HiCAD

UNLIMITED CAD PERFORMANCE **DEVELOPED BY ISD**

What's new?

Version 2021

News Overview

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THE WORLD OF CAD AND PDM SOLUTIONS

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Basics

Discontinuations

Discontinuation of Windows® 7 and 8
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 2012, Windows Server 2008 R2 or older) will then no longer be supported either. Windows 8, too, is no longer supported.

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. Please also read the information given in the **Conversion of Old Itemisations** topic.

Discontinuation of "old" OpenGL versions
From HiCAD 2021 on, only OpenGL version 4.3 is used in all HiCAD modules. Until now this was only the case with the module **HiCAD Point Cloud**. This means that HiCAD 2021 can only be run on computers whose graphics card supports OpenGL Version 4.3 or higher. To avoid possible problems with onboard graphics cards, we recommend using a stand-alone graphics card.

Discontinuation of the old figure format (FIG)
The following notes on FIG-FGA conversion are not necessary if HELiOS is used in conjunction with the HELiOS Vault Server.
Since HiCAD 2017 we support FGA as figure format (before that FIG). As of HiCAD/HELiOS 2021 or HELiOS 2021 as an update for HiCAD 2019/2020, we require that all figures stored using HELiOS have been converted to the new format. For the conversion of existing 2-D FIG files, the tool **Converter_FIG_To_FGA.exe** is available in the exe directory of the HiCAD installation.
If there are still unconverted FIG files in the HELiOS document stock at the time of the database update to HELiOS 2600.0, you will be notified of the outstanding conversion of these data before the database update. In this case, the conversion must be carried out before or at the latest directly after the update using **Converter_FIG_To_FGA.exe**.

Discontinuation of the "old" Create detail drawing function
With HiCAD 2012, the previously valid functionality of workshop drawing in Steel Engineering has been expanded to a function for general drawing derivation. The former functions for detail drawing creation in Steel Engineering were previously still available in the **Drawing** menu under **Detail drawing** . As of HiCAD 2022 (version 2700.0), these functions are no longer supported.

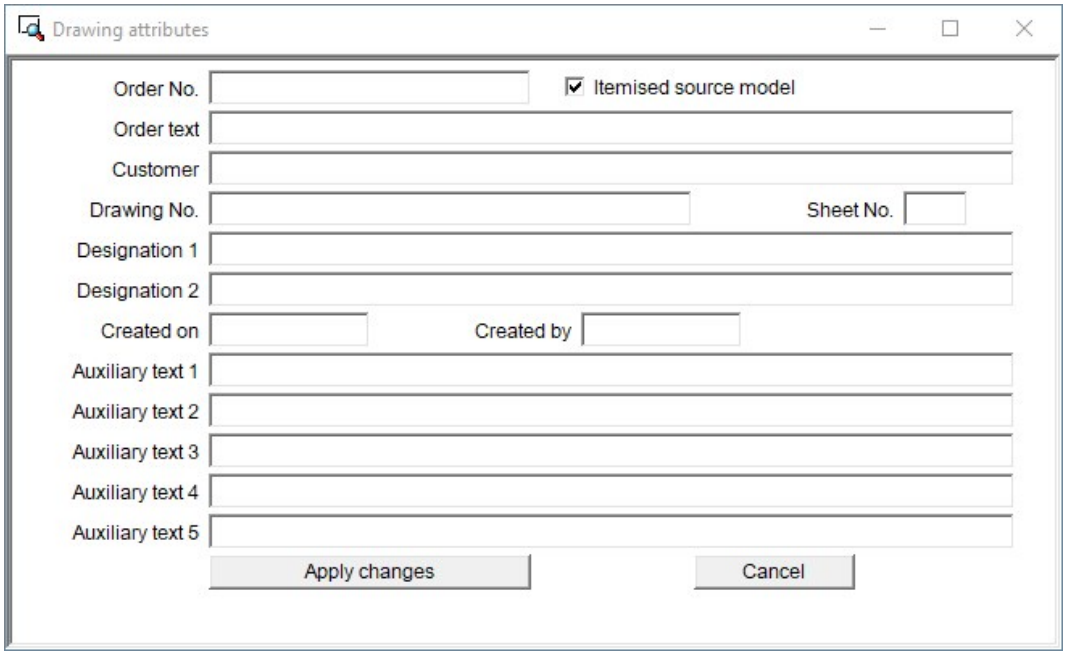
Discontinuation of length dimensions deviating from mm
HiCAD 2021 only supports constructions with the unit of measurement mm. The unit of measurement is saved in the corresponding SZA, FGA or KRA file when saving a drawing or a part. Attempts to open drawings with a different unit of measurement or to insert parts with a unit of measurement different from mm are blocked.

Discontinuation of HELiOS 32-Bit and HiCAD Viewer 32-Bit
As of HELiOS 2022 (Version 2700.0), no 32-Bit version will be available for HELiOS and the HiCAD Viewer. However, the interface to 32-Bit applications such as Office will still be possible and is not affected by the discontinuation of the 32-Bit installation of HELiOS.

Service Pack 1 2021 (V 2601)

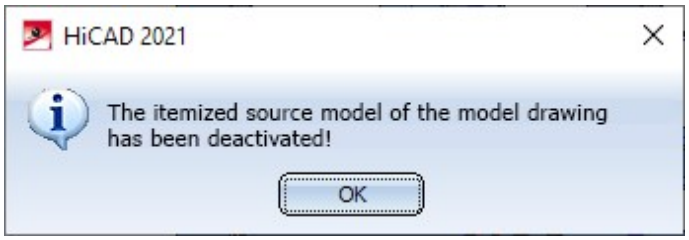
Itemised source models

New in the **Drawing attributes** dialogue window is the checkbox**Itemised source model**.



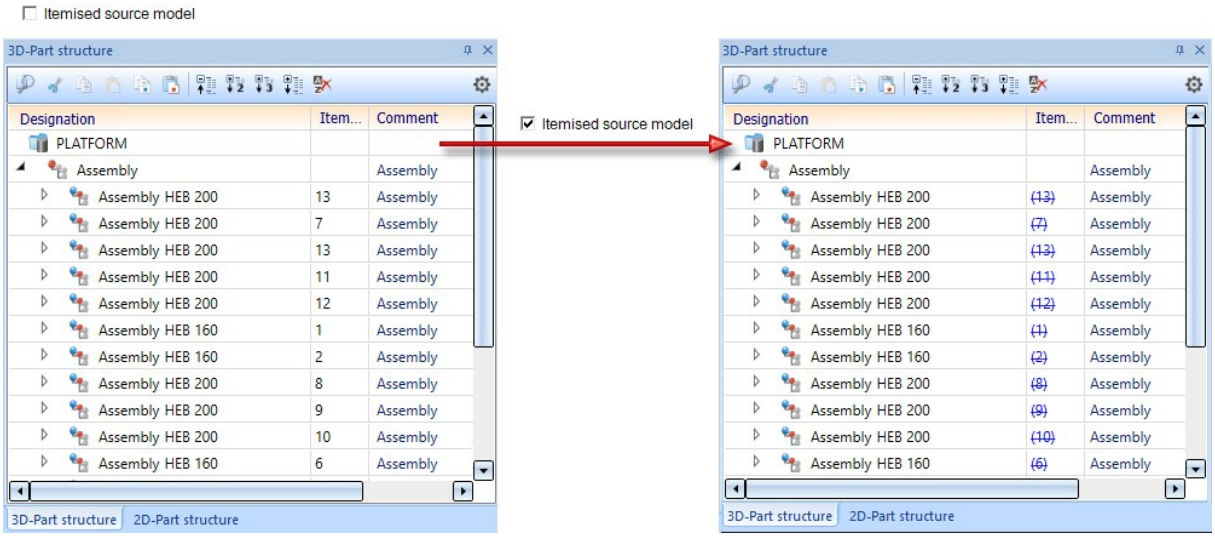
This checkbox (Attribute #PMOD) determines whether a model drawing is an itemised source model or not. An itemised source model is a model drawing in which itemisation is allowed. The ISD default setting is that new model drawings are always itemised source models, i.e. the checkbox is active by default.

Model drawings that are not itemised source models cannot be edited with the **Automatic itemisation** functions. This also applies to deleting any existing item numbers. If an **Automatic itemisation** function is called, the following message appears:



Deactivating the checkbox is useful, for example, if you want to disable itemisation in model drawings when passing them on to third parties.

If a model drawing with item numbers subsequently becomes an itemised source model by activating the checkbox, all existing item numbers become invalid.



The item numbers can then only be updated by performing a new itemisation.

Please note:

If you are not able to find the **Itemised source model** checkbox in your **Drawing attributes** window, e.g. after installing an update, you can insert it as follows:

- Close HiCAD.
- Open the **brw_szene.hdx** file, that is located in the HiCAD SYS directory, with an editor.
- There, insert the following line:

  <H>::TEXT=""::ATTR="#PMOD"::EDIT="YES"::ALIGN="RIGHT"::E_WIDTH="250"::E_LIMIT-T="250"::TYP="CHECKBOX"

  e. g.

```
HDB
# Konfigurationsdatei f. die Ausgabe der Konstruktionsattribute
# <D> - Dokumentenstamm
# <T> - Teielestamm
# <H> - HiCAD-Attribut

# --------------------------------
# Projektdaten
# --------------------------------
<H>::TEXT=""::ATTR="_SZNATTRS01"::EDIT="YES"::E_WIDTH="200"::E_LIMIT="200"::TYP="WCHAR"
<H>::TEXT=""::ATTR="#PMOD"::EDIT="YES"::ALIGN="RIGHT"::E_WIDTH="250"::E_LIMIT="250"::TYP="CHECKBOX"
<H>::TEXT=""::ATTR="_SZNATTRS02"::EDIT="YES"::E_WIDTH="500"::E_LIMIT="500"::TYP="WCHAR"
<H>::TEXT=""::ATTR="_SZNATTRS03"::EDIT="YES"::E_WIDTH="500"::E_LIMIT="500"::TYP="WCHAR"



# --------------------------------
# Zeichnungsdaten
# --------------------------------
<H>::TEXT=""::ATTR="_SZNATTRS04"::EDIT="YES"::E_WIDTH="300"::E_LIMIT="300"::TYP="WCHAR"
<H>::TEXT=""::ATTR="_SZNATTRS05"::EDIT="YES"::E_WIDTH="40"::E_LIMIT="40"::E_POS="RIGHTTOP"::TYP="WCHAR"
<H>::TEXT=""::ATTR="_SZNATTRS06"::EDIT="YES"::E_WIDTH="500"::E_LIMIT="500"::TYP="WCHAR"
<H>::TEXT=""::ATTR="_SZNATTRS07"::EDIT="YES"::E_WIDTH="500"::E_LIMIT="500"::TYP="WCHAR"
<H>::TEXT=""::ATTR="_SZNATTRS09"::EDIT="YES"::E_WIDTH="100"::E_LIMIT="100"::TYP="WCHAR"
<H>::TEXT=""::ATTR="_SZNATTRS08"::EDIT="YES"::E_WIDTH="100"::E_LIMIT="100"::E_POS="RIGHTTOP"::TYP="WCHAR"

<H>::TEXT=""::ATTR="_SZNATTRS10"::EDIT="YES"::E_WIDTH="500"::E_LIMIT="500"::TYP="WCHAR"
<H>::TEXT=""::ATTR="_SZNATTRS11"::EDIT="YES"::E_WIDTH="500"::E_LIMIT="500"::TYP="WCHAR"
<H>::TEXT=""::ATTR="_SZNATTRS12"::EDIT="YES"::E_WIDTH="500"::E_LIMIT="500"::TYP="WCHAR"
<H>::TEXT=""::ATTR="_SZNATTRS13"::EDIT="YES"::E_WIDTH="500"::E_LIMIT="500"::TYP="WCHAR"
<H>::TEXT=""::ATTR="_SZNATTRS14"::EDIT="YES"::E_WIDTH="500"::E_LIMIT="500"::TYP="WCHAR"

```

- Save the file.
- After you restart HiCAD, the checkbox will be available.

When working with the **Drawing Management**, the checkbox **Itemised source model** is not available. Use the ITEMISATIONMODEL attribute there for this purpose.

Toggle visibility of dimensions

In the **Visualisation** toolbar, the previous function "Toggle visibility of dimensions" has been renamed **Toggle visibility of drawing dimensions**. With this function, as of SP1, only all 2-D and 3-D drawing dimensions of the model drawing can be hidden/shown with one click. This no longer applies to parameter dimensions!

Import of HASCO and STRACK standard parts

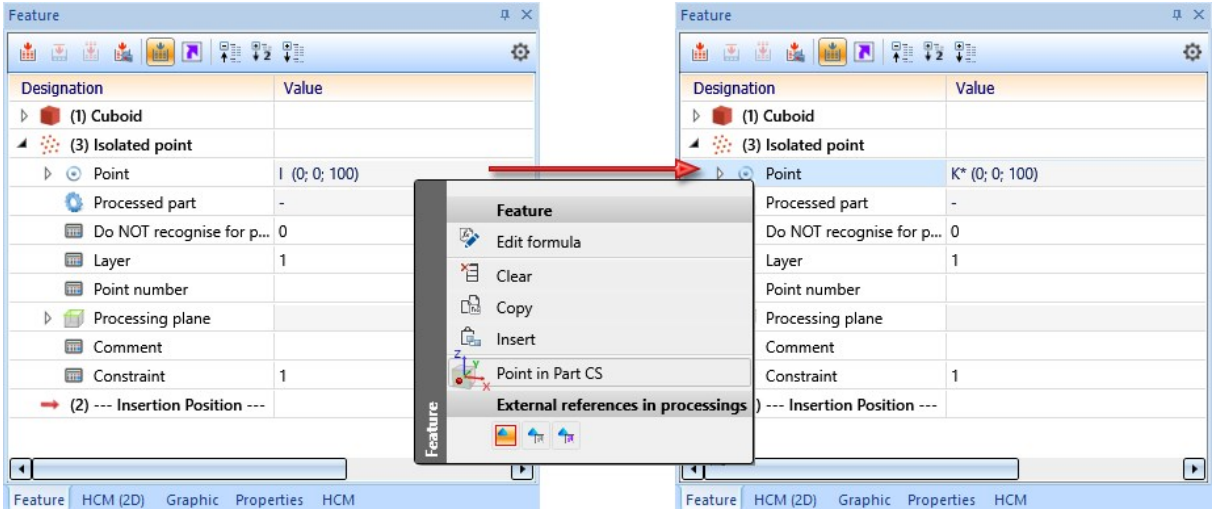
The functions for inserting HASCO and STRACK standard parts have been removed from the HiCAD user interface. The insertion of these standard parts is now only possible via the corresponding macros in the HiCAD sub-directory MAKRO2D:

- 2d3dteil_einfueg1hasco.mac
- 2d3dteil_einfueg1strack.mac

As of HiCAD 2022, importing via the macros and via the API will also no longer be possible!

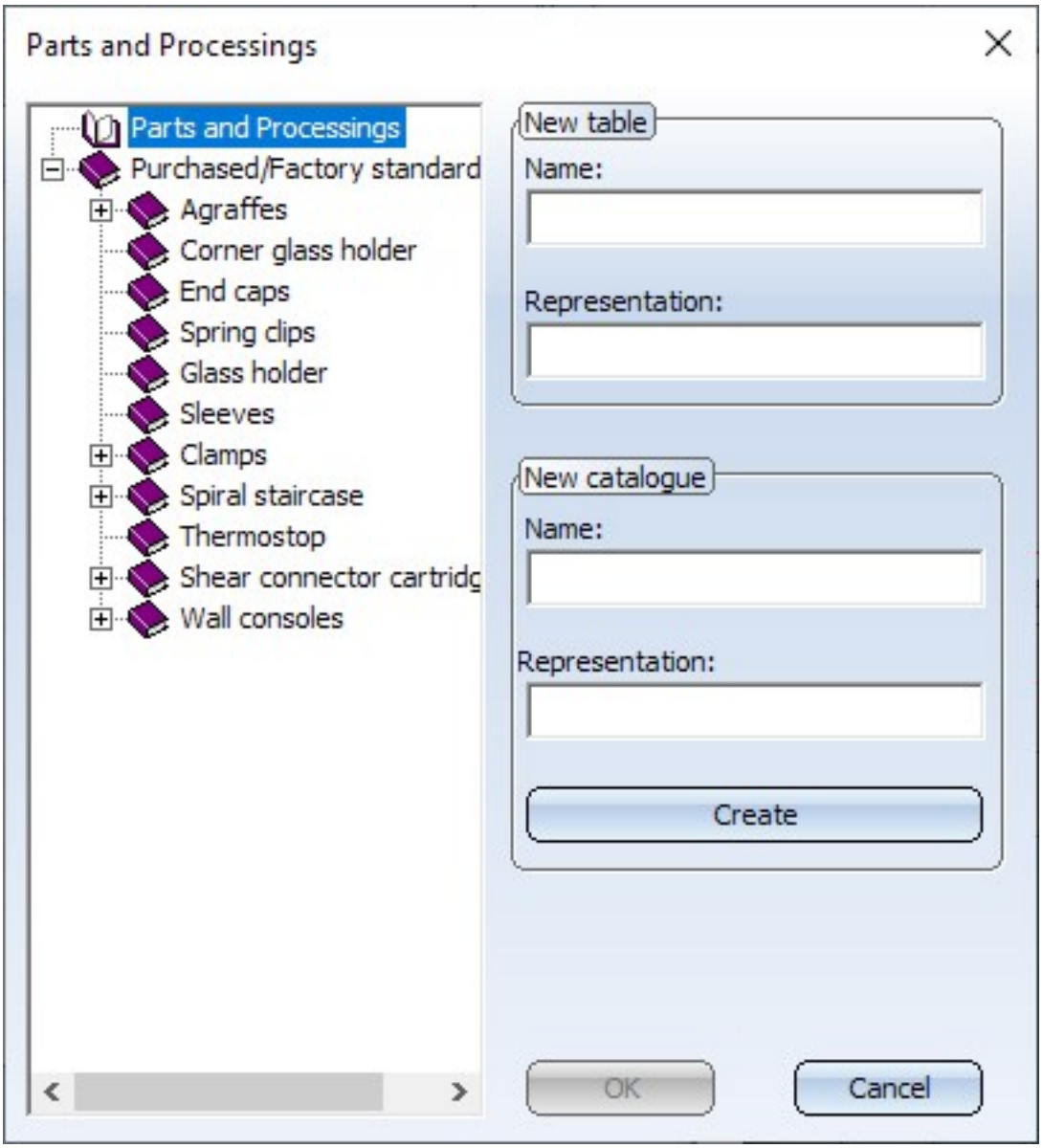
Point in Part CS

With the point option **Point in Part CS** the coordinates of a point are described by the coordinates in the part coordinate system. This means that the coordinates of the point are converted if necessary. This function is only available in the feature log and there in the context menu for points.



Save parts as variants

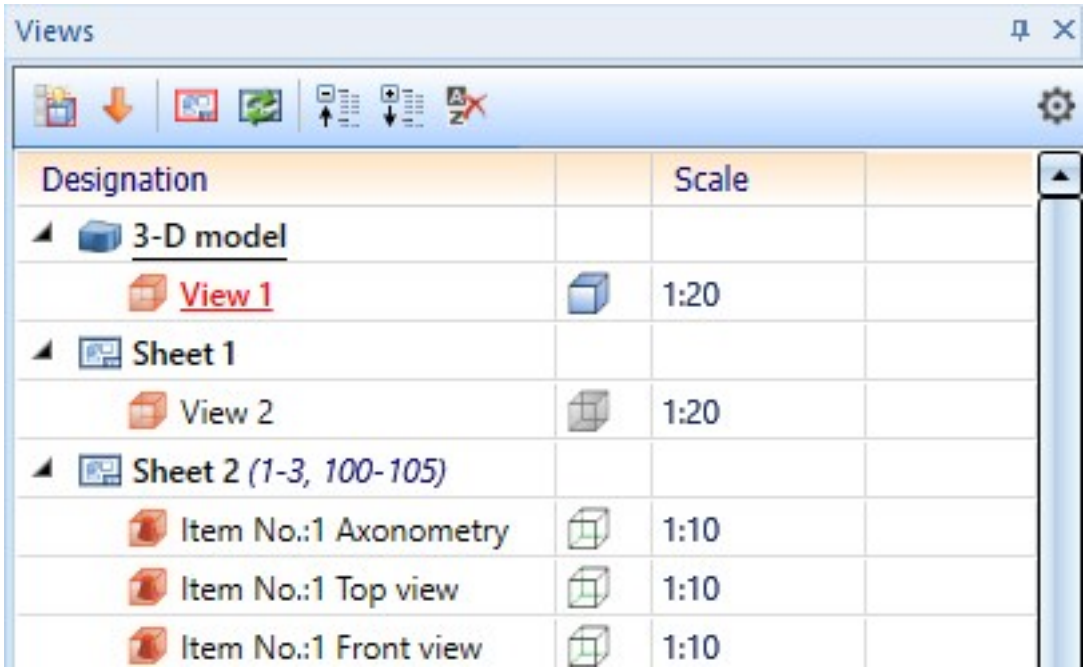
When saving parts as variants with a catalogue entry, from HiCAD 2021 SP1 it is also possible to store them in the catalogue **Factory standards > Purchased/Factory standard parts**.



Derived drawings

Drawing target - Existing model drawing

If the model drawing (**To existing drawing**) is selected as the drawing target for derived drawings, the workshop or detail drawings are created in the sheet areas of the current model drawing. The item number of the part with detail drawing on the sheet will appear in the sheet name. If more than one part in the sheet area has a detail drawing, the sheet name will contain a list of item numbers, e.g. 100-105 or 101,103,109 etc.



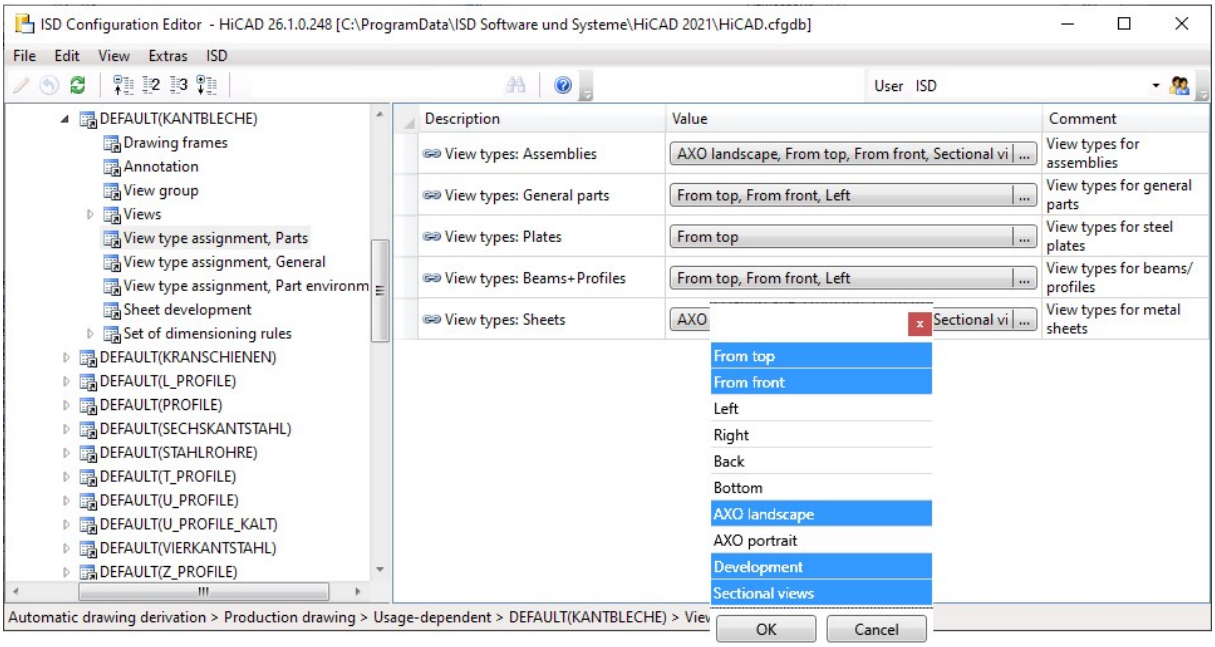
Please note:

- Drawing sheet names assigned by you will be replaced without any further confirmation prompt.
- The sheet name is automatically assigned as described above. This behaviour is not configurable.

Views of Sheet Metal parts

In the default settings for sheet metal parts in the drawing derivation, the view from the left has been deactivated. This affects the setting in the Configuration Editor at **Automatic drawing derivation > Production drawing > Usage-dependent > DEFAULT(KANTBLECHE) > View types: Sheets**.

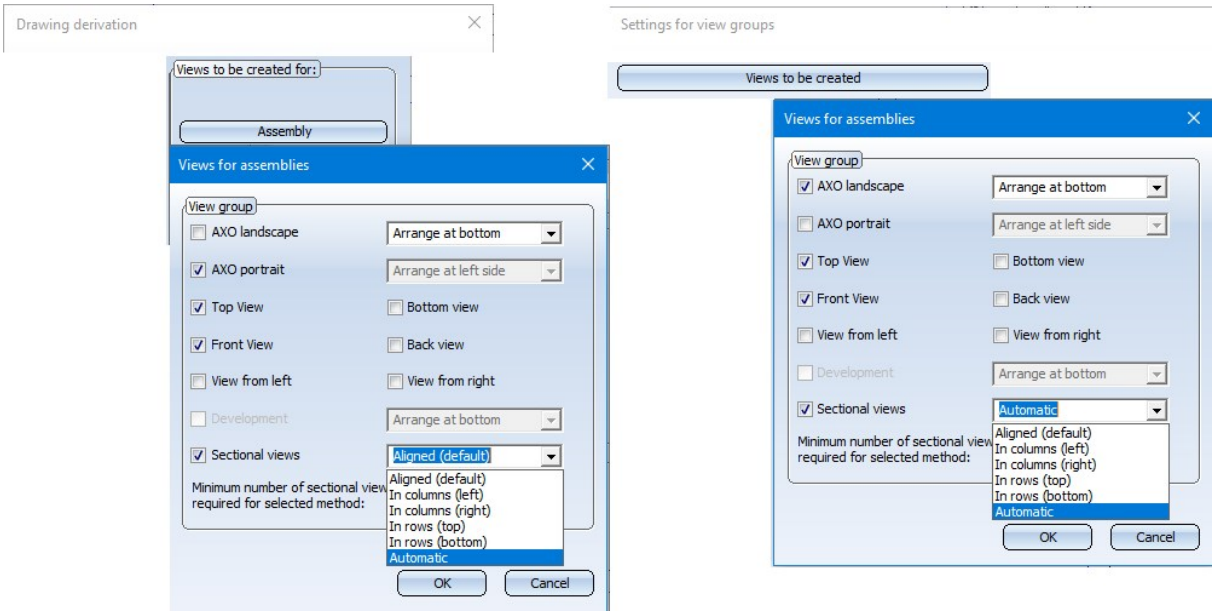
This means that the view from the left is not automatically generated for sheet metal parts during drawing derivation.



Optimised arrangement of sectional views

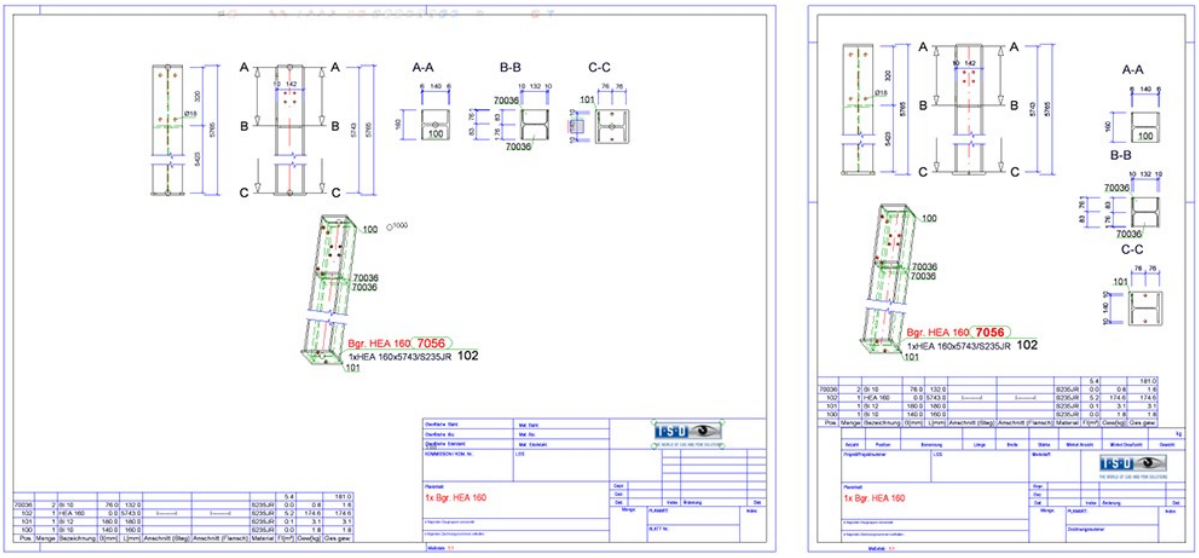
As of HiCAD 2021 SP1, the option **Automatic** is also available for the arrangement of sectional views in the drawing derivation. This affects

- the settings for **Drawing derivation** under **Views to be created for...** and
- the settings for views in the function **Change settings for active view group** .



If this setting is selected, HiCAD tries to place the section views in such a way that the display is as clear as possible. The following aspects are taken into account:

- An aligned display is preferred.
- If the sectional views are not aligned, they are placed as far to the right as possible.
- The distances between the views are chosen in such a way that the representation fills as much of the surface as possible.

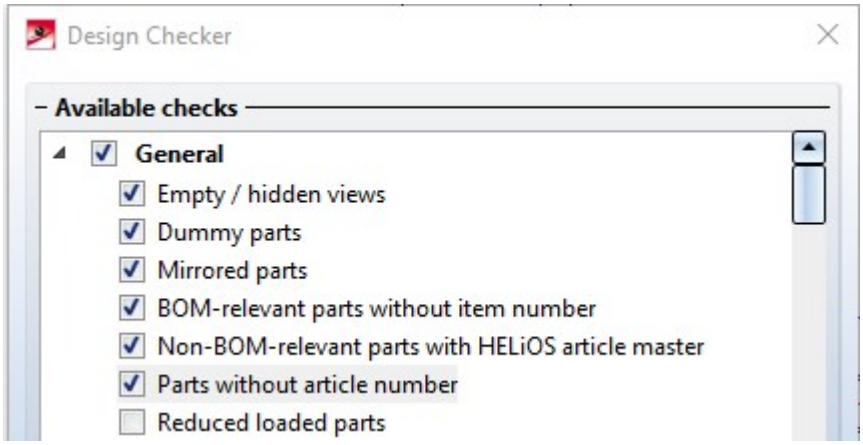


Left: Previous versions, Right: From HiCAD2021 SP1

Design Checker

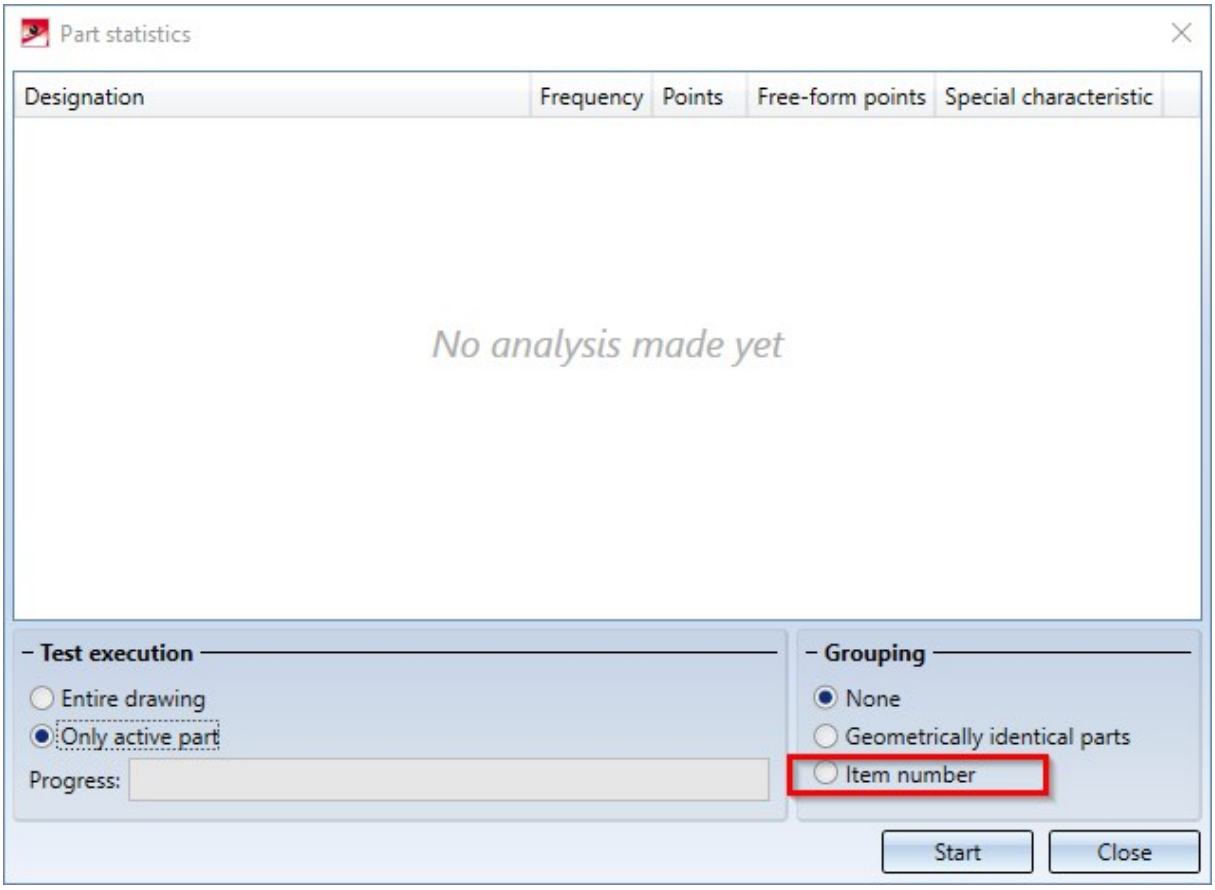
In the **General** section the following checks have been renamed as follows:

- Parts without name → Parts without article number
- Data model → Reduced loaded parts

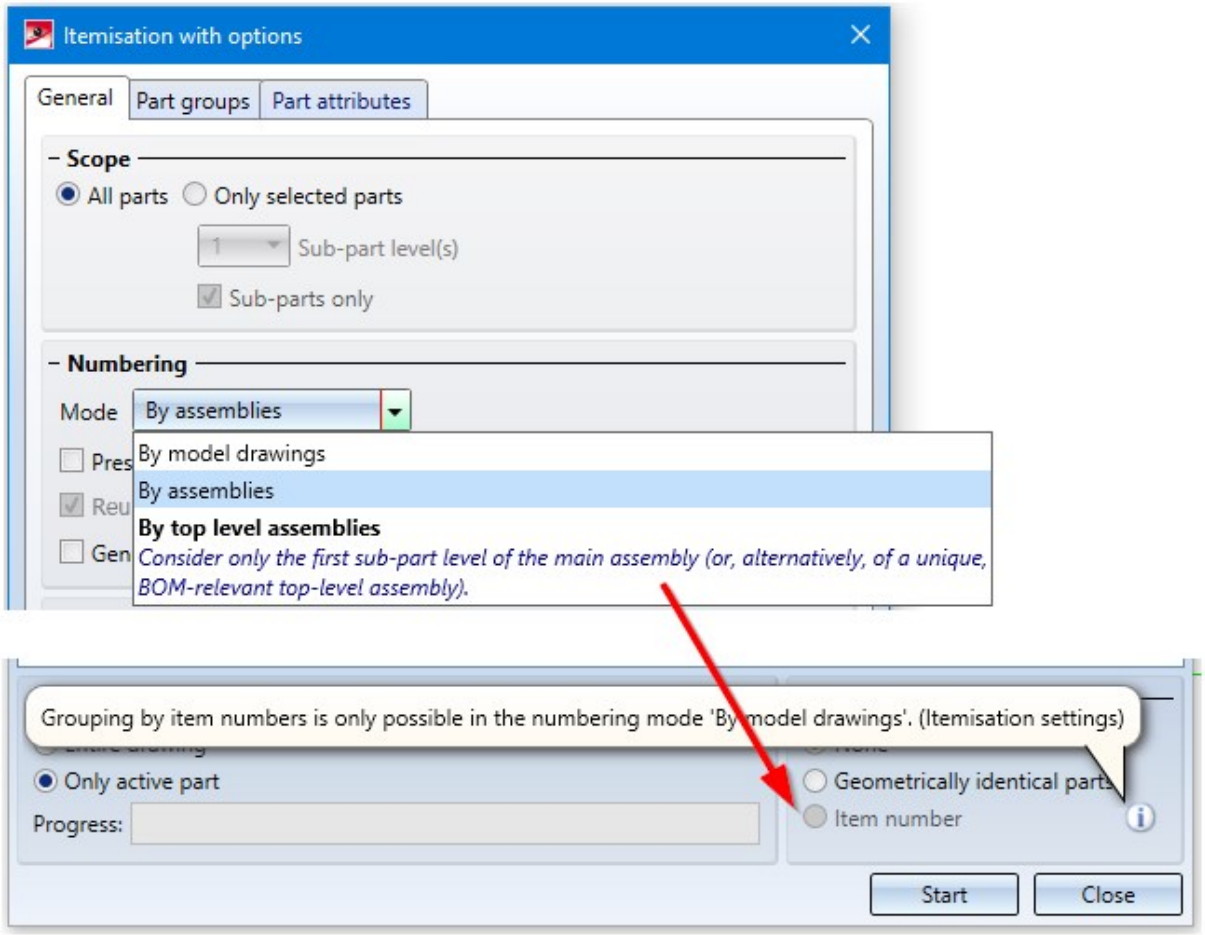


Part statistics

As of SP1, it is also possible to group the result list by item numbers in the function **Part statistics** .



However, grouping by item numbers is only supported if the itemisation was done in the **By model drawings** mode. Otherwise the option **Item number** is greyed out.



Drawing frames / Title blocks

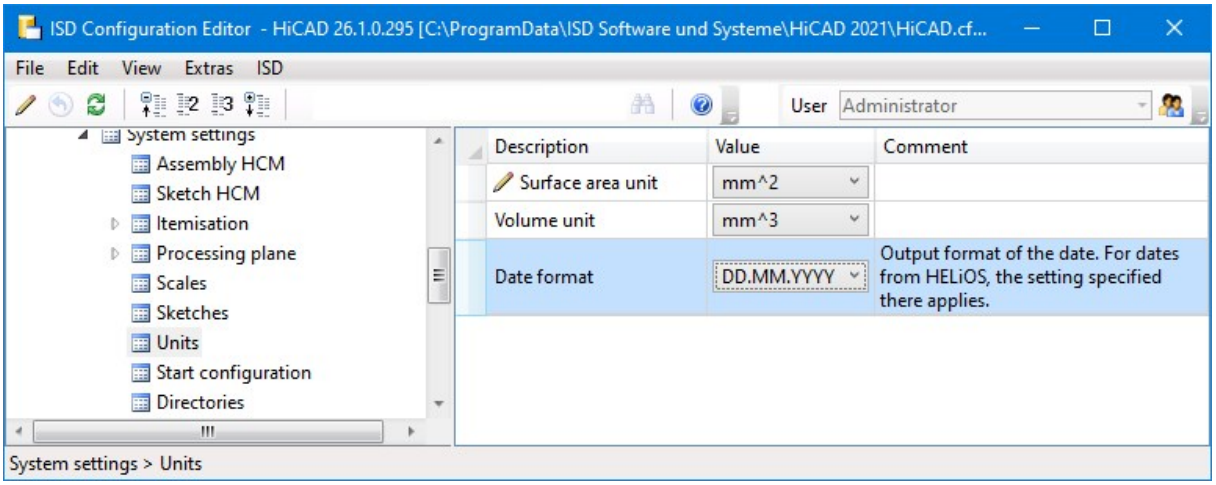
Drawing sheet name

In addition to the drawing sheet number, the drawing sheet name can now also be output in the title block of the drawing frames. The new special attribute $BLATTNAME is available for this purpose. This attribute is taken into account when filling in the title block with the function **Complete title block, with attributes** or when filling in the title block via the HELiOS. For this purpose, the file **SchrFe_ausAttr.dat** in the HiCAD sys directory must be adapted accordingly.

Output	Indication in SchrFe_ausAttr.dat		Variable in HiCAD title block
	Attribute	Title block text	
Name of the current drawing sheet	$BLATTNAME	HIC_Blattname	@*@20,D.HIC_Blattname@*@

Date / Print date

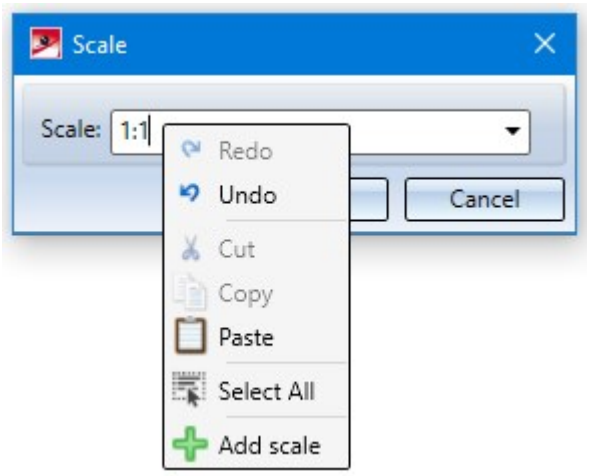
- When outputting the print date via the variable @PLOD@, the **Date format** set in Configuration Editor at **System settings > Units** is taken into account.



- The selected date format is also taken into account by the string function **DAT$**. This function is used when filling in the title block with the function **Complete title block (without DB)** (**Drawing > Extras >Tools**).

Manage drawing scales

The function **Manage drawing scales** can also be called directly within certain scale functions. To do this, right-click in the scale field in the dialogue of the corresponding function and select the function **Add scale** in the context menu.



In this way, further individual scales can be defined for the current model drawing without having to leave the current function. This is possible for the following functions:

Main scale

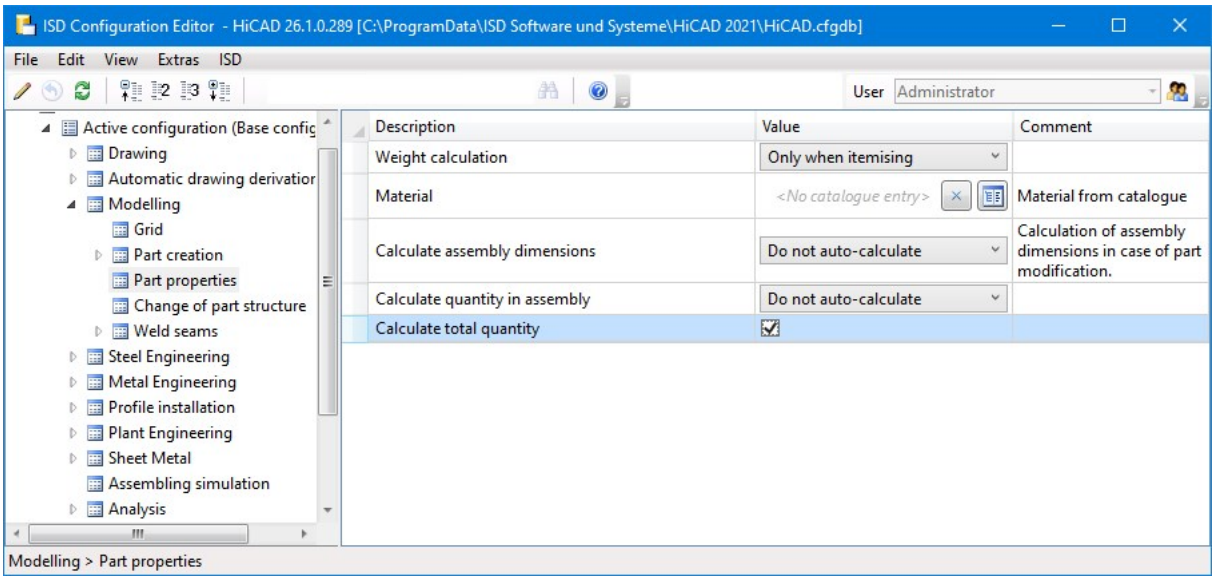
Change view scale

2-D part scale and

Detail view, Sphere/Cuboid.

Attribute "Total quantity"

New in the Configuration Editor at **Modelling > Part properties** is the checkbox **Calculate total quantity**.

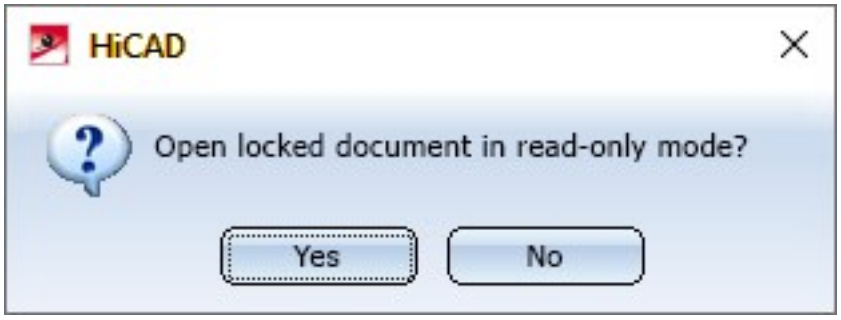


If the checkbox is active, the number of times each part appears in the model drawing is determined during itemization (only in itemization mode **By model drawings**). This value is assigned to the attribute **Total quantity (%06)** and automatically updated when editing the parts, e.g. when deleting or repeating parts. In very large constructions, this update can have a negative impact on performance, for example when deleting assemblies. In such cases, the checkbox can be deactivated. If this is the case, then - after a restart of HiCAD - the next time the drawing is loaded, the attribute **Total number** of the affected parts will be deleted instead of calculated.

Opening write-protected files via the Vault Server

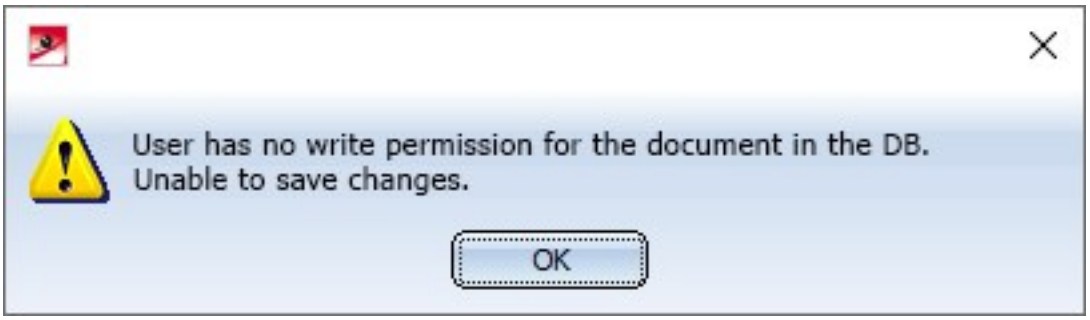
If an SZA file is opened via the Vault Server, which is locked there for writing, then it cannot be saved later either.

When opening such a file, HiCAD displays a corresponding warning message:



If you select **Yes**, the file will be opened in read mode. If you select **No**, the opening of the file will be cancelled.

If an attempt is made to save a file opened in read mode, the following message appears:



Major Release 2021 (V 2600)

The new Renderer - Update to OpenGL Version 4.3

HiCAD uses OpenGL for graphics output - up to now in version 2.0. From HiCAD 2021, only OpenGL version 4.3 will be used in all modules. Up to now, this only applied to 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 that you no longer use these as of HiCAD 2021, as the performance of these chips is insufficient for complex designs.

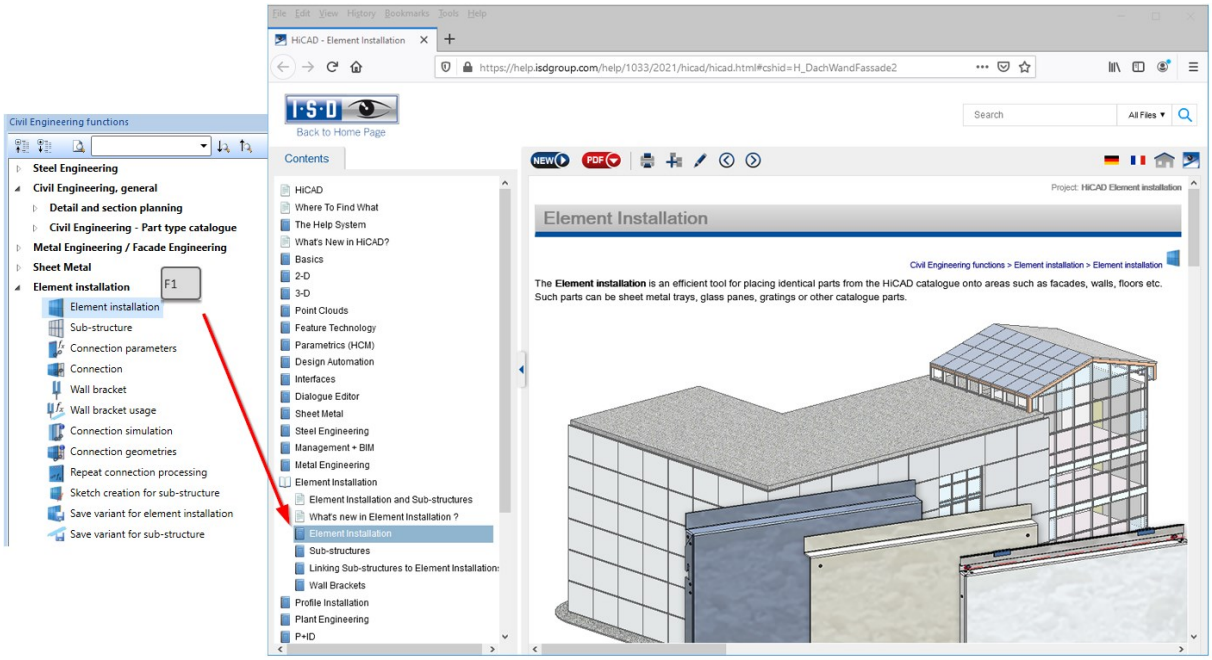
Decisive factors in the development of the new renderer were the extended range of functions in graphics output through the use of newer OpenGL functions and the possibility of being able to realise noticeable performance improvements in graphics output in the future. With HiCAD 2021, graphics output is accelerated in width, 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. The full performance gain will be achieved with HiCAD 2021 SP1.

Licensing

From HiCAD 2021 onwards, the HiCAD Basic module **Education Edition Premium** for schools and teaching institutions also contains the Extension module **HiCAD Point Cloud**.

Direct Help

The direct help for the functions of the **Civil Engineering functions** docking window can now also be called up with **F1**.

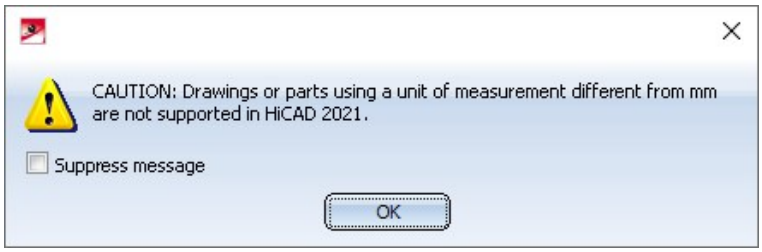


Scale and unit of measurement

Unit of measurement in the drawing

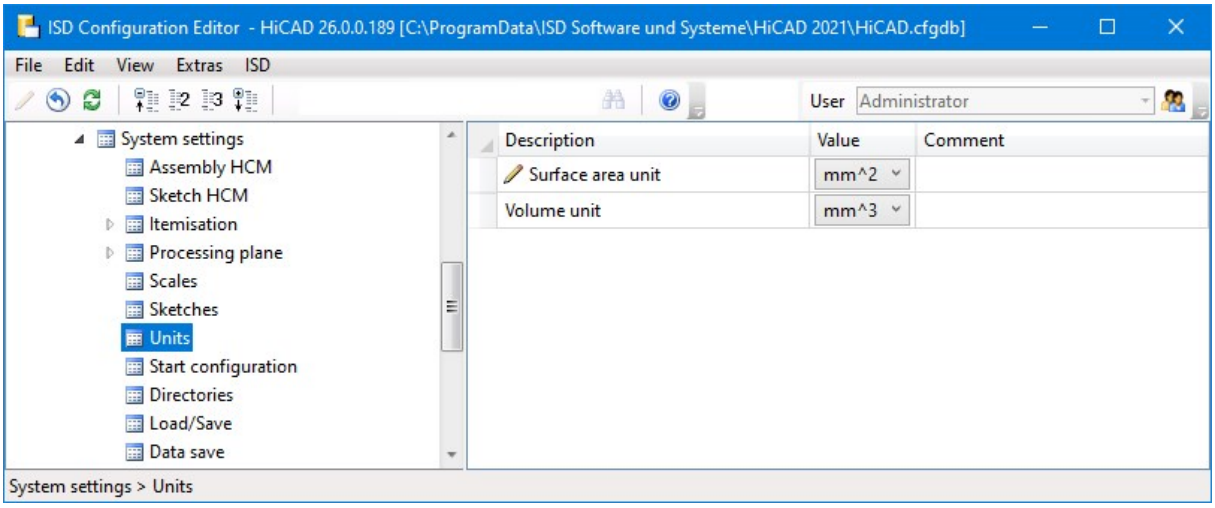
HiCAD 2021 only supports drawings with the unit of measurement mm. The previous function "Unit of measurement" is no longer available. This also applies to the parameter **Unit of measurement** in the Configuration Editor at **System settings > Miscellaneous**.

The unit of measurement is saved in the corresponding SZA or KRA file when saving a drawing or a 3-D part. If you try to open constructions with a different unit of measurement or insert parts with a unit of measurement different from mm, an error message is displayed:



Surface area and volume unit

In addition to the unit of measurement of the model drawing, it is possible to specify for the surface areas (Attribute §10) and volumes (Attribute §20) in which unit of measurement they are to be output in the ICN, in the attribute masks, in the HiCAD Viewer, in bills of materials and in the transfer to HELiOS. The default setting for surface areas is square metre m^2, for volumes cubic millimetre mm^3.



For model drawings and parts created with an earlier HiCAD version (before HiCAD 2021), you can specify in the Configuration Editor at **System settings > Miscellaneous** how to proceed when opening such model drawings and parts for surface area and volume calculation.

The ISD default setting for surface areas is square metres, for volume **Active unit**, i.e. cubic millimetres).

Units of measurement, Scale, Tolerance		
Angle unit	deg.	
Unit of measurement	mm	
Scale 1 :	1	
Relative tolerance	0.001	Referring to unit of measurement; >= 1.E-5; <= 0.1
Unit of measurement of areas when loading old drawing files (before HiCAD 2021)	m**2	
Unit of measurement of volume when loading old drawing files (before HiCAD 2021)	Active unit	

According to our experience in discussions with customers, people usually work with the default settings provided by the ISD, so that in the vast majority of cases nothing has to be changed in the settings when loading designs from earlier HiCAD versions.

If you have worked with changed default settings for surface areas and volumes in a previous HiCAD version, then these will be adopted during an update installation to HiCAD 2021 at **System settings > Miscellaneous**, so that you do not have to change anything in this case either.

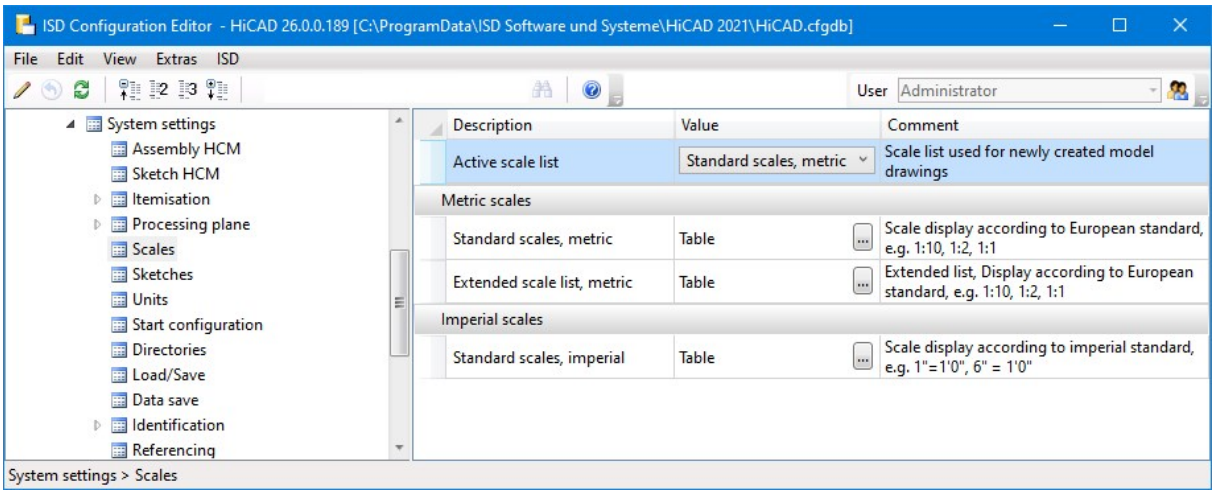
If you load model drawings from third parties that were created with an earlier HiCAD version, then the settings used there for the unit of measurement of surface areas and volumes must be known and adjusted at **System settings > Miscellaneous** if necessary, as otherwise deviations may occur. For example, if the setting used for model drawing differs from m^2 then internally all surface attributes will be scaled to m^2 and then scaled back to the display unit in the display.

An example:

With an earlier HiCAD version a cuboid was constructed with a side length of 100 mm. The unit of measurement for the surface area was cm^2. If you load this model drawing with the default settings shown above, 600 is displayed as the surface area of the cuboid, i.e. 600 m^2. This is of course not desired. In this case, at **System settings > Miscellaneous** the parameter **Unit of measurement for surface areas...** to cm^2 so that the display in HiCAD shows the correct value of 0,06 m^2.

Scale lists

HiCAD supports both metric scales commonly used in Europe and imperial scales used in North America. Which scales are available for selection in HiCAD is defined in the Configuration Editor at **System settings > Scales**. There you can define different **Scale lists** and specify which of these lists should be the active scale list. This list will then be used by default for new model drawings in HiCAD.



Three scale lists are predefined by the ISD:

- Standard scales, metric (1)
- Extended scales, metric (2) and
- Standard scales, imperial (3)

(1)

Scale factor	Displayed text
1:200	1:200
1:100	1:100
1:50	1:50
1:20	1:20
1:10	1:10
1:5	1:5
1:2.5	1:2.5
1:2	1:2
1:1	1:1
2:1	2:1
5:1	5:1
10:1	10:1
20:1	20:1
50:1	50:1
100:1	100:1

(2)

Scale factor	Displayed text
1:1	1:1
1:2	1:2
1:2.5	1:2.5
1:5	1:5
1:7.5	1:7.5
1:10	1:10
1:20	1:20
1:40	1:40
1:50	1:50
1:100	1:100
1:200	1:200
2:1	2:1
5:1	5:1
10:1	10:1
50:1	50:1

(3)

Scale factor	Displayed text
1:1536	1/128" = 1'-0"
1:768	1/64" = 1'-0"
1:384	1/32" = 1'-0"
1:192	1/16" = 1'-0"
1:128	3/32" = 1'-0"
1:100	1:100
1:96	1/8" = 1'-0"
1:64	3/16" = 1'-0"
1:48	1/4" = 1'-0"
1:32	3/8" = 1'-0"
1:24	1/2" = 1'-0"
1:16	3/4" = 1'-0"
1:12	1" = 1'-0"
1:8	1-1/2" = 1'-0"
1:4	3" = 1'-0"
1:2	6" = 1'-0"
1:1	1'-0" = 1'-0"
2:1	2:1
4:1	4:1
8:1	8:1
10:1	10:1
100:1	100:1

The default active scale list is **Standard scales, metric**.

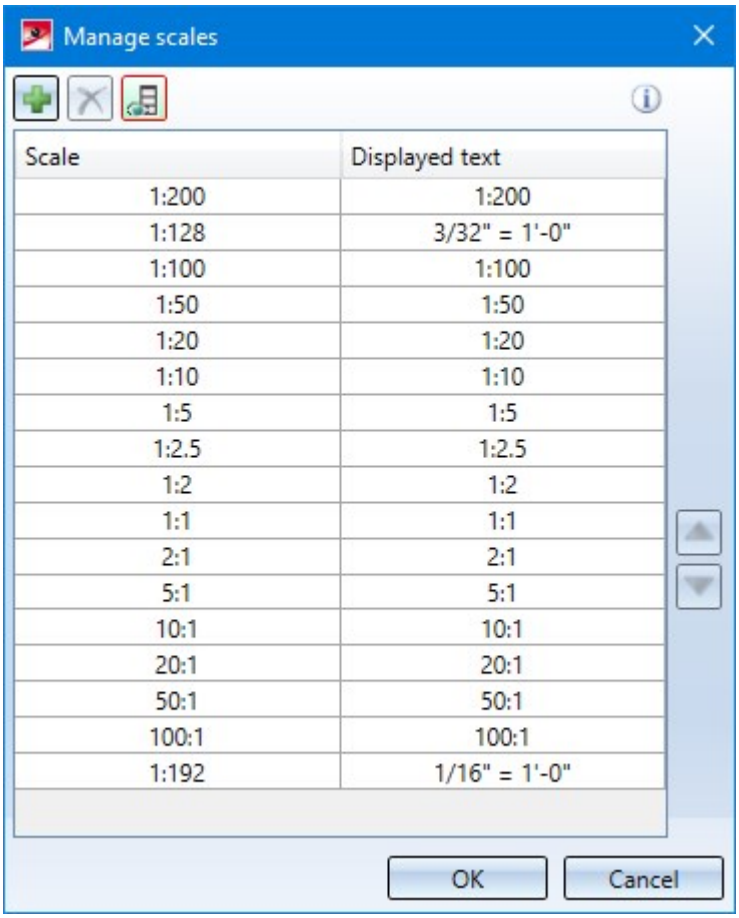
Please note:
The predefined scale lists can be edited and additional scale lists can be defined. You can find out how this works in the **Create and Edit Scale Lists** topic.

Manage drawing scales

New at **Drawing > Properties > Scale** > ... is the function **Manage drawing scales**.

With this function you can change the active scale list for the current drawing. The changed scale list is then only valid for the current drawing and is saved together with the drawing.

After calling the function the dialogue window **Manage drawing scales** is displayed.



The column **Scale** describes the mapping ratio. The column **Displayed text** contains the text that is displayed in the selection box for scales in the various functions instead of the corresponding scale. This text may differ from the text in the **Scale** column. In this way, country-specific designations for scales can be stored - as shown in the image above.

The order of display in the selection boxes can be changed with the arrow keys on the left of the dialogue window.

Add scale

Inserts a new line. Then enter the scale and the display text. Please note that the scale must be unique. If this is not the case, the symbol appears in the corresponding row.

Remove scale

Removes the scale in the currently selected row.

Show/hide displayed text

Shows/hides the column with the displayed text.

If the dialogue window is closed with **OK**, the changed scale list is taken over for the current drawing and saved with the drawing.

Itemisation

Conversion of previous itemisation

As of HiCAD 2021, the mode "Itemisation to HiCAD 2017" is no longer available.

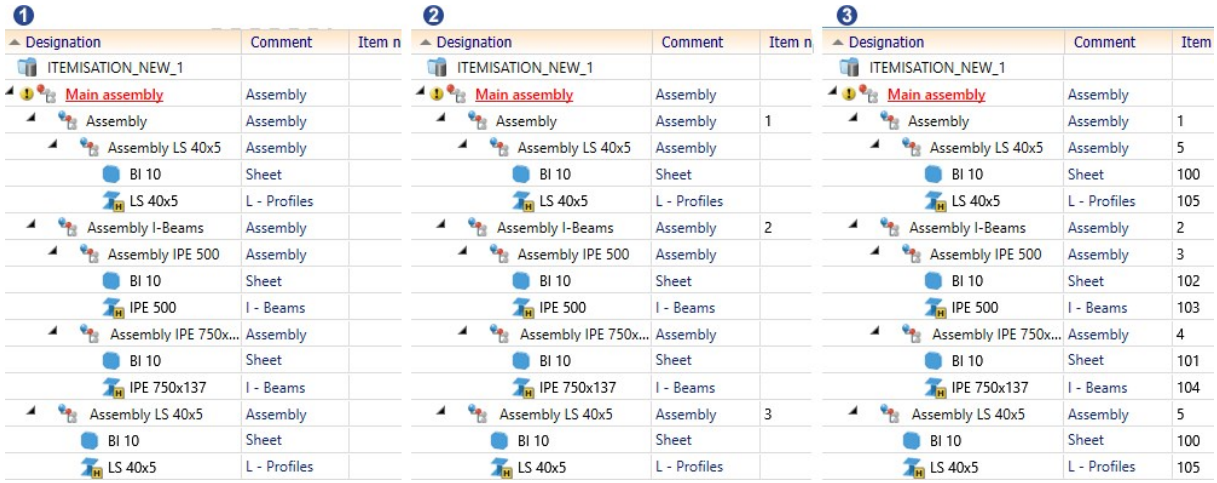
In drawings created with an earlier HiCAD version and positioned there - at least partially - with the old itemisation mode, the previous itemisation must be converted accordingly before using the itemisation functionality in HiCAD 2021. This can be done

- with the function Switch to standard itemisation used since HiCAD 2018 or
- directly when calling an itemisation function.

Itemisation by top level assemblies

In the Itemisation, with options dialogue box, the new mode **By top level assemblies** is available on the **General** tab under **Numbering**.

In this mode, only the first sub-level of the main assembly is taken into account. This mode is also possible if a unique, BOM-relevant top level assembly exists instead of the main assembly.



(1)

Designation	Comment	Item n
ITEMISATION_NEW_1		
Main assembly	Assembly	
Assembly	Assembly	
Assembly LS 40x5	Assembly	
Bl 10	Sheet	
LS 40x5	L - Profiles	
Assembly I-Beams	Assembly	
Assembly IPE 500	Assembly	
Bl 10	Sheet	
IPE 500	I - Beams	
Assembly IPE 750x...	Assembly	
Bl 10	Sheet	
IPE 750x137	I - Beams	
Assembly LS 40x5	Assembly	
Bl 10	Sheet	
LS 40x5	L - Profiles	

(2)

Designation	Comment	Item n
ITEMISATION_NEW_1		
Main assembly	Assembly	
Assembly	Assembly	1
Assembly LS 40x5	Assembly	
Bl 10	Sheet	
LS 40x5	L - Profiles	
Assembly I-Beams	Assembly	2
Assembly IPE 500	Assembly	
Bl 10	Sheet	
IPE 500	I - Beams	
Assembly IPE 750x...	Assembly	
Bl 10	Sheet	
IPE 750x137	I - Beams	
Assembly LS 40x5	Assembly	3
Bl 10	Sheet	
LS 40x5	L - Profiles	

(3)

Designation	Comment	Item
ITEMISATION_NEW_1		
Main assembly	Assembly	
Assembly	Assembly	1
Assembly LS 40x5	Assembly	5
Bl 10	Sheet	100
LS 40x5	L - Profiles	105
Assembly I-Beams	Assembly	2
Assembly IPE 500	Assembly	3
Bl 10	Sheet	102
IPE 500	I - Beams	103
Assembly IPE 750x...	Assembly	4
Bl 10	Sheet	101
IPE 750x137	I - Beams	104
Assembly LS 40x5	Assembly	5
Bl 10	Sheet	100
LS 40x5	L - Profiles	105

(1) Without item numbers, (2) Itemised by top level assemblies, (3) Itemised by model drawings.

Important:
If this mode is used when converting previous itemisation, then no subordinate assemblies will be converted.

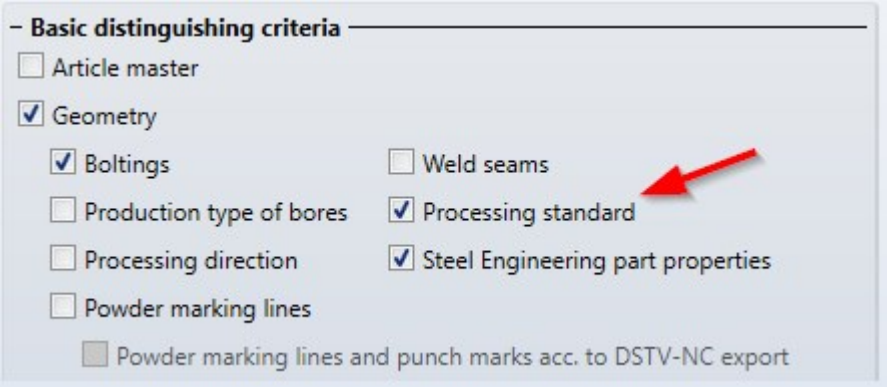
Itemisation by assemblies

Until now, assemblies were only considered "identical" when using the itemisation by assemblies if they were the same referenced part. From HiCAD 2021, they are also considered identical if they are geometrically identical. Geometrically identical means that the assemblies are also structurally identical and the attributes to be compared are identical on all levels. Any existing item numbers of sub-assemblies are not taken into account.

Identical part search - Processing standard

Bores - even those created as subtractions - are considered to be identical even if they have different representations. An exception are bores that are only represented as axes.

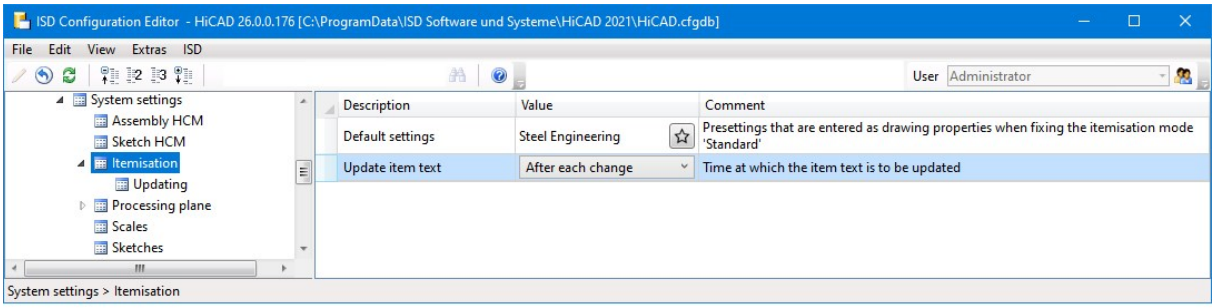
If holes and corresponding processings are to be considered the same only if their catalogue/table IDs match, you can achieve this by activating the new checkbox **Processing standard**.

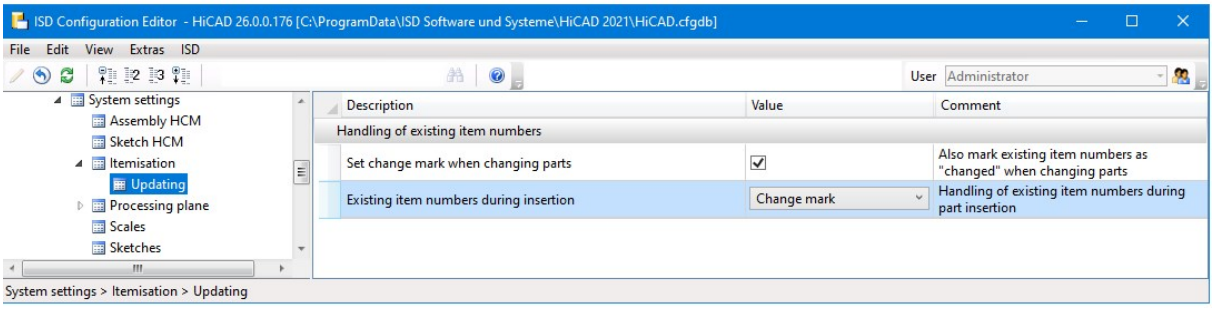


You can find an example here.

Parameters for itemisation in the Configuration Editor

From HiCAD 2021 on, the parameters for the default itemisation can be found in the Configuration Editor at **System settings > Itemisation**.





Article master, Itemisation

The function **Article master, Itemisation** for article master synchronisation during itemisation and saving only applies when using the **Management + BIM** module. For this reason, the function at **Drawing > Itemisation/Detailing > Attr...** > **Article master, Itemisation** has been moved to **Management + BIM > Production > Attr...** > **...**.

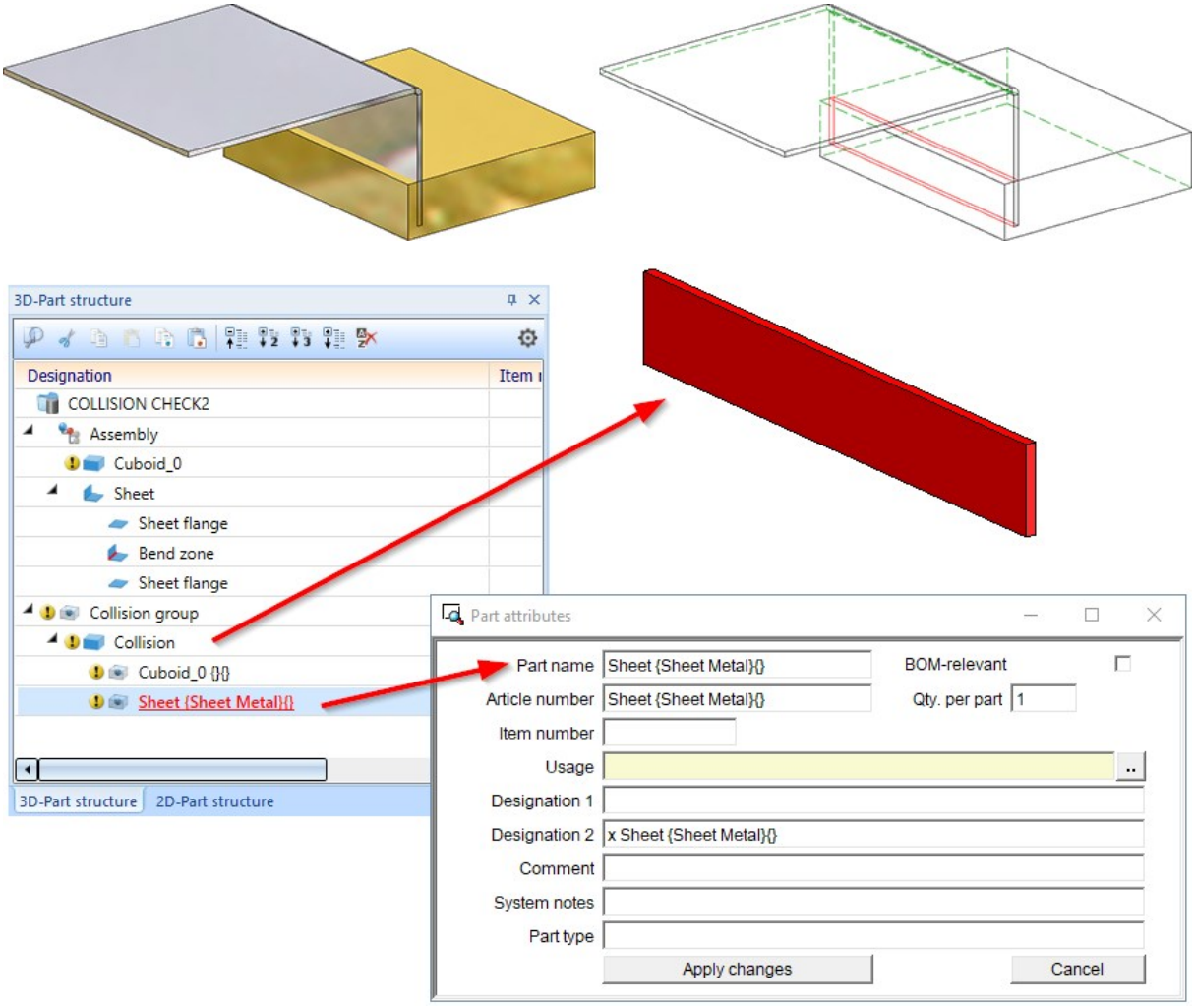
Collision check

The generation of collision parts via the **Collision check** functions has been slightly modified.

For each collision, a collision part called **Collision** of the type Solid is created, with the geometry of the collision. This contains the part "copies" of the two parts causing the collision. These copies are dummy parts that have no geometry and only limited part attributes. These two dummy parts serve with the names of the parts causing the collision. This serves to clarify the cause of the collision more quickly, since the collision part may be very small and often almost invisible visually. As a rule, the name of the source part displayed in the ICN, including article number and comment, is used as the part name and article number of these dummy parts. An exception are collision parts that are not "real" parts. In this case, the name of the superordinate part is used for the dummy part. All collision parts are combined under the name **Collision group**. This applies to threads, sheet metal flanges, bend zones and sub-parts of series beams and profiles.

Example

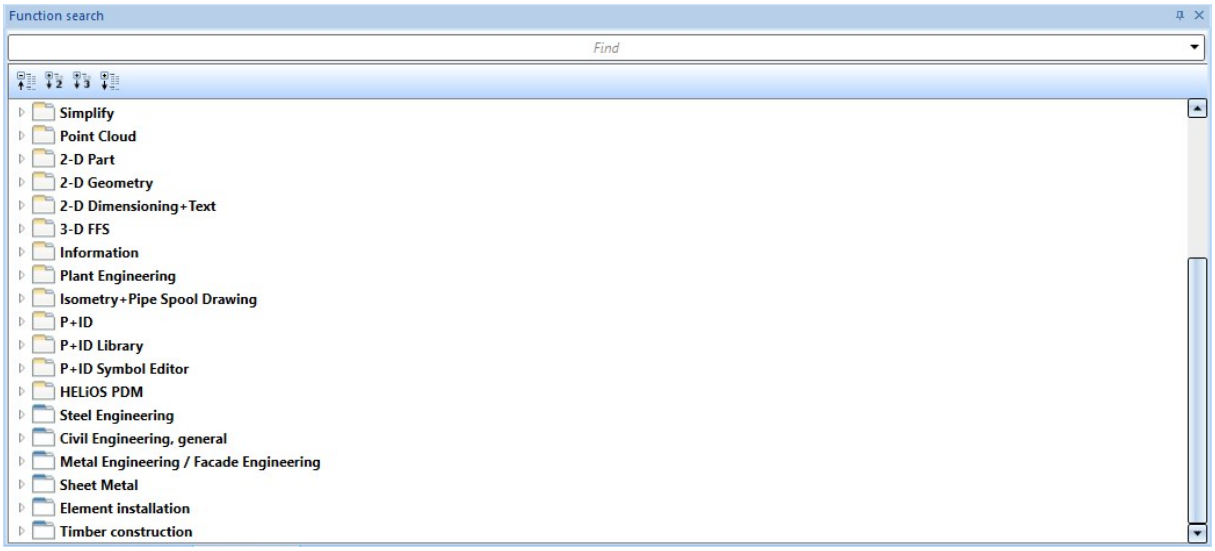
In the image below, the sheet metal part - or more precisely the flange - collides with the cuboid.



Function search

HiCAD offers a very extensive range of functions. Therefore, it can happen that one is looking for a certain function but does not know exactly on which Ribbon the function can be found. In such cases, the new **Function search** docking window can help.

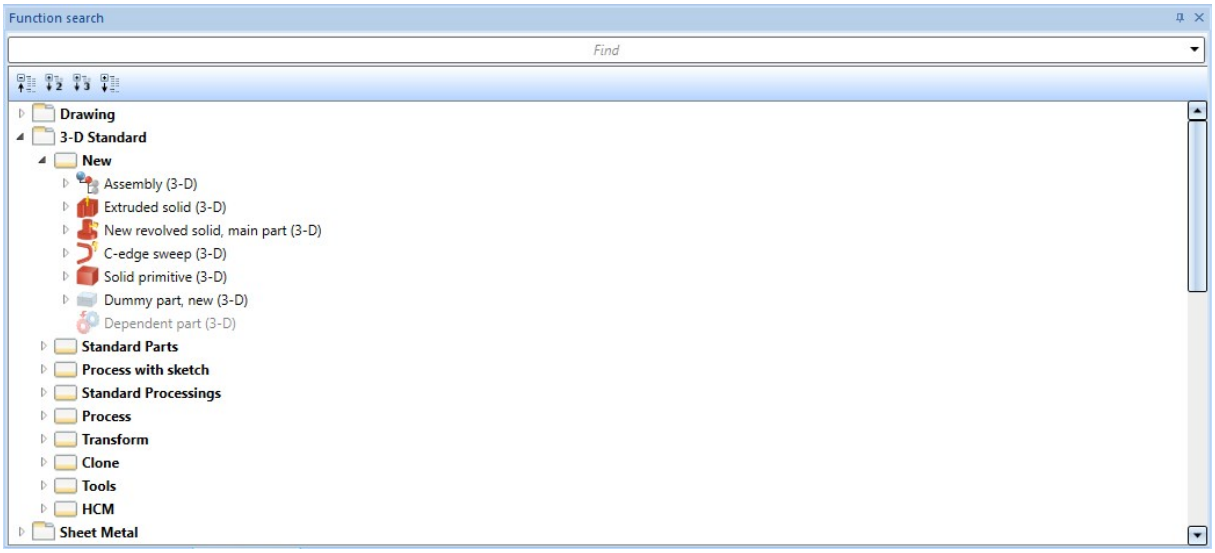
In the docking window, all Ribbon tabs and the function groups of the docking window **Civil Engineering functions** are listed.



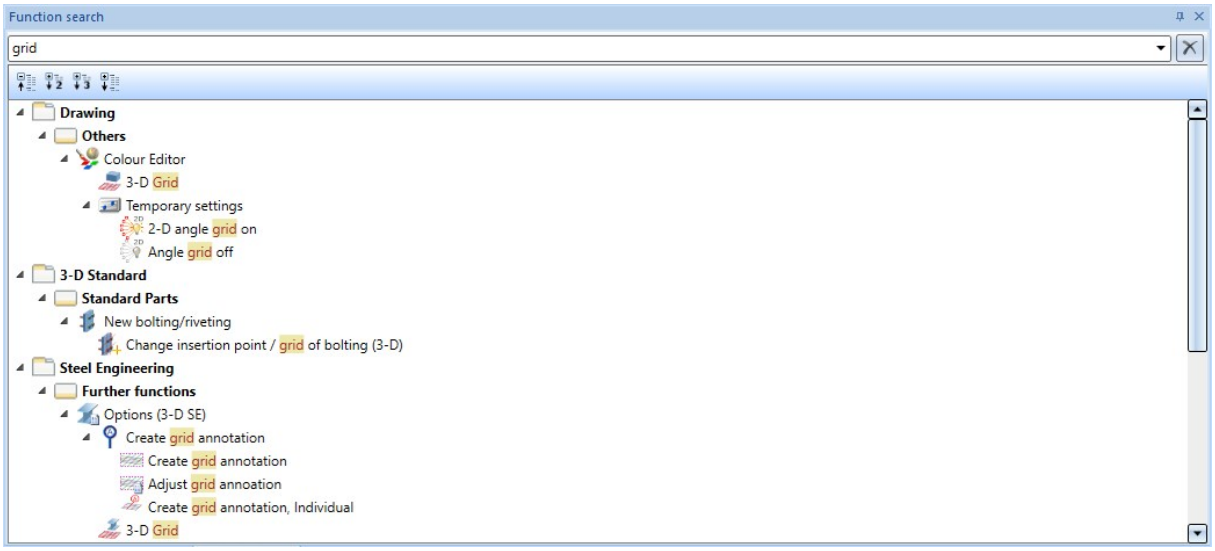
The display can be influenced with the toolbar functions as follows:

- collapses the display and shows only the first level of the Ribbons / Civil Engineering functions
- shows the first two levels of the Ribbons / Civil Engineering functions
- shows the first three levels of the Ribbons / Civil Engineering functions
- shows all levels of Ribbons / Civil Engineering functions

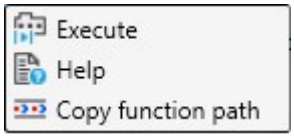
In addition, the display of individual Ribbons / Civil Engineering functions can be expanded further by clicking on a symbol, i.e. the function groups of a Ribbon or the functions of a pull-down menu are also listed, for example. Accordingly, these can be hidden again by clicking on the symbol.



If you enter a search term in the docking window, HiCAD lists all functions whose name contains the specified search term. The search is carried out in all ribbon bars as well as in the functions of the docking window **Civil Engineering functions**.

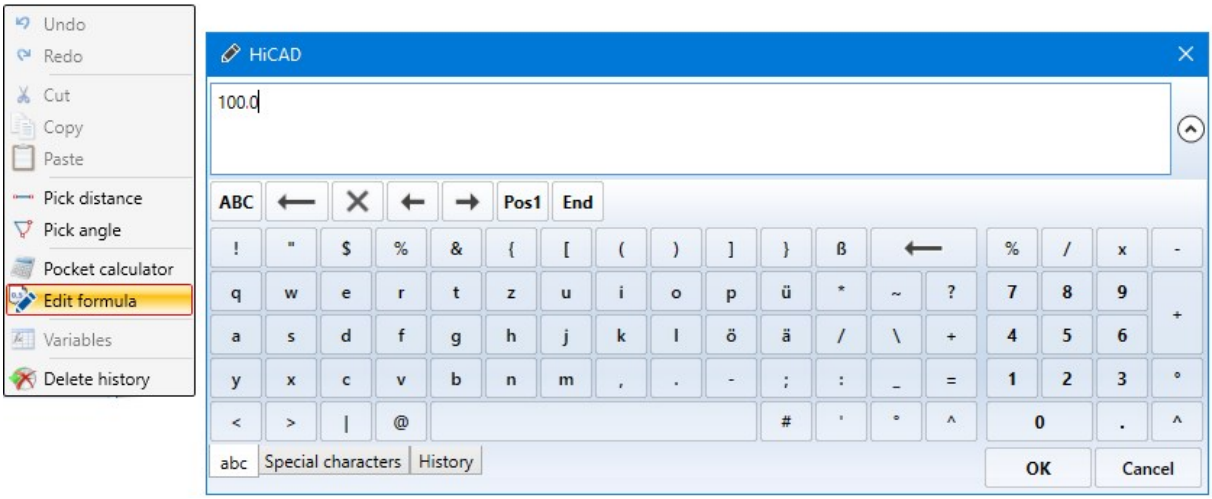


If you right-click on a function, the function can either be called up directly (Execute), or you can open the Help for this function, or Copy the function path.



Edit formula

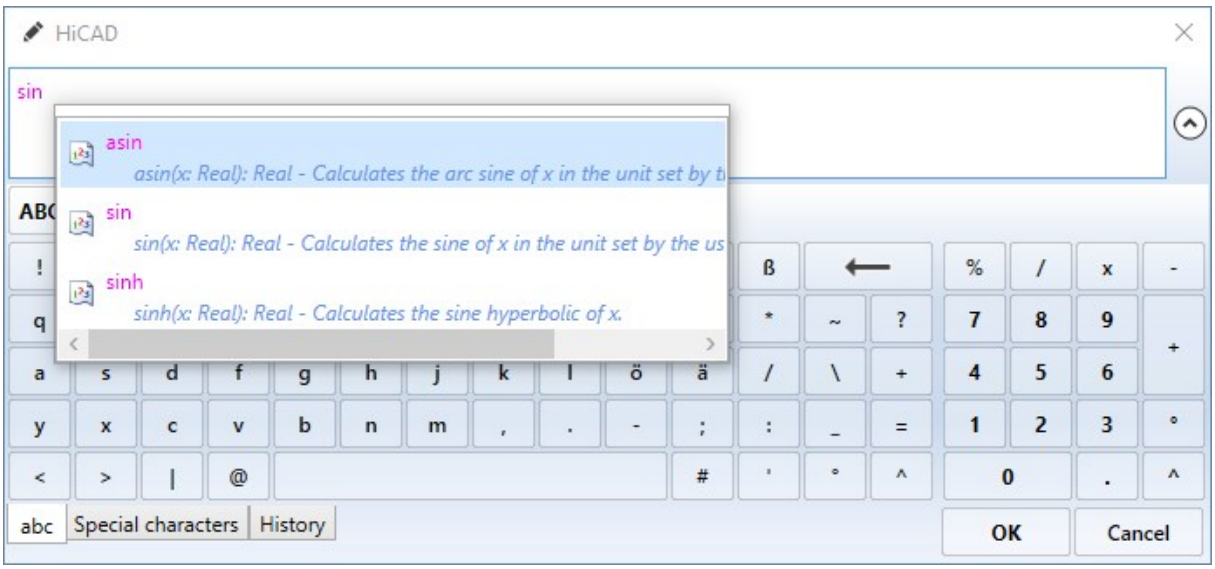
For entering values in dialogue windows, the function **Edit formula** is available from HiCAD 2021 in addition to the calculator. This function supports you in entering and editing more complex formulae. You activate the function by pressing the right mouse button in the value input field of a dialogue window and selecting the function in the context menu.



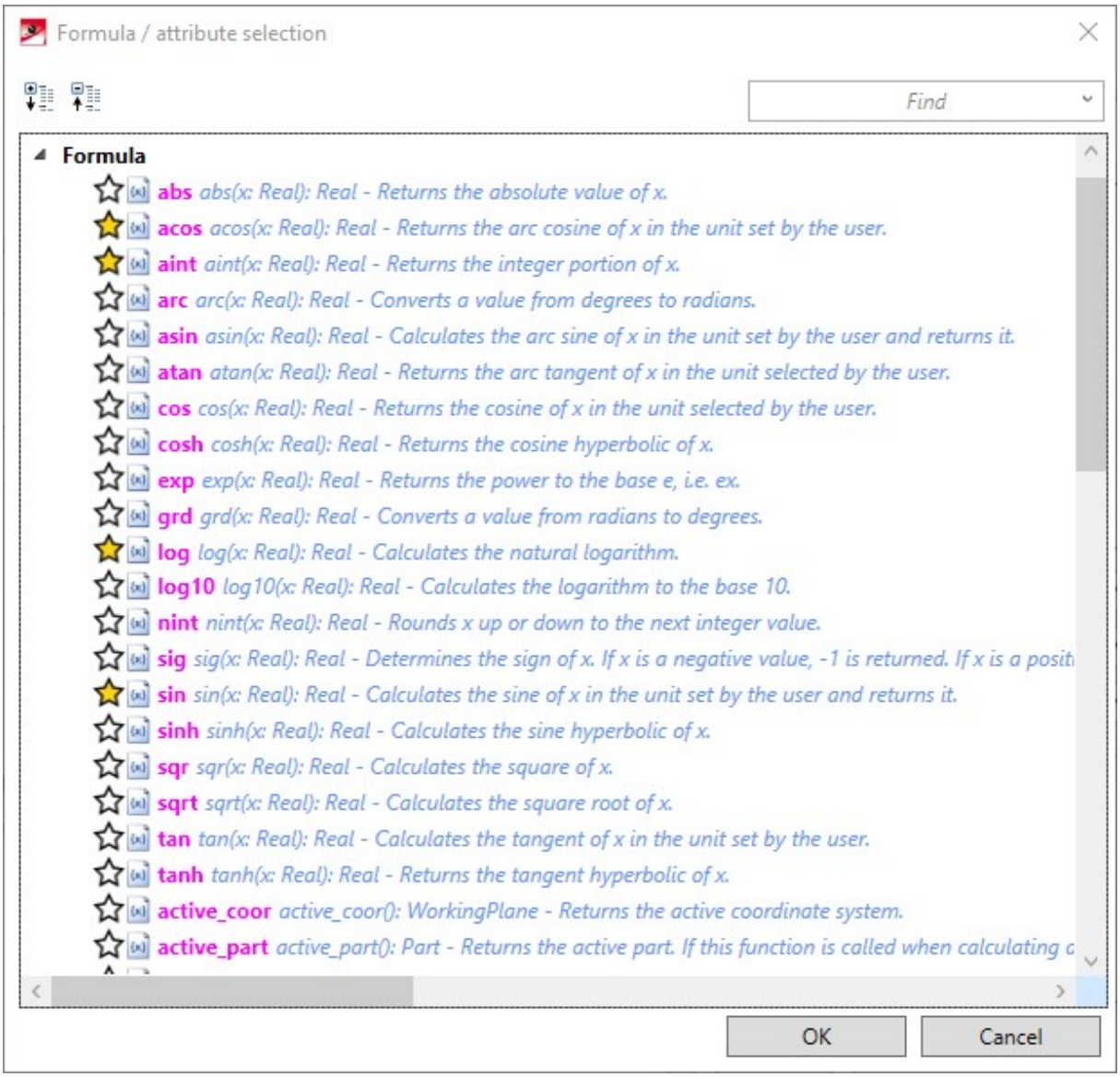
You can select the layout of the editor's keyboard using the tabs at the bottom of the window:

abc	normal keyboard with numeric keypad
Special characters	Scrollable keyboard with upper/lower case letters and special characters
History	last used entries

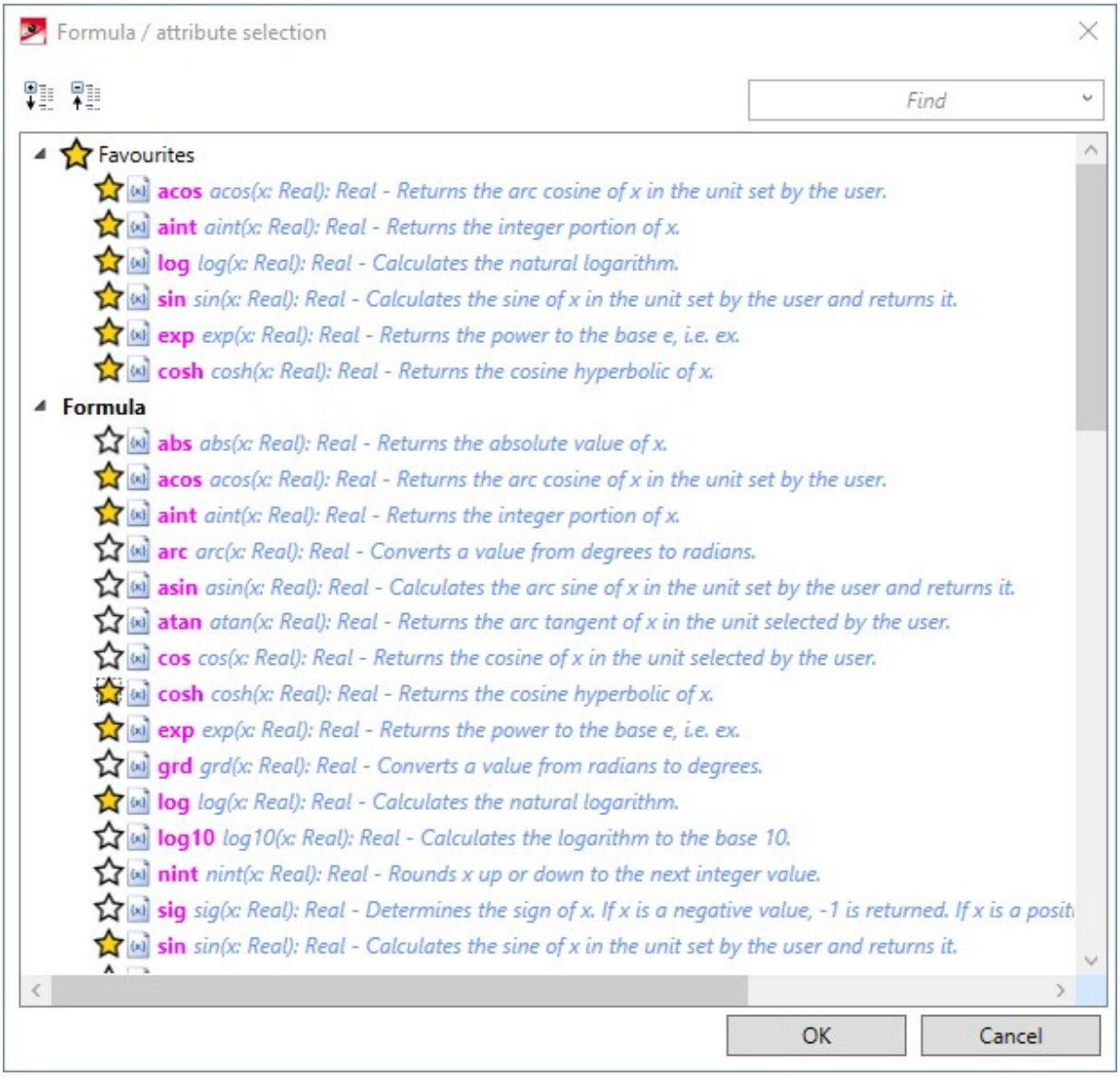
As soon as you enter something in the editor's input field, HiCAD automatically searches for mathematical functions whose name contains the string you entered. For example, if you enter ***sin***, all functions are offered for selection whose name contains the string ***sin***.



If you press CTRL+K in the input field of the editor, a list of all supported mathematical functions is displayed.



Frequently used functions can also be marked as favourites here for faster access. To do this, simply click on the ☆ symbol next to the function name. The white symbol is then replaced with a yellow ★ symbol. Functions marked in this way are listed in the selection window under **Favourites**.

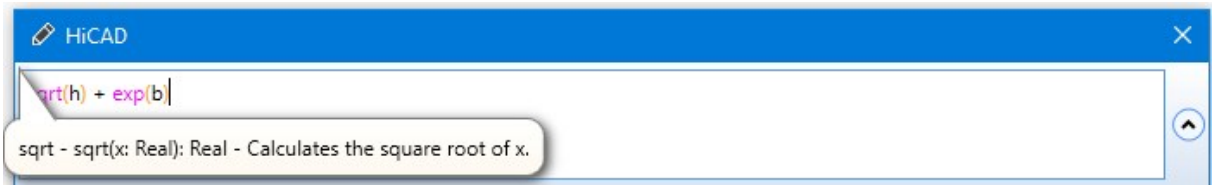


To remove an attribute from the list of favourites, simply click on the corresponding symbol - either directly in the list of favourites or in the list of functions.

The search field at the top right of the dialogue window can also be used to search for functions.

A function is selected by double-clicking on the function name in the selection list. The function is entered in the editor's input field and you can add the desired function argument (in brackets) and extend the formula as desired. Nested formulas are also possible.

If you point the cursor to a function name of the formula, corresponding information is displayed as a tooltip.



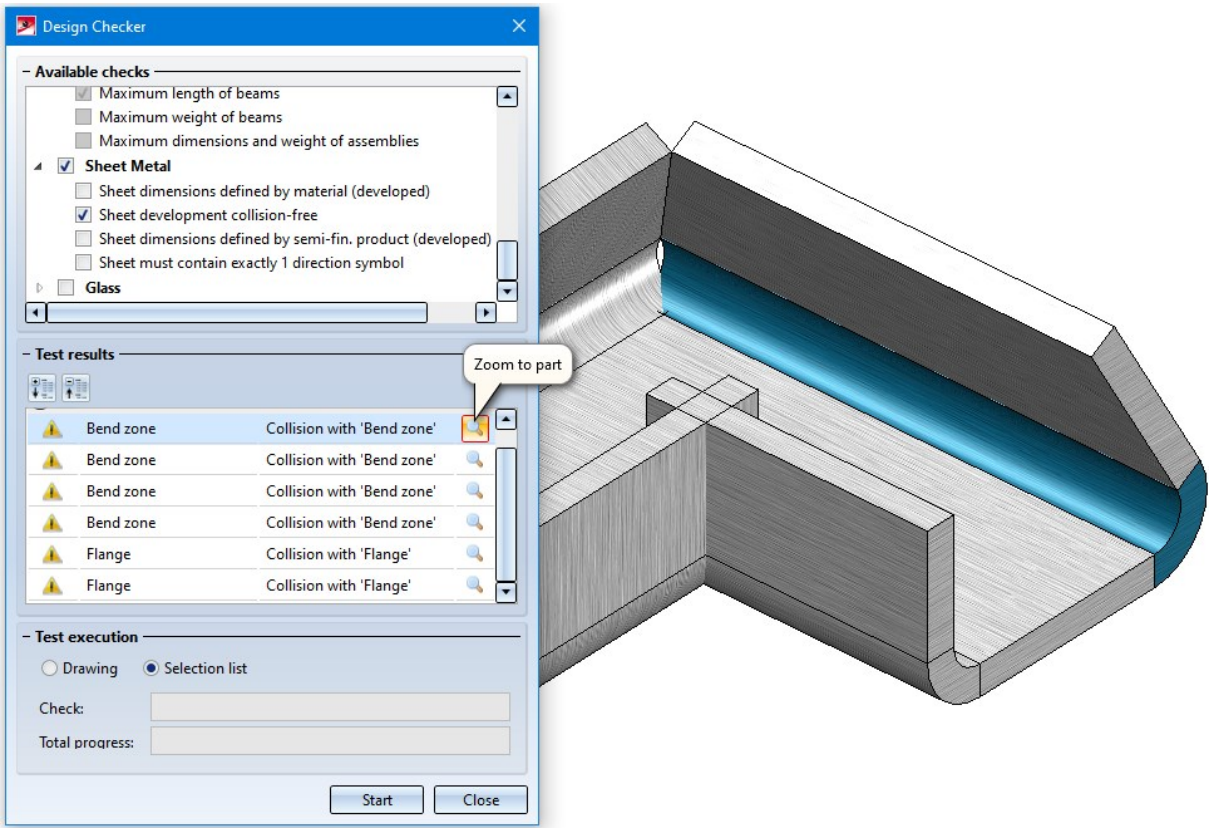
With **OK** the formula is taken over into the input field of the current function.

Design Checker

Non-BOM-relevant parts with HELiOS article master

The new test **Non-BOM-relevant parts with HELiOS article master** is available in the **Design Checker** under **General**.

If a model drawing contains parts with article master data, some of which are BOM-relevant and some of which are not, there may be problems when transferring the product structure to HELiOS. With the new test, all parts can be found that have an article master but are not BOM-relevant.



Length and weight of beams and profiles

The test Maximum length and weight of beams has been split into two tests:

- **Maximum length of beams** and
- **Maximum weight of beams**

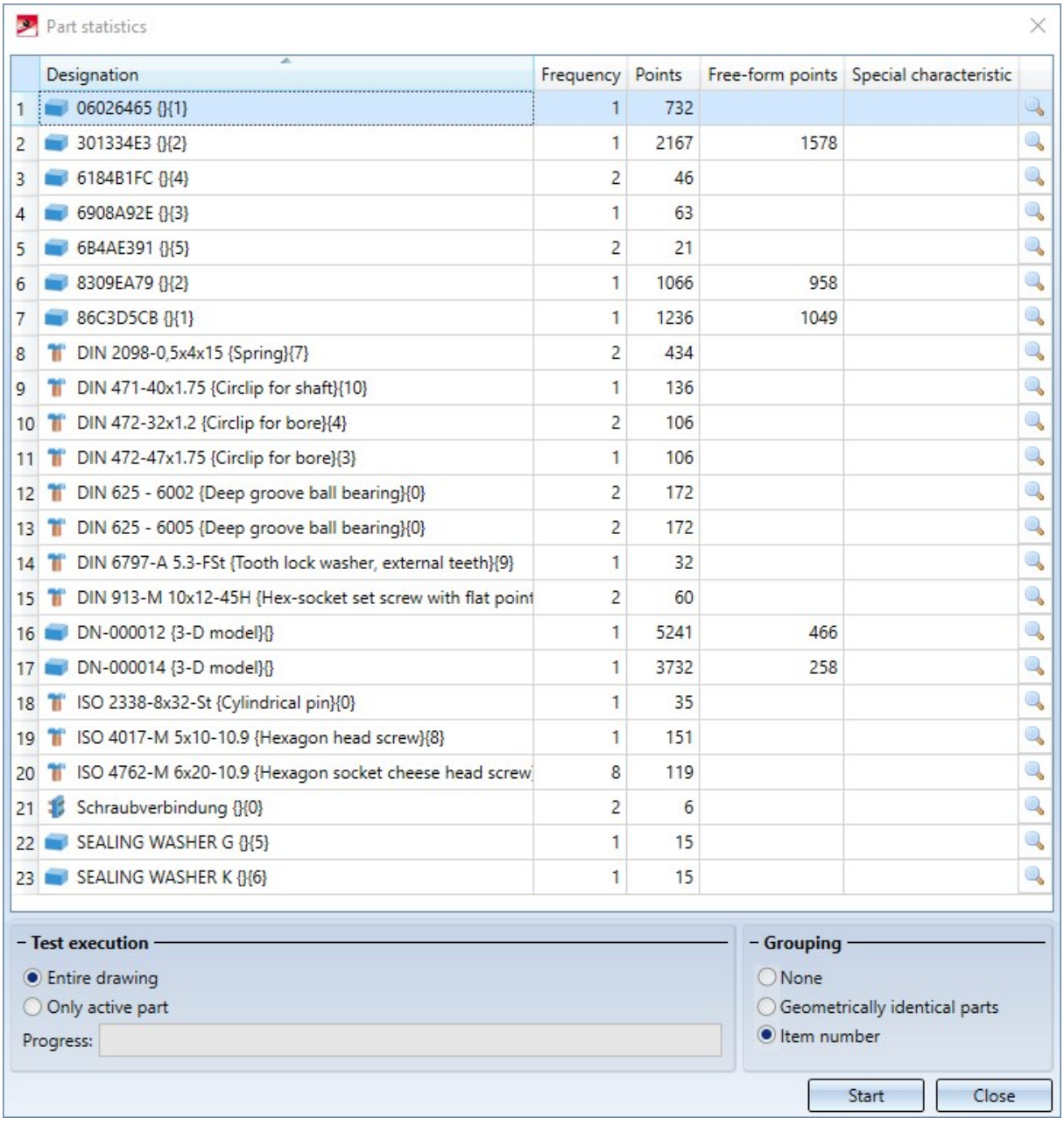
Collision free sheet development

This test checks sheet metal parts for collisions between flanges or bend zones during a development or bending simulation.

Part statistics

With the new function **Part statistics** you can find parts and assemblies that have a particularly negative influence on the performance of the current model drawing. These are, for example, parts with many free-form points or surface parts. This makes it easier to optimise the corresponding parts/assemblies for faster work. You can find the function at **Information > 3-D, Further**.

After calling the function, the dialogue window shown is displayed:



Test execution

With the options under Test execution you can determine

- whether the entire drawing or only the active part is to be tested and
- whether geometrically identical parts should be grouped or not. Grouping identical parts can provide a better overview, especially in complex drawings.

Output columns

Designation	corresponds to the designation configured in the ICN. The default is Article number (or the part name), Comment (attribute **Designation1**) and the Item number of a part..
Frequency	Shows how often a part appears in the model drawing (if the option **Grouping of geometrically identical parts** is active).
Points	Number of points a part consists of.
Free-form points	Number of free-form points. These points are created, for example, when working with free-form surfaces or when filleting parts.
Special characteristic	At the moment only parts with free surfaces are evaluated here, i.e. parts without volume. In this case, **Surface body** is displayed as a special characteristic. For all other parts the field is empty.

The columns can be sorted in ascending and descending order by clicking on the column header. The default sorting is by points.

With a click on **Start**, the part statistics are determined. A magnifying glass symbol is displayed behind each part in the output area. Clicking on this symbol activates the corresponding part in the model drawing, enlarges it and highlights it in colour (**Special colour Marking 3**). If another part is selected with the magnifying glass symbol, this part is added to the current part list.

While the dialogue window is open, all HiCAD functions can be called up, i.e. you can edit the current part list, e.g. optimise or approximate surface parts etc.

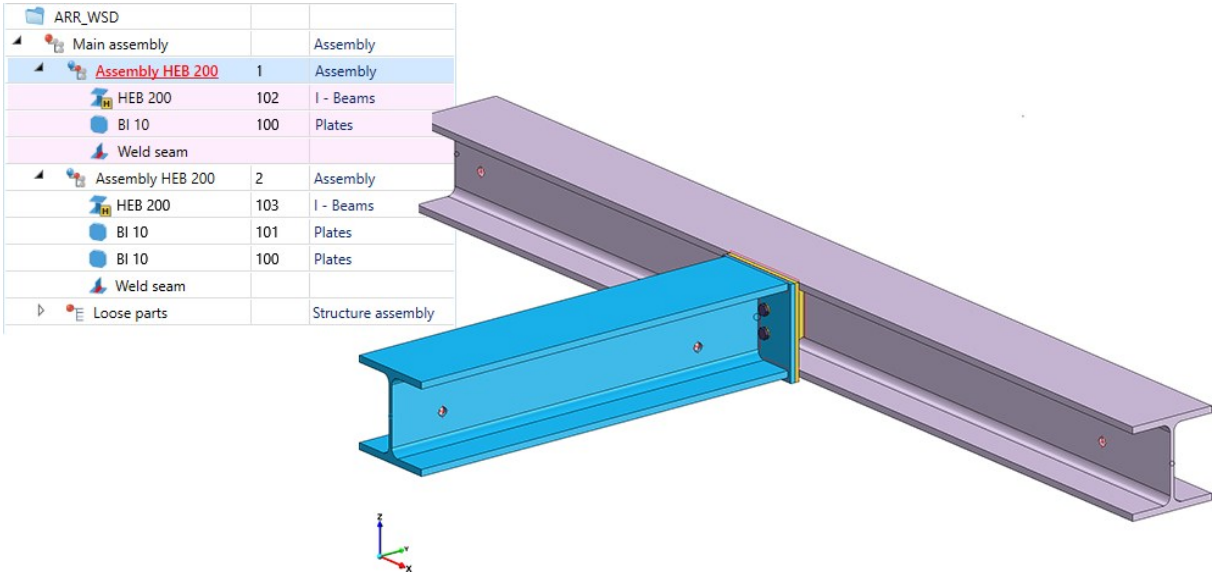
Drawing derivation

Arrangement of views

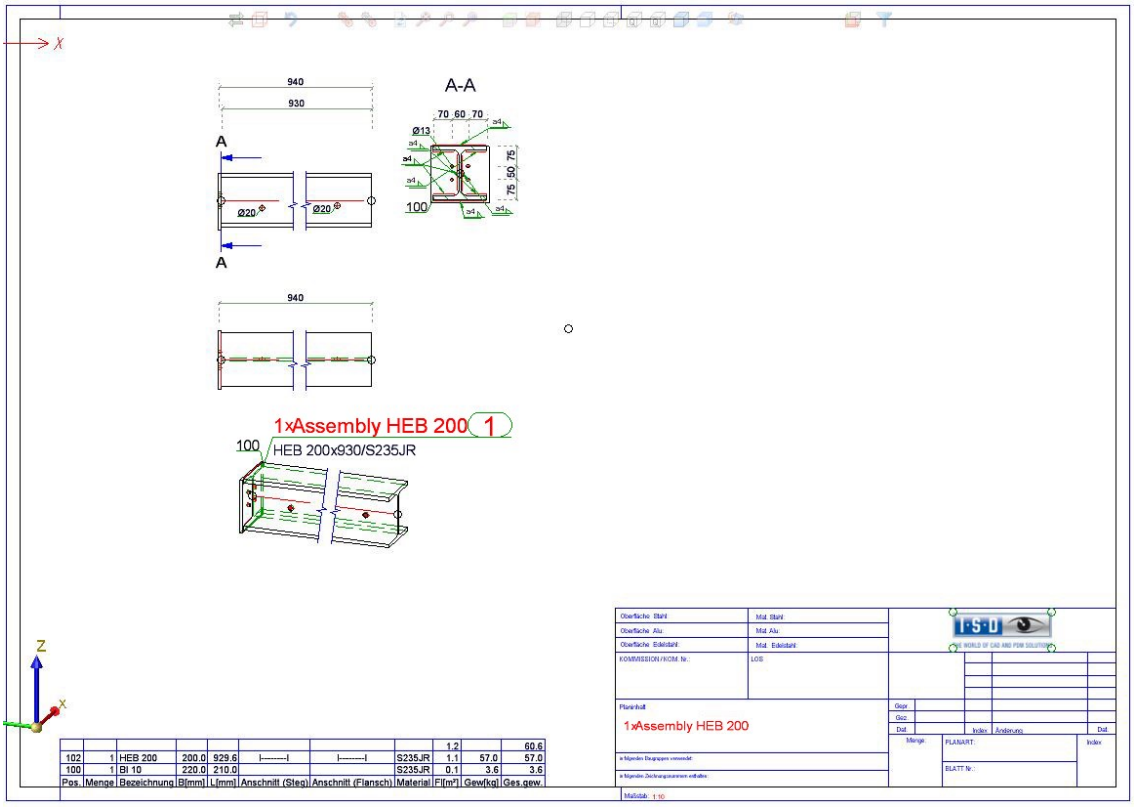
The arrangement of views in production drawing has been improved. Views are now distributed more clearly on the drawing sheet and the automatic selection of the sheet size or view scale has been improved so that the space on the sheet is used optimally.

An example:

In the model drawing shown, the production drawing for the selected assembly has been created - once with HiCAD 2020 and once with HiCAD 2021.

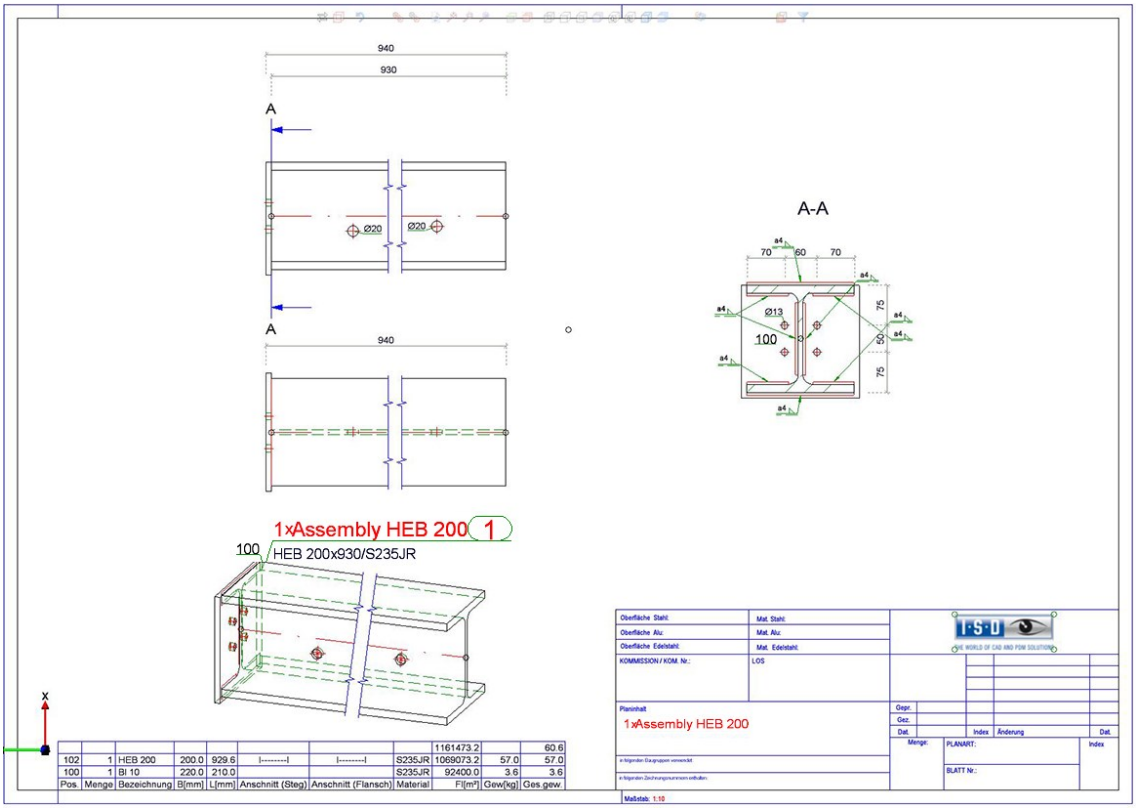


Production drawing in HiCAD 2020



Pos.	Menge	Bezeichnung	B[mm]	L[mm]	Anschnitt (Steg)	Anschnitt (Flansch)	Material	Fl[m²]	Gew[kg]	Ges.gew.
102	1	HEB 200	200.0	929.6	I--------I	I--------I	S235JR	1.1	57.0	57.0
100	1	Bl 10	220.0	210.0			S235JR	0.1	3.6	3.6

Production drawing in HiCAD 2021



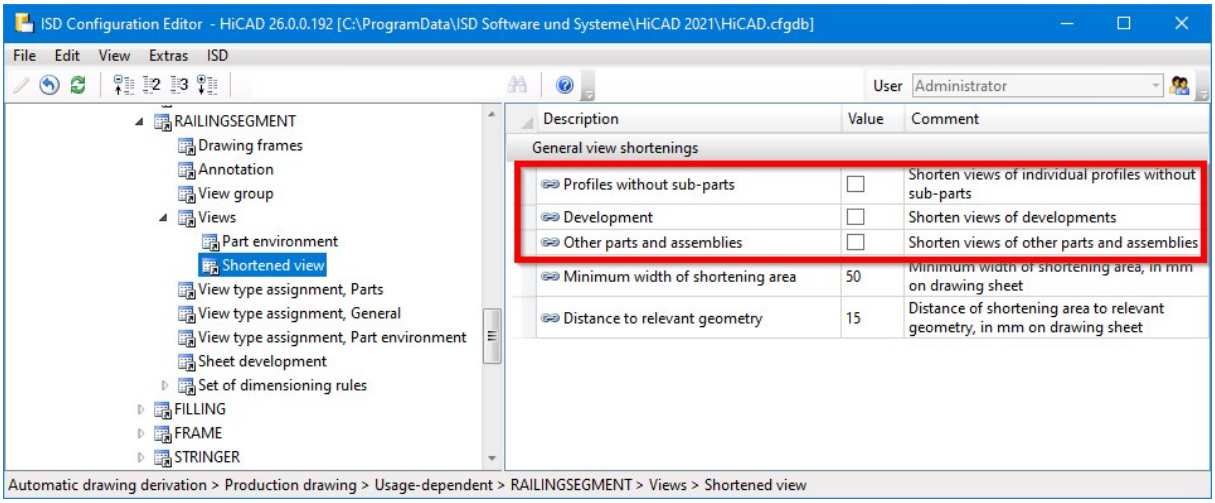
Pos.	Menge	Bezeichnung	B[mm]	L[mm]	Anschnitt (Steg)	Anschnitt (Flansch)	Material	Fl[m²]	Gew[kg]	Ges.gew.
102	1	HEB 200	200.0	929.6	I--------I	I--------I	S235JR	1161473.2 1069073.2	57.0	57.0
100	1	Bl 10	220.0	210.0			S235JR	92400.0	3.6	3.6

Railings

The **RAILINGSEGMENT** configuration for railings in the drawing derivation has been changed. Views are now not automatically displayed shortened here. For this purpose, the following settings have to be made in the Configuration Editor at

Automatic drawing derivation > Production drawing >Usage-dependent > RAILINGSEGMENT > Views > Shortened view

the checkboxes have been deactivated.



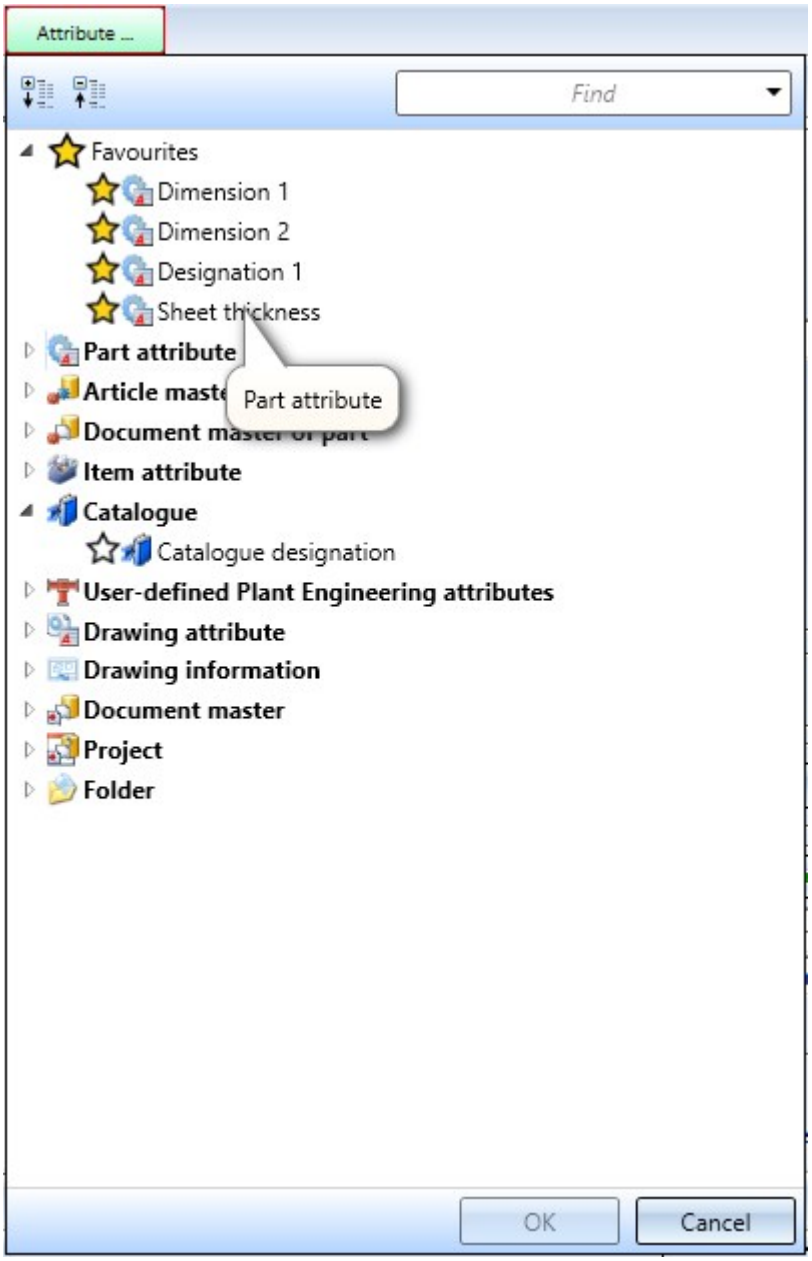
2-D

Service Pack 1 2021 (V 2601)

Attributes in annotations and text blocks

Improved attribute selection

Each attribute is now marked with a symbol indicating the attribute group.When you move the cursor over an attribute, a tooltip will show you to which attribute group the attribute belongs: .



Favourites

If attributes are marked as favourites in the Annotation Editor, the list of favourites is now automatically expanded when the **Attribute...** button is clicked.

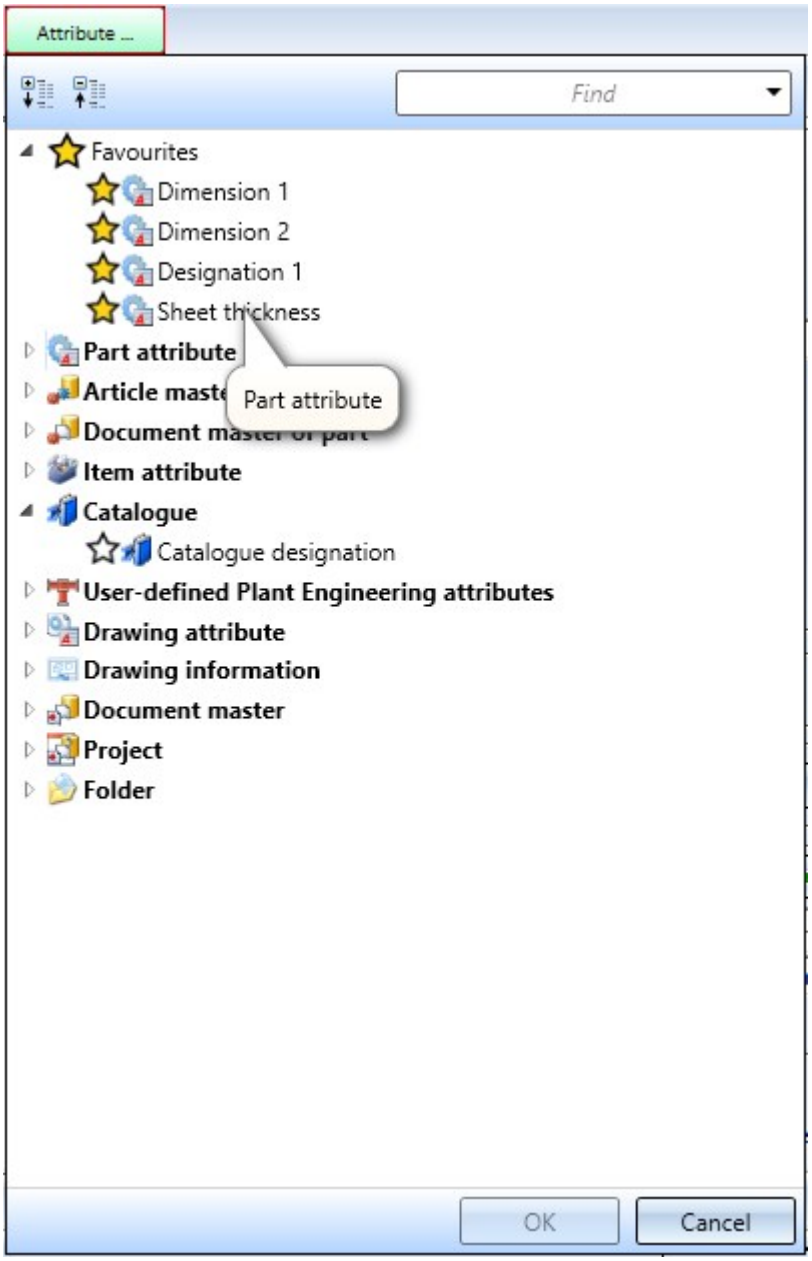
Expanded attribute selection

From SP1 onwards, further attributes can be used in 2-D annotation tags and text blocks:

- Drawing information, e.g. name and number of the drawing sheet on which the text or annotation is located,
- HELiOS item attributes of the part,
- attributes from the HELiOS document master of the part,
- attributes of the HELiOS folder to which the model drawing belongs.

Please note:

A different view scale exists only if the view scale and main scale are different and the 2-D part is below the view in the 2D Part structure of the ICN. The sheet scale exists only if all views on the sheet have the same scale. Not relevant are sectional/detail views, views of sheet developments and axonometries in workshop drawings. The sheet name exists only if a name has been assigned to the sheet.



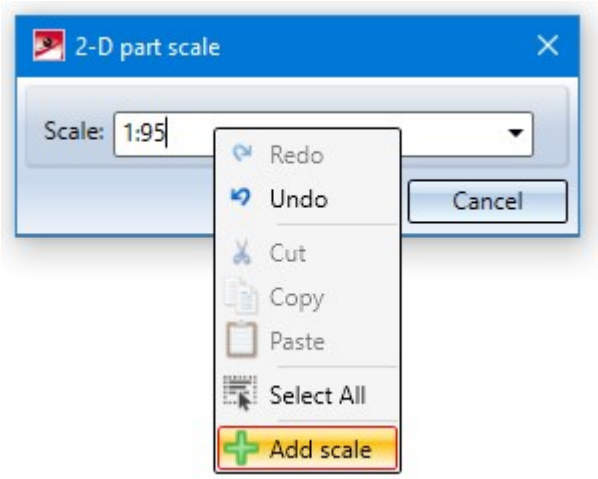
(1) Sheet view active
(2) 2D part structure in ICN
(3) Part 002_001 is on the same plane as "View 2", therefore no information about the View is displayed
(4) Part 001_001 is below "View 2", therefore Sheet and View information is displayed

HELiOS attributes in annotations

If a model drawing contains annotations or text blocks with HELiOS attributes, these are automatically updated from SP1 when the drawing is loaded. This behaviour can be switched off in the Configuration editor. For this purpose, the checkbox **Update annotation tags with HELiOS attributes** is available at **System settings > HELiOS**.

Manage drawing scales

The **Manage drawing scales** function can also be called directly within certain scale functions, e.g. **Part scale** . To do this, right-click in the scale field within the corresponding function and select the **Add scale** function in the context menu.



In this way, further individual scales can be defined for the current model drawing without having to leave the current function.

Date in title block

When the print date is output via the @PLOD@ variable, the **Date format** set in the Configuration Editor at **System settings > Units** is taken into account.

The selected date format is also taken into account by the DAT$ string function. This function is used when filling the title block with the **Complete title block** (without DB) function (at Drawing > Others > Extras >Tools) or in macros.

Automatically break up cloned 2-D parts

Connected, cloned 2-D parts can no longer be regenerated for some time.

When loading SZA, FGA and FIG, old connected cloned 2-D parts are automatically broken up as of HiCAD 2021 SP1.

Therefore the Break up **Break up 2-D cloning** (Drawing > Others > Extras > Tools > ...) is no longer available.

Import of HASCO and STRACK standard parts

The functions for inserting HASCO and STRACK standard parts have been removed from the HiCAD user interface. The insertion of these standard parts is now only possible via the corresponding macros in the HiCAD subdirectory MAKRO2D:

- 2d3dpart_insert1hasco.mac
- 2d3dteil_einfueg1strack.mac

As of HiCAD 2022, importing via the macros and via the API will also no longer be possible!

Major Release 2021 (V 2600)

Unit of measurement for model drawings

HiCAD 2021 supports only model drawings with the unit of measurement mm. The previous **Unit of measurement** function is no longer available. This also applies to the **Unit of measurement** parameter in the Configuration Editor at **System settings > Miscellaneous**.

The unit of measurement is saved in the corresponding SZA or KRA file when saving a model drawing or a 3-D part. When trying to open model drawings with a different unit of measurement or insert parts with a unit of measurement different from mm, an error message is displayed.

Scales

The scale can be selected from a selection box in the corresponding HiCAD function dialogs. Which scales are available in the selection boxes was previously defined in the **SZENE-MASSSTAB.TXT** files in the HiCAD sub-directory **MAKRO2D**. As of HiCAD 2021, **Scale lists** can be edited and defined in the Configuration Editor at **System settings > Scales**. In addition, you can define which scale list is to be used by default for new model drawings.

Unit of measurement centesimal/GON withdrawn

The unit of measurement Centesimal/GON for angular dimensions is an auxiliary unit for indicating the angular width of planar angles, which could only be established in surveying. Therefore, it was removed in HiCAD 2021.

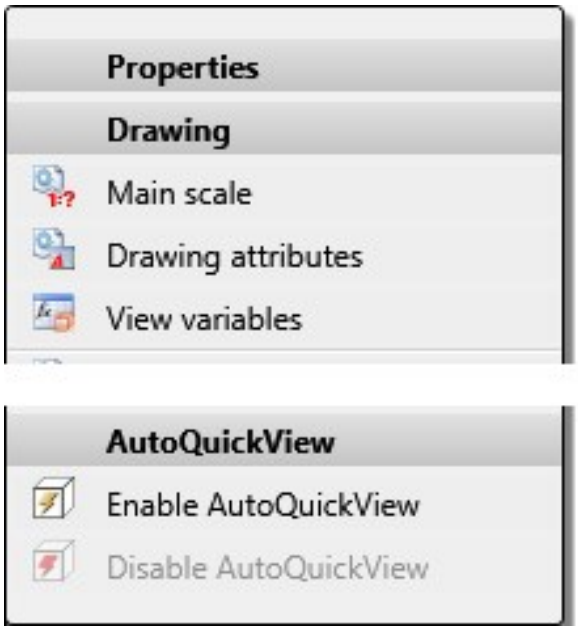
3-D

Service Pack 1 2021 (V 2601)

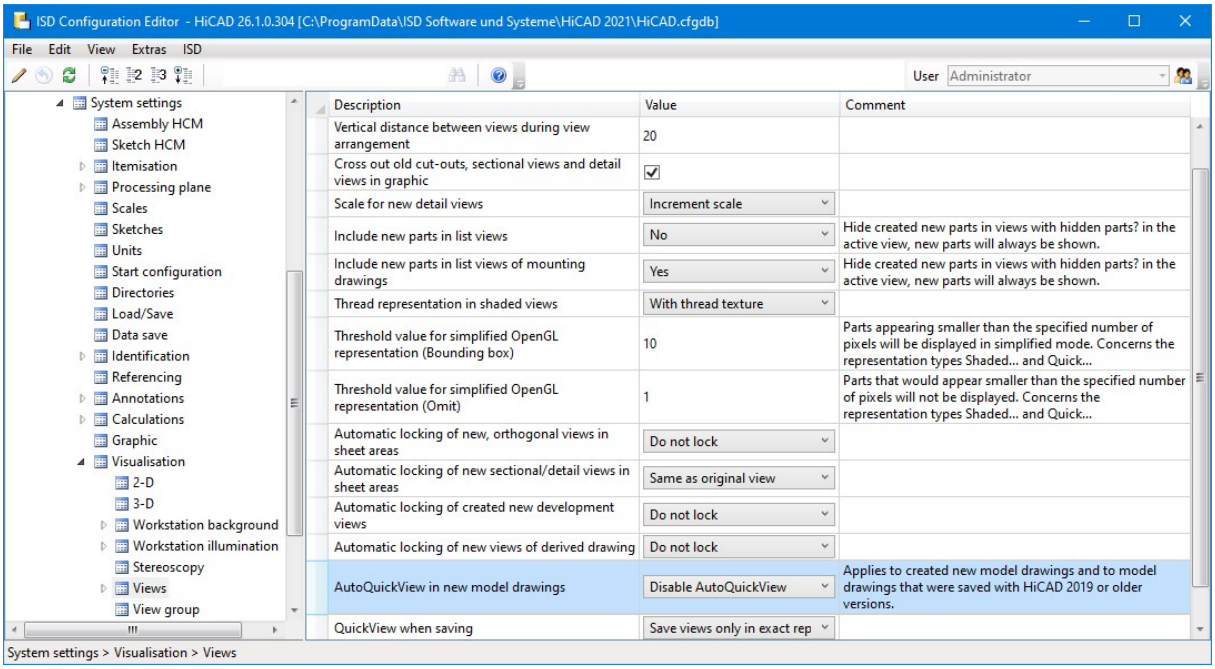
AutoQuickView

The previous user guidance texts in the context menu of the drawing at **Properties > AutoQuickView** as well as in the **Properties** function group of the **Drawing** Ribbon tab have been changed to emphasise the automatic aspect more clearly.

The function **Enable AutoQuickView** switches the AutoQuickView on. This automatism sets one or more views to QuickView mode - at the given time - in order to skip HiddenLine and Glass model calculations.



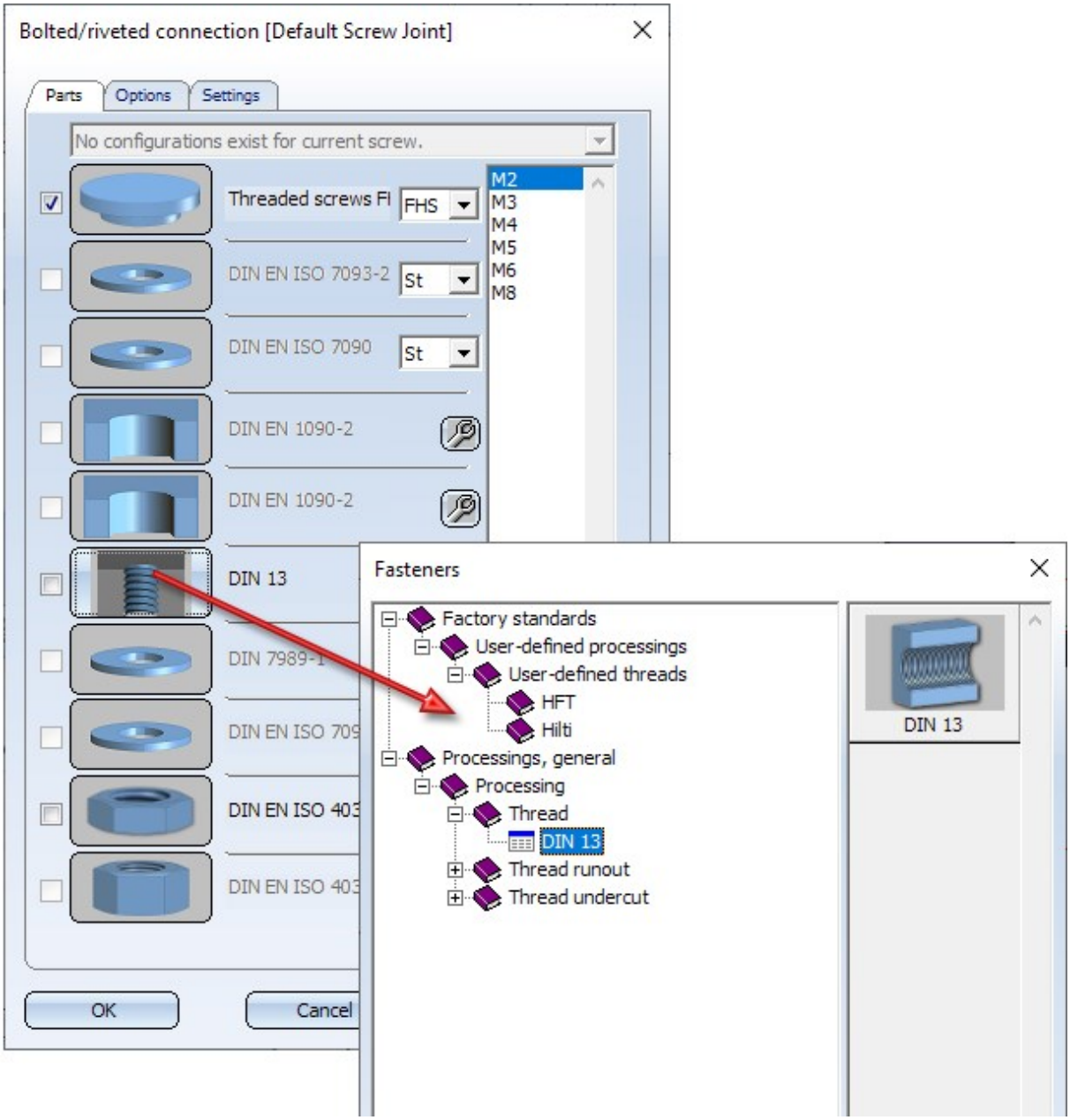
The change of the user guidance text also concerns the settings in the Configuration Editor.



Standard Parts / Standard Processings

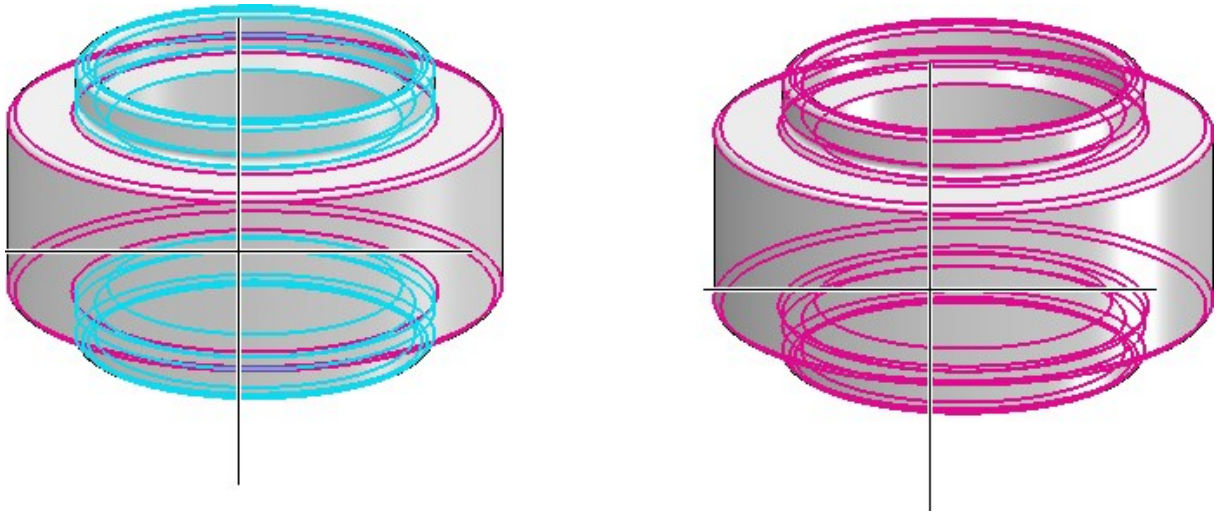
Boltings - User-specific threads

From SP1 onwards, user-defined threads can be selected from the catalogue **Factory standards > User-defined processings > User-defined threads** - depending on the selected screw/bolt type - when inserting boltings in your model drawing.


Bolted/riveted connection [Default Screw Joint]
Parts
Options
Settings
No configurations exist for current screw.
Threaded screws Fl
FHS
M2
M3
M4
M5
M6
M8
DIN EN ISO 7093-2
St
DIN EN ISO 7090
St
DIN EN 1090-2
DIN EN 1090-2
DIN 13
DIN 7989-1
DIN EN ISO 709
DIN EN ISO 403
DIN EN ISO 403
OK
Cancel
Fasteners
Factory standards
User-defined processings
User-defined threads
HFT
Hilti
Processings, general
Processing
Thread
DIN 13
Thread runout
Thread undercut
DIN 13

Spherical plain bearing

The revision of the display of bearings started with HiCAD 2021 has been continued in SP1 with the revision of the spherical plain bearings and spherical plain bearing rod ends. The new parts allow the selection of all edges and surfaces and present themselves as one part in every respect. This applies, for example, to highlighting and part selection..



Left: Highlighting in HiCAD 2020, Right: in HiCAD 2021 SP1.

The new representation, however, only affects spherical plain bearings that are integrated into the drawing from HiCAD 2021 SP1. When loading drawings with spherical plain bearings from a version before HiCAD 2021 SP1, the "old" representation is used.

Process / model parts

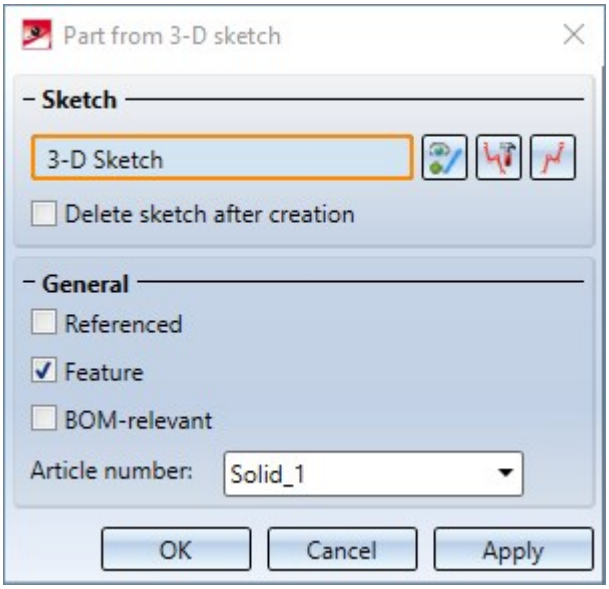
New part from 3-D sketch

With the new function **New part from 3-D sketch** you can derive a solid from a 3-D sketch. This solid consists of planar surfaces and the body edges are the lines of the sketch.

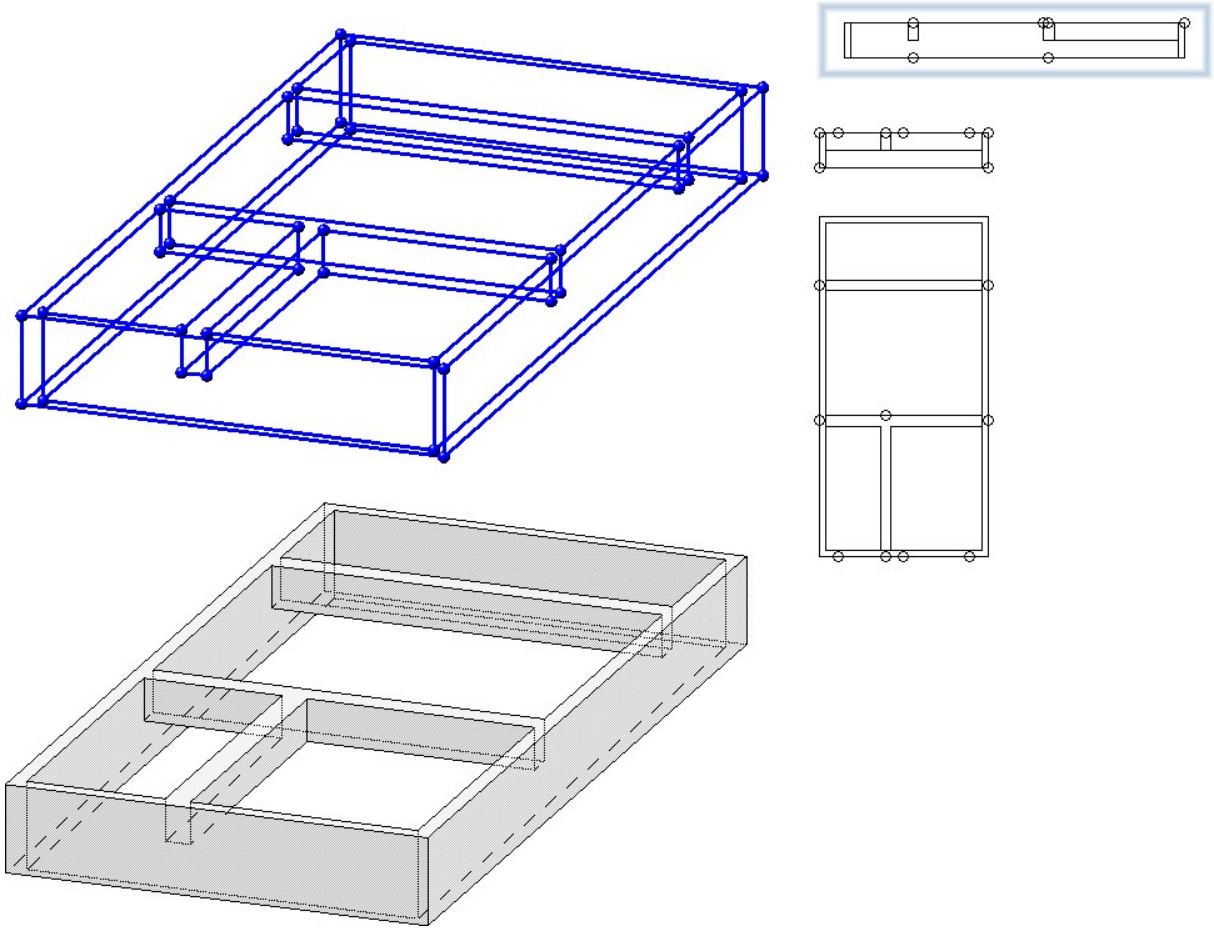
The 3-D sketch must fulfil the following conditions:

- The sketch may only contain straight lines. These must be real lines. For example, straight Nurbs curves and auxiliary geometry are not allowed.
- The lines of the sketch must not intersect or coincide (lie on top of each other).
- The sketch must result in a meaningful solid.
- At least three lines must converge at each line end. Any auxiliary geometry that may exist will not be considered here. Ends with one or two lines are evaluated as errors and displayed in the graphic. In this way, even small gaps in the sketch can be found quickly.

After calling the function, the **Part from 3-D Sketch** dialogue window is displayed and HiCAD prompts you to select a line of the 3-D sketch.



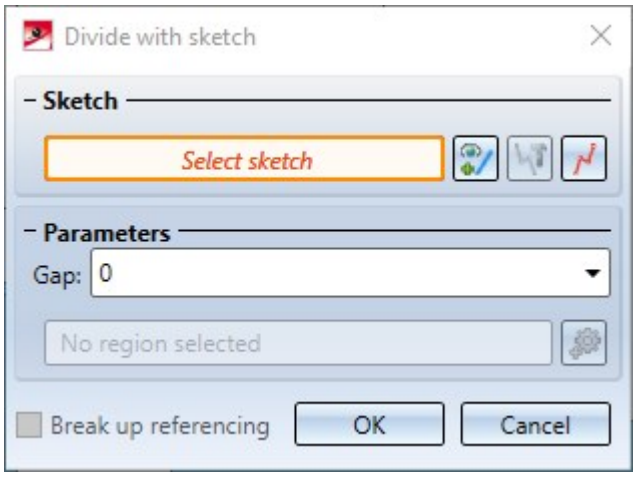
If it is possible to create a solid on the basis of the selected 3-D sketch, then a preview of the part is automatically displayed in the design. With a click on the **Apply** button (or on **OK** or by pressing the middle mouse button) the part is created.



If the creation of the 3-D part cannot be carried out, then the symbol is displayed at the **OK** button. If the selected sketch does not meet the requirements, the symbol appears next to the field for sketch selection. If you point with the cursor to one of the symbols, you will get further information. The corresponding lines and surfaces are visualised in the construction and highlighted in red. You can then correct the sketch or cancel the function.

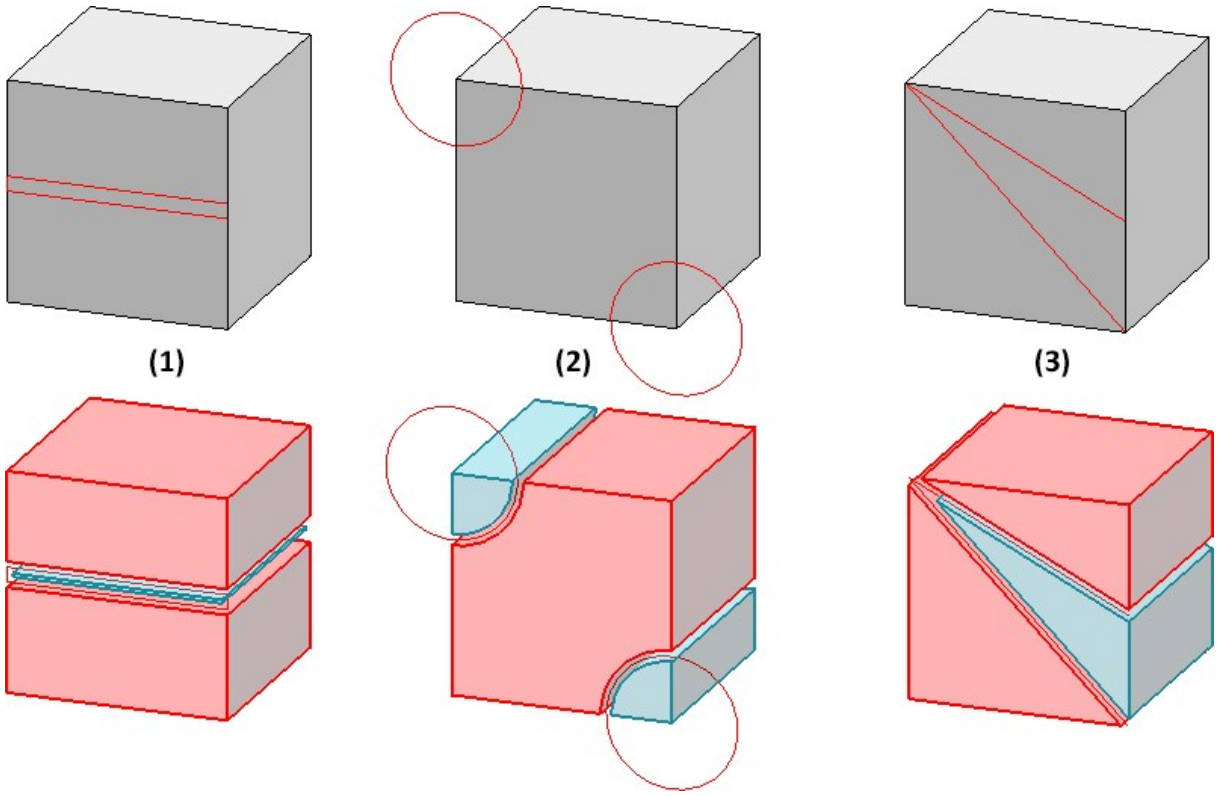
Divide part

The **Divide** function has been completely revised.

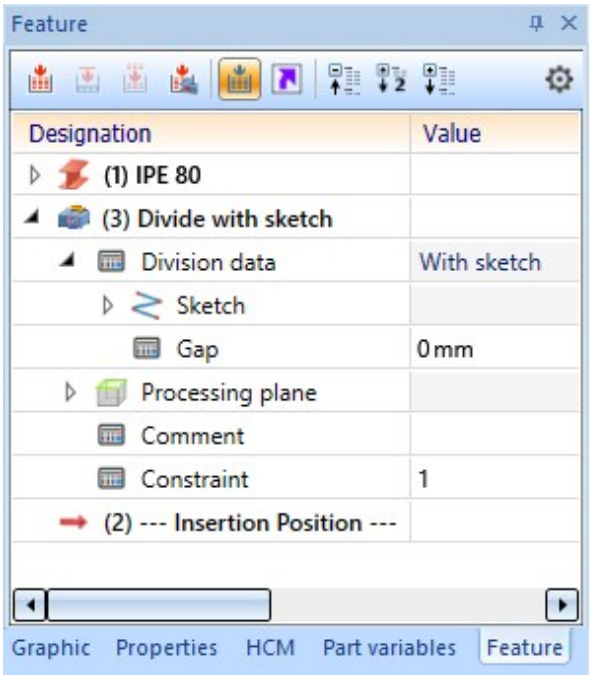


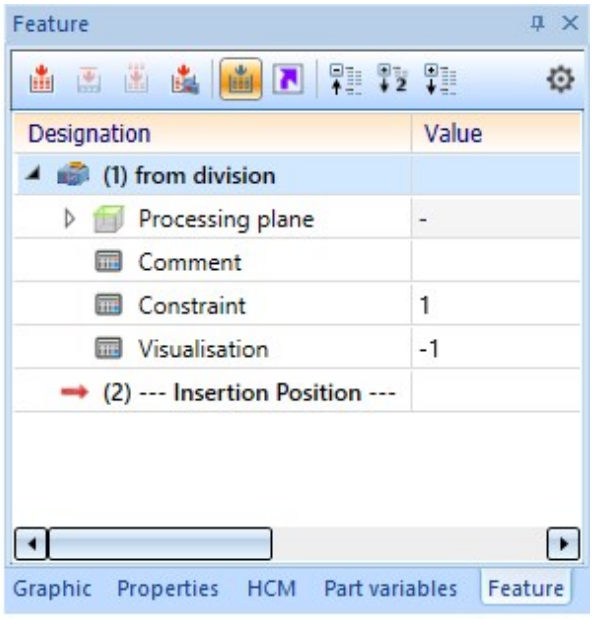
With this function you divide a part based on a planar sketch. Each line of the sketch is a section line that divides the part into two section regions. Connected lines are treated as one line. This means, for example, that a sketch consisting of three connected lines also divides a part into two cutting regions. Each section region can consist of several parts, whereby the number of parts is determined by the actual number of lines.

The illustration shows a cuboid with three different sketches and the resulting division. In all cases, two parts are created. In (1) and (3) the leading part consists of two segments in case (2) of one segment.



The main feature **Divide with sketch** is assigned to the leading part (highlighted red when executing the function). With this feature the sketch can be edited later and the gap can be changed. All other parts created are assigned the feature **from division**.

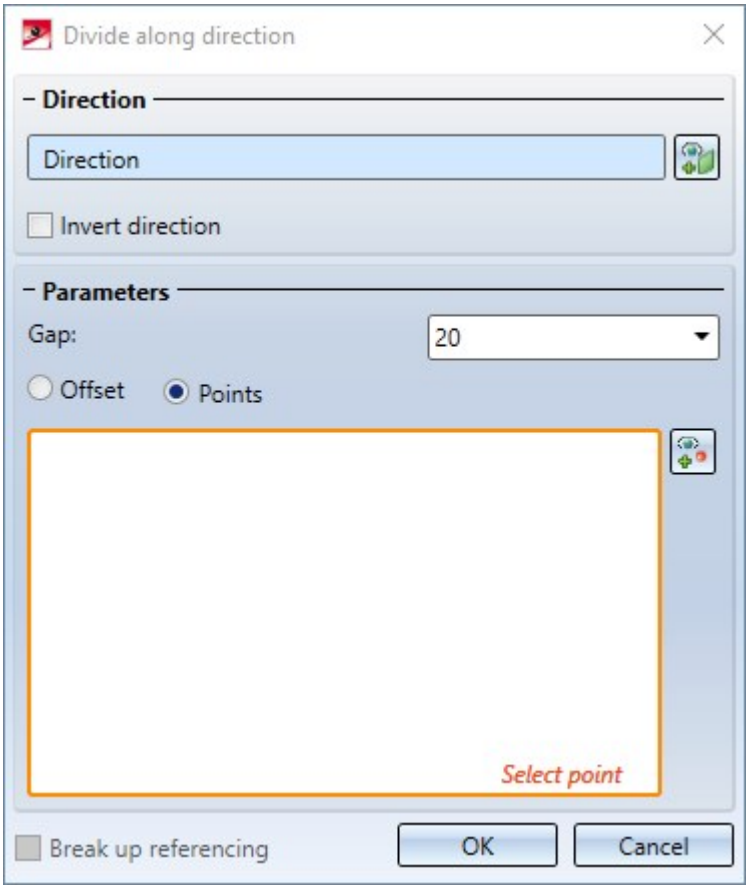


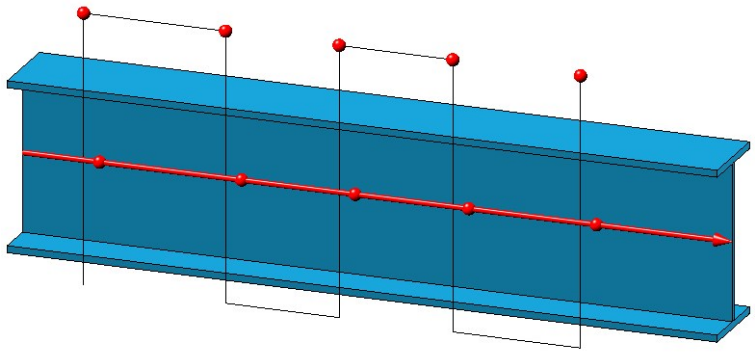


Divide along direction

The functions **Divide along direction** and **Divide along direction** (Steel Engineering > Lengthen > Divide > ...) have been revised

- The display of the cut planes in the preview is omitted. Only the division direction and the division points are visualised.
- Dividing can now also be done by selecting points.

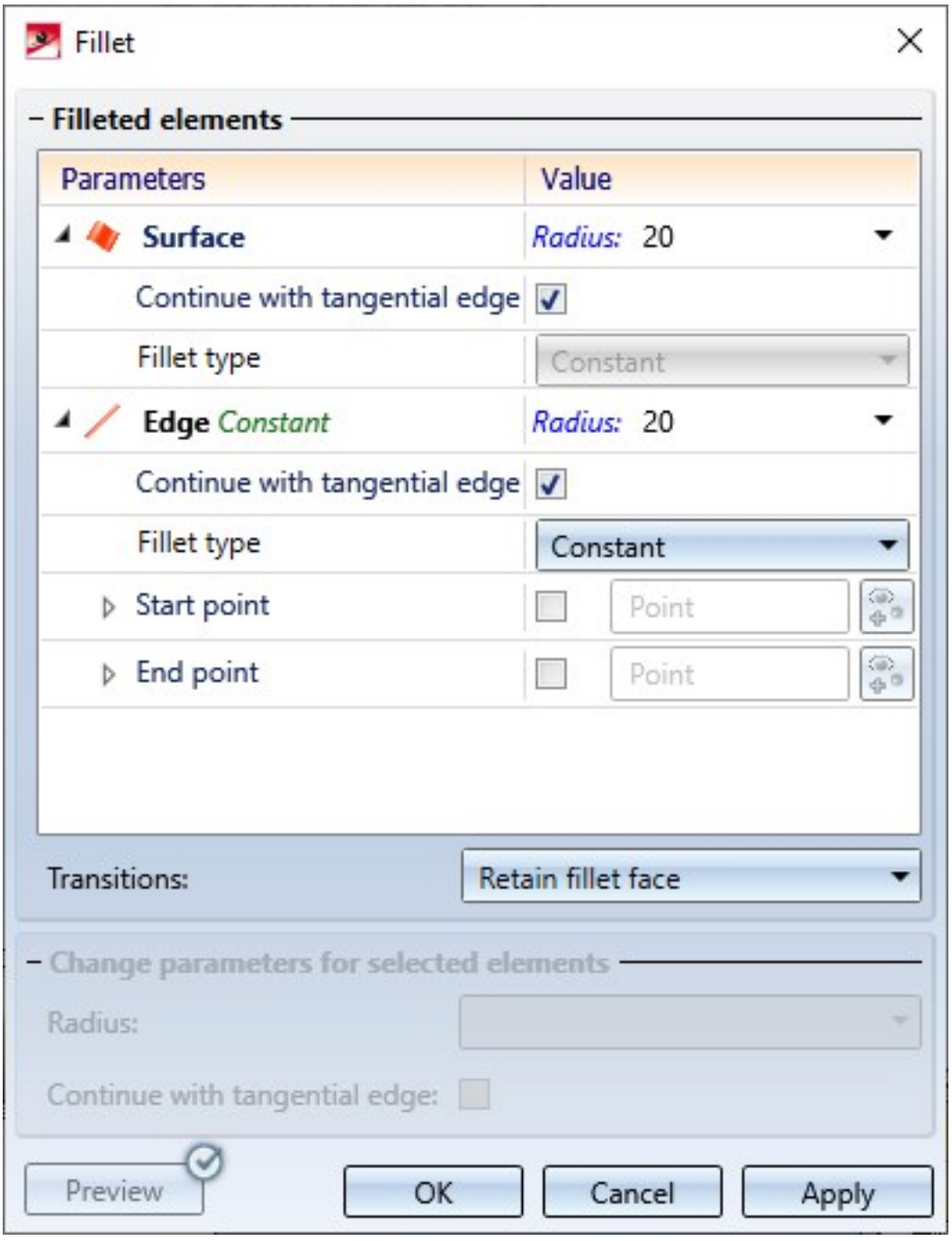




- If the active 3-D part is a beam or profile, the beam/profile axis is automatically suggested as the division axis when the function is called up via the context menu for 3-D parts. The division direction depends on the cursor setting when the function is called.

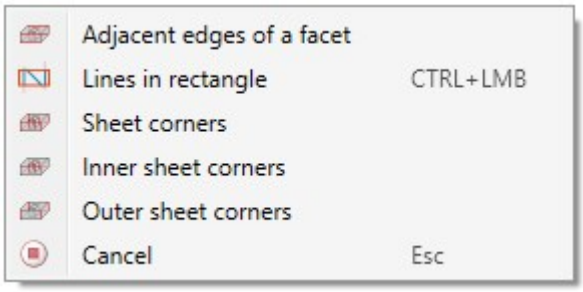
New Fillet dialogue

The **Fillet** dialogue window has had a redesign, now containing additional selection options for elements to be filleted, an improved preview, changed parameters, extended value input options etc.



New and enhanced options:

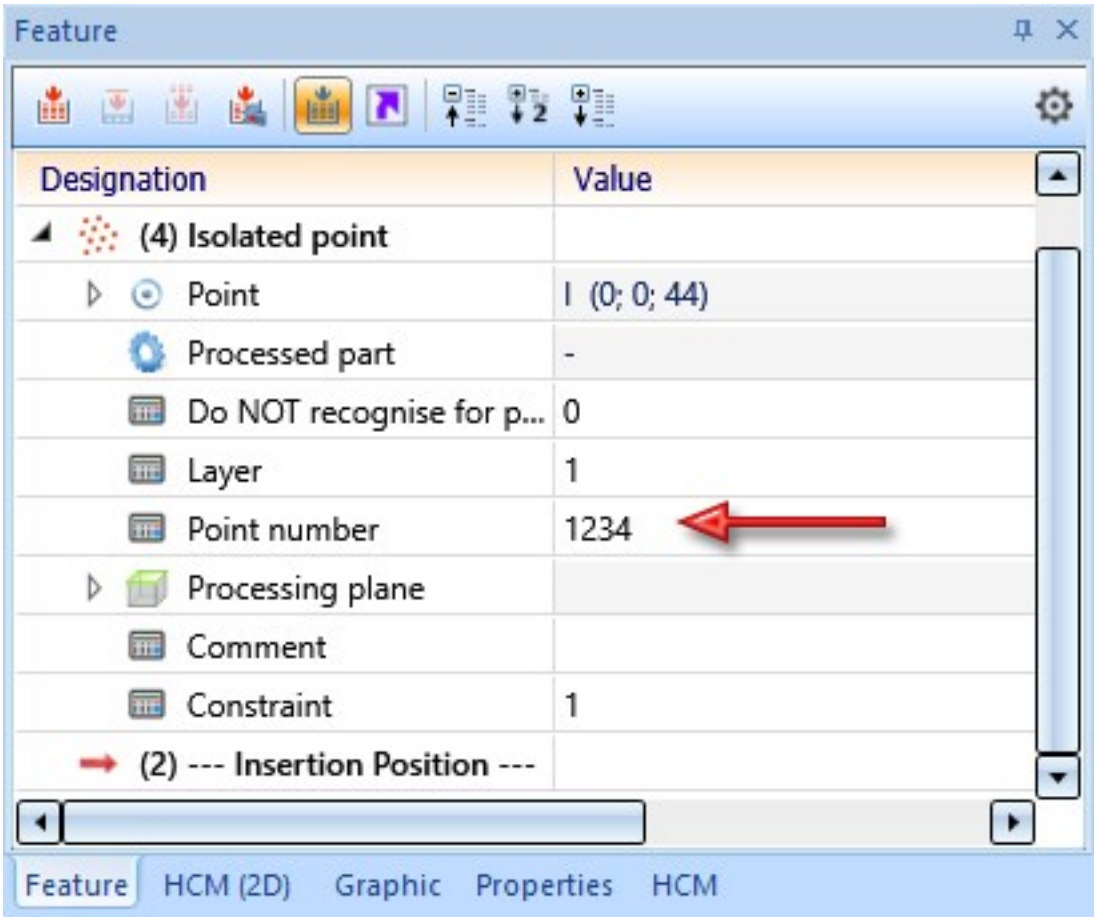
- In addition to the selection of individual edges and surfaces, additional elements can now be selected for filleting via a context menu:



- For edges, the edge direction is visualized. This depends on the cursor setting when selecting the edge.
- Intermediate points for variable filleting can be specified either as a percentage of the length of the edge to be filleted or as a point option. Points that are determined via a point option do not have to be on the composite edge. They are projected automatically.
- It is also possible to fillet segments of edges. The start and end points of a segment can be specified as a percentage or as a point option, just like the intermediate points in variable filleting. However, this applies only to constant filleting!
- Variables and formulas can be used in all input fields.
- The preview has been improved and can now be updated either automatically or by clicking.
- Multiple selection is possible when changing parameters of elements that have already been selected for filleting.
- In Element snap mode, the **Fillet** function is now also available in the context menu.
- The **Rework 3-D corners** function is no longer available as of HiCAD 2021 SP1.

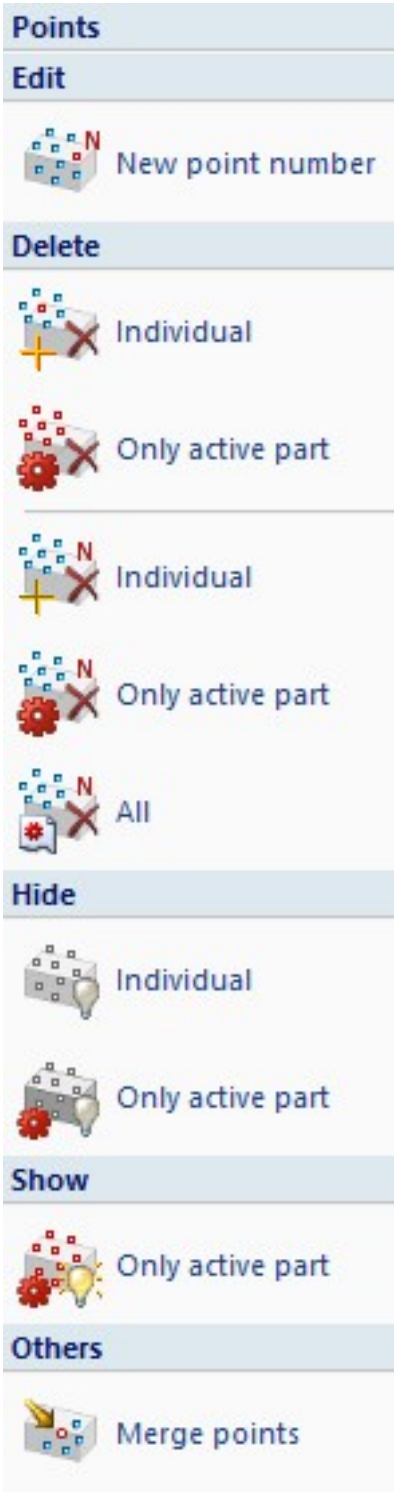
Isolated points - Point number

Previously, when assigning a point number to an isolated point, a feature called **Assign point number** was entered into the feature log of the corresponding part - at the same level as the **Isolated point** feature. This behaviour has changed with SP1. When creating an isolated point, the feature parameter **Point number** is now always entered into the feature Isolated point. If a point number is assigned, then the parameter value is adjusted accordingly. Due to this new behaviour, the deletion of point numbers is now also taken into account during feature recalculation.



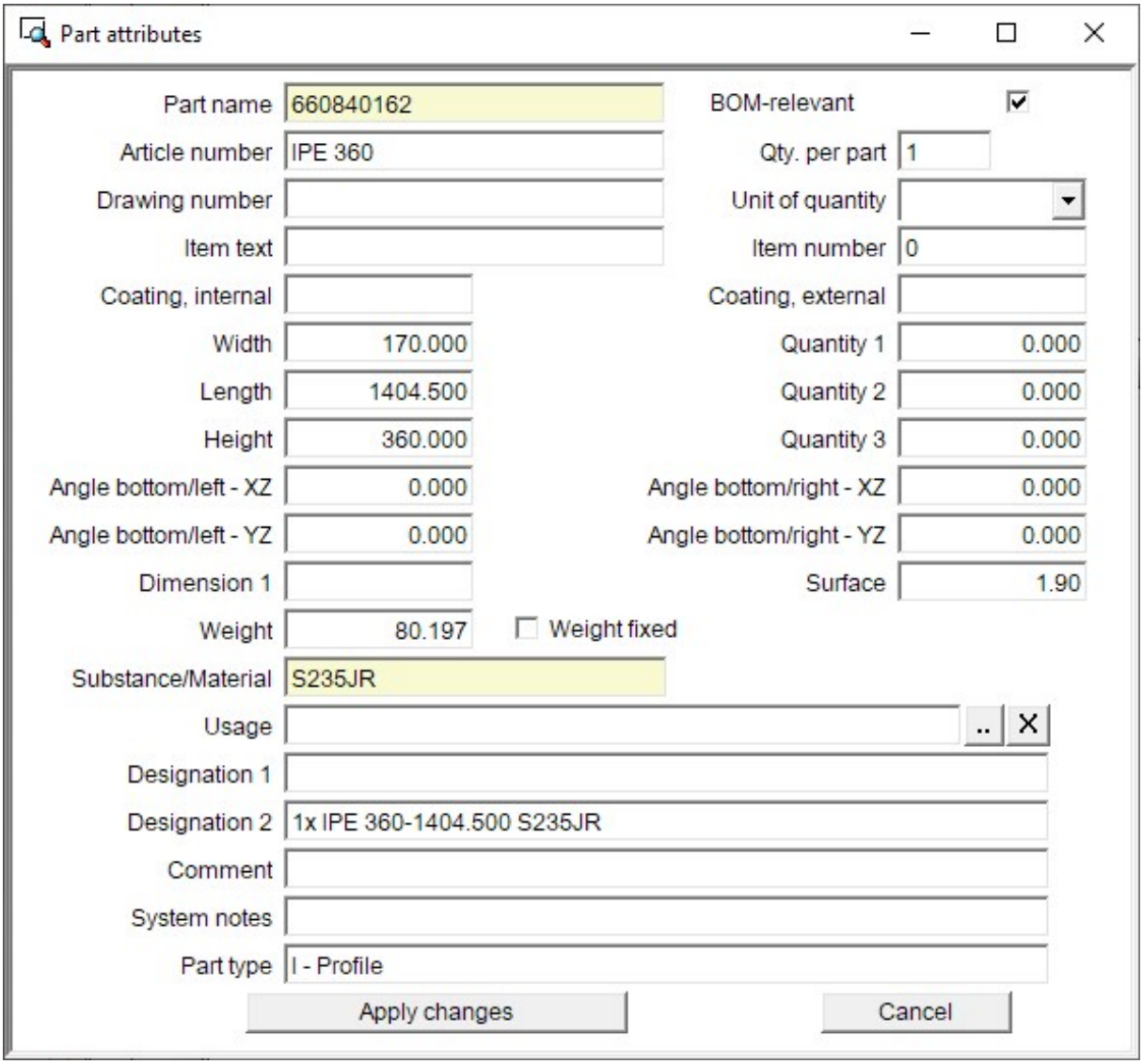
Further information on this topic can be found in the news for Feature Technology.

In this context, the texts in the menu **3-D Standard > Tools > Point** have also been standardized.



Part attributes - Delete usage

In the Part attributes masks, fields highlighted in light yellow are fields that are locked for manual input. This applies, for example, to the **Usage** type. Here you only have the option of selecting a usage type from the catalogue. For several of these fields, the content can now be deleted by clicking on the button..



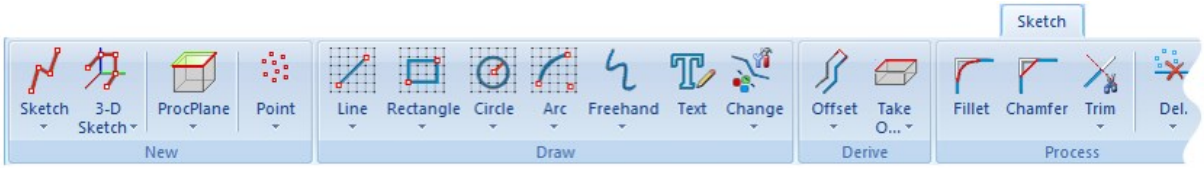
Sketches

Combined Sketching Tools

The sketching tools for planar sketches and 3-D sketches have been unified and combined into one function. This means that a sketching tool with a modern look and modern functions is now also available for planar sketches. The **Sketching Tool** has also been significantly expanded, so that many functions that could previously be called up separately are now replaced by the new Sketching Tool.

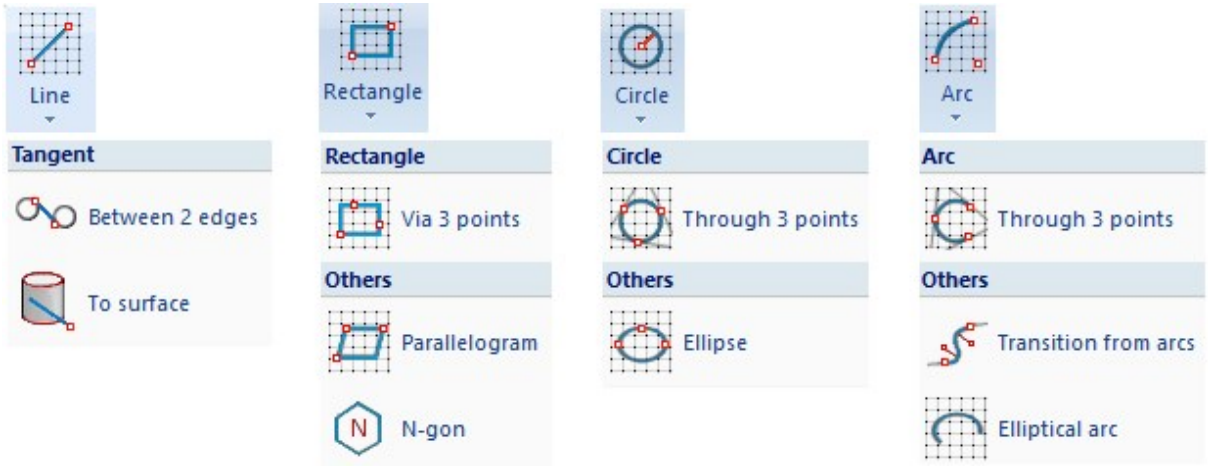
This concerns the following sketching elements:

- Lines and polylines,
- Rectangles and Parallelograms,
- Circles and ellipses, and
- Arcs of circles and ellipses

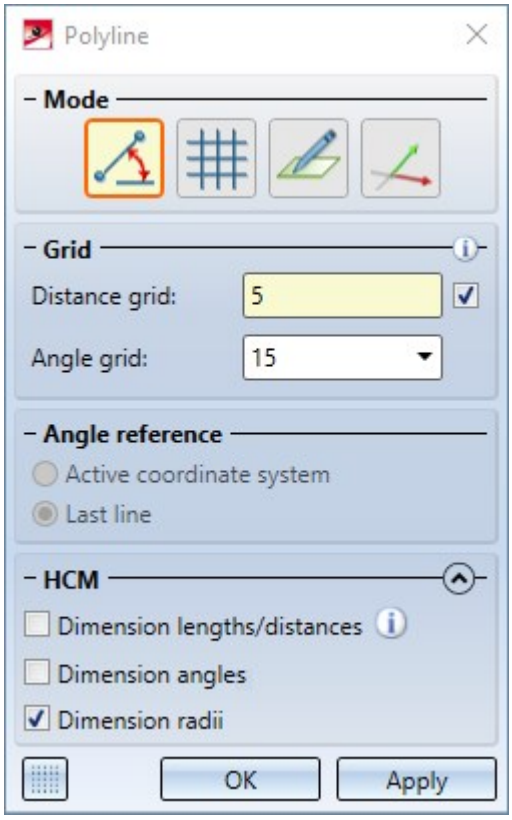


Changed Ribbon tab

The Sketching Tool is available for all drawing functions whose function icon has a grid as background.



Changed menu structure



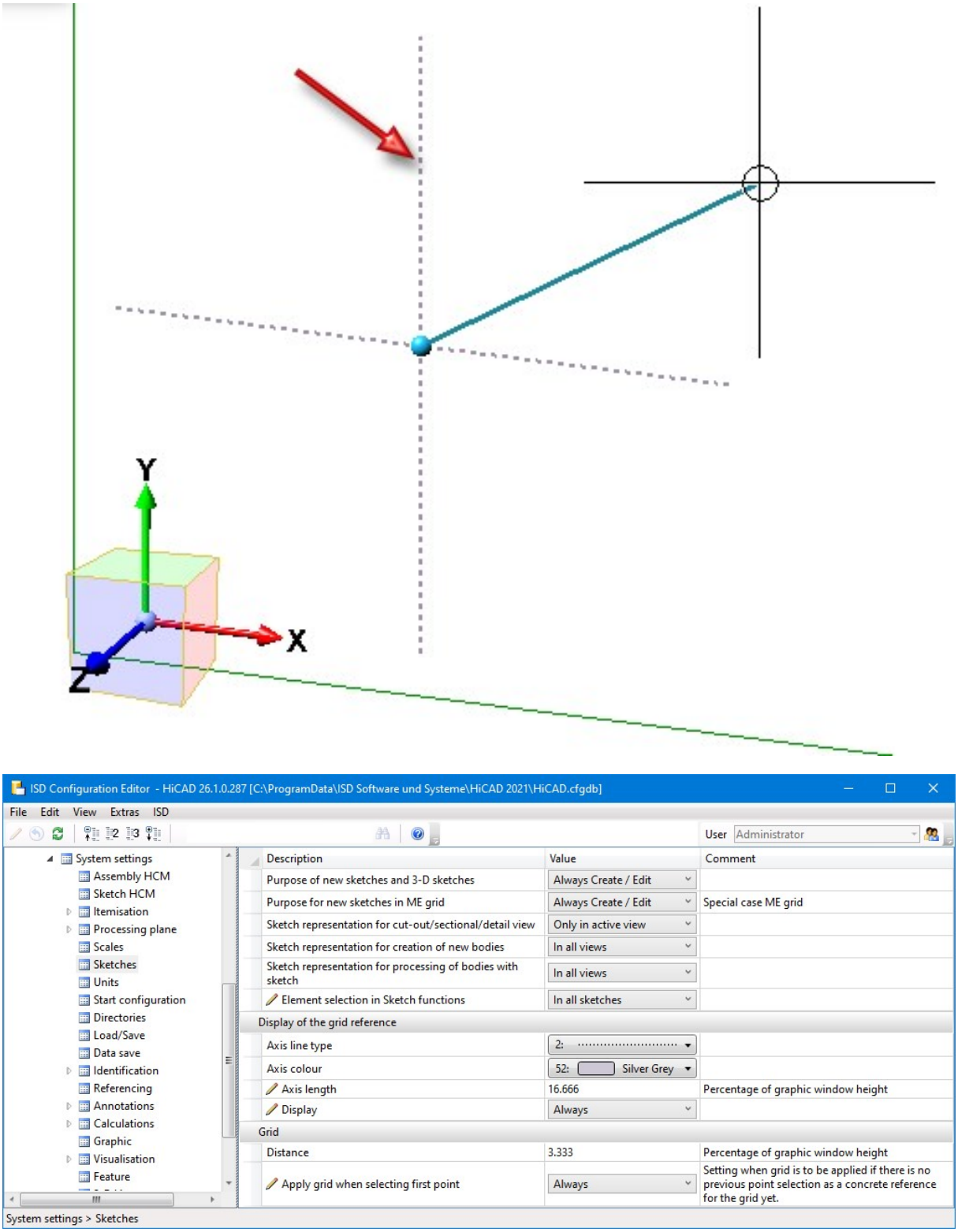
Dialogue window of the Sketching Tool for polylines

The following advantages are particularly noteworthy:

- All functions work with grids - where it makes sense.
- Individual distance grids can be used.
- Switching from drawing on a grid to free drawing can be done directly with the Sketching Tool. There is no need to start an additional function.
- Arcs can now be attached to a line directly by selecting the last point.
- New polylines within a sketch can be started directly without having to call the function again.
- HCM constraints can be set automatically directly at the drawing. This can be activated / deactivated in the dialogue window for each sketch element. While drawing sketch elements, this automatism can also be switched off for individual elements by simultaneously holding down the SHIFT key.

Display of the grid reference when sketching

The display of the grid reference when creating 2-D and 3-D sketches with the Sketching Tool can be set in the Configuration Editor at **System settings > Sketches** and there in the section **Display of the grid reference**.

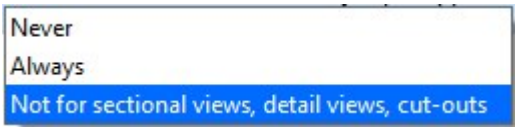


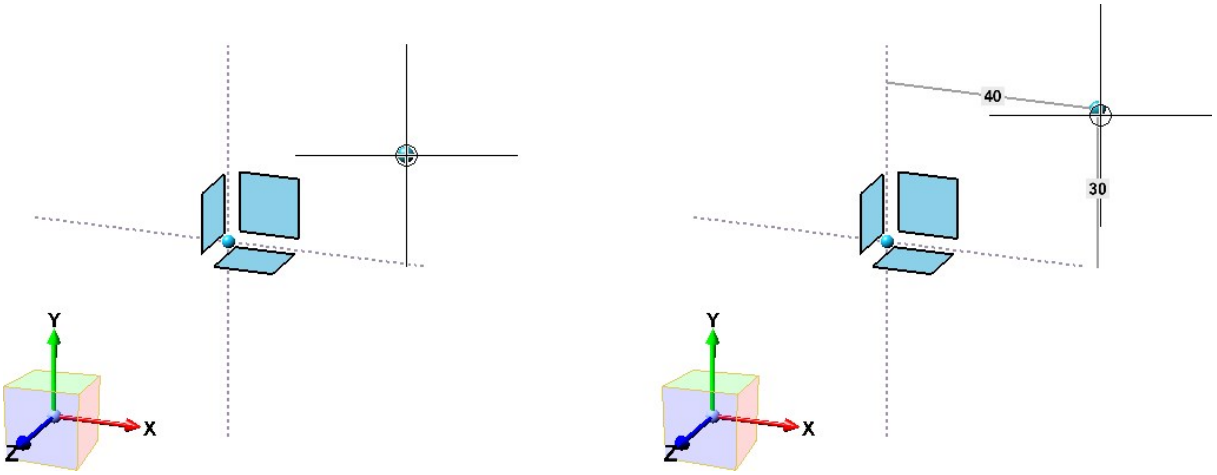
The image shows the ISD default settings.

You can set the line type and colour of the axes and the axis length. At **Display** you can choose when the grid reference should be displayed: .

The grid of the first point and the fineness of the distance grid can also be set. The setting for the distance will take effect when HiCAD is restarted.

The following settings are possible for the first point:



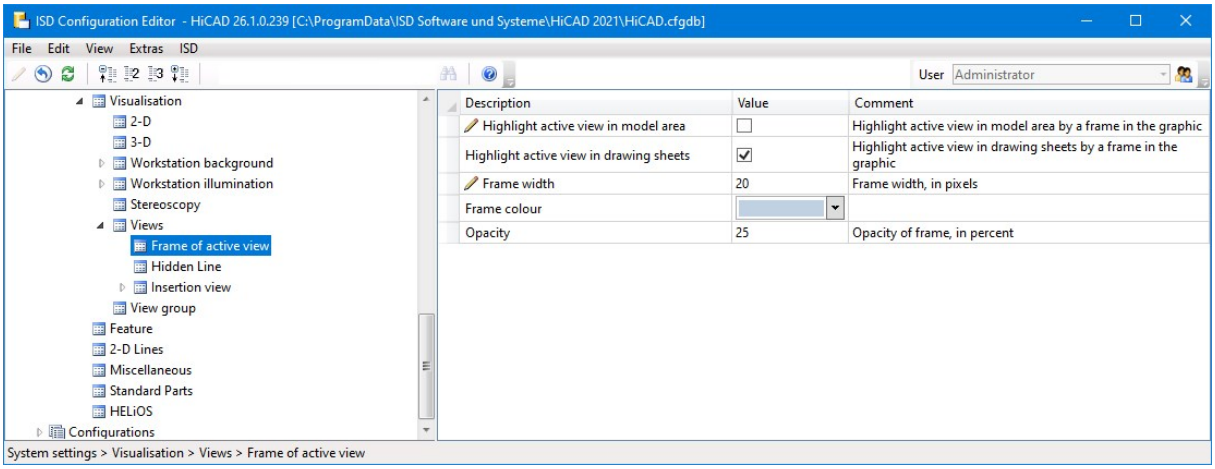


First point, without (left) and with (right) grid display

Views

Representation of the active view

In the Configuration Editor at **System settings > Visualisation > Views > Frame of active view** it is now possible to define for the model space as well as for the drawing sheets (sheet views) whether the active view should be highlighted by a coloured frame in the drawing.



If one of the **Highlight active view in...** checkboxes is deactivated, the active view is displayed by a thin, dashed frame in the Special colour **Marking 1**. This corresponds to the display of the active view before HiCAD 2021.

Zoom to sectional / detail view

Since HiCAD 2021, a sectional or detail view belonging to the active view can be displayed as large as possible with the functions **Zoom to sectional view** or **Zoom to detail view**. The sectional or detailed view then becomes the active view.

From SP1 onwards this also applies if the sectional view is on a different sheet than the original view.

To use the zoom functions, right-click on the annotation, designation or direction symbol in the original view of a sectional/detail view.

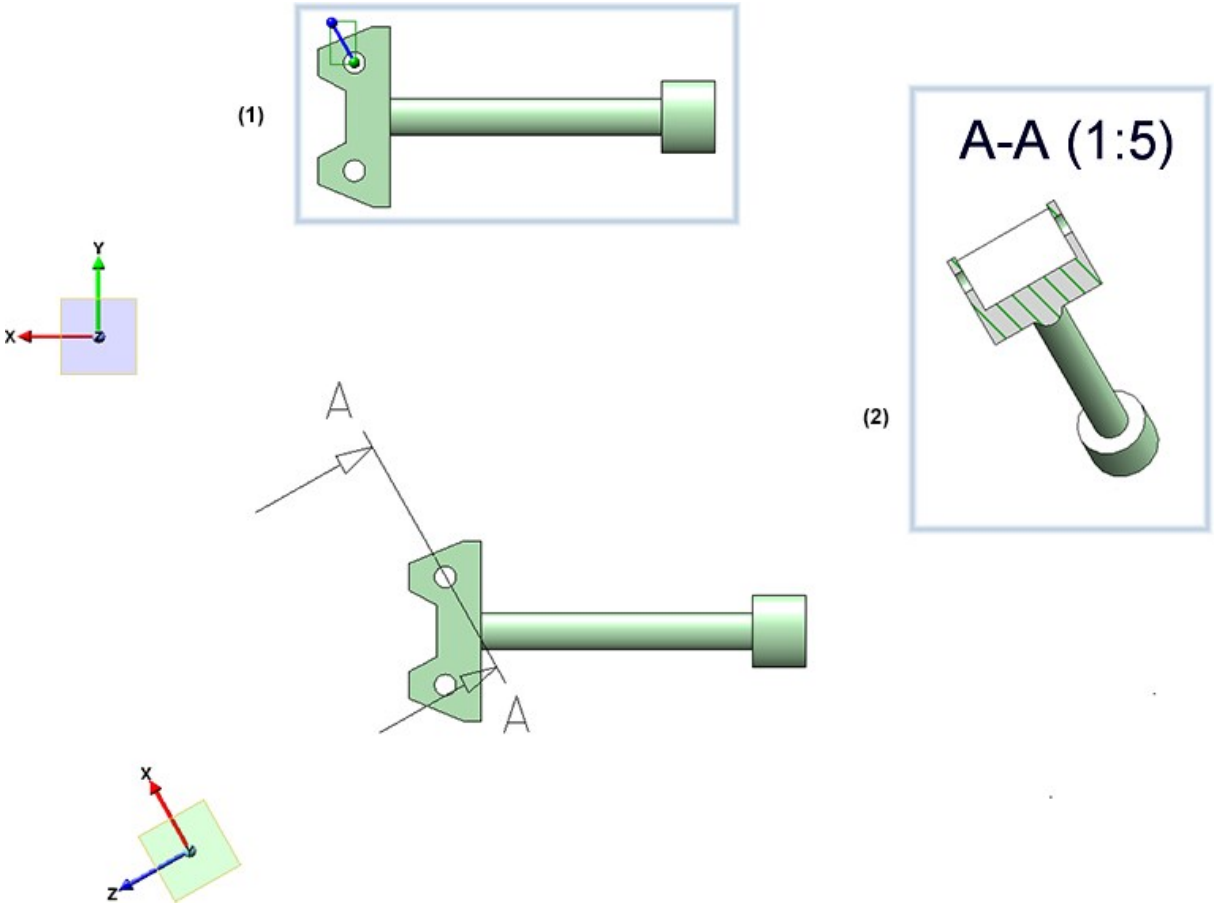
Change of section path in sectional view

If the viewing direction in sectional views is perpendicular to the leading cut plane, then it will be tracked from SP1 onwards when the section path is changed manually, i.e. it will also be perpendicular to the cut plane after the change. This is especially important for variable-controlled models.

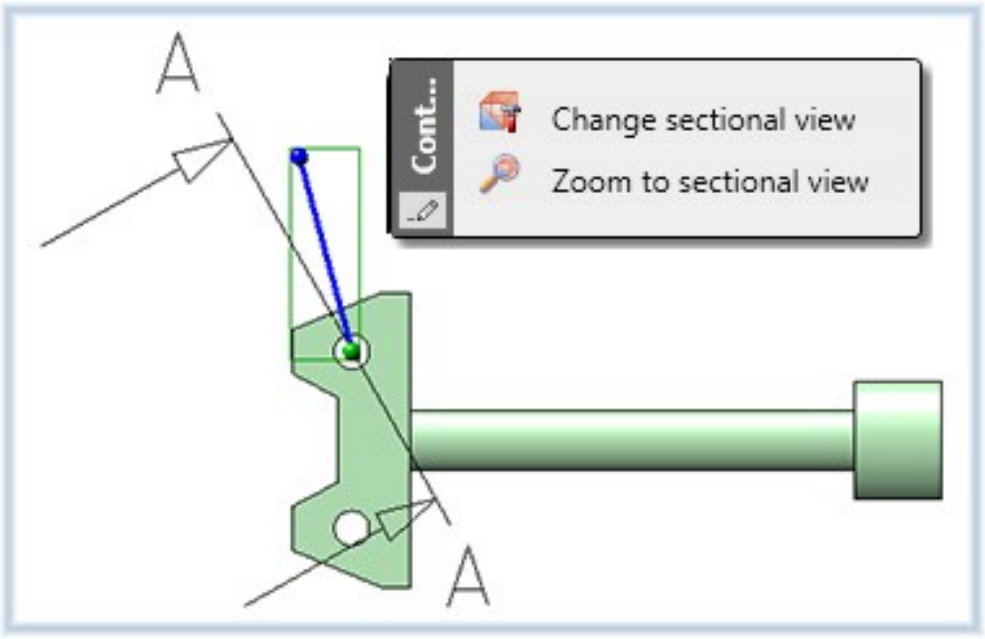
However, if you have changed the viewing direction manually, then it will not be changed.

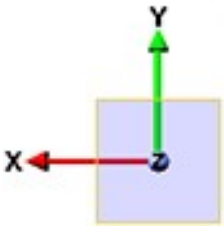
Example:

The image shows the original view with the sketch for the section path (1) and the generated sectional view (2).

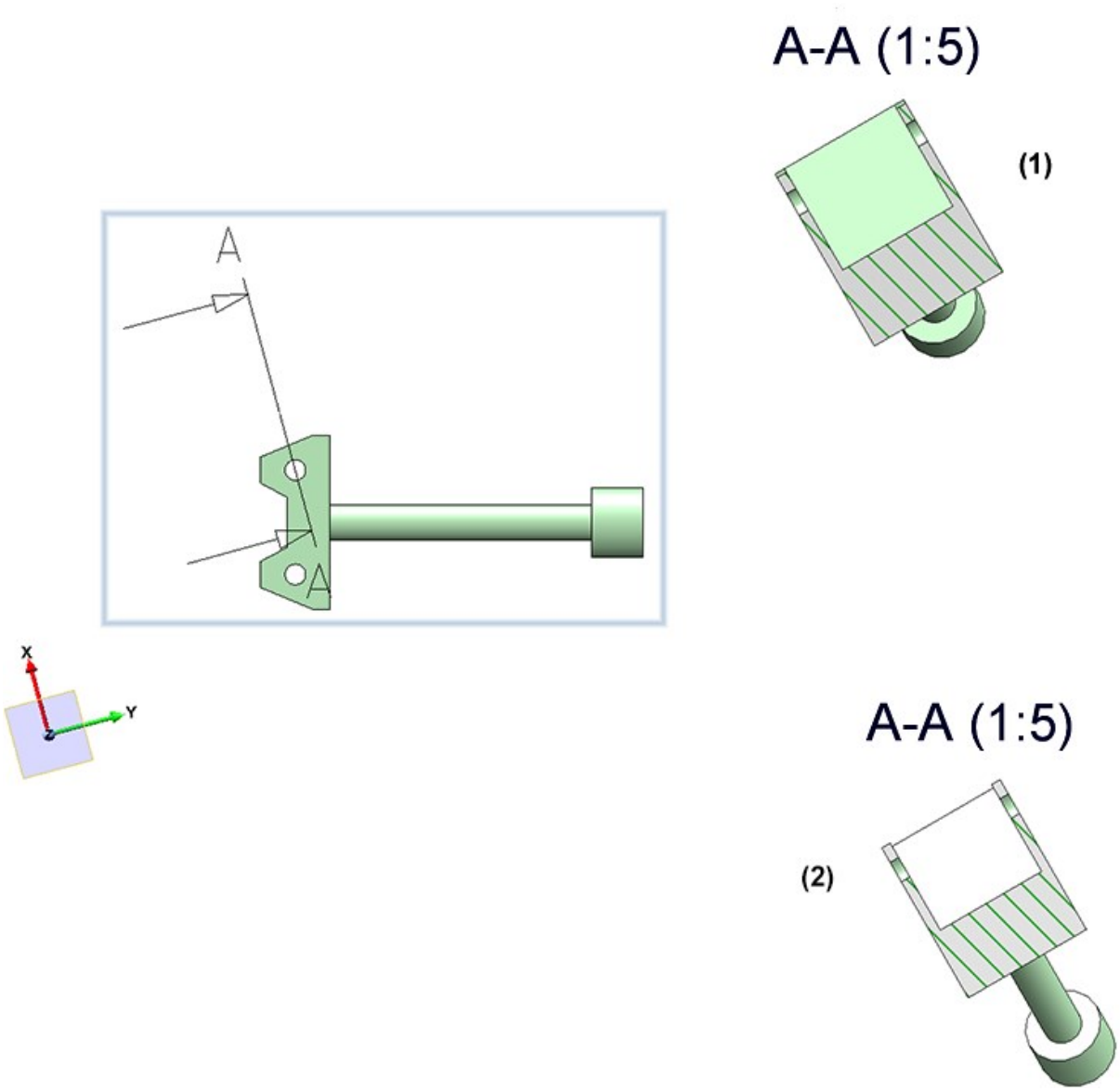


Now the section path is changed by changing the sketch.





The following image shows the result in HiCAD 2021 SP1 (1) and in HiCAD 2020 (2).



Plane position of detail view when changing the section path

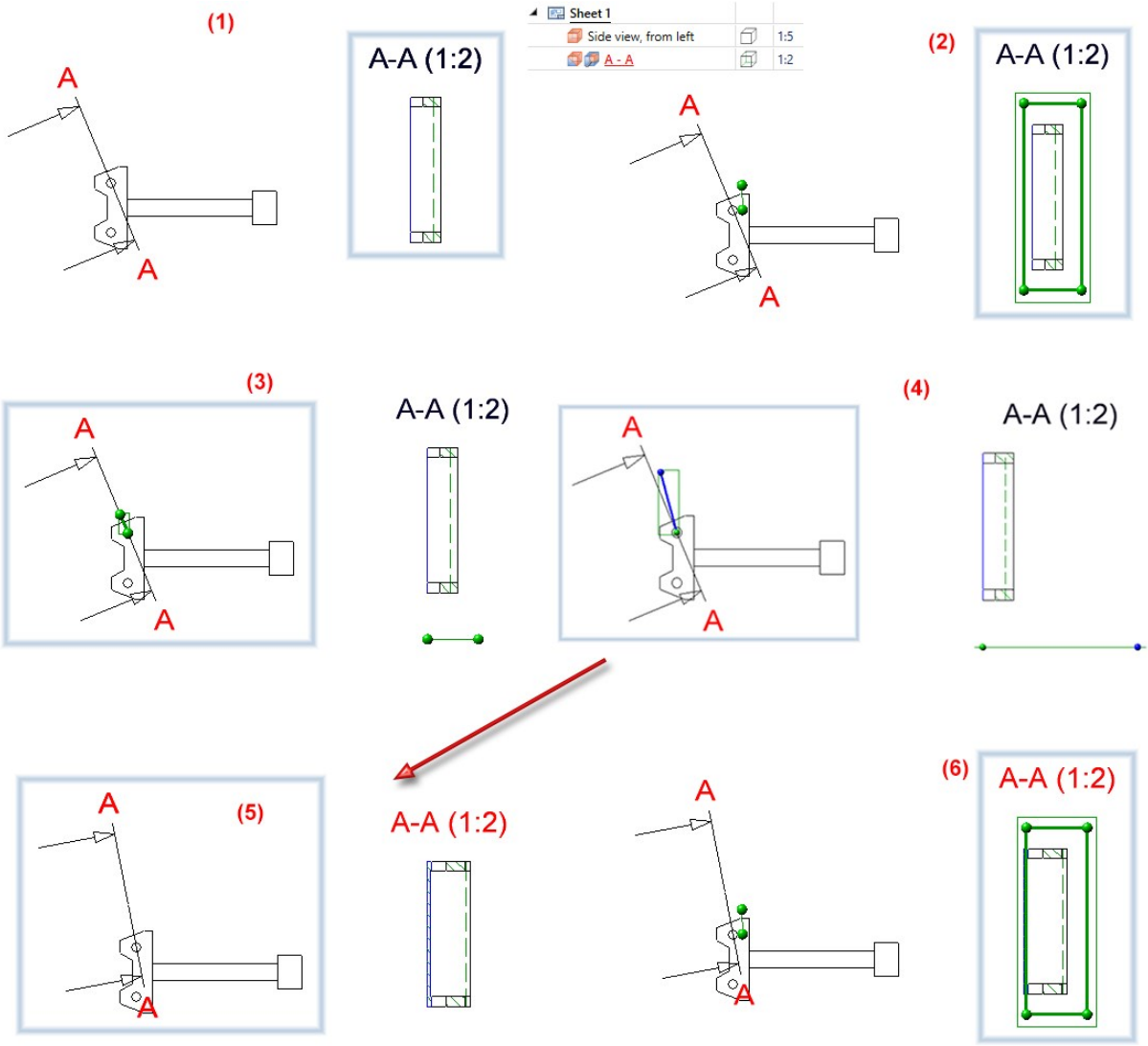
For limited section views, the limiting sketch (i.e. the sketch for the detail shown) is in the leading cut plane. This is usually also the case when you have extracted a detail (with the sketch as detail section) from a section view.

If the leading cut plane now changes, then from SP1 the sketch for the detail section is adjusted so that it is again in the cut plane.

Example:

The image shows a limited sectional view (1) and the sketch plane in the detail view (2).

Now use the function **Change sectional view** to change the sketch (3) as shown (4). (5) shows the changed section path, (6) the sketch plane after calling the function **Change detail view**.



This enhancement is especially important for variable-controlled models.

Position of detail views when moving the detail

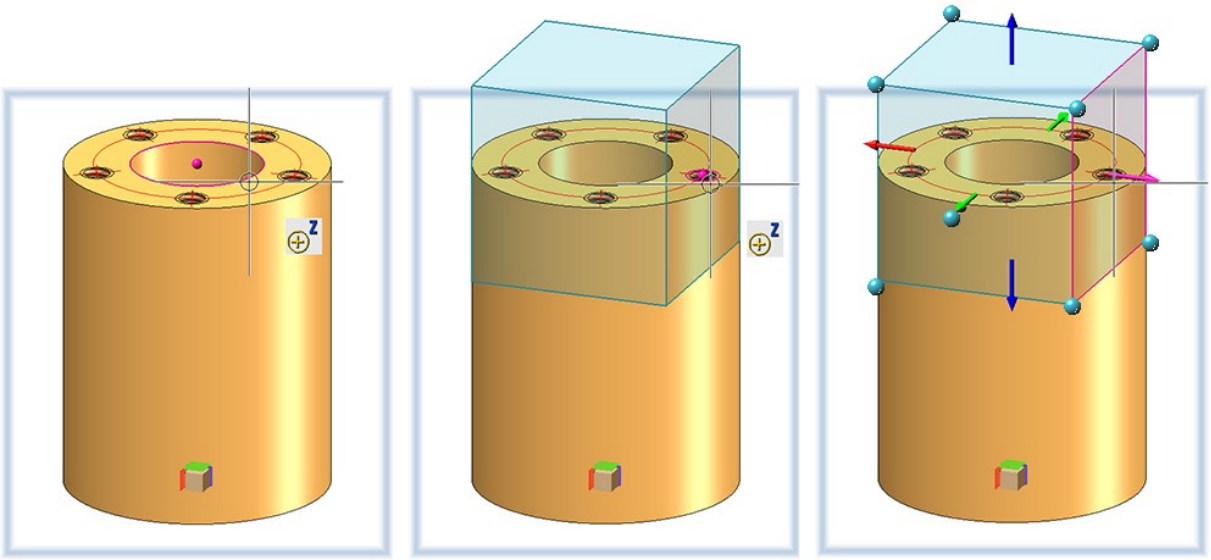
Until now the position of detail views and limited sectional views changed in an undesired way when the section in the model moved, e.g. due to the entered HCM constraints.

From SP1 onwards, the centre of gravity of the sketch or - in the case of detailed view cuboid/sphere - of the cuboid/sphere is used as a reference point and remains in this position in the drawing if possible.

This enhancement is particularly important for variable-controlled models.

Adapt bounding box size for detail views

With the function **Detail view, Cuboid/sphere**, point symbols are displayed at the corners of the bounding box from SP1.



The box size can thus also be influenced by dragging the point symbols . You can use the point options of the Autopilot and the **Point options** menu.

Dimensioning / Annotations / Text

Delete standard dimensions

With the new function **Delete standard dimensions, In active sheet** all dimensions in the views of the active sheet can be deleted in one step. You can find the function at **3-D Dimensioning + Text > Delete**.

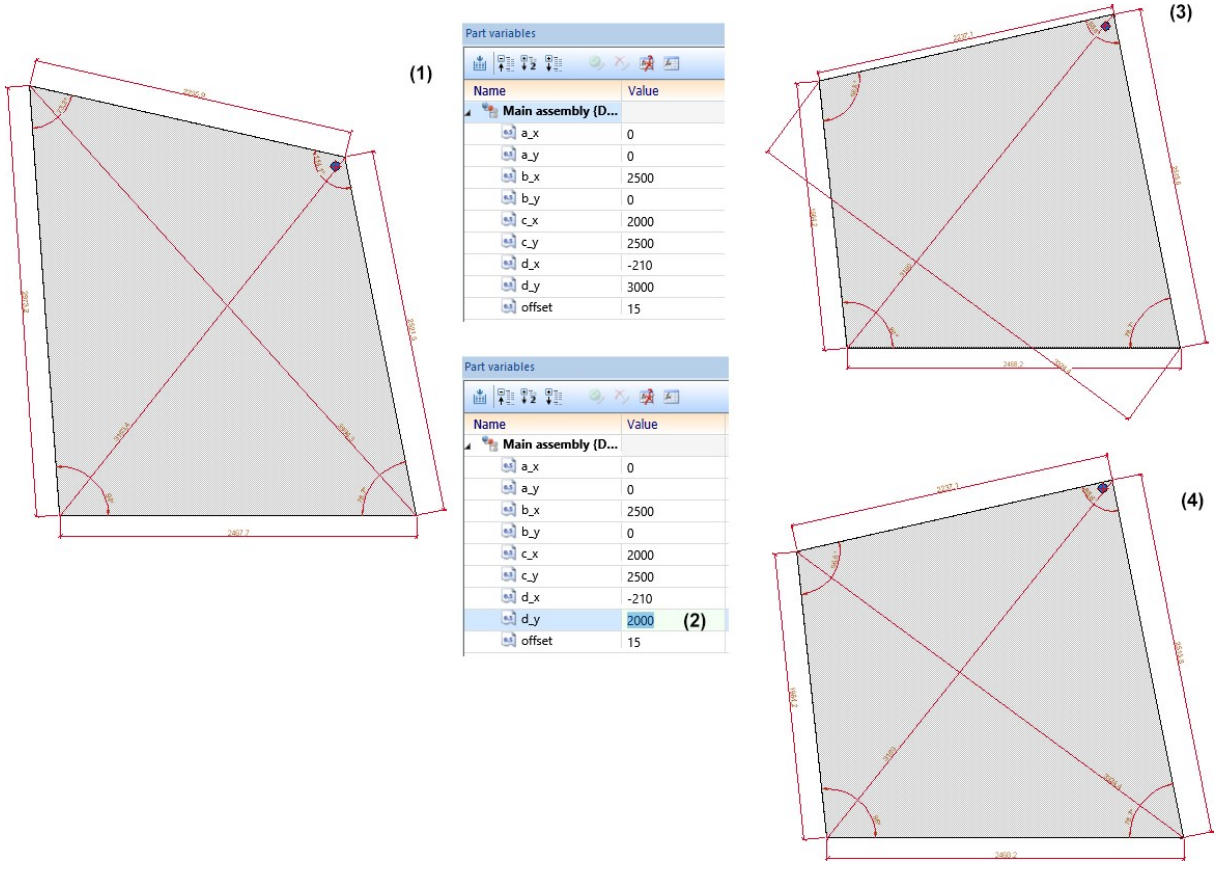
Behaviour of drawing dimensions when changing the reference geometry

If the form and position of parts to which drawing dimensions refer changes, then the position of the dimensions may also change. Previously, individual dimensions were placed in relation to their first base point. When the geometry was changed, the new dimension was placed in such a way that the length of the dimension auxiliary line of the first base point was again as in the unchanged state. In many cases this led to rather unaesthetic results. As of SP1, the placement logic of the measurements has changed. The length of the shortest projection line after the change should correspond to the length of the shortest projection line before the change. The case can also occur that the shortest projection line is assumed at different base points.

Example:

The image shows a dimensioned assembly (1) with different part variables. Now the value of one variable is changed, e.g. d_y from 3000 to 2000.

(3) shows the subsequent dimensioning in HiCAD 2020, (4) the result in HiCAD 2021 SP1.



The behaviour described above applies analogously to structural dimensions. As a rule, the smallest distance between the base point and the dimension line is also retained here, whereby all base points of the structure are to be used for this.

Attributes in annotations

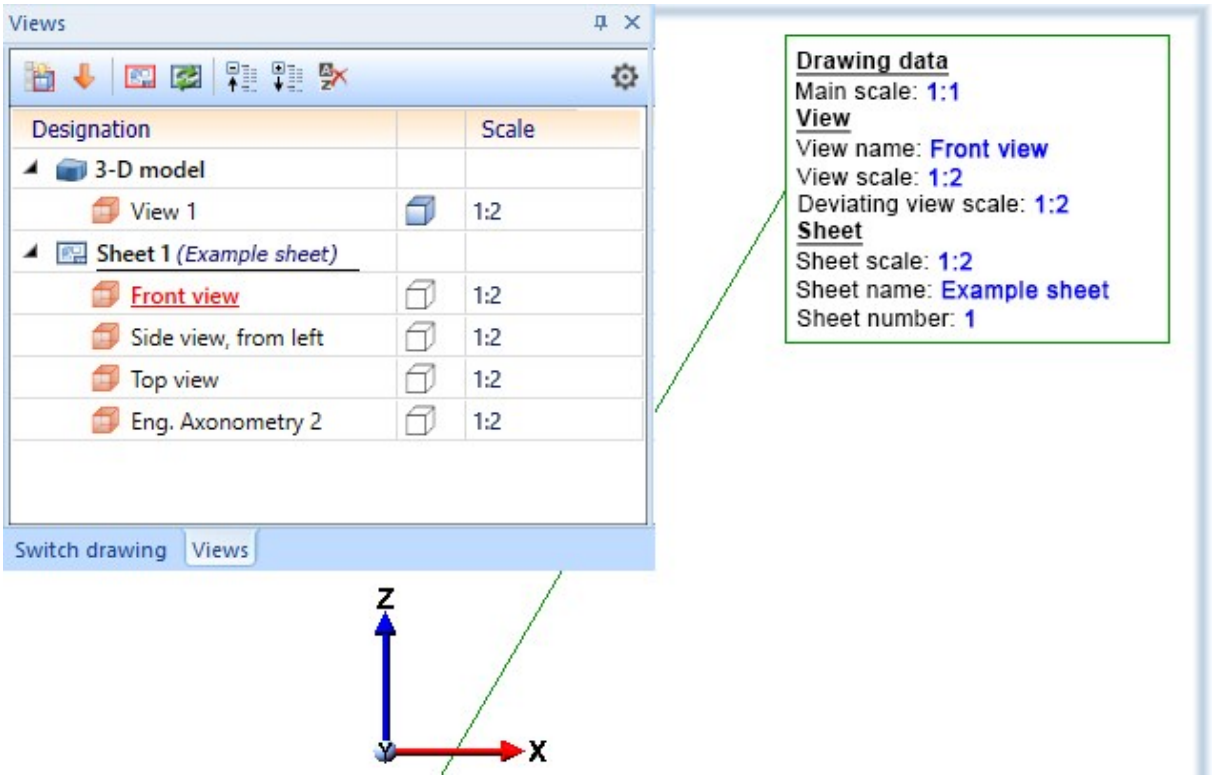
Extended attribute selection in annotations

From SP1 onwards, further attributes can be used in 3-D annotation tags:

- HELiOS item attributes of the part,
- Attributes from the HELiOS document master of the part,
- Drawing property, e.g. main scale, name and number of the drawing sheet on which the annotation is located and name and scale of the view and
- Attributes of the HELiOS folder to which the model drawing belongs.

Note on drawing attributes:

A different view scale only exists if the view scale and main scale are different. The sheet scale only exists if all views on the sheet have the same scale. Not relevant are sectional and detail views, views of sheet metal part developments and axonometries in workshop drawings. The sheet name only exists if a name has been assigned to the sheet.



Expanded attribute selection in annotations

Fom SP1 on, further attributes can be used in 3-D annotation tags:

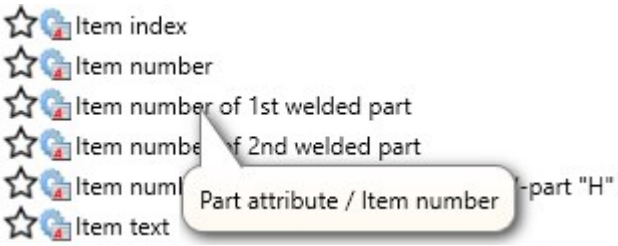
- HELiOS item attributes of parts,
- Attributes from the HELiOS document masters of parts,
- Drawing information, e.g. name and number of the drawing sheet on which the annotations have been created and
- Attributes of the HELiOS folder to which the model drawing belongs

Improved attribute selection

Each attribute is marked with a symbol indicating the attribute group.

	Part attribute	HiCAD
	Attribute from the article master of the part	HELiOS
	Attribute from the document master of the part	HELiOS
	Item attribute of the part	HELiOS
	Catalogue e.g. catalogue designation	HiCAD
	User-defined Plant Engineering attribute	HiCAD
	Drawing attribute	HiCAD
	Drawing information, e.g. view scale, sheet name and sheet number of the drawing sheet on which the annotation has been made	HiCAD
	Document master of the model drawing	HELiOS
	Project attribute	HELiOS
	Folder attribute	HELiOS

When you move the cursor over an attribute, a tooltip will be displayed, showing you to which attribute group the attribute belongs.

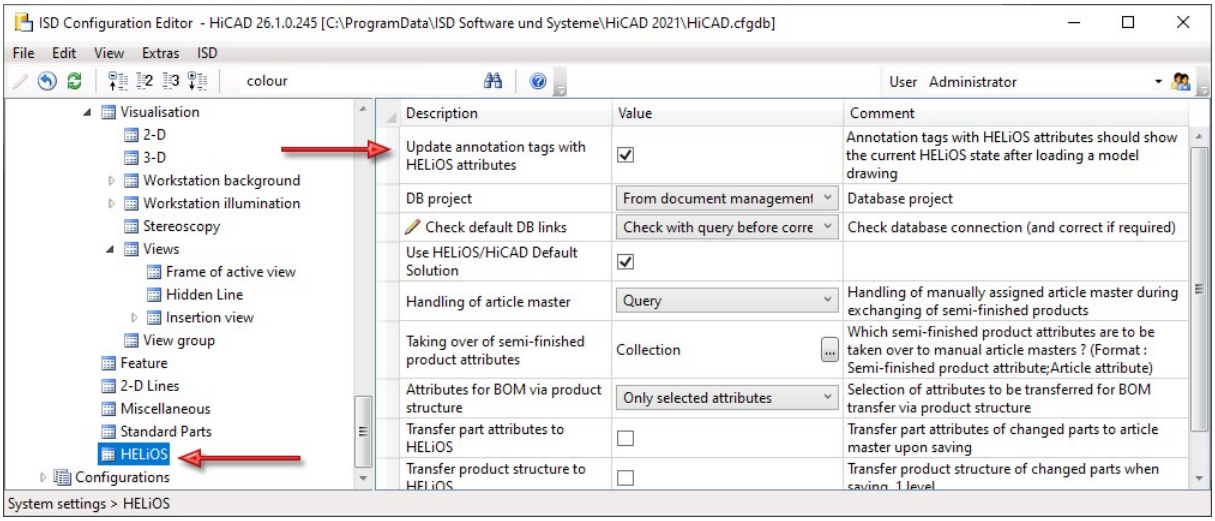


Favourites

If attributes are marked as favourites in the Annotation Editor, the list of favourites is automatically expanded when you click on the **Attributes** button.

HELiOS attributes in annotations

If a model drawing contains annotations with HELiOS attributes, then from SP1 these are automatically updated when the drawing is loaded. This behaviour can be switched off in the Configuration Editor at System settings > HELiOS. There you can deactivate the **Update annotation tags with HELiOS attributes** checkbox.

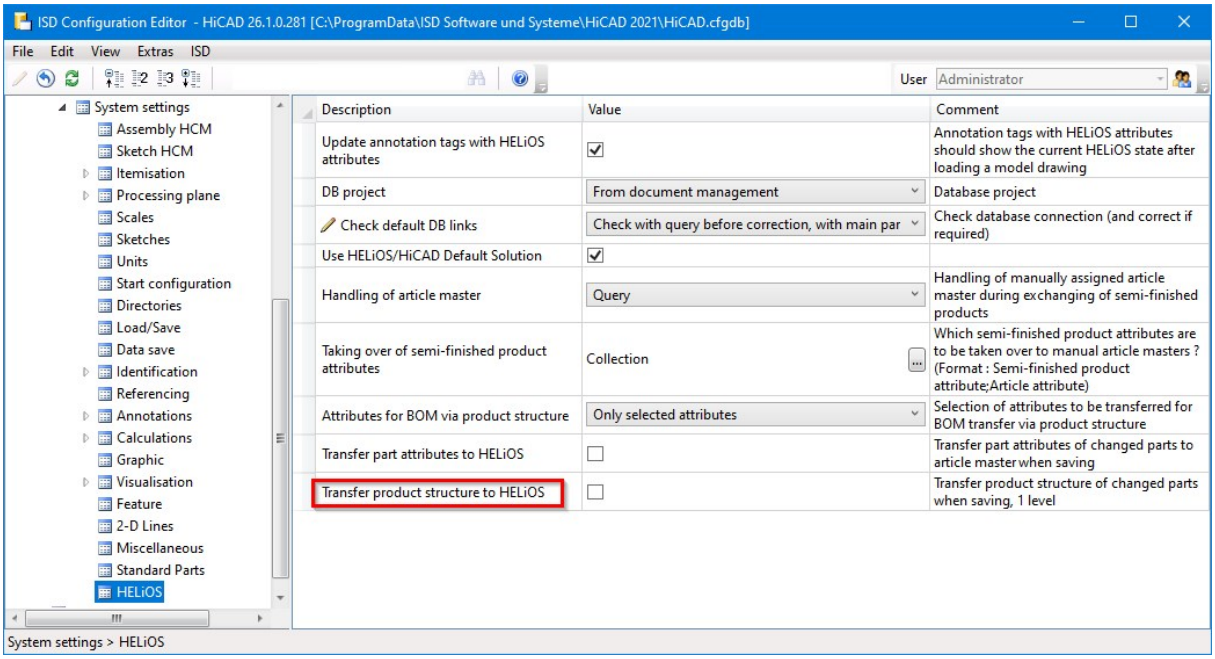


Already released drawings will not be updated!

Referencing

Product structure to HELiOS

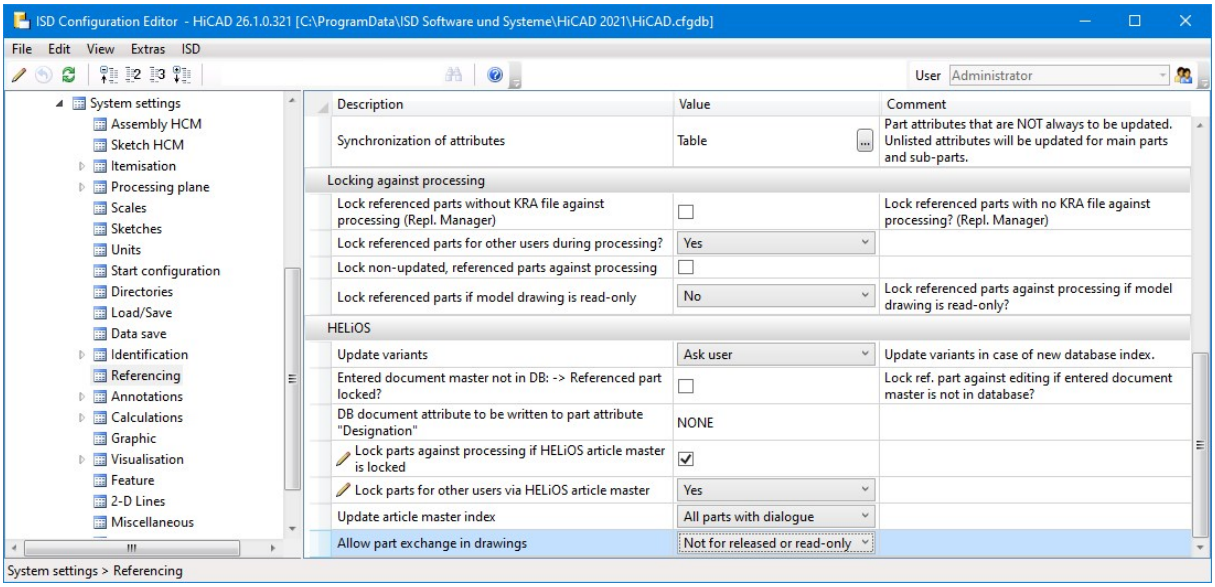
In the Configuration Editor (ISDConfigEditor.exe), the checkbox **Transfer product structure to HELiOS** is available at **System settings > HELiOS**. If this checkbox is activated, the product structure (**1 level**, i.e. on the first hierarchy level) is automatically transferred to HELiOS when saving parts changed in HiCAD (with article master assignment or with BOM relevance). In this case, the **Product structure to HELiOS** checkbox in the **Save referenced parts** dialogue is active but greyed out. This means that it cannot be deactivated in the dialogue.



Note:
The above configuration setting is only possible if both HiCAD 2021 and HELiOS 2021 have been updated to Service Pack 1. If one of the two applications is available in a version lower than V 2601, the option is not available.

Allow part exchange in drawings

If you use HiCAD in conjunction with HELiOS, the settings for part exchange from the HELiOS **Database** Options on the **HiCAD** tab were previously taken into account when exchanging referenced parts. These settings can now be found in the Configuration Editor at **System settings > Referencing** in the **HELiOS** section.



For users with a HiCAD version before 2021 SP1 (2601), the **Options** in HELiOS are still available.

Major Release 2021 (V 2600)

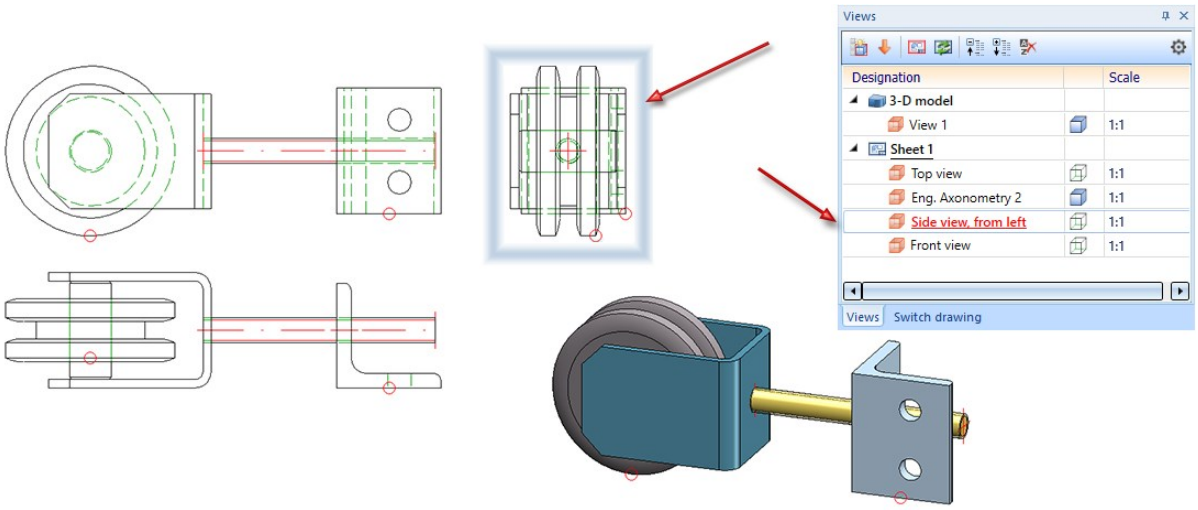
Views

Stereoscopic representation

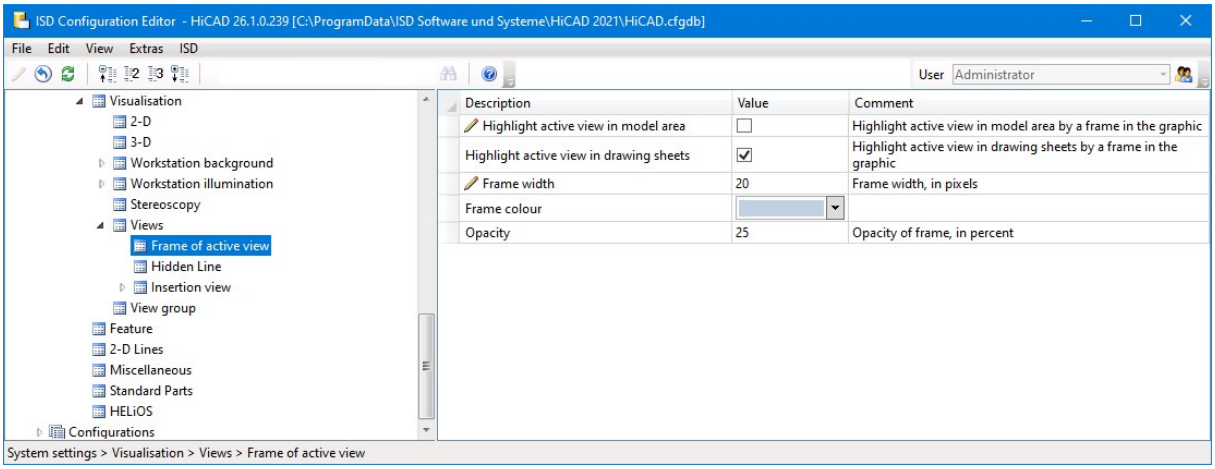
The function **Stereoscopic representation** (previously available at **Views > Properties > Stereo**) is no longer available as of HiCAD 2021.

Representation of the active view

The active view is now represented in the model drawing by a coloured, wider view frame.



The frame display of the active view - colour, line width and transparency of the frame - can be preset in the Configuration Editor at **System settings > Visualisation > Views > Frame of active view**. You determine the colour by specifying the RGB values. There you can also set whether the active view should be marked or not. The image below shows the ISD default settings.



Maximum number of views and Sheet areas

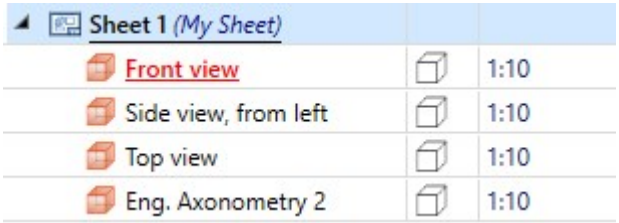
As of HiCAD 2021, a maximum of 9998 Sheet areas and views can be created. The Model area is included in the number of Sheet areas.

Delete all Sheet areas

If all Sheet areas are selected when calling the **Delete** function, they will be deleted without confirmation and a new Sheet 1 will be created - without views.

Change sheet names

When creating new sheets, the name **Sheet *n*** is given by default, where *n* is a consecutive number. With the function **Change sheet name** , a sheet name can also be specified, which is then displayed in brackets behind the sheet designation in the view ICN.

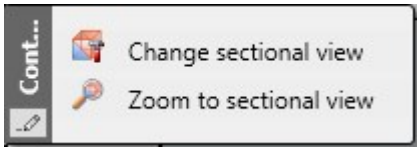


Sheet names can also be used for sheets in the production drawings created with the drawing derivation function. These are retained even when the drawing is updated.

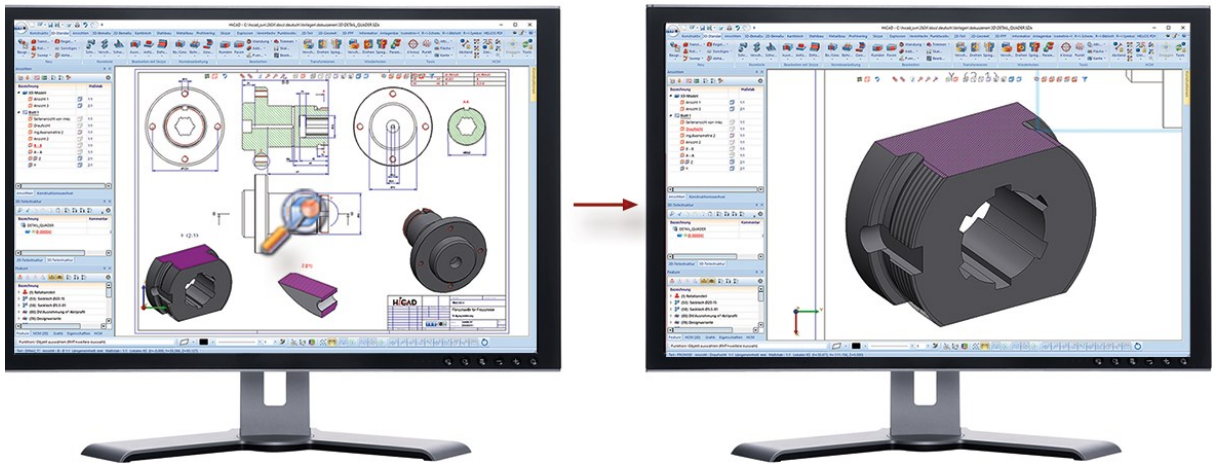
Zoom to sectional/detail view

If you right-click the annotation, the designation or the direction symbol in the original view of a sectional view and then call the

Zoom to sectional view function, the corresponding sectional view is displayed as large as possible and activated.



Similarly, the **Zoom to detail view** function zooms and activates the corresponding detail view accordingly.

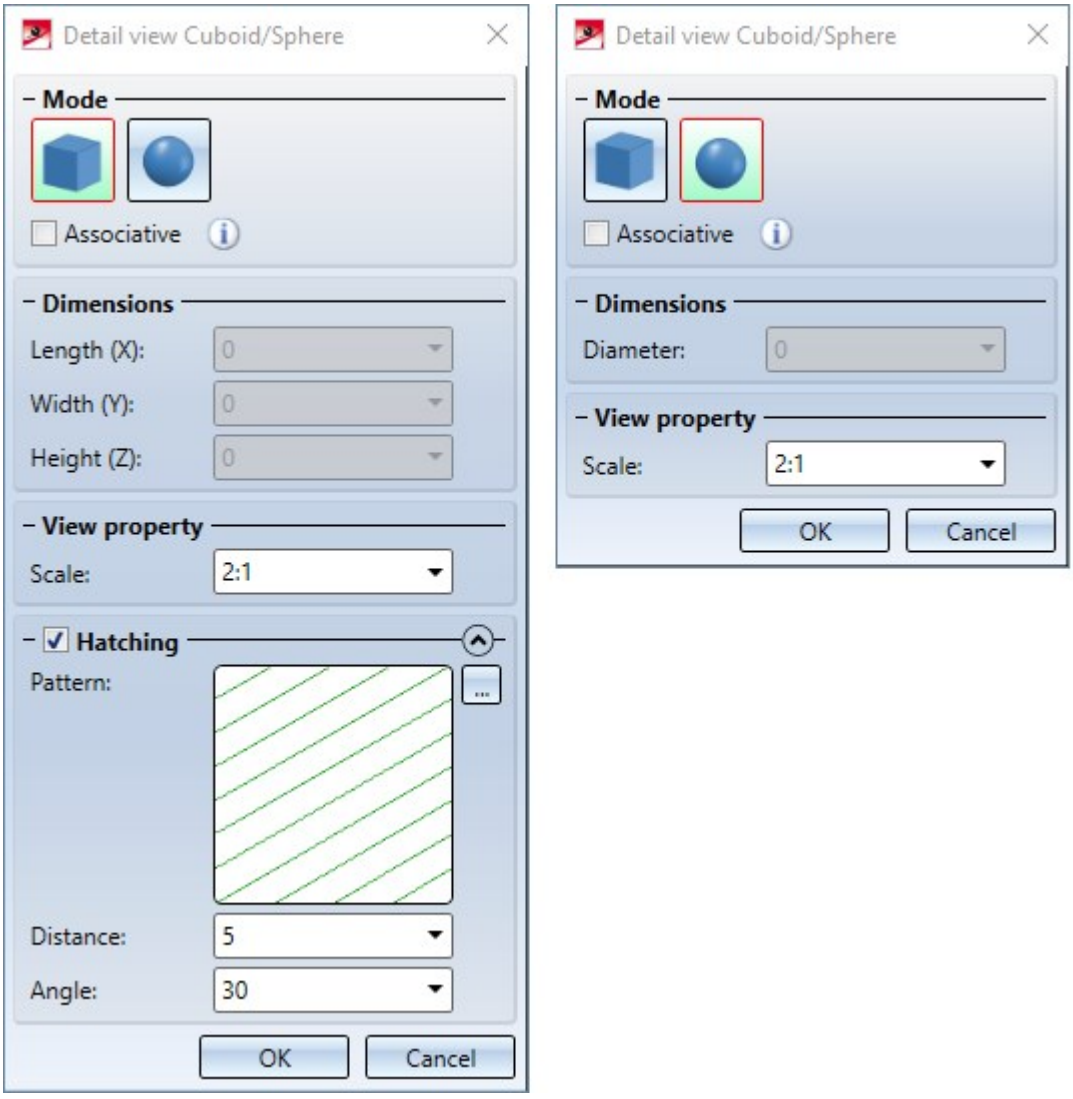


Detail view, Cuboid/Sphere - Associative behaviour

If one or more
HiCAD Point options are used to define the cuboid, then the checkbox **Associative** can be used to define how the detail view should behave if the corresponding points are moved due to changes in the construction.

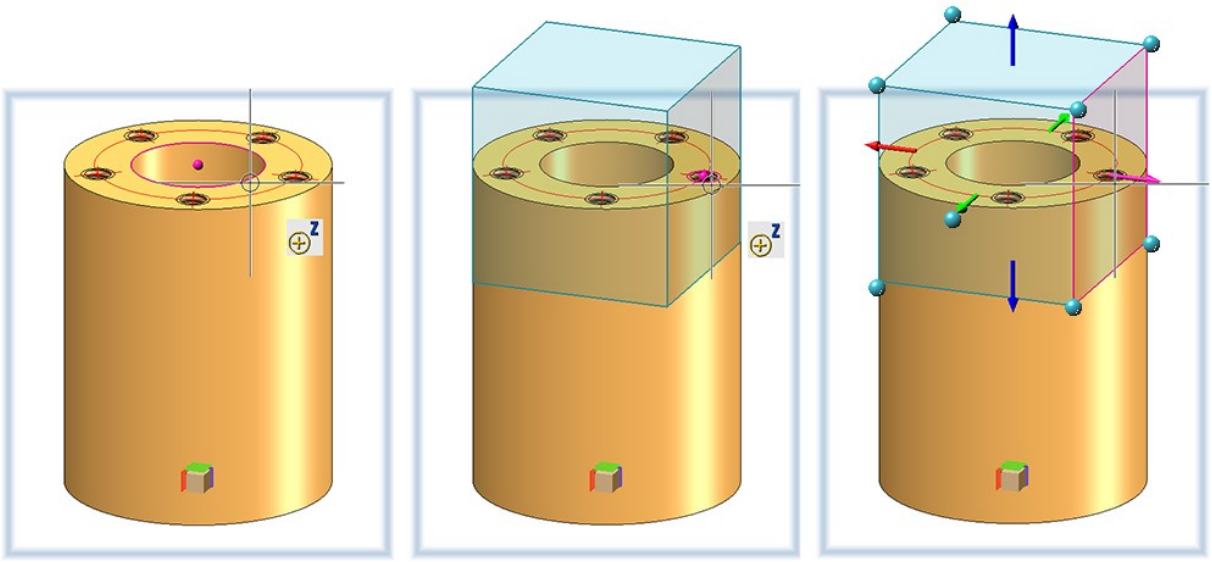
If the checkbox is active, the detail section follows the changes, it can, for example, "go along" with a moved geometry.

This is not the case if the checkbox is inactive (default setting). The detail view always shows the same spatial section, regardless of whether the original geometry is still in that position.

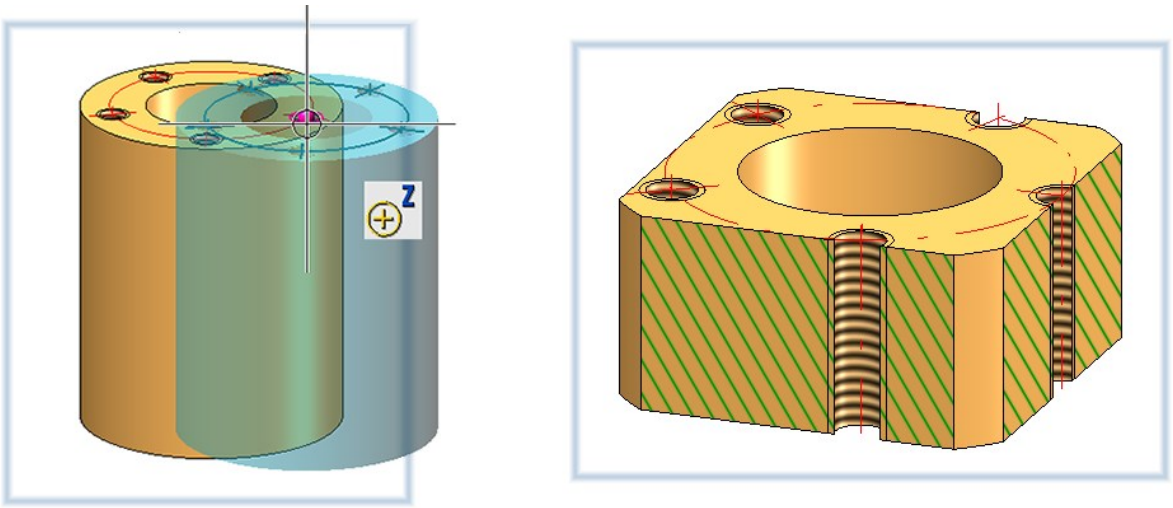


An example:

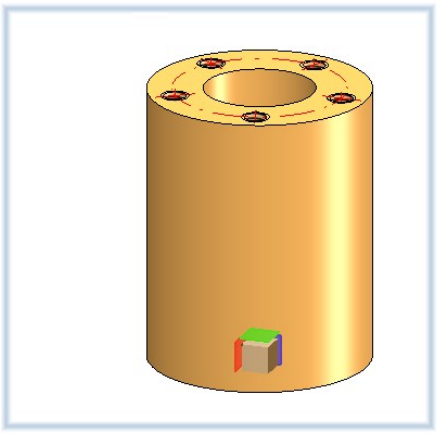
The image below shows the **Detail view, Cuboid** of a cylinder with bore and threaded holes. The centre of the cuboid is in the centre of the cylinder bore. The size of the cuboid is determined by the centre of the right-hand threaded hole.

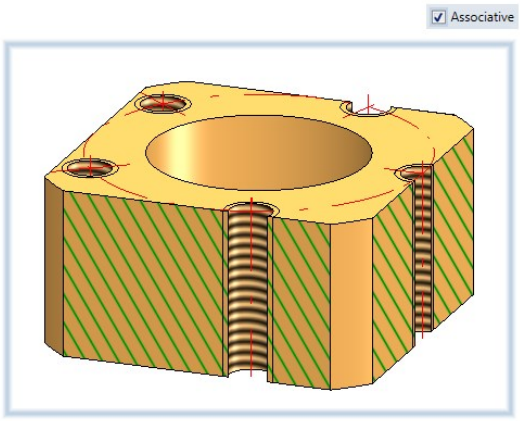


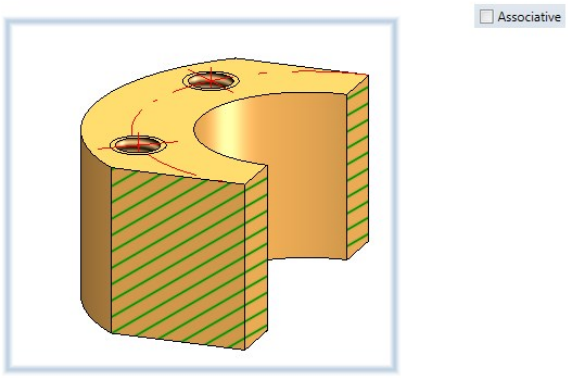
Now the cylinder is moved to the right.



The following image shows the difference between associative and non-associative behaviour after updating the detail view.

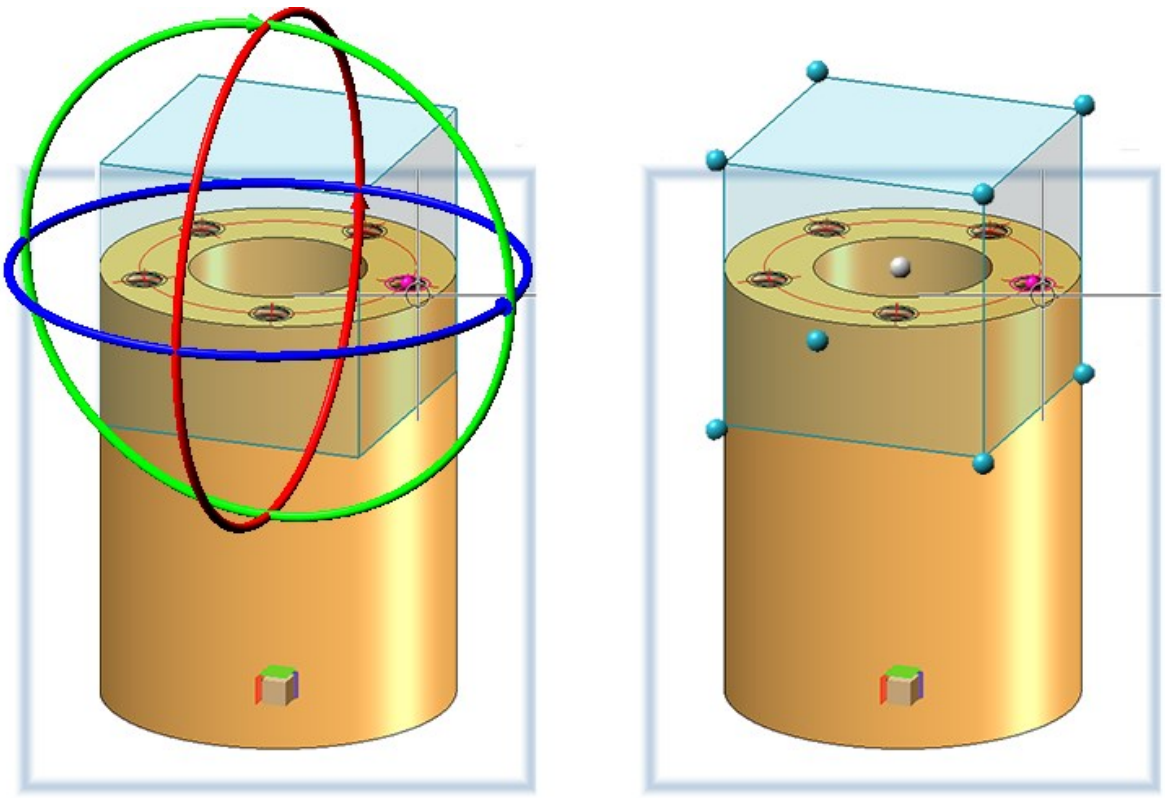






Detail view, Cuboid/Sphere - Rotations

In addition to scaling, i.e. dragging the individual side surfaces, you now also have the option of rotating or moving the cuboid. The corresponding functions are available in the context menu after pressing the right mouse button. If rotate is selected, the cuboid can be rotated around an axis by selecting the rotation circle of the respective axis with the cursor and then rotating it dynamically with the cursor. When moving, select one of the marked points on the cuboid and drag the point to the new position.



If you want to move the sphere, click on the centre point and drag the sphere to the desired position.

Discontinuation of shading type

Up to now, the shading type **Flat** could be used as the simplest and fastest shading method. By changing the renderer to OpenGL version 4.3, the graphics output in HiCAD is accelerated in width, 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.

For this reason, the shading method **Flat** is no longer needed. The function **Properties > Shading type** has therefore been removed from the context menu for views.

Show/Hide Parts

Up to now the functions

Hide parts in active view and

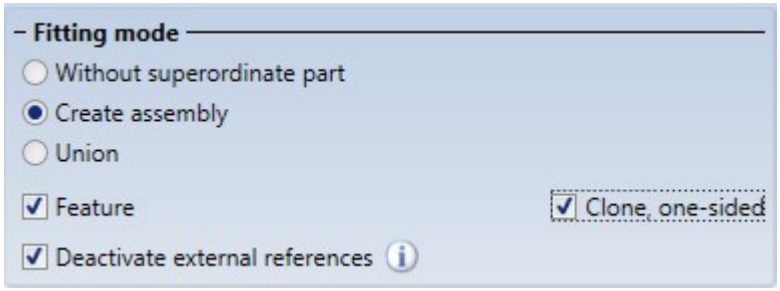
Show parts inactive view

could only be called up via the context menu for parts. As of HiCAD 2021, the functions are also available in the **Views** Ribbon tab at **Parts > Hide > ...** .

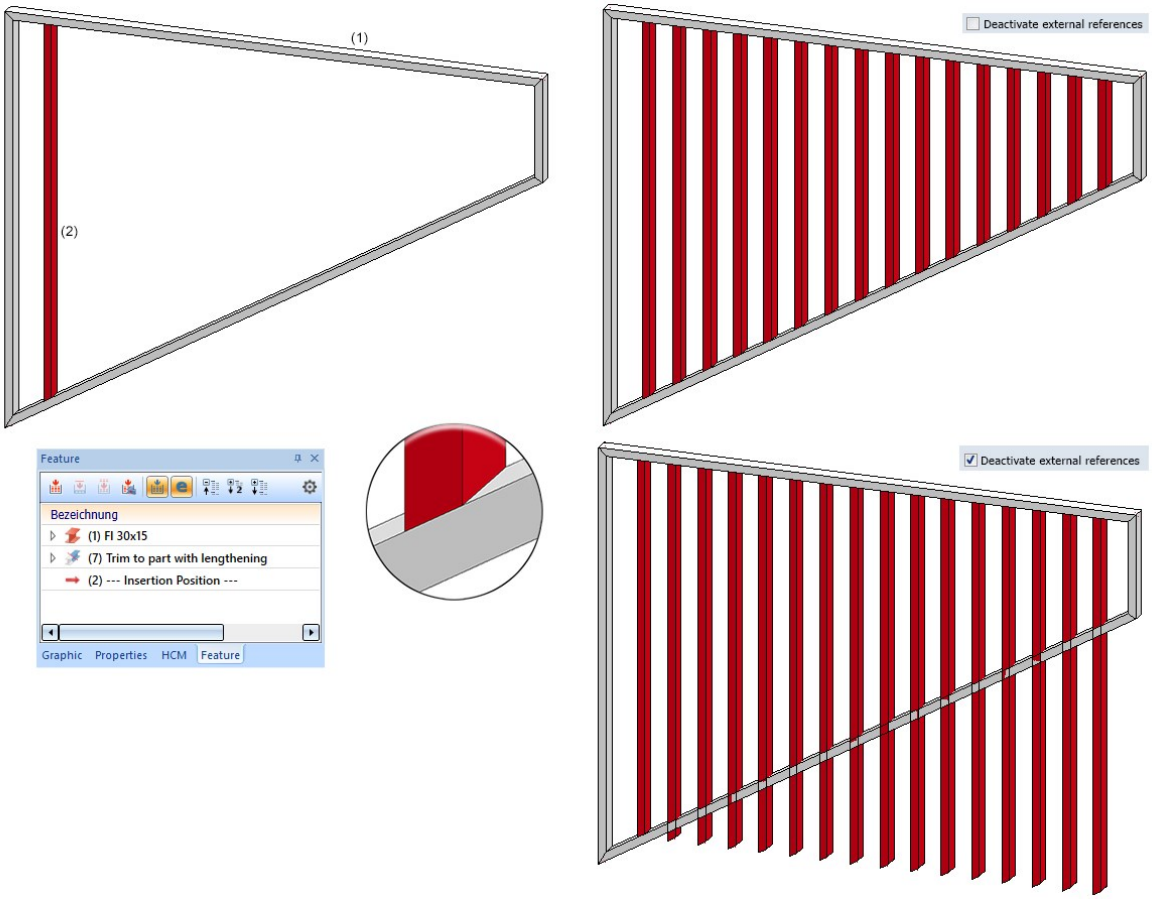
Process parts

Parametric cloning with external references

During designing, reference is often made to other drawing objects when inserting parts, for example when you want to fit a hole relative to the centre of a hole in another part, or cut a beam on another part. This reference between two parts is called an external reference. If the part to be cloned (copied) has feature parameters with an external reference, you can now use the checkbox **Deactivate external references** to specify whether the external references are to be taken into account during cloning or not. By default, the checkbox is inactive. This corresponds to the previous standard behaviour when cloning parts. If the checkbox is active, the external reference of the original is taken over for all cloned parts.

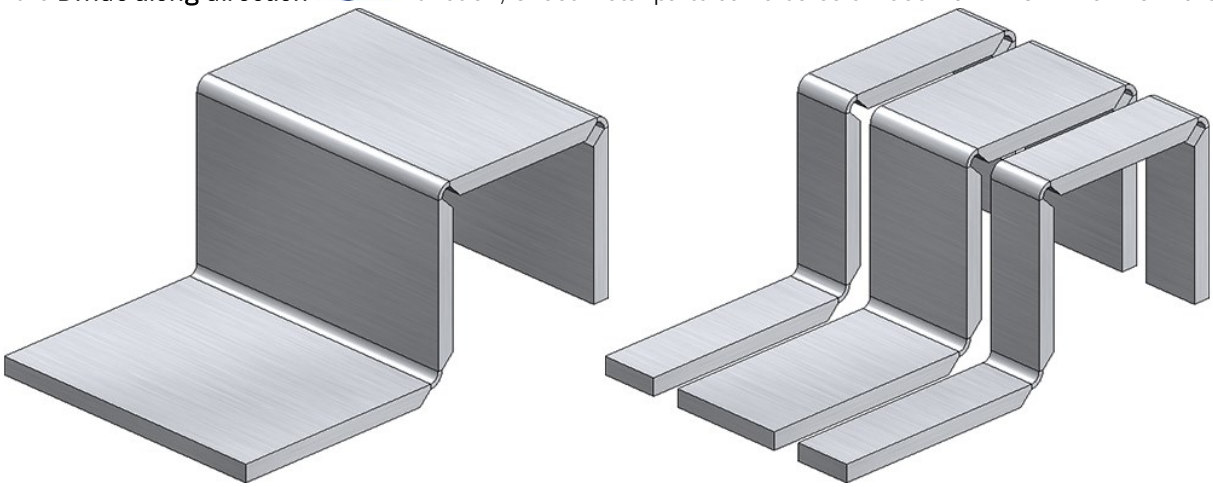


The image below shows a framework (1) with a trimmed filling rod (2) on the left. This filling rod is to be cloned (copied) by linear repetition in such a way that the framework is evenly filled with trimmed filling rods. Due to the cut, there is an external reference between the filling rod (2) and the lower framework beam. On the right you can see the result with and without the use of external references.

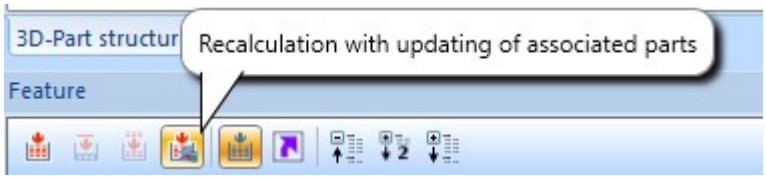


Divide along direction

With the **Divide along direction** function, Sheet Metal parts can also be divided from HiCAD 2021 onwards.



- The division can now also be executed by pressing the middle mouse button instead of the **OK** button.
- Feature editing has been improved:
  - The feature of the initial part has been renamed to **Divide along direction**.
  - For updating the parts after changes of the original part, the function **Recalculation with updating of associated parts** is now available directly in the toolbar of the feature window in the ICN.



  - If the feature Divide along direction of the original part is deactivated, the features from division of the other segments are automatically deactivated as well.

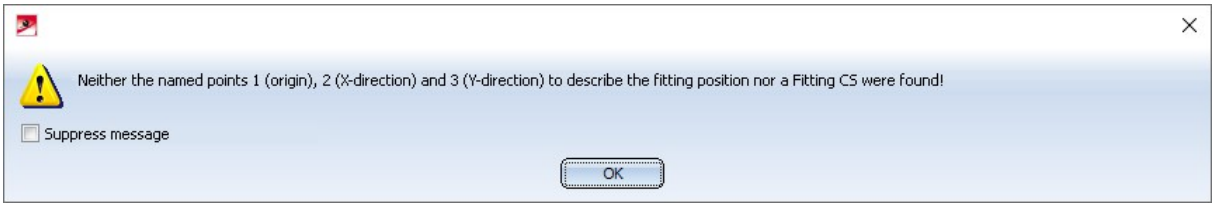
- For dividing beams and profiles, the function **Divide along direction** is available in the **Steel Engineering** Ribbon at **Lengthen > Divide**, which works essentially analogous to the 3-D function **Divide along direction** except for the following differences:

  - With the 3-D function, 3-D solids, Sheet Metal parts, beams and profiles can be divided, with the Steel Engineering function only beams and profiles.
  - The 3-D function always processes the active part. The steel engineering function, on the other hand, prompts you to select the beam to be processed.
  - The Steel Engineering function remains active after the division, i.e. the next beam to be divided can be selected. The beam selection is ended with the middle mouse button.

Standard Processings, Standard Parts and Boltings

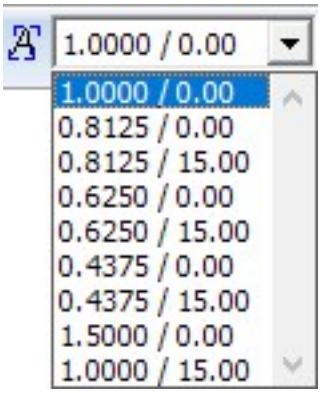
Inserting purchased/factory standard parts

Purchased/factory standard parts and moulding tools are also placed in the model drawing via the Fitting CS of the corresponding KRA file or via three isolated and named points defined in the KRA file (Point 1 for the origin, Point 2 for the X-direction, Point 3 for the Y-direction). If neither these points nor a Fitting CS are defined, the following message appears from HiCAD 2021 onwards:



Lettering with HiCAD fonts

When using the Lettering function, texts of HiCAD fonts were always displayed in italics. From HiCAD 2021 on, it is possible to specify whether these texts should be displayed in italics or not. This is determined by the **Aspect ratio** and the **Inclination angle**. When a HiCAD font is selected, a corresponding selection box is displayed in the upper toolbar of the dialogue window.

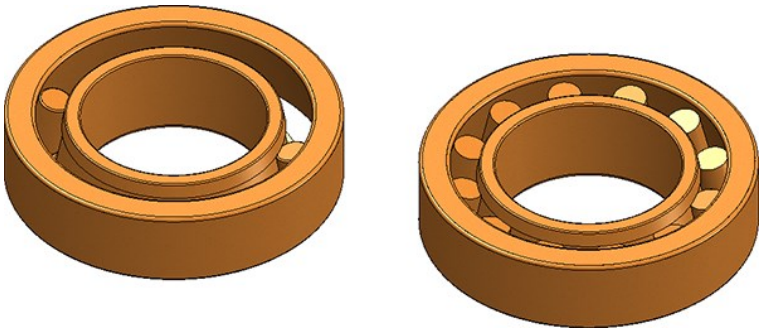


The box determines the typeface and the inclination of the text when using HiCAD fonts. If the aspect ratio is less than 1, the text is compressed. Similarly, with an aspect ratio greater than 1 the text is stretched. With an inclination of 0.00 degrees the text is displayed normally. With an inclination of 15.00 degrees the text is displayed in italics.

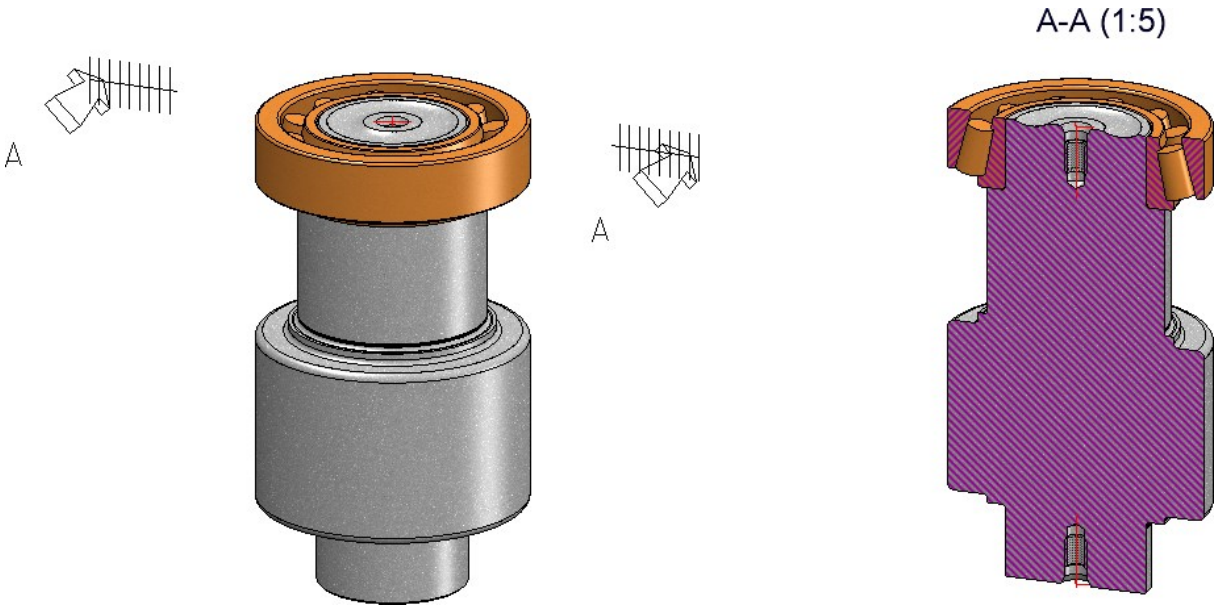
Geometric representation of bearings

The geometric representation of bearings has been significantly improved. Both in simple and exact representation, all ball / rolling elements are now shown. The increase in the number and position of the rolling elements also results in technically useful sectional views with improved hatching, which is opposite for the inner and outer ring.

The following figure shows the representation of a tapered roller bearing (DIN 722).



Left: Before HiCAD 2021, Right: From HiCAD 2021 onwards



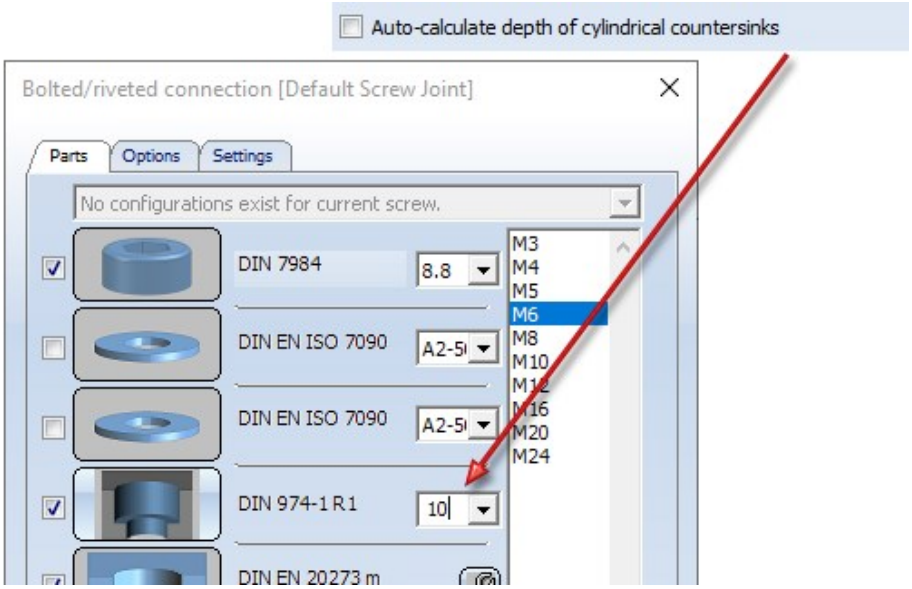
Sectional view with tapered roller bearing

Another improvement is that parametric constraints are retained when the size of the bearing is changed and do not have to be reassigned.

Boltings - countersink depth for cylindrical countersinks

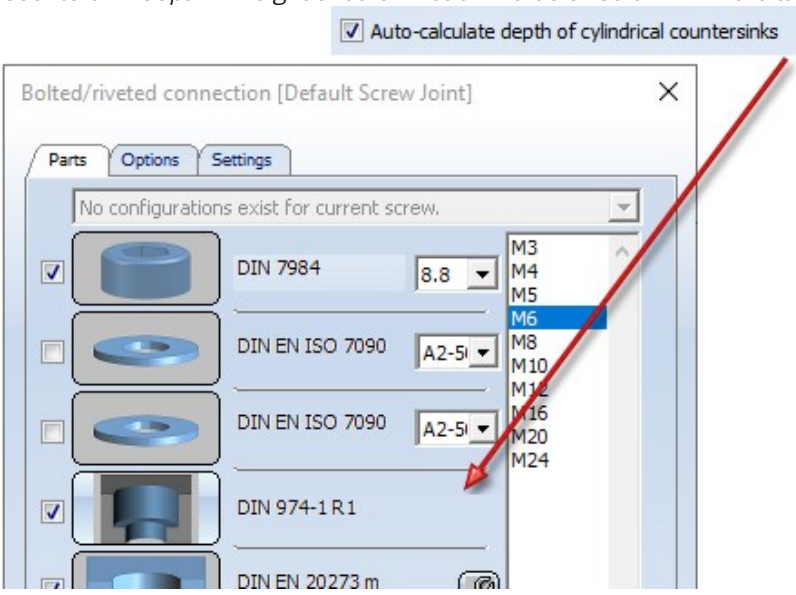
When inserting bolted connections, the countersink depth of cylindrical countersinks can now either be entered manually or determined automatically by HiCAD. For this purpose, a corresponding checkbox has been added to the **Options** tab.

- If the checkbox is inactive (default), the countersink depth is determined by the value entered on the **Parts** tab.

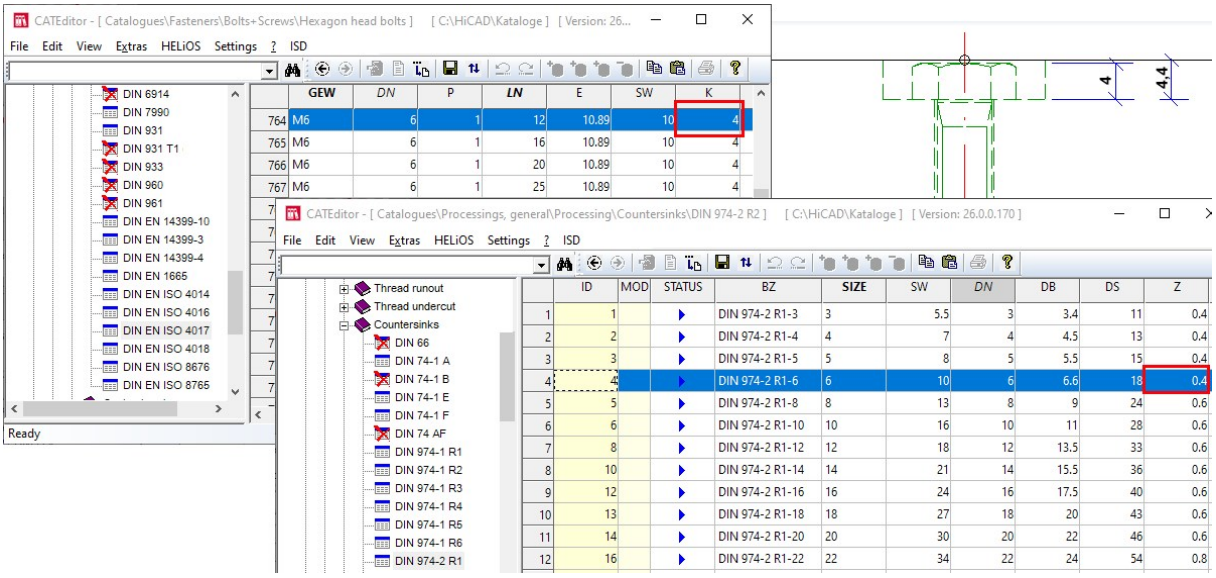


- If the checkbox is active, then no input field for the countersink depth will be displayed on the **Parts** tab, but the countersink depth will be automatically calculated by HiCAD as follows:

  Countersink depth = Height of screw head + Value of Column Z in the table of the countersink

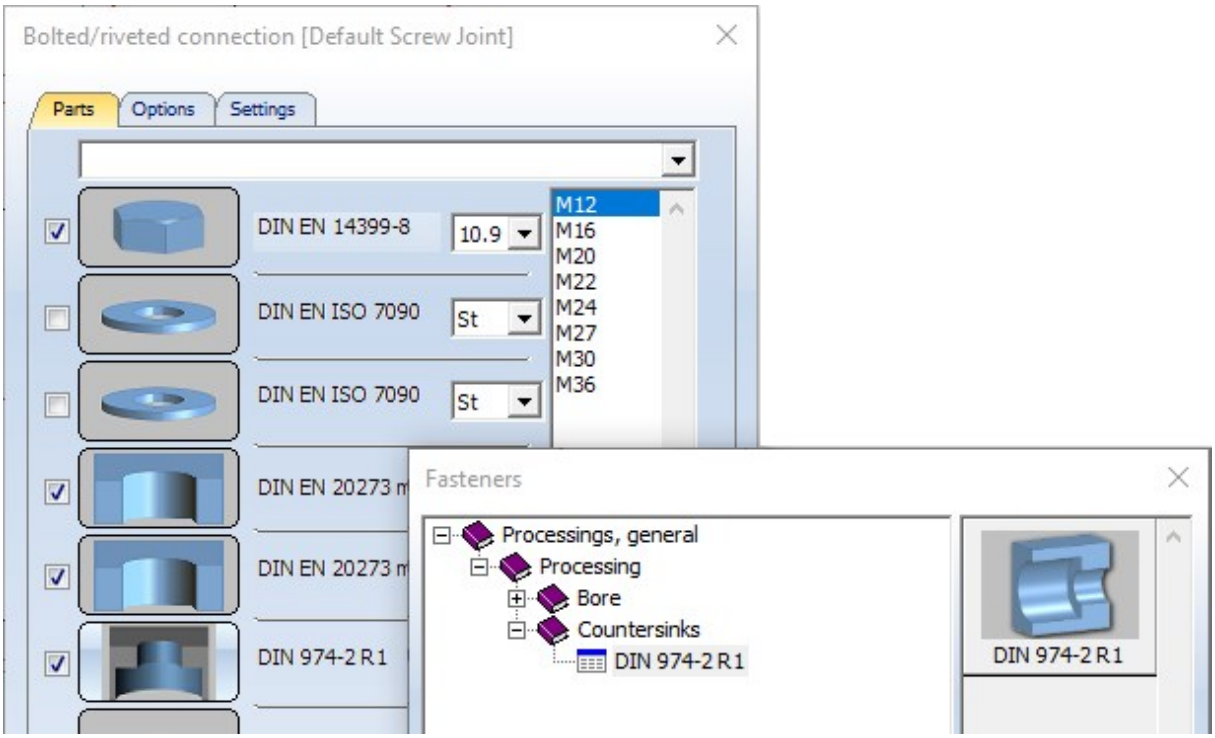


  For example, if - as shown above - a screw with nominal diameter 6 is used, then the height of the screw head will be 4 and the value Z in the table of countersink DIN 974-2 R1 will be 0.4. The countersink depth will thus be 4.4.



Standardised countersinks

When installing bolted connections, the corresponding standard countersink is now available for bolts according to DIN/ISO.



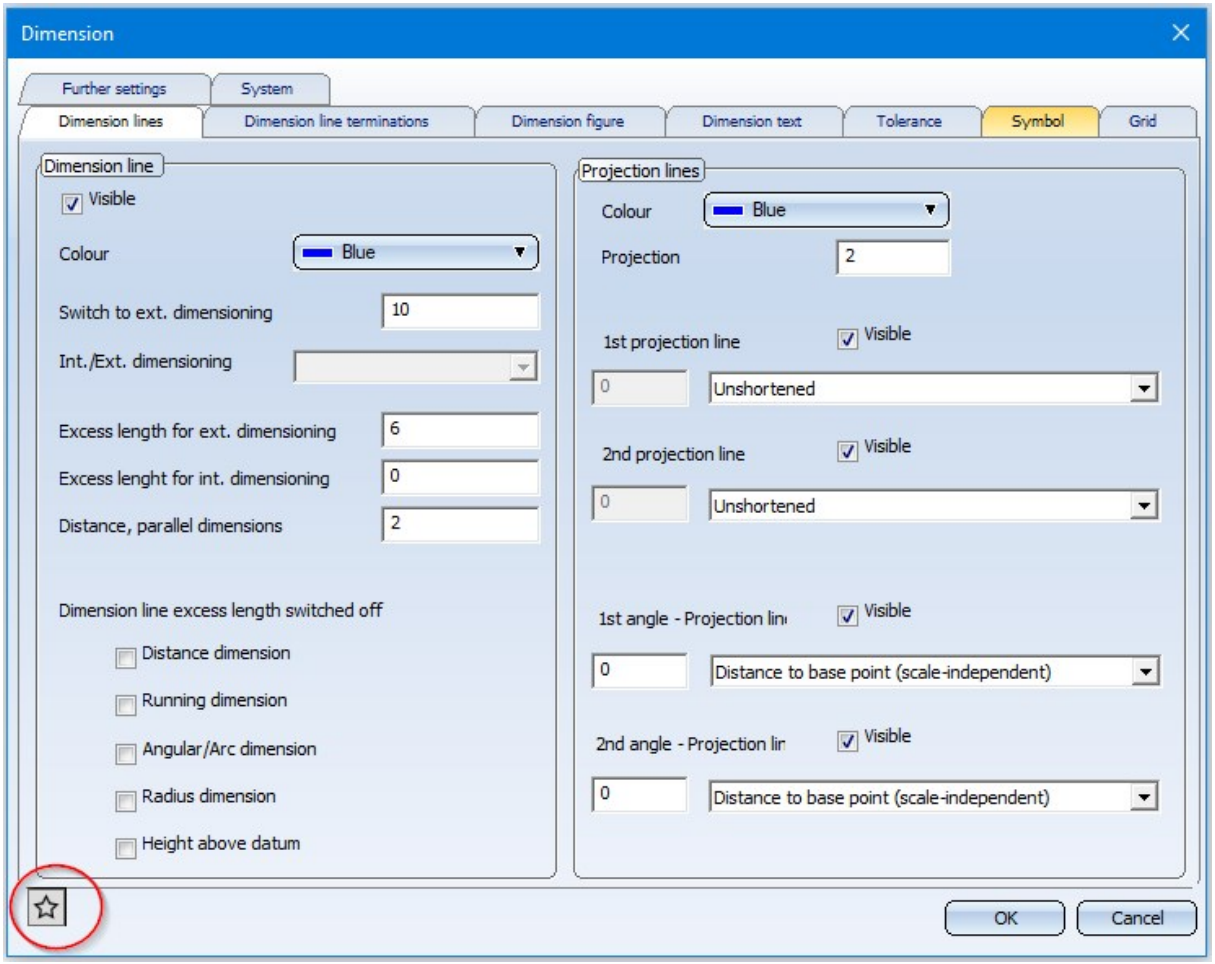
Dimensioning

Save dimension parameters as Favourites

The parameter settings for the functions

- Set parameters for new general dimensions
- Set parameters for new parametric dimensions , and
- Set parameters for new constraint dimensions

can now be saved as favourites for a later reuse. To do this, click on the symbol in the dialogue window. Read more about managing of favourites in the HiCAD Basics Help at Manage Favourites.



Favourites are stored in the **Favourites** sub-folder of the folder in which the HiCAD configuration database is located. If you have installed HiCAD from the red DVD with the ISD default settings, this is the folder **ProgramData\ISD Software und Systeme\HiCAD** 2021. For each functional area in which you have saved favourites, a corresponding sub-folder is created in the FAVOURITES folder, For 3-D dimensioning this is the DIMENSIONING\SETTINGS folder. The favourites of the various parameter settings are saved there in further sub-folders:

Default	Favourites for general dimensions	
Parameters	Favourites for parametric dimensions	
HCM	Favourites for constraint dimensions	

More information on favourites management can be found in the **Manage Favourites** topic in the HiCAD Basics Help.

Change dimension parameters, Individual

The function **Change dimension parameters, Individual** has been changed. After calling the function, several dimensions to be changed can now be selected. With the right mouse button the selection of the dimensions is ended and the dialogue window **Dimension parameters** is displayed.

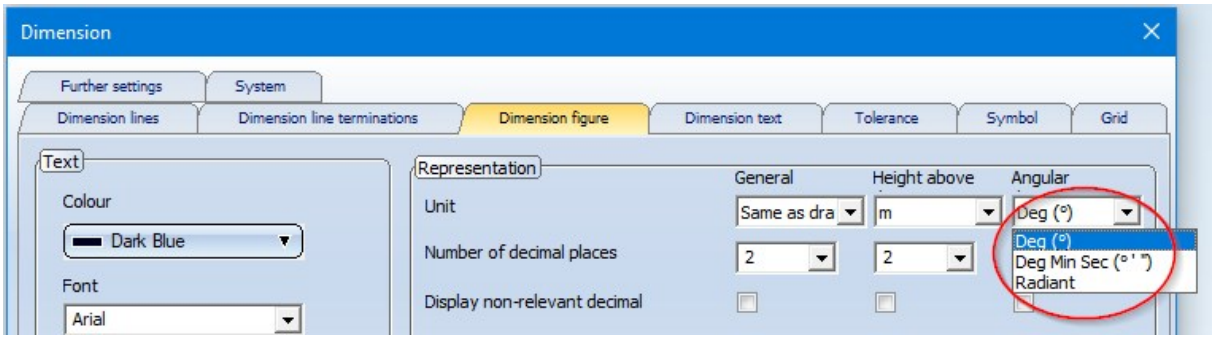
The function with the same name in the context menu for dimensions remains unchanged, i.e. only the individual dimension is changed here.

Units of measurement for angular dimensions

In the parameter settings of the functions

- Set parameters for new general dimensions
- Set parameters for new parametric dimensions , and
- Set parameters for new constraint dimensions

the unit **Centesimal degree** (bzw. GON) for angular dimensions is no longer available.

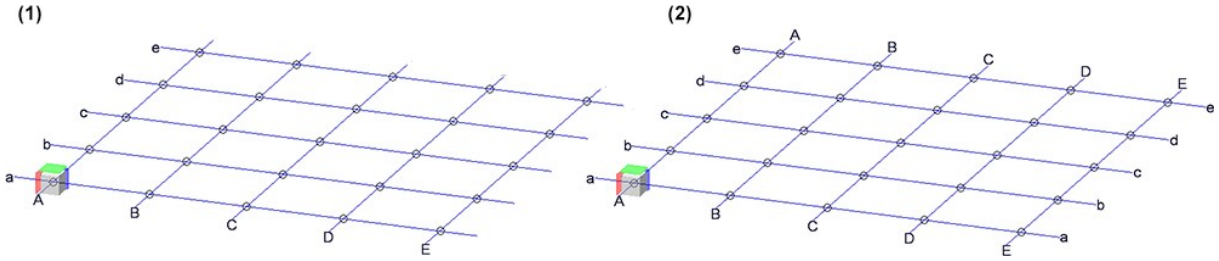


This also applies to the settings in the Configuration Editor at

- Drawing > Annotations > Dimensioning, 3-D > interactive dimensions,
- Drawing > Annotations > Dimensioning, 3-D > Parametric dimensions, and
- Drawing > Annotations > Dimensioning, 3-D > HCM dimensions.

Grid annotations

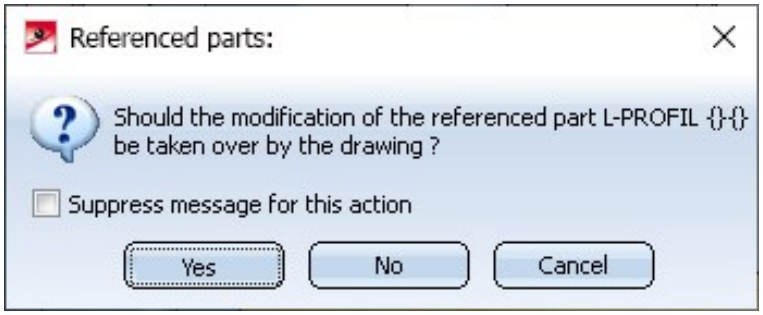
The 3-D grid function now inserts grid annotations on both sides of the grid.



(1) up to HiCAD 2020, (2) as of HiCAD 2021

Copying internally referenced parts

To reuse internally referenced parts, you can simply copy the part. To do this, you can also use the **Copy to HiCAD Clipboard** and **Paste from HiCAD Clipboard** functions in the ICN toolbar. Please note the following: If an internally referenced part or an assembly with internally referenced parts is inserted via the HiCAD clipboard from a construction into another construction in which internally referenced identical parts to the internally referenced parts of the clipboard already exist, then HiCAD asks whether these parts are to be adapted to the version in the drawing, e.g.



The checkbox **Suppress message for this action** is new here. Via this checkbox it can now be determined whether the update must be confirmed explicitly for each of the parts or not. If the checkbox is inactive (default setting), the query was made for each corresponding part - as before HiCAD 2021. For assemblies with many internally referenced parts, this can be very annoying and time-consuming.

If the checkbox is active, then a subsequent click on **Yes** causes the update to be carried out for all corresponding parts without further query. If **No** is clicked, the corresponding parts from the clipboard remain unchanged.

Only if the parts in the drawing actually deviate from the corresponding parts of the clipboard will the query be displayed again.

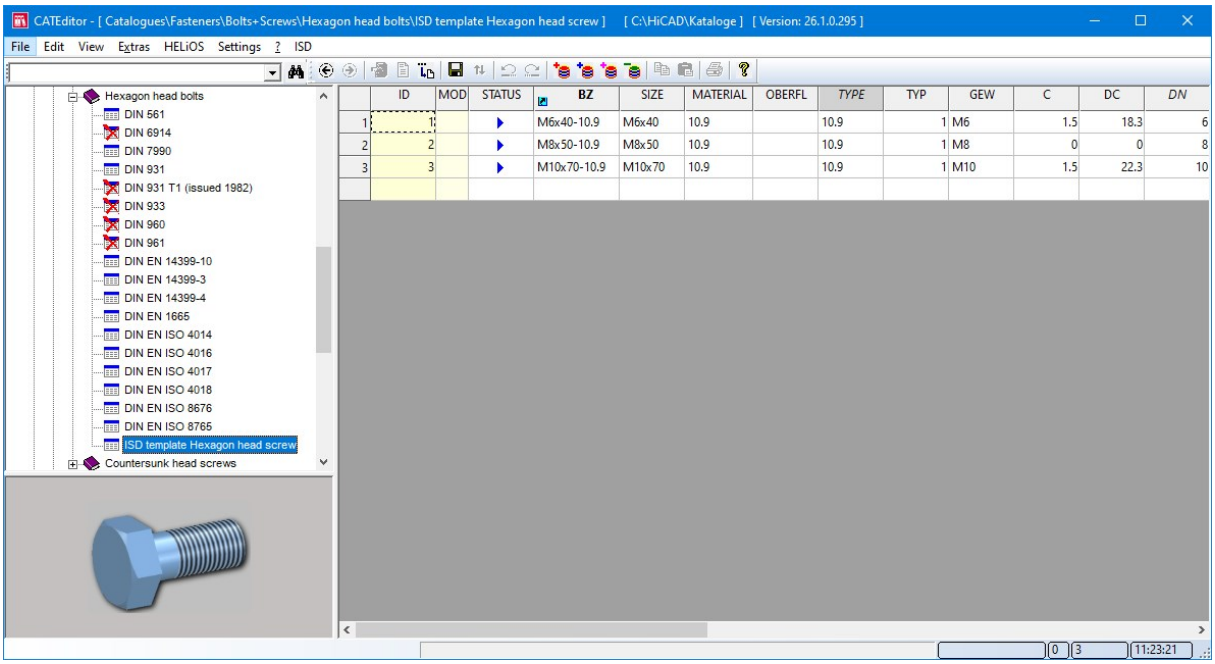
Catalogue Editor

Service Pack 1 2021 (V 2601)

ISD template for hexagon head screws

The table **ISD template hexagon head screw** is available in the catalogue **Fasteners > Bolts+Screws > Hexagon head bolts**.

This table contains a predefined and parametrically designed 3-D part that you can use as a basis for new hexagonal bolts.



In order to be able to use the ISD template as a basis for your own standards tables, you must first create a copy of the template. In the process, a copy of the KRA file is also created automatically. You can then extend the copy of the table with further data records. Or you can use the KRA file to define individual hexagonal bolts.

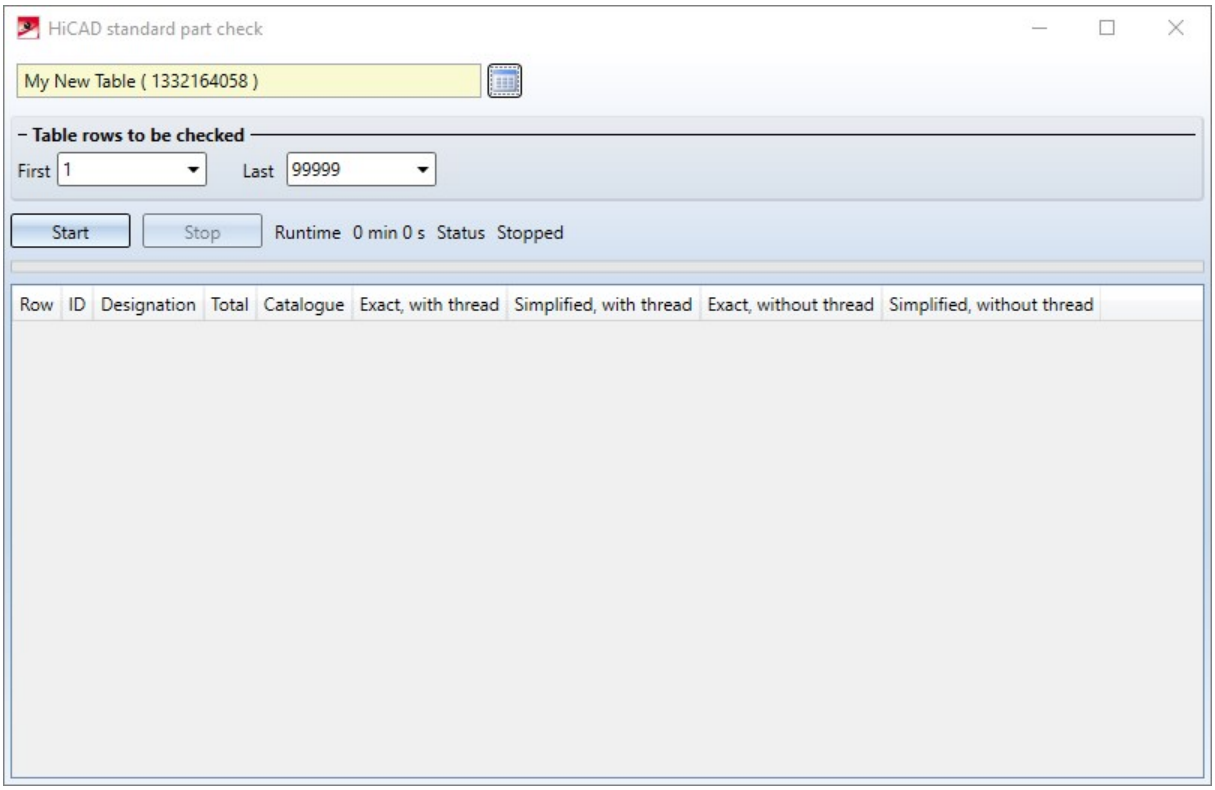
Standard part check

HiCAD offers you the possibility to test newly created or extended standard part tables. A corresponding macro is available for this purpose.

To start this macro:

1. Press **CTRL+8.**
2. Change to the HiCAD sub-directory **MAKRO3D**.
3. Double-click on the macro name **StandardPartTest.mac**.

The **Standard part check** dialogue window is displayed.



At the top of the dialogue window click on the symbol and select the table to be tested in the catalogue structure.

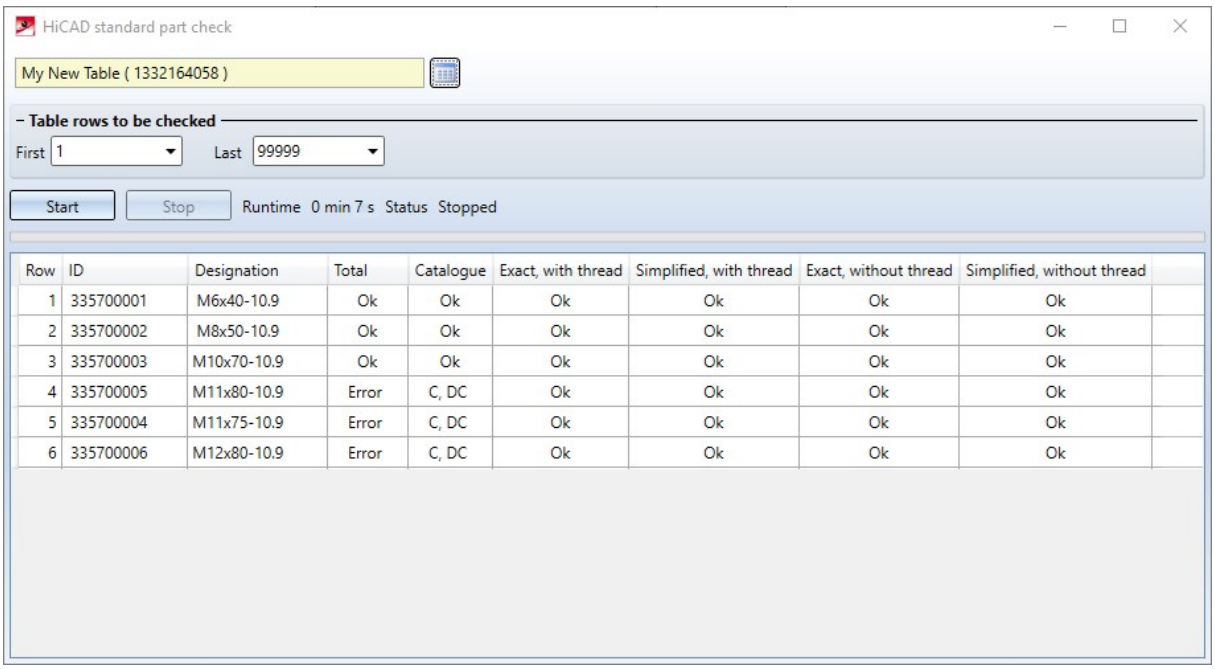
In the **Table rows to be checked** area, determine the range to be tested by entering the first and last row number of the desired range. If you enter 1 and 999999, the whole table will be checked.

Click on **Start** to start the standard part check. A new drawing is created in HiCAD with the name of the table ID, e.g. 1330340533. The checked standard parts are - if possible - automatically inserted into this drawing.

The results of the check are listed in the dialogue window:

- The column **Total** shows whether a problem was detected for a table row.
- The **Catalogue** shows whether there are problems with the consistency of the values in the table.

- The remaining columns refer to the insertion in the four different display options. A collision check is also performed for each part created.



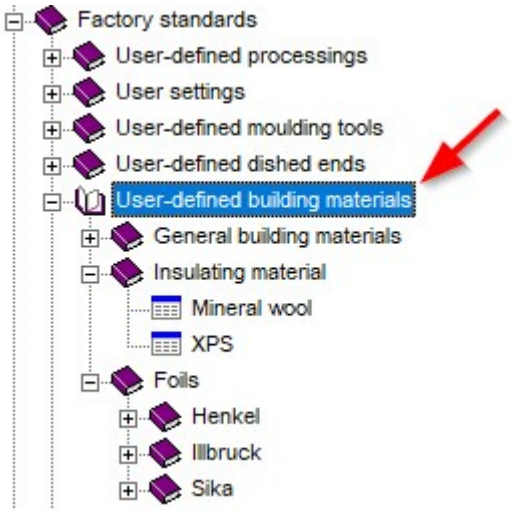
Row	ID	Designation	Total	Catalogue	Exact, with thread	Simplified, with thread	Exact, without thread	Simplified, without thread
1	335700001	M6x40-10.9	Ok	Ok	Ok	Ok	Ok	Ok
2	335700002	M8x50-10.9	Ok	Ok	Ok	Ok	Ok	Ok
3	335700003	M10x70-10.9	Ok	Ok	Ok	Ok	Ok	Ok
4	335700005	M11x80-10.9	Error	C, DC	Ok	Ok	Ok	Ok
5	335700004	M11x75-10.9	Error	C, DC	Ok	Ok	Ok	Ok
6	335700006	M12x80-10.9	Error	C, DC	Ok	Ok	Ok	Ok

After the end of a check or in case of an abort with **Stop**, the created parts will be sorted geometrically - by type of representation - in the drawing, so that they can also be checked again manually. To do this, you must close the **Standard part check** dialogue in order to be able to work in the drawing.

Changed catalogue structure

New under **Factory standards** is the catalogue **User-defined building materials**. This catalogue contains the ISD catalogues **General building materials**, **Foils** and **Insulating material**, which were previously available under **Factory standards > User-defined semi-finished products > Civil Engineering**.

The catalogue **Factory standards > User-defined semi-finished products** is no longer available for a new installation.



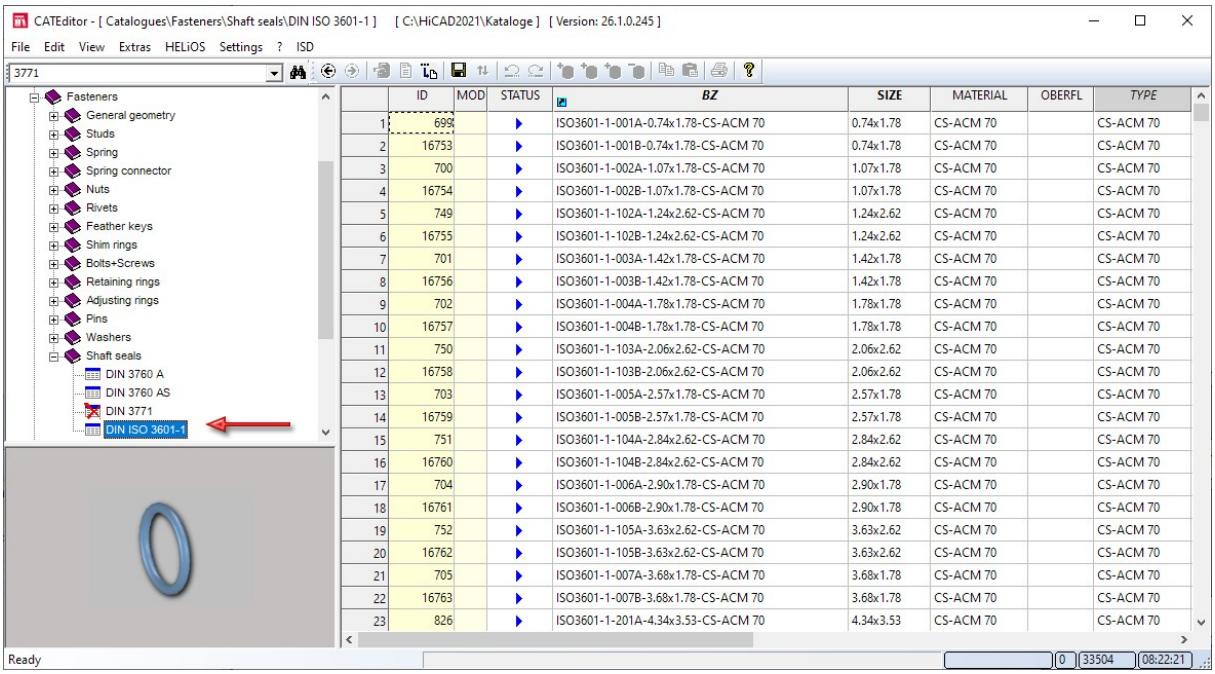
For an update installation, proceed as follows:

- The known ISD sub-catalogues from **Factory standards > User-defined semi-finished products > Civil Engineering** are moved to the catalogue **Factory standards > User-defined building materials**.
- If the catalogue **Factory standards > User-defined semi-finished products** is empty afterwards, it will be deleted.
- If the catalogue **Factory standards > User-defined semi-finished products** contains customer-specific tables, they are renamed USER_HALFZEUGE_TO_DELETE..... The catalogue User-defined semi-finished products remains. In this case, you should move your file to other catalogues and then delete the User semi-finished products folder.

Shaft seals DIN ISO 3601-1

The standard DIN 3771 for shaft seals has been withdrawn and replaced by DIN ISO 3601-1. This is taken into account in the HiCAD catalogues:

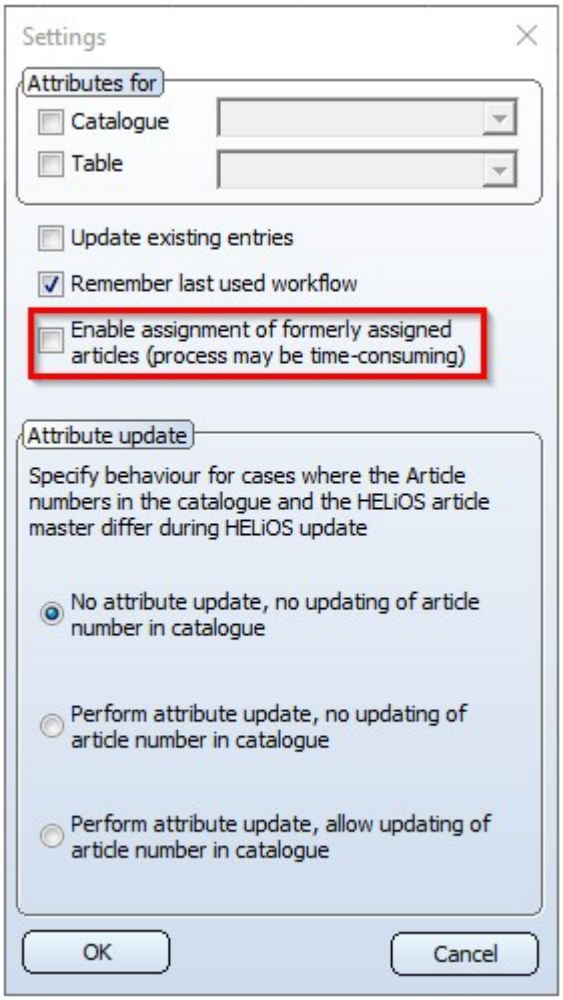
- The table DIN_3771 has been marked as obsolete.
- A new table for shaft seals according to DIN ISO 3601-1 has been added. You can find this under **Fasteners > Shaft seals**.



	ID	MOD	STATUS	BZ	SIZE	MATERIAL	OBERFL	TYPE
1	699			ISO3601-1-001A-0.74x1.78-CS-ACM 70	0.74x1.78	CS-ACM 70		CS-ACM 70
2	16753			ISO3601-1-001B-0.74x1.78-CS-ACM 70	0.74x1.78	CS-ACM 70		CS-ACM 70
3	700			ISO3601-1-002A-1.07x1.78-CS-ACM 70	1.07x1.78	CS-ACM 70		CS-ACM 70
4	16754			ISO3601-1-002B-1.07x1.78-CS-ACM 70	1.07x1.78	CS-ACM 70		CS-ACM 70
5	749			ISO3601-1-102A-1.24x2.62-CS-ACM 70	1.24x2.62	CS-ACM 70		CS-ACM 70
6	16755			ISO3601-1-102B-1.24x2.62-CS-ACM 70	1.24x2.62	CS-ACM 70		CS-ACM 70
7	701			ISO3601-1-003A-1.42x1.78-CS-ACM 70	1.42x1.78	CS-ACM 70		CS-ACM 70
8	16756			ISO3601-1-003B-1.42x1.78-CS-ACM 70	1.42x1.78	CS-ACM 70		CS-ACM 70
9	702			ISO3601-1-004A-1.78x1.78-CS-ACM 70	1.78x1.78	CS-ACM 70		CS-ACM 70
10	16757			ISO3601-1-004B-1.78x1.78-CS-ACM 70	1.78x1.78	CS-ACM 70		CS-ACM 70
11	750			ISO3601-1-103A-2.06x2.62-CS-ACM 70	2.06x2.62	CS-ACM 70		CS-ACM 70
12	16758			ISO3601-1-103B-2.06x2.62-CS-ACM 70	2.06x2.62	CS-ACM 70		CS-ACM 70
13	703			ISO3601-1-005A-2.57x1.78-CS-ACM 70	2.57x1.78	CS-ACM 70		CS-ACM 70
14	16759			ISO3601-1-005B-2.57x1.78-CS-ACM 70	2.57x1.78	CS-ACM 70		CS-ACM 70
15	751			ISO3601-1-104A-2.84x2.62-CS-ACM 70	2.84x2.62	CS-ACM 70		CS-ACM 70
16	16760			ISO3601-1-104B-2.84x2.62-CS-ACM 70	2.84x2.62	CS-ACM 70		CS-ACM 70
17	704			ISO3601-1-006A-2.90x1.78-CS-ACM 70	2.90x1.78	CS-ACM 70		CS-ACM 70
18	16761			ISO3601-1-006B-2.90x1.78-CS-ACM 70	2.90x1.78	CS-ACM 70		CS-ACM 70
19	752			ISO3601-1-105A-3.63x2.62-CS-ACM 70	3.63x2.62	CS-ACM 70		CS-ACM 70
20	16762			ISO3601-1-105B-3.63x2.62-CS-ACM 70	3.63x2.62	CS-ACM 70		CS-ACM 70
21	705			ISO3601-1-007A-3.68x1.78-CS-ACM 70	3.68x1.78	CS-ACM 70		CS-ACM 70
22	16763			ISO3601-1-007B-3.68x1.78-CS-ACM 70	3.68x1.78	CS-ACM 70		CS-ACM 70
23	826			ISO3601-1-201A-4.34x3.53-CS-ACM 70	4.34x3.53	CS-ACM 70		CS-ACM 70

Enable assignment of formerly assigned articles

If you have cancelled the assignments between catalogue entries and HELiOS article masters and would like to restore them when transferring them again, you only have to activate the checkbox **Enable assignment of formerly assigned articles** in the **Settings**.



Please note:

- Please note that activating the checkbox can be time-consuming depending on the number of the corresponding catalogue entries.
- If the checkbox is not activated (ISD default setting), assignments that have been deleted will not be restored when they are transferred again.

Orientation of tray panels

Tray panels for **Profile Installation** were previously aligned according to the manufacturer's drawings. In some cases this led to these tray panels being installed the wrong way round. This has been corrected in SP1 so that the following tray panels are now stored in the catalogue correctly aligned:

- Factory standards\Series\Roof Wall Facade\Room-closing profiles\Fischer\Liner trays
- Factory standards\Series\Roof Wall Facade\Room-closing profiles\Montana\Liner trays
- Factory standards\Series\Roof Wall Facade\Room-closing profiles\Münker\Liner trays
- Factory standards\Series\Roof Wall Facade\Room-closing profiles\Profiltec Bausysteme\Liner trays
- Factory standards\Series\Roof Wall Facade\Room-closing profiles\SAB\Liner trays \Liner trays IJsselstein
- Factory standards\Series\Roof Wall Facade\Room-closing profiles\SAB\Liner trays \Liner trays Niederaula
- Factory standards\Series\Roof Wall Facade\Room-closing profiles\Salzgitter Bauelemente\Liner trays

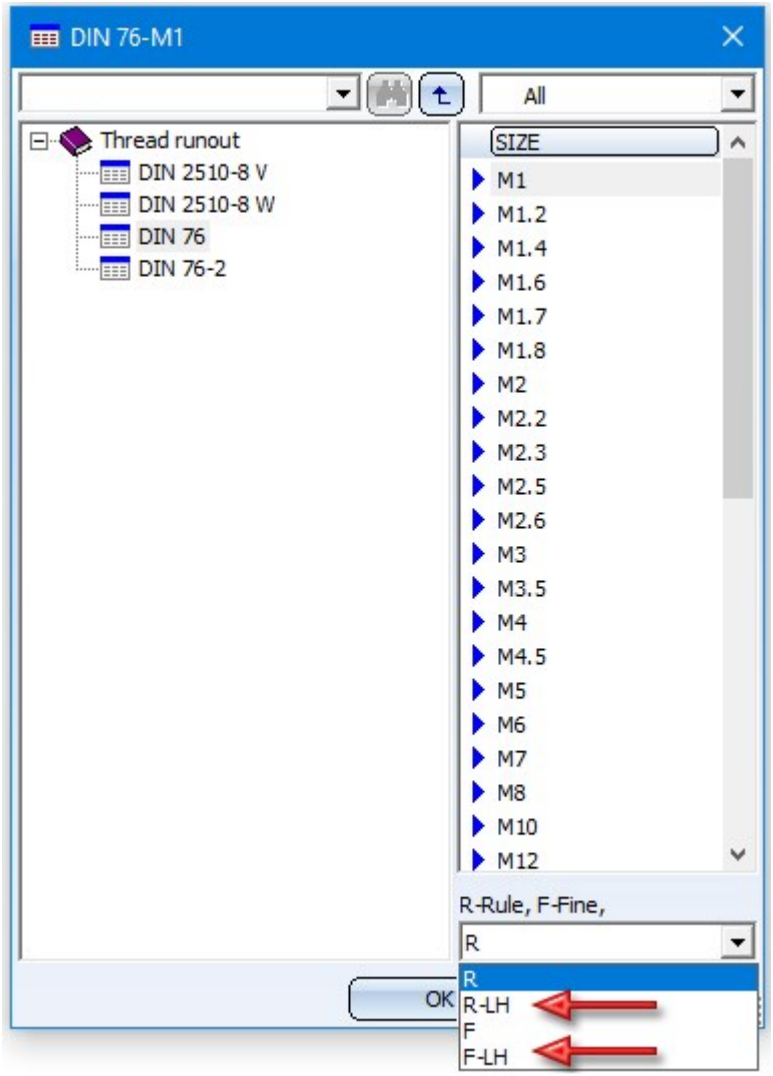
These changes only affect newly created profile installations. Existing profile installations remain in their original state even after recalculation.

If you want to change from the old to the new tray panels in an "old" profile installation, edit the profile installation and select the desired tray panel from the catalogue.

Major Release 2021 (V 2600)

Thread runout DIN 76, LH thread

In the table **DIN 76** at **Processings, general > Processing > Thread runout** you can now also find left-handed threads (Type F-LH and R-LH).



Welded studs DIN EN ISO 13918 FD and MD

New in the catalogue **Fasteners > Studs > > Welded studs** are the tables:

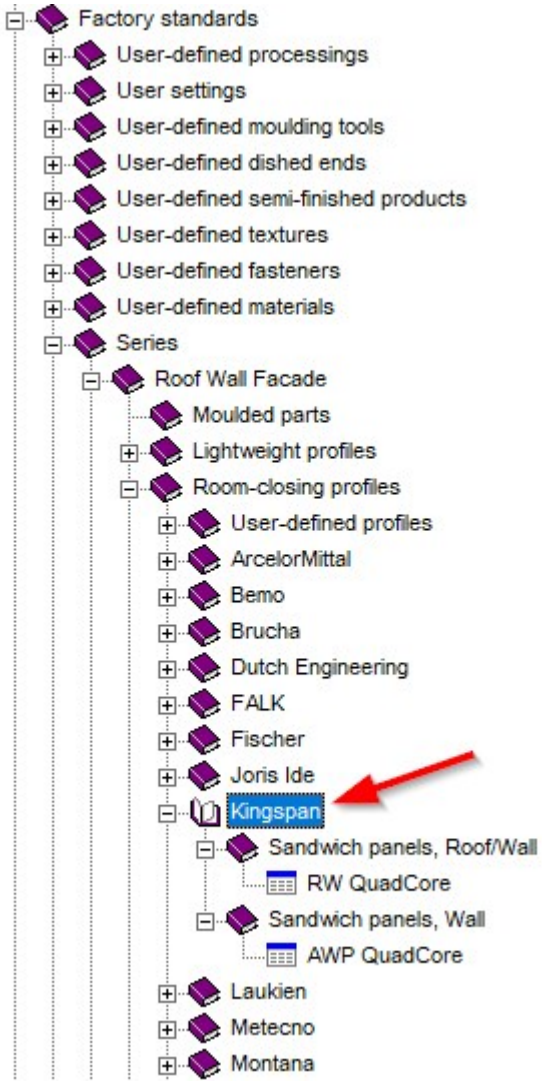
- DIN EN ISO 13918 FD and
- DIN EN ISO 13918 MD.

Kingspan Sandwich elements for Roof + Wall

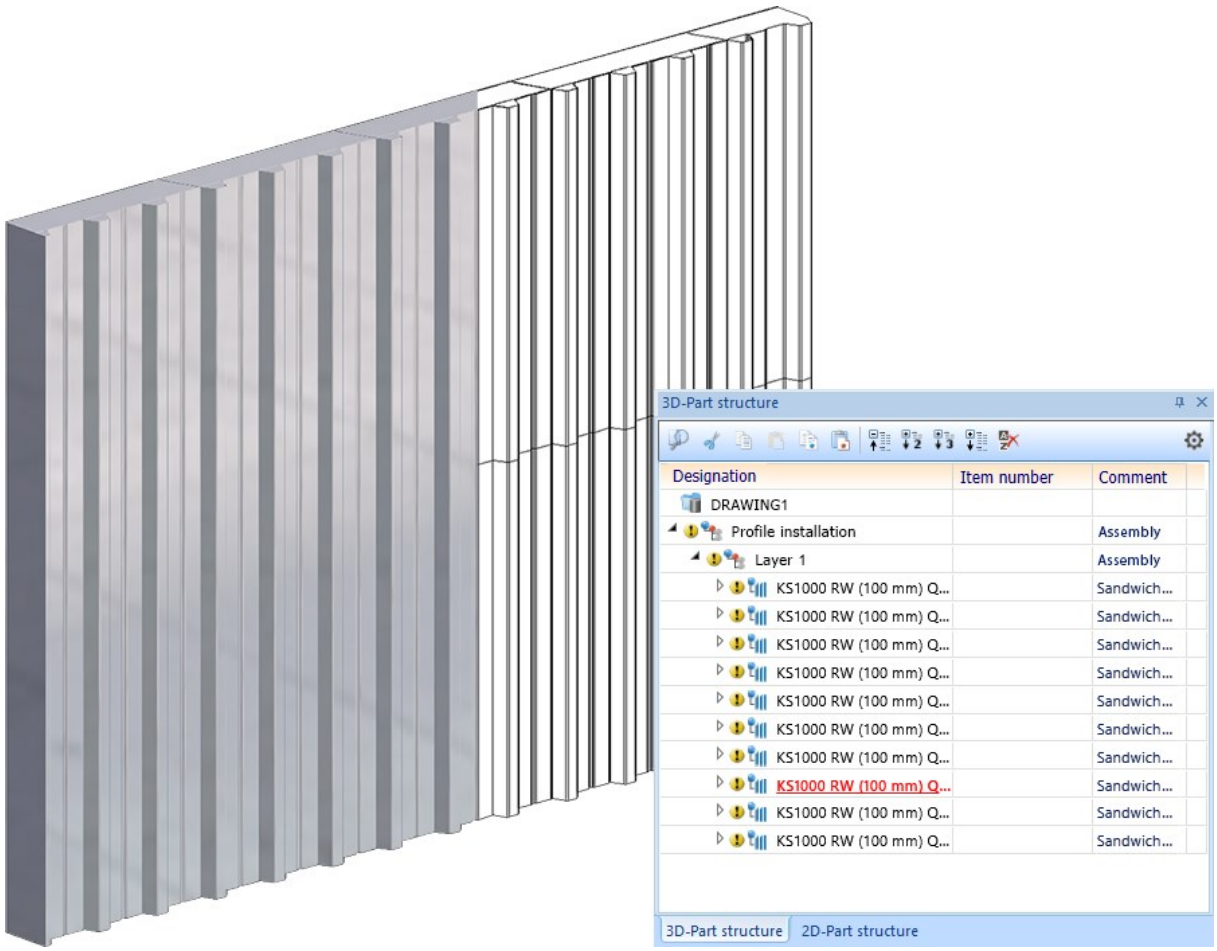
The catalogue **Factory standards\Series\Roof Wall Facade\Room-closing profiles** has been extended with sandwich panels for roof and wall of the company Kingspan:

- **Kingspan\Sandwich panels, Roof/Wall\RW QuadCore** and
- **Kingspan\Sandwich panels, Wall\AWP QuadCore**

These can be used, for example, for profile installation or the insertion of series in Steel Engineering.



In this context, the **Duromers** table in the **Materials\Plastics** catalogue has been expanded to include the rigid foam **QuadCore®** from Kingspan.



Example of a profile installation with Kingspan sandwich panels RW QuadCore

New milling groove forms

In the catalogue **Factory standards > Composite panels, groove form** the table **Standard groove form** has been expanded. This table is used, among other things, for attaching and folding flanges with milling edge zones:

V 90°	for angles up to 90°
V135°	for angles greater than 90° and less than or equal to 135 °
Rectangle 10	for angles greater than 135 ° (as of HiCAD 2021: > 160°)

New as of HiCAD 2021 are:

- Groove forms V 160 ° for angles up to 160°
- Groove forms V 90°, V 135°, V160° as milling cutter for acute folds

	ID	MOD	STATUS	BZ	W	B	D	BW_MIN	BW_MAX	SORT	ICON
1	4		▶	V 90°	90	3	0.8	0	90	1	V90.bmp
2	6		▶	V 135°	135	2	0.8	0	135	2	V135.bmp
3	7		▶	V 160°	160	2	0.8	0	160	3	V160.bmp
4	8		▶	V 90° acute	90	0.05	0.01	0	90	6	V90 Spitz.bmp
5	9		▶	V 135° acute	135	0.05	0.01	0	135	7	V135 Spitz.bmp
6	10		▶	V 160° acute	160	0.05	0.01	0	160	8	V160 Spitz.bmp
7	1		▶	Rectangle 10	0	10	1.5	135	180	10	Rect10.bmp
8	3		▶	Rectangle 15	0	15	1.5	135	180	11	Rect15.bmp
9	2		▶	Rectangle 14	0	14	1.5	135	180	12	Rect14.bmp
10	5		✘	V 135°	135	3	0.8	0	135	100	V135.bmp

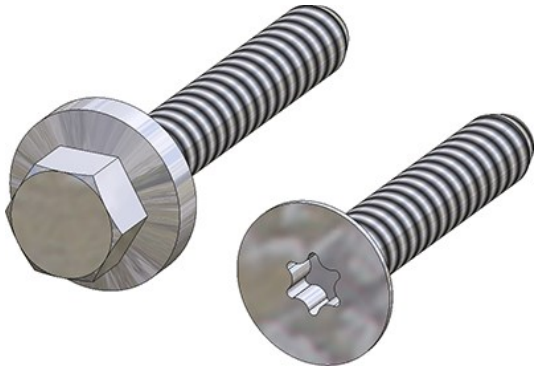
The milling cutters for acute angled folds are designed for 2-D drafts. There, tray panels are conceived as simplified "paper models". This is helpful if you design using a paper model and want to use the groove forms for acute angled folds. In HiCAD, the cutters with priority 1-3 are used by default. So to use the cutters for acute angled folds you have to change their priority.

For more information, see the Attach Flange and Bend Zone topic in the Sheet Metal Online Help.

Ejot® sealing screws JZ3 and JZ3-S

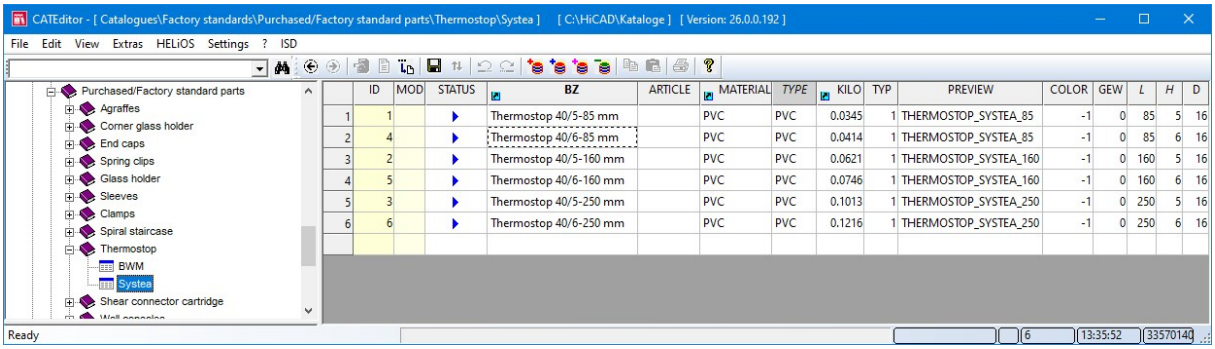
The catalogue **Factory standards > User-defined fasteners > User-defined bolts+screws > Ejot** contains the following new screws:

- EJOT® sealing screw JZ3 (thread-cutting, with hex head and pre-mounted sealing washer)
- EJOT® sealing screw JZ3-S (thread-cutting, ,with countersunk head).



SYSTEA® Thermostop elements

The catalogue **Factory standards > Purchased/Factory standard parts > Thermostop** now contains new elements by Systea®.



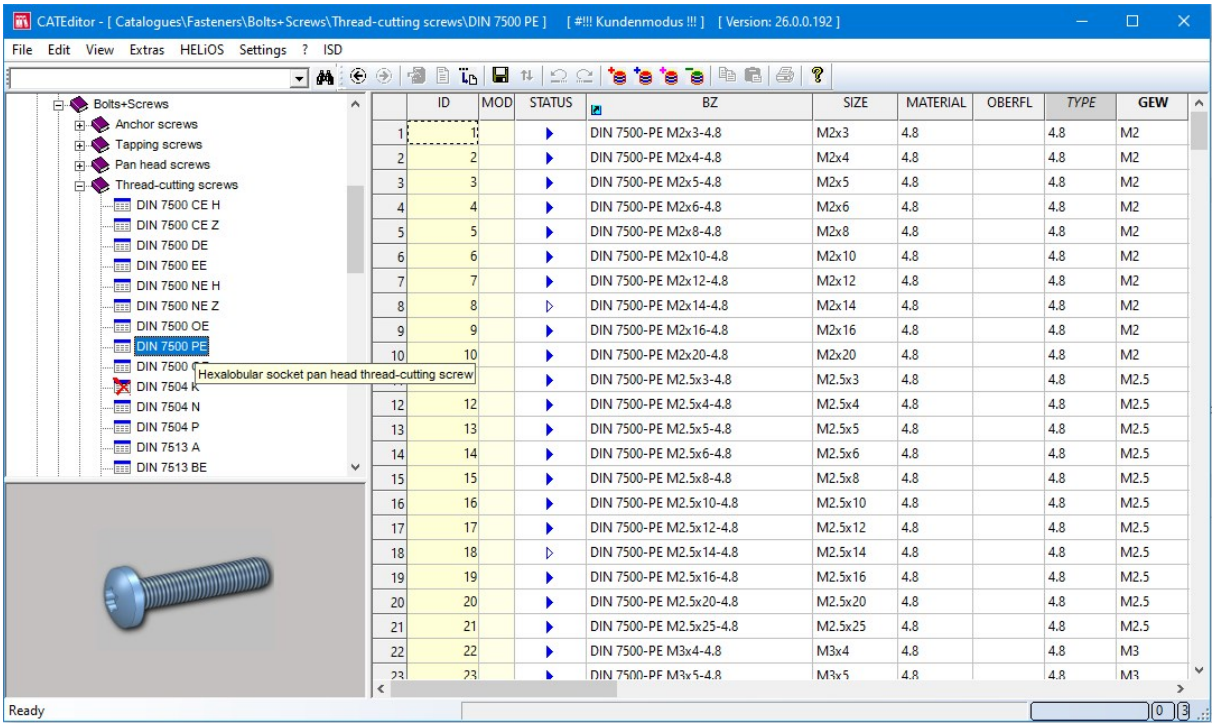
Preview images for data records in tables

If a table is opened in the Catalogue Editor, a preview image is displayed at the bottom left of the dialogue. If a data record is activated in the table, the image belonging to this record and specified in the ICON column is displayed as a preview image. From HiCAD 2021 on, the following procedure is used here:

- If the ICON column is empty or not available, the image of the corresponding table is still displayed as a preview image.
- If there is an entry in the ICON column, this must be a valid file name. In this case, the preview image defined by the specified file is displayed for the table row. If the file name is incorrect, no preview image is displayed for this table row.

Improved tool tips for bolts and screws

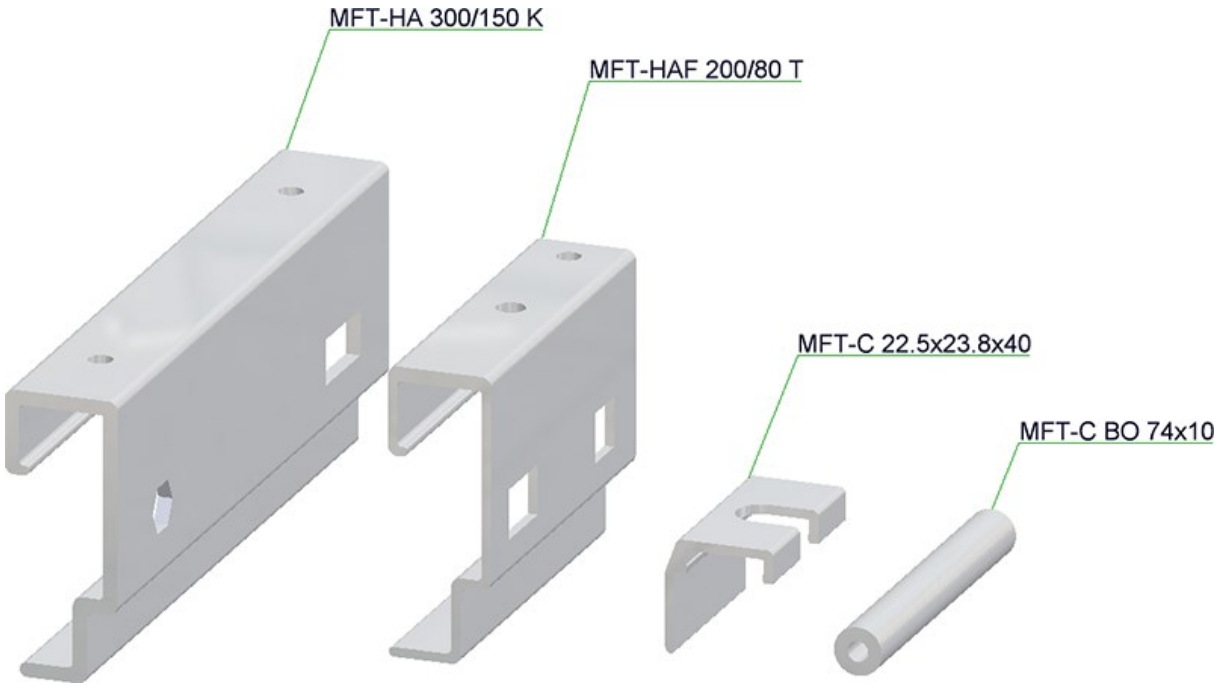
The tooltips that are displayed when moving the cursor over a table in the Catalogue Editor structure have been improved and made more meaningful for bolts and screws.



More Hilti standard parts for sub-structures

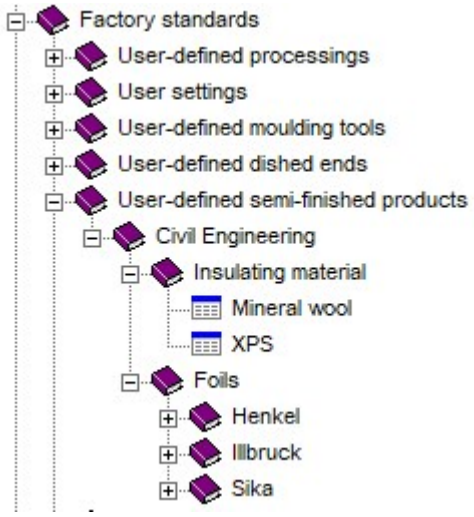
The catalogue **Factory standards > Purchased/Factory standard parts** contains new Hilt parts for sub-structures:

Catalogue	Table	
Agraffes > Hilti > MFT-H 300	■ MFT-H 300 K ■ MFT-H 300 RL92 ■ MFT-H 300 T	Agraffes for undercut fastening with Hilti MFT-HP 300 profiles
Agraffes > Hilti > MFT-H 200	■ MFT-H 200 K ■ MFT-H 200 RL92 ■ MFT-H 200 T	Agraffes for undercut fastening with Hilti MFT-HP 200 profiles
Clamps > Hilti	■ Clip MFT-C ■ Facade studs MFT-C BO	Safety clips and studs for suspended facades with Hilti MFT-CP profiles



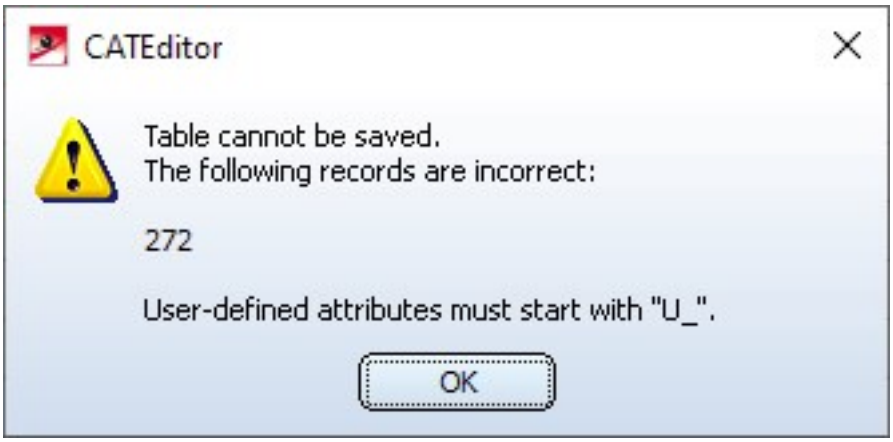
Changed catalogue structure for building materials

From HiCAD 2021 onwards, semi-finished products for civil engineering, such as foils or insulations, can be found in the catalogue **Factory standards > User-defined semi-finished products**.



Customer-specific system attributes

In practice, there is often a desire to use own, customer-specific system attributes. Until now, this was only possible through the staff of the ISD. As of HiCAD 2021, customers can now also do this themselves. However, it should be noted here that the names of customer-specific attributes must always begin with **U_**. If this is not the case, the table cannot be saved. In this case, a corresponding error message appears:



Customer-specific system attributes can be used like other system attributes in attribute masks, for annotation, etc.

Rectified Salzgitter catalogue entries

The catalogue entries for Salzgitter profiles contained errors: The tables Sandwich panels, roof and Sandwich panels, wall from Salzgitter (to be found in the catalogue at the path Factory standards > Series > Roof Wall Facade > Room-closing profiles > Salzgitter Building elements) contained incorrect weights and U-values. This has now been corrected.

Profiles that have already been installed are not automatically corrected; instead, they must be updated manually.

For manually installed profiles, recalculation is not sufficient. Instead, the profile type must first be changed to another profile and then back again.

For profiles that have been installed via the Profile Installation, such a change is not sufficient. Instead, at least the layer on which the profiles are installed must be deleted and recreated. However, it is more advisable to recreate the entire profile installation.

Point clouds

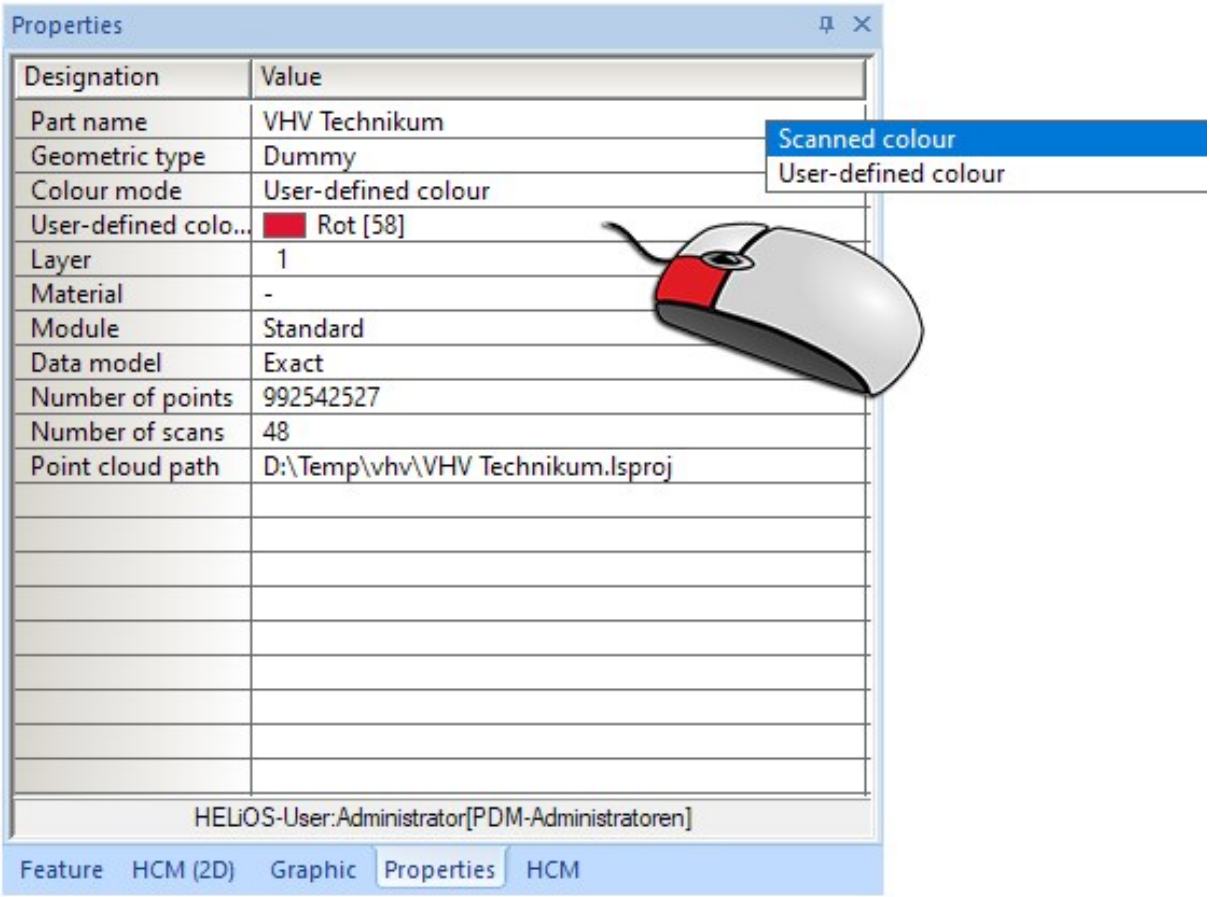
Service Pack 1 2021 (V 2601)

Colour of point clouds

From HiCAD 2021 SP1 onwards a false colour representation can be selected for point clouds. This is particularly interesting for point clouds that do not have their own colour information.

The colour is assigned via the ICN window **Properties** and there via the rows **Colour mode** and **User-defined colour**.

With the colour mode you have the choice between scanned and user-defined colour. To select the mode, double-click in the column Value of the line Colour mode. You can then change the colour mode.



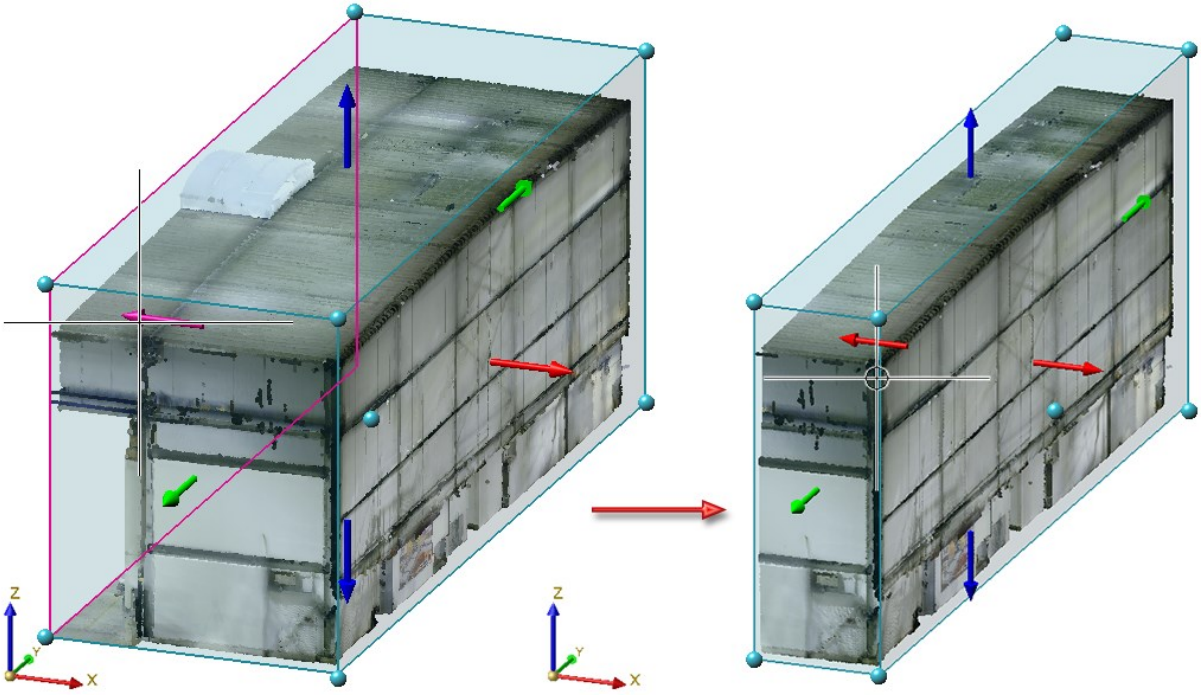
To assign a user-defined colour to the point cloud, double-click in the **Value** column of the corresponding row and then select the desired colour.

Exchanging identical point cloud references

If several point cloud parts have a missing reference when loading a model drawing, it is often the same reference. In this case, the reference does not have to be selected again for each part, but all parts with identical reference are updated accordingly. This means that it is sufficient to select the new reference for one of these parts.

Clipping boxes

When defining and editing clipping boxes, point symbols are now also displayed at the corners of the box. By dragging these symbols, the clipping box can also be changed. This way you can also use point options of the **Auto-pilot** and the **Point options** menu.

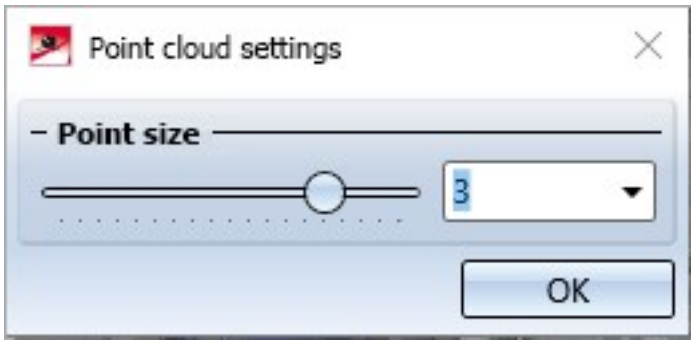


Major Release 2021 (V 2600)

Point cloud settings

New on the **Point cloud** Ribbon tab is the **Point cloud settings** function. With this function the representation of the points of a point cloud can be reduced and enlarged. This can be useful in order to obtain an image that is as "flat" as possible, even when zooming in closer to a point cloud.

After calling the function, the **Point cloud settings** dialogue window is displayed:



Here you can change the point size dynamically with the slider, by selecting a value in the selection box, or by direct value input. The changed size is immediately visualized in the construction and can be accepted with **OK**. The value for the point size must be between 2 and 10. The default setting at the start of HiCAD is 3, during the current HiCAD session the last used value.

Example:

The image below shows a point cloud with point size 3.

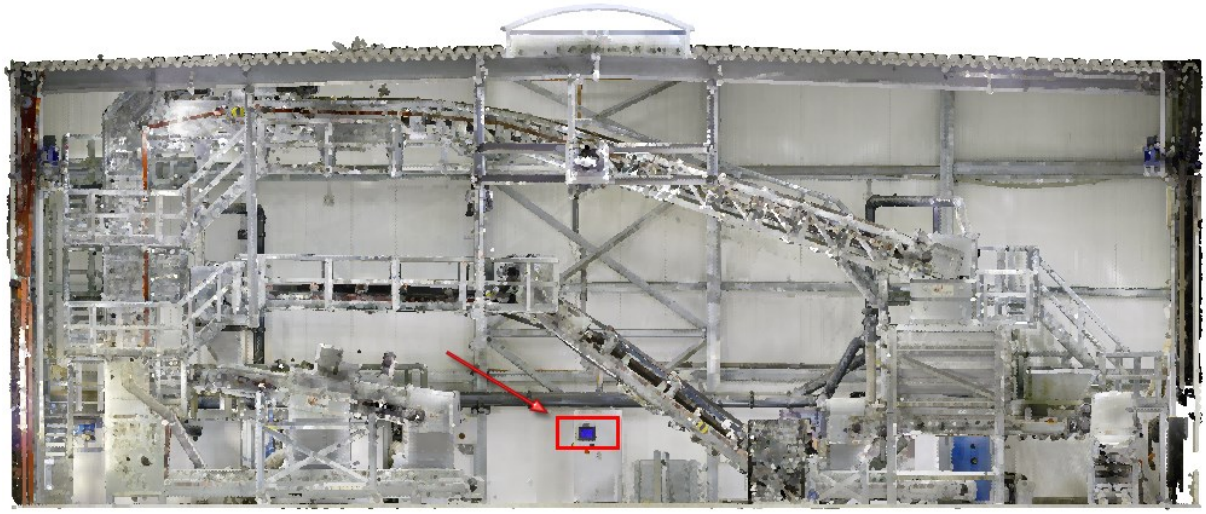
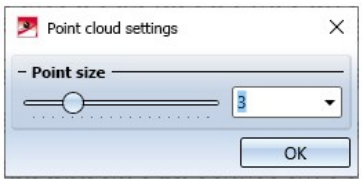
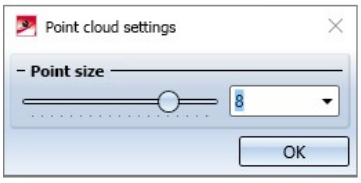


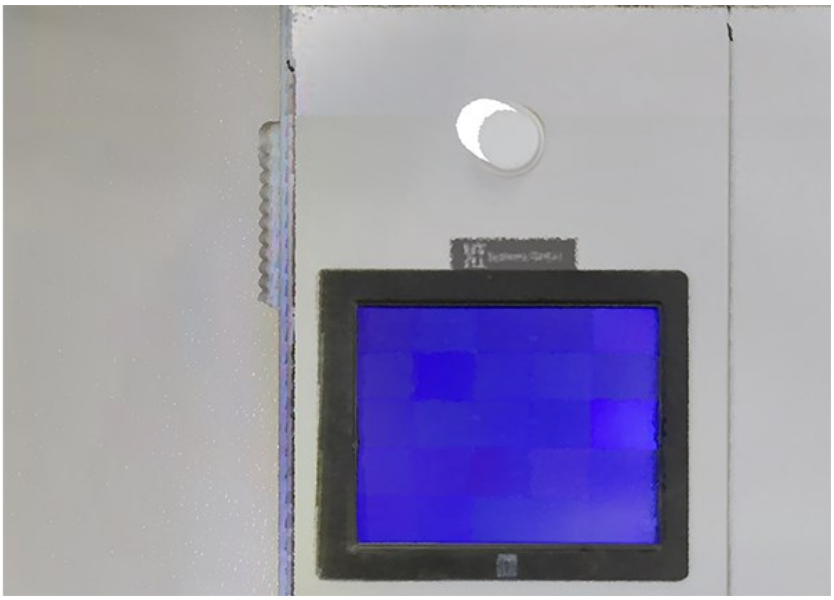
Image: VHV Anlagenbau GmbH, Hörstel

The section marked in red is to be zoomed in several times.



If you now increase the point size to the value 8, you will get the following representation.



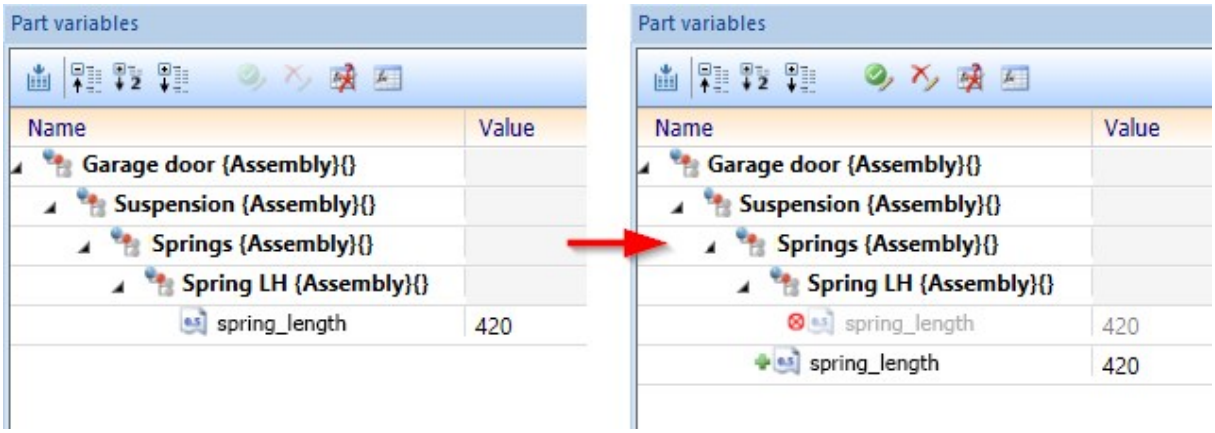


Feature Technology

Service Pack 1 2021 (V 2601)

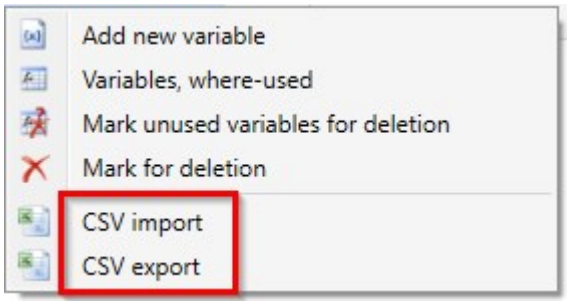
Drag & Drop, Import and Export in the Part variables docking window

The Part variables window of the ICN has been extended by further functions: It is now possible to move part variables within an assembly structure by Drag & Drop.



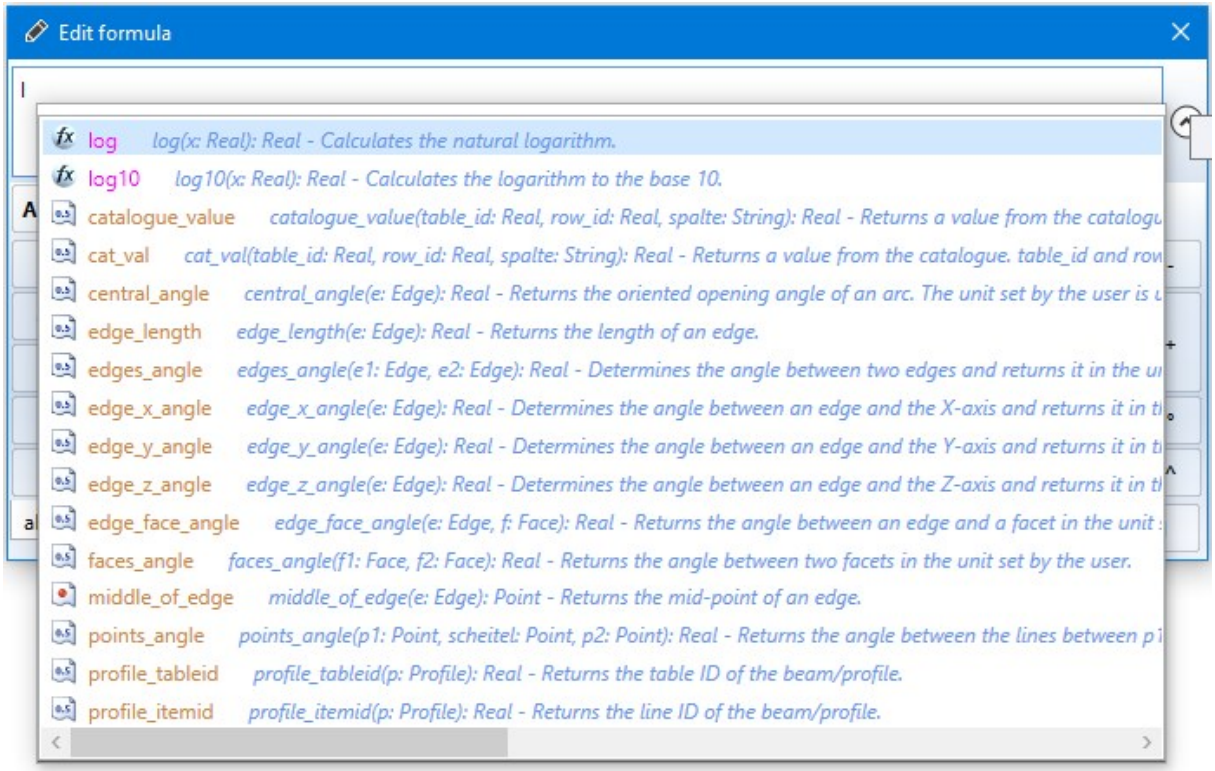
Example of a Drag & Drop operation: Here the variable spring_length was moved from the Assembly "Spring LH" to the Assembly "Springs".

Furthermore, the commands CSV export and CSV import are available in the context menu of the parts or assemblies.



Improved auto-completion in the Formula Editor

The auto-completion for variables and functions in the HiCAD **Formula Editor** has been optimised. The suggestions are now displayed in one line, which makes long lists clearer. The selected suggestion can be accepted by pressing **Enter** or **Tab** or by simply clicking with the left mouse button. In addition, different symbols are displayed in front of functions and variables to make the different types clearer..



Performance

The recalculation of assemblies that have been parameterised with the Part HCM has been accelerated by up to a factor of 7.

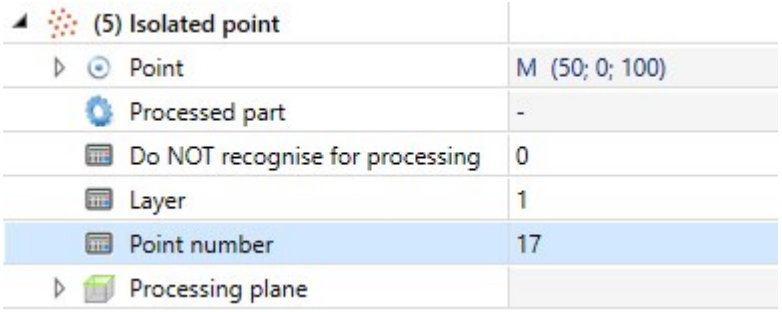
Configuration Editor: Renamed parameter

At **System settings > Feature** in the Configuration Editor, the parameter "Use external references" has been renamed to **Activate external references for new parts** to better highlight the effects of this option.

Point numbers on isolated points

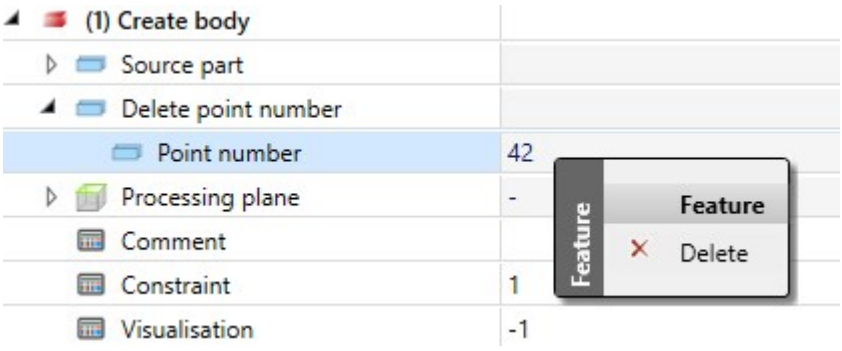
Previously, when creating an isolated point with a point number, two entries were always created in the feature log: first an **Isolated point** feature for creating the point, and then a **Point number** feature for assigning the point number.

This behaviour has now been changed so that when a point number is assigned, the **Point number** attribute of the **Isolated point** feature is now set instead of creating a separate log entry for it.



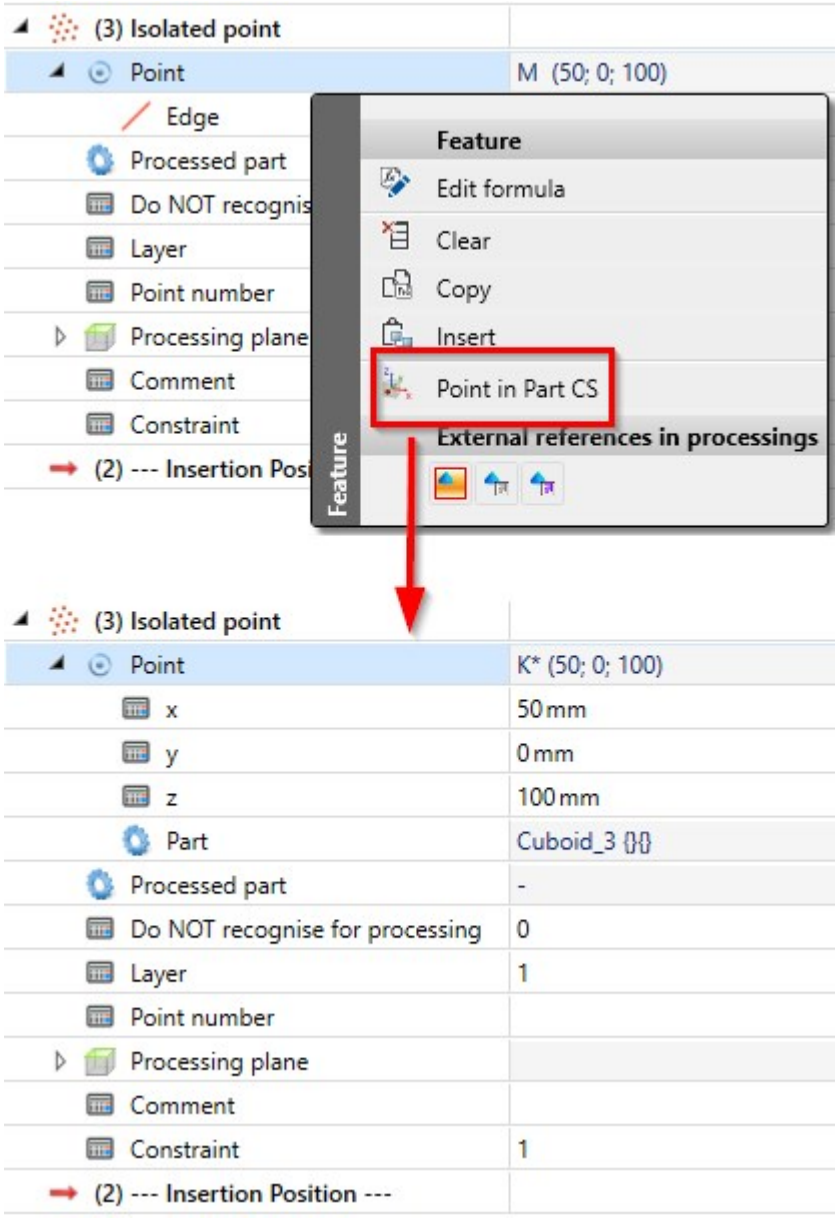
This is also handled accordingly by the **Delete point number** functions, so that they now empty the **Point number** attribute. This has the consequence that the deletion of point numbers remains even after a feature recalculation.

Special cases here are parts whose feature log was activated when points and point numbers were already created. If you delete point numbers in this situation, they will be entered as a separate **Delete point designation** entry at the **Create body** feature. Here you can delete individual deleted point numbers via the **Delete** context menu functions or delete the entire list via the **Clear** function, whereby the point numbers are restored at the latest with the next feature recalculation.



Point in Part CS

It is now possible to convert point options of features to absolute coordinates in the part coordinate system. For this purpose, the new function **Point in Part CS** is available in the context menu of a point option of a feature, which converts the previously selected point.

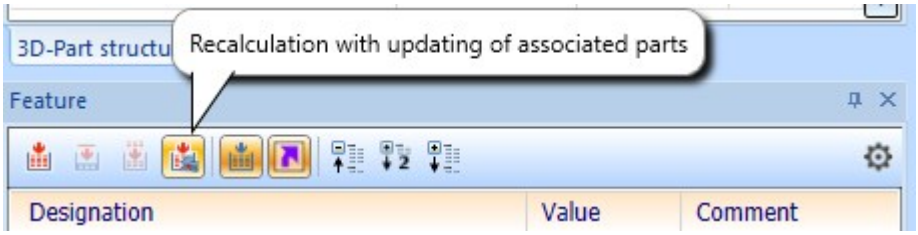


Top: Point, selected with point option I; Bottom: Point after calling the **Point in Part CS** function.

Major Release 2021 (V 2600)

Recalculation with update of associated parts

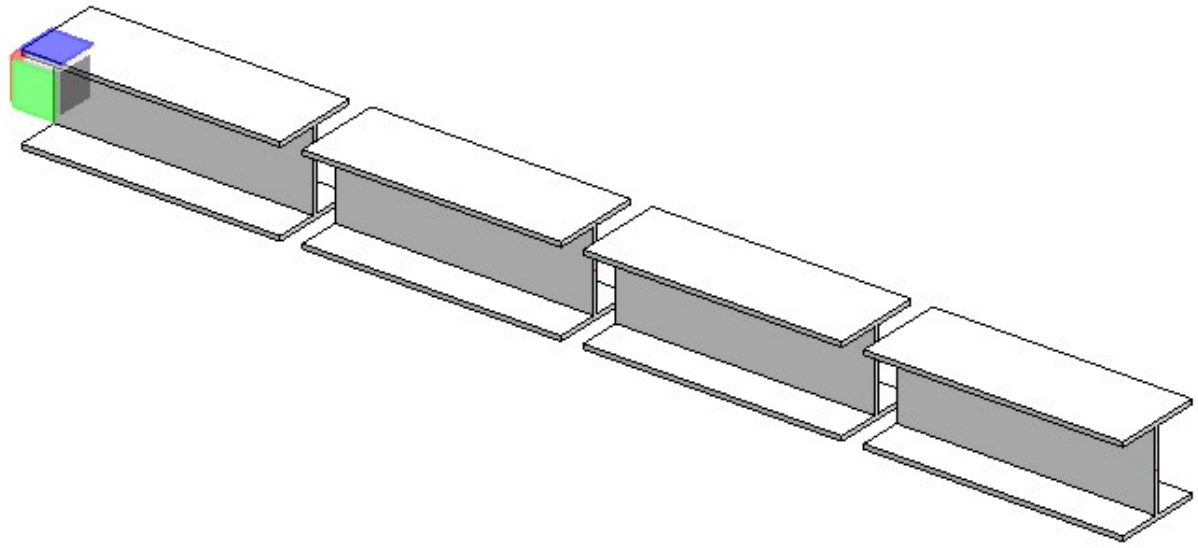
In the case of functions that concern several parts and their feature logs (such as the connection between an element installation and a sub-structure, the individual parts created by the function **Divide part along direction** or beams/profiles connected to each other via steel engineering connections), it was only possible to recalculate completely in a complicated and time-consuming manner when changes were made to original parts. To simplify this procedure, the new function **Recalculation with update of associated parts** has now been introduced.



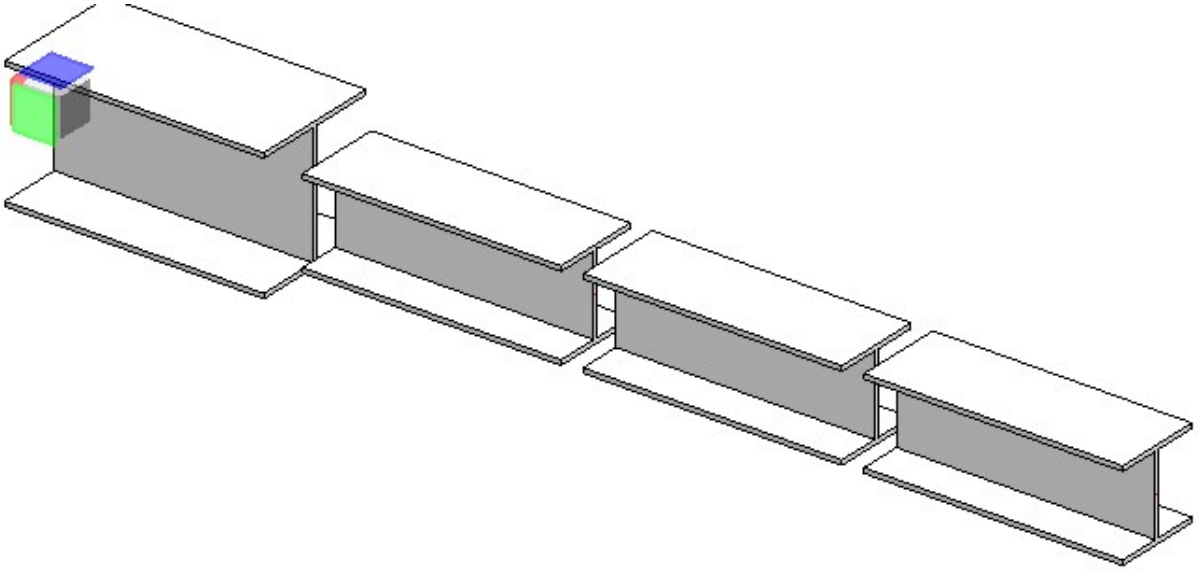
This initially behaves like a normal **Recalculation**. However, all parts connected to the current component are also updated afterwards, which means that changes to the original part then also affect the connected parts.

Example 1:

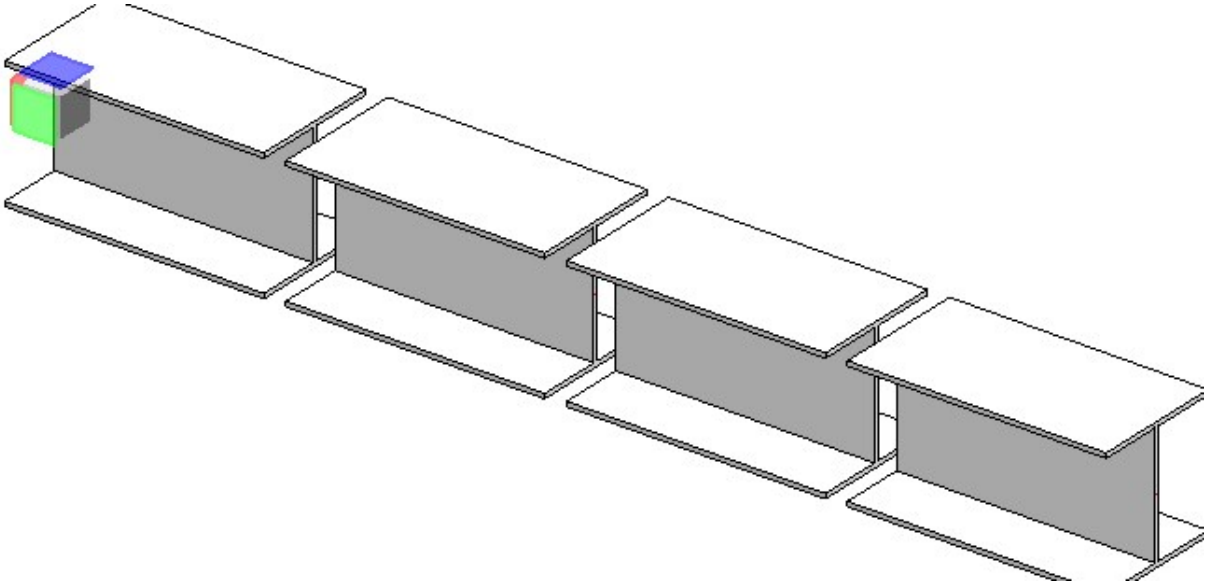
A steel engineering beam was divided into four parts with the function **Divide along direction**.



The feature log of the first part is now used to change the type of the first beam: This change only affects the selected part:

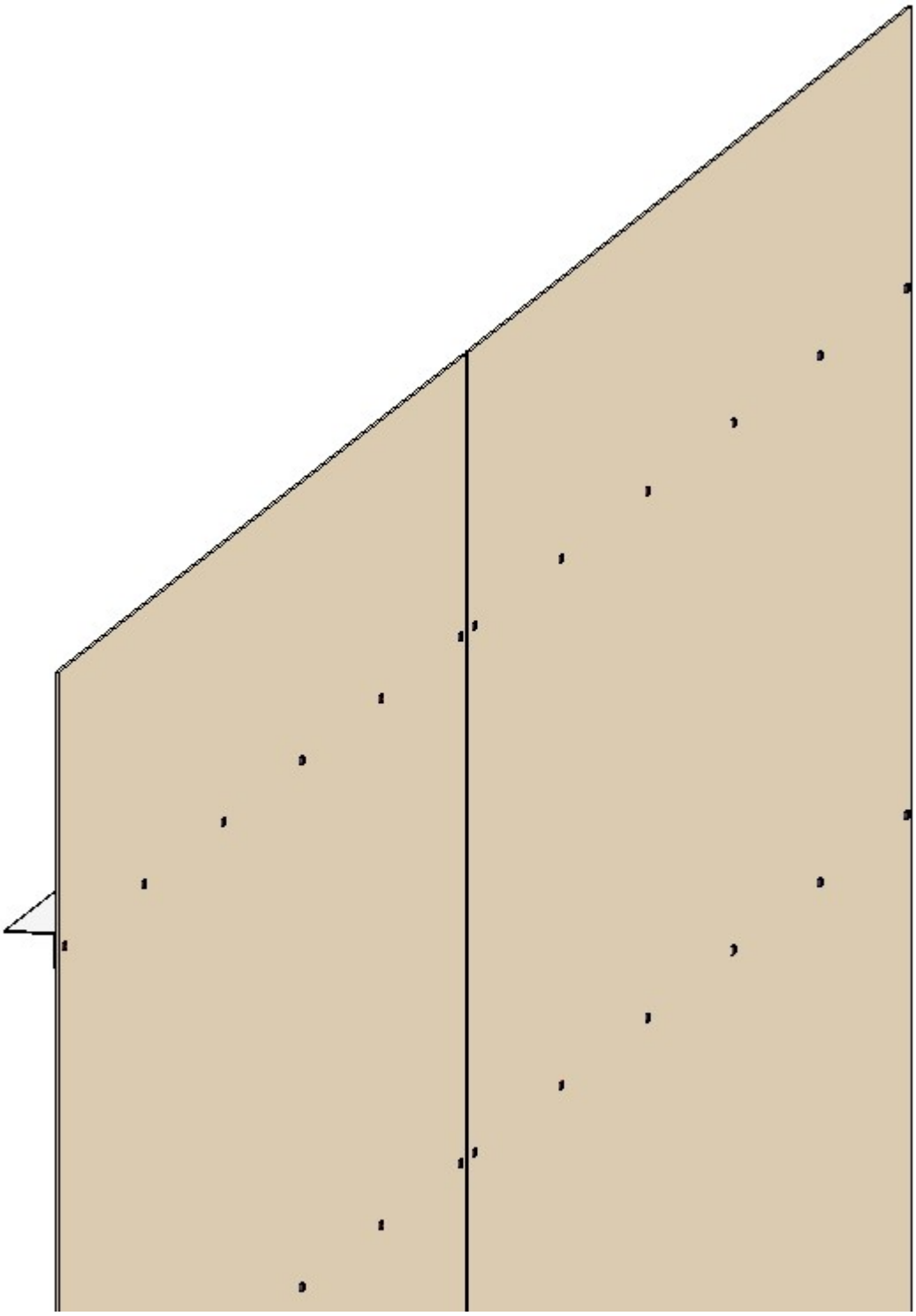


Only after selecting the function **Recalculation with update of associated parts**, the other parts also receive the changed beam type:

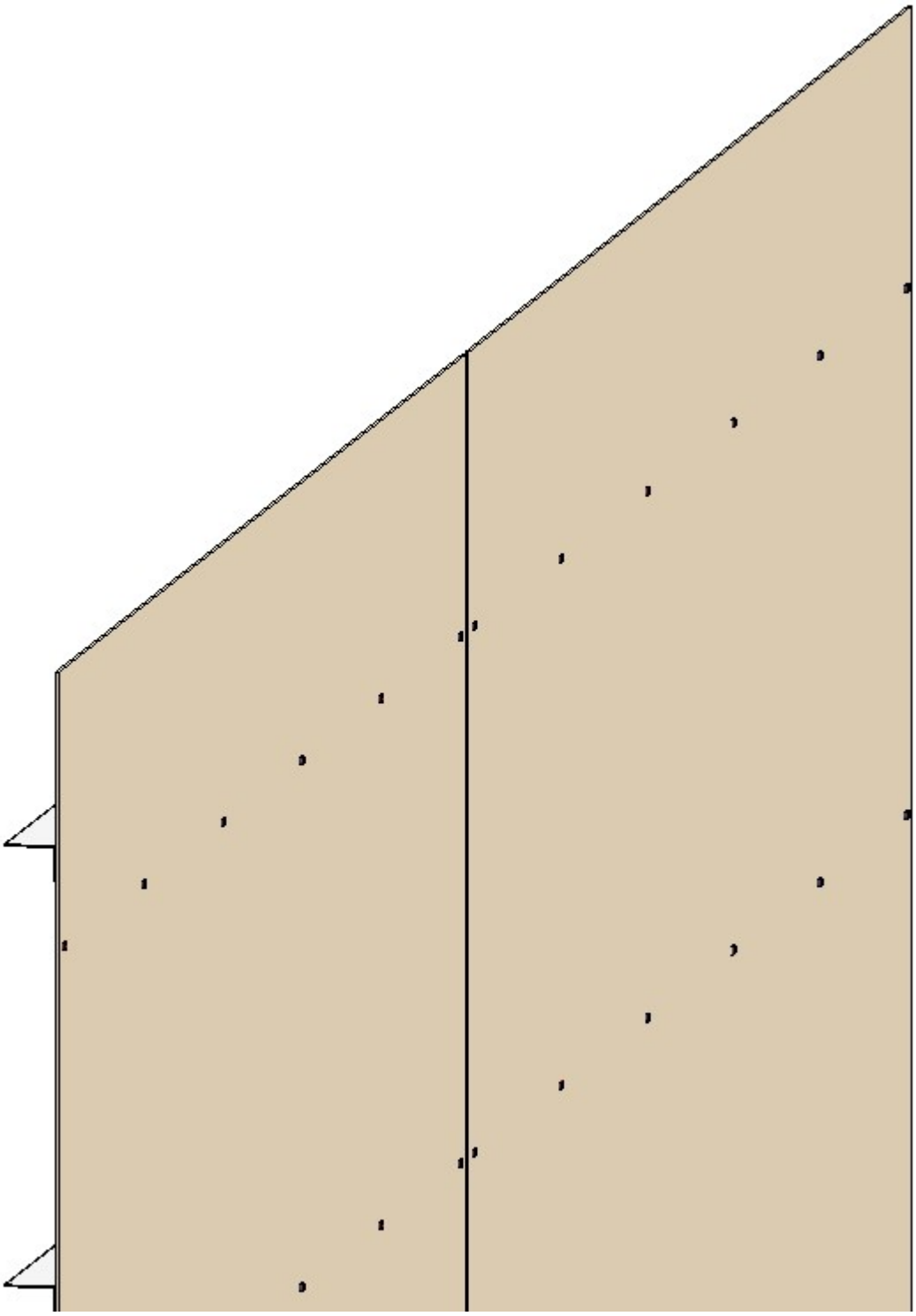


Example 2:

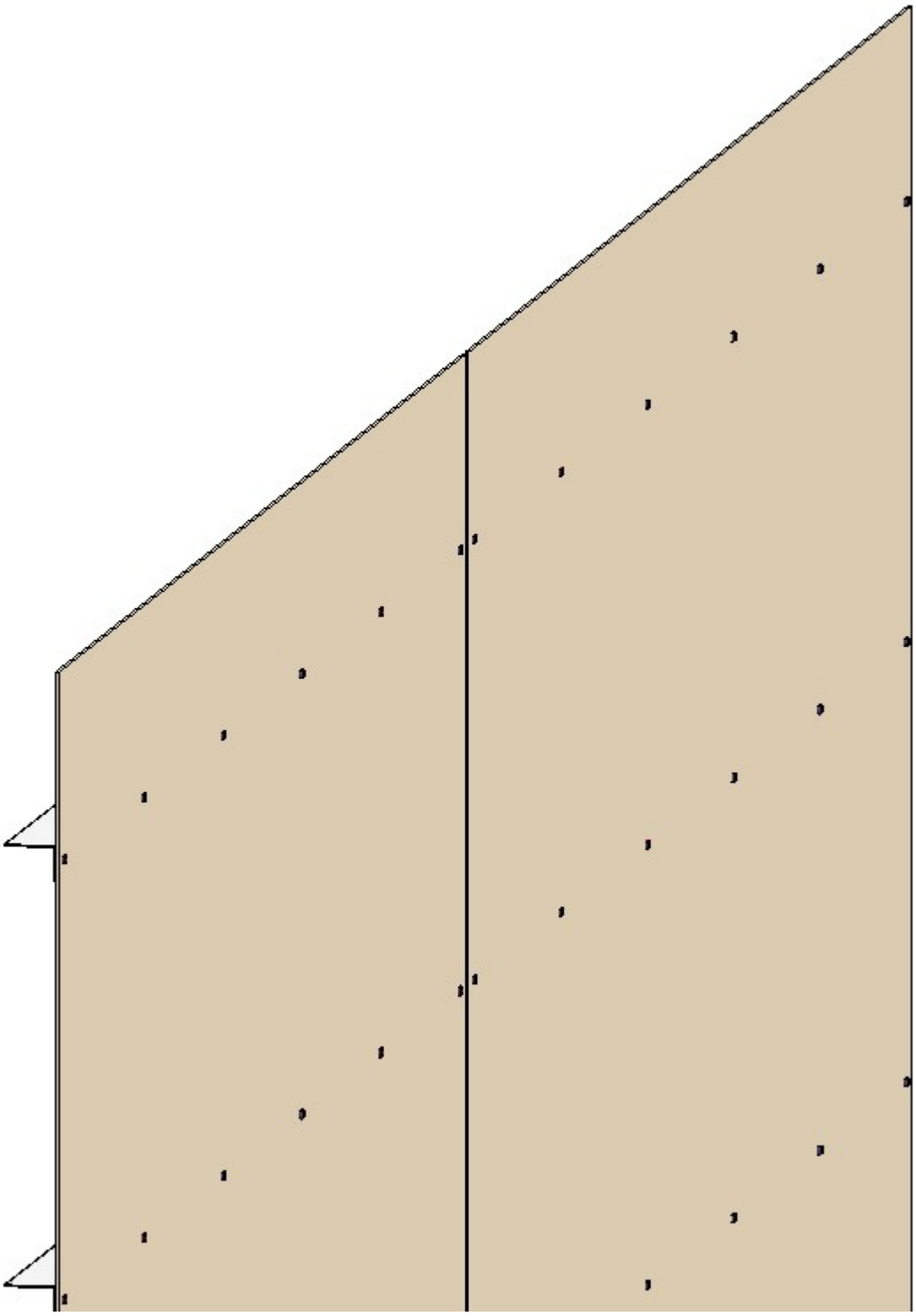
A connection consisting of an element installation with panels of the type ALUCOBOND, riveted and a sub-structure of Hilti MT-F profiles:



The sub-structure is changed so that the profiles now have smaller distances to each other. The rivets and bores in the element installation, which were created via the feature **Connection**, remain unaffected by this change and are still in the old, now still incorrect position:



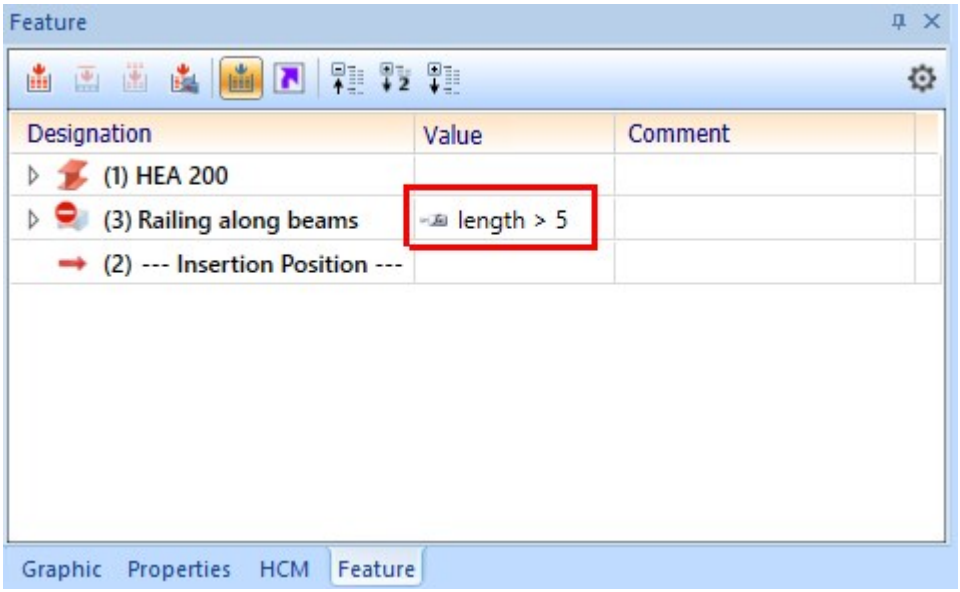
After calling up the function **Recalculation with updating of associated parts**, the rivets and bores are placed in the correct positions again:



Improvements for the Feature log

The following improvements have been introduced when working with feature logs:

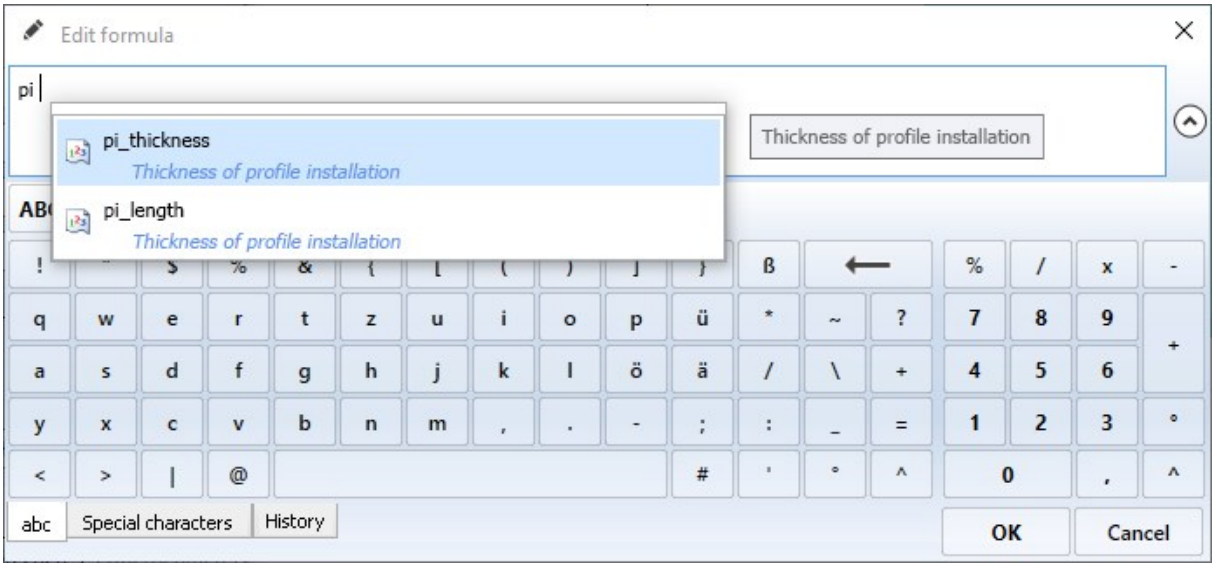
- For feature steps that contain a formula as a **Constraint**, this is now displayed as a **Value** directly at the feature step if the **Multi column structure** option is activated. From here, it can also be edited directly by double-clicking. However, the values 0 and 1 are not displayed.



- If several feature steps are selected, the context menu functions **Activate / deactivate feature** are now also selectable if only a part of the selected steps can be activated / deactivated.
- For feature steps that are deactivated via the Constraint entry, the icon of the feature step behind the "Deactivated" icon is now greyed out making it easier to recognise deactivated feature steps:

AutoCompletion for part variables

The redesigned **HiCAD Formula Editor** now also offers auto-completion for part variables.



Currently, the auto-completion only takes place in selected dialogues and then only if the respective part is already inserted in the drawing.

Example: Create a new base sheet via **Sheet Metal > New > Base sheet**, insert it into an assembly with part variables or provide it with part variables yourself, then open the dialogue window by double-clicking on the **Base sheet** entry in the Feature log and activate the Formula Editor by right-clicking on one of the input fields and selecting **Edit formula**. The available variables are then suggested to you as auto-completion.

A complete example of how to work with the Formula Editor can be found here.

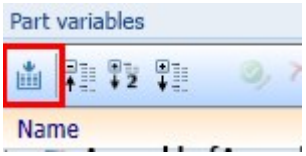
Modulo operator for Double values

The previous `%` operator for calculating the modulo (remainder in case of division, Example: `10 % 3` = 1) only works with integer values. In order to be able to work with decimal numbers as well, the new function `modulo(a; b)` was introduced, which accepts both integer and double values. The existing `%` operator remains unchanged for reasons of backwards compatibility.

Easier working with variables

The following functions have been added when working with variables in the **Part variables** docking window::

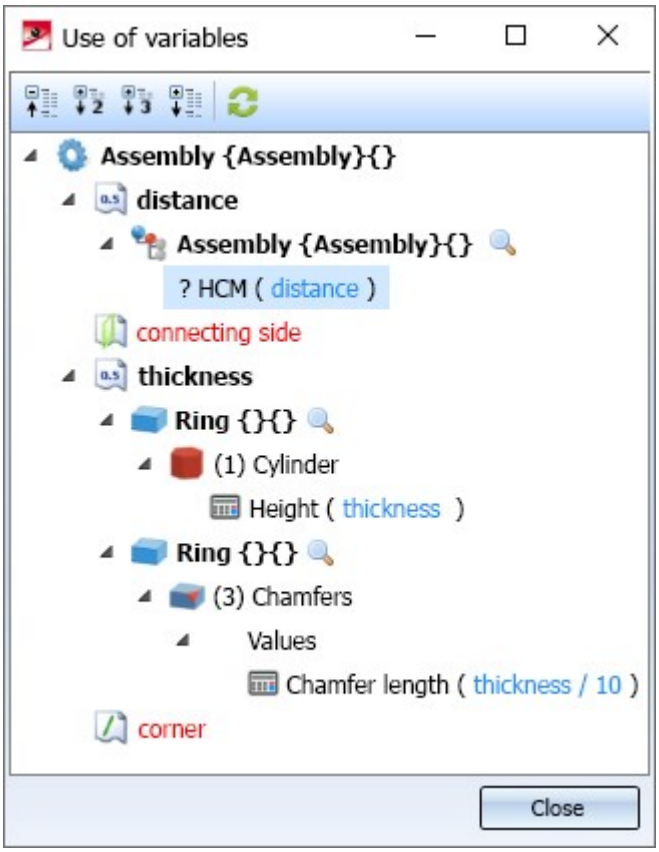
- Since the recalculation of the feature log when changing variables is also controlled by the **Automatic recalculation** function, a button for activating or deactivating the automatic recalculation has been added to the toolbar of the Part variables dialogue window.



- Double-clicking on the value of a variable now directly opens the calculator and thus enables a comfortable change of the value.
- The functions Cut, Copy, Paste and Delete are now also directly accessible via the key combinations CTRL+X, CTRL+C, CTRL+V and DEL.

Optimized "Use of variables" dialogue window

The **Use of variables** dialogue window has been optimized.



The innovations include:

- While the **Use of variables** dialogue is open, work with HiCAD can continue in the background.
- In the toolbar, the function **Update** was added, with which the use of variables can be updated without having to reopen the dialogue.
- Behind the components in which a variable is used, a magnifying glass symbol is now displayed which, when clicked, centres the corresponding component in the construction and zooms in.
- The labels of the individual uses are now more compact: Instead of "Name=Height Formula=thickness", "Height (thickness)" is now displayed.
- Identical feature elements are now grouped together in the tree.
- This is also displayed for variables that are used in an HCM model.

Display of element variables

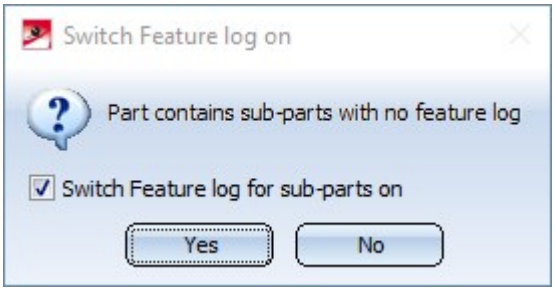
When working with element variables, the elements they reference are now highlighted in the drawing as soon as an element variable is selected in the Part variables docking window.

Combined functions "Enter body creation feature" und "Switch feature on"

The **Enter body creation feature** function has been removed. Instead, the **Switch feature on** function has been extended so that it now automatically adds a body creation feature if the selected part contains a geometry. If no geometry is present, simply the feature log is activated as before.

Switch feature on, with sub-parts

If you call the **Switch feature on** function for a part that contains sub-parts that also do not have a feature log, HiCAD now offers the possibility to directly activate the feature logs of the sub-parts as well. The following dialogue window appears for this purpose:



If the selected part does not contain any sub-parts or if they all already have an active feature log, this dialogue window does not appear.

Units of measurement for variables

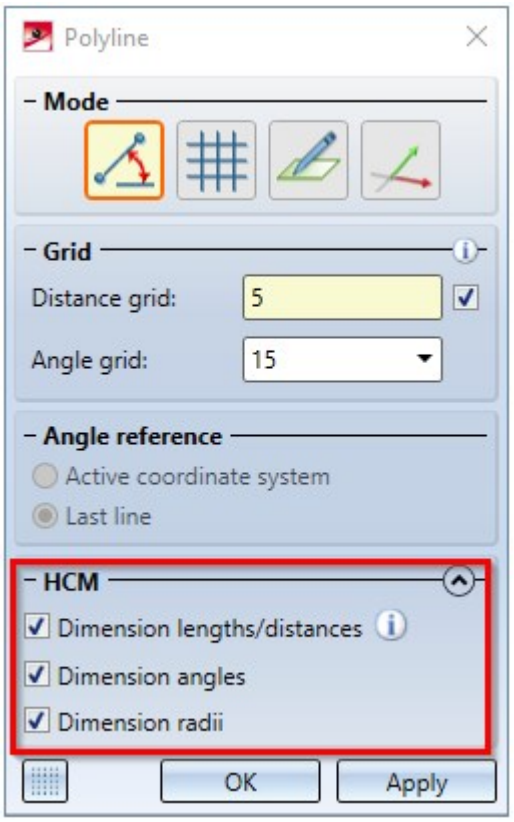
HiCAD 2021 supports only model drawings with the unit of measurement mm. Drawings with with any other unit of measurement cannot be opened.

HCM

Service Pack 1 2021 (V 2601)

Automatic generation of HCM constraints during sketching

In the course of changing the sketch functions, the automatic creation of HCM constraints has also been adjusted. Thus, in the **Settings** dialogue window, it is now only generally to be set whether automatic creation of HCM constraint is desired during sketching. The other settings, which constraints exactly are to be generated, have been moved to the dialogues for the individual sketch functions. This now makes it possible to quickly switch the generation of constraints on and off during designing.



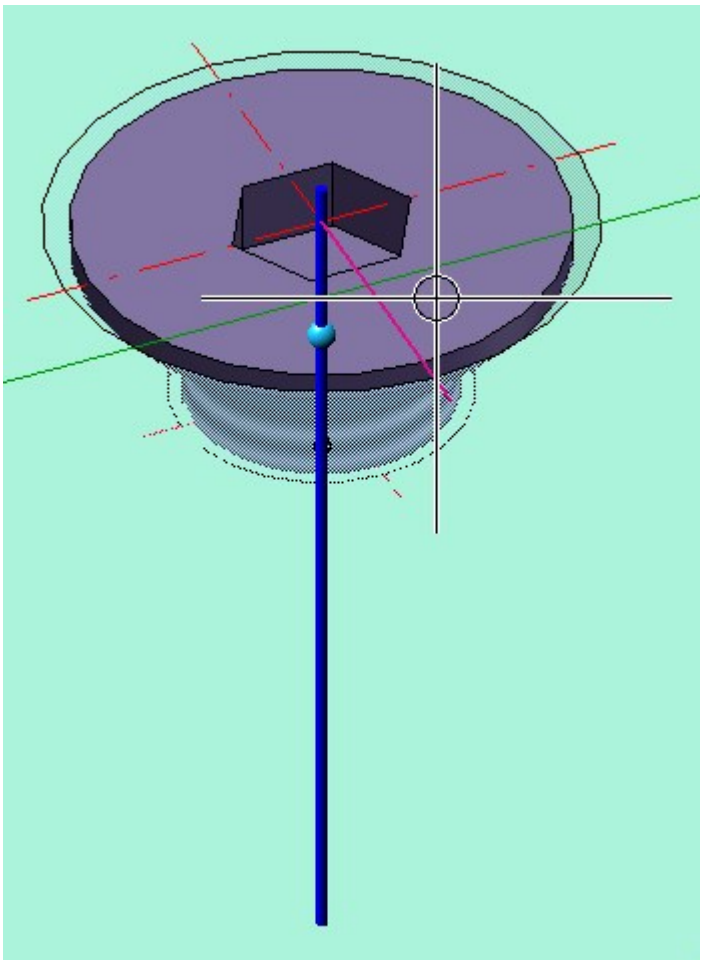
Visibility of dimensions no longer refers to parameter dimensions

In the **Visualisation** toolbar the function **Toggle visibility of dimensions** has been renamed to **Toggle visibility of drawing dimensions** and now no longer affects parameter dimensions, but only toggles the visibility of 2-D and 3-D drawing dimensions.

Improved Z-axis display for fasteners

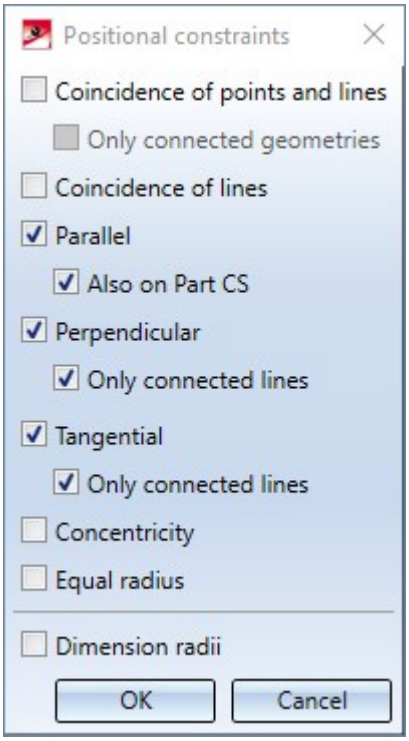
If you want to couple a part with a fastener via an HCM constraint, the Z-axis of the fastener is often of interest. This is highlighted by HiCAD at this time and can be selected directly by mouse click.

What is new in Service Pack 1 is that the Z-axis is now extended during display and protrudes a little beyond the part both at the front and at the back. This makes it much easier to select the Z-axis without having to rotate the view first.



New automatic positional constraints

When constructing sketches, the **Automatic positional constraint** function is available, which automatically generates positional constraints. This function has been extended and can now generate more types of positional constraints.



The new positional constraints are:

- Coincidence of points and lines
  - Optionally only for connected geometries
- Coincidence of lines
- Concentricity
- Equal radius

Major Release 2021 (V 2600)

Improved setting of constraints in the part HCM

When setting constraints in the Part HCM that refer to cylinders, spheres, cones or tori, the recognition of the desired orientation has been improved. This is particularly noticeable with **Tangent** constraints. Here, it is now always attempted to set the side of the part that was clicked on when setting the constraint tangentially. Afterwards, the orientation can still be changed via the Orientation selection field in the HCM constraint dialogue.

Improvements for dragging in Sketch-HCM

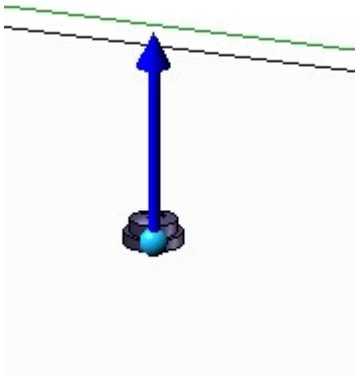
The following improvements have been made for dragging in the Sketch-HCM:

- If the snap mode **Element snap** is active, sketch elements can be dragged directly via Drag & Drop without having to call up the **Drag** function first.
- Points can now be dragged onto the following elements:
  - Edges
  - Coordinate system axes
  - The origin of the coordinate system
  - Centre points

  A **Coincidence** constraint is automatically created, with the following exceptions:
  - If the SHIFT key is pressed when dropping and the SHIFT key was not pressed when dragging started.
  - If the **Enter constraints** option is deactivated in the HCM settings via **Sketch > HCM > Tools** > **Settings**. If the constraint is an external reference, the **Use external references** option is also observed.
- When dragging points and line elements, they are attached to the cursor during dragging.
- When dragging a circle (not the centre of the circle), the centre of the circle remains fixed and the radius changes.
- When attempting to drag a fully defined sketch, a corresponding message appears.

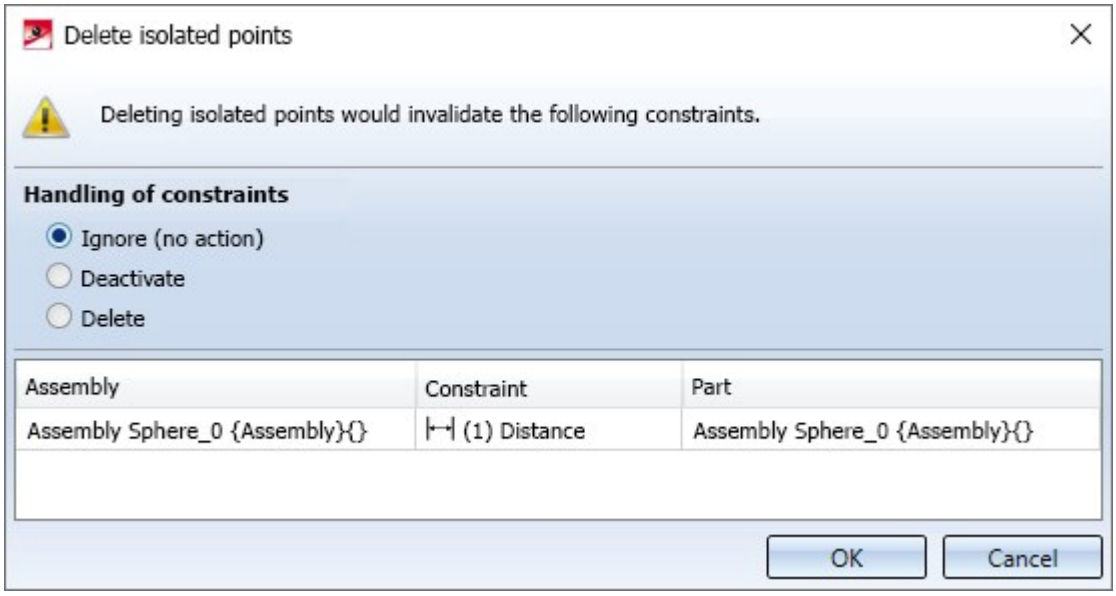
HCM constraints for fasteners

When assigning HCM constraints that refer to fasteners such as screws, reference is often made to the origin of the part coordinate system or the Z-axis of the screw. To simplify this step, HiCAD now automatically displays the origin and Z-axis of a fastener when you touch it with the mouse pointer during the assignment of an HCM-constraint. This display remains for a short time afterwards, allowing you to select origin or Z-axis.



Clearer error messages

In many places in the HCM system, displayed error messages have been revised and are now much clearer than before and also offer different options for dealing with the respective error:



For example, when deleting an isolated point referenced in an HCM constraint, it is displayed which parts in which assembly are affected via which constraint. In addition, it is offered to deactivate the corresponding constraints, to delete them or to ignore this error (which would then leave the constraints unfulfilled).

Change dimension figure and Move dimension figure for sketches functions made available again

The functions

- Sketch > HCM > Smart dimensioning > Change and
- Sketch > HCM > Drag > Move dimension line

have been made available again, especially to simplify work in the view area.

Improved operability of the context menus in the ICN

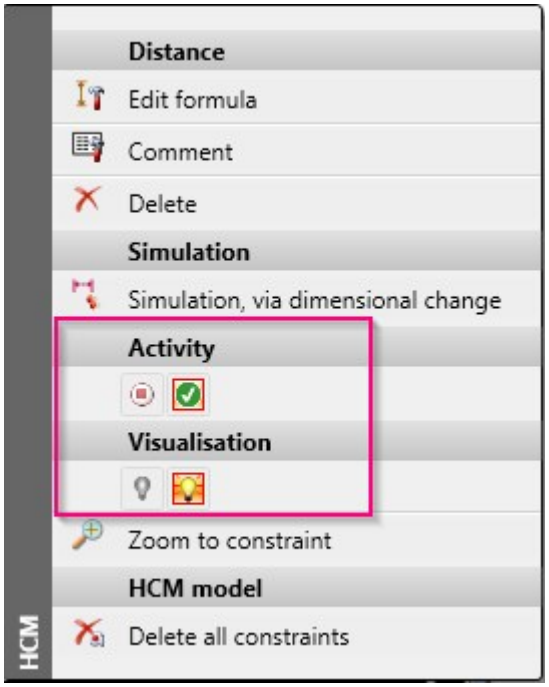
Via the context menu of the HCM area in the ICN, it is conveniently possible to change the properties of several HCM constraints simultaneously. Thus, several constraints can be selected in the tabular list and collectively deactivated via the context menu, for example.

Until now, only functions that could be applied to all selected HCM constraints were offered in the context menu, e.g. the function **Hide constraint** was only offered if none of the selected constraints was already hidden. This behaviour has now been adapted so that this function is always offered and can therefore also be applied if some of the selected constraints are already hidden..

The functions affected by this change are:

- Activate / deactivate constraint
- Activate / deactivate constraints of a part
- Use / do not use external references
- Show / Hide constraint
- Show / Hide part
- Allow / lock rotation (for Concentricity)
- Planarity of a spline
- Specify spline properties

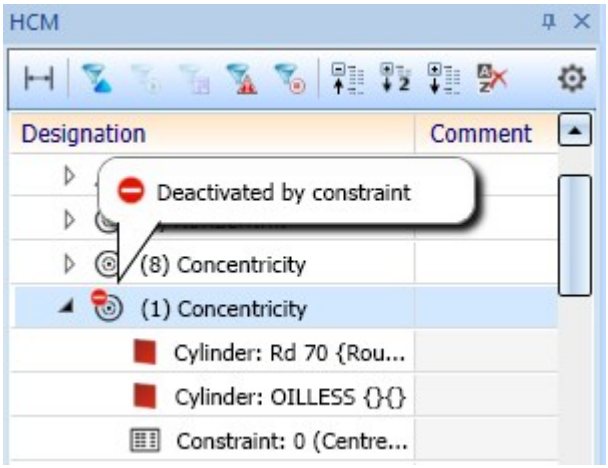
In addition, these functions are now displayed in the context menu as buttons only with icons without annotation:



If a state applies to all selected constraints, the corresponding button is marked with a red frame. (In the example, all constraints are activated, so the button **Activate constraints** is marked with a red frame under **Activity**). If a state is mixed for the selected constraints, no such frame is displayed (in the example, the selected constraints are partly hidden and partly shown, therefore no frame is displayed in the **Visualisation** area).

Display of deactivated constraints

HCM constraints that have been deactivated by another **Constraint** are now indicated by a new symbol:



Units of measurement

HiCAD 2021 only supports model drawings with the unit of measurement mm. Model drawings with any other unit of measurement cannot be opened.

Configuration Management

Service Pack 1 2021 (V 2601)

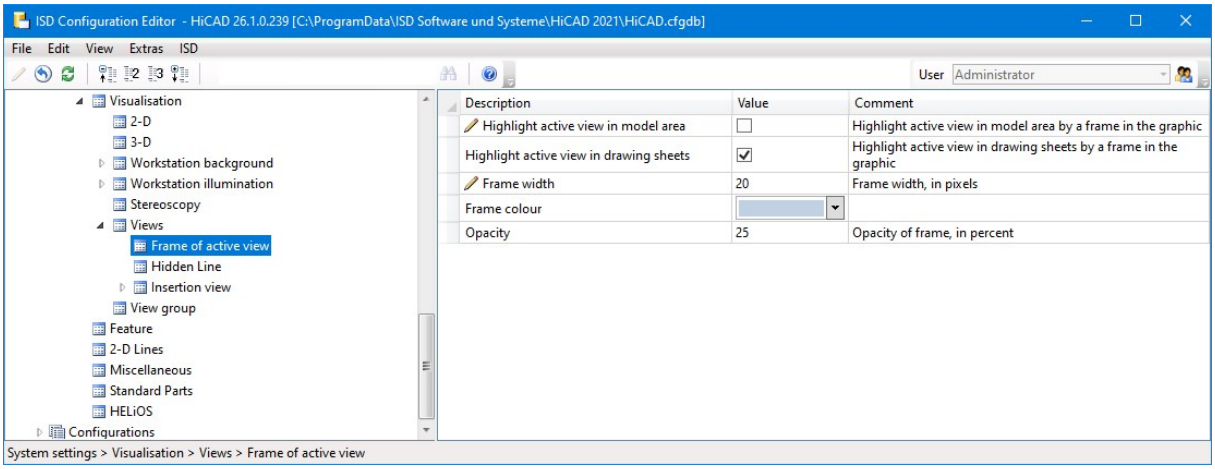
AutoQuickView

The previous user guidance texts in the context menu of the drawing at **Properties > AutoQuickView** as well as in the **Properties** function group of the **Drawing** Ribbon tab have been changed to emphasise the automatic aspect more clearly.

The change of the user guidance text also concerns the settings in the Configuration Editor at **System settings > Visualisation > Views > AutoQuickView in new drawing**.

Representation of the active view

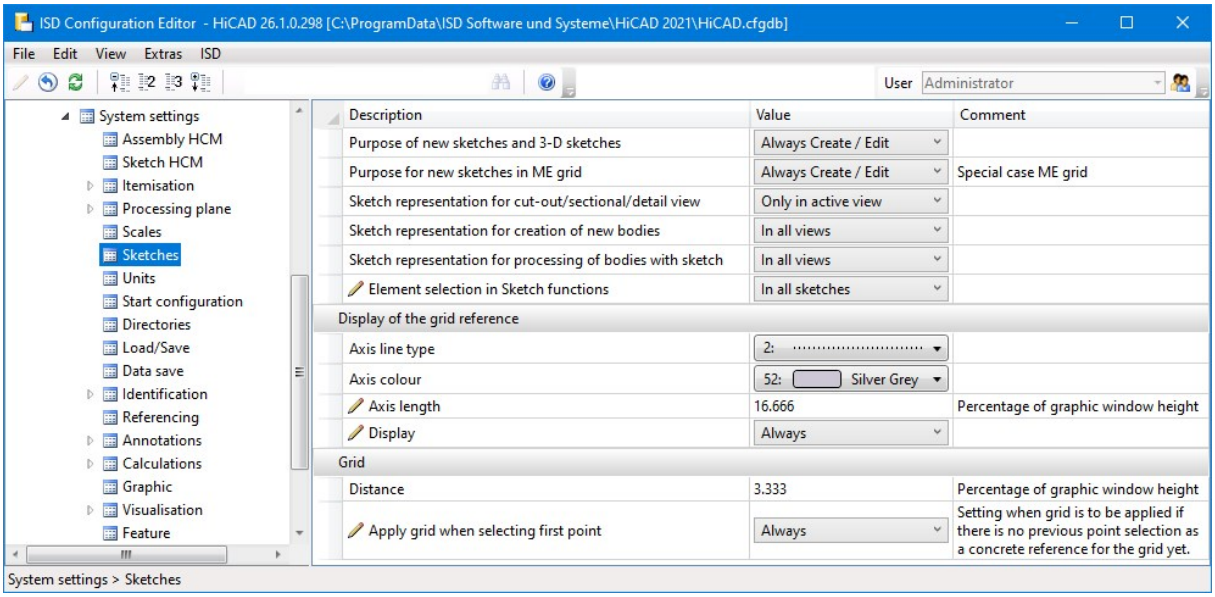
In the Configuration Editor at **System settings > Visualisation > Views > Frame of active view** it is now possible to define for the model space as well as for the drawing sheets (sheet views) whether the active view should be highlighted by a coloured frame in the drawing.



If one of the **Highlight active view in...** checkboxes is deactivated, the active view is displayed by a thin, dashed frame in the Special colour Marking 1. This corresponds to the display of the active view before HiCAD 2021.

Display of grid reference during sketching

The display of the grid reference when creating planar sketches and 3-D sketches with the sketching tool can be set in the Configuration Editor at **System settings > Sketches** and there in the section **Display of grid reference**.



The image above shows the ISD default settings.

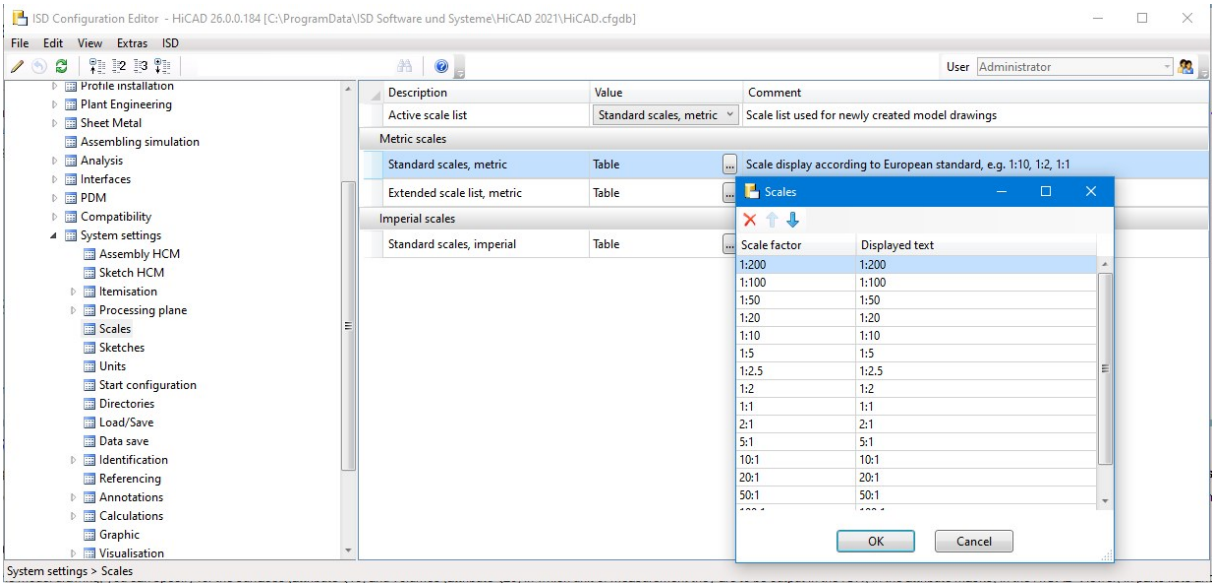
Previously, when using HiCAD in conjunction with HELiOS, the part exchange settings from the **HELiOS Database Options** on the **HiCAD** tab were taken into account when exchanging referenced parts. These settings can now be found in the Configuration Editor at **System Settings > Referencing** in the **HELiOS** section.

Major Release 2021 (V 2600)

Scale lists

The scale can be selected from a selection box in the corresponding HiCAD function dialogues. Which scales are available in the selection boxes was previously defined in the **SZENE-MASSSTAB.TXT** files in the HiCAD sub-directory **MAKRO2D**. As of HiCAD 2021, **Scale lists** can be edited and defined in the Configuration Editor at **System settings > Scales**. In addition, you can define which scale list is to be used by default for new model drawings.

The settings from the STBETZNG_MASSSTAB.DAT file are no longer relevant for HiCAD 2022. The file is therefore no longer available from HiCAD 2022 (Version 2700).



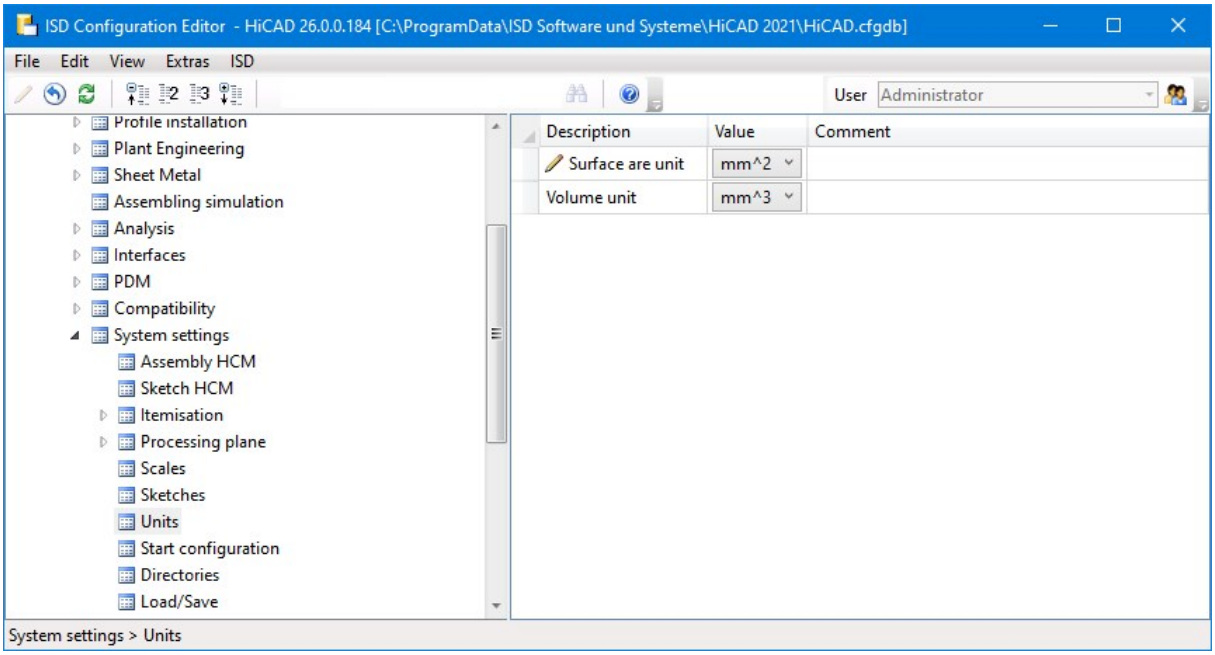
Unit of measurement for model drawings

HiCAD 2021 supports only model drawings with the unit of measurement mm. The previous **Unit of measurement** function is no longer available. This also applies to the **Unit of measurement** parameter in the Configuration Editor at **System settings > Miscellaneous**.

The unit of measurement is saved in the corresponding SZA or KRA file when saving a model drawing or a 3-D part. When trying to open model drawings with a different unit of measurement or insert parts with a unit of measurement different from mm, an error message is displayed.

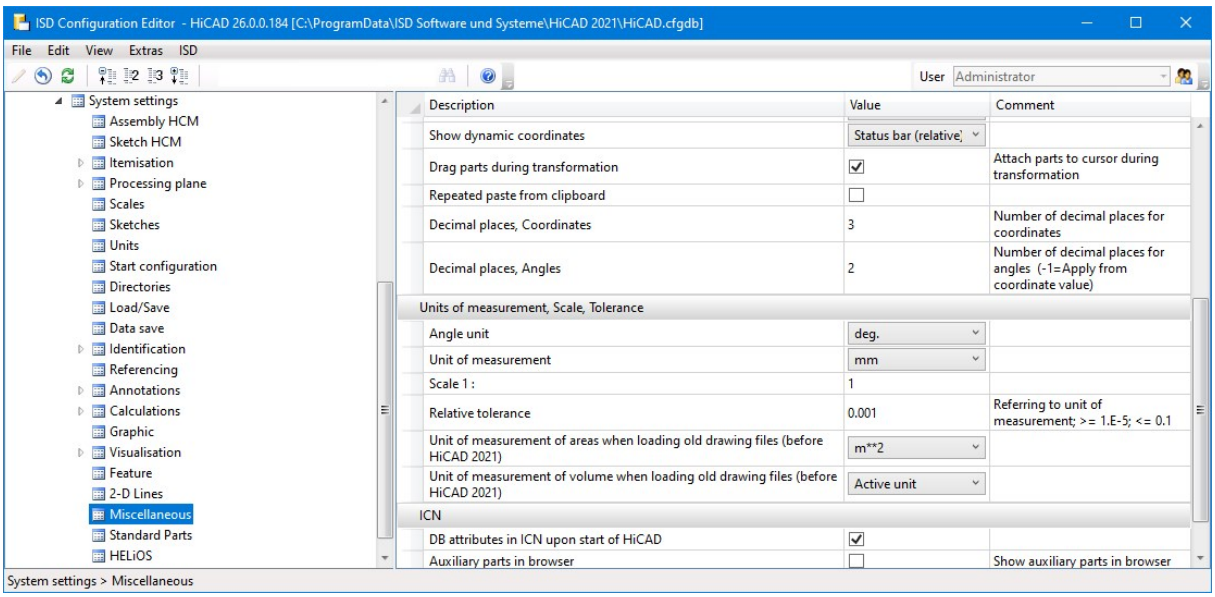
Unit of measurement for displaying surfaces and volumes

In addition to the unit of measurement of the model drawing, you can specify for the surfaces (attribute §10) and volumes (attribute §20) in which unit of measurement they are to be output in the ICN, in the attribute masks, in the HiCAD Viewer, in parts lists and when transferred to HELiOS. As of HiCAD 2021, you define this in the Configuration Editor at **System settings > Units**. The ISD default setting for surfaces is square metre m^2, for volumes cubic millimetre mm^3.



Unit of measurement of surfaces/volumes when loading "old" model drawings

For model drawings and parts created with an earlier HiCAD version (before HiCAD 2021), you can define in the Configuration Editor at **System settings > Miscellaneous** how to proceed when opening such constructions and parts for surface and volume calculations. The default setting for surfaces is square meters, for volumes **Active unit** i.e. cubic millimetres).



Centesimal/GON unit for angular dimensions withdrawn

The unit of measurement Centesimal/GON for angular dimensions is an auxiliary unit for indicating the angular width of planar angles, which could only be established in surveying. Therefore, it was removed from the following entries in HiCAD 2021:

- Drawings > Annotations > Dimensioning, 3-D > Interactive dimensions > Unit of measurement, Angular dimensions
- Drawings > Annotations > Dimensioning, 3-D > Interactive dimensions > Unit, 2nd dimension figure, Angular dimensions
- Drawings > Annotations > Dimensioning, 3-D > Parametric dimensions > Unit of measurement, Angular dimensions
- Drawings > Annotations > Dimensioning, 3-D > Parametric dimensions > Unit, 2nd dimension figure, Angular dimensions
- Drawings > Annotations > Dimensioning, 3-D > HCM dimensions > Unit of measurement, Angular dimensions
- Drawings > Annotations > Dimensioning, 3-D > HCM dimensions > Unit, 2nd dimension figure, Angular dimensions

Stereoscopic representation

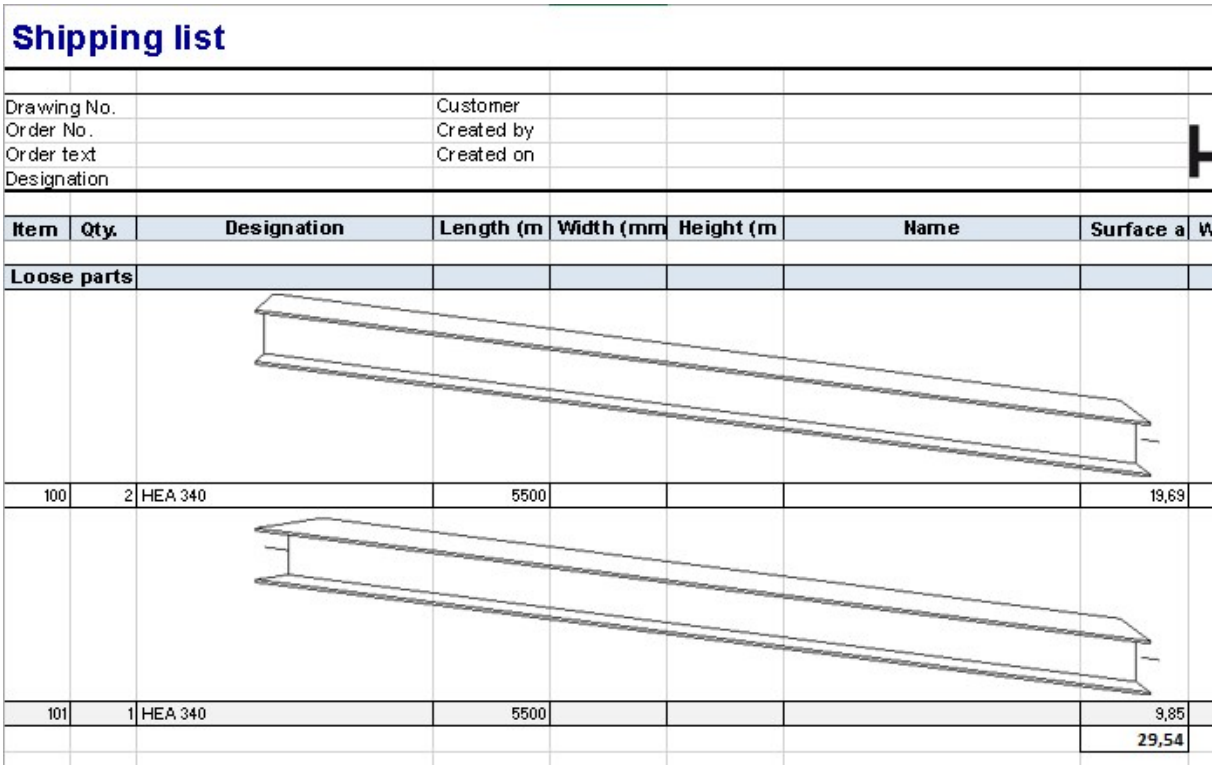
The function **Stereoscopic representation** (previously available at **Views > Properties > Stereo**) is no longer available as of HiCAD 2021. Therefore, the settings in the Configuration Editor at **System settings > Visualisation > Stereoscopy** have been removed.

Bill of Materials / Report Manager

Service Pack 1 2021 (V 2601)

Steel Engineering Excel template: Shipping list, short, with image

Another worksheet has been added to the Excel template **HiCAD_Stahlbau**: **Shipping list, short, with image**. It contains the same information as **Shipping list, short**, but additionally contains a picture of the parts.



Shipping list

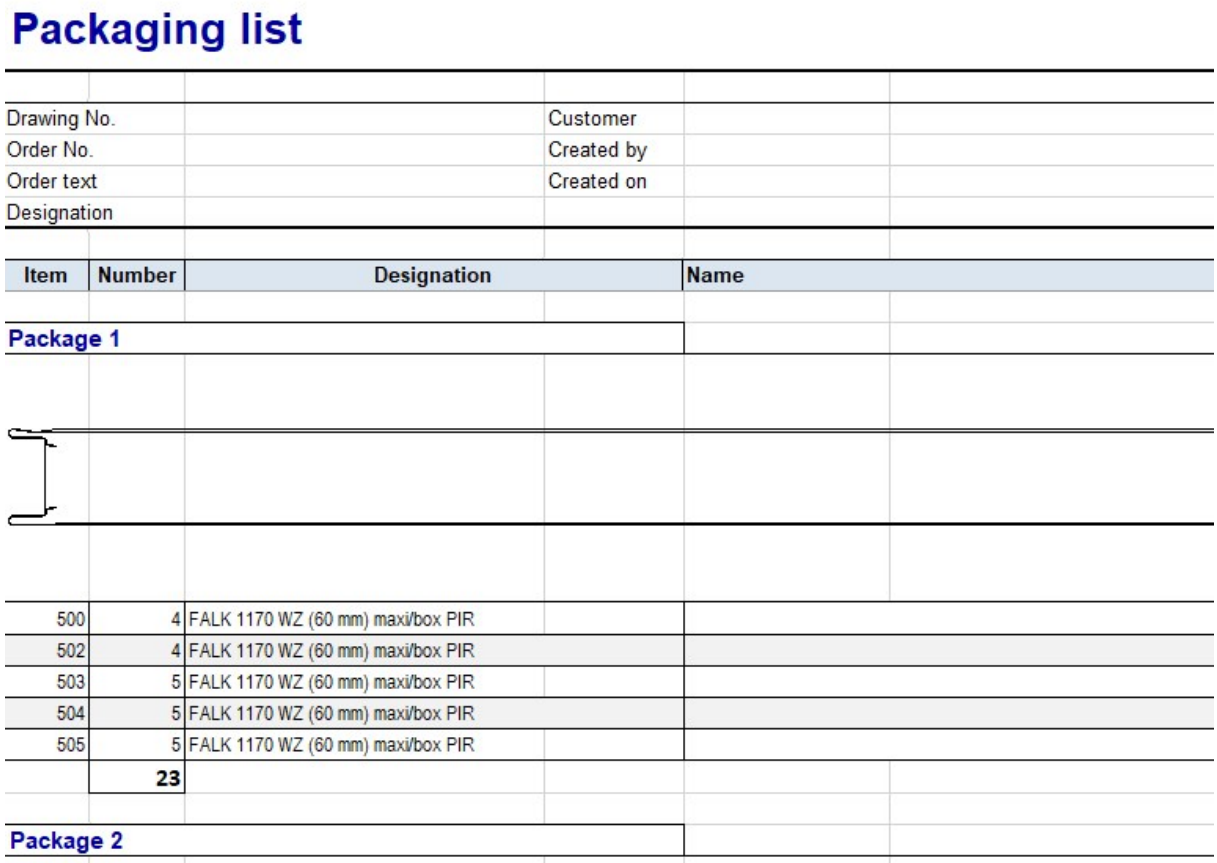
Drawing No.		Customer	
Order No.		Created by	
Order text		Created on	
Designation			

Item	Qty.	Designation	Length (m	Width (mm	Height (m	Name	Surface a	W
Loose parts								
100	2	HEA 340	5500				19,69	
101	1	HEA 340	5500				9,85	
							29,54	

Major Release 2021 (V 2600)

Packaging lists - Profile cross sections in Excel export

When exporting a packaging list for a profile installation to Excel, the profile cross-sections of the affected profiles are now displayed on the **Packaging list** spreadsheet.



Packaging list

Drawing No.		Customer	
Order No.		Created by	
Order text		Created on	
Designation			

Item	Number	Designation	Name
Package 1			
500	4	FALK 1170 WZ (60 mm) maxi/box PIR	
502	4	FALK 1170 WZ (60 mm) maxi/box PIR	
503	5	FALK 1170 WZ (60 mm) maxi/box PIR	
504	5	FALK 1170 WZ (60 mm) maxi/box PIR	
505	5	FALK 1170 WZ (60 mm) maxi/box PIR	
	23		
Package 2			

For this purpose, you must use the Report Manager configuration HICAD_STAHLBAU.2600.0.

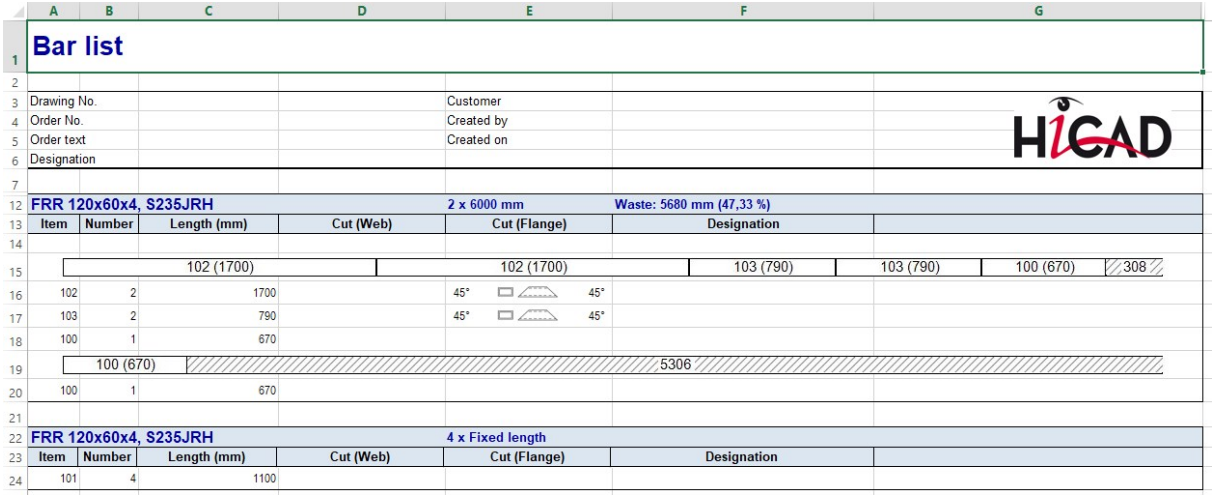
Bar lists - Optimized arrangement

When creating bar lists, the distribution of beams/profiles to the individual bars has been improved. This can lead to fewer bars being necessary to obtain the required beams/profiles.

Bar list - Beams and profiles with fixed lengths

In practice, beams or profiles are often used in model drawings, e.g. purchased parts, which are ordered for production with a fixed length. These beams/profiles should not appear in the bar list or must be shown separately.

In HiCAD, this can now be regulated via the attribute **Fixed length**(%FIXLEN), which must be assigned to the corresponding beams/profiles. If this attribute has the value 0 or is not set at all, the fixed length is not taken into account in the bar list. If the value is 1, then these beams/profiles are listed separately in the bar list.



Bar list

Drawing No.		Customer	
Order No.		Created by	
Order text		Created on	
Designation			

FRR 120x60x4, S235JRH — 2 x 6000 mm — Waste: 5680 mm (47,33 %)

Item	Number	Length (mm)	Cut (Web)	Cut (Flange)	Designation
		102 (1700)		102 (1700)	103 (790) | 103 (790) | 100 (670) | 308
102	2	1700		45° 45°	
103	2	790		45° 45°	
100	1	670			
		100 (670)		5306	
100	1	670			

FRR 120x60x4, S235JRH — 4 x Fixed length

Item	Number	Length (mm)	Cut (Web)	Cut (Flange)	Designation
101	4	1100			

By default, the attribute is not set, i.e. it is not included in the attribute masks. If you want to use this attribute, you must manually extend the attribute mask for beams and profiles by changing the files

- **BRW_3DTeil_Profil.HDX** (part attributes for beams and profiles) and
- **BRW_3DTeil_Profil_H.HDX** (Parts for beams and profiles which are assembly main parts)

accordingly (see also **Steel Engineering - What's New?**).

Interfaces

Service Pack 1 2021 (V 2601)

Unification of the interface names

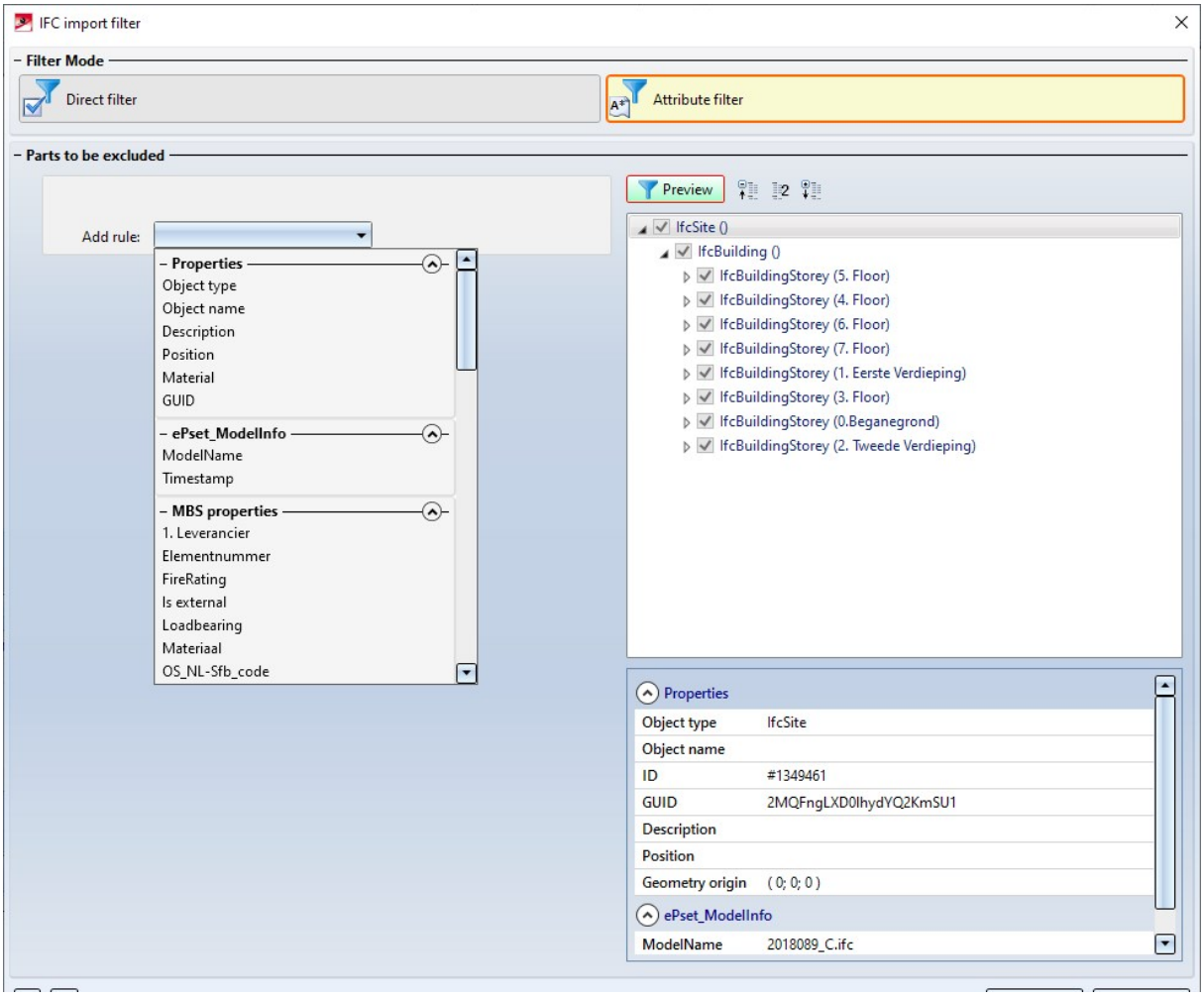
With SP1 the interface names in the different functions and dialogs have been unified, e.g.:

- Pro/E -> Creo Parametric
- JTOpen -> JT
- Unigraphics ->NX

An overview of the interfaces can be found here.

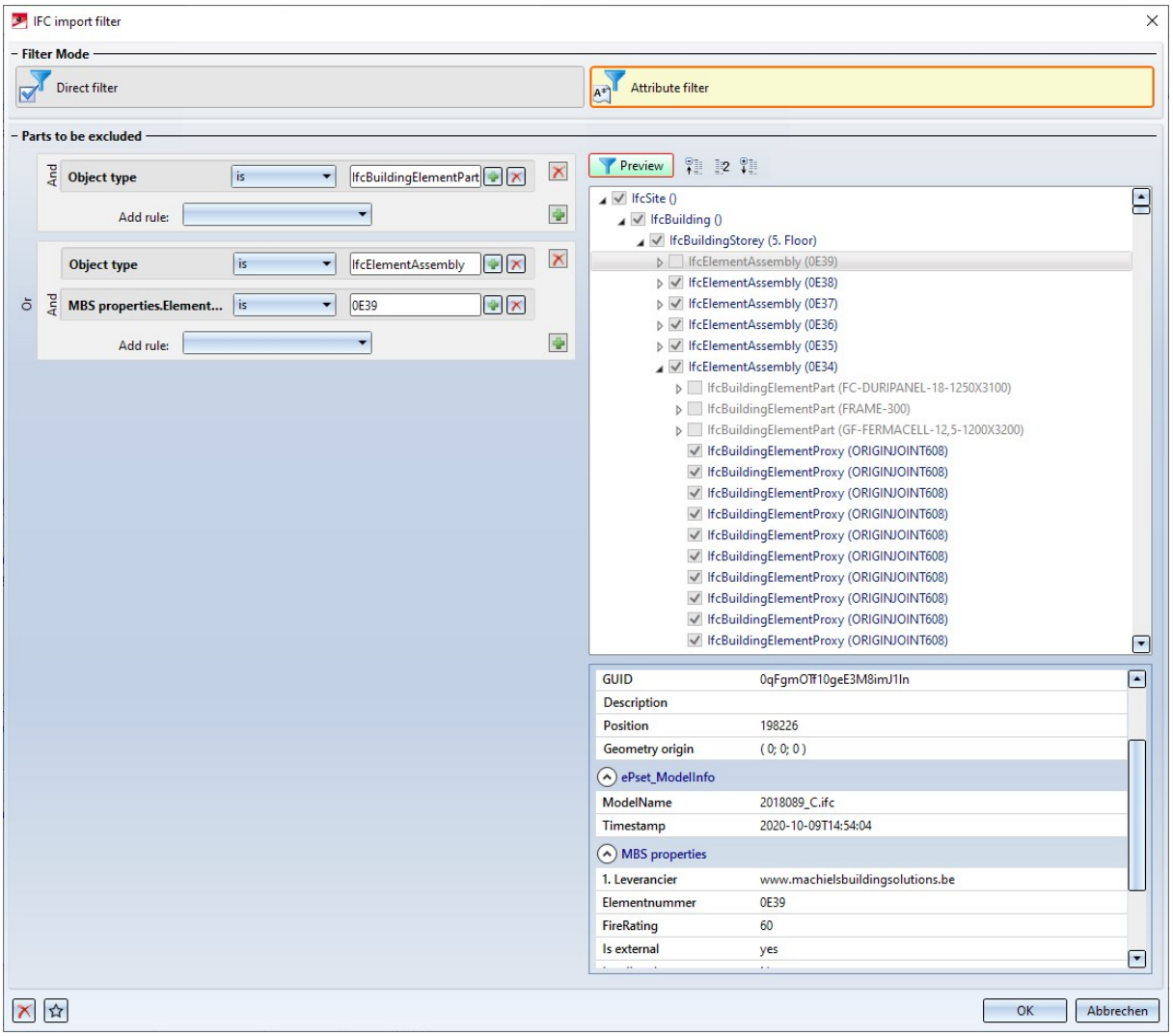
IFC Import - Enhanced attribute filters

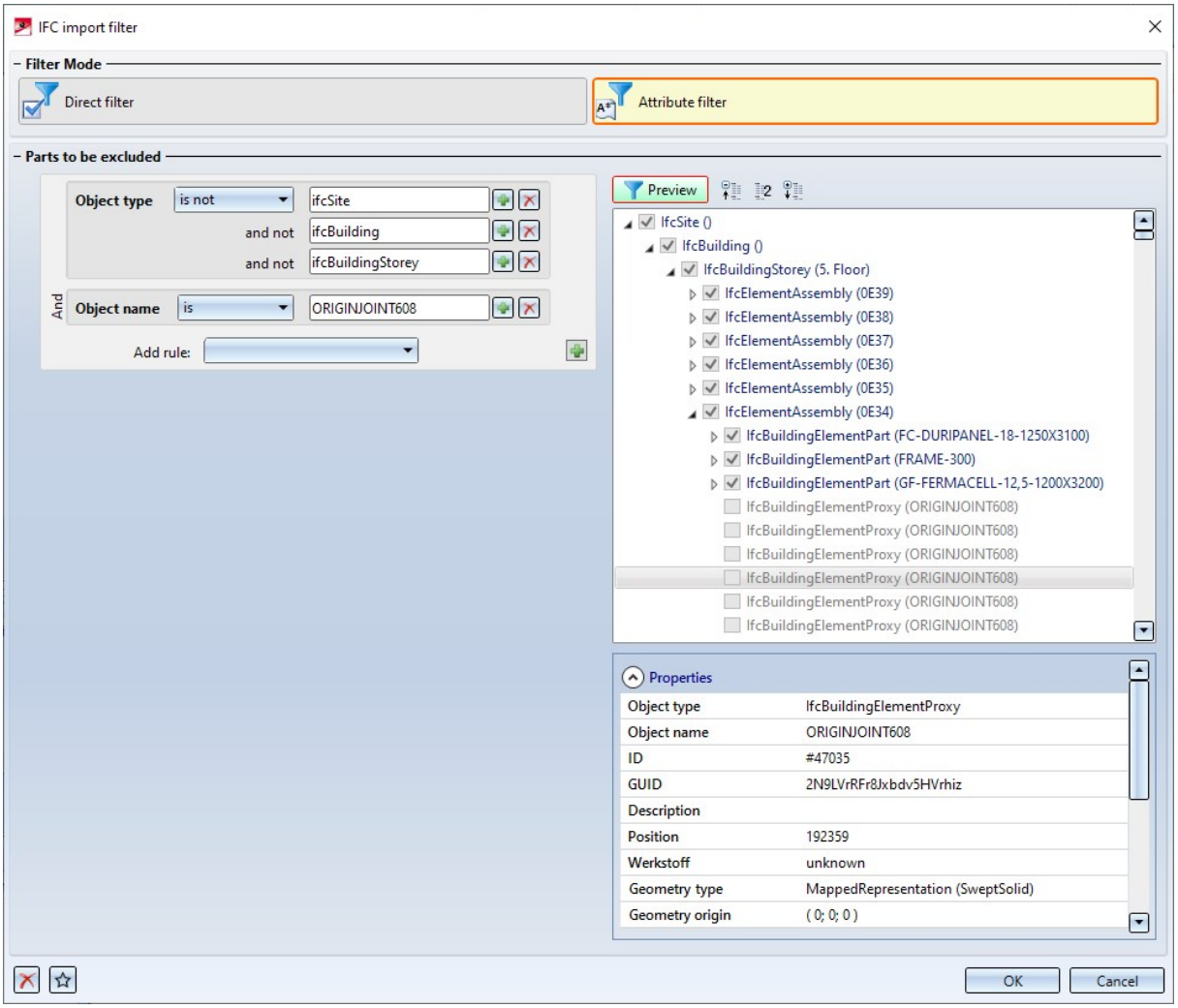
As of SP1, the properties **Position** and **Material** are additionally available as attribute filters. Furthermore, the file can now also be filtered by Property Sets. These property sets depend on the source system and the respective file. Several filter criteria can be combined with AND or OR.



This now covers the following scenarios, for example:

- Exclude all assemblies with the name xxx including the auxiliary parts as well as all parts with a certain criterion from the remaining assemblies
- Importing all assemblies with the name xxx except parts with certain criteria





- Contents of the **Properties** window can be copied with CTRL+C or via the context menu and pasted into the filters with CTRL+V.

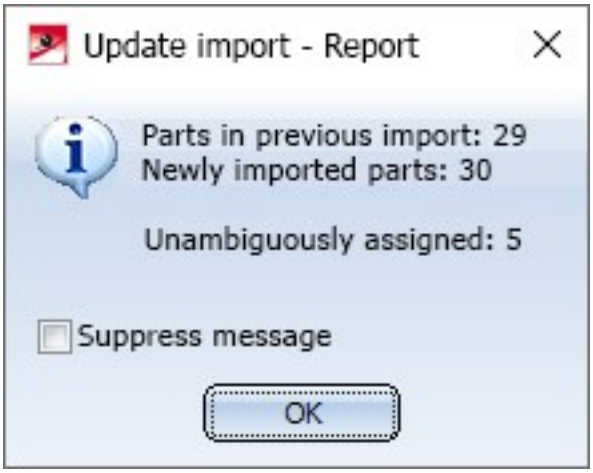
Update imported 3-D parts

The **Update import, via 3-D interface** function has been extended in SP1:

After the update, an evaluation of the update is displayed with the following information:

- parts in the previous import,
- newly imported parts and
- unambiguously assigned parts among them.

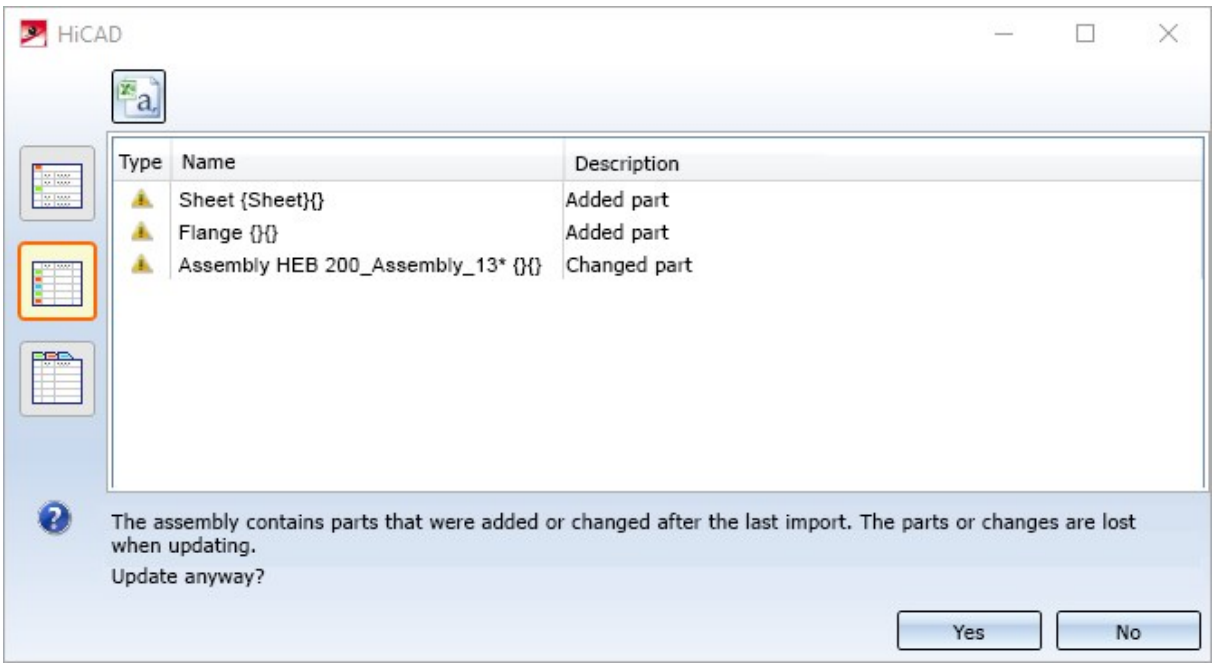
This dialogue is used to illustrate to what extent it was possible to transfer the attributes from the old parts to the new ones and whether there were changes in the number of parts.



In the following cases, a warning message is issued before the update:

- In HiCAD, parts of the old import have been changed or deleted.
- In HiCAD, new parts have been inserted into the assembly of the old import.

All corresponding parts are listed in the dialogue window. Only parts that are visible in the ICN are taken into account.



The icons on the left of the dialogue window can be used to vary the display. The log can also be exported as a CSV file via the icon at the top of the window.

If the dialogue window is closed with **OK**, the update is started. Thereby the parts and changes in the list will be lost!

Also new:

- View-specific properties will be preserved after the update.
- Set scales will be preserved after the update.
- Copies of imported parts can be updated like originals.
- Imported parts are marked in the ICN with the symbol. If you point to the symbol with the cursor, you will get further information about the corresponding part, e.g. the imported file format and the HiCAD version with which the part was imported. As of SP1, the name of the file (including the path) and - in the case of IFC files - the IFC version of the import are now also displayed.

BabCAD

The **BabCAD - DXB creation** function (**Drawing > Save/Print > Save as > Conversion**) is no longer available from SP1.

BabCAD files cannot be opened via the HiCAD Start Center.

Major Release 2021 (V 2600)

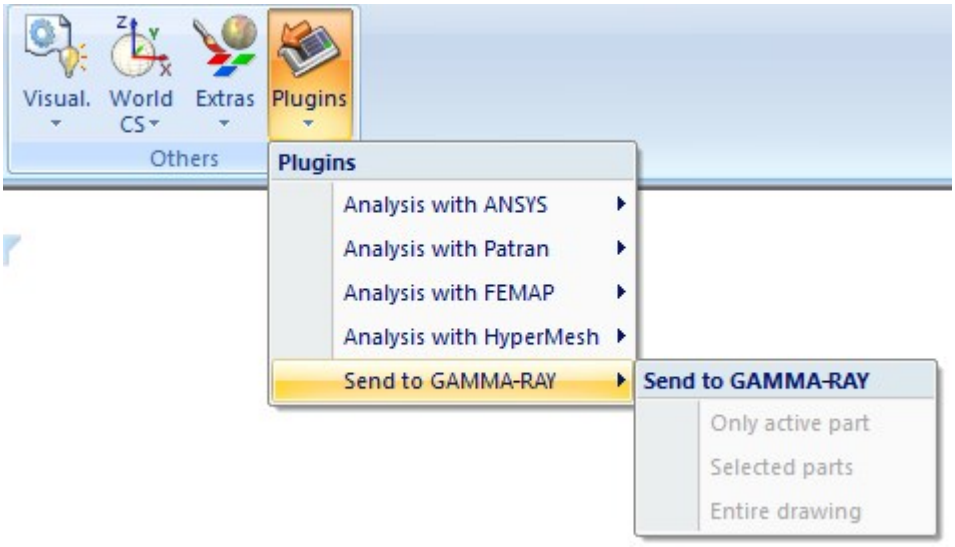
Update to CADfix 12 SP 1.1

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

- CATIA V5-6R2020
- NX 1899
- SOLIDWORKS 2020
- Inventor 2020
- 3-D DXF/DWG 2019
- Parasolid V32.1
- Creo Parametric 7

Direct call of GAMMA-RAY

If the rendering program **GAMMA-RAY** is installed on your computer, you can also export parts of your model drawing into the TGF format via the Drawing Ribbon bar and open the files automatically in GAMMA-RAY. The **Send to Gamma-Ray** menu is available under **Others > Plugins** > ....



With the functions of this menu you can

- the active part,
- all selected parts or
- all parts of the drawing

automatically as TGF file in GAMMA-RAY.

Normally, HiCAD finds the program GAMMA-RAY automatically. The path of the EXE file can be preset in the Configuration Editor at **Interfaces > GAMMA-RAY**.

NCX - Automatic DXF file generation

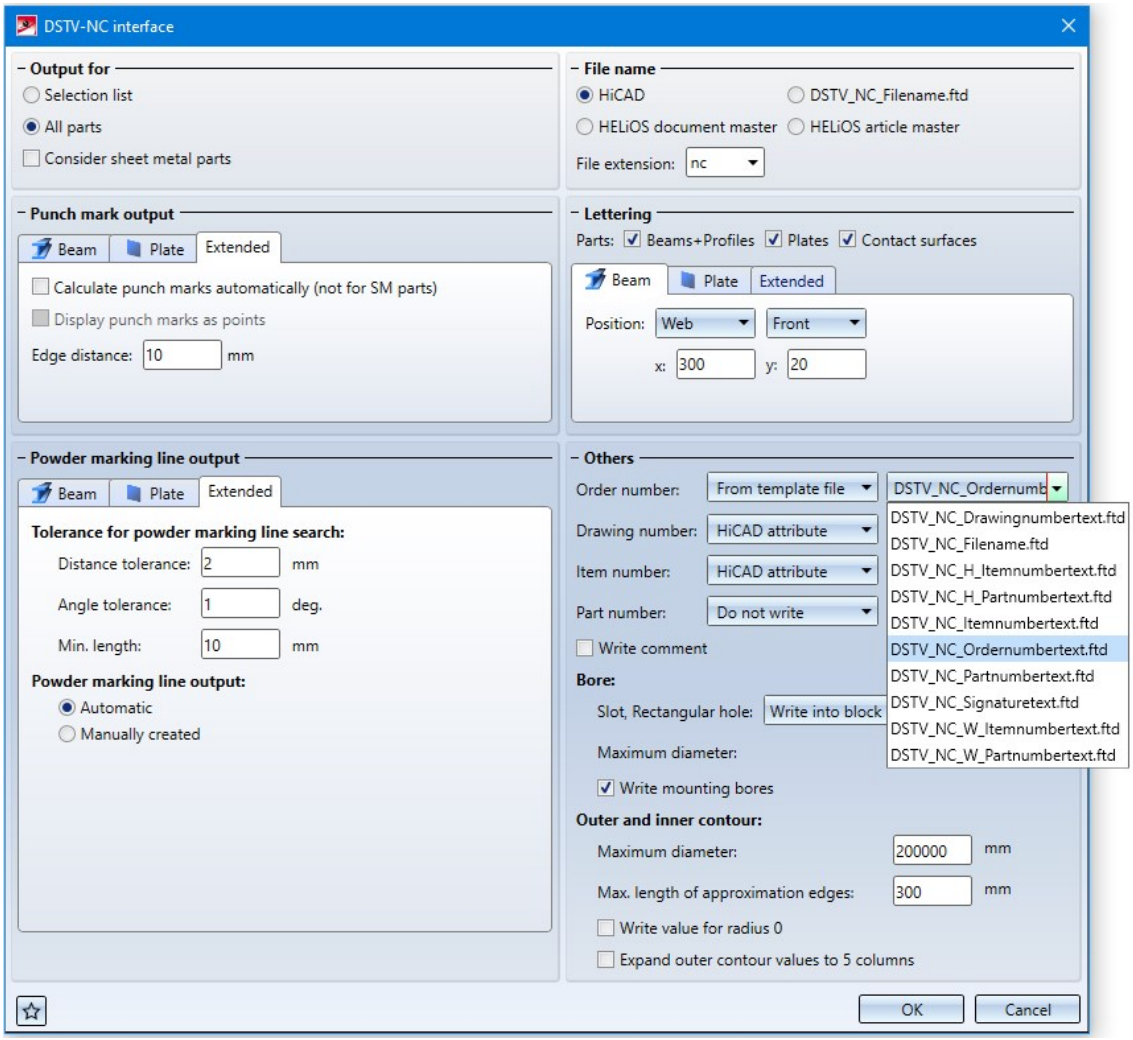
When saving model drawings in NCX format, the cross-sections of beams, profiles, plates and sheets can be automatically saved as DXF files. Until now, flat steels and steel plates were excluded from this, since in many cases the machine software automatically displays a rectangular profile if DXF data is missing. However, there is also machine software that displays a hollow profile in such cases. In order to ensure the correct display of the cross-section, HiCAD 2021 and higher now also creates a DXF file of the cross-section for flat steels and steel plates.

NavisWorks - Export via API

Export to NavisWorks is also possible via the HICAD API - both in NCX and NWD format. The format is selectable by the extension in the transferred file name. Please note that the export as NWD file is only possible if a licensed version of NavisWorks is available on the computer.

DSTV-NC Export - Template files

In the **DSTV-NC** export, all FTD files of the HiCAD sys directory whose name starts with DSTV_NC_ are now available for selection under **Others** for determining the order number and drawing number from a template file. These can also be your own templates that you have saved using the **3-D Dimensioning+Text > Text > LLine** > **Change font** function, e.g. DSTV_NC_Ordernumbertext1.FTD etc.



Export of sheet metal developments by views

As of HiCAD 2021, an export of sheet metal developments by views is now also possible. To do this, use the **3-D Export (STEP, 3D PDF...) by views** function in the context menu of a view. To activate the menu, right-click on the pink dotted view frame.

This is valid for STEP and all CADFIX formats.

Updating of 3-D imports in the current model drawing

3-D parts/assemblies that you insert into the current model drawing using the function **Drawing > Insert part > Exp.** > **3-D Import** will be marked in the ICN with the symbol.

With the new function

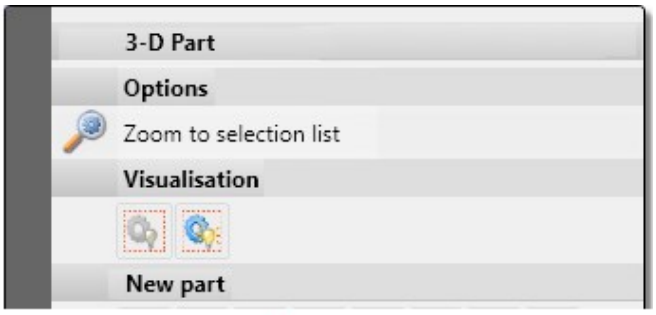
Update import

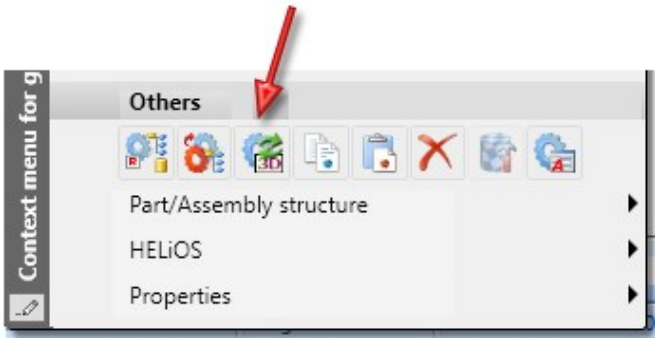
you can replace these parts/assemblies with other imported files. You can access the function by right-clicking on the marking, or via the context menu for parts.

The prerequisite for the update is that the top assembly of the first import still exists and the part on which the update is called is below this assembly or is itself this assembly. The assembly of the old import is completely exchanged for the newly imported one. This means in particular that parts that were added to the assembly after the import are lost during the update.

You can access the function

- by right-clicking on the marking of an imported part in the ICN or
- by right-clicking on an imported part in the ICN or in the drawing. In this case the context menu for 3D parts is activated. There you will find the function in the **Others** sub-menu.





Then, select the desired file for the replacement.

Calling the function is possible on every part marked with the symbol. After calling the function, select the replacement file. This file can also have a different name than the first import. After selecting the file, the dialogue window with the corresponding import options is displayed. The options you selected when you first imported the file to be updated are preset here. These can be changed if necessary. Click on **OK** to start the update.

Please note:

- Only parts/assemblies imported with HiCAD 2021 can be updated, as the corresponding import options are only available for these parts. The dialogue for updating is started with the import options of the first import of the file, the options for the new import can be changed in the dialogue if required.
- The visibility of unchanged parts is maintained during the update.

Further information is provided in the **Update Imported 3-D Parts** topic.

Sheet Metal

Service Pack 1 2021 (V 2601)

Connect sheets

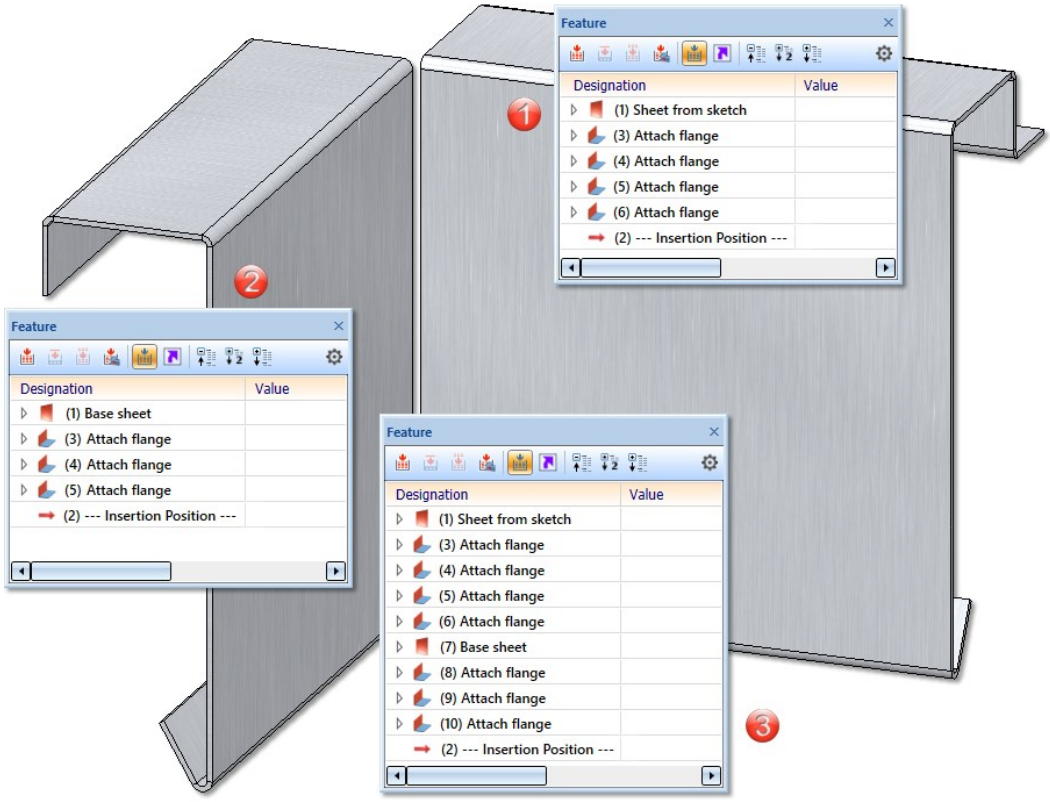
With the new function **Connect sheets** (**Further functions > Extras** > ...) you can connect bent sheets with an own feature log. The prerequisite is that the sheet thickness, the material or semi-finished product match and the base sheet was created with one of the following functions:

- Base sheet,
- Sheet along sketch,
- Sheet from sketch.

First activate a sheet and then call the function. Then select the sheets to be attached. Finish the selection by pressing the middle mouse button.

The feature log of the added sheets is hung under the log of the active sheet and the added sheets are deleted. The creation feature and the other feature steps such as **Attach flange** are retained.

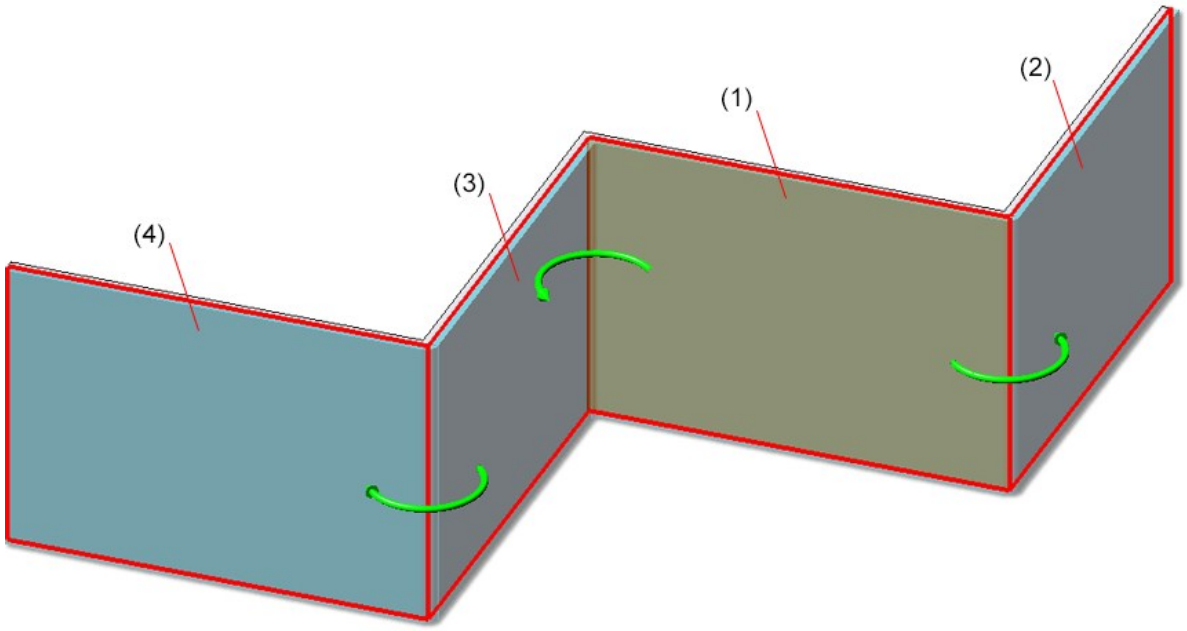
In the part structure, the flanges and bend zones are also attached to the active sheet. The added sheets are deleted from the part structure.



(1) Active sheet; (2) Added sheet; (3) Feature log after the connection. The feature of the second sheet is added.

Sheet from surface

The automatic search on one side has been added to the function **New sheet from surface** . This means that sheet-like parts that have been imported via 3-D interfaces, for example, can only be searched on one side of the surface..



(1) Start surface (base sheet) for automatic search on one side ; (2, 3, 4) connected sheets

Extended settings for sheet development

The tab **Milling+Folding - Extended** has been added to the **Extended settings** for sheet development. The following options have been moved from the **Milling+Folding** tab:

- Show milled area
- Adjust milling tool paths for production
- Lengthen milling lines

New is the option

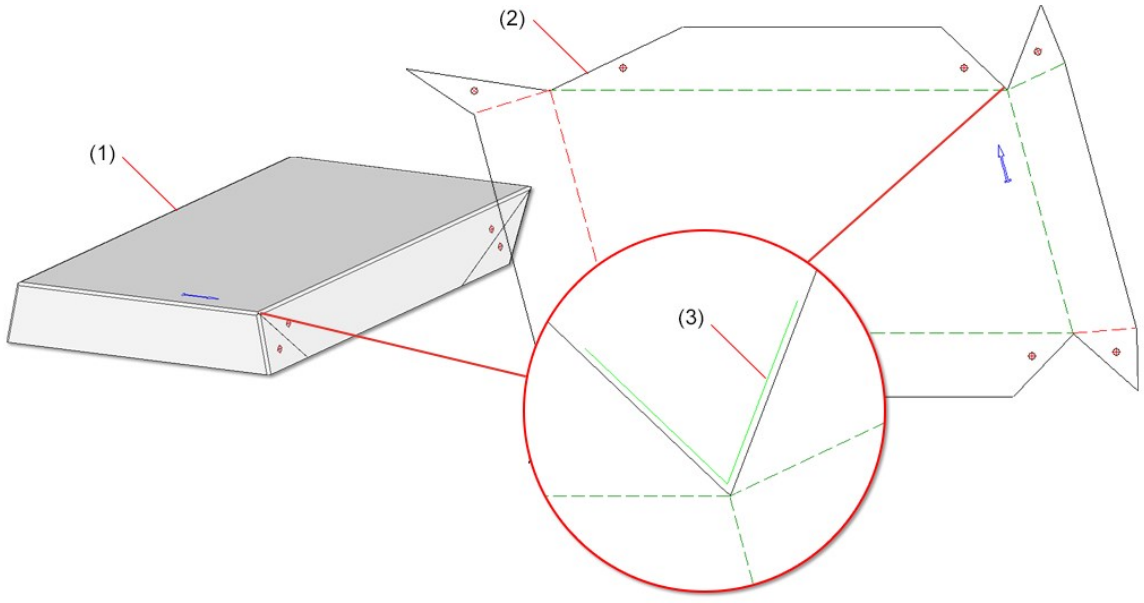
- Paths for acute inner corners

Path for acute inner corners

In acute corners, the outer contour cannot be milled exactly with the standard milling cutter. In practice, a cutter with a smaller diameter is used here. In the development calculation, paths for the treatment of acute corners can optionally be generated on a separate layer.

To display the path, check the option **Paths for acute inner corners**. Then enter the angle up to which the path is to be generated. You can also set the path length, layer, color and line type.

The path does not lie exactly on the outer contour (distance of 0.01 mm) to prevent it from disappearing during export.



(1) Sheet metal part, (2) Development, (3) Enlargement of the path for the acute inner corners with a path length of 0.3

Views of Sheet Metal parts

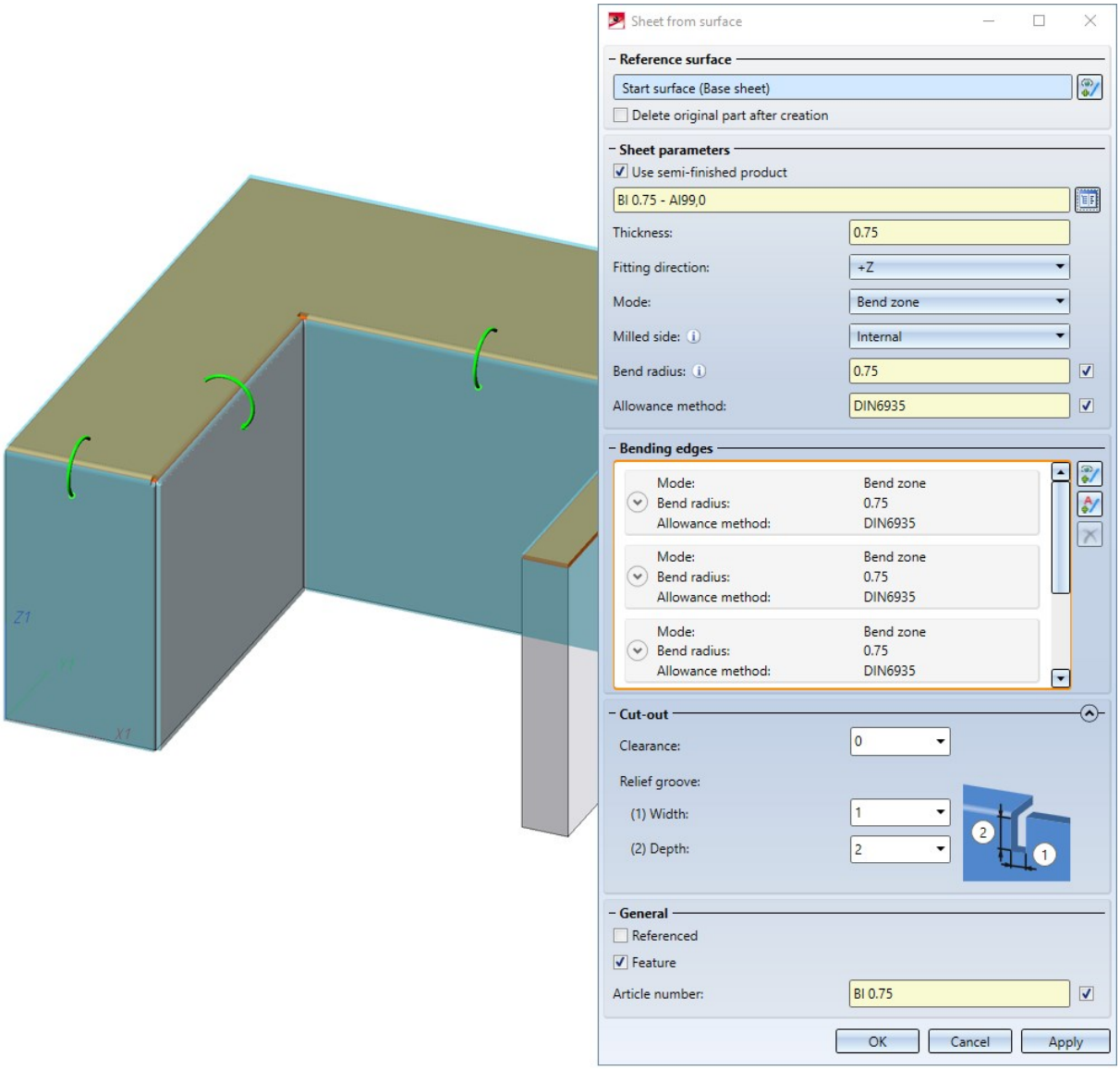
The view from the left has been deactivated in the default settings for Sheet Metal parts in the drawing derivation. This affects the setting at **Automatic drawing derivation > Production drawing > Usage-dependent > DEFAULT (KANTBLECHE) > View type assignment, Parts > View types: Sheets**.

This means that the view from the left is not automatically generated for Sheets Metal parts in the drawing derivation.

Major Release 2021 (V 2600)

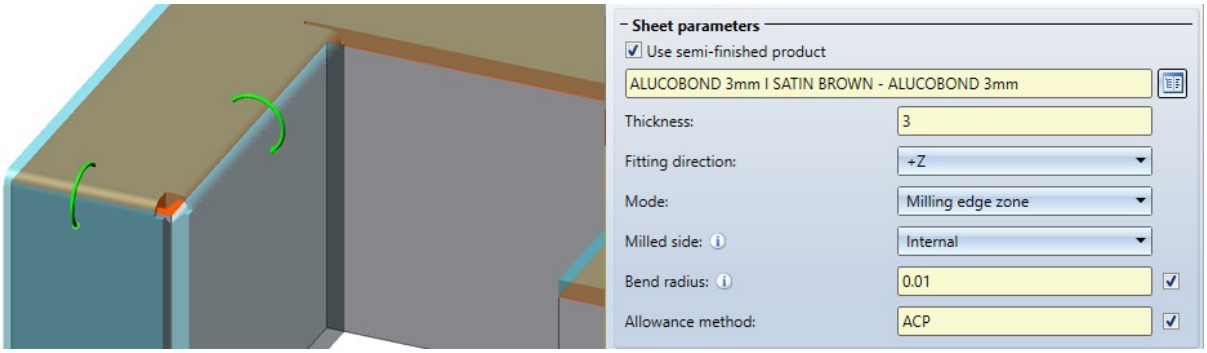
New sheet from surface

Due to the complete revision of the function **New sheet from surface** you can now construct a new sheet metal part with bending zones or milled edge zones from the surfaces of a 3-D part. After selecting the function, the following dialogue window appears.



1. Identify a planar surface of the 3-D part as the start surface. This surface is highlighted and becomes the base sheet.

If you want to identify another surface, select the icon for identifying the base surface in the **Reference surface** area.



2. Enter the sheet Thickness, Fitting direction and Mode (Bend zone or Milling edge zone) in the dialogue window.

If you choose a semi-finished product from the Catalogue Editor, the sheet thickness is not requested. The Thickness, the Material and, if applicable, the Article number are then taken from the Catalogue Editor for the sheet metal. In the **General** area you can also change the article number for the selected semi-finished product. If you want to take over the bending radius, which is only evaluated in the **Mode: Bend zone**, from the Catalogue Editor, activate the **From semi-finished product** checkbox next to the input field of the bend radius. In the **Bending edges** area, you can also change the bend radius individually afterwards for selected semi-finished products.

The **Fitting direction** determines the side to which the sheet thickness is applied. The surface of the 3-D part serves as a reference.

In the **Mode** field you can choose between Bend zone and Milling edge zone. The milled edge zone is used in practice for composite panels. The **Milled side**, which is only evaluated in the **Mode: Milling edge zone**, refers to the position of the 3-D part. The setting is adopted as a default for the complete sheet metal part and can be changed individually in the **Bending edge** area or via the feature for each milled edge zone.

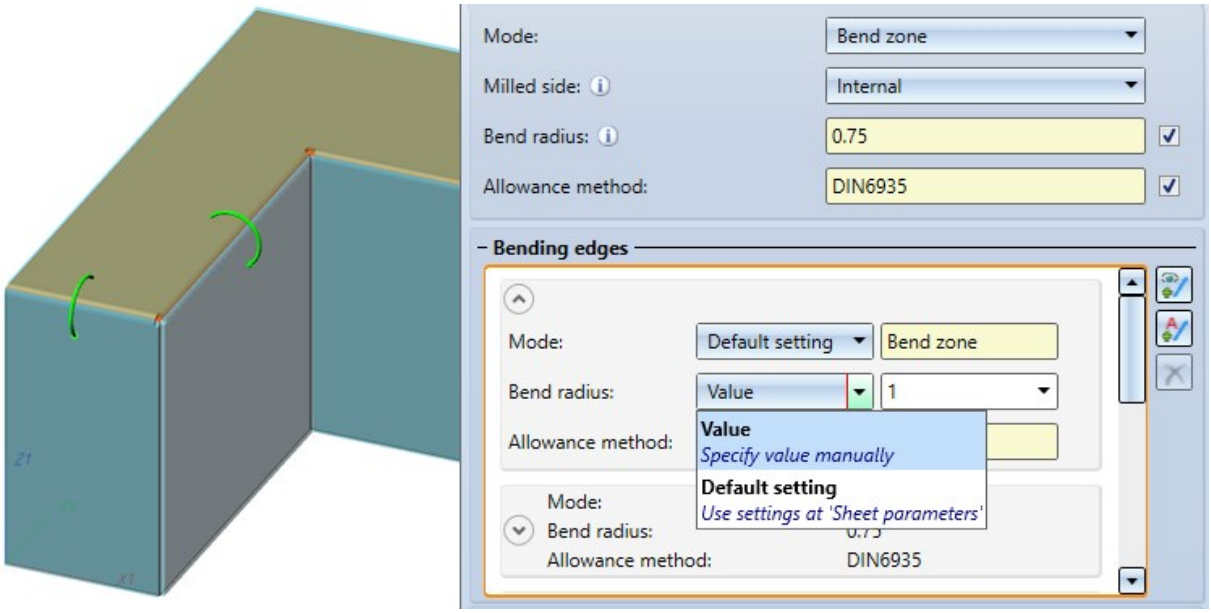
3. Select the bending edges .

This area is used to select the bending edges and for individual settings. Deviations from the default setting that you have set in the **Sheet parameters** area can be made here.

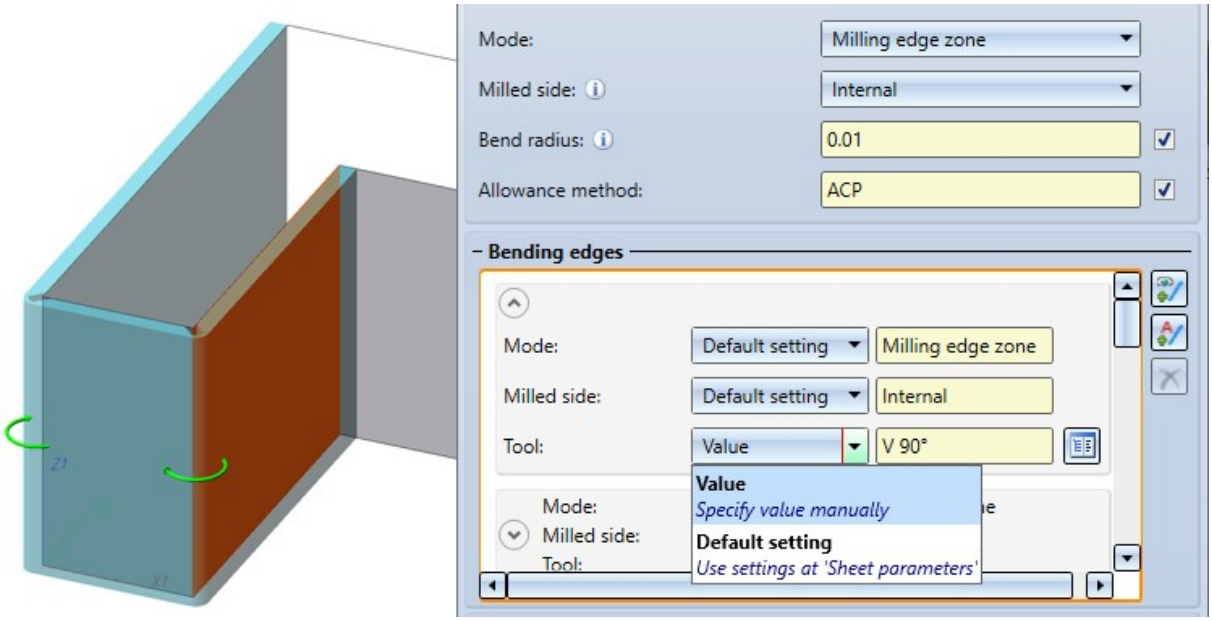
You can edit the selection of bending edges with the following icons:

- Use this icon to select edges or cylinder segments.
- With the AutoSearch option search, all possible bending edges are taken into account.
- The selected bending edge is removed from the list. You can select the edge in the dialogue window or in the drawing. Multiple selection is possible by selecting while holding down the CTRL key.

If you want to change e.g. the radius or the allowance method for a bend zone, expand the area with this icon .

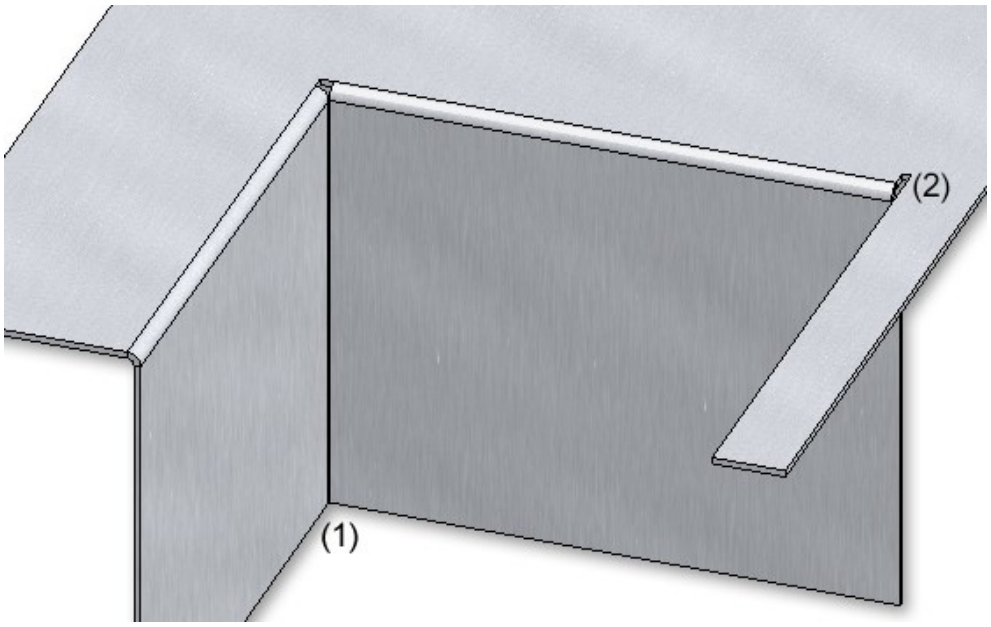


For the Bend zone the **Mode**, the **Bend radius** and the **Allowance method** become editable if you select **Value** instead of **Default setting**.



For the Milling edge zone, the **Mode**, the **Milled side** and the **Tool** (from the catalogue) become editable if you select **Value** instead of **Default setting**.

4. Enter the values for **Clearance** and **Relief groove**.



(1) Clearance, (2) Relief groove

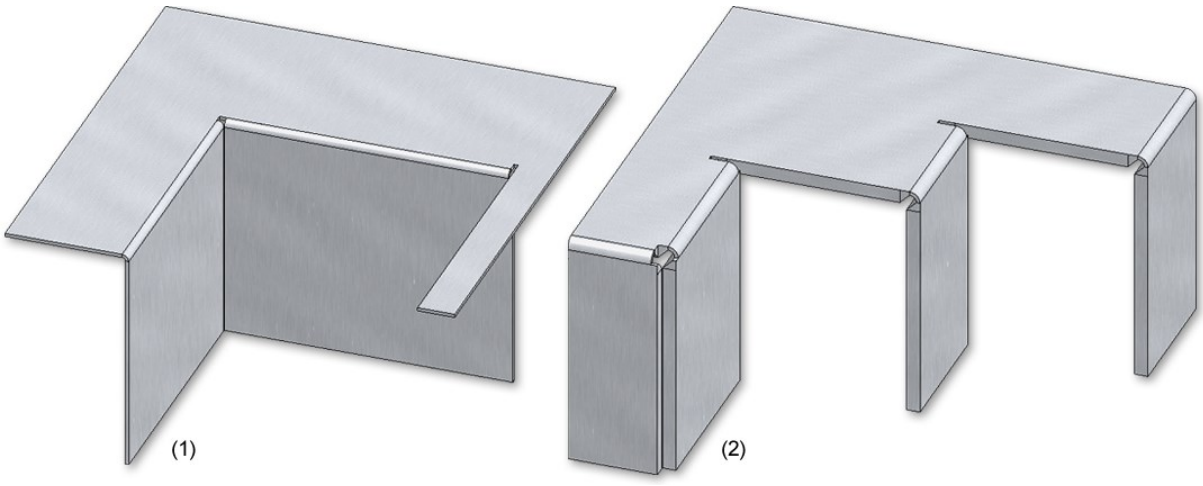
5. Exit the dialogue window with **OK**.

Once you have entered all the necessary data, the new sheet can be inserted. If you select **Apply** or press the middle mouse button (MMB), the sheet metal part will be inserted, but the dialogue window remains open - in contrast to **OK**.

Please note:

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

If the function cannot be executed with the entered data, this symbol . appears on the **OK** button. Here, too, move the cursor over the symbol to display the error message.



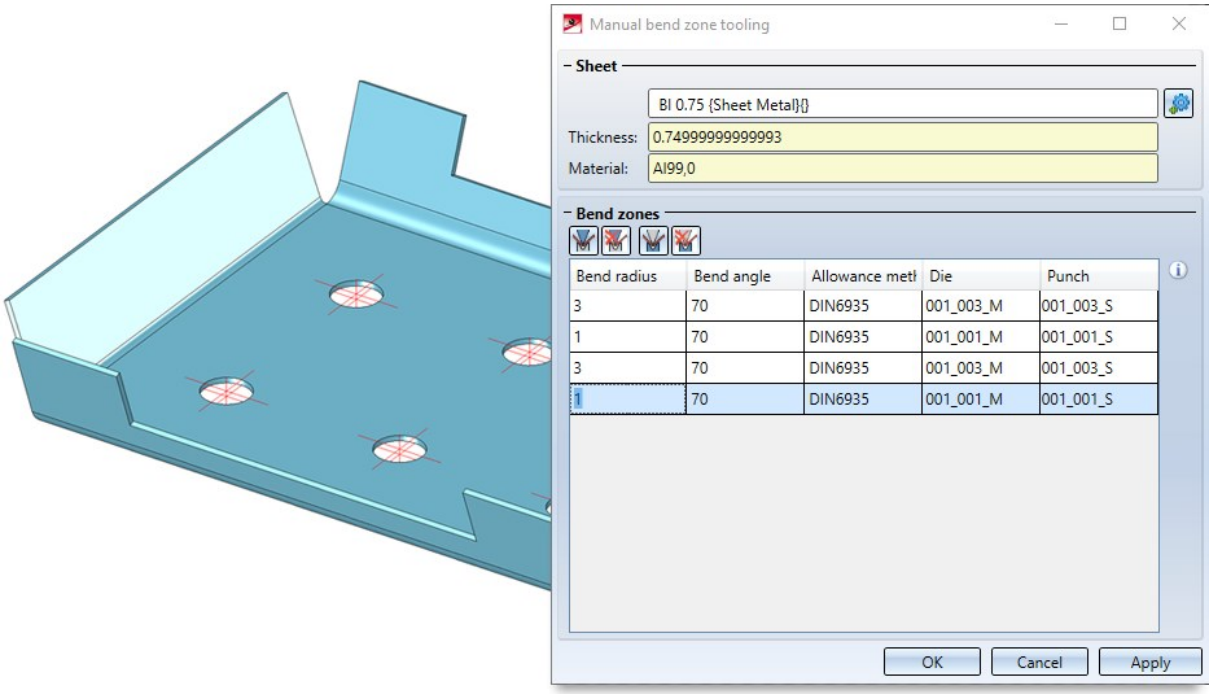
(1) Sheet Metal part with bend zone
(2) Sheet Metal part with milling edge zone

Bend zone tooling

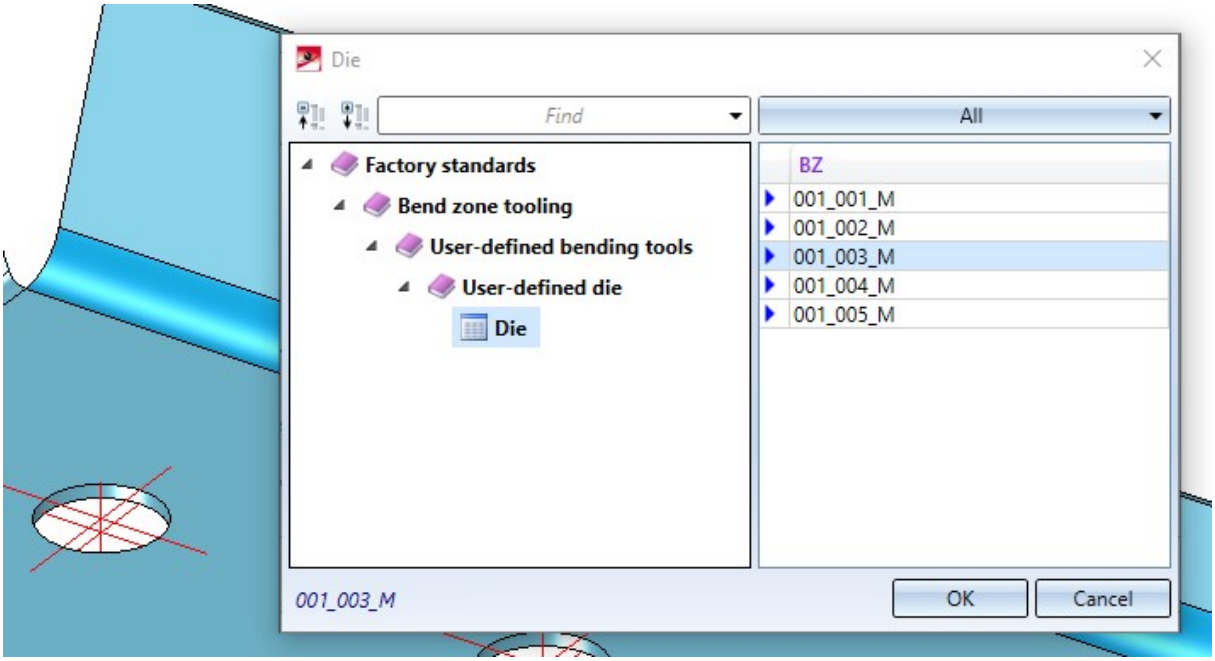
With the **Bend zone tooling** function you control the assignment of bending tools to bend zones. In the HiCAD Sheet Metal module, the angle, the bending radius and the allowance method are defined for bend zones. In practice, the bend radius depends, for example, on the material, the thickness and also the bending tools used. To take this into account, you must first enter the data of the tools in the Catalogue Editor. Then the function can be called and the data stored in the catalogue can be assigned. The data set in this way can then be output in the development.

Assigning the bending tools

After calling the function, the input mask appears. The bend zones of the active part are displayed. If you want to identify another sheet metal part, select the icon for identification in the **Sheet** area.



To assign the tools, activate the bend zone. If you hold down the CTRL key, you can activate several bend zones by clicking on them. Then select the icon for e.g. Die or Punch . The data stored in the catalogue are then offered for selection.

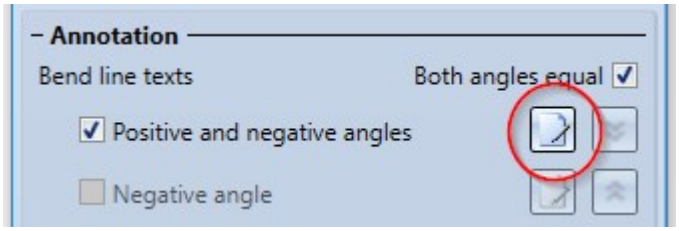


If you have entered all the necessary data, the assignment can be applied.

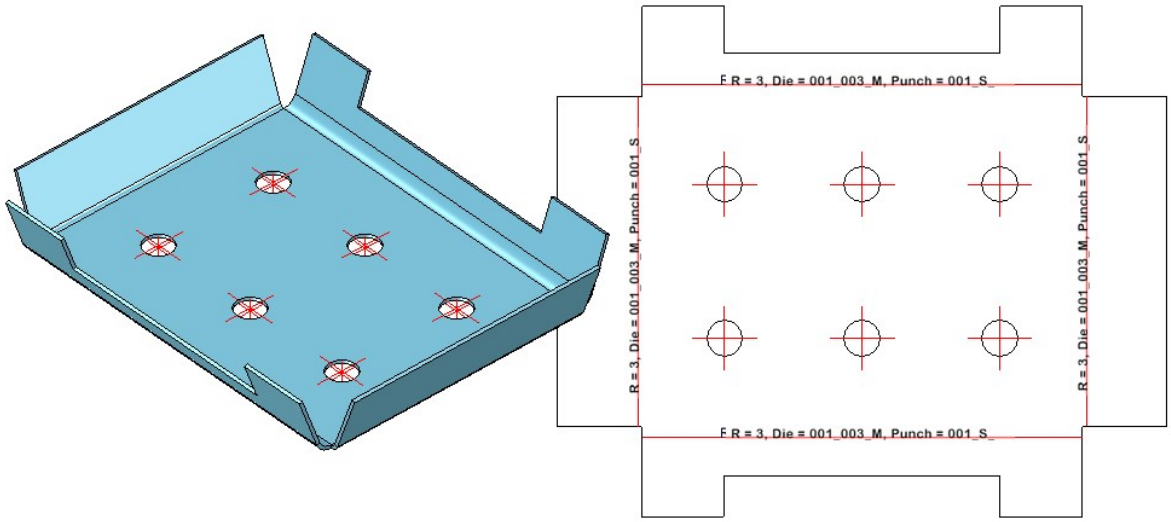
Creating developments with output of die and the punch

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

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

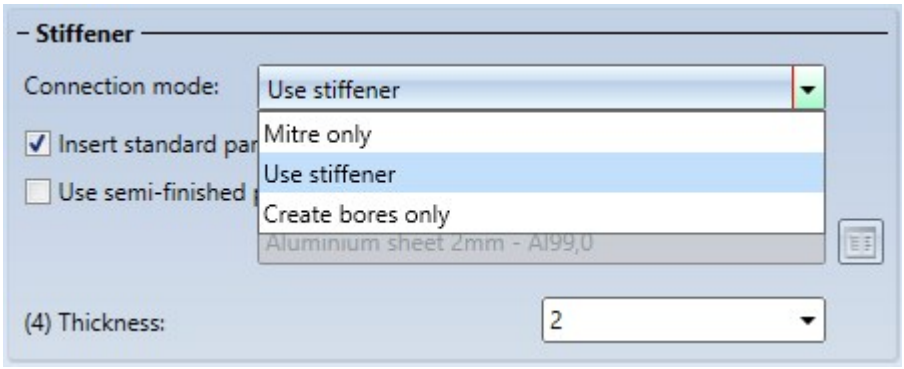


If a bend zone tooling exists for the bend zone, the names of these tools can now be inserted into the bend line text, via the attributes **Die** and **Punch** in the Text Editor.

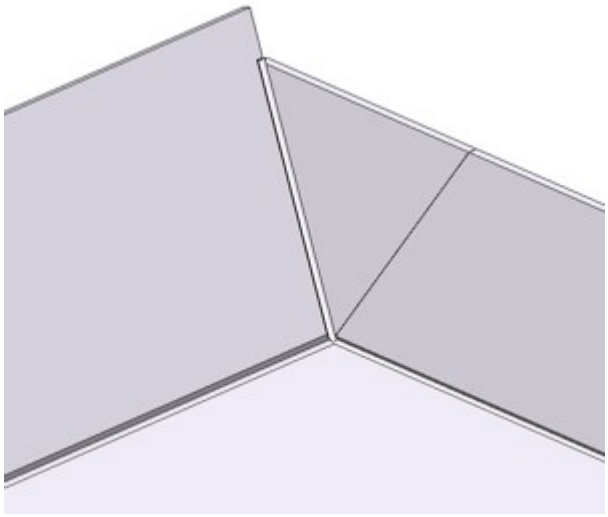


Design Variant: Sheet corner with stiffener

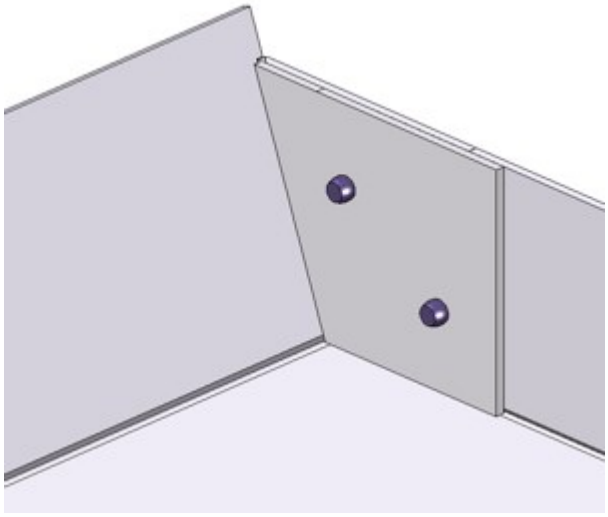
When installing the design variant **Sheet corner with stiffener**, you now have several options to choose from when using the stiffener.



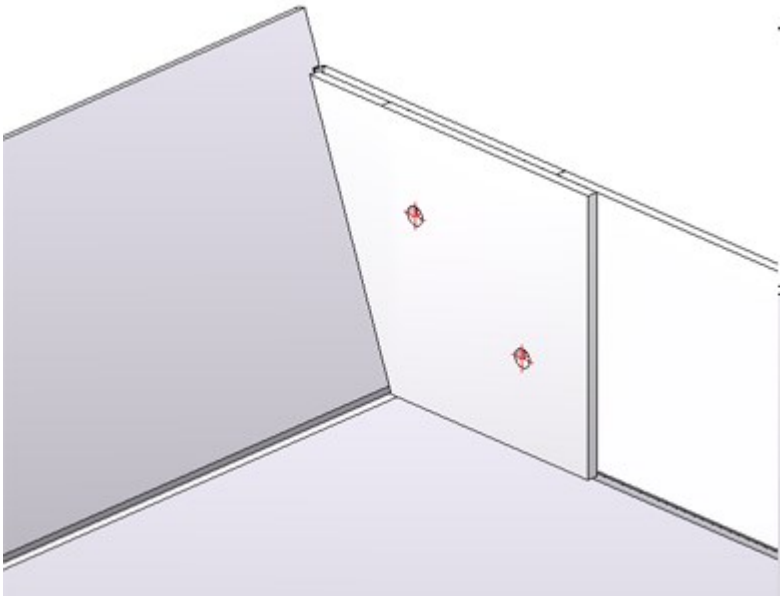
Mitre only



Use stiffener

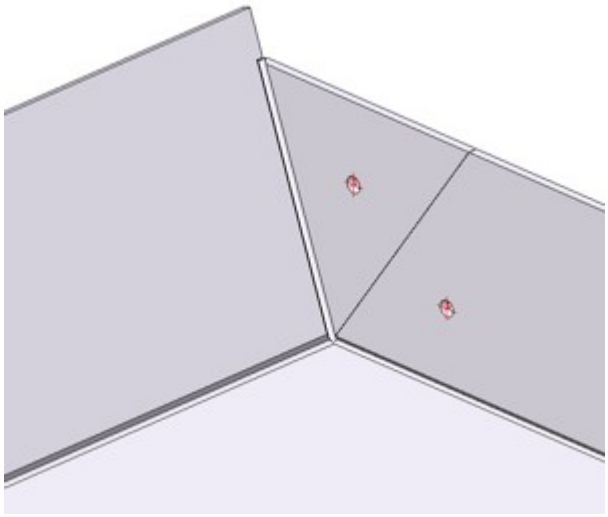


Use of stiffener with the option **Insert standard parts**



Use of stiffener without the option **Insert standard parts**

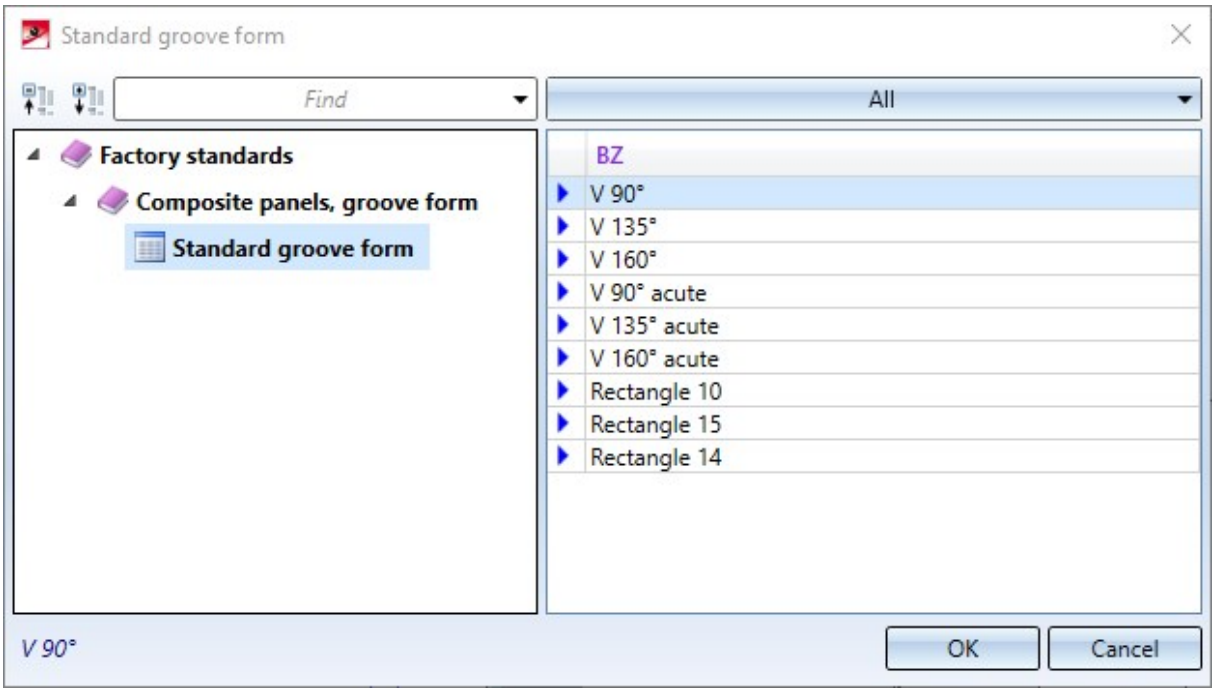
Create bores only



The ISD 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. The option is only active if you have selected **Use stiffener** in the connection mode.

Extension for Milling groove forms

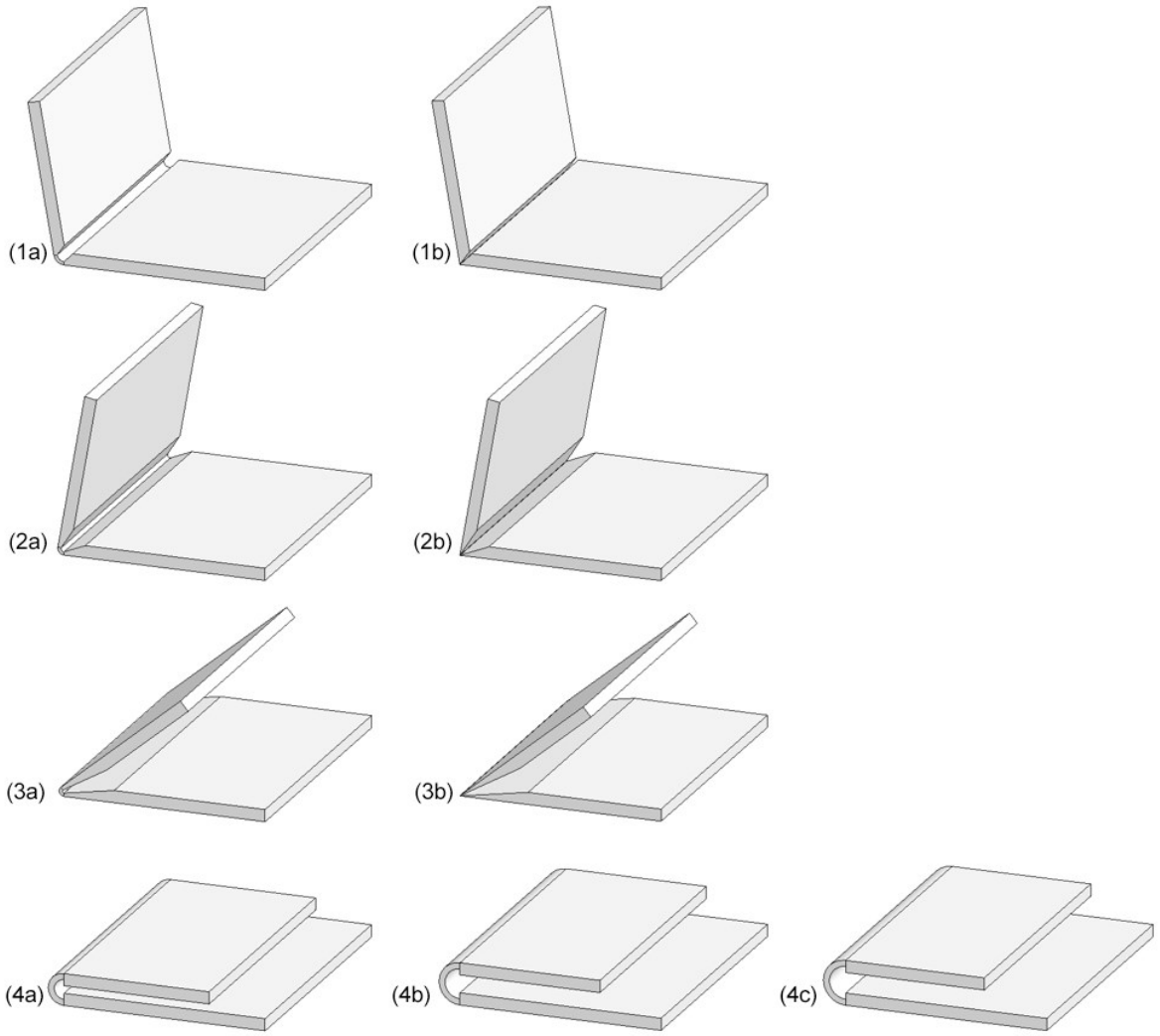
In the catalogue **Factory standards > Composite panels, groove form** the table **Standard groove form** has been extended. This table is used for attaching and folding of flanges



The new milling cutters for acute-angled attachments or foldings are intended for 2-D drawing. There, cassettes and trays are represented as simplified paper models. If you design in the paper model and want to use the groove shapes for acute-angled attachments or foldings automatically, you have to change the priority in the catalogue editor. E.g. groove form **V 90°** gets a **6** in column **SORT** and **V 90° acute** gets a **1**. If you have set the check mark for **Tool** when bending, a tool is automatically loaded depending on the selected bending angle and the assignment in the catalogue (column SORT).

In HiCAD, the milling cutters with the priority (SORT column) 1-3 and 10 are used by default.

Bend angle	**Groove form**	**SORT** (assignment to bend angle)
≤ 90°	V 90 °	1
91° bis 135°	V 135°	2
136° bis 160°	V 160°	3
161° bis 180°	Rectangle 10	10



(1a) V 90°, (1b) V 90° acute ,
(2a) V 135°, (2b) V 135° acute ,
(3a) V 165°, (3b) V165° acute ,
(4a) Rectangle 10, (4b) Rectangle 14, (4c) Rectangle 15

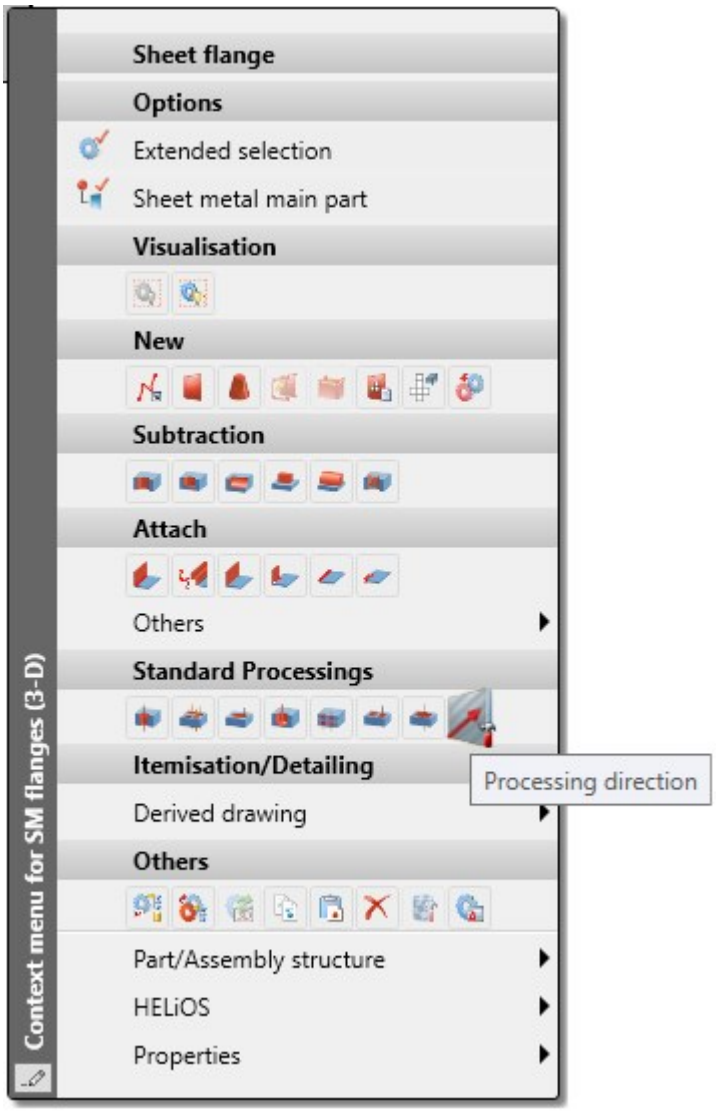
Optimized user guidance

Edit allowance method

The ABWEditor (Development Editor) from the EXE directory of your HiCAD installation is available for editing or creating the allowance methods for sheet metal developments. You can now also call the Editor via the **Sheet Metal** Ribbon tab, at **Sheet development > AllMet** > **ABWEditor** .

Processing direction

If you activate a sheet metal flange with the right mouse button, the function for creating a Processing direction now appears in the context menu.



Insert moulding tools

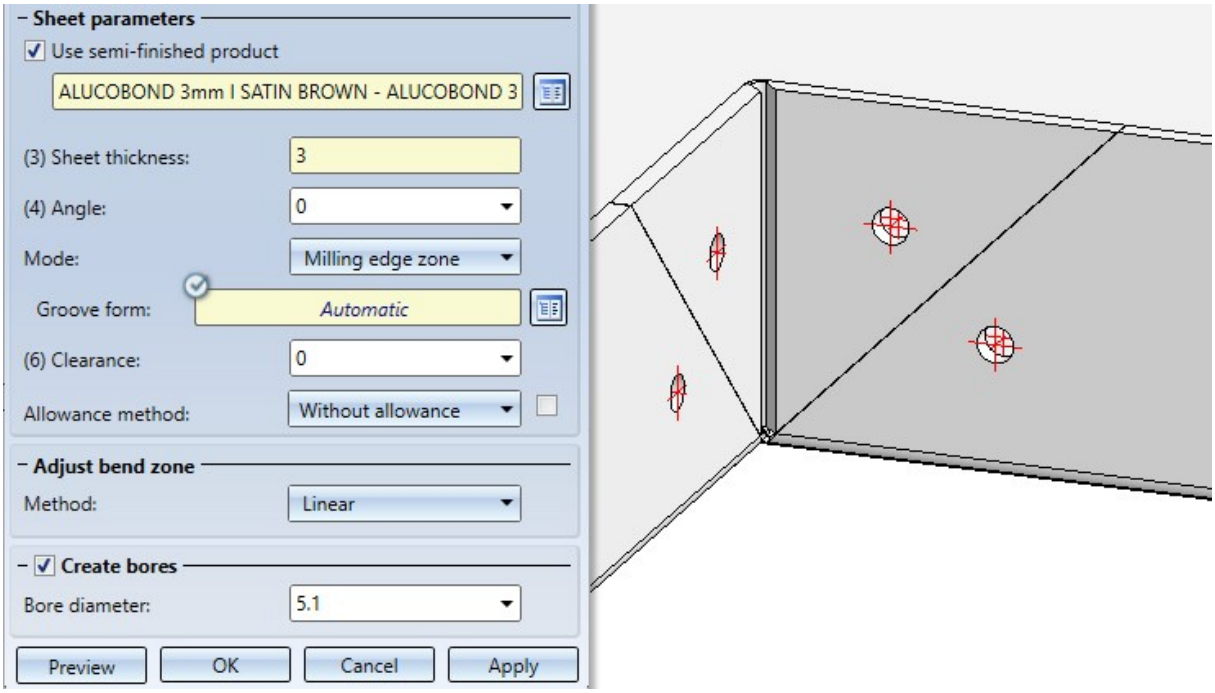
Purchased/factory standard parts and moulding tools are also placed in the model drawing via the Fitting CS of the corresponding KRA file or via three isolated and named points defined in the KRA file (Point 1 for the origin, Point 2 for the X-direction, Point 3 for the Y-direction). If neither these points nor a Fitting CS are defined, a corresponding message appears from HiCAD 2021 onwards.

Welded corner

Create bores

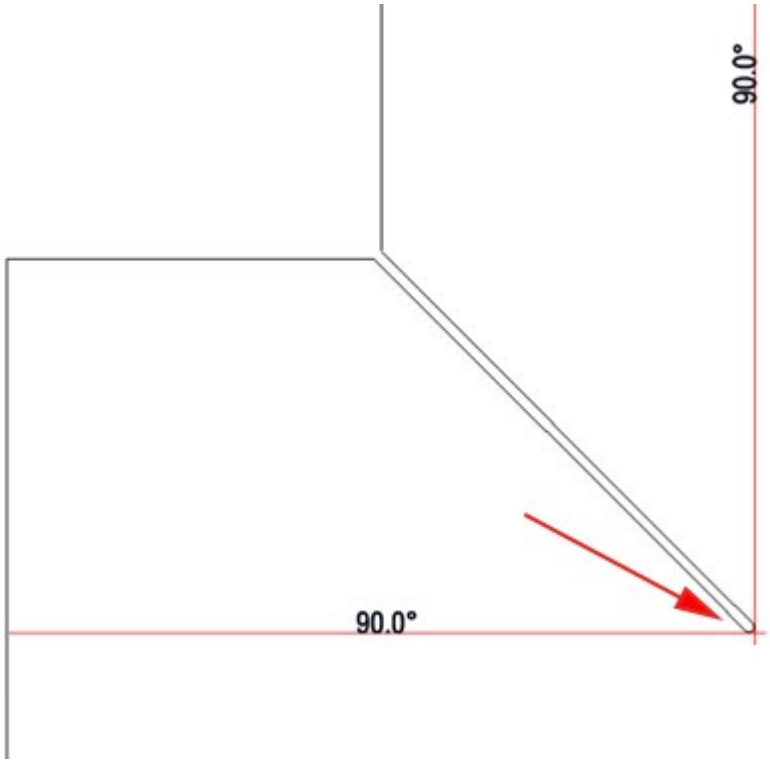
Since composite panels are usually bonded or riveted, you now have the option of inserting bores in the **Welded corner** function. The prerequisite is the use of a composite panel for the semi-finished product.

If you have selected a composite sheet for **Semi-finished products**, you can activate the option **Create bores** and select a bore diameter.



Process outer contours

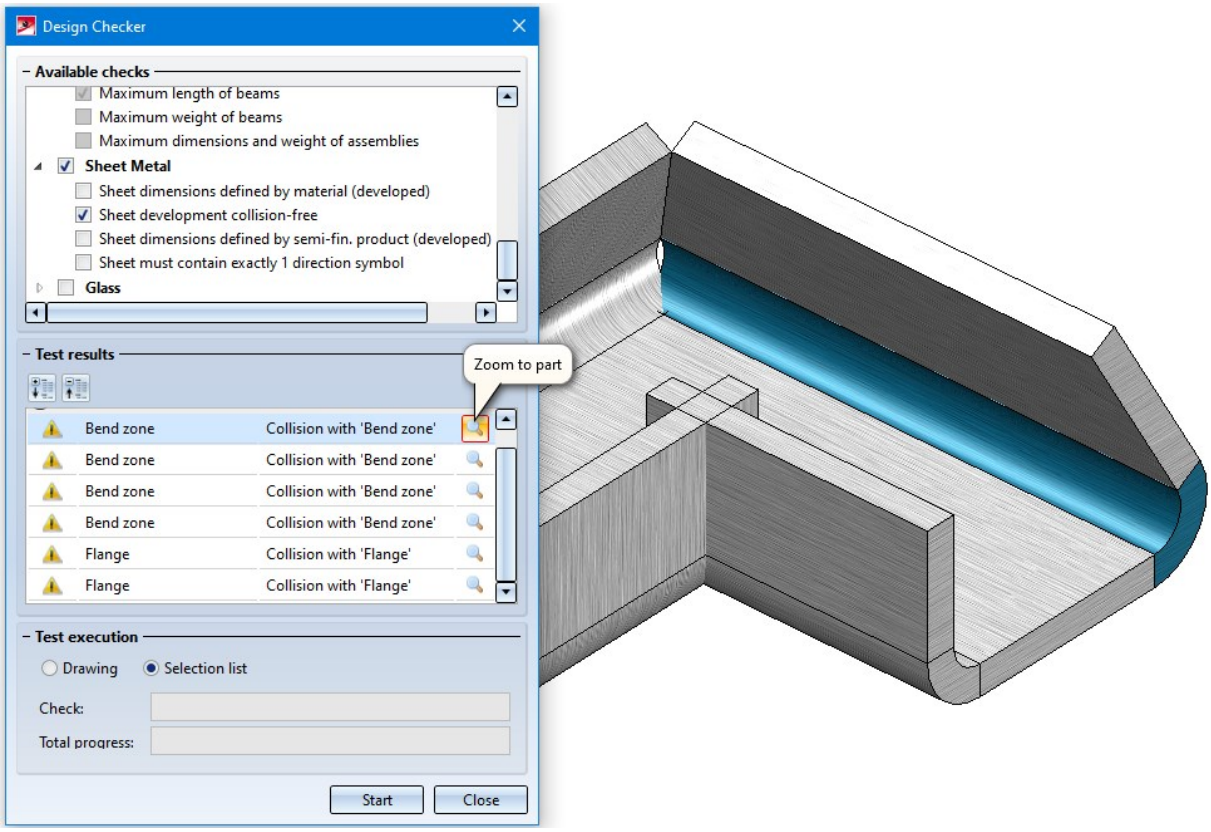
If you have activated **Milling edge zone** as a mode when installing the welded corner, the outer contour of the sheet in the area of the bonded / riveted edges is excluded for the milling cutter.



Development with subtraction for milling cutter

New test for sheet developments in the Design Checker

The **Design Checker** can be used to check sheet metal parts for collisions between flanges or bend zones during a development or bending simulation.



As of HiCAD 2021, it is now also possible to **export sheet metal developments by views**. To do this, use the function **3-D Export (STEP, 3D PDF...) by views** in the context menu of a view. To activate the menu, right-click on the view frame.

This applies to STEP and all CADFIX formats.

Steel Engineering

Service Pack 1 2021 (V 2601)

Connections

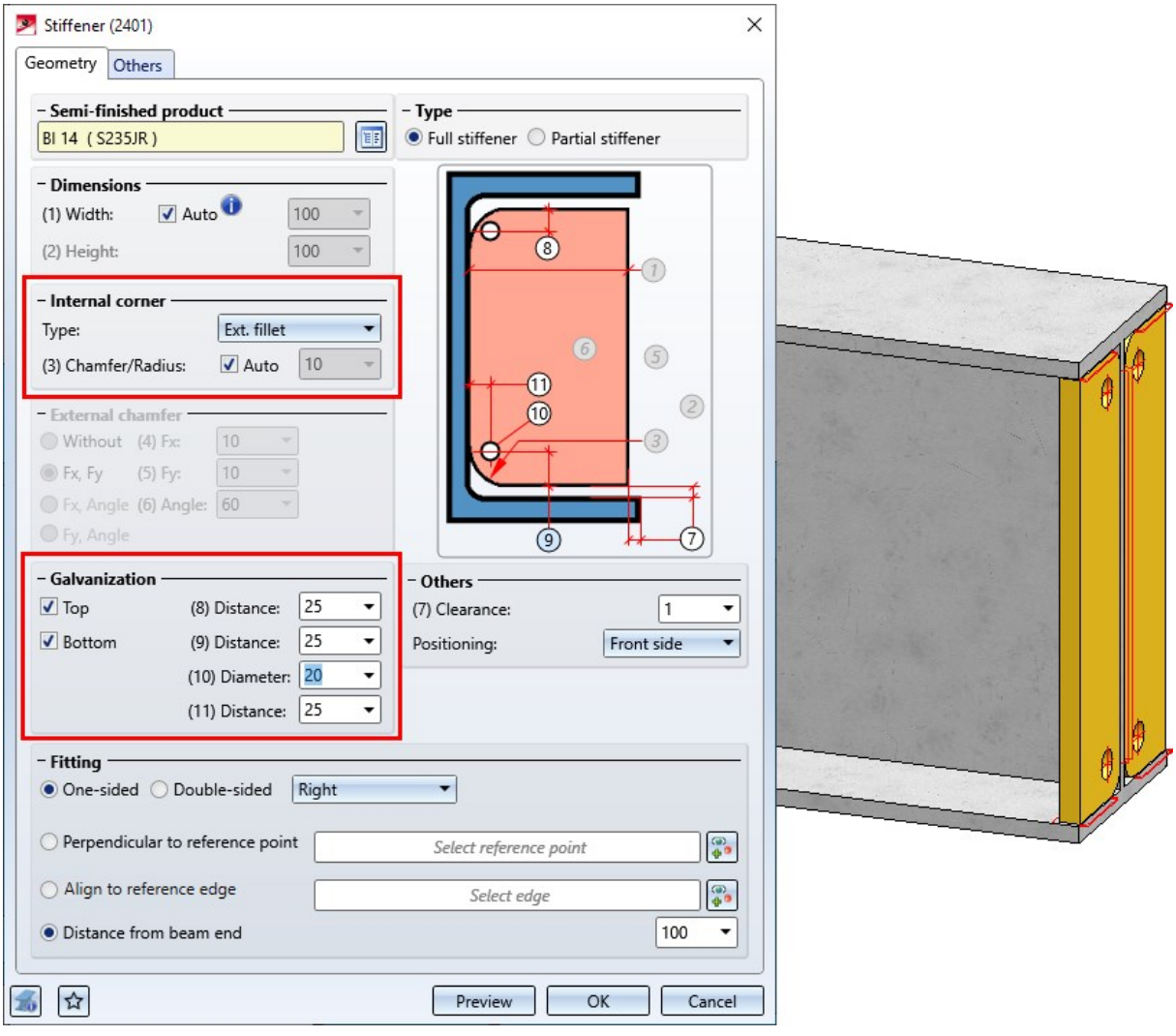
Stiffeners with holes for galvanisation

If, for the connections:

- Stiffener (2401),
- Beam to web, with 2 plates and stiffener (1211) and
- Front plate connection to flange (2330)

Ext. fillet is selected in the **Type** field under **Internal corner** when creating the stiffeners, then you can determine from SP1 whether the stiffeners are to be provided with holes for galvanisation.

By activating the checkboxes, determine which holes - top and/or bottom - are to be created and enter the hole diameter as well as the minimum distance to the web and flange.



Edit connection / Design Variant

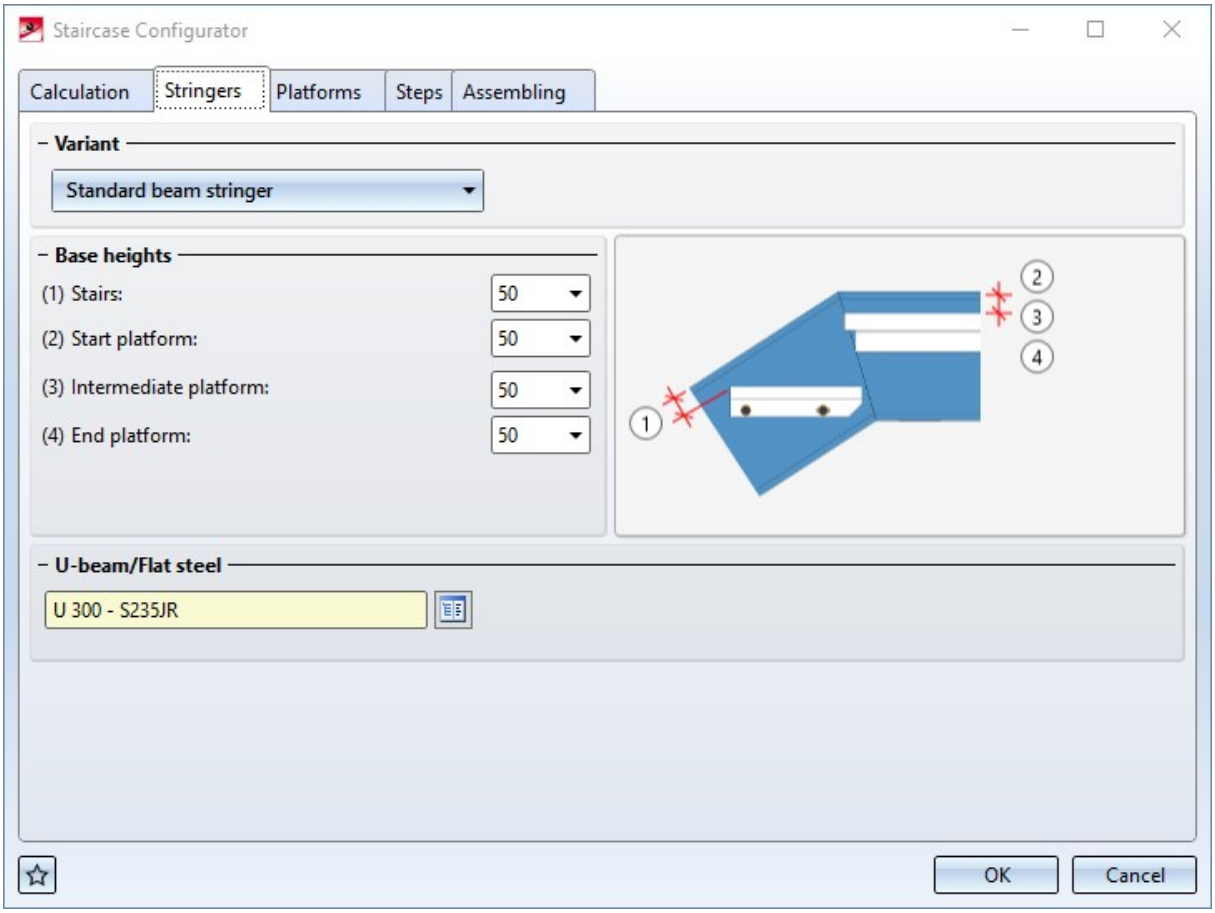
From SP 1 onwards, the **Edit connection/Design Variant** function remains active after changing a connection or a Design Variant. This means you can directly edit another connection without having to call up the function again. Simply identify an edge of the connection or Design Variant.

To end the function, press the middle mouse button.

Stairs + Railings

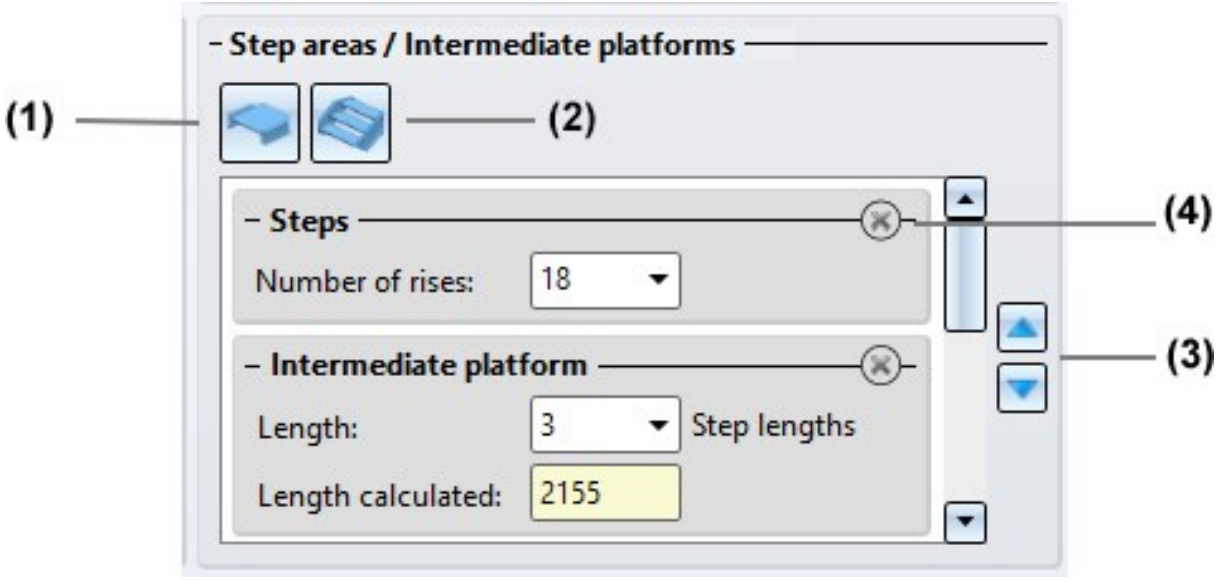
Staircase Configurator - Stringers from elongated plates

From SP1 onwards the Staircase Configurator offers elongated plates as stringers for staircases.



Staircase Configurator - Steps / Intermediate platforms

In addition to platforms at the stair start and stair end, several intermediate platforms are now possible. It should be noted that intermediate platforms must be added depending on the DIN standard. For example, according to DIN 18065, a maximum of 18 steps without intermediate platforms is possible for building staircases. This means that the staircase may have to be divided into several staircase segments, whereby an intermediate platform must be installed between two staircase segments.

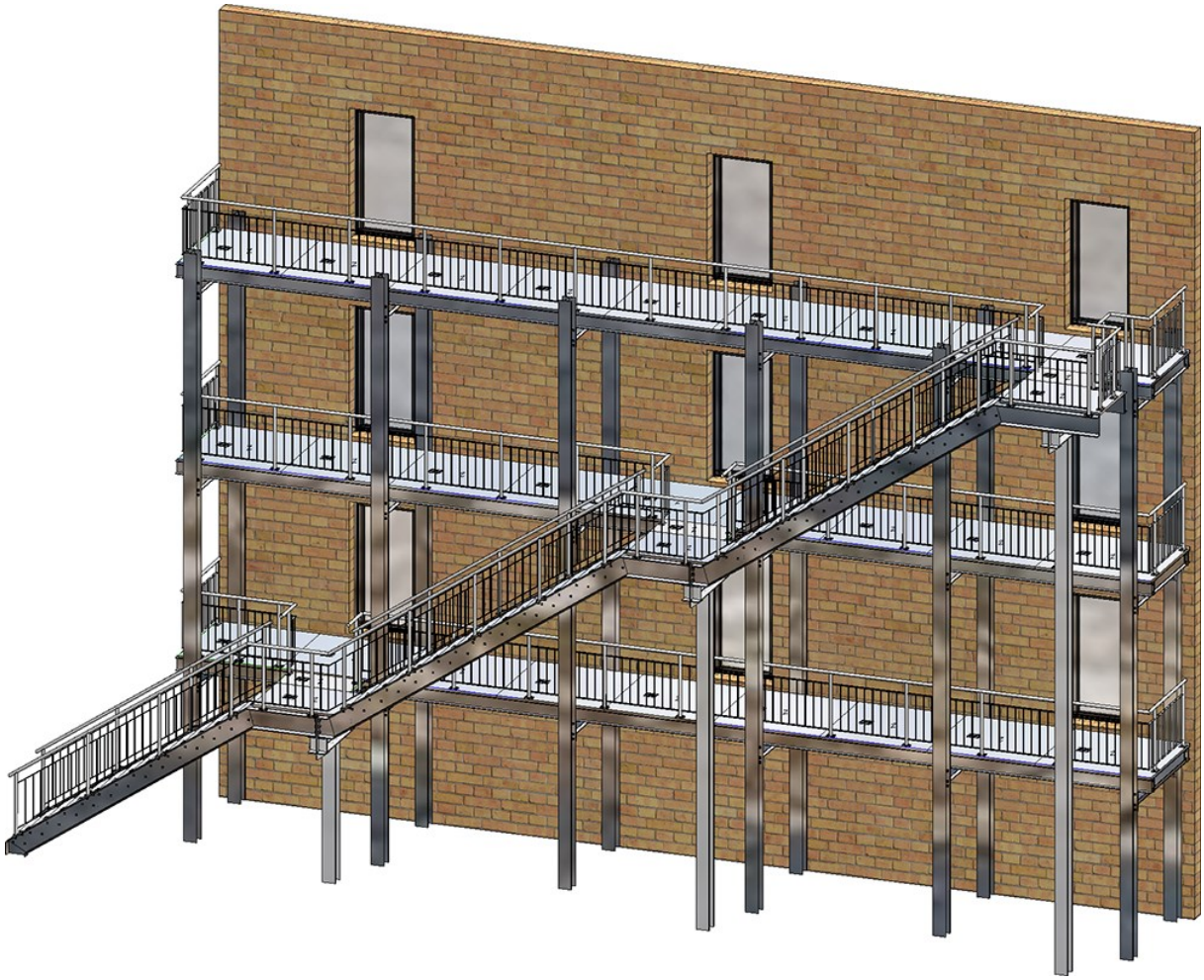


(1) Click on to insert an intermediate platform. Enter the length of the platform in step lengths. Note that the last element in this area must be a stair segment and that several platforms must not follow each other.

(2) Click on to insert a stair segment (steps). Enter the number of rises. Note that there must always be a platform between two stair segments.

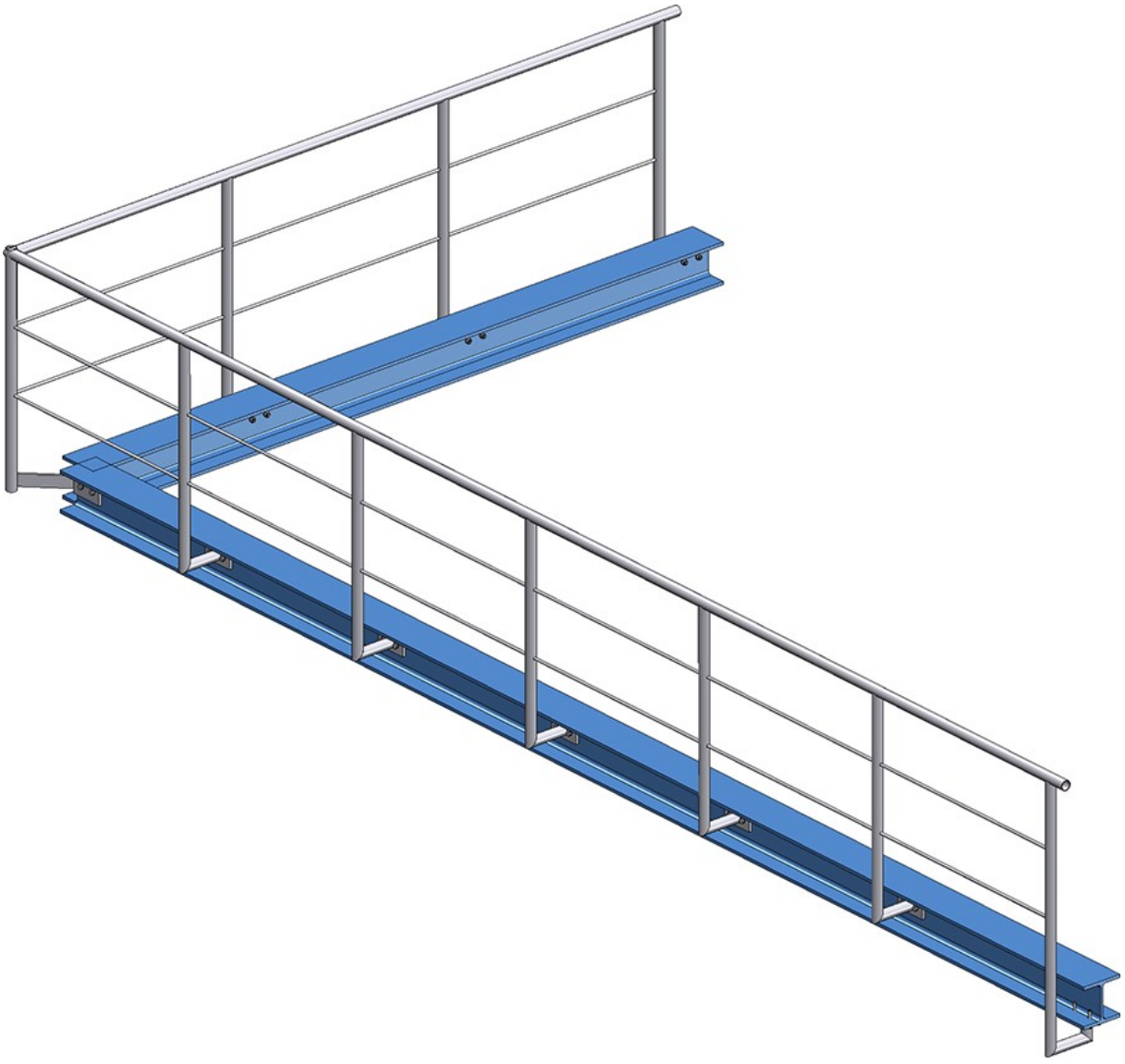
(3) Click on the delete symbol to remove a stair segment or an intermediate platform.

(4) Use the arrow symbols to change the order of the stair segments and intermediate platforms.



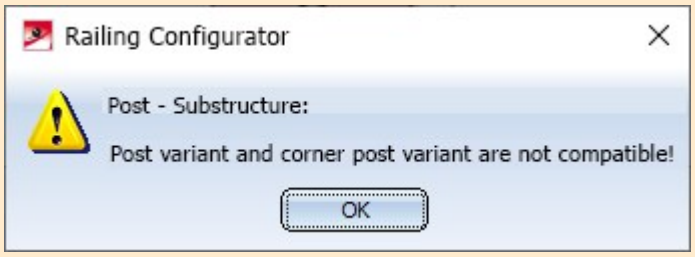
Railing Configurator - Posts - Substructure

- The post connection for the substructure can now be determined separately for
  - start, intermediate and end posts and
  - corner posts.
- Start and end posts can also be determined individually, i.e. independently of the intermediate posts. To do this, you must activate the corresponding checkbox at the top of the dialogue window.



Railing with lateral connection of the start, corner and intermediate posts and connection from bottom for the end post

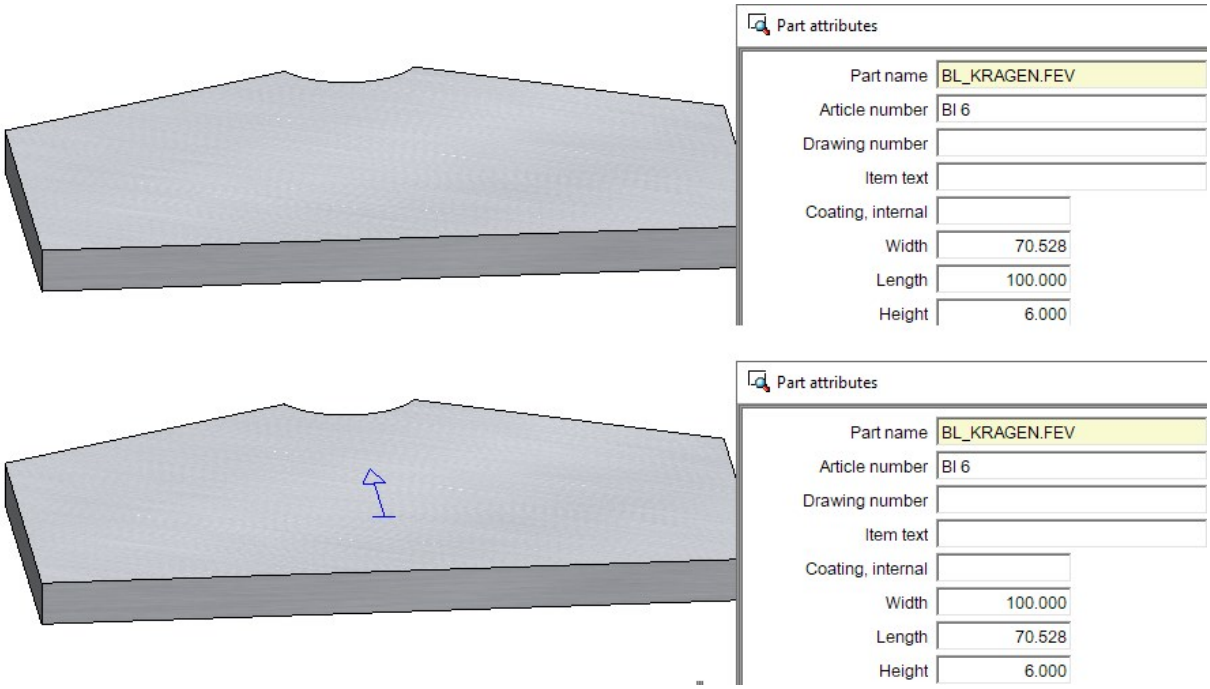
Please note the following when selecting the post connections: It is only possible to combine the variants **Post connection, lateral** and **Post connection, bottom**. If, for example, you have selected the variant **Post connection, lateral** for a corner or intermediate post and the variant **Post connection, top** for the start or end post, then the installation is not possible and a corresponding error message appears, e.g.



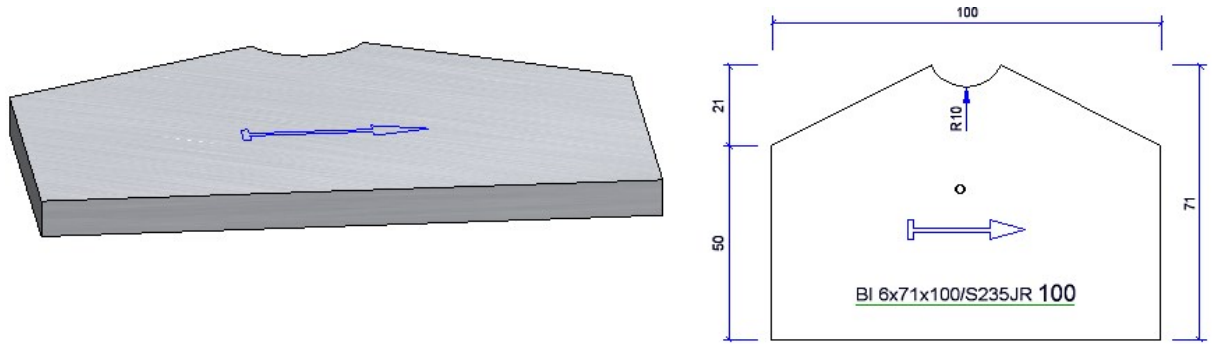
Beams + Plates

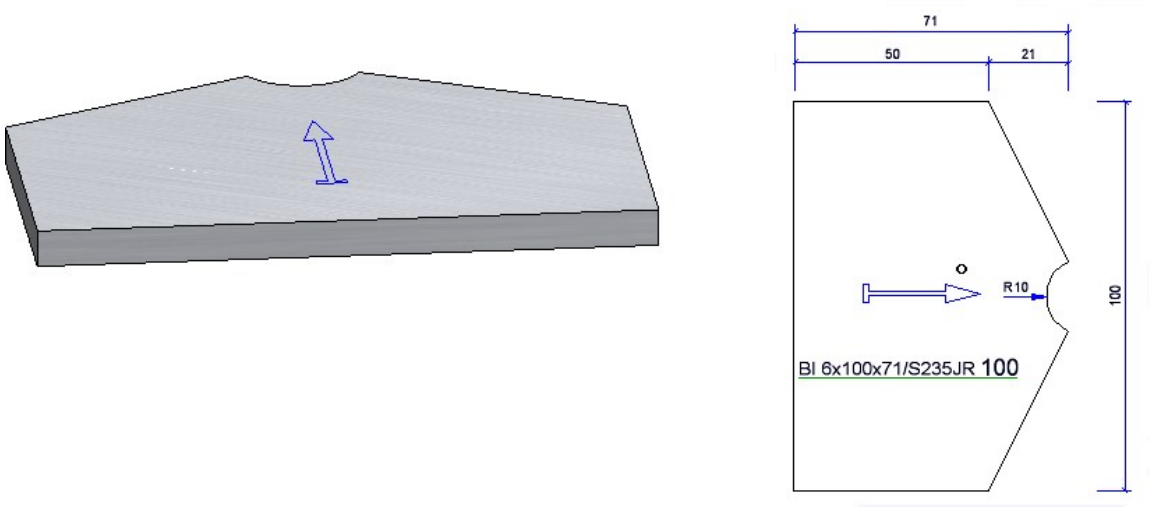
Steel Engineering plates with processing direction

As with Sheet Metal parts, the calculation of the length and width of Steel Engineering beams now also takes into account whether a processing direction is assigned to the plate.



The processing direction also affects the position of the plate in the workshop drawing.





Prototype beams from catalogue

When installing prototype beams, an article master assigned to the beam in the catalogue editor was not taken over until now, since the dimensions can be changed individually during installation. For this reason, two new functions are available as of SP1:

Prototype beams from catalogue

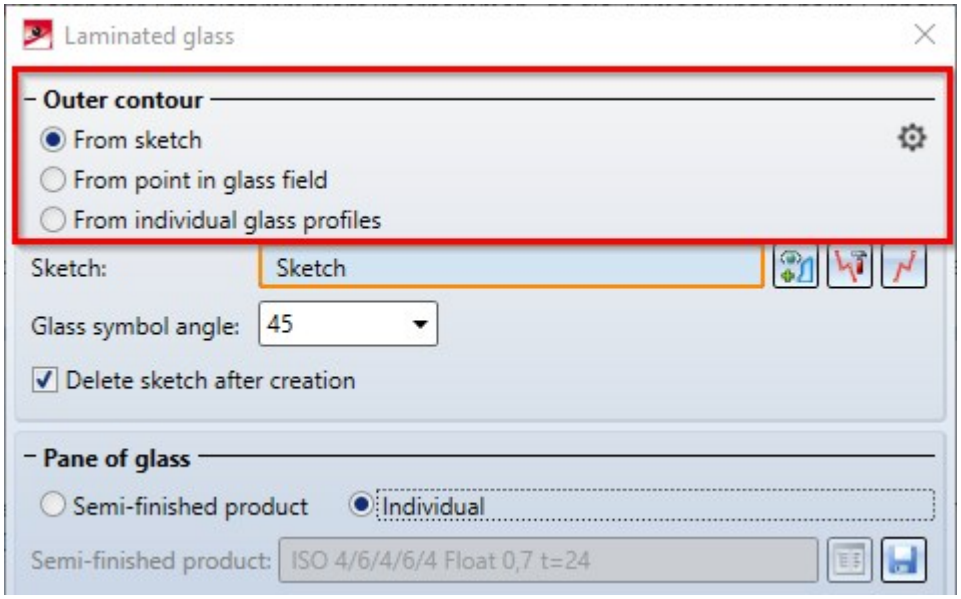
Sub-part, Prototype from catalogue

In contrast to the previous functions for prototype beams, here prototype beams from the catalogue are installed, whose dimensions (except for the total length) cannot be changed in the dialogue. If assigned, the article master is also taken over.

Laminated glass

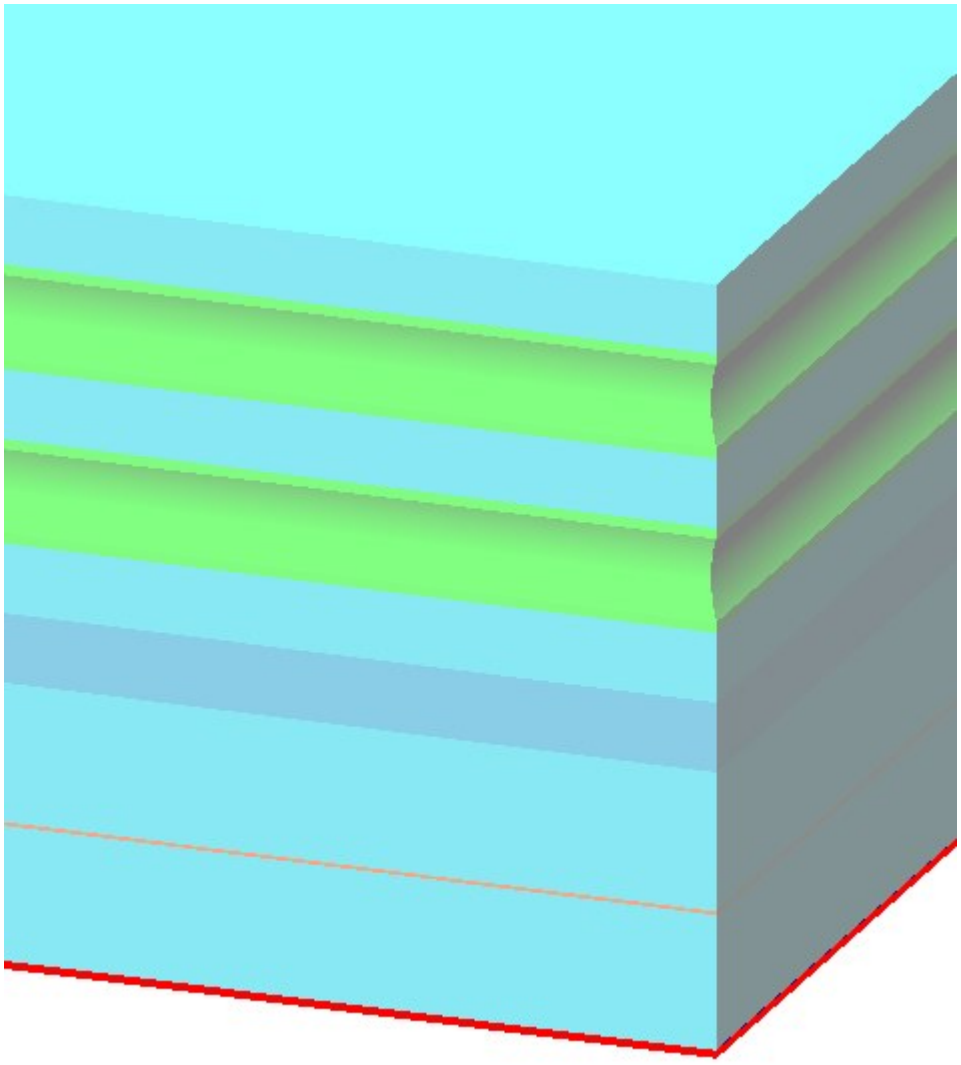
Options for selecting the outer contour

In the **Laminated glass** dialogue window, the three options **From sketch**, **From point in glass field** and **From individual glass profiles** are now directly available for selecting the outer contour. The functions behind these points were already available before, but some of them could only be accessed via the context menu. Now they can be selected directly in the dialogue window:



Unified colours in the preview

The colours in which the individual layers of a laminated glass are displayed in the preview view in the drawing have been unified. Glass panes are now always displayed in blue. If several glass panes follow one another directly, a slightly different shade of blue is used alternately. Foils and spacers are displayed in green. If a layer is selected in the Laminated glass dialogue window, it is highlighted in dark blue.



Others

Shipping list, short, with image

The Excel template **HiCAD_Stahlbau** has been extended by another worksheet: **Shipping list, short, with image**. This shows the same information as **Shipping list, short**, but additionally contains a picture of the parts.

see also Report Manager - What's new?

Part attributes - Delete usage

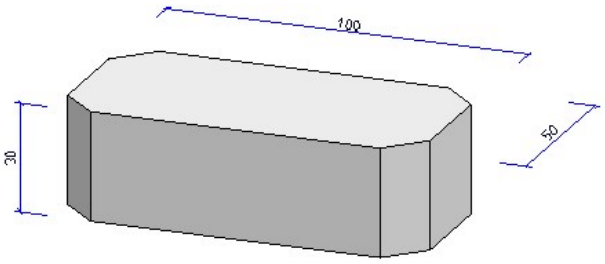
In the Part attributes masks, fields with a light yellow background are blocked for manual entries. This applies, for example, to the usage type. Here you can only select a usage type from the catalogue. As of HiCAD 2021 SP1, the content of several of these fields can now be deleted with a click on the button.

Steel Engineering plates - Attribute: Designation 2

In the case of steel engineering plates, the width was previously output before the length in the **Designation 2** attribute. From HiCAD 2021 SP1 this behaviour has been changed. In the same way as for Sheet Metal parts, the length is now output before the width, i.e.

Previously: {total number}x {part number}x{width}x{length} {material designation}

From SP1: {total number}x {part number}x**{length}x{width}** {material designation}



Properties

Designation	Value
Part name	BL_RECHTECK_F_WERT.FEV
Geometric type	Solid
Colour	Oyster White [40]
Layer	1
Material	S235JR
Module	Steel Eng. (Plate)
Data model	Exact
BOM-relevant	Yes
Item number	0
T_Sachnummer	TN-27756
T_Benennung	Sheet
T_Normbez.	BI 6
T_Freigabe	In progress
Article number	BI 6
Designation 1	
Designation 2	1x BI 30x50.000x100.000 S235JR
Part type	Plates

Properties

Designation	Value
Part name	BL_RECHTECK_F_WERT.FEV
Geometric type	Solid
Colour	Oyster White [40]
Layer	1
Material	S235JR
Module	Steel Eng. (Plate)
Data model	Exact
BOM-relevant	Yes
Item number	0
T_Sachnummer	TN-27756
T_Benennung	Sheet
T_Normbez.	BI 6
T_Freigabe	In progress
Article number	BI 6
Designation 1	
Designation 2	1x BI 6x100.000x50.000 S235JR
Part type	Plates

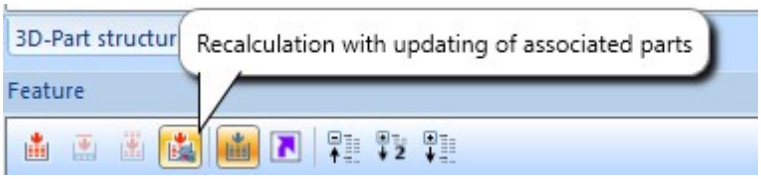
If you load drawing files created with a Version older than HiCAD 2021 SP1, you must perform a feature recalculation of the plates to adapt the **Designation 2** attribute.

Major Release 2021 (V 2600)

Connections

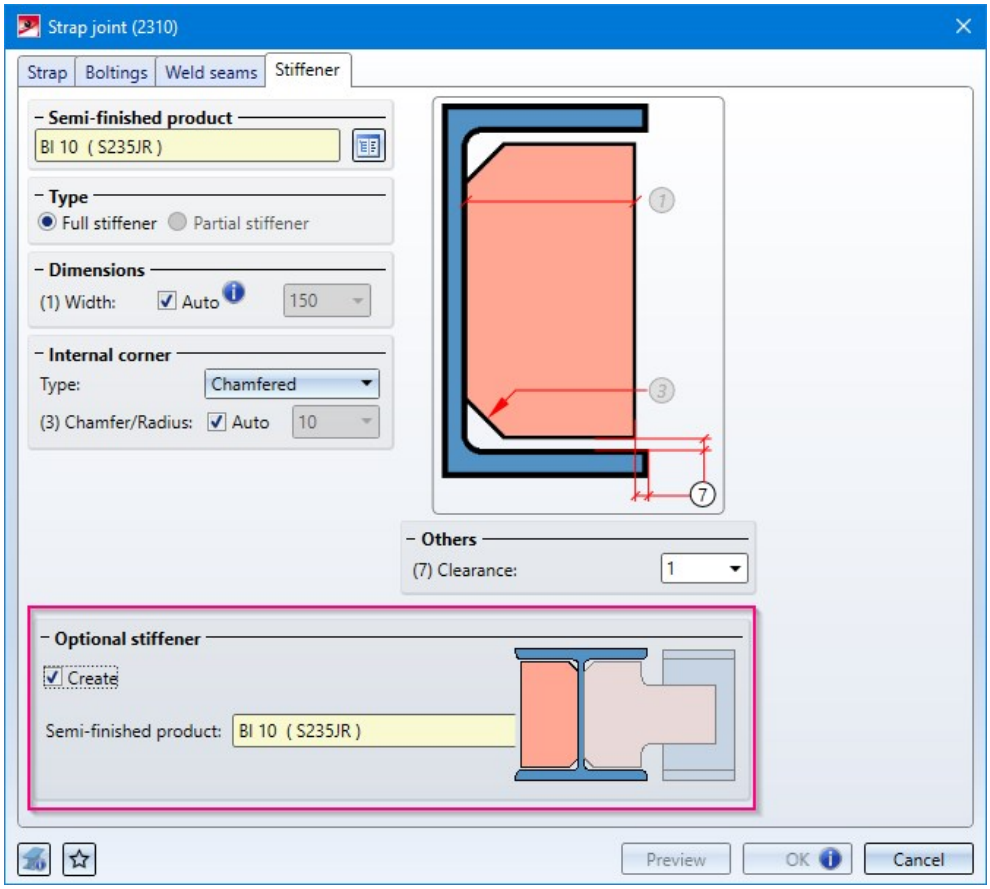
Update connections

To update connections, you can also use the feature function **Recalculation with updating of associated parts** , which can be found in the toolbar of the Feature window in the ICN.

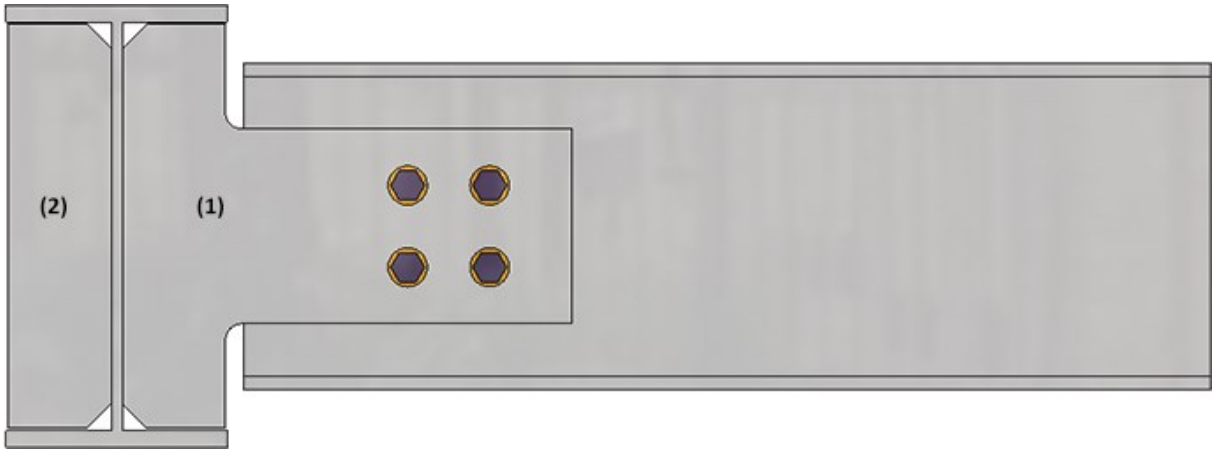


Strap joint (2310) with optional stiffener

For the **Strap joint (2310)** with stiffener, a second stiffener can be optionally installed. The **Stiffener** tab has been extended accordingly.



The **Type** selected under **Internal corner** is used for the optional stiffener. If the stiffener is chamfered, for example, this also applies to the optional stiffener..



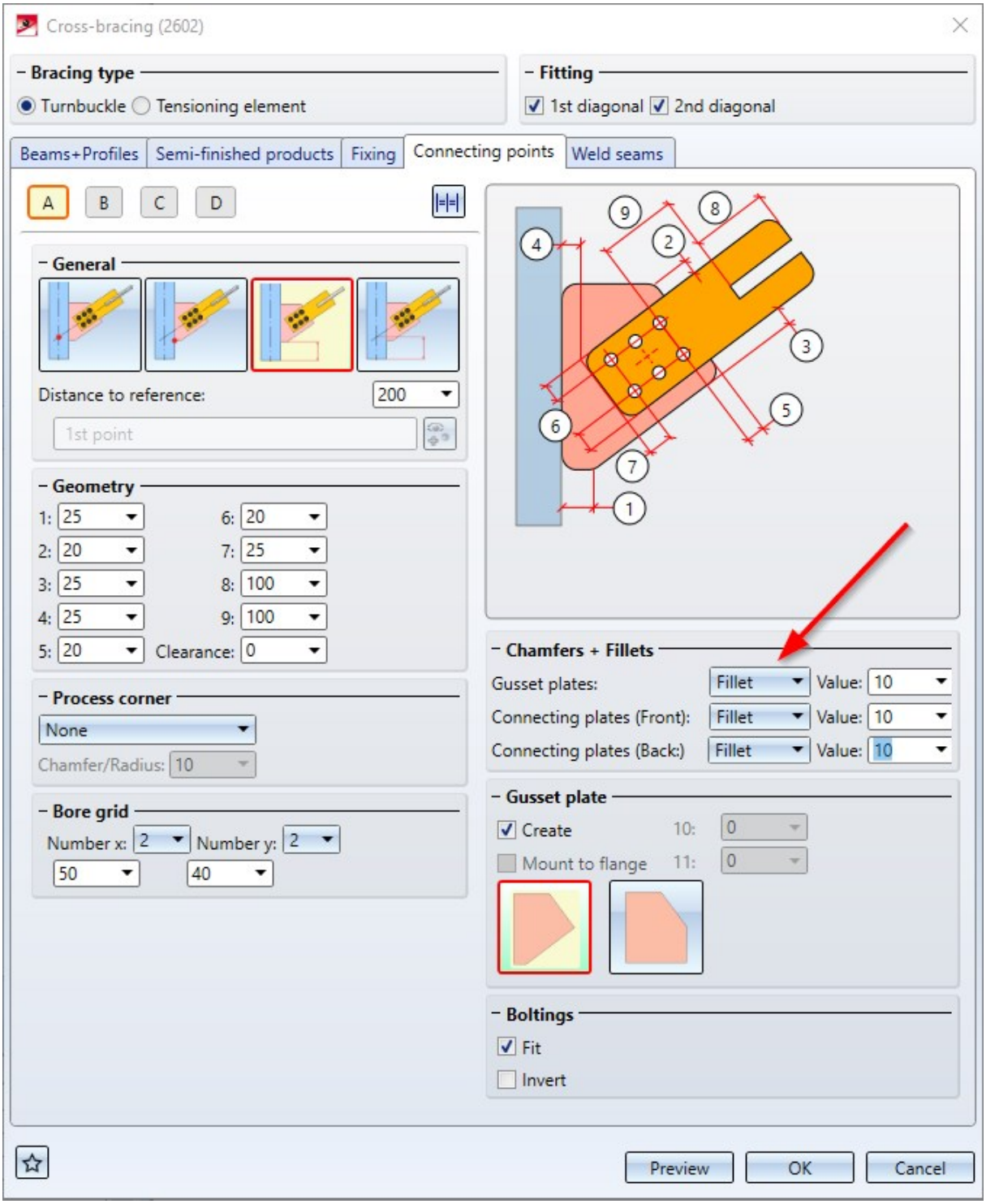
(1) Stiffener, (2) Optional stiffener

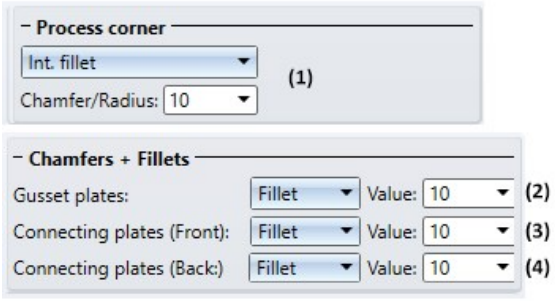
Cross-bracing - Chamfering gusset plates and connecting plates

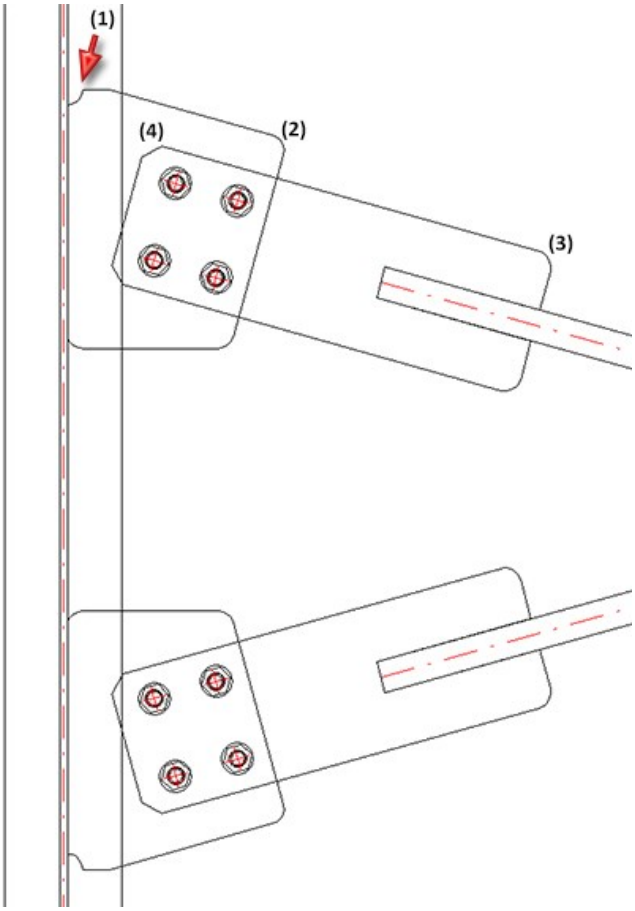
Gusset plates and connecting plates created for cross-bracings could previously only be filleted. As of HiCAD 2021, chamfering is now also possible here. This affects

- the gusset plates in **Cross-bracing (2601)** and
- the gusset plates and connecting plates in **Cross-bracing (2602)**.

The dialogue windows have been adapted accordingly.




Process corner
Int. fillet
Chamfer/Radius: 10
(1)
Chamfers + Fillets
Gusset plates: Fillet Value: 10 (2)
Connecting plates (Front): Fillet Value: 10 (3)
Connecting plates (Back:) Fillet Value: 10 (4)


(1)
(4)
(2)
(3)

Base plate + Anchor plate (2101) - Extensions

For the connection **Base plate + Anchor plate (2101)** , various extensions are available with HiCAD 2021.

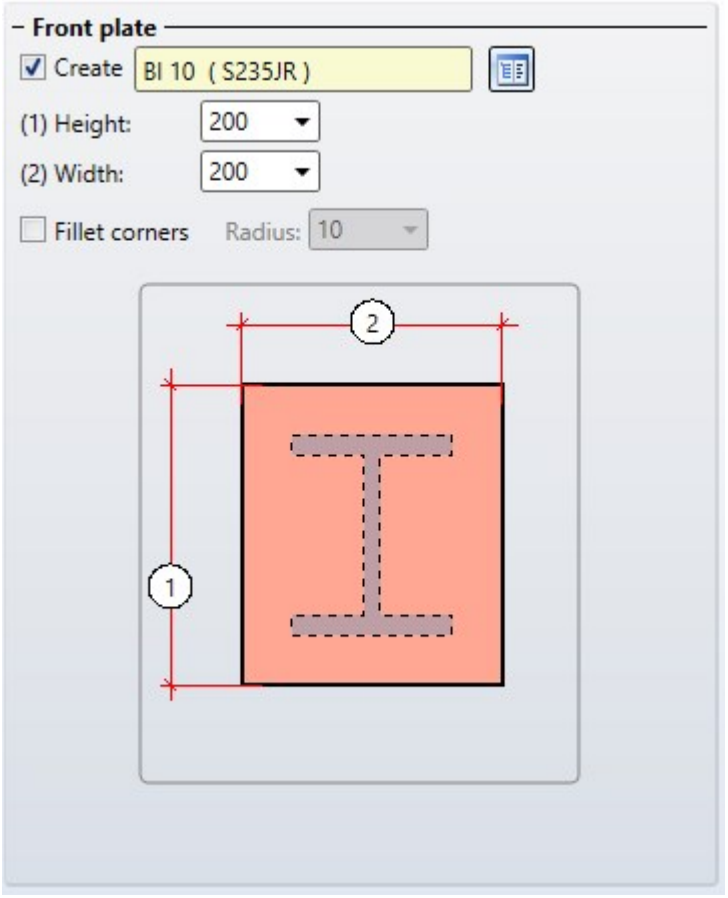
Changed part structure

The part structure of the created parts has changed as follows:

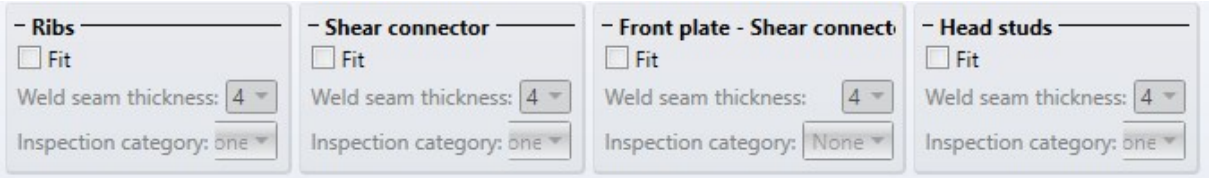
- The assembly with the shear anchor is now called **Anchor plate**.
- The welded plate is installed as an individual part and is not assigned to an assembly as before.

Shear connector with front plate

The shear connector can now be equipped with a front plate - also rounded. The **Shear connector** tab has been extended accordingly.

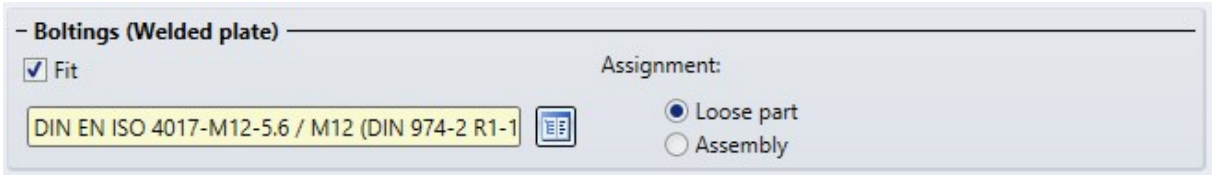


If the **Create** checkbox is active, you can specify on the **Weld seams** tab whether weld seams are to be installed between the front plate and the shear connector.

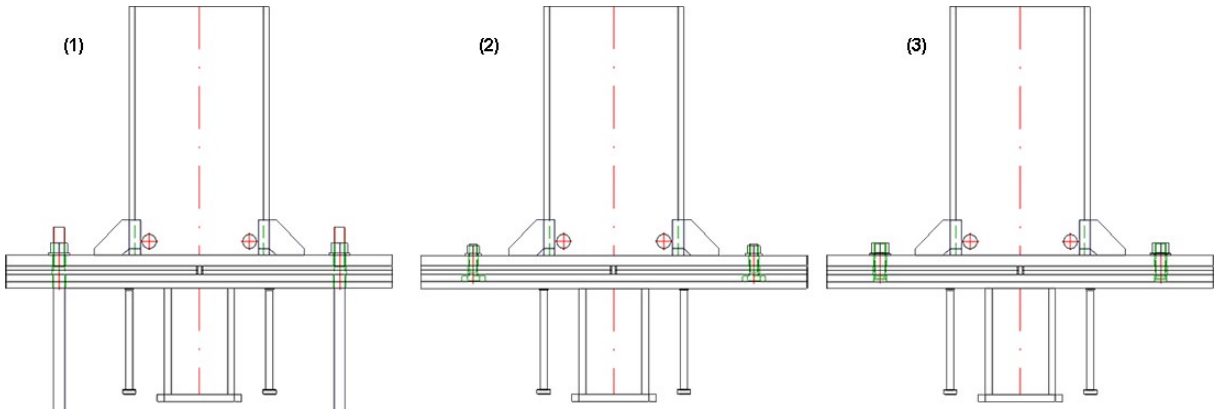


Fixing

On the **Fixing** tab, under **Boltings (Base plate)**, you can select another type of fixing.



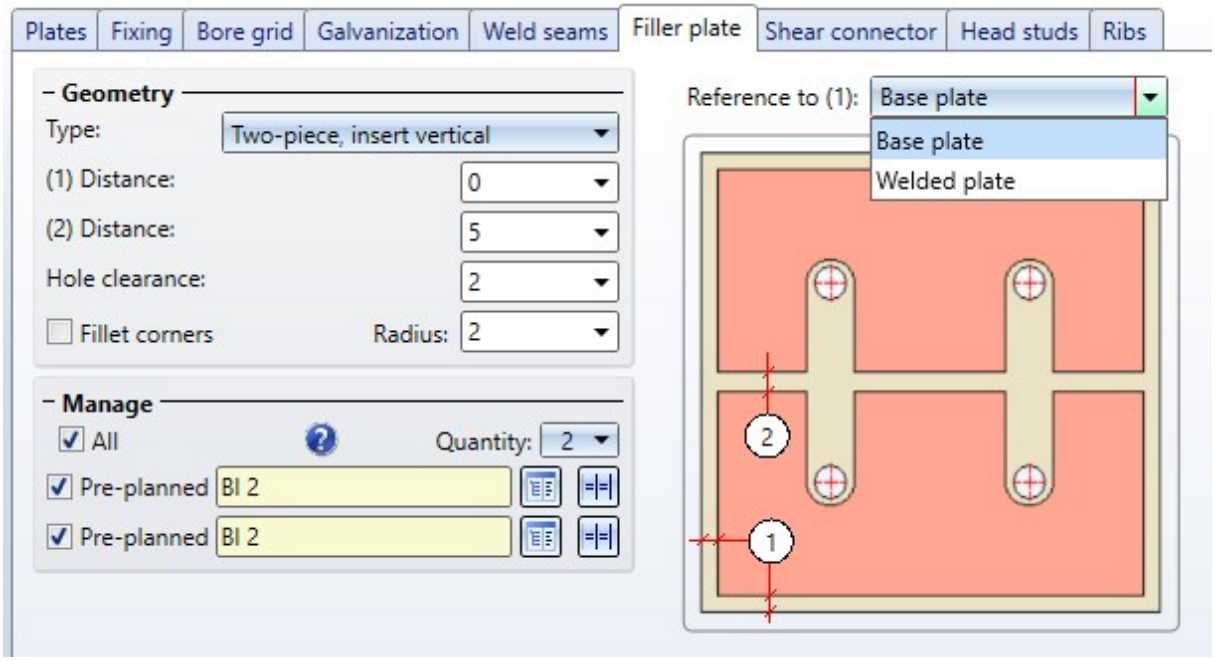
If the checkbox **Fit** is active, the fastening is done by a screw connection countersunk into the welded plate from below, i.e. only welded plate and base plate are screwed together.



Fixing (1) Boltings (Base plate), (2) Boltings (Welded plate), (3) Threaded stud

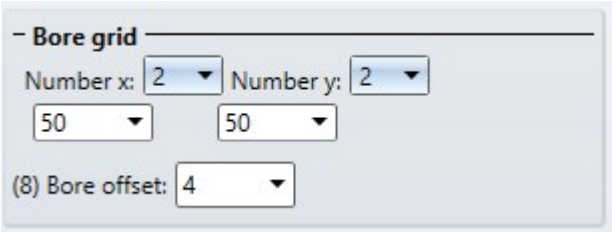
Size of the filler plates

On the **Filler plate** tab you can now choose whether the size of the filler plates should be based on the base plate or the welded plate.

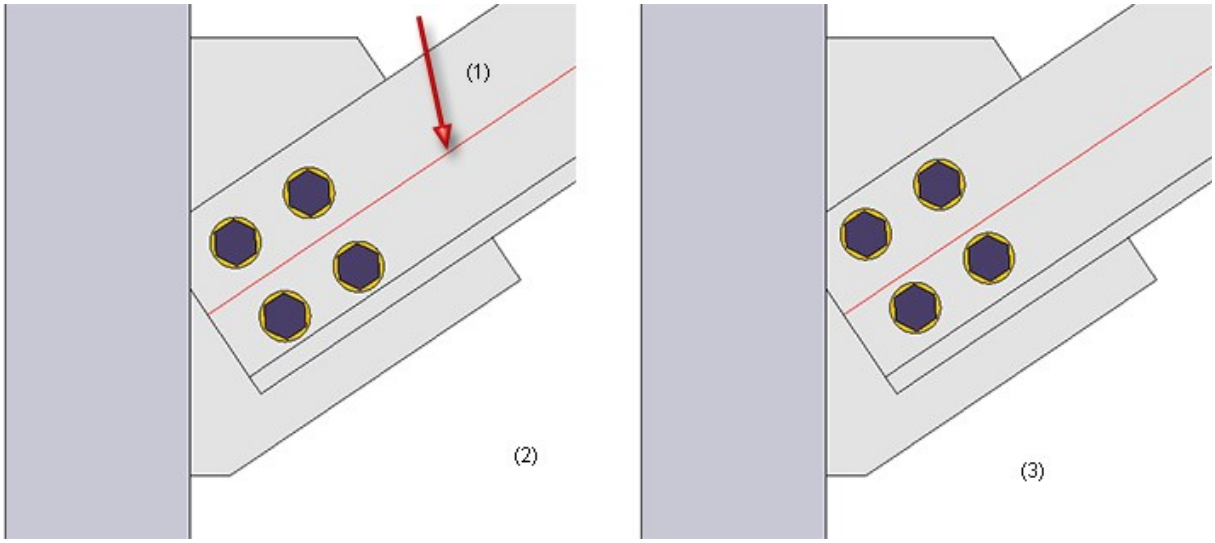


Cross-bracing (2601) - Horizontal offset of bore grid

For the **Cross-bracing (2601)**, an offset for the bore grid can now be specified on the **Connecting points** tab and thus influence the position of the bore grid.



The default setting is 0, i.e. the centre of the bore or bore grid lies on the centre axis of the diagonal bracing profiles. In certain situations, for example with L-profiles, this is often not desired. In such cases, the position of the bore grid can be influenced by specifying an offset.



(1) Centre line of L-profile, (2) Without offset, (3) With offset

Series beams

Kingspan Sandwich elements for roof and wall

The catalogue **Factory standards > Roof Wall Facade > Room-closing profiles** now contains the following new sandwich elements for roofs and walls by the company Kingspan:

- **Kingspan >Sandwich panels, Roof/Wall > RW QuadCore** and
- **Kingspan >Sandwich panels, Wall > AWP QuadCore**.

In this context, the **QuadCore®** rigid foam from **Kingspan** has been added to the **Duromere** table in the **Materials > Plastics** catalogue.

See also **Catalogue Editor - What's New**?

Improved generation of notches on beams with CUTOUT contour

The algorithm for creating notches on CUTOUTs has been revised and now delivers better results than in previous versions.

This initially only affects newly created notches; notches created with previous versions remain at the previous level even after a feature recalculation. To have these notches calculated with the new algorithm, you must delete them and then create them again.

Process beams

Exchange standard beams

When exchanging beams, the material of the beam is not changed, provided that the material of the beam to be exchanged is available for the exchanged beam.

An example:

A drawing contains a type I 80 beam with material S355J0. This beam is to be exchanged with an IPE beam. In this case, the material S355J0 is automatically set when selecting the exchanged beam in the catalogue. If the material for the exchanged beam is not available, the standard material of the exchanged beam is set.

Divide beams

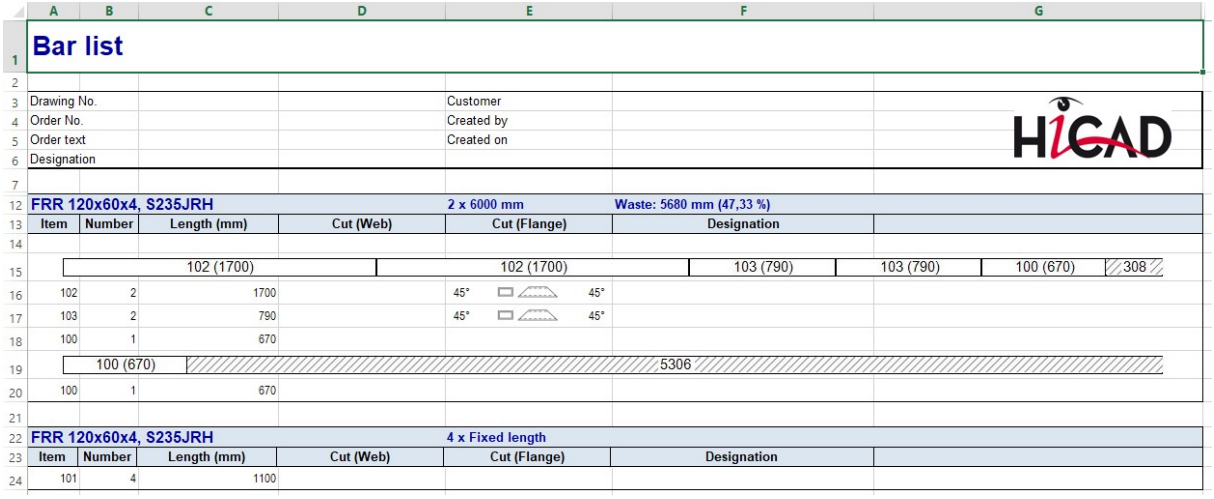
For dividing beams along a direction the function **Divide along direction** is now available at **Steel Engineering > Lengthen > Divide**. This function essentially works in the same way as the 3-D function **3-D Standard > Process > Trim > Divide along direction** function, except for the following differences:

- The beam to be divided must be selected after the function call.
- The beam axis is automatically suggested as the division axis, whereby the division direction is determined by the selection point.
- After the division, the function remains active, i.e. another beam can be selected directly for division. The beam selection is ended by pressing the middle mouse button.

Fixed length beams

In practice, model drawings often contain beams, e.g. purchased parts, which are ordered for the production with a fixed length. These beams should not appear in the bar list or must be shown separately.

In HiCAD this can now be regulated via the attribute **Fixed length** (%FIXLEN), which must be assigned to the corresponding beams. If this attribute has the value 0 or is not set at all, the fixed length is not taken into account in the bar list. If the value is 1, then these beams are listed separately in the bar list.



Bar list

Drawing No.		Customer	
Order No.		Created by	
Order text		Created on	
Designation			

FRR 120x60x4, S235JRH — 2 x 6000 mm — Waste: 5680 mm (47,33 %)

Item	Number	Length (mm)	Cut (Web)	Cut (Flange)	Designation
		102 (1700)		102 (1700)	103 (790) 103 (790) 100 (670) 308
102	2	1700		45° 45°	
103	2	790		45° 45°	
100	1	670			
		100 (670)		5306	
100	1	670			

FRR 120x60x4, S235JRH — 4 x Fixed length

Item	Number	Length (mm)	Cut (Web)	Cut (Flange)	Designation
101	4	1100			

By default, the attribute is not set, i.e. it is not included in the attribute masks. If you want to use this attribute, you must manually extend the attribute mask for beams by adding the files

- **BRW_3DTeil_Profil.HDX** (part attributes for beams) or
- **BRW_3DTeil_Profil_H.HDX** (part attributes for beams that are assembly main parts)

accordingly.

In case of a new installation of HiCAD (from HiCAD 2021), the attribute fixed length is automatically evaluated when creating Excel BOMs with the configuration file HiCAD_Stahlbau.2600.0. For this purpose, the system file for the BOM item data **rm_h_exportpart.hdb** must be supplemented with the line

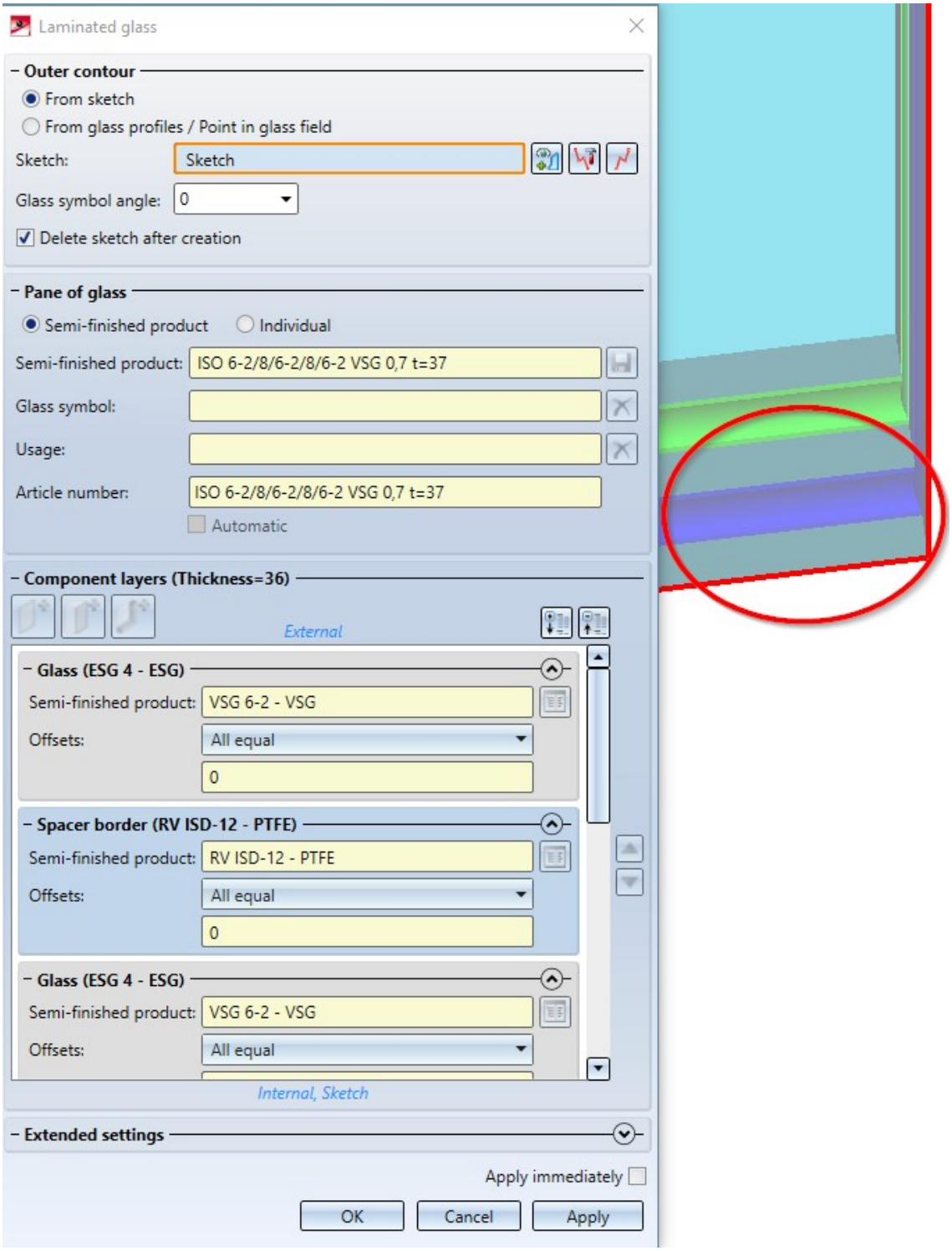
<H>::TEXT=""::ATTR="%FIXLEN"::TYP="INTEGER"::ALIGN="LEFT"::EDIT="NO"

has been adapted accordingly. In the case of an update installation, you may have to add this line to the file manually.

Laminated glass

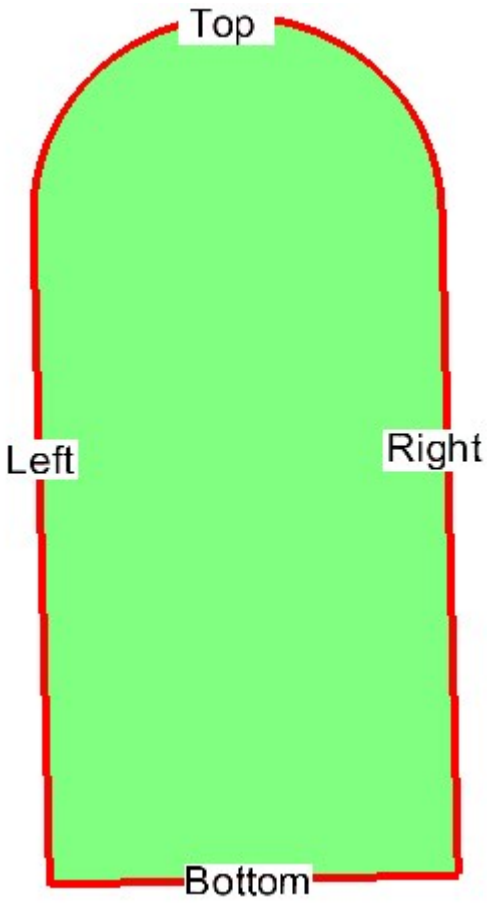
Highlighting of active layer

If you construct a laminated glass with an individual layer structure, the corresponding layer is now also highlighted in colour in the drawing when you select a layer in the **Laminated glass** dialogue window.



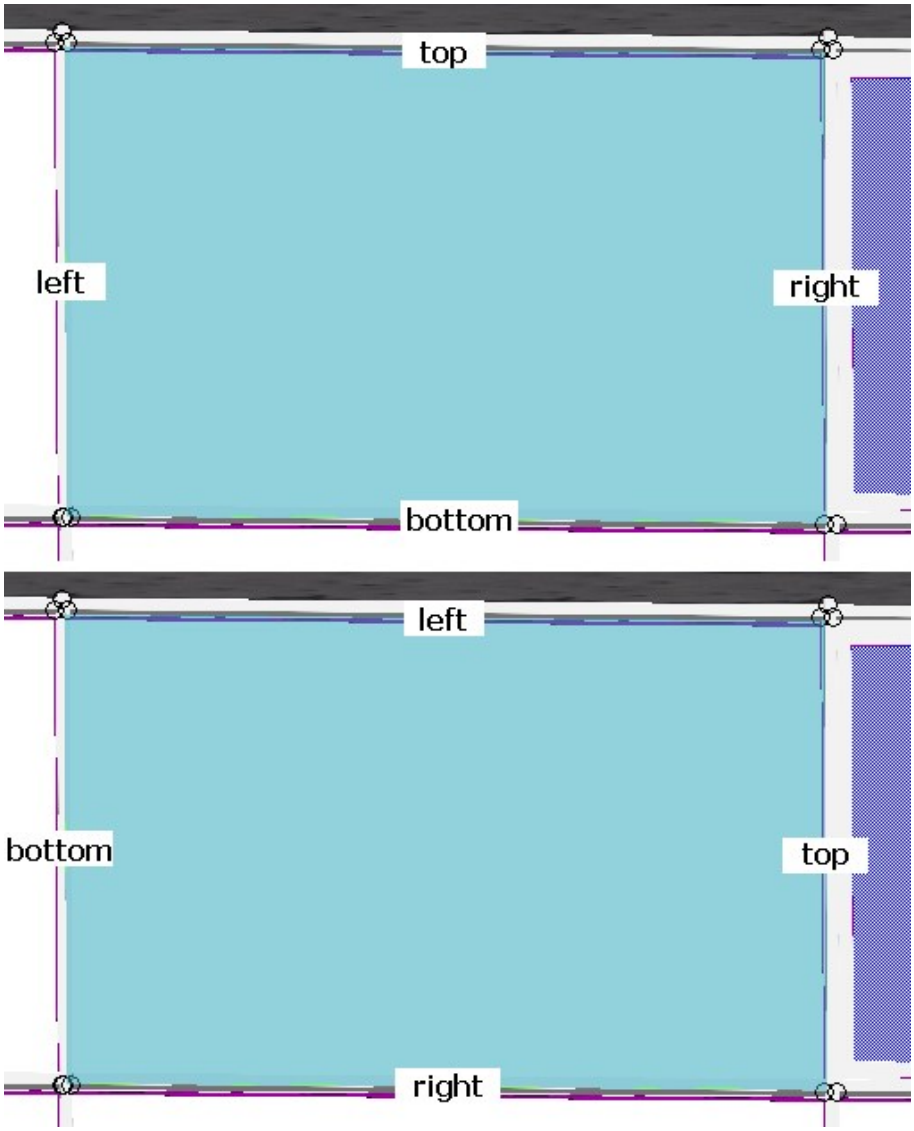
Direction indication during installation

During insertion / editing of laminated glass panes, the directions left, right, top and bottom are now displayed in the drawing on the glass.



Rotation angle of glass symbol

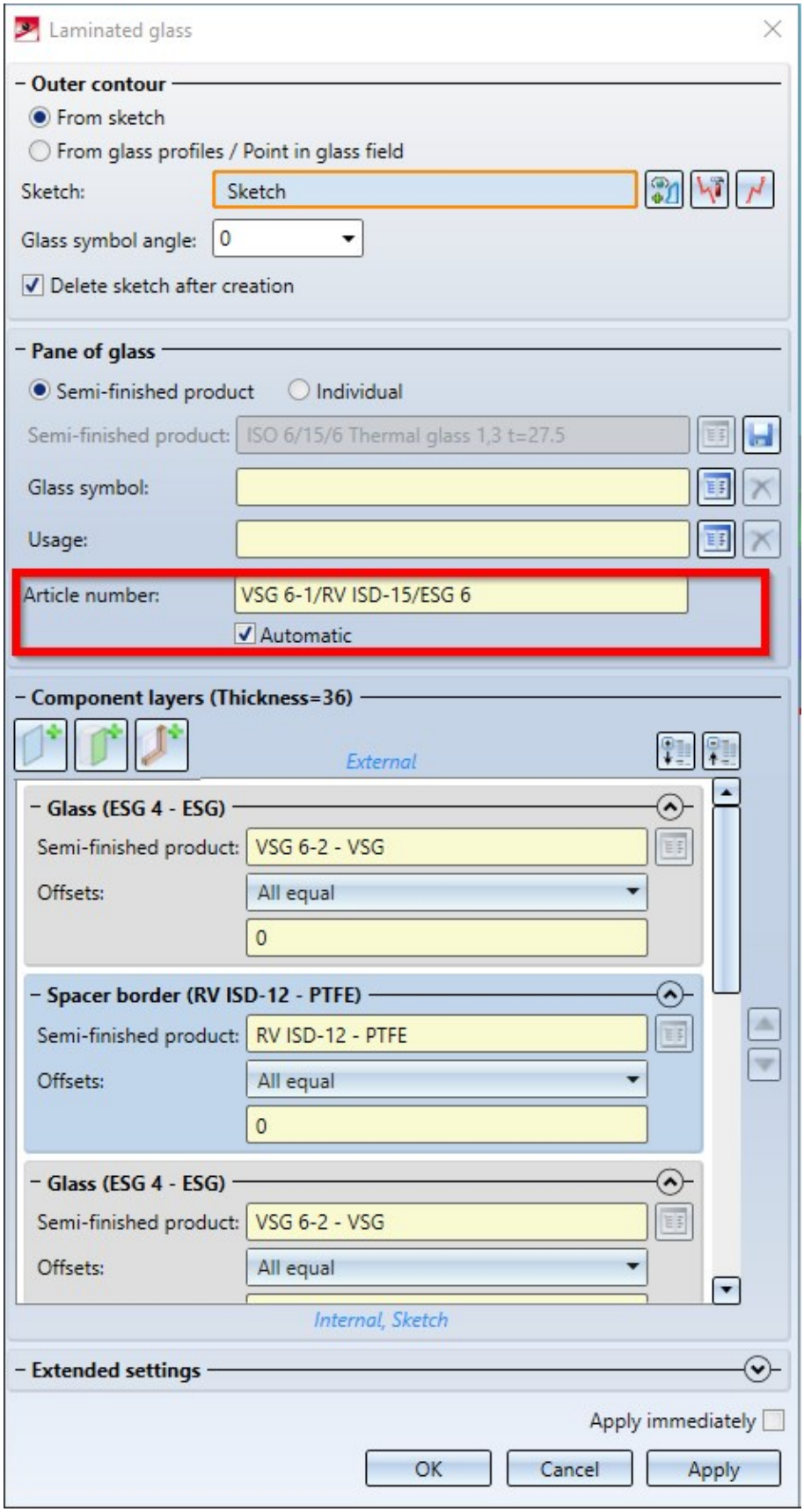
In order to be able to influence the alignment of the glass and thus the angle of rotation of the glass symbol, the option **Glass symbol angle** has been added to the dialogue. With this option the alignment can be adjusted exactly to the degree. The direction display in the construction is adjusted accordingly.



Top: Glass symbol angle 0; Bottom: 90.

Automatic assignment of an article number

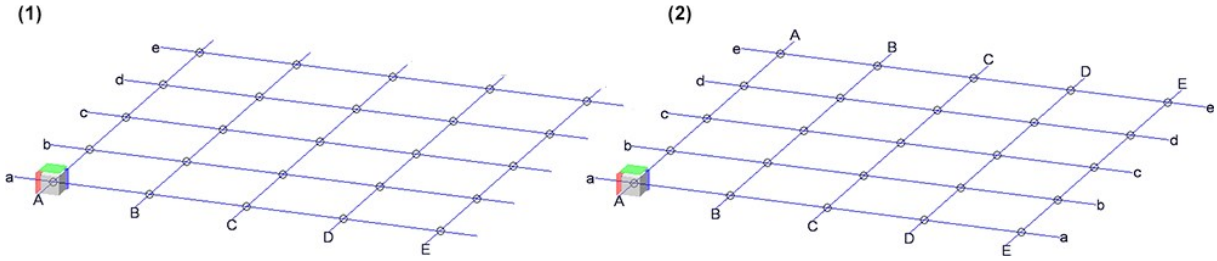
It is now possible to automatically generate an article number from the layers used in the individual construction of a laminated glass panel. For this purpose, there is a new checkbox **Automatic** in the dialogue window **Laminated glass** under the **Article number** field. If this is activated, the article number is generated automatically by combining the designations of the individual layers separated by a separator to form an article number.



The character "/" has been set as separator; this can be changed in the **Configuration Editor** at **Steel Engineering > Products > Laminated glass pane > Separator for auto-generated article number of a laminated glass pane** if desired.

3-D grid

When working with the 3-D grid function, grid annotations are now inserted on both sides.



(1) up to HiCAD 2020, (2) from HiCAD 2021 onwards

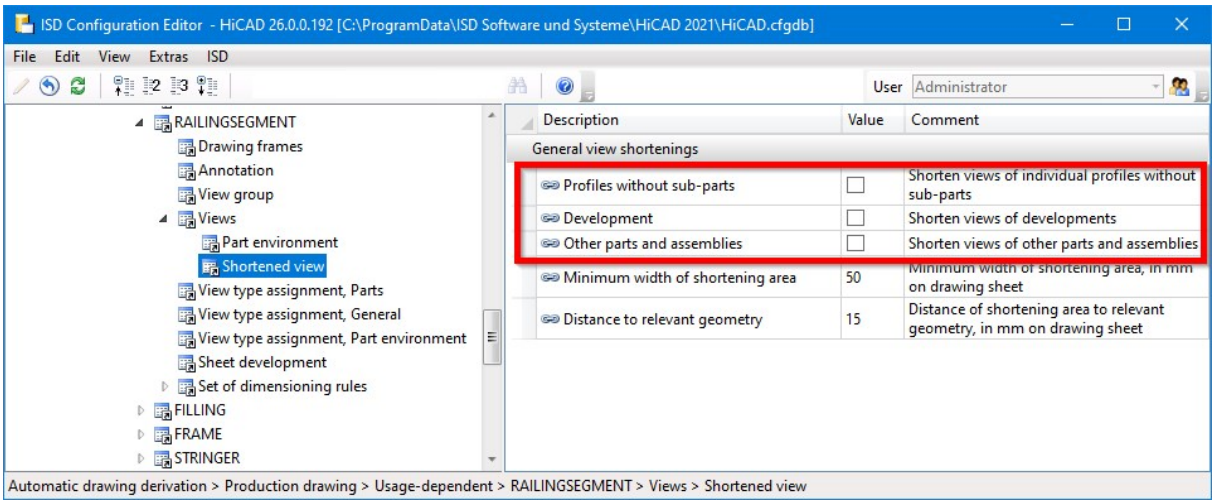
Stairs and railings

Railings in derived drawings

The configuration **RAILINGSEGMENT** for railings in the drawing derivation has been changed. Views are now not automatically shortened here. For this purpose, in the Configuartion Editor at

Automatic drawing derivation > Production drawing > Usage-dependent > RAILINGSEGMENT > Views > Shortened view

the checkboxes have been deactivated.



Spiral staircase

With the new **Spiral staircase** tool in the **Civil Engineering functions** docking window the first draft of a configurator for spiral staircases is available. However, it does not yet have the full range of functions yet. In future HiCAD versions, this tool will be expanded into a fully functional configurator with different variants for newels, stringers, platforms, railings and steps, special dimensioning rules for drawing derivation, etc.

However, the current configurator can already be used to visualise spiral staircases, for example in Plant Engineering drawings as a placeholder for such prefabricated staircases.

The image below shows a spiral staircase with two floors (1) and (2), which has been created with the default settings of the ISD.

(2)

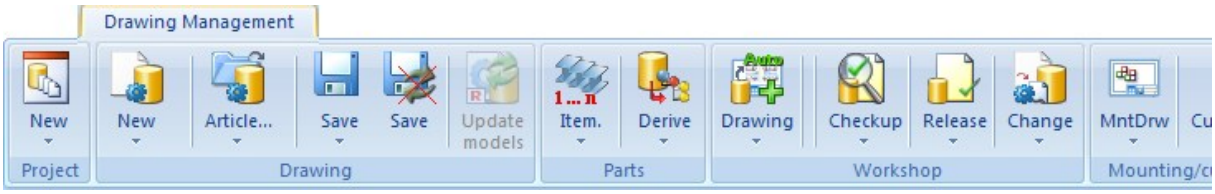
(1)

Drawing Management

Service Pack 1 2021 (V. 2601)

New Ribbon name

The **Management + BIM** Ribbon has been renamed to **Drawing Management**.



The settings in the Configuration Editor can be found under **PDM > Drawing Management**.

Automatic updating/creation of drawings via part filter

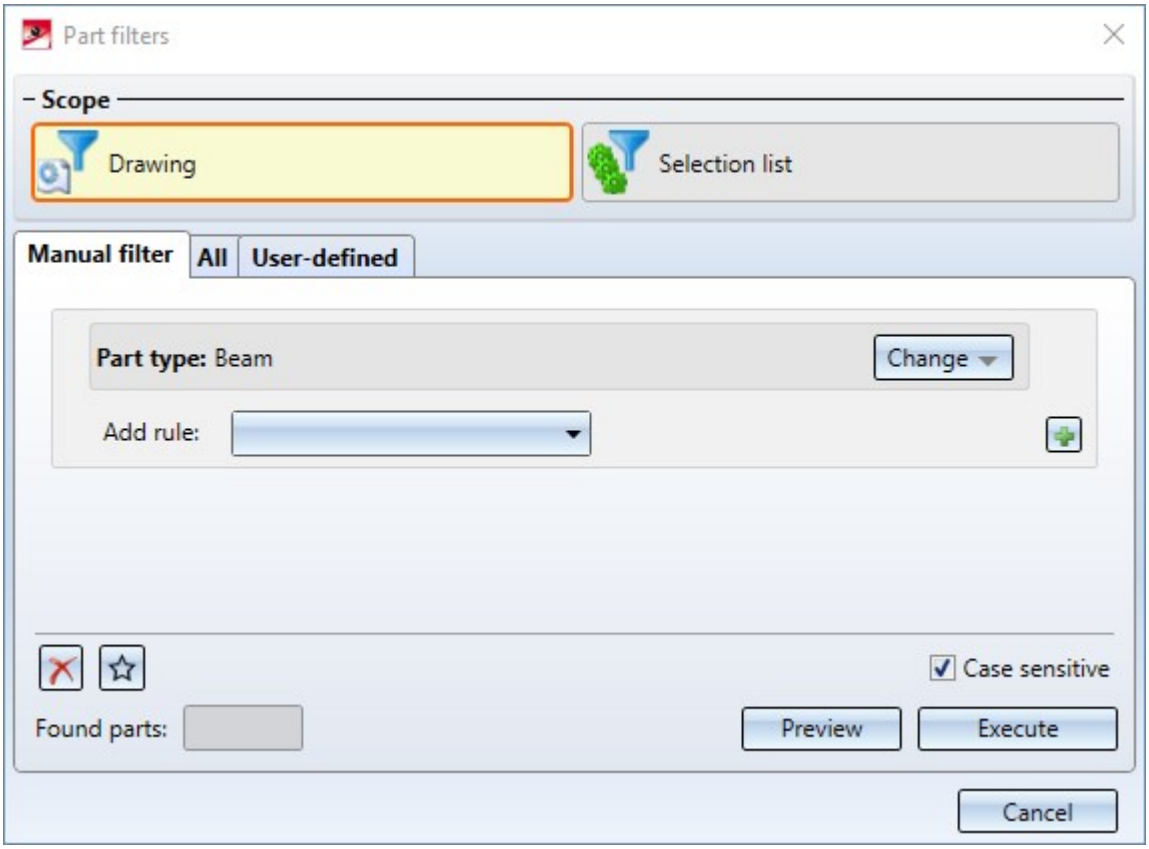
The automatic updating and creation of drawings can now also be restricted via parts filters. For example, you can search for specific beams or plates and update/create drawings only for these parts.

Two new functions are available for this at **Drawing Management > Workshop > Drawing**:

Via part filter

Via filter with HELiOS selection

After calling the functions, the **Part filter** dialogue window is displayed.

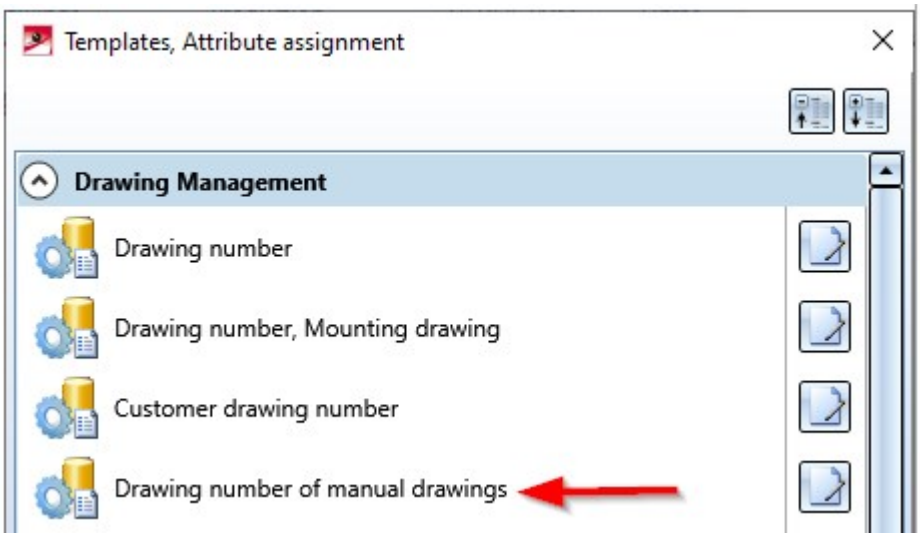


Detailed information about the part filters can be found here.

Template for attribute assignment of manually created drawings

A corresponding template for attribute assignment is now also available for manually created drawings:

BIM_PDM_ManualDrawingnumberGeneration.ftd.



This template can be used to configure drawing numbers of manually created drawings. The number is assigned to the HELiOS attribute **DRAWINGNUMBER_TEXT**.

The default setting is: **M - {HEL_PROJNUMMER (Project)}-{DRAWINGNUMBER (Document master)}**.

Drawing frame / Title block

The **View scale** and the **Sheet name** can now also be output in the title block in the Drawing Management. It is not necessary to change the file schrfe_aus_Attr.dat or wsd_titleblock.dat - as is the case when working without Drawing Management - but it is sufficient to adjust the title blocks in the drawing frames by using the variable belonging to the respective special attribute in the title block.

The following special attributes are available:

Output	Attribute	Variable in HiCAD title block*
Total number of drawing sheets of the model drawing	%BLATTANZAHL	@*@3,D.HIC_Blattanzahl@*@
Number of the active drawing sheet	%BLATTNUMMER	@*@3,D.HIC_Blattnummer@*@
Name of the active drawing sheet	$BLATTNAME	@*@3,D.HIC_Blattname@*@
Main scale of the model drawing	$MASSSTAB	@*@10,D.HIC_Massstab@*@
View scale**	$VIEWSCALE	@*@10,D.HIC_ AnsMassstab@*@

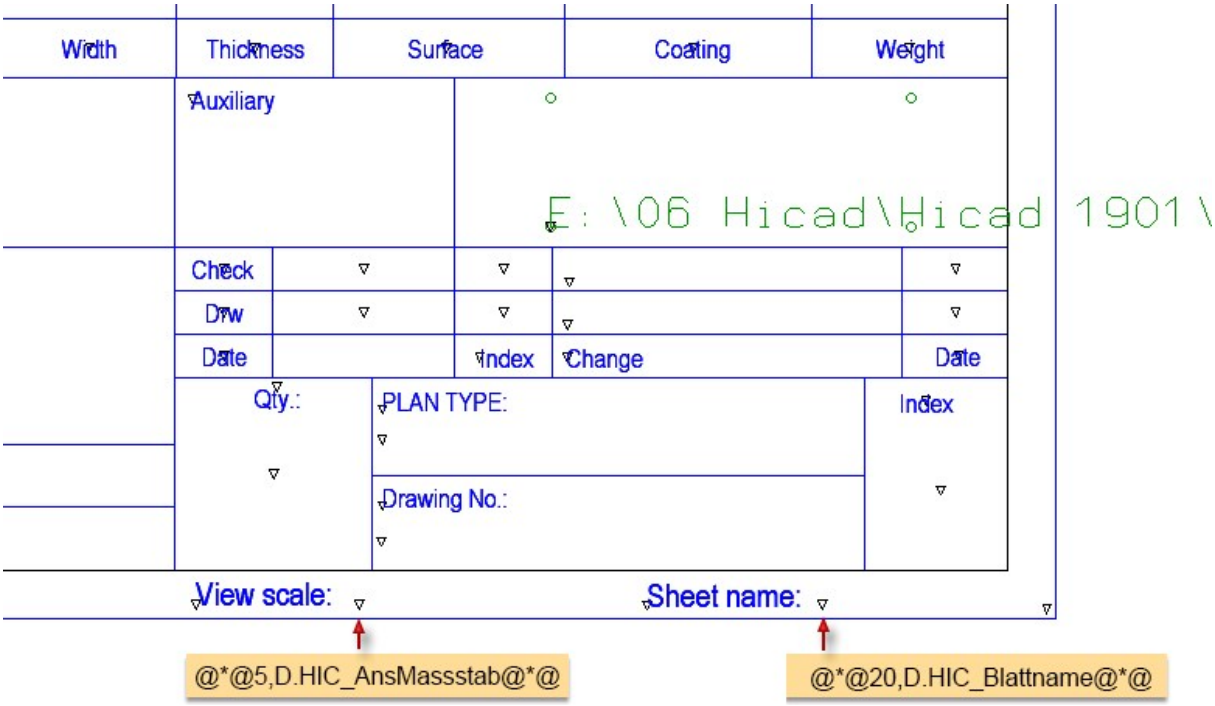
* The length values are example values

** The view scale is only output if all relevant views of the drawing sheet have the same scale. Relevant are all views except axonometric workshop drawing views, views of developments of Sheet Metal parts and detail views.

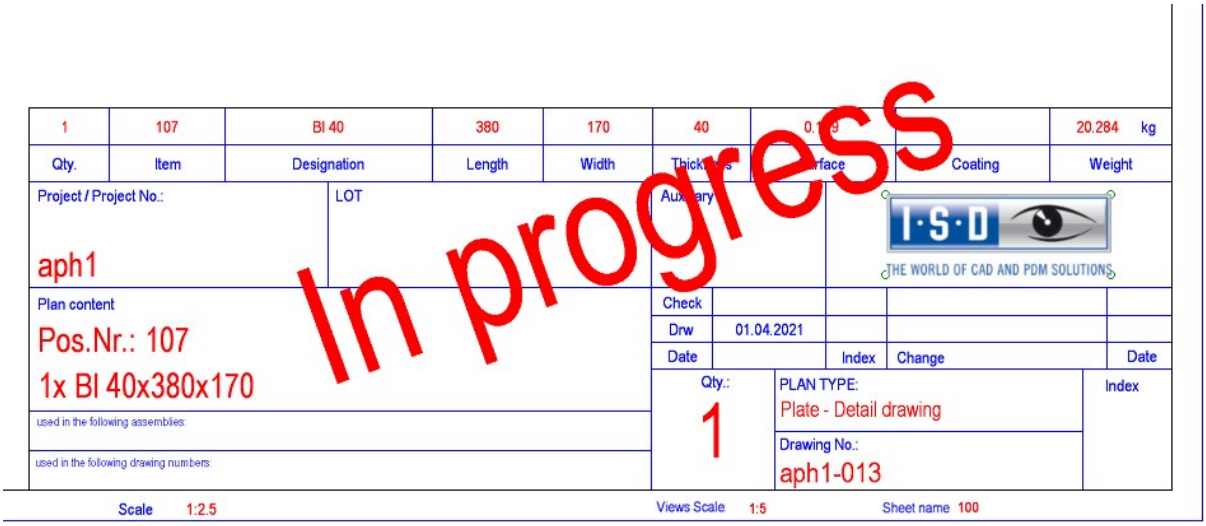
Using the prefix **HIC_** ensures that this data is always evaluated from the HiCAD sources when updating the drawings.

For information on how to use these attributes in the title block, see the section Variables/Attributes in the title block.

In the following example, the drawing frame DINA4H_BLECHE.FGA has been extended by the view scale and the sheet name.

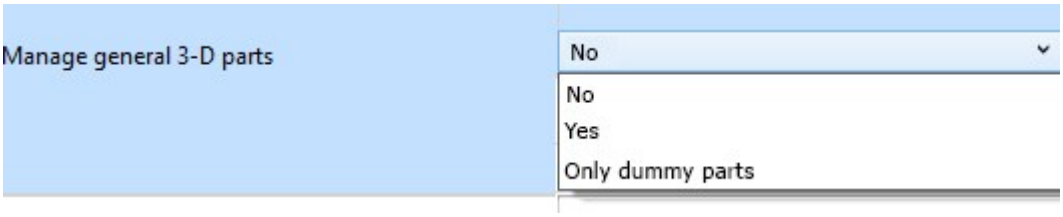


In the drawing derivation, the view scale and the sheet name are then output in the individual detail drawings for Steel Engineering plates.



Managing dummy parts

Analogous to general 3-D parts, from SP1 the automatisms of the Drawing Management can also be used for dummy parts. For this purpose, the selection box in the Configuration Editor at **PDM > Drawing Management > Manage general 3-D parts** has been extended by the option Only dummy parts.



With this option, dummy parts (e.g. foil, insulation, etc.) can also be managed and taken into account in bills of materials.

Update / create new drawings for all parts

For the functions

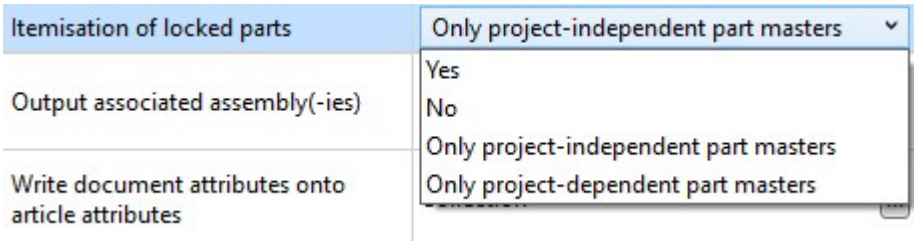
All and

All with HELiOS selection

the processing sequence has changed. Analogous to the **Drawing** function, existing drawings are now updated first and then new drawings that are still missing are created.

Itemisation of locked parts - Renamed options in Configuration Editor

For the **Itemisation of locked parts** parameter at **PDM > Drawing Management**, the texts in the selection box have been changed:



Lock HELiOS Functions for Objects of Drawing Management

In some cases it may be useful to protect objects of the drawing management against editing with functions of HELiOS Desktop or the **HELiOS PDM** Ribbon in HiCAD. For example, it can be ensured that for parts and assemblies that are relevant for drawing management, the workflow functions, index management etc. of HELiOS Desktop cannot be used, but only the functions of the **Drawing Management** Ribbon.

To achieve this, proceed as follows:

Open the file **kst_isd.ini** in the HiCAD sys directory with an editor. In the section **[Module]** change the line

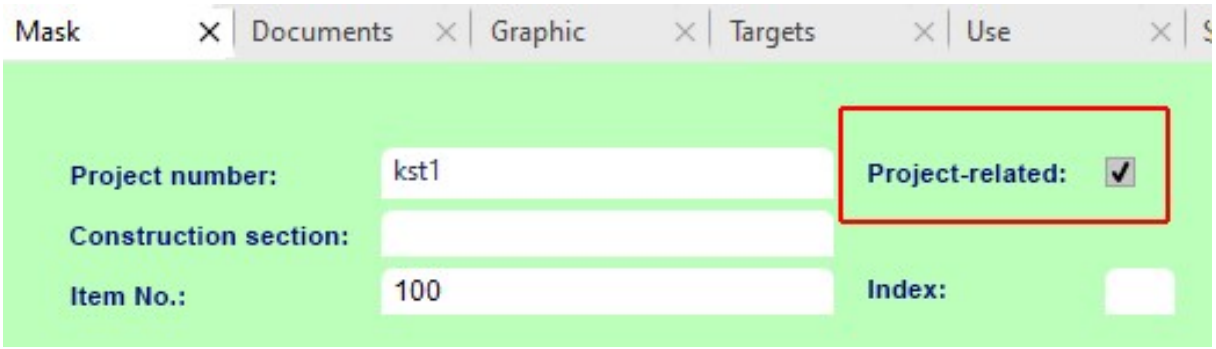
IsdStdHicadBim1 = Active="0" AllowedVersions="1" to

IsdStdHicadBim1 = Active="1" AllowedVersions="1"

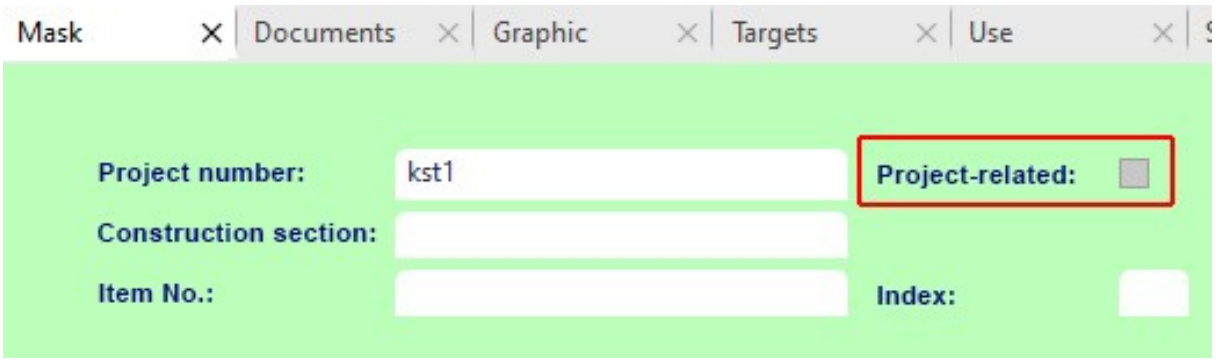
```
[Module]
;->Attention! The following order of the KstExt modules to be loaded
; determines the standard calling order of the KstExt trigger functions!
;->For temporarily deactivating a KstExt module use parameter Active,
; e.g., to deactivate module Hel2Komp, write:
;  Hel2Komp = Active="0"
; The default Activity is 1 (that means yes).
;->To allow selected versions of a KstExt module use parameter AllowedVersions,
; e.g., to allow the versions "1" and "2a", write:
;  Hel2Komp = AllowedVersions="1","2a"
; The default allowed version is "1".
;->To load a KstExt module only on selected hosts use parameter LoadingHosts,
; e.g., to load module Hel2Komp only on host dedtm066, write:
;  Hel2Komp = LoadingHosts="dedtm066"
; Normally, all hosts load all active KstExt modules.
;->All parameters can be used in combination,
; e.g., to explicitly specify activity and allowed versions for module Hel2Komp, write:
;  Hel2Komp = Active="1" AllowedVersions="1"

IsdStdHicadBim1 = Active="1" AllowedVersions="1"
Hel2Komp = Active="1" AllowedVersions="1"
```

This setting then has the effect that for parts and assemblies the attribute STAHLBAUAUTO is taken into account when calling the corresponding functions of the HELiOS Desktop:

- Drawing Management-relevant parts/assemblies, i.e. project-related parts/assemblies cannot be managed via the HELiOS Desktop (e.g. workflow functions; index management etc.).



- Project-independent items in the drawing management environment can be managed in HELiOS Desktop.



This applies analogously to the corresponding functions of the **HELiOS PDM** Ribbon in HiCAD.

Major Release 2021 (V 2600)

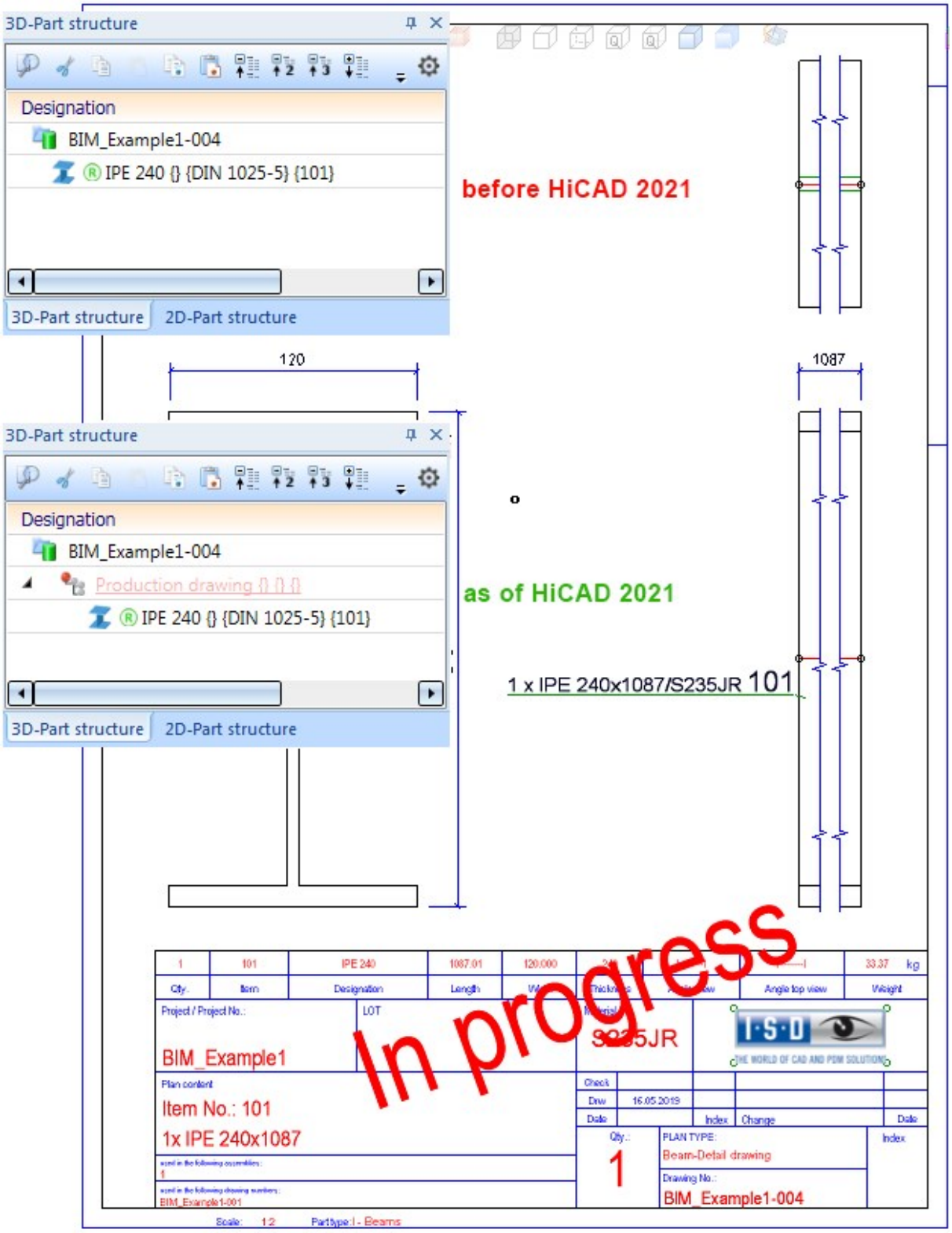
Part annotation and attribute assignment

The functions at **Drawing > Itemisation/Detailing** are now also available in the **Production** function group of the **Drawing Management** Ribbon tab:

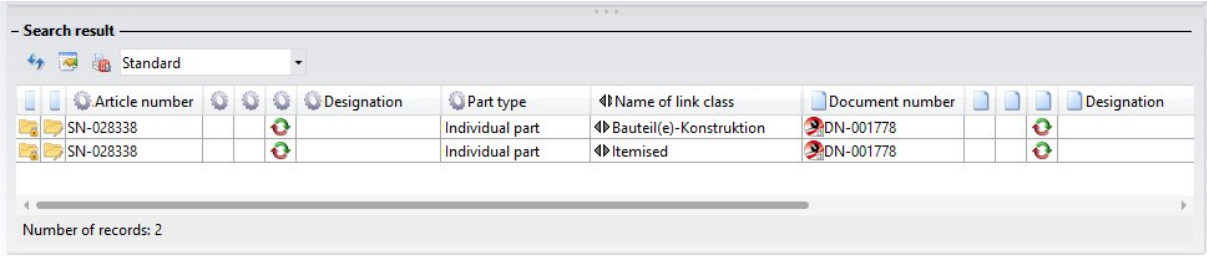
Attributes		
	Edit part annotation templates	Settings for part annotation in workshop drawings. The function is identical to the function with the same name in the **Itemisation/Detailing** function group of the **Drawing** tab.
Click on to open a pull-down menu with the following functions:		
	Edit templates for attribute assignment	Allows you to adjust the templates for attribute assignment of the Drawing Management as well as in the DSTV-NC export. The function is identical to the function with the same name at **Drawing > Itemisation/Detailing > Attr...** .
	Article master, Itemisation	Definition of the item attributes to be automatically generated during the itemisation/identical part search.

No main assemblies in detail drawings

Until now, an article master for the main assembly was always created during the automatic creation of detail drawings. Since this has led to problems in various cases - for example, when forwarding to ERP systems - the creation of the article master for main assemblies in the detail drawings will be omitted from HiCAD 2021.



Note that the detail drawing and the part have the same item master. The following illustration shows the result list in HELiOS.



Metal Engineering

Service Pack 1 2021 (V 2601)

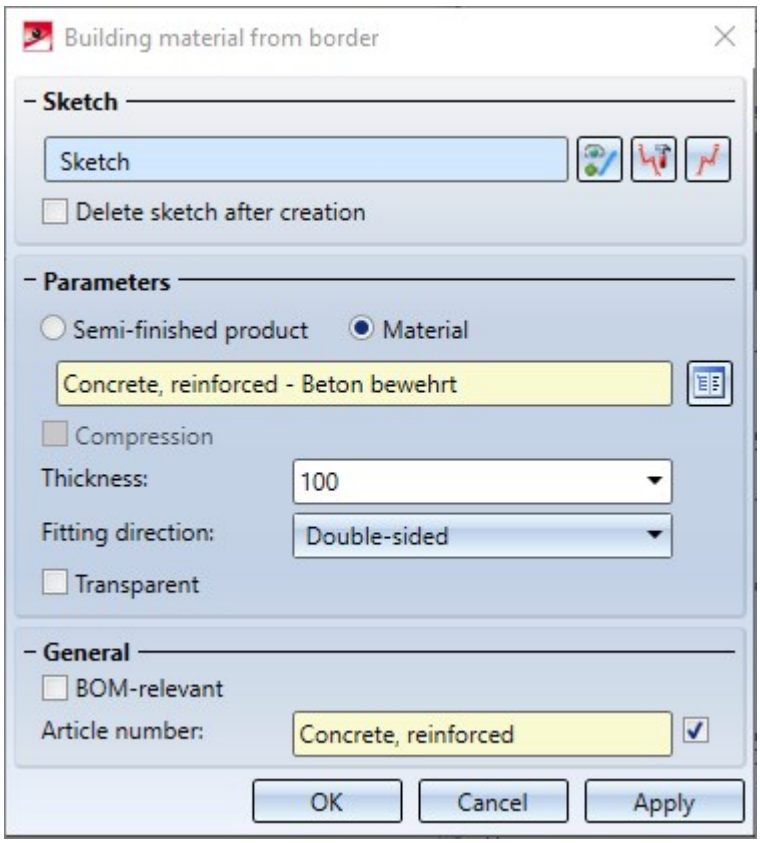
Changed catalogue structure

The supplied sub-catalogues at "Factory standards > User-defined semi-finished products > Civil Engineering" have been moved to the catalogue **Factory standards > User-defined building materials**. If you had stored customer-specific catalogues or tables at **Factory standards > User semi-finished products**, this catalogue was renamed to **USER_HALBZEUGE_TO_DELETE** to allow you to move your data to a new location in the catalogue structure yourself. If, on the other hand, the catalogue was empty, it was automatically deleted.

Building materials

With the new functions **Building material along sketch**, **Building material from cross-section** and **Building material from border** it is now possible to create general building materials in a drawing.

The operation of the functions is based on the corresponding functions for creating **Insulations**.



Article numbers for foils

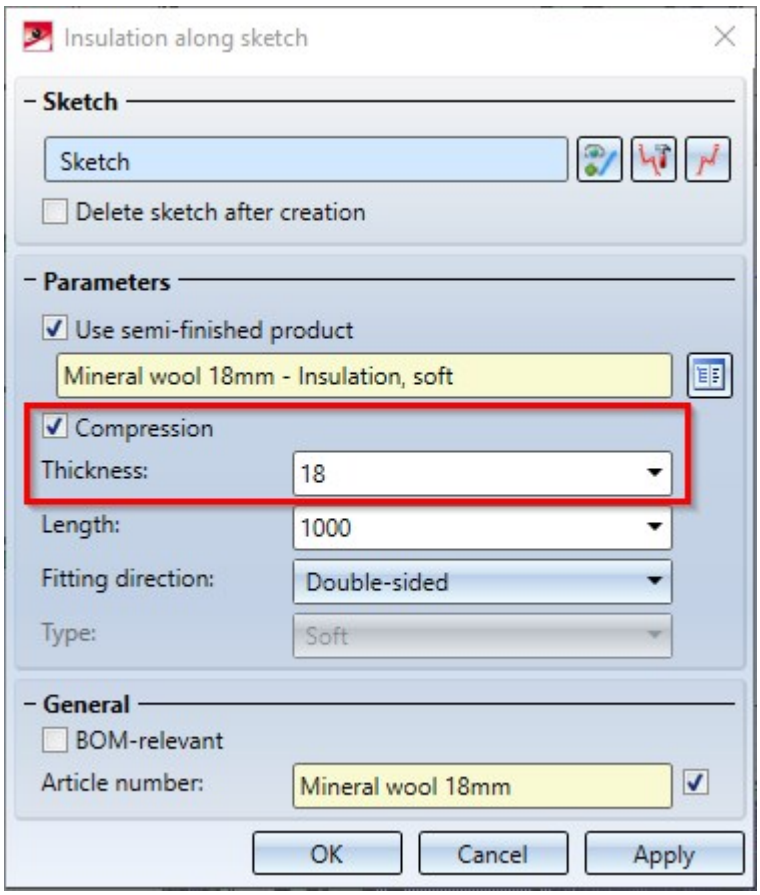
In the **Foil** dialogue you can now influence the **Article number** of the created foil. You can either enter your own article number or, if you use a semi-finished product from the catalogue, take the article number from the catalogue.

Insulations along c-edges

For the installation of an **Insulation** based on a cross-section, in addition to using a sketch, you now have the option of automatically deriving a contour from a section view. To do this, instead of selecting a sketch, you can instead identify a point in the section view. Starting from this point, a closed contour is then searched for.

Compressed insulations

With the functions **Insulation along sketch** and **Insulation from border**, it is now possible when using a semi-finished product to compress it to a different thickness. An example of this would be if insulation material with a width of 60 mm is to fit into a gap with a width of 55 mm.



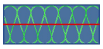
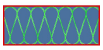
Major Release 2021 (V 2600)

Renamed Civil Engineering function

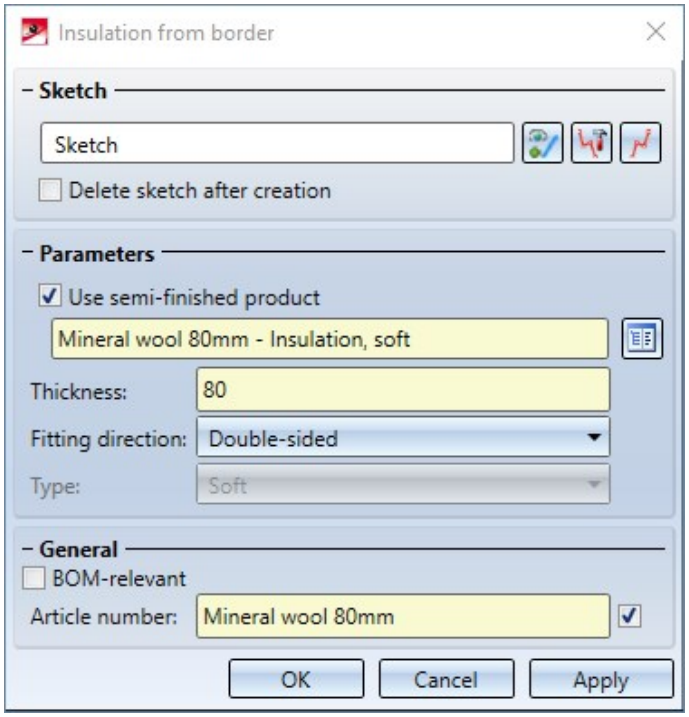
In the **Civil Engineering functions** docking window, the path **Civil Engineering, general > Civil Engineering - Sectional views (3-D)** has been renamed to **Civil Engineering, general > Detail and section planning**.

New Insulation function

The function for creating insulation has been split into three sub-functions:

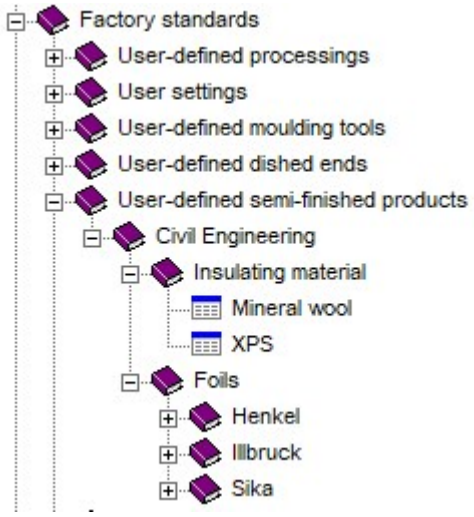
-  **Insulation along sketch**
-  **Insulation from cross-section**
- **Insulation from border**

You can find this function in the **Civil Engineering functions** docking window at **Civil Engineering, general > Detail and section planning**. A sketch serves as the basis for creating the insulation. For **...along sketch**, this sketch must contain a tangential, open composite edge. The other two functions require a closed composite edge. In the dialogue window you can either use a semi-finished product from the catalogue or enter the parameters of the insulation manually.



Changed catalogue structure for building materials

From HiCAD 2021 onwards, semi-finished products for civil engineering, such as foils or insulations, can be found in the catalogue **Factory standards > User-defined semi-finished products**.

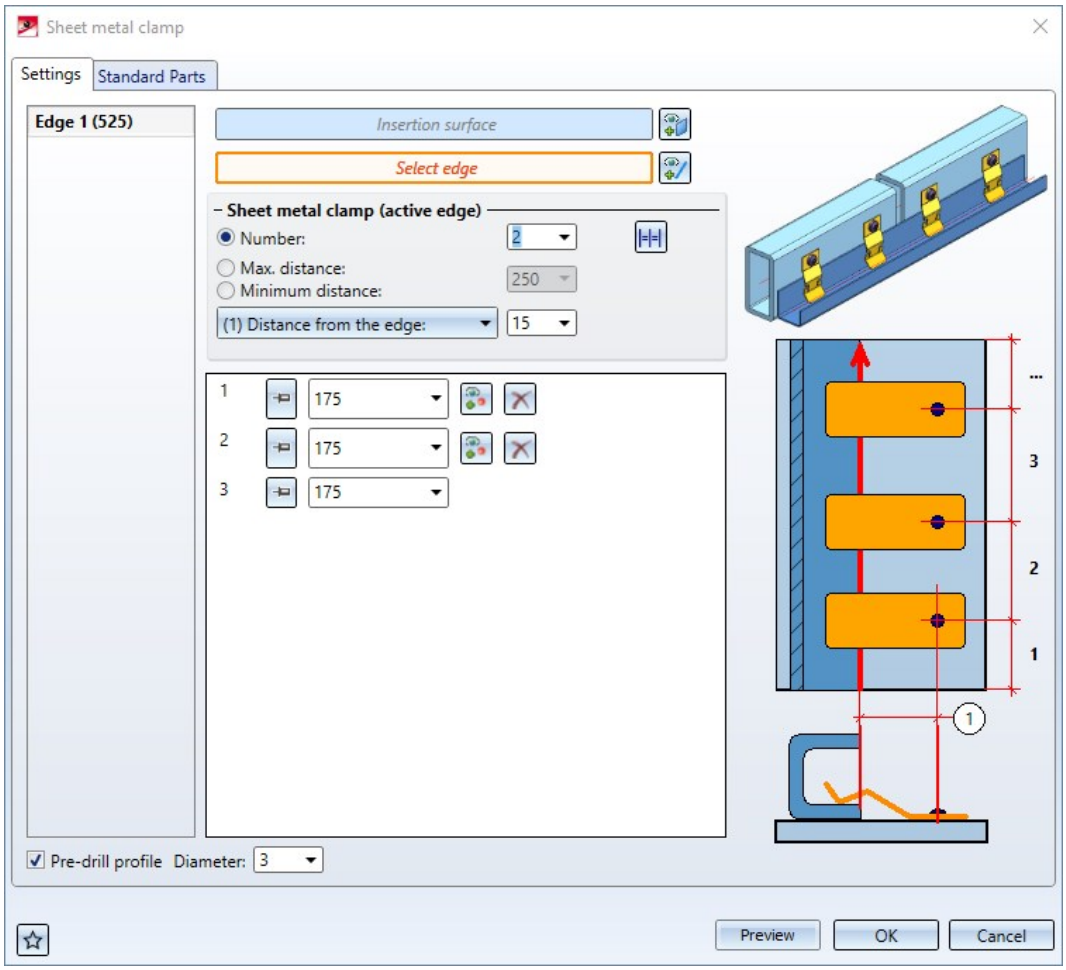


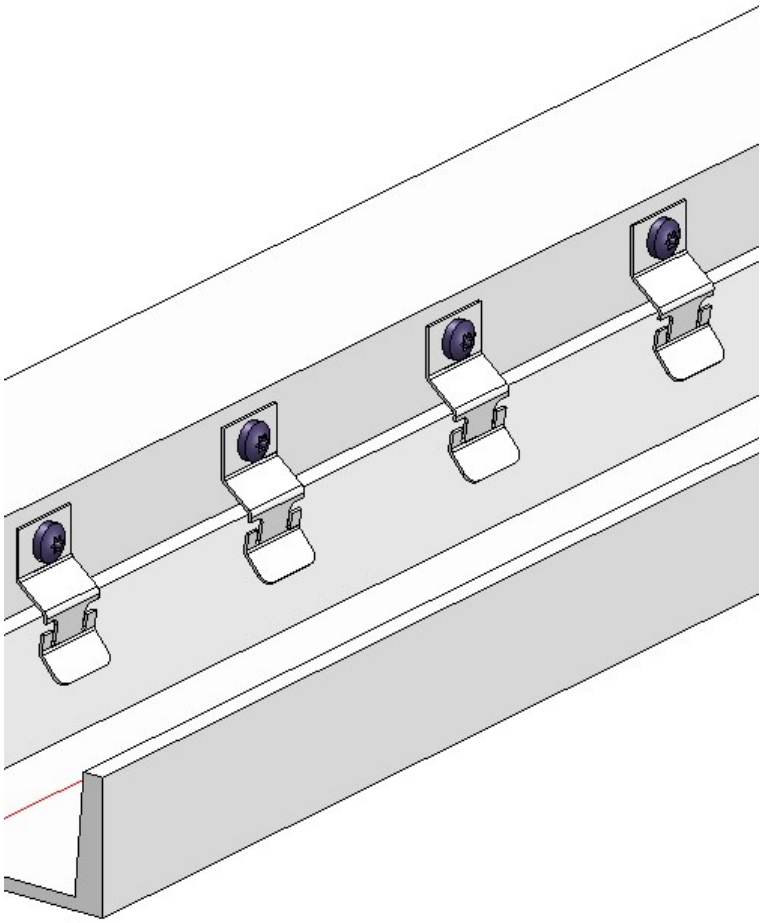
Glass insertion in non-closed glass fields

With the **Laminated glass** function, it is now possible to install glass in non-closed glass fields. For this purpose, either a point in the glass field can be identified or, after clicking the right mouse button and selecting the option Select glass profiles directly, two glass profiles can be selected, which may also lie parallel.

Sheet metal clamps

The new **Sheet metal clamp** function allows you to connect two or more profiles with the help of sheet metal clamps. In practice, this method is often used to mount cladding sheets to profiles.



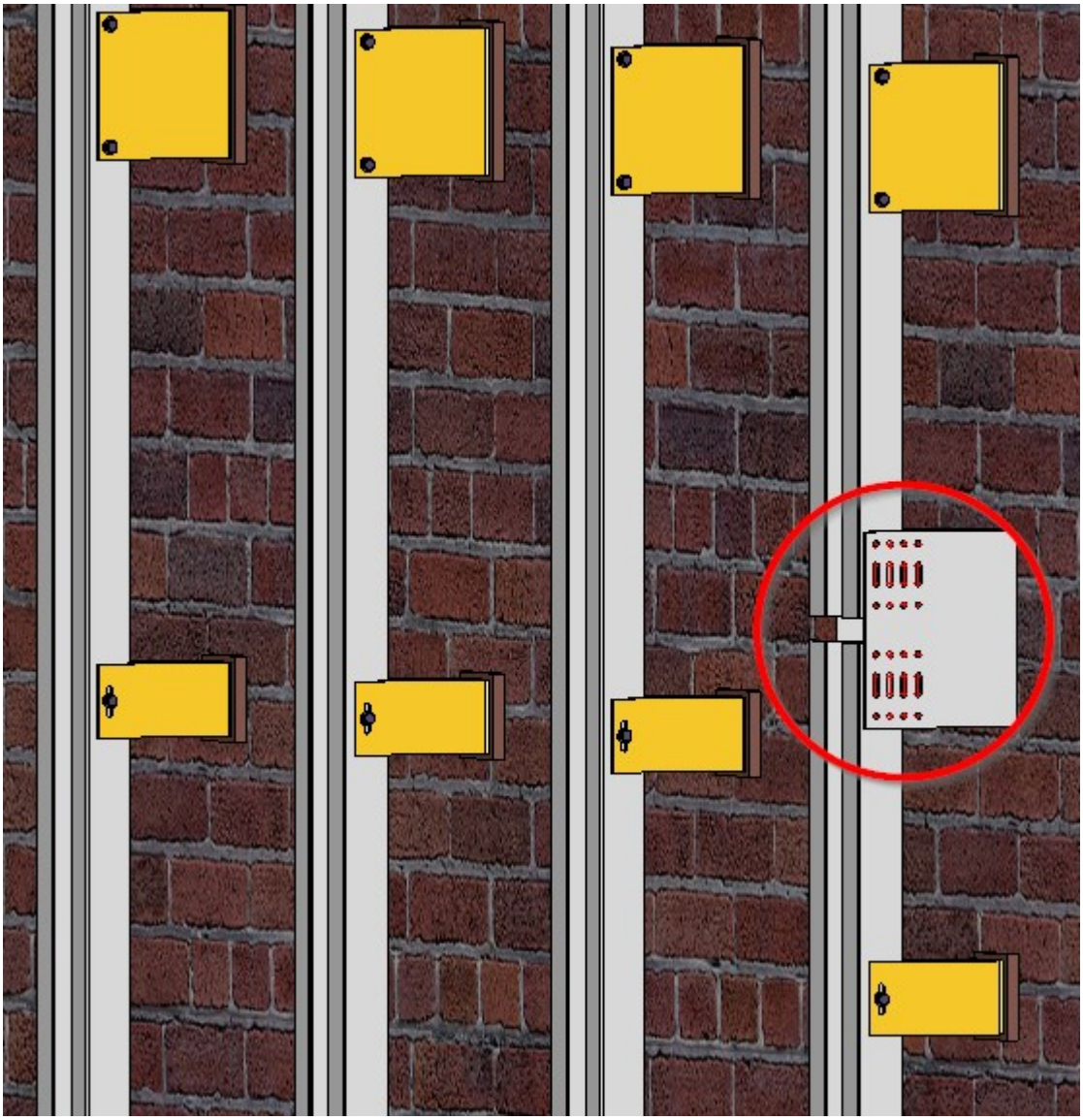


Element Installation

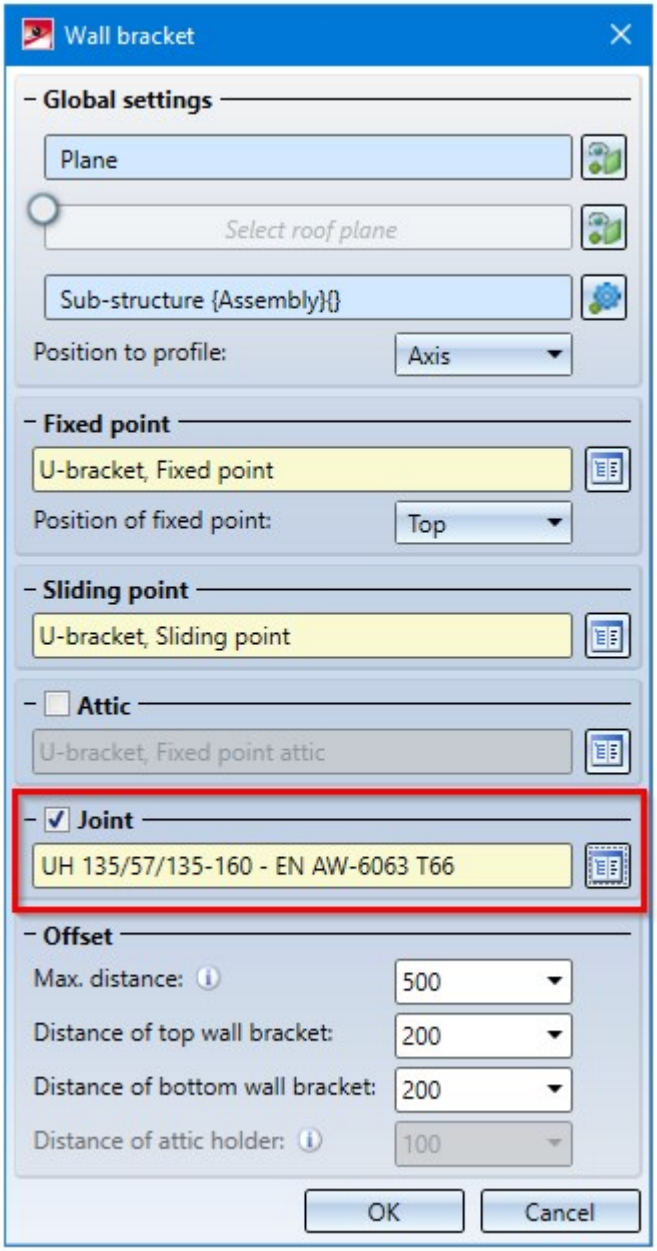
Service Pack 1 2021 (V 2601)

Wall brackets for joints

In the **Wall bracket** function, it is now possible to assign specific wall brackets for joints. These are installed via the joint and taken into account in the distance calculations.



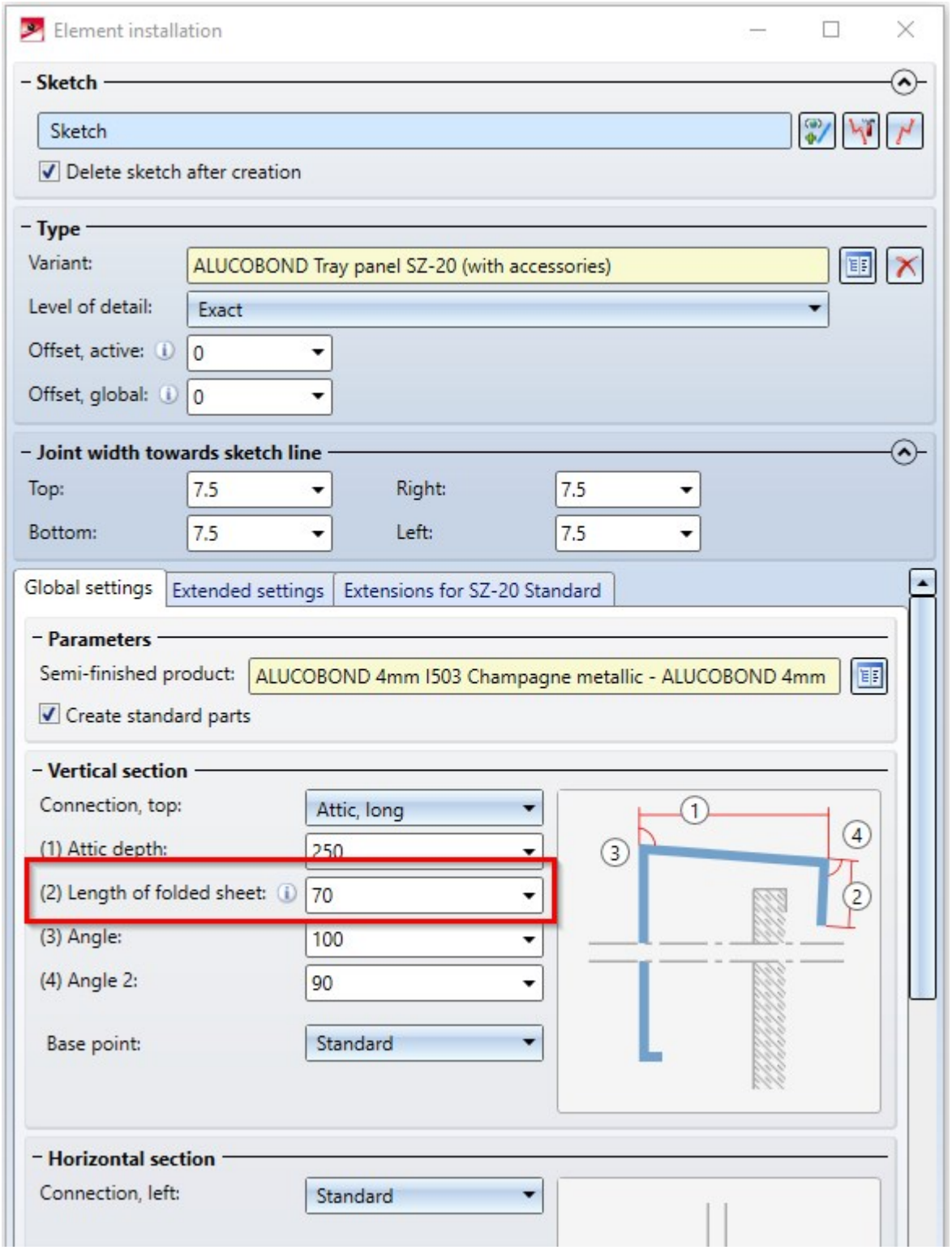
To do this, you can select the desired wall bracket from the catalogue in the **Wall bracket** dialogue window under **Joint**. If the **Joint** checkbox is not activated, joints are not taken into account and the function behaves as before.



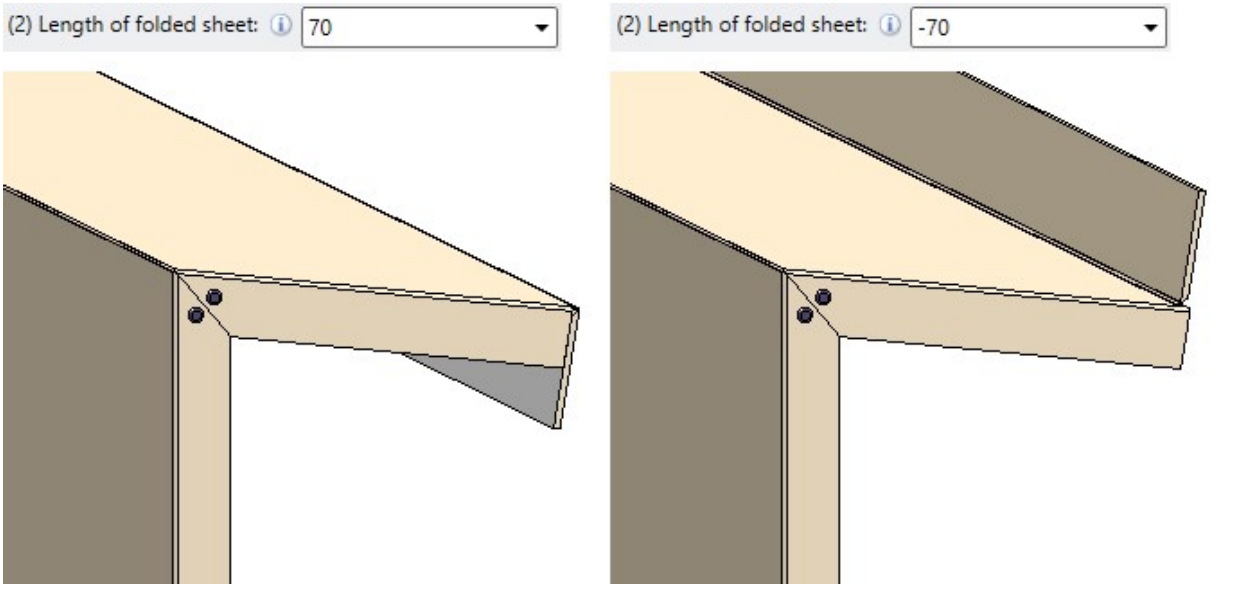
ALUCOBOND panels with flanges folded upwards

For selected ALUCOBOND panels, it is now possible to have the rearmost flange bent upwards as an alternative for some connections. The following panel types and connections are affected by this:

- ALUCOBOND SZ 20
  - Attic, long
  - Attic, short
  - Window connection, Parapet
- ALUCOBOND suspended
  - Attic, long
  - Window connection
- ALUCOBOND bonded
  - Attic, long
  - Attic, short
- ALUCOBOND riveted
  - Attic, long
  - Attic, short


Element installation
Sketch
Sketch
Delete sketch after creation
Type
Variant:
ALUCOBOND Tray panel SZ-20 (with accessories)
Level of detail:
Exact
Offset, active:
0
Offset, global:
0
Joint width towards sketch line
Top:
7.5
Right:
7.5
Bottom:
7.5
Left:
7.5
Global settings
Extended settings
Extensions for SZ-20 Standard
Parameters
Semi-finished product:
ALUCOBOND 4mm I503 Champagne metallic - ALUCOBOND 4mm
Create standard parts
Vertical section
Connection, top:
Attic, long
(1) Attic depth:
250
(2) Length of folded sheet:
70
(3) Angle:
100
(4) Angle 2:
90
Base point:
Standard
1
2
3
4
Horizontal section
Connection, left:
Standard

For this purpose, a negative value can be entered in the option (2) Length of folded sheet when creating the element installation for the mentioned connections. If the value 0 is entered, no flange is created.



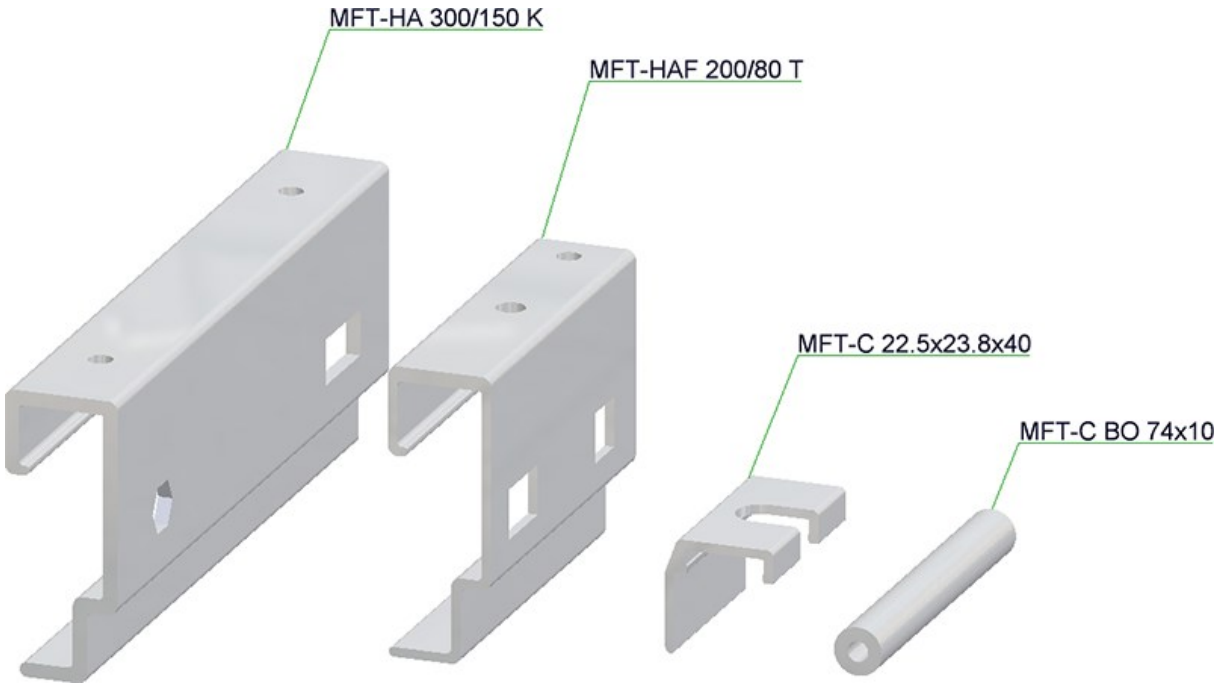
Example - Left: Regular folding; Right: Folding with negative value

Major Release 2021 (V 2600)

New Hilti accessory parts for sub-structures

Further Hilti accessory parts have been added to the catalogue **Factory standards > Purchased/Factory standard parts** :

Catalogue	Table	
Agraffes> Hilti > MFT-H 300	■ MFT-H 300 K ■ MFT-H 300 RL92 ■ MFT-H 300 T	Agraffes for undercut fixing with Hilti MFT-HP 300 profiles
Agraffes > Hilti > MFT-H 200	■ MFT-H 200 K ■ MFT-H 200 RL92 ■ MFT-H 200 T	Agraffes for undercut fixing withHilti MFT-HP 200 profiles
Clamps > Hilti	■ Clip MFT-C ■ Facade studs MFT-C BO	Fixing clips and studs for suspended facades with Hilti MFT-CP profiles



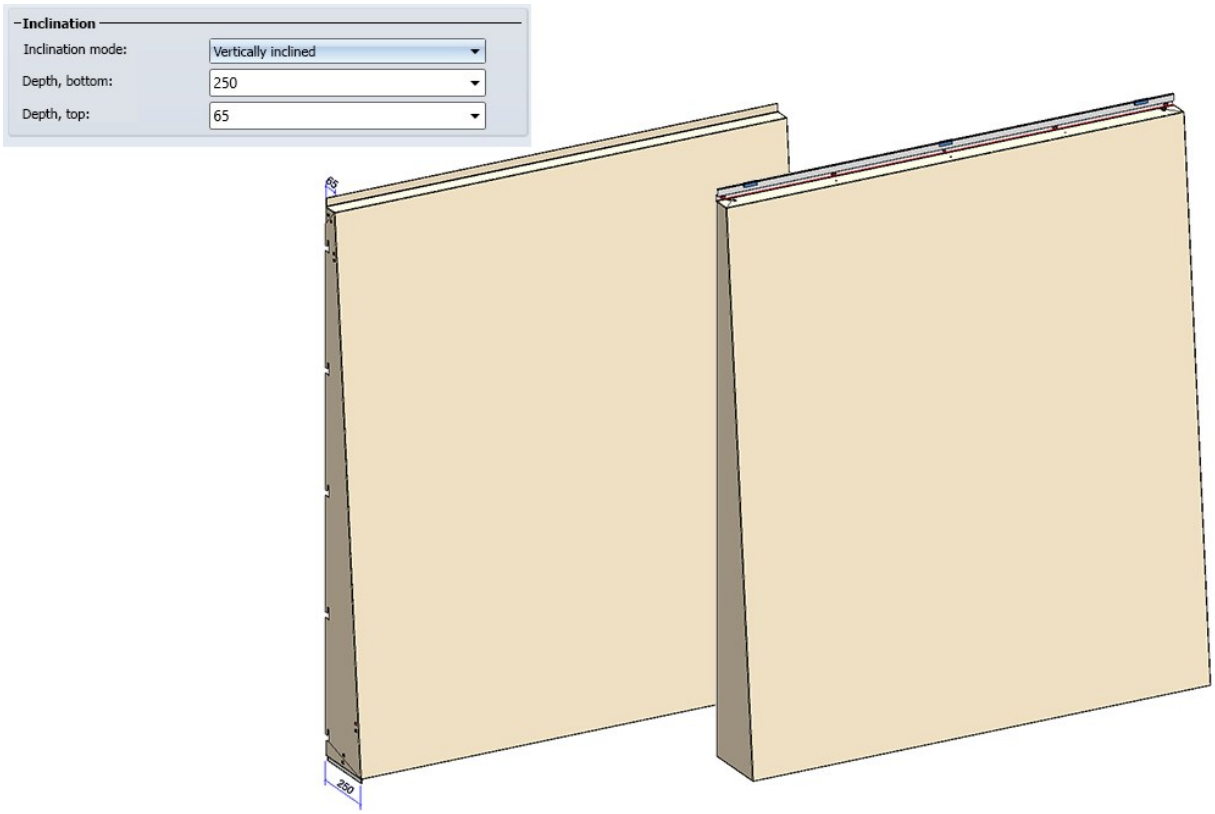
ALUCOBOND® installation elements

ALUCOBOND® suspended and SZ20

Until now, the base surface of suspended panels and SZ-20 panels was always parallel to the sketch plane. As of HiCAD 2021, the panels can be inclined horizontally or vertically - but only if the **Standard** or **Without flanges** type

is selected for the **Horizontal/Vertical section**.

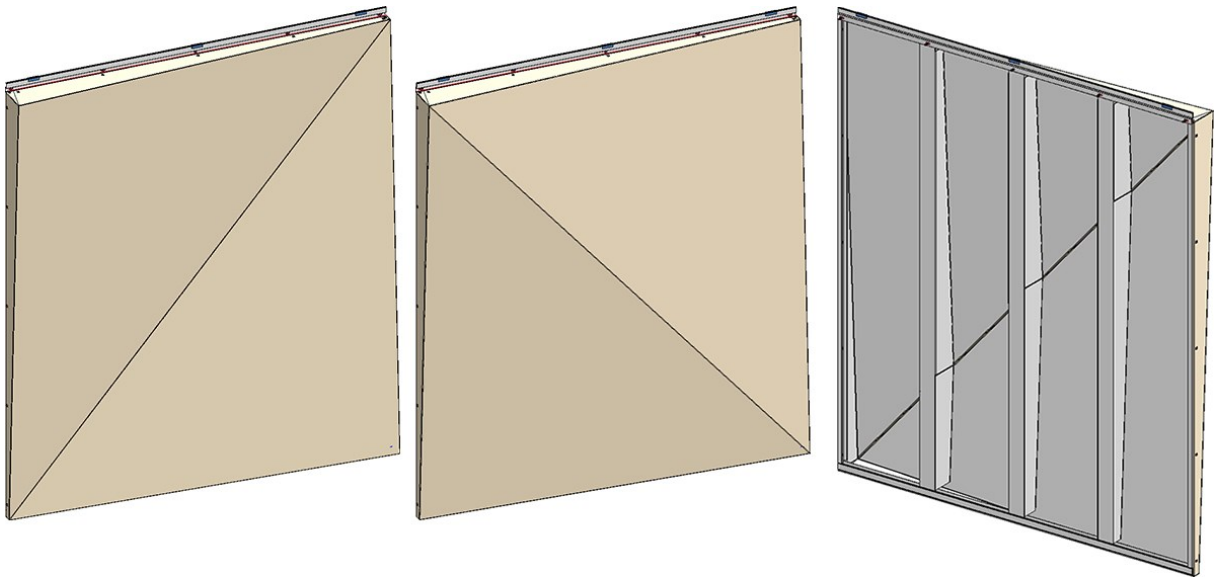
The **Extended settings** tab has been extended accordingly.



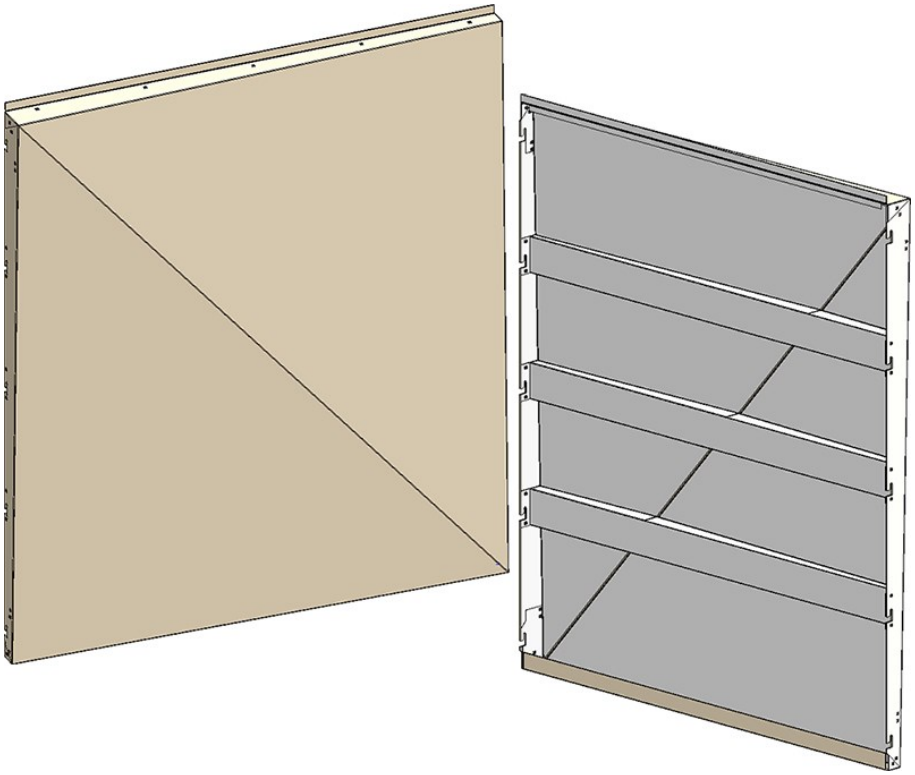
ALUCOBOND® Delta 2

ALUCOBOND® SZ-20 panels and suspended ALUCOBOND® panels can also be installed with a spatially folded base surface. Two new variants will be available for this from HiCAD 2021 onwards:

- ALUCOBOND® Delta 2 SZ20 and
- ALUCOBOND® Delta 2 suspended.



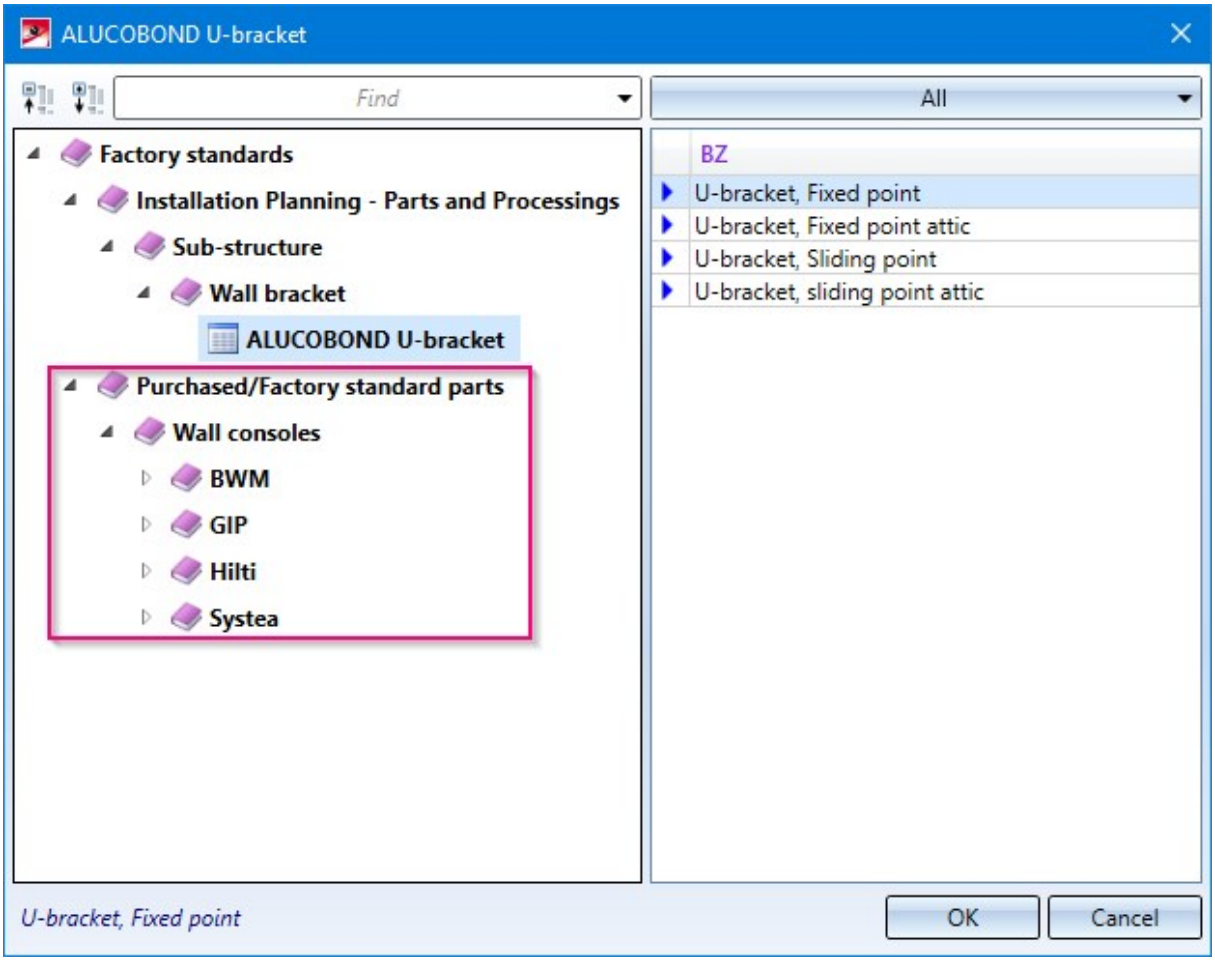
ALUCOBOND® Delta 2 SZ20



ALUCOBOND® Delta 2 suspended

Wall brackets: Purchased parts can be used

When creating wall brackets, it is now no longer only possible to use wall brackets from the catalogue path **Factory standards > Installation Planning - Parts and Processings > Sub-structure > Wall bracket**, but also from the path **Factory standards > Purchased/Factory standard parts > Wall consoles**.



Profile Installation

Service Pack 1 2021 (V 2601)

Orientation of tray panels

Tray panels for **Profile Installation** were previously aligned according to the manufacturer's drawings. In some cases this led to these tray panels being installed the wrong way round. This has been corrected in SP1 so that the following tray panels are now stored in the catalogue correctly aligned:

- Factory standards\Series\Roof Wall Facade\Room-closing profiles\Fischer\Liner trays
- Factory standards\Series\Roof Wall Facade\Room-closing profiles\Montana\Liner trays
- Factory standards\Series\Roof Wall Facade\Room-closing profiles\Münker\Liner trays
- Factory standards\Series\Roof Wall Facade\Room-closing profiles\Profiltec Bausysteme\Liner trays
- Factory standards\Series\Roof Wall Facade\Room-closing profiles\SAB\Liner trays \Liner trays IJsselstein
- Factory standards\Series\Roof Wall Facade\Room-closing profiles\SAB\Liner trays \Liner trays Niederaula
- Factory standards\Series\Roof Wall Facade\Room-closing profiles\Salzgitter Bauelemente\Liner trays

These changes only affect newly created profile installations. Existing profile installations remain in their original state even after recalculation.

If you want to change from the old to the new tray panels in an "old" profile installation, edit the profile installation and select the desired tray panel from the catalogue.

Attribute for uncut surface areas

In addition to the attribute `DWF_PROFILE_AREA` which contains the surface area of the cut profile, profiles created with the **Profile installation** function now also contain the attribute `DWF_UNCUT_PROFILE_AREA`, which contains the uncut area of a profile.

This corresponds to the formula "Construction width x Length".

Major Release 2021 (V 2600)

Rectified Salzgitter catalogue entries

The catalogue entries for Salzgitter profiles contained errors: The tables Sandwich panels, roof and Sandwich panels, wall from Salzgitter (to be found in the catalogue at the path Factory standards > Series > Roof Wall Facade > Room-closing profiles > Salzgitter Building elements) contained incorrect weights and U-values. This has now been corrected.

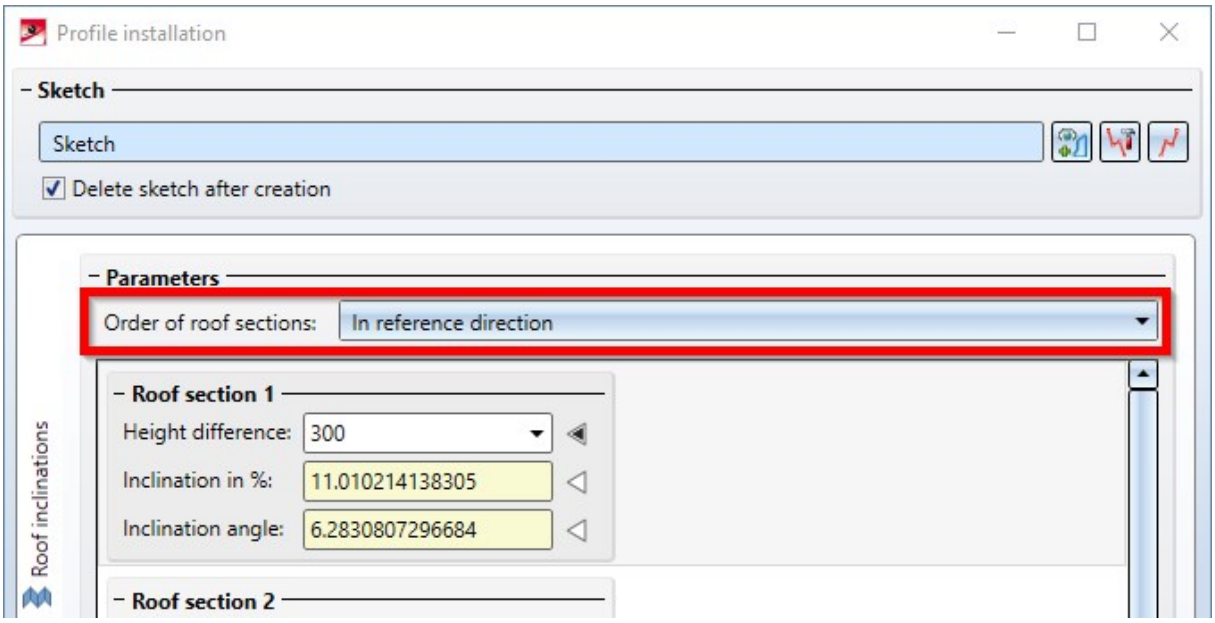
Profiles that have already been installed are not automatically corrected; instead, they must be updated manually.

- For manually installed profiles, recalculation is not sufficient. Instead, the profile type must first be changed to another profile and then back again.
- For profiles that have been installed via the Profile Installation, such a change is not sufficient. Instead, at least the layer on which the profiles are installed must be deleted and recreated. However, it is more advisable to recreate the entire profile installation.

Changing the order of the roof sections for roof inclinations

When creating a profile installation with roof inclinations, the individual roof sections are always numbered in the direction of the Y-axis of the sketch. The only exception to this is when the edges of the roof inclinations are exactly parallel to the Y-axis. In this case, the sections are numbered in the direction of the X-axis of the sketch.

In the **Roof inclinations** area of the **Profile installation** window it is now possible to reverse the numbering of the sections. To do this, the option **Order of roof sections** is available, which offers the choices **In reference direction** (corresponds to the previous numbering in the Y- (or X-) axis direction of the sketch) and **In opposite direction**, which numbers the sections in the opposite direction to the Y/X direction.

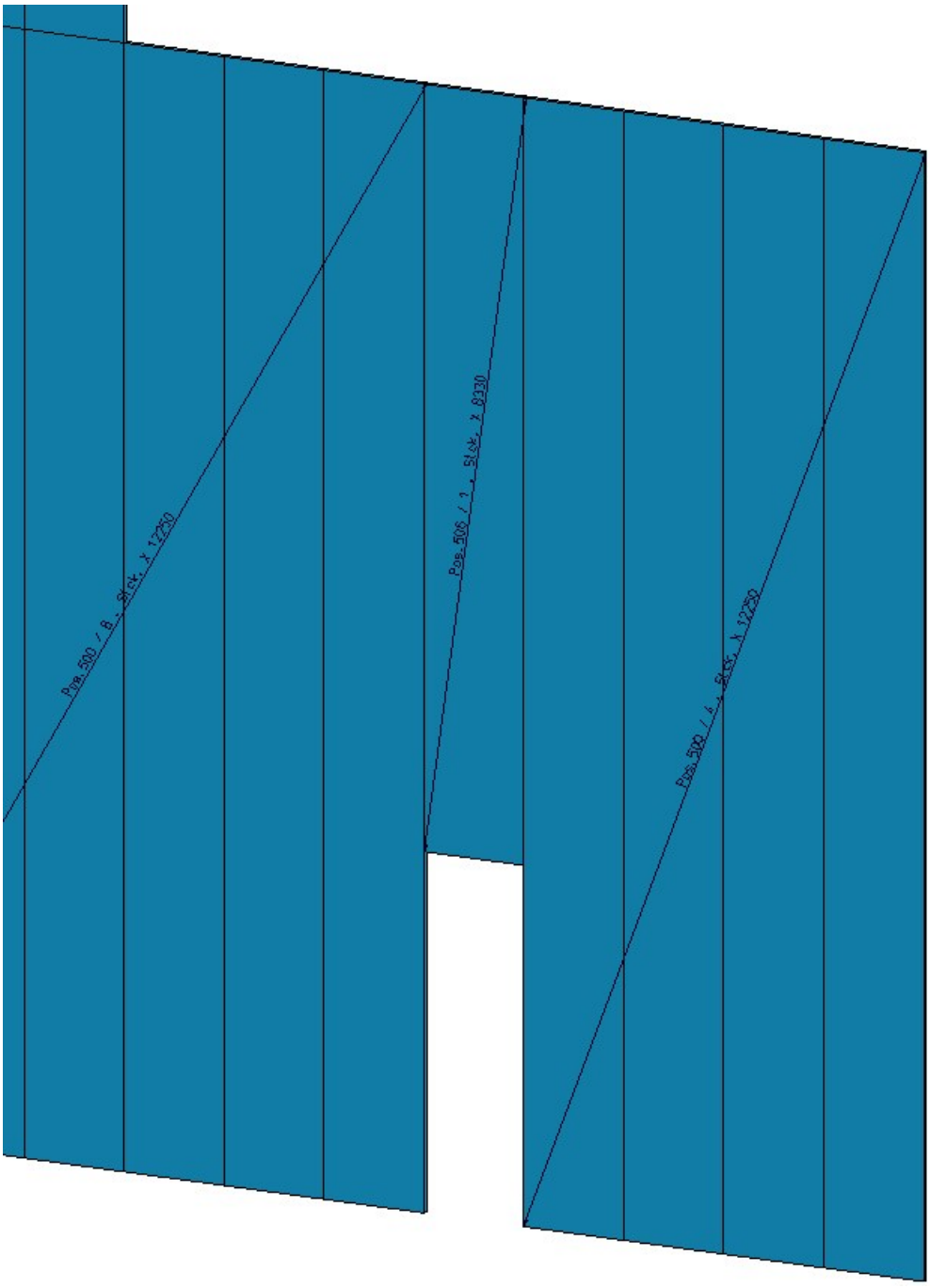


This distinction is useful because the beginning of the first roof section is always at the height of the sketch plane and all further surfaces measure their inclination or height from this value.

Display of profile designations in the drawing

Until now, it was only possible to obtain the industry-standard display of profile groups consisting of profiles with the same length and from the same package by creating workshop drawings. Now the new view tools **Add designations** and **Delete designations** have been added, which display these designations for the currently selected layer directly in the drawing, or remove this display again.

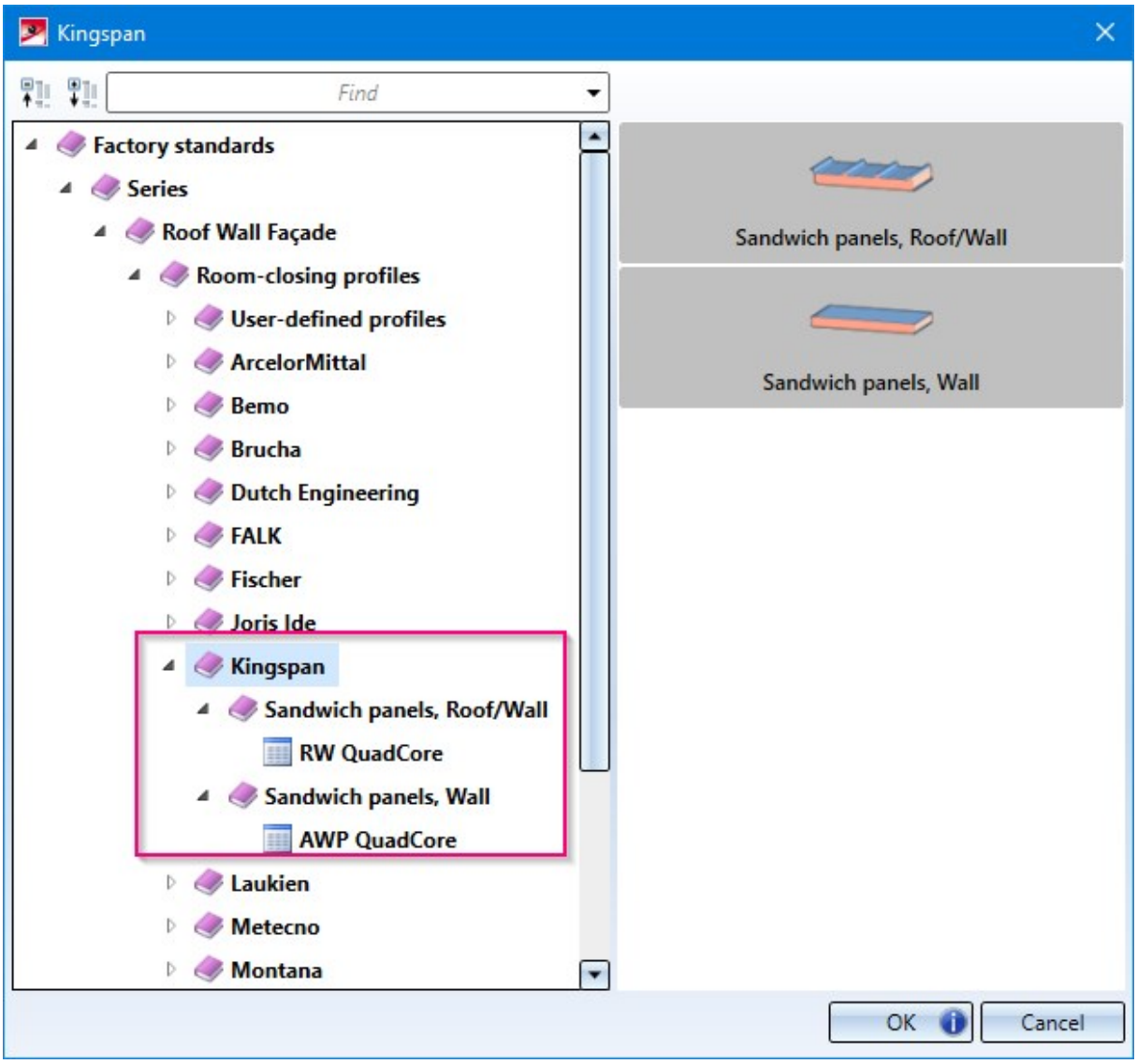
This display only takes place in the view mode **Contour, uncut**. If another view mode is active when the**Add designations** function is called, the system automatically switches to the **Contour, uncut** mode.



Kingspan Sandwich elements

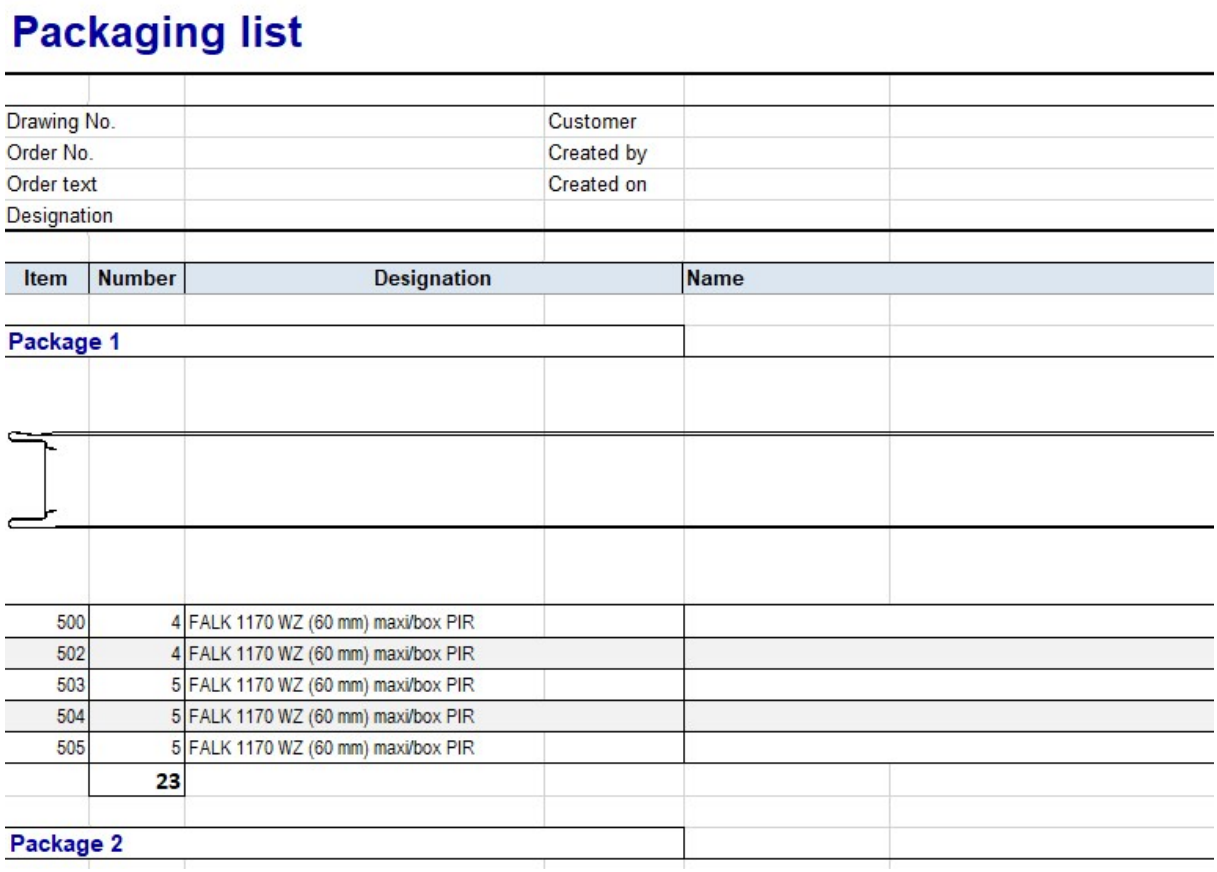
The catalogue **Factory standards > Roof Wall Facade > Room-closing profiles** now contains the following new sandwich elements for roofs and walls by the company Kingspan:

- **Kingspan >Sandwich panels, Roof/Wall > RW QuadCore** and
- **Kingspan >Sandwich panels, Wall > AWP QuadCore**.



Packaging list: Profile cross-sections in Excel export

When exporting the packaging list to an Excel spreadsheet via the **Report Manager**, the profile cross-sections of the profiles concerned are now shown on the Packaging list spreadsheet.



Packaging list

Drawing No.		Customer	
Order No.		Created by	
Order text		Created on	
Designation			

Item	Number	Designation	Name
Package 1			
500	4	FALK 1170 WZ (60 mm) maxi/box PIR	
502	4	FALK 1170 WZ (60 mm) maxi/box PIR	
503	5	FALK 1170 WZ (60 mm) maxi/box PIR	
504	5	FALK 1170 WZ (60 mm) maxi/box PIR	
505	5	FALK 1170 WZ (60 mm) maxi/box PIR	
	23		
Package 2			

For this you need to use the Report Manager configuration HICAD_STAHLBAU.2600.0.

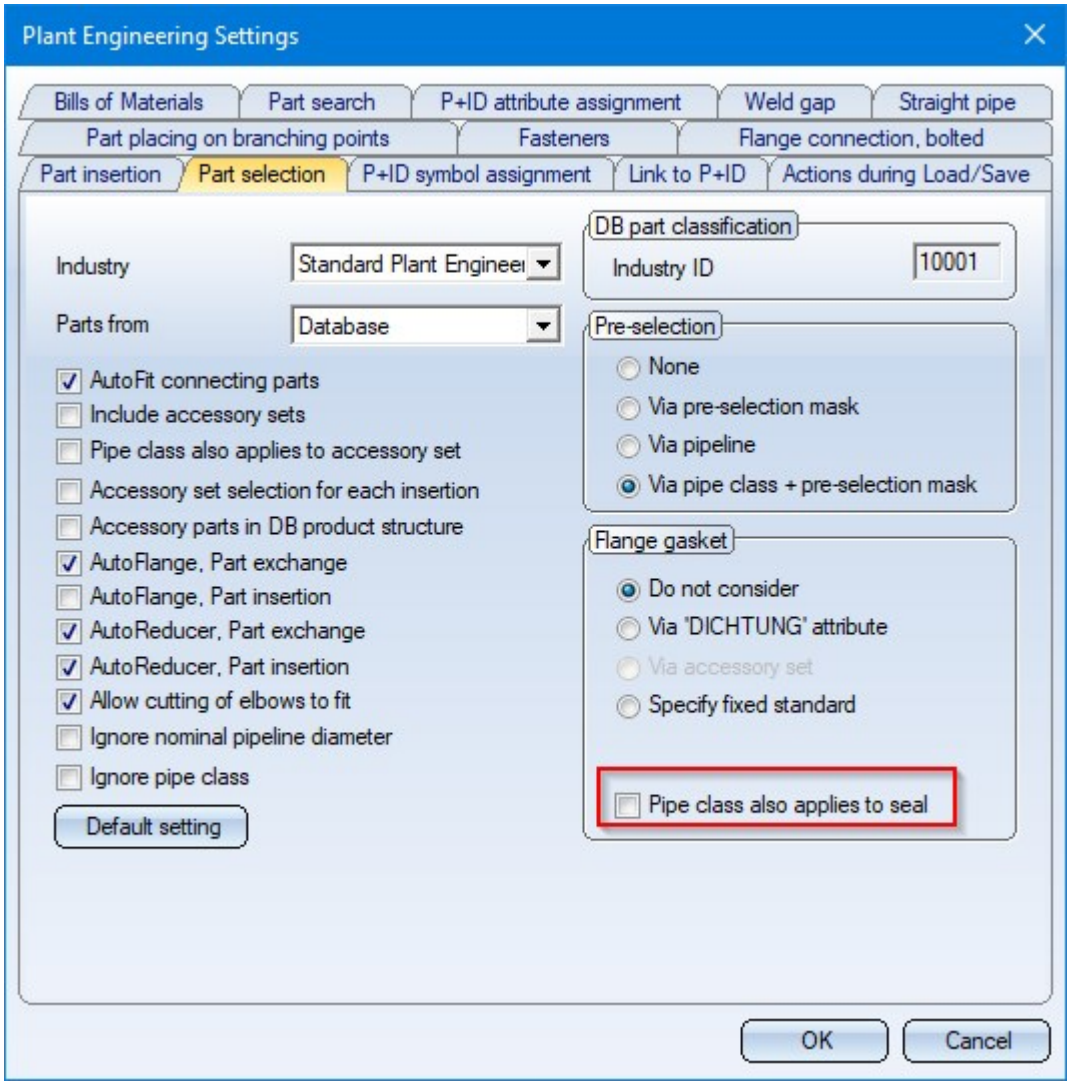
Plant Engineering

Service Pack 1 2021 (V 2601)

Plant Engineering Settings - Part selection

Part selection

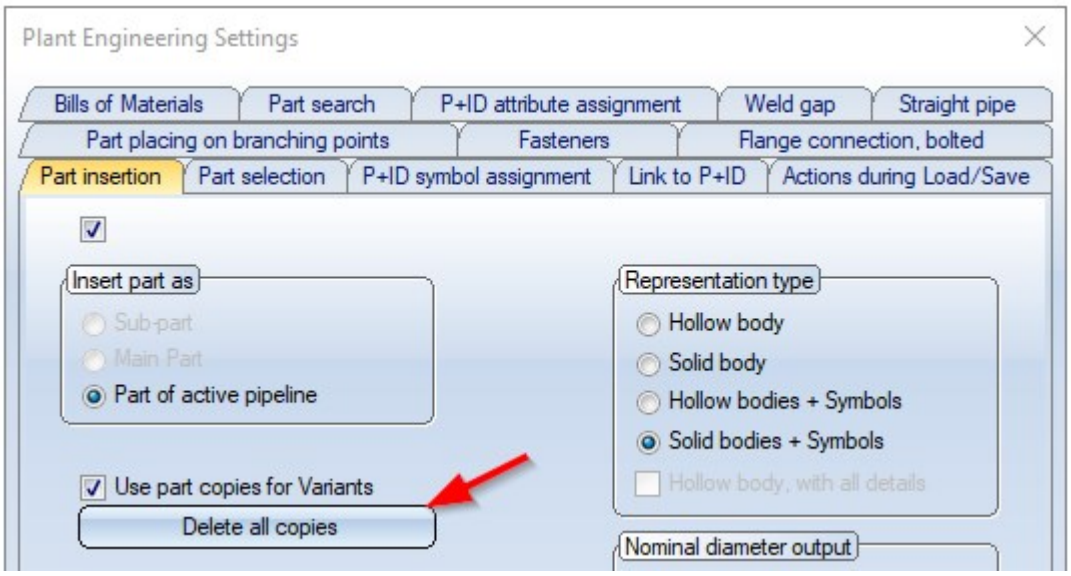
In the settings for **Part selection** under **Flange gasket**, the **Pipe class also applies to seal** checkbox used to be available only if the **Do not consider** option was activated. As of HiCAD 2021 SP1, the checkbox is always available.



Part insertion - Part copies

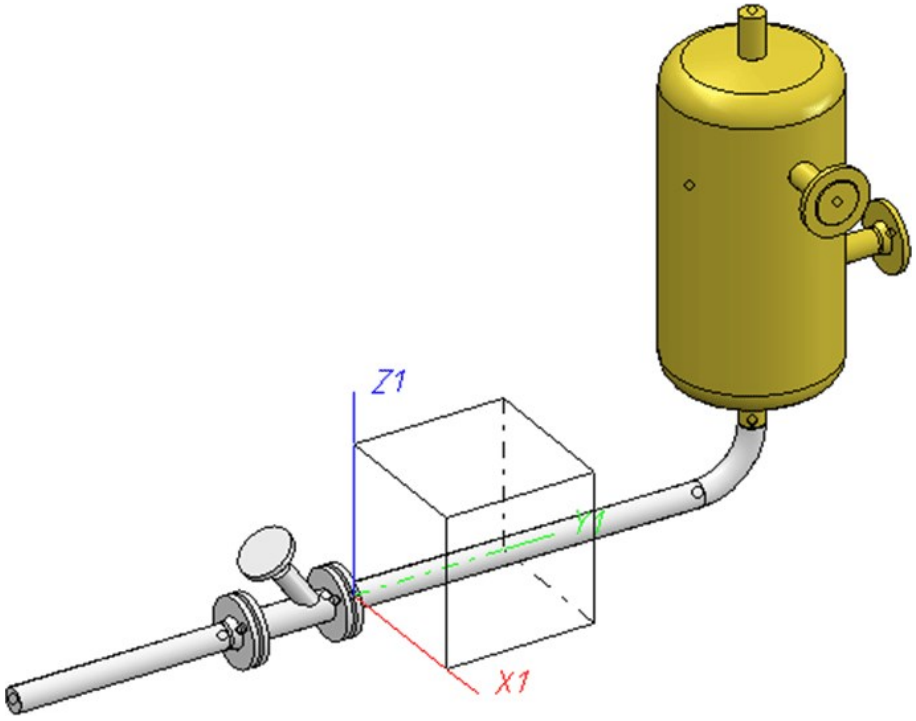
With HiCAD 2021 SP1 the way the variants are stored in the cache has changed.

If you have previously used part copies for pipe parts in variants to speed up installation, you must click the **Delete all copies** button once in the **Part insertion** tab of the **Plant Engineering Settings** dialogue window.



Elbows - Insertion options

The "Adjust to displacement" insertion type is no longer available for elbows in the **Pipe parts** function. This insertion type was usable for the following and comparable situations:

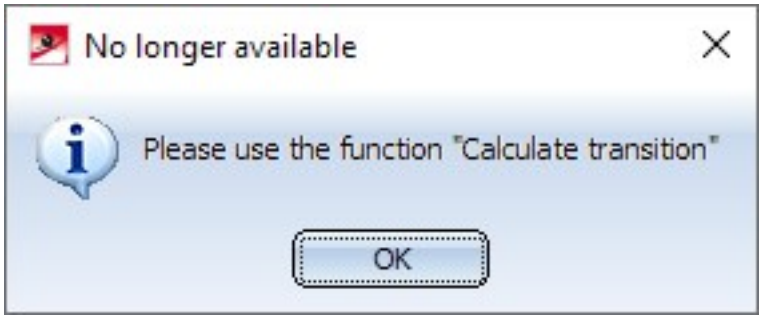


- The straight pipe runs in the Y-direction, the vessel is perpendicular to the XY-plane.
- Now the vessel is to be displaced a little in the X direction.

As of HiCAD 2021 SP1, the **Calculate transition** function, which has been expanded accordingly, must be used in such cases.

Please note:

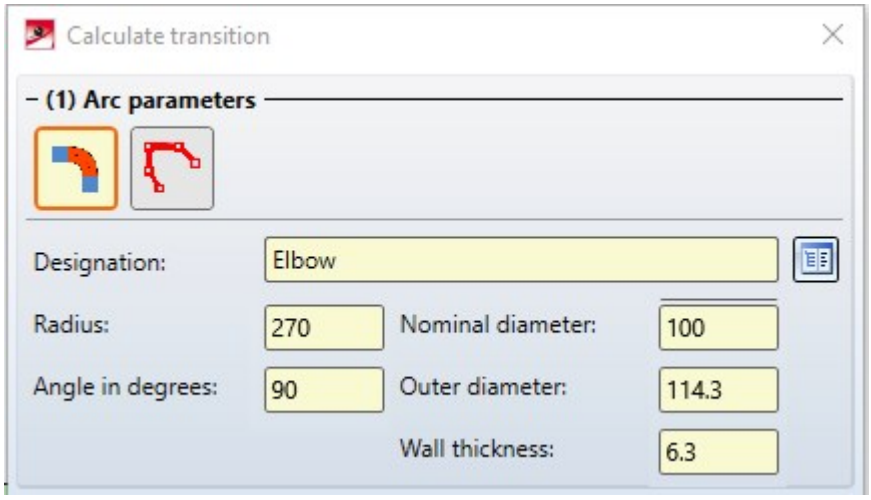
The parts and insertion types provided by the **Pipe parts** function are defined in the file **rohrbt_list2.dat** in the HiCAD sub-directory **makroanl**. If you have adjusted this file manually, then the insertion type "Adjust to displacement" may still be available after an update installation. In this case, the following message is output when this insertion type is selected.



Calculate transition

Select elbows

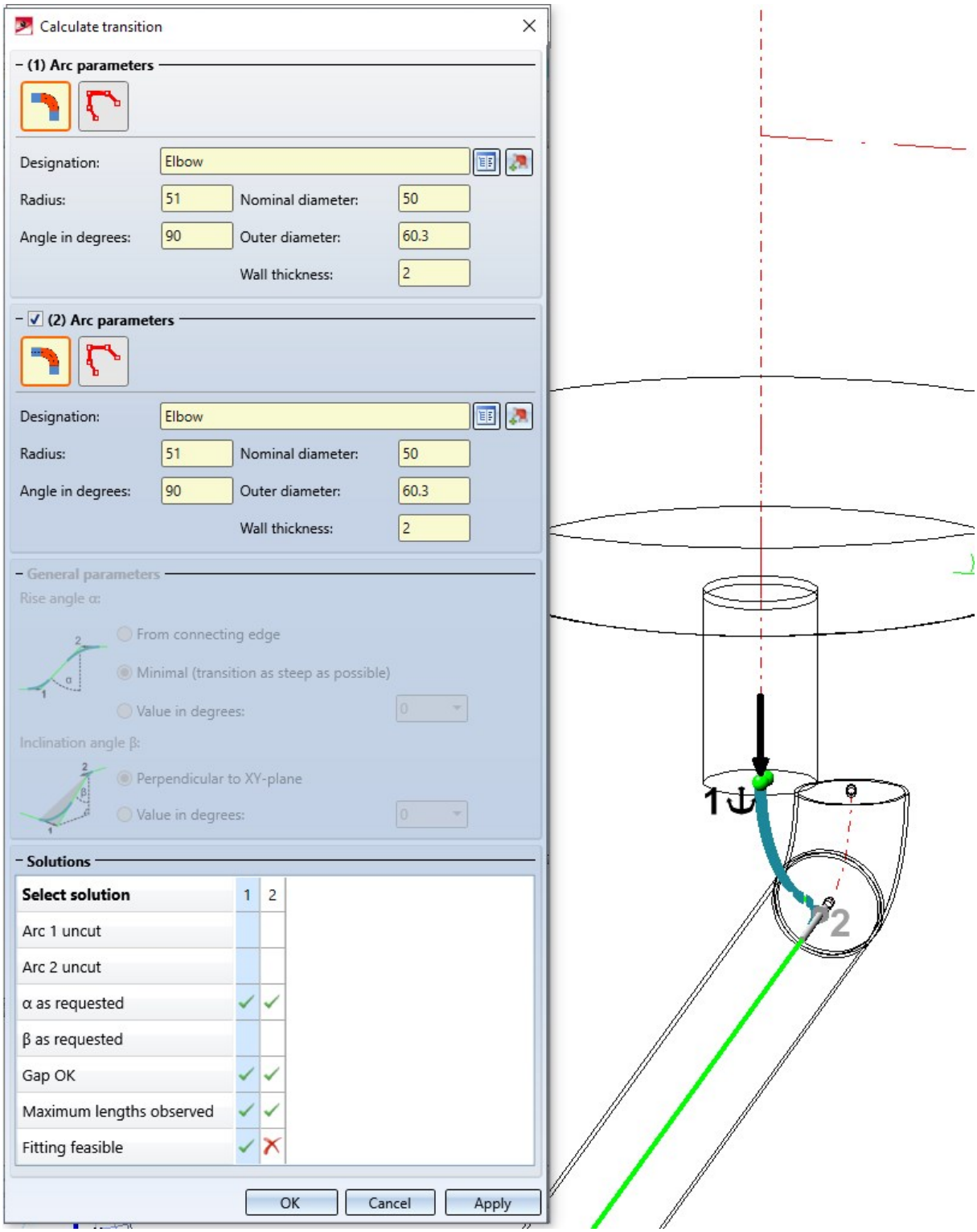
With the **Calculate transition** function, the elbows to be set no longer have to be defined via a component search, but can also be selected directly in the construction. For this purpose the button is available in the dialogue window. When you click this button, this type of arc selection is active and the symbol display changes to . HiCAD then prompts you to select the desired elbow in the drawing. After selecting an elbow, its parameters are displayed in the dialogue window.



The elbow selection can be cancelled by clicking on the symbol again.

Fixed connecting points

Transitions are calculated between connecting points, i.e. the connecting points are connected by a transition. If the connecting points can be moved along the direction of the connection, this can be influenced by the rise angle α and the inclination angle β. In practice, however, it is sometimes necessary to calculate a transition where at least one of the connecting points is fixed. In particular, points on vessels are always considered fixed. If you select a connecting point that cannot be moved, it will be marked with an anchor as of HiCAD 2021 SP1 (in the example at Connection 1)):



Note that the **General parameters** area in the dialogue is greyed out as soon as a fixed connecting point is selected as the connection for the transition calculation.

For transitions whose connecting points are not fixed, additional solutions are now offered - compared to previous HiCAD versions - in which the connecting points are assumed to be fixed.

Adjust to displacement

In the **Pipe parts** function, the **Adjust to displacement** insertion type is no longer available for elbows since HiCAD 2021 SP1. This is now completely taken over by the **Calculate transition** function.

The **Adjust to displacement** installation type covered a concrete application: A component to which a pipe is connected is displaced by a few centimetres so that a gap is created between the connection of the component and the pipe.

You can now compensate for such a gap with the **Calculate transition** function by essentially following the example under Fixed connecting points, i.e.

1. Move the vessel to fit
2. Select **Calculate transition**
3. Click on **Select elbow** and select the elbow in your drawing
4. Select the connection on the vessel
5. Select the connection at the end of the pipe - but before the - now misaligned - elbow
6. Select a valid solution and confirm with **OK**

This solution is superior to the previous insertion type **Adjust to displacement**, especially if the pipeline has a downgrade. This case was not supported by the insertion type.

Elbow orientation from pipeline route

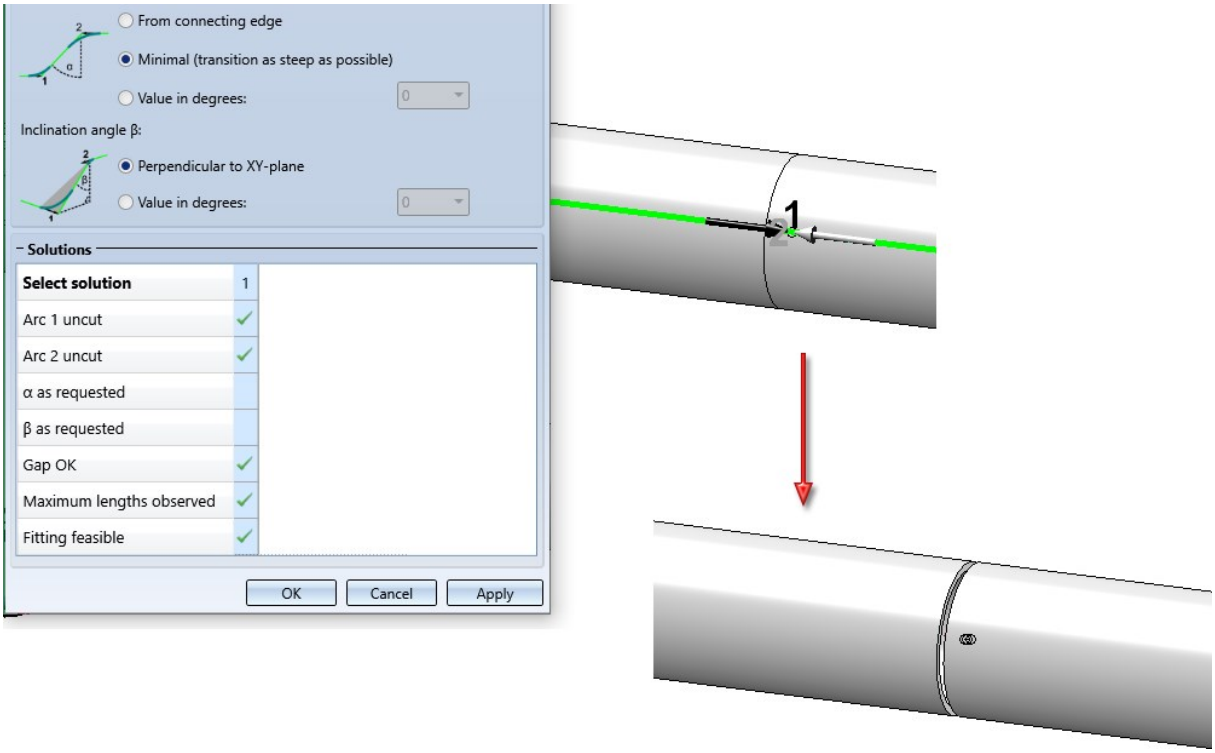
The **Calculate transition** function allows you to select connecting points where parts are already connected. Before HiCAD 2021 SP1, these parts were only removed when the transition is created.

As of HiCAD 2021 SP1, the connecting directions of the connected parts (and also guidelines) are examined. The first direction that deviates from the selected connection direction is now understood as an indication of the desired elbow orientation.

Additional suitable solutions are then calculated and offered.

Set weld seams subsequently

The **Calculate transition** function can also be used to set a weld seam gap subsequently, where the gap is set according to the settings for weld seams:



Create guideline as sketch

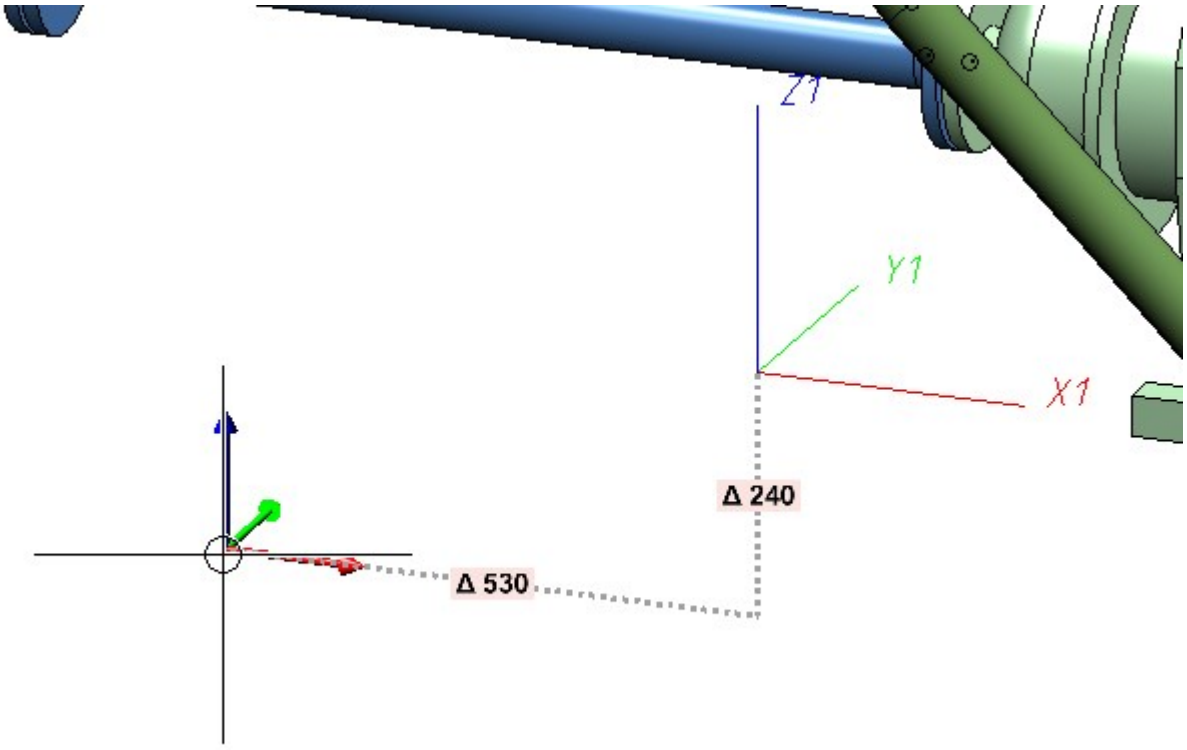
Guidelines have been created as **3-D sketches** in Plant Engineering since HiCAD 2021. The **Create guideline** function is used for this purpose.

3-D sketches require a part coordinate system to which, for example, the grid during sketching or possibly assigned HCM constraints refer. In general, when creating a 3-D sketch, it is necessary to specify a plane in which the sketch is to be drawn. This determines the part coordinate system of the 3-D sketch (also sketch coordinate system).

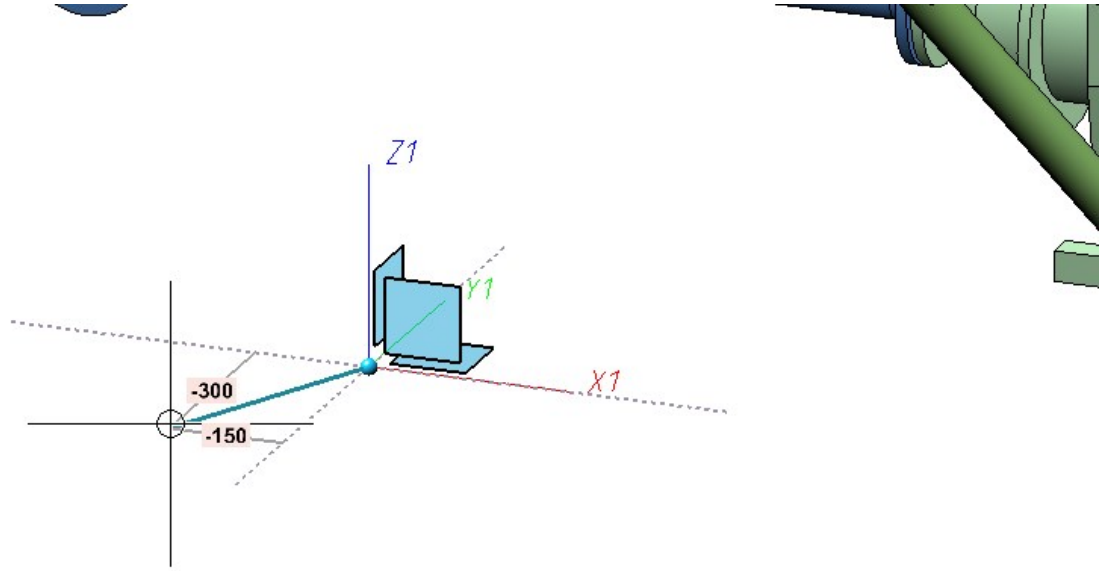
During pipeline planning, this step tends to inhibit the workflow, so as of SP1 - in contrast to 3-D sketches - this step can be bypassed for guideline sketches. Nevertheless, such a sketch coordinate system is also required for guideline sketches, since the origin of this coordinate system is used for the calculation of HCM constraints that relate the edges of a sketch to each other.

> Therefore, unlike the **3-D Sketch** function, when creating guideline sketches, the first point determined is also the origin of the sketch coordinate system.

When you call the **Create guideline** function, a coordinate system tripod is displayed at the cursor and its distance to the origin is additionally represented by auxiliary lines.

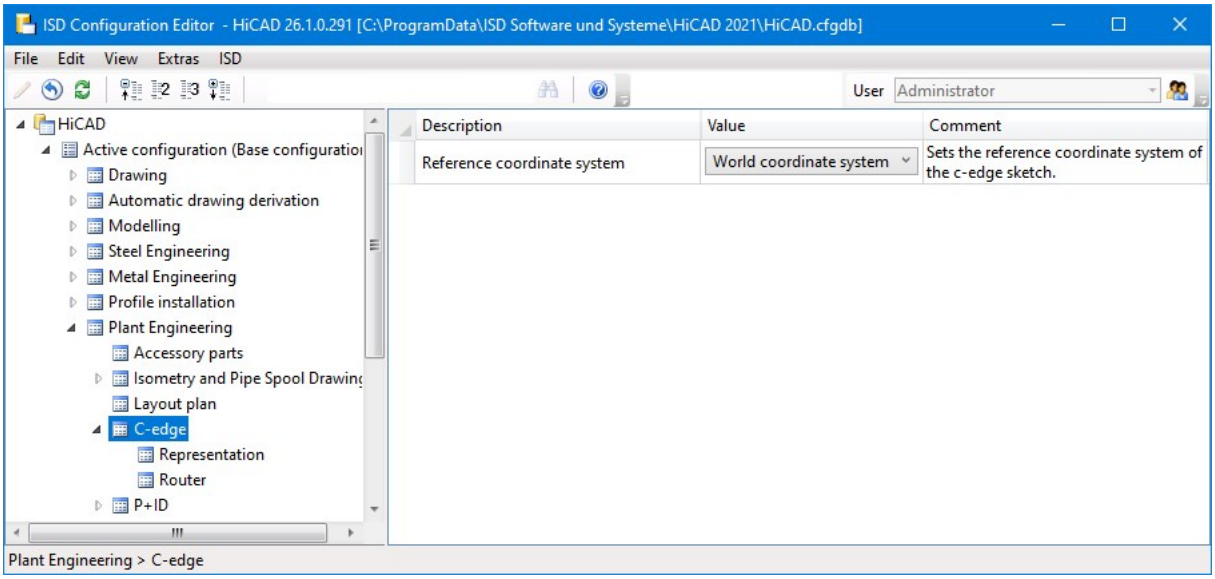


If you now select a point, the sketch coordinate system is the coordinate system previously indicated by the tripod at the cursor. Specifically, in the example above, the coordinate system tripod represents a moved version of the world coordinate system. Thus, the world coordinate system was used as the basis for the sketch coordinate system, with only its origin moved. **This behaviour can be changed in the Configuration Editor**.



After the determination of the first point, the usual drawing of the sketch begins.

The behaviour regarding the sketch coordinate system of a guideline sketch can be defined in the Configuration Editor at **Plant Engineering > C-edge** via the **Reference CS** parameter.

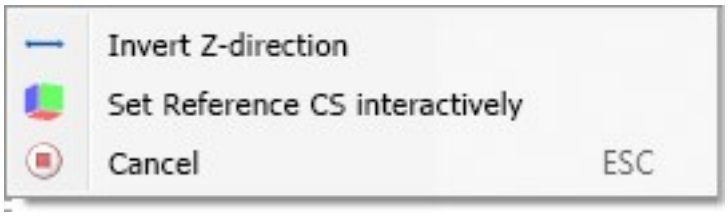


The following options are available:

- World CS
  The sketch coordinate system of a guideline sketch is always set to the world coordinate system, moved to the first point of the sketch. This is the default setting.
- Local CS
  The sketch coordinate system of a guideline sketch is always set to the local coordinate system, also moved to the first point of the sketch.
- Set interactively
  As when creating a regular 3-D sketch, you are prompted to define a sketch coordinate system interactively. So this option disables the use of the first sketch point as the coordinate origin.

For the **World CS** and **Local CS** options, pipe part connecting points are handled in a special way. If you select such a point as the first point of the sketch, the sketch coordinate system is selected in such a way that its Z-axis points out of the connection belonging to the connecting point.

If you have selected the **World CS** or **Local CS** options in the Configuration Editor, the following context menu is available to you when selecting the first sketch point (see Settings in the Configuration Editor):

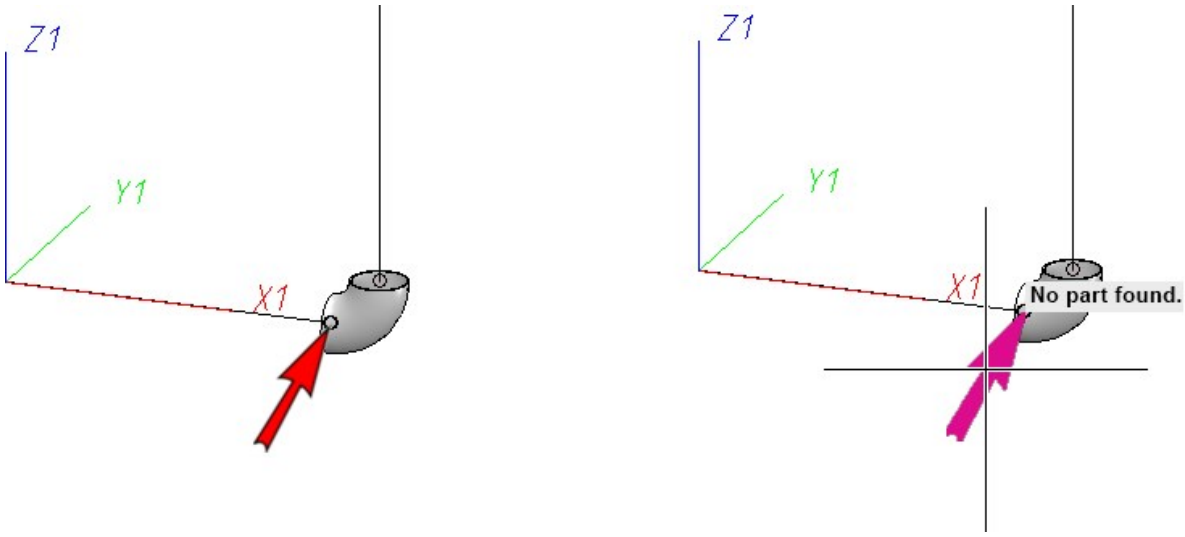


Invert Z-direction	You can use this option to make the Z-axis of the sketch coordinate system point in the opposite direction. Especially at connecting points, the Z-axis can thus point into the connection instead of out.
Set Reference CS interactively	As with the regular sketch function, you will be prompted for the sketch coordinate system. So you always have the possibility to get full flexibility in choosing the coordinate system, if that should be necessary.
Cancel	Ends the function.

Tip:
If you select the option **Local CS**, you can use the Plant Engineering functions **Set on connection** and **Set on guideline** to easily set the origin of the sketch coordinate system relative to a pipe connection.

Automatic placing of parts on guidelines

If there was an error during automatic placing of parts, the error position is now indicated by a red arrow in your drawing after placing. If you move the cursor over the arrow, the corresponding error message is also displayed.



The message *Marked parts prevent execution* is displayed.

To switch off the arrow and the message, the active selection must be ended. This is done, for example, by clicking on the arrow, by clicking with the middle mouse button, with the ESC key, by selecting **Cancel** in the context menu (right mouse button) or by clicking on a function in the Ribbon.

Dynamic route change

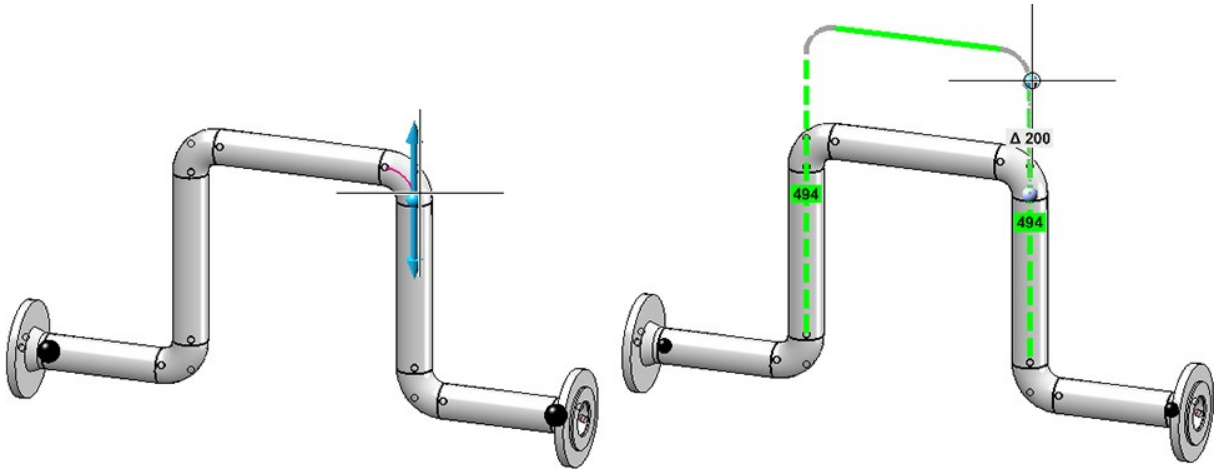
Changed display

Previously, the route change highlighted the entire selected pipeline network. This could sometimes lead to the fact that the essential changes to the pipeline route were difficult to recognise.

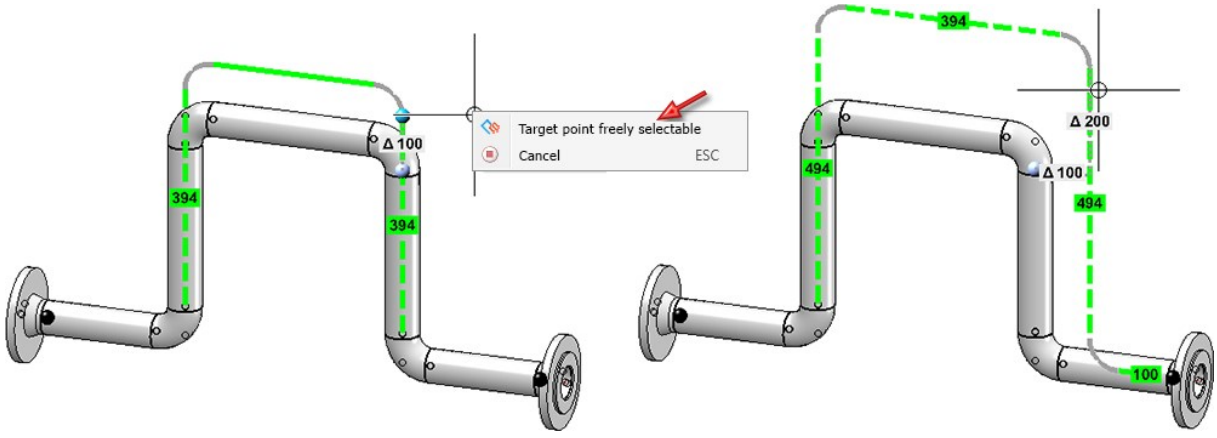
As of HiCAD 2021 SP1, only the parts that are changed in length or position are now highlighted.

Connection direction

While selecting the start point, the connection direction is now clearly shown as an arrow for each point. This means that after selecting the start point you can only change the pipeline route in the direction shown.



If you want to deactivate this restriction, press the right mouse button after selecting the start point and select **Target point freely selectable** in the context menu that then appears. Now you can freely select the direction.



If you want to reactivate the restriction, select **Restricted target selection** in the context menu.

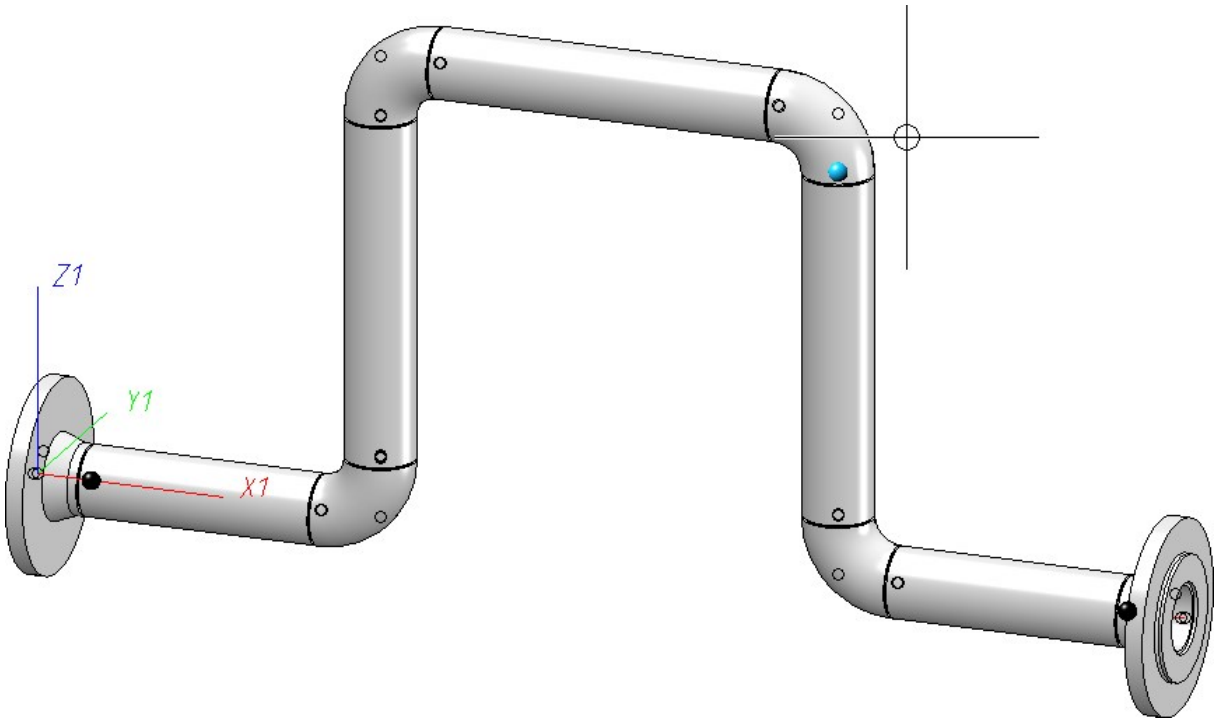
Fixed points

The dynamic route change allows a very flexible adjustment of a pipeline route. However, full flexibility is often a hindrance, because having many possibilities also means being able to make many potential design errors. For this reason, with HiCAD 2021 SP1, some changes have been made to sensibly limit the flexibility of the dynamic route change.

Automatic fixed points

After selecting a start point HiCAD now automatically displays a pre-selection of fixed points. Fixed points are set at the ends of the pipelines, whereby each fixed point is located behind the last part that allows a change in length. Usually these are the straight pipes.

For example, if the blue point were chosen as the starting point in the pipeline shown, the two black points would be set as fixed points.

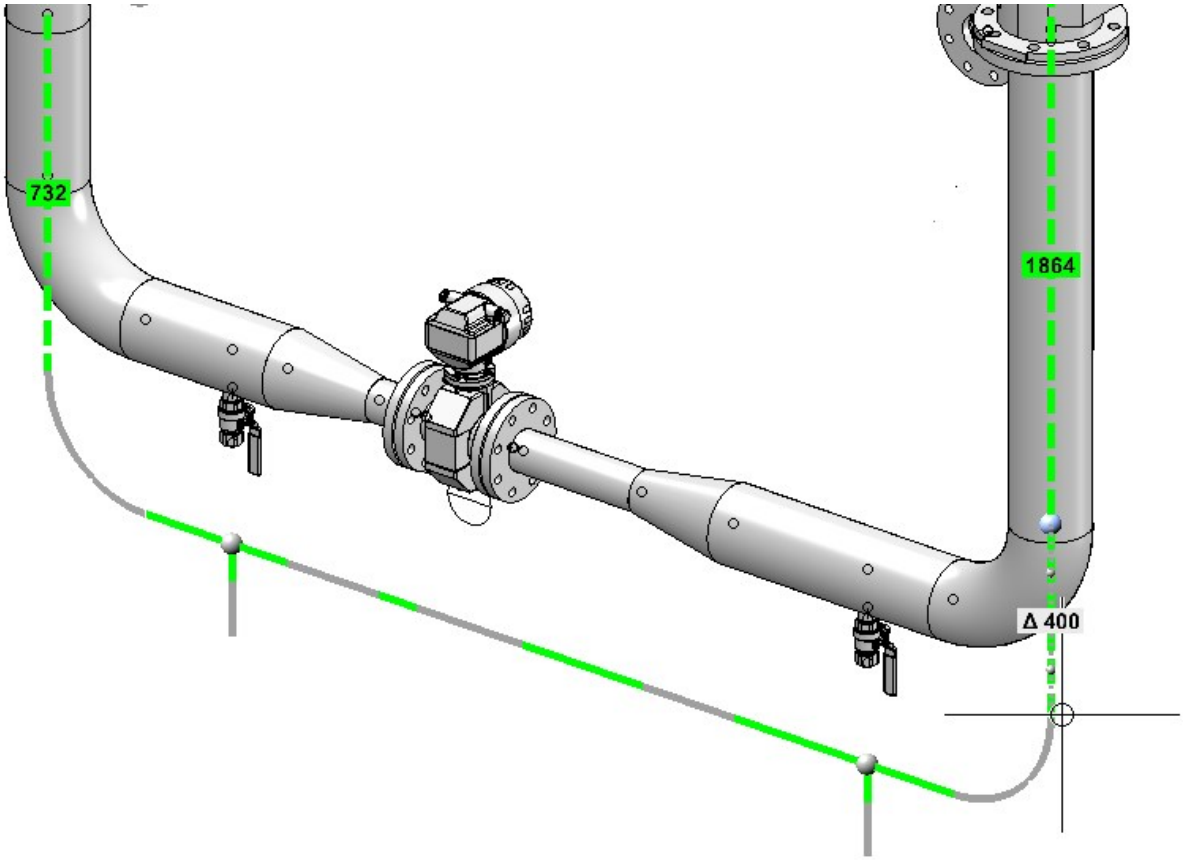


Exceptions:

In the following cases, pipe ends are not fixed:

- The pipeline consists only of parts whose length cannot be changed.
- The only part of the pipeline whose length can be changed is an inserted (plugged-in) pipe part.

In the situation shown below, you could therefore pull the pipeline down directly without first having to remove fixed points on the two inserted ball valves.



Automatically set fixed points can be deactivated by clicking on them again in the fixed point mode. In addition, you can set any fixed points.

Fixed points are managed separately for each start point. This means that if a new start point is selected, the set fixed points also change.

A special role is played by fixed points that are set before the start point is determined. These are used as fixed points for all subsequently selected start points. If such points are present when the start point is selected, then no automatic fixed points are assigned.

Major Release 2021 (V 2600)

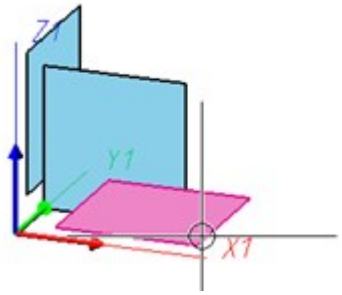
Guidelines

Create guidelines as a 3-D sketch

Guidelines are created as a 3-D sketch in plant engineering from HiCAD 2021. The **Create guideline** function is used for this purpose.

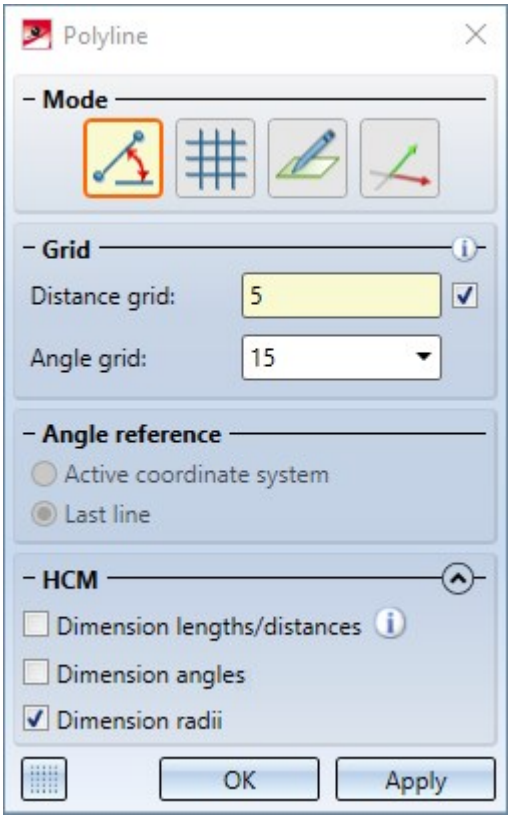
The function works in two steps:

1. A new guideline is created. This corresponds to creating a new 3-D sketch. That is, you must first determine the plane in which the guideline is to be drawn. This determines the part coordinate system.



2. After creating the guideline, the **Sketch polyline** function is automatically started and with it the 3-D sketching tool. The sketching tool facilitates the construction in that HiCAD - starting from the respective last point of a polyline - automatically displays auxiliary lines along a predefined grid and - depending on the selected mode - shows the corresponding angles, distances or radii at the cursor. On this grid you can, for example, determine the direction and length of a line with a mouse click by moving the cursor accordingly.

   Since 3-D sketches - like other 3-D parts - have a part coordinate system, the plane in which drawing takes place can be changed at will during drawing. This drawing plane is always parallel to the selected plane and runs through the last determined point.



The guideline is only created when you actually draw lines.

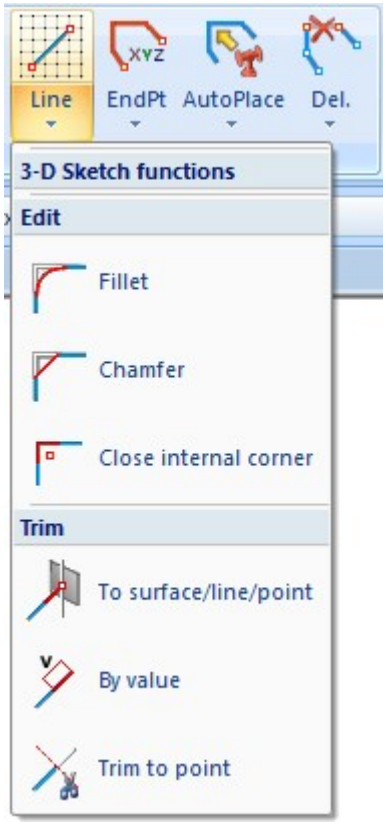
The former Guideline Editor is still available via **Plant Engineering > Guideline Tools > EndPt > Create guideline** . This also applies to the earlier editing functions.

Please note:

Unlike regular 3-D sketching functions, the **Create guideline** and **Sketch polyline** functions on the **Plant Engineering** Ribbon undo the entire change at once and not each added edge individually as in regular sketches.

Edit guidelines

To edit guidelines you can now use the functions from the **Sketch** ribbon. In addition, a selection of frequently used functions for editing guidelines is also available in the **Sketch polyline** sub-menu .

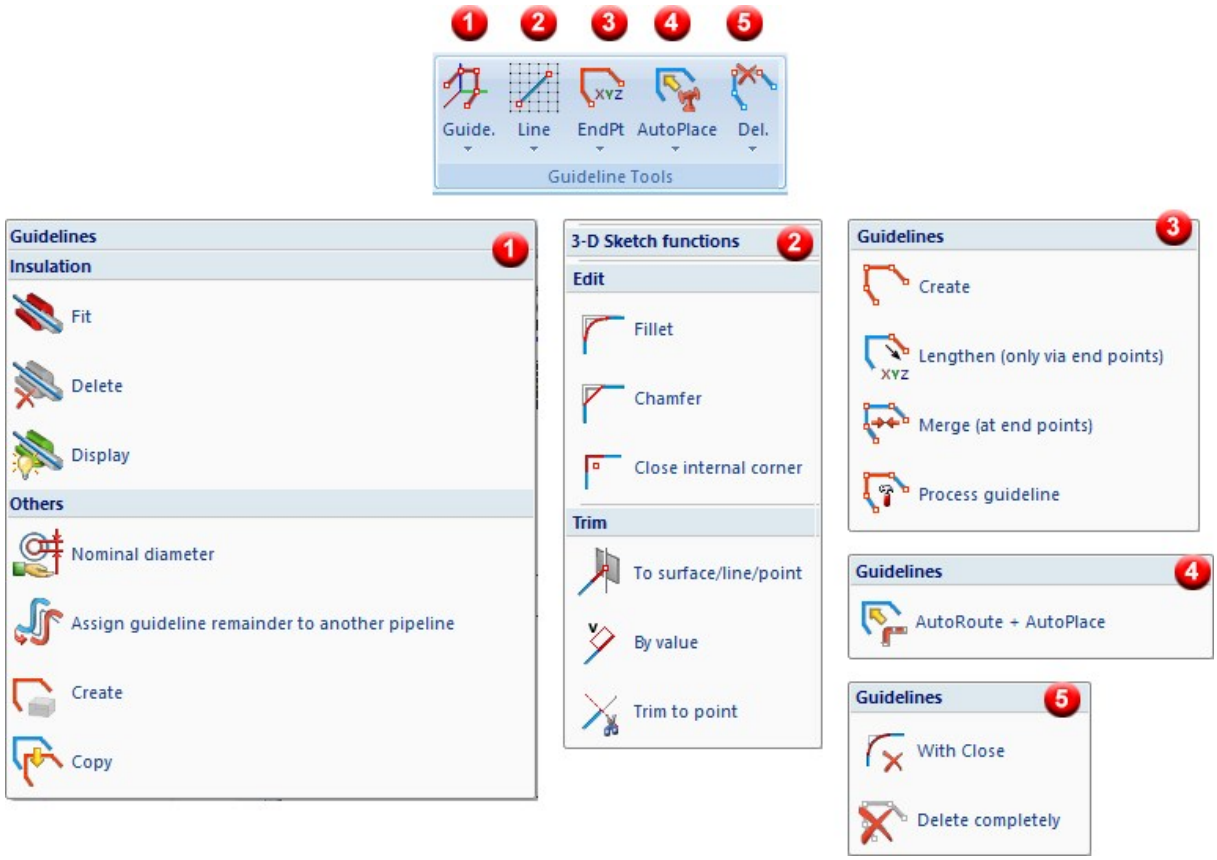


The previous functions for filleting / cutting off of a corner have been replaced by the corresponding sketch functions **Fillet / Chamfer**.

The function **Mark guideline** is also no longer available. Instead, use the function **Sketch > Tools > Sort GE > Mark**.

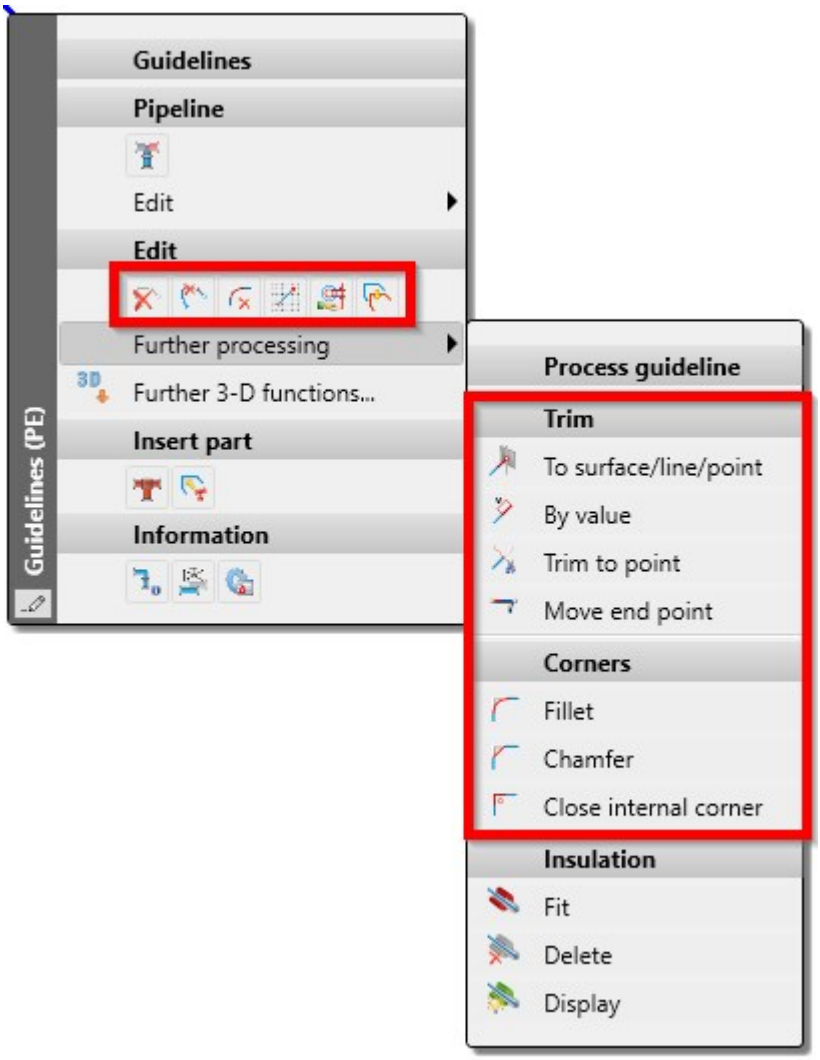
Changed function group

Due to the creation of guidelines as a 3-D sketch, the **Guideline Tools** function group has changed as shown:



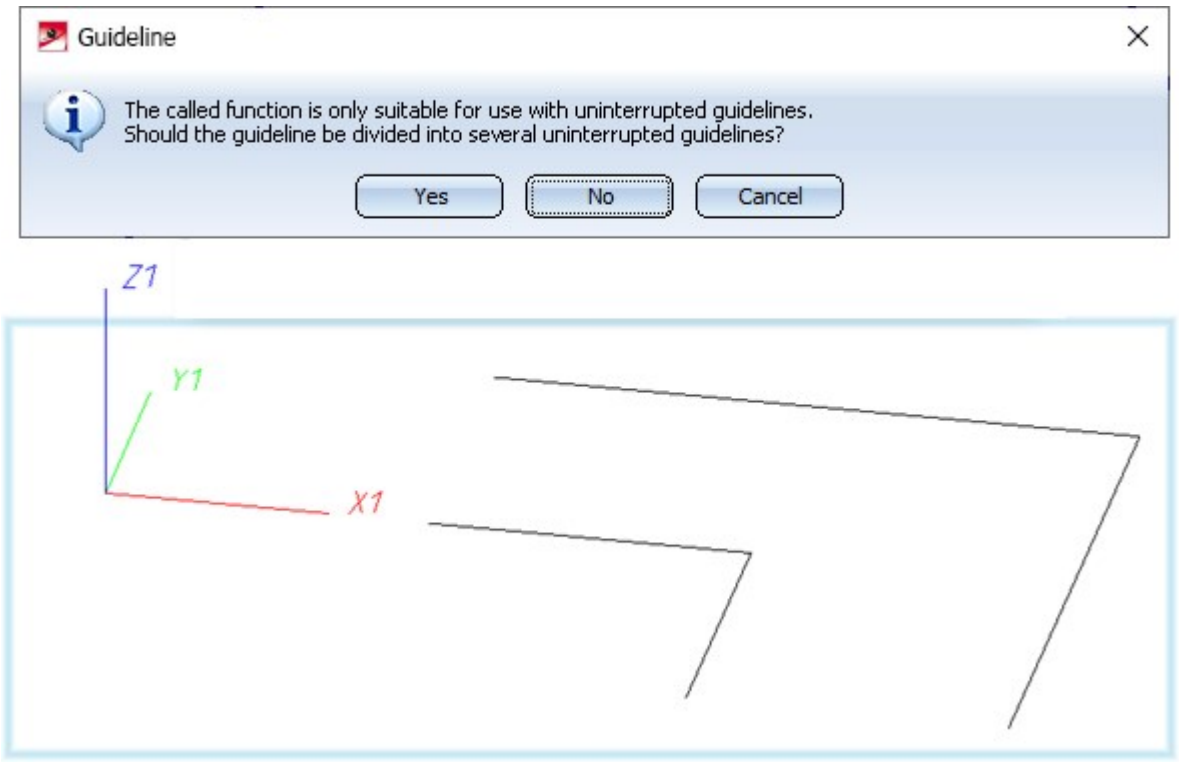
Changed context menus

The context menus for guidelines have also been changed accordingly.



Non-contiguous guidelines

Guidelines that are created with the Guideline Editor must - in contrast to the creation as a sketch - be created contiguously, i.e. without gaps. If the editor is used on a non-contiguous guideline, a corresponding warning message is displayed.

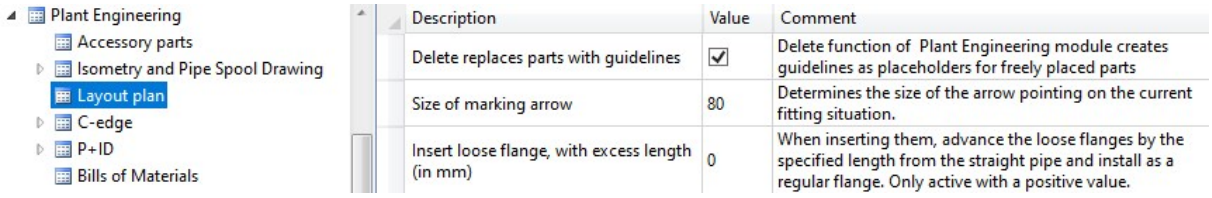


If you confirm the query with **Yes**, the guideline is divided into several guidelines, each consisting of the contiguous components of the original guideline (in the above Example 2). You can then select one of the divided guidelines for use with the Editor.

If you select **No**, the guideline will not be changed and you can select another guideline.

Automatic placing of parts on guidelines

With HiCAD version 2019 SP2 (Version 2402), the automatic assignment of guidelines has had a redesign. The previously valid behaviour could previously be used further by activating the checkbox **AutoPlace like in version 2401 and older** in the Configuration Editor at **Plant Engineering > Layout plan**. This option has been removed from the Configuration Editor with the release of HiCAD 2021.



Standardized part search

When searching for parts, the syntax for search queries used to slightly differ depending on whether you searched something in the catalogue or in HELiOS. This is now standardized for all data input in 3-D Plant Engineering. This applies e.g. to the pre-selection mask for part insertion and the dialogue for editing catalogue pipe classes.

The new syntax is basically the same as in HELiOS, with only slight differences.

The detailed rules can be found at Rules for part search.

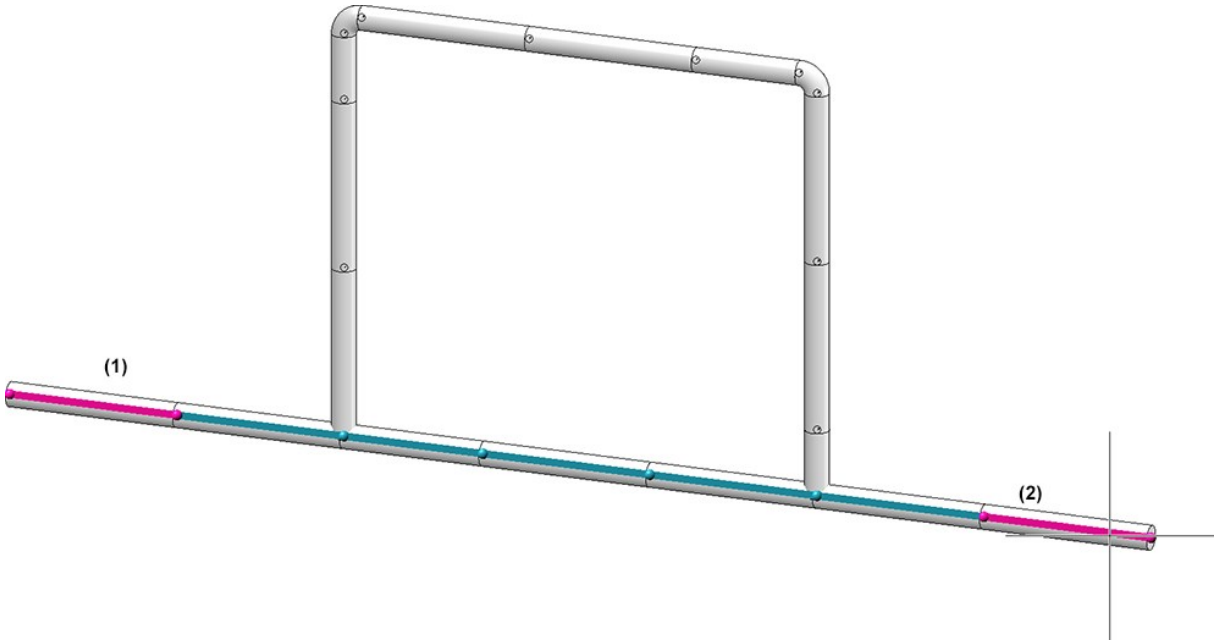
Delete parts

When deleting parts, guidelines can optionally be generated. These guidelines are then located where the deleted parts were before. In many cases this is helpful, but not for very narrow parts at the end of the pipe. This is especially true for guidelines that were previously generated when deleting gaskets. These were often covered by the indicated centre line of the adjacent flange and could therefore only be deleted separately with difficulty.

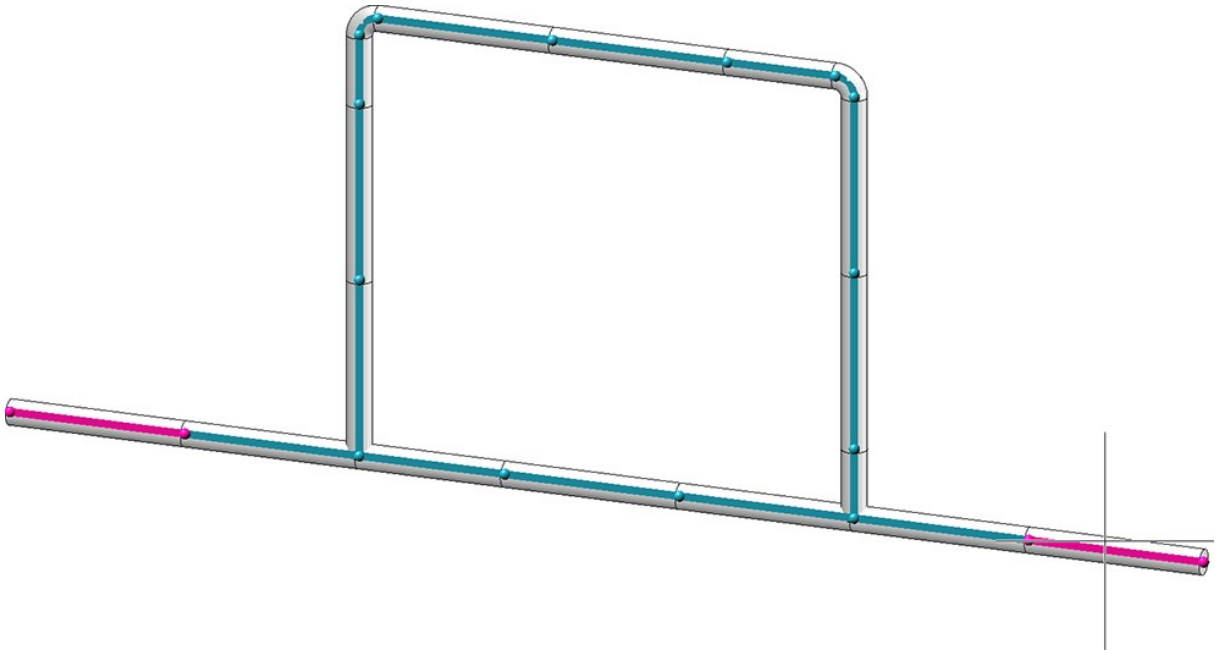
From HiCAD 2021, no more guidelines will be generated at the end of the pipeline for gaskets and loose flanges with excess lengths.

Select part chains

The behaviour of the function **Select part chain** has been changed. Previously, all parts on any path between the selected start point and end point were selected. From HiCAD 2021 onwards, only the parts on the shortest path from start to end are selected.



The previous behaviour can still be used. The new function **Select part network** is available for this purpose.



DIN 11853: Changed part type

The part type of N11853-3_KL.VAA has been changed from **Fastener, unsymmetric** to **Fastener, symmetric**.

When updating to HiCAD 2021, this variant must be newly entered into an existing HELiOS database in order to adapt the classification there as well.

This also applies to the variants

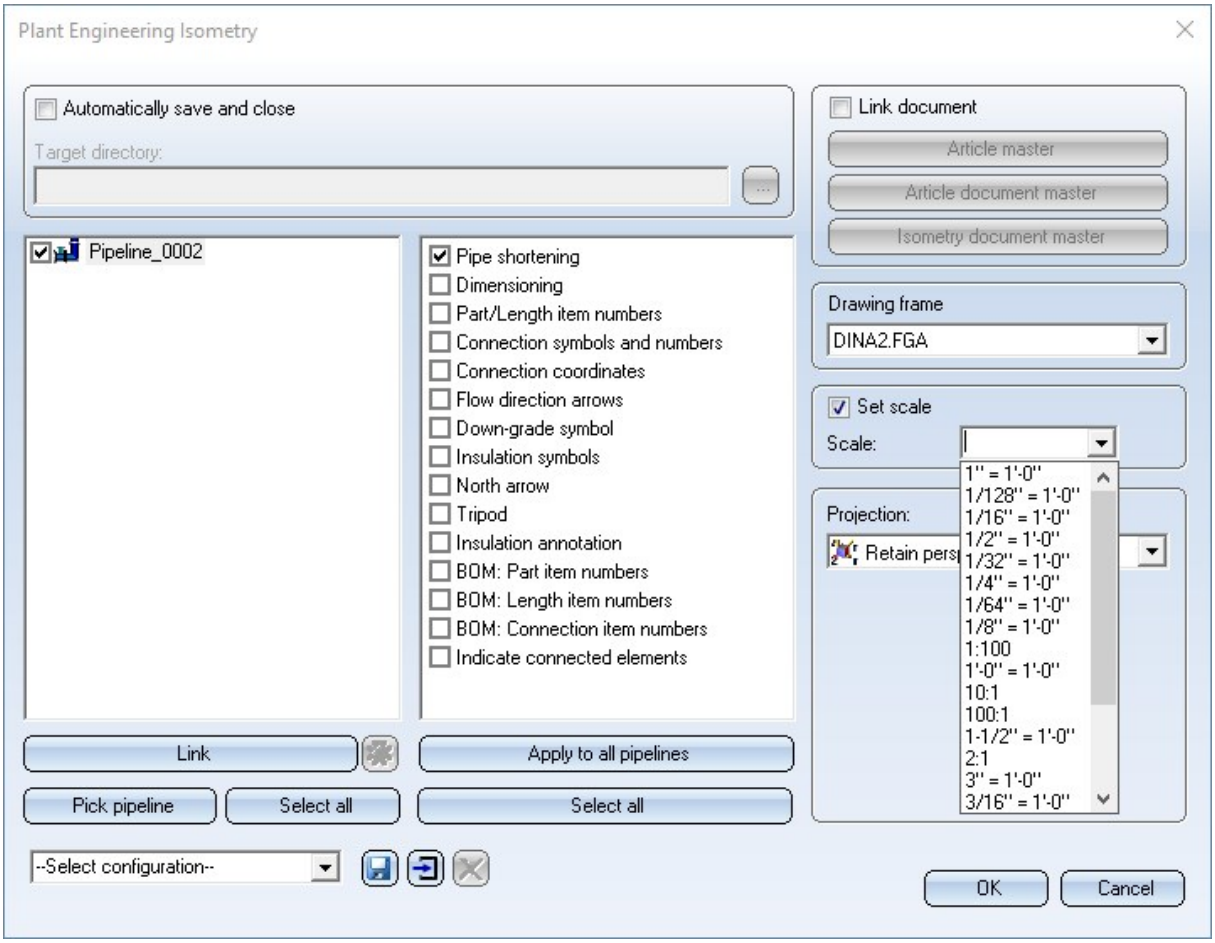
- N11853-3_BBK.VAA
- N11853-3_BK.VAA
- N11853-3_BNK.VAA
- N11853-3_NK.VAA

These refer in their connection IDs to N11853-3_KL.VAA , so that their connection type definition must also be adjusted.

Use the tool **DbPlantDataImport.exe** for the adjustment. Delete the previous files and then enter them into the database again.

Scale in isometry and pipe spool drawing

The settings in the Configuration Editor at **System settings > Scales** are also taken into account when selecting the scale in the isometry and in the pipe spool drawing, i.e. the scale can now also be selected in the selection box. The dialogue windows have been adapted accordingly. In particular, imperial scales can also be used here.



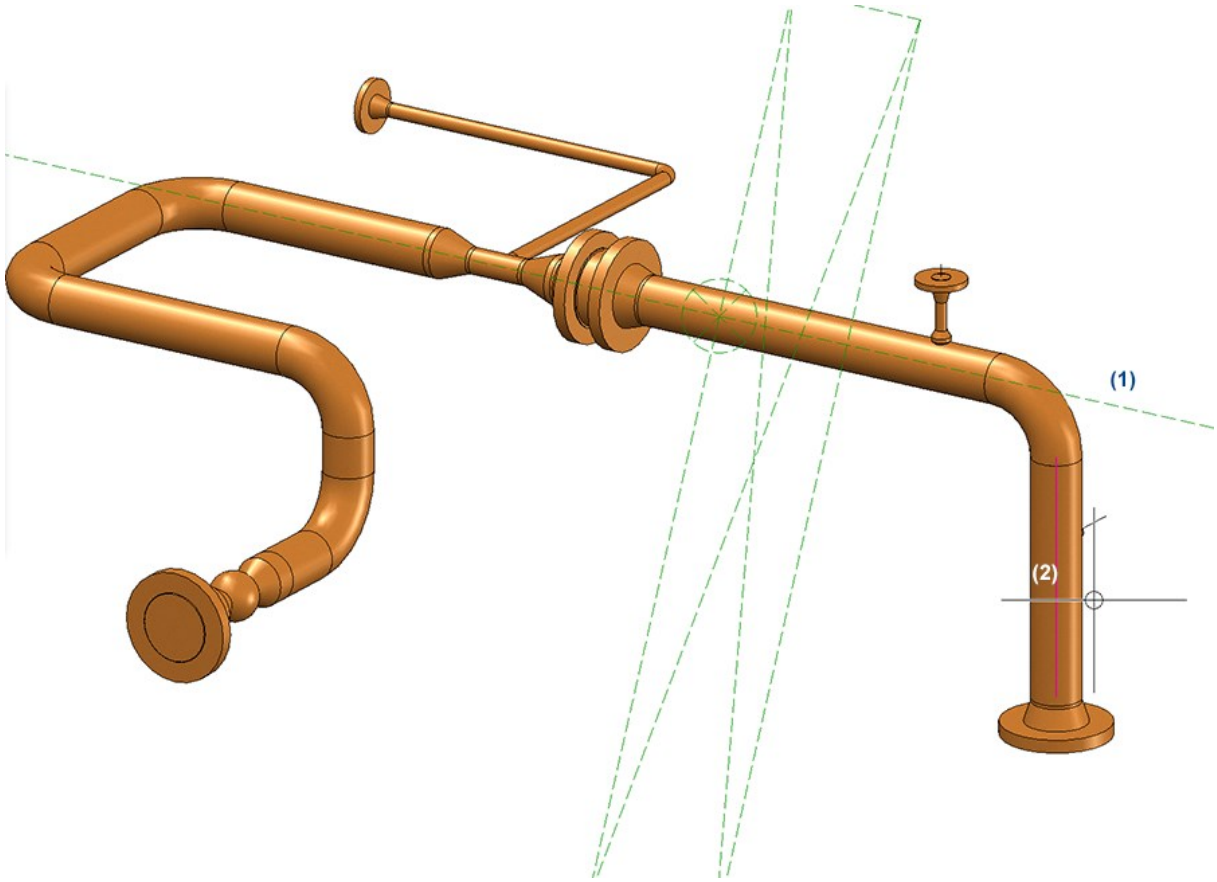
Viewer

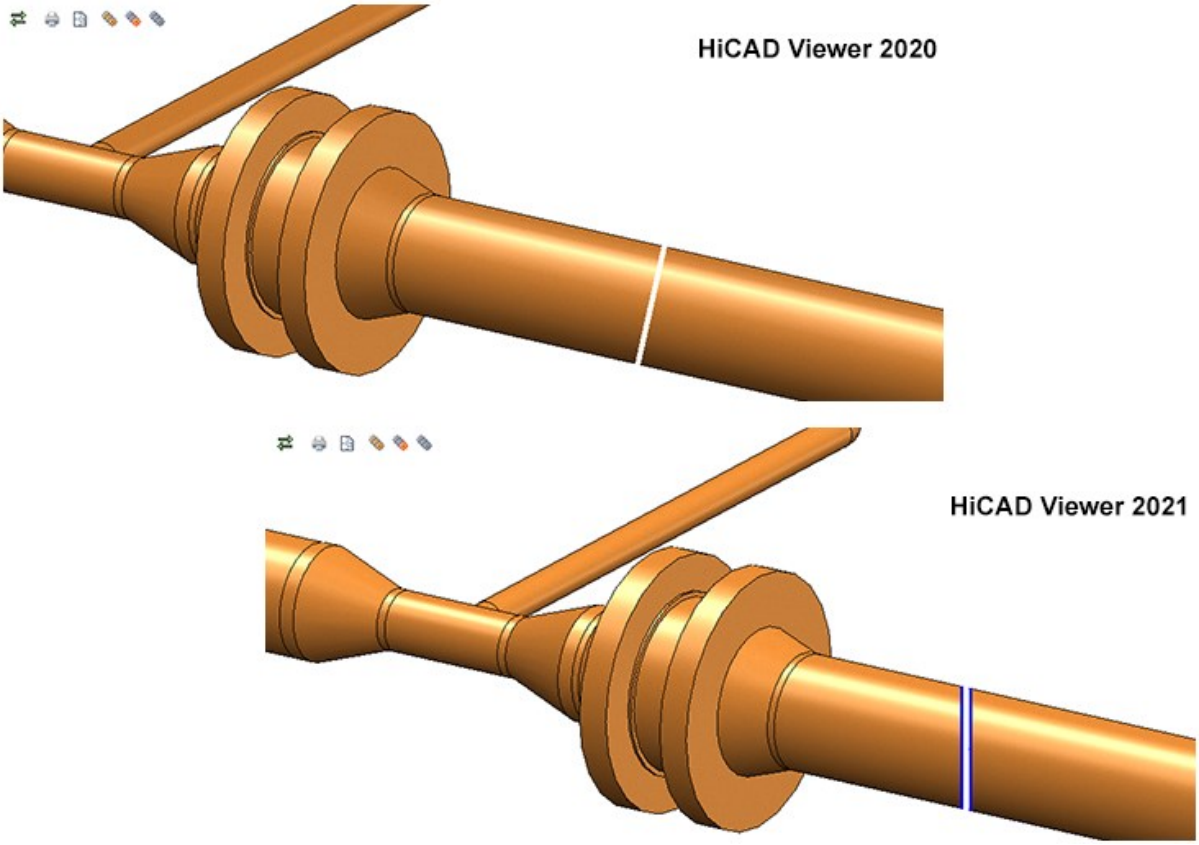
Major Relase 2021 (V 2600)

Representation of shortened views

Changes in the section direction of shortened views, e.g. perpendicular to the shortening direction, were not displayed in the HiCAD Viewer until now. This is possible from HiCAD 2021.

In the image below the section direction (2) perpendicular to the shortening direction (1) has been selected.




HiCAD Viewer 2020
HiCAD Viewer 2021

Notes on HELiOS Updates

For an update to HELiOS 2021 (Version 2600) from a version older than 2500 a central update of the supplied HELiOS database is required.

Since conflict may occur during the update process in case of inconsistent data stocks, you should do the following:

- Data backup before updating

  Make sure that a data backup was made before carrying out the update of your HELiOS database.
  For the backup, either use the HELiOS Database Creator (further information can be found in the Installation Notes) or your SQL Server Application.
  In case of any questions, or if you need any help with regard to your customized system architecture , contact the ISD Hotline.

- Log file for update

  If any conflicts occur during the update, these will be recorded in the log file **HeliosDbUpdate.txt** (in the system path **%appdata%\ISD Software und Systeme\HeliosDbUpdate**).
  Have this file ready when contacting the ISD Hotline in case of an unsuccessful update, so that they can help you solve the problem and make a successful update.

- New mask format

  Please read all notes on the new mask format introduced with HELiOS 2020 !

- MultiCAD interfaces

  If you are working with 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.

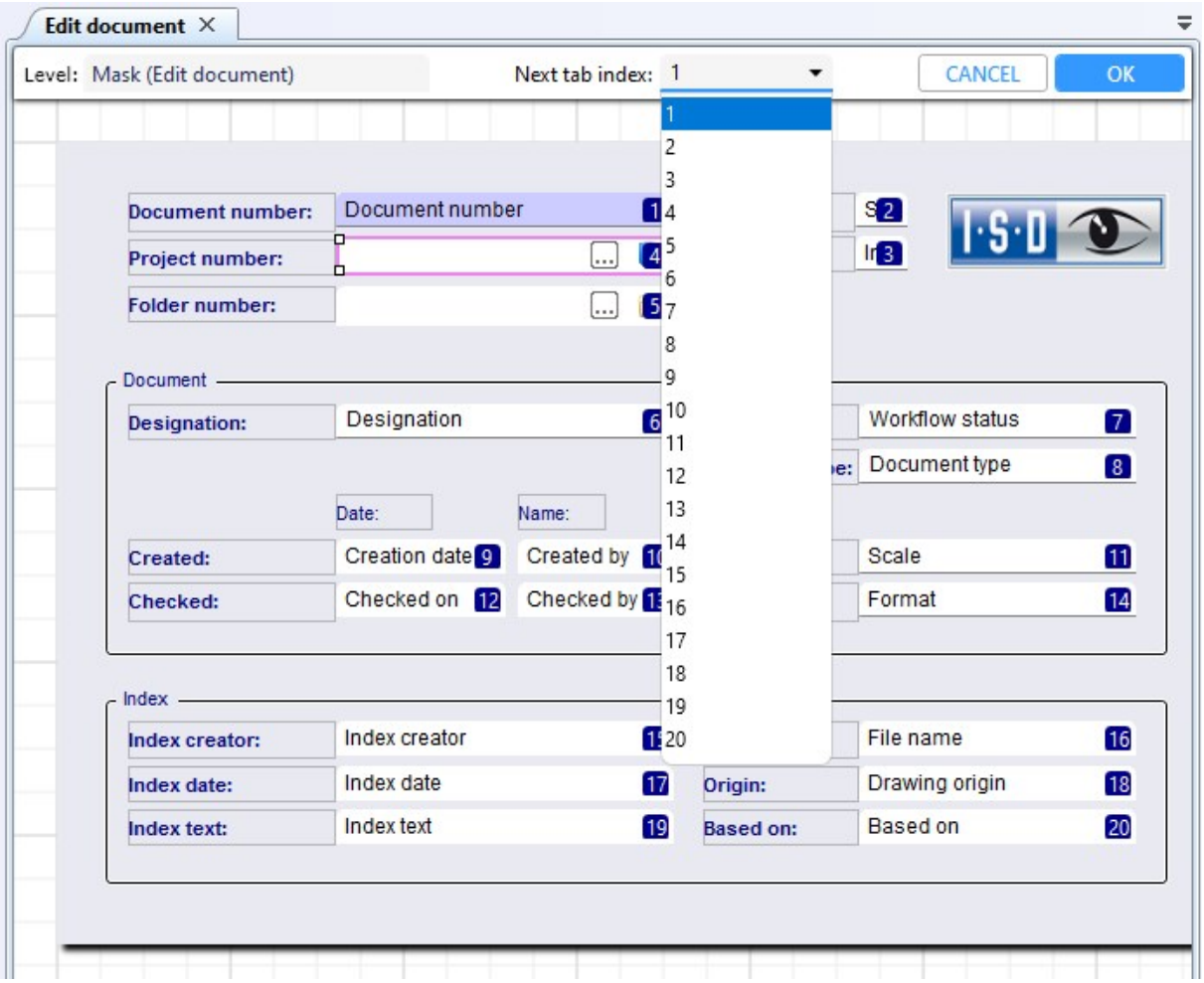
HELiOS Desktop

Service Pack 1 2021 (V 2601)

Mask Editor

Tab order

Among the new features in the **Mask Editor** is an improved option to define the order of tabs in mask elements. The new approach is even more convenient and simpler in its operation.

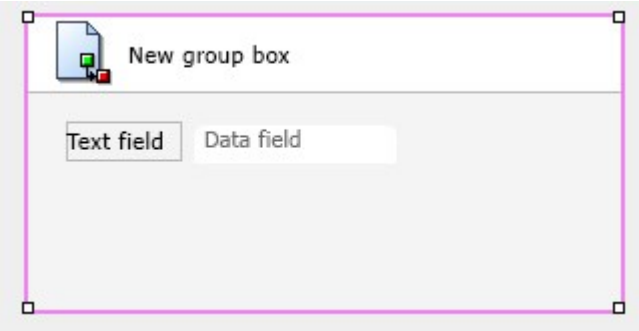


Group frames and group boxes

In addition to being able to draw a frame around elements to visually group them together, you can also insert actual group boxes into masks whose elements behave in a grouped manner.

This also allows, for example, that moving and positioning group boxes affects all associated elements. Elements in a group frame must have been marked for this in each case.

Also, group boxes can contain header icons.



Furthermore, "grouping" means that the tab order of elements within a group box is treated as a related group, while within the entire mask context the group box receives a tab index for itself.

Additional new and enhanced functions are planned for future releases.

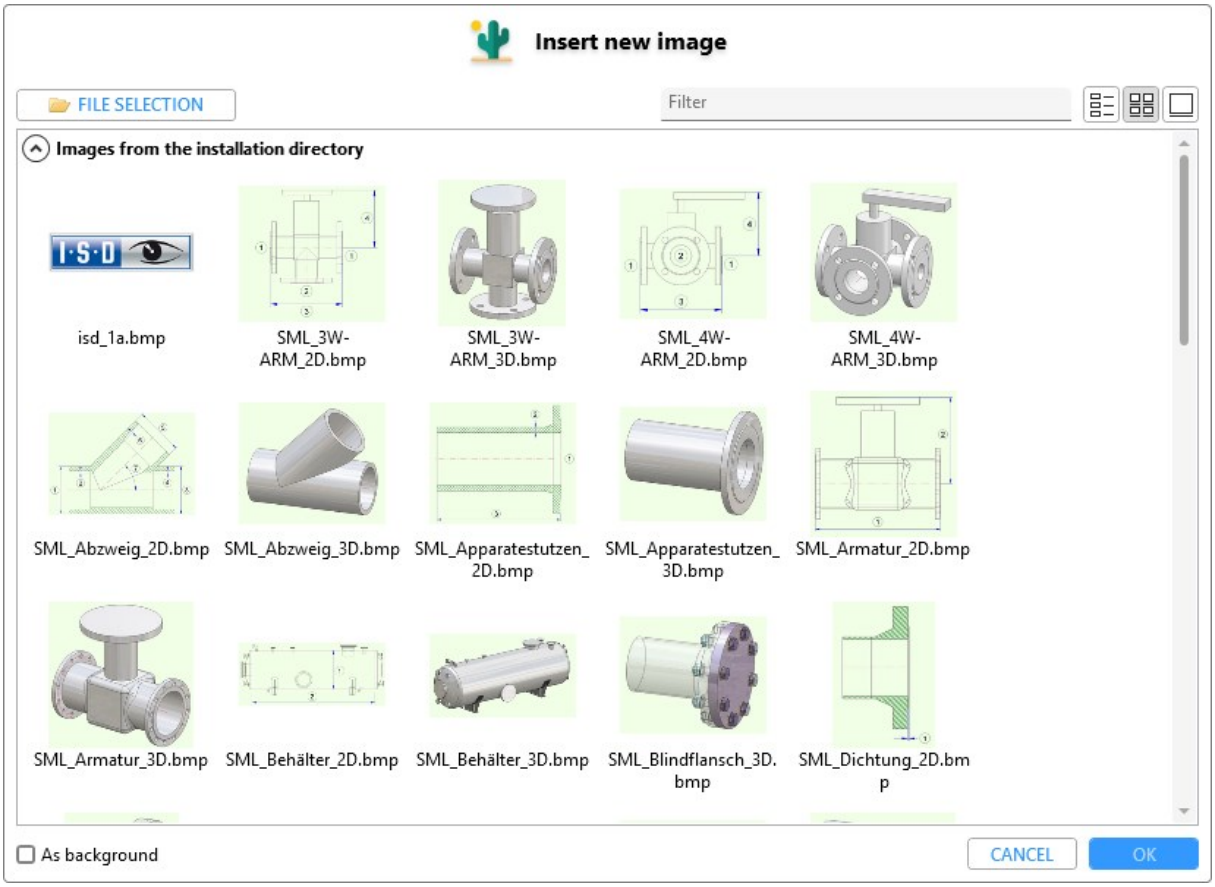
Insert new image

The functionality for integrating graphics into HELiOS masks using the mask editor has been extended and improved.

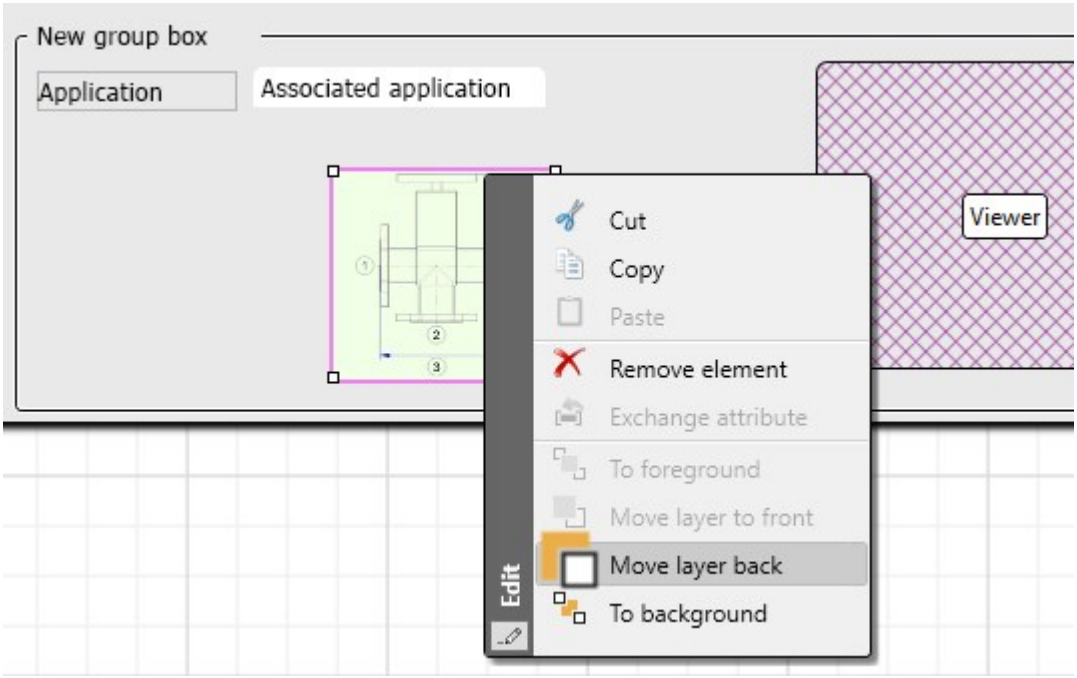
Within the selection dialogue you can adjust display type of the images and search for file names with a **Filter** input field.

In addition to the default selection list of the HELiOS installation directory, it is also possible to import image files from other storage paths.

All common image file formats can be used.



Images in masks can also be moved forwards or backwards, e.g. in the case of "overlaps", using the context menu of the right mouse button on the visibility layer.



The control for changing the transparency of mask objects has been improved with Service Pack 1.

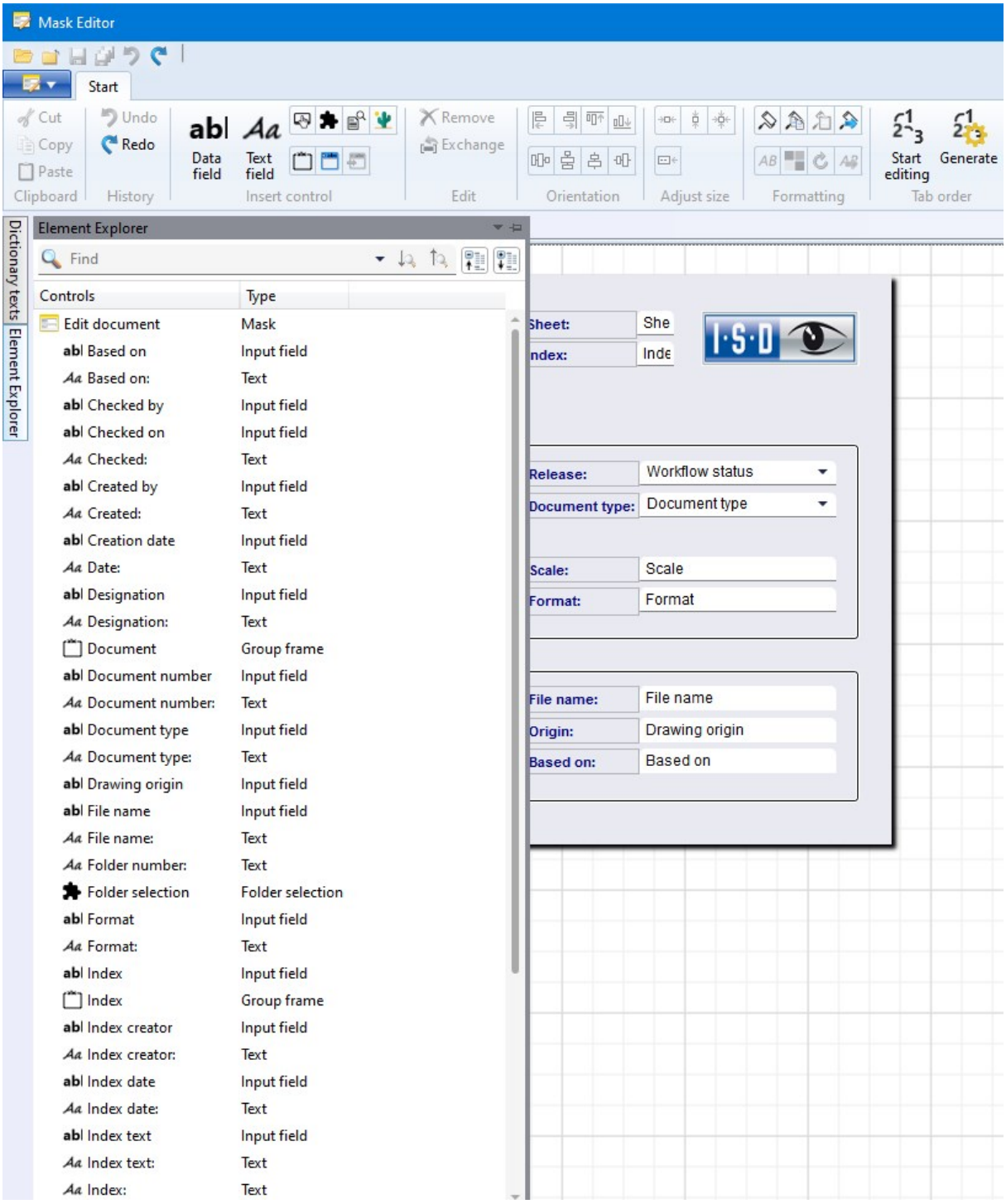
By holding down the SHIFT key, you can scale images while maintaining the size ratio.

Element Explorer

In the new **Element Explorer** of the **Mask Editor** you can see a listing of all objects of the active mask. The overview is hierarchically arranged and also includes elements that may be "hidden" by other objects in the mask itself.

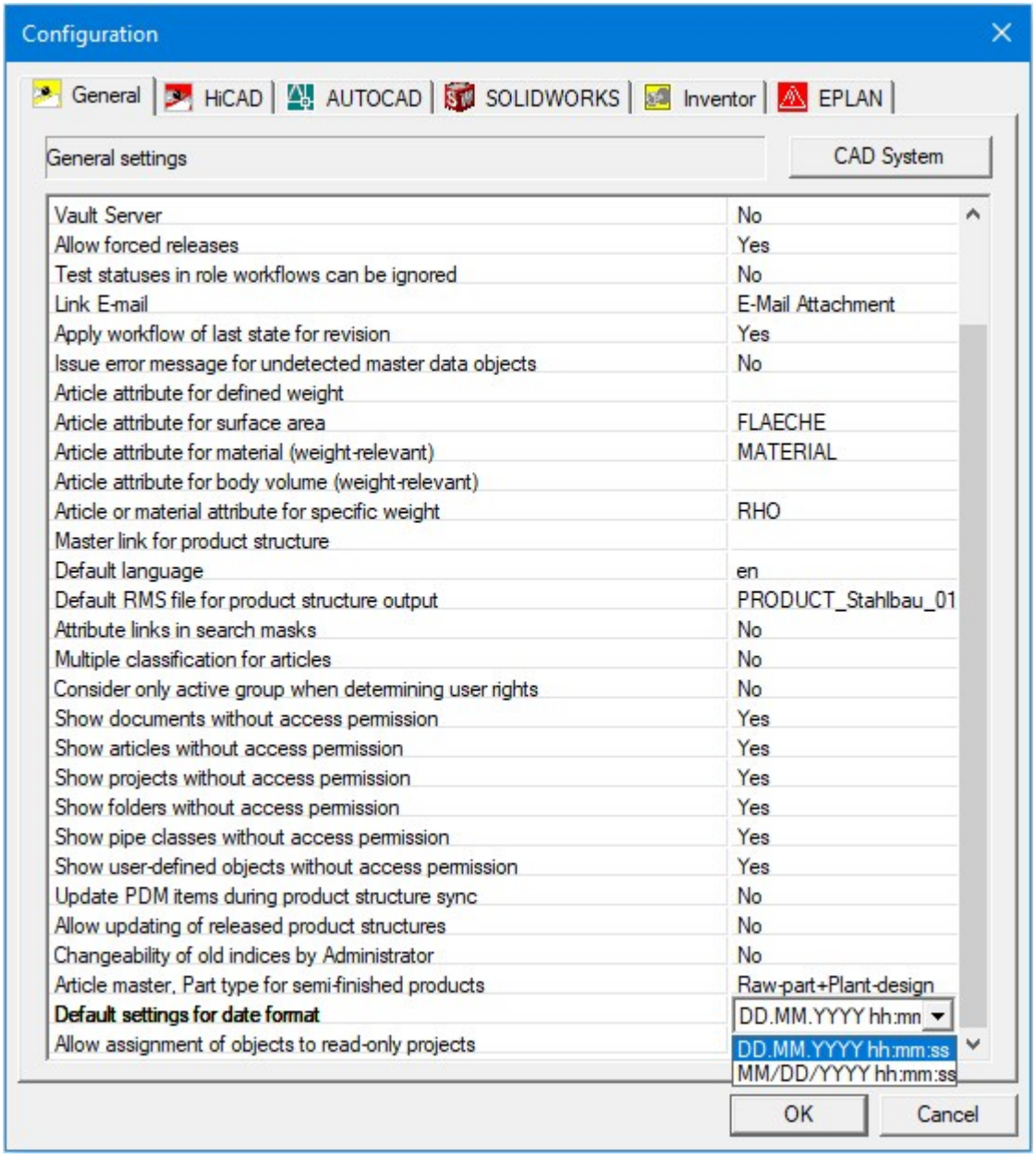
By clicking on a mask element in the **Element Explorer** overview, the corresponding element is set active in the mask and displayed in the **Properties** window.

By selecting and moving elements you can influence the tab order. The possibility of multi-selection is also supported.



Date format

In the **Database Configurations** of the **HELiOS Options**you have the option to change the default representation of time or date information to the American format with the option **Default settings for date format**.



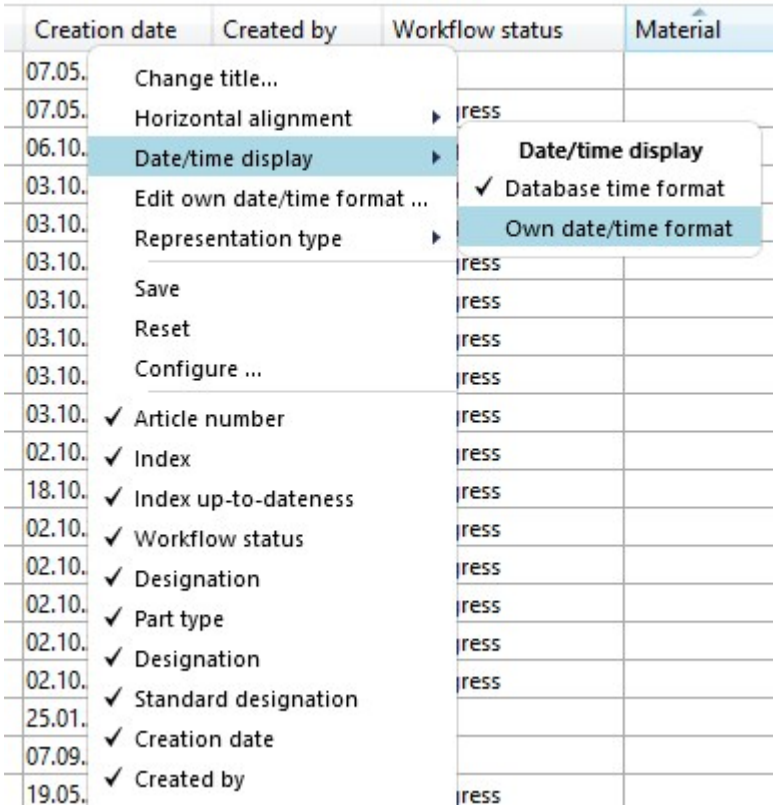
Please note:

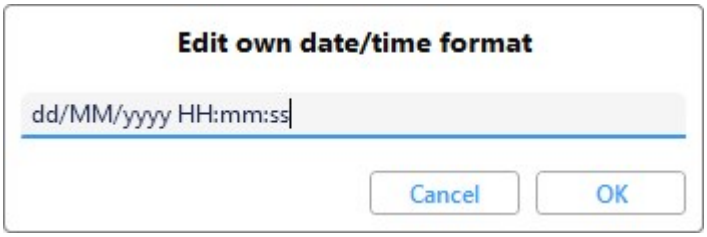
HELiOS must be restarted after the change for the new setting to take effect.

With regard to the time representation in HELiOS, you can generally make further settings.

For example, in result lists you can switch from **Database time format** (this is the above-mentioned database setting) to **Own time format** via the context menu of the right mouse button on the header of date fields under the menu item **Time display**.

If you have changed the display for the column to your own time format, you can edit it by clicking on **Edit own date/time format...** in the context menu below. in the context menu below.





Please also read the information given in the topic Time Display.

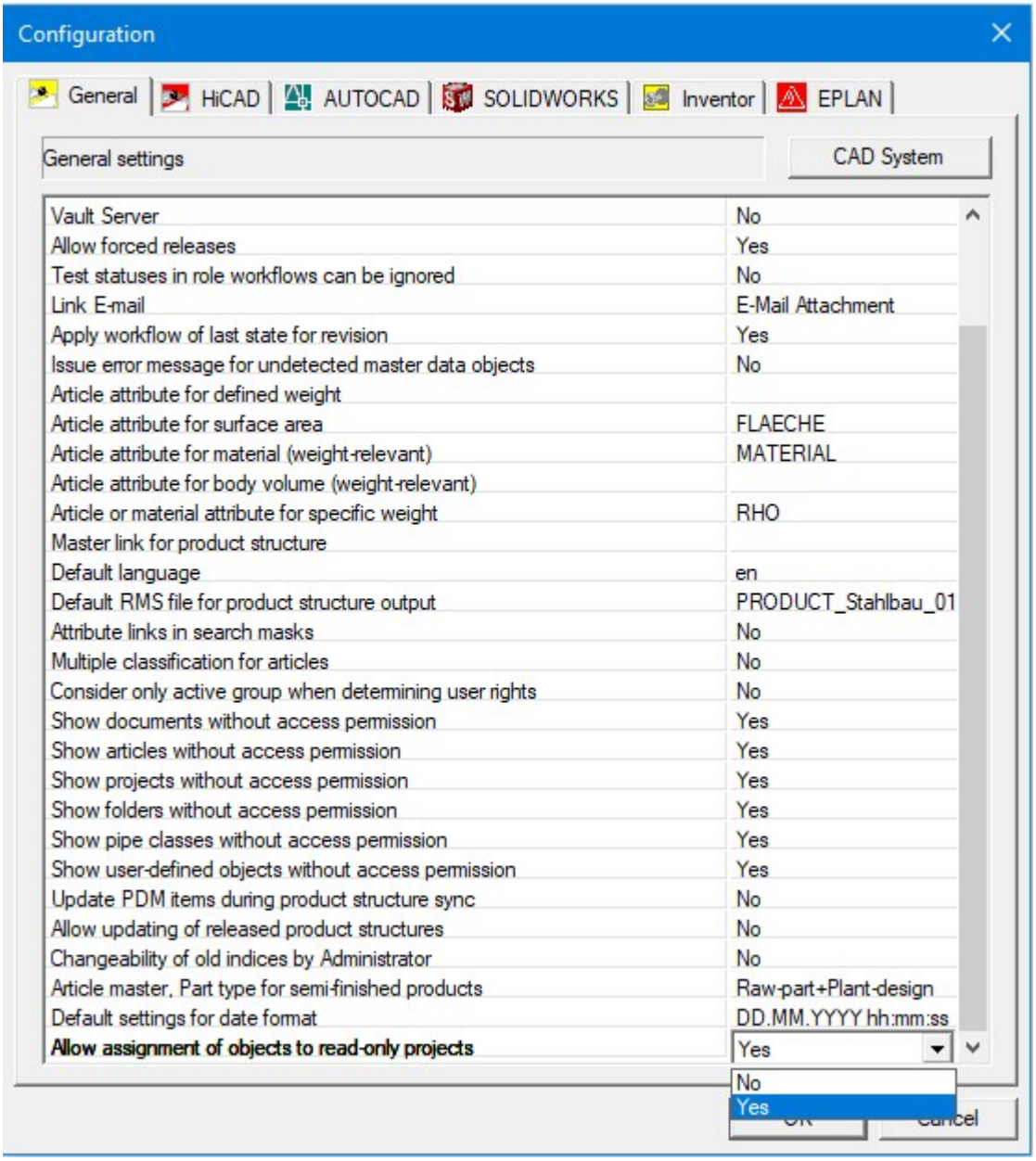
HELiOS Options: Allow assignments of objects to read-only projects

In the **Database** configuration of the **HELiOS Options** you will find the new setting Allow assignments of objects to read-only projects.

In principle, further objects can also be added to read-only projects (e.g. by Drag&Drop) if this option is set to **Yes** (default setting). To prevent this, you can set **No** via the pull-down menu.

Please note:

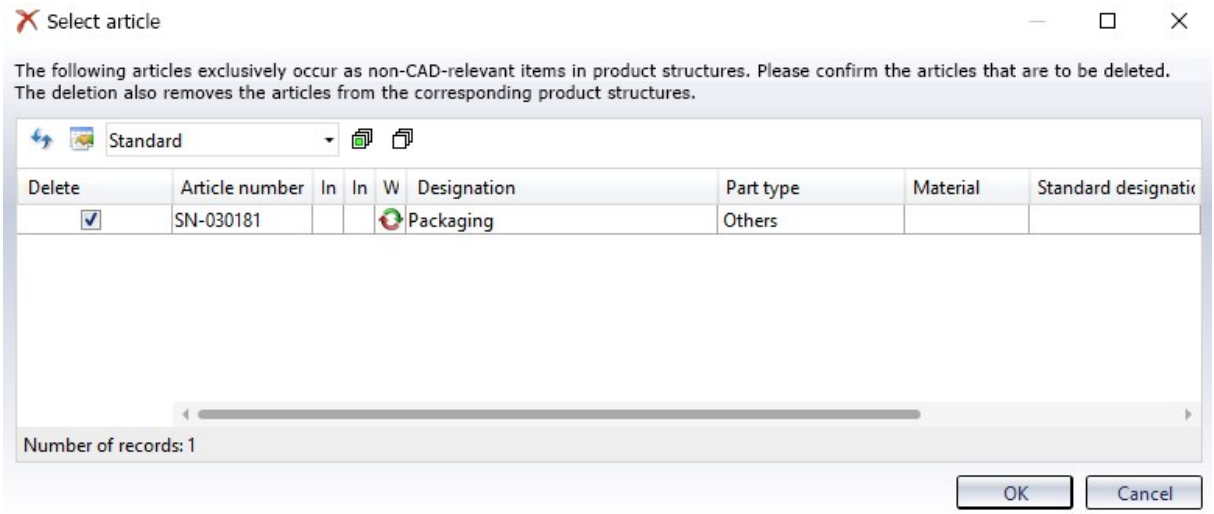
This also prevents the creation or assignment of sub-projects. Editing or object assignments below sub-projects of a read-only project that are not themselves read-only are still possible, however.



Deleting non-CAD-relevant articles

When deleting articles, their use in HELiOS product structures is checked. If there are articles in product structures whose header is not to be deleted, a CAD relevance check is also carried out for them.

If articles were entered exclusively non-CAD-relevant in product structures (whereby they can be deleted and simultaneously removed from all shoring), you will receive a warning message with the option to confirm a respective deletion for the corresponding articles (and thus also the deletion of all existing shoring in product structures).



The prerequisite is, of course, that the logged-in user has the appropriate authorisation for the process.

If problems occur during the deletion process, a corresponding error dialogue will be displayed.

Workflow: Old index

If you are working with an outdated index in the Workflow window for a HELiOS object, you will see a corresponding note in the title line of the dialogue, like in other HELiOS windows.

Please note:

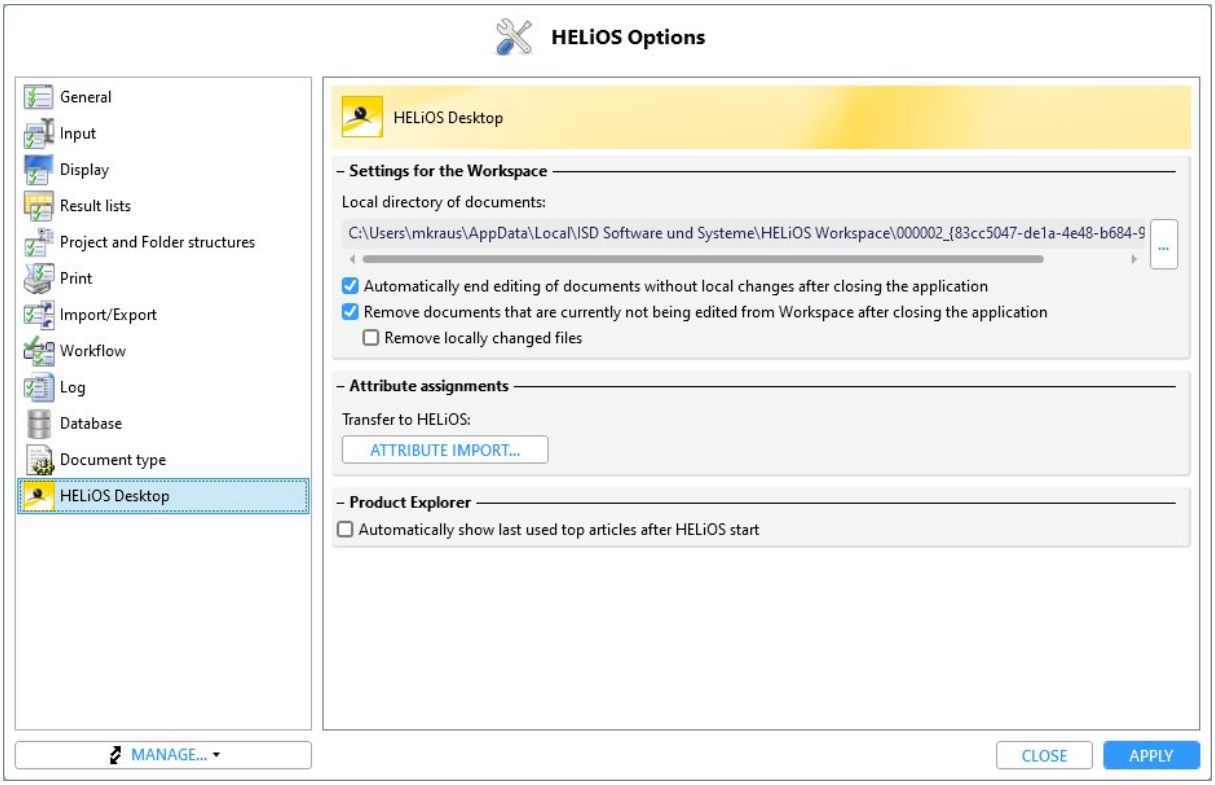
Workflows with old indices cannot be edited.

Major Release 2021 (V 2600)

HELiOS Options

As of HELiOS 2021, you will find the HELiOS Options for the HELiOS Desktop as well as for linked applications such as MS Office or MultiCAD applications in a centralised dialogue.

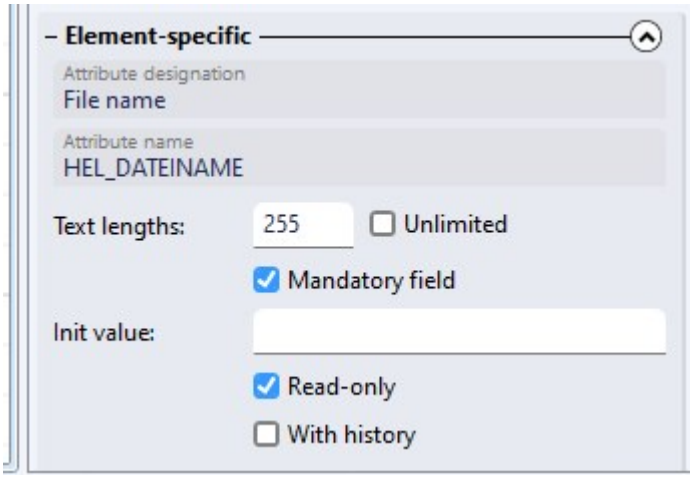
At the start, the submenu window belonging to the application from which the options were called is displayed as default. This means that if you call up the options from the HELiOS Desktop, the application-specific setting options for the HELiOS Desktop are displayed directly, from a linked Office or MultiCAD application the HELiOS Options specific to this application are displayed, and so on.



Mask Editor

Mandatory fields

Since HELiOS 2600 it is possible to mark certain elements in the Mask Editor as mandatory fields.



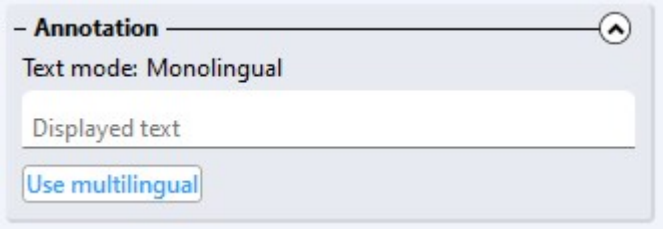
This makes it possible to allow certain actions, such as assigning articles or documents to classes, only if a certain database field of the class mask is not filled with a value. If this information is still missing, saving the corresponding mask is then not possible.

Working with mandatory attributes at mask level instead of attribute level enables you to use certain attributes differently for different classes. For example, an inspection attribute can be set as a mandatory attribute for the article class of a certain part type with a high safety standard, but remain optional for the article classes of other part types.

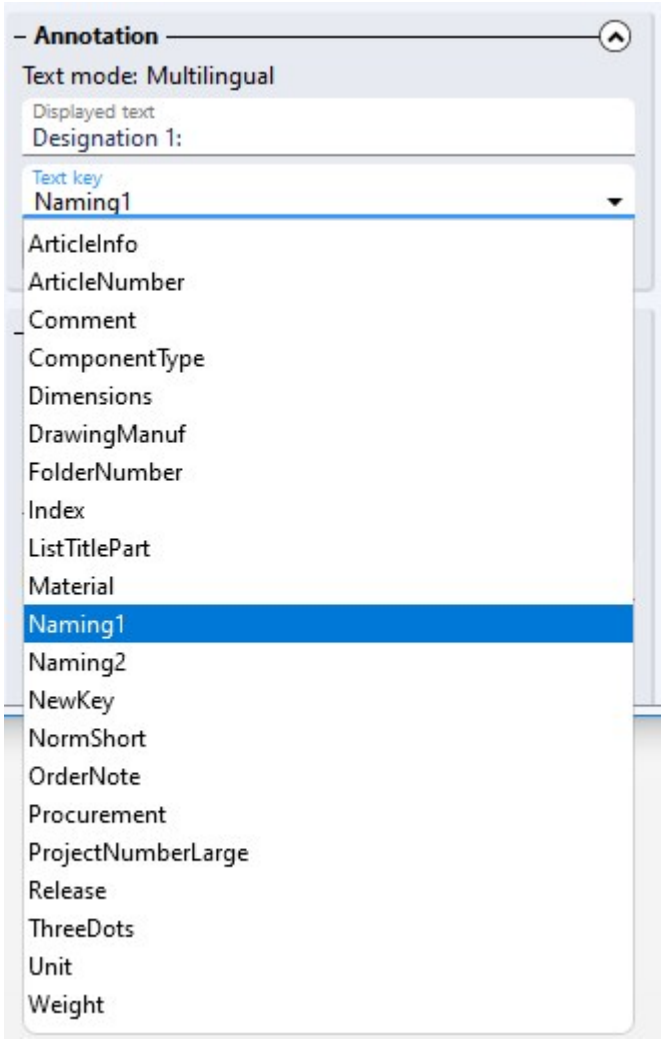
Multilingual text objects

Another new feature in the **Mask Editor** is that you can create text objects in several languages and manage translated texts (so-called "Dictionary texts").

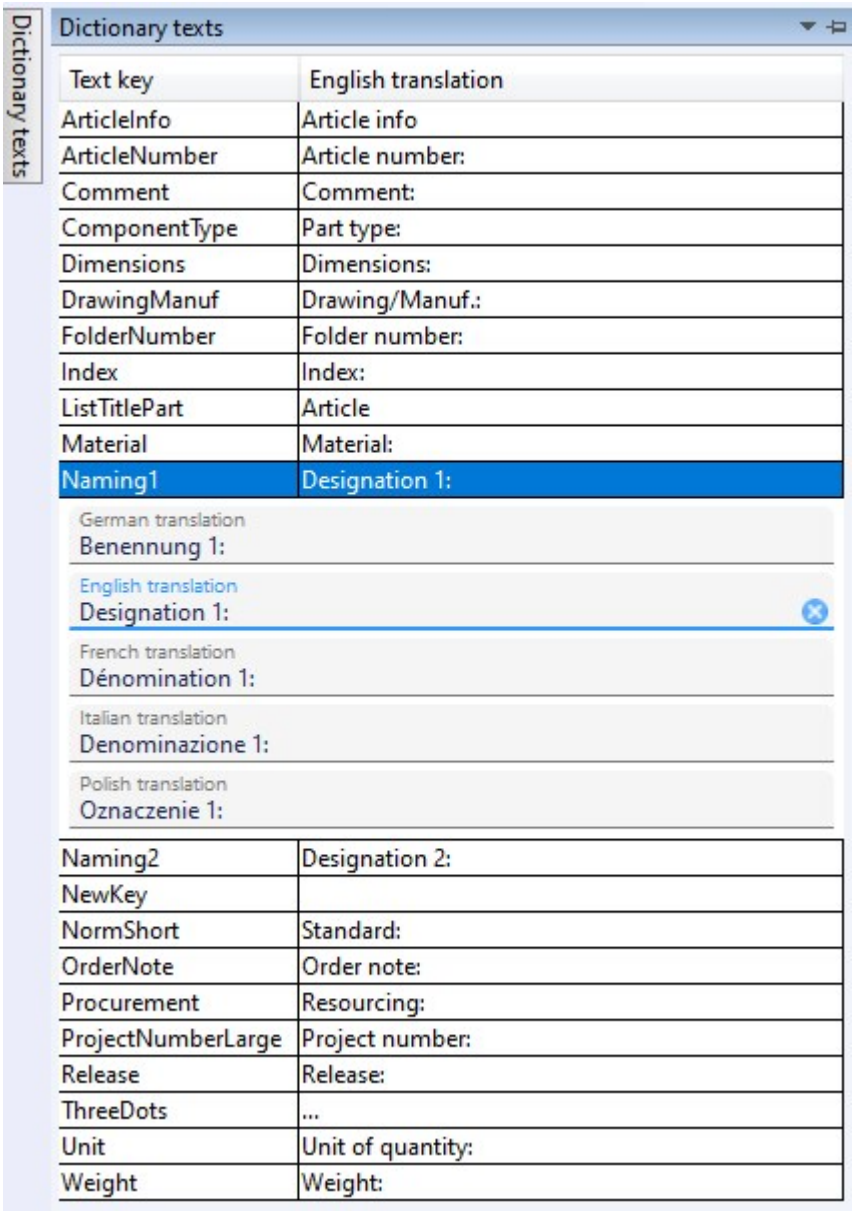
To do this, all you have to do is activate the **Use multilingual** button in the **Properties** window under **Text** with a mouse click when creating or editing corresponding elements.



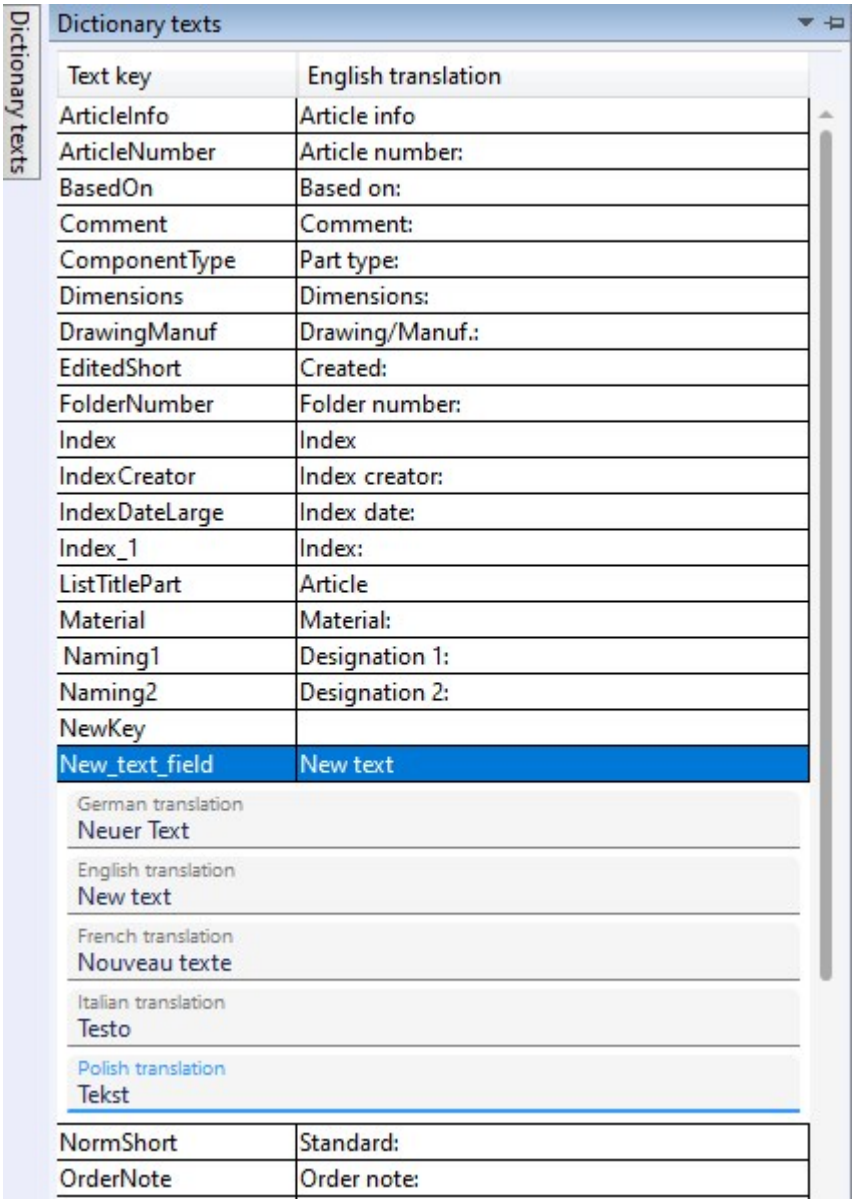
The **Text mode** of the element is changed from **Monolingual** to **Multilingual** and a field for selecting a **Text key** appears.



On the left in the mask editor you will see the vertical tab **Dictionary texts**, with which you can open and close a window. There you can click on a text key to display all the translatable dictionary texts available in the system or relevant for the active mask, and you can also change them manually by clicking in a field.



Newly created fields, for example under the text key "Text", can also be managed multilingually directly there.


Dictionary texts
Dictionary texts
Text key
English translation
ArticleInfo
Article info
ArticleNumber
Article number:
BasedOn
Based on:
Comment
Comment:
ComponentType
Part type:
Dimensions
Dimensions:
DrawingManuf
Drawing/Manuf.:
EditedShort
Created:
FolderNumber
Folder number:
Index
Index
IndexCreator
Index creator:
IndexDateLarge
Index date:
Index_1
Index:
ListTitlePart
Article
Material
Material:
Naming1
Designation 1:
Naming2
Designation 2:
NewKey
New_text_field
New text
German translation
Neuer Text
English translation
New text
French translation
Nouveau texte
Italian translation
Testo
Polish translation
Tekst
NormShort
Standard:
OrderNote
Order note:

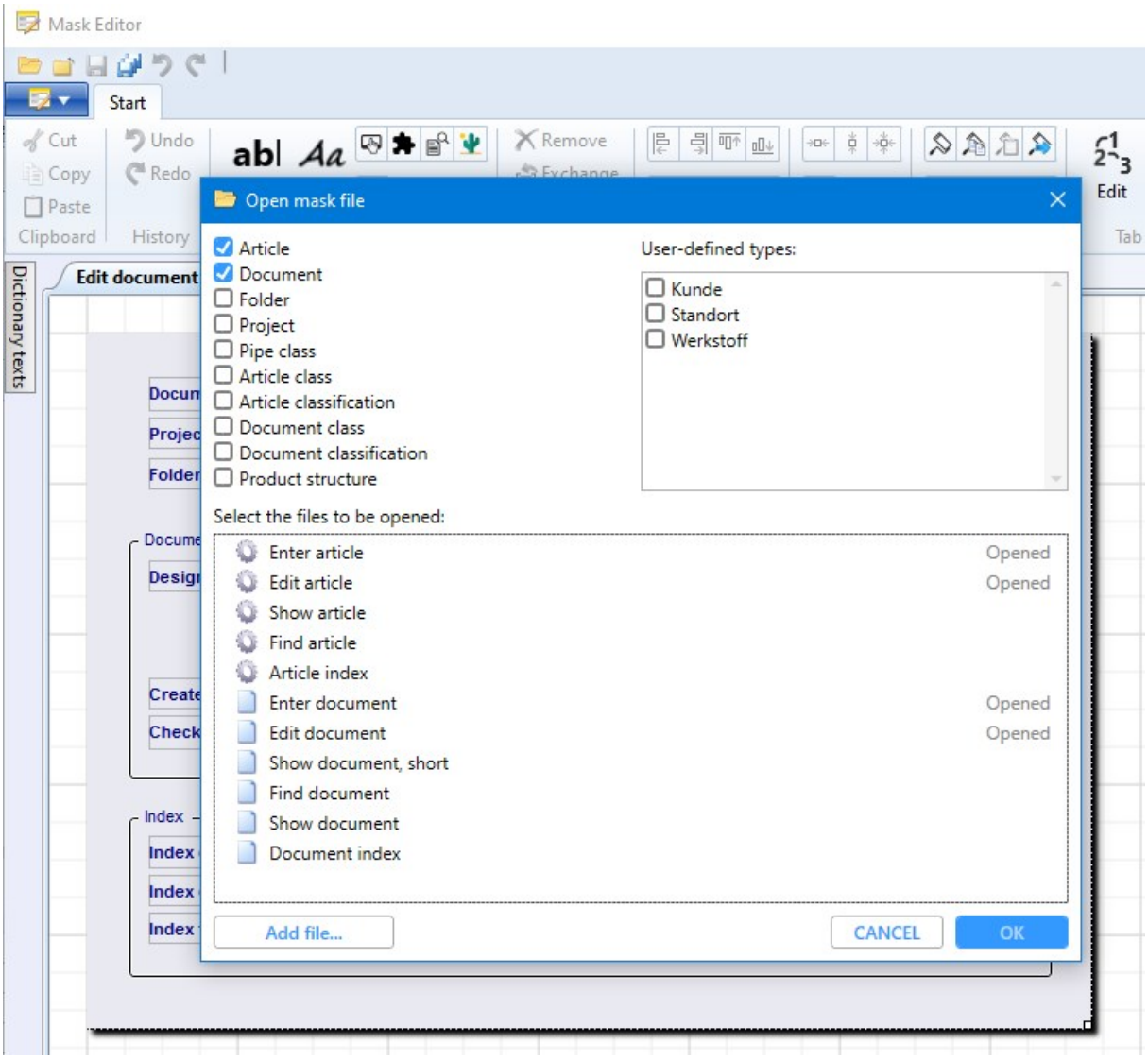
Further improvements

The new mask editor introduced with HELiOS 2020 has received some further improvements and minor enhancements.

For example, you can select masks in the selection window by typing the first letters of the name, similar to the Windows Explorer.

Masks that are already open are marked as such in the selection window.

HELiOS remembers the currently defined selection of .mask file object types and user-defined types and restores them the next time you run the selection dialogue.



In addition, if you should be working with a large mask in a small mask editor window, you can centre it in scroll viewer mode by pressing the space bar.

HELiOS / HELiOS Viewer 32 Bit

As of HELiOS 2021 (Version 2600), the 32 Bit installations for HELiOS and the HELiOS Viewer are no longer available in the download area of the ISD Wiki. If a 32 Bit installation is required, please contact ISD Support by e-mail at support@isdgroup.com.

As of HELiOS 2022 (Version number 2700.0), a 32 Bit installation for HELiOS and the HELiOS Viewer will no longer be available. However, the interface to 32 Bit applications such as Office will still be possible and is not affected by the discontinuation of the 32 Bit installation of HELiOS.

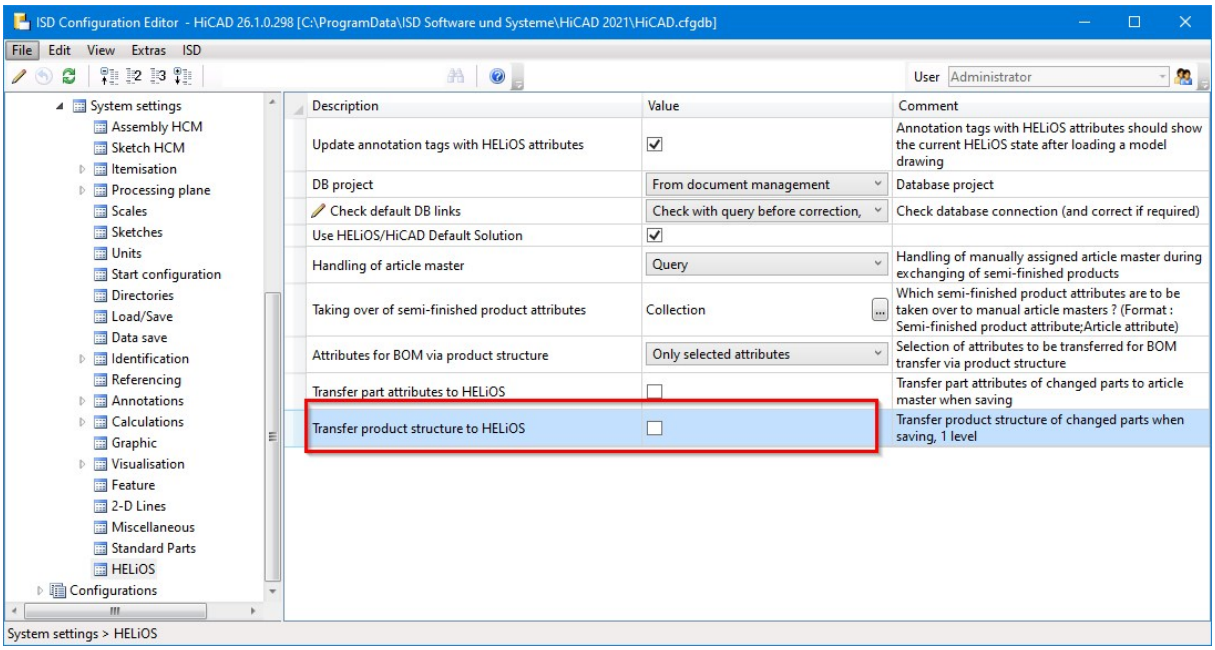
HELiOS in HiCAD

Service Pack 1 2021 (V 2601)

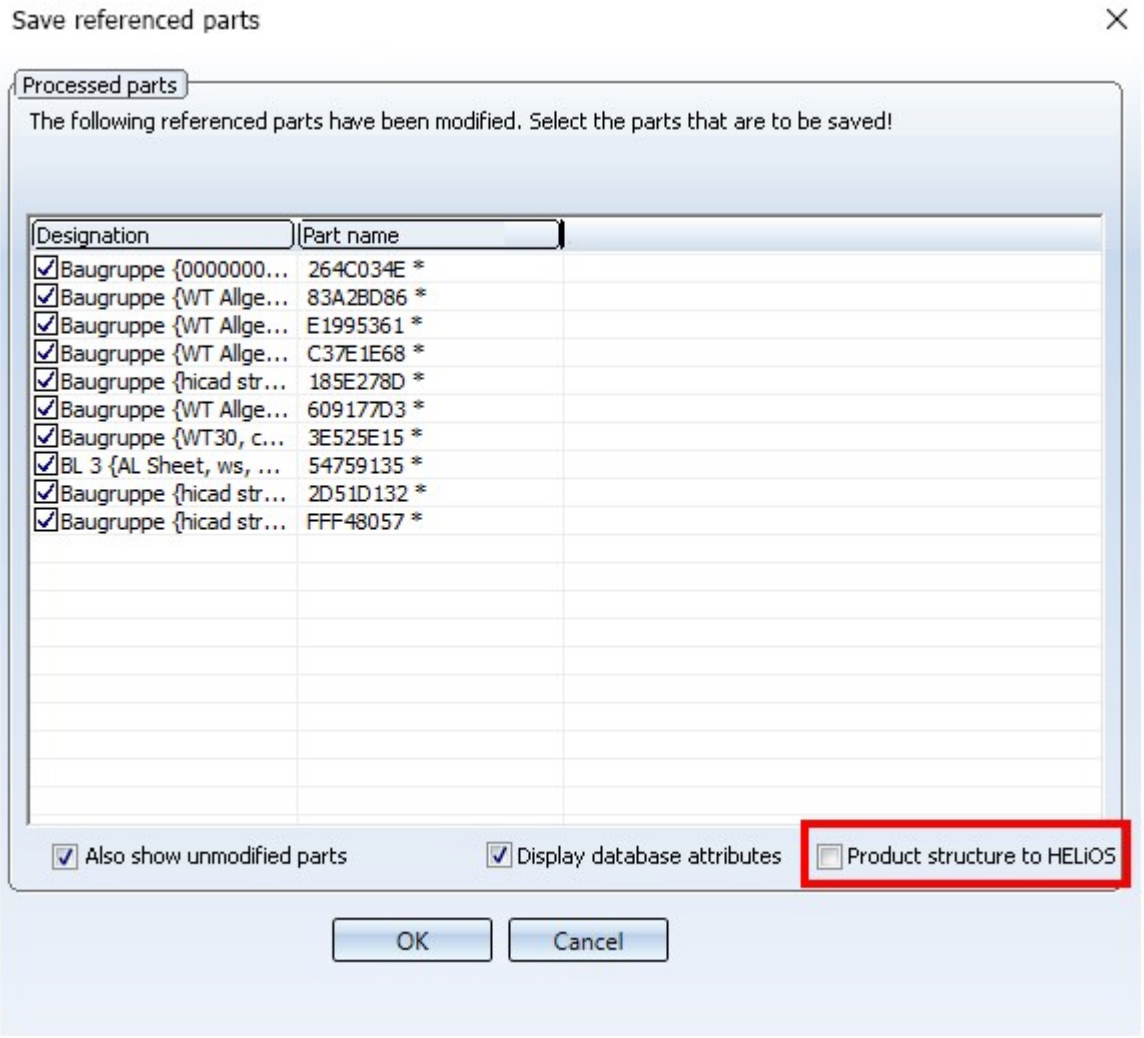
Improved product structure transfer

In the **Configuration Editor** you will find a new **HELiOS** option that take effect when automatically transferring the product structure from HiCAD to HELiOS:

- **Transfer product structure to HELiOS**: Activate this checkbox to ensure that when saving in HiCAD, modified parts with article master assignment and with BOM relevance are automatically transferred to HELiOS as a 1-level structure.



When saving referenced parts in earlier versions of HiCAD, you could ensure that the modified product structure is transferred to HELiOS by activating the **Product structure to HELiOS** checkbox.



If the above-mentioned setting has been activated in the **Configuration Editor**, the option is grayed out at this point and cannot be deactivated, since the product structure is always transferred when saving.

As of HELiOS 2021 Service Pack 1, only **1 level** of the product structure will be transferred, and no longer all sub-part levels of it.

Please note:

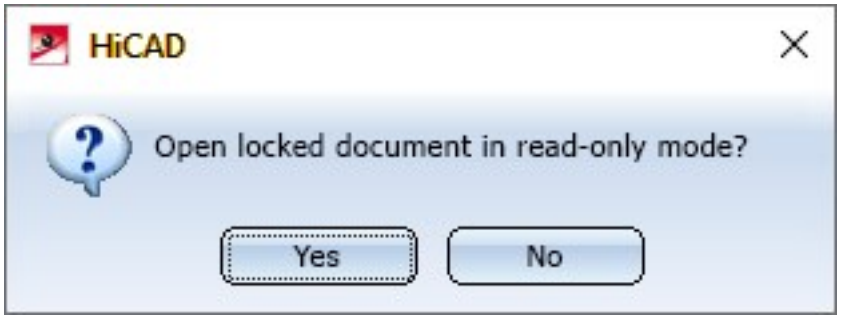
The above configuration setting is only possible if both HiCAD 2021 and HELiOS 2021 have been updated to Service Pack 1.

If one of the two applications is available in a version lower than V 2601, the option is not available.

Opening write-protected files via the Vault Server

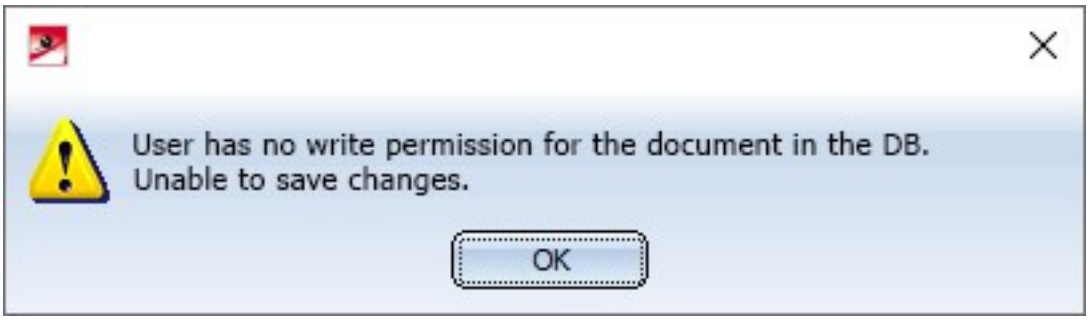
If an SZA file is opened via the Vault Server, which is locked there for writing, then it cannot be saved later either.

When opening such a file, HiCAD displays a corresponding warning message:



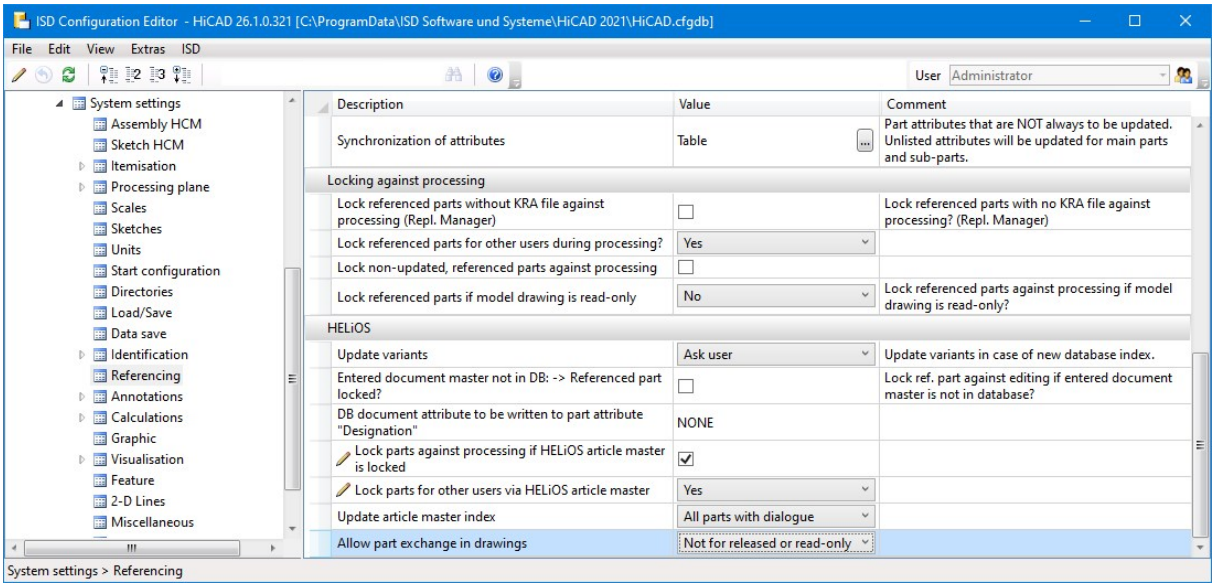
If you select **Yes**, the file will be opened in read mode. If you select **No**, the opening of the file will be cancelled.

If an attempt is made to save a file opened in read mode, the following message appears:



Configuration Editor: Allow part exchange in released drawings

The HELiOS option for setting whether parts of already released drawings can still be exchanged can be found in the **Configuration Editor** as of HiCAD 2021 SP1 (V 2601).

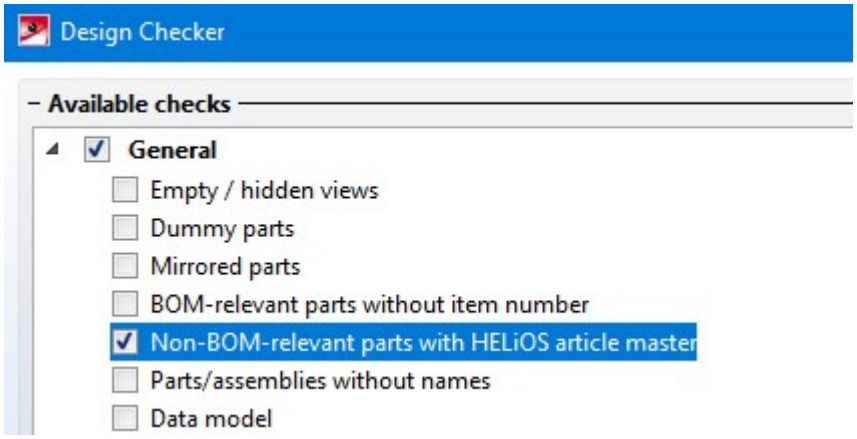


Major Release 2021 (V 2600)

New Design Checker option: Non-BOM-relevant parts with HELiOS article master

In the Design Checker of HiCAD the new test **Non-BOM-relevant parts with HELiOS article master** is available under **General**.

If a model drawing contains parts with article master data, some of which are BOM-relevant and some of which are not, there may be problems when transferring the product structure to HELiOS. With the new test all parts can be found that have an article master but are not BOM-relevant.



You start the Design Checker in HiCAD at **Information > 3-D, Further > Design Checker**.

Discontinuation of the old figure format .FIG

Important:

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

Since HiCAD 2017 we support FGA as figure format (before that FIG). As of HiCAD/HELiOS 2021 or HELiOS 2021 as an update for HiCAD 2019/2020, we require that all figures stored using HELiOS have previously been converted to the new FGA format. To convert existing 2-D FIG files, the tool Converter_FIG_To_FGA.exe is available in the exe directory of the HiCAD installation.

If there are still unconverted FIG files in the HELiOS document database at the time of the database update to HELiOS 2600.0, you will be informed of the outstanding conversion of these files before the database update. In this case, the conversion must be carried out before or at the latest directly after the update using Converter_FIG_To_FGA.exe.

HELiOS MS Office Interface

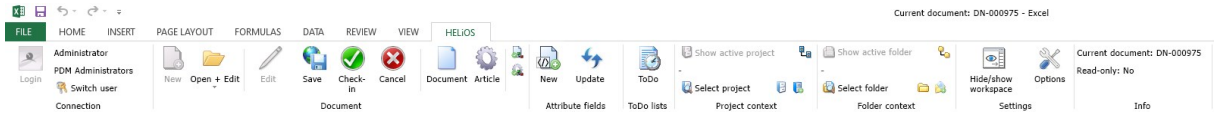
Service Pack 1 2021 (V 2601)

Enhanced Office interface

With the update to HELiOS 2021 Service Pack 1, you will find a revised and extended HELiOS Ribbon menu in the applications Word, Excel, and PowerPoint when Office interface is activated.

New additions include HELiOS functions for editing document and article master data and their workflow status.

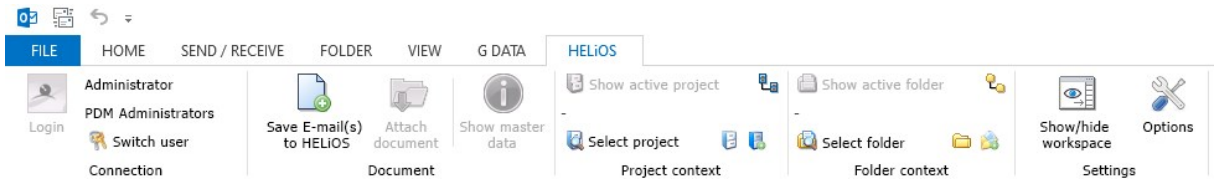
In addition, the functionality in the project context and folder context has been enhanced.



Enhanced Outlook interface

The Ribbon menu bar of the Outlook interface is also available in a revised and extended form as of Service Pack 1.

One of the new features is that you can also call the HELiOS workspace from the Outlook context in order to edit or delete E-mails stored in the database at that location.



Major Release 2021 (V 2600)

Discontinuation of Office 2010

As of HELiOS 2021 (Version 2600), the interface to Office versions older than MS Office 2013 will no longer be supported.

This means that Office 2010 will no longer be supported.

Legal notes

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HiCAD

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HiCAD_2601

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