HiCAD

UNLIMITED CAD PERFORMANCE **DEVELOPED BY ISD**

HiCAD Profile Installation

Version 2022

Adding own profiles

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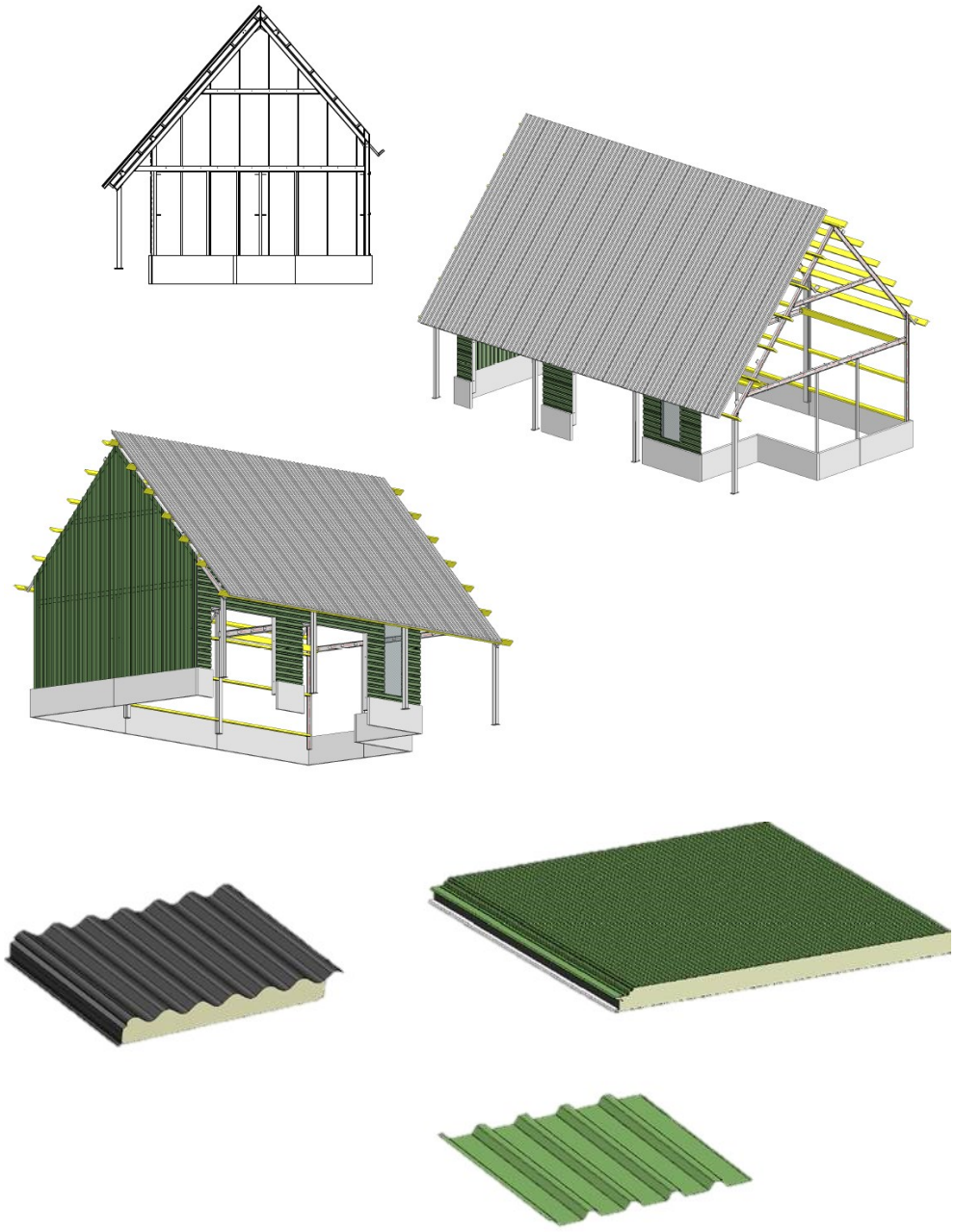
THE WORLD OF CAD AND PDM SOLUTIONS

TOC

Adding Own Profiles

Profiles for Profile Installation are listed in tables that are in turn contained in catalogues. Typically, one catalogue exists per supplier, which in turn contains further catalogues for different types of profiles. In these catalogues you will find one table per profile series, in which the individual profiles are listed. This table must contain particular columns for the Profile Installation module.

This chapter shows you how to add your own profiles to catalogues in order to use them for the “Profile installation” module.



Sketch-based Profiles

The catalogue entries that are relevant for Profile Installation can be found in the catalogue at **Factory standards > Series > Roof wall facade > Room-closing profiles**.

There you will find already some profiles by various manufacturers. You can add your own profiles at **User-defined profiles** if desired.

The cross-sections of the profiles have been stored as HiCAD sketches. The relevant chapters explain how such sketches and the corresponding catalogue entries are created in order to add new profiles.

For this, two steps are required:

- Creation of the sketches and an
- Entry in the catalogue

Sketches for Sketch-based Profiles

The sketches for the profiles must lie in the XY-plane. When creating the sketch, the currently active coordinate system will be taken over. Therefore, the top view should be active when you create the sketch.

For technical reasons, the mirroring of a profile sketch via **3-D Standard > Transform > Mirror part** is not allowed here. Instead, use the function **Sketch > Transform > Mirror GE**.

The sketch must contain a closed polyline, which represents the cross-section of the profile to be created. In particular, an open polyline without thickness is not allowed.

Often,the drawings of profile manufacturers exist in the form of DXF or DWG files. These can be imported to HiCAD and taken over into a sketch with the functions at **Sketch > Derive > Take over edges**. When working with such drawings you should start with applying the function **2-D Geometry > Tools > Sort GE > Entire drawing**. This will clean up smaller technical defects in the drawing such as gaps between the edges.

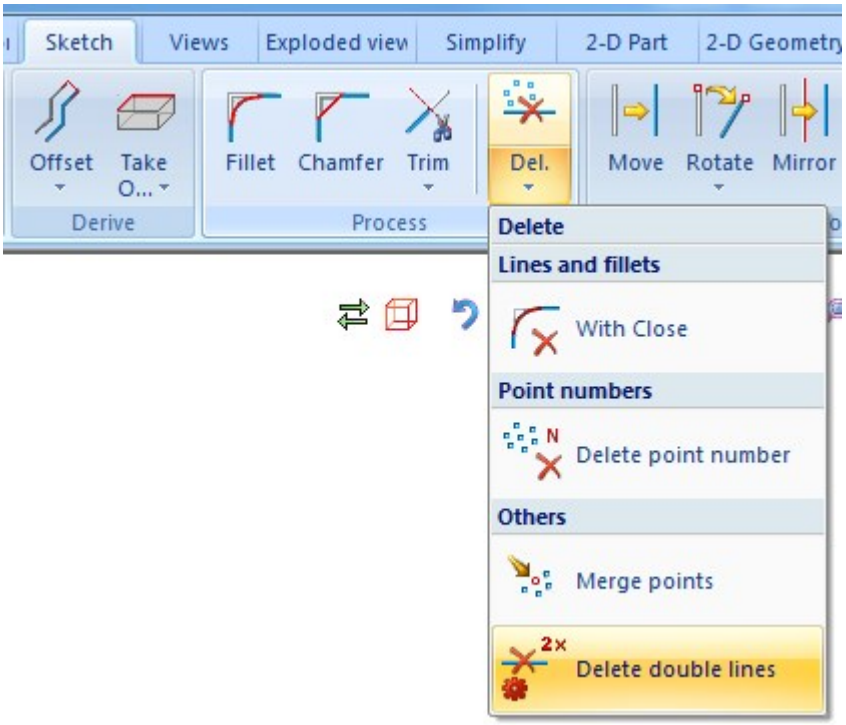
The upper profile side is aligned to the "front" in the profile installation (if you have activated the **Negative direction** checkbox, the alignment will be to the back). "Front" refers to the direction into which the Z-axis of the coordinate of the sketch used for profile installations points.

The origin of the sketch will become the profile axis when the part is created from it. This axis should be located inside the profile, ideally near the centre and on a significant point. If you want to move the origin of the sketch (and thus the axis of the resulting profile) in a single-part profile, you can use the function **Sketch > New > Processing plane > New origin**.

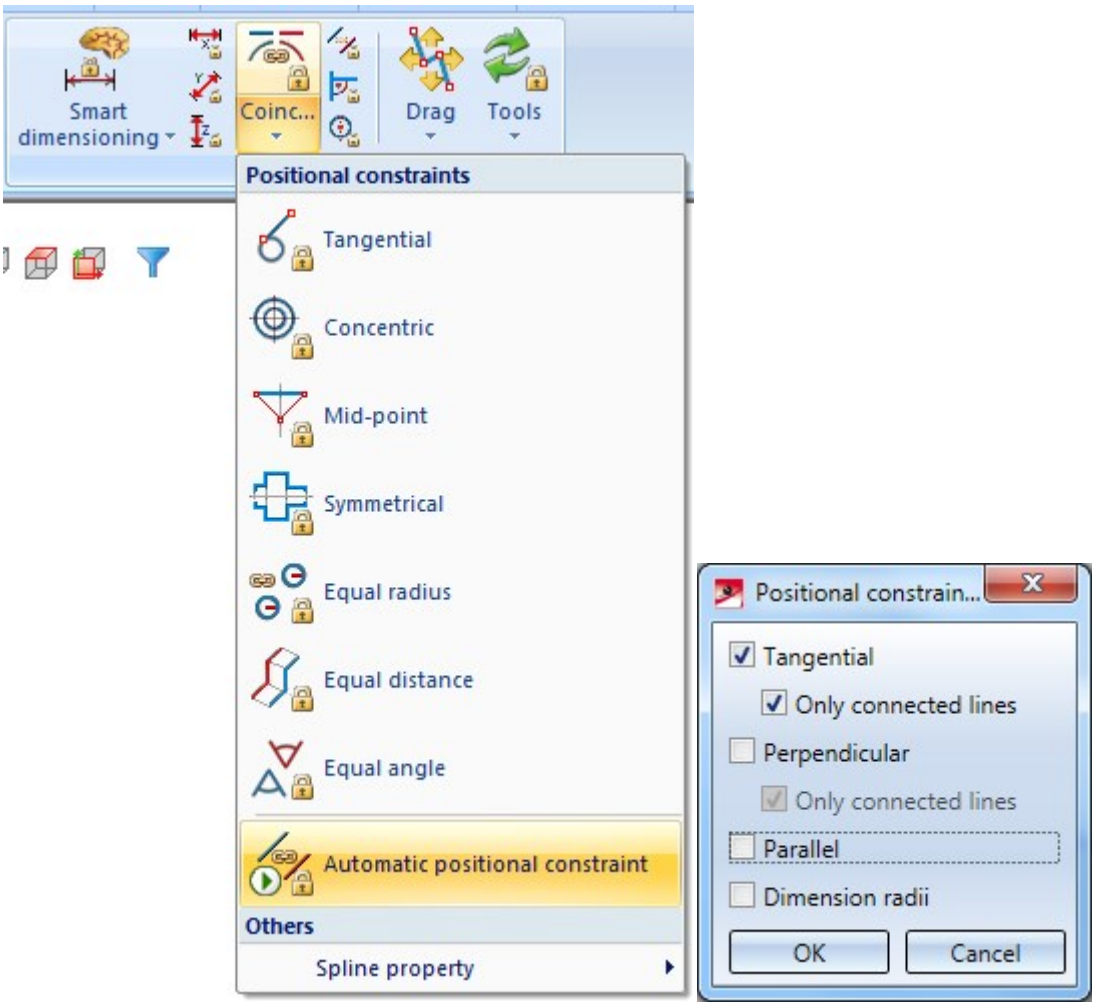
Quality requirements for profile cross-sections

There are some quality requirements with regard to profile cross-sections:

- The cross-sections must not have nay gaps. Otherwise, the profile cannot be created.
- Double line (i.e. two lines on top of each other that appear to be only one line) should be avoided. Otherwise, various HiCAD functions will not work correctly. To delete double lines, you can use the function **Sketch > Process> Del. > Delete double lines**.



- There must not be any self-intersections within polylines. Otherwise, the profile cannot be created. Intersections of different parts, e.g. of neighbouring profiles or individual elements of multi-part profiles (sandwich profile, see below) are not really forbidden, but should be avoided. Otherwise, errors in the representation may occur, such as disappearing profile edges, or edges that are not displayed correctly.
- To allow tangential edge transitions to be hidden in drawings, "almost tangential" transitions will not be sufficient. You should therefore check whether all of the transitions are actually genuinely tangential. You can show tangential transitions with the help of a HCM function: To do this, activate the sketch and choose **Sketch > HCM > Coinc.** > **Automatic positional constraint**.



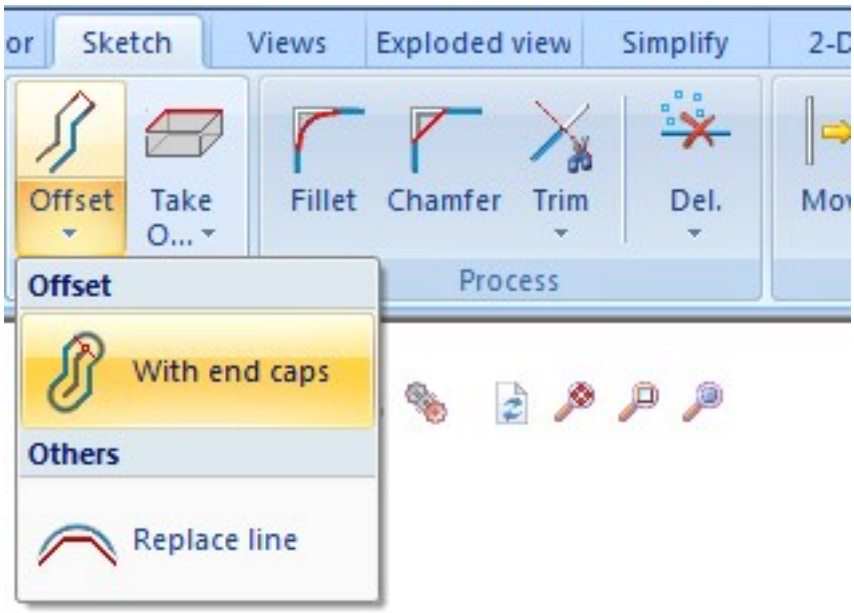
In the dialogue, activate only the **Tangential** checkbox and the **Only connected lines** checkbox beneath it. When you click **OK**, all tangential transitions will be marked with an icon. If any non-tangential edges are found, the resulting edges of which are to be "hideable", this can be corrected by means of a tangential HCM constraint.

- Lines that should actually run exactly horizontally or vertically, but do in fact not fully comply with this requirement, should be avoided. Sometimes, drawings of manufacturers contain lines that deviate only minimally (less than 1 degree). In the resulting profiles, this can already lead to an incorrect behaviour of functions (e.g. subtractions).
- It is desirable that the dimensions specified by the manufacturer are kept to in the drawings. Only in this way can it be ensured that the dimensions that are applied to the resulting profiles are actually identical with the dimensions in the drawing.

Also, it is recommended to apply the function **Sketch > Tools > Sort GE** before saving the sketch. This will eliminate small errors that can occur while the sketch is drawn.

Apply sheet thickness and HCM constraints to polyline

If only one polyline without a sheet thickness exists, HiCAD offers various tools to create this sheet thickness. If the existing polyline is to represent the centre line, you can use the function **Sketch > Derive > Offset > izze > Ableiten > Offset > Offset with end caps**.



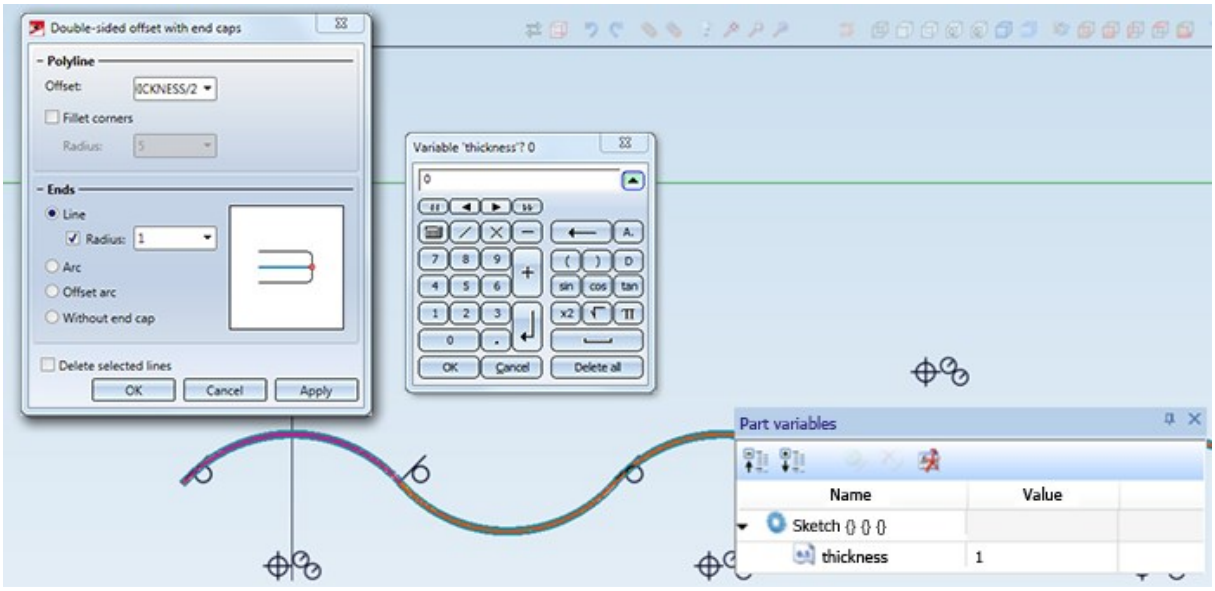
The entered offset will then be added to both sides of the polyline. So, if you want to apply a sheet thickness of 1 mm, you need to enter an offset of 0.5 here.

If you want to apply the offset only to one side of the polyline, you should use the function **Sketch > Derive > Offset, via point and value**. However, the end caps will not be connected in the process. This must be done manually afterwards. This one-sided offset is particularly recommended for tray panels.

Also, sketch-based profiles offer the option to parameterise the geometry via HCM constraints. For cross-sections of one-part profiles such as trapezoidal profiles or corrugated profiles, identical basic forms with different sheet thicknesses are often required. Instead of creating and saving these thicknesses one by one, it is more convenient to create the sheet thickness as a parameter. These part variables can later be assigned values via catalogue columns. In this way it is possible to visualize a multitude of geometries by means of only one sketch.

For this to happen, you need to enter a variables name instead of a numerical value when entering the sheet thickness or other parameterisable values. This name will then be automatically entered as a part variable. The part variable can later be assigned a value via a same-named catalogue column.

Here, too, please note that when applying double-sided offsets, the value will be applied to both sides, starting from the centre line. Therefore, it makes sense to enter, e.g. a value of "THICKNESS/2" for these sketches. In this way, a thickness of 0.5 mm from the catalogue is applied to both sides of the centre line, resulting in the desired total thickness of 1 mm.



Multi-part profiles

For multi-part profiles such as sandwich profiles, one sketch needs to be created for each individual part. the sketches are grouped in HiCAD with the help of a dummy part, resulting in a structure that looks as follows in the ICN:

Designation	Item number	Comment
DRAWING1		
ONDAFIBRE 3003 B (40 mm)...		Sandwich panel
Insulation		Insulation
Outer sheet		Sandwich panel, outside sheet
Inner sheet		Sandwich panel, inside sheet

The profile axis is created in the origin of the uppermost dummy part. In contrast to sketches, there is no particular function for placing of the origin for dummy parts.

However, to determine the position of the origin, you can use the property that the World CS is always used whenever a dummy part is created. So, to use a particular point on the geometry as the origin of the uppermost dummy part, you first need to move the entire geometry in such a way that this point coincides with the origin of the World CS. Then, create a new dummy part.

Other properties

In addition to the geometry, the sketches and the uppermost dummy part can have other properties.

- The **Part name** for profiles is taken from the catalogue column BZ, i.e. a manual naming is not necessary.
- The **Article number** is used for default settings as the displayed name in the ICN. Since the article number of the sketch and the uppermost dummy part will e taken over to the resulting profile, the name of the profile should be entered here.
- For the uppermost part (i.e. depending on the profile type, the uppermost dummy part or the sketch) activate the **BOM-relevance**. This property will then be transferred to the resulting profile.

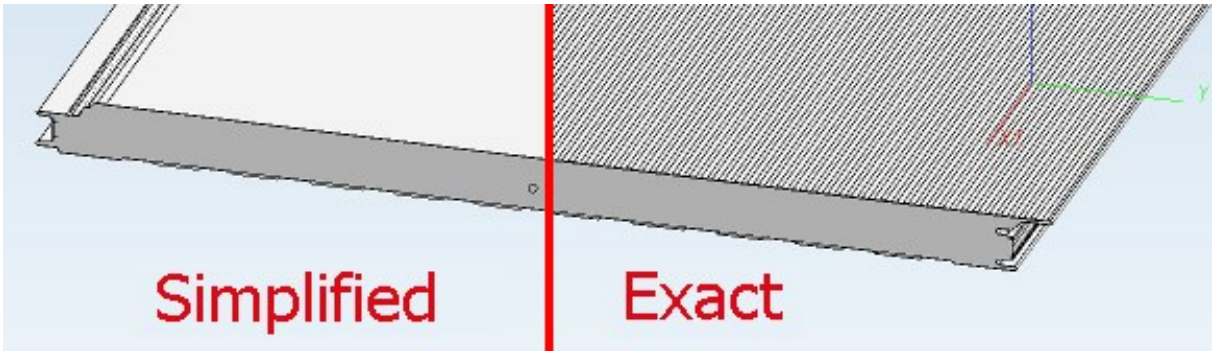
- For the sketches or the uppermost dummy part you should also specify **Usages**. In the catalogue, there are already a number of suitable entries. For instance, at **Civil Engineering > Profile installation > Elements** you can find **R/W trapezoidal profile** and **R/W panel**; at **Civil Engineering > Profile installation > Element parts** you can find the usages **R/W inner sheet**, **R/W outer sheet** and **R/W insulation** for sandwich profiles. In particular the usage for the elements of sandwich profiles are mandatory, since this information is required to apply the insulation material and the inside and outside colours to the corresponding parts.
- Similar to the usages, you can specify a **Part type** for sketches or the uppermost dummy part. Here you use no catalogue entries, but you can enter free texts here. For the most frequent cases special abbreviations are available, which can also be automatically translated. These are listed in the table below:

Part type in sketch property	Resulting part type in profile
TT(TrapezoidalProfile)	Trapezoidal profile
TT(SinusoidalProfile)	Sinusoidal profile
TT(LinerTray)	Liner tray
TT(SpecialProfile)	Room-closing special profile
TT(SandwichPanel)	Sandwich panel
TT(SandwichPanelOutsideSheet)	Sandwich panel-Outer sheet
TT(SandwichPanelInsideSheet)	Sandwich panel-Inner sheet
TT(DWF_clad_insulation_name)	Insulation

  If these abbreviatons are not used, the TT() is not necessary either! The entering of the part type is not mandatory when inserting own profiles. So, assign this value only if it is really required.
- No manual entering of Material and Weight for sketches will be necessary - this will be done via the catalogue.
- You have the option to create transparent profiles. To do this, place the uppermost part on Layer 40.

Exact and simplified representation

HiCAD offers the option to switch between an exact and a simplified representation. This option is used for surface profiles to switch the representation of the surface structures of profiles on or off. This option can be useful for performance reasons.



To ensure that this switching works properly, one or several parts of the profile cross-must contain sections for both cases. This is achieved by creating a dummy part (instead of a sketch) which contains two sketches - one with the name SIMPLE and one with the name EXACT. SIMPLE and EXACT must have been entered for both the part name and the article number of the sketch. The simplified and the exact cross-sections will be placed into these sketches.

All further properties such as **Usage** and **Part type** are assigned to the dummy part, so they needn't be entered twice (once for the SIMPLE and once for EXACT sketch).

Here is an example of a sandwich profile structure, where the Outer sheet and the Inner sheet also contain simplified geometries, but not the Insulation and the Sealing cord:

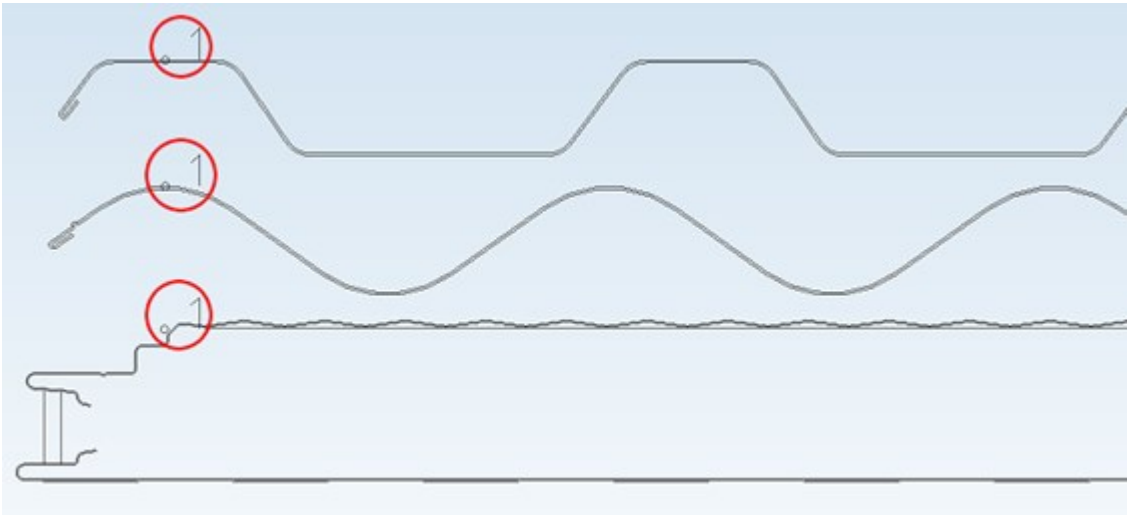
Designation	Item number	Comment
DRAWING		
Ondatherm 2000 B 60		TT(SandwichPanel)
Insulation		TT(DWF_clad_insulation_name)
Sealing cord		
Outer sheet		TT(SandwichPanelOutsideSheet)
EXACT		TT(SandwichPanelOutsideSheet)
SIMPLE		TT(SandwichPanelOutsideSheet)
Inner sheet		TT(SandwichPanelInsideSheet)
EXACT		TT(SandwichPanelInsideSheet)
SIMPLE		TT(SandwichPanelInsideSheet)

Insertion point

Profiles often possess visually prominent points, an easy and convenient placing of which on facades would be desirable, e.g. the middle of the first rib on trapezoidal or sinusoidal profile, or the joint between two sandwich profiles. The Profile Installation module enables you to create a situation where it is not the left edge of a profile, but such a special point that is located in a selected start point of the installation.

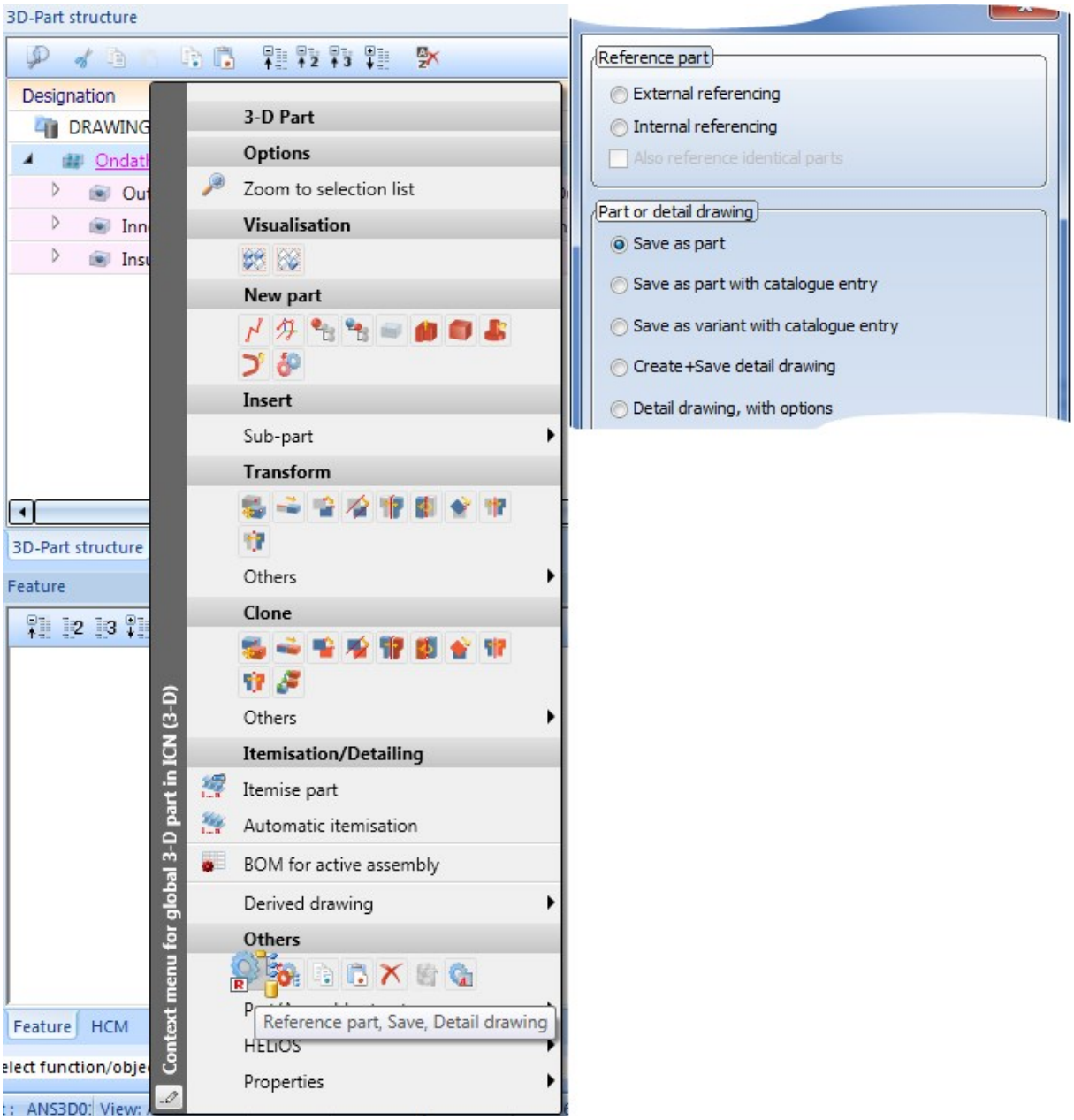
For this to happen, first create an isolated point with the function **Sketch > New > Point**. The point should be named "1" with the function **Sketch > New > Point > Point number**. It designates the moving of the profile during its creation with the Profile Installation module. The point must be created in the uppermost part - i.e. in the sketch in case of single-part profiles, or in the dummy part in case of multi-part profiles.

This behaviour can be deactivated in the dialogue. These points will then be ignored, and the start point will be on the left profile edge instead.



Save

To save a profile cross-section, the entry for the superordinate dummy part or the sketch must be marked in the ICN. Right-click the entry and choose **Others > Reference part, Save, Detail drawing**. In the dialogue window, choose **Save as part**.



The file path should be in the HiCAD installation directory at "…\Kataloge\Werksnormen\Baureihen\Dach Wand Fassade\Raumabschließende Profile“.

Catalogue entry

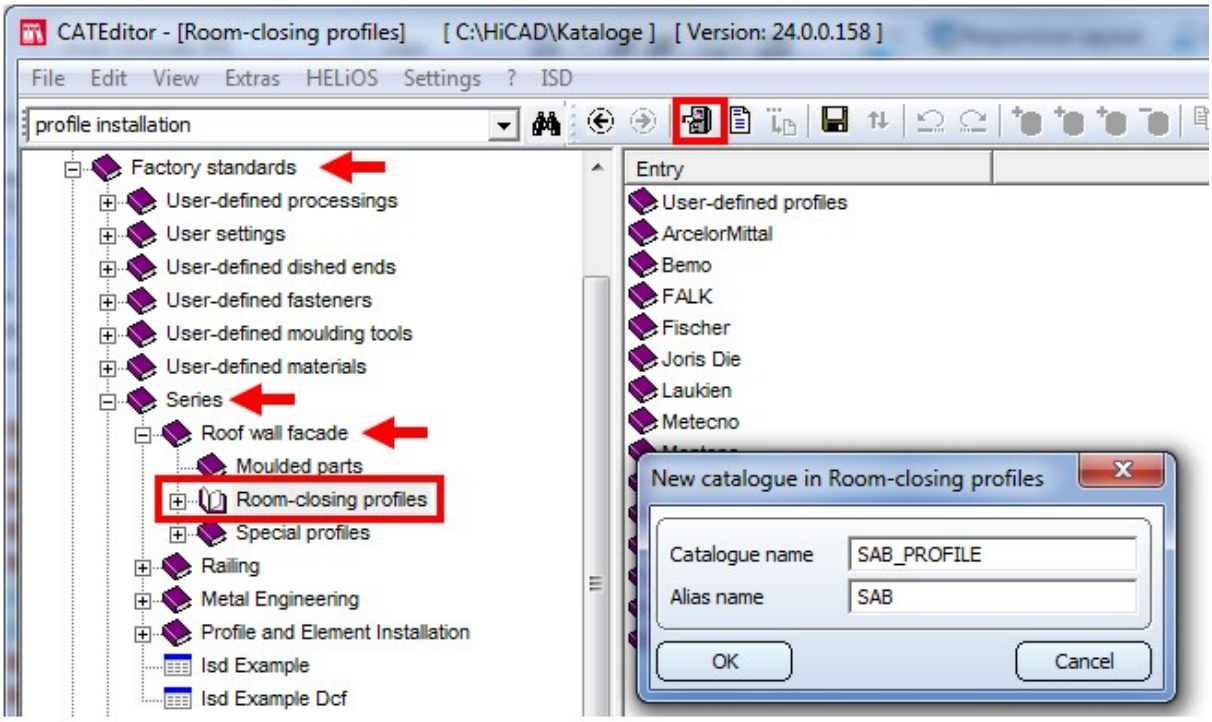
The second step required for a creation of own profiles is the creation of a catalogue entry.

Catalogue Entries for Sketch-based Profiles

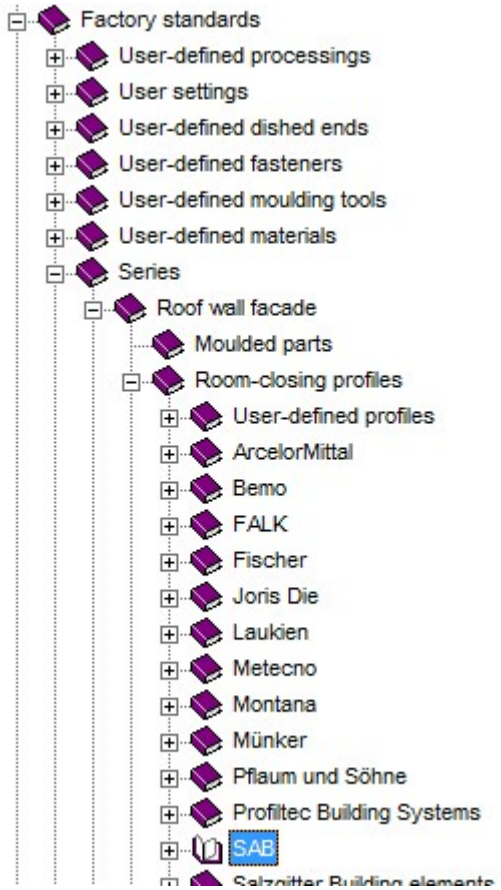
Profiles are listed in tables, which are contained in catalogues.

Typically, there is one catalogue per manufacturer, containing further catalogues for the different types of profiles. In these catalogues you find one table per profile series, in which the individual profiles are listed.

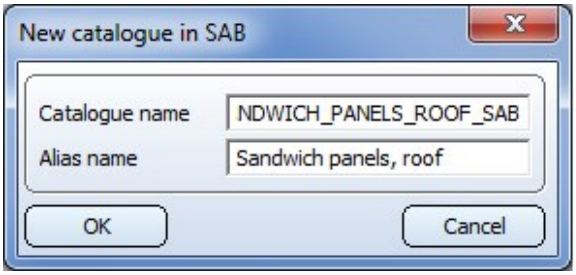
The following example shows the creation of a new catalogue for SAB profiles.

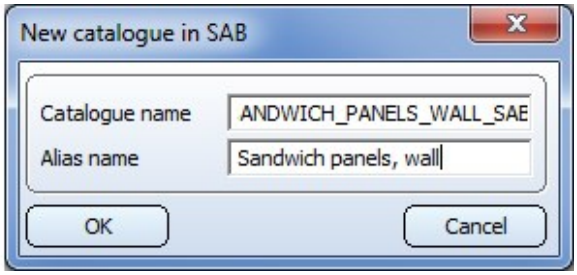


Use only the name of the manufacturer as the Alias name.

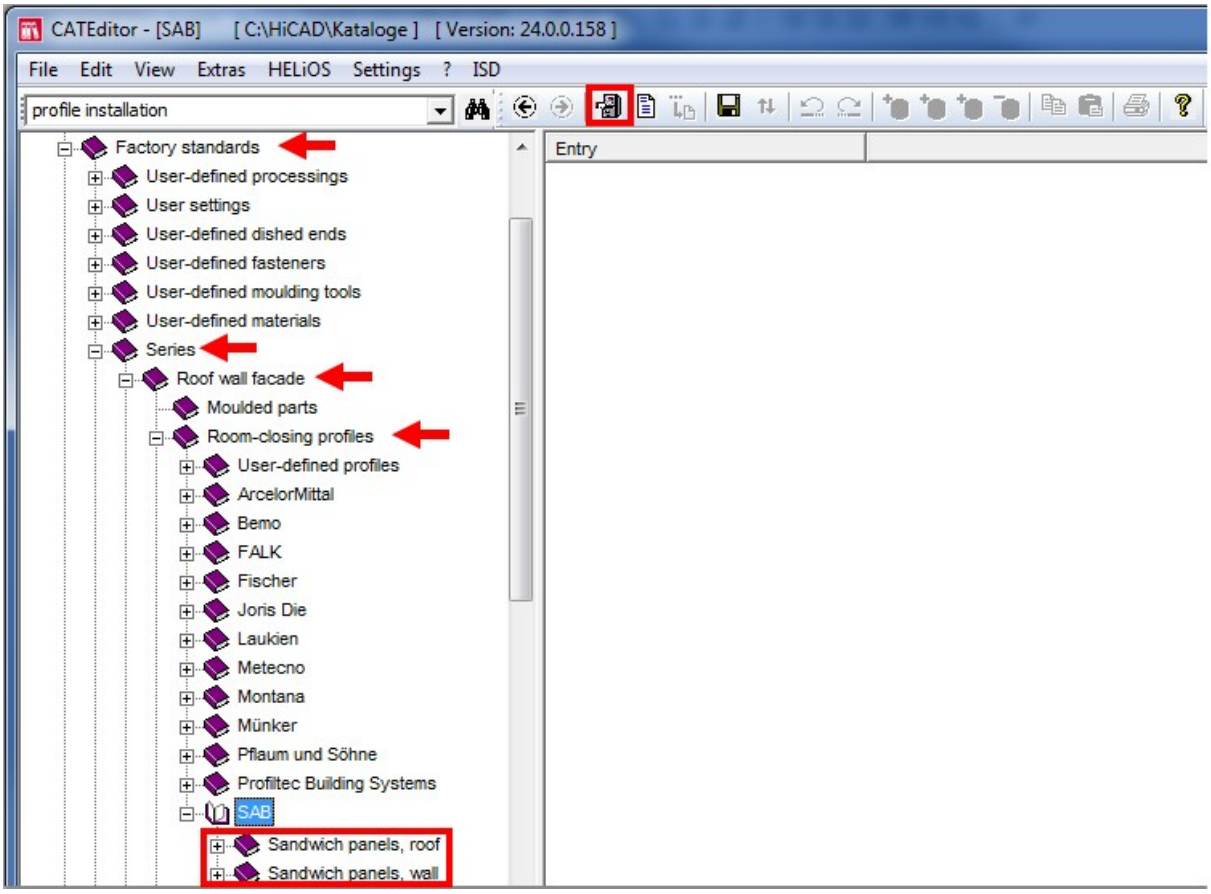


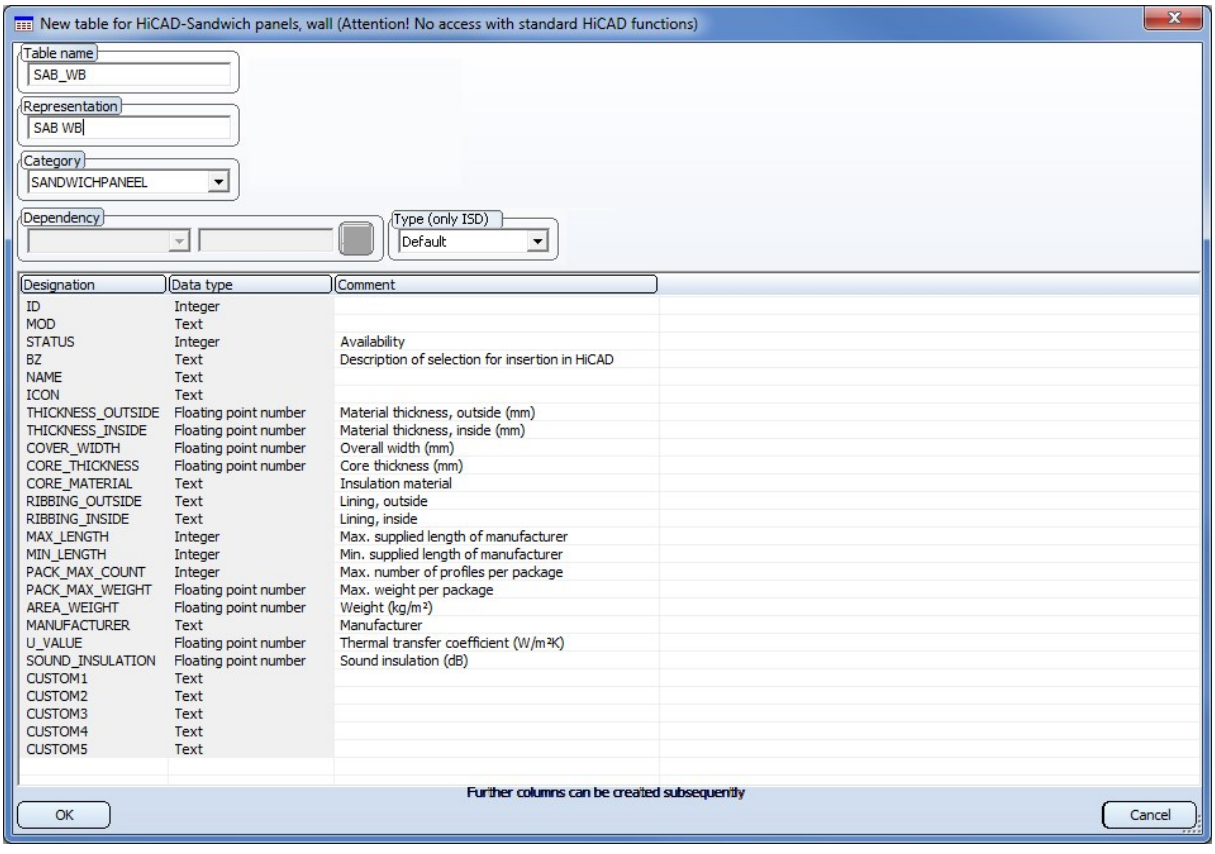
Now, select the new SAB catalogue. Beneath it, create further catalogues. For the catalogue name, use a combination of the profile designation and the supplier.

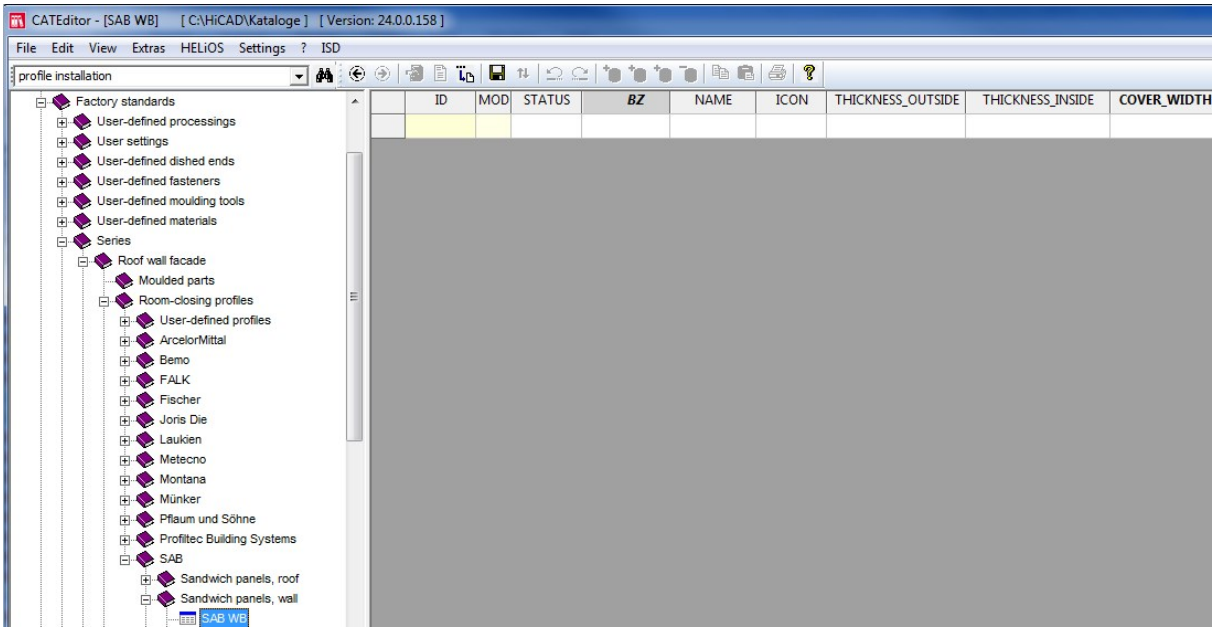




After creating the two catalogues, enter a table for a special product:


CATEditor - [SAB] [C:\HiCAD\Kataloge] [Version: 24.0.0.158]
File Edit View Extras HELiOS Settings ? ISD
profile installation
Factory standards
User-defined processings
User settings
User-defined dished ends
User-defined fasteners
User-defined moulding tools
User-defined materials
Series
Roof wall facade
Moulded parts
Room-closing profiles
User-defined profiles
ArcelorMittal
Bemo
FALK
Fischer
Joris Die
Laukien
Metecno
Montana
Münker
Pflaum und Söhne
Profiltec Building Systems
SAB
Sandwich panels, roof
Sandwich panels, wall
Entry


New table for HiCAD-Sandwich panels, wall (Attention! No access with standard HiCAD functions)
Table name
SAB_WB
Representation
SAB WB
Category
SANDWICHPANEEL
Dependency
Type (only ISD)
Default
Designation | Data type | Comment
ID | Integer
MOD | Text
STATUS | Integer | Availability
BZ | Text | Description of selection for insertion in HiCAD
NAME | Text
ICON | Text
THICKNESS_OUTSIDE | Floating point number | Material thickness, outside (mm)
THICKNESS_INSIDE | Floating point number | Material thickness, inside (mm)
COVER_WIDTH | Floating point number | Overall width (mm)
CORE_THICKNESS | Floating point number | Core thickness (mm)
CORE_MATERIAL | Text | Insulation material
RIBBING_OUTSIDE | Text | Lining, outside
RIBBING_INSIDE | Text | Lining, inside
MAX_LENGTH | Integer | Max. supplied length of manufacturer
MIN_LENGTH | Integer | Min. supplied length of manufacturer
PACK_MAX_COUNT | Integer | Max. number of profiles per package
PACK_MAX_WEIGHT | Floating point number | Max. weight per package
AREA_WEIGHT | Floating point number | Weight (kg/m²)
MANUFACTURER | Text | Manufacturer
U_VALUE | Floating point number | Thermal transfer coefficient (W/m²K)
SOUND_INSULATION | Floating point number | Sound insulation (dB)
CUSTOM1 | Text
CUSTOM2 | Text
CUSTOM3 | Text
CUSTOM4 | Text
CUSTOM5 | Text
Further columns can be created subsequently
OK
Cancel



Save the created table.

The following categories have be defined beforehand, including the necessary columns for the tables:

- Trapezoidal profile
- Sinusoidal profile
- Room-closing special profile
- Liner tray
- Sandwich panel

However, you have the option to expand the tables by adding further columns.

Meaning of the columns for Sandwich panels

Designation	Content	Data type
BZ	Designation of the profile; is made the Article number of the profile	Text
NAME	Path to the part with the sketches (see Step 1), relative to the Werks-normen (Factory standards) catalogue	Text
THICKNESS_ OUTSIDE	Material thickness, outside (mm)	Floating point num-ber
THICKNESS_INSIDE	Material thickness, inside (mm)	Floating point num-ber
COVER_WIDTH	Installed width (mm)	Floating point num-ber

CORE_THICKNESS	Core thickness (mm)	Floating point number
CORE_MATERIAL	Insulating material; will be applied to the part with the usage **R/W insulation**	Text
RIBBING_OUTSIDE	Outside ribbing (line pattern)	Text
RIBBING_INSIDE	Outside ribbing (line pattern)	Text
MAX_LENGTH	Maximum profile length	Integer
MIN_LENGTH	Minimum profile length	Integer
PACK_MAX_COUNT	Maximum number of profiles in one package	Integer
PACK_MAX_WEIGHT	Maximum weight of onepackage	Integer
AREA_WEIGHT	Weight referring to the installed width (kg\m²)	Floating point number
MANUFACTURER	Name of manufacturer; can be output in BOMs	Text
U-VALUE	Thermal transfer coefficient	Floating point number
SOUND_INSULATION	Acoustic insulation of the part (dB)	Floating point number

Meaning of the columns for single-part profiles

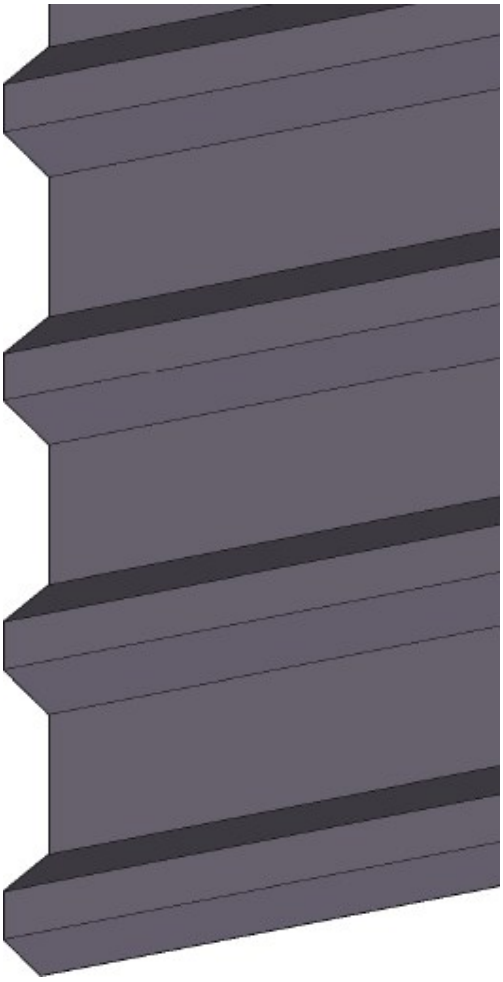
Designation	Content	Data type
BZ	Designation of the profile; is made the Article number of the profile	Text
NAME	Path to the part with the sketches (see Step 1), relative to the Werksnormen (Factory standards) catalogue	Text
THICKNESS	Material thickness (mm); controls the sheet thickness in sketches with HCM constraints	Floating point number
COVER_WIDTH	Installed width (mm)	Floating point number
HEIGHT	Height (mm)	Floating point number
SUITABILITY	Suitability: 0=Roof, 1=Wall, 2=Roof/Wall, 3=No information	Integer
MATERIAL	Material, applied to part	Text
MAX_LENGTH	Maximum profile length	Integer
MIN_LENGTH	Minimum profile length	Integer
PACK_MAX_COUNT	Maximum number of profiles in one package	Integer
PACK_MAX_WEIGHT	Maximum weight of one package	Integer

AREA_WEIGHT	Weight referring to the installed width (kg\m²)	Floating point number
MANUFACTURER	Name of manufacturer; can be output in BOMs	Text

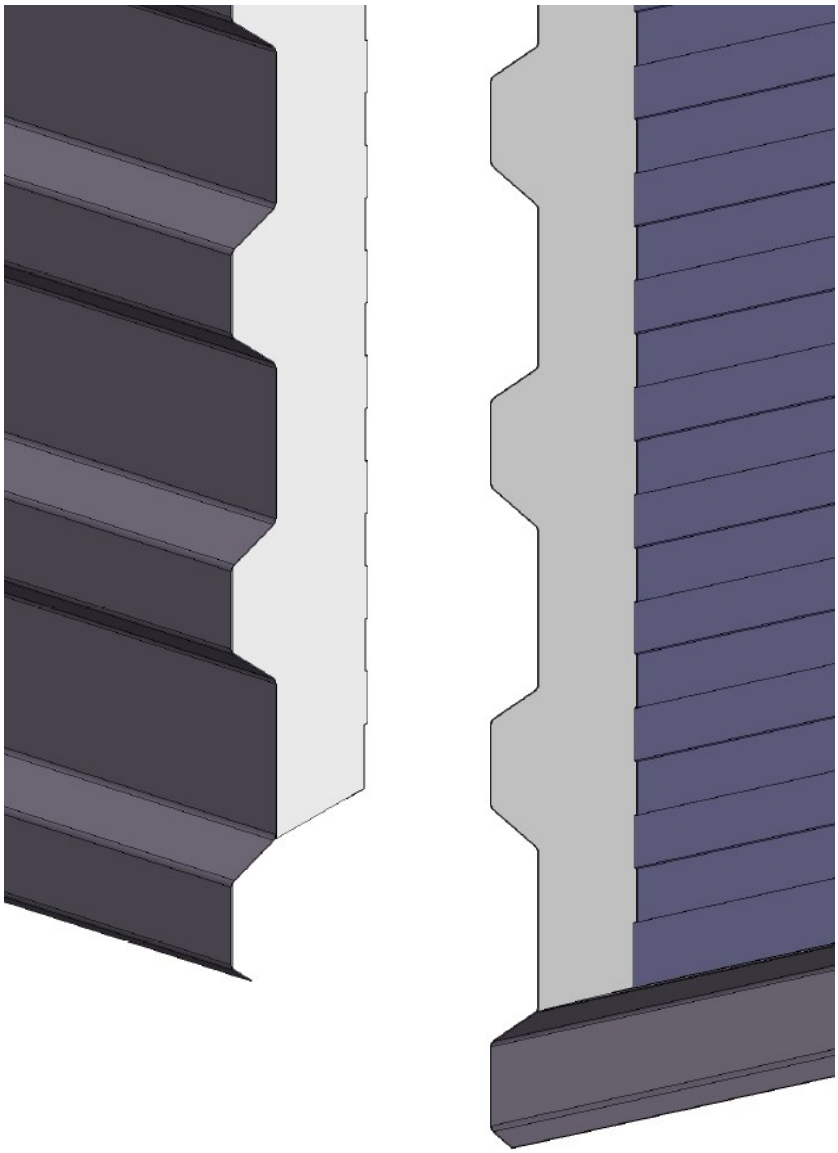
To be able to create a profile, only the columns BZ, NAME, COVER_WIDTH and maybe THICKNESS would be, theoretically speaking, sufficient. Further entries determining the weight, material, manufacturer or similar, should, however, also be entered, because otherwise these properties cannot be generated correctly. Some columns, such as RIBBING_OUTSIDE, U-VALUE will not be used for profile installation and serve only the purpose of providing information in the catalogue.

Examples From Practice

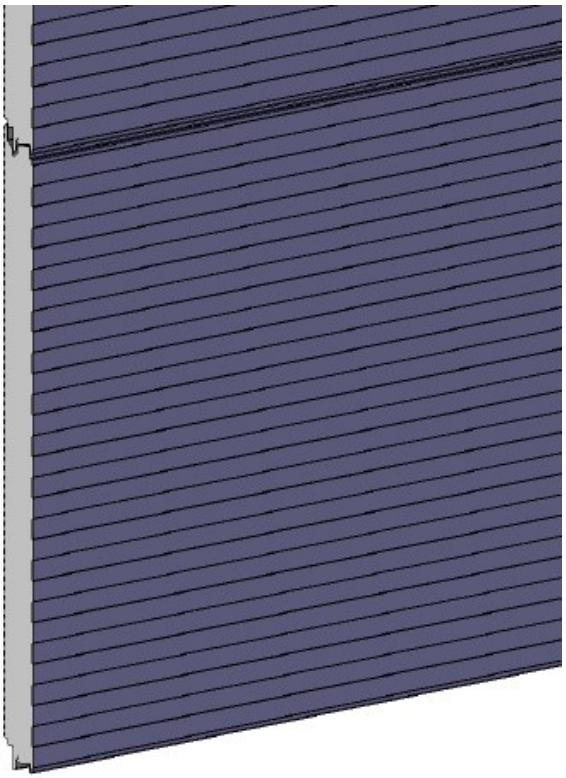
Hardemann: Single skin wall 32 1000 DW

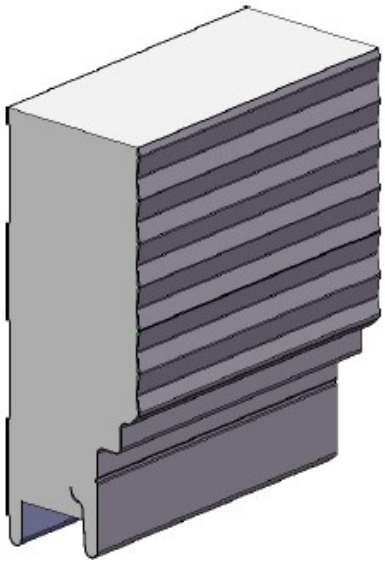


Kingspan: Double skin roof panels KS1000 FC



Falk: Double skin wall 1060 WB





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