eCabinet

User Guide

Version 4.2 SP2

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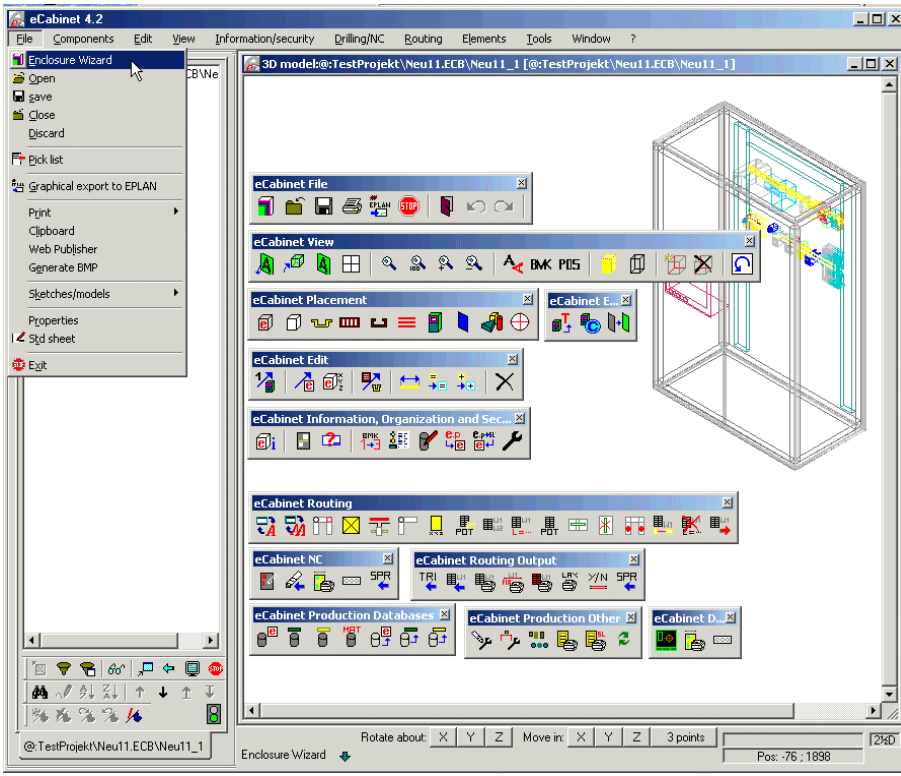
1 eCabinet system principles

1.1 Starting eCabinet

To start **eCabinet**, double-click its program icon. The eCabinet program window opens with a menu bar and a number of toolbars.



1.2 eCabinet toolbars

eCabinet works entirely through toolbar commands. The toolbars group commands by function:

- File (project and drawing commands)
- Placement (components)
- View (display options)
- Edit (modify components)
- External Enclosure Templates (create and use cabinet templates)
- Information, Organization and Security (device, database and parts list options)

1.2.1 eCabinet File toolbar

Commands on the File toolbar, from left to right:

Create and edit project

Close project drawing

Save project drawing

Print project drawing

Graphical DXF export to EPLAN

Exit eCabinet

Exit command

Undo last change

Restore last undone change

1.2.2 eCabinet Placement toolbar

Commands on the Placement toolbar, from left to right:

	Open device browser and place device
	Insert universal part
	Place mounting rail
	Place duct
	Place cable clamp rail
	Place busbar system
	Select and place enclosure from database
	Place mounting panel
	Place user-defined component
	Place user-defined holes

1.2.3 eCabinet View toolbar

Commands on the View toolbar, from left to right:

	Activate component and rotate into view, hiding other components
	Switch to ISO view
	Activate component
	Switch to enclosure views
	Enable zoom (select a detail for viewing)
	Disable zoom (display entire drawing)
	Zoom in (150%)
	Zoom out (75%)
	Reposition text
	Display device tag labels
	Display item number labels
	Enable shading
	Disable shading
	Refresh hidden lines
	Delete hidden lines
	Rotate view

1.2.4 eCabinet Edit toolbar

Commands on the Edit toolbar, from left to right:

	Copy enclosure
	Change device placement
	Move device
	Move enclosures, ducts and rails
	Change length of ducts and rails
	Change plant designation
	Change location designation
	Delete

1.2.5 eCabinet External Enclosure Templates toolbar

Commands on the External Enclosures toolbar, from left to right:

Create enclosure template from drawing

Insert Rittal cabinet from Cadenas

Define mounting area

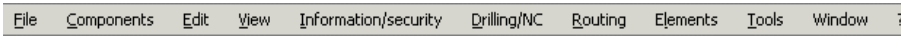
1.2.6 eCabinet Information, Organization and Security toolbar

Commands on the Information, Organization and Security toolbar, from left to right:

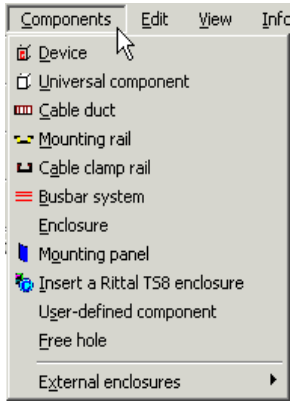
	Device information
	Place blocking surface
	Check drawing: Find surplus devices
	Renumber components
	Import parts data
	Update components in databases
	Update project parts list
	Change project assignment
	Parameter settings

1.3 eCabinet menu bar

All eCabinet commands can also be accessed in a menu bar. If a different menu bar is active when you start eCabinet, click the Options menu, choose Menu Bar, select ‘eCabinet’ and click [OK] to install the eCabinet menu bar.

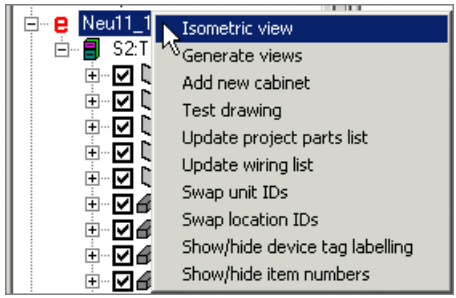


The menus are arranged to match the toolbars, and the commands on each menu match those on the corresponding toolbar.



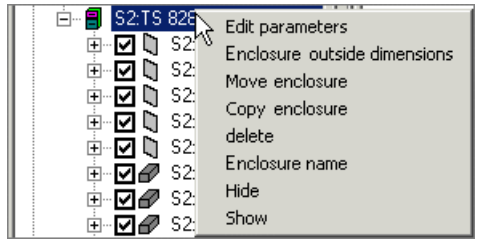
1.4 eCabinet Navigator node

The eCabinet project is shown in the Navigator under a separate node with a red **e** icon. Right-click the icon to open a context menu with the main view and project commands.

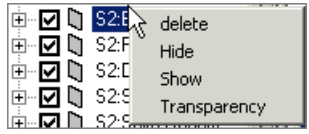


The component nodes further down the tree also have context menus:

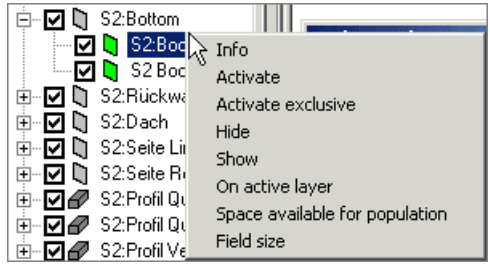
Enclosure node



Cabinet component node



Component layer node



1.5 Selecting commands in eCabinet

Various ways of selecting commands are described in this manual. Commands selected on a toolbar or in a main menu are shown as follows:

File **Enclo ure Wizard**	eCabinet **En losure Wizard**
Left: Command on a toolbar The name of the toolbar is shown above the icon, to the right of which is the name of the command.	*Right: Command on a menu* The name of the menu bar is shown above a symbol for a menu bar, to the right of which is the name of the command.

Commands selected from secondary parts of the user interface such as dialogues are shown as follows:

Enclosure Wizard | Create a new enclosur project

Enclosure Wizard | Create new project

Selecting commands in the Navigator

Commands that are only available on context menus in the Navigator are shown with a Navigator symbol:

Enclosure node | Cabinet dimensions

Using freehand symbols to select commands

A freehand symbol is a way of selecting a command by moving the mouse in a specific figure with the left mouse button pressed down. If a commands can additionally be selected with a freehand symbol, the figure is shown:

Enclosure Wizard

2 Creating and editing projects and drawings

A project in eCabinet contains all data used in designing and laying out an enclosure. Each project is kept in a project directory identified by the name of the project with an .ECB extension. A project directory contains an EPLAN parts database, project control files, an EPLAN parts list file (EPLAN.PBF), an EPLAN administration list, and eCabinet drawings.

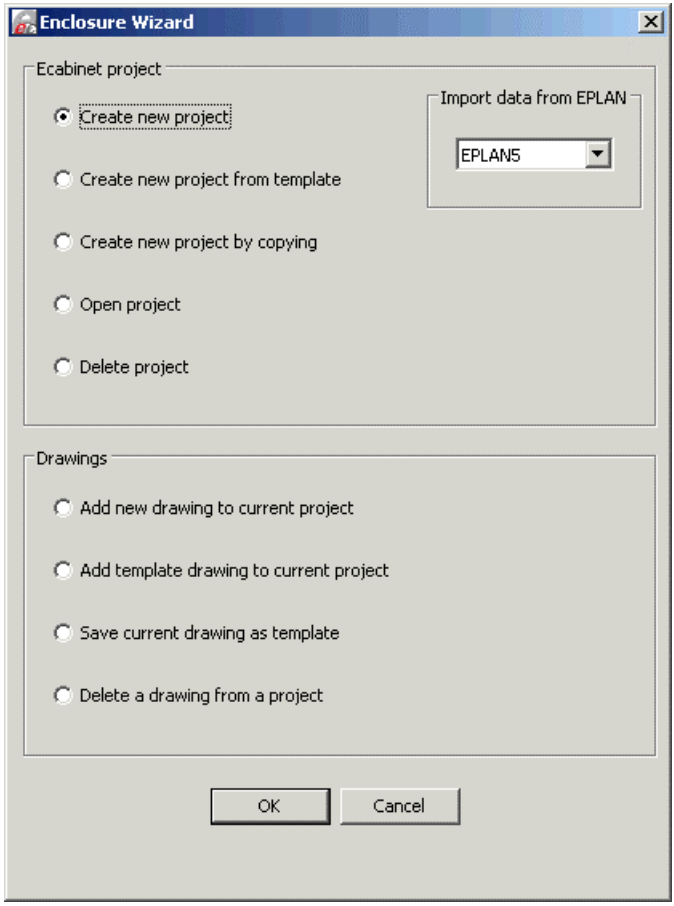
Only one project can be open at a time. It is not possible to have multiple projects open for editing.

File	eCabinet
Enclo ure Wizard	**File | Enclosure Wizard**

Freehand symbol

Enclosure Wizard

The Enclosure Wizard contains all commands needed for creating and editing eCabinet projects and for adding drawings and enclosures to them.


Enclosure Wizard
Ecabinet project
Create new project
Create new project from template
Create new project by copying
Open project
Delete project
Import data from EPLAN
EPLAN5
Drawings
Add new drawing to current project
Add template drawing to current project
Save current drawing as template
Delete a drawing from a project
OK
Cancel

2.1 eCabinet projects

2.1.1 Creating a new enclosure project

Enclosure Wizard | Create new project

File | Enclosure Wizard | Create new project

You can create an enclosure project inside an existing EPLAN project or on its own without reference to an EPLAN project.

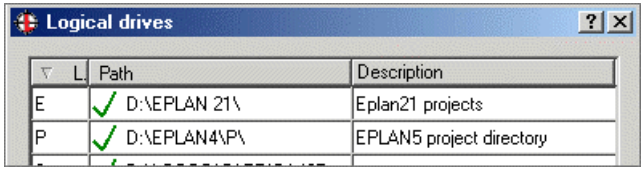
2.1.1.1 Creating a project inside an EPLAN project

In the 'Import data from EPLAN' box, select the EPLAN version you want to import EPLAN component data from:

- EPLAN 5
- EPLAN 21

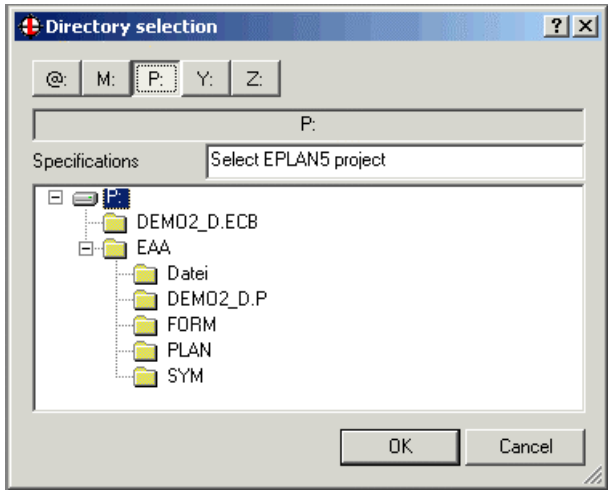
Select a project drive

A logical drive (e.g. 'P') should be assigned in eCabinet so that it points to the EPLAN project directory.

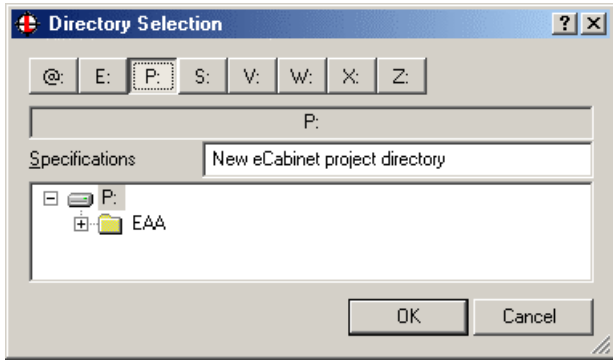


The data in the EPLAN project directory can be imported into the enclosure project.

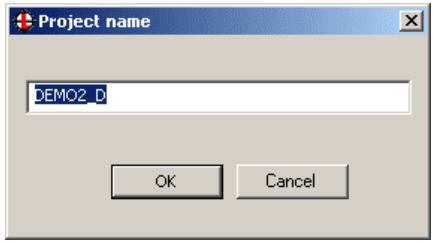
Choose the drive that is assigned to the EPLAN project directory:



Click [OK] to go on to the directory selection, where you choose the directory in which the eCabinet project will be stored.

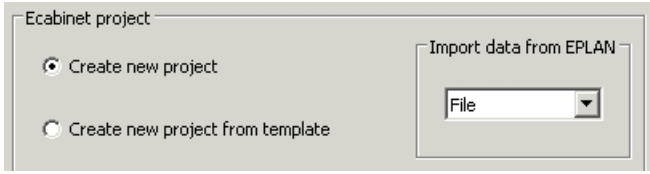


Click [OK] to go on and name your project. You can either accept the name of the EPLAN project or type a name of your own.



2.1.1.2 Creating a project from a file

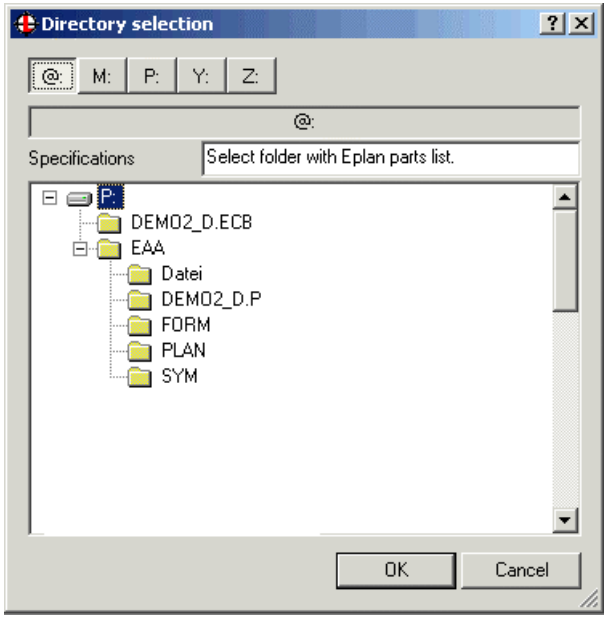
In the 'Import data from EPLAN' box, choose 'File'. Then click [OK] to create a project that is based on a parts list file but is not related to an existing EPLAN project.



Select folder containing parts list

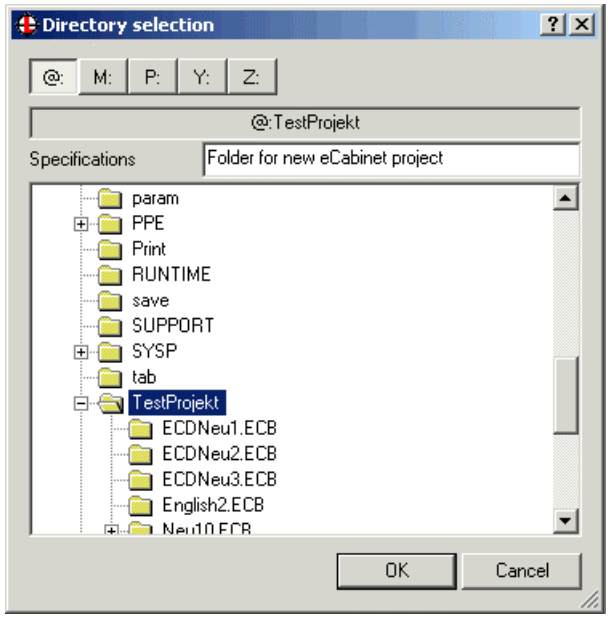
Select the directory containing the source data – that is, a parts list (EPLAN.PBF or FORM.ASC) with data for enclosure components.

If you want to create a new folder, right-click to open the context menu and choose 'Create new folder'.



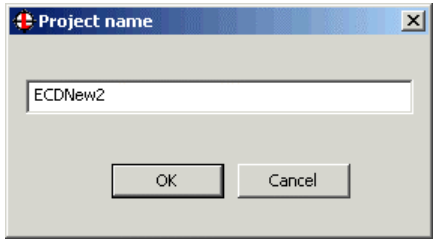
Select folder for eCabinet project

Choose a folder in which the custom eCabinet project will be stored.



Name the project

In the Project Name dialogue, type a name for the enclosure project.

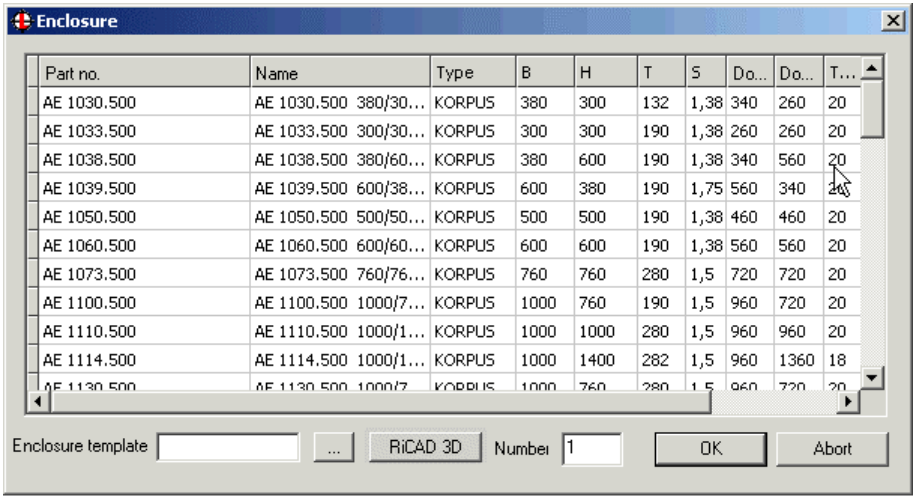


Click [OK] to save the named project in the selected folder. In the example shown, a subdirectory ECDNeu2.ECB is created in the selected folder, TestProjekt.

A drawing is automatically created and a dialogue opens for you to select a basic enclosure from the database.

2.1.1.3 Selecting an enclosure from the database

Once you have named the project, the enclosure models in the eCabinet database are displayed in a list.



In this window, you can:

- Select a standard enclosure as the base unit for your drawing.
- Select a cabinet from a template drawing.
- Specify how many copies of the selected enclosure you want to insert in your drawing.

If you decide not to place an enclosure in your drawing, click [Cancel].

Selecting from the database

Select an enclosure in the list. If you want to insert two or more cabinets of the same type, enter the number in the Number box.

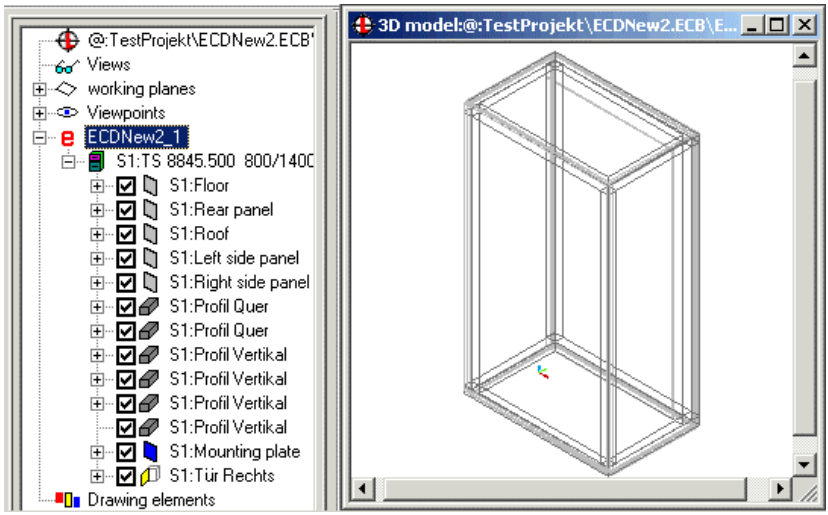
Selecting an enclosure template

Click the button next to the Cabinet Template box to open a file selection dialogue and select an enclosure drawing you have stored as a template for use as a standard enclosure model in addition to those in the database.

The name of the selected drawing is copied into the box.

Placement point

You are next asked to specify a placement point for inserting the enclosure in your drawing. The cabinet is placed with its rear bottom left corner at the specified point and displayed to fill the window in the 3D model.



With this enclosure as your starting point, you can now go on to add drawing elements and equipment.

2.1.2 Creating a new enclosure project from a template

Enclosure wizard | Create new project fr m template

File | Enclosure wizard | Create new proj ct from template

This command creates a new project directly from a selected template drawing. The procedure is the same as for creating a new project in an existing EPLAN project or from a file, except that there is no selecting a basic enclosure from the database.

- Select the project type and then an EPLAN project or parts list file.
- Select a folder to create the project in.
- Name the project.
- Instead of the database browser, a file selection dialogue opens for you to select a template drawing. Select the cabinet you want.
- Specify a placement point for inserting the cabinet in your drawing.

2.1.3 Creating a new enclosure project by copying

Enclosure wizard | Create new project by copying

File | Enclosure wizard | Create new proj ct by copying

This command copies an existing enclosure project. It copies all eCabinet data from the source project to a new EPLAN project or new parts list.

The procedure depends on whether you choose to copy the data from a file or from an EPLAN project.

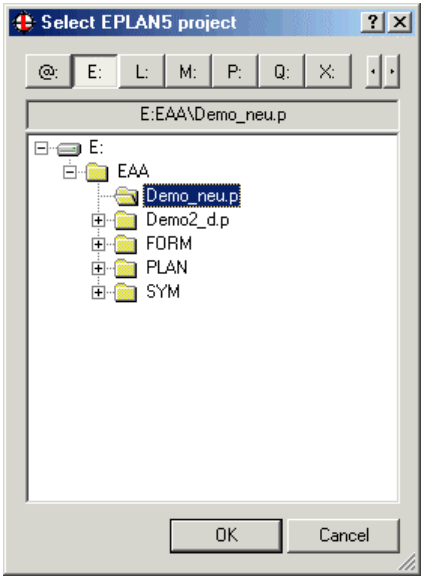
Copying EPLAN data from a file

- Select the folder containing the EPLAN parts list (EPLAN.PBF or FORM.ASC).
- Specify a folder to create the project in.
- Name the project.
- Select an existing eCabinet project to copy.
- A new project is created in the specified folder and with the specified name.

Copying data from EPLAN 5/EPLAN 21

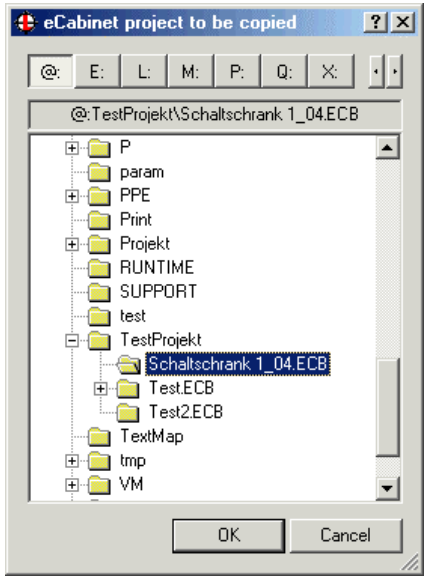
Select an EPLAN project

First, select the EPLAN 5 project from which you are going to copy data to an enclosure project.



Select an enclosure source project

Next, select an existing enclosure project directory to copy (enclosure project directories can be recognized by the .ECB extension):



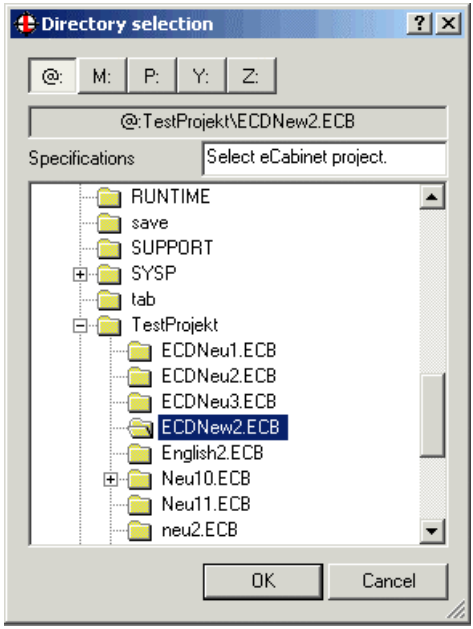
The drawings in the copied enclosure project are opened in the new EPLAN 5/21 target project.

2.1.4 Opening an existing enclosure project

Enclosure wizard | Open project

File | Enclosure wizard | Open project

Select the project you want:



Click [OK] to confirm your selection and load the project drawing.

2.1.5 Deleting an existing enclosure project

Enclosure wizard | Delete project

File | Enclosure wizard | Delete project

In the directory list, select the project you want to delete. Click [OK] and answer the confirmation prompt to delete the project together with all associated drawings and system files.

2.2 eCabinet drawings

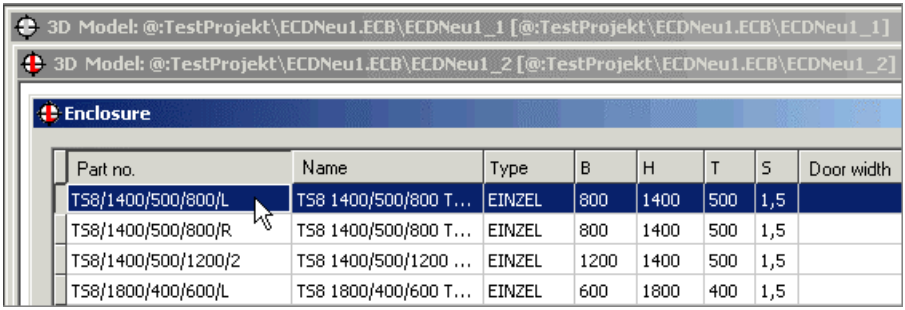
2.2.1 Adding a new drawing to a current project

This command adds a drawing to an existing enclosure project, for when you want an enclosure project to include more than one drawing. First, be sure that the existing project is already loaded.

Enclosure wizard | Add new drawing to current project

File | Enclosure wizard | Add new drawing to current project

The new drawing is given the name of the project followed by a sequential number such as _2. The enclosures in the database are then listed for selection.



Part no.	Name	Type	B	H	T	S	Door width
TS8/1400/500/800/L	TS8 1400/500/800 T...	EINZEL	800	1400	500	1,5	
TS8/1400/500/800/R	TS8 1400/500/800 T...	EINZEL	800	1400	500	1,5	
TS8/1400/500/1200/2	TS8 1400/500/1200 ...	EINZEL	1200	1400	500	1,5	
TS8/1800/400/600/L	TS8 1800/400/600 T...	EINZEL	600	1800	400	1,5	

Select an enclosure model and specify a placement point to insert the cabinet in your drawing.

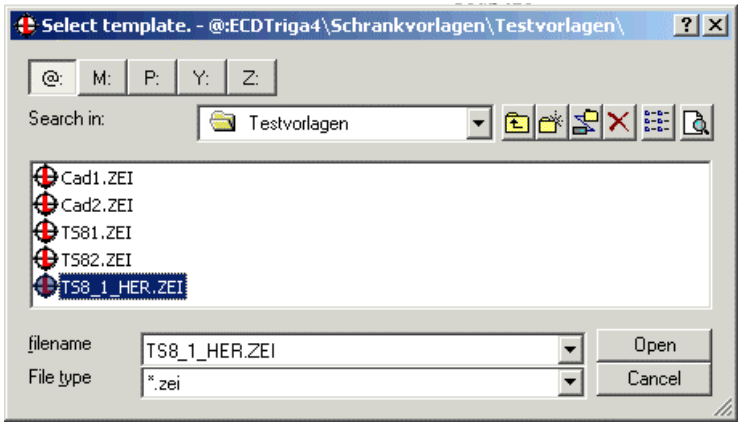
The additional cabinet drawings will be opened automatically the next time you open your project.

2.2.2 Adding a template drawing to the current project

You can add project drawings to your project from template drawings.

	Enclosure wizard | Add template drawing to current project
	File | Enclosure wizard | Add template drawing to current project

Select the drawing you want in the templates directory.



Click [Open] to open the template drawing in the current project, rename it and save it to the project directory.

2.2.3 Saving a current drawing as a template

This command saves the active drawing in the currently loaded project as a template.

For example, you could save a drawing containing a partly fitted enclosure as a template for use as a starting point in other projects.

Enclosure wizard | Save current drawing as template

File | Enclosure wizard | Save current drawing as template

Choose the directory you want and enter a file name. Click [Save] to save the drawing in the selected directory.

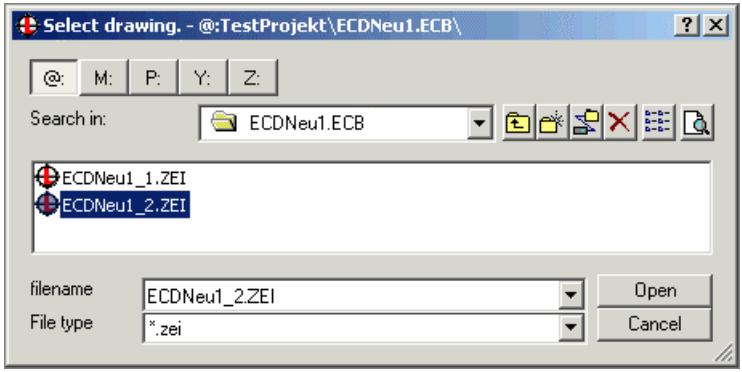
2.2.4 Deleting a drawing from an enclosure project

This command removes a drawing from a project that is not currently loaded. If you select the command when a project is loaded, you are prompted to close it first.

Enclosure wizard | Delete drawing from a project

File | Enclosure wizard | Delete drawing from a project

The drawing files in the selected project are listed.

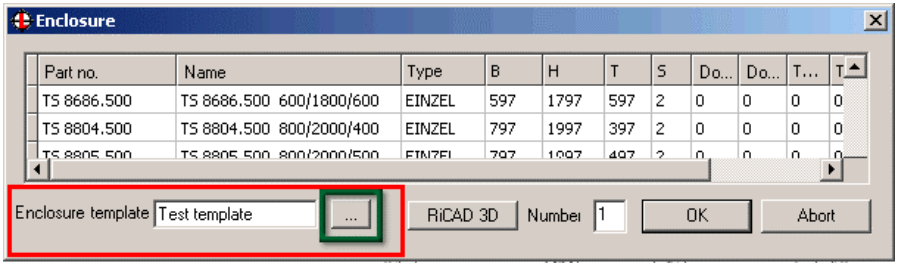


Click [Open] and answer the confirmation prompt to delete the selected drawing from the project directory and to remove all reference to it in the project file.

2.3 External enclosure templates

2.3.1 Creating an enclosure template from a drawing

When placing enclosures, you can either choose a model from the database or look in an enclosure template.



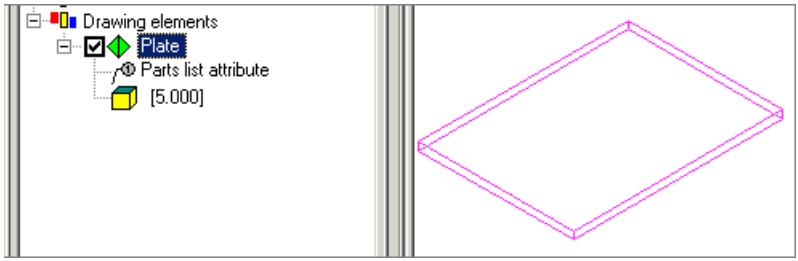
An enclosure template is a fully or partly fitted enclosure, or an enclosure component, stored in a separate drawing.

Saving a sketch or a drawing as an enclosure template

An enclosure template does not belong to a project; instead, it is a separate template that can be incorporated into a project.

In a sketch or drawing, draw an enclosure (or insert one from a database or another template) or a component such as a mounting panel. The cabinet or panel can be unpopulated or it can be fitted with other components.

Alternatively, you can import a model using a 3D interface such as SAT.

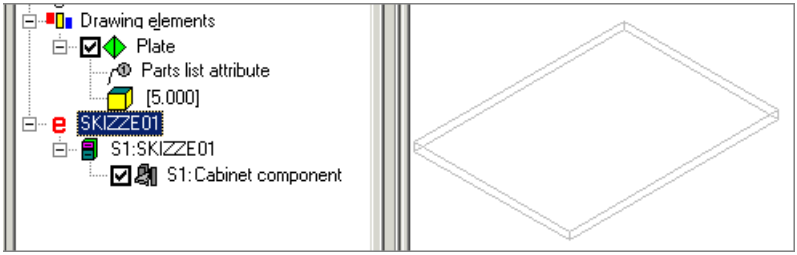


Now create a template from your sketch or drawing:

External enclosure templates

Create template from drawing or sketch

In the Navigator, the drawing is displayed under an eCabinet node as an enclosure drawing containing a cabinet component:



On this component, specify one or more mounting panels (see **Specifying mounting panels in enclosure templates**).

Save the drawing in the directory @:\ECD Triga4\Schrankvorlagen.

Now load a project. To use the template, incorporate it into your project, go to the Enclosure Wizard menu and choose **Add template drawing to current project**.

Enclosure wizard | Add template drawing to current project

You can now use the template when you place an enclosure in your drawing.

Place

Select and place enclosure from database

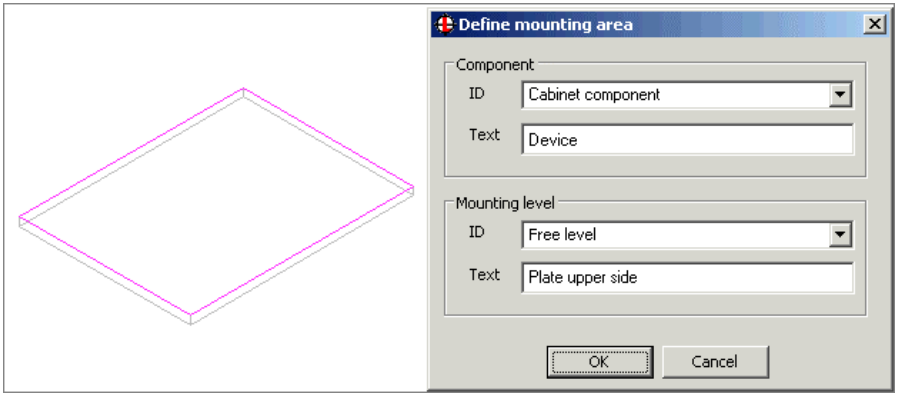
2.3.2 Specifying mounting panels in enclosure templates

A mounting panel is an area for mounting other components. Specifying a mounting panel turns a graphical surface into a logical surface that is integrated into the eCabinet hierarchy. It is then available for manipulation by special eCabinet activation, display and placement commands.

External Enclosure Templates

Define mounting area

[Ctrl]-click to pick the surface you want. The surface is highlighted and you can name the component and the surface in a dialogue.



ID/Text (Component)

ID Selection of pre-set names designating the template component as a whole. An ID must be chosen or at least left as the default ID of 'Cabinet component' because it is used to identify the part in the components database and elsewhere.

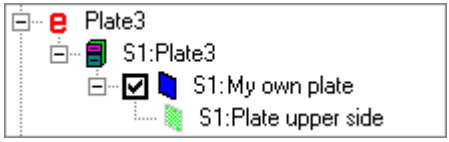
Text The user-defined name entered here is shown in the Navigator and as a designation in drawing views.

ID/Text (Mounting Level)

ID This identifies the surface of the template component on which devices can be mounted. An ID must be chosen or at least left as the default ID of 'Cabinet component' because it is used to identify the part in the components database and elsewhere.

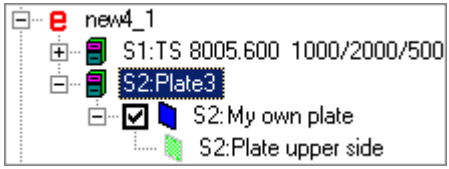
Text The user-defined name entered here is shown in the Navigator and as a designation in drawing views.

Click [OK] to assign the specifications to the template component. The ID/text hierarchy is shown in the Navigator:



In this example, an enclosure template **Platte3** ('Panel 3') is made up of a mounting panel **Meine Platte** ('My panel') with one active mounting level, **Platte oben** ('Panel top').

The names are also shown when the template is inserted in a project:



Editing a mounting panel

The items under 'Component' can be modified at any time. Right-click the component and choose 'Define component' on the context menu.

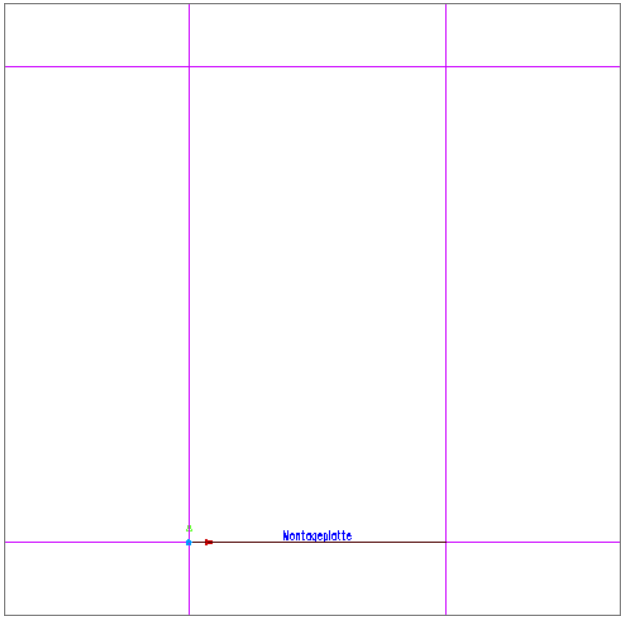
Choose the 'Move' item on the menu if you want to move the mounting panel by specifying an offset to the right, upward or forward.

Space available for population

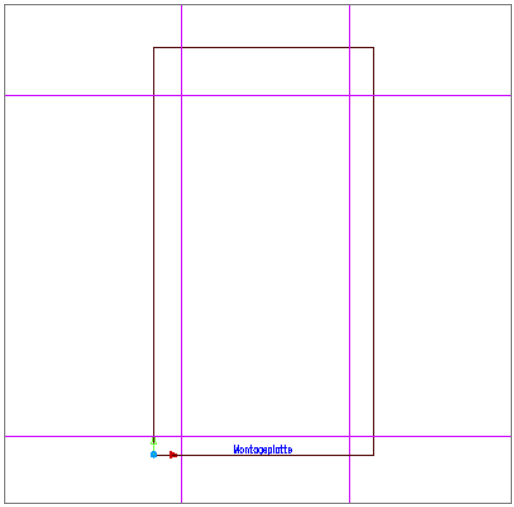
For a given area (such as a mounting panel or a door), the ‘space available for population’ is the area on which components can be mounted. This is normally the entire area (such as the entire mounting panel).

Use this command if you want to change this so reduce the area available for mounting components.

First, make certain that only the affected area is active. On the component’s Navigator node, choose ‘Space available for population’. Two vertical and two horizontal lines are displayed, representing the space available.



Tap one of the lines to move it and adjust the available space:



Field size

The ‘field size’ determines the size of a field (such as a mounting panel) from a manufacturing point of view. It does not matter whether the mounting panel or other component is placed in the drilling machine with or without fitted parts. The field size tells the machine the size of the component being processed.

The command works the same as ‘Space available for population’.

2.3.3 Inserting a Rittal TS8 enclosure

External Enclosure Templates

Insert Rittal TS8 cabinet from Cadenas

If a Cadenas standard part system with Rittal TS8 enclosures is installed on your computer, you can use these as enclosure templates.

Click the toolbar button to start Cadenas and select and place the enclosure you want.

2.4 Using custom 3D models as enclosure templates

Users often want to supplement the enclosures provided in the eCabinet database and in RiCAD 3D with their own enclosure models.

Custom enclosures can be added manually to the eCabinet database, but this method is not suitable if you want to incorporate a detailed enclosure model from another system.

For such cases, eCabinet has commands for importing models and integrating them with the eCabinet database.

Requirements

To import a custom model into eCabinet, you need:

- eCabinet 4.2
- 3D interface
- 3D model of the enclosure

Custom enclosures models can be imported into eCabinet version 4.2 or later. The models must be in a supported 3D data format.

Formats supported by eCabinet 4.2:

- SAT (interface included in eCabinet 4.2)
- DWG (interface included in eCabinet 4.2)
- STEP (interface has to be purchased separately)

Note: The available information on an imported enclosure depends on the software used to create it. Only what is exported (into a file) will later be available for use in eCabinet.

2.4.1 Importing a 3D enclosure model

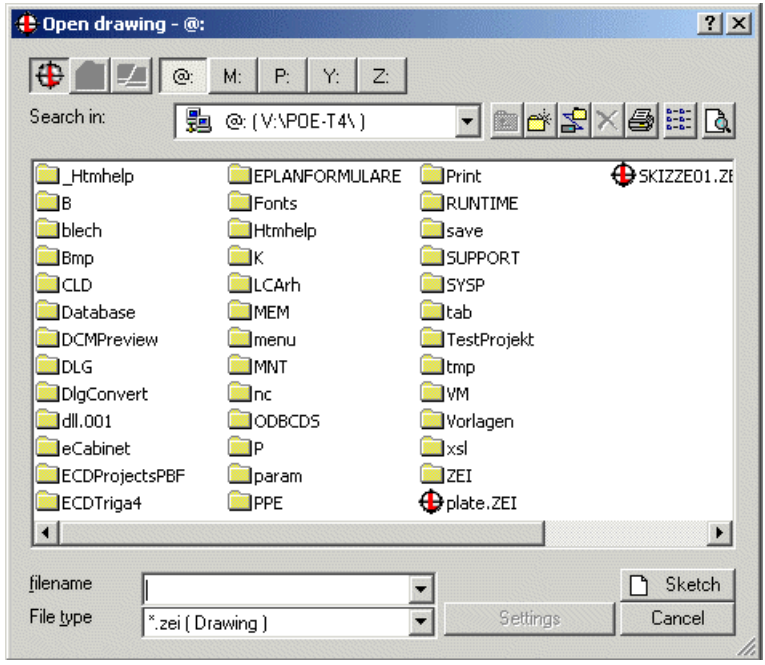
Copy the 3D model file to your drawing directory, Z. You may want to create a subdirectory for the purpose (e.g. '3D models').

Example:

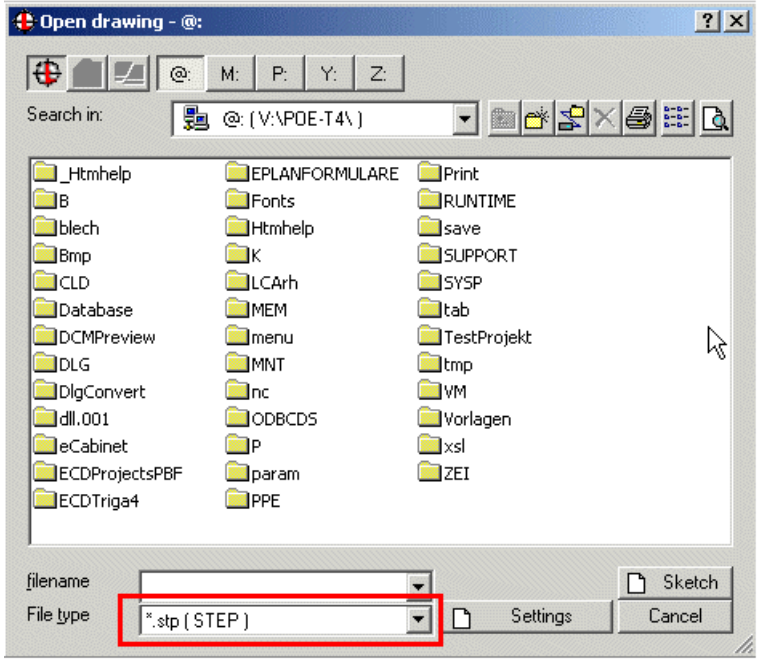
This example uses a file called Schaltschrank1.stp, containing a model of an enclosure in STEP format.

The file is in a subdirectory, 'Z:3D Modelle\'.

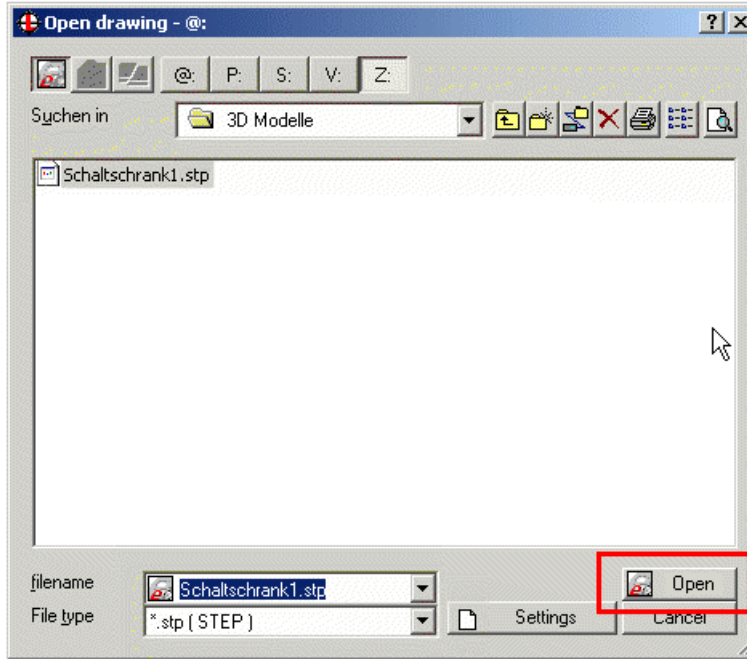
Start eCabinet and, on the menu, choose FILE - OPEN.



Change the file type to '*.stp (STEP)'.

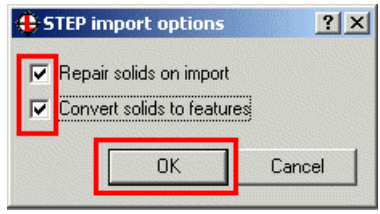


Double-click the ‘3D Modelle’ directory to open it, then select the STEP file you want.



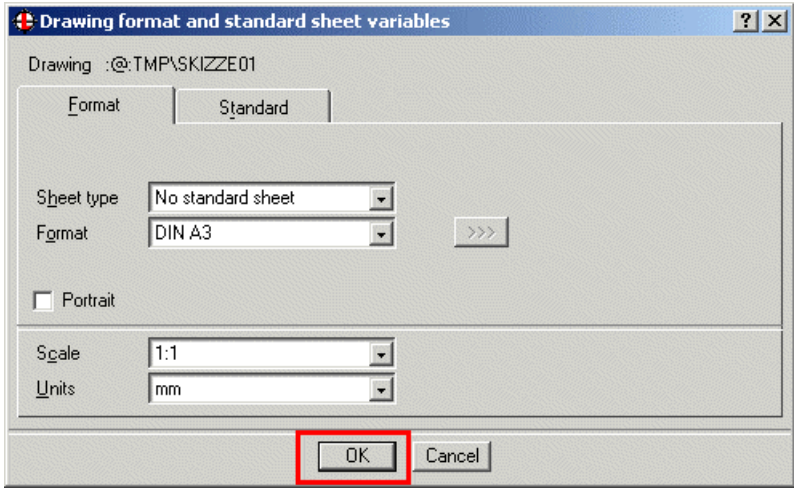
Choose [Open].

In the next dialogue, called ‘Import options’, select both options.



Click [OK].

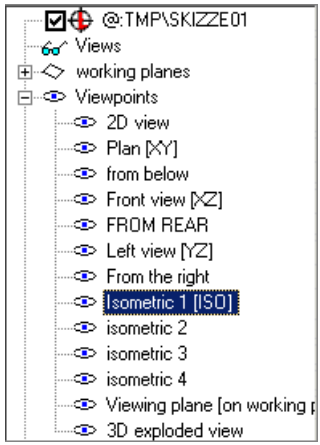
In the standard sheet dialogue parameters, just click [OK] without making any changes.

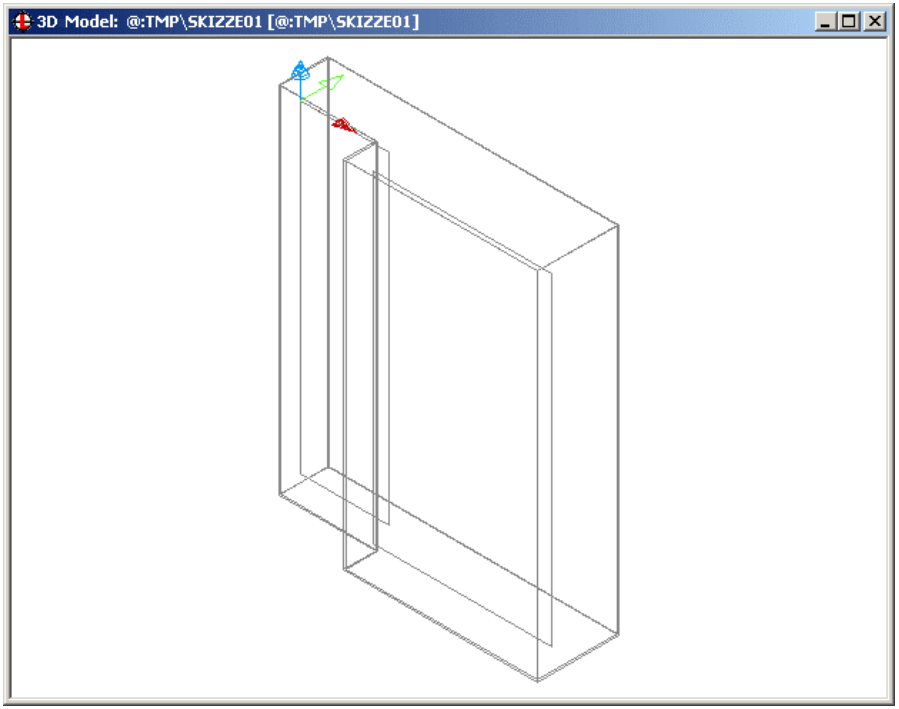


eCabinet will now create a new drawing and import the enclosure model. This may take some time depending on the size, detail and scope of the model.

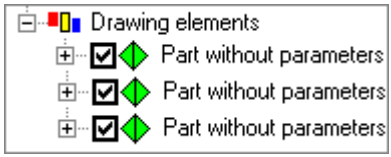
When 'Select function' is displayed on the status bar, eCabinet has finished opening the model.

In the Navigator, double-click 'Isometric 1 (ISO)' under 'Viewpoints' to display the enclosure.



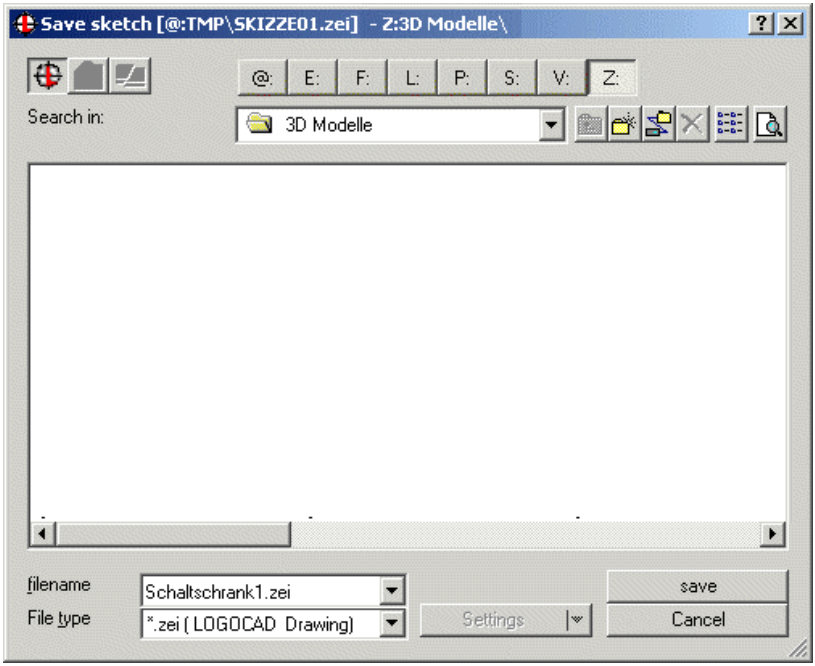


The Navigator also shows the individual solids that make up the enclosure. In this example, these are three parameterless components (an enclosure and two mounting panels).



eCabinet has now finished importing the custom enclosure.

Save the drawing (e.g. in ‘Z:\3D Modelle’).



Note: This is not yet a finished eCabinet enclosure. The drawing saved here is only an intermediate stage.

2.4.2 Correcting the orientation of the enclosure

When they are first imported, all solids have the same orientation in 3D space as they had in the program used to create the model.

You will probably need to correct the enclosure's orientation for eCabinet.

In the example, the enclosure is rotated 180° from where it should be. The rear panel and mounting panel are currently at the front.

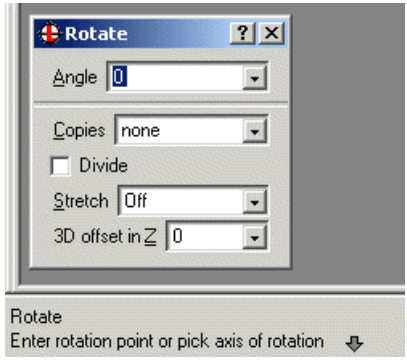
This must be corrected before going on any further.

In this example, the enclosure must be rotated through 180° about the Z-axis.

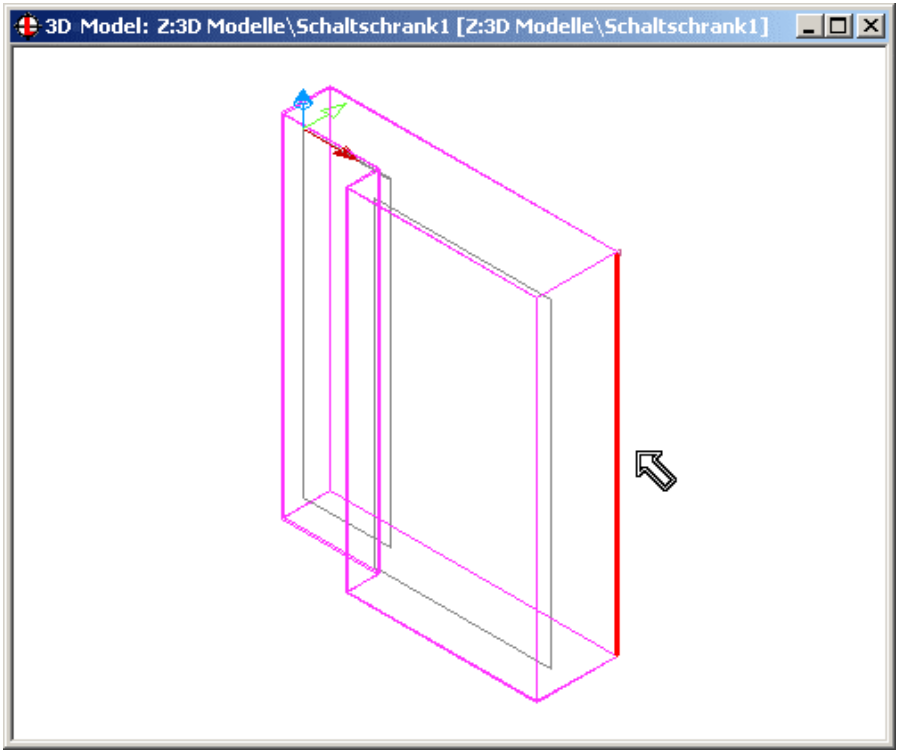
Note: A different model might require different treatment. The corrections needed depend on the model and the conditions that applied when it was exported from the program it was made in.

Choose the Rotate command on the menu by selecting Edit – Sketches/Models – Rotate.

Edit | Sketches/Models | Rotate



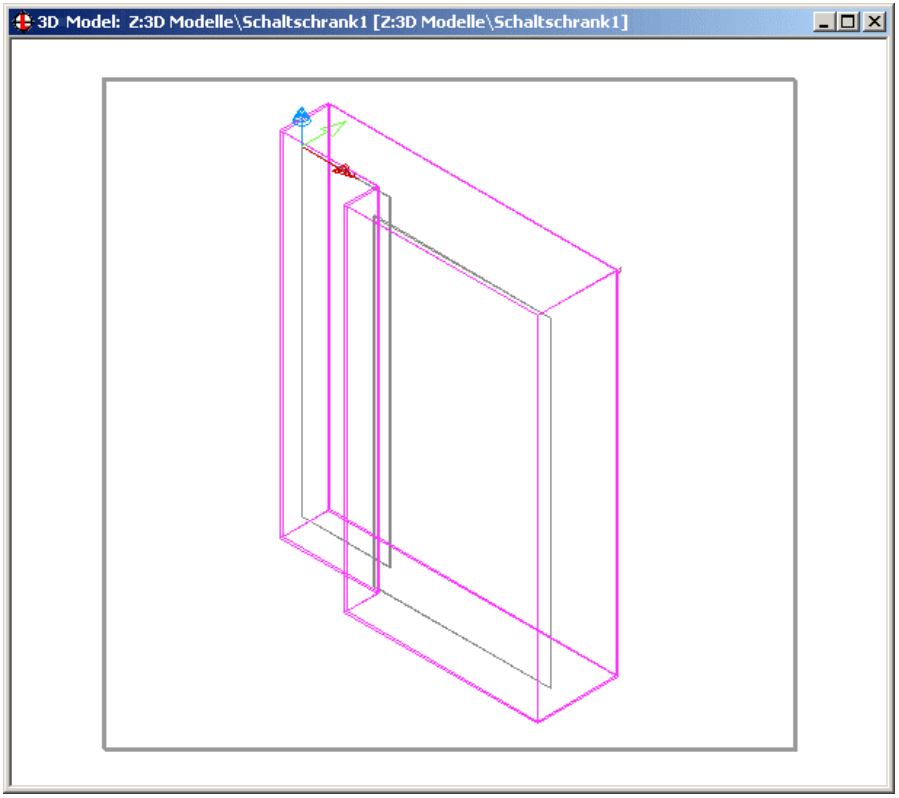
First, select the rotation axis. A suitable axis in our example is the right-hand vertical edge of the enclosure. To select this edge, press and hold down the CTRL key, point at the edge and click once.



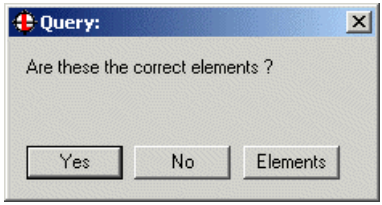
Type '180' for the rotation angle in this example.

Now to select the elements to be rotated. These are the two solids in our example (the enclosure is one solid and the two mounting panels are two instances of the other).

Using rectangle selection mode, select the elements you want.

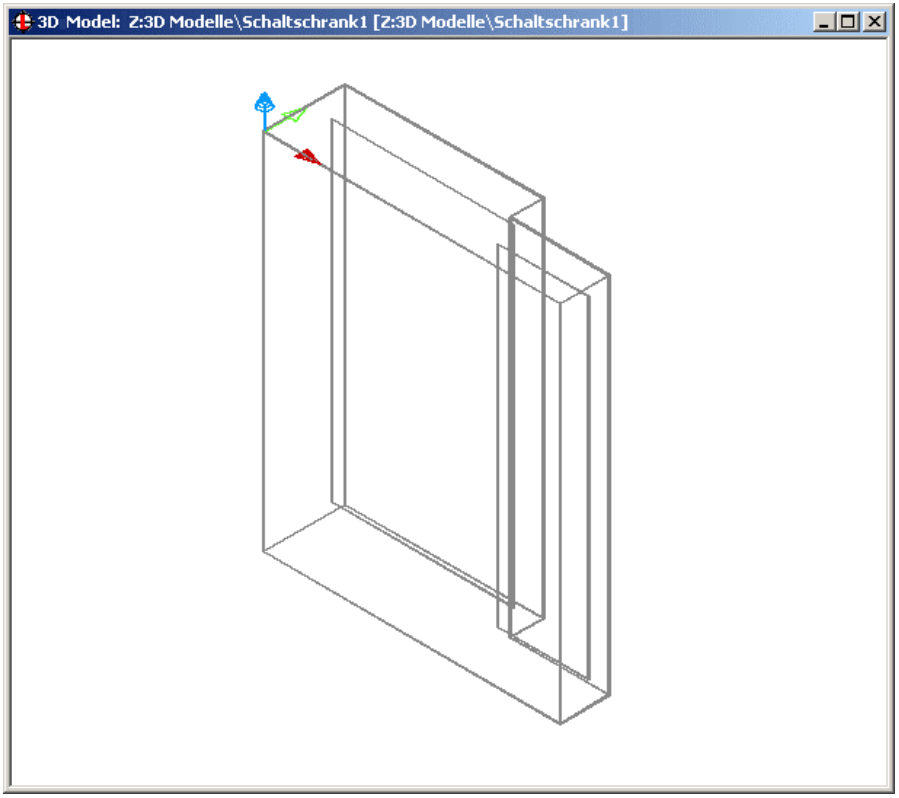


Click inside the rectangle to confirm your selection.



If you are happy with the selection, choose [Yes].

All solids in the example are rotated to their proper orientation in 3D space.



Save the drawing in its current form.

2.4.3 Creating an enclosure template from an imported model

The various solids in the drawing are not yet enclosure components: they are just solids without any logical information.

The drawing itself is not yet an eCabinet enclosure drawing.

We will now do something about this.

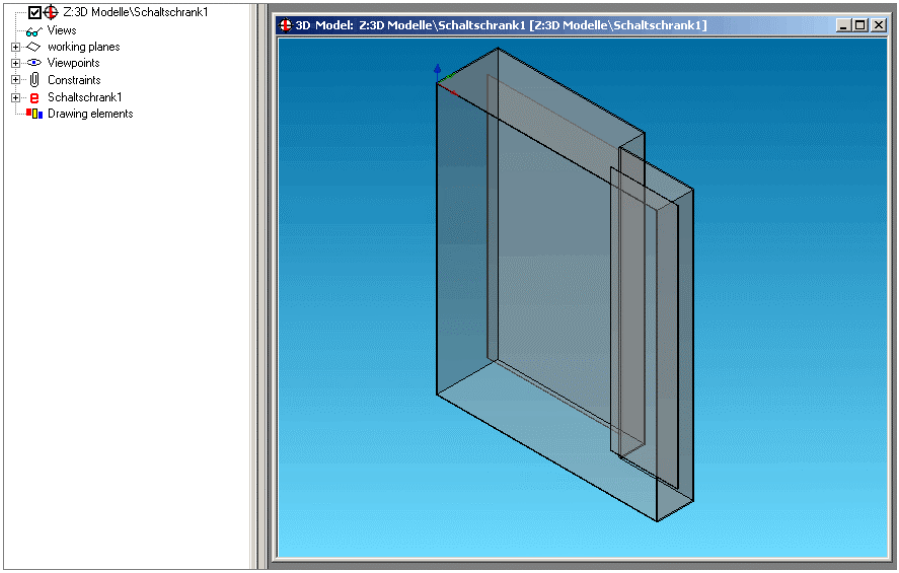
Note: In the next step, the drawing will be made into an enclosure drawing and all solids in it will become components of the enclosure. Because this step cannot be undone, you should first save your drawing in its current form so that you can come back to this point at any time.

Choose the Create Template command.

External enclosure templates

Create template from drawing or sketch

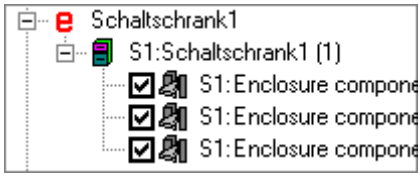
Components | External enclosures | Create template



The changes are displayed straight away in the drawing, with shading enabled. The separate solids are automatically made into enclosure components and so take on the transparency settings specified for such components in the eCabinet parameters.

The enclosure object hierarchy is automatically displayed in the Navigator (you may need to refresh the Navigator to see it).

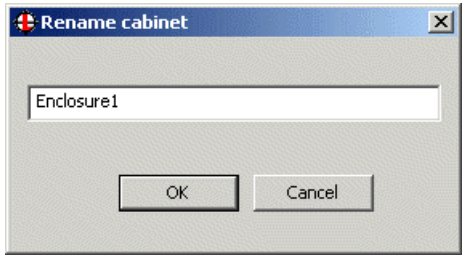
In our example, the result is the three enclosure components in ‘Schaltschrank1’.



2.4.4 Specifying enclosure components

First of all, you can rename the enclosure.

To do this, right-click the enclosure in the Navigator and choose ‘Cabinet – Name’.



Once you have changed the name, choose [OK].

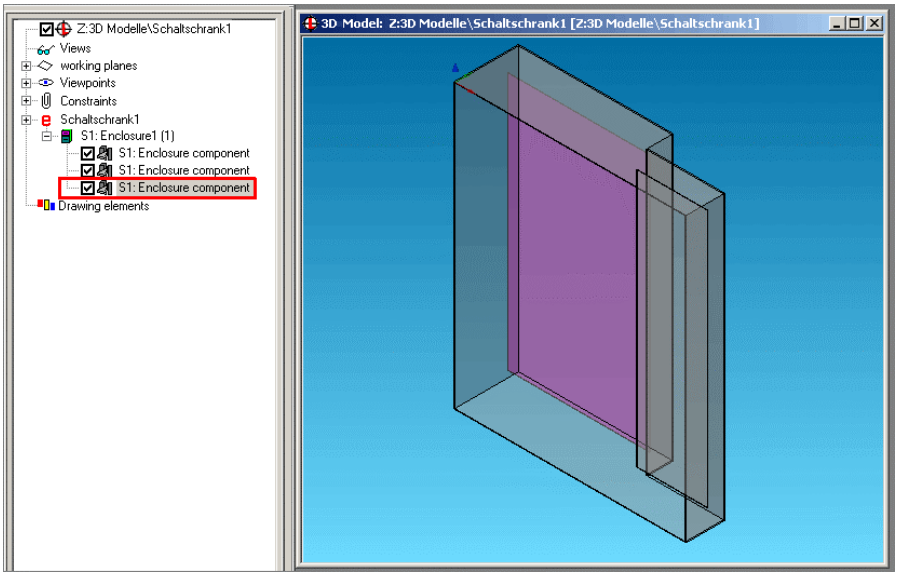
So far, three ‘general’ enclosure components are specified in the example.

In reality, they are an enclosure and two mounting panels.

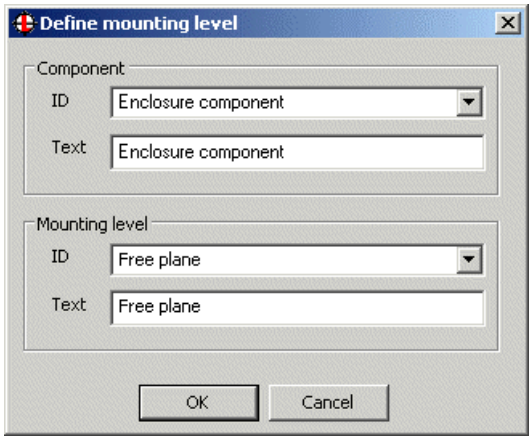
We will next designate the two mounting panels in the example.

Select the lower component in the Navigator.

Note: Selecting a component in the Navigator also selects the corresponding solid in the drawing.

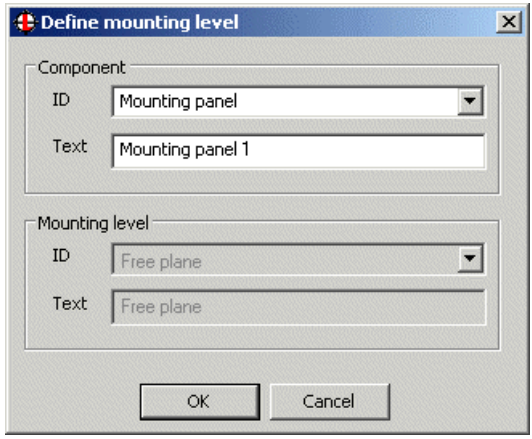


Right-click the component to open the context menu, and choose ‘Define component’.

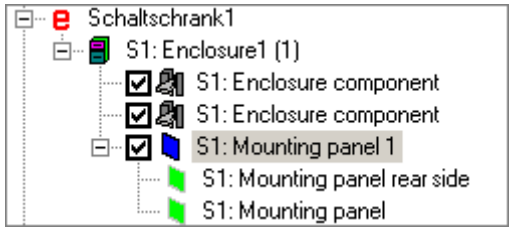


Under 'Mounting part ID', select the type of component – 'mounting panel' in our example.

You can also enter a descriptive name under which the component will be stored.



In the case of a mounting panel, the two surfaces available for population are designated automatically.

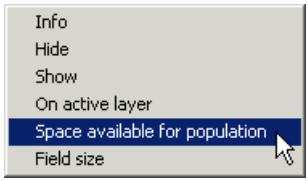


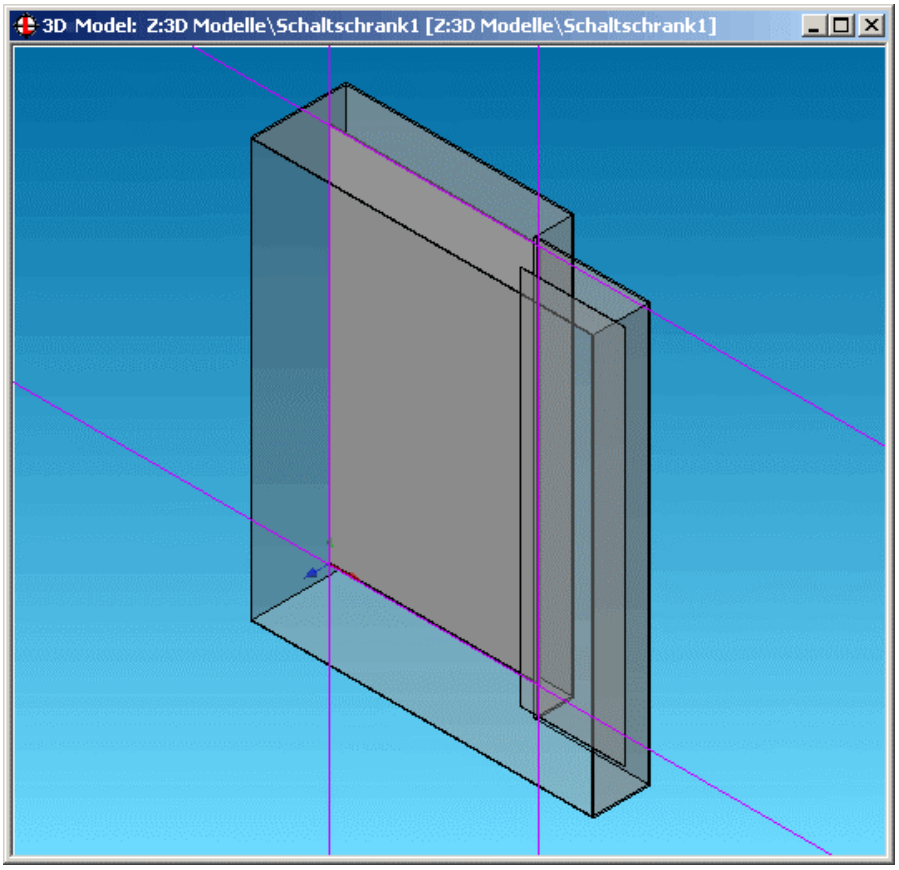
Note: Space available for population

For a given surface (like a mounting panel or a door), the 'space available for population' is the area where components can be mounted. This is normally the entire surface (for example an entire mounting panel).

Use this command if you want to change this and reduce the area available for mounting components.

Right-click the mounting panel in the Navigator and choose ‘Space available for population’.

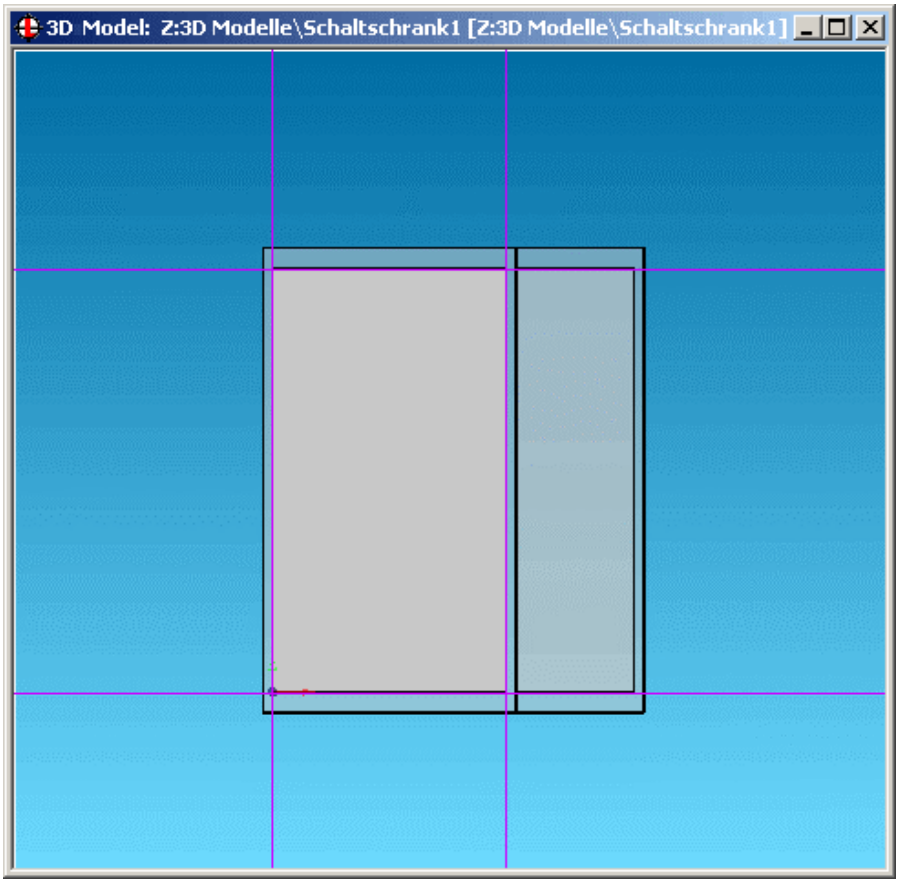




The straight lines at the top, bottom, left and right mark out the space available for population. To restrict the available space further, click one of the lines, move it and click again to fix it in its new location.

Tip:

Before you use this command, rotate the drawing to view the mounting panel from the front. This makes it easier to see what is going on as you work.



Note: Field size

The 'field size' determines the size of a routing field (such as a mounting panel) from a manufacturing point of view. It does not matter whether the mounting panel or other component is placed in the drilling machine with or without fitted parts. The field size tells the machine the size of the component being processed.

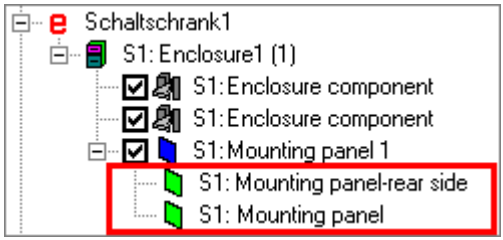
The command works the same as 'Space available for population'.

Right-click the mounting panel in the Navigator and choose 'Field size'.

Specify the field size as described above.

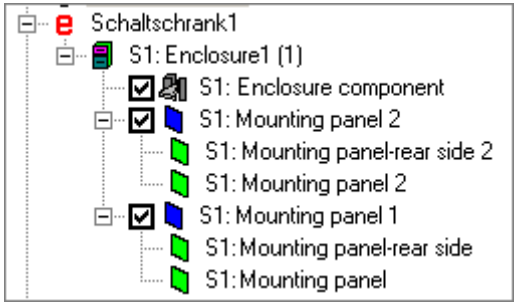
Once you have specified the space available for population and the field size, the component is fully defined and ready for use in eCabinet.

This is indicated in the Navigator by the black boxes around the symbols for the mounting panels.



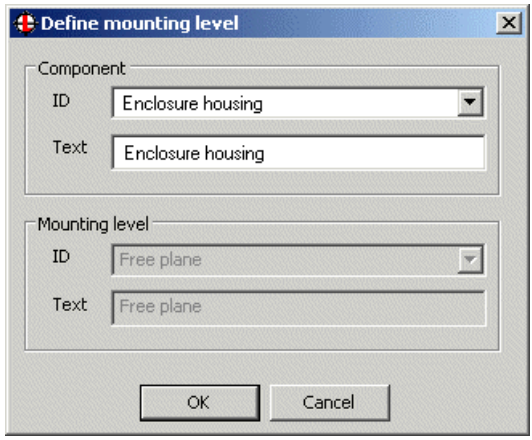
Now repeat for the second mounting panel in our example.

The result looks like this:



We will now deal with the enclosure.

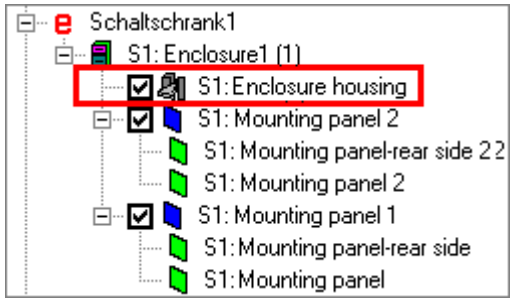
Right-click to open the context menu, and select ‘Define component’.



First, select ‘Enclosure’ for the type of component under Component ID.

Now type a description.

Choose [OK].



With an enclosure, unlike a mounting panel, the mounting levels are not automatically designated. The next step is to identify these manually.

Various parts of an enclosure (such as the top, side, rear and bottom panels) can be made available for mounting components.

Note: In our example, we will specify the right side panel and the top. The same procedure applies for all surfaces.

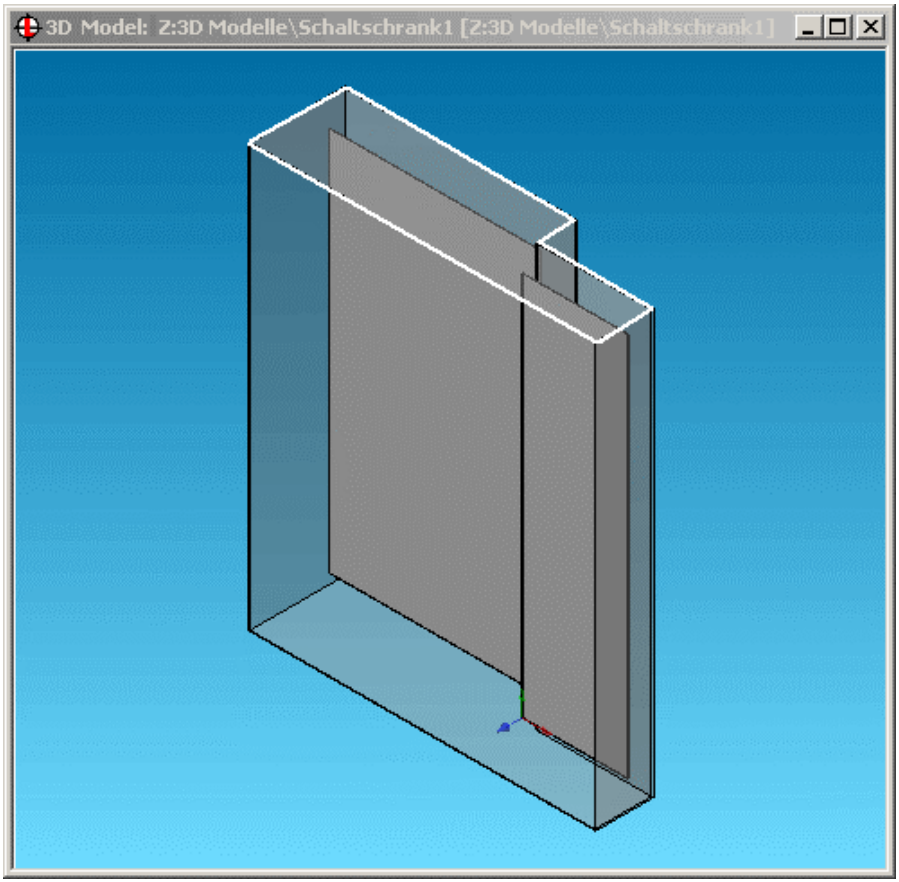
To specify mounting levels:

On the toolbar, select the Define Mounting Area command.

External enclosure templates
Define mounting area
Components | External enclosures | Defin : mounting area

Note: Surfaces must be visible – and so capable of being selected – for them to be made into mounting levels. If the surfaces you want are not visible, select an appropriate view of the enclosure before selecting this command.

Select the command and click the top panel of the **enclosure**.

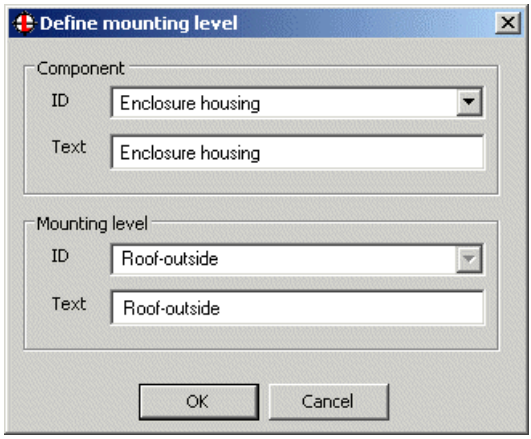


The panel is highlighted so you can verify your selection.

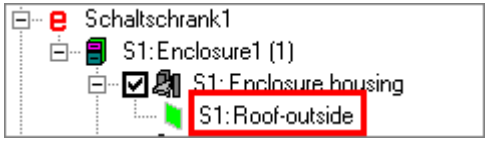
The component type has been specified earlier, which is why the Component ID section is already filled in. It also already has a description.

Note: The ID and description can be altered at this point if required.

Now specify the mounting level and type in a description for it. The description is displayed in the Navigator.

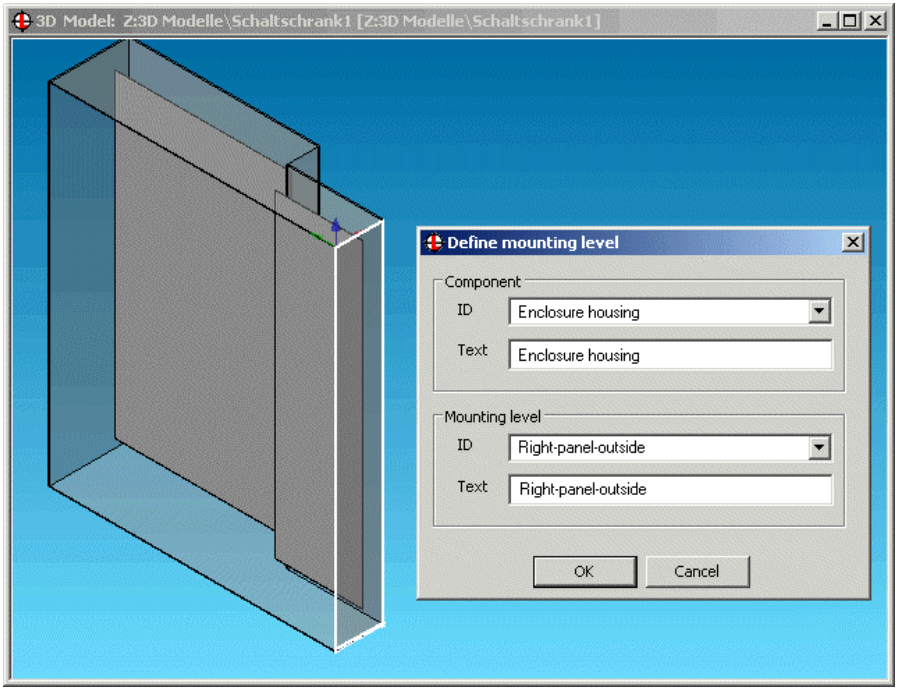


Choose [OK].

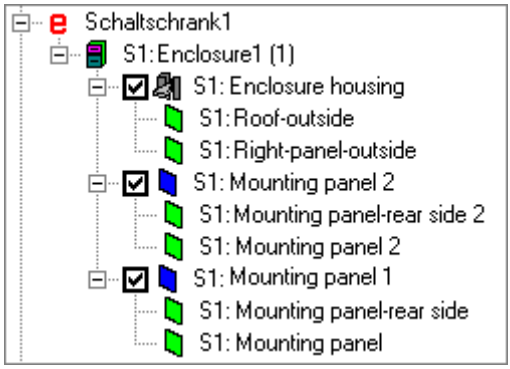


Specify the space available for population and the field size for the new mounting level, as described earlier.

Repeat for the right-hand side panel.



The result looks like this:



All required mounting levels are now specified.

2.4.5 Specifying an origin for the enclosure

So that the enclosure can be precisely placed in drawings, its append point for placement must be carefully specified.

The append point is specified as the origin of the enclosure drawing.

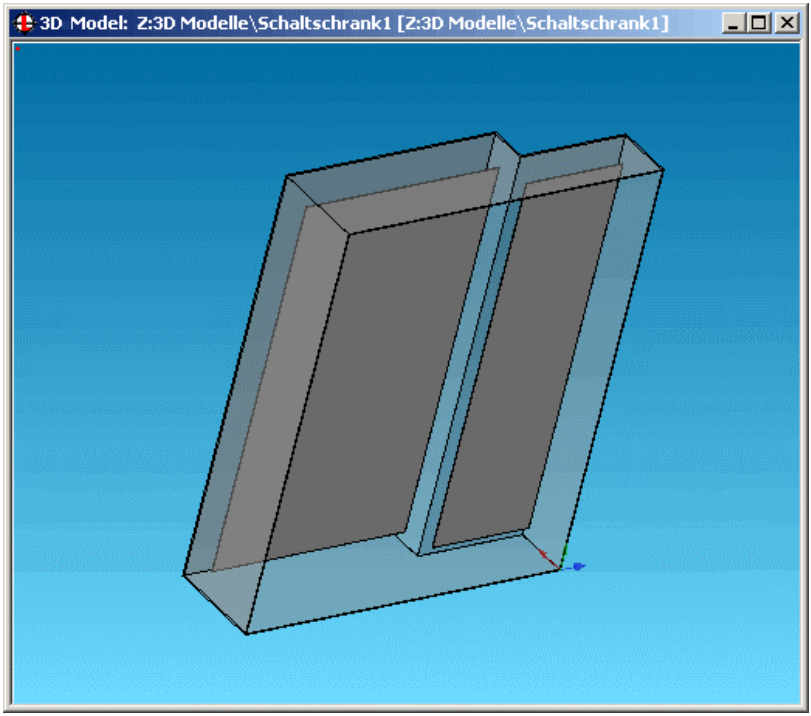
The append point of an enclosure should always be at the bottom left of the rear panel.

To specify the append point:

View

Rotate view

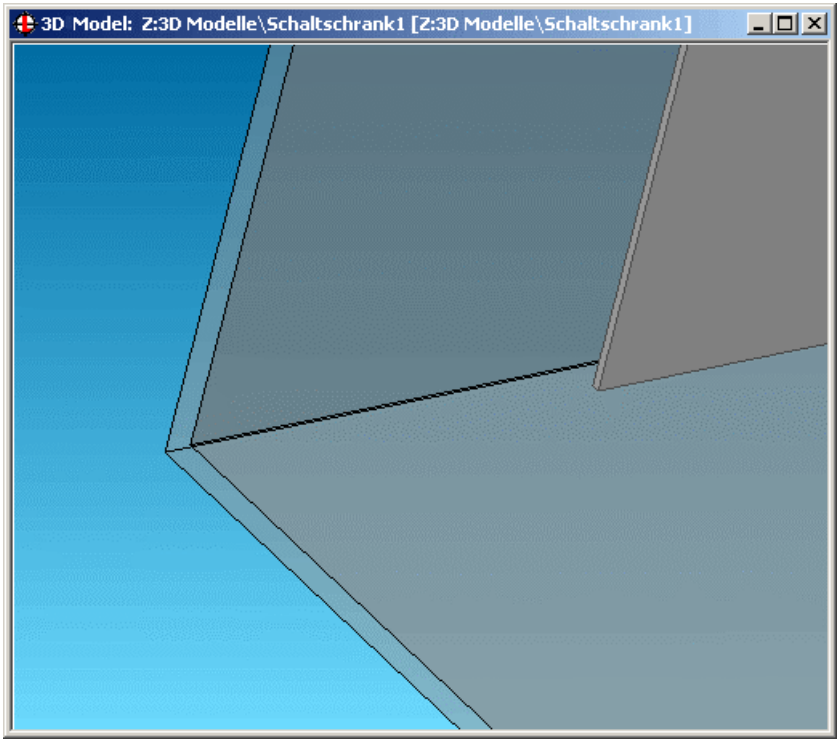
- Rotate the view so you can see the bottom left corner of the rear panel.



View

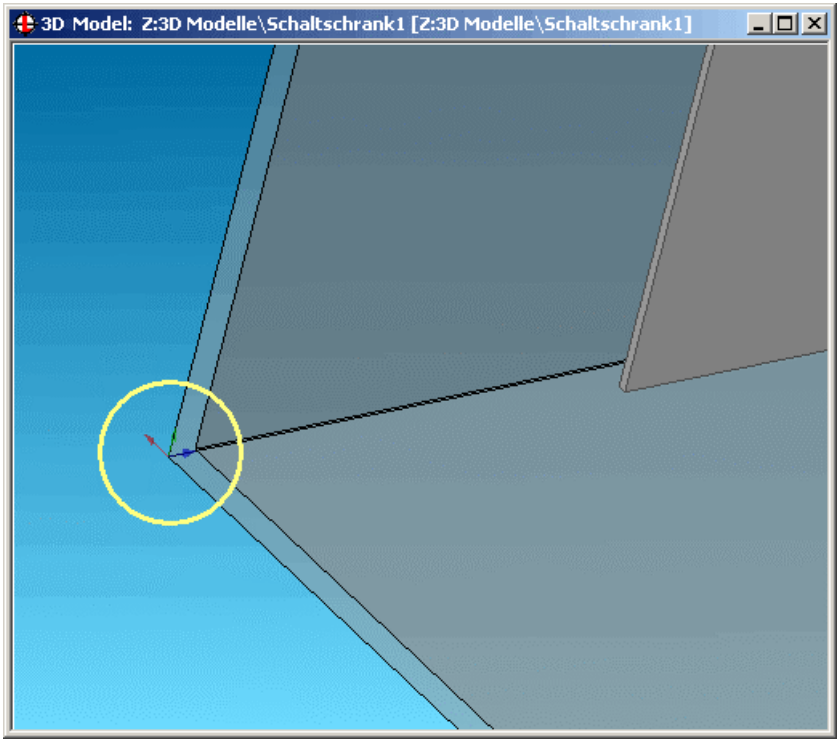
Zoom

- Zoom in on the corner you want.



- Select the corner by using point search (with no command active). The coordinate system is moved to the corner.

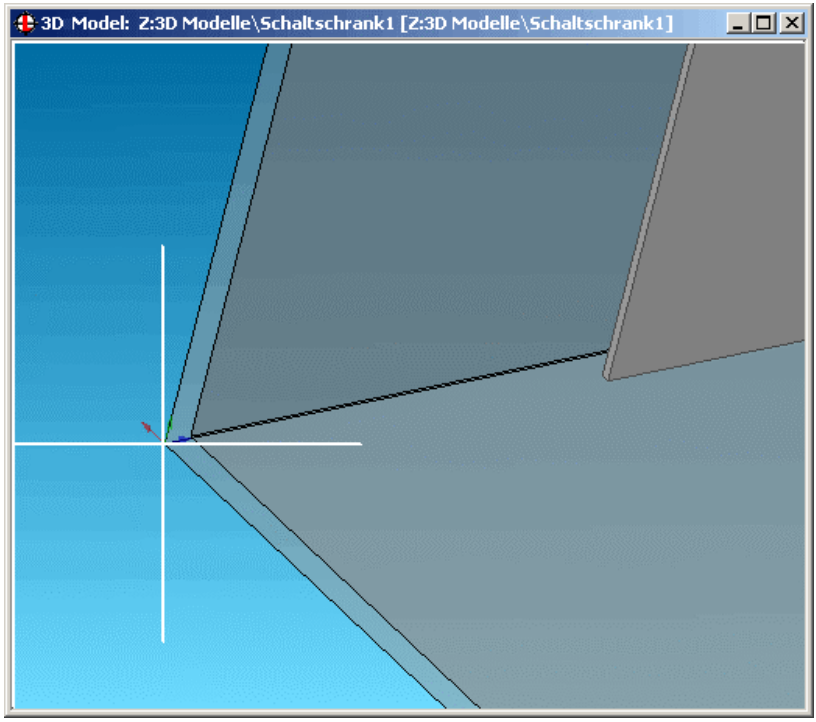
  Note: 'Point search' means press and hold the left mouse button until the point is selected.



Note:

If the coordinate cross will not move to the corner point, the default 'novice' mode is probably still enabled. The coordinate system cannot be moved manually in this mode. To disable novice mode, on the menu, select Tools – Options – Working Planes Auto/Manual. You will then be able to move the coordinate cross.

- On the menu, choose Elements | Drawing Origin.
- Use point search to select the corner you want.



- Press [RETURN] to confirm the selected origin.

2.4.6 Saving an enclosure as a template

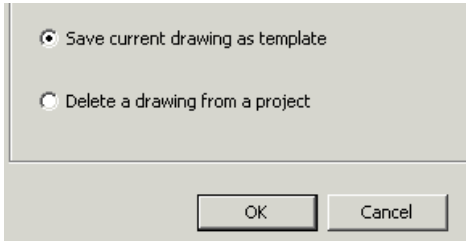
The enclosure is almost finished. The final step is to make it available to e-Cabinet for use as an enclosure template.

Enclosure wizard | Save current drawing as template

File | Enclosure wizard | Save current drawing as template

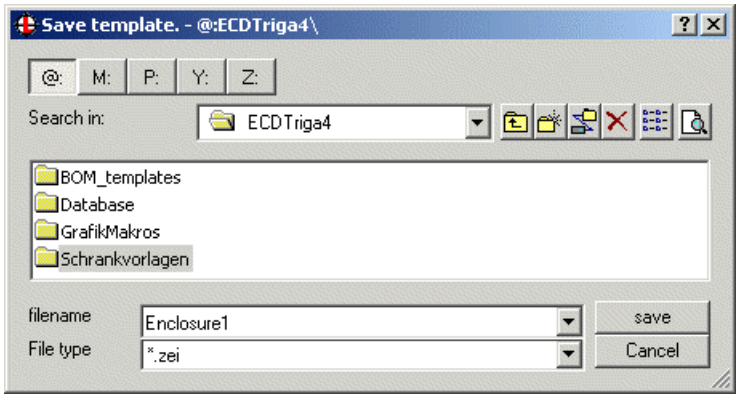
Open the enclosure wizard.

Select the ‘Save current drawing as template’ option.



Choose ‘OK’.

eCabinet automatically opens the appropriate directory. Type a name for the template drawing.

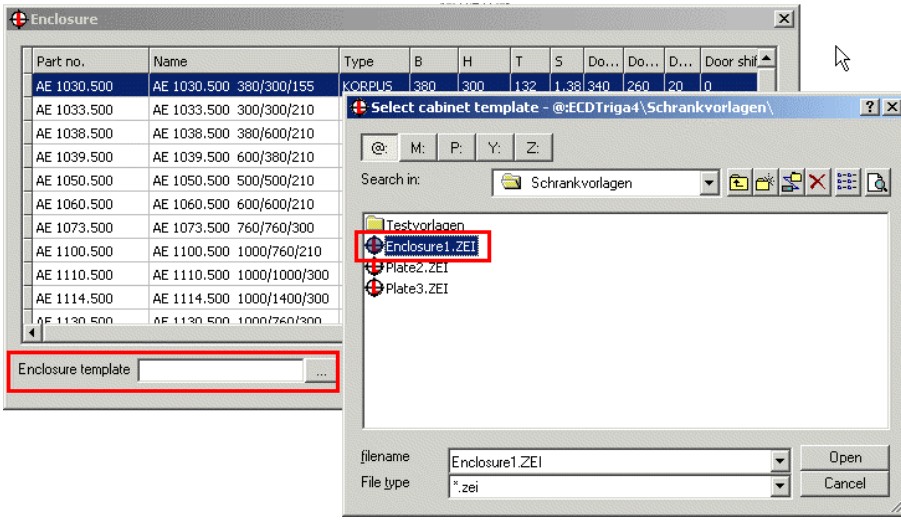


Choose [Save].

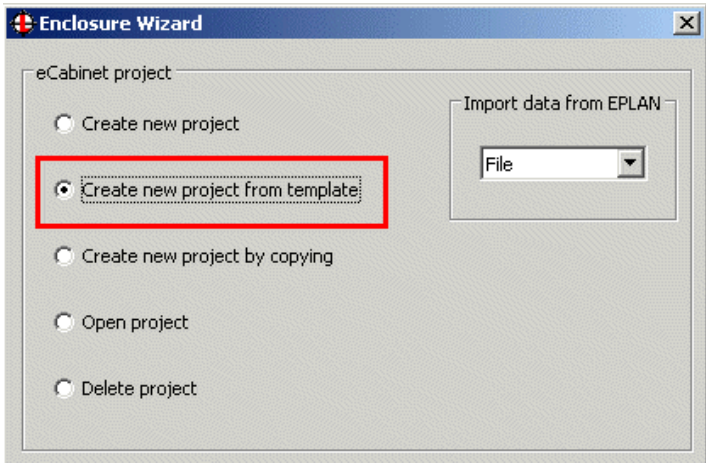
The new enclosure is immediately available for use in eCabinet projects.

The enclosure can be:

- Placed like a 'normal' enclosure using the 'Place enclosure' command.



- Used as a template for new projects.



2.5 RiCAD 3D enclosures

RiCAD 3D is a program provided by Rittal for integrating Rittal CAD data into a design drawing.

eCabinet is closely integrated with RiCAD 3D.

Initially, the TS8 and AE enclosure series from the RiCAD 3D catalogue can be used in eCabinet 4.2 Service Pack 1.

You can select an enclosure straight from the catalogue and insert it in an eCabinet project.

The Rittal accessories included in the RiCAD 3D catalogue cannot yet be incorporated into eCabinet.

The following RiCAD 3D enclosures can be used in eCabinet projects:

- TS8 modular enclosures
- TS8 corner enclosures
- TS8 electronics enclosures
- TS8 enclosures for modular front design
- AE compact enclosures

2.5.1 Installing RiCAD 3D

2.5.1.1 Running RiCAD 3D from CD

RiCAD 3D is supplied on a CD. There is no installation routine. The program does not need to be installed. You simply insert the CD and start work.

This has the advantage that you can insert and use the CD as needed.

The downside is that every user needs a CD or has to share one with others.

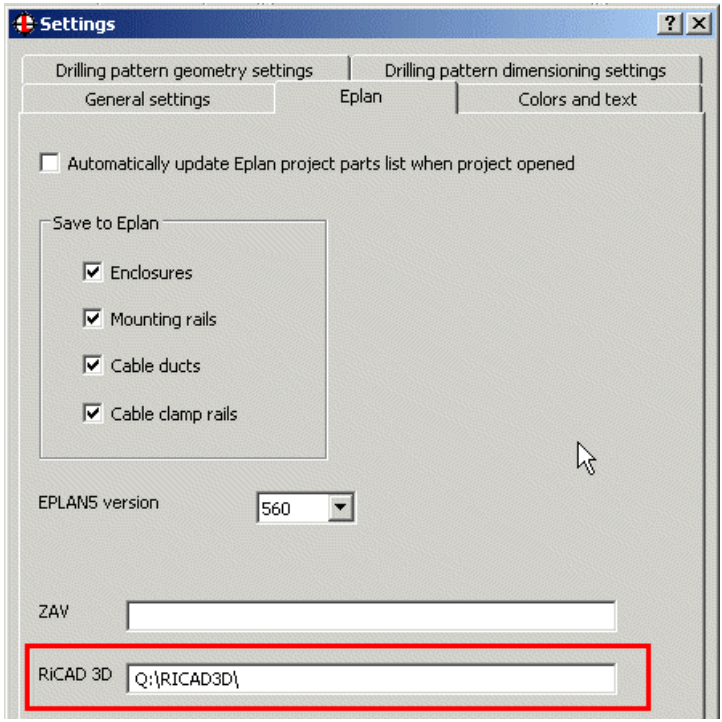
2.5.1.2 Running RiCAD 3D from hard disk

The contents of the RiCAD 3D can be copied to any hard disk directory. The program can then be run from that directory at any time.

This has the advantage that every network user can have direct access to the same data. It also drastically reduces the administrative effort of installing updates.

To copy the RiCAD 3D to hard disk:

- Insert the RiCAD 3D CD. If the program automatically opens, close it.
- Open Windows Explorer and click your CD drive.
- Select the entire contents of the RiCAD 3D CD and then choose Edit – Copy.
- Create a new directory on a network drive or on your hard disk, for example Q:\RiCAD3D.
- Click the new directory, and choose Edit – Insert. Windows will now copy the entire contents of the CD to the new directory.
- Switch to (or start) eCabinet and select the parameter settings under Information/Security – eCabinet Parameters.
- Choose the Eplan tab.



- Change the RiCAD 3D parameter to the new directory.

Click [OK] to save the new parameter setting.
Repeat this parameter setting on all workstations.

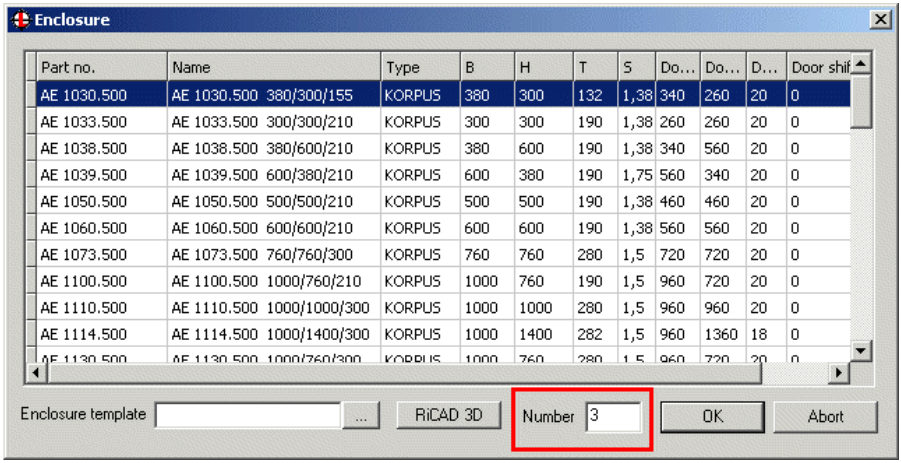
2.5.2 Using RiCAD 3D data

There are three ways to access enclosure data in order to insert an enclosure in eCabinet:

- Enclosure from the eCabinet database
- Enclosure from a template
- Enclosure from the RiCAD 3D enclosures catalogue

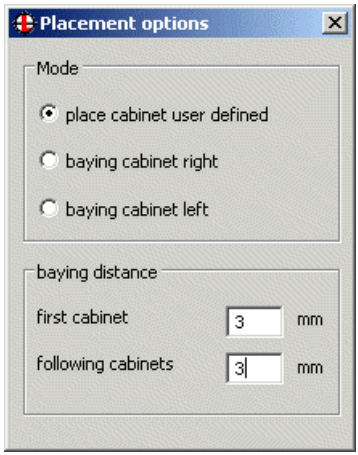
These options are available whether you are creating a new eCabinet project or inserting an enclosure into an existing project.

When you select the placement command, the first option in the dialogue specifies the number of **enclosure**s you want to insert:



Next, click the **[RICAD 3D]** button To select and place a RiCAD 3D enclosure.

A dialogue opens for you to specify the placement mode and row spacing:



To choose freely where to place the enclosure, select ‘Place enclosure user defined’ and specify a placement point.

Alternatively, choose the ‘baying enclosure left’ or ‘baying enclosure right’ options to place one or more enclosures (as specified in ‘Quantity’) so they are next to each other.

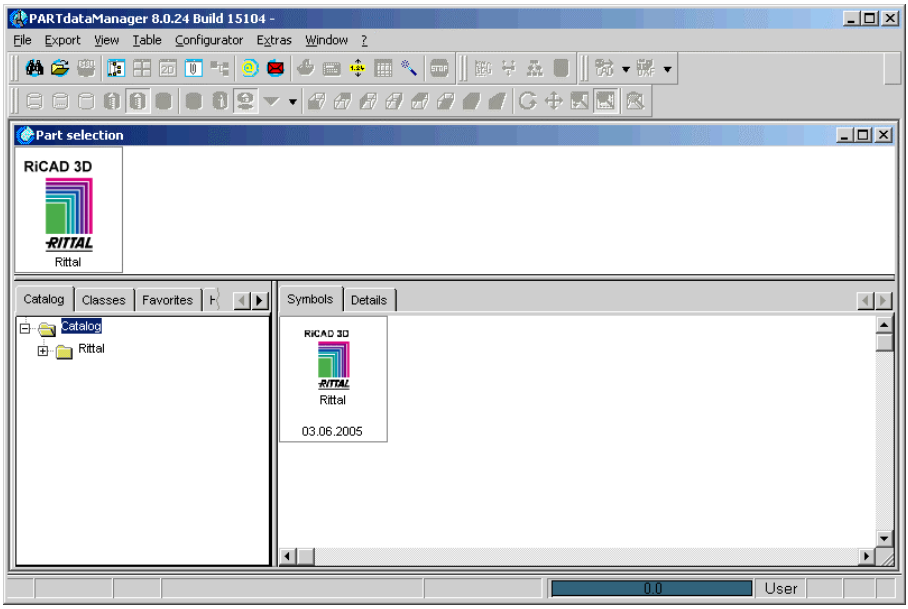
Note: Enter the value for the spacing by hand.

In the status bar, you are prompted to click the enclosure or profile.

The placement point is always the bottom left rear corner of the enclosure.

Specify the placement point.

RiCAD 3D now starts up.

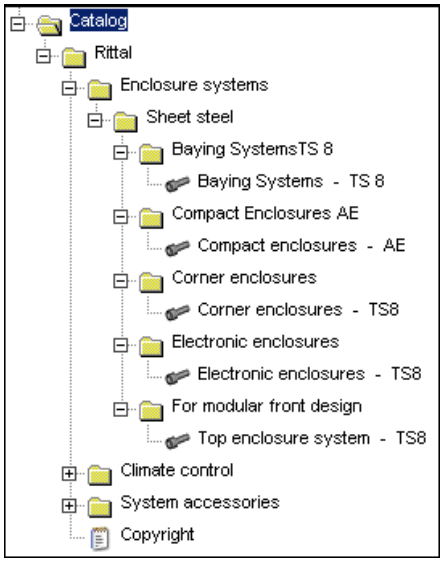


The next step is to select an enclosure.

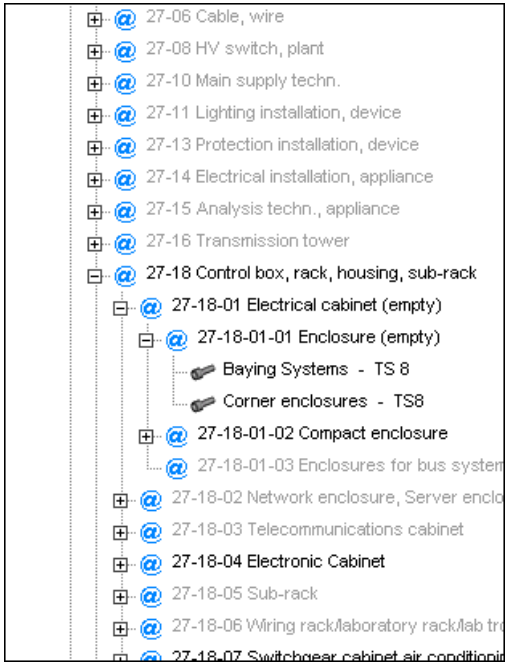
2.5.3 Finding an enclosure in RiCAD 3D

There are several ways to find and select an enclosure:

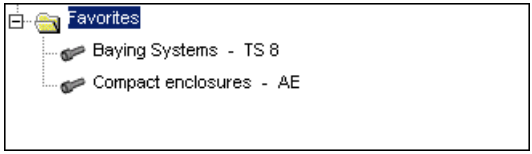
- **Browse catalogue**



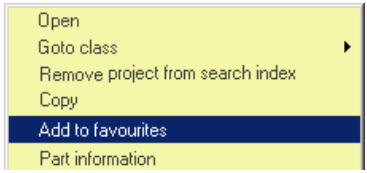
- **Classes**



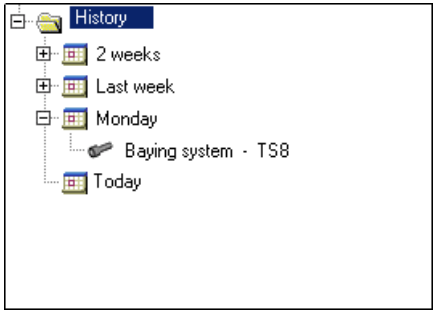
- **Favourites**



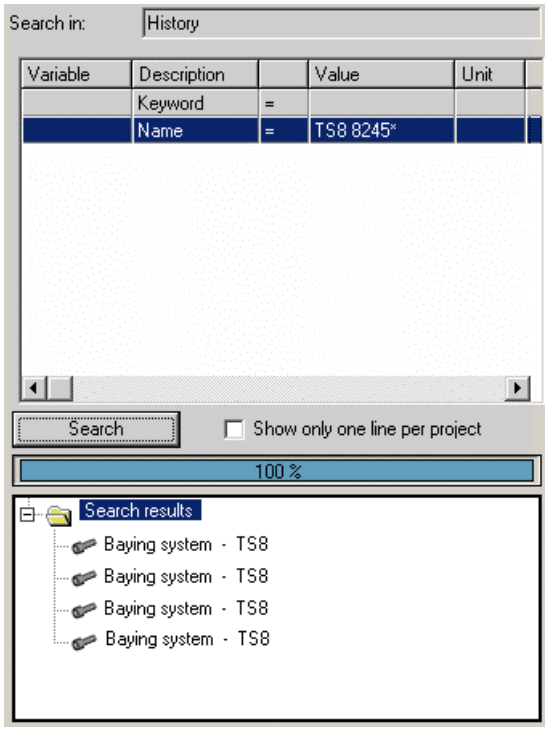
Note: To create a favourite, right-click an enclosure in the hierarchy.



- **History**



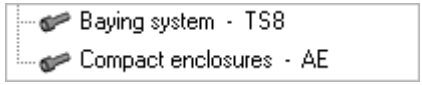
- **Search**



Type a search expression (for example a name) and click [Search].

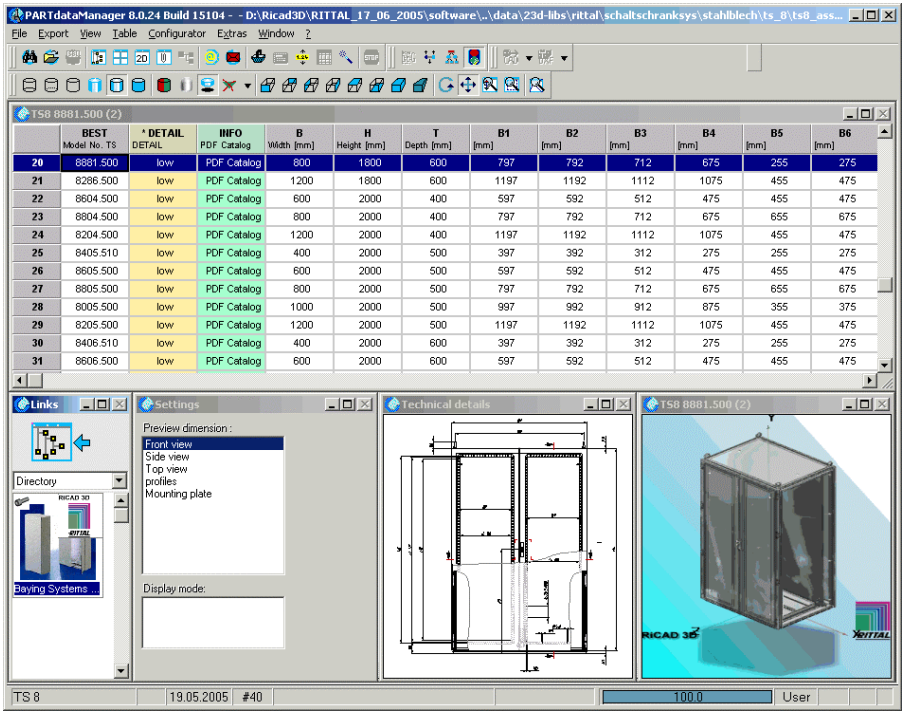
The search results are shown in the lower area.

The search result is a list of objects matching the type of search:



To select one of the enclosures, double-click it in the list.

RiCAD 3D then does calculations on the selected model and displays a further selection:



	BEST Model No. TS	* DETAIL DETAIL	INFO PDF Catalog	B Width [mm]	H Height [mm]	T Depth [mm]	B1 [mm]	B2 [mm]	B3 [mm]	B4 [mm]	B5 [mm]	B6 [mm]
20	8881.500	low	PDF Catalog	800	1800	600	797	792	712	675	255	275
21	8286.500	low	PDF Catalog	1200	1800	600	1197	1192	1112	1075	455	475
22	8604.500	low	PDF Catalog	600	2000	400	597	592	512	475	455	475
23	8804.500	low	PDF Catalog	800	2000	400	797	792	712	675	655	675
24	8204.500	low	PDF Catalog	1200	2000	400	1197	1192	1112	1075	455	475
25	8405.510	low	PDF Catalog	400	2000	500	397	392	312	275	255	275
26	8605.500	low	PDF Catalog	600	2000	500	597	592	512	475	455	475
27	8805.500	low	PDF Catalog	800	2000	500	797	792	712	675	655	675
28	8005.500	low	PDF Catalog	1000	2000	500	997	992	912	875	355	375
29	8205.500	low	PDF Catalog	1200	2000	500	1197	1192	1112	1075	455	475
30	8406.510	low	PDF Catalog	400	2000	600	397	392	312	275	255	275
31	8606.500	low	PDF Catalog	600	2000	600	597	592	512	475	455	475

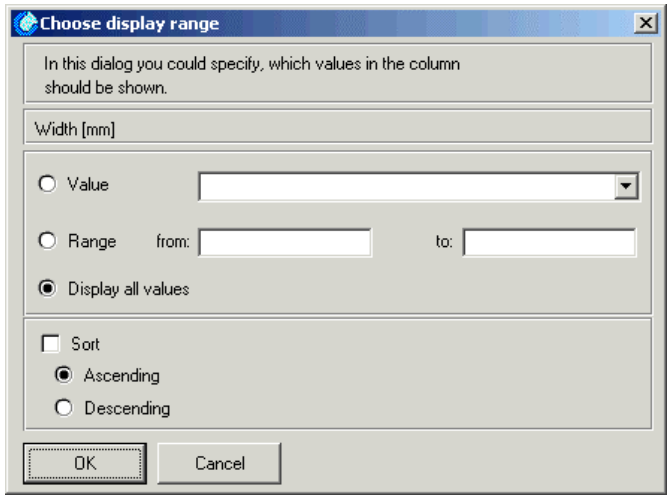
The upper area shows all enclosures of the selected model.

If you want to narrow down the selection, there are various ways to do so:

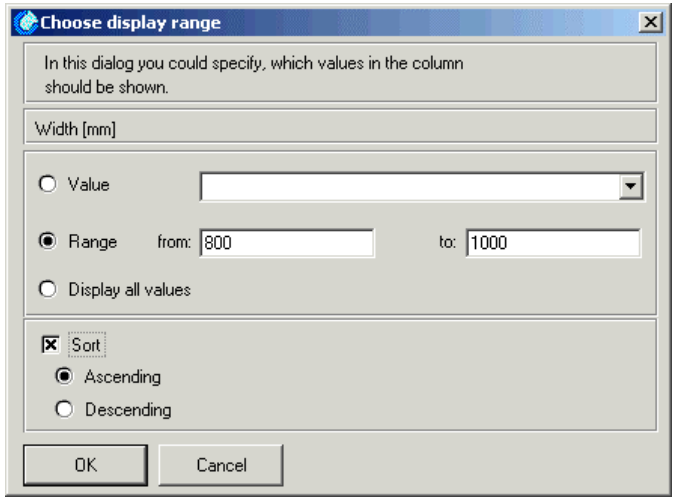
The main ways of filtering the selection are as follows:

- **Filtering by column headings**

Simply click a column heading. For example, if you want to select an enclosure by its width, click the column heading W for Width.



You can now filter the selection to show all enclosures matching a specific width or width range. You can also specify the sort order.



Click [OK] to sort and redisplay the list. The current filter is displayed in the column heading.

TS8 8881.500 (2)

	BEST Model No. TS	* DETAIL DETAIL	INFO PDF Catalog	B (800-1000) Width [mm]	H Height [mm]	T Depth [mm]
1	8845.500	low	PDF Catalog	800	1400	500
2	8865.500	low	PDF Catalog	800	1600	500
3	8884.500	low	PDF Catalog	800	1800	400
4	8885.500	low	PDF Catalog	800	1800	500
5	8880.500	low	PDF Catalog	800	1800	500
6	8886.500	low	PDF Catalog	800	1800	600
7	8881.500	low	PDF Catalog	800	1800	600
8	8804.500	low	PDF Catalog	800	2000	400
9	8805.500	low	PDF Catalog	800	2000	500
10	8806.500	low	PDF Catalog	800	2000	600
11	8808.500	low	PDF Catalog	800	2000	800
12	8826.500	low	PDF Catalog	800	2200	600
13	8084.500	low	PDF Catalog	1000	1800	400
14	8080.500	low	PDF Catalog	1000	1800	400
15	8005.500	low	PDF Catalog	1000	2000	500
16	8006.500	low	PDF Catalog	1000	2000	600

A number of different filters can be combined:

TS8 8881.500 (2)

	BEST Model No. TS	* DETAIL DETAIL	INFO PDF Catalog	B (800-1000) Width [mm]	H (1400-2000) Height [mm]	T Depth [mm]
1	8845.500	low	PDF Catalog	800	1400	500
2	8865.500	low	PDF Catalog	800	1600	500
3	8884.500	low	PDF Catalog	800	1800	400
4	8084.500	low	PDF Catalog	1000	1800	400
5	8080.500	low	PDF Catalog	1000	1800	400
6	8885.500	low	PDF Catalog	800	1800	500
7	8880.500	low	PDF Catalog	800	1800	500
8	8886.500	low	PDF Catalog	800	1800	600
9	8881.500	low	PDF Catalog	800	1800	600
10	8804.500	low	PDF Catalog	800	2000	400

To remove a filter, right-click the column-heading.

- **Filtering by individual values**

Click a value, for example **W = 800**.

TS8 8880.500 (2)

	BEST Model No. TS	* DETAIL DETAIL	INFO PDF Catalog	B (800-1000) Width [mm]	H Height [mm]
1	8845.500	low	PDF Catalog	800	1400
2	8865.500	low	PDF Catalog	800	1600
3	8884.500	low	PDF Catalog	800	1800
4	8084.500	low	PDF Catalog	1000	1800
5	8080.500	low	PDF Catalog	1000	1800
6	8885.500	low	PDF Catalog	800	1800
7	8880.500	low	PDF Catalog	800	1800
8	8886.500	low	PDF Catalog	800	1800
9	8881.500	low	PDF Catalog	800	1800
10	8804.500	low	PDF Catalog	800	2000
11	8805.500	low	PDF Catalog	800	2000
12	8005.500	low	PDF Catalog	1000	2000

The list is filtered and sorted. The current filter is displayed in the column heading.

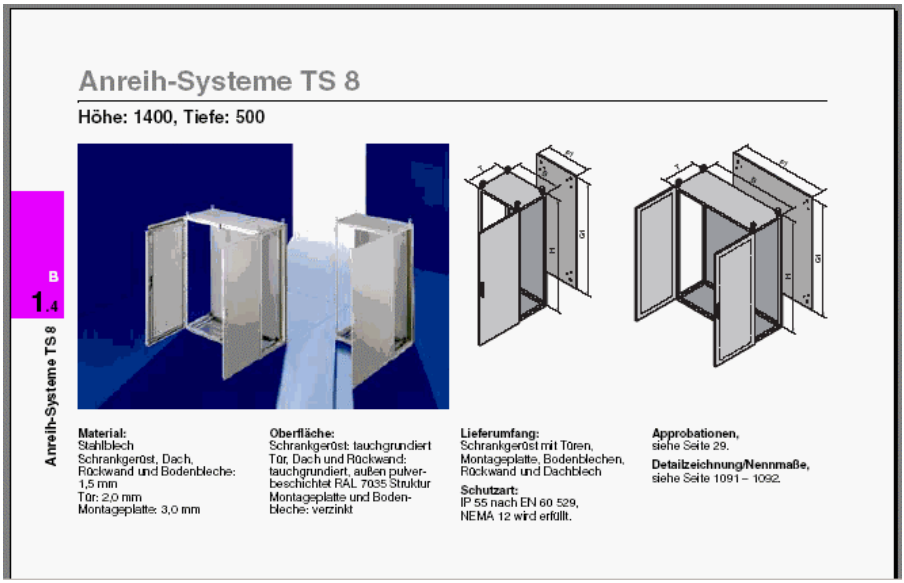
Filter criteria can be combined.

To remove a filter, right-click the column-heading.

2.5.4 Additional information on RiCAD 3D enclosure models

Additional information is available on the enclosure models in RiCAD 3D. To display it, click **[PDF Catalogue]**.

Note: To use this feature, you need to have Acrobat Reader installed.

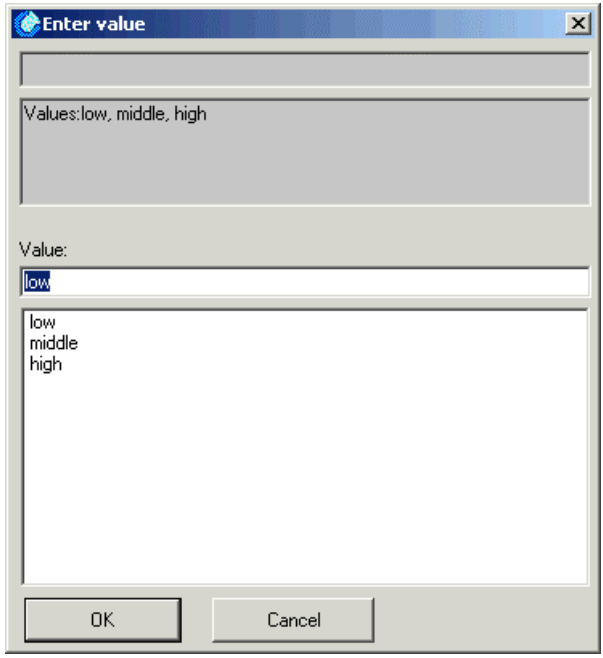


Information on the selected product is displayed from the Rittal manual.

2.5.5 Detail level for displaying parts in RiCAD 3D

You have the option of specifying a detail level in RiCAD 3D. Simply click the box provided for the purpose:

The available detail levels are low, medium and high.



Note: Parts are currently always transferred to eCabinet at 'low' detail, regardless of the setting in RiCAD 3D.

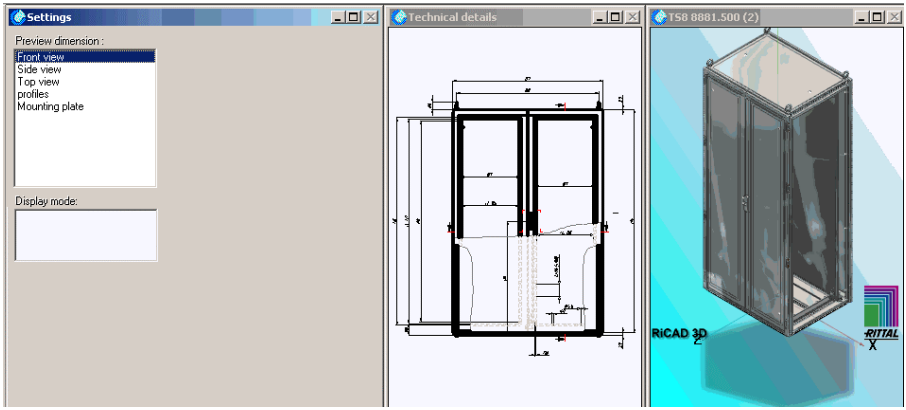
2.5.6 Transferring an enclosure to eCabinet

Once you have filtered the selection, you can select the enclosure you want. To do this, simply click the appropriate line number.

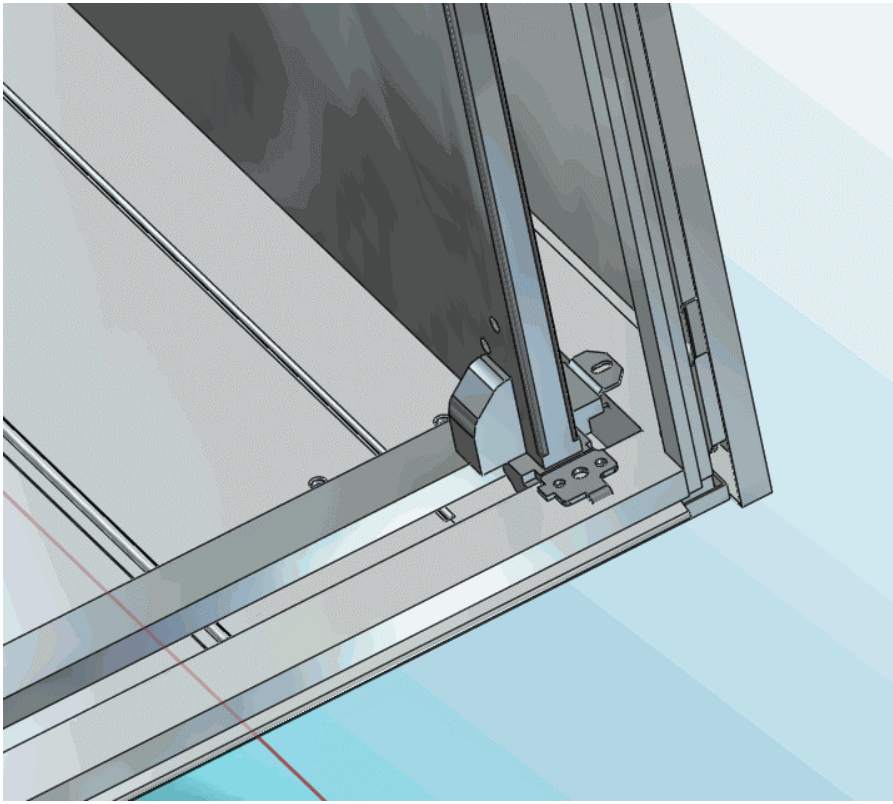
TS8 8884.500 (2)

	BEST Model No. TS	* DETAIL DETAIL	INFO PDF Catalog	B (800-1000) Width [mm]	H Height [mm]	T Depth [mm]
1	8845.500	low	PDF Catalog	800	1400	500
2	8865.500	low	PDF Catalog	800	1600	500
3	8884.500	low	PDF Catalog	800	1800	400
4	8084.500	low	PDF Catalog	1000	1800	400
5	8080.500	low	PDF Catalog	1000	1800	400
6	8885.500	low	PDF Catalog	800	1800	500
7	8880.500	low	PDF Catalog	800	1800	500
8	8886.500	low	PDF Catalog	800	1800	600
9	8881.500	low	PDF Catalog	800	1800	600
10	8804.500	low	PDF Catalog	800	2000	400

The selected enclosure is shown in various views in the lower portion of the screen.

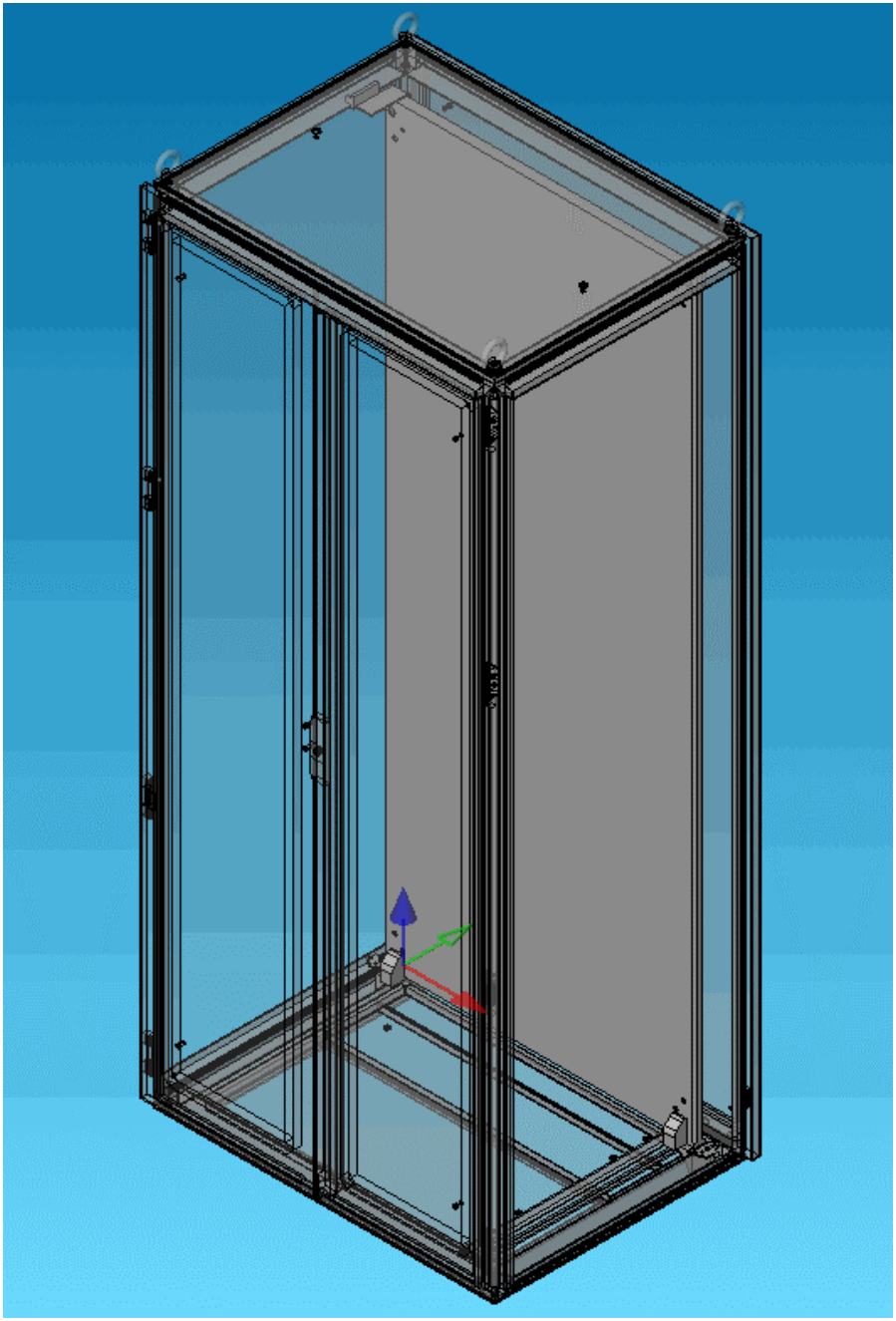


All views can be enlarged in the usual way. Additionally details can be made visible when required.



To transfer the enclosure to eCabinet, choose Export – Export on the menu or click the Export button.

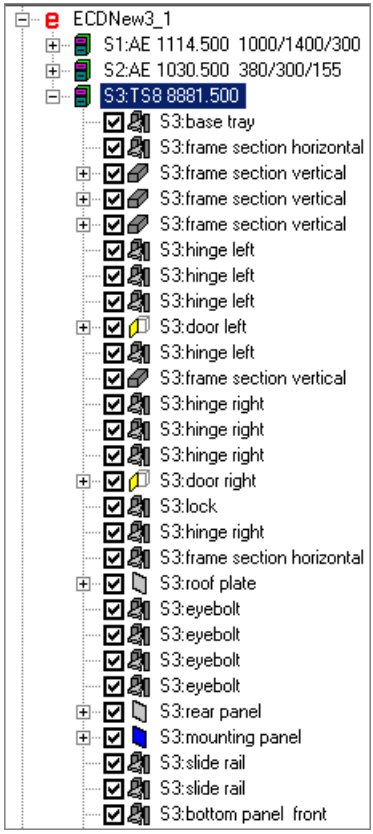
The system automatically returns to eCabinet to insert the selected enclosure.



Note:

As well as importing a graphical representation of the enclosure, eCabinet also integrates it into its component hierarchy. That is, the enclosure is given logical data that identifies it as an enclosure, and its components are tagged and treated as enclosure components. eCabinet can automatically tell the difference between a mounting panel, a door, a profile, etc.

The hierarchy is displayed in the Navigator:



You can work with an enclosure imported from RiCAD 3D exactly as you do with any other enclosure from the eCabinet database.

A RiCAD 3D enclosure has the major advantage, however, of being far more detailed and its data having been provided directly by the manufacturer.

2.6 Baying enclosures

eCabinet can arrange one or more enclosures next to others in the drawing.

There are three easy-to-use options for placing enclosures:

- Place enclosure user defined
- Baying enclosure left
- Baying enclosure right

The spacing between enclosures is taken from the database and can be modified before placement.

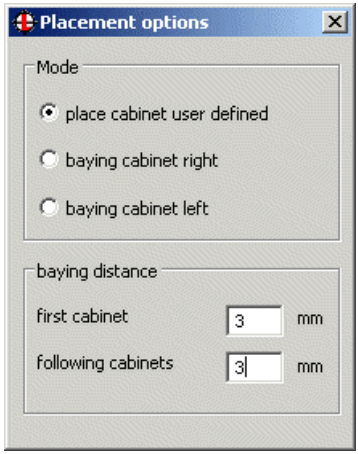
Any side panels can be removed when arranging enclosures in a row.

These features can be used with:

- Enclosures from the eCabinet database
- Enclosure templates
- Enclosures from RiCAD 3D

Enclosure templates are always placed with the quantity set to 1.

A dialogue opens when you select an enclosure for placement in a drawing:



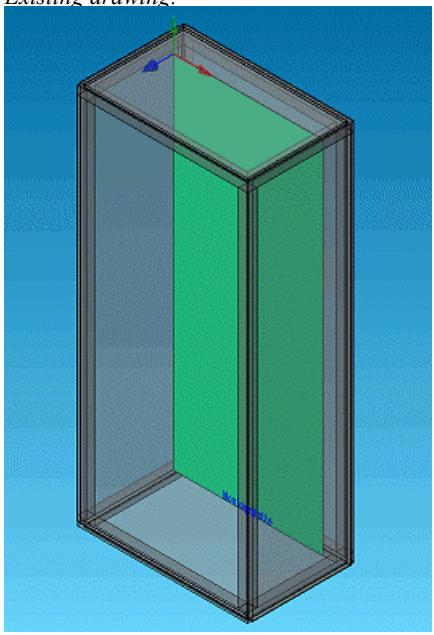
You can still place the enclosure at any location by selecting the 'Place enclosure user defined' option and entering a placement point.

Alternatively, choose the 'Baying enclosure left' or 'Baying enclosure right' options to place one or more enclosures (as specified in 'Quantity') so they are next to each other.

Note: Any number of enclosures from the eCabinet database or RiCAD 3D can be placed at a time by specifying a quantity. Enclosure templates are always placed one at a time.

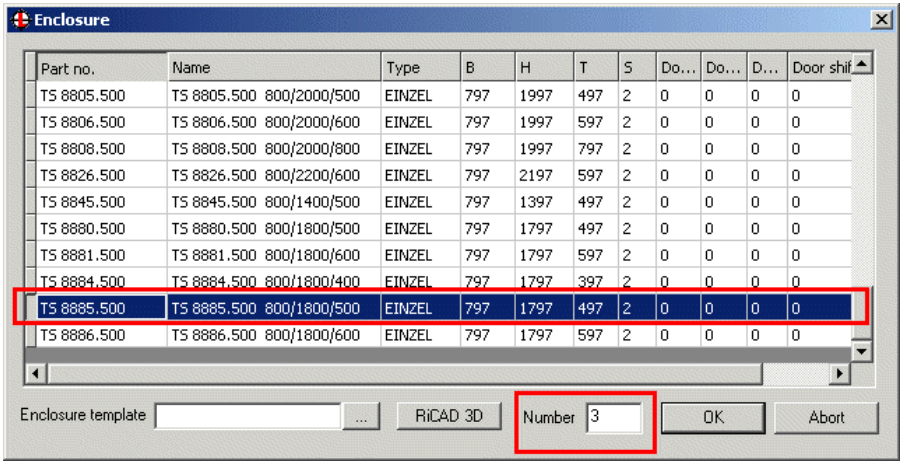
Example

Existing drawing:

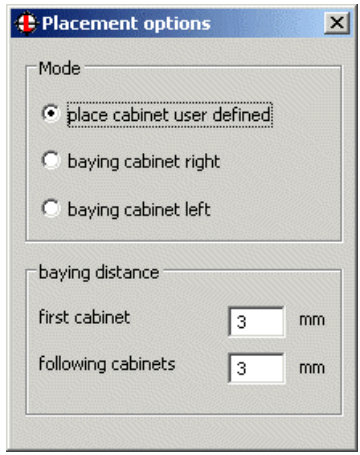


To place three TS8 enclosures so that they adjoin the TS8 enclosure already in the drawing:

- On the menu, choose Components – Enclosures and then 'Place Enclosure'.
- Choose an enclosure from the database and specify the quantity.



- Choose [OK] to confirm. A dialogue is then displayed:



- Under ‘Mode’, select ‘Baying enclosure right’.

Note: The spacing between enclosures is taken from the database and can be modified before placement.

- In the status bar, you are prompted to click the enclosure or profile.

  Click the enclosure in the drawing. The new enclosure will be placed next to the enclosure you click.

- A message is displayed:

- Choose whether to remove the right-hand side panel of the enclosure already in the drawing (because a new enclosure will adjoin it).

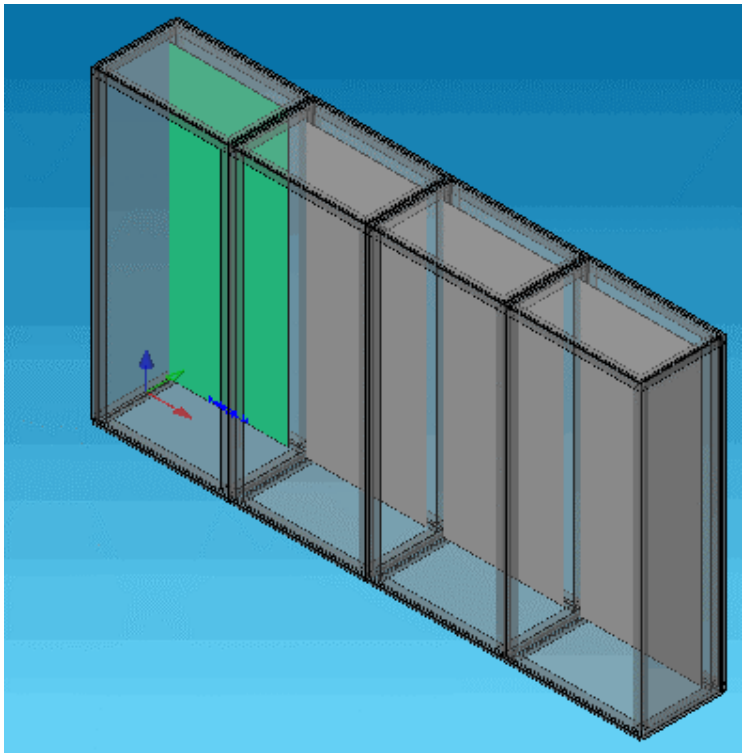
  If you select [Yes], the right side panel of the existing enclosure and the left side panel of the first new enclosure will be omitted when the new enclosure is added.

  If you select [No], the first new enclosure will be placed with all side panels intact and the enclosure already in the drawing will remain unchanged.

Note: When you insert two or more enclosures, their side panels are always omitted. If this is not what you want, place the enclosures individually.

- Once you have made your selection, the enclosures are inserted in the drawing.

Result:



3 Activating and displaying components

3.1 Activating a component

One of the basic features of working in eCabinet is that a component is always subordinate to another component. The other component can be part of the main enclosure, an auxiliary component such as a mounting rail, or an electrical component that has already been placed in the drawing.

Before placing a component for the first time, its reference component must be activated.

View	eCabinet
Activate component	**View | Activate component**

The activation method can be toggled between two modes:

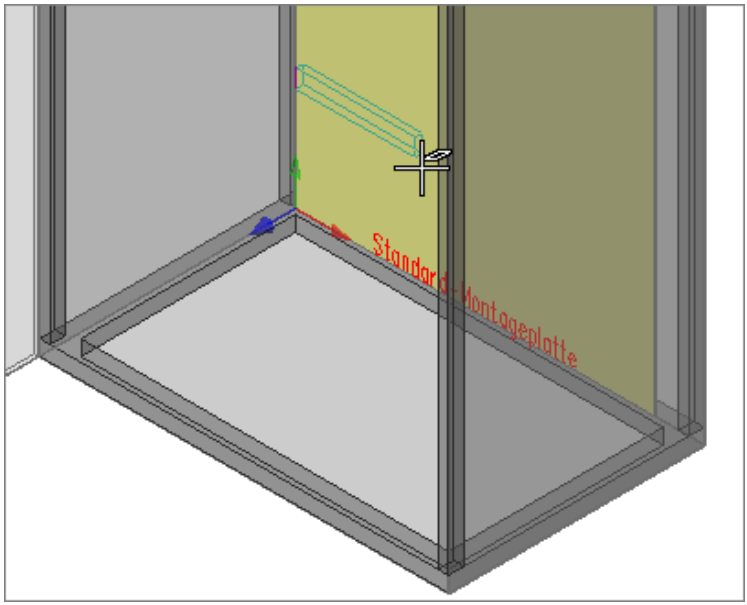
Edge	A component is activated by holding down [Ctrl] and tapping one its edges. This is the usual method and is best suited for mounting panels, side panels and doors.
Surface	A component is activated by holding down [Ctrl] and tapping one of its surfaces.

Hold down the [Ctrl] key and tap an edge or surface of a component.

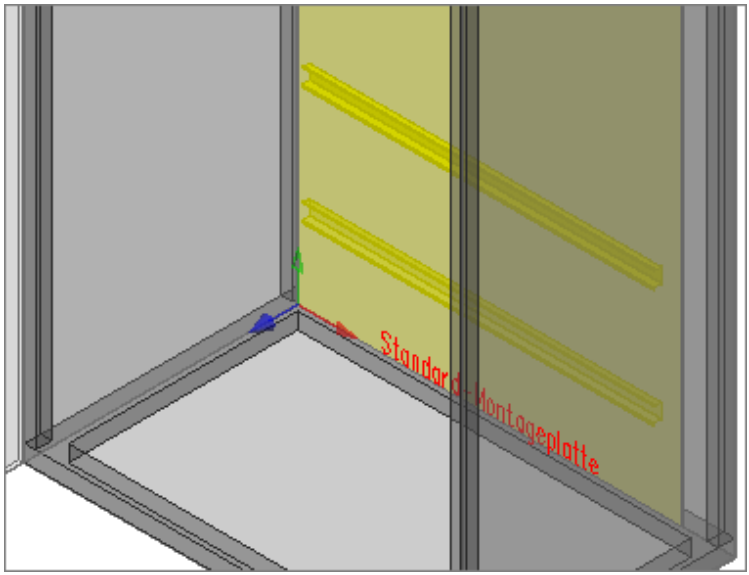
The view in modelling space and the viewing plane remain unchanged, but the working plane is set to the reference component so that subsequent input when placing a component takes effect on that plane.

The colour of the activated component can be selected as a parameter.

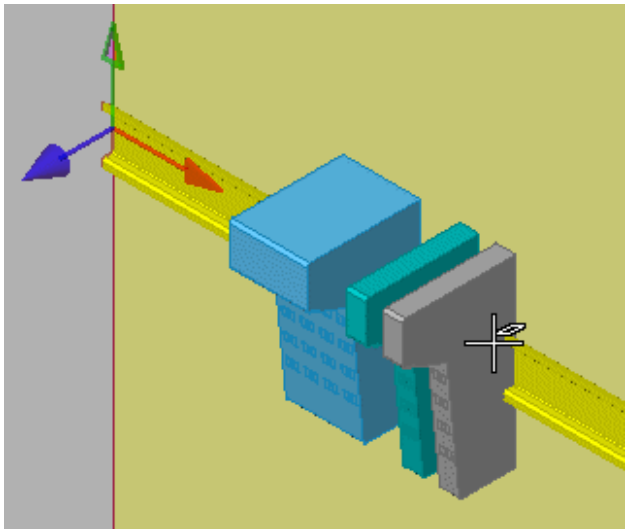
When you place a subsequent component, such as a mounting rail, it is highlighted in colour and the name of the activated component is displayed.



Mounting rail placed on mounting panel:

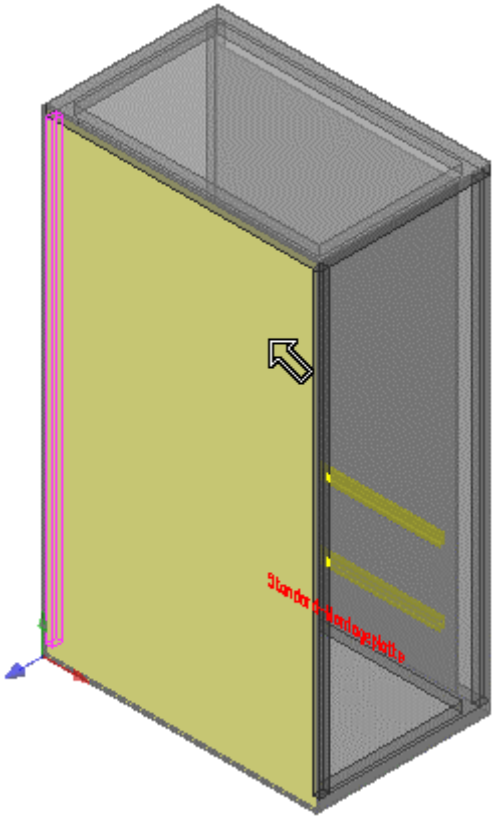


The mounting rail can now be fitted with other components. If the mounting rail is activated, all subsequent components are placed in its working plane.



Special case: Inward-facing surfaces in a closed cabinet

When components are activated by tapping a surface, note that surfaces located behind others are only available for activation when they are visible. One situation in which this effect becomes noticeable is when the enclosure door is closed. If the wanted surface is not visible, tapping it will select the visible surface located in front of it, so an attempt to activate the rear panel of an enclosure when the door is closed will result in the door being selected instead:

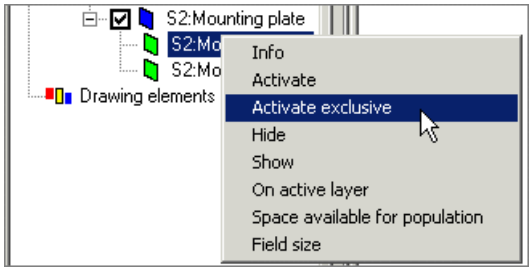


In such cases, switch to Edge mode and try activating the rear panel by tapping one of its edges. If this fails, hide the door and use the 'Activate component exclusively' command.

View toolbar | **Activate component exclusively**

Navigator: Another easy way to activate any component is by using the Navigator.

The context menu for each surface of a component includes 'Activate' and 'Activate exclusive' commands:



3.2 Activating a component exclusively

The 'Activate component exclusively' command activates a component as a reference component for use when placing other components and at the same time displays it 'exclusively' from the front: All other components of the enclosure not related to the activated component or not placed on it are hidden.

Activation and exclusive display of a component is not saved with the drawing.

View	eCabinet
Activate component exclusively	**View | Activate component exclusively**

The activation method can be toggled between two modes:

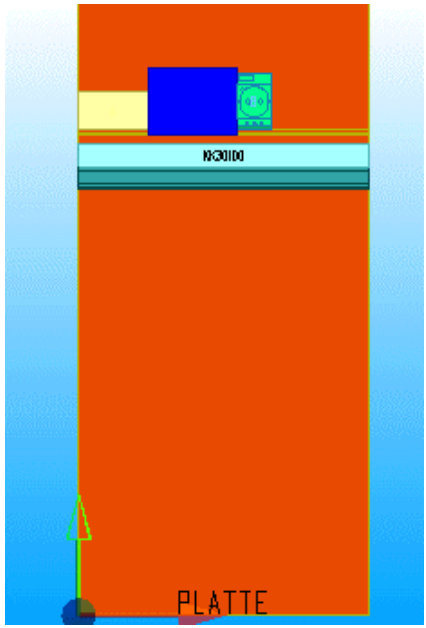
Edge	A component is activated by holding down [Ctrl] and tapping one its edges. This is the usual method and is best suited for mounting panels, side panels and doors.
Surface	A component is activated by holding down [Ctrl] and tapping one of its surfaces.

Hold down the [Ctrl] key and tap an edge or surface of the component you want to activate.

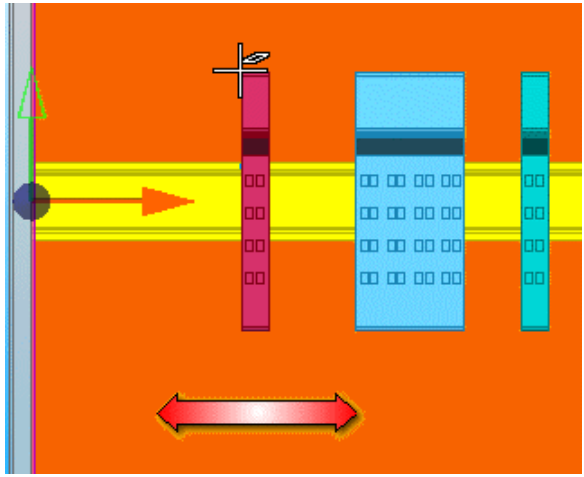
The activated component, e.g. a mounting panel, is shown from the front. All side panels, profiles, doors and components fitted to them are hidden. Components belonging to the activated component remain visible.

The colour of the activated component can be selected as a parameter.

When a mounting panel is activated, the ducts, mounting rails and other components attached to it remain visible.



Components can be moved during placement on the mounting rail:



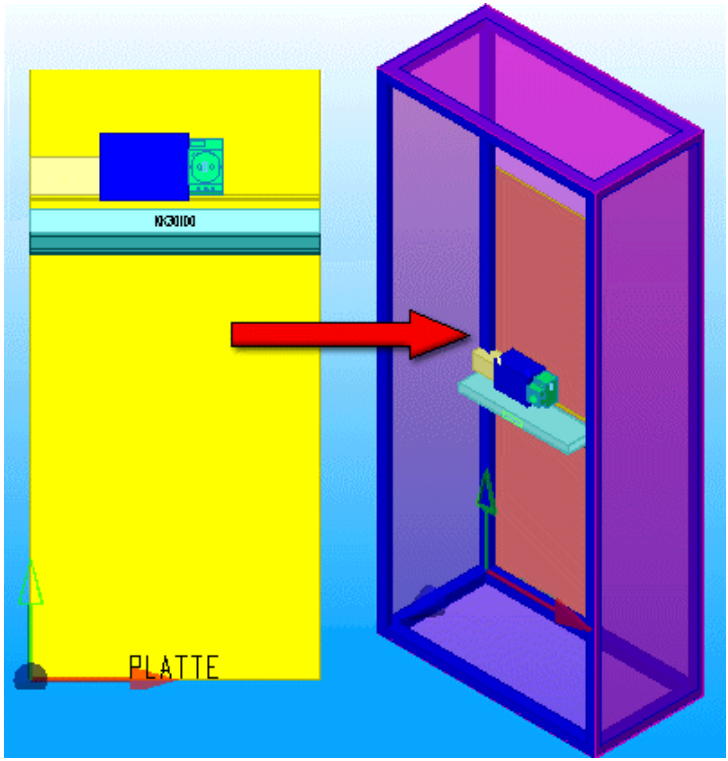
3.3 Isometric view of all parts

The Isometric View command switches back from an exclusive front view of the activated component to an isometric view of the modelling space. All parts and placed components are shown.

View	eCabinet
Chan e to ISO view	**Vie w | Isometric view**

The activated component remains activated.

Change to isometric view



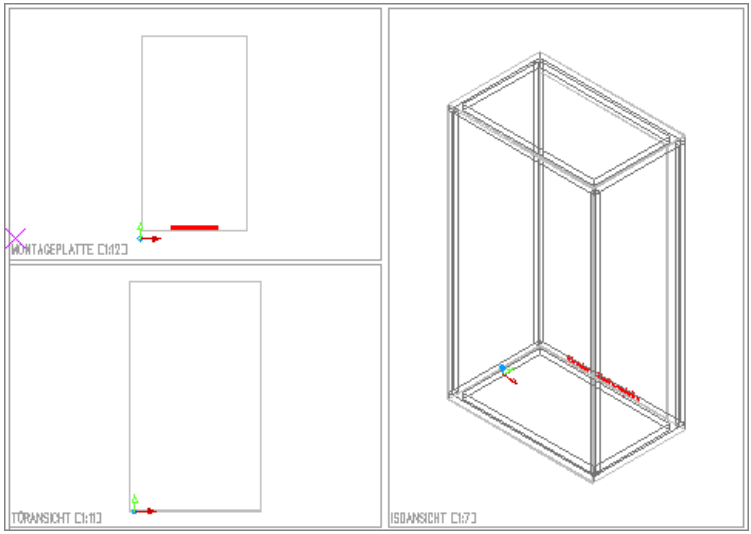
3.4 Enclosure views

Views are standardised elevations on 3D space. A drawing can contain several views. Enclosure views can only be used for displaying a drawing; they are needed in particular for DXF drawing data exports to EPLAN. Unlike other ways of displaying a drawing, editing is not possible in enclosure views; the commands for activating components are not available.

View	eCabinet
Enclo ure views	**Vie w | Enclosure views**

eCabinet includes three predefined views to display the enclosure or parts of it at various scales:

- Front view of the door/cabinet front
- Front view of the mounting panel
- Isometric view of the enclosure



The views are shown under the Views node in the Navigator and are saved with the drawing.

Selecting or clearing the checkbox next to a view icon shows or hides the view. Hidden views are not included in DXF data exports.

After a DXF export, you can switch back to another viewing mode.

3.4.1 Creating custom views

The default enclosure views can be supplemented with additional standard views and various types of custom views.

View | 3D views | Create views

Create standard views	This command creates one or more pre-set views for the formats A0 to A4, portrait or landscape.
Create new view	Creates a single view. The type of view is selectable.
Create perspective	This command creates a perspective view with a vanishing point.
Create detail view	A detail view shows an enlargement of a cut-out – a specific area of the drawing. It is specified by dragging a box to mark out a source area and is placed as a target area.

For detailed information on views, see the LOGOCAD TRIGA 3D Manual, Chapter 2 or in Help, TRIGA 3D Manual, ‘Views in 3D’.

3.5 Zoom

The four zoom buttons on the eCabinet View toolbar help find your way around the drawing in any view (isometric view, exclusive component view or enclosure views).

Enable zoom

Zooms a specific area in the drawing window.

If two or more drawings are open, tap the window you want to zoom first.

Tap to define the bottom left corner of the area you want to zoom, move to the top right corner and tap again.

The specified area is enlarged to fill the window.

Disable zoom

Returns to displaying the entire drawing.

If two or more drawings are open, tap the window you want to restore first.

The entire drawing is displayed again.

Zoom in (150%)

Zooms-in the centre of the current view in the drawing window to 150%.

Tap the drawing window you want.

All views can be zoomed, including enlarged views created with the Enable Zoom command.

Zoom out (75%)

Zooms-out the centre of the current view in the drawing window to 75%.

Tap the drawing window you want.

All views can be zoomed, including details and the entire drawing.

3.6 Labels

Many objects can have labels in eCabinet, including devices, ducts, various rails and user-defined components. These labels are permanently assigned to each component and are placed automatically.

The behaviour of the various types of label can be configured before and after placement. These settings affect all components that have already been inserted in drawings and all that are inserted in future.

Labels on components already in the drawing can also be modified in various ways. These include specifying a font size, fixed positioning, moving to a user-defined position, and showing and hiding individual labels independently of the global visibility setting.

3.6.1 Global parameter settings for labels

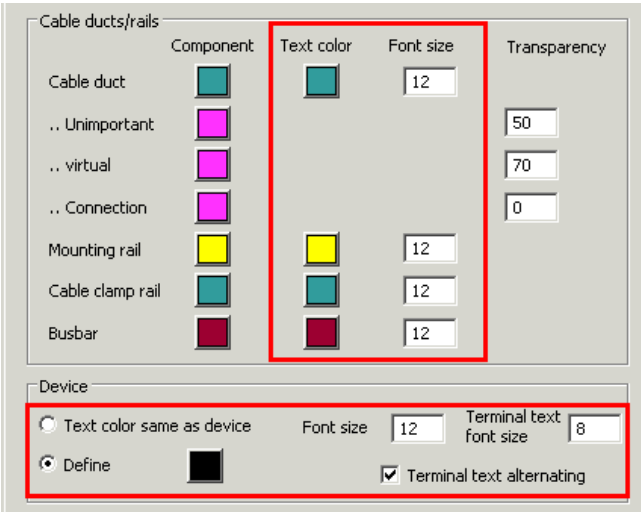
Global settings can be specified for text labels in the eCabinet parameters.

Information, organization and security

Parameters

Information/security | eCabinet parameters

Click the Colours and Text tab



In the marked areas, you can change the font colour and the font size for individual components.

Note: These settings only affect new components placed from now onwards.

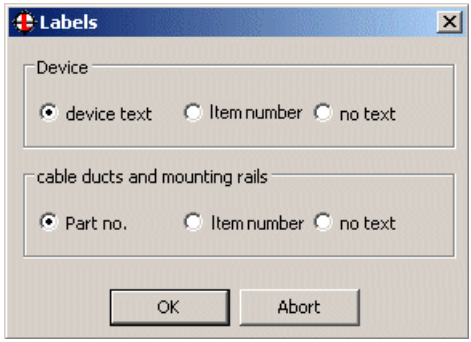
Once you have clicked [OK], the settings are saved and will continue to apply the next time you start eCabinet.

Note: All settings are stored in a file, ecdapp.ini. This .ini file is stored in the ECDTRIGA4 subdirectory of the start directory. Settings can be transferred from one workstation to another by copying this file.

3.6.2 Visibility of labels

The visibility of labels can be controlled separately for devices and ducts/rails.

Select the command on the menu bar, under View – Configure Text.



You can choose separately for devices and ducts/rails whether to display:

- Device tags
- Item numbers
- No labelling

Note:

'Devices' include:

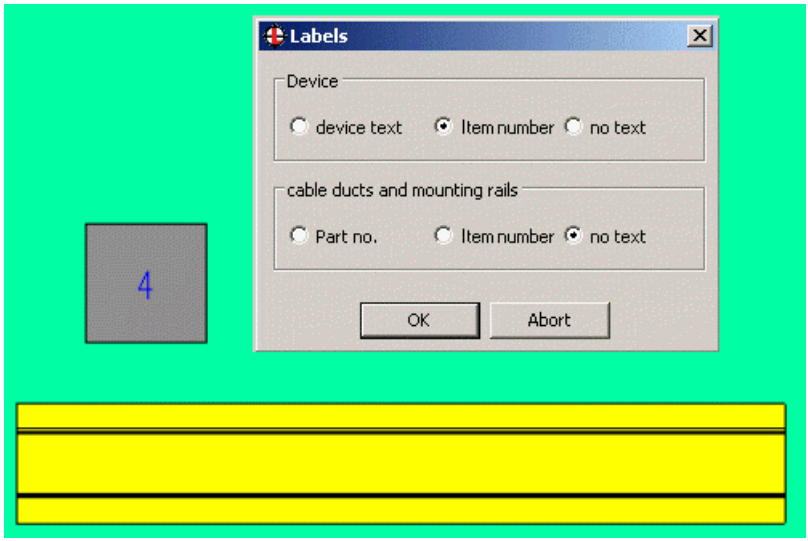
- Devices from the parts list (placed using the browser)
- Universal components
- User-defined components

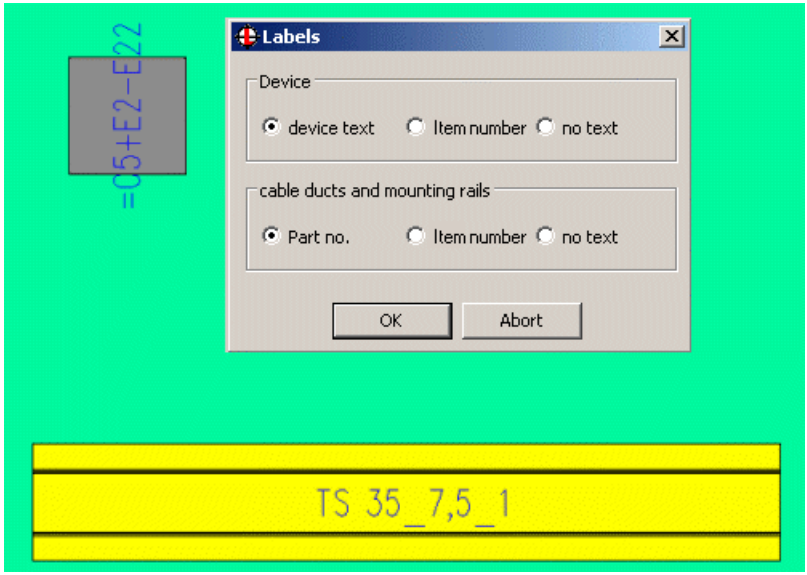
'Ducts and rails' include:

- Ducts
- Mounting rails
- Busbar systems
- Cable clamp rails

These settings are applied to the currently open project as soon as they are confirmed.

Each project has its own settings.





3.6.3 Editing existing labels

Labels on placed components can be modified at any time.

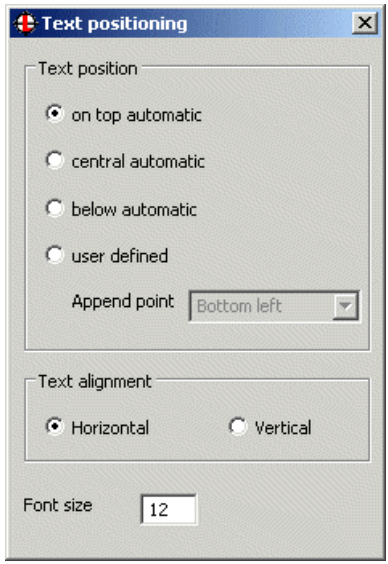
The modifiable parameters are:

- Font size
- Alignment
- Fixed position

It is also possible to specify a user-defined position.

Note: Only the format and location of the labels are modified, not the text.

On the menu, select View – Reposition Text.



The ‘automatic above’, ‘automatic centred’ and ‘automatic below’ positioning options can be applied to several components simultaneously.

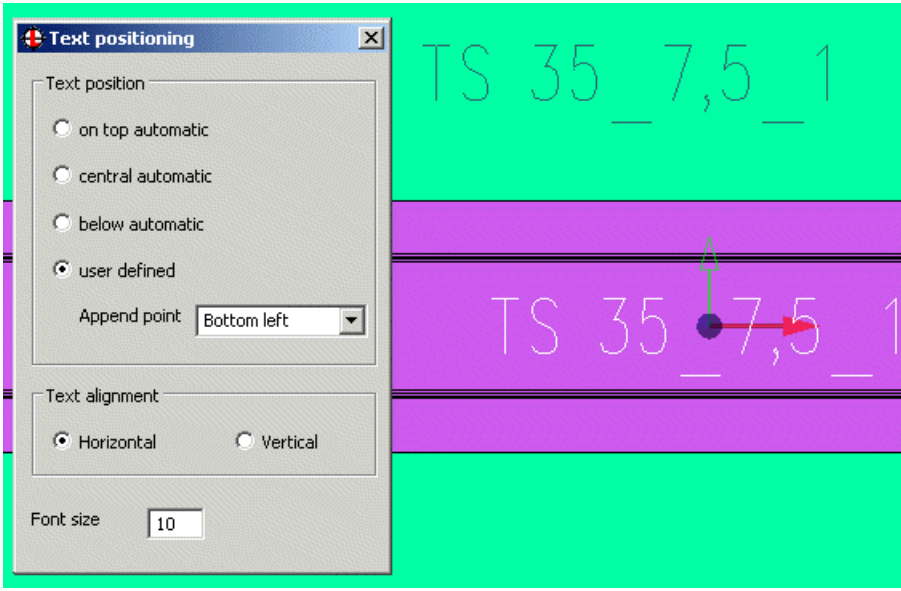
The ‘freely selectable’ option can only be used on one component at a time.

Choose the options you want, then select the components to apply them to. For multiple components, select them in rectangle selection mode.

To freely move a component's label, choose the 'freely selectable' option and specify an append point.

Click the component. You can now move the label with the cursor and click again to set it down.

The alignment and append point can still be modified.



If you alter the length of a duct or a rail, the label returns to its original position. The same happens if you exchange a duct or rail.

3.6.4 Showing and hiding labels

You can show and hide labels of components that have already been placed in a drawing.

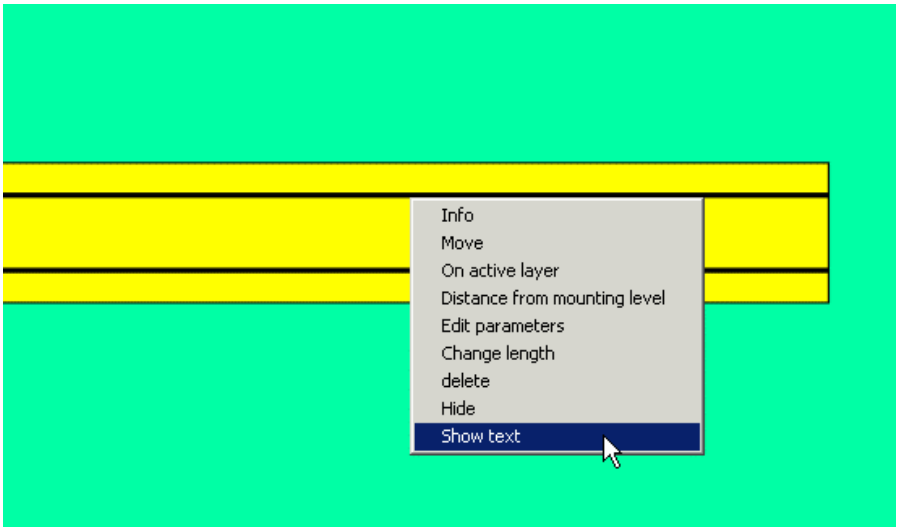
The command applies for one component at a time.

You will find it on the context menu that opens when you right-click a component.

Example:

Label shown:

Label hidden:



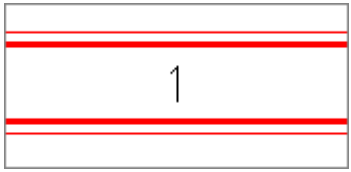
3.7 Displaying component labelling

3.7.1 Item number labelling

View

Display item number labelling

By default, components are labelled with their item number from the parts list associated with the project. Clicking the button a second time hides component item numbers.

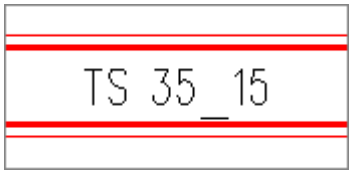


3.7.2 Device tag labelling

View

Show device tag labelling

Switches component labelling to device tags. Clicking the button a second time hides component device tags.



3.7.3 Renumbering components

Information, Organization and Security

Renumber components

eCabinet sequentially numbers all components as they are placed in an enclosure. Subsequently added components are given a number at the end of the sequence. Gaps in the numbering resulting on component deletion are not automatically closed. Use the Renumber command to remove such discrepancies.

3.7.4 Repositioning label text

The positioning of label text on components is modifiable. For the sake of better readability, it may be necessary to change the position, alignment and font size of specific component labels from the default.

View

Reposition text

A dialogue opens with three options:

Text position	Placement of the text relative to the component: • Centre (default) • Above the component • Below the component
Text alignment	• Horizontal (default) • Vertical
Text size	• 12 pt (default) • Custom value

Modify the settings and click [OK] to confirm. The dialogue closes and you can pick the components you want to apply the settings to.

Individual components can be picked by tapping them one after another until you press [ESC] to stop.

Multiple components can be picked by specifying a box that encloses them. Enter a point for the bottom left corner of the box, then press and hold the left mouse button for about 1 sec. Move to the top right corner and tap to enter a point for that corner. To confirm:

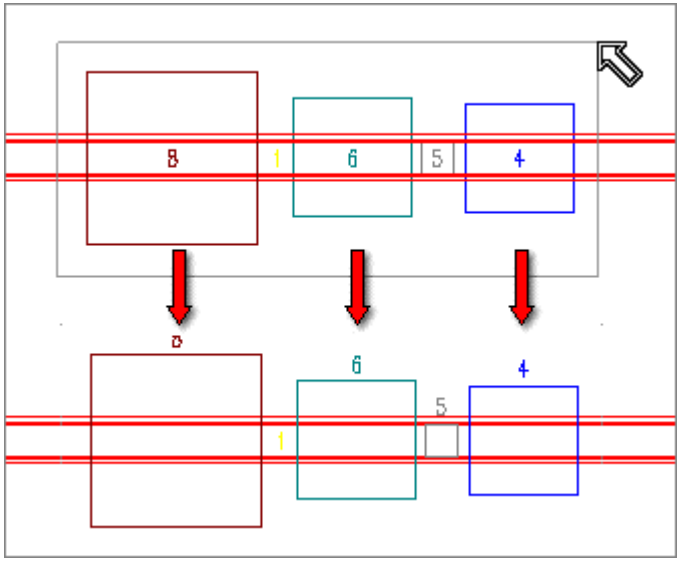
- Tap inside the box. This selects all components that are fully inside the box.
- Tap next to the box. This selects all components that the box touches, including ones outside it.

You are then prompted to confirm your selection or to add other elements by tapping them.

Choose [Yes] to apply the specified modification to the labels of all selected components.

Example:

In the picture, the components inside the box are selected and their label texts are changed from centred to above.

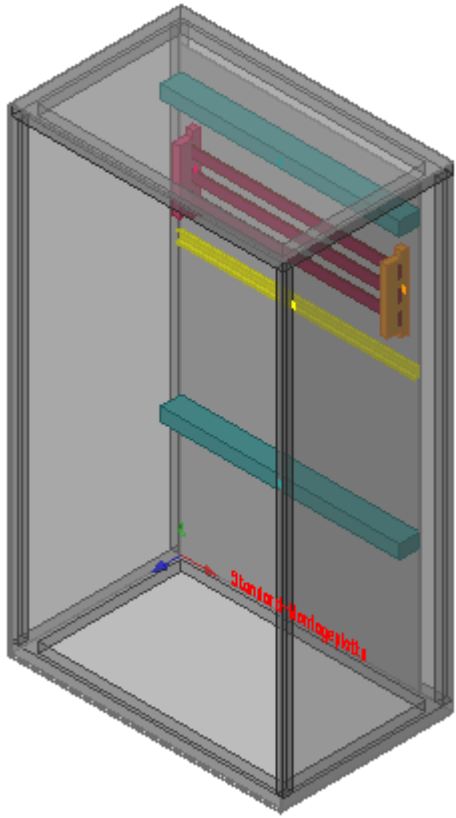


3.8 Showing/hiding shading

View **Shading on**
View **Shading off**

Shaded view shows the enclosure and its components with coloured surfaces. The top, side panels and door are 50% transparent. You can continue to work without restriction in shaded view.

The transparency can be adjusted in the eCabinet parameters.

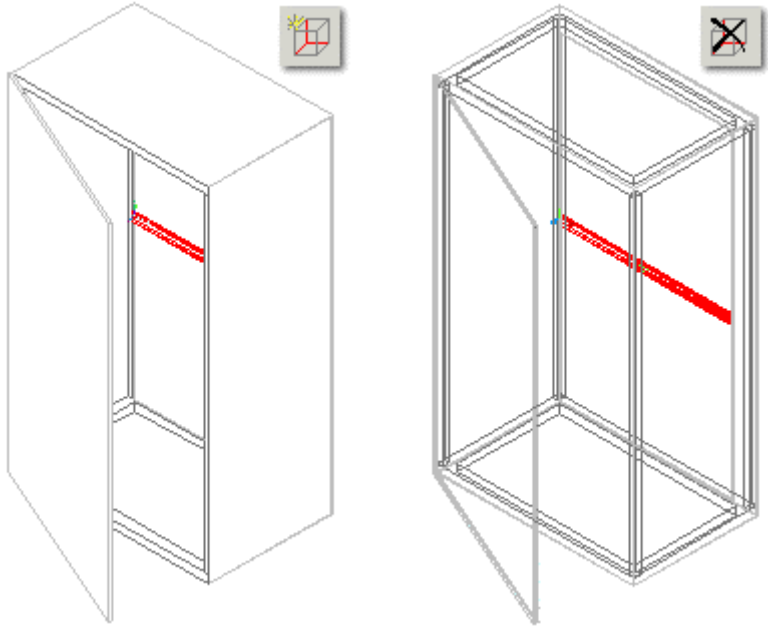


3.9 Showing/hiding hidden edges

View **Recalculate hidden edges**
View **Delete hidden edges**

Hidden edges are edges of components that are concealed by surfaces between them and the viewer. The default view is isometric view with hidden edges displayed; the enclosure and all components appear transparent.

When you change to a different view, the information on which edges are visible in isometric view is lost. Correct display of the non-transparent parts is restored by recalculation. Deleting redisplays the hidden edges.



3.10 Rotating a view

'Rotate view contents' rotates the contents of a view or the model about all three axes.

View

Rotate view

Specify the view and centre of rotation

First, specify the view to rotate and the point about which it is to be rotated.

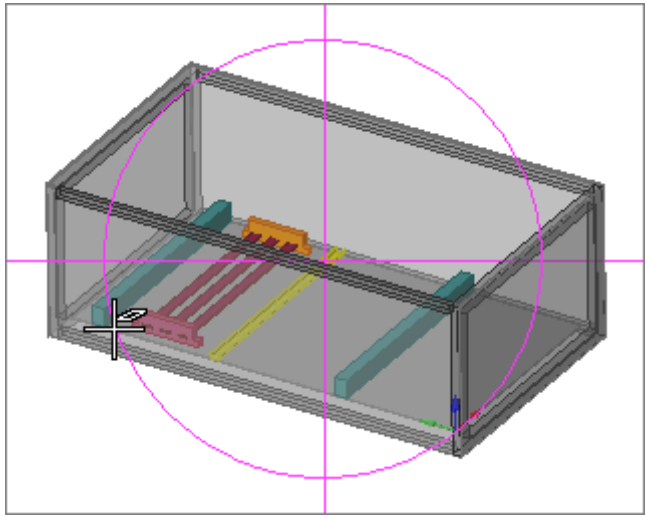
Tap	The centre of rotation is the centre of the space shown in the view.
<Search>	<Search> any point to make it the centre of rotation.
[M]	Press the M key to select the middle of the geometry as the centre of rotation.

The centre of rotation is marked with a cross.

Specify the rotation range

The possible rotation range is 360 degrees in each direction, i.e. it has the form of an imaginary sphere. There are two ways to control the rotation:

1) Tap	Tap next to the centre of rotation to determine the size of the sphere. This also affects the rotation speed: the bigger the sphere, the slower the rotation of the view contents. The direction of rotation is determined by where you tap the sphere. Press [Return] to accept your settings; the rotation stops.
2) Drag	Tap once to determine the size of the sphere and **hold down the mouse button/pen**. You can now rotate the view contents on the imaginary sphere by moving the mouse or pen. When you let go of the mouse button/pen, the view stops rotating and is left at its current position. If you let go while moving, the rotation continues, leaving the view contents rotating around the specified point.



Rotating with keyboard control

If you want to rotate the view contents manually in shaded or wireframe view, first specify a centre of rotation. Then specify an angle for the rotation increment.

For example, if you enter 10°, each time you press the left or right cursor key and each time you tap in the view causes the view contents to be rotated by 10°.

The PgUp and PgDn keys move the view contents closer or further away.

Stopping rotation

There are two ways to stop a view rotating:

[Esc] Restores the view to its original orientation.

[Return] Retains the current orientation.

Resetting a rotated view

To reset a rotated view to isometric view, right-click in the edge of the view and select the 'ISO view' option on the context menu.

4 Placing and modifying structural components

Structural components are elements that support and provide routing for electrical components and cables. They include:

- Enclosures
- Mounting rails
- Ducts
- Cable clamp rails
- Busbar systems

4.1 Adding a new enclosure to the current drawing

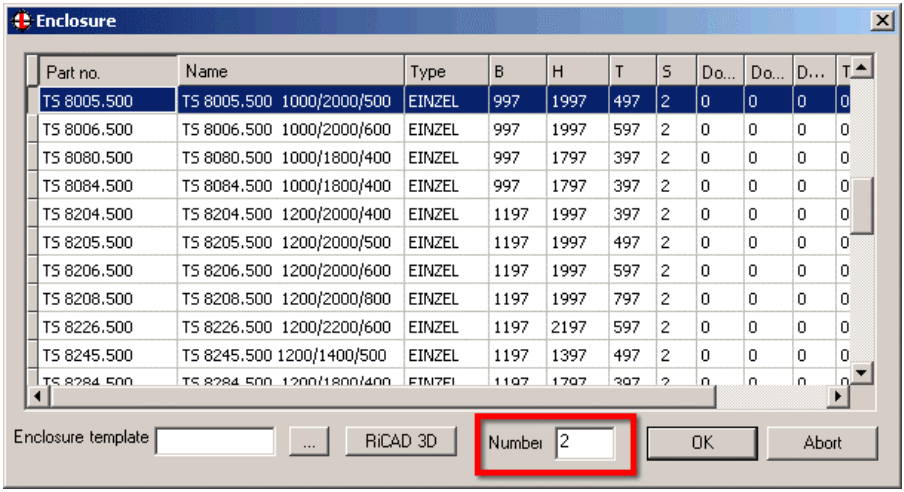
Placement

Select and place enclosure from database

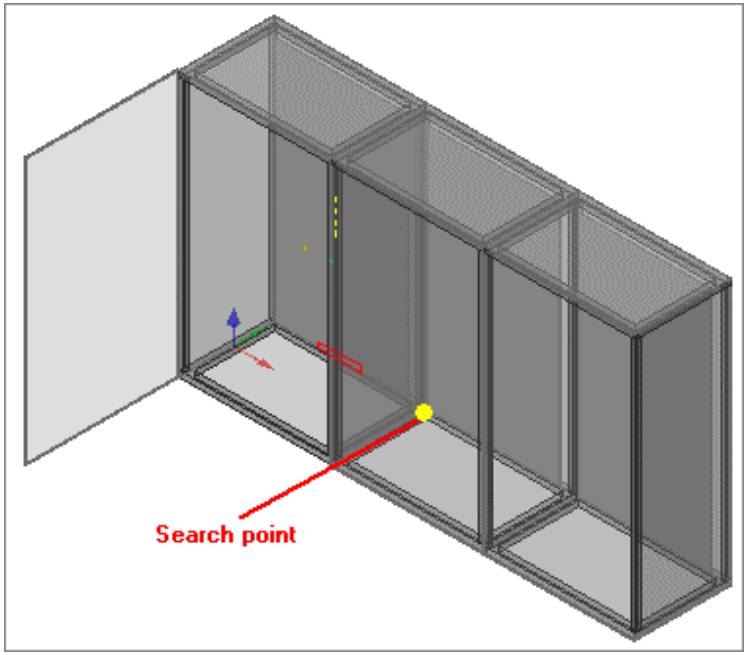
Components | Enclosure

Use this command to insert one or more additional enclosures in a loaded drawing.

Choose the model you want and type a quantity (2 in this case):



To place an enclosure flush with its predecessor, <search> for a bottom corner.

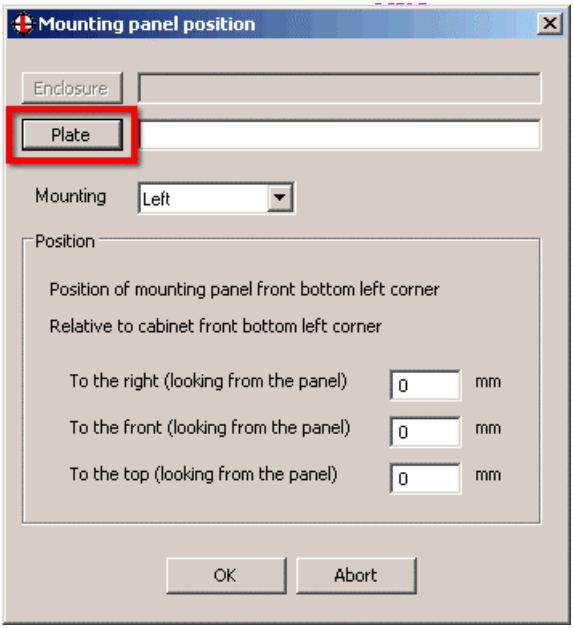


4.2 Placing a mounting panel

There is no need to activate a component before placing a mounting panel. A mounting panel is assigned to the enclosure and by default is placed at the bottom left of the rear panel.

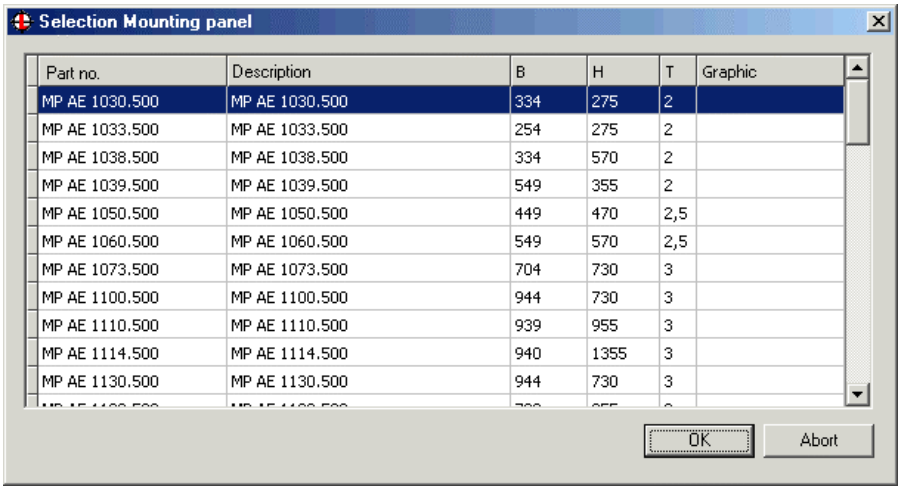
P acement		
Place mounting panel	**Components** | **Mounting panel**	

Tap or stroke the cabinet or a profile belonging to the cabinet to select it for placement of the mounting panel. Then specify the model and position of the mounting panel in the dialogue.

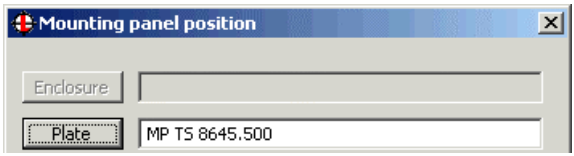


The [Panel] button opens a list of mounting panels in the database for selection.

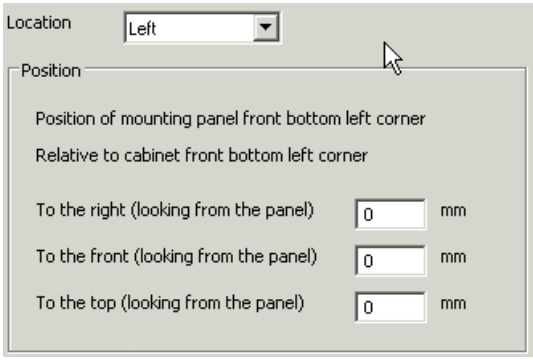
Click the panel you want to select it. This selects the entire row.



Click [OK] to select the panel. The model is inserted in the placement dialogue.



Placement options



Location

Default: Pre-set location on all standard enclosures (always at rear).

Position

The exact position relative to the placement point at bottom left can be specified using the X, Y and Z offsets:

- Offset towards right
- Offset towards front
- Offset towards top

Example

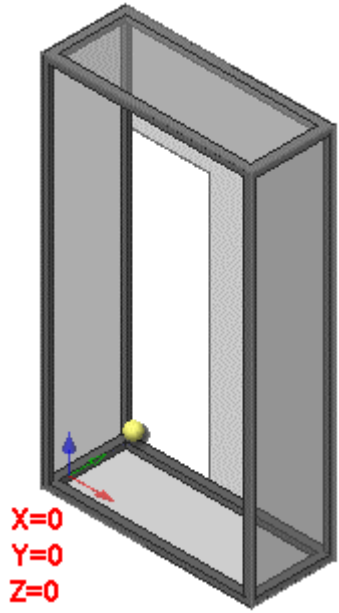
The figures show mounting panels placed with different offsets.

Mounting panel placed with

To right = 0

To front = 0

To top = 0

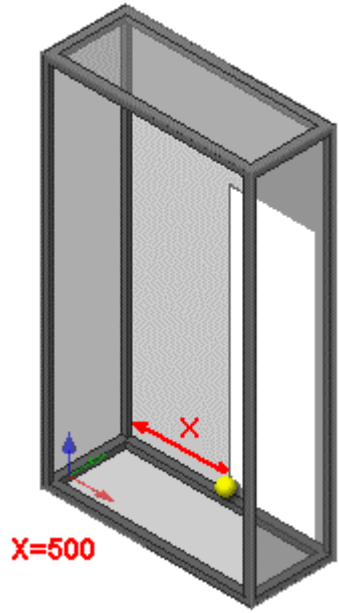


Mounting panel placed with

To right = 500

To front = 0

To top = 0

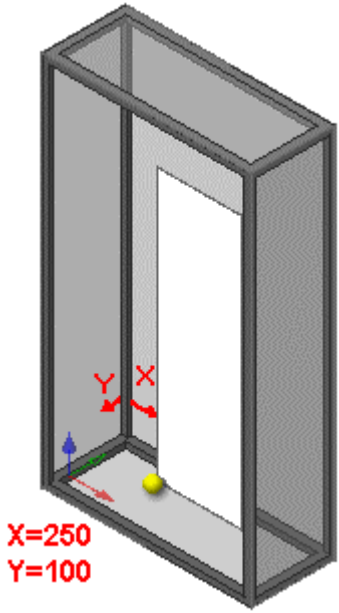


Mounting panel placed with

To right = 250

To front = 100

To top = 0

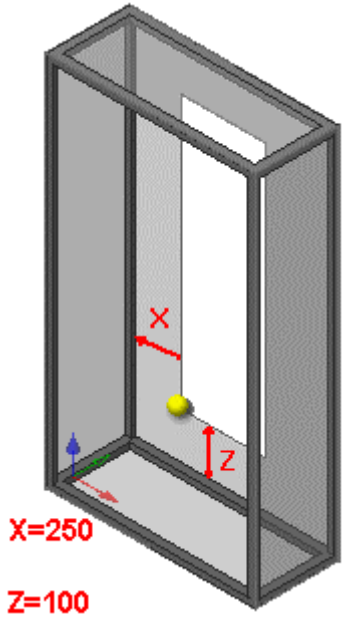


Mounting panel placed with

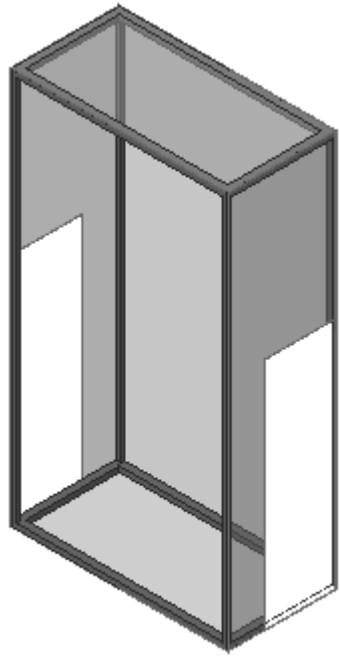
To right = 250

To front = 0

To top = 100



Mounting panels with location left and right

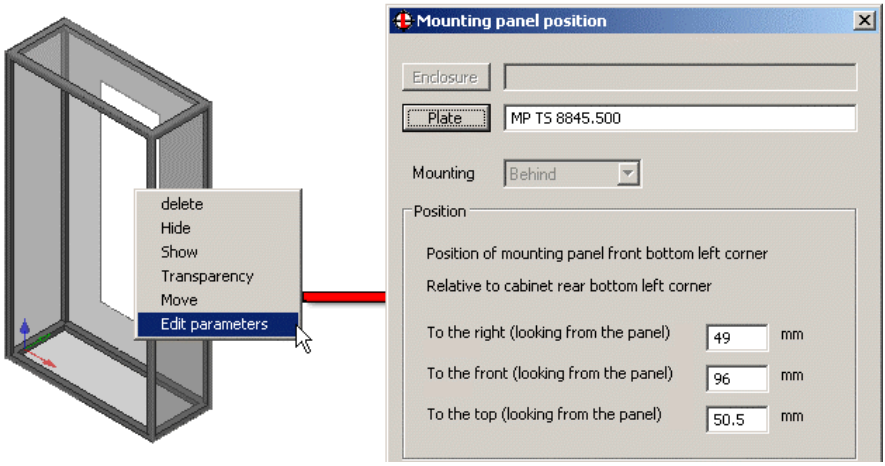


4.2.1 Modifying mounting panels after placement

A mounting panel's properties and parameters can be modified at any time after placement. Right-click the mounting panel in your drawing and choose 'Edit parameters'.

Drawing | Right-click mounting panel | Edit parameters

Navigator | Right-click mounting panel Edit parameters



Modifiable parameters:

- Panel model (select from database)
- Location
- X, Y and Z offsets

If there are other components on the mounting panel, they move with it when you change the location, angle or offset.

4.2.2 Editing a mounting panel in the database

The properties of mounting panels in the database can be modified. The panel will appear in all selections in its modified form.

Information, Organization and Security

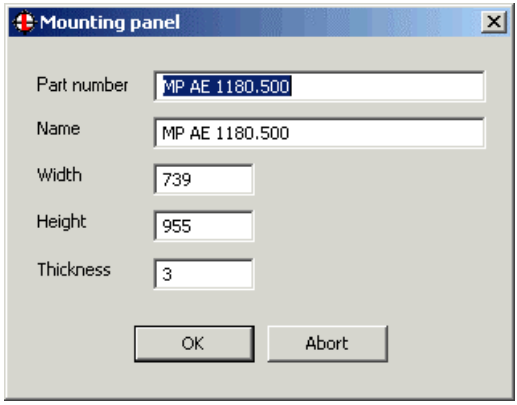
Update Components in Databases | Mount ng panels

Information/Security | Databases | Mount ing panels

Select a panel from the list.

[Edit]

Click [Edit] to open the mounting panel's edit dialogue and modify its properties.



Part no.	Part number in the database
Name	Name as displayed in the database browser
Width	Width of the mounting panel in mm
Height	Height of the mounting panel in mm
Thickness	Material thickness in mm

4.2.3 Creating a mounting panel in the database

Information, Organization and Security

Update Components in Databases | Mount ng panels

Information/Security | Databases | Mount ing panels

[New]

Click [New] to create a new mounting panel. The same dialogue is used for creating a mounting panel as for editing its properties. Type the values you want for the new panel.

Part no.	Part number in the database (required)
Name	Name as displayed in the database browser
Width	Width of the mounting panel in mm
Height	Height of the mounting panel in mm
Thickness	Material thickness in mm

Click [OK] to add the entry to the database. It will be listed for selection the next time you place a mounting panel.

4.2.4 Deleting a mounting panel from the database

Information, Organization and Security

Update Components in Databases | Mount ng panels

Information/Security | Databases | Mount ing panels

[Remove]

Click [Remove] and then [OK] to remove the selected mounting panel from the database. It will no longer be listed for selection.

4.3 Placing a mounting rail

A mounting rail is usually attached to a mounting panel or a cabinet profile. Before placing one on a mounting panel or a side panel, activate the component it will be assigned to.

Activate component and rotate view to show it front on, hiding all other components

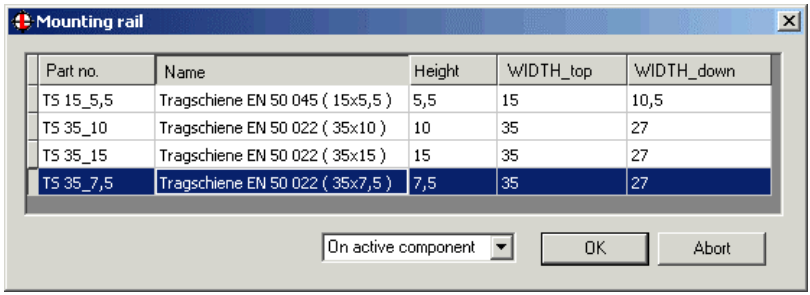
Activate component

Select and place component

Place mounting rail

Components | Mounting rail

The database contains various sizes of mounting rail. Click a record in the database browser to select the one you want.



There are two ways to activate a component:

Edge — To activate a component (to select it as the component on which the mounting rail will be placed), [Ctrl]-click one of its edges.

Surface — To activate a component, [Ctrl]-click one of its surfaces.

Mounting rails are always placed vertically or horizontally. There are several ways to enter the first placement point:

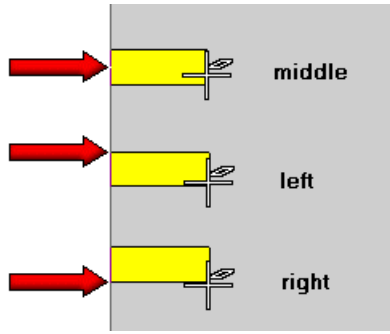
Point — <Search> for a point on a mounting panel or a cabinet profile or enter a user-defined point.

Construction — Enter X and Y values relative to the selected reference point.

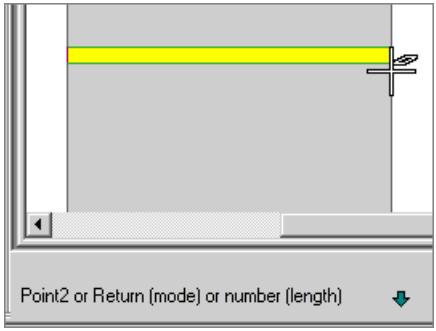
Stroke parallel component — If the mounting panel already features a mounting rail or a parallel duct, etc., you can place the new mounting rail parallel to it by stroking the existing component.

In either case, once you have entered the first point:

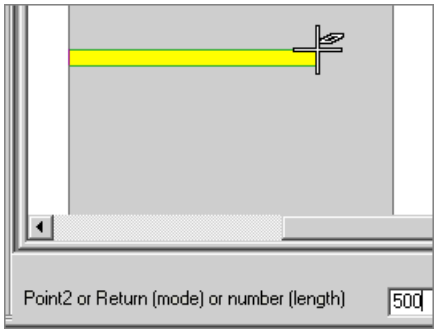
- Press [Return] to toggle the append point at the cursor from left to middle to right:



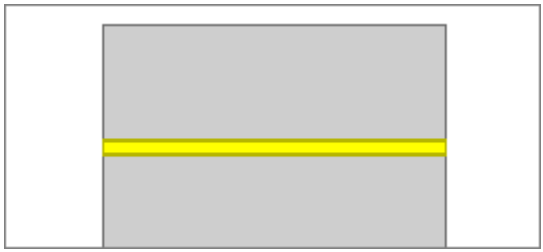
- Enter a user-defined second point or choose one by <search>ing:



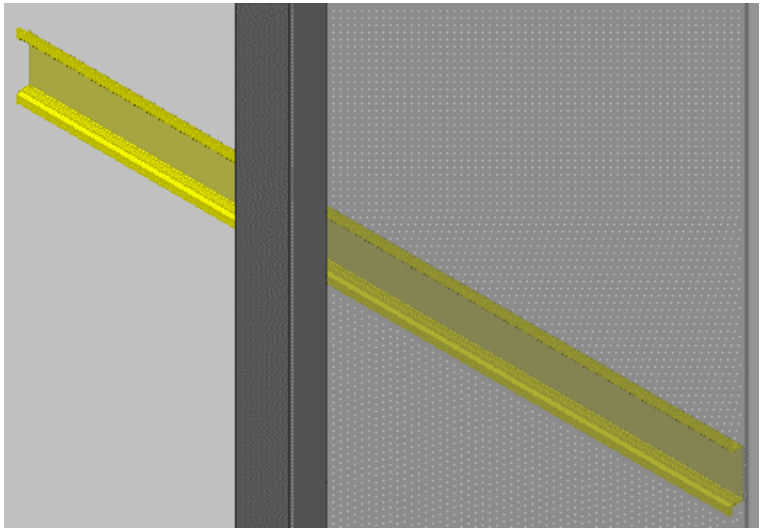
- Or enter a number to specify the length of the mounting rail:



The mounting rail is placed on the active mounting panel.



Isometric view of the rail:



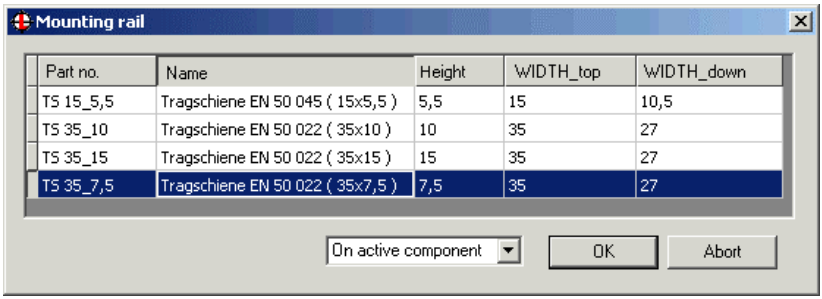
4.3.1 Creating mounting rails in parallel

The ability to place mounting rails in parallel to existing parts means you can very quickly fit a mounting panel with precisely placed rows of mounting rails.

Place mounting rail

Components | Mounting rail

Select the mounting rail you want from the database.



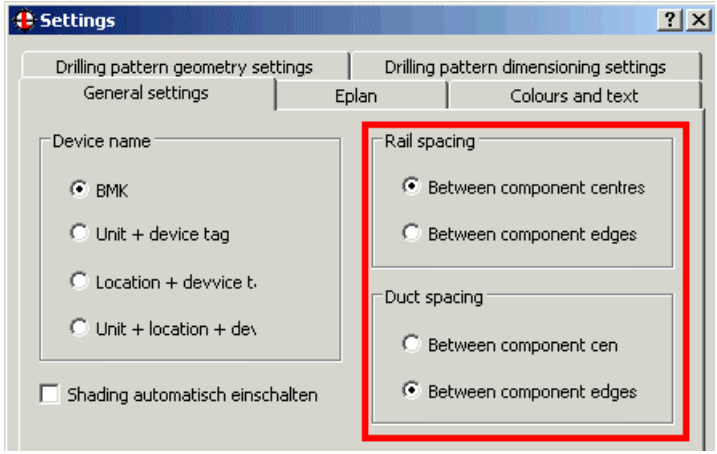
Click [OK] to confirm your selection.

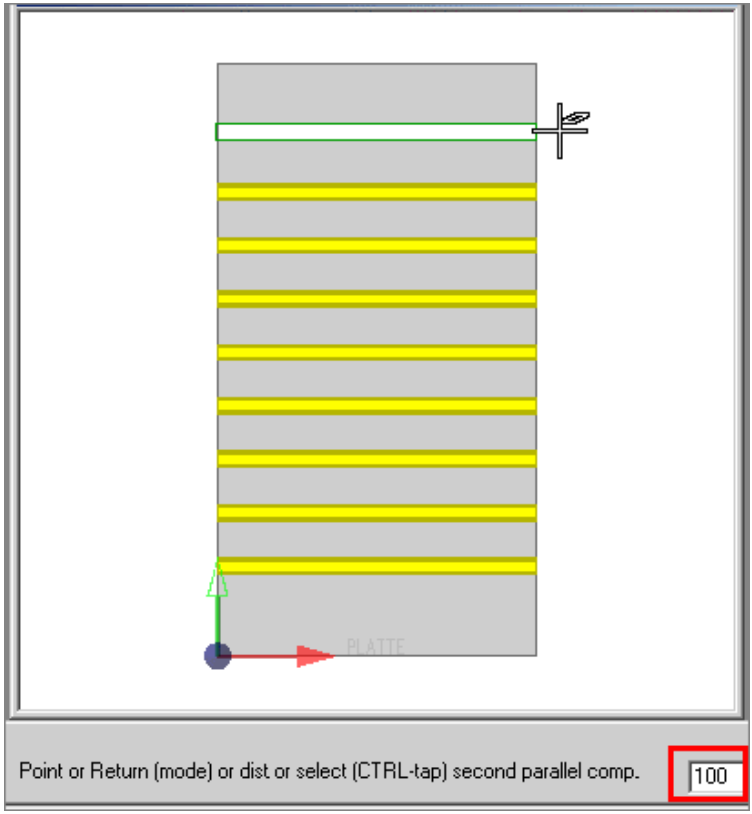
Press and hold the [Ctrl] key and tap a rail to use its length and position. The new rail can now be moved with the cursor. You can specify a placement point manually or determine it by entering a distance. You can also change the append point for the new rail (right-left-middle).

Calculation of spacing

A parameter specifies how the spacing is measured:

- Between centres
- Between edges



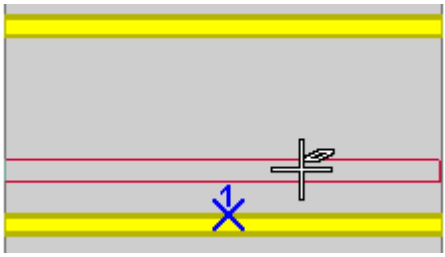

PLATTE
Point or Return (mode) or dist or select (CTRL-tap) second parallel comp.
100

Placing a mounting rail centred between parallel components

Pick a second parallel component (this can also be a duct or another rail) to place a new rail exactly centred between it and the first one.

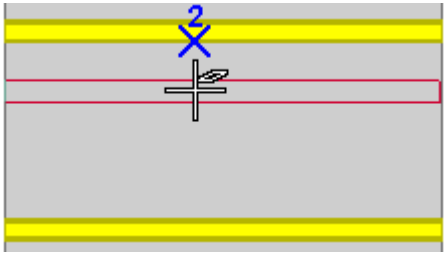
First point

[Ctrl]-tap an edge of the first rail.



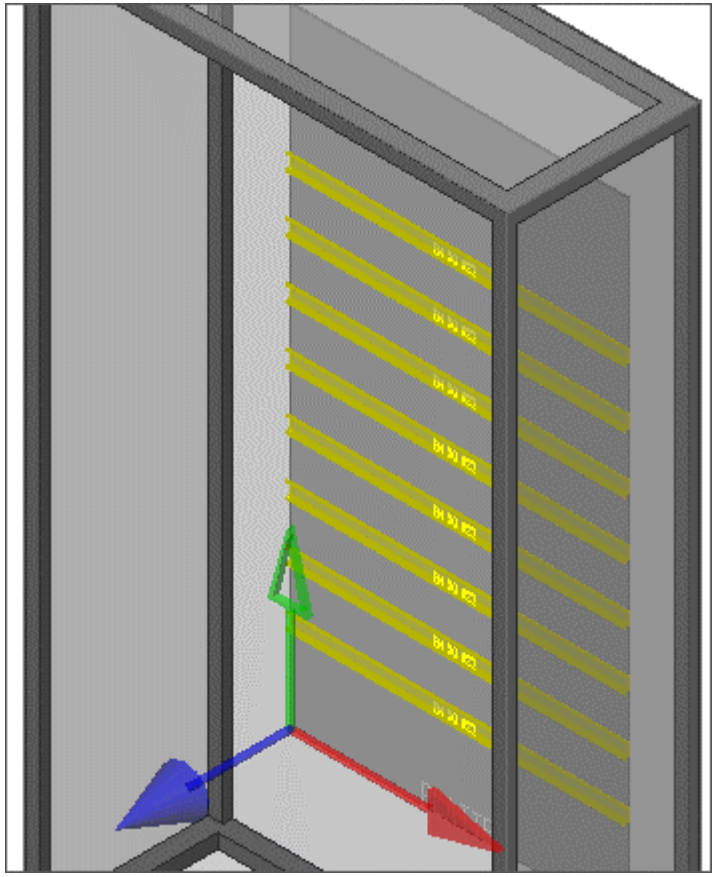
Second point

[Ctrl]-tap an edge of the second parallel rail.



The new rail is placed in the middle between the two.

Isometric view



4.3.2 Editing a mounting rail

There are two ways to modify a mounting rail:

- Modify the length using 'Modify component length'
- Changing the model by choosing a different one from the database

Right-click mounting rail | Edit parameters

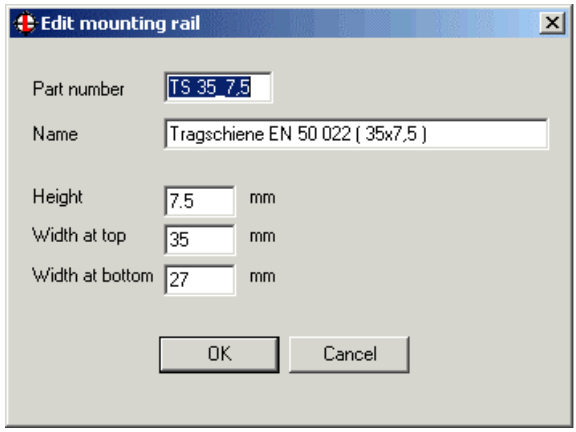
Choose a different model in the database browser. Click [OK] to replace the mounting rail with the selected model.

Editing/creating a mounting rail model in the database

Database | Mounting rails | [Edit]

Select the rail in the database browser.

- Click [Edit] to open a dialogue and modify the rail's parameters.
- Click [New] to open a dialogue and enter new parameters.



4.4 Placing a duct

Before placing a duct, you need to activate the component it will be assigned to.

Activate component and rotate view to show it front on, hiding all other components

Activate component

Select and place component

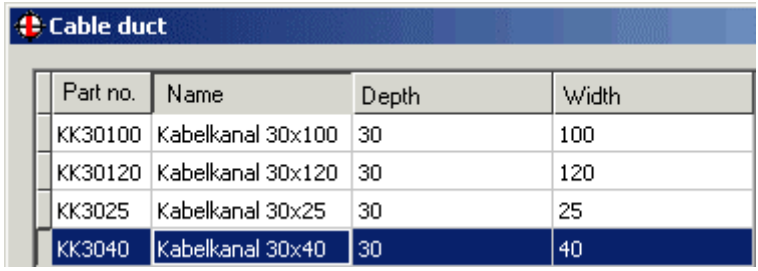
Place duct

Ducts are drawn in the same way as mounting rails. The same features apply:

- Select a model in the database browser
- <Search> for a point or enter a user-defined point
- Use [Return] to toggle the append point between middle/left/right
- <Search> for a second point or enter a user-defined point
- Alternatively enter a number to specify a length
- [Ctrl]-tap a component to create parallel copies (use [Return] to toggle the append point)
- [Ctrl]-tap two parallel components to place a duct between them

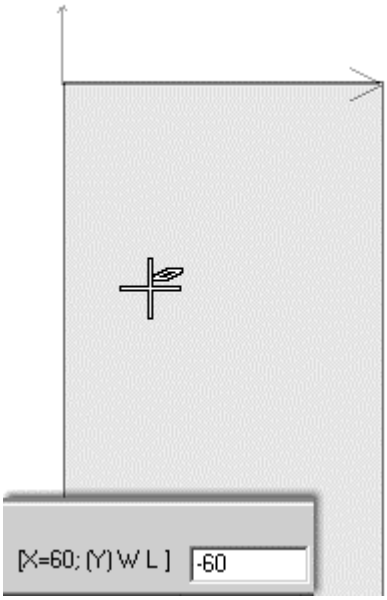
Drawing with ducts

Select the duct you want in the database browser.



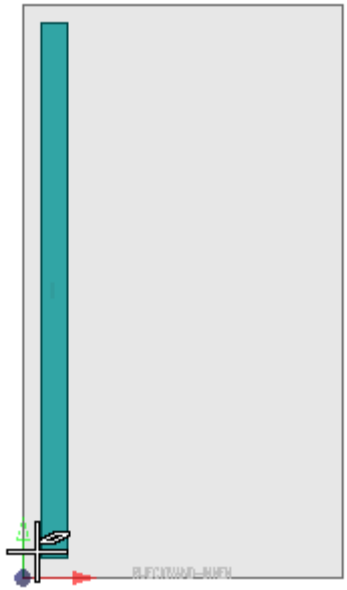
Ducts should be positioned slightly inward from the components they are mounted on to avoid overlaps and collisions. To ensure this, use relative reference points:

Press and hold [Alt Gr] and tap the top left corner of the rear panel. A coordinate cross is displayed there. For the first placement point, enter the relative coordinates X = 60 and Y = -60.

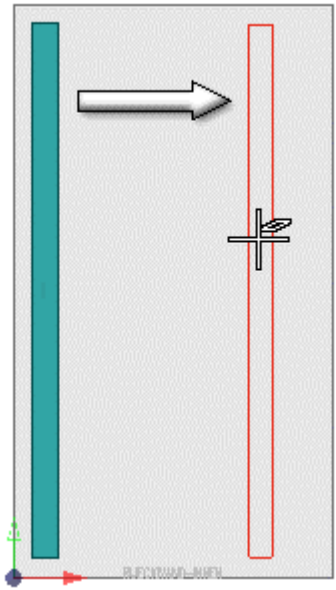


Press [Return] to switch the append point to the right so that the duct is definitely inside the enclosure.

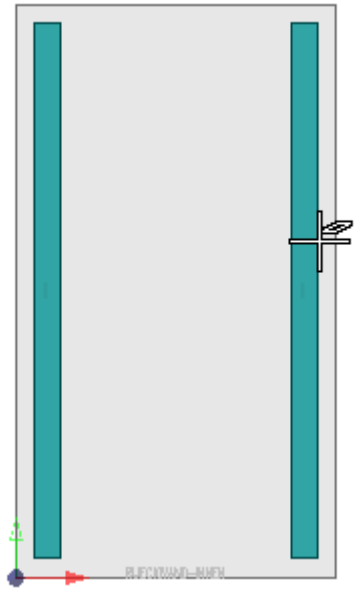
Set down the second point at a suitable distance from the bottom edge.



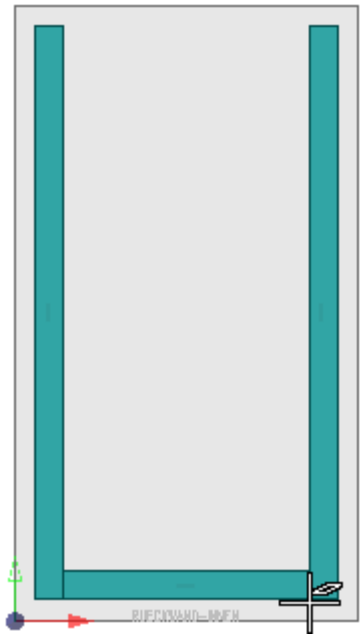
For the next duct, press and hold the [Ctrl] key and tap the first to create a parallel copy.



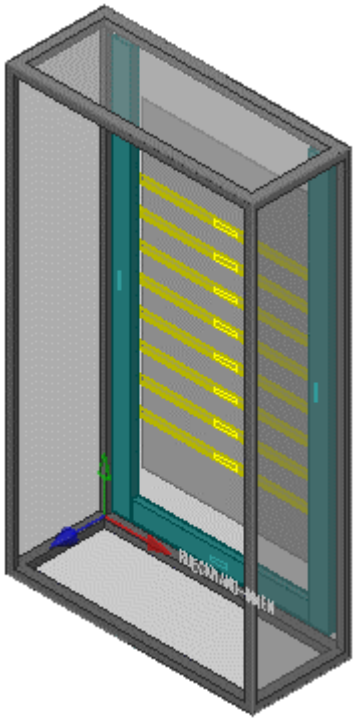
Switch the append point to the right and use the top right corner of the rear panel as a relative reference point with x = -60, Y = -60.



Complete the structure with a cross-link between the two vertical ducts (append point at left).



The isometric view shows the ducts on the mounting panel between the surrounding profiles.



4.4.1 Editing a duct

There are two ways to modify a duct:

- Modify the length using 'Modify component length'
- Changing the model by choosing a different one from the database

Right-click duct | Edit parameters

Choose a different model in the database browser. Click [OK] to replace the duct with the selected model.

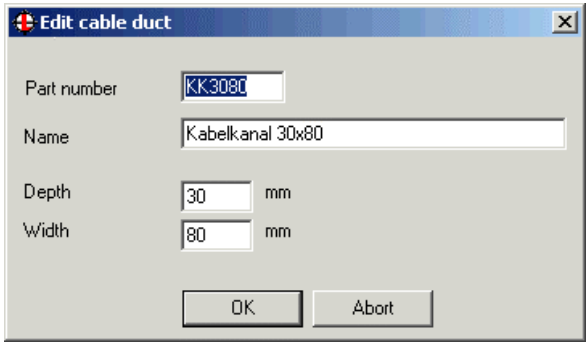
Editing/creating a duct model in the database

Right-click duct | 3D object parameters

Database | Ducts | [Edit]

Select a duct in the database browser.

- Click [Edit] to open a dialogue and modify the duct's parameters.



- Click [New] to open a dialogue and enter new parameters.

4.5 Placing a cable clamp rail

Place cable clamp rail

The cable clamp rail model is selectable in the database. These components are used to relieve strain on cables and can be fitted at various places in a cabinet. The placement method differs according to the location:

- On a mounting panel (placement on current component)
- On a user-specified plane

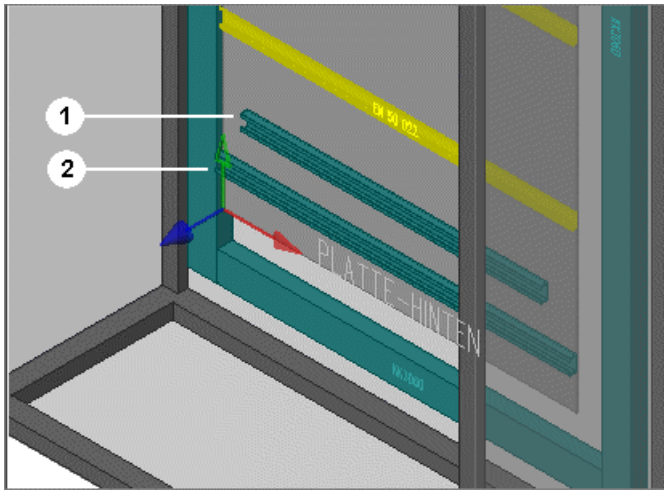
Placement on the current component

Activate the component you want, for example the mounting panel.

Activate component and rotate view to show it front on, hiding all other components

Activate component

Place the cable clamp rail by entering two points (**1**) or [Ctrl]-tap a parallel component (**2**).

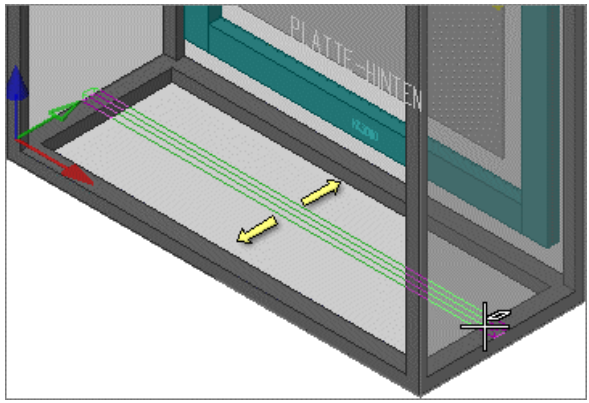


Placement on a user-specified plane

In the database browser, select the ‘free’ placement option. You are prompted to tap a surface, for example a floor profile on one side. Finally, enter the two endpoints for the clamp rail. The clamp rail is placed in the same plane as the specified surface.

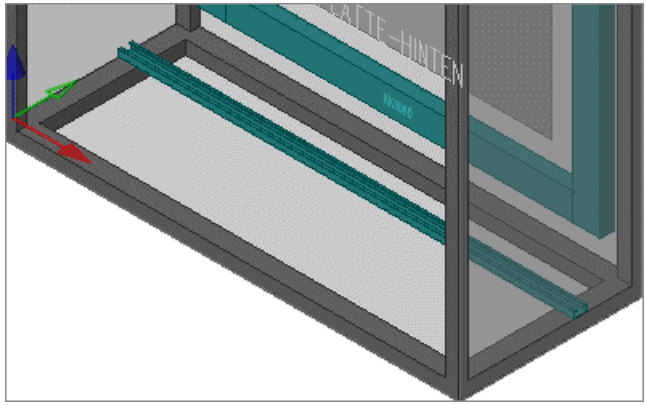
Parallel copies

When creating parallel copies, the clamp rail can only be moved in the current range.

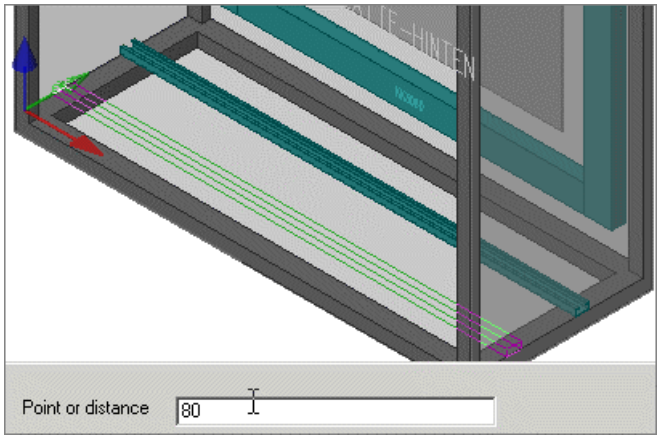


The append point at the cursor is in the middle and can be toggled between middle, left and right by pressing [Return].

Enter a point to place the clamp rail.



Alternatively, instead of entering a placement point, enter a spacing value. The next clamp rail is then displayed at the specified distance.



4.6 Placing busbar systems

Before placing a busbar system, you need to activate the component it will be assigned to.

Activate component and rotate view to show it front on, hiding all other components

Activate component

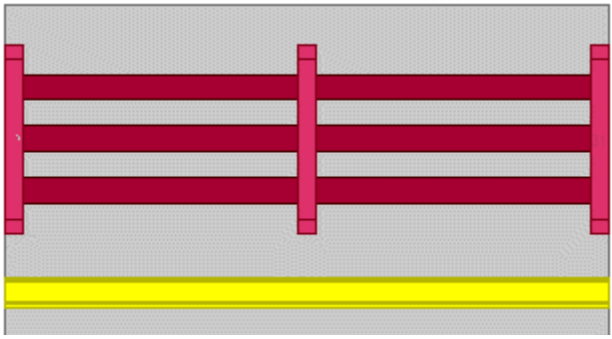
Select and place component

Place busbar

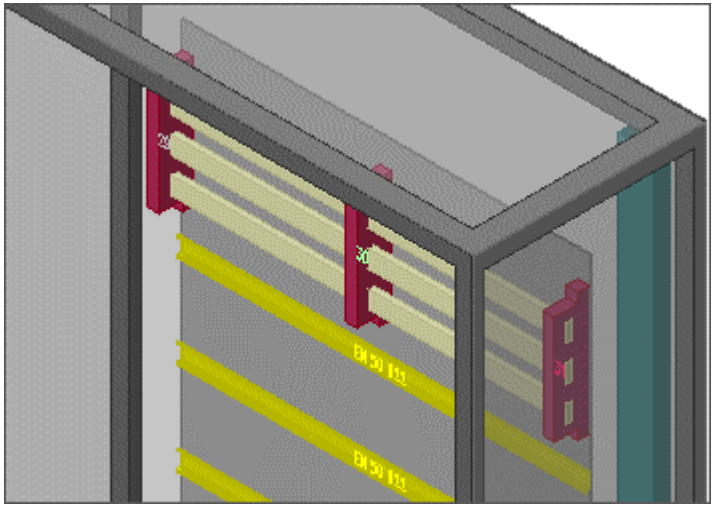
A busbar is placed similarly to a mounting rail:

- Select a model in the database browser
- Enter point 1
- Use [Return] to toggle the append point between middle/left/right
- Enter point 2 to place the busbar at a user-specified location
- Or enter a length
- Stroke a first parallel component to accept its orientation and length
- Stroke a second parallel component to place the busbar in the middle between the two

Once it has been placed, a busbar system's parameters can no longer be modified, but it can be moved and its length can be changed.



Busbar placed on mounting rail



Isometric view of busbar

4.7 Placing a user-specified component

User-specified components are specified in eCabinet and are part of the project hierarchy. They are represented in the drawing as a 3D CAD object by a graphical macro.

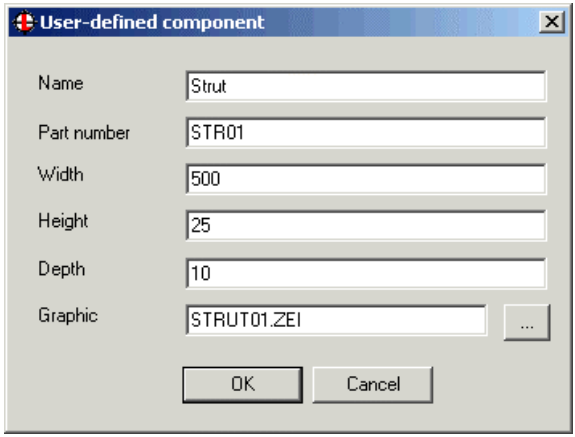
Typical uses are screws, bearings, eyelets, supports, struts, etc.

P ace

User-defined component

Component | U er-defined component

Make the necessary entries in the dialogue. The graphical macro must exist as a drawing object; if none is entered, a rectangular solid with the specified dimensions is placed.



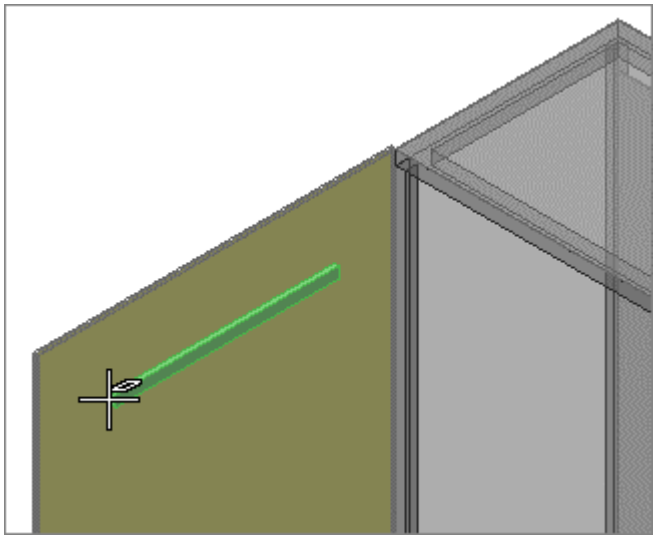
Placement options

Placement mode	Change active component • Edge • Surface • User-specified surface
Corrections	• Horizontal offset • Vertical offset • Mounting depth
Append point	• Bottom/middle/top • Left/middle/right
Collision checking	Can be disabled if needed. User-specified components can then be placed on top of and inside each other.

Placing

The component moves with the cursor and can be placed at any point within the current plane. Because it is part of the project hierarchy, it is included in any movement of the enclosure or component on which it is mounted.

User-specified component after placement (strut on rear of door)



4.8 Placing user-specified holes

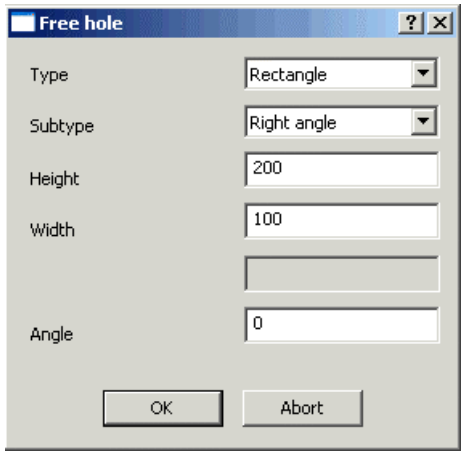
User-specified holes are openings in structural components. The openings can have various shapes and diameters. You need to activate a component before you can add an opening.

P acement **User-defined holes**	**Components | U ser-defined holes**

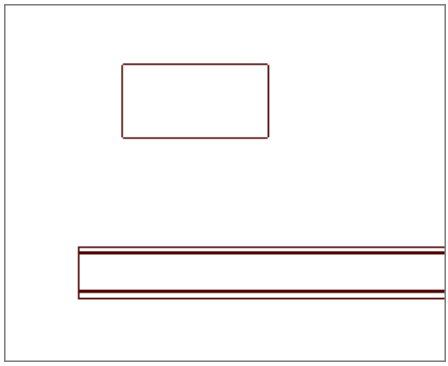
User-defined holes are available in various shapes:

- Drilled hole
- Thread
- Rectangle

Enter the parameters in the dialogue according to the selected type of hole.

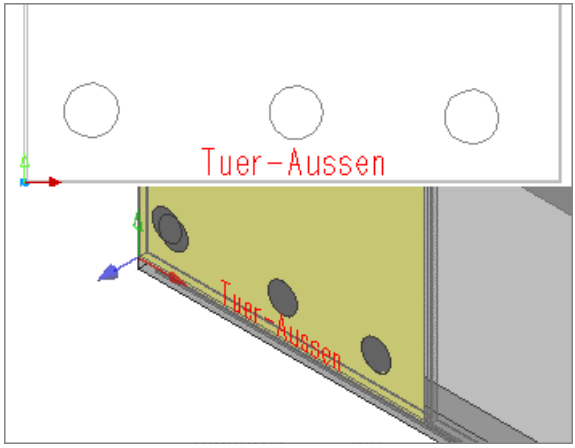


To place the hole, select a point on the active surface (e.g. mounting panel).



Example: Round hole

Enter a diameter in the dialogue and click [OK]. Place the hole with the append point at the middle at the location you want on the active component.



4.9 Changing the length of components

The ‘Change component lengths’ command lengthens or shortens structural components such as mounting rails, ducts, cable clamp rails and busbars.

Change component lengths **Edit | Change length**

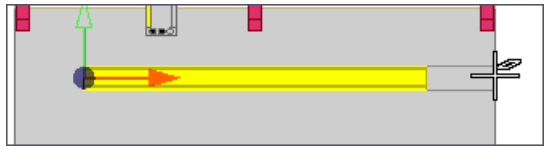
Tap the component you want to select it. The modification always applies to the end nearest to where you select the component.

There are two ways to modify the length of a component:

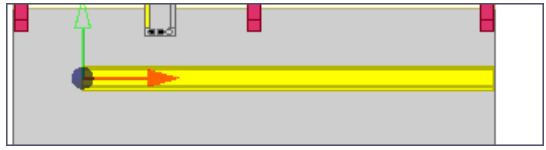
- By entering a point
- By entering a number

Changing the length of a component by entering a point

Pick the mounting rail by tapping the right-hand half of it. The right-hand end can now be modified.

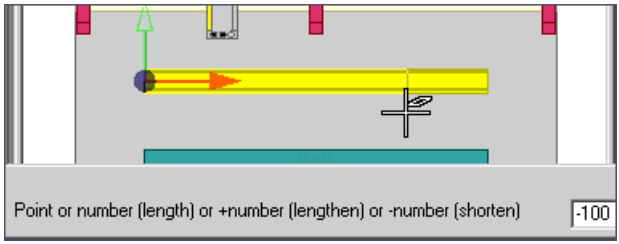


Enter a point or use <search> to find one. The component will be lengthened or shortened to the specified point.

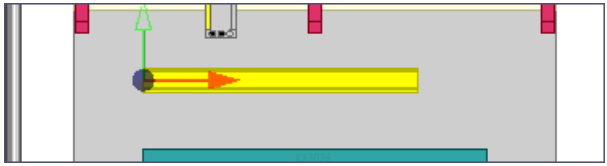


Changing the length of a component by entering a measurement in mm

After picking the component, enter a positive number to lengthen it or a negative number to shorten it.



In this example, the component is shortened by the specified length of 100 mm.



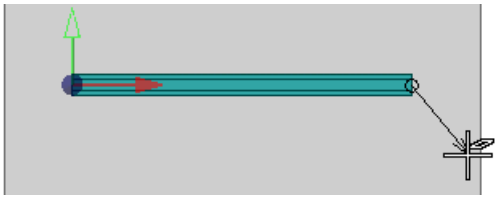
4.10 Moving a structural component

Move components	**Edit | Move components**

The Move command moves a component by its append point in a user-specified direction (diagonally), in the X direction only, or in the Y direction only. Movements in X and Y can be specified by entering X and Y offsets.

Moving in a user-specified direction (diagonally)

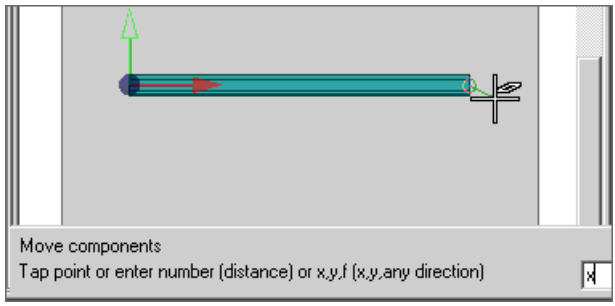
Tap the component you want to move.



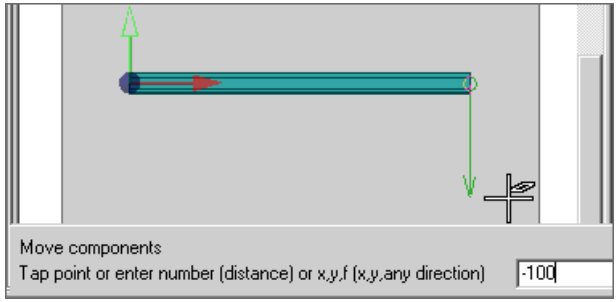
The default direction of movement is vertical. Enter an end point for the movement operation or use <search> to choose one. The component is moved so that its append point is at the specified end point.

Movement in X or Y

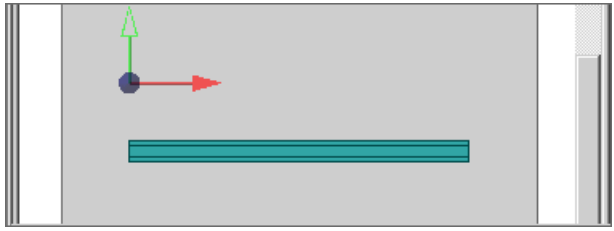
Tap the component you want to move. Pre-select the direction by typing **X**, **Y** or **F** (for 'free') and press [Return] to confirm.



The cursor is now constrained to the selected direction. Now enter the distance by which you want to move the component.



The component is moved in the chosen direction by the specified distance.



Press **F** if you want to return to a user-specified direction.

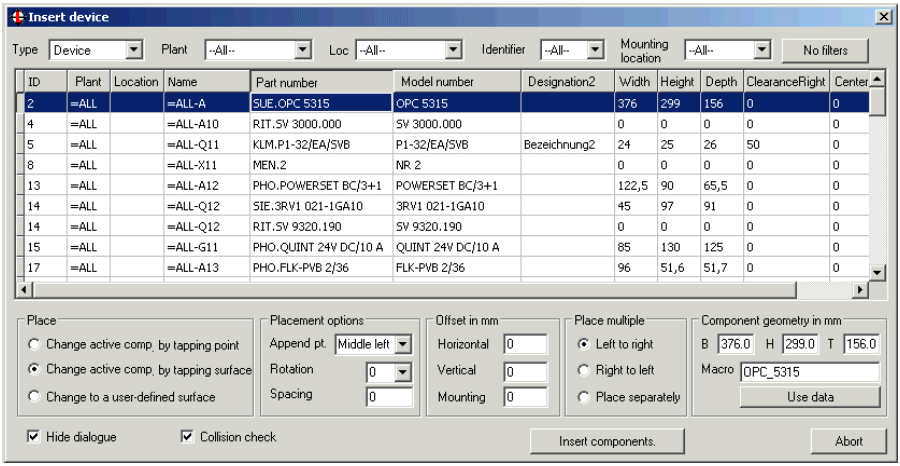
5 Placing and modifying devices

5.1 Placing and modifying a device

The electrical components that can be placed in an enclosure are determined in an EPLAN project and transferred to eCabinet. The transferred information is listed in the device browser. This also has all the options needed for placing devices in drawings.

Placement

Place devices



The dialogue has three areas

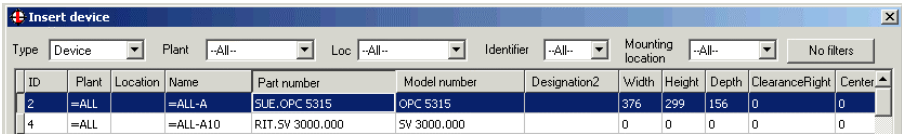
Filter Filters the list of devices by unit, location, identifier or mounting location; filter criteria can also be used in combination.

Components List of all available devices in the project list or those matching the current filter.

Placement options Settings for placement method, append point, offset, quantity, etc.

5.1.1 Using filters in the device browser

The five filters in the device browser limits the list of available devices to ones that match specific criteria such as unit, location, identifier or mounting location. Filter criteria can also be used in combination.



Any filters are cleared when you close the dialogue; the next time you open it, all available components are displayed again.

5.1.2 Selecting devices in the device browser

The middle area of the device browser lists all available devices in the project parts list or those which match the current filter. Devices that have already been placed in the drawing are no longer listed.

ID	Plant	Location	Name	Part number	Model number	Designation2	Width	Height	Depth	ClearanceRight	Center
2	=ALL		=ALL-A	SUE.OPC 5315	OPC 5315		376	299	156	0	0
4	=ALL		=ALL-A10	RIT.SV 3000.000	SV 3000.000		0	0	0	0	0
5	=ALL		=ALL-Q11	KLM.P1-32/EA/SVB	P1-32/EA/SVB	Bezeichnung2	24	25	26	50	0
8	=ALL		=ALL-X11	MEN.2	NR 2		0	0	0	0	0
13	=ALL		=ALL-A12	PHO.POWERSET BC/3+1	POWERSET BC/3+1		122,5	90	65,5	0	0
14	=ALL		=ALL-Q12	SIE.3RV1 021-1GA10	3RV1 021-1GA10		45	97	91	0	0
14	=ALL		=ALL-Q12	RIT.SV 9320.190	SV 9320.190		0	0	0	0	0
15	=ALL		=ALL-G11	PHO.QUINT 24V DC/10 A	QUINT 24V DC/10 A		85	130	125	0	0
17	=ALL		=ALL-A13	PHO.FLK-PVB 2/36	FLK-PVB 2/36		96	51,6	51,7	0	0

Tap the device you want to select it. Its parameters are copied to the bottom part of the dialogue. You can also choose multiple devices by holding down [Shift] or [Ctrl] as you select them. They will be placed in the order they are selected.

A height, width and depth should be specified for each component in the parts database.

5.1.3 Placement options when placing devices

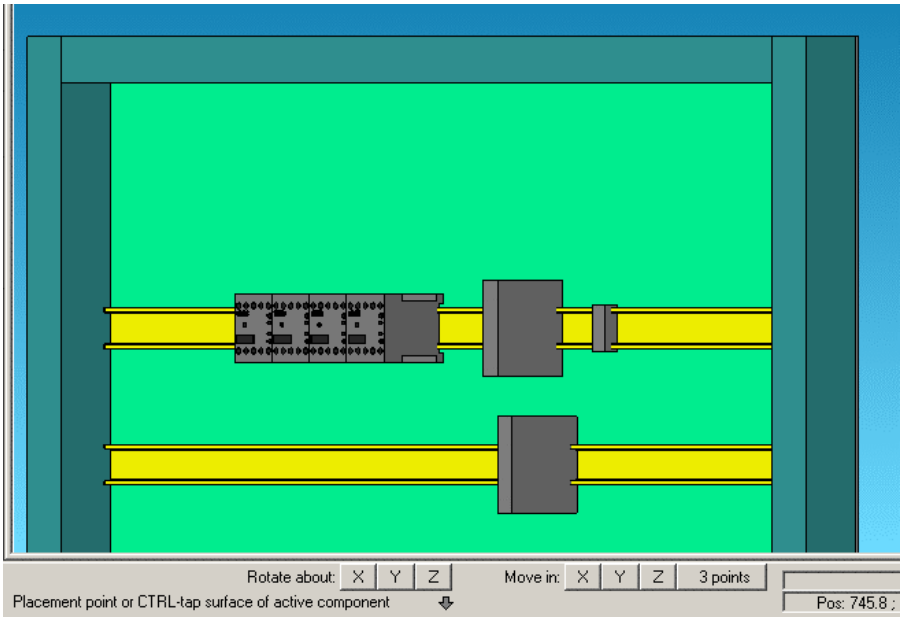
Placement mode

Before you place a device, it is possible to change the structural component (e.g. mounting rail) on which the device will be mounted. The 'Insert device' dialogue specifies how the active component is selected.

Change active component by tapping edge

To change the active component, [Ctrl]-tap one of its edges. The device can then only be moved on the active component and is placed by entering a point.

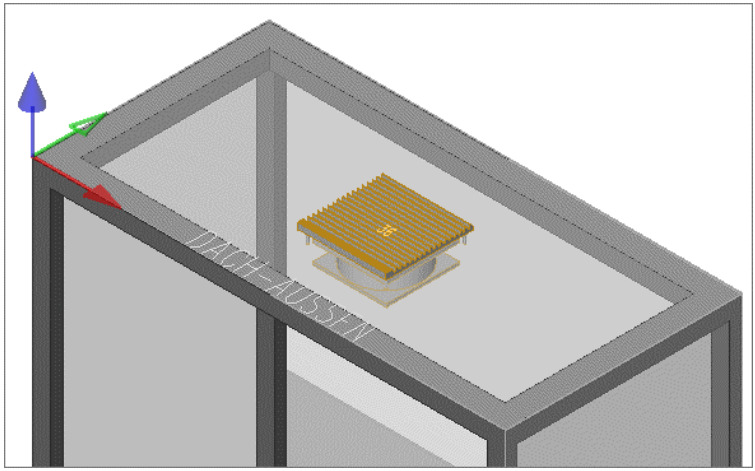
The figure shows the active component being changed from the upper to the lower mounting rail:



Change active component by tapping surface

This placement mode can be used by flat structural parts such as mounting panels, side panels and doors. [Ctrl]-tap the surface. The device can then be moved in the same plane as and placed on the picked component.

Example: Fan in the top panel:

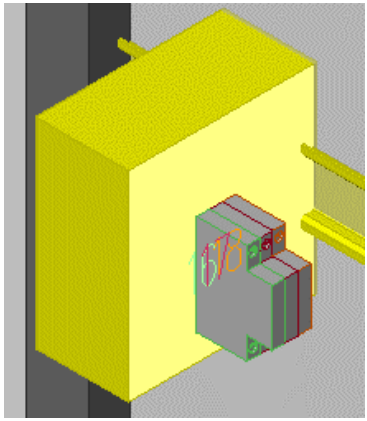


Change to a user-defined surface

This placement mode is used by way of exception to place devices on surfaces of other components.

[Ctrl]-tap the surface.

This mode only remains active for placement of one component. It then switches back to edge mode.



Append point

The default append point for inserting devices is middle left. It is assumed that subsequent components will be placed from left to right.

There are nine possible positions for the append point.

Offset

The three offset values specify how far from the selected append point the device is placed.

Horizontal	Positive values: offset to the right
Vertical	Positive values: upward offset
Mounting depth	Positive values: forward offset

Geometry

Width/height/depth	Dimensions of the device
Spacing	Specifies the spacing between devices when several are placed at the same time.
Rotation	Devices can be rotated in steps of 90° on placement.
Graphical macro	Name of the graphical drawing defining the device. If there is not a specific graphical macro, a default box is placed with the specified dimensions.

Direction of placement for multiple components

When multiple components are selected, the devices can be placed beginning from the placement point from left to right (default) or from right to left.

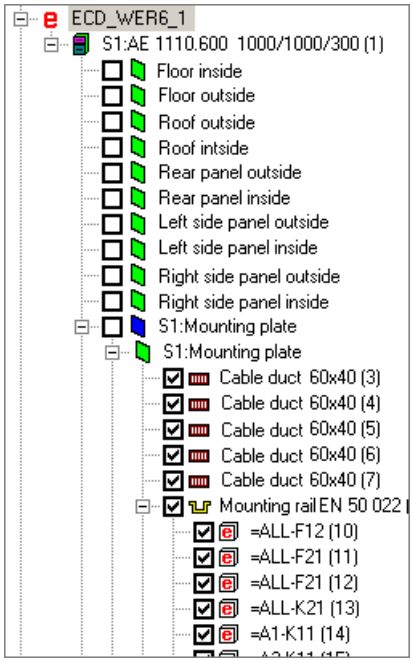
Collision checking

Collision checking prevents components from being placed in and around each other. It is always enabled by default and should only be disabled in exceptional instances.

Inserting/placing devices

Click [Insert devices]. The device selected in the list is displayed with the current placement options and can be moved around the screen. Enter a point to finally place it in the drawing.

Placed devices are assigned to the active mounting plane. The hierarchy is shown in the Navigator.

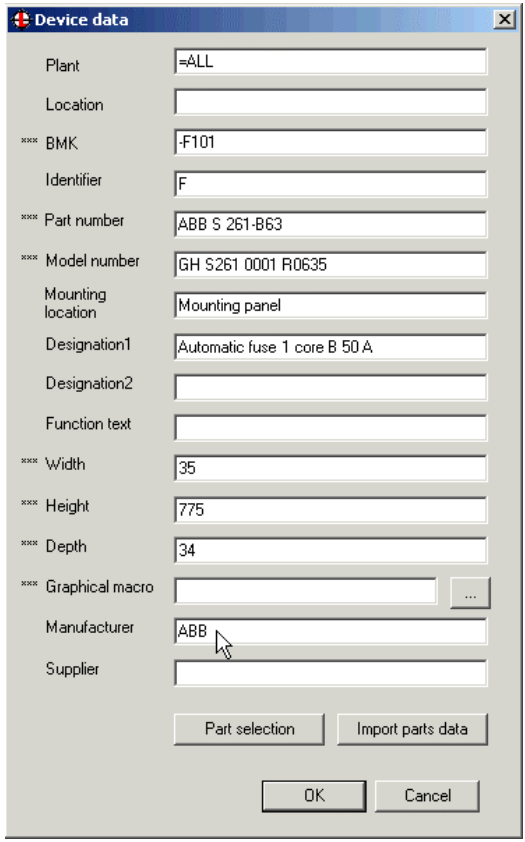


5.2 Inserting a universal part

Universal parts are devices that are not in the EPLAN parts list. They are specified and assigned information when selected and placed in a drawing like a device.

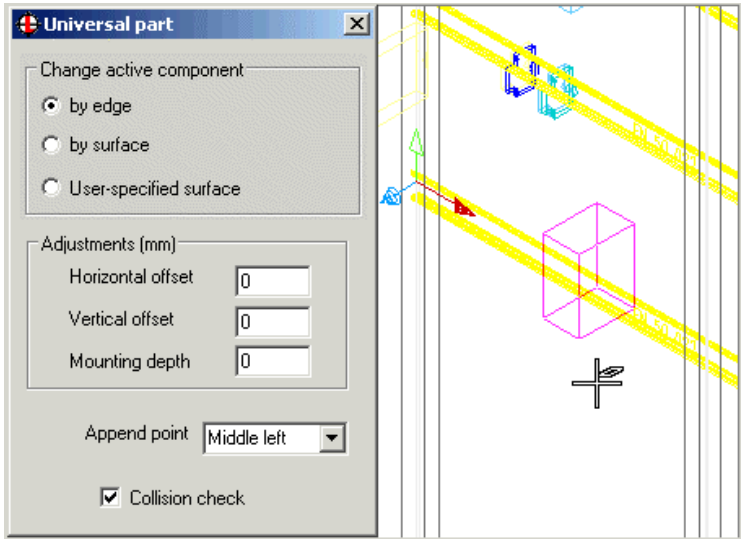
Placement

Insert universal part



Enter the parameters in the dialogue as you want them. The device tag, article no., model no. and height/width/depth or graphical macro fields must be filled; the other fields are optional.

Click in the drawing where you want to place the component. A dialogue opens with the usual placement options for you to select the append point, active component and placement direction for multiple components.



5.3 Specifying a blocking surface

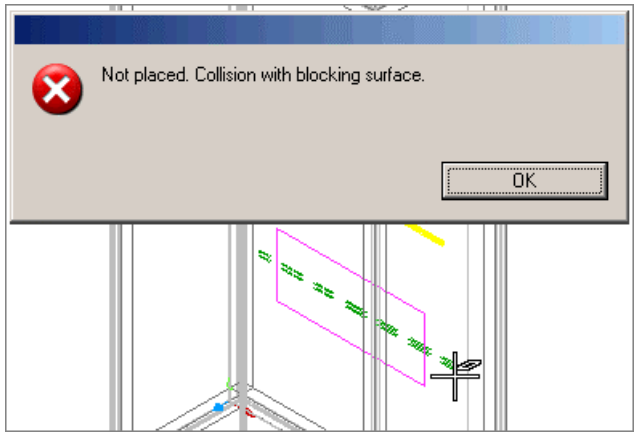
Blocking surfaces are areas of mounting planes (e.g. mounting panels) where no components or devices can be placed. They are set aside for adding components, fasteners, drilled holes etc. at a later stage.

Information, Organization and Security

Place blocking surface

First activate the component that is to contain the blocking surface. Specify the blocking surface by entering two corner points of a box. You are then asked if you want to make the blocking surface part of the component's master data. Choose 'Yes' if you want the mounting panel to be given the blocking surface whenever it is used in a drawing.

Any attempt to place a component on the blocking surface is detected by the collision checking routine.



Deleting a blocking surface

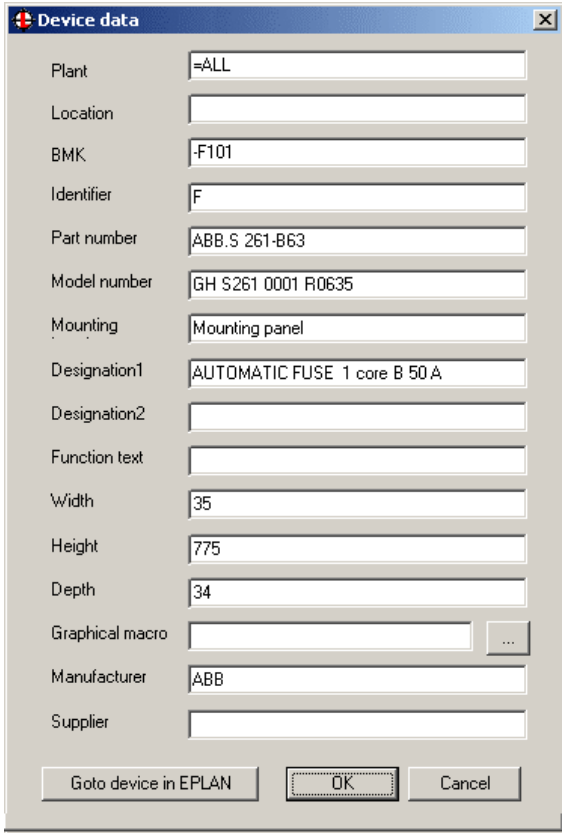
To remove a blocking surface, right-click it to open the context menu and choose 'Delete'. You are asked if you want to remove the blocking surface from the component's master data.

5.4 Displaying device information

Information, Organization and Security

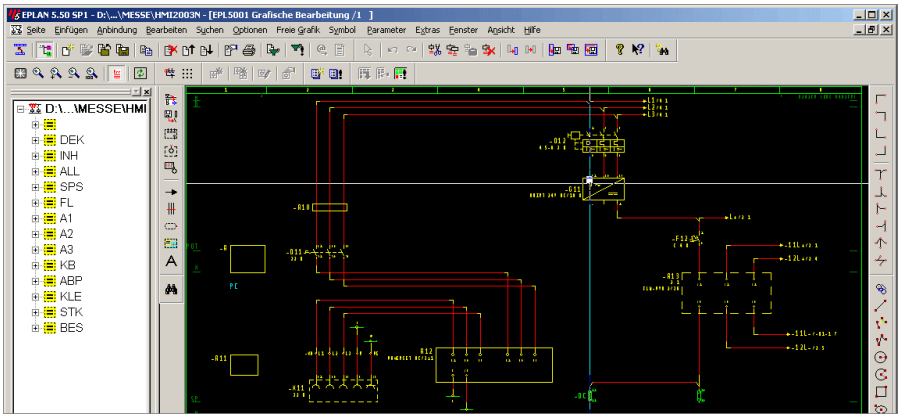
Component information

Tap a device in your drawing. The dialogue shows all information from the parts list or in the case of a universal part, all parameters entered when the part was created.



5.4.1 Go to component in EPLAN

Click [Go to component in EPLAN] in the Device Information dialogue to go straight to the component in your EPLAN project. If the component is used several times in your drawing, a selection is displayed. Select the device you want. The appropriate page is opened in EPLAN with the cursor on the selected device.



5.5 Modifying devices

5.5.1 Place devices again

Edit

Place devices again

This command moves devices that have already been placed in a drawing. The dialogue contains the same options as are available in the device browser when inserting devices:

- Activation method for the reference component
- Horizontal/vertical offsets
- Append point (not applicable for parts placed with a graphical macro)
- Placement sequence for multiple components
- Placement mode for multiple components (retain spacing/realign)
- Collision checking

Enter the values you want and tap the component to be modified.

5.5.2 Adjusting the position of a device

Edit

Move device

You can move a device in a drawing by using ‘Move device’ to specify adjustments. Enter the values you want in the dialogue:

- Horizontal offset
- Vertical offset
- Mounting depth offset
- Rotate by 90°, 180° or -90°

Then tap the component to be moved. It is repositioned in accordance with the specified values.

5.5.3 Moving a component

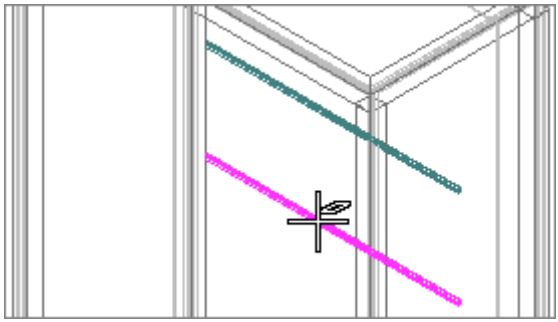
This command moves a component in a drawing in X, Y or a user-specified direction or by a specified distance.

Edit

Move component

Tap the component you want to move. If you want to move several components at once, select them in rectangle mode.

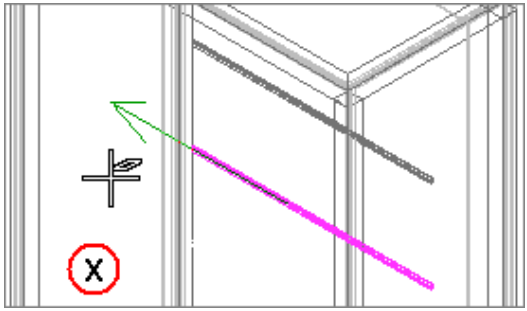
Tap the append point of the component to which the movement will relate.



Direction of movement

When you first select the command, the default direction of movement is along the Y axis. You can see this from the fact that when you move the cursor, an arrow is displayed can only move up and down.

[X] key Changes the direction of movement to the X axis. The arrow at the cursor can only be moved left and right.

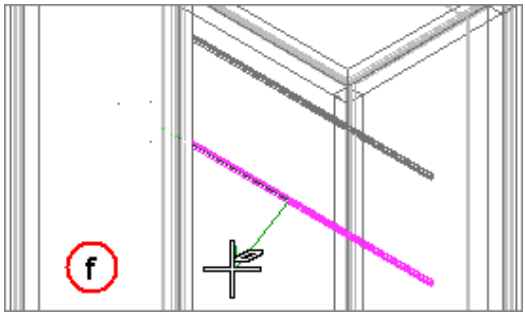


[Y] key

Changes the direction of movement to the Y axis. The arrow at the cursor can only be moved up and down.

[f] key

Changes the direction of movement to ‘free’. The arrow at the cursor can be moved in any direction.

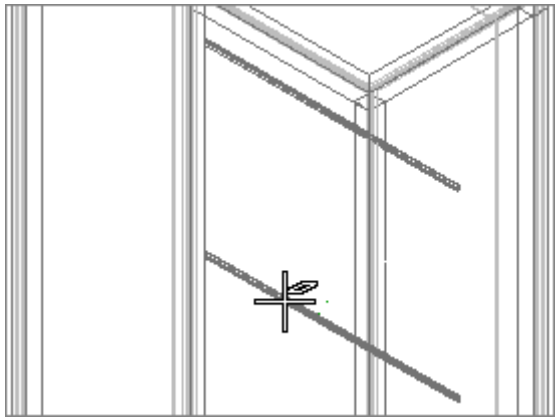


End point for movement

Once you have chosen the direction of movement, enter an end point. You can do this by entering:

- A point
- A number (positive or negative)

The selected components are moved to the new position.



The command then remains active so that you can immediately go on to move the component again or a different component.

5.5.4 Copying an enclosure

Use this command to create a copy of an entire enclosure including its structure and fittings within the same drawing.

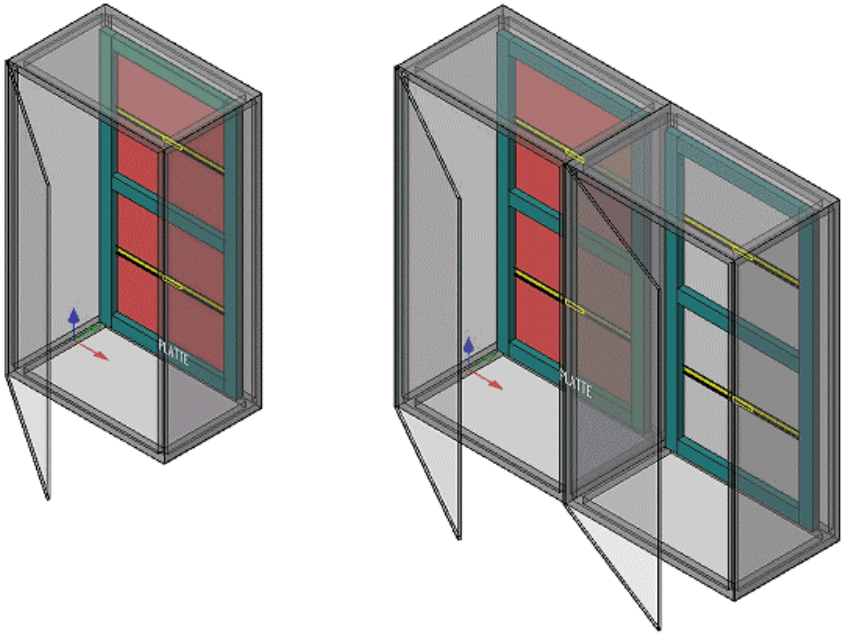
Before copying an enclosure, change to isometric view so that it is fully visible.

Edit

Copy enclosure

Tap the enclosure to be copied and enter a placement point for the copy. You can enter a new unit and location designation in a dialogue. Confirm the dialogue with [OK].

The copy is created with this point at the rear left.



5.5.5 Changing the plant designation/unit ID

Edit

Change plant designation

Changes the plant designation/unit ID throughout the project. In the dialogue, the old (current) designation is shown at the top. Type the new designation below.

5.5.6 Changing the location designation/ID

Edit

Change location designation

Changes the location designation/location ID throughout the project. In the dialogue, the old (current) designation is shown at the top. Type the new designation below.

5.5.7 Deleting devices

Edit

Delete

Single component

To delete a device or a structural component from the drawing, choose the Delete command and tap the device or component.

If other components are associated with the selected one, a confirmation prompt prevents the associated components form being deleted by mistake. Choose 'Yes' if you want to delete all associated components as well.

Multiple components selected in rectangle mode

To delete multiple components, hold down the left mouse button slightly longer and drag a rectangle around the components you want to delete. Enter a point for the second corner of the rectangle.

- If you click *inside* the rectangle to confirm, all elements that are fully inside it will be deleted.
- If you click *next to* the rectangle to confirm, all elements within, overlapping or touching the rectangle will be deleted.

Multiple components selected in polygon mode

Instead of a rectangle, you can also use a polyline to mark out the elements to be deleted. Select the Delete command, press the left mouse button and hold it down until a prompt appears asking you to enter a point for a polygon.

Now enter a number of points connected with polygon lines, pressing [Return] when you have done to complete the polygon. Finally, select the elements to be deleted as in rectangle mode, by tapping inside or next to the polygon.

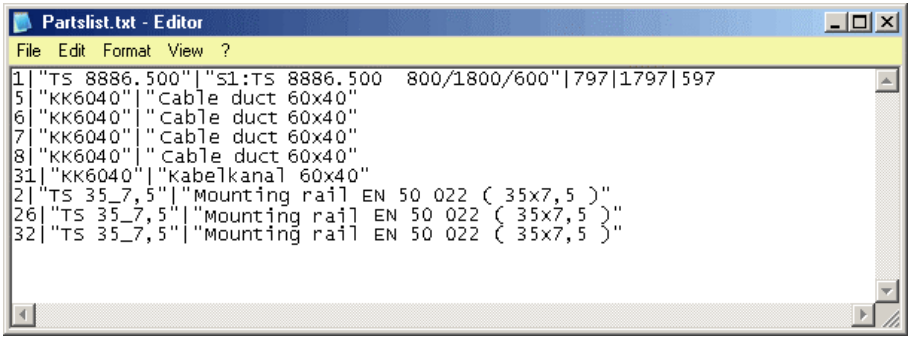
6 Project and system organization

6.1 Projects parts list

Many different types of object can be inserted in eCabinet drawings – enclosures, ducts, rails, devices, drilled holes, etc.

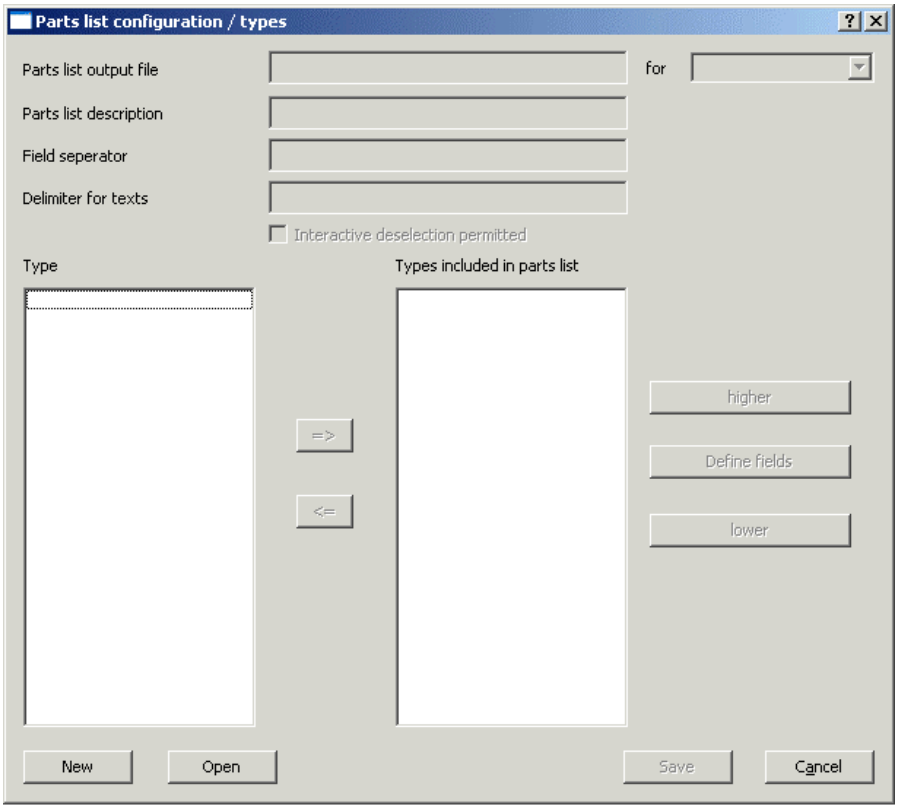
The project parts list can be used to produce a range of different analyses covering a whole project. The output can be controlled using configuration files to generate different types of analyses and so different lists.

Parts lists are written to a file and stored in the project directory.

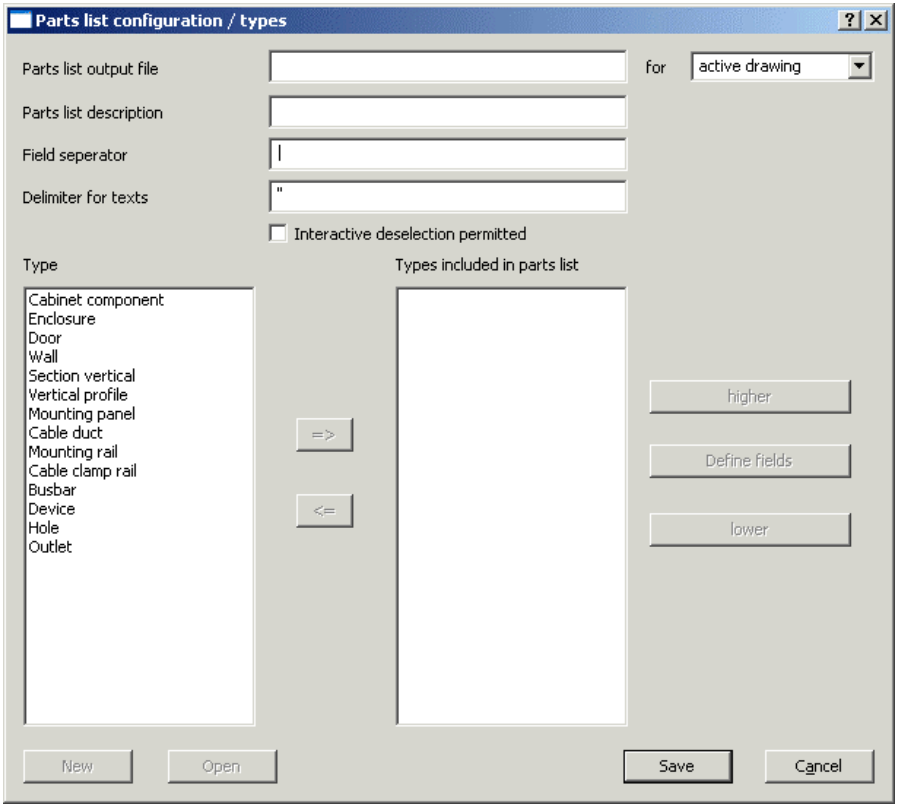


6.1.1 Project parts list: Configuration

Each user can have a custom set of parts list settings. The configuration is stored in a .BOM file. To configure the parts list, choose Edit – Project Parts List/ASCII on the menu and select the **Configuration** command.



Click the [New] button to create a new configuration.



First, enter a file name for the parts list.

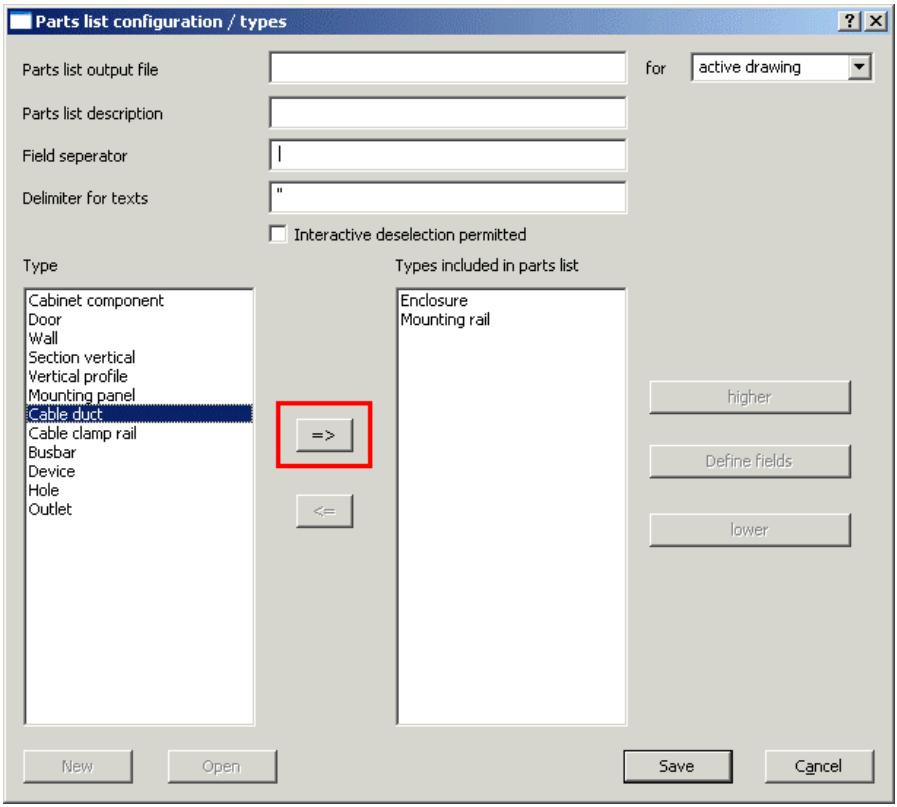
Note: Only specify the file name (for example 'partslist.txt'), not the directory. The file will be automatically saved in the project directory.

If required, type a description for the parts list.

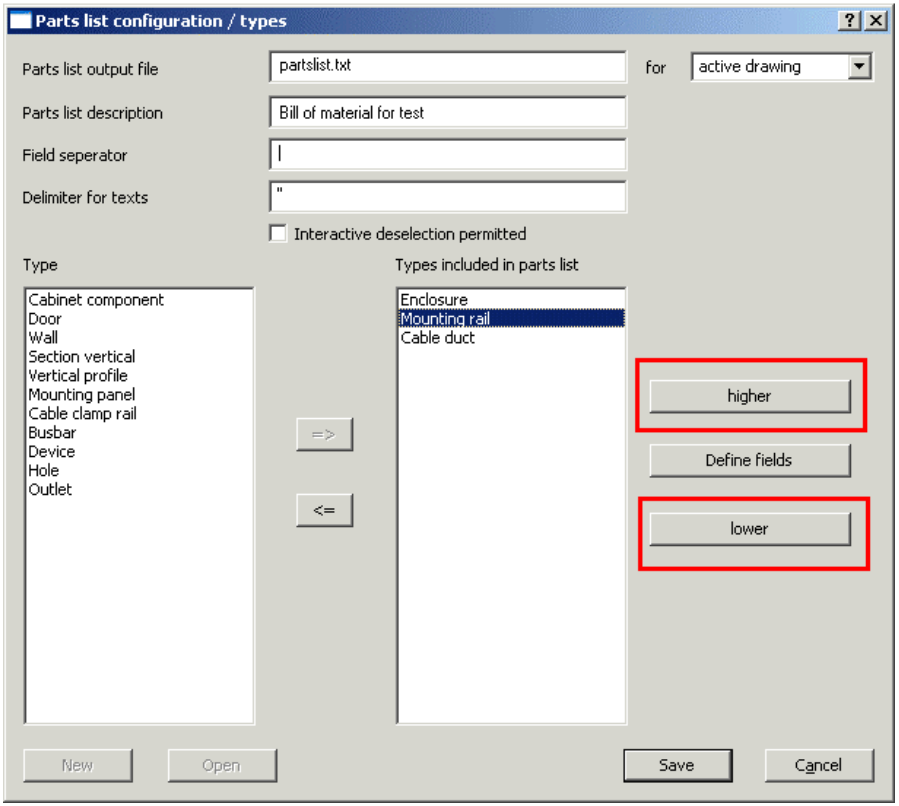
You can also choose the separator character for all text parameters and for all fields in printed output.

Under 'Type List', you see a list of all object types that are used in eCabinet and can be included in a parts list.

Select the types you want to include in the parts list and click the [=>] button. Repeat until all the types you want are selected.

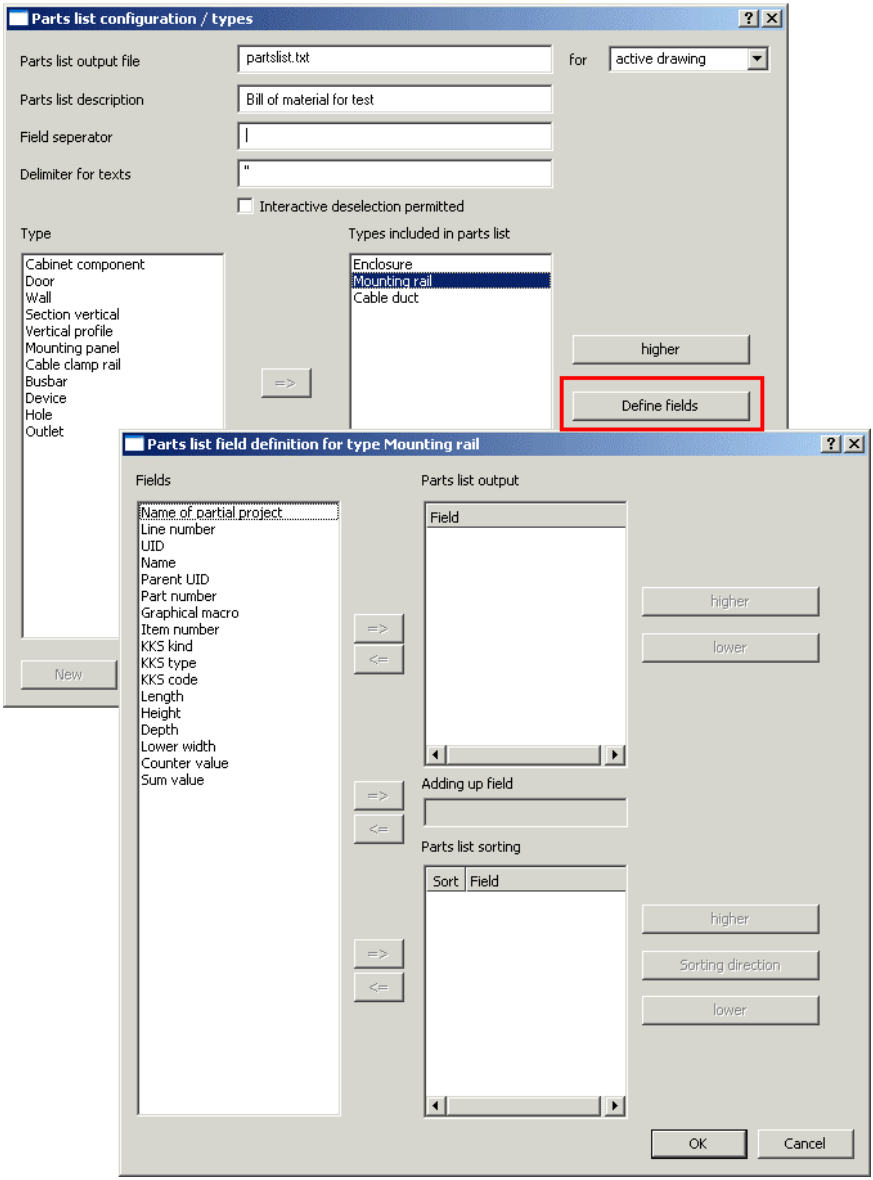


Use the [up] and [down] buttons to change the order of items in your selection.



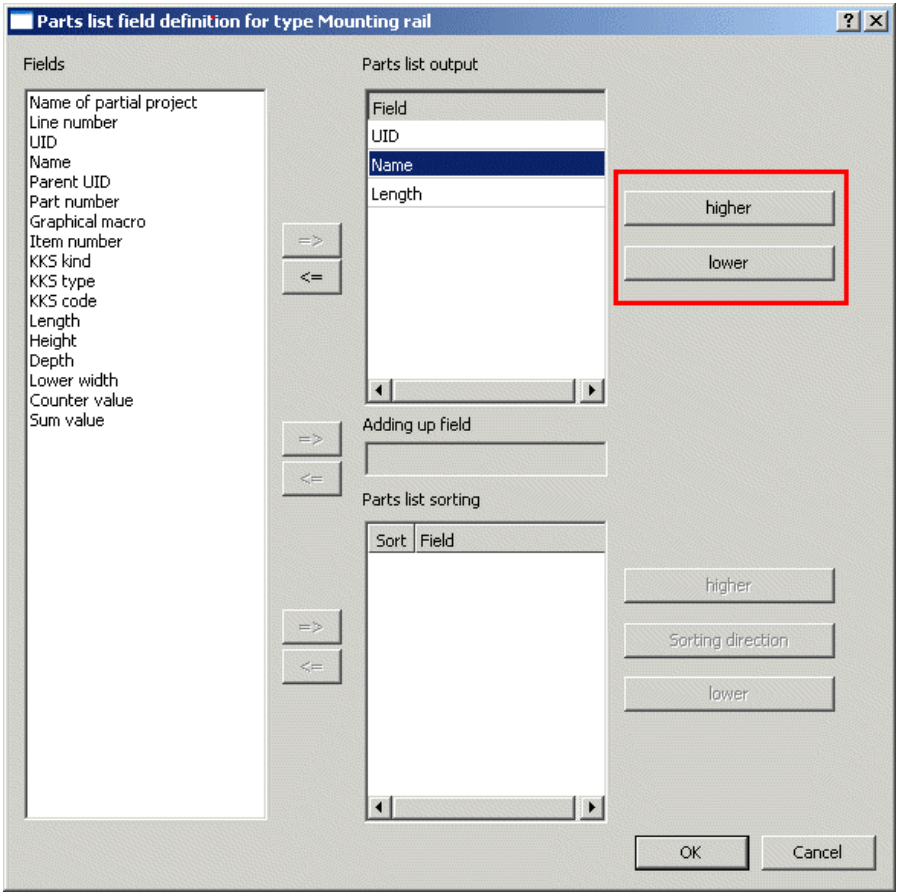
Use the [Define fields] button to specify the object properties to analyse for the parts list.

Select the object you want to select and click [Define fields].

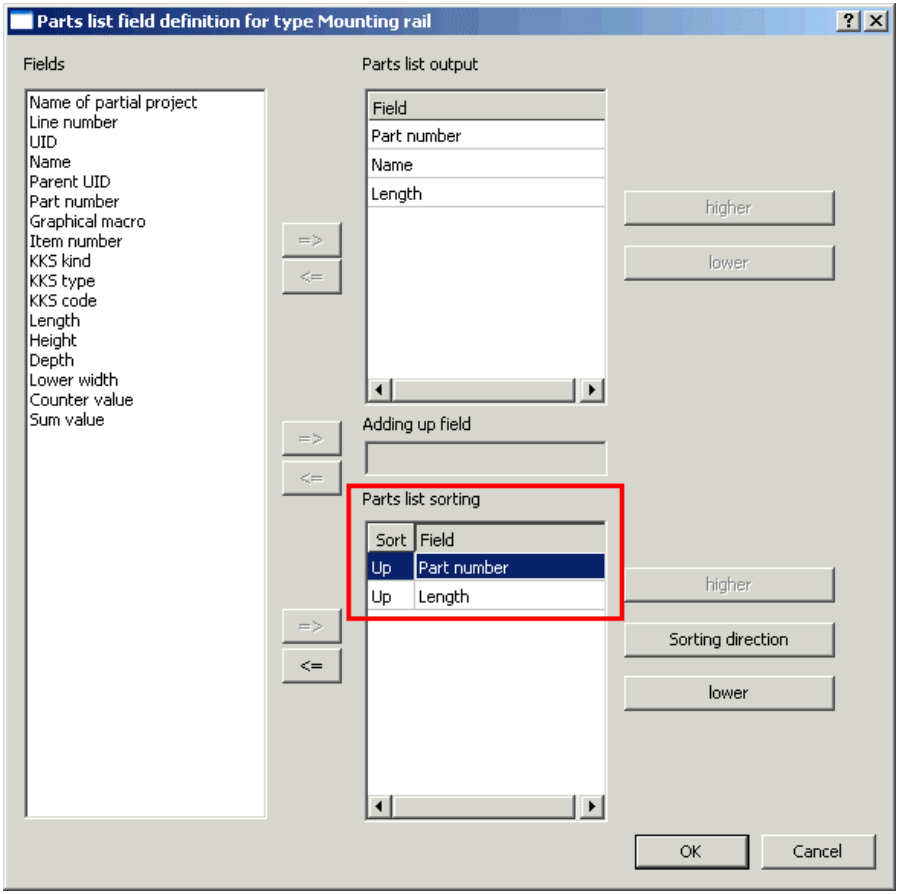


From the available fields, select the ones you want and click the [=>] button to copy them to your selection for parts list output.

Use the [up] and [down] buttons to change the sequence of items in the printed parts list.



Under ‘Sort order’, specify the sequence of object types in the printed output. In this example, this means the criteria by which items within the ‘duct’ object type are sorted in the parts list.



In our example, ducts are sorted first in ascending order of part number and then (for ducts with identical part numbers) in ascending order of length.

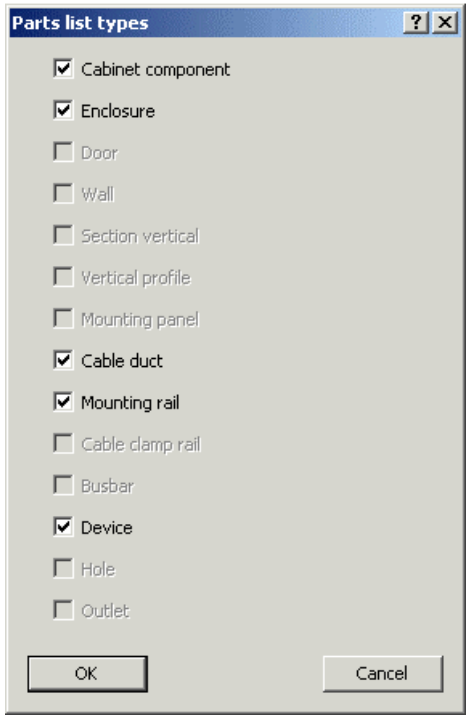
The [Sort direction] button selects between ascending and descending order.

The [up] and [down] buttons change the sort order of the selected fields.

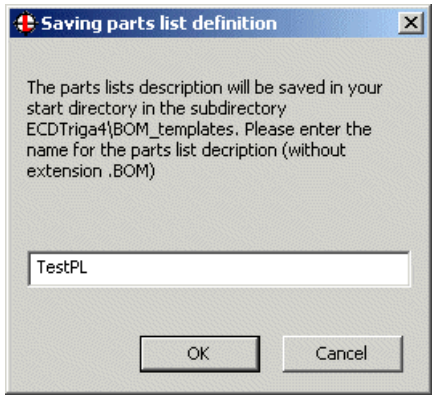
Once you have finished specifying parameters for the current object type, choose ‘OK’.

Repeat for all selected object types.

Note: Parts lists generated from this configuration are affected by the ‘Interactive deselection permitted’ option. If this option is enabled, you can deselect object types that would normally be included in the parts list about to be generated.



Click [Save] to save the configuration.



Type a name for the configuration file.

Note: Type the name only, without the '.BOM' extension.

All configurations are automatically stored to the directory @:ECDTRIGA4\BOM_Templates\

Note: You can re-open this configuration for editing at any time by clicking [Open].

6.1.2 Totals in project parts lists

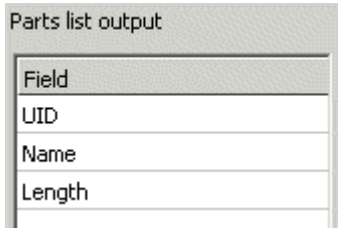
Totals can be computed automatically and inserted in the project parts list. The following applies:

- Totals are always specified by object type.
- The criteria for calculating a total are all fields under 'Parts list – Sort order'. A total is only calculated if these fields are all identical.
- For a total to be calculated correctly, the 'Total' fields must be included in the list of parts list output fields.
- Under 'Totalling field', specify the field to use to calculate the total.

Example

In the example, the total length of ducts with the same part number will be calculated. The printout will show every duct part number together with the total length.

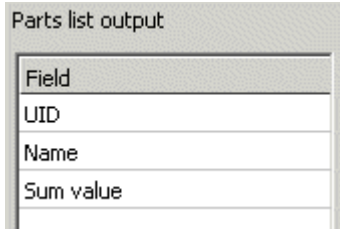
Go to ‘Part lists – Output’.



As in the previous example, this parts list shows the part number, name and length.

If you want to show the total length (i.e. a calculated total), the Length field is no longer needed.

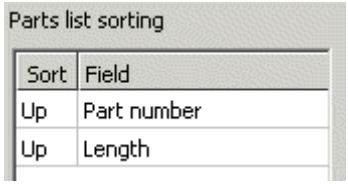
Instead, you need a Total field. This will show the total length.



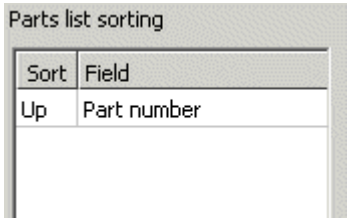
You next need to specify the field to be totalled – in this example the length.

Go to ‘Totalling field’ and select the length.

In the example, the part number and the length are selected under ‘Parts list – Sort order’.



This selection needs to be changed, because it would mean only ducts with the same part number and the same length are included in the total. For the total length of all ducts with the same part number, the length needs to be deselected.



Done! The generated parts list file will show a combined entry with the total length for each duct part number.

Save this configuration.

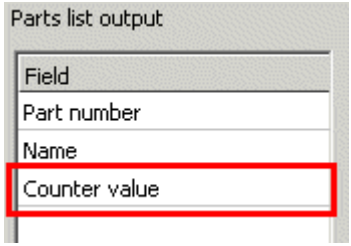
Note: All fields specified for totalling under ‘Parts list – Sort order’ must match.

6.1.3 Counting in projects parts lists

Besides calculating totals, it is also possible to count items. The following applies:

- Quantity counts are always specified by object type.
- The quantity counted is always the number of units.
- The criteria for a quantity count are all fields specified under ‘Parts list – Sort order’. Items are only counted if these fields are all identical.
- For a count to be produced correctly, the ‘Counter’ field must be included in the list of parts list output fields.

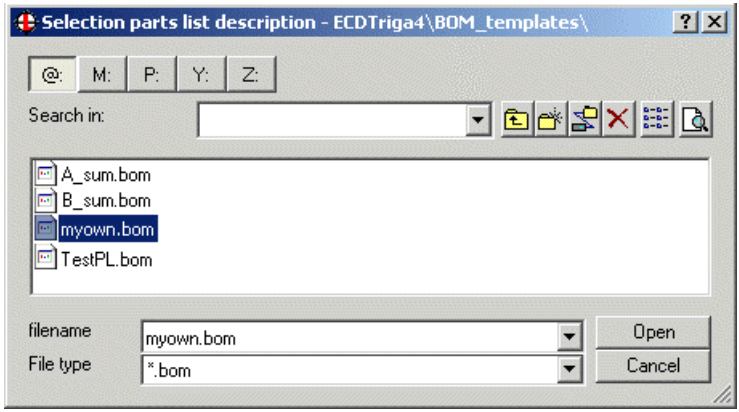
If you select ‘Counter’ under ‘Parts lists – Output’, the quantity will be counted automatically in accordance with the current settings.



6.1.4 Generating a project parts list

Open the eCabinet project.

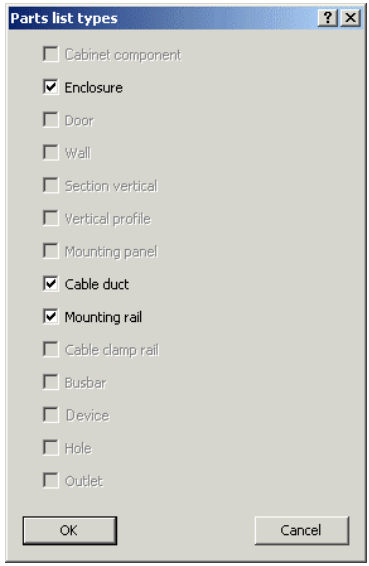
To generate a parts list, choose Edit – Project Parts List/ASCII on the menu and select the 'Generate project parts list' command.



Choose a configuration (.BOM) file, then choose [Open].

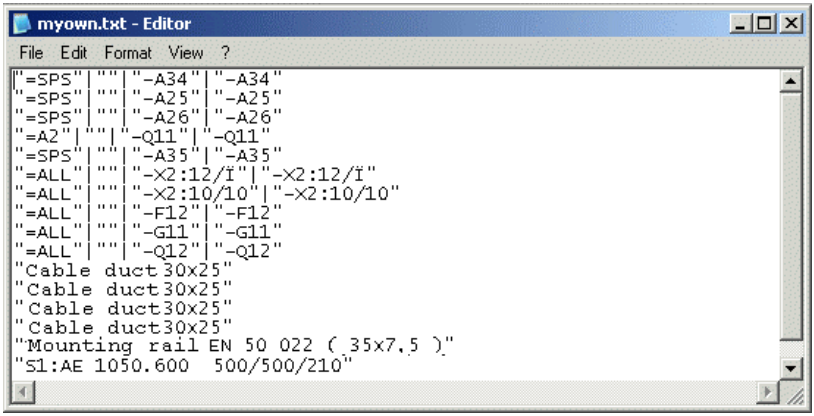
If the 'Interactive deselection permitted' option is selected in the configuration, you will first see a dialogue.

In the dialogue, you can deselect specific types of object before generating the parts list.



Choose ‘OK’.

A parts list is now generated in the project directory as a file in accordance with the selected configuration.



6.1.5 Updating project parts lists

Information, organization and security

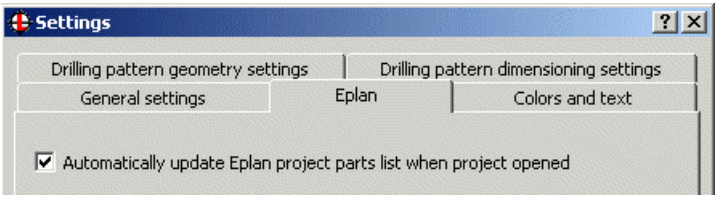
Update project parts list

When there has been a change to the EPLAN project, the project parts list can be updated.

This makes new devices available for selection and placement in the device browser.

Surplus devices already placed in a drawing can be automatically removed.

You can stipulate in the eCabinet parameters that the parts list is automatically updated when you open the project.



6.2 Importing parts data

Information, Organization and Security

Import parts data

If EPLAN data import from an EPLAN 5 or EPLAN 21 project is preselected in the eCabinet project, this command can be used to import the parts data used in EPLAN.

The data can be used for specifying universal parts that are not in the database because they are not specified in the EPLAN project.

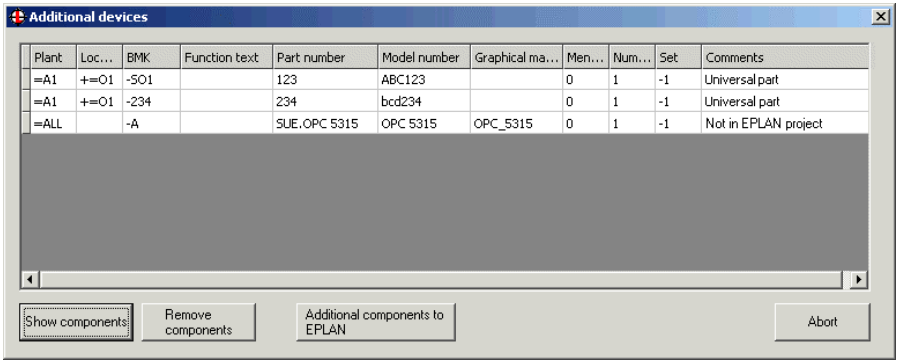
6.3 Checking a drawing

Checking a drawing finds any devices that are in the eCabinet project but not in the EPLAN project and hence not in the parts list. Such devices can be universal parts specified using parts data.

Information, Organization and Security

Check drawing

The project is searched and the devices found are listed in a browser:



Select a component in the list. There are three options for working with the listed components:

[Show components]

Highlights the component in the drawing.

[Remove components]

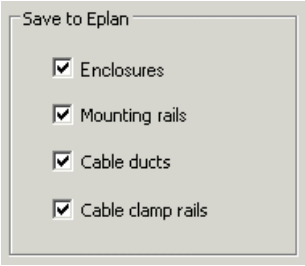
Removes the selected component from the drawing.

[Additional components to EPLAN]

Transfers all selected components to the EPLAN project. The components will be in the project parts list the next time it is generated.

Enclosures, ducts, rails and busbars can also be transferred back to the EPLAN project.

You can control how this command works in the eCabinet parameters.



6.4 Importing project parameters

As of eCabinet 4.1 Service Pack 1, it is possible to import project parameters from EPLAN projects to eCabinet projects.

The following project parameters are imported:

- Project name
- Customer
- Plant designation
- Order number/commission
- Project manager
- Date of last modification
- Time of last modification
- Company name (address 1)
- Company postal address (address 2)
- Company postal town (address 3)

These parameters are automatically copied from the relevant EPLAN project when the parts list is generated or updated.

If the eCabinet project type is ‘File’, it is also possible to copy these parameters via the .PBF file.

Using project parameters in eCabinet

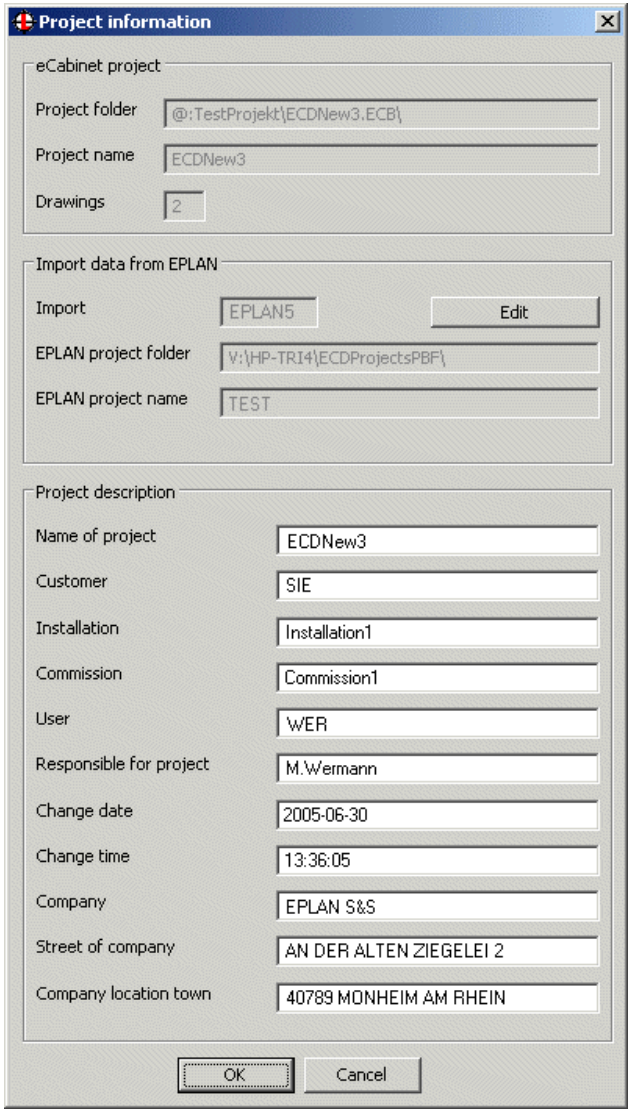
The project parameters copied when generating or updating a parts list are stored in the eCabinet project.

You can use these parameters in various ways.

6.4.1 Project information

The parameters stored in the eCabinet project can be viewed by using the Project Information command.

Select the command on the menu under Information/Security – Change Project Assignment.



Note: You can modify the parameter values here, for example to control output in the standard sheet or other documents. Note, however, that the data will be overwritten whenever you select 'Update project parts list'.

6.4.2 Output to an eCabinet standard sheet

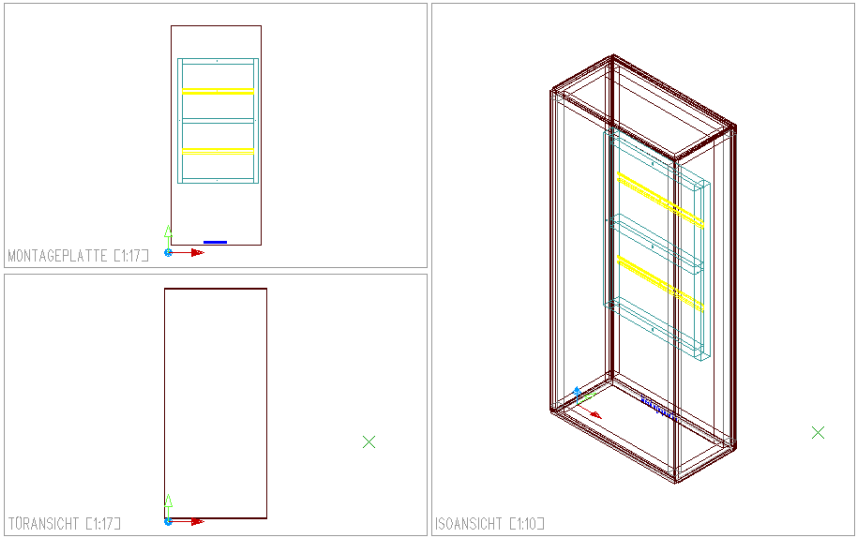
The parameters stored in the eCabinet project can be visualized at any time in a project drawing as a standard sheet that matches the drawing.

The standard sheet is displayed in the views.

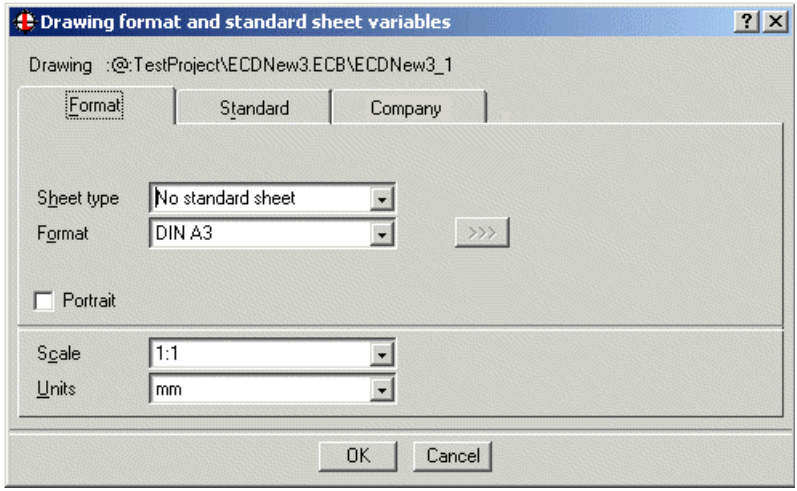
To enable the standard sheet:

- Create enclosure views by choosing View – Enclosure Views.

  Alternatively, you can switch straight to the views if they already exist (e.g. via the Navigator).

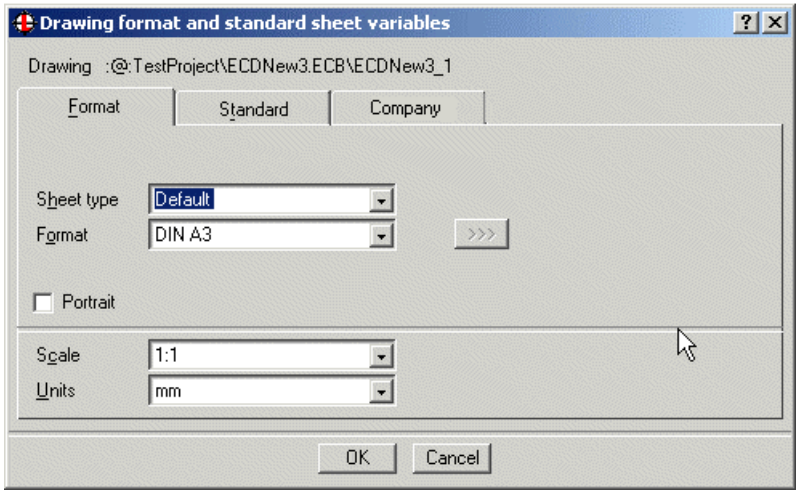


- On the menu bar, select File – Standard Sheet.

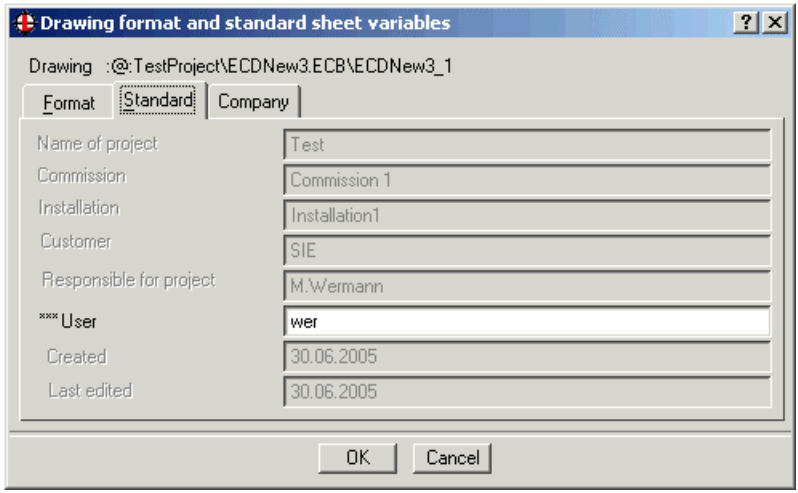


The current setting is displayed.

- If you want to configure a standard sheet, change the 'Sheet type' setting and either select the 'Default' standard sheet or a custom one of your own.

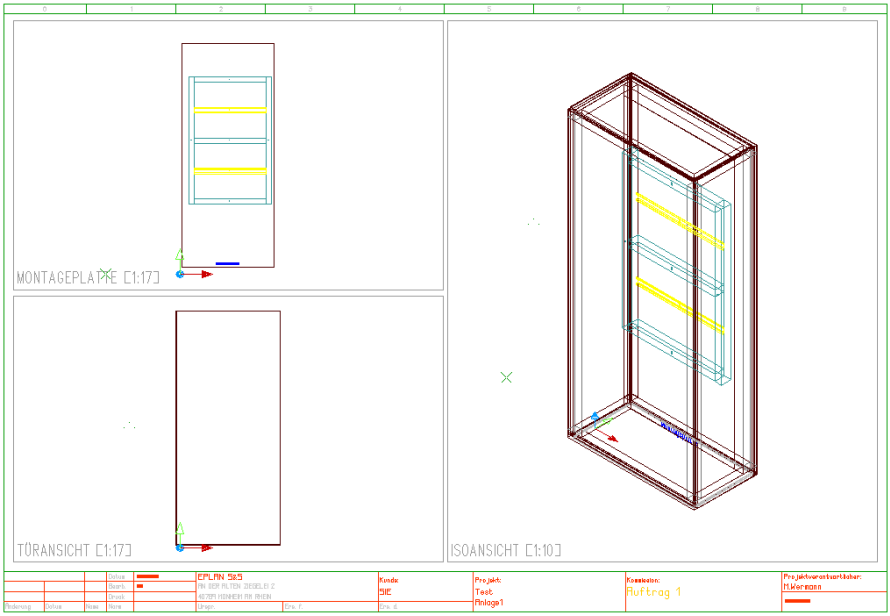


- The Default and Company tabs display the standard sheet variables. These are all automatically copied from the project parameters and cannot be edited here except for the user name.



Note: You can edit these parameters in Information/Security – Change Project Assignment. The standard sheet variables will change automatically in line with the changes you make. However, the data will be overwritten every time you select 'Update project parts list'.

The standard sheet with the specified variables is displayed when you choose [OK]. It is updated whenever you modify the project parameters.



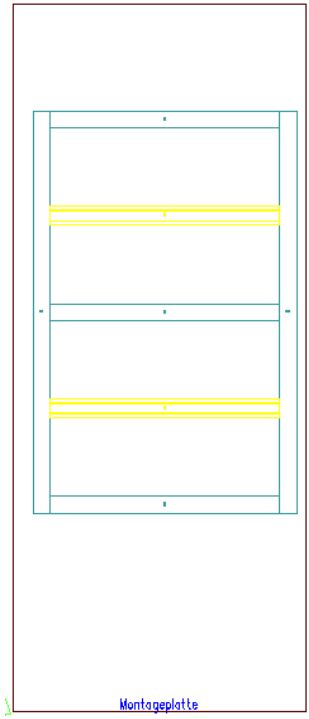
6.4.3 Output to a symbol

The project parameters stored in an eCabinet project can additionally be 'stamped' in any printable area of your drawing.

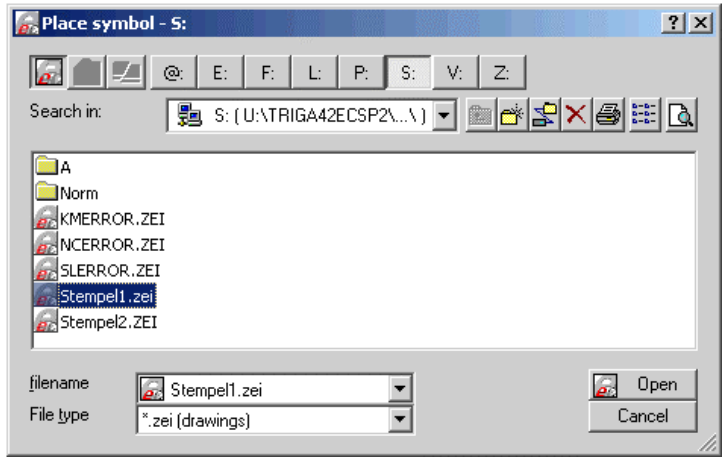
This means you can label all printed parts with the project parameters.

To place a stamp in the area to be printed:

- Go to the enclosure views and enlarge the editable area of the drawing.



- On the menu, choose Elements – Place Part – Symbol.



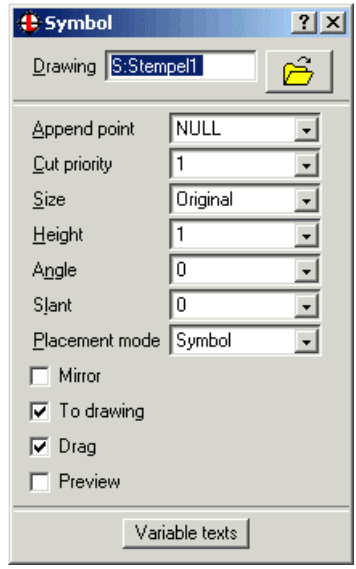
- Logical drive S contains two stamp symbols:

  Stempel1.zei - Symbol with project properties, with table

  Stempel2.zei - Symbol with project properties, without table

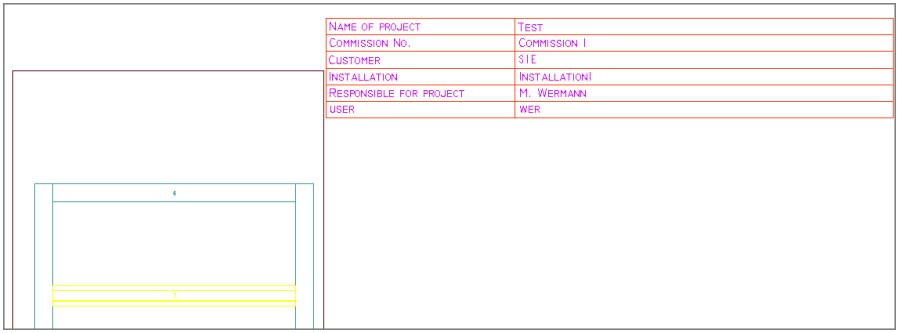
- Select the stamp you want and choose [Open]. A preview of the symbol is displayed at the cursor.

- Modify the symbol placement parameters before placement as needed:



The main parameters involved here are the append point, the angle and the size.

Place the stamp where you want it in the drawing.



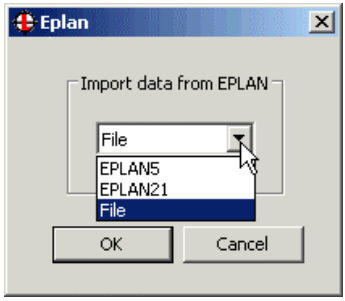
6.5 Changing project information/project assignment

Information, Organization and Security

Change project assignment

A dialogue opens showing the data for the current project.

The EPLAN data import method for the project parts list can be modified:



6.6 Updating components in databases

Information, Organization and Security

Update components in databases

Various types of component can be edited in the eCabinet components database:

- Enclosures
- Mounting panels
- Mounting rails
- Ducts
- Busbars
- Adapters

To edit the data, select a component type in the 'Edit data records' dialogue. For each type, a browser opens with a list of the components in the database.

Editing options

There are three options for editing a component in the browser:

[New]	Creates a new part. Specifications and values are entered for each type of component in a specification dialogue. The size and layout of the dialogue varies according to the type of part. Click [OK] or [Apply] to write the new values to the database.
[Edit]	Opens the specification dialogue for a component selected in the browser for editing. Click [OK] or [Apply] to write your changes to the database.
[Remove]	Prompts for confirmation and then removes the selected component from the database.

6.7 eCabinet parameters

The eCabinet system parameters govern a wide range of display, data handling and display features in eCabinet.

Information, Organization and Security

Parameters

6.7.1 Parameters: General

Device name

Parameter setting for the appearance of the device name:

- Device tag (default)
- Unit + device tag
- Location + device tag
- Unit + location + device tag

Rail spacing

Reference point for spacing stipulated when creating parallel copies of rails. The spacing can be:

- Between component centres
- Between component edges (default)

Duct spacing

Reference point for spacing stipulated when creating parallel copies of ducts. The spacing can be:

- Between component centres
- Between component edges (default)

6.7.2 Parameters: Eplan

Automatically update Eplan parts list when project opened

With the default setting, a newly generated parts list from EPLAN is not automatically updated in the eCabinet project.

The ‘Update project parts list’ command can be used to perform this task manually when needed.

Save to Eplan

Parameter setting for components saved to the EPLAN project using the ‘Additional components to EPLAN’ command. With the default setting, all components are selected:

- Enclosures
- Mounting rails
- Ducts
- Cable clamp rails

EPLAN 5 Version

To accommodate different parts list file formats, the installed EPLAN 5 version can be specified here:

- 5.50
- 5.60
- 5.70

6.7.3 Parameters: Colours and text

Colours and transparency can be specified for various components and devices. Text colours and text sizes can also be changed.

Enclosure

- Component colour for individual components
- Transparency (%) for individual components

Ducts/rails

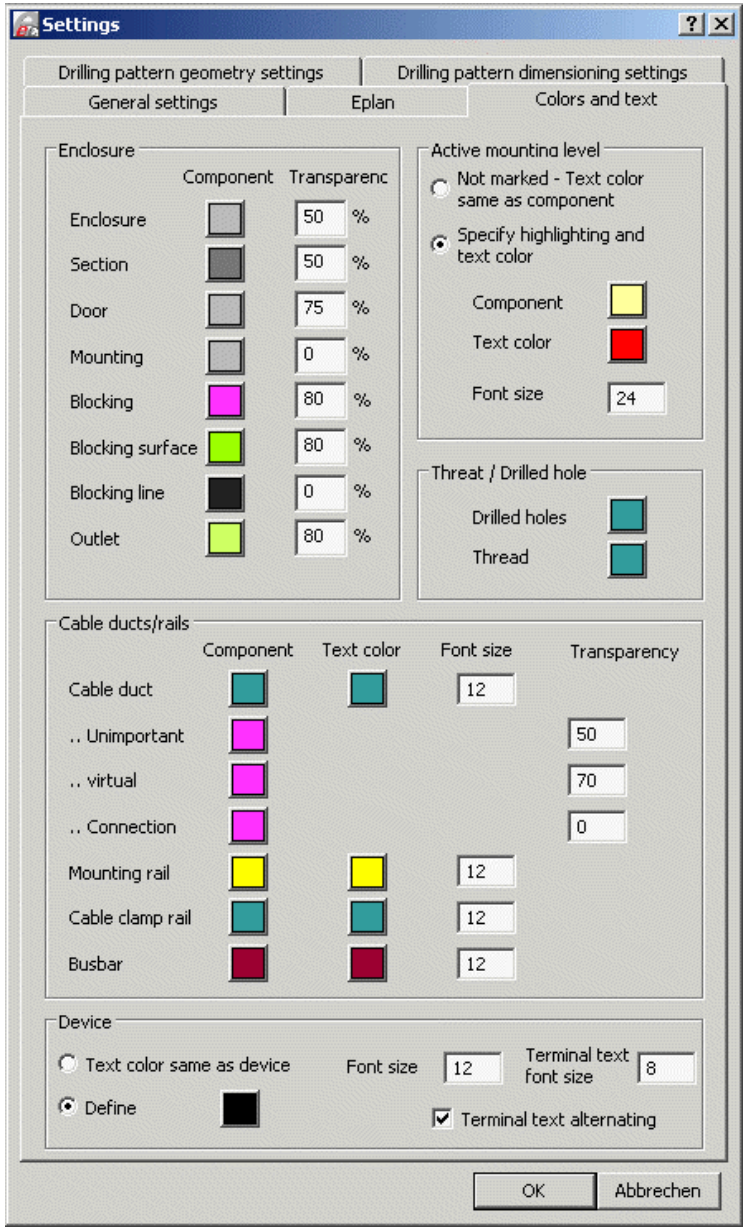
- Component colour for entire component
- Text colour for component labelling
- Text size for component labelling, in drawing units

Devices

- Text colour as for component or specified
- Text size in drawing units
- Text size for terminals, in drawing units
- Text on terminals in alternating colours

Active mounting level

- Component colour
- Text colour
- Text size in drawing units
- No highlighting (text colour same as component)


Settings
Drilling pattern geometry settings
Drilling pattern dimensioning settings
General settings
Eplan
Colors and text
Enclosure
Component
Transparenc
Enclosure
50 %
Section
50 %
Door
75 %
Mounting
0 %
Blocking
80 %
Blocking surface
80 %
Blocking line
0 %
Outlet
80 %
Active mounting level
Not marked - Text color same as component
Specify highlighting and text color
Component
Text color
Font size
24
Threat / Drilled hole
Drilled holes
Thread
Cable ducts/rails
Component
Text color
Font size
Transparency
Cable duct
12
.. Unimportant
50
.. virtual
70
.. Connection
0
Mounting rail
12
Cable clamp rail
12
Busbar
12
Device
Text color same as device
Font size
12
Terminal text font size
8
Define
Terminal text alternating
OK
Abbrechen

6.7.4 Parameters: Drilling pattern geometry

Parameter settings for displaying the geometry of 2D drilling patterns:

Margins	Minimum distance between the drilling pattern geometry and the edge of the drawing area, top/bottom/ left/right
Layers	Drawing layers for geometry elements of panels, silhouettes, holes/threads and other elements
Text sizes	Size of device tag and dimensioning text

6.7.5 Parameters: Drilling pattern dimensioning

Specifies text labelling (symbols) for dimensioning drilling pattern components:

- Hold diameter: Ø
- Thread diameter: M
- Width: B
- Height: H
- Chamfer: X
- Fillet radius: R
- Hexagon side length: a
- Octagon side length: a

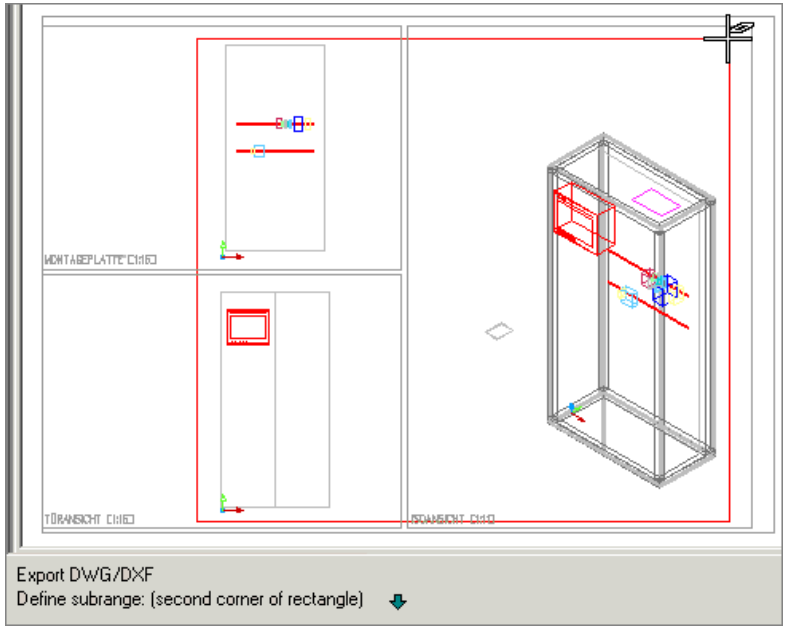
7 Graphical export to EPLAN

When work on an enclosure is completed in eCabinet, the final step is to export the graphical plan to EPLAN. The plan is stored in DXF format in the associated EPLAN project directory and integrated into the project. A new page of type 'Free graphic' is automatically created in EPLAN. The drawing is transferred to EPLAN as it is seen on screen.

File	eCabinet
Grapl ical export	**Fil | Graphical export**

7.1 Selecting an area for graphical export

Enter two points to enclose the area to be exported in a rectangle.



After the second point has been entered, the area is displayed again and you are prompted to confirm that the correct area is enclosed.

Answer ‘No’ if you want to specify the rectangle again.

Enter ‘Yes’ to have the selected area automatically exported to EPLAN, creating a graphical page in EPLAN or a DXF file (project type ‘file’).

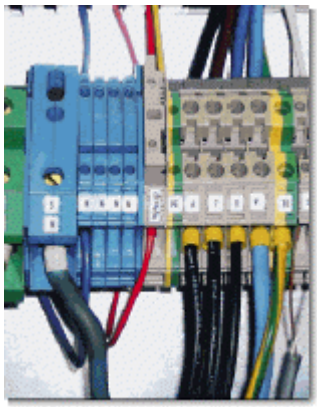
The page number is requested beforehand in eCabinet.

8 Cable routing

8.1 Routing: Basics

The eCabinet Routing module allows you to determine the following information based on a layout created in eCabinet basic and the wiring information in the circuit diagram:

- Wire lengths
- Routing
- Preparation of wire ends
- Duct fill levels



This information can be put to further use in various ways.

- **Output to a wiring centre (a machine for making up wires), to:**

  - Cut wires to length
  - Prepare wire ends
  - Label wires

- **Output to a file, stating:**

  - Source/target with contact information
  - Routing direction
  - Wire lengths
  - Preparation of wire ends
  - Wire bundling

- **Printouts**

  - Wiring parts list
  - Fixed lengths
  - Remaining parts list

Routing toolbar

The routing commands are on the eCabinet Routing toolbar:

eCabinet menu bar

You will find the routing commands on the eCabinet menu bar under ‘Routing’.

8.2 Wiring list

The wiring list is usually copied directly to the eCabinet project from the circuit diagram. This is done automatically when the eCabinet project is created.

You can updated the wiring list manually at any time.

On the menu, select Information/Security – Update Wiring List.

If the project type is EPLAN5 or EPLAN21, a wiring list matching the circuit diagram in its current form is automatically generated in EPLAN and exported.

If the project type is 'File', a wiring list (VDR_ECD_TRIGA4.txt) is loaded from a directory. The wiring list is in the same directory as the parts list.

8.2.1 Manually edited wiring lists

In projects where the project type is EPLAN5 or EPLAN21, the respective version of EPLAN is responsible for the wiring list. The circuit diagram and the features supported by EPLAN5/21 determine the make-up of the wiring list.

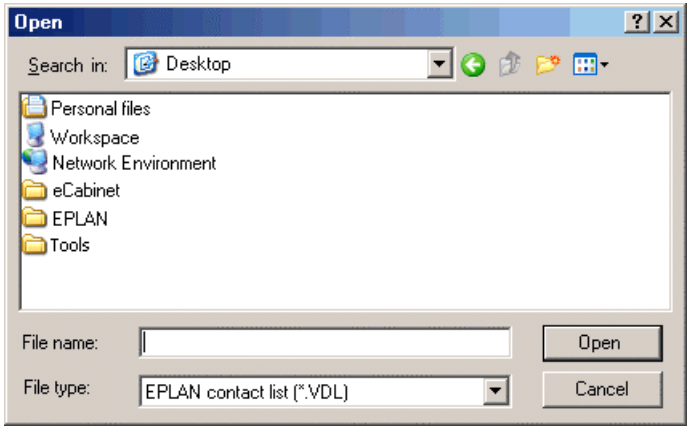
A wiring list can be edited manually before importing into eCabinet. This can be done in a simple text editor. The structure of the wiring list must not be modified, but the content can be changed. The file can have any name.

In this way, you can effect changes such as removing from the wiring list connections that are not wires or wires that you do not wish to have routed by eCabinet.

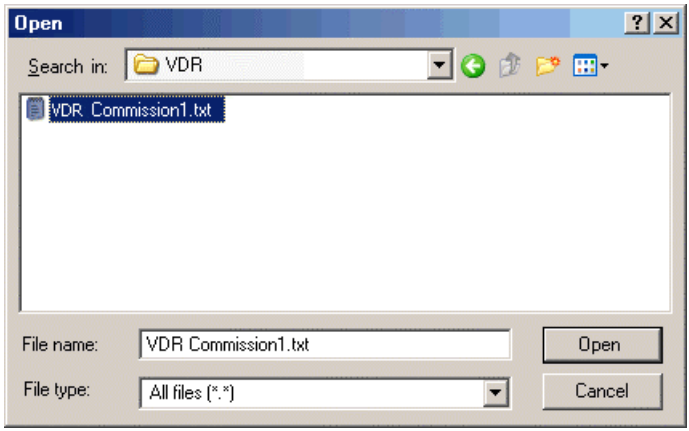
Note: A manually edited wiring list can <u>not</u> be imported in the usual way using the 'Update wiring list' button because this would automatically overwrite the edited wiring list with the data from EPLAN. A special command is therefore provided for importing custom wiring lists into eCabinet.

Be sure to save the manually edited wiring list in a separate directory so that it is not inadvertently overwritten by an automatic update.

Then, on the menu, choose Routing – Import a Wiring List.



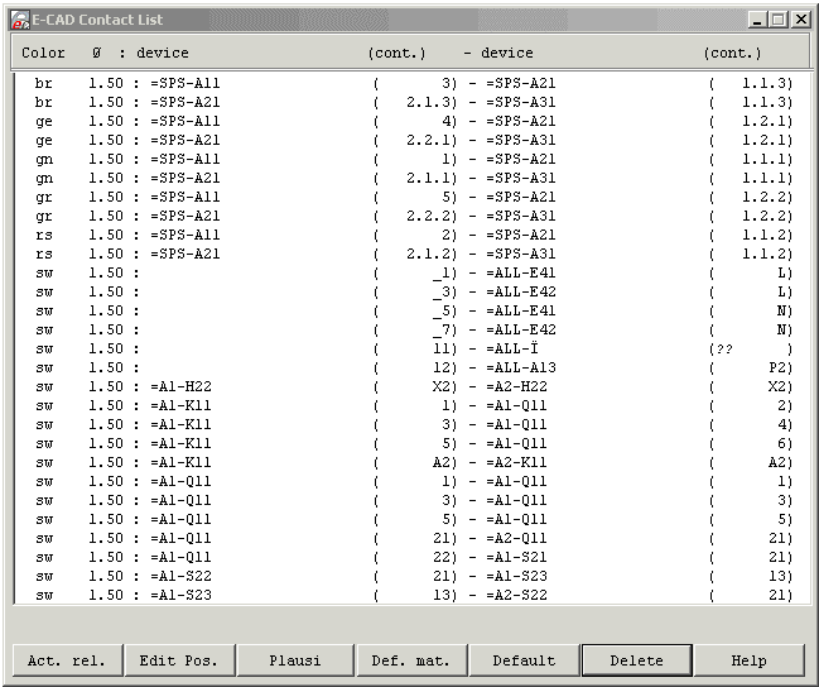
Change the file type to *.* and select your edited wiring list.



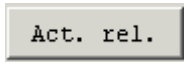
Click ‘Open’. The wiring list is imported and can now be used for routing.

8.2.2 Working with the wiring list

Open the wiring list on the menu bar by selecting Routing – Display Wiring List.

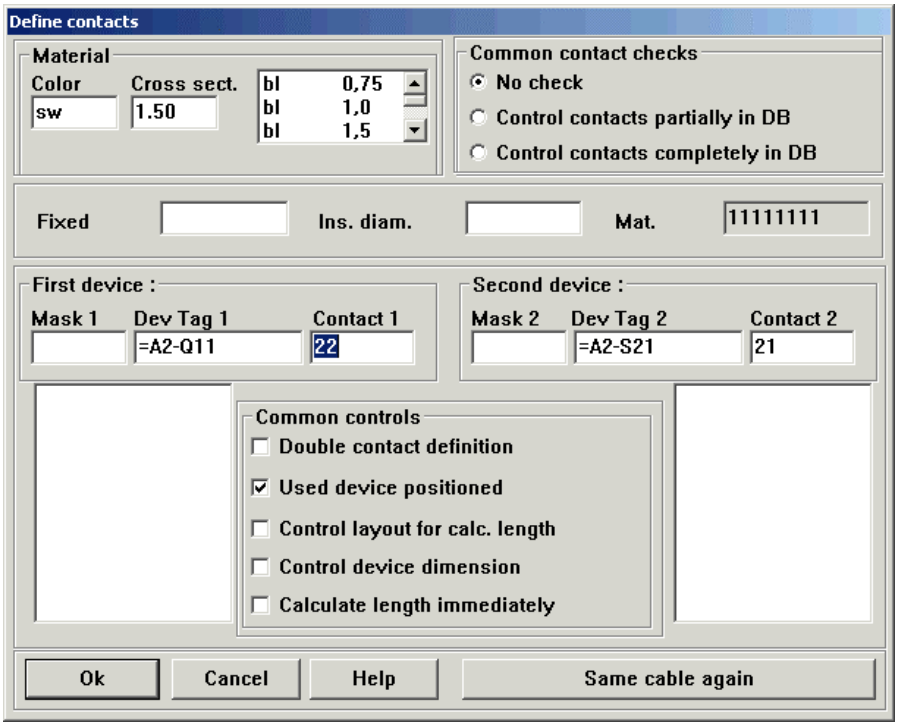


There are various commands for working with the wiring list:

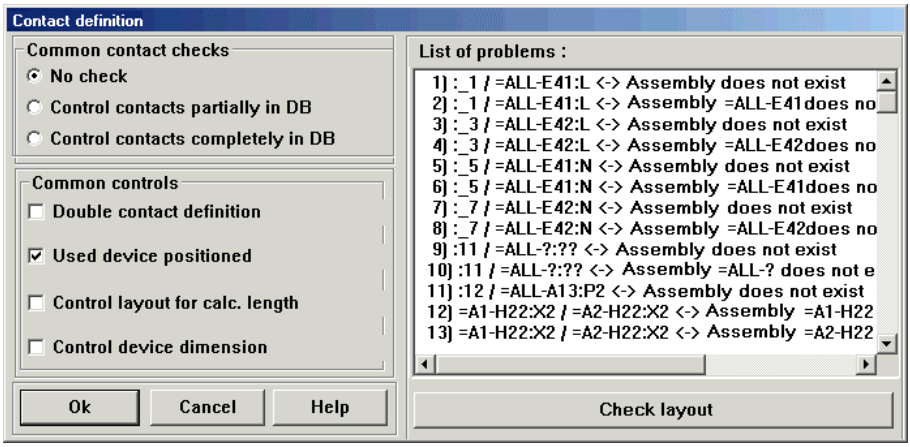


Update material allowed in the current wiring list.

Edit a selected item in the wiring list.

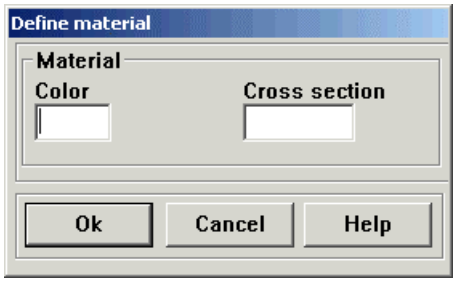


Check the current layout, allowing any problems to be eliminated before running the length calculation.



Def. mat.

As the EPLAN wiring list can be imported without material assigned, a command is provided for assigning material (colour and cross-section) at any time.



Select a connection for modification, click [Assign material] and specify a new colour and cross-section.

Default

Modify all items that have been assigned a colour and cross-section in EPLAN. The colour or cross-section is replaced on all items using **'???'** or **'??'**.

Delete

Delete all selected items.

8.3 Parts data

During the routing process, eCabinet accesses the parts data for the devices to be connected.

The following applies:

- There is not a general requirement to enter contact data, because the default settings can be used.
- Contact data can be entered in the parts master data. These are automatically used in routing.

8.3.1 Production databases

The production databases (Routing and NC) are Paradox databases. The database files are kept in the directory PDOXSRC.

In a standard installation of eCabinet 4.2, this directory is under C:\Programme\EPLAN\eCabinet42\Triathlon\.

Note: It is possible for the directory to be moved to a different location. Information on why and how this is done is provided in the eCabinet 4.2 installation instructions.

The databases in the PDOXSRC directory are as follows:

Database name	Content
Baugrup.DB	Information on devices including contact data for routing
Bohren.DB	Information on drilling patterns for devices
Kanal.DB	Information on ducts
Schiene.DB	Information on mounting rails
SammelSchiene.DB	Information on busbars
Material.DB	Information on wires to be used

All device contact data are stored in the database Baugrup.DB.

8.3.2 Entering article data for routing

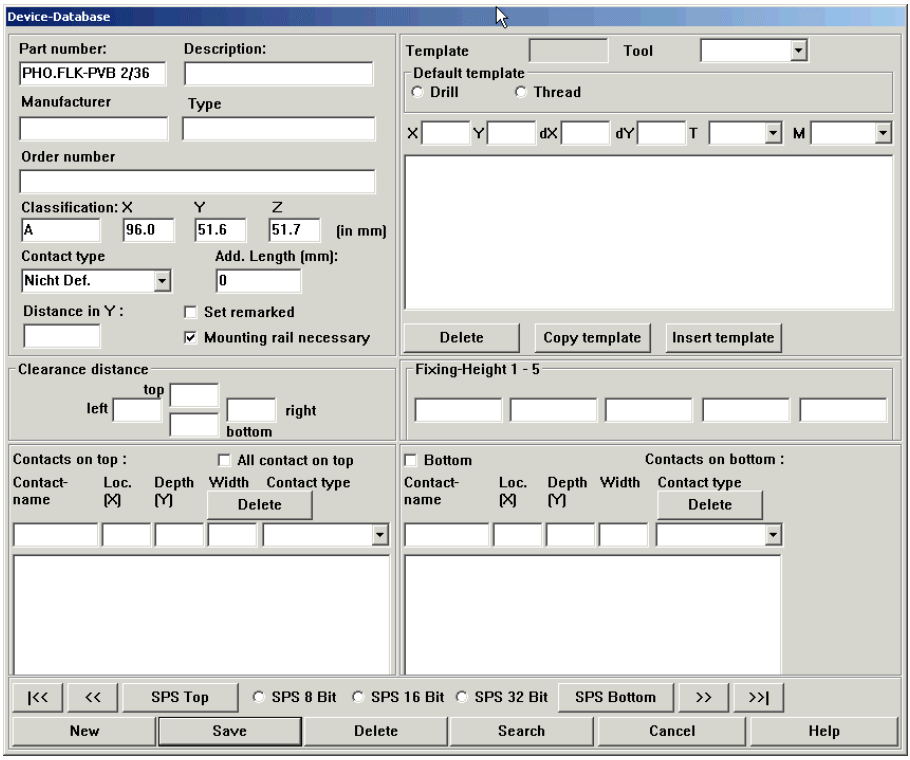
Every device used in eCabinet Basic has a unique part number.

If there are no data on a part in the production databases, a database record is automatically added for it when a matching device is placed in a enclosure drawing.

Any special data needed for routing (e.g. contacts) only need to be entered once per part number.

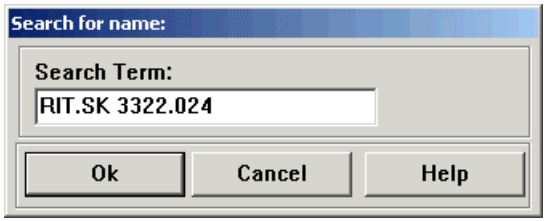
To edit production part data:

- **Editing part data in the production database for devices**



A dialogue is displayed, showing the first part (in alphabetical order).

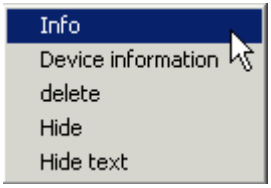
To select a different part for editing, use the cursor keys or choose [Search].



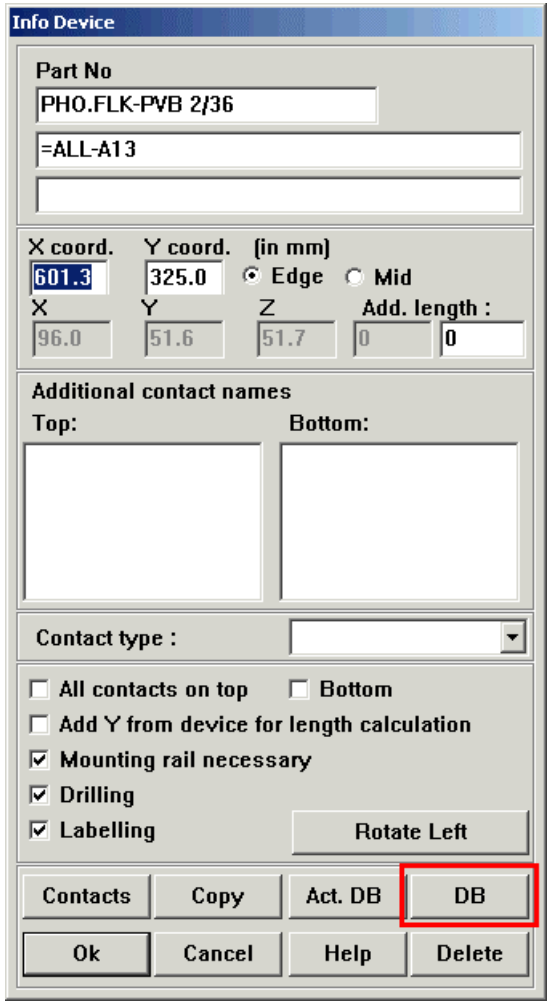
To find the part, simply type its part number.

Note: The full part number is needed – wildcards are not supported.

- **Editing part data by right-clicking a device in the enclosure drawing and selecting 'Information' on the context menu**

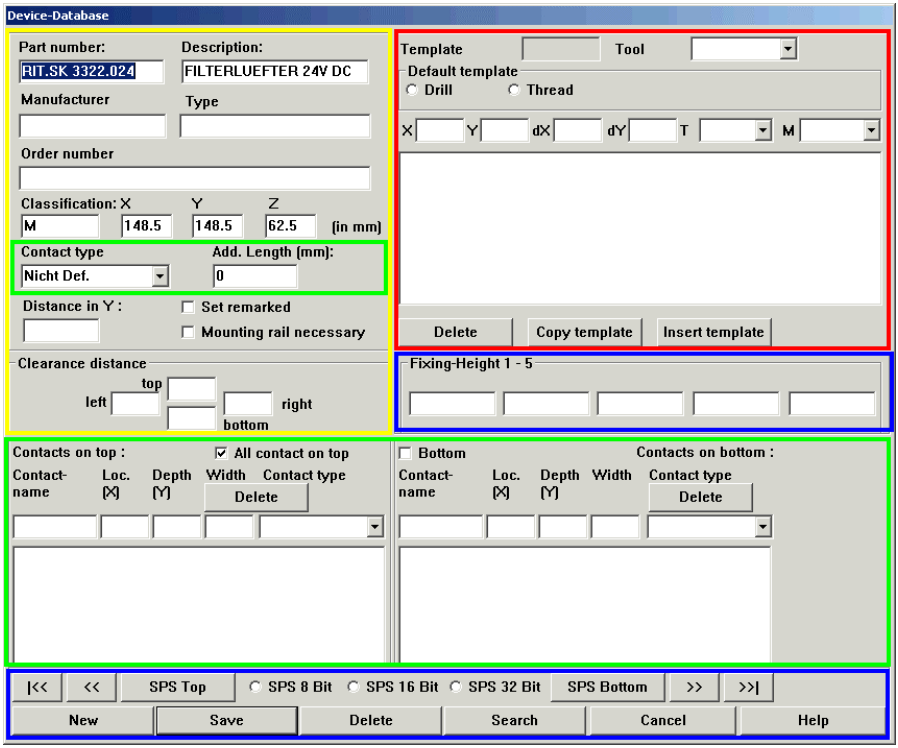


The following dialogue opens:



Click the [DB] button to open the parts database. The part corresponding to the selected device is automatically selected.

The dialogue used for displaying the article data is laid out as follows:



Colours in the dialogue:

Yellow General part data

Red Drilling data

Blue Not used in eCabinet

Green Data relevant to routing

In the following, we will only deal with data relevant to routing.

8.3.2.1 General contact data

General contact type

‘General contact type’ is a global setting for all contacts on the part.

All wires connected to a contact on this part (including wires for which no details are specified) are automatically assigned the contact type specified here unless the contact itself is specified with a different contact type.

Note: On devices where only one conductor end sleeve or double end sleeve can be attached to each contact, the default general contact type is a double end sleeve. If two or more wires are connected to a contact on such a device during routing, this contact type remains active. If only one wire is connected, the contact changes from double end sleeve to crimping (a normal conductor end sleeve).

Note: The available contact types are always customer-specific and are defined in the file schalt.ini.

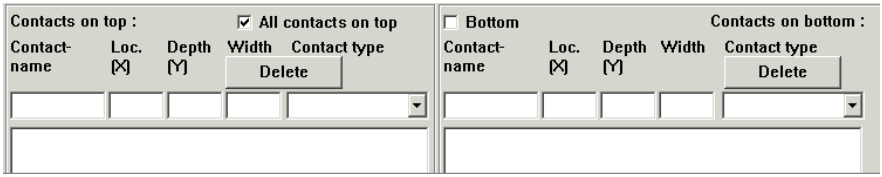
It is essential for this to be configured on every system. A standard installation of eCabinet 4.2 is not sufficient.

Additional length

If you enter a numeric value under Additional Length, the specified amount is automatically added to the length of every wire that is connected to a contact on the part concerned.

This setting applies for all contact points on the device.

8.3.2.2 Detailed contact data



A general distinction is made between top and bottom contacts. As a result, you need to designate contacts as top or bottom.

Basic definitions

Use this option to specify whether all contacts are top or bottom.

If the precise position of contacts is not important, this setting will be all that is needed in most cases.

Note: It is possible to select both options, but it the only meaningful choices are top, bottom or neither.

Enabling ‘All contacts top/bottom’ overwrites and invalidates any detailed contact data that are specified separately.

Detailed contact data

The following parameters can be specified separately for top and bottom contacts:

- **Contact name**

A contact name can be up to 9 characters long.

The contact name must match that of the contact in the wiring list.

- **X coordinate**

X coordinate of the contact.

The origin is always at the left hand side of the device.

The coordinate is always specified in millimetres.

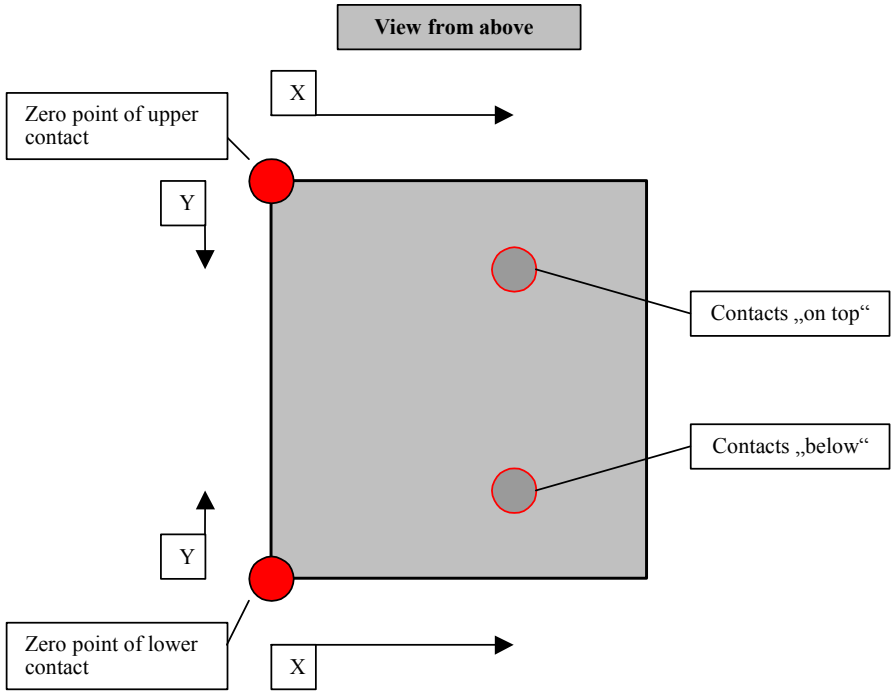
- **Y coordinate**

Y coordinate of the contact.

The origin is the top of the device on a ‘top’ contact and the bottom of the device on a ‘bottom’ contact.

The coordinate is specified in millimetres.

A Y coordinate does not need to be specified for contacts on the device periphery.



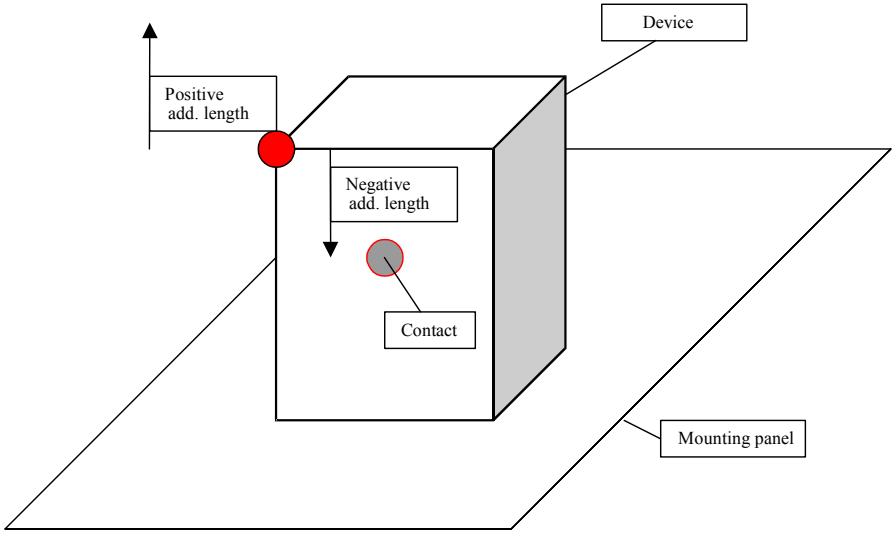
- **Additional length**

When a wire is connected to the contact, the specified additional length is automatically added to the wire's calculated length provided that the X and Y coordinates of the contact are exactly specified.

By default, the wire length is calculated as if the contact is at the front edge of the device (along the Z axis) – allowing for the maximum distance from the mounting panel or rail. The wire can thus be connected at any level, even if the distance in Z is not specified.

You can specify a Z coordinate (distance from the mounting panel or rail) by entering a value in Additional Length.

Another possible use for this setting is where inaccessible contacts require special routing and hence a longer wire.



- **Contact type**

The ends of wires connected to the contact are given the specified contact type.

Once you have specified the parameters for the contact, press [Return] to copy them to the appropriate contact list. If there already is a contact with the specified name, it is assumed that you want to update the existing contact with the new parameter values.

If you click an item in the contact list to select it, its data are automatically copied to the input line.

To delete an item, select it and click the [Delete] button.

8.4 General routing rules

8.4.1 Automatic contact detection

The connections between devices are determined when the circuit diagram is created. This information is copied to eCabinet by way of the wiring list.

For each part made of a given device, it is possible to specify a unique location (top, bottom), position (distance from left, distance from the top/bottom) and additional length for a contact.

A general rule also applies, under which **uneven** contacts are at the **top** and **even** contacts at the **bottom**.

This automatic contact detection can be enabled separately for each project (default: enabled).

Contacts that do not conform with this rule must be specified as such in the master data.

8.4.2 Calculation of wire lengths

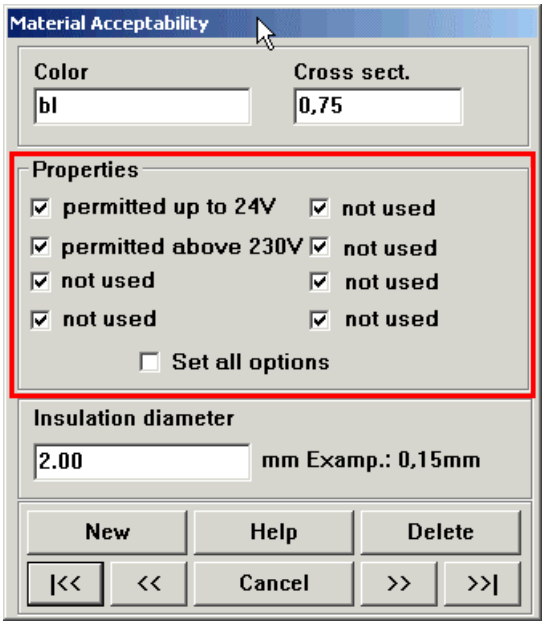
Wire lengths are essentially calculated in two steps:

- Step 1: Compute a network of nodes
- Step 2: Compute the wire length for each possible routing path

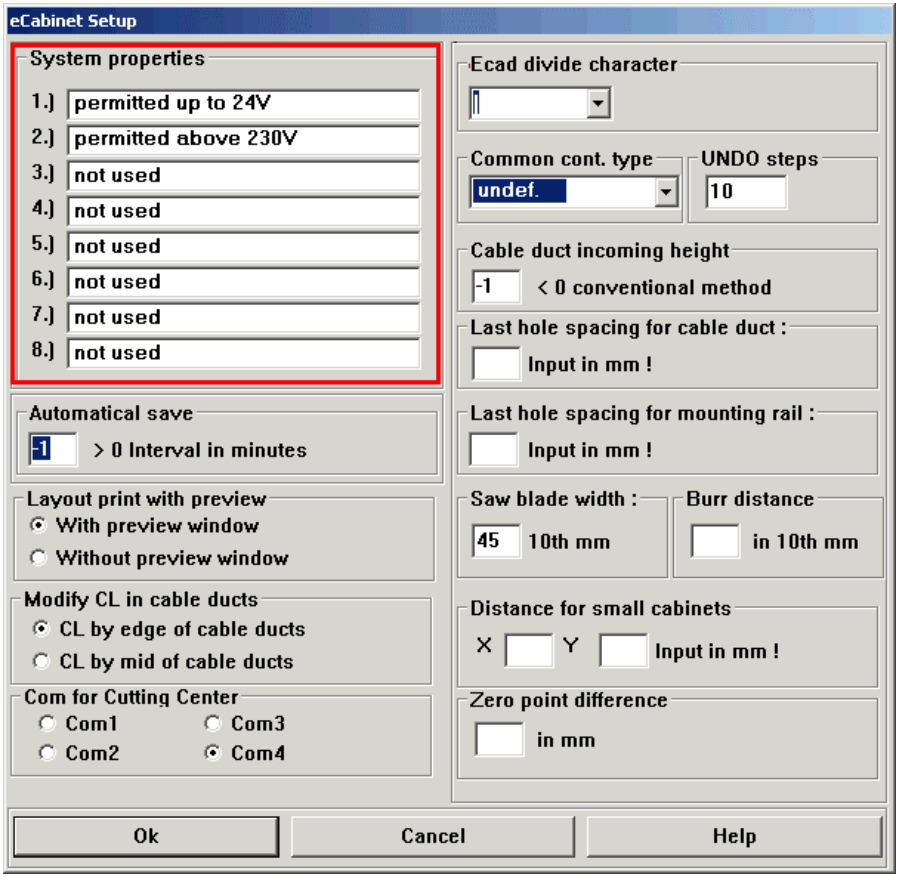
8.4.2.1 Material properties for wires

During the routing process, the material properties recorded in the materials database are taken into account by comparing them with the project-specific duct properties.

Materials database



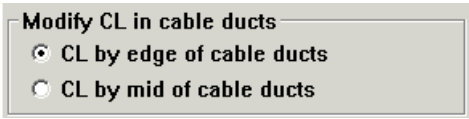
These properties can be specified globally under ‘Settings (Production)’.



8.4.2.2 Length calculation tolerance

You can specify a tolerance for length calculation by running the length calculation via duct corners or duct centres and specifying the height at which a wire enters the duct system.

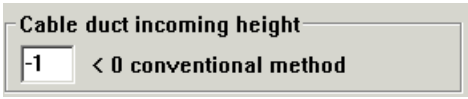
Length calculation over duct corners/duct centres



The node network is generated based on duct coordinates, using the specified X and Y coordinates and the length of each duct. These data are used to determine the shortest path. This is equivalent to length calculation over duct corners. The heights and widths of ducts are not taken into account.

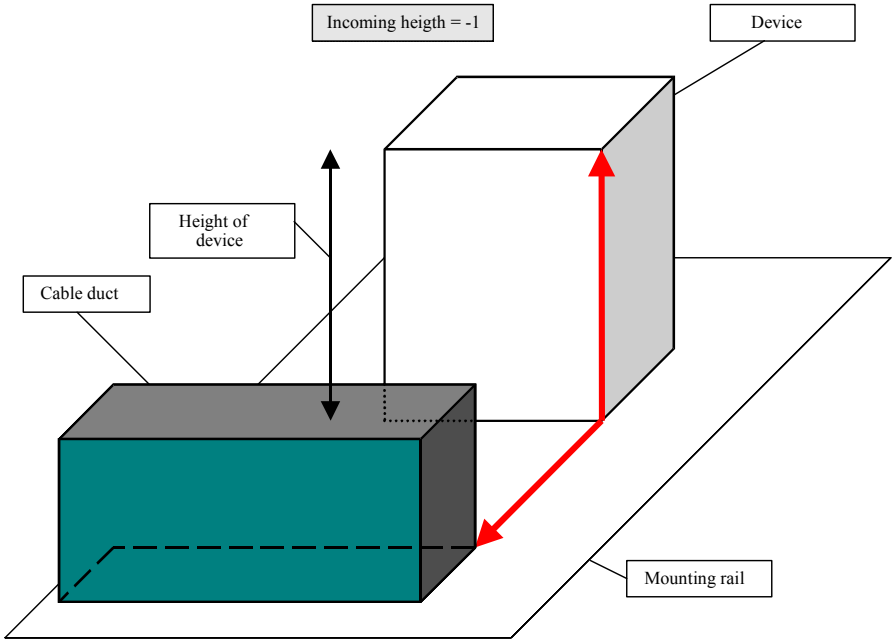
Alternatively, you can stipulate length calculation along duct centres by setting the appropriate parameter. The length calculation is done using the node network as before and the resulting path then adjusted to the duct centres. Only the width of ducts is taken into account.

Entry level into the duct



This parameter can be used to weight the path from the device to the first duct.

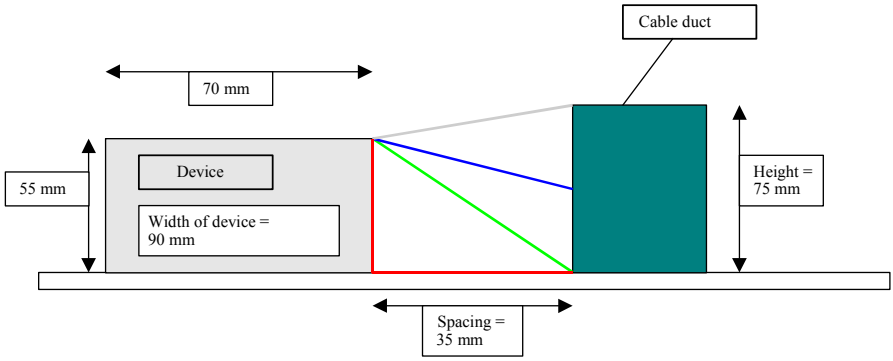
If the value is -1, length calculation is done via corners. The vertical position of the device and the distance to the first duct are added in their entirety to the length of the wire.



If the parameter is set to a value from 1 to 99, the direct length is calculated based on the relative heights of the device and the duct.

The duct height is weighted by the specified percentage.

Example



Case	Entry height	Wire length computation	Wire length
1	-1	L= 35 mm + 55 mm + 90 mm	180 mm
2	1	L= √ ((35 mm)² + (55 mm - 1% * 70 mm)²)+ 90 mm	154.6 mm
3	50	L= √ ((35 mm)² + (55 mm - 50% * 70 mm)²)+ 90 mm	130.3 mm
4	99	L= √ ((35 mm)² + (55 mm - 99% * 70 mm)²)+ 90 mm	127.8 mm

Case 1 or 2 is usually used in practice. Cases 3 and 4 are more theoretical values, provided for comparison purposes.

For conservative length calculation resulting in the least possible wiring effort, you would select Case 1 (value -1). All wires will then be calculated exactly as stipulated.

If you specify your contact data very precisely with little margin for error, you would generally use Case 2 (value 1).

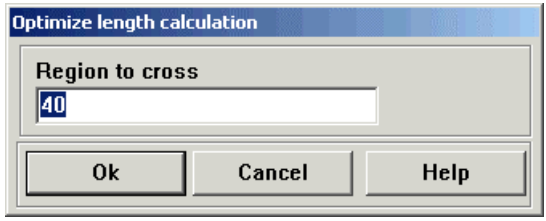
8.5 Optimization commands for ducts

The width of cable ducting is initially of secondary importance in length calculation. It is important for ducts placed in the eCabinet drawing to be assigned the crossing points they need in the layout for routing. The top left coordinates of each duct are used for this purpose.

8.5.1 Optimize duct nodes

The Optimize Duct Nodes command ensures that the necessary crossing points are present in a stipulated region of the duct system.

Select the command on the menu under Routing – Optimize Duct Nodes.



Note: The default value for the range to be bridged is determined using the largest current duct width.

8.5.2 Check contact area

All specified ducts can be rendered transparent for easier identification of problem layout regions.

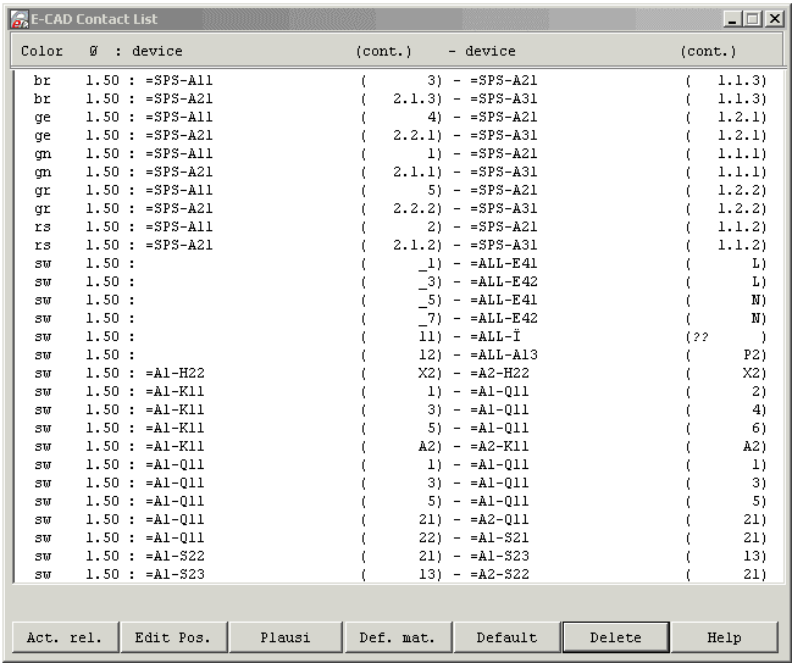
Select the command on the menu under Routing – Check Contact Area.

You can then see if the layout needs to be optimized manually or by using the optimization commands described above.

8.5.3 Display fill level

More connections mean more wires inside the ducting. As a result, there is a risk of running out of space, especially at duct nodes. The Display Fill Level command shows how much duct space is used at nodes. Circles are displayed with their diameter representing the amount of space taken up. To use the command:

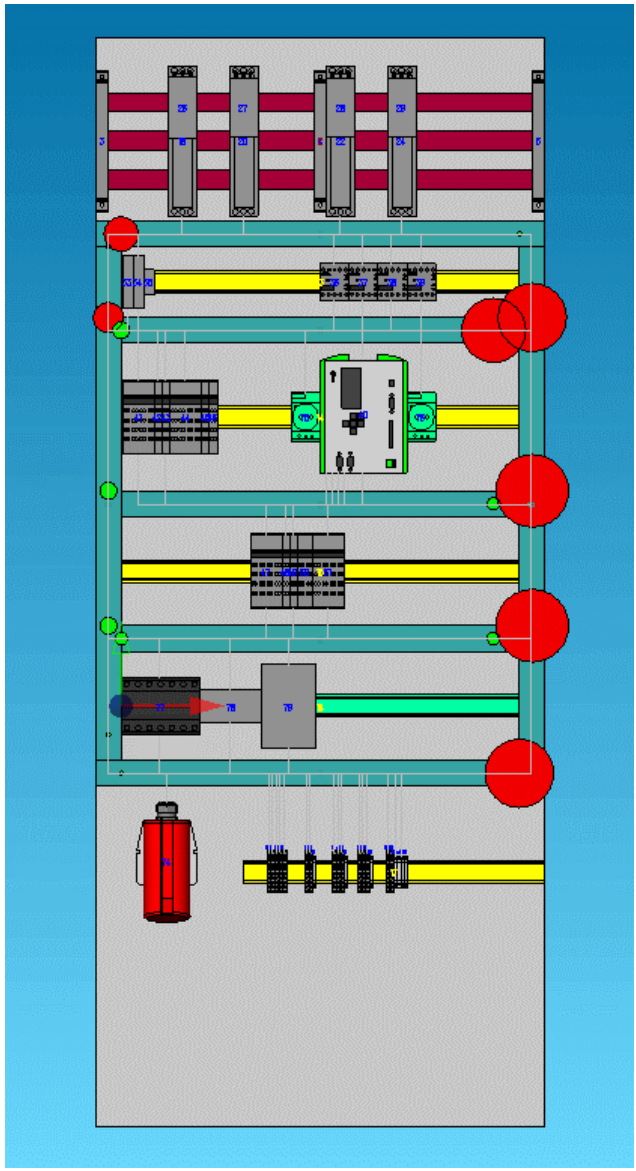
1. Display the wiring list



2. Click the [Act. rel] button to update the permitted materials and calculate the fill factors.

3. Run the length calculation

4. On the menu, select Routing – Display Fill Level.



Note: The fill factor is affected by the insulation diameter recorded in the materials database.

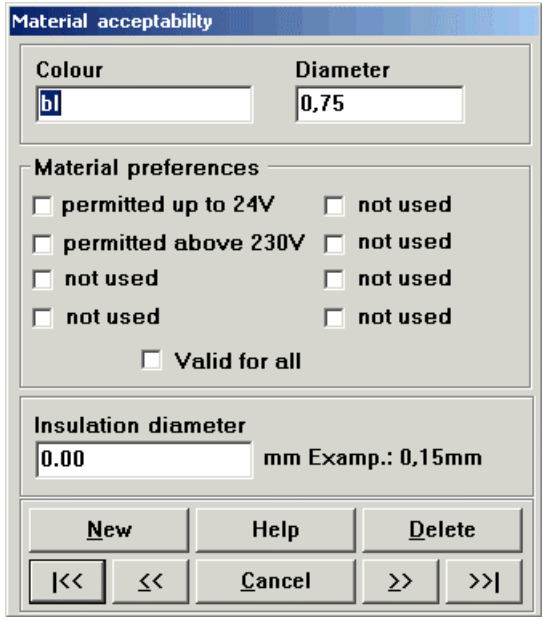
8.6 Global material properties

Global material properties can be specified that affect the routing in ducts.

The material properties correspond to the master data for wires and contain the set of possible cross-sections and colours.

You can specify material properties under Information/Security – Settings (Production). A maximum of eight different properties can be specified.

By default, all eight properties are valid when a duct is placed in an eCabinet drawing. The available options can be restricted afterwards. For example, a duct can be limited to certain voltages by editing the appropriate properties box.



Note: When you run the length calculation, material properties are matched with duct properties of the same name. A duct is only used in a possible routing path and included in the length calculation process if the two sets of properties match.

Note: The specified insulation diameter affects the fill factor, which is used to display space utilization at duct nodes.

8.7 Placing outlets

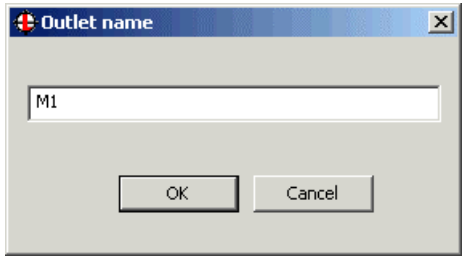
Generally, the problem of length calculation can be solved with a two-dimensional enclosure or working surface layout.

There are, however, situations that cannot be handled in this way – for example if a wire is to be routed straight to a side panel from a mounting panel.

To place an outlet, select Routing – Place Outlets on the menu.

An outlet can only be placed on a duct. Select the command and then click the duct you want to create an outlet in. The outlet moves with the cursor until you click again to set it down at the precise location you want.

You are then asked to name the outlet:



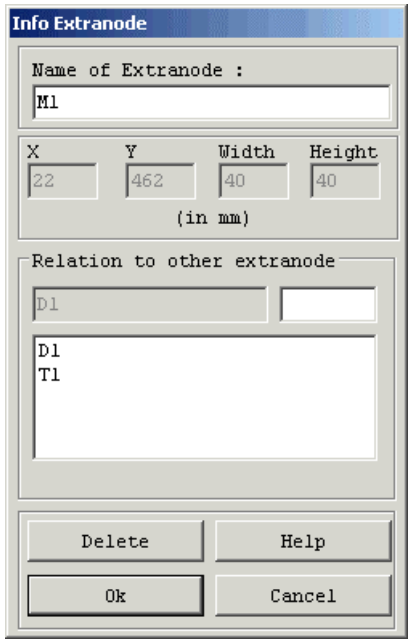
Type a name and choose [OK].

For each outlet, two inlets – two branching points – can be specified and assigned.

Note: All these objects are included in global length calculation. If any routing fields are specified, they are only included if the outlets and matching inlets are in the specified fields.

To assign an outlet to an inlet, open the context menu by right-clicking the outlet in the drawing or in the Navigator and choose ‘Assign’.

A dialogue is displayed:



The upper part of the dialogue shows basic information on the outlet.

The lower part is used to specify distances to other outlets in the drawing. Two additional inlets can be specified for each outlet, but one is enough for the usual purpose of running a cable from one unit to another.

Click the inlet you want in the list to copy it to the input line. Now specify a distance for the inlet by typing a distance in mm. When you check the properties of the inlet, you will see that the distance from the outlet is automatically stated.

8.8 Specifying connection ducts

By default, only ducts within the applicable routing field (e.g. mounting panel) are included in length calculation.

All ducts specified as connection ducts are also included.

Note: Ducts are always tagged as connection ducts for a single calculation operation and should be returned to normal afterwards. When working with multiple routing fields, the increased number of connections can slow down the calculation process. It is best to make a habit of checking the layout and making the necessary adjustments before each length calculation.

Connection ducts are shown in a different colour in the layout. You can choose the colour in the eCabinet parameters.

First, go ‘Edit – SuperEdit (Multiple Selection)’ and select the ducts you want to make into connection ducts.

Then press function key **[F11]** or choose ‘Routing – Set Connection Duct’ on the menu.

The ducts are now connection ducts and will be included as such in the length calculation that follows.

8.9 Duct (un)important for length calculation

When you run a length calculation for two or more routing fields, some ducts may be left without any wires through them. You can allow for this by tagging these ducts as unimportant before running the calculation.

This reduces the number of permutations in the network and hence the computation time.

First, go ‘Edit – SuperEdit (Multiple Selection)’ and select the ducts you want to make into connection ducts.

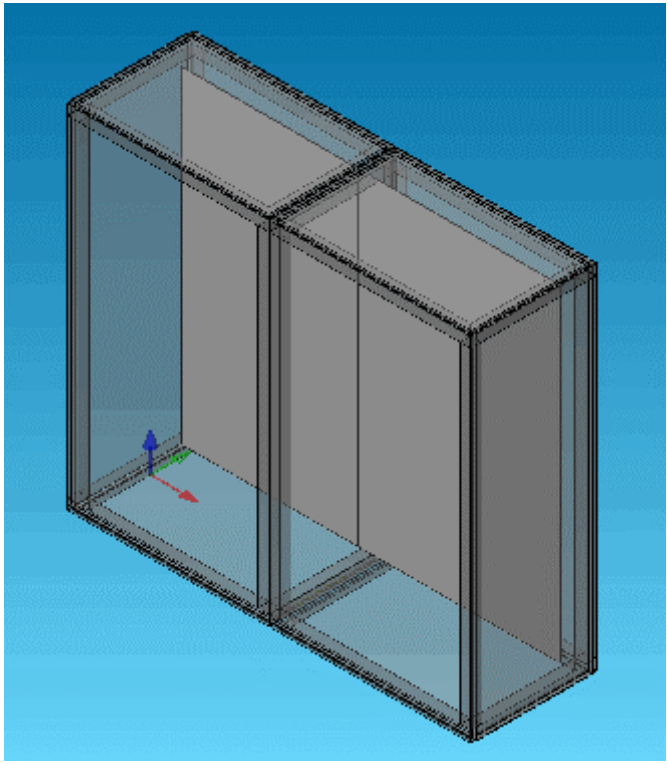
Then press function key **[F12]** or choose ‘Routing – Duct Unimportant’ on the menu.

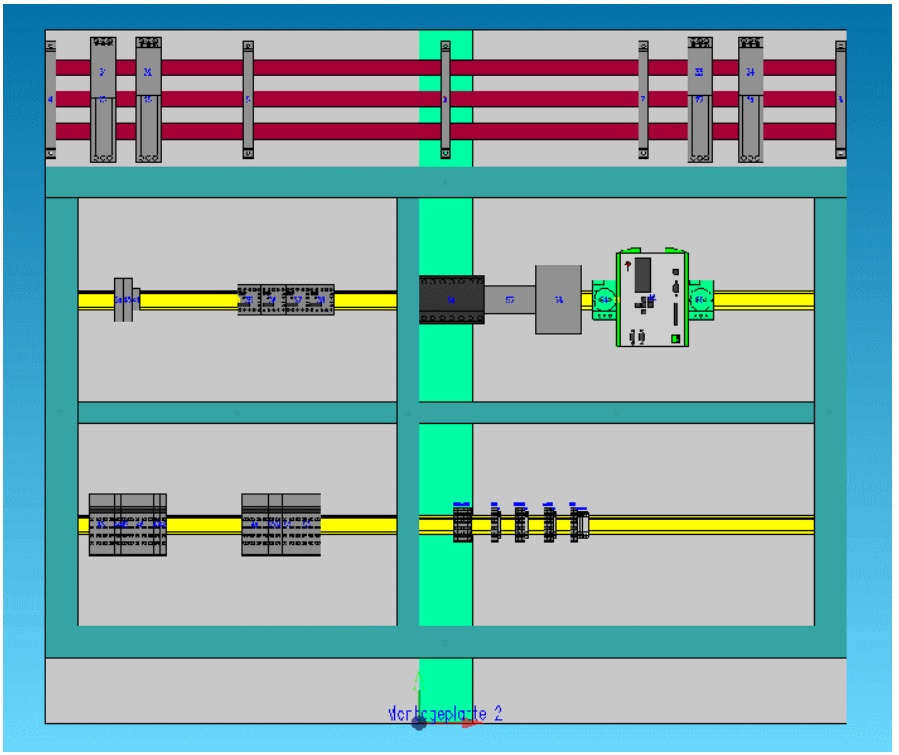
The duct is now tagged as unimportant and will not be included in the length calculation that follows. It is displayed with a different colour on the screen. You can choose the colour in the eCabinet parameters.

Note: The ‘connection duct’ property overrides the ‘unimportant’ property, so a connection duct tagged as unimportant will still be included in length calculation.

8.10 Virtual routing fields

A virtual routing field can be used to combine two directly adjacent routing fields into one. The typical example is a number of mounting panels at the same height in neighbouring enclosures.





Normally, one routing field is created for each mounting panel. By creating a virtual routing field, you can combine a number of mounting panels like the three shown here and run the length calculation for all of them in a single operation.

To place the virtual routing field, select Routing – Create Routing Field on the menu.

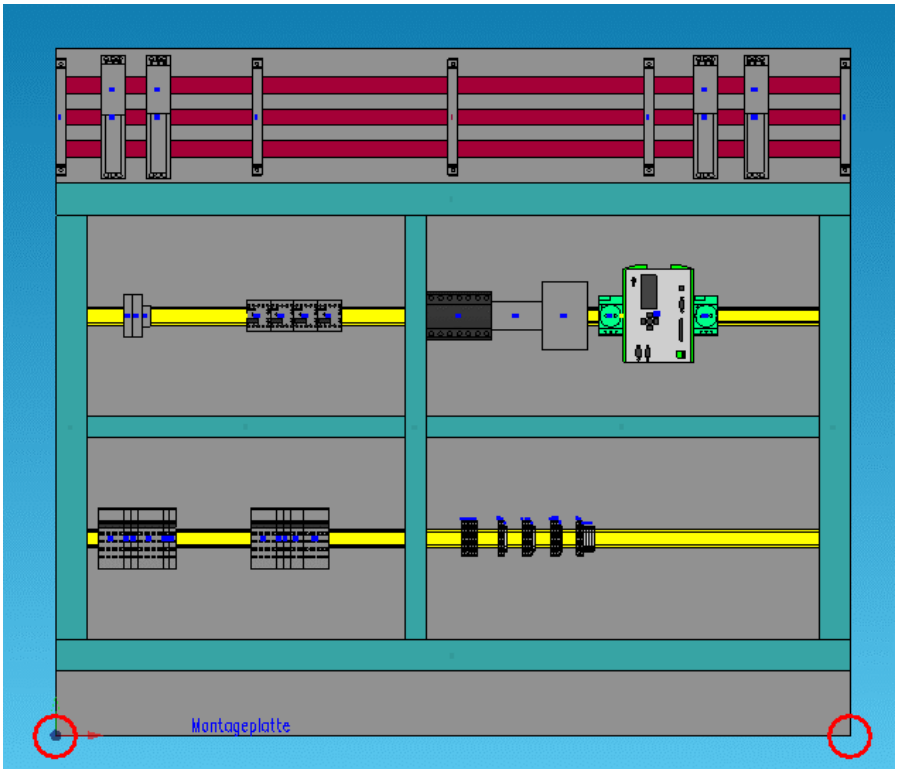
You are asked to select the X coordinate of the left-hand corner of the new routing field.

Select the bottom left corner of the left-hand mounting panel.

Now select the bottom right corner of the right-hand mounting panel.

The program next asks for a name for the routing field. Type a name and choose [OK].

The routing field is displayed as a coloured solid in the drawing.

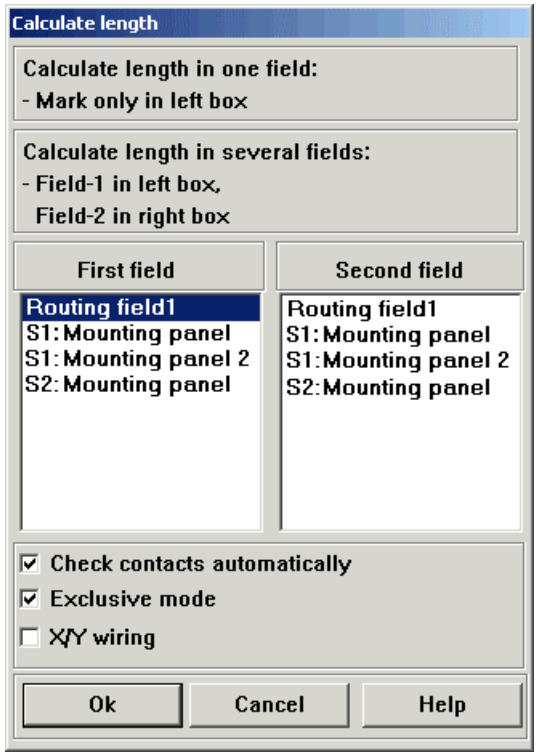


Points specifying a routing field

Note: Creating a virtual routing field does not remove the need to make ducts that span more than one routing fields into **connection ducts**.

Run the automatic length calculation.

The virtual routing field is now listed for selection with the other routing fields.



Even if you limit the length calculation for the selected routing field, all mounting panels comprising the virtual routing field will be routed.

Tip:

If you want to use the virtual routing field feature but still want the mounting panels to be displayed separately, you can hide the virtual routing field in the Navigator.

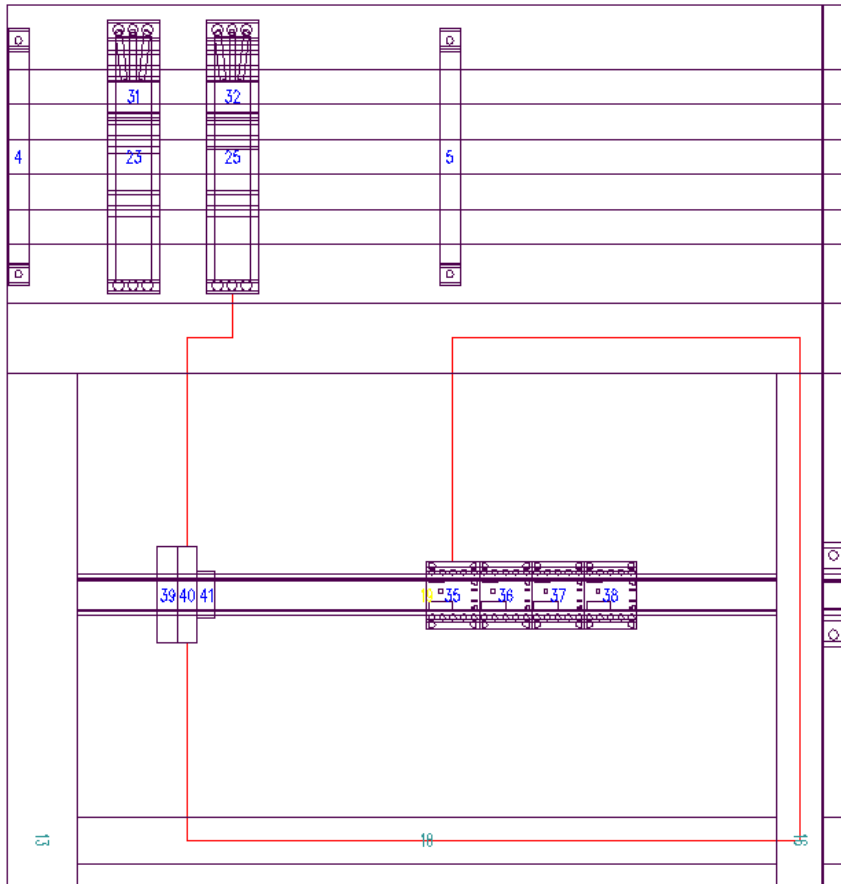
8.11 Placing a blocking line

Blocking lines can be used to influence the length calculation.

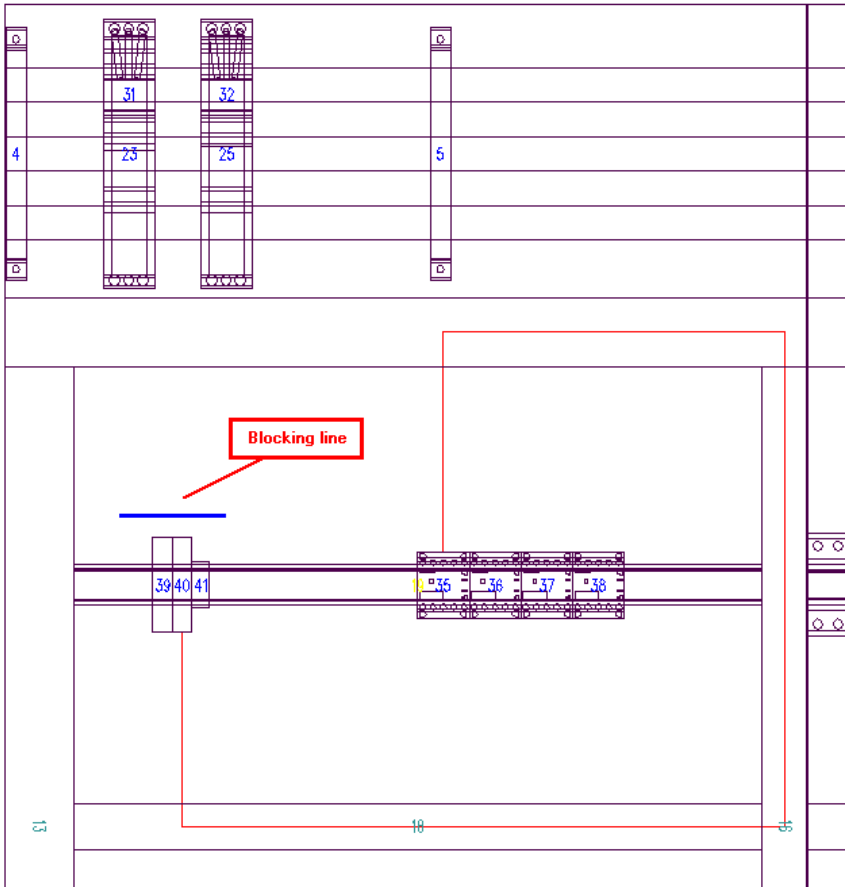
Blocking lines are laid parallel to ducts and block a wire's path between a device and a duct.

Example:

Layout without blocking line



Layout with blocking line: Wire is not allowed to pass



8.12 Length calculation for part of a routing field

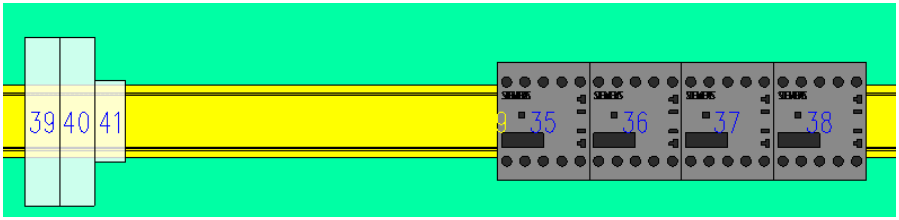
Length calculation can be performed for all routing fields or selected routing fields. You can narrow down the selection even further by limiting it to specific devices.

If you select individual devices before running the length calculation, wires will only be computed for the selected devices.

The devices must be selected before running the length calculation, by using the ‘SuperEdit – Multiple Selection’ command. The devices must be selected when the length calculation starts.

Select the command on the menu by choosing ‘Edit – SuperEdit (Multiple Selection)’.

Then click the devices you want to select.



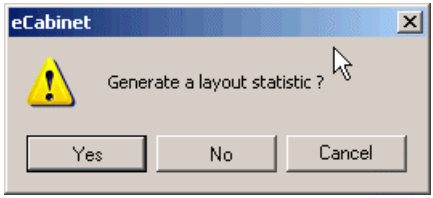
Now run the length calculation as normal. The lengths of wires are only calculated for the selected devices.

8.13 Automatic length calculation

Select the automatic length calculation command on the menu by choosing Routing – Route Wires Automatically.

8.13.1 Layout analysis

Before you run the automatic length calculation, you have the option of generating a layout analysis:



The layout analysis warns you of certain types of layout problem that might affect the length calculation you are planning to run.

The first table in the analysis shows all items that cannot be included in the calculation. These are items where one of two devices to be connected is not in the layout. Length calculation is not possible in such cases.

A second and third table in the printout shows devices for which the specified length cannot be computed because the wire will not go through the available ducting. That is, there is no duct for the wire from the devices in question. A separate analysis is produced for devices -1 and devices -2.

Example:

A device is to be connected from underneath, but there is no duct beneath it.

The fourth and fifth tables list all items for which the contact length for the specified terminal cannot be uniquely determined. This analysis is also produced separately for devices -1 and devices -2, for example when two contacts on a unit have the same name.

The remaining tables are provided for information only and do not affect the routing.

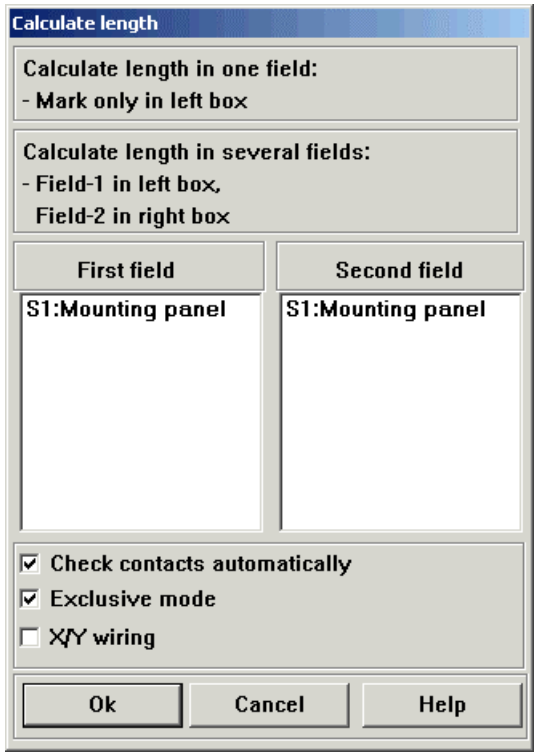
The next four tables show which contacts on devices -1 or devices -2 connect to two, three or more wires. If a double end sleeve is specified in the master data as a device's contact type and there are two or three wires on one contact, the wire end is only stripped for one double end sleeve. If there is only one wire on a contact on such a device, the contact type is automatically changed to crimping. This is done in the machine's parts list program.

The last table lists connections where the material does not match – where either the cross-section or the wire colour changes on a connection.

Uncalculable positions				
Color	**ø**	**From device/contact**	**To device/contact**	**Rem.**
sw	1.50	=FL-H21:X2	=SPS-A32:1.2.1	
sw	1.50	=FL-H22:X1	=SPS-A22:1.1.1	
sw	1.50	=FL-H22:X2	=FL-S21:21	
sw	1.50	=FL-S21:13	=SPS-A23:1.1.1	
sw	1.50	=FL-S21:14	=FL-S22:14	
sw	1.50	=FL-S21:22	=SPS-A22:1.2.1	
sw	1.50	=FL-S22:13	=SPS-A23:1.1.2	
sw	1.50	=FL-S22:14	=SPS-A23:1.2.1	
sw	1.50	=SPS-?:??	=SPS-A11:?	
sw	1.50	=SPS-A11:6	SE:??	
sw	1.50	=SPS-A11:L	=ALL- A13:-A13:+	
sw	1.50	=SPS-A21:1.1.4	SE:??	
sw	1.50	=SPS-A21:2.1.4	SE:??	
sw	1.50	=SPS-A21:4.1.1	=ALL- A13:-A13:+	
sw	1.50	=SPS-A21:4.1.2	=ALL- A13:-A13:+	
sw	1.50	=SPS-A21:4.2.2	=ALL- A13:-A13:+	
sw	1.50	=SPS-A24:2.2.1	=A1-H22:X1	
sw	1.50	=SPS-A24:3.2.1	=A2-H22:X1	
sw	1.50	=SPS-A24:4.2.1	=A3-H22:X1	
sw	1.50	=SPS-A31:4.1.1	=ALL- A13:-A13:+	
sw	1.50	=SPS-A31:4.1.2	=ALL- A13:-A13:+	
sw	1.50	=SPS-A31:4.2.2	=ALL- A13:-A13:+	
sw	1.50	=SPS-A35:2.1.1	=A1-S21:22	
sw	1.50	=SPS-A35:2.1.4	=A1-S22:22	
sw	1.50	=SPS-A35:2.2.1	=A1-S23:14	
sw	1.50	=SPS-A35:2.2.4	=A2-S21:22	
sw	1.50	=SPS-A35:3.1.1	=A2-S23:14	
sw	1.50	=SPS-A35:3.1.4	=A3-S21:22	
sw	1.50	=SPS-A35:3.2.1	=A2-S22:22	
sw	1.50	=SPS-A35:3.2.4	=A3-S23:14	
sw	1.50	=SPS-A35:4.1.1	=A3-S22:22	
sw	1.50	SE:??	=SPS-A31:1.1.4	

Contact used twice in Assembly1				
Color	**ø**	**From device/contact**	**To device/contact**	**Rem.**
sw	1.50	:11	=ALL-?:??	
sw	1.50	=A1-H22:X2	=A2-H22:X2	
sw	1.50	=A1-S22:21	=A1-S23:13	
sw	1.50	=A1-S23:13	=A2-S22:21	
sw	1.50	=A2-H22:X2	=A3-H22:X2	

8.13.2 Setting parameters for automatic length calculation



Automatic contact detection

This parameter is enabled by default. See above for an explanation.

Exclusive mode

In this mode, once the current network has been determined, every possible path is checked. Because this can take a very long time, exclusive mode can be disabled. Doing so excludes a large number of paths from the calculation if the layout satisfies the following criteria:

- The duct system is fully closed: Once a horizontal duct has been used with specific X/Y coordinates, higher ducts are ignored if device A is higher and device B is lower than the selected horizontal duct.

- Once a horizontal duct has been used with specific X/Y coordinates, lower ducts are ignored if device A is lower and device B is higher than the selected horizontal duct.
- The same applies for vertical ducts.

X/Y wiring

A common alternative to the conventional and widely used duct routing system is the Lütze wiring system. In this system, ducts in the drawing do not route wiring: they merely represent transition points for wires from devices. The path from any such point to a similar point on the way to another device is travelled in X and Y to determine the required wire length. By definition, wire length optimization is not possible with this system.

8.13.3 Running the length calculation

For each connection in the wiring list that is capable of being calculated, an item is added to the wires parts list. Items calculated on an earlier run are kept and are not recalculated.

Connections in the wiring list that cannot be calculated are not highlighted. You can check the results of the length calculation by comparing the wires parts list with the wiring list. The program also gives you the option of running a layout immediately before the length calculation.

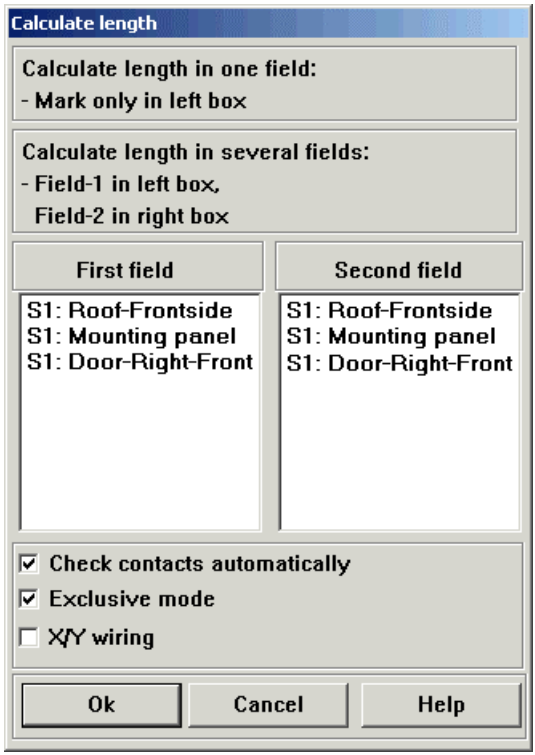
Connections in the wiring list that have already been calculated are marked with a '*'.

sw	1.50 : =SPS-A35	(3.1.4) - =A3-S21	(	22)
sw	1.50 : =SPS-A35	(3.2.1) - =A2-S22	(	22)
sw	1.50 : =SPS-A35	(3.2.4) - =A3-S23	(	14)
sw	1.50 : =SPS-A35	(4.1.1) - =A3-S22	(	22)
sw	1.50 : SE	(??) - =SPS-A31	(	1.1.4)
*br	1.50 : =SPS-A21	(2.1.3) - =SPS-A31	(	1.1.3)
*ge	1.50 : =SPS-A21	(2.2.1) - =SPS-A31	(	1.2.1)
*gn	1.50 : =SPS-A21	(2.1.1) - =SPS-A31	(	1.1.1)
*gr	1.50 : =SPS-A21	(2.2.2) - =SPS-A31	(	1.2.2)
*rs	1.50 : =SPS-A21	(2.1.2) - =SPS-A31	(	1.1.2)
*sw	1.50 : =A1-K11	(1) - =A1-Q11	(	2)
*sw	1.50 : =A1-K11	(3) - =A1-Q11	(	4)
*sw	1.50 : =A1-K11	(5) - =A1-Q11	(	6)
*sw	1.50 : =A1-K11	(A2) - =A2-K11	(	A2)

8.13.3.1 Length calculation for specific routing fields

Specifying routing fields results in length calculation automatically being performed for specific routing fields or for all fields. 'Fields' are mounting panels, doors, side panels, etc.

eCabinet specifies routing fields automatically.



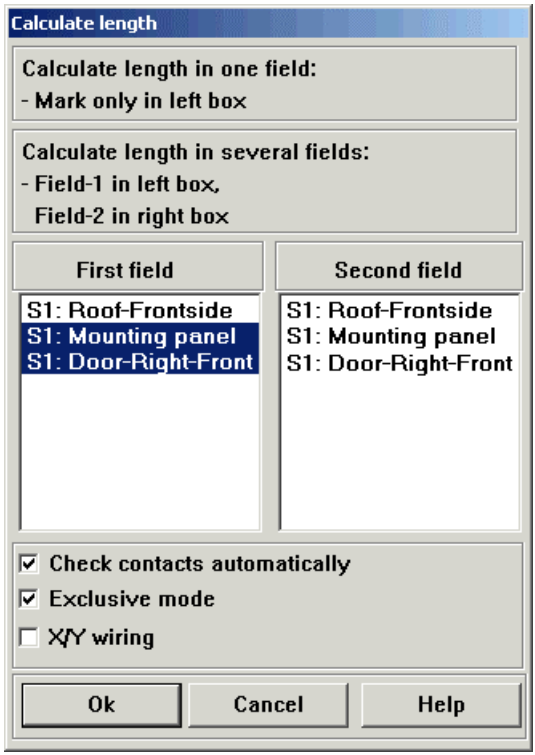
The list shows all populated routing fields.

Only devices and ducts in the selected field are included in the length calculation.

Note especially that ducts are only included if they are entirely inside the specified field.

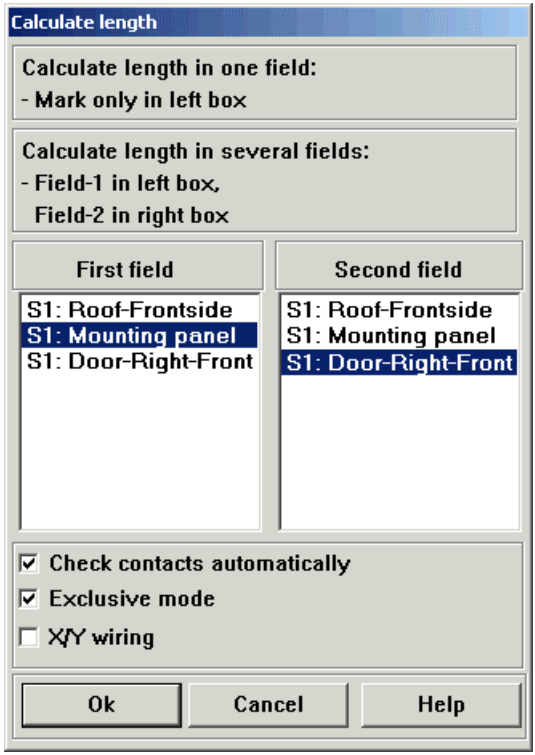
Note: To run the length calculation for specific fields, only select fields in the left-hand list.

Multiple fields can be selected in the left-hand list. To do this, hold down the [CTRL] key while you click the field names you want.



8.13.3.2 Length calculation across multiple routing fields

If routing fields are selected in both the left-hand and the right-hand list, the length calculation automatically takes in all the selected fields. Every combination of the fields is calculated.



Only devices and ducts in the selected fields are included in the length calculation.

Note especially that ducts are only included if they are entirely inside the specified field.

The length calculation additionally includes all ducts that have been tagged as connection ducts, for example between neighbouring mounting panels.

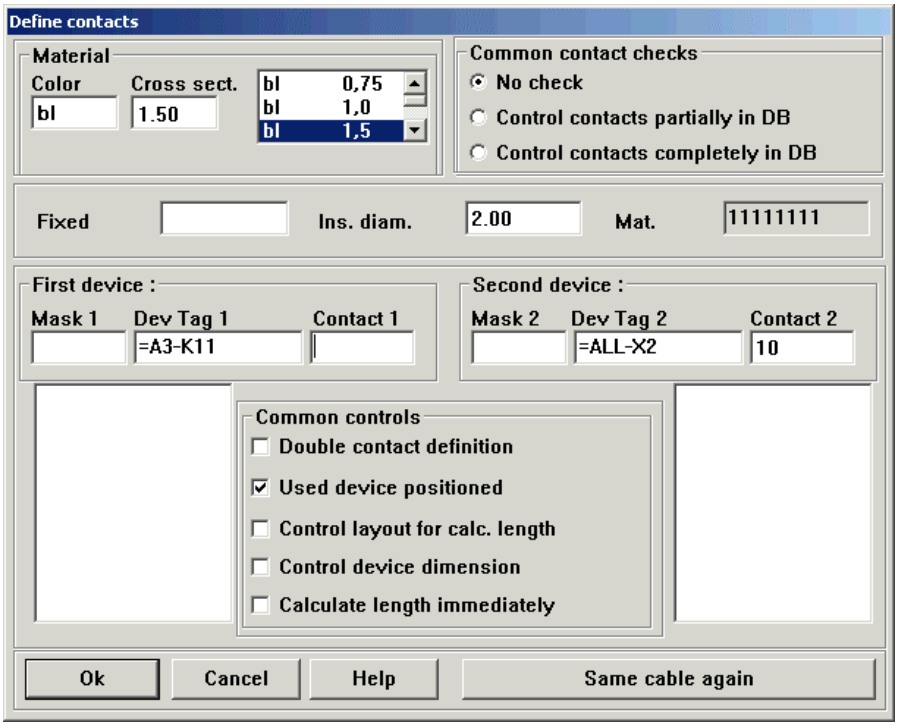
Note: Length calculation across multiple routing fields only calculates the lengths of wires that cross between fields. The connections between fields are specified as outlets before running the length calculation.

8.14 Manual routing

In addition to wires based on wiring information from EPLAN and included in the wiring list, you can also add and route wires manually.

On the menu, select Routing – Specify Wires Manually, and click two devices in the drawing to select them. A connection is created between the two selected devices.

Once you have clicked the two devices to select them, a dialogue opens for you to specify the two contact names and the wire colour and cross-section. The colour and cross-section are retained from one time to the next so that you only need to specify them if they change.



If you want to route two or more wires from device A to device B, use the [Another Wire] button. This saves you having to select the devices for a second time.

8.14.1 Plausibility checking

The parameters that can be enabled in the manual routing dialogue are explained in the following.

Calculate length immediately

If this parameter is enabled, length calculation is performed immediately when wiring is added. This can cause slight delays when working with large layouts. Disabling this parameter stops length calculation from being performed. New contacts are added to the contacts list after plausibility checking in the layout.

[OK]

Checks the plausibility of the entered data. This is immediately followed by length calculation (if enabled) and the contact is added both to the contacts list and to the wires parts list.

If length calculation cannot be performed for any reason, a message is displayed and the contact in question is not added to the contacts list or wires parts list.

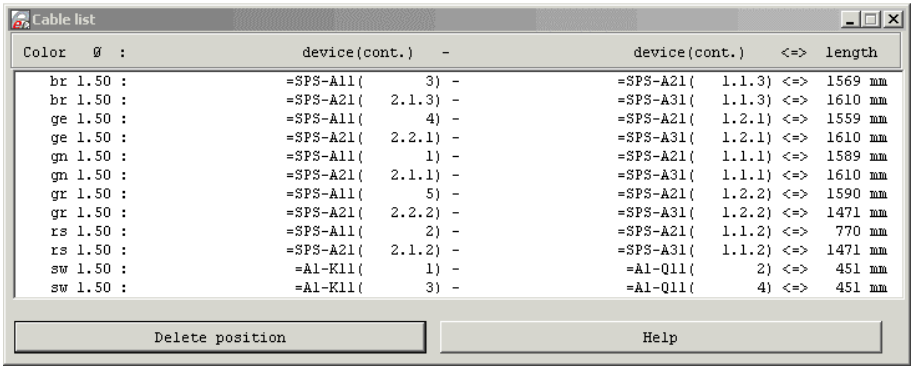
[Cancel]

Ignores the most recent changes and cancels the length calculation.

8.15 Display wires parts list

Once length calculation has been carried out, all calculated items are copied from the wiring list to the wires parts list. Each item now additionally includes a length in mm.

To view the wires parts list, select Routing – Display Wires Parts List on the menu (or press the [F2] key).



[Delete item]

All items added to the wires parts list can be deleted.

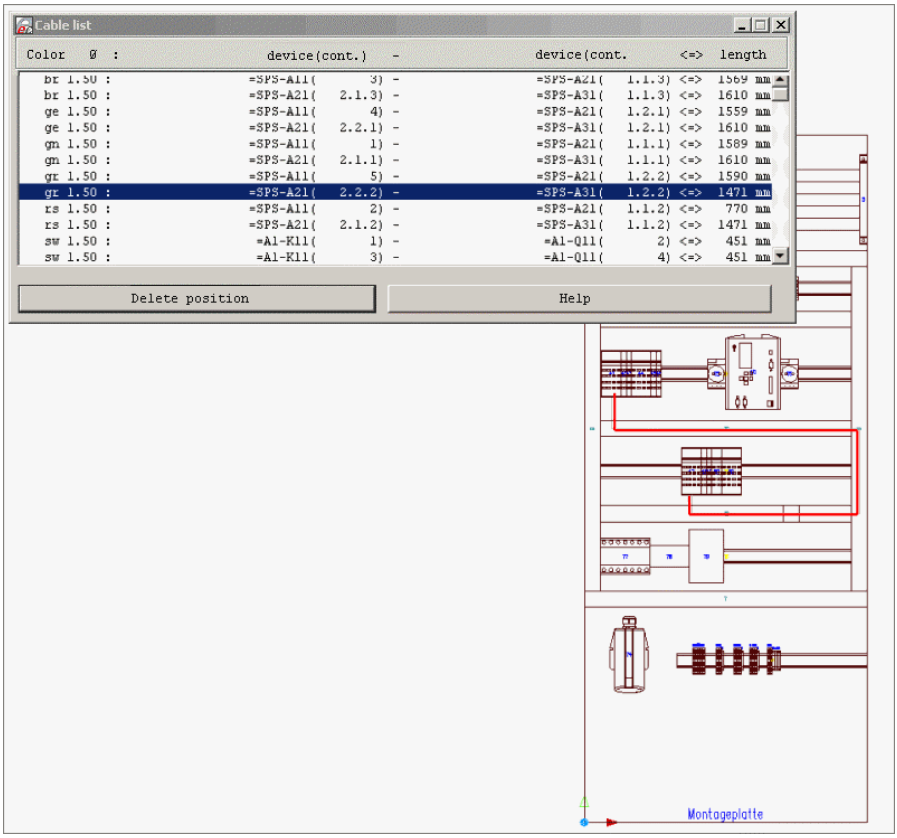
For example, you might want to delete a wire to stop it from being exported to and made up by the TRIATHLON wiring centre.

A wire's path can be graphically displayed and viewed in the layout by selecting the wire in the wires parts list. All wires must be hidden first (not deleted!).

To hide all wires, select Routing – Routing Output – Show/Hide Wires on the menu.

By default, all wires are displayed. This command can be used to hide all wires (and a second time to show them again).

When all wires are hidden, open the wires parts list and select the wire you want. The selected wire is displayed in the layout and you can see its path.



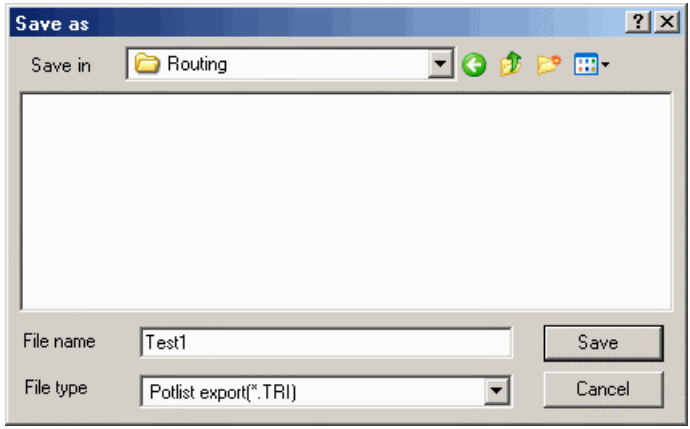
Note: Wires that run across two or more routing fields are not displayed graphically.

8.16 Exporting to the TRIATHLON wiring centre

Once you have completed an enclosure and run the length calculation in e-Cabinet, you can transfer the data needed to make up the wires to the TRIATHLON wiring centre.

On the menu, select Routing – Routing Output – Export Triathlon Cable Centre.

A file is generated with a .TRI file name extension.



This file contains all the information needed to make up the wires.

All other parameters relating to the material to be processed are taken from the materials database.

Note: Although projects are usually modified many times before production, the full current data is always exported to the machine. The decision whether to treat the data as an entirely new parts list or as an update is made on the machine.

The prepared data from eCabinet can be used to make up wiring to various criteria:

- Full project
- Specific material selection
- Specific routing field
- Across routing fields
- Specific assembly
- Specific contact
- Specific loop

8.17 Printed routing output

8.17.1 Print wires parts list

On the menu, select Routing – Routing Output – Print Wires Parts List.

The wires parts list shows the information passed on to the TRIATHLON wiring centre for making up wires. One use for the printout is to issue it with the made-up wires so that the wiring technician can record any problems in the remarks column. Once the work is complete, the list containing the technician's notes is returned to a superior for checking.

The wires parts list is headed with identifying information such as the project name.

Each list additionally has a data header. This shows the project parameters and so clearly identifies the job to which the list belongs.

The number of pages in the printout is shown at the top left of the first page. Each page also shows the type of list and project name at bottom left and the page number at bottom right. A new table header is printed at each change in material or colour.

Total pages number: 5

eCabinet parts list - Project: Z:Projekte\Routing\Füllfaktor\Fuellfaktor

Color: br Diameter: 1.50					
Color	**ø**	**From-assembly/contact**	**To-assembly/contact**	**Length(mm)**	**Rem.**
br	1.50	=SPS-A11:3	=SPS-A21:1.1.3	1569	
br	1.50	=SPS-A21:2.1.3	=SPS-A31:1.1.3	1610	
Insulation diameter : 0.00			**Total length : 3.17 m**		

Color: y Diameter: 1.50					
Color	**ø**	**From-assembly/contact**	**To-assembly/contact**	**Length(mm)**	**Rem.**
ge	1.50	=SPS-A11:4	=SPS-A21:1.2.1	1559	
ge	1.50	=SPS-A21:2.2.1	=SPS-A31:1.2.1	1610	
Insulation diameter : 0.00			**Total length : 3.17 m**		

Color: gr Diameter: 1.50					
Color	**ø**	**From-assembly/contact**	**To-assembly/contact**	**Length(mm)**	**Rem.**
gn	1.50	=SPS-A11:1	=SPS-A21:1.1.1	1589	
gn	1.50	=SPS-A21:2.1.1	=SPS-A31:1.1.1	1610	
Insulation diameter : 0.00			**Total length : 3.17 m**		

Color: grey Diameter: 1.50					
Color	**ø**	**From-assembly/contact**	**To-assembly/contact**	**Length(mm)**	**Rem.**
gr	1.50	=SPS-A11:5	=SPS-A21:1.2.2	1590	
gr	1.50	=SPS-A21:2.2.2	=SPS-A31:1.2.2	1471	
Insulation diameter : 0.00			**Total length : 3.17 m**		

Color: red Diameter: 1.50					
Color	**ø**	**From-assembly/contact**	**To-assembly/contact**	**Length(mm)**	**Rem.**
rs	1.50	=SPS-A11:2	=SPS-A21:1.1.2	770	
rs	1.50	=SPS-A21:2.1.2	=SPS-A31:1.1.2	1471	
Insulation diameter : 0.00			**Total length : 3.17 m**		

Color: bl Diameter: 1.50					
Color	**ø**	**From-assembly/contact**	**To-assembly/contact**	**Length(mm)**	**Rem.**
Date:		2005-07-16	Project name:	ECD_WER	
User:		wer	Commission :	Monheim	
Respons. for project:		M.Wermann	Customer:	WER	
EPLAN S&S		AN DER ALTEN ZIEGELEI 2	Installation:	eCabinet HMI2005	
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8.17.2 Print wiring list

On the menu, select Routing – Routing Output – Print Wiring List. The wiring list contains all information imported from EPLAN and used in length calculation.

Total pages number: 8

eCabinet Contact list - Project Z:Projekte\Routing\Füllfaktor\Fuellfaktor

Color: sw Diameter: 1.50				
Color	**ø**	**From assembly/contact**	**To assembly/contact**	**Rem.**
sw	1.50	:_1	=ALL-E41:L	
sw	1.50	:_3	=ALL-E42:L	
sw	1.50	:_5	=ALL-E41:N	
sw	1.50	:_7	=ALL-E42:N	
sw	1.50	:11	=ALL-?:??	
sw	1.50	:12	=ALL-A13:P2	
sw	1.50	=A1-H22:X2	=A2-H22:X2	
sw	1.50	=A1-Q11:22	=A1-S21:21	
sw	1.50	=A1-S22:21	=A1-S23:13	
sw	1.50	=A1-S23:13	=A2-S22:21	
sw	1.50	=A2-H22:X2	=A3-H22:X2	
sw	1.50	=A2-L1:??	=A2-Q11:1	
sw	1.50	=A2-L2:??	=A2-Q11:3	
sw	1.50	=A2-L3:??	=A2-Q11:5	
sw	1.50	=A2-Q11:22	=A2-S21:21	
sw	1.50	=A2-S22:21	=A2-S23:13	
sw	1.50	=A2-S23:13	=A3-S22:21	
sw	1.50	=A3-L1:??	=A3-Q11:1	
sw	1.50	=A3-L2:??	=A3-Q11:3	
sw	1.50	=A3-L3:??	=A3-Q11:5	
sw	1.50	=A3-Q11:22	=A3-S21:21	
sw	1.50	=A3-S22:21	=A3-S23:13	
sw	1.50	=ALL- A13:-A13:+	=FL-A11:US2	
sw	1.50	=ALL- G11:-G11:+	=ALL-F12:1	
sw	1.50	=ALL- L -L+:??	=ALL-F21:1	
sw	1.50	=ALL- M21:-M21:+	=ALL-X1:2	
sw	1.50	=ALL-?:??	=ALL-A12:PE	
sw	1.50	=ALL-A13:-	=FL-A11:GND	
sw	1.50	=ALL-A13:P1	=ALL-S22:13	
sw	1.50	=ALL-A41:L	=ALL-X22:L	
sw	1.50	=ALL-A41:N	=ALL-X22:N	
sw	1.50	=ALL-A41:PE	=ALL-X22:PE	
sw	1.50	=ALL-E41:L	=ALL-E42:L	
sw	1.50	=ALL-E41:N	=ALL-E42:N	

Date:	2005-07-16	Project name:	ECD_WER
User:	wer	Commission :	Monheim
Respons. for project:	M.Wermann	Customer:	WER
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8.17.3 Print remaining parts list

On the menu, select Routing – Routing Output – Print Remaining Parts List.

The remaining parts list contains all items in the wiring list at the time of printing except those for which lengths have already been calculated.

Total pages number: 4

eCabinet Remaining parts list - Project Z:Projekte\Routing\Füllfaktor\Fuellfaktor

Color: sw Diameter: 1.50				
Color	**ø**	**From assembly/contact**	**To assembly/contact**	**Rem.**
sw	1.50	:_1	=ALL-E41:L	
sw	1.50	:_3	=ALL-E42:L	
sw	1.50	:_5	=ALL-E41:N	
sw	1.50	:_7	=ALL-E42:N	
sw	1.50	:11	=ALL-?:??	
sw	1.50	:12	=ALL-A13:P2	
sw	1.50	=A1-H22:X2	=A2-H22:X2	
sw	1.50	=A1-Q11:22	=A1-S21:21	
sw	1.50	=A1-S22:21	=A1-S23:13	
sw	1.50	=A1-S23:13	=A2-S22:21	
sw	1.50	=A2-H22:X2	=A3-H22:X2	
sw	1.50	=A2-L1:??	=A2-Q11:1	
sw	1.50	=A2-L2:??	=A2-Q11:3	
sw	1.50	=A2-L3:??	=A2-Q11:5	
sw	1.50	=A2-Q11:22	=A2-S21:21	
sw	1.50	=A2-S22:21	=A2-S23:13	
sw	1.50	=A2-S23:13	=A3-S22:21	
sw	1.50	=A3-L1:??	=A3-Q11:1	
sw	1.50	=A3-L2:??	=A3-Q11:3	
sw	1.50	=A3-L3:??	=A3-Q11:5	
sw	1.50	=A3-Q11:22	=A3-S21:21	
sw	1.50	=A3-S22:21	=A3-S23:13	
sw	1.50	=ALL- A13:-A13:+	=FL-A11:US2	
sw	1.50	=ALL- G11:-G11:+	=ALL-F12:1	
sw	1.50	=ALL- L -L+:??	=ALL-F21:1	
sw	1.50	=ALL- M21:-M21:+	=ALL-X1:2	
sw	1.50	=ALL-?:??	=ALL-A12:PE	
sw	1.50	=ALL-A13:-	=FL-A11:GND	
sw	1.50	=ALL-A13:P1	=ALL-S22:13	
sw	1.50	=ALL-A41:L	=ALL-X22:L	
sw	1.50	=ALL-A41:N	=ALL-X22:N	
sw	1.50	=ALL-A41:PE	=ALL-X22:PE	
sw	1.50	=ALL-E41:L	=ALL-E42:L	
sw	1.50	=ALL-E41:N	=ALL-E42:N	

Date:	2005-07-16	Project name:	ECD_WER
User:	wer	Commission:	Monheim
Respons. for project:	M.Wermann	Customer:	WER
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Note:

As with the wires parts list, it is useful to issue the remaining parts list together with the made-up wires. The wiring technician can see what is theoretically left to do in order to finish the control panel, and has the full set of information generated from EPLAN.

Between them, the wires parts list and the remaining parts list contain all connections within a given control panel.

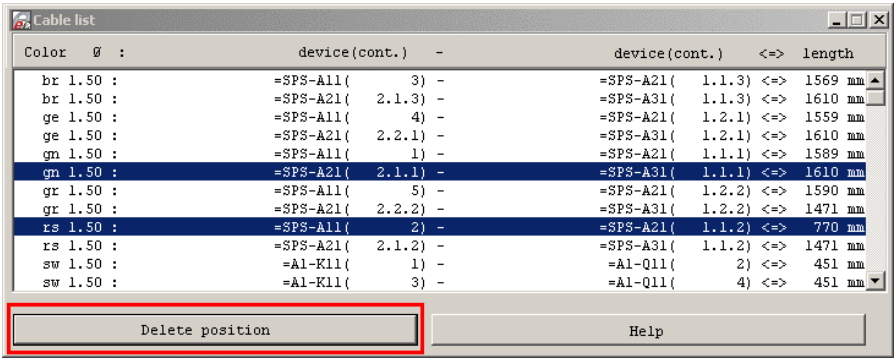
8.18 Delete wires parts list

The wiring parts list generated after length calculation can be deleted.

To delete the wires parts list, select Routing – Delete Wires Parts List on the menu.

All items in the wires parts list are deleted together with the generated paths and the graphical routing.

It is also possible to delete individual connections in the displayed wires parts list: Select one or more connections and click the [Delete Item] button.



Note:

If a device is moved or deleted in the layout, all wires associated with the device are automatically deleted.

If a duct is moved, deleted or its length modified, all wires associated with the duct are automatically deleted.

9 Drilling

Drilling toolbar

The commands for creation and output of drilling patterns are on the eCabinet Drilling toolbar.

eCabinet menu

On the eCabinet menu bar, the commands are found under Drilling/NC.

9.1 Displaying drilling view

The 'Display drilling view' command switches to drilling view. All drilling points are displayed in this view. Selecting the command again hides all drilling points.

The drilling view must be displayed in order to create drilling templates.

Drilling **Displa y drilling view**	eCabinet **Dr lling/NC | Display drillin g view**

9.2 Creating drilling templates

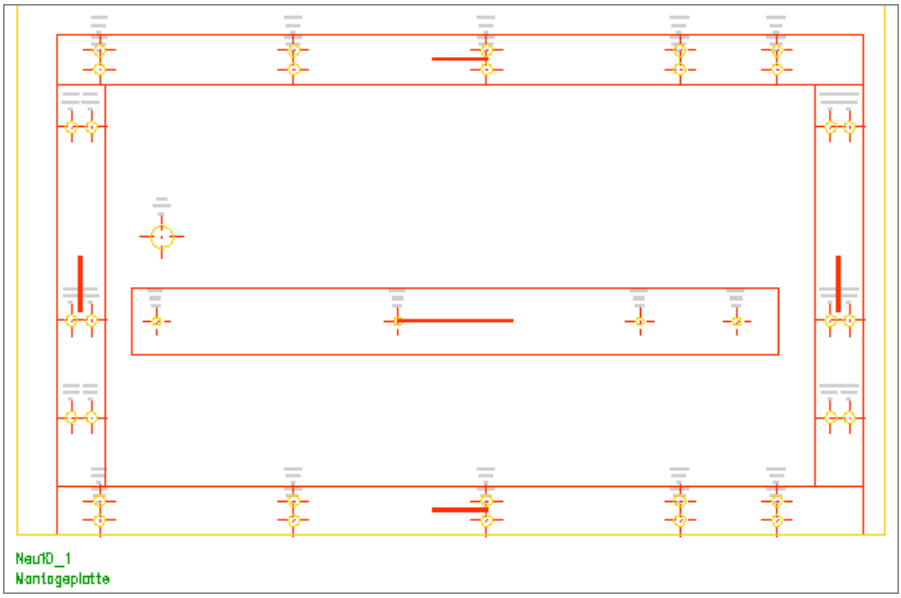
Drilling templates are 1:1-scale 2D drawings that map the coordinates for all drilling points for ducts, mounting rails and fittable components on a mounting panel. Printed out in scale 1:1, they can be used straight away as templates for drilling work.

To create drilling templates, drilling view must be active.

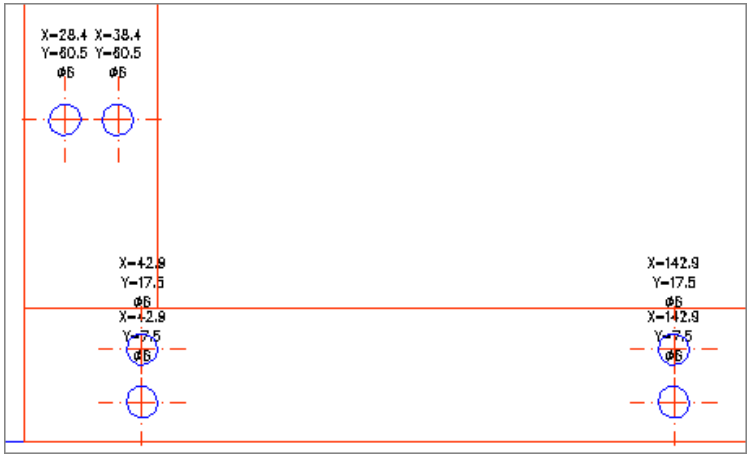
Drilling	eCabinet
Creat drilling templates	**Dr ling/NC | Create drilling te nplates**

The command works automatically. It creates a 2D drawing with drilling points and their coordinates for each fitted panel in the project and saves it under the name of the described component, e.g. MOUNTING-PANEL, in a subdirectory of the project directory.

Complete drilling pattern for a mounting panel



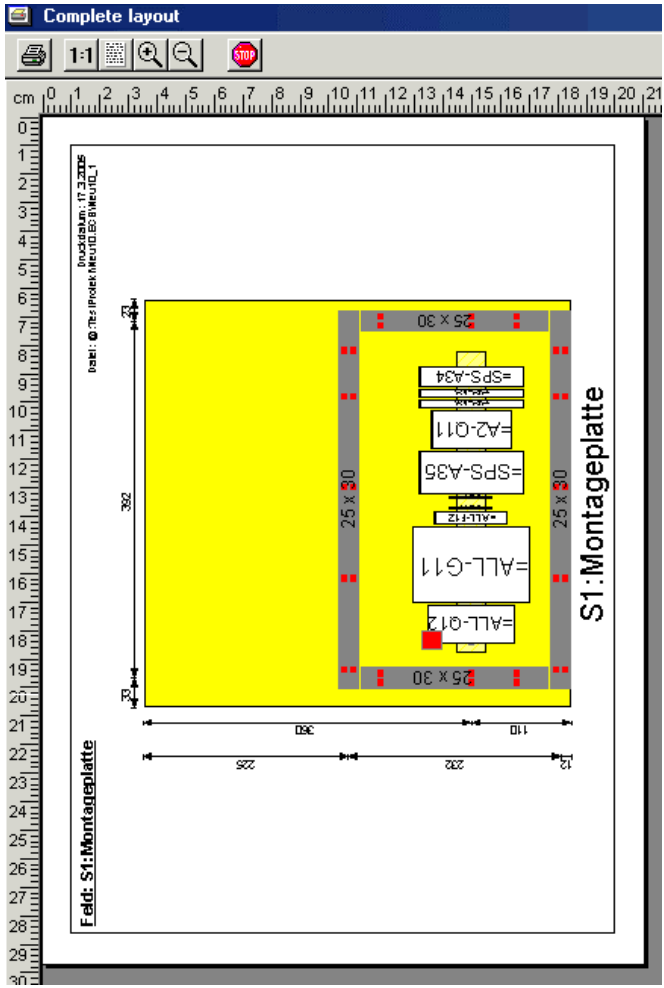
Detail view with coordinates for the drilling points



9.3 Printing a drilling layout

Drilling	eCabinet
Print drilling layout	**Drilling/NC | Print drilling layout**

Specify the sheet format, orientation and scope (current view or entire project). The drilling layout generated from the drawing is loaded in the print browser, from which it can be printed (not to scale).



9.4 eCabinet NC

NC toolbar

The eCabinet NC commands for controlling NC drilling machines are on the eCabinet NC toolbar:

eCabinet menu bar

On the eCabinet menu bar, the commands are found under ‘Drilling/NC’.

9.4.1 Specifying a blocking surface for drilling

A blocking surface excludes a specified area from the determination of drilling coordinates. Drilled holes inside blocking surfaces are not included in drilling layouts or in NC data and are not drilled.

NC	eCabinet
Spe ify blocking surface (hol s)	**D illing/NC | Specify bl cking surface (holes)**

When you have finished specifying a blocking surface, you are asked if you want to add it to the master data for the mounting panel.

To specify the blocking surface, enter two corner points of a rectangle. Alternatively, after entering the first point, enter the X and Y coordinates of the second as a number.

The blocking surface is shown in the Navigator under the mounting panel. It can be removed by right-clicking to open the context menu and selecting ‘Delete’.

9.4.2 Exporting drillings to machine

This command passes the coordinates of the drillings in the layout as control data to an NC drilling system.

NC	eCabinet
Expoı t drillings to machine	**Dril ing/NC | Export dril-ling: to machine**

Enter a file name in the File Save As dialogue. Select the machine type in the File Type box. A file is then generated with the appropriate format and extension:

- PPR drill (*.PPR)
- Steinhauer drill (*.CNS)
- Elpromatik DXF (*.DXF)
- Standard DXF (*.DXF), machine-independent

9.4.3 Exporting a workshop project

A workshop project contains full information on all components and devices in the project, including their location and wiring. The file is imported into the Triathlon system for use with control panel and wiring harness fitting machines.

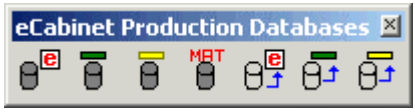
NC	eCabinet
Expoı t workshop project	**Dril ing/NC | Export wor :shop project**

Enter a file name in the File Save As dialogue. The file is given an .SPR extension.

10 Working with production master data

Production Databases toolbar

The configuration dialogues for working with production master data and for importing into an eCabinet project are accessed from the eCabinet Production Databases toolbar:



10.1 Editing master data

10.1.1 Device master data (production)

Production Databases

Device master data (production)

The production data for all devices in the project are displayed in a large dialogue. Data can be modified and new data can be added. Drilling data are also specified here on a global basis.

The [>>] and [<<] buttons go to the next or the previous record.

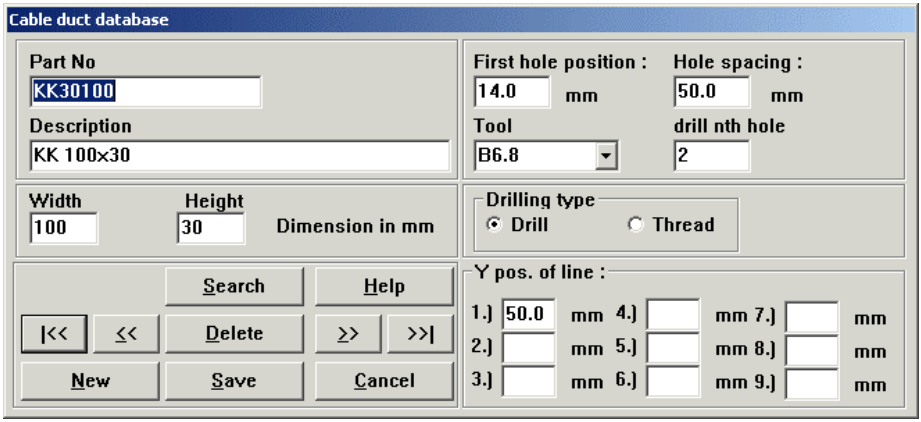
10.1.2 Duct master data (production)

Production Databases

Duct master data (production)

The production data for all ducts in the project are listed with all information on the component and hole/drilling spacing. The data can be modified and new data can be added.

The [>>] and [<<] buttons go to the next or the previous record.



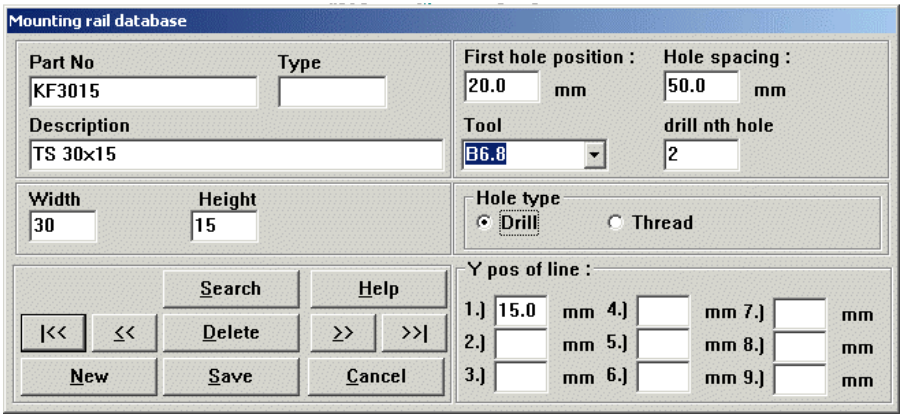
10.1.3 Mounting rail master data (production)

Production Databases

Mounting rail master data (production)

The production data for all rails in the project are listed with all information on the component and hole/drilling spacing. The data can be modified and new data can be added.

The [>>] and [<<] buttons go to the next or the previous record.



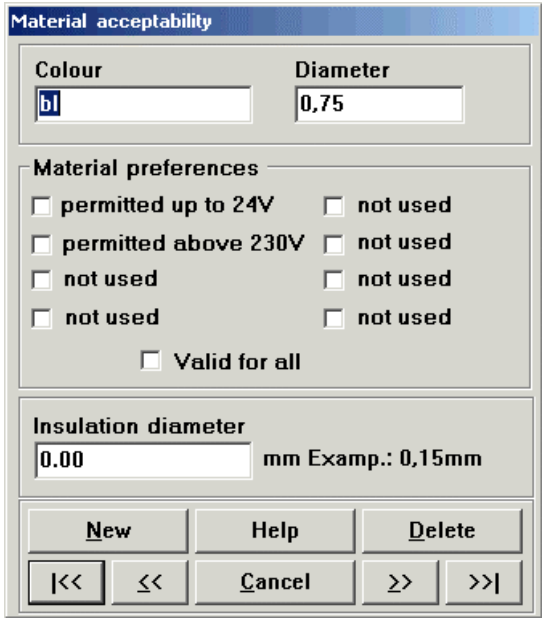
10.1.4 Material acceptability master data

Production Databases

Material acceptability master data

Lists the material data for wires. The data can be modified and new data can be added.

The [>>] and [<<] buttons go to the next or the previous record.



10.2 Importing master data

The production master data for devices, mounting rails and ducts are kept in the Triathlon database. If they are changed there, the data need to be reimported to apply the changes to objects in drawings. The import commands for the various types of component apply modified master data to objects in eCabinet drawings.

10.2.1 Importing device master data

Production Databases

Import device master data

Applies modified master data on specific devices to the corresponding objects in a drawing.

10.2.2 Importing duct master data

Production Databases

Import duct master data

Applies modified master data on ducts to the corresponding objects in a drawing.

10.2.3 Importing mounting rail master data

Production Databases

Import mounting rail master data

Applies modified master data on specific mounting rails to the corresponding objects in a drawing.

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