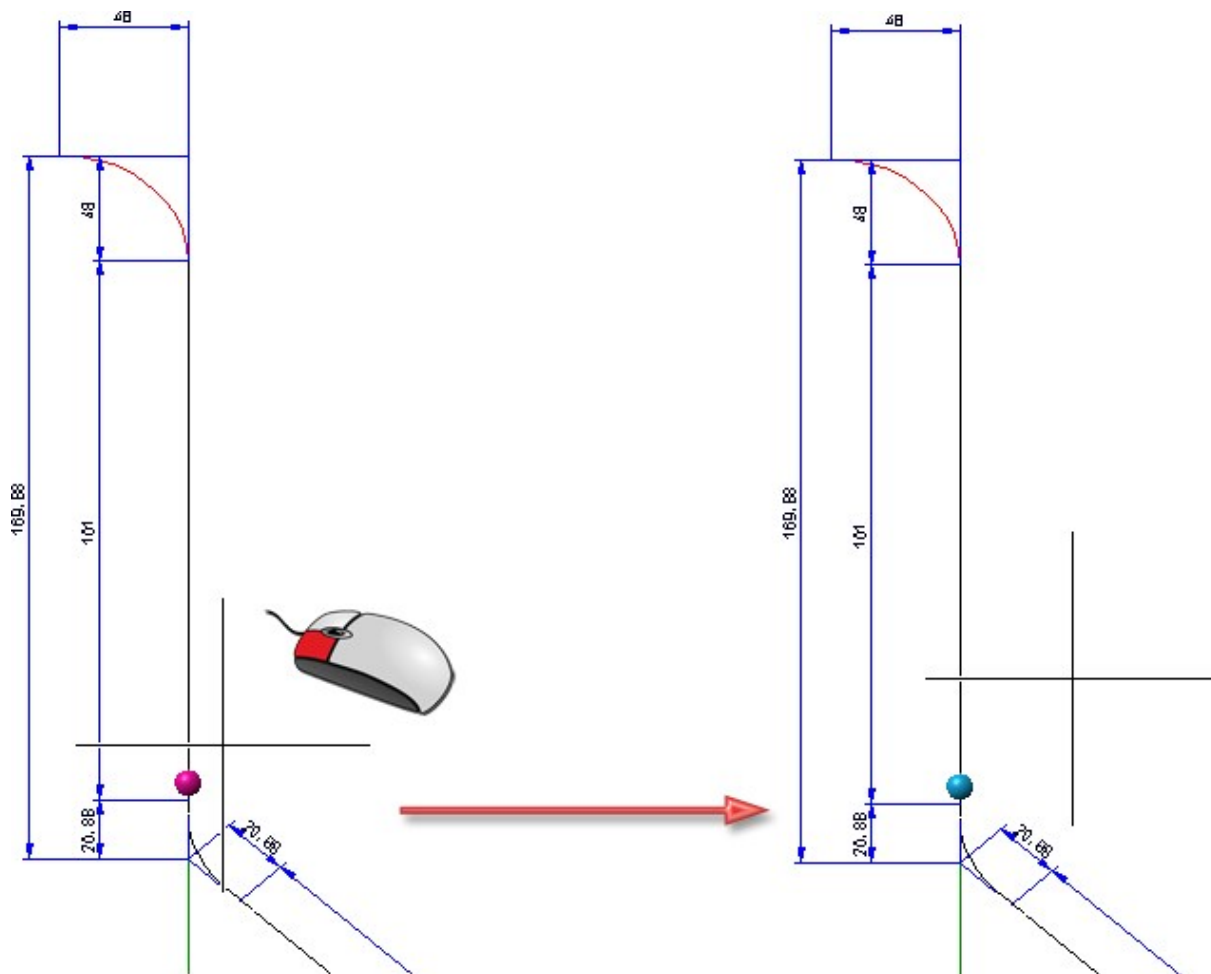
Connection type - Assigning of Data record and place of construction: (1) no assignment yet, (2) with assignment

### Divide isometry - Visualisation of division points



When you use the **Specify division**  function, the division points will now be graphically visualized. When you move the cursor over the isometry, possible division points will be marked in a different colour (Marking colour 1).

Left-click to accept a marked point as a division point. When you move the cursor away from the division point, its colour will change (Marking colour 3).





If an already selected division point is selected again, the related division will be removed.

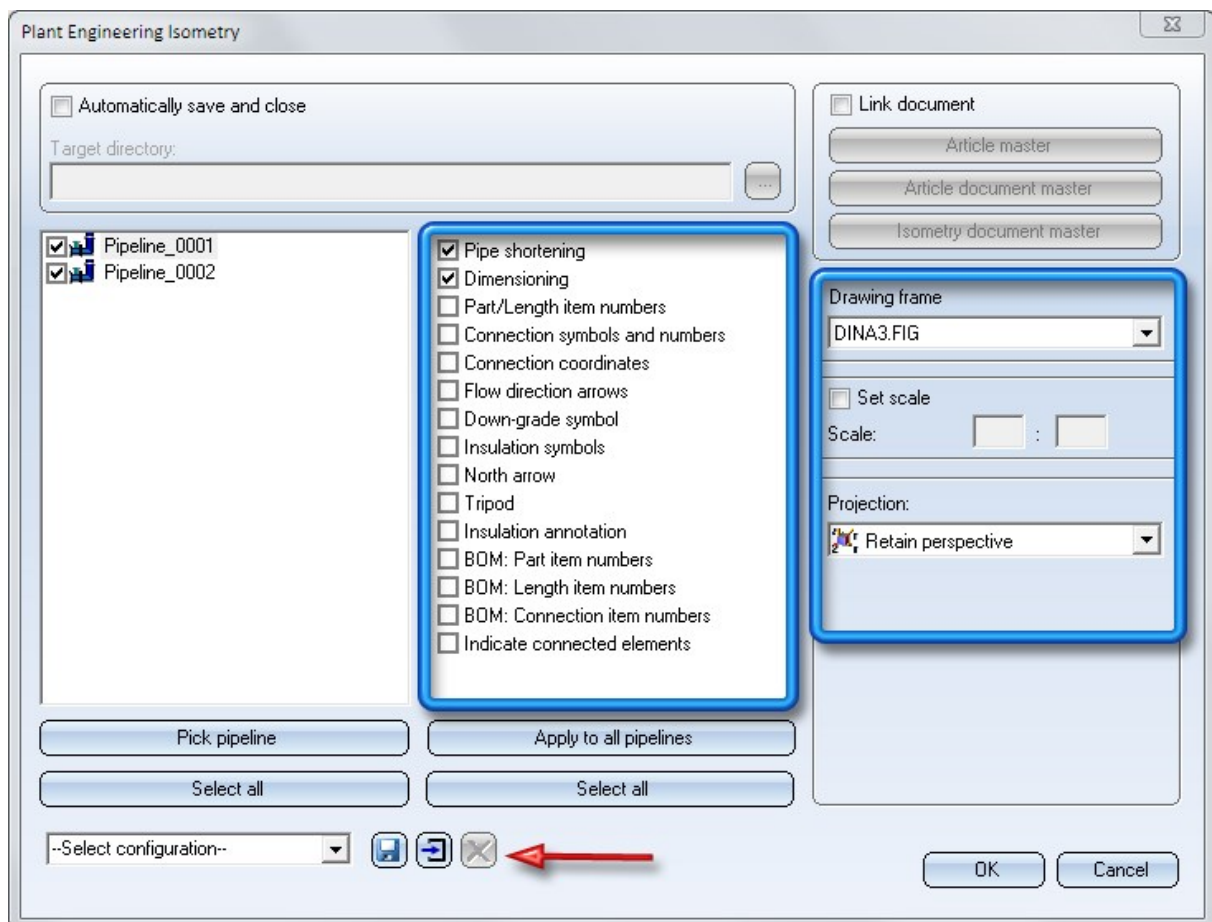
Right-click or press ESC to cancel the function without applying the selected division points. Press the middle mouse button to apply the selected division points and exit the function.

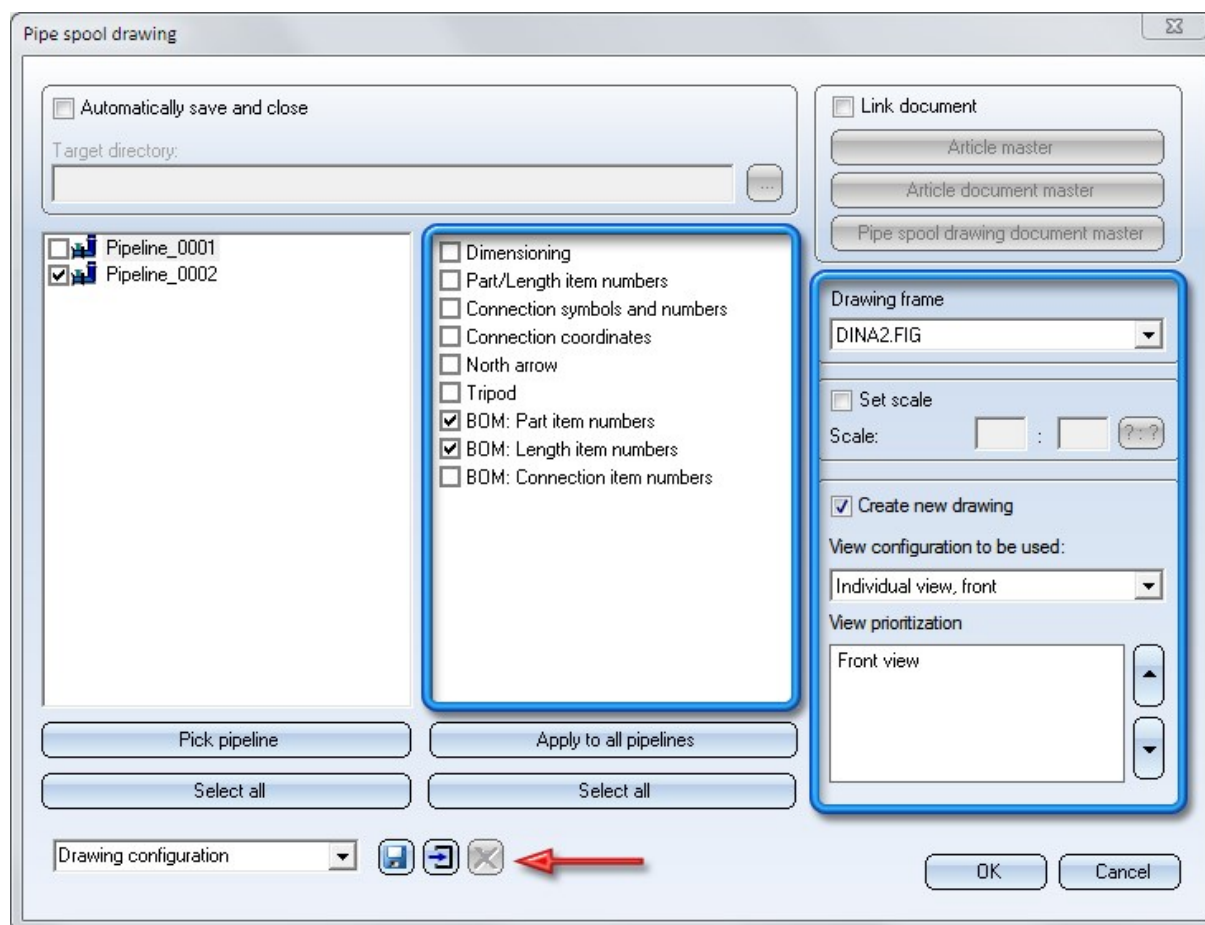
## User-defined configuration for Isometry and Pipe Spool Drawing

In previous versions, the settings in the start dialogue of Isometry and Pipe Spool Drawing creation were saved for each pipeline in the drawing. Besides this drawing configuration, HiCAD now offers the additional option to save and re-use user-defined settings as XML files independent of the drawing.

For this purpose, the dialogue windows of the functions AutoGenerate drawing (Iso)  and Generate pipe spool drawing  functions have been expanded accordingly.

The Isometry / Pipe Spool Drawing settings shown in blue frames in the below images can now be saved and re-used as configurations.






## Drawing configuration

The settings specified for this pipeline will be saved, together with the drawing, for each pipeline of the drawing when the isometry or the pipe spool drawing is saved. These are the so-called drawing configurations.


## User-defined configuration

Besides the drawing configurations, you can also specify your own settings, independent of the the drawing, as XML files and reuse them. To do this, click the **Save configurations**  icon and enter the name of the XML file. The following HiCAD sub-directories have been preset for the configuration files:


- **PlantParts\IsoConfig** for isometries and
- **PlantParts\SpoolConfig** for pipe spool drawings.

User-defined configurations can be reused at any time via the selection box.

## Default configuration

And, finally, you have also the option to define a default configuration. To do this, specify the desired settings and then click the **Save as default configuration**  icon. The current settings will then be used as default settings for the generation of isometries and pipe spool drawings. The file **DefaultConfig.xml** is located, like the user-defined configurations, in the HiCAD sub-directory **PlantParts\IsoConfig** for isometries or **PlantParts\SpoolConfig** for pipe spool drawings, respectively.

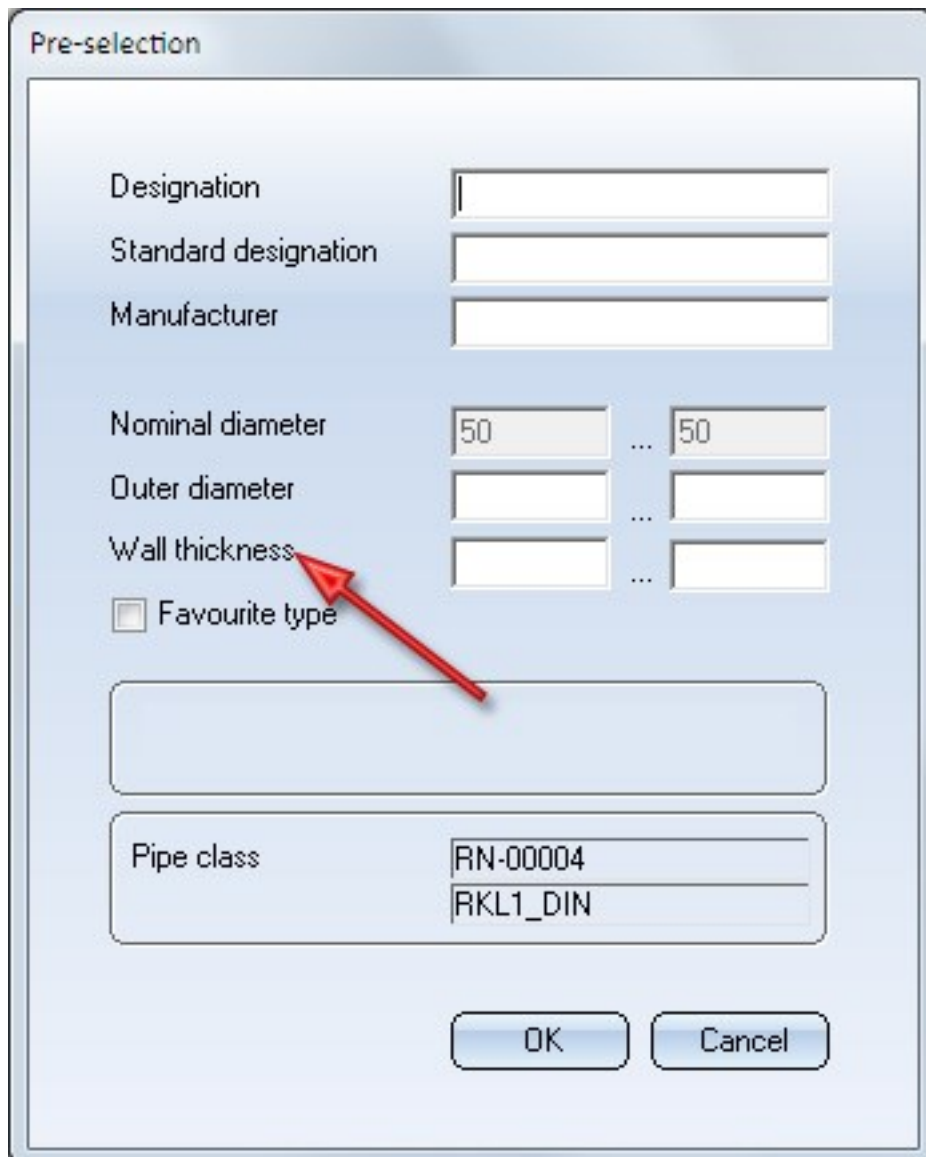
## Delete configurations

User-defined configurations can be deleted by choosing the corresponding configuration from the selection box and clicking the **Delete configuration**  icon. Drawing configurations and default configurations cannot be deleted with this function.

## Settings

### Pre-selection

The option to restrict part search by a predefined wall thickness has been made available again. For this purpose, the Pre-selection dialogue window has been expanded accordingly.



The image shows a 'Pre-selection' dialog box with the following fields and controls:

- Designation**: A single-line text input field.
- Standard designation**: A single-line text input field.
- Manufacturer**: A single-line text input field.
- Nominal diameter**: Two numeric input fields, both containing '50', separated by an ellipsis (...).
- Outer diameter**: Two empty numeric input fields, separated by an ellipsis (...).
- Wall thickness**: Two empty numeric input fields, separated by an ellipsis (...). A red arrow points to this field.
- ☐ **Favourite type**: A checkbox.
- Pipe class**: A list box containing 'RN-00004' and 'RKL1\_DIN'.
- Buttons**: 'OK' and 'Cancel' buttons at the bottom right.

## Major Release 2016 (V 2100)

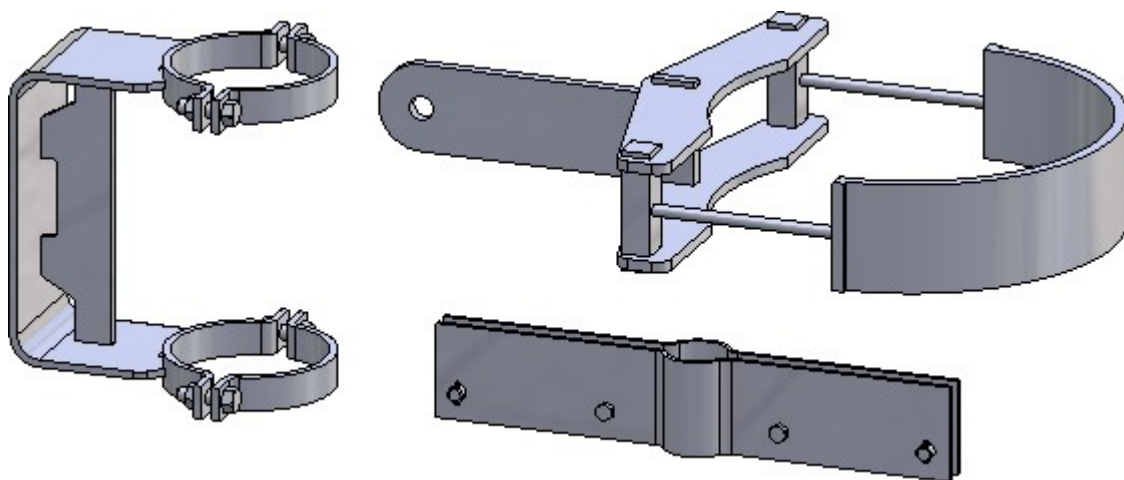
### New part variants

#### Lisega Pipe clamps

The following Lisega Pipe clamps of the product group 4 are available as part variants (VAA files):

- Lisega Horizontal clamp Type 42
- Lisega Horizontal clamp Type 43
- Lisega Horizontal clamp Type 44
- Lisega Horizontal clamp Type 44 with strap
- Lisega Horizontal clamp Type 44 with belt
- Lisega Pipe way Type 49,
- Lisega Pipe way Type 49 for high temperatures
- Lisega Vertical clamp Type 45,
- Lisega Vertical clamp Type 46

The file LISEGA\_Pipe\_Supports\_PG4.lst (in the HiCAD subdirectory PLANTPARTS) contains a list of all new LISEGA part variants.



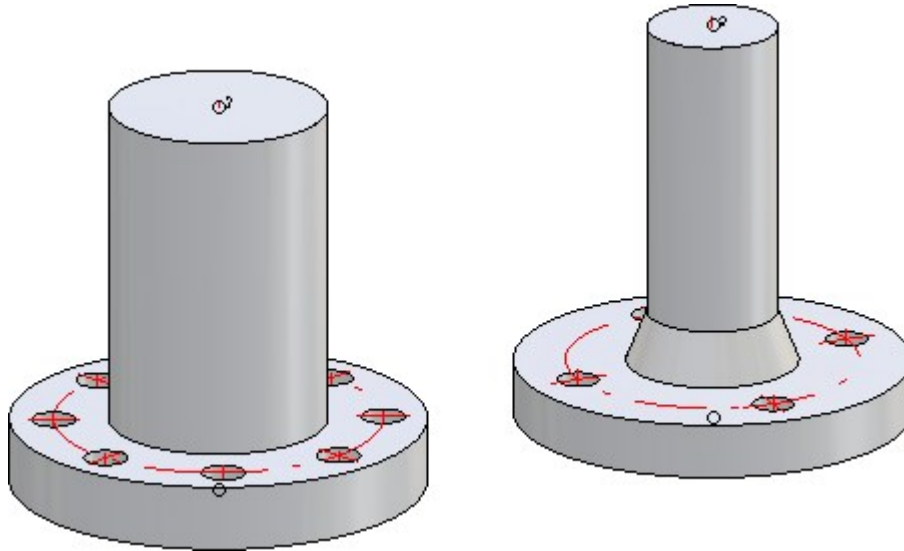
In case of an update, these parts may be entered into the database or the HiCAD catalogue, respectively.

#### Further DIN and ASME Flange connectors

- Flange connector 150 ASME B16.5
- Flange connector 1500 ASME B16.5
- Flange connector 2500 ASME B16.5
- Flange connector 300 ASME B16.5
- Flange connector 400 ASME B16.5
- Flange connector 600 ASME B16.5

- Flange connector 900 ASME B16.5
- Flange connector DIN 28115 PN10
- Flange connector DIN 28115 PN25

The file ASME\_16.5-LWN\_Nozzles.lst (in the HiCAD subdirectory PLANTPARTS) contains a list of all new ASME Flange connectors, the file DIN\_28115\_Nozzles.lst a list of the new DIN Flange connectors.

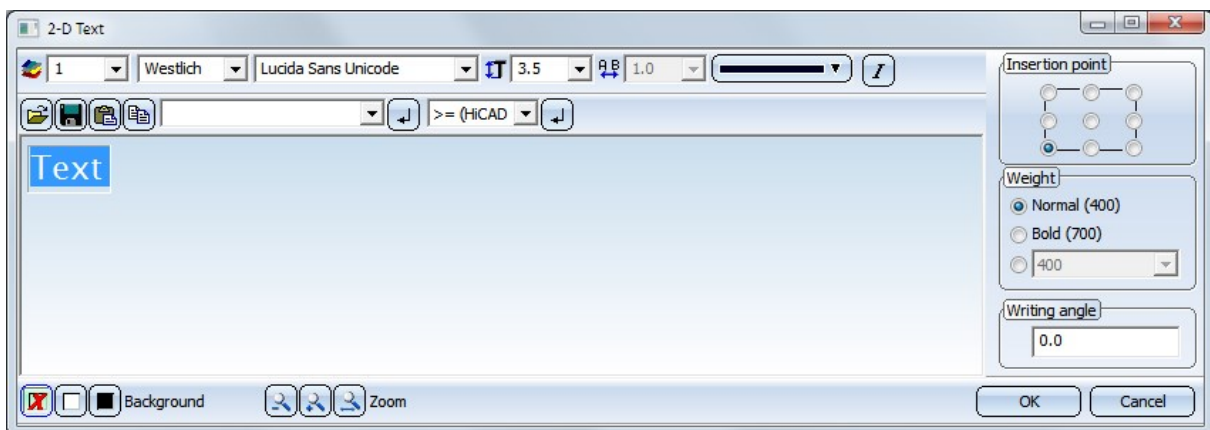


In case of an update, these parts may be entered into the database or the HiCAD catalogue, respectively.

## Isometry / Pipe spool drawings

### Text parameters - Changed dialogue

In all Setting dialogues offering a **Text parameters** button (e.g. **Isometry/Pipe Spool Drawing > Settings > Texts/Lines** or **... > Lists**), the following dialogue will now be used:





## Revised rise triangles functionality

Amongst other things, pipeline isometries in the Plant Engineering module serve the purpose of visualizing complex, three-dimensional pipeline constructions in such a way in 2-D, that this visualisation still gives a good impression of the spatial pipeline routes. If a straight pipeline does not run parallel to a coordinate axis, rise triangles can be inserted to convey a better impression of the spatial position of such a pipeline. Here, you distinguish between vertical and horizontal rise triangles. To improve the visual representation of pipeline isometries, the **Automatic dimensioning** tab in the **Isometry: Settings** dialogue offers significantly enhanced options for for rise triangles.

**Isometry: Settings**

**Automatic dimensioning** | Texts/Lines | Optimise positioning | Connections | Lists

**Dimension line distances**

1st: 25 mm    Further dimension: 20 mm    **Rise triangles: 25 mm**

☐ Only write dimension figure next to rise triangles

☐ Determine reference line acc. to part diameter

**Dimension figure height**

2.5 mm

**Vertical rise triangles**

☒ Draw triangle    Line parameters: 5.0 85.0

Range of angles: 5.0 85.0

☐ Dimension hypotenuse

☒ Hatch triangle

Hatching line spacing: 2.0 mm

Triangle distance: 3.0 mm

Complete hatching

Triangle length: 20 mm

Triangle surface area: 2.0 cm<sup>2</sup>

**Horizontal rise triangles**

☒ Draw triangle    Line parameters: 5.0 85.0

Range of angles: 5.0 85.0

☒ Dimension hypotenuse

☒ Hatch triangle

Hatching line spacing: 2.0 mm

Triangle distance: 3.0 mm

Complete hatching

Triangle length: 20 mm

Triangle surface area: 2.0 cm<sup>2</sup>

**Position of dimension lines**

☐ Always in vertical plane

☐ Dimensions in all views

☐ Repeat dimensions

**Dimension types**

☒ Part dimensions

☐ Dimensions for all parts

☐ Also dimension weld gaps

☒ Dimensions for points of action

☒ Corner dimensions

☒ Dimensions for points of action

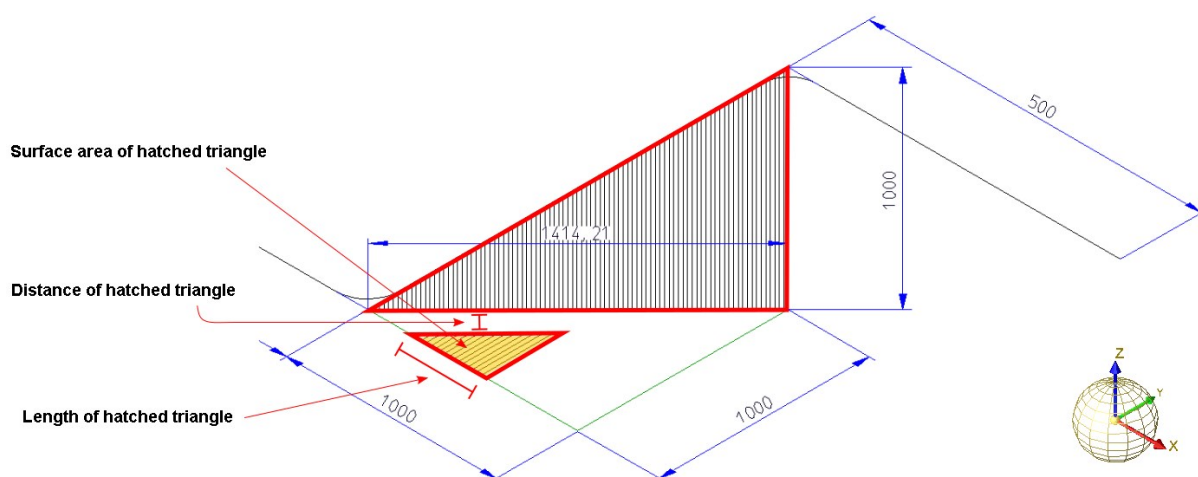
Default

OK Cancel

## Hatchings of rise triangles

Previously, rise triangles were always drawn with complete hatching. As a possible result, the rise triangle may be optically too prominent in the drawing. Also, complete hatchings can often overlap with other drawing elements: For example, the vertical rise triangle in the image below overlaps with the length dimensioning of the triangle cathetus.

You have now the option to give rise triangles a partial hatching. For example, the horizontal rise triangle in the image below has a partial hatching. The hatching itself has a triangular form, which is mathematically similar to the outer, unhatched green rise triangle further down the page. The size of the hatched triangle can either be determined by specifying the **Triangle length** or the **Triangle surface area** - separately for horizontal and vertical rise triangles. If you select the **Triangle surface area** option for the hatching, the length of the corresponding hatching will be calculated by HiCAD accordingly. The effect of this setting is that an acute angle between hypotenuse and start cathetus leads to a longer hatching distance than an obtuse angle. Visually, the hatching in both rise triangles are equally intense. The result is a significantly enhanced impression of the three-dimensional course of the pipeline.



Vertical and horizontal rise triangle

The vertical and horizontal hatching angles are automatically calculated to provide a best possible visual representation. However, if you prefer angles deviating from the automatically calculated angles, you can specify the desired difference angle in the Configuration Editor, at **Plant Engineering > Isometry and Pipe spool drawing**:

| Dimensioning                        |   |
|-------------------------------------|---|
| Horizontal hatching angle deviation | 0 |
| Vertical hatching angle deviation   | 0 |

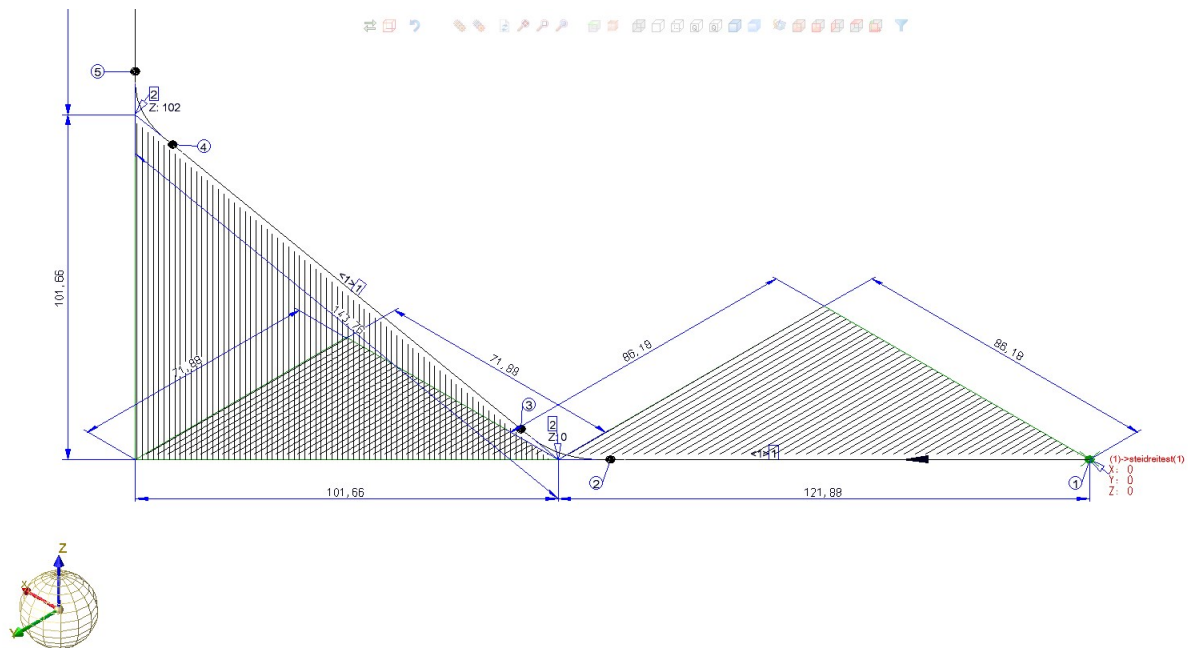
## Offset triangles in front of pipelines if possible

If a straight pipe is not located in one of the three basic coordinates, you can, for the sake of a better spatial representation, insert two rise triangles:

- a horizontal rise triangle in the XY-plane to convey an impression of the horizontal route, and
- a vertical rise triangle within the plane created by the Z-axis and the projection of the straight pipeline onto the XY-plane to convey the vertical route

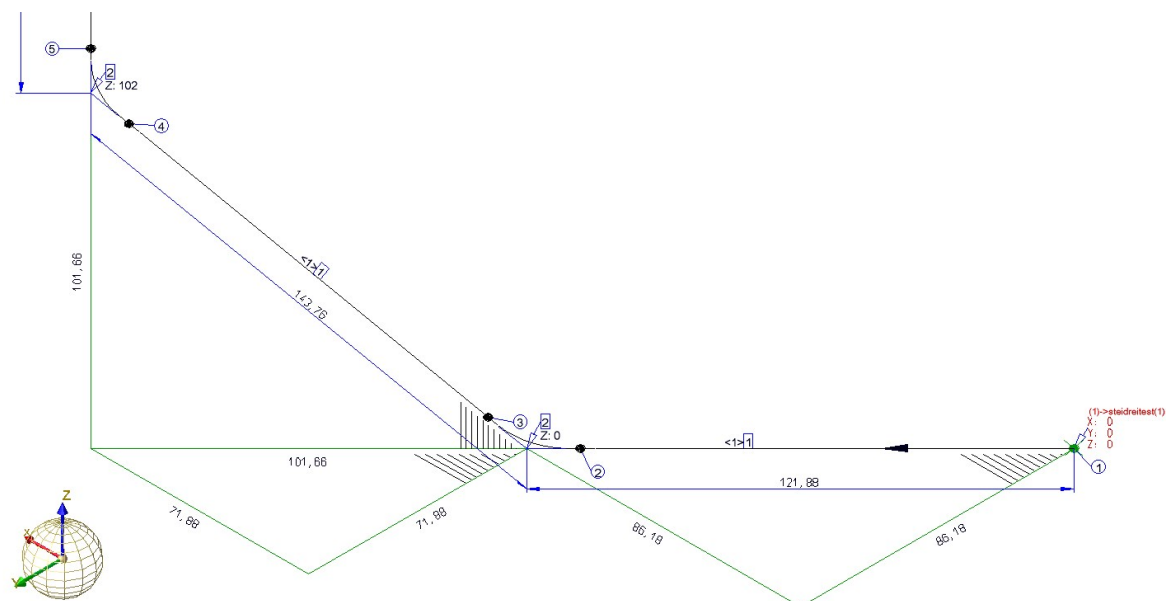
Previously, overlappings of the horizontal and the vertical rise triangles could occur in particular in isometric projections 1 and 4; especially if the decision was made to give a hatching to both rise triangle types, this could result

in a very confusing graphical representation - totally contrary to the original idea of providing a better spatial representation.



Before HiCAD 2016: Unsuccessful, complete hatching and no separate dimensioning

Now, If the two rise triangles overlap, the horizontal rise triangle will be mirrored (flipped) at its hypotenuse, thus avoiding an intersection. If a pipeline isometry has several areas with two rise triangles, the number of all overlaps of rise triangles will be determined in a first step. If more that half of the rise triangles overlap, all horizontal rise triangles in the complete pipeline will be flipped. The advantage of this procedure is that the cathetus between start point and base point of all horizontal rise triangles either runs in the direction of the positive Y-axis or in the direction of the negative Y-axis. Thanks to this consistent orientation, the observer does not need to re-orientate him/herself for each rise triangle and gets a better spatial impression of the situation.

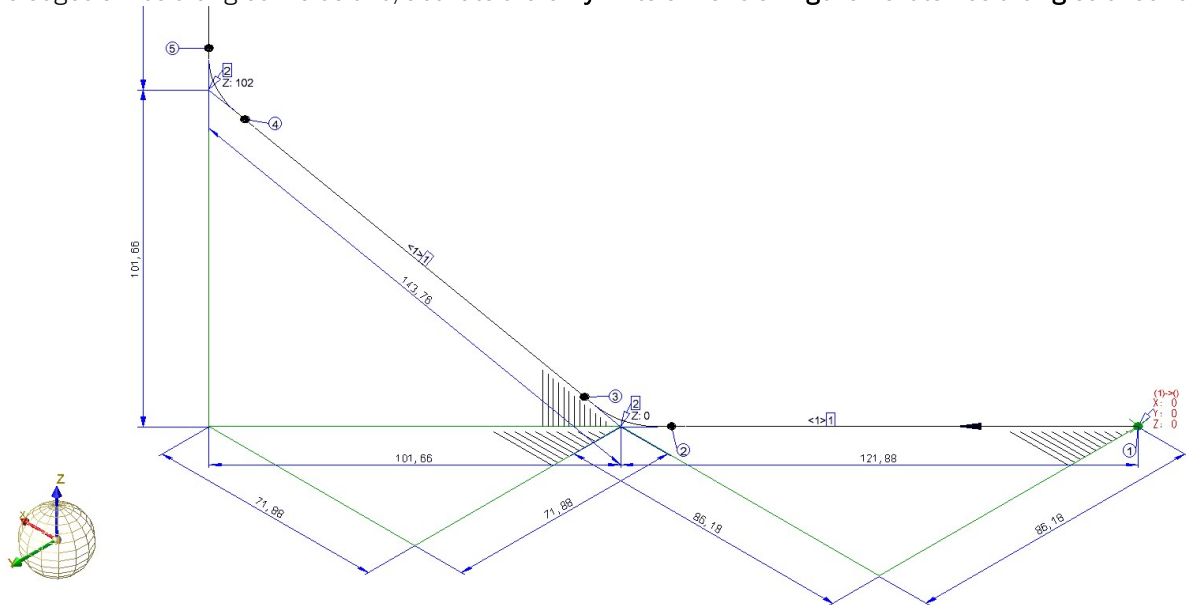


HiCAD 2016: Horizontal triangles flipped, separate dimensioning (no liner dimensions) for rise triangles and implied hatching

### Dimensions directly at triangle; introduction of a separate dimension distance

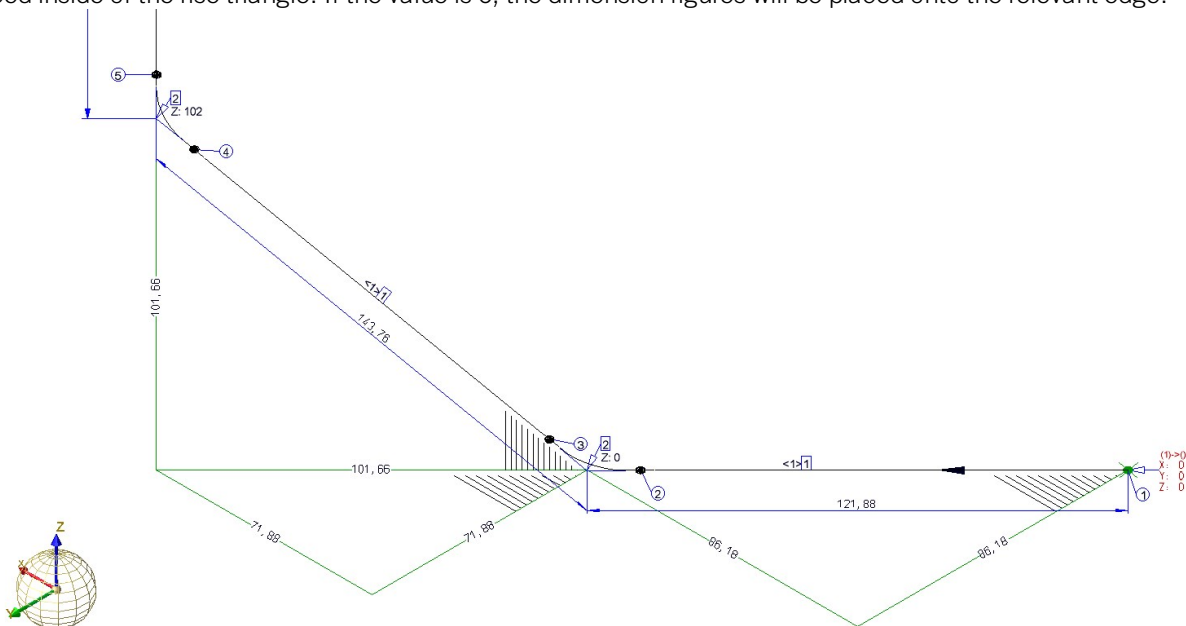
Normally, each of the three edges of a rise triangle obtains a linear dimension. The dimension figure of such a linear dimension indicates the length (in mm) in the corresponding spatial direction. The dimensioning of all rise triangle edges by means of linear dimensions as well as the many other dimensions can sometimes make the representation of pipeline isometries confusing.

To avoid this, HiCAD offers the option to place only dimension figures (instead of complete linear dimensions) next to the edges of rise triangles. To do this, activate the **Only write dimension figure next to rise triangles** checkbox



Linear dimensions on rise triangles - No special dimensioning for rise triangles

When the linear dimensions are removed, the dimensioning of the rise triangles is much clearer, and the overall view is less confusing. The dimension figure is placed centrally next to the relevant edge. The distance to this edge can be entered as a separate dimension distance into the **Rise triangles** input field. If the distance is positive, the dimension figure will be placed outside of the rise triangle, if the distance is negative, the dimension figure will be placed inside of the rise triangle. If the value is 0, the dimension figures will be placed onto the relevant edge.



Rise triangles on edges only with dimension figure

If the **Only write dimension figure next to rise triangles** checkbox is deactivated, linear dimensions will be used for the rise triangles, but with the dimension distance entered in the **Rise triangles** checkbox. If the checkbox is deactivated, and the same dimension distance is set for **Rise triangles** and **1st dimension line**, the behaviour will be the same as in older HiCAD versions (before HiCAD 2100.0).

## BOMs

### New template format for BOMs

The template format for the output of BOMs to Microsoft Excel has been revised. The new templates supplied with HiCAD offer more output options than the old ones. Furthermore, it is now possible to create customized templates for BOMs conveniently via adjustment of an Excel template. Already existing templates in the old format can, of course, still be used.

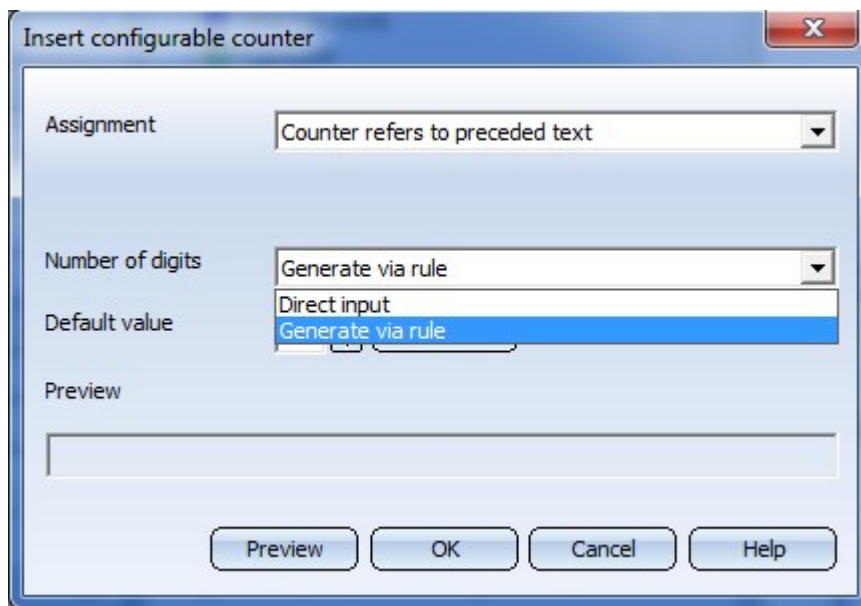
Extensive information on the new BOM templates can be found in the Help of the Report Manager.

## P+ID

### Service Pack 2 2016 (V 2102)


#### Configurable counter

The dialogue for the insertion of configurable counters has been expanded. The **Number of digits** input field now offers the new selection option **Generate via rule**.



The generation expressions with counter (configurable counters) can be used:

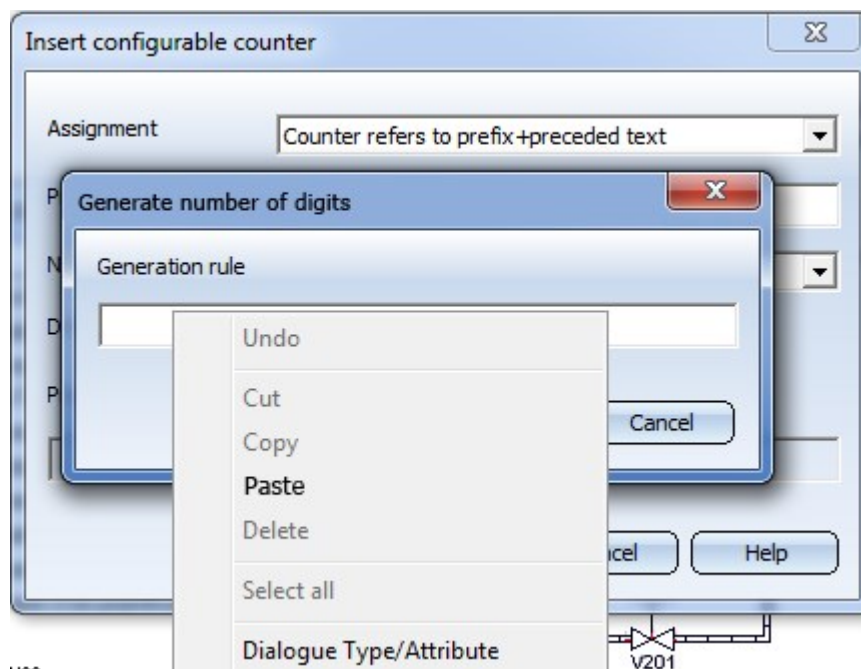
- when editing multiple symbols
- for data mask objects (DMO)
- when editing data masks in the P+ID
- when editing data masks in the Symbol Editor

To open the dialogue, right-click on an input field in a symbol dialogue, and select **Dialogue Type/Attribute** in the context menu. There, double-click the  **Configurable counter** entry.

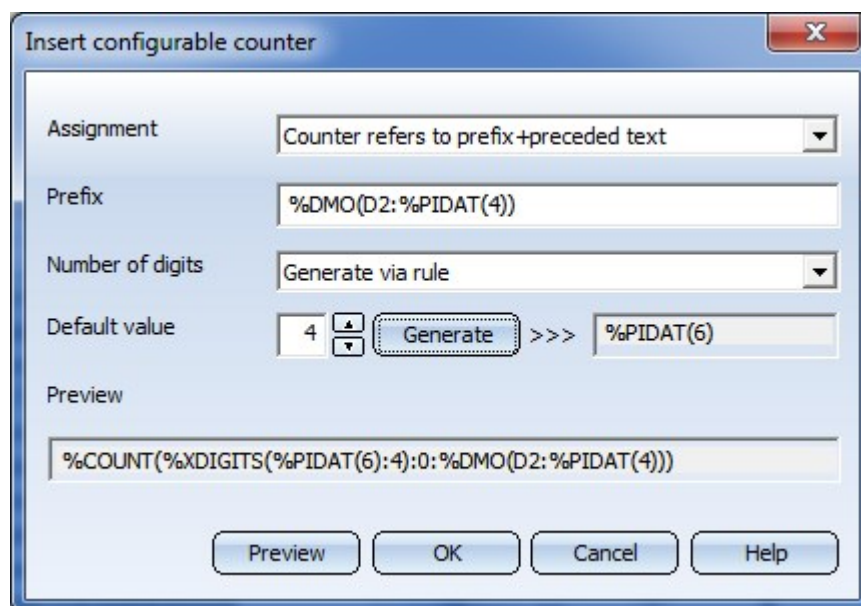
In the **Number of digits** field, select the option **Generate via rule**.

With the option **Generate via rule** you can determine the number of digits of the text key **%XDIGITS**. After selecting this option, click the **Generate** button. In the context menu of the input field in the displayed dialogue window, you can then right-click, choose **Dialogue type/Attribute** and insert a desired Generation rule.



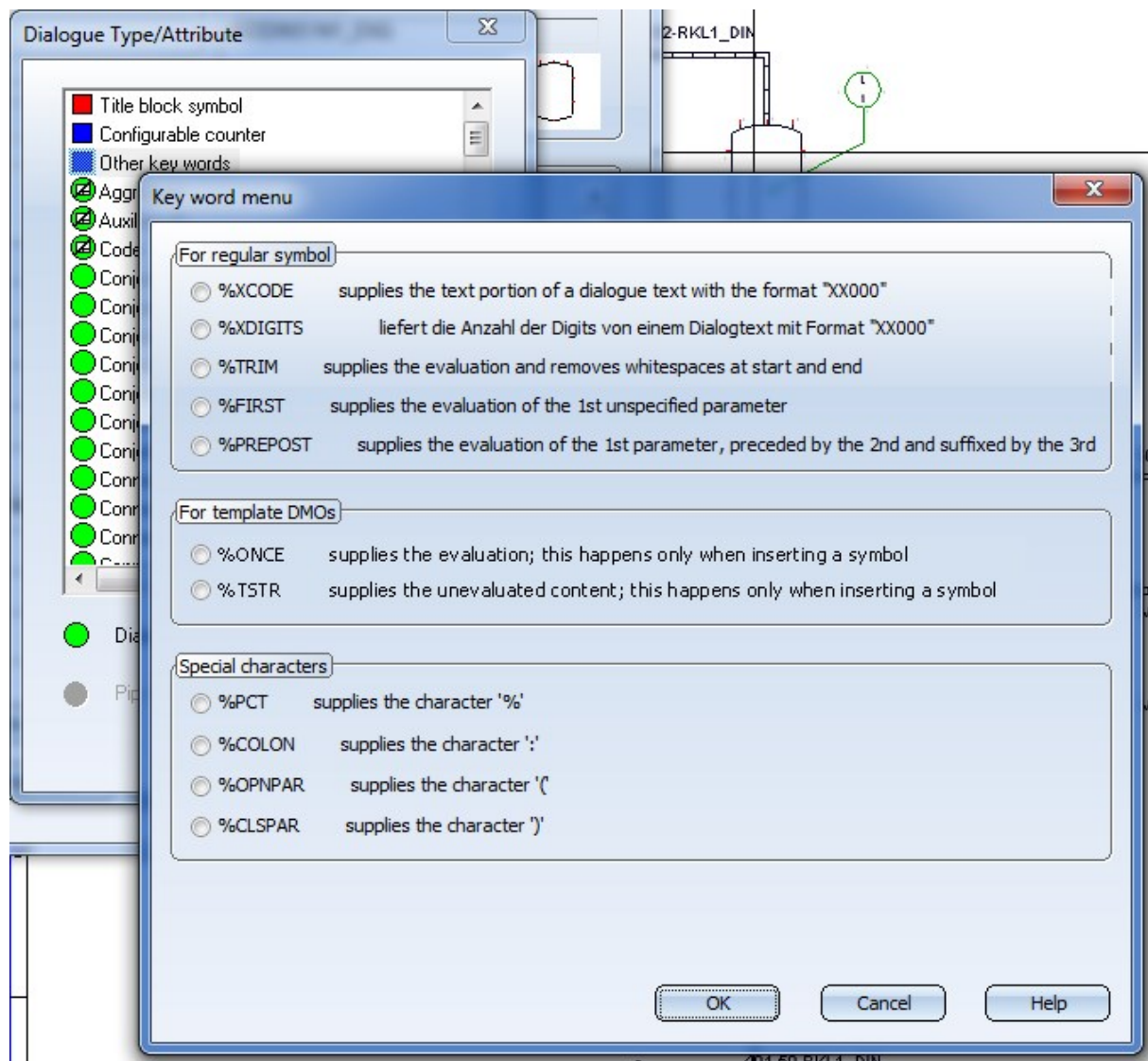


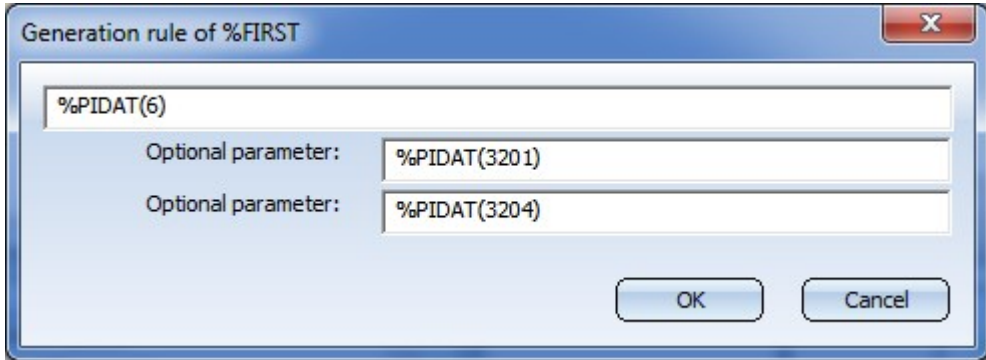
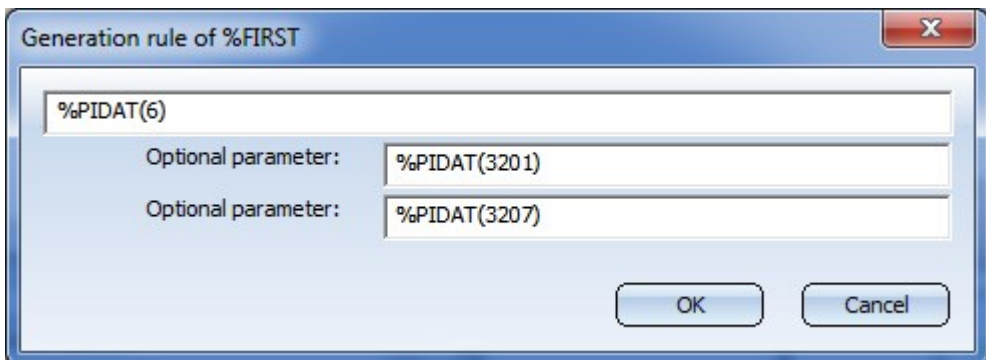
After confirming with **OK**, the rule will be displayed (e.g. %PIDAT(6)).



Further text keys can be called by double-clicking the ■ **Other key words** icon in the **Dialogue Type/Attribute** dialog window.

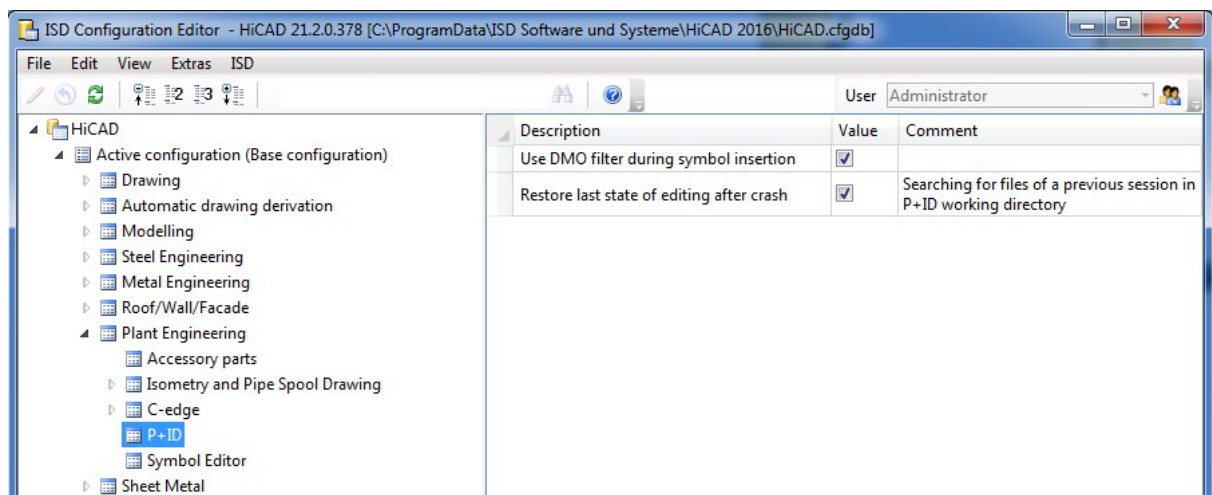




|                                |  |
|--------------------------------|--|
| <b>%XCODE,<br/>%XDIGITS</b>    | After selecting the key word you can call the context menu function <b>Dialogue type/Attribute</b> in the displayed dialogue window.<br>e.g. %XCODE(%PIDAT(6))   |
| <b>%TRIM,<br/>%ONCE, %TSTR</b> | After selecting the key word you can call the context menu function <b>Dialogue type/Attribute</b> in the displayed dialogue window.<br>e.g. %TRIM(%PIDAT(3200))   |
| <b>%FIRST</b>                  | <p>After selecting the key word you can call the context menu function <b>Dialogue type/Attribute</b> in the displayed dialogue window.</p>  <p>e.g. %FIRST(%PIDAT(6):%PIDAT(3201):%PIDAT(3204))</p>     |
| <b>%PREPOST</b>                | <p>After selecting the key word you can call the context menu function <b>Dialogue type/Attribute</b> in the displayed dialogue window.</p>  <p>e.g. %PREPOST(%PIDAT(6):%PIDAT(3201):%PIDAT(3207))</p> |
| <b>%PCT</b>                    | Via this key the character % will be displayed after evaluation.   |
| <b>%COLON</b>                  | Via this key the character : will be displayed after evaluation.   |
| <b>%OPNPAR</b>                 | Via this key the character ( will be displayed after evaluation.   |
| <b>%CLSPAR</b>                 | Via this key the character ) will be displayed after evaluation.   |

## Security prompt after crash

In the Configuration Editor at **Plant Engineering > P+ID** you can specify the procedure after a crash.



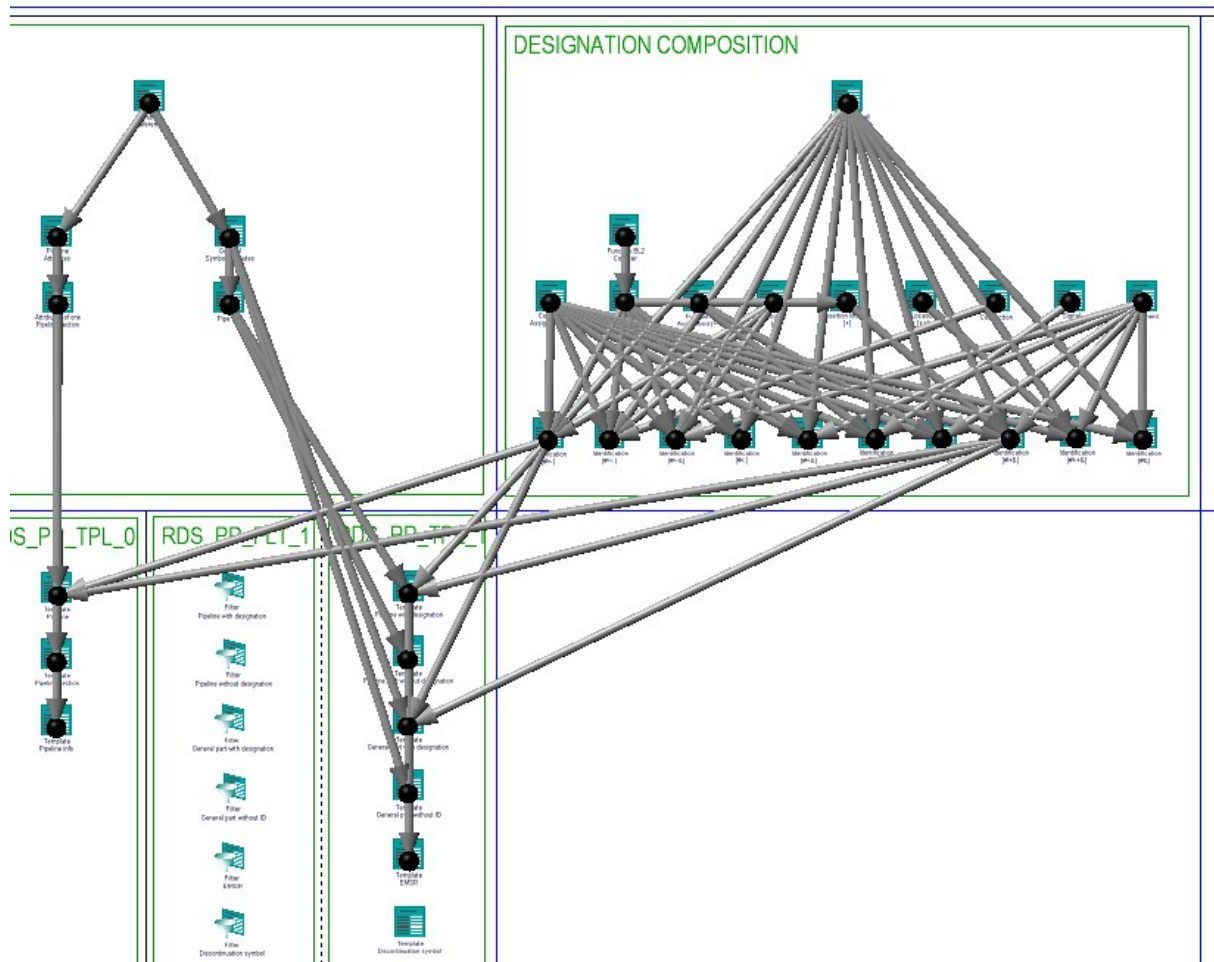
If you deactivate the **Restore last state of editing after crash** checkbox here, HiCAD will not search for files from previous sessions in the working directory. If the checkbox has been activated, the following query will be displayed:

Working directory contains data from a disrupted editing action. Do you want HiCAD to try to load this editing state?

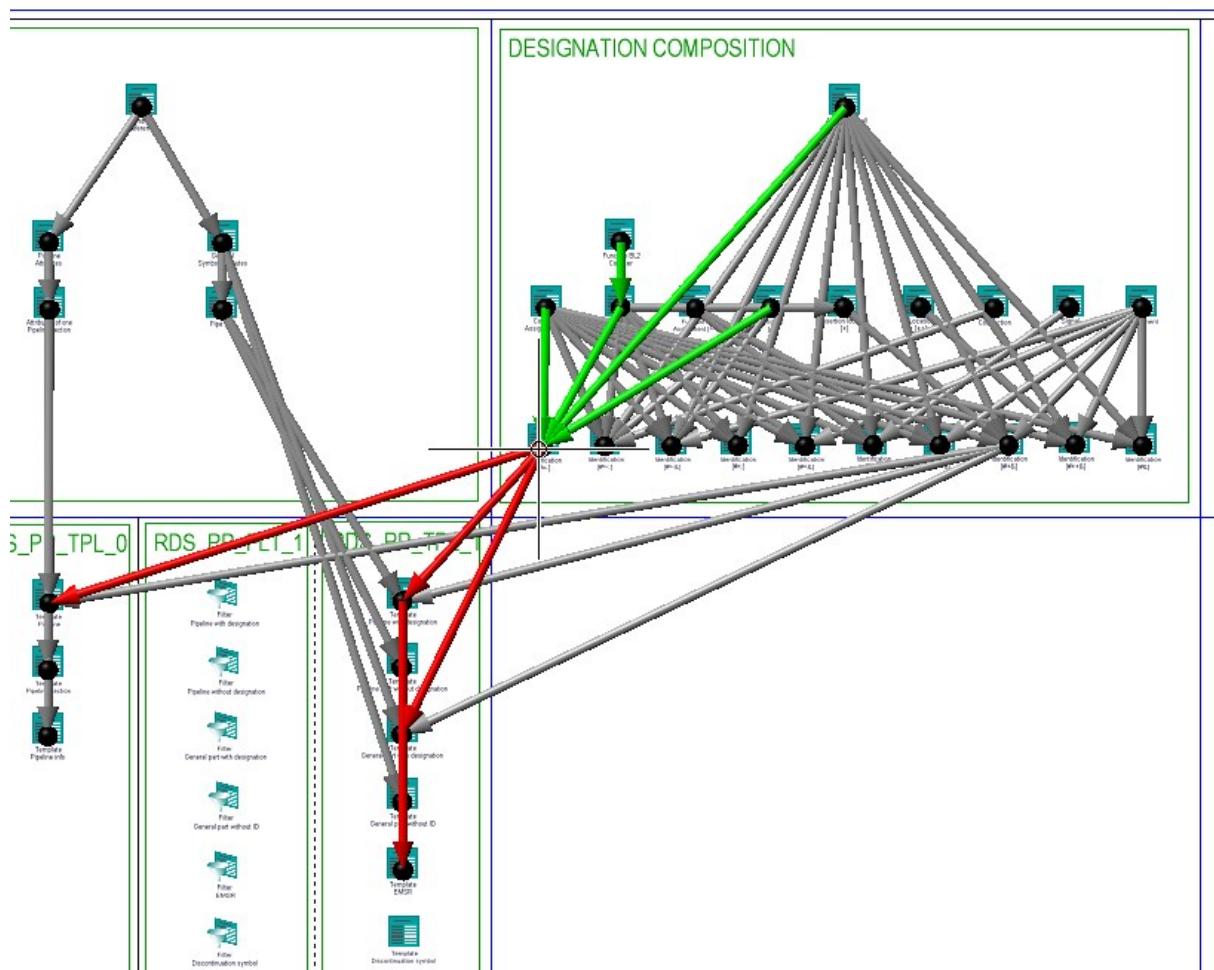
## Visualize derivation relations in symbol libraries

You have now the option to visualize the derivation relations between data mask objects in a DMO library with the help of inheritance graphs. You can scroll over the inheritance graph and highlight the inheritance paths, starting from the node where the cursor is located, in a different colour. Also, you can define new derivation relations to the current node in the process.

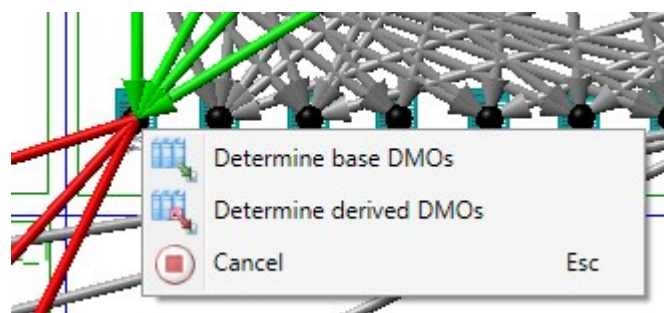
When you call the function, the inheritance graph will appear as shown below:



When you move the cursor to a node, the inheritance paths from this particular node will be highlighted in a different colour.

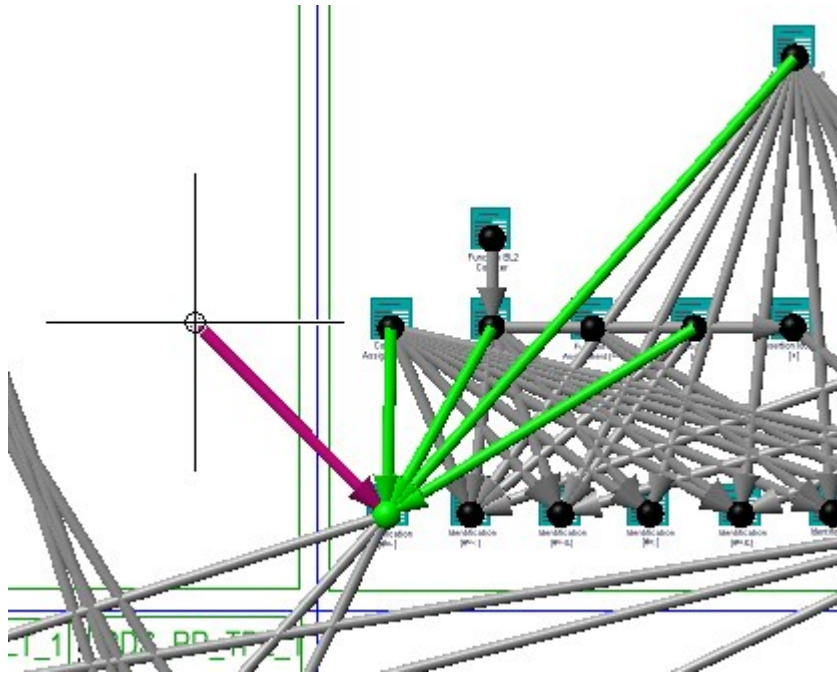


You can right-click and call the functions **Determine base DMOs** and **Determine derived DMOs**. These functions allow you to define further base DMOs or derived DMOs from a particular node, while the inheritance graph will remain visible.

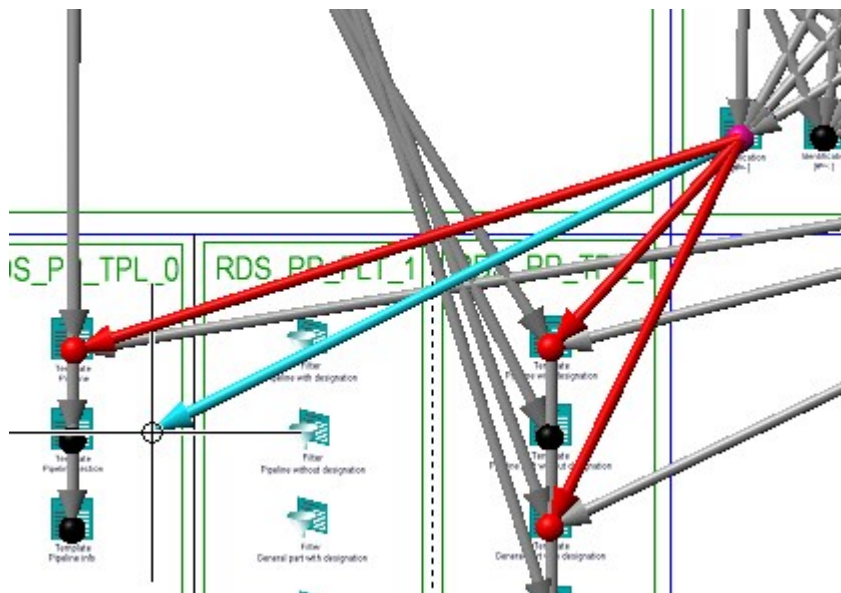


If further base DMOs are to be set, the following arrow will be visible:





If further derived DMOs are to be set, the following arrow will be visible:



Press the middle mouse button or the ESC key to cancel the function.

## Service Pack 1 2016 (V 2101)

### Derive symbols within DMO libraries

It is now possible to derive data mask objects (DMOs) from other DMOs according to a modular principle. This is particularly useful if you want different templates to have the same attributes.

## RDS-PP attributes

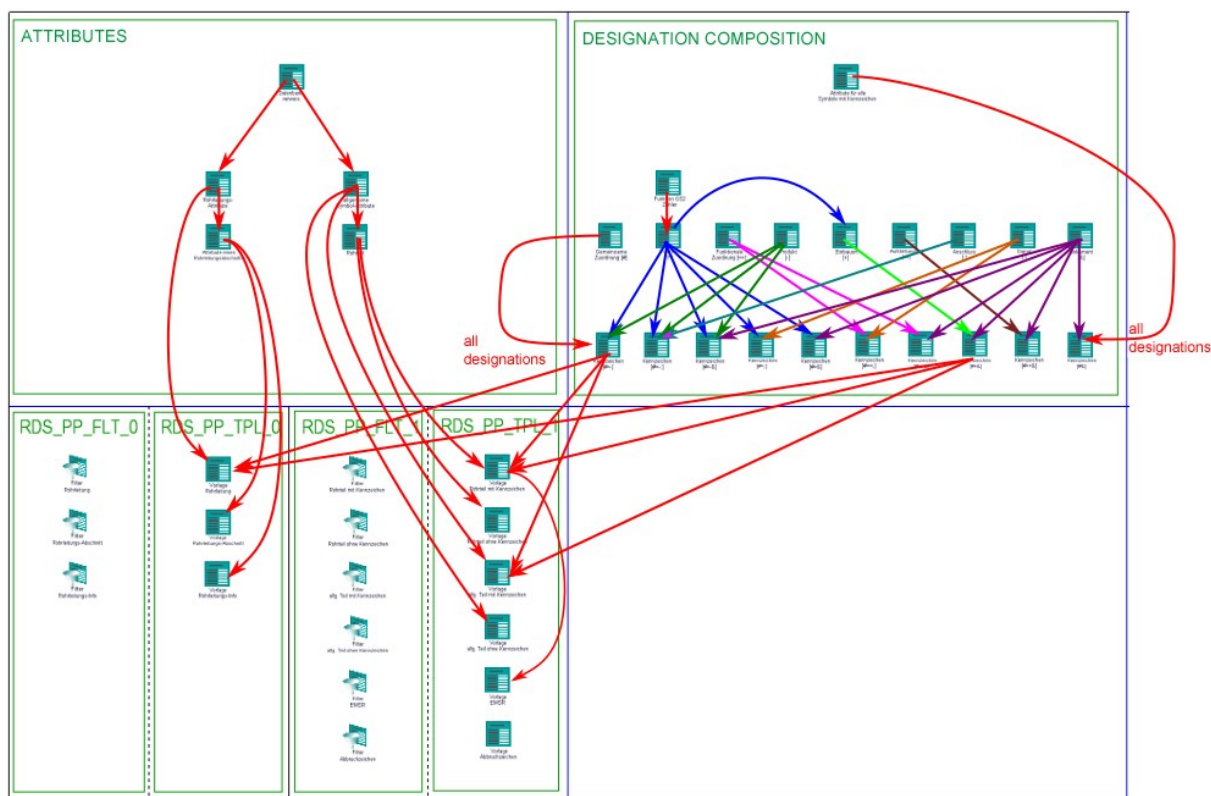
To enable you to work with the RDS-PP designation system, a number of new attributes has been defined. These can be combined into complete RDS-PP designations. As the attributes are able to depict the RDS-PP in a fairly detailed manner, it will be sufficient in most cases to directly specify values for the combining attributes.

- BL stands for "Breakdown level".
- LC stands for "Letter Code", i.e. the letter (non-numerical) portion of a breakdown level.
- DTG stands for "Digits", i.e. the numerical portion of a breakdown level.

## RDS-PP filter/templates library

The filter/template library supporting the designation according to RDS-PP is the ISD\_RDS\_PP library. This library makes extensive use of symbol derivation.

*The image below illustrates the defined relations between derivations:*



The actual DMO templates are located, together with their filters, in the symbol groups

- RDS\_PP\_FLT\_0
- RDS\_PP\_TPL\_0
- RDS\_PP\_FLT\_1
- RDS\_PP\_TPL\_1

These DMO templates derive from Base DMOs, which are subdivided into two groups.

In the **ATTRIBUTES** group, general attributes are defined. For example, it is ensured here that attributes for the database link exist.

In the **DESIGNATION COMPOSITION** group the RDS-PP designation composition is emulated using the derivation mechanism.



## Symbol libraries for RDS-PP designation

For the RDS-PP designation the symbol libraries **RDS\_PP\_SYMO**, **RDS\_PP\_SYM1** and **RDS\_PP\_SYM2** are available. They largely correspond with the libraries **ISDDINSYMO**, **ISDDINSYM1** and **ISDDINSYM2**, except for some changes to the attributes.

For symbols to which a designation according to IEC/PAS 62400 was assigned, this designation can be found on the attribute **Function BL2(LC)**, i.e the "Letter Code" portion of "Breakdown Level" 2 of the "Function" aspect.

Furthermore, you need to activate the **From templates library** option on the **Symbol** tab of the **Settings** dialogue window.

Also, the symbols have the two attributes **Designation1 (RDS-PP)** and **Designation2 (RDS-PP)**. These will obtain values from the template DMOs of the RDS-PP filter/template library.

## Using RDS-PP designations in P+IDs

To be able to use the RDS-PP designation defined in **ISD\_RDS\_PP**, you need to integrate the libraries **RDS\_PP\_SYMO**, **RDS\_PP\_SYM1**, **RDS\_PP\_SYM2** and **ISD\_RDS\_PP** into your project.

To do this, choose **Settings** on the **P+ID** tab. In the dialogue window, open the **Configuration** tab. Now, click **Add** to integrate the libraries into the P+ID. In addition, you must deactivate the **From templates library** option on the **Symbol** tab.

Furthermore, you need to add the template DMOs of the RDS-PP filters/templates library to the DMO sheet of the project, to ensure that they will actually be used when the symbol is inserted. Call the **Insert several** function and select the library.

## RDS-PP attributes in title block libraries

In HiCAD P+ID the sheet-related attributes are stored on the title block symbol and can thus, for instance, also be addressed via the text key **%FRAME**. The attributes of the title block symbols of the symbol library **ISD\_FRAME\_ENG.SZA** contains the following entries for RDS-PP:

- Conjoint Designation (RDS-PP)
- Function GSO
- Function GS1
- Functional allocation (RDS-PP)
- Point of installation (RDS-PP)
- Point of installation GSO
- Location (RDS-PP)
- Document (RDS-PP)

## Enhanced generation rules

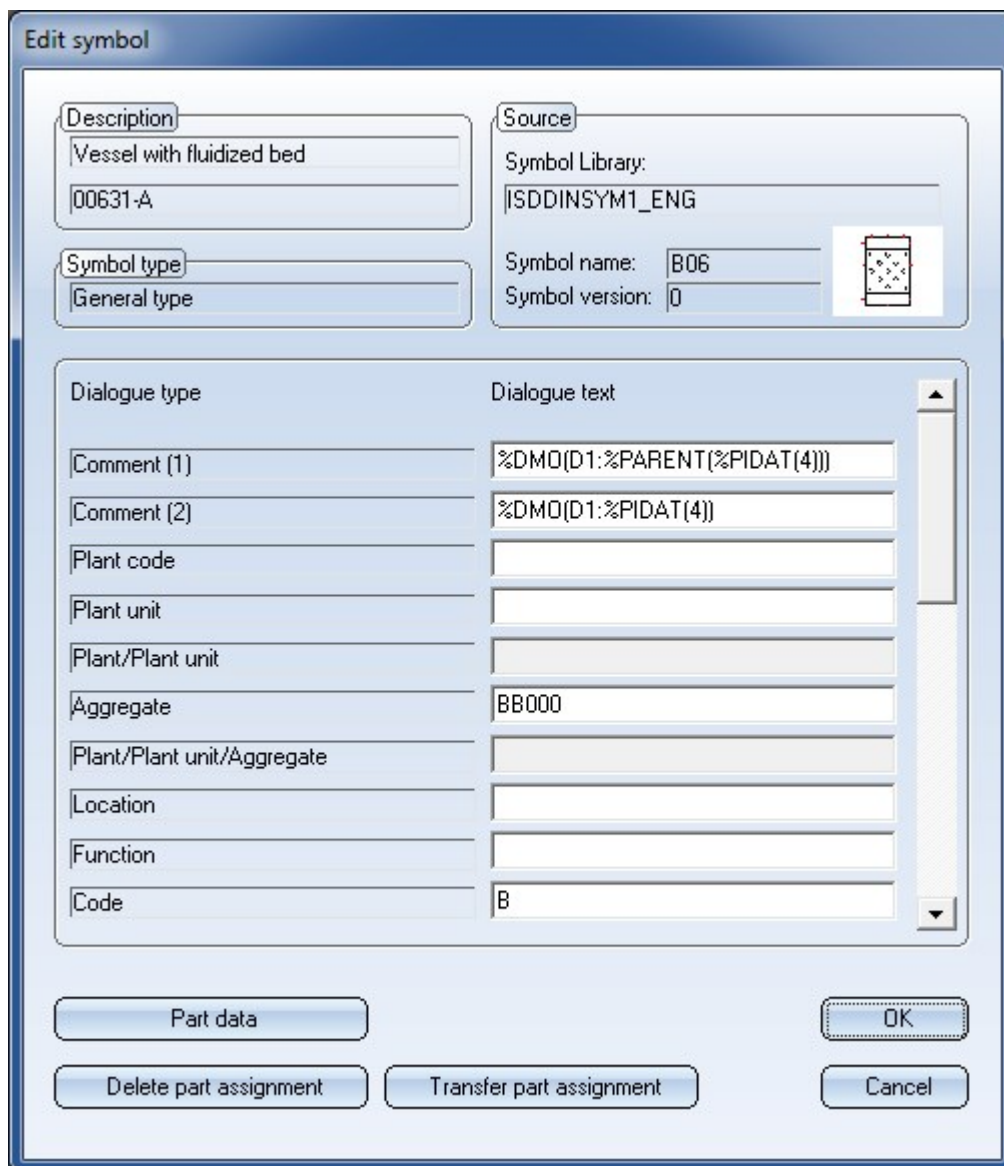
The new keyword **%TRIM()** removes the whitespaces at the start and the end of evaluated generation rules.

## Reset %COUNT text counters

If you have activated the **From templates library** option (on Symbol tab of Settings dialogue) for the selection of a designation system, a list with all counters containing the keyword **%COUNT** will be displayed after selection of the scope (Entire project, Same sheet number, ...). Select one or several rows in the list and confirm your selection with **OK**. The selected text counters will then be reset, i.e. counters with **%COUNT** will be set to 0 and recalculated.

## Embed DMOs in modules


Portions of a P+ID can be saved as so-called "modules". When symbols with the keyword **%DMO** were contained in the module, references to the DMOs were not embedded in the module in previous versions; as of Version 2101 the references to DMOs will be preserved.



The "Edit symbol" dialog box is used to configure a symbol. It contains the following fields and sections:

- Description:** Vessel with fluidized bed, 00631-A
- Symbol type:** General type
- Source:**
  - Symbol Library: ISDDINSYM1\_ENG
  - Symbol name: B06
  - Symbol version: 0
- Dialogue type / Dialogue text table:**

| Dialogue type              | Dialogue text               |
|----------------------------|-----------------------------|
| Comment (1)                | %DMO(D1:%PARENT(%PIDAT(4))) |
| Comment (2)                | %DMO(D1:%PIDAT(4))          |
| Plant code                 |                             |
| Plant unit                 |                             |
| Plant/Plant unit           |                             |
| Aggregate                  | BB000                       |
| Plant/Plant unit/Aggregate |                             |
| Location                   |                             |
| Function                   |                             |
| Code                       | B                           |
- Buttons:** Part data, Delete part assignment, Transfer part assignment, OK, Cancel

When inserting the module with the **Insert module**  function, the embedded DMOs will then be taken over into the DMO sheet of the new project.



### Please note:

Only the regular DMOs will be embedded in the module, but not the DMO filters.

## Generation key %FRAME for DMO

During editing of DMOs the **Title block symbol** option is now also available in the **Dialogue type/Attribute** selection list of the **Edit symbol** function.

## Symbol identification

The **Symbol** tab of the **Settings** dialogue window contains the new **Attribute** button for the definition of identifiers for different designation systems.

When you click this button, the following dialogue will be displayed:

| Row   | Code | Dialogue type              | Role |
|-------|------|----------------------------|------|
| 1     | 1    | Plant code                 | 2    |
| 2     | 5    | Plant unit                 | 2    |
| 3     | 4    | Code                       | 1    |
| 4     | 2    | Function                   | 2    |
| 5     | 8    | Gauge function             | 2    |
| 6     | 9    | Aggregate                  | 2    |
| 7     | 16   | Plant/Plant unit/Aggregate | 2    |
| 8     |      |                            |      |
| 9     |      |                            |      |
| >> 10 |      |                            |      |

Buttons: Delete row, OK, Cancel

Here you can select several attributes (dialogue types) serving as symbol uniqueness. **1** in the **Role** column stands for the main dialogue type, **2** for sub-dialogue type. Up to 10 dialogue types can be defined here. Please note that greyed out dialogue types cannot be removed with the **Delete row** function. During insertion or editing of a symbol these attributes will be checked for uniqueness, if the ☒ **Check symbol identification for uniqueness** checkbox has been activated.

The main dialogue types serving as symbol identifiers will, for instance, be used in symbol search result lists.

## Major Release 2016 (V 2100)




## Cancel drawing of connection lines

If the start of a connection line is drawn in the wrong direction, an error message will be displayed and the connection line jumps back to the first connecting point.

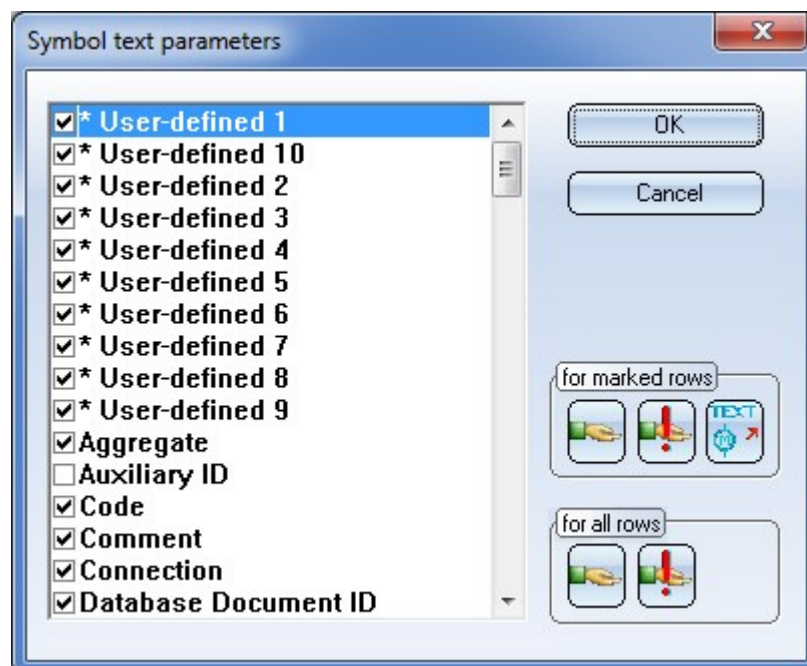
During drawing of a connection, you can press the middle mouse button to jump back to the previous point. If the connection direction at the other end is not allowed, the connection line will jump back to the previous point. In older versions, the complete connection line was deleted.

## Dialogue for text parameter pre-settings

The possibilities of text parameter pre-settings for symbol dialogue texts were extended, and the corresponding dialogue window was changed accordingly. You will find the functions for the specification of text parameters at

- P+ID > Settings > Preset text parameters 
- P+ID Library > Settings > Preset text parameters 
- R+I-Symbol > Settings > Preset text parameters 

After activating the icon, the **Symbol text parameters** dialogue window will be displayed.



OK

Save all changes and close dialogue window.

Cancel

Discard all changes and close dialogue window.



for  
marked  
rows

**Only active when P+ID  
symbol in Symbol  
Editor**

Reset the parameters of the dialogue types with activated checkbox ☒ to default values.



for  
marked  
rows

**Only active when P+ID  
symbol in Symbol  
Editor**

Define the parameters of the dialogue types with activated checkbox ☒ as default values.



for  
marked  
rows

**Only active when P+ID  
symbol in Symbol  
Editor**

Apply the parameters from the graphics texts of the dialogue types with activated checkbox ☒, if corresponding text placeholders exist.



for  
all rows

Define the parameters of all dialogue types as default values.



for  
all rows

Reset the parameters of all dialogue types to default values.

On the left side of the dialogue window, all dialogue types are listed that are given in the current P+ID configuration via the assigned attribute and dialogue type definitions.



### Please note:

The text parameters for the dialogue types are saved to the file *SymAttrTextParam.xml*, in the same sub-directory as the attribute and dialogue type definitions specified in the selected configuration, e.g. ..\PID\Lib-s\AttDef\ISDAttDef\_eng.

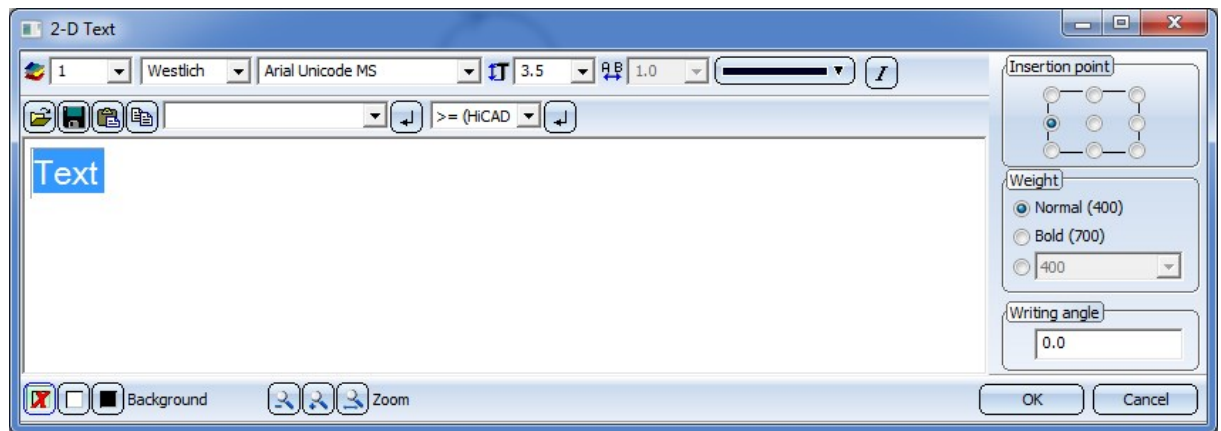
It depends on the currently loaded P+ID object which dialogue types are activated ☒ in the list:

- A P+ID sheet: All dialogue types occurring in the data masks of the symbols on the sheet
- A P+ID library: All dialogue types occurring in the data masks of the symbols in the library
- A P+ID library symbol: All dialogue types occurring in the data mask of the symbol

If you right-click a dialogue type in the list, you will obtain the following context menu. The selected option refers exclusively to the selected symbol type.

**Edit parameters** Opens the Text Editor:

*Example*



The text parameters of the selected dialogue type are displayed and can be changed.

**Set parameters to default values**

Resets the text parameters of the selected dialogue type to the default values.

**Transfer parameters**

Allows you to transfer the text parameters of the selected dialogue type to other dialogue types. Another dialogue window will then be opened. Proceed as follows:

1. In the left list, mark the dialogue types to which the text parameter pre-settings are to be transferred.
2. In the right list, mark the parameters that you wish to transfer. Unmarked parameters will be left unchanged. Use the corresponding button to set or delete all checkmarks in the list.
3. Now, apply the selected text parameters with **OK**. If you select **Cancel**, the presets will be left unchanged. In both cases you will be redirected to the **Symbol text parameters** dialogue window.

**Take over parameters from text**

Allows you to take over the text parameters from a 2-D graphic text of your choice. After calling the function, use the crosshairs to select the text in the drawing the text parameters of which you want to take over. After selecting the text, the Text Editor will be opened, showing you the selected text and its parameters, and offering them for selection. You can apply the parameters with **OK**, or discard them with **Cancel**.

**Define parameters as default values**

Allows you to define the text parameters of the selected dialogue type as its default values.

## Text parameters for KKS textbox

The text parameter setting on the **Symbol** tab of the P+ID **Settings** dialogue window is no longer available. Now, the preset text parameters of the dialogue types **Plant/Plant unit** and **Aggregate** will be used.

## Extended generation rules

The new generation rule contains text keys the purpose of which is a breaking up of character strings such as XX000. The text has the format Format XX0000 and serves, for example, as a code for a symbol.

Example 1:

**%XCODE(%PIDAT(4))** returning the following after evaluation: BR, with (Dialogue type: 4) BR0000 being the code.

Example 2:

**%XDIGITS(%PIDAT(4):3)** returning the following after evaluation: 4, with (Dialogue type: 4) BR0000 being the code.

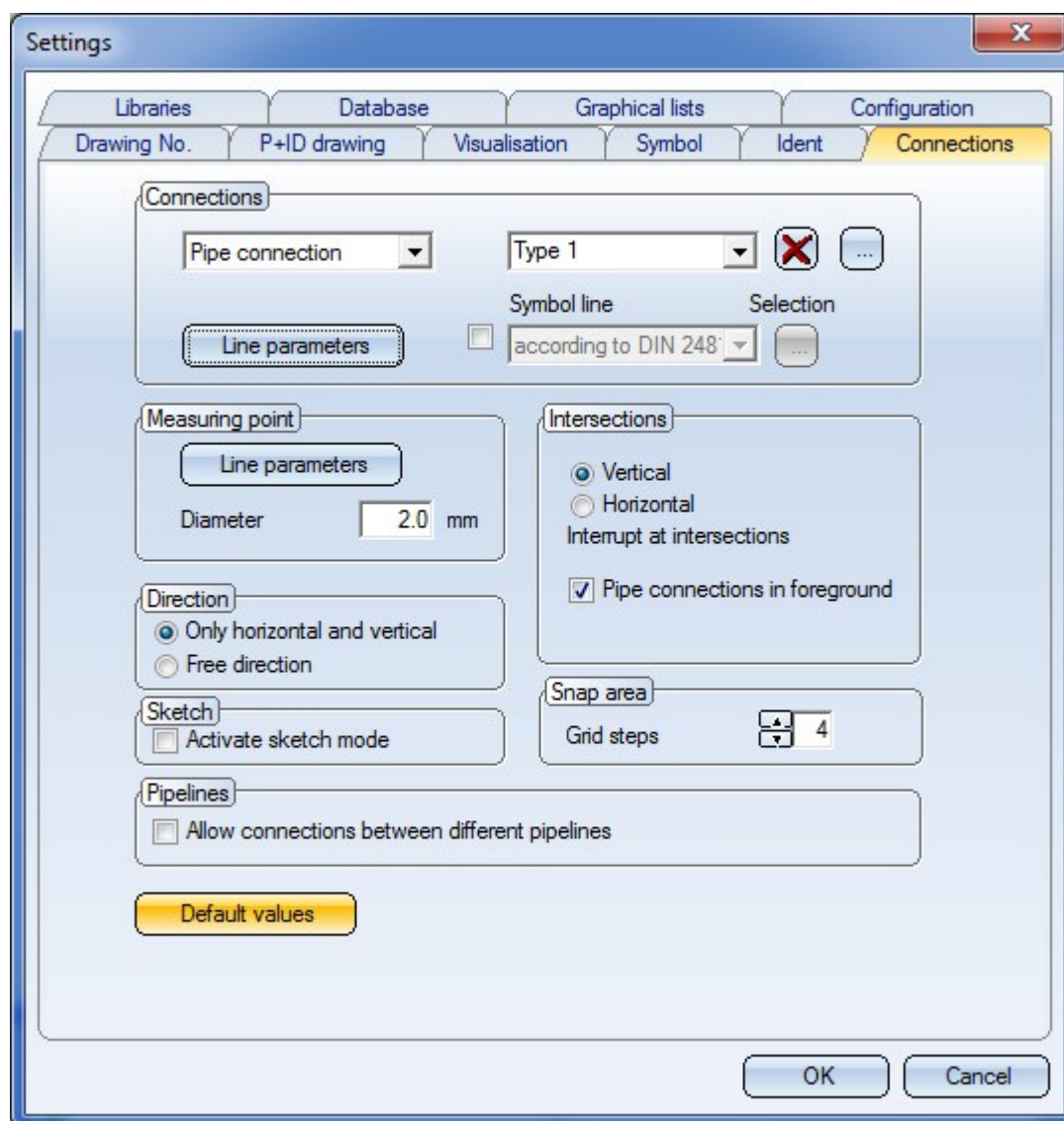
Example 3:

**%XDIGITS(%PIDAT(4):3)** returning the following after evaluation: 3, with (Dialogue type: 4) BR being the code.

## Settings: Connection definitions

The behaviour of **Default values** and **Cancel** buttons on the **Connections** tab of the P+ID **Settings** dialogue window has changed with regard to the connection definitions (see Online Help).

*Example:*



When you click on **Default values**:



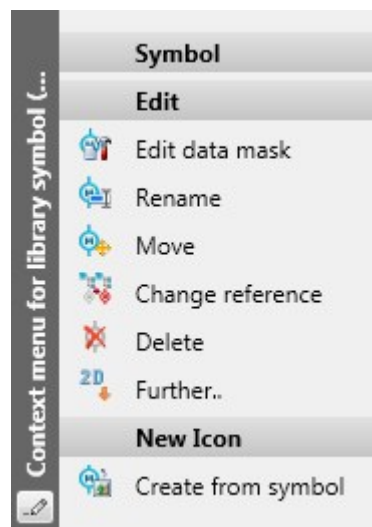
- Connection definitions created by the user will remain unchanged.
- Only connection definitions with names existing in the default connection definitions will be reset to the default parameter values.

When you click on **Cancel**:

- Previously created connection definitions will not be deleted again.

## Edit data mask in symbol library

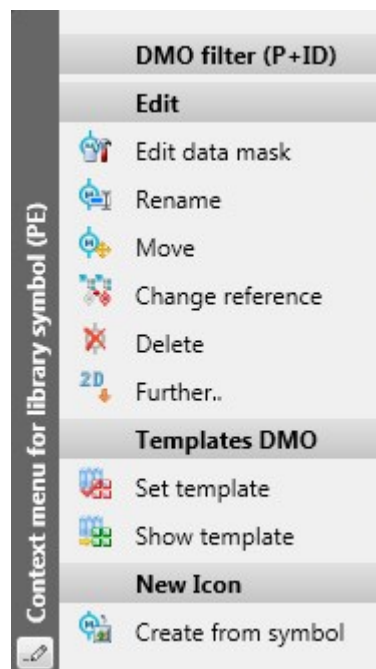
To adjust the attributes of a symbol in a data mask library, it was formerly necessary to load the symbol into the Symbol Editor first. This required a switching of drawing that had a somewhat disruptive effect on the working process. Now, the context menu of a symbol contains the **Edit data mask** function:



This function opens the dialogue for an editing of the mask, with the restriction that connection attributes cannot be changed. The number of connections influences the graphical representation of the symbol, with still can only be changed via the Symbol Editor.

## Data mask object filter

The context menu for data mask object filters contains two additional functions in the new **Templates DMO** menu section:



These functions facilitate the initialisation of the attributes **Templates DMO** and **Templates DMO library**:

**Edit symbol**

Symbol name:  Description:   
 Symbol type:

| Row | Code | Dialogue type         | Default dialogue text |
|-----|------|-----------------------|-----------------------|
| 1   | 40   | Comment (1)           |                       |
| 2   | 40   | Comment (2)           |                       |
| 3   | 31   | DMO-ID                |                       |
| 4   | 32   | DMO name              |                       |
| 5   | 2014 | Target symbol name    | _PIPE1                |
| 6   | 2010 | Templates DMO         | T_PIPESECT            |
| 7   | 2011 | Templates DMO library | ISDDINDMO0_ENG        |
| 8   | 2012 | Templates DMO ID      |                       |
| 9   |      |                       |                       |
| 10  |      |                       |                       |

Insert row Delete row OK Cancel

The two attributes are used when adding a data mask object (DMO) filter to a P+ID, in order to automatically find the matching template data mask object, add it, and link it to the data mask object filter via a reference to the DMO ID.

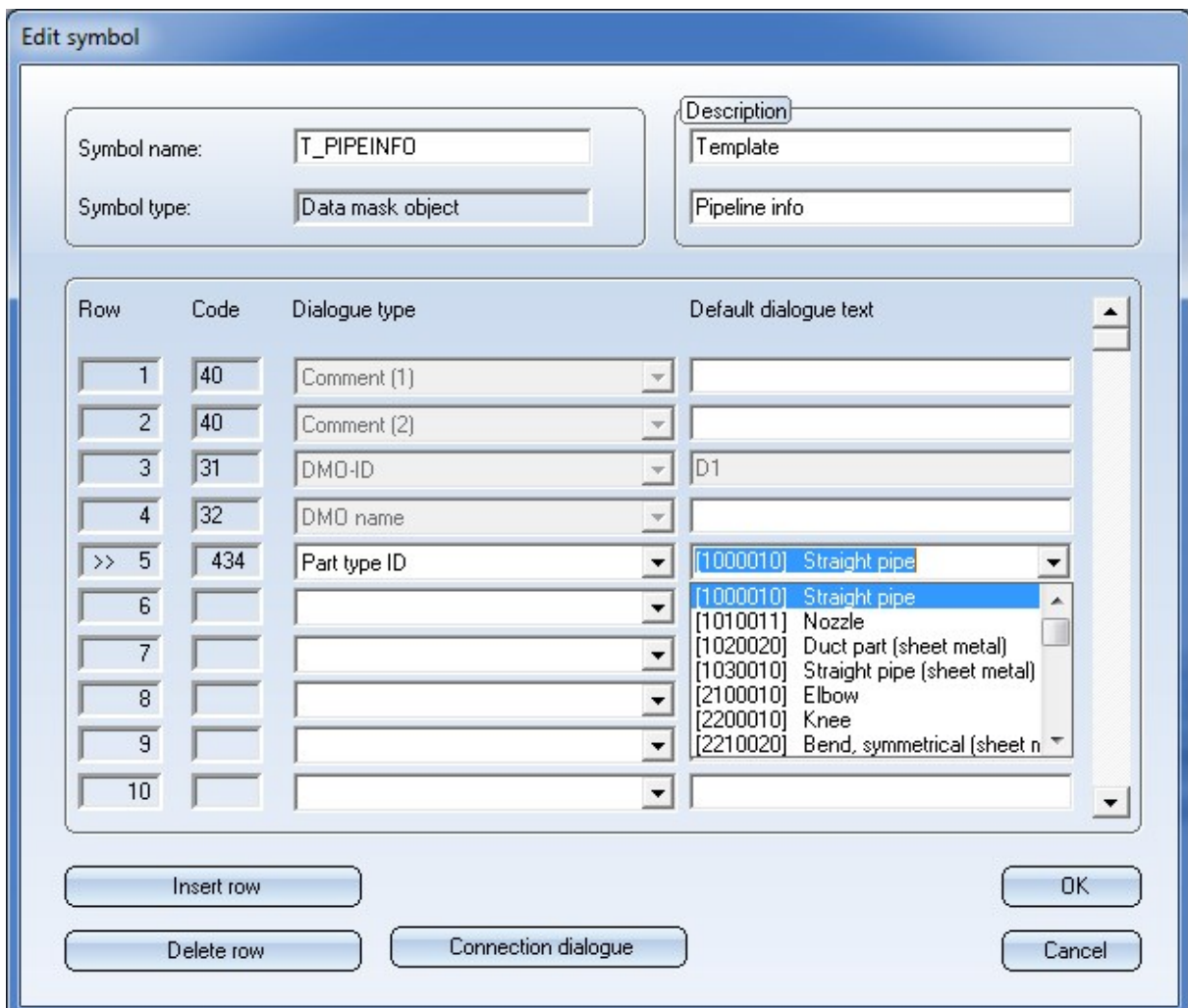
The **Set template** function expects the selection of a data mask object symbol, the name of which is entered in the **Templates DMO** attribute. The currently edited library is entered into the **Templates DMO library** attribute. If no data mask object was selected by mistake, a corresponding reminder will be displayed.

The **Show template** function evaluates the attributes **Templates DMO** and **Templates DMO library**, find the matching data mask object and highlights it both in the drawing and in the ICN. If the filter refers via **Templates DMO library** to a different library, a corresponding message will be displayed.

If the entered data mask object could not be found, a corresponding message will be displayed.

## Part type ID selection in 'Edit symbol' dialogue

In previous versions, a separate dialogue was shown for the initialization of the value for the **Part type ID** attribute. This dialogue has been replaced by a combo box.



The 'Edit symbol' dialog box is shown with the following fields and table:

Symbol name:

Symbol type:

Description:

Pipeline info:

| Row  | Code | Dialogue type | Default dialogue text                 |
|------|------|---------------|---------------------------------------|
| 1    | 40   | Comment (1)   |                                       |
| 2    | 40   | Comment (2)   |                                       |
| 3    | 31   | DMO-ID        | D1                                    |
| 4    | 32   | DMO name      |                                       |
| >> 5 | 434  | Part type ID  | [1000010] Straight pipe               |
| 6    |      |               | [1000010] Straight pipe               |
| 7    |      |               | [1010011] Nozzle                      |
| 8    |      |               | [1020020] Duct part (sheet metal)     |
| 9    |      |               | [1030010] Straight pipe (sheet metal) |
| 10   |      |               | [2100010] Elbow                       |
|      |      |               | [2200010] Knee                        |
|      |      |               | [2210020] Bend, symmetrical (sheet n  |

Buttons: Insert row, Delete row, Connection dialogue, OK, Cancel

## New attribute types

There are two new attribute types, which facilitate the applying of data mask object filters (DMO filters) to specific symbols.

If you use data mask object filters, HiCAD will check upon insertion of a symbol in a P+ID whether the search patterns on the attributes of the filter match the attributes of the inserted symbol. If this is the case, the attribute values of the template that belongs to the filter will be copied to the symbol.

This means that it will be specified in the filter to which symbols the attributes of a template are to be copied. The new attribute types **Target symbol name** and **Target symbol type** facilitate the restriction of a filter to a specific range of symbols.

In the **Target symbol name** field you can directly enter the name of a symbol or enter a matching search pattern. For example, a filter which contains the search pattern ?EMSR? as the value for the **Target symbol name** attribute applies to the gauge symbols in the supplied library ISDDINSYM1\_ENG.sza.

The **Target symbol name** attribute is handled in a special way: As the name is a very specific attribute of a symbol, it is assumed that the template attributes have priority over other template attributes, and that these can be overwritten, without causing any conflicts.

In the **Target symbol type** field you can specify a symbol type for which a data mask object filter is to take effect, i.e. for instance, only for pipe part symbols. The available symbol types can be selected from a combo box:

| Row  | Code | Dialogue type      | Default dialogue text       |
|------|------|--------------------|-----------------------------|
| 1    | 40   | Comment (1)        |                             |
| 2    | 40   | Comment (2)        |                             |
| 3    | 31   | DMO-ID             | D1                          |
| 4    | 32   | DMO name           |                             |
| >> 5 | 2013 | Target symbol type | [20] Pipeline part          |
| 6    |      |                    | [01] General type           |
| 7    |      |                    | [03] Discontinuation symbol |
| 8    |      |                    | [09] Title block            |
| 9    |      |                    | [20] Pipeline part          |
| 10   |      |                    | [21] Pipeline               |
|      |      |                    | [16] Data mask object       |
|      |      |                    | [17] Data mask filter       |

## Designation according to templates library

The P+ID module has supported two plant engineering designation systems, namely, according to DIN and according to KKS. For both systems, the rules for attribute value assignment were little flexible, which is, on the one hand, a useful thing as it helps avoid errors, but on the other hand, prevents the utilisation of user-defined, deviating designation systems.

Thanks to the support of data mask object filters (DMO filters), you can now not only handle the existing designation systems in a much more flexible way, but also define your own designation systems.

### Existing designation systems

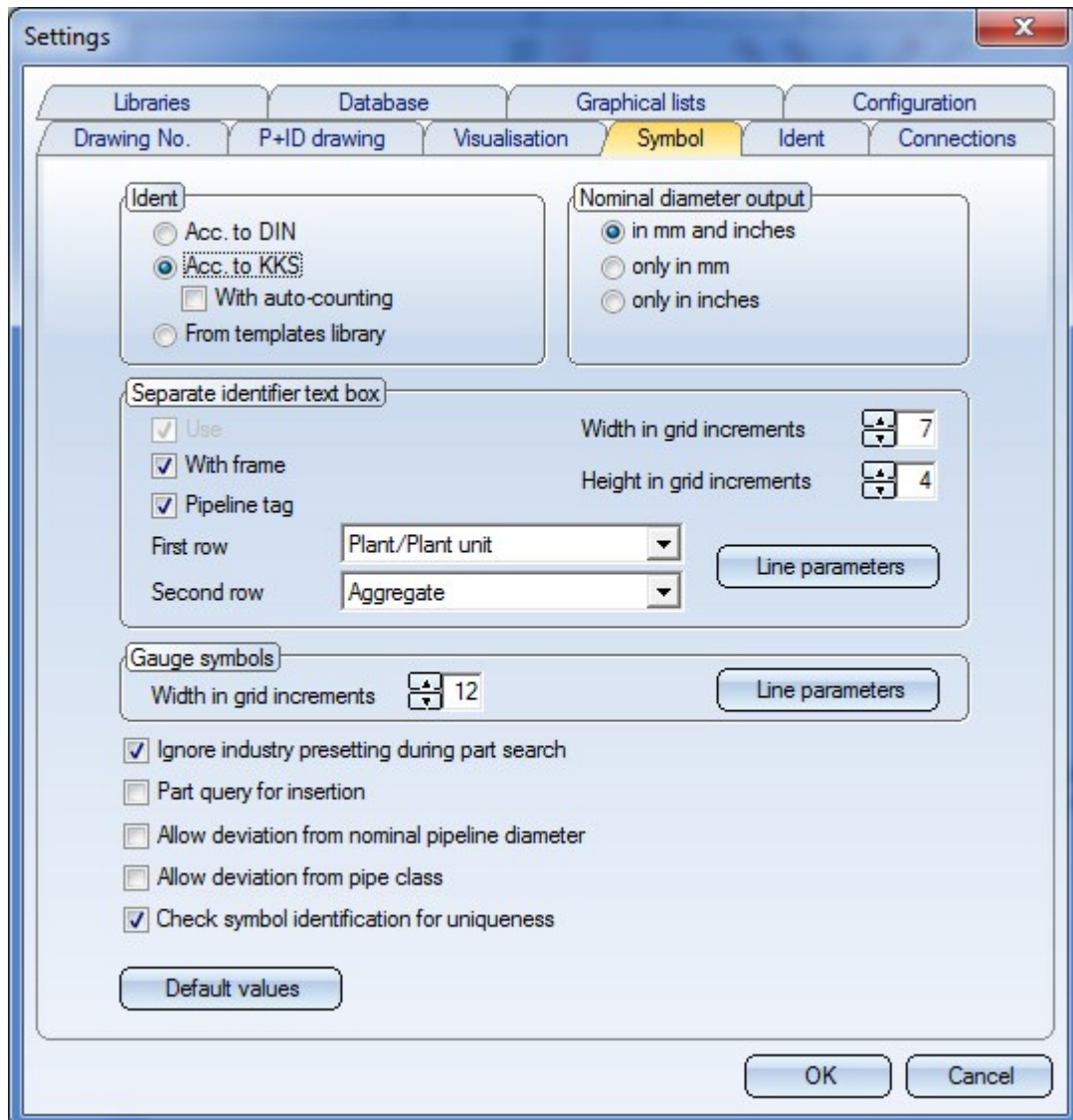
The existing designation systems become more flexible, as they are now handled subordinately to the text key expressions, which can be copied from the data mask object templates from a templates library to the symbol attributes. An example:

If you use the KKS designation system, the attribute **Plant / Plant unit** will be composed of the attributes **Plant code** and **Plant unit**. If now a text key expression exists on the attribute **Plant/Plant unit**, i.e. %PARENT(%PIDAT(6)), for example, HiCAD will recognize this constellation and refrain from using the fixed mechanism. Instead, the text key expression will be evaluated, which results in the applying of the value of **Plant/Plant unit** of the superordinate symbol.

This means that the text key mechanism will override the fixed mechanism.

### User-defined designation systems

If you want to use neither of the two fixed mechanisms, activate the **From templates library** option beneath **Ident** on the **Symbol** tab of the **Settings** dialogue window.



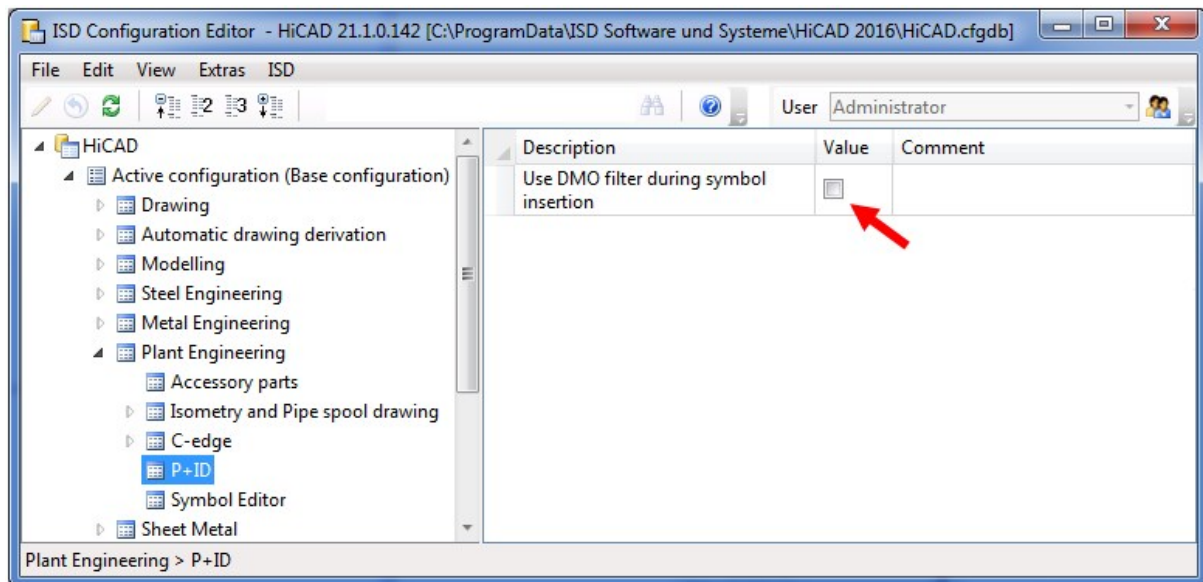
The screenshot shows the 'Settings' dialog box with the 'Symbol' tab selected. The 'Ident' section has three radio buttons: 'Acc. to DIN', 'Acc. to KKS' (selected), and 'From templates library'. There is also a checkbox for 'With auto-counting'. The 'Nominal diameter output' section has three radio buttons: 'in mm and inches' (selected), 'only in mm', and 'only in inches'. The 'Separate identifier text box' section has checkboxes for 'Use', 'With frame', and 'Pipeline tag'. Below these are dropdown menus for 'First row' (set to 'Plant/Plant unit') and 'Second row' (set to 'Aggregate'). There are 'Line parameters' buttons next to these dropdowns. The 'Gauge symbols' section has a 'Width in grid increments' spinner set to 12 and a 'Line parameters' button. At the bottom, there are several checkboxes: 'Ignore industry presetting during part search' (checked), 'Part query for insertion' (unchecked), 'Allow deviation from nominal pipeline diameter' (unchecked), 'Allow deviation from pipe class' (unchecked), and 'Check symbol identification for uniqueness' (checked). A 'Default values' button is at the bottom left, and 'OK' and 'Cancel' buttons are at the bottom right.



## Globally deactivate data mask object filter via the Configuration Editor

When a symbol is added in a P+ID, HiCAD will check whether a data mask object filter matches this symbol. If this is the case, the attribute values of the corresponding template will be copied to the attributes of the symbol.

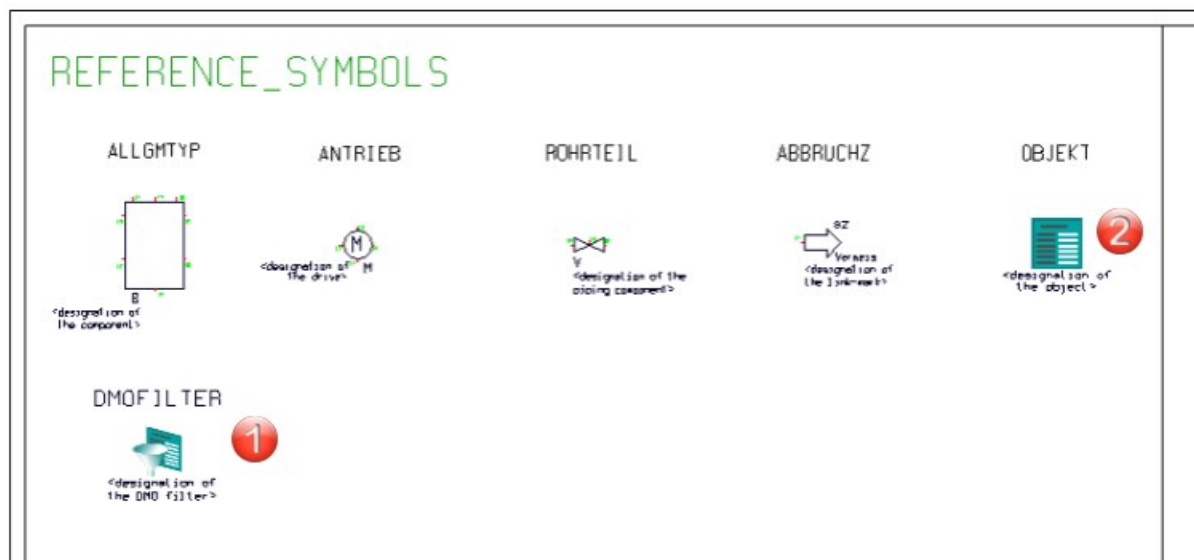
If you do not want this, you have the following options: You can either remove all data mask object filters from the project, or deactivate this mechanism globally in the Configuration Editor, at **Plant Engineering > P+ID > Use DMO filter during symbol insertion**.



## Creation of user-defined template libraries

The first steps for the creation of a user-defined templates library are identical to those for the creation of a normal symbol library. First, create a new P+ID library, which needs to be of the type "DMO library" to enable an adding of data mask objects and data mask object filters.

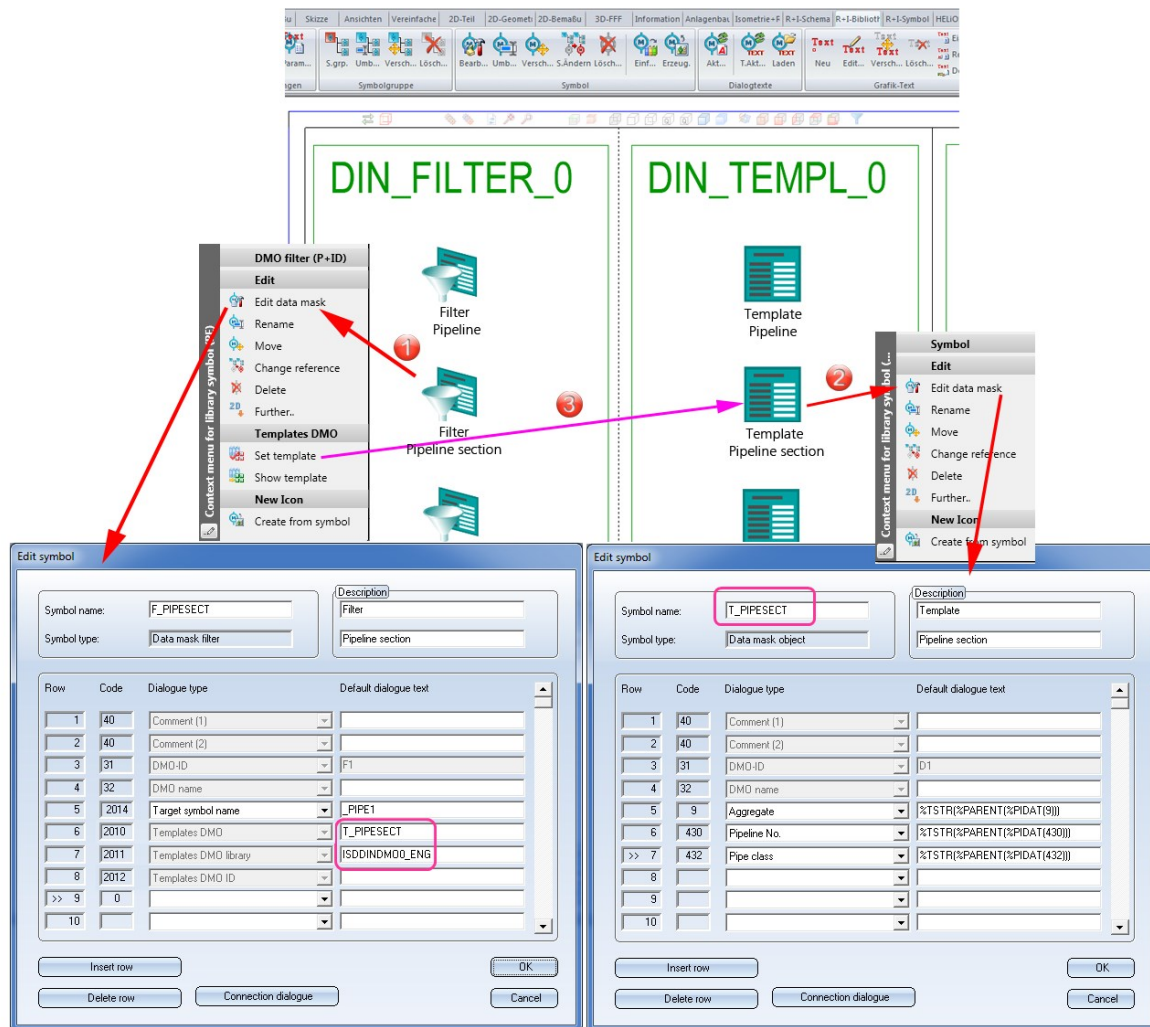
Then, add data mask objects and data mask object filters, preferably by selecting the corresponding reference symbols from the library ISDREFSYM.SZA:



1) Symbol for data mask object filter

2) Symbol for data mask object

The following image illustrates the further procedure:



1. Search patterns determining to which symbols a filter is to be applied can be assigned to the attributes of a data mask object filter. For this you use the **Edit data mask** function in the context menu or the Symbol Editor.
2. Any texts or text keys can be assigned to the attributes of the Templates DMO. To do this, use the **Edit data mask** function in the context menu or the Symbol Editor.
3. To ensure that the matching Templates DMO will automatically be provided and linked to a DMO filter that is added in a P+ID, you need to specify in the library which filter belongs to which template. This assignment can be conveniently defined with the **Set template** function in the context menu. This function sets the value of the **Templates-DMO** for the **Symbol name** of the Templates DMO object. Furthermore, the name of the library you have just edited will be assigned to the attribute **Templates DMO library**. If you prefer the assigning to a template in a different library, fill in the attribute manually.

## Supplied filter/templates libraries

Two filter/template libraries are supplied, namely, ISDDINDMOO\_ENG.SZA and ISDKKSDMOO\_ENG.SZA. These emulate the designation system according to DIN and the KKS designation system.

Both libraries match the supplied symbol libraries. In particular, the template data mask objects are initialized with attributes that provide the designation and counter predefinitions from the supplied symbol libraries.



## Default libraries

The DMO libraries ISDDINDMO\_GER.SZA and ISDDINDMO\_ENG.SZA have been added to the supplied default libraries stored in the configuration files config\_ger.dat and config\_eng.dat which are integrated into a new P+ID.

These configuration files are located at PID\Admin\PidConfig of the HiCAD installation directory

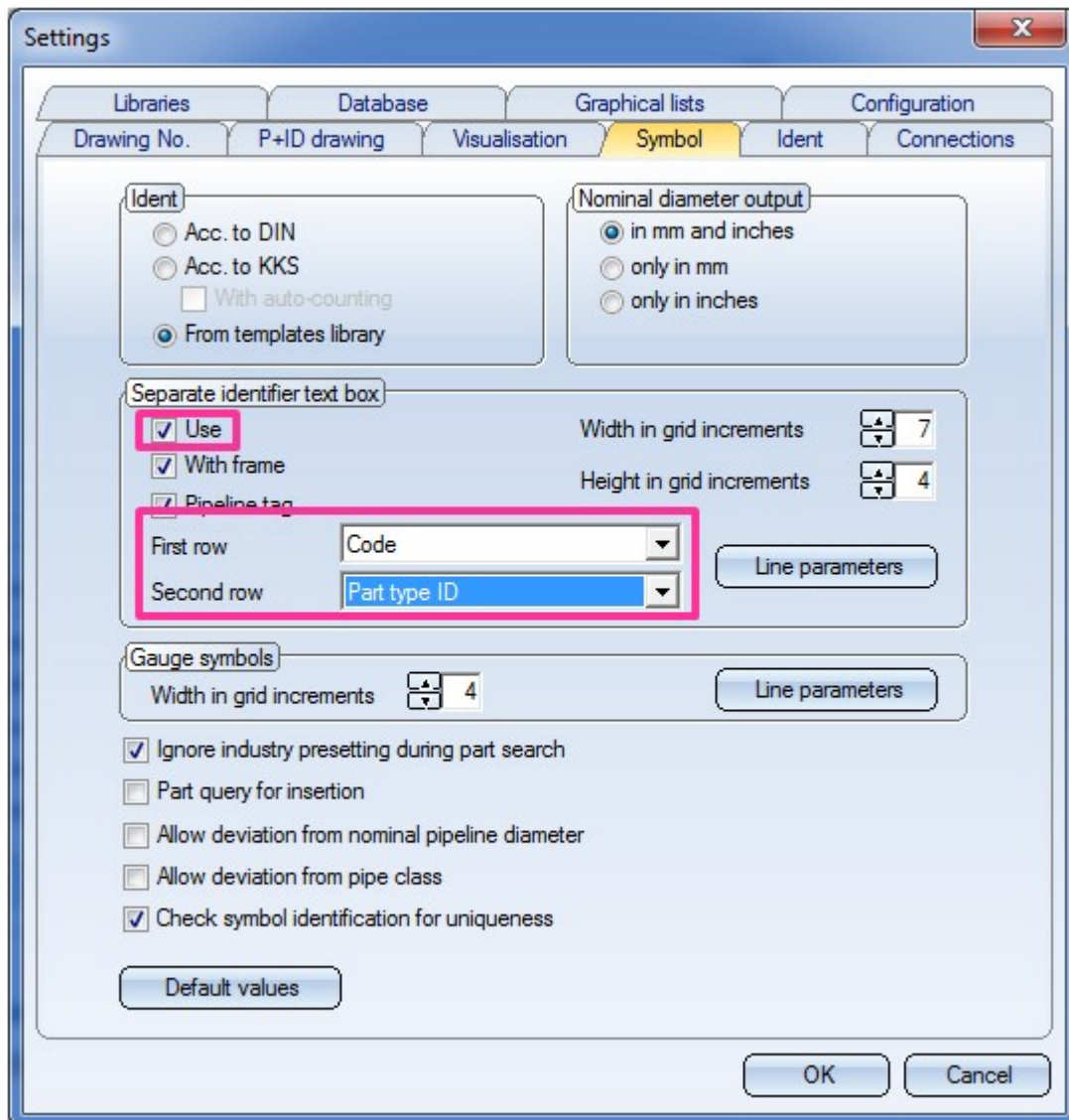
.

## Separate identifier text box in the Symbol settings

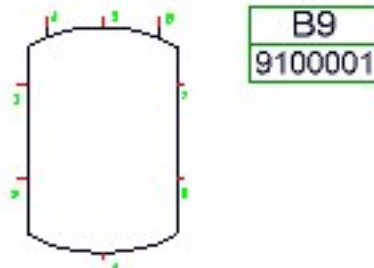
The 2-line KKS text box was previously only available if the KKS system was activated. To be able to use this text box also when working with Templates DMOs, their availability has been enhanced.

Previously, the attributes that were entered into the text box were predefined. They can now be used in a more flexible way via the corresponding settings on the **Symbol** tab.

Example:






A Vessel that was inserted with the settings shown above would look as follows:



## New functions for the management of data mask objects

You can use the functions **Edit sheet for data mask object** or **Edit DMO filter** to specify which data mask objects or data mask object filters are to be available in a P+ID. These functions allow you to add individual data mask objects or data mask object filters.

Three new functions for the handling of Templates DMOs have been added at **P+ID > P+ID Sheet > DMO** (dropdown menu). These functions affect the availability of data mask objects.

|   |                           |   |
|---|---------------------------|---|
|  | <b>Insert<br/>several</b> | Displays a list of all templates libraries that exist in the P+ID. When you select some of them and confirm with <b>OK</b> , all data mask objects that exist in these libraries will be added to the available data mask objects. Already existing data mask objects will not be changed or overwritten.   |
|  | <b>Reset<br/>several</b>  | Displays a list of all templates libraries, at least one data mask object exists in the P+ID. When you now select one or several of these libraries, the available data mask objects will be overwritten with the state that was specified in the libraries.<br><br>This function enables you to conveniently undo your own changes, or apply changes to libraries. |
|  | <b>Delete<br/>several</b> | Displays a list of all templates libraries, at least one data mask object exists in the P+ID. When you now select one or several of these libraries, the corresponding data mask objects will be removed from the list of available data mask objects.  |

## Hierarchy of data mask objects

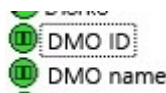
It is now possible to subordinate one DMO to another DMO. At the same time, a DMO can be superordinated as a so-called "Parent DMO" to other DMOs. A Parent DMO can, however, not be subordinated to another DMO.

Use the **Edit sheet for data mask object** function to display a list of DMOs in the DMO sheet in the dialogue window. With the functions Set parent, Show parent and Remove parent you can edit the hierarchy.

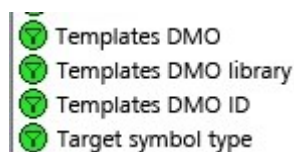
Use the corresponding function in the context menu of a subordinate DMO to apply the attribute value of its Parent DMO via generation rule with the key word **%PARENT**.

## New attribute icons

Icons for DMO and DMO filter:



Icon for DMO filter:



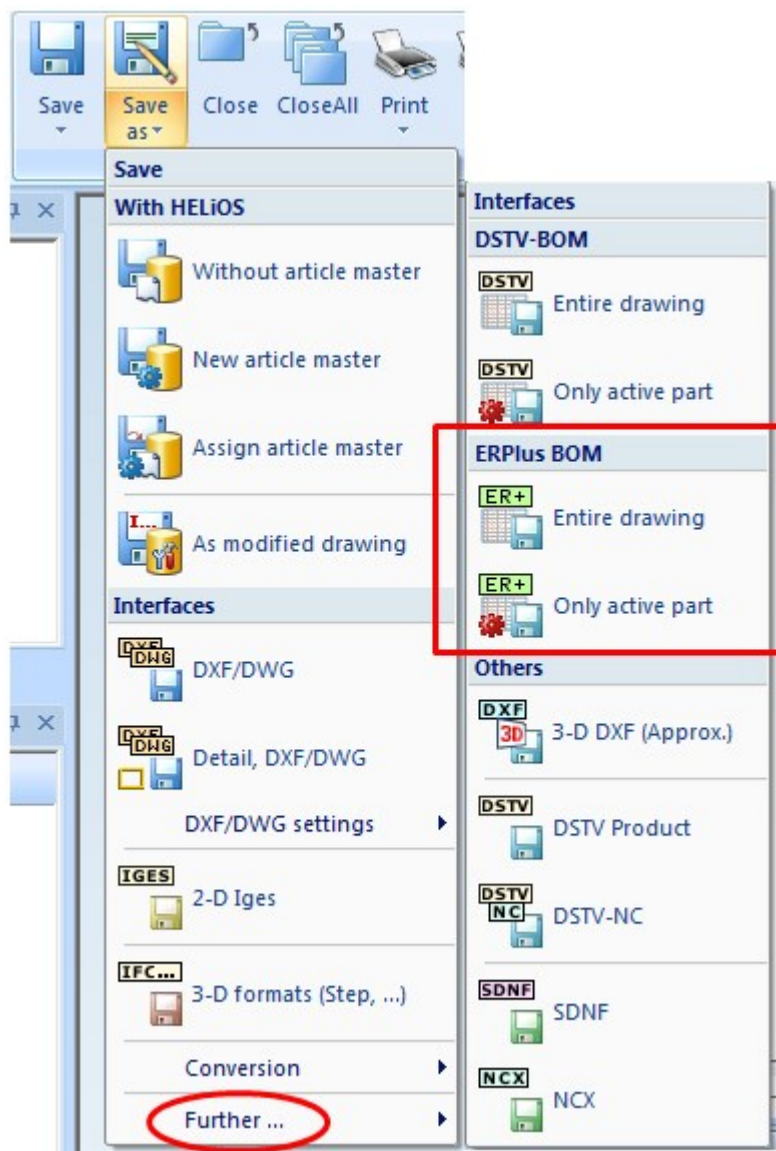
These icons are shown, for example, in the context menu dialogue **Dialogue type/Attribute**.

## Interfaces

Service Pack 2 (V 2102)

### ERPlus BOM

HiCAD now offers functions for the generation of DSTV-BOMs for the ERP system **ERPlus**.



### DSTV-NC export

The dialogue window for DSTV-NC export has been revised and expanded by options for the configuration of the order number, drawing number, item number and part number that can be specified directly in the dialogue window.

Please note the following change:

In older HiCAD versions, only the configuration files **DSTV\_NC\_Itemnumbertext.ftd** for the **Item number** and **DSTV\_NC\_Partnumbertext.ftd** for the **Part number** were used.

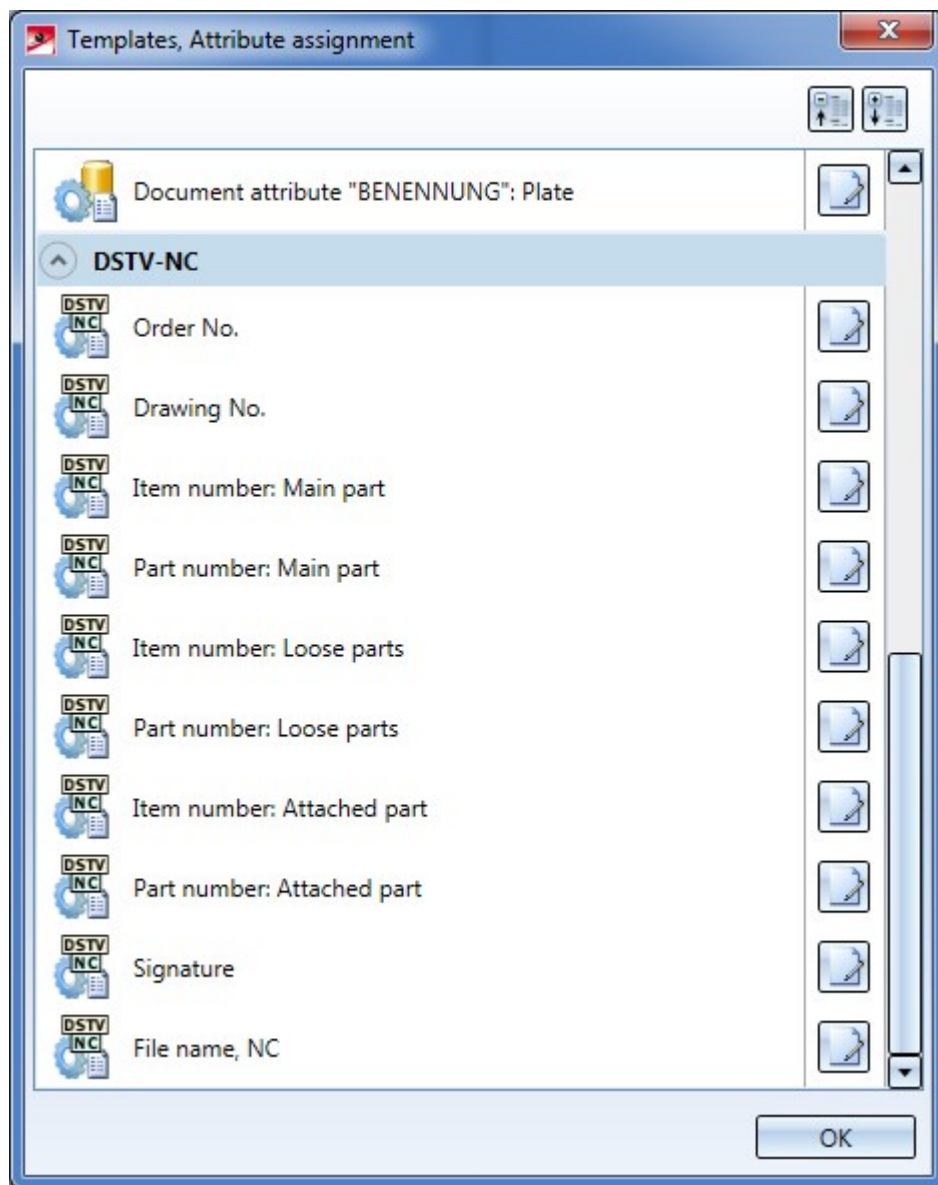
As of HiCAD 2016 Service Pack 2 (Version 2102) you distinguish between Main part (H part), Sub-part (W part) and Loose parts, for which separate configuration files exist.

If you have worked with the corresponding configuration files in older versions, their content must be copied to those for Main parts and Sub-parts: For the Item number **DSTV\_NC\_H\_Itemnumbertext.ftd** (main part) and **DSTV\_NC\_W\_Itemnumbertext.ftd** (Sub-part) or for the Part No. **DSTV\_NC\_H\_Partnumbertext.ftd** (Main part) and **DSTV\_NC\_W\_Partnumbertext.ftd** (Sub-part), respectively.

**DSTV\_NC\_Itemnumbertext.ftd** and **DSTV\_NC\_Partnumbertext.ftd** each refer to Loose parts that do not belong to any assembly.

Please also note that the configuration file **stb\_parameter.dat** that was used in older versions does no longer exist. The settings for the item number of the assembly can now be specified in the Configuration Editor.

Choose **Steel Engineering > Further functions > Settings** > **Templates, Attribute assignment** to adjust the templates for attribute assignment for the DSTV export.



The templates for attribute assignment for DSTV-NC export are FTD files, too; they are located in the sys directory of your HiCAD installation. Supplied with HiCAD are the templates listed below:

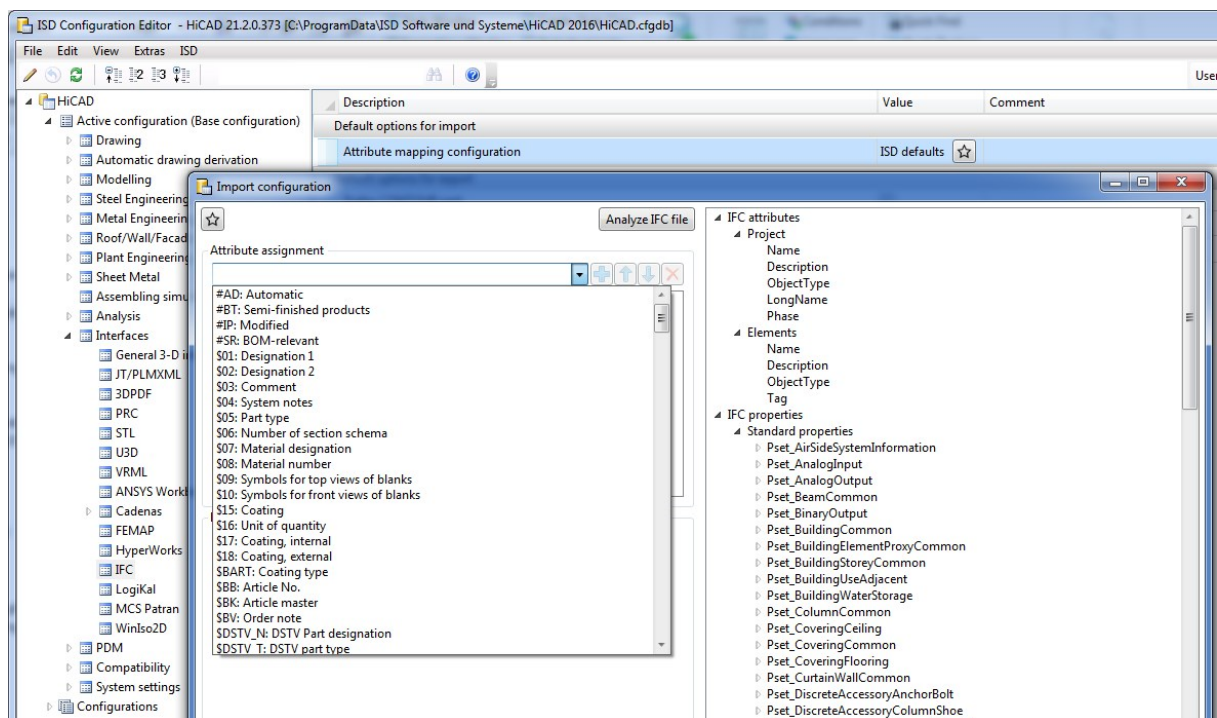


| Templates file                        | Settings for                      | Pre-setting   |
|---------------------------------------|-----------------------------------|---|
| DSTV_NC_<br>ORDERNUMBERTEXT<br>.FTD   | Order number                      | %TS(TEXT760)<br><br>%TS(TEXT760) stands for the content of the text key TEXT760, which is the text <b>Order number:</b>           |
| DSTV_NC_<br>DRAWINGNUMBERTEXT<br>.FTD | Drawing number                    | %TS(TEXT1AN113)<br><br>%TS(TEXT1AN113) stands for the content of the text key TEXT1AN113 which is the text <b>Drawing number:</b> |
| DSTV_NC_H_<br>ITEMNUMBERTEXT .FTD     | Item number for<br>main parts     | {Item number (Part attribute)}  |
| DSTV_NC_H_<br>PARTNUMBERTEXT<br>.FTD  | Part number<br>for main parts     | not allocated   |
| DSTV_NC_<br>ITEMNUMBERTEXT .FTD       | Item number for<br>loose parts    | {Item number (Part attribute)}  |
| DSTV_NC_<br>PARTNUMBERTEXT<br>.FTD    | Part number for<br>loose parts    | not allocated   |
| DSTV_NC_W_<br>ITEMNUMBERTEXT .FTD     | Item number for<br>attached parts | {Item number (Part attribute)}  |
| DSTV_NC_W_<br>PARTNUMBERTEXT<br>.FTD  | Part number for<br>attached parts | not allocated   |
| DSTV_NC_<br>SIGNATURETEXT .FTD        | Signature                         | {Item number (Part attribute)}  |
| DSTV_NC_FILENAME<br>.FTD              | File name - NC                    | {Item number (Part attribute)}  |

## IFC: Attribute mapping

During the data transformation between IFC and HiCAD the attribute values from the original data will be transferred to permissible attribute values of the target system.

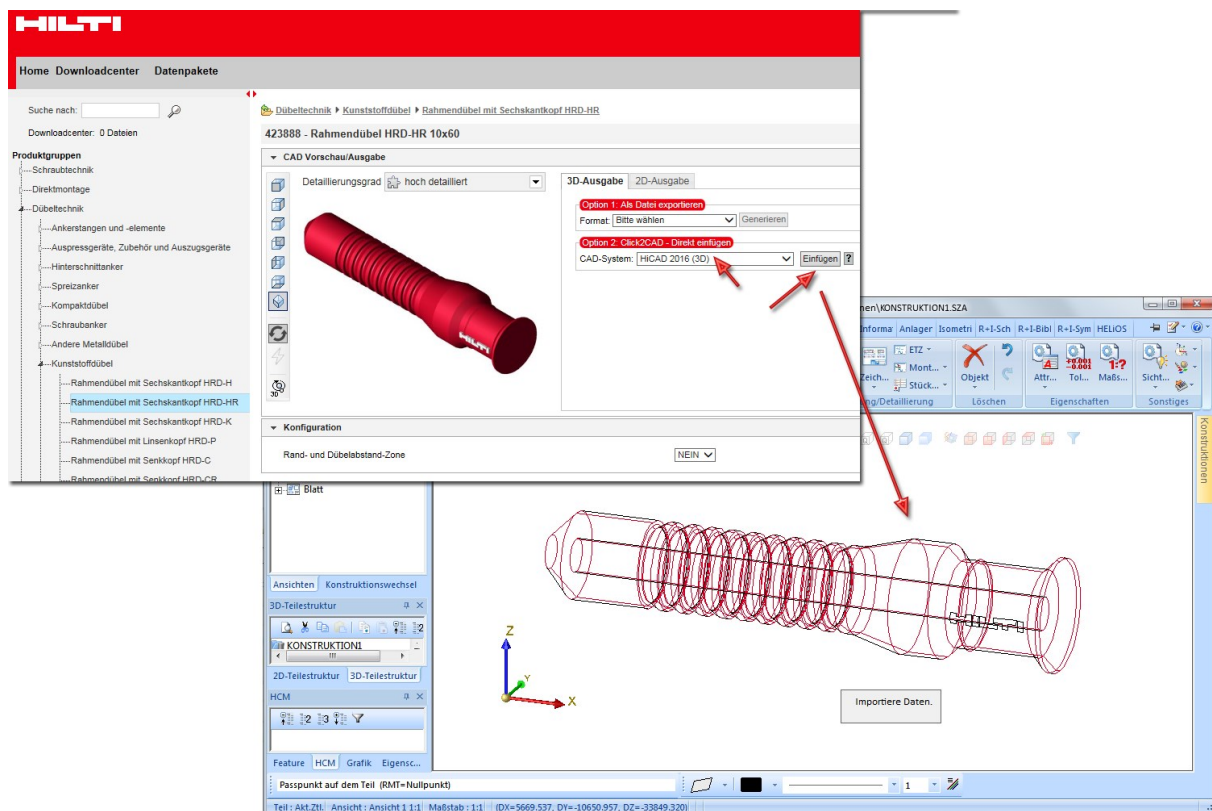
The rules according to which the Attribute mapping configuration is to be carried out can be defined for the import and the export of IFC files.



## Interaction of CADCLICK with HiCAD

The ISD Group cooperates with the company KiM GmbH (<http://www.cadclick.com/>). Parts in supplier catalogues, e.g. HILTI dowels, can now directly be inserted in HiCAD via the CADclick solutions.

The following example shows what such an integration could look like:



Select the desired part, choose the target CAD system **HiCAD** and click on the part insertion button on the right. A download onto the local Client will then take place on the background (file format: STEP/SAT).

In HiCAD the *Importing data..* message will be displayed. In the preview of the part, specify a point on the part, and then specify the position of this point in the drawing. If desired, you can insert the part multiple times. To end the insertion, press the middle mouse button.

**Please note:**

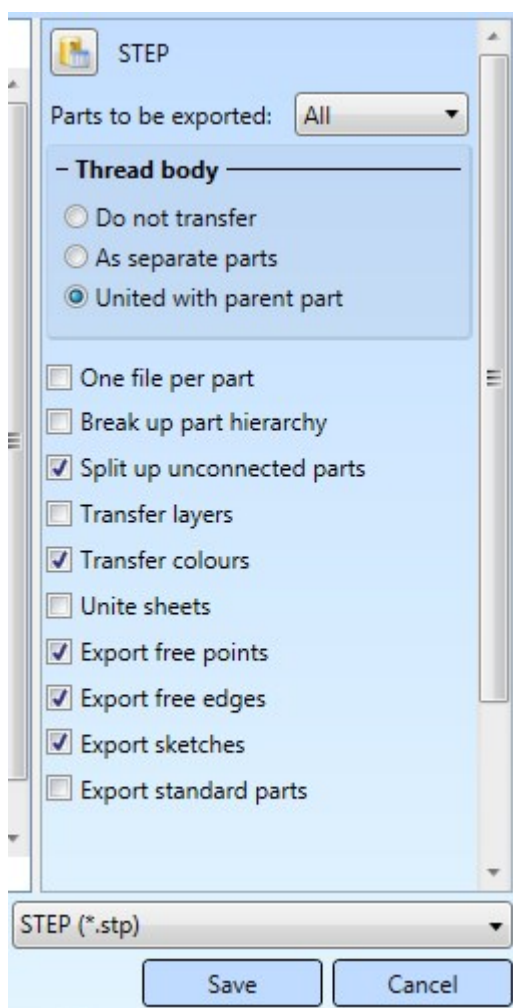


During part insertion, the order number of the part will be transferred to the HiCAD system attribute **Order No.** (Attribute name **\$ON**) . If you want the order number to be displayed also in the Part attributes mask for Standard Parts, you need to expand the system file BRW\_3DTEIL\_NORMTEIL.HDX accordingly.

## Service Pack 1 2016 (V 2101)

### STEP: Export of standard parts

The STEP interface now offers the additional option activate or deactivate the export of standard parts.



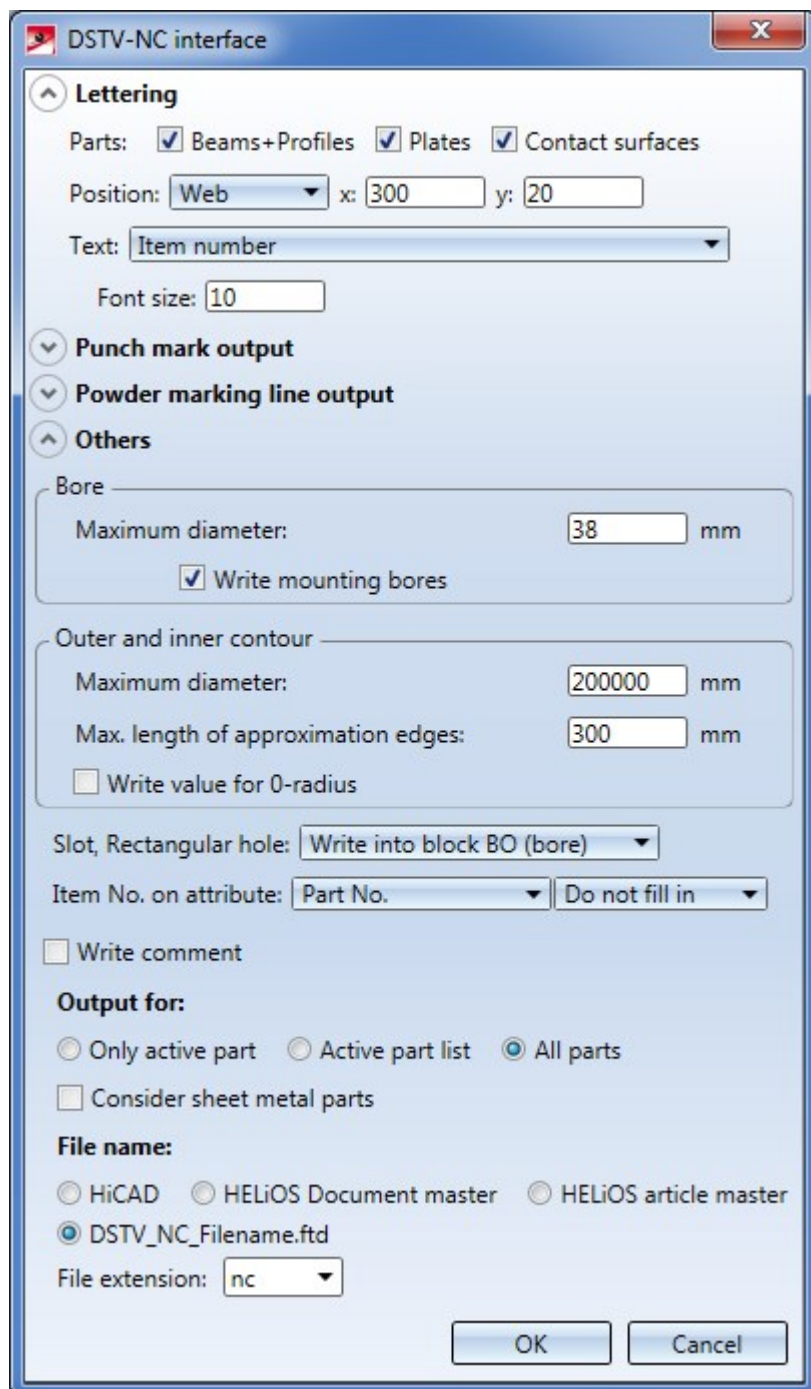
## DSTV BOM: Decimal places for shipping dimensions, Assigning of mounting screws

You can now specify for the export of the DSTV BOM whether mounting screws are (in contrast to workshop screws) to be grouped beneath the Z-set instead of the H-set.

Furthermore, you can now specify the number of decimal places for shipping dimensions.

## DSTV-NC: Configurable file name

The DSTV-NC interface now offers the option to export, besides the HiCAD name or HELiOS data, the file name via a .ftd configuration file.



The image shows the 'DSTV-NC interface' dialog box. It has a title bar with a close button. The dialog is organized into sections with expandable/collapsible headers. The 'Lettering' section is expanded, showing options for 'Parts' (Beams+Profiles, Plates, Contact surfaces), 'Position' (Web, x: 300, y: 20), 'Text' (Item number), and 'Font size' (10). Below it are 'Punch mark output' and 'Powder marking line output' sections, both collapsed. The 'Others' section is expanded, showing 'Bore' settings (Maximum diameter: 38 mm, Write mounting bores checked), 'Outer and inner contour' settings (Maximum diameter: 200000 mm, Max. length of approximation edges: 300 mm, Write value for 0-radius unchecked), 'Slot, Rectangular hole' (Write into block BO (bore)), 'Item No. on attribute' (Part No., Do not fill in), 'Write comment' (unchecked), 'Output for:' (Only active part, Active part list, All parts), 'Consider sheet metal parts' (unchecked), 'File name:' (HiCAD, HELiOS Document master, HELiOS article master, DSTV\_NC\_Filename.ftd), and 'File extension:' (nc). At the bottom are 'OK' and 'Cancel' buttons.

**DSTV-NC interface**

**Lettering**

Parts: ☒ Beams+Profiles ☒ Plates ☒ Contact surfaces

Position:  x:  y:

Text:

Font size:

**Punch mark output**

**Powder marking line output**

**Others**

Bore

Maximum diameter:  mm

☒ Write mounting bores

Outer and inner contour

Maximum diameter:  mm

Max. length of approximation edges:  mm

☐ Write value for 0-radius

Slot, Rectangular hole:

Item No. on attribute:

☐ Write comment

**Output for:**

☐ Only active part ☐ Active part list ☒ All parts

☐ Consider sheet metal parts

**File name:**

☐ HiCAD ☐ HELiOS Document master ☐ HELiOS article master

☒ DSTV\_NC\_Filename.ftd

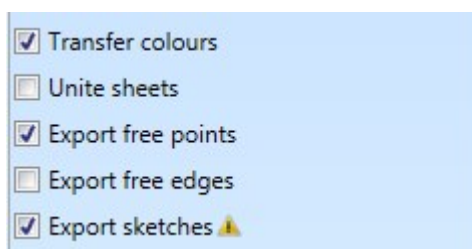
File extension:

OK Cancel

### Warning symbol when exporting sketches

Many 3-D interface export formats offer the option to include sketches in the export. In this case, it is recommended to activate the checkboxes **Export free points** and **Export free edges**; otherwise, the sketch may not be exported completely.

If you forget to activate those two checkboxes, a warning symbol will appear next to the **Export sketches** checkbox:



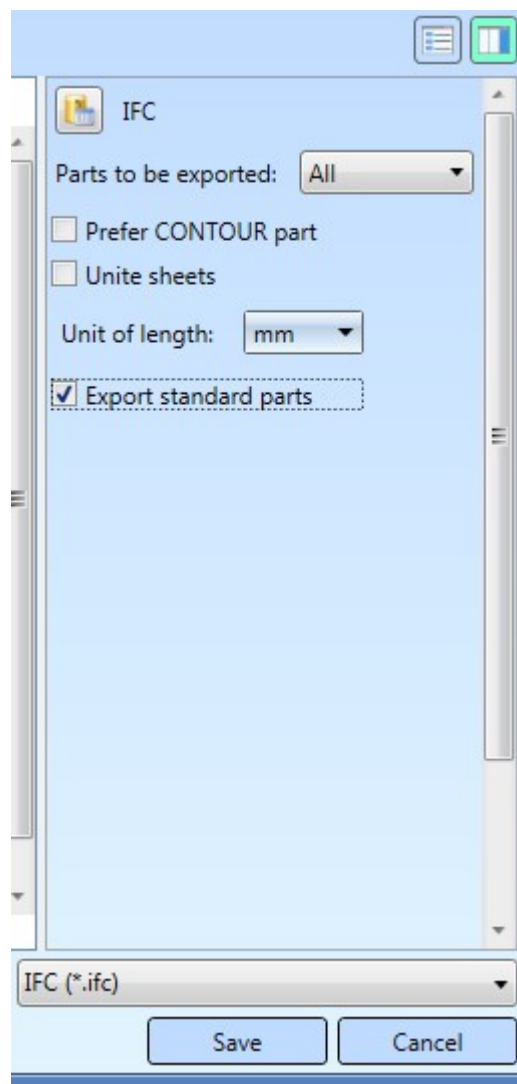
### PLMXML

The XML-based accompanying format PLMXML is a supplement to the JT interface. It allows the defining and storing of product structures and process data of complex assemblies in a readable form.

Import and export options are handled in the same way as in the JT interface.

### IFC Export: Standard Parts

For the IFC export you can specify whether Standard Parts (bolts, screws, roller bearings, springs etc.) are to be included in the export or not by activating or deactivating the Export standard parts checkbox.



## 2D-DXF - Taking into account of UOMs

Thanks to an optimisation of the 2D-DXF interface, the unit of measurement (British or American inches) will now be taken into account during import and export (via the \$MEASUREMENT variable).

## Update CADfix 10.0 SP2

Thanks to the CADfix update to Version 10.0 SP2, the import format version Inventor 2016 is now available in HiCAD 2016 SP1.

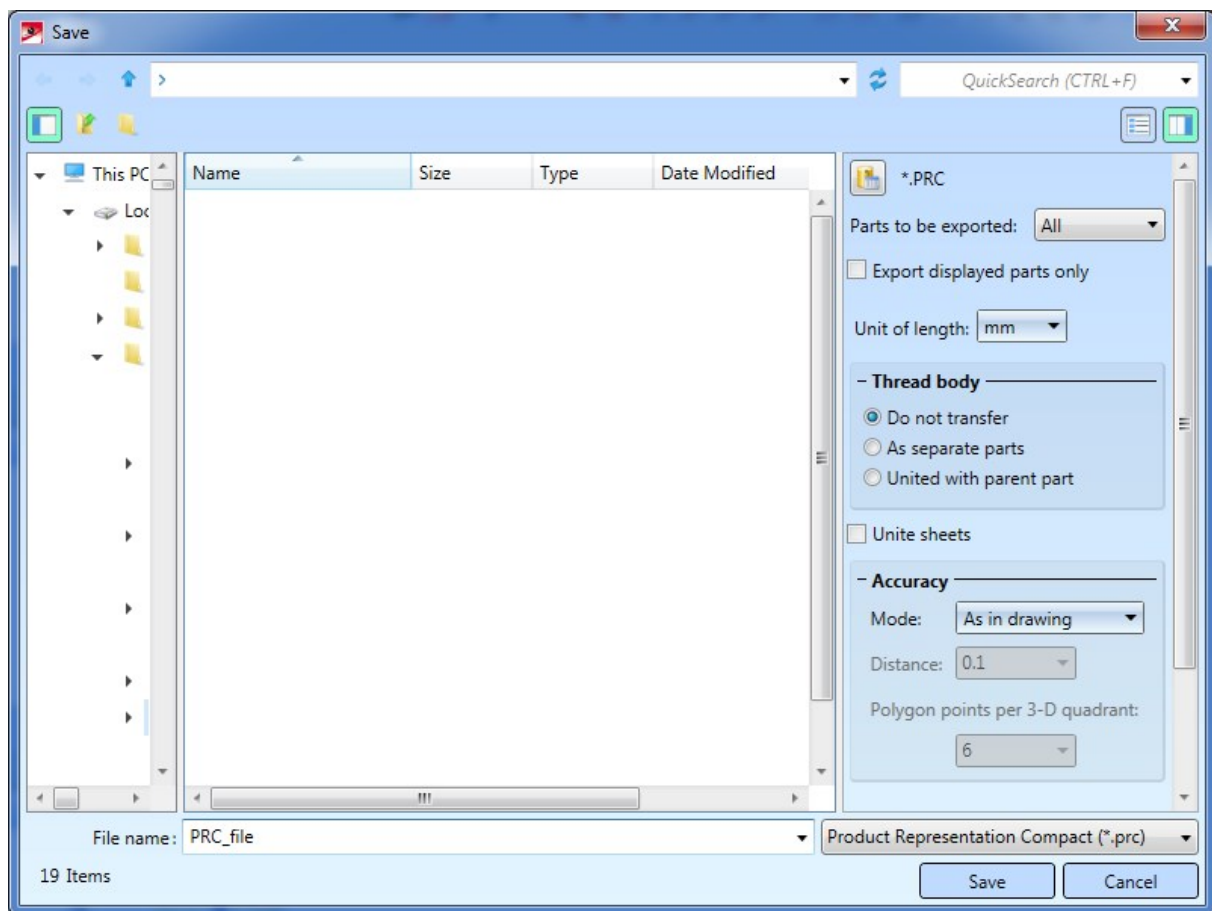


## Major Release 2016 (V 2100)

### 3DPDF, Universal 3D and Product Representation Compact

A new export format, **Product Representation Compact (.prc)**, is now available. It can be exported in the same way as the formats 3DPDF (.pdf) and Universal (\*.u3d).

All three file formats offer various options for the export of thread bodies and the unioning of sheets.



### IFC

A significant performance increase could be achieved for IFC export: Depending on the size of the drawing to be exported, exports now take place 4 - 5 times faster.

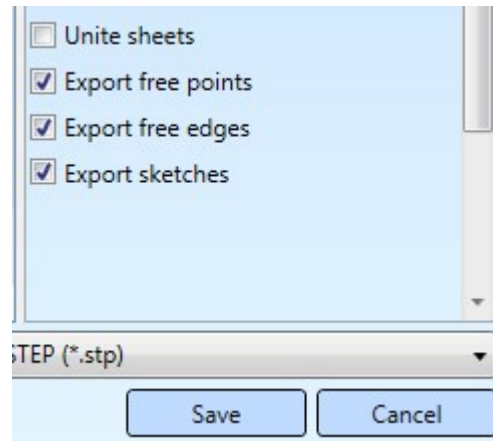
## STEP

The STEP export functionality now offers the additional option to include sketches into the export.



### Please note:

If the **Export sketches** option has been selected, you should also activate the Export free edges and the Export free points options, to prevent that an "empty sketch" will be exported by mistake.

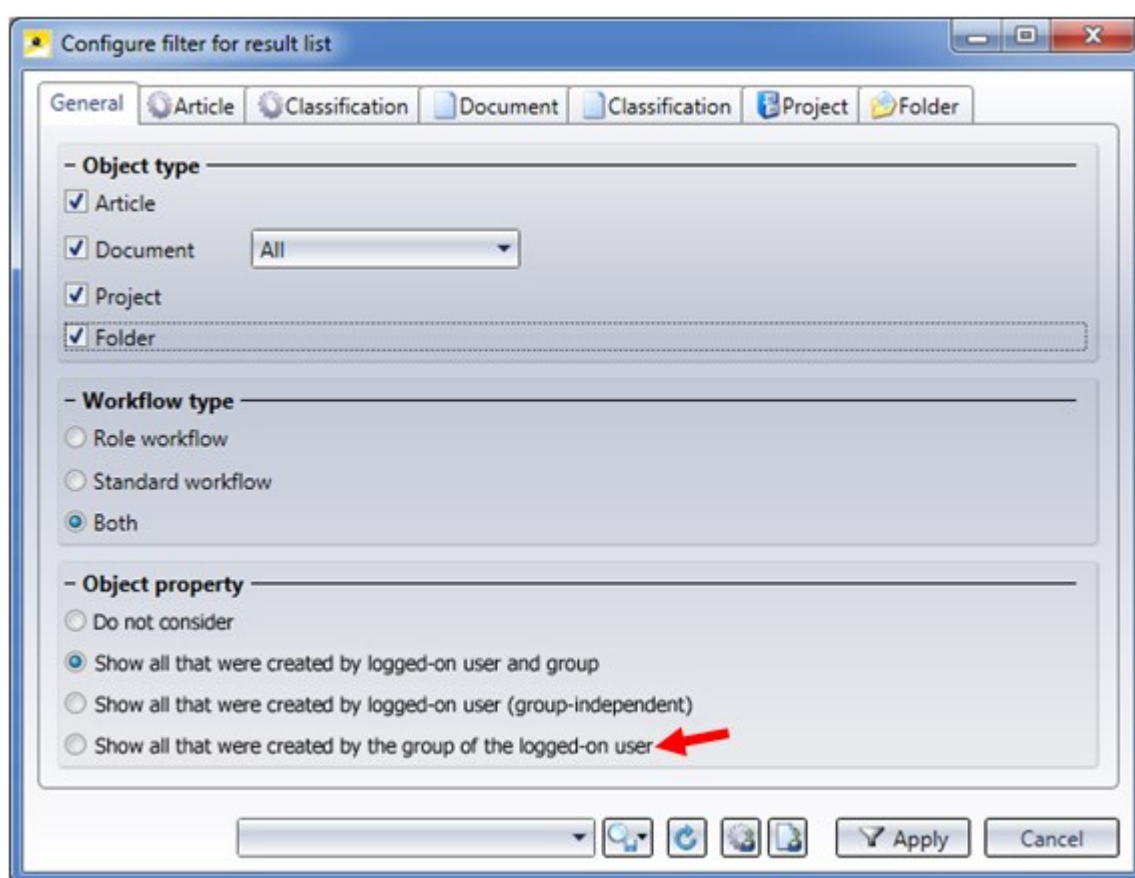


# HELiOS Desktop

Service Pack 2 2016 (V 2102)

## ToDo list - Enhanced result list filter configuration options

The result list filter configuration options for ToDo lists have been enhanced: In the **Object property** area of the **Configure filter for result list** dialogue window you will now find a new option, allowing you to display all objects created by the group of the logged-on user (i.e. not only those that were created by the user him-/herself, but also those that were created by other members of his/her group).



## Editing in result lists

Similar to the editing of objects via the editing mode of their respective detail masks, attributes can also be edited directly in result lists.

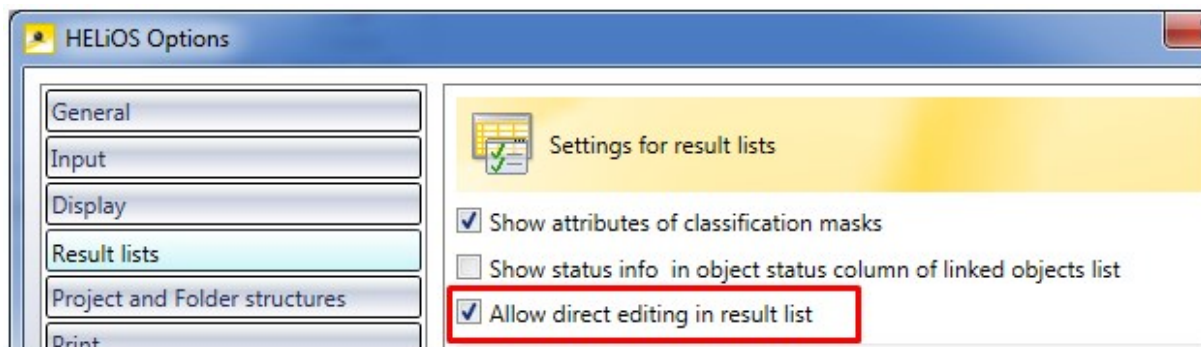
Just click to edit a result list row. You can, for instance, change the document number or article number, as well as the folder and project names. Attributes such as Designation, the Document type or the Part type can also directly be changed in the result list in this way, without having to call the editing window first.

| Articles Documents Documents and Articles Objects |              |       |       |             |               |                |
|---|--------------|-------|-------|-------------|---------------|----------------|
| Standard  |              |       |       |             |               |                |
| R#  | Document No. | Sheet | Index | Designation | Document type | Release status |
|   | DN-000384    | 1     |       | Draft       | HiCAD Draw    | In Progress    |

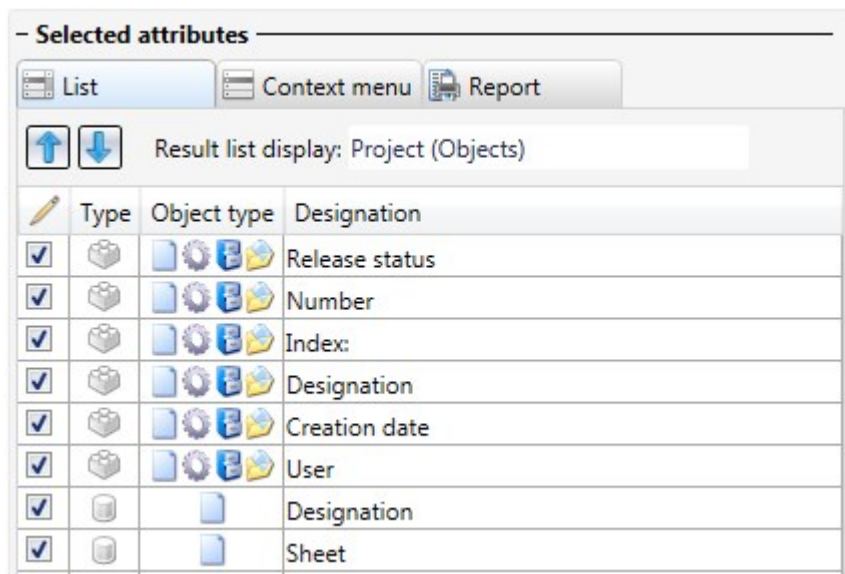
  


| Articles Documents Documents and Articles Objects |           |       |                        |                       |             |
|---|-----------|-------|------------------------|-----------------------|-------------|
| Standard  |           |       |                        |                       |             |
| R#  | Number    | Index | Designation            | Part type             | Designation |
|   | SN-000001 |       | Slip-on gear mechanism | Product               |             |
|   | SN-000002 |       | Clamping element       | Assembly              |             |
|   | SN-000003 |       | Housing assembly       | Assembly              |             |
|   | SN-000004 |       | Gear wheel assembly    | Assembly              |             |
|   | SN-000005 |       | Pinion assembly        | Asserr                |             |
|   | SN-000006 |       | Gearbox housing        | Assembly              |             |
|   | SN-000007 |       | Gearbox cover          | Drawing               |             |
|   | SN-000008 |       | Gear wheel             | Individual part       |             |
|   | SN-000009 |       | Pinion                 | Loose part            |             |
|   | SN-000010 |       | Guide sleeve           | Others                |             |
|   | SN-000011 |       | Spring guide sleeve    | Part                  |             |
|   | SN-000012 |       | Lock ring              | Product               |             |
|   | SN-000013 |       |                        | Raw-part+Plant-design |             |
|   |           |       |                        | Standard-part         |             |

The possibility to edit result list rows can be set globally in the HELiOS Options.

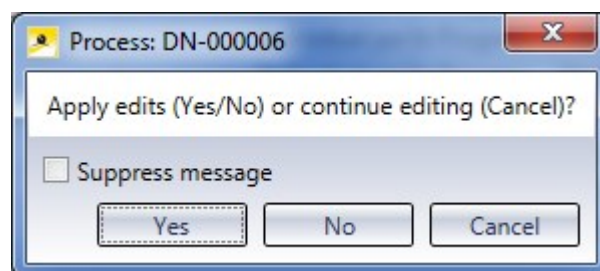


If a direct editing of result lists is generally allowed in HELiOS, you can also specify for individual attributes within the result lists whether they are to be editable or not.



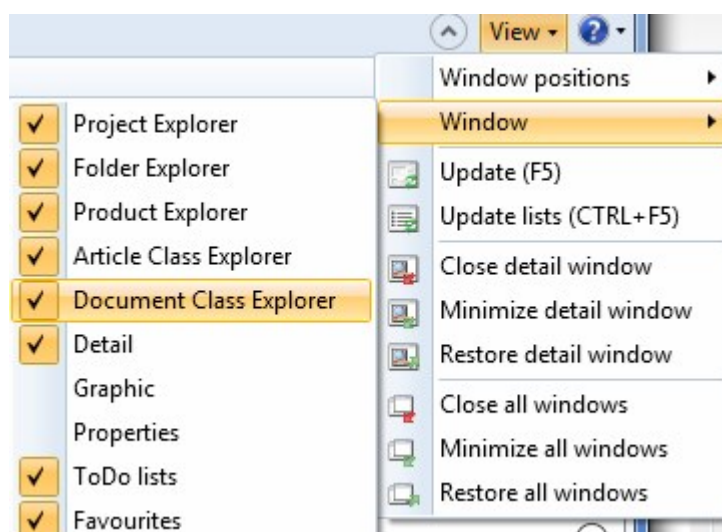
If the option has been deactivated, the checkbox column beneath the pen symbol  will not be displayed in the result list configuration.

The security prompt that is displayed after direct editing of result list rows can be suppressed and switched on again if desired:



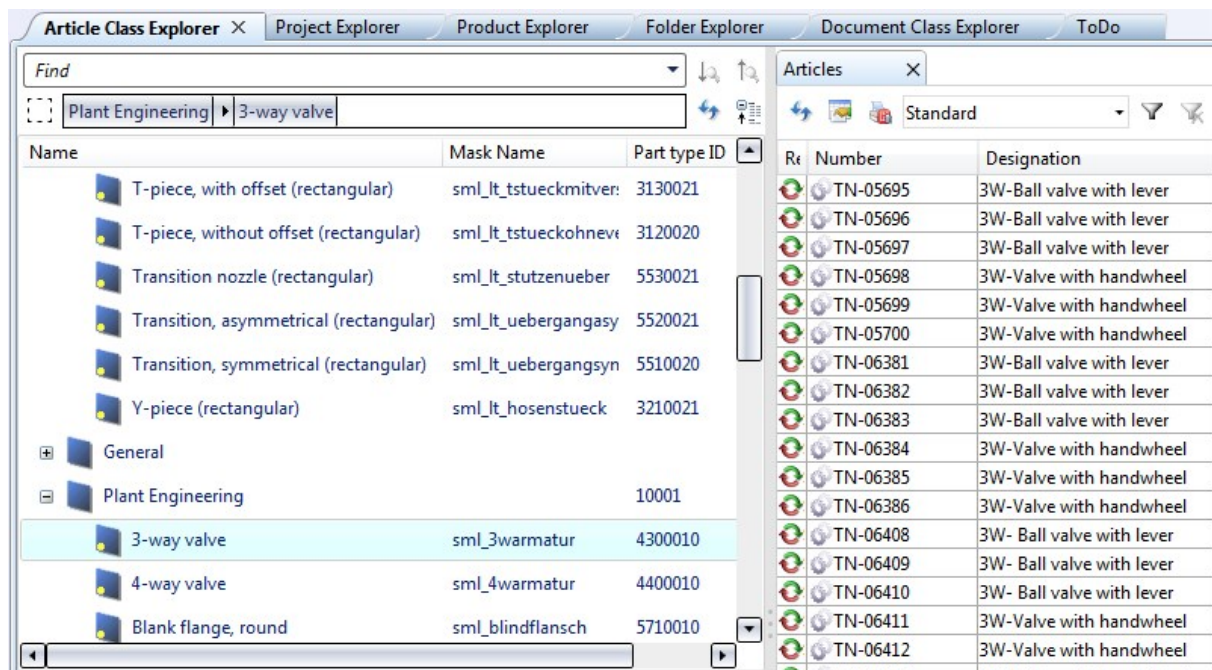
## Article and Document Class Explorer

In addition to the Product, Project and Folder Explorer you can now add the new **Article Class Explorer** and **Document Class Explorer** to the user interface: Choose **View > Window** at the top right of the HELiOS screen and activate the required Explorer.

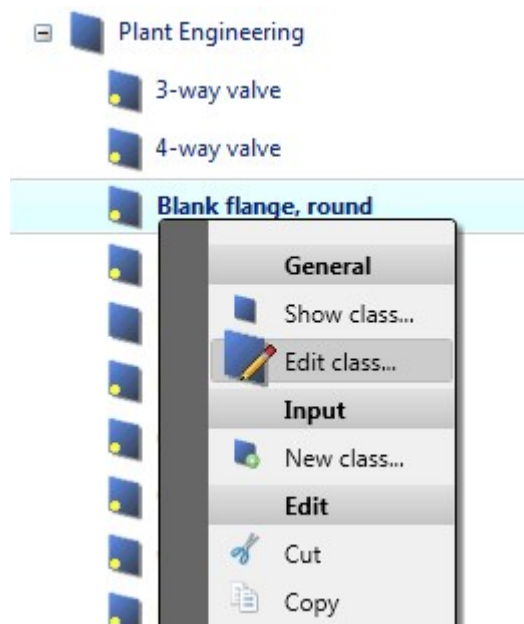




The tree structure of the respective Explorer contains all classes and sub-classes that were created for the corresponding object type. Identify a class in the Explorer and the result list will display all articles and documents that have been assigned to this class.



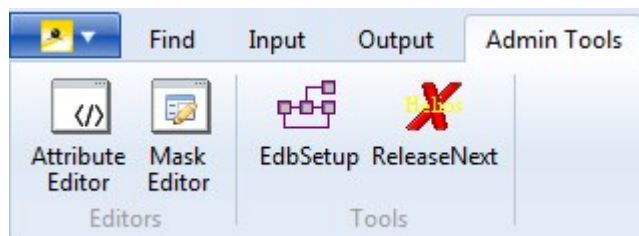
Right-click to show and/or edit the classes, or to add new ones.





## Admin Tools

The **Admin Tools** tab now contains some additional functions: You can now also open the User and Workflow management tool **EDBSETUP** and the **ReleaseNext** tool for the listing and unlocking of documents directly from the HELiOS Desktop.

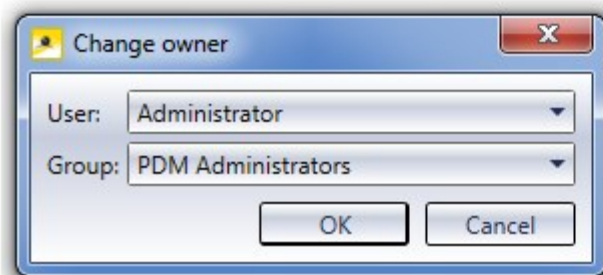
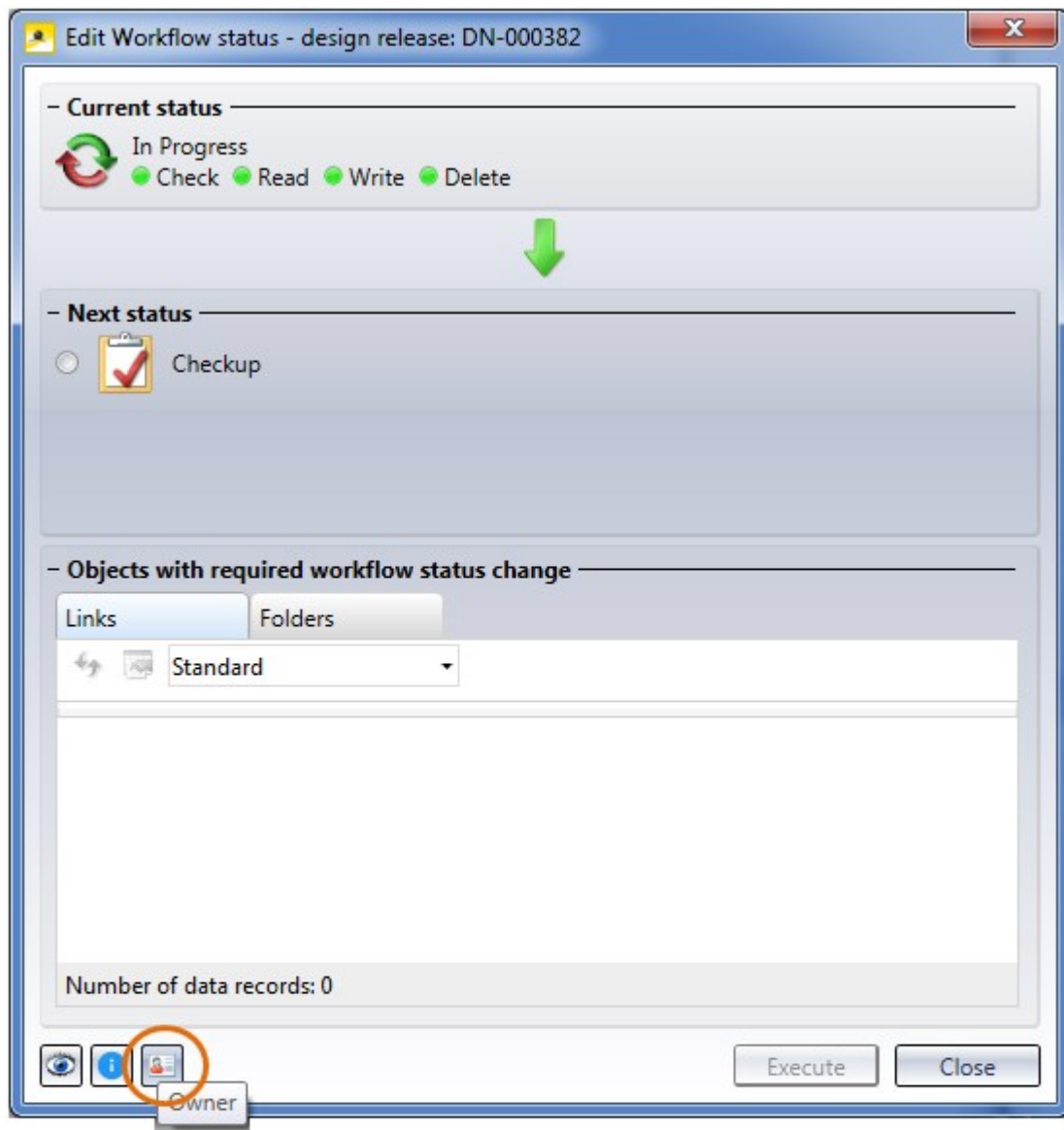


The functions for Classification that were available in the **Admin Tools** tab of older versions have been withdrawn due to the introduction of the new Class Explorers.

The **Admin Tools** tab is only visible if you have logged in as Administrator.

## Workflow: Change owner

If you work with the classic Workflow (i.e. not with the Role Workflow), you have the option to change the owners and groups of Workflows in the Edit workflow status dialogue window:

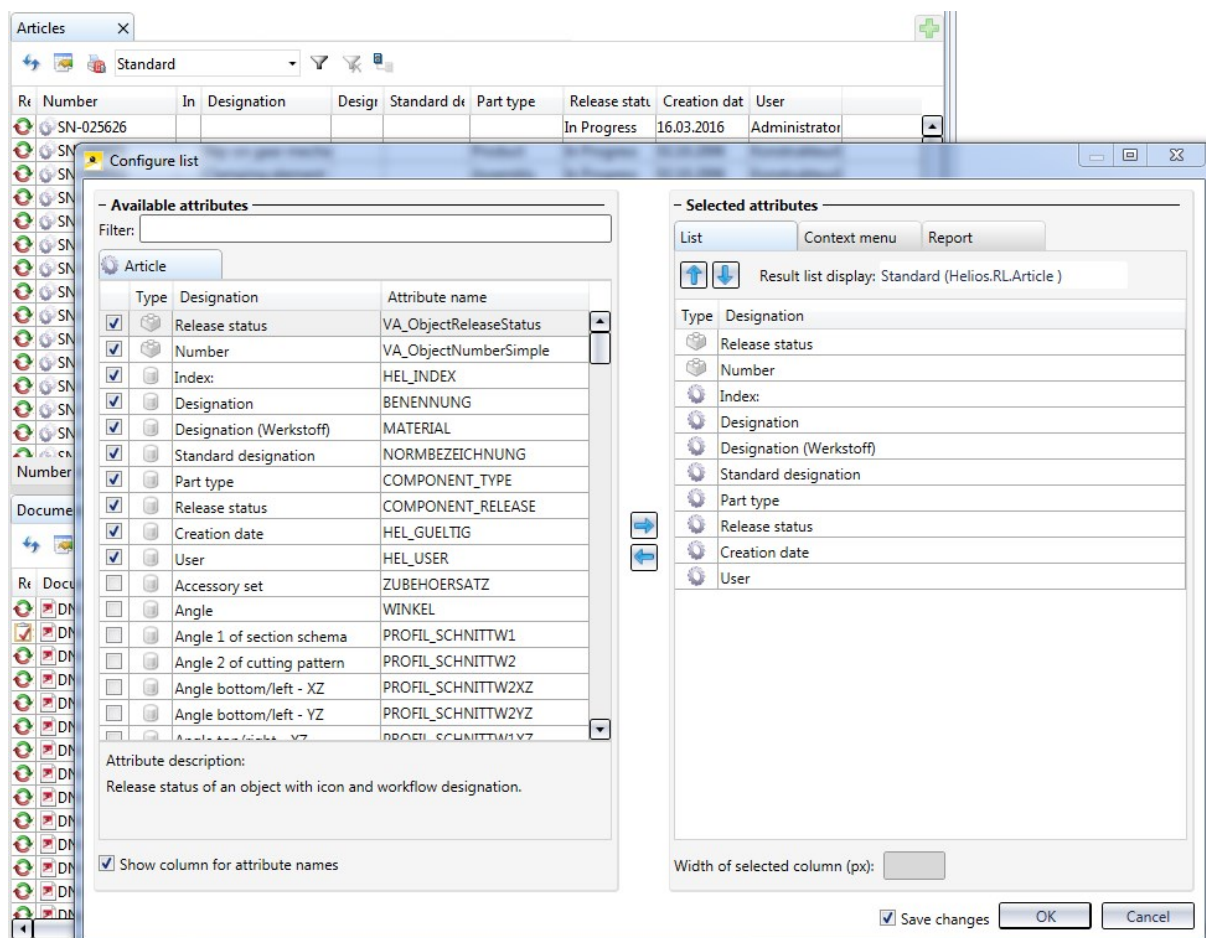


## Service Pack 1 2016 (V 2101)

### New result list configuration options

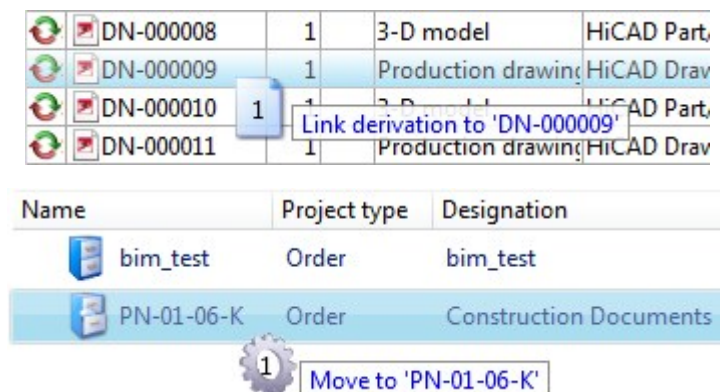
The configuration of result lists is now even more convenient and easier to handle; some additional options are available, too.

#### Configure list

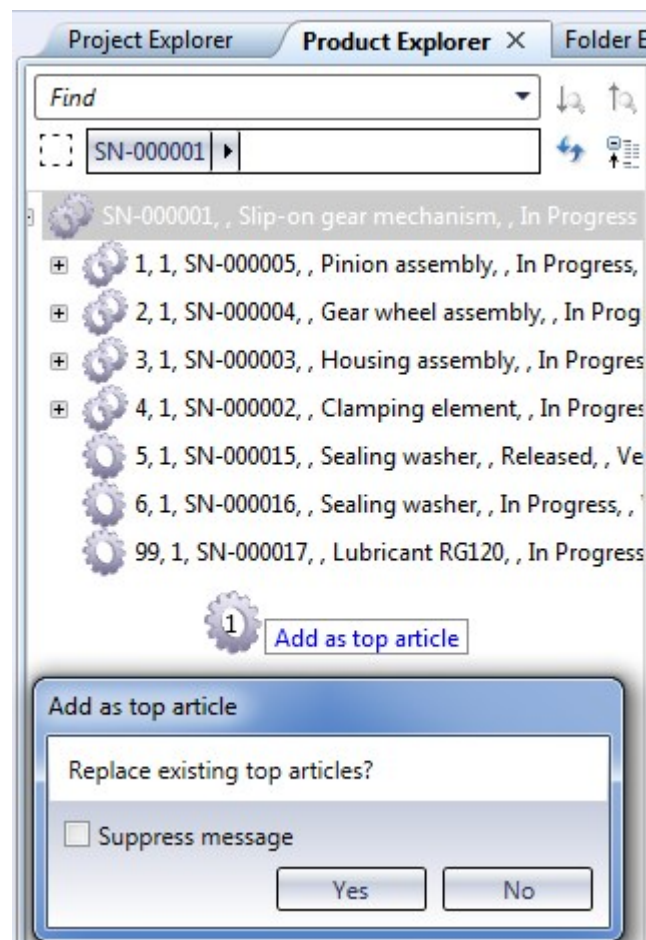


## Product Structure: Drag & Drop behaviour, tooltip infos

During all Drag & Drop actions, descriptive tooltip texts will be displayed:



If you drag an article into the Product Explorer where a Product Structure already exists, HiCAD will ask you whether the existing Product Structure is to be replaced with the new one, or if the top article is to be added to the existing Explorer contents.



On the **Display** tab of the HELiOS Desktop you can activate or deactivate this dialogue. If the corresponding checkbox has been deactivated, has been disabled, the contents of the Product Explorer will always be replaced with the top article.

In this way you can also drag multiple selected articles into the Product Explorer.

HELiOS Options: Display tab

**HELiOS Options**

General  
Input  
**Display**  
Result lists  
Project and Folder structures  
Print  
Import/Export  
Workflow  
Log  
Database  
Document type  
Help

**Settings for display of windows**

**– Projects**  
Project selection:  
☒ List form  
☐ Structure

**– Folders**  
Folder selection:  
☒ List form  
☐ Structure

**– Object detail**  
☐ Always show in foreground  
☒ Show in foreground in search windows  
☐ Show tabs for classification masks  
☐ Show new contents in same window

**– Article detail**  
Shown usage levels:  
☒ All  
☐ Last set levels

**– Masks**

**– Product Explorer**

**– Drag & Drop**  
☐ Show query for Move operation  
☒ Confirmation of product structure window clearing

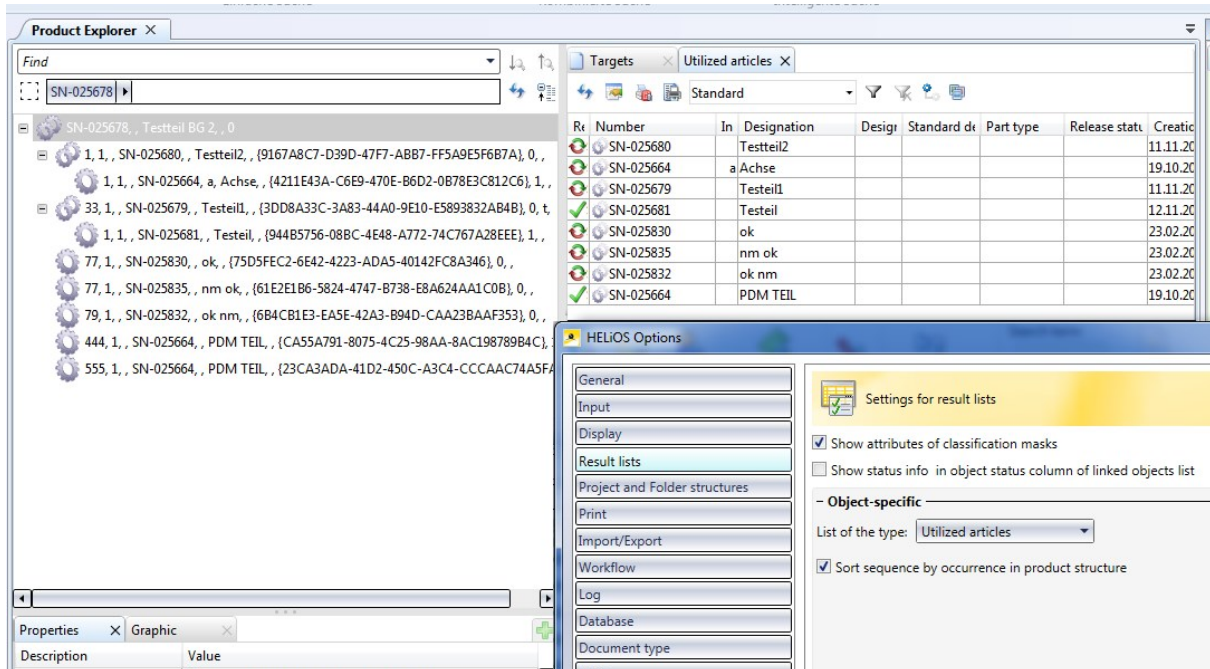
OK Cancel



## Product Structure: Sort sequence of Utilized articles

On the **Result lists** tab of the of the **HELiOS Options** dialogue you can specify for lists of the type **Utilized articles** whether these utilized articles in the Product Structure context are to be shown with the same sorting as in the Product Structure when they are called for the first time.

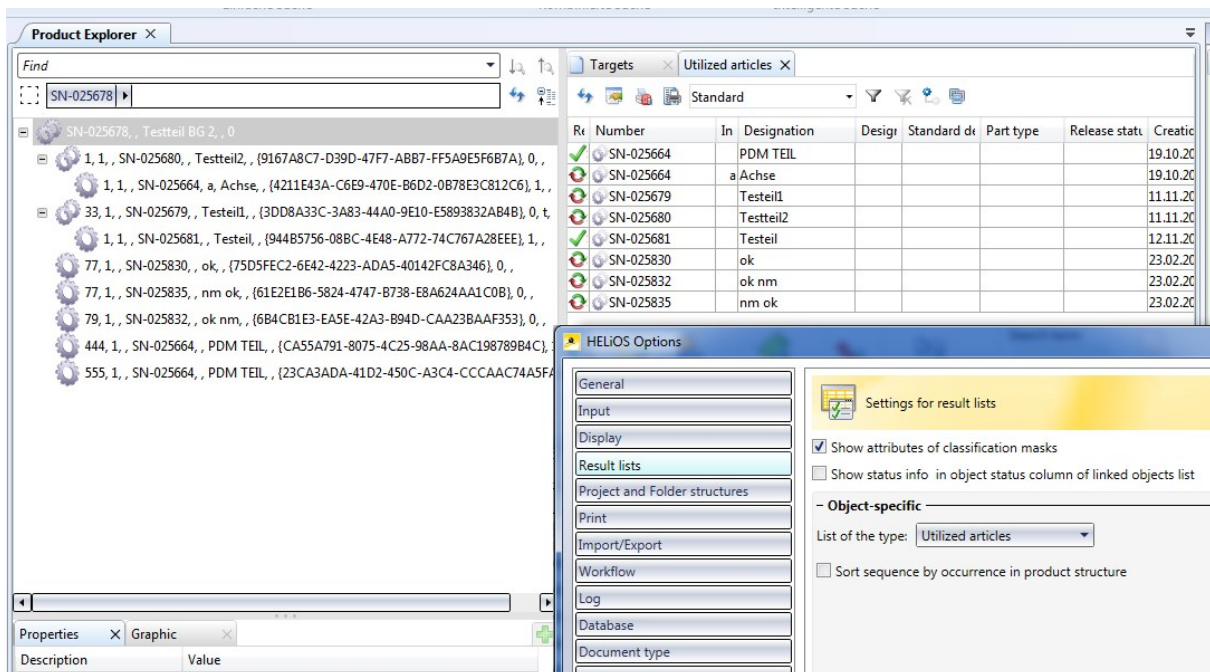
*Sort sequence of articles by occurrence in product structure*



The screenshot shows the HELiOS Options dialog with the 'Result lists' tab selected. The 'Settings for result lists' section has the 'Sort sequence by occurrence in product structure' checkbox checked. The 'List of the type' is set to 'Utilized articles'.

| Re | Number    | In | Designation | Design | Standard de | Part type | Release statu | Creation |
|----|-----------|----|-------------|--------|-------------|-----------|---------------|----------|
|    | SN-025680 |    | Testteil2   |        |             |           |               | 11.11.20 |
|    | SN-025664 | a  | Achse       |        |             |           |               | 19.10.20 |
|    | SN-025679 |    | Testteil    |        |             |           |               | 11.11.20 |
|    | SN-025681 |    | Testteil    |        |             |           |               | 12.11.20 |
|    | SN-025830 |    | ok          |        |             |           |               | 23.02.20 |
|    | SN-025835 |    | nm ok       |        |             |           |               | 23.02.20 |
|    | SN-025832 |    | ok nm       |        |             |           |               | 23.02.20 |
|    | SN-025664 |    | PDM TEIL    |        |             |           |               | 19.10.20 |

*Do not sort sequence of articles by occurrence in product structure*



The screenshot shows the HELiOS Options dialog with the 'Result lists' tab selected. The 'Settings for result lists' section has the 'Sort sequence by occurrence in product structure' checkbox unchecked. The 'List of the type' is set to 'Utilized articles'.

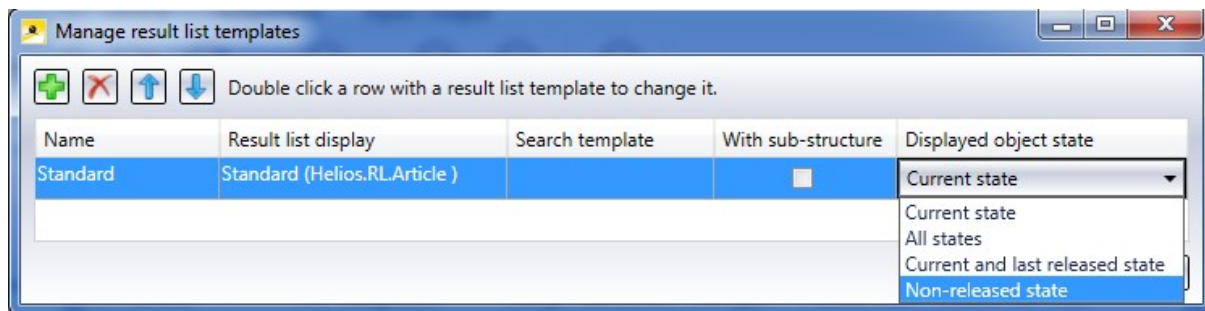
| Re | Number    | In | Designation | Design | Standard de | Part type | Release statu | Creation |
|----|-----------|----|-------------|--------|-------------|-----------|---------------|----------|
|    | SN-025664 |    | PDM TEIL    |        |             |           |               | 19.10.20 |
|    | SN-025664 | a  | Achse       |        |             |           |               | 19.10.20 |
|    | SN-025679 |    | Testteil    |        |             |           |               | 11.11.20 |
|    | SN-025680 |    | Testteil2   |        |             |           |               | 11.11.20 |
|    | SN-025681 |    | Testteil    |        |             |           |               | 12.11.20 |
|    | SN-025830 |    | ok          |        |             |           |               | 23.02.20 |
|    | SN-025832 |    | ok nm       |        |             |           |               | 23.02.20 |
|    | SN-025835 |    | nm ok       |        |             |           |               | 23.02.20 |



## Utilized articles: Displayed object state (index)

In the result list templates management for **Utilized articles** of the Product Structure you can make a selection concerning the displayed object state:

- **Current state:** Displays the current versions of the objects in the list.
- **All states:** Displays all versions of objects in the list.
- **Current and last released state:** The current and last released state of objects will be displayed in the list.
- **Non-released state:** All versions of objects which have not been released yet will be displayed in the list.



## Utilized articles in structure version

The list of Utilized articles has been added to the **Product structure versions** window in the form of a new tab.

The working approach with the sort sequence and the displayed object state replaces the older context menu functions **Parts in 1-level assembly**, **Parts in assembly** and **Parts with non-released index**.

## Structure version

Product structure versions

Temporary

Displayed object index: ☒ Highest index ☐ Released index

SN-000001, Slip-on gear mechanism, In Progress

1, 1, SN-000005, Pinion assembly, In Progress, Ersatzteil

2, 1, SN-000004, Gear wheel assembly, In Progress, Ersatzteil

3, 1, SN-000003, Housing assembly, In Progress, Ersatzteil

4, 1, SN-000002, Clamping element, In Progress, Ersatzteil

5, 1, SN-000015, Sealing washer, Released, Verschleißteil

6, 1, SN-000016, Sealing washer, In Progress, Verschleißteil

99, 1, SN-000017, Lubricant RG120, In Progress, getrennt verpacken, Ersatzteil

Properties

Utilized articles

Standard

Standard

Save as...

Manage...

| Rt | Number    | In | Designation       | Design | Standard d | Part type      | Relea  |
|----|-----------|----|-------------------|--------|------------|----------------|--------|
|    | SN-000002 |    | Clamping element  |        |            | Assembly       | In Pro |
|    | SN-000003 |    | Housing assembly  |        |            | Assembly       | In Pro |
|    | SN-000004 |    | Gear wheel assemb |        |            | Assembly       | In Pro |
|    | SN-000005 |    | Pinion assembly   |        |            | Assembly       | In Pro |
|    | SN-000017 |    | Lubricant RG120   |        |            | Others         | In Pro |
|    | SN-000015 |    | Sealing washer    |        |            | Individual par | Releas |
|    | SN-000016 |    | Sealing washer    |        |            | Individual par | In Pro |
|    | SN-000017 |    | Lubricant RG120   |        |            | Others         | In Pro |

Number of data records: 8

Original

Current state of product structure as saved

SN-000001, Slip-on gear mechanism, In Progress

1, 1, SN-000005, Pinion assembly, In Progress, Ersatzteil

2, 1, SN-000004, Gear wheel assembly, In Progress, Ersatzteil

3, 1, SN-000003, Housing assembly, In Progress, Ersatzteil

4, 1, SN-000002, Clamping element, In Progress, Ersatzteil

5, 1, SN-000015, Sealing washer, Released, Verschleißteil

6, 1, SN-000016, Sealing washer, In Progress, Verschleißteil

99, 1, SN-000017, Lubricant RG120, In Progress, getrennt verpacken, Ersatzteil

Properties

Utilized articles

Utilized items

Standard

Standard

Save as...

Manage...


| Rt | Number    | In | Designation       | Design | Standard d | Part type      | Relea  |
|----|-----------|----|-------------------|--------|------------|----------------|--------|
|    | SN-000002 |    | Clamping element  |        |            | Assembly       | In Pro |
|    | SN-000003 |    | Housing assembly  |        |            | Assembly       | In Pro |
|    | SN-000004 |    | Gear wheel assemb |        |            | Assembly       | In Pro |
|    | SN-000005 |    | Pinion assembly   |        |            | Assembly       | In Pro |
|    | SN-000017 |    | Lubricant RG120   |        |            | Others         | In Pro |
|    | SN-000015 |    | Sealing washer    |        |            | Individual par | Releas |
|    | SN-000016 |    | Sealing washer    |        |            | Individual par | In Pro |

Number of data records: 7

News Overview

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## Enhanced HELIOS-URL

The context menu function  **Copy HELIOS URL** saves the Head ID of a HELIOS object. Via the Head ID, the corresponding HELIOS object can then also be called externally (e.g. from an external software) by means of the HELIOS protocol (helios://).

Also, the HELIOS protocol can be used in conjunction with the document number, article number or project number to identify and call the corresponding HELIOS objects (Documents, Articles, Projects).

## Improved preview image concept

Due to problems and instabilities in connection with external viewer software that occurred in the past, the preview image concept for graphic previews in detail windows of HELIOS has been improved.

Now, neutral formats are used, which offers a number of advantages:

- Through avoiding of third party software, the stability of HELIOS will no longer be compromised.
- No external components need to be installed and set up.
- Neutral formats are also very suitable for other processes, e.g. for Redlining.
- Neutral formats can also be generated for various document formats. This means that a consistent preview mechanism exists for those formats, including a consistent operation (e.g. of the Viewer integrated in HELIOS).

Settings for the preview are still specified in the system file hel\_preview.ini.

In the first step, the HELIOS Add-in for Inventor has been adjusted. The HELIOS Add-in saves the neutral formats for previews when passing documents to the HELIOS Vault Server.

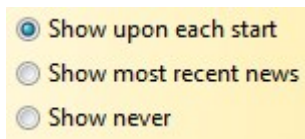
## Major Release 2016 (V 2100)

### The 'News' window

In order that you can take full advantage of the numerous new features and enhancements offered by HELiOS 2016, check out the videos explaining the new and changed HELiOS functionalities in our new 'News' window.

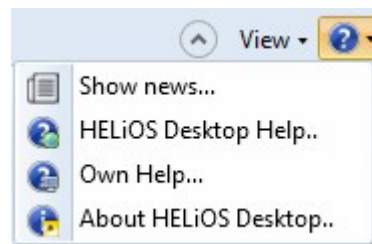
We will provide you with such videos with each Major Release and Service Pack to keep you constantly informed in the best possible way.

Please note that you can configure the display of the News window at the bottom left of the window:



We recommend using the **Show most recent news** option to ensure that you will never miss any news about HELiOS.

If you have selected the **Show never** option you can call the News window again at any time and change the display option by clicking on the "?" symbol at the top right and selecting **Show news**.



### Terminological changes, Glossary

Some terms in the UI of HELiOS 2016 have been changed on the request of our customers.

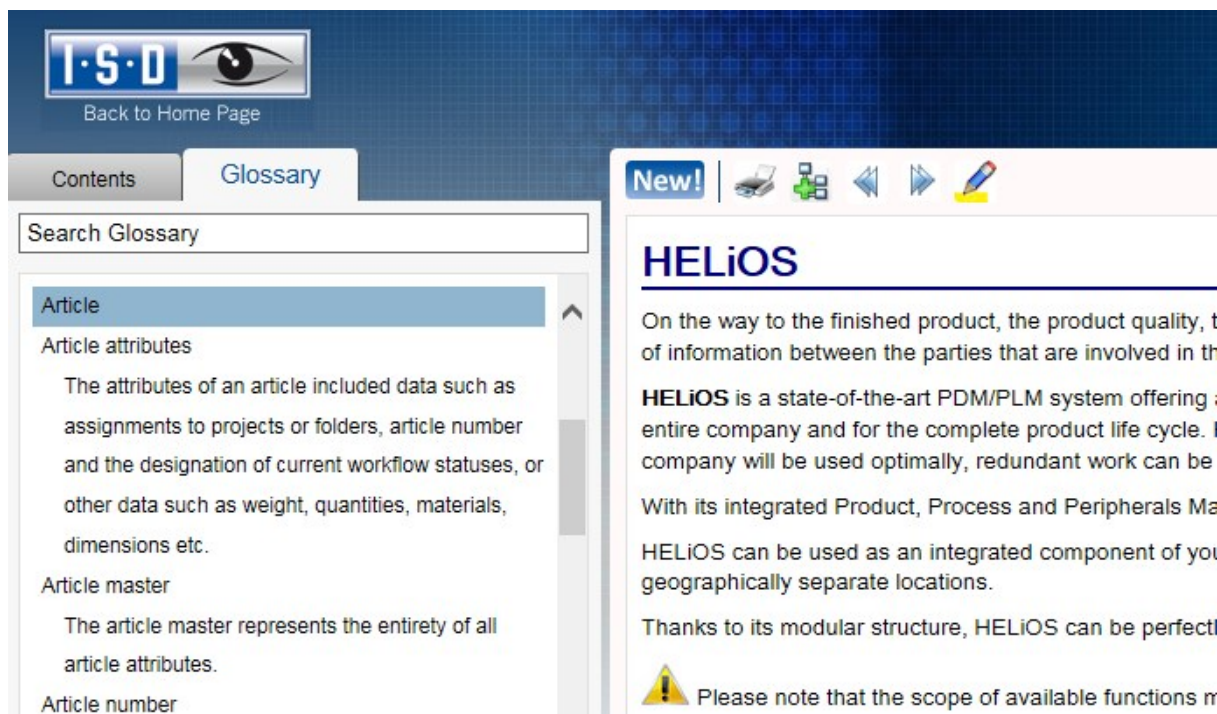
To emphasize the difference between CAD parts and PDM parts, the HELiOS "Parts" have now be renamed to **Articles**, since these are articles in the HELiOS database. "Part masters" are now called **Article masters**, "part master data" are now called **Article attributes**.



Please note that due to technical backgrounds, designations such as "Part (R)" and link classes such as "Bauteil-Teilegeometrie" ("Part-Part geometry") or "Bauteil-Dokument" ("Part-Document") in the standard database supplied with HELiOS continue to exist for the moment.

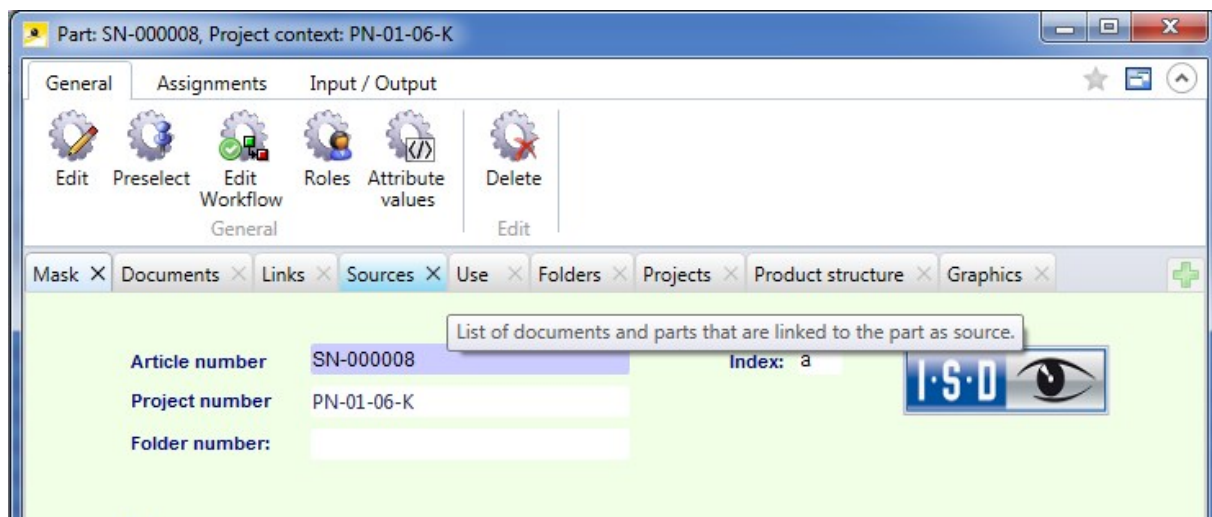
The function for a revision of Articles or Documents with the assigning of a new index to them is now called **Create index**. A "Modified drawing" still describes the process of a modification.

An overview of HELiOS-specific terms is now provided in an additional **Glossary** tab in the Online Help.



## Changes in the UI

Together with some adjustments of the HELiOS Desktop UI, some tooltips have also been revised. If you move the cursor over a UI element, e.g. a tab, a tooltip containing a short description will be displayed.



When you move objects, HELiOS will ask you confirm the Move operation:

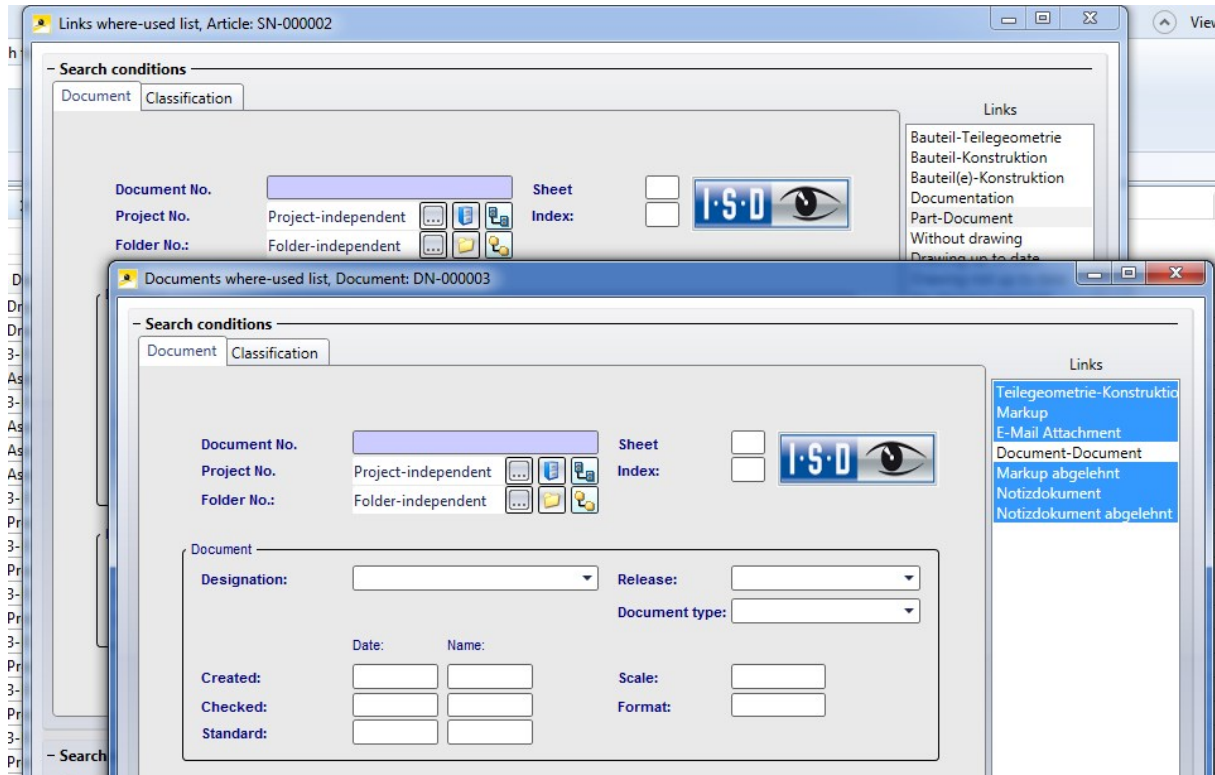


If desired, you can suppress this message by activating the same-named checkbox, or, on the **Display** tab of the **HELiOS Options**, by deactivating the **Show query for Move operation** checkbox beneath **Drag & Drop**.

Furthermore, the overall performance of the HELiOS Desktop UI has been increased. This applies in particular to masks using suggested values, the Utilized articles list and the transfer of very large product structures from the CAD system (up to 6 times faster now).

The search mask technology has been incorporated into the links where-used lists for documents or articles. Right-click the desired part or document and select the function in the context menu.

*Search masks for links where-used lists*





## Representation of assemblies in result lists

You have now various options for the representation of article numbers in the article result lists:













In the default setting, there is no graphical difference between individual part articles and assemblies (articles with subordinate items). The corresponding attribute is "Number (VA\_ObjectNumber)".

If you want this difference to be shown, you should activate the attribute "Number (VA\_ObjectnumberWithIcon)" to be displayed in the result list.

|                                     |                                  |
|-------------------------------------|----------------------------------|
| <input type="checkbox"/>            | NPS3 (inch) (NPS3_INCH)          |
| <input type="checkbox"/>            | Number (VA_ObjectNumber)         |
| <input checked="" type="checkbox"/> | Number (VA_ObjectnumberWithIcon) |

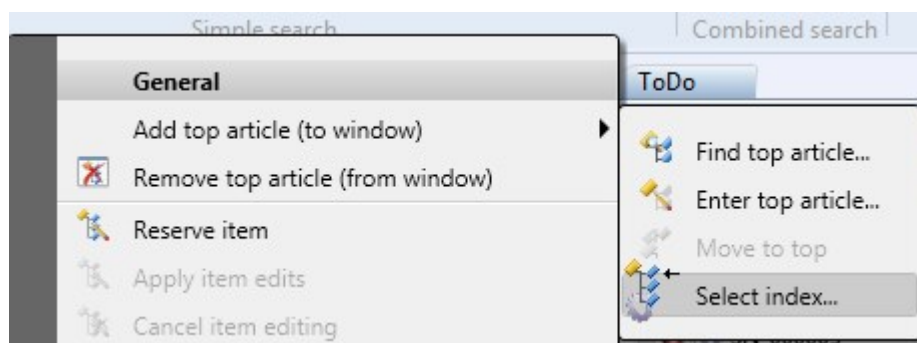
This slightly more performance-intensive option displays individual parts as a gearwheel symbol, and assemblies as a double-gearwheel symbol.

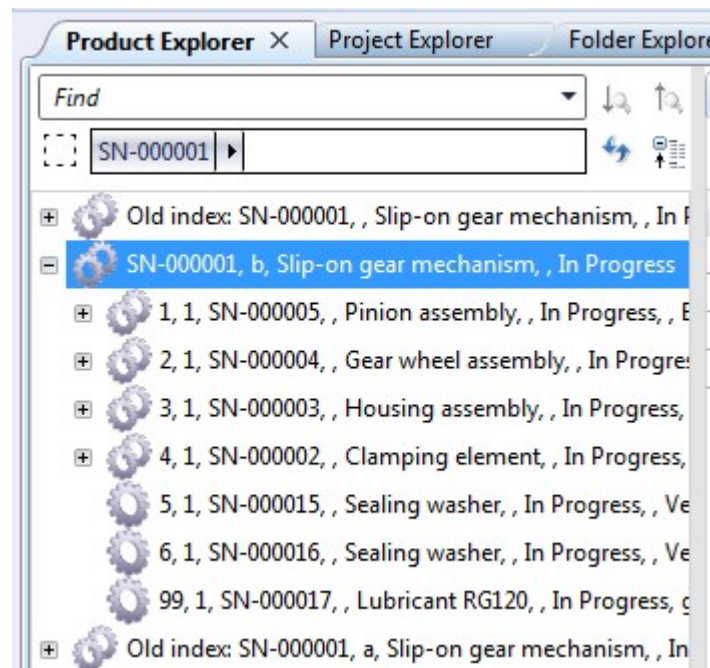
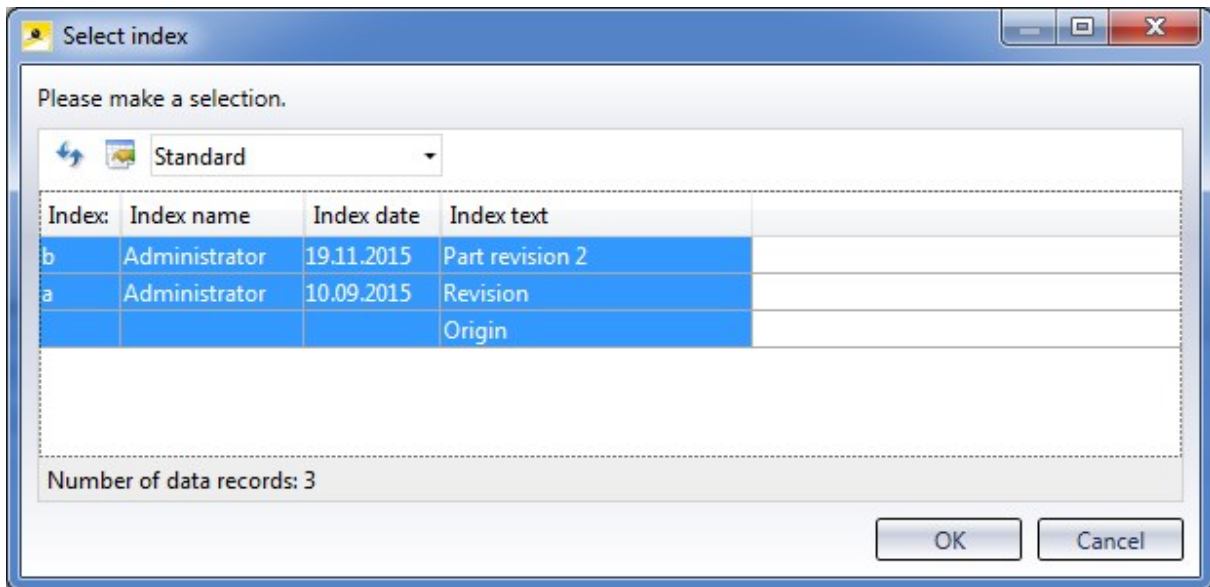
In the image below, the leftmost column is the "Number (VA\_ObjectNumber)" column, and the column to the right of it is the "Number (VA\_ObjectnumberWithIcon)" column:

| Number  | Number  | Part type | Designation         |
|---|---|-----------|---------------------|
|  SN-000003   |  SN-000003   | Assembly  | Housing assembly    |
|  SN-000004   |  SN-000004   | Assembly  | Gear wheel assembly |
|  SN-000005   |  SN-000005   | Assembly  | Pinion assembly     |
|  SN-000006  |  SN-000006  | Part      | Gearbox housing     |
|  SN-000007 |  SN-000007 | Part      | Gearbox cover       |
|  SN-000008 |  SN-000008 | Part      | Gear wheel          |

## Product structure

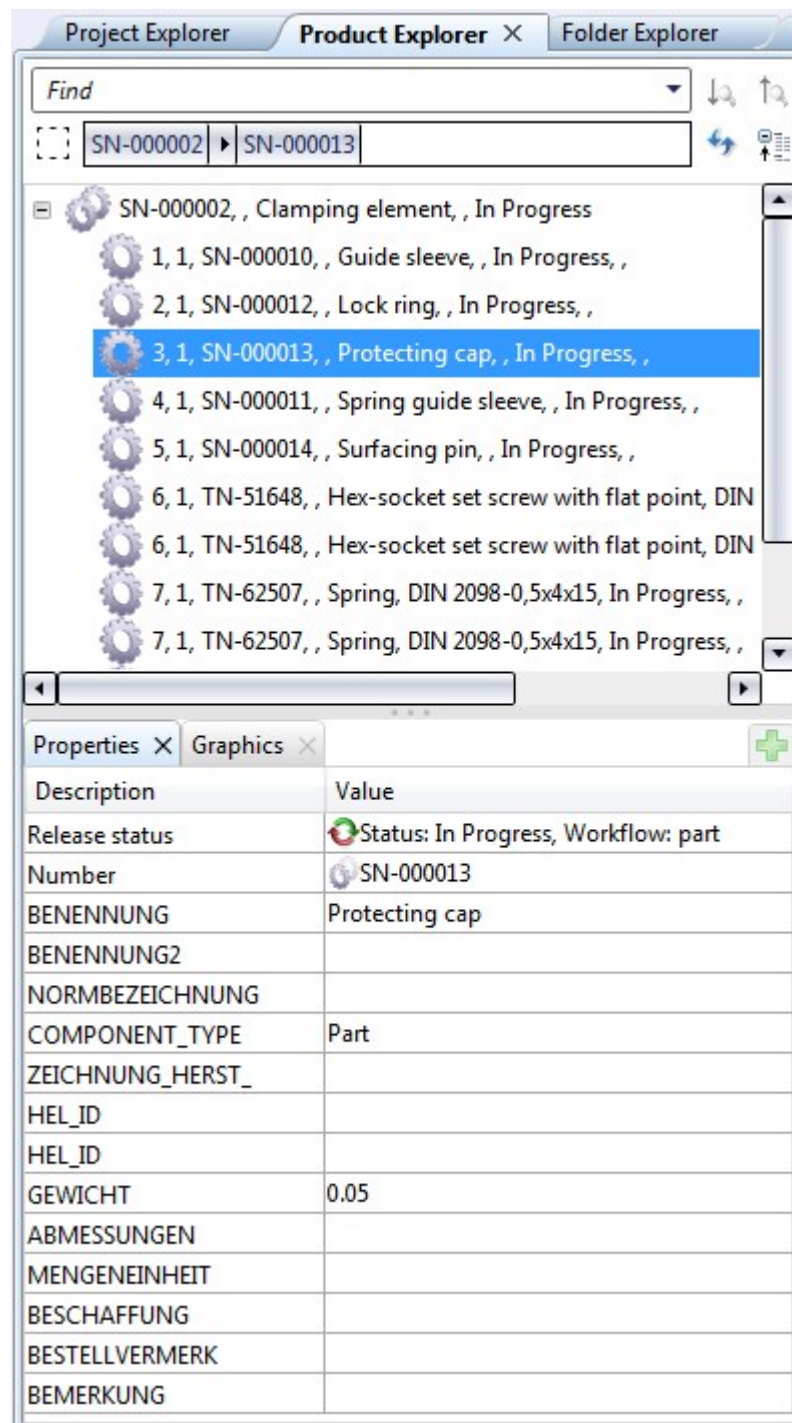
When selecting an index for the top article in the Product Explorer you can also make multiple selections. All selected top articles will be loaded into the Product Explorer one below the other. Old revision indices will be indicated as such.





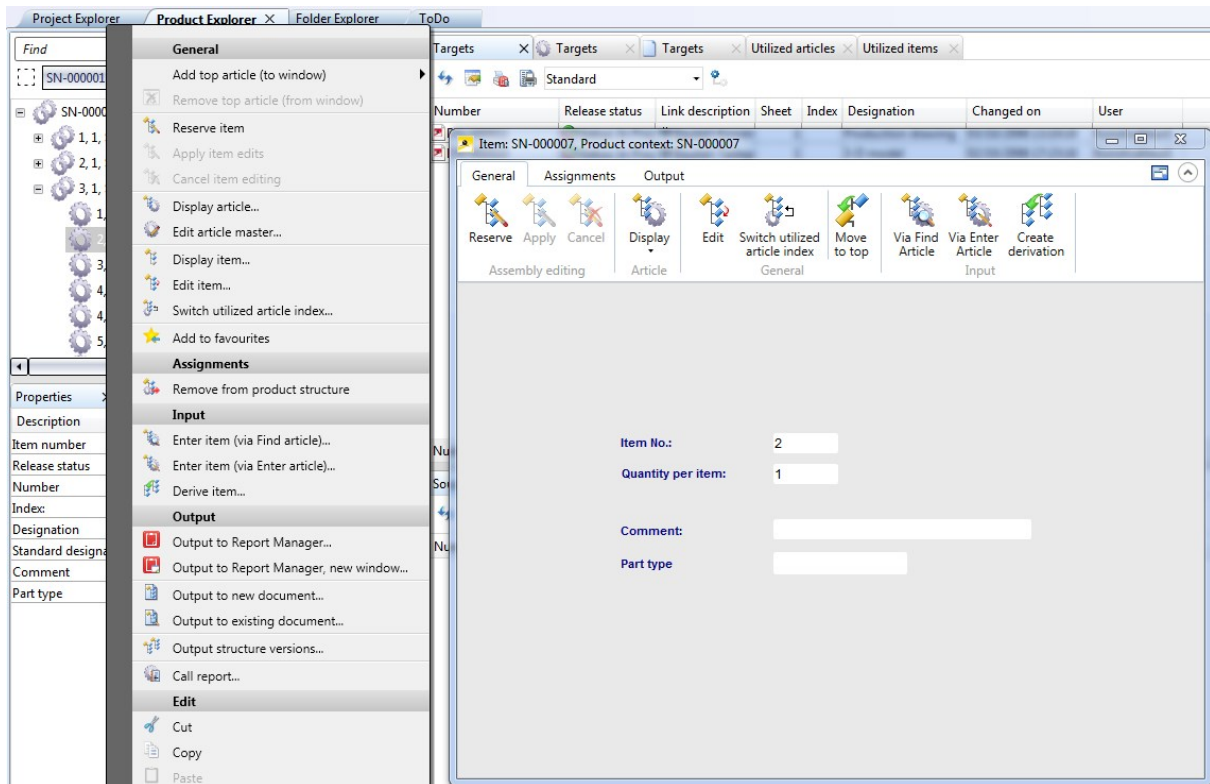
Also, the **Properties** window of the Product Explorer displays (usage) attributes not only for super-ordinate items, but also for the sub-items below them.

## Product Explorer and Properties window



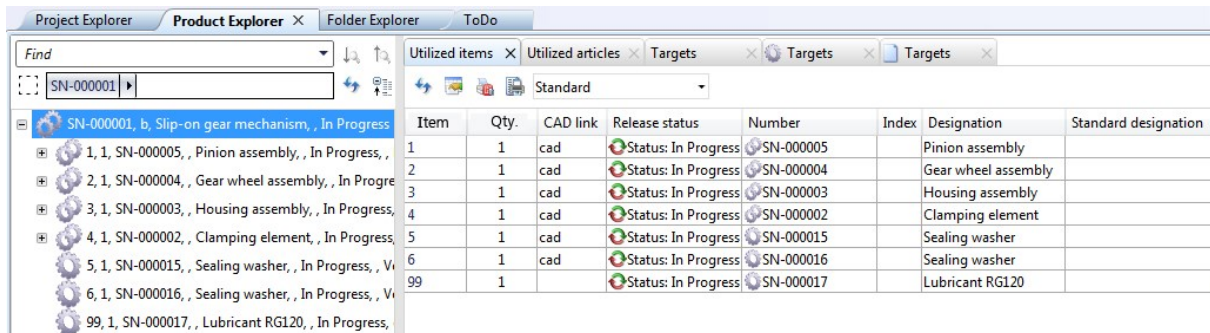
The new product detail window allows you to display and edit items of the product structure, similar to the detail windows of articles, documents etc.

### Display item: Product detail window



Besides the Utilized articles tab, you can also call the **Utilized items** tab in the result list area of the Product Explorer and in detail windows. The list in this tab replaces the Product Editor (context menu functions **Display in Editor** and **Process in Editor**) of older HELiOS versions.

### Utilized items



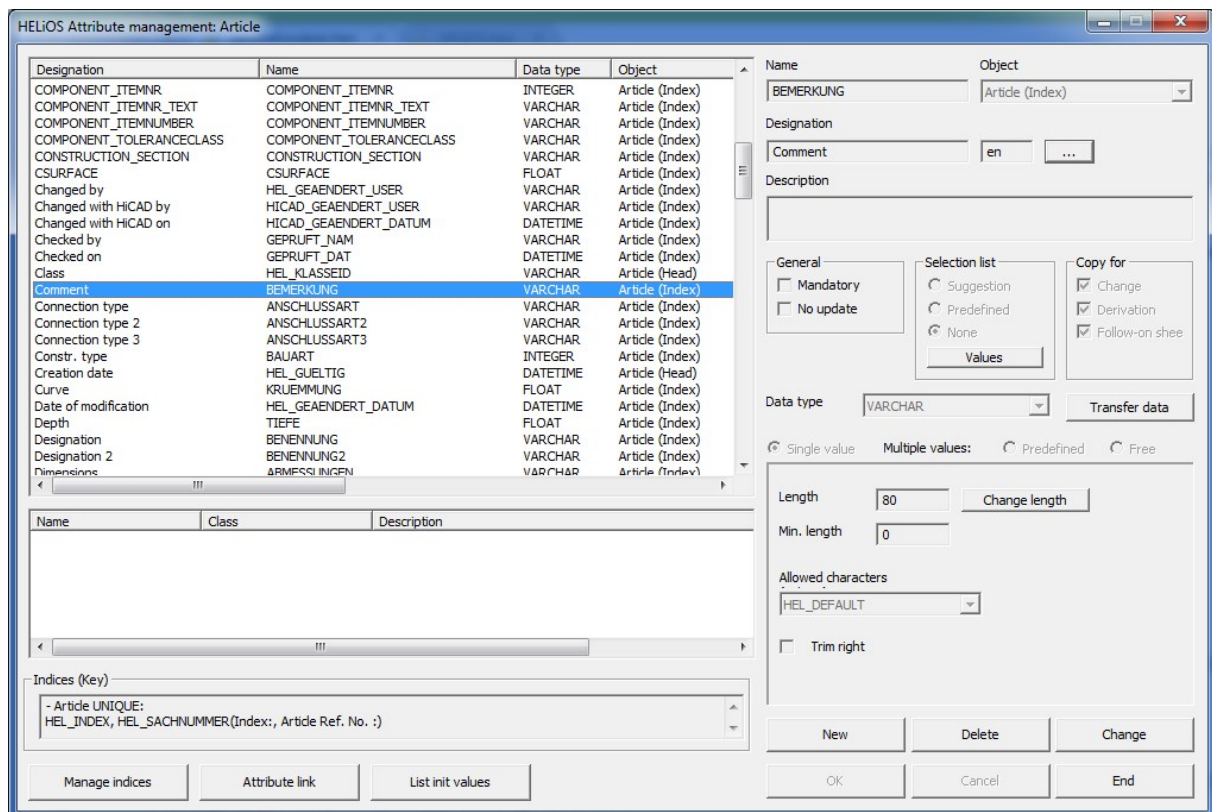
The screenshot shows the HELiOS Desktop interface with the 'Utilized items' tab selected in the Product Explorer. The table displays a list of items with their quantities, CAD links, release statuses, numbers, indices, designations, and standard designations.

| Item | Qty. | CAD link | Release status      | Number    | Index | Designation         | Standard designation |
|------|------|----------|---------------------|-----------|-------|---------------------|----------------------|
| 1    | 1    | cad      | Status: In Progress | SN-000005 |       | Pinion assembly     |                      |
| 2    | 1    | cad      | Status: In Progress | SN-000004 |       | Gear wheel assembly |                      |
| 3    | 1    | cad      | Status: In Progress | SN-000003 |       | Housing assembly    |                      |
| 4    | 1    | cad      | Status: In Progress | SN-000002 |       | Clamping element    |                      |
| 5    | 1    | cad      | Status: In Progress | SN-000015 |       | Sealing washer      |                      |
| 6    | 1    | cad      | Status: In Progress | SN-000016 |       | Sealing washer      |                      |
| 99   | 1    |          | Status: In Progress | SN-000017 |       | Lubricant RG120     |                      |



## Attribute Editor

In the Attribute Editor you can assign attribute designations in different languages to attributes. You can add as many languages as you like, and also change the designations of already existing attributes. The predefined designation will be preserved in the system, while the displayed designations can be repeatedly changed or deleted, for as many languages as you like.



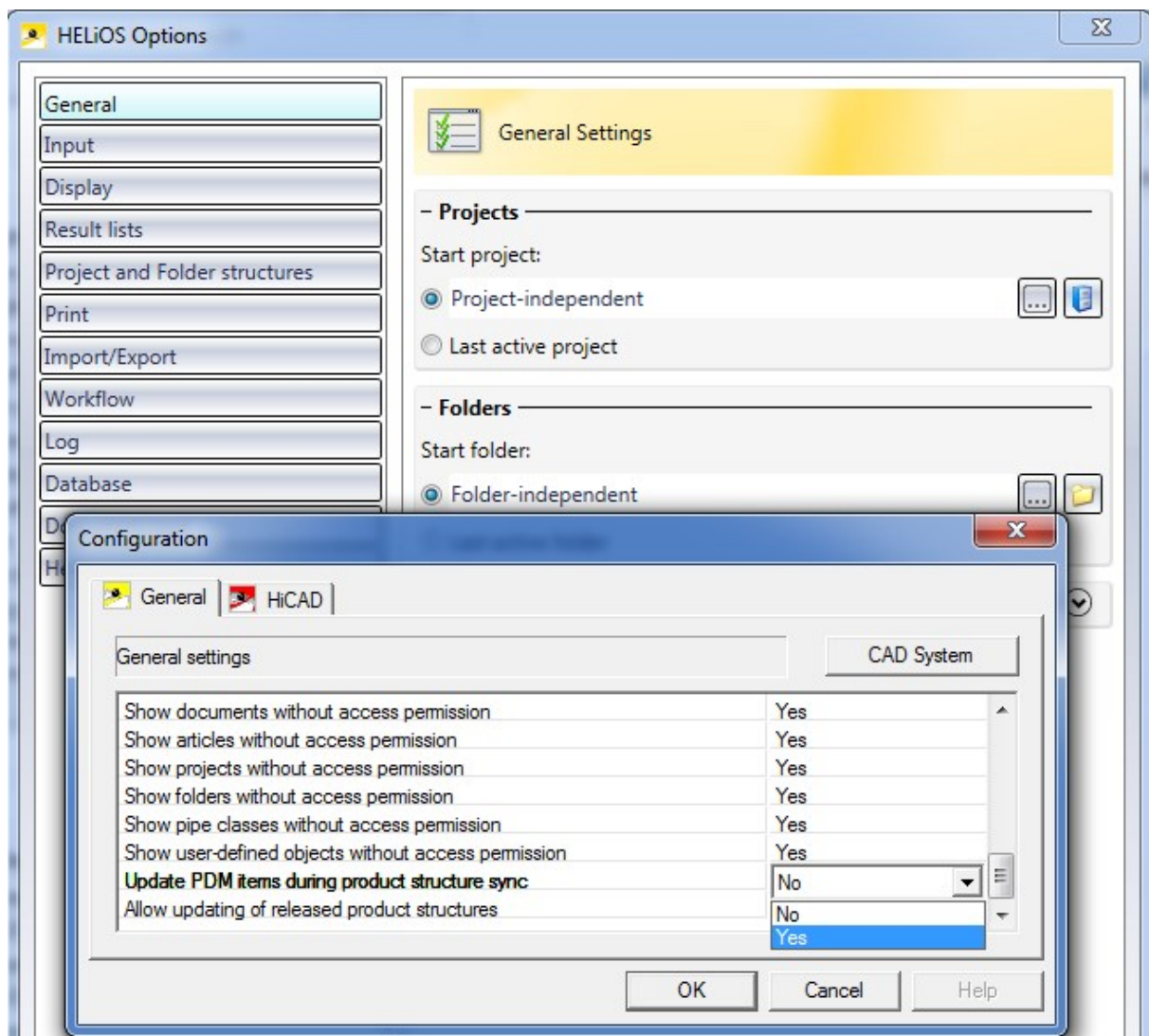
## Updating of PDM parts in the product structure

New settings in the Database tab of the HELiOS Options dialogue window of the HELiOS Desktop allow you to update

- individual PDM parts with a new index and
- released product structures.

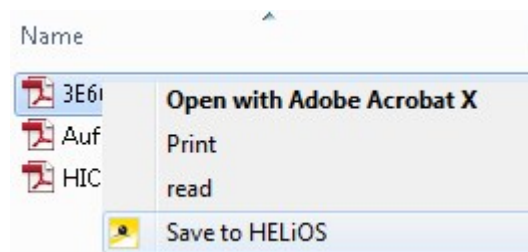
The option **Update PDM items during product structure sync** simplifies the product structure synchronisation between CAD and PDM insofar as it will be checked here whether PDM parts have a new index and if so, the product structure will be updated accordingly. This means that the version of the article does not need to be exchanged manually with the current version.

If you set the **Allow updating of released product structures** to "Yes" (default: "No"), modified PDM parts (with new index) can be updated in the product structure.



## Save to HELiOS from Windows Explorer

As of HELiOS 2016 you can save files of any type directly via the Windows Explorer to HELiOS by means of the new **Save to HELiOS** function.

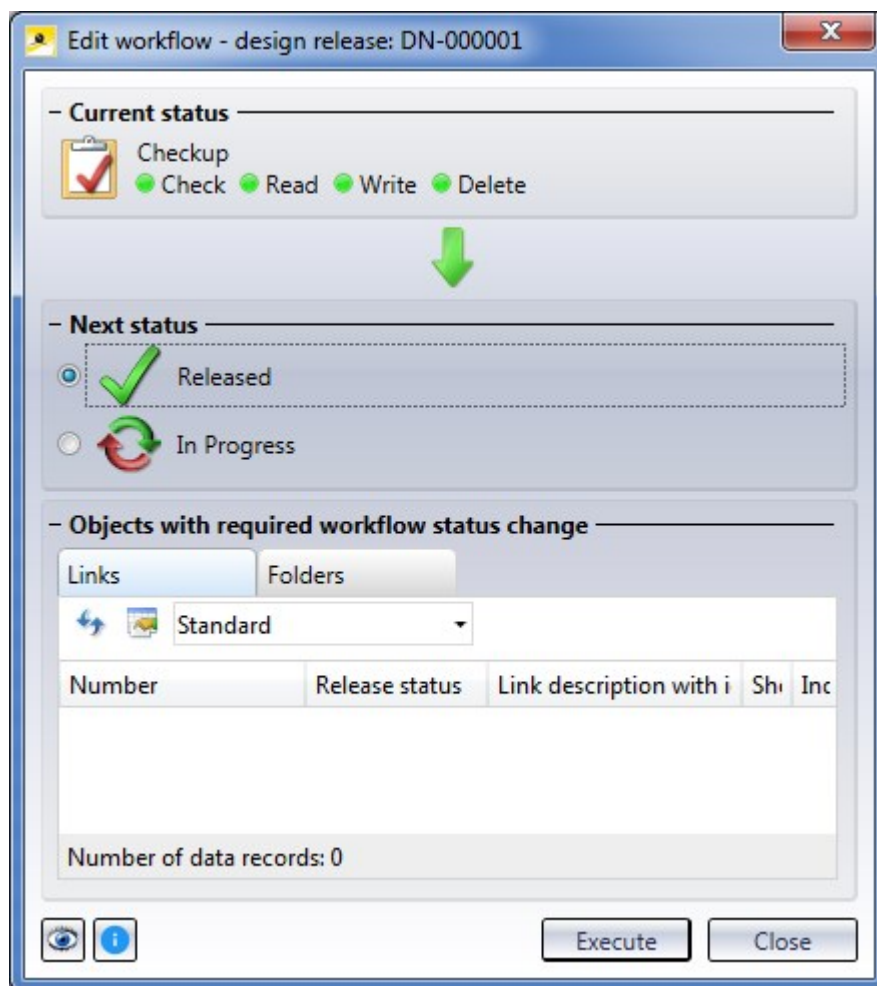




## New workflow dialogue

The new dialogue for the execution of workflow steps offers the option to assign the same workflow step to a selection of multiple documents or articles. A prerequisite for this is that all selected objects have the same object type, the same workflow and the same workflow status.

Also, several steps can be executed in immediate succession, i.e. the window will not be closed in the process.



## HELiOS in HiCAD

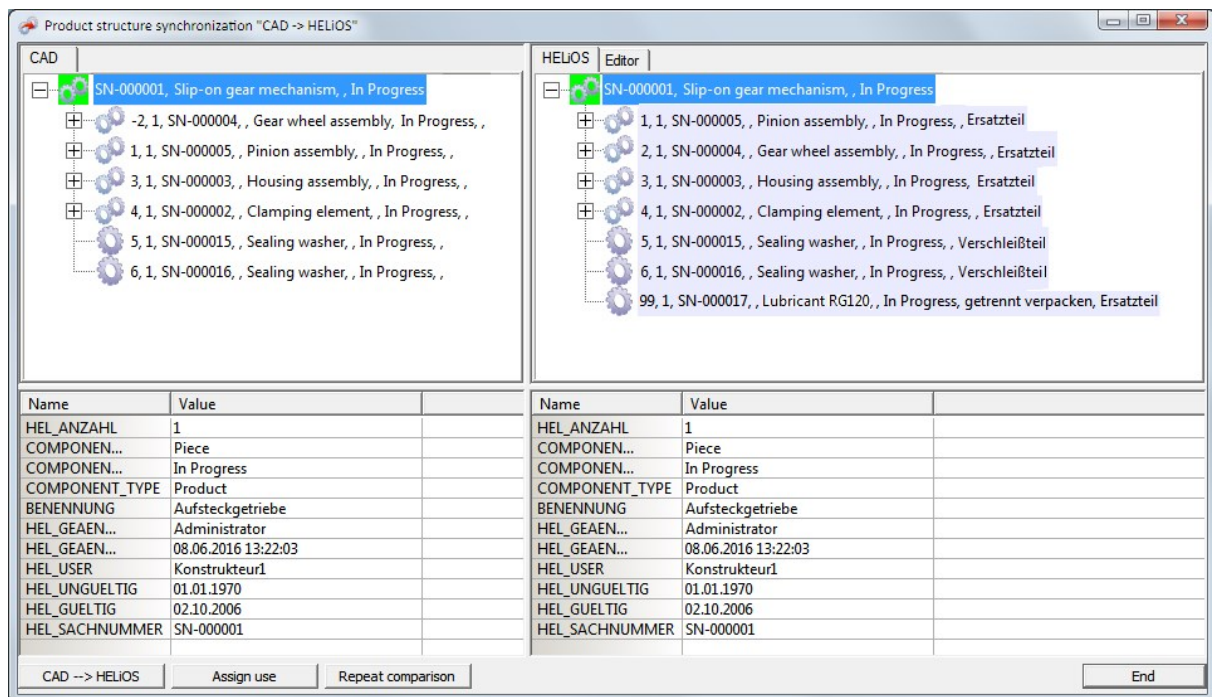
### Service Pack 2 2016 (V 2102)

#### Product structure synchronization

The Product structure synchronization functionality in HiCAD uses Look & Feel of the product structure in HELiOS 2016 SP2.

Changes in the product structure will be indicated by meaningful icons.

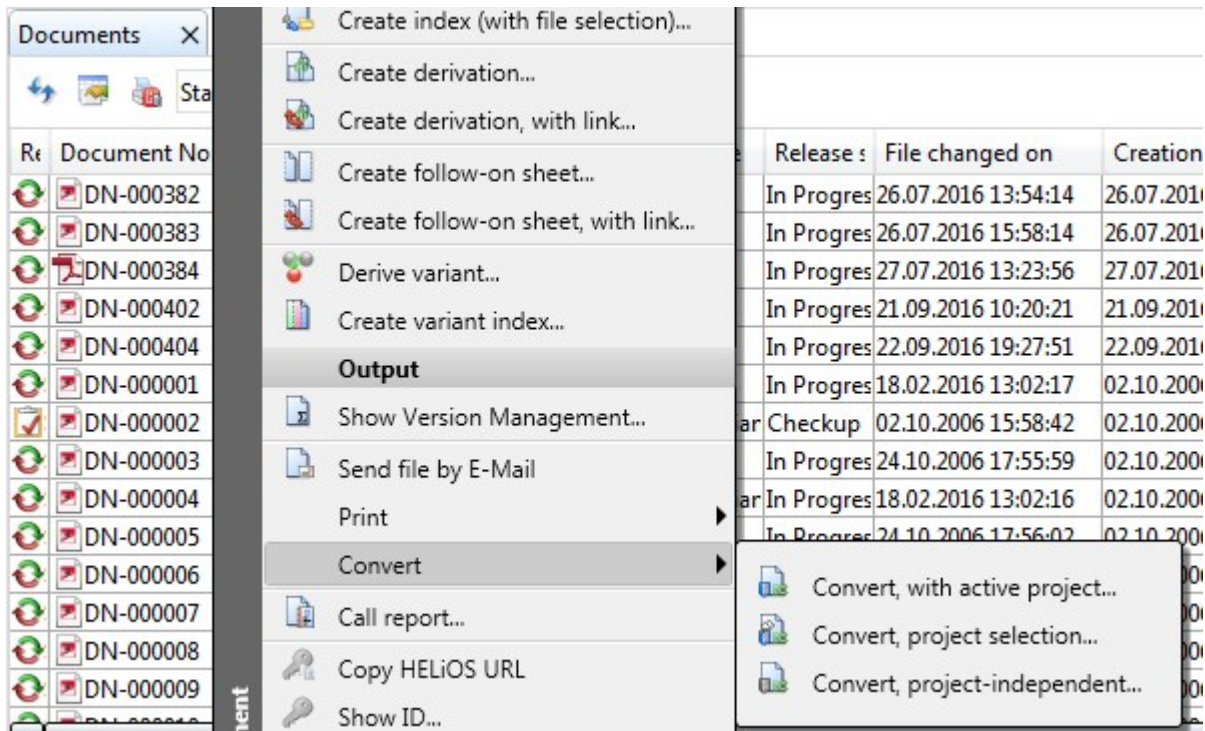
Thanks to the integrated result lists for utilized articles and items the product structure can also be displayed integrated as a BOM and offers many editing options.



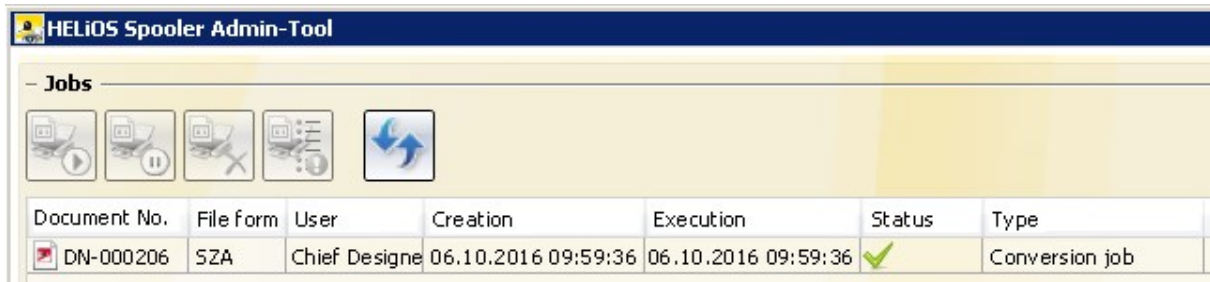
#### Other Export formats in the HELiOS Spooler

HiCAD drawing files (.SZA) files that have been saved as HELiOS documents to the database can be converted to other formats from HELiOS.

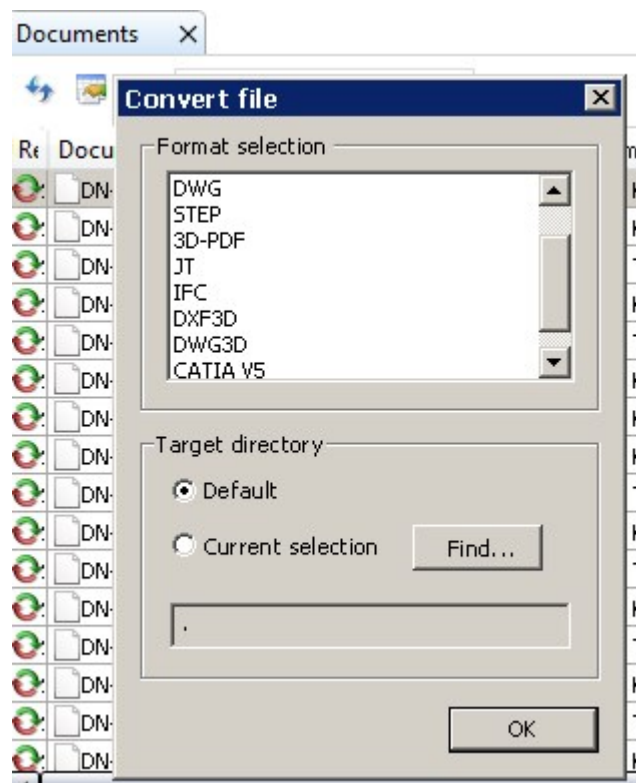
This can take place in the HELiOS Desktop, or via the context menu of HELiOS documents in HiCAD.



In the process, the data will be sent to the **HELiOS Spooler** which will perform the conversion and, if required, automatic post-processings and the management of the foreign format files.



The selection list of the suggested formats can be individually adjusted and expanded. Besides the selection of standard formats (DXF, DWG, STEP, 3D-PDF and JT) you can also integrate conversions to IFC, DXF3D, DWG3D, ACIS, PARASOLID, CATIAV4, CATIAV5, PROE, VDAFS and IGES3D.



Further information about file conversion can be found in the **HELiOS** Online Help.

## Service Pack 1 2016 (V 2101)

### Load via HELiOS

The HELiOS **Combined search** mask for the loading of drawings now offers the option to load documents in read-only mode.

Multiple selections are also possible, i.e. you can load several selected documents as drawings into HiCAD.

*Combined search mask*

**Combined search**

Search conditions

Article Classification Document Classification

Document number: [text box] Sheet: [checkbox] [I.S.D. icon]

Project number: Project-independent [icon] Index: [checkbox]

Folder number: Folder-independent [icon]

Document

Designation: [dropdown] Release: [text box]

Date: [text box] Name: [text box] Document type: [dropdown]

Created: [text box] Scale: [text box]

Checked: [text box] Format: [text box]

Standar: [text box]

Index

Index creator: [text box] File name: [text box]

Index date: [text box] Origin: [text box]

Text: [text box] Based on: [text box]

Links

- Bauteil-Teilegeometrie
- Bauteil-Konstruktion
- Bauteil(e)-Konstruktion
- Documentation
- Part-Document
- Without drawing
- No drawing required
- Drawing up to date
- Drawing released
- Drawing checklist
- Drawing not up to date
- Deleted
- Drawing part
- Itemised
- PDF data up to date
- PDF data not up to date
- Without NC-data
- DSTV-NC data up to date
- DSTV-NC data not up to date
- Without DXF data
- DXF data up to date
- DXF data not up to date
- Without STEP data
- STEP data up to date
- STEP data not up to date
- Unprocessed part
- Construction unit
- DXF drawing up to date

Search result

Standard

| Article No. | Index | Designation            | Standard designatic | Release     | Link description with icon | Document No. | SI |
|-------------|-------|------------------------|---------------------|-------------|----------------------------|--------------|----|
| SN-000001   |       | Slip-on gear mechanism |                     | In Progress | ↔ Bauteil-Konstruktion     | DN-000027    | 1  |
| SN-000001   |       | Slip-on gear mechanism |                     | In Progress | ↔ Bauteil-Konstruktion     | DN-000028    | 1  |
| SN-000002   |       | Clamping element       |                     | In Progress | ↔ Bauteil-Konstruktion     | DN-000003    | 1  |
| SN-000003   |       | Housing assembly       |                     | In Progress | ↔ Bauteil-Konstruktion     | DN-000005    | 1  |
| SN-000004   |       | Gear wheel assembly    |                     | In Progress | ↔ Bauteil-Konstruktion     | DN-000006    | 1  |
| SN-000005   |       | Pinion assembly        |                     | In Progress | ↔ Bauteil-Konstruktion     | DN-000007    | 1  |
| SN-000006   |       | Gearbox housing        |                     | In Progress | ↔ Bauteil-Konstruktion     | DN-000009    | 1  |

Number of data records: 26

☒ Load, read-only

Find OK Cancel

### Title block update upon release

Upon release of a drawing from HiCAD, the CAD document will automatically be saved again with the updated title block (Workflow step). In this way it is ensured that the correct and most recent status will be shown everywhere

(i.e. also in the viewer).

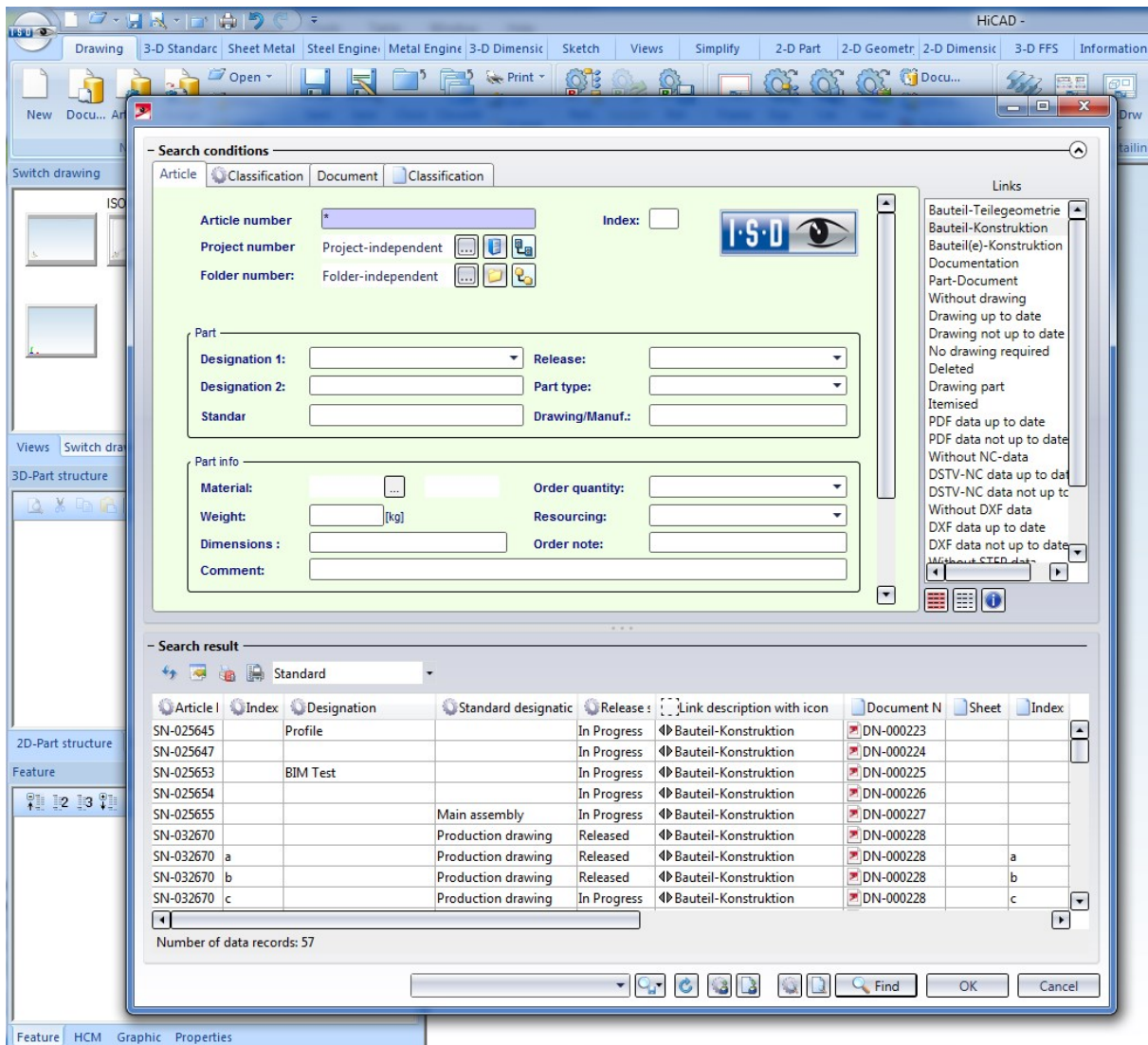
This behaviour had already been implemented for the management of Steel Engineering drawings and some customer-specific adjustments. It has become a standard with the release of HiCAD 2016 Service Pack 1.



## Major Release 2016 (V 2100)

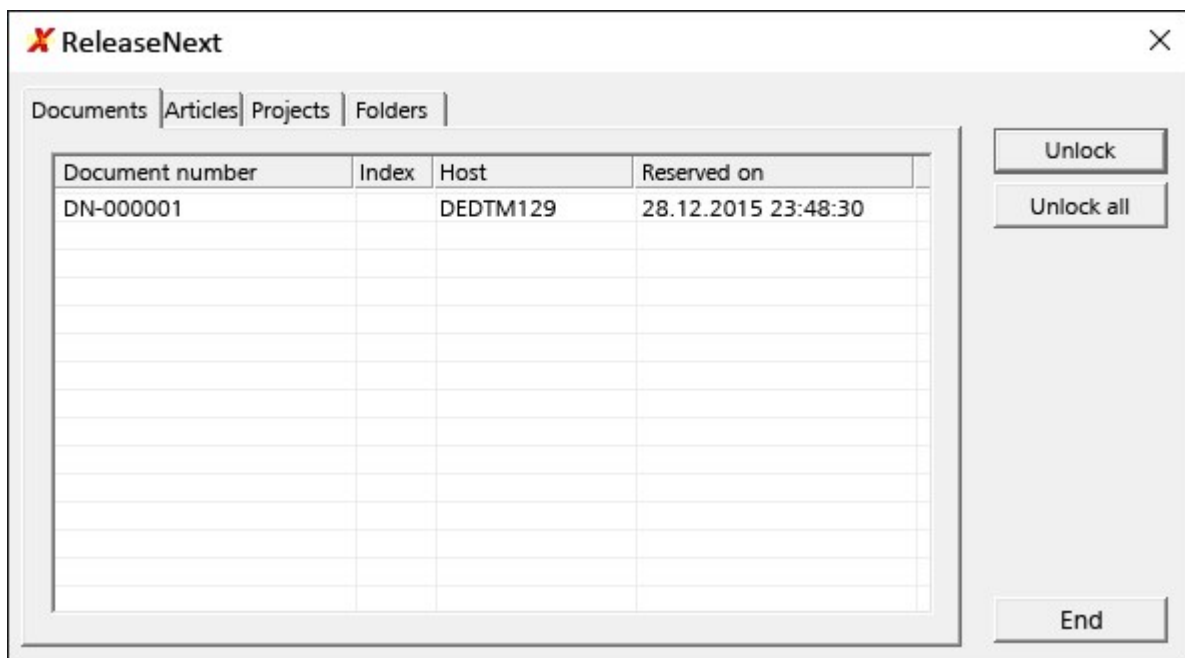
### New HELiOS masks

In HiCAD 2016 you will work with new HELiOS dialogue windows (e.g. search and input masks) offering a wider range of functions and enhanced user-friendliness.



### Date and time in dialogue for unlocking of HELiOS objects

The dialogue for the unlocking of reserved HELiOS objects (the "ReleaseNext" tool) that may be called upon start of HiCAD in conjunction with HELiOS, now has an additional column for the date and time of the reservation. This allows an administrator to see for how long an object has been reserved.



## Product structure synchronisation

- In the database options of the HELiOS Desktop, user with administrator rights can specify whether individual PDM parts in the product structure with a new index and released product structures can be automatically updated each. When the default setting has been chosen, this is not possible.
- If you set the **Database option Update PDM items during product structure sync** to **Yes**, the product structure will be updated accordingly when new indices of PDM articles exist. This means that no manual exchanging of the older version with the revised version will be required.
- If you set the **Allow updating of released product structures** to **Yes** (default: "No"), modified PDM parts (with new index) can be updated in the product structure.

## HELiOS Plot Manager

### Service Pack 2 2016 (V 2102)

#### Other export formats

The conversion of SZA files via the HELiOS Spooler is now also possible for the following target formats:

- IFC
- 3D-DXF
- 3D-DWG
- ACIS
- Parasolid
- Catia V4 und Catia V5
- ProE
- VDAFS
- Iges

The set up takes place on the Client in the system file hel\_fileconvert.ini.

#### Enhanced HELiOS Spooler Add-in for AutoCAD

The HELiOS Spooler Add-in for AutoCAD now offers the option to convert files.

For instance, you can automatically convert DWG files to DXF files via the HELiOS-Plot Management.

#### HELiOS Spooler - New PostProcessing variables

New variables for the postprocessing of print jobs are now available.

These allow special mappings, for instance, you can assign conversions by particular users to particular directories via the variable "HEL\_USERNAME".

## HELiOS MS Office Interface

### Service Pack 2 (V. 2102)

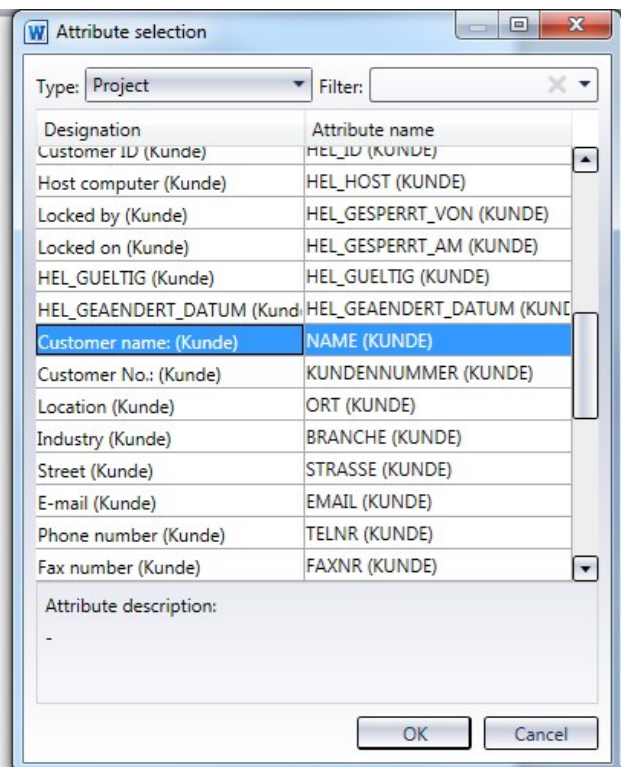
#### Expanded attribute selection

The attribute selection dialogue for the placing of HELiOS attributes into your Office document has been adjusted to the latest standards of the HELiOS Desktop.

This provides you with a wider range of attributes that can be inserted into title blocks. This also includes reference attributes such as the Material ID.

Furthermore, you can use the Filter function to search for specific attributes.

| HELiOS Attributes |               |
|-------------------|---------------|
| Designation       | Word document |
| Customer          |               |



### Major Release 2016 (V 2100)

#### Office 2016


The Office interface of HELiOS 2016 can be used with the latest Office Version 2016. Currently supported are Office 2010, 2013 and 2016 (32Bit and 64Bit).


#### ToDo List


The ToDo list of the HELiOS Desktop can now also be called via Office applications.

## MS Word with HELiOS ToDo list

HELiOS

 New

 Update

 ToDo list

Current document: DN-000391  
 Read-only: no
















Attribute fields

ToDo lists

Info

ToDo

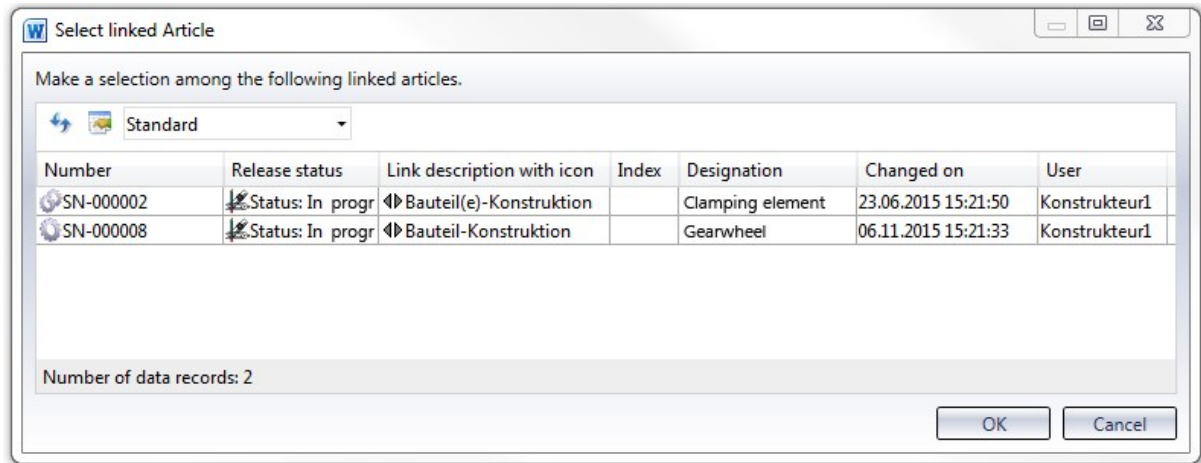
Standard

| Re  | Number    | VA_Object    | VA_Object   | Responsible   | Workflow name      |
|---|-----------|--------------|-------------|---------------|--------------------|
|    | DN-000226 | 2014-02-20_t | Administrat | Administrator | design release (R) |
|    | DN-000227 | 2014-03-20_t | Administrat | Administrator | design release (R) |
|    | DN-000229 | 2014-05-21_t | Administrat | Administrator | design release (R) |
|    | DN-000241 | 2014-06-18_j | Administrat | Administrator | design release (R) |
|    | DN-000248 | DN-000469    | Administrat | Administrator | design release (R) |
|    | DN-000251 | 2014-09-24_z | Administrat | Administrator | design release (R) |
|    | DN-000252 | 2014-09-25_j | Administrat | Administrator | design release (R) |
|    | DN-000253 | 2014-09-26_c | Administrat | Administrator | design release (R) |
|    | DN-000254 | 2014-10-01_t | Administrat | Administrator | design release (R) |
|    | DN-000256 |              | Administrat | Administrator | design release (R) |
|    | DN-000303 | 2015-02-23_j | Administrat | Administrator | design release (R) |
|    | DN-000304 | 2015-02-23_z | Administrat | Administrator | design release (R) |
|   | DN-000305 | 2015-02-24_j | Administrat | Administrator | general document ( |
|  | DN-000306 | 2015-02-24_c | Administrat | Administrator | general document ( |
|  | DN-000323 | 2015-03-30_z | Administrat | Administrator | general document ( |

Number of data records: 61

## Linked articles

When you open Office documents that are linked to several articles, an additional dialogue for the selection of the article context will be displayed.









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