HiCAD

UNLIMITED CAD PERFORMANCE **DEVELOPED BY ISD**

What's new?

Version 2020

News Overview

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THE WORLD OF CAD AND PDM SOLUTIONS

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Basics

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

Discontinuation of “old“ HiCAD itemisation
As of HiCAD 2019 the “old” itemisation, i.e. the itemisation that was used up to HiCAD 2017, will only be available for model drawings that were already itemized with these functions. From HiCAD 2021 onwards, only the “new” itemization will be supported.

Discontinuation of "old" OpenGL versions
From HiCAD 2021 on, only OpenGL version 4.3 is used in all HiCAD modules. Until now this was only the case with the module **HiCAD Point Cloud**. This means that HiCAD 2021 can only be run on computers whose graphics card supports OpenGL Version 4.3 or higher. Please also read the corresponding notes shown below.

Discontinuation of the old figure format (FIG)
Since HiCAD 2017 we have been supporting FGA as figure format (before FIG). Starting with HiCAD 2021 (Version 2600.0), we assume that the figures stored in HELiOS have been converted to the new format, or convert the still existing figures automatically as part of the database sub-version update to Version 2600.0.

Service Pack 2 2020 (V 2502)

The new Renderer - Update to OpenGL Version 4.3

HiCAD uses OpenGL for graphics output - so far in Version 2.0. As already announced, starting with HiCAD 2021 only OpenGL version 4.3 will be used in all modules. Up to now this was only valid for the **HiCAD Point cloud** module. This means that HiCAD 2021 will only run on computers whose graphics card supports OpenGL Version 4.3 or higher. The use of onboard cards is also still possible - provided they meet the corresponding requirements. However, we recommend not to use them from HiCAD 2021 on, since the performance of these chips is insufficient for large and complex model drawings.

Decisive factors in the development of the new Renderer were the extended range of functions for the graphics output through the use of newer OpenGL functions, and the possibility to realize noticeable performance improvements in the graphics output in the future. With HiCAD 2021, the graphics output will be generally accelerated, which is reflected in a higher frame rate (frames/second). Viewing functions such as zooming, rotating or panning are thus significantly faster and appear even more fluid.

In order to give you the opportunity to get to know the new Renderer in advance, it is already delivered with Service Pack 2 for HiCAD 2020 and can be activated on a computer-related basis if required. It already offers significant performance optimizations in particular situations.

Our request:

If you are updating to HiCAD 2020 SP2, you should also - at least on some workstations - activate the new Renderer (if the graphics card supports OpenGL 4.3).

With HiCAD 2020 SP2 you still have the possibility to switch between both Renderers. This way you can help to detect possible problems with the new Renderer at an early stage. Of course, this renderer has already undergone extensive testing as part of our quality assurance process, but its use in daily practice can provide us with further important information. This will ensure that future updates to HiCAD 2021 or newer versions can be carried out smoothly.

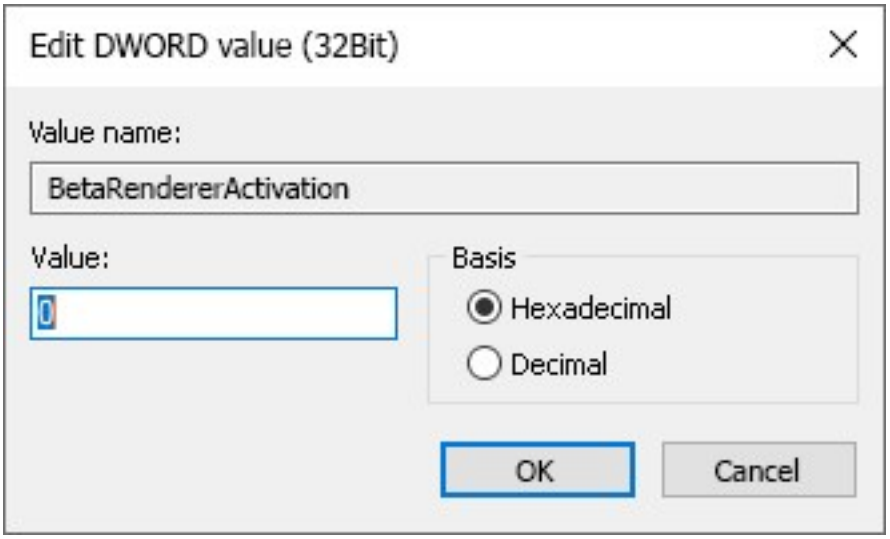
We thank you in advance for your kind support!

Proceed as follows to activate/deactivate the new Renderer:

To switch between the old and new Renderer you first have to adjust the Windows Registry as follows:

In the Registry, switch to **Computer\HKEY_CURRENT_USER\SOFTWARE\ISD Software und Systeme\HiCAD 2020**.

Right-click on the **Feedback** entry there and create a new entry of type DWORD value (32Bit) called **BetaRendererActivation**.



Supported are the following values:

- 0 The setting from the Configuration Editor will be used - **System settings > Graphic > Use OpenGL Beta Renderer**
- 1 The new Renderer will be activated - regardless of the setting in the Configuration Editor
- 2 The new Renderer will be activated - regardless of the setting in the Configuration Editor

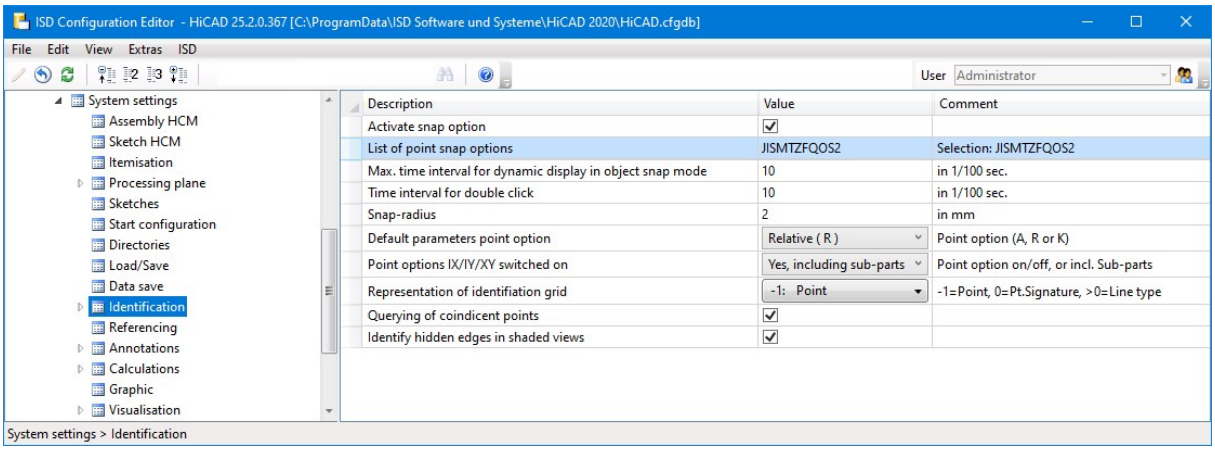
GUI and operation

New Autopilot Settings toolbar

The **Autopilot Settings** toolbar at the bottom of the HiCAD user interface can now be used to control which points the Autopilot offers as snap points. Switching is possible at any time - even during the execution of a function. In the toolbar the possible snap points are highlighted in colour.

Using this bar you can control which point options should be visible on the Autopilot by activating / deactivating the symbols. For example, if you click on , the snapping of isolated points is deactivated.

The currently selected Autopilot settings apply to the active HiCAD session. Which settings should be active when restarting HiCAD can be defined in Configuration Editor at **System settings > Identification** with the parameter **List of point snap options**.



The ISD default setting is JISMTZFQOS2, i.e. snap points are isolated points, end points, genuine intersection points, centre points, tangential points, centroids, perpendicular base points, quad points, on-line points and theoretical intersection points.

If you click on the symbol in the **Autopilot Settings** toolbar, the settings from the Configuration Editor are restored to defaults.

Please note:

- The Autopilot Settings apply to 2-D and 3-D point options.
- Point options that are deactivated from the active function cannot be activated in the Autopilot Settings either.
- The **Autopilot Settings** toolbar can be activated/deactivated via **Settings > Toolbars**.

New Visualisation toolbar

The previous toolbar "CS display" has been extended with additional functions for fast fading in and out of objects and renamed **Visualisation**. With the functions of this toolbar you can switch on or off the visibility of coordinate systems, isolated points, dimensions, HCM constraints etc. with one click. In certain situations - especially with large model drawings - this can make editing easier. For example, you can hide all dimensions of a model drawing without having to call a function in the context menu or ribbon first.

The following functions are new here:

Toggle visibility of isolated points (Drawing)

All isolated points of the model drawing can be shown or hidden with one mouse click.

Toggle visibility of dimensions (Drawing)

All 2-D and 3-D dimensions of the model drawing can be shown or hidden with one mouse click. This applies both to drawing dimensions and parametric dimensions.

Toggle visibility of dimensions (3-D Part HCM)

Toggle visibility of constraints (3-D Part HCM)

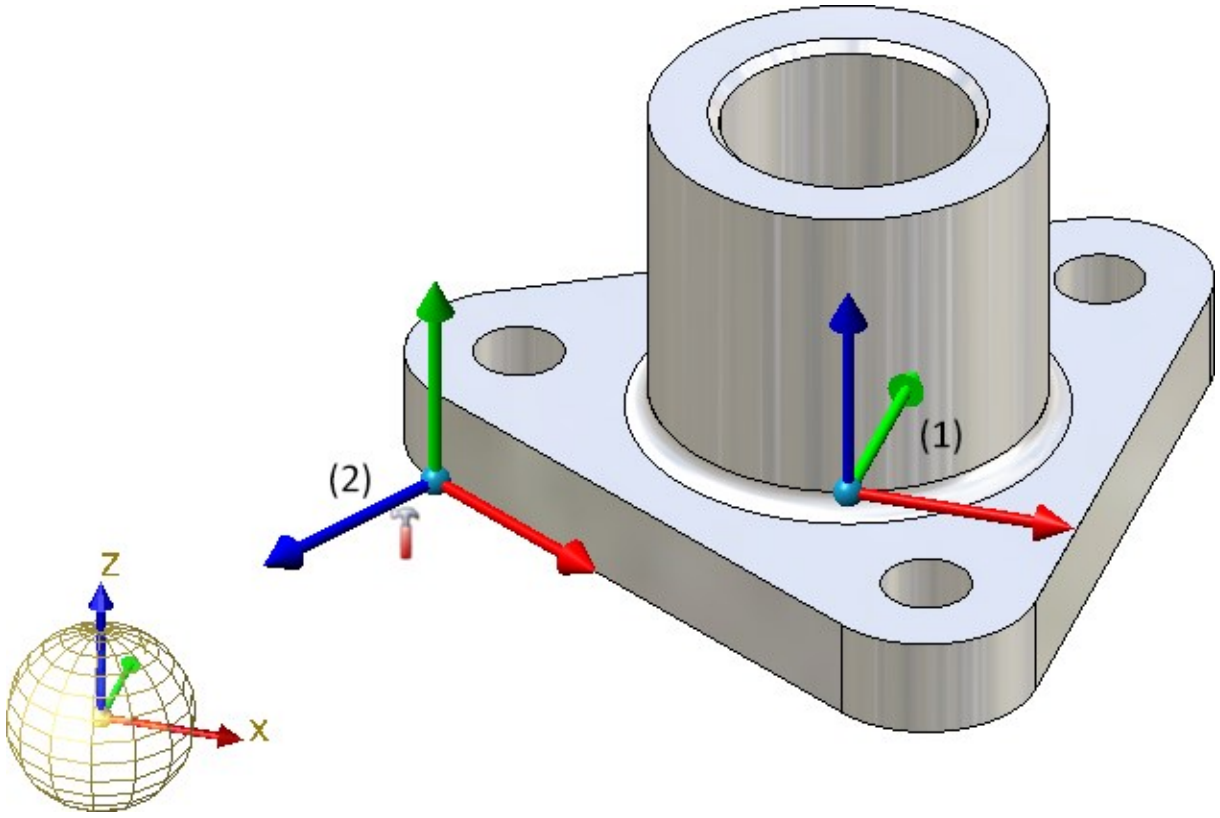
Toggle visibility of degrees of freedom (3-D C-Edge HCM)

Toggle visibility of dimensions (3-D C-Edge HCM)

Toggle visibility of constraints (3-D C-Edge HCM)

CS display - Fitting CS

The spatial representation of the Fitting CS (2) has been revised to make it easier to distinguish it from the Part CS (1).



New symbol for part with free edges

In the part structure display of the ICN the symbol for Part with free edges has been changed. The new symbol looks like this:

New pocket calculator

The HiCAD Pocket calculator has had a redesign.

In many cases, HiCAD suggests a default value, which you can apply directly or correct.

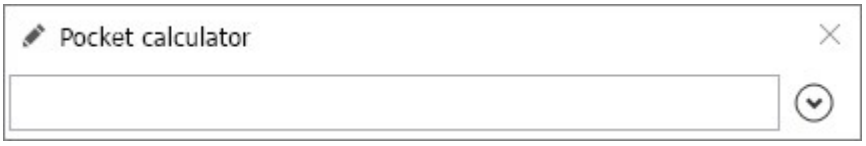
You complete your input by

- pressing the left mouse button or
- pressing the ENTER key of the keyboard or
- clicking on the OK button.

Special keys:

Is used to take distances and angles from existing construction objects. Alternatively, you can also call these functions in the context menu, which you activate by pressing the right mouse button in the input field of the calculator. The functions of the distance and angle menu displayed here are the same as the identically named functions in the **Information** menu. The **3-D Distances** menu also provides the **Tracing lines** function, which enables you to transfer the spacing of the tracing lines of a steel engineering beam.

Hides the keyboard area of the pocket calculator.



Docking windows - Zooming of contents

The contents of various docking windows can be enlarged or downsized dynamically by zooming. This is possible for the following docking windows:

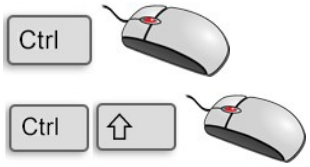
- Views,
- 2-D/3-D Part structure,
- Feature,
- HCM (2-D) und HCM (3-D),
- Part variables,
- Exploded view.

Zoom contents of one docking window

To enlarge or downsize the contents of a single docking window, point the mouse cursor inside the window and turn the mouse wheel while holding down the CTRL key.

Zoom contents of all docking windows

To enlarge or downsize the contents of all docking windows, turn the mouse wheel while holding down the CTRL and the SHIFT key.



Reset zoom

The zoom of the active window can be reset by pressing the CTRL and middle mouse button simultaneously. To reset the zoom of all windows, press CTRL, SHIFT and the middle mouse button accordingly.

Easier element snap

As long as no function is active, pressing and holding the SHIFT key from SP2 on automatically activates the element snap function - independent of the selected identification mode. This provides faster access to the context menus of individual geometry elements and is also a practical way to find a Feature in the ICN from within the geometry - especially in large and complex model drawings.

North American scale

Scales are used in HiCAD as

- Main scale of a model drawing,
- Scale of a view,
- Scale of a 2-D part and
- Scale of steel construction part drawings.

The scale can be selected from a selection box in the corresponding HiCAD function dialogues. In addition, in many cases it is also possible to enter a scale directly. The scales available in the selection boxes of the HiCAD functions are defined in the file **SZENE-MASSSTAB.TXT** in the HiCAD subdirectory **MAKRO2D** or for steel construction component drawings in the file **StbEtZng_MASSSTAB.DAT** in the HiCAD subdirectory **sys**. Please note that HiCAD supports the European scale logic by default, i.e. scale specifications in the form n:m, e.g. 1:1, 1:10, 5:1 etc.

SZENE_MASSTAB.TXT

```
1:1
1:2
1:2.5
1:5
1:10
1:20
1:50
1:100
1:200
2:1
5:1
10:1
20:1
50:1
100:1
```

StbEtZng_MASSSTAB.DAT

```
#
# Einzelteilzeichnung: Massstab
# -----------------------------
# 1. Zeile: Bezeichnung
# 2. Zeile: Massstabsfaktor
#
1:100
0.01
1:75
0.0133
1:50
0.02
1:25
0.04
1:20
0.05
1:10
0.1
1:5
0.2
1:2,5
0.4
1:2
0.5
1:1
1.0
2:1
2.0
5:1
5.0
10:1
10.0
```

Default scales

However, the US scale logic differs from that in Europe. Therefore HiCAD offers the possibility to extend the above mentioned files by defining additional scale specifications. For examples of typical scale specifications in North America, refer to the following table:

1/128"=1'-0"	1/8"=1'-0"	1"=1'-0"
1/64"=1'-0"	1/4"=1'-0"	-1/2"=1'-0"
1/32"=1'-0"	3/8"=1'-0"	3"=1'-0"
1/16"=1'-0"	1/2"=1'-0"	6"=1'-0"
3/32"=1'-0"	3/4"=1'-0"	1'-0"=1'-0"

' foot, " Inch, 1 foot = 12 inches, 1 inch = 2,54 cm

Conversion table		
Scale	Factor	**Decimal**
1'=1'-0"	1:1	1.0
6"=1'-0"	1:6	0.5
1-1/2"=1'-0"	1:8	0.125
1"=1'-0"	1:12	0.08333
3/4"=1'-0"	1:16	0.06250
1/2"=1'-0"	1:24	0.41667
3/8"=1'-0"	1:32	0.031250
1/4"=1'-0"	1:48	0.020833
3/16"=1'-0"	1:64	0.015625
1/8"=1'-0"	1:96	0.010417
3/32"=1'-0"	1:128	0.007813
1/16"=1'-0"	1:192	0.005208
1/32"=1'-0"	1:384	0.002604
1/64"=1'-0"	1:768	0.001302
1/128"=1'-0"	1:1536	0.000651

Changing the file SZENE-MASSSTAB.TXT

To structure the file, any number of empty lines and comments can be inserted, whereby comment lines must begin with the character #. Each scale specification must be in a separate line. The following notations are supported for the scale specification:

Notations		Examples
1.	**Specification in scale format n:m**	1:10 1:1 2:1 5:1
2.	**Specification as factor**	0.1 1 2 5
3.	**As factor with display text** The factor is at the beginning of the line, the display text in brackets behind it. The display text is used by HiCAD in dialogues and information outputs.	2.5 (2‘-6“ = 1‘-0“) 1 (1`-0“ = 1‘-0“)
4.	**In scale format with display text** The scale is at the beginning of the line, the display text in brackets behind it. The display text is used by HiCAD in dialogues and information outputs.	1:12 (1“=1‘-0“) 1:48 (1/4“ = 1‘-0“)

The notations 3 and 4 can be used to define US scale specifications, for example.

Example of an expanded SZENE-MASSSTAB.TXT file

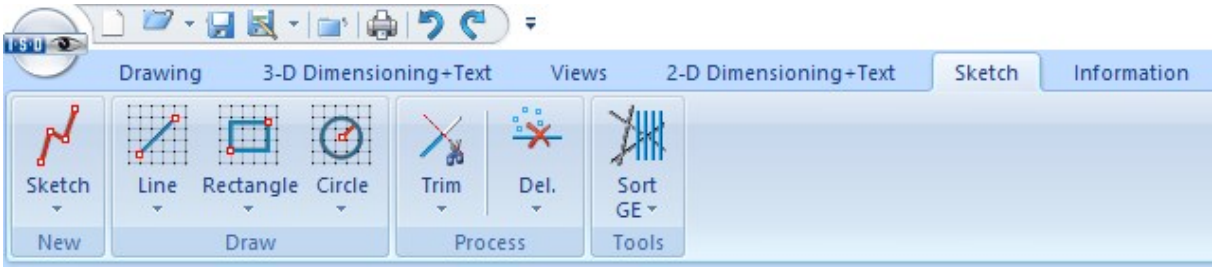
SZENE_MASSTAB.TXT | StbEtZng_MASSSTAB.DAT

```
# Maßstäbe / scale definitions
# Europäisch / European
1:1
1:2
1:2.5
1:5
1:10
1:20
1:50
1:100
1:200
2:1
5:1
10:1
20:1
50:1
100:1

# US
1:1536 (1/128"=1'-0")
1:768 (1/64"=1'-0")
1:384 (1/32"=1'-0")
1:192 (1/16"=1'-0")
3:384(3/32"=1'-0")
1:96 (1/8"=1'-0")
1:48 (1/4"=1'-0")
3:96 (3/8"=1'-0")
1:24 (1/2"=1'-0")
3:48 (3/4"=1'-0")
1:12 (1"=1'-0")
3:24 (1-1/2"=1'-0")
0.25(3"=1'-0")
0.5 (6"=1'-0")
1 (1'-0"=1'-0")
```

Further information can be found in the Definition of Scales topic.

Enhanced HiCAD VI / AV module

With the HiCAD VI / AV Viewer solution, HiCAD drawings and also and drawings imported via interfaces can be visualised. Furthermore, the module supports the plot generation/output and data export. In order to support the creation and editing of sketches for sectional and detailed views, the corresponding functions are now available on the **Sketch** Ribbon tab.

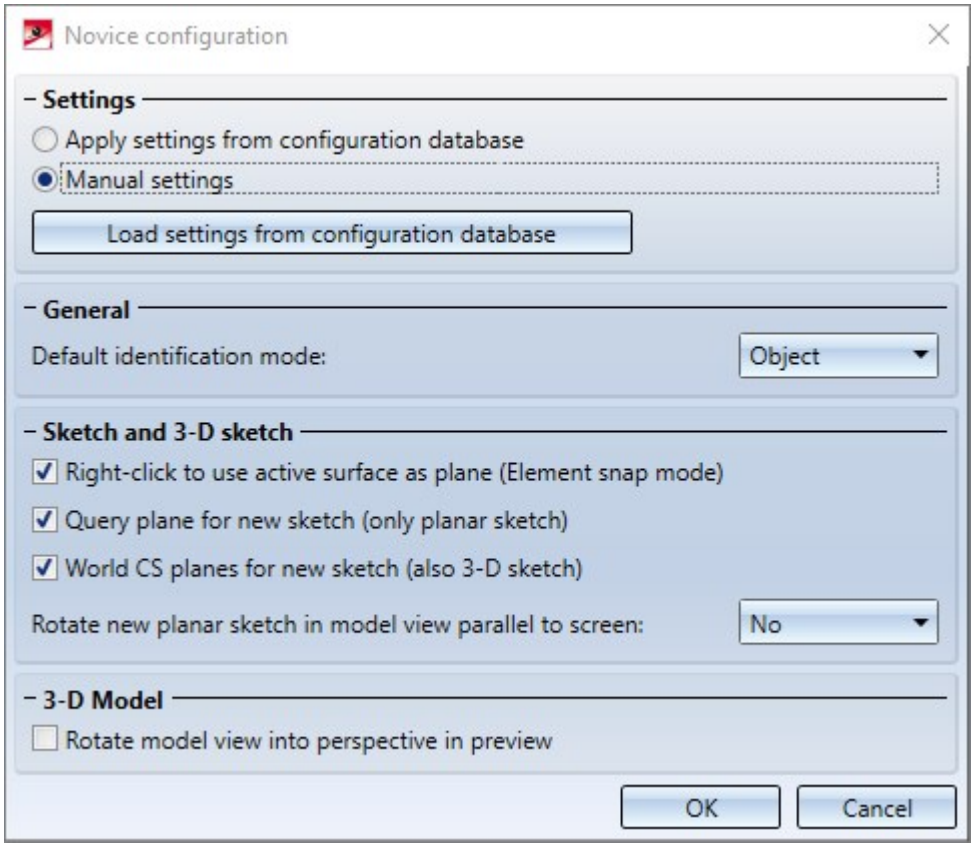


Functions		Effect
	Create new sketch and sub-menu functions	Allows you to draw a new sketch
	Sketch polyline and sub-menu functions	Inserts polylines, tangents and straight lines
	Sketch rectangle and sub-menu functions	Inserts rectangles and polygons
	Sketch circle and sub-menu functions	Inserts full circles
	Trim to point and sub-menu functions	Trims/edits sketch elements
	Delete lines and isolated points and sub-menu functions	Deletes line elements and points
	Sort GE and sub-menu functions	Sorts graphical elements of the active sketch and provides further tools

Novice configuration - - Settings for sketches

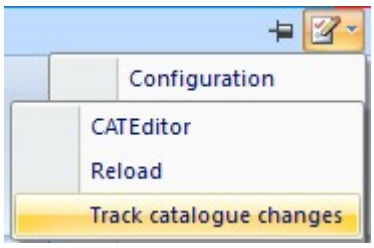
The settings for sketches in the **Novice configuration** previously only affected the creation of new sketches with the **New sketch in plane** and **New 3-D sketch** functions, and the functions at 3-D Standard > Process with sketch. However, they did not apply to other functions that use sketches to create or process parts, and also allow you to create new sketches. Here, the sketch plane was always the XY-plane of the world coordinate system.

As of SP2, the behavior when creating a new sketch has been standardized, that is, the settings of the Novice configuration dialogue window affect not only the creation of new sketches with the New sketch in plane and New 3-D sketch functions, but also all functions that use sketches to create or process parts, as well as all functions that allow you to create new sketches. This applies to such functions as Add and Subtract, deriving sheet metal parts from sketches, element installation, and much more. This also allows these functions to query the determination of the sketch plane before creating a new sketch



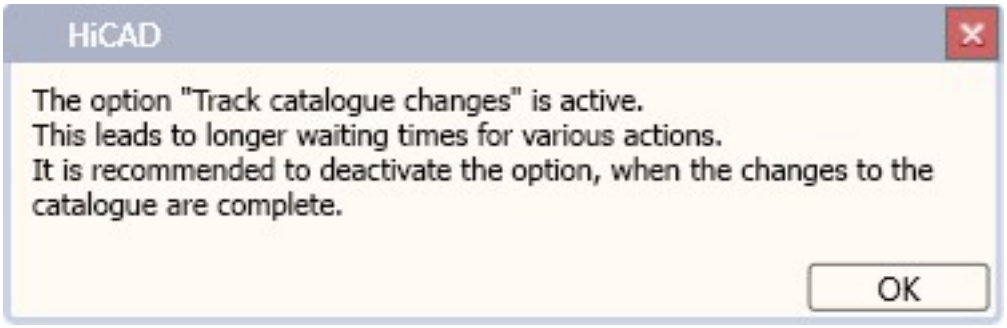
Track catalogue changes

Up to now, the Configuration Editor at **System settings > Catalogues** could be used to define how catalogue changes should be handled in HiCAD. From HiCAD 2020 SP2 onwards these settings are no longer available in the Configuration Editor . Instead, the switch **Track catalogue changes** is now available at **Settings** in the **Catalogue** menu.



If the switch is active, the catalogues are regularly checked during a HiCAD session to ensure that they are up-to-date. This actuality check can, however, significantly impair performance in some situations. If the switch is active, then this is indicated when HiCAD is started. By default, the switch is inactive by default, that is, the catalogues are loaded in HiCAD only once, when starting HiCAD. After that, the system does not check whether more recent data is available. To update the catalogues after changes, the function **Reload** is available. This offers the possibility to update catalogues after changes without having to leave HiCAD and without having to accept performance losses due to the constant check for up-to-dateness.

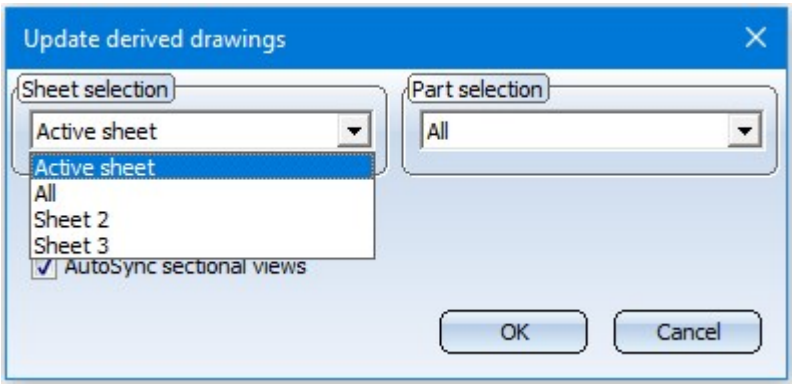
The current status of the switch (active/inactive) is entered into the Windows Registry. If the switch is active, the following message is displayed when starting HiCAD.



Drawing derivation

Update derived drawings - active sheet

When updating derived drawings, it is now easier to update the active sheet. The selection box has been extended accordingly.



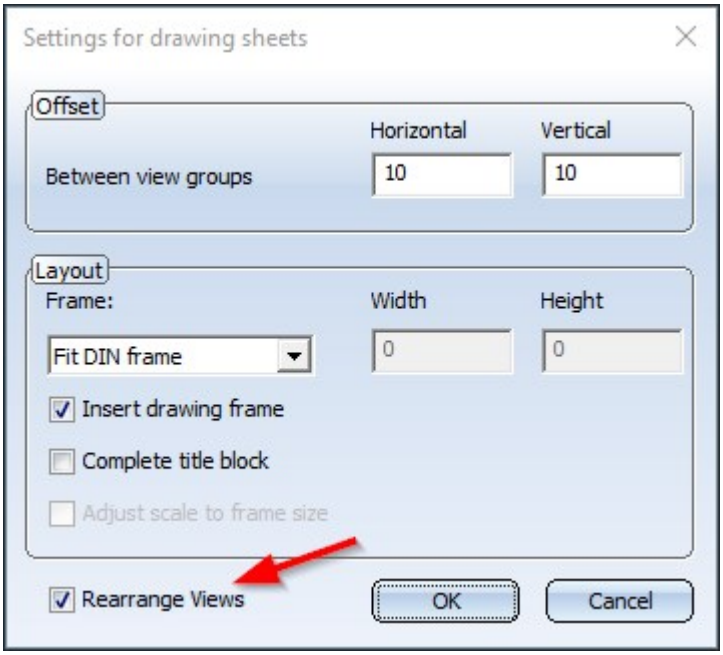
Change settings - Rearrange views

If the settings were changed in an existing workshop drawing and then the drawing was updated, HiCAD always rearranged all views. From HiCAD 2020 SP2 onwards this can be set individually. For this purpose the **Rearrange views** checkbox has been added to the dialogue windows of the following functions:

Change settings, Active drawing sheet

Change settings, active view group

Change settings, active view

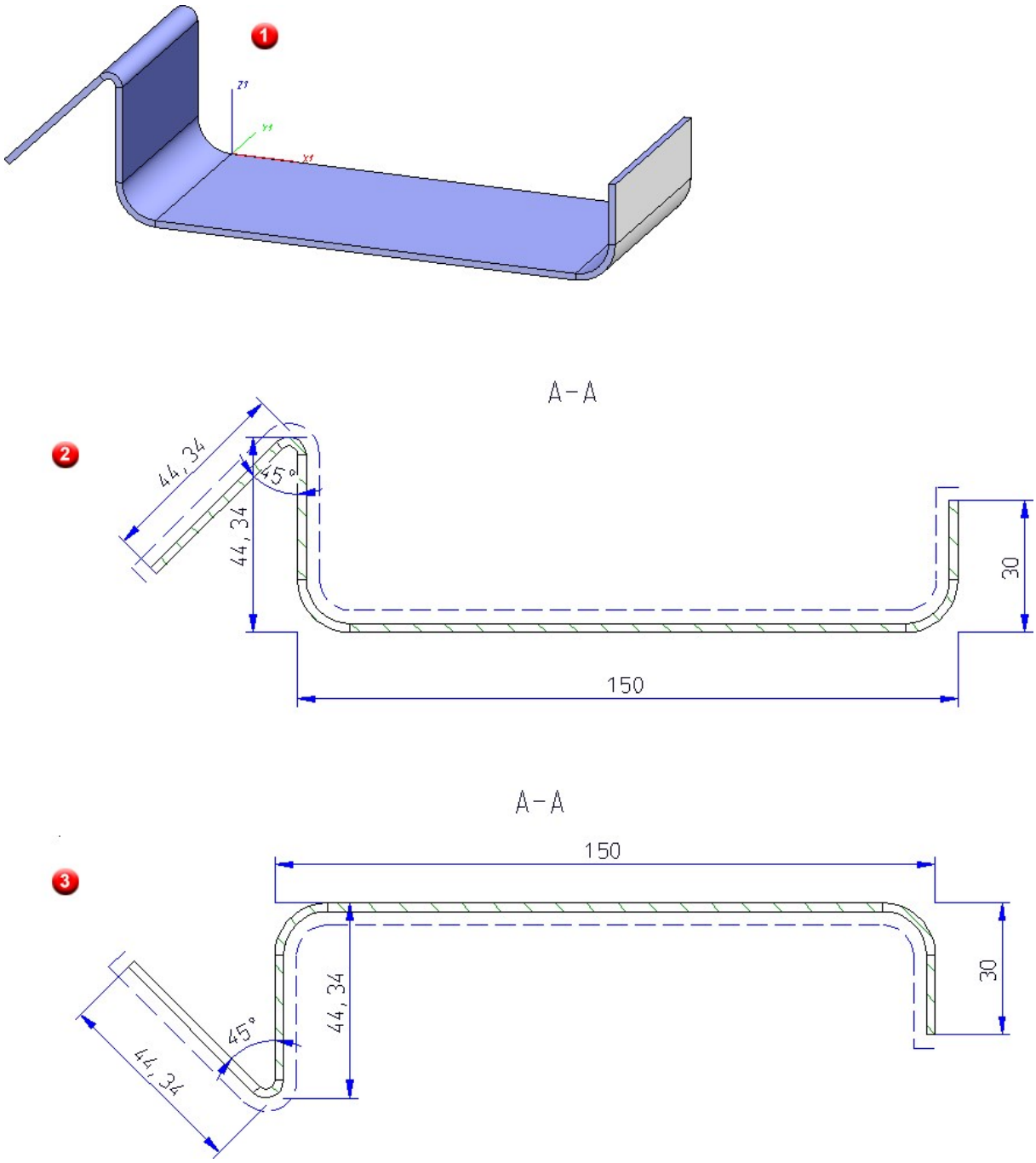


Please note:

If the changed settings add further views, these may have to be arranged manually.

Sectional views of sheet metal parts

From HiCAD 2020 SP2 onwards, the viewing direction of the sectional views of sheet metal parts in the workshop drawing is to the right or downwards. Aligned cuts are accordingly to the right or below.

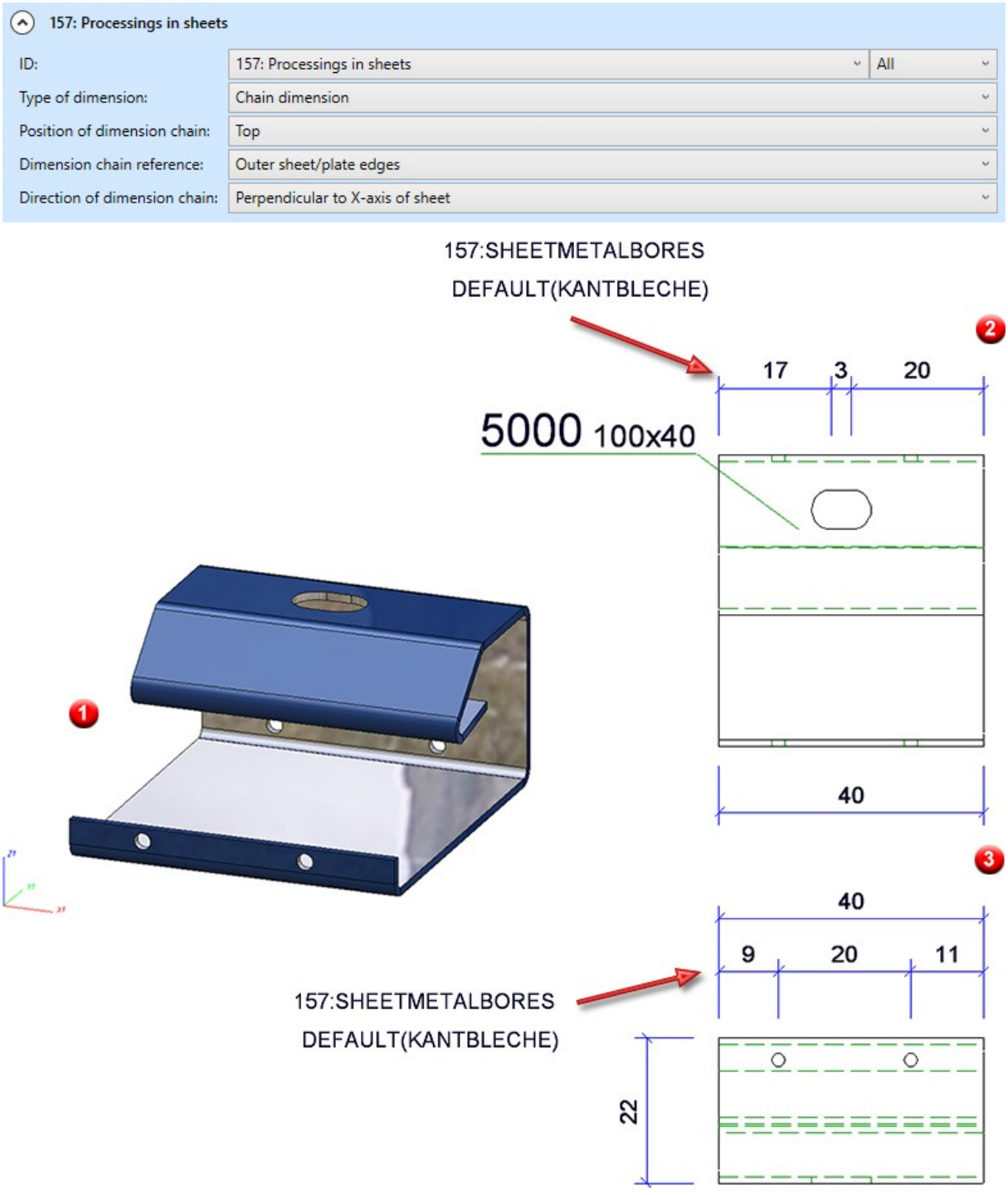


(1) 3-D model in axonometry, (2) sectional view in workshop drawing before HiCAD 2020 SP1, (3) sectional view in workshop drawing from HiCAD 2020 SP1 onwards

Please note that updating your existing drawings can change the position of the sections.

Dimensioning of sheet metal parts

In the configuration for the usage "Sheets" **DEFAULT(KANTBLECHE)**, Dimensioning rule **157: Processings in sheets** is now additionally preset for the front view and the top view as follows:



(1) 3-D model, (2) front view and (3) top view in workshop drawing

Optimisation of base points

Since SP1, for automatically dimensioned developments of sheet metal parts, the (unshortened) projection lines are only drawn to the nearest point on the object to be dimensioned, i.e. to the next point on the contour. As of SP2 this also applies to sheet metal parts, steel engineering plates and glass panes.

Tangential transitions when dimensioning contours

The selection of base points has changed for the automatic dimensioning of contours. This change applies to applies to sheet metal parts, steel engineering plates and glass panes.

Until now, when dimensioning contours, the end points of the individual edges were dimensioned. However, this is usually not desired in practice. From SP2 onwards, the following base points are therefore taken into account when dimensioning contours:

Outer contours

- Lines running perpendicular to the dimension line
- Extreme points of outer contours that lie in the dimensional direction (also points lying on an arc or freeform line)
- All points with a bend

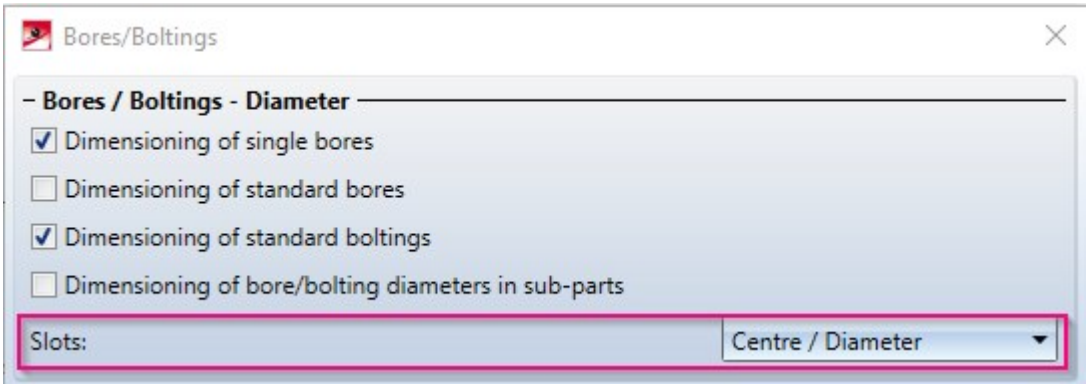
  Except for inverse curves, where the centre lies on both adjacent lines.
- Theoretical intersection points of curves in the outer contour

  For each arc that is tangentially adjacent to a straight line on both sides, the theoretical intersection point of the two lines is dimensioned if the angle of the two lines is greater than 50 degrees (i.e. not for very acute angles only). When merging with other dimensions, theoretical intersections are removed first, which affects the length of the projection lines.

Bores/Subtractions

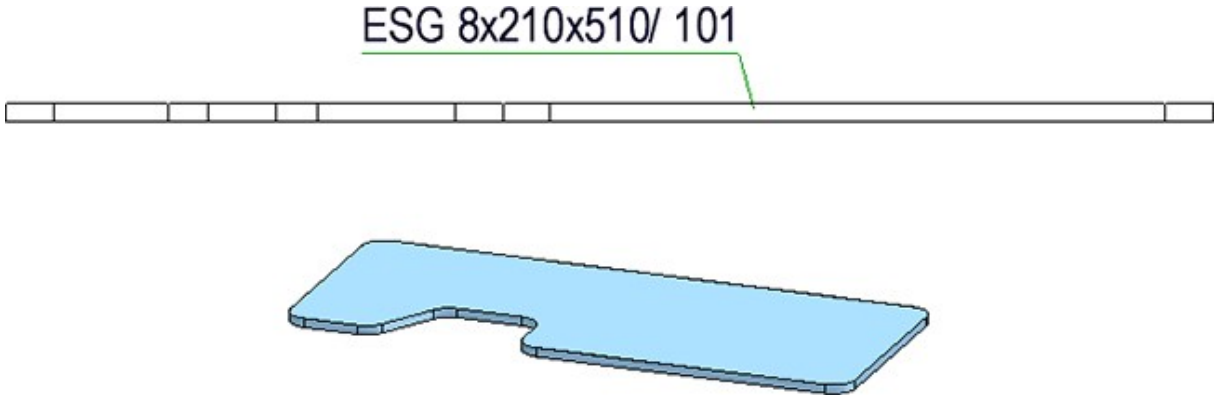
- Centres of circles, bores and slots

  Depending on the setting specified at **Drawing > Itemisation/Detailing > Dim... > Dimensioning Settings**

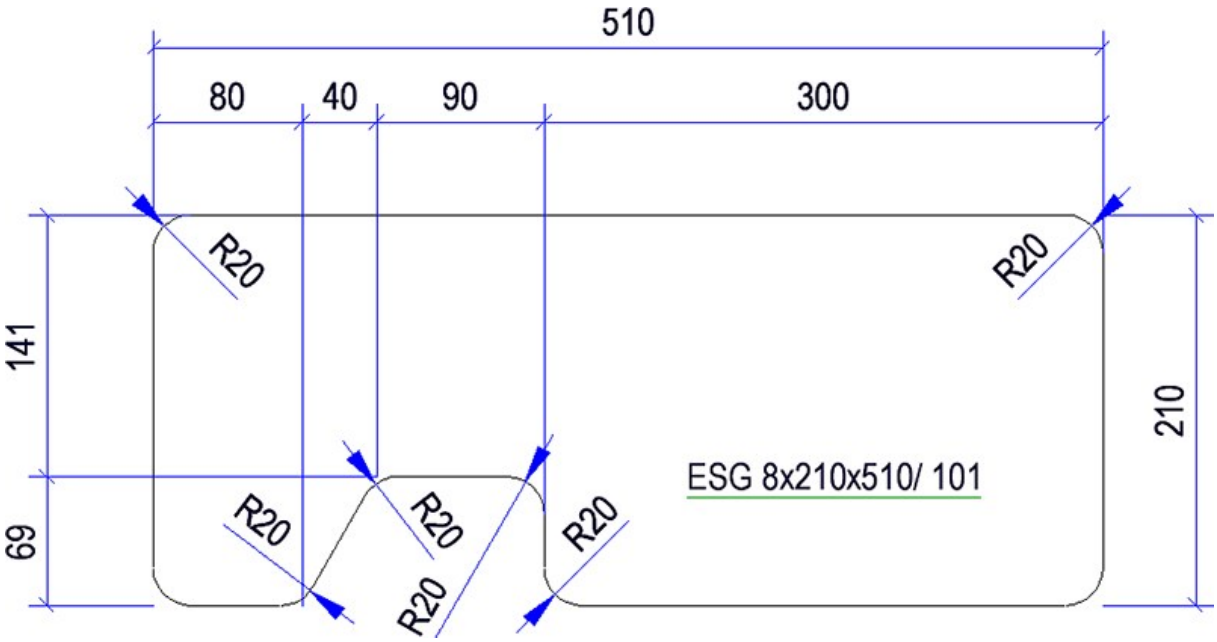


- Lines, points and theoretical intersection points as for outer contours

This change concerns the dimensioning rules for outer contours and bores/subtractions of sheet metal parts (and their developments), steel engineering plates and glass panes.



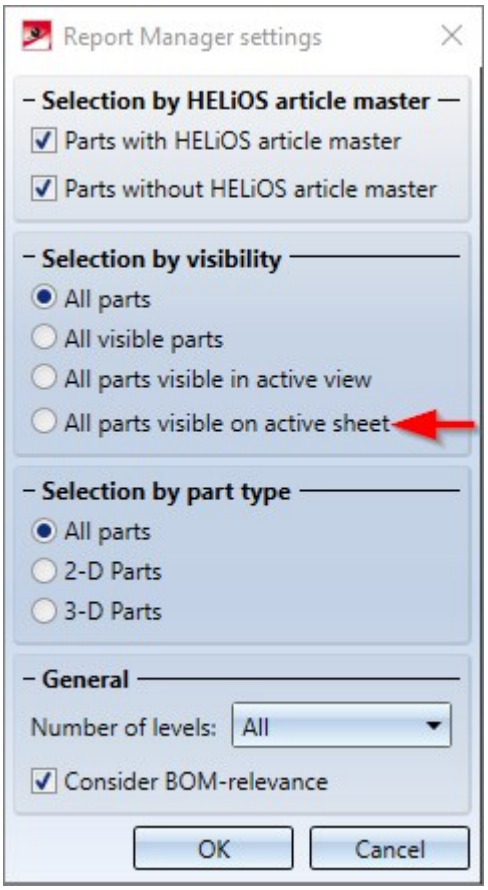
Dimensioning of outer contour, before HiCAD 2020 SP2



Dimensioning of outer contour with HiCAD 2020 SP2

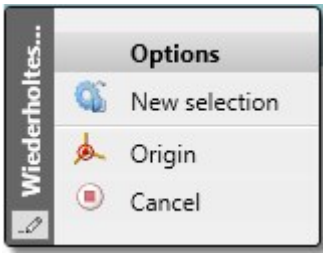
Bills of materials - All visible parts on active sheet

When creating BOMs, it is now possible to consider only the visible parts of the active sheet area. Therefore the settings for BOMs have been extended accordingly.



Insert part, via standard part catalogue

The function **Drawing > Insert Part > Cat** has been enhanced. When HiCAD asks yout to determine tfitting poiint in the drawing, you can right-click to activate the displayed context menu:



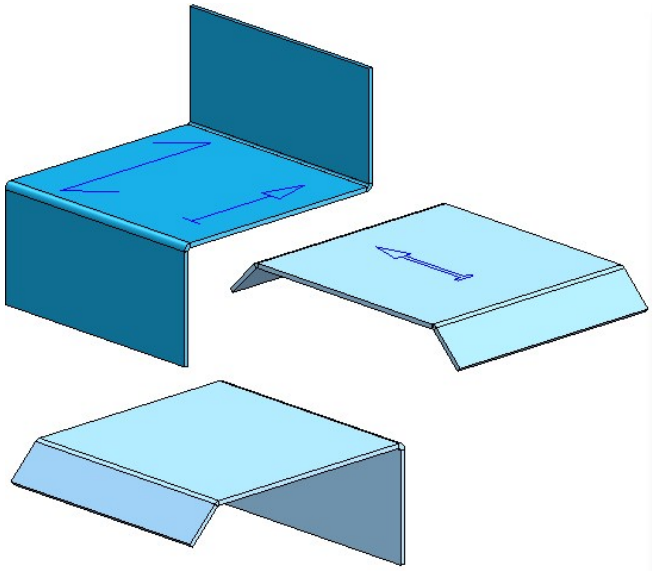
Here you have the possibility to select another catalogue part, to take over the origin of the coordinate system as target point or to cancel the function.

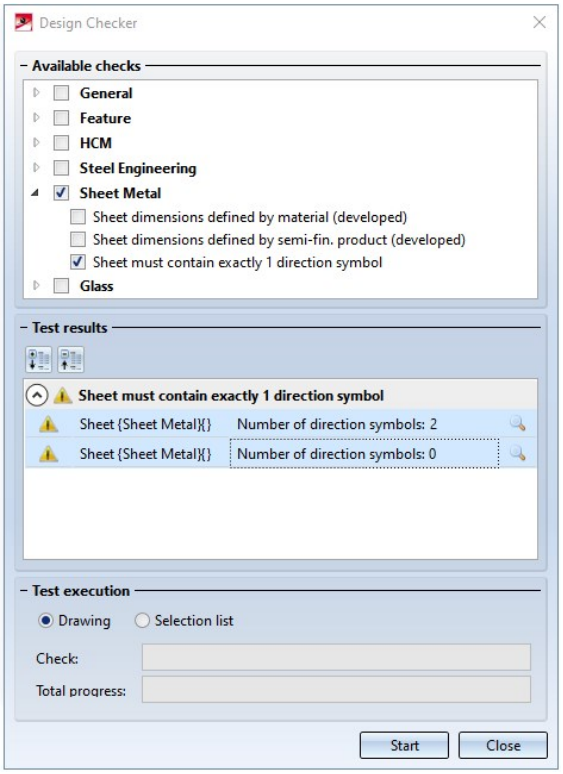
The function remains active after the insertion of a catalogue part. You can install the selected catalogue part several times or reactivate the context menu shown above by pressing the right mouse button.

You end the function with the middle mouse button or with the **Cancel** function in the context menu.

Design Checker - New check for sheet metal parts

The Design Checker offers a new check for sheet metal parts called **Sheet must contain exactly 1 direction symbol**. This check finds all sheet metal parts with no or more than one direction symbol.

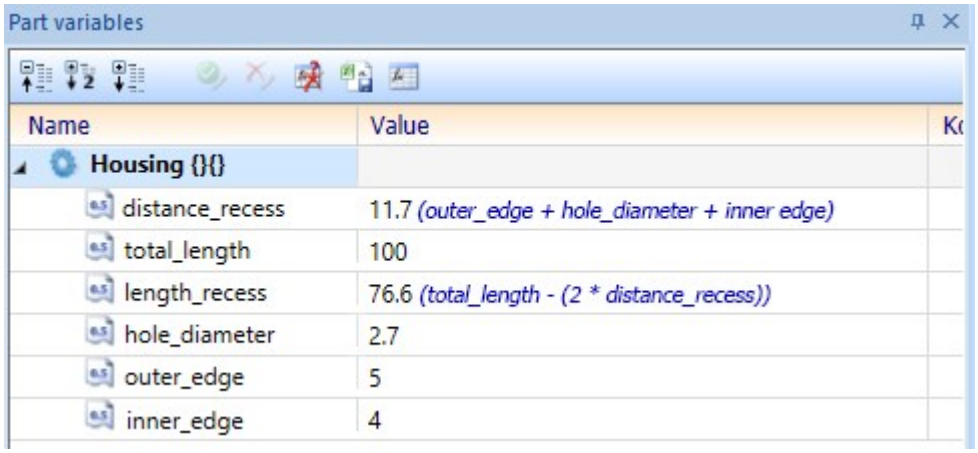




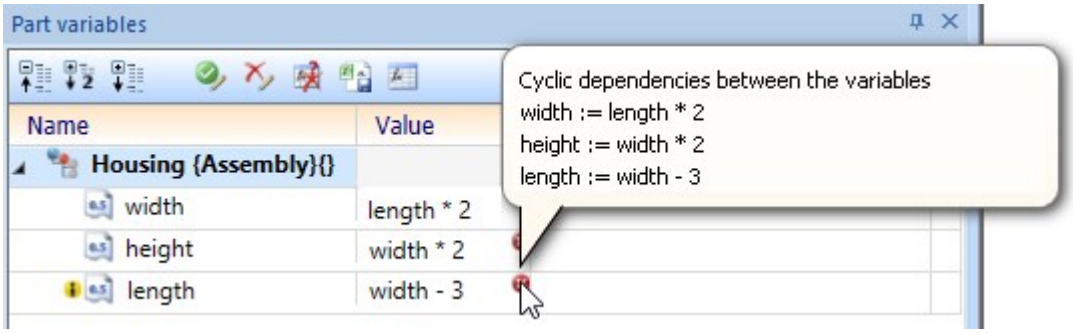
Nested variables and formulas

It is now possible to specify a formula that contains additional variables when defining the value of a variable. Especially with complex parts, the variable structure can become much clearer.

Thereby variables may be nested arbitrarily deeply into each other:



However no cyclic structures may occur (variable a refers to variable b, which refers to variable a), since these cannot be computed. In this case the automatic calculation of the formula values is no longer possible. The last values are retained and an error mark is displayed at the affected variables, which shows an explanatory tooltip when the cursor is moved over them:



In this case, remove the cyclic dependency by editing the formulas to remove the error markings and reactivate the automatic calculation of the formula values.

Service Pack 1 2020 (V 2501)

Restore HiCAD GUI to default

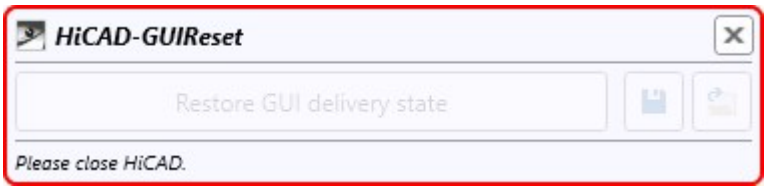
If you want to reset the HiCAD user interface, for example because ICN windows are no longer displayed although they have been activated with **Settings > Docking windows** or because incorrect icons or texts are displayed in the HiCAD toolbars or ribbons, use the contains the tool

HiCADGUIReset.exe

tool in the HiCAD EXE directory. This tool resets the entire HiCAD user interface, i.e. all corresponding settings in the Windows Registry and in the %APPDATA% directory, to the state after a new HiCAD installation.

After starting the tool, the following dialogue window is displayed.

HiCAD should be closed before calling the tool. If this is not the case, a corresponding message is displayed:



Then close HiCAD. HiCADGUIReset does not have to be closed before!

You start the reset by clicking the **Restore GUI delivery state** button. With the functions **Save GUI settings** and **Load GUI settings** the current settings can be saved and reloaded at any time. The file format is hicadgui. This is also useful if you use HiCAD at different workstations, e.g. on a laptop and on a computer with two monitors. You can simply save the settings of both workstations and then load the respective settings file you need.

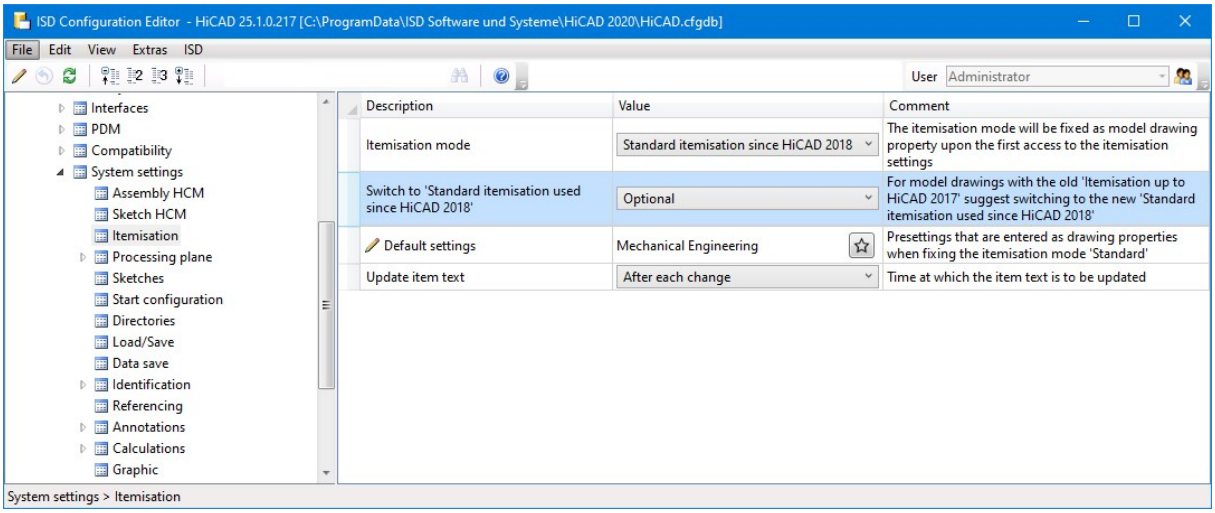
Before you start the reset you should always create a backup of your current settings.

Please remember that after a restoring of the GUI to default, user-defined toolbars will be no longer available!

Itemisation

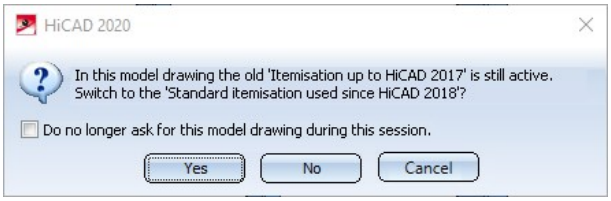
Switch to new standard itemisation

As of HiCAD 2020 SP1, you can use the Configuration Editor to define the procedure for starting the itemization in a model drawing that still uses the itemization up to HiCAD 2017. To do this, open the Configuration Editor and choose **System settings > Itemisation > Switch to 'Standard itemisation used since HiCAD 2018'**.



The following options are available here:

- **Never**
  No message or query will be displayed. This means that the itemisation up to HiCAD 2017 will still be used.
- **Optional**
  A message will be displayed, informing you that the itemisation up to HiCAD 2017 will still be used.



  You then have the option of determining whether or not the itemization should be converted to the standard itemization. To **Cancel** the itemization function, use the same named option.
- **Mandatory**
  Here, too, the message will be displayed, informing you that the itemisation up to HiCAD 2017 will still be used.

  Here, however, you only have the possibility to switch to the standard itemisation with **OK** or to **Cancel** the itemization function.

The default setting is **Optional**.

Sheet Metal parts with visible side

Sheet Metal parts that have an visible side identifier assigned to them are included in the identical part search. For example, if the model drawing contains two Sheet Metal parts of the same size, one of which has an ID for the visible side identifier, then the sheets are not considered identical and will receive different item numbers.

Enhanced ICN

New ICN window for views

The ICN window for views has been completely revised and makes working with many views much easier. Model and sheet areas are now marked by new symbols. Like the parts structure, the view structure is now also displayed - optionally - in multiple columns. For each view an icon for the respective view type, the view name, an icon for the view mode and the scale of the view are displayed. Also new is the extended toolbar, which allows you to create and update sheet areas without having to activate a context menu first. The automatic sorting mode is also worth mentioning, which ensures that sheet views are sorted correctly according to their name. This mode can be switched on and off.

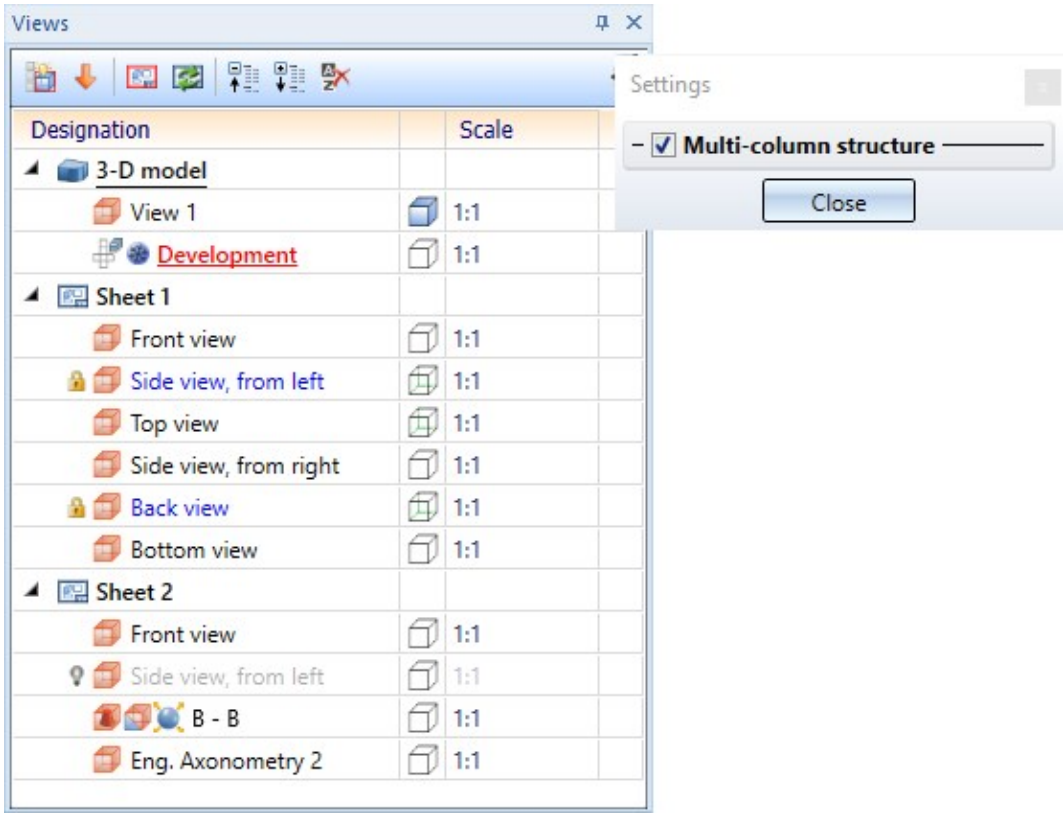
Also new is that certain views are marked with appropriate symbols. This applies to views that are hidden in the model drawing, views that have been "frozen", so-called "QuickViews", and sheet developments for which the synchronization was locked with the function **Suppress sheet update** in the context menu of sheet development.

The view is hidden in the model drawing.

The view is "frozen", i.e. it is visible but cannot be edited.

This symbol indicates sheet metal developments for which the adjustment was locked with the **Suppress sheet update** function in the context menu of the sheet metal development.

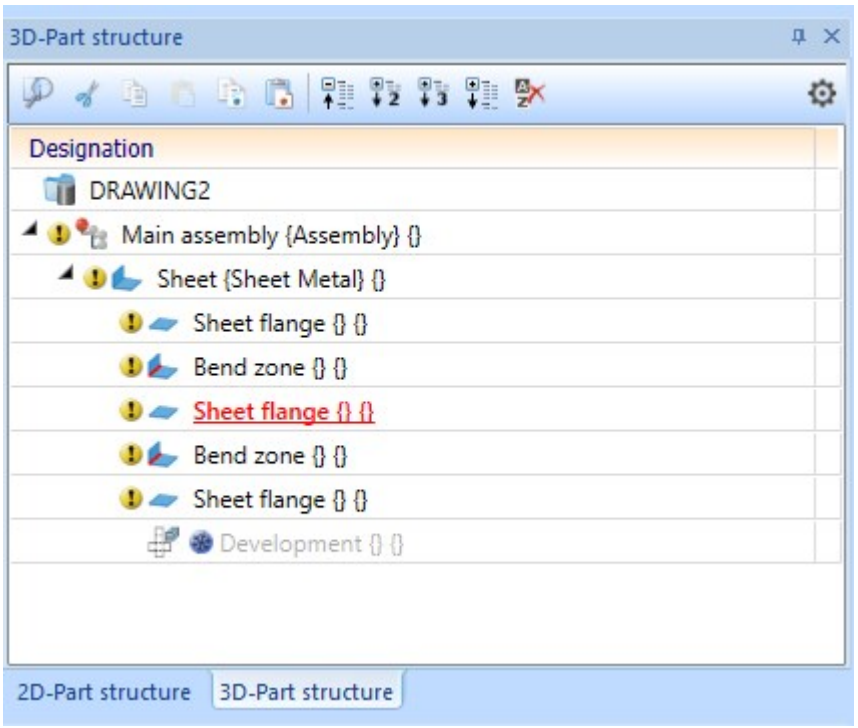
This symbol indicates that the view is a QuickView .



If, for example, a view is a list view, a sectional view and also an exploded view at the same time, then several symbols will be displayed

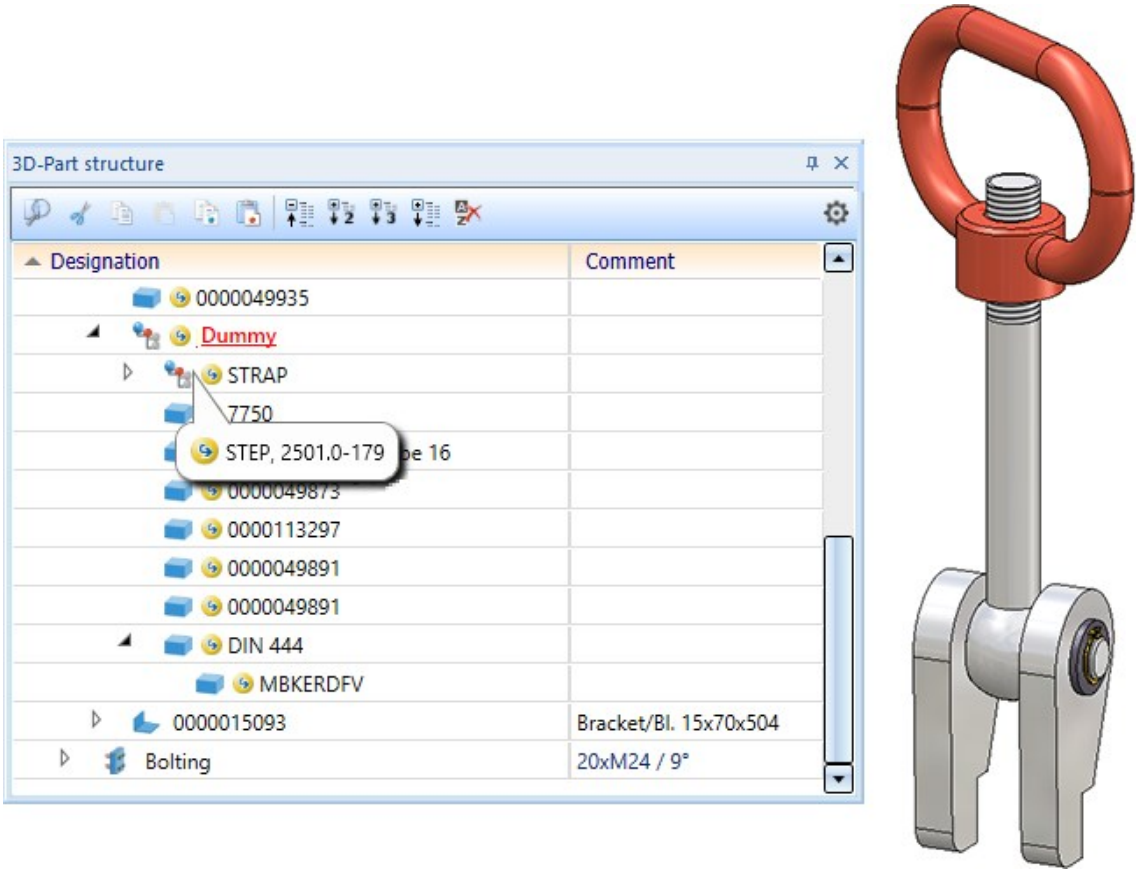
Marking of sheet developments

Sheet developments for which updates were prevented with the **Suppress update** function in the context menu of sheet development are now indicated in the ICN by the symbol.



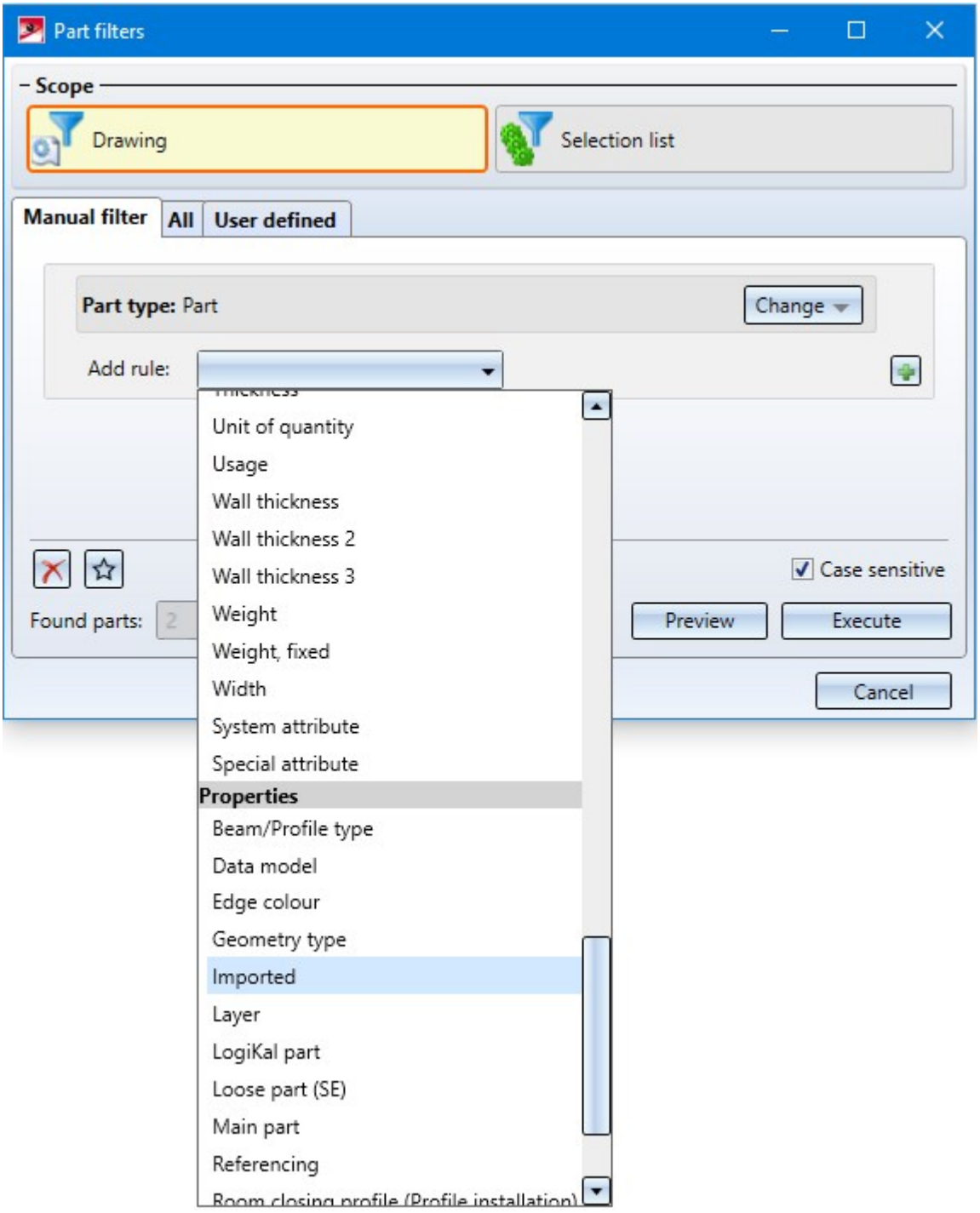
Marking of imported parts in the ICN

All parts that you insert into the model drawing using the **3-D Import** function are marked with the symbol in the ICN. If you place the cursor over the symbol, you will obtain further information on the relevant part, such as the imported file format and the HiCAD version with which the part was imported.



Part filters - Imported parts

With the **3-D Import** function, parts inserted into the model drawing can also be searched with the **Find** function in the transparent toolbar. The part filter dialogue has been extended accordingly.



Repeated paste from clipboard

Previously, pasting from the HiCAD clipboard was performed "in a loop", i.e. after pasting from the clipboard, the function remained active, so that you could paste the clipboard contents several times at different positions in the construction.

From HiCAD 2020 SP1 onwards, this behaviour can be defined in the Configuration Editor at **System settings > Miscellaneous** by activating / deactivating the **Repeated paste from clipboard** checkbox .

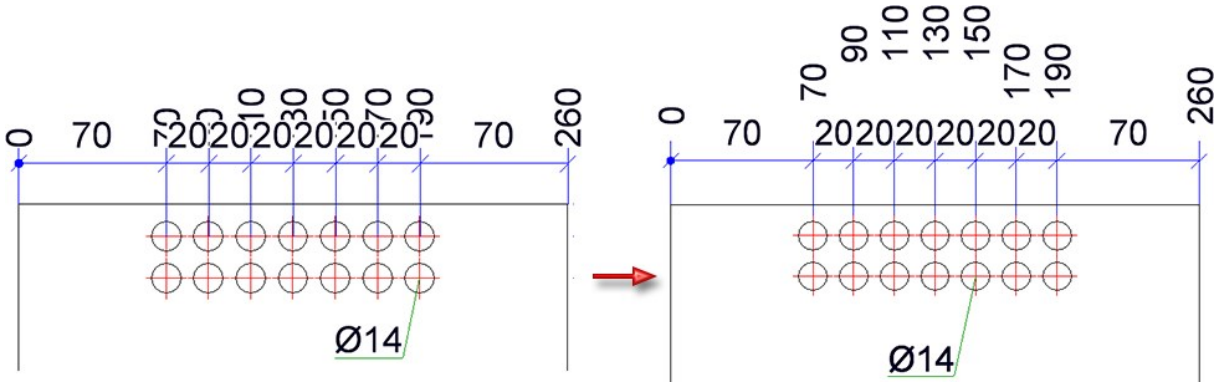
- If the checkbox is active, the **Paste from HiCAD Clipboard** function remains active after the first paste, so that you can insert the clipboard content several times at different positions in the model drawing. You end the function with the middle mouse button.
- If the checkbox is inactive, the **Paste from HiCAD Clipboard** function will be automatically ended after the first paste. This is the default setting for new installations as of SP1.

During an update installation, the previous behaviour remains unchanged, i.e. repeated pasting remains the default setting.

Derived drawings

Readability of running dimension chains

The creation of running chain dimensions has been improved by preventing dimension figure collisions.



AutoSave - Time interval per model drawing

As of HiCAD 2020 SP1, the time interval of the AutoSave mechanism **per opened model drawing** applies, i.e. HiCAD remembers the elapsed time of the current model drawing in the case of a drawing switch. For the model drawing that you switch to, the time interval runs from the first change to the model drawing or from a remaining time previously noted for this model drawing.

An example:

The time interval is set to 10 minutes and you are working with three model drawing times K1, K2 and K3.

Timer		Time remaining
[00:00]	You have loaded K1 and for for 9 minutes in K1.	K1 = 1 min.
[09:00]	You switch to an empty drawing thumbnail, load K2 and work on K2 for 11 minutes.	
[19:00]	After 10 minutes the time interval for K2 has expired and an interim backup of K2 is created.	K2 = 9 min.
[20:00]	You switch back to K1 and work on K1 for 4 minutes.	
[21:00]	After 1 minute, an interim backup of K1 is created.	
[24:00]	You switch to an empty drawing thumbnail, load K3 and work on K3 for 12 minutes.	K1 = 7 min.
[34:00]	After 10 min. the interim backup of K3 takes place.	
[36:00]	You switch to K2 and work on K2 for 13 minutes.	K3 = 8 min.
[45:00]	After 9 min. K2 is backed up.	
[49:00]	You switch to K1.	K2 = 6 min.
etc.		

Others

Delete parts

If parts of the model drawing are deleted via the context menu in the ICN or in the model drawing, the part that was active before the part was deleted is then active. If you delete the active part, the first part of the part structure is active.

The same applies to multiple selection. This means that when you delete a multiple selection, the part that was active before the multiple selection was deleted is then active. If the active part belongs to the deleted parts list, the first part of the parts structure is then active.

Coordinate System display

In the **CS display** toolbar, the symbol for part orientation has been changed:

Major Release 2020 (V 2500)

New Mask Editor in HELiOS 2020

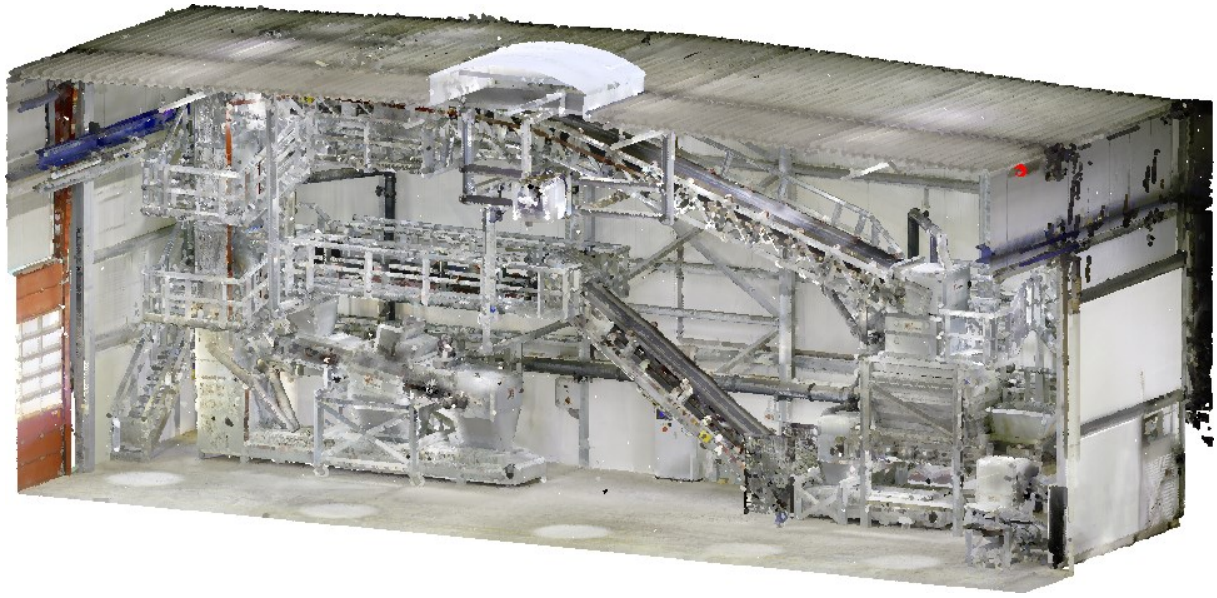
Due to a complete revision of the Mask Editor for HELiOS 2020 the previous mask format has been changed from .MSK to .XML. In addition, mask files are now managed outside the installation directory, resulting in some changes in the system architecture that administrators should urgently consider:

- The masks of older versions that were previously stored in the \sys\ directory of the installation are automatically converted to the new XML format by an update installation of an older HELiOS version to HELiOS 2020 (V 2500.0) or higher and stored at **%Programdata%**.
- At %**Programdata%**, the masks are stored in a version-specific path, i.e. for HELiOS 2020 this path could look like this: "C:\ProgramData\ISD Software und Systeme\HELiOS 2020\GUI\Masks". In a later version, the version identifier in the path will be correspondingly higher.

Please read the installation instructions before updating to HiCAD / HELiOS 2020.

Licensing

- The new Extension Module **HiCAD Point Cloud** provides an associative solution for the integrated representation of point clouds in HiCAD. Besides the import of common point cloud formats, the imported point cloud can be placed and clipped in HiCAD. When drawing CAD geometries, you can also refer specifically to point information from the point cloud. A prerequisite for using the new module is either one of the Basic Modules HiCAD Creator or HiCAD Solution.

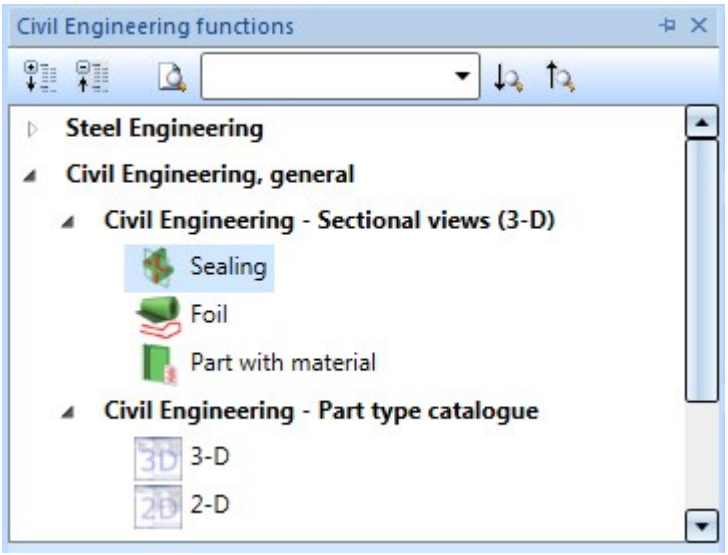


Example of a Point Cloud imported in HiCAD (Image: VHV Anlagenbau GmbH, Hörstel)

Highly enlarged section of the point cloud shown above

An example of a point cloud can be found here.

- The civil engineering functions (in the docking window under **Civil Engineering General**) are now also available in the following extension modules:
  - Sheet Metal Professional
  - Beams
  - Metal Engineering
  - Steel Engineering
  - Steel Engineering, Stairs + Railings
  - Profile Installation
  - Element Installation



- The basic module **HiCAD Spooler** now contains the same range of functions as **HiCAD Solution**, e.g. the functions for itemisation and drawing derivation. Furthermore, the module can be linked with the same extension modules as HiCAD Solution, for instance with the **HiCAD Beams** module.

Performance

- The HiCAD dimensioning functions are now much faster thanks to an internal change to saving by bodies.
- In large model drawings and assemblies, UNDO actions after cloning of assemblies can now be carried out much faster.
- With the release of HiCAD 2020, both the changing of item numbers and the updating of item number tags have been speeded up significantly.

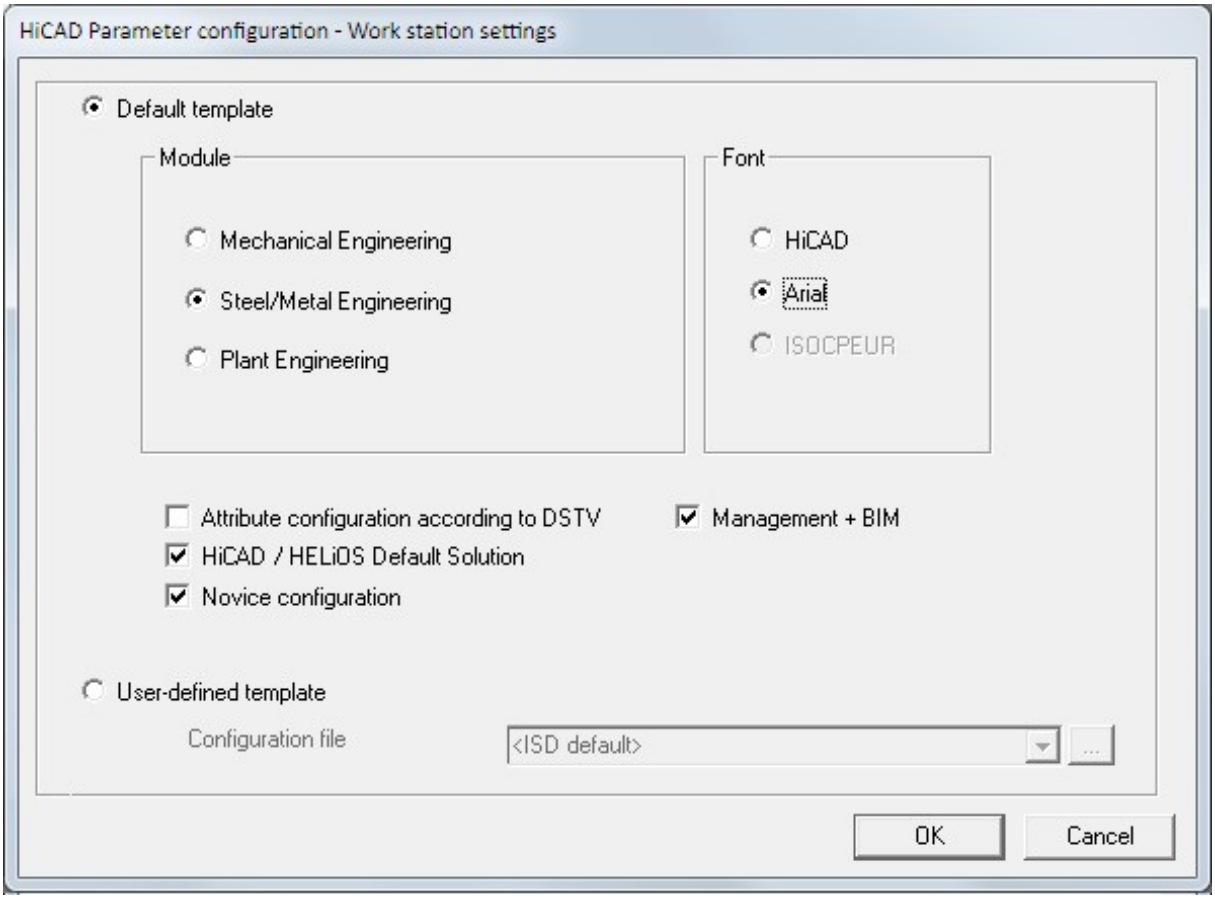
AutoSave

The AutoSave function has been upgraded particularly in regard to the backup time:

- The automatic backup timer is enabled only after a drawing has been fully loaded and all updates have been completed.
- When saving a drawing, the timer will be reset.
- The symbols are enlarged.

ISOCPEUR font

HiCAD 2020 supports the ISOCPEUR font, which is, for example, recommended by the Austrian Steel Engineering Association. If this font has been installed on your computer or network, it can be (like the Arial font) selected in the Parameter configuration dialogue window (ParKonfigComp.exe / ParKonfigUser.exe) as the default font. Otherwise, this font is greyed out in the dialogue window.



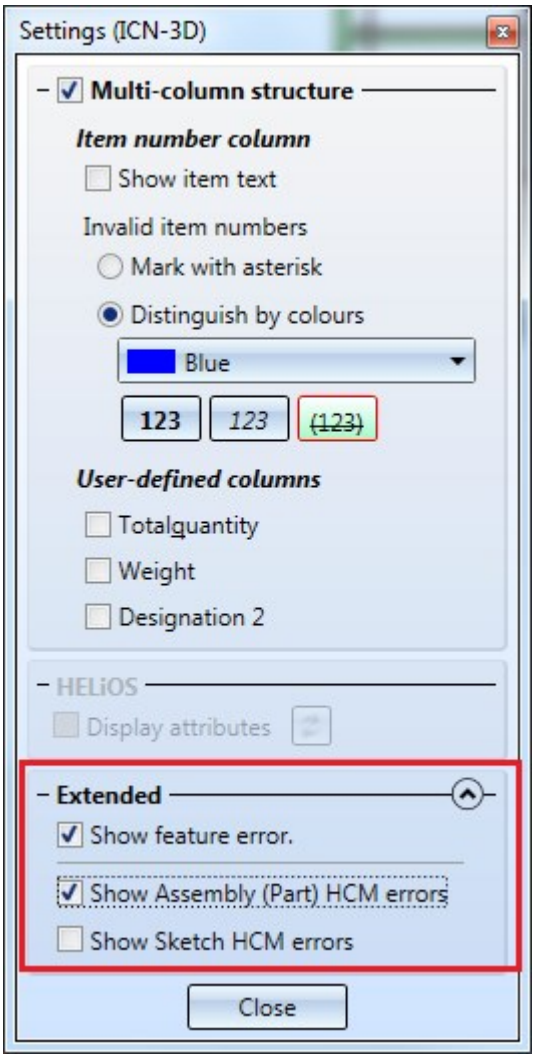
Please note:
The ISOCPEUR font is not part of HiCAD's scope of delivery and will therefore not be installed together with HiCAD.

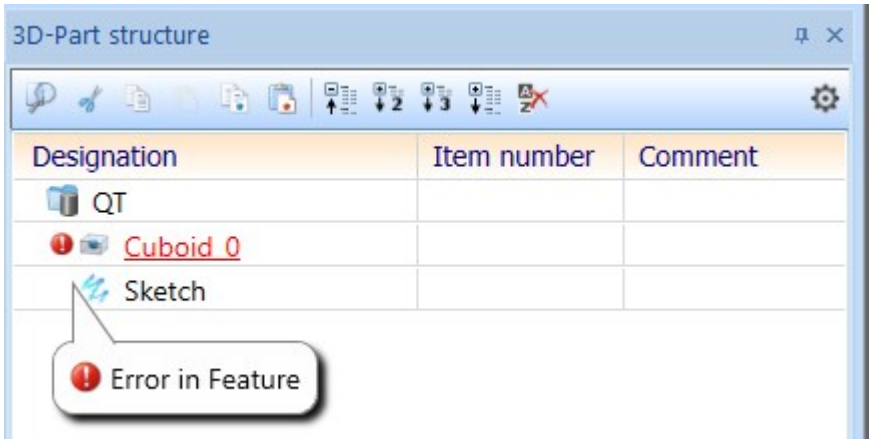
Important:

By running the parameter configuration, customer-specific settings, flags, drawing frames etc. are changed to settings predefined by ISD.

Enhanced ICN features

The Settings for the 3-D part structure allow to indicate parts with feature error or HCM constraints with error in the part structure.



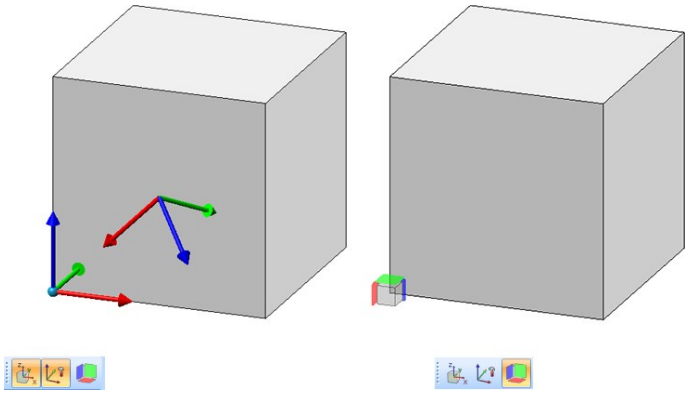


Switch on/off representation of Coordinate Systems

For a better use of coordinate systems, HiCAD 2020 and higher allows you to switch on and off the part and fitting coordinate system of the currently active part directly via the corresponding symbols at the bottom of the user interface. As an alternative, you can either use the F8 key for the part coordinate system or SHIFT + F8 for the fitting coordinate system.

When switched on, a plastic representation of the respective coordinate system is displayed.

The following graphic shows an example with the part and installation coordinate system switched on.



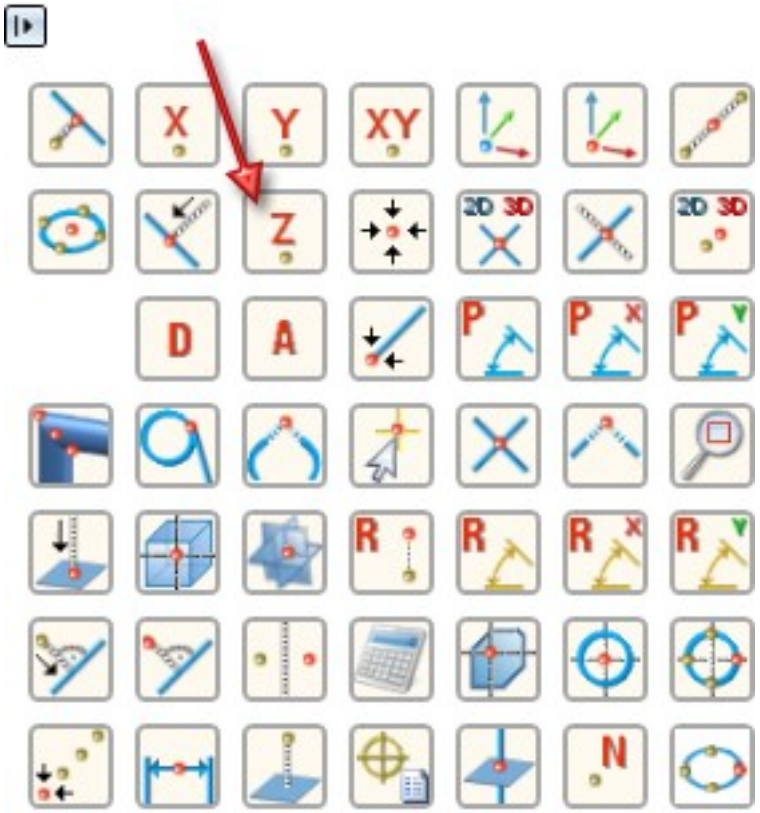
Furthermore, the orientation of a part, which you have defined with the **Part orientation** function in the context menu for parts or the profile orientation can be shown/hidden. To do so, click on the symbol

Profile orientation or Alignment of active 3-D part, alternatively use the F6 key. This is only possible if the setting **Switch on/off with F6 key**under **System settings > Visualisation > Show alignment of active 3-D part in drawing** or **System settings > Visualisation > Indicate orientation of active Steel Engineering beam** in the Configuration Management has been activated.

Point options

Absolute Z-position (AZ)

Analogous to the point options **AX** und **AY** the point option **(AZ) Absolute Z-position** is now available. This option determines a point by specifying its Z-coordinate and accepting the X- and Y-coordinates of the last point.



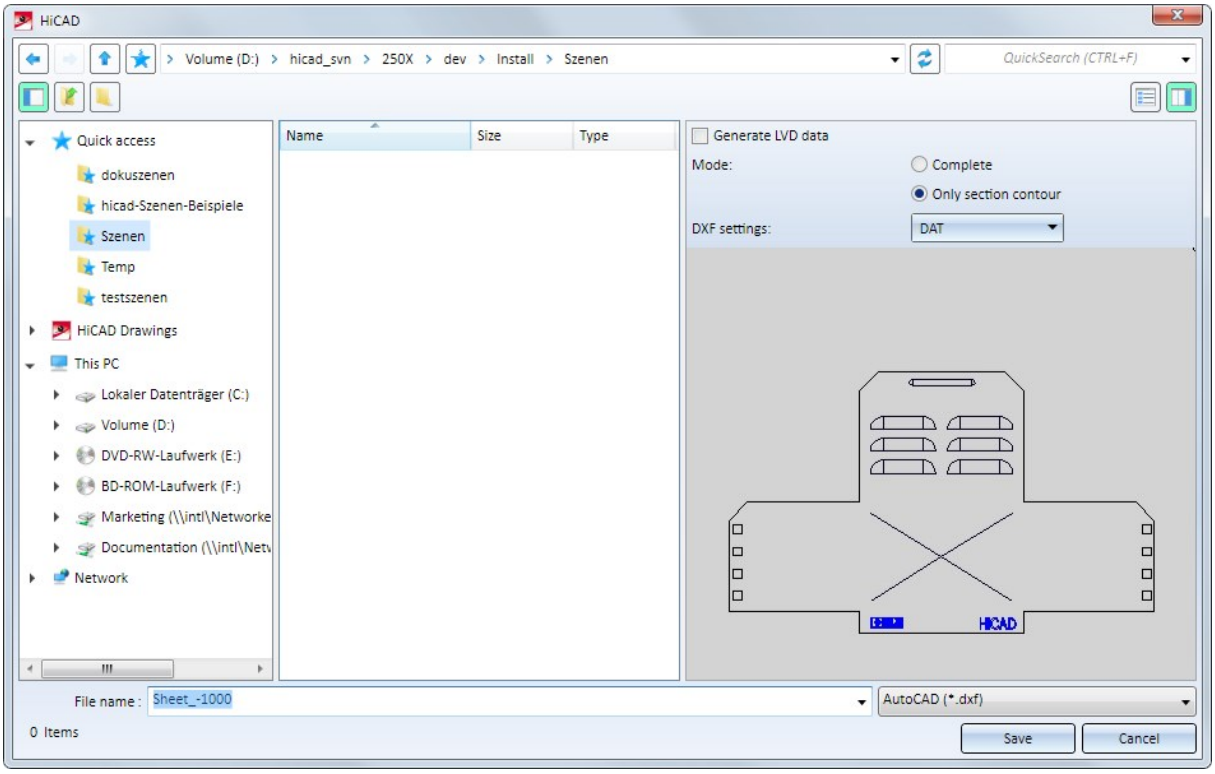
Point from Point Cloud (PW)

The option **(PW) Point from Point Cloud** is required for working with **HiCAD Point Cloud** module.

Directories

Quick Access for directory selection

As with the dialogue window for loading and saving of files, selected directories can now also be added to the **Quick Access** folder, e.g., for DXF export of Sheet Metal parts.



Improved designations of directories in HiCAD Settings

The designations of the HiCAD directories in the dialogue window of the function > Settings > Directories have been improved.

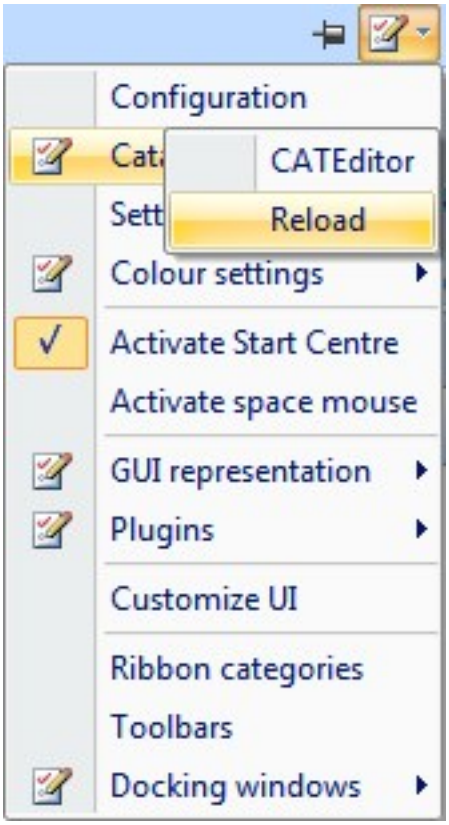
ID	Directories	Designation
	C:\HiCAD\sys	System directory
A:	C:\HiCAD\szenen	Drawings
B:	C:\HiCAD\norm	Standard parts (2-D)
C:	C:\HiCAD\szenen	Model drawing (default directory)
D:	C:\HiCAD\ega	Preview images
E:	C:\HiCAD\szenen	Drawings
F:	C:\HiCAD\szenen	Drawings
G:	C:\HiCAD\szenen	Drawings
H:	C:\HiCAD\szenen	Drawings
I:	C:\HiCAD\szenen	Drawings
J:	C:\HiCAD\szenen	Drawings
K:	C:\HiCAD\material	Material directory
L:	C:\HiCAD\REFTEILE1	Referenced parts (default directory)
M:	C:\HiCAD\REFTEILE2	Referenced parts (exchange directory)
N:	C:\HiCAD\szenen	Drawings
O:	C:\HiCAD\pneu	Pneumatics symbols (2-D)
P:	C:\HiCAD\plo	Temp. plot files
Q:	C:\HiCAD\szenen	Drawings
R:	C:\HiCAD\makroabw	User guidance Sheet Metal
S:	C:\HiCAD\szenen	Drawings
T:	C:\HiCAD\szenen	Drawings
U:	C:\HiCAD\szenen	Drawings
V:	C:\HiCAD\szenen	Drawings
W:	C:\HiCAD\etesym	E-Tech. symbols (2-D)
X:	C:\HiCAD\temp	Temp. interface files
Y:	C:\HiCAD\custom	Files with customized programming
Z:	C:\HiCAD\sys	System directory
0:	C:\HiCAD\makro2d	User guidance 2-D
1:	C:\HiCAD\makro3d	User guidance 3-D
2:	C:\HiCAD\makrost3d	User guidance Steel Engineering
3:	C:\HiCAD\makromod	User guidance, HCM
4:	C:\HiCAD\szenen	Drawings
5:	C:\HiCAD\makroanl	User guidance, Pipeline planning
6:	C:\HiCAD\PlantParts	Parts, Pipeline planning
7:	C:\HiCAD\PID\Modules	P+ID modules
8:	C:\HiCAD\PID\Projects	P+ID projects
9:	C:\HiCAD\PID\Libs	P+ID libraries
#:	C:\HiCAD\temp	Automatic data backup, other temporary

Update catalogues

In the Configuration Editor at **System settings > Catalogue > Track catalogue changes** you can now determine how to proceed in case of catalogue changes in HiCAD. The default setting is **Do not track**, that is, the catalogue data will be loaded only once when you start HiCAD. After this, HiCAD will no longer check whether newer data exist. The reason for this is that a check for up-to-dateness of catalogues in HiCAD can significantly deteriorate the performance in some situations.

As of HiCAD 2020, the new **Reload** function offers the option to update catalogues after applying changes to them without having to close HiCAD, and without having to put up with drops in performance due to repeated checks for up-to-dateness.

You find the new function at **Settings > Catalogue**.

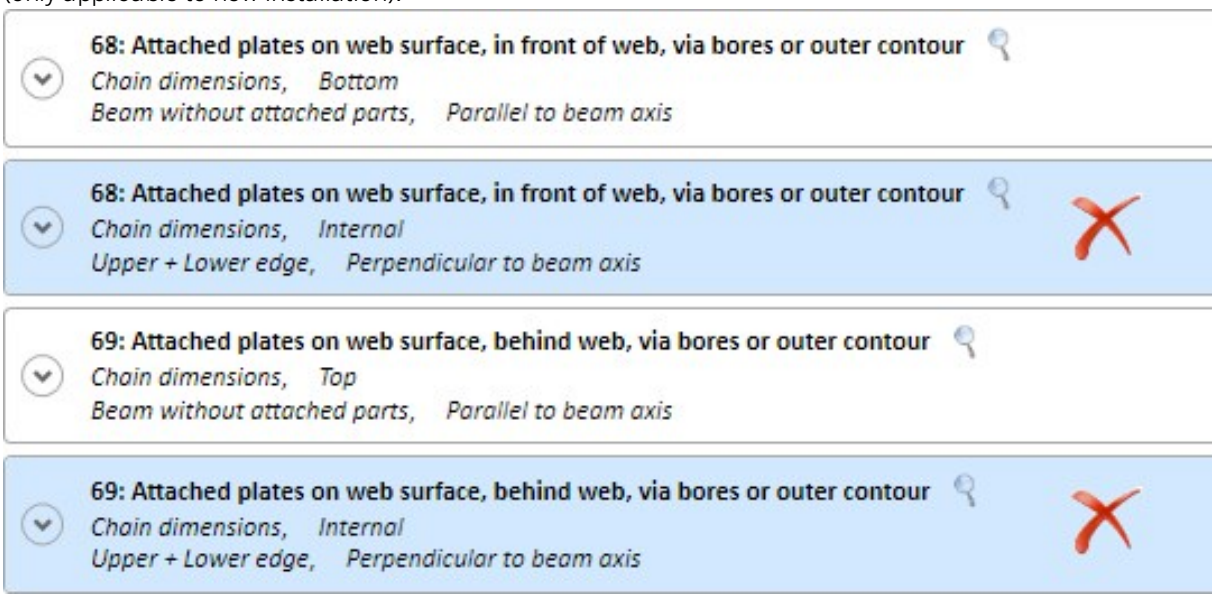


Drawing derivation

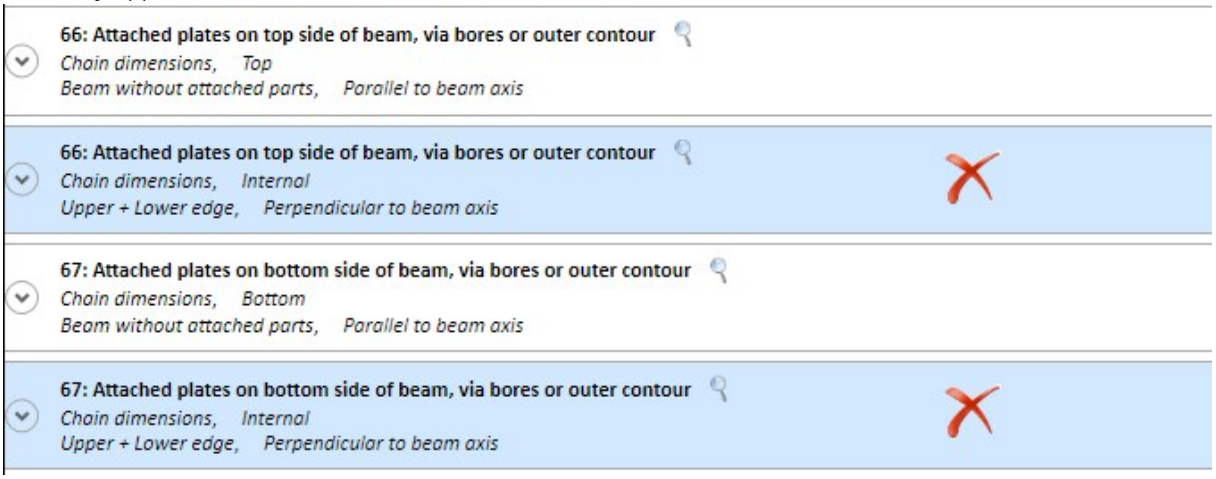
Dimensioning rules

- In the basic settings for the Usage "Sheet Metal" in the Configuration Editor (Configuration **DEFAULT (KANTBLECHE)** the dimensions with the ID
  - 135: Parametric linear dimensions of sheets (SHEETMETAL_PARA_DISTANCE) and
  - 136: Parametric angular dimensions of sheets (SHEETMETAL_PARA_ANGLE)

  have been removed from all views. As a result, die dimensioning rules of this configuration has changed (only applicable to new installation).
- In the settings for the Usages **Assembly column** and **Assembly girder** in the Configuration Editor (Configuration ASSEMBLY_COLUMN and ASSEMBLY_BEAM) the following dimensions have been removed from the **Top view** (only applicable to new installation):



- In the settings for the Usages **Assembly column** and **Assembly girder** in the Configuration Editor (Configuration ASSEMBLY_COLUMN and ASSEMBLY_BEAM) the following dimensions have been removed from the **Front view**(only applicable to new installation):

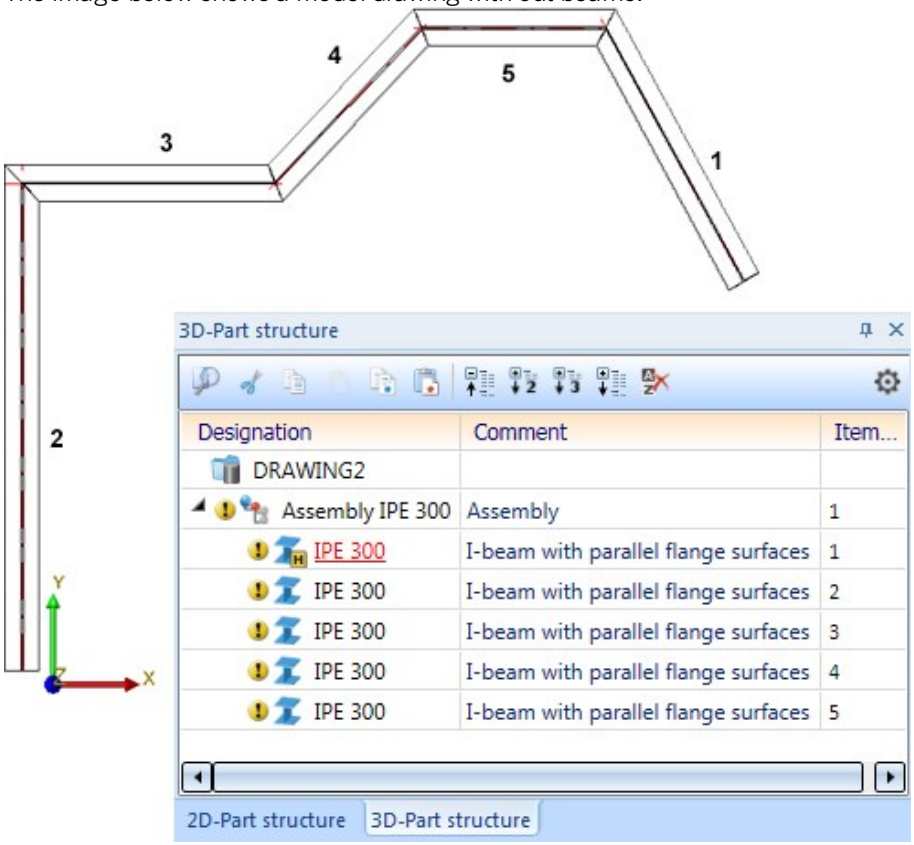


- For the dimensioning rules
  - 64: Length of cut at beam/profile start (SECTIONAL_ LENGTH_BEGIN) and
  - 65: Length of cut at beam/profile end (SECTIONAL_ LENGTH_END)

  the dimension type **Individual dimension** is now also permissible, in addition to the dimension type **Chain dimension**. If you choose **Chain dimension** as the dimension type, the chain of dimension will be lengthened according to the selected reference, and becomes a chain dimension. If you choose **Individual dimension**, only the length of the cut will be dimensioned. When updating model drawings from versions older than 2020, only the cut will be dimensioned, and no chain dimension will be created.

  An example:

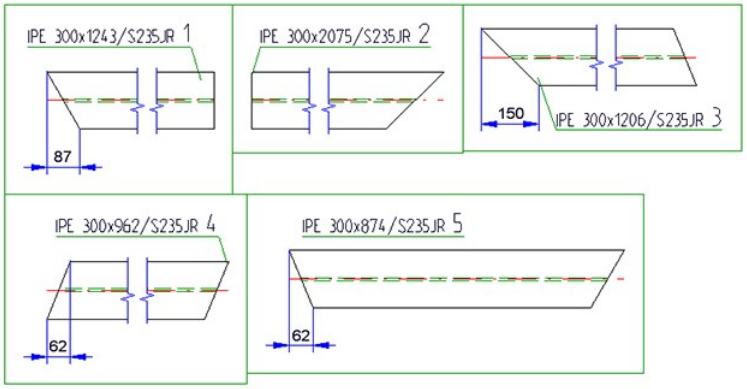
  The image below shows a model drawing with cut beams.



  From this model drawing, a drawing was derived, considering only the beams and generating the top views:

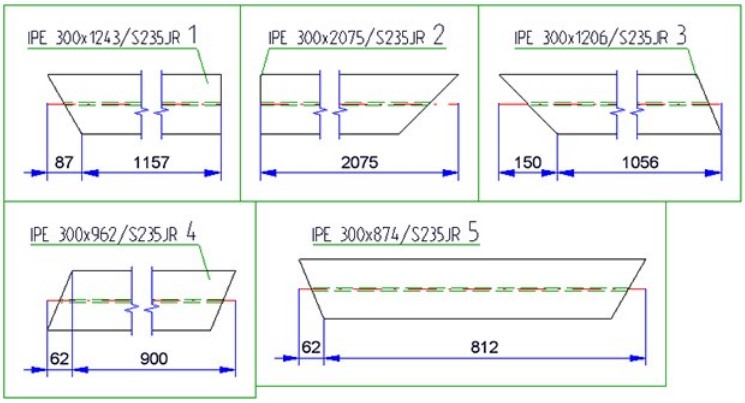
64: Length of cut at beam/profile start

ID:	64: Length of cut at beam/profile start
Type of dimension:	Individual dimension
Position of dimension chain:	Automatic
Dimension chain reference:	System axes
Direction of dimension chain:	Parallel to beam axis



64: Length of cut at beam/profile start

ID:	64: Length of cut at beam/profile start
Type of dimension:	Chain dimension
Position of dimension chain:	Automatic
Dimension chain reference:	System axes
Direction of dimension chain:	Parallel to beam axis



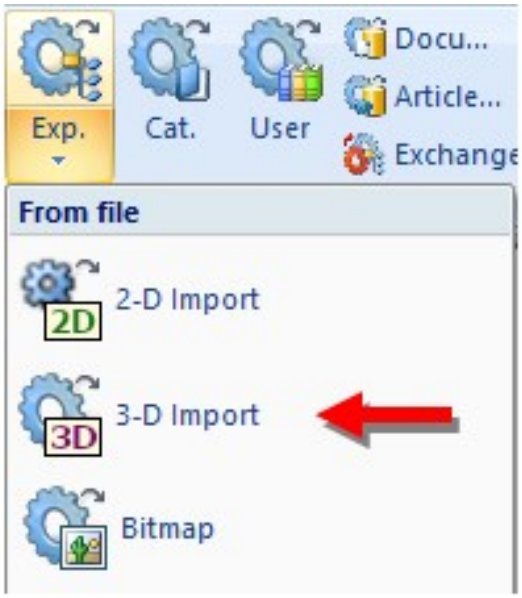
Import of foreign formats

Insert 3-D foreign formats as parts in current model drawing

Data in the 3-D formats:

- STEP (*.stp, *.step)
- STL (*.stl) (the STL file must exist in the ASCII format!)
- IGES (*.igs, *.iges)
- VDAFS (*.vda)
- CATIAV4 and CATIAV5 (*.MDL, *.Model, *.EXp, *.ISO, *.DLV3, *.CATPart, *.CATProduct)
- ACIS (*.SAT)
- Parasolid (*.x_t, *.xmt_txt, *.x_b)
- ProE (*.prt)
- Unigraphics (*.prt)
- SOLIDWORKS (*.sldasm, *.sldprt)
- Inventor (*.ipt, *.iam)
- AutoCAD (*.dxf, *.dwg)
- JT (*.jt)
- PLMXML (*.plmxml)
- IFC (*.ifc, *.ifczip)

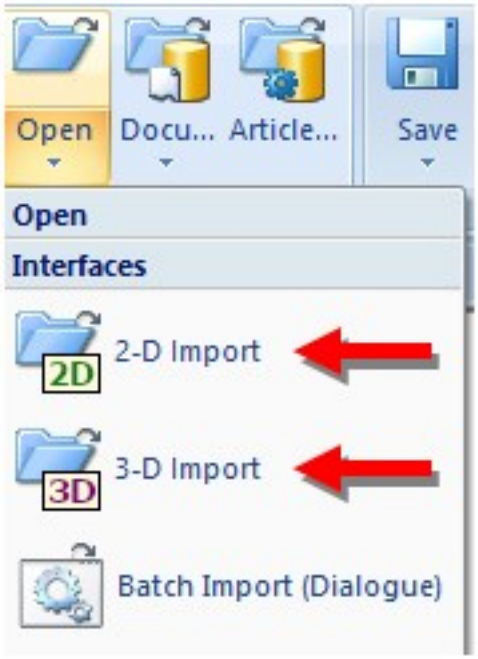
could previously only be imported to a new HiCAD drawing with the **Drawing > New/Open > Open** > **3-D Import** function. If you want to insert data in these formats to the current drawing, now choose the renamed function **Drawing > Insert Parts > Exp.** > **3-D Import**.



The parts will then be inserted in the current drawing as an assembly with the name of the selected file.

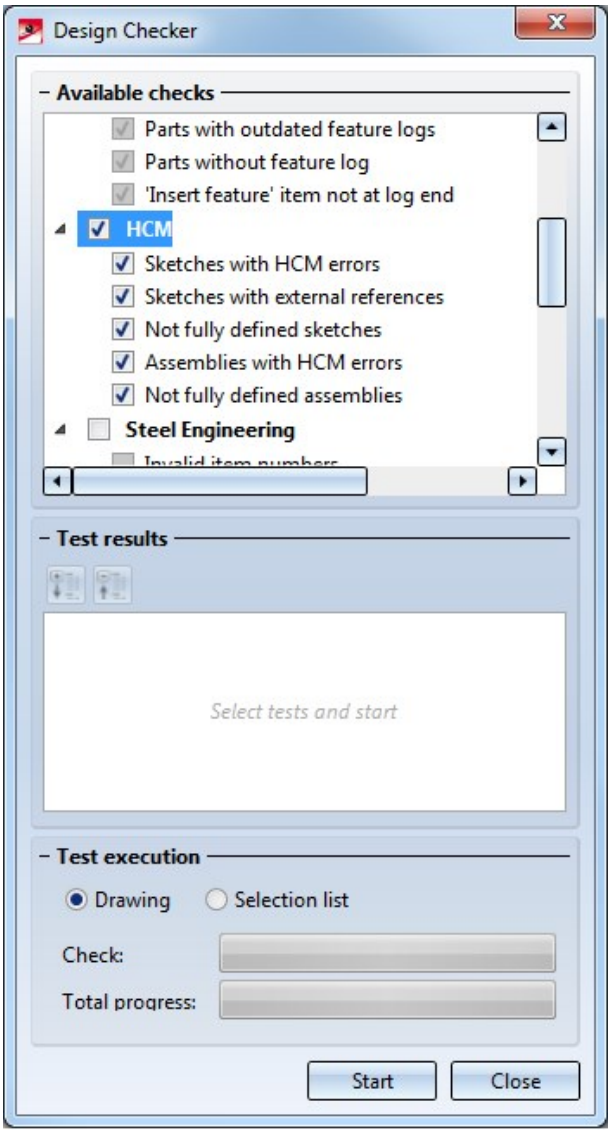
Import model drawings

When importing 2-D/3-D model drawings via interfaces, HiCAD 2020 distinguishes between 2-D and 3-D imports. For this purpose the old function **Drawing > New/Open > Open > STEP, IGES, ...** has been split into the 2 new functions called 2-D Import and 3-D Import.



HCM models in the Design Checker

The **Design Checker** now offers the following additional tests for HCM models.



- **Sketches with HCM errors**
  lists all sketches containing incorrect HCM references.
- **Sketches with external references**
  lists all sketches containing HCM constraints with external references.
- **Not fully defined sketches**
  lists all sketches that are not fully defined.
- **Assemblies with HCM errors**
  lists all assemblies containing incorrect references.
- **Not fully defined assemblies**
  lists all assemblies containing a HCM model that is not fully defined.

2-D

Service Pack 2 2020 (V 2502)

New Visualisation toolbar

The previous "CS display" toolbar has been extended with additional functions for fast hiding and showing of objects and renamed to **Visualisation**. With the new functions you can, among others, enable/disable the visibility of isolated points and 2D dimensions with one click. This allows for example to hide all dimensions of a drawing without having to call a function from the context menu or ribbon first..

Toggle visibility of isolated points

AAll isolated points and the point identification of the construction can be switched on/off with one click.

Toggle visibility of dimensions

All 2-D and 3-D dimensions of a drawing can be switched on/off with one click. This applies to both drawing dimensions and parameter dimensions.

New Autopilot settings toolbar

The **Autopilot settings** toolbar below the drawing area can now be used to control which points the autopilot offers as snap points. In the toolbar the possible snap points are highlighted.

With this bar you can control which point options should be visible on the Autopilot by activating / deactivating the icons. For example, if you click on , the snapping of isolated points is disabled.

The currently selected Autopilot settings apply to the active HiCAD session. You can define which settings should be active when restarting HiCAD in the Configuration Editor at **System settings > Identification > List of point snap options**.

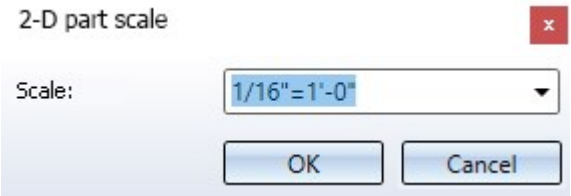
If you click on the icon in the **Autopilot settings** toolbar, the settings from the Configuration Editor are restored.

North American scale

Scales in HiCAD are used as

- main scale of a model drawing,
- Scale of a view,
- Scale of a 2-D part, and
- Scale of Steel Engineering detail drawings.

The scale can be selected from a selection box in the corresponding HiCAD function dialogues. In addition, in many cases it is also possible to enter a scale directly. The scales available in the selection boxes of the HiCAD functions are defined in the file **SZENE-MASSSTAB.TXT** in the HiCAD subdirectory **MAKRO2D** for 2D parts. Please note that HiCAD supports the European scale logic by default, i.e. scale specifications in the form n:m, e.g. 1:1, 1:10, 5:1 etc



However, the US scale logic differs from that in Europe. Therefore HiCAD offers the possibility to extend the file **SZENE-MASSSTAB.TXT** accordingly by defining further scale specifications.

SZENE_MASSTAB.TXT | StbEtZng_MASSSTAB.DAT

```
# Maßstäbe / scale definitions
# Europäisch / European
1:1
1:2
1:2.5
1:5
1:10
1:20
1:50
1:100
1:200
2:1
5:1
10:1
20:1
50:1
100:1

# US
1:1536 (1/128"=1'-0")
1:768 (1/64"=1'-0")
1:384 (1/32"=1'-0")
1:192 (1/16"=1'-0")
3:384(3/32"=1'-0")
1:96 (1/8"=1'-0")
1:48 (1/4"=1'-0")
3:96 (3/8"=1'-0")
1:24 (1/2"=1'-0")
3:48 (3/4"=1'-0")
1:12 (1"=1'-0")
3:24 (1-1/2"=1'-0")
0.25(3"=1'-0")
0.5 (6"=1'-0")
1 (1'-0"=1'-0")
```

Example of an expanded SZENE-MASSSTAB.TXT file
' foot, " inch, 1 foot = 12 inches, 1 inch = 2,54 cm

Changing the file SZENE-MASSSTAB.TXT

To structure the file, any empty lines and comments can be inserted, whereby comment lines must begin with the # character. Each scale specification must be in a separate line. The following notations are supported for the scale specification.

Notation		Examples
1.	**Specification in the scale format n:m**	1:10 1:1 2:1 5:1
2.	**Specification as factor**	0.1 1 2 5
3.	**As factor with display text** The factor is at the beginning of the line, the display text in brackets behind it. The display text is used by HiCAD in dialogues and info messages.	2.5 (2‘-6“ = 1‘-0“) 1 (1`-0“ = 1‘-0“)
4.	**In the scale format with display text** The scale is at the beginning of the line, the display text in brackets behind it. The display text is used by HiCAD in dialogues and info messages.	1:12 (1“=1‘-0“) 1:48 (1/4“ = 1‘-0“)

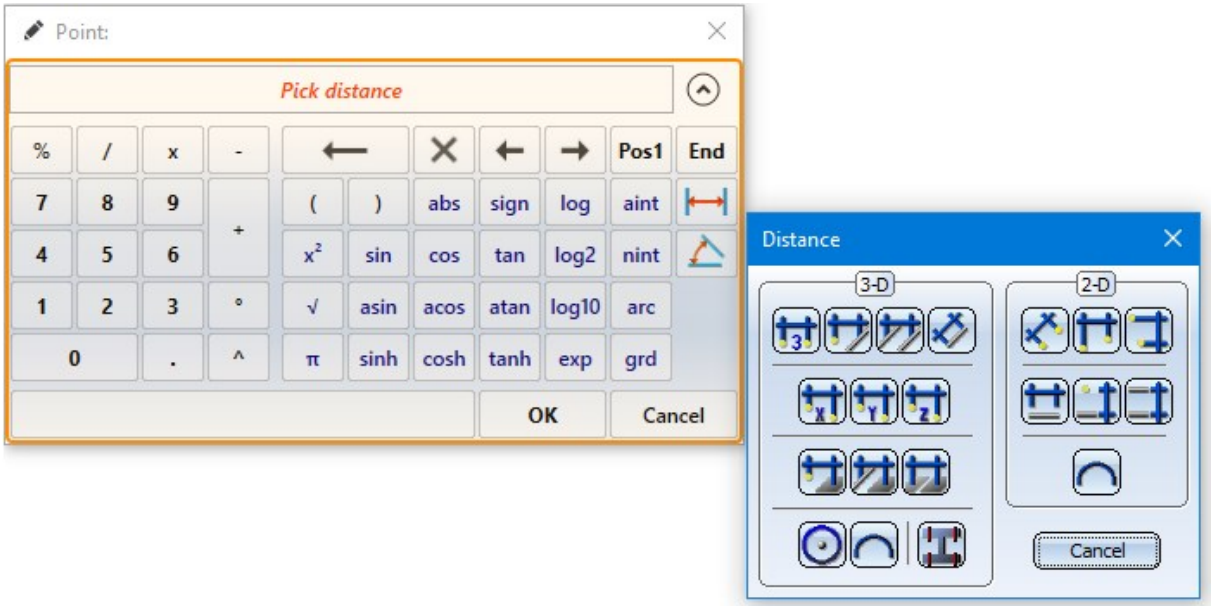
Notations 3 and 4 are suitable for US scale specifications.

Please note:

Further information on **Definition of Scales** can be found in the same named topic in the Basics Help.

New pocket calculator

The HiCAD pocket calculator has been redesigned.



In many cases HiCAD offers a default value which you can either apply or modify. You end the value input

- by clicking the left mouse button, or
- by pressing the ENTER key on your keyboard, or
- by clicking on the **OK** button.

Special keys:

The two buttons for Distances and Angles are used to transfer from existing construction objects.

Alternatively, you can call these functions in the context menu, which you activate in the calculator input field with the right mouse button. The functions of the Distance and Angle menu displayed here correspond to the functions with the same name in the **Information** menu.

Service Pack 1 2020 (V 2501)

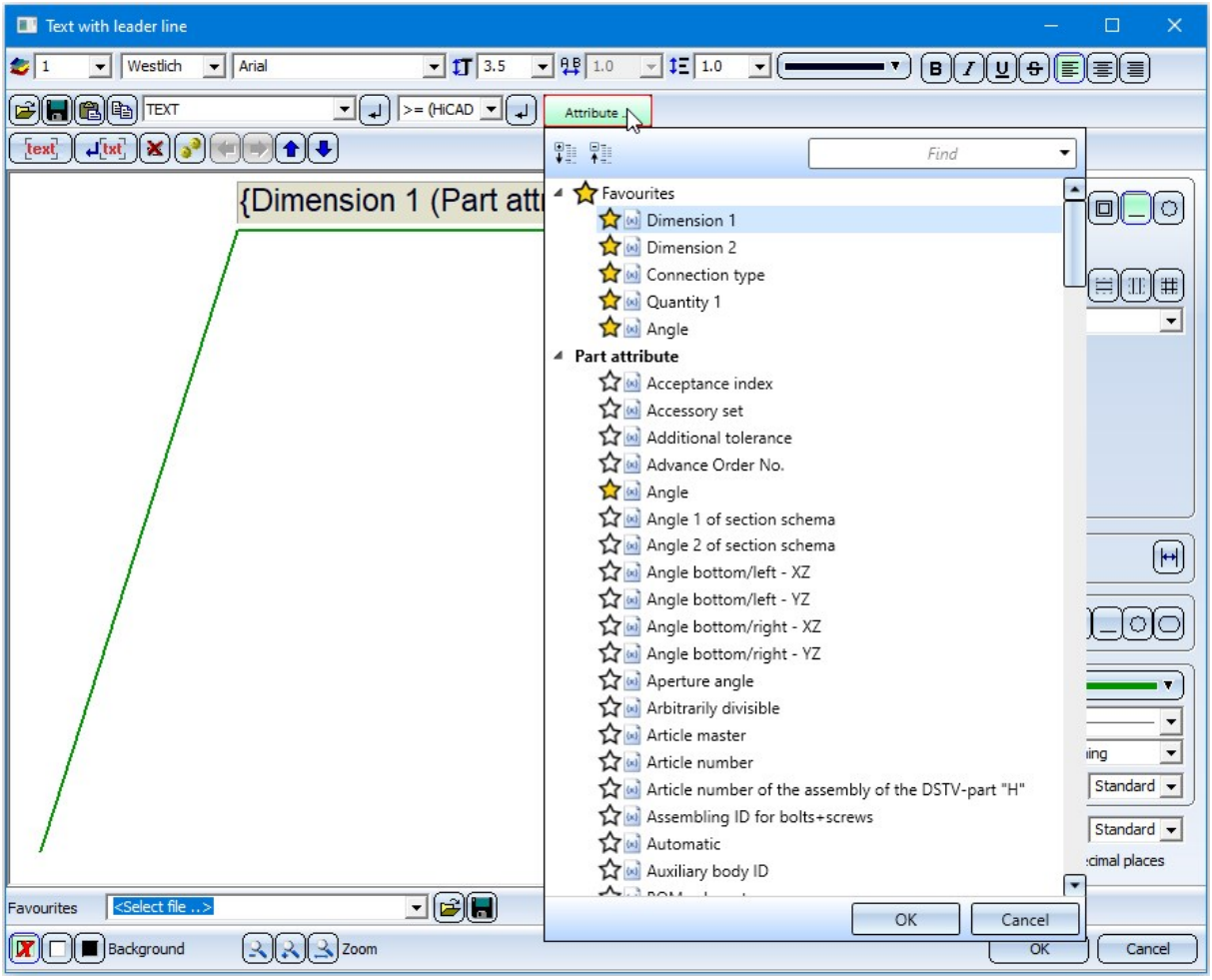
Point from point cloud

The point option **(PW) Point from point cloud** is now also available in the 2-D point options menu. The point option can also be called via the keyboard by pressing the C key.

Mark attributes in annotations as Favourites

When selecting attributes for annotations, you can mark frequently used attributes as favourites for faster access.

To do this, simply click on the symbol next to the attribute name. The symbol then turns yellow . Attributes marked in this way are listed in the selection window under **Favourites**.



To remove an attribute from the favorites list, simply click on the corresponding symbol - either directly in the Favourites list or in the attributes list.

Further point options of the Autopilot during sketching

When drawing with the 2-D geometry functions, the Autopilot - depending on the cursor position - now also displays the point options

(O) Online on edge through point and

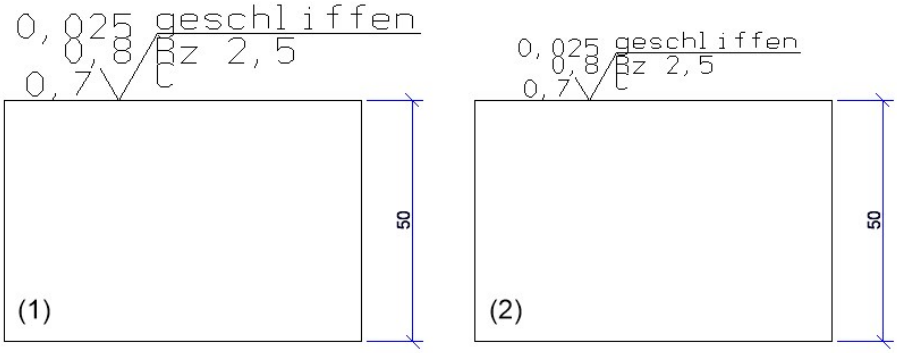
(F) Perpendicular base point

is displayed when the cursor is near a straight line.

If the cursor is near a circle, the **(F) Perpendicular base point** and **(T) Tangential point** options are now also displayed. For arcs and ellipses the behaviour is the same.

Standard-compliant information on surface finish

From SP1 onwards, all information on the surface finish is displayed in the same font size and line width. After a new installation of HiCAD, the default setting for the text height of the surface symbols in the file SURFSYM.INI 3.5 mm.



(1) before HiCAD 2020 SP1
(2) from HiCAD 2020 SP1 onwards

Insert pixel graphics

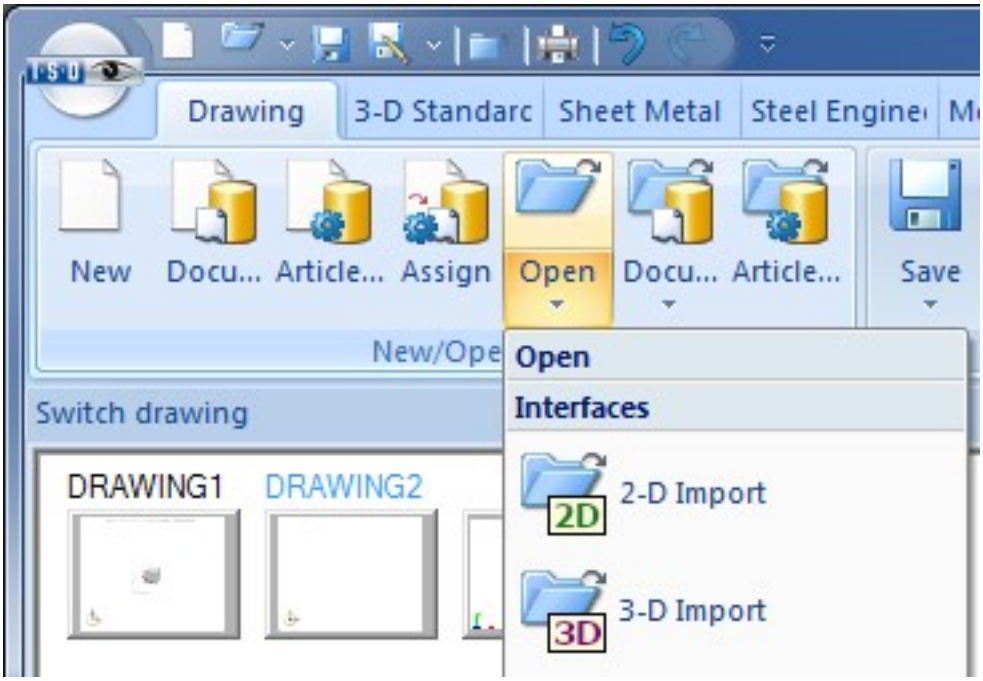
Pixel graphics - such as company logos - in the formats BMP, GIF, PCX, TIF and JPG as well as EMF files (Enhanced Meta Files) can be inserted either with the function **Drawing > Insert Part > Exp... > Bitmap** or with CTRL+V from the clipboard into the current HiCAD model drawing. They are assigned to the 2-D part structure and have the name _B_M_P__.

The internal behaviour of these two procedures was different until now. From HiCAD 2020 SP1 onwards the two procedures have been unified and stabilized. This means, in particular, that checking the 2-D part structure via **Information > Check > 2-D Geometry** no longer indicates errors if the model drawing contains pixel graphics.

Major Release 2020 (V 2500)

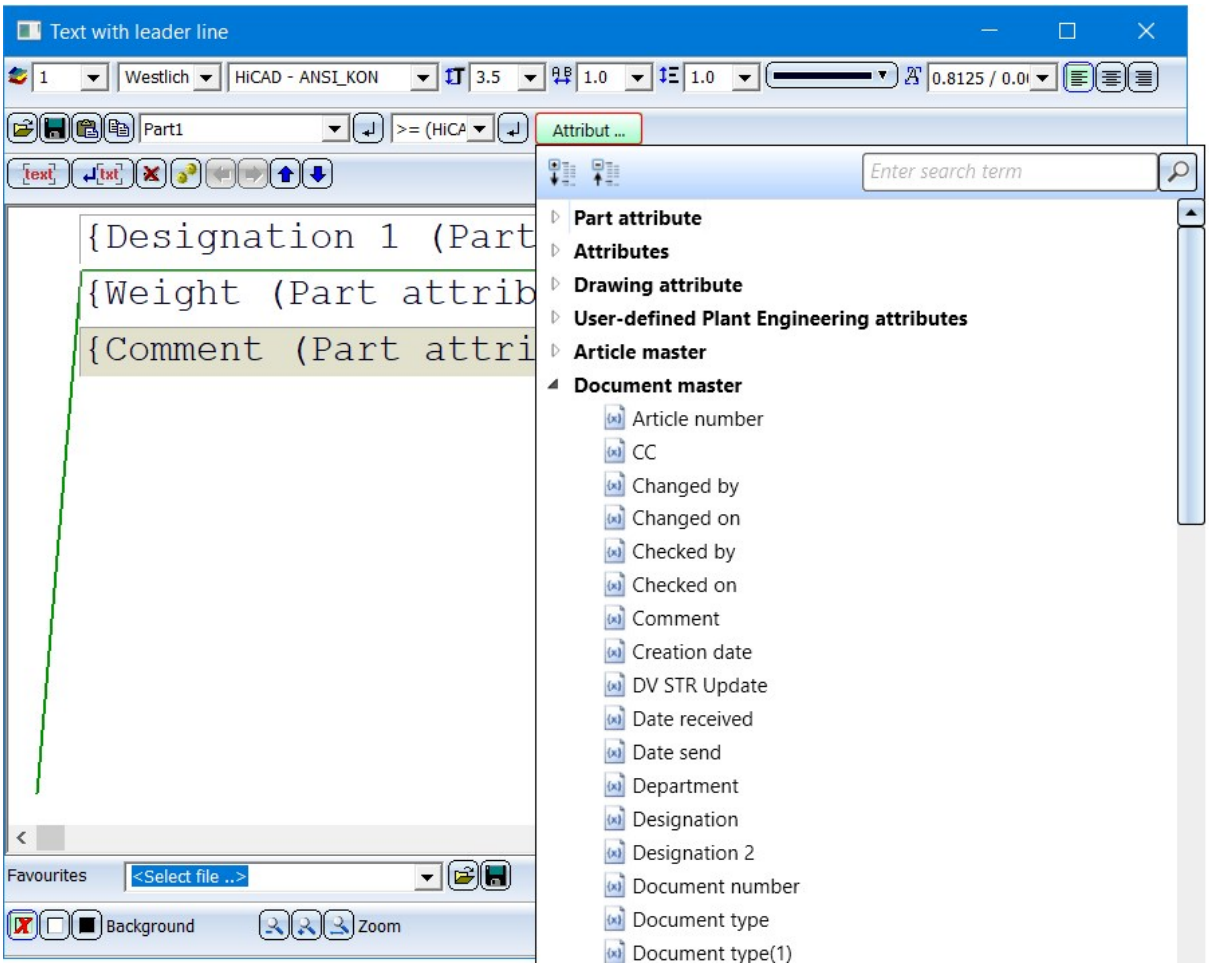
Import drawings

When importing 2-D/3-D drawings via interfaces, HiCAD 2020 will distinguish between 2-D and 3-D imports. For this purpose the previous function **Drawing > New/Open > STEP, IGES, ...** has been split into two new functions:



Changed dialogue for attribute selection

The **Attributes** button is now available for selecting attributes in the label editor. After clicking on this button the selection list for attributes is displayed, which is divided into different attribute groups.



2-D text settings

The default settings for fonts can now be set separately for 2-D and 3-D text in the Configuration Editor. The corresponding parameters can be found there at **Drawing > Annotations Text**.

3-D

Service Pack 2 2020 (V 2502)

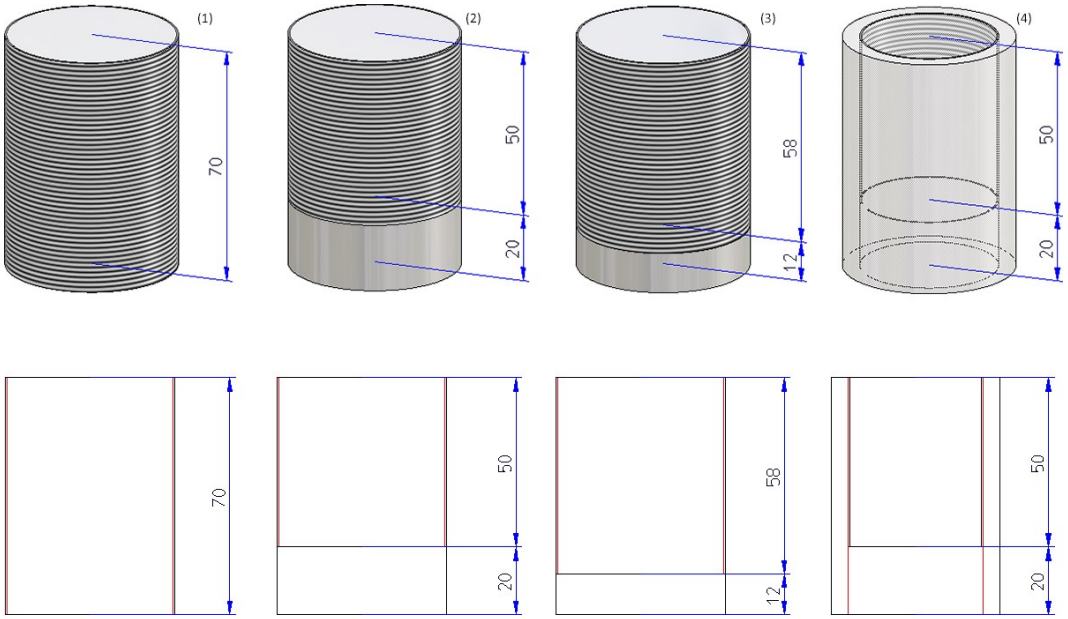
Model and process parts

Internal/external thread with runout

In the **Internal/external thread** function, thread runouts are now also possible.

The following options are available for runout length determination :

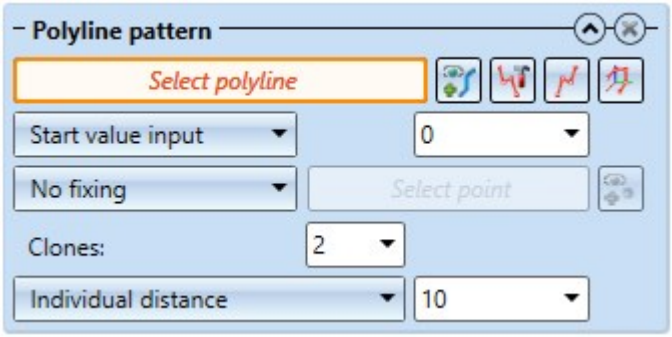
- **Value, as part of thread length**
  The runout has the length specified in the **Value** field. The specified thread length is shortened accordingly.
- **Value, in addition to thread length**
  This is only possible with the length options **Fixed length** and **To point**. The specified value determines the length of the runout, the thread length does not change. If the sum of thread length and runout length is greater than the possible full thread length, the runout is automatically shortened.
- **Pitch factor, as part of thread length**
  The specified factor is multiplied by the thread pitch. The result is the runout length. The specified thread length is shortened accordingly.
- **Pitch factor, in addition to thread length**
  This is only possible with the length options **Fixed length** and **To point**. The specified factor is multiplied by the thread pitch. The result is the runout length, the thread length does not change. If the sum of thread length and runout length is greater than the possible full thread length, the runout is automatically shortened.



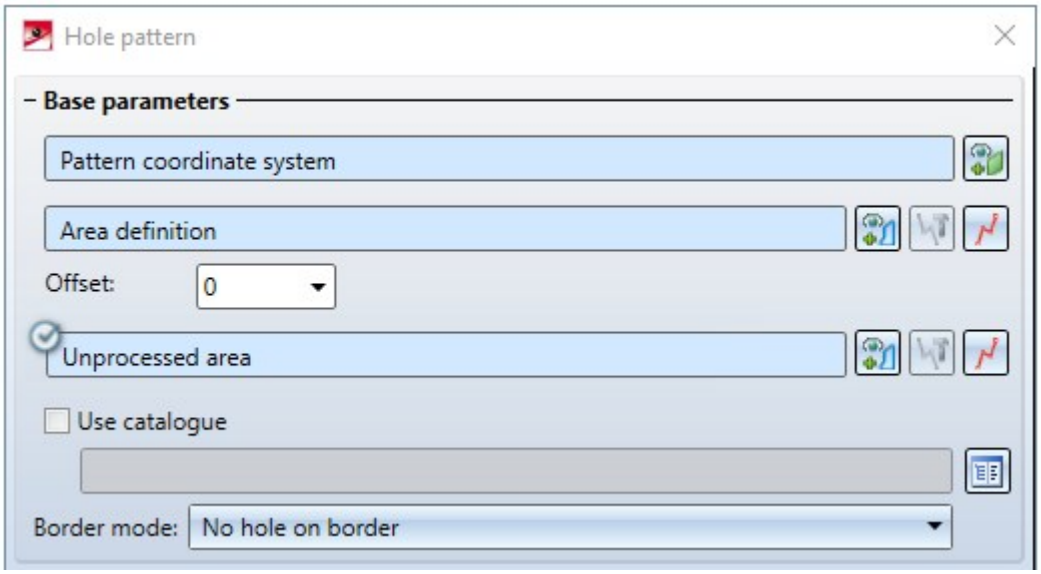
(1) Thread DIN 13 -M50x1.5, (2) Runout length 20, (3) Runout factor 8, (4) Internal thread DIN 40x1.5 with runout

Changed dialogue for Clone parts (parametric) function

For the **Clone > Param.** function, the sketch selection dialogue for linear patterns has changed slightly. The **Process sketch** , **New sketch in plane** and **New 3-D sketch** functions can now be activated directly in the top level of the dialogue.

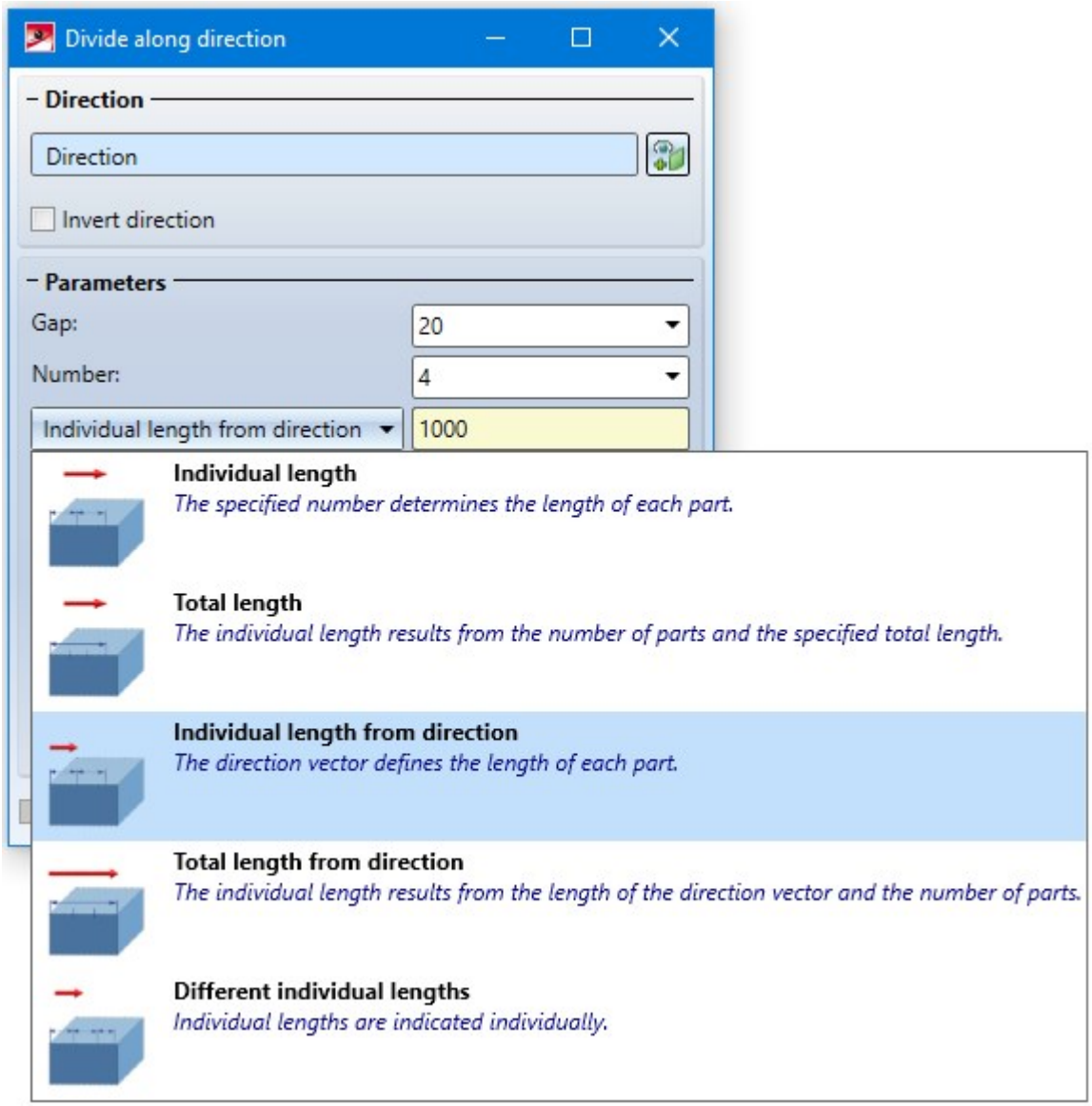


Similarly, the dialogue for the **Hole pattern** function has been modified for the area definition and the unprocessed area.



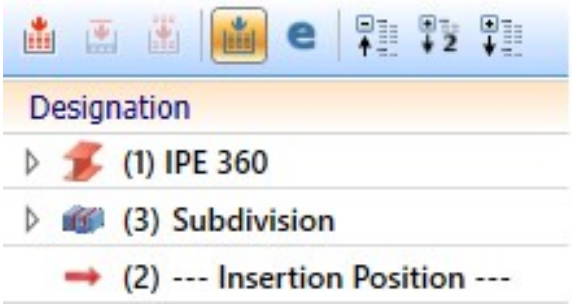
Divide along direction

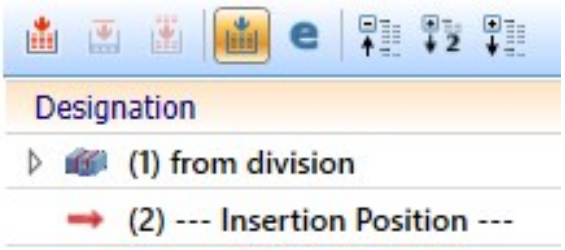
The new **Divide along direction** function at **3-D Standard > Process > Trim > ...** allows you to divide parts of the type Solid into several sections along one direction. Different division options are available, for example, the individual parts can have different lengths. In addition, a corresponding feature is created with this function so that the division can also be edited later.



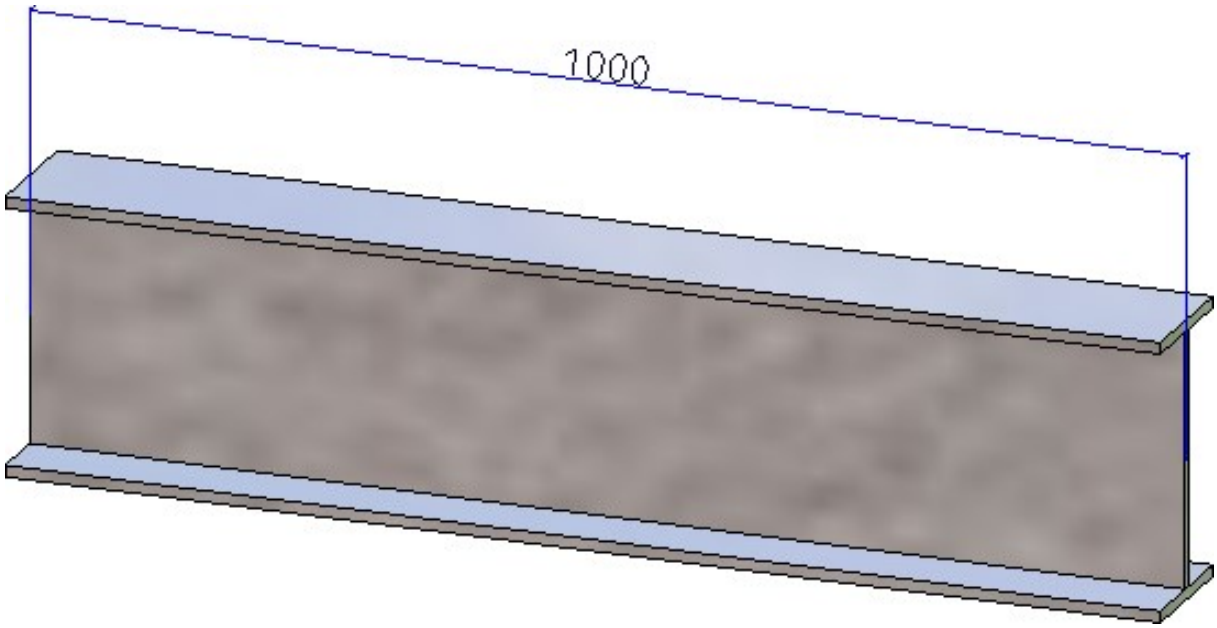
The direction, each division point and each gap are visualized in the drawing.

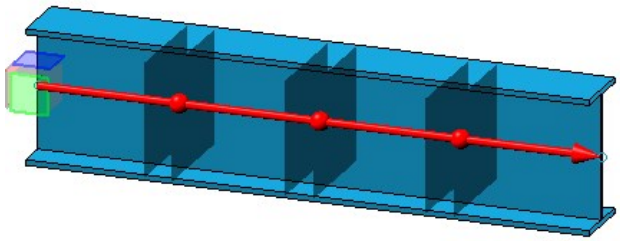
Each part is a separate part in the part structure - on the same hierarchy level as the initial part and with the same name. The initial part is assigned a feature with the name **Subdivision**, all other parts are assigned a feature with the name **from division**. The division can be edited by double-clicking on one of the features. All parts are adjusted accordingly.

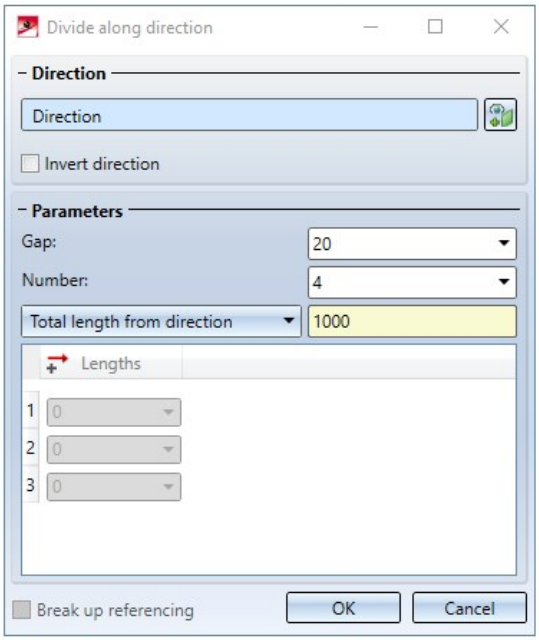


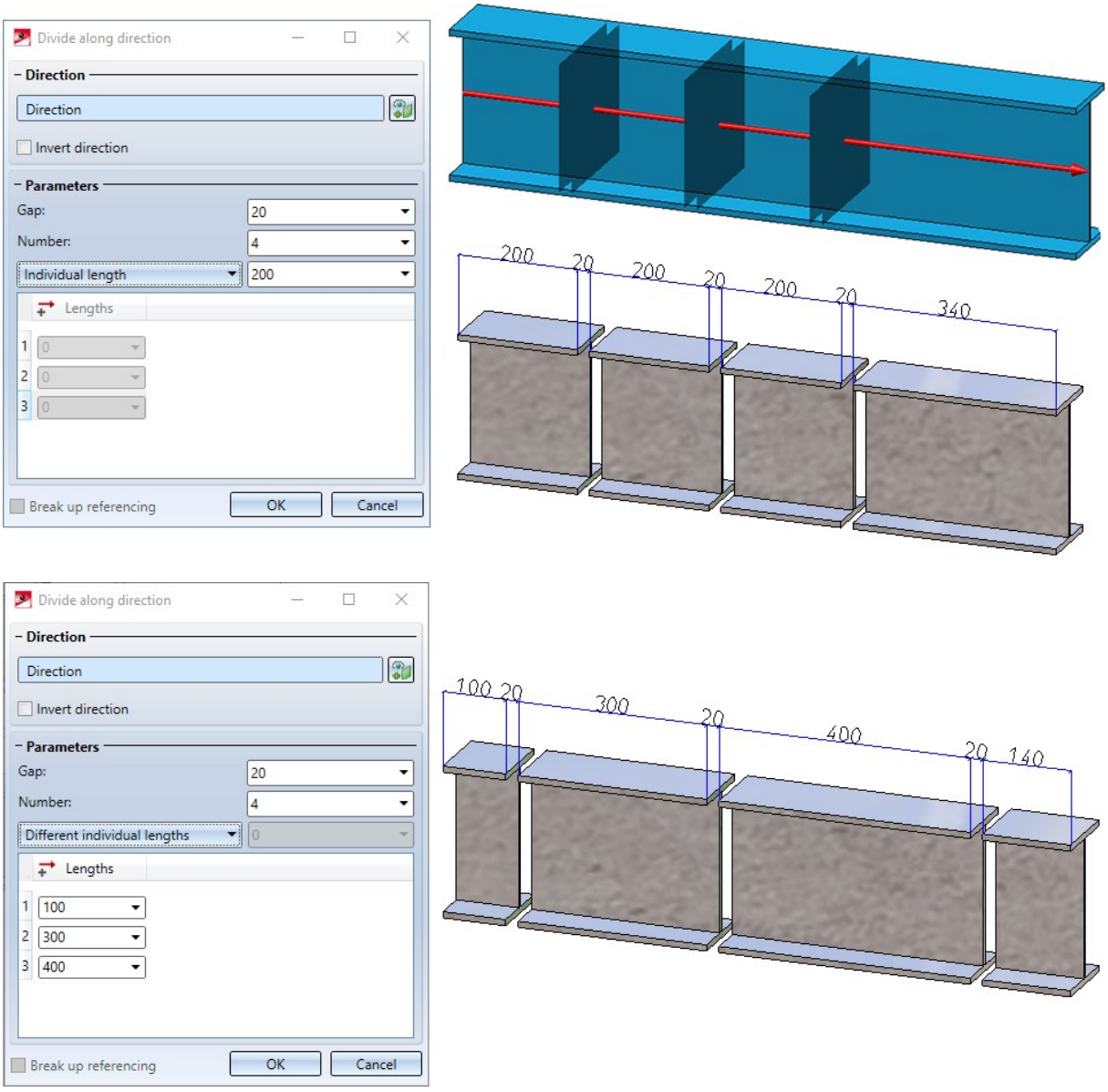


Example: Dividing a beam

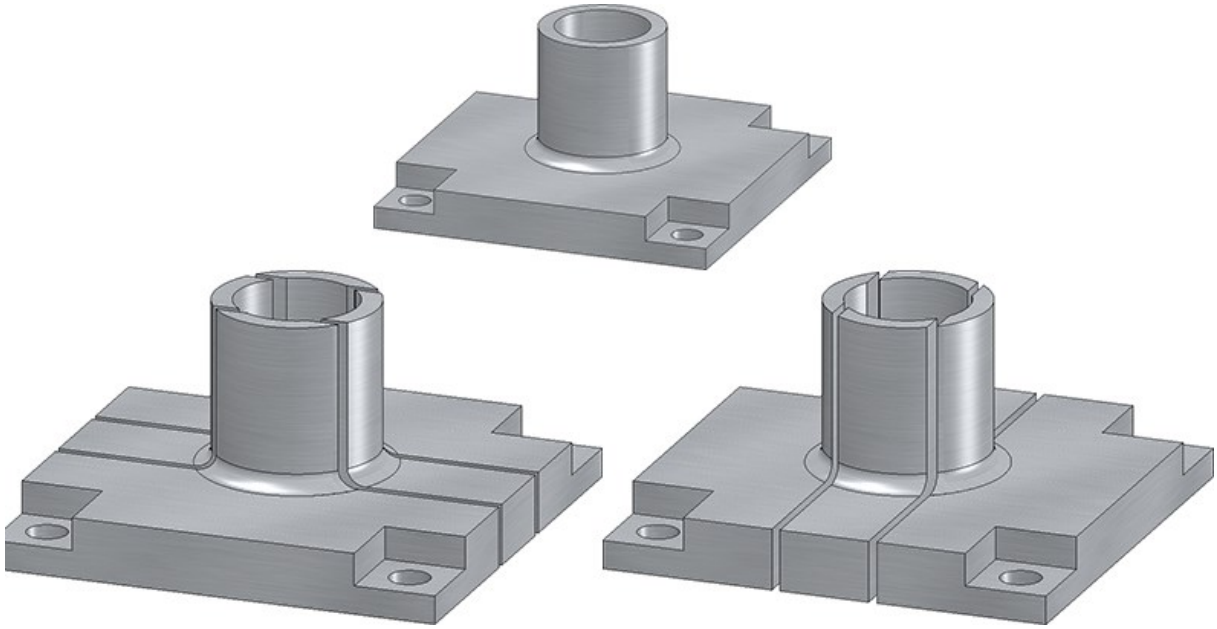








Example: Divided 3-D part



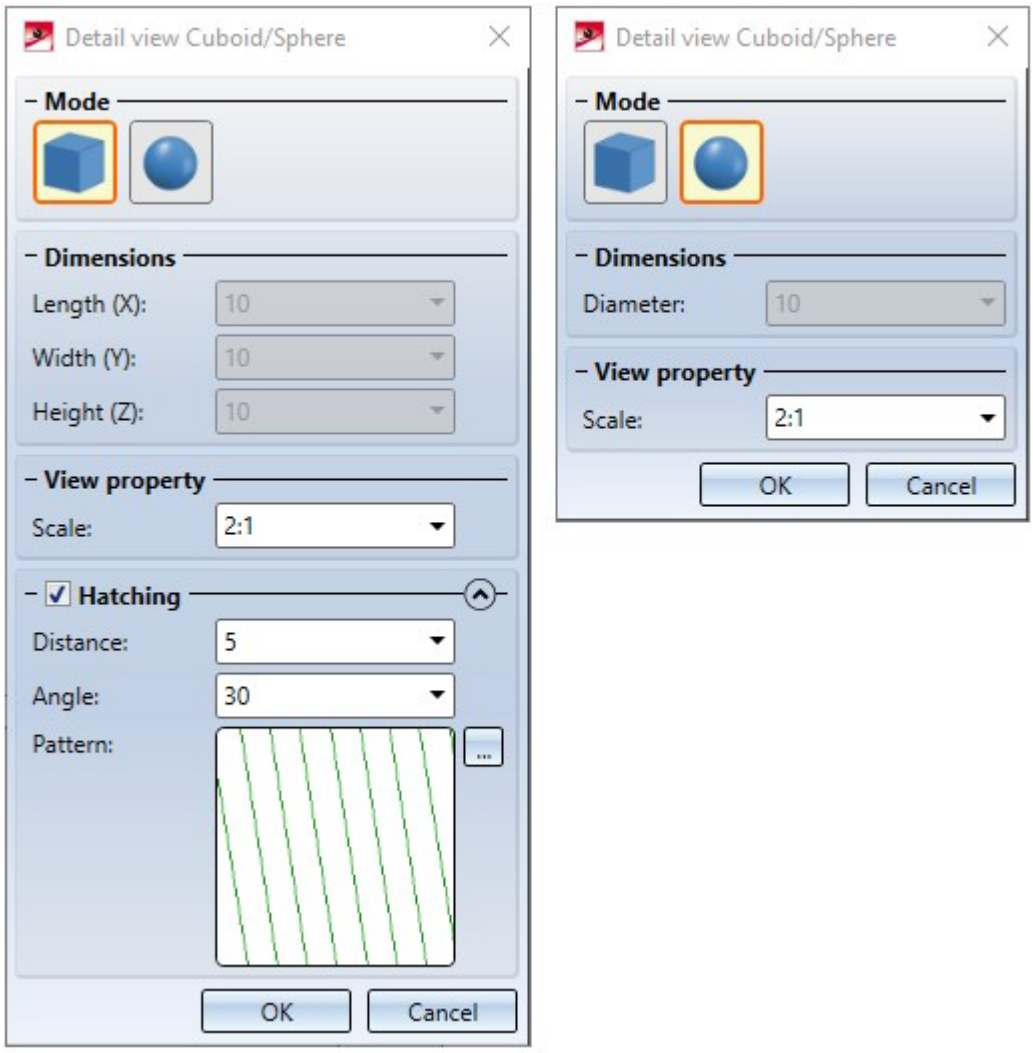
The function can also be found at **Steel Engineering > Lengthen > Divide**.

If you make changes to the initial part on features coming before the **Subdivision** feature, only the initial part will be recalculated. The division is performed at the same position relative to the part coordinate system. This means that neither the division points nor the parts are recalculated. If you want to adapt all sections to the changes, use the **Update** function in the feature context menu of the division.

Views

Detail view Cuboid/Sphere - New dialogue window and change options

The previous functions "Detail view Cuboid" and "Detail view Sphere" have been combined to form the new function **Detail view Cuboid/Sphere** . You can switch between both modes in the **Detail view** dialogue window..



The new dialogue is much more user-friendly. For example, the cuboid and sphere are now visualized in the preview and can be drawn dynamically with the cursor.

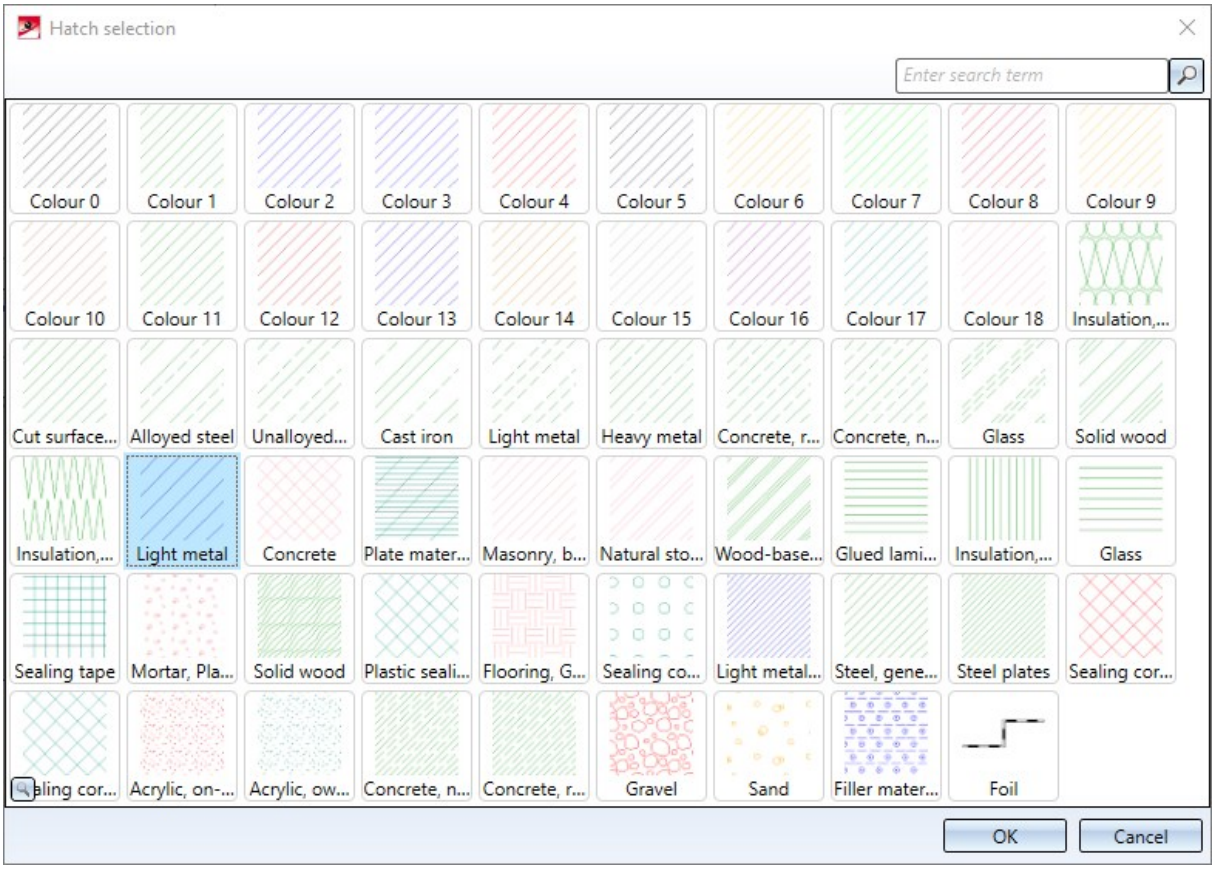
From SP2 on it is also possible to change a Detail view Cuboid / Sphere subsequently. To do this, select the **Change detail view** function in the **Views** Ribbon or in the context menu for views. Afterwards the corresponding dialogue window is displayed.

New dialogue window for hatching selection

The selection of hatch patterns in view functions has been revised. This affects the following functions:

- New sectional view
- Change sectional view
- Create detail view
- Change detail view
- New cut-out
- Change cut-out
- Hatching in sectional view and cut-out

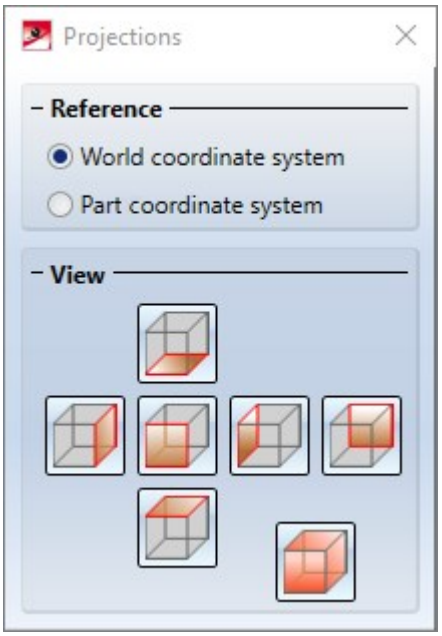
The **Hatch selection** dialogue is now displayed for selecting the hatch pattern. If you point the cursor on one of the patterns, more information about the hatch pattern is displayed. By clicking on the magnifying glass symbol in the lower left corner you can also zoom the display of the patterns.



If the **Cut surface hatching** parameter is set to **According to material**, the hatching selection in the dialogues has no effect.

New dialogue window when creating views

The dialogue for selecting the projection when creating a new view has been replaced in SP2.



This concerns the following functions:

Views > New > Standard

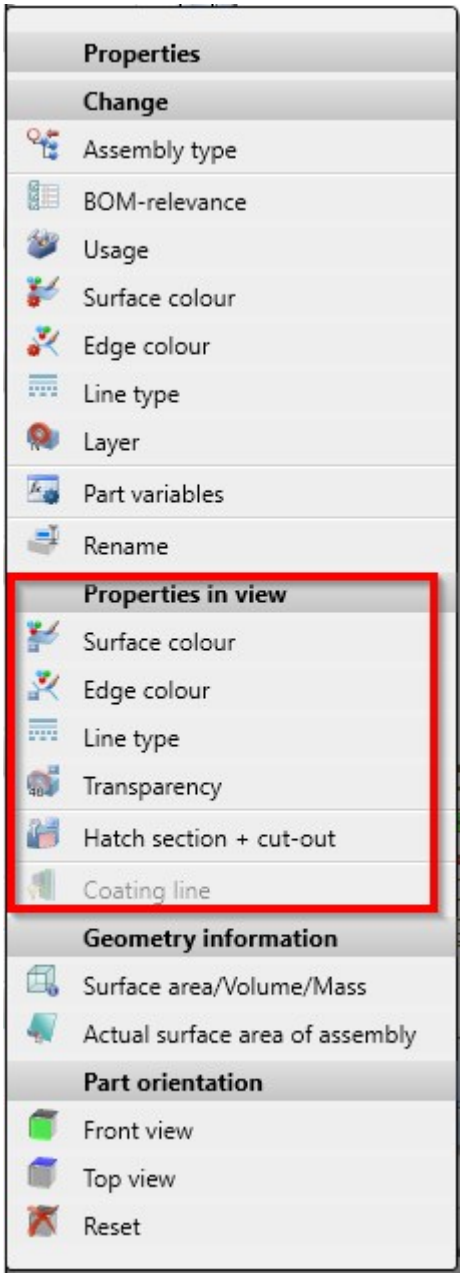
Views > New > List

Views > New > List > List from ref. view

Views > New > List > Active part list

View properties of assemblies in context menu

As for other 3-D parts, the **Properties in view** area is now also available for assemblies in the context menu for assemblies beneath **Properties**.



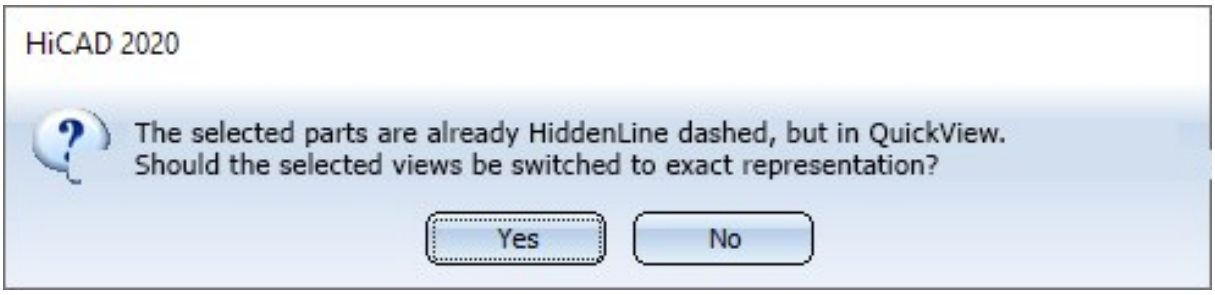
These functions allow you to define how the assembly is displayed in the active view.

Change representation in multiple views

If several views are selected in the ICN, the representation of all selected views can be changed in one step.

Note the following:

1. The progress bar is view-comprehensive. This also applies to aborting hidden line calculations with the ESC key. Views that have not yet been calculated are then displayed in QuickView.
2. If the display function is called with the CTRL key pressed, the views are always switched to exact representation. This also applies when the automatic QuickView is active.
3. If several views are selected, all of which already have the desired display, but are in QuickView, HiCAD asks whether the views should be switched to exact representation.



Hide parts in multiple views

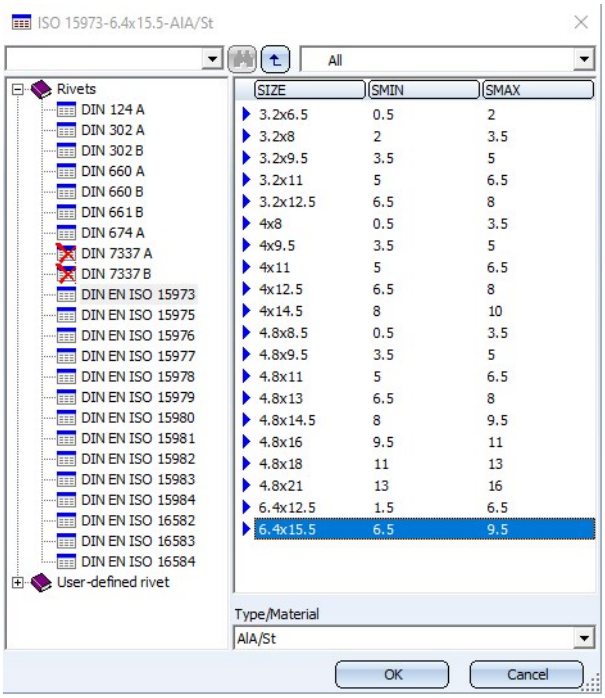
Parts can now be hidden in all currently selected views in one step. To do this, you first select all required views in the ICN. Then select the parts to be hidden in the ICN or in model drawing and choose **Part list, Hide in active view** from the context menu (right-click). The parts are then hidden in all views that are selected in the ICN.

Standard parts / Standard processings

Grip lengths for rivet insertion

When inserting rivets, the grip length range of a sub-type is now also displayed. For this purpose, the tables for rivets in the catalogue at **Fasteners > Rivets** and at **Factory standards > User-defined fasteners > User-defined rivets** have been extended by the minimum (SMIN) and maximum (SMAX) grip length.

When rivets are installed, these values are displayed in the selection window. If a sub-type is selected, the dialogue window **Grip length** is preset with the value SMAX of the selected sub-type. This value can be changed, but must not be less than SMIN and not greater than SMAX.



For update installations the grip length will not be displayed.

Change standard processings

Standard processings, which you create with the functions at **3-D Standard > Standard Processings**, can also be edited from SP2 onwards by double-clicking on the corresponding entry in the feature log. This applies to all standard processings for which a catalogue selection is possible as well as through holes and letterings.

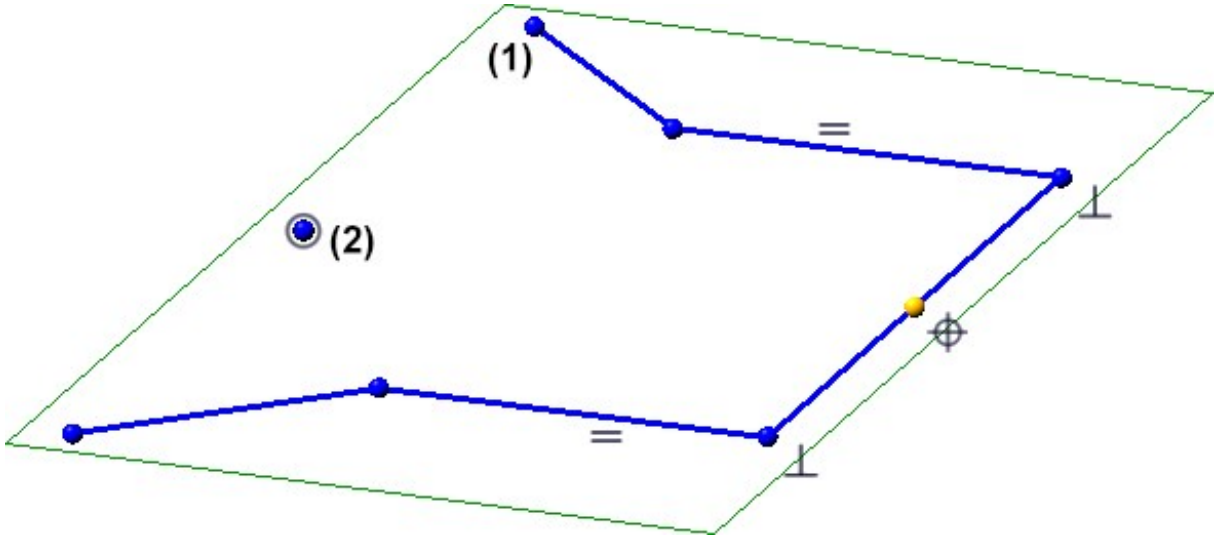
Sketches

Enhanced HiCAD VI / AV module

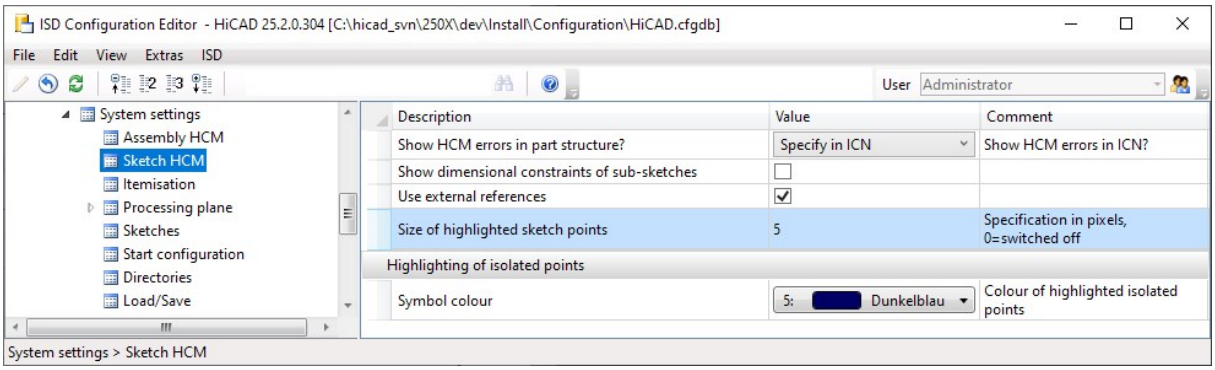
With the HiCAD VI / AV viewer solution, HiCAD constructions and drawings from third-party systems can be visualised. Furthermore, the module supports the plot generation/output and data export. In order to support the creation and editing of sketches for sectional and detailed views, the corresponding functions are now available on the **Sketch** Ribbon tab of the module.

Highlighting isolated points

If the **Enter constraints** checkbox is active in the HCM settings for sketches or if a sketch is parameterized manually, the sketch points are highlighted in colour. As of SP2, this also applies to isolated points in a sketch. Start and end points of sketch lines are marked by a filled circle (1), isolated points of the sketch by a filled circle with a ring (2). This applies to planar sketches and 3-D sketches.



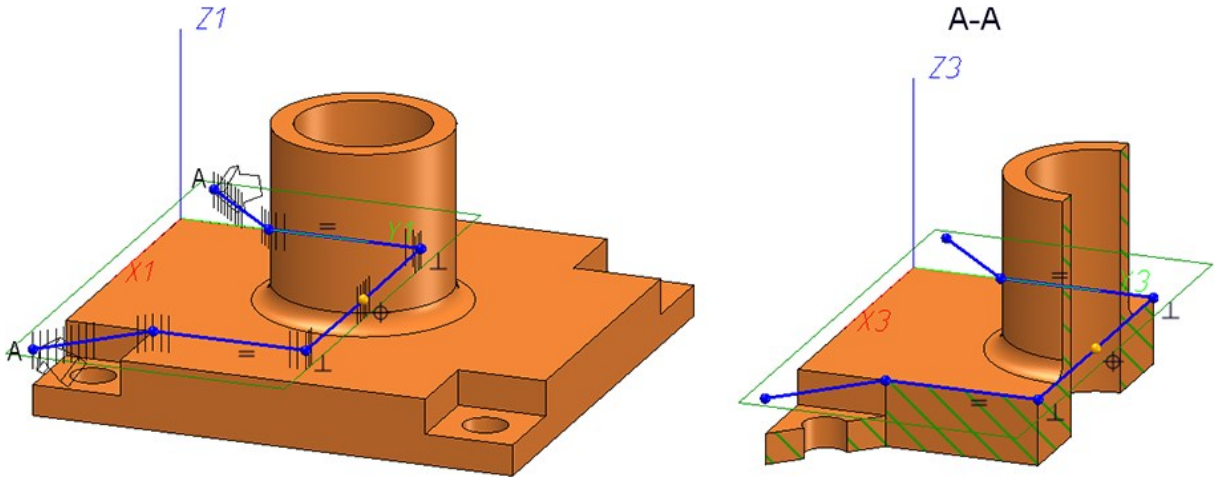
You define the size and colour of the highlighting in Configuration Editor at **System settings > Sketch HCM**.



If sketch points are not to be highlighted, set the parameter **Size of highlighted sketch points** to 0.

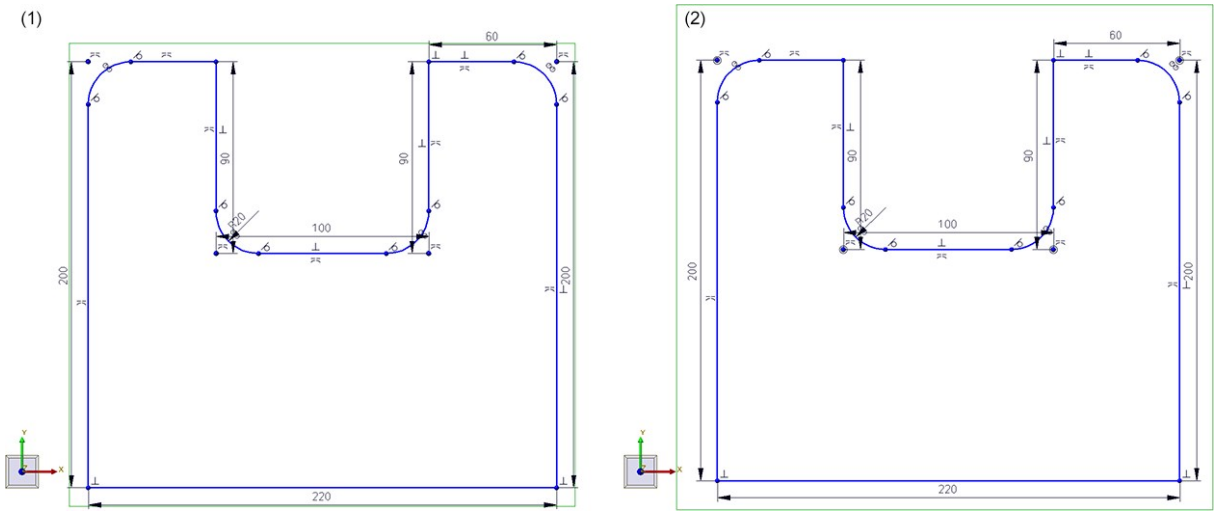
Parameterized sketches in sectional views

If a parameterized sketch with the purpose **Create/Edit** is active in a sectional view, the line end points of the sketch and the degrees of freedom are also visualized there.



Size of sketch plane

HiCAD places a green frame around the geometry of a sketch, representing the plane of the sketch and completely enclosing the geometry of the sketch. From HiCAD 2020 SP2 onwards, HCM positional and dimensional constraints of the sketch are also taken into account.



(1) before HiCAD 2020 SP2, (2) as of HiCAD 2020 SP2

3-D sketching tool - Changed arc creation

When inserting arcs, the arc radius and the opening angle are displayed at the cursor from SP2 onwards. If you want to use a specific radius, press the space bar or select the **Value input** function and enter the arc radius. You can determine the end point of the arc either by moving the cursor or with a point option.

First select the radius of the arc. To do this, drag the arc so that the desired radius is displayed and drop the cursor. The direction in which the arc is drawn (tangential or non-tangential) depends on the click point when the function is selected.

If you want to use a specific radius, press the space bar or select the function **Value input** and enter the arc radius.

You determine the end point of the arc either by placing the cursor (the opening angle is displayed) or with a point option.

If the last determined point is selected again, the drawing of an arc is automatically started.

If you specify a direction, the arc is drawn in the plane that is spanned by the specified direction and the tangent of the last edge at the connecting point.

Extended point options of the Autopilot

The point options

- Tangential point (T)
- Online through point (O)
- Theoretical intersection point (S2)
- Perpendicular base point (F)

offered by the Autopilot are also available for plane sketches and 3-D sketches. The option F is only displayed if several points have to be selected within a function, for example when selecting a direction. In addition, the following restrictions must be observed depending on the situation:

Planar sketches

	T	F	O	S2
Sketch > Draw > Rectangle > Free	✗	✗	✓	✓
Sketch > Draw > Ellipse	✗	✗	✓	✓
Sketch > Draw > Ellipse > Partial ellipse	✗	✗	✓	✓

3-D sketches

	T	F	O	S2
Sketch > New > Point	✗	✗	✓	✓
Sketch > Draw > Freehand	✗	✗	✓	✓
Sketch > Draw > Text	✗	✗	✓	✓

Direction/Axis

When choosing a direction/axis over two points, T, F, O and S2 are offered at the second point, but not at the first point. This affects the following Sketch functions:

- Sketch > Draw > Freehand > Transition curve
- Sketch > Derive > Take O... > Directed
- Sketch > Transform > Move
- Sketch > Transform > Rotate
- Sketch > Transform > Rotate > Move+Rotate
- Sketch > Transform > Mirror
- Sketch > Transform > Scale
- Sketch > Clone > Move
- Sketch > Clone > Rotate
- Sketch > Clone > Rotate > Move+Rotate
- Sketch > Clone > Mirror
- Sketch > Clone > Scale

C-edge HCM

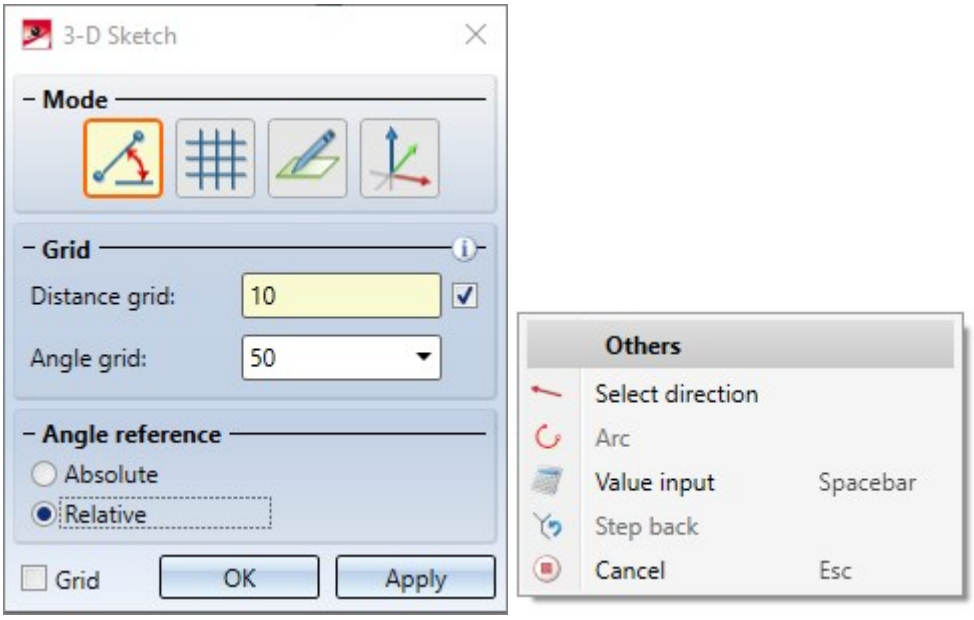
Here S and S2 are offered for all functions, except for dragging.

Service Pack 1 2020 (V 2501)

Sketches

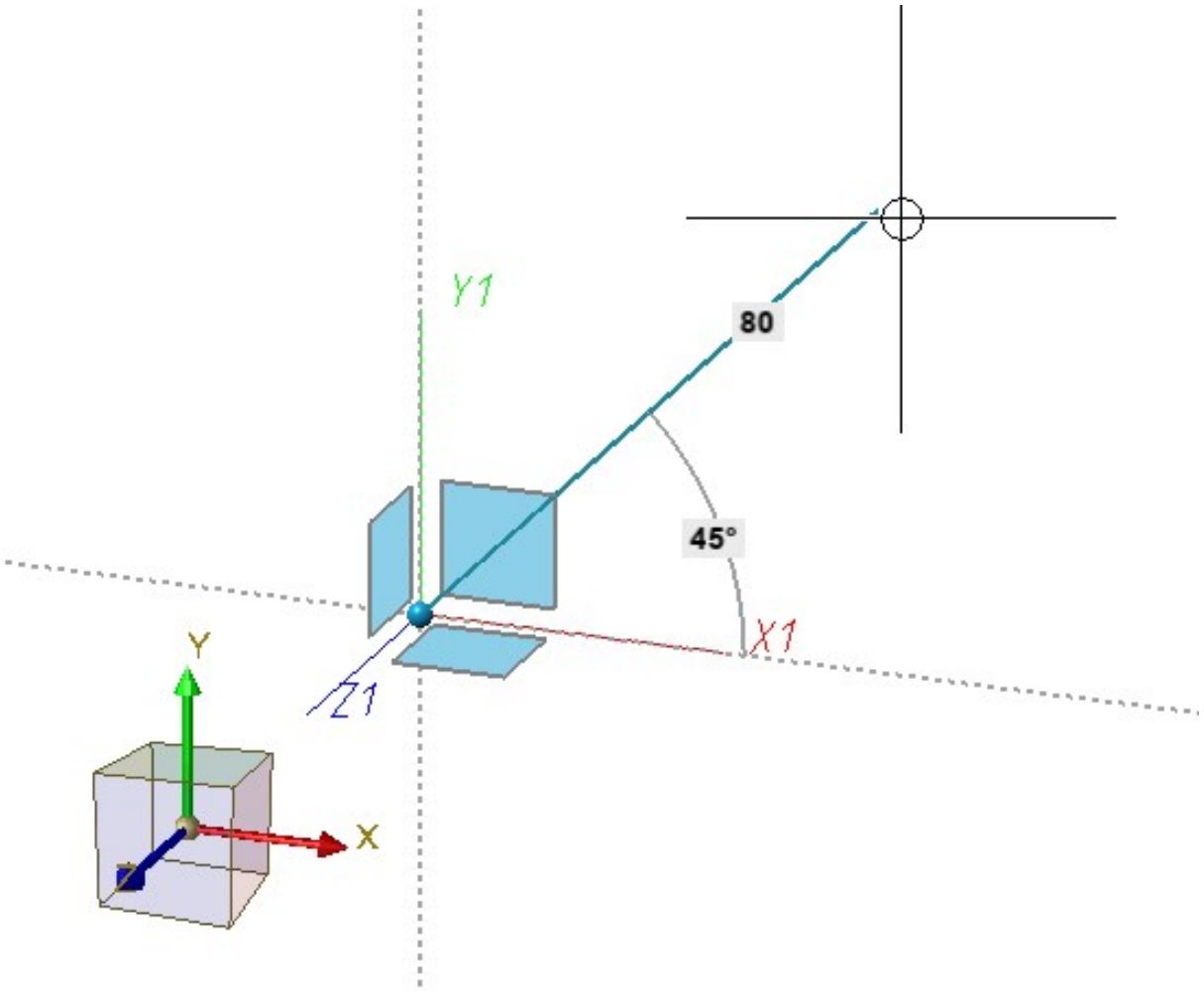
Enhanced 3-D sketching tool

With SP1 the 3-D sketching tool has been significantly enhanced. The dialogue window and the context menu have changed as well:

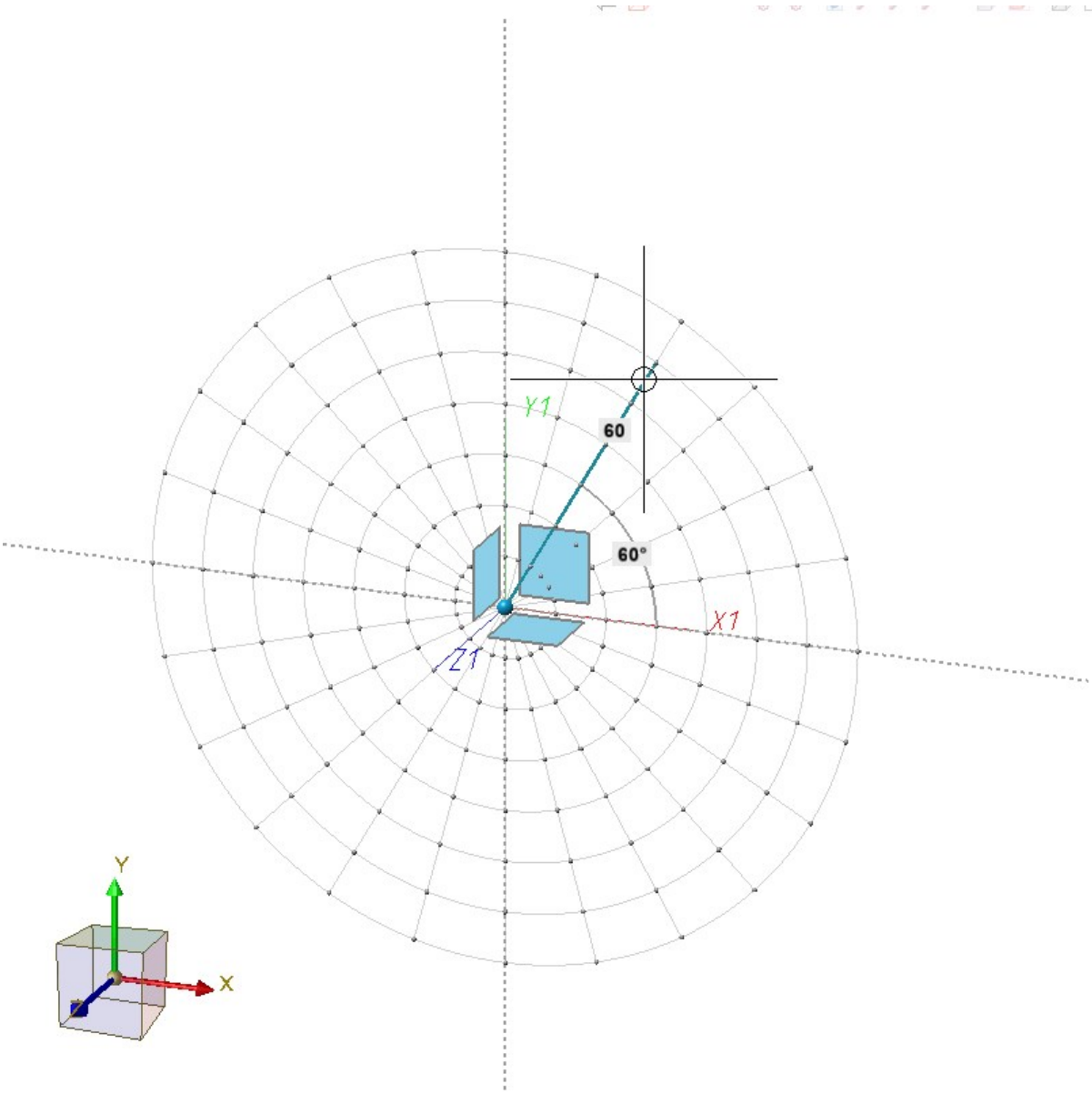


- The "Direction+ Distance" sketching mode is no longer available. Instead, you now use the **Select direction** function in the context menu to change the direction.
- If you want to enter a distance, length or radius using the keyboard, you can now also use the **Value input** function of the context menu instead of the space bar.
- If you activate the checkbox next to the **Distance grid** field in the dialogue window, the value for the distance grid is automatically adjusted when you zoom the drawing. This is the ISD default setting.

- The display of distances and angles during sketching has been improved.



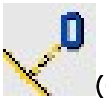
- By activating the **Grid** checkbox, grid lines and points can be displayed as tools for determining angles and distances, e.g.



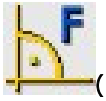
- If a polyline is continued, it is also possible to draw an arc directly.
- When selecting the starting point, the autopilot also offers the point option **O Online on edge** and when determining the following points, both **O** and **F Perpendicular base point**. This depends on the cursor position.
- Connecting points of Plant Engineering parts are given special consideration during sketching. If the first point is a connecting point, then this point is used to define a grid direction. This means that the following lines are drawn in this direction.

More point options of the Autopilot during sketching

For both planar sketches and 3-D sketches, point options



(O) Online to edge through point as well as

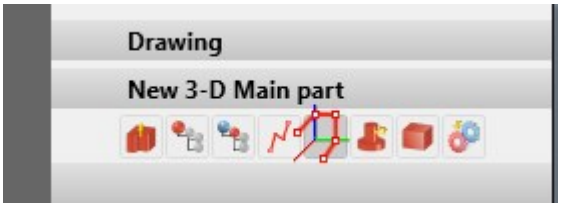


(F) Perpendicular base point

are now also displayed by the Autopilot - depending on the cursor position - when the cursor is close to a straight line.

Others

- The function **3-D Sketch** is now also available in the context menu for drawings.



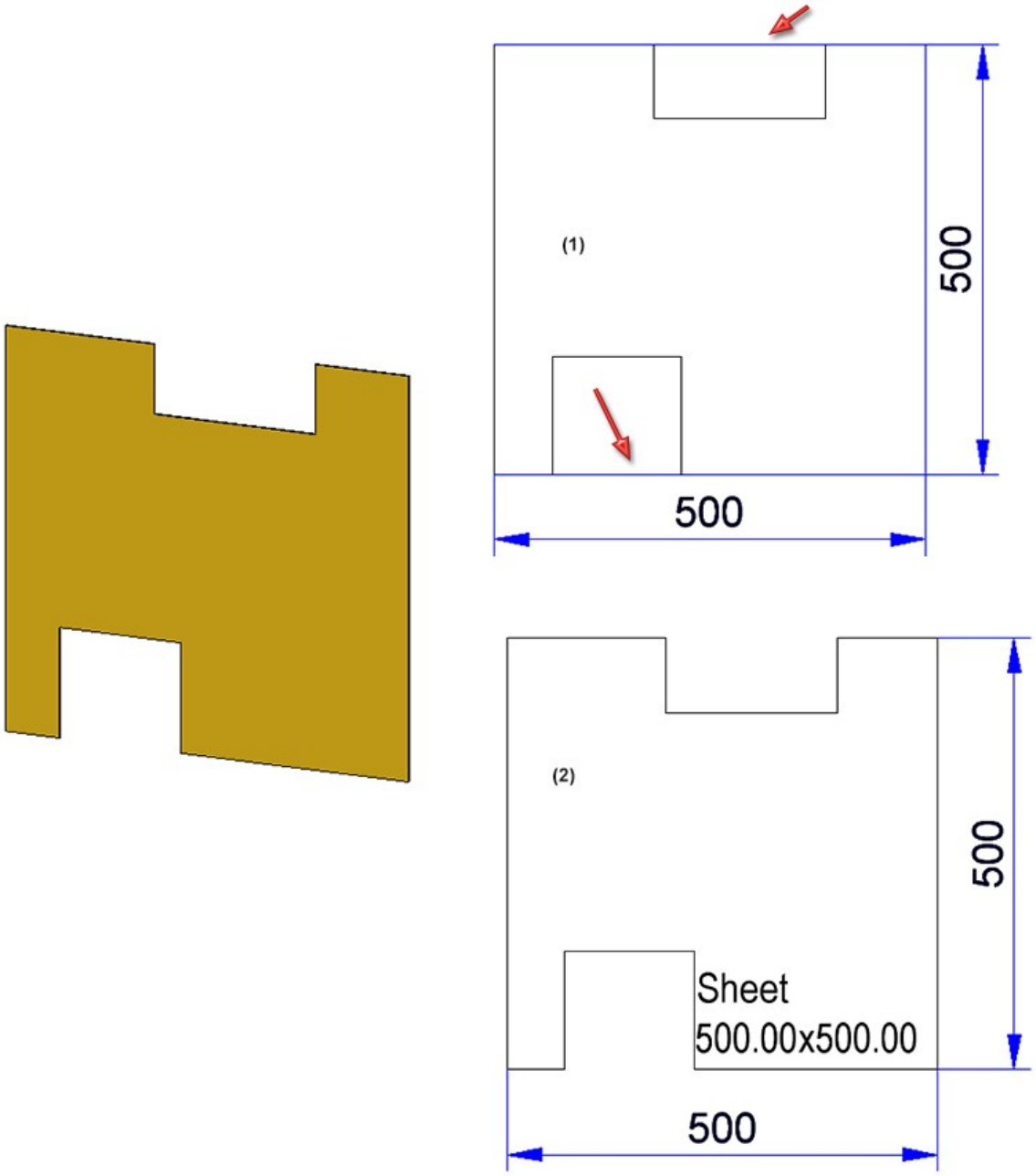
- If the automatic assignment of constraints is activated in the Sketch HCM settings (**Sketch > HCM > Tools > Settings**), the start and end points of the sketch lines and arcs are represented by small blue points from SP1 onwards. The size of the points can be set in the Configuration Editor at **System settings > Sketch HCM** with the parameter **Size of highlighted sketch points**.

Dimensionings and Annotations

Automatic dimensioning - Base point optimisation

For automatically generated dimensions, from HiCAD 2020 SP1 onwards, the projection line (for unshortened projection lines) is only drawn to the nearest point on the object to be dimensioned, i.e. to the next point on the contour. This applies to automatically dimensioned developed sheets.

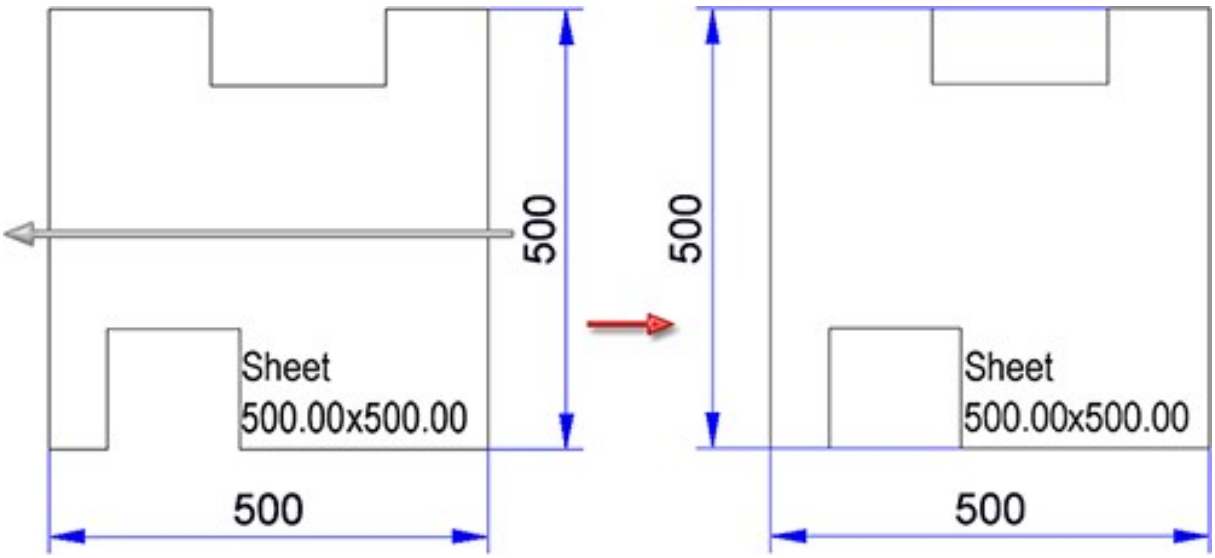
The following figure shows an folded sheet with material subtractions and the automatically dimensioned development of the sheet.



(1) before HiCAD 2020 SP1, (2) from HiCAD 2020 SP1 onwards

Please note:

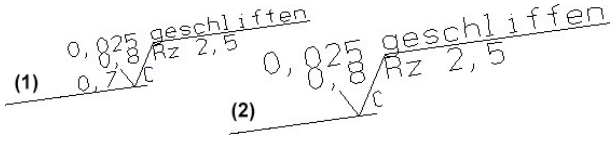
If an automatically generated dimension is changed manually, the optimization described above may not be possible in some cases. This is the case, for example, if you drag the right dimension completely to the other side in the example shown.



Standard compliant information on surface finish

From SP1 onwards, all information on the surface finish is displayed in the same font size and line width. After a new installation of HELiOS, the default setting for the text height of the surface symbols in the surfsym.ini file is 3.5 mm.

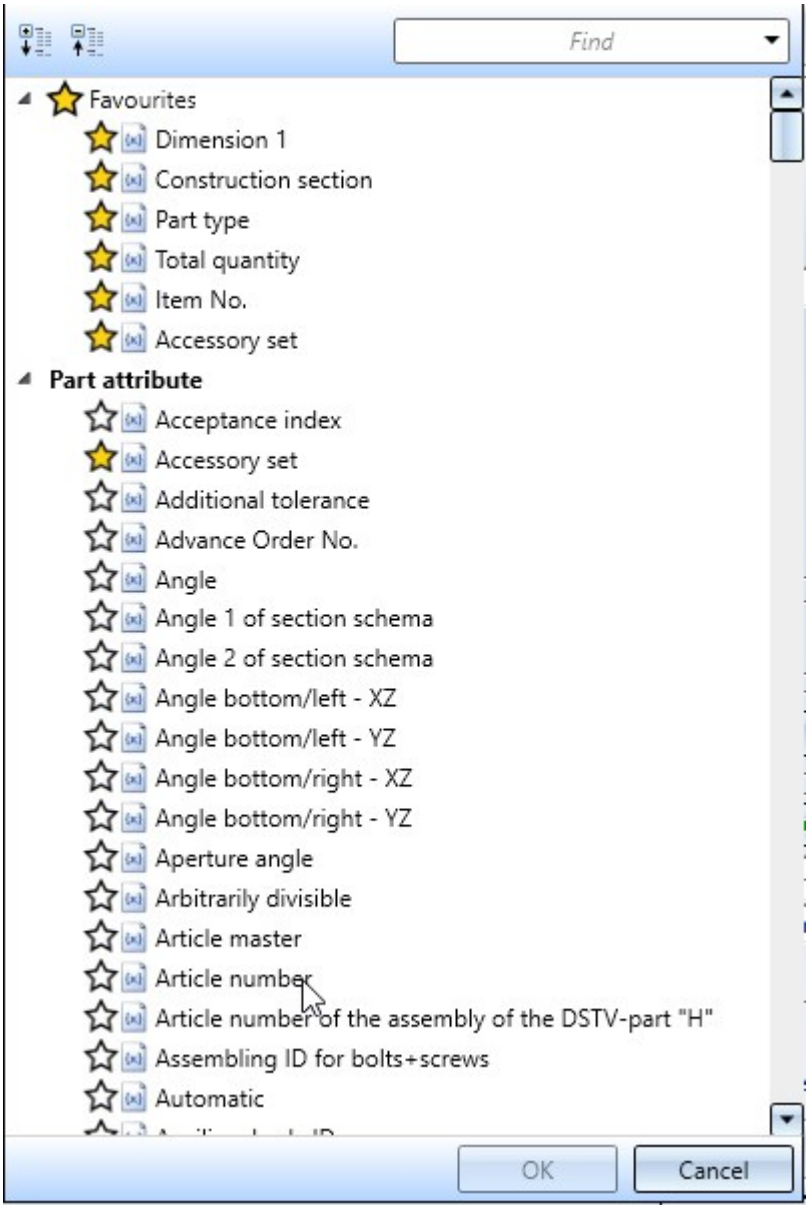
```
[A1_TEXT] =  ;0,025;0,05;0,1;0,2;0,4;0,8;1,6;3,2;6,3;12,5;25;50;N1;N2;N3;N4;N5;N6;N7;N8;N9;N10;N11;N12;
[A2_TEXT] =  ;0,025;0,05;0,1;0,2;0,4;0,8;1,6;3,2;6,3;12.5;25;50;N1;N2;N3;N4;N5;N6;N7;N8;N9;N10;N11;N12;
[B1_TEXT] =  ;
[B2_TEXT] =  ;%ts(SURFSYM_GRINDED);%ts(SURFSYM_RAW);%ts(SURFSYM_MILLED);%ts(SURFSYM_CLEANED);
[F1_TEXT] =  ;Rz ;Rz 0,4;Rz 1;Rz 2,5;Rz 4;Rz 6,3;Rz 10;Rz 16;Rz 25;Rz 40;Rz 63;Rz 100;Rz 160;
[F2_TEXT] =  ;
[E_TEXT]  =  ;0,1;0,2;0,3;0,4;0,5;0,6;0,7;0,8;0,9;1;1,5;2;

[FONT2D] = 1 0 0 1 -1 1 ; 3.5 0 0 0
[FONT3D] = 1 0 0 1 -1 1 ; 3.5 0 0 0

[HEIGHT_TABLE_H]    = 2.5;3.5;5;7;10.14;20;
[HEIGHT_TABLE_H1]   = 3.5;5;7;10;14;20;28;
[HEIGHT_TABLE_H2]   = 8;11;15;21;30;42;60;

[TREATTYP_ICON_PATH] = /makro3d/image/

[LINECOLOR2D] = -1
[LINECOLOR3D] = -1

[ELEM_REF_ERROR_COLOR] = 6


[SYMBOLCOLOR2D] = -1
[SYMBOLCOLOR3D] = -1
[HAS_ROOTSYMBOL] = 1

[DONT_REVERSE] = 0
```



(1) from HiCAD 2020 SP1 onwards, (2) before from HiCAD 2020 SP1

Mark attributes in annotations as favourites

When selecting attributes for annotations, you now have the option to mark frequently used attributes as favourites for a faster access. Simply click on the ☆ symbol next to the attribute name. The symbol then changes its colour to yellow ★. Attributes marked in this way are listed in the selection window under **Favourites**.



To remove an attribute from the Favourites list, simply click on the corresponding ★ symbol - either directly in the Favourites list or in the attributes list.

Views

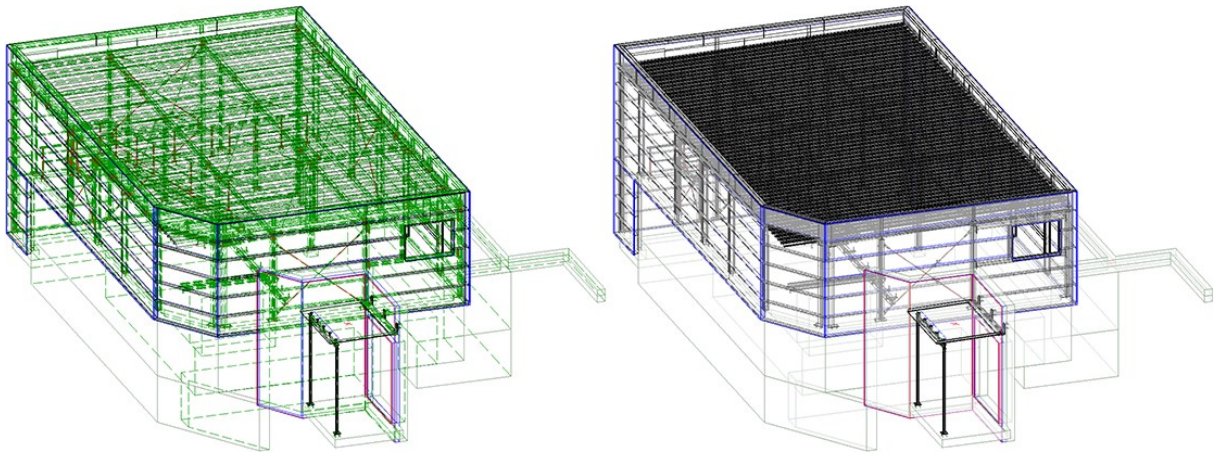
Automatic QuickView

For complex model drawings, especially when updating views in **HiddenLine** mode, waiting times may occur. In order to reduce these waiting times, HiCAD offers the possibility to temporarily use the corresponding **Quick HiddenLine** representation - the so-called QuickView - instead of the exact HiddenLine calculation for views with long calculation times. This considerably increases performance when editing and updating views. Situations in which this mode is very useful include

- the opening of model drawings with updating of referenced parts or
- switching from the Model view to a sheet area with many views.

The table below shows which QuickView belongs to which exact representation:

Exact representation	Associated QuickView
HiddenLine	Quick HiddenLine
HiddenLine dashed	Quick HiddenGrey
Shaded with HiddenLine	Shaded with edges
Glass model	Wireframe (not separately selectable)

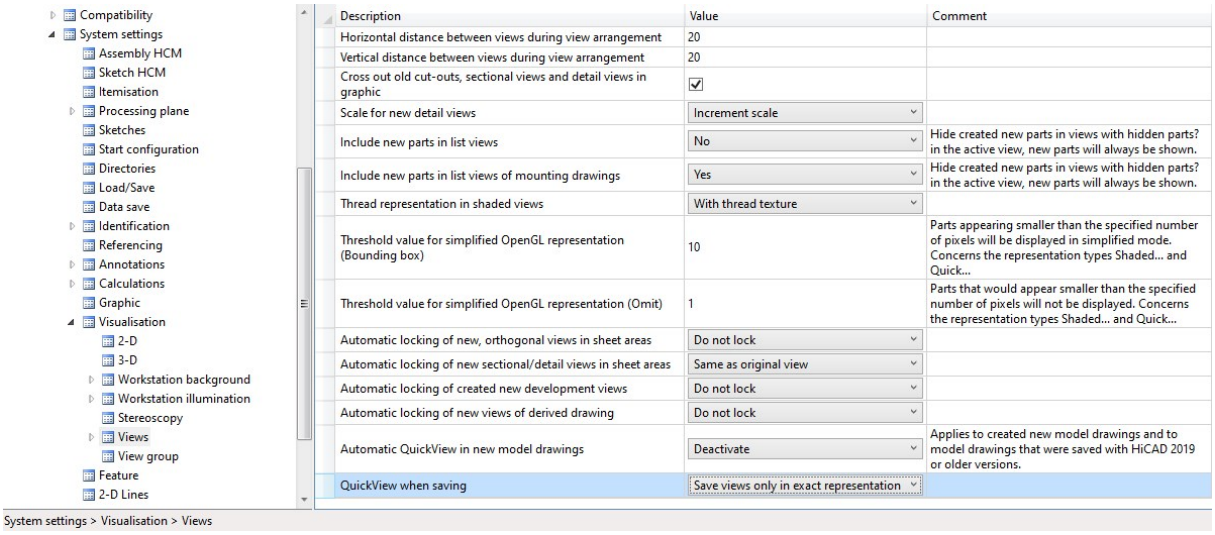


Left: Exact representation - HiddenLine dashed; Right: QuickView - Quick HiddenGrey (Image: Metallbau Wilhelmer Projekt GmbH, Kolbnitz, Austria)

The QuickView can be activated model drawing-dependent, i.e. you can switch the mode on or off depending on the requirements or size of the model drawing. This can be done either automatically or manually. In addition, you can define how the system should proceed when saving if the model drawing contains QuickViews.

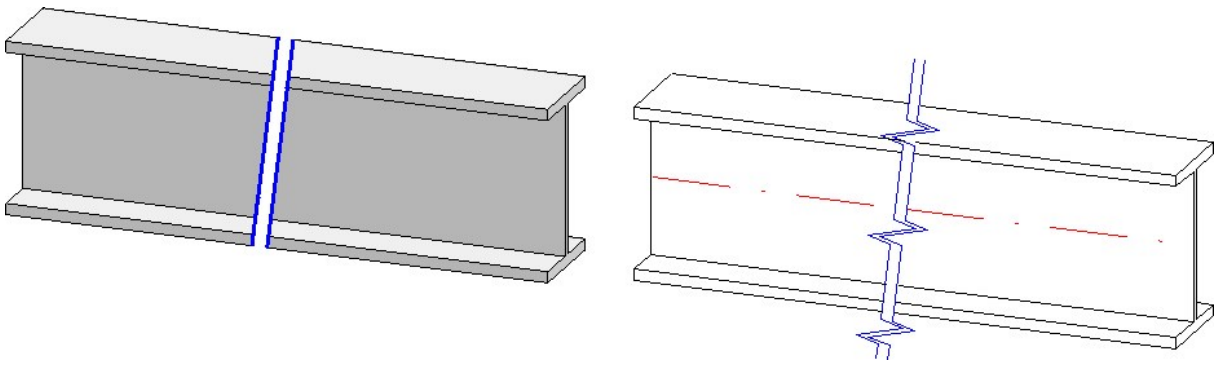
The following parameters are available in the Configuration Editor at **System settings > Visualization > Views**:

- **Automatic QuickView in new model drawings**
  This setting determines whether the automatic QuickView should become active for newly created model drawings and for model drawings created with HiCAD 2019 or earlier. If the setting **Activate** is selected, HiCAD automatically decides when the QuickView is required. The I default setting is **Deactivate**, that is, the QuickView must be activated manually if required.
- **QuickView when saving**
  Use this parameter to specify how to proceed when saving if the model drawing contains views in QuickView. The default setting is **Save views only in exact representation**.



Representation of shaded views with shortening

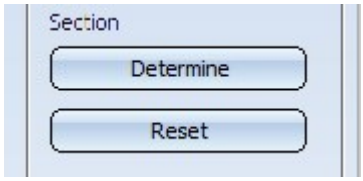
Since HiCAD 2020, it has also been possible to shorten views for shaded views (Representation type **Shaded with/without edges** or **Quick Hidden Line / Quick Hidden-Grey**). However, the shortening marks in shaded views have not been displayed clearly enough until now. As of SP1, these are now highlighted in the colour selected in the shortening parameters for the break lines.



Please note in this context that only the colour is taken into account for shortening shaded views.

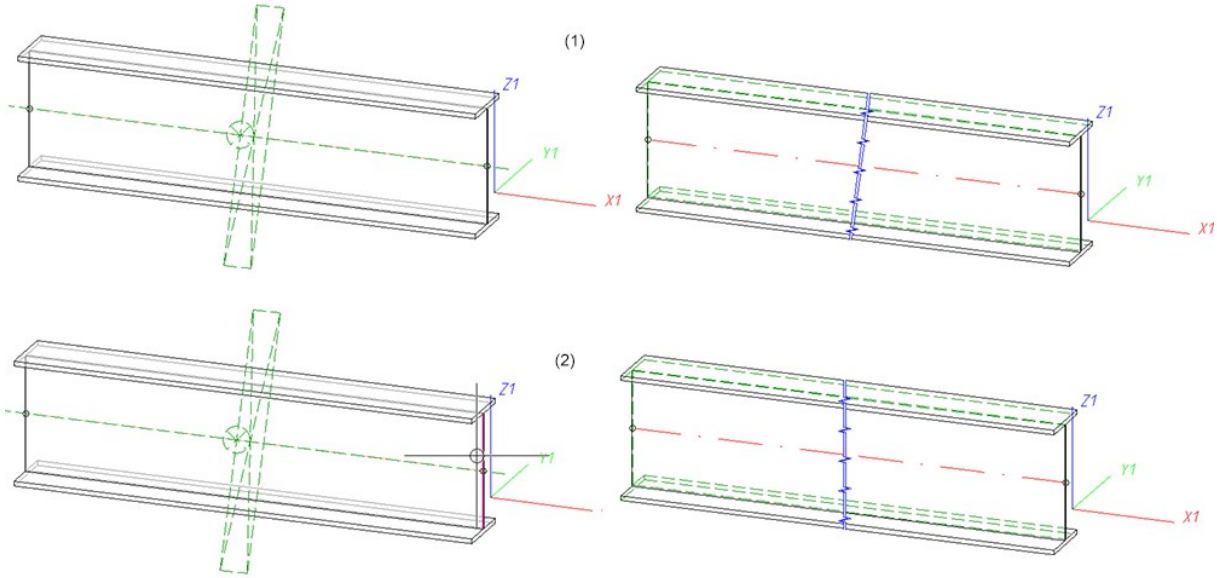
Determine the section direction for views containing shortenings

From HiCAD 2020 SP1 onwards, you have the option of individually defining the section direction, i.e. the direction in which the break lines run. The **Shortened view** dialogue window has been extended accordingly:



After adding a shortening area, click on **Determine**, and then define the direction by specifying two points, selecting an edge or selecting a surface. When selecting a surface, the surface normal is used. If you have defined a section direction, the break lines run parallel to the projection of the section direction into the screen plane. Otherwise, the section is always orthogonal to the direction of shortening, i.e. the direction in which the partial areas are pushed together.

Click on **Reset** to cancel the section direction.



(1) Shortening without section line; (2) Shortening with perpendicular edge as section line

Part properties

Alignment of flanges and bend zones

Since no workshop drawing can be created for sheet metal flanges and bend zones, the alignment selected in the context menu for 3-D parts via **Properties > Part orientation** always refers to the main sheet metal part from HiCAD SP1 onwards.

Alignments of sheet metal flanges and bend zones that were set in older HiCAD versions (before 2020 SP1) can be deleted using the **Reset** function. The main sheet metal part must be active. The view alignment of the sheet metal main part is displayed for these sheet metal straps and bend zones.

Non-dimension relevant assemblies and parts

Not all parts are relevant for the dimensions of assemblies. For this reason, HiCAD offers the option of marking such parts as not relevant for dimensioning. These parts are then ignored in all superordinate parts and assemblies when calculating their dimensions.

To mark a part as "non-dimension relevant", the part attribute Ignore for dimensions (attribute name: #NDR, attribute type: Integer) must be assigned to it. If this attribute has the value 1, it is considered "non-dimension relevant".

To use this attribute, you must manually expand the attribute masks and/or properties windows of the ICN. This attribute is not preset in the masks and windows included in HiCAD's scope of delivery.

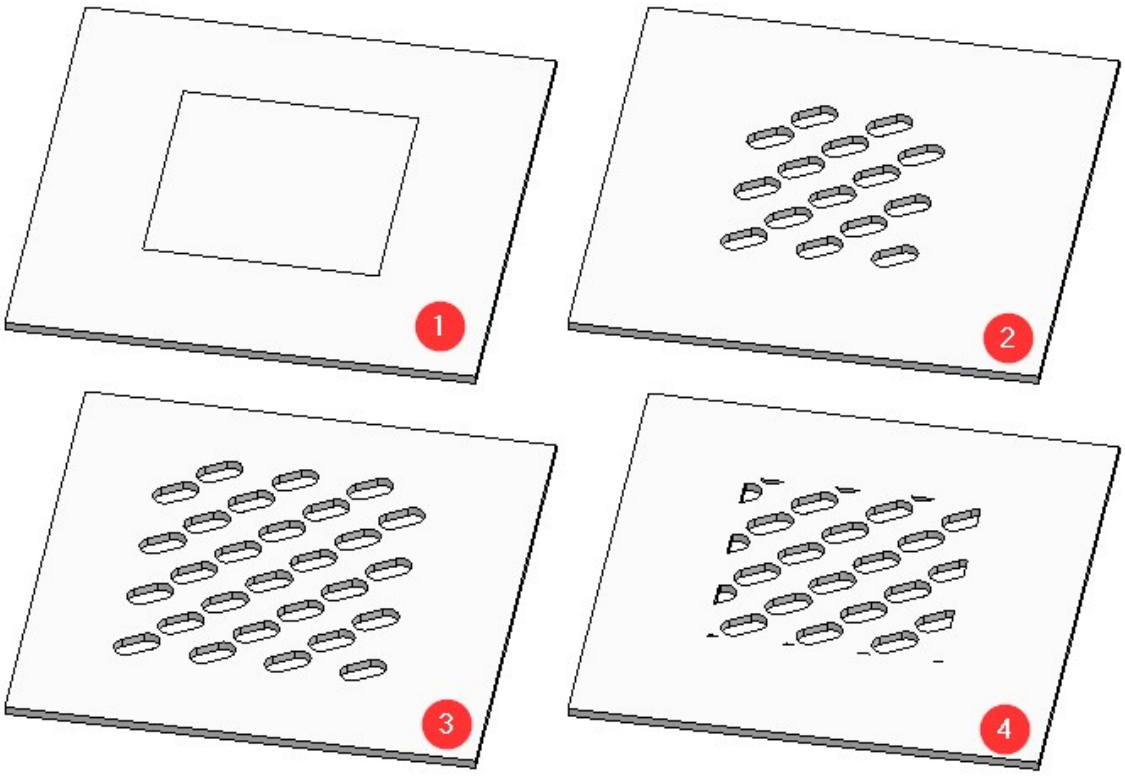
You can find an example here.

Standard processings

Selectable border mode for hole patterns

For the **Hole pattern** function you can now specify how to handle holes that cut the border of the processed area and are therefore only partially inside the processed area. The **Border mode** option is available for this purpose. The possible options are:

- **No hole on border**: Only holes are created that lie completely within the processed area (corresponds to the behaviour of older versions).
- **Complete hole on border**: Every hole that lies partly or completely within the processed area is created completely.
- **Intersect hole with border**: Holes that lie only partially within the processed area are only partially created.

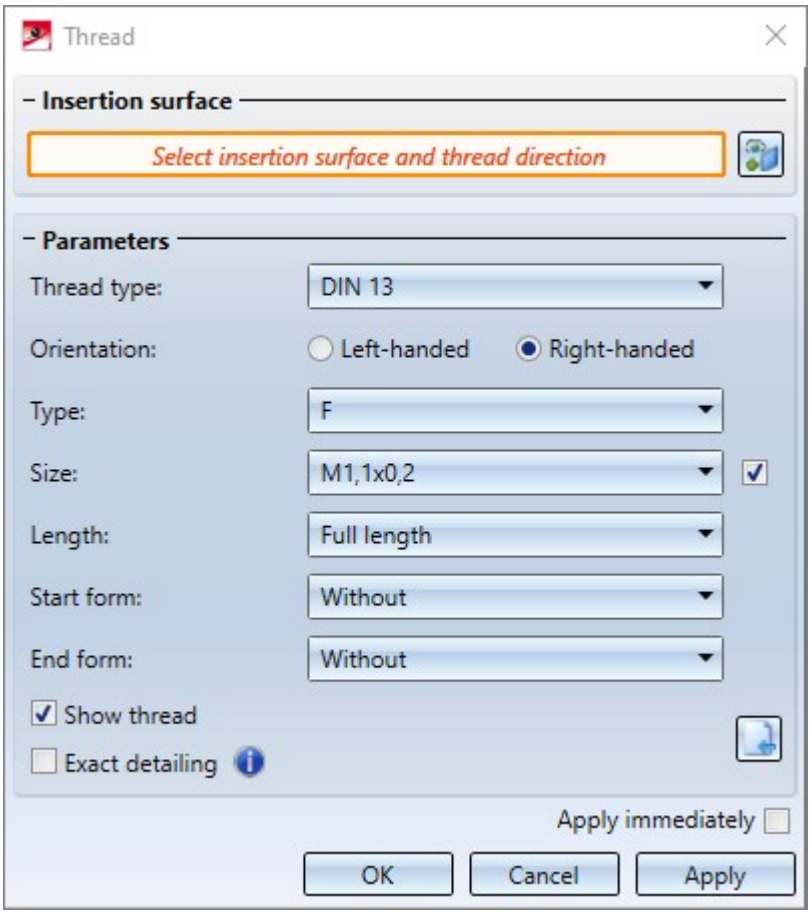


(1) Template: Hole with area to be processed; (2) No hole on border; (3) Complete hole on border; (4) Intersect hole with border

Extended Thread dialogue window

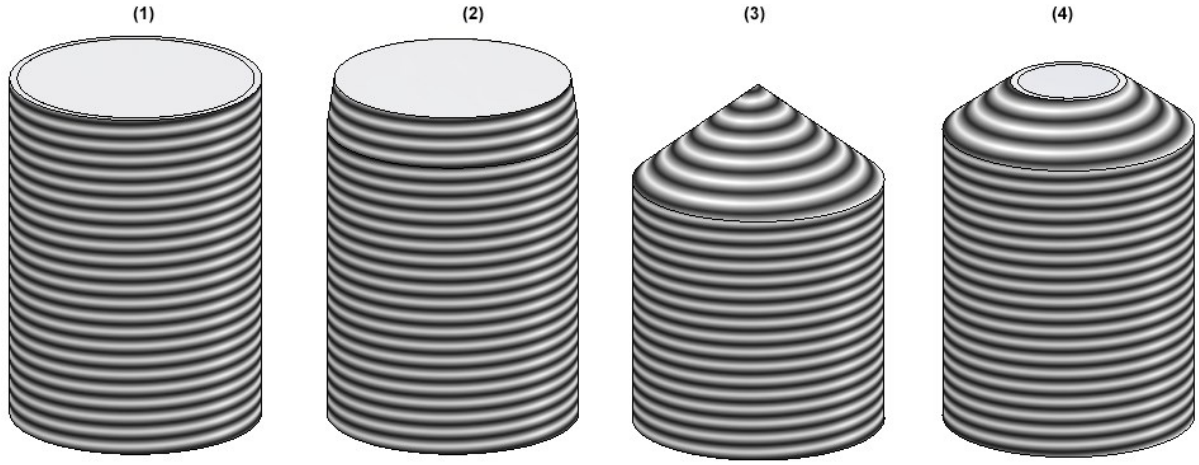
The **Internal/external threads** function has been extended:

- Instead of a cylinder surface you can now also choose an edge on one side of the cylinder.
- The thread texture will be displayed in a preview.
- You can now define the start and the end of the thread.

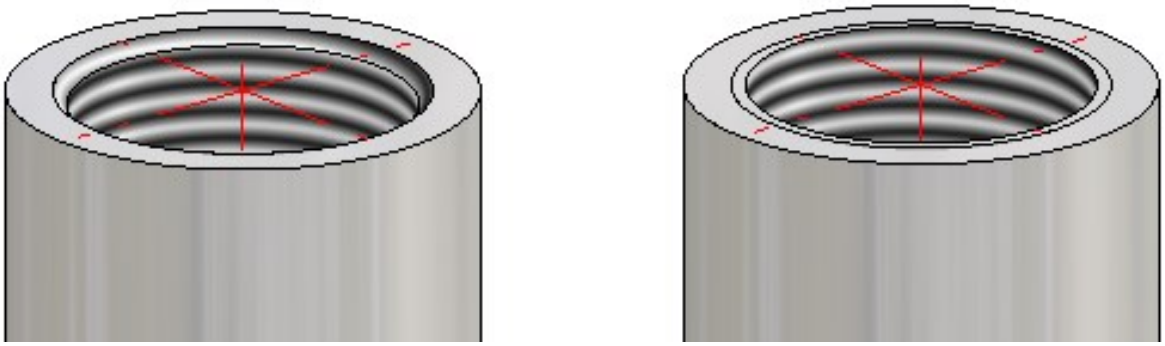


The following options are available:

External thread	Internal thread
Without	**Without**
Chamfer Enter the length of the chamfer or choose **Symmetric**, if you want the chamfer to be created with an angle of 45 degrees. In this case the chamfer length will be calculated automatically. The chamfer will only be created if the **Exact detailing** checkbox has been activated.	**Countersink** The countersink will only be created if the **Exact detailing** checkbox has been activated.
Cone point Enter the length of the cone to its point.	
Tapering Enter the length and the diameter of the tapering.	



(1) Without, (2) Chamfer, (3) Cone point, (4) Tapering



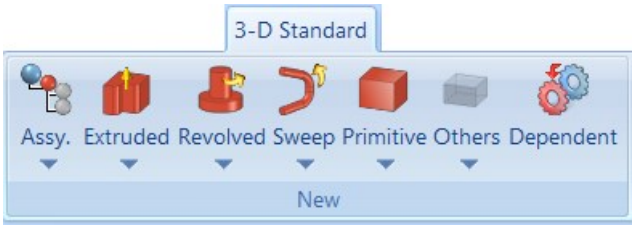
Internal thread with and without countersink

Major Release 2020 (V 2500)

Create parts

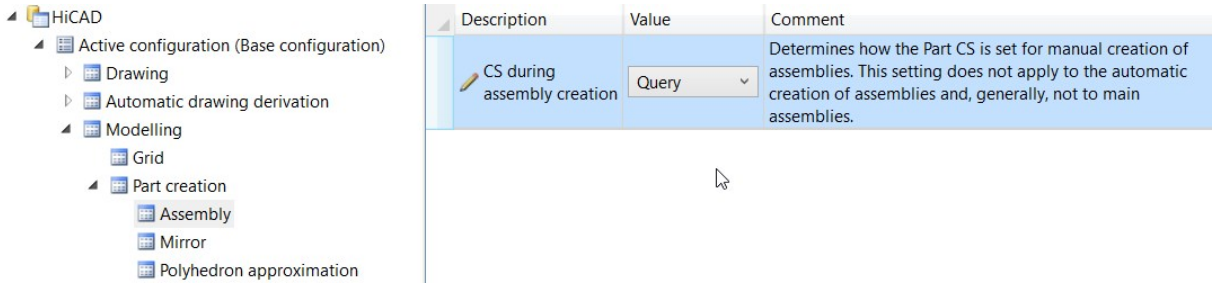
Dependent parts

The **Dependent part** function has been moved into the **New** function group of the **3-D Standard** Ribbon tab.



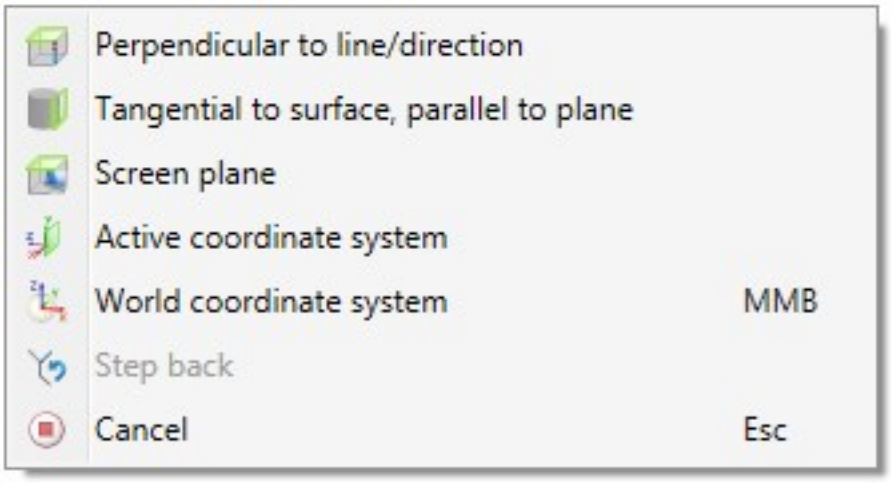
Part CS of assemblies

Like any other part, assemblies, too, have a part coordinate system. This coordinate system can either (as previously) be automatically determined by HiCAD, or (from HiCAD 2020 onwards) determined by you during creation of a dummy assembly or during forming of an assembly from existing parts. For this purpose, a new parameter is available in the ISD Configuration Editor at **Modelling > Part creation > Assembly**.



The following settings are possible:

- **Standard CS** (default)
  The Part CS will be determined automatically by HiCAD and corresponds to the World CS. There will be no further queries. This behaviour is identical to that of previous versions.
- **Query**
  If this setting is active, HiCAD will ask you to choose a plane, as you would do for a processing plane. This plane is the top view plane of the Part CS. You have the following options to determine this plane:
  - You determine the plane by identifying, in any order, points, edges, surfaces and also processing planes in the model drawing. The standard CS will then be rotated in such a way that the XY-plane lies in the selected plane.
  - Press the middle mouse button to choose the World CS.
  - Right-click to activate a context menu with further functions.



The functions of the menu work in the same way as the same-named functions for processing planes.

The settings in the Configuration Editor do not apply to main assemblies. For created new main assemblies the Part CS always corresponds to the World CS.

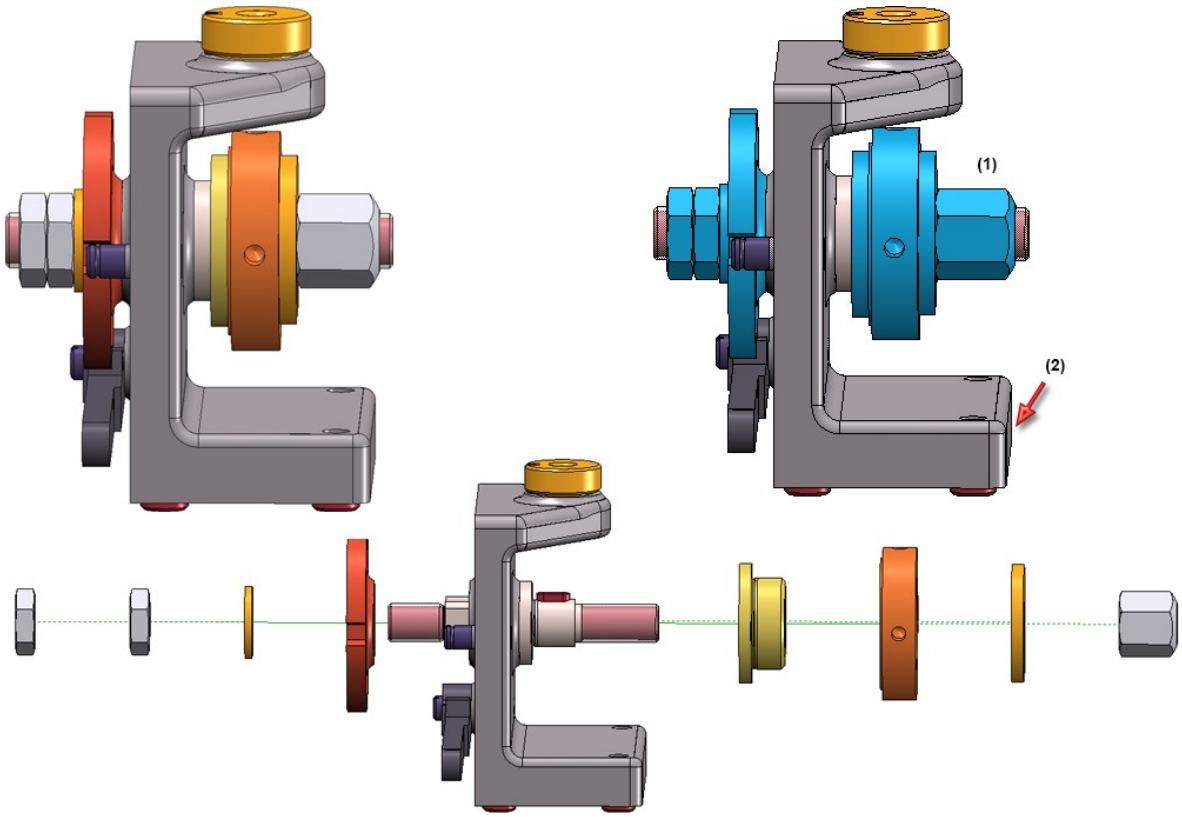
Views

Exploded view with AutoDirection

HiCAD 2020 features a new movement type called **AutoDirection** on the Exploded view Ribbon tab.

This function allows you to define displacements with automatic direction in the Exploded view mode. As with the displacement, several parts are linearly displaced simultaneously. In contrast, these parts are not moved with the same distance, but are distributed along the selected direction relative to a certain part (the so-called fixed part). This makes it possible to move several parts simultaneously in positive and negative direction in one step - depending on the position of the parts in relation to the selected fixed part.

In the image below, take a look at the initial drawing shown on the left. We want to achieve that the selected parts (1) - highlighted blue in the figure - are moved on both sides of the fixed part (2) in one step, depending on their position relative to part (2). The desired result is shown in the figure below. This can be achieved with the **AutoDirection** function.



New icons for Show/Hide parts

The icons for the **Show/Hide parts** functions in 3-D views have been replaced with new and more descriptive icons. This concerns both the functions on the Views Ribbon and the functions in the context menu for views. This concerns the following functions:

Hide parts in active view, Individual (Views > Parts > Hide)

Hide all parts in active view and show individually (Views > Parts > Off/On)

Show parts in active view, Individual (Views > Parts > Show)

Show all parts in active view (Views > Parts > ShowAll)

Visibility of textures in the HiCAD Viewer

The settings for the representation of textures (at **Views > Representation > Shaded** > **Shaded representation**) will now be considered for the representation of textures. This means that whether and in which view the textures will be shown in the Viewer will depend on the above settings.

Settings for shaded representation

The functions **Shaded representation, active view** and **Shaded representation, all views** have been renamed and now have new icons:

Settings for shaded representation, Active view

Settings for shaded representation, All views

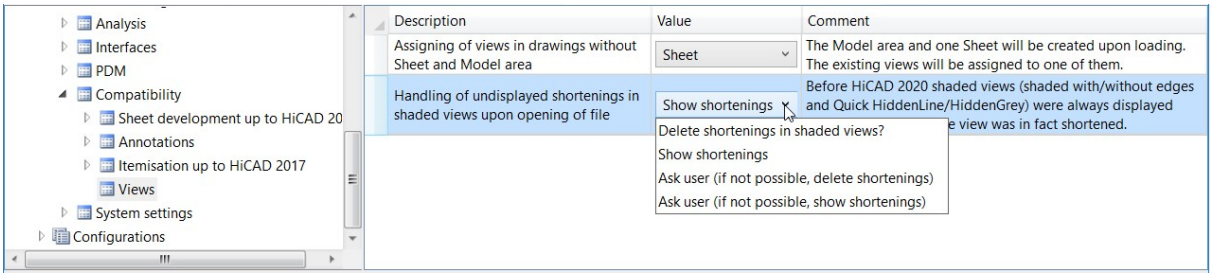
Shortened views

In HiCAD 2020 a shortening of views is also possible for shaded views.

Up to now, views were created with the representation type

- Shaded with/without edges or
- Quick HiddenLine / Quick HiddenGrey

is always shown unshortened, even if the view was actually shortened. For drawings created with versions before HiCAD 2020 and containing such views, the Configuration Editor at **Compatibility > Views** allows you to define how to open these drawings.



The default setting is **Show shortenings**.

Simulation

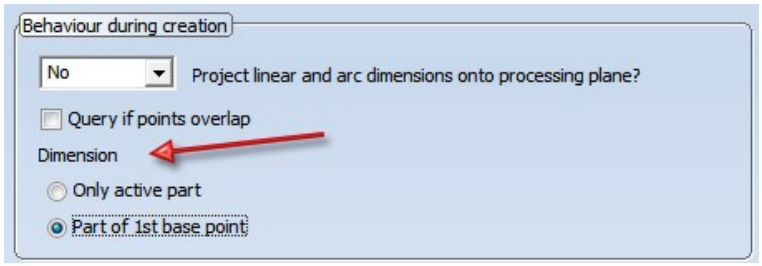
Discontinuation of "old" assembling simulation

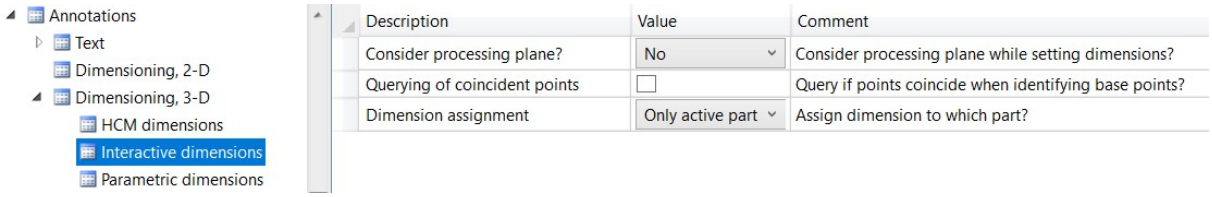
The old functions **Assembling simulation** and **Simulation (Compatibility)** that could be reached via the toolbars (**Settings > Toolbars** have been discontinued and will be no longer available as of HiCAD 2020. For the creation of simulations, please use the functions of the **Simulation** docking window.

Dimensioning

Dimension assignment

In previous versions, dimensions were always assigned to the active part during their creation. From HiCAD 2020 onwards, you can assign the dimensions either to the active part or to the part to which the 1st dimension base point belongs. This behaviour can be determined via the **Set parameters for new general dimensions** function, on the **System** tab of the dialogue window.



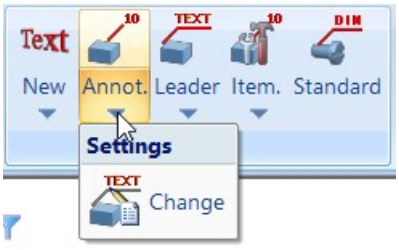


The parameters of this tab can be preset via the Configuration Editor, at **System settings > Annotations > Dimensioning, 3-D > Interactive dimensions > Dimension assignment**. The default setting is **Active part**.

Texts+ Annotations

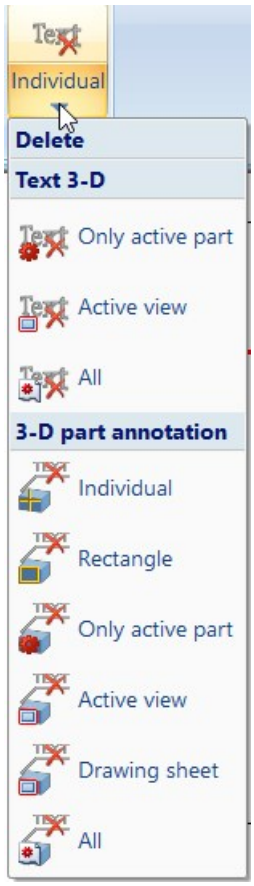
Part annotation settings

The **Part annotation Settings** function can now be found at **3-D Dimensioning+Text > Text > Annot.** **> ...** .



Delete part annotations

The Delete functions for part annotations can now be found in the pull-down menu of the **Delete text** function:

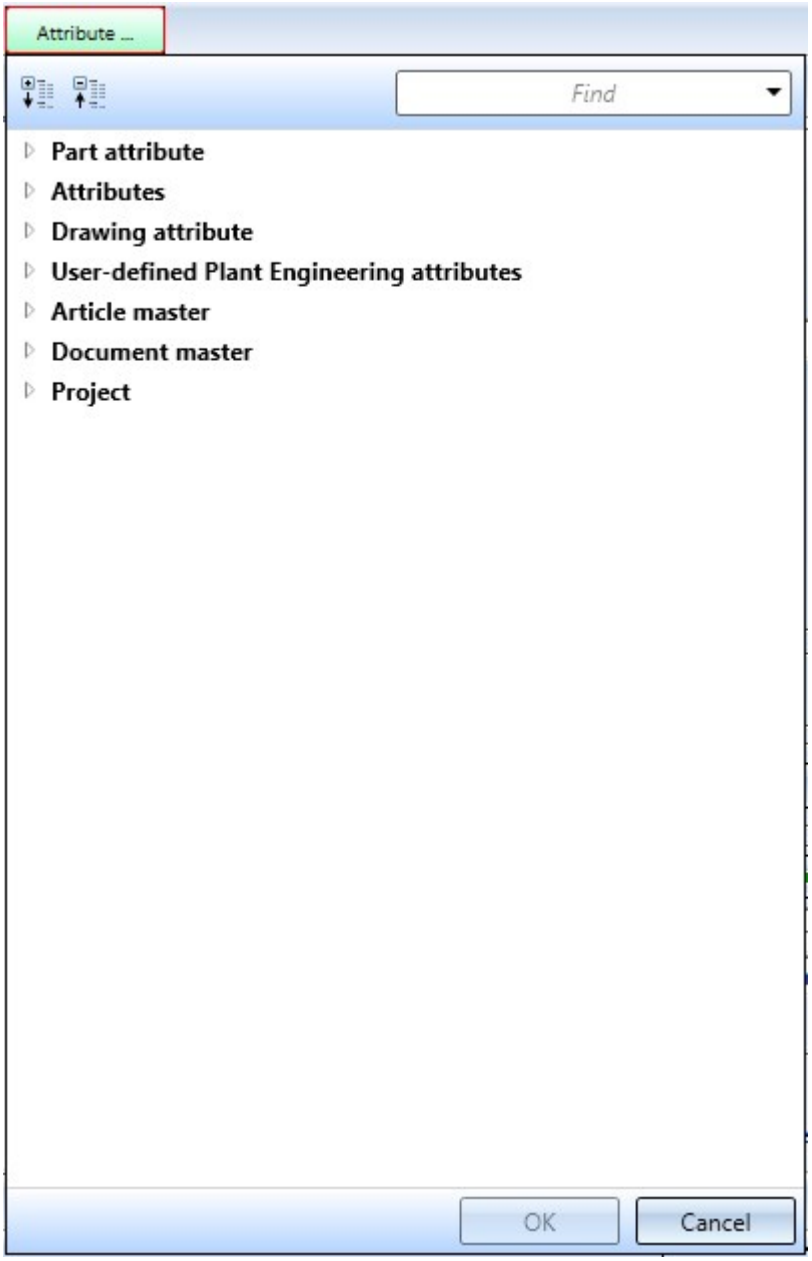


New in this menu is the function **Delete 3-D part annotation, drawing sheet** function. Use this function to delete all annotations on the active drawing sheet.

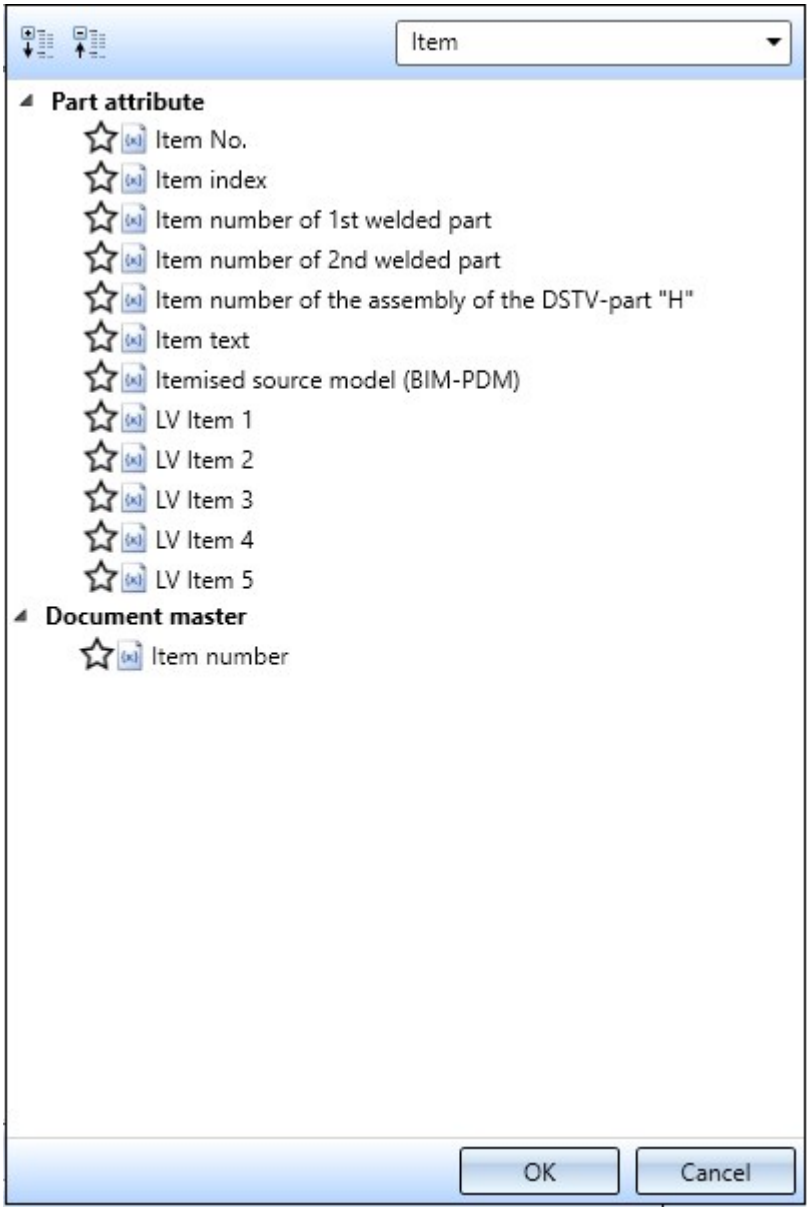
Changed attribute selection dialogue

HiCAD and HELiOS attributes can now be selected directly in the Annotation Editor. To do this, click on the **Attributes** button and select the desired attribute from the selection list.

The list is divided into the different attribute groups

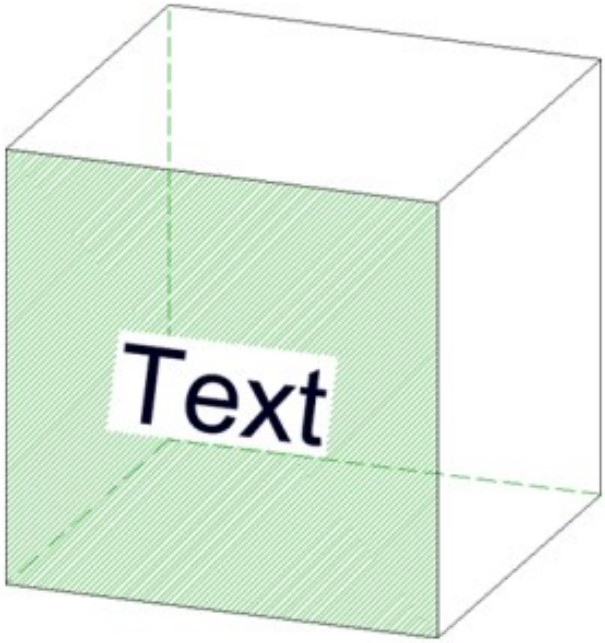


The selection list can be expanded, collapsed and searched for specific entries.


Item
Part attribute
Item No.
Item index
Item number of 1st welded part
Item number of 2nd welded part
Item number of the assembly of the DSTV-part "H"
Item text
Itemised source model (BIM-PDM)
LV Item 1
LV Item 2
LV Item 3
LV Item 4
LV Item 5
Document master
Item number
OK
Cancel

Revised 3-D text

- The handling of 3-D texts has been revised. From HiCAD 2020 onwards, texts that are newly inserted into the model drawing can not only be selected via their reference point, but also via any point on the text. This facilitates selection and increases performance. In addition, the text is automatically generated with an exclusion.



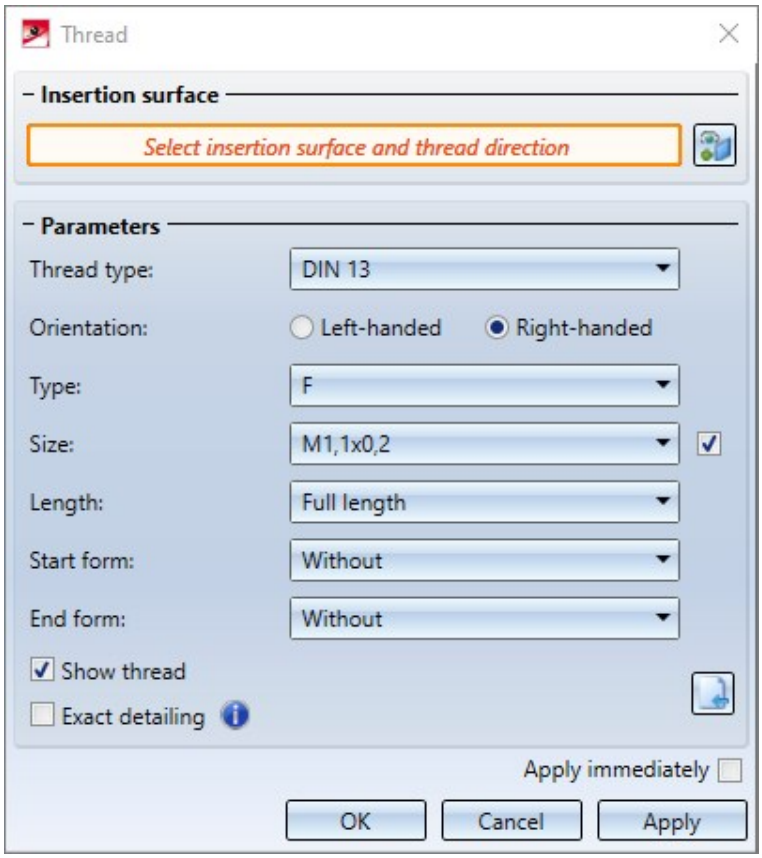
- The functions
  - **Copy active text and move via 2 points (3-D)** and
  - **Move text, via 2 points (3-D)**

  are no longer available.
- The default settings for fonts can be set in the Configuration Editor, separately for 2-D and 3-D texts, at **Drawing > Annotations > Text**.

Standard parts/processings

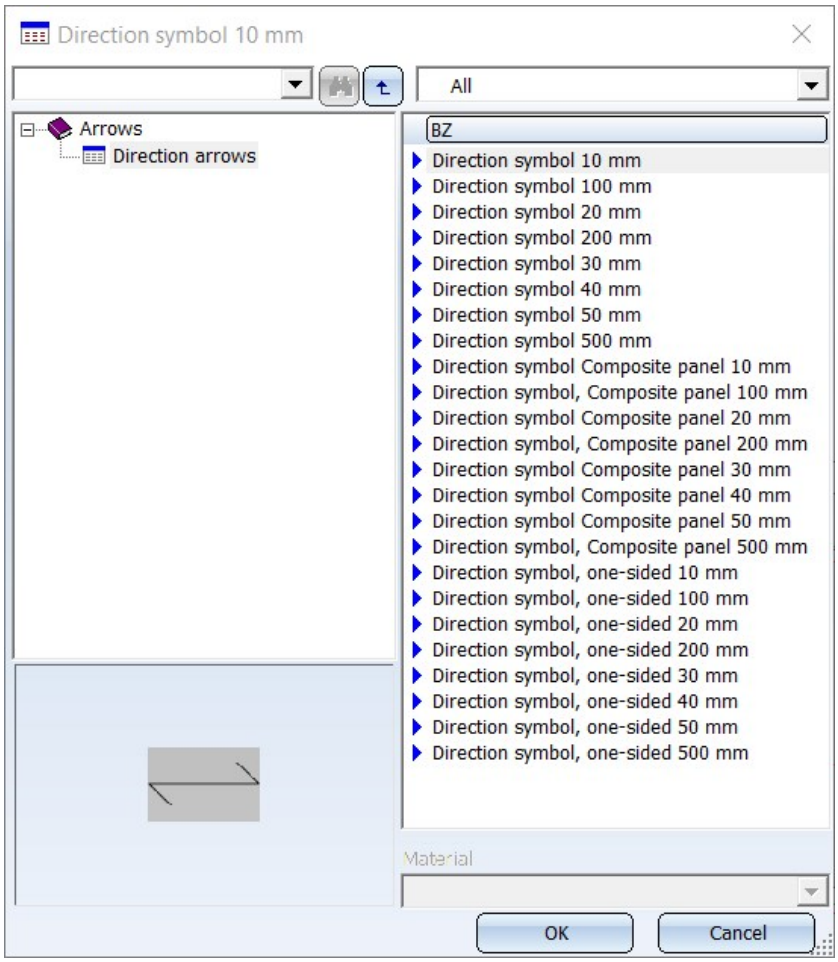
Revised Threads function

The function **3-D Standard > Standard Processings > Thread** for the creation of internal and external threads has been revised completely. and outer



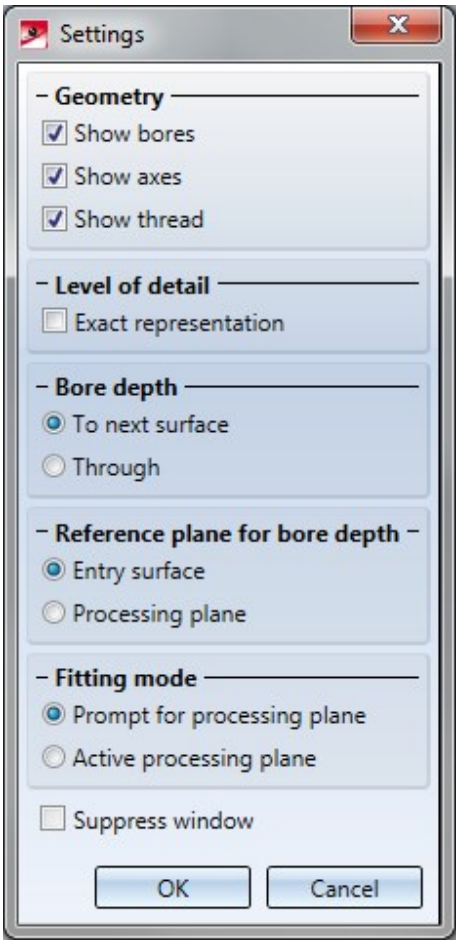
Processing direction

New symbol sizes are available in the table **Direction arrows** (in the catalogue **Factory standards > Symbols > Arrows**). These are available to you when you choose the **Processing direction** function.

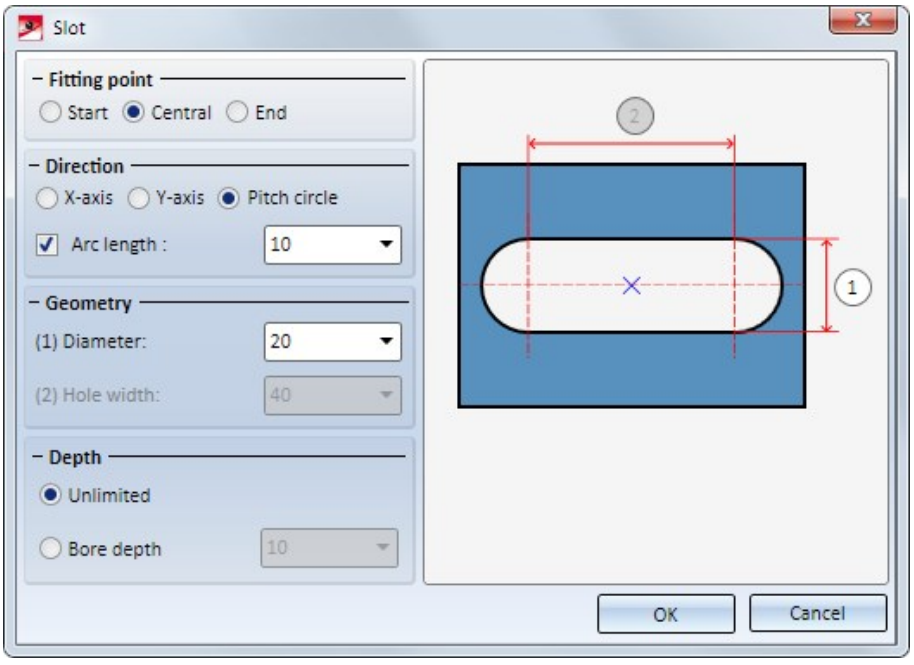


Standard processings / Slots

- In the Settings for Standard Processings the **Through** dialogue text has been replaced with **Unlimited**.



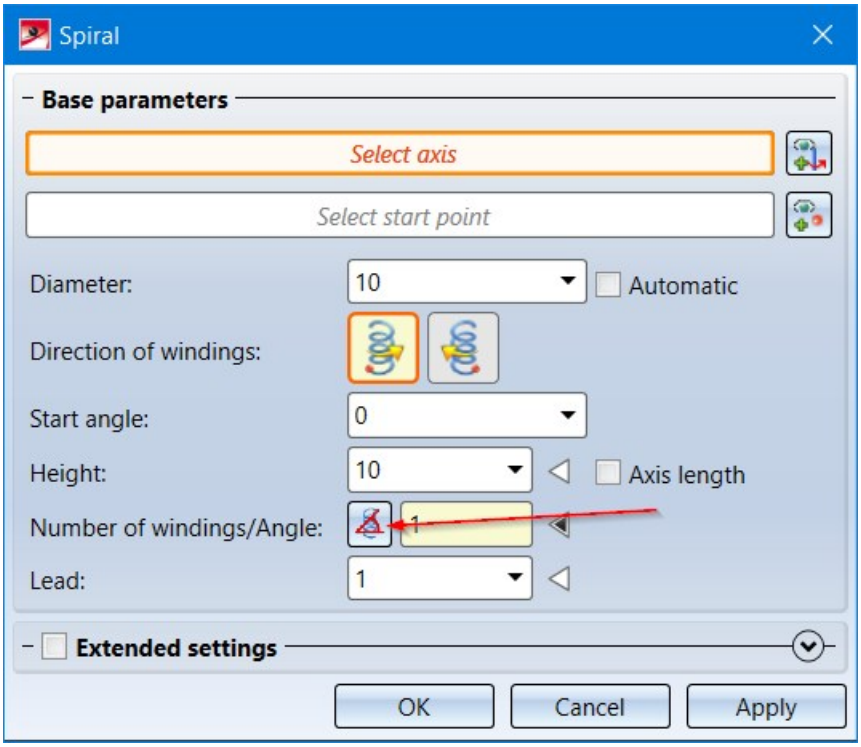
- In the dialogue window of the **Slot** function the **Through** checkbox has been replaced with the options **Unlimited** and **Bore depth**.



Sketches

Spirals

In the dialogue window of the **Spirals** function you can now also specify a winding angle instead of a number of windings. Use the and symbols to switch between the options.

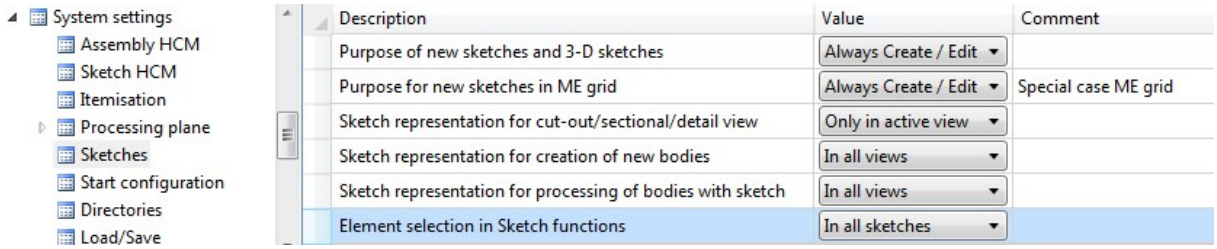


If this symbol is visible, the **Number of windings** mode is active. Enter the number of the required windings. The winding angle will be calculated automatically.

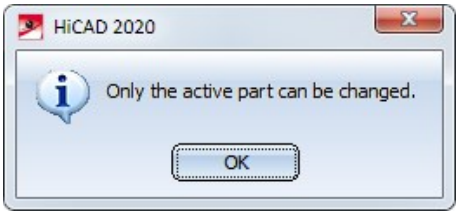
If this symbol is visible, the **Winding angle** mode is active. Enter the required winding angle. The number of windings will be calculated automatically.

Element selection in Sketch functions

In some drawings in can happen that sketches or sketch lines are hidden by other sketches or sketch lines in front of them.To prevent accidental deleting or trimming of the wrong sketches or sketch lines, HiCAD offers you the option to determine whether you can only select lines of the active sketch, or the lines of all sketches when you use these functions. This behaviour can be determined in the Configuration Editor at **System settings > Sketches > Element selection in Sketch functions**. The default setting is **In all sketches**.

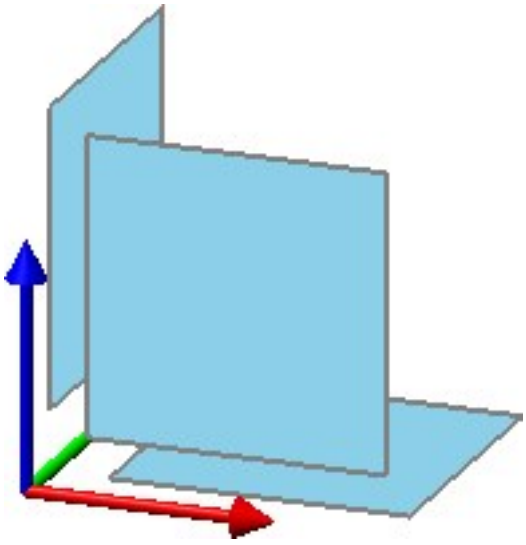


If the setting **Only in active sketch** is active, the following message will be displayed when you select a line of a non-active sketch with the corresponding functions:



Preview of processing planes when creating sketches

If the World CS planes for new sketch (also 3-D sketch) checkbox is active in the **Novice configuration**, a preview of the planes of the world coordinate system will be displayed when creating new sketches. You can then simply use the cursor to select one of the default planes displayed as a preview.

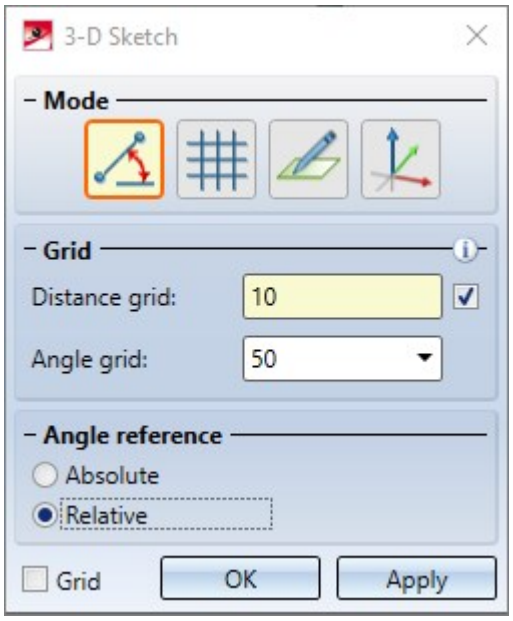


This preview is now considerably larger.

New Sketching Tool for 3-D sketches

Similar as for planar sketches, the new 3-D Sketching Tool is now available for 3-D sketches as a convenient tool for creating three-dimensional polylines.

If a 3-D sketch is active, the **3-D Sketching Tool** will be automatically started when you choose the **Line** (Sketch > Draw > Line) function.



The Sketching Tool simplifies the design process by automatically displaying auxiliary lines along a predefined grid starting from the last point of a line and displaying the corresponding angles, distances or radii at the cursor depending on the selected mode. On this grid you can, for example, determine the direction and length of a line with a mouse click by appropriate cursor movements. Since 3-D sketches - like other 3-D parts - have a part coordinate system, you can change the plane in which you are drawing as you wish while drawing. This drawing plane is always parallel to the selected plane and runs through the last point determined.

Model and process parts

Hide points/crosshairs

The functions

Hide point and

Hide crosshairs

will now remain active after selecting a point/crosshairs edge, i.e. you can select further points/crosshairs edges to be hidden. Press the MMB to exit the function.

Wrap function moved

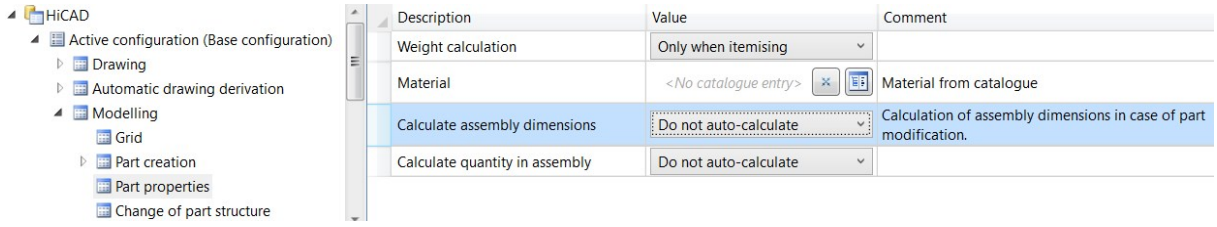
The **Wrap** function can now be found in the pull-down menu of the **Deform** function.

Automatic calculation of assembly dimensions

For assemblies, the dimensions of the parts belonging to the assembly can be used to automatically generate the attributes Length, Height and Width of the assembly and enter it in the assembly attributes. All solid parts belonging to the assembly are included in the calculation, with the exception of standard and purchased parts with the insertion type **Site assembly**.

Excluded from the automatic calculation are structure assemblies.

Whether and when the assembly dimensions are calculated when parts are changed can be set in the Configuration Editor, namely at **Modelling > Part properties > Calculate assembly dimensions**.



The following settings are possible there:

- Do not calculate automatically
  The attributes of the assembly are not updated. This is the default setting.
- Only during itemisation
  The attributes are only updated during itemisation.
- Always
  The attributes of the assembly are updated immediately, i.e. after each change to the components or after adding further components.

To calculate the dimensions, a bounding box for the assembly is determined internally, i.e. the smallest cuboid that completely encloses the parts of the assembly to be considered.

1. The orientation of the bounding box also plays a role in the calculation of the dimensions. This is determined according to the following priority:
2. Alignment of the changed or newly added part, i.e. the view selected in the context menu for Parts at Properties > Part orientation.
3. Alignment of the assembly main part
4. Coordinate system of the assembly main part
5. Coordinate system of the assembly itself

The first option that is available is used. Since each assembly has a part coordinate system, there is always a usable orientation and the dimensions can be calculated.

The assignment of coordinate system axes to the values is

- x: Length
- y: Width
- z: Height

If the part orientation (point 1 or 2) is used, length/height is derived from the front view and the width is perpendicular to it. This assignment is not configurable.

Catalogue Editor

Service Pack 2 2020 (V 2502)

Derive new standard parts from existing standard parts

As of SP2 you have the possibility to derive new standard parts from existing standard parts. Currently, tables of the following catalogues can be copied:

- Processings, general,
- Semi-finished products,
- Fasteners, and
- Factory standards.

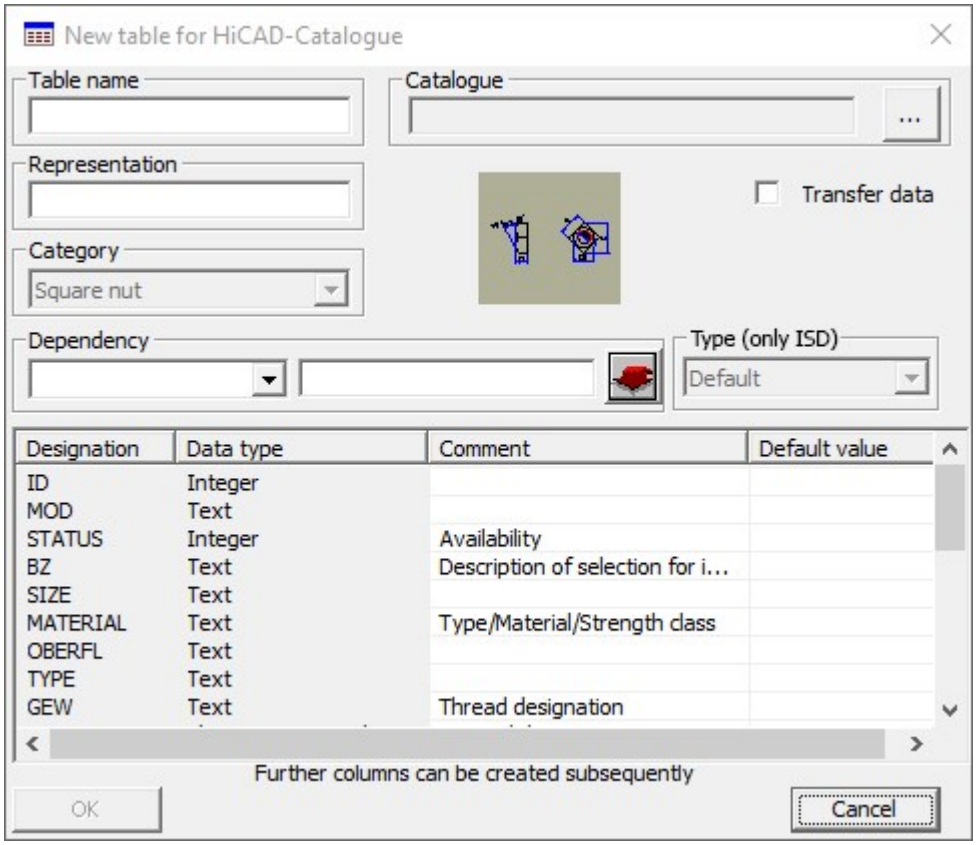
This means in particular that you can use all the geometries and tables for fasteners and semi-finished products created by the ISD to derive your own tables for standard parts with similar geometry. For example, you can quickly derive tables for bolts/nuts of a similar DIN from an already existing table of bolts/nuts of a specific DIN. To do this, you use the new **Copy table** function to copy an existing table and then modify the copy.

Standard parts created in this way behave in the same way as the standard parts in the original table and can be inserted in exactly the same way. For example, they can be inserted with the same function, their display can be switched analogously and - in the case of Fasteners - they can also be used in boltings.

The IPT file of the copy is always stored in the **Factory standards** catalogue and can only be assigned to meaningful catalogues in the Catalogue Editor. This depends on the standard part type of the output table.

For example, if you have selected the DIN 557 table in the **Fasteners > Nuts > Square nuts** catalogue, the copy can only be assigned to the **User-defined fasteners > User-defined nuts** catalogue and its sub-folders.

If you have copied a table from the **Semi-finished products > Beams + Profiles > I-Beams** catalogue, the copy can only be assigned to the **Factory standards > Factory beams** folder and its sub-folders.



The dialogue window of the **Copy table** function.

Track catalogue changes

Up to now, you could define in the Configuration Editor at **System settings > Catalogues**

- how catalogue changes should be handled in HiCAD and
- how catalogue tables should be handled.

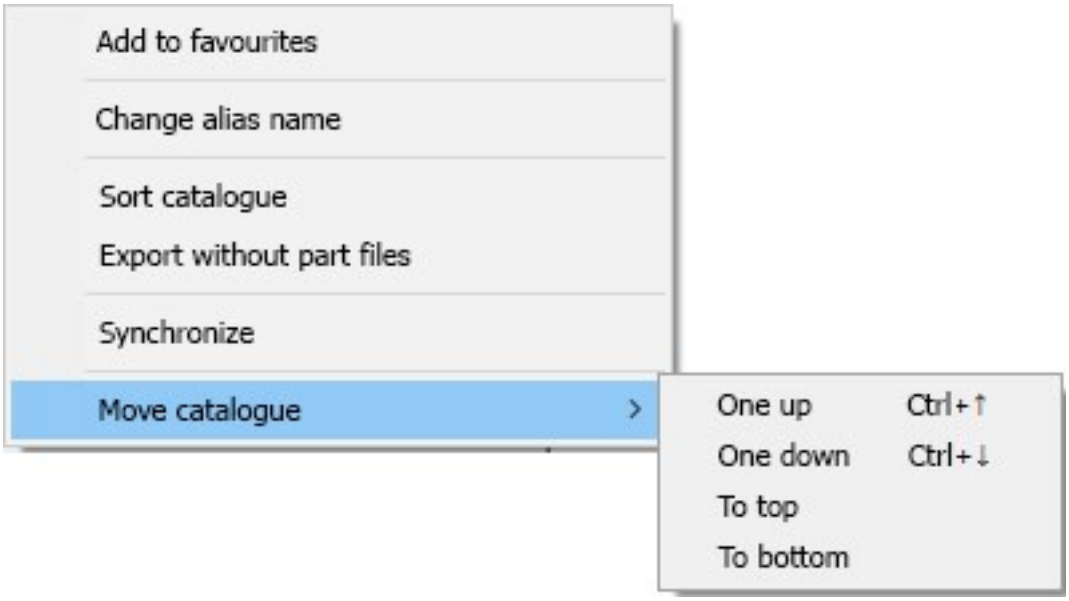
With HiCAD 2020 SP2, these settings have been removed from the configuration management. The parameter "Handling of catalogue tables" has been omitted without replacement. Instead of the parameter **Track catalogue changes**, HiCAD provides a corresponding switch at **Settings > Catalogue**.

Change order of catalogues

The order of the catalogues can now be changed if necessary. This affects both the display in the Catalogue Editor and the catalogue selection in HiCAD. For example, you can put the catalogues that are important to you and most frequently used by you in the front positions.

The following options are available for changing the sequence:

- You click with the cursor on the name of the catalogue to be moved and move the selected catalogue up or down with the key combinations CTRL + Move Up or CTRL + Move Down.
- You activate the context menu for catalogues by clicking with the right mouse button on the name of the catalogue to be moved and use the functions at **Move catalogue**.



Please note:

- The order you have set will be retained during a catalogue update.
- Moving is only possible within the same level, i.e. a catalogue cannot leave its level.

HILTI HSA-Segment anchor

Segment anchors are single parts when inserted in HiCAD, whereas the real anchor consists of several parts. As of HiCAD 2020 SP2, the corresponding catalogue table contains the material of the expanding part in the TYPE column.

The table HSA Segment Anchors under **Factory standards > User-defined fasteners > User-defined anchors > Hilti** has been extended by anchors of the type

- **HSA-R2** and
- **HSA-BW**.

ALUCOBOND® semi-finished products

The ALUCOBOND® semi-finished products of 3A Composites GmbH available in the catalogue have been extended by semi-finished products for North America. As of SP2 the following tables are available:

Catalogue	Table
Textures and Colours > Colours	ALUCOBOND_US_FARBEN
Factory standards > Sheets > ALUCOBOND US	■ ALUCOBOND ANODIZED ■ ALUCOBOND BRUSHED ■ ALUCOBOND COLOR-SHIFTING ■ ALUCOBOND METALLIC ■ ALUCOBOND MICA, ■ ALUCOBOND MIRROR ■ ALUCOBOND SOLID ■ ALUCOBOND TEXTURED ■ ALUCOBOND PATTERN ■ ALUCOBOND WOODGRAIN
Factory standards > Surface treatment > Profile and element Installation > ALUCOBOND US	■ Coating ALUCOBOND ANODIZED ■ Coating ALUCOBOND BRUSHED ■ Coating ALUCOBOND COLOR-SHIFTING ■ Coating ALUCOBOND METALLIC ■ Coating ALUCOBOND MICA, ■ Coating ALUCOBOND MIRROR ■ Coating ALUCOBOND SOLID ■ Coating ALUCOBOND TEXTURED ■ Coating ALUCOBOND PATTERN ■ Coating ALUCOBOND WOODGRAIN

Fasteners USA

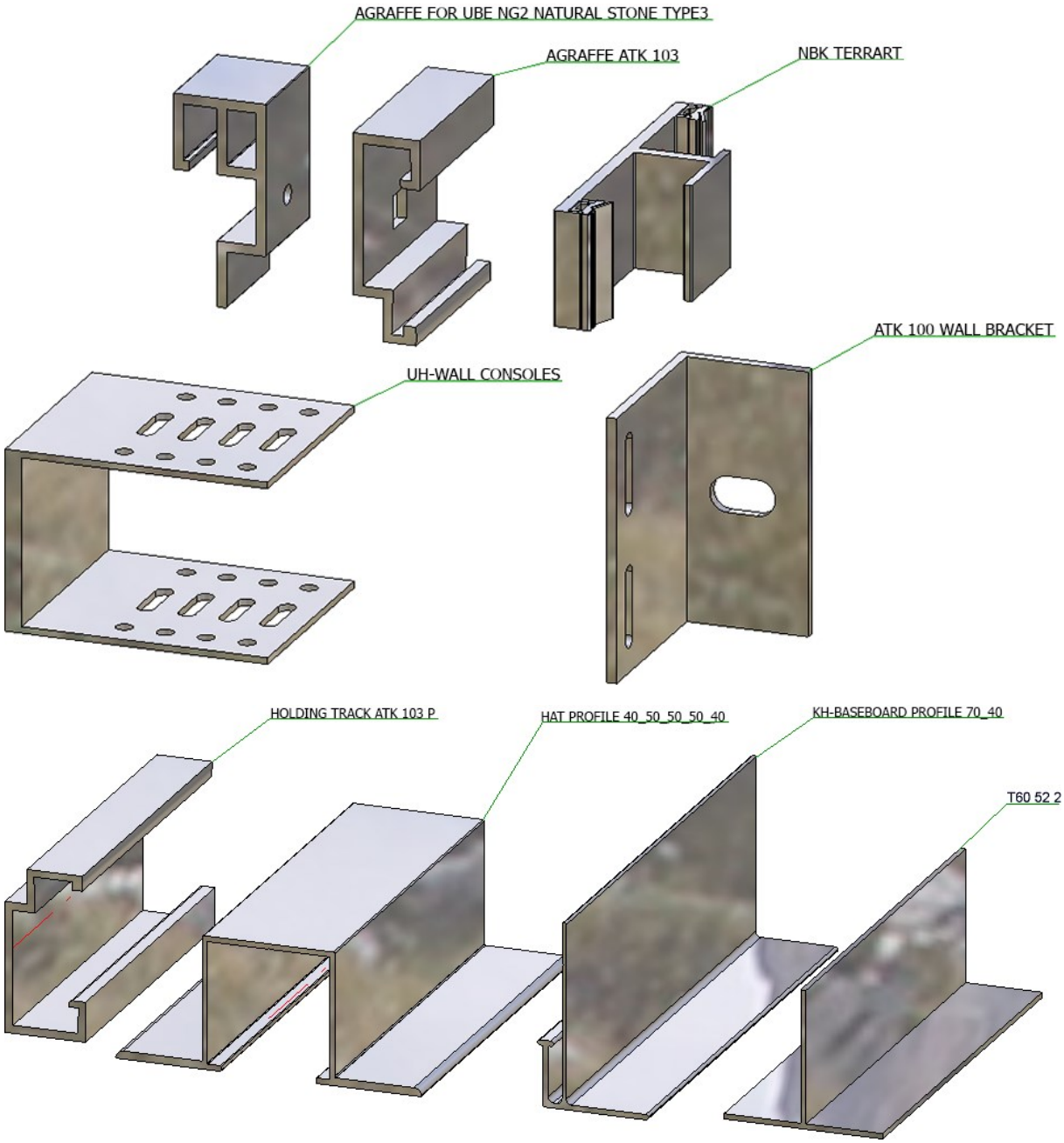
Multiple fasteners for the American market have been added to the catalogue **Factory standards > User-defined fasteners**:

Catalogue	Table
User-defined rivets / Fastenal	■ Rivet 120° Countersunk Head ■ Rivet Dome Head
User-defined rivets /Rivet King / Multigrip / MultigripRT "Aesthetic"	■ Dome Head MGRT ■ Large Flange MGRT ■ Countersunk MGRT
User-defined bolts+screws / Fastenal / Self-Drilling Screws	■ Phillips Drive Pan Head
User-defined bolts+screws / Fastenal / Self-Drilling Screws	■ Hex-Unslotted Hex Washer Head
User-defined bolts+screws / Fastenal / Sheet Metal Screws	■ Indented Hex Washer head Slotted
User-defined bolts+screws / SENCO	■ Rex Drive Flat Pan Head

SYSTEA and BWM profiles

The **Factory standards** catalogue has been extended by special profiles for Roof Wall Facade and Purchased/Factory standard parts from Systea and BWM:

Catalogues	Table
Special profiles by Systea and BWM	
Factory standards / Series / Roof Wall Facade / Special profiles / BWM	■ ATK 103 ■ Hat profiles ■ ATK 106 SZ 20 ■ Support profiles
Factory standards / Series / Roof Wall Facade / Special profiles / Systea	■ Hat profiles ■ Mounting profiles ■ Holding tracks ■ UBE NG2
Agraffes	
Factory standards / Purchased/Factory standard parts / Agraffes / BWM	■ ATK 103
Factory standards / Purchased/Factory standard parts / Agraffes / Systea	■ UBE NG2
Clamps	
Factory standards / Purchased/Factory standard parts / Clamps/ Systea	■ NBK Terrat
Thermostop	
Factory standards / Purchased/Factory standard parts / Thermostop	■ BWM
Wall consoles	
Factory standards /Purchased/Factory standard parts / Wall consoles / BWM	■ L-Wall bracket ■ U-Holder Type S ■ U-Holder Type N
Factory standards / Purchased/Factory standard parts / Wall consoles / Systea	■ L-Wall consoles ■ U-Wall consoles



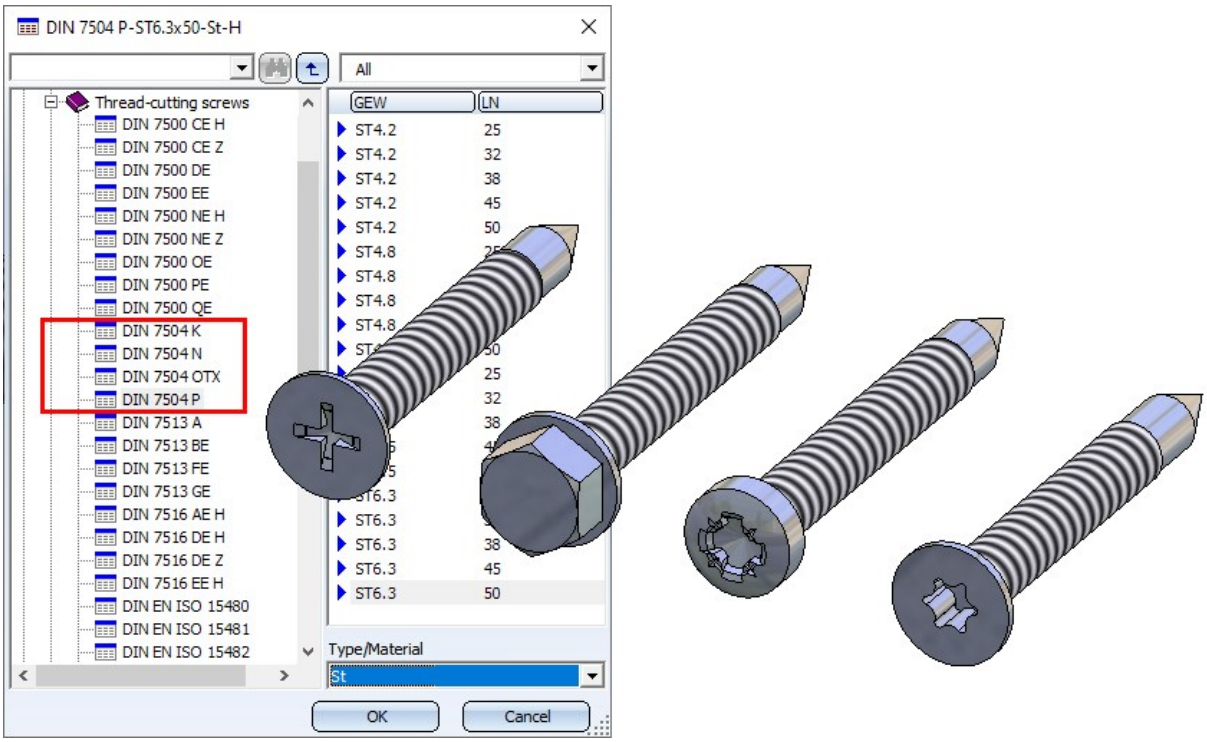
The special profiles can be inserted via **Steel Engineering > New > Cat.**, the other parts via **3-D Standard > Standard Parts > BoltScrew > Purchased/Factory standard parts**.

Service Pack 1 2020 (V 2501)

Thread-cutting screws

New in the catalogue **Fasteners > Bolts+Screws > Thread-cutting screws** are the screws of the type

- DIN 7504 P Cross-recessed thread-cutting screw PH),
- DIN 7504 N (Oval head thread-cutting screw),
- DIN 7504 K (Hexagon head thread-cutting screw) and
- DIN 7504 (Thread-cutting screw OTX).



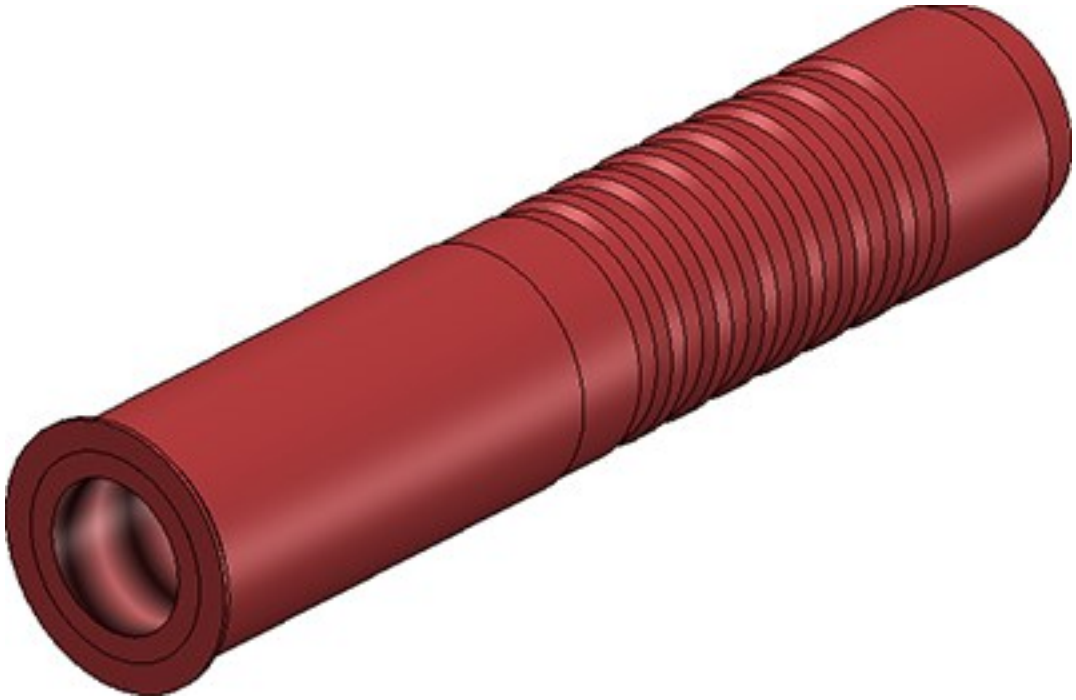
HILTI anchors

- New in the Catalogue Editor at **Factory standards > User-defined fasteners > User-defined anchors > Hilti > HSA Segment anchor** are the HILTI expansion anchors.
- New at **Factory standards > User-defined fasteners > User-defined anchors > Hilti** are the tables:
  - **HUS HILTI Screw anchors** and
  - **HSC-I Undercut anchors**.
- Also new are the HILTI shear connector cartridges **HVU2** and **HVU-TZ**. You can find the corresponding tables at **Factory standards > Purchased/Factory standard parts > Shear connector cartridge > HILTI**. \HVU2 (ID 335701298) and HVU-TZ (ID 335701299). The corresponding 3-D parts are BOM-relevant dummy parts.

HILTI plastic dowels

New in the Catalogue Editor at **Factory standards > User-defined dowels > Hilti** are the following tables with HILTI plastic dowels:

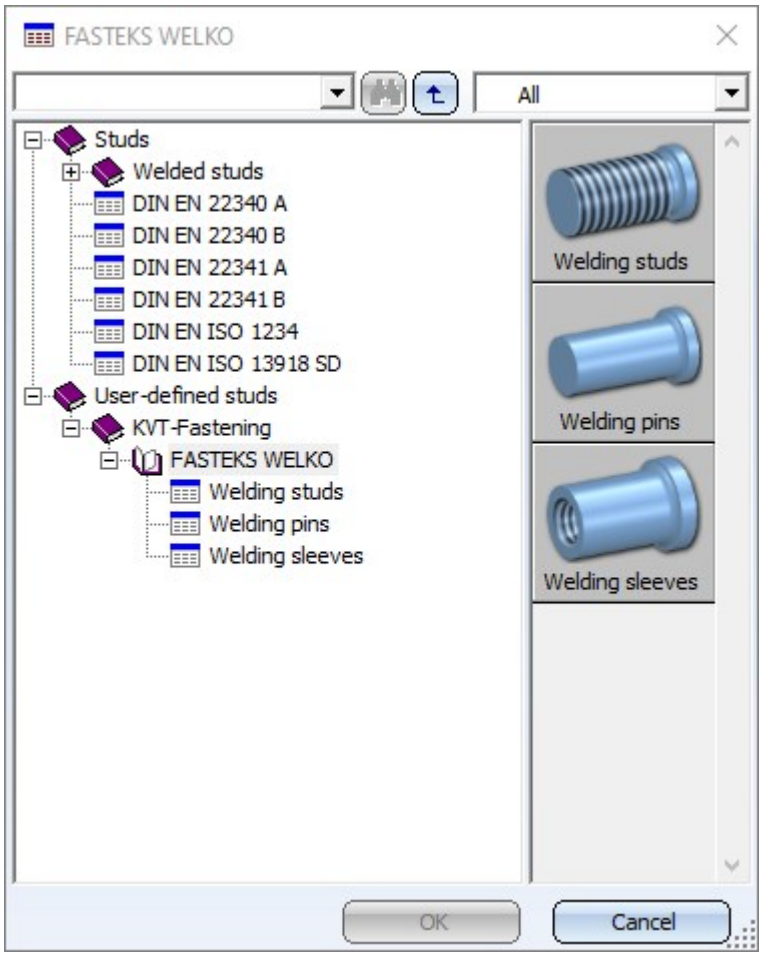
- **HUD-1** and
- **HPS-1**.



Also, wood screws according to DIN 96 are now available especially for these dowels. You can find them at **Fasteners > Bolts+Screws > Wood screws**.

WELKO fasteners

New in the Catalogue Editor are WELKO studs, pins and sleeves. You can find them at **Factory standards > User-defined fasteners > User-defined studs > KVT-fastening > FASTEKS WELKO**:



Fabco and HFT fasteners

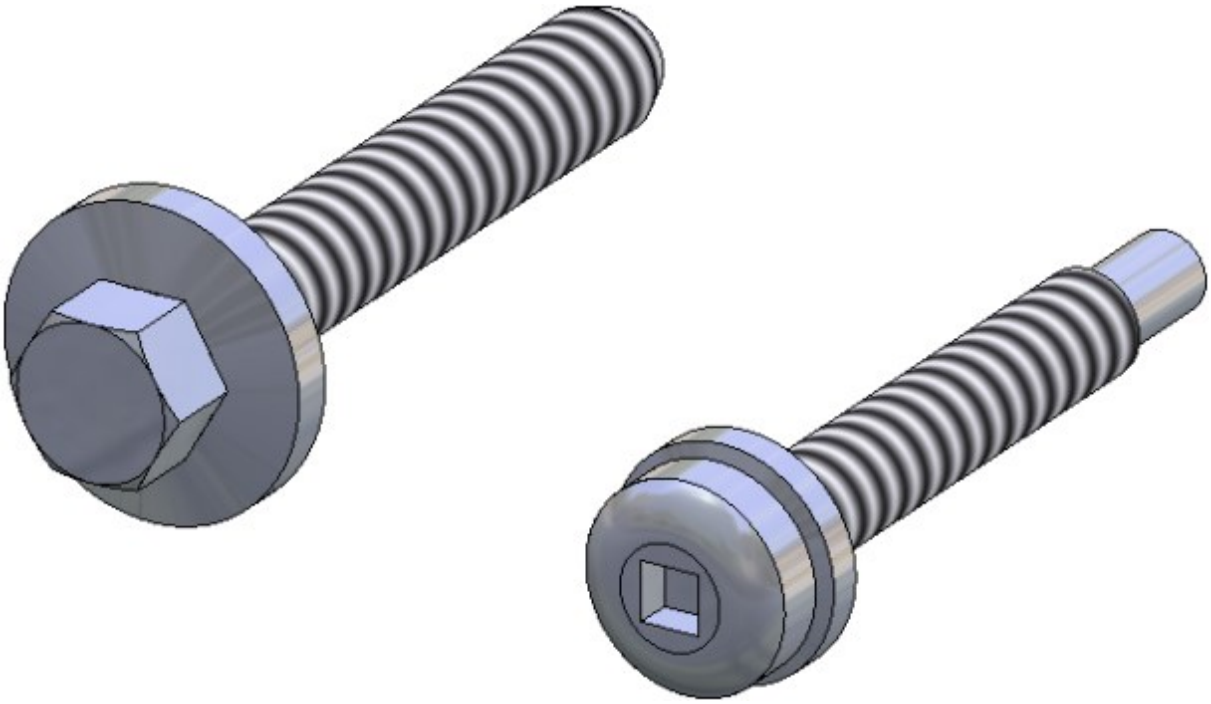
New in the catalogue **Factory standards > User-defined fasteners > User-defined bolts+screws > HFT** are the following tables:

Fabco special screws for roof and wall:

- E-VS A
- E-VS FC
- E-X A
- E-X BZ

HFT facade screws:

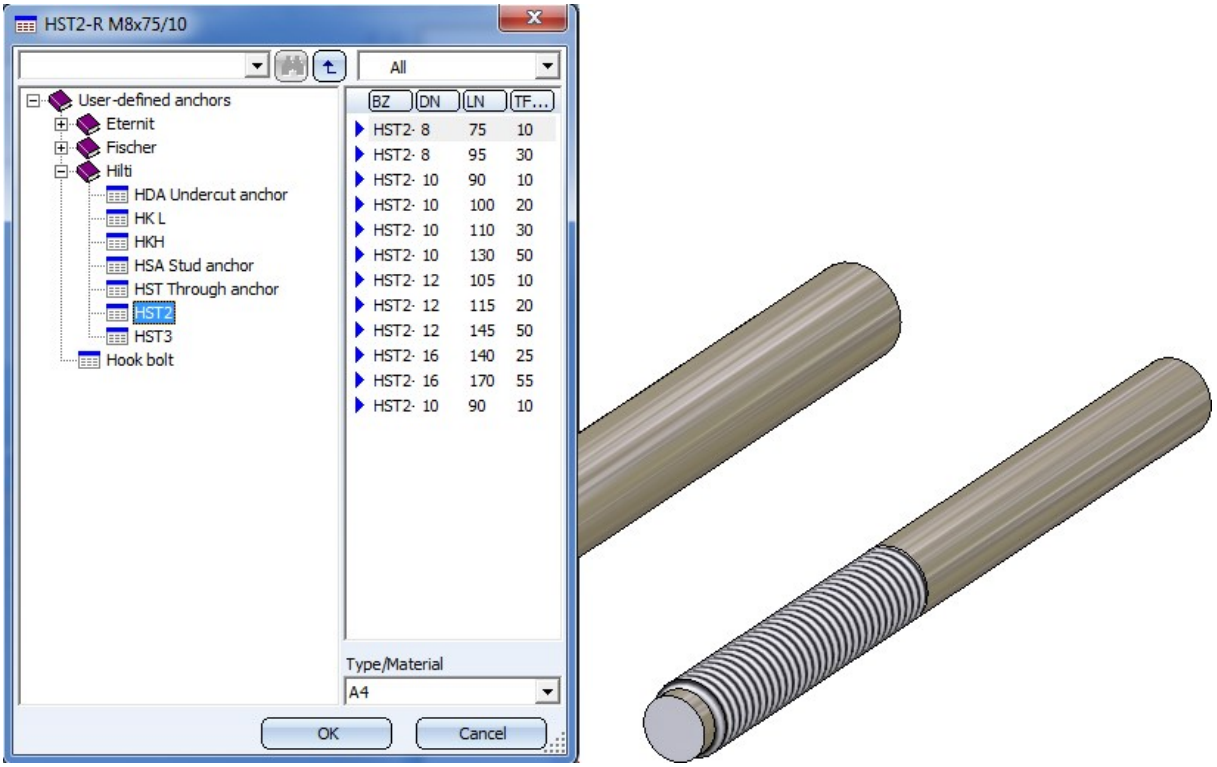
- E-X BR RS SEC 12
- E-X BR 2 SEC 12
- E-VS 8 A



Major Release 2020 (V 2500)

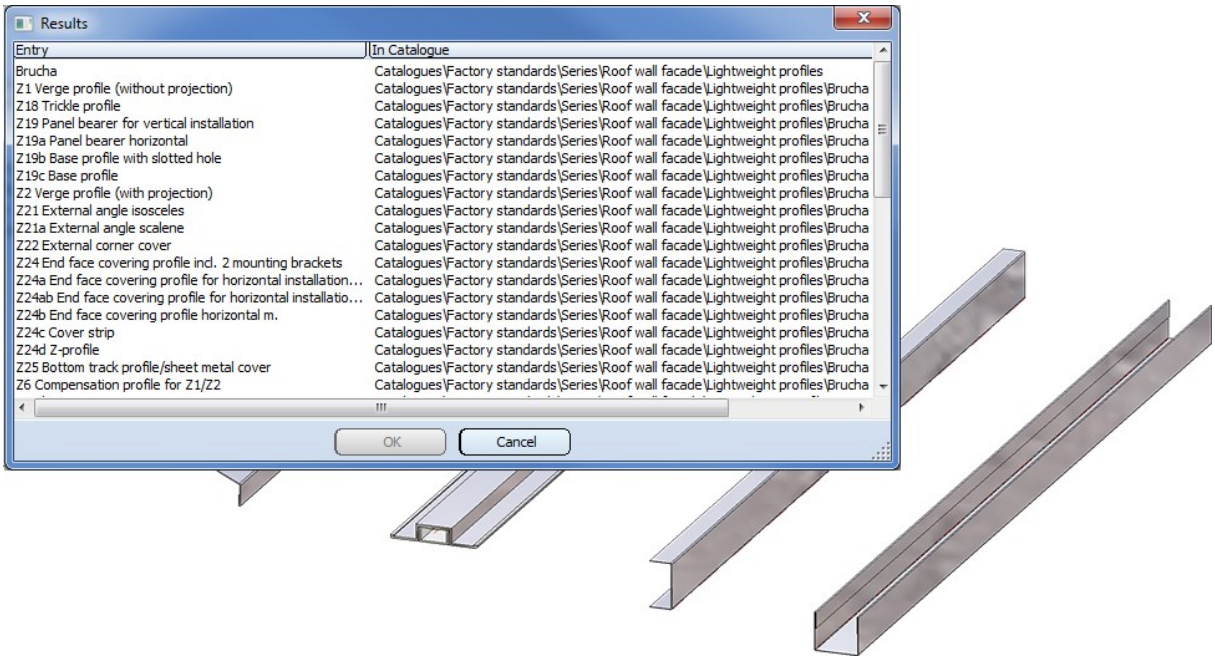
HILTI anchors

New in the catalogue **Factory standards > User-defined fasteners > Usere-defined anchors > Hilti** are the anchors of the types HST2 and HST3.



BRUCHA folded profiles

In the catalogue **Factory standards > Series > Roof wall facade > Lightweight profiles** you will find new profiles of the company BRUCHA for profile installation.



JIS Beams - Standard G 3192

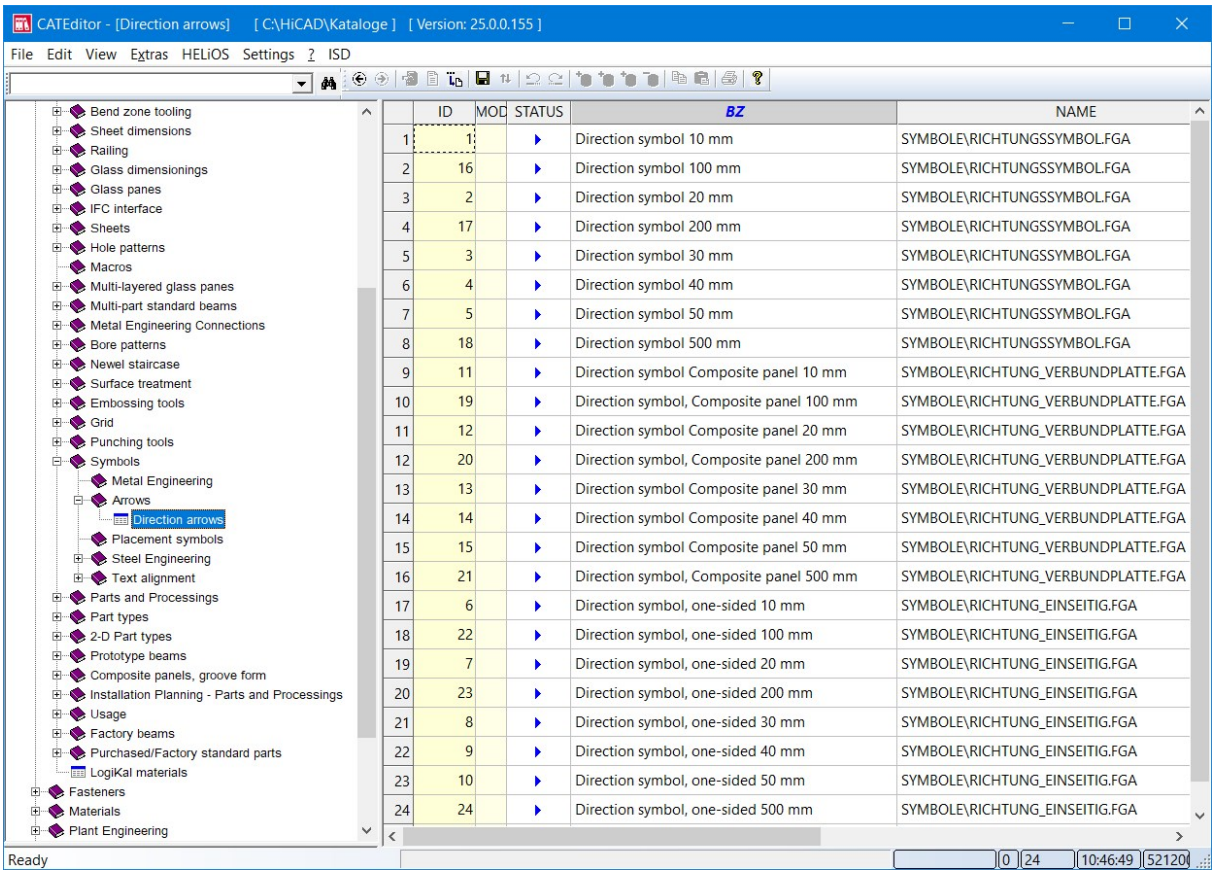
The JIS standard G 3192 contains in some cases, besides the standard description, an annex where further cross-section geometries are stored. In those cases where HiCAD obtains the geometry from the annex, an additional table with the standard geometries exists. The names of these tables have a supplement called **(Standard)**, the tables with the geometries from the annex have the supplement ***(Annex n)***, with *n* being the number of the annex. This applies to the following tables:

- **Semi-finished products > Beams+Profiles > I-Beams**
  - JIS G 3192 H (Standard)
  - JIS G 3192 I (Standard)
  - JIS G 3192 ISB (Annex 5)
  - JIS G 3192 ISC (Annex 6)
- **Semi-finished products > Beams+Profiles > L-Beams**
  - JIS G 3192 L (Annex 7)
  - JIS G 3192 LE (Annex 1)
  - JIS G 3192 LE (Standard)
  - JIS G 3192 LU (Annex 2)
  - JIS G 3192 LU (Standard)
  - JIS G 3192 L (Standard)
- **Semi-finished products > Beams+Profiles > U-Beams**
  - JIS 3192 CH (Annex 4)
  - JIS 3192 U (Standard)
- **Semi-finished products > Beams+Profiles > T-Beams**
  - JIS 3192 T (Annex 9)
  - JIS 3192 T (Standard)

By activating/deactivating the tables in the **CatalogueMaker** you can determine whether JIS beams are to be inserted according to standard description or according to annex.

Additional direction arrows

New symbol sizes are available in the table **Direction arrows** (in the catalogue **Factory standards > Symbols > Arrows**). These are available to you when you choose the **Processing direction** function.



	ID	MOD	STATUS	BZ	NAME
1	1			Direction symbol 10 mm	SYMBOLE\RICHTUNGSSYMBOL.FGA
2	16			Direction symbol 100 mm	SYMBOLE\RICHTUNGSSYMBOL.FGA
3	2			Direction symbol 20 mm	SYMBOLE\RICHTUNGSSYMBOL.FGA
4	17			Direction symbol 200 mm	SYMBOLE\RICHTUNGSSYMBOL.FGA
5	3			Direction symbol 30 mm	SYMBOLE\RICHTUNGSSYMBOL.FGA
6	4			Direction symbol 40 mm	SYMBOLE\RICHTUNGSSYMBOL.FGA
7	5			Direction symbol 50 mm	SYMBOLE\RICHTUNGSSYMBOL.FGA
8	18			Direction symbol 500 mm	SYMBOLE\RICHTUNGSSYMBOL.FGA
9	11			Direction symbol Composite panel 10 mm	SYMBOLE\RICHTUNG_VERBUNDPLATTE.FGA
10	19			Direction symbol, Composite panel 100 mm	SYMBOLE\RICHTUNG_VERBUNDPLATTE.FGA
11	12			Direction symbol Composite panel 20 mm	SYMBOLE\RICHTUNG_VERBUNDPLATTE.FGA
12	20			Direction symbol, Composite panel 200 mm	SYMBOLE\RICHTUNG_VERBUNDPLATTE.FGA
13	13			Direction symbol Composite panel 30 mm	SYMBOLE\RICHTUNG_VERBUNDPLATTE.FGA
14	14			Direction symbol Composite panel 40 mm	SYMBOLE\RICHTUNG_VERBUNDPLATTE.FGA
15	15			Direction symbol Composite panel 50 mm	SYMBOLE\RICHTUNG_VERBUNDPLATTE.FGA
16	21			Direction symbol, Composite panel 500 mm	SYMBOLE\RICHTUNG_VERBUNDPLATTE.FGA
17	6			Direction symbol, one-sided 10 mm	SYMBOLE\RICHTUNG_EINSEITIG.FGA
18	22			Direction symbol, one-sided 100 mm	SYMBOLE\RICHTUNG_EINSEITIG.FGA
19	7			Direction symbol, one-sided 20 mm	SYMBOLE\RICHTUNG_EINSEITIG.FGA
20	23			Direction symbol, one-sided 200 mm	SYMBOLE\RICHTUNG_EINSEITIG.FGA
21	8			Direction symbol, one-sided 30 mm	SYMBOLE\RICHTUNG_EINSEITIG.FGA
22	9			Direction symbol, one-sided 40 mm	SYMBOLE\RICHTUNG_EINSEITIG.FGA
23	10			Direction symbol, one-sided 50 mm	SYMBOLE\RICHTUNG_EINSEITIG.FGA
24	24			Direction symbol, one-sided 500 mm	SYMBOLE\RICHTUNG_EINSEITIG.FGA

Multi-language entries in BZ column

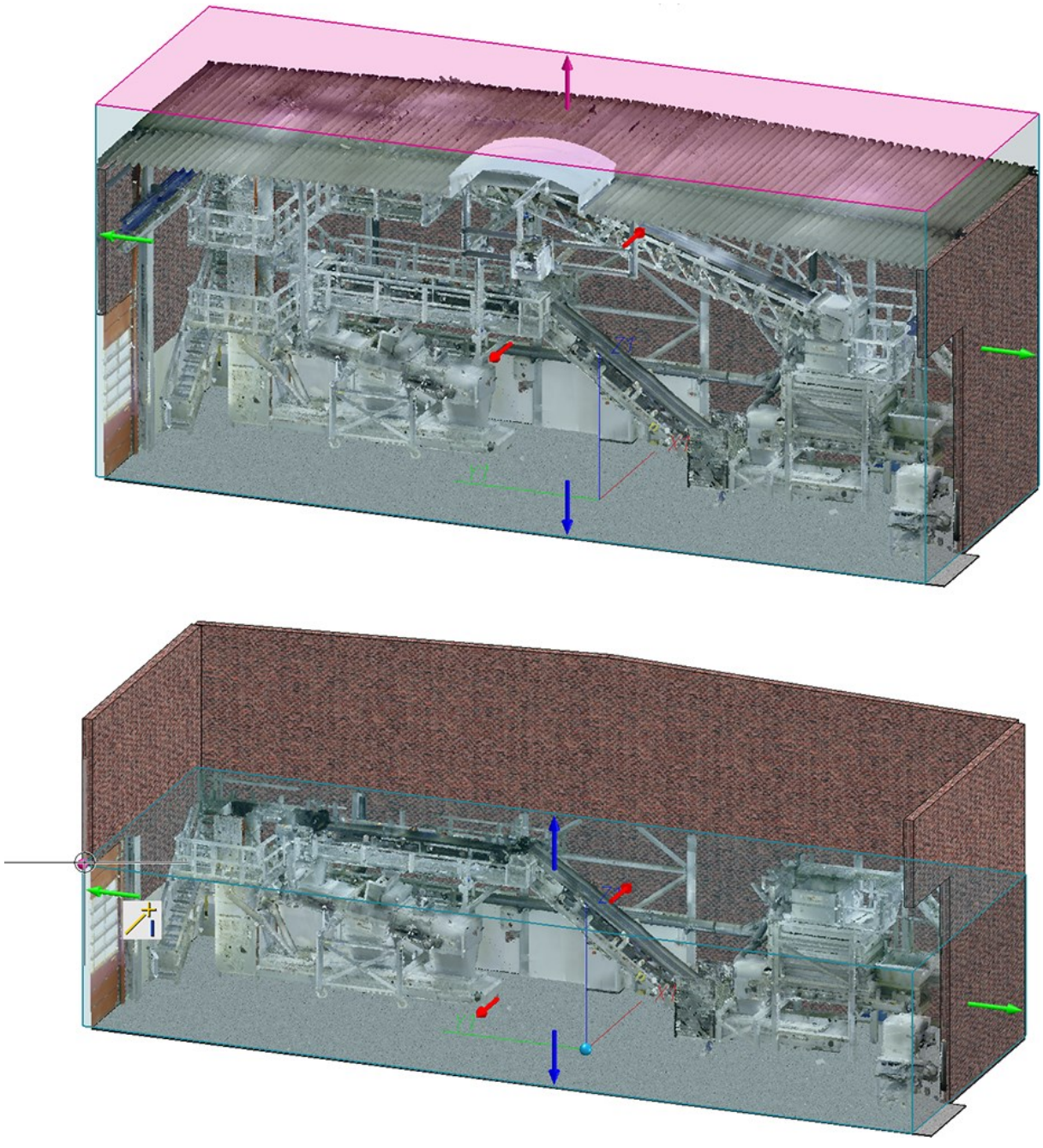
The designations in the column **BZ** now exist in different languages.

Point clouds

Service Pack 2 2020 (V 2502)

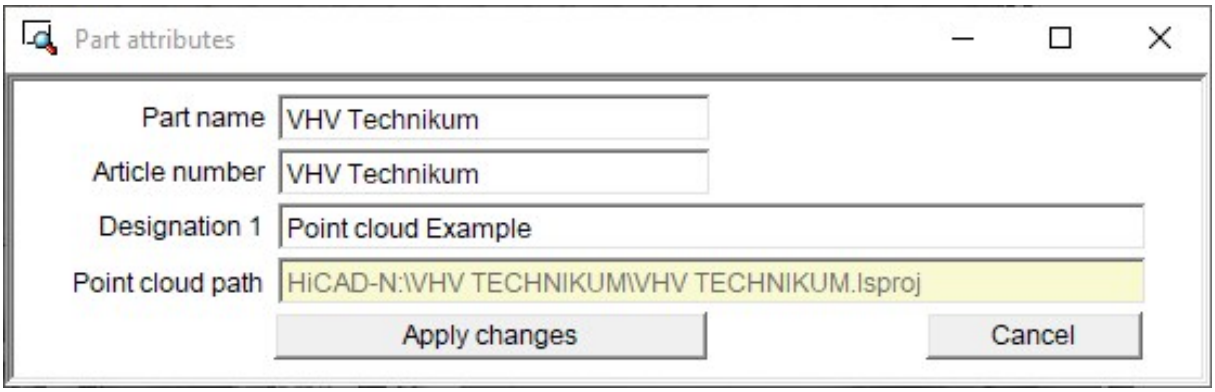
Create / edit clipping boxes

Both when creating and editing clipping boxes, the autopilot now displays snap points. This allows you to set the edges of clipping boxes to exactly one point. You can also use the options of the Point Options menu to set the clipping box edges, especially the **(PW) Point from point cloud** point option.



Part attributes of point clouds

The **Part attributes** dialogue window for point clouds has been optimized.



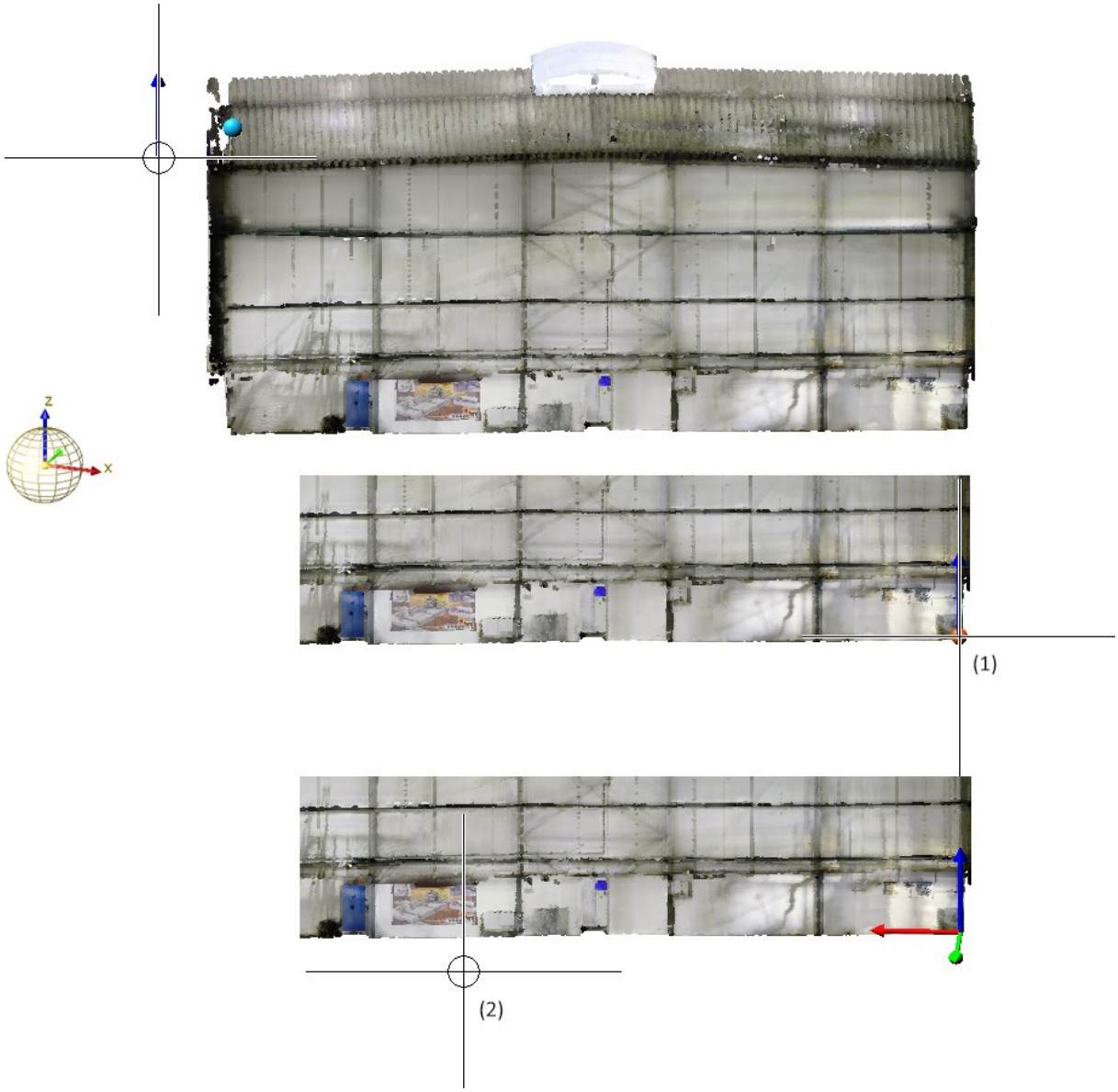
Align point clouds

New in the **Point cloud** Ribbon tab is the function **Align point cloud** . This function allows you to change the alignment selected when inserting a point cloud into the model drawing.

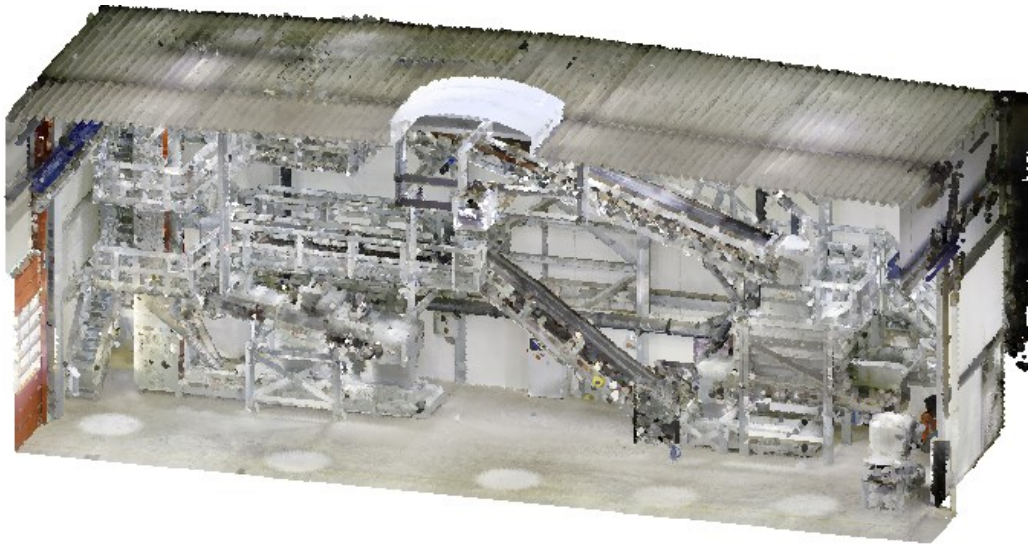
The alignment is changed by defining two points.

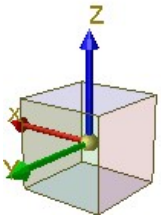
1. The first point defines the new origin of the part coordinate system.
2. The second point determines the X-direction of the part coordinate system.

The determination of the two points and the installation direction are graphically visualized in the drawing. To determine the points, you can also use the point option **Point cloud point (PW)** (not automatically active). Point cloud points always lie on the visible part of the point cloud and are represented by a red point on the cursor.



Result:



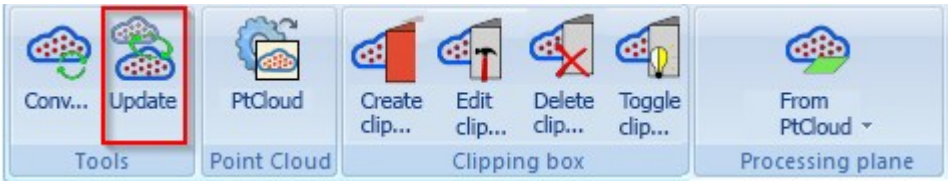


Tip:
The **Point cloud point (PW)** point option is not automatically active when determining the points required to perform this function. In certain situations, it may therefore be useful to determine isolated points before alignment in order to better select the origin and X-direction of the point cloud. Isolated points can also be point cloud points.

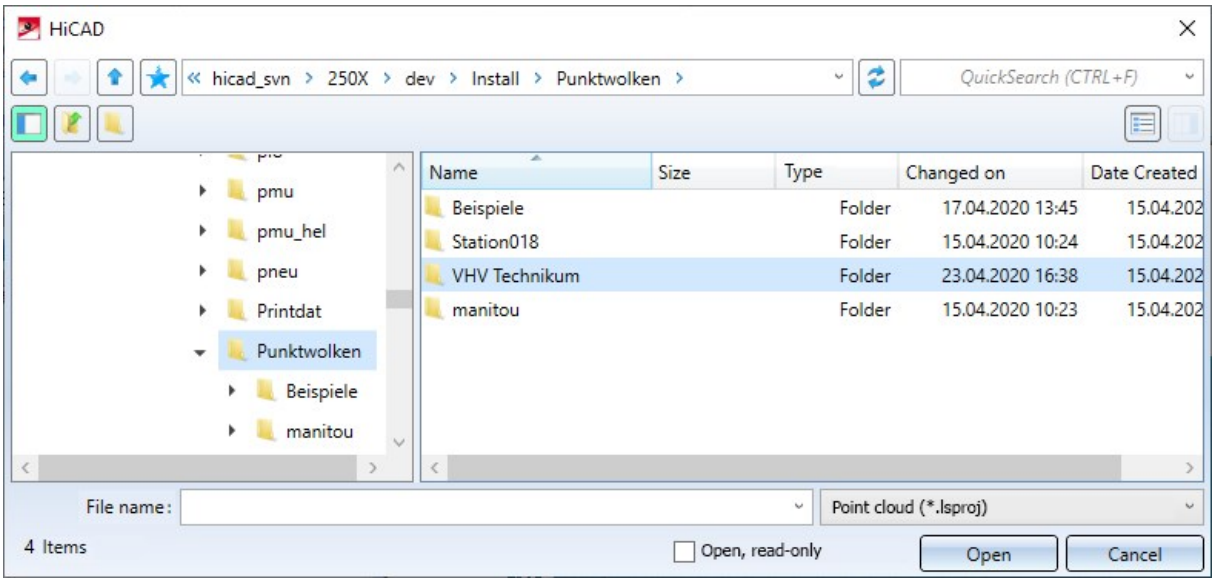
Service Pack 1 2020 (V 2501)

Update point cloud references

New on the **Point Cloud** Ribbon tab is the **Update point cloud references** function.



This function allows point cloud references to be updated, for example to replace point clouds imported into a model drawing with other point clouds or to change the path where the point cloud was stored.



Select the new point cloud or point cloud path in the dialogue window and click **Open**.

The point cloud will be replaced or the point cloud reference will be updated.

Improved updating of point clouds

HiCAD model drawings or parts with point clouds do not contain the point cloud data but only a reference to the corresponding project point cloud, i.e. to the path in which the point cloud is located during import. If you transfer files with point clouds to a third party, i.e. to another HiCAD workstation, then you must also pass on the folder with the project point cloud in addition to the SZA or KRA file. At the same time, it had to be ensured that this folder also existed on the "new workstation". From HiCAD 2020 SP1 on, the updating of point cloud references when opening a corresponding SZA or KRA file has been improved.

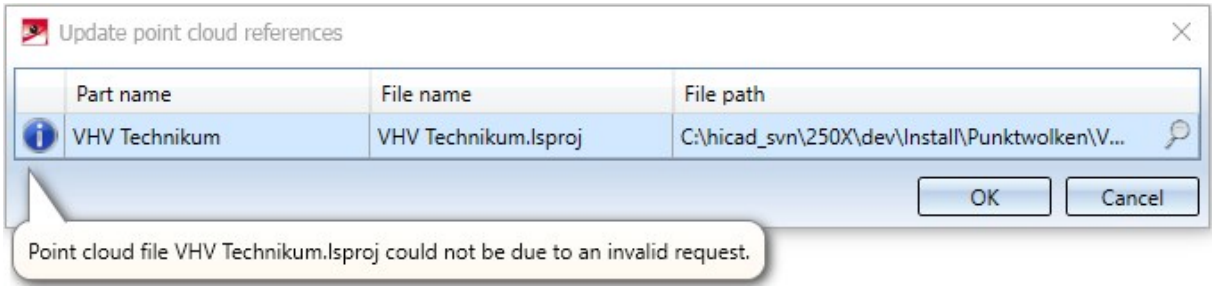
- If the directory set as drive N in the filegrup.dat is part of the point cloud path, the point cloud will be found.
- If the directory set for drive N is moved, the point cloud is still found within this folder. However, this applies only if the point cloud has been added in a version from HiCAD 2020 SP1 onwards. This does not apply to already existing model drawings created with HiCAD 2020.

An example:

The directory C:\HiCAD\Point clouds is set as drive N. The point cloud file VHV Technikum.lsproj is located in the folder C:\HiCAD\Punktwolken\Examples\User\VHVTechnikum The model drawing with the point cloud is saved.

The model drawing is then opened on a computer on which the same directory is set for drive N. The point cloud file VHV Technikum.lsproj is located there in the folder C:\HiCAD\Punktwolken\VHVTechnikum. **-> The point cloud is found**.

If HiCAD does not find a point cloud contained in a model drawing when loading it, the **Update point cloud references** dialogue window is automatically displayed. If you move the cursor over the symbol, further information will be displayed.



After clicking on the magnifying glass symbol and selecting the desired path / file, close the dialogue window with **OK**.

The point cloud reference will be updated and the point cloud will be exchanged if necessary.

Point from point cloud

- The point option **(PW) Point from point cloud** is also available in the 2-D Point options menu.
- The point option can also be accessed from the keyboard by pressing the **C** key.

Point Cloud Converter

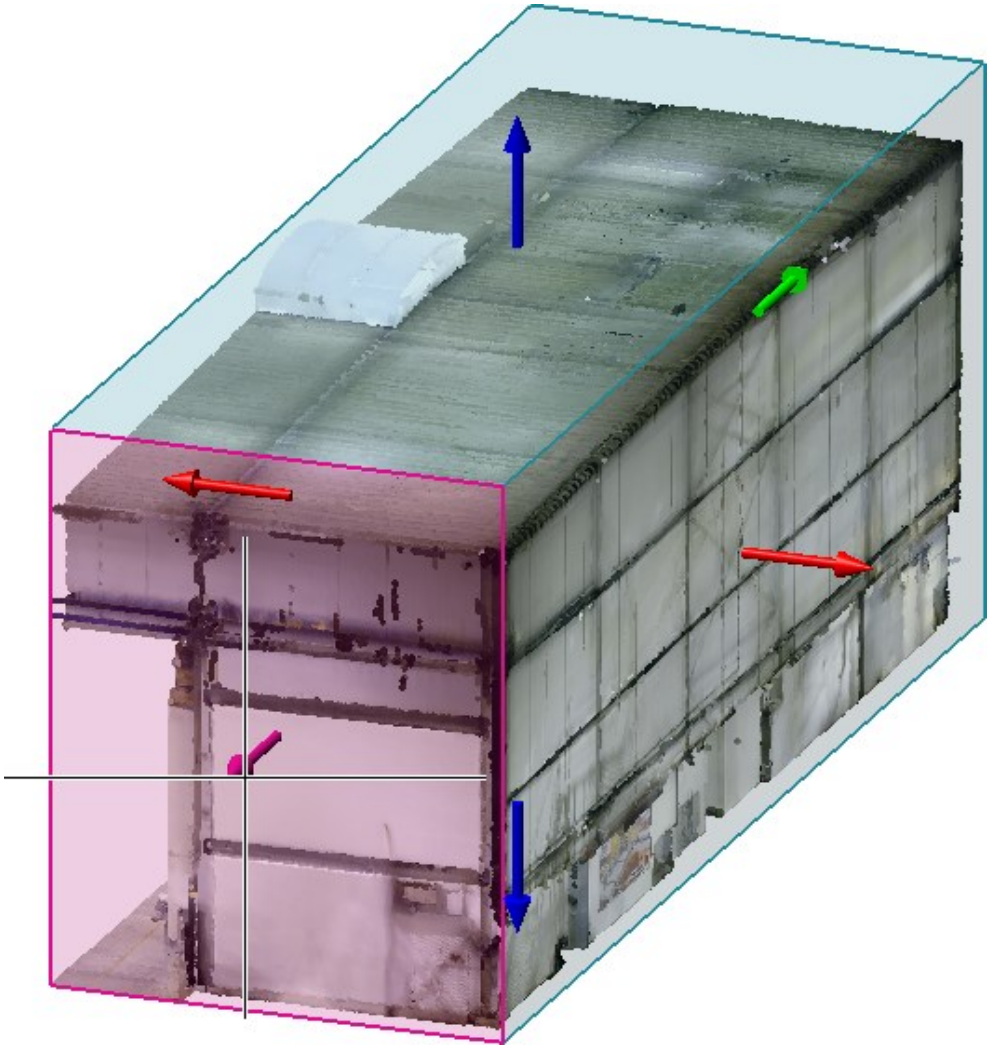
When calling the **Point Cloud Converter** (PointCloudConverterConsole.exe) from the command line, the parameter for colour/intensity of point clouds is no longer available

```
C:\hicad\exe>PointCloudConverterConsole.exe /?
Possible parameter values
Allowed options:
  -? [ --help ]                    produce help message
  -p [ --project ] arg             full path of the new Project
  -i [ --input ] arg               full path to file which should be
                                   converted
```

The import of E57 files uses the colour by default.

Improved clipping

When creating and editing clipping boxes, if you point the cursor at a directional arrow, the corresponding area will now be marked and you can reduce or enlarge the box by dynamically dragging the area in the selected direction.



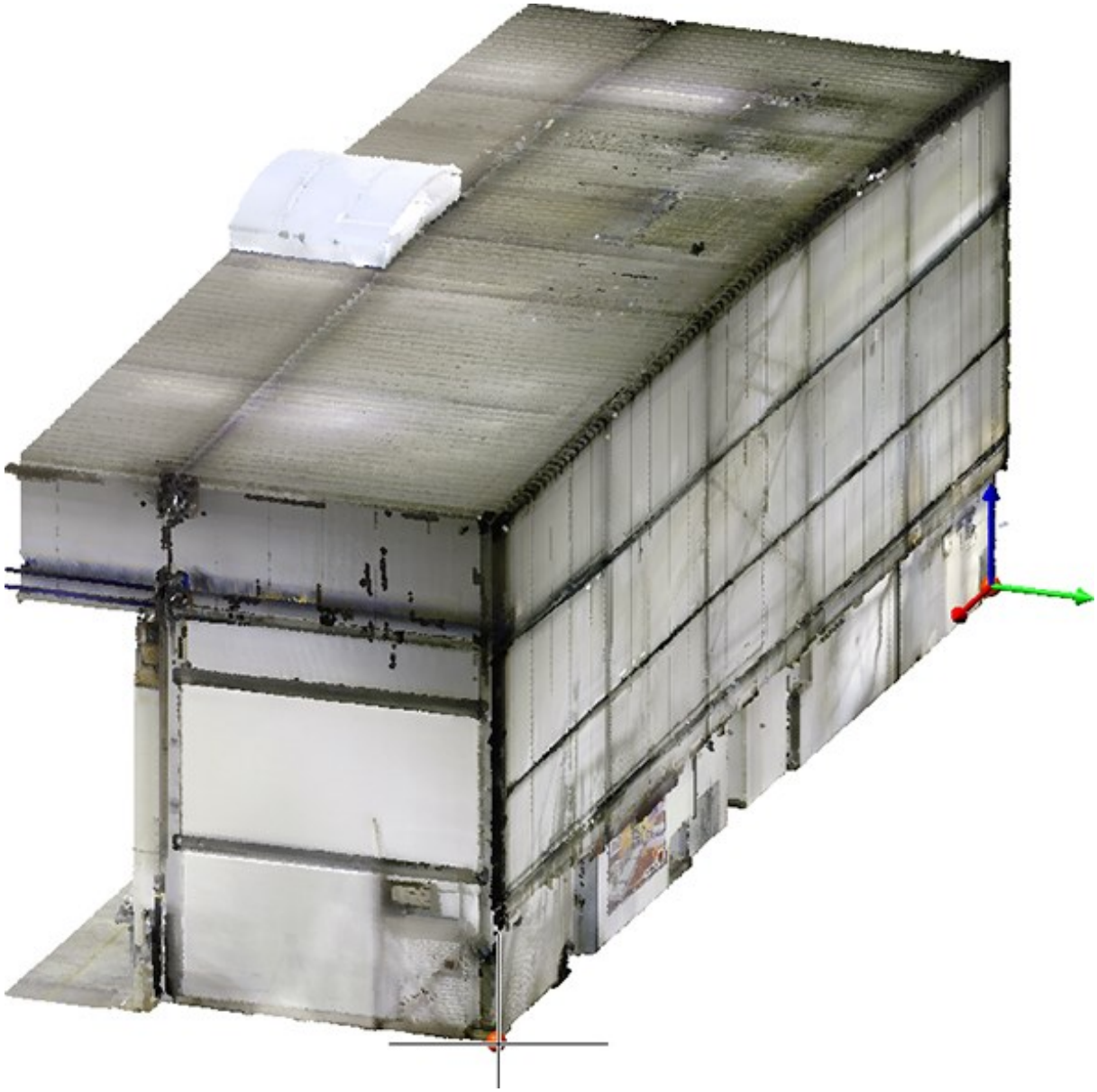
Fitting direction determination for import

To align the point cloud in the construction, HiCAD now requests the determination of two point cloud points after importing a point cloud. These are always located on the visible part of the scanned object. These points, together with the Z-direction of the point cloud, define the part coordinate system. To determine the point cloud points, HiCAD automatically activates the point option (PW) Point from point cloud.

1. Specify the first point cloud point. This defines the origin of the Part CS.
2. The second point cloud point determines the X direction of the Part CS.

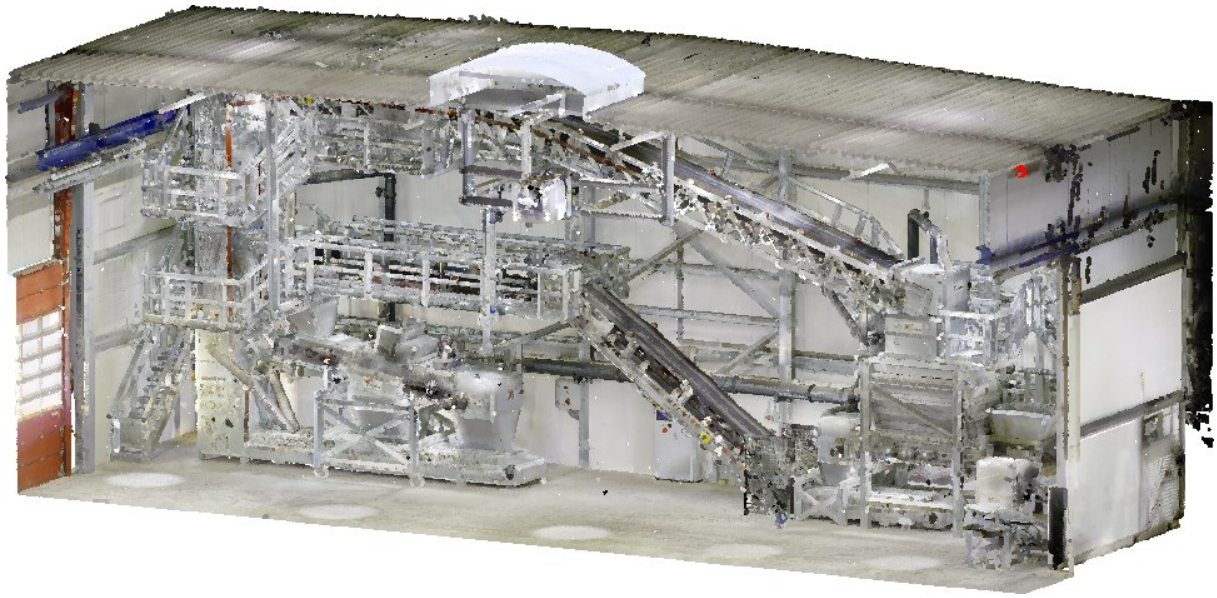
The determination of the two points and the installation direction are graphically visualized in the model drawing. Point cloud points are represented by a red point.

Determining of first point (Image: VHV Anlagenbau GmbH, Hörstel)



Determining of second point and visualisation of fitting direction (Image: VHV Anlagenbau GmbH, Hörstel)

After determining the points, the point cloud is rotated so that the Part CS coincides with the World CS.



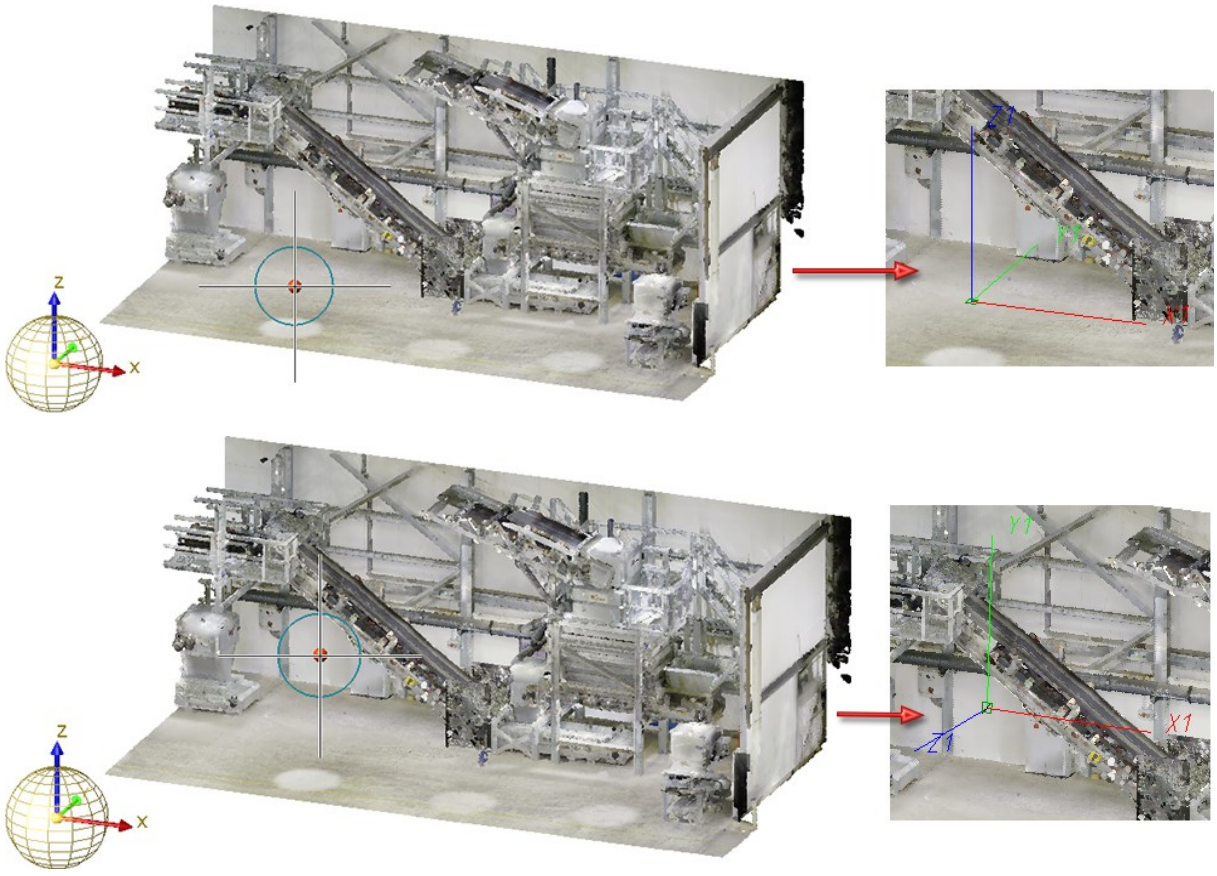
Example of a point cloud imported to HiCAD with Part CS (Image: VHV Anlagenbau GmbH, Hörstel)

Plane from point cloud

The new **Select plane from point cloud** function allows you to place a processing plane directly in a point cloud. This simplifies design and machining in the spatial view, for example, if a new building is to be fitted into an environment or a plant is to be installed in a room.

To define a processing plane, move the cursor to the desired position in the point cloud. HiCAD symbolizes the area around the current cursor position by a circle. By clicking with the mouse, the marked area is used to define the processing plane. Proceed as follows:

- If the plane is more vertical, then the X-axis is chosen so that it is perpendicular to the Z-axis of the world coordinate system. The Y-axis is chosen so that it forms a positive scalar product with the Z-axis of the world coordinate system, i.e. the angle is between 0° and 90°.
- If the plane is more horizontal, then the X- and Y-direction of the world coordinate system are taken over.



Please note:
When determining the processing plane you should make sure that the circle around the click point contains only points of the desired processing plane. To avoid unwanted results, it may be useful to zoom in before determining the click point. You can also use the mouse wheel while the circle is displayed.

Click to open a pull-down menu with the function **Plane from point cloud, with X-axis** . Here you define the plane by two additional points - the origin and a point on the X-axis.

Feature Technology

Service Pack 2 2020 (V 2501)

Updating dependent parts in locked assemblies upon loading

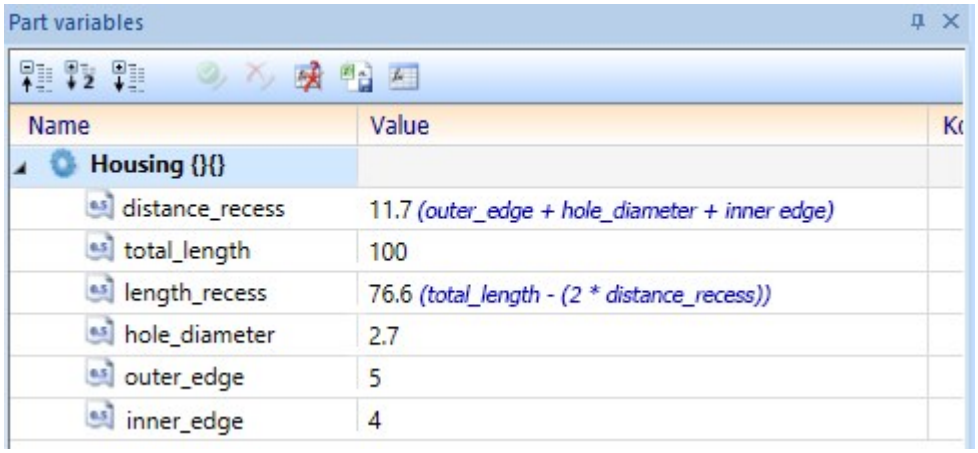
If a dependent part is contained in a locked assembly, it is now possible to have this part updated when loading the drawing. This was not possible before.

Any other update of the part is still not possible.

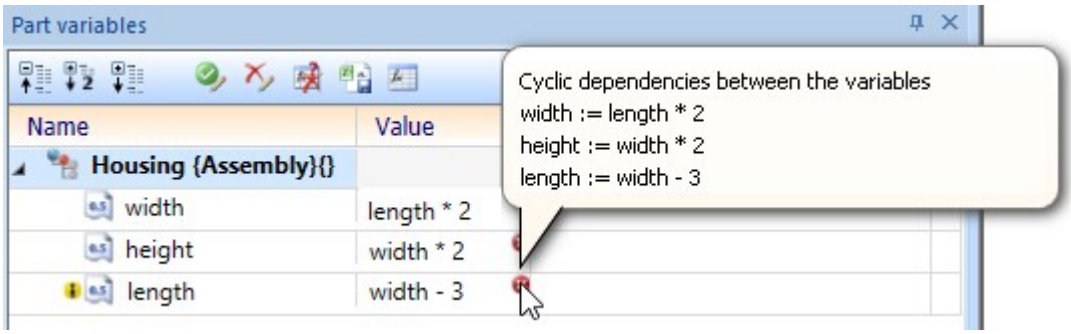
Nested variables and formulas

It is now possible to specify a formula that contains additional variables when defining the value of a variable. Especially with complex parts, the variable structure can become much clearer.

Thereby variables may be nested arbitrarily deeply into each other:



However no cyclic structures may occur (variable a refers to variable b, which refers to variable a), since these cannot be computed. In this case the automatic calculation of the formula values is no longer possible. The last values are retained and an error mark is displayed at the affected variables, which shows an explanatory tooltip when the mouse pointer is moved over them:

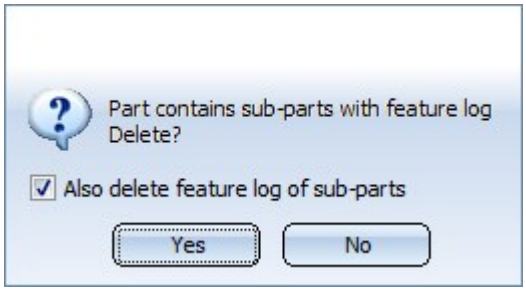


In this case, remove the cyclic dependency by editing the formulas to remove the error markings and reactivate the automatic calculation of the formula values.

Delete feature log of part with sub-parts

For instance, if the feature log of an assembly is deleted through the **Delete feature log** function, then the feature log of all subordinate parts is always deleted as well.

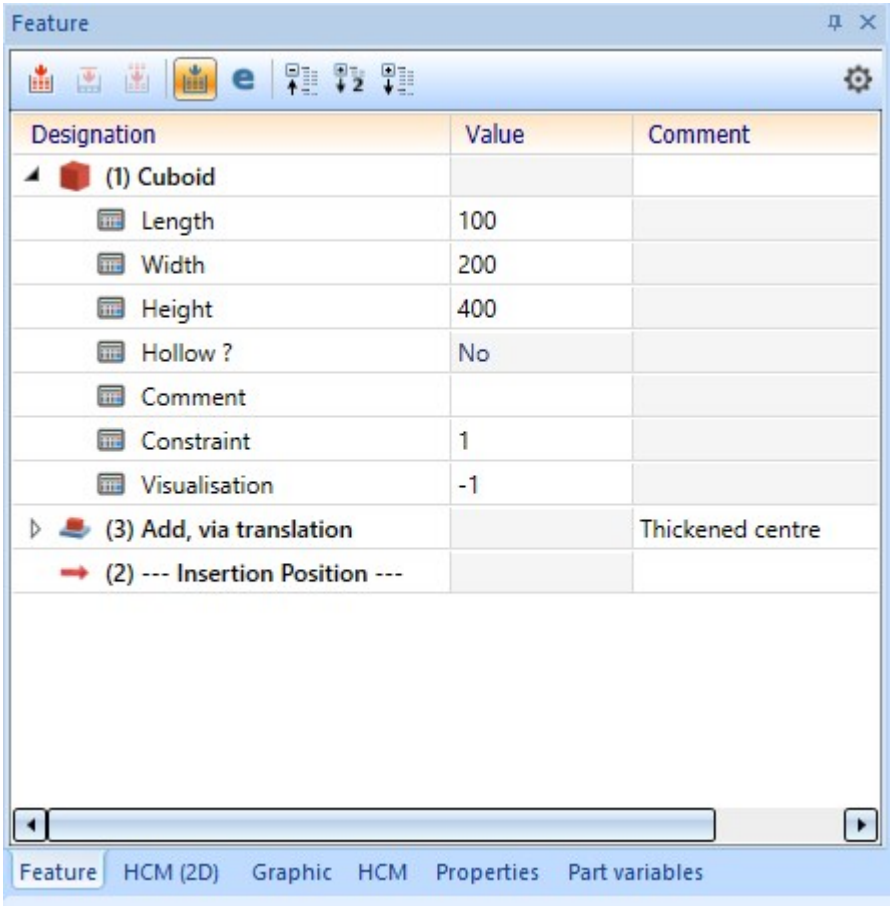
Service Pack 2 and higher now displays the following information dialogue:



This dialogue allows you to cancel the deletion process by negating the query. In addition, you can choose the **Also delete feature log of sub-parts**option to have the feature logs of all sub-parts deleted at the same time (which was previously the default) - or you can leave them as they are by deactivating the option.

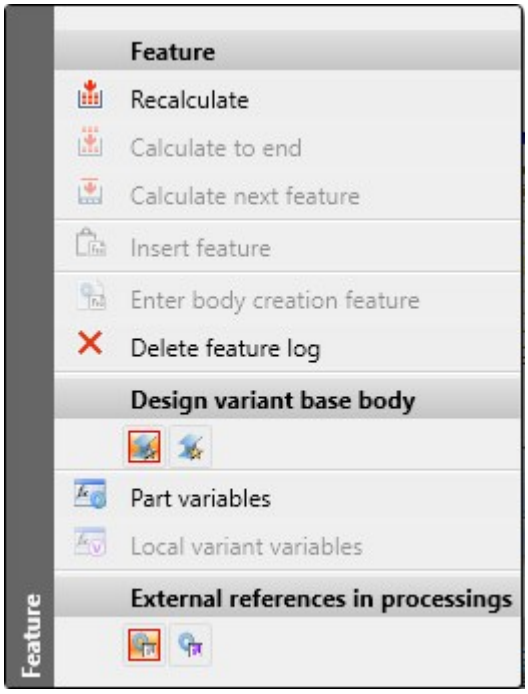
Redesigned display of feature log

The window for displaying the feature log in the ICN has been revised and now presents itself in the form of a table analogous to the newly designed 3D part structure since HiCAD 2020.



This change was accompanied by many additional convenient functions. For example, the toolbar now not only provides direct access to the functions **Recalculate**, **Calculate next feature** and **Calculate to end**, but the options **Automatic recalculation** and **Use external references** can also be directly activated and deactivated here.

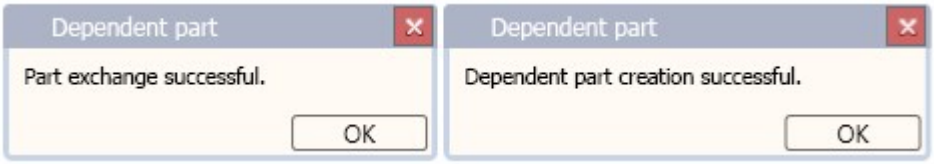
Also the context menus have been reworked and now have a new design.



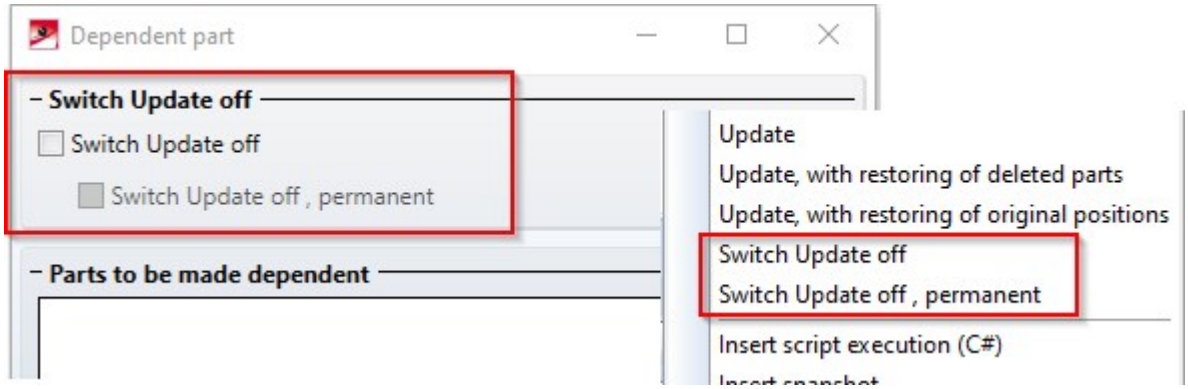
Service Pack 1 2020 (V 2501)

Improvements for dependent assemblies

- The **Dependent part** function in the **3-D Standard** tab is now greyed out as long as no part is selected to which this function can be applied.
- In addition, an information window appears after the function has been executed. This window disappears automatically even without clicking **OK** in order not to interrupt the workflow.



- The functions for temporary or even permanent switching off the update of dependent parts can now not only be found in the context menu of the feature entry but can also be activated after double-clicking the feature entry in the **Dependent part** dialogue window.



Weld seam tags for automatically derived drawings without feature log

When the feature log is not activated, weld seam tags are now also created for automatically derived drawings. Previously, when automatically deriving a drawing via an API variant such as the **Staircase Configurator** or the **Cross-bracings**, it was possible that weld seam tags could not be created for all welds seams.

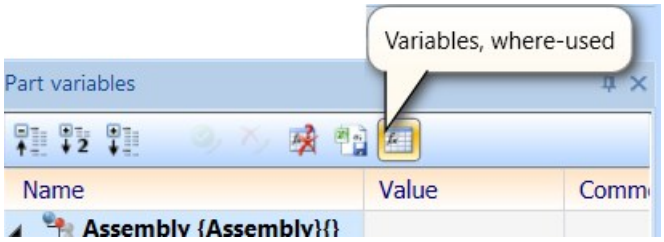
This behaviour has now been corrected so that corresponding weld seam tags are generated for all weld seams as expected. For older model drawings, it may be necessary to call the **Recalculate feature** function once before deriving the drawing.

Major Release 2020 (V 2500)

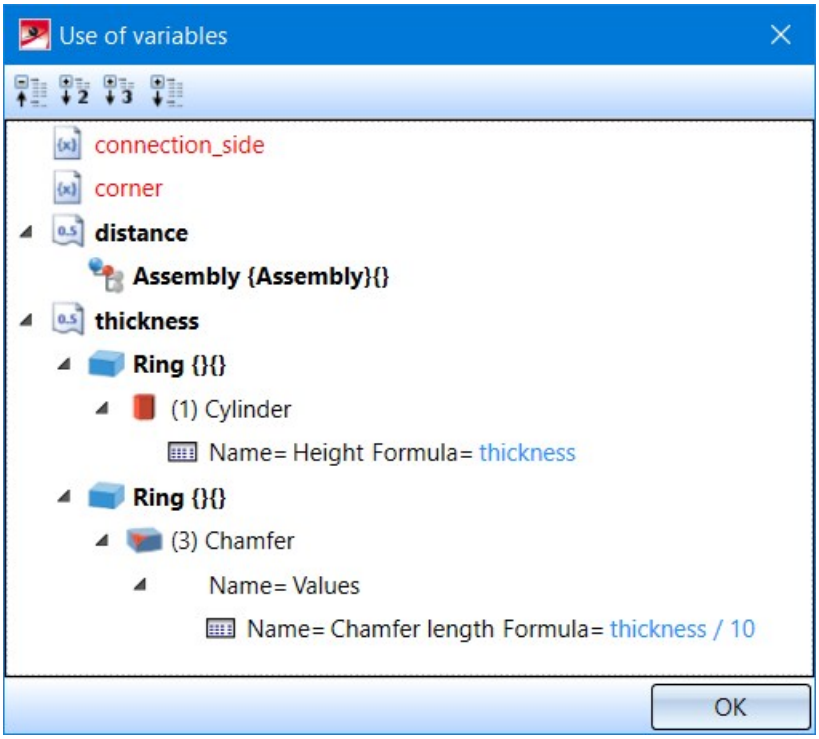
Further improved Use of variables dialogue window

The **Use of variables** window has been revised, now offering even more comfort in its use. The modifications are as follows:

- A button is now available in the toolbar of the **Part variables** docking window, which directly opens the **Use of variables** dialogue window.



- Icons are now displayed in the **Use of variables** window: The icon preceding a variable indicates the type of the variable; in the same way, the respective icons are displayed preceding the parts, features and feature parameters.
- Unused variables are displayed in red.
- Formulas containing the respective variable are displayed in blue.

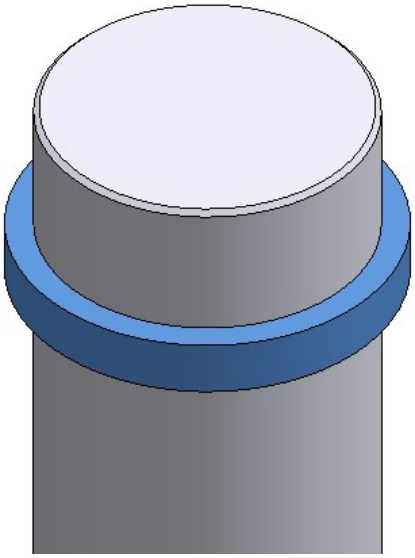


Access to variables of other parts

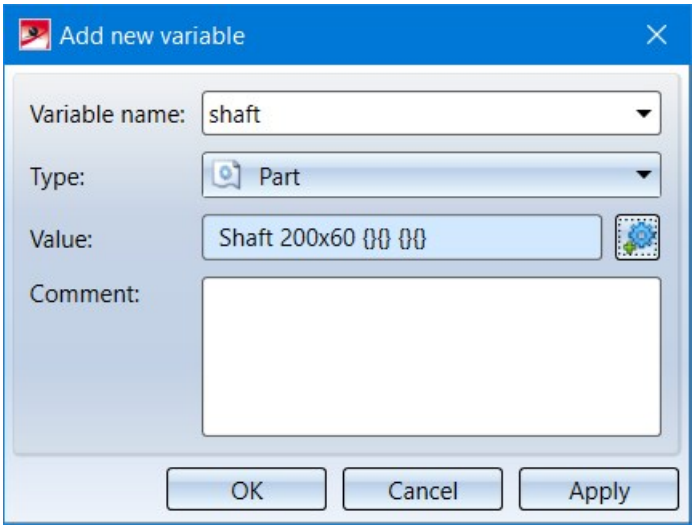
It is now possible to access variables of any other parts within formulas. This allows part variables to be used independently of the part structure. In particular, the variables of a part can be accessed within views.

Example:

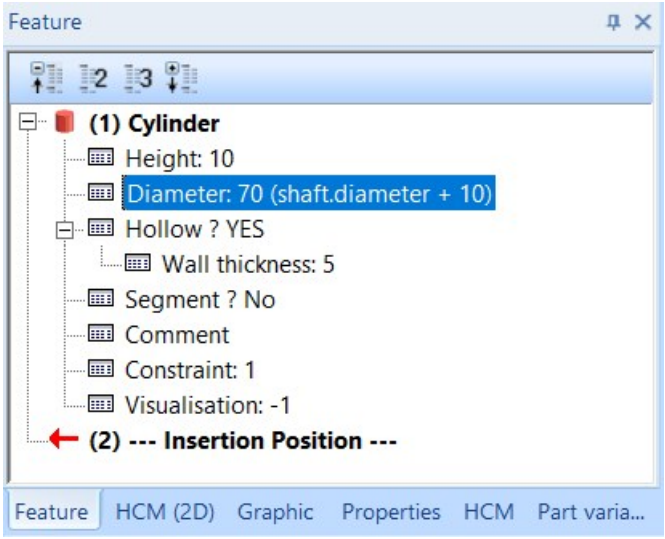
The diameter of the ring should depend on the diameter (part variable `diameter`) of the shaft.



For this purpose, the ring contains a part variable of type part with the name `shaft` that refers to the shaft.



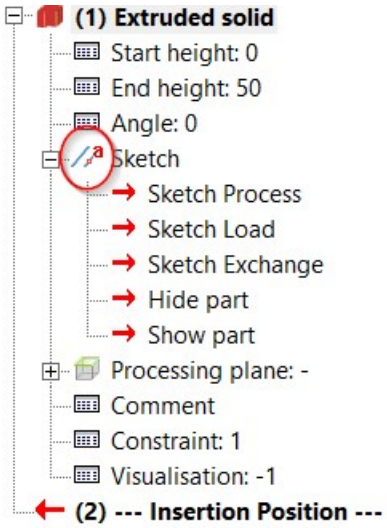
The value of the variable `diameter` of the part `shaft` can then be accessed in a formula by using `shaft.diameter`.



If the ring is now to be mounted on another shaft, only the `shaft` variable must be adjusted so that it refers to the new shaft (this must, of course, contain a `diameter` variable, otherwise the formula cannot be evaluated).

Display of sketches with external references

For entries in the feature log that refer to sketches, the sketch icon now directly indicates whether the sketch contains external references.



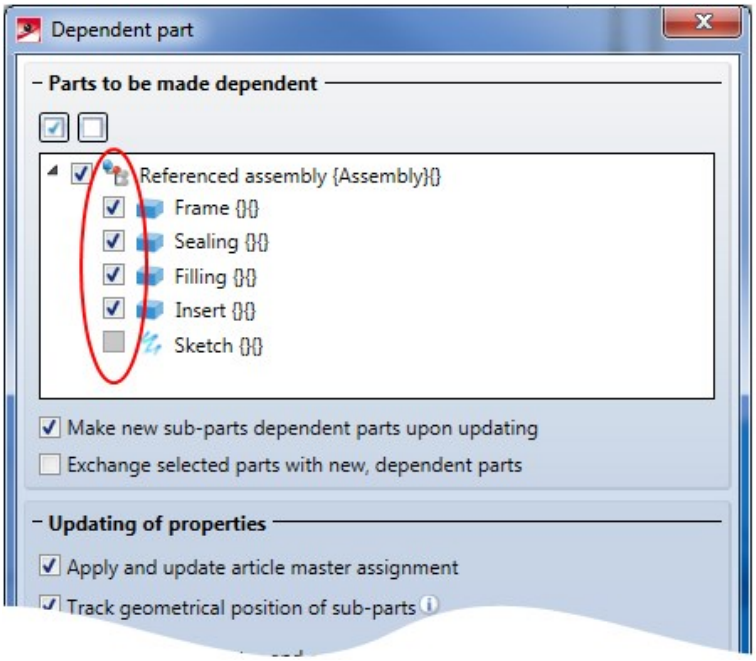
- symbolizes a sketch with external references that were actually used.
- symbolizes a sketch that contains external references that were not used, for example, because the use of external references was deactivated.

With existing drawings it can happen that the symbol is only displayed correctly after a feature recalculation.

Merging of "Dependent part" and "Dependent assembly" function

The functions **Dependent part** and **Dependent assembly** have been merged into one, new function. This new function is called **Dependent part**which, however, behaves like the former **Dependent assembly** function, while offering the following, additional options:

- You can now also choose single parts as base parts, as long as they are referenced.
- You can choose whether the selected parts are to be exchanged with the dependent parts, or if the latter are to be additionally created.
- The function can now be found in the **New** function group of the **3-D Standard** tab of the Ribbon.
- In the part selection list, the icons for the respective part types are now also displayed.



New API functions

Two new functions are available for working with API variants:

- `Sketch.CreateAutoConstraints()` corresponds to the calling of the function **Sketch > HCM > Coinc... > Automatic positional constraint**.
- `Sketch.CreateAutoDimensions()` corresponds to the calling of the function **Sketch > HCM > Smart dimensioning > Automatic dimensional constraint**.

In both cases the selected function will be carried out with the default settings, i.e. without any further queries or offering any configuration options.

Weld seam tags without Feature log

It is now possible to create and edit weld seam tags even without an activated Feature log. Up to now, when automatically deriving a drawing to an API variant such as the **Staircase Configurator** or the **Cross-bracing**, it was possible that weld seam tags could not be created for all weld seams

For older constructions, it may be necessary to call the **Recalculate feature** function once before deriving the drawing.

Parametrics (HCM)

Service Pack 2 2020 (V 2502)

More meaningful error messages for inconsistent HCM models

When you performed an action that makes an existing HCM model inconsistent, a rather general error message used to appear:

In HiCAD 2020 SP1 some errors have already been provided with new error messages. In Service Pack 2, further cases are added:

- If a constraint is generally not possible (e.g. Coincidence between two cylinders with different radii).
- When a part is moved to another assembly that is not included in the current assembly. In this case, the HCM constraint is now moved to a suitable assembly so that it can continue to be active. If this should not be possible, a message window appears.
- When deleting a part to which HCM-constraints refer.
- When transforming parts.
- When fixing a part that has constraints to another fixed part.
- When adding constraints that would redefine the model.
- When updating referenced parts.
- When a part is changed by a feature recalculation so that a constraint can no longer be resolved.
- When deleting an isolated point to which a constraint refers.

New Visualisation toolbar

The functions **Toggle visibility of dimensions (3-D Part HCM)**, **Toggle visibility of degrees of freedom (3-D Part HCM)**, **Toggle visibility of constraints (C-Edge-HCM)** and **Toggle visibility of degrees of freedom (C-Edge-HCM)** have been removed from the Ribbons **3-D Standard**and **Sketch** , and can now be found on the new **Visualisation** toolbar.

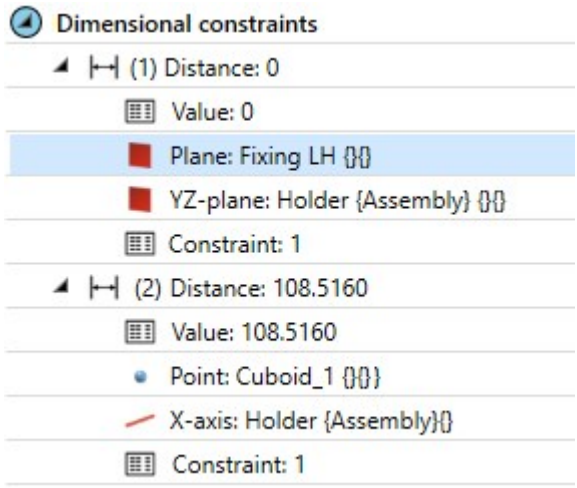
Visualization of the sketch plane

For sketches in the model drawing, the sketch plane belonging to the sketch is always displayed as a green rectangle in the drawing. Until now, this rectangle was always dimensioned so that all line elements of the sketch fit into it. However, HCM dimensions and symbols were not taken into account, so they were displayed partly on and partly off the sketch plane.

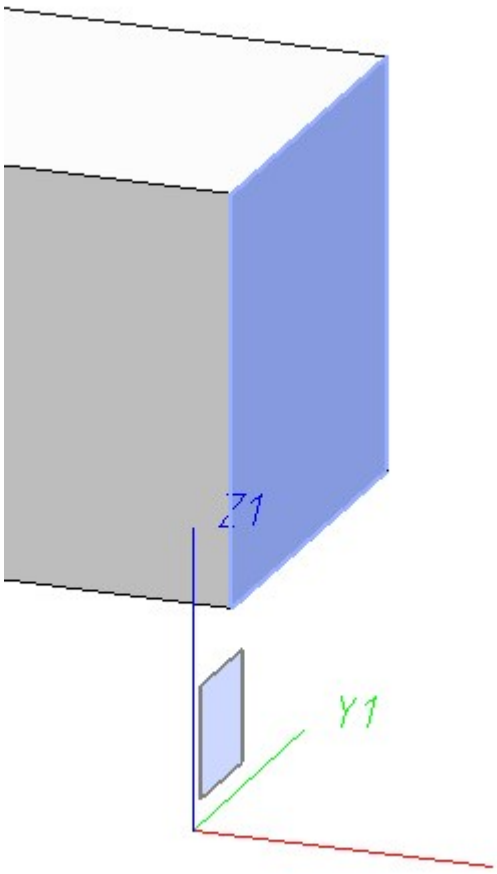
This behavior has now been corrected so that these entities are also completely enclosed by the rectangle in the sketch plane.

Coordinate system elements in the ICN

HCM constraints that refer to a coordinate system element such as a plane or an axis are now displayed in the ICN in a shortened form as "XY-plane" instead of e.g. "XY coordinate system plane". Similarly, axes are now displayed as "Z-axis" instead of "Z coordinate system axis". This makes the listing of conditions in the ICN more compact and clearer.



In addition, the affected coordinate system elements are now also highlighted in the construction when a corresponding entry in the ICN is selected.

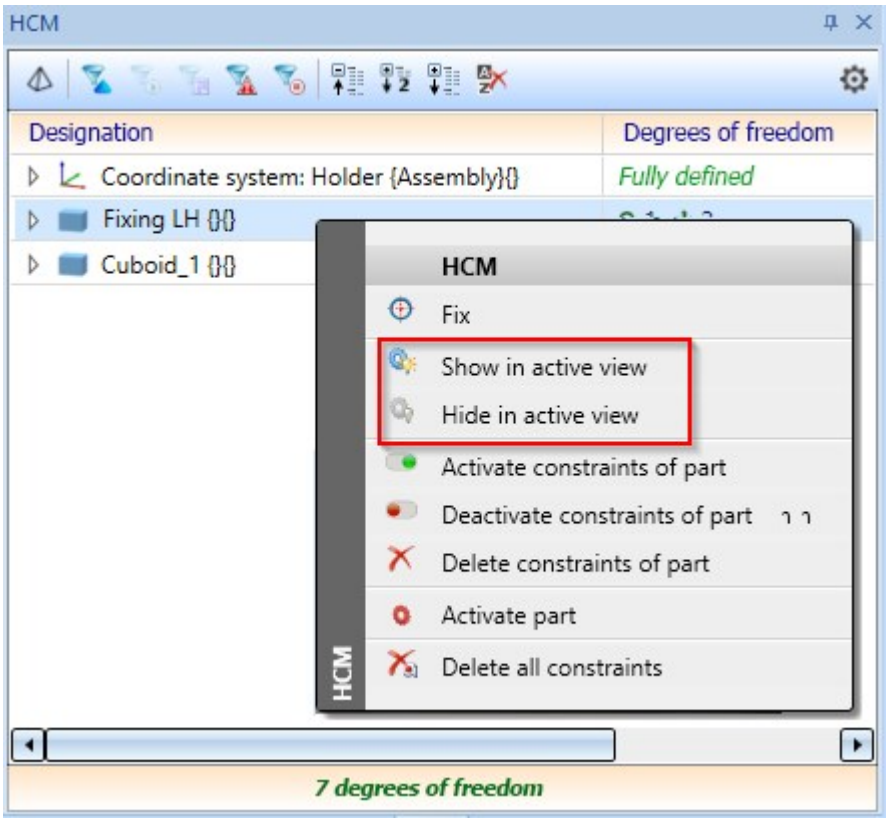


Stable area distance

For distance constraints with area definition, the distance may be changed by the HCM within this definition. Up to now, the HCM has often used this option. However, as of SP2, the distance is only changed if there is no other way to calculate the HCM model.

Show/hide parts in the ICN

If the HCM-ICN is set to display **By geometries**, the **Show in active view** and **Hide in active view** functions can now be found in the context menus of the parts. This allows for quick and convenient part display and hiding without having to search through the part structure first.



Service Pack 1 2020 (V 2501)

Part HCM

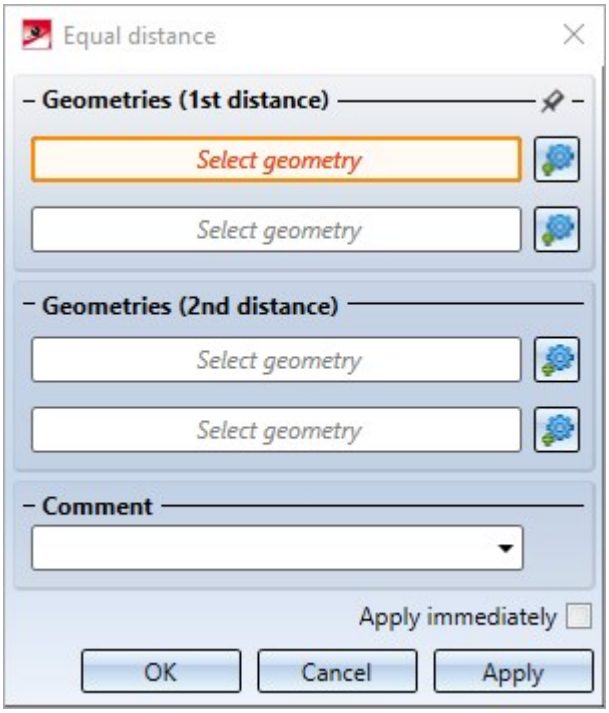
Improved placing of HCM dimensions

The algorithms for offsetting HCM dimensions have been improved to improve the offsetting of HCM dimensions. The improvements include:

- Dimensions in 3-D sketches now align with the sketch coordinate system, not the world coordinate system.
- For surfaces, if possible, the base points of dimensions are now selected so that no bend in the dimension line will be necessary any more.
- The plane direction for dimensions is now selected as for drawing dimensions.
- The same context menu as for drawing dimensions is available when placing dimensions.

Creation of Equal distance and Equal angle constraints via dialogue window

The Part HCM constraints **Equal angle** and **Equal distance** are now also created via the dialogue. The pin in the upper right corner of the dialogue (/) can be used to keep the 1st distance and/or the 1st angle even after clicking **Apply**. This allows for the convenient creation of several equal constraints one after the other.

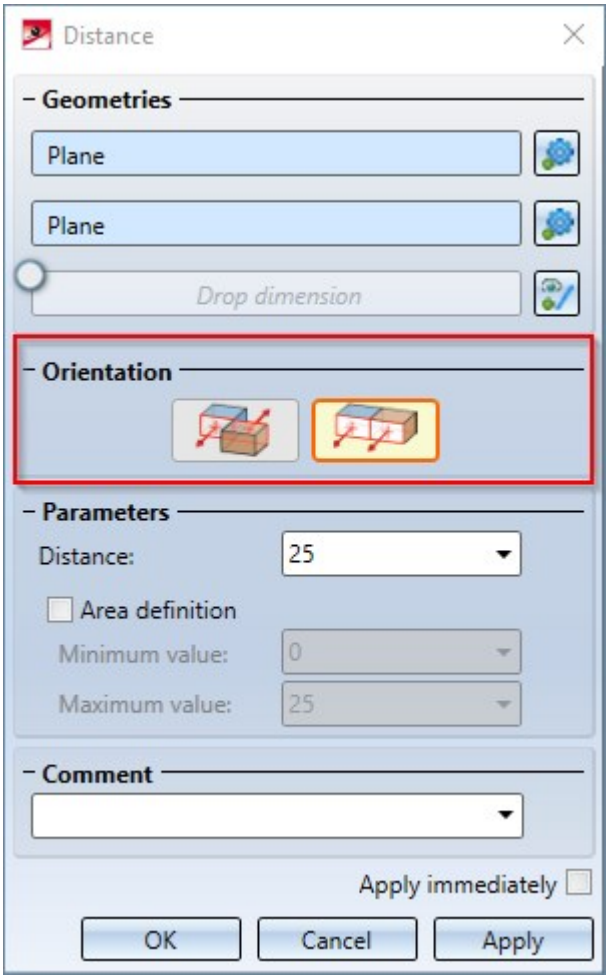


Changing constraints

To change a constraint in the 3-D Part HCM, the corresponding dialogue window can now be opened by double-clicking on the constraint in the ICN. In this way, changes can now be made directly in the dialogue window.

Part HCM: Display and change of orientation of a constraint via dialogue window

The HCM constraints **Coincidence**, **Distance**, **Parallelism** and **Concentricity** dialogue windows now contain an additional **Orientation** section. Here, after selecting suitable geometries, the current orientation is displayed. By clicking on the respective button, the orientation can be adjusted. A real-time preview is displayed in the model drawing.

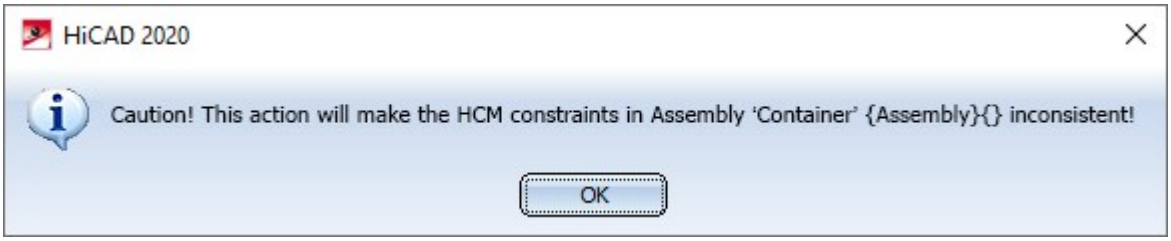


This section in the dialogue window replaces the old **Orientation** function.

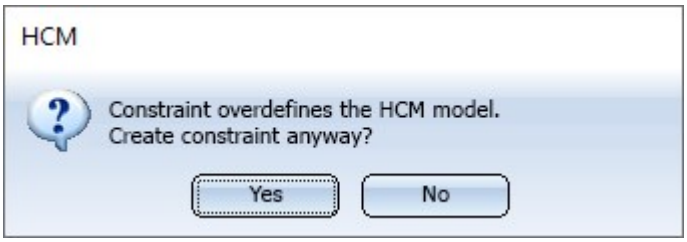
Error messages for problematic constraints

When an action such as creating or modifying an HCM constraint creates inconsistencies or other problems, a message box is now displayed to explain the error. If possible, suggestions for action to solve the problem are also offered directly:

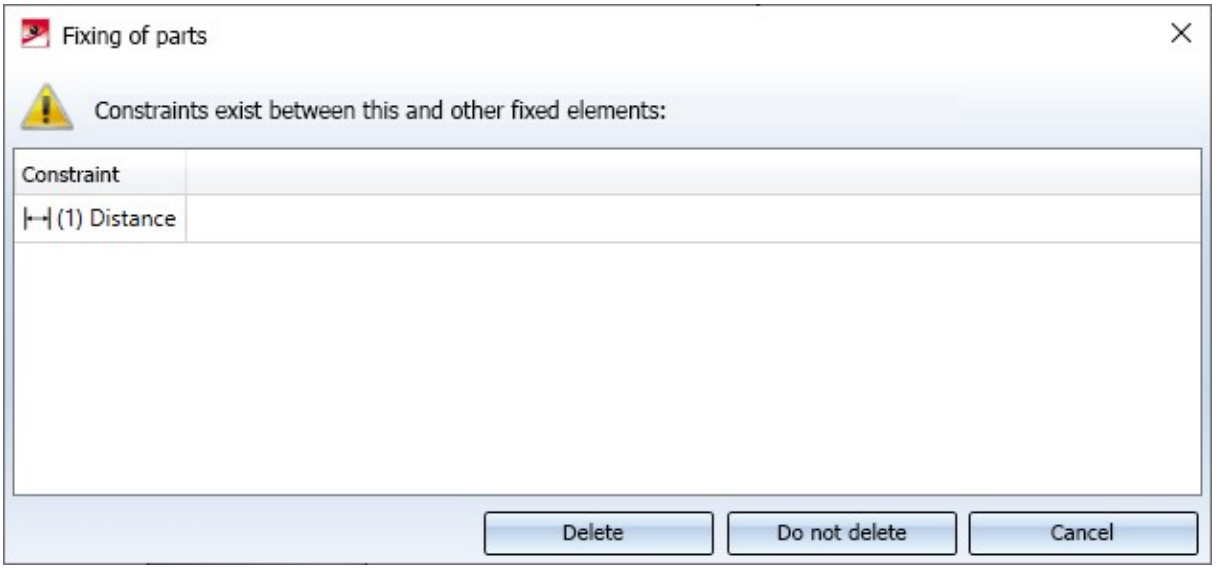
- If an action on an HCM constraint causes a superordinate assembly to become inconsistent:



- If a model is overdefined by a new constraint:



- If a part is fixed that is already bound to another fixed part via constraint, a list of constraints is displayed. You can choose between the following actions: **Delete** removes these constraints and fixes the part.**Do not delete** leaves the constraints and fixes the part anyway. **Cancel** leaves the old constraints unchanged, but does not create a new fixing either.



HCM entries in the ICN

ICN: Expanded and collapsed entries are kept

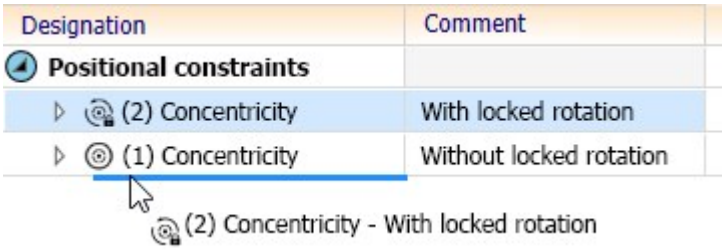
If you open or close positional and dimensional constraints of a part in the ICN, this state is now also maintained when the active part is changed. This means that when you later switch back to a part, the same entries will be expanded and collapsed as before the switch.

References to deleted parts

HCM references that refer to a deleted part now show the name of the deleted part in the ICN. This can be helpful to identify such constraints later.

Change order of constraints in ICN

The order of the HCM constraints in the **ICN** can now be changed by Drag & Drop.

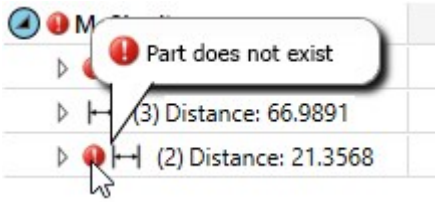


HCM constraints can only be moved within their group. Accordingly, positional constraints cannot be moved to the Dimensional constraints group and vice versa; likewise, in the **Sort by geometry** view, constraints can be moved only within their part.

Newly created constraints are always inserted at the end of the list.

Tooltips for error and warning icons

If an HCM model contains problematic constraints (for example, constraints on parts that no longer exist or over-defined parts), error or warning icons are displayed in the ICN to draw attention to these. These icons have now been added to the tooltips. This makes it easier to find out the reason for the error or warning.



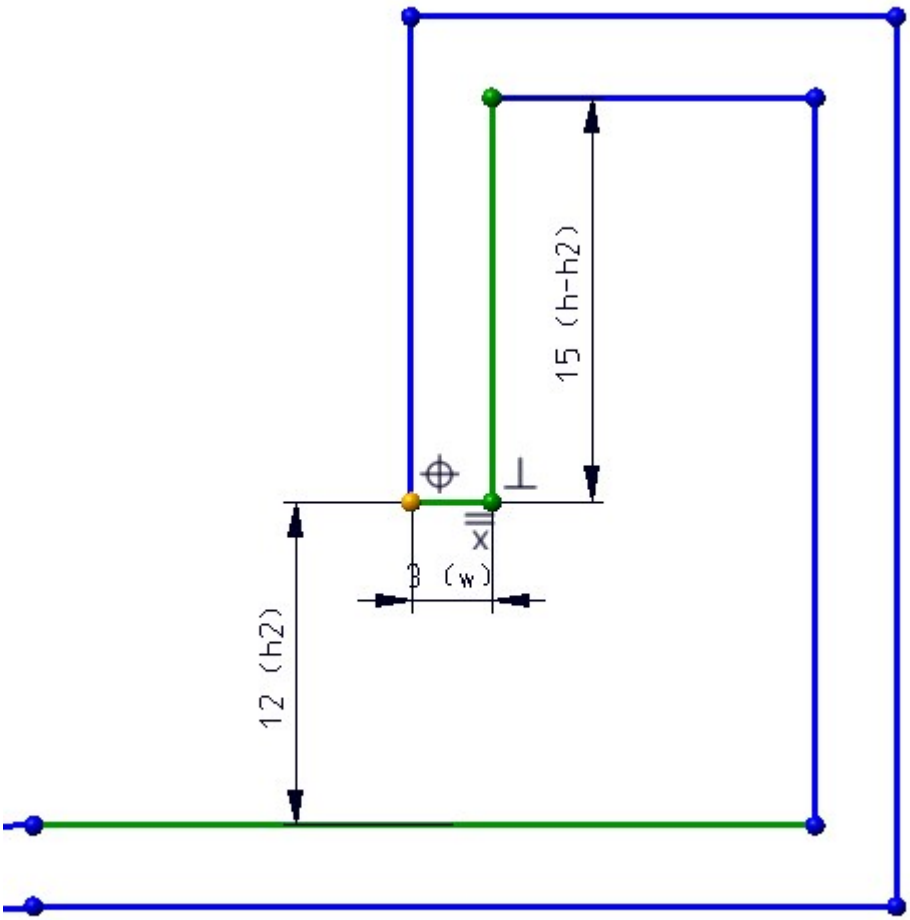
3-D sketching tool: Automatically generated HCM constraints

When using the new **Direction+Length** mode of the 3-D sketching tool the following HCM constraints are automatically created:

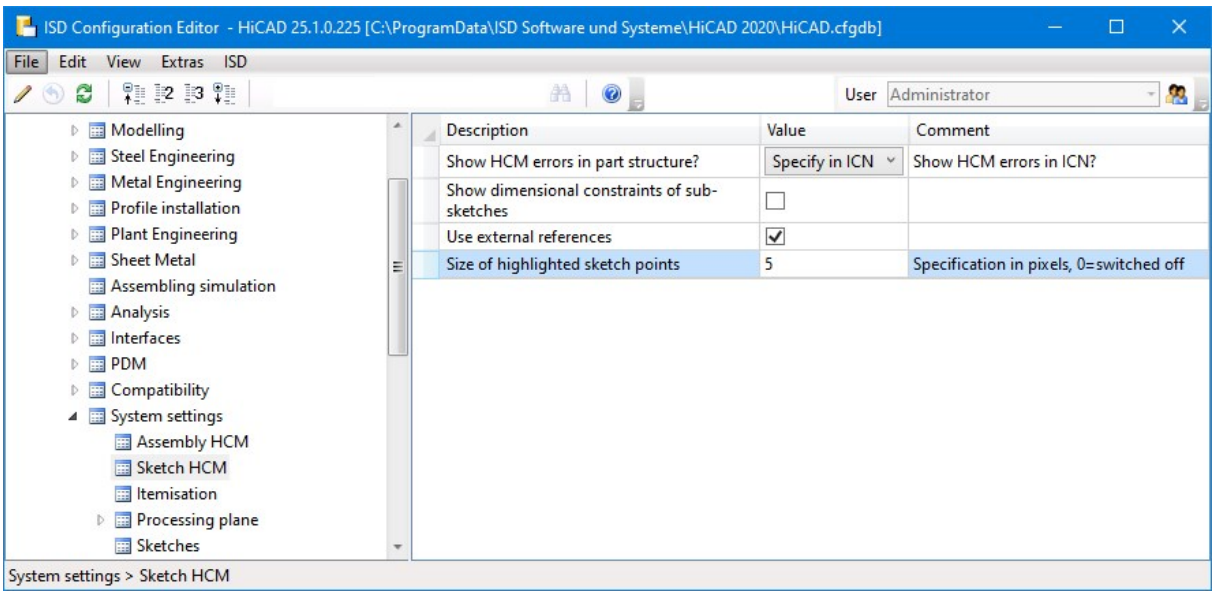
- If an existing edge or cylinder is selected as direction, a Parallelism constraint is set.
- If no parallelism to a coordinate system axis is set, a Perpendicular constraint to the plane is set.
- If the line is tangential to the last line and this was an arc, a Tangent constraint is created.

Display of points in sketches and 3-D sketches

If a sketch or 3-D sketch is selected as the active part, the end points of lines contained in the sketch are now highlighted in the model drawing, according to the logic of the HCM: Fully defined points are highlighted in green, fixed points in yellow. Overdefined points are highlighted in red. All other (underdefined) points are highlighted in blue.



The size of the point can be changed in the **Configuration Editor** at **System settings > Sketch HCM > Size of highlighted sketch points**. If you set the size to 0, the points will no longer be highlighted.



Sketch HCM

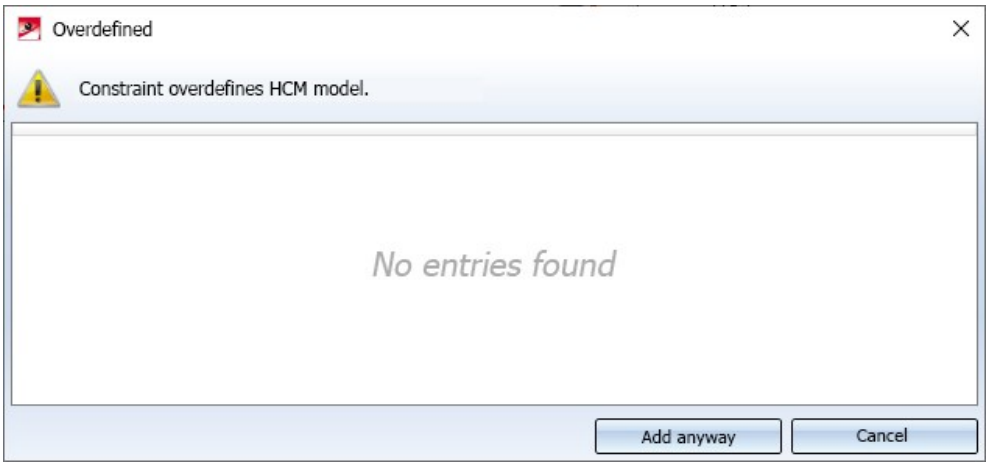
Point option M

When setting HCM constraints in sketches and 3-D sketches, the point option **M (Mid-point of line/edge)** is now offered again by the Autopilot.

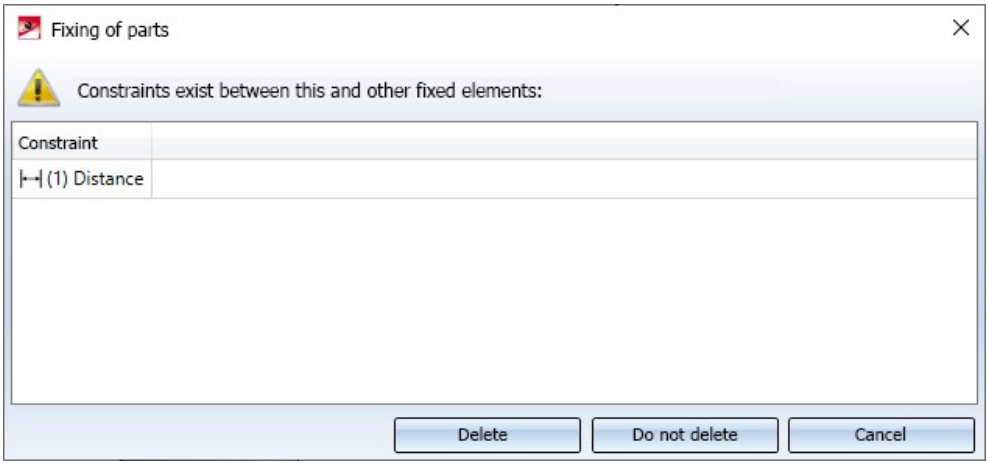
Error messages for problematic constraints

When an action such as creating or changing an HCM constraint creates inconsistencies or other problems, a message box is now displayed to explain the error. If possible, suggestions for action to solve the problem are also offered directly:

- If a condition overdefines the HCM model, a dialog indicates this problem. You have the following options for dealing with it: **Add anyway** nevertheless creates the constraint, which means that the HCM model is now overdefined. **Cancel** will not create the constraint.



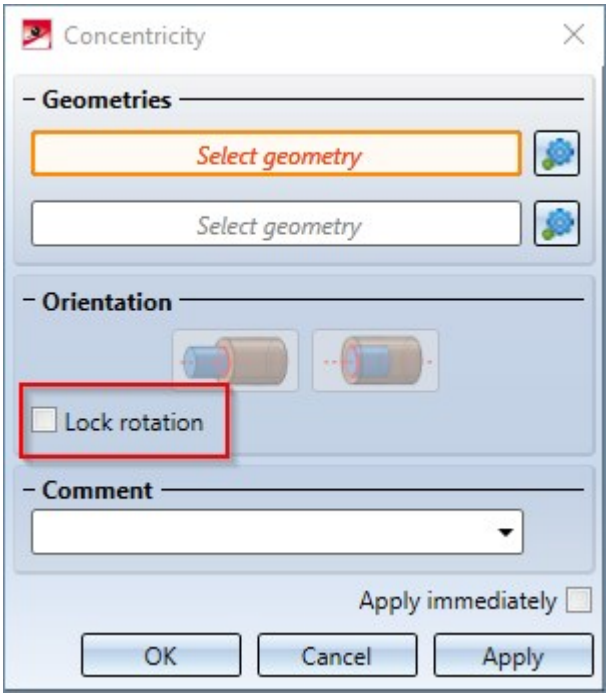
- If a sketch element is fixed that is already linked to another fixed element by constraints, a list of conditions appears. You can choose from the following actions: **Delete** removes these constraints and creates the fixing. **Do not delete** leaves the constraints and creates the new fixing. Cancel leaves the old constraints unchanged, but does not create a new fixing either.



Lock rotation for Concentricity constraints in Assembly HCM

When creating a **Concentricity** constraint in the assembly HCM, the rotation can now be optionally locked. This makes it easier, for example, to achieve full definition when screws are present.

The **Concentricity** dialogue window now features a **Lock rotation** checkbox.



The rotation lock can only be set between geometries such as two edges, but not between an edge and a point. In this case a corresponding message is displayed.

In the ICN a **Concentricity** constraint with rotation lock is indicated by a changed icon:

Designation	Comment
Positional constraints	
(1) Concentricity	Without locked rotation
(2) Concentricity	With locked rotation

In the context menu of **Concentricity** constraints in the ICN the new functions **Lock rotation for concentricity** and **Unlock rotation for concentricity** are now available.

Influence of positive/negative sign on dimensional constraints in Assembly HCM

The influence of the positive/negative sign on the direction of dimensional constraints was standardized.

- For distance constraints, at least one of the geometries must be a surface. If the distance constraint refers to several surfaces, the first selected surface is used as reference surface. A positive distance then points away from the body to which this surface belongs. Accordingly, a negative distance causes it to be measured "into the body".
- For angle constraints the current value is always positive.

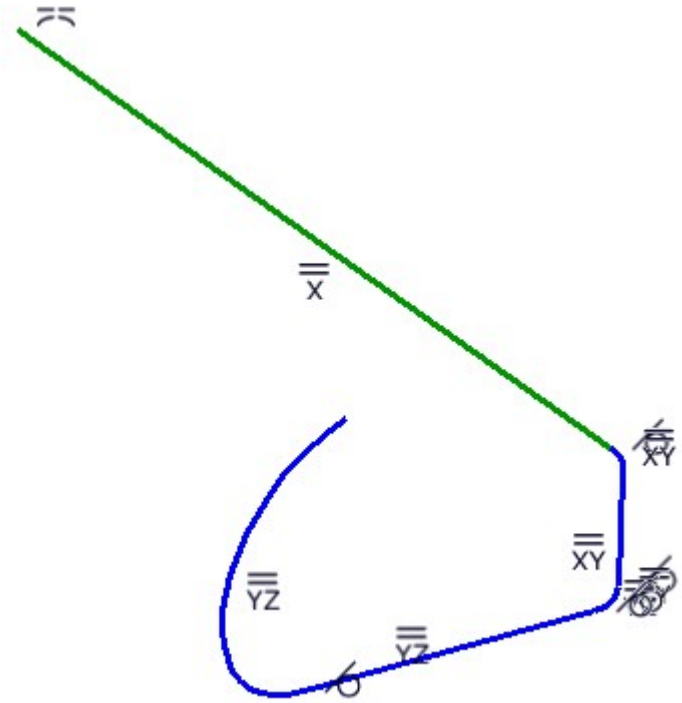
When creating a new constraint or editing an existing one, the current value is now always given with the correct sign. A change of the sign then causes the alignment of the measurement to be changed accordingly. If the sign is not changed, the alignment remains unchanged.

For distance constraints that do not relate to surfaces (for example, the distance between two points), changing the sign has no effect on the model. If you enter a negative value here, a warning is displayed.

Major Release 2020 (V 2500)

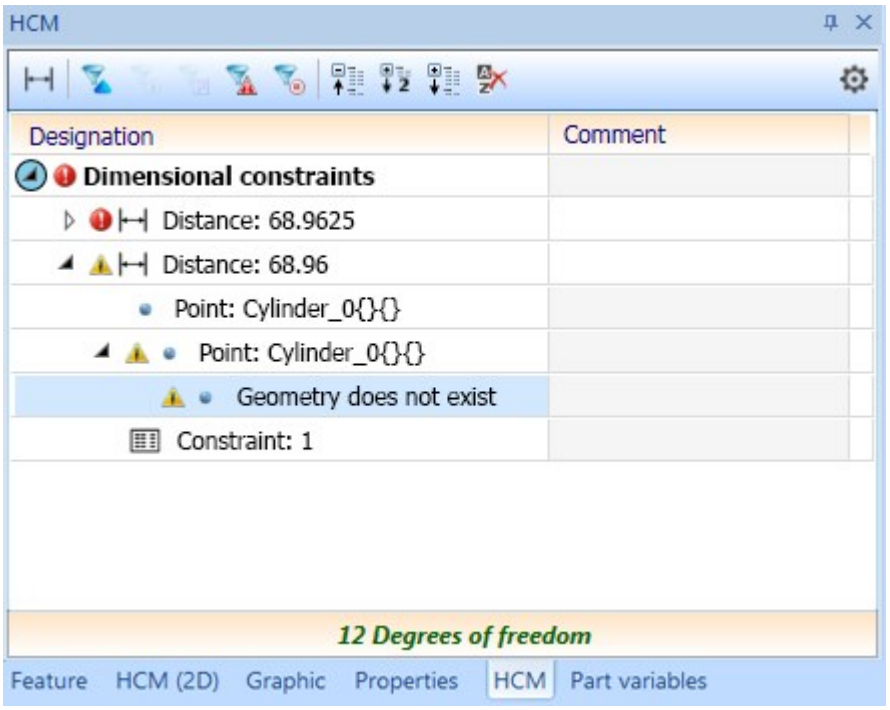
Automatic assignment of HCM constraints in the new 3-D sketching tool

The new sketching tool, which is available for the quick creation of 3-D sketches, can automatically assign HCM constraints. As an example, a short tutorial is available on how to create a 3-D sketch with HCM constraints as a basis for a handrail support in just a few steps.

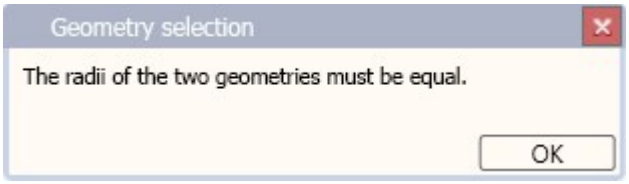


Error messages in the Part HCM

Up to now, references to missing components in the HCM were displayed as **Errors**. Now a special handling for parts that are disabled by a feature constraint has been introduced. Such parts are also listed with the message Geometry does not exist, but instead of as errors only as a **Warning**. The reason for this is that HCM constraints that refer to deactivated parts can also be considered as deactivated and therefore do not represent an error.



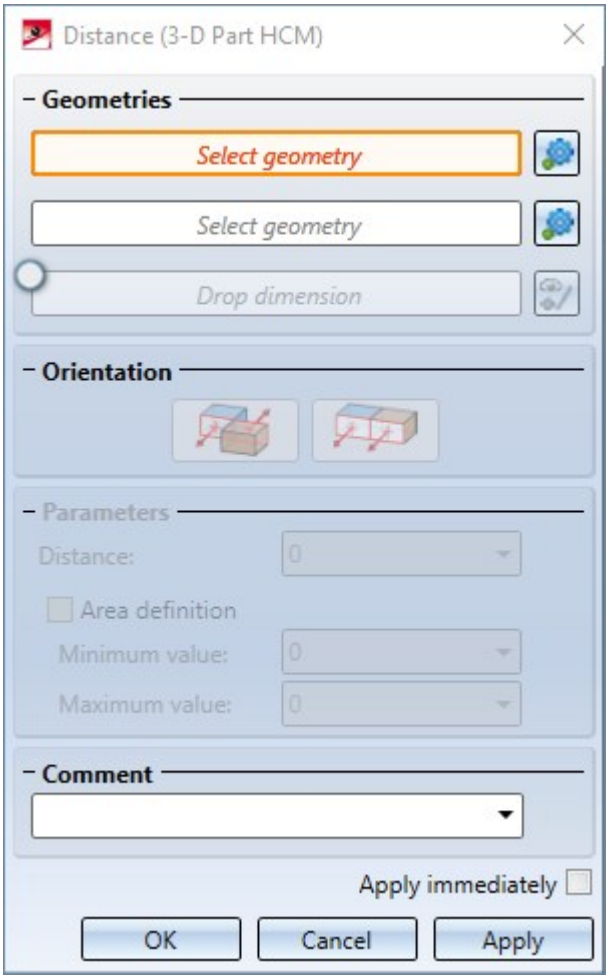
In addition, a more meaningful error message is now issued if you assign a **Coincidence** constraint between two cylinders with different radii.



Revised assigning of positional and dimensional constraints for assemblies

The assignment of positional and dimensional constraints between parts of an assembly has been revised. The changes include:

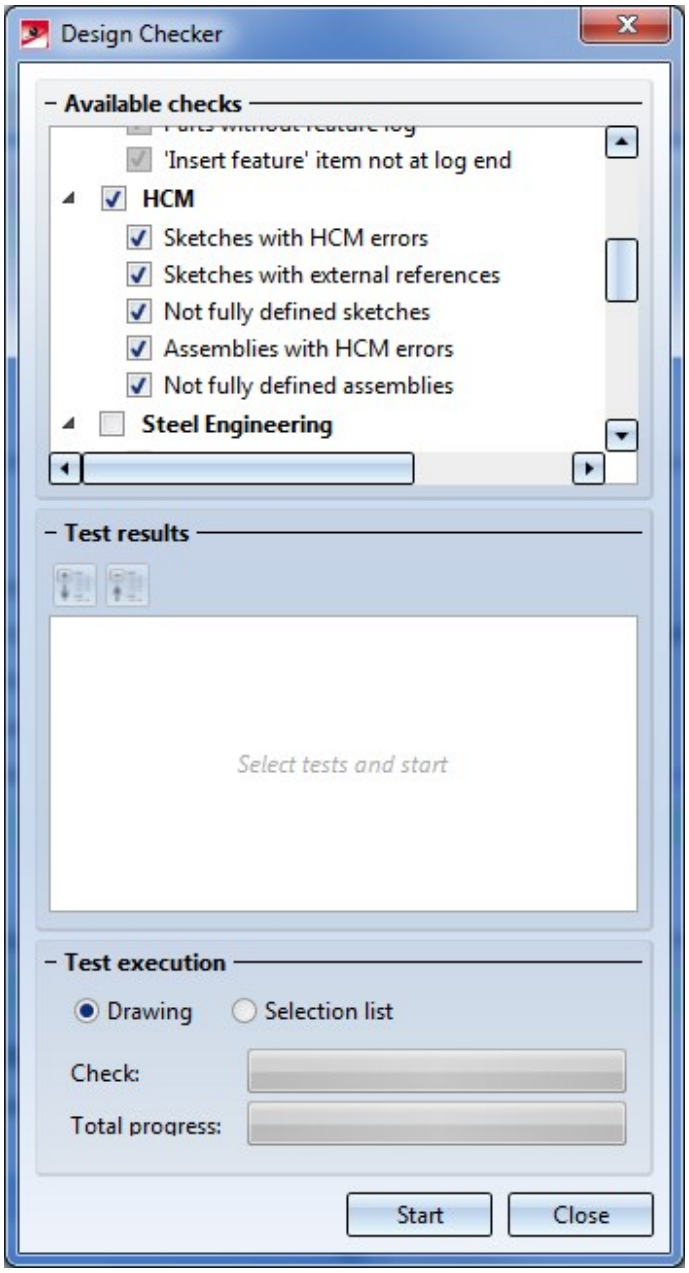
- When creating constraints, a dialogue window is now displayed which queries the relevant geometries and allows for further settings such as the direct assignment of a comment for this constraint.



- The **Distance with min/max. value** and **Angle with min/max. value** functions have been integrated to the **Distance** and **Angle** functions.
- After selecting the first part for a constraint, the constraint is now automatically displayed transparently to simplify the selection of an underlying part.
- It is now also possible to temporarily hide the area currently under the cursor by pressing the **ALT** key when selecting a geometry. By pressing the **Ctrl+ALT** key combination it is possible show such a hidden area again.
- The **Change dimension**, **Move dimension line**, **Equal parameter** and **Equal direction** functions have been removed from the Ribbon.
- By pressing the **F4** key and the entry **Show/Hide coordinate system** in the context menu the part coordinate system can be shown and hidden.

HCM tests in the Design Checker

The following new tests that allow you to examine the model drawing for specific situations are now available in the Design Checker:



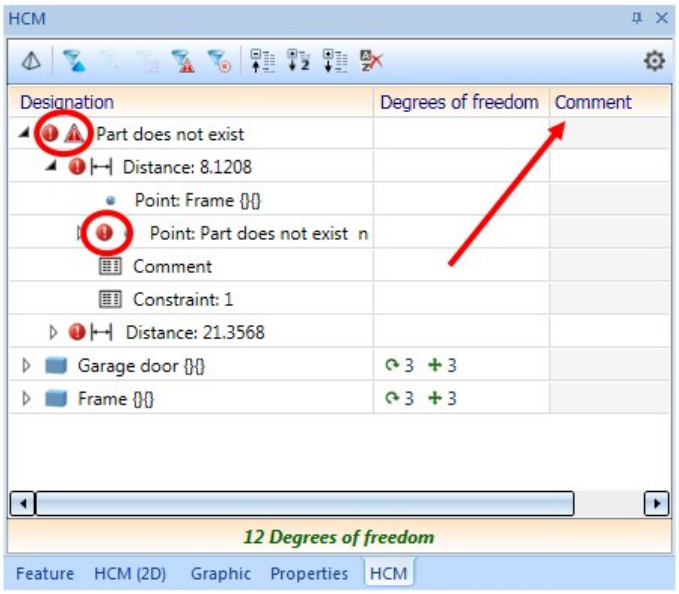
- **Sketches with HCM errors** lists all sketches that contain incorrect references.
- **Sketches with external references** lists all sketches that contain HCM constraints with external references.
- **Not fully defined sketches** lists all sketches that are not fully defined.
- **Assemblies with HCM errors** lists assemblies that contain errors, such as incorrect references or over-defined HCM models.
- **Not fully defined assemblies** lists all assemblies containing a HCM model that is not fully defined.

Performance

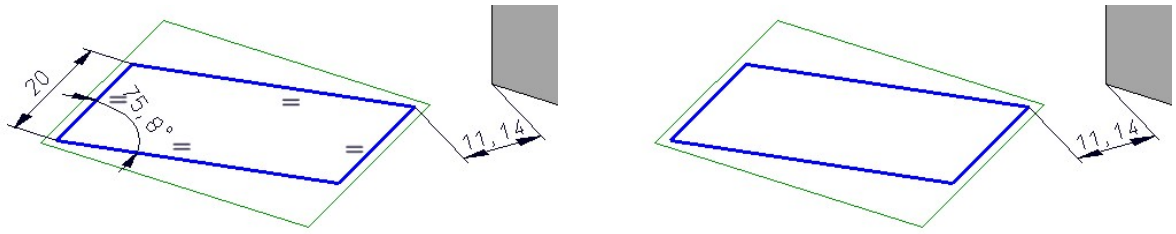
Thanks to internal optimizations the performance during working with large model drawings with very many HCM constraints could be further improved.

Further improved HCM constraint display in the ICN

The display of HCM constraints in the ICN has been further improved:



- The **Comment** column is now also displayed in the **Sort by geometries** view.
- The icons for incorrect constraints are now displayed next to the icons for the affected constraints/geometries.
- The filter function for constraints in the ICN now also affects the display of constraints in the model drawing. For example, if you activate a filter **Constraints with external references** in the ICN, then only these constraints will be displayed in the drawing.



Left: Display of all constraints; Right: Filter "Constraints with external references"

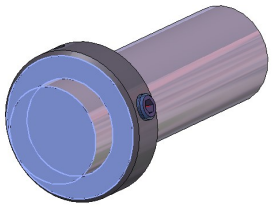
Equal distance constraint applied to surfaces

In the Assembly HCM, the **Equal distance** constraint could not yet be applied to surfaces. This constraint has been changed to allow for the use of surfaces.

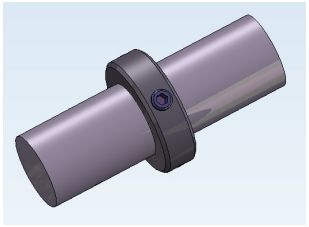
However, it must be considered that the parts must already have a suitable position relative to each other when assigning the constraint. Otherwise, an error message will be displayed:

Example:

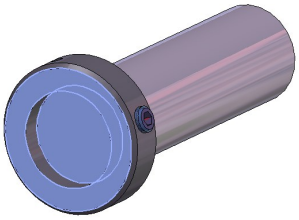
The ring should be positioned centrally on the shaft. For this purpose, the **Equal distance** constraint is to be applied between the outer surfaces of the ring and the outer surfaces of the shaft. The position of the ring is correct in this example because it is already located between the outer surfaces of the shaft.



Result:

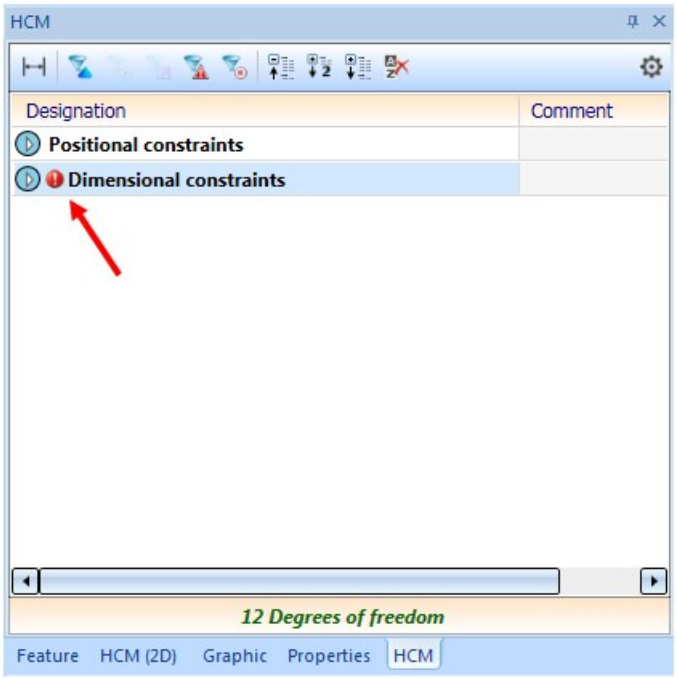


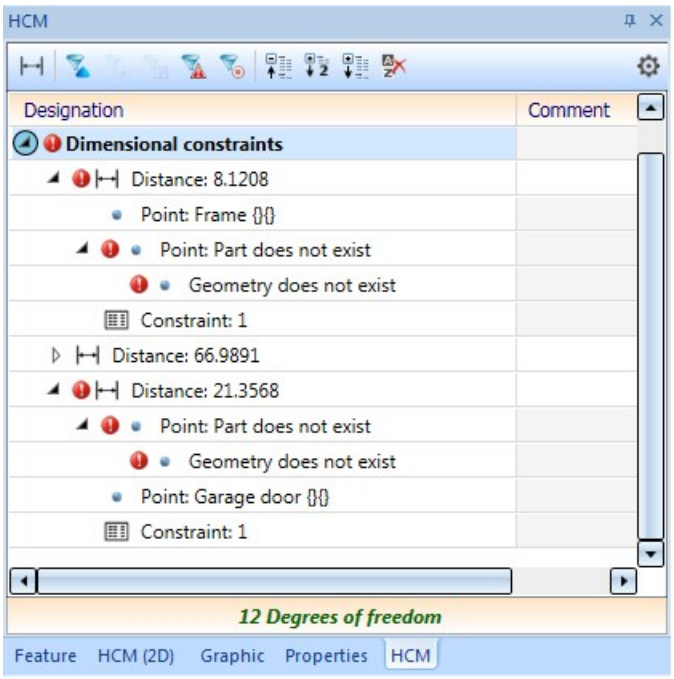
If, however, the ring is not located between the outer surfaces of the shaft, the constraint cannot be resolved and the above error message will be displayed.



Show constraints with errors in structure

If a HCM model contains constraints which cannot be evaluated because of errors, you can now click on the red error mark to expand the structure in such a way that all errors are shown.

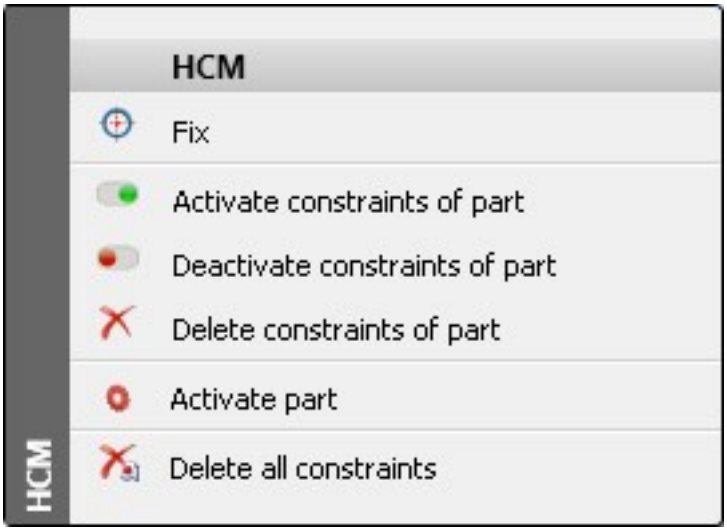




Delete and deactivate the HCM constraints of a part

In the Assembly HCM you have now the option to activate, deactivate or delete all constraints of one part at once.

To do this, switch the display of the HCM window in the ICN to **Sort by geometries** and right-click the desired part to open the following context menu:



Here you will find the new functions **Activate constraints of part**, **Deactivate constraints of part** and **Delete constraints of part**.

Tooltips for HCM constraints

The tooltips that appear when you move the cursor over the icons of the HCM functions on the Ribbons now contain a short description of the respective function.

Function	Tooltip
Coincidence	Coincidences of C-edge elements
Align to CS axis	Align lines to coordinate system axes
Tangent	Tangential transition of distance, circle/circular arc, sphere
Concentricity	Symmetry around a common centre
Mid-point	Fix point in the middle between 2 geometry elements of a sketch
Symmetric	Symmetry between distances, circles/circular arcs, points
Equal radius	Equal radius between circles/circular arcs
Equal distance	Equal distance between sketch segments
Equal angle	Equal value between angles
Parallel	Parallel alignment of edges, surfaces
Perpendicular	Perpendicular position of distances and planes to each other
Fix	Fix/Unfix geometry
Forced fixing	Forced fixing, no transformation possible

Configuration Management

Service Pack 2 2020 (V 2502)

Track catalogue changes

Up to now, the Configuration Editor at **System settings > Catalogues** could be used to define how catalogue changes should be handled in HiCAD. From HiCAD 2020 SP2 onwards these settings are no longer available in the Configuration Editor . Instead, the switch **Track catalogue changes** is now available at **Settings** in the **Catalogue** menu.

If the switch is active, the catalogues are regularly checked during a HiCAD session to ensure that they are up-to-date. This actuality check can, however, significantly impair performance in some situations. If the switch is active, then this is indicated when HiCAD is started. By default, the switch is inactive by default, that is, the catalogues are loaded in HiCAD only once, when starting HiCAD. After that, the system does not check whether more recent data is available. To update the catalogues after changes, the function **Reload** is available. This offers the possibility to update catalogues after changes without having to leave HiCAD and without having to accept performance losses due to the constant check for up-to-dateness.

The current status of the switch (active/inactive) is entered into the Windows Registry. If the switch is active, the following message is displayed when starting HiCAD.

Service Pack 1 2020 (V 2501)

Switch to new standard itemisation

As of HiCAD 2020 SP1, you can use the Configuration Editor to define the procedure for starting the itemization in a model drawing that still uses the itemization up to HiCAD 2017. To do this, open the Configuration Editor and choose **System settings > Itemisation > Switch to 'Standard itemisation used since HiCAD 2018'**.

Major Release 2020 (V 2500)

ABWPOL.DAT settings moved to Configuration Editor

The settings of the ABWPOL.DAT file have been moved to the Configuration Editor (ISDConfigEditor.exe).

You now find the settings for the neutral axis for approximative sheet development at **Sheet Metal > Default setting**.

Assigning of views for model drawings without Sheet and Model area

The settings

- Assigning of views in drawings without Sheet and Model area
- Handling of shortenings displayed as unshortened in shaded views

have now their own node in the Configuration Editor, at **Compatibility > Views**.

The detail drawings created with the Management+BIM module are always stored on Sheet 2 of the drawing. When printing via the HELiOS/HiCAD Spooler, all sheets will be printed - depending on the setting - i.e. also Sheet areas without production drawings such as Sheet 1.

By setting the parameter **Delete unused Sheet areas** to **Yes**, it is now possible to specify in the Configuration Editor at **PDM > Management+BIM > Production drawings** that Sheet areas with no production drawings are deleted. The default setting is **No**.

When creating/updating external CAM data in the formats DSTV-NC, DXF, DWG, NCW/NCX and STEP, and drawing data in the formats DXF, DWG and PDF, the corresponding files can also be automatically exported to the Windows file system if required. The appropriate parameters are available in the Configuration Editor at **PDM > Management+BIM > External production documents**.

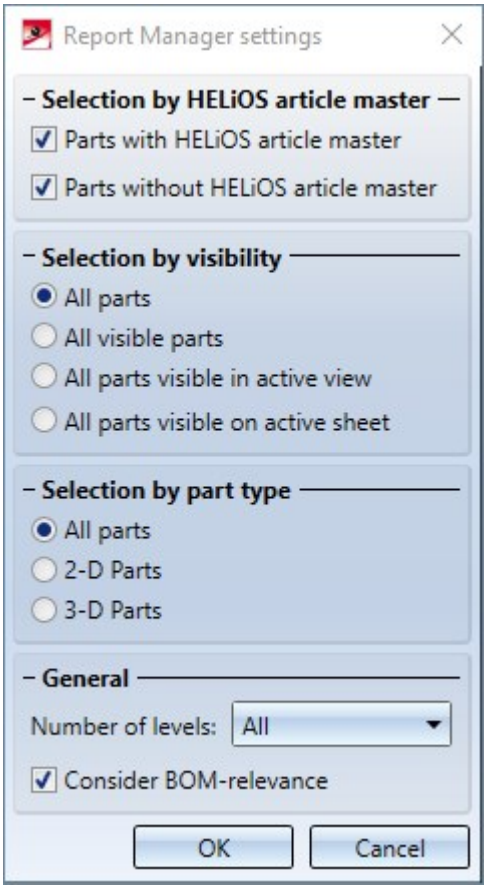
By default, the Management+BIM module works project-related, i.e. the managed parts are usually clearly assigned to one project. When installing parts with reuse in other projects, such as purchased parts, you must assign the article master manually, as these parts are ignored by the BIM automatisms.

Bill of Materials / Report Manager

Service Pack 2 2020 (V. 2502)

BOMs - All visible parts on active sheet

When creating Bills of Materials it is now possible to consider only the visible parts of the active sheet area. For this purpose the **Settings for BOMs** in HiCAD have been extended accordingly.



Service Pack 1 2020 (V 2501)

Steel Engineering BOMs - handling of structure assemblies

Steel Engineering BOMs now allow you to define in the Excel template (HiCAD_Stahlbau.2501.0.xlsx) how BOM-relevant structure assemblies should be handled in the shipping list. For this purpose the row **Ignore structure assemblies** is available on the sheet **Settings**.

	A	B	C	D	E	F	G
1	**Script file**	HiCAD_Stahlbau.2501.0.cs					
2	**Export raw data**	false					
3	**Language**	en					
4							
5	**Consider site assembly**	false	If active, boltings inserted on-site will be declared as loose parts in the shipping				
6	**Projection method for cut symbols**	1	1: European Projection 3: American projection				
7	**Assembly image height**	150	Height of assembly images in shipping lists , 0: Deactivate image output				
8	**Assembly image width**	0	Width of assembly images in shipping list, 0: Total table width				
9	**Assembly image representation**	QuickHiddenLine					
10	**Ignore structure assemblies**	false	If active, structure assemblies on Level 1 are ignored in the shipping list and only their sub-parts are output				
11	**Sheet metal part height**	150	Height of Sheet Metal part images in Sheet Metal part list				
12	**Combine loose parts**	true					

If this setting is set to **true**, then all structure assemblies on the first level of the part structure are ignored in the shipping list and only the sub-parts of these assemblies are taken into account.

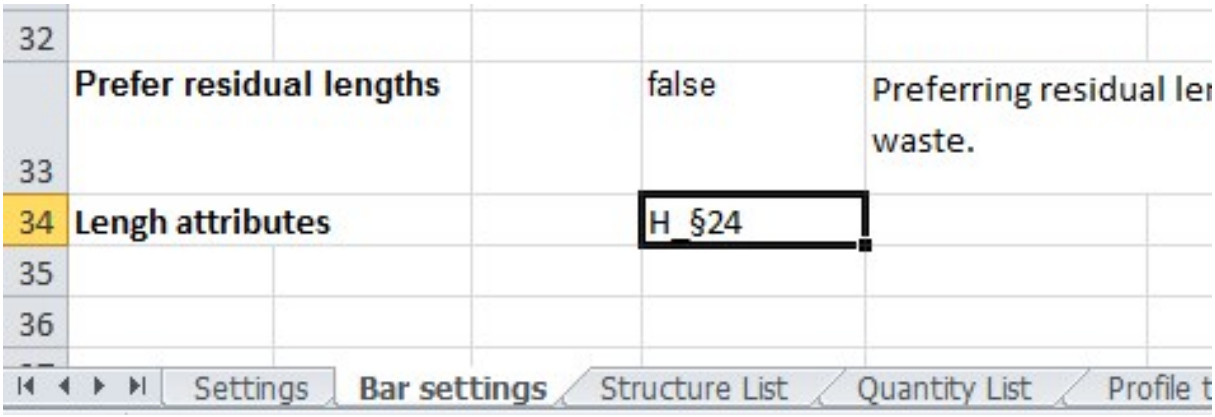
The default setting is **false**.

Major Release 2020 (V 2500)

Steel Engineering BOMs - Selectable length attribute for bar optimization

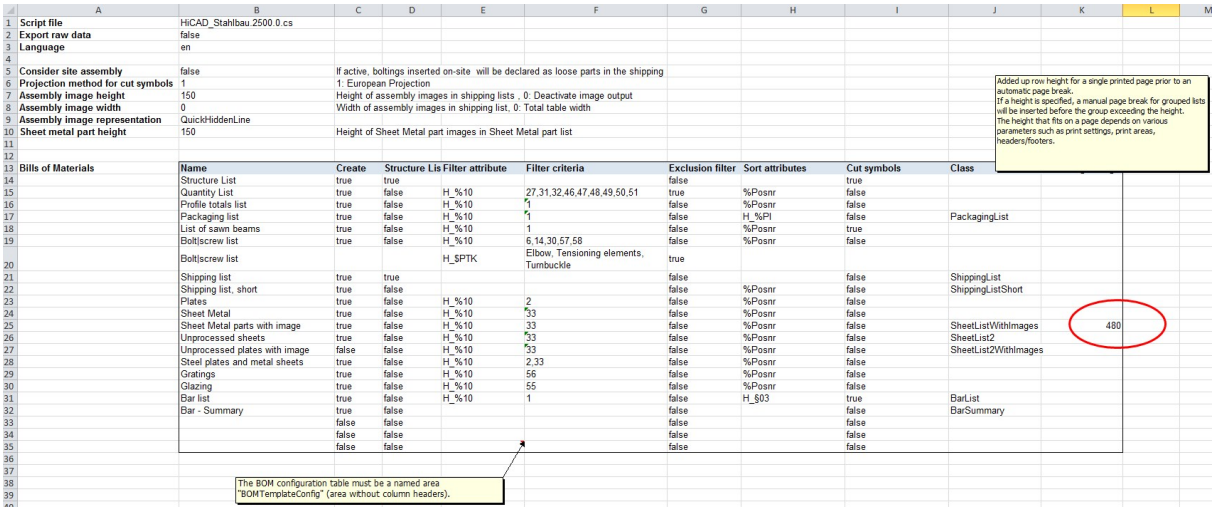
For bar optimization in Steel Engineering BOMs you can now choose in the corresponding Excel template (HiCAD_Stahlbau.2500.0.xlsx), which attribute is to be used for length determination. For this purpose you can find the **Length attribute** option on the **Bar settings** table sheet.

Previously the attribute `H_§03` (Profile length) was used here. The new default value for this option is `H_§24` (Trimmed length).



Steel Engineering BOMs - New parameter for Sheet Metal parts with image

To be able to control the automatic page break when printing the sheet **Sheet Metal parts with image**, you have now the option to set the maximum height (added up row height) for automatic page break on the **Settings** sheet of the Excel template file (e.g. HiCAD_Stahlbau.2500.0.xlsx). If a height is specified, a manual page break for grouped lists will be inserted before the group exceeding the height.



	A	B	C	D	E	F	G	H	I	J	K	L
1	**Script file**	HiCAD_Stahlbau.2500.0.cs										
2	**Export raw data**	false										
3	**Language**	en										
4												
5	**Consider site assembly**	false	If active, boltings inserted on-site will be declared as loose parts in the shipping									
6	**Projection method for cut symbols**	1	1: European Projection									
7	**Assembly image height**	150	Height of assembly images in shipping lists , 0: Deactivate image output									
8	**Assembly image width**	0	Width of assembly images in shipping list, 0: Total table width									
9	**Assembly image representation**	QuickHiddenLine										
10	**Sheet metal part height**	150	Height of Sheet Metal part images in Sheet Metal part list									
11												
12												
13	**Bills of Materials**	**Name**	**Create**	**Structure Lis**	**Filter attribute**	**Filter criteria**	**Exclusion filter**	**Sort attributes**	**Cut symbols**	**Class**		
14		Structure List	true	true			false		true			
15		Quantity List	true	false	H_%10	27,31,32,46,47,48,49,50,51	true	%Posnr	false			
16		Profile totals list	true	false	H_%10	1	false	%Posnr	false			
17		Packaging list	true	false	H_%10	1	false	H_%Pl	false	PackagingList		
18		List of sawn beams	true	false	H_%10	1	false	%Posnr	true			
19		Bolt|screw list	true	false	H_%10	6,14,30,57,58	false	%Posnr	false			
20		Bolt|screw list			H_$PTK	Elbow, Tensioning elements, Turnbuckle	true					
21		Shipping list	true	true			false		false	ShippingList		
22		Shipping list, short	true	false			false	%Posnr	false	ShippingListShort		
23		Plates	true	false	H_%10	2	false	%Posnr	false			
24		Sheet Metal	true	false	H_%10	33	false	%Posnr	false			
25		Sheet Metal parts with image	true	false	H_%10	33	false	%Posnr	false	SheetListWithImages	480	
26		Unprocessed sheets	true	false	H_%10	33	false	%Posnr	false	SheetList2		
27		Unprocessed plates with image	false	false	H_%10	33	false	%Posnr	false	SheetList2WithImages		
28		Steel plates and metal sheets	true	false	H_%10	2,33	false	%Posnr	false			
29		Gratings	true	false	H_%10	56	false	%Posnr	false			
30		Glazing	true	false	H_%10	55	false	%Posnr	false			
31		Bar list	true	false	H_%10	1	false	H_§03	true	BarList		
32		Bar - Summary	true	false			false		false	BarSummary		
33			false	false			false		false			
34			false	false			false		false			
35			false	false			false		false			

Added up row height for a single printed page prior to an automatic page break.
If a height is specified, a manual page break for grouped lists will be inserted before the group exceeding the height.
The height that fits on a page depends on various parameters such as print settings, print areas, headers/footers.

The BOM configuration table must be a named area "BOMTemplateConfig" (area without column headers).

The maximum height depends on various parameters such as print settings, print areas, headers/footers.

To calculate the height you can switch, on an Excel sheet with more or less the same row heights (e.g. Structure list) and a sufficient number of filled in rows, to the **Page break preview** mode on the **View** tab. The height results from

- from the number of rows up to the first page break,
- minus the rows that are repeated on every printed page,
- multiplied with the row height.

The default value is 480 (32 rows* Row height 15).

HiCAD GUI Creator

Major Release 2020 (V 2500)

Default values for Toolbox objects

For Toolbox objects of the type EditBox, ComboBox, CheckBox and RadioButtons as well as CatalogEntry, default values can be set from HiCAD 2020 onwards.

CatalogEntry

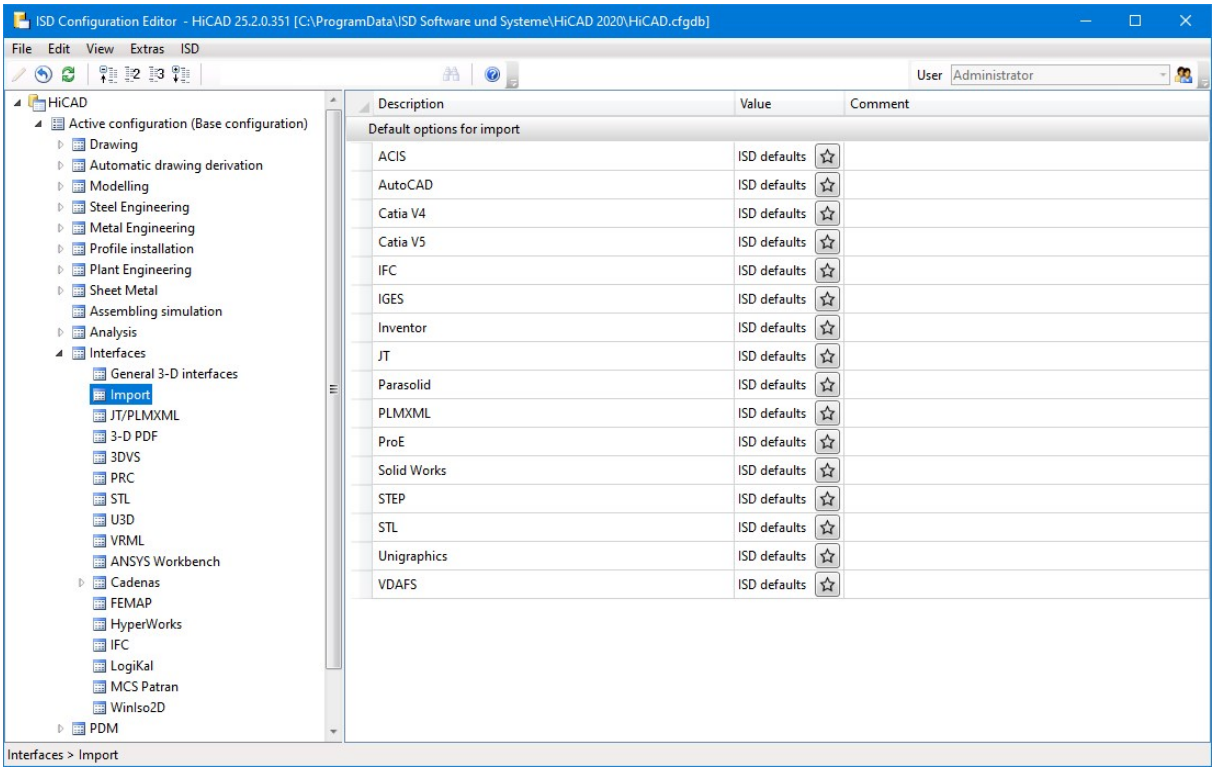
Variables for catalogue entries that were assigned to a installation element or railing component and have not been not defined as list variables are now automatically converted to list variables when the variant is saved. The text object **CatalogEntry** will then be automatically used for these variables when they are inserted into a user dialogue

Interfaces

Service Pack 2 2020 (V 2502)

Favourites for 3-D import

Thze settings for the **3-D Import** function can now be saved as favourites. To do this, click on the symbol in the dialogue window. You can find more information on favourites management in the in the Manage Favourites topic of the HiCAD Basics Help. You can specify which favourite is used as the default in the import dialogue in the Configuration Editor at **Interfaces > Import**.



To preset a favourite for a file format, just click on the symbol and choose the desired favourite.

When starting the import dialogue, please note that the last selected settings for the respective file format are used as default. If you have not made any changes to the settings of the favourite defined in the configuration management, the favourite settings will be used.

Due to the favourites management, the following settings in the Configuration Editor are not applicable:

- **Interfaces > General 3-D interfaces > Default options for import**
- **Interfaces > STL > Default options for import**
- **Interfaces > IFC > Default options for import** except for the **Attribute mapping configuration**

For import via Drag&Drop from the Explorer, the import dialogue is now also displayed. This dialogue is preset with the file to be imported and the corresponding options for the file format are displayed. This does not apply to the import of 3-D DXF/DWG.

3-D export by views

From SP2 on the context menu for views provides the function **3-D Export (STEP, 3D PDF...) by views** . Analogous to the 3-D export of model drawings, only the active view is exported here.

Export by views is possible for the following formats:

- 3D PDF
- Universal 3D
- Product Representation Compact
- 3DViewStation
- Step
- Autocad (DXF, DWG)
- IGES
- VDAFS
- CATIAV4
- CATIAV5
- ACIS
- Parasolid
- ProE
- JT
- PLMXML

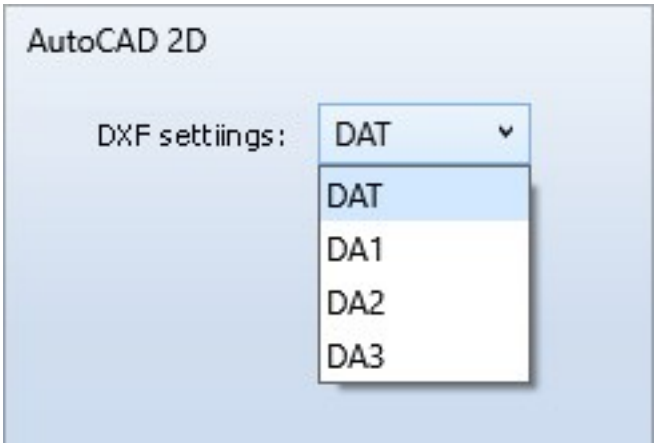
Please note:

- Only parts that are visible in the view are exported.
- The export is also possible for sectional views.
- For exploded views only the export by views to 3D PDF, Universal 3D, Product Representation Compact and 3DViewStation is possible. During normal export, the parts are exported in their original position.
- Views of developed sheets cannot be exported.

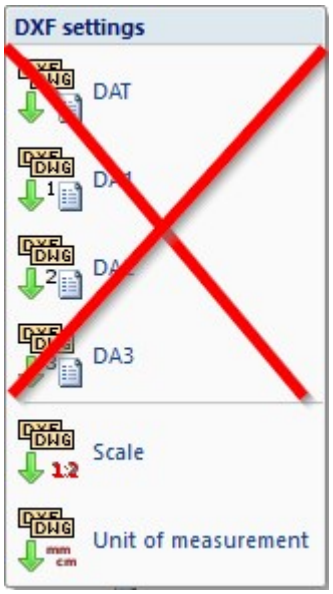
2-D import

AutoCAD DXF/DWG

When importing 2-D files in the DXF/DWG format, the conversion file **ACADHCAD** can now be selected for import directly in the import dialogue.



As a result, the functions formerly available at **Drawing > New/Open > Open** > **Settings** have been withdrawn.



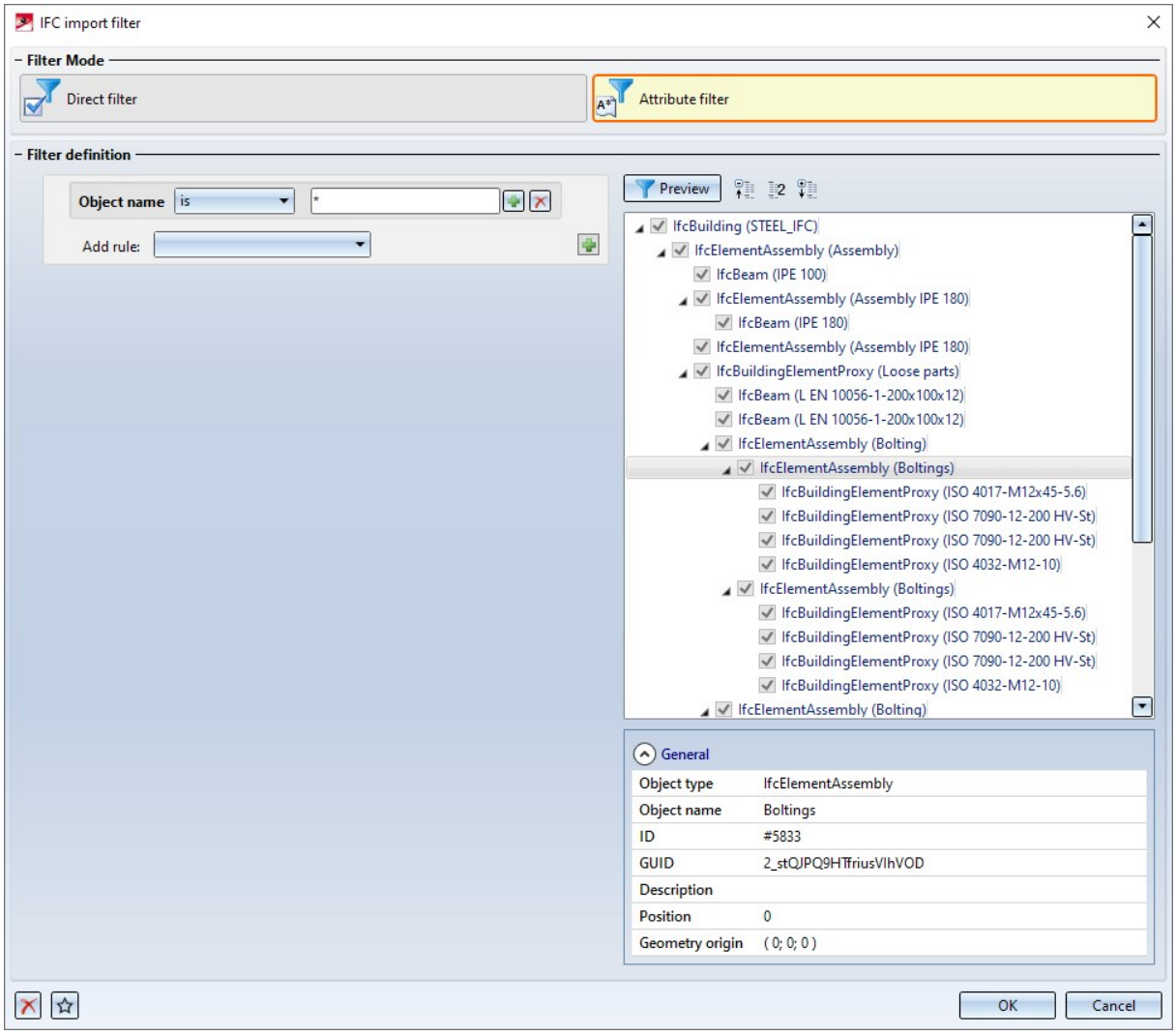
2-D import via the HiCAD Start Centre

BabCad (DXB), PC-Draft (V11) and ProCad (DXP) files can no longer be loaded via the Start Centre. The corresponding import functions can be found at **Drawing > New/Open > Open** > **Further**.

Starting with SP2 the Start Centre supports the import of ME10 files (MI) and files from Technovision (GAF).

IFC import

When importing IFC files, the IFC objects to be imported can now also be selected by using attribute filters. The filter mode Attribute Filter is available for this in the **IFC import filter** dialogue window. Here you can filter the structure according to the attributes **Object name**, **Object type**, **Description** and **GUID**. You can also define one or more rules that must be fulfilled for an object to be selected. These rules can be linked with AND or OR.



Please note:
You can use either direct filters or attribute filters. A combination of direct and attribute filters is not possible.

IFC export - Part filter for geometry merging

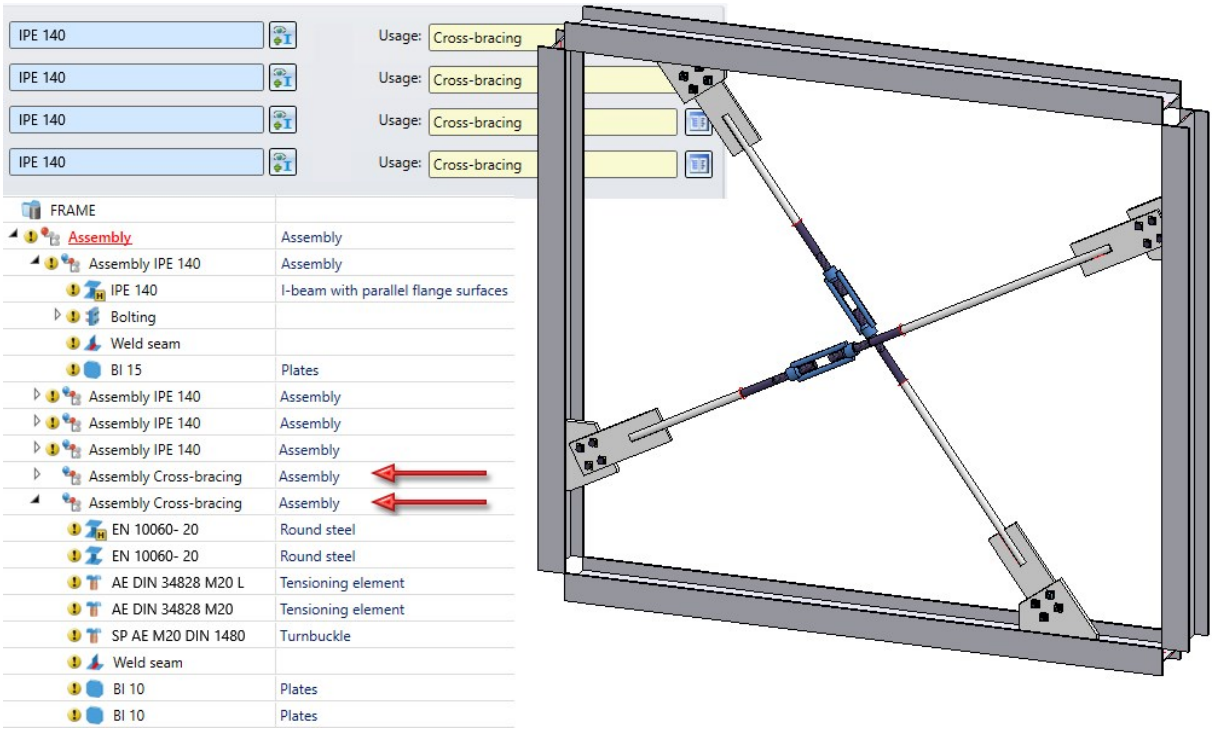
This new parameter allows you to specify during **IFC export** that main and sub-parts are combined into one part during export - without Boolean operations.

To use part filters, activate the checkbox **Part filter for geometry merging** and then - after clicking the symbol - the part type. All parts filters defined with the **Search via toolbar (Part filter)** function are supported. The following part filters are already predefined:

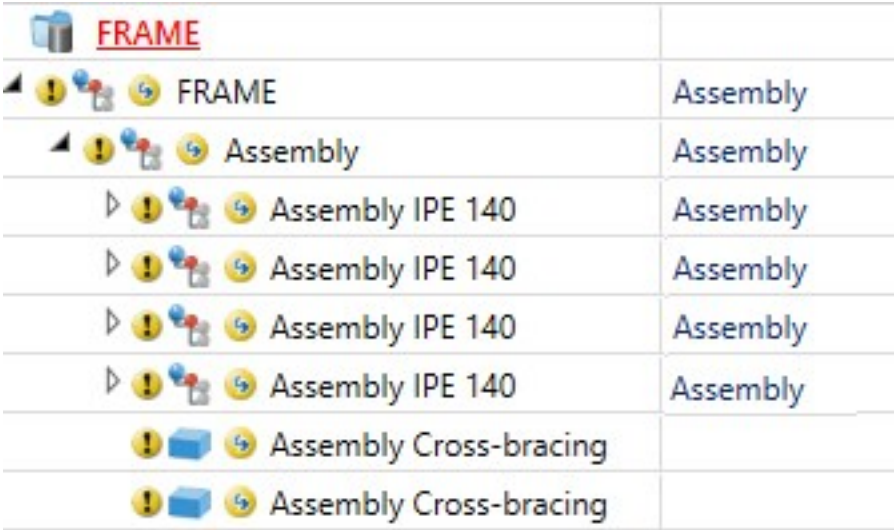
Industry	Part type	
General	Referenced part	
	Sketch	
	Solid	
Plant Engineering	Straight pipe	
	Elbow	
	Pipeline	
Sheet Metal	Sheet	
Mechanical Engineering	Standard part	
Steel Engineering	Facade assembly	■ Mullion or transom assembly ■ Mullion assembly ■ Mullion joint ■ Transom assembly
	Steel Engineering assembly	■ Stabilizing pipe ■ Console ■ Cross-bracing ■ Frame ■ Column ■ Girder
	General part	■ Glass
	Facade part	■ Rubber seal ■ Isolator ■ Clamping strip or cover tray ■ Mullion or transom
	Steel Engineering part	■ Flat steel ■ Shaped pipe ■ Grating ■ I-beam ■ L-Profil ■ Round pipe ■ Round steel ■ Plate ■ Steel beam or profile ■ T-beam ■ U-beam ■ Z-profile

Example:

A Cross-bracing (2602) has been installed between four beams. **Cross-bracing** was chosen as usage for the beams in the cross- bracing dialogue.



The two marked assemblies then have the usage **Cross-bracing**. If the model drawing is exported as an IFC file using the part filter **Steel Engineering > Steel Engineering assembly > Cross-bracing**, the parts of the assemblies will be combined into one part each.



Service Pack 1 2020 (V 2501)

Marking of imported 3-D parts in the ICN

All parts that you insert into the model drawing with the **3-D Import** function will be marked with the symbol in the ICN. If you move the cursor over the symbol, further information on the corresponding part will be displayed, e.g. the imported file format and the HiCAD version with which the part was imported.

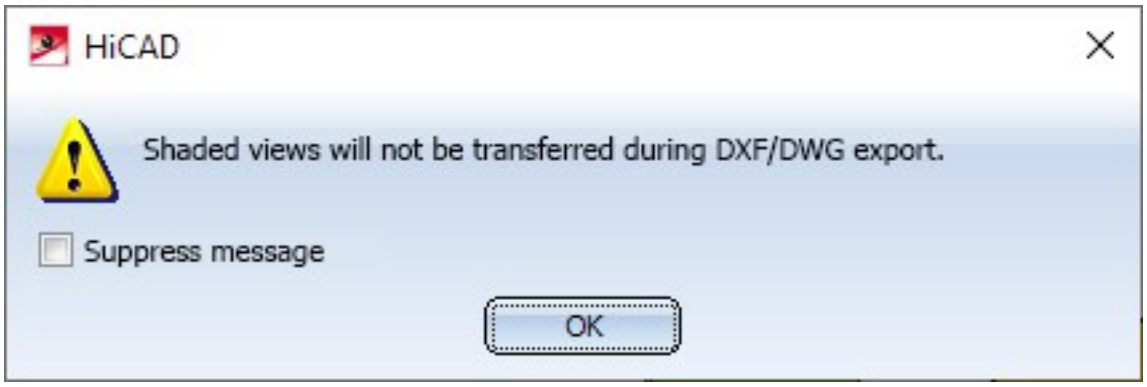
Save as DXF/DWG

When saving model drawings or image details with the functions

DXF/DWG and

Detail, DXF/DWG

a message will now be displayed if the selected area contains shaded 3-D models.



BOM-relevance for IFC import

When importing IFC files, the BOM-relevance of parts and assemblies is automatically removed.

Changed menu for 2-D import

- The import of DXF/DWG files is now done by the function **2-D Import** .
- The import of PC Draft files (.V11), Babcad files (.DXB) and Procad files (.DXP) is now performed via the corresponding functions at **Drawing > New/Open >Open** **> Further > ...** :

Major Release 2020 (V 2500)

Update to CADfix 12 SP1

With the update to CADfix 12 SP1 the following format versions are now available in HiCAD 2020:

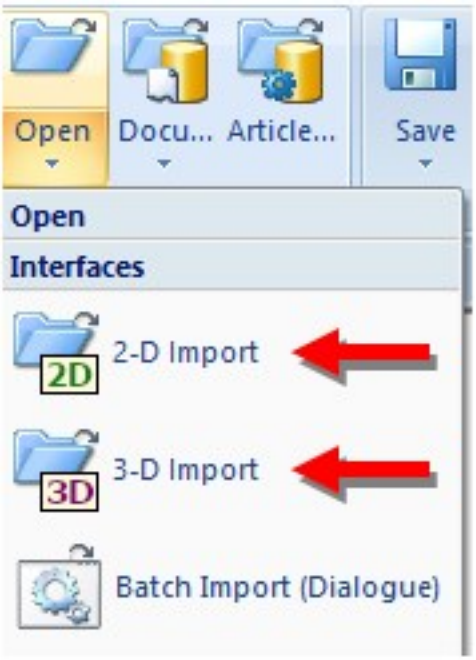
- CATIA V5 – V5V62019
- NX 12
- SolidWorks 2019
- Inventor 2019DXF/DWG 2018

Import files via interfaces

When importing 2-D/3-D model drawings via interfaces, HiCAD 2020 distinguishes between 2-D and 3-D imports. For this purpose the old function **Drawing > New/Open > Open > STEP, IGES, ...** has been split into the following 2 new functions:

2-D Import and

3-D Import :



Insert 3-D foreign formats as parts in current drawing

Data in the 3-D formats:

- STEP (*.stp, *.step)
- STL (*.stl) (the STL file must exist in the ASCII format!)
- IGES (*.igs, *.iges)
- VDAFS (*.vda)
- CATIAV4 and CATIAV5 (*.MDL, *.Model, *.EXp, *.ISO, *.DLV3, *.CATPart, *.CATProduct)
- ACIS (*.SAT)
- Parasolid (*.x_t, *.xmt_txt, *.x_b)
- ProE (*.prt)
- Unigraphics (*.prt)
- SOLIDWORKS (*.sldasm, *.sldprt)
- Inventor (*.ipt, *.iam)
- AutoCAD (*.dxf, *.dwg)
- JT (*.jt)
- PLMXML (*.plmxml)
- IFC (*.ifc, *.ifczip)

could previously only be imported to a new HiCAD drawing with the **Drawing > New/Open > Open** > **3-D Import** function. If you want to insert data in these formats to the current drawing, now choose the renamed function **Drawing > Insert Parts > Exp.** > **3-D Import**..

IFC import - Filter option and multiple selection with CTRL or SHIFT key

If the **Filter** option is active during IFC import, another dialogue will be opened after selecting **Open**, enabling you to select/deselect IFC classes. From HiCAD 2020 onwards you can also make multiple selections of classes using the CTRL or SHIFT key.

3DPDF - Enhancements

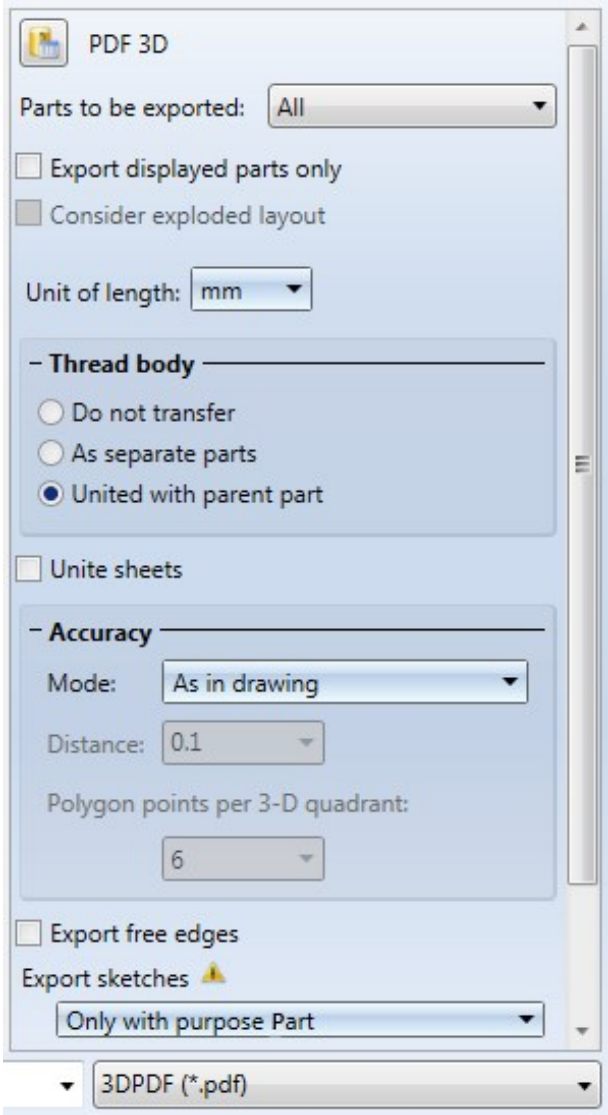
SDK Update to 2.18.1

The SDK (Software Development Kit) for 3DPDF export has been updated to Version 2.18.1.

Taking into account of free edges and sketches

For PDF-Export free edges and free sketches are now taken into account. This behaviour can be specified in the export parameters.

Furthermore, you can now select or sketches whether all sketches are to be taken into account, or only sketches to which the purpose **Part** has been assigned.



Sheet Metal

Service Pack 2 2020 (V 2502)

New bend zone, with shortening

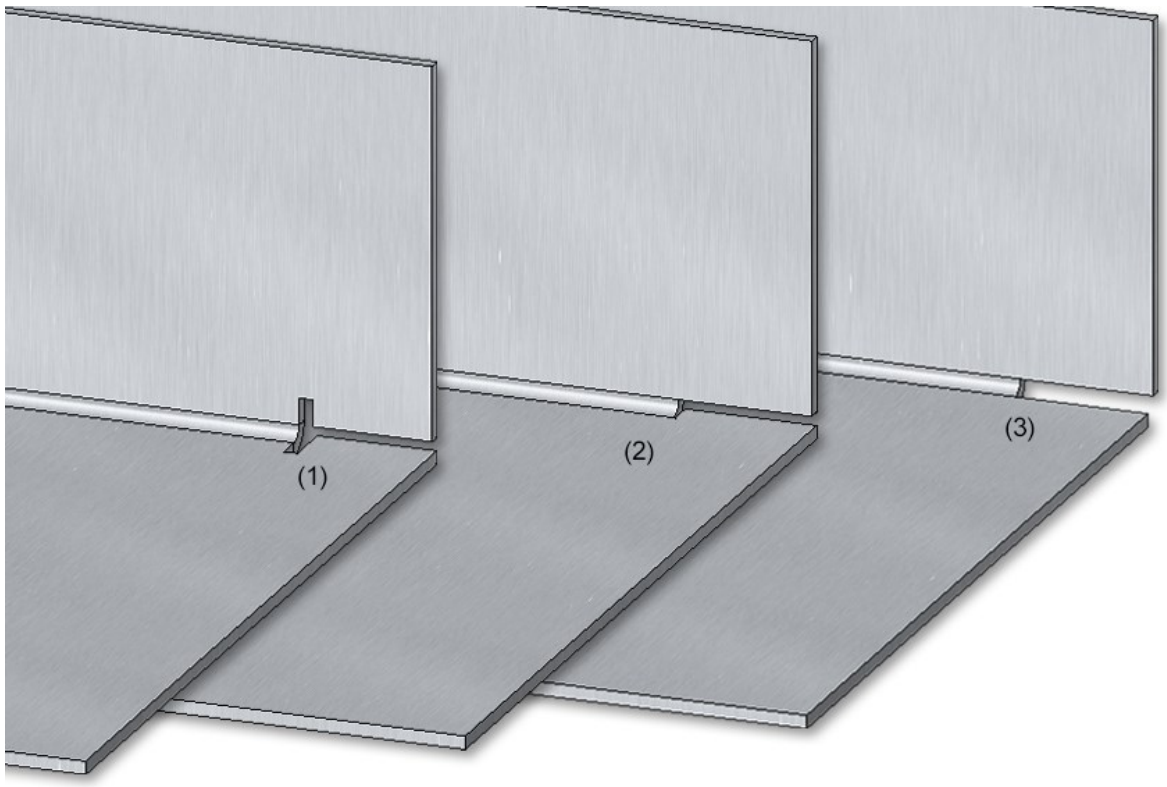
The new user-friendly **Insert bend zone** function brings together the previous functions in the **Bend zone** function group on the **Sheet Metal** tab. A major advantage is the insertion of milling edge zones for composite sheets.

The **Insert bend zone** function allows you to subsequently join the flanges of a sheet metal part to a bend zone. The sheets are shortened so that the height or length does not change. The amount of the shortening depends on the bend radius.

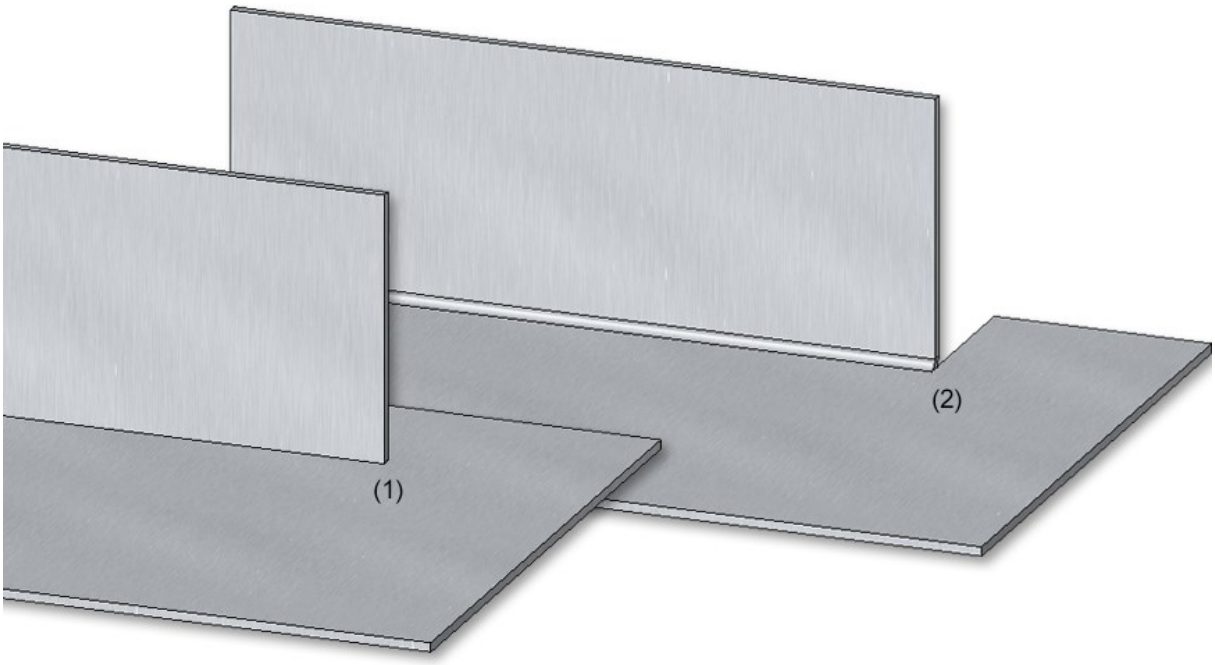
The length of the bend zone can be determined by 2 points. To avoid shortening the connecting sheet you can choose between clearance and a relief groove.

If the width of the connecting sheet is different, a cut is inserted. The depth of the sheet on which the attachment is made is retained.

All entries are recorded in the feature log and can be altered there later.



(1) Bend zone with relief groove
(2) Bend zone with clearance
(3) Bend zone with shortened flange, without relief groove

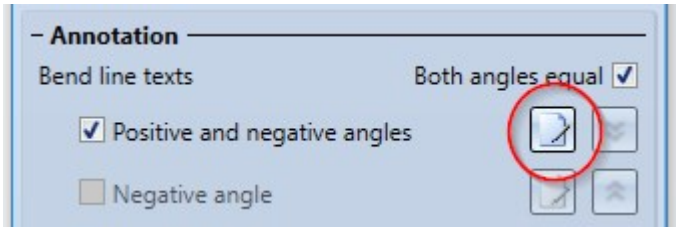


(1) Original situation
(2) Different flange widths with cut and clearance

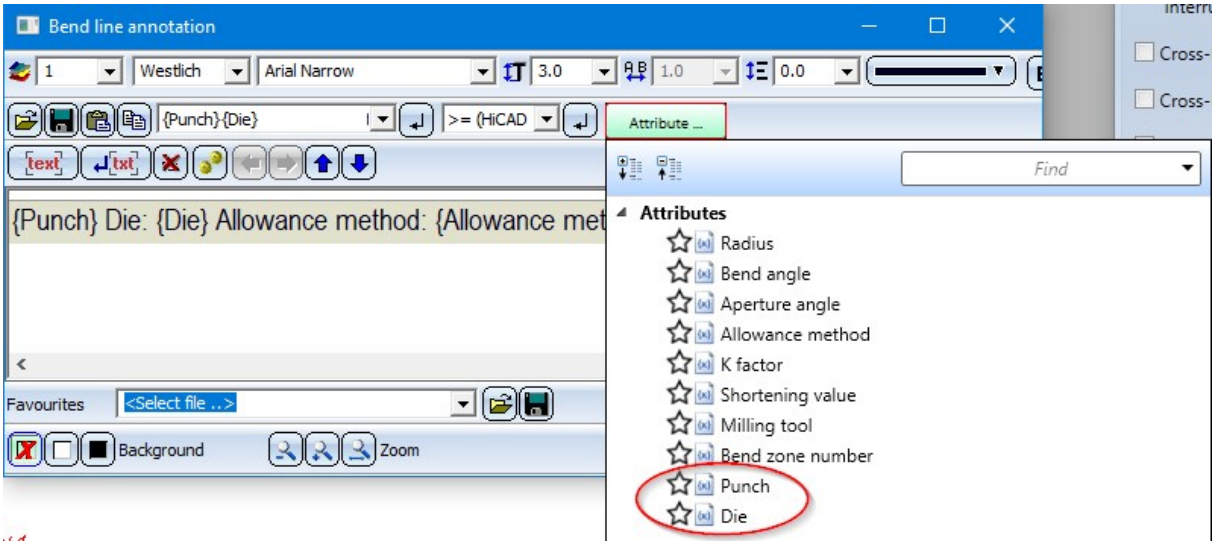
Sheet metal development with die and punch

The bend tools **Die** and **Punch** defined in the LVD **Bend zone tooling** can now be displayed in the development bend line text.

The selection is made in the development settings when editing the bend line texts.

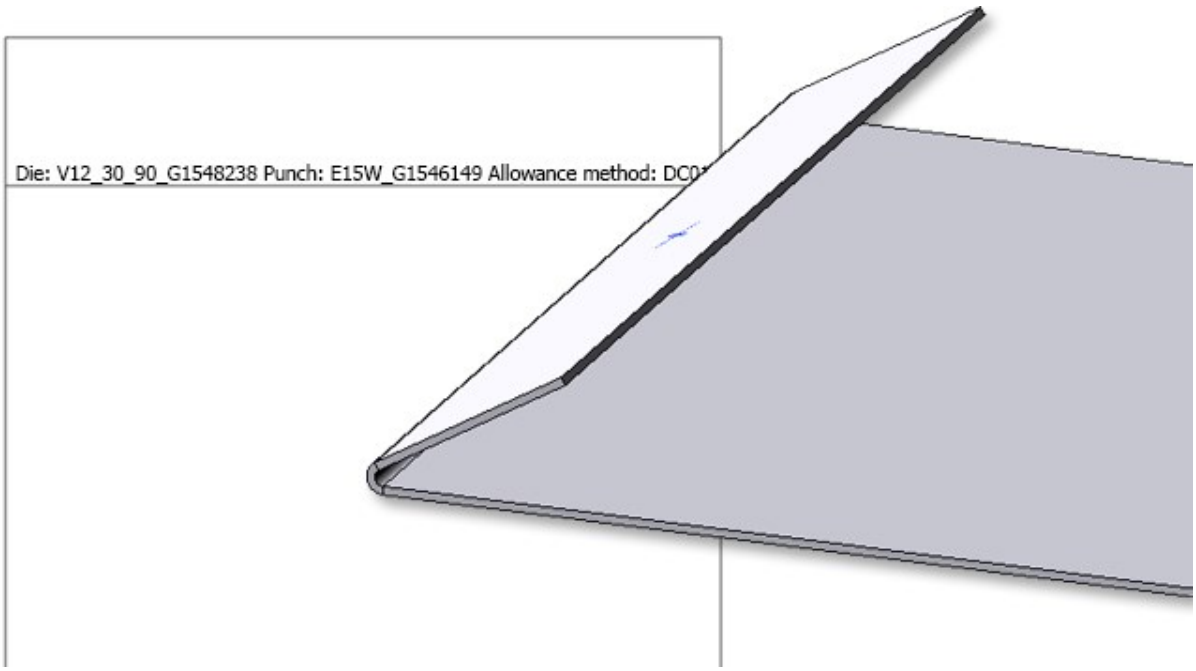


Then select the **Die** and **Punch** attributes in the text editor.



If an assigned tool exists for the bend zone, the names of these tools can now be inserted in the bend line texts.

The names are the entries in the **BZ** column in the corresponding bend tool assignment tables "LVD Die - Table" and "LVD Punch - Table" from the Catalogue Editor.

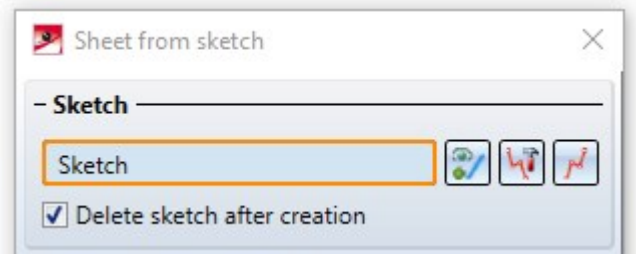


Changed dialogue for functions with sketch selection

The dialogues of functions requiring the selection of a sketch,

- Sheet from sketch
- Sheet along sketch
- Flange along sketch

have been slightly modified. The functions **Process sketch** and **New sketch in plane** can now be accessed immediately in the dialogue window.



Design Checker - New check for Sheet Metal parts

The Design Checker has been extended by a new check called **Sheet must contain exactly 1 direction symbol**. This allows you to find all sheet metal parts that are not assigned exactly one direction symbol.

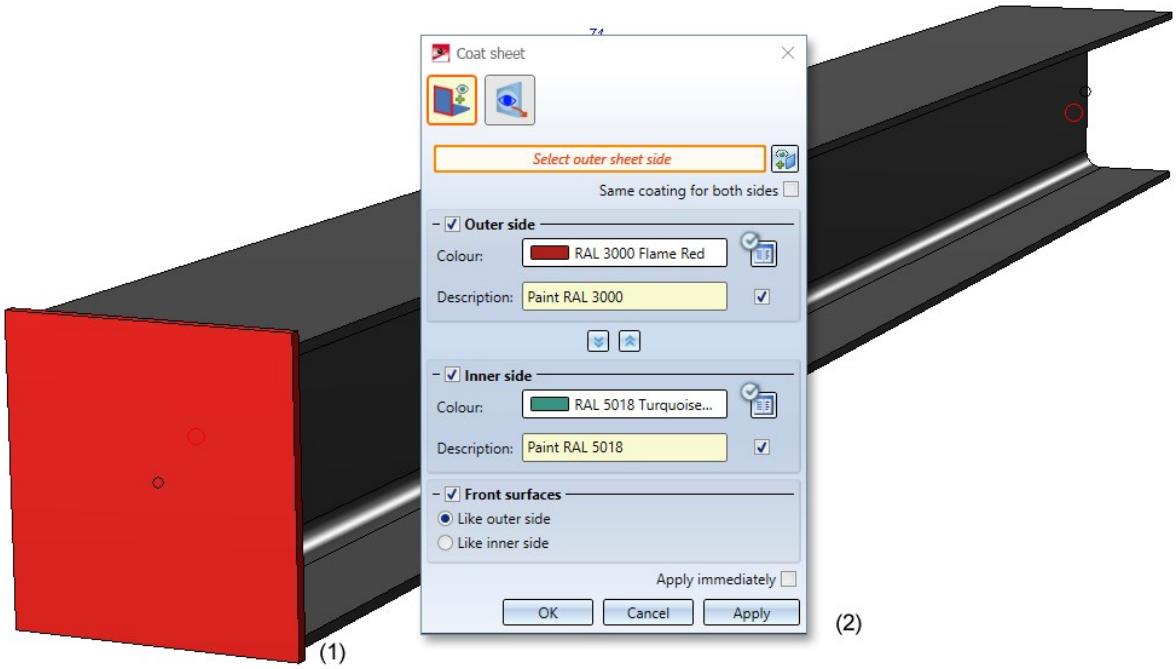
Steel Engineering plates can be coated like Sheet Metal parts

From HiCAD 2020 SP2 on, Steel Engineering plates can be coated like Sheet Metal parts. Therefore the functions

- Visible side
- Delete visible side
- Coating
- Extend coating

can now be found in the **Further Functions** function group of the **Steel Engineering** tab.

In the **Process** function group of the **Sheet Metal** tab, these functions have been combined in the **Coating** function.



(1) Steel Engineering plate,
(2) Coating settings

Service Pack 1 2020 (V 2501)

Sheet development

Add and subtract in context menu for sheet development

The new processing functions for 3-D developments can be activated by right-clicking on the development and selecting the **Process development** function .

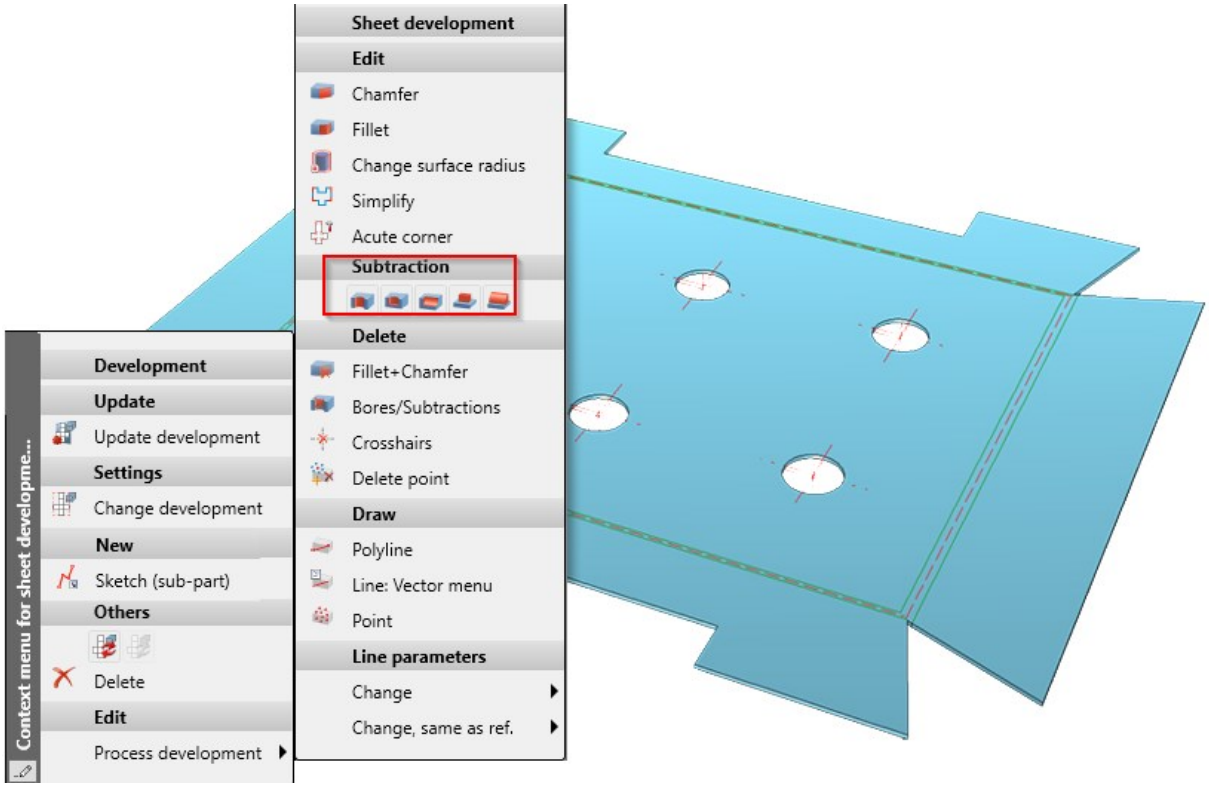
With the following functions:

- Subtract, via translation
- Subtract, via translation and depth
- Subtract, via rotation
- Add, via translation
- Add, via rotation

you can insert holes, create subtractions, attach sweep bodies and cut parts on the basis of a planar sketch.

If you have selected the **New sketch** option in the dialogue window, HiCAD automatically activates the **Sketch** tab. Use the functions on this tab to create the sketch. For sketch-based functions, you can use the **Information** menu functions during creation or processing of the sketch.

Only the closed polylines of a sketch are considered.



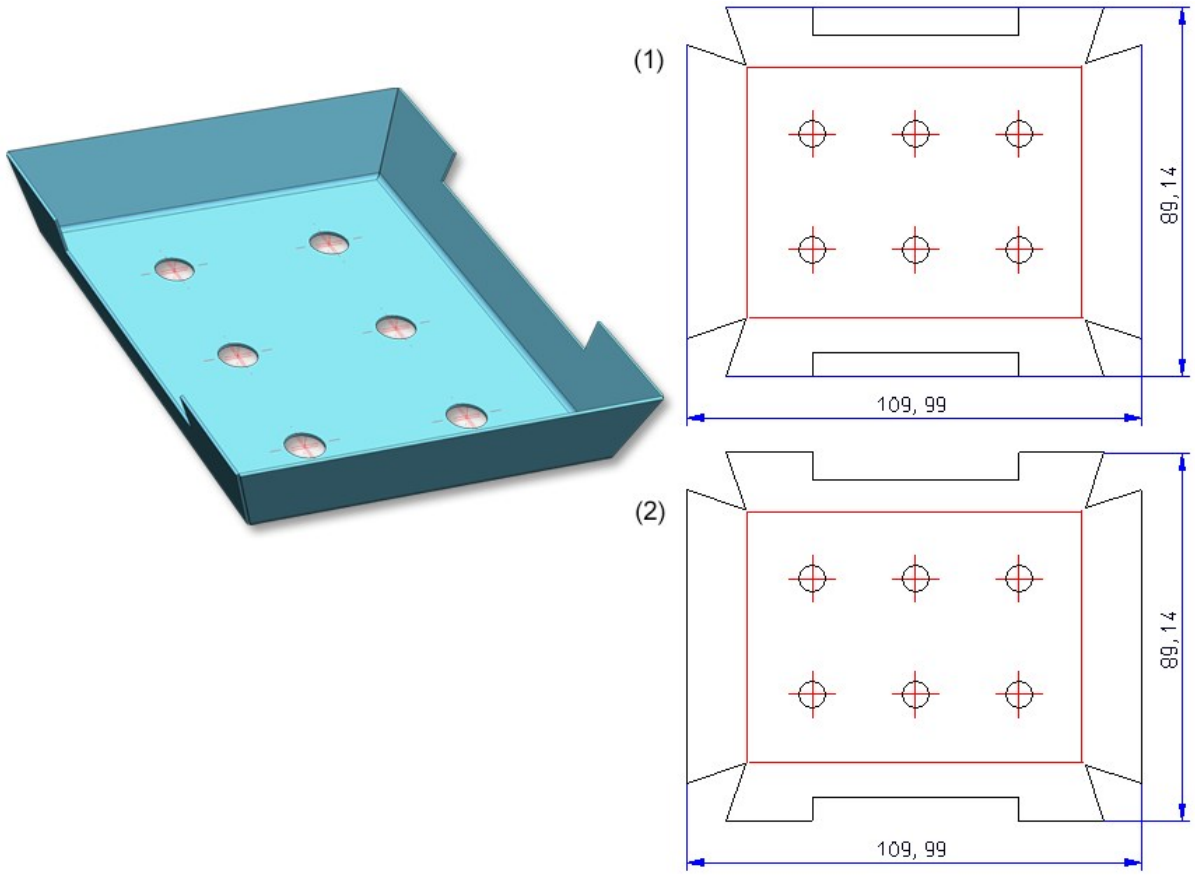
Mark sheet developments with suppressed updates in ICN

With the function **Suppress sheet update** (right click on a development) you "freeze" the development for changes. This means that changes to the sheet metal part will not be transferred to the development. In the ICN these developments are marked with a star symbol.

Automatic dimensioning - Optimised base points

For automatically generated dimensions, such as in sheet developments, from HiCAD 2020 SP1 onwards, the (unshortened) projection line is only drawn to the nearest point on the object to be dimensioned, i.e. to the next point on the contour.

The image below shows a Sheet Metal construction with subtractions and the automatically dimensioned development of the sheet.

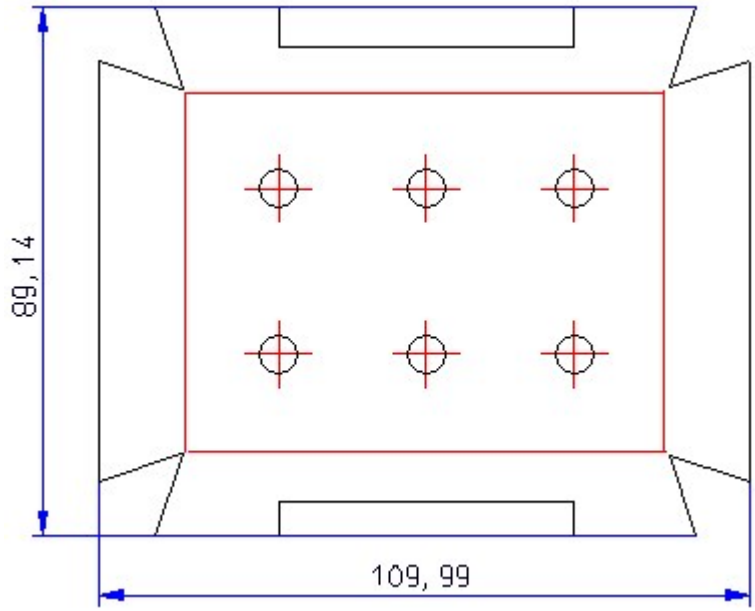


(1) Automatically dimensioned development, older versions

(2) Automatically dimensioned development, from HiCAD 2020 SP1 onwards

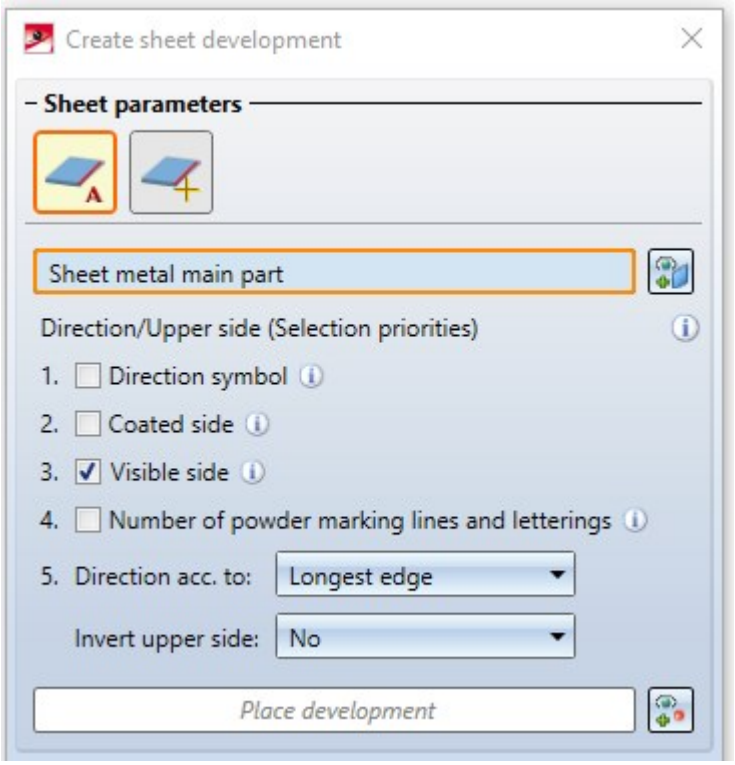
Please note:

If an automatically generated dimension is changed manually, the optimization described above may not be possible in some cases. This is the case, for example, if you drag the dimension on the right completely to the other side in the example shown below.



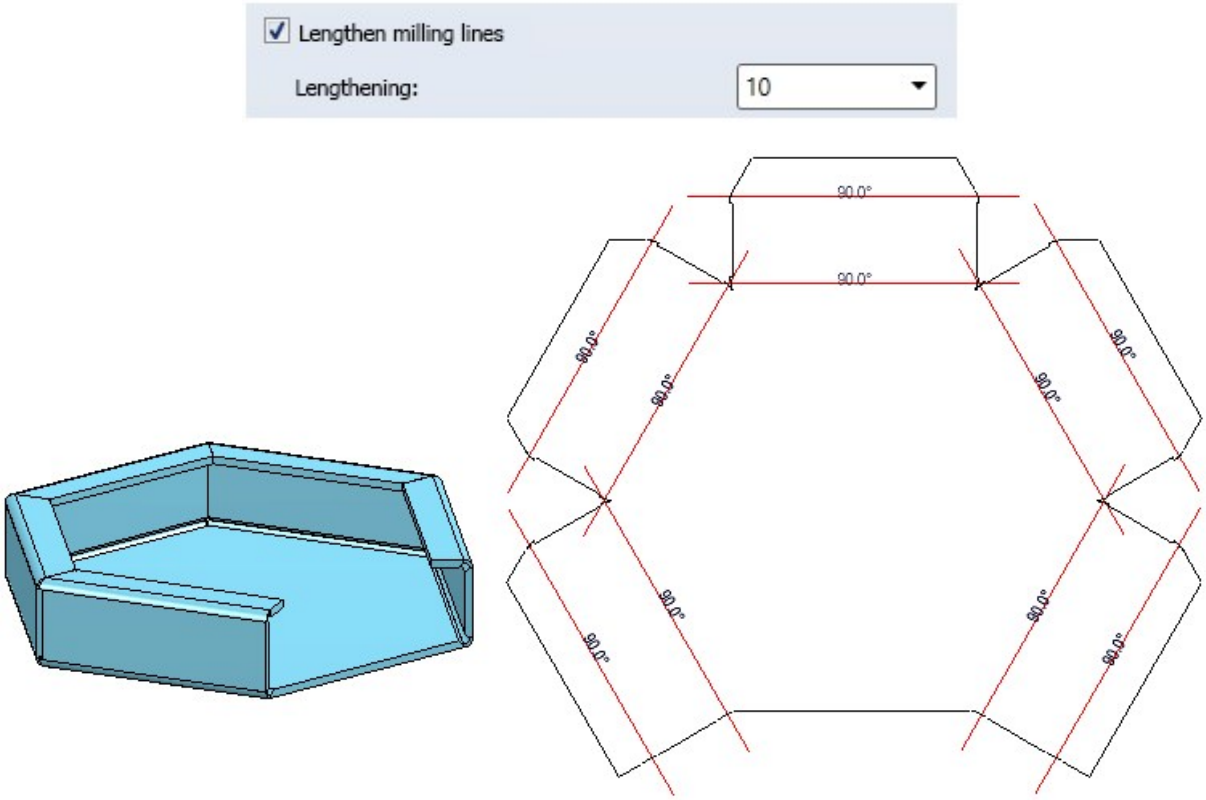
Development considers definition of outer sheet side

When choosing the Sheet alignment:Automatic option for a sheet development, the definition of the visible sheet side can now be taken into account. This is done with the new **Determine visible side** function in the **Process** function group.



Lengthening of milling edges in developments

In order to fold Sheet Metal cassettes with milling edge zones better and more easily, the milling edge zones are often extended a little at the intersection points. You have the possibility for milling edge zones to extend the bend line on the **Milling+Folding** tab of the **Extended settings** dialogue window. There, activate the **Lengthen milling lines** checkbox and enter a value.

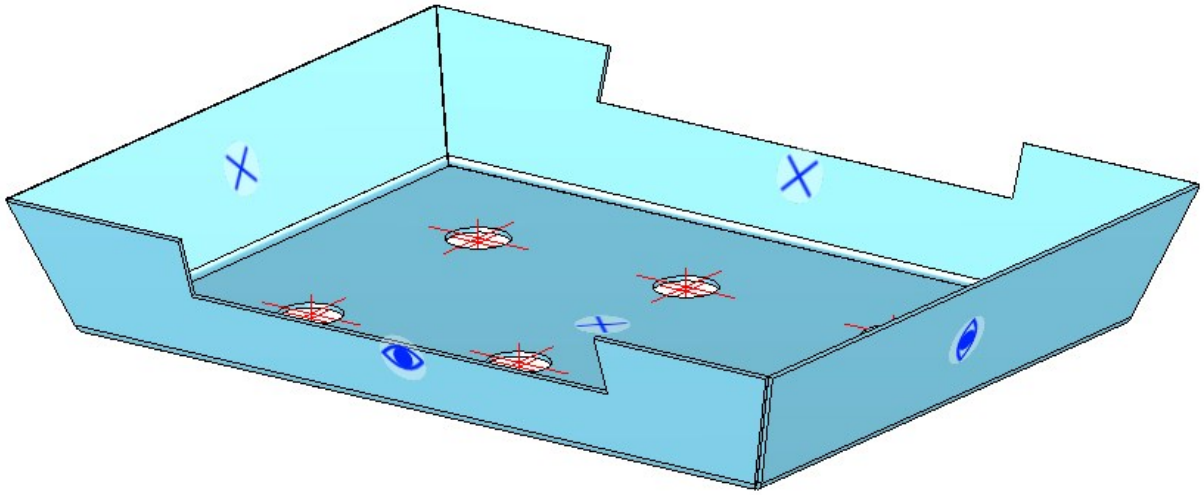


Lengthening of bend lines in the development

New visible side of Sheet Metal part

The new **Determine visible side** function lets you define the upper side of a Sheet Metal part. This side can then be taken into account during development or coating. Identify the side using a surface or two edges. Then you can determine the visible side for further Sheet Metal parts or execute the function with the middle mouse button.

For deleting, the **Delete visible side** function is available in the pull-down menu.



The visible side is marked with these icons . The other side is marked with X icons .

Sheet Metal parts with visible side

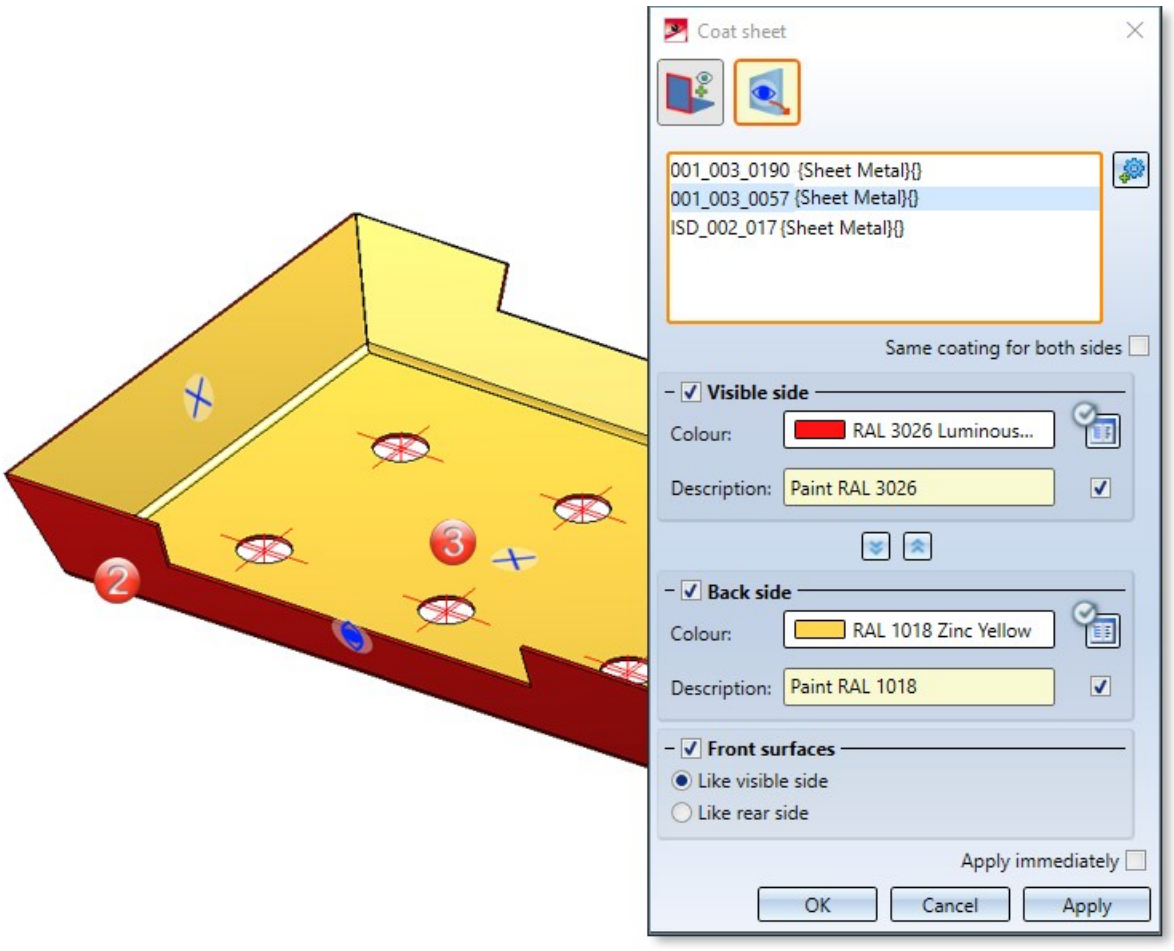
Sheet Metal parts that have been assigned an identifier for the visible side are taken into account for identical part search. For example, if the model drawing contains two Sheet Metal parts of the same size, one of which has an identifier for the visible side, the sheets will not be considered to be identical and are assigned different item numbers.

Coat Sheet Metal parts with multiple selection

Sheets that were previously marked with the **Determine visible side** function can now be taken into account when coating the sheets with multiple selection .

If you have called the coating dialogue, the **Via visible side** option is now available. You can then add several sheets with a visible side to the list and assign the coating colour (RAL, NCS and System colour or Drawing colour) and the coating type Visible side, Back side and Front surface) with **Apply**.

Right-click on a Sheet Metal part to remove it from the list. If you want to select more sheets, select the icon for identification.



(1) Select Sheet Metal parts via visible side
(2) Visible side
(3) Back side

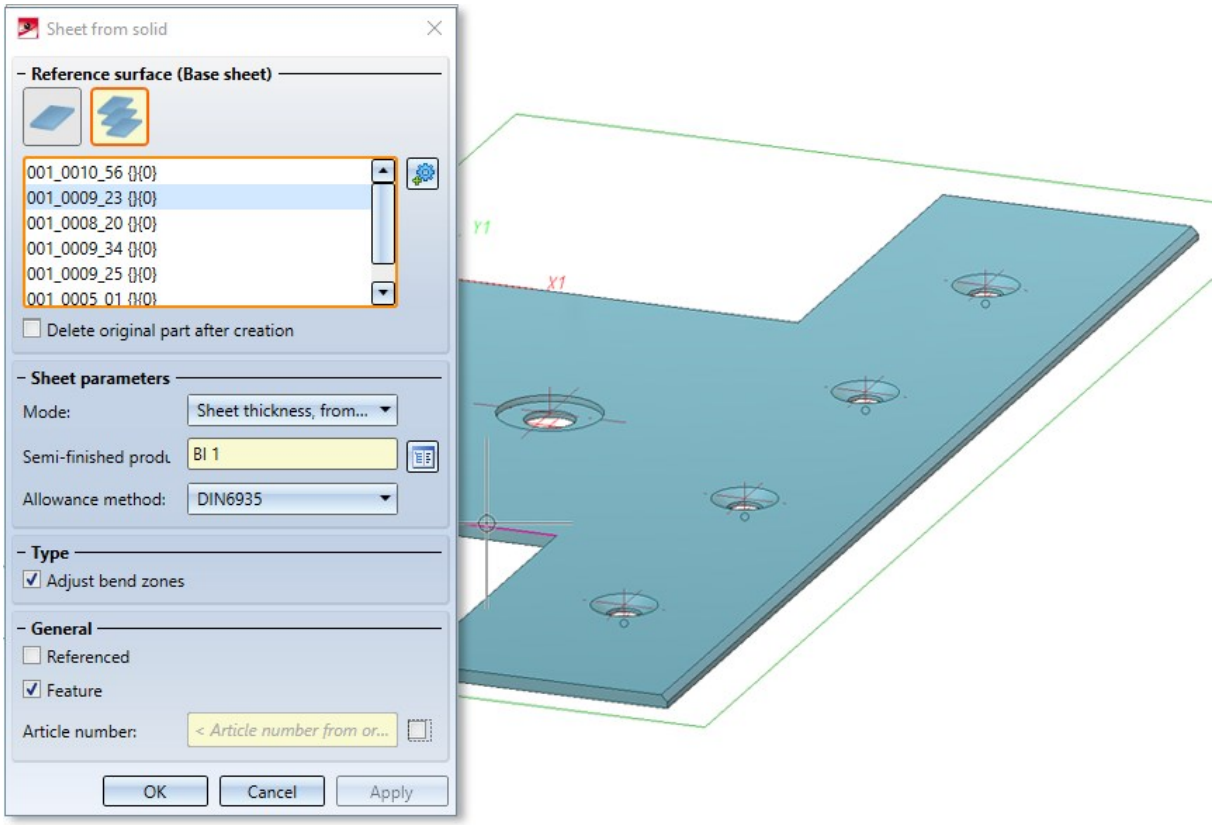
Sheet from solid

The **Sheet from solid** function has been reworked for convenient generation of Sheet Metal parts. In addition to individual selections, multiple selections are now also possible. Here you can select the parts in the model drawing after calling the function. If you have activated an assembly or several main parts in the ICN before you call the function, these parts also appear in the list of active parts.

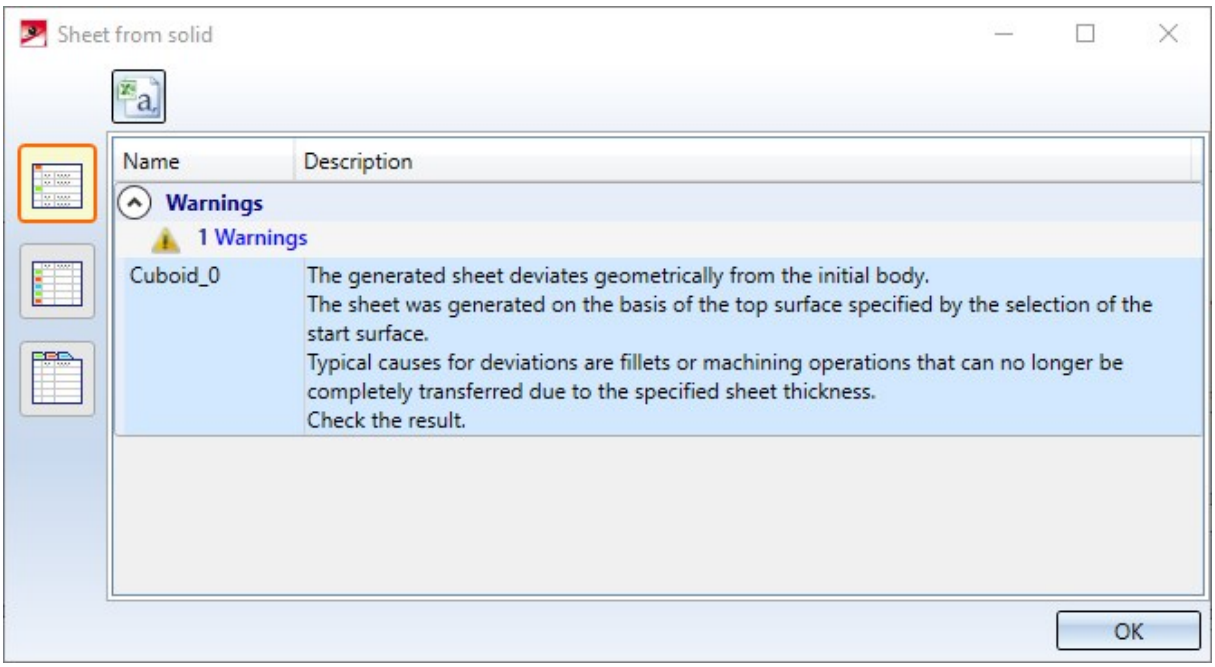
The preview of the new sheet is also displayed. Choose **Mode** to select the evaluation type for sheet thickness.

Sheet thickness from semi-finished product	In this mode you select the sheet thickness with semi-finished product.
Automatic sheet thickness from semi-finished product	The sheet thickness is automatically selected to match the selected semi-finished product.
Sheet thickness, from solid	The sheet thickness is taken from the solid.
Sheet thickness, value input	The sheet thickness is entered directly.

If the initial part contains chamfers, these are transferred to the sheet metal part..



If deviations from the geometric initial body occur during generation, they are listed after the sheet metal parts have been generated:

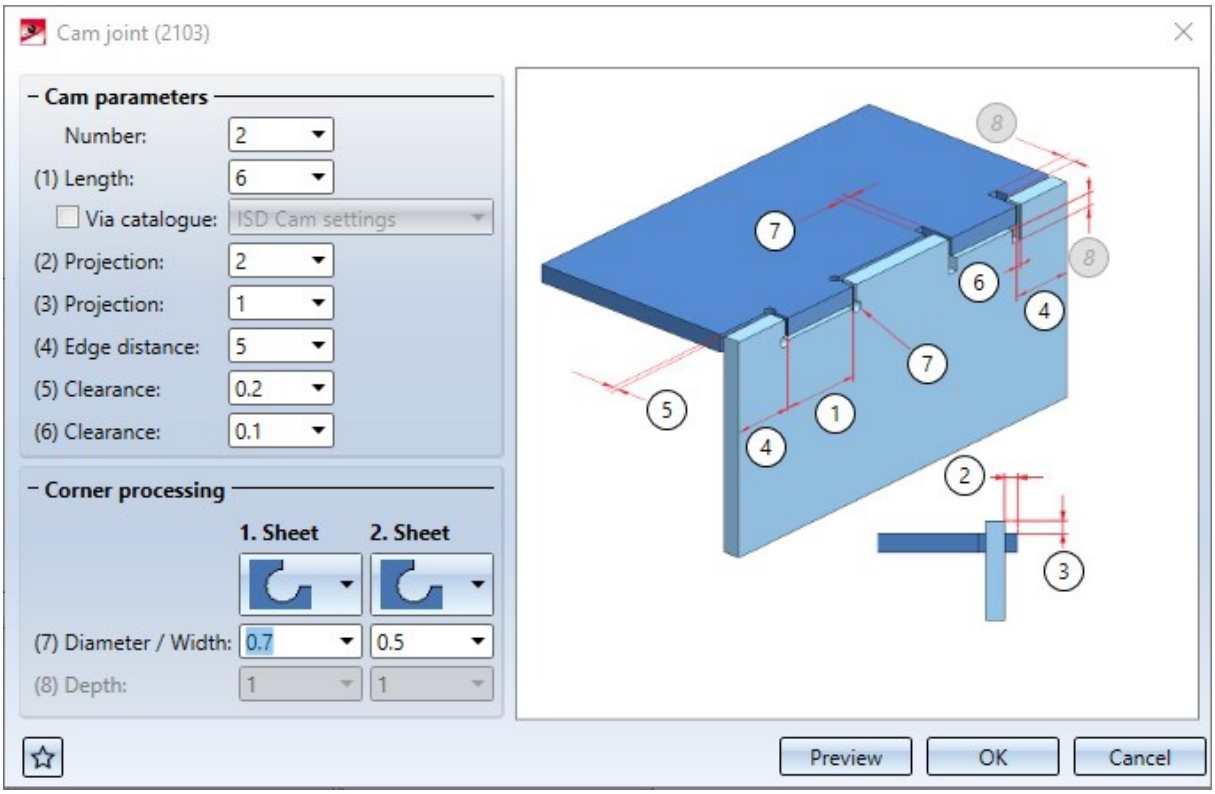


Favourites in dialogue windows for Design Variants

For the Sheet Metal Design Variants the Favourites management is now also available.

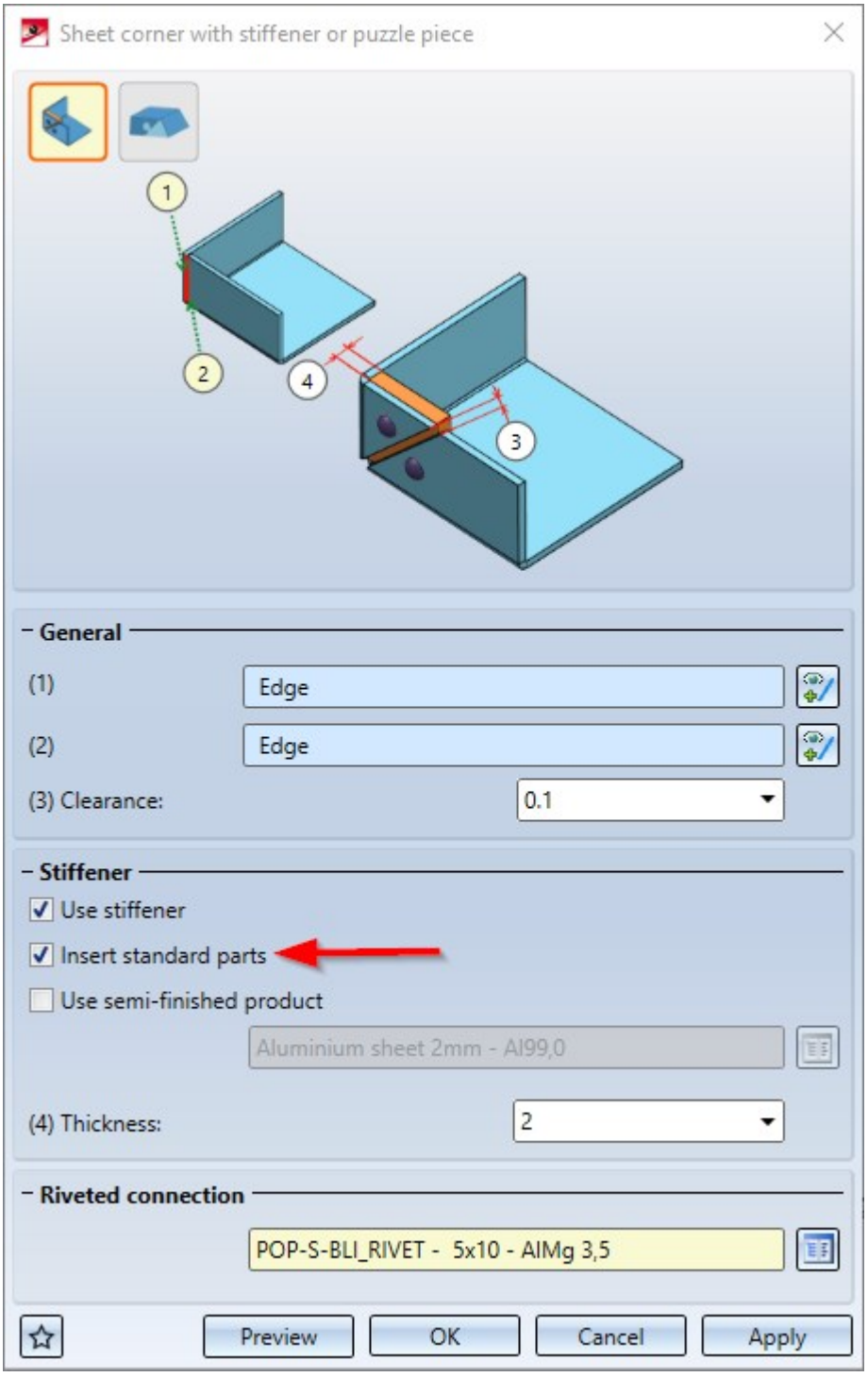
The settings of the various dialogue windows can be saved as favourites and reused at any time. To do this, click on the ☆ symbol at the bottom left of the dialogue window.

The Favourites supplied with the software are only examples and may have to be adapted to your company's specific needs. You can find more information on Favourites Management in the **HiCAD Basics** Help.



Optional standard part insertion

When using the design variant **Sheet corner with stiffener** it is now possible to choose whether standard parts (rivets, screws etc.) should be installed or not. For this purpose, the checkbox **Insert standard parts** has been added to the dialog box. The default setting is that standard parts are created, i.e. the checkbox is active



If the checkbox is not active, the standard parts will not be listed in the BOM.

No view-defining coordinate system on flanges and bend zones

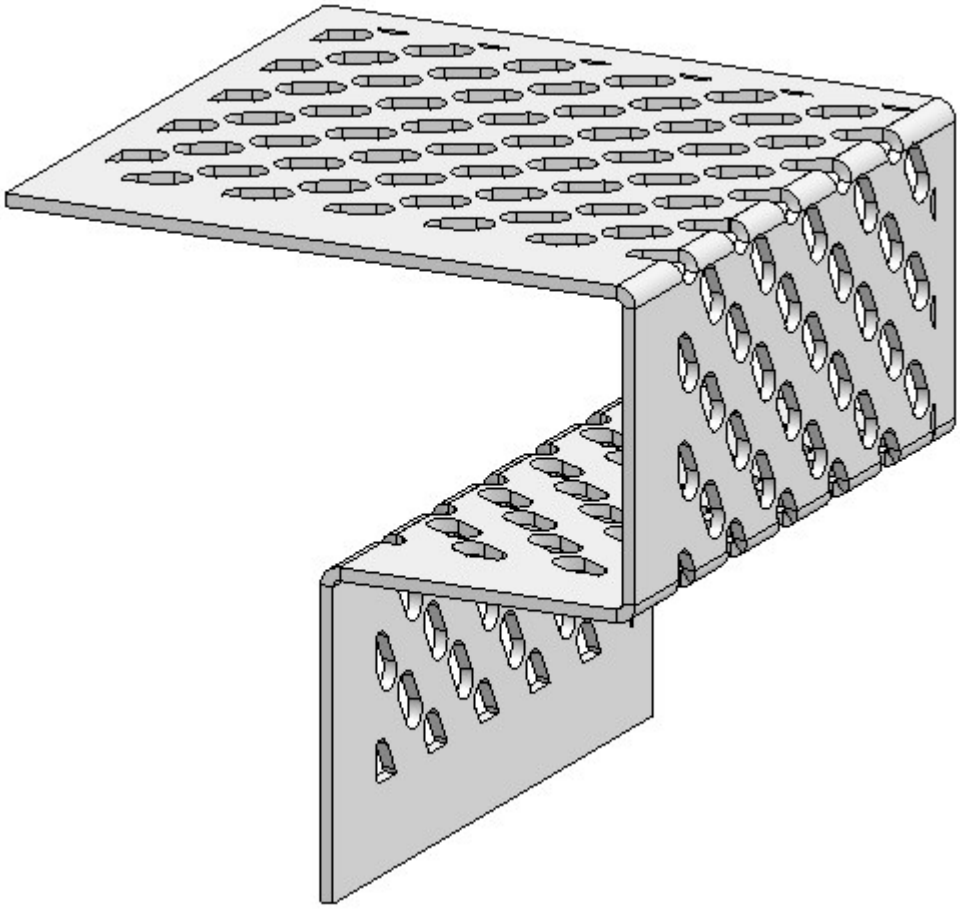
For Sheet Metal parts, you can use the **Properties > Part orientation** function in the context menu (right-click) to define which view is to be the **Front view** or the **Top view** for the orientation of derived drawings (workshop drawings).

Since workshop drawings cannot be created for sheet flanges and bend zones, the orientation from HiCAD 2020 SP1 onwards always refers to the sheet metal main part.

You can delete orientations of sheet flanges and bend zones that were set in older versions using the **Assignment reset** function. For resetting, the Sheet Metal main part must be active. The view orientation of the main part is displayed for these sheet flanges and bend zones.

Hole patterns on sheet developments

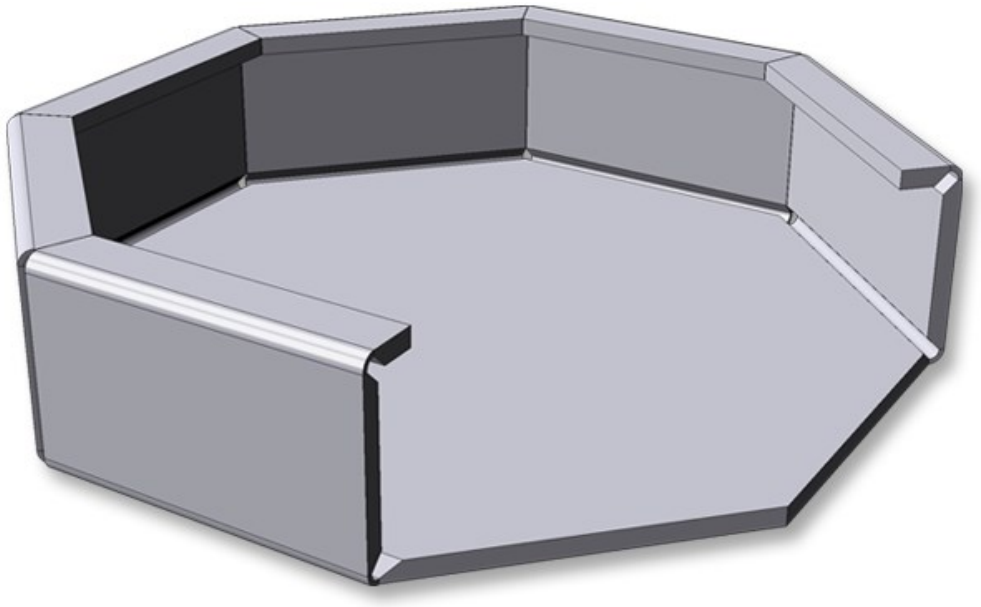
Hole patterns can now also be applied to sheet developments.



Major Release 2020 (V 2500)

Sheet Metal cassette with milling edge zone

You can now use the **Sheet Metal cassette** Design Variant to create Sheet Metal cassettes with milling edges zones and inverted milling edge zones. To do this, simply select the desired **Mode** on the **Attach flange** tab of the dialogue window.



Sheet Metal cassette with milling edge zone

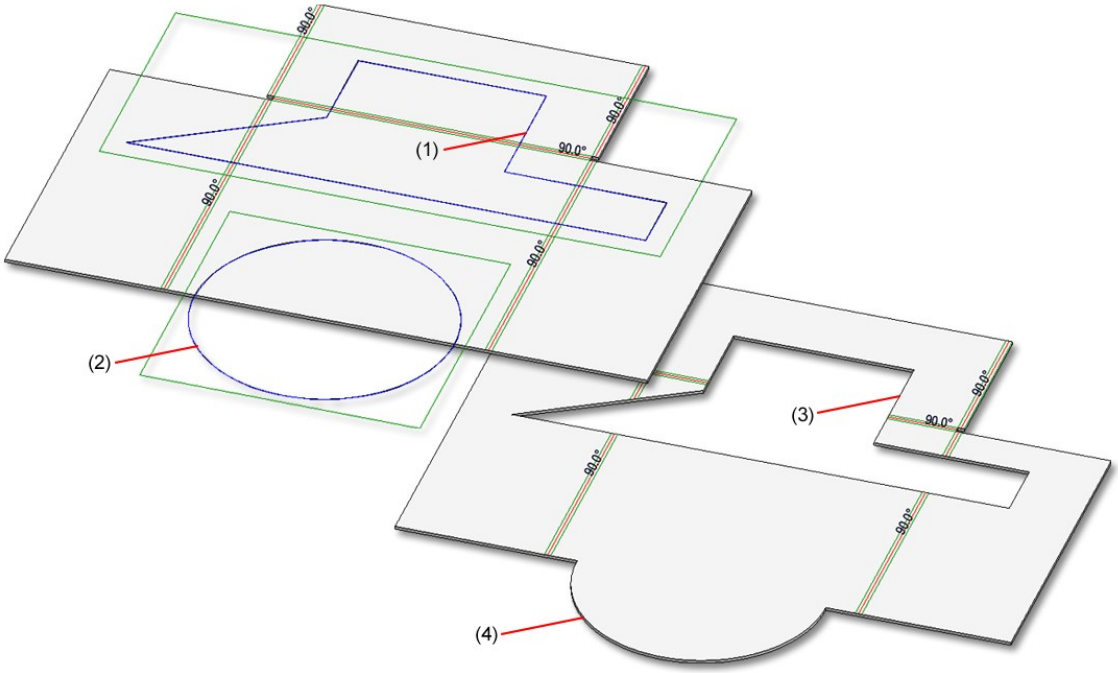
Sheet developments

Process sheet developments

Sheet developments can now be edited with assigned sketches. To do this, you can select the new **Sketch (sub-part)** function by right-clicking on the development in the menu that appears before processing. After placing a processing plane, draw a sketch, for example, for a subtraction. The ICN now displays the development with the sub-part called **Sketch**. This allows you to activate the sketches you have created in advance when you process the development, for example, with the 3-D function **Subtract, with translation**. The processing plane is visible if it is switched on via **Show/Hide elements** for the view. If the sketch is generated from the dialogue window of the processing function, it is also located in the ICN under the development.

With the following functions on the **3-D Standard** tab you can now also process developments:

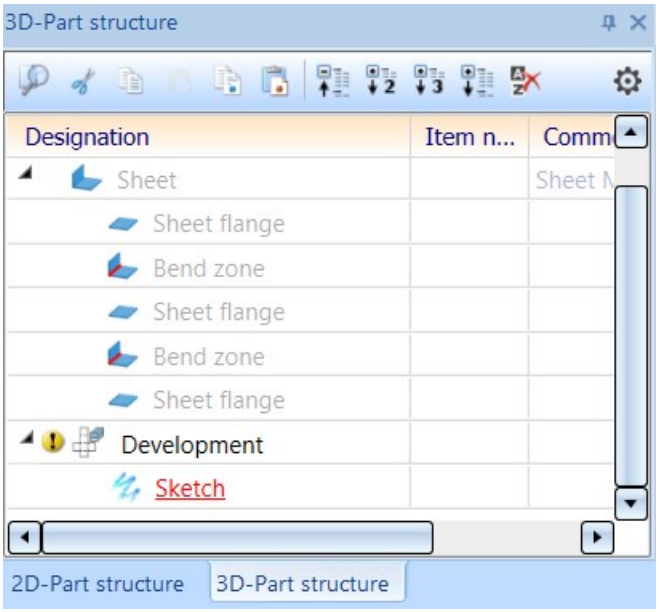
- Subtract, with translation
- Subtract, with rotation
- Add, with translation
- Add, with rotation



(1) Sketch for subtraction
(2) Sketch for addition
(3) Subtraction
(4) Addition

Developments in the ICN

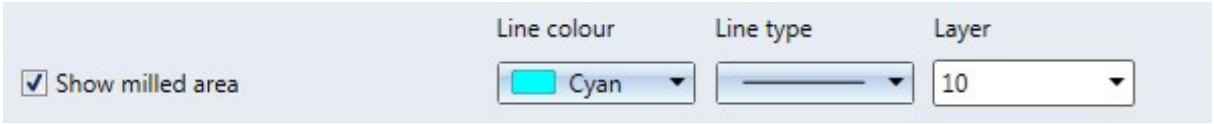
The development now also appears in the ICN below the Sheet Metal part and can contain sketches as sub-parts. If the development is active the sheet metal part is greyed out. Activate the development in the drawing or by selecting the view.



Show milled area

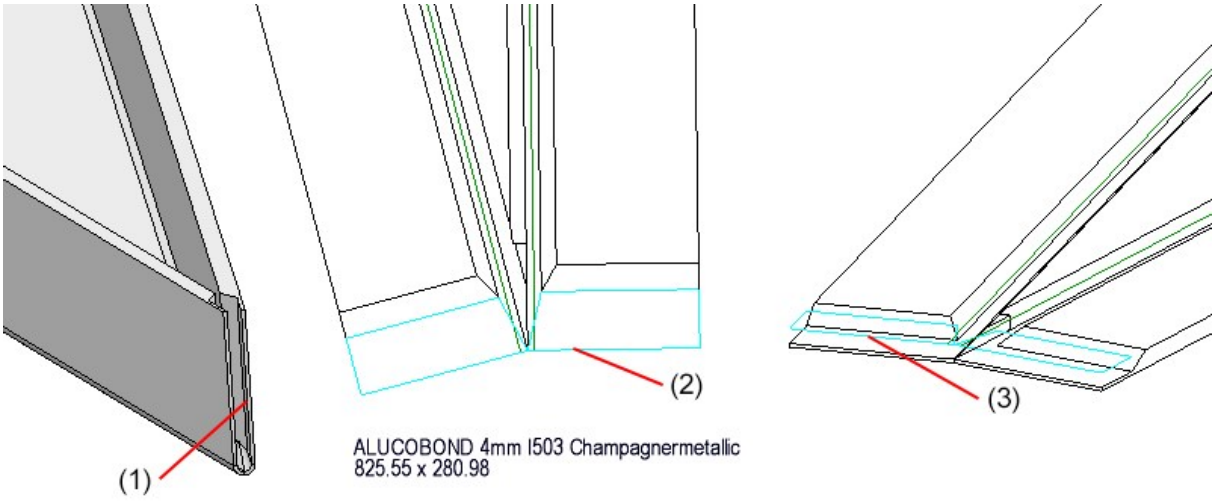
You use the **Show milled area** option to mark exclusions that you have defined, e.g., with the 3-D function **Subtract part, via translation** . For the triangular SZ-20 panel in Element Installation, too, the material in the "acute corner" will be excluded (milled away). The milled area will be recognized when creating the development. That is, facets which are no cover facets, which run parallel to the cover surface, and which have no standard processings form the milled (i.e. subtracted) area.

To prepare this area for machining, for machine control during DXF output, a layer must be defined for this milled area.



If you activate this option, the edges of the milled area will be entered in the development. The edges are sorted and oriented as specified for inner cycles. Milled areas are combined across milling edge zones.

The generation is activated in the development Favourite for Cobus NCAD: Color 10, Continuous line, Layer 10 In HCADACAD_COBUS.DAT the layer assignment MILLING is defined for the DXF export.



(1) Exclusion in Sheet Metal panel
(2) Exclusion in development (top view)
(3) Exclusion in development (Isometry)

Moveable bend line text

If you move the mouse over a bend line text during 3-D development, the text will be displayed in magenta. If you now move the cursor over the text, press and hold down the leftmouse button, you can move the text. If you move the bend line text along the bend line, the position will now be retained after updating the development.

Export of sheet developments

Optional taking into account of processings for ToPs GEO export

If you choose ToPS GEO, you can determine via the **Export forming edges** option, whether processings (forming tools, cross-breaks) are to be considered for export.

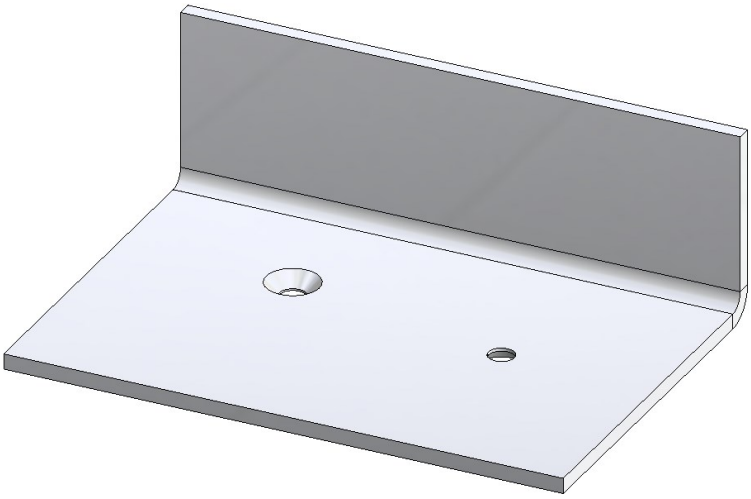
Determine polyline orientation for DXF export of Sheet Metal parts

When generating the DXF export files for Sheet Metal blanks, the orientation of the polylines in the created DXF file will now always be always the same (i.e. always clockwise or always anticlockwise), since CAM programs require these for determining the outer/inner side when generating the milling tool paths.

The orientation of closed lines can be set when exporting sheet metal developments. In the Configuration Editor at **Sheet Metal > Sheet development** you can set a clockwise (activate checkbox) or anticlockwise orientation (deactivate checkbox) for the inner and outer contours.

Sheet from solid

When generating a sheet from a solid with two countersinks with a conical surface (e.g. DIN74) on 2 sides, the geometry is correctly recognized regardless of the start surface.



Sheet from solid, with 2 countersinks (DIN74) on two sides

The old **Sheet from solid** function has been replaced with the same-named new function at **Sheet Metal > New**. The old function could still be accessed by right-clicking on a bend zone and choosing New > From solid.

ALUCOBOND®SZ-20 tray panels: Lateral connection with additional flange

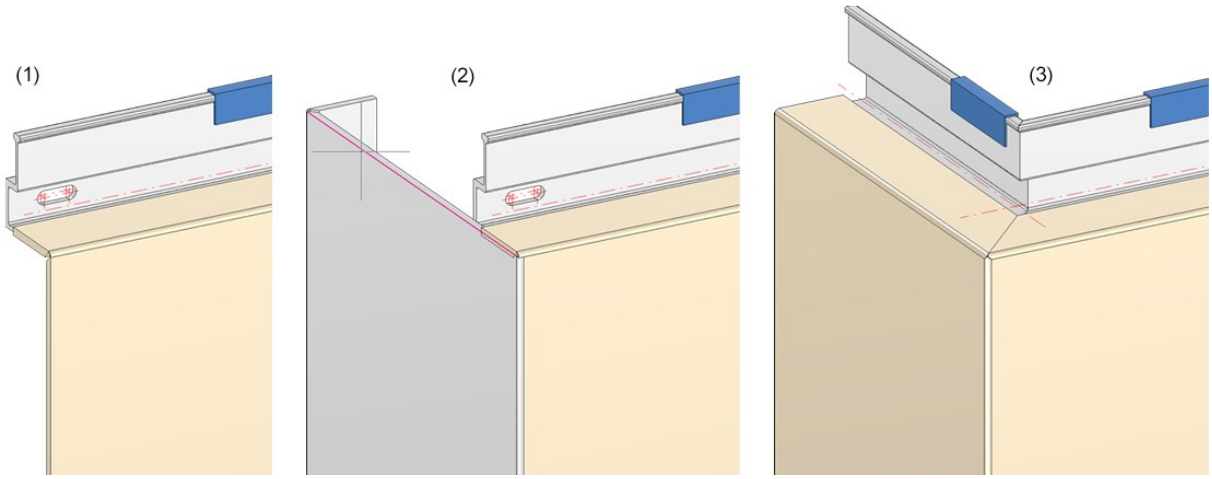
When using ALUCOBOND® SZ-20 tray panels, there is sometimes the wish to create a connection with a flange at the side of the panel.

This application cannot be realized directly via the Element installation dialogue window!

To create such a connection, proceed as follows instead:

1. Insert the SZ-20 panel via element installation, but select the setting **Without flanges** for the left and/or right connection.
2. Attach the new flange on the open side of the panel (optionally with 35mm side flange or side flange with the selected tray panel depth).
3. Start the Design Variant **Flange for for SZ20** under **Sheet Metal** in the **Civil Engineering functions** docking window.
4. Then select the upper or lower longitudinal edge of the previously bent new flange.

Example:



(1) Element installation without flanges
(2) Attaching of flange with 35mm side flange, calling of the Design Variant and selection of the highlighted edge
(3) Result

Please make sure to observe the following instructions:

The macro only applies to ALUCOBOND®SZ-20 tray panels where the standard connection is selected for the vertical section.

The side connection is installed without stiffeners.

No automatic connection to the sub-structure is possible, therefore the checkbox **Create screws for sub-structures** in the **Extended settings** of the **Element installation** should be inactive.

Favourites in Design Variant dialogue windows

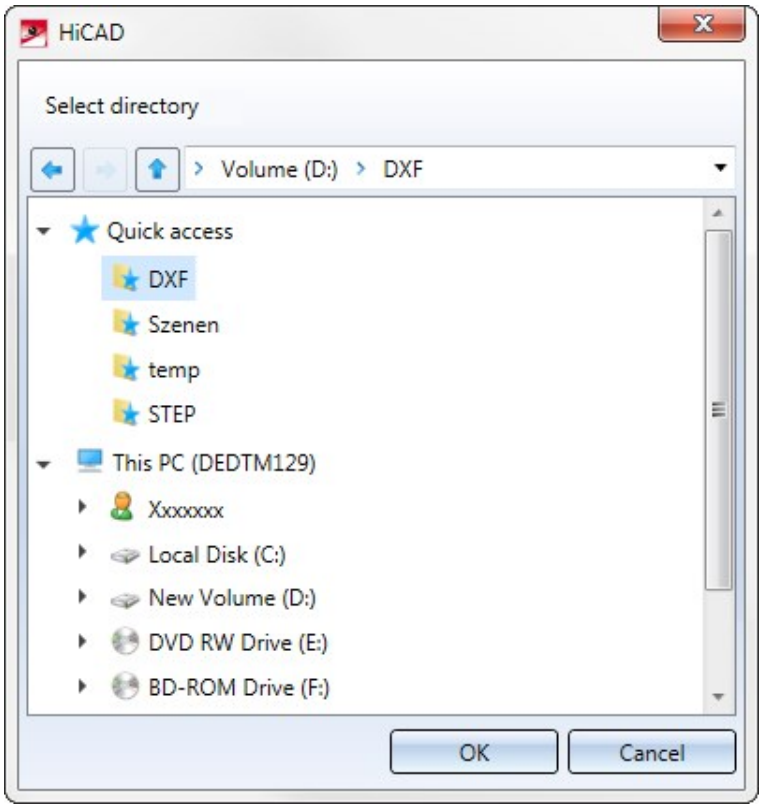
The settings on the dialogue window of the **Sheet corner with stiffener or puzzle piece** Design variant can be saved as favourites and re-used at any time. To do this, click on the ☆ symbol at the bottom left corner of the dialogue window. More on favourites management can be found in the Manage Favourites topic of the **HiCAD Basics** Help chapter.

Improved info bar texts

The user guidance texts in the info bar at the bottom left have been improved for the functions **Cut off corner** and **Create base sheet** for an easier operation.

Quick Access for directory selection

As with the dialogue for loading and saving of files, selected directories can now also be added to the **Quick Access** folder, e.g., for DXF export of Sheet Metal parts.



ABWPOL.DAT settings moved to Configuration Editor

The settings of the ABWPOL.DAT file have been moved to the Configuration Editor (ISDConfigEditor.exe).

You now find the settings for the neutral axis for approximative sheet development at **Sheet Metal > Default setting**.

Processing direction

New arrow sizes have been added to the **Direction arrows** table (in the catalogue **Factory standards > Symbols > Arrows**. They are available when you choose the **Processing direction** function. (3-D Standard > Standard Processing > Bore > Processing direction).

Discontinuation of old base sheet macro

From HiCAD 2021 onwards, the calling of the old Sheet Metal base sheet macro 'abwgrundblech.mac' will no longer be supported.

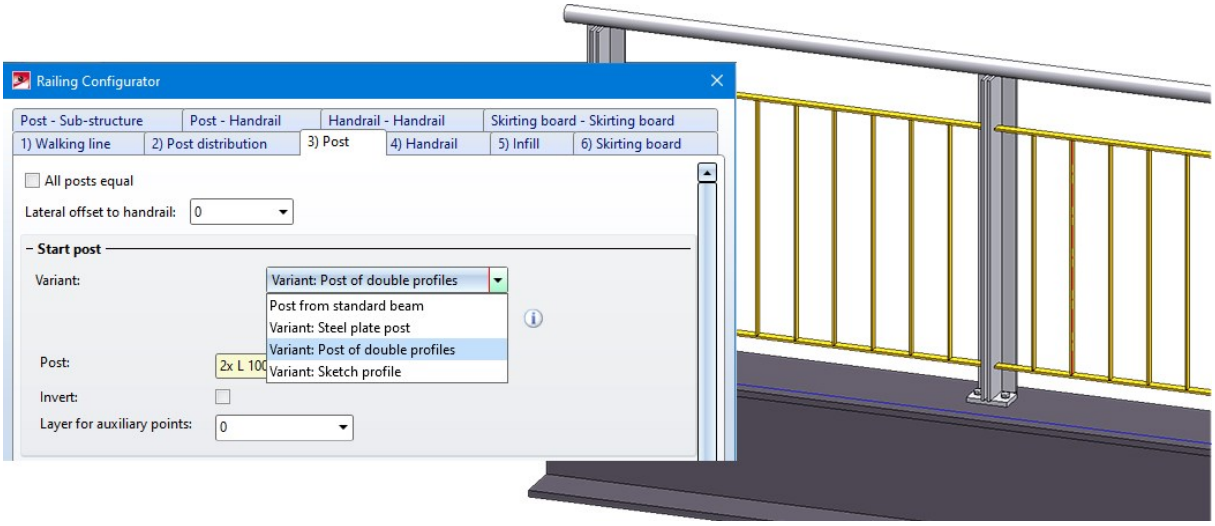
Steel Engineering

Service Pack 2 2020 (V 2502)

Railings

- Posts from double profiles

When creating railings, double profiles can also be selected from catalogue **Factory standards > Railing > Railing profiles > ISD Examples - Double profiles** from SP2 onwards.



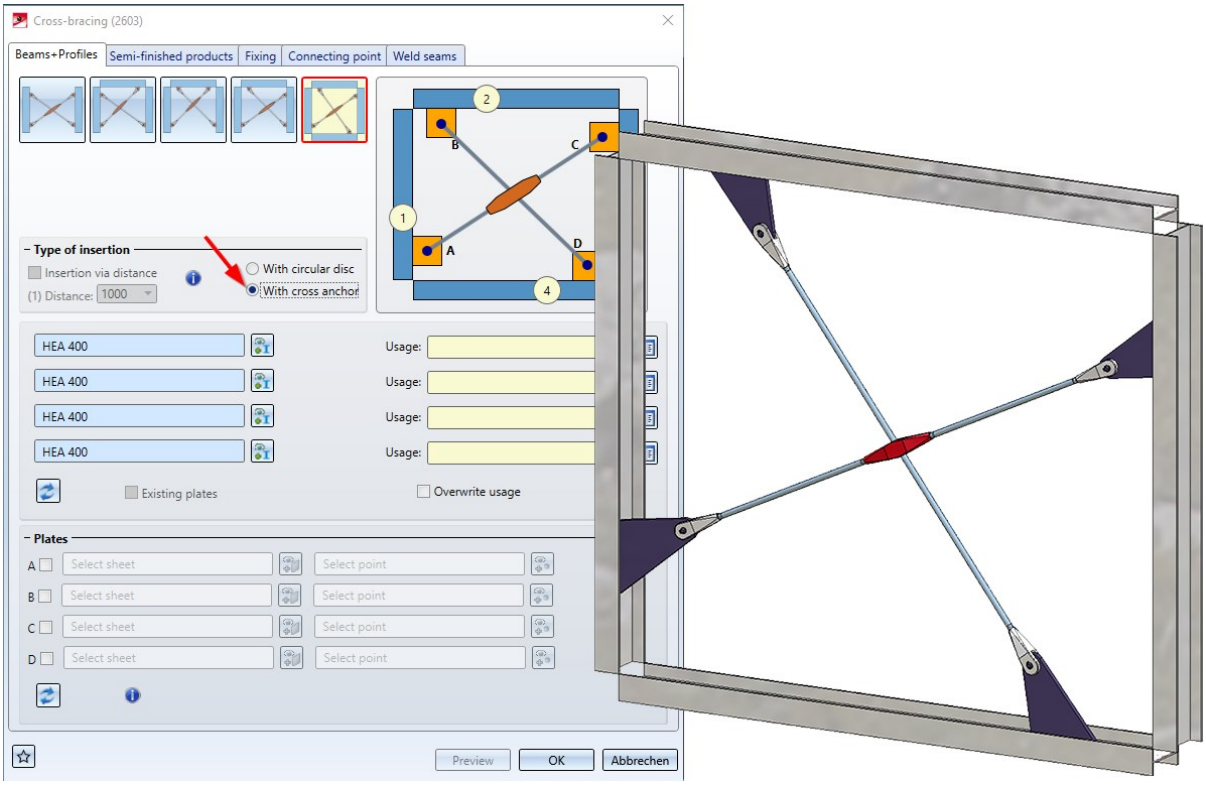
Connections

Cross-bracing between two perpendicular beam (2602)

With the Design Variant 2602 it is also possible from SP2 onwards to connect two perpendicular beams to each other with a diagonal using a cross-bracing.

Cross-bracing Besista with cross anchor (2603)

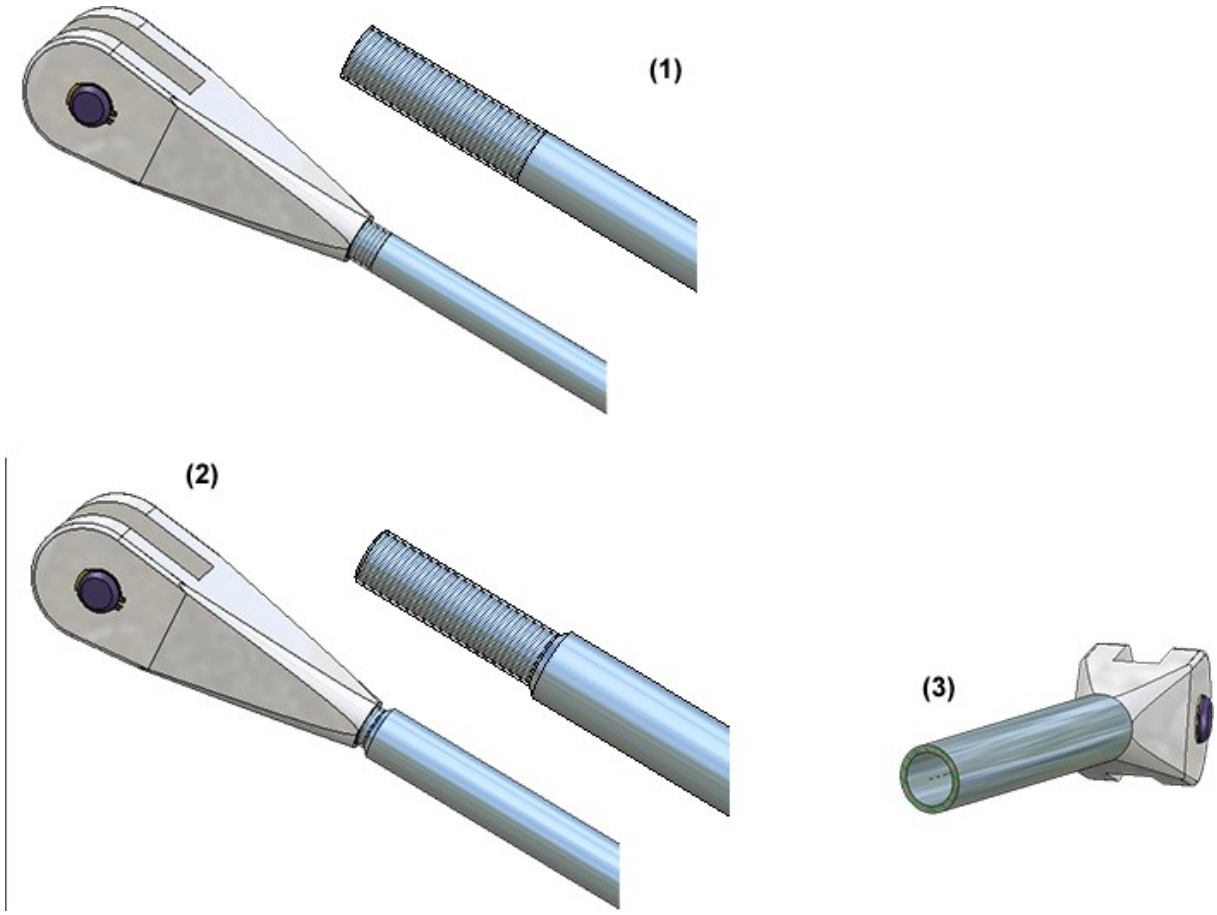
The Design Variant 2603 has been extended by the insertion type **With cross anchor**. It is now possible to connect two, three or four beams/plates with a tension rod system consisting of tension rod and rod anchor and a Besista cross anchor. The cross-bracing can also be attached to plates that already exist on the beams.



Stabilizing pipe connection (2702)

The new Stabilizing pipe connection (2702) Design Variant creates a stabilizing pipe connection of the type BESISTA between 2 beams or plates. You can choose between the following options:

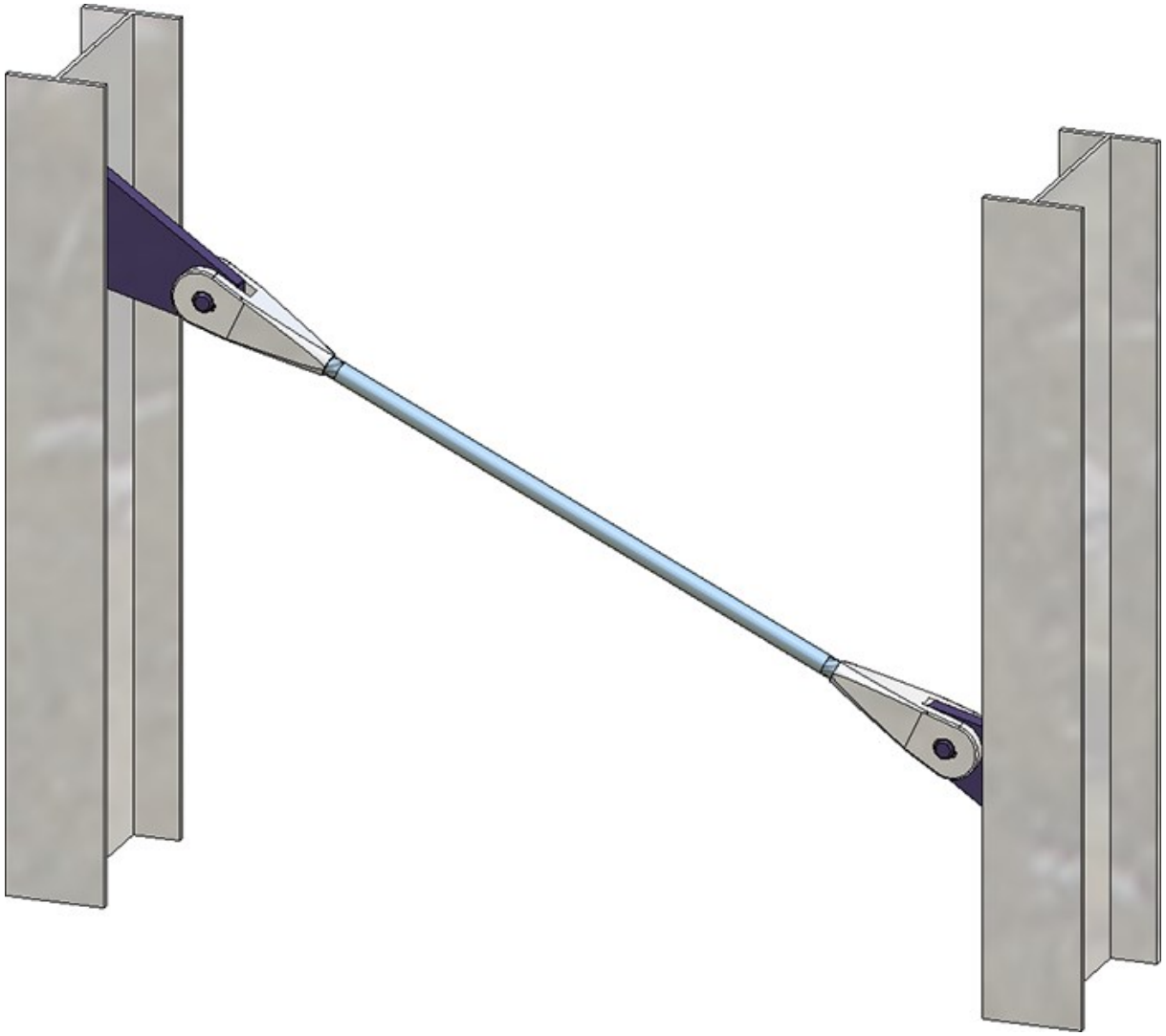
- Tension rod system (Besista)
- Pressure rod system, Solid rod
  consisting of 1 Besista solid round steel rod with conical ends, 2 Besista rod anchors, 2 Besista studs and retaining rings. The round steel rod and the rod anchors are welded together.
- Pressure rod system, Steel pipe
  consisting of a steel pipe, 2 Besista rod anchors, 2 Besista studs and retaining rings. The steel pipe and the rod anchors are welded together.



(1) Tension rod system, (2) Pressure rod system: Solid rod(3) Pressure rod system: Steel pipe, with sectional view

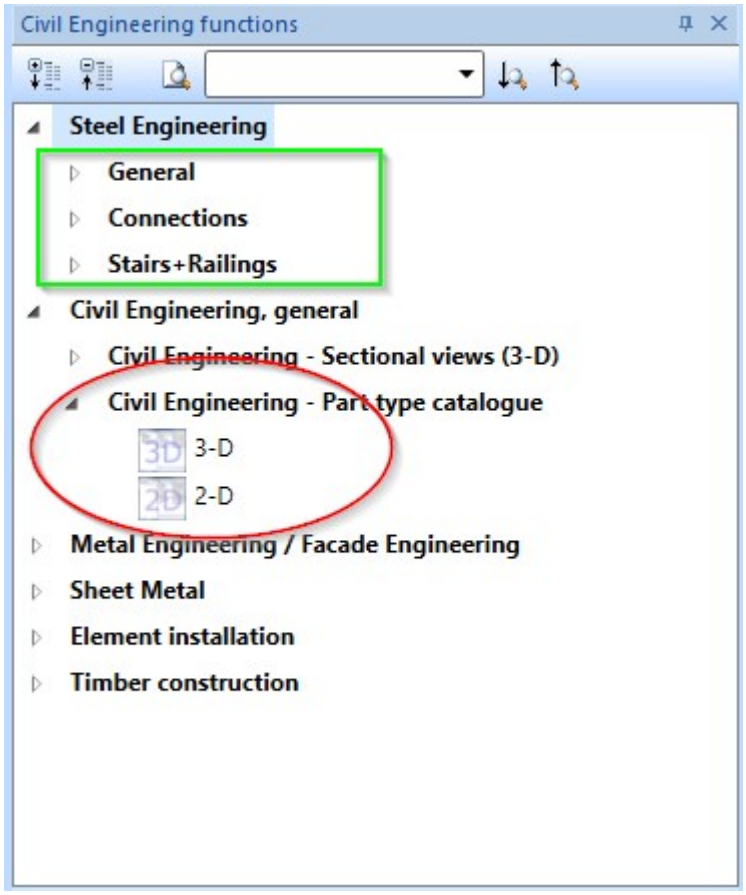
When connecting beams, the stabilizing pipe connection normally already provides the gusset plates. However, you also have the option of attaching the stabilizing pipe to plates that already exist on the beam.

Please note that when fixing to plates, the plates must lie in one plane.



Macro-based connections

The macro-based connections for gusset plates and base plates have been removed from the **Civil Engineering functions** docking window and are no longer available in SP2. To insert them from the Part type catalogue, use the **Civil Engineering, general** entry in the tree structure.



Steel Engineering plates - Visible side and coating

The functions

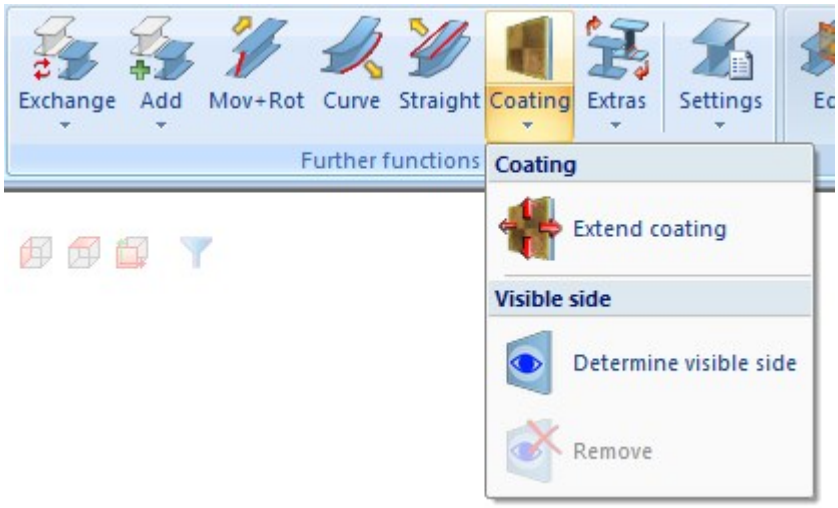
Coating

Extend coating

Visible side

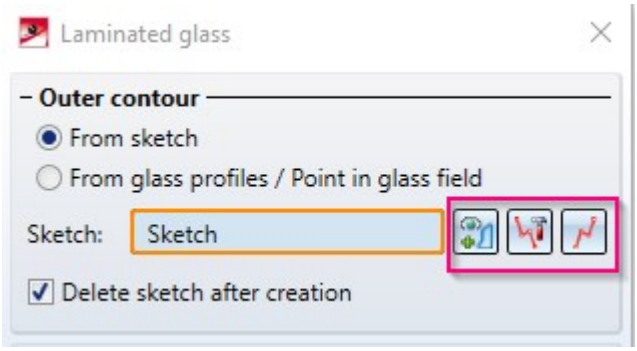
Remove visible side

can now also be used for Steel Engineering plates. As of SP2 you can find these functions in the Steel Engineering Ribbon at Steel Engineering > Further functions.



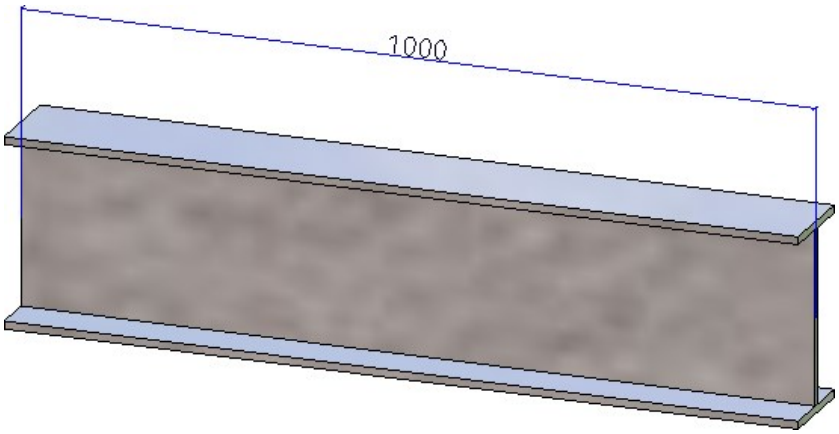
Sketch selection in the Laminated glass dialogue window

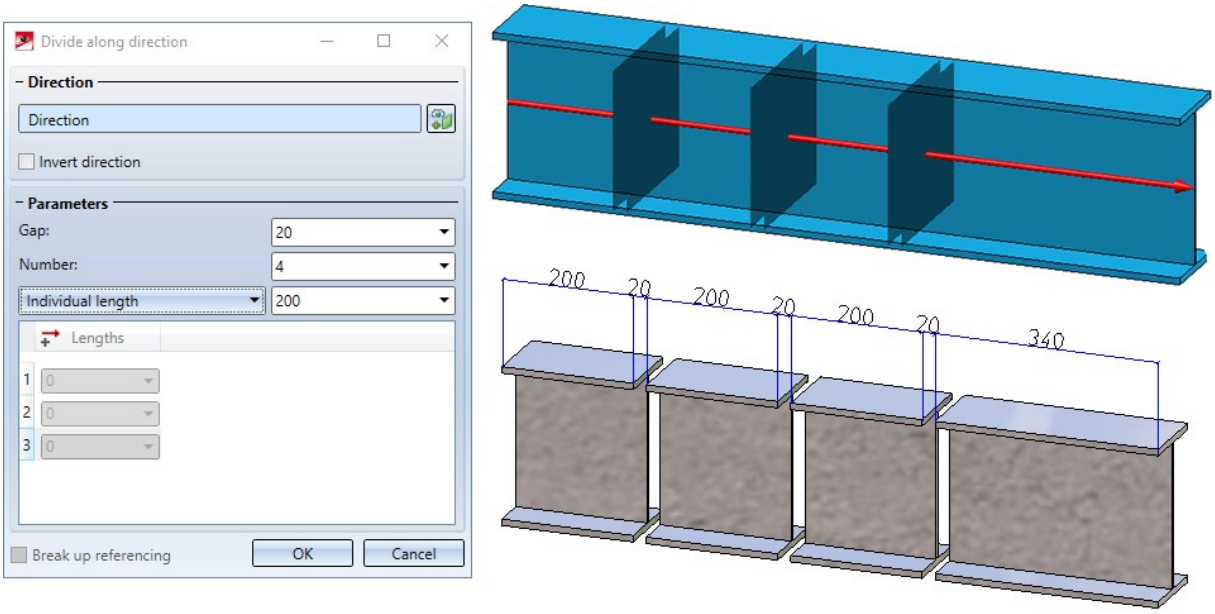
The **Process sketch** and **New sketch in plane** buttons are now available directly in the **Laminated glass** dialogue window, and no longer in a pull-down menu.



Divide along direction

The new function **Divide along direction** at **Steel Engineering > lengthen > Divide** allows you to divide any 3-D part of the type Solid into several parts along a freely selectable direction. Various division options are available, for example the individual parts can have different lengths. In addition, this function creates a corresponding feature so that the division can be edited later.





This function can also be found at **3-D Standard > Process > Trim**.

More convenient creation of sketch-based series beams and profiles

When creating your own sketch-based series beams/profiles, it is now possible to conveniently prepare the beams/profiles using the functions **Assign attributes** and **Series cross-section, Save as catalogue part** and save them to the catalogue.

Service Pack 1 2020 (V 2501)

Connections

Favourites management

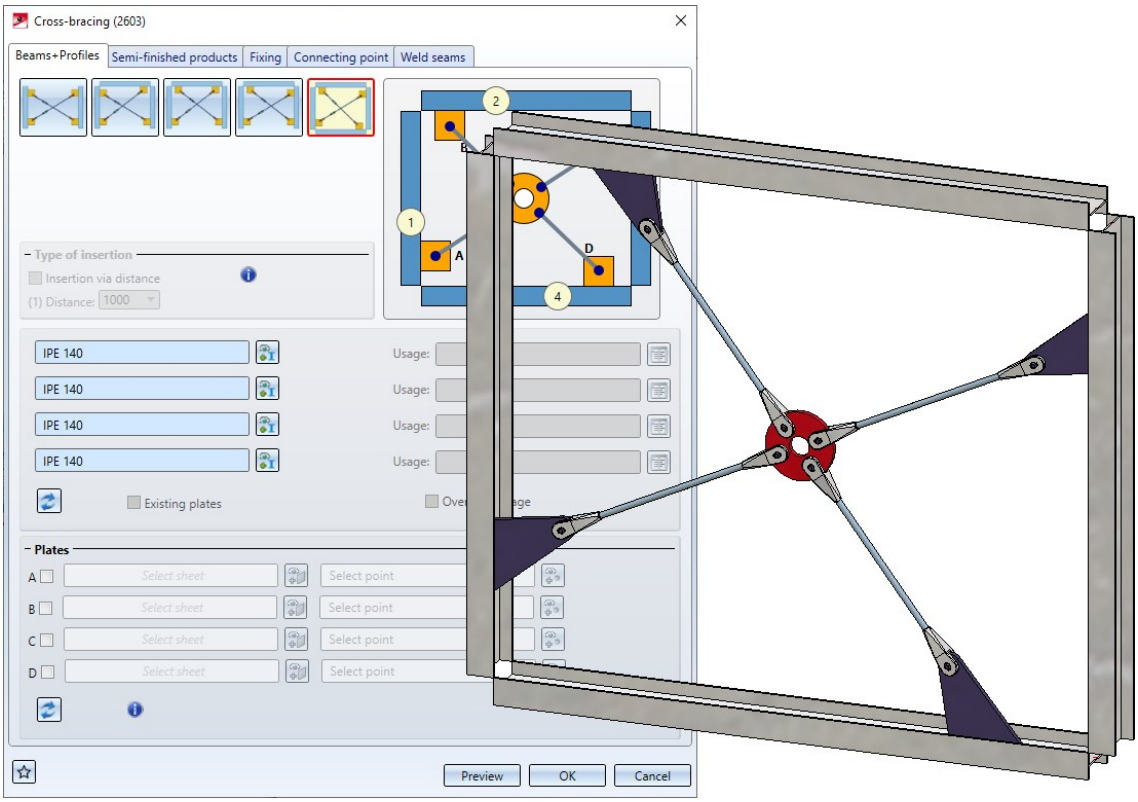
For all connections based on design variants and when creating stairs and railings, the settings of the respective dialogue window can be saved as favourites and reused at any time. To do this, click on the symbol at the bottom left of the dialogue window to activate the context menu. For more information on Favourites management, read the Manage Favorites topic ofm the HiCAD Basics Help.

Strap joint (2310)

In the dialogue window of the **Strap joint (2310)** a new tab for **Weld seams** is now available.

Cross-bracing (2603)

This new design variant connects two, three or four beams or plates with a cross-bracing of the type "Besista with Circular disc". Here the beams/plates are connected to a circular disc by a Besista tension rod system consisting of rod and anchor. The cross-bracing can also be attached to plates that are already present on the beams.



BOM templates for Steel Engineering

From HiCAD 2020 SP1 onwards, the BOM templates for Steel Engineering:

- hicad_stahlbau_01
- hicad_stahlbau_02
- hicad_stahlbau_03
- hicad_stahlbau_04

are no longer available. They have been replaced by a new template, hicad_stahlbau, in the HiCAD configuration. This applies in particular to the usage-dependent setting in the Configuration Editor at **Automatic drawing derivation > Production drawing > Usage-dependent > ... > View group > File name: BOM**.

For update installations everything remains unchanged.

Please note:

Derived drawings of earlier HiCAD versions in which these BOM templates are used will not be changed.

Others

Steel Engineering BOMs - handling of structure assemblies

Steel Engineering BOMs now allow you to define in the Excel template (HiCAD_Stahlbau.2501.0.xlsx) how BOM-relevant structure assemblies should be handled in the shipping list. For this purpose the row **Ignore structure assemblies** is available on the sheet **Settings**.. (see also Report Manager - What's New?)

Changed default settings for Notches

For the settings defined in the Configuration Editor at **Steel Engineering > Notch** the default value for distance from the web has been changed to 0. The previous default value was 10.

Major Release 2020 (V 2500)

Indication of beam orientation

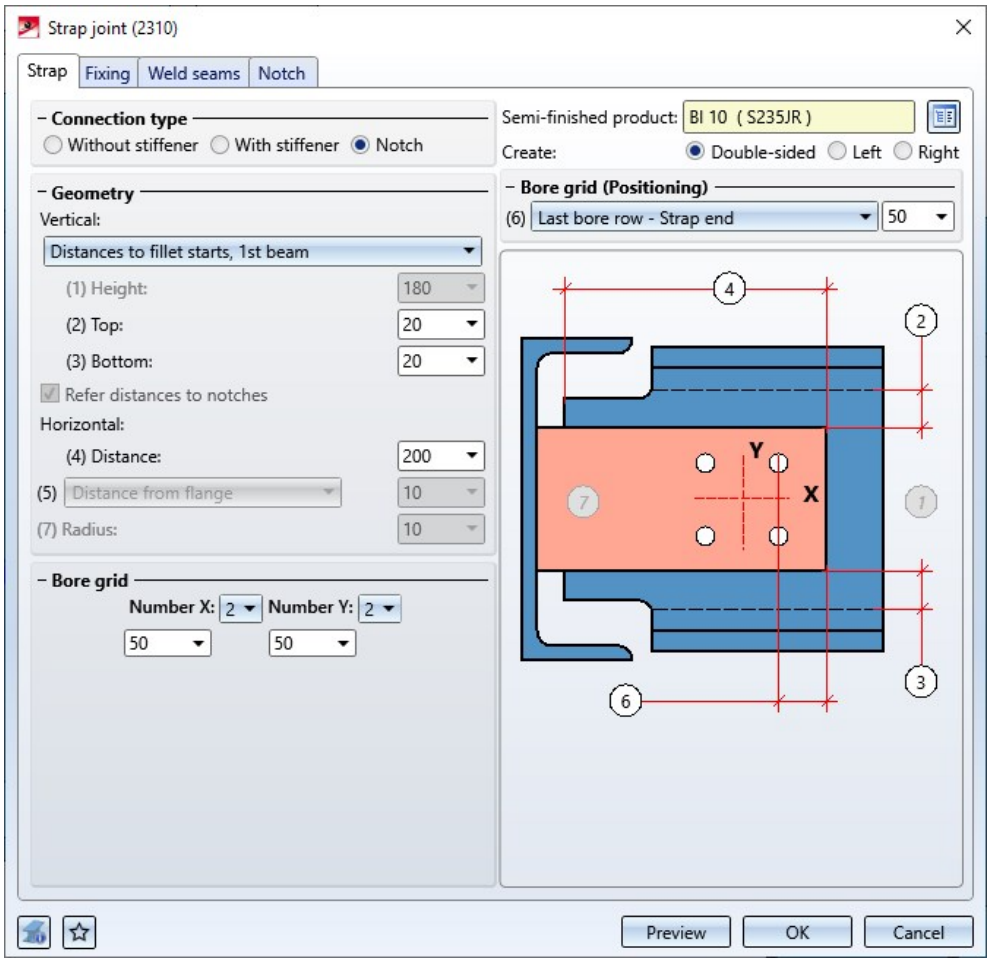
If the parameter **Indicate orientation of active Steel Engineering beam** is set to **Switch on/off with F6 key** in the Configuration Editor at **System settings > Visualization**, the display can alternatively be switched on/off via the **Coordinate systems** toolbar (at the bottom of the dialogue window) by clicking on the

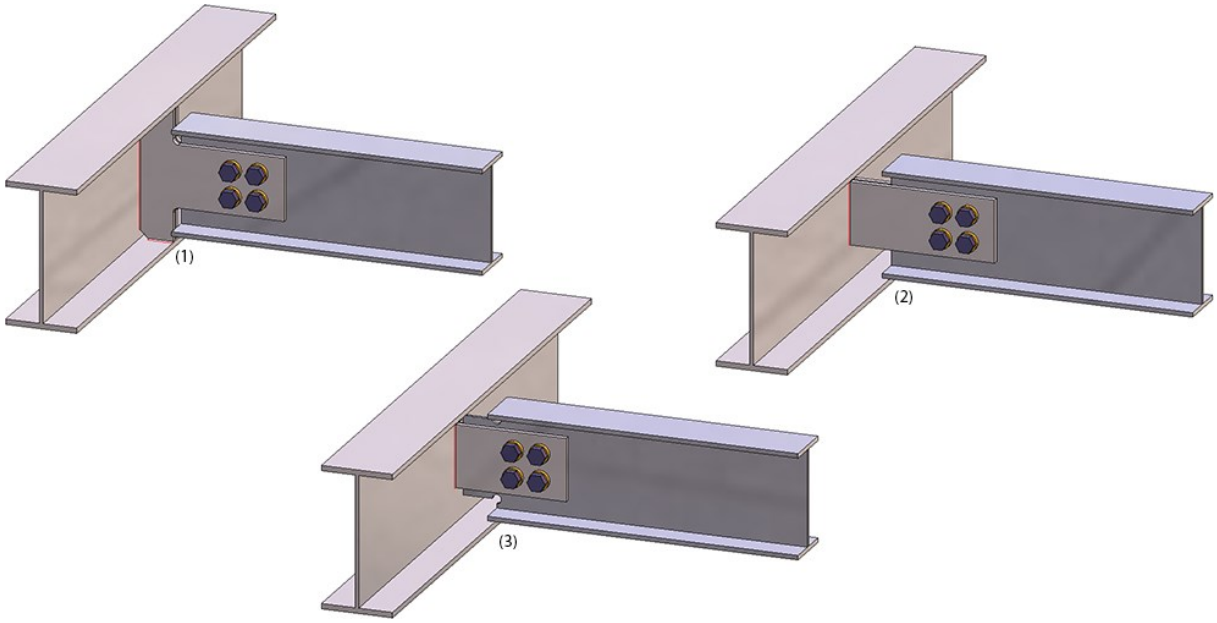
Show beam orientation or drawing alignment of the active 3-D part [on/off] symbol.

Connections

Strap joint (2310)

The previous Strap joint (1310) connection has had a redesign and has been replaced with the new **Strap joint (2310)**.

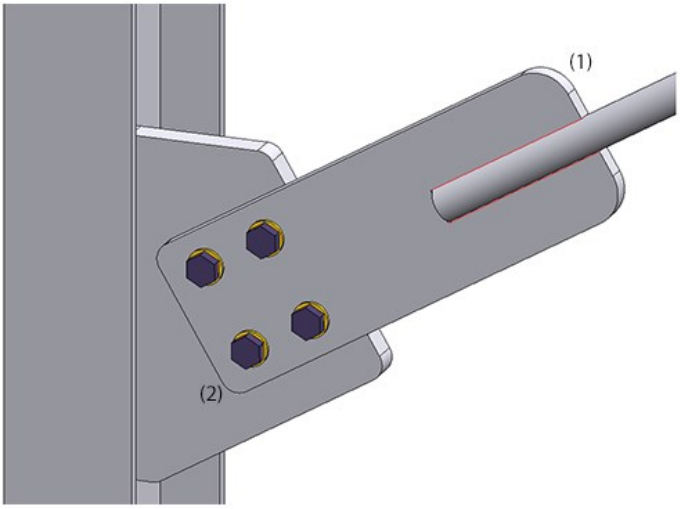


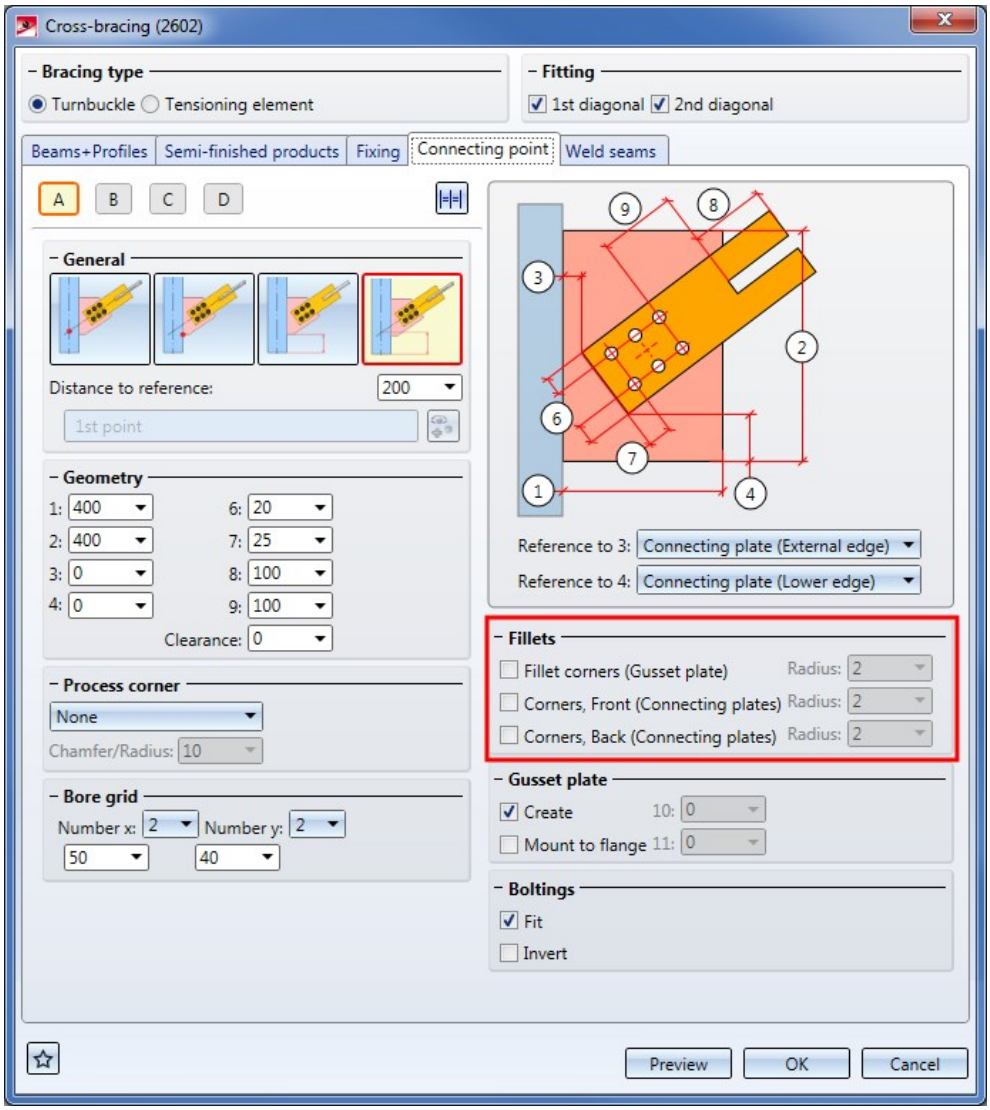


(1) With stiffener; (2) Without stiffener; (3) With notch

Cross-bracing (2602)

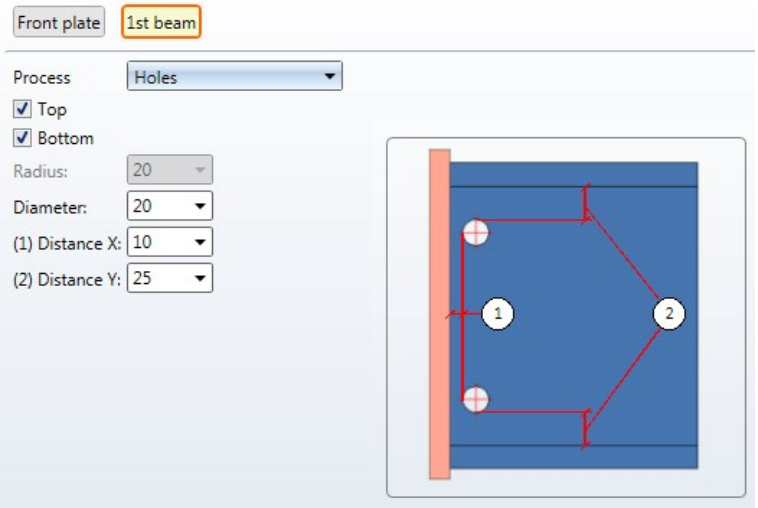
For the **Cross-bracing (2602)** with turnbuckle, different rounding radii for the front (1) and rear corners (2) of the connection plates can now be specified on the Connecting point tab.





Galvanisation

For all connections with a **Galvanisation** tab in the dialogue window, the graphic for the 1st/2nd beam with the processing type **Holes** has been changed. The distance "X" here is the distance to the web, the distance Y is the distance to the flange.

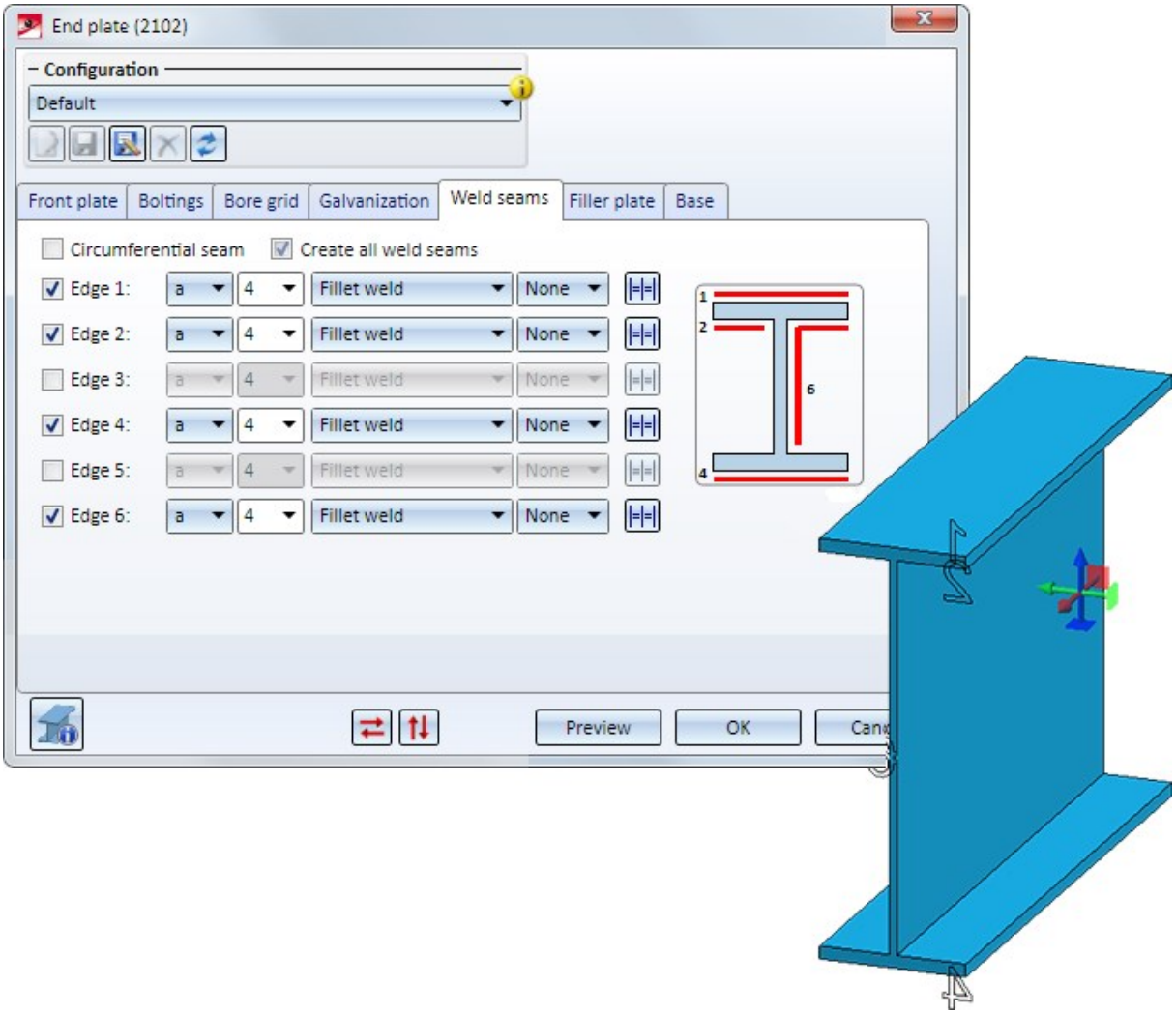


This change concerns the following connections:

- Base plate + Anchor plate (2101)
- End plate (2102)
- Purlin joint, 2 plates with mitre cut (2201)
- Beam to web with 2 plates and stiffener (1211)
- Column connection, Frame corner (2203)
- Column connection, Frame corner (2204)
- Front plate connection to web/flange (2320)
- Front plate connection to flange (2330)

Weld seam insertion

- For all connections that support weld seam insertion, the edges are now visualized according to the settings on the **Weld seams** tab. This allows a better assessment of the weld seam positions.

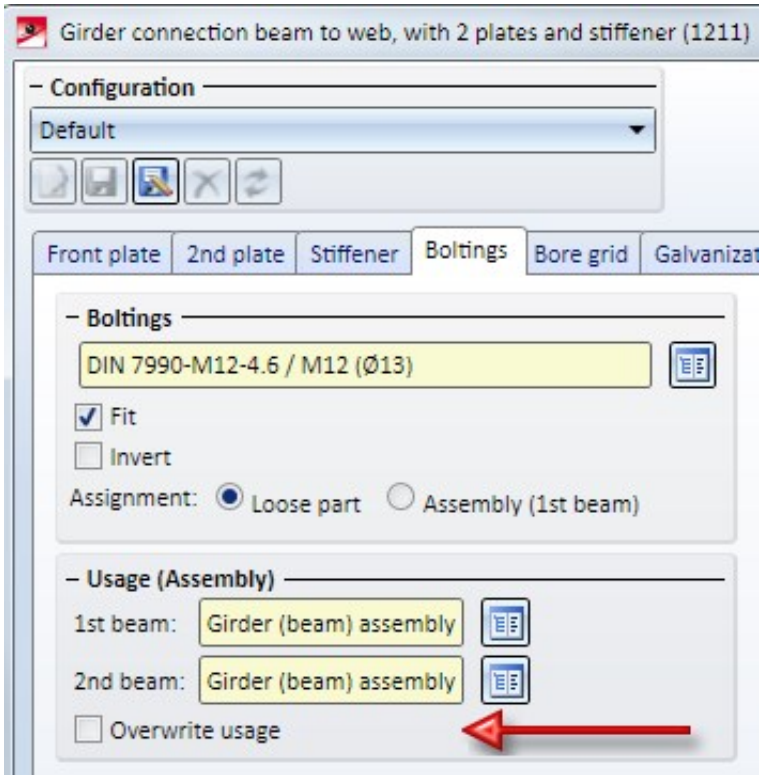


- From HiCAD 2020 onwards, you can specify the weld seam width for HV and HY seams in all connections that support weld seam insertion. For continuous weld seams, enter the value 0.

Usage of assemblies

Many connections create assemblies, or add further parts to already existing assemblies. For example, for Connection 1211 (Beam to web with 2 plates and stiffeners) the front plate and the corresponding weld seams of the first connecting plate will be assigned to the assembly of the first beam to be connected. The 2nd plate and the stiffener including the associated weld seams for the second connecting plate will be assigned to the assembly of the beam to which the connection is made.

A usage can be assigned to these assemblies in the dialogue window of the connection. If one of these assemblies was already assigned a usage type, the usage type specified in the dialogue window was ignored in previous versions. From HiCAD 2020 onwards, the **Overwrite usage** checkbox can be used to specify whether the existing usage is to be kept or replaced by the usage type selected on the **Bolting** tab (or **Fixing** or **Beam** tab, depending on the connection).

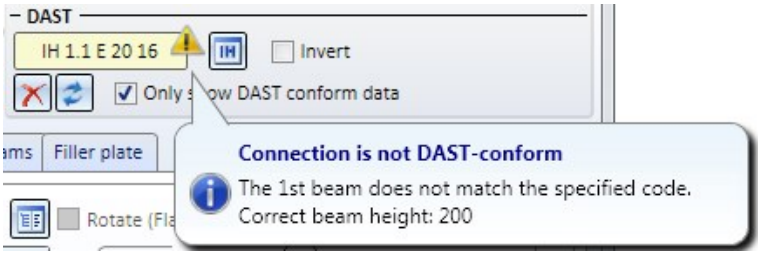


This functional enhancement concerns the following connections:

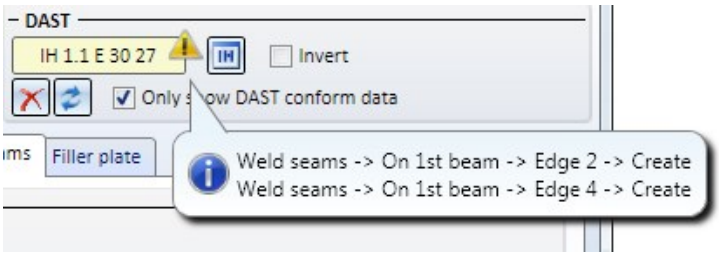
- Base plate + Anchor plate (2101)
- End plate (2102)
- Purlin joint (2201)
- Stiffener (2401)
- Beam to web with 2 plates + stiffener (1211)
- Front plate connection to web/flange (2320)
- Front plate connection to flange (2330)
- Front plate connection (double-sided) to web (2322)
- Column connection, Frame corner (2204)
- Column connection, Frame corner (2203)
- Angle connection, one-sided (1305)
- Angle connection, double-sided (1306)
- Cross-bracing (2601)
- Cross-bracing (2602)
- Stabilizing pipe connection (2701)

DAST connections

- If a DAST connection is changed, HiCAD checks whether the connection is still DAST-compliant. If this is not the case, the DAST selection field will be marked with the symbol from HiCAD 2020 onwards. If you point to this symbol with the cursor, a corresponding message is displayed, e.g.:



- For DAST connections, in addition to fillet welds, the other weld seam types (double fillet weld, HY and HV weld, circumferential weld) can also be selected. If one of these weld seam types is selected, the connection is no longer DAST-compliant. In this case, the DAST selection field will be marked with the symbol, e.g.:



Stairs and Railings

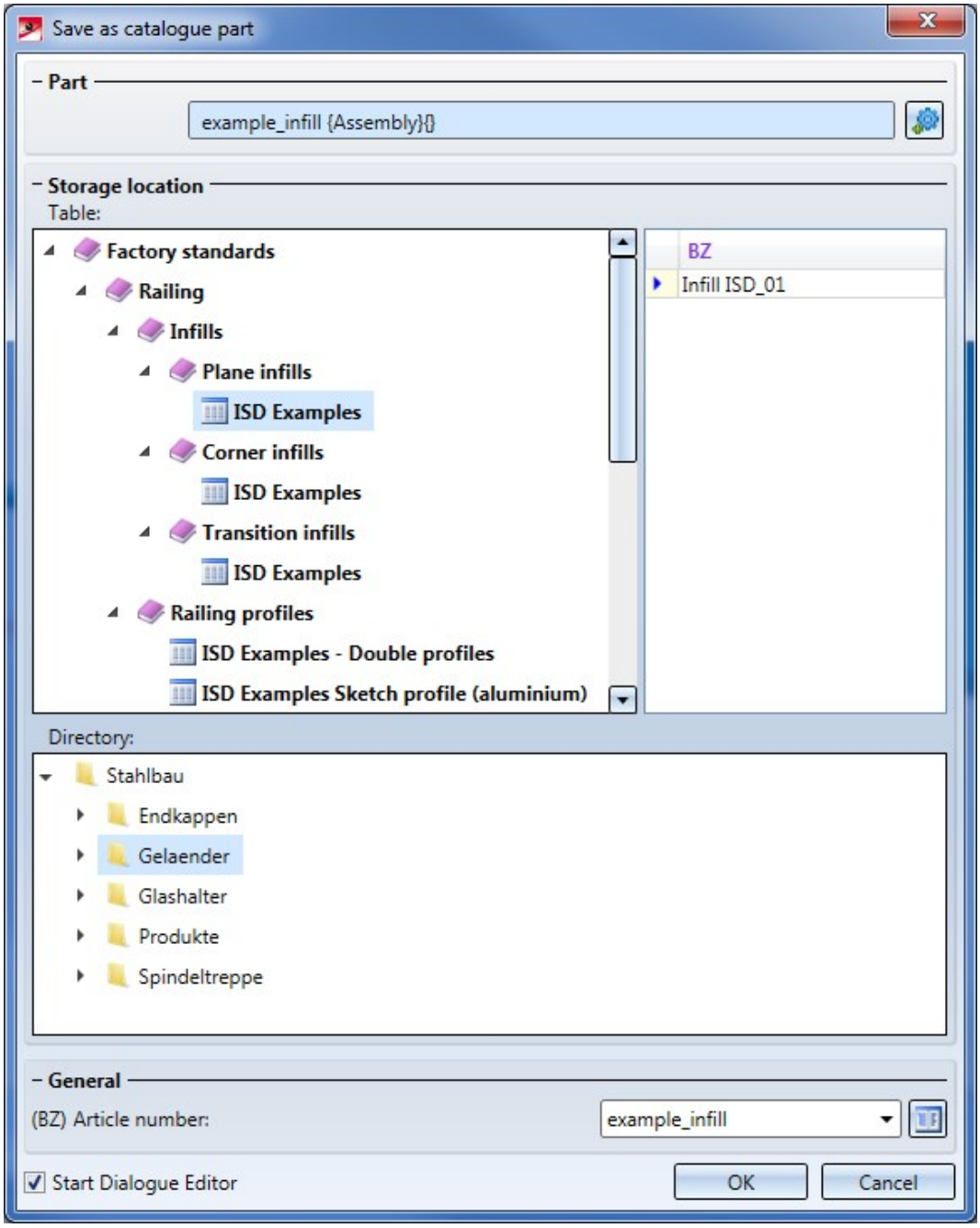
Railing Configurator

The **Post - Sub-structure** tab has been revised, now featuring an optimised alignment of the fields, improved explanatory graphics, etc.

Variant Editor for Railings

From HiCAD 2020 onwards, the HiCAD Dialogue Editor (HiCAD GUI Creator) can be used to create user-specific dialogues for railing variants - analogous to installation elements. The Variant Editor function for railings has therefore been adapted accordingly.

If a parameterized part is active, this function can be used to create a corresponding variant with a list of the variables used as a CSV file. This file is required for creating user dialogues with the HiCAD Dialogue Editor, which can be started directly from the dialogue of the function.



Variables for catalogue entries assigned to a railing component and not defined as list variables are now automatically converted to list variables when the variant is saved. The text object CatalogEntry is then automatically used for these variables when inserting them into a user dialogue.

Semi-finished products as sub-items for series beams

Semi-finished products can be automatically transferred to HELiOS as a structural sub-item of an individual part. This is set in the Configuration Editor (ISDConfigEditor), at **PDM > HiCAD-HELiOS interface > Semi-finished product article as sub-item**.

From HiCAD 2020 on, series beams are also taken into account.

Steel Engineering - Management + BIM

Service Pack 2 2020 (V 2502)

Total quantity of a part in an assembly (COMPONENT_REFASSEMBLY)

In the Configuration Editor, at **PDM > Management+BIM**, you can define whether the associated assemblies, e.g. welded assemblies, are to be determined for individual parts (beams, profiles, plates, sheets, general parts). The parameter **Output associated assemblies** is available for this purpose.

If this parameter is set to **Yes**, the system determines the associated assembly as well as how often the part occurs in this assembly. This data is assigned to the HELiOS article attribute COMPONENT_REFASSEMLY. This can be useful for the BOM output, for example. The ISD default setting is **No**.

How exactly the attribute is assigned can be defined using the template file **BIM_PDM_ReferenceAssembly.ftd**.

This file is copied into the HiCAD sys directory during the installation of HiCAD for Management+BIM - provided that the parameter configuration is suitable!

In this file, the attribute COMPONENT_REFASSEMBLY is preset as follows:

(1)	The superordinate assembly is searched for.
(2)	HELiOS article attribute COMPONENT_ITEMNR_TEXT This attribute is determined by the template file BIM_PDM_ItemnumberGeneration.ftd.
(3)	**Space** and **(**
(4)	HiCAD part attribute Qty. in assembly (%13)
(5)	**x** and **)**

For example, if the assembly with item number 1 contains the part with item number 102 three times, the value of the COMPONENT_REFASSEMBLY attribute is

1 (3x)

If a part exists in several assemblies, the attributes are separated by semicolons, e.g.

1 (3x); 3 (1x); 6 (4x)

This display can also be customized. To do this, you can open the file BIM_PDM_ReferenceAssembly.ftd, for example, with the Annotation Editor, change it as desired, and then save it again under the same name.

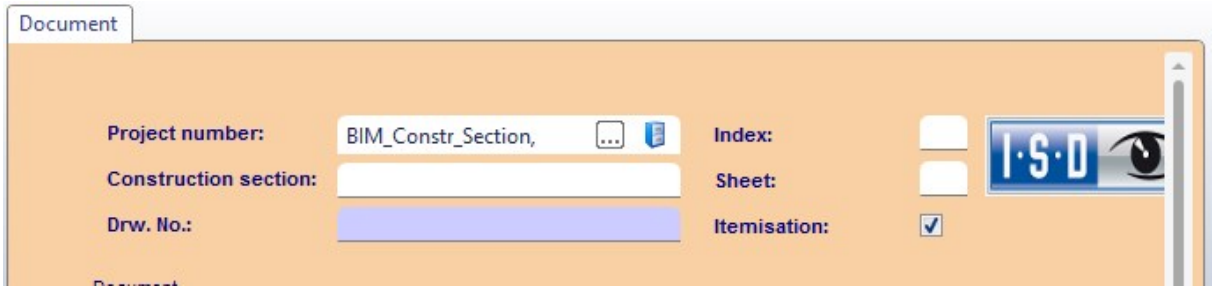
If you also want to display the associated assembly and total quantity of a part in the title block of the single part drawings, use the _SZNATTRRIQ attribute in the title block of the drawing frames. For this purpose, enter **@*@65,S._SZNATTRRIQ@*@** as placeholder text in the title block (with a maximum text length of 65 characters).

Output construction section lists in drawing

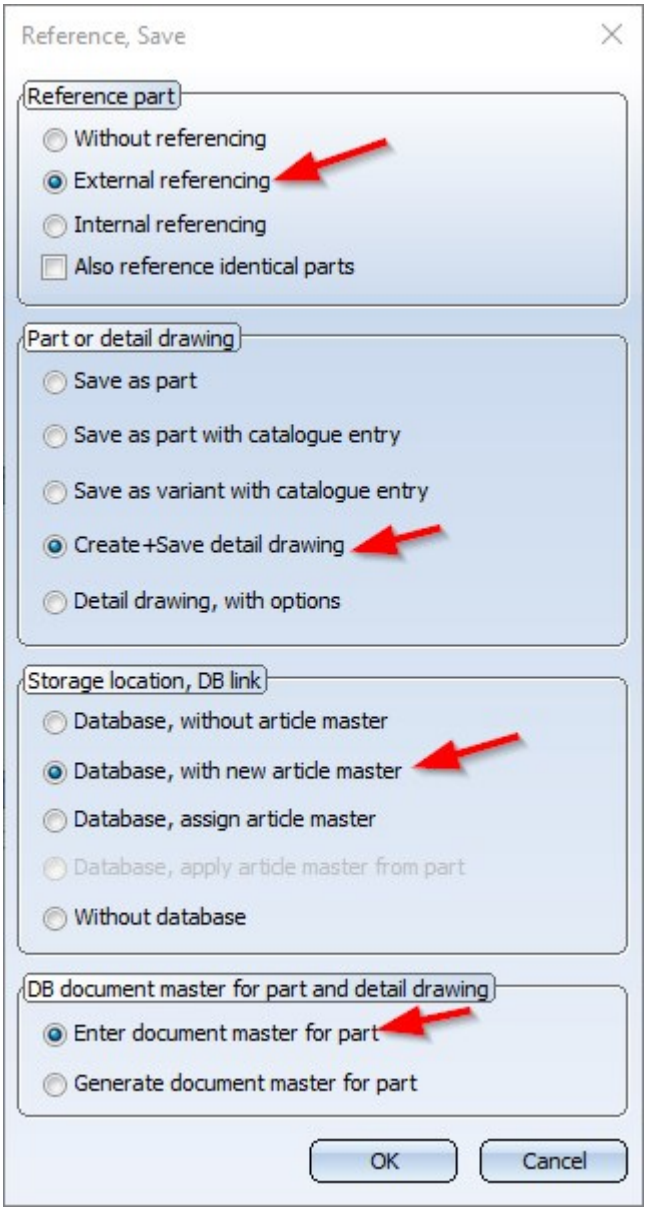
If you work with different construction sections, this function allows you to insert a construction section list into the drawings of the individual parts. This construction section list shows how often a part is present in the different construction sections. For this purpose the new function **Output construction section lists in drawing** is available at **Management + BIM > Workshop > Drawing**.The function can be used in the 3-D model as well as in the detail drawings.

To use the function, the following requirements must be met:

- In the Configuration Editor at **PDM > Management + BIM**, the parameter **Output associated assembly(-ies)** must be set to **Yes**.
- The model drawing must be divided into construction sections.
- The itemisation must be carried out in the overall model, i.e. the **Itemisation** checkbox must be active in the document master of the model drawing.



- The construction sections must be saved as **externally referenced detail drawings**. For each construction section in the article and document master the attribute Construction section must be assigned and the checkbox Itemisation must be deactivated.



- The production drawings must be created in the overall model.

Once the production drawings have been created, a construction section list can be generated using the **Output construction section list in drawing** function.

- If you have called the function in the complete model, then first select the construction sections in which the part is to be searched for and the desired individual parts.
- If you have called the function in the detail drawing, then select the construction sections.

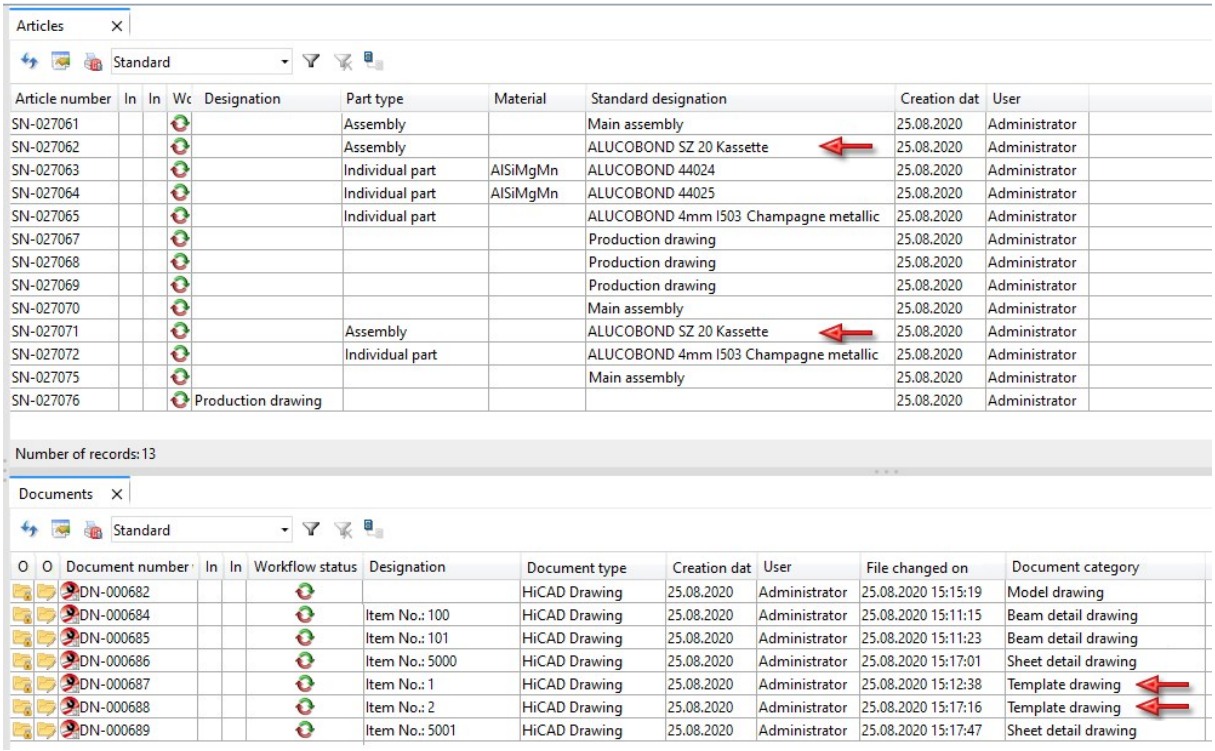
When updating the drawings, the list will be deleted again!

In this context, please also note the information on the **_SZNATTRRIQ** attribute. The use of this attribute in the title block of the drawing frames causes the system to display the assembly assignment and total number of a part in the title block of the detail drawings.

Management of template drawings

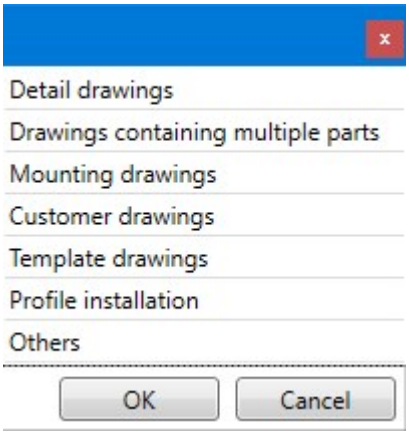
Starting with SP2, template drawings for Element installation can also be managed with BIM.

If a BIM-managed model drawing installation contains elements that have a corresponding template file assigned to them, the corresponding template drawings are automatically created and managed in HELiOS when the drawing is derived. The document type of these drawings is **Template drawing**. Like other drawings, template drawings can be automatically updated, checked and released when changes are made. Furthermore, the same links are used as for production drawings.



Rework of template/profiles installation drawings via script

When saving drawings created with the functions of the **Management + BIM** module, a custom script can be executed to automatically rework the drawings. From SP2 on, this is also possible for template and profile installation drawings. For this purpose, the settings in Configuration Editor at **PDM > Management + BIM > Production drawings > Rework of drawing** have been extended.



The correcponding script files are

- UpdatePI.cs (Profile installation) and
- UpdateTD.cs (Template drawings).

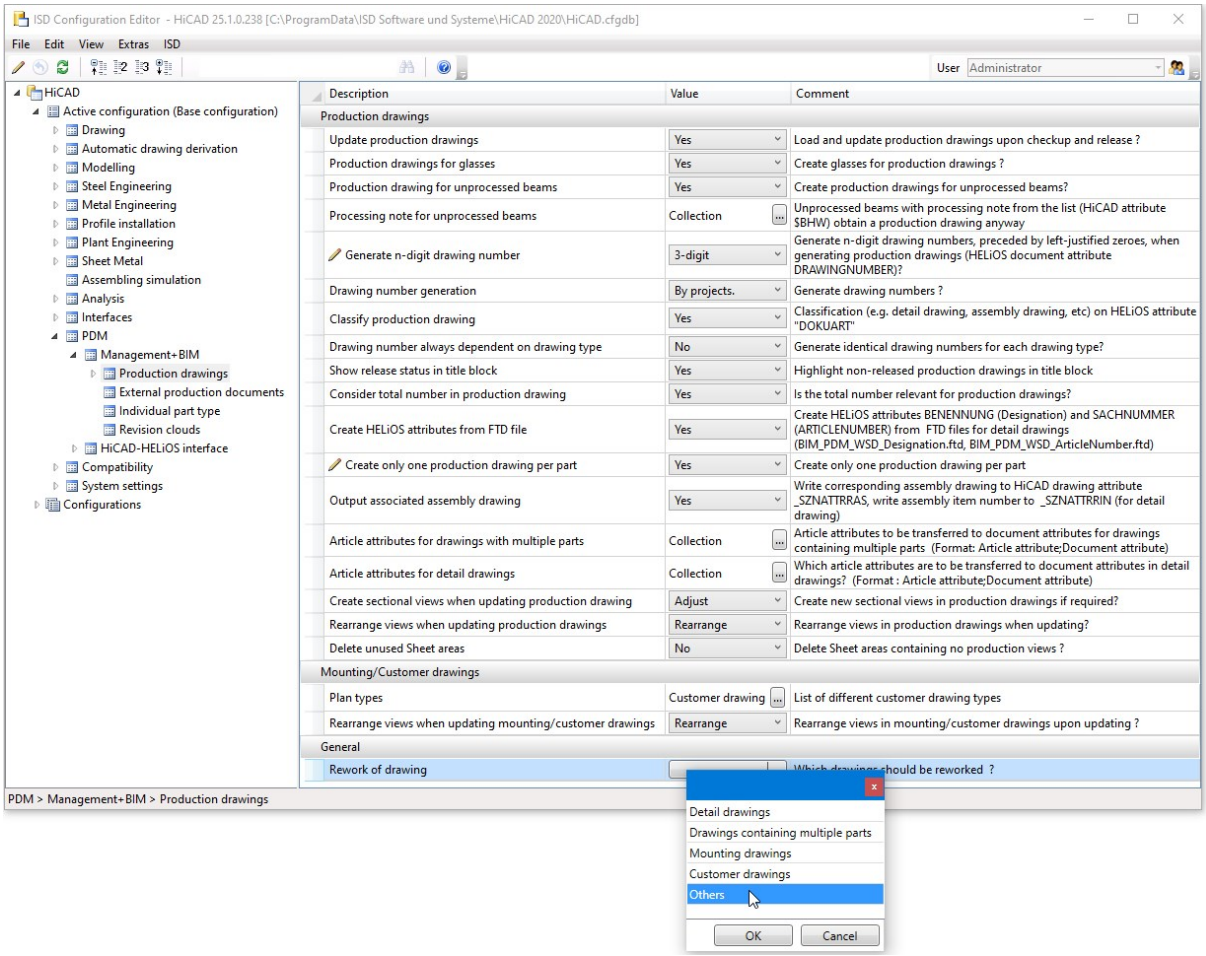
Service Pack 1 2020 (V 2501)

Type of part referencing

In the Configuration Editor at **PDM > Management+BIM**, the **Type of part referencing** parameter does no longer exist. This means that the parts of the automatically generated drawing are referenced unilaterally with the initial model drawing and are marked accordingly in the ICN with the Ⓡ symbol. If changes are made to the model, the drawing can be updated automatically. This does, however, not apply vice versa.

Script for rework of model drawings

Similar to the manufacturing documents created with the Management+BIM module, a script for rework can now be automatically executed when saving model drawings controlled with Management+BIM. For this purpose, the **Rework of drawing** selection box in the Configuration Editor at **PDM > Management+BIM > Production drawings** has been expanded.



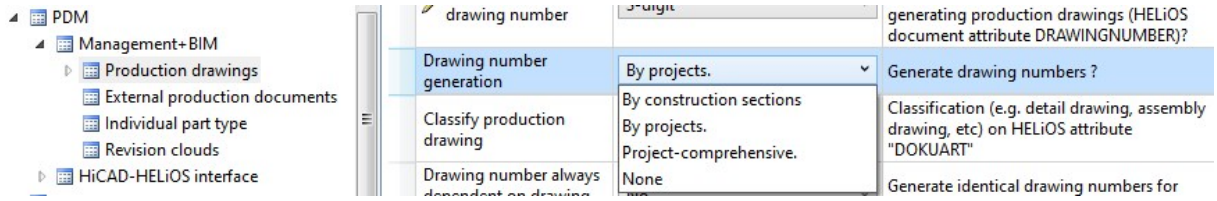
"Others" are all HiCAD model drawings that do not comply with other classifications, such as 3-D models, files created with the **Reference part, Save, Detail drawing** function etc.

Drawing number generation

If you have selected the generation of n-digit drawing numbers in Configuration Editor at **PDM > Management+BIM > Production drawings**, you can use the parameter **Drawing number generation** from HiCAD 2020 SP1 onwards to specify how the numbering is to take place:

- **By production sections** (consecutively within one production section),
- **By projects** (consecutively within one project), or
- **Project-comprehensive** (spanning several projects).

Select **None** to deactivate numbering completely.



The default setting is **By projects**.

Output associated assemblies

In the Configuration Editor at **PDM > Management+BIM**, you can now define whether the associated assemblies, e.g. welded assemblies, should be determined for individual parts (beams, profiles, plates, sheets and general parts). The parameter **Output associated assemblies** is available for this purpose. If this parameter is set to **Yes**, the association is determined and assigned to the HELiOS article attribute COMPONENT_REFASSEBMLY. This can be useful for the BOM output, for example.

The default setting is **No**.

The COMPONENT_REFASSEMBLY attribute is assigned the item number of the first parent assembly of the part. If a part exists in several assemblies, the item numbers of the parts are separated by semicolons.

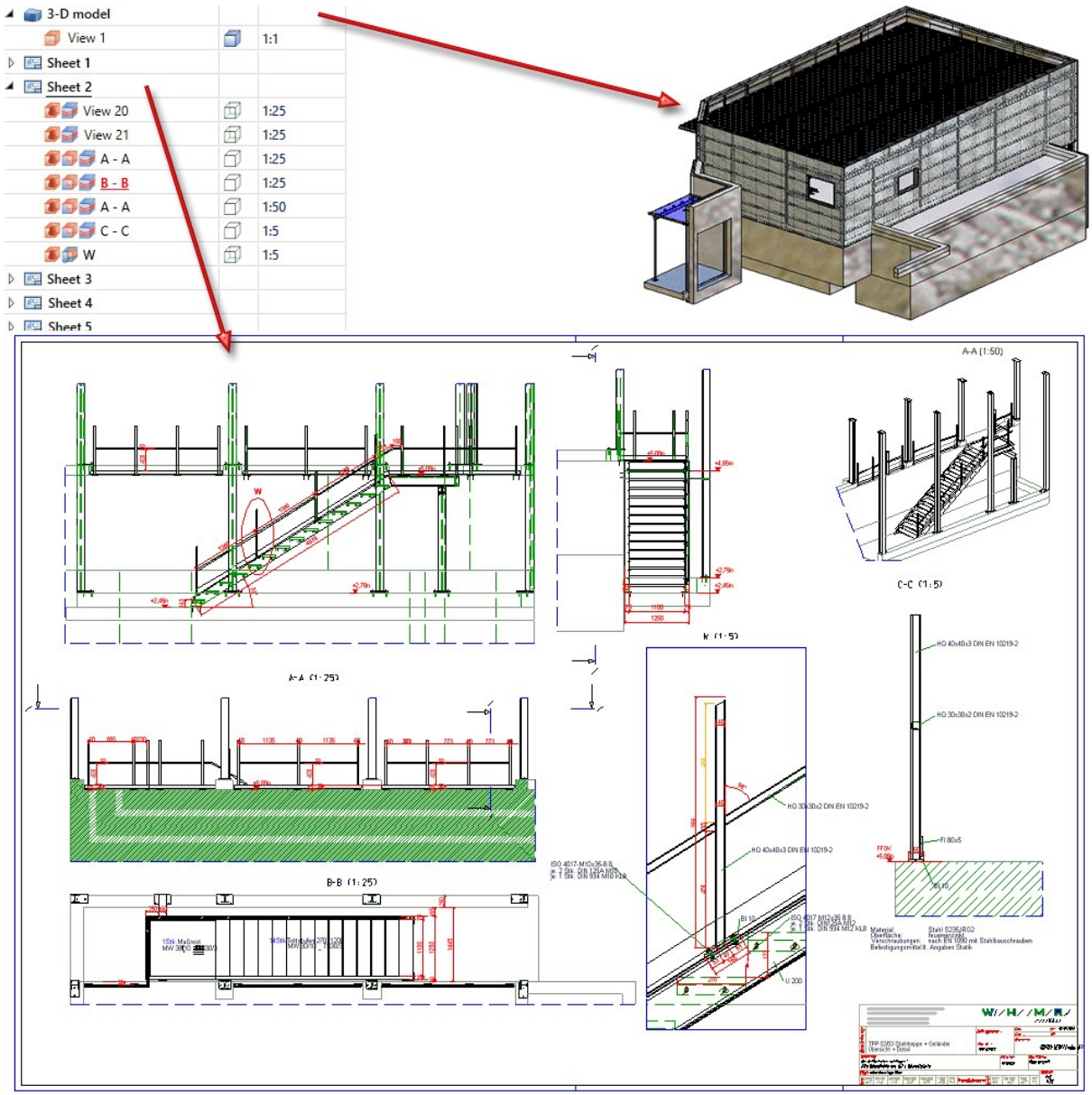
Articles

Standard

Article number	In	In	W	Designation	Part type	Material	Standard designation	Creation dat	User	COMPONENT_REFASSEMBLY	COMPONENT_ITEMNR_TEXT
SN-026675					Assembly		Main assembly	14.07.2020	Administrator		
SN-026676					Assembly		Assembly IPE 200	14.07.2020	Administrator		1
SN-026677					Assembly		Assembly IPE 200	14.07.2020	Administrator		2
SN-026678					Assembly		Assembly IPE 200	14.07.2020	Administrator		3
SN-026679					Individual part	S235JRG2	IPE 200	14.07.2020	Administrator	1 (2x); 2 (2x); 3 (2x); 4 (2x)	100
SN-026680					Individual part	S235JRG2	Z 30	14.07.2020	Administrator	2 (1x); 3 (1x)	101
SN-026681					Individual part	S235JRG2	Z 30	14.07.2020	Administrator	2 (1x); 3 (1x)	102
SN-026682							Production drawing	14.07.2020	Administrator		1
SN-026683							Production drawing	14.07.2020	Administrator		
SN-026684							Production drawing	14.07.2020	Administrator		100
SN-026685							Production drawing	14.07.2020	Administrator		101
SN-026686							Production drawing	14.07.2020	Administrator		102
SN-026688					Assembly		Assembly IPE 200	14.07.2020	Administrator		4
SN-026691							Production drawing	14.07.2020	Administrator		4

AutoCreate General Production Documents

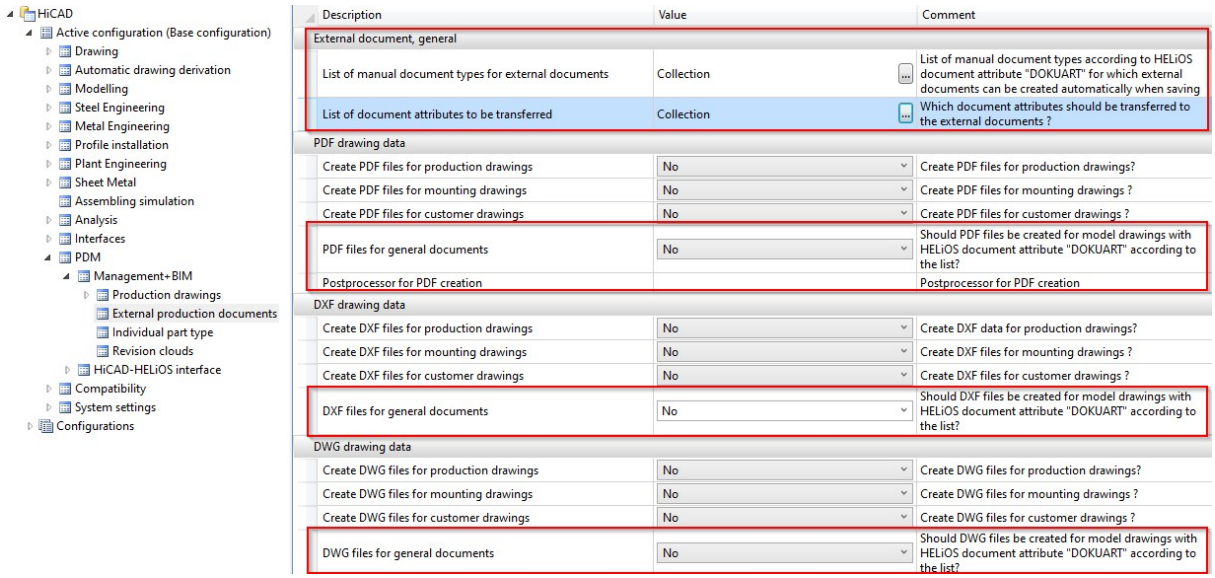
In practice, it often happens that overview drawings of a model drawing are created manually in a separate sheet area. These drawings contain, for example, sectional views, cut-outs, shortened views, detail views, etc., and are intended to illustrate how the model is realized. These sheet areas are not taken into account when creating external production documents.



Drawing by Metallbau Wilhelmer Projekt GmbH, Kolbnitz, Austria

In order to be able to manage these documents together with the corresponding model in HELiOS, HiCAD offers the possibility to automatically create a PDF, DXF and/or DWG file of the respectively active model or sheet area directly when saving a model drawing.

Whether and which formats are generated is determined via the Configuration Editor at **PDM > Management + BIM > External production documents**, where you will find the required parameters.

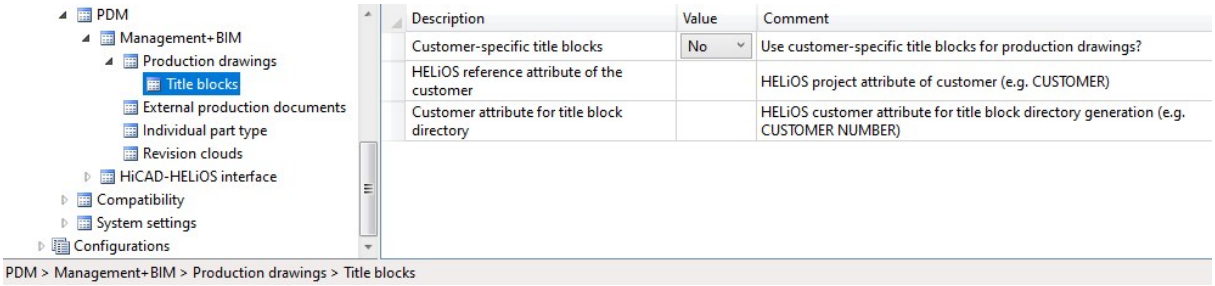


User defined objects (UTO)

The user-defined object type (UTO) Location UTO is no longer used by HiCAD in drawing derivations. This means that when the database is configured with the HicadBIMConfig.exe tool, the attributes belonging to this object type are no longer transferred.

Customer-specific and usage-dependent title blocks

In the Configuration Editor, you can now use the settings at **PDM > Management + BIM > Production drawings > Title blocks** to specify that customer-specific and/or usage-dependent title blocks are to be used in production drawings.



HiCAD's scope of delivery includes corresponding title block templates and the corresponding drawing frames. These templates and frames can be found in the HiCAD subdirectory **templates\Module_3DSteel\BIM\szen-en\BIM**:

Drawing frame		Title block templates for detail drawings	Used for
ST_DINA0.FGA		ST_TB_DEFAULT_ASSEMBLY.FGA	Assembly
ST_DINA1.FGA		ST_TB_DEFAULT_BOLTEDASSEMBLY.FGA	Mounting assembly
ST_DINA2.FGA		ST_TB_DEFAULT_GRATING.FGA	Grating
ST_DINA3.FGA		ST_TB_DEFAULT_MAINASSEMBLY.FGA	Main assembly
ST_DINA3_KANTBLECHE.FGA		ST_TB_DEFAULT_PANE.FGA	Glass
ST_DINA4H.FGA		ST_TB_DEFAULT_PLATE.FGA	Steel Engineering plate
ST_DINA4H_BLECHE.FGA		ST_TB_DEFAULT_PROFILE.FGA	Beam, Profile
ST_DINA4H_PROFILE.FGA		ST_TB_DEFAULT_PURCHASEDPART.FGA	Purchased part
ST_DINA4Q.FGA		ST_TB_DEFAULT_SHEET.FGA	Sheet Metal part
		ST_TB_DEFAULT_WELDEDASSEMBLY.FGA	Welded assembly
		ST_TB_DEFAULT.FGA	Others

Based on these templates, you can create additional title blocks, for example for certain types of parts. In this way, you can use different title blocks for different profile types, for example.

Major Release 2020 (V 2500)

Neuer Maskeneditor in HELiOS 2020

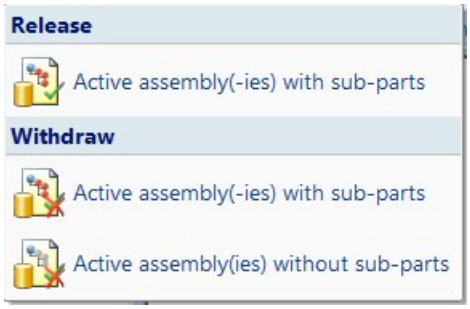
Due to the complete revision of the Mask Editor in HELiOS 2020, the previous mask format has been changed from .MSK to .XML. In addition, mask files are now managed outside the installation directory, resulting in some changes in the system architecture that Administrators must consider:

- The masks of older versions (before V 2500.0) that were previously stored in the SYS directory of the installation are automatically converted to the new XML format in case of an update installation and stored at %Programdata%.
- The new masks at %Programdata% should not be overwritten under any circumstances, as these represent the "delivery status", which will be updated by later update installations. If masks are edited and changed with the Mask Editor, you must store and call them at %Appdata%.

Before updating to HiCAD / HELiOS 2020, it is important that you read the Installation Notes !

Withdraw releases

The release of drawings, e.g. in case of a premature release, can now also be revoked by the administrator. To do this, two new functions have been added to the **Release** function menu.



Withdraw release of active assembly(-ies), with sub-parts

This function can be used to undo already released drawings of the active part-/assembly including all sub-parts. Multiple selection is also possible.

Only the administrator can withdraw releases. HELiOS therefore displays the HELiOS Login dialogue window. After the administrator has logged in, the release is executed. The HELiOS Login dialogue window is then displayed again. Then log in again with your user data.

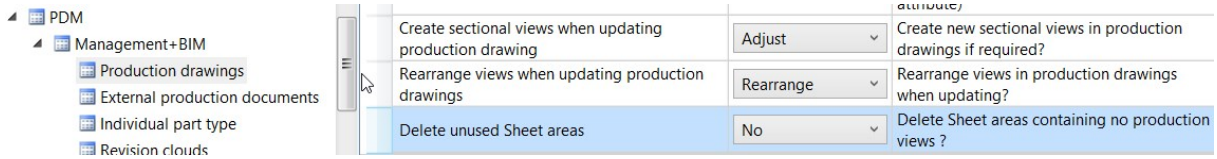
Withdraw release of active assembly(-ies), without sub-parts

As with the previous function, this function can be used to withdraw already released drawings of the active part-/assembly. However, the drawings of sub-parts are not considered here. Multiple selection is also possible here.

Delete unused Sheet areas

The detail drawings created with the **Management+BIM** module are always stored on Sheet 2 of the drawing. When printing via the HELiOS/HiCAD Spooler, all sheets are printed - depending on the setting - i.e. also sheet areas without production drawings such as Sheet 1.

By setting the **Delete unused sheet areas** parameter to **Yes** you can now specify that sheet areas without production drawings are deleted. The default setting is **No**.



Automatic export of CAM and drawing data to the Windows file system

When creating/updating external CAM data in the formats DSTV-NC, DXF, DWG, NCW/NCX and STEP and drawing data in the formats DXF, DWG and PDF, the corresponding files can also be automatically exported to the Windows file system if required. The Configuration Editor provides the appropriate parameters for this purpose at **PDM > Management+BIM > External production documents**:

External part data	
Automatic CAM data export	If this parameter is set to **Yes**, the CAM data are automatically exported from HELiOS to the Windows file system to the folder specified at **Windows export path** when creating/updating the CAM data. Depending on the file type, the files are stored in a separate sub-folder, e.g: The files are automatically named according to the configuration file **BIM_PDM_CAMExport.ftd** (in the HiCAD sys directory). Default setting is the part attribute **Item number**. The default setting for automatic CAM data export is **No**.
Windows export path	Here you specify the path for automatic CAM data export.

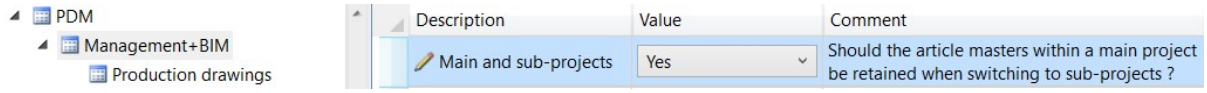
Drawing data export	
Automatic drawing data export	If this parameter is set to **Yes**, the drawing data are automatically exported from HELiOS to the Windows file system to the folder specified at **Windows export path** when creating/updating the drawing data. Depending on the file type, the files are stored in a separate sub-folder, e.g: The files are automatically named according to the configuration file **BIM_PDM_WSDExport.ftd** (im HiCAD sys-Verzeichnis). The default setting for the designation is: Article master attribute **COMPONENT_DRAWINGNUMBERTEXT-** Part attribute **Item number**. The default setting for automatic drawing data export is **No**.
Windows export path	Here you specify the path for automatic drawing data export.

Important: The files imported to the Windows file system will not be managed in HELiOS.

Working across projects

By default, **Management+BIM** works project-related, i.e. the managed parts are usually clearly assigned to one project. When installing parts with reuse in other projects, such as purchased parts, you must assign the article master manually, as these parts are ignored by the BIM automatisms.

As of HiCAD 2020, this behaviour can be changed in the Configuration Editor at **PDM > Management+BIM** by setting the **Main and sub-projects** parameter to **Yes**.



This enables a working across several projects. This means that when you work with main and sub-projects, the article masters of the main project are retained when you switch to a sub-project and vice versa. This also applies when you copy parts from a sub-project to the main project via the clipboard and vice versa.

The following prerequisites must be met for working across projects:

- The **Main and sub-projects** parameter must be set to Yes.
- The project masks must contain the Boolean attribute **ROOTPROJECT**. To do this, you must extend the masks manually using the HELiOS Mask Editor.
- The **ROOTPROJECT** attribute must be set to **Yes** for the main project and to **No** for the sub-projects.

For an example, see the Working with Main and Sub-projects topic.

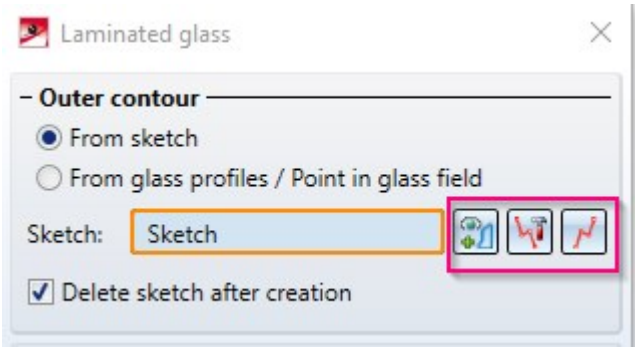
Please note that working across projects is only possible within the 1st level beneath the main project!

Metal Engineering

Service Pack 2 2020 (V 2502)

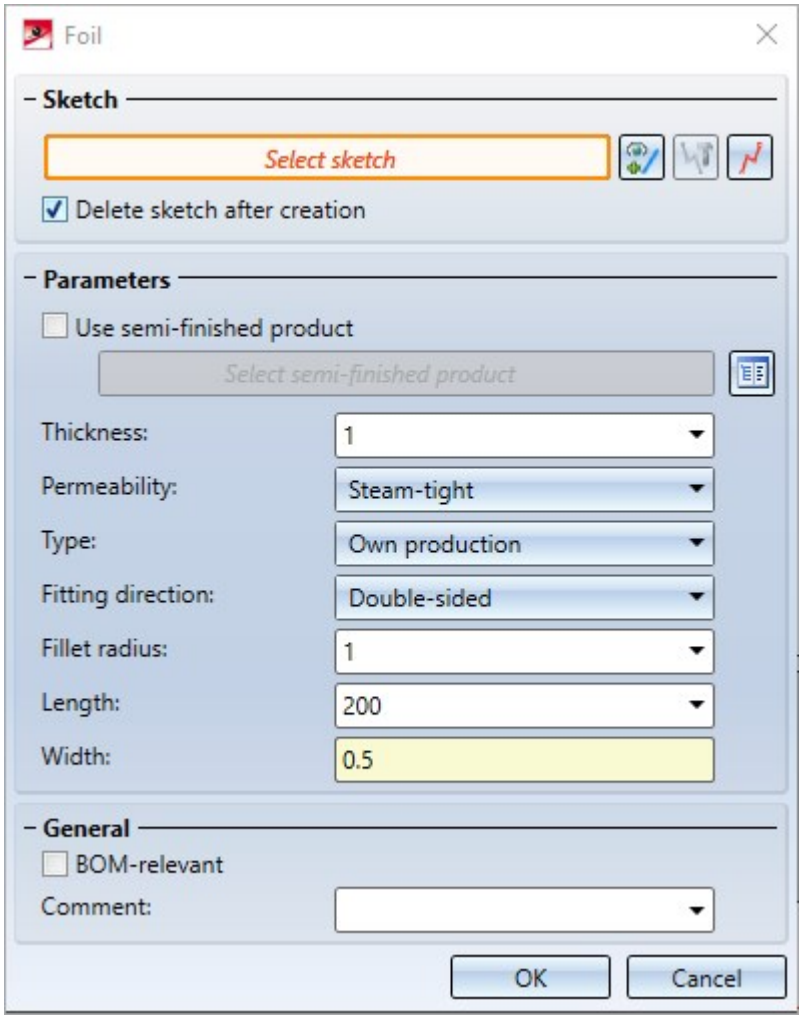
Sketch selection in the Laminated glass dialogue window

The **Process sketch** and **New sketch in plane** buttons are now available directly in the **Laminated glass** dialogue window, and no longer in a pull-down menu.



Revised Foil function

The function **Foil**, available in the **Civil Engineering functions** docking window at **Civil Engineering, general > Civil Engineering - Sectional view (3-D) > Foil**, has been revised. It is now possible to select or create a sketch directly from the dialogue. Foils can be taken as semi-finished products from the catalogue and the effects of the individual settings are visualized as a real-time preview directly in the model drawing.



Service Pack 1 2020 (V 2501)

Connections - Favourites management

For all connections based on design variants, the settings of the respective dialog window can be saved as favorites and reused at any time. To do this, click on the symbol at the bottom left of the dialogue window to activate the context menu. For more information on Favourites management, read the Manage Favorites topic ofm the HiCAD Basics Help.

Element Installation

Service Pack 2 2020 (V 2502)

Open template drawing

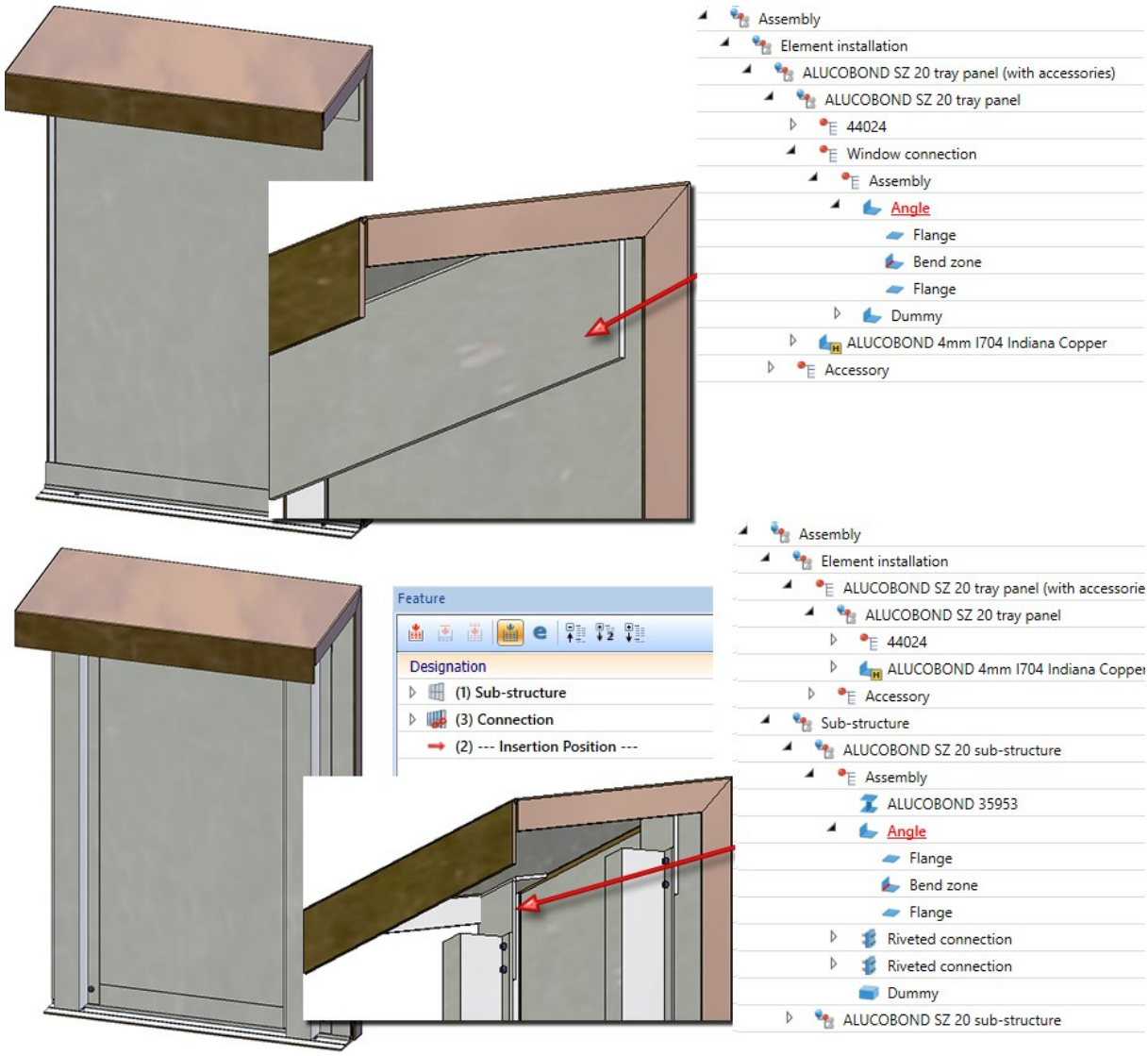
The new function **Open template drawing** is now available for working with template drawings. After you have created drawings for an element installation using the **Template drawing** function, you can first select any element in the model drawing and then call the **Open template drawing** function. This will open the drawing belonging to the selected part.

Changed dialogue for sketch selection

Dialogues requiring the selection of a sketch have been slightly changed. The functions **Process sketch** and **New sketch in plane** can now be activated directly in the dialogue.

ALUCOBOND® SZ 20 tray panels - Window connection

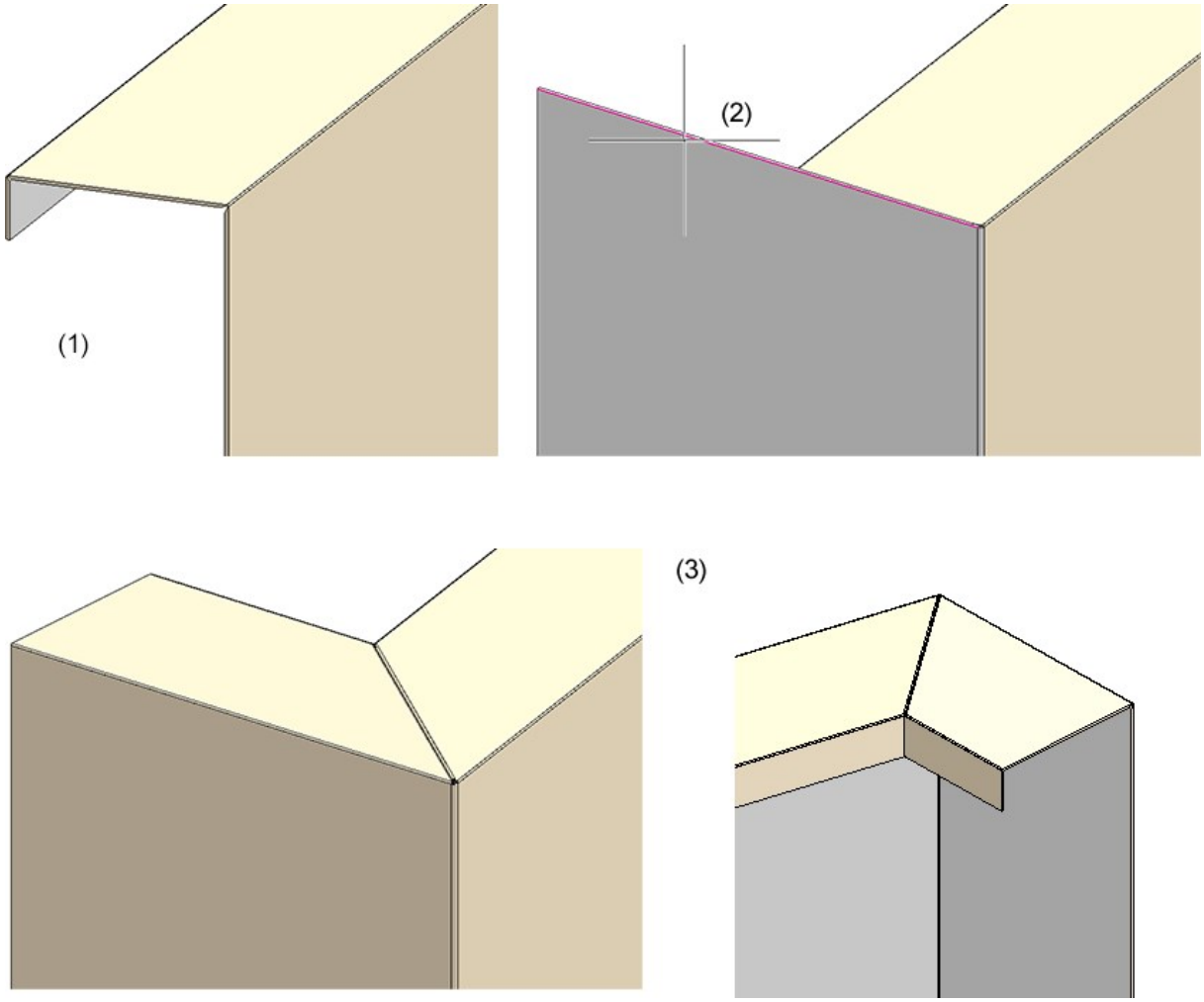
When installing ALUCOBOND SZ-20 tray panels with window connection, the "L-bracket" was previously part of the panel. From SP2 onwards, it is only created by the connection with the sub-structure. It is designed as a "short piece" on each sub-structure and is not continuous.



Top: Part structure of the tray panel before HiCAD 2020 SP2, Bottom: Part structure as of HiCAD 2020 SP2

Design Variant - Flange for SZ20

When using ALUCOBOND SZ-20 tray panels, there is sometimes the desire to create a connection with a flange for the lateral connection, so that a "U" is created. This cannot be realized directly via the Element installation dialogue window, but via the Design Variant **Flange for SZ20** in the **Civil Engineering functions** docking window, beneath **Sheet Metal**. Until now, this variant could only be applied to SZ-20 tray panels with the setting Vertical section: Standard. As of SP2, it is now possible for all connection types, e.g. for **Attic, short**.

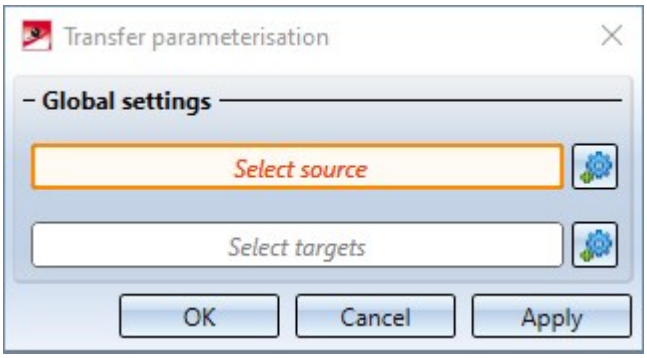


(1) Element installation: Attic, short, without flanges; (2) Attaching of flange and calling of design variant with selection of the marked edge; (3) Result

Transfer parameterisation

New in the **Civil Engineering functions** docking window at **Element installation** is the function **Transfer parameterization**. With this function, the parameters of an installation element can be transferred to other installation elements. However, this is only possible if the installation elements are based on the same variant. This means, for example, that the parameters of an ALUCOBOND SZ-20 tray panel cannot be transferred to a suspended ALUCOBOND panel.

When you call the function the following dialogue window is displayed:



First select the installation element whose parameters are to be transferred. Click on the icon to change the selection.

Then select the installation element to which the parameters are to be transferred. Multiple selection is also possible.

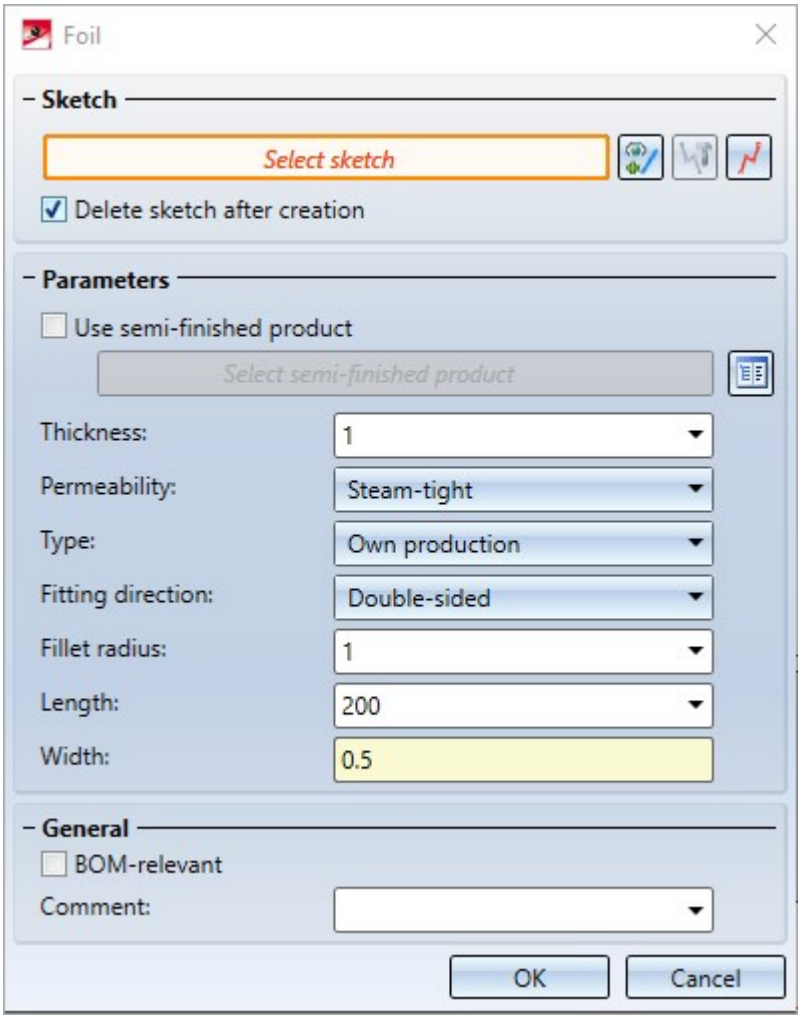
Click on **OK** or **Apply** to transfer the parameters. When using **Apply**, the dialogue window remains open.

Service Pack 1 2020 (V 2501)

Template drawings

If you have applied changes to elements in an element installation, such as additional bores or subtractions, it may be necessary to create individual drawings of these elements for the production department. For large element installations, many elements can be affected, which would require a lot of time to create and adapt the individual drawings.

This is where the new **Template drawings** function can help. It is now possible to create drawing templates for the different sub-types of self-generated variants and then automatically generate drawings for the individual elements of an element installation.



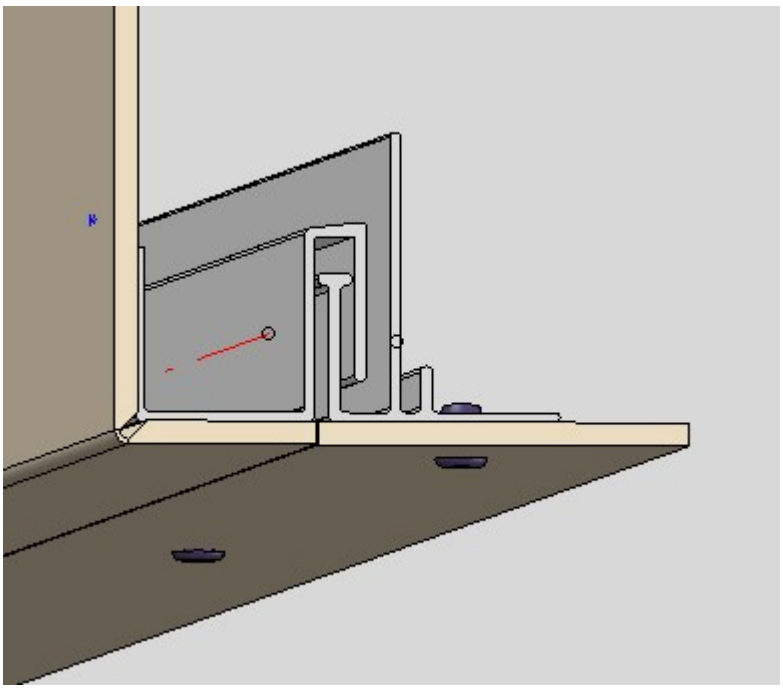
Here you have the choice under Part selection whether you want to create drawings for all elements of the active element installation or only for certain parts.

To use this function, it is first necessary to create and configure the templates to be used. For a description of the necessary steps, please refer to the Setting Up Template Drawings topic. There is also an Example available.

ALUCOBOND® SZ-20 panels and sub-structures

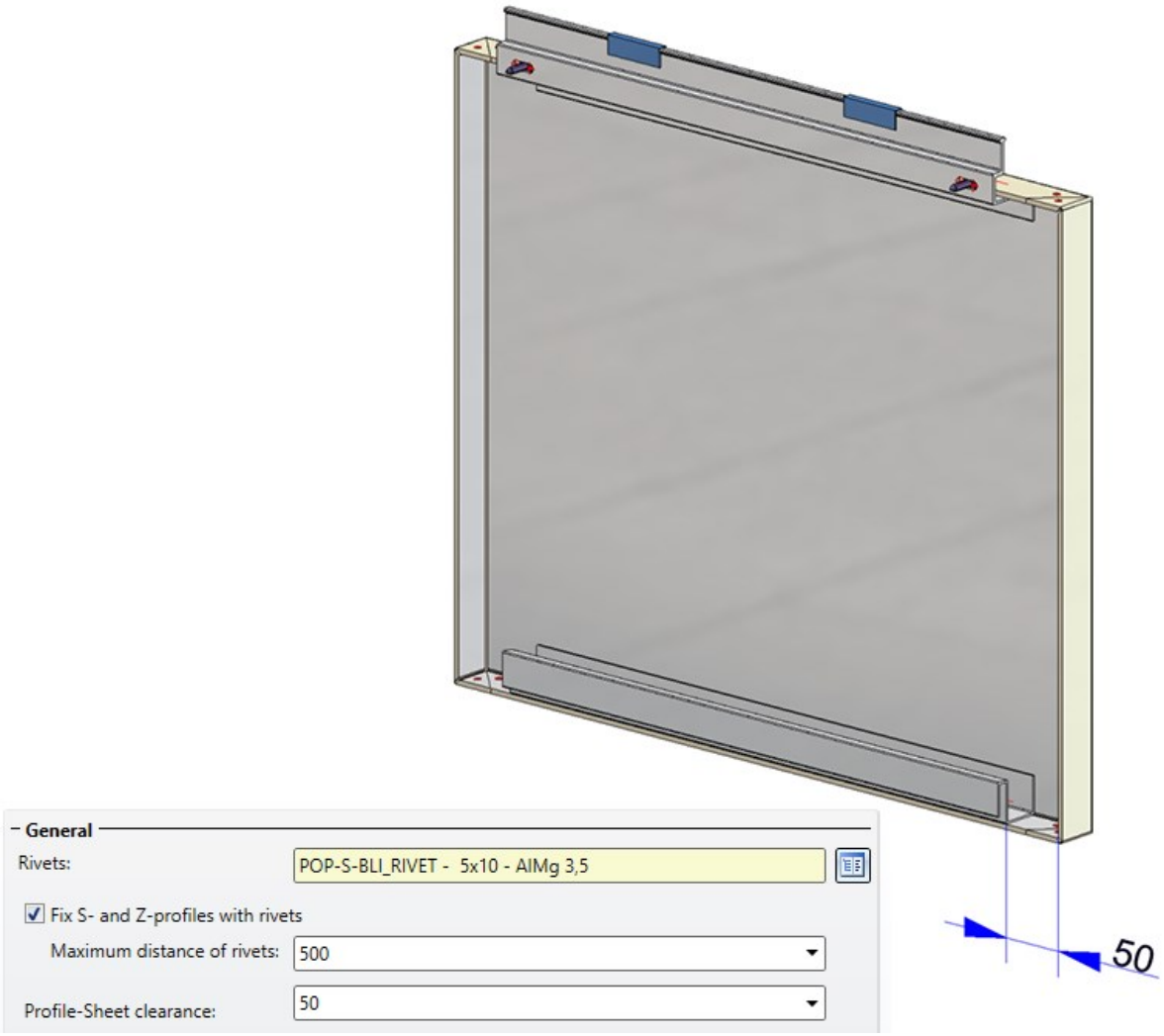
Window connection with base profile and additional sheet

SZ-20 panels can now obtain a window connection with a base profile and additional sheet at the base point. The length of the additional sheet can be chosen freely.



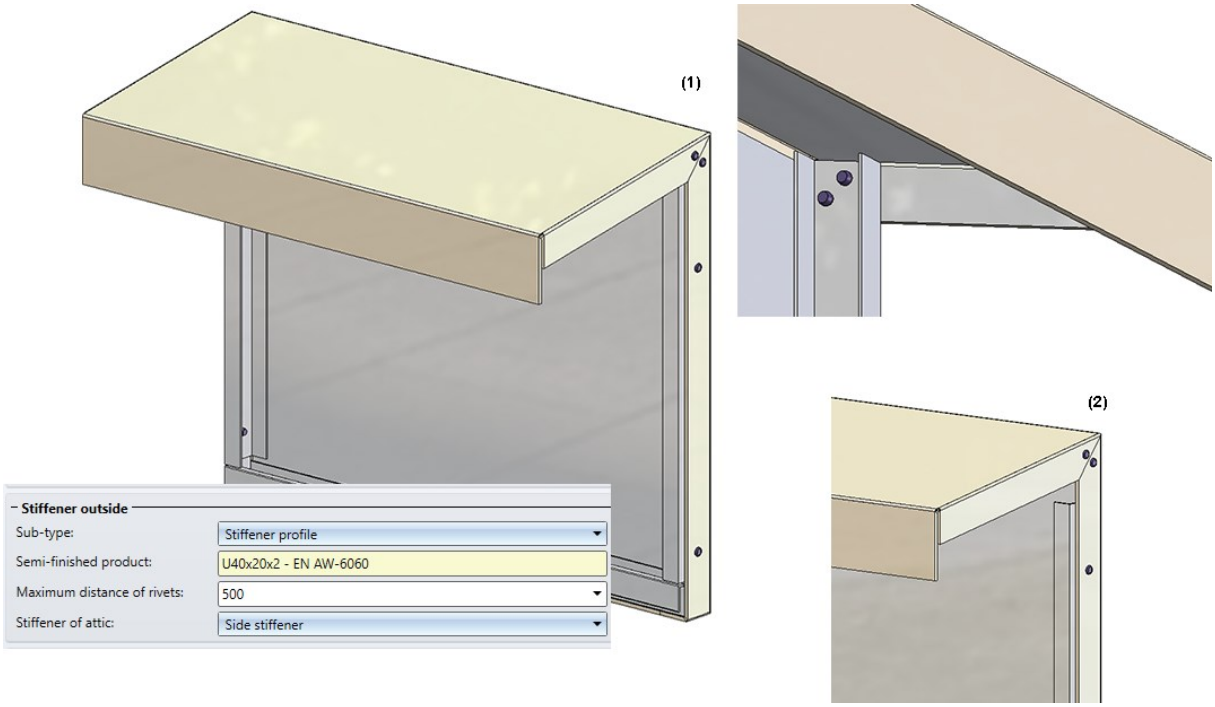
Length of S- and Z-profiles

The length of S- and Z-profiles (suspension profiles) can now be influenced by specifying a clearance between profile and sheet.



Attics - Stiffener outside

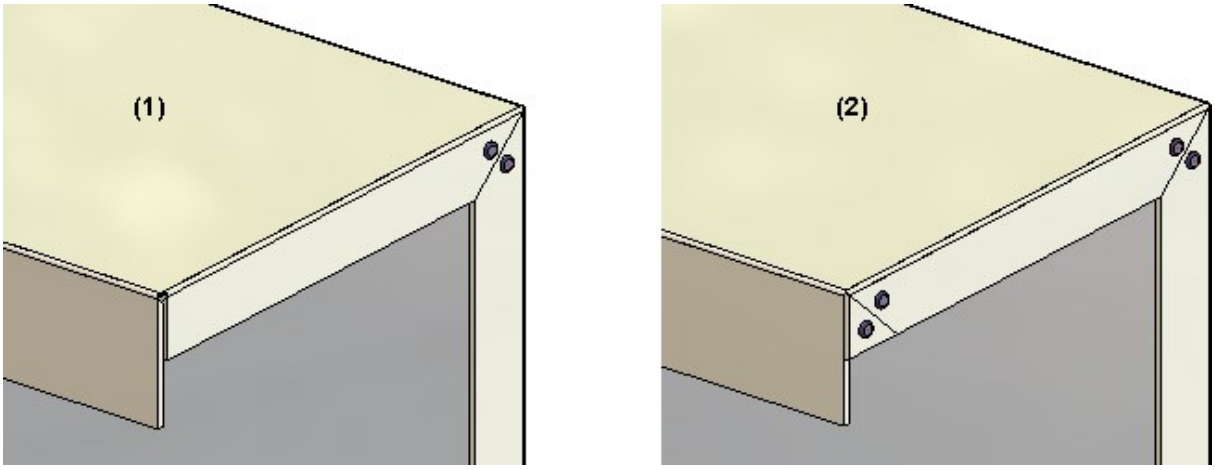
For attic connections, it is now possible to lengthen the **Stiffener outside** to the attic and rivet it to the flanges. To do this, select the setting **Stiffener of attic: Side stiffener**.



(1) Stiffener, lengthened up to the attic; (2) Stiffeners on attic = Corner sheet

Attic connections with and without lug

Attic connections can be created with or without lug. For this purpose the tab **Extensions for SZ-20 standard** has been extended accordingly.



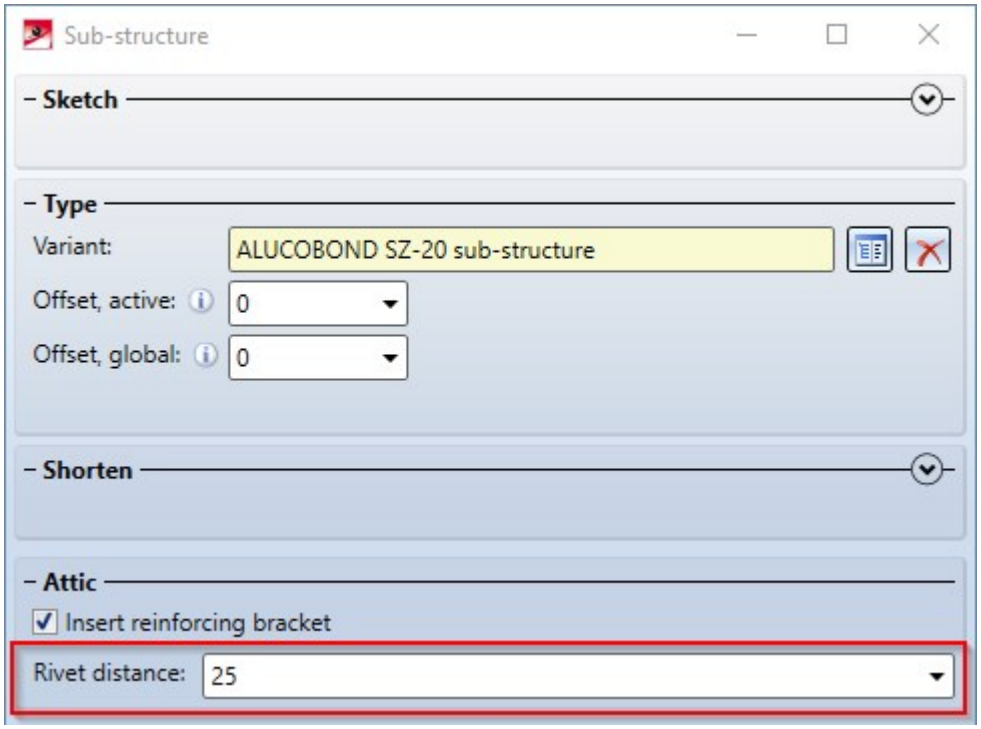
(1) With lug; (2) Without lug

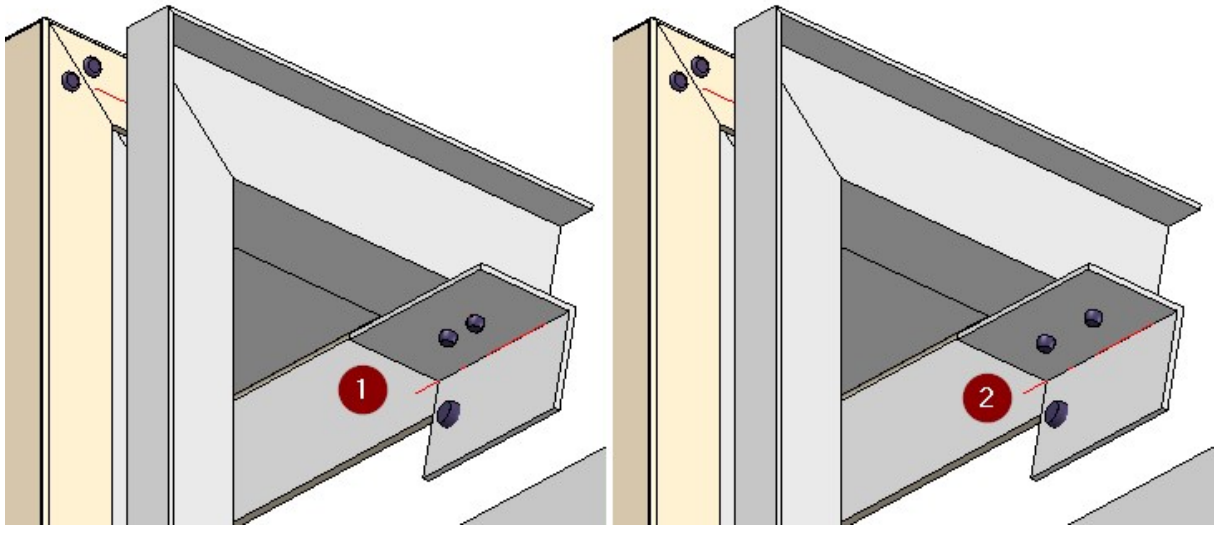
Window connection without side flanges

As with attic connections, it is now also possible to insert the window connection without flanges.

Configurable rivet distances on attic stiffeners

When installing an SZ-20 attic with reinforcement bracket, the rivet distance can now be configured in the sub-structure settings.

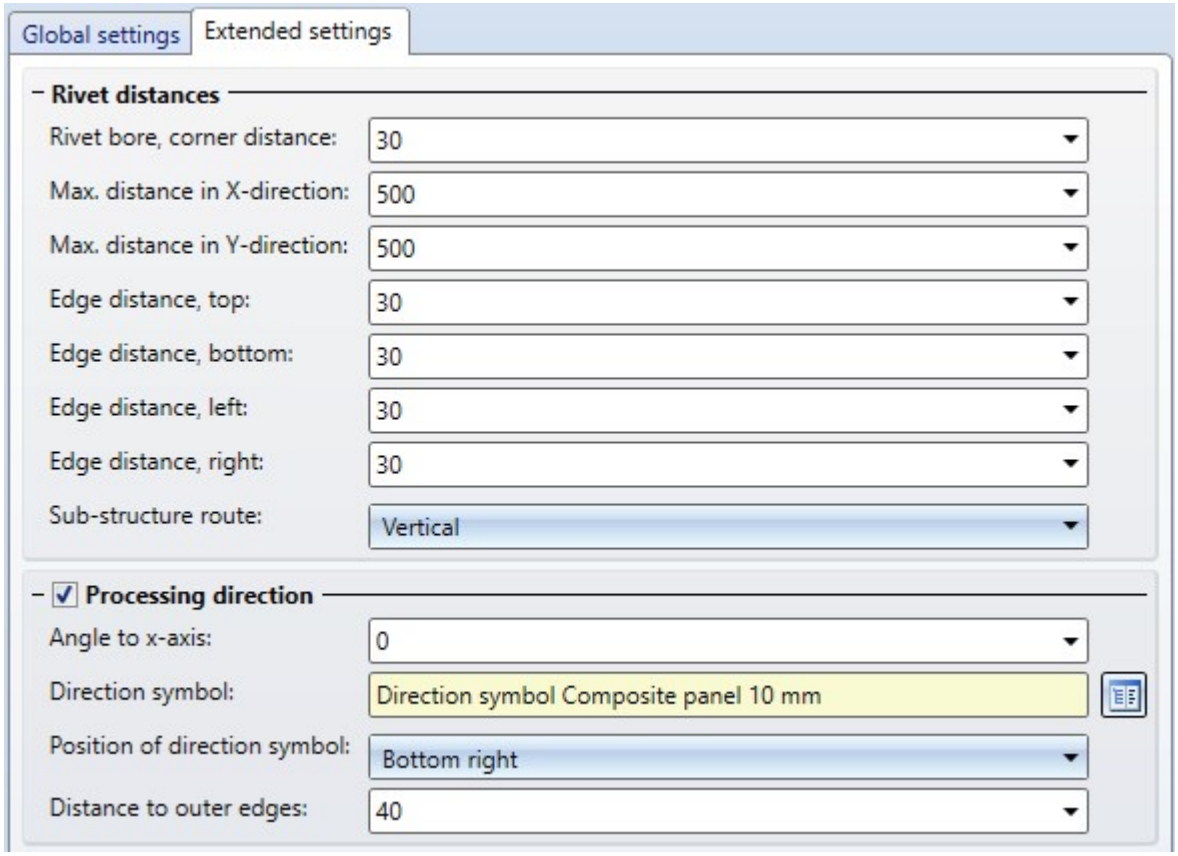


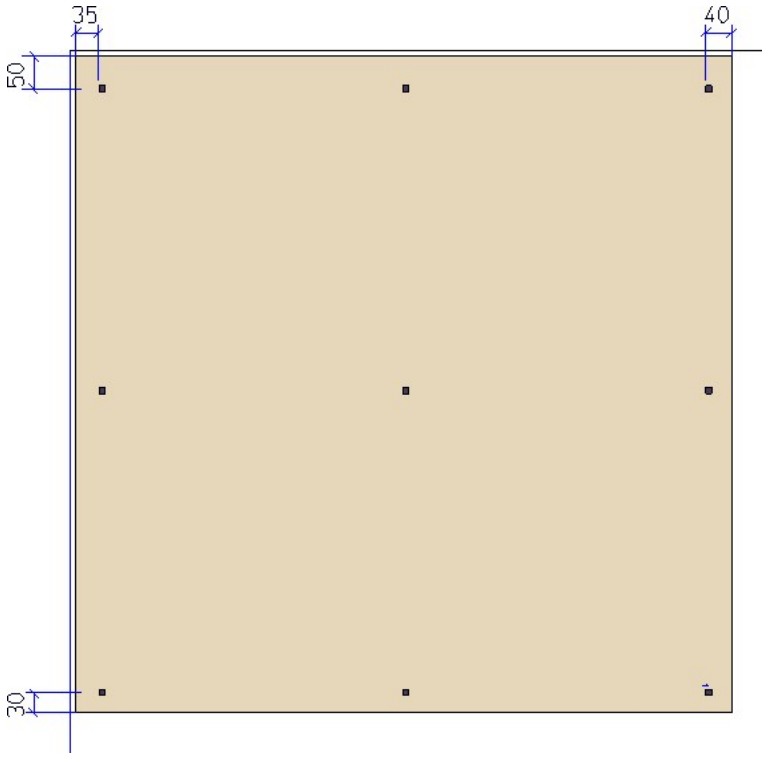


(1) Rivet distance: 20mm, (2) Rivet distance: 25mm

ALUCOBOND® riveted - Individual edge distances

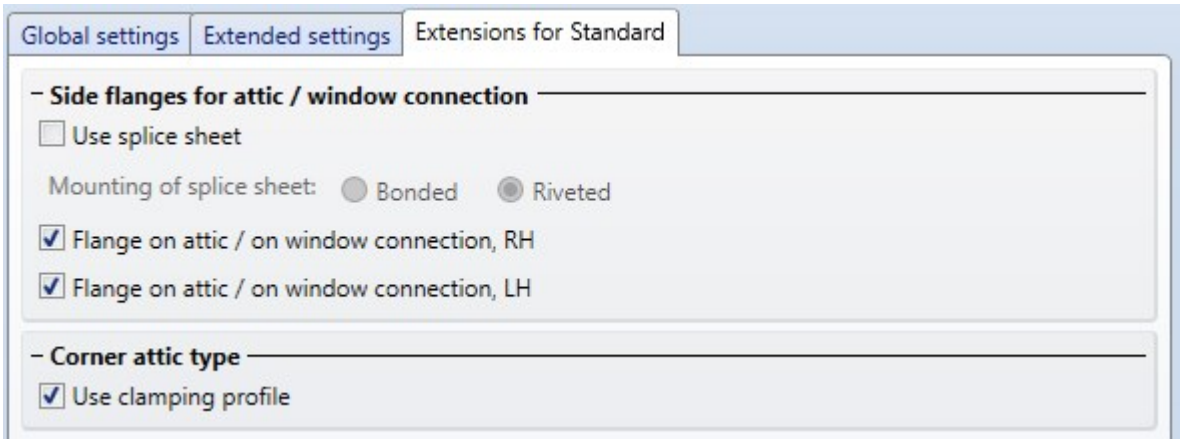
For riveted ALUCOBOND® panels, the top, bottom, left and right edge distance can now be specified on the **Extended settings** tab.

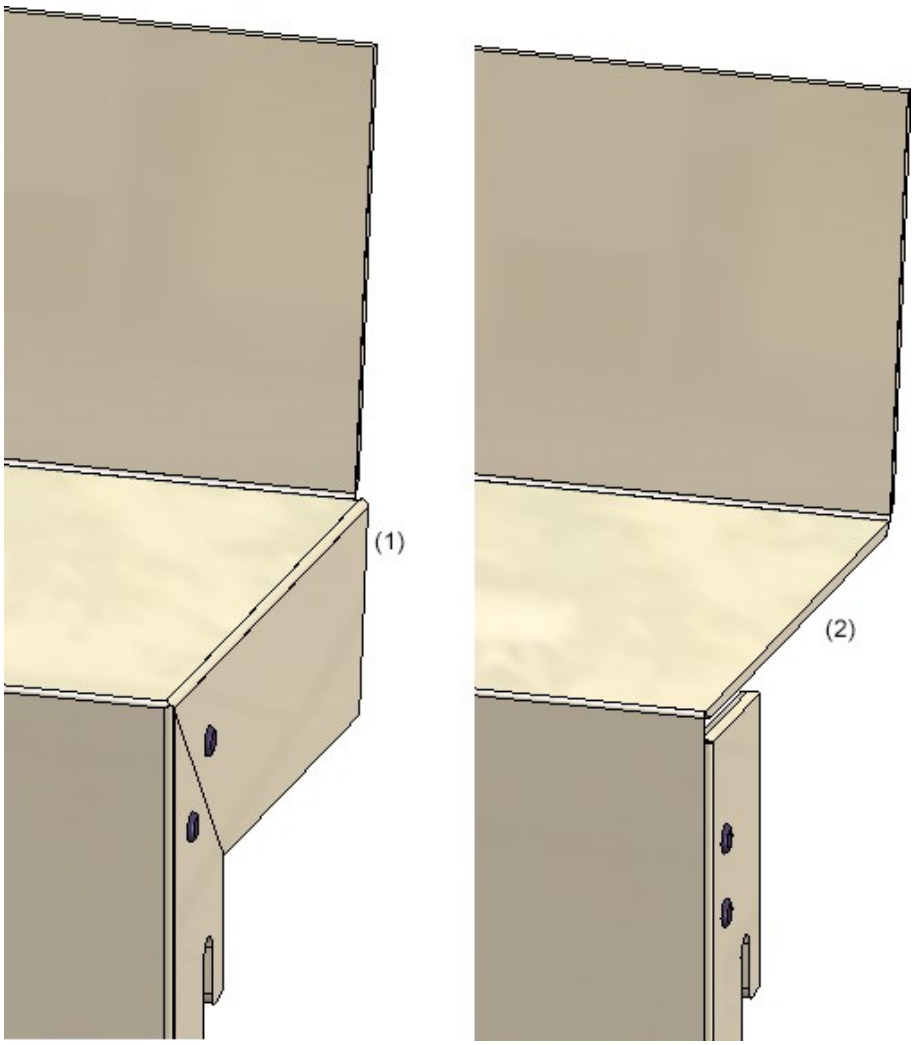




ALUCOBOND® suspended - Window connection without side flanges

As with attic connections, it is now possible to insert the window connection without flanges when using the suspended ALUCOBOND® panels.

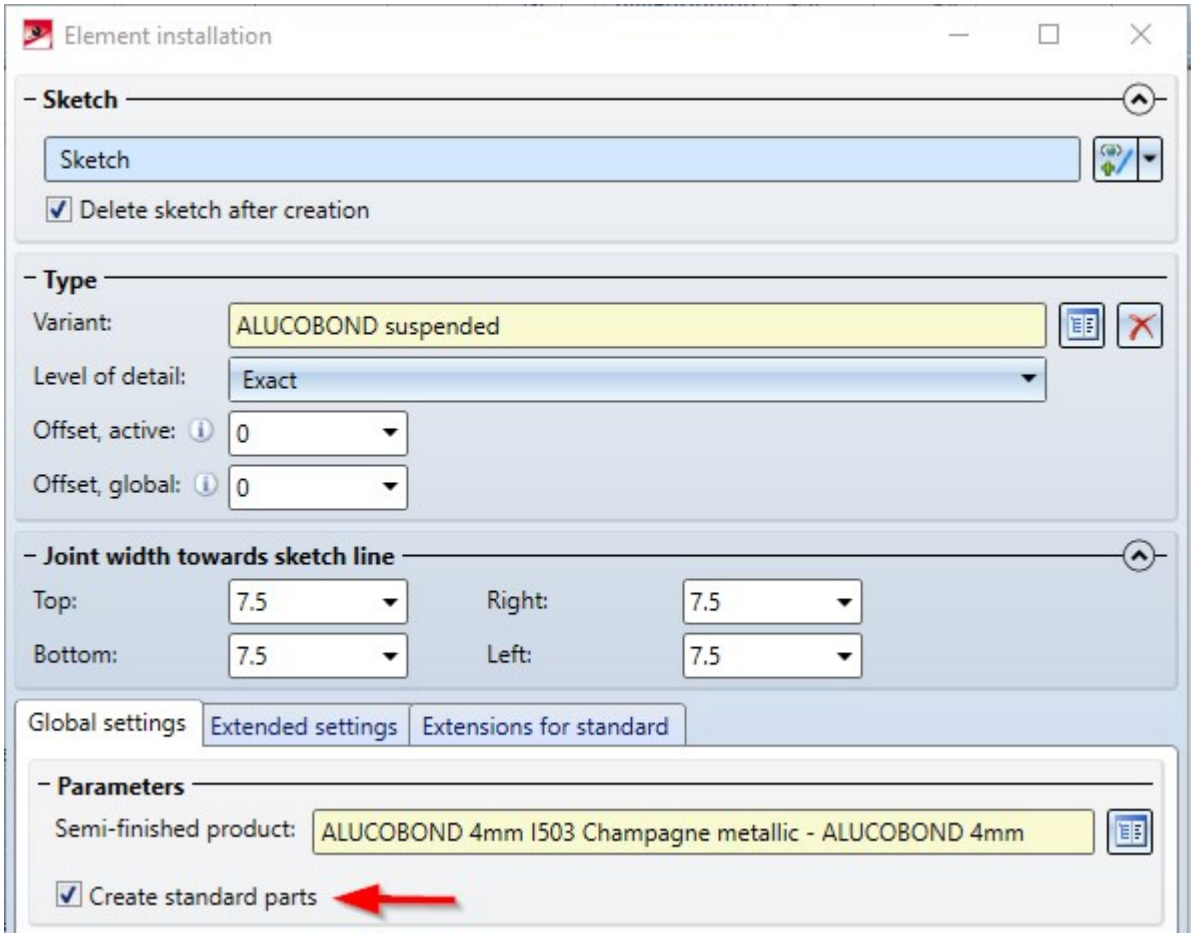




Window connection with (1) and without ohne (2) side flange

Optional insertion of standard parts

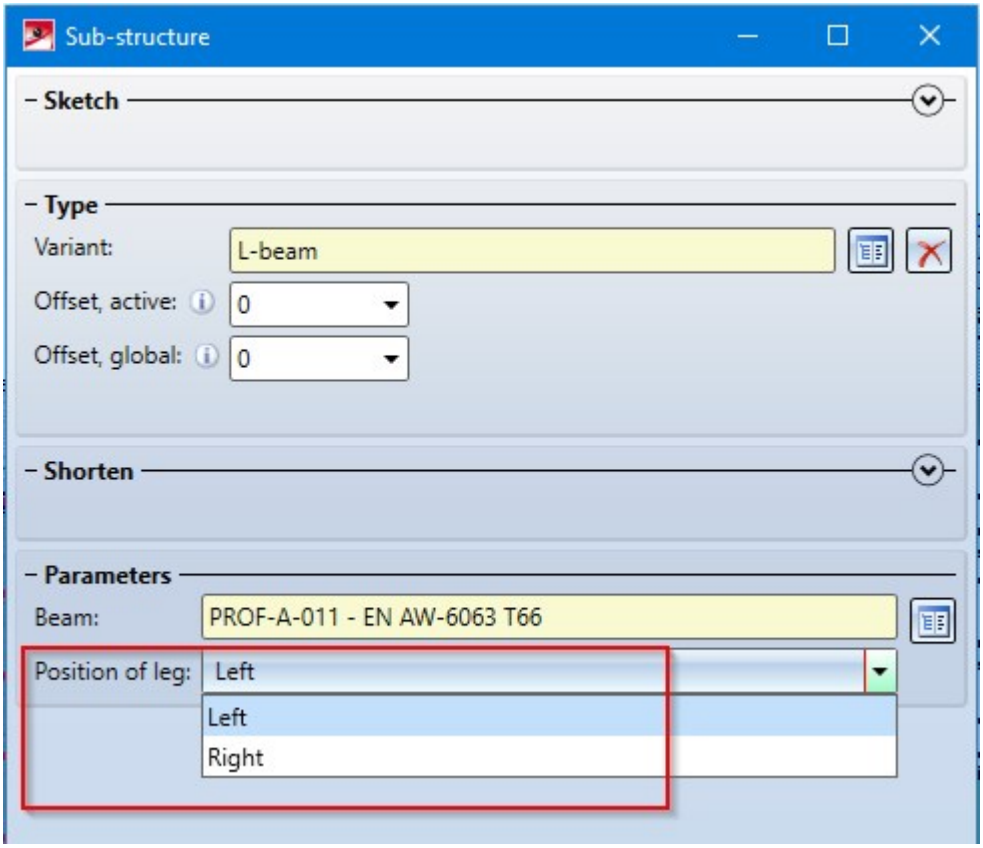
In the element installation it is now possible to choose whether standard parts (rivets, bolts etc.) should be installed or not. For this purpose the **Create standard parts** checkbox has been added to the corresponding dialogue windows. The default setting is that standard parts are created, i.e. the checkbox is active.

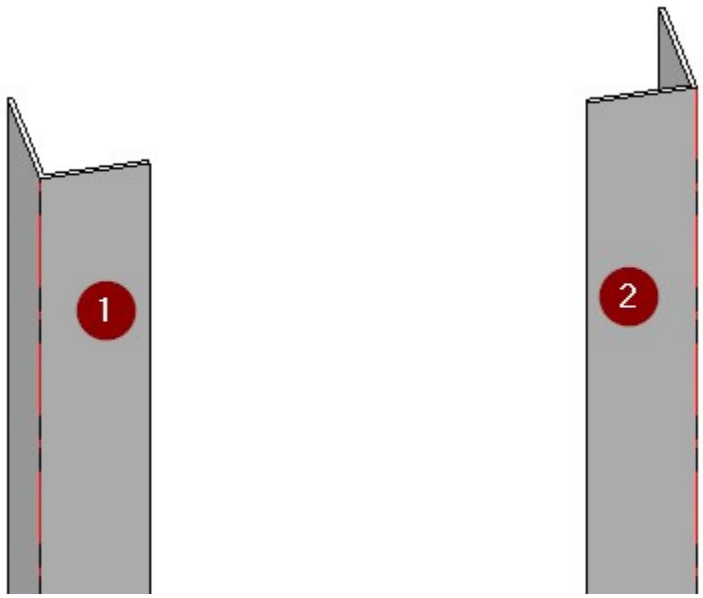


If the checkbox is inactive, the standard parts are not listed in the bill of materials.

Leg position for GIP L-profiles

When creating a sub-structure consisting of GIP L-profiles, it is now possible to choose whether the legs of the profiles should be on the left or right:

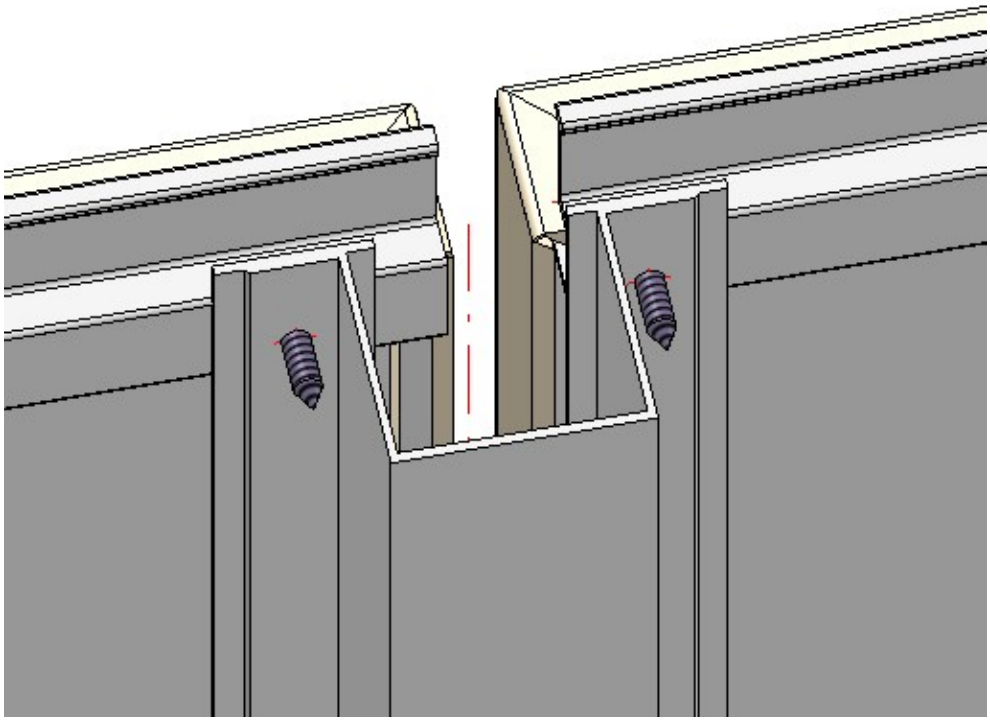




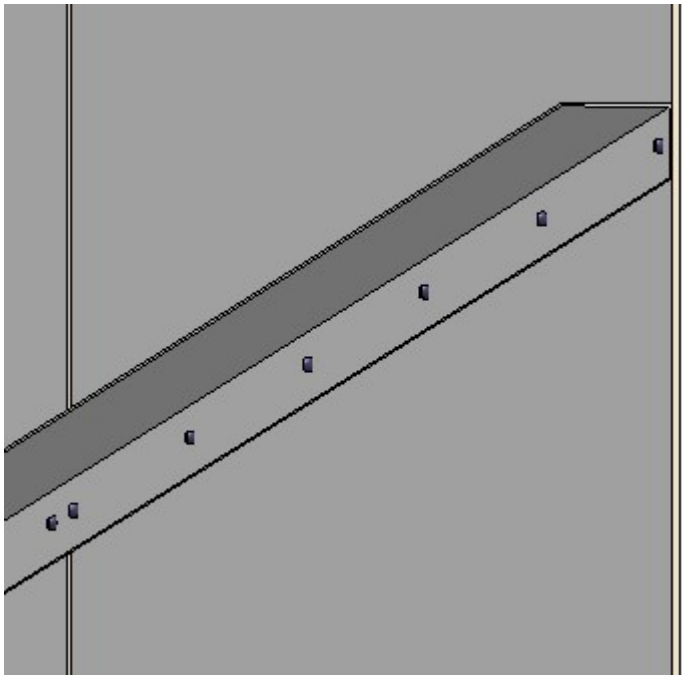
Front view: (1) Leg on the left; (2) Leg on the right

HILTI profiles for ALUCOBOND facades

The HILTI profiles MFT-L, MFT-T and MFT-H SZ-20 are now available for the sub-structure of ALUCOBOND riveted and ALUCOBOND SZ-20 facades. Using the Connection function, the element installation and the sub-structure can be connected to each other, creating the necessary processing and fastenings.



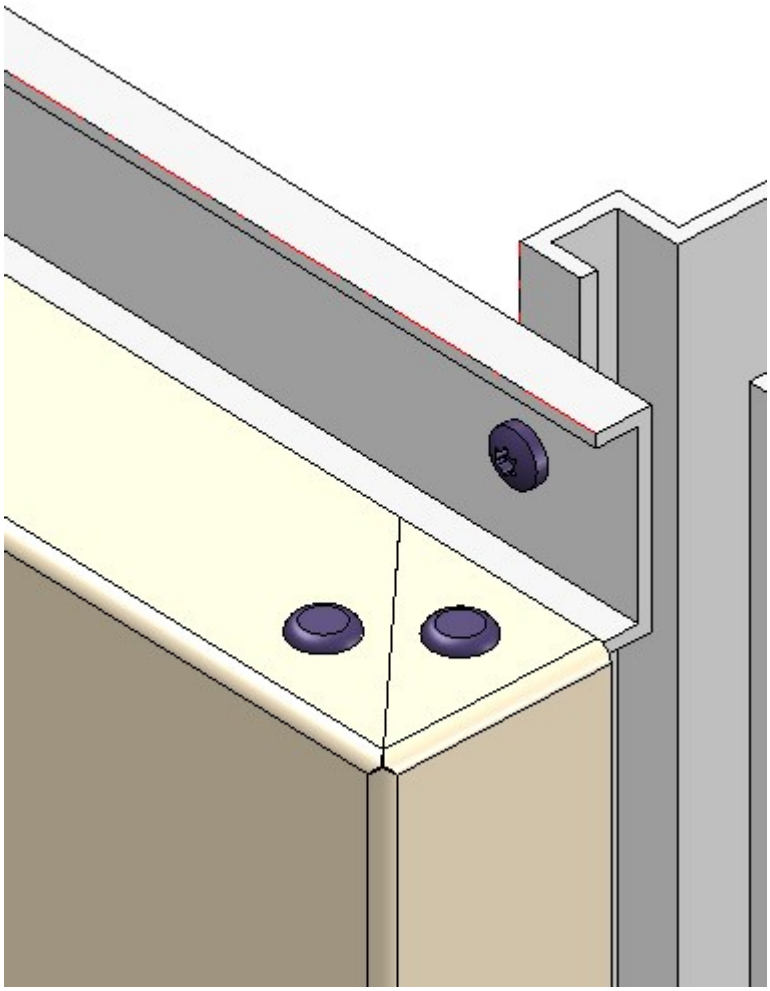
ALUCOBOND SZ-20 facade on a HIILTI MFT-H SZ-20 sub-structure



ALUCOBOND suspended facade on a HILTI MFT-L-sub-structure

CS 20 Window end profile

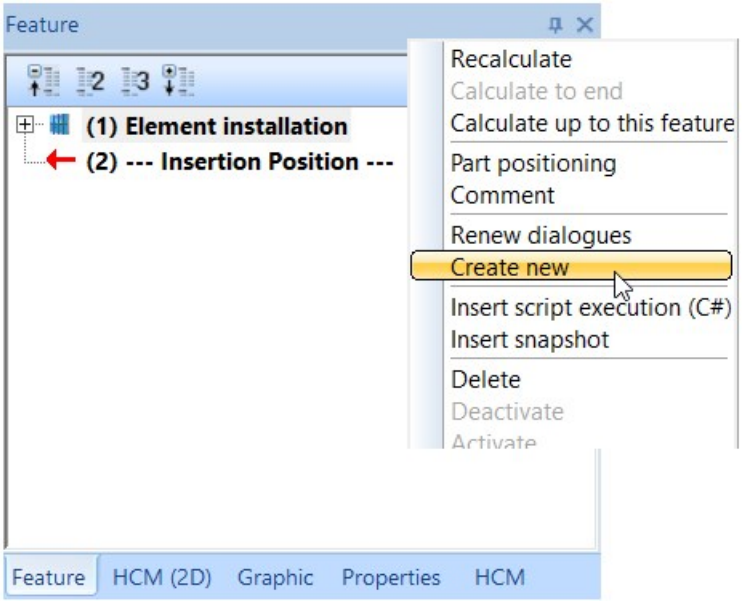
In the Catalogue Editor at **Series > Roof wall facade > Special profiles > Systea > CS 20** you can now find the **CS 20 Window end profile**.



Major Release 2020 (V 2500)

Element installation - Create new

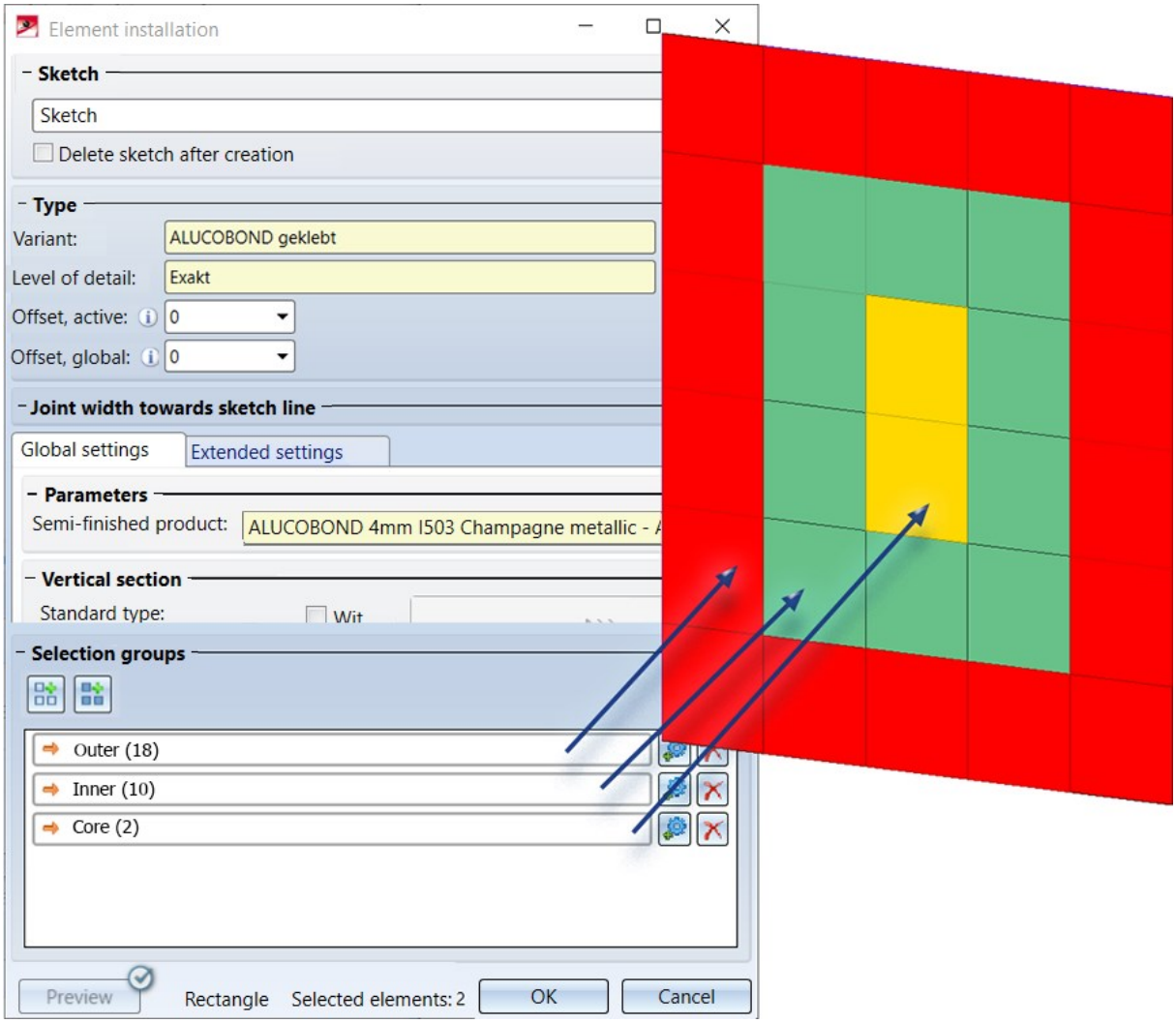
The **Create new** button which was previously available in the Element installations dialog when editing element installations has been removed from the dialog window. This special function is now available in the context menu of the **Element installation** feature.



Before you start please read the notes in the Edit Element Installation topic.

Working with selection groups

When installing elements - especially for complex facades - it is often the case in practice that certain installation elements have the same options. To make it easier to select the areas to be occupied (rectangular or polygonal sketch areas) and to edit installation elements with the same parameters, HiCAD offers the possibility to work with so-called selection groups. The advantage is that you can group installation elements with the same parameters and then select or deselect all elements of this group with one click. The **Selection groups** area is available in the Installation elements dialogue window.



In the image above, 3 selection groups have been defined - one for the red areas, one for the green areas and one for the yellow areas.

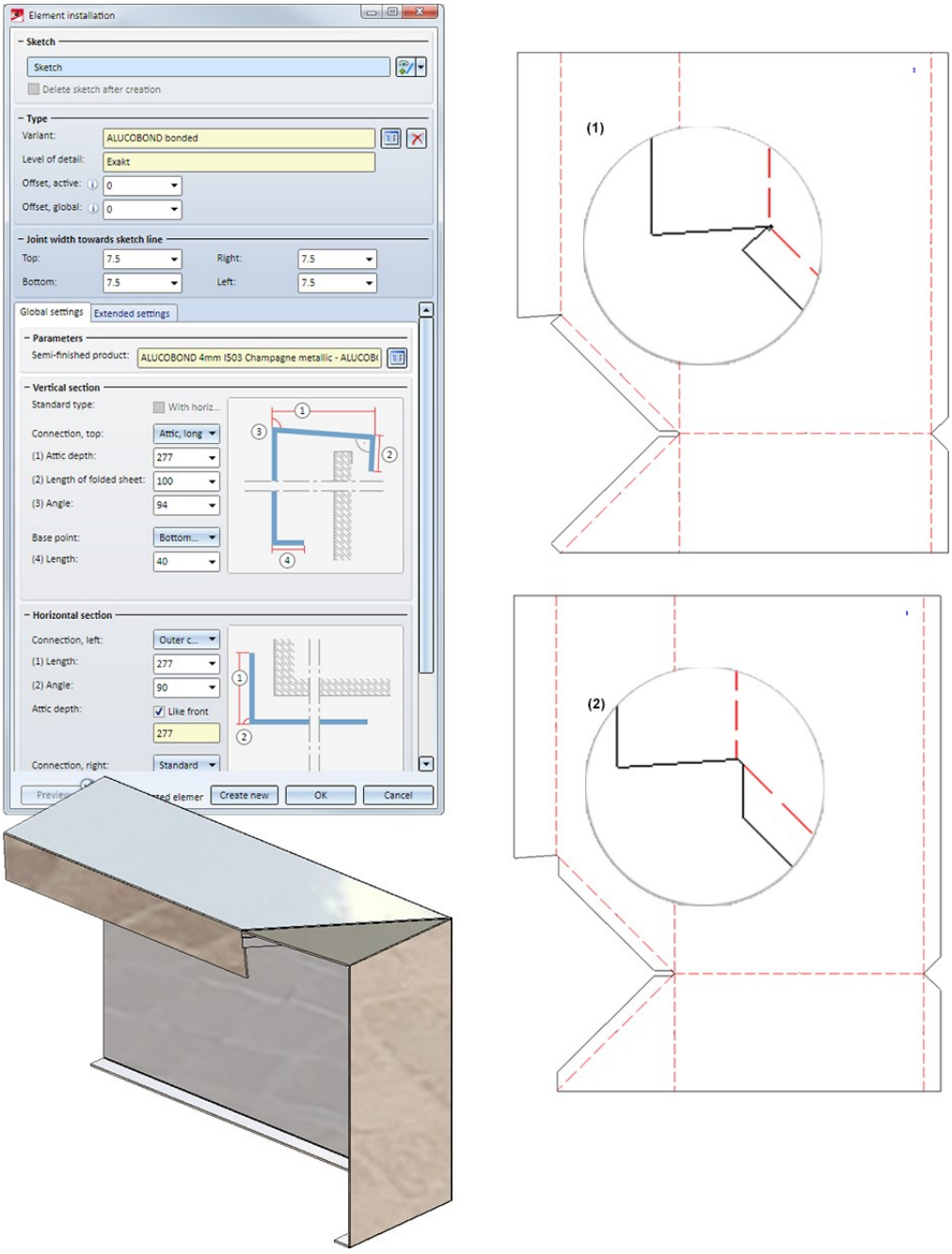
Detailed information on working with selection groups can be found here.

Selection groups can also be applied to **Sub-structures**.

ALUCOBOND® panels - General

Bend zone end suitable for milling

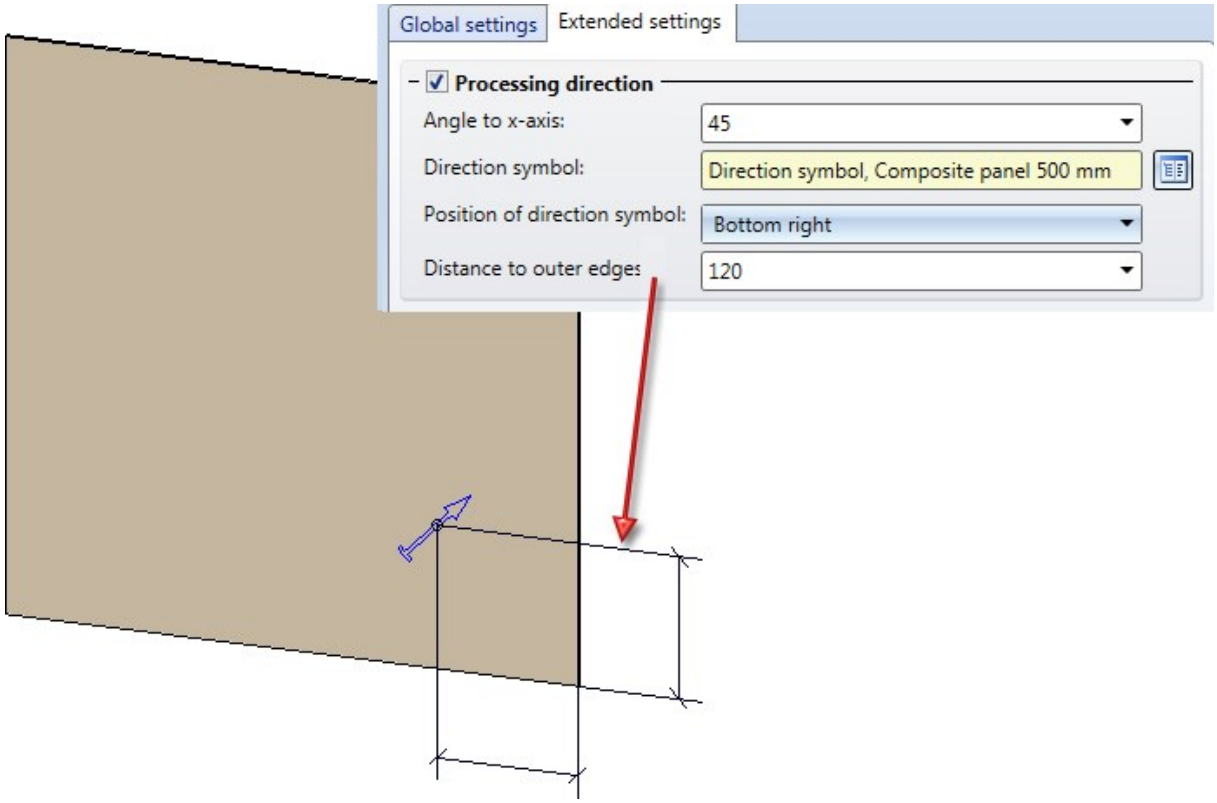
As with the ALUCOBOND SZ-20 tray panels, bend zone ends suitable for milling are now also created for the other ALUCOBOND panels - according the Sheet Metal function **Trim corner**.



Example of a bonded ALUCOBOND tray panel with development - (1) HiCAD 2019, (2) HiCAD 2020

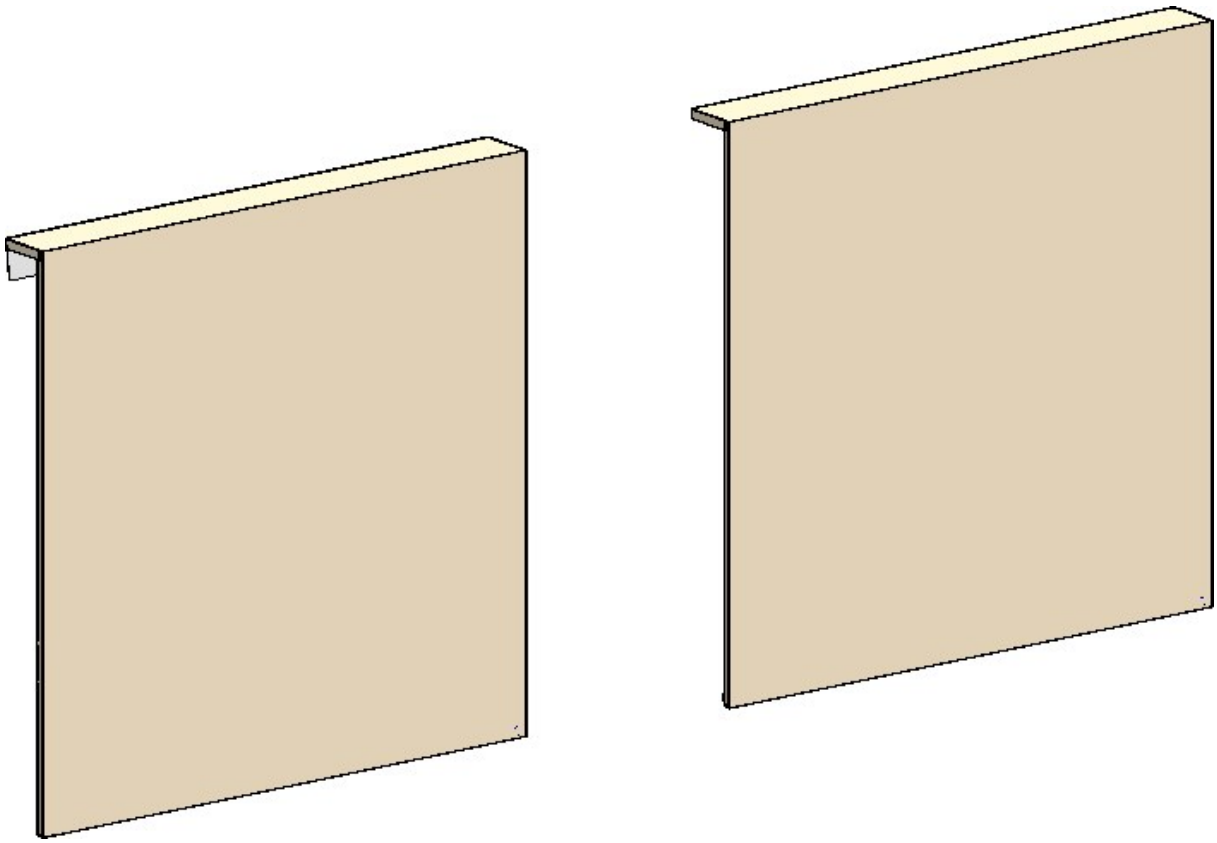
ALUCOBOND® panels - Processing direction and symbol

For all ALUCOBOND panels you can now select the symbol for the processing direction, determine its position, and its distance to the outer edges of the panels. The **Extended settings** tab has been expanded accordingly for this purpose.



Attic connection without last flange

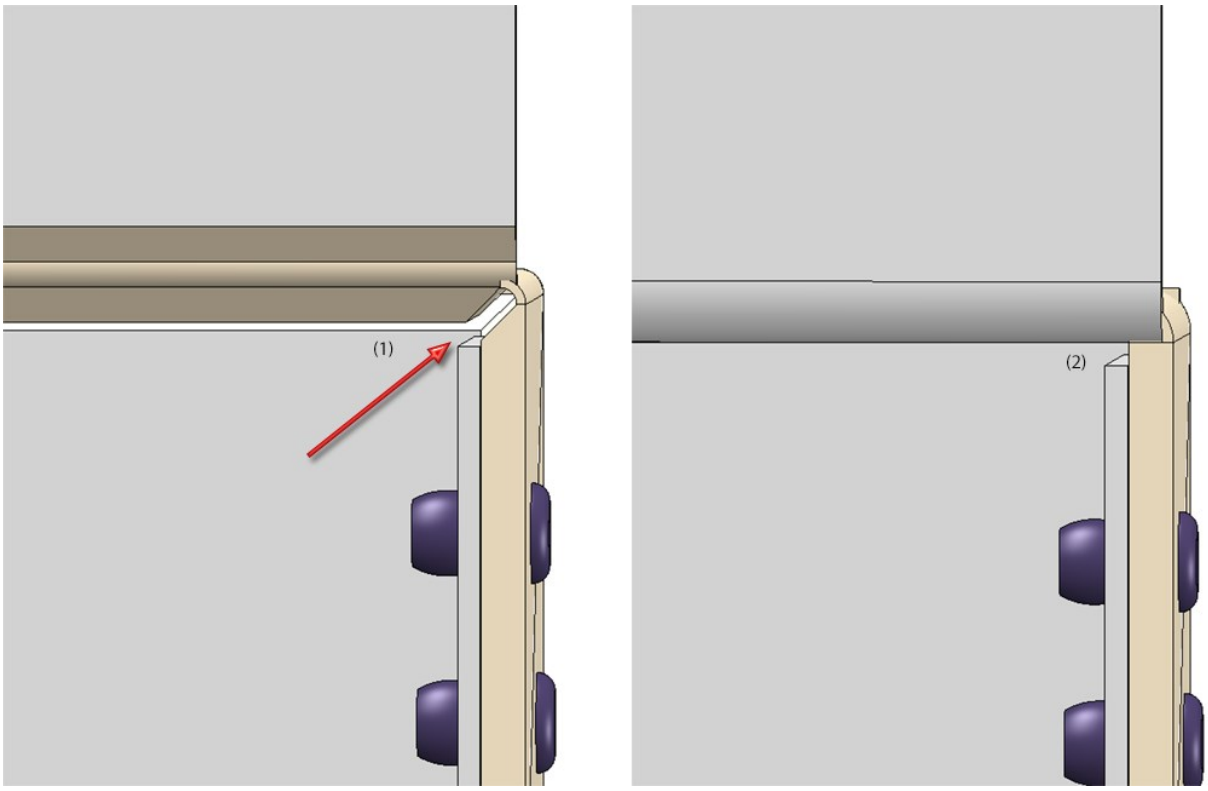
With all ALUCOBOND panels, the parapet connection can also be installed without the last flange. To do this, simply enter the value 0 in the field **Length of folded sheet**.



SZ-20 with attic connection - Left: with last flange; Right: without last flange

All bend edges as milling edges

From HiCAD 2020 onwards, the bend edges of all ALUCOBOND panels are implemented as milling edges.



Example of a suspended ALUCOBOND panel - (1) HiCAD 2020, (2) before HiCAD 2020

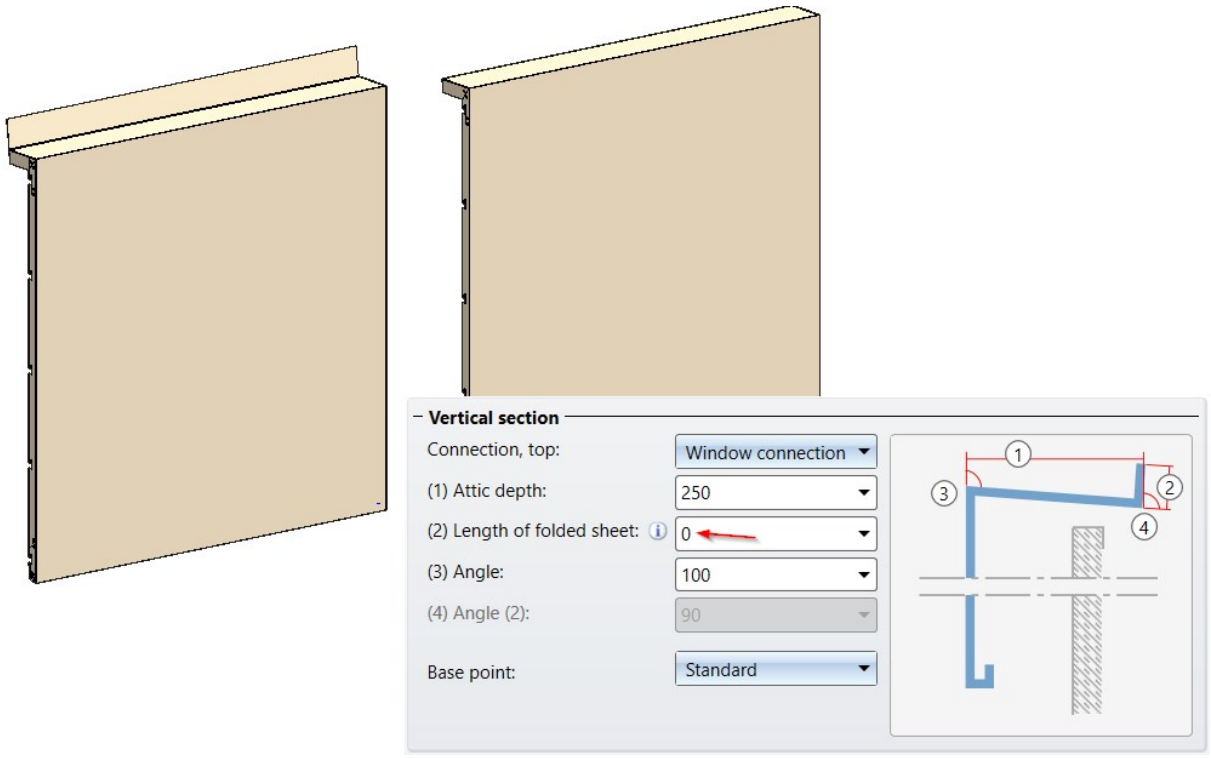
Mitre cut with milling edge

For all ALUCOBOND panels you can now specify that the mitre cut in the sheet development is displayed with a milling edge. To do this, the checkbox **Mitre cut with milling edge** has been added to the **Extended Settings** tab.

ALUCOBOND® panels, suspended

Window connection without flanges

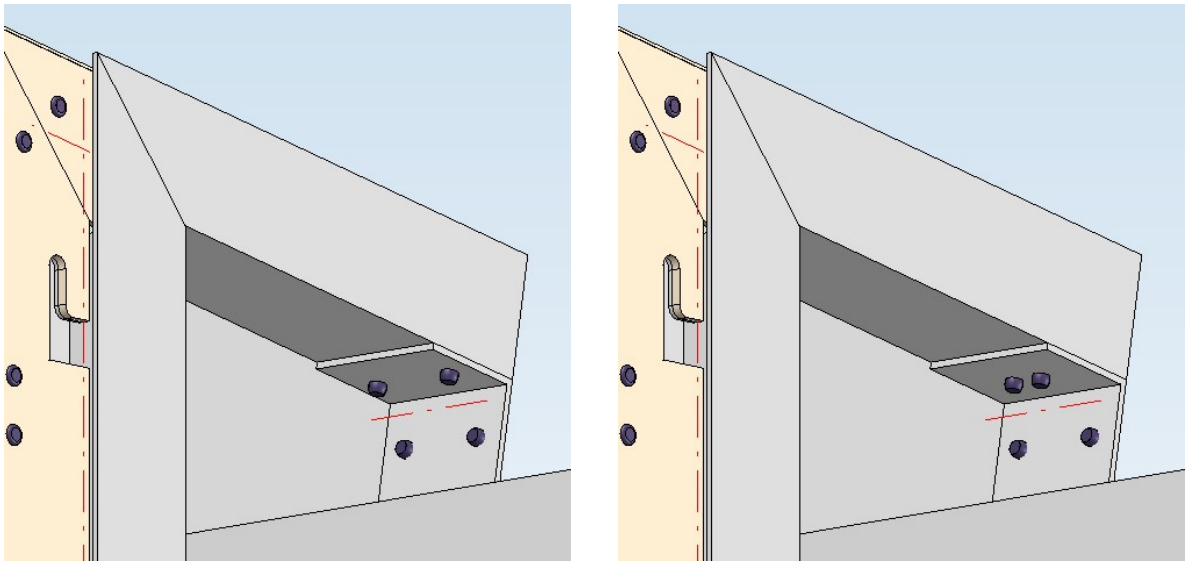
From HiCAD 2020 onwards, it is possible for suspended ALUCOBOND panels to create the window connection even without a rear flange. To do this, simply enter the value 0 in the **Length of folded sheet** field.



Window connection with flange (left) and without flange (right)

Sub-structure - Rivet distances on attic reinforcing bracket

When creating an ALUCOBOND sub-structure of the the type suspended, with reinforcing bracket on the attic, the **Rivet distance** for mounting the reinforcing bracket to the sub-structure can now be adjusted.



Left: Rivet distance 40mm (default value); Right: Rivet distance 15 mm

ALUCOBOND® SZ-20 tray panels

Installation on triangular areas

In the case of ALUCOBOND SZ 20 tray panels laid on a triangular area, the material is removed in the acute corner. In order to prepare this area for machine processing, the corresponding location in the DXF file must be defined as a layer.

HiCAD 2020 automatically recognizes this area for subtraction during sheet metal development by searching for all facets that are not cover facets, are parallel to the cover surface and are not standard machining surfaces. These facets form the area for subtraction. The edges surrounding the area are entered in the development The edges are sorted and oriented as specified for internal cycles.

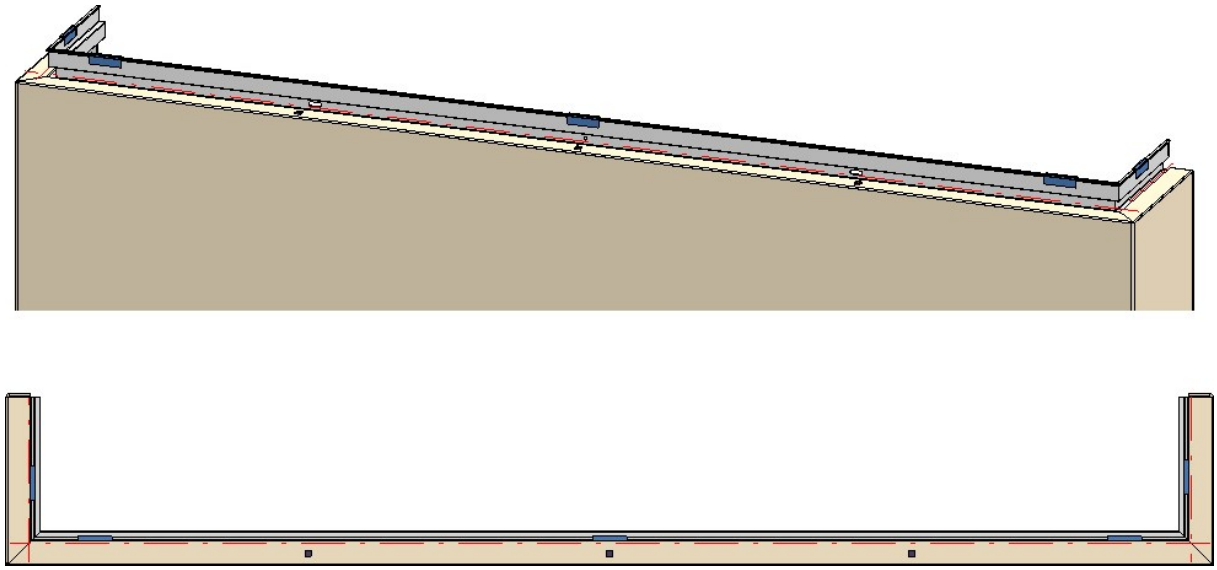
The display parameters for the subtraction area in the development can be defined in the parameters for Sheet Metal development at Milling edges.

Optional insertion of bolts for the sub-structure

The insertion of the screws for the sub-structure is now optional. For this purpose, a corresponding checkbox has been added to the **Extended Settings** tab.

Lateral connection with additional flange

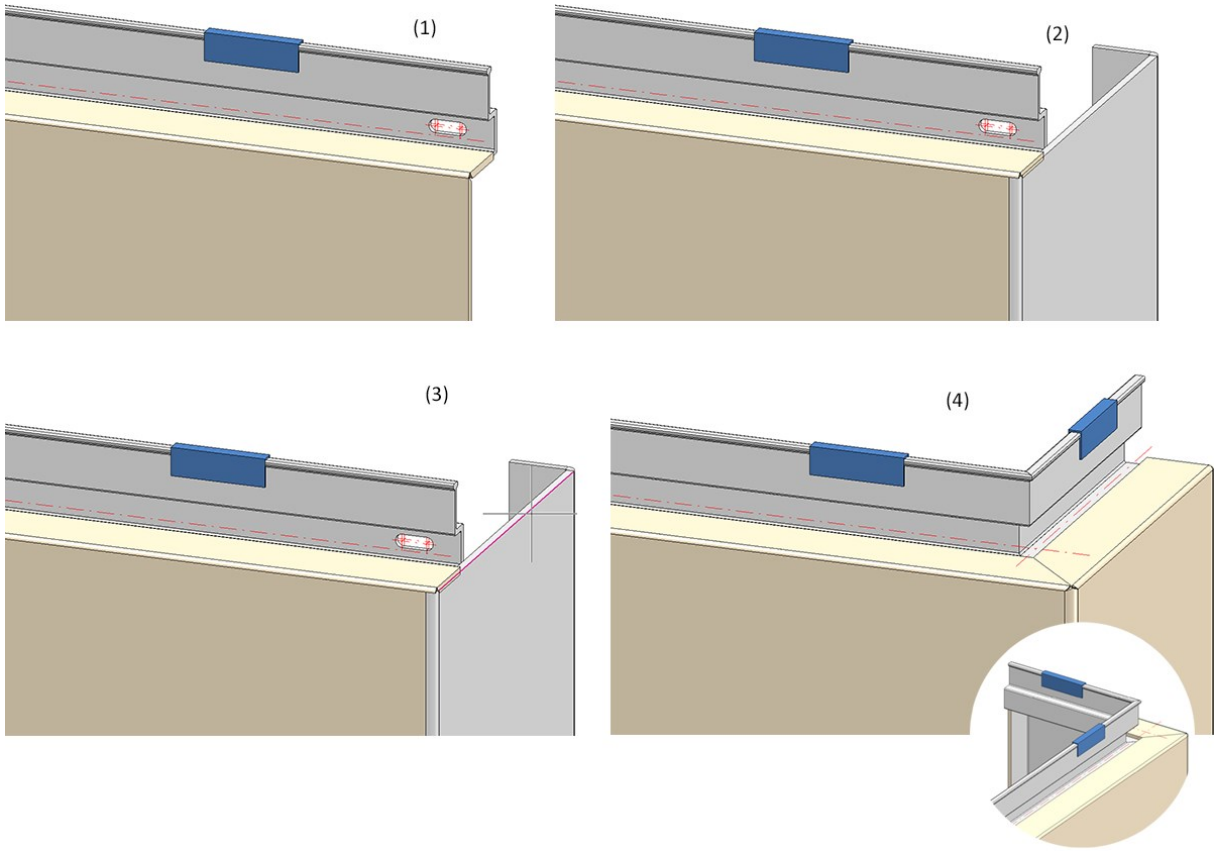
When using ALUCOBOND SDZ 20 tray panels, there is sometimes the desire to create a connection with a flange there as well when connecting from the side, thus creating a "U".



This use case cannot be realized directly via the Element installation dialogue window!

- To create such a connection, proceed as follows instead:
- Construct the SZ-20 panels using the Element installation function. However, select the setting **Without flanges** for the left and/or right connection.
- Attach the new flange on the open side of the panel (optionally with 35 mm lateral flange or lateral flange with the selected panel depth).
- In the **Civil Engineering functions** docking window at **Sheet Metal**, choose the Design Variant **Flange for SZ-20.**
- Then select the upper or lower longitudinal edge of the previously attached new flange

Example:



(1) Element installation without flanges, (2) Attaching of the flanges with 35mm lateral flange, (3) Calling the Design Variant and selection of the marked edge, (4) Result

Please observe the following rules:

- The macro is only valid for ALUCOBOND SZ-20 panels where the standard connection is selected for the vertical section.
- The side connection is installed without a stiffener.
- No automatic connection to the sub-structure is possible, therefore the checkbox **Create screws for sub-structures** on the **Extended settings** tab of the **Element installation** dialogue window should be inactive.

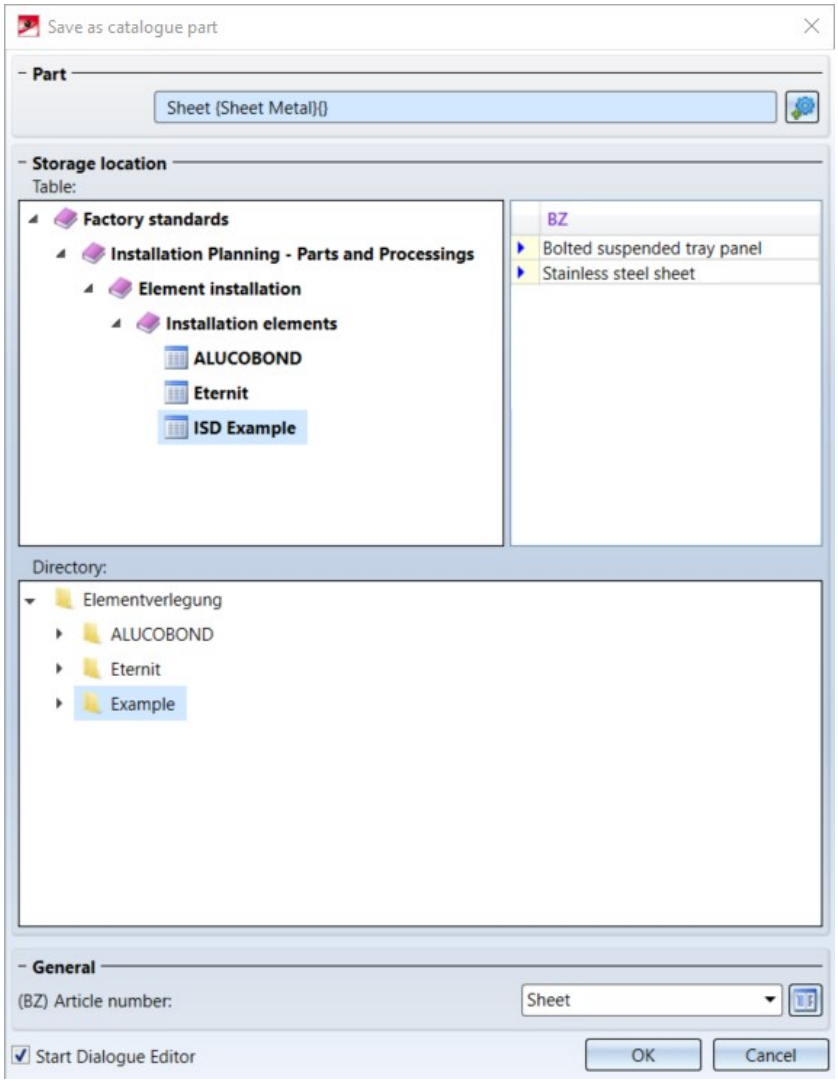
Save variants for Element installation /Sub-structure

For the saving of variants for element installations and sub-structures, you will find the following functions in the **Civil Engineering functions** docking window at **Element installation**:

Save variant for element installation

Save variant for sub-structure

This function automatically creates the corresponding catalogue entries and the CSV files with the variables used in a variant. These CSV files are required if you want to individually design the dialogue windows for installing customer-specific variants using the HiCAD Dialogue Editor (HiCAD GUI Creator). The Editor can be started automatically after saving a variant.



User-specific dialogue windows

- Variables for catalogue entries assigned to a relocation element and not defined as list variables are now automatically converted to list variables when the variant is saved. The text object **CatalogEntry** is then automatically used for these variables when they are inserted into a user dialogue.
- If you have subsequently changed the user dialogue of an already installed variant, the dialogue of this variant can be updated accordingly. To do this, right-click the **Element installation** or **Sub-structure** feature in the feature log of the corresponding element installation or sub-structure. The context menu provides the function **Renew dialogues**. Use this function to update the user dialogue displayed when editing the feature.

Reset wall brackets and sub-structures

If you have made manual changes to a sub-structure or to wall brackets (for example, by drilling holes in a sub-structure or by moving or deleting individual wall brackets), these changes are retained even if these parts are recalculated.

However, if you want to discard the changes you have made manually and restore the sub-structure or the wall brackets to their original state, you can now use the appropriate functions:

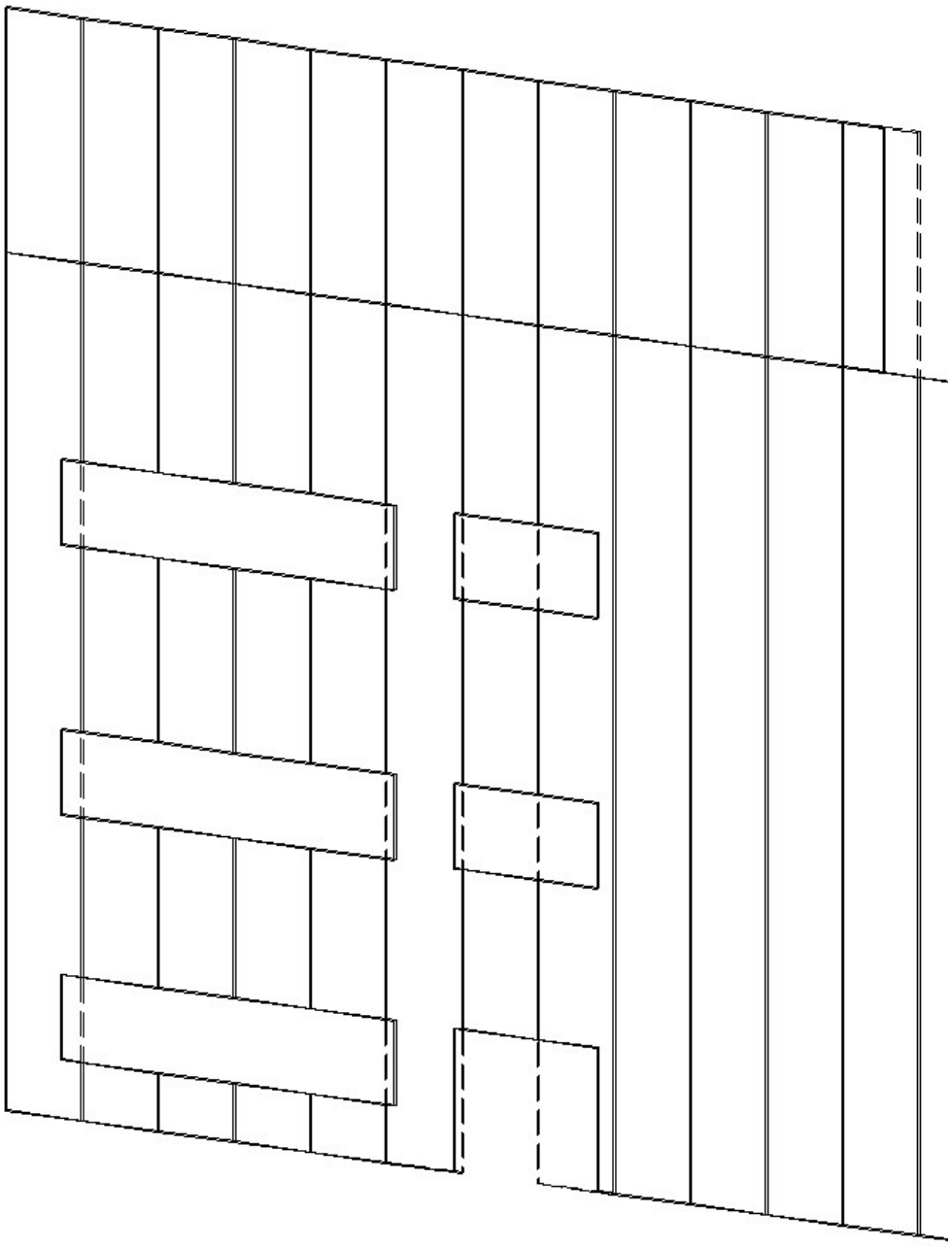
- For **Sub-structures** you will find the menu item **Create new** in the context menu of the **Sub-structure** feature entry . This function deletes all existing parts of a substructure and then inserts them again. This has the advantage that variants that have been changed in the catalogue in the meantime are then also used in the current version in the drawing. However, during this step, further editing and dimensioning of the parts of the sub-structure is lost.
- For **Wall brackets**, the **Restore** function is available in the context menu of the **Wall bracket** entry in the feature log. This retains the installed parts and only moves them back to the correct position if necessary. Deleted wall brackets are also created again.

Profile Installation

Service Pack 2 2020 (V 2502)

New view option: Contour, combined

The **Profile installation** tab now offers the new Contour, combined mode in addition to the previous view modes. This mode is a combination of the modes Contour, cut and Contour, uncut. Cut contours are represented by continuous lines, uncut contours by dashed lines.

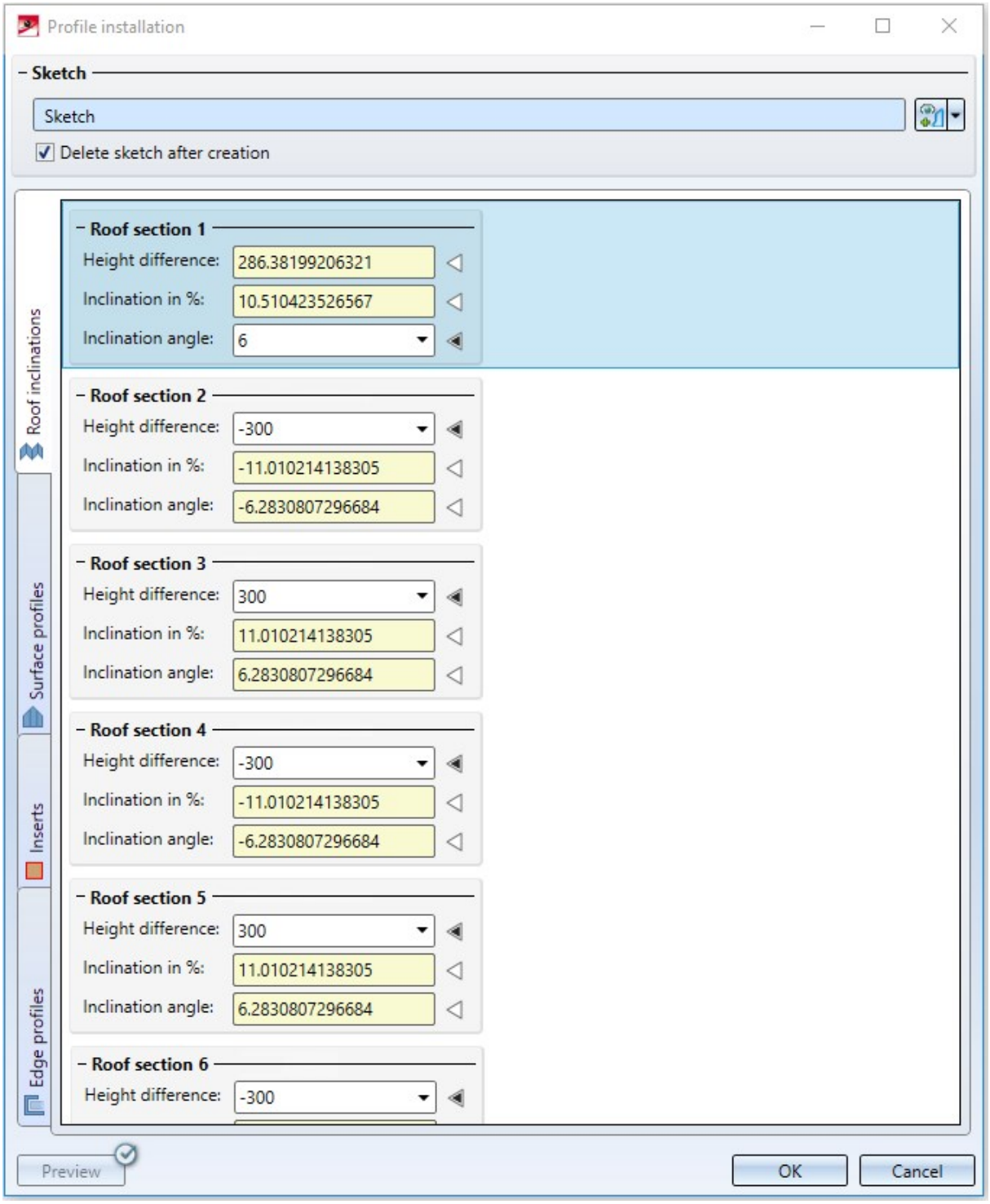


Selecting and creating sketches in the "Profile installation" dialogue

The **Process sketch** and **New sketch in plane** functions of the Profile installation dialogue window can now no longer be found in a context menu, but are displayed directly in the dialogue window:

Roof inclinations: Definition via inclination angle

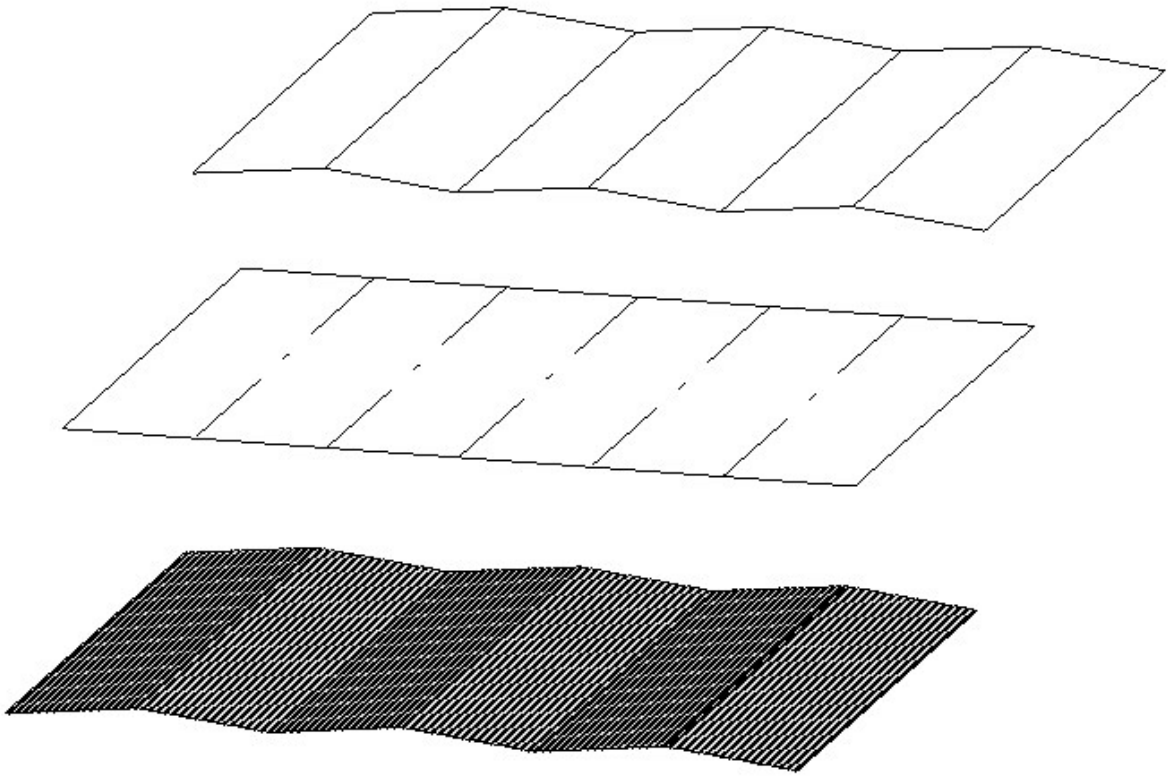
When creating roof inclinations, it is now possible to specify the inclination angle as well as the inclination in % and the height difference. The other values are then calculated automatically.



Service Pack 1 2020 (V 2501)

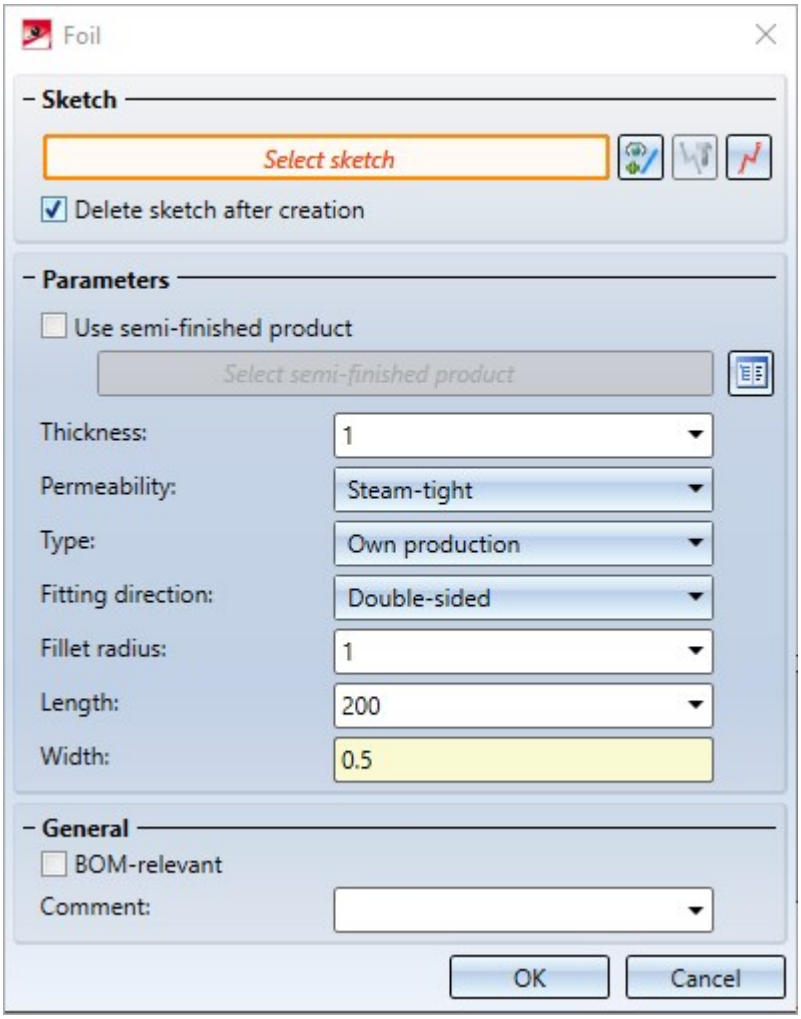
Creation of inclined roof sections

When creating a profile installation on roof surfaces, it is now possible to conveniently define the inclinations of the individual roof sections in the **Profile installation** dialogue window, thus creating a continuous profile installation on a single sketch instead of having to create several inclined sketches.



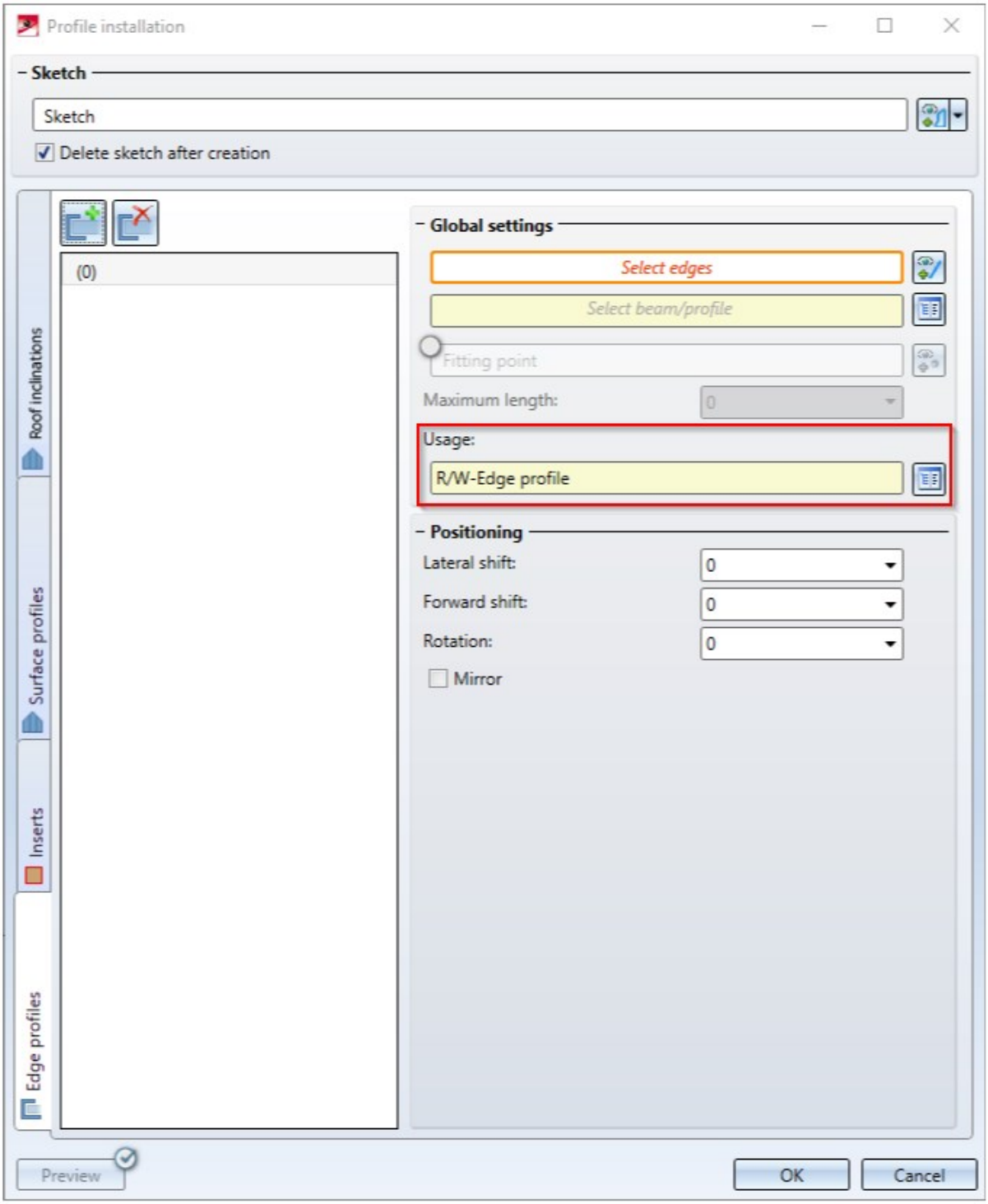
Top: Drawing consisting of several three-dimensional sketches for planning a flat roof with inclined sections before HiCAD 2020 SP1. Bottom: Sketch for the same project, with HiCAD 2020 SP1.

For this purpose, the new **Define inclined roof sections** function is available, with which, as with the **Define transverse joint edges** function , edges of a sketch can be defined as inclined roof sections. The sections can then be selected from the **Roof inclination** tab of the **Profile installation** dialogue window and applying slopes to them.



Usage for edge profiles

When installing edge profiles you can now define a **Usage** for those edge profiles in the **Profile installation** dialogue window.



Attributes for the installed width of the room closing profiles

The installed width of room closing profiles is now written to the DWF_COVER_WIDTH attribute. This can then be output in workshop drawings or BOMs, for example.

Major Release 2020 (V 2500)

Edge profiles from multiple sketches

Self-created **Edge profiles** may now consist of several sketches, similar to **Surface profiles**. In this case, they must be placed in a dummy part. During installation, the individual sketches are then turned into separate parts.

Freely selectable fitting point for surface profiles

By default, the first surface profile is created flush with the start point. In some cases, however, it may be necessary to cut away part of the first profile and only install part of the profile. An example of this would be tongue and groove systems on sandwich profiles.

Therefore it is now possible to select a **Fitting point** for profile installations where the first profile will be cut off.

Deactivate preview

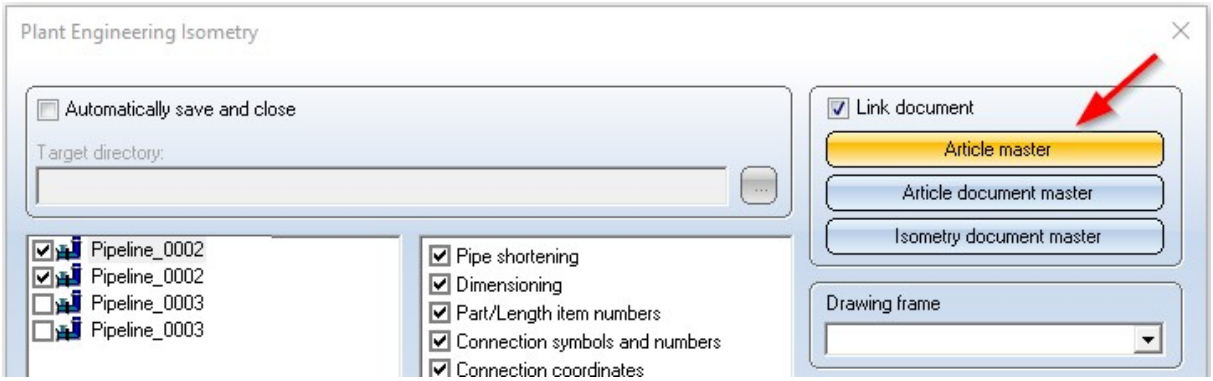
In the **Profile Installation** dialogue window, you can now control the generation of the preview. On the one hand, you now have the option of switching between exact and simplified profiles for the preview as well. On the other hand, you can now deactivate the automatic generation of a preview and only display a preview on demand.

Plant Engineering

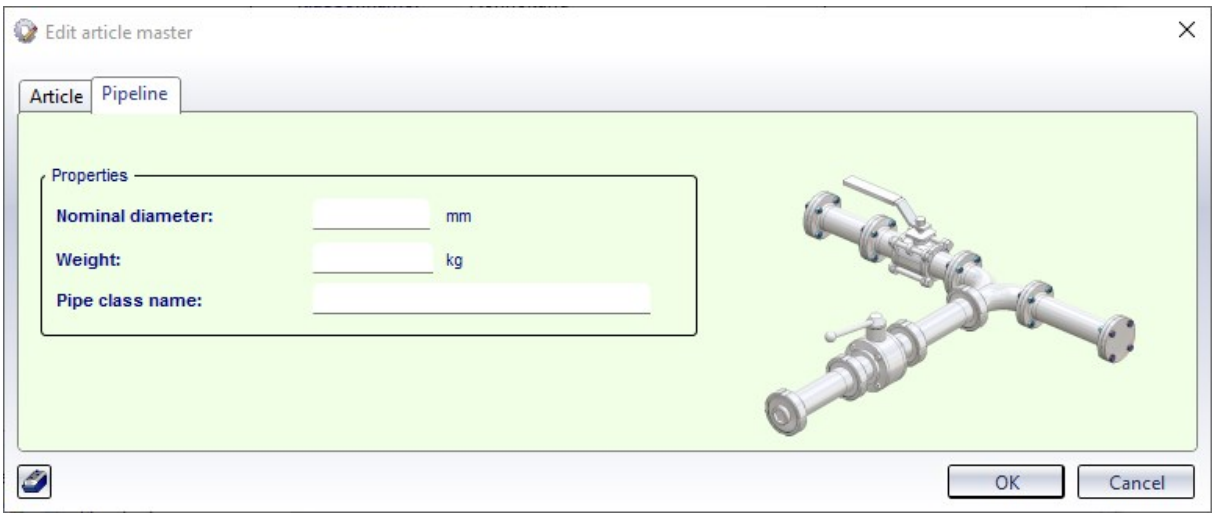
Service Pack 2 2020 (V 2502)

Classify pipeline article

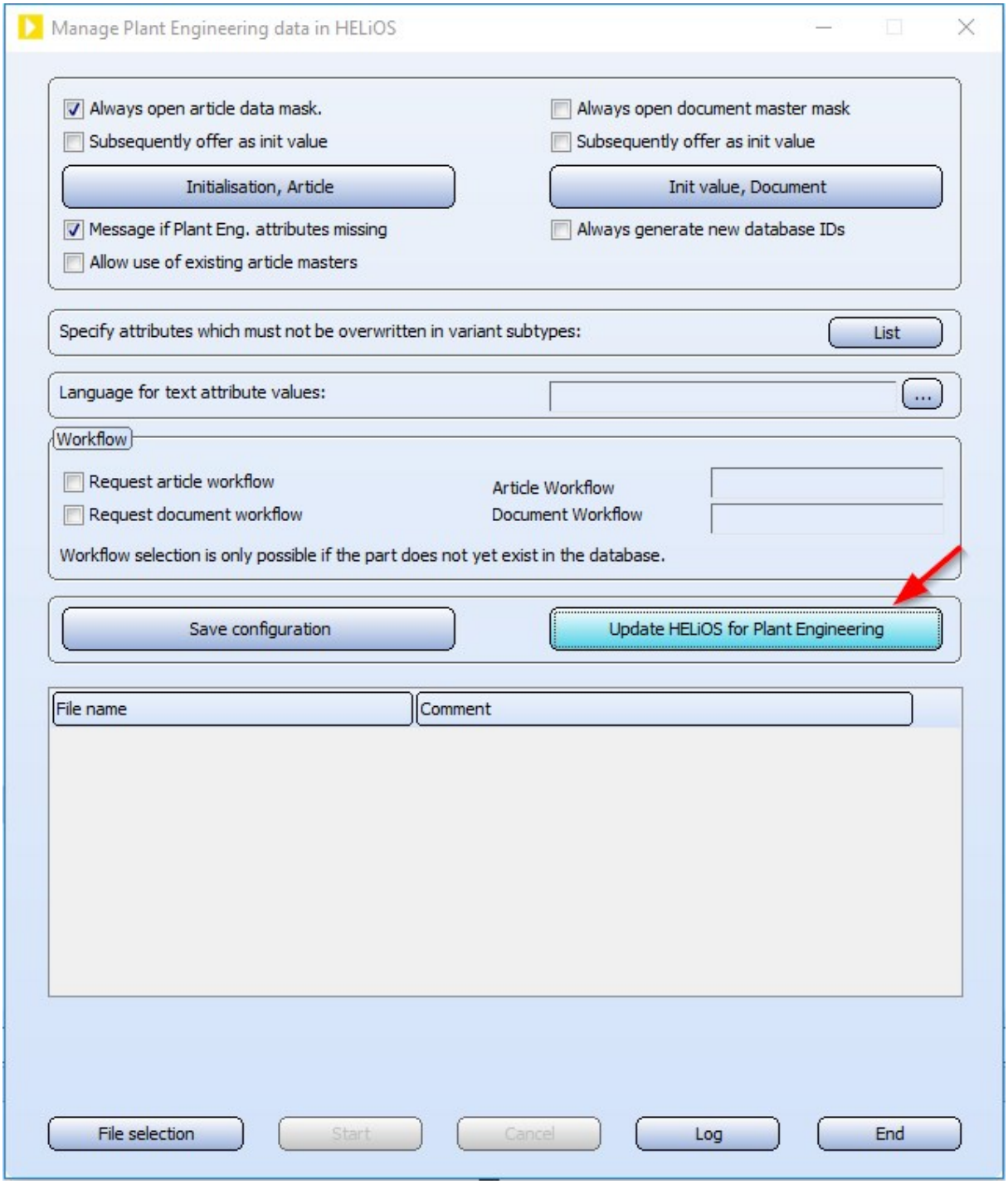
If you select the creation of an article master when generating isometrics or piping plans, the article master in HELiOS is assigned the new classification **Pipeline** as of HiCAD 2020 SP2. However, this only applies to articles that you generate as of HiCAD 2020 SP2.



For the classification **Pipeline**, a corresponding input mask and the new HELiOS attribute PIPE_CLASS_NAME for the pipe class name are available in HELiOS.



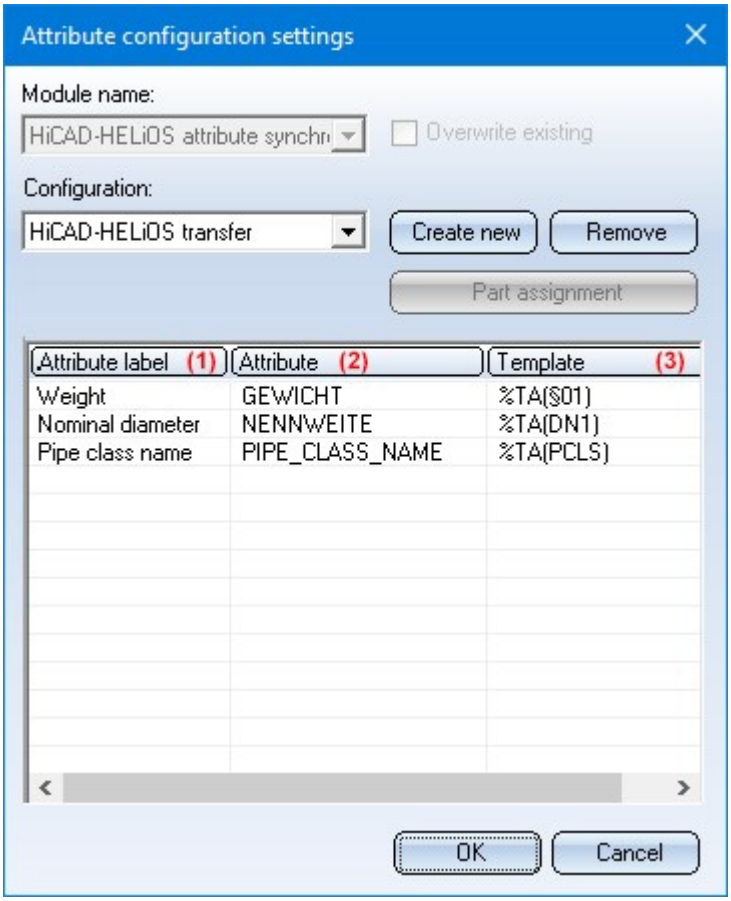
To use the new mask and the new attribute you must first update HELiOS using the tool **DbplantDataImport.exe** and then restart HiCAD.


Manage Plant Engineering data in HELiOS
Always open article data mask.
Subsequently offer as init value
Initialisation, Article
Message if Plant Eng. attributes missing
Allow use of existing article masters
Always open document master mask
Subsequently offer as init value
Init value, Document
Always generate new database IDs
Specify attributes which must not be overwritten in variant subtypes:
List
Language for text attribute values:
...
Workflow
Request article workflow
Request document workflow
Article Workflow
Document Workflow
Workflow selection is only possible if the part does not yet exist in the database.
Save configuration
Update HELiOS for Plant Engineering
File name
Comment
File selection
Start
Cancel
Log
End

Automatic transfer of the pipeline attributes

The attributes of a pipeline can also be automatically transferred from HiCAD to HELiOS when saving. To do this, proceed as follows:

1. On the **HELiOS PDM** Ribbon tab at **Others > Link > ...**, select the function **Article master sync when saving**. Specify the assignments shown there.



(1) Attribute, (2) Target attribute, (3) Creation scheme
End the function with **OK**.

2. Since the assignments only become effective after a restart, close HiCAD.
3. In the Configuration Editor at **System settings > HELiOS**, activate the **Transfer part attributes to HELiOS** checkbox.
4. The next time you start HiCAD, the settings you made previously will apply.

Please note:
If you want to subsequently assign the classification **Pipeline** to existing articles, you must change this manually. To do this, click on the **Classification** symbol in the dialogue window for the respective article and then activate the **Pipeline** checkbox at **Plant engineering** in the Classification dialogue window. You can then assign/change the attributes Nominal diameter, Weight and Pipe class name.

Move pipe parts

At **Plant Engineering > Part Tools > Copy > ...** you can find the new **Move pipe parts** function.

This function works similar to copying of parts. The difference is that here straight pipes are not moved and the parts to be moved must form a straight string. In addition, after moving parts, an attempt is made here to fill the gap with straight pipes.

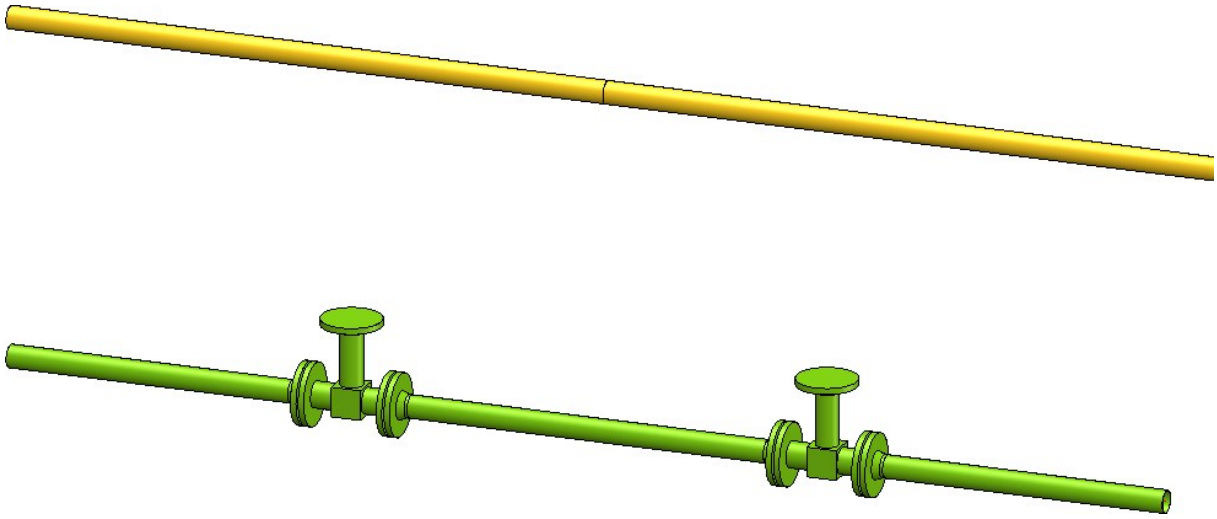
Before you call the function, you must first select the parts to be moved. This can be done either manually or with the **Select part chain** function. The selection of the parts to be moved may contain straight pipes, but they are not moved with them.

After calling the function, HiCAD prompts you to select a so-called handle for moving the part. This is the point over which the move is placed in the drawing. The allowed attachment points are highlighted in colour in the drawing.

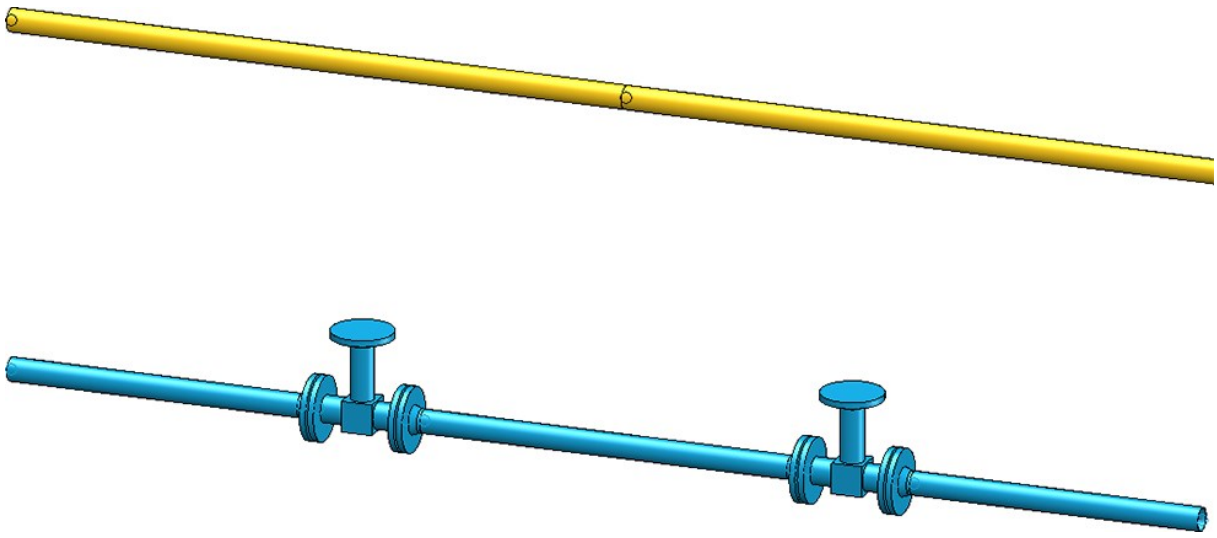
If you have selected the handle, all parts lying on a dashed line running through this point will be attached to the cursor. Then, determine the target point of the move. If necessary, a dynamic route change is automatically carried out at the target point to make room for the moved parts.

An example:

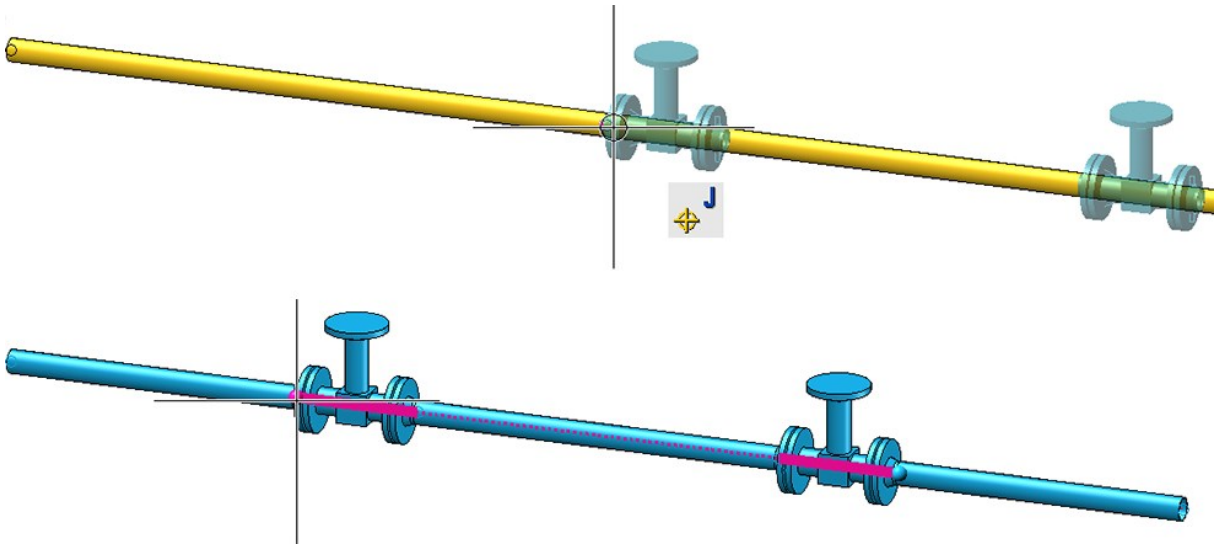
Let's take a look at the drawing shown below. The two valves are to be moved to the upper pipeline.



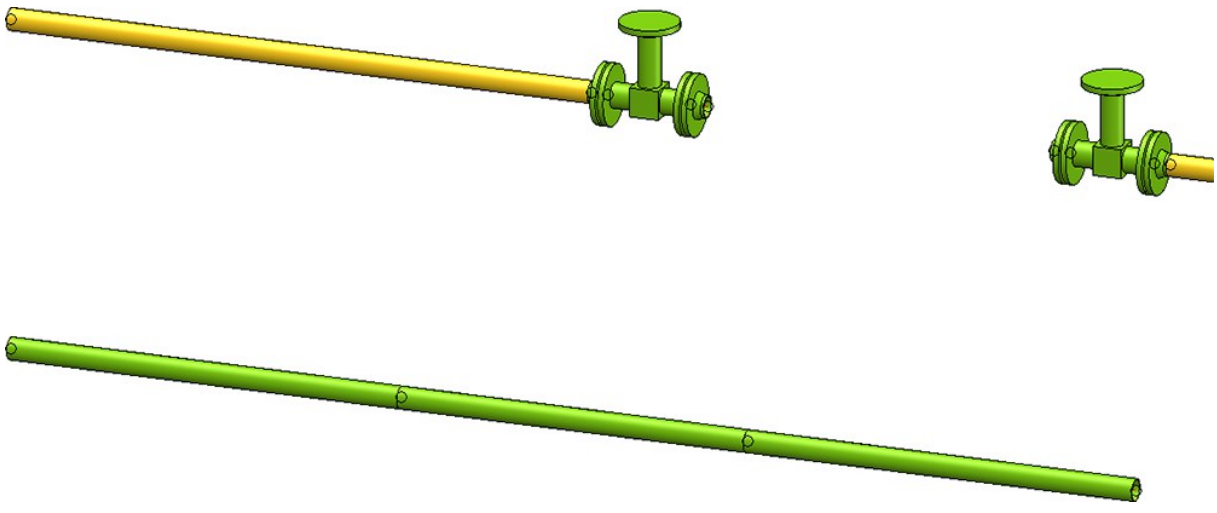
Select the desired part chain. Alternatively one could also select the two valves manually, which, however, would take more mouse clicks.



The possible handles of the part are highlighted. Select a handle and specify the target point for the displacement.

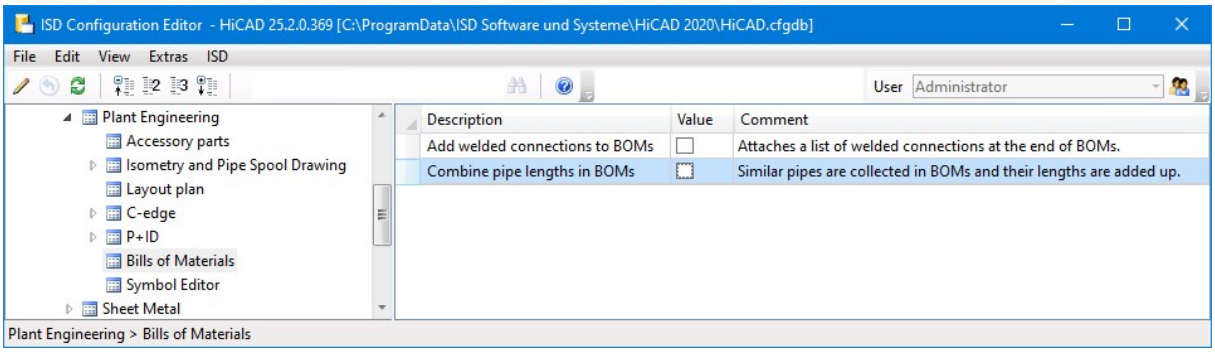


Result:



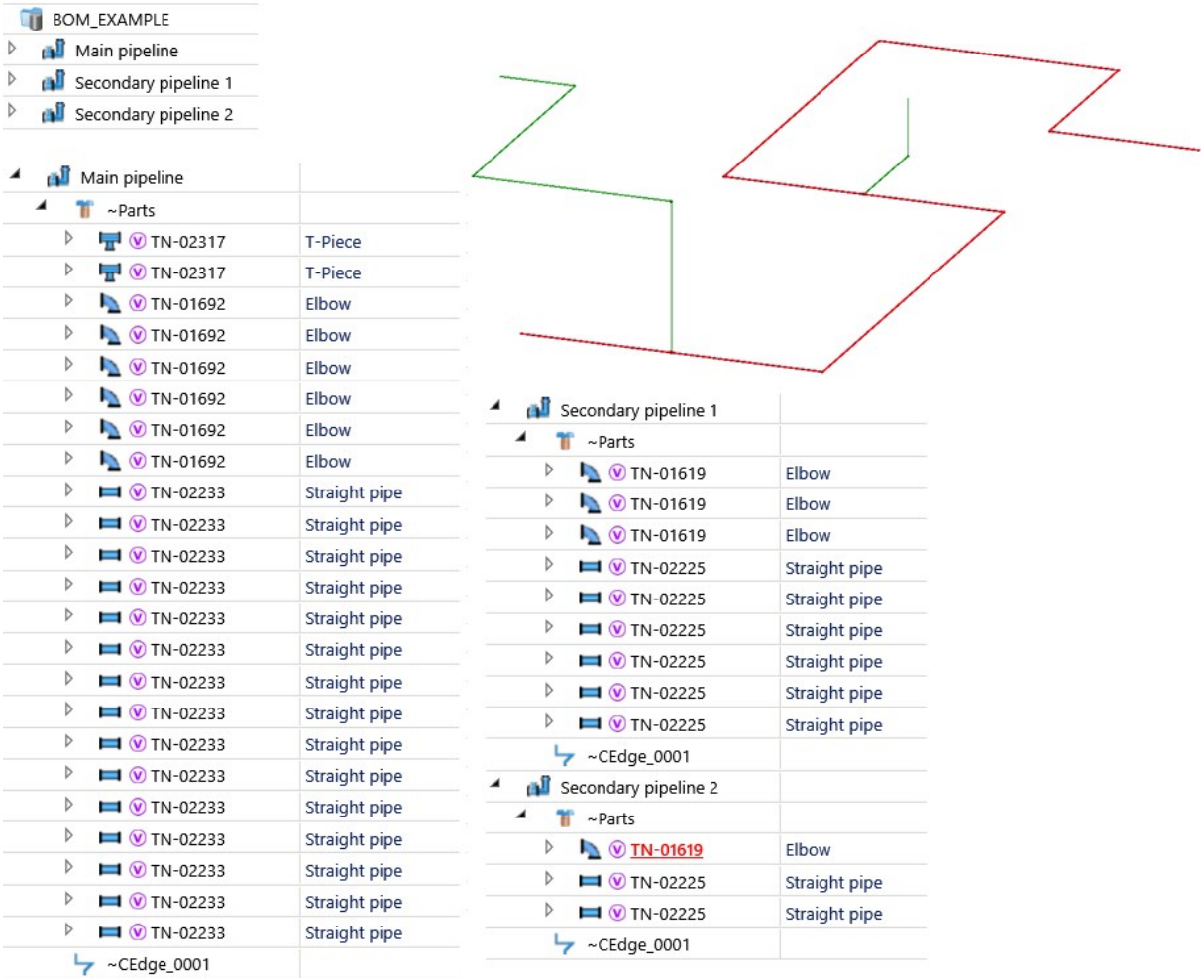
Combined BOM / length list

When generating bills of materials, the lengths of similar straight pipes can be combined. The **Combine pipe length lists in BOMs** checkbox is available in the Configuration Editor at **Plant Engineering > Bills of materials**.

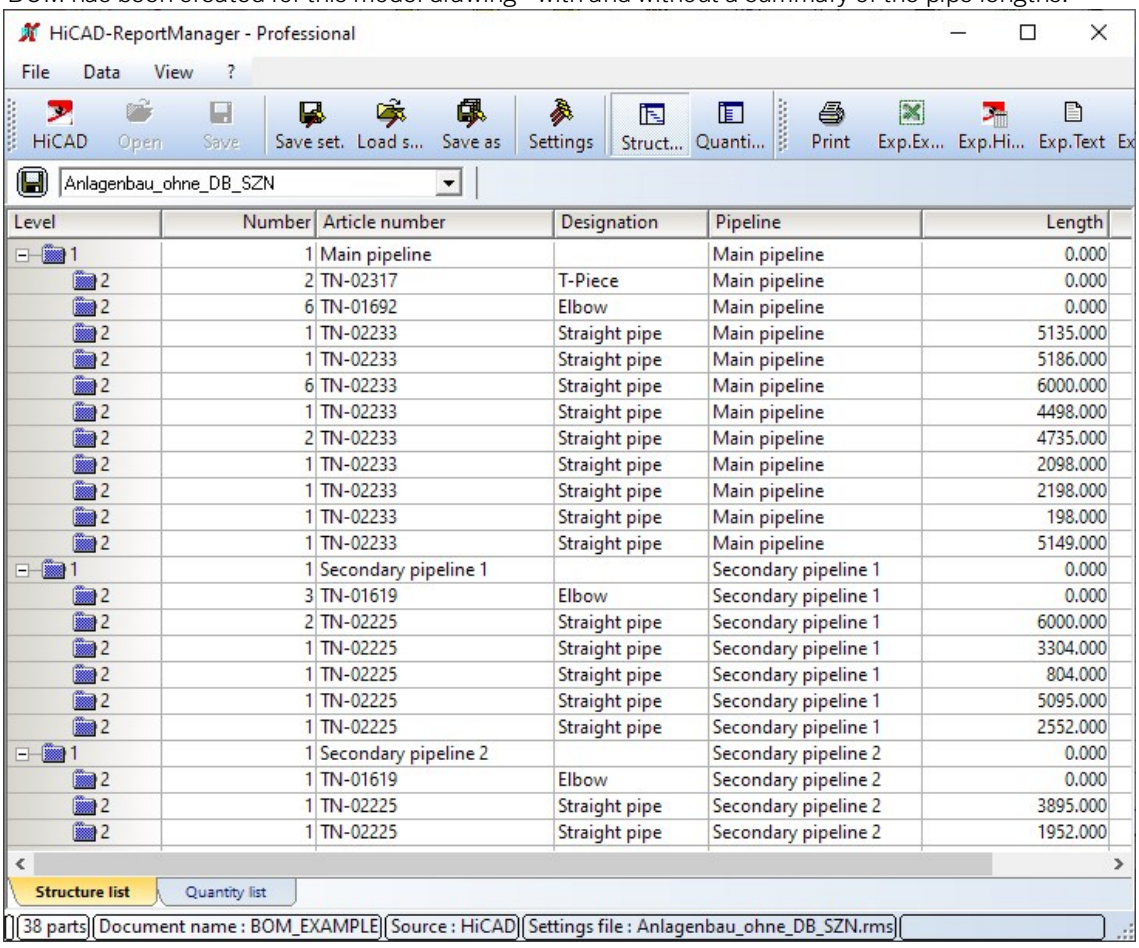


If the checkbox is active, similar pipes are combined to a single pipe when generating bills of materials. The length of this pipe is equal to the sum of the individual pipe lengths. By default, the checkbox is inactive.

The following figure shows a construction with three pipelines - one main pipe (red) and two secondary pipes (green).

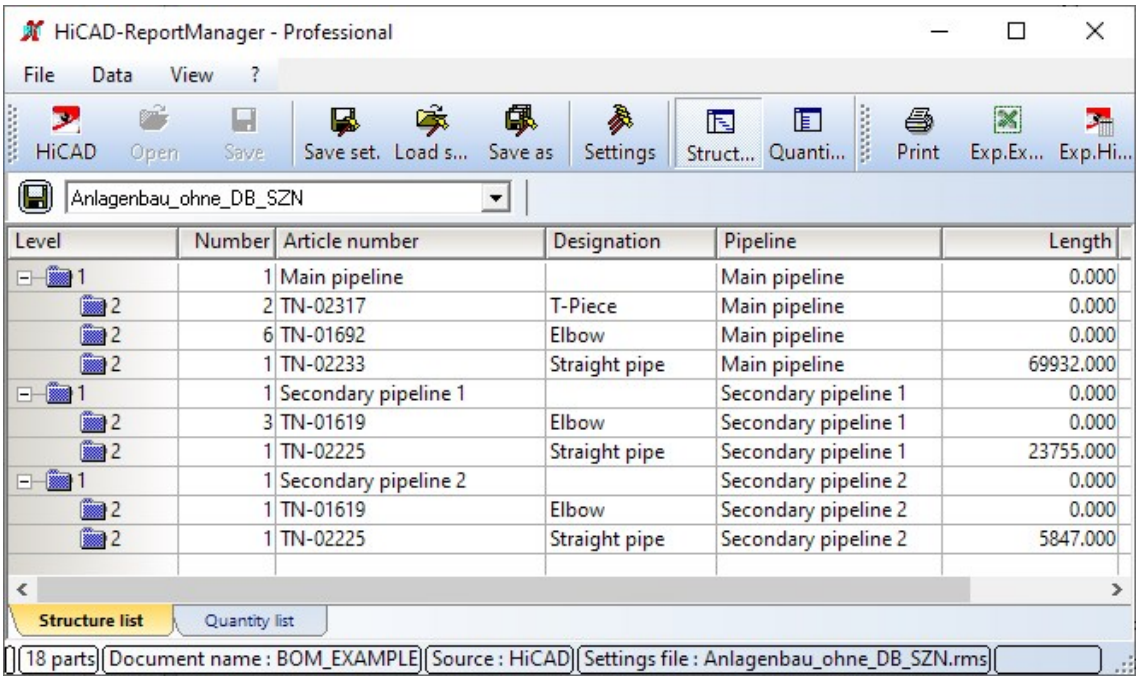


The BOM has been created for this model drawing - with and without a summary of the pipe lengths.



Level	Number	Article number	Designation	Pipeline	Length
1	1	Main pipeline		Main pipeline	0.000
2	2	TN-02317	T-Piece	Main pipeline	0.000
2	6	TN-01692	Elbow	Main pipeline	0.000
2	1	TN-02233	Straight pipe	Main pipeline	5135.000
2	1	TN-02233	Straight pipe	Main pipeline	5186.000
2	6	TN-02233	Straight pipe	Main pipeline	6000.000
2	1	TN-02233	Straight pipe	Main pipeline	4498.000
2	2	TN-02233	Straight pipe	Main pipeline	4735.000
2	1	TN-02233	Straight pipe	Main pipeline	2098.000
2	1	TN-02233	Straight pipe	Main pipeline	2198.000
2	1	TN-02233	Straight pipe	Main pipeline	198.000
2	1	TN-02233	Straight pipe	Main pipeline	5149.000
1	1	Secondary pipeline 1		Secondary pipeline 1	0.000
2	3	TN-01619	Elbow	Secondary pipeline 1	0.000
2	2	TN-02225	Straight pipe	Secondary pipeline 1	6000.000
2	1	TN-02225	Straight pipe	Secondary pipeline 1	3304.000
2	1	TN-02225	Straight pipe	Secondary pipeline 1	804.000
2	1	TN-02225	Straight pipe	Secondary pipeline 1	5095.000
2	1	TN-02225	Straight pipe	Secondary pipeline 1	2552.000
1	1	Secondary pipeline 2		Secondary pipeline 2	0.000
2	1	TN-01619	Elbow	Secondary pipeline 2	0.000
2	1	TN-02225	Straight pipe	Secondary pipeline 2	3895.000
2	1	TN-02225	Straight pipe	Secondary pipeline 2	1952.000

BOM without summary of pipe lengths



Level	Number	Article number	Designation	Pipeline	Length
1	1	Main pipeline		Main pipeline	0.000
2	2	TN-02317	T-Piece	Main pipeline	0.000
2	6	TN-01692	Elbow	Main pipeline	0.000
2	1	TN-02233	Straight pipe	Main pipeline	69932.000
1	1	Secondary pipeline 1		Secondary pipeline 1	0.000
2	3	TN-01619	Elbow	Secondary pipeline 1	0.000
2	1	TN-02225	Straight pipe	Secondary pipeline 1	23755.000
1	1	Secondary pipeline 2		Secondary pipeline 2	0.000
2	1	TN-01619	Elbow	Secondary pipeline 2	0.000
2	1	TN-02225	Straight pipe	Secondary pipeline 2	5847.000

BOM with summary of pipe lengths

Please note:

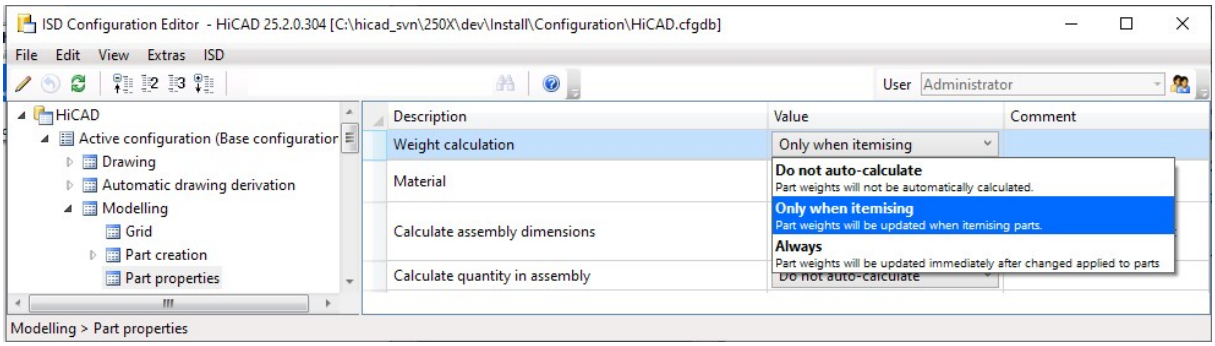
- Except for the length and weight of the pipe, the remaining pipe attributes of a single pipe in the summary are used. These are usually the attributes of the first pipe (in the part structure) of the combination.
- Pipes are combined according to their structure in the structure list. This may differ from the pipeline structure, for example, if pipelines are not marked as BOM-relevant.

Calculate weight of pipeline

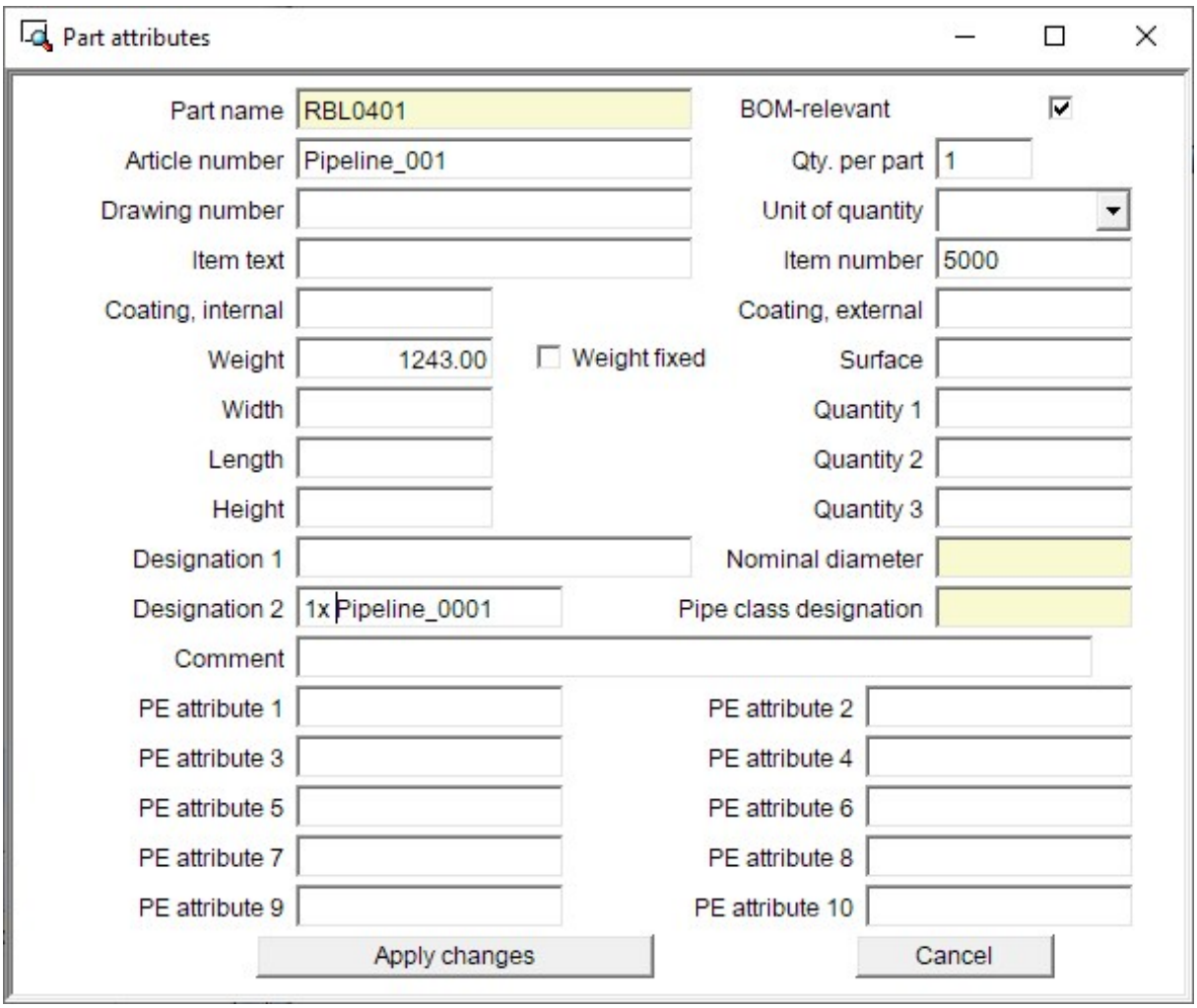
Weights of pipe parts of a pipeline are summed up from SP2 and transferred to the corresponding pipeline. For the calculation of the weights, pipelines are treated in the same way as assemblies. This means in particular

- Only weights of parts that are BOM-relevant are totalled.
- The weight of a pipe part is multiplied by the entry in field **Qty. per part**. An empty field is equivalent to the value 0.
- The pipeline must also be BOM-relevant for the weight to be entered.
- The weight of accessory sets is not taken into account.

When the weight calculation of assemblies and thus also of pipelines is performed can be changed in the Configuration Editor under **Modelling > Part properties > Weight calculation**. The ISD default setting is **Only when itemising**.



The attribute mask for pipelines has also been extended by the **Weight fixed** checkbox. If this is active, the entered weight is not overwritten during a weight calculation. The checkbox is inactive by default.

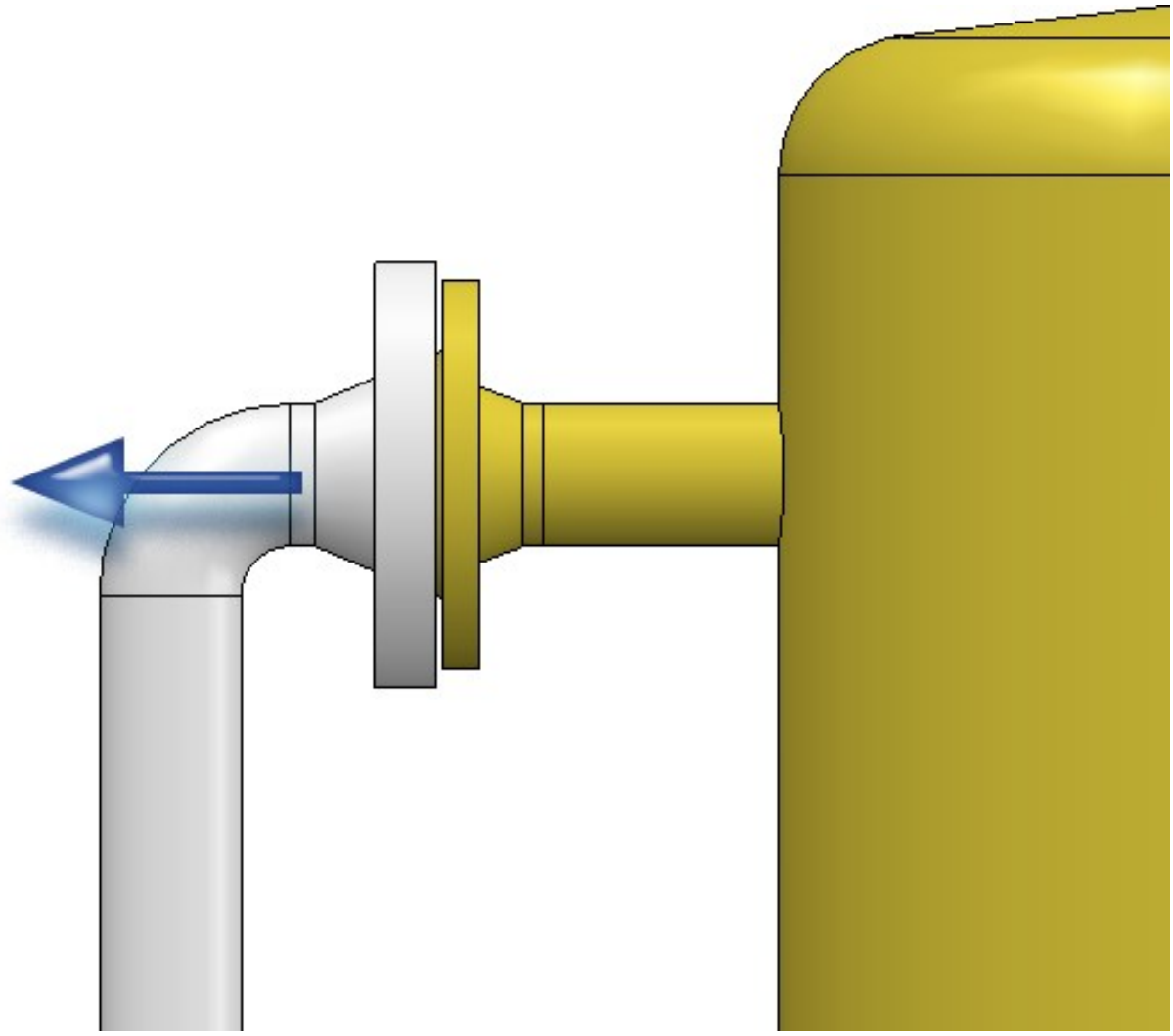


Dynamic route change

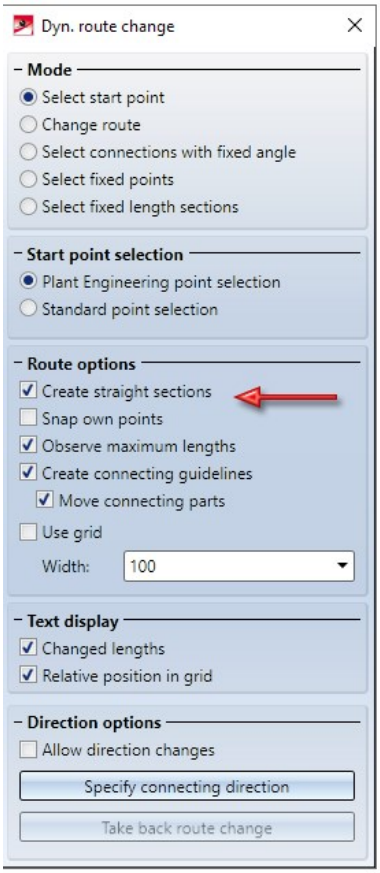
Create straight sections

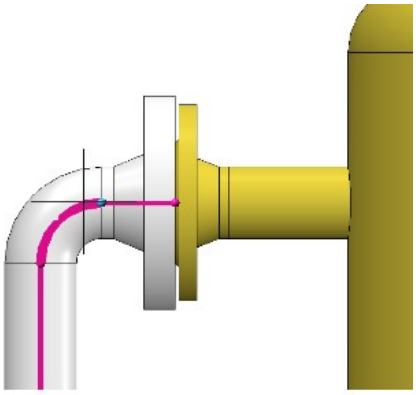
Previously, it was only possible to change the lengths of existing pipes with the dynamic route change. In many cases, however, it is desirable to be able to create additional straight sections by changing the route. This is possible from HiCAD 2502 on. For this purpose the dialogue of the function has been extended by the checkbox **Create straight sections**.

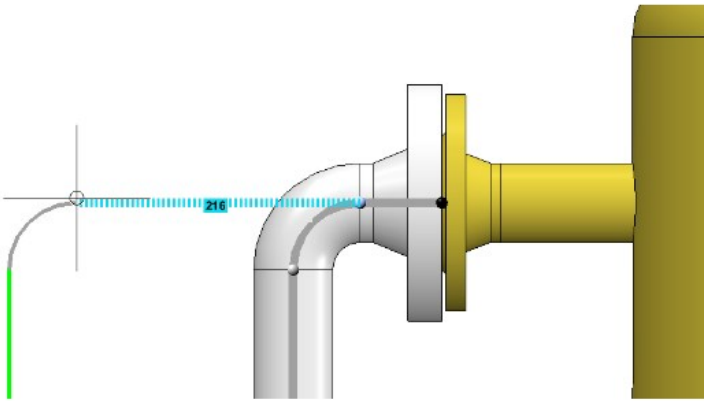
A typical use case is the increase of the distance between two parts, such as the illustrated elbow and the flange.

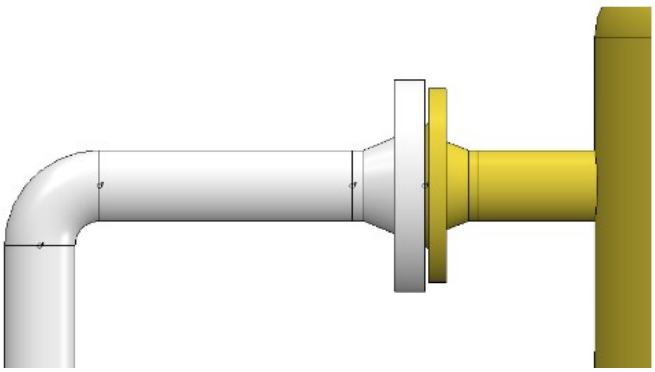


At this point you can reach your target as follows:



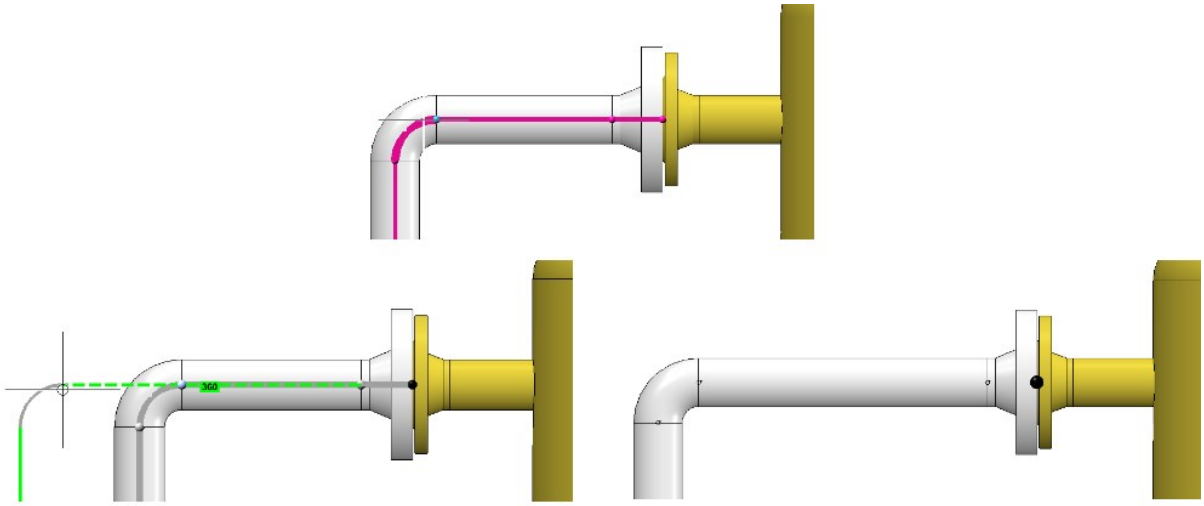




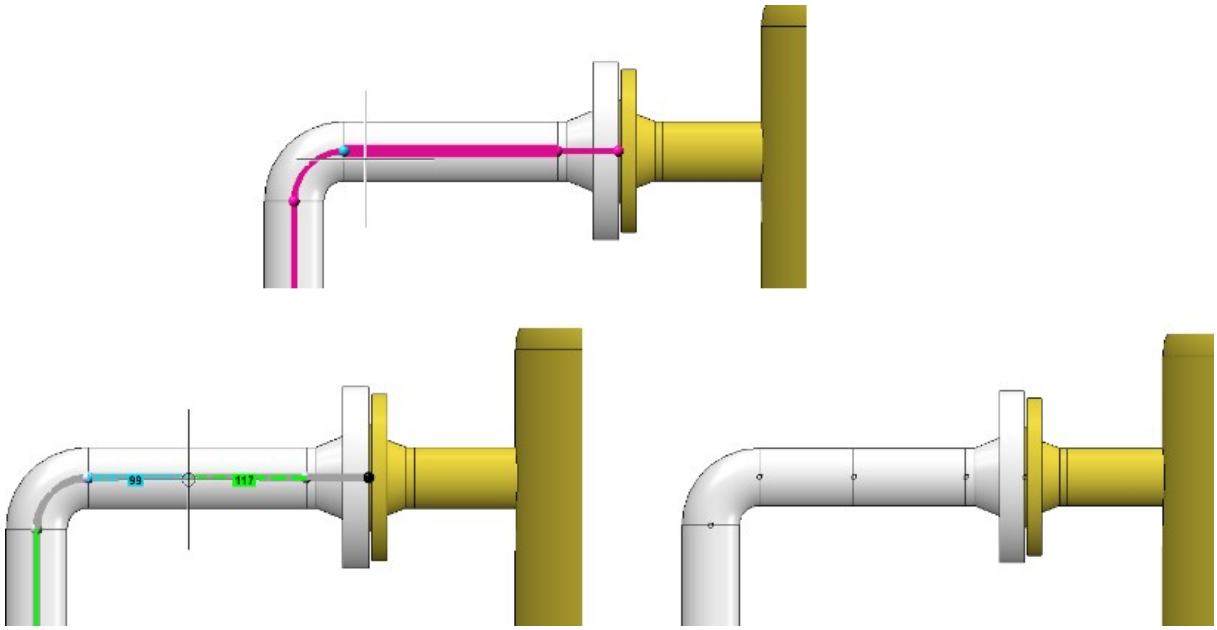


Simply pull the elbow away from the flange. A piece of straight pipe appears between the elbow and the flange. Please note that it is important to know on which side of a connection the cursor is located when selecting the connection.

We select the connection at the elbow once more. Note the different results depending on which side the connection is selected from.



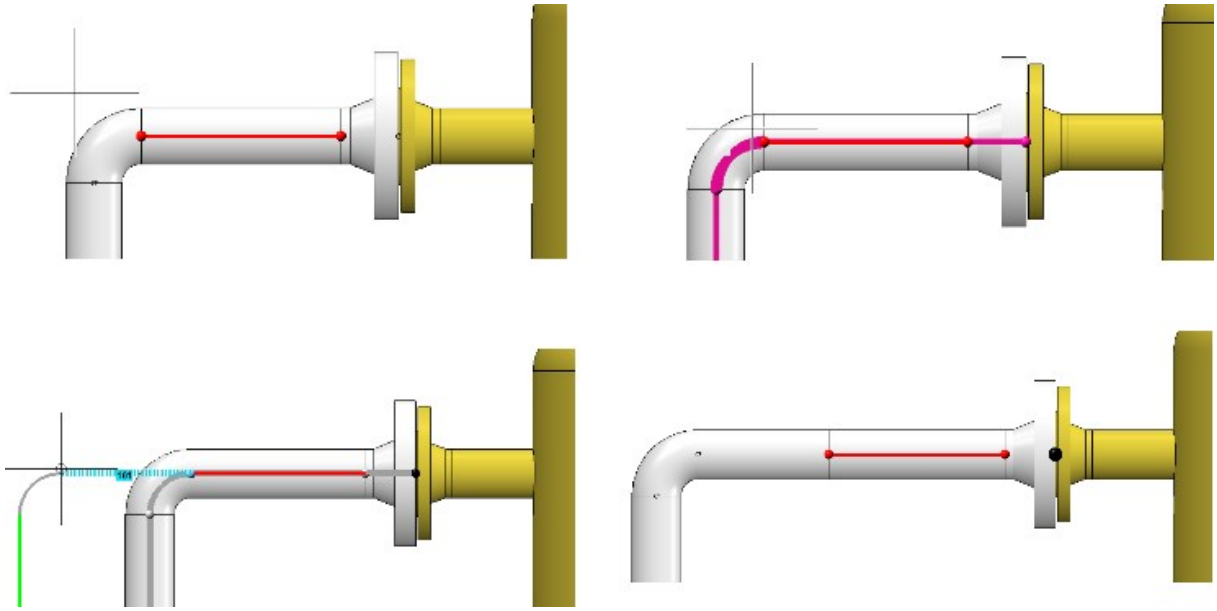
Elbow side



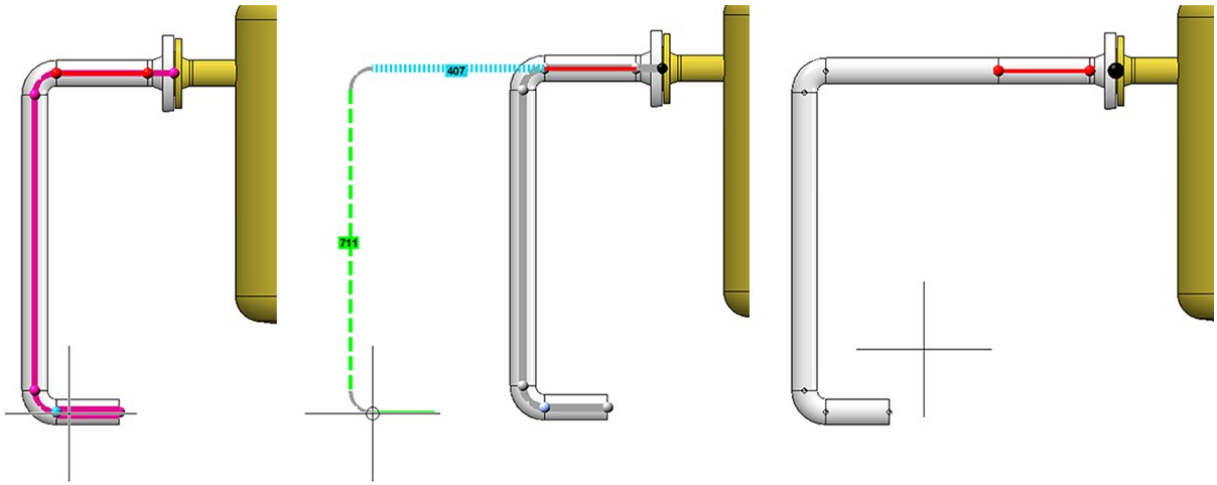
Pipe side

If you pull the elbow again, the previously created straight pipe will be extended. If, on the other hand, you drag the straight pipe, a new straight section is created between the straight pipe and the elbow.

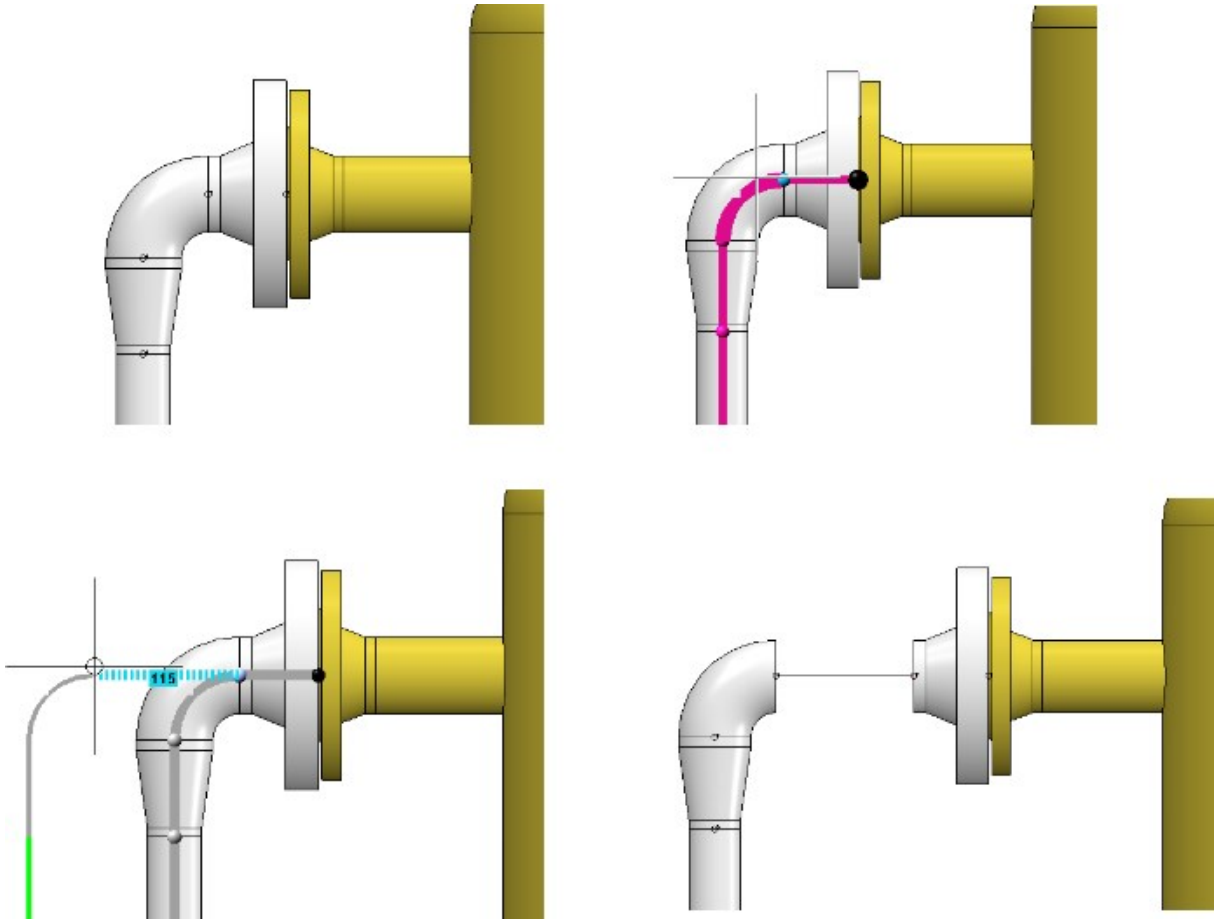
In general, dragging on a connection causes a new straight section to be created. Unless a straight pipe follows which may be changed in its length. Then its length is increased until its maximum length is reached. From then on a new straight section is created. If you had fixed the length of this straight pipe, the behaviour would be the same as that of a straight pipe whose maximum length has been reached, i.e:



If you push a connection, no new straight sections are created there. However, a new straight section can be created at another location if the target position would otherwise not be reachable, which was forced by the length-fixed section in the following:



The newly created straight pipe is created as a modified copy of an already existing straight pipe. The existing straight pipes within the same pipeline are used to determine the straight pipe to be copied. If one is found that matches the connection conditions, a length-adjusted copy of this pipe is installed. If such a pipe is not found, a leading edge pull section is inserted instead. In the following example a straight pipe with a nominal diameter of 50 is required, but after the reduction only pipes with a nominal diameter of 32 follow:



Limitations:

- Straight sections can only be generated at welded connections. Plugged, screwed, socket-welded and flanged connections are not supported. Mitred welds and component connections are also not supported. The former can be found on segment bends, the latter on inserted pipes.
- Straight sections are created within a pipeline, not at the transition to components.
- A newly created straight section is only assigned a straight pipe if a suitable cylindrical pipe exists within the same part chain without further processing, which is then used as a copy template. Pipes with mitre cuts and inserted tubes are not valid copy templates.

Select target point

Until now, there were two methods of target point selection in the route change:

- Plant Engineering point selection
  Automatic recognition of the connecting points and adoption of the connection direction for directional adjustments
- Standard point selection
  Common HiCAD point selection with all available point options - but without automatic recognition of the connection directions.

From SP2 on, there is only one point selection, which basically corresponds to the usual HiCAD point selection. However, points named here will also be checked to see if they are Plant Engineering connection points. For such points, the connection direction is still determined and the option **Snap own points** in the dialogue window of the route change is effective.

Another new feature is that the target point can always be selected with the left mouse button.

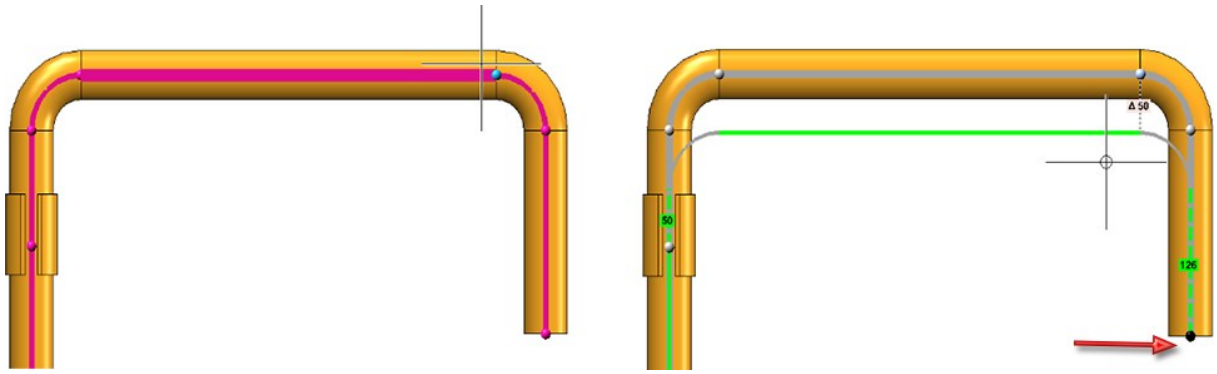
Select fixed point

The pre-selection of fixed points in the route change has been revised. It now depends on the side of the connection where the cursor is located when selecting the point to be moved.

A common application of the route change is to move the end of a pipeline without directly grabbing the end point. Until now, in such a case the free ends of the pipeline were provided with fixed points, which had to be removed manually:

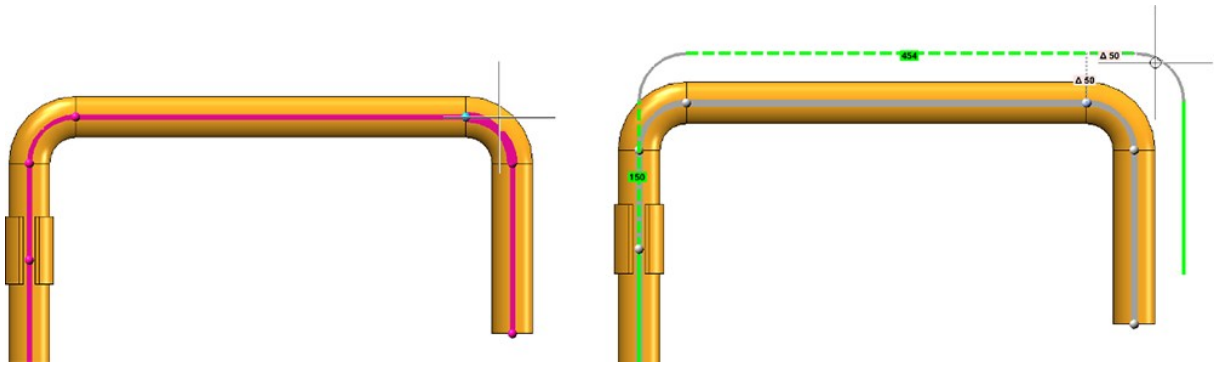
From SP2 on, no more fixed points are set at those free ends of the pipeline that follow on the side of the cursor.

Case 1: Selection of the start point on the pipe side.



A fixed point is displayed.

Case 2: Selection of the start point on the sheet side

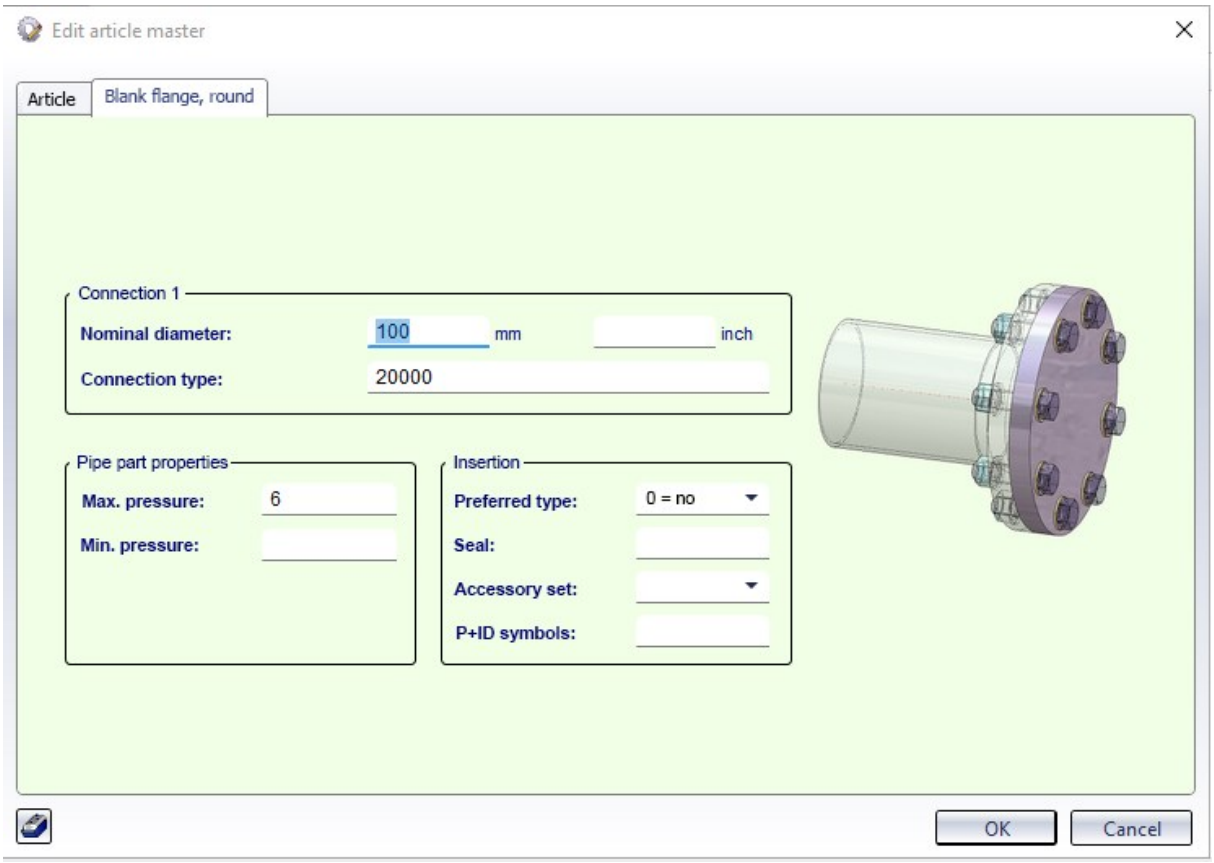


No fixed point is displayed (if there is a connected component at the end of the pipeline, a fixed point is of route created there).

Minimum pressure of pipe parts

Since HiCAD 2020 SP1, the pressure range has been a search criterion for all component types (except weld seam gaps). In this context, the HELiOS attributes and masks for the classification of components in Plant Engineering have now been adapted in such a way that a pressure range can be defined.

- The previous attribute DRUCK (Pressure) describes the maximum nominal pressure.
- The DRUCK_MIN (Min. pressure) attribute for the minimum pressure is new.



Service Pack 1 2020 (V 2501)

Part Tools

Modified Ribbons and Pull-down menus

The **Part Tools** function group now contains two new functions:

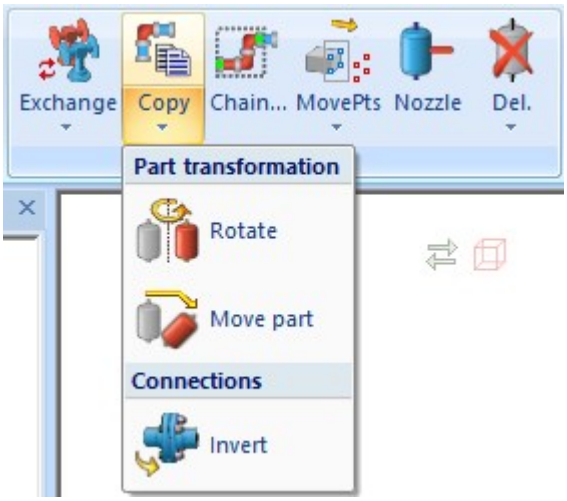
Copy pipe parts and

Select part chain .

In this context the functions

- Rotate part,
- Move part and
- Invert fastener or bolted flange connection

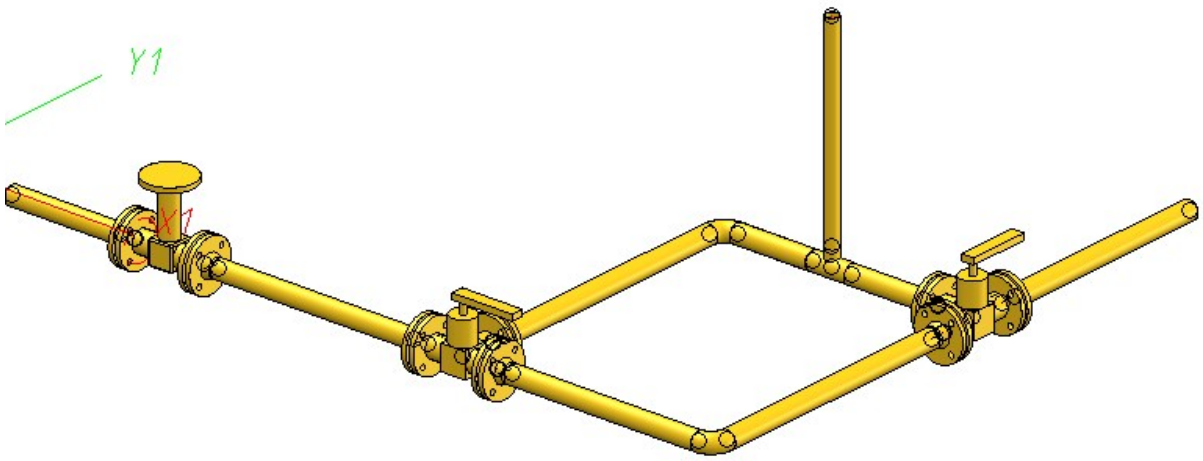
have been moved to the PullDown menu of the **Copy parts** function,; also the former **Move+Rotate part** function has been renamed to **Move part**.

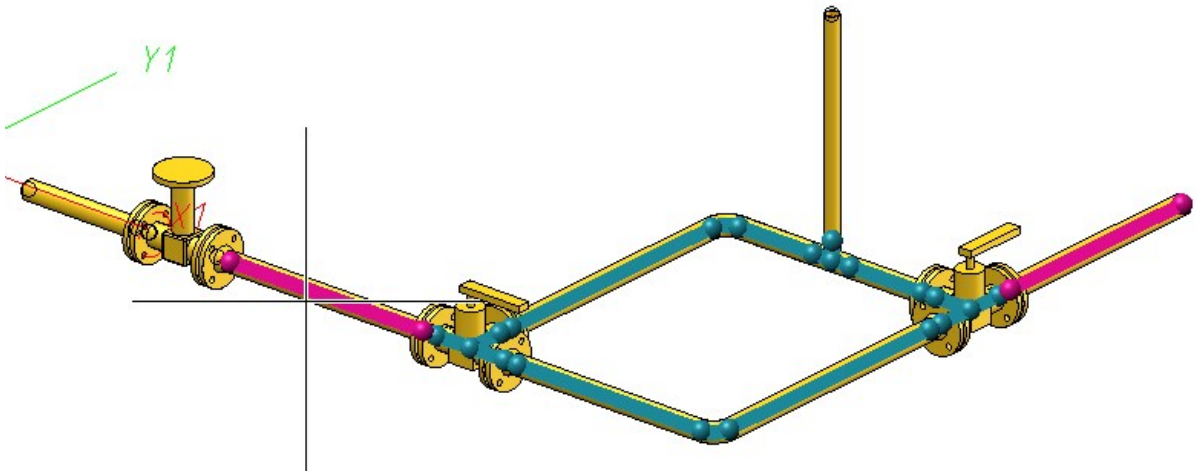


Select part chain

The new **Select part chain** function allows you to select complete part chains. To do this, you first select two pipe parts which serve as start and/or end points of the part chain. HiCAD then selects all parts which are located on a connection between the start and end of the chain.

A preview is displayed while selecting a chain. Start and end of the chain are displayed magenta (Special colour: Marking 1) and the additional parts of the chain are displayed blue (Special colour: Marking 3).





Copy parts

With the new **Copy parts** function you can copy a selection of pipe parts into a pipeline. Parts from several pipelines can be copied into a single target pipeline. Source and target pipeline can also be identical.

Before you call the function, you must select the parts to be copied - either in the ICN or in the model drawing. The part to be copied will then either be the active part or the active part list. You can also use the **Select part chain function** to select the parts.

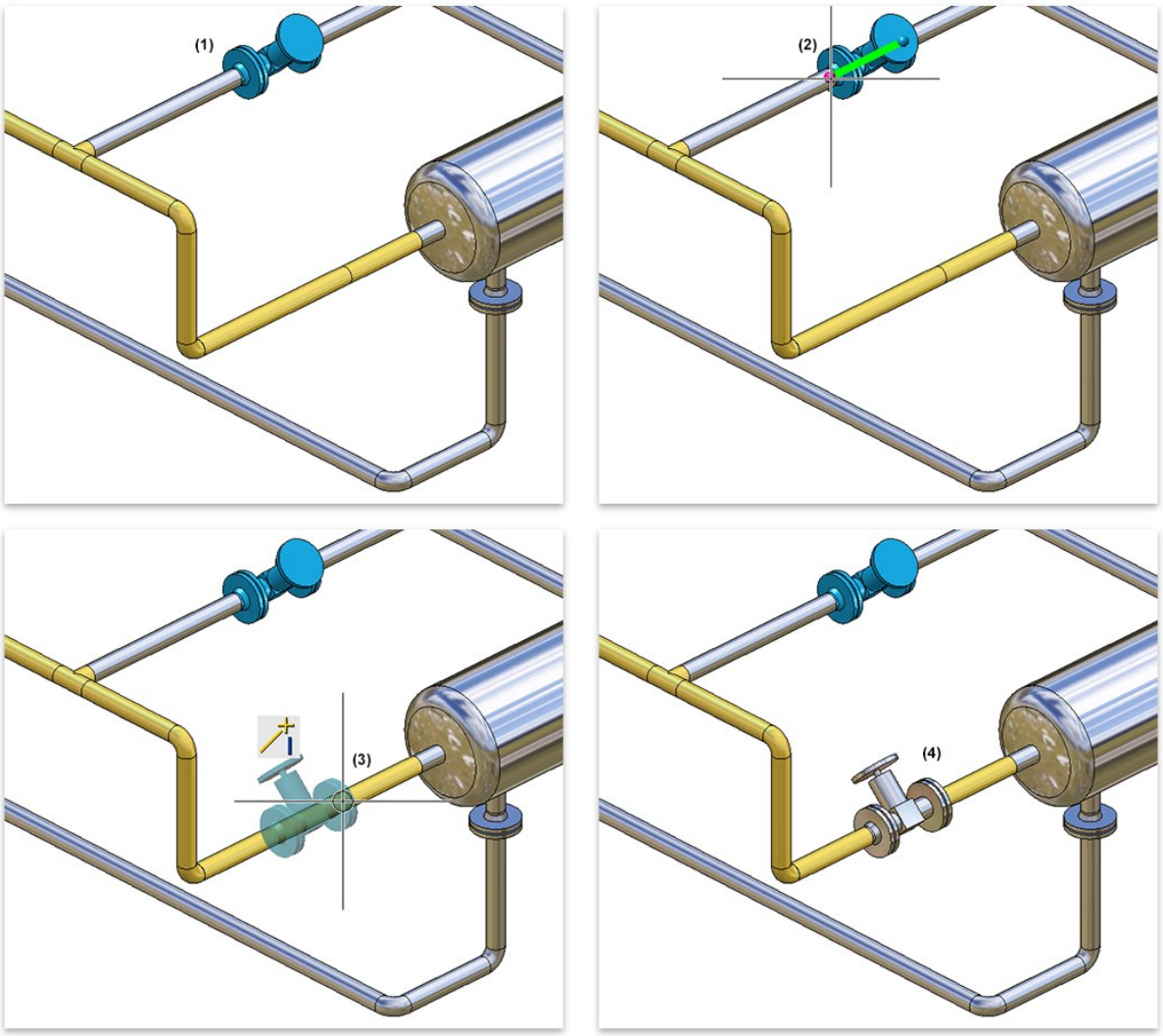
The copying of parts is done in several steps:

1. **Select handle**
   First you define the handle, i.e. the point via which the copy will be placed in the model drawing. You can select all free connecting points of the part list.
2. **Select target point**
   Here the position is selected where the parts to be copied are to be insterted. A preview of the parts to be copied is displayed at the current cursor position. Three different types of target points are possible.

   - Copy onto a connection point
   - Copy onto a guideline
   - Copy onto a free point

   After determining the target point, it may be necessary to perform a dynamic route change - for example, if a connection between two parts has been selected. HiCAD attempts to determine the target of the route change itself.
3. **Select target pipeline**
   From the selection in the second step, it is not always clear to which pipeline the parts are to be copied (e.g. when connecting to a component or copying between two pipelines). In this case the target pipeline must be selected.

In the example below, the highlighted blue parts (1) have been copied into the yellow pipeline.



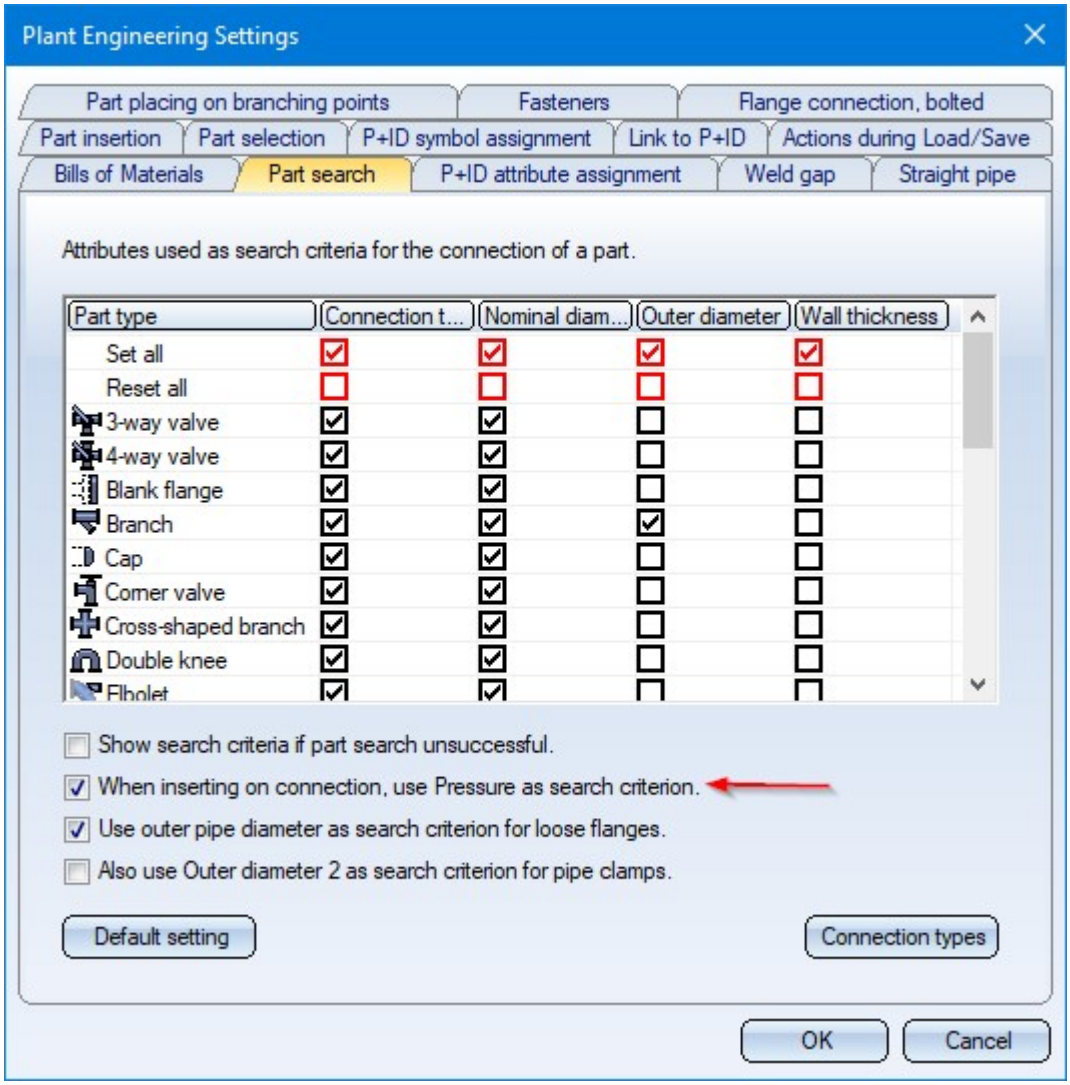
(1) Part selection, (2) Selected connecting point, (3) Target point determination, (4) Result

Plant Engineering Settings

Part search

Previously, a pressure range was only used as a search criterion when searching for seals. As of HiCAD 2020 SP1, the Pressure range is a search criterion for all part types (except for weld seam gaps). On the **Part search** tab in the **Plant Engineering Settings** dialogue window the checkbox **Use Pressure as search criterion for flange connections** has therefore been renamed to **When inserting on connection, use Pressure as search criterion**.

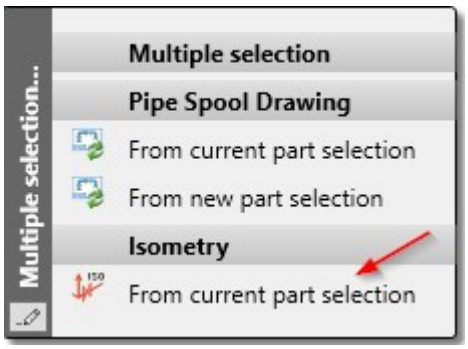
If a part is to be connected to another part, then - if this option has been set and a value for the **Pressure** attribute has been assigned to the other part - this pressure is used as search criterion when searching for the part to be connected.



Isometry and Pipe plan

Multiple selection when starting the isometry

If you have selected several pipe parts or pipelines in the model drawing, then the isometry can also be started from HiCAD 2020 SP1 via the **Multiple selection** context menu. To do this, right-click after your multiple selection and choose **Isometry - From current part selection** in the context menu.

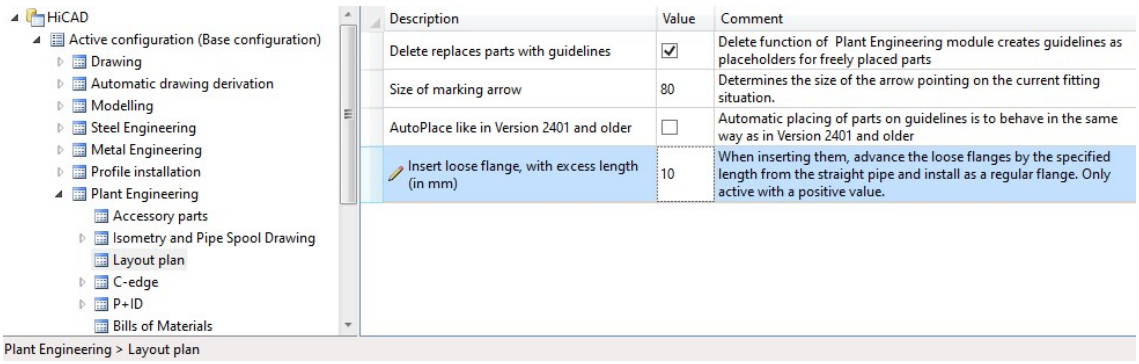


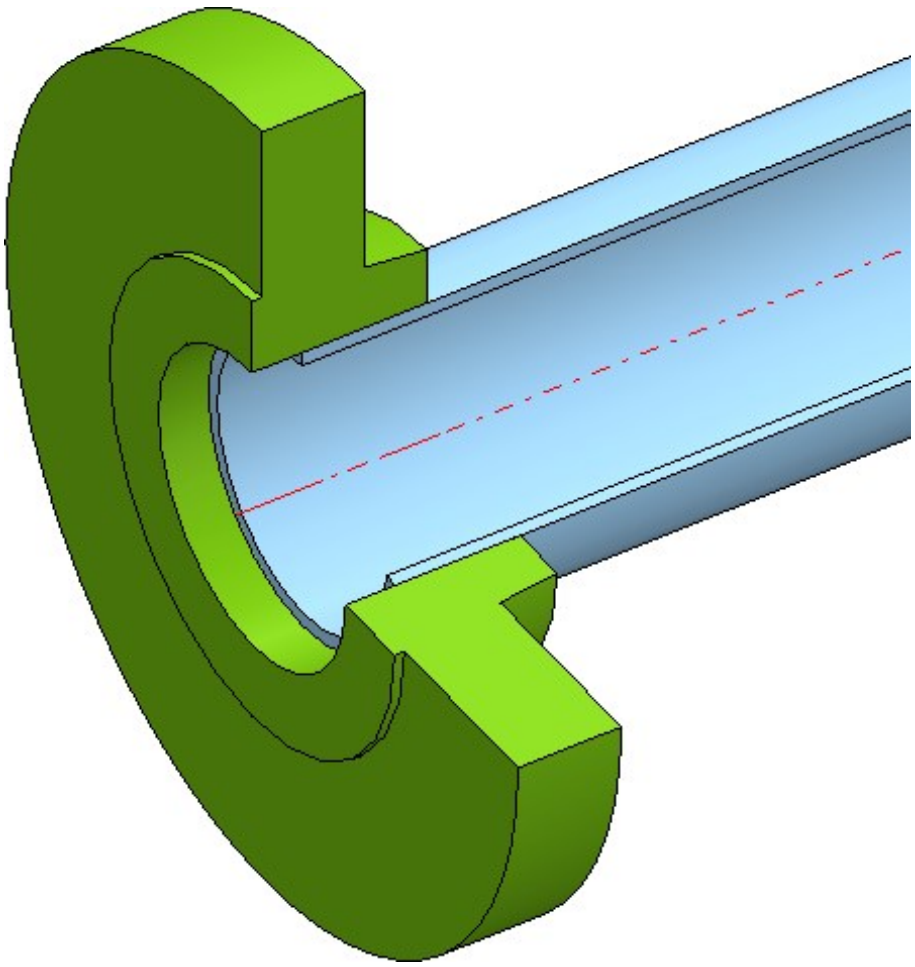
In the **Plant Engineering Isometry** dialogue window, the checkboxes of all pipelines belonging to the selection will then be active or all pipelines to which the parts of the multiple selection belong.

Parts

Insert loose flange as regular flange

Loose flanges can be inserted as regular flanges from SP1 onwards. The loose flange is pushed down from the straight pipe by a fixed excess length. This value can be set in the Configuration Editor at **Plant Engineering > Layout plan > Insert loose flange** as with excess length.





If you enter a value of 0 or a negative value, loose flanges will be installed as usual. If a positive value is entered, then each loose flange is installed like a regular flange.

In particular, when connecting to a straight pipe, it is not necessary to connect with the first point as usual for loose flanges, but with the second point.

The default setting is 0.

Insert placeholder

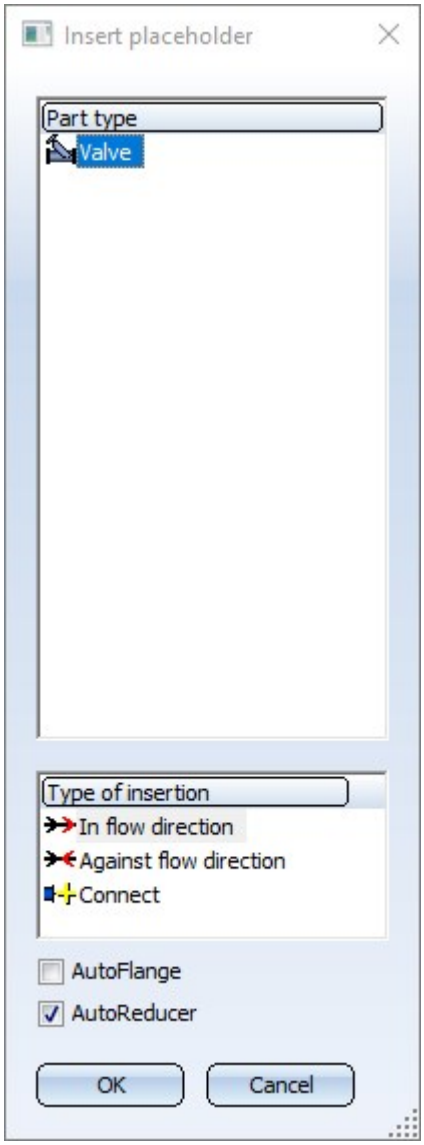
As of HiCAD 2020 SP1 it is possible to insert placeholder valves in a pipeline. The new function

Insert placeholder

has been made available for this purpose.

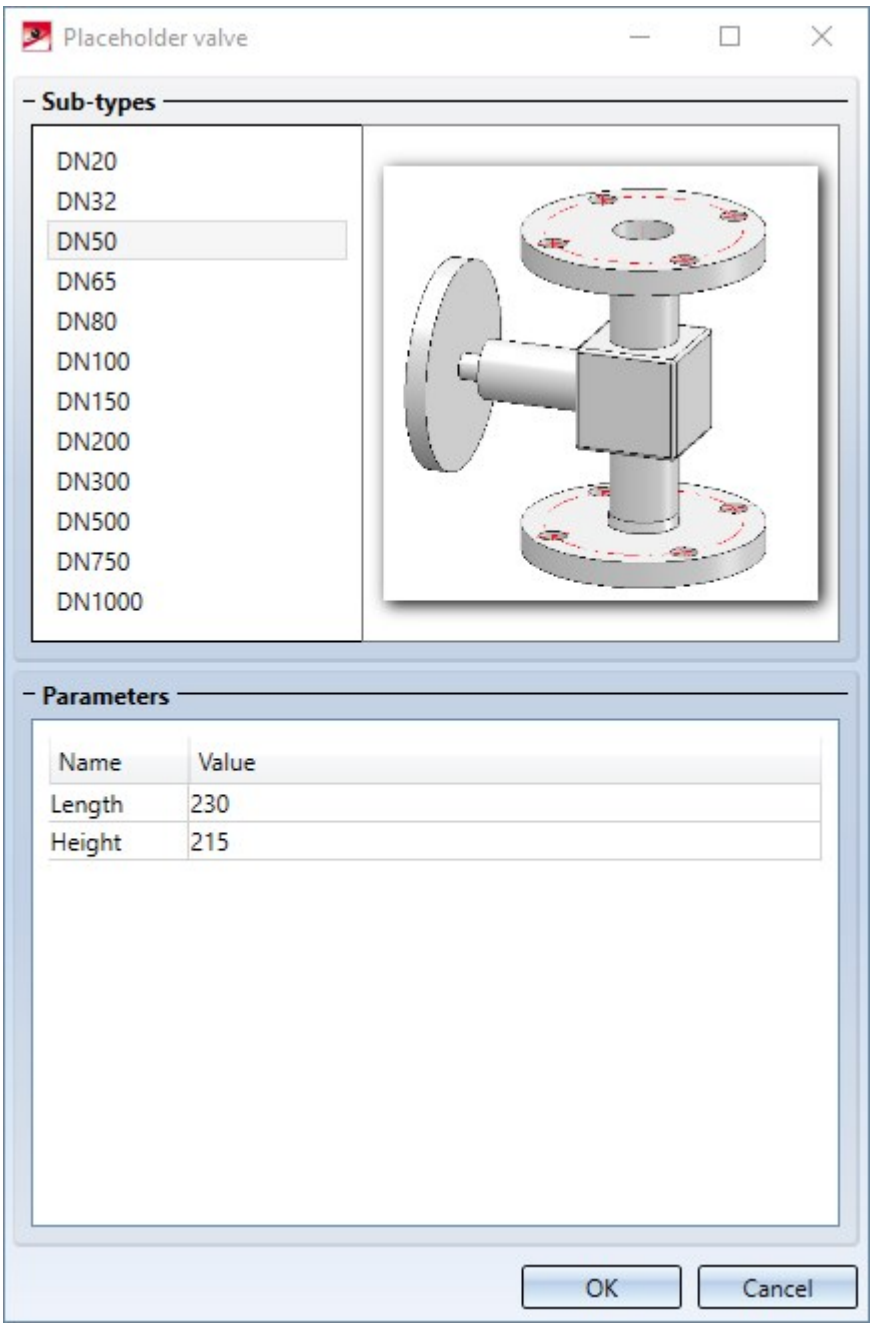
is available for this purpose.

In this way, useful valves can be inserted in the planning process even if the actual valve has not yet been determined or is not yet available. The placeholders ensure that meaningful BOMs and isometries can also be created in the planning phase.



The insertion of a placeholder valve is identical to the insertion of a regular Valve. The only difference is that the part is not selected via HELiOS or the HiCAD catalogue. Instead, a dialogue window is displayed in which various nominal diameters can be selected under **Sub-types**.

After calling the function, the nominal diameter that best matches the nominal diameter of the selected pipeline is automatically selected here.

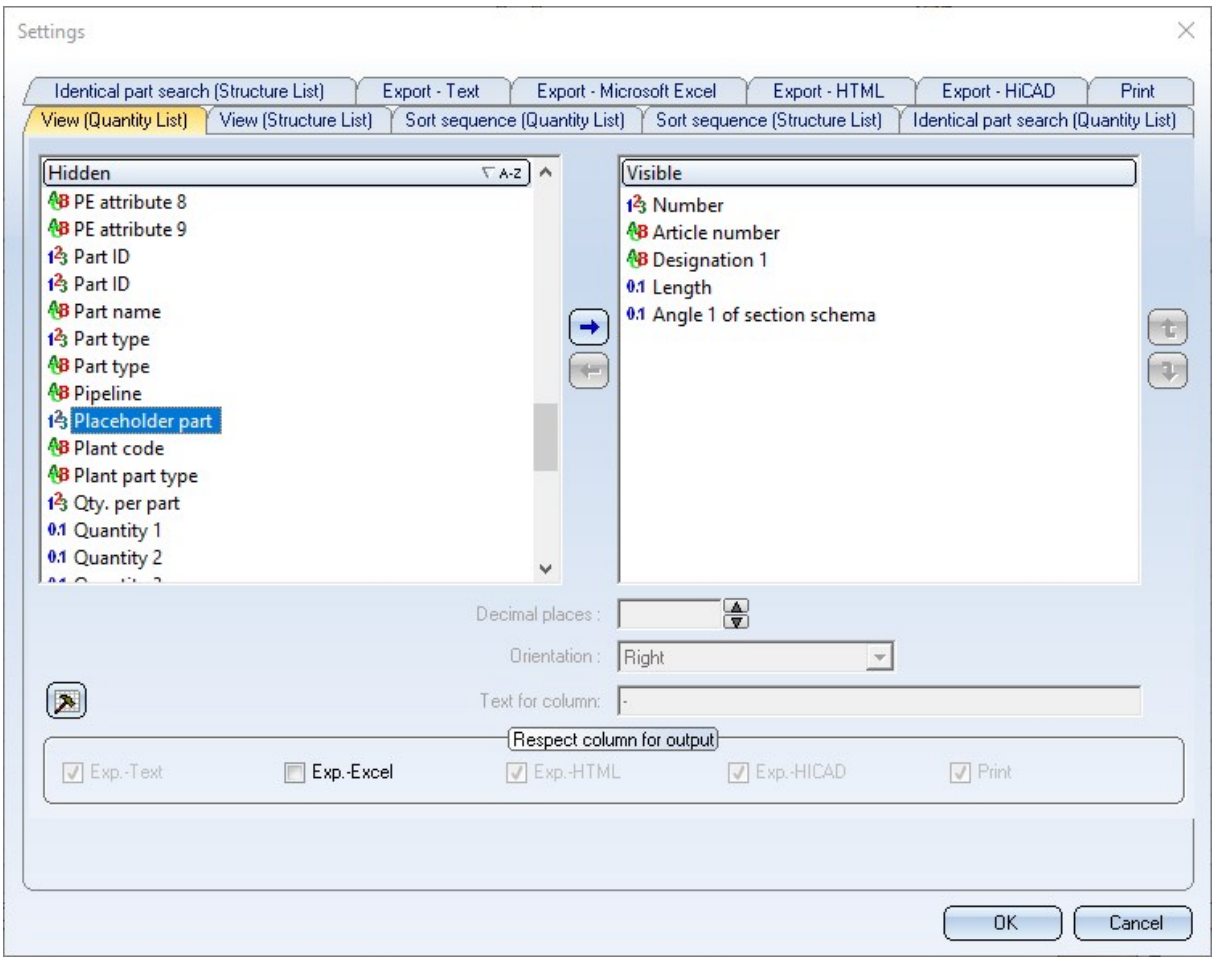


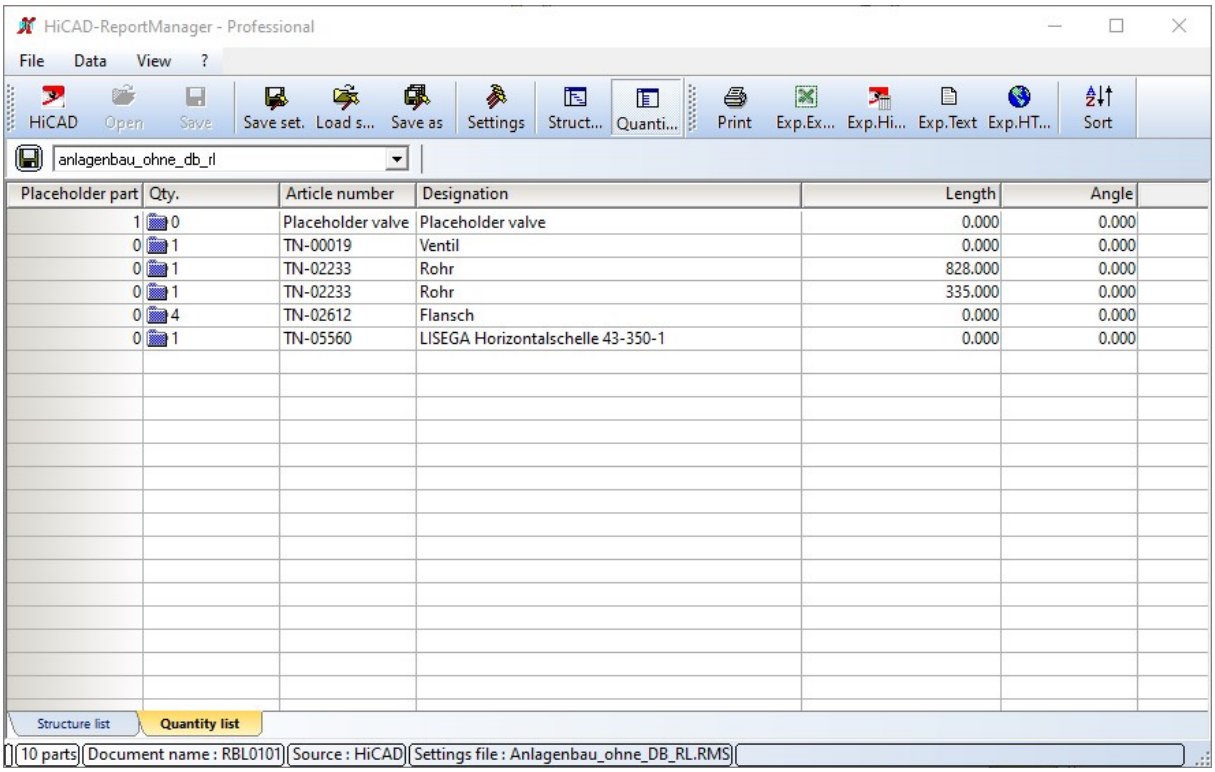
For each nominal diameter, the values for the length and height of the valve are preset in the **Parameters** area. These values can be adjusted individually.

Placeholder valves automatically receive the article number **Placeholder valve** and the corresponding names.

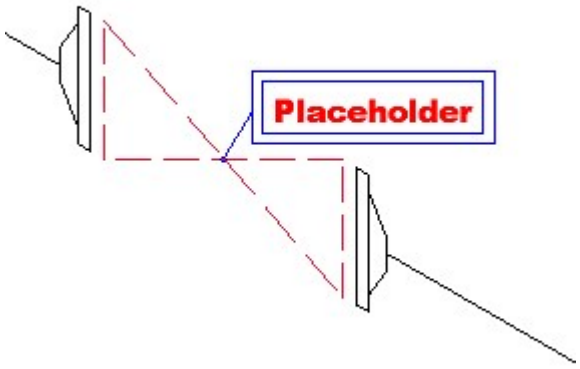
To make it clear in BOMs which valves are only placeholders, the BOM attribute **Placeholder part** is available for Plant Engineering BOMs. If this attribute has the value 1, then the part is a placeholder.

In the example shown below, the Quantity List in the Report Manager has been extended by the column **Placeholder part**.

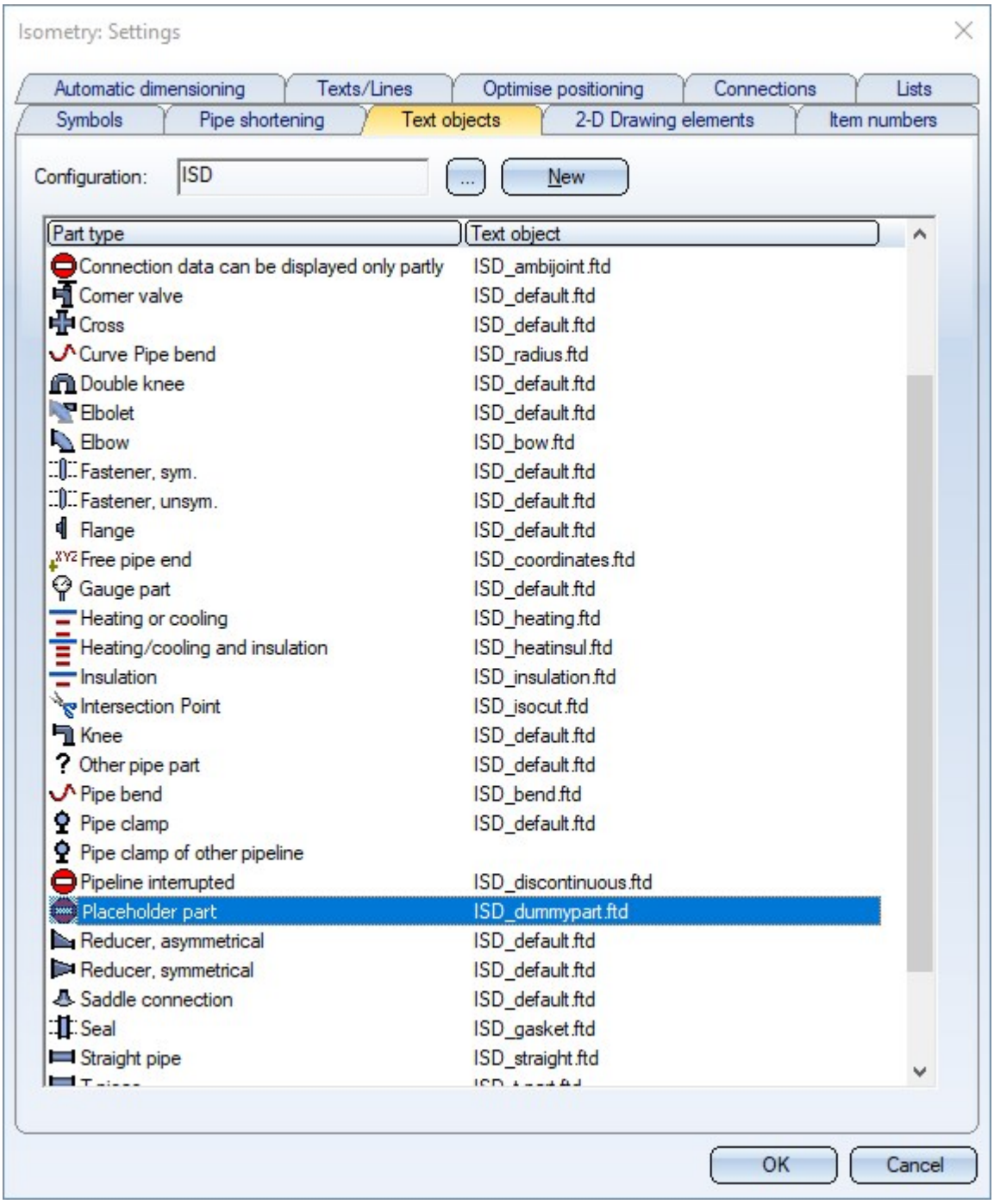




In the isometry, placeholder valves are shown as red dashed lines and have a designation tag with the text **Placeholder**.



The designation tag can be customized in the isometric settings on the **Text objects** tab if desired.

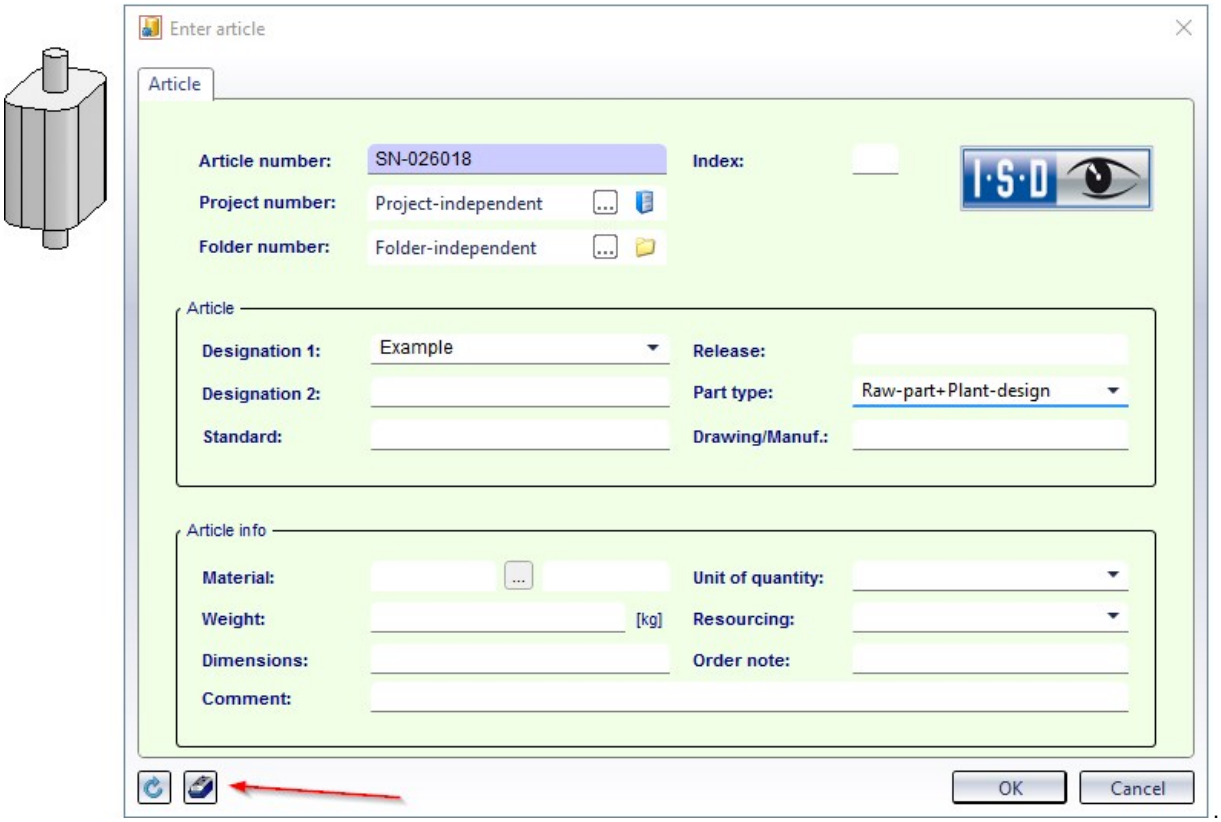


Directly insert classified .KRA files

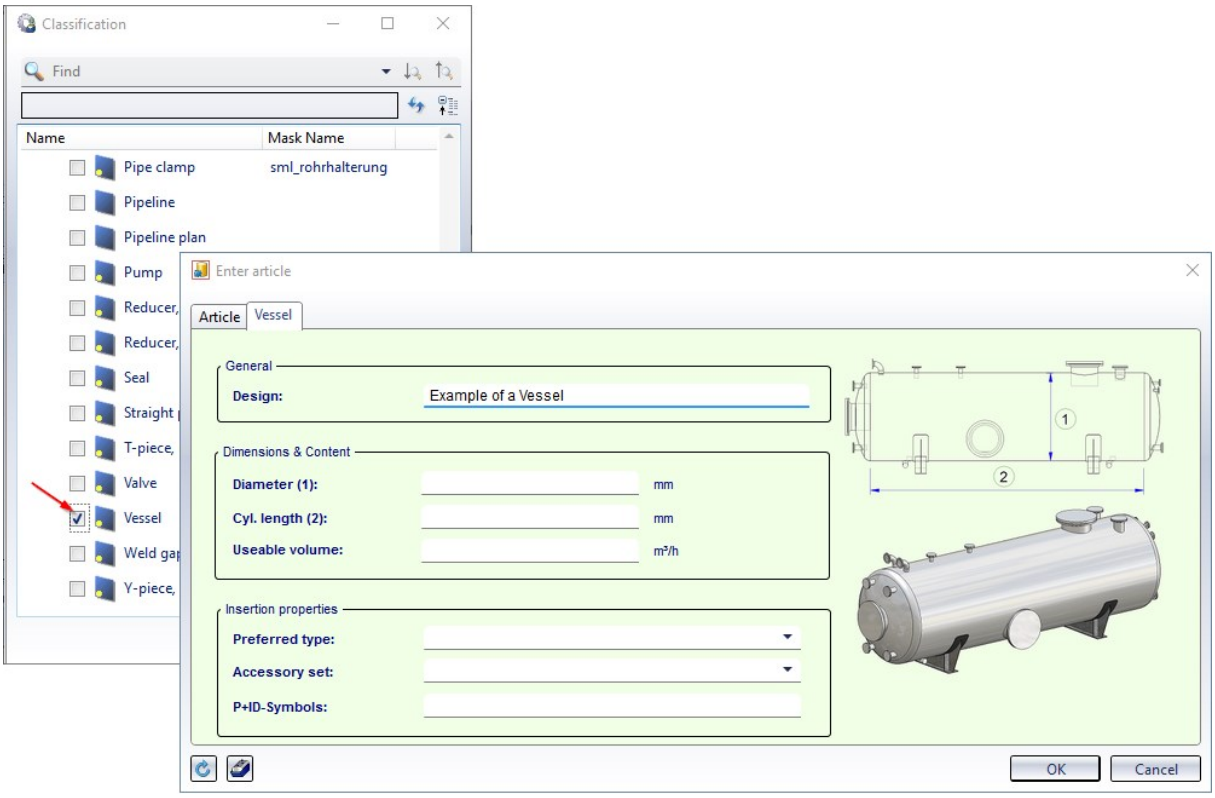
When components are installed, KRA files that are classified as components in HELiOS can now also be installed directly.

An example:

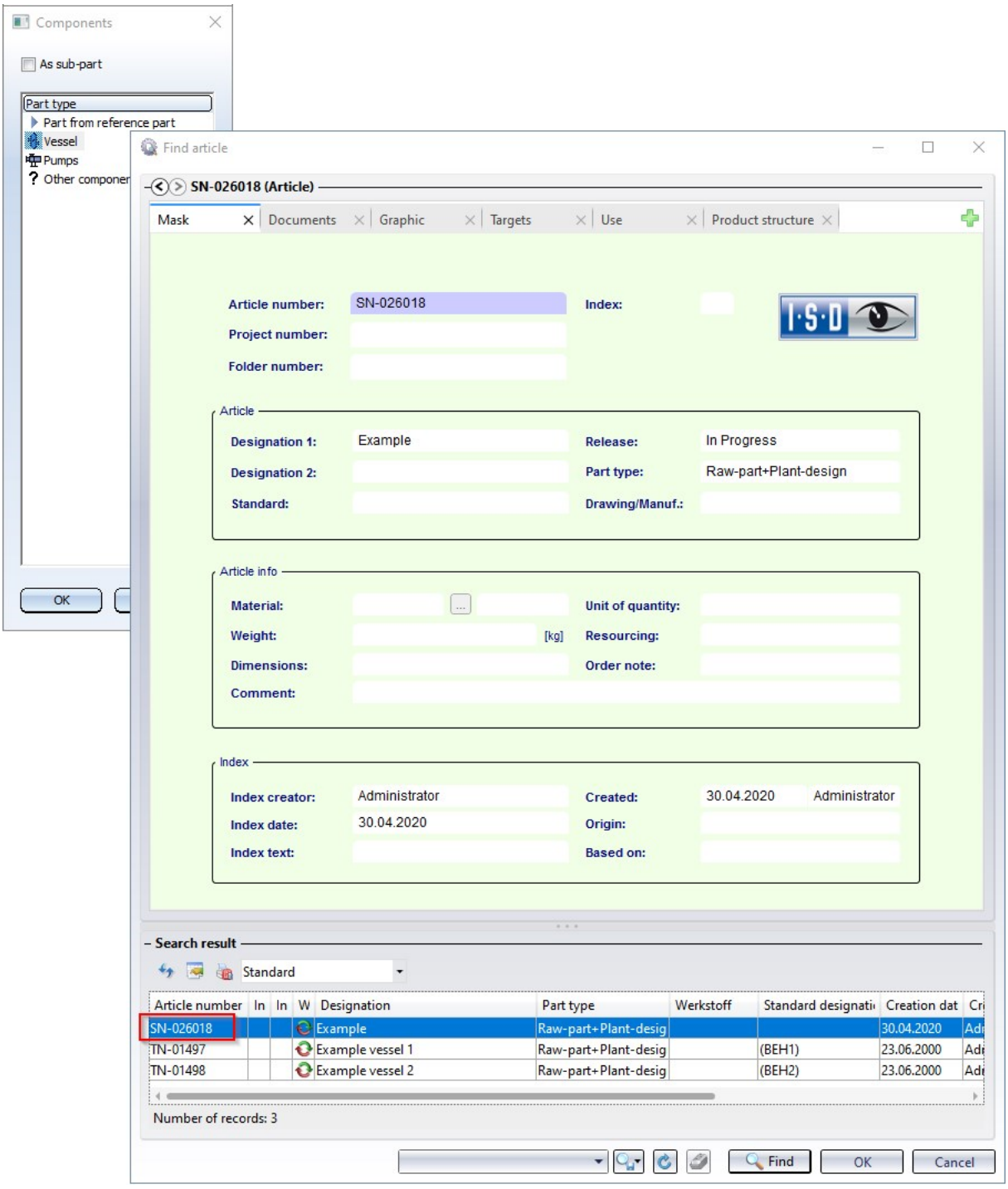
The displayed component is to be classified as a component of type **Vessel**. For this purpose, an article master is first assigned and then - before leaving the dialogue window - the button **Classification** is clicked.



After selecting the component type, the vessel data can be specified.



If the entry **Vessel** is now selected when installing a component, the KRA file saved above is available.

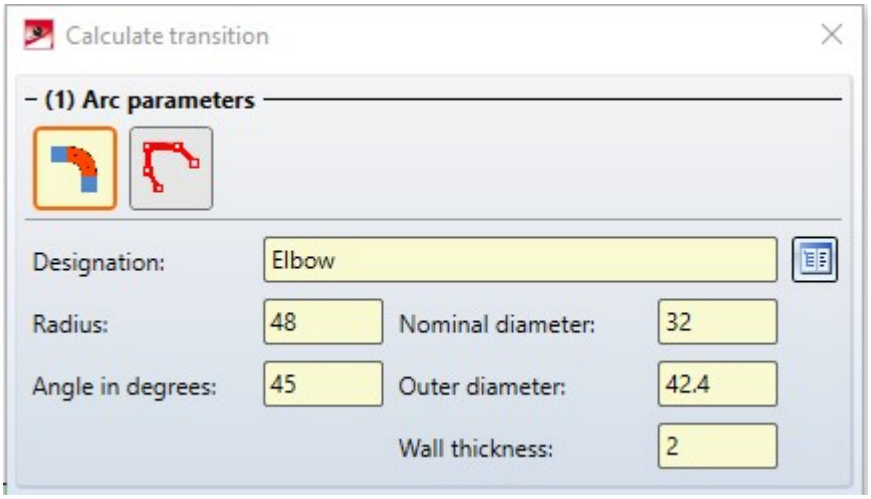


Pipeline Tools

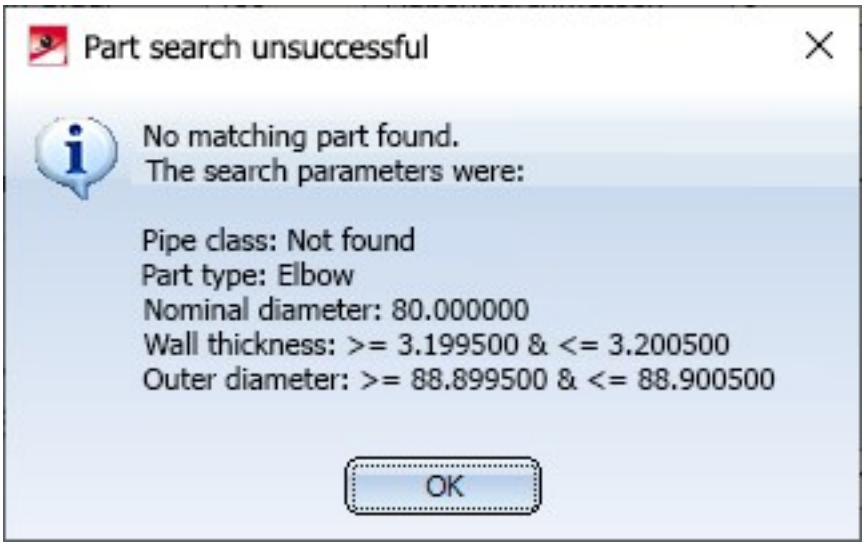
Calculate transition: Improved dialogue

The dialogue of the **Calculate transition** function has been improved:

- Additional information about the selected elbows is now displayed: Nominal diameter, Outer diameter and Wall thickness. Furthermore, the **Designation** field has been enlarged.



- If the part search fails, a corresponding message is displayed. The part search uses as search parameters values from the selected connections, namely those that you have defined for Elbows on the **Part search** tab of the **Plant Engineering Settings** dialogue window. The pipe class is also taken into account. If no suitable part is found, a corresponding message is displayed:



Major Release 2020 (V 2500)

Restructured Ribbon and context menus

- The **AutoRoute guideline and AutoPlace parts** function can now be found in the pull-down menu of the **AutoPlace parts on guidelines** function.
- The **Set Plant Engineering ID** function has been moved to the pull-down menu of the **Plant Engineering Settings**.
- The previous functions for the editing of pipelines have been divided into 2 groups:
  - Functions that change the structure of pipelines can be found at **Plant Engineering > Pipeline Tools > Change** > **...** .
  - Functions that change the properties of pipelines can be found at **Plant Engineering > Pipeline Tools > Assign** >**...**.
- The functions of the **Check** menu can now be found in the pull-down menu of the **Collision check** function at **Plant Engineering > Pipeline Tools > Coll..** >**...**.

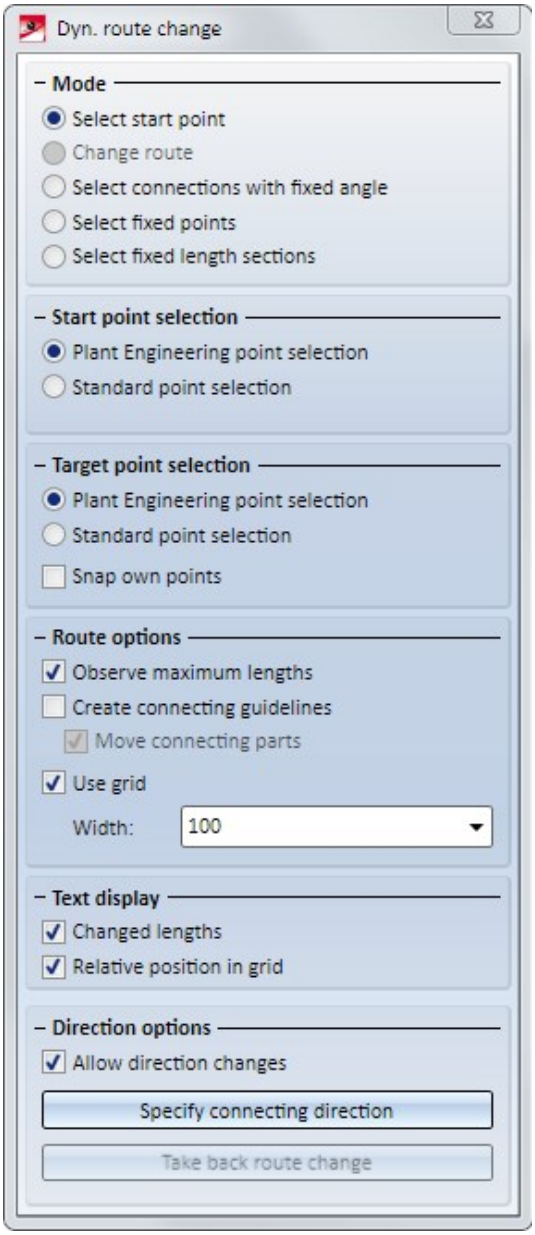
Pipelines

Activate pipeline

As of HiCAD 2020, the active pipeline is determined via the active part. This means: If the active part belongs to a pipeline, then this pipeline is considered the active pipeline. Otherwise, no pipeline is active. The previous functions for activating pipelines are therefore no longer available from HiCAD 2020 onwards.

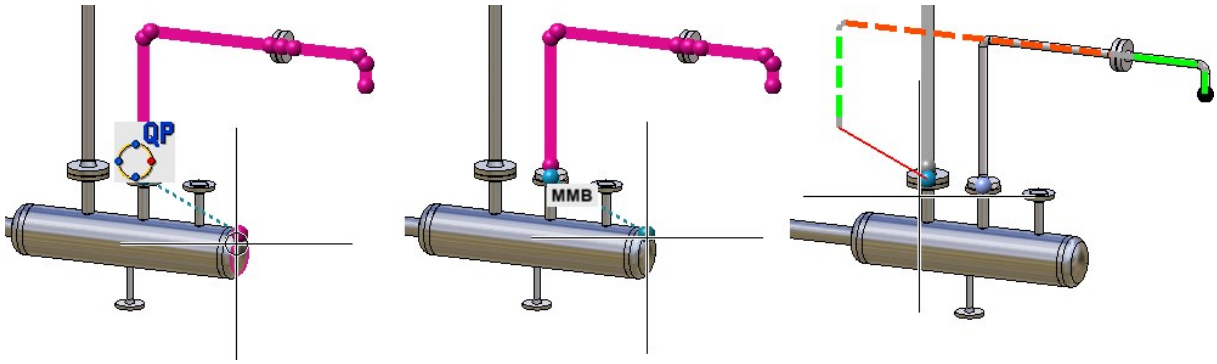
Change pipeline route

Previously, only Plant Engineering connecting points were permitted as start points for route changes. As of HiCAD 2020, any points can be selected, i.e. also points that were determined via point options.



- In the **Plant Engineering point selection** mode, the route change behaves as before.
- In the **Standard point selection** mode, you can select any point as the start point. In order to be able to clearly define the sections of the pipeline to be changed, the route change must nevertheless refer to a Plant Engineering connecting point. Therefore, HiCAD automatically suggests a Plant Engineering connecting near the selected point after the selection of the start point. This is indicated by with a **MMB** note.
  - If you want to use the suggested point, just click the middle mouse button.
  - Otherwise, simply select another Plant Engineering connecting point with a left-click.

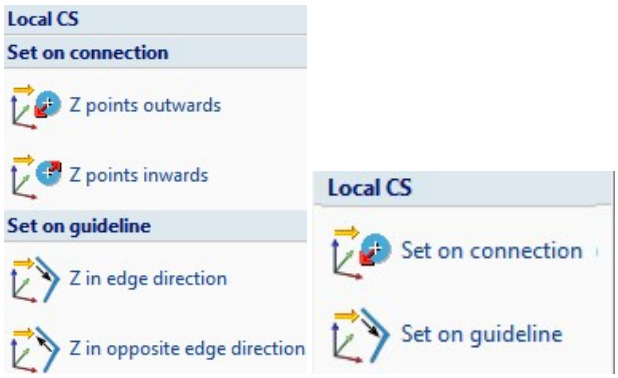
  During the route change, an auxiliary line is drawn between the selected start point and the Plant Engineering connecting point. This auxiliary line is displayed in red (Special colour X-axis) if the starting point cannot reach the target point.



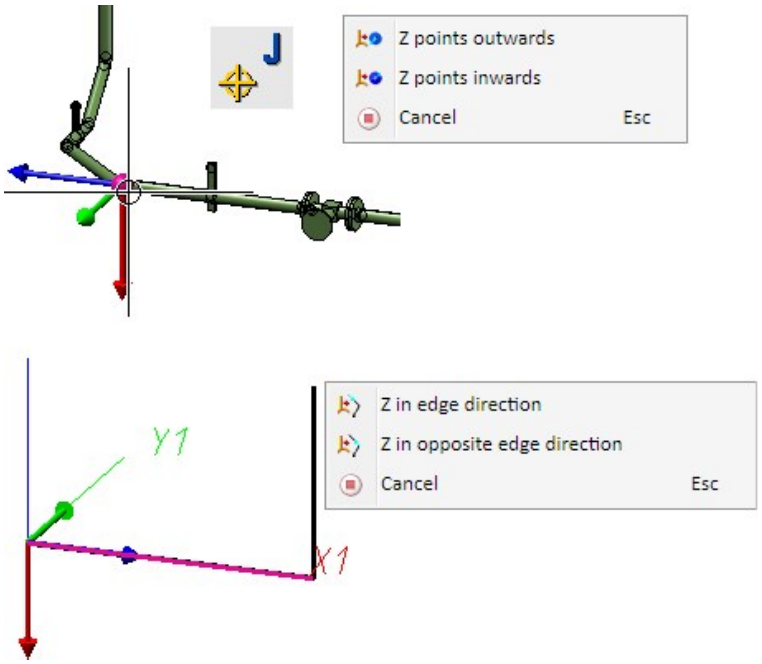
Settings

Set Local CS

The setting of a local coordinate system has been simplified in HiCAD 2020. The selection of the direction of the coordinate system no longer takes place when starting the function, but can be set, after a right-click, via the respective context menu. In addition, HiCAD now displays a preview of the coordinate system.



Left: Up to HiCAD 2019 Right: As of HiCAD 2020

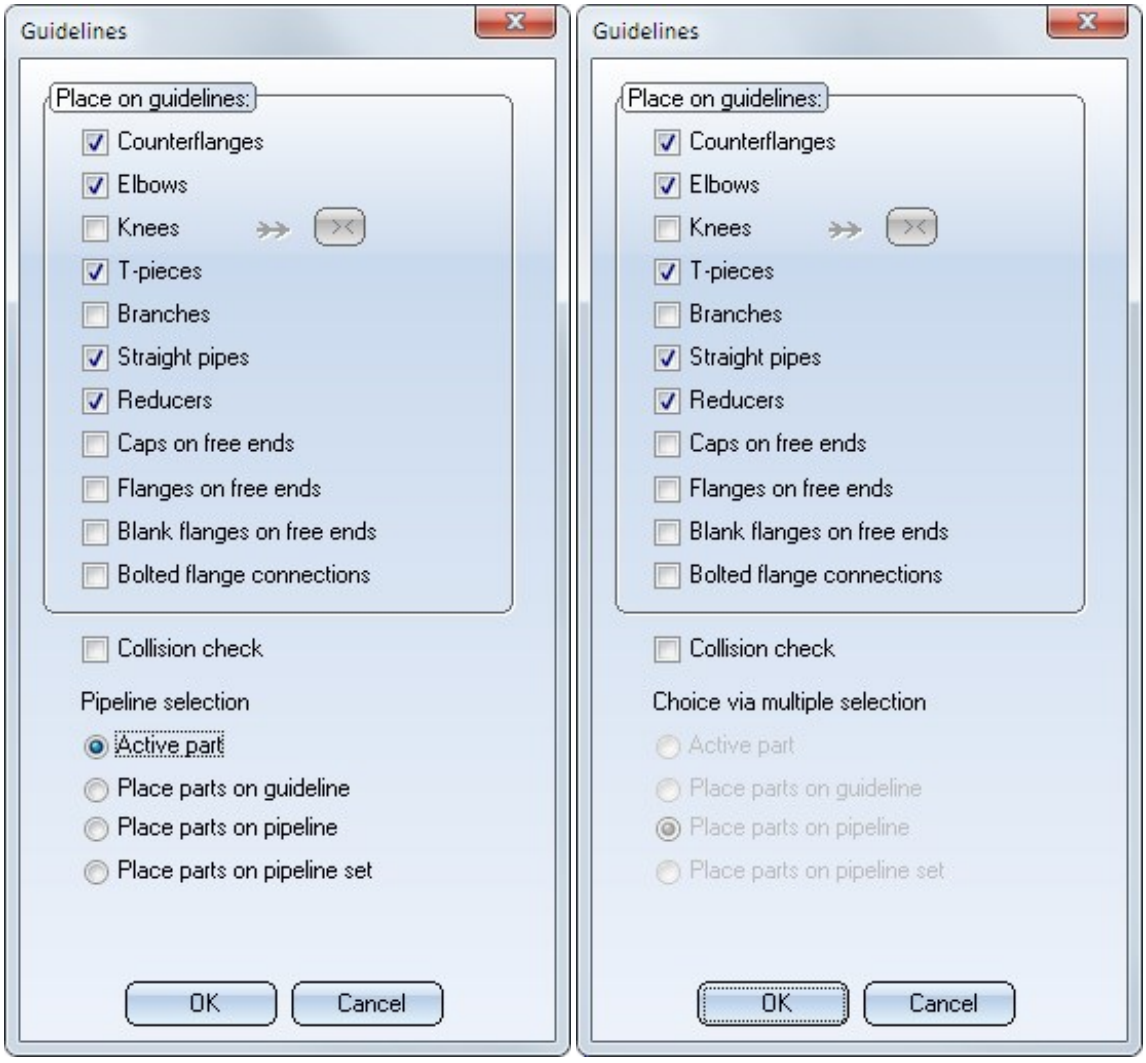


Preview and context menus

Guidelines

Pipeline selection for automatic placing of parts

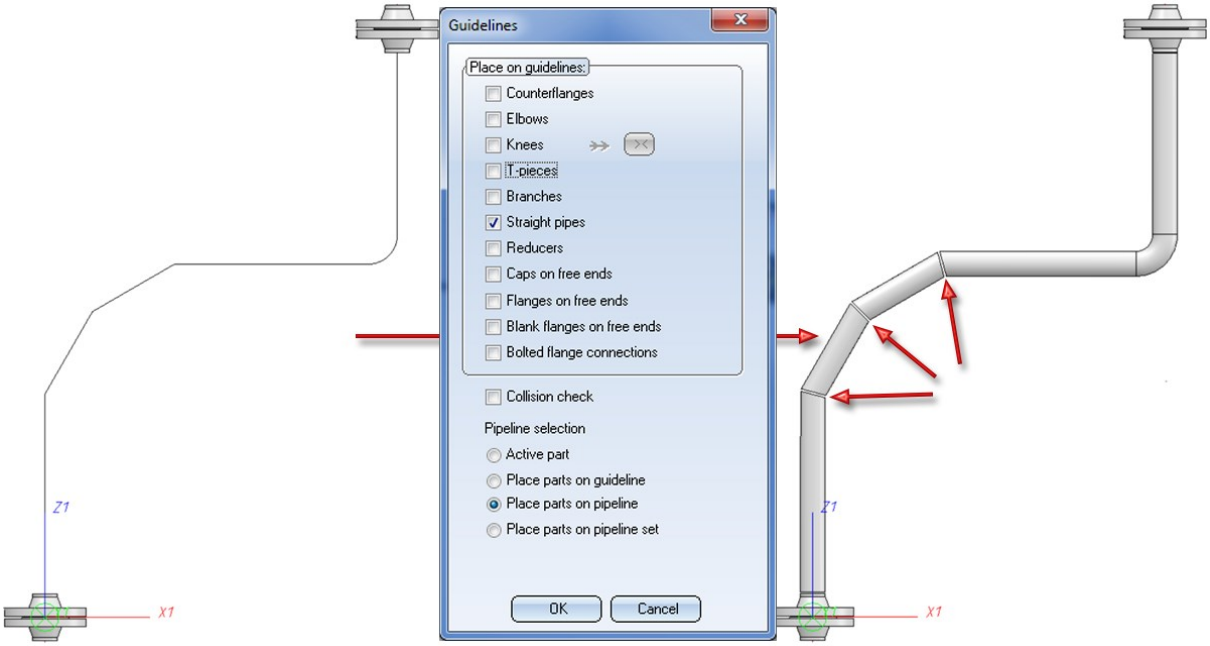
The selection of the active pipeline for the automatic placing of parts on guidelines has been revised. New here is the **Active part** option. Here the pipeline to be assigned is determined via the active part. If the active part is a guideline, parts will be placed only on this guideline. Otherwise the parts will be placed on the superordinate pipeline. If the active part has no superordinate pipeline, the dialogue behaves as with the **Place parts on pipeline** option.



In addition, the automatic part placing also supports multiple selections from HiCAD 2020 onwards. If several parts are selected, of which at least one is a sub-part of a pipeline, the options are greyed out and the elements to be assigned are determined via the multiple selection. Analogous to the selection via the active part, parts will be placed on individual guidelines. For each other selected part, parts will be placed on the respective superordinate pipeline.

Mitre cuts during automatic placing of parts

If you select neither elbows nor knees, but straight pipes, mitre cuts will be applied to the straight pipes:



Parts - New automatisms for parts

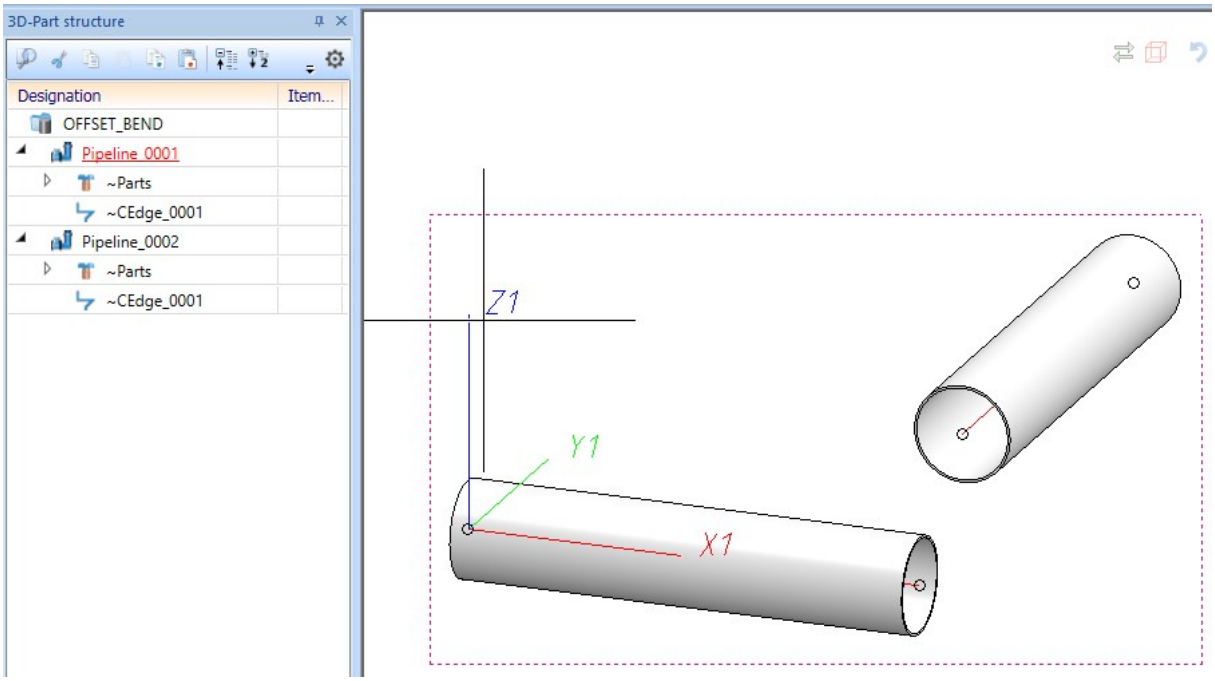
From HiCAD 2020 onwards, two new functions for part automatisms are available in the pull-down menu of the **Change** function (Plant Engineering > Pipeline Tools > Change > ...):

Calculate transitions and

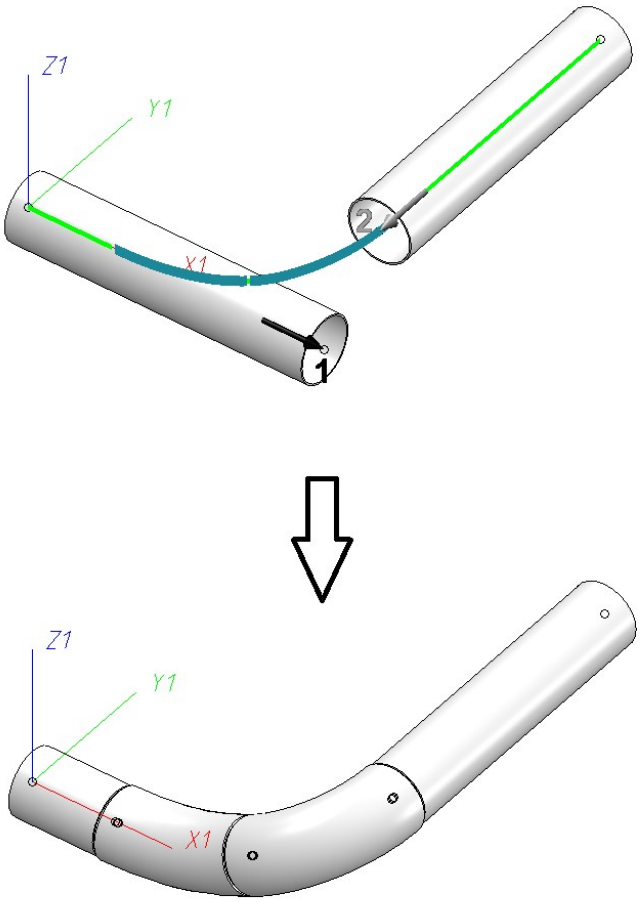
Edit arcs

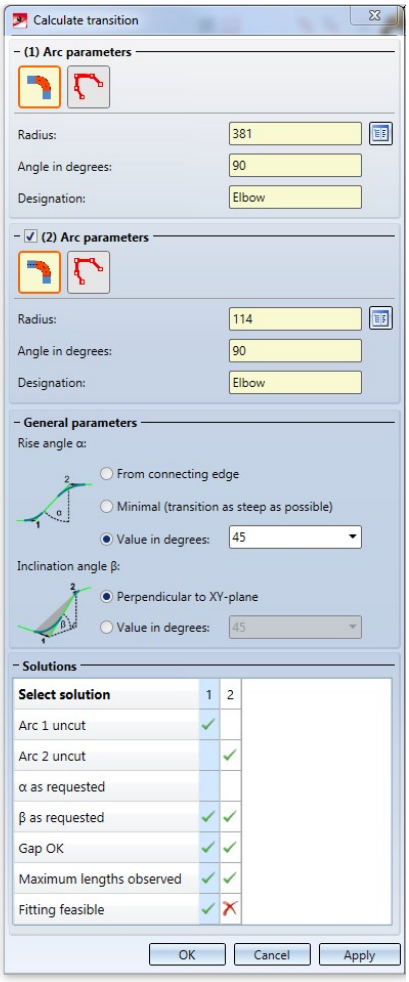
Calculate transitions

Until now, HiCAD did not provide a simple way to calculate and place arcs for offset transitions. The problem here consists in connecting two pipelines using suitably cut bends, as in the following example:



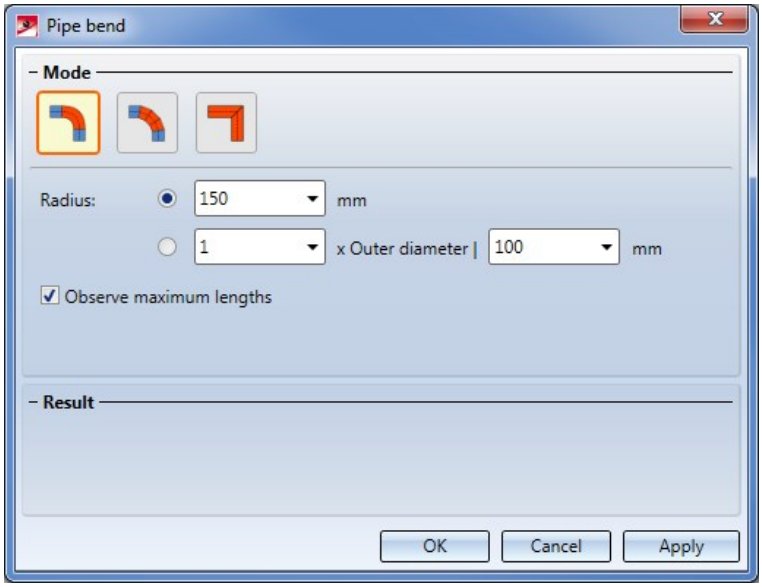
From HiCAD 2020 onwards, this problem can be solved with the new **Calculate transition** function:





Edit arcs for bent pipes

With this Editor you can modify arcs in Plant Engineering. You can change arcs for bent pipes, insert segment arcs and perform mitre cuts. The Editor requires a working with guidelines and can also modify unoccupied guideline edges.



Three modes are supported:

Transition by bending

Transition by segments

Transition by mitre cut

Delete counter-flanges

When deleting valves, counter-flanges in the same pipeline are now also deleted.

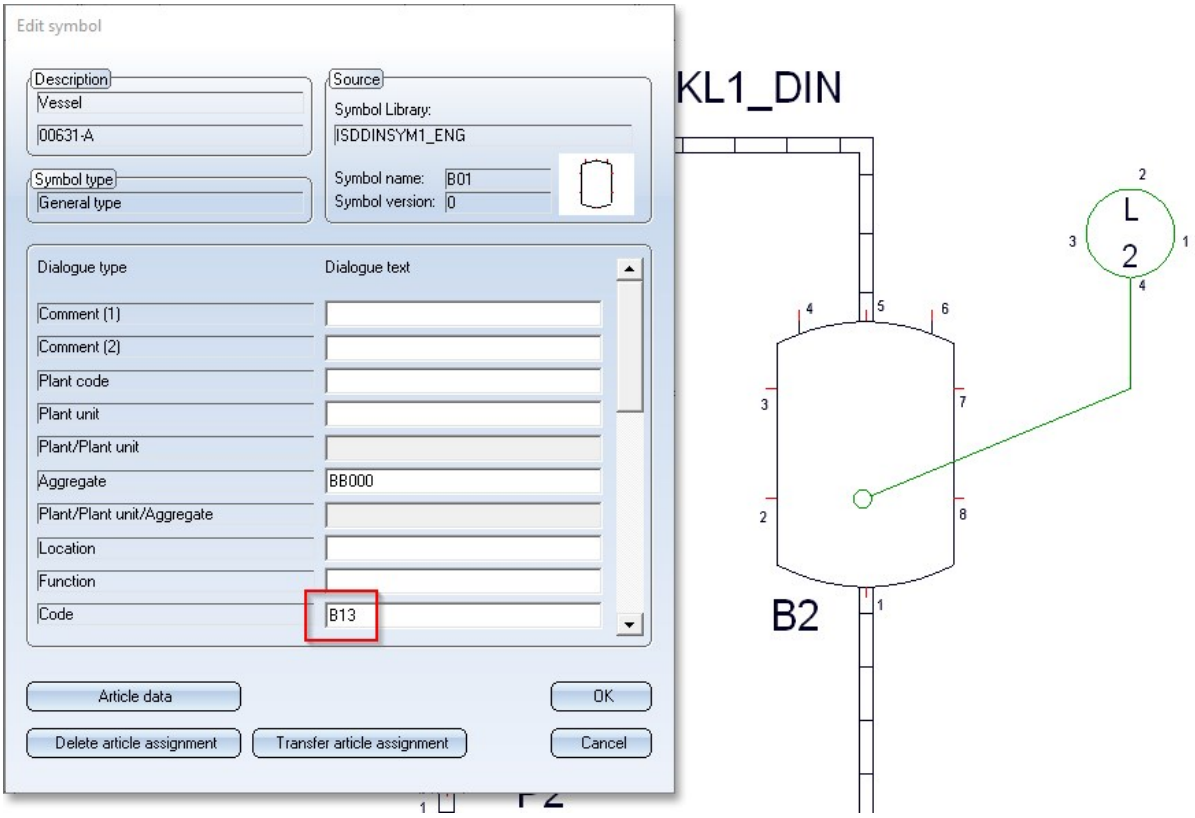
P+ID

Service Pack 1 2020 (V 2501)

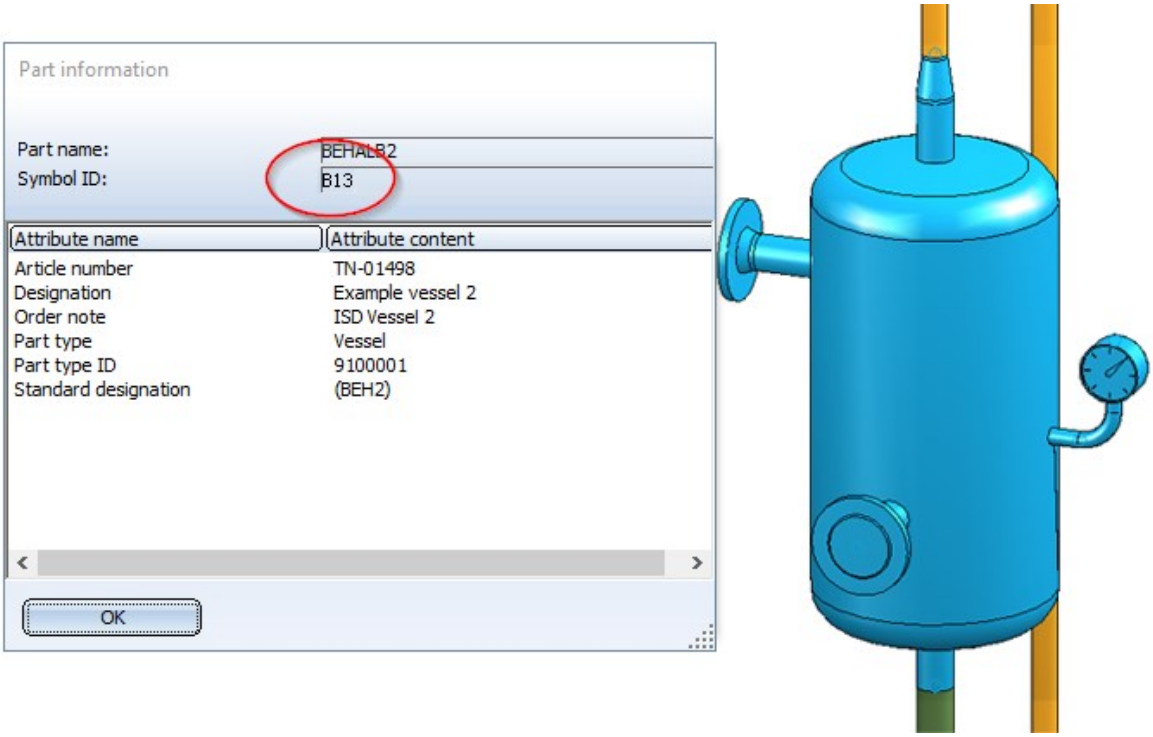
Take over P+ID symbol code change to 3-D layout plan

From HiCAD 2020 SP1 onwards, symbol code changes in the P+ID are taken over to the assigned 3-D layout plan.

If the **Edit symbol** function in P+ID is used to change the Code of a symbol, the change is visible when switching to the **Assigned 3-D layout plan** if the **Part information** of the corresponding 3-D part is displayed



Changes made with the **Edit symbol** function are transferred to the 3-D layout plan.

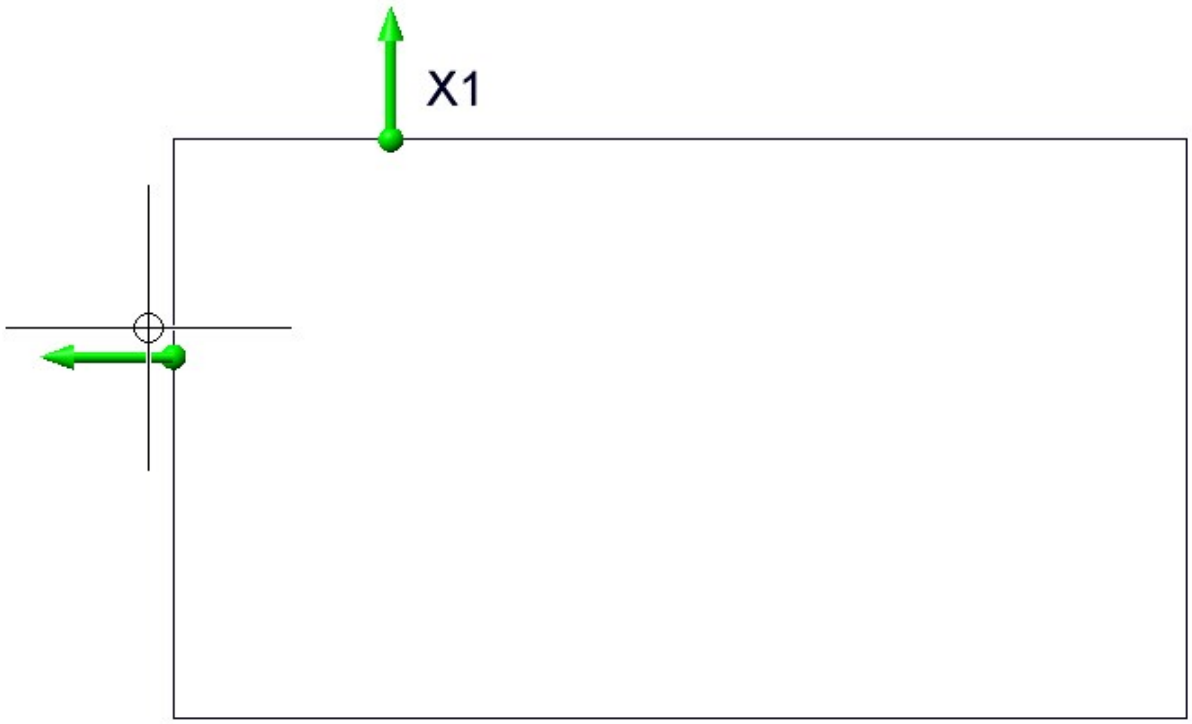


Choose the **Part information** function (click with the right mouse button on the part) to display the changed Code in the 3-D lay-out plan in Plant Engineering.

Major Release 2020 (V 2500)

Indicated connecting direction when creating own symbols

Setting new connections is now supported by graphic plug-ins. After selecting a specific connection direction, an arrow is generated in the corresponding direction which can only be placed at grid points. Connections that have already been placed are highlighted with an arrow. This new feature is available with the P+ID Symbol Editor as well as when editing a P+ID.



Note in the image above that the cursor does not coincide with the start point of the arrow because the arrow "jumps" to grid points.

Support of nominal diameters in millimetres and inches

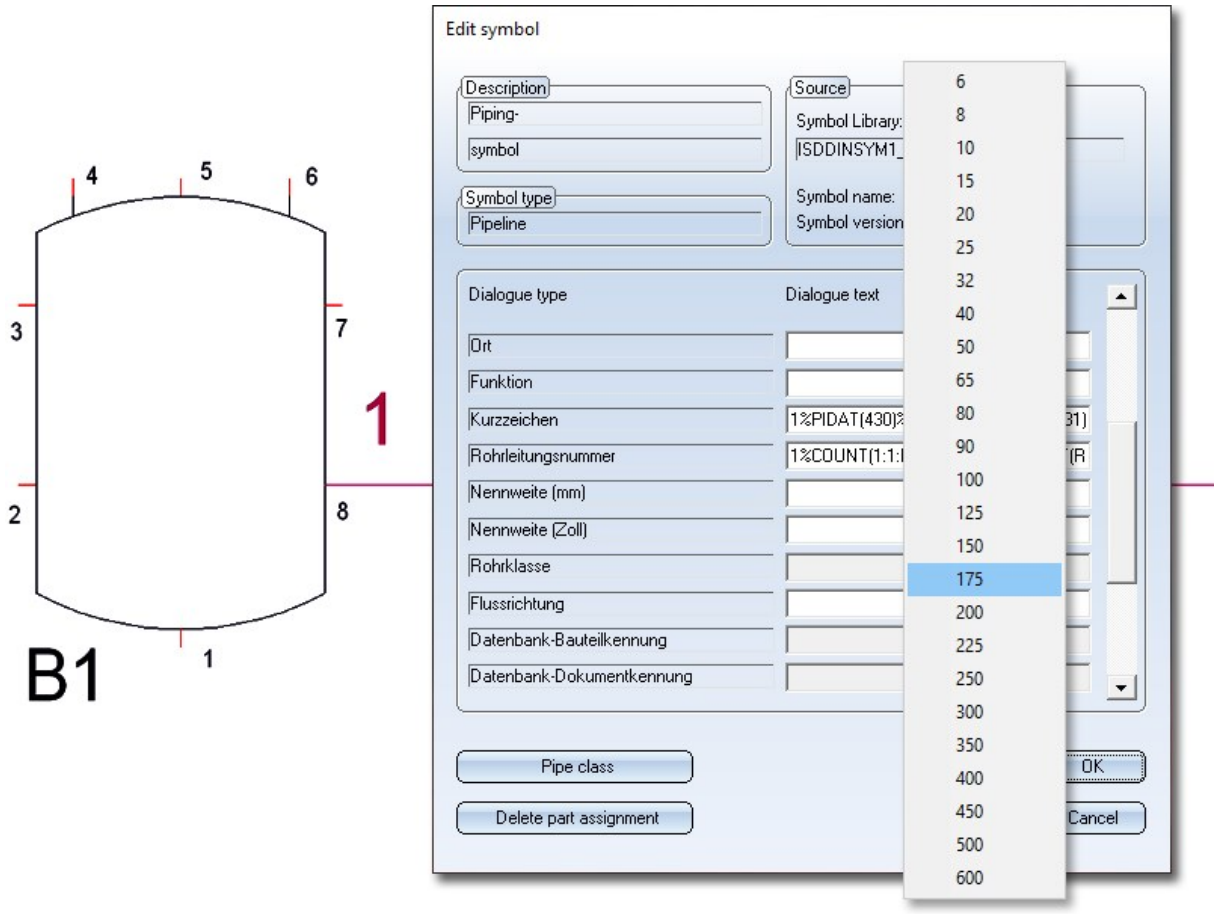
From now on there are two different nominal diameter attributes in the P+ID, one in millimetres and one in inches. The two attributes are linked to each other according to the assignment list in the file ANINCHTABLEW.DAT (HiCAD EXE directory / SYSTEM). By default, the nominal diameter attribute in inches has been added to the delivered libraries. In older libraries it should be added. Simply open a library, add an additional inch attribute to all symbols with a nominal diameter attribute and save the library. An addition of the nominal diameter attribute in inches is also done in older projects. In this case it is also sufficient to open and save the project. An addition to the line:

4310 82 0 Nominal diameter (inch)

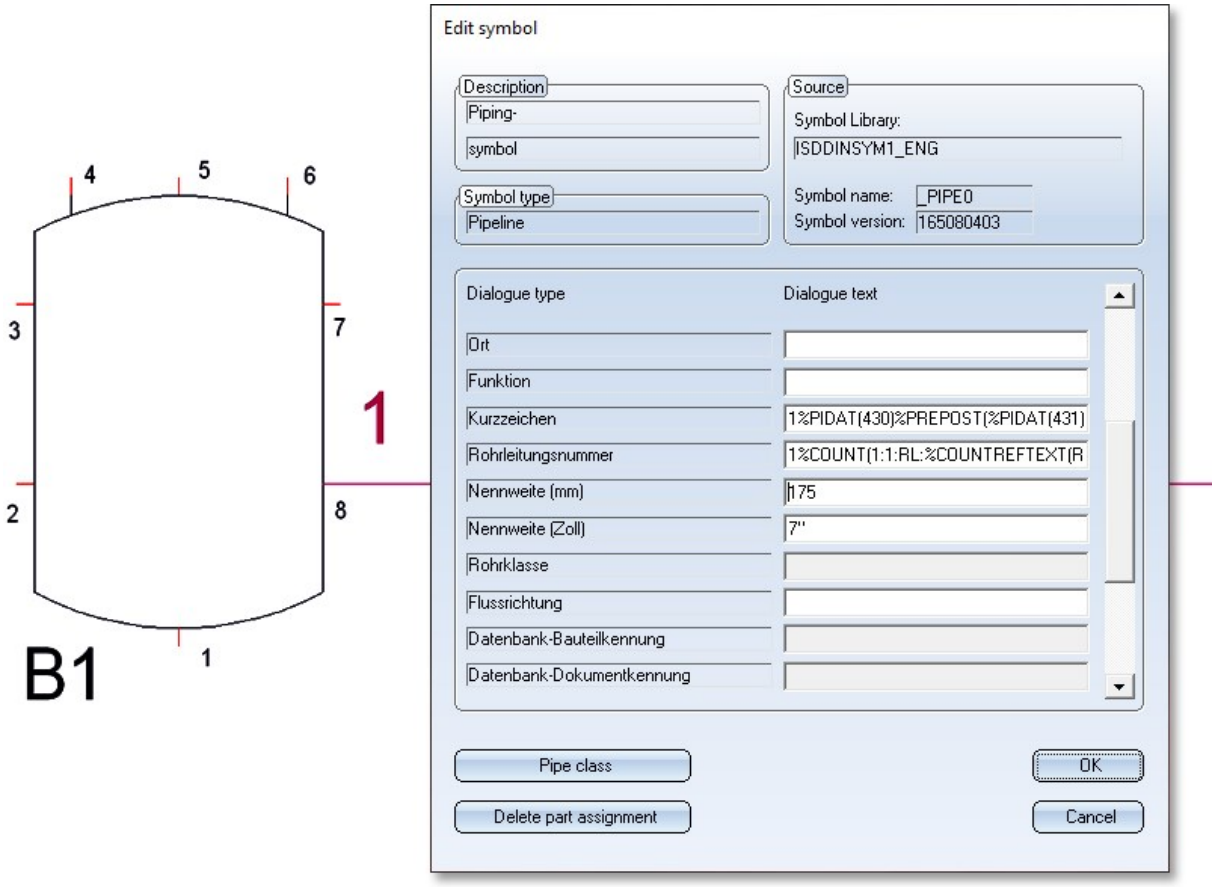
(tabulator characters between blocks) in the FLDLGCODE.dat files in the ISDAttDef folders is necessary if the corresponding files provided by the ISD are not used. An addition to the nominal diameter attribute in inches is also used in older projects. Here too, it is sufficient to open and save the project. Depending on the size of the project, this process can take a few minutes.

Before creating a new project, all libraries should be updated.

When entering a value into one of the two Edit boxes for the nominal diameter of a symbol, an automatic conversion takes place in the nominal diameter for the unprocessed Edit box, provided the entry is in the above mentioned assignment list. When you right-click on an Edit box for a nominal diameter attribute a selection list according to the assignment list will be displayed.



When a specific entry is selected, the corresponding entry in inches is automatically entered in the provided Edit box .



For symbols with several nominal diameters, a corresponding automatic conversion will be made, too.

Viewer

Major Relase 2020 (V 2500)

Visibility of textures

The settings of the HiCAD functions

Settings for shaded representation, Active view und

Settings for shaded representation, All views

for the representation of textures (at **Views > Representation > Shaded** **> Shaded representation**) will now be considered for the representation of textures. This means that whether and in which view the textures will be shown in the Viewer will depend on the above settings.

Notes on HELiOS Updates

Major Release 2020 (V 2500)

Important Notes on Updates for Administrators

For updates to HELiOS 2020 (Version 2500 or higher) a central update of the supplied HELiOS database is required.

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

- Data backup before updating

  Make sure that a data backup was made before carrying out the update of your HELiOS database.
  For the backup, either use the HELiOS Database Creator (further information can be found in the Installation Notes) or your SQL Server Application.
  In case of any questions, or if you need any help with regard to your customized system architecture , contact the ISD Hotline.

- Log file for update

  If any conflicts occur during the update, these will be recorded in the log file **HeliosDbUpdate.txt** (in the system path **%appdata%\ISD Software und Systeme\HeliosDbUpdate**).
  Have this file ready when contacting the ISD Hotline in case of an unsuccessful update, so that they can help you solve the problem and make a successful update.

- New mask format

  Please read all notes with regard to the new mask format in HELiOS 2020 !

- MultiCAD interfaces

  If you are working with an Inventor or SOLIDWORKS interface and HELiOS, please note that before installing an update of an older version to HELiOS 2020 (Version 2500) or higher, some adjustments may have to be made before the update.
  In this case, please contact the Consulting department of the ISD Group.

HELiOS Desktop

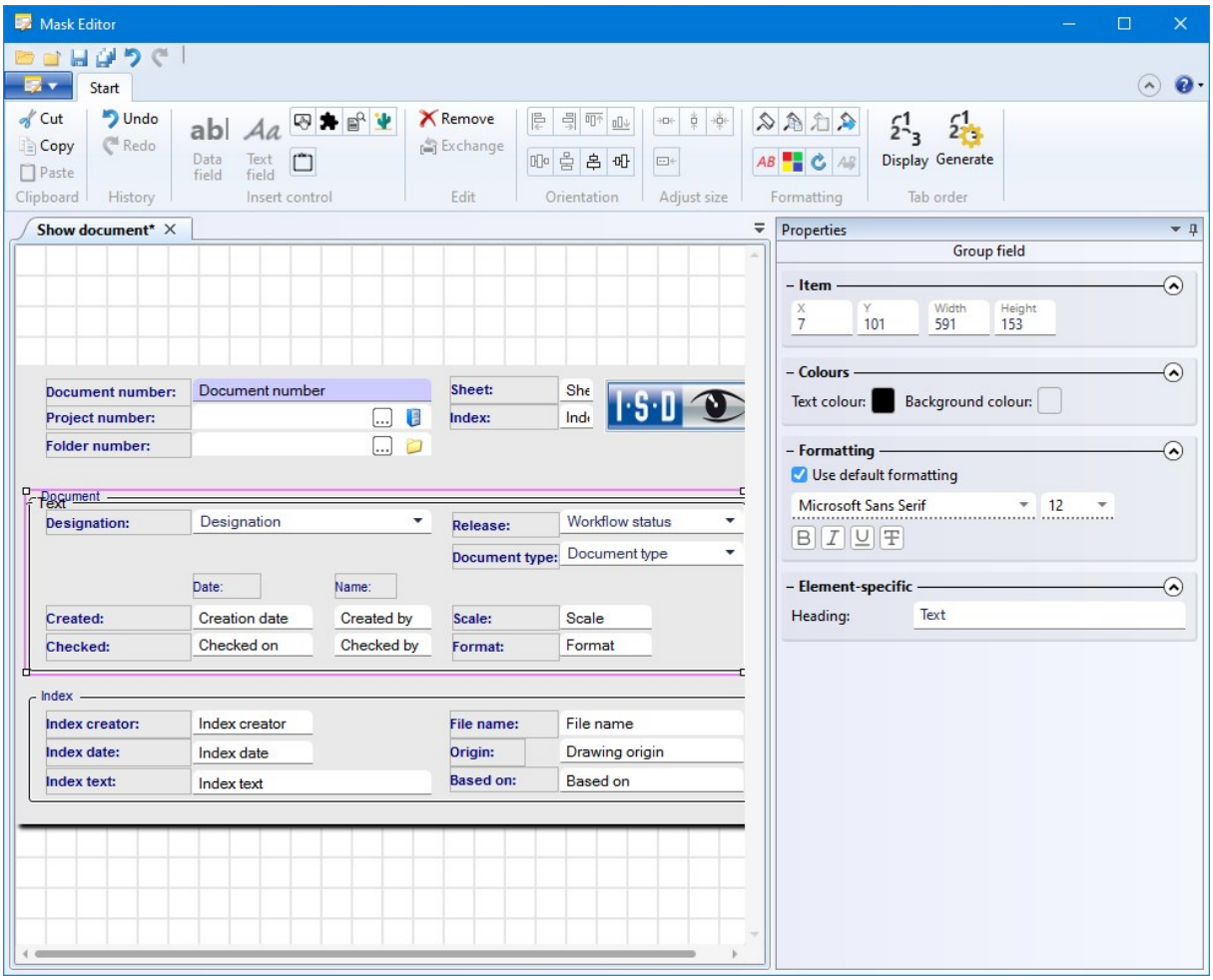
Service Pack 2 2020 (V 2502)

Enhanced Mask Editor

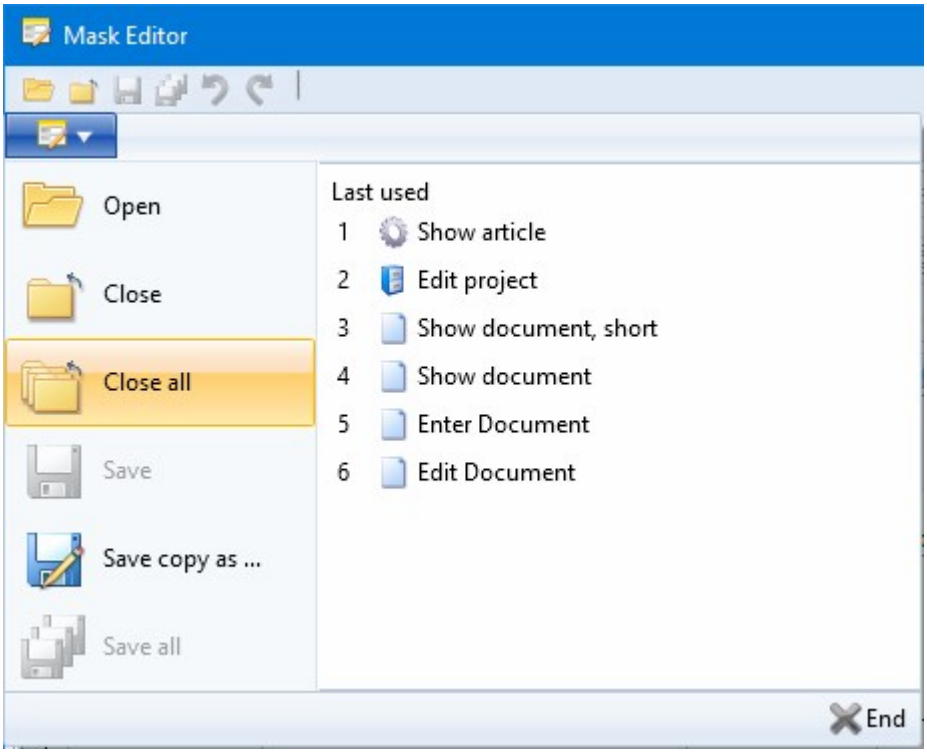
The new Mask Editor introduced with HELiOS 2020 has been further revised and enhanced with new features.

The **Transfer formatting** function allows you to transfer the formatting of a specific element to a selection of other mask elements.

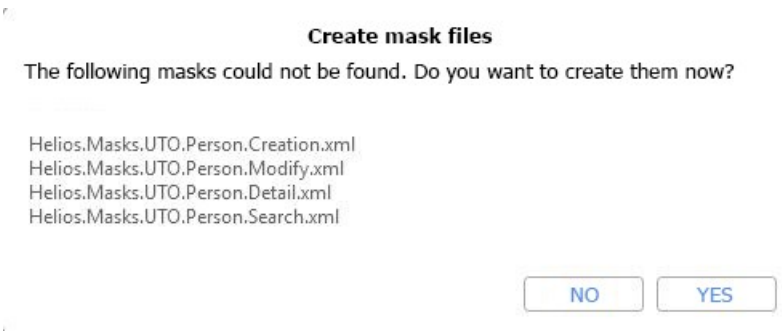
The also new **Exchange** function can be used to change the attribute stored in a data field via a selection dialogue.



The range of functions for opening, closing and saving mask files has also been extended, including a display of last used files.

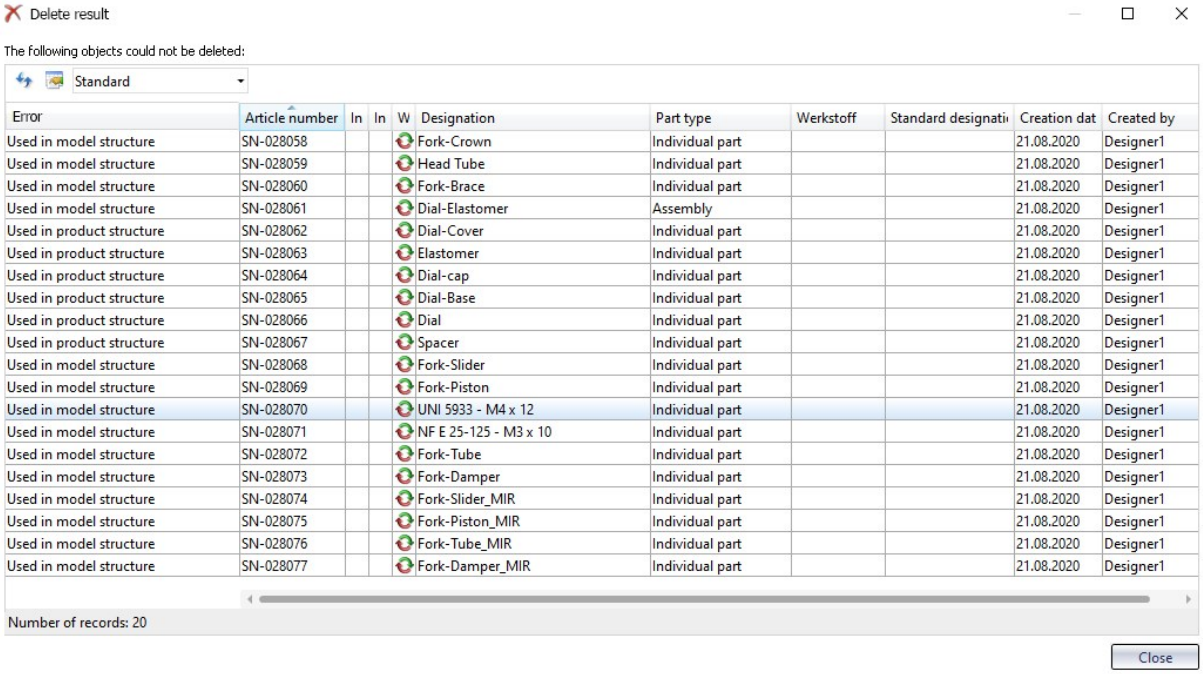


If new **user-defined object types** have been created in the **Attribute Editor**, you will be asked to create new mask files for them after calling or restarting the Mask Editor. Confirm the query with **YES** to execute the automatic process.



Intelligent "Delete" behaviour of objects

If HELiOS objects (Articles, Documents, Projects, Folders, ...) cannot be deleted - e.g. because the user does not have the right to delete them, or the corresponding edited objects are reserved for export, or are built into the product structure and/or model structure - then these objects are displayed in a corresponding result list after the deletion process, which shows the reason why the deletion process cannot be carried out for each object.



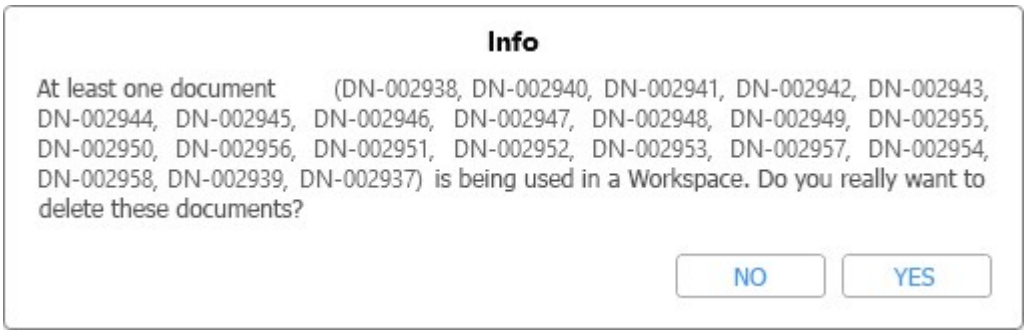
Error	Article number	In	In	W	Designation	Part type	Werkstoff	Standard designati	Creation dat	Created by
Used in model structure	SN-028058				Fork-Crown	Individual part			21.08.2020	Designer1
Used in model structure	SN-028059				Head Tube	Individual part			21.08.2020	Designer1
Used in model structure	SN-028060				Fork-Brace	Individual part			21.08.2020	Designer1
Used in model structure	SN-028061				Dial-Elastomer	Assembly			21.08.2020	Designer1
Used in product structure	SN-028062				Dial-Cover	Individual part			21.08.2020	Designer1
Used in product structure	SN-028063				Elastomer	Individual part			21.08.2020	Designer1
Used in product structure	SN-028064				Dial-cap	Individual part			21.08.2020	Designer1
Used in product structure	SN-028065				Dial-Base	Individual part			21.08.2020	Designer1
Used in product structure	SN-028066				Dial	Individual part			21.08.2020	Designer1
Used in product structure	SN-028067				Spacer	Individual part			21.08.2020	Designer1
Used in model structure	SN-028068				Fork-Slider	Individual part			21.08.2020	Designer1
Used in model structure	SN-028069				Fork-Piston	Individual part			21.08.2020	Designer1
Used in model structure	SN-028070				UNI 5933 - M4 x 12	Individual part			21.08.2020	Designer1
Used in model structure	SN-028071				NF E 25-125 - M3 x 10	Individual part			21.08.2020	Designer1
Used in model structure	SN-028072				Fork-Tube	Individual part			21.08.2020	Designer1
Used in model structure	SN-028073				Fork-Damper	Individual part			21.08.2020	Designer1
Used in model structure	SN-028074				Fork-Slider_MIR	Individual part			21.08.2020	Designer1
Used in model structure	SN-028075				Fork-Piston_MIR	Individual part			21.08.2020	Designer1
Used in model structure	SN-028076				Fork-Tube_MIR	Individual part			21.08.2020	Designer1
Used in model structure	SN-028077				Fork-Damper_MIR	Individual part			21.08.2020	Designer1

The deletion processes are executed in an optimal sequence, i.e. taking into account any existing references in the model or product structure.

Please note:

Under **Error**, only the first error that HELiOS encountered during the deletion process is displayed. That is, there may be other reasons why the object cannot be deleted immediately after the error has been corrected.

Furthermore, even if the Workspace is closed, you will be informed if objects to be deleted are currently in the Workspace.

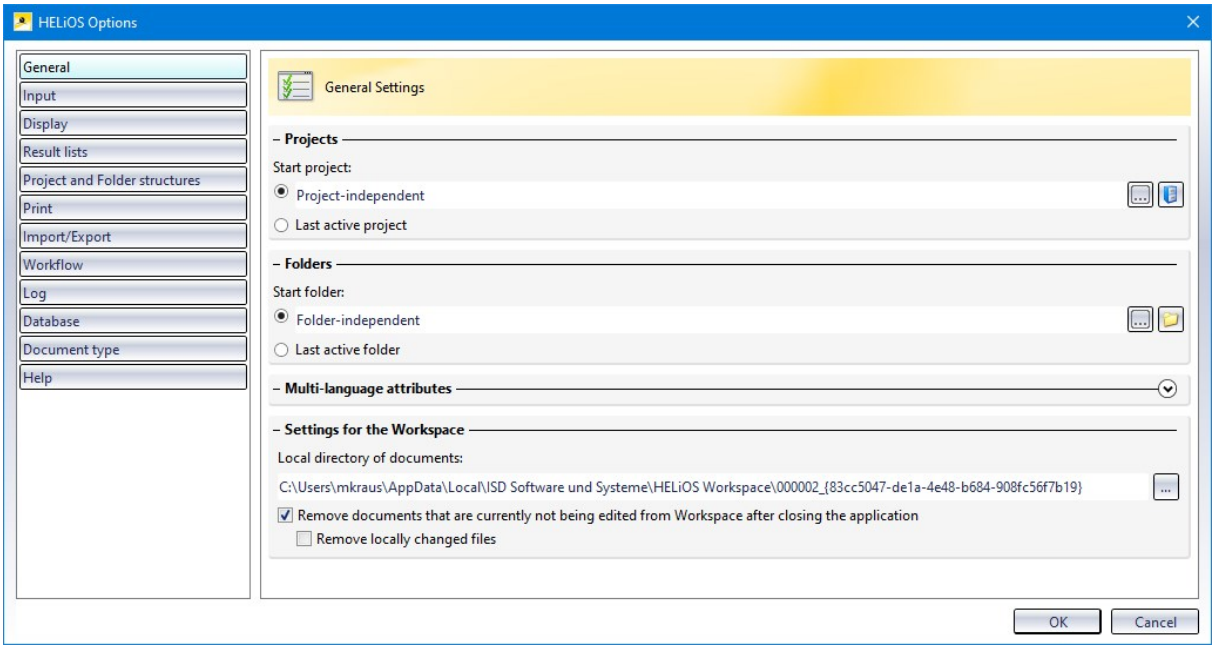


Settings for the Workspace

On the **General** tab of the **HELiOS Options** you can now find the new **Settings for the Workspace** area.

As with the known setting options in the area of MultiCAD interfaces you can also change the local session directory for the Workspace of the HELiOS Desktop, if desired.

You can also specify whether documents that are not currently being edited, or whether locally changed documents are to be removed from the work area when the program is closed.

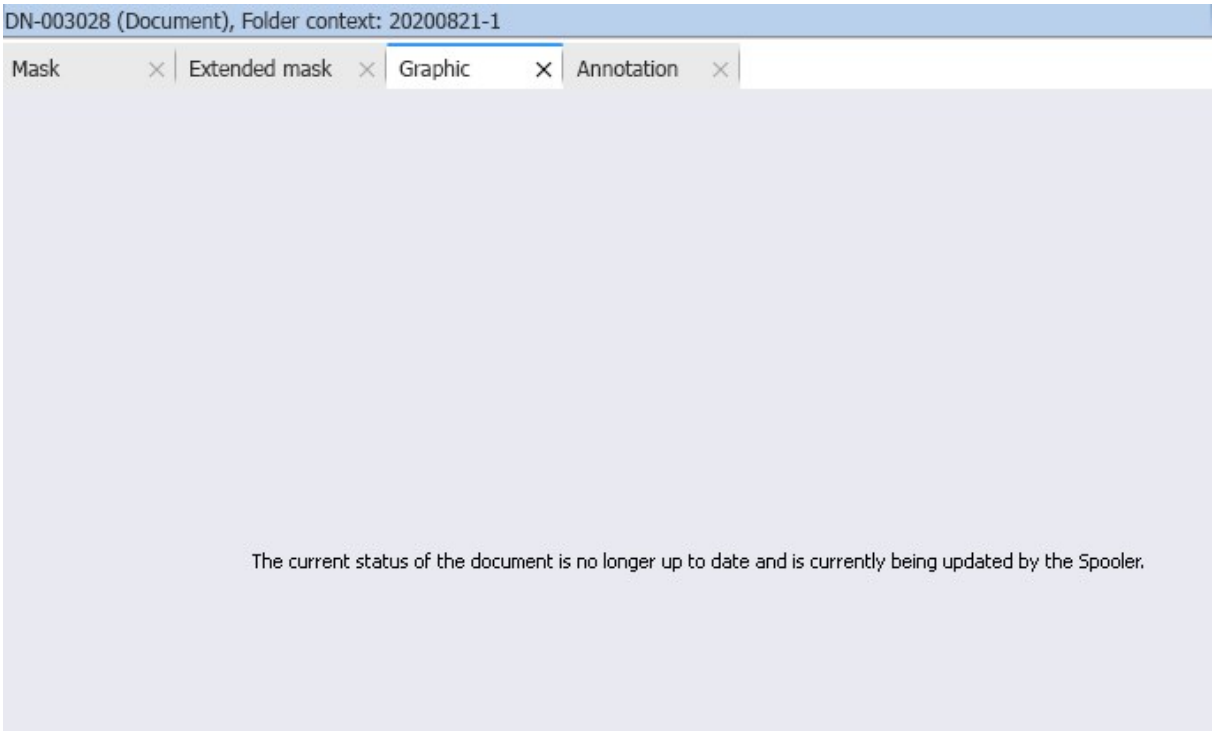


Marking "empty" archive documents when loading, viewing and exporting

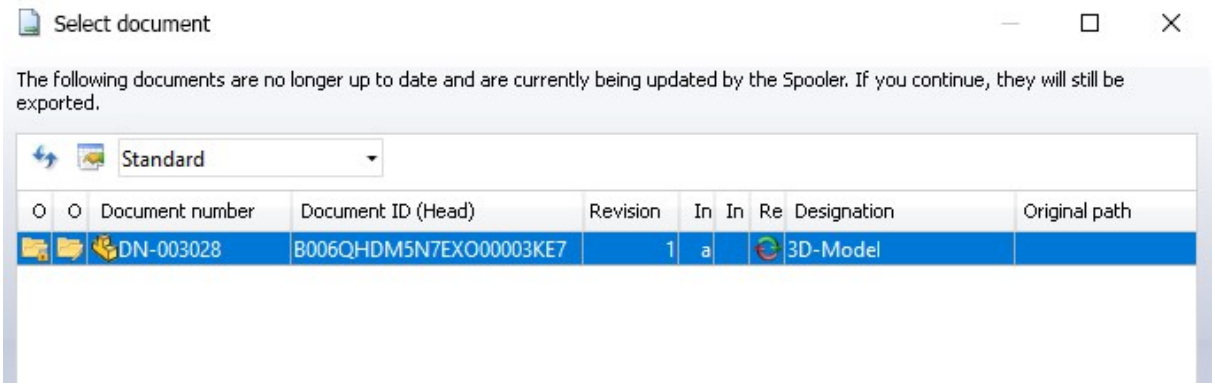
When sending print or conversion jobs via KST, which subsequently update an existing document master in HELiOS, the corresponding document in HELiOS can be marked as "obsolete".

A common example of this is the updating of neutral formats via the spooler. Another example is customer-specific adaptations that automatically generate archive documents upon release. Here it can happen that these archive documents are displayed as "empty" due to errors, outdated document statuses or a release or conversion by the HELiOS Spooler still running in the system.

Such documents are appropriately indicated when the HELiOS document master data function is called:



Also when exporting such documents, as well as when calling actions such as **Create notes document**, **Create mark-up** or **Open with (...)** you will receive a corresponding warning message:



After the document is updated by the Spooler, the corresponding identifier will be removed again.

Major Release 2020 (V 2500)

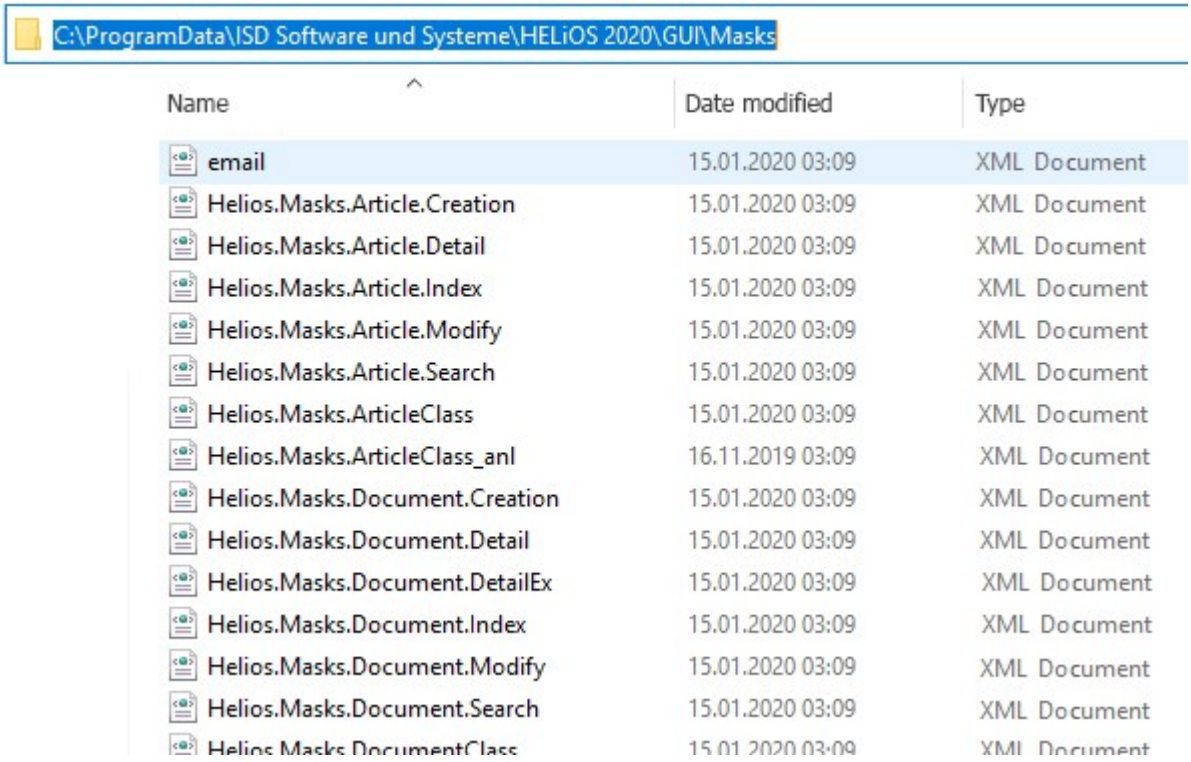
New mask format and central mask configuration

In HELiOS 2020, the previous mask format has been changed from .msk to .xml, in addition to the revision of the **Mask Editor**.

In addition, mask files are now managed outside the installation directory, resulting in some changes in the system architecture that administrators should urgently consider:

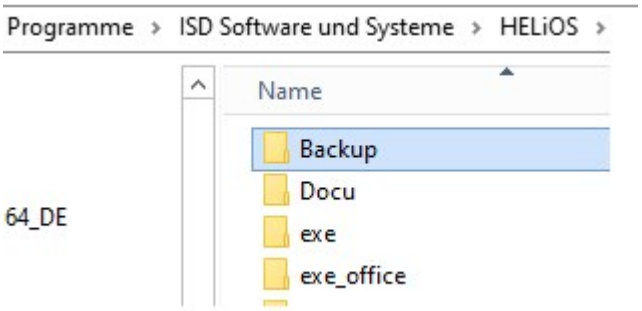
The masks of older versions that were previously stored in the \sys\ directory of the installation are automatically converted to the new XML format by an update installation of an older HELiOS version to HELiOS 2020 (V 2500) or higher and stored at**%Programdata%**.

At %**Programdata%**, the masks are stored in a version-specific path, i.e. for HELiOS 2020 this path could look like this: "C:\ProgramData\ISD Software und Systeme\HELiOS 2020\GUI\Masks". In a later version, the version identifier in the path will be correspondingly higher.



This conversion or directory is not dependent on whether the local or central directory was selected for the interface configuration (see below).

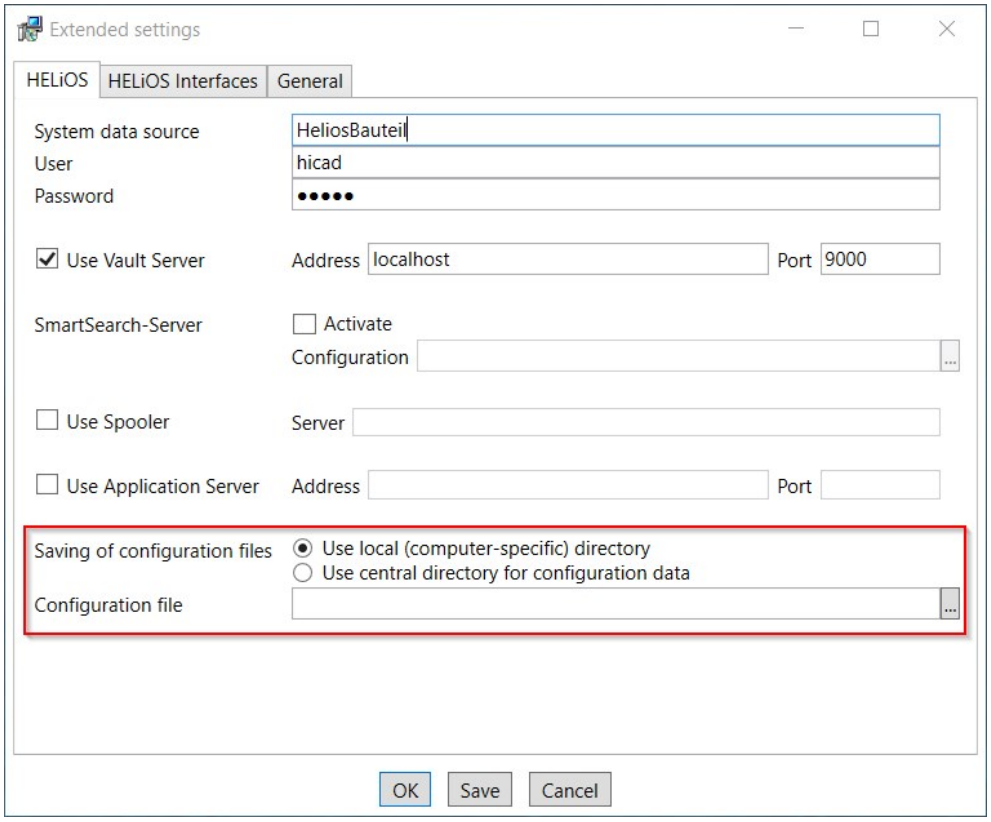
The old configuration file can then be found in a **Backup** directory that was created below the installation folder.



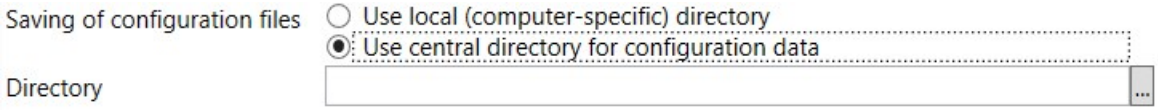
The new masks at **%Programdata%** should not be overwritten in any case, because these are the "delivery status", which may be updated by later update installations. If masks are edited and changed with the mask editor, they are stored at **%Appdata%** and are called up by HELiOS when working in corresponding masks from this directory.

This means: If masks are to be distributed uniformly to different workstations, they must be saved under %Appdata% and not (!) under %Programdata%.

The optional query for UI configuration during a new, update or change installation has also been extended to distinguish between a local and a central directory (across computers). This means that you can also store the interface configuration for several work centers in a central directory instead of having to "distribute" it to the individual local directories.

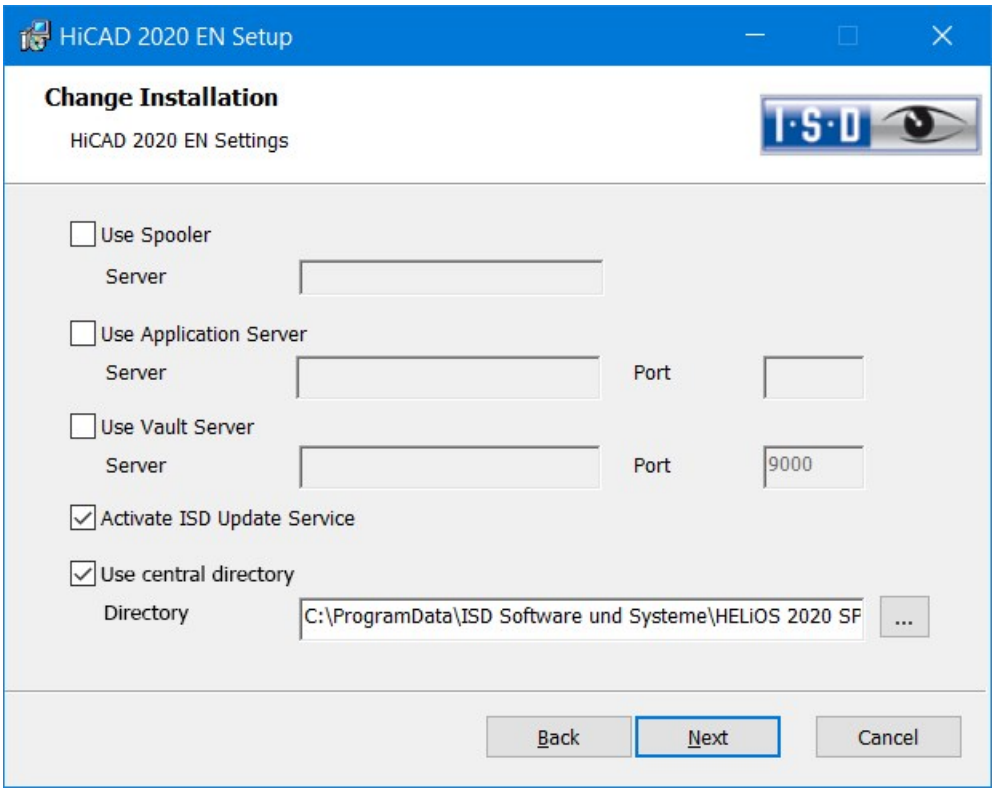


If you select the option **Use central directory for configuration data** in the new or update installation, you must select a **Directory**, e.g. network drive, via Windows Explorer instead of the local**Configuration file** in .zip format after clicking on the ... symbol in the selection dialogue.



Please also note the following:

- A **Change** installation of HELiOS or HiCAD with HELIOS offers the possibility to change to a central configuration directory later. However, the subsequent selection of a local configuration file is not provided at this point. If you want to change the local configuration afterwards, you should use the tool **HELiOSConfigImport.exe**.



- Settings files that are provided by the administrator in a central directory must be selectively copied to this directory. Due to the dependency on files, the directory structure provided by HELiOS must be observed. This must correspond to the structure under **%APPDATA%\ISD Software und Systeme\HELiOS 2020**. If you have any further questions, please contact the Consulting department of the ISD Group.
- If the set configuration directory is not accessible at runtime, HELiOS behaves as if the directory is empty. This means that the program starts without updating the UI configuration.
- When updating HELiOS for an older HiCAD version, the option to set the configuration directory is missing. This can only be adjusted via the registry. If you have any further questions, please contact the Consulting department of the ISD Group.

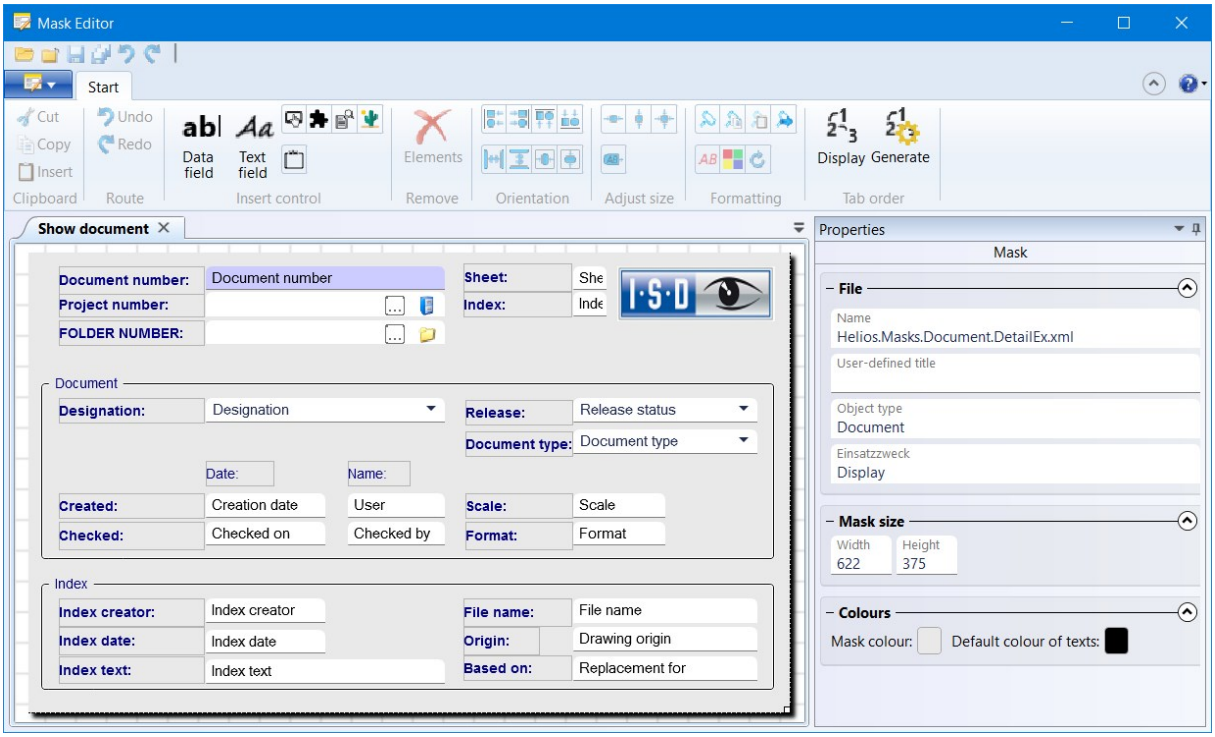
If problems occur during the conversion to the new mask format in the course of an update installation, the mask converter **MskToXmlUpdate** can be used.

New Mask Editor

With HELiOS 2020 (version 2500) you will find a new **Mask Editor** on the **Admin Tools** tab of the HELiOS Desktop, which has been adapted to the latest state of the art with a modern Look & Feel and thus offers some innovations and advantages compared to the previous version.

In addition to a modern Ribbon design, which also supports operation at high resolutions, this has laid the foundation for further expansion of the masks and the Editor in 2020.

The revision of the Mask Editor has also led to a conversion of the mask format to XML. Mask files are now managed outside the installation directory.



Start search with F3

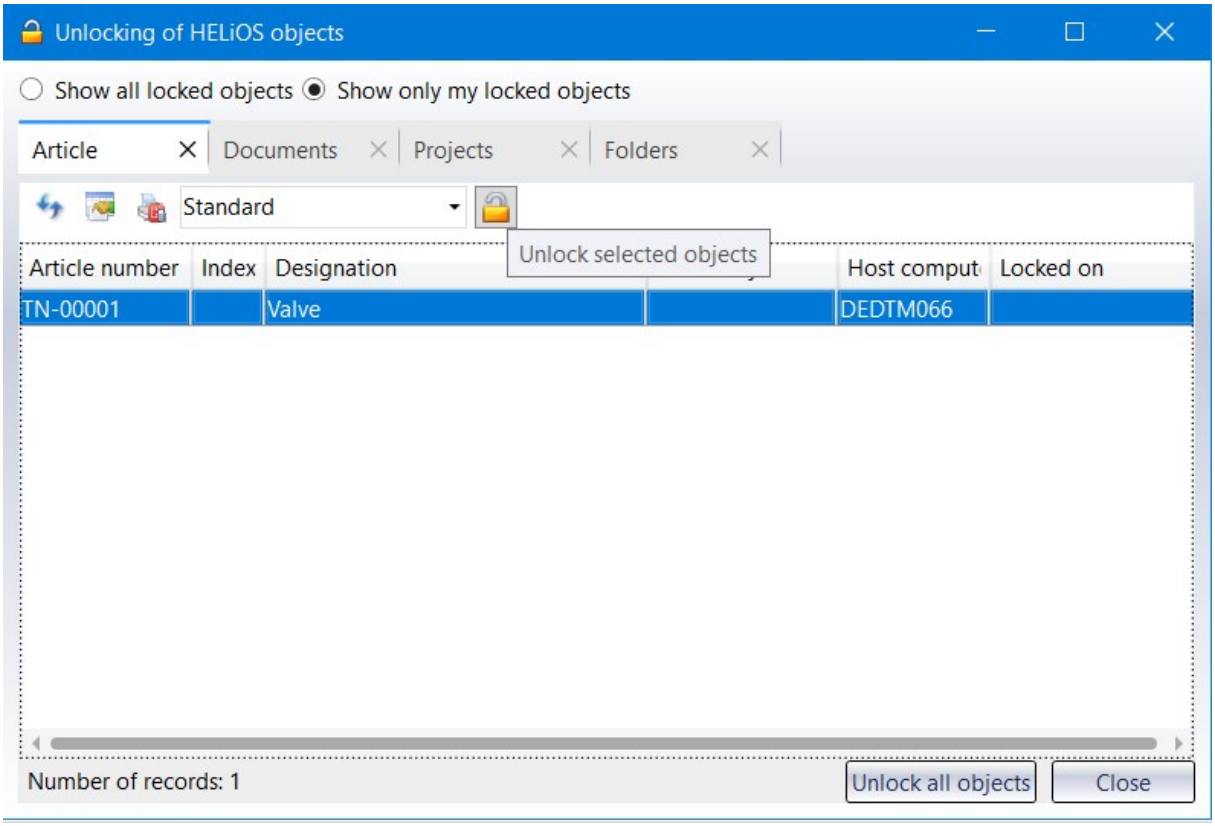
To avoid having to switch between keyboard (entering the search criteria) and mouse (initiating the search) when searching for a specific object in the HELiOS Database (Document, Article, Project, Folder, ...), you can also use the F3 key to perform the search.

You must therefore click on **Find** with the mouse in the corresponding, active search mask, but you can also execute the search using the keyboard with F3 for manually entered criteria.

ReleaseNext: Distinguishing between all objects and Admin objects

In the Admin tool **ReleaseNext** for unlocking HELiOS objects that are being edited by other instances, you can now distinguish between two options:

- **Show all locked objects**: In the corresponding object tab (Articles, Documents, Projects or Folders), all entries that were locked in HELiOS are listed (user-independent).
- **Show only my locked objects**: Only the objects that were locked by the logged on Administrator are listed.

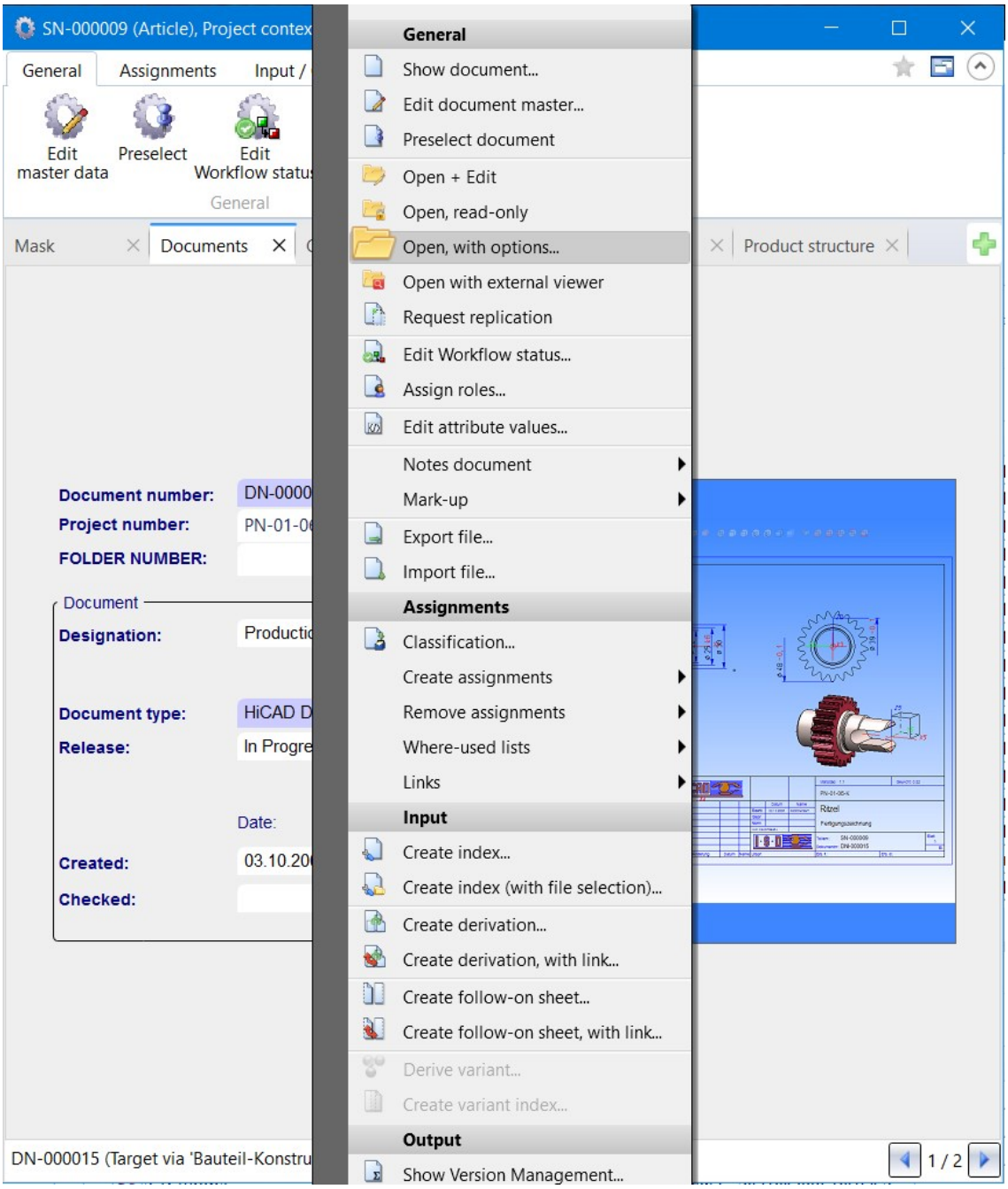


HELiOS in HiCAD

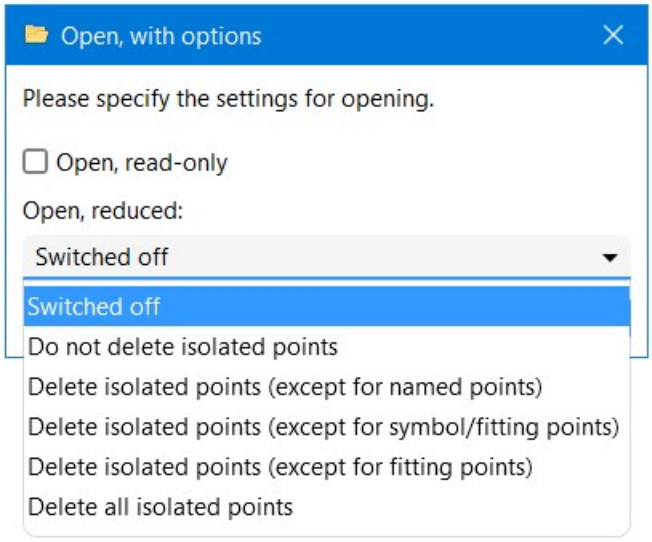
Major Release 2020 (V 2500)

Open, with options: Reduced loading

In the context menu and in the detail window ribbon of a HELiOS document, you will find the new function **Open, with options** at General, in addition to the familiar functions **Open + Edit** and **Open, read-only.**



If the document is an HiCAD drawing file (SZA format), an extended dialogue with additional options for reduced loading appears. This means that you can optionally select how to proceed when loading with isolated points:



You can leave the **Open, reduced** option **Switched off** at this point or **Delete all isolated points** or **Delete isolated points except for**:

- named points
- symbol / fitting points
- fitting points

You can also activate the **Open, read-only** checkbox in this dialogue if the document should only be loaded for viewing and not for editing.

Please note:

- The **Open, with options** function does not work for multiple selections, but only for single SZA documents.
- If the document opened with options is not an SZA file but, for example, an Office document, it is opened without another intermediate dialogue.
- If the file type of a document is not known or cannot be assigned to an application for opening, a Windows query may appear asking in which application the document should be opened.

HELiOS Spooler

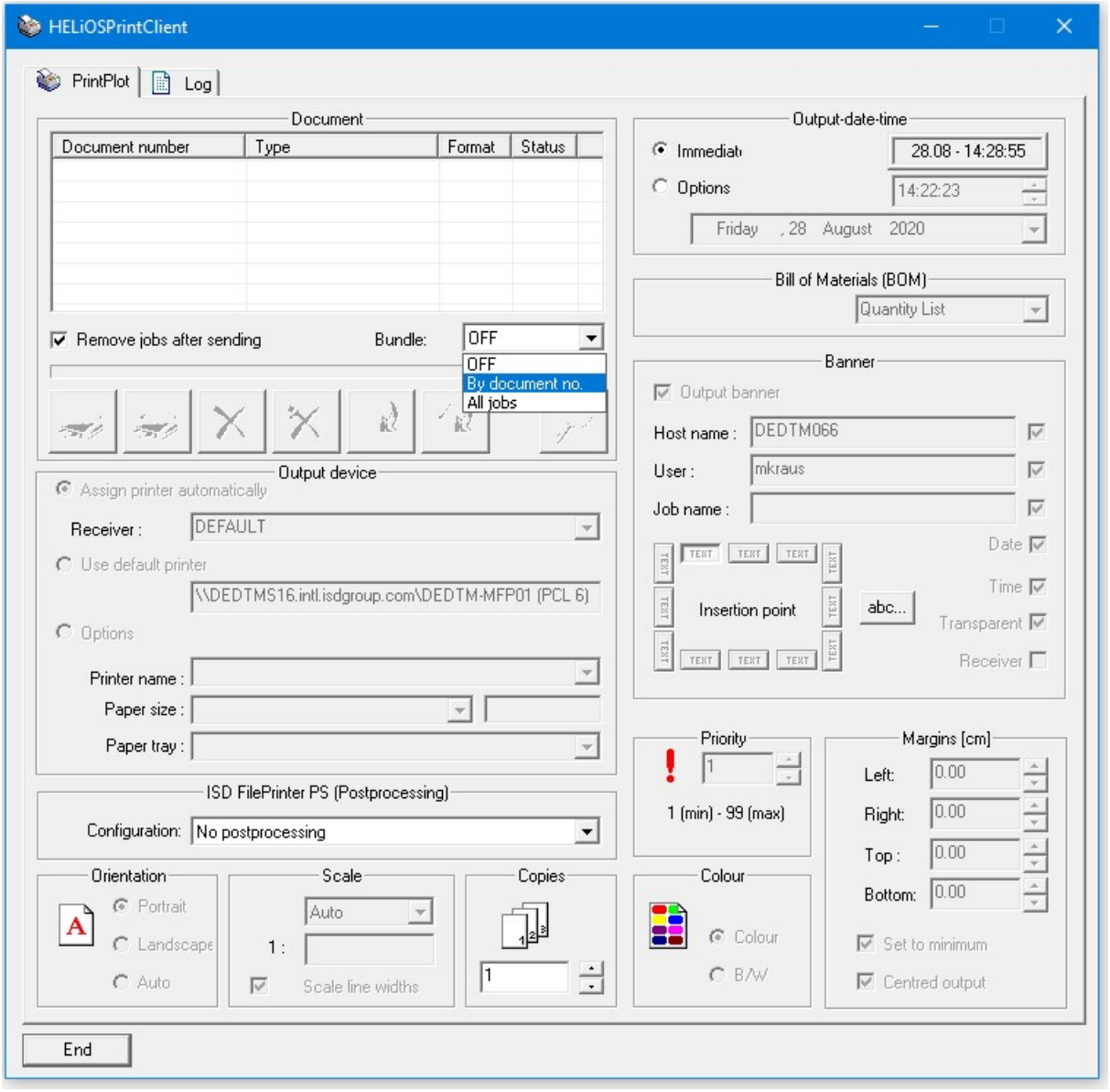
Service Pack 2 2020 (V 2502)

Print Client: Bundling of jobs by document number

The option to bundle jobs in the **PrintPlot** menu part of the **HELiOS Print Client** has been extended:

In addition to the default setting where print jobs are not processed in bundles (**Bundle: Off**), you can now choose between **Bundle: All jobs** and **Bundle: By document no**.

If the **Bundle: By document no.** option is active, documents (e.g. SZA files with several sheet areas each) can be converted into separate PDF documents, where a PDF document contains a separate sheet for each HELiOS sheet area.

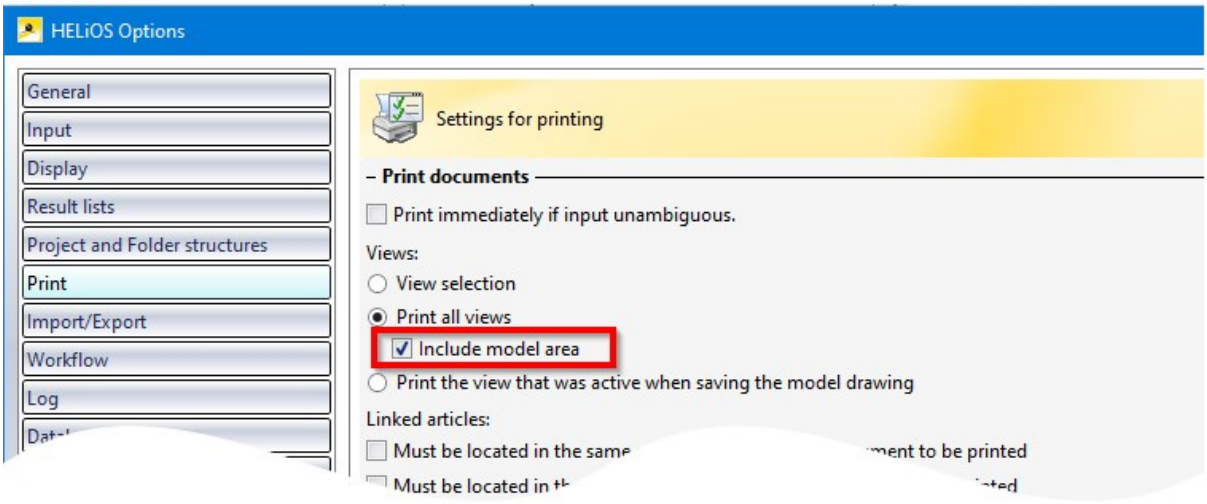


HELiOS Options: Include model area

In the **HELiOS Options** for printing documents you find the new sub-menu item **Include model area** under **Print all views**.

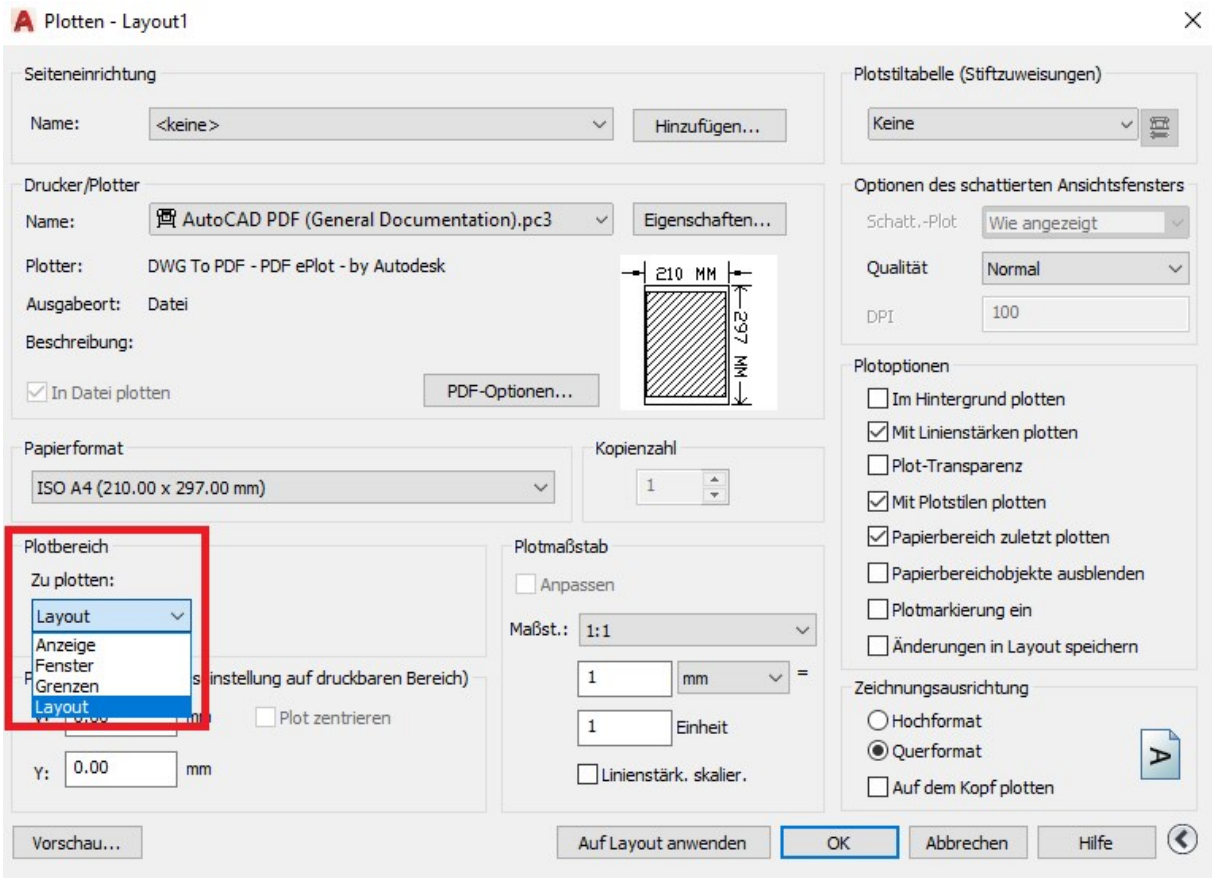
This option is relevant if you work with a CAD system (such as HiCAD or AutoCAD) that makes a distinction between model and drawing area.

If the option is not activated, all sheet areas of the drawing are transferred when the print jobs are transferred to the Report Manager. If this option is activated, the model area is also transferred.



AutoCAD plot area definition

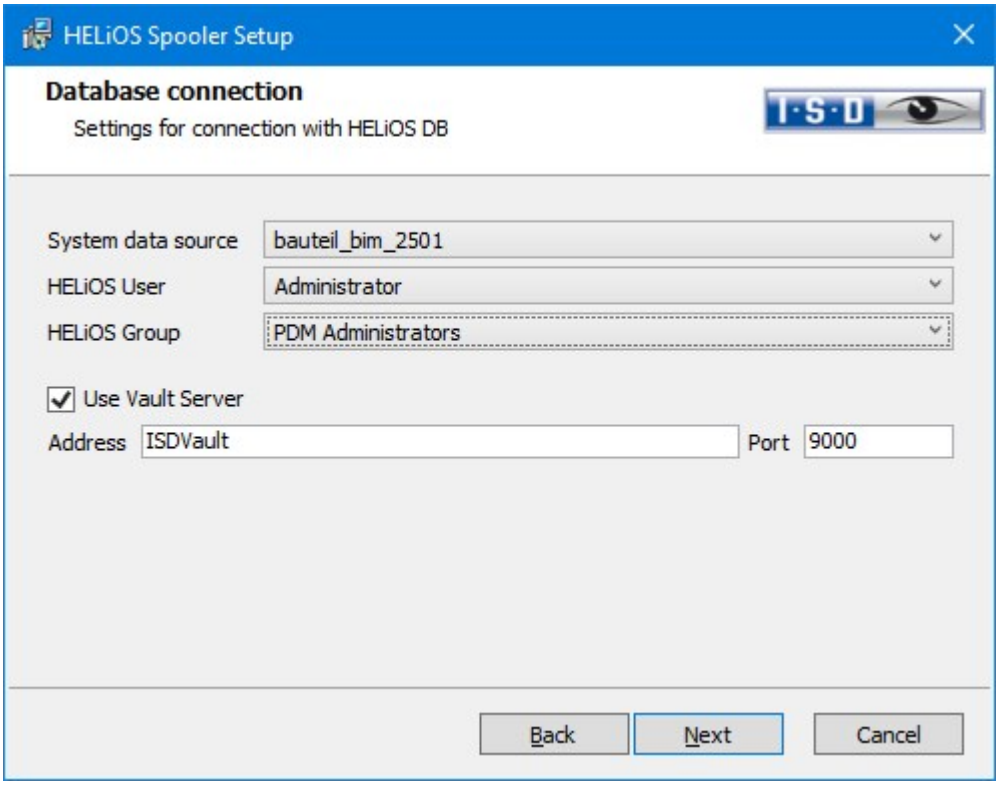
When printing files from AutoCAD, the user can also select the **plot area**.



With Service Pack 2 (Version 2502) the configuration options of MultiCAD formats of the **HELiOS Spooler** have been extended in such a way that the AutoCAD plot area, which distinguishes between model and layout, can be taken into account accordingly when printing AutoCAD files.

Installation

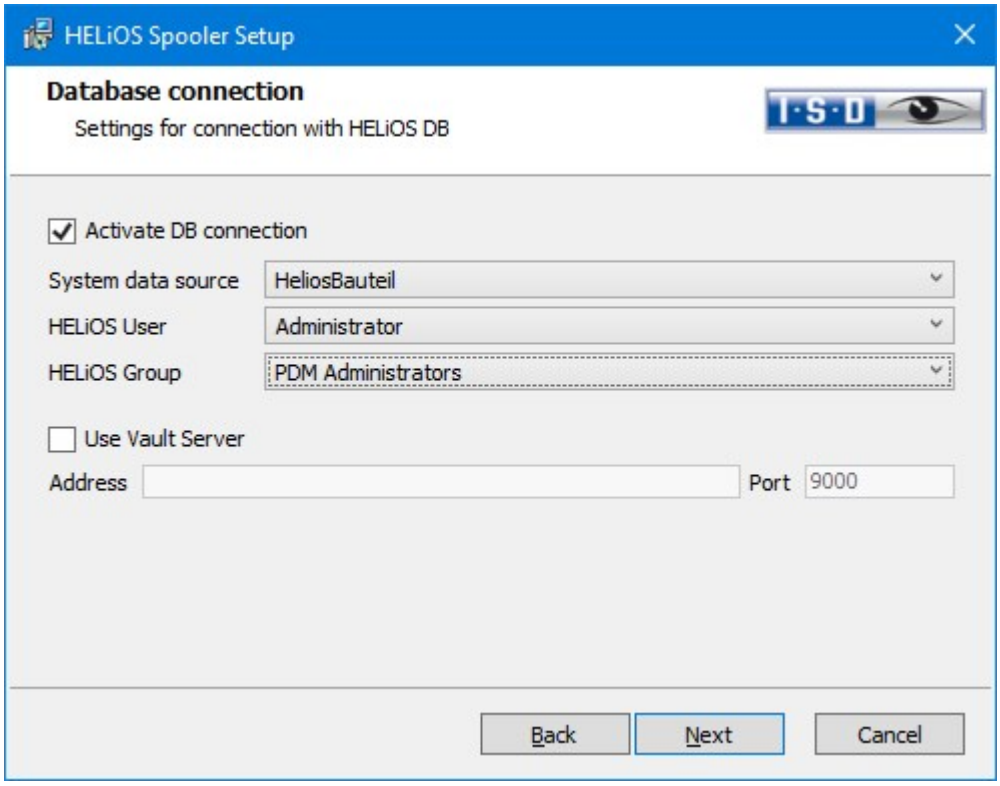
When installing the HELiOS Spooler, please note that the database connection specification, which was optional in earlier versions or only required for certain interfaces, will be mandatory starting with HELiOS 2020 Service Pack 2 (Version 2502).



Service Pack 1 2020 (V 2501)

Extension of the installation

When working with MultiCAD interfaces please note the following extension to the installation: By setting up the database connection at this point, an even more stable operation of the HELiOS-MultiCAD coupling is guaranteed.



Major Release 2020 (V 2500)

Notes on Update installations/ New postprocessing

The Spooler will be extended in such a way that the following information for bundled PDF export will be available for postprocessing:

<MultiSheetCurIdx> (%HEL_MULTISHEETCURIDX%) - Current sheet index for multiple sheets within one job (if applicable, otherwise always 0).
<MultiSheetEndIdx> (%HEL_MULTISHEETENDIDX%) - Last sheet index for multiple sheets within one job (if applicable, otherwise always 0).
<MultiSheetOutputName> (%HEL_MULTISHEETOUTPUTNAME%) - Output file name without file extension; for multi-sheet printouts identical with <FileName>, but without attached sheet index.

In case of an update installation, you have to copy some system files to the right places, which will not be over-written by it.

These are the files

- ISD FilePrinter? PS.BatchPDF.bat (executes the postprocessing)
- ISD FilePrinter? PS.BatchPDF.PostProcess (contains the calling of the postprocessing)

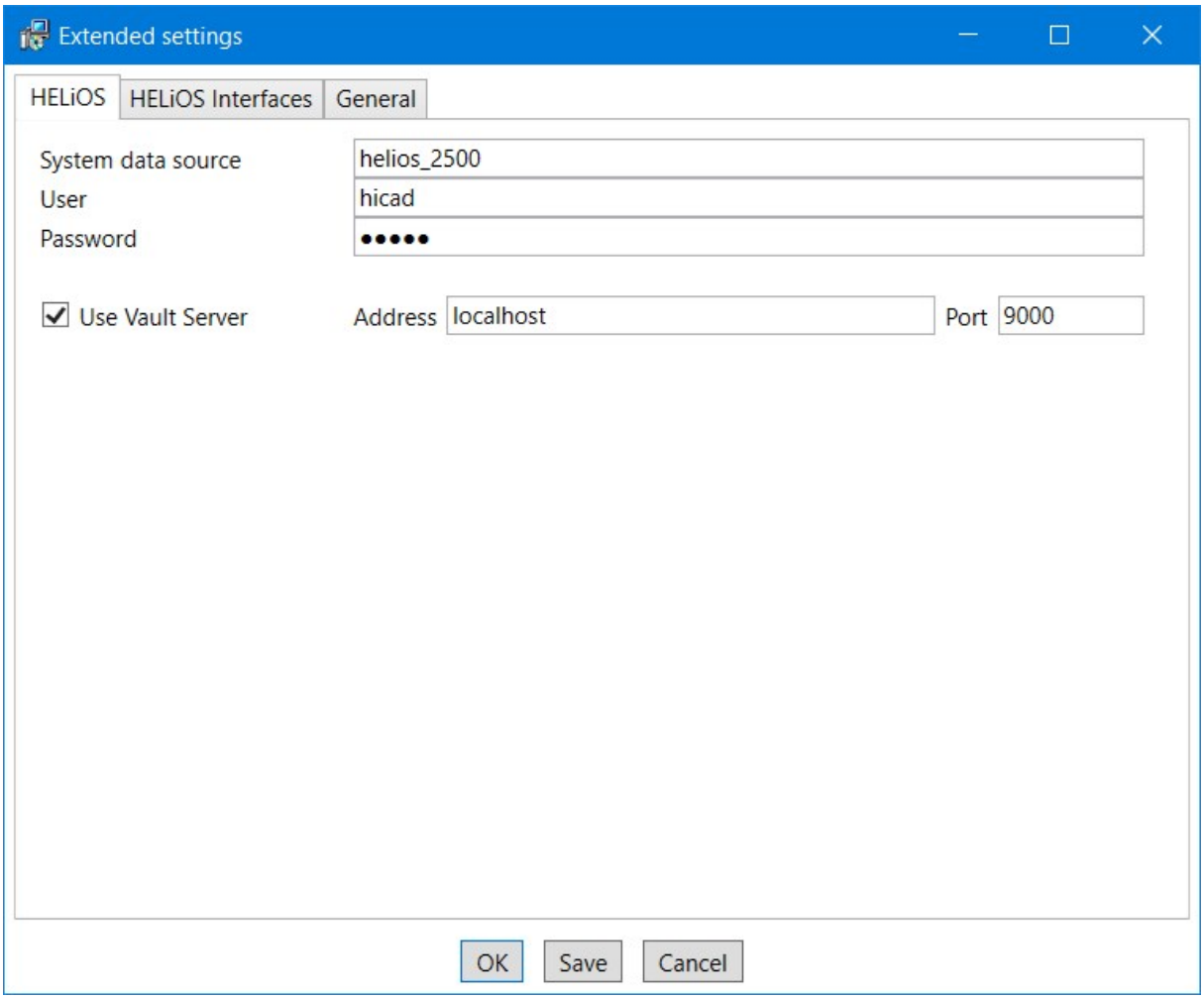
which you have to transfer from the installation directory of the Spooler (by default: C:\Program Files\ISD Software und Systeme\HELiOS Spooler\Defaults\FilePrintPostprocess\) to the target directory

C:\ProgramData\ISD Software und Systeme\HELiOS Spooler 2019 SP2\FilePrintPostprocess\.

A new installation of the HELiOS Spooler in Version 2500 (HELiOS 2002) is not affected by this.

"Change" option for MultiCAD settings

Not only can you configure the database and Vault Server connection for a CAD application (HiCAD, AutoCAD, Inventor, SOLIDWORKS) linked to HELiOS during a new or update installation of the HELiOS Spooler, but also via a **Change** installation.

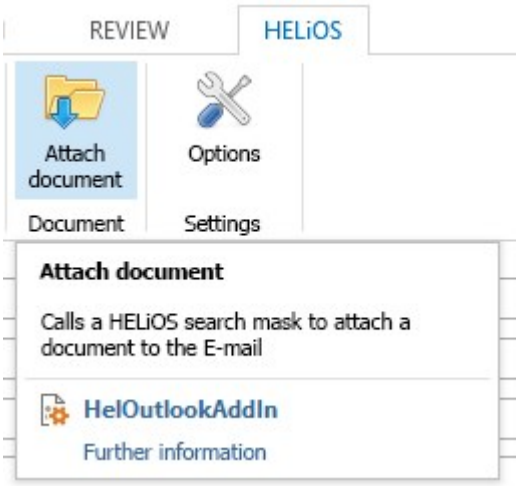


HELiOS MS Office Interface

Service Pack 2 2020 (V 2502)

Add attachments to E-mails

When you write, forward or reply to an E-mail in Outlook, the **HELiOS** Ribbon of the E-mail window also provides the **Attach document** function for attaching files via the **HELiOS** document search.



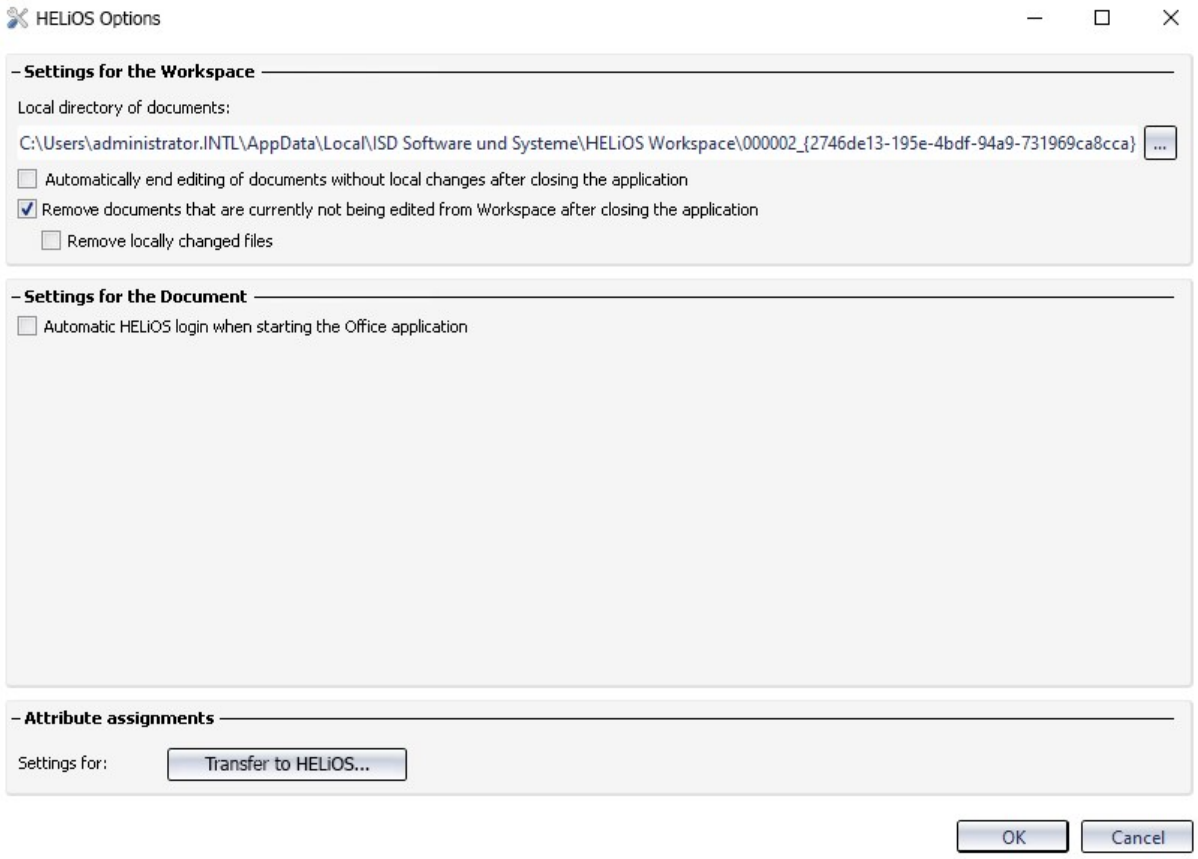
Please note:

If you are currently using Office 2010, the HELiOS interface will only work in separately opened E-mail windows, but not when the E-mail is docked in Outlook.

HELiOS options with automatic login

With the update to HELiOS 2020 Service Pack 2, an Options dialogue is available in the Office interface, as is already known from the HELiOS-MultiCAD interfaces.

In addition to settings for the Workspace and for the import of attribute mappings, you can activate the option **Automatic HELiOS login when starting the Office application** under **Settings for document** to prevent the HELiOS login dialogue from preceding each start of an Office application.



Service Pack 1 2020 (V 2501)

Transfer of forwarded E-mails as separate documents

When transferring forwarded E-mails to HELiOS, you can decide whether the forwarded E-mail should be transferred as a new HELiOS document (e.g. because the content has changed) or whether the original e-mail already stored in the HELiOS database is sufficient and the transfer of the forwarded E-mail is therefore not necessary.

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HiCAD

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HiCAD_2502

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