HiCAD

Metal Engineering Training

HIC-TR-GB-MTE-211

Metal Engineering Training

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1 Quick Overview

Text Section + Exercises Section

Contain numerous Metal Engineering exercises.

Glossary

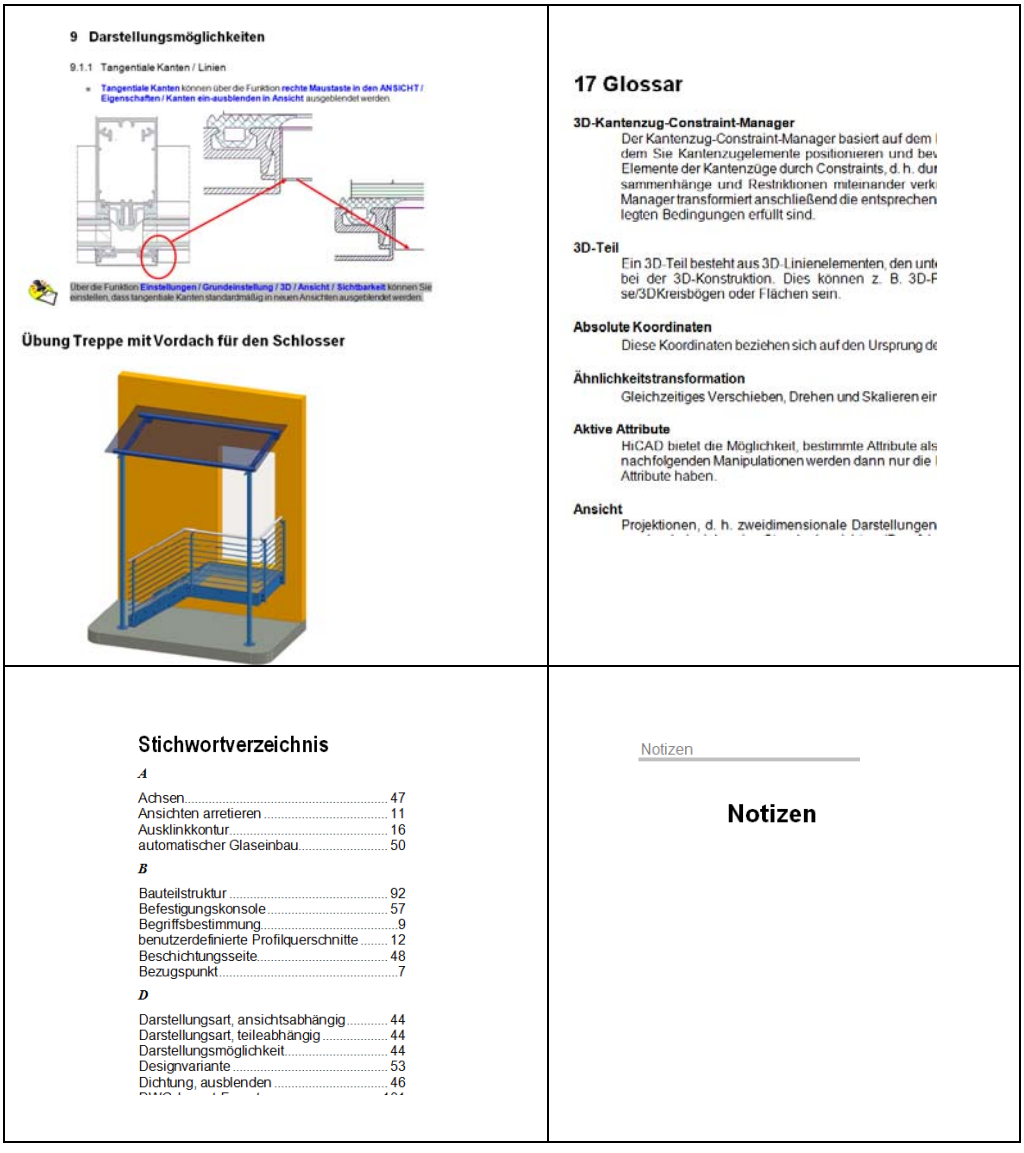
Describes and explains the basic CAD terms.

Index

Contains an alphabetical list of the training book contents, with references to the corresponding pages.

Notes

Empty space where you can make your own notes on the Metal Engineering Training.



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2 Training Content and Requirements

The “HiCAD Metal (Glass/Facade) Engineering, Advanced“ training is a follow-up training, building on the knowledge conveyed in the “HiCAD Metal (Glass/Facade) Engineering, Basic“ training. It addresses users who already have a good knowledge of the HiCAD modules 2-D, 3-D, Sheet Metal and Steel Engineering, which is required for this training.

Besides explaining the special functions for 2-D and 3-D glazing constructions, this training is meant to convey the **optimal designing methods for the complete construction of complex Metal Engineering objects in 3-D, and the connection of elements required in drawings and the actual construction.**

The training will convey

- practice-oriented, efficient methods for the designing of Metal Engineering objects,
- methods for the designing of wall connections in 3-D,
- methods for the designing of Sheet Metal parts in 3-D for wall connections and transitions,
- methods for the designing of geometries in 3-D that are already exist on-site.

Besides the designing in 3-D, the training also explains the creation of customer/architect/production documents. These include:

- the deriving of sectional views,
- the creation of overviews and details,
- the generation of dimensioned profile production drawings,
- glass pane drawings for orders,
- sheet developments, or
- Bills of Materials (BOMs)

on the basis of the data of the 3-D drawing.

If desired, advanced functionalities, e.g. for

- the definition of multi-part standard beams and profiles, or
- the creation of user-defined catalogues for frequently required purchased parts and recurring assemblies

can also be explained in the training.

To enable a simplified insertion of beam and profile cross-sections from different facade system suppliers, the interface with the software program **LogiKal** by the company **ORGADATA** will be used in the training. Users who do not have this software and/or the corresponding HiCAD interface, or customers using self-defined beam and profile systems will be shown how to define their own beam and profile cross-sections.

3 FAQ

Experience has shown that some questions arise particularly frequently. To spare you unnecessary browsing through the pages we will begin with 13 FAQs:

Question 1: What are the functions of the mouse buttons?

a) Left mouse button (LMB): Select function / confirm entry (e.g. a radius)

b) Middle mouse button (MMB): End function / cancel

c) Right mouse button (RMB): Context menu (you can right-click on an icon, a part, the drawing area, a dimensioning, a text and numerous other objects).

Question 2: How do I select a point option?

a) All point options represented by *one* letter (R, A, D, W, P, N) can be selected via the corresponding keyboard button.

b) Press RETURN key on the keyboard

c) Right-click just when HiCAD suggests a point option

d) (Quick) double left-click

Question 3: How do I draw a closed, i.e. uninterrupted polyline even if various point options are required:

a) Left-click on the drawing area and select new point option

b) Overwrite previous point option with new point option (only possible for point options represented by one or two letters)

c) Right-click > Polyline > Continue, and select new point option

Question 4: Which point options do I select if a reference point is required?

Select “Relative-Return“ (REL RET). Just press the “R“ key for point option „Relative“, then confirm by pressing the RETURN key.

Question 5: Which types of re-use are available?

a) Clipboard

b) Drawing > Insert Part > Exp. > From drawing, Via document master

c) Drawing > Insert Part > Insert part, via DB document master

d) Drawing > Insert Part > Insert main part, via Standard Part catalogue

e) Insert Part > User Library

In this context please also note how you save parts to a catalogue, resp. to the database.

Question 6: How do I close a contour in a 3-D Sketch?

a) Sketch > Tools > Sort GE in active sketch

b) Information > 3-D, Further > Line > Open polylines

c) Information > 3-D, Further > Line > Self-intersection

After further changes, select “Sort GE in active sketch” again.

Question 7: I want to convert a 2-D part to a 3-D part. A closed contour is required for this. How do I make sure that the contour of the 2-D part is closed?

a) 2-D Geometry > Tools > Sort GE

b) 2-D Geometry > Change > Delete > Others > Double lines

c) Information > 2-D, Further > Line > Polyline

After further changes, select “Sort GE” again.

Question 8: How do I perform a displacement by a particular value with a displacement vector?

Start point: Select an arbitrary point in the drawing area. End point: Relative.

Question 9: How do I change the grid in Sketch functions?

a) "–" key: refine the grid; (e.g. 5 mm steps instead of 10 mm steps).

b) "+" key: coarsen the grid; (e.g. 50 mm steps instead of 10 mm steps).

c) Space bar: Direct value entry.

Question 10: How do I convert a HiCAD drawing to a graphic?

a) CTRL + C > Select format and size > Select detail / Select background colour

b) CTRL + W > Select detail / …

c) "PRINT" key

d) Activate the "Drawing" tab and select Save/Print > SaveAs > Conversion > Create Enhanced Metafile (EMF). You can also select a superordinate part with a *right-click*.

Question 11: How do I load the automatic "10 Minutes AutoSave" option?

Drawing / Open / Local Disk / hicad / temp / switch to "SZN" extension in the textbox „Files of type" / select **hicad_N1**. During this process, N is the number of the drawing on which you have worked. If you want to load the drawing represented by the third thumbnail in the „Switch drawing " tab, load **hicad_31**; if you have worked on the drawing represented by the first thumbnail, select the drawing **hicad_11**.

Question 12: How do I exchange the ISD logo in the drawing frame with my own company logo?

a) Drawing > Insert Part > Insert drawing frame > Do not complete title block > select a frame, e.g. DIN A0 > insert this frame in your drawing.

b) In the "2-D Part structure" tab of the ICN, right-click on the name of the *drawing*. Select the "Auxiliary parts On/Off" function, then select "Auxiliary parts On".

c) Delete the sub-part with the old logo and create a new sub-part (e.g. with the name "Logo") below the "SCHRIFE" entry. Draw the new logo there, e.g. with lines, texts etc. You can also create a new sub-part to the part "Logo", draw a company logo as a closed contour there and hatch it with very narrow hatching line spacing (e.g. 0.05 – an example for this would be the "HiCAD eye" symbol). Save the part "Logo" as *Part*.

d) Activate the entire drawing frame and save it as 2-D part without database.

   i) with the name "DINA0" suggested by the computer, if you are creating a frame for your own company

   ii) with a different name, if you are creating a frame for a customer, for whom you regularly create drawings

e) Now load, one after another, frames DIN-A1 to DIN-A4. For each of them you can delete the sub-part with the old logo and load your own company logo saved in step c), as a sub-part to the part „SCHRIFE". Then continue with step d).

Question 13: How do I achieve that HiCAD does not suggest any links when loading a drawing from the database?

During part-oriented working you assign a part master to parts and drawings. Drawings are typically loaded with links in the process. During document-oriented working you can switch off the suggested links permanently by setting the following entry in the system file ALGPAR.DAT to 0:

Use Helios/HiCAD default configuration - Helios/HiCAD-Standardlösung benutzen.

4 Metal Engineering-Specific Terms

The section below lists and explains the Metal Engineering-specific technical terms that will be used in this training. These are as follows:

Inserts

Glass panes, windows, doors, casements etc.

Facade

The planar, glazed area of a glazing construction.

Glass

All glass types, e.g. single glass panes, insulating glass etc.

LogiKal

A facade planning program by the company ORGADATA.

Mullions

The vertical metal beams in a facade.

Transoms

The horizontal metal beams in a facade. These are connected with the mullions or with other transoms.

Beam/Profile group

A set of several individual 3-D beams and/or profiles that can be created and processed together (e.g. mullions/transoms complete with cover strips, sealings, etc.).

Beam/Profile cross-section

2-D cross-section of a set of individual beams/profiles (e.g. mullions/transoms complete with cover strips, sealings, etc.) that form a beam/profile group in 3-D when being inserted.

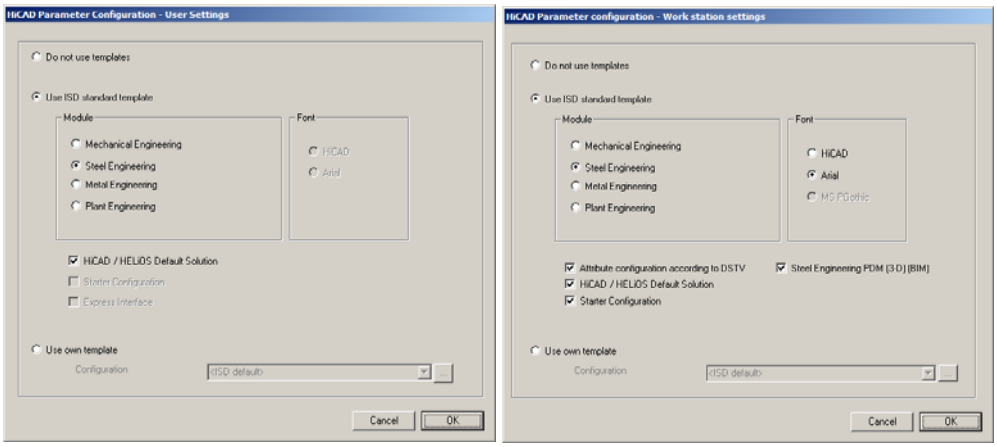
Glazing construction

All different types of glazing constructions, e.g. facades, winter gardens, translucent roofs, windows, doors etc.

5 Pre-Settings: General Notes

5.1.1 HiCAD – Parameter Configuration for Glazing Constructions

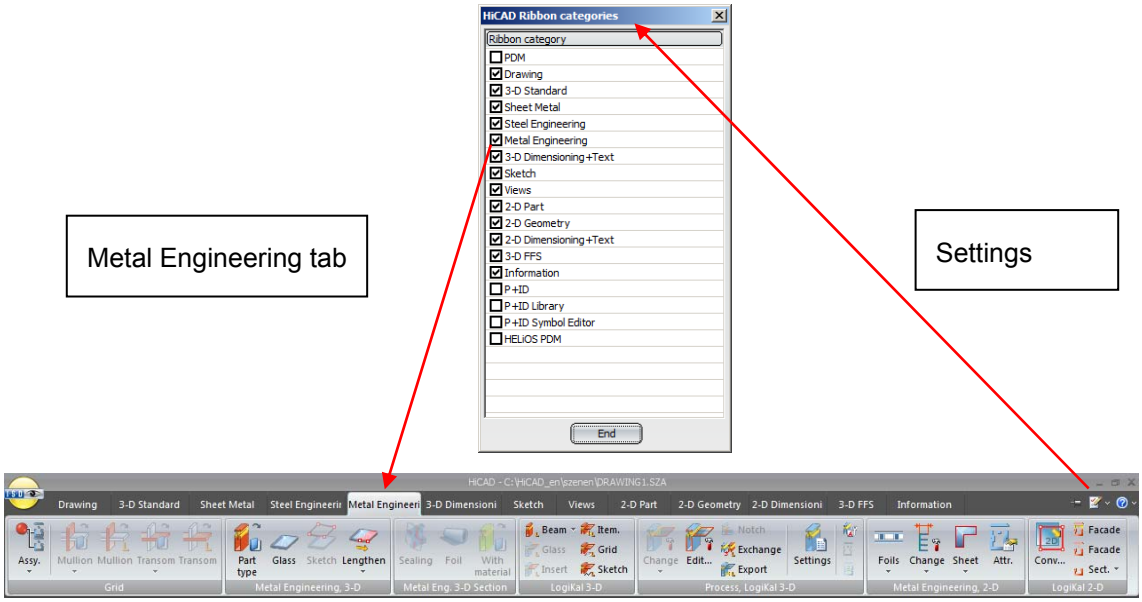
For this training the **Metal Engineering** option needs to be activated in the **HiCAD Parameter configuration** for the user and the workstation.



The parameter configuration can be changed subsequently with the programs <hicad installation directory>\exe\ ParKonfigComp.exe (Workstation configuration) and <hicad installation directory>\exe\ ParKonfigUser.exe (User configuration).

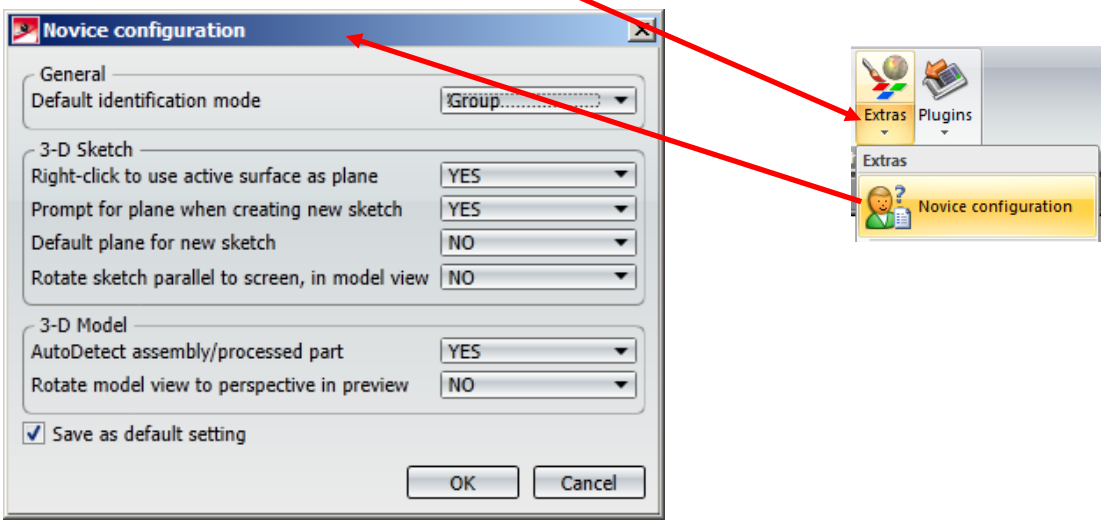
5.1.2 The “Metal Engineering” Tab

To access the functions for 2-D and 3-D Metal Engineering as well as for Facade Engineering, you need to open the **Metal Engineering** tab on the HiCAD Ribbon. To be able to do this, select **Settings > Ribbon categories** and activate the **Metal Engineering** checkbox.



5.1.3 Novice Configuration

The so-called **Novice configuration** makes it easier for you to learn how to create and process beam/profile groups. New users will be safely guided through the options. To activate the **Novice configuration**, open the **Drawing** tab and select **Others → Extras → Novice configuration**.

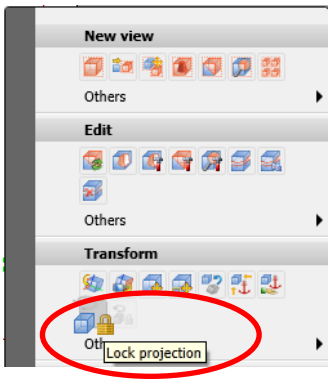


5.1.4 Lock Views in Sheet Area

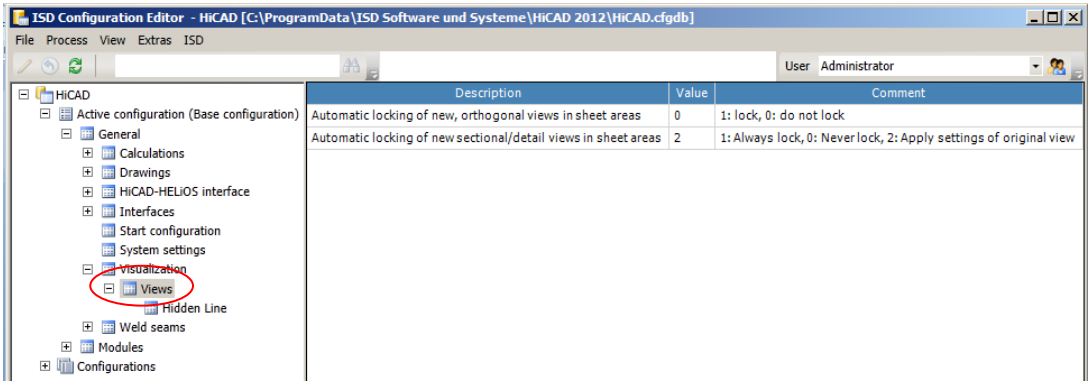
The locking of views prevents an unintentional rotating of views during drawing creation. This option allows you to work in the same way as in 2-D while still using the benefits of 3-D design.

Lock / unlock views manually:

- Right-click view frame (pink dotted frame around drawing object) and select “Transform” > “Lock projection” (“Unlock”).



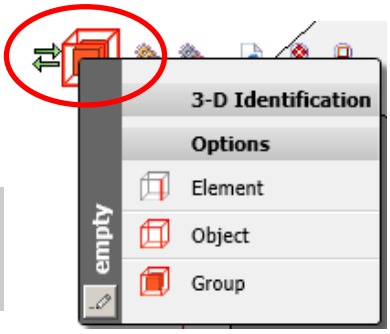
Global setting for view configuration <hicad installation directory>\exe\ ISDConfigEditor.exe.



5.1.5 “Group Snap” Mode

If this snap mode has been activated, Steel Engineering functions and beam/profile functions will not just refer to the selected object, but to the entire corresponding beam/profile group (if any). This facilitates the processing of complete beam/profile groups (activate, transform, clone etc.) significantly.

You activate the “Group snap” mode via the transparent toolbar, by right-clicking the 2nd icon from the left and selecting “Group”.



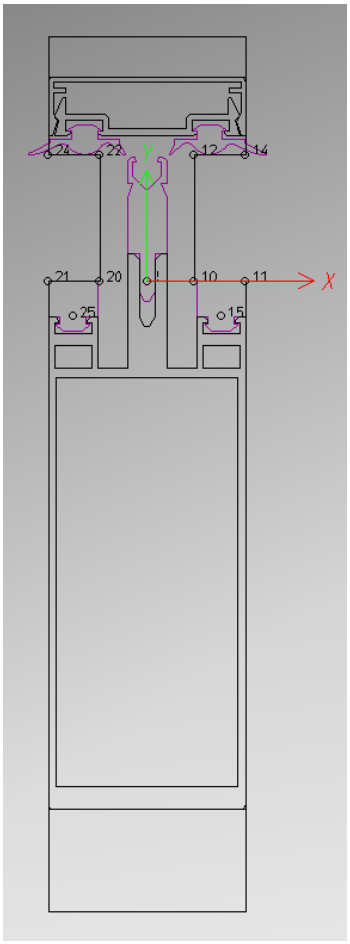
If you want to make the “Group snap” the default setting, open the **Drawing** tab and select **Others > Extras > Novice configuration**. In the dialogue, select “Default identification mode: Group”.

6 Glazing Construction with User-Defined Beam/Profile Cross-Sections

6.1 Create User-Defined Cross-Sections

It is possible for users to create their own beam/profile groups and apply the available Metal Engineering functions to them as usual.

In this exercise an imported mullion from the SCHÜCO series FW50+ is transferred and processed.



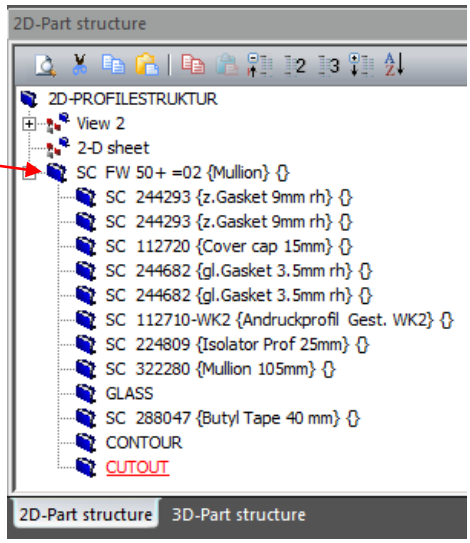
6.1.1 Create 2-D Part Structures

The definition of a user-defined beam/profile group takes place via the creation of a 2-D part with the part structure as shown in the screenshot below.
Options for part creation can be found on the **2-D Part** tab of the HiCAD Ribbon.

Create structure of user-defined beam/profile

Main beam/profile (e.g. SC FW 50+ mullion)

- **Profile 1**
- **Profile 2**
- **Profile 3**
- **Profile …**
- **Glass** … Glass insertion points
- **Contour** … Simplified contour of the entire beam/profile
- **Cutout** … Notching contour for beams/profiles to be attached



- EXACT geometries will be created for each single cross-section in the 2-D part **EXACT**.
- SIMPLE geometries will be created for each single cross-section in the 2-D part **SIMPLE**.
- CONTOUR geometries will be created for the complete cross-section in the 2-D part **CONTOUR**.

It is recommended to use only the exact geometry of the single cross-section, and one contour for the complete cross-section. Firstly, the creation will require less time, and secondly, it will be possible to switch the representation types depending on the respective view.

For “View all” views, the contour representation will often be sufficient (which also allows a higher quality of representation), whereas in sectional views an exact representation would be required.

Copy the Part name to the Article number field

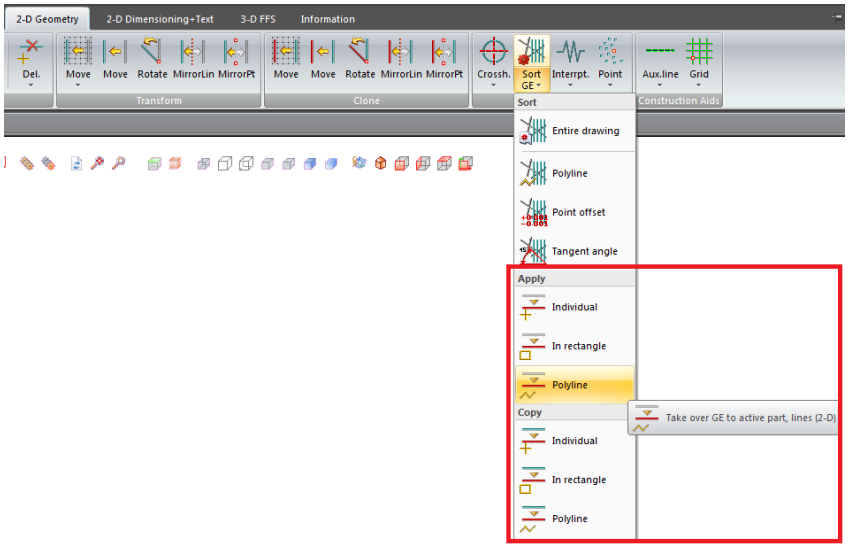
- It is easiest to copy it in the Part attributes window.
- For the sake of clearness, the Part type can also be specified. It is displayed in the ICN and makes an identification of the parts easier.

6.1.2 Import Cross-Section Geometry

Insert the file ***SC FW 50+ =02.FIG*** from the directory HiCAD\Szenen in an empty drawing file.

Take the contours of the 2-D figure over into prepared part structure, as described in 6.1.1.

To do this, open the **2-D Geometry** tab and select **Tools > Sort GE > Apply/Copy > Take over/Copy GE to active part**

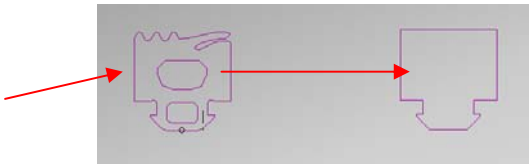


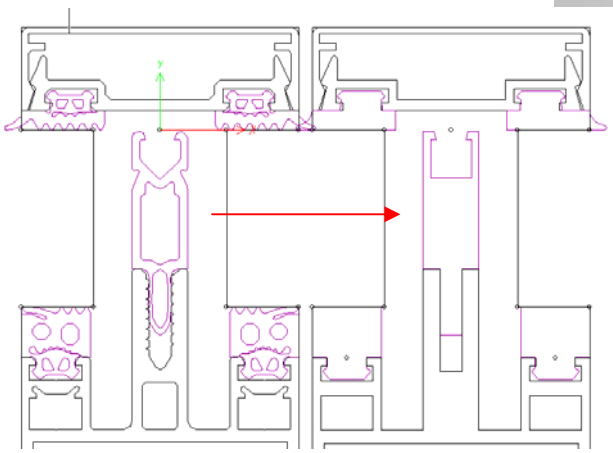
6.1.3 Clean Up Cross-Sections

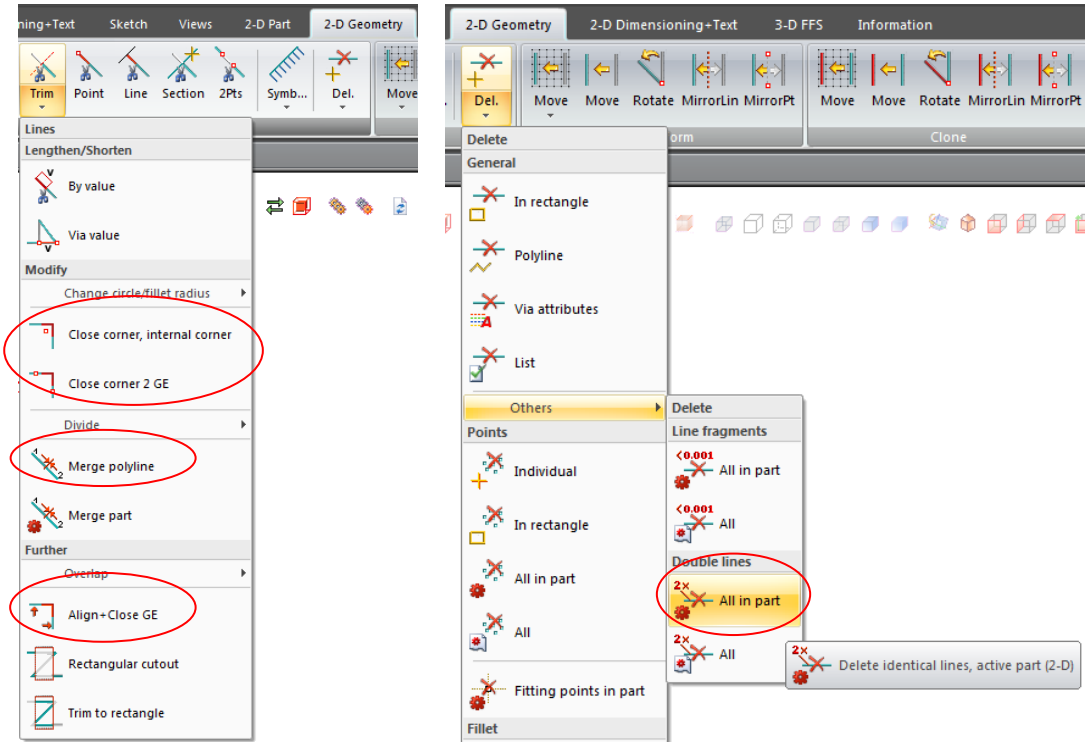
Check the geometries for errors and clean them up using the 2-D standard functions if required.

You should consider the following:

- Avoid collisions of 2-D parts (otherwise the colliding edges will be hidden in 3-D Hidden Line representation!)
- Delete double lines
- Delete overlaps
- Close corners
- Close polylines
- Simplify contours where possible







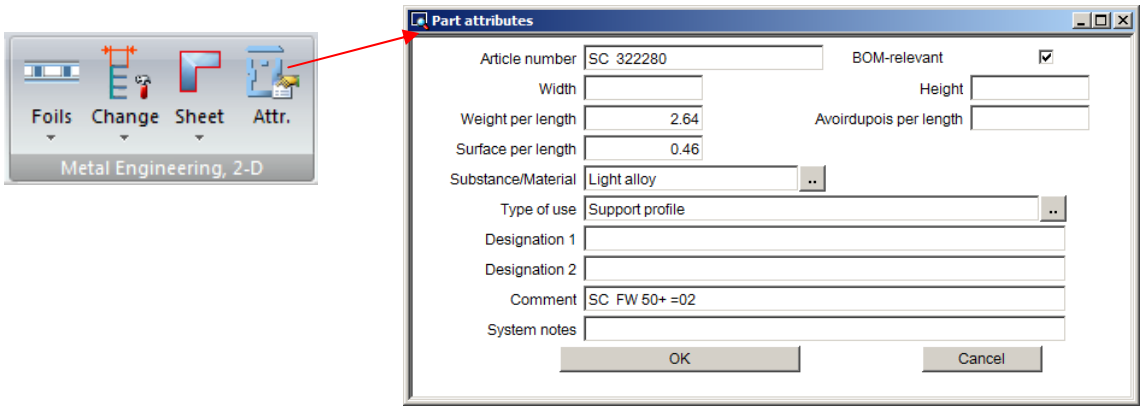
Many geometries of older drawings contain small, approximated circular arcs. If possible, these should be replaced by circular arcs running out tangentially, in order to obtain a higher quality of representation of the views.

6.1.4 Part Attributes / BOM Relevance

Assigning of attributes, Option 1

To set the required part attributes, open the **Metal Engineering** tab and select **Metal Engineering, 2-D → Define cross-section properties**.

The relevant attributes will be queried and assigned to the 2-D part.



Now use this function to assign the properties for the single profiles and the “superordinate” part.

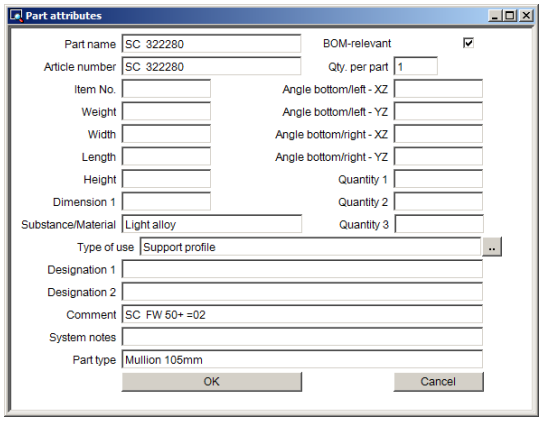
The corresponding materials can be found in the **Civil Engineering** materials folder.

- Activate the BOM-relevant checkbox if desired.

Assigning of attributes, Option 2

To set the required part attributes, open the **Steel Engineering** tab and select **Further functions → Settings → Series cross-sections, Assign attributes**.

The relevant attributes will be queried and assigned to the 2-D part.



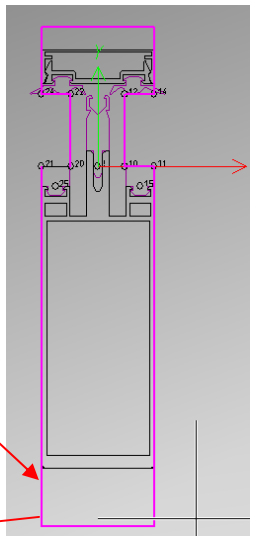
Now use this function to assign the properties for the single profiles and the “superordinate” part.

The corresponding materials can be found in the **Civil Engineering** materials folder.

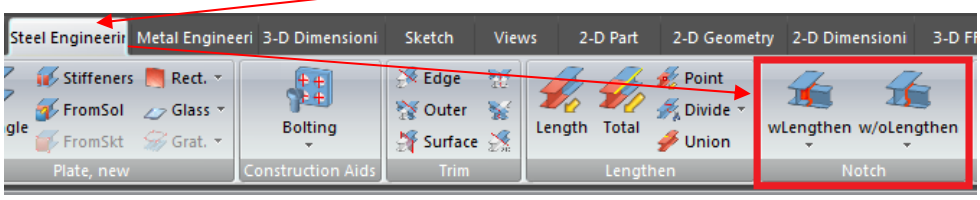
- Activate the BOM-relevant checkbox if desired.

6.1.5 Notching Contour for Attached Beams/Profiles

The geometry for the notching of attached beam/profile groups is defined as a closed polyline in the part **CUTOUT**.



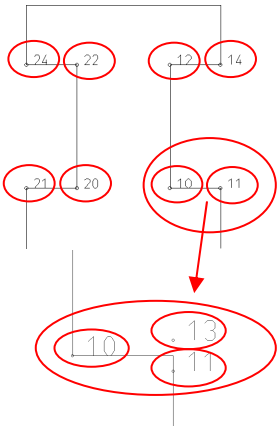
In the 3-D beam/profile group, the CUTOUT figure is invisibly included. To create a notch, use the same-named Steel Engineering functions.



6.1.6 Glass Insertion Points

The definition of glass insertion points takes place via the creation of appropriately numbered isolated points in the part **GLASS**.

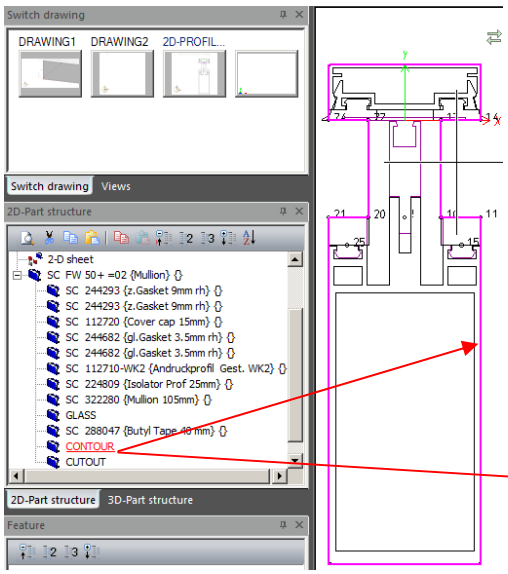
The point numbers are defined as follows:



- Points 10 and 20 define the glass inset
- Points 11 and 21 define the glass insertion direction
- Points 12 and 22 define the glass thickness
- Points 14 and 24 are required for the option “Change glass in insert”. These points are not mandatory.

- Optional is the definition of Points 13 and 23. Between Points 11 and Point 13 you can define a tolerance (as for Points 21 and 23). The definition of these points is not mandatory.

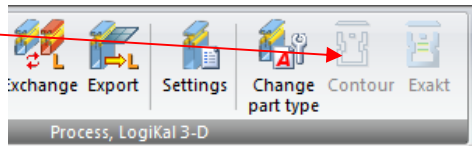
6.1.7 Contour Geometry for Contour Representation of Complete Beam/Profile Groups



In the 2-D cross-section of a beam/profile group, create the part CONTOUR, and draw the outer contour for the complete beam/profile group in the part. Also draw the glass insertion depth here.

In this way, the created 3-D beam/profile groups can be displayed as contours by means of the function **All Metal Engineering assemblies → Contour representation**.

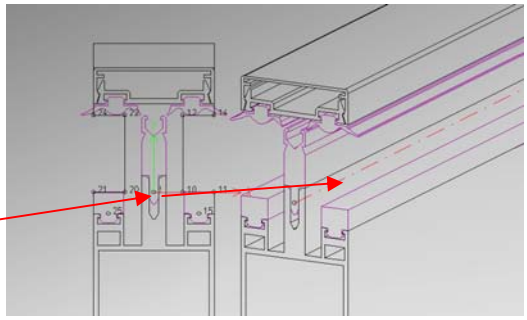
This does not only increase graphical performance, but also leads to optimised representation in overviews and easier element identification in 3-D drawings.



6.1.8 Define Insertion Axis

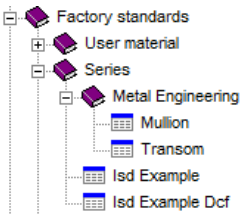
On the desired point, create an isolated point in the superordinate beam/profile. Via this point, the complete beam/profile can be moved to the origin of the coordinate system (A 0 0).

When you save the drawing, the origin will define the insertion axis.
It is recommended to choose a point on the **inner** or **outer** glass plane.

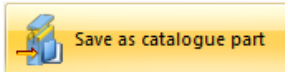


6.1.9 Save Beam/Profile Group to Catalogue

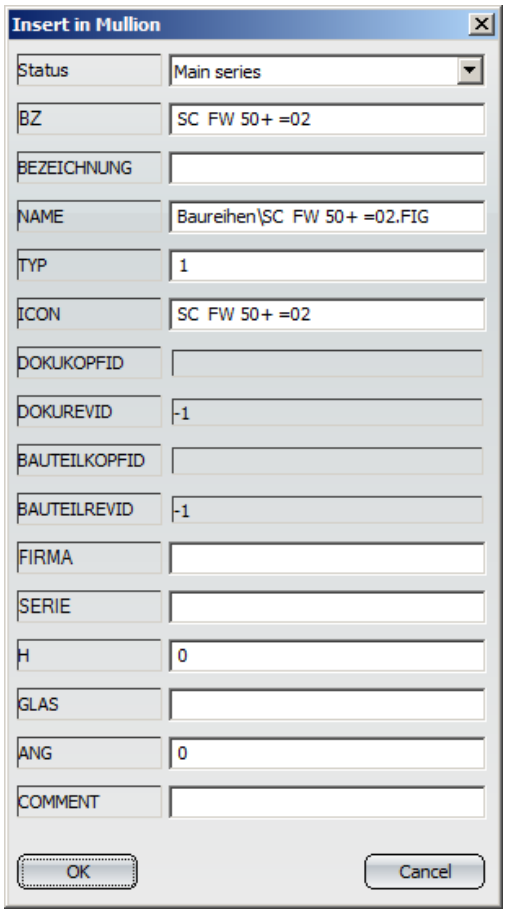
In the Catalogue Editor you can create your own tree structure under Factory standards\Series\Metal Engineering\ … .



To do this, open the **Steel Engineering** tab and select **Further functions → Settings → Series cross-sections → Save as catalogue part**



- Select the part (entire profile at isolated point)
- Specify the storage location of the 2-D file (e.g. Catalogues\Series\Metal Engineering\...)
- Choose the desired table of the catalogue
- Enter the corresponding attributes in the input fields



Availability

Description for selection when inserting the part in HiCAD

Value for Article number

Storage location of part

1= Fixed 2-D/3-D part (Default)

Name of icon file in Image sub-folder

Beam/profile manufacturer

Beam/profile series

Beam/profile height

Glass thickness

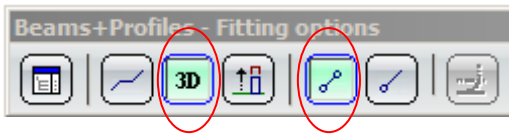
Inclination angle

Comment

6.2 Insert User-Defined Beam/Profile Cross-Sections

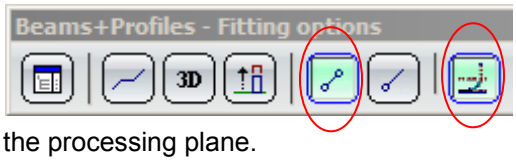
6.2.1 Free Insertion

The free insertion takes place in the same way as the insertion of Steel Engineering beams, with the known functions. The insertion points can be picked via already designed parts, or can be constructed with the help of point options **R, P**, etc.



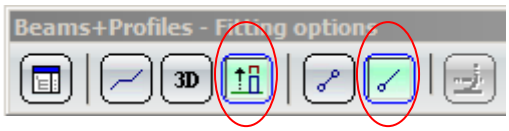
6.2.2 Insertion with Sketch

The insertion via sketch takes place in the same way as the insertion of Steel Engineering beams, with the known functions. Prior to insertion you need to define a processing plane. The beam/profile will be placed with its insertion axis into the processing plane.



6.2.3 Insertion Perpendicular to Processing Plane

The insertion perpendicular to processing plane takes place in the same way as the insertion of Steel Engineering beams, with the known functions. Prior to insertion you need to define a processing plane. The beam/profile will be placed perpendicular on the processing plane, via specification of one start point and a length.

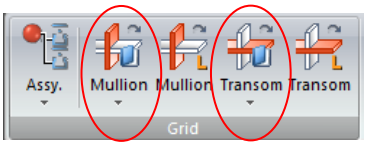


6.2.4 Insertion on Grid via User-Defined Beam/Profile Cross-Sections

In the following chapter, user-defined beam/profile cross-sections are to be placed on a Metal Engineering grid (use Metal Construction plugin). Prior to this, the steps described in Chapter ***8 Create Facade via Metal Engineering Grid (Metal Engineering PlugIn)*** need to be performed.

1. The creation of the grid is described in Chapter ***8 Create Facade via Metal Engineering Grid (Metal Engineering PlugIn)***. First, perform the steps in this chapter.

2. Now use the **Mullion** or **Transom from catalogue** function to place the beams/profiles on the sketches. You can also use “collection” functions for the selection of equally aligned axes.



- CTRL + Left-click = Collection/deselection of individual (sketch) axes
- SHIFT + Left-click = Automatic collection of equally aligned (sketch) axes

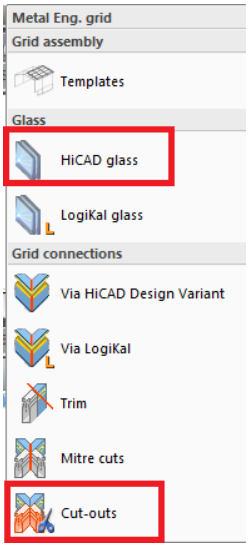
3. Right-click to open the HiCAD dialogue. In the Metal Engineering folder you can find prepared beams and profiles.

4. Use the **Cut-outs** option for the notching of the mullions/transoms: On the **Metal Engineering** tab, select **Grid → Assy. → Cut-outs**.

5. The automatic insertion of glass panes across the complete facade can be performed via **Metal Engineering → Grid → Assy. → HiCAD glass**.

6. Here it is important that you define a suitable glass pane, or select a glass pane with the previously specified glass thickness (see Chapter *6.1.6 Glass Insertion Points)*

7. Create several views in the Sheet area and familiarize yourself with the representation options.



7 Creating Glazing Constructions with LogiKal

7.1 Pre-Settings for Beam/Profile Groups via LogiKal

Before you can use the interface with LogiKal, you need to define some pre-settings. For example, you need to specify where HiCAD can find the LogiKal installation, from where the beam/profile cross-sections will be loaded etc. ...

7.1.1 Interface Settings (Registry Entries 32/64 BIT)

To ensure a proper functioning of the interface between HiCAD and LogiKal you need to make the following settings in the Registry:

- Click the Windows **Start** button, enter **regedit** in the input field at the bottom and confirm with ENTER.

- Create an **"Orgadata"** key under **HKEY_LOCAL_MACHINE/SOFTWARE**, with the string

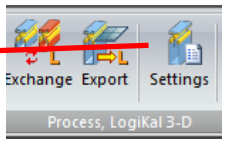
  LogiDll REG_SZ C:\LOGIKAL\LogiDll64.dll

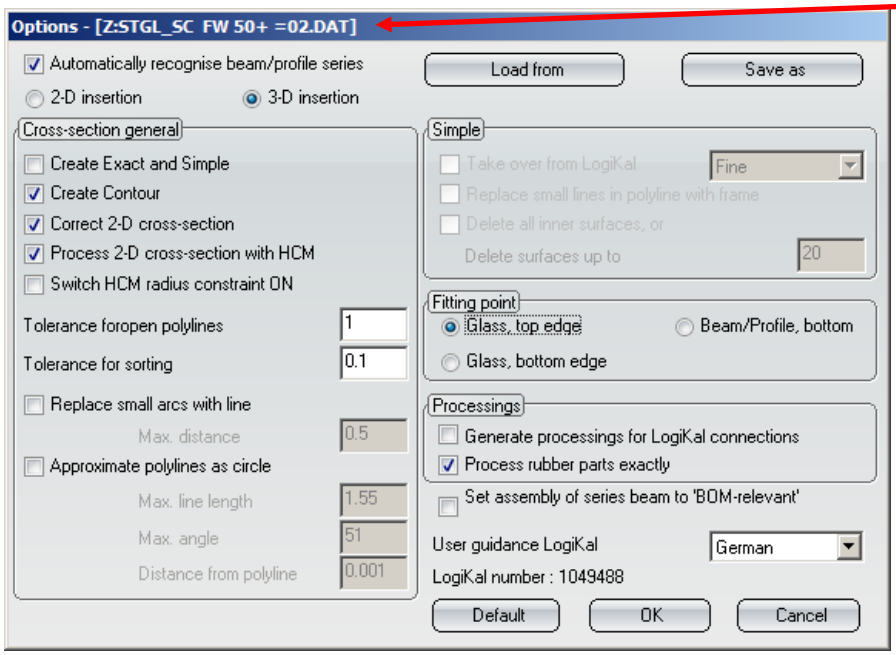
- Also create an **"Orgadata"** key under **HKEY_LOCAL_MACHINE/SOFTWARE/WOW6432Node**, with the string

  LogiDll REG_SZ C:\LOGIKAL\LogiDll.dll

7.1.2 Metal Engineering Settings

The Metal Engineering settings are used for the optimisation of LogiKal beams and profiles. To access the setting options, select **Metal Engineering → Process, LogiKal 3D → Metal Engineering Settings**.

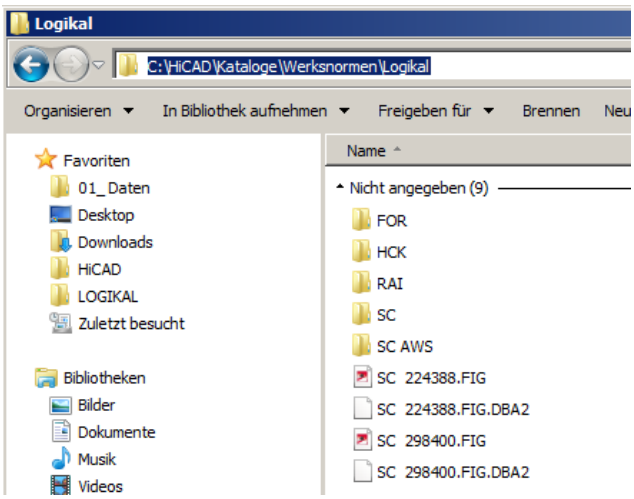




- **Load from...** Use this button to load the beam/profile series to be inserted from the HiCAD/sys sub-directory of the HiCAD installation, and to explicitly specify insertion functionalities.
- **Automatically recognise beam/profile series...** When you insert a beam/profile series via LogiKal, this will be recognised in HiCAD, and the insertion will be optimised appropriately. The insertion options for the different beam/profile series are managed via the buttons in the upper window area.
- **Cross-section, general…** As HiCAD uses the beam and profile library of LogiKal (Logidll.dll), and these 2-D cross-sections do not always have a closed contour, these 2-D cross-sections can be auto-reworked with the “Cross-section, general” option.
- **Simple…** Use this option to additionally auto-create a SIMPLE figure. You can switch between the SIMPLE and the EXACT figure using the Steel Engineering functions. (see also Chapter **7 Representation Options**)
- **Fitting point…** Here you define the fitting point of the beam/profile group. Normally, this should be “Glass edge, top” or ”Glass edge bottom”, to ensure that the correct glass plane will be used when the mullions and transoms are inserted.
- **Processings…** Here you define whether processings should be generated on mullions and transoms for LogiKal notchings.
- **Set assembly of series beam to BOM-relevant...** The single beams and profiles, such as mullions/transoms, cover strips, pressure plate profiles, etc... are always BOM-relevant. Use this option to set also the superordinate assembly to BOM-relevant.
- **User guidance LogiKal…** Here you can select the language for the LogiKal window opened in HiCAD.

7.1.3 Simplified Beam/Profile Cross-Sections in the HiCAD Catalogue

The insertion of LogiKal beams and profiles normally takes place via the Logidll.dll and uses the exact beam/profile cross-sections by LogiKal. As these cross-section may contain gaps or double lines, you have the option to save simplified/corrected beam/profile cross-sections in your HiCAD installation, at **...\Catalogues\Factory Standards\LogiKal**.



When you insert LogiKal beams and profiles, HiCAD will check whether the corresponding figures are available in the **...\Catalogues\Factory standards\LogiKal** directory and will use them. If the figures do not exist, the exact figures by LogiKal will be used.

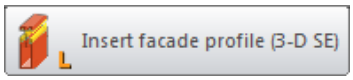
To ensure that always the newest beam and profile cross-sections will be used, they will obtain an appropriate timestamp.

.

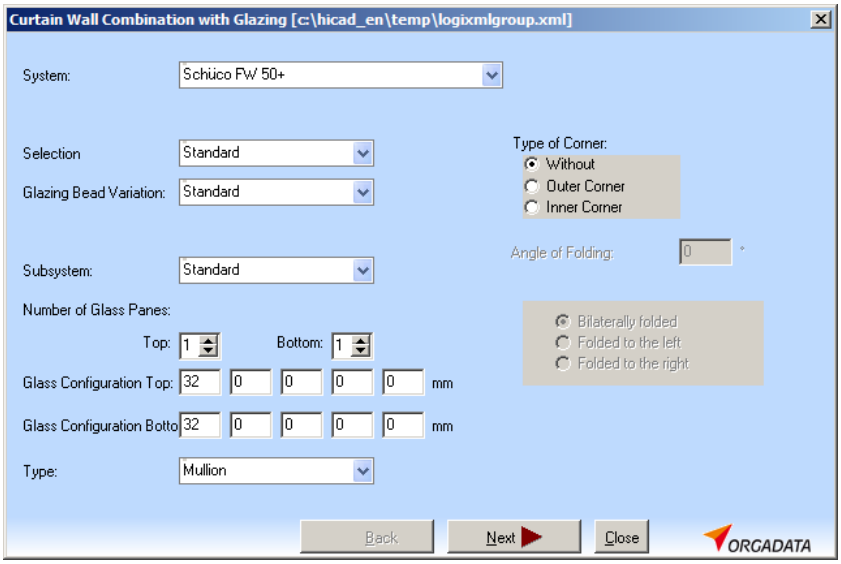
7.2 2-D/3-D Interface with LogiKal

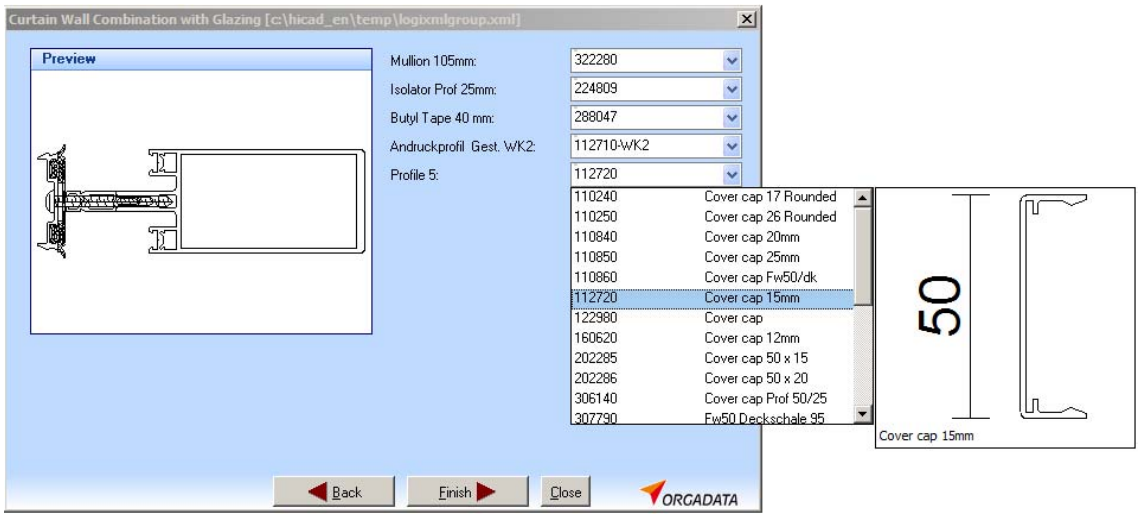
7.2.1 2-D/3-D Import of Individual Beam/Profile Cross-Sections

The **Insert facade profile** function in the **LogiKal 3-D** function group enables you to use and process the beam/profile systems of all facade system manufacturers available in LogiKal.



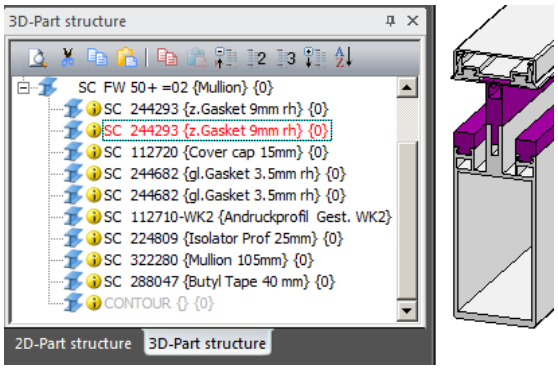
- On the **Metal Engineering** tab, select **Logikal 3-D → Insert facade profile**.
- In the LogiKal dialogue by ORGADATA you can then configure the desired beam/profile cross-section.





- After clicking **Finish** the beam/profile group can be inserted in your drawing with the standard Steel Engineering functions.

After inserting the beam/profile group in the drawing, a main part (which is not BOM-relevant, see 7.1.2 Metal Engineering Settings) with sub-parts such as Sealing, Pressure plate profile, Mullion etc. ... will be created in the 3-D Part structure of the ICN. These sub-parts are BOM-relevant and contain all BOM-relevant information.

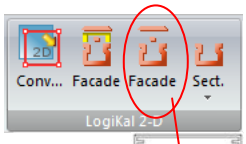


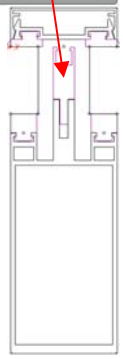
Besides the part structure, beam/profile manufacturers, beam/profile systems, designation, article number and part type will be generated into the part attributes and can be displayed in the ICN.

The glass insertion points and the cut-out figure are invisibly included in the part structure. They will be automatically evaluated during glass insertion/notch creation.

2-D Insertion

The option **Insert facade cross-section** in the **LogiKal 2-D** function group enables you to use and process beam/profile systems in 2-D, from all facade system manufacturers that are available in LogiKal.

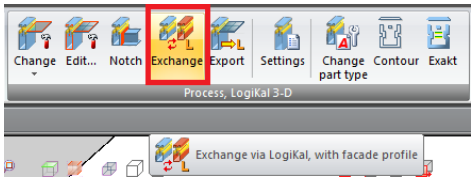




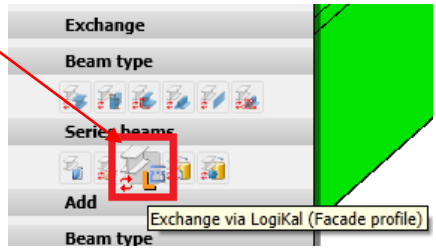
- On the **Metal Engineering** tab, select **LogiKal 2-D → Insert facade cross-section**.
- The beam/profile cross-sections are configured in the same way as the 3-D beams and profiles.

7.2.2 Exchange Beams/Profiles via LogiKal

If beams and profiles are modified during the construction phase by an architect, project supervisor or structural engineer, one or several beams/profiles can be exchanged via LogiKal.



Right-clicking the items in the **Exchange via LogiKal** selection list allows a multiple selection of beams and profiles.



This option is only available for 3-D beam/profile groups.

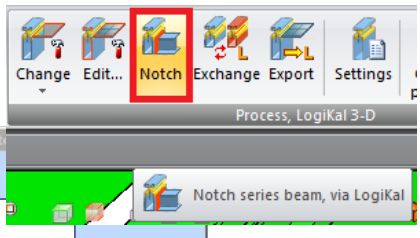
7.2.3 Mullion/Transom Connection via LogiKal 3-D Interface

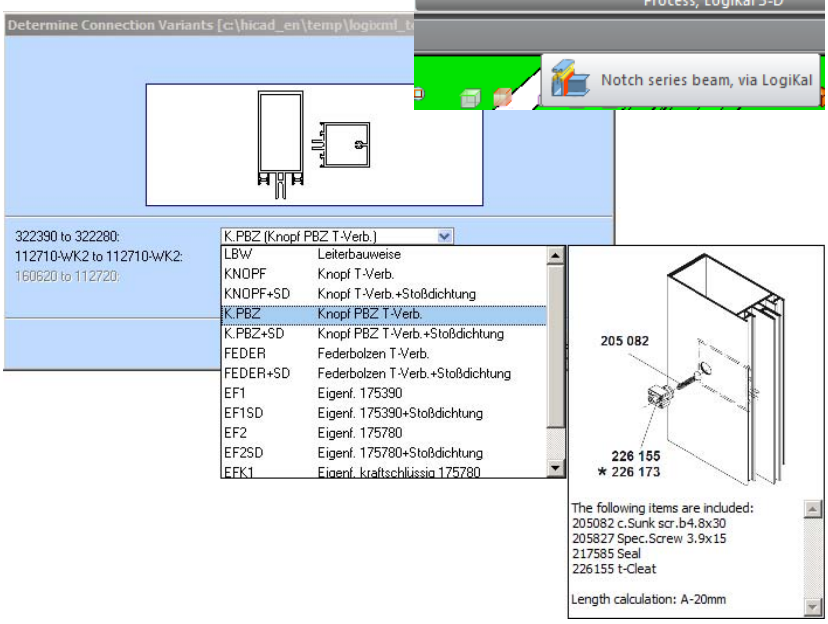
You can apply mechanical processings in HiCAD to beam/profile combinations selected via LogiKal in order to create mullion/transom connections. The accessory parts will be created as dummy parts with BOM data.

Caution:

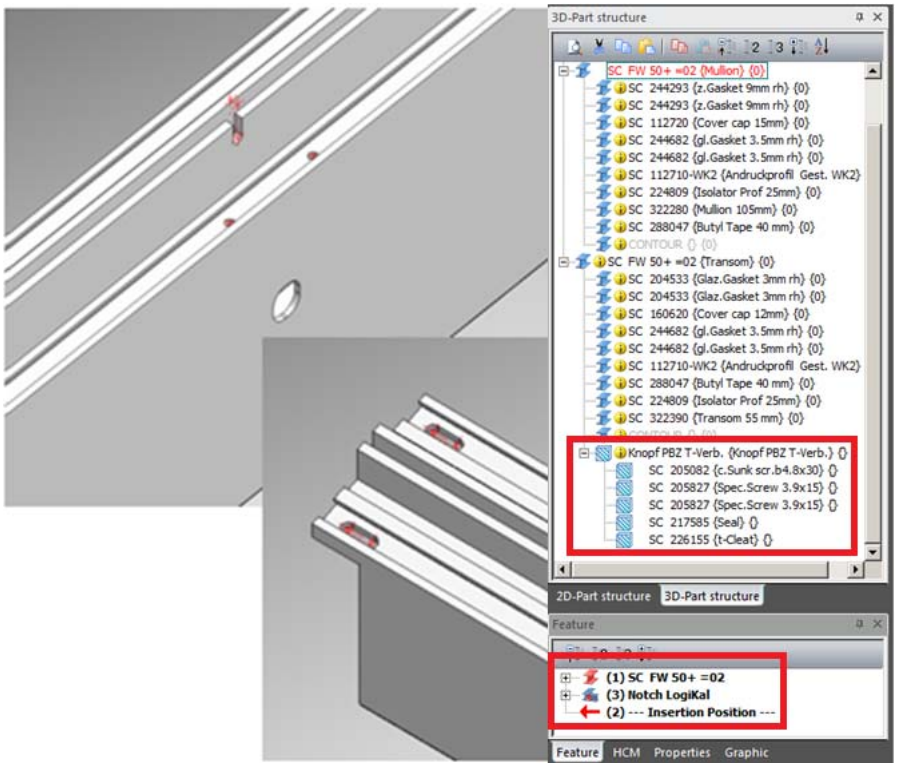
When the connections are created, it will not be verified whether they actually match the current specifications of the manufacturer! Therefore, the created connections must always be checked by the user and, if required, adjusted manually according to the beam/profile manufacturer's specifications.

- To create mullion/transom connections via LogiKal, select **Metal Engineering → Process, LogiKal 3-D → Notch series beam, via LogiKal**.
- First select the transom, and then select the mullion.
- After this, you can choose the desired connection situation via the LogiKal selection dialogue.

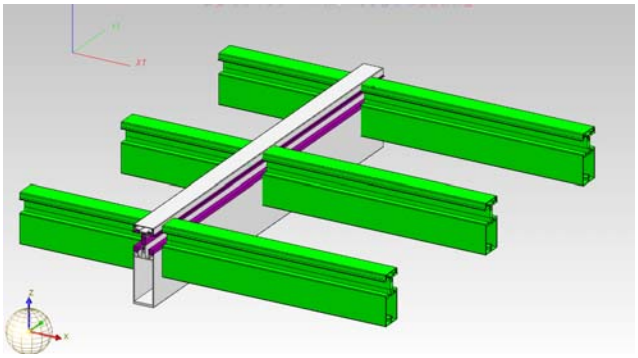




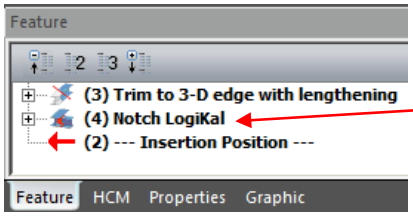
- After selecting the connection elements, the connection will be automatically created, complete with all processings and accessories.



- For mullion/transom notchings, a multiple selection of transoms is possible. Please note that the “Group snap” mode needs to be activated for this. Press and hold down the CTRL key and left-click to select, one after the other, all transoms that should be attached to a particular mullion. A selection list is created in the ICN, and all selected beams are highlighted in green in the drawing area. Then, choose the **Notch via LogiKal function**, and finally, select the mullion with a mouse-click. The connections will then be created automatically.



To **change** the position of the transom on the mullion, you need to delete the feature log entries for the notchings on mullion and transom.

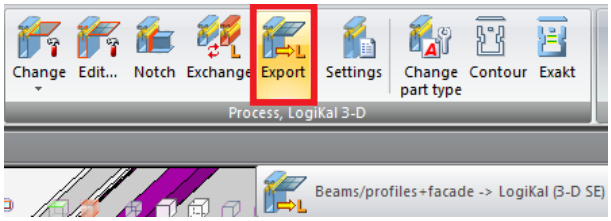


Right-click feature log entry and select Delete

7.2.4 Transfer of Beams/Profiles to LogiKal

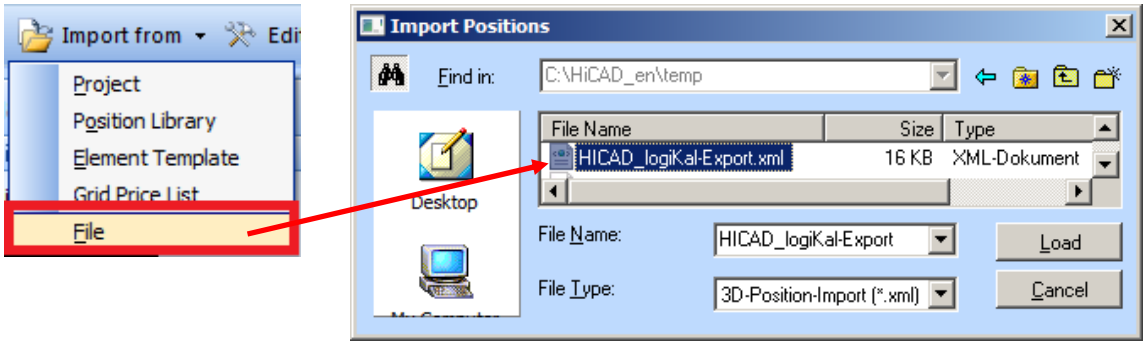
If you insert individual beams and profiles via LogiKal in HiCAD and process them, you have the option to export the beams/profiles including all processings and accessories to LogiKal.

- On the **Metal Engineering** tab, select **Process, LogiKal 3-D → Beams/profiles+facade -> LogiKal**



- The beams/profiles can be collected with left mouse clicks in the drawing area or in the ICN. You end the selection by pressing the middle mouse button. An export file (**HiCAD_logiKal-Export.xml**) will be created in the **...\temp** directory of the HiCAD installation.

- This **HiCAD_logiKal-Export.xml** file can be imported to LogiKal. For this you need to start LogiKal and activate a project. The “Import from file” option will then be available. Go to the HiCAD\temp directory and select the file **HiCAD_logiKal-Export.xml**.

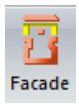


To be able to use this option, the 3-D Interface of the company ORGADATA needs to be activated.

If any changes are applied to the HiCAD drawing, the beams/profiles must be re-exported to LogiKal.

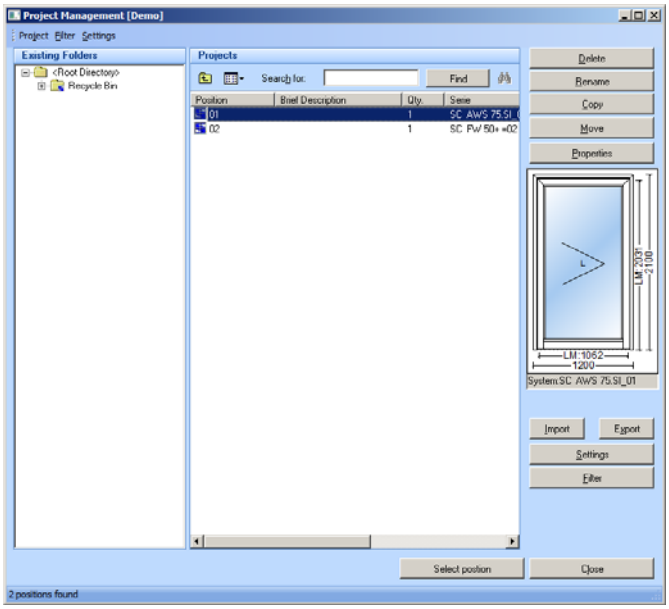
7.2.5 2-D/3-D Import of Existing LogiKal Items

If window or facade elements have already been created in LogiKal, these can be imported to HiCAD in 2-D or 3-D.

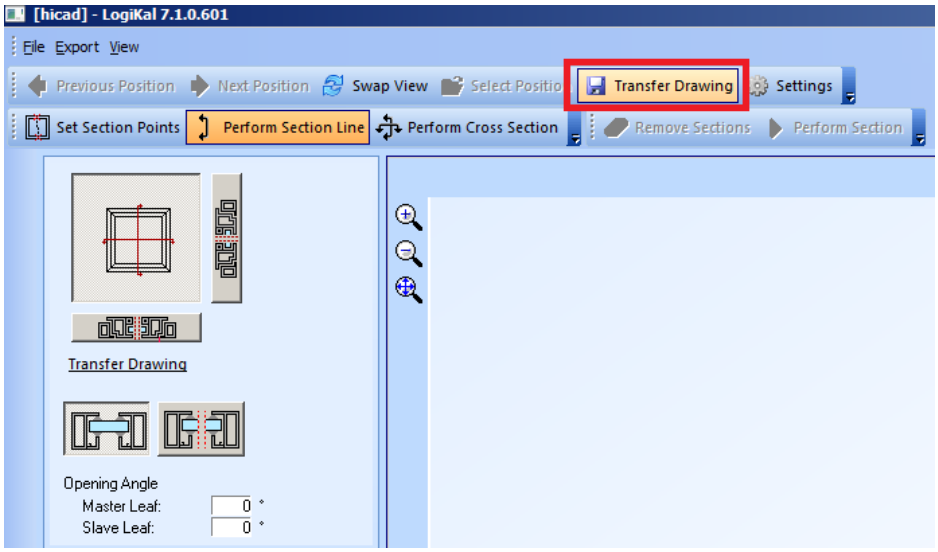


- Import of a 2-D facade element

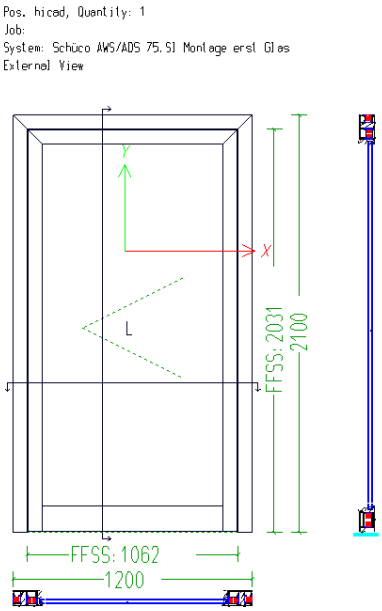
Select **Metal Engineering → LogiKal 2-D → Insert facade + inserts from existing LogiKal project (2-D)**. The project management dialogue of LogiKal opens, enabling you to select the desired item.



When the desired item has been selected, several view options can be chosen. You can decide whether you want to import the complete view with section details, or only the view, or only the sections, a shortened or unshortened view etc. After configuring the views select **Transfer drawing** to get back to HiCAD.

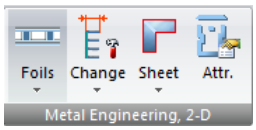


When importing the 2-D views you will be prompted to specify a fitting point on the part. When making the insertion in the HiCAD drawing you can directly specify the correct position. Furthermore, you will be asked to specify the rotation angle and the scale of the 2-D view.



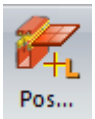
Changes can be made at any time using the HiCAD 2-D functions. However, such changes will have **no** effect on the corresponding item in LogiKal. To keep changes in HiCAD and LogiKal up to date, the item needs to be changed in LogiKal and then re-imported to HiCAD.

For the construction of drawings for approval containing wall connections, the Metal Engineering, 2-D functions and the general 2-D Geometry functions are available.



3D

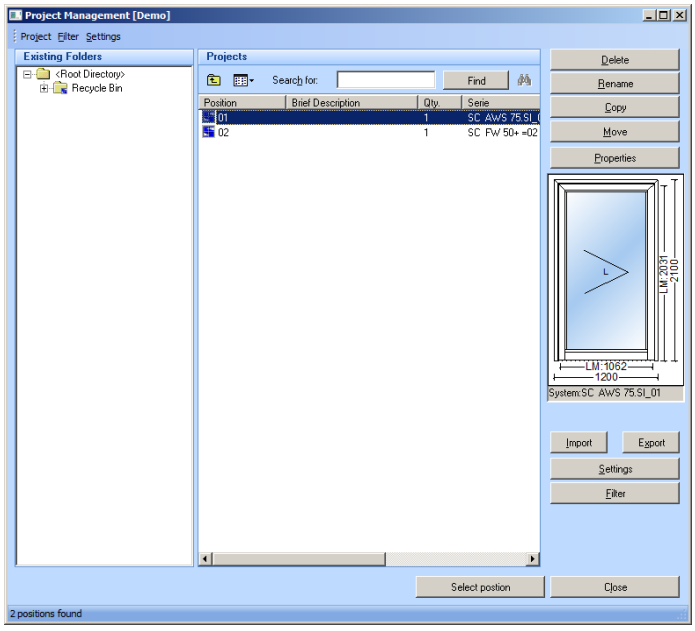
- Import of a 3-D window/facade element



Before importing the element, you need to define a processing plane, into which the window/facade element will then be imported. This will prevent a subsequent rotating or moving of the part. The import is performed according to the active coordinate system. If no processing plane is active, the element will be imported according to the World Coordinate System.

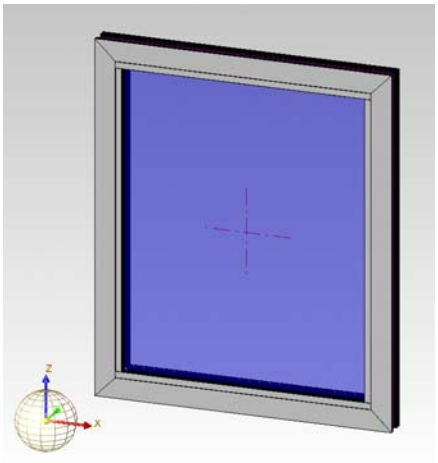
LogiKal elements are aligned as follows: The X-axis always refers to the width of the elements, the Y-axis to their height and the Z-axis to their depth!

To access the function, select **Metal Engineering → LogiKal 3-D → Insert facade + inserts from existing LogiKal project (3-D SE)**. The project management dialogue of LogiKal opens, enabling you to select the desired item.



When importing the 3-D window/facade element, you will be prompted to directly specify a fitting point on the part. Left-click twice to directly specify the correct position when making the insertion in the HiCAD drawing.

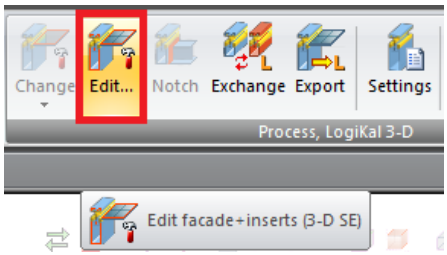
Alternatively, you can right-click twice to place the element in the origin of the processing plane.



Changes can be performed at any time using the HiCAD **3-D Standard** and **Steel Engineering** functions. However, such changes will have **no** effect on the corresponding item in LogiKal. To keep changes in HiCAD and LogiKal up to date, the item needs to be changed in LogiKal and then re-imported to HiCAD.

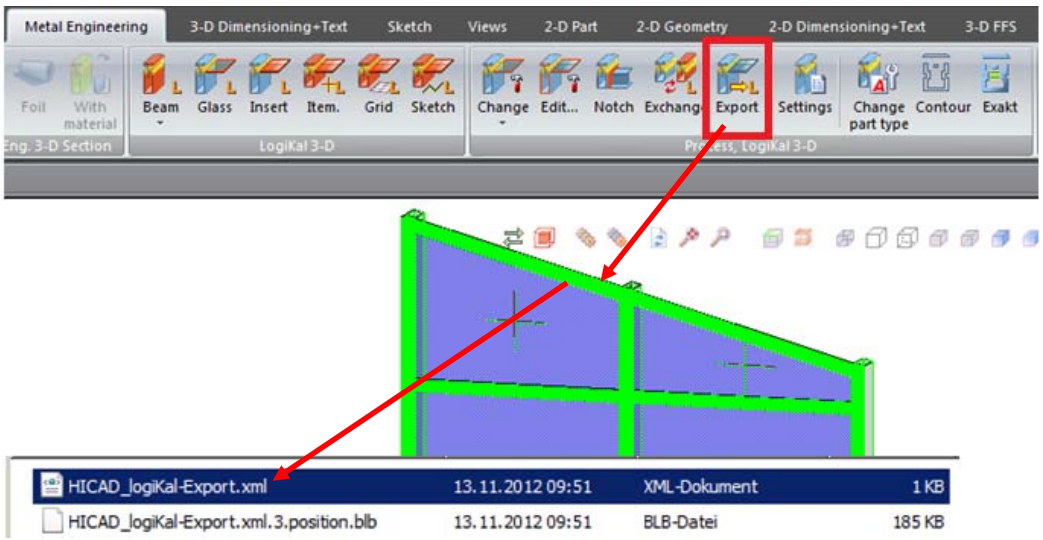
Changes, Option 2

You can use the **Edit facade + inserts (3-D SE)** function to change the element (dimensions, glass etc.) via LogiKal. It will be updated in HiCAD immediately.

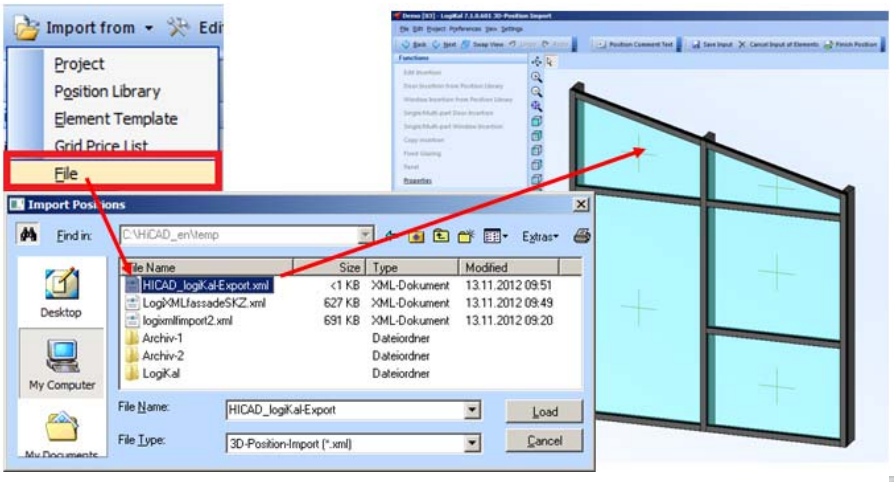


Caution: When using this option, the changed facade/window element will only be available in HiCAD, but not in LogiKal. At this point, there are two different items. In such cases the item needs to be deleted in LogiKal, and then the HiCAD item needs to be imported to LogiKal.

- The facade/window element can be exported to LogiKal with the **Beams/profiles+facade -> LogiKal** function. For this to happen you need to select a *part of the facade/window element*, **or** to the *complete assembly* selected in the 3-D Part Structure (confirm by pressing the middle mouse button). After this, a **HiCAD_logiKal-Export.xml** file will be created in the **HiCAD\temp** directory.



- To perform the import, start LogiKal first. Use the LogiKal function **Import from: File** to import the **HiCAD_logiKal-Export.xml** file from the **HiCAD\temp** directory. The xml file contains all information that was previously entered during the creation of the item via LogiKal.



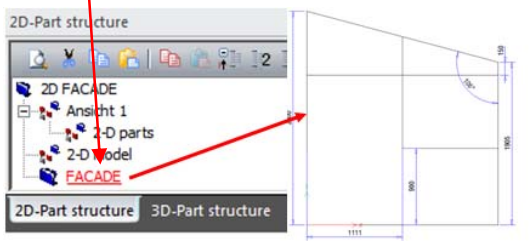
For the construction of drawings for approval containing wall connections, all HiCAD functions are available. In addition, you can find further, optimised tools for the processing of sections in the **Metal Eng. 3-D Section** function group of the **Metal Engineering** tab.

7.2.6 2-D/3-D Facade/Insert with Sketch

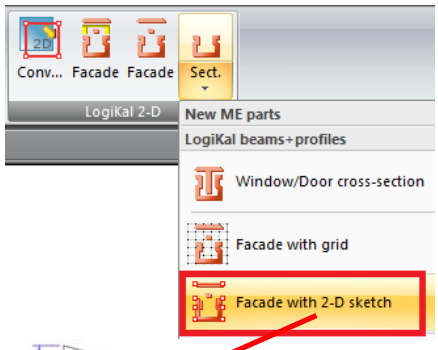
2D

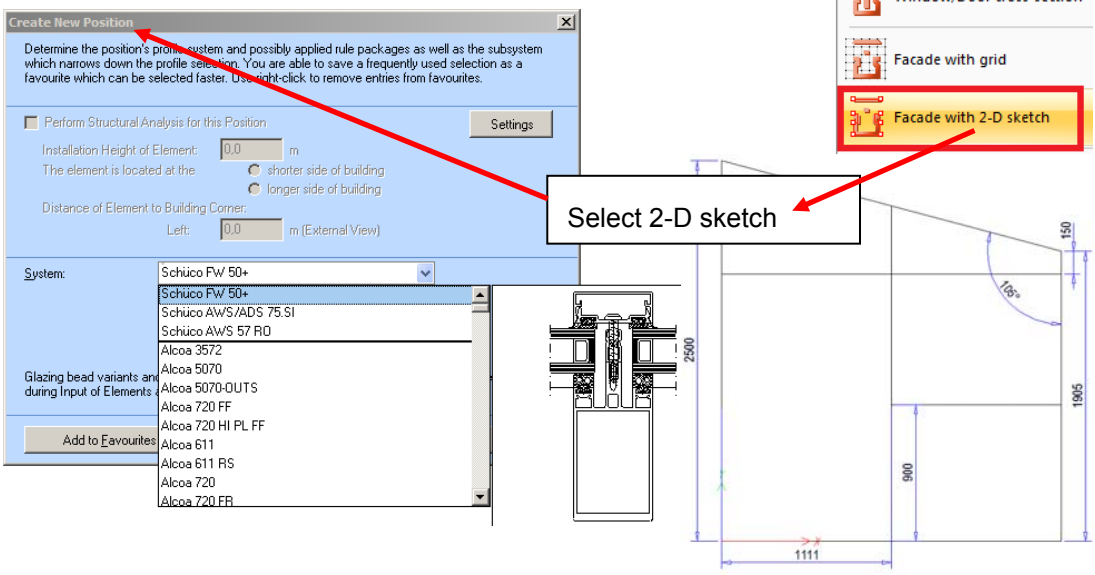
Facade with 2-D sketch

- Option **2-D Facade/Insert with sketch**
- Prior to this, a **New 2-D part** must have been created in the 2-D Part structure. The part needs to be active.
- After this, use the 2-D sketching tools in the **2-D Geometry** tab to draw the outer contour and the divisions (axes) of the facade/the insert.

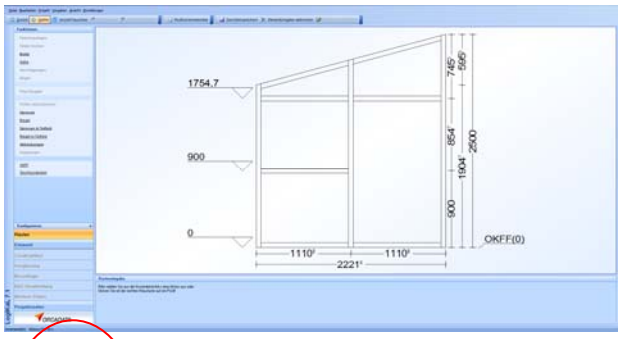


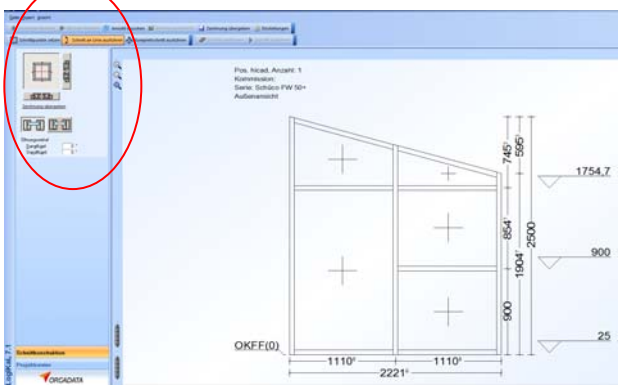
- On the **Metal Engineering** tab, choose **LogiKal 2-D → Facade with 2-D sketch** and select the sketch. A LogiKal dialogue opens, enabling you to select a beam/profile system.





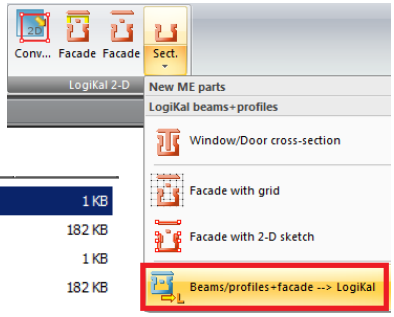
- After this, LogiKal is launched automatically and the beams/profiles will be placed on the sketch in LogiKal.
- The facade/window element can then be configured with the usual LogiKal functions.
- Before performing the import to HiCAD, LogiKal asks which views / sectional views should be imported.
- Select the LogiKal function **Transfer drawing**, which closes LogiKal and imports the 2-D element to HiCAD. You will be asked to specify a fitting point (left-click twice).



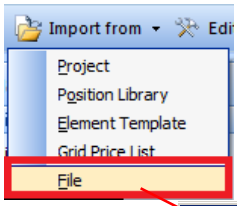


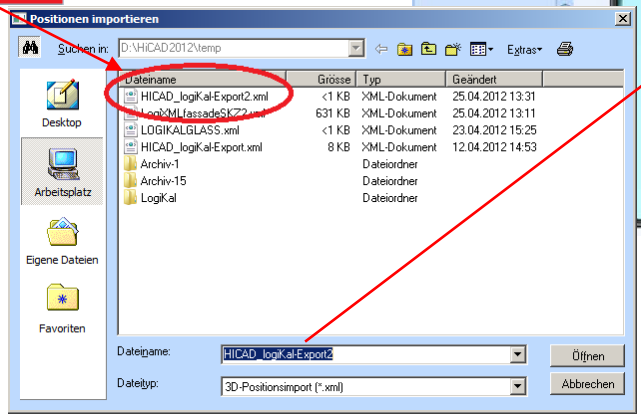
When you choose this procedure, the facade/window element will only exist in HiCAD, but not in LogiKal.

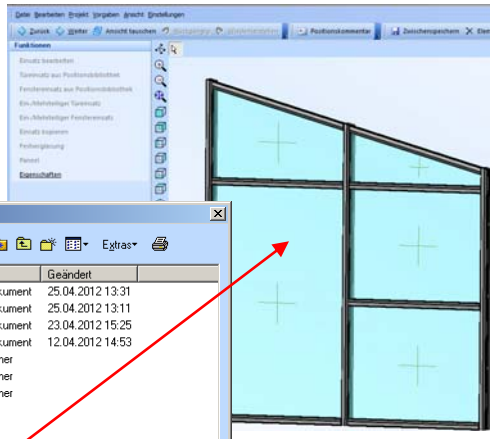
- Use the **Beams/profiles + facade -> LogiKal** function to export the facade/window element to LogiKal. When you left-click the 2-D facade/window element after selecting the function, an export file will be created in the **HiCAD\temp** directory.



- To perform the import, start LogiKal first. Use the LogiKal function **Import from: File** to import the **HiCAD_logiKal-Export2.xml** file from the **HiCAD\temp** directory. The xml file contains all information that was previously entered during the creation of the item via LogiKal.





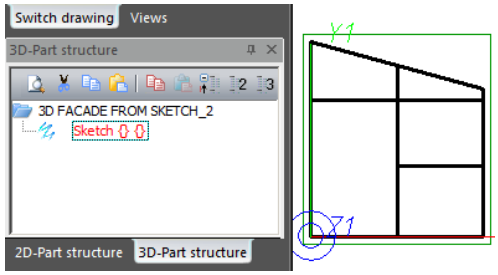


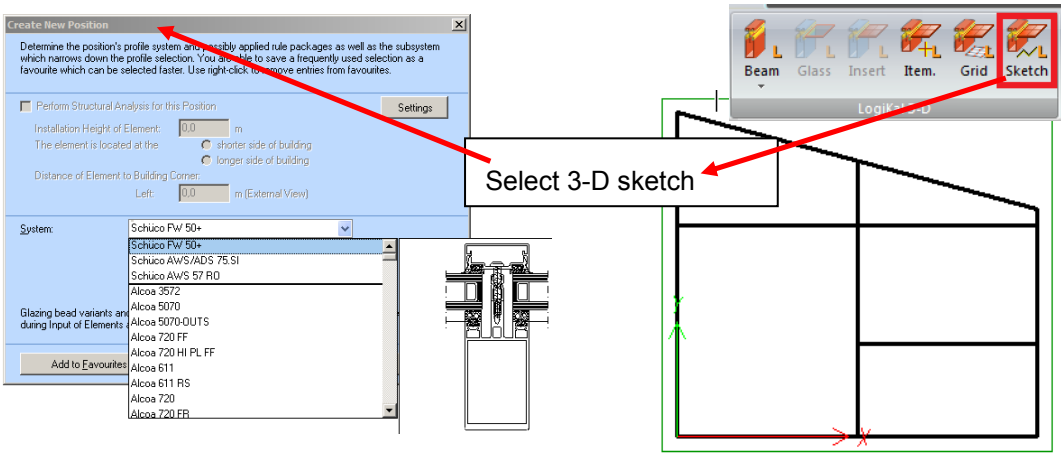
Changes can be made at any time using the HiCAD **2-D Geometry** functions. However, such changes will have **no** effect on the exported item in LogiKal. To keep changes in HiCAD and LogiKal up to date, the 2-D facade first needs to be deleted in HiCAD and LogiKal, and the 2-D creation sketch needs to be changed. Then, configure it via LogiKal again, using the **Facade with 2-D sketch** function, import it to HiCAD and, finally, re-export/import it to LogiKal.

3D

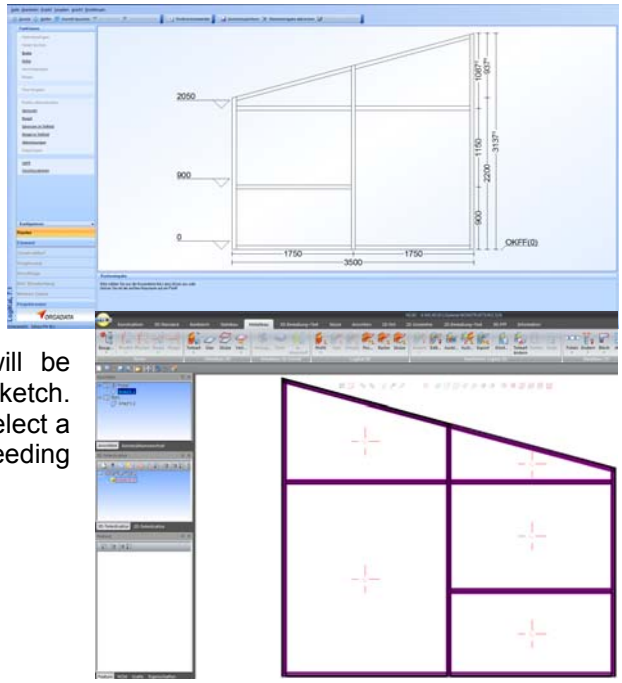
Sketch

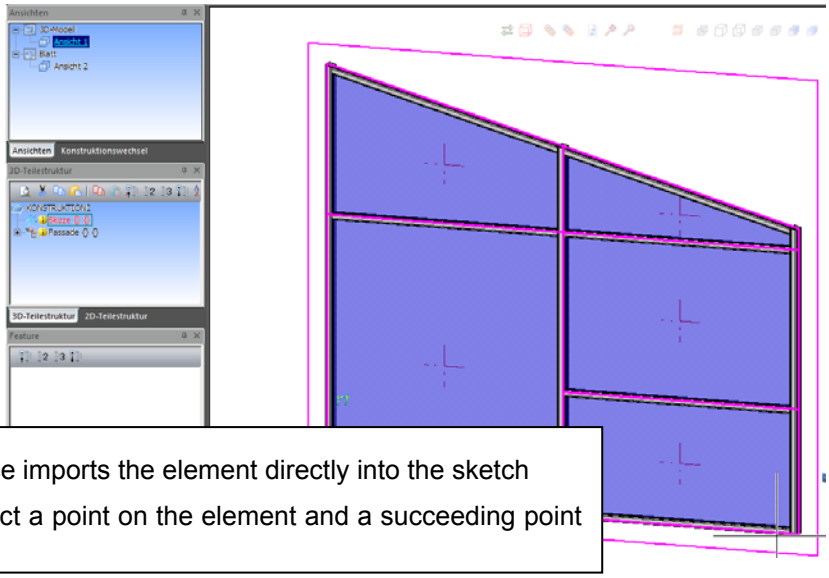
- Option **3-D Facade/insert with sketch**
- Create a sketch in a plane: Open the **Sketch** tab and select **New → Create new sketch in plane**. A sketch will be created in the 3-D Part structure and the sketching functions will be active.
- Use the sketching functions to draw the outer contour and the divisions (axes) of the facade/window element.
- On the **Metal Engineering** tab, select **LogiKal 3-D → Facade/insert with sketch**. In the LogiKal dialogue, choose a beam/profile system.





- After this, LogiKal is launched automatically and the beams/profiles will be placed on the sketch in LogiKal.
- The facade/window element can then be configured with the usual LogiKal functions.
- When performing the import, HiCAD will ask you to specify a fitting point on the part; if you **right-click twice**, the element will be directly imported into the sketch. Alternatively, you can left-click to select a point on the element, and a succeeding point in the drawing.

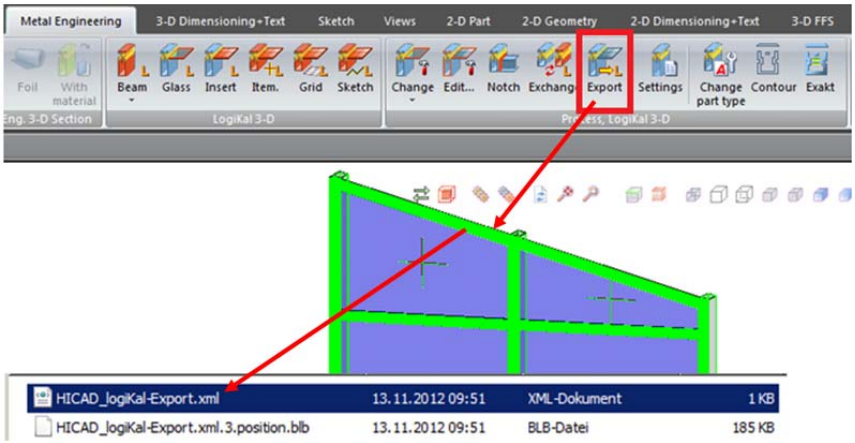




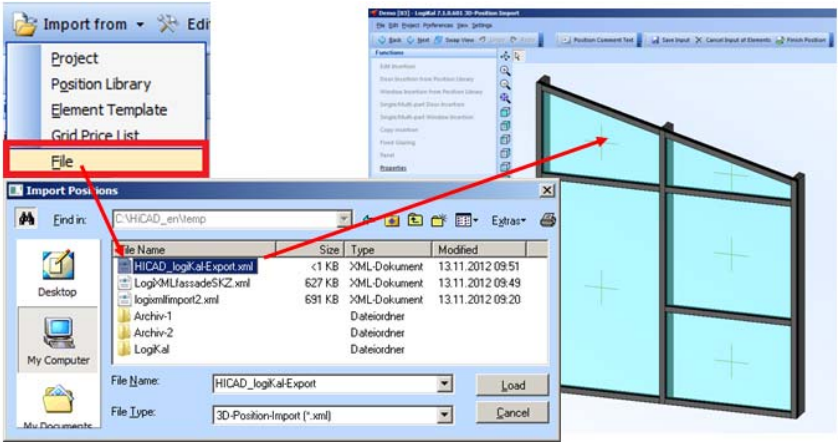
1) Left-clicking twice imports the element directly into the sketch
2) Left-click to select a point on the element and a succeeding point in the drawing

If you select this procedure, the facade/window element will only be available in HiCAD, but not in LogiKal.

- The facade/window element can be exported to LogiKal with the **Beams/profiles+facade -> LogiKal** function. For this to happen you need to select a *part of the facade/window element*, **or** to the *complete assembly* selected in the 3-D Part Structure (confirm by pressing the middle mouse button). After this, a **HiCAD_logiKal-Export.xml** file will be created in the **HiCAD\temp** directory.

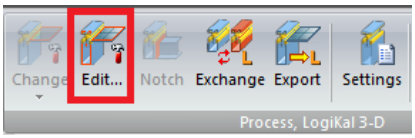


- To perform the import, start LogiKal first. Use the LogiKal function **Import from: File** to import the **HiCAD_logiKal-Export.xml** file from the **HiCAD\temp** directory. The xml file contains all information that was previously entered during the creation of the item via LogiKal.

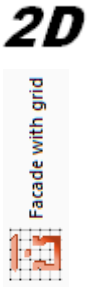


Changes to the beams/profiles can be made at any time using the HiCAD **3-D Standard** and **Steel Engineering** functions. However, such changes will have **no** effect on the exported item in LogiKal. To keep changes in HiCAD and LogiKal up to date, the 3-D facade first needs to be deleted in HiCAD and LogiKal, and the creation sketch needs to be changed. Then, configure it via LogiKal, using the **Facade with sketch** function, import it to HiCAD and, finally, re-export/import it to LogiKal.

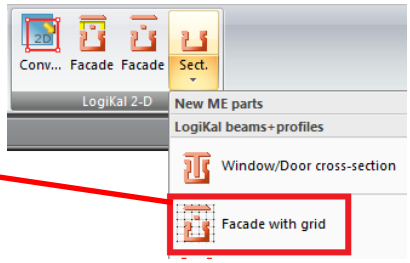
- Use the **Edit facade + inserts (3-D SE)** function to change the element via LogiKal. You cannot change the outer dimensions here, as these are pre-defined by the sketch. Glass thickness, inserts, etc. can be conveniently changed via LogiKal.

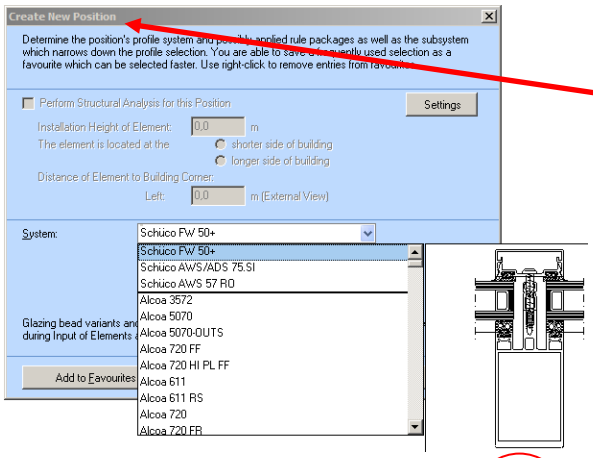


7.2.7 2-D/3-D Facade/Insert with Grid

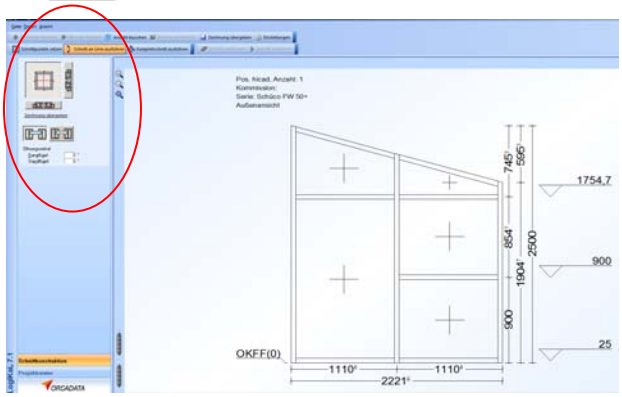


- Option **2-D facade/insert with grid**
- When you select this option, a LogiKal dialogue will be opened, enabling you to select the desired beam/profile system. LogiKal will be launched directly afterwards.
- The facade/window element can be configured with the usual LogiKal functions.



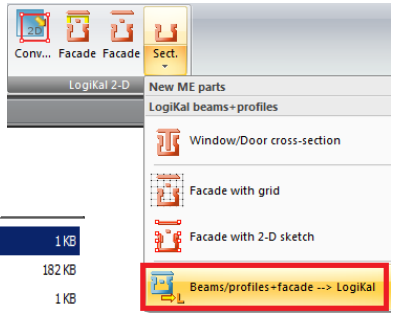


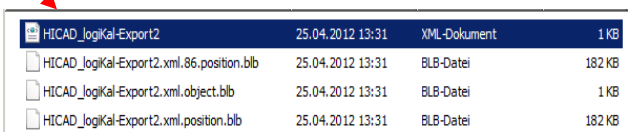
- Before performing the import to HiCAD, LogiKal asks which views / sectional views should be imported.
- Select the LogiKal function **Transfer drawing**, which closes LogiKal and imports the 2-D element to HiCAD. You will be asked to specify a fitting point (left-click twice).



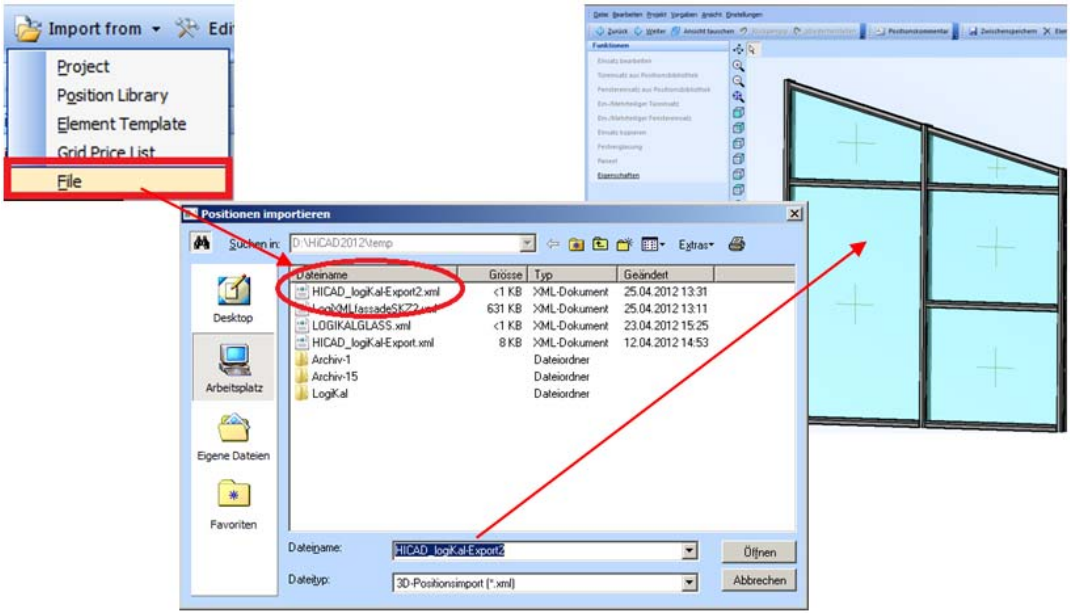
If you select this procedure, the facade/window element will only be available in HiCAD, but not in LogiKal.

- Use the **Beams/profiles + facade -> LogiKal** function to export the 2-D facade/window element to LogiKal. When you select the 2-D facade/window element with a left-click, an export file will be created in the **HiCAD\temp** directory.





- To perform the import, start LogiKal first. Use the LogiKal function Insert from file to import the **HiCAD_logiKal-Export2.xml** file from the **HiCAD\temp** directory. The xml file contains all information that was previously entered during the creation of the item via LogiKal.



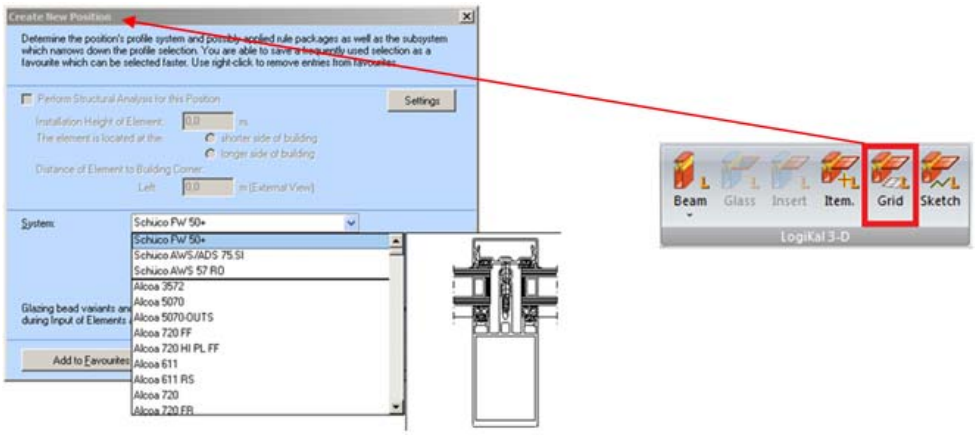
Changes can be made at any time using the HiCAD **2-D Geometry** functions. However, such changes will have **no** effect on the exported item in LogiKal. To keep changes in HiCAD and LogiKal up to date, the 2-D facade first needs to be deleted in HiCAD and LogiKal, and the 2-D creation sketch needs to be changed. Then, configure it via LogiKal again, using the "Facade with 2-D sketch" function, import it to HiCAD and, finally, re-export/import it to LogiKal.

- Option **3-D Facade + insert with grid**

Before importing the element, you need to define a processing plane, into which the window/facade element will then be imported. This will prevent a subsequent rotating or moving of the part. The import is performed according to the active coordinate system. If no processing plane is active, the element will be imported according to the World Coordinate System.

LogiKal elements are aligned as follows: The X-axis always refers to the width of the elements, the Y-axis to their height and the Z-axis to their depth!

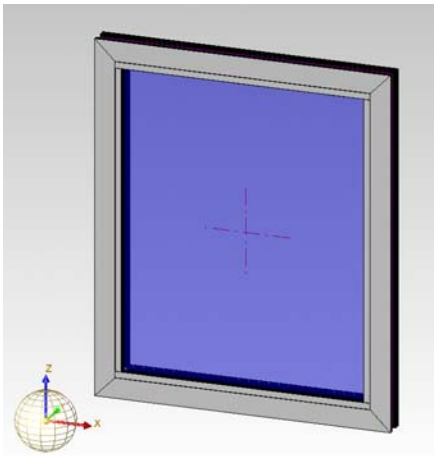
To access the function, select **Metal Engineering → LogiKal 3-D → Insert facade + insert with grid**. A LogiKal dialogue opens, enabling you to select the desired beam/profile system. LogiKal will be launched directly afterwards.



- You configure the facade/window element with the usual LogiKal functions.

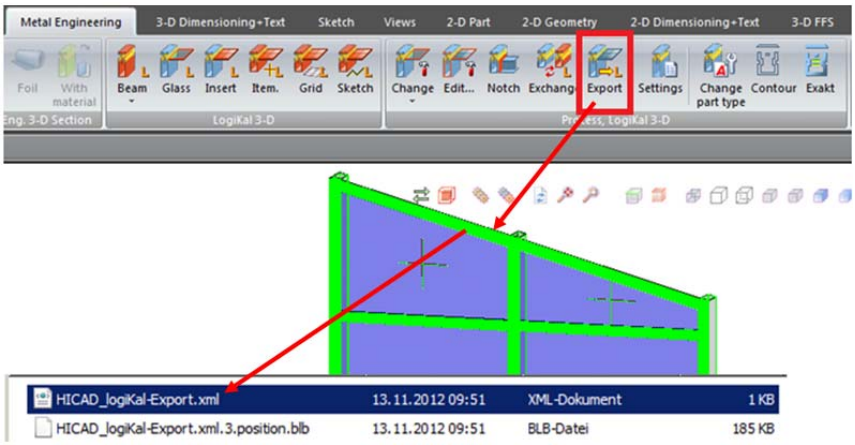
When importing the element that was configured in LogiKal, you will be asked to specify a fitting point on the part. You can then directly determine the correct position for the insertion in the HiCAD drawing by left-clicking twice.

Alternatively you can right-click twice to place the element in the origin of the processing plane.

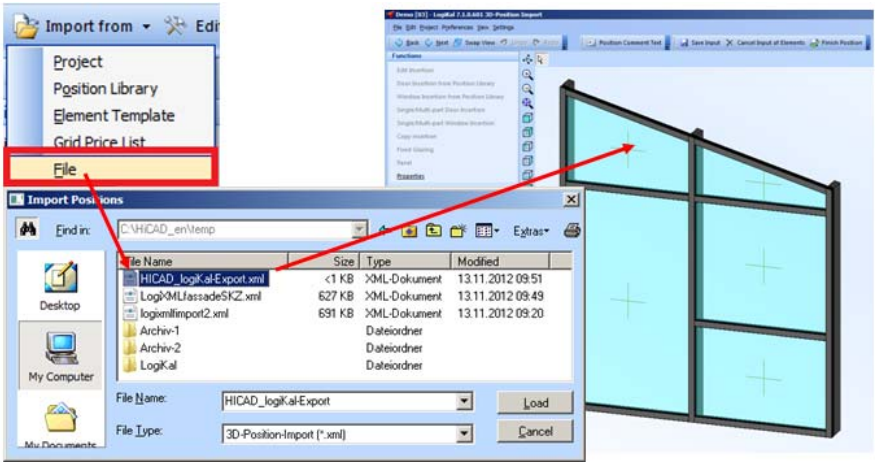


If you select this procedure, the facade/window element will only be available in HiCAD, but not in LogiKal.

- The facade/window element can be exported to LogiKal with the **Beams/profiles+facade -> LogiKal** function. For this to happen you need to select a *part of the facade/window element*, **or** to the *complete assembly* selected in the 3-D Part Structure (confirm by pressing the middle mouse button). After this, a **HiCAD_logiKal-Export.xml** file will be created in the **HiCAD\temp** directory.

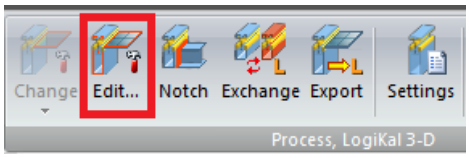


- To perform the import, start LogiKal first. Use the LogiKal function **Import from: File** to import the **HiCAD_logiKal-Export.xml** file from the **HiCAD\temp** directory. The xml file contains all information that was previously entered during the creation of the item via LogiKal.



Changes to the beams/profiles can be made at any time using the HiCAD **3-D Standard** and **Steel Engineering** functions. However, such changes will have **no** effect on the item in LogiKal. To keep changes in HiCAD and LogiKal up to date, the 3-D facade needs to be deleted in LogiKal. In HiCAD, the item then needs to be edited via LogiKal with the function **Edit facade + inserts (3-D SE)**.

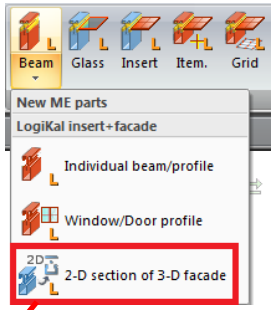
All parameters (dimensions, glass etc.) can be changed. The facade/window element changed in LogiKal replaces the element existing in HiCAD.



Caution: If you select this procedure, the changed facade/window element will only be available in HiCAD, but not in LogiKal. If you want the element to be available in LogiKal as well, you need to re-export/import it to LogiKal.

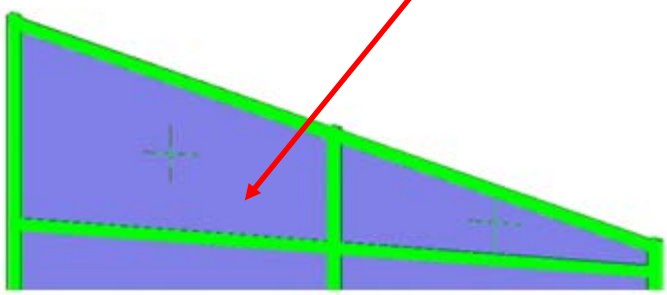
7.2.8 2-D Section of a 3-D LogiKal Facade

This option is available via **Metal Engineering → LogiKal 3-D → Beam → 2-D section of 3-D facade**.

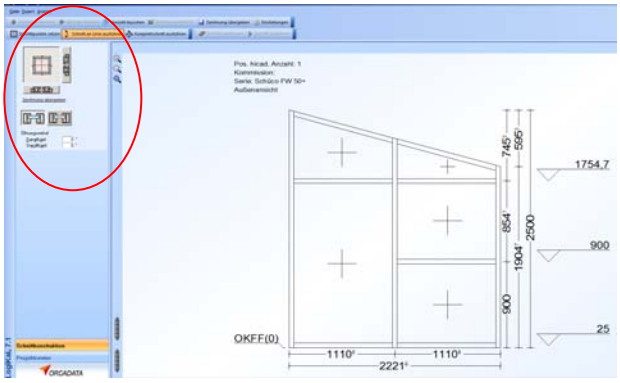


For this option a 3-D LogiKal item needs to be available in HiCAD. It must have been created with the following options:

- **Insert 3-D facade + inserts from existing LogiKal project**
- **3-D facade/insert with sketch**
- **3-D facade/insert with grid**



When you choose the **2-D section from 3-D facade** function and select the 3-D LogiKal item in HiCAD, LogiKal will be launched, displaying the sectional view of the drawing.



Now specify the 2-D views/sectional views that should be exported to HiCAD.

Select the LogiKal function **Transfer drawing**, which closes LogiKal and imports the 2-D views to the desired positions in HiCAD. You will be asked to specify a fitting point (left-click twice).

7.2.9 Change Glass in Insert

To access this function, select **Metal Engineering → Process, LogiKal 3-D → Change glass in insert**.

To be able to use this function, a glazing construction must have been created from single beams/profiles or via the HiCAD grid.

For this, the glass insertion points 14 and 24 are mandatory (see Chapter ***6.1.6 Glass Insertion Points***). These points will be automatically and invisibly supplied in the following cases:

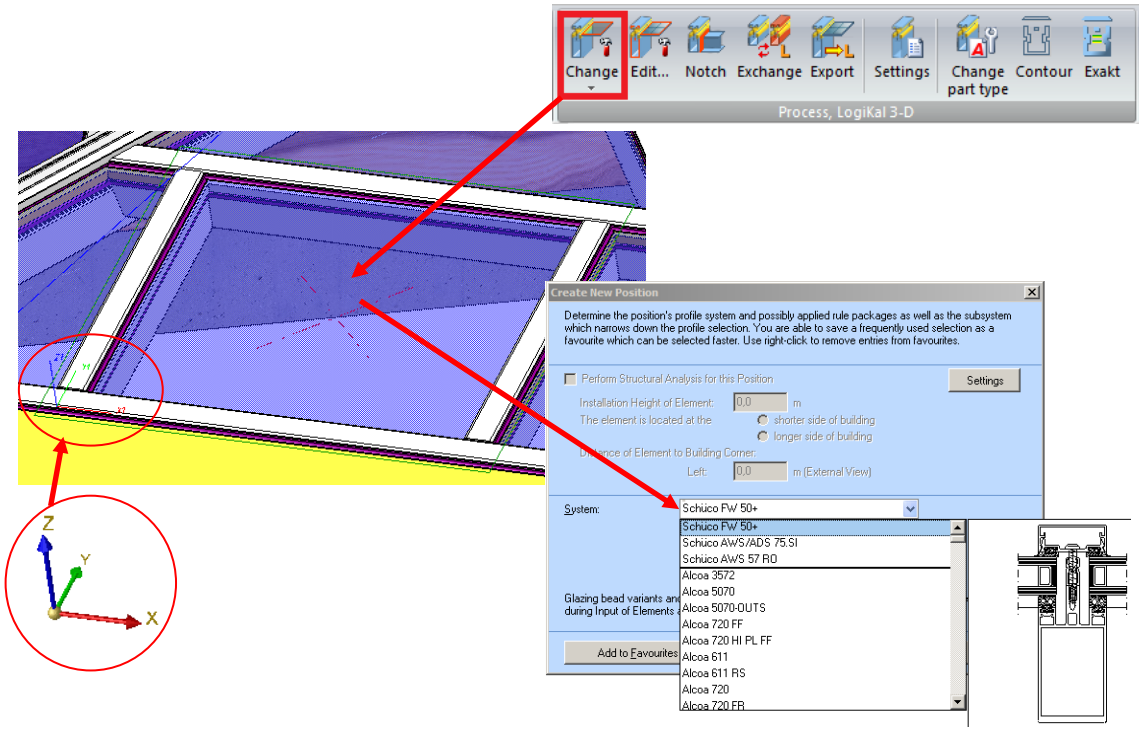
- If a facade construction is created from individual beams/profiles as described in Chapter ***7.2.1 3-D Import of Individual Beam/Profile Cross-Section***, these points will be automatically supplied.
- If a facade construction is created via the HiCAD grid option, as described in the Chapter ***6.2.4 Facade via Metal Engineering Grid (Metal Engineering Plugin)***, these points will be automatically supplied. If you use your own series beams/profiles, these points need to be created manually. In case of a use via LogiKal these points will be automatically supplied.

Before executing the **Change glass in insert** function, you need to define a processing plane.

LogiKal elements are aligned as follows: The X-axis always refers to the width of the elements, the Y-axis to their height and the Z-axis to their depth!

Then select the HiCAD glass with a mouse-click. A LogiKal dialogue will be displayed, enabling you to choose the desired beam/profile system for the insert. LogiKal will be launched directly afterwards, so that you can configure the insert with the usual LogiKal options.

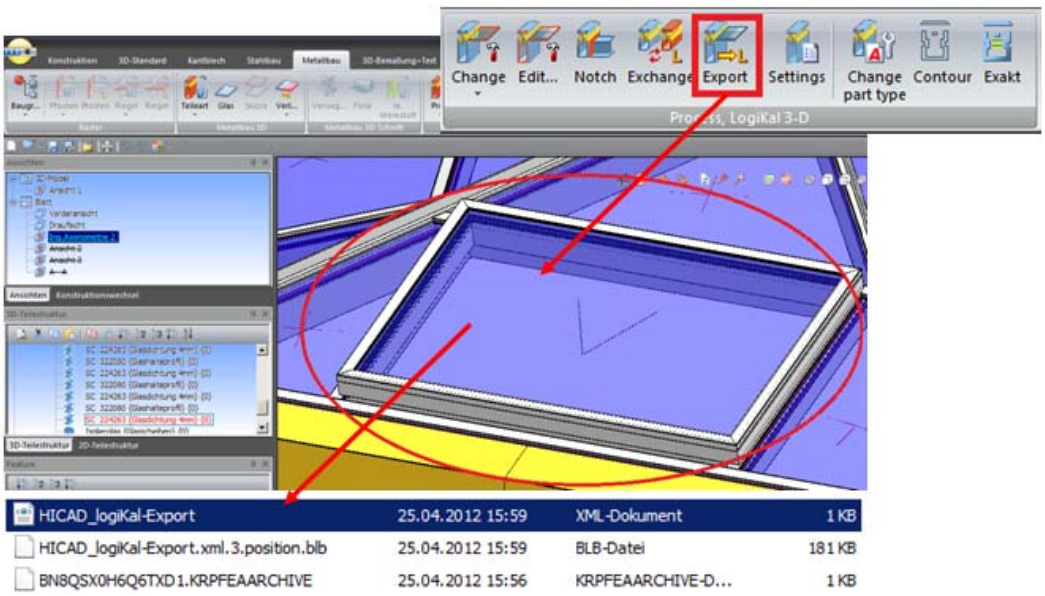
After finishing the procedure, the glass will be deleted automatically, and the LogiKal insert will be created in the plane of the glass.



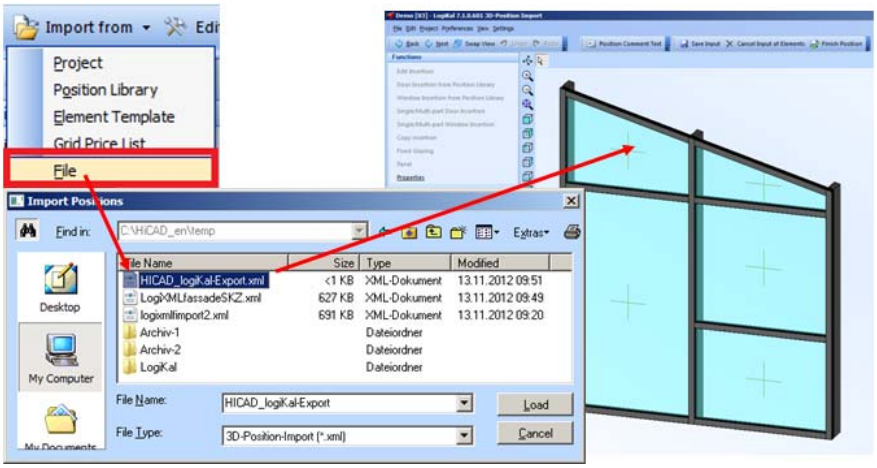
Caution: If you select this procedure, the changed facade/window element will only be available in HiCAD, but not in LogiKal. If you want the element to be available in LogiKal as well, you need to export/import it to LogiKal. See also Chapter ***7.2.10 Export Facade/Insert to LogiKal.***

7.2.10 Import Facade/Insert to LogiKal (3-D)

- The facade/window element can be exported to LogiKal with the **Beams/profiles+facade -> LogiKal** function. For this to happen you need to select a *part of the facade/window element*, **or** to the *complete assembly* selected in the 3-D Part Structure (confirm by pressing the middle mouse button). After this, a **HiCAD_logiKal-Export.xml** file will be created in the **HiCAD\temp** directory.



- To perform the import, start LogiKal first. Use the LogiKal function **Import from: File** to import the **HiCAD_logiKal-Export.xml** file from the **HiCAD\temp** directory. The xml file contains all information that was previously entered during the creation of the item via LogiKal.

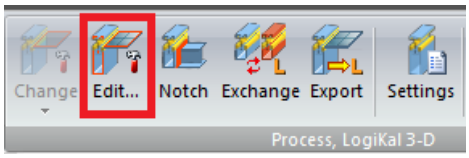


The LogiKal 3-D export can be used for the following options:

- 7.2.1 3-D import of individual beam/profile cross-sections
- 7.2.5 3-D import of existing LogiKal items
- 7.2.6 3-D facade/insert with sketch
- 7.2.7 3-D facade/insert with grid
- 7.2.12 Facade via Metal Engineering grid (Metal Eng. Plugin)

7.2.11 Edit Facade/Insert via LogiKal (3-D)

Use the **Edit facade + inserts (3-D SE)** function to change the element (dimensions, glass, etc.) via LogiKal. It will be directly updated in HiCAD.



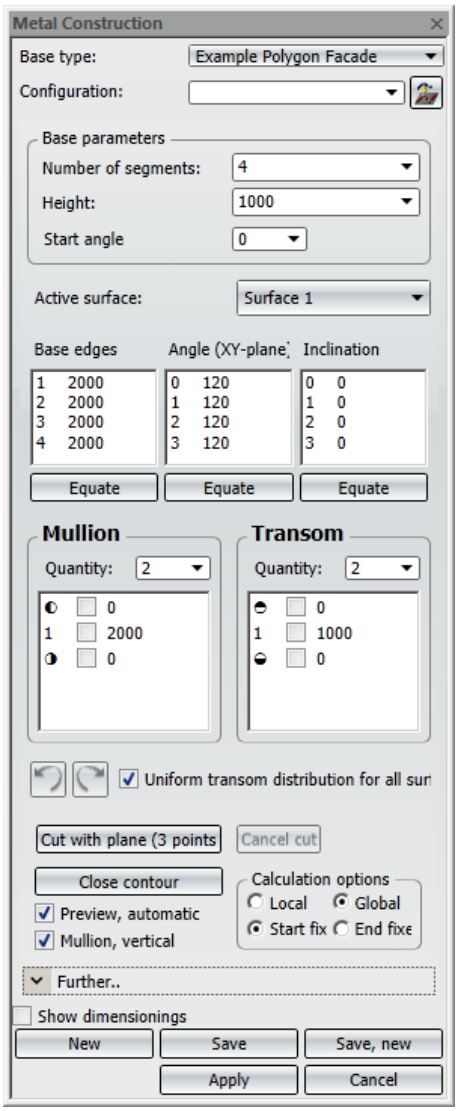
Caution: If you select this procedure, the changed facade/window element will only be available in HiCAD, but not in LogiKal. If you want the element to be available in LogiKal as well, you need to delete the item in LogiKal, re-export the HiCAD item, and import it to LogiKal. See also Chapter ***7.2.10 Export Facade/Insert to LogiKal.***

Edit facade + insert via LogiKal can be used for the following options:

- 7.2.5 3-D import of existing LogiKal items
- 7.2.6 3-D facade/insert with sketch
- 7.2.7 3-D facade/insert with grid

7.2.12 Place Beams/Profiles on Grid via LogiKal

In this chapter, beams and profiles are placed on a Metal Engineering grid (Metal Construction plugin) via LogiKal. Prior to this, the steps described in Chapter ***8 Facade via Metal Engineering Grid (Metal Construction Plugin)*** need to be performed.



1. The creation of the grid is described in Chapter ***8 Facade via Metal Engineering Grid (Metal Construction Plugin)***. Please perform the steps in this chapter beforehand.

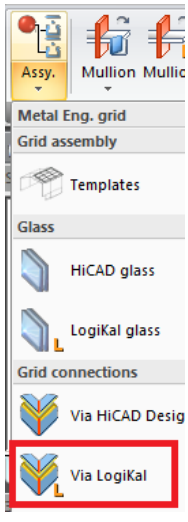
2. On the **Metal Engineering** tab, select **Grid → Mullion** or **Transom via LogiKal** to place the beams/profiles on the sketches. You can also use "collection" functions for the selection of equally aligned axes:

   - CTRL + left-click = Collection/deselection of individual (sketch) axes
   - SHIFT + left-click = Automatic collection of equally aligned (sketch) axes

3. Right-click to open a LogiKal dialogue enabling you to configure the desired mullions/transoms. Bent mullions are automatically recognised and the hip angles will be initialised with a corresponding value in the LogiKal dialogue.

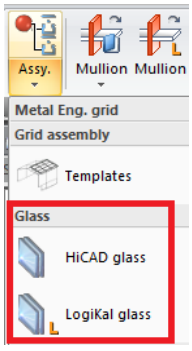
4. The beams/profiles will be automatically placed on the sketches.

5. Mullion/Transom notching = 90°: Via LogiKal. **Metal Engineering → Grid → Assy. → Grid connections, via LogiKal**.



6. Mullion/Transom notching ≠ 90° (Schifter notch): Via HiCAD Design Variant. **Metal Engineering → Grid → Assy. → Grid connections, Via HiCAD Design Variant.**

7. Glass insertion can be performed automatically and across the complete facade via automatically via **Metal Engineering → Grid → Assy. → HiCAD glass** or **LogiKal glass**.



8. Here it is important that you define a suitable glass pane, or select a glass pane with the glass thickness that was previously specified in the LogiKal dialogue.

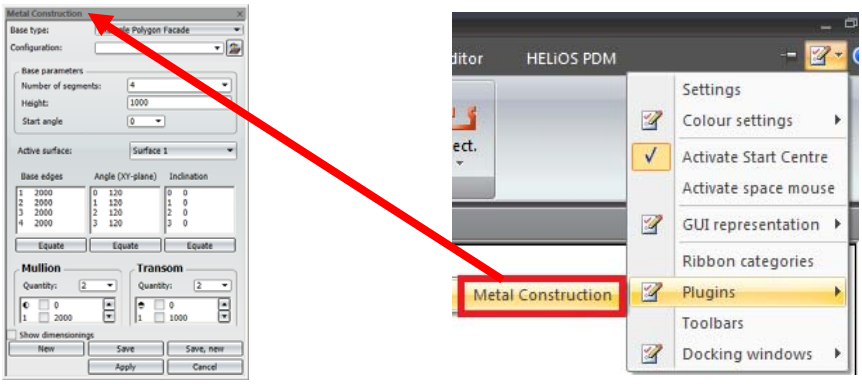
Use the grid connection via LogiKal for mullion/transom notchings = 90°, and the grid connection via HiCAD Design Variant for mullion/transom notchings ≠ 90° (Schifter notch).

8 Facade via Metal Engineering Grid (Metal Construction Plugin)

This chapter describes the creation of facades with the help of the Metal Engineering grid (Metal Construction plugin).

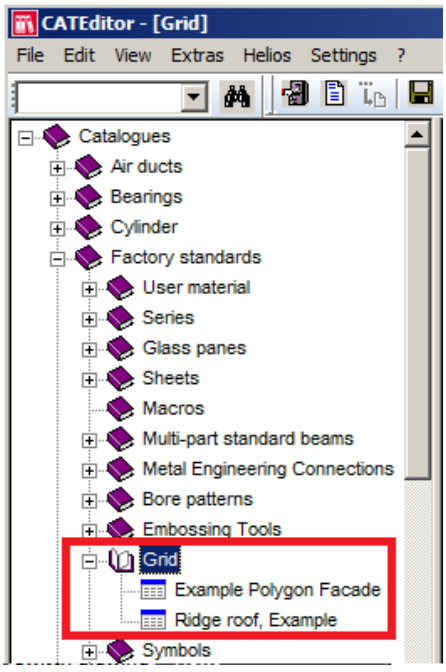
For a convenient working with the **3-D planning grid**, Metal Engineering grid templates are provided in the **Metal Construction** plugin.

You activate the **Metal Construction** plugin via **Settings** > **Plugins**.



The Metal Construction plugin enables the configuration of individual facade contours via the Grid functionality with only a few mouse clicks, and assign mullions, transoms and glass panes to them.

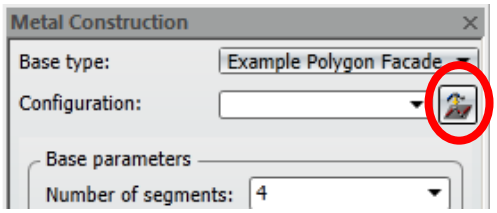
For this purpose you can choose between the two templates “Ridge roof” and “Polygon facade” (saved in the HiCAD Catalogue) from the Base type selection box at the top of the plugin window.

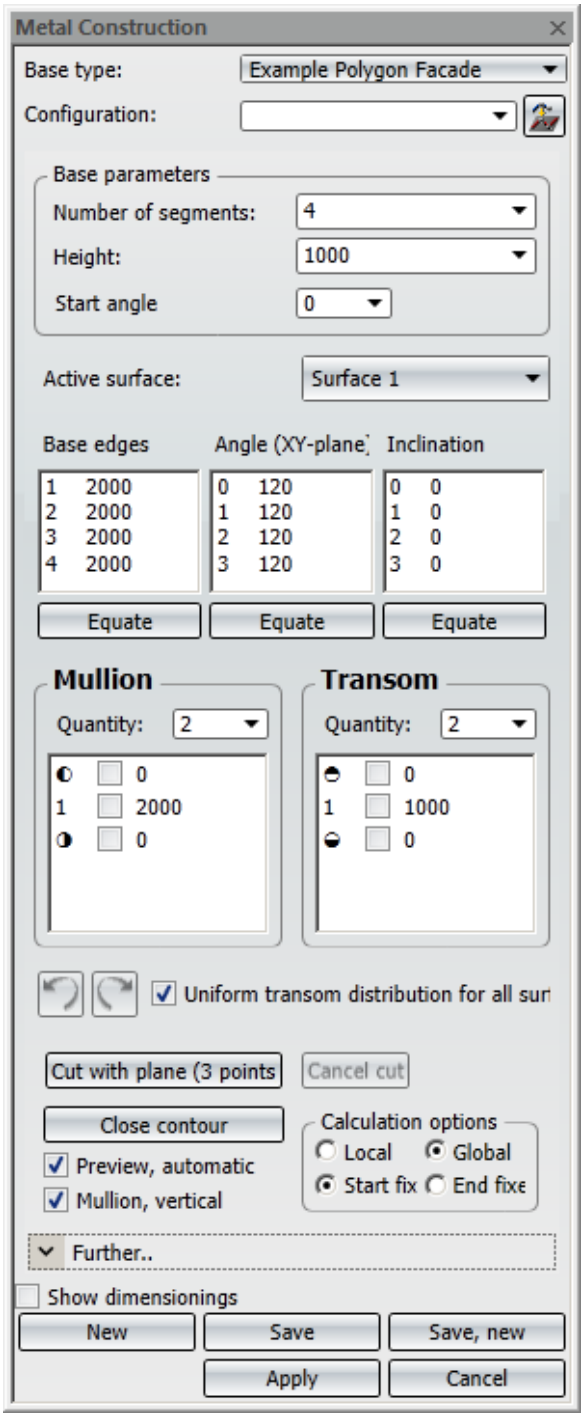


The sample templates in the Catalogue Editor can be accessed via **Catalogues → Factory standards → Grid**.

In the **Configuration** selection field you can enter the names for the specific sample facades you created. These can then be saved to the catalogue and reloaded at any time.
At the bottom of the plugin window you can apply or save changes, create a new configuration, or save an existing configuration with a different name by clicking the corresponding buttons.

Use the **Select sketch** button to apply the facade contour of a sketch drawn in the top view. Click the button, then identify the sketch and, after generation of the facade, change the dimension parameters in the plugin navigator if required.



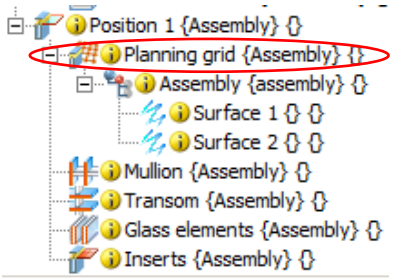


1. In the Base parameters area, enter the Number of segments and the Height. A grid consisting of connected sketches in the desired form will then be available to you.

2. Apply the sketches to your drawing.

3. If required, modify the sketches with the usual standard functions, e.g. to obtain oblique transoms.

4. Select **Grid → Grid assembly, new** Create the predefined assembly structure. In the structure browser, move the grid assembly into the “Planning grid” assembly.



5. Activate the **Planning grid** assembly.

6. Use the **Mullion** or **Transom from catalogue** function to place the beams/profiles onto the sketch edges.

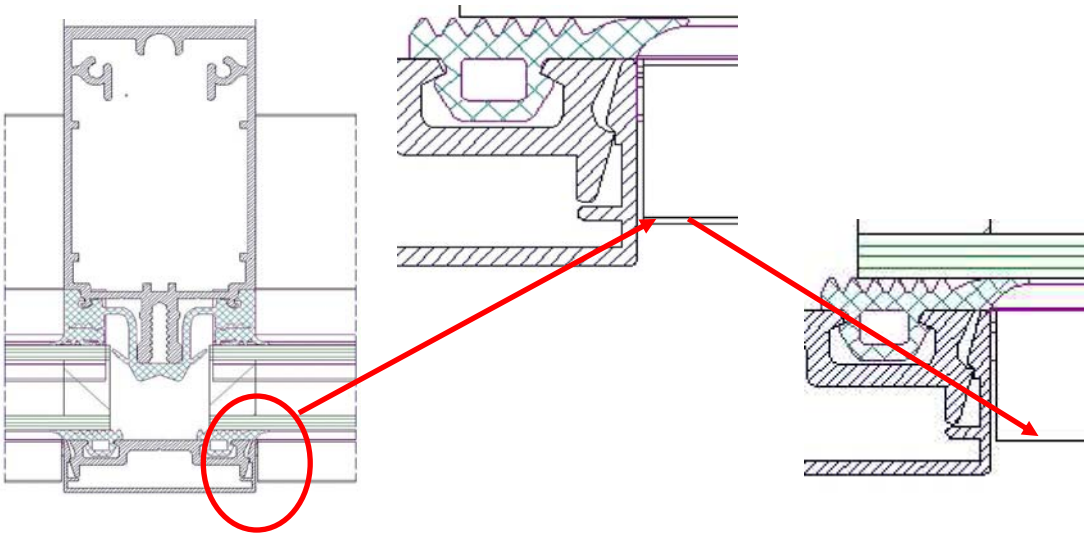
The placing of beams/profiles on grids is described in the following chapters:

- 6.2.4 Insertion on Grid via User-Defined Beam/Profile Cross-Sections
- 7.2.12 Place Beams/Profiles on Grid via LogiKal

9 Representation Options

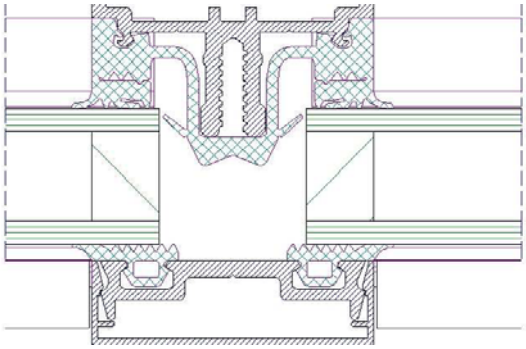
9.1.1 Tangential Edges/Lines

- **Tangential edges** can be switched off as follows: Right-click the view frame (pink dotted rectangle) and select **Properties → Show/Hide edges in view**.



Select **Settings / Basic settings / 3-D / View / Visualisation / Hide tangential edges: Yes** in you want tangential edges to be hidden in new views by default.

- To hide the edges behind the glass, you need to change the transparency of the glass in the sectional view. To do this, right-click and select **Properties → Properties in view, Transparency → Not transparent**.



9.1.2 View-Dependent/Part-Dependent Control of Representation Type

In all of the previous examples, the EXACT representation was used for the insertion of beam/profile groups. In some cases, however, a SIMPLE representation of the beam/profile groups would be preferable:

- In overall views, very many single edges would be rather confusing. A simplified contour would be sufficient.
- Important points might not be available for identification (e.g. beam/profile corners when there are radii, top/bottom edges of glass insertion surfaces etc.).

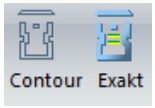
Therefore, HiCAD offers the following three representation types:

Representation type	**Description**	**Examples of use cases**
EXACT representation	■ All individual beams and profiles of a beam/profile group are displayed ■ Each individual beam/profile is displayed as exactly as possible ■ This is the default setting for newly inserted beam/profile groups	■ Cut beams/profiles in 1:1 sectional views ■ Representation in enlarged detail views (perspective views, shaded details etc.) ■ Checks of 3-D design drafts
CONTOUR representation	■ Only one CONTOUR part is displayed for the complete beam/profile group ■ The individual elements of the beam/profile group are hidden via list view functions ■ A view- and part-dependent switching between CONTOUR and EXACT representation is possible	■ Beam/profile representation in overall views ■ For an easier identification in design drafts
SIMPLE representation	■ All individual beams and profiles of a beam/profile group are displayed ■ Each individual beam/profile is displayed with a simplified geometry ■ By default, the generation of simplified geometries is deactivated in the LogiKal interface configuration ■ A part-dependent switching between EXACT and SIMPLE is possible ■ The parts are displayed in all views of a drawing either exact or simplified	■ Performance enhancements for large drawings during the design draft phase

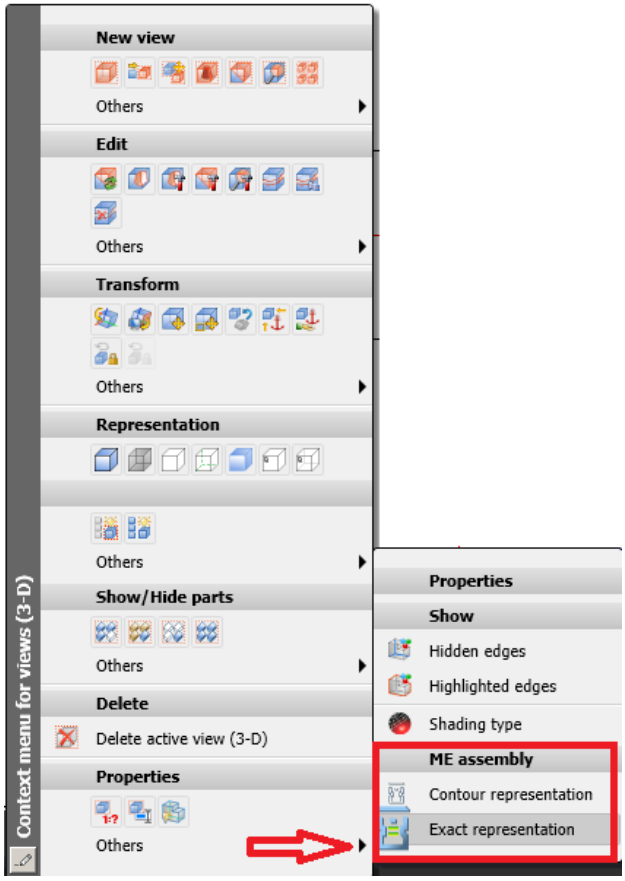
9.1.3 Switching Between Exact and Contour Representation

Functions for a view-dependent, direct switching between **EXACT** geometry and **CONTOUR** representation are available, so that you do not need to use the general Hide/Show functions for this. You can access these functions as follows:

- In the **Process, LogiKal 3-D** function group of the **Metal Engineering** tab.



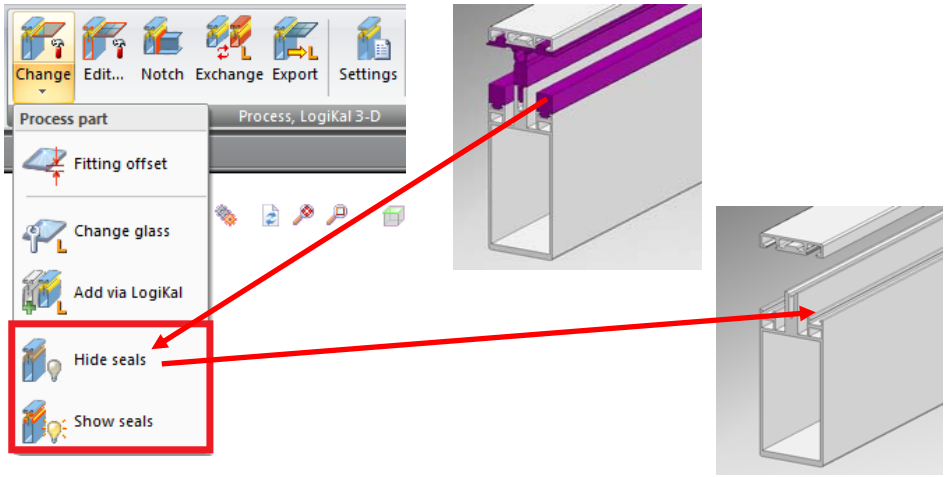
- Open the context menu for views (right-click pink dotted view frame) and select **Others → ME assembly.**



9.1.4 Hide Seals and Rubber Parts

Seals and rubber parts can be hidden and redisplayed during designing.

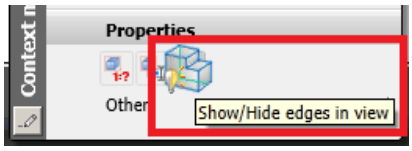
- On the **Metal Engineering** tab, select **Process, LogiKal 3-D → Change → Hide seals**.



9.1.5 View-Dependent Hiding/Showing of Axes and Tangential Edges

In drawing views you often want profile axes, tangential edges of beam/profile radii etc. to be invisible, i.e. hidden. However, their visualisation is rather useful during designing in 3-D.

To hide or show edges in views, use the **Hide/Show edges in view** function in the context menu for views.

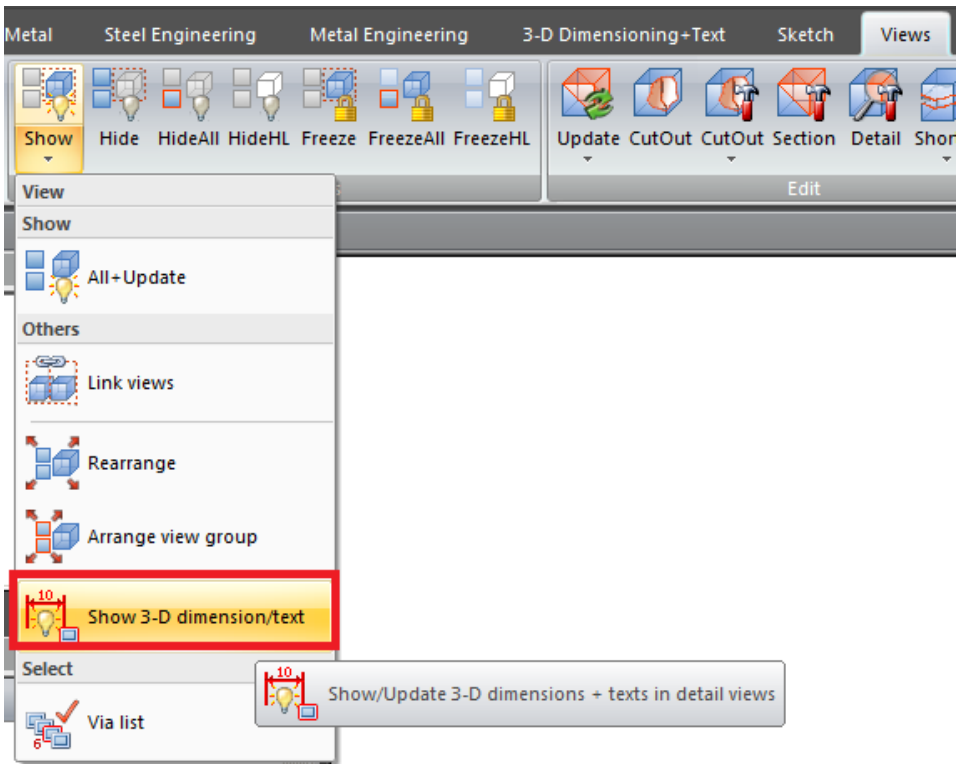


9.1.6 Show Facade Sections from Detail Views + Show Dimensions/Texts

The creation of complete sections through facades was formerly achieved via the creation of large-scale sectional views and the shorting of views.

If further functionalities for sectional views are required, e.g. drawing- or sheet area-comprehensive, associative dimensioning, the following procedure can be used:

- Create an unshortened sectional view with any scale
- Add annotations and dimensionings in the unshortened sectional view
- Create detail views for the visible areas of the complete section, and arrange the detail views in the drawing(s)
- Show all dimensions and annotations of the unshortened sectional view in the detail views (open the **Views** tab and select **View Functions → Show → Show 3-D dimension/text**). In the active detail view, all 3-D dimensions and part annotations of the reference view that lie within the detail will be shown/updated

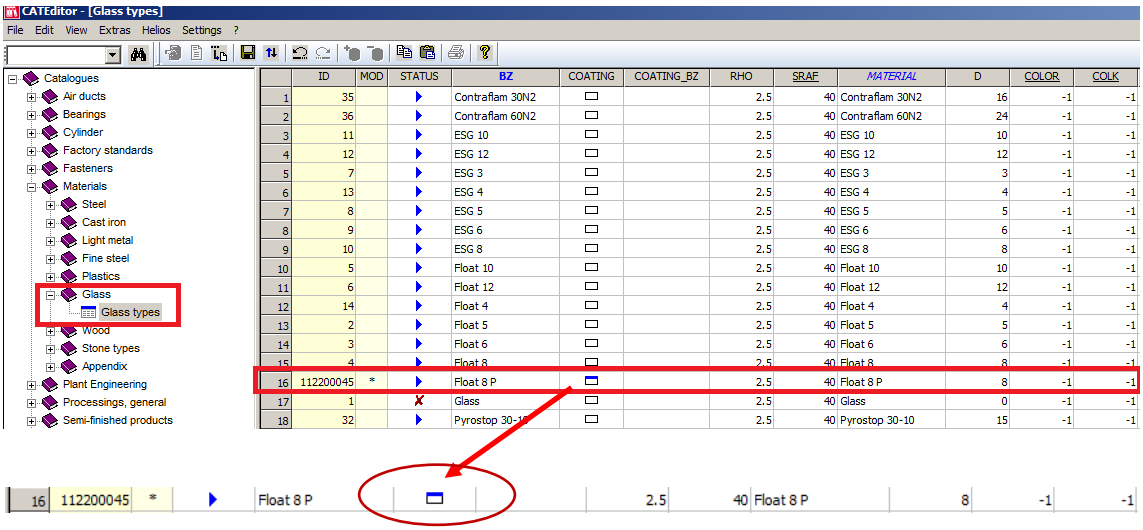


10 Glass Functions

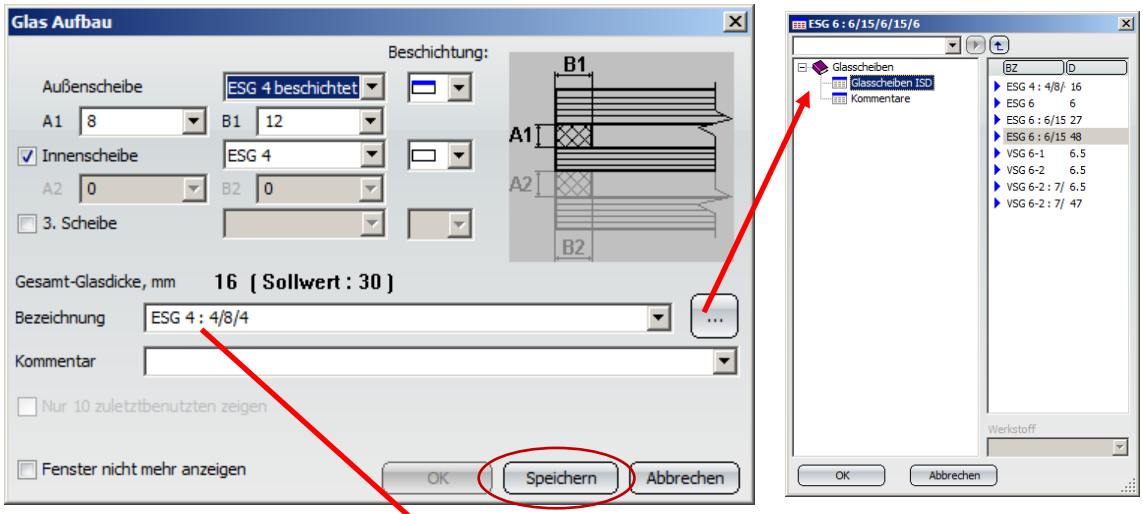
10.1.1 Glass Insertion Dialogue and Settings

To expand the list of glass types, open the Catalogue Editor and go to Materials → Glass → Glass types.

For example, if you copy Row 7 with the ID 13 and paste it at the bottom, you have the option to select a coating side for this glass pane in the “Coating” column.



In the glass insertion dialogue the coating side will be evaluated as follows:



The coating will be considered for itemisation and identical part search, i.e. the workshop drawings of the glass panes will always be created from the exterior view.

The glass structure needs to be given a Designation and saved to the HiCAD catalogue.

The glass cross can be hidden/shown via the following files:

- HiCAD/sys … stb_parameter.dat → Line 90 (Display glass ID)

```
Display glass id (cross in the glass centre of gravity) (0 = no, 1 = yes) : - Glaskennung darstellen (Kreuz im Glasschwerpunkt) (0 = nein, 1 = ja) :
1
```

- HiCAD/sys … stb_parameter.dat → Line 199 (Position of glass ID)

```
Position of glass id (0 = centre, 1 = outside, -1 = inside, 2 = inside and outside) : - Position der Glaskennung
0
```

10.1.2 Glass Insertion in 3-D

HiCAD offers a function for the 3-D insertion of glass panes in mullion-transom constructions. The glass structures are managed in catalogues. Further, customer-specific catalogues, tables and glass panes can be added at any time.

The **Insert glass** function can be found in the **Metal Engineering, 3-D** function group.

When you click on a glass field, the glass dimensions will be automatically calculated on the basis of the limiting profiles, by taking the glass insets (see also 6.1.6 Glass Insertion Points) of each limiting profile into account.

After selecting the function, you can also right-click first, and then left-click to “collect” the profiles forming the frame.

After selection of the glass field (left-click a point on the glass field; if required: right-click to select individual profiles) you need to define the glass structure by specifying the individual glass panes, the gap width and the edge compound width.

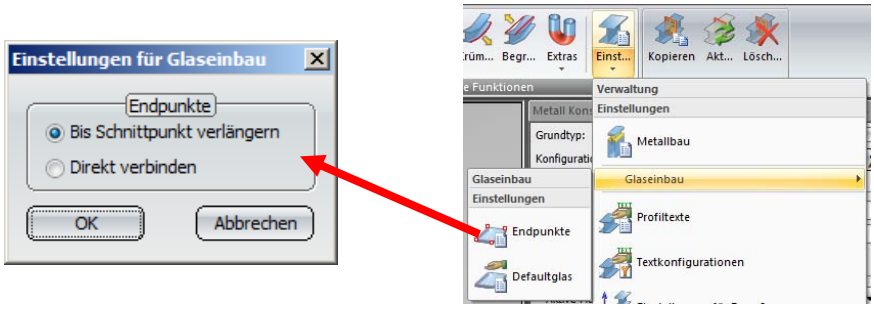
10.1.3 Automatic Glass Insertion – Directly Connect Beam/Profile End Points

When a 3-D glass insertion is performed, the axes of the limiting beams/profiles will be lengthened to an intersection point to enable the calculation of the glass sketch. This means that a glass insertion is even possible if the beam/profile axes do not directly intersect.

However, if no intersection points of the beam/Profile axes can be calculated, a glass insertion via the standard option will not be possible. Such cases can occur, for example, if no limiting profile exists for one or several sides of a glass.

To still enable an automatic insertion (i.e. without having to define a sketch manually), you can use the Steel Engineering function for the handling of end points:

Open the **Steel Engineering** tab and select **Further functions → Settings → Glass insertion → End points**.



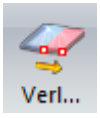
10.1.4 Hidden Line Representation, View-Dependent Glass Pane Transparency

Normally, all parts located on Layer 40 will be transparent. This is the default setting for glass panes, i.e these will be displayed correctly in shaded views.

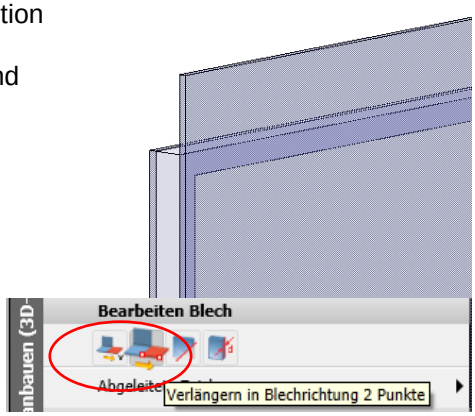
However, since no Hidden Line calculation is performed for transparent parts, a transparent representation in sectional views would cause problems.

Therefore, use the function **Properties → Properties in view, Transparency** to activate/deactivate the transparency for each view, independent of Layer 40.

10.1.5 Stepped Glass



The creation of stepped glass is performed with the function **Metal Engineering → Metal Engineering, 3-D → Lengthen…**. To lengthen a glass, select it at its edge and lengthen it via value entry or via the picking of points.



Alternatively via right-click on glass pane:

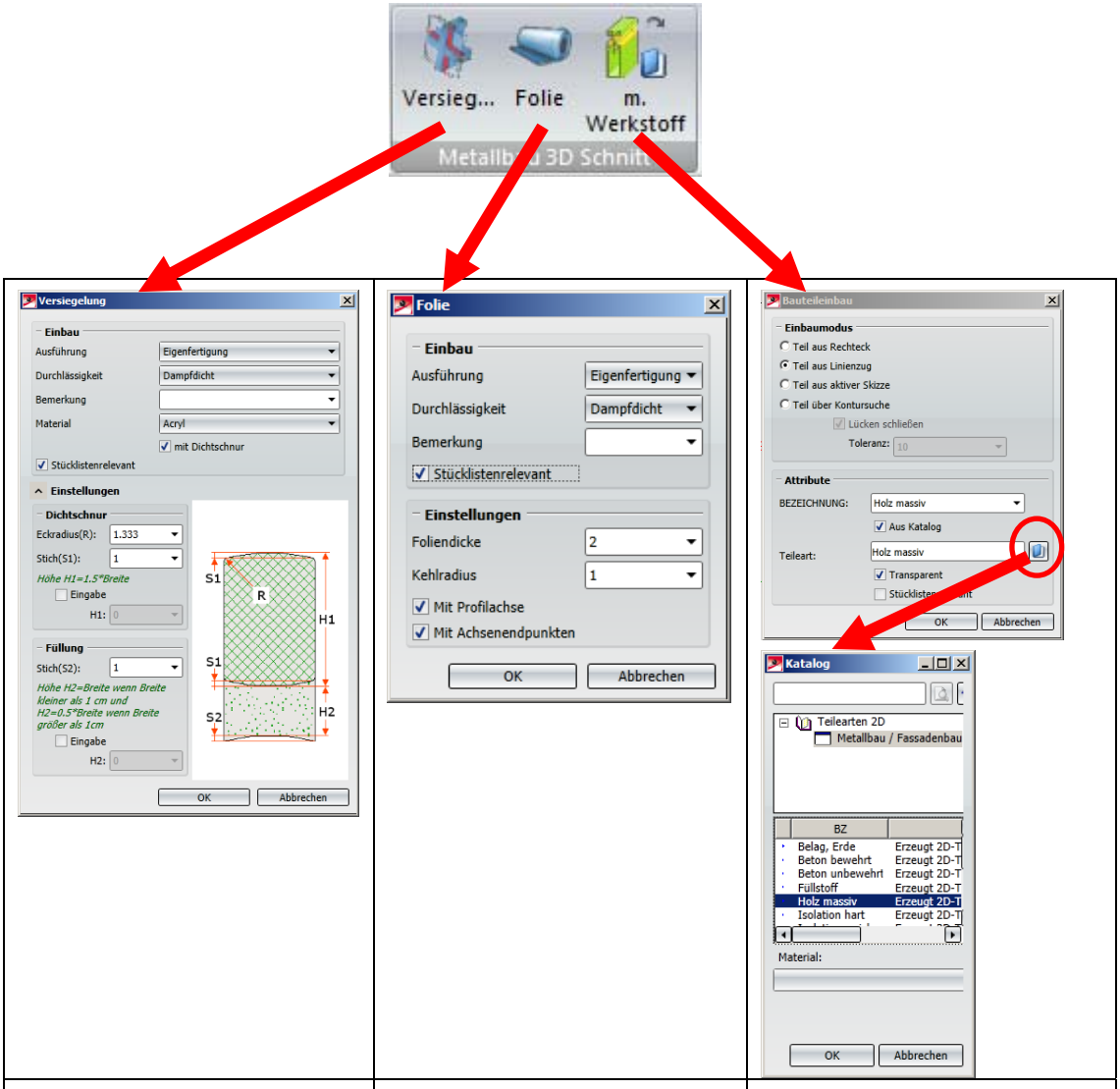
Select **Sheet Processing: Lengthen in sheet direction** (value), or **Lengthen in sheet direction, 2 points**.

11 Create Wall Connections (3-D)

- 3-D sectional views for detail planning

The tools for detail planning can be found at **Metal Engineering → Metal Eng., 3-D Section**. These functions can **only** be used in sectional and detail views. You work in the same way as in 2-D drawings while using the benefits of 3-D design at the same time. The parameters for annotations are automatically entered in the part attributes.

The tools for detail planning can only be used in sectional and detail views. For previously created sketches, please make sure that the processing plane actually lies in the sectional view!



The insertion takes place via the picking of two edges. The edges do not need to run parallel.	The insertion requires a previously creates sketch. Use the “Foil” option to identify the sketch, and then change the desired foil parameters. During insertion you will be asked to specify the direction of the foil thickness.	Several insertion options are possible (see screenshot on next page). Select the desired part types via the catalogue.

Fitting mode:

- Part from rectangle → Create a rectangle from two diagonally opposite points in the sectional/detail view. After this, HiCAD automatically generates a solid body in positive and negative Z-direction. After updating the sectional/detail views the correct hatchings will be displayed.

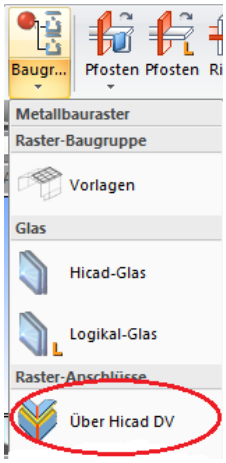
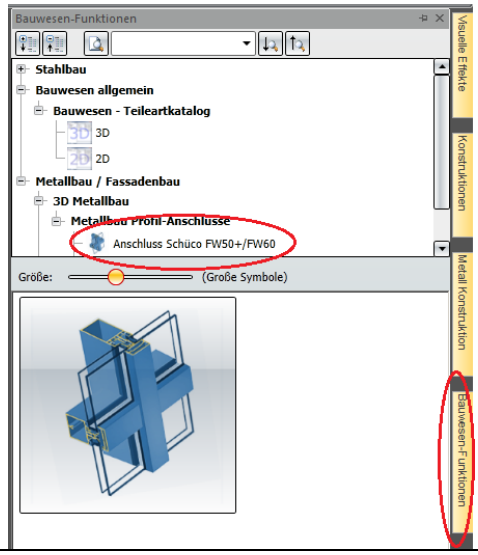
- Part from polyline → Draw a polyline in the sectional/detail view. Please make sure that the polyline is closed, i.e. does not have any gaps. After this, HiCAD automatically generates a solid body in positive and negative Z-direction. After updating the sectional/detail views the correct hatchings will be displayed.

- Part from active sketch → Draw a sketch (closed polyline) in the sectional/detail view. Please make sure that the polyline is closed, i.e. does not have any gaps. After this, HiCAD automatically generates a solid body in positive and negative Z-direction. After updating the sectional/detail views the correct hatchings will be displayed.

- Part via contour search → This option is normally used for the part type “Insulation soft/hard”. Click the desired area of a sectional/detail view. HiCAD auto-generates a sketch with the boundary edges and creates a solid body in positive and negative Z-direction. You can specify a tolerance to directly close gaps that are smaller than the entered value. After updating the sectional/detail views the correct hatchings will be displayed.

12 HiCAD Design Variants for Metal Engineering

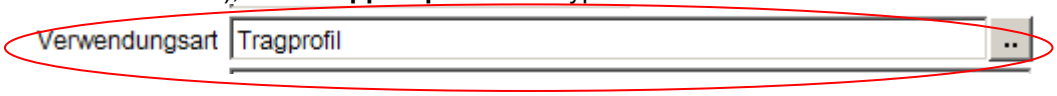
12.1 HiCAD Mullion-Transom Notching

Connection via grid	Individual insertion
Metal Engineering → Grid → Assy. → Via HiCAD Design Variant - see also Chapter 7.2.12 Place Beams/Profiles on Grid via LogiKal 	**'Civil Engineering functions' docking window → Metal Engineering/Facade Engineering → 3-D Metal Engineering → Metal Engineering beam/profile connections** - see also Chapter 7.2.1 3-D Import of Individual Beam/Profile Cross-Sections via LogiKal 

For the HiCAD Design Variant you normally use the glass insertion points (Chapter ***6.1.6 Glass Insertion Points***) and the additional **Points 15 and 25**. The Type of use of the mullion and transom needs to be set to **Support profile**.

Assigning of the Type of use:

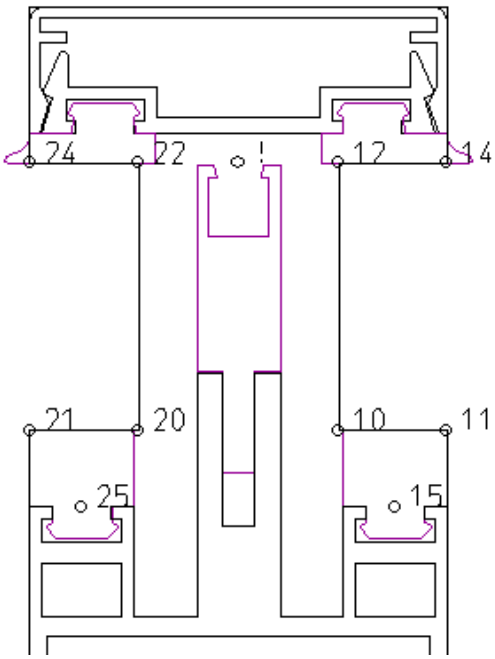
In the Part attributes window (double-click beam/profile, or right-click beam/profile and then select “Part attributes”), enter **Support profile** for Type of use.

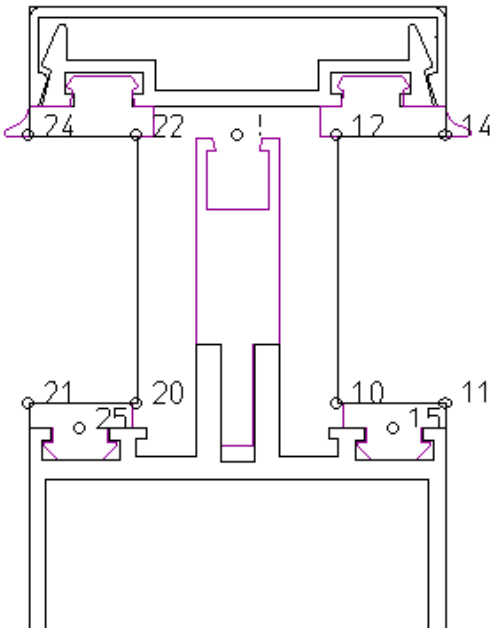


The definition of the glass insertion points and the 15 and 25 takes place via the creation of isolated points with corresponding point numbers.

The point numbers are defined as follows:

- Point s10 and 20 define the glass inset → The points must be located in the 2-D part GLASS
- Point s11 and 21 define the direction of the glass insertion → The points must be located on the 2-D part GLASS
- Point s12 and 22 define the glass thickness → The points must be located on the 2-D part GLASS
- Point s14 and 24 are required for the “Change glass in insert” option. These points are not mandatory → The points must be located in the 2-D part GLASS
- Points 15 and 25 define the position of the mounting bores on mullion and transom → The points must be located in the 2-D part MULLION and TRANSOM, respectively.

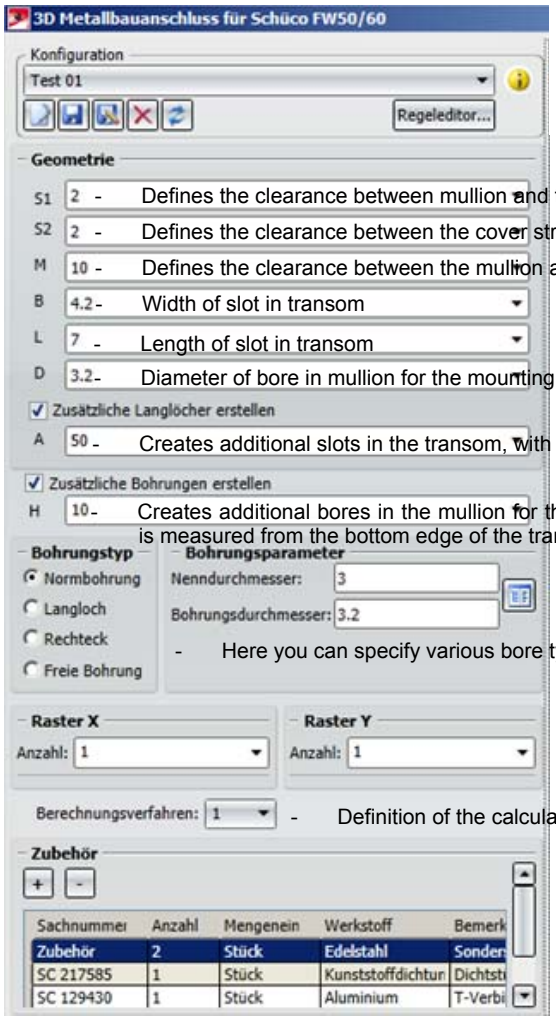




The Metal Engineering Design Variant can be used for the following design tasks:

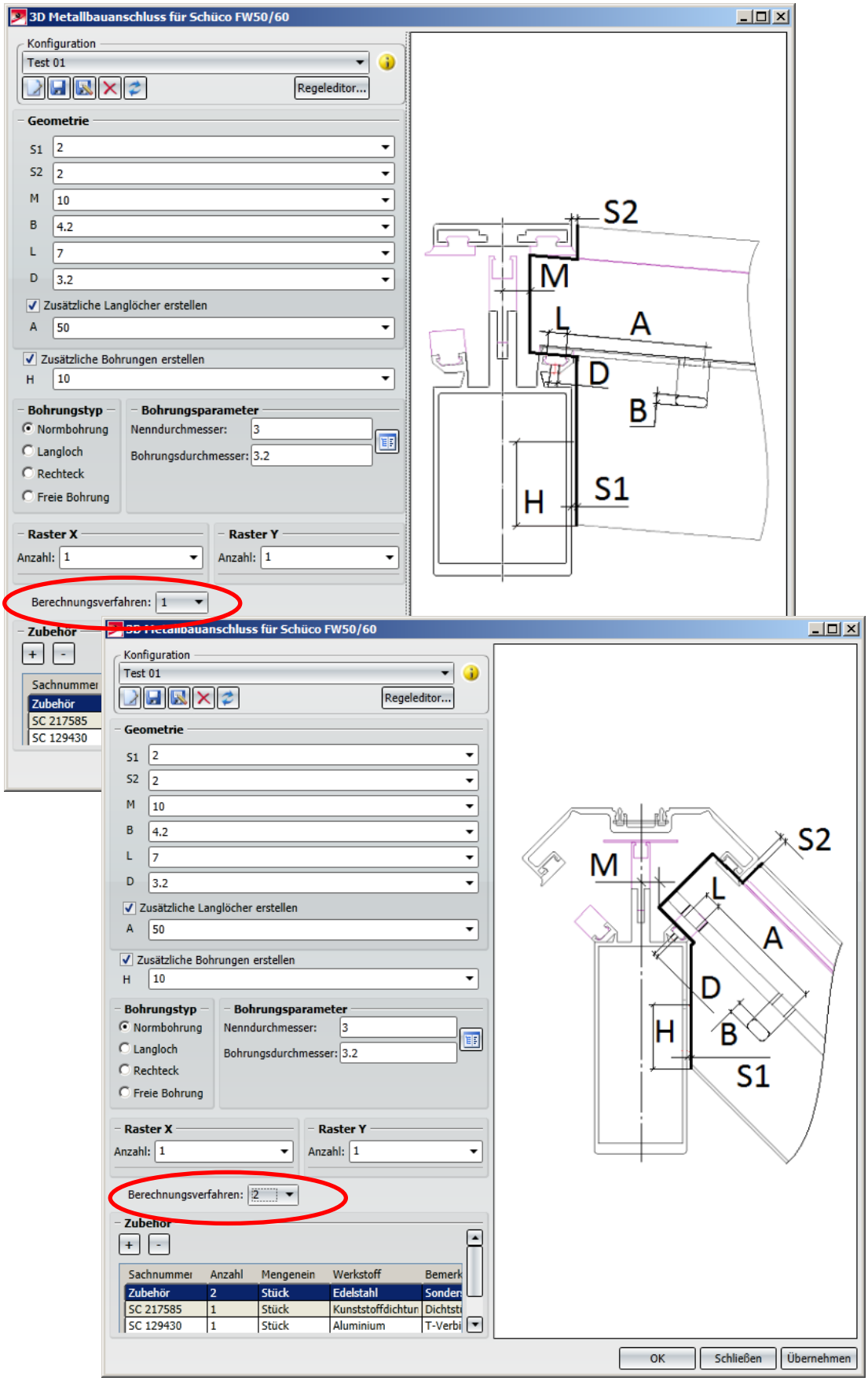
- ***6.1 Create User-Defined Beam/Profile Cross-Sections***
  - Here you need to create Points 15 and 25, in addition to the glass insertion points, in the part Mullion and Transom, respectively.
- ***7.2.1 3-D Import of Individual Beam/Profile Cross-Sections*** via LogiKal
  - Here you need to save the individual mullions and transoms (not the entire beam/profile group) together with Points 15 and 25 to the …Catalogues\Factory standards\LogiKal directory beforehand ***(→ 7.1.3 Simplified Beam/Profile Cross-Sections in the HICAD Catalogue)***.
- ***7.2.12 Facade via Metal Engineering Grid (Metal Engineering Plugin)***
  - For the placing of beams and profiles on grids you can use either user-defined beam/profile cross-sections, or beam/profile cross-sections imported from LogiKal (see previous Chapters ***6.1*** und ***7.2.1***).

For the creation of the 3-D Metal Engineering connection you can use two calculation methods:
Calculation method 1 is used for connections where the cover strips have the same width as the support profile.
Calculation method 2 is used for connections where the cover strips are bent (angle-shaped).

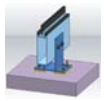


- Here you can create and save user-defined or manufacturer-specific connections.
- Here you can specify various bore types with a bore pattern.
- Definition of the bore pattern for the selected bore type.
- Definition of the calculation method (see next screenshots)
- Here you can create accessory parts that are required for the connection, and save them together with the configurations.

For insertion, first select the transom, and then select the mullion.



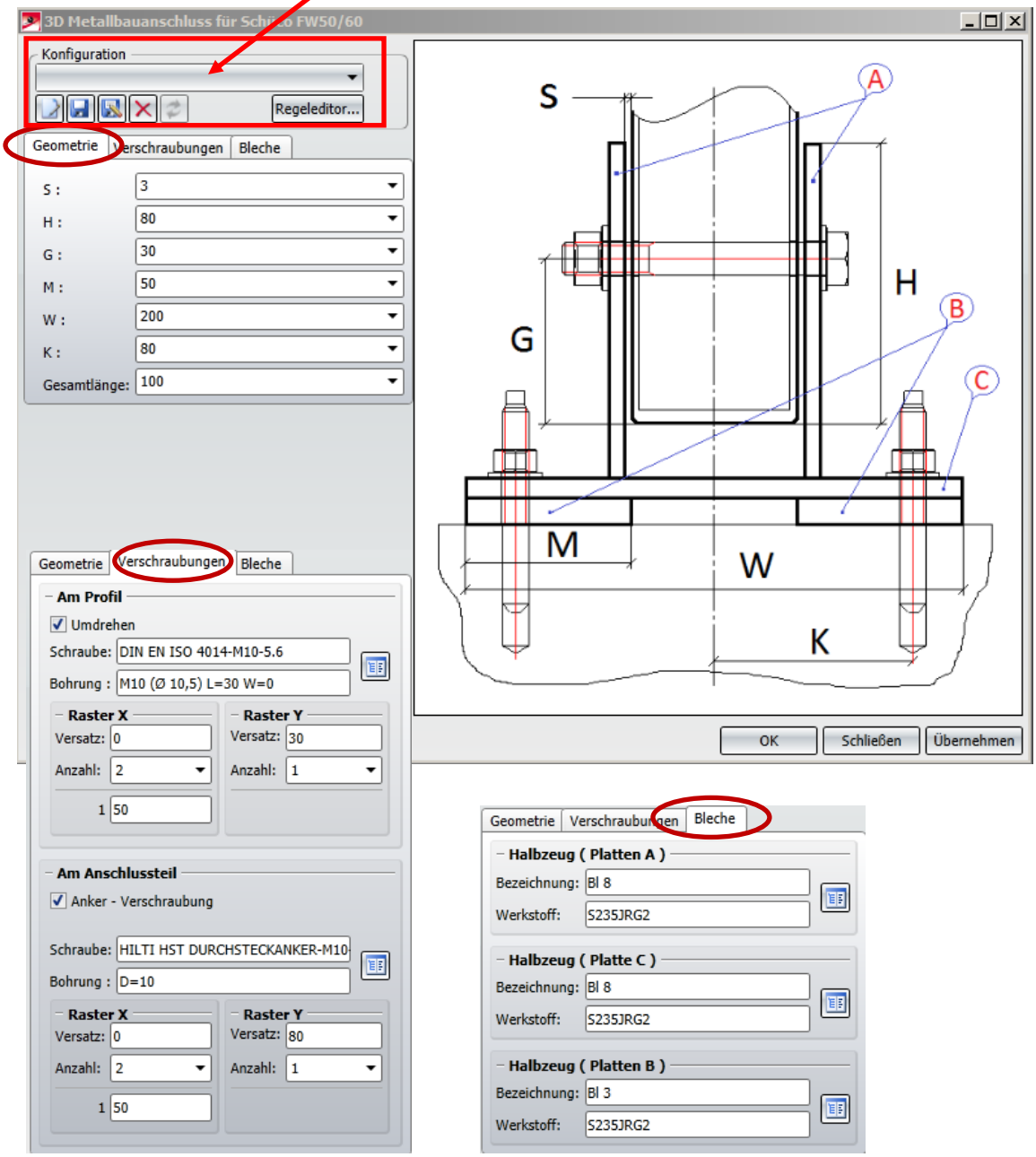
12.2 Fixing Bracket for Intermediate Ceilings



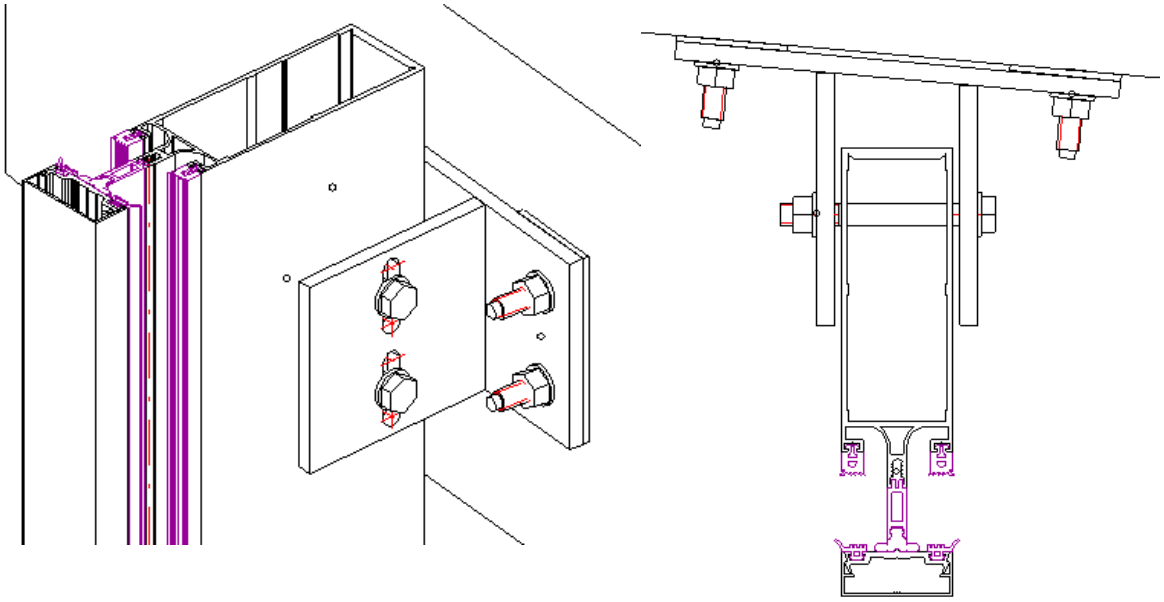
Use this Design Variant to attach facade mullions to building structures. This variant can not only be used for aluminium facade profiles, but also for hollow steel beams.

First select the mullion, then select the building structure, and finally select the insertion point. The dialogue will then be opened, enabling you to configure the bracket.

The different configurations can be saved as templates.



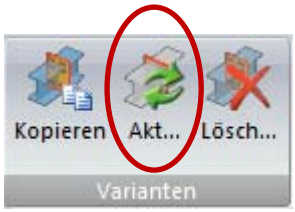
The intermediate ceiling can also be placed at a specific angle to the mullion!



Changes:

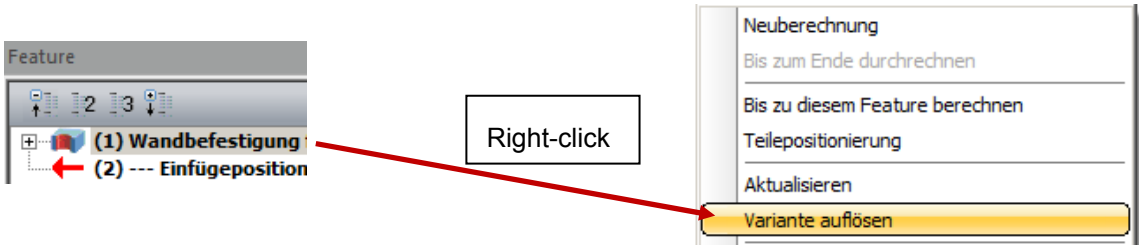
- If the width of the mullion or the position of the facade is changed subsequently, the connection needs to be recalculated via the Feature log, and all connection constellations will be automatically adjusted to the new situation.
- To change bolting parameters, such as slots, bolt sizes etc. subsequently, double-click the corresponding Feature log entry. You can then change the Feature log entry as desired.

The updating of the connection can either be performed via feature recalculation, or via the Steel Engineering module (Variants →Update).



The Feature log entry “Wall mounting” can be broken up into its individual working steps:

- Right-click Feature log entry
- Select “Break up variant”



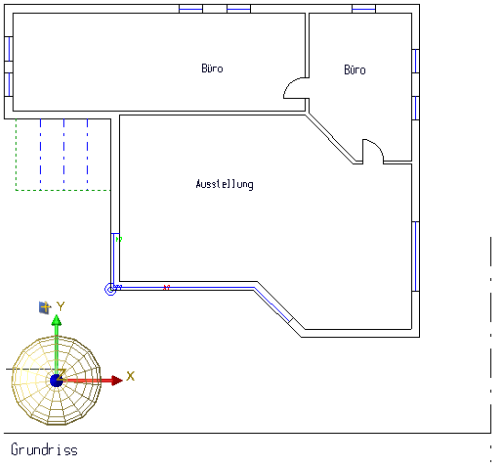
13 Exercise: Create Metal Engineering Facade Construction

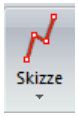
13.1 Construction of the On-Site Scenario via 2-D DWG Data

HiCAD provides the Metal Engineer with a tool that enables the representation of the on-site situation, and a production-oriented integration of Metal Engineering constructions into the displayed scenario.

1) In the present example an imported 2-D dxf/dwg file is transferred to a 3-D CAD drawing. To do this, first load the file **BUILDING2010.DWG** from the "Szenen" sub-directory of your HiCAD installation.

   ➢ Open the Drawing tab and select → New/Open → Open → Interfaces, DXF/DWG...



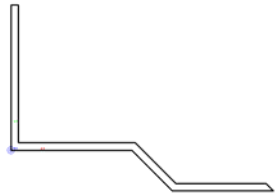


2) To construct the wall, first switch to the top view according to the World Coordinate System. Create a sketch (the processing plane will be placed in the screen plane).

3) Use the **Copy 2-D polylines** function to copy the 2-D lines to the 3-D sketch.

   ➢ Activate the Sketch tab and select Derive → Copy → Copy 2-D Polyline.

   Now process the sketch in such a way that a closed polyline will be the result (see screenshot).

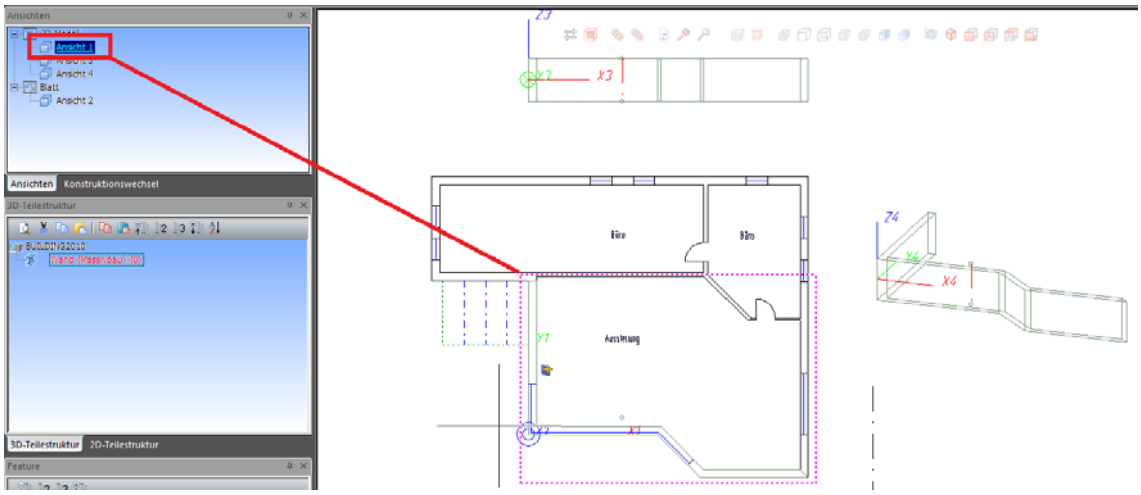


4) From this sketch, create a Wall (masonry) with a height of 2000 mm, using the **Solid Construction** catalogue (Metal Engineering → Part type → Solid construction).

5) Create two further views:

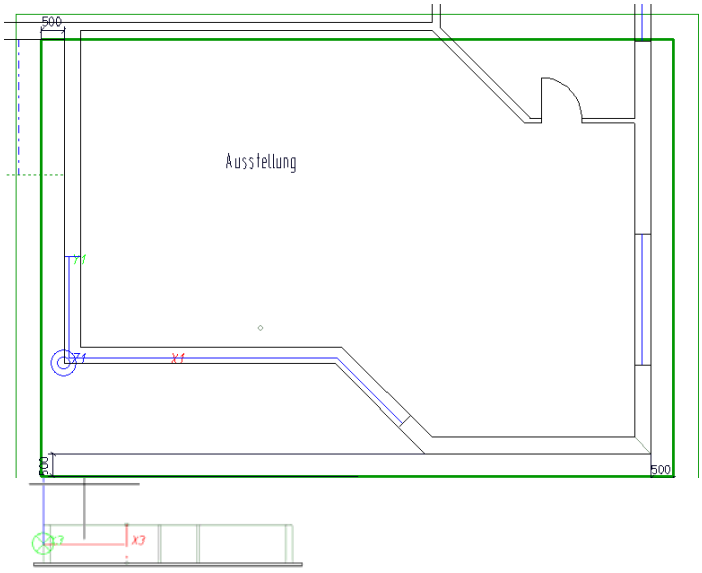
   - A linked front view (right-click on pink dotted view frame, or open Views tab)
   - A axonometric view (right-click on pink dotted view frame, or open Views tab)

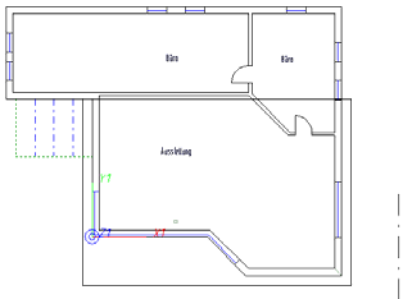
When working with several views you always need to activate the view that you want to process. To do this, click the view frame of the desired view. The active view is indicated in the Views tab of the ICN.

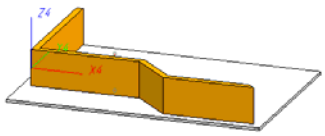


6) Creation of the base plate

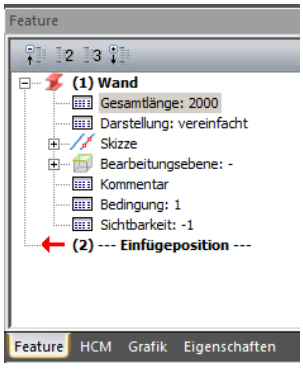
- Activate top view.
- Create sketch and use the sketching options to draw a rectangle for the base plate. For an exact positioning of the sketch you can also use HCM options.
- Right-click sketch and create extruded part with a height of **-150 mm**.
- The base plate will be automatically represented in all views.
- Familiarize yourself with the different representations of views, such as Hidden Line, Shaded with edges etc... .





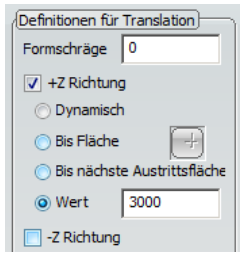


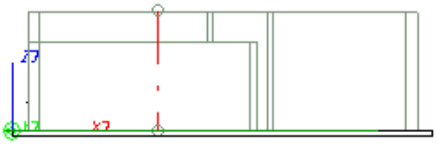
7) In the Feature log of the “Wall”, double-click the “Total length” entry to change the height to **4000 mm**.

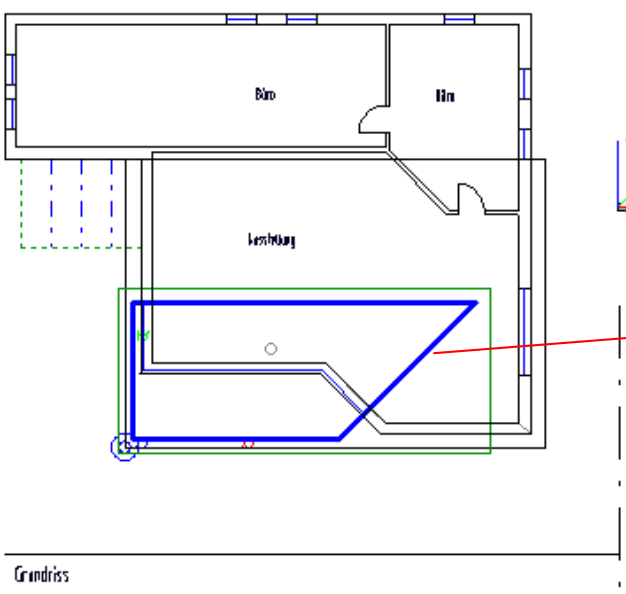


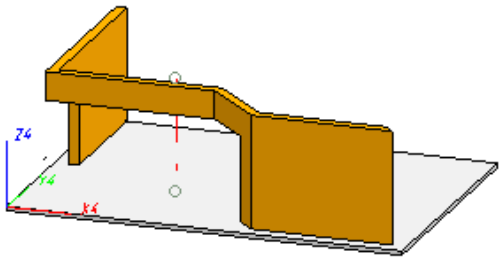
8) Apply subtraction via translation to facade

- Activate top view
- Create sketch and place processing plane on base plate. Use the sketching options to create a material subtraction via translation. The subtraction must be made by means of a closed polyline.
- Activate the part “Wall” (Solid construction).
- Open the 3-D Standard tab, select the “Subtract with translation + depth” function (“Process with sketch” function group), and apply the previously created sketch (subtraction contour). For the depth specification, set the value in **+Z direction** to **3000 mm** and confirm with **Subtract**.



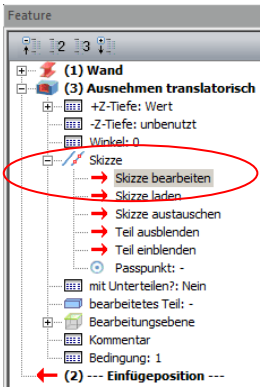




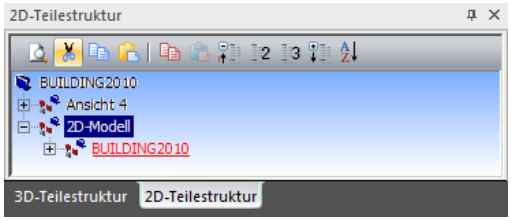


Subtraction contour

9) The subtraction contour can be changed via the Feature log entry “Sketch, Process” at any time. You can also use HCM options here.



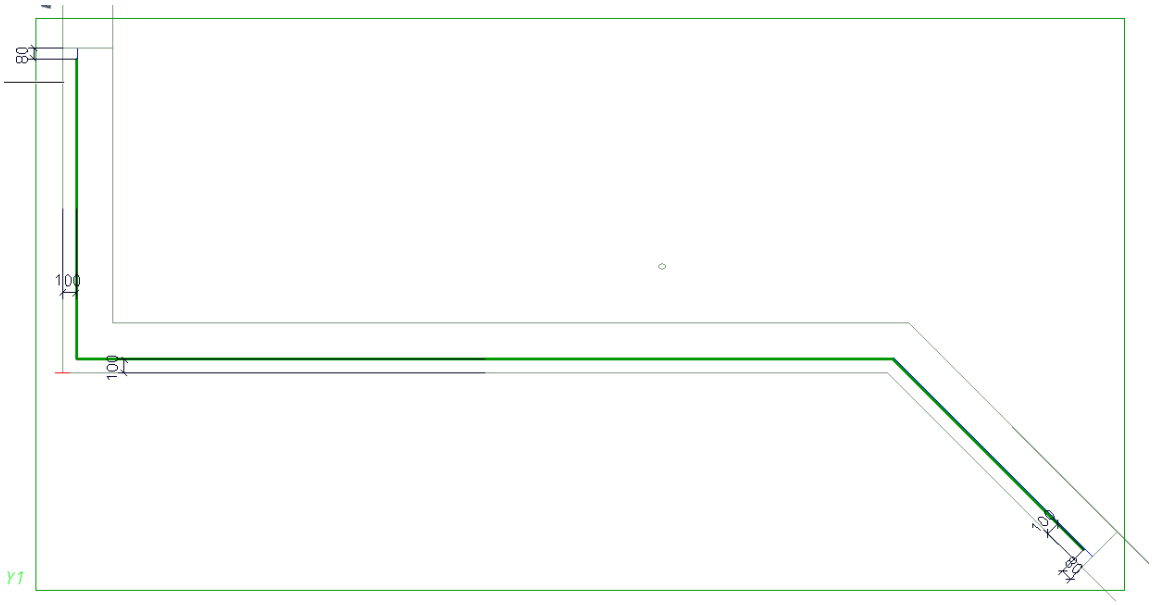
10) In the 2-D Part structure, subordinate the part “BUILDING2010” to the view “2-D Model”. In this way you achieve that the 2-D drawing will only be displayed in the “3-D Model, View” area, but not in the Sheet areas.



13.2 First Draft of the Facade

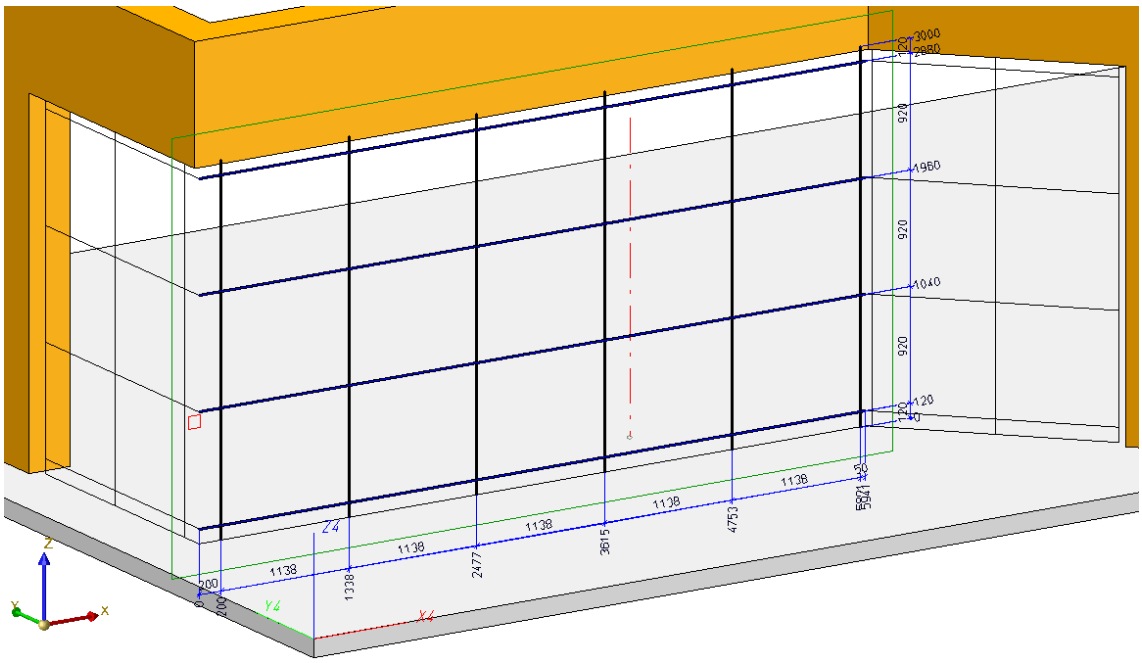
11) Draw the route of the facade with the help of a sketch. Assume that the facade route is based on the outer edge of the glass.

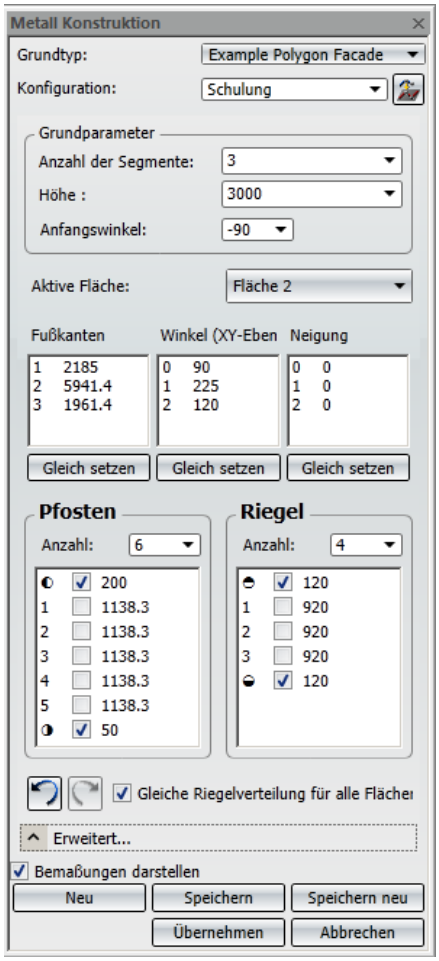
- Activate the top view.
- You create the sketch with a processing plane placed on the base plate, and use the sketching options to draw the route of the facade. You can also use HCM options here.



12) After drawing the route of the facade, use the Metal Construction plugin to create a facade planning grid. Proceed as described in Chapter **8 Facade via Metal Engineering Grid (Metal Construction Plugin)**.

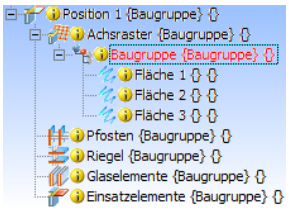
- Take the dimensions of the facade from the screenshots below:





13) After applying the created grid to you drawing, create a grid assembly.

➢ Activate the Metal Engineering tab and select Grid → Grid assembly, new.

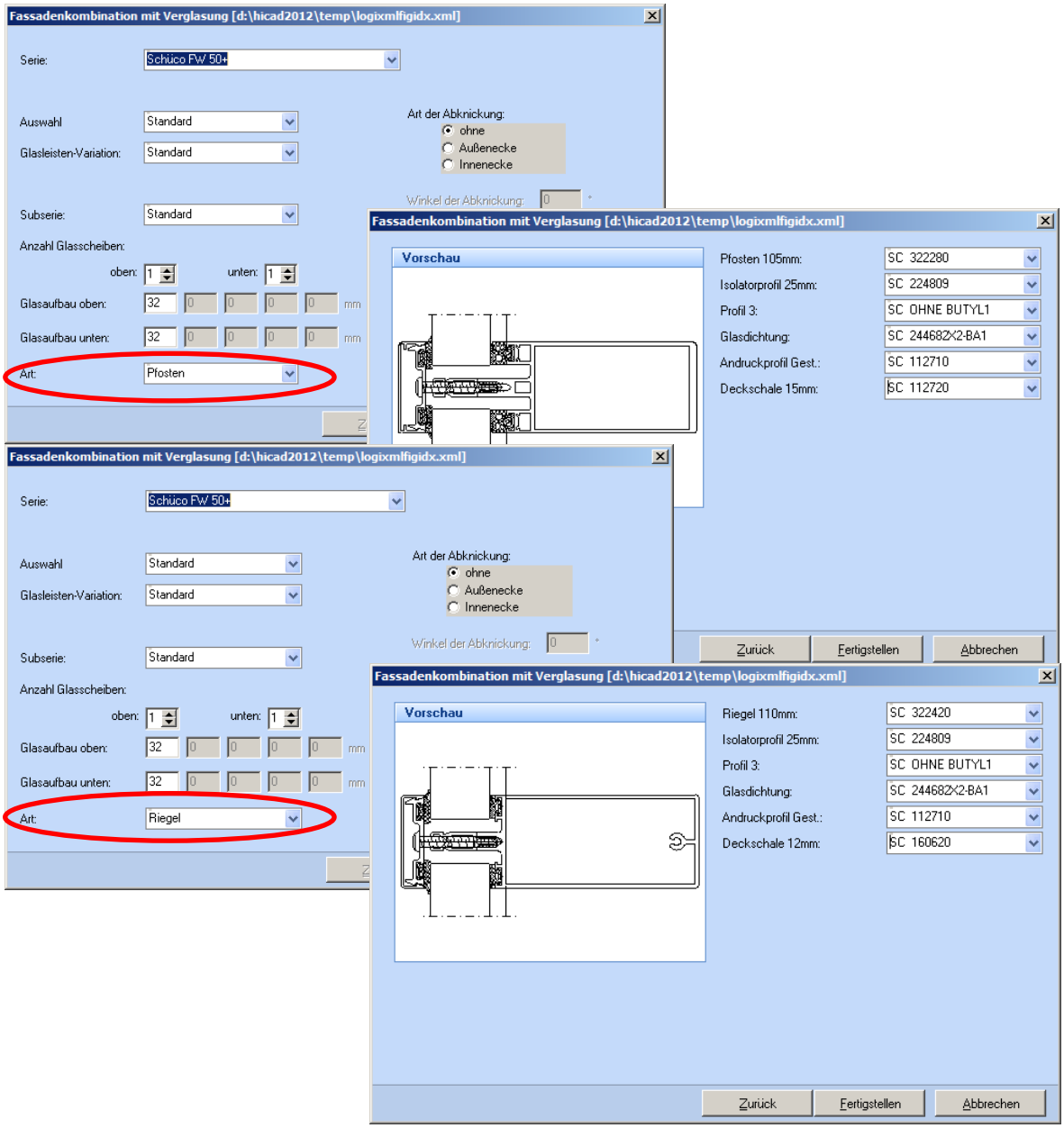


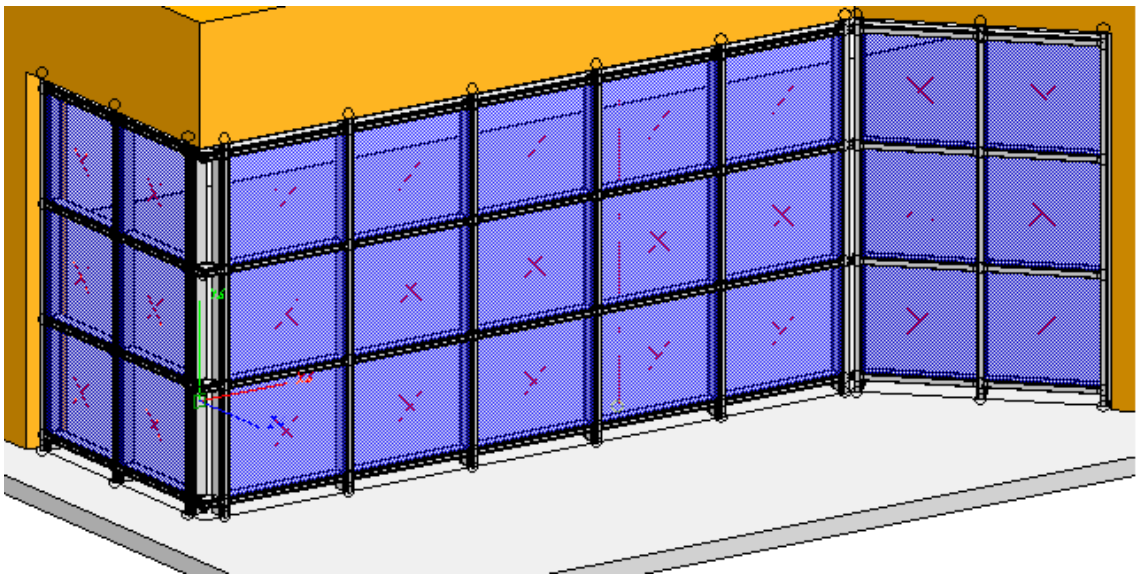
- Then subordinate the sketches of the grid to the Planning grid assembly.

14) <u>For the placing of parts on the grid, proceed as described in the following chapters:</u>

- For a placing of beams/profiles on the grid via **LogiKal**, please read the information given in Chapter 7.2.12 Place Beams/Profiles on Grid via LogiKal. Please also consider the beam/profile insertion points described in Chapter **7.1.2 Metal Engineering Settings**.
- For a placing of beams/profiles on the grid via **user-defined beam/profile cross-sections**, please read the information given in Chapter 6.2.4 Insertion on Grid via User-Defined Cross-Sections. Please also consider the beam/profile insertion points described in Chapter **6.1.8 Define Insertion Axis**.

- For the placing of parts on the grid via LogiKal, you can use the following beam/profile combinations:



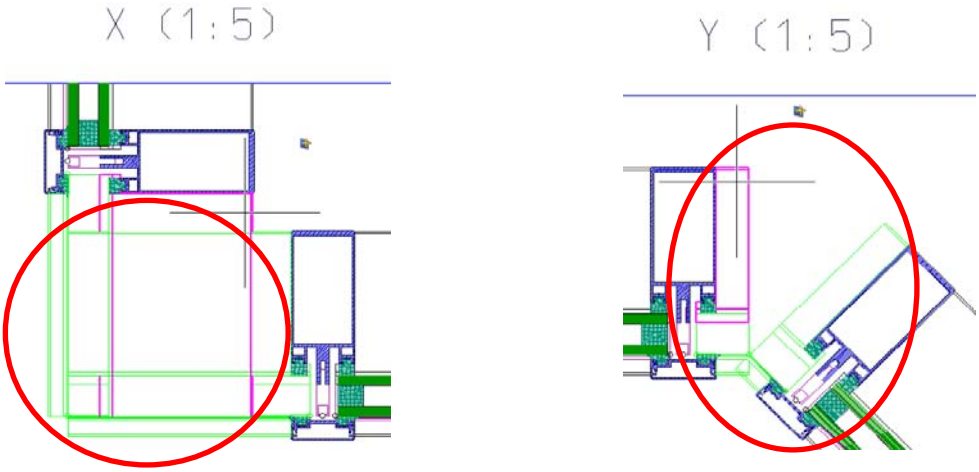


13.3 Create Drawing for Approval

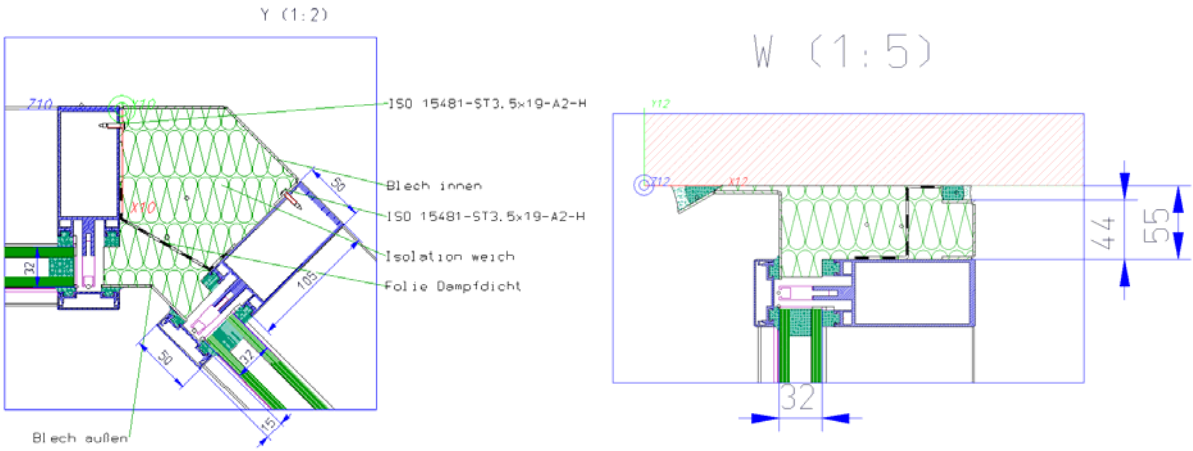
15) For the creation of a drawing for approval, switch to the **Sheet** area, insert a DIN-A1 drawing frame, change the **Main scale 1:25** and create several **Sectional and detail views**.

➢ Open the Views tab and select New → New sectional view / New detail view.

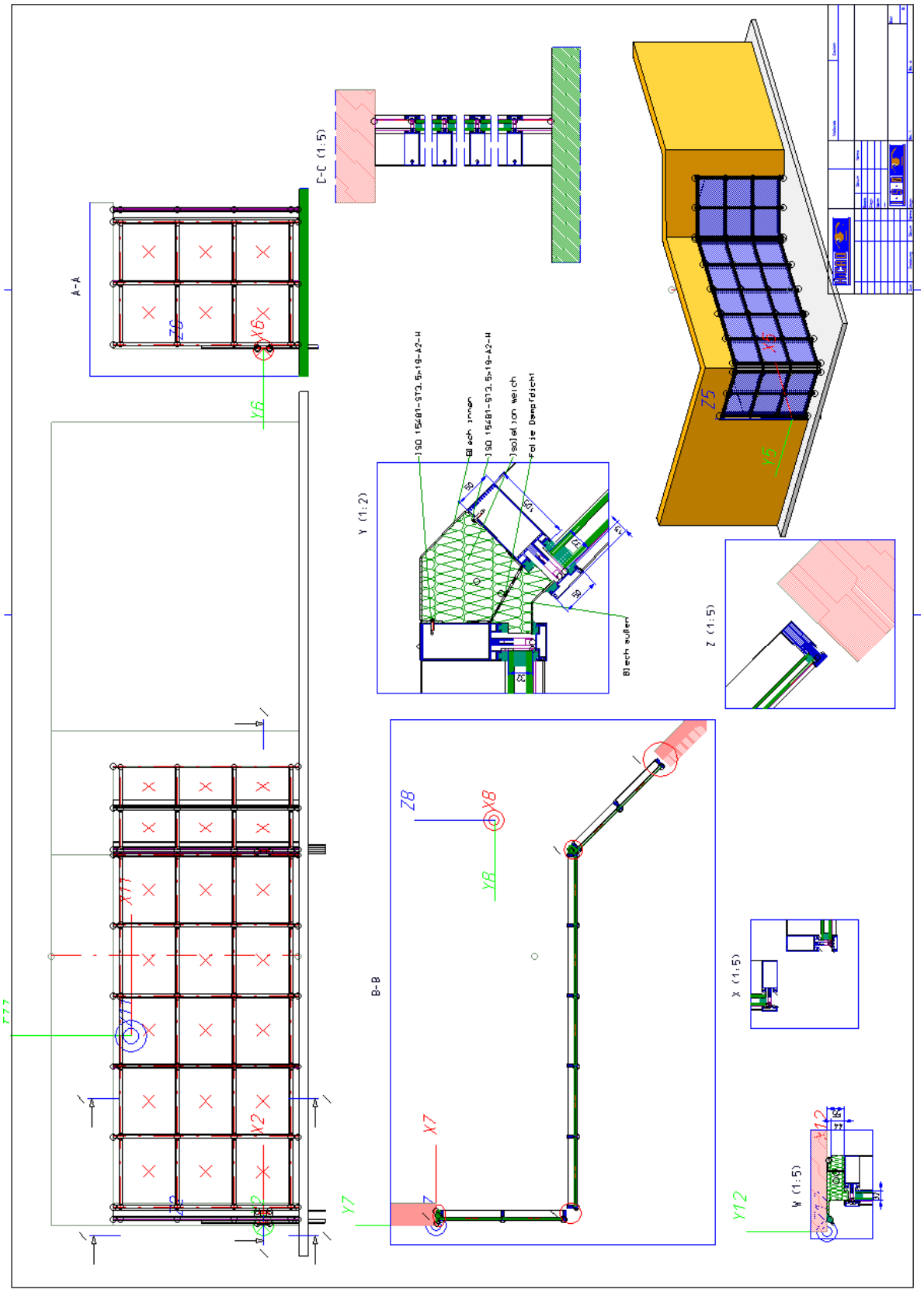
- Change the view scale of the sectional and detail views to 1:5, the axonometric view to 1:50.
- Switch the view representation to Hidden Line
- In the detail views **X** and **Y,** the superfluous profiles can be deleted. These superfluous profiles were created during the placing of elements onto the grid.



- When deleting the profiles, please make sure that the Group snap option is active. (see Chapter **5.1.5 "Group Snap" Mode**.
- Now proceed as described in Chapter **11 Create Wall Connections (3-D)** to construct the wall connections with plates, foils, insulations, sealings etc. …

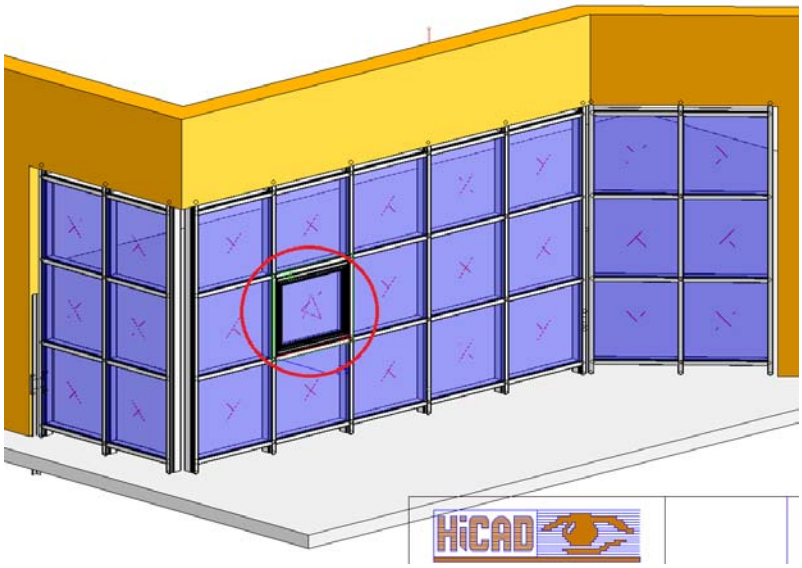


- Then, dimension and annotate the drawing for approval.


C-C (1:5)
A-A
Y (1:2)
Z (1:5)
X (1:5)
W (1:5)
B-B
Blech außen

13.4 Fit Window Insert into Facade

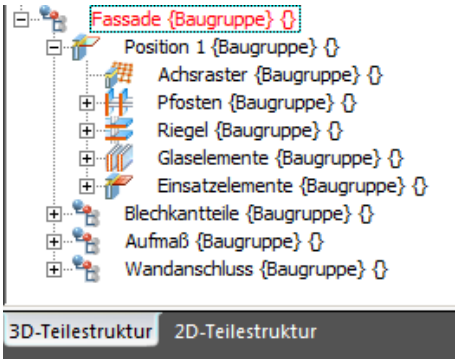
- Proceed as described in Chapter 7.2.9 Change Glass in Insert.



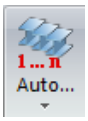
13.5 ICN Structure

Before itemisation and workshop drawing creation, the ICN structure needs to be arranged appropriately. The screenshot on the right shows an example of such a structure.

- To create a new assembly, open the **3-D Standard** tab and select **New → Assembly**.
- You can move the 3-D parts in the 3-D Part structure via Drag & Drop.

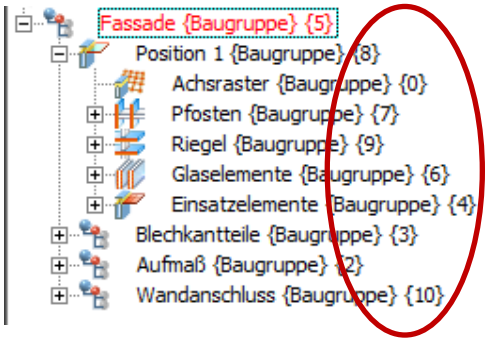


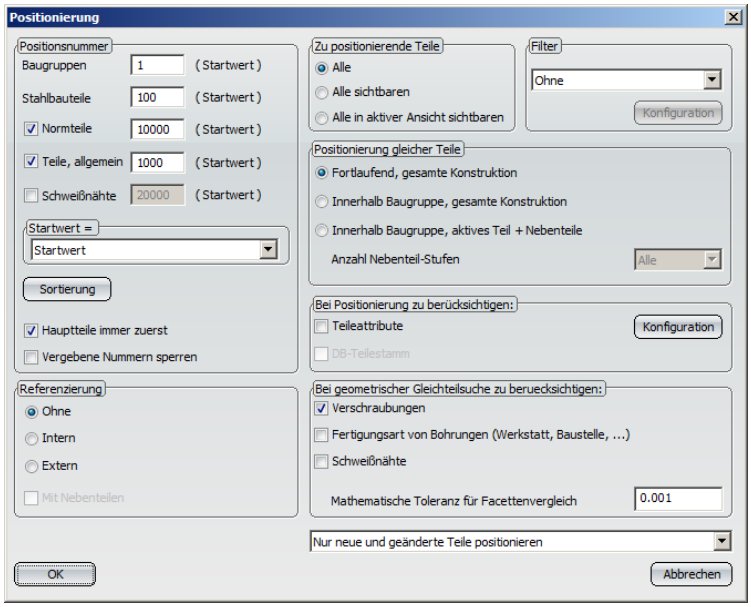
13.6 Itemisation / Workshop Drawings



To be able to create a workshop drawing you need to itemize your CAD drawing beforehand. To do this, open the **Drawing** tab and select **Itemisation/Detailing → Itemisation.**

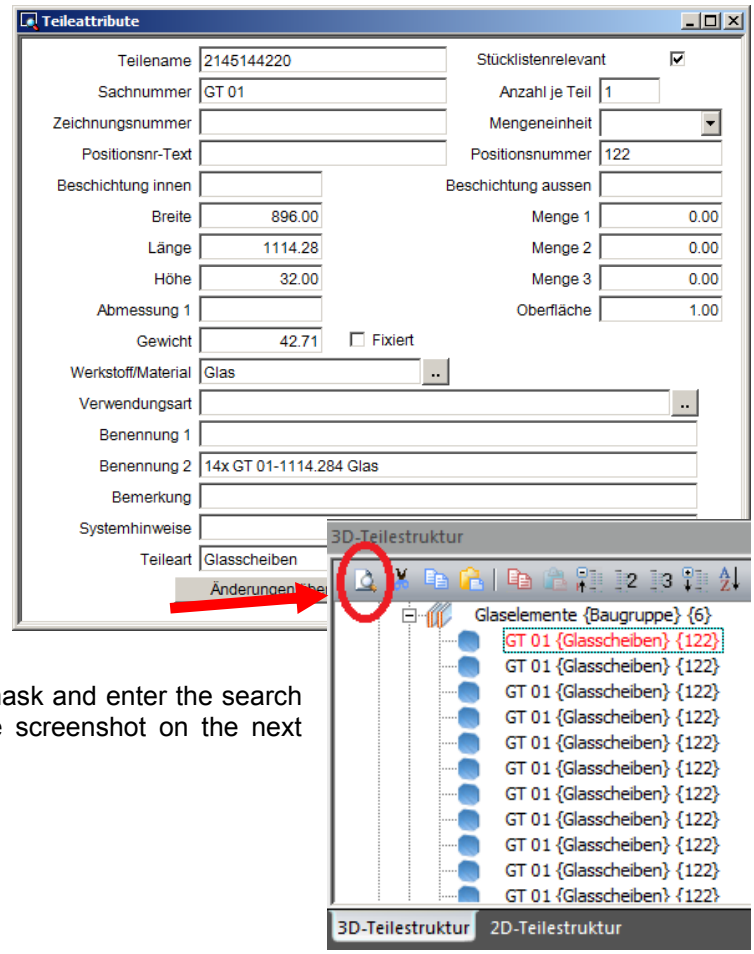
- During itemisation, an automatic (geometrically) identical part search takes place.
- The item numbers will be written after the parts and assemblies in the ICN structure.
- Choose **Drawing → Itemisation/Detailing → Auto → ...with options** to open the Itemisation dialogue window.
- Here you can specify the desired settings for itemisation.

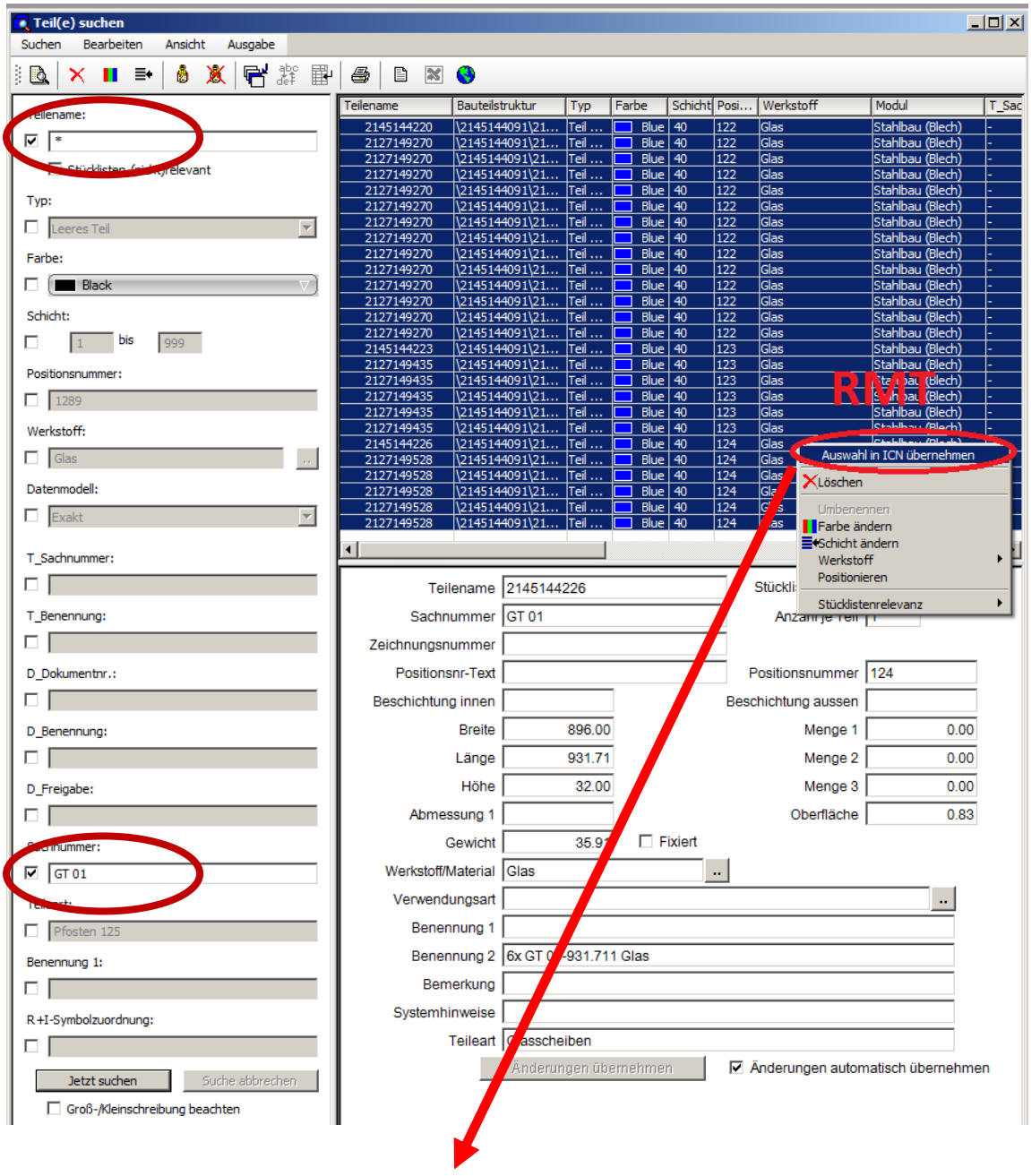




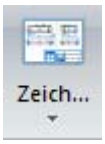
Create plan of model glass panes:

- First step: Open the Part attributes dialogue window of the glass pane (click the cross symbol on the glass pane or right-click the part in the ICN or in the drawing area and select Properties → Part attributes.
- The Part attributes window provides information on the active part. This information can also be evaluated in the Bill of Materials (BOM).
- The **Article number GT 01** is to be used as a filter criterion.
- Then, open the search mask and enter the search criteria, as shown in the screenshot on the next page.



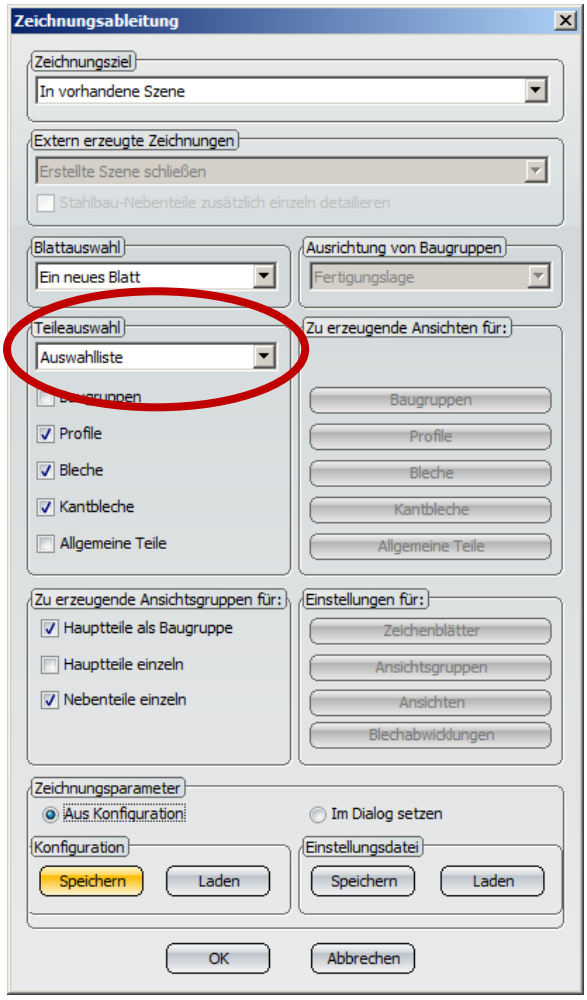


- Select “Take over selection in ICN” to create a selection list in HiCAD (this selection list can also be created manually in the ICN).



- For the active selection list, select the **Drawing derivation** (Drawing → Itemization/Detailing → Drawing derivation).

- Fill in the dialogue for drawing derivation as follows:

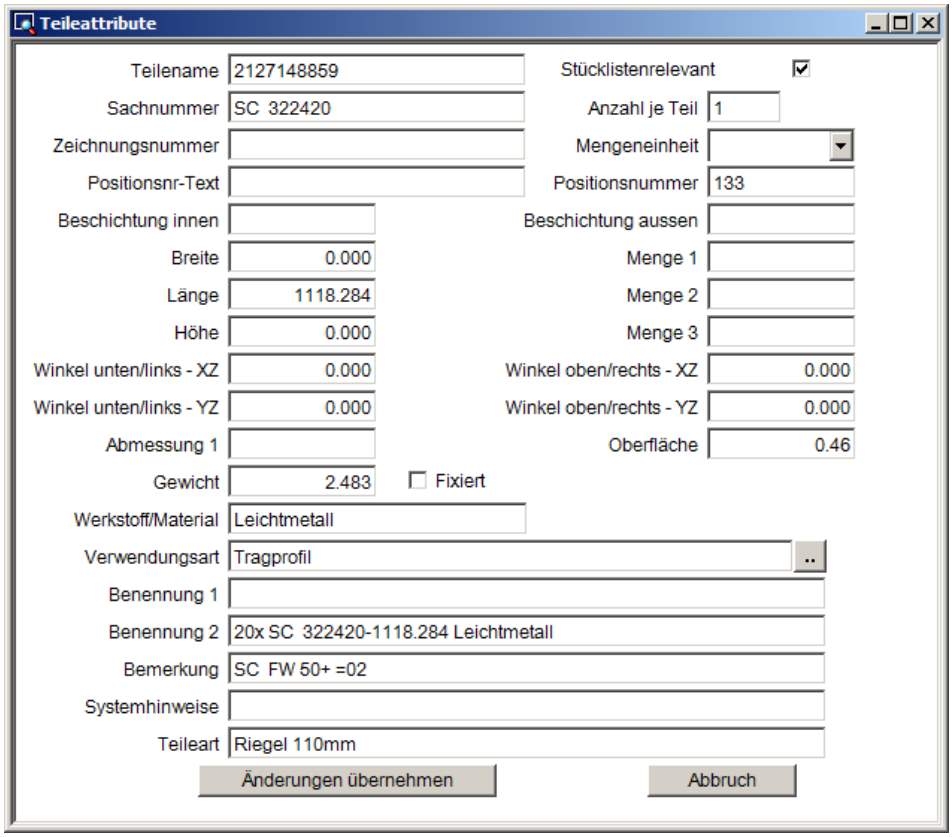


- On the Views tab of the ICN, create a second Sheet area containing the model glass panes.

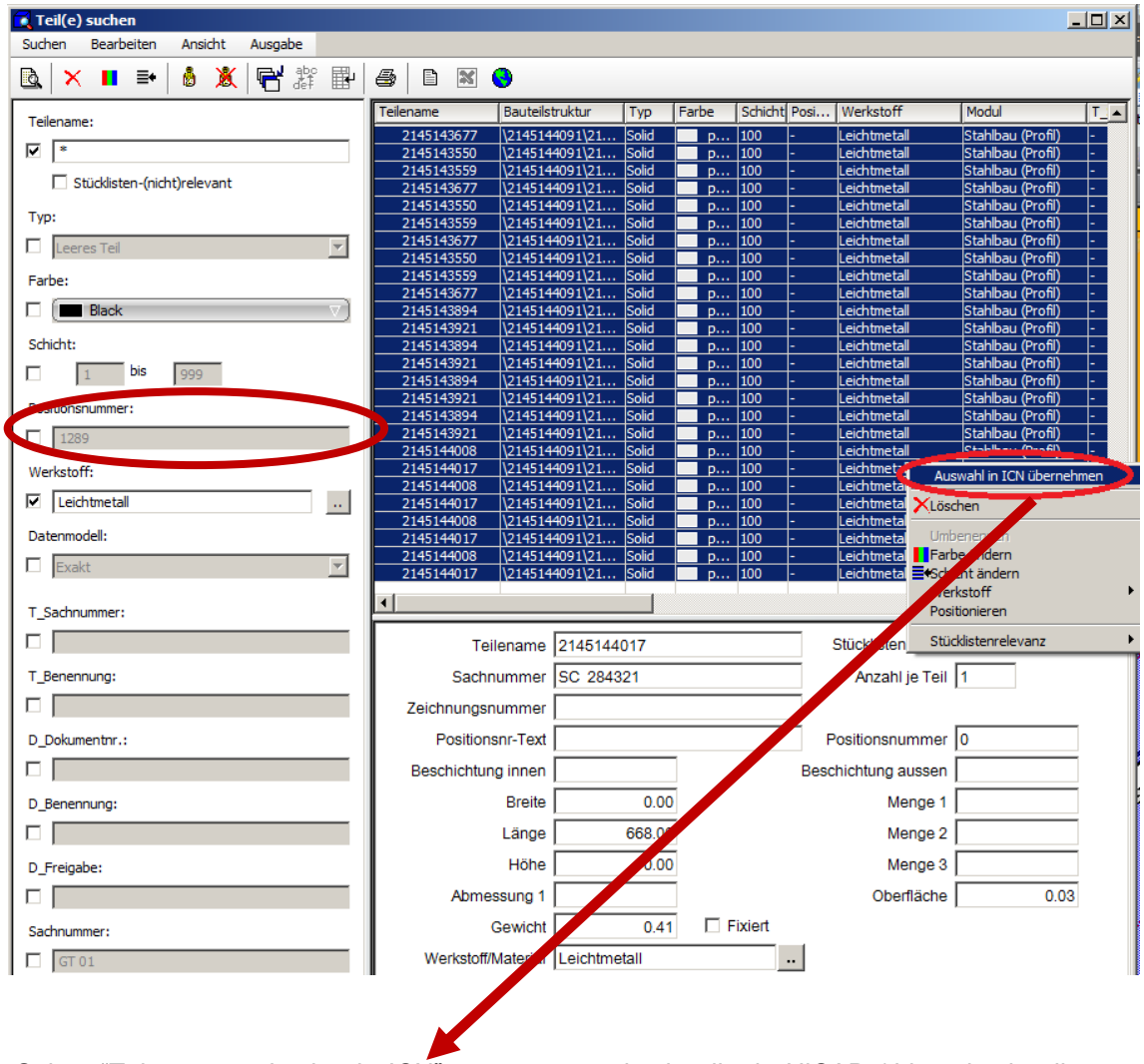
- To insert the BOM for the glass Sheet area, proceed as follows: Right-click on the **Glass elements** assembly, and select **Bill of Materials, for active part** function. In the **Report Manager Configuration** area, pick the “Glass” option from the list box, and select **Output to HiCAD**.

Create production drawing for the transoms:

- The procedure is the same as for the creation of the model glass panes.
- First, open the Part attributes window of the transom.

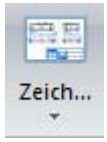


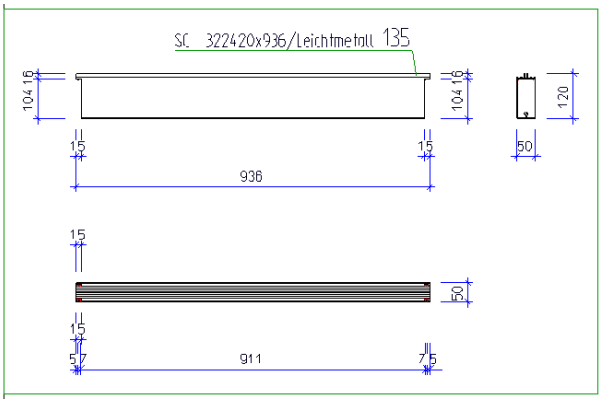
- You can filter by Article number or Part type (as for the model glass panes). This procedure enables you to create only the production documents for the transoms in the next step. The same procedure can be applied to the mullions etc.
- In our example we will filter by Material, so that all light metal parts will be created in one Sheet area.
- Open the HiCAD search mask and fill in the search mask as follows:



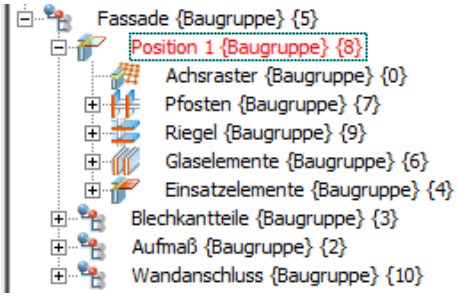
Select "Take over selection in ICN" to create a selection list in HiCAD (this selection list can also be created manually in the ICN)

- For the active selection list, select the **Drawing derivation** (Drawing → Itemization/Detailing → Drawing derivation).
- To apply the settings for the production drawings, proceed in the same way as for the model glass panes. After this, detail drawings will be created on Sheet 3 for all light metal parts.





- To insert the BOM, mark the assembly “Position 1”, create a BOM of the active part, and import it to Sheet area 3.



13.7 Transfer to LogiKal (only available in conjunction with the LogiKal interface)

For the transfer of elements to LogiKal please read the information given in the following chapters:

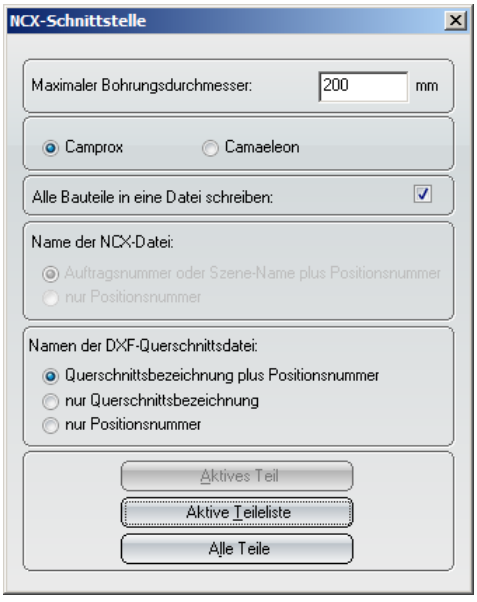
- **7.2.4 Transfer of Beams/Profiles to LogiKal**
- **7.2.10 Export Facade / Insert to LogiKal (3-D)**

13.8 Machine Control (Profile Machining Centre PBZ)

Machine control can take place in two ways:

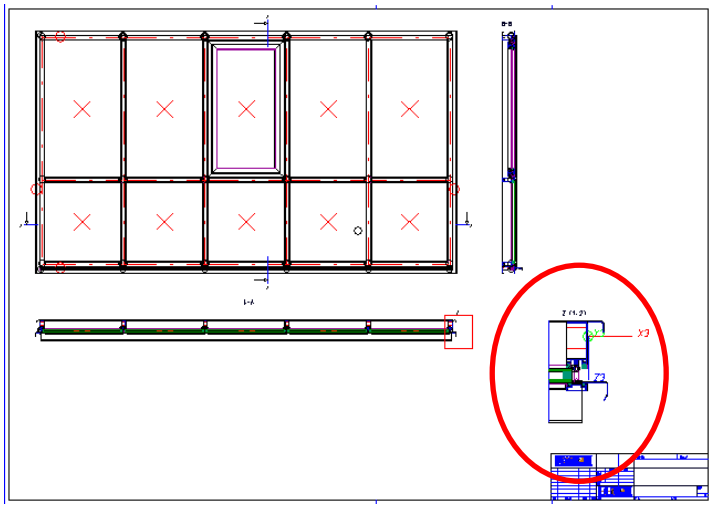
- For machine control via **LogiKal** the reading of the following chapters is recommended:
  - 7.2.4 Transfer of Beam/Profiles to LogiKal
  - 7.2.10 Export Facade / Insert to LogiKal (3D)

- **HiCAD NCX Export** file. Please note that the drawing needs to be itemized for this procedure.
  - Save as → **NCX**



14 Exercise: Section and Detail Planning

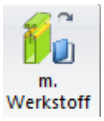
For this exercise, please load the HiCAD drawing **Glass_Facade.SZA** in the "Szenen" directory of the HiCAD installation. This example will show you how easily a detail planning of the wall connection for the approval drawing can be performed.



Please also note the information given in Chapter ***11 Create Wall Connections (3-D)*** !

In this exercise we want to create the wall connection in the detail view **Z (1:2)**. Proceed as follows:

- Activate the detail view in the ICN structure.
- Select **Metal Eng., 3-D Section → Part with Material**.

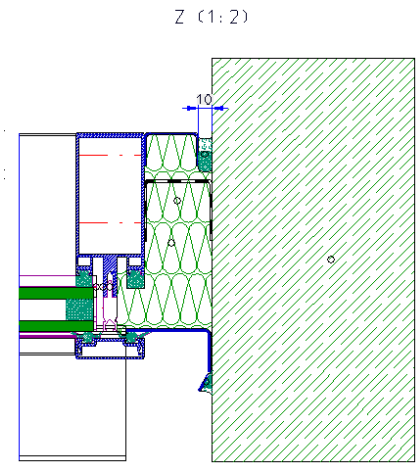


- Fitting mode: Part from rectangle, Part type: Concrete, reinforced → OK.
- Sketch a rectangle via 2 diagonally opposite points. Change the exact dimensions via the Feature log entry "Sketch, Process". The sketches can be conveniently modified via the C-Edge-HCM.
- Select **Metal Eng., 3-D Section → Sealing**.

- Choose the desired options, such as Steam-tight, In-house production etc. → OK.
- For insertion it will be sufficient to identify two edges in the sectional/detail view. The edges do not need to run parallel. The sealing cord will be created automatically.
- The insertion of the Part type "Insulation, soft" also takes place via the **Part with Material** option.

- Fitting mode: Part via contour search, Tolerance: 2, Part type: Insulation, soft.
- Left-click the area. Contour + 3-D part will then be created automatically.
- Before creating a **3-D Foil** we have to draw a sketch indicating the route of the foil.

- After selecting the **Foil** option it will therefore be sufficient to identify the polyline of the previously created sketch. Before inserting the foil, you will be asked to specify the insertion direction of the foil (i.e. the direction of the foil thickness).



- After updating the sectional view, the correct hatchings will be displayed.
- In the other views the parts are hidden.
- After showing the parts in the front view, they can be processed further with Steel Engineering functions, such as Trim, Change length etc. …
- For part annotation, all set parameters will be read out automatically.

15 Exercise: Staircase with Glass Canopy

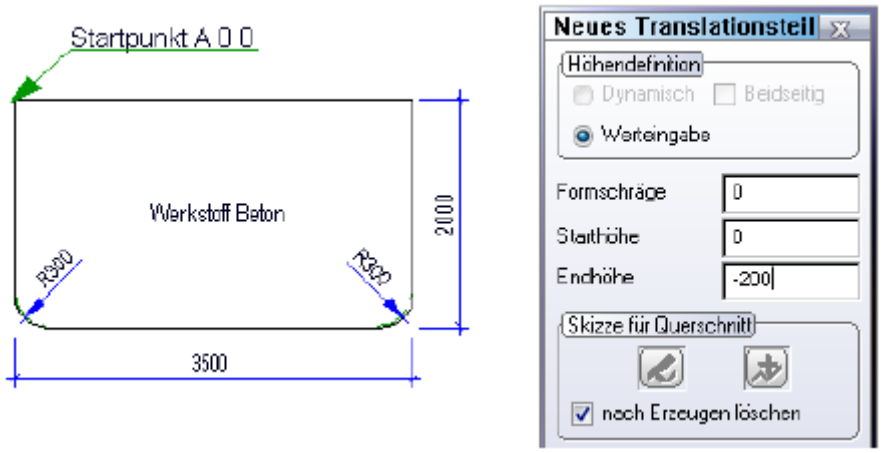
15.1 Learning targets

- Create a 3-D CAD drawing of the geometries existing on-site
- Insert beams and profiles in the drawing via the Catalogue
- Familiarise yourself with the different insertion options
- Place beams and profiles in certain positions to each other and duplicate them
- Process beams and profiles and bolt them together
- Create and process Steel Engineering plates and glass panes
- Store and use purchased parts in the Catalogue
- Create documents for customers, architects, production

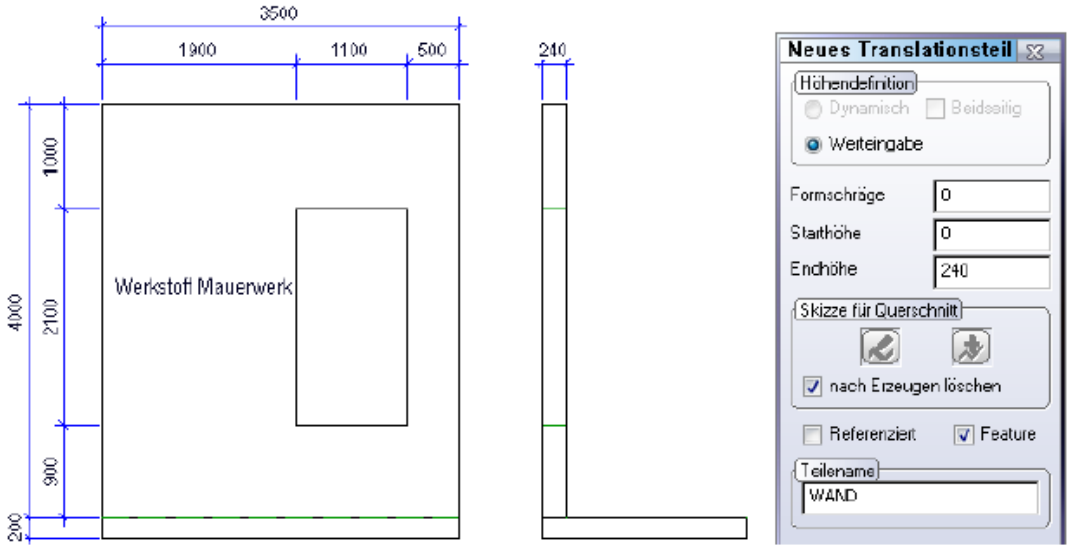
15.2 Create the Existing On-Site Situation

First, use extruded solids to construct the existing on-site scenario.

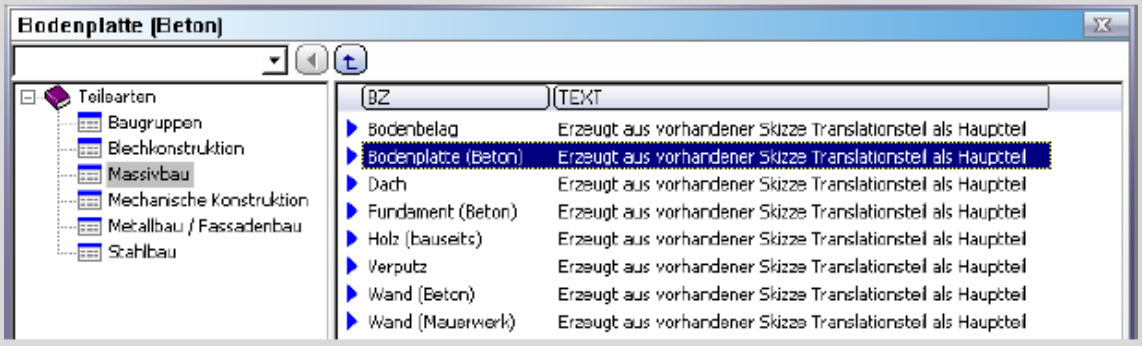
- Create the sketch of the base plate in the XY-plane of the World Coordinate System.



- Create the sketch form the wall in the XZ-plane of the World Coordinate System.

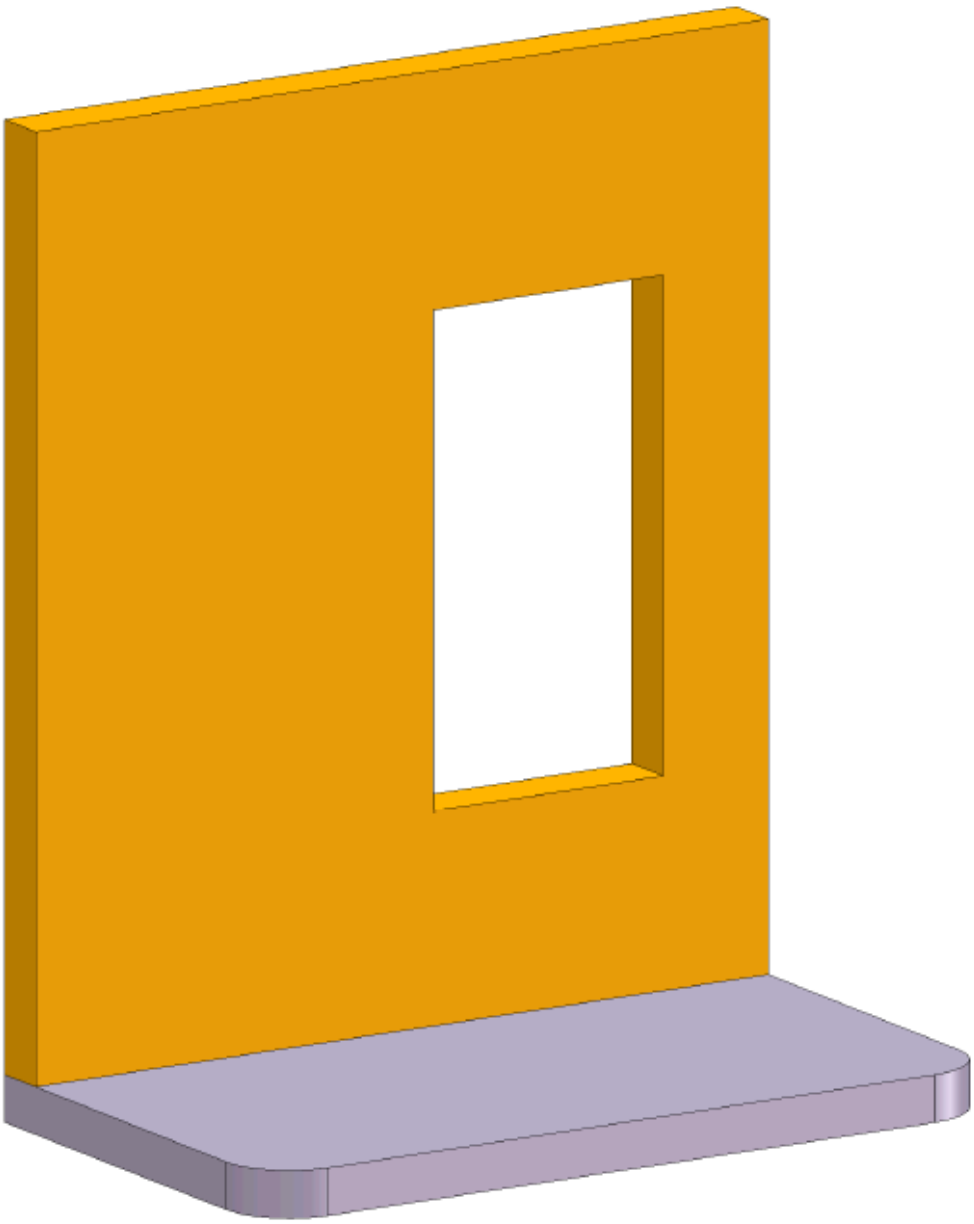


HiCAD offers the option to change the part type of extruded parts via the part type catalogue. Open the **Metal Engineering** tab and select **Process, LogiKal 3-D → Change part type**.
The procedure assigns a colour and a material to the parts. You can adjust these settings to your individual requirements in the Catalogue Editor, and modify them if desired. This also ensures that the correct hatching code will be used in the (later created) sectional views of the on-site elements.



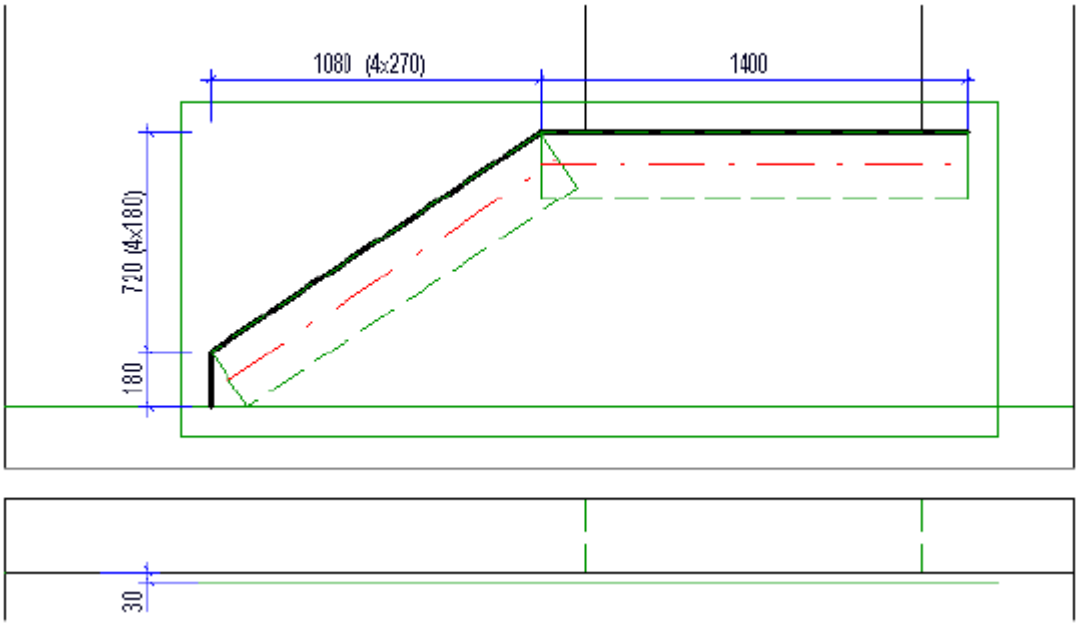
Use the HCM constraints for the determination of door cut-outs, in order to be able to conveniently apply later, more precise dimension specifications to your drawing if required.

15.3 On-Site Structures

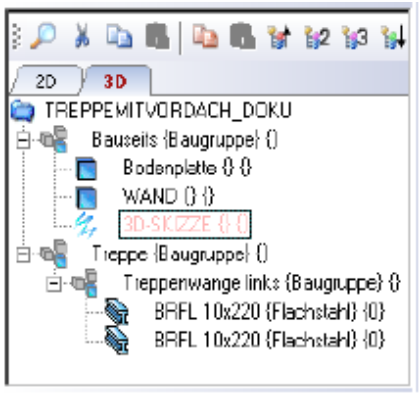


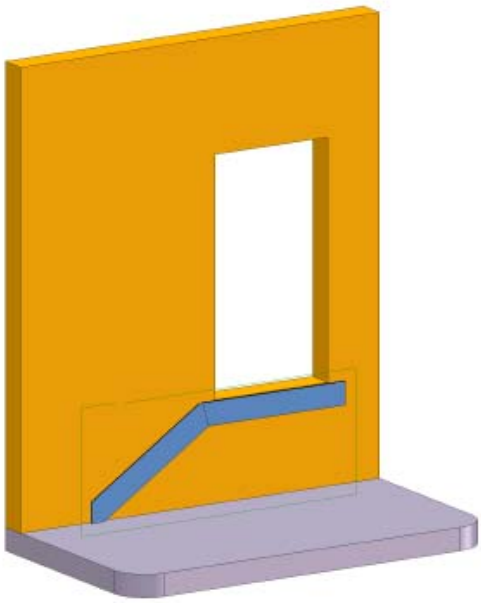
15.4 Staircase

For the staircase it is recommended to draw the contour, i.e. the division of the steps, by means of a sketch at the location of insertion. Place a sketch on the wall and move it 30 mm away from the wall. Draw the contour of the staircase into this sketch.



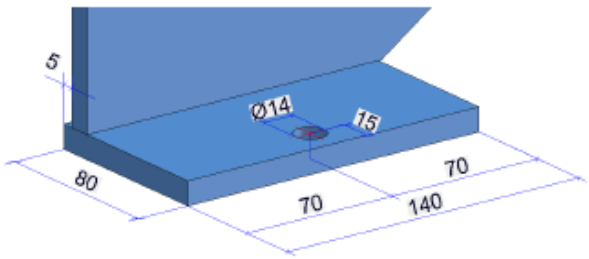
- Place the wide flat steel 10x220 in **3-D** via the end points of the sketch. Switch the fitting position to **Top axis** and rotate the profile accordingly.
- **Mitre** the wide flat steel at its joint, and trim it at its start using the **Trim to edge** function (the result can be seen in the right screenshot below).



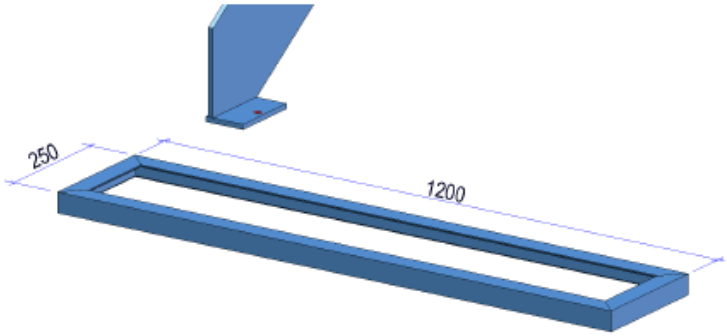


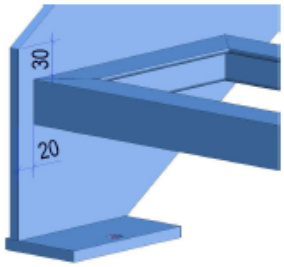
It is recommended to structure your parts into assemblies right from the start. This will enhance the clearness and facilitate the identification of parts. Parts can be conveniently moved via Drag & Drop, or by cutting and pasting them in the structure browser.

- Use a flat steel 80x10 with a bore D14 for the base plate of the stair stringer.



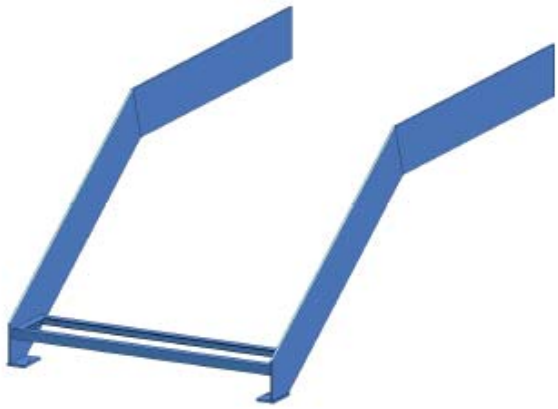
- Use the **Trim beam, to surface** function to cut the stringer to the base plate.
- An angled steel frame made of 40x5 L-profiles forms the sub-construction for the steps. Create an assembly “Step”. You can initially construct the frame freely in space, and then move the “Step” assembly it to the desired position with the **Transform →**



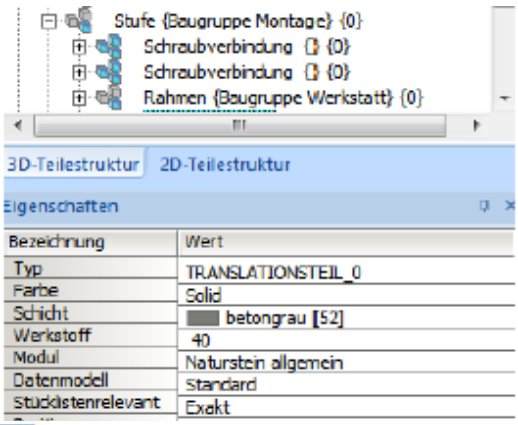


Move, 2 points function.

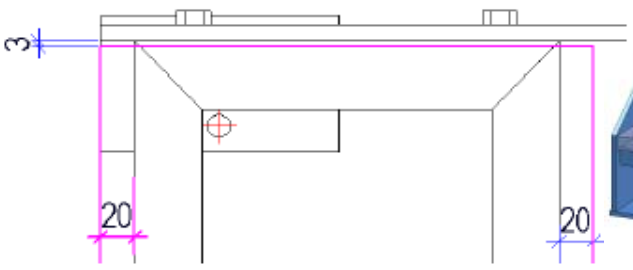
- Now copy the “Stair stringer left” assembly to obtain the other side of the staircase.
- Rotate the base plate using the function **Transform → Rotate part, via 2-D point** in the top view of the glass model.
- Change the designation of the “Stair stringer, left” to “Stair stringer, right”.

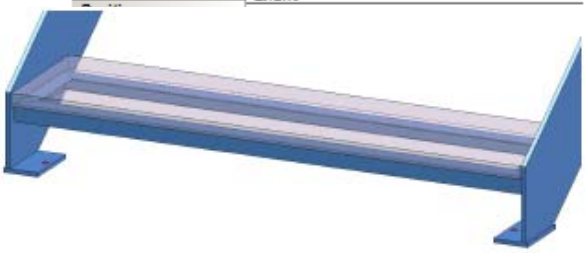


- Now create the actual step in the assembly “Step” as an extruded solid (Height: 30 mm) that is placed directly on the angled steel frame. Define the geometry of the sketch via the HCM (20 mm projection, 3 mm distance from stair stringer). Assign the desired material and an appropriate designation.



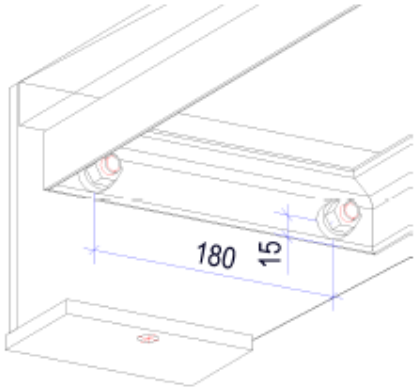
The transparency of the step is obtained by assigning Layer 40 to it (via Properties).



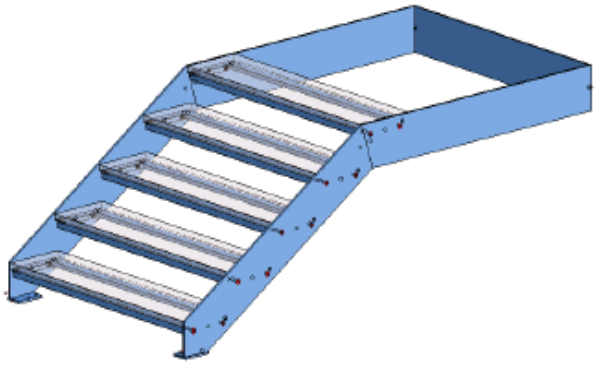


Alternatively you can also define “Steel Engineering plates” with the Material “Granite” and the corresponding thickness in the Catalogue Editor. The advantage of this is that workshop drawings of these plates can thus be created.

- Now mount the step onto the stair stringer, using the “Linear grid” option for bolt insertion, with a Steel Engineering bolting according to DIN 7990.
- Assign the boltings to the assembly “Step”.

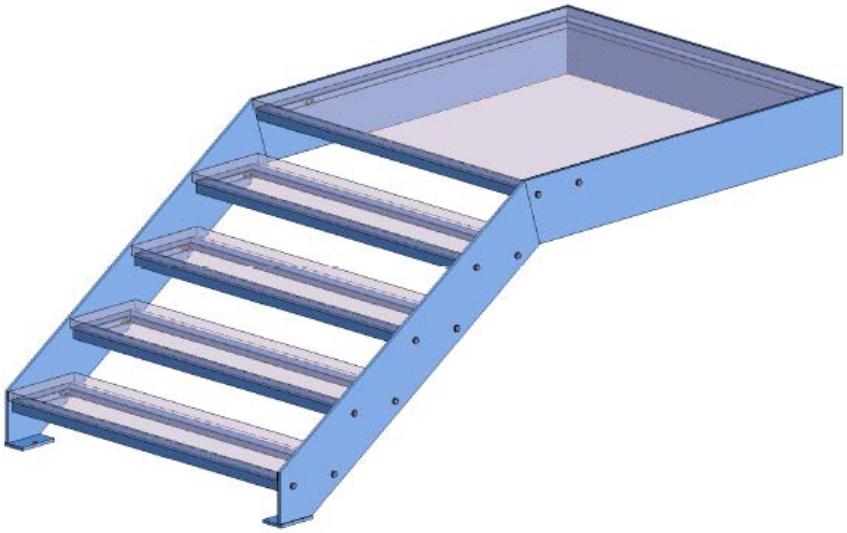


- Copy the assembly “Step” including both boltings 5 times, using the **Clone + Move, 2 points** function, taking the specified rise ratio (270/180) into account.
- For the first point, click anywhere in free space.
- For the second point: R 270 0 180.
- Connect the two stair stringers at the platform end with a wide flat steel 220x10, flush via the inner corner points, and assign it to an assembly “Platform finish”.



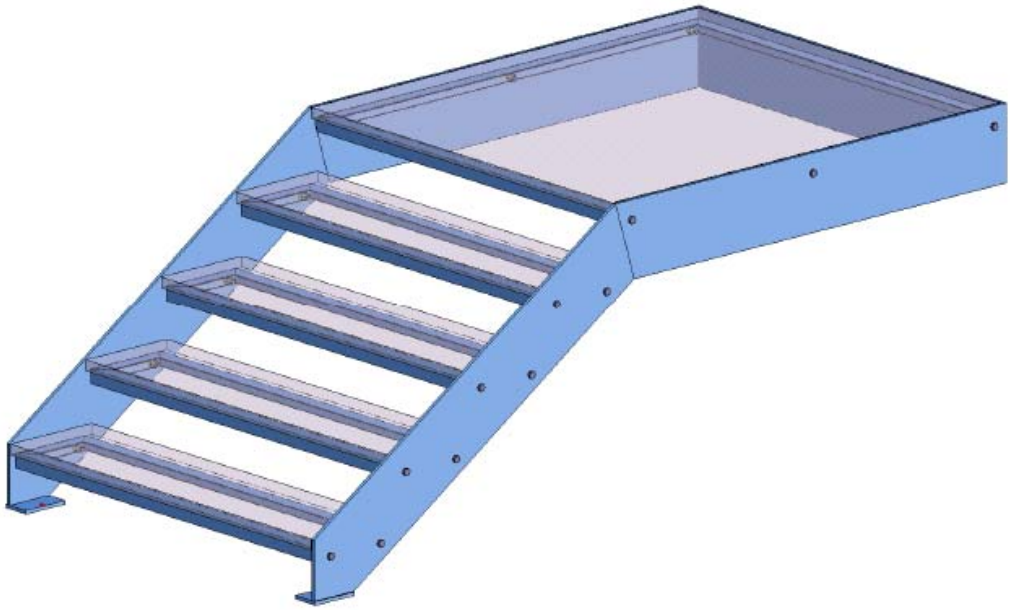
- Proceed as follows to adjust the size of the platform step:

1. Move the angle profile at the back of the step to the back stringer of the platform.
2. Update mitre – via Feature recalculation of the assembly, with external references switched on (right-click in Feature tab of ICN and select External references...: Use)
3. Adjust the step size via Sketch: Process (use HCM options)
4. Rename the upper assembly “Step” to “Platform”.



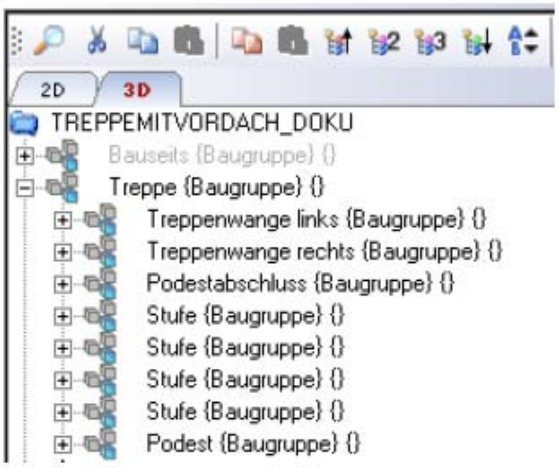
To update the assignment of parts to a bolting, open the **3-D Standard** tab and select **Standard Parts → Bolting → Part assignment – New, manually** or **New, automatically.**

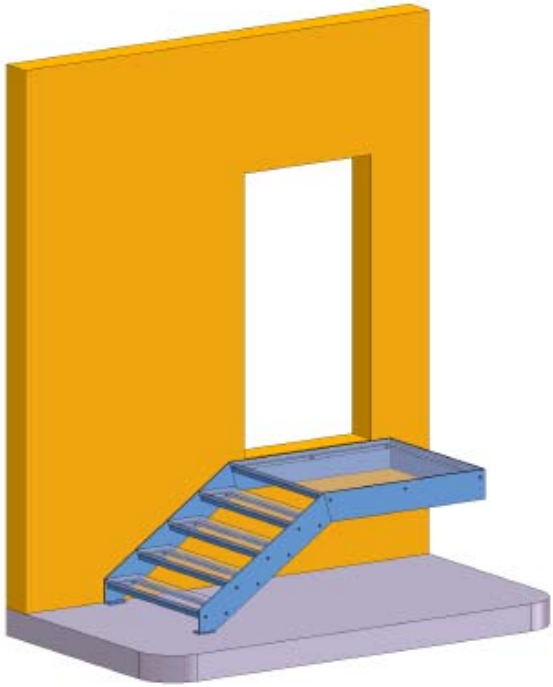
- After updating the bolt assignment, change the platform bolting grid to 3 bolts with a distance of 650, and select the mid-point of the profile as the fitting position (as for the step).
- Alternatively, you can use the "Even" grid function. Define the distances as maximum distances, and the number of bores will be calculated. You can also enter the number of bores, and the distances will be calculated.



You can use existing boltings as templates for new boltings to be set. Right-click the corresponding bolt and select **Boltings, Apply standard**.

- Check the part structure once again.

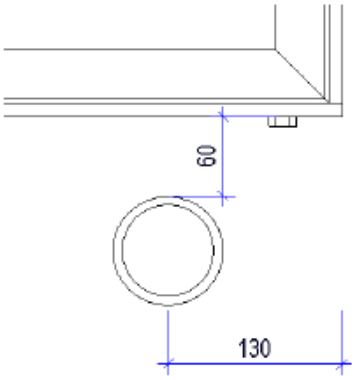




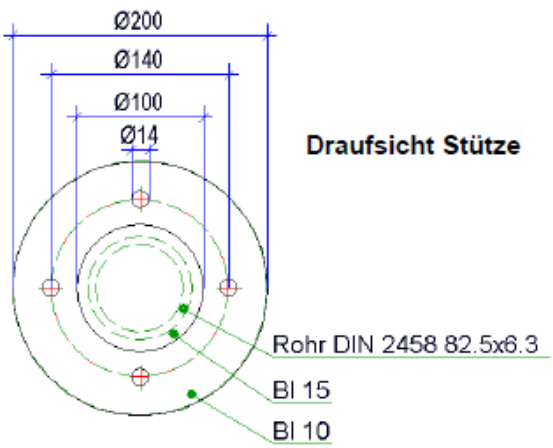
15.5 Support Posts

In most cases, the easiest way to insert support posts is vertical to the processing plane.

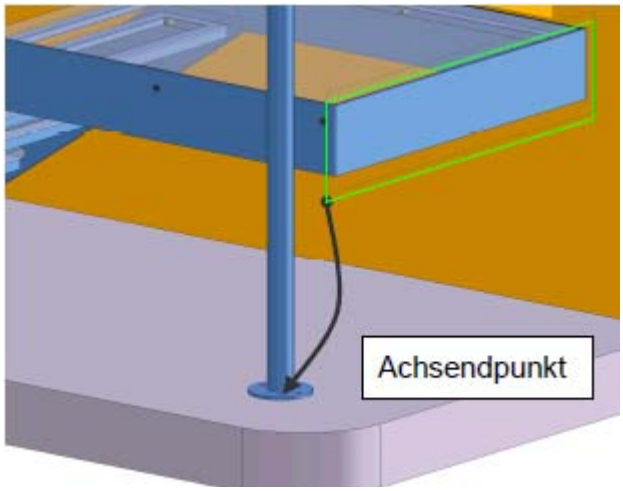
- Delete all processing planes and switch to Top view, Hidden Line.
- Crate an assembly “Support post rear”.
- Place a pipe with the dimensions 82,5 x 6,3 x 3090 relative to the platform edge: **R -130 -60**. Switch the fitting position to **Top axis**.



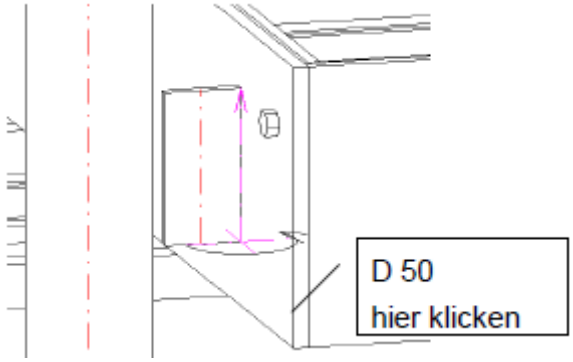
- On the **Steel Engineering** tab, select **Plate, new → Rect. → Round plate, Circular form** and create the10 mm support post foot plate with a diameter of D=200 mm on the base plate, including the bores.
- Proceed likewise to create the 15 mm cover plate with a diameter of D=100 and place it on the pipe.
- Adjust the support post length to the foot plate.



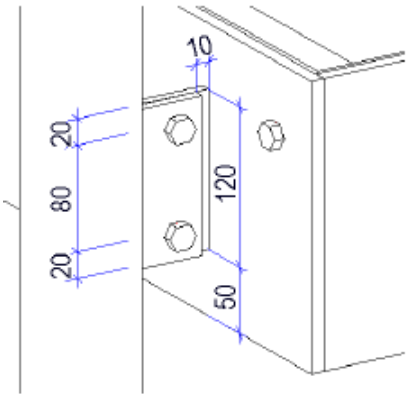
- The front support post can be obtained by copying the rear support post with a distance of 2420 mm.
- Rename the assembly to “Support post front”.
- To connect the support post with the stair stringer, first create a processing plane at the rear side of the platform stringer.
- Press and hold down the left mouse button and move the **plane** onto the axis end point of the support post.



- 2 flat steels with the dimension 50x8x120 serve as fixing straps for the staircase.
- Use point option D 50 to project the first fitting point from the vertical platform edge to the existing processing plane.
- Place the second point directly via the sketch function at a distance of 120mm
- Use the appropriate insertion options and rotation angles to get the flat steel into the correct position at its fitting location.

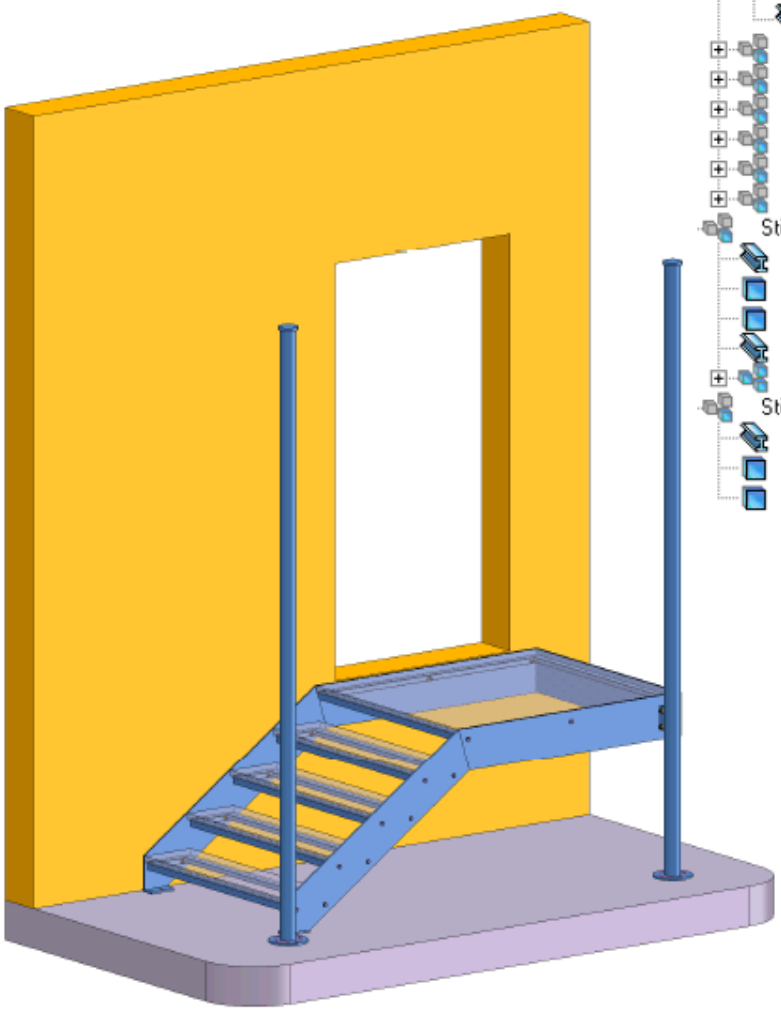


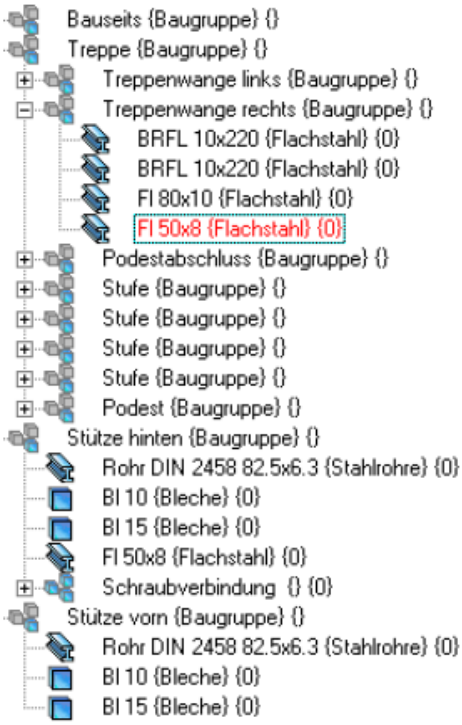
- You obtain the second flat steel by means of the **Clone + Move, 2 points** option.
- The bolting will be inserted according to DIN 7990 at the centre of the contact surface.



Points located in the centre of a contact surface can be snapped at the diagonally opposite corner points, via the point option (**M2) Mid-point between 2 points**.

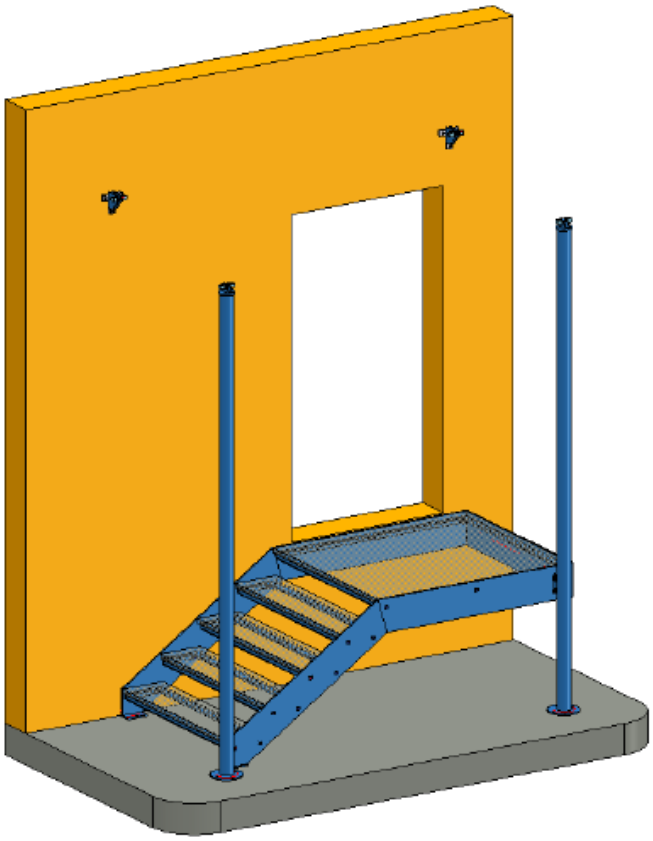
- Assign the new parts to the corresponding assemblies in the part structure.

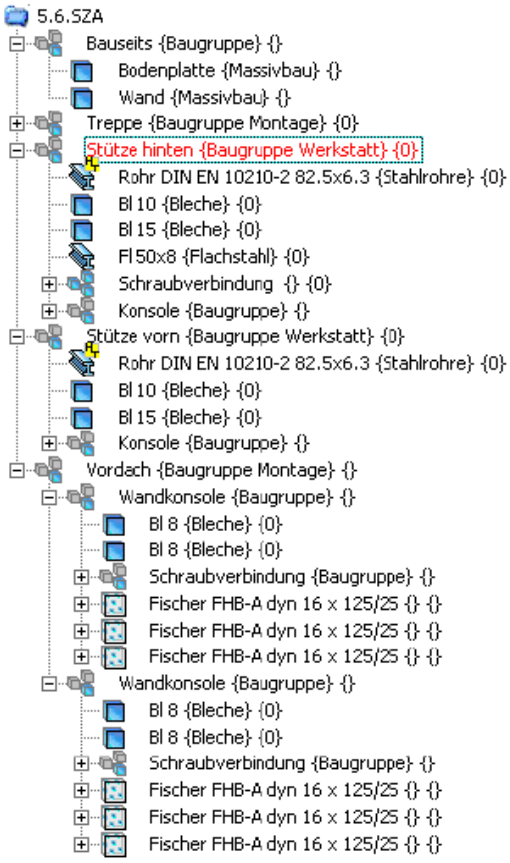




15.6 Canopy

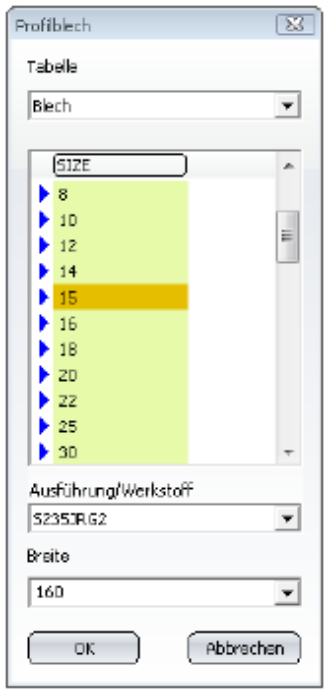
- To ensure a consistent position of the parts during designing, use the **Insert part, via Explorer (2-D/3-D)** function to import to import the assembly “Canopy.kra” first.
- Activate the World Coordinate System. This enables you to place the fixing brackets on the support posts, via the origins (right-click twice), directly at the correct position in your drawing.
- Assign the “Fixing bracket support post” assemblies to the corresponding “Support posts”

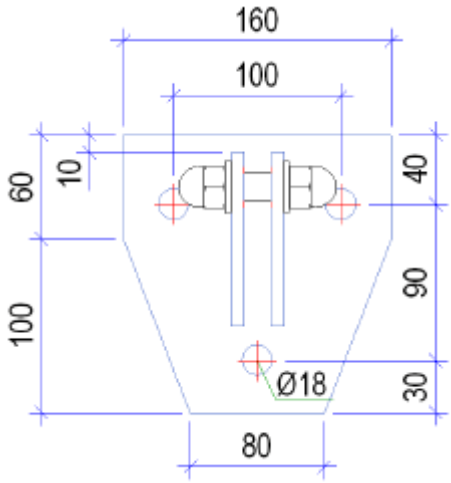




in the assembly structure.

- Create the base plates for the fixing brackets on the wall.
- Use the **Elongated plate** function for this. The function combines functionalities for Steel Engineering plates and for beams and profiles.
- All required dimensions can be taken from the drawing shown below.

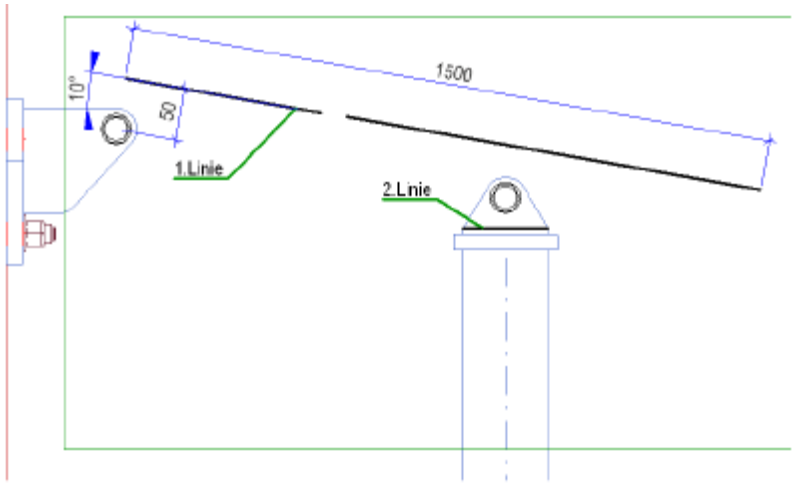




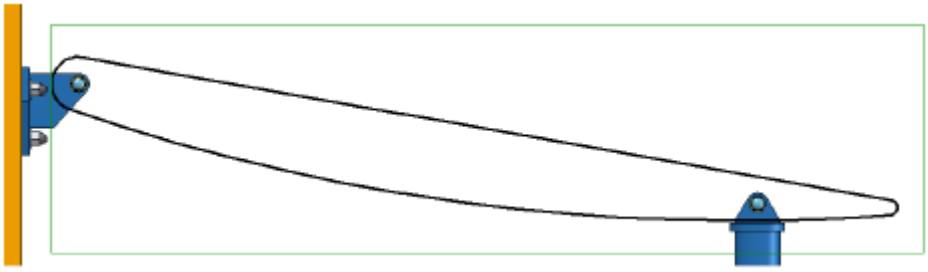
- To construct the sword-shaped ceiling support plates shown below, you require a horizontal plane on the support post axis.

To shape the form of the plates, use the **Plate from sketch (3-D SE)** function in the context menu of the sketch.

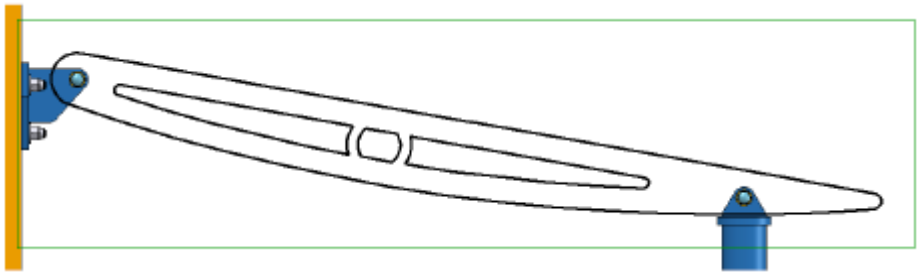
- First draw the following 2 lines:



- Connect the ends of the lines with the **Arc – 2 edges** function.
- The result should look as follows:

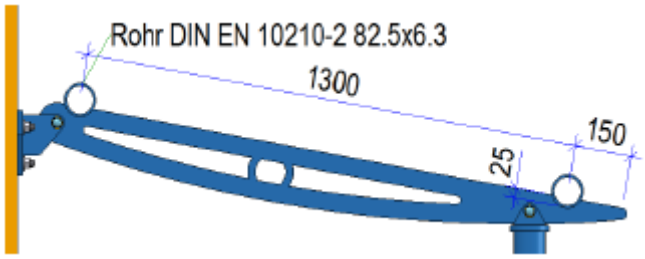


- Now create the inner polyline via an offset with a distance of 45mm. Fillet the ends with a radius of R10. If desired, add an ornament in the centre within the sketch.
- It might be necessary to apply the **Sort GE** function, to ensure that the inner polyline will be closed.

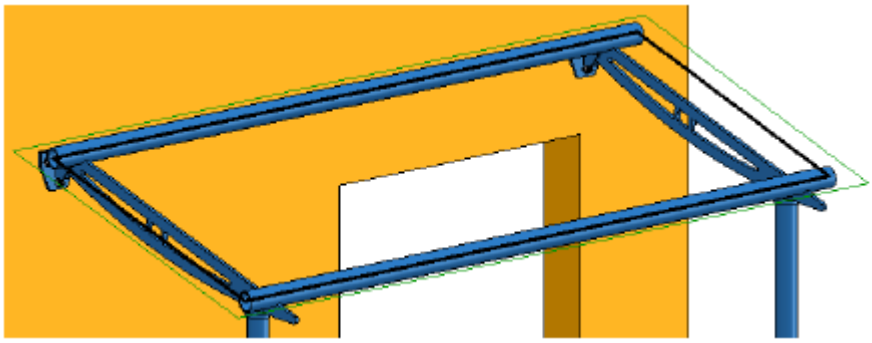


- Create a 15 mm Steel Engineering plate from the sketch.
- You can delete the sketch afterwards. If desired, you can process the sketch via the feature log at any time.
- Copy the plate for the rear post, and assign the wall plates to the fixing brackets.

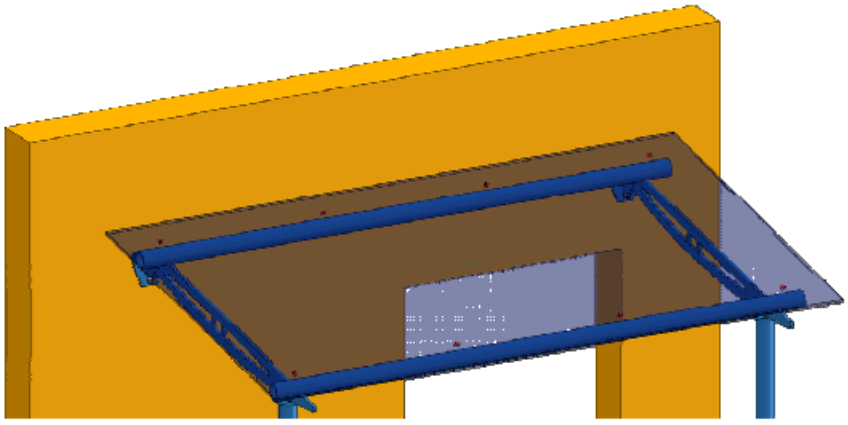
- Add 2 horizontal support pipes with a total length of 2680mm as shown in the screenshot below.
- Place the pipes in central position.
- Notch the two plates appropriately.



- For the inclined glass ceiling, first create a sketch in the plane of the two pipe axes.
- Draw the outer glass contour (polygon) via the diagonally opposite J-points of the support pipes.



- To define the distance of the glass pane, move the sketch in Z-direction 82.5/2+30 upwards. In practice, the distance is, of course, determined by the glass holders used.
- Open the **Steel Engineering tab** and select **Plate, new → Glass → Glass, from sketch** to create the glass of VSG 16-2. Place in your drawing by right-clicking twice.
- Delete the sketch and assign the glass pane to the “Canopy” assembly.
- The lateral projection of the glass pane is 100 mm, the projection at the top is 150mm, and the projection at the bottom is 200mm. The projections are realised by right-clicking the glass and selecting **Sheet processing → Lengthen in sheet direction** (via value) in the context menu.
- If desired, the bores can be directly applied to the glass.
- Use the “Even” grid option for this.



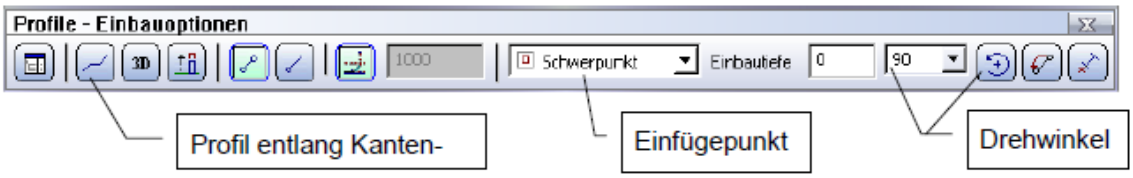
The geometries for the required glass holders are in practice obtained by an appropriate supplier and are not part of the training.

15.7 Railing

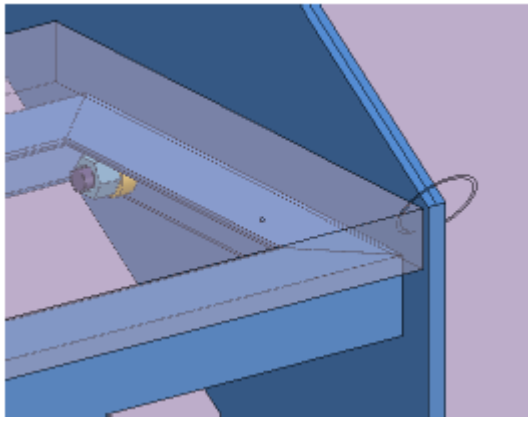
- You create the railing by placing a profile along a composite edge.
- Create an assembly “Railing”, and, within it, create a 3-D sketch.
- For the creation of the composite edge, switch to the **Sketch** tab and select **New → 3-D Sketch**.
- To draw the guideline for the railing cross bars and the handrail (centrally on the stair stringer), open the **Sketch** tab and select **Draw → Line → New polyline**.
- Adjust the length at the wall (45mm distance), close the corners and fillet the entire polyline with a radius R80.

If you do not want to fillet the composite edge completely, open the **Sketch** tab and select **Tools → Sort GE.**

- Place the pipe for the handrail (DIN ISO 1127 – 42.4x2.0 – 1.4301) along the composite edge.

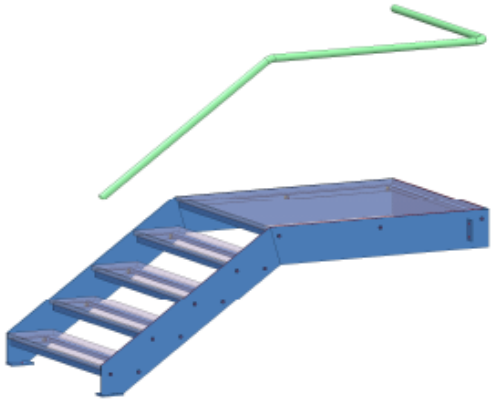


- After selecting the guideline, the cross-section will be projected up to the end of the stair stringer. At this point, the insertion point and the rotation angle can still be adjusted. Left-click (see info bar for user guidance) to generate the profile.

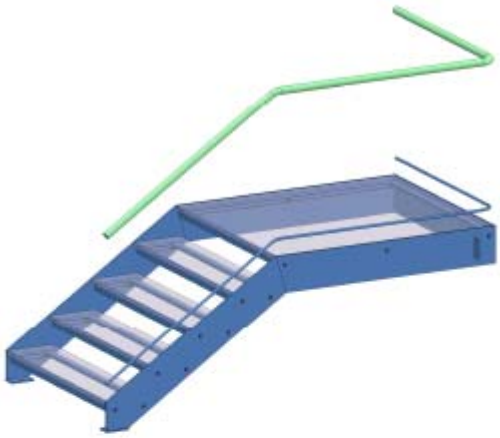


- Load the composite edge via the Feature log in order to re-use it for the creation of the railing cross bars. Move the 3-D sketch one level up in the part structure.

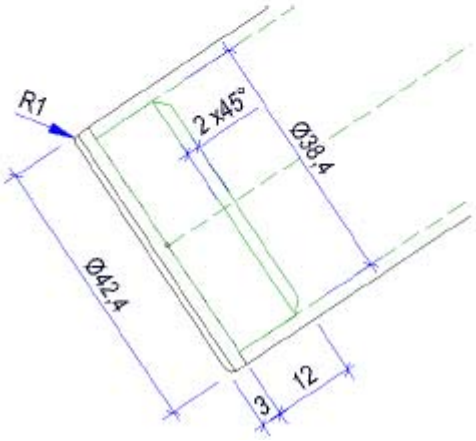
- Move the handrail to the correct height. (1000-42.4/2 upwards).



- Proceed likewise for the creation of a railing cross bar of round steel 16, move it 100 mm upwards, and fillet the ends with a radius of 8 mm.

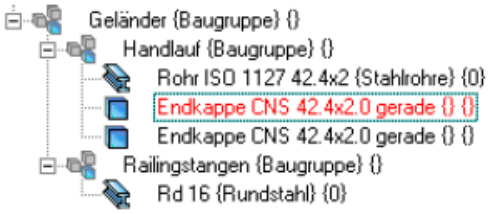


- The end caps of the handrail can be created from appropriately shaped solid primitives.
- After placing a processing plane onto the pipe end, you can place the solid primitives (cylinders) in an appropriately aligned way.
- Merge the cylinders into one part, and assign a designation and a material.
- Alternatively you can also use the “Revolved solid” part creation.



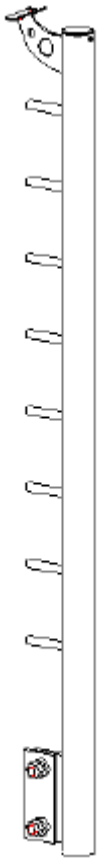
To spare you a tiresome aligning of purchased parts, please make sure that you save the parts with a suitable, practice-oriented processing plane when saving them to the catalogue.

- Copy the end cap via the Clipboard to the other end of the pipe.



- Insert the prepared railing post “Railing post.kra” from the Explorer via coordinate system origins (right-click twice) into a separate drawing file.

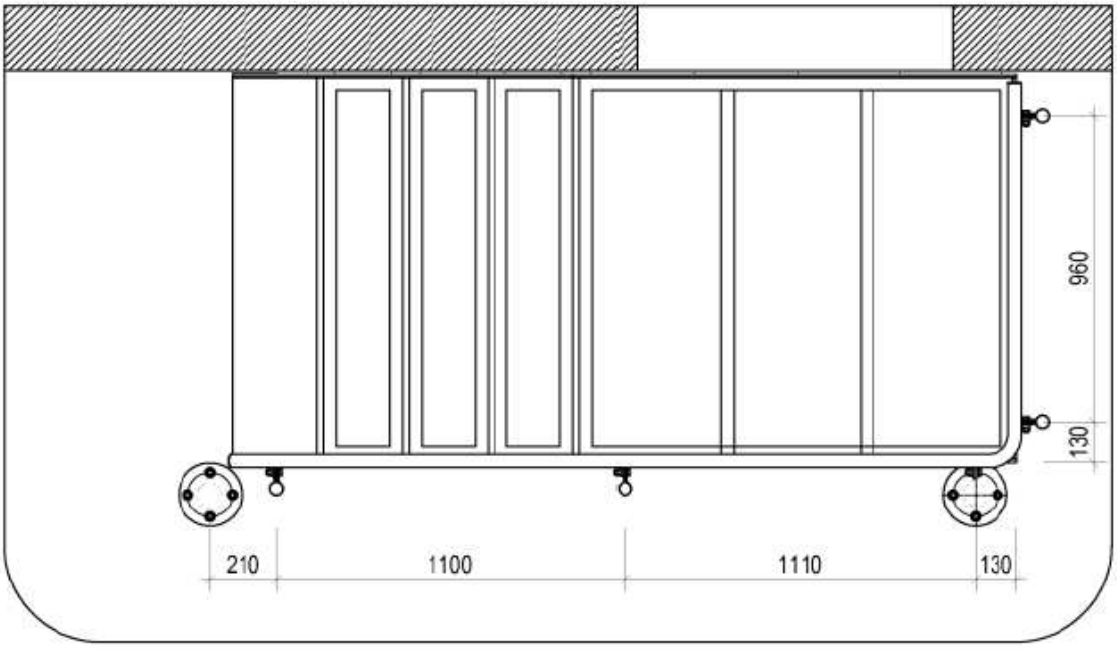
In many companies, railing posts are re-used again and again, with slightly modified shapes. It is therefore recommended to create one sample post and save it to the catalogue.



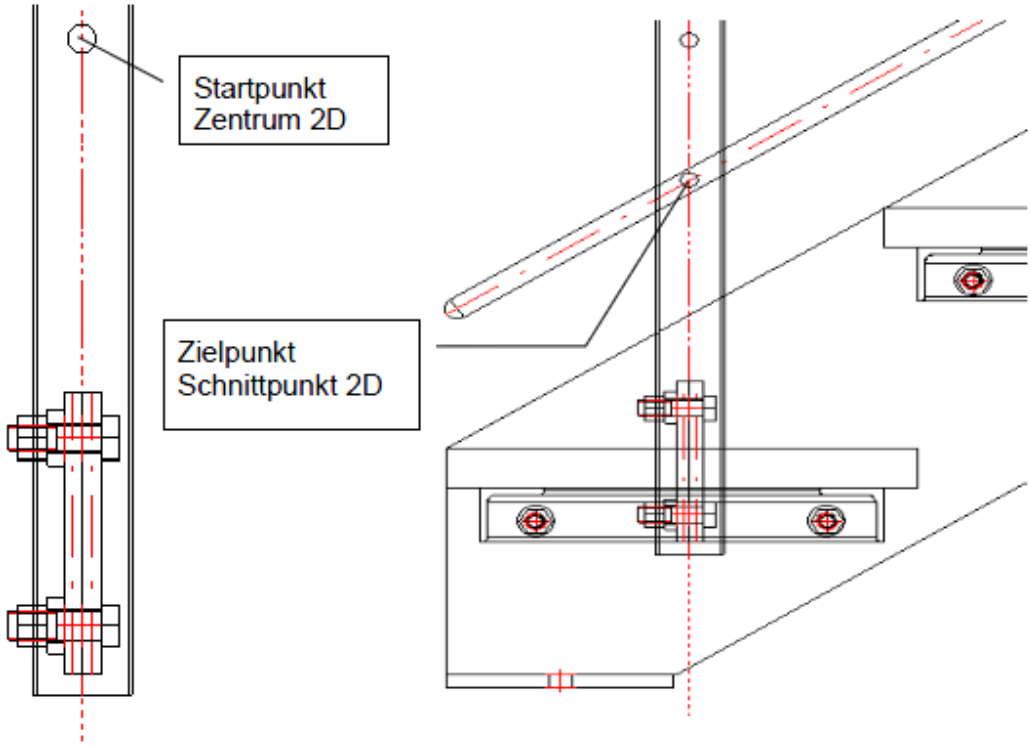
- Switch to Hidden Line representation, Side view from left, so that a corresponding preview image will be created when you save the part.
- Save the post to your catalogue.
- In the ICN, right-click the assembly “Railing post” and select **Others → Reference part, Save, Detail Drawing → Save as part, with catalogue entry**. Save to Factory standards/Parts and Processings/Table: “Example Tab-1”.

- Now insert the prepared post from your catalogue into your staircase construction.

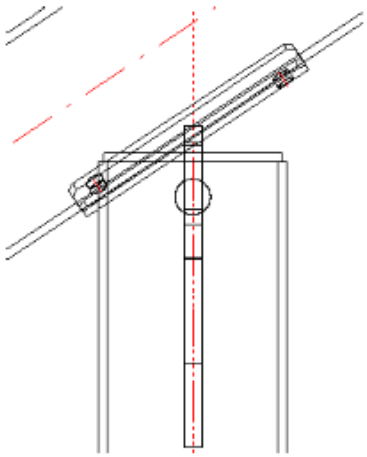
- Pay attention to the active processing plane.



- Create the lower railing post initially at the same height as the platform posts.
- For the vertical alignment of the post, use the **Transform → Move part, 2-D via 2 points (3-D) function**.
- To do this, switch to Front view, Glass model representation and create a vertical 2-D auxiliary line.

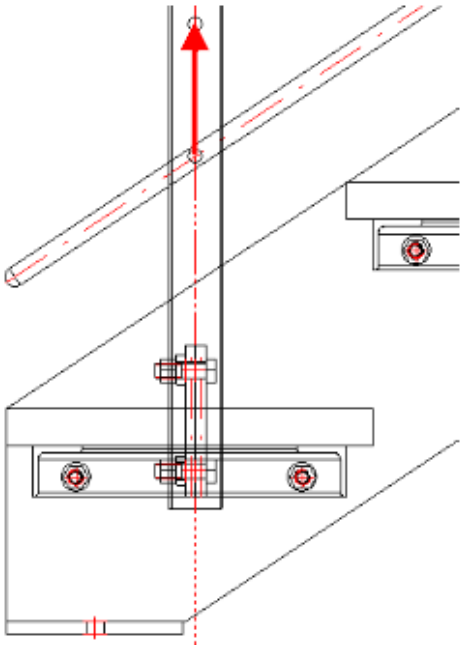


- The inclination requires an adjustment of the handrail bracket, and of the rotation angle of the half-shell with the same function.



- Assign the two outer fixing straps of the posts in the structure browser via multiple selection to the stair stringer (and platform stringer, respectively).

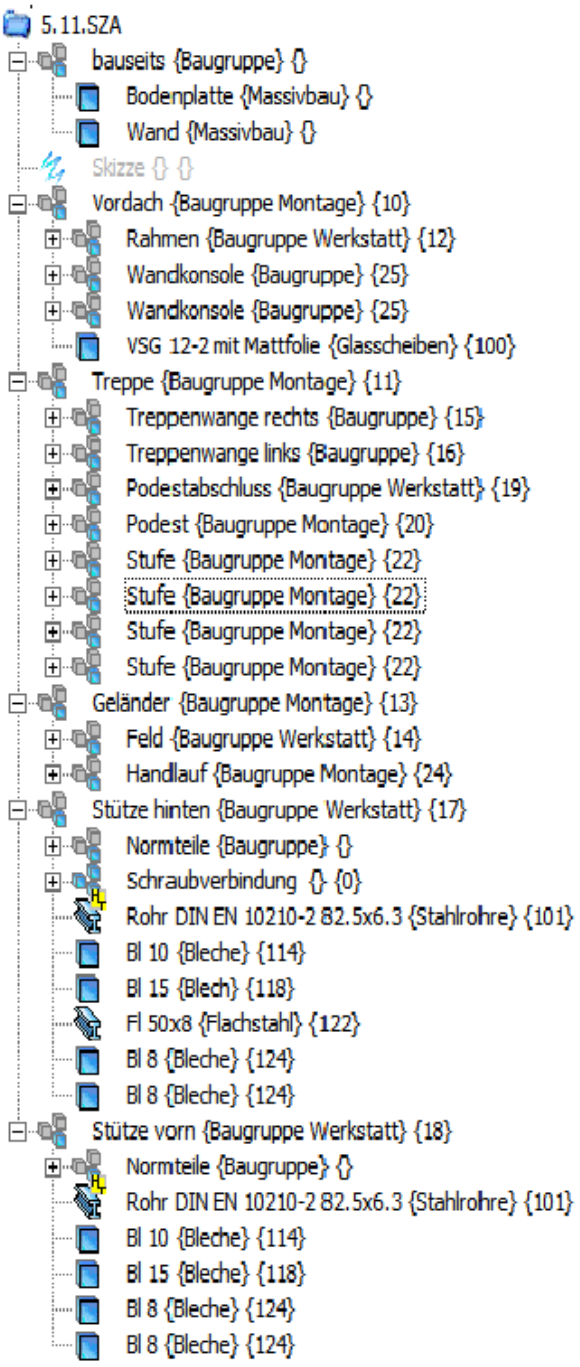
- Copy the railing cross bar using the **Clone → Clone + Move part, 2-D via 2 points (3-D)** option. Number of copies: 8 (create assembly).
- Delete the 2-D auxiliary line.



15.8 Part Structure

- Finally, check the part structure and BOM-relevance of the parts and assemblies in your CAD drawing.
- Steel Engineering beams, Steel Engineering plates, Standard Parts and Assemblies are set to BOM-relevant by default.
- Use Cut & Paste to insert your parts at the correct position within the part structure.

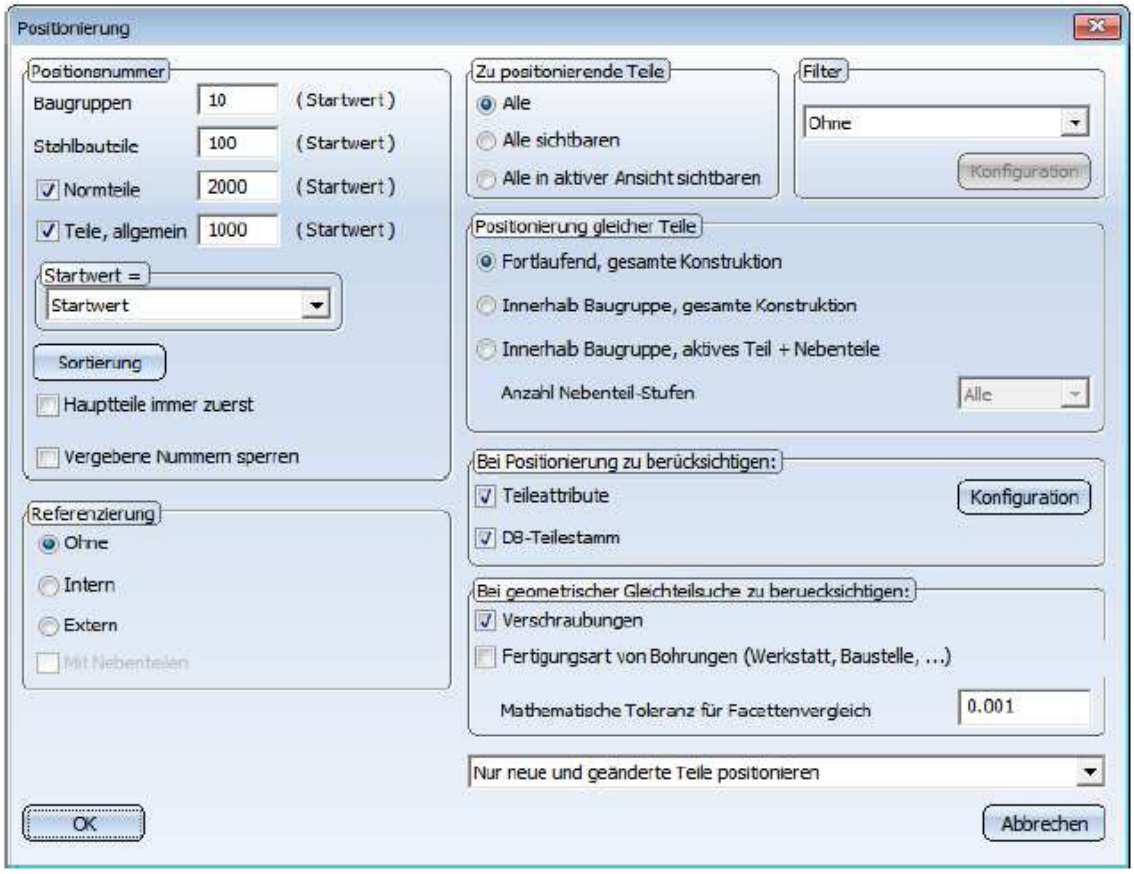
A correct part structure is a prerequisite for a practice-oriented output of Bills of Materials (BOMs) in the Report Manager. This enables you, for example, to output mounting lists for fixing elements in structure lists. The structures in HiCAD will also provide valuable assistance for assembling preparation.



15.9 Itemisation

The creation of Bill of Materials (BOMs) and Workshop drawings requires a previous itemisation of the parts in your CAD drawing. The functions for itemisation can be found in the **Itemisation/Detailing** function group of the **Drawing** tab.

- Itemise your drawing with the **Itemisation → ...with options** function (in the pull-down menu of the **Auto** icon).
- Specify the settings as shown below and confirm with OK:



The following itemisation functions are available:

Itemisation (**Auto** icon)	Automatically assigns item numbers to all parts, using the current settings.
Update	
...with options	Automatically assigns item numbers to all parts. The settings specified in the **Itemisation** dialogue window will be applied.
...individual parts	Assigns item numbers to individual parts. 1. Identify the part. 2. Specify the item number. 3. If you want to check whether the item number already exists, activate the **Check conflicts with existing item numbers** checkbox. 4. If you also want to itemise parts that are identical with the active part, activate the **Also itemise identical parts** checkbox. If the active part already has an item number, it will be displayed and can be changed. It will also be shown whether any identical parts exist.
Change parameters	Changes the default setting for itemisation
Information	
Select parts with same item no.	Finds and highlights all parts having the same item number as the identified part.
Select parts with specified item no.	Finds and highlights all parts with a specified item number.
Delete item numbers	
Part	Deletes the item numbers of individual parts. The item numbers of the sub-parts belonging to the active part will however be retained.
Part+sub-parts	Deletes the item numbers of individual parts, including the item numbers of the sub-parts belonging to the active part.
All	Deletes the item numbers of all Steel Engineering parts.

15.10 Bill of Materials – Report Manager

Bills of Materials (BOMs) can be created for complete CAD drawings, or only for the currently active part. Depending on the current settings for BOMs, HiCAD will collect the corresponding information and display it in the Report Manager.

The functions for BOM creation can be found in the **Itemization/Detailing** function group of the **Drawing** tab.

When you call the function, the **Bill of Materials (BOM)** dialogue window will be displayed. The window allows a rapid creation of BOMs and consists of the following options and input fields:

- Attribute source
- Report Manager Configuration
- Execute
- BOM type
- Variables file...

Make the desired settings, and start the BOM creation with OK. The Report Manager will then be launched automatically. Whether you will be shown a structure list or a quantity list depends on the BOM type you selected in the **Bill of Materials (BOM)** dialogue window.

- At this point your trainer will explain the different output and setting options of the Report Manager to you.
- A detailed description of the individual function can be found in the Online Help of the Report Manager.
- Use the **Bill of Materials, for active part** function for the creation of an assembly layout. Activate the desired assembly beforehand.
- For the assembly “Railing” the template file Hicad_Stahlbau01.rms has been used, which led to the result shown below. The template files enable you to create and save list templates that are tailored to your individual requirements.

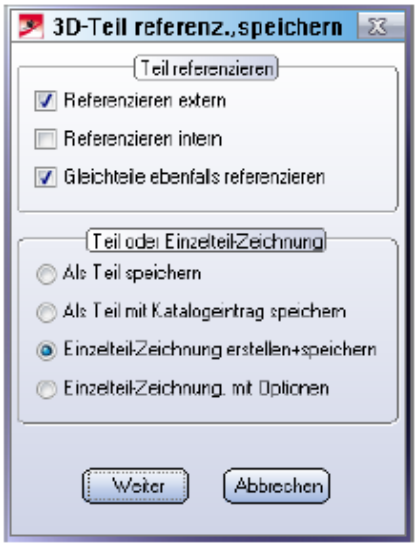
									7.5		69.4
800	2	Endkappe	0.0	0.0			1.4301		0.0	0.1	0.3
129	4	Bl 2	40.0	40.0			S235JRG2		0.0	0.0	0.1
128	32	Rd DIN 1013-1 12	0.0	57.0	|-------|	|-------|	S235JRG2		0.1	0.1	1.6
127	4	Bl 4	70.0	72.3			S235JRG2		0.0	0.2	0.6
125	4	Fl 50x8	0.0	120.0	|-------|	|-------|	S235JRG2		0.1	0.4	1.5
111	8	Rd DIN 1013-1 16	0.0	3838.3	|-------|	|-------|	S235JRG2		1.5	6.1	48.5
110	1	Rohr ISO 1127 42.4x2	0.0	3852.3	|-------|	|-------|	1.4301		5.1	7.8	7.8
102	4	Rohr DIN 2458 42.4x2	0.0	1127.6	|-------|	|-------|	S235JRH		0.6	2.2	9.0
Pos.	Menge	Bezeichnung	B[mm]	L[mm]	Anschnitt (Steg)	Anschnitt (Flansch)	Material	Benennung	Fl[m²]	Gew[kg]	Ges.gew.

15.11 Layouts and Workshop Drawings

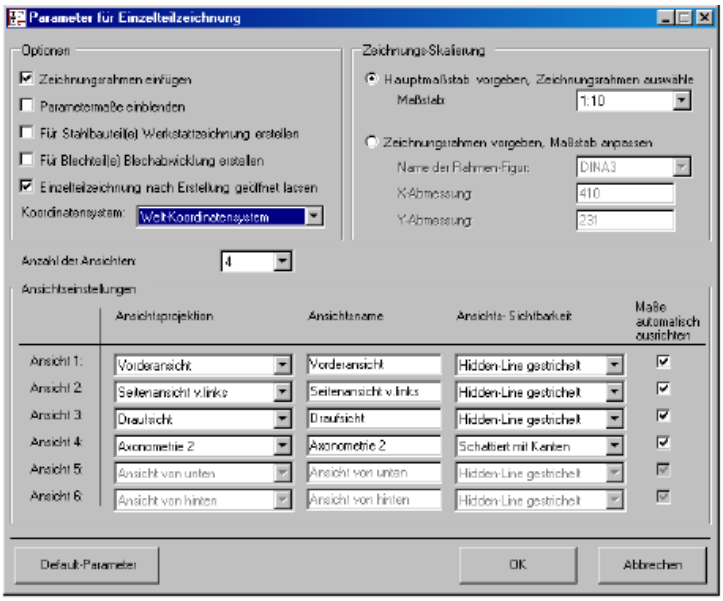
- In practice, several types of plans and drawings are required. These are, for example:
  - Advance drafts
  - Plans for approval (for builders and architects)
  - Workshop plans (individual parts and assemblies)
  - Installation plans

- In this training you will first create the present general plan. Use the techniques for view creation you have learnt in the 3-D Basics training.
- Insert the description in the plan by means of the “2-D text block” function. Please consider the part assignment of the 2-D parts in the structure browser.

Text blocks can be saved as template files in the Editor and re-used at a later time.



- HiCAD offers several options for workshop drawing creation.
- For smaller CAD drawings, the workshop drawings can be created within one drawing spread over several sheets.
- Larger CAD drawings need to be divided into several areas (assemblies), which are then saved to external drawing files. The external assemblies need to be referenced externally, if changes to the original CAD drawing can be expected, which would require a synchronization of the parts.
- The function **Reference, Save, Detail drawing** can either be accessed via the **Reference** function group of the **Drawing** tab, or via the context menu of the part or assembly.
- Specify, via the options in the dialogue, whether you want to reference the assembly or the part.
- The detail drawing generation can be performed either directly or with options.
- If you select **...with options**, you can specify various settings before creating the detail drawing.



- Give the drawing a name and save it.
- Within the drawing you can now create further views.

Workshop drawings

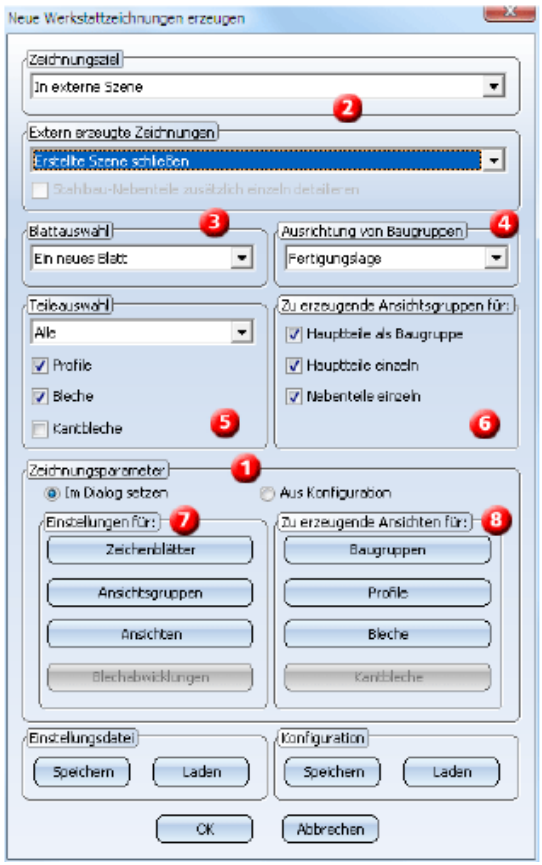
Please note the following:

- Change referenced assemblies always in the original, complete CAD drawing, and save the changed, referenced assemblies with the referencing functions.
- When applying changes to your drawings, please remember to re-itemize your orginal, complete CAD drawing before saving the assemblies.
- The changes in the original, complete CAD drawing will then also be transferred to the drawing files containing the workshop drawings.

Descriptive, well-structured workshop drawings and detail drawings are required for a reliable planning and production. The automated creation and management of these important documents in HiCAD is particularly suitable for large Steel Engineering assemblies:

- All detail drawing views are managed in one view group, including the view-comprehensive parameters.
- All views of a view group will be linked to each other.
- Each view group can contain several, auto-generated views per part or assembly.
- HiCAD automatically arranges the views, view groups, item numbers and annotations, dimensionings etc. in a collision-free and space-saving manner.
- Production drawings can be automatically spread over several sheets with separate drawing frames if desired.
- Assembly BOMs can be automatically integrated in the workshop drawing if desired.

To create a new workshop drawing, activate the **Drawing** tab and select **Itemization/Detailing → Drawing derivation**. The **Derived drawing** dialogue window will be displayed.



Proceed as follows:

1. Select the target for the workshop drawing from the **Drawing target** list box.
2. In the **Part selection** area, select the part types that you want to be transferred to the workshop drawing by activating/deactivating the corresponding checkboxes.
3. In the **View groups to be created for...** area, activate/deactivate the checkboxes for the parts for which you want to create view groups.
4. Specify the **Settings for** drawing sheets, view groups and views.
5. In the **Views to be created for ...** area, determine which views for assemblies, beams, plates etc. you want to be combined in a view group.

Once you have specified all required settings, close the window with **OK**.

If you have chosen the setting **Part selection: Select, individually**, HiCAD will now prompt you to identify the parts to be included in the workshop drawing. To end the part selection, press the middle mouse button.

The workshop drawing will be directly created as a Sheet view - either in the original drawing or in a new drawing. It can be opened in the **Views** tab of the ICN.

Please note:

- To create a workshop drawing in the current Steel Engineering drawing, first itemize the parts of the Steel Engineering drawing with the same-named function.
- The inserted Sheet views can be deleted via the context menu for views (either in the ICN or in the drawing area).
- View groups will always be displayed within a rectangular frame that encloses all views of the group. This frame can be adjusted to the size of the view group with the **Update derived drawing** function.
- Bolts and sub-parts will be dimensioned automatically. Plates will be rotated 90 degrees if their extension in Y-direction is greater than in X-direction.
- 3-D weld seams can be automatically annotated in the workshop drawings. This behaviour can be activated or deactivated in the **Settings for views** dialogue window.
- For countersinks (e.g. DIN74), the standard, nominal diameter and (if any) auxiliary notes will be displayed in the generated workshop drawing.
- When you call the function again, the last specified settings will apply. Use the **Derive → Change settings** options to change the settings of an already existing workshop drawing subsequently.
- To influence the arrangement of the views, open the **Views** tab of the Ribbon and select **View Functions → Show > ...** .
- When creating the workshop drawing views, the current projection will be recognised as the so-called “Required position”. This enables you to restore the original position at any time after rotating the view.

More information on these functions can be found in the Online Help.

15.11.1 Exercise: General Plan

- The template can be found in the appendix of the training book.
- Create the corresponding views on a 1:20 scale on Sheet 1. Dimension the views, add item numbers and annotations.
- Insert a DIN A2 drawing frame.

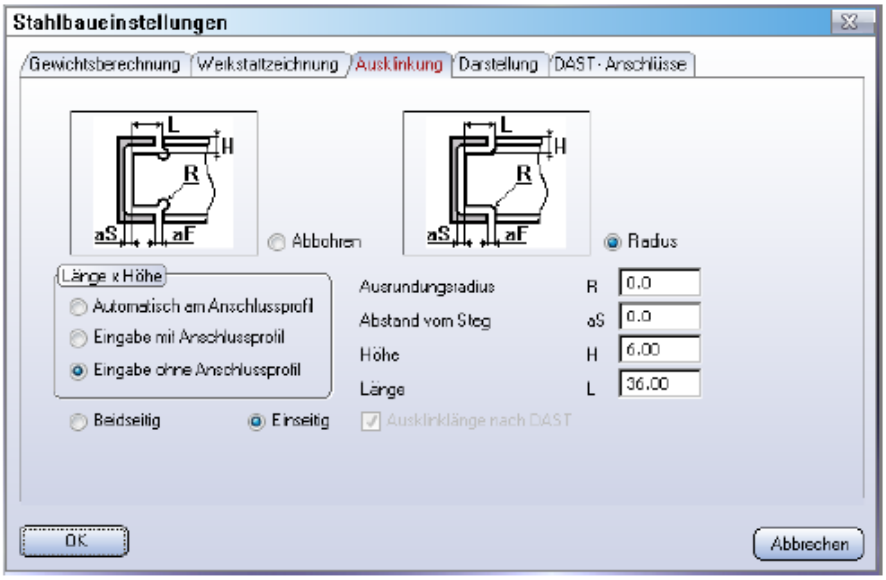
15.11.2 Exercise: Workshop Drawing for Stair Stringer

- Create a workshop drawing of the assembly “Stair stringer right”.
- The layout should include drawing frame, BOM, detail views of individual parts of the assembly, and auxiliary, production-relevant dimensions.
- Use the automatic workshop drawing generation in conjunction with individual views.

15.12 Additional Exercise “Staircase”

15.12.1 Platform Extension

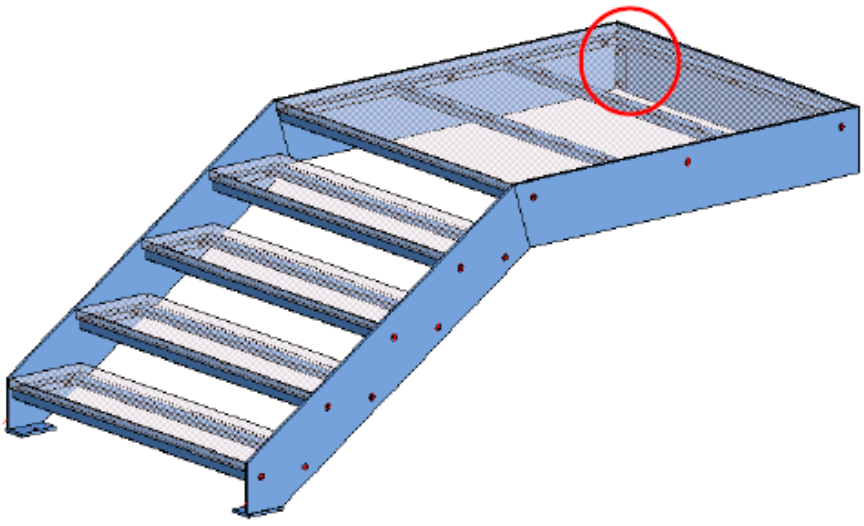
- Add a T-beam to the platform construction, via the mid- points at the inner side of the L-profiles. In such cases, the Notch function offers you the option to re-define the notching via new parameters.
- Apply 8 mm chamfers to the T-beam ends for a production-oriented shape.



- Move the T-beam 220 mm to the rear, and copy+move it 440 mm in opposite edge direction.

15.12.2 Bolt Finished Platform Together

- Finally, connect the two stair stringers with the rear platform stringer. The parts should again be assigned to the corresponding assembly.



- If required, bolt the L-profile at its mid-point to the rear platform stringer.

16 Import / Export Interfaces

- Besides a bidirectional LogiKal interface (see Chapter **7 Creating Glazing Constructions with LogiKal**), HiCAD offers a number of other Import and Export Interfaces:

16.1 NCX Export for Machine Control (Profile Machining Centre PZB)

- The NCX Export Interface saves beam and profile lengths, cutting angles and processings to a NCW file.

16.2 3D-PDF Export

- The 3D-PDF file can be read with any free PDF Reader. The advantage of PDF files is that they do not disclose the know-how contained in the drawing, but still provide the builder or architect with sufficient information for the realisation of the project.

16.3 IFC Import/Export

- The Industry Foundation Classes (**IFC**) constitute an open civil engineering standard for the digital calculation of building models.
- IFC is supported by many software programs for the purpose of building data exchange. Fields of application include 2-D/3-D CAD, or statics and energy calculations. Data exchange takes place by means of IFC files with the extension *.ifc.

16.4 2-D and 3-D DXF/DWG Import/Export

16.5 Step/Iges Import/Export

- 3-D Solids Interface

16.6 Statics Interface

- **DSTV** and **IFC**

17 Glossary

3-D C-Edge Constraint Manager
The C-Edge Constraint Manager is based on the 3-D HCM and is a tool enabling you to position and move composite edge elements. For this purpose, the c-edge elements are linked to each other through so-called “constraints”, i.e. dimensional and logical relations and restrictions. The C-Edge Constraint Manager then transforms the appropriate elements in such a way that the defined constraints are fulfilled.

3-D Part
A 3-D part consists of 3-D graphical elements (short: “GE”), which constitute the lowest organisational units of a 3-D drawing. These can, for instance, be 3-D points, 3-D (poly)lines, 3-D circles/arcs or surfaces.

3-D Polyline
a 3-D polyline is a structurally coherent (i.e. continuous and uninterrupted) succession of individual 3-D lines. The end point of the previous line coincides with the start point of the next line.

Absolute coordinates
Coordinates referring to the point of origin of the current coordinate system (short: “CS).

Similarity transformation
Simultaneous moving, rotating and scaling of a part.

Active attributes
HiCAD offers the option to define particular attributes as active attributes. For all subsequent manipulations only elements with these attributes will be taken into account.

Additional modules
Modules that serve as supplements to other modules. You can, for instance, supplement the Steel Engineering module with the Metal Engineering module.

View
Projections, i.e. two-dimensional representations of 3-D objects. One distinguished between standard views (top view, side view, front view), isometric and axonometric views. View generation is based on projection principles. *See also Projection methods.*

Arithmetic expressions
Valid algebraic combinations of constants, numeric variables, arithmetic operators and brackets.

Assembly
Several combined (“assembled”) parts under one superordinate part. This superordinate part serves only the purpose of structuring (enabling, for instance, a simultaneous moving of the sub-parts below it, or the assigning of part master data to the superordinate part) and can therefore be an empty dummy part, i.e. it needs not necessarily contain lines, dimensionings or similar.

Associativity
“2-D/3-D Associativity” basically describes the possibility of a combined and simultaneous working in 2-D and 3-D. A HiCAD drawing files can both contain 2-D and 3-D objects. 3-D views can be added to technical drawings in 2-D, 2-D objects can be used as a basis for 3-D drawings etc.

Attributes
Characteristics such as colour, line type and line thickness, influencing the appearance and shape of elements.

Autopilot
A tool for the identification of points and parts. If the Autopilot is switched on, the cursor provides information on snappable points, (e.g. start points, end points or mid points of edges, perpendicular base points, intersection points or centre points of circles) when it is moved over the drawing objects. These points can be taken over directly, i.e. without activating a point option.

Basic elements
The lowest organisational units of a HiCAD drawing. In 2-D, these are graphic elements (GE); in 3-D, these are edges, surfaces and 3-D points.

Beam processing functions
Functions enabling you to cut, notch, move, rotate, mitre-cut or divide beams and profiles.

Bill of Materials (BOM)
List of all parts of which a product or an assembly is made up, indicating quantities and designations of the parts.

C-Edge Constraint Manager
see 3-D C-Edge Constraint Manager.

Clone
Parts which are required several times in a drawing can be copied. These copies are called “clones” in HiCAD. You can choose whether you want to combine all parts into one part or if you want to create one superordinate part above the copied parts instead.

Closed contour
Continuous contour with coinciding start and end point.

Collision check
Function in the Information tab that checks whether there are collisions of parts in the drawing. If so, the detected collision is shown in the drawing and in the graphics window of the ICN.

Colour table
Enables you to pre-set the assigning of available colours.

Construction Aids
Pre-defined parts or processings facilitating the construction. In the Steel Engineering module, these are, e.g. boltings, base plates, bracings, connections, corner plates, rigde joints, stairs, steel halls etc.

Contour
Contours link graphic elements that are optically but not structurally coherent – they need not belong to the same part.

Contour edges
“Apparent” edges of the analytical 3-D model.

Contour hatching
Hatches a surface defined by a contour. Lines integrated in the boundary line may belong to different parts.

Copy
see Clone

Crosshairs
Cursor symbol which is moved by the mouse, indicating the current position on the screen.

Cursor
See Crosshairs

Cut
Cutting of beams in longitudinal direction, along the entire beam length.

DAST connection
Steel engineering connections according to DSTV/DASt.

Data structure
The HiCAD drawing structure. The highest organisational unit in HiCAD is the **Drawing**. The Drawing contains **Main parts** which may in turn consist of **Sub-parts**. The individual parts consist of the lowest organisational units: In 2-D, these are the graphic elements (GE) and in 3-D, these are edges, surfaces and 3-D points. *See also 2-D part, 3-D part, Drawing.*

Default value
The value of a parameter which is set when the program is started.

Detail
An arbitrary drawing detail can be defined and inserted separately on the screen in any scale.

Display lists
Graphic card drivers for an accelerated output of graphic elements on the screen.

Drawing
Drawing file in HiCAD with the file extension .SZA. A Drawing is the highest organisational unit in HiCAD. It is composed of 2-D and/or 3-D parts. These can be assemblies, parts or bores. Each drawing is assigned an unambiguous name, enabling the user to retrieve it.

Drawing detail
The detail of a drawing that is displayed on the screen (alterable by the “Rectangular zoom window” and “View all” functions.

Drawing layout
Arrangement of several HiCAD drawings on one drawing sheet for print output.

Engineering axonometry
Parallel projection with an unshortened representation of the y-axis and z-axis and a shortened representation of the x-axis. The x-axis stands at under angle of approx. 41°, the y-axis under an angle of 7° to the horizontal of the screen plane. The z-axis stands perpendicular on it.

Error tolerance
Rounding errors in numerical values. Point coordinates, e.g., can be checked for a specific tolerance. Errors are calculated as the difference between the exact and the rounded value. The error tolerance, i.e. the allowed maximum deviation, can be preset within a specific range of values. Two points are recognised as one if the distance between them is less than the given value.

ESM (European Solid Modeller)
ISD’s self-developed 3-D kernel.

Feature
When designing in 3-D, Feature Technology is active by default. Feature Technology records all processing steps (“Features”) in a Feature log. Activate the Feature tab in the ICN to view the processing steps applied to the active part. With the help of Feature Technology you are enabled to conveniently change parts via the Feature log, and even correct processing steps that were applied long ago, without having to repeat these steps once again. You can also derive Feature variants from the Feature log. If Feature Technology is switched on, HiCAD automatically generates the parametric dimensions of the part. These are visible if the part and the Feature step are active. Such parametric dimensions provide, besides the Feature log, an additional option to change parts. *See also Parametric dimensions.*

File group
Short term for a HiCAD directory path, defined in the FILEGRUP.DAT file.

File name
The name under which the operating system (e.g. Windows or UNIX) manages the HiCAD data, such as CAD-drawings, parts, macros etc.

Fitting points
Points used for reference, e.g. when transforming parts.

Fitting types
In the Steel Engineering module you can fit parts in the drawing with the fitting types „Fit beam in space“ or „Fit beam perpendicular to processing plane“. If both fitting types are DEactivated, you can fit the beam parallel to the processing plane; if the fitting type „Fit beam perpendicular to processing plane“ is active, the beam will be perpendicular to the active processing plane. If the fitting type „Fit beam in space“ is active, you can fit beams in any direction, independent of the active processing plane. To specify the end point of the fitting, you can use the point option „Relative“ or directly select the end point on an existing beam.

Freehand symbol technique
Special input method based on the definition of a symbol and assigning it to a specific command.

Generating rules
Set of rules enabling an automatic generation of numbers or names for many fields of the database mask, via the program genummer.exe. Normally, you let your computer generate the file name.

Graphic elements (GE)
Lowest organisational 2-D unit. Graphic elements can be points, symbols, lines, circles/arcs or conic sections.

HCM HiCAD Constraint Manager
Parametric designing on the basis of Constraints, linking individual objects to each other by means of geometrical relations.

Identical part search
Automatic detection of parts with an identical geometry in the drawing. HiCAD performs the identical part search during the itemisation of a drawing. All detected identical parts in the drawing are assigned the same item number, only one detail drawing is created for identical parts, and are displayed as only one part in the workshop drawing. *Also see Item numbers.*

Identification
Selection of an object with the cursor.

Intelligent Cursor
Context-sensitive cursor providing you with functions appropriate to current processing requirements. If you process objects, for instance, you are prompted to select the object you want to process. When you now move the cursor on an object, this object is highlighted and can be selected for processing by a mouse-click. Only those functions which are useful for the processing of this object are then displayed.

Interfaces
Software modules enabling data interchange with other programs.

Isolated points
Independent points that can be inserted freely into the drawing. They need not be located on a drawing object. Isolated points can be used as auxiliary point during drawing creation, or as fitting points during transformations.

Item numbers
Item numbers can be assigned automatically or manually before creating a Bill of Materials (BOM), Detail drawing or Workshop drawing. Item number *tags* are not absolutely necessary, they do however facilitate the assigning of item numbers to the corresponding part in the drawing. Itemised parts will appear in the BOM and enable the detection of identical parts in the drawing. Identical part search ensures that all identical parts detected in the drawing are assigned the same item number. only one detail drawing is created for identical parts, and are displayed as only one part in the workshop drawing.

Kinematics
Functions for modelling, simulating and analysing complex freely definable linkages.

Kinetostatics
Analysis function for bearing and element stress.

Layer
The working with layers can be compared to the working with transparent foils on a conventional drawing board.

Level
see Overlap and Overlap level.

Library
A “pool” of parts (part library), macros or variants (macro library). A library can be added to the screen menu. In this case, the calling of the parts, macros or variants takes place via selection of the appropriate library entry.

Local 3-D coordinate system
Temporary 3-D coordinate system, which can also be rotated.

Logical relational expression
Simple arithmetic comparison, e.g. a > b (a is greater than b). The value of the expression is TRUE or FALSE.

Logical variable
Variables which always return either TRUE or FALSE as a result. They are frequently used in macro technology, especially in loop and IF conditions.

Macro
Sub-program accessing HiCAD functions and creating variants in cooperation with the user. Macros serve the purpose of reproduction, i.e. the repetition of permanently returning HiCAD processes.

Main part
The second-highest organisational unit after the Drawing. A Main part can be an Assembly or an individual part. An unlimited number of Sub-parts can be assigned to a Main part to provide the logical structure of a real construction. *See also Part structure and Sub-part.*

Mark-up
Use this function to add comments and correction suggestions to a document, without having to change the original document. All comments added via mark-up are identified by user name and date and can thus be retraced at any time. When you use this function, a file with the extension *.RED is created. You can pass this file to other users and view it with an external Viewer.

Mask
A form that is temporarily displayed for the input or selection of data. Masks usually contain input fields or selection fields for parameter specification, as well as explanatory texts.

Model area
A view in which all other views, sectional views, detail views, drawing frames etc. are hidden. This enables a fast and structured working. Besides, you can rotate the model view without any repercussions on the production views in the sheet area. *See also Sheet area.*

Mouse
A device that is manually moved over the table, thus moving and controlling the crosshairs on the screen.

Move+Rotate objects
Simultaneous moving and rotating of objects.

Multiple selection
Selection of several parts in the ICN or in the drawing while holding down the CTRL or SHIFT key.

Named points
Points to which a name has been assigned. This is possible for start points and end points of lines and arcs, as well as for isolated points.

Natural coordinates
All 2-D data contained in the drawing refer to the Cartesian coordinate system. These coordinates are also referred to as natural coordinates.

Object selection
A choosing of objects for processing, either via identification, name entry, or with the structure browser.

Overlap level
You can also create overlaps in 2-D by assigning so-called “overlap levels” to parts. *See also Overlap contour and Overlaps*.

Parametric dimension
Dimensionings appearing on the active part if the Feature is switched on. If you right-click a dimensioning, you are enabled to change the part. *See also Feature.*

Parametric Variant Technology
Parameterised geometric form, formulated as a procedure; e.g. the representation of a nut M10 according to DIN 934, 936.

Part
A drawing consists of very many graphical objects, such as lines, texts, dimensionings, etc. All of these graphical objects are assigned to parts. One part can, for instance, be called IPE 300 or Bl 100x200x1, another part can be called „Flange“. All lines, dimensionings, texts and processing symbols of the IPE-beam are assigned to the part IPE 300, all lines, dimensions etc. of the flange are assigned to the part „Flange“. If you move, rotate, mirror or delete such a part, all dimensionings, texts and processing symbols are also moved, rotated, mirrored or deleted. If you create a part as main part or sub-part depends on whether the part is an individual, independent part, or if it belongs to another part. For instance, an individual haunched plate in a detail drawing would, as an independent part, be created as a main part. If this haunched plate is welded to a steel beam in an assembly drawing, it would be a sub-part of an assembly part (consisting of beam and plate). *See also Main part, Sub-part.*

Part hatching
Hatching of a surface bordered by a closed polyline. All lines of the border need to belong to one part.

Part list
Multiple selection of parts. The aim is to process several parts in one step.

Part structure
Subdivision of the drawing into main parts and sub-parts. The part structure describes the logical composition of the real construction. It constitutes, for instance, the basis for the automatic Bill of Materials (BOM), the automatic creation of subtractions for part hatchings, and part overlaps (in the 2-D module).

Patterning
Special hatching of a surface, with arbitrary symbols instead of lines for the hatching.

Pixel graphic
Pictures constructed of a regular array of dots (pixels). Pixel graphics are created when you scan pictures. Common formats are BMP, JGP, GIF, PCX and TIF.

Point options
Functions for the specification of point coordinates. The functions are subordinated to all other functions, i.e. they can be activated within other functions.

Point specification
See Point options.

Polyline
A polyline is a structurally coherent (i.e. continuous and uninterrupted) succession of individual lines, circular, elliptical, hyperbolic or parabolic curves. The end point of the previous line coincides with the start point of the next line.

Projects
A means to effectively arrange, file and archive drawings and parts. Projects can be real working projects, they can however also be used to file, e.g., all drawings created in one year, by one specific department, by one specific customer etc. Projects can in turn contain individual sub-projects.

Project-independent
A project created by the computer. All drawings are automatically assigned to the project "Project-independent", even if you assign a drawing to a specific project. This project named "Project-independent" cannot be deleted. Therefore you can delete all projects that you created yourself, without losing any drawings: Even after deletion, they still exist in the project named "Project-independent".

Redo
Reverses the previous UNDO.

Reference point
The point to which a later subsequent transformation or point specification refers. *See also Fitting point.*

Relative coordinates
The coordinates refer to a point in the drawing that is not necessarily the origin of the coordinate system. For many functions, this reference point is automatically defined by the last set point. Reference points can however also be defined by the user.

Report Manager
Program for the creation of individual Bills of Materials (BOMs) based on the HiCAD part/assembly structure or the HELiOS product structure. The BOM provides information on the individual components of a product, as well as on their quantity, names, item numbers etc. You can also set filters, define different sort criteria or export to Excel or HTML format.

Representation
HiCAD offers several options for the representation of the 3-D parts in a drawing, e.g. shaded, as glass model, with or without hidden edges).

Screen coordinate system (Screen CS)
Coordinate system the XY-plane of which is the screen.

Screen menu
The area(s) of the graphic screen containing the icons, symbols and texts for function calls.

Screen plane
The XY-plane of the screen coordinate system.

Screenshot
Saving of the current screen or screen details as pixel graphic.

Sheet area
The area in which you can create and name production views. These can be normal views, sectional views, detail views, drawing frames etc. You can also create several sheet areas, e.g. one for the quotation drawing, one for the production drawing etc. *See also Model area.*

Sketches
3-D part with free edges that lies in a plane. Sketches can be used, e.g., for the derivation of extruded solids, revolved solids, bores, subtractions or C-edge sweeps. The name "3-D SKETCH" can be changed if desired. After creating a sketch the appropriate sketch functions are displayed. The sketch is drawn in the active processing plane or, if no processing plane is available, in the XY-plane of the active coordinate system. Before drawing a closed contour in the sketch, you can use the "New plane, …" functions to define or activate a different processing plane.

Snap circle
If snap mode is active, points within the radius of the displayed circle are automatically detected.

Snap mode
If this mode is active, points can be specified directly, without calling a point option. *See also Autopilot.*

Snap radius
Determines the size of the snap circle.

Solid primitives
Basic 3-D bodies such as cuboids, cylinders, spheres etc.

Status bar
Displays information about the current program status, i.e. the name of the active part, active view and active coordinate system, etc.

String
A sequence of alphanumeric and special characters.

Structure browser
Graphic representation of the part structure, allowing you to process the structure.

Sub-part
A part that is subordinated to another part (Main part or Sub-part). *See also Part structure and Main part.*

System variable
HiCAD variables with fixed values.

Unit of measurement
Preset unit of measurement (mm, cm, etc.) for all value entries.

Undo
Reverses the previous action.

Transformation
Umbrella term for positional and size changes of drawing parts, such as displacements, rotations, mirrorings, enlargements, downsizings etc.

Trim
Limits or shortens lines with respect to given conditions.

User-specific variable
Variables which have been defined by the user and to which, depending on the type, a numerical value or a character string has been assigned.

Value entry
Input of numbers, arithmetic expressions and variables.

Variables memory
Function enabling a rapid definition and saving of variables.

Variant
You use Variant Technology to create “Variants”. The creation of 2-D or 3-D variants makes sense if you frequently draw similar parts, differing only in their dimensions, the number of bores, etc. After calling a variant you only need to specify the required parameters (width, height, number of bores etc.), and HiCAD then draws the part automatically. Parts created with Variants (or Macros) can be simple sheets or plates, but also complex assemblies.

Vector graphic
An image the data structure of which consists of vectors. In this type of graphic you can delete, move, change or scale the individual graphic elements. Vector graphics usually require less memory than pixel graphics.

View All
Displays the entire drawing on the screen.

Window
The division of the screen into several processing windows, enabling you to process various details of a drawing.

Workflow
Defined working process enabling you to define the steps that documents, folders, projects or parts need to run through in a company, and to control their handling. Such a process can, for instance, be the way from the concept to the realisation of a product. The number of the test steps that an object has to pass through can be defined via the Workflow chart. Please note that a Workflow chart can only be defined for objects of the object type 2-D/3-D drawing, part, folder, project, general document, DXF, IGES or NC program. Workflows can be pre-defined or user-defined. The access rights are linked to the object names and determine whether an object can be viewed, created, changed or deleted. You can use the "User“ function to define the rights of the individual users.

Zoom
Functions enabling you to enlarge or downsize the objects on the screen.

ZTL file
A 2-D drawing file with the extension ZTL, used in older HiCAD versions.

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