

SEE Electrical 3D Panel News

SEE Electrical 3D Panel V1R4 Service Pack 2

Table of Contents

SEE Electrical 3D Panel News	1
Table of Contents	2
I. Changes and Enhancements Made in SEE Electrical 3D Panel V1R4 SP2	3
I.A. New Features	3
I.B. Enhancements.....	4
II. Changes and Enhancements Made in SEE Electrical 3D Panel V1R4 SP1	6
II.A. New Features	6
II.B. Enhancements.....	7
III. Changes and enhancements made in SEE Electrical 3D Panel V1R3 SP1	9
III.A. New Features.....	9
III.B. Enhancements	12
IV. Changes and enhancements made in SEE Electrical 3D Panel V1R2	16
IV.A. New Features.....	16

I. Changes and Enhancements Made in SEE Electrical 3D Panel V1R4 SP2

I.A. New Features

New snap functionality: **Object**

You can activate the **Object** snap functionality when moving or inserting components to see how they align with other objects in the 3D drawing. The red dotted line shows they align with the outer edges of another object, while the green dotted line shows they are aligned with an inner section of the object.

New snap functionality: **Plane**

When the **Plane** snap functionality is active, components are automatically rotated to match the active plane.

Clipping view

You can now define a clipping view through the **Define** button in the **Main** tab. The clipping view makes it easier to look at components and their parts that could otherwise be concealed by other components. The **Show** button allows you to switch between the clipping view and the standard view.

Export wires to CSV file

It is now possible to export wires information to a CSV file, which can be used with Microsoft Excel to present the information in a table structured format.

Hide route paths

The new **Hide Route Path** button in the **Cabinet** tab toggles the visibility of route paths. The route paths are still active, but they are hidden from view. Click the button again to show them.

Lock route paths

You can now lock route paths you have selected for wires to avoid re-routing them when re-routing all wires. To lock route paths, select a wire and right-click it to open the pop-up menu. Then go to **Wires** > **Change route path**. Select one of the available paths and tick **Lock route path** to lock it.

Move with base point

You can now determine a base point for a component when you move it for more precision. The **Move with Base Point** command is accessible via the pop-up menu. Distances are calculated to the base point of the component.

I.B. Enhancements

Possibility to show boundary of components in drill holes charts

You can now show the boundaries of components when generating drill holes charts. To access the option, select the cabinet and right-click it to open the pop-up menu. Click the **Generate Drill Holes Charts** command to open the **Drill holes charts preview** window and then click the **Settings** button.

Improved fonts and position of component attributes

The font and position of attributes texts have been improved to make sure they are clearly visible at all times.

Drawing rails and channels has been simplified

To draw a rail or a channel just select their dimensions and colour from the respective button and click in the drawing window to place the rail in its desired location. To increase the length of the rail or channel simply click on one of the green rectangles at both ends of the rail or channel and move it in the desired direction until you reach the right length. Alternatively, after clicking the green rectangle, you can press the space bar key to type in the length of the rail or channel.

Draw rails and channels now remember your last selection

The **Draw Rail** and **Draw Channel** buttons now remember your last selection.

Use the “b” key to cycle through pre-set reference points

Press the “b” key when inserting or moving components to cycle through pre-set reference points at the four corners of the component and its centre. Reference points can help you place objects with more precision.

Improved **Property** card

The **Property** card has been improved and tidied to include only relevant information. Some of the options from the **Property** card of wires have been moved to the **Wire Settings** dialogue, which is called by clicking  in the bottom right corner of the **Wire** panel in the **Cabinet** tab.

Improved visual representation of overloaded channels

Activating the **Check Overloaded** functionality in the **Cabinet** tab now highlights the overloaded channels. You can also use the **Information** button to get more information about the channels.

Route only through channels and route paths

This option has been added to the **Wire Settings** dialogue, which is called by clicking  in the bottom right corner of the **Wire** panel in the **Cabinet** tab. When the option is ticked, wires are only routed if channels or route paths are available.

II. Changes and Enhancements Made in SEE Electrical 3D Panel V1R4 SP1

II.A. New Features

New options in the **Component Generation Wizard**

When creating a component through the **Component Generation Wizard** and opting to define it as "mounted on rail", you are enabled to specify offset distances on the Z and Y axis of the rail. If the created component is to be "mounted on back plate", you can choose whether drill holes are to be automatically generated

Possibility to update and download libraries via external servers

For this purpose, two new pop-commands are available in the **Symbols** pane: **Check update** and **Check update for all libraries**.

- ✓ **Check update** allows you to update a particular library, selected in the pane.
- ✓ **Check update for all libraries** allows you to update several or all existing libraries, as well as to download new ones.

Possibility to lock/unlock libraries

To make a library in the **Symbols** pane inaccessible for modifications, a new pop-up command: **Lock** was introduced. Respectively, for locked libraries the command **Unlock** is available. Respectively, you can unlock previously locked libraries.

Possibility to delete and rename 3D drawing templates

Introduced via the new pop-up commands: **Rename** and **Delete**, which are available in the dialogue for 3D drawing template selection.

New component types

For specifying the type of a component, during component creation, you have the following new types at your disposal: *Connector*, *PLC Main*, *Relay Coil*, *Terminal Stop* and *Terminal Separator*.

Ability to import 3D files by "drag & drop"

You can now import *.step*, *.dwg* and *.iges* files by dragging them from your file explorer into the drawing window of the *SEE 3D Panel*

Possibility to choose a solid or mesh graphical representation for a 3D object to draw
Introduced via the **Create Mesh Entity** and **Create Solid Entity** options available in the new **Settings** dialogue, which is called by clicking  on the **3D Objects** panel of the **Drawing** tab.

II.B. Enhancements

Possibility to assign work instructions to drill contours

You can now select predefined work instructions or enter custom values for them in the **"Description"** fields of the drill contours **Property Cards**.

Multi-line descriptions from SEE Electrical fit on one line in 3D Panel

Descriptions that take up multiple lines in a *SEE Electrical* diagram are now positioned on a single line in *SEE 3D Panel*.

Double-clicking an extremity in the **Nets** pane highlights it on the drawing

When you double-click on an extremity (tag) in the **Extremities** tab of the **Nets** pane, the corresponding component is highlighted on the 3D drawing. If the respective net is already routed, the connection point, through which the component is connected with the created wire, is indicated by a red dot.

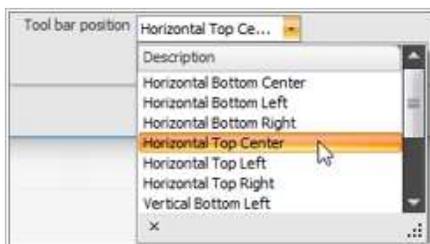
Possibility to modify Dimension texts

The **Text** section of the dimensions **Property Cards** enables you to modify at any time the size and font of the text parts of the already inserted Dimensions.

Possibility to reposition the 3D viewing tools on different sides of the drawing window

The main toolbar, containing the 3D viewing and navigation tools, which is by default available on top of the drawing window, can now be displayed on different positions within the window.

This is possible via a new setting introduced in the **General** tab of the **Global Settings** dialogue:



Aspect ratio preserved when resizing rails

When you are resizing a rail via the **Dynamic scale** command, instead of stretching out the symbol, the functionality resizes it appropriately by keeping its original aspect ratio.

Connection names and product tags shown on mouse hover

When you hover over a connection point, the respective connection name and product tag (of the component) are displayed next to the cursor.

Default routing directions for connections

When a connection symbol is being created, a default direction is suggested, according to the direction of the last connection created.

New end fitting settings

Three new columns have been added to the **End fitting mapping** tab of the **Wire Settings** dialogue: "**Color**", "**End-Fitting instructions**" and "**Stripping length**".

Description attributes hidden on the automatically created drill holes a

The description attributes of the automatically generated drill holes are now not displayed by default on the drawing window.

Possibility to edit block attributes before creating a block

An attribute inserted in the drawing window is possible to be modified before becoming part of a block via its **Property Card**.

Possibility to launch debugger on any object on the drawing window

The debugger, launched by pressing **Ctrl+D**, can now be used to view the data structure of any object you have selected.

III. Changes and enhancements made in SEE Electrical 3D Panel V1R3 SP1

III.A. New Features

Ability to define the shape of the drill contours

It is now possible to choose the shape of the drill contours. Execute the **Assembly > Create Drill Contour** command, and select the shape of the drill contour (circle/rectangle/polylines) from the available drop down menu.

Add possibility to delete a symbol library

You can delete user-defined elements or folders from the symbol library by right-clicking on them and selecting the **Delete** command from the context menu.

Auto Route Wires

The option "**Auto Route Wires**" which is found in the PropertyCard of the page is also available in the pages which are using Net routing template.

Balance the routing between connection points with same connection text

A new option available in **Wire Settings** called "Balance wires". When activated, if a device has two connections points with the same name, the wires will be balanced between these points.

(For example - first wire will go to first connection, second wire to second connection, third wire to first connection, fourth wire to second connection, etc.)

Drill hole chart support

When a drill hole view is generated, a corresponding table containing the Tag, XLoc, YLoc and Size is created. This feature is accessible when you right-click on a mounting plate and execute the **Generate Drill Holes Charts** pop-up command.

Hide connecting arrows

The command **Assembly > Connection > Show Connection** shows/hides component's connection arrows which are part of the block.

Maximum number of wires on a connection point

The attribute is found in the **Property Card/Connections** field of the selected element. The value represents the maximum number of wires that can be connected to the particular connection point.

"-1" means that the number is infinite.

New attribute for Connection point -> Connector type

The attribute is found in the "**Property Card/Connections**" field of the selected component. It allows you to select a connector type from the drop-down list.

New commands in the Commands pane

The **Update general Block attributes** command has been added. It allows you to select an attribute, define a value for it, and apply this modification to all components containing this attribute.

The **Update special connection informations** command has been added. It allows you to define the connection properties and apply this change to all such connections.

New option for visualization of the inserted elements

The new **Flat** view has been added in the **Main** tab. It allows you to see the geometric elements and their basic lines without any shadows or materials (in case such have been added).

New search tool available directly in the drawing window

The new search tool allows you to perform a search, based on the Product, Location or Function of the elements inserted in the drawing window. To activate the search tool press **Ctrl+F**.

New tab in the Wire Settings window

The **End fitting mapping** tab has been added in the **Wire Settings** window. This tab allows you to define the end-fitting configuration of the wires, as well as to add description for each wire. You are also allowed to import an XML file with already defined end-fitting settings or to export the currently defined settings.

New tool for creation of channels

The new **Draw channel** feature, available in the **Cabinet** tab, allows you to draw a channel and define its colour, width, and depth.

Option to filter wires only for selected component(s)

New command called **Filter wire for selected component** available in the context menu after right-clicking on a particular wire in the **Wires** list. After executing it, only the wires connected to the component remains in the list.

Overwrite with the selected entity

New context command called **Overwrite with selected** is available in the **Symbols** pane. It allows you to replace a library component with a selected one from the drawing window.

Possibility to add predefined connection points

The new **Add Predefined Connection** feature, located in the **Assembly** tab, allows you to insert an already defined connection point to a component. Six predefined connection points are available.

Possibility to change the icon size

It is now possible to change the size of the icons displayed in the **Symbol** pane. The available sizes are 16X16 pixels, 28X28 pixels and 32X32 pixels. The options are available in the context menu that appears on right-click on an empty area in the **Symbols** pane.

Possibility to check overloaded channels

The new **Check Overload** feature checks the capacity of each channel inserted in the drawing window. If a channel is overloaded, it is colored in red.

Possibility to check the available shortcuts and hotkeys

The new **Hotkeys and Shortcuts** feature has been added in the **Main** tab of *SEE Electrical 3D Panel*. This feature displays a list of all available shortcuts and hotkeys.

Possibility to define Work Instructions for the drill contours

The **Drill Holes Work Instructions** window, available when you click on the  icon at the bottom right corner of the **Drill Holes** section (**Assembly** tab), allows you to add or remove work instructions for the generated drill holes. You can also add detailed work instructions, by typing them directly into the "WorkInstruction.txt" file, located in the templates folder of *SEE Electrical 3D Panel*.

Possibility to edit blocks

The new **Edit Block** feature, located in the **Assembly** tab, allows you to modify the definition of the selected block. You can change all block references contained in the definition of the block. If the block is connected to a symbol, you can also change the symbol definition.

Possibility to import IGE symbols

The new **Import IGE Symbol** pop-up command (available in the **Symbols** pane) allows you to import symbols in "*.see3dc" format. The symbols are imported either in an existing symbol library or in new one, depending on the information contained in their header.

Possibility to insert equipment from the Equipment Catalog

It is now possible to insert equipment directly from the Equipment Catalog via the **Insert From Equipment Catalog** button that has been added in the **Components** pane.

Possibility to show/hide the tag text

The new **Hide Text Attrib.** feature, located in the **Assembly** tab, allows you to display/hide the tag text of the devices inserted in the drawing window.

Show connected channels

The **Cabinet > Channel Path > Show Connected Channels** command allows you to see which channels are connected to each other. To check the connectivity, select one channel and execute the command. The connected channels will be colored in yellow.

Validate component command

The **Assembly > Component > Validate Definition** command allows you to validate the selected component. The functionality checks different component parameters: **color** (taken from the parent assembly); **construction planes** and their **number**, defined **connection points**; names of the **connection points**; **snap points**; **drill holes** (for inserting the component to a rail).

Wire color mapping

From the **Wire Settings > Color mapping** tab the user can set the corresponding colors according to the colors' code names. There are also two additional buttons to Add/Remove color relations.

III.B. Enhancements

API command tree duplicate folders

The commands **Export wire to XML** and **Export logical links** are now in one command folder called "Export/XML" (**Commands** pane).

Change of the default cursor colour

The default cursor colour has been changed, so that it is easier to see the cursor on the grid.

Changes in the behavior of the Cut command

You are not allowed to use the **Cut** command or the **Ctrl+X** shortcut on symbols that have been loaded directly from the 2D diagram.

"Color by parent check" option

The "Color by parent check" from **Validate Definition** command is now working properly.

Context menu is available for all components

Previously there were problems with the availability of the context menu for some components like rails and channels. Also the context menu now is shown exactly where the mouse cursor is.

Enhancement in the wire Property Card

When a single wire is selected, its Property Card now displays not only the drawing properties of the selected wire, but also the actual wire properties (number, length, size, diameter, color, segregation code, etc.).

Eyeshot 9 update 1

All example projects are using EYSHOT 9 now.

Eyeshot 9 update 2

Not possible to de-serialize old drawings/models and symbols libraries.

Eyeshot 9 update 3

Resolved the crash when right-clicking on the Component explorer to get the context menu.

Improvement of the helper text visibility

A yellow background has been added to the helper text (that appears when using certain tools) to improve its visibility on the drawing window.

Link to the News document

The **New** button, located in the **About SEE Electrical 3D Panel** window, allows you to open the News file of *SEE Electrical 3D Panel*.

Location(+) and Function(=) inheritance

If the parent component have set location(+) and function(=) attributes the child components inherit these attributes.

Modifications of the icons in the Explorer pane

The icons in front of the devices in the **Explorer** pane change in accordance to whether the device is hidden or not.

Navigation commands in preview window

It is possible to zoom in/out, rotate, etc. in the preview window when choosing a component to insert from the Symbols pane.

New command for working plane

New command **Create from surface** available in the **Block Planes** window. It allows you to create a working plane from a symbol flat surface.

New connection point property

The new **Wire Marking** property has been added in the **Connections** sub-section of the Property Card. It allows you to choose between four wire marking options – To Up, To Down, To Left, To Right.

New fields in the Wires pane

The new **Comments** field allows you to insert comments for the respective wire. A new field shows whether the wire is present in the drawing window. If there is a red check mark in front of the wire, the wire is used (present in the drawing window).

Object snap distance global settings

On the **Global Settings > General** tab you can set the "*Snap distance*", which is measured in pixels. This distance determines when the cursor will "jump" to the snap point, previously enabled in the snap settings (EndPoint, MidPoint, etc.). The smaller the value the closer you have to move the cursor to detect the snap point.

Pop-up command modifications

The **Hide all except selection and its children** pop-up command has been removed from the pop-up menu, available at right-click in the drawing window. The command is available only in the **Explorer** pane context menu.

Possibility to add drill hole contour to an existing component

It is now possible to add a drill contour to an inserted component via the command **Create Drill Contour**.

Possibility to control the language with XML file

If a xml file with the name "Lagugage.xml" exists in the same location as See3D.v1.2.dll/exe then the 3D module will ignore the default settings and will use the xml file to load the language.

Possibility to define the text and table settings for the drill holes chart

The new **Drill Chart Text Setting** window, available when you click on the **Settings** button in the **Drill holes charts preview** window, allows you to define the dimensions and the location of the table, as well the size of the text.

Possibility to display report after wire routing process ends

The new "**Show route log after the route process is finished**" option (available in the **Wire Settings** window > **Settings** tab) allows you to choose whether to show a report when the routing process finishes.

Possibility to select an entire assembly

The command **Main > Select > Select Assembly** allows you to select at once every component that belongs to a single assembly.

Possibility to use the Select All and Invert Selection features in the Component load functions

When loading a product/location/function it is possible to select all products/locations/functions or to invert your selection via the corresponding buttons or via the available shortcuts – **Ctrl+A** and **Ctrl+I** respectively.

Preview of large files

Symbol files larger than 10MB are showing as red square in the Symbols preview window.

SEE Electrical 3D Panel marks the connection, while you are editing its parameters

A temporary red dot appears at the beginning the connection, when you are editing its parameters. This allows you to see exactly which connection you are editing.

Show connected channel

Show Connected Channels command is working properly now with all types of channels.

Small improvements in the Wire Settings window

New values have been added in the **Line Weight** dropdown list, allowing you to define more precisely the weight of the lines.

Zoom menu bar in the drill hole preview windows

It is now possible to modify the view in the **Steinhauer Drill Holes Preview** and **Drill holes charts preview** windows via the **Zoom**, **Pan** and **Zoom Fit** functionalities, located at the top of the respective window.

IV. Changes and enhancements made in SEE Electrical 3D Panel V1R2

For running *SEE Electrical 3D Panel V1R2*, *SEE Electrical Expert V4R2 SP2* (4.42.A) or higher, or *SEE Electrical V8R1 SP2* (8.1.2.18) is required.

IV.A. New Features

- ✓ Snap to points outside the active plane is allowed, but projected into the active one.
- ✓ Optimized SeeDbEquipments.
- ✓ Option to hide wires and texts in the 2D View.
- ✓ Make Event when a Component relation is changed – for example, when moved from one rail to another.
- ✓ Possibility to store the original symbol name on block definition.
- ✓ Version control when opening a project or loading a symbol database.
- ✓ Possibility to show pin markers, like in *SEE Electrical*, when navigating to a component.
- ✓ Clicking on a wire selects also the wire name in the wire control list.
- ✓ Feature tree: new setting for enabling / disabling feature tree loading.
- ✓ Possibility to perform search on the folder names in symbol libraries within the **Symbols** pane.
- ✓ New column in the **Components** pane to show the corresponding symbol names.
- ✓ New block property to define if components are scalable or not.
- ✓ New global settings for grid colours.
- ✓ Method to update all open drawings with the new settings.
- ✓ Possibility to control the colour of the vector drawings via an option in the print settings.
- ✓ Possibility to edit Net and Wire attributes.
- ✓ Possibility in Net mode to update the 3D drawing with the new wires added in the 2D diagram.
- ✓ New commands in the pop-up menu: **N-Copy X** and **N-Copy Y**.
- ✓ Possibility to re-route wires generated in Net mode.
- ✓ Option for displaying labels (Component name + Function + Location) in the 2D views.
- ✓ Group ID in SeeDbEquipment to enable loading of an entire terminal strip or other components that need to be loaded together.
- ✓ New structure to manage assemblies like terminal strips.
- ✓ Functionality for report creation.
- ✓ Equipment loaded dynamically when inserted, and deleted via events.
- ✓ New possibilities in the features explorer.
- ✓ Highlighted node in the feature tree corresponds to the current selection in the drawing.
- ✓ New attribute for Symmetric drill whole generation.