CADvizor 2025

User's Guide

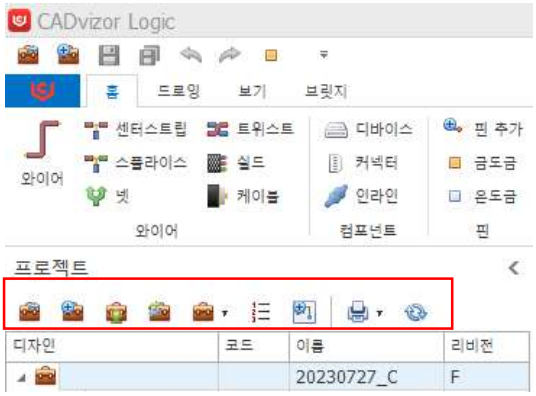
[Logic Module]

Table of Contents

Logic

CADvizor Logic is a powerful tool that supports electronic circuit design and simulation, enabling efficient performance of complex circuit design and verification. Provides a variety of part libraries and tools to shorten design time and ensure quality through design rule checks. This guide describes the functionality and usage of CADvizor Logic's circuit design.

Project



The first step in the drawing design is to create a project and add a diagram. You can check the related menus on the Project tab and have the following functions.

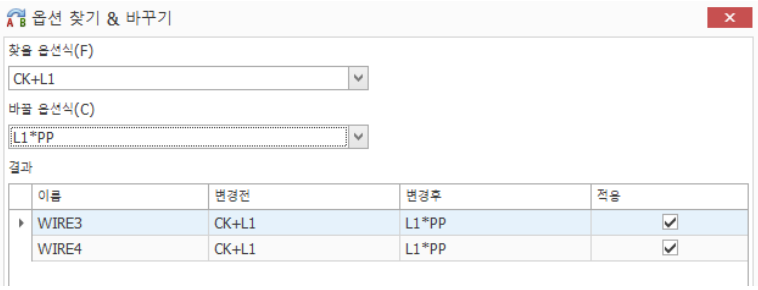
Open a project: Import the projects stored in the Database, display a list, and select the project to import to the Project tab workspace.

Create Project: To create a new project as a function to create it, create it through the Create Project window.

Importing a project: Importing an extension (.cpf) file as a function to share and enter projects from other companies.

Project Other Features: Project Other features provide the following.

- Close
  Close the project in the project list.
- Delete Deletes the created project
  .
- Change the existing settings for the editing
  project. You can re-establish names, revisions, template settings, and user privileges.
- Copy
  Copy Copies an existing created project.
- Manage revisions of revision
  projects. The revision is sequentially (A, B, C) (1, 2, 3) and design change history management can be performed efficiently.
- If the cleanup
  operation does not end normally or the network environment is unstable, the project will not open normally, which can be normalized through the corresponding "clean up" function.
- You can export
  an export project in .jcpf (JSON format) or .cpf (binary) format. It will be used to bring up projects in the future.
- Imports a project of a file saved as an import
  cpf.
- Find and replace options You can change



  the whole thing by finding options for the option expression specified in the circuit object in the project drawing. Specifying the option expression to be found will allow you to browse all applicable circuit objects and change and check them with new option expressions.

Add diagram:

Create through the Create diagram window as a function to create a new diagram for the selected project.

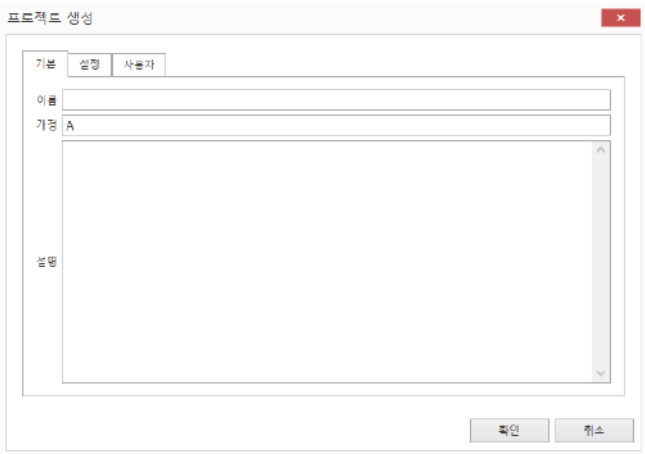
Print:

Prints all diagrams included in the selected project.

Create a project

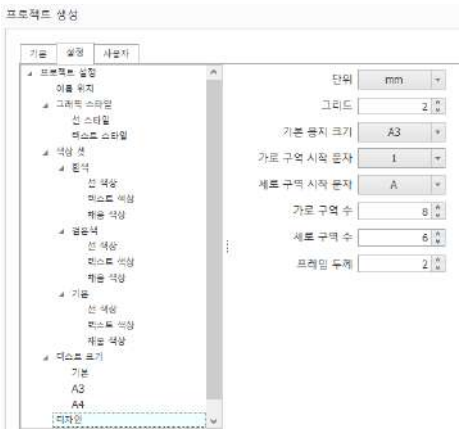
Create a project. For project creation, you can specify default settings for the project by default, settings, and user settings.

Main tab



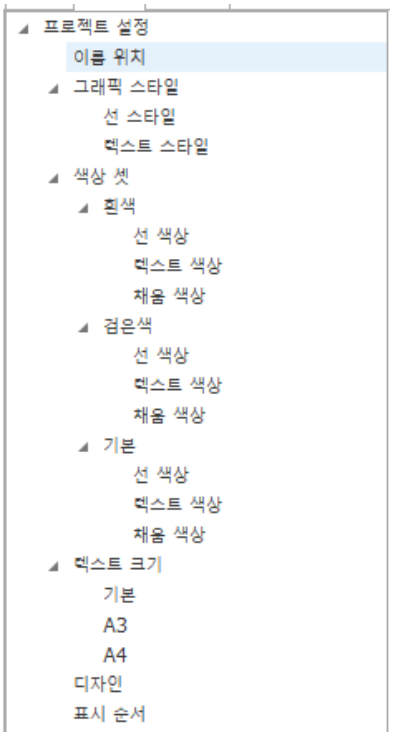
When you create a project, on the Default tab, enter **the** name of the project. **"Revision**" allows you to systematically manage design changes during the project. Revisions allow you to compare changes to previous versions to prevent errors and maintain design quality. Also, if you need to record your project history or additional information, you can enter it in the **"Description**" column.

Settings tab



Settings allow you to specify initial **settings** that will be applied when using circuit objects and graphics-related features. When creating circuit objects, you can set overall drawing styles such as initial name location, text size, and color.

Project Settings



- **The name location** allows you to specify the location of the circuit object name that you want to create. You can specify up, down, and center, and a name is created at the location you specify for each circuit object.

- **In line style**, you can specify the style of the circuit objects that you want to create, such as wires, net, devices, plugs, and pins. Displayed in the drawing according to the

specified settings, such as dotted or solid lines, or line thickness increase or decrease.

- **Text styles** allow you to specify font styles for font circuit objects.

- **The color set** allows you to specify colors for each drawing color. Currently, CADvizor can specify the appearance of the drawing in three main, black, and white colors, and you can specify the color for each theme: dot color, selection color, and ghost color in the drawing. You can specify the initial color of circuit objects (devices, connectors, pins
, etc.) in line color, text color, and fill color.

- **The text size** allows you to specify the size of the text that is generated when a circuit object is created. It is reflected as much as the value of the specified font and is generated in the drawing.

- **The design** proceeds with the overall setting of the drawing. Sets the initial status of the drawing, such as spacing between grids, paper size, horizontal zone start character, vertical zone start character, etc

- Wire **Name** tab in the display order. This tab allows you to specify which property values you want to line up and are reflected in order as you create the wire. Front-back decorative characters separate the characters that precede and follow their properties. If you select HARCODE and add '['']' to the front and back decorations, it is expressed as WIRE1 [H] 0.5 R [A + B] AVSS [H] by adding the front and back decorations to the harness code 'H'.

■ The Off-page tab lets you set the way a phrase **is represented when** a shared **object (for example, WIRE2)** spans different diagrams.

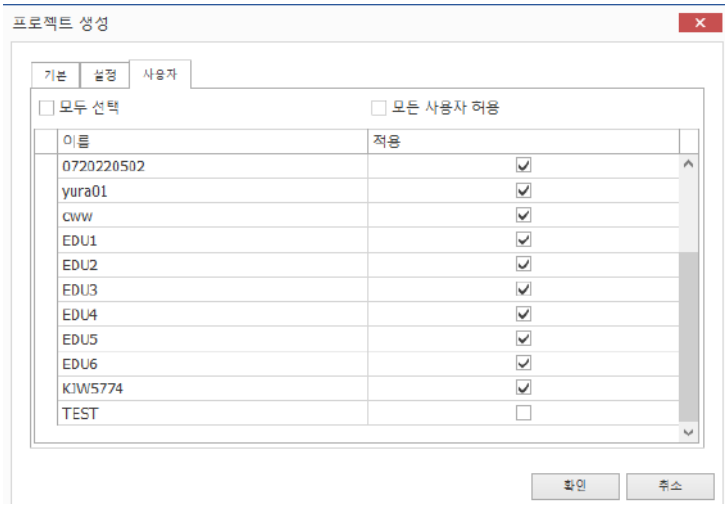
For example,

■ Page separator ':'

■ Object separator '/'

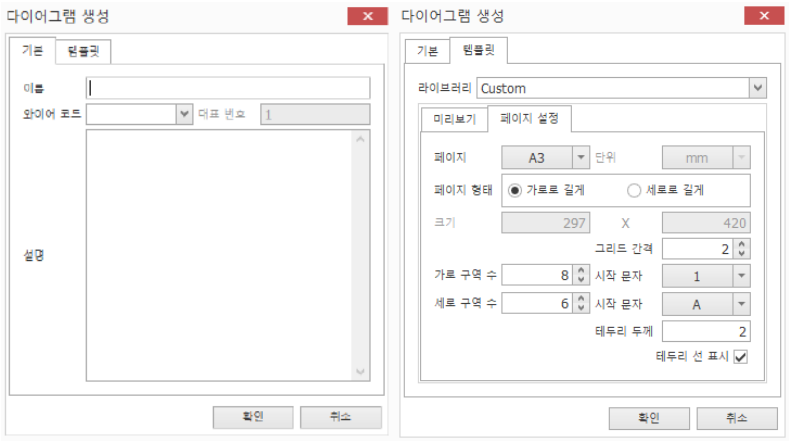
When set to , it is expressed as a0:P1/PIN1/WIRE2 and means "P1 connector in a0 diagram, WIRE2 generated by PIN1".

User tab



The **Users** tab allows you to grant users permission to access the project. If you check the application, you will be granted permission to edit the project, and if you release it, you can't edit it, you can only check it.

Add a diagram



Set the diagram for which you want to configure the project. Add diagrams are divided into the main **tab** and the templates **tab.**

Main tab

Enter the name of the diagram and the wire **code.** The wire code specifies a representative circuit name, **and** when a wire is generated, **the number increases sequentially from number** one. By default, the wire code is designated as WIRE, and the wire is generated as WIRE1 and WIRE2. If WR is entered into the wire code, the wire name is created based on the wire code with WR1, WR2.

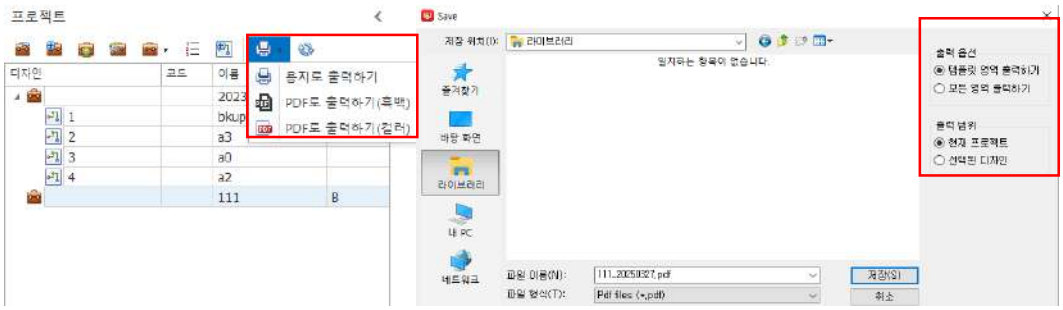
Templates tab

You can set the size and shape of the diagram. Specifies the section to be displayed in the

template.

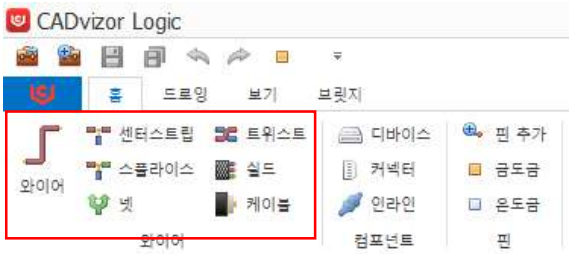
See Project-Project Creation-Project Setup-Design for more information.

PDF & Paper Output



You can export or print the results that you have worked on in a drawing as a PDF**.** PDF **output (black and white/color) or print** is possible using the button**, and all** activated **diagrams must be terminated** before use. You can select a template **area or all areas** at the output, and you can specify the current **project or the selected design** as the output range.

Wire



Wires are electrical connections that represent signal flows between symbols. One of the most fundamental elements of circuit design can have a variety of properties (name, color, cross-sectional area, material, etc.) rather than just a line. Functional elements are meaningful when used in simulation, DRC inspection, wire list generation, BOM linkage, etc. and logically connected to other circuit objects.

Create wire (short key: W)

There are many ways to create wires, and the way they are can vary depending on the situation.

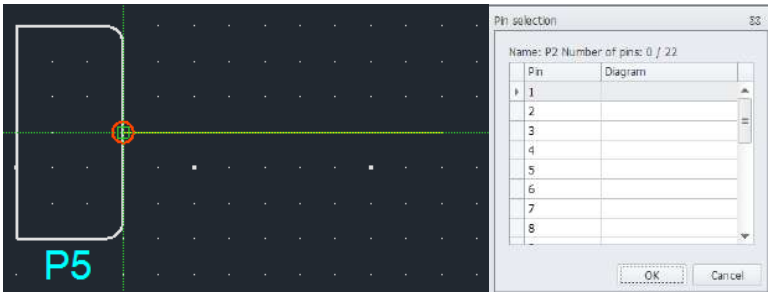
- **On connectors without library information**, a normal pin is created and a wire is also created.
- **For connectors with library information,** a separate 'pin mapping' window allows you to select the pin to connect to, and a wire is created upon selection.

In addition, you can use the wire button in the wire layout at the top or use **the 'W' shortcut** to create wires, providing the following options.

- Press **and** hold **the 'Shift' key** and create → wires align horizontally or vertically.
- Press **and** hold **the 'Ctrl' key** to create a wire automatically to avoid devices on the → path.

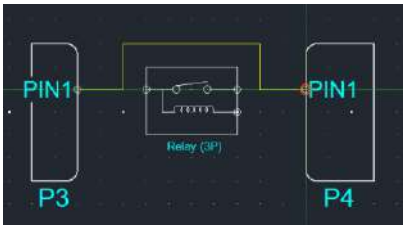
When creating a wire on the connector

After selecting a wire in the Top Home **tab** or using **the 'W' shortcut,** left-**click** on the diagram to specify a starting point, **then left-click again at the location you want** to create or press the "Enter" key to create a wire.



When you create a wire on a connector that does not have library information, the normal **pin and wire are generated at the same time.** By default, wires are generated in the form **of "WIRE1" and pins in the** form **of "Pin1" and automatically change according to** the existing wire **code name** in the diagram.When you create a wire on a connector with library information, a Pin **Selection window** appears, allowing you **to select** pins to connect to. The wire **is automatically generated** with the selected pin number.

Create AutoWire

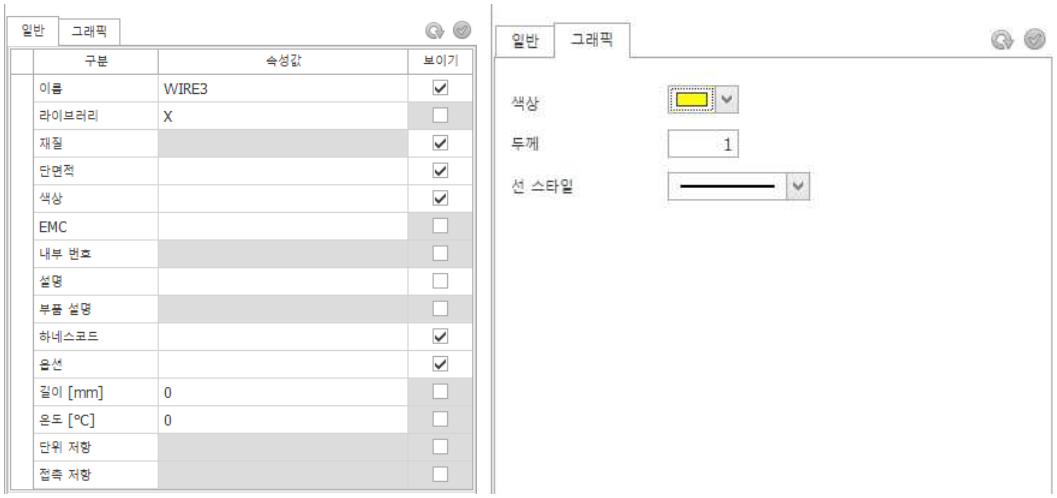


When **you press the 'Ctrl' key in the Wire creation state and move the mouse cursor, the** wire **automatically repositiones the devices** in the path**.**

Delete the wire

Select the wire you want to delete. You can press the "Del" key or find the "Delete" button on the mouse right-click menu tab and delete the selected wire from the drawing.

Enter Wire Properties

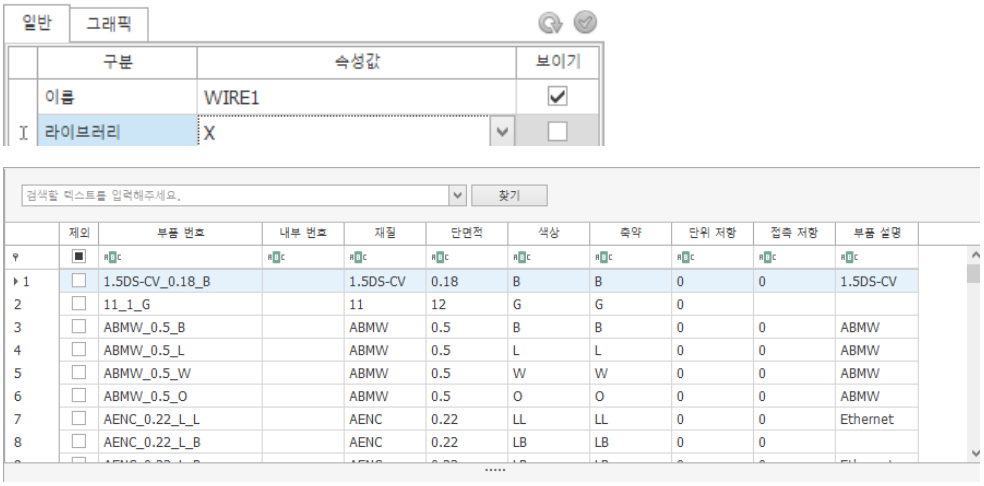


Wire properties consist of a general **tab** and a graphical **tab.** The way to check the properties window is to double-click or right-click a wire and check the properties window from the menu.

- **The General tab** displays data values based on registered library information. At initial creation, the wire is created in the form "Wire1" without library information by default, and when you specify the library, the stored data is reflected in the wire properties.
- You can find more information about library properties in the Library Guide.
- The **Graphics tab** lets you edit graphics-related information, such as the color, thickness, and line style of the wire.

If you have wire library information, when expressed in a drawing, the property values are displayed next to the name. In this case, you can specify what information you want to hide and show in the "View" column of the property through the checkbox.

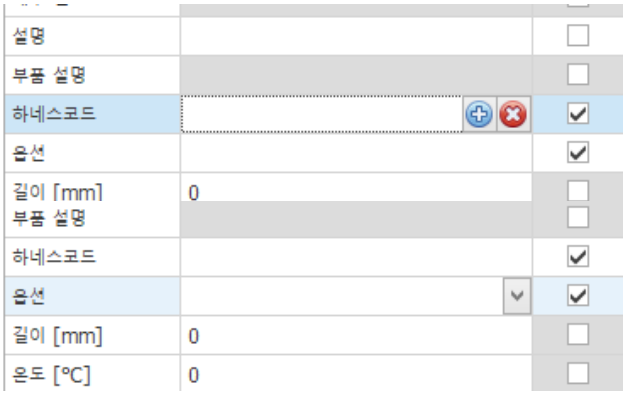
Enter the library



You can open a library window by clicking on the library compartment. This window allows you to link pre-registered library information from the library module to the wire and is available in the Properties pane.

If a library exists on the wire, the data information is displayed next to the name of the drawing wire, as shown in the image above. You can determine whether to display or not the data in the Properties pane check box. You can change the order of displaying wire property data information under Project-Project Creation – Project Setup-WireName.

Enter harness code & options



Based on **a** list of **pre-registered harness** code items **& options,** you **can enter** that information **into** the wire **properties.** This allows data related to harness codes and options to be reflected in wire properties

Option Filter

The optional filter function is a value entered into the wire and allows you to add and delete wires according to the manufacturing specifications when manufacturing the wiring harness. Basically, if there is no option value, it will go in unconditionally regardless of the production specification. The formula symbol is as follows.

- "+" Formula symbol: Same as logical condition "OR".
- "*" Formula symbol: Same as logical condition "AND".
- "^" Formula symbol: Same as logical condition "NOT".

"(", ") " Formula symbol: Determine the logical condition first of the formulas in parentheses.

Center strip & splice

Center strip

The center strip is a method of adding a new wire based on the central axis to the generated wire and combining it into a single path. By default, the center strip is automatically created when a new wire is created from the mainline wire, even if it is not selected from the Home tab.

- The name of the center strip follows the name of the wire that is being arranged, and the new wire is connected based on that center strip.
- On the Home tab, select a center strip, and specify the wire you want to create to create the center strip. You can continue to create ESC until you press or cancel it.

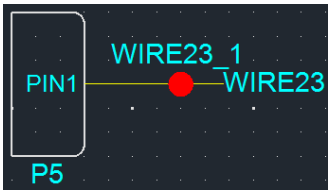
Splice

Splice is the concept of cutting the wiring from a wire and connecting the ends. You can create a splice on a wire by selecting it on the Home tab.

- Unlike the center strip, creating a splice distributes one wire into two wires.
- When you create a splice on "Wire1", both sides are divided into "Wire1 and Wire2" based on that splice.

How to create a center strip

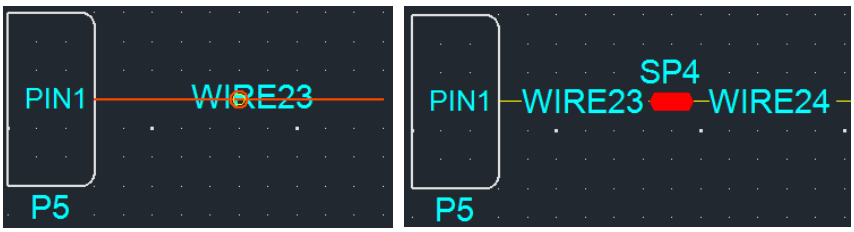
The method of **creating a center strip by creating a wire** is shown in the figure above. If you create a WIRE24 on top of a WIRE23 that is already created, a center strip called WIRE23_1 is created.



When **creating** a center **strip on the Home tab,** select **the center strip button in** the wire

layout at the top and click on W**IRE23** to create the center strip.

How to create a splice



On the Home tab, **under** Wire **Layout,** select **the splice button**. With the splice function enabled, selecting a specific point above the pre-**generated WIRE23 creates** a spl**ice** and forms a new wire as shown on the right.

Multi-core

Multicore is one of the important factors when designing drawings with multicircore cables that combine multiple strands of wires together. Multicore elements include twists, shields, cables, etc., and two or more wires can be selected and used.

- **Twist**:
  A method of twisting two or more wires together, usually used to minimize interference between wires, reduce signal interference, and increase the strength of wires.

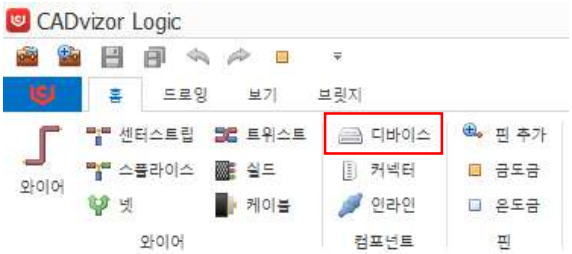
  When you connect the library to the twist in **Bundle Size:**Logic, the outside diameter of the connected twisted library is used for the Bundle Size calculation and if not connected, the Bundle Size is calculated by the outside diameter + twist weight of each of the twisted wires. In addition**, multicores except** for the twist require **a library connection**, and if they are not connected, MFG will not include them in the Bundle Size at all and will display a warning window.

- **Shield**: Metallic protective film to protect signal integrity
- **Cable**: Capability to safely transfer power from one point to another when multiple wires are wrapped in an insulator or protective film

Here's how to use multi-core.

- Select at least two applicable wires
- Hop Tab → Click to apply multi-core [Twist, Shield, Cable] button
- Multi-core is not possible with shared wire

Device

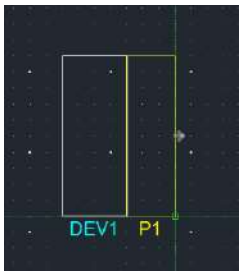


The device displays device parts that perform electrical and electronic functions. Add parts to the circuit diagram, connect the wiring, and link the layout to synchronize the location **information and the wiring path**. Devices include input**, output, and process**ing capabilities to support design tasks, allowing you to visualize and **output** designed models **in** physical **form.** The device must be pinned or pin-mapped.

Creating a Device

You can select the component category device button in the top menu or use the shortcut 'D' to create the device. After selection, the device can be placed by clicking the mouse "left click → left click" in the drawing.

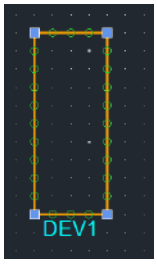
Create a connector on the device



The connector can **be placed on** the **device outline** by specifying the direction you want to create. This allows you to align connectors to suit your circuit configuration and design wiring connections efficiently.

How to create a device pin

Device Pin Generation

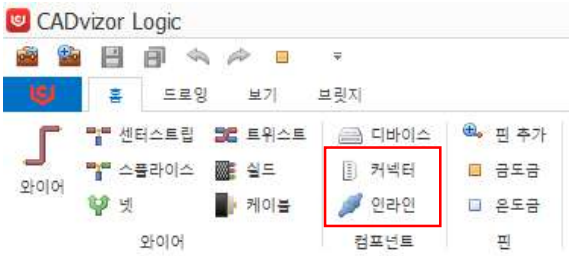


Select the device that you want to generate the pin, then click the shortcut 'P'. Then, you can create pins in any **groove** you want.

Automatically create device pins

Hold down 'Ctrl' and automatically create pins when you create the device.

Connectors



It is an element that connects or interfaces various parts or objects together, which is important in maintaining the structural, functional, and logical continuity of the design. Provides the electrical signal transmission path of the circuit, defines connections between parts, defines relationships between circuit objects in the design, and simulates behavior.

Creating Connectors

You can use  the connector button on the top component tab or use the shortcut 'C' to create the connector. The connector you create can be connected to the device or flexibly used as a "share" function.

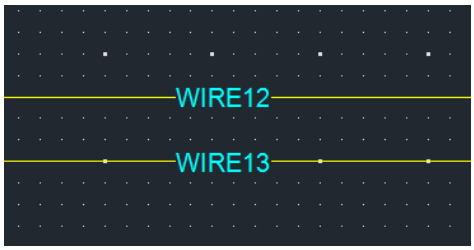
- **Left-click** and **then left-click** back to the location you want to create to create a connector.
- When initially created, the connector is created with a name of the default form, such as "P1", and the name increases as much as it is created.
- You **can add** a registered **library** later or use the Utility-Library Hotkey feature to create a connector **with the library present from** scratch**.**
- In addition, various **connector pin generation methods** are provided, allowing the user to flexibly select and use them according to the method he or she wants.

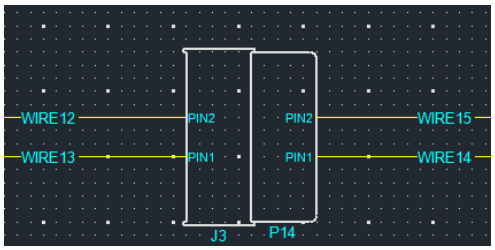
In-line connector

An in-line connector is a device that connects two or more wires or circuits, and supports fast and efficient installation and connection through straight connections. The plug is connected in the form of a pin protruding, and the jack has a hole to accommodate the pin. These combinations provide interconnectivity and ensure accurate contact between connectors and wires. This method maintains a reliable electrical connection and provides reliable performance in a variety of environments.

Create inline connector

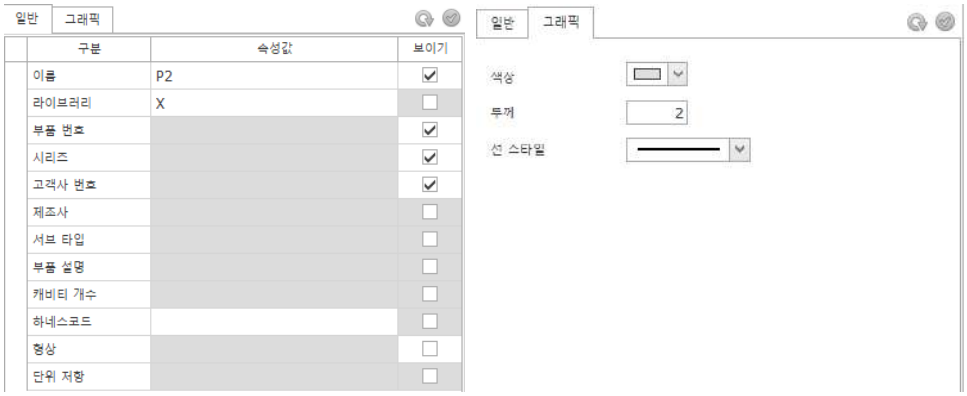
You can use  the buttons on the top components tab or use the shortcut 'I' to create an inline connector.





An in-line connector can be created on a wire already located in the drawing, and pins are automatically generated and placed in that location based on the contact point with the wire. This process automatically sets up pin numbers and electrical connections, saving design time. The pin arrangement of the connector is automatically adjusted according to the set specifications, and positioning is possible after placement. The electrical connection is automatically formed so that the circuit diagram can be easily tracked. This enables efficient and accurate design.

Enter connector properties



Connector properties can be accessed in two ways.

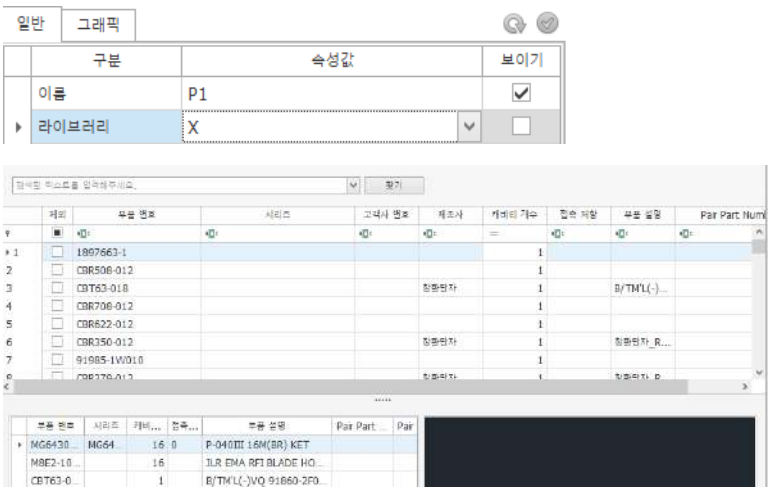
- Select Connector → Right-click → Properties
- Connectors Double Click

The Properties pane has the General tab and the Graphics tab.

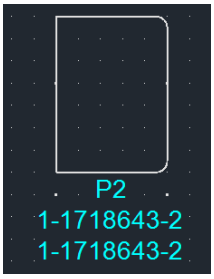
- **The General tab** provides connector connection information and library link functionality.Details of library property information can be found in the Library.
- On **the Graphics tab**, you can edit visual effects such as color, thickness, and style.

You can also check data values, change libraries, rename them, code harnesses in the Properties pane, and decide whether or not you want to show the property values in the drawing.

Enter the library

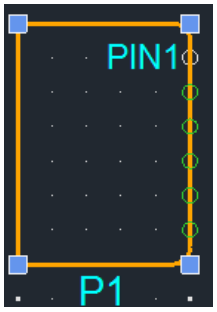


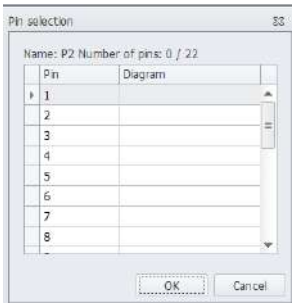
You can open a library window by clicking on the library compartment. This window allows you to map library information pre-registered in the library module to the connector and verify that the data has been entered in the Properties window.



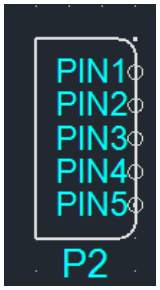
If a library exists in the connector, the data information is displayed along with the connector name in the drawing. This information can **be** selected **whether** or **not** to be displayed through **the check box in** the Properties **pane,** and can be hidden or displayed as needed. This allows you to adjust the readability of the drawing and selectively display only the desired information.

Create connector pins





- After selecting the connector that you want to generate the pin **using the single pin generation method**, press the shortcut 'P' to display the groove where the pin can be generated in ghost state.
- If the connector **has** a library, the "Pin Selection" window will appear, and you can specify which pins to generate.

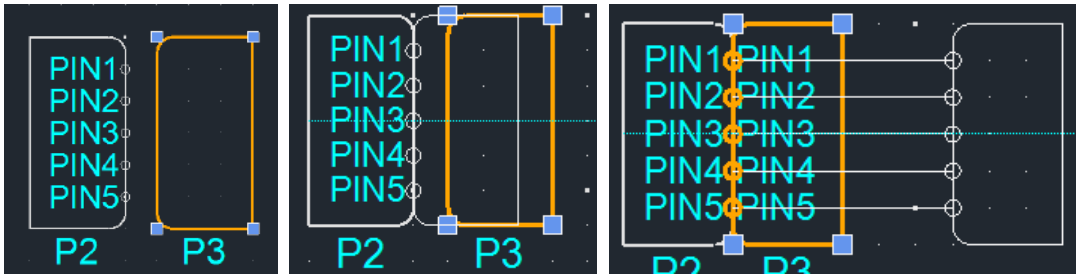


- **The multi-pin generation** method is a method of generating multiple pins simultaneously.

- When you create the connector, press and hold the "Ctrl" key to automatically create the maximum pin you can create.
- The existing method is to generate by specifying pins one by one, but this method can generate multiple pins at once.

Create connector pins & wires

This is a way that can be used when **there is a connector with a pin created** to help you easily **and quickly connect pins and wires between** the connectors. This feature is useful for quickly connecting pins and wires.



1. Create and select a new P3 connector when a P2 connector with a pin is present.
2. If the P3 connector is positioned according to the outline position of the P2 connector with the 'Shift' pressed, pins are created in a 1:1 format on the lines where the two connectors face off, resulting in a pin in P3 as well.
3. With pins present on both connectors, press 'Shift' again and move one connector's position to see that a new wire is created on the pin between the two connectors.

Create a shared circuit object

Share functionality is the ability to share circuit objects (devices, connectors, wires, etc.) to improve design **effic**iency**, consistency, and data management**.

How to switch share

- **(After selecting a circuit object) Right-click → Toggle share →**
- Alternatively, you can switch by **clicking** the Toggle Share button in the top edit layout.

Characteristics of shared objects

- Resource **sharing** is possible within the project.
- It can be created and managed in **the shared list on the left menu bar.**

- **You can link the source object and** access it from multiple diagrams.
- **Maintain data consistency** and use the same element without duplication.

Share transition

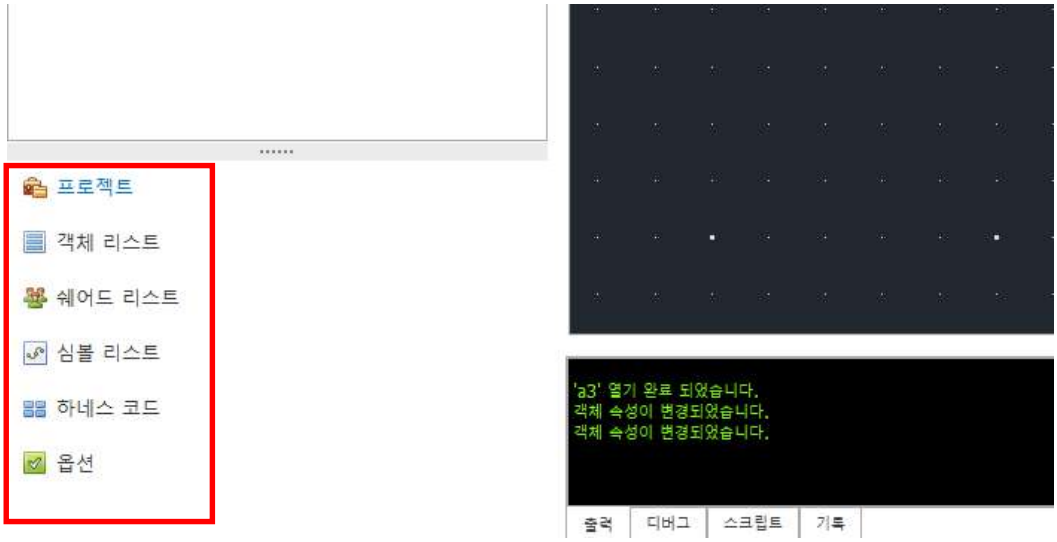
After selecting the circuit objects (wire, device, connector, multicore, etc.) you want to specify a share, the "**Share→ Transition**" option is available from the right-click menu.

When specified as shared, the circuit object is given a special mark, and the object is automatically updated in the shared **list** so that other diagrams can share the same resources.

Create share

Objects converted to shared can be used at any time within the project and can be accessed from other diagrams, making collaboration easier. It can also be freely imported and deployed within the same project, helping to increase the efficiency of circuit design. Share objects can be found in the Data List-Share List menu on the left and can be created in a drawing by double-clicking on the list.

Data list



The lower left corner of the main screen provides a menu that provides a quick, at-a-glance view of all projects, shared lists, symbol lists, and lists of objects in the diagram. Selecting a circuit object from this list allows you to immediately move to it or create a new one, and also provides the ability to manage harness codes and options.

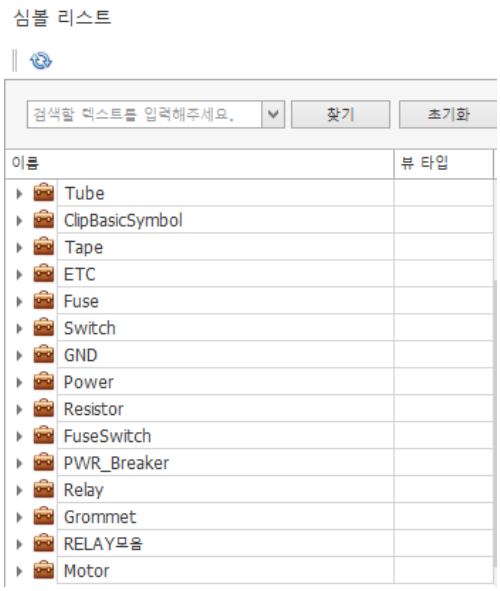
Object List

Displays all circuit objects present in the current drawing, and is a tab that allows you to view and manage a list of all objects, including devices, symbols, connectors, inline, and so on at a glance. Double-click on an object to zoom to that location for easy management.

Share list

A list that manages shared objects, allowing you to view and use the converted shared objects within the project. It is accessible from anywhere in the diagram, and the right-**click → Find** feature allows you to navigate the location of the object. You can also add shared objects to a drawing by double-**clicking** or right-**clicking → creation.**

a symbol list

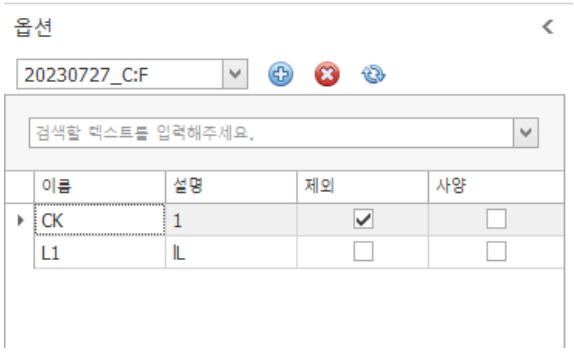


Pre-registered list of elements can be checked, including fuses, switches, ground, relays, etc. Double-click to create it in a drawing and preview the shape of the element in the View screen at the bottom of the list. You can create elements on the Symbol tab of the Library tab, which are organized by category in the corresponding symbol list. To add elements from the symbol list, it can be configured directly from the symbol, and it is updated to the list after the task.

harness cord

Most electrical equipment are designed by defining the part where the wiring harness is assembled as a code, and this is called the harness code. This tab allows you to register a harness code (for example, harness code: A, item: MAIN) to connect the harness code type to the connector or wire. Click **Renew** code **at** the top of the harness code tab to call the list, and you can manage the harness code through the Add, Delete, and Combo boxes. Click the button to update the harness code list and to add harness code items with the button.

options



If the wiring harness of the electrical equipment must be produced in a variety of specifications, the optional **definition** can be used to support multiple specifications of the product based on a single drawing.

Small-scale electrical equipment is produced in a single specification, so no separate option setting is required, but if large-scale production is required, the design is carried out through options. CADvizor Logic allows you to define options on wires, which **allows you to produce wiring harness products of various specifications in** a single **drawing.** Click the button to update the list of options and to add options with the button.

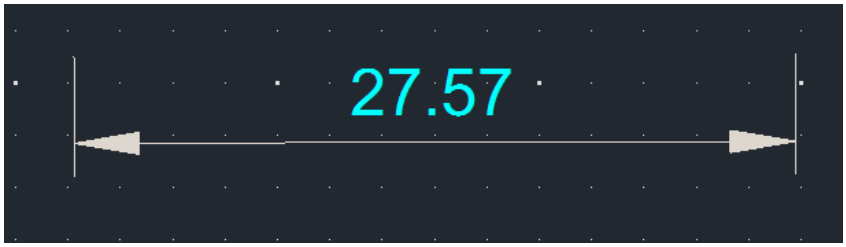
cycle

When designing a drawing, you can use a variety of tools to provide visual effects rather than just circuit objects, increasing the readability of the design and delivering information more clearly.

a line of instruction

Use indicators to clearly indicate or display specific areas in the drawing. You can create a directive by clicking in the Cycle Layout on the top tab, and it is available in arrow and text format by default.

a line of measurement



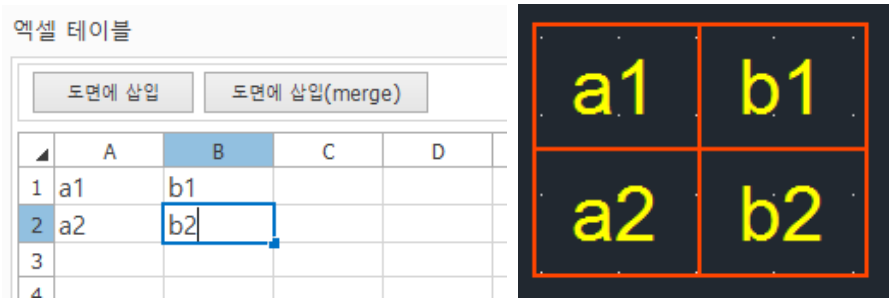
Dimension lines are measurement information that visually represents the length, angle, radius, diameter, etc. of an object and are essential elements in a design drawing. It is used to deliver the correct size during fabrication and inspection. You can create a dimension line by clicking in the Period layout on the top tab, and the components of the dimension line are as follows:

- Dimension Mainline: Keyline indicating measured values
- Dimension auxiliary line: an extension that connects the dimension line to the object
- Dimension values: Measurements displayed on the dimension line

Text

You can create a text box in any location in the drawing. You can click in the top tab cycle layout to create text and preset size, color, and font.

Insert Excel



This is an Excel sheet that can be created in the form of a table in the drawing. Click in the cycle layout of the top tab to import the Excel table window, and you can copy data values from an external Excel program or add data directly to the cell. When you are done, you can create what you have done in the drawing by clicking the "insert in drawing" button.

Edit

The ability to edit objects in a drawing. Use the Edit tab to make your drawing design more intuitive.

Disconnecting the connection

Function to disconnect the wiring elements. Select the elements you want to release and click in the top edit layout to release them. Below are some examples.

- If the Device is connected to a Connector
- Wire is connected to the Connector
- Net is connected inside Deviec

Disconnect the connection and switch to an independent circuit object.

Grip Point

The Add and Delete Grip Points feature can be applied to specific segments and is available by clicking in the top edit layout.

To add **additional** grip points, take the mouse cursor around the segment. In additional positions, a square-shaped grip point appears, and clicking on that position creates a grip point.

To delete **or** delete**,**
you can remove a square grip point by placing the mouse cursor around the segment end point and clicking it when it is activated.

Image

You can create an image by clicking a button in the edit layout at the top. Position the image with the first click, and resize with the second click. Supported image formats are JPG, BMP, and PNG.

Text original location

Restore the name position of the wiring elements to the system preferences. If you want to return the text to its original location after changing the position of the text during drawing design, click the button  in the top edit layout to automatically sort the name to its existing location.

Getting the end right

A function that aligns the ends of the wires to the same position. If the ends are uneven when creating or managing multiple wires, you can organize them all at once without individual adjustments. After selecting the wires to sort, click  the button in the top edit layout and select the end to be the reference, and all wires are adjusted to their end points.

Connector geometry

Even if the connector has a library and a shape exists, it is not displayed when it is created in the drawing by default. However, if you click on the  feature in the top edit layout to activate the feature, you can see the shape of the connector in the drawing.

a twist

The ability to invert selected shapes along horizontal or vertical axes. After you select the shape you want to invert, you can use the buttons  in the edit layout to select the axis, which reverses the shape based on the selected axis.

scale

The ability to shrink or enlarge the selected shape. After selecting the buttons in Shape and Edit Layout, you can adjust the scale by clicking the reference point and moving the mouse, or you can adjust it to the desired size by entering a value directly into the text box  . However, the text is excluded from the scale target.

Spinning

Click the button in the edit layout  to rotate the shape in 90-degree increments.

Customization

The function of aligning the selected features. After selecting all the features to sort, click the button in the edit layout  to select the alignment direction from left, right, top, and bottom. When you select a direction, the shapes are aligned based on that direction.

Sorting

This feature is available by clicking a button in the edit layout.  The ability to sort for selected features, if the customization is to align the baseline of the features, then the alignment is the task of sending the features forward or backward. This feature consists of a total of four options:
forward, forward, forward**,** backward, backward**.**

Group

The ability to group or release selected shapes. This feature only supports raw figures and does not support wiring elements. Select the features you want to apply and group them  by buttons in the edit layout.

Utilities

It is located at the top right of the home tab with a layout that brings together features to help design drawings.

Find it

Search for wiring elements present in the drawing. You can write down what you want to find and set up where to find, options, and more to navigate. It can be used as  Ctrl+F or as a button.

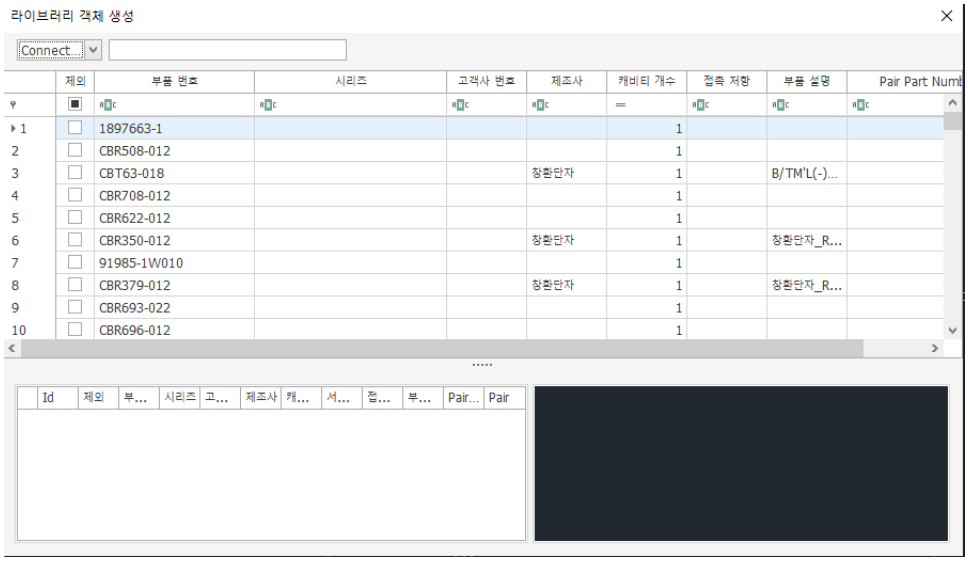
Print

Prints the drawing.

harness cord

This button is a harness code multi-input function. Specify the same harness code for all selected objects.

Library Hotkey

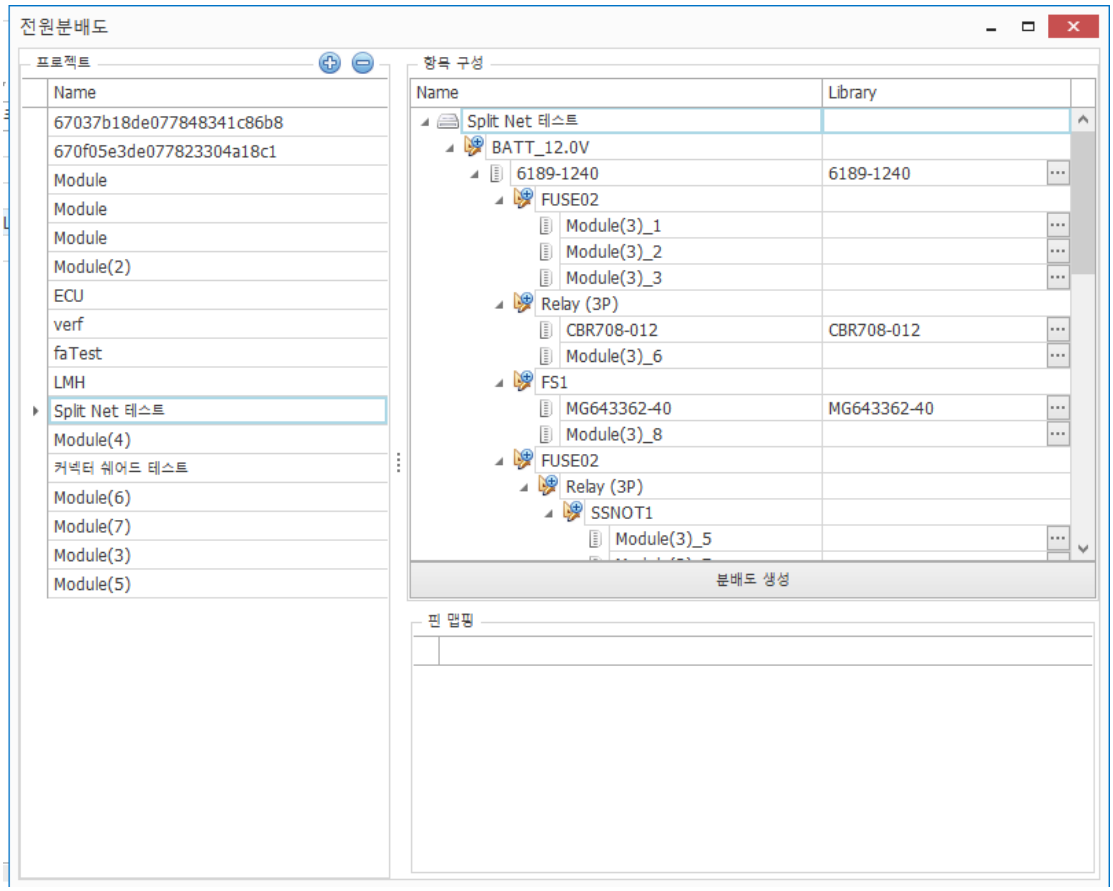


To open a window, click a button in the appropriate top utility layout. Previously, applying libraries to Wire and Connector required access to the created object through a property window, and if there were multiple objects, you had to open and close repeatedly. However, this feature makes it easier to apply libraries to multiple objects by enabling separate library windows.

The form remains open until it is closed, and you can use it as follows: Select the object you want to apply the library to, and repeat the process of specifying the library. You can select Connector and Wire through the combo box at the top left, **and if** no object **is selected**, you can also create a connector or wire with that library directly in the drawing.

Power distribution chart

The power distribution chart is a visual representation of how power is distributed to each component or device within an electrical circuit or electrical system. It is a design showing the flow of electricity from the power supply to each element. Instead of drawing the circuit design of the power supply directly, if you pre-configure it in the form of a tree as shown in the picture below and specify the connection relationship between each connector and the element, it is automatically generated in the drawing.



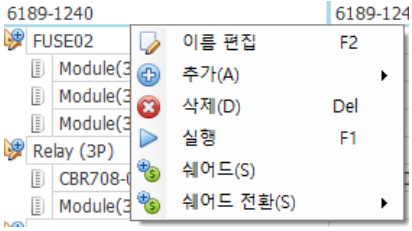
Create a project

You can create, remove, and edit projects using the button at the top or right-click.

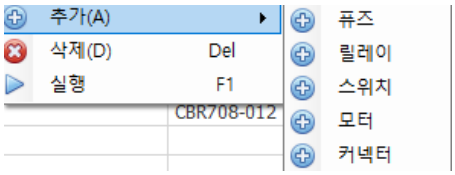
Organize items

The power distribution of the project is also configured. It can be configured in a tree format, with the project name at the top, followed by power, connector, and device. You can connect the library by clicking the button in the Connector Library column. The basic configuration is set to have connectors under the power and elements under the connectors, and right-click to activate the tab that allows you to configure the tree.

■ For the Edit Name connector, you
can edit the name separately.



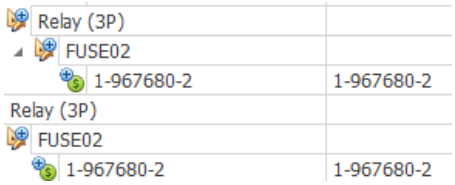
■ You can configure
additional power distribution charts.
When you select the connector or element you want to configure, call the list registered in the library to configure the tree.



■ Delete tree components are deleted
. If you delete an element of the list, the sub-items are also removed, and the pin list is initialized.

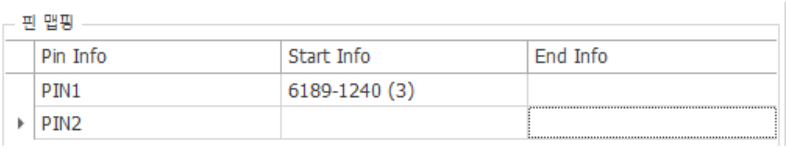
■ Create a power distribution diagram in
the drawing that you have configured.

■ Share. Specifies the selected connector.
The shared connector can be applied to other connectors. Through this function, it can be reused in multiple locations or configured by connecting with common data.

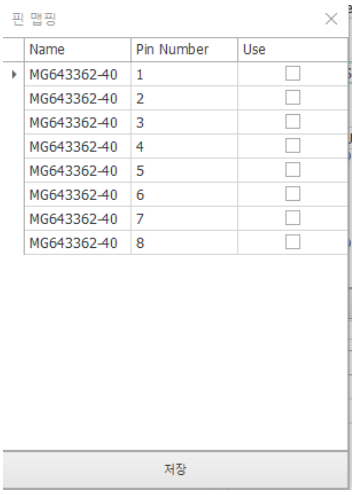


■ Share Conversion Share Conversion function
ality allows you to convert to a share connector. When you specify a share, you will share all the information (pin mapping).

pin mapping

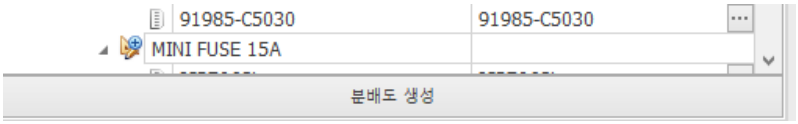


Select a connector or element configured in the power distribution to view the pin mapping table in the tab below. PinInfo is the information of the Pin currently held by the selected element and StartInfo is the pin information associated with the element at the parent node. The picture above illustrates that pin (3) of the 6189-1240 connector is connected to PIN1 of the element. End Info represents the end point of the pin. For example, if you select a blank space in the End Info column of PIN2 above, the following selection window appears.



You can check all the pins on the MG643362-40 connector and check the check box in the Use column to complete the connection.

Create Distribution Chart



When the power distribution chart configuration is completed, the circuit is automatically generated in the drawing through the Create Distribution Chart button.

Windows

List horizontally

This feature is available when more than one diagram is active. When you click the button in the window layout, it is activated, and the original drawing tab shows that only one screen was visible at a time, divided horizontally and divided into the screen by the number of open diagrams. This feature allows you to view and work with multiple diagrams simultaneously.

List vertically

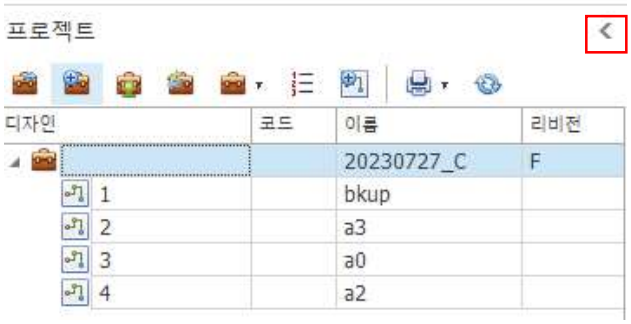
When clicking the button in the window layout , it is the same as the horizontal listing function above, but this time, it is divided vertically.

Back to tab group

This feature can be utilized after using the horizontal or vertical listing function. Restore the diagram split by the buttons in the window layout back to the top tab.

Panel

Project

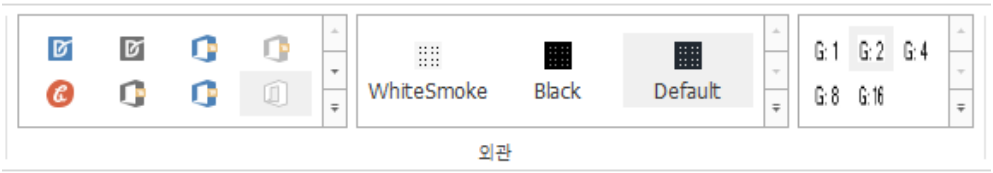


The ability to hide or re-display the left project tab. By default, the Project tab is always visible, but if you want to hide the tab, you can click it at the top of the project tab or click a button in the top layout panel to the left side of the Hide**&Show.** To re-display the Hidden Projects tab, you can reactivate it by clicking on the button of the same function.

Output window

This function determines whether or not to activate the console window at the bottom. It can be determined by the button, and the function is displayed in the corresponding console window in text format on what works within the project. It usually displays information about system changes, such as activating diagrams or storing them.

appearance



Themes

You can specify a background theme for CADvizor. In the tab on the left, in addition to the commonly seen White & Black themes, there are various types of themes, such as Office and VS, which change the color of the window and font depending on the type you choose. WhiteSmoke, Black, and

Default on the right change the theme of the drawing to the selected color.

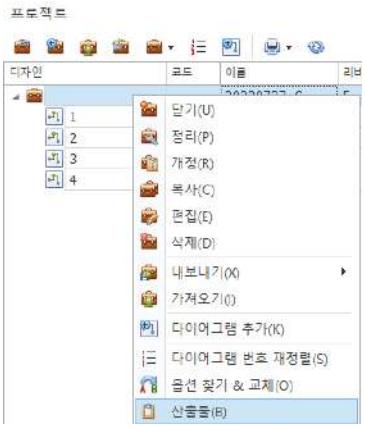
Grid

Grid acts as a baseline by which objects can be aligned and placed in a drawing. This Grid allows the precise placement of circuit objects. The corresponding tab lets you adjust the alignment criteria of the drawing by setting the Grid interval.

By default, the Grid interval is set to 2, and the user can select values such as 4, 8, 16, and so on. This interval determines how far apart objects are placed in the drawing, increasing the accuracy of the design and reducing interference between objects.

After setting the Grid interval, it is generated to meet this criterion when creating circuit objects, allowing for more precise and consistent placement.

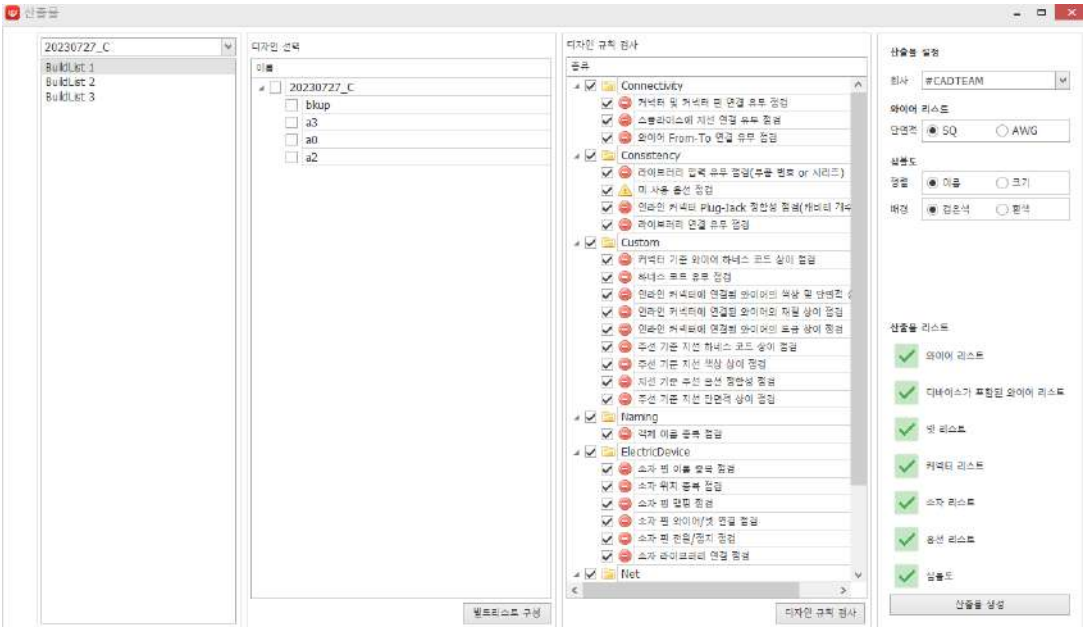
DRC & Output



On the Project tab, you can access "Right-click → Output" to create output based on the information you have designed. This feature allows you to design drawings within a project, then perform a Design Rule Check (DRC) check based on the design information and view the work history.

Output generation is an important part of the design process, and after design is completed, errors are found and corrected through various verification operations. DRC checks are steps to ensure that the design conforms to the rules, which can ensure the quality of the final design. Work history also allows you to determine what modifications or tasks have occurred, making project management easier.

Build List Configuration & DRC



Design rule checking (DRC) is a very important process in circuit design. This process ensures the accuracy of the design and improves the quality of the drawing by detecting errors in advance.

1. Build list configuration:
   Before you can proceed with the design rule check, you must configure the build list. A build list is a bundle for outputting a circuit diagram of a specific specification for each item. This allows you to select the design to conduct the inspection. When you select a project to run a design rule check and activate the check box for that design, the build list is complete.

2. Design Rule Check Item:
   The DRC checks multiple items to identify design errors. Key checks include:

   - **Check circuit object connection**: Ensure that the designed circuit object is connected correctly.
   - **Check library presence**: Verify that the parts used are defined in the library.
   - **Check harness code**: Make sure harness code is applied correctly.
   - **Duplicate name search**: Check for objects with the same name.
   - **Check mapping**: Verify that the designed objects are mapped correctly.

3. Warning and error messages:
   If an alert or error is found during the DRC check, a message will be sent out. This message shows the location of the problematic drawing, and you can click the log to find and track it. This allows you to fix the problem quickly.

4. The importance of pre-checking:
   Design rule checking is a must-do task before you get the output. DRC can help ensure the quality of the final design by proactively detecting and correcting errors. This is a very important validation task for improving the quality of drawing design.

Therefore, early detection and correction of design errors through DRC is an important process to improve circuit design quality and ensure product reliability.

Types of DRC

Design rule checks are classified into six **main categ**ories, and you can specify whether each item is inspected or not through a checkbox.

If a warning or error occurs during the examination, you can view the log for the problem. This allows you to check and supplement what you missed during the design process **to create a more accurate and reliable drawing.**

Connectivity

Check connectors and connector pin connections:
Check that the pin generated in the connector is connected properly

Check for splice branch connections:

- Wires to be connected to the splice are missing.
- If the wires connected to the splice are inconsistent because they have different option definitions.
- If there is a discrepancy error in wire type such as thickness, material, current capacity, etc.

Check wire from-to connectivity:

- Error for unconnected wires (in levitation status)
- Option Conflict Error
-

Consistency

Check library input (part number or series):

- The registered part does not have a part number.
- The series information to which the part belongs is missing.

Check unused options:

Make sure that certain wires or parts are clearly used in which options.

In-line connector Plug-Jack consistency check (number of cavities):

- When the pin placement of Plug and Jack is set differently.
- When the number of cavities in Plug and Jack does not match.
- Option conflict error.
- When the series or manufacturer of Plug and Jack is different.

Check library connectivity:

If the circuit object in the drawing does not have a library.

Custom

Check connector reference wire harness cord differences:

- When the harness cords of wires connected to the same connector are different.
- When the same part but not consistent because they have different harness codes.
- Depending on the option, the harness code is defined differently, resulting in a collision.

Check for harness cords:

- If the harness cord is not entered in the wire, connector, or wiring element.

Check for color and cross sectional differences in wires connected to inline connectors:

- The cross-sectional area of the wire does not meet the current capacity of the connector or exceeds/believes the specification.
- The color and cross-sectional area do not match or do not match each other.
- If the color of the wires connected to the inline connector does not match.

Check for material degradation of wires connected to inline connectors:

- If the materials of wires connected to the inline are different for reasons such as electrical compatibility, mechanical consistency (durable, resistant, sagging), etc.

Check the plating phase of the wire connected to the inline connector:

- If the plating of the wire connected to the inline connector is different due to problems such as electrical performance degradation, corrosion occurrence, mechanical instability, etc.

Check mainline reference branch harness cord difference:

- The main line has a harness cord defined, but the branch line does not have a harness cord.
- If the harness cord of the main line and the branch line is different.
- If there is an error in the consistency of the harness cord at the splice or junction.
- If the harness code in the diagram is not automatically updated.

Check the color difference of the main line reference branch line:

- Whether there is a color mismatch between the main line and the branch line due to possible confusion, wiring errors, and difficulties in maintenance.
- Color consistency at splice or bifurcation point.
- If the main line has color but the branch line is not specified in color.

Check the consistency of branch line reference arrangement options:

- If the options of branch line and main line are different.
- If the main line has an option but the branch line does not have an option.
- When the option is changed, the relevant wires in the diagram are not reflected properly.

Check for differences in cross-sectional area of mainline reference branch lines:

- When the cross-sectional area of the branch line and the main line is not properly established.
- Cross-sectional mismatch at splice and bifurcation points.
- If you have a cross-sectional difference in excess of a particular current capacity.

Naming

Check duplicate object names:

- The circuit object's name is not unique and overlaps with another object.

ElectricDevice

Check duplicate element pin names:

- The same pin name exists within the same part.

Check the location of the element:

- When more than one element is placed in the same coordinate.
- The minimum gap between parts is not maintained.
- If electrical interference is likely to occur.

Check element pin mapping:

- The library data and the actual pin information are inconsistent.
- If the power and GND pins are incorrectly connected.

Check the element pin wire/net connection:

- If the pin is not connected to the wire.
- An incorrect signal line is connected to the pin of the element.

Check element pin power/ground:

- The power and ground pins are not connected to the circuit.
- No voltage is supplied to the VCC pin, or the GND pin is not grounded.

Check the connection to the element library:

- The output pin is set to the input pin.
- The signal pin is connected to a power source.

Net

Redundant check of four names:

- When used repeatedly in different circuits.

Check Net Drive Source:

- The drive source is not connected to the four.
- When connected to an unnecessary circuit or incorrect part.
- Multiple drive sources are connected to the same net at the same time.

Check the four and pin connection:

- If the pin is not connected to four.
- If you are connected to a wrong or unnecessary net.
- Physically connected, but power and signal four are wrong.

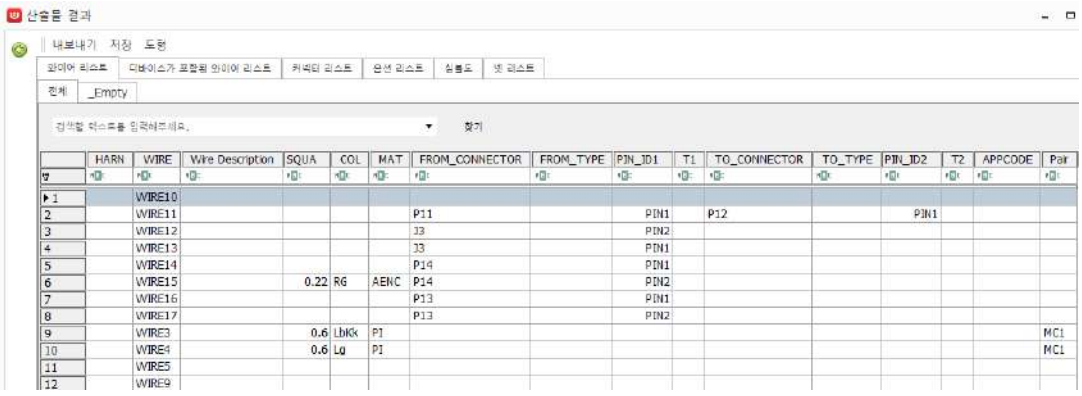
Check for net connectivity:

- Four is not connected to any pin
- It is connected to the wrong part or to the wrong signal & ground net.
- If there is an element that may cause an electrical collision.

4 Thickness check:

- When four thicknesses that do not meet the standard are used.

output



	HARN	WIRE	Wire Description	SQUA	COL	MAT	FROM_CONNECTOR	FROM_TYPE	PIN_ID1	T1	TO_CONNECTOR	TO_TYPE	PIN_ID2	T2	APPCODE	Par
1		WIRE10														
2		WIRE11					P11		PIN1		P12		PIN1			
3		WIRE12					J3		PIN2							
4		WIRE13					J3		PIN1							
5		WIRE14					P14		PIN1							
6		WIRE15		0.22	RG	AENC	P14		PIN2							
7		WIRE16					P13		PIN1							
8		WIRE17					P13		PIN2							
9		WIRE3		0.6	LbKk	PI										MC1
10		WIRE4		0.6	Lg	PI										MC1
11		WIRE5														
12		WIRE9														

If the DRC check has been completed normally, you can now proceed with the output output. The deliverables list contains a variety of reports and data files, which provide a variety of information about the designed circuits. The outputs are mainly available in Excel format, and some are stored in specific formats. The list of key deliverables is as follows:

1. Wire List:
   A list containing information about all designed wires, which allows you to check the connection and characteristics of the wires.
2. Wire List with Device—
   A list that contains information about the wires associated with the device, which allows you to track the wires associated with the device.

3. Netlist—A list that provides information about all net defined in the circuit. The net represents the electrical connection of the circuit, which allows you to check the status of the connected net.

4. Connector list:
   A list containing information about the designed connector, which allows you to view the connector's connection information and related data.

5. Element list:
   A list of all elements (parts) used, providing details for each element.

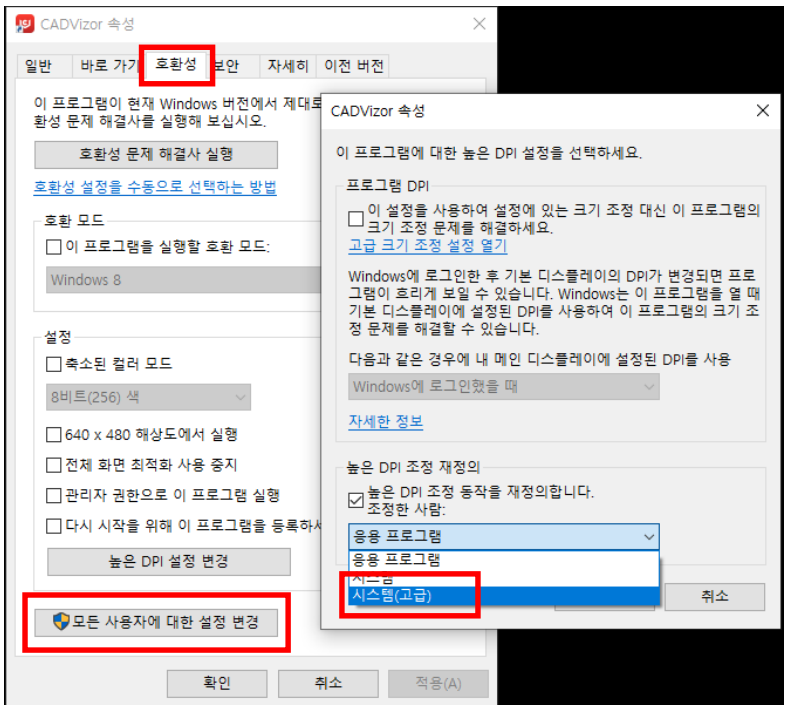
6. Option list:
   A list of options selected in the design, including electrical specifications of the wire, shielding, voltage and current.

7. Symbol diagram:
   A list that outputs diagrams of designed symbols, which are usually stored in .dxf format. This format can be used in CAD programs to help you visually check your drawings.

All of these outputs are classified by harness code so that parts belonging to a particular harness code can be checked and managed separately. By outputting the output in Excel format, data can be easily analyzed and managed, and the symbol stored in .dxf format can be utilized for CAD drawings.

Configuration Settings

Low resolution display resolution settings



If the resolution is low, the display resolution settings must be specified separately. Here's how to set up your system.

1. Right-click the desktop CADvizor execution icon
2. Click "Properties" in the menu
3. "Compatibility" tab → "Change high DPI settings" in the Properties pane
4. Set the "Override High DPI Adjustment" part of the Settings window
   - ■ "Override High DPI Adjustment Behavior" Check
   - ■ Change to "Application" → "System (Advanced)"
5. Desktop → Right-click →→ Display Settings
6. Change the setting values in the text app and other items resize menu (set according to your screen)

Network connection

If a program is not used for a long time or the network environment is unstable, the connection to the server may be lost, resulting in limitations in functionality. At this time, if the program does not work properly or needs to be reconnected during the operation, you can use the account

information by pressing the button at the top right to reconnect. This will help you resolve connectivity issues and keep working smoothly.

Contact CADvizor

Room A-1106, U-Space, 660, Daewangpangyo-ro, Bundang-gu, Seongnam-si, Gyeonggi-do

Yura IT Business Division Development Team 2

13494

Phone: 070-7878-7082

Website: http://www.cadvizor.com

Sales Information

If you want to purchase additional CADvizor products, please contact your agency or head office directly.

Technical Support

If you have any technical inquiries about the product and any inconvenience in using it, please contact us by e-mail below.

Email: cadvizor@gmail.com

Feedback

If you have any improvements or errors regarding CADvizor, please feel free to contact us.