CADvizor 2025

User's Guide

[MFG Module]

Table of Contents

New features and introductions

1

Based on the CADvizor platform, the MFG module provides simple and practical features for manufacturing drawings. This chapter introduces new features and how to use them.

Logic-MFG Sync

After creating MFG projects and drawings that connect Logic deliverables, use when Logic design changes occur with all item (hannes code) drawings design completed. It is a function that reflects the modifications of the Logic project in the design information of the drawings in the MFG project.

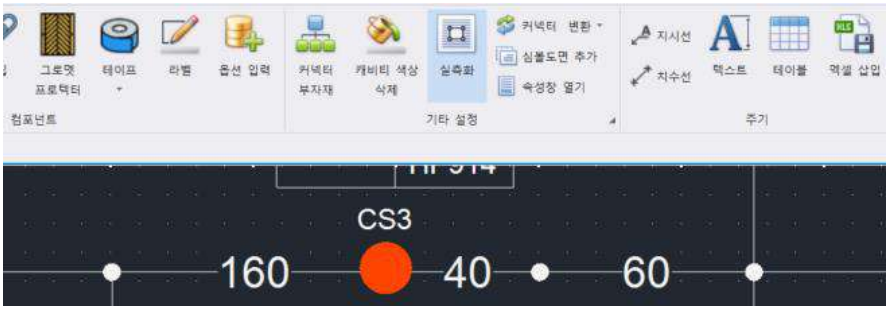
Complete all modifications in the Logic project and create and save the output. Go to the MFG module and import the MFG project you want to modify, select the project and click the [Logic Sync] button at  the top to automatically update the design information for all drawings included in the MFG project.

When you open the revised drawing to check the connector list, the Used column of the deleted connector shows 'X' and the Used column of the newly created connector is empty. The drawing does not change automatically, so the user must locate and delete the deleted connector and insert a new added connector.

Realization

After creating a layout, modifying the length text of a particular segment translates to a virtual length. In this case, to change the segment length to the measurement size to match the length text, use the Realization function.

For example, if you arbitrarily reduce the length of the segment shape of the drawing to draw a measurement drawing of size A0, the length text value remains the same. In other words, even if the segment representation length on the drawing is reduced, the length text is represented as the actual length.



To restore the drawing to its actual production drawing size in the future, use the Realization feature to convert the length of all segments of the drawing to the actual measurement value.

Delete cavity color

This function controls the wire color displayed in the cavity position within the connector symbol in the MFG drawing. It can be used for color discrimination when not managing wires in color or when printing black and white.

The feature is available through the Other Settings > Cavity Color button on the top bar and supports three options.

- Turn off the color

  Remove the wire color of the cavity and display only the wire name.

- Color display

  Restore the removed wire color to its original state and display it again.

- Color Text

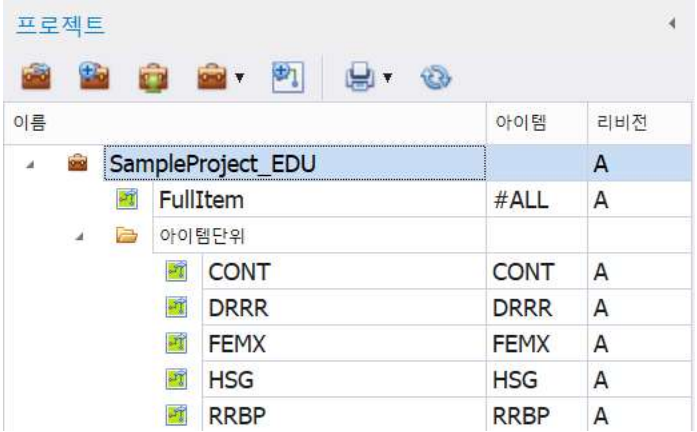
  The color display of the cavity is removed in the same way as 'Turn off the color', and the color of the wire is displayed under the name of the wire in text. However, the text color name is represented in a normal position only when the wire name is less than 6 characters.

MFG 2

The CADvizor MFG workspace is a graphical user interface designed for a variety of user workflows that outlines the basic components of the workspace. Describe how to access design components and design manufacturing drawings using the groove, drawing, view, analysis, and bridge tabs at the top. We start with how to create MFG projects and diagrams as the first step in utilizing the workspace.

Create projects and diagrams

Describes how to create a project that you need to work with and how to create drawings that are included in the project. The project includes a manufacturing drawing for each item and a folder to classify them.



Screen Configuration

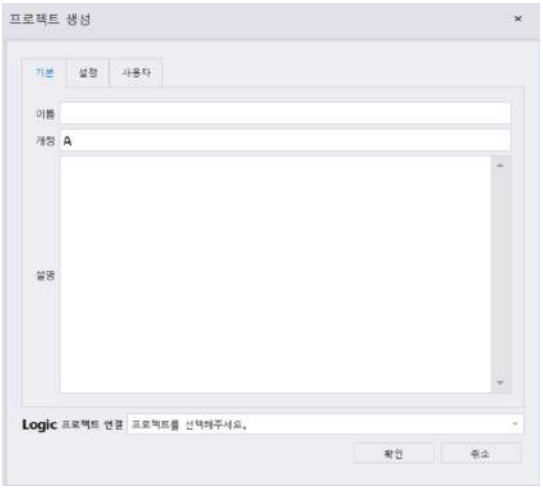
The function buttons in the Project Management panel include Open, Create, Import, Update, Other Features, Drawing, Print, Logic Syn from the left. The description of each button is shown in the table below.

function	Description
Open a project	Import projects stored in the cloud, display a list, select the project you want to import, and load it into the workspace.
Create a project	Displays a window that creates a new project.
Import Project	Import the local extension (.cpf) file into the workspace.
Project Other Features	It provides other functions such as closing, copying, and exporting projects.
Create Drawings	Create a new drawing for the selected project.
Print	Prints all drawings included in the selected project.
[Logic-MFG Sync]	Import the latest output of the Logic project linked to the selected project and update the design information of all drawings included in the project.

Create a project

If you have an existing project, you can access it by clicking the first button in the "Projects" panel  on the left side of the workspace. In the "Choose a Project" pane, select and import the project you want to import.

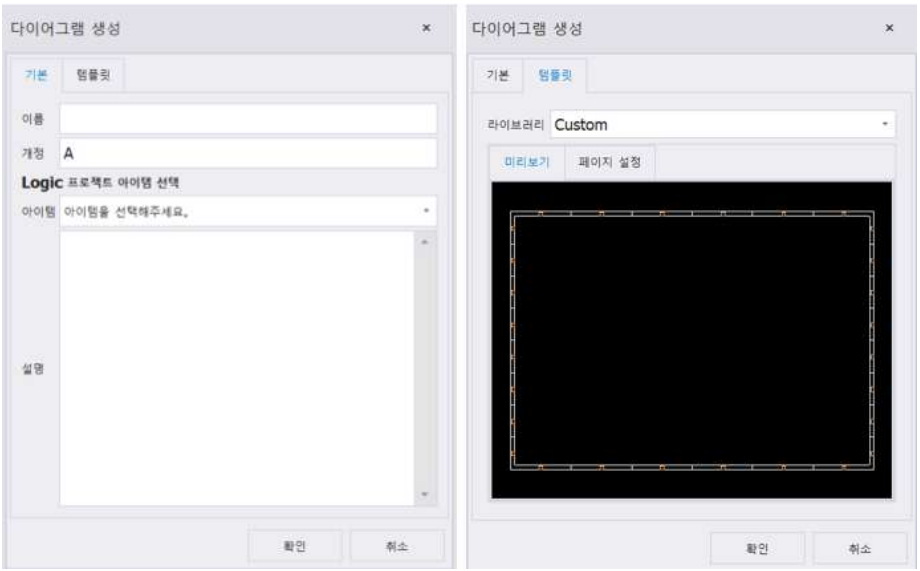
To create a new project, click the button  to open the Create Project window and proceed with the settings.



Create a project name and select the Logic output data that is pre-stored as "Logic Project Connection". On the Users tab, you can grant permission to edit per account.

Create a diagram

After  you select the project you created, access the Create Diagram window by clicking the top or clicking "Add Drawings" in the context menu where you right-clicked the project.



Create a drawing name and select the item in the manufacturing drawing classified by the harness code assigned to each object in the Logic project. On the Templates tab, select a template pre-saved in the library, or select a template to use through page settings.

Other Features

Copying is possible through the revision function of the project and drawing. An abnormal down of a project you are working on, such as an Internet connection error, can be opened by using the Cleanup function through  the button.

Drawing Information

Click the top left end "Design Information" button to view the data in the current open drawing. You can check the design information corresponding to the item selected in Diagram Creation among the selected Logic outputs when creating the project. It is a list of information about wires, connectors, and part numbers, and contains all the information necessary for the drawing design.

Design Information

Design information consisting of wire list, connector list, and part number list is stored as data linked to drawing and drawing left panel information and is reflected in real time when the window is closed. By default, it is editable, and if you click the button at the top, 

it becomes unedited 

. You can "add" and "delete" through the right-click menu.

- **The wire list** contains all the information about the wire, including the From-To path, joint, option, and multicore. Note that the connector name must be on the connector list and the joint and option must conform to the form to be applied to the system. Color values must exist in the library and two colors must be written consecutively without any special symbols. (Example: RB, BrO, LaPp)

- **The connector list** shows the location name of the connector set by Logic and the part number of the connector registered in the library, and all Library columns must be marked with an "O" for a normal drawing design. This indicates that the connector is connected to the library, and if there is no "O", all connectors can be connected to the library through the  "Connect Library" button at the top. If that part number exists in duplicate in the library, it cannot be connected and can be selected directly from the library by clicking on the cell.

- **The part number list** provides information about the combination of options that Logic applied to the wire. "Full Option", a P/NO with all options registered by Logic, is automatically registered. After registering a new P/NO, click on the option cell to register a P/NO for the new option combination by creating or deleting an "O" mark. Right-click "Edit Options" to add new options or exclude existing ones.

layout

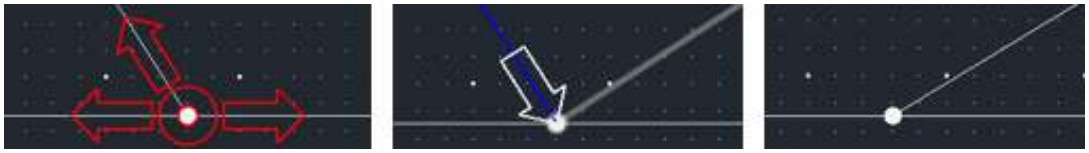
Once you have completed the 3D path design, you can see the branching method and length values for each bundle, which allows you to design a layout for manufacturing drawings. Bundle refers to the path through which wires pass, and in MFG, it is used in the same sense as a segment. It can be designed through the layout categories in the top menu bar.

Create Segment

Create by using the segment button or  shortcut "S" in the layout category of the top menu bar. Click the start and end points of the segment in the drawing to open the Enter Segment Length window. The start and end points are oriented only, and the actual length to be created is entered into the input window. The length value is a measurement size, and clicking OK creates a measurement-sized segment in the drawing. The length text information is placed in the center of the segment as shown below and is a true measurement design that matches the length of the segment exactly.

Modify Segment

If you select a node as a segment redirect method in the measurement design and click the arrow generated, you can only adjust the segment's orientation while the length is maintained.



The wire generated when segment was created is the actual length, and the following is how to change to a virtual length design with different values of actual length and text length.

- Modify length text directly: If you double-click the segment length text to change the value, it is applied as a virtual length that does not affect the actual segment length.
- Adjust segment: After selecting a segment, clicking one of the arrows at both ends will keep the length text intact and apply to a virtual length that changes only the direction and length of the segment.

Add a Node

It can  be created through the Add Node button or the shortcut "N" to generate nodes on top of the segment you created. Certain sub-materials, such as clips and tapes, are created on top of nodes, so nodes must be generated in advance.

Other Features

Other features in the layout include branch diagram editing and path navigation, and the description of

each feature is as follows.

function	Description
Spinning	When you select a segment to move with the reference node, the segment's length remains fixed.
Separation	With one node and one segment selected, press Enter to separate the layout based on that node.
Merge	Merges the first selected node and the second node.
Moving	By selecting one node, you can move all the connected layouts, and you can also use the shortcut "M".
Path navigation	When you select a node and press Enter, it outputs the total length and number of segments of the connected layout between the selected nodes.

Creating Connectors

The MFG module draws for each item and inserts a connector at the end of the bundle in the layout. The **Topology page** describes how to manage layouts that connect different items with inline connectors.

The connectors are connected to the library in the [Design Information - Connector List] and can be inserted into the drawing through the left panel connector list **tab** of the MFG main window.

Insert the connector

Double-click the connector you want to insert on the Connector List tab to highlight all nodes in the drawing layout. In this state, clicking the end node of the layout sets the position of the selected connector in the drawing, and clicking again determines the position of the connector. Connectors are represented by default in text form, and representations include text, symbols, and table formats. Each expression method is as follows.

the way of expression

The icon at the top right of the Connector List tab indicates how the connector will be represented in the current drawing. For example, in the figure above, it is set in a 'text' manner, and when the icon is clicked, it changes to a 'symbol' manner. Click again to switch to the 'table' method.

Create a joint

Insert General Joints

When all connectors are inserted into the drawing to fit the layout through the [Design Information - Connector List], wires with both From-To connectors in the wire list are shown as 'O' in the Used item and the length value (Len) of the wire is automatically determined.

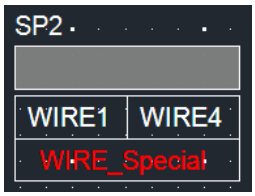
On the other hand, wires with joints in the path are marked with an unconfirmed length value (Len). In this case, right-click the wire in the Design Information - Wire List and select the [Joint] menu to access the joint insertion window and determine the length value (Len) by specifying the joint position on the drawing.

In the joint insertion window, the main and branch lines are automatically set depending on the type of joint selected (e.g., center strip, splice) and can be modified arbitrarily by the user. Click the OK button to enter a mode where you can specify the joint position on the segment. When you mouse over the segment, the length value (Len) of the wire list is reflected and displayed in real time. When you click above the segment with the mouse, a joint is created in that location, and the length value is immediately determined.

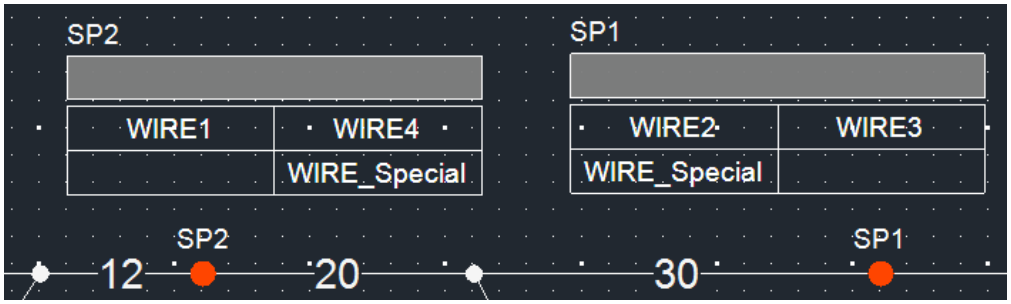
Insert special path joint

A special path joint means that one or more **of** the wires connected are **jointed at** both **ends.** The method of opening the joint insertion window and the insertion procedure are the same as normal joints, but for special path joints, wires are not automatically set in the "Branch Wire" list, so **you** must select **a wire with both ends joint** from the right "Wires" list and add it to the left before inserting it.

If a wire with **both ends joint** is inserted into the drawing only one joint, the wire will be shown in red as shown below. This means that the From-To information has not yet been determined on the drawing and is a visual warning indicating that the wire connection is incomplete.



If the joint at the other end is inserted into the drawing after checking the "Branch wire" list, the wire name that was marked red will be changed to white and the wire connection will be confirmed. This also determines the length (Len) value of the wire in the left wire list.



generation of subsidiary materials

Secondary materials are used to assemble, finish, protect, etc. by assisting this material, and to increase design completeness and to facilitate the manufacturing process. It is expressed by filling in dimensions, creating symbol(symbol) and Bill of Materials(BOM) in the manufacturing drawings.

BOM is a report that summarizes the list of all parts and subsidiary materials used in the design drawing, and contains information such as product name, specification, quantity, and material. This allows designers to pinpoint the parts they need and prevent errors in the assembly process.

The shape of the subsidiary material you want with the [Symbol] **function**, **the** [Library – ETC] **function** registers **attribute** information (part number, manufacturer, type, etc.) to manage it efficiently, and can be quickly imported and used when designing manufacturing drawings.

Add Zig

The jig maintains the wiring path of the wiring harness to help ensure that the wiring is properly positioned without tangling.

■ **Create Zig**

1. Click the Zig button in the Component category on the top menu bar.
2. Double-click the desired jig in the jig input window.
3. Select multiple nodes and type Enter. → Jig Inserted Done

■ **Change jig**

1. Click the Change Zig button at the bottom of the UI.
2. Select several jigs you want to change and enter Enter.
3. Double-click the new jig in the jig input window. → Finished changing the selected jig

Add Tube

The tube protects and organizes wiring and improves electrical safety.

1. Click the Tube  button in the Component category on the top menu bar.
2. In the Tube Input window, click the desired shape.
3. Enter **Size, Length, Option, and Taping** information at the bottom and click the OK **button.**
   - **Size is 0 or blank**: Reflected from the built-in calculation formula.
   - **If Length is 0 or blank**: Reflected as segment length.
4. Select multiple segments and enter Enter. → Tube Insertion Completed

Add Clips

Clips secure wiring to prevent damage to wiring due to vibration and impact, and to facilitate organization and maintenance.

1. Click the Clip button in the Component category on the top menu bar  .
2. Select the desired shape in the Clip Input window.
3. Review the list of information registered in the library and select the items that meet the required specifications.
4. If the selected shape requires Section information, enter the corresponding value and click the OK **button.**
5. Select one node you want and enter Enter. → Clip Insertion Completed

Add Grommet/Protector

The grommet protects the wiring from contacting sharp edges when **it passes through holes in the panel or body.**

The protector **covers and protects** the entire wiring **or a specific section** from external shocks, vibrations, heat, friction, etc.

■ **Importing and grouping objects**

1. Import the desired shape in D**XF/DWG format**.
2. Select the object as drag.
3. Click → in  the Edit  category on the top menu bar

■ **Create Grommet/Protector**

1. Select the bundled object and click the Grom**met/Protector** button in the components

category on  the top menu bar.

2. Select **Type (Grommet/Protector)** in the input window.
3. Review the list of information registered in the library and select the items that meet the required specifications.
4. Click **the OK button**.
5. Click one segment you want. → Grommet/Protector Insertion Completed

■ **Add subsidiary materials**

1. Right-click the inserted shape.
2. On the menu that appears, click **the Add** Sub**material** (**S**) option.
3. When you select and add a sub-material, the sub-material applies to that shape.

Add Tape

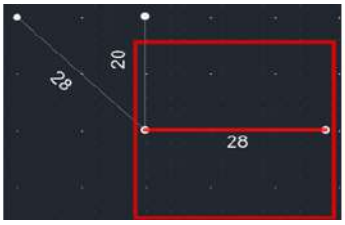
Tape covers the wiring for protection, insulation, fixation, noise reduction, etc.

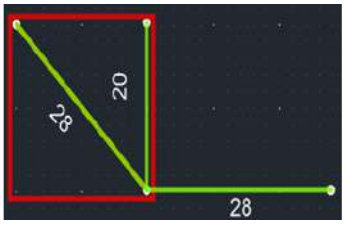
1. Create Tape

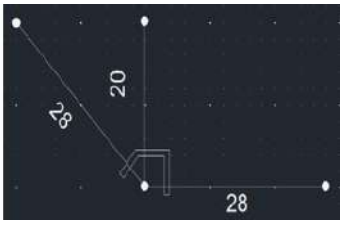
On the top menu bar, under Component categories, click the Tape  button.

2. **How to create by type**

- **Cross-Tape**: Click on one node you want. → Cross Tape Insertion Completed
- **Marking Tape**: Select the marking you want from the list → Click the OK **button** → Click one node you want. → Marking tape inserted complete
- **Bundle tape**: Select multiple segments you want to bind and type Enter.Draw the bundle mark with your mouse.After you're done,
  enter
  Enter. → Bundle tape inserted complete







Add connector sub-materials

The connector subsidiary materials serve to strengthen the connector's coupling force, waterproof, protective, insulating, aligning, and fixing.

How to add connector sub-materials

1. Click the connector you want.
2. On the top menu bar, under the components category, click the Connector  **Submaterial** button.
3. In the Sub-subsidiary Input window, load the registered information in Library – Connector.
4. Click the Apply button to be included in the SubMatrials list.
5. No sub-materials are registered in Library – Connector, or to add sub-materials: Check the list of information registered in the library and
   select the appropriate specifications.You can add it yourself by pressing the Add button.
6. Press **the OK button** to insert the sub-material into the drawing.

Key Features

In the left window of the MFG drawing, there are various functional panels that can be managed and inspected, including [Design Information] and drawing information. These panels allow you to identify

design components and efficiently check the completeness and accuracy of the drawings.

Connector list

It provides the [Create Connector] function to insert the connector into the drawing, and at the same time, it is a panel that can manage the connectors present in the drawing.

When you select a connector from the list of connectors, the wire information connected to each pin of that connector is available in the Connector Details window at the bottom.

If the Used item is blank, the connector indicates that it is not inserted into the drawing. When the connector is inserted into the drawing, 'O' will be displayed in the Used item to clearly check whether it is inserted or not.

The Connector List panel provides a variety of features for connectors with the Used item marked 'O'. Right-click on the connector to use the following key features.

- Find the location of the corresponding connector on the drawing
- Check the wire list connected to the corresponding connector
- Begin your drawing to determine the path of all wires or specific wires connected to the connector

The Route Initiation menu supports two methods.

- **Path View**: Visual representation of only the From-To section of the wire in the drawing
- **View all connection paths:** Visual representation of extended connection paths, including not only From-To of wires, but also other electrically connected wires at opposite ends (e.g., bicompression, joint, etc.)

When a joint (center strip, splice) is inserted into the drawing, a joint name is added at the bottom of the connector list, and 'X' is displayed in the Used item. This is an indication to distinguish that the item is a joint and not a normal connector. Joint items can also be inspected with the same function as regular connectors.

Wirelist

When you perform [Connector Generation] or [Joint Generation] in the drawing, if both of the From-To information of the wires connected to the part exist in the drawing, the length value (Len) of the wire is automatically calculated and reflected in the wire list in real time.

For wires marked with 'O' the Used entry provides the following features.

- **View circuit path:** Visual representation of only the From-To section of the selected wire in the drawing

■ **View all circuit connection information:** Visual representation of the entire connection path, including not only the From-To of the selected wire, but also other electrically connected wires at the end (e.g., twin pressure, joint, etc.)

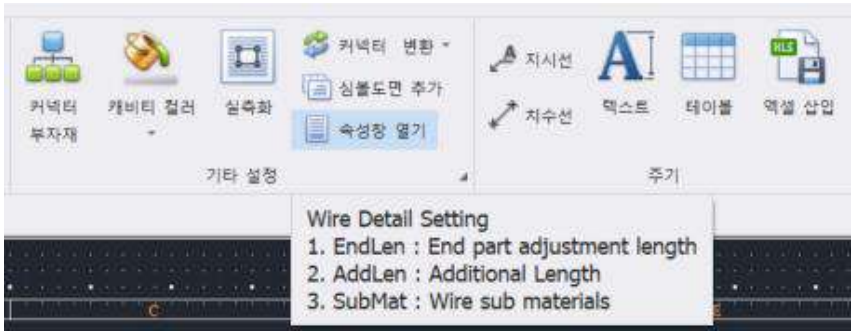
The wire cost information area at the bottom of the wire list panel provides total length and cost information for wires of the same specification based on the material, square (cross-sectional area), and color of the wire. This information is calculated and automatically displayed based on the unit price (based on 1m) entered for the wire in that condition in the library.

Part Number List

Configuring the part number in the Design Information - Part Number List automatically updates the part number list panel displayed in the left pane. Right-click a part number entry in the updated list to view or output the drawing and BOM for that part number.

Wire Detail Setting

The ability to add extra wire length and end length is accessible through the opening of the properties window of other settings.



Click Open Properties window to open a new window to the right of the drawing, which consists of two tabs.

■ In the first tab, you can check the connector and pin number information connected to each wire end, and you can enter the end length and subsidiary materials individually for each end.

■ On the second tab, you can enter an extra length for the entire wire. It is possible to enter individually for each wire, and the values entered through the button at the top right can be applied collectively to all wires based on percentage or absolute values.

Other Features

Save as a drawing file (DXF/PDF)

The ability to store manufacturing drawings designed by MFG in DXF or PDF format on a local PC.

To save as a drawing file

1. On the top menu bar, under Design **Data** Category, click **the Save** as File  button.
2.  Select the D**XF or PDF** format from the drop-down menu to save it.

Insert Label

Labeling segments within a design drawing makes it easy to identify and manage specific objects or areas in complex drawings. By separating specific objects or areas, confusion can be reduced and work efficiency can be increased during processing, assembly, and inspection.

How to insert a label

1. Click the Label button  in the Component category on the top menu bar.
2. In the Label Input window, create the desired label in the Name field and click the OK **button.**
3. Select one segment you want. → Label Insertion Completed

Enter Options

The ability to **specify an optional expression on** a wire in Logic, **and then an** additiona**l** optional **expression** on a subsidiary material in the MFG. In the part number list, the corresponding option for each part number (P/NO) is marked as **"O",** and you can see a drawing of objects that meet the option conditions of the part number (P/NO). More information **is available in** [Core Features – Part Number List].

How to Enter the Op-Shop

1. On the top menu bar, under Component Category, click **the Enter**  Options **button.**
2. In the Options Input pane, click the → OK button to set the desired option expression.
3. Select the objects to give you the option and type Enter. → Option input completed

Indication and Dimension Lines

■ **Indication lines** are used to intuitively express design intentions by clearly communicating a description of a particular object or area.

How to insert the indication line

1. Click the Indicator 

button in the Period category on the top menu bar.
2. Create a directive with a mouse click.
3. Create the text you want to enter in the instructions, and then click the OK button. → Instruction Line Insertion Completed

■ **Dimension lines** are used to help design accurate drawings by specifying the exact size of objects.

To insert a dimension line

1. Click the Dim**ension Line button** 

in the Period category on the top menu bar.
2. Click two points with your mouse to measure the dimensions.
3. Click with your mouse in the desired location. → Dimension line insertion completed

Insert text

You can create and place text anywhere you want in the drawing.

To insert text

1. Click the text button  in the Period category on the top menu bar.
2. In the text entry window, create what you want to enter.
3. Set the size, color, and font → Click the OK button → Place it in the desired location in the drawing.
4. Duplicate insertion is possible, enter the ESC key after all insertion is complete. → Text Insertion Completed

Insert Excel

You can create and place an Excel sheet at the desired location in the drawing.

How to insert Excel

1. Click **the Insert** Excel **button**  in the Period category on the top menu bar.
2. Create data in Cell in the spreadsheet entry window. You can copy and paste data from Excel on your local PC.
3. Click **the Insert into or merge** into a **drawing** button.
4. Place in the desired location in the drawing. → Excel insertion completed

Image

You can create and place an image in a drawing where you want it.

To insert an image

1. In the Edit category on the top menu bar, click the Image  button.
2. Locate the file (.jpg, .bmp, .png) to import from your local PC and double-click it.
3. Specify the location with the first click and the size with the second click. → IMAGE INSERTED

Text original location

Restores the wiring elements' names to their original system settings.

Text Where to Go

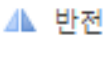
1. Click Wiring Elements.
2. In the Edit category on the top menu bar, click the **Where**  Text button.

Edit Shape Information

The object shape information editing function is located in the editing category of the upper menu bar and allows you to modify the shape information of the object. Descriptions of the features are described in the order of inversion**, rotation, scale, fit, group, and alignment**, and the usage of each feature is as follows.

■ **a twist**

The inversion function is to invert the selected shapes along the vertical/horizontal axis.**Invertible items**
: nodes, segments, connectors, joints

1. Click the shapes of the object you want to invert.
2. Click **the Invert** 

button.
3. Select the axis that you want to invert with the mouse.
4. The shape is inverted based on the selected axis.

■ **Spinning**

The rotation function rotates the selected shapes in units of 90 degrees.**Non-rotatable items**
: nodes, segments, connectors, joints

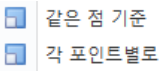
1. Click the shapes of the object you want to rotate.
2. Click **the Turn** 

button.
3. Click the reference point to rotate.
4. After rotating to the desired angle, click the mouse to apply it.

■ **scale**

Scale function is the ability to shrink/enlarge selected features.**Non-scalable items**: nodes, segments

1. Click the shapes of the object you want to scale.
2. Click **the Scale** 

button.
3. From the drop-down menu, select 

criteria for the same point or for each **point.**

   - **Same point basis**: A method of resizing around a single reference point. All objects are reduced or enlarged in the same proportion around the reference point
   . - **By** each
   point: The method of resizing based on the selected individual point (coordinates). Each point can be deformed at a different magnification, so the ratio of the shape may vary.

4. Click the reference point to scale.

5. Adjust the scale with your mouse or enter a value directly in the text box.

■ Customization

The customization function is to align the selected features to the reference line.

1. Select the features of the object to align with the baseline.

2. Click **the Custom**  button.

3. From the drop-down menu, select a baseline from one of the left**, right, up, and down.**

4. The objects are aligned to the selected baseline.

■ Group

A group function is the ability to group selected features, or to release already tied features.
Not applicable to wiring elements.

1. Select the shapes of the objects to group/ungroup.

2. Click **the Group** button  .

3. From the drop-down menu 

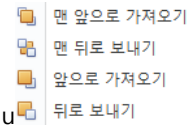
, select Tie **or** Untie**.**

■ Sorting

The alignment function is to adjust the front and rear order of the selected features.
Wiring elements are not applied because the layers are fixed.

1. Click one feature of the object to be sorted.

2. Click  **the sort** button.

3. From the drop-down menu 

, select **Import** Forward **or Send Back.**

Report

Once the MFG drawing design is complete, the report can be output based on the final design information. These functions can be executed through the [BOM] button and  the [Product Number Output] button  located in the report category of the main menu bar.

BOM

 Run through the [BOM] button, and based on the designed drawing, you can output a list of materials required for manufacturing preparation and a report for sub-operation.

The following three reports are available through this feature.

■ Material BOM

Provide the total length of all materials and subsidiary materials inserted in the MFG drawing, and wires. Wires are divided by material and square (SQ) basis and are calculated by total length for wires of the same specification.

■ Wire BOM

Report for utilizing the total wire prepared by the material BOM in the cutting stage, providing the length of each individual wire in detail.

■ SubList

It is used as a machining reference for sub-work before putting the cut wire into the full-scale manufacturing process. The terminal and seal information required at both ends of each wire is provided as a connector library basis, and the wires in the bimodal or joint (center strip, splice) relationship are visualized and provided as arrow-shaped connections.

Item number output

Depending on the optional configuration, you can save the drawings and BOM data for each part number created in a file format.

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.