IGE-XAO

SEE Electrical Expert V4R3
What's new?

Table of Contents

1 - New Features

1.1   Cable Installation Panel

The Cable Installation Panel is available only in the Layout sheets. This command gives you the possibility to display the routing inside the cabinet, mark the wires as installed or uninstalled and generate a report of the installation progress. To launch the Cable Installation Panel execute the **Tools > Cable Installation Panel** command.

Be aware that you can also sort the wire list according to your personal preferences via the available pop-up commands.

1.2   Generating a Log File in Batch Mode Using Windows System Variables

It is now possible to generate a Log file in Batch Mode using Windows system variables. All depends from the value of the LOGFILE parameter.
- If the LOGFILE parameter value points to a certain user profile, a Log file will be generated in the Logs directory of the given user.
- If the LOGFILE parameter value points to a certain user shell folder, a Log file will be generated in the Logs directory, located within the specified user shell folder.

1.3   Printing Layout Sheet with Generated Routes

The **"Routes"** option now allows you to print the Layout sheet(s) together with the generated routes. To enable this feature go to the **File** menu, select the **Print** command, click the **Visibility** button and enable the **"Routes"** option.

Keep in mind that the *"Display Routing"* option, located in the **View** section (Layout sheet type) of the *Settings* window has to be enabled for the routes to be printed.

1.4   Pasting Elements from SEE Viewer to SEE Electrical Expert

Elements, such as texts, connections, symbols, etc., copied from *SEE Viewer* can be inserted into *SEE Electrical Expert* via the **Paste** command.

1.5   Switching Pins within Logical Connector

It is possible to replace a pin inserted in a logical connector by another pin, given that the necessary symbols are specified in the ***Connector and Pin Association for Insertion*** Method within the **Connectors** category of the ***Methods*** window. Select the pin (inserted in a logical connector), right-click and execute either **Change to Backshell** or **Change to pin**, depending on the change you want to make.

1.6   Assigning / Changing Validity

It is possible to assign or change the validity of texts, drawings, graphical locations, OPR symbols and symbols without connection points. To do so, select the respective entity, right-click and execute the **Change Validity…** command. Within the **Validity Builder** window, you can assign a new validity or edit the existing one.
See also the Variants Management Improvements section.

1.7   Exporting Sheets in JSON Format

This new feature allows conversion of project data in a language independent text format. It uses a library provided with the application to describe the exported entities. You can export Functional Diagram(s) (Connection Mode project) or Wiring Diagram(s) (Wire Mode project). To export in JSON Format execute the **File > Export JSON File(s)** command.

1.8   Exporting a Simulation File

This new component converts Electrical Diagram into set of logical equations, which describe the Electrical Diagram behavior. To export a Simulation file execute the **File > Export Simulation File** command. The file is exported in XML format.

1.9   Exporting Symbols or Illustrations in SEE Electrical Jigboard Format

This functionality converts symbols, defined in *SEE Electrical Expert* environment, into symbols that can be used in *SEE Electrical Jigboard* (symbols in XML format). To convert symbols in SEE Electrical Jigboard Format, execute the **Export > Format of SEE Electrical Jigboard** pop-up command (on folder or symbol's level in the **Symbol Explorer** window).

1.10 Undo/Redo in Multiuser Mode

In Multiuser mode the **Undo/Redo Explorer** now displays a list of all actions executed by all the users working on the same project (a project stored on SQL Repository on a given server location). Depending on the options specified in **SEE Electrical Expert Configuration** window, you are allowed to undo or redo an action performed by you or the other user(s). See also the New Options in "SEE Electrical Expert Configuration" Window section.

1.11 New Toolbar Icon

The new "Highlight Text and Drawings" icon () has been added in **Sheet Explorer** window, and in the **Visibility** toolbar of the main *SEE Electrical Expert* window.
-    The "Highlight Text and Drawings" icon, located in the **Sheet Explorer** window, displays only the text and the drawings in **Preview** pane of the selected sheet, while hiding everything else.

- The "Highlight Text and Drawings" icon, located in the **Visibility** toolbar, highlights the text and the drawings in the currently opened sheet, without hiding the remaining part of the sheet.

A new setting in the **Line and Color** method allows you to define the highlight colour for the texts and drawings.

1.12 New Controls Method

The **Controls** Method has been added in the **Project** section of the *Methods* window. The method allows management of the default values of the controls. You can define whether the control will be started when the project is closed, whether the control is mandatory, as well as the priority of the control.

1.13 New Tab in the Connector and Pin Association for Insertion Method

The *Equipment connectors attributes management* tab, located in **Connector and Pin Association for Insertion** Method (**Connectors** section of the *Methods* window), allows you to choose whether to synchronise the equipment attributes or the equipment attributes to be inherited by the connector.

1.14 New Harness Working Modes in the "Methods" Window

The **Harness** section in the *Methods* window now allows you to choose between two Harness working modes:
- **General mode**
- **Manufacturing mode**

Working in **Manufacturing mode** allows to define the exact position on the branch of the inserted accessories and protections, relative to the branches extremities – connective equipment or nodes. The insertion of harness accessories depends on the chosen working mode. The working mode affects the following symbols with Harness Design Behaviour and functions: Harness Accessory Adapter, Harness Accessory Clamp, Harness Accessory Label, and Harness Accessory.

The *CAD Properties* window of a harness protection also displays different attributes depending on the chosen working mode. The work in **Manufacturing mode** forbids the insertion of a harness protection on a branch without defined length.

1.15 New Options in the "Settings" Window

The new *"Execute 'Controls at close' even if project was not modified"* option, located in the **General** section, launches the controls, defined as "**Control at close**", when closing the project even if no modification in the project was made.

The new *"Black Box Selection"* option, located in the **Schematic** section of the **Coordinates & Cursor** category, allows you to define whether to select the Black Box by clicking anywhere within it, or by clicking only on its contours.

The new *"Tag status color"* area and its respective options, located in the **Schematic**, **Harness** and **Layout** sections of the **View** category, allow you to define different colors for the differently defined tags (Frozen tag, External tag, and Manual tag).

The new *"Recalculate order in case of tag modification"* option, located in the **Schematic/Synoptic/Harness/Terminal Strip** sections of the **Edit** category, allow choosing whether to keep the tag order or compute this order when the tag is modified.

1.16 Modifications in SEE Electrical Expert Configuration Tool

New Options in SEE Electrical Expert Configuration Tool

The new *"Turn on support for high-DPI resolutions"* option, located in the **Editor Appearance** section of the **General** category, allows you to enlarge all items within *SEE Electrical Expert.* For the option to take effect you also have to set a custom scaling level from the Control Panel of the Operating System.

The new *"Application language"* option, located in the **Editor Appearance** section of the **General** category, allows switching the initially installed language, in case your protection key allows multi-language use and the corresponding language package is installed.

The new *"Allow multiuser undo"* option, located in the **General Settings** section of the **Multi-User** category, allows you to define whether other users can use **Undo** command in the same project in Multiuser mode.

The new *"Length of operation history"* field, located in the **General Settings** section of the **Multi-User** category, allows you to define for how long the operation history in Multiuser mode will be stored.

New Sections in SEE Electrical Expert Configuration Tool

The new **Logging Settings** section, located in the **General** category, allows you to define the settings for the logging of the application behavior.

The new **Advanced** section, located in the **Security** category, allows you to define whether to start *SEE Electrical Expert* with Translation license.

1.17 New Tabs in the Electric Data Explorer

The new *Errors* tab lists all errors generated during the different electrical processes. You are allowed to delete an error, mark an error as corrected/not corrected (via a context menu on the respective error), or filter the displayed errors (list only warnings, or only corrected errors).

The new *Sheets* tab displays a tree view of the currently opened project and allows you to open the selected sheet, or to open the full *Sheet Explorer.*

1.18 New Fields in the Equipment Catalog

The following new fields have been added in the Equipment Catalog:**"Start Block"**, **"Intermediary Block"**, and **"End Block".** Those fields have assigned respectively the functions "PLC: START BLOCK", "PLC: INTERMEDIARY BLOCK" and "PLC: END BLOCK". They enable you to assign one or several blocks to the equipment code.

1.19 New Tagging Mode

The **"Semi-Automatic Tagging Without Uniqueness"** tagging mode option is now available at project/template creation via the **File > New Project** command **> Properties** button. In this tagging mode, it is possible to use duplicated components tags, because the tag of the components is composed from the hidden system attribute DEVICE_ID and the component Tag, which is visible.

1.20 New Connection Point Attribute

The new **Default Contact Order** attribute, located in **CAD Attributes** window of a connection point (in the **Symbol Editor**), allows you to define a default integer value for the **"Contact Order"** attribute of the wire extremity that will be connected to this connection point. This value will be visible in the Schematic sheet only if you insert the $CONTACT_ORDER metacommand.

1.21 New Attributes in "Wire Extremities" Window

The **"Extraction information"** and **"Soldered Extremity"** attributes have been added in the **Wire Extremities** window.

- The **"Extraction information"** attribute displays information about performed extraction, retrieved by the **$Extraction_Info** metacommand.
- The **"Soldered Extremity"** allows you to define whether the wire extremity to be soldered or not.

2 - Improvements to Existing Commands

2.1   Ability to Assign Keyboard Key for Symbol Rotation

You are allowed to rotate a symbol to 90, 180 or 270 degrees. However, it is now possible to define a keyboard shortcut for symbol rotation. To do so, execute the **Options > Customize** command **>** select the *"Other"* category, then select "Increase angle of inserting symbol" in the *Commands* pane and assign the shortcut and activate it.

2.2   Automatic Tagging on Harness Nodes

The harness nodes are now automatically tagged. The tag increments automatically on each node. Each separate harness is tagged separately. To make the tagging visible activate the Electrical Point Visibility ( ).

2.3   Automatic Zoom on Searched Entities

When you perform a search for text, attribute or equipment code (via the **Edit > Search > Text or Attribute** or **Edit > Search > Equipment Code** commands), the sheet where the element is located opens, the element is zoomed and yellow flag points at it.

2.4   AWG Standard Improvements

Activating the AWG standard from the **Standards** Method now enables you to:
- Import an external file that extends the list of gauges (**Import** button);
- Clear the imported file (**Clear** button);
- Export the currently active values (**Export** button).

2.5   Wire Mode Migration Process Improvements

The migration to Wire Mode process (launched when a normal project is saved as project of type "Wire Connectivity") checks the base project for errors and warnings and decides whether to convert the project.
- If errors are detected during the conversion, the conversion process is stopped and the project/template is not saved. Such error is the existence of forbidden elements (tags, numbers, etc.) in the base project.
- If only warnings are detected, the conversion process continues and the project/template is converted to type "Wire Connectivity". The conversion process can detect the following warning types:
    - Deletion of Unconnected Wires;
    - Wires Branched at Point;
    - Superposed Symbols.
  For each warning type, the conversion process takes actions according to the set rules, so that the project is converted and saved.

2.6   Variants Management Improvements

You are now allowed to edit the existing variant definitions xml file or to create a new one.

- If no variant definitions xml file has been loaded, clicking on the **Edit Variants** button in the *Variants Management* tab of the *Project Properties* opens an empty *Validity Builder* window and allows you to create new variant definitions.
- If a variant definitions xml file has been loaded, clicking on the **Edit Variants** button opens the *Validity Builder* window with the loaded variants definitions and allows you to edit them.

2.7   Printing Area Improvements

When trying to print part of a sheet (*"Selected Area"* radio button in *Print Manager* window), without having defined the printing area first, an information message reminding you about this appears. To proceed, define the printing area.

2.8   Improvement in the "CAD Properties Window"

The **Tag** of the selected element, which was previously displayed as title of the *CAD Properties* window, has been moved into a separate **Tag** field.

- The title of the window now displays *CAD Properties*, instead of the element's tag*.*
- The content of the **Tag** field can be copied, but cannot be modified.

2.9   Improvements in the "Control Process Summary" Window

The *Control Process Summary* report window, which appears after clicking the **Execute** button from the *Control Process* window, has been improved. The *Control Process Summary* report window now displays detailed information about all controls (Signal, Device, Terminal Strip and Connector, Cabling, Synoptic, Harness, and Illustration controls).

2.10 Improvements in the "Update of symbols, illustrations and connections inserted in sheets" Window

The *"Definitions to Update"* area has been moved from the *Process* tab to the *Application Domain* tab. A new option allowing you to update the connections inserted in the sheet has been added in the *"Elements to Update"* area.

The new *"Signals"* area (with its respective options), has been added into the *Application Domain* tab. It allows you to define which signal attributes to be updated.