

ElectricalOM CAD Plan Design Add-on Module

Table of contents

Introduction	4
Workspace and Right-click Menu	5
Move and Copy to Page	8
Select Similar and Select Similar with options...	9
CAD Plan Design Main Screen	11
Toolbar No.1	12
Toolbar No.2	21
Toolbar No.3	22
XRef Settings	22
Export and Print Settings	25
Symbols Summary	29
Material Lists	31
Multiline Settings	36
Other settings	37
Toolbar No.4	40
Zooming	41
Object Snaps	43
End Point	44
Mid Point	45
Nearest Point	46
Perpendicular Point	48
Intersection Point	49
Centre Point	50
Quadrant Point	51
Node	52
Tangent Point	52
Extension Point	53
Measure length and area	53
Symbols Area	54
Creating a new symbol	57
From scratch	58
Using an existing symbol	58
Using an existing shape from drawing area	59
Editing and deleting an existing symbol	60
Quick rename and Quick change category	61
Importing symbols	62
Synchronise symbols	63
Side Panel	64
Pages Tab	65
Layer Options	71
Page Details	74
Page Frame	74
Design Tab	76
Electrical Design Tools	77
Draw Symbol	78
Draw Symbols Array	81
Draw Wiring	83

Drawing a reference wire	86
Wire by Selection	90
Fix Wiring Connections in Page	92
Update Labels	93
Update Symbols Label from Schematic	93
Set Labels Prefix To Schematic	94
Insert Legend and Cable Schedule	95
Page Symbols/Wiring Legend	96
Page Spaces/Zones Legend	97
Cable Schedules	97
Multilines	98
Architectural Tools	99
Reference from ElectricalOM Circuits	102
Create Circuits via CAD Plan Design	102
Assign CAD Plan Design symbols to ElectricalOM circuits	107
Dissociate symbols	109
Select all symbols and wires of a circuit	111
Assigned Measurements	112
Properties Tab	114
Circuit Groups Tab	115

Introduction

CAD Plan Design add-on Module for ElectricalOM enables the user to create electrical layout drawings in 2D and 3D with simple actions.

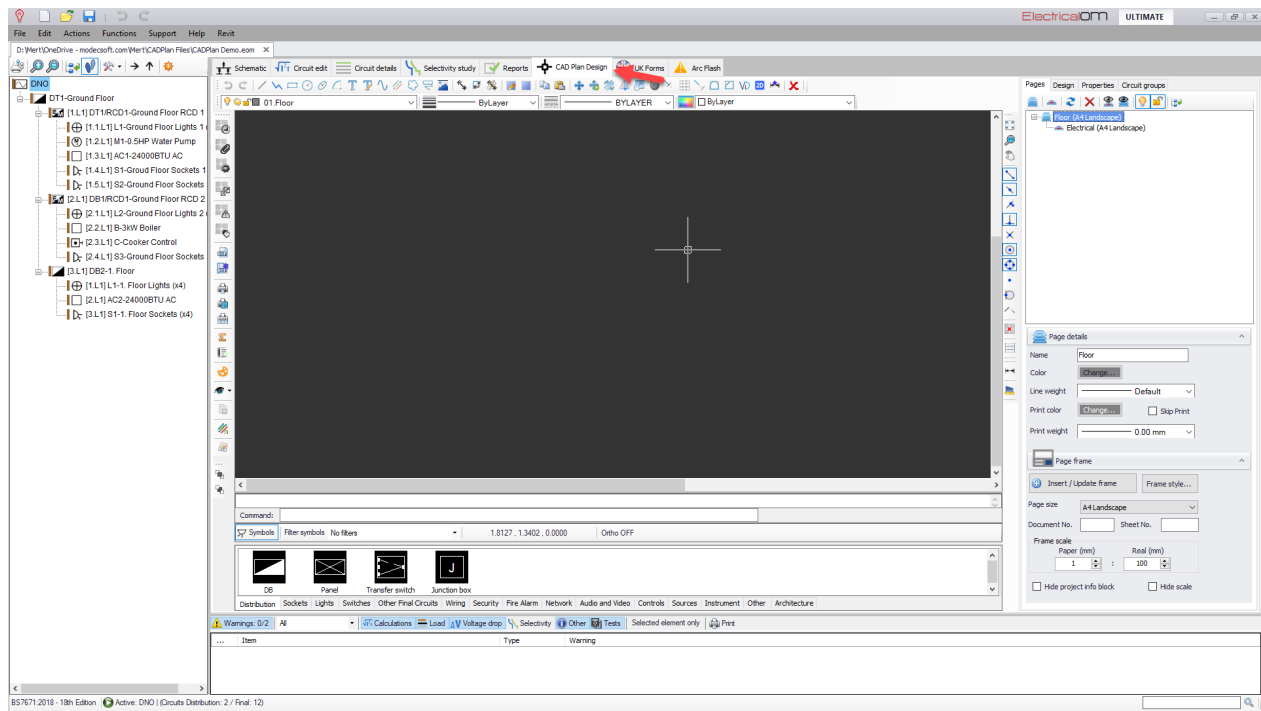
Users may import external references, create new objects on a CAD environment with the help of all the required CAD tools like snapping, etc., create layers for different purposes, use built-in or personal symbols to represent components, create reports like List of Materials, and export/print the outcome.

The main power of CAD Plan Design module is the ability to link the layout drawing to a system already created by ElectricalOM core software. Users can assign circuits from the core software to CAD Plan Design environment, and share the properties in both directions e.g. cable lengths can be estimated from the CAD Plan drawing and reflected to the original design, circuit prefixes can be populated from the original design on to the CAD Plan Design drawing.

CAD Plan Design add-on module requires to be purchased separately from the core ElectricalOM software at our Add-Ons web page.

The screenshot shows the ElectricalOM website interface. At the top, there is a navigation bar with the following items: 'ElectricalOM', 'News', '18th Edition', 'UK Training', 'Pricing', 'Support', and 'Contact Us'. A red arrow points to the 'Support' dropdown menu, which is open and shows 'Purchase License' and 'Add-on Modules' options. Below the navigation bar, there is a main content area with a red header 'CAD Plan Design Add-on'. The content area contains two video thumbnails. The first thumbnail is titled 'Automate the electrical design with CAD Plan Design add-on' and shows a 3D electrical layout. The second thumbnail is titled 'Watch how you can import drawings in CAD Plan Design Add-on' and shows a 2D electrical drawing. At the bottom of the page, there is a red banner for 'Autodesk Revit® Synchronization tool - Available only for ElectricalOM Ultimate'.

After the purchase, there is no need to download any additional files. The add-on will automatically be added to your modules tabs on the first run of ElectricalOM.

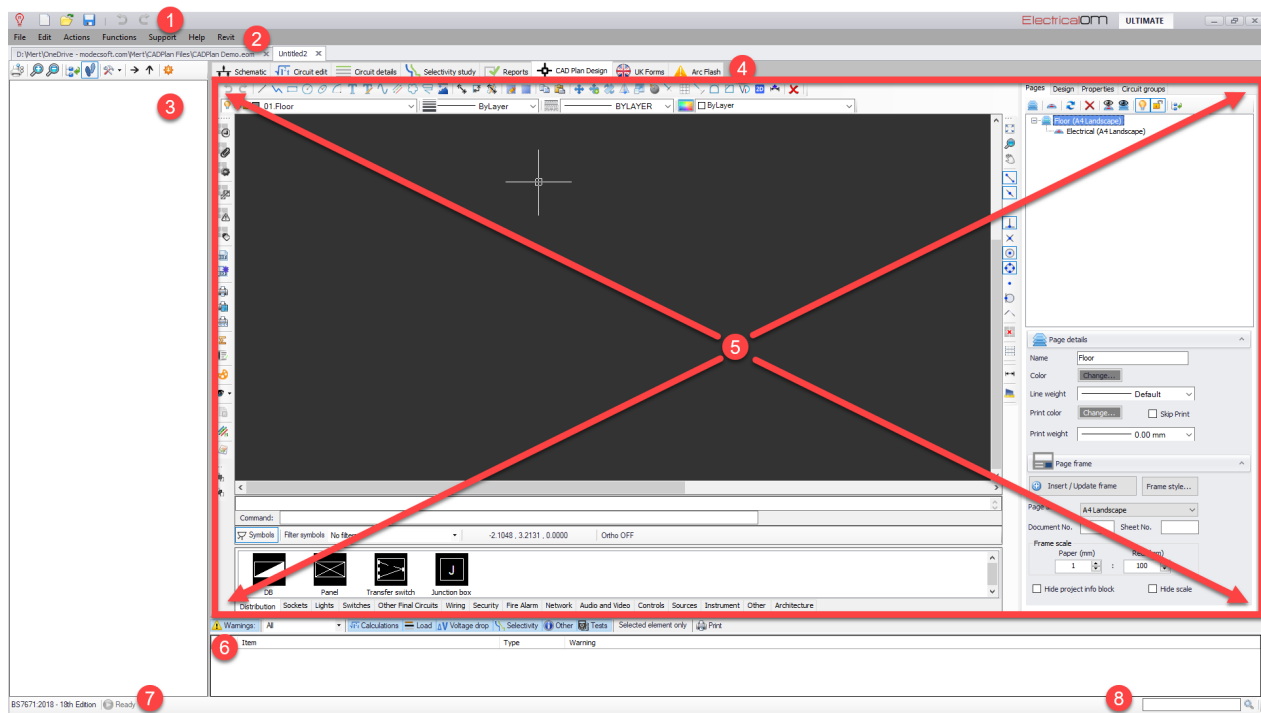


Workspace and Right-click Menu

Workspace

The main screen is divided into different areas which serve different specific functions; some are purely informative, some are the ones where the user inputs related data, and some are for navigation purposes. The user needs to know exact functions of all these areas in order to build up and design the electrical installation effectively leading to an accurate modelling of the system. Main screen areas are listed and also shown below:

1. Quick Access Toolbar: This toolbar includes shortcut buttons for basic functions.
2. Menu Bar: This bar includes various menu tabs.
3. System Tree Section: This section shows the system designed as a tree representation.
4. Module Selection Bar: This bar includes various module tabs.
5. CAD Plan Design Module: This is the working area for CAD Plan Design module.
6. Warnings Section: Warnings related with the system being designed will be shown here.
7. Status Bar: This area indicates some software settings being used for calculations.
8. Search Box: User can run a text based search by typing in a keyword in this box.

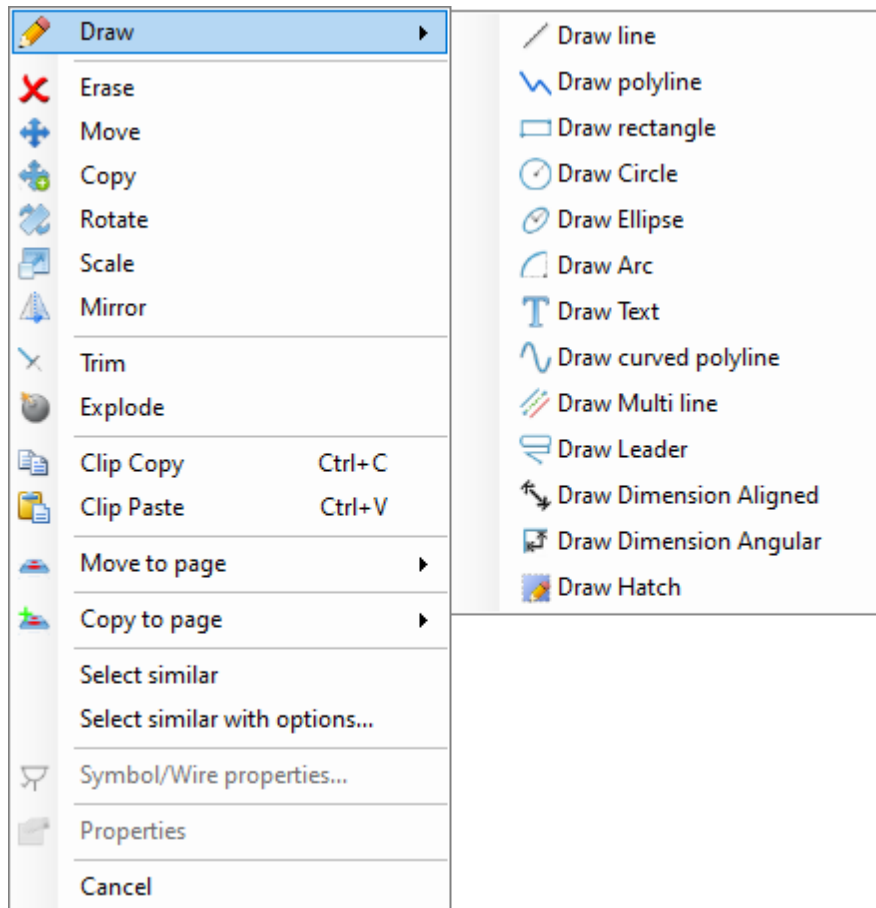


For further details about screen areas other than CAD Plan Design module, please refer to our [Online Manual for ElectricalOM](#).

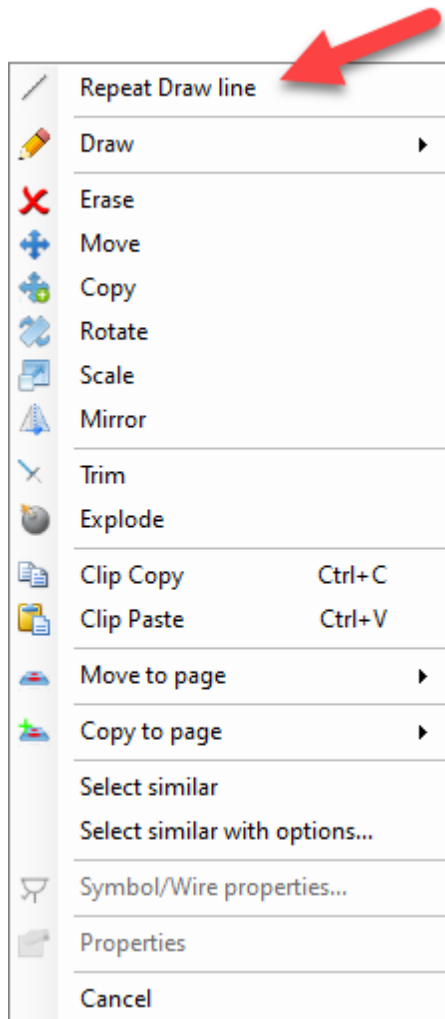
Right-click Menu

The right-click menu of CAD Plan Design provides a faster experience accommodating most used commands. Most of the options are the same as the ones already discussed in Toolbar 1 section of this manual.

Clip Copy and Clip Paste options copy and paste any selected item as expected.



However, there are a few options which are unique to the right-click menu. First of these is the repeat the last command feature. Once a command is executed, this will be saved, and next right-click menu will present this command at the top of the menu.

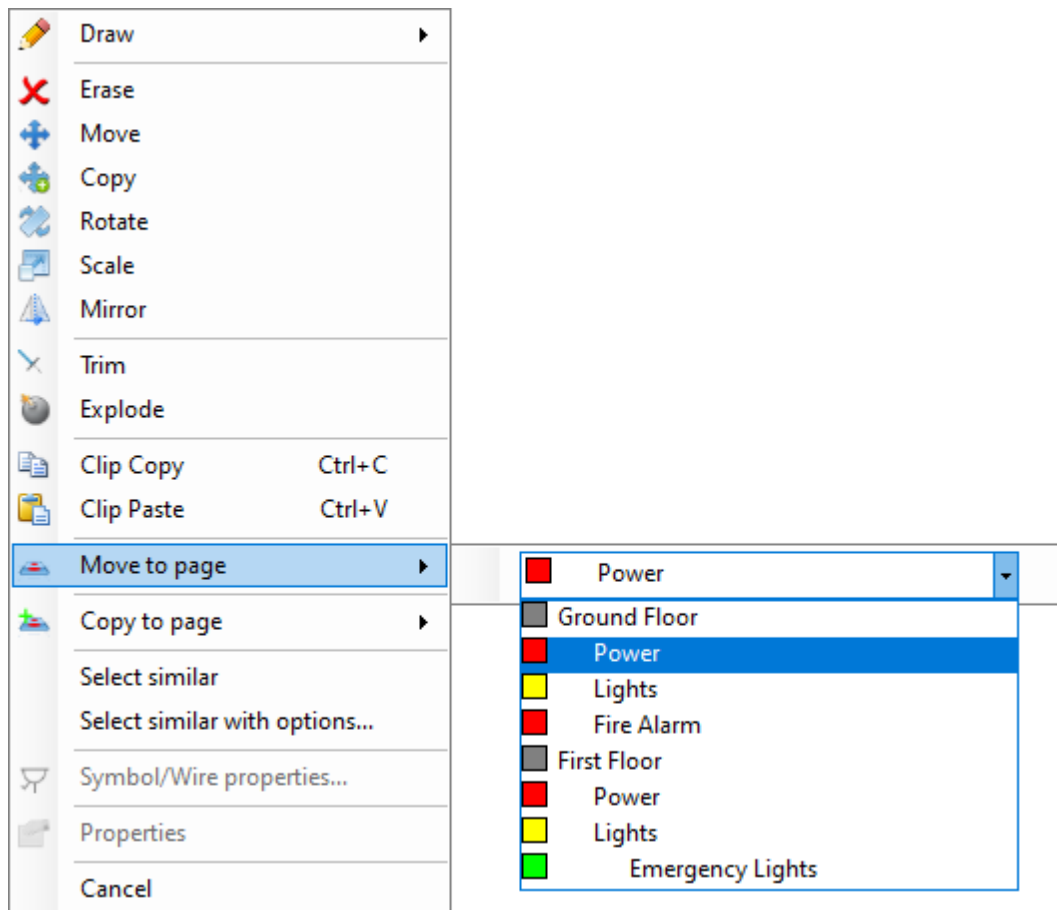


Other options are:

- [Move and Copy to page](#)
- [Select similar and Select similar with options...](#)

Move and Copy to Page

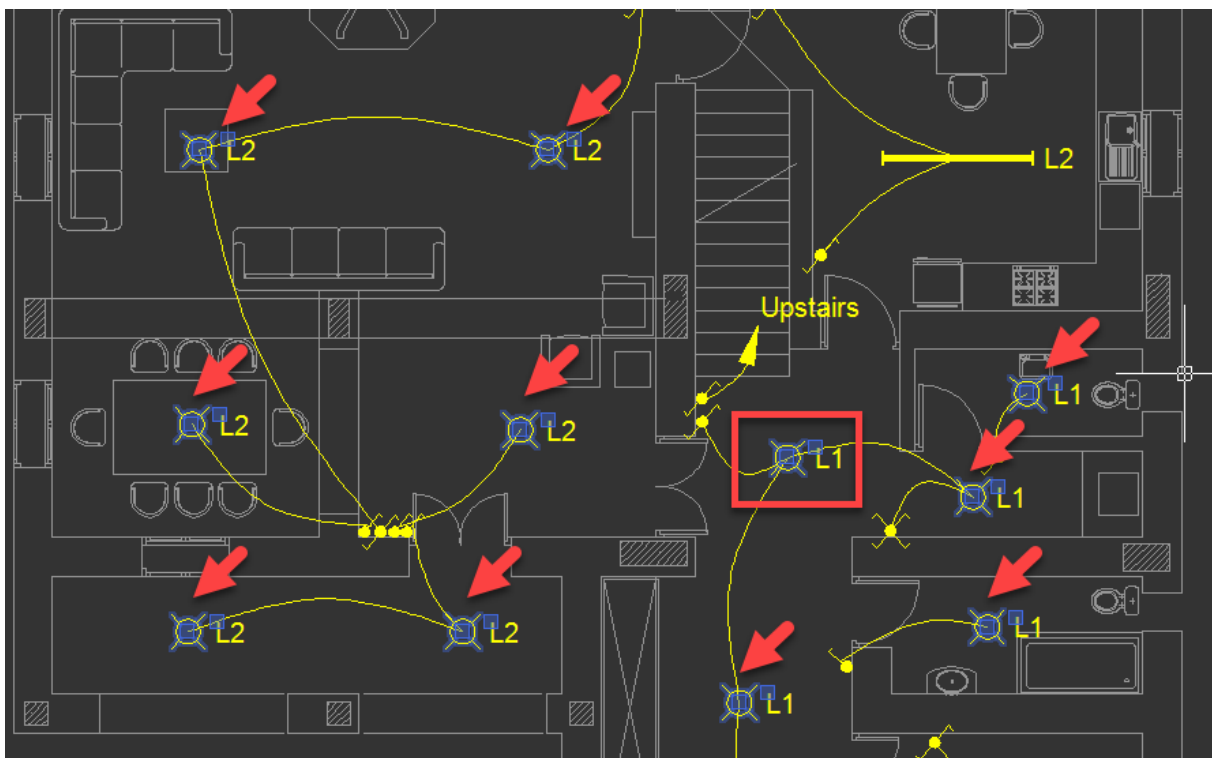
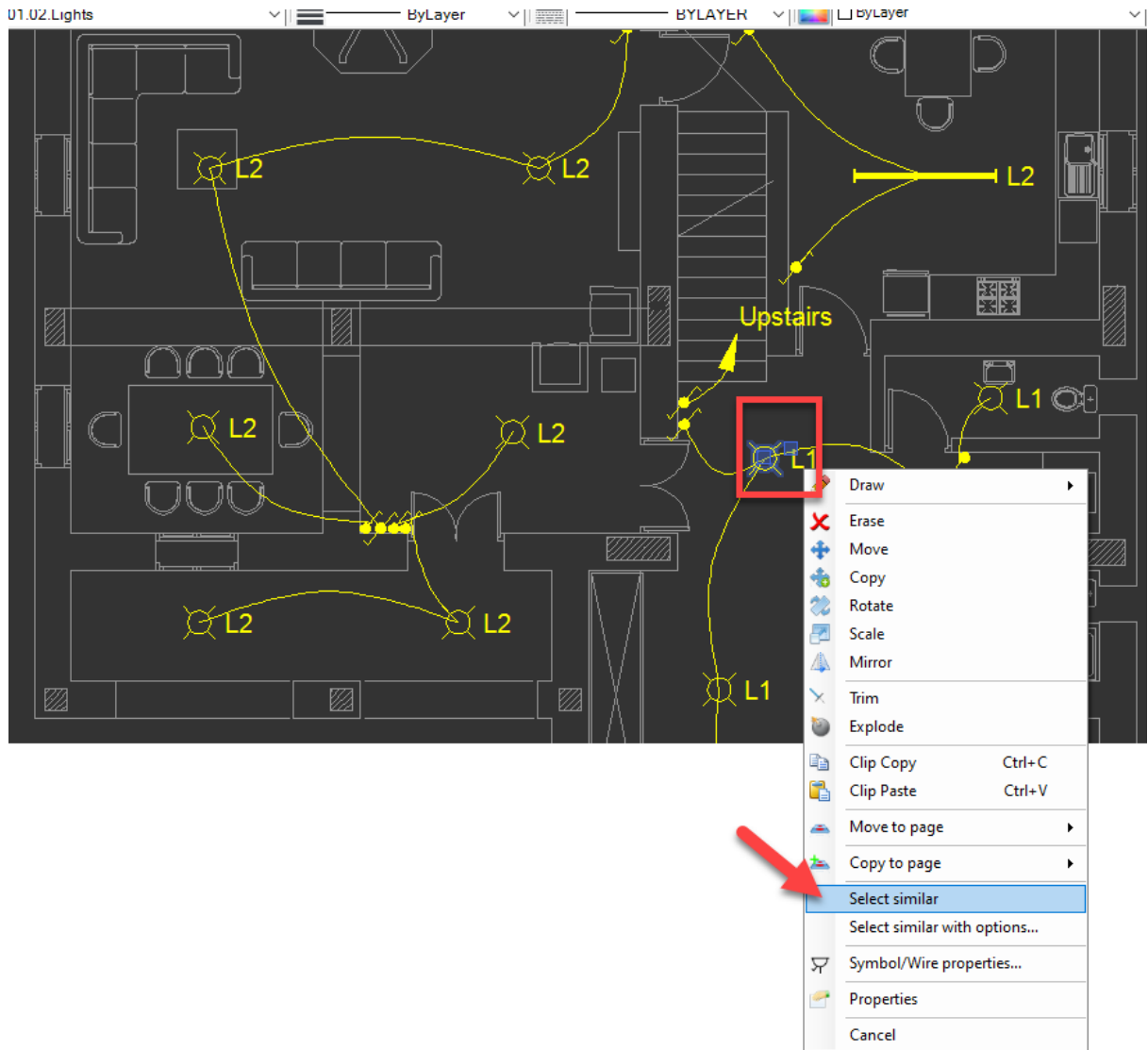
Using these two options, it is possible to move or copy any selected object to a desired layer. Once the option is selected, a drop-down list will be displayed where the user can select the required layer to move or copy the selected object(s).



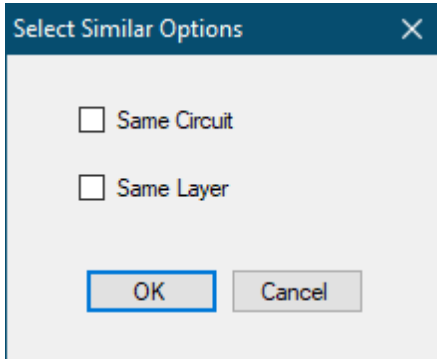
Select Similar and Select Similar with options...

It is possible to expand a selection to cover the whole project or layer by using these two options.

- Select similar will directly select the similar symbols assigned to the circuits on the active layer. The selection will also include other layers if there are any symbols belonging to the circuits on the current layer.



- Select similar with options... on the other hand will present two preference to the user: Same Circuit and Same Layer

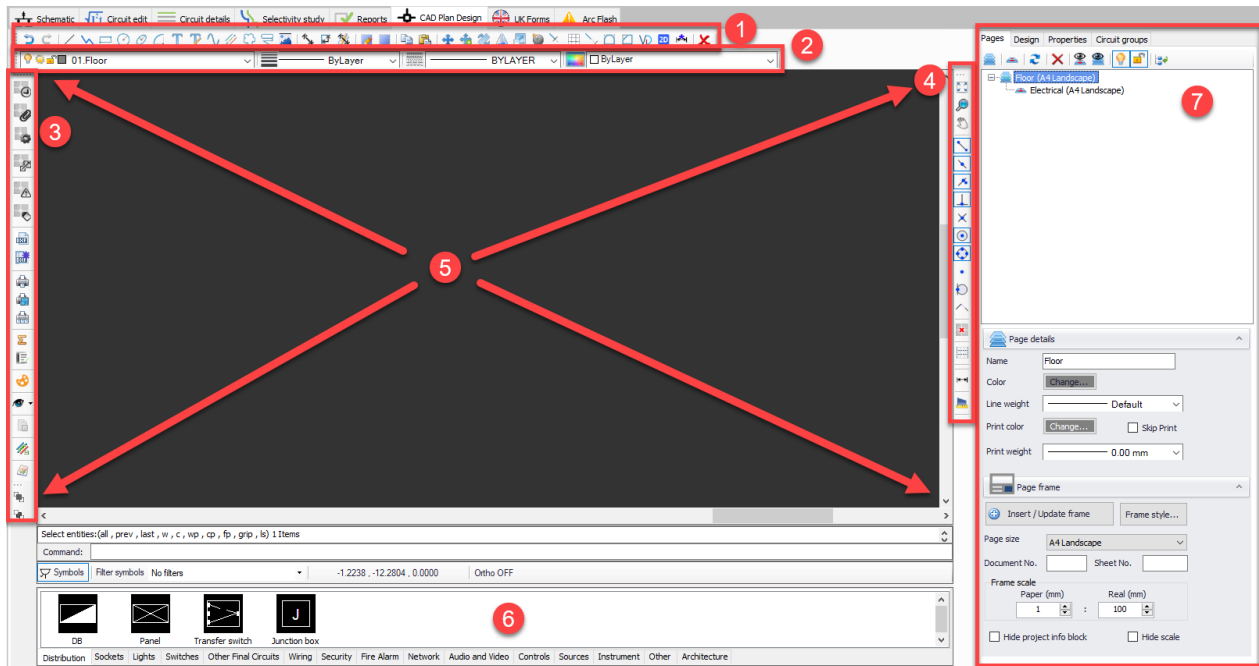


These options can be selected simultaneously. Same circuit will select the similar symbols assigned to the same circuit only. Selected only this options will not limit the selection with the active layer. Same Layer will select similar symbols on the current layer only. This option will ignore the circuit assignation. Selecting both options will select similar symbols which are assigned to the same circuit and will also limit the selection with the current layer.

CAD Plan Design Main Screen

CAD Plan Design module screen consists of:

- 1 to 4. Toolbars: Toolbars accommodates requited tools to complete a drawing. Further details are located within following chapters.
5. Drawing area: This is the working area of CAD Plan Design module. All the items are selected from this area.
6. Symbols area: This area accommodates the symbols can be used within a drawing. It works with drag and drop philosophy.
7. Side Panel: This panel consists of various tabs which will be detailed in following chapters.



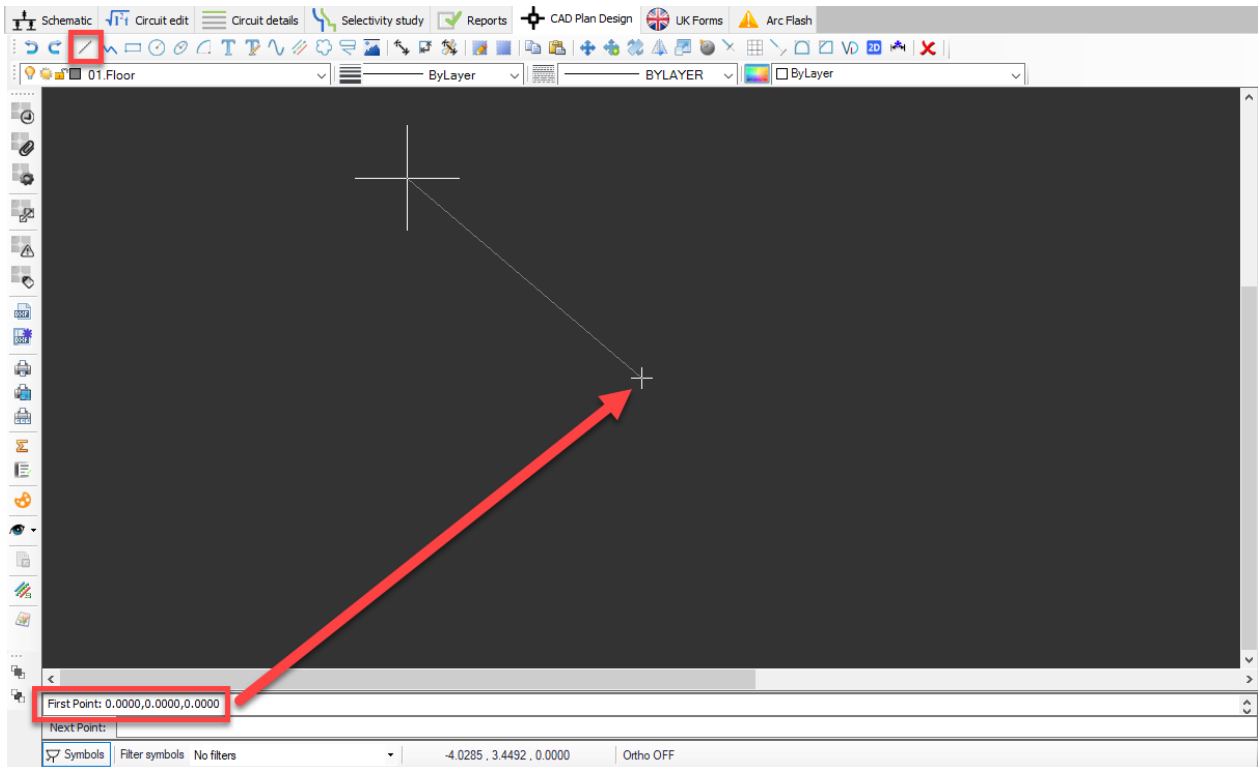
Toolbar No.1



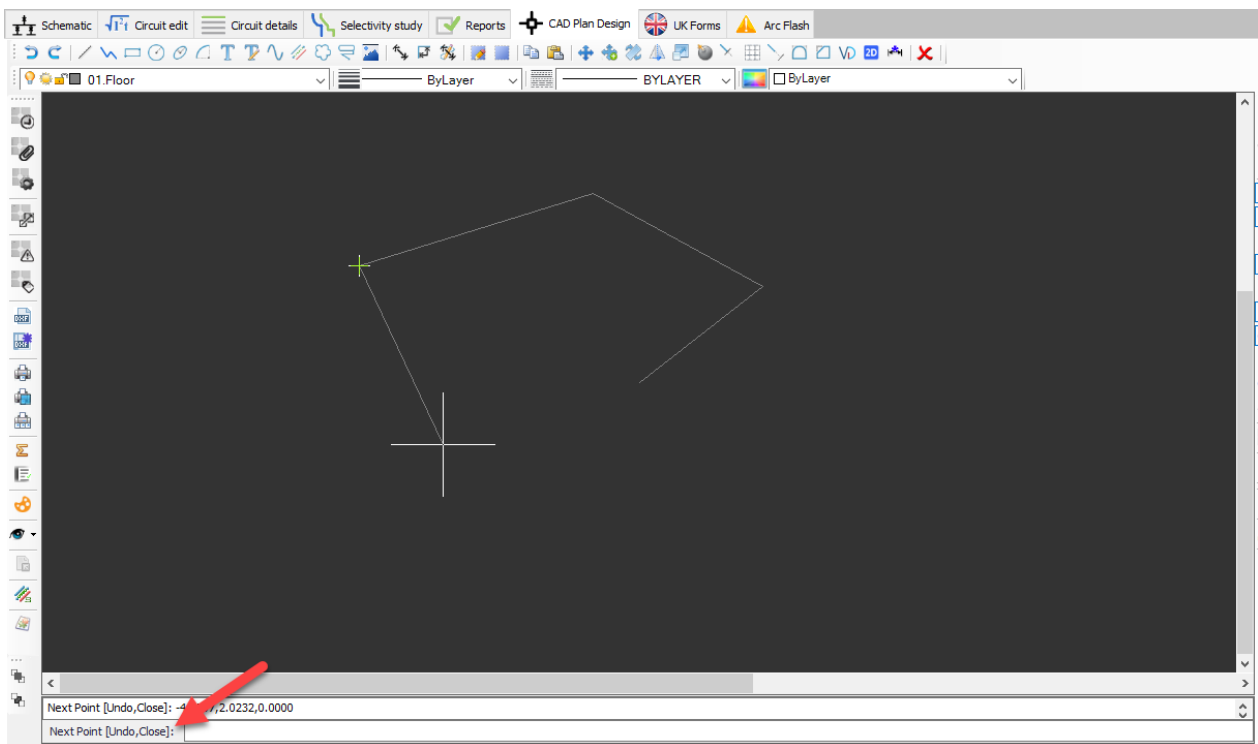
Toolbar 1 is focused on drawing, and manipulation of items in the drawing area. Keyboard key F8 will enable or disable the orthogonal mode which only lets the user to draw on X or Y axis regardless of the cursor position.

Toolbar No.1 buttons are, In order from left to right:

- **Undo/Redo:** These buttons take back/repeat the last action. They can be used multiple times to take back/repeat multiple actions.
- **Draw Line:** This button is used to draw a line on drawing area. Once this button is click ElectricalOM will wait for an input from the user to define the starting point. It can be done either by selecting a point from the drawing area by clicking on the left mouse button or entering a coordinate using keyboard.



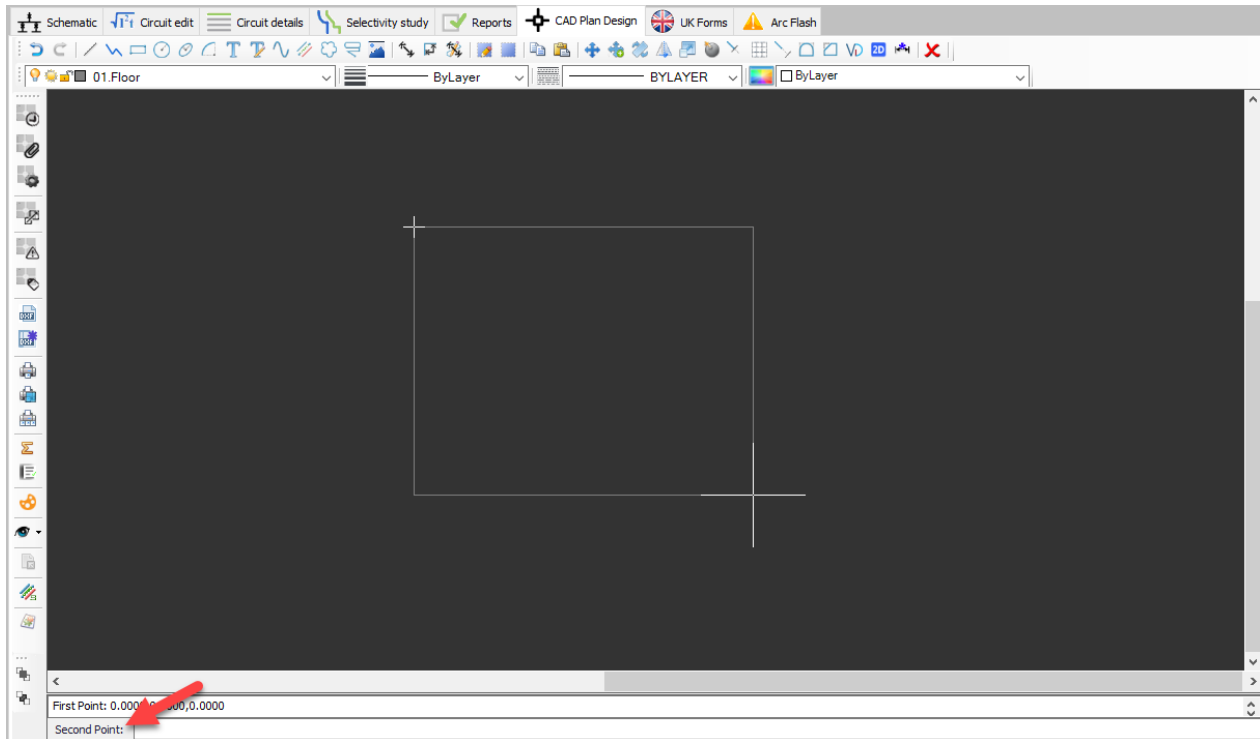
It is possible to continue drawing lines by selecting points continuously or entering coordinates one after the other. **Esc** key will stop the process.



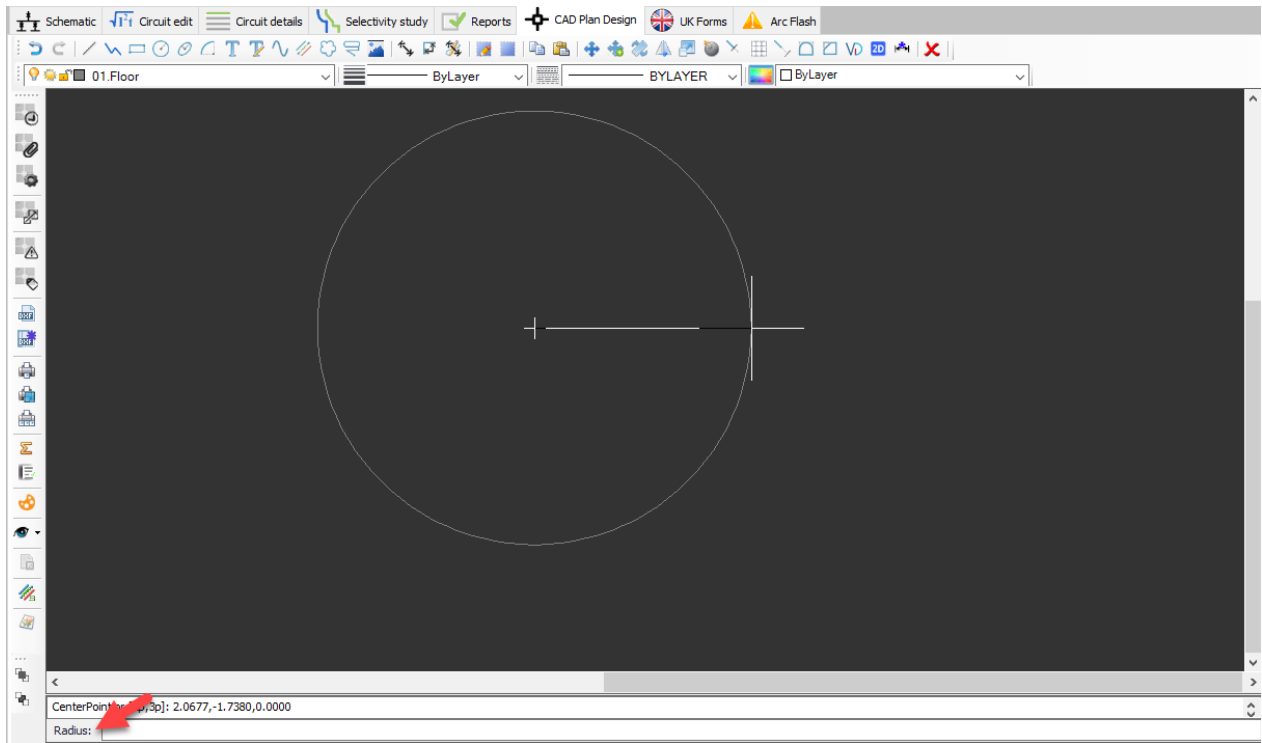
- Draw Polyline: This button is used to draw a polyline on the drawing area. Usage is similar to drawing a line, however, the result will be a single item rather than multiple

lines.

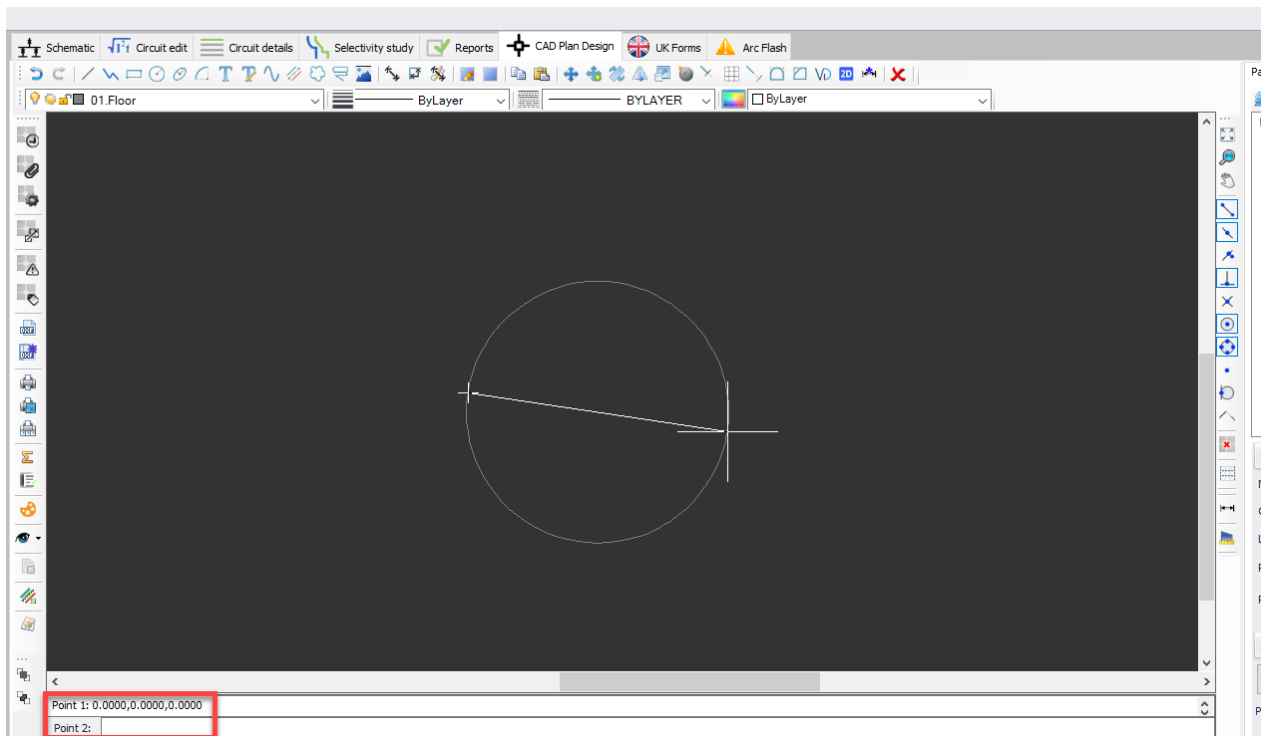
- Draw rectangle: This button is used to draw a rectangle on the drawing area. Once clicked, ElectricalOM will first wait for a start point entry, then, for a second point entry.



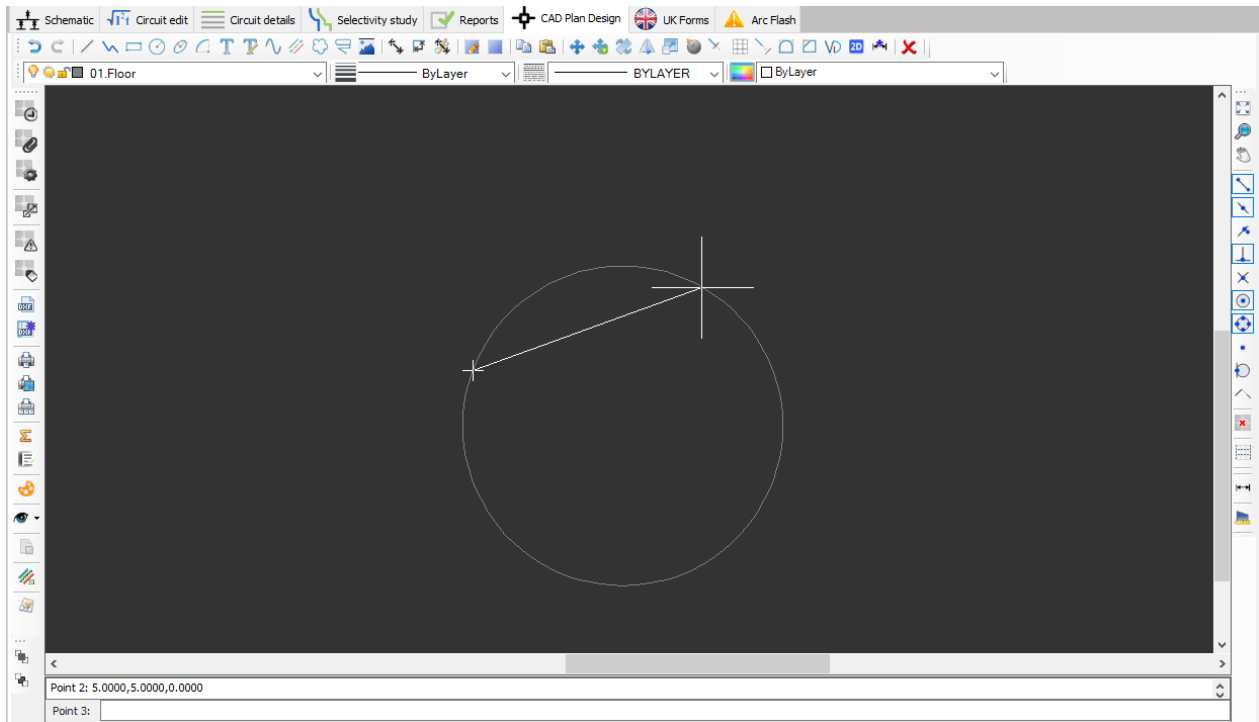
- Draw circle: This button is used to draw a circle on the drawing area. Once clicked, a centre point entry will be expected, however, the user may force a 2 point or a 3 point definition of the circle by typing in 2P or 3P using the keyboard. If centre point is defined, then, ElectricalOM will wait for a radius.



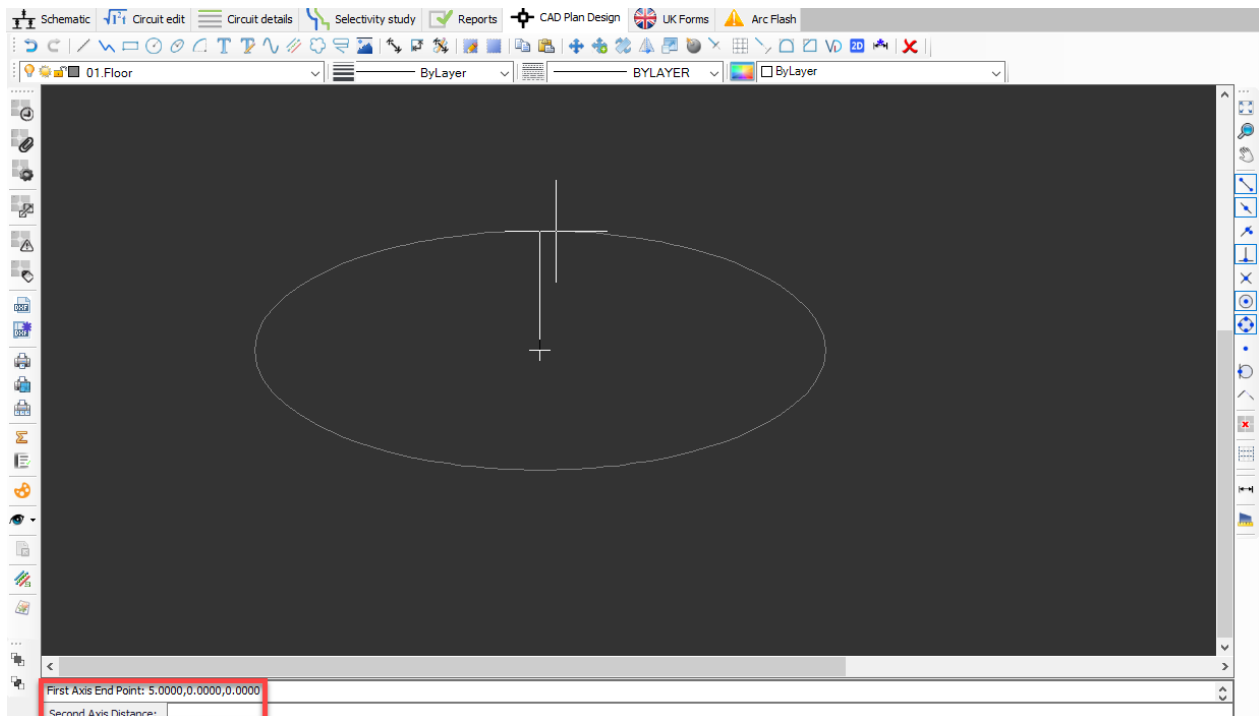
2 point option will wait for a Point 1, and then, for Point 2.



3P option will expect three points to be defined by the user to draw a circle.

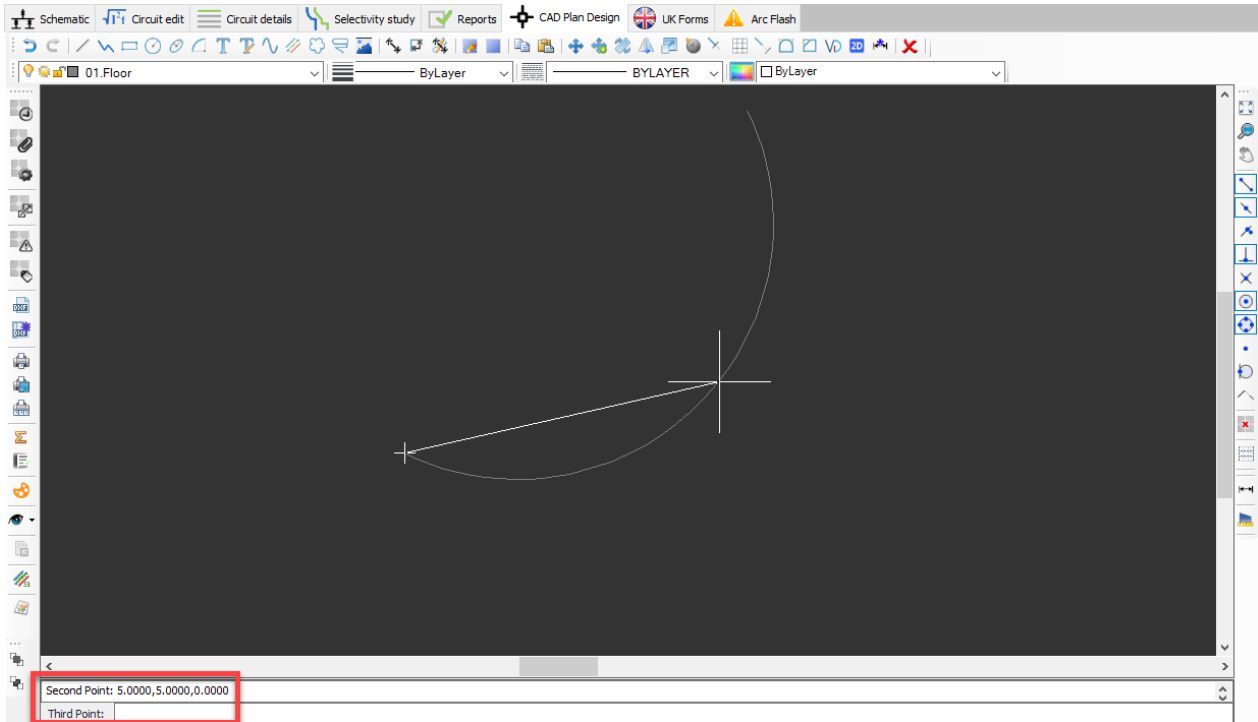


- Draw ellipse: This button is used to draw an ellipse on the drawing area. ElectricalOM requires a start point, a First axis length, and a second axis length to be defined by the user.

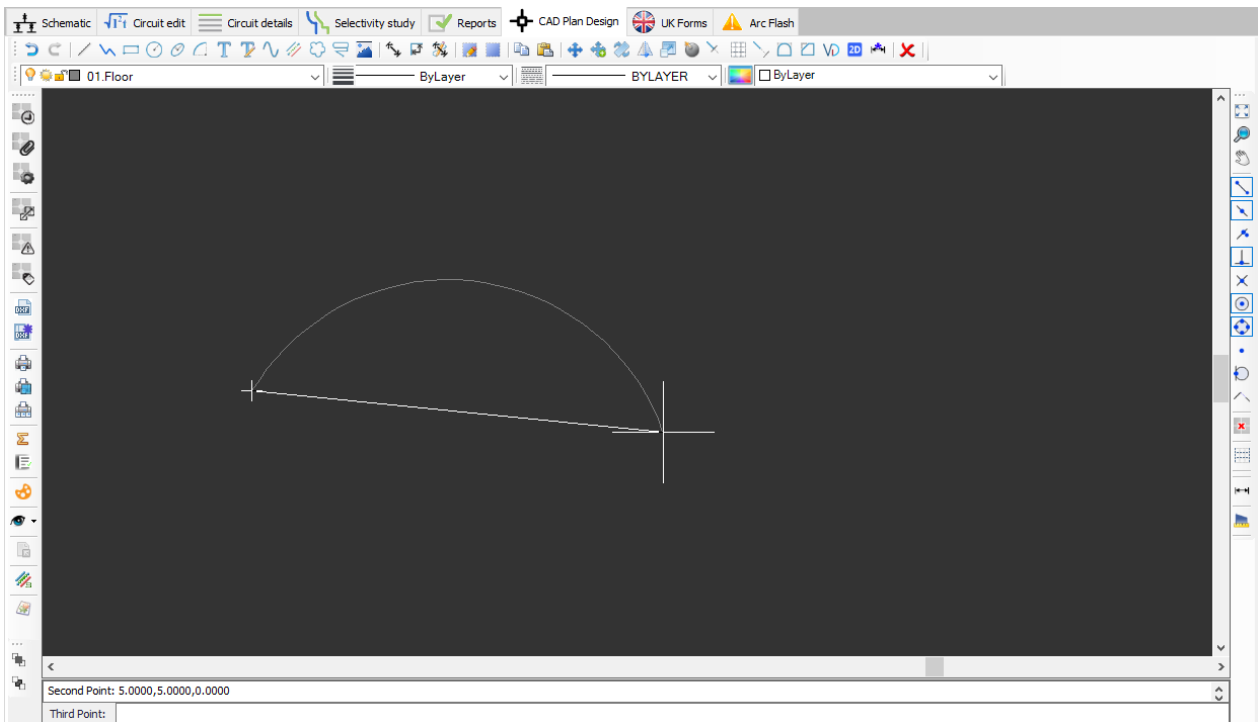


- Draw arc: This button is used to draw an arc on the drawing area. Arc has three ways of definition. Default way will expect the user to define a centre point, a radius, a start

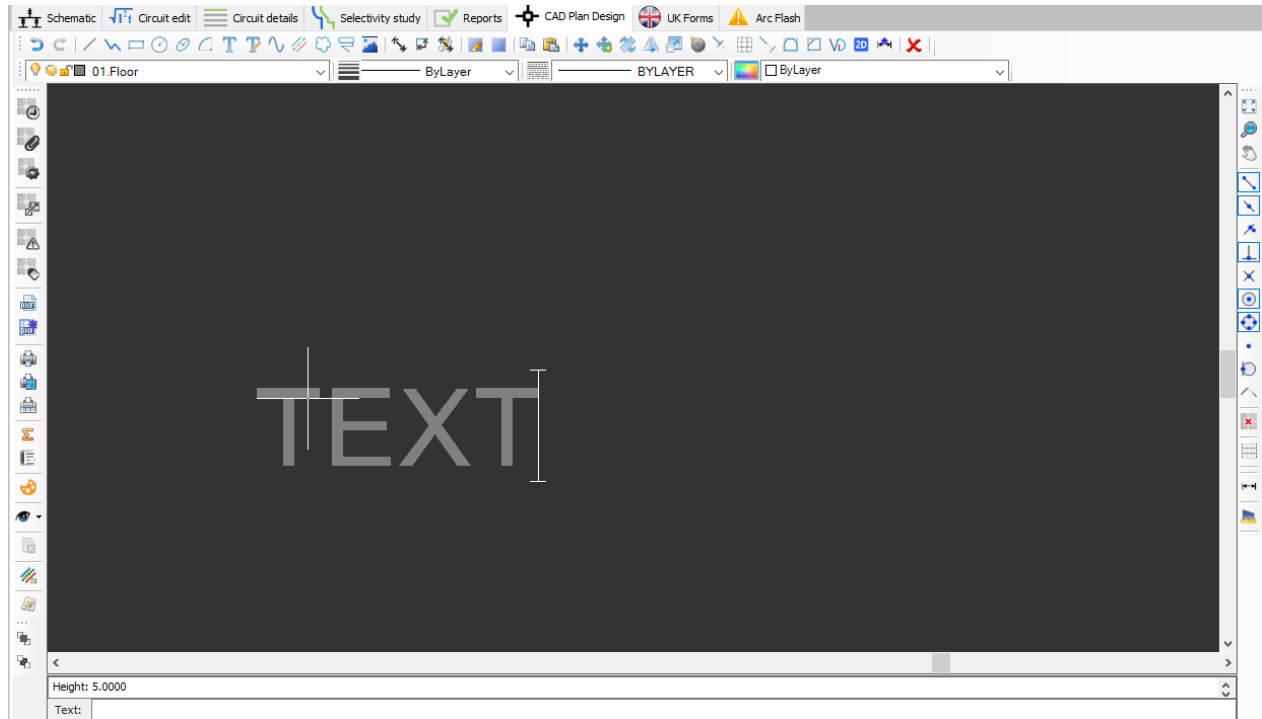
angle and an end angle. 3P option will require a start point, an end point and a third point to finalise the drawing.



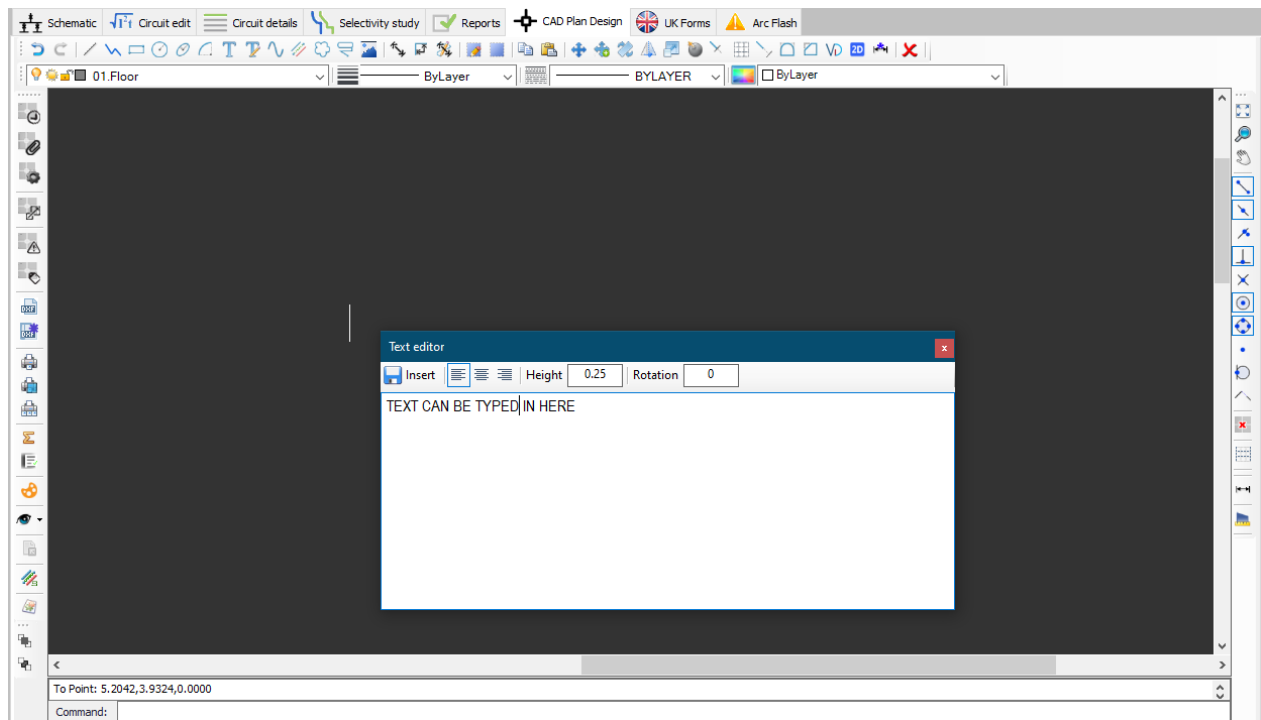
E3P option will also requires three points to be defined, however, this time the last entry is the end point.



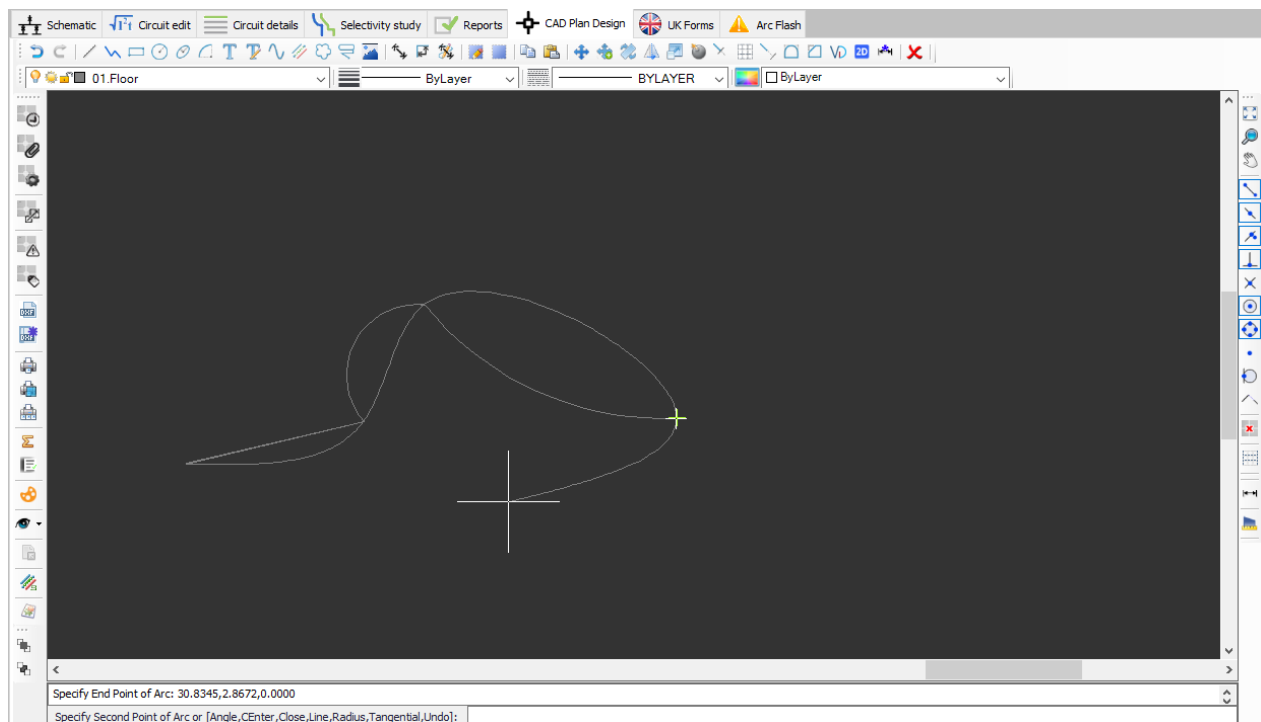
- **Insert text:** This button is used to insert a text on the drawing area directly. Once clicked, the user must define the insertion point for the text and the height of the text. Then, required text can be entered using the keyboard. Once the user finished typing, right click will finalise the process.

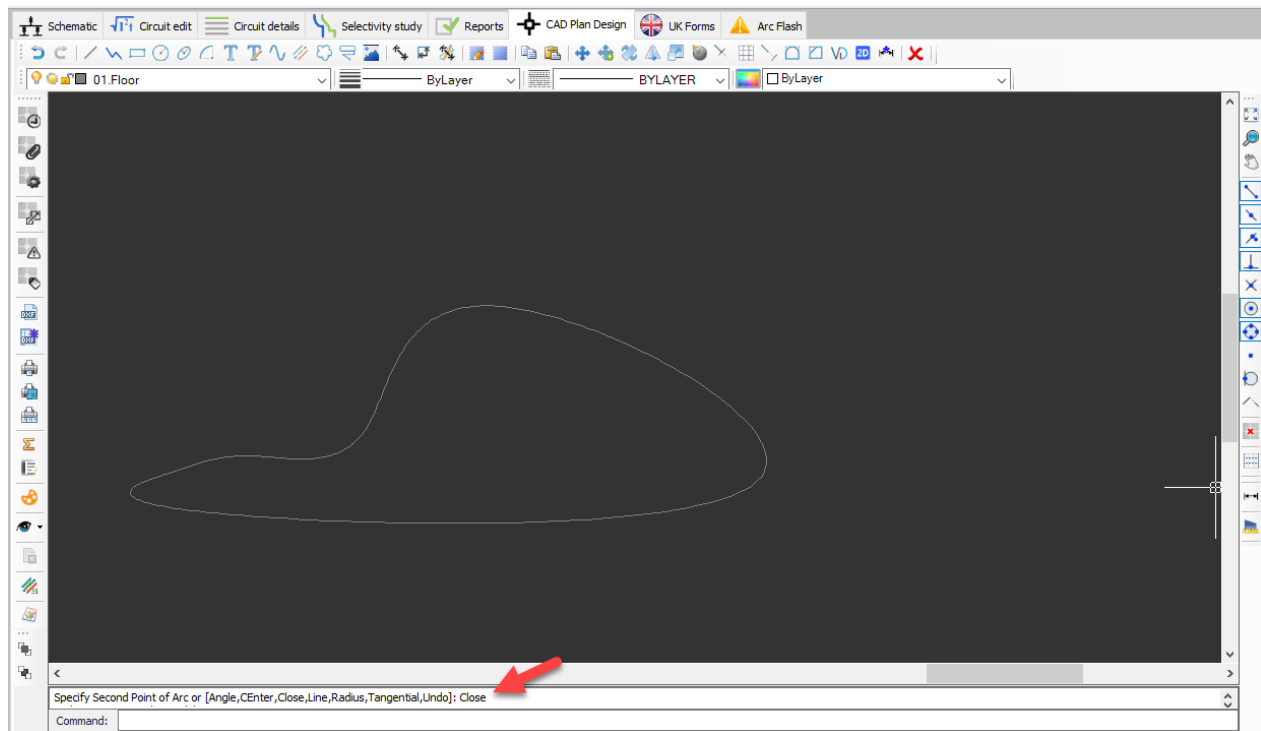


- **Insert text with editor:** This button is used to insert a text using a text editor window. Similar to insert text, this button will also wait for an insertion point to be defined, only this time a text editor window will appear to define text properties. Insert button will insert the text to the defined location.



- Draw curved polyline: This button is used to draw a spline on the drawing area. This button will wait for a start point to be defined, first. Then, user will have various option to draw spine. Typing in desired options will expect user to define specific points. Once drawing of the spline is finished, user may like to leave the drawn shape as it is, or may choose to join the first and the last points by typing in Close.

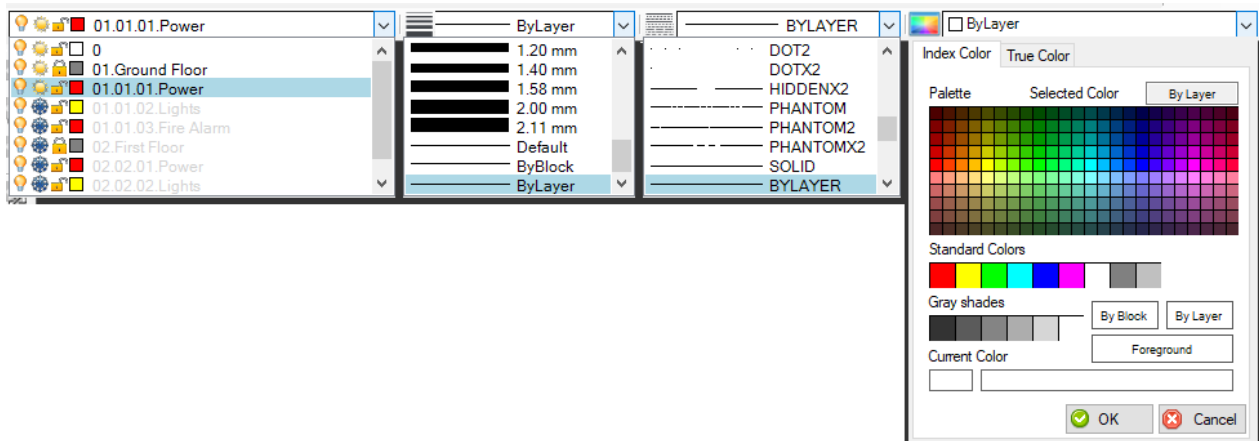




- Draw multi-line: This button is used to draw a multi-line on the drawing area. For multiline settings, see [Multiline Settings](#).
- Draw revision cloud: This button is used to draw a revision clouds on the drawing area.
- Draw leader: This button is used to draw a leader on the drawing area.
- Insert image: This button is used to insert an image on the drawing area.
- Draw dimension aligned: This button is use to insert a dimension aligned with the reference item.
- Draw dimension angular: This button is used to insert an angular dimension related with the reference item.
- Dimension styles: This buttons displays a windows to set dimension properties.
- Draw hatch: This button is used to draw a hatch.
- Figure hatch fill: This button displays a windows to set hatch properties.
- Clip copy: This button is used to copy selection.
- Clip paste: This button is used to paste already copied items on to the drawing area.
- Move: This button is used to move selected items.
- Copy: This button is used to copy and move selected items.
- Rotate. This button is used to rotate selected items.
- Mirror: This button is used to mirror selected items.
- Scale: This button is used to scale selected items.

- Explode: This button is used to explode selected items.
- Trim: This button is used to trim a selected item from a reference point.
- Array: This button is used to create an array from a selected item.
- Extend: This button is used to extend a selected item to a reference point.
- Fillet: This button is used to fillet the edges of two items.
- Chamfer: This button chamfers the edges of two items.
- Join: This button is used to join two items.
- Flatten: This button is used to flatten 3D items. In other words, it is used to convert 3D to 2D.
- Fix dimensions: This button will convert an external reference's dimensions from inches to metres.
- Erase: This button is used to delete selected items.

Toolbar No.2



Toolbar 2 is focused on temporary setting to be applied for the drawing action. From left to right:

- Layer list will list the layers available. When a layer is selected, any drawing will be one on this layer.
- Line weight list will set the line thickness, and Line type will set the line type for the active drawing session.
- Line colour will set the colour for the active drawing session.

The settings above do not affect the default layer settings but only the current drawing action. In order to change default settings, please refer to [Page Details](#).

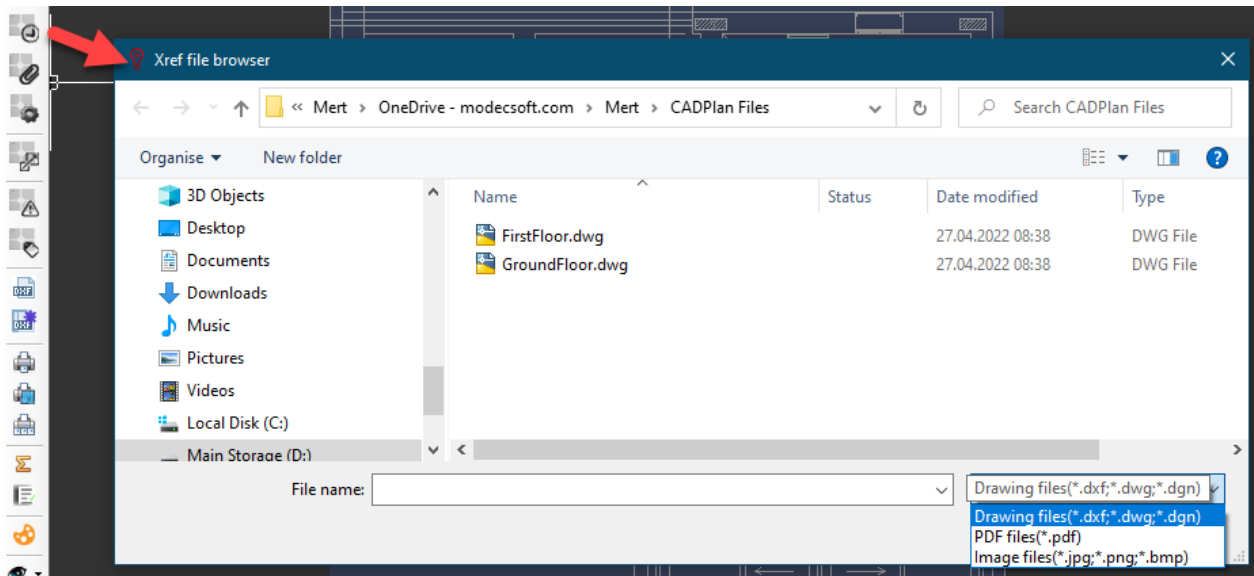
Toolbar No.3



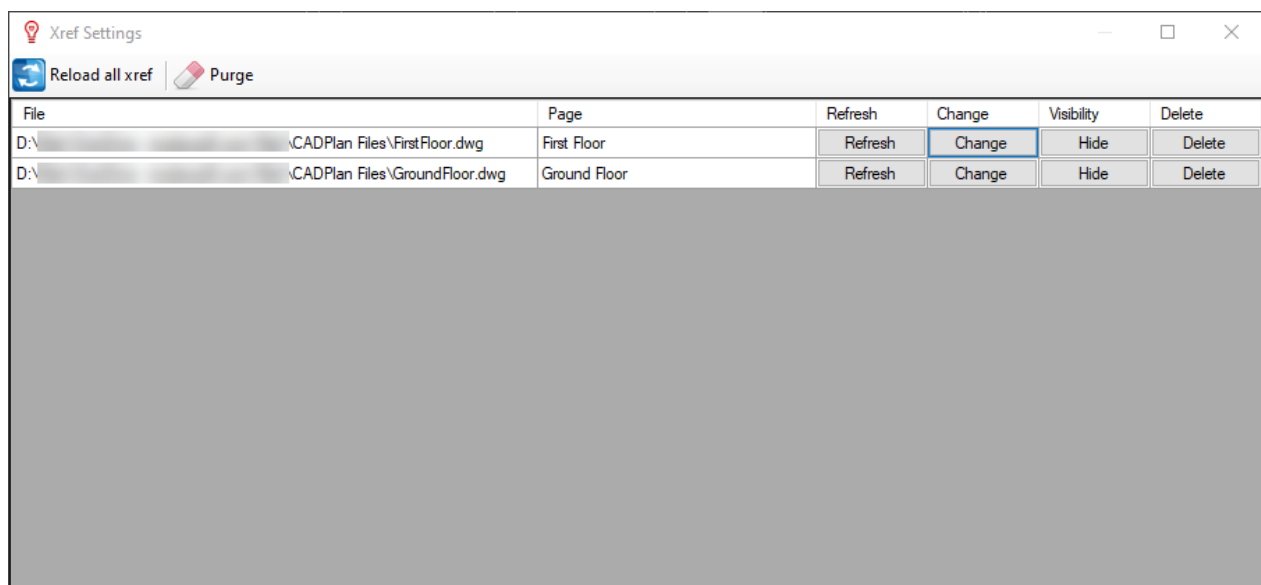
Toolbar 3 is located on the left hand side of the drawing area. It is mainly focused on external references, export and print, reports, and multiline settings. For further detail refer to specific chapters.

XRef Settings

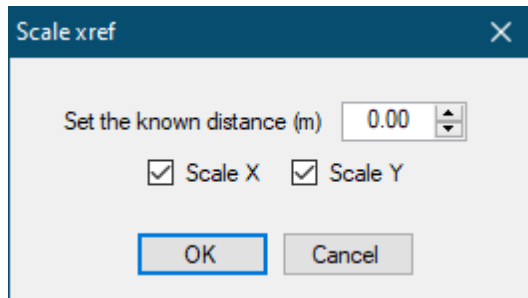
- **New XRef:** This button will initiate a browser window, and lets the user to insert an external reference to the drawing area. CAD files, PDF files and image files can be imported by using this option.



- **Bind XRef:** This button will bind the selected external reference to the CAD Plan Design. Once an external reference is bound, it becomes a part of CAD Plan Design drawing and the user may modify it. However, it will not be considered as an external reference any more.
- **XRef Settings:** This button will list all the external references. User may reload external references individually by using Refresh button, or collectively by using the Reload all xref button. Purge button will remove any unused entities from the selected external reference. Change button will open a browser window where the user may select a different file to replace the current file. Hide button will hide the selected external reference. Delete buttons will delete the selected external reference.

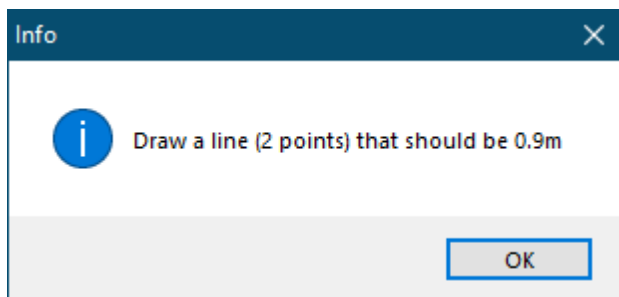


- Scale XRef: This option will display the Scale xref window. Using the tick boxes at the lower area it is possible to select to scale X and Y axes individually.

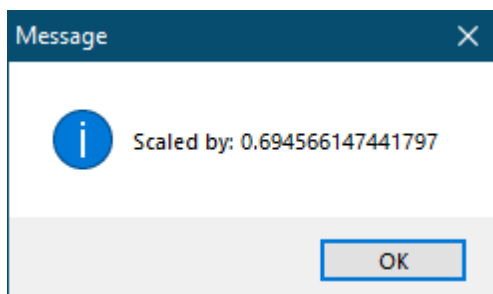


Scaling works as stated below:

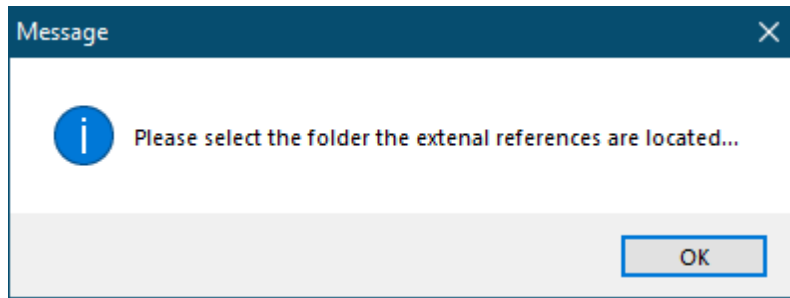
- A known length must be type in to Set the known distance (m) field. After OK button is clicked,
- two point must be selected from the drawing area, for the aforementioned length



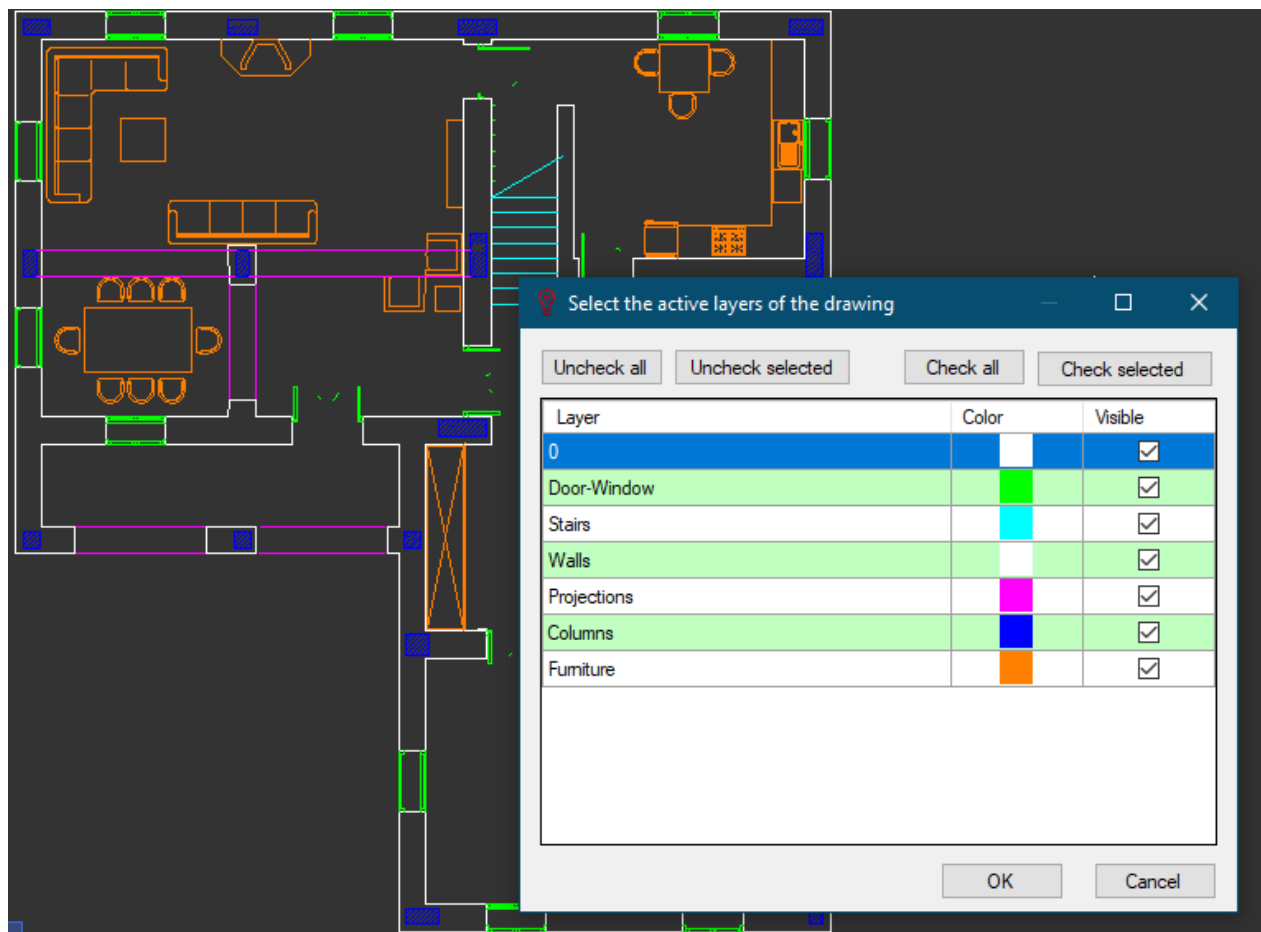
- Once the second point is selected, CAD Plan Design will scale the external reference so the selected two points correspond to the length defined at first step



- Fix XRefs: If the external references are moved to a different location, then, this option will let the user to update a new location for external references.

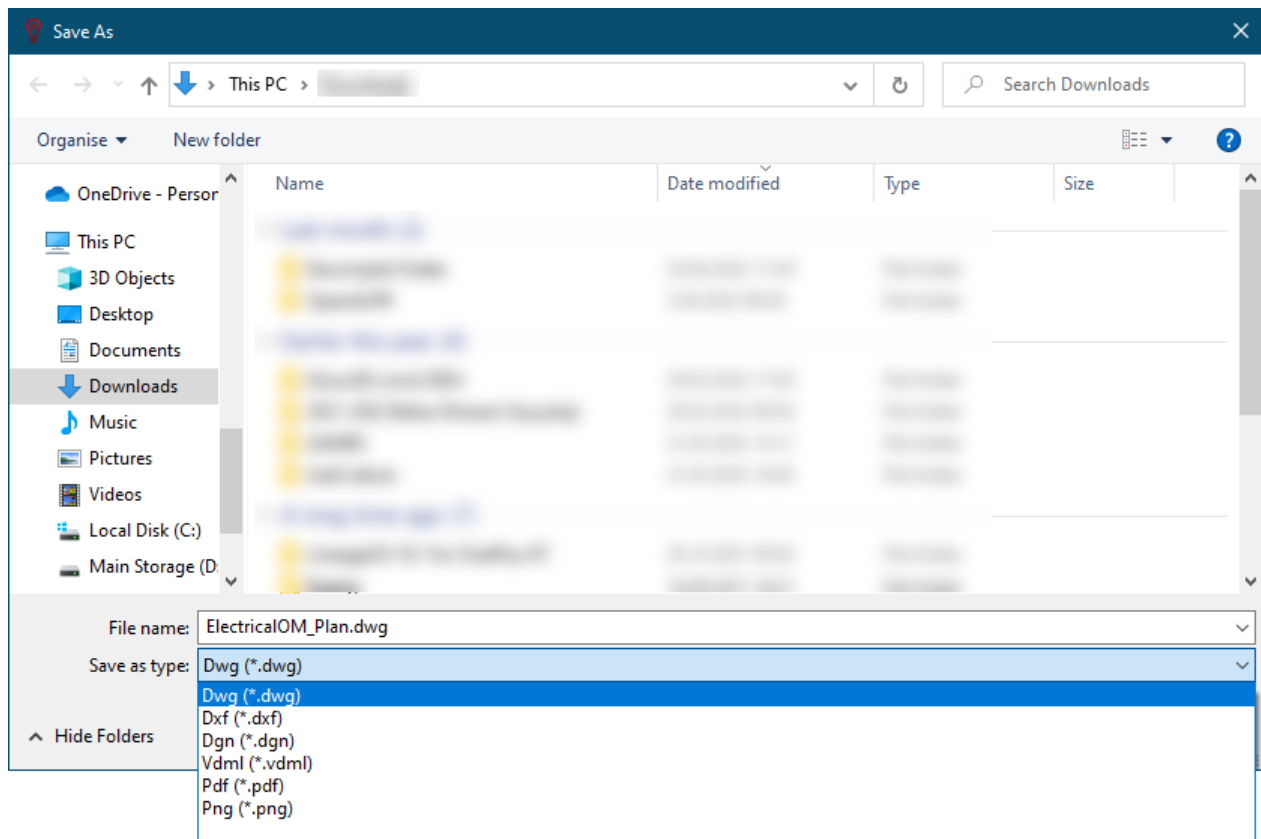


- XRef Layers: This button will display the layer information embedded to the external reference, if applicable. Using the Select the active layers of the drawing window, it is possible to modify colours, and choose to display or hide individual layers using the Visible tick boxes.

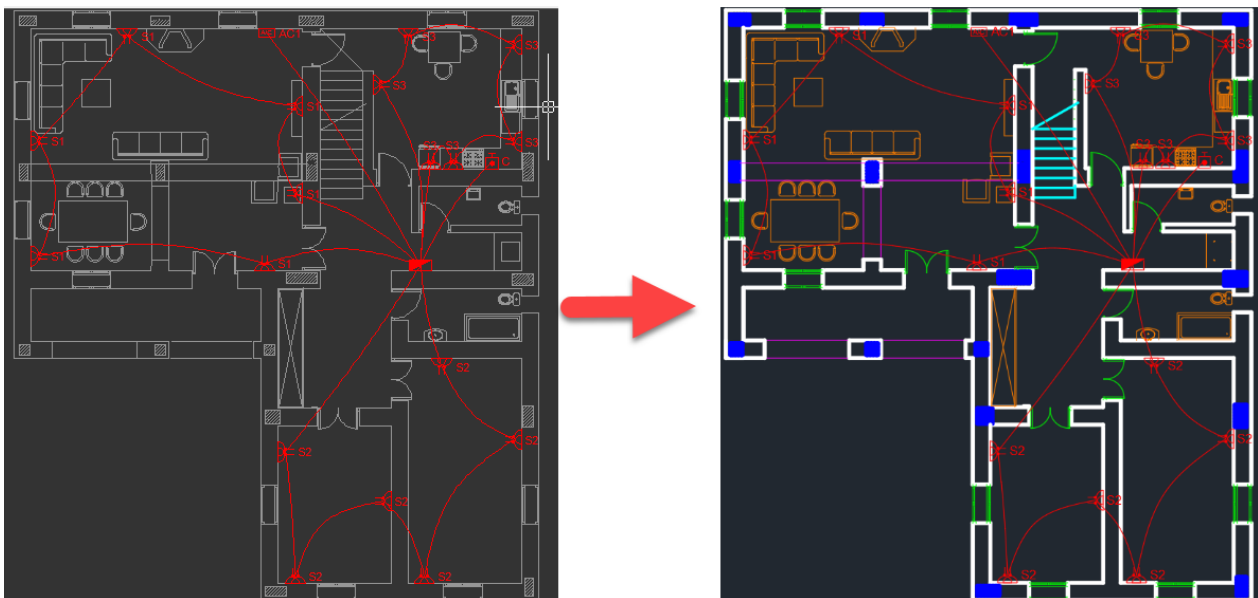


Export and Print Settings

- Export drawing: This will initiate the Save As dialogue window where the user can select a location and a file extension for the exported file. Export drawing will export what is shown at the drawing area of CAD Plan Design and nothing else. So, before exporting, user must select the appropriate layer from the Pages tab.

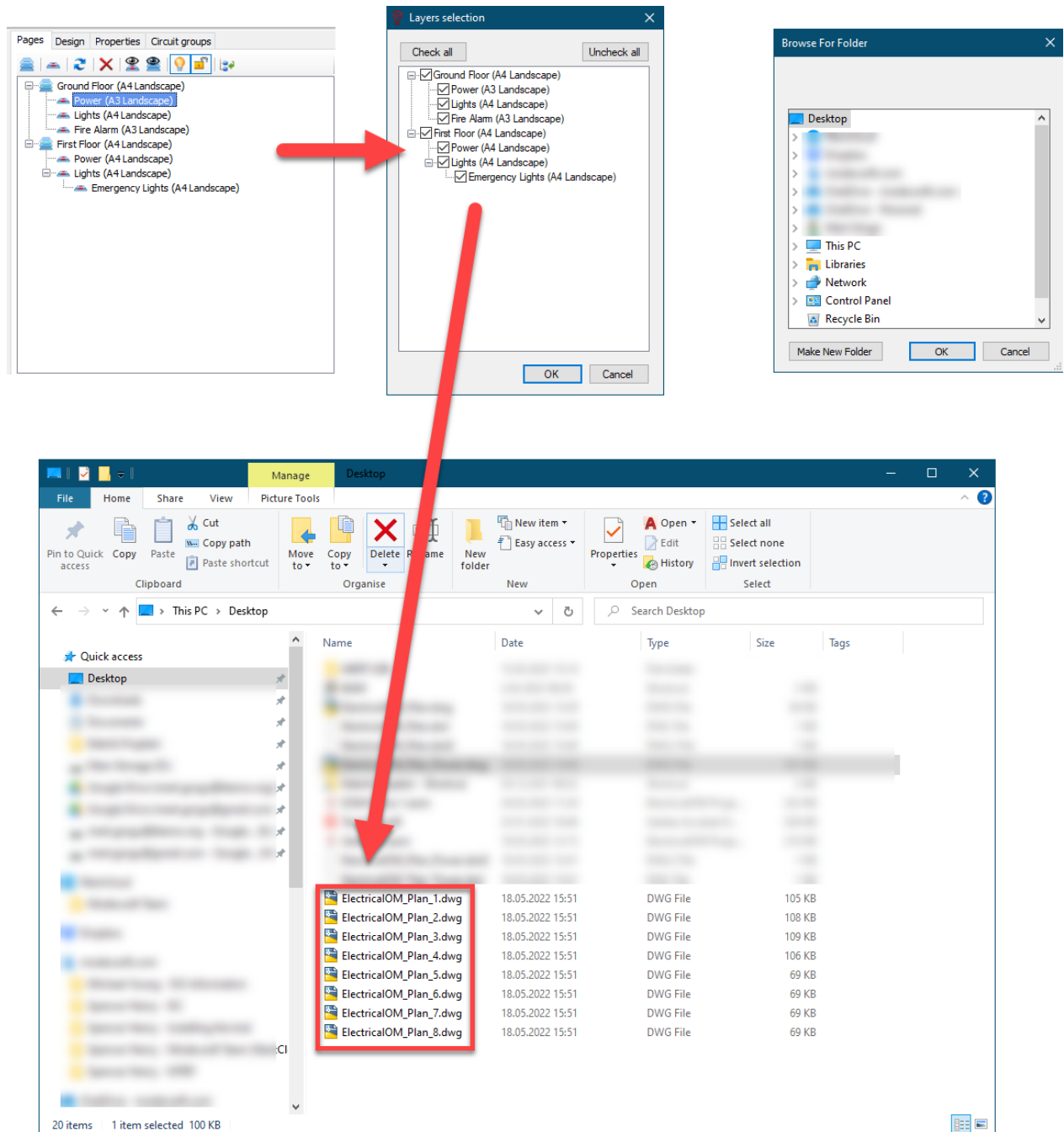


While exporting the drawing, original layer data, if applicable, will be maintained. As an example, the original layer colours, line weights, etc. are kept as seen on the right hand side below.

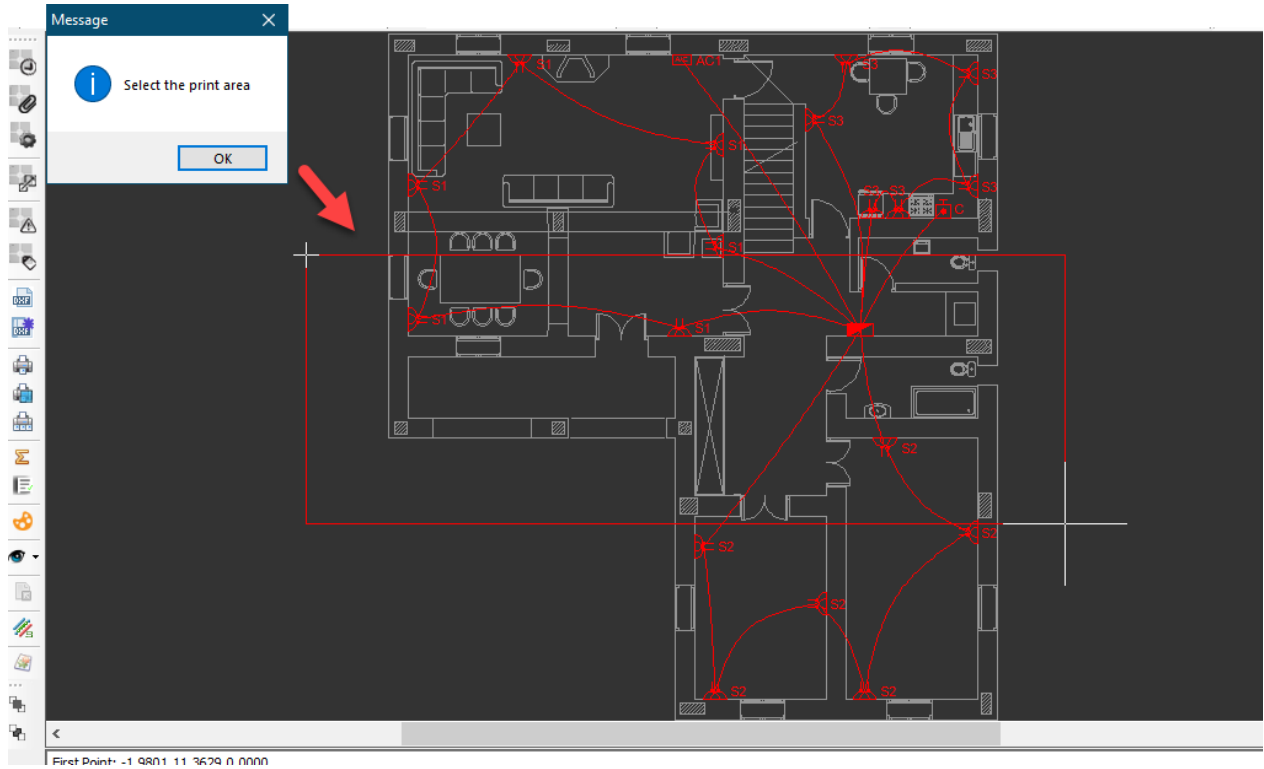


- Export selected: Similar to Export drawing, this option will also export CAD Plan Design drawings, however, this option will let the user choose layers using tick boxes.

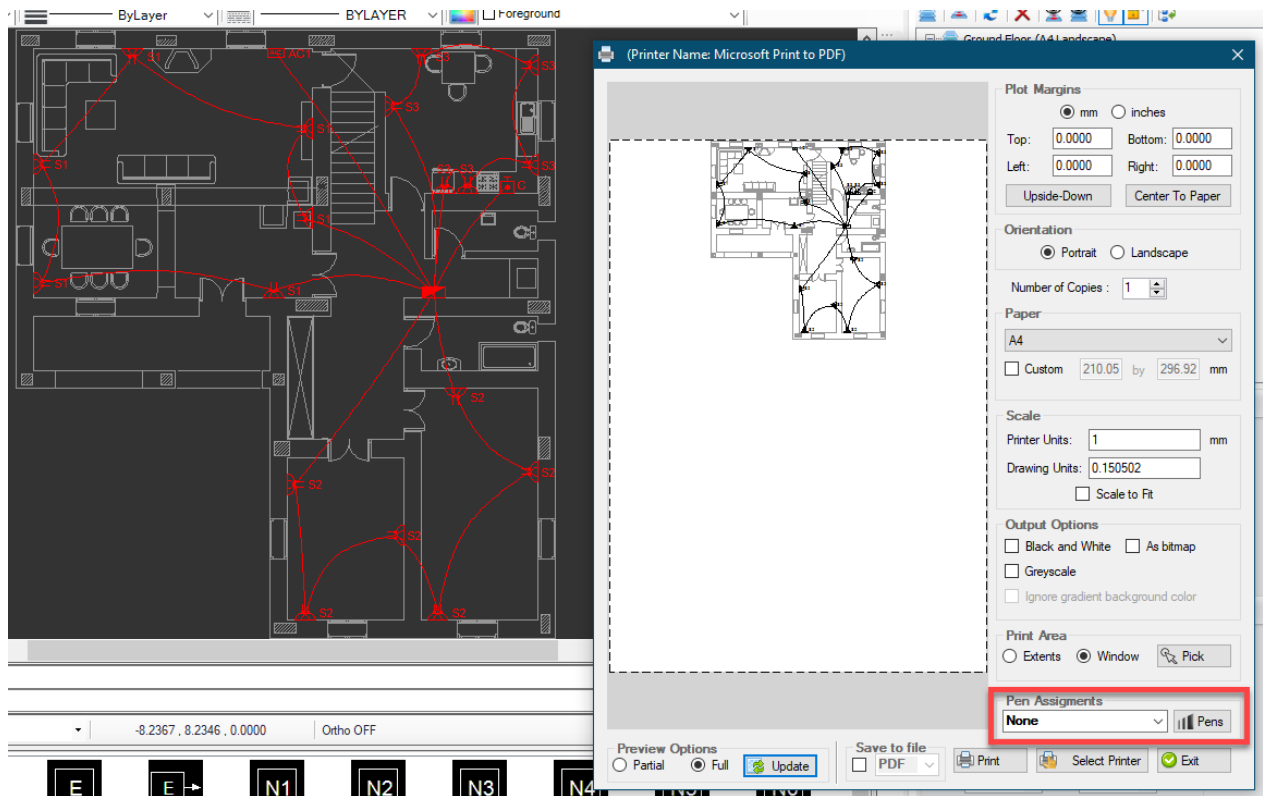
Once the required layers and location for output files are selected, CAD Plan Design will export each layer as a separate .dwg file. Exporting will consider CAD Plan Design module's layer visibility settings.



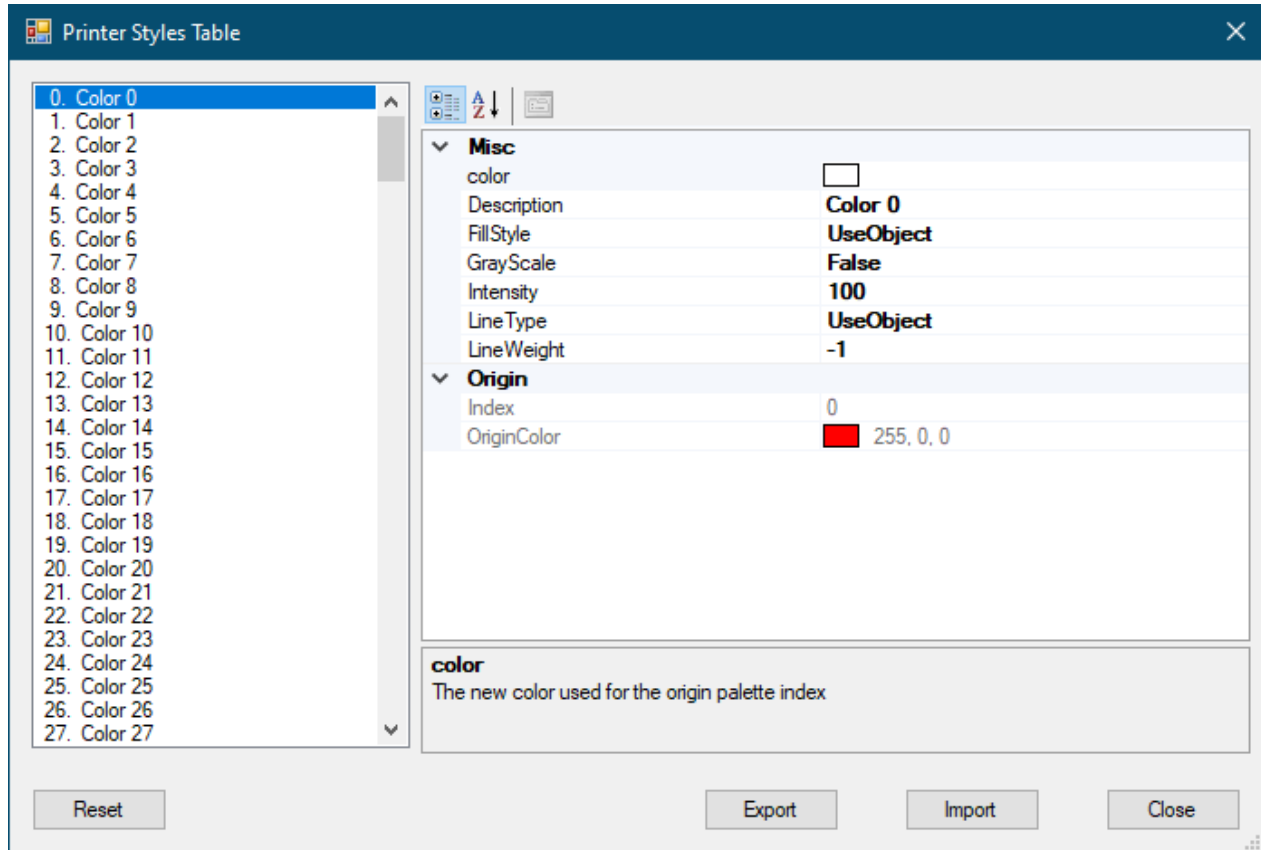
- Print page: This button will open up a print preview window with the current visual at the drawing area of CAD Plan Design module.
- Print page window: This button will display a Message window and expect the user to draw a rectangle which indicates the print area. After the rectangle is defined, the same print preview windows will appear.



- Print with settings: This button will display print options where the user may set various print options.



Print options window also lets the user to define custom pen setting. Once Pens button is clicked, CAD Plan Design will display the Printer Style Table window where settings related with each colour can be modified. Reset button will revert all the settings back to their original state.



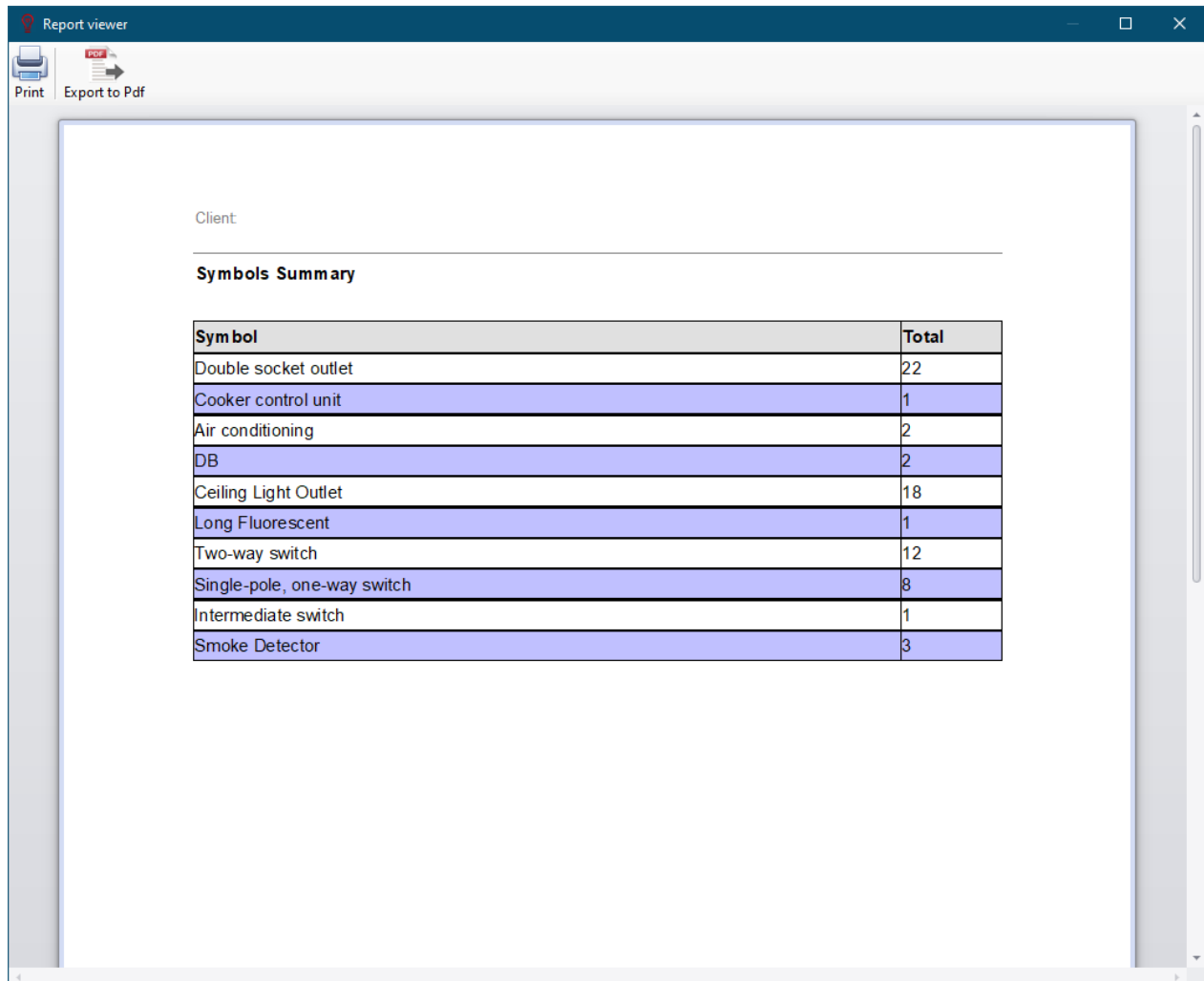
Symbols Summary

Symbols summary button will populate all the symbols used within the project as a list and display it. There are two formats to choose from: Vertical Summary, and Horizontal Summary. The difference is shown below.

Symbols Summary	
Symbol	Total
Double socket outlet	22
Ground Floor - Power	18
First Floor - Power	4
Cooker control unit	1
Ground Floor - Power	1
Air conditioning	2
Ground Floor - Power	1
First Floor - Power	1
DB	2
Ground Floor - Power	1
First Floor - Power	1
Ceiling Light Outlet	18
Ground Floor - Lights	13
First Floor - Lights	5
Long Fluorescent	1
Ground Floor - Lights	1
Two-way switch	12
Ground Floor - Lights	11
First Floor - Lights	1
Single-pole, one-way switch	8
Ground Floor - Lights	5
First Floor - Lights	3
Intermediate switch	1
Ground Floor - Lights	1
Smoke Detector	3
Ground Floor - Fire Alarm	3

Symbols Summary										
Page	Double socket outlet	Cooker control unit	Air conditioning	DB	Ceiling Light Outlet	Long Fluorescent	Two-way switch	Single-pole, one-way switch	Intermediate switch	Smoke Detector
Grand Totals	22	1	2	2	18	1	12	8	1	3
Totals Ground Floor	18	1	1	1	13	1	11	5	1	3
Power	18	1	1	1	13	1	11	5	1	
Lights										
Fire Alarm										3
Totals First Floor	4		1	1	5		1	3		
Power	4		1	1						
Lights					5		1	3		

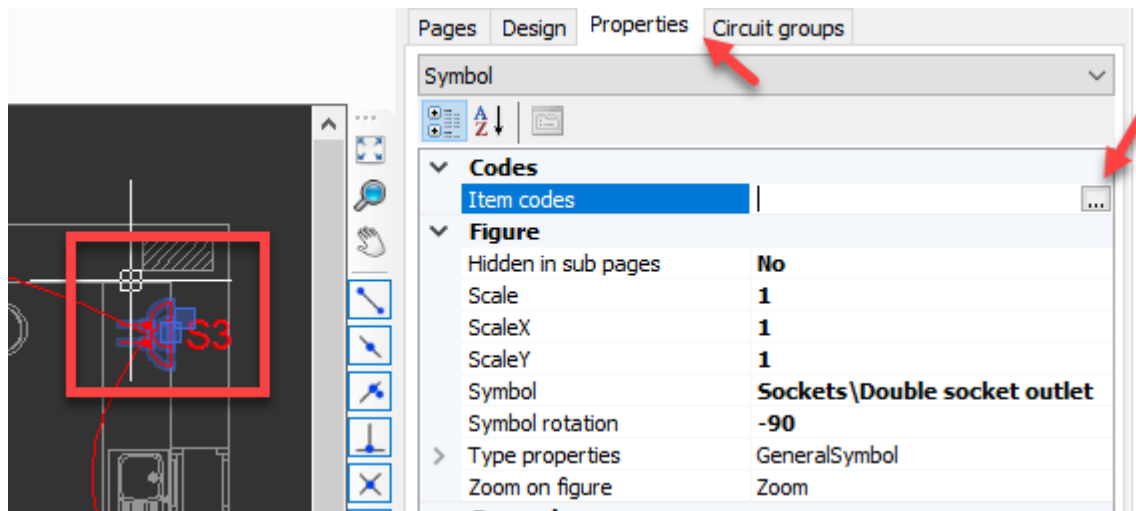
Export to Excel button will export the list as the selected format. Symbols Summary Report will display all the symbol used as a list, where it can be printed or exported to PDF using the appropriate buttons on the toolbar at the top.



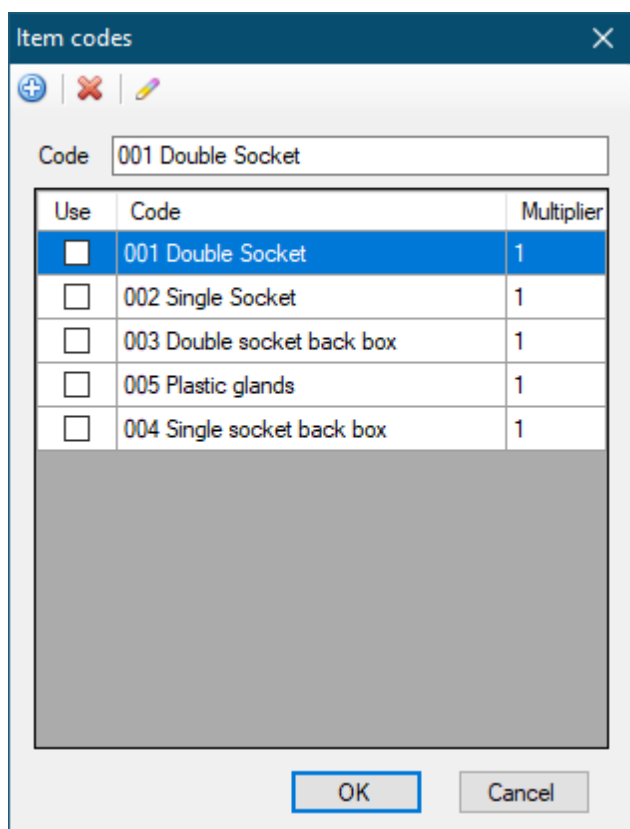
Material Lists

It is possible to create material lists within CAD Plan Design environment. This is a separate feature than ElectricalOM's Bill of Quantities, and it is independent from ElectricalOM data.

In order to create a material list, Properties tab's Item codes field is used. A single item code, or multiple item codes can be assigned to every individual symbol, which means if the user assigns codes to a symbol, this will not be applied to any other symbol not even to other occurrences of the same symbols.

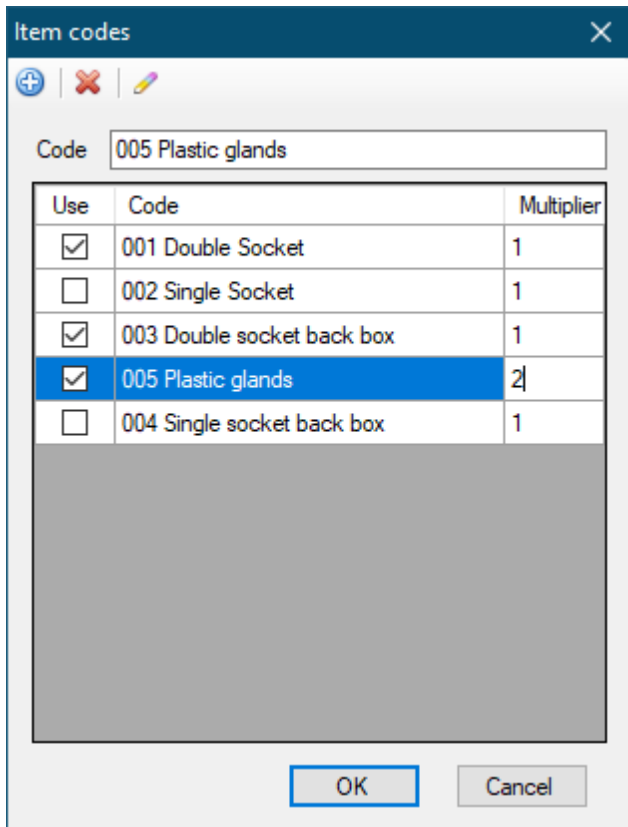


Three dots button of Item codes field will initiate the Item Codes List window. Toolbar accommodates Add, Delete, and Rename buttons. These buttons will add a new code to the list, delete a code from the list, and rename a priorly selected code. Use tick boxes can be used to assigned each item to the selected symbol. Multiplier column can be used to define a quantity for the related item. Multiplier value will only be saved for the selected symbol, and will be reset to one for other occasions.

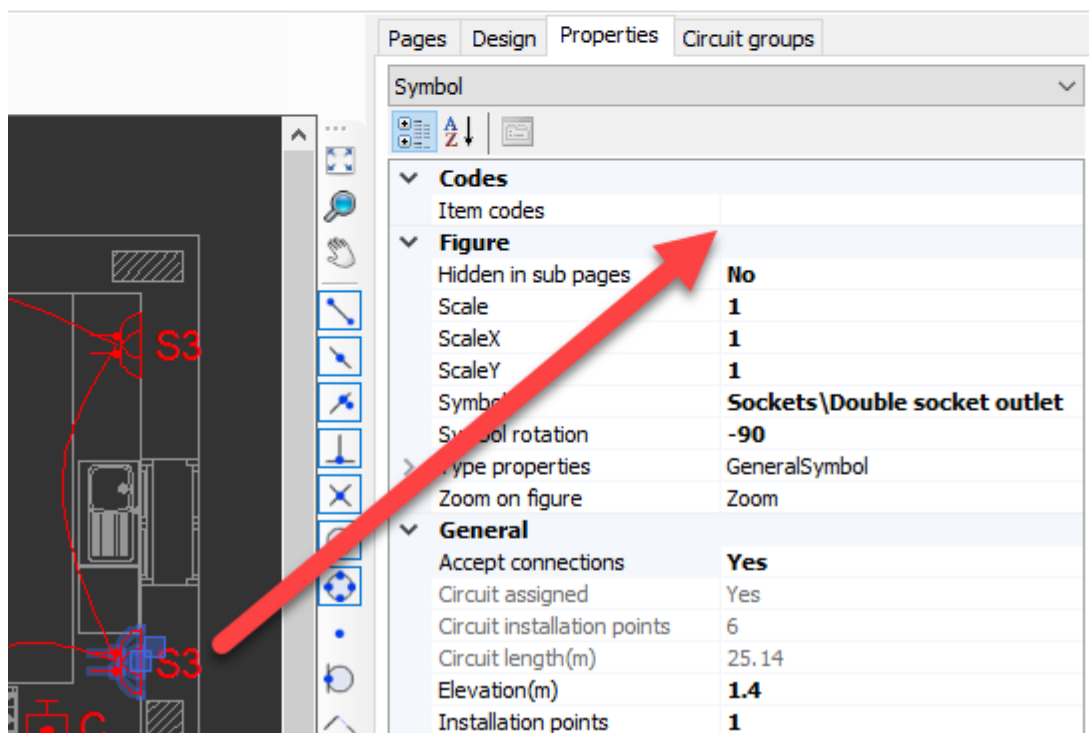
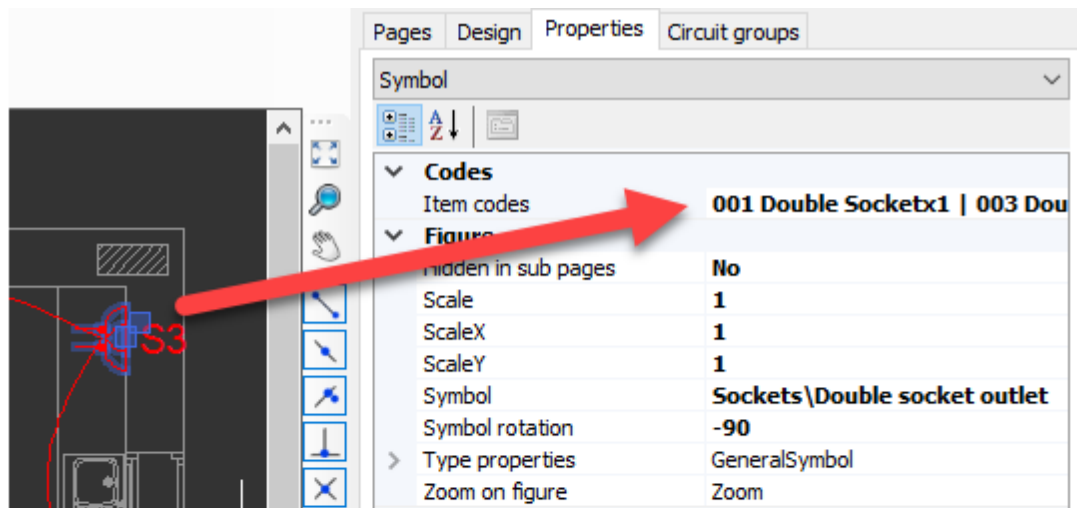


As an example, we created various items as seen above. We will assign 001, 003, and

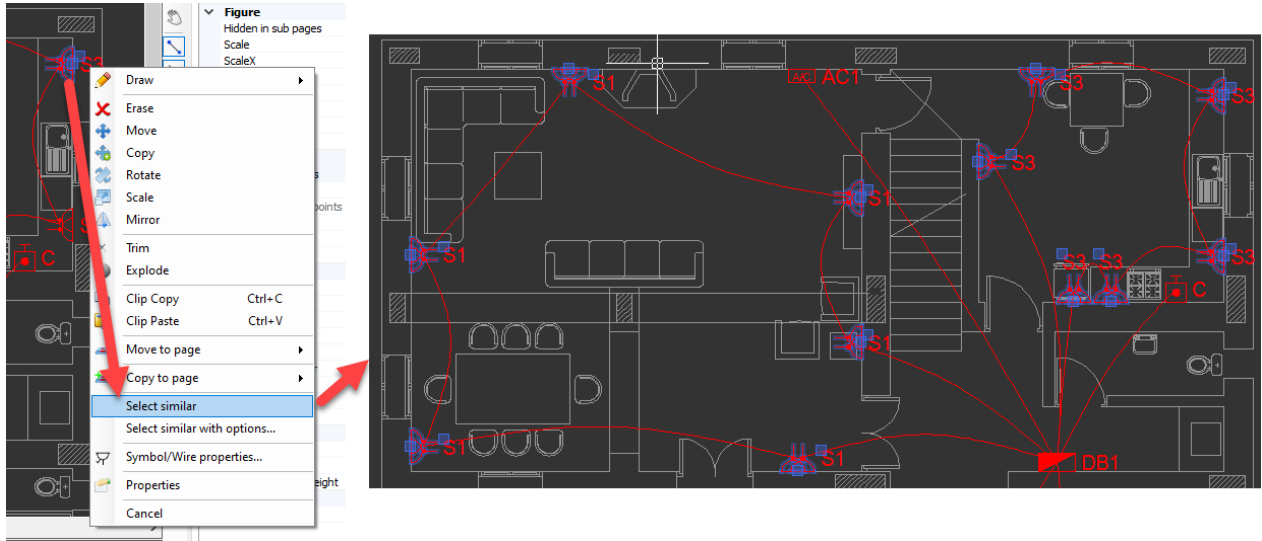
005 for a double socket symbol. We also will define a multiplier, 2, for 005.



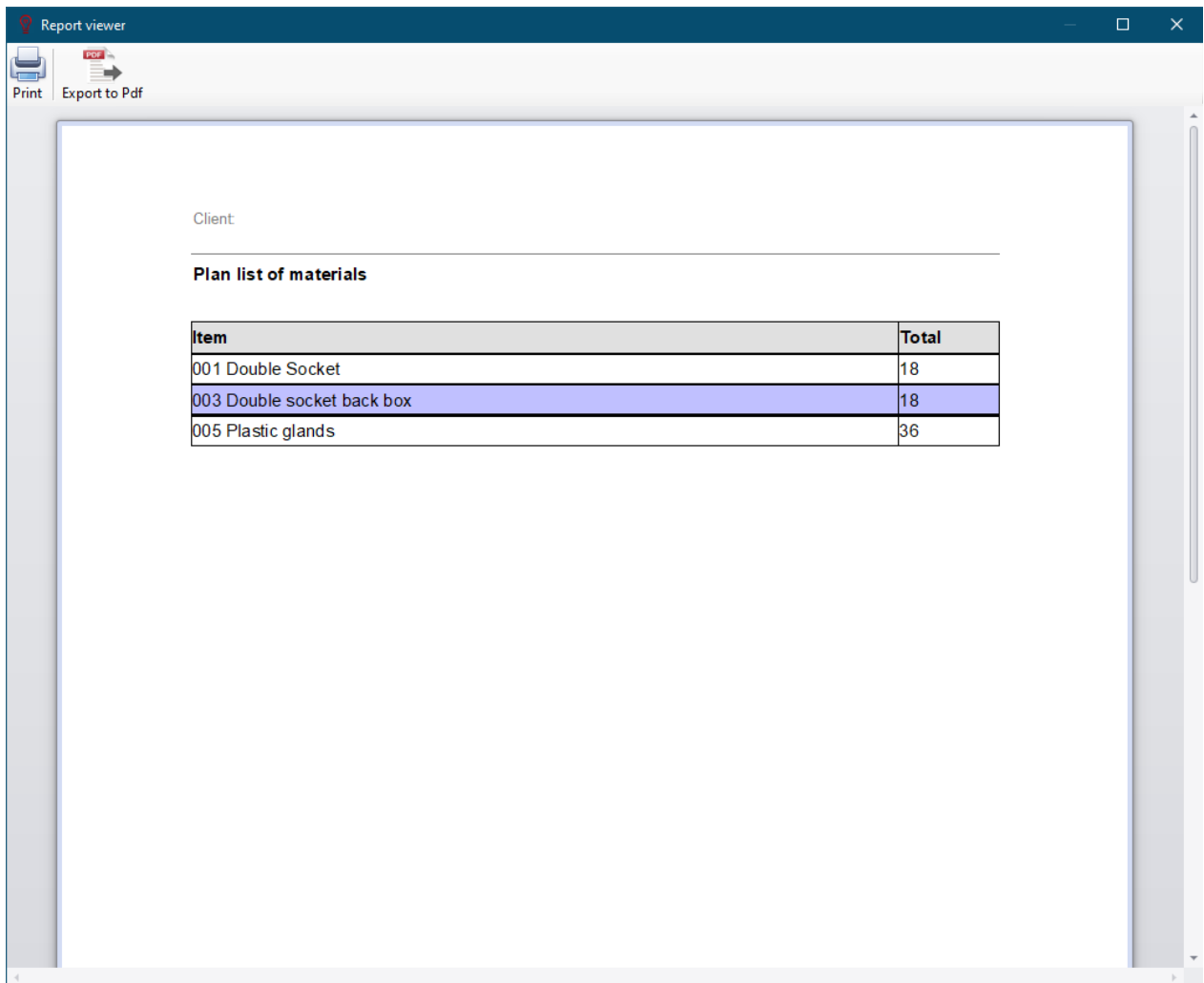
It can be seen that this is applied only to the selected symbol but no other occurrences of the same symbol.



As a note, it is possible to select all the symbols which are the same using Select similar option from the right-click menu and assign items for all of them at the same time.



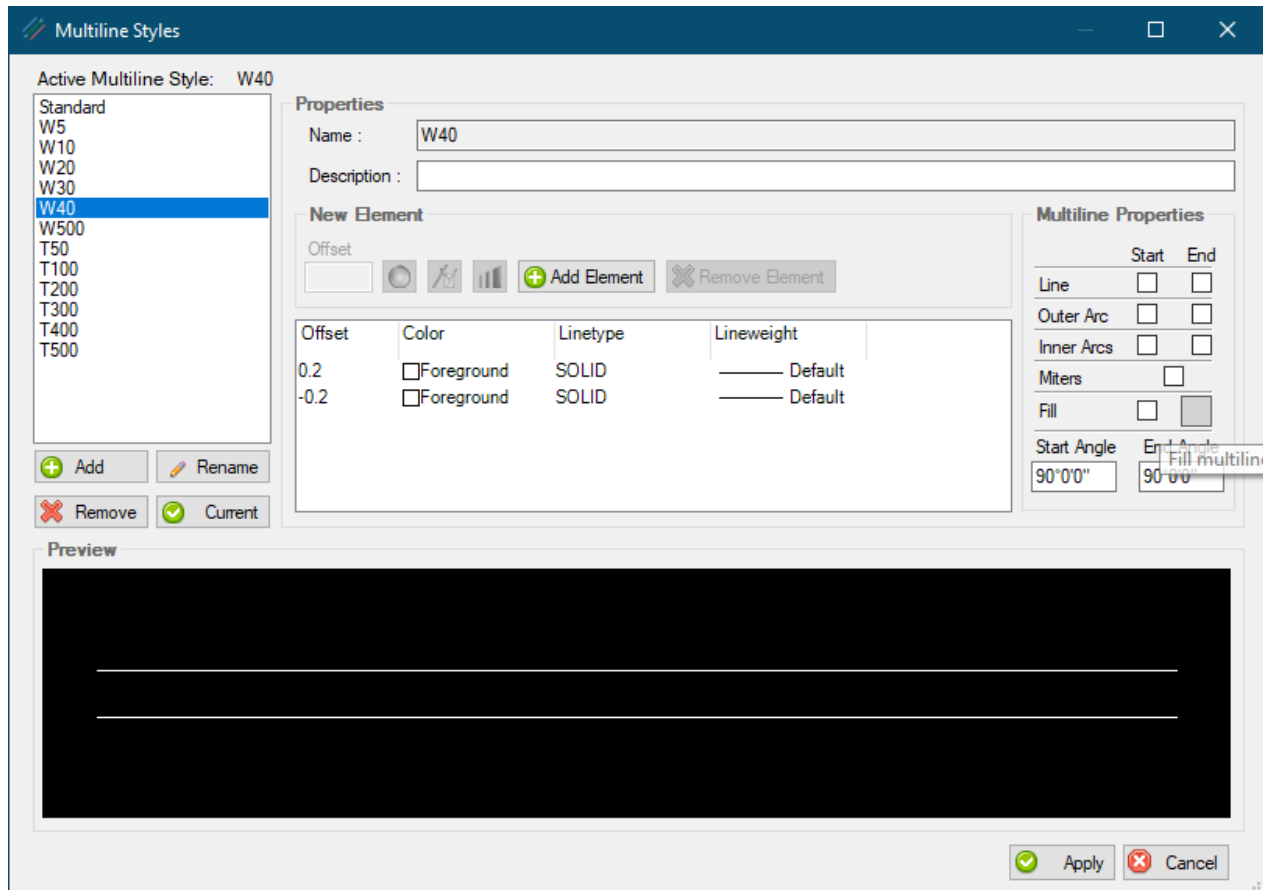
Once item codes are defined and assigned to symbols, Materiel List option can be used to display items used within he project. Print and Export to Pdf button can be used to print, and export the list as Pdf.



Multiline Settings

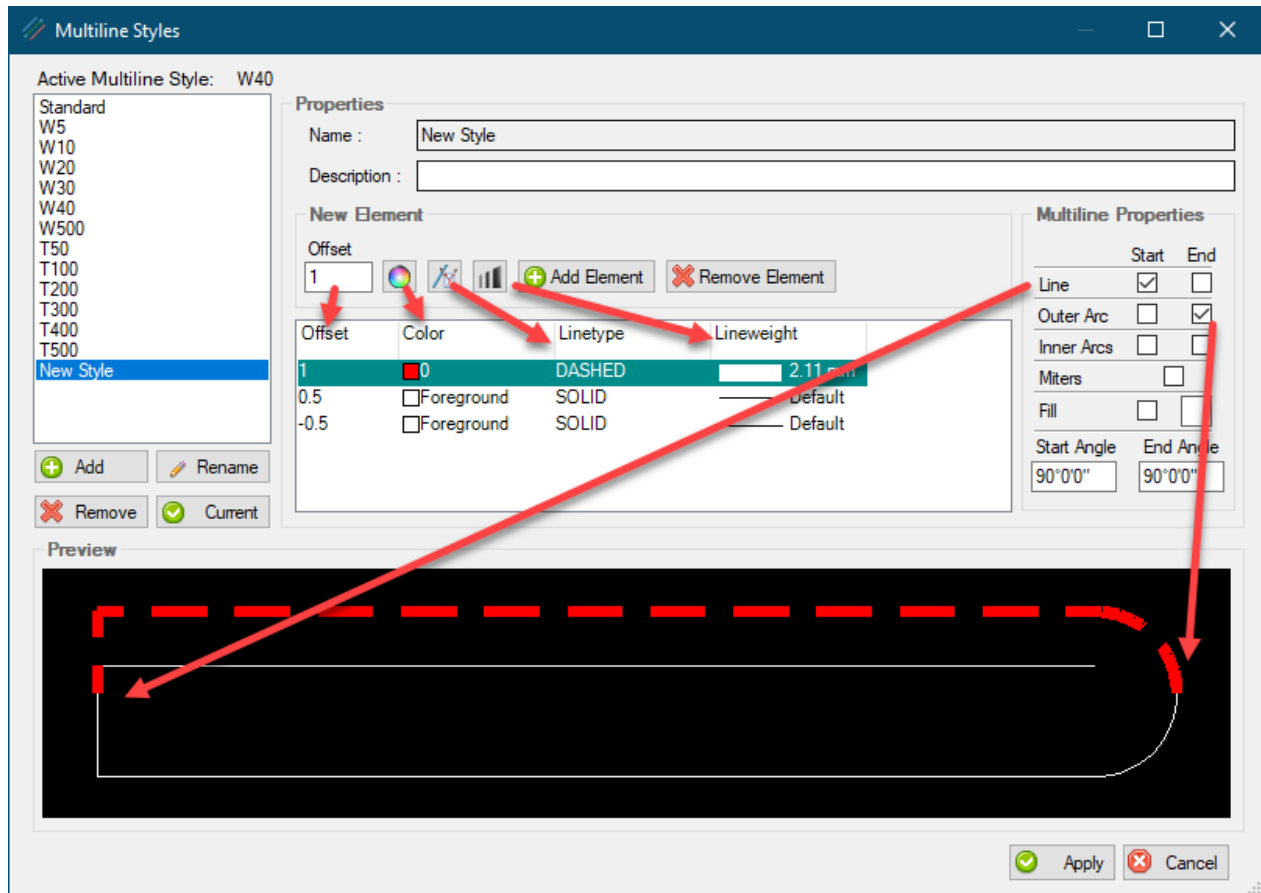
multilines are basically used to draw multiple lines at once to represent various architectural entities, however, in context of electrical design, they are also used to represent cable trays which makes the multiline settings quite important.

multiline styles button will initiate the multiline Styles window. This window is used to define different multiline styles to be used within the CAD Plan Design project.



All available styles are listed on the left hand side list. On top of this list, current style is indicated. Below the list, there are various buttons. Add, and Remove buttons are used to add a new multiline style, or deleted one from the list. Rename button will let the user to rename the selected style. Current button will set the selected style as active style. Current style is used by Draw multiline button.

New element and Multiline Properties sections can be used to define or modify the appearance of a style.

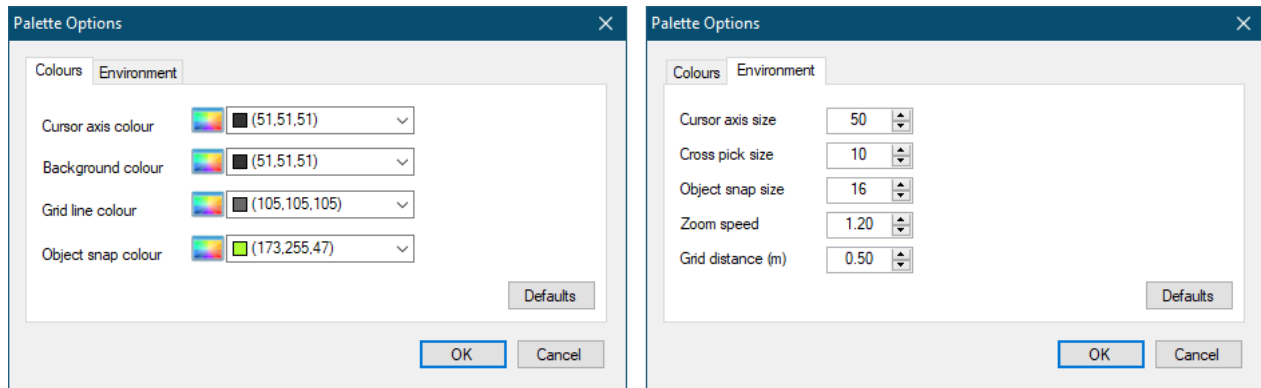


- Add Element button will add a new line to the style where Remove Element will remove the selected line from the style.
- Offset value will dictate the offset distance from the 0 point of the multiline.
- Colour, line type, and line weight options will define how the line will look.
- Tick boxes under the Multiline Properties section will add various feature to the multiline style.
- Each change will also be displayed within the Preview area as they are done.
- Apply button will apply the changes, however, this will not change the current style.

Other settings

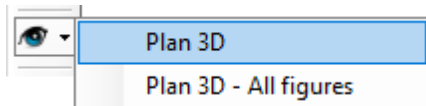
Palette options

Visual properties of CAD Plan Design module can be modified using Palette Options. Usage is straight forward, and settings can be reset by using the Defaults buttons.



Plan 3D settings

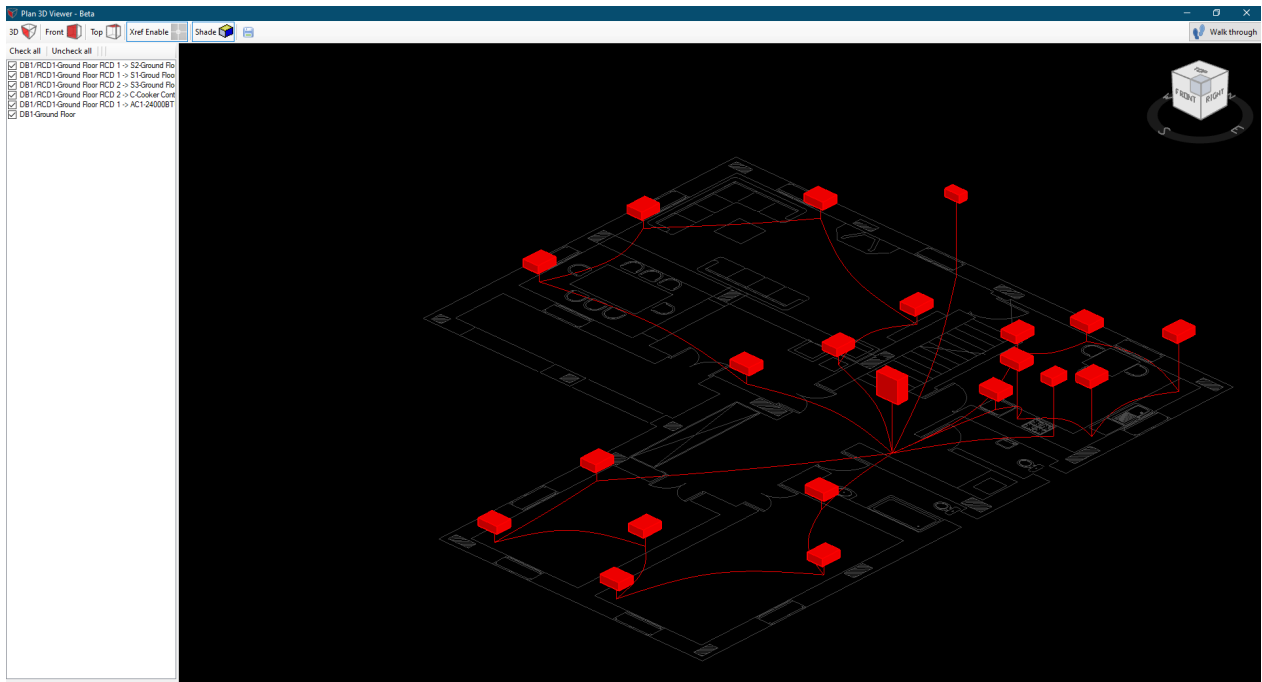
CAD Plan Design has the ability to display the drawing in 3D. The height information is gathered from the Properties tab. There are 2 options for ^3D view: First one, Plan 3D, will display only the electrical components of the selected layer, where the second option, Plan 3D - All figures will also display any non-electrical shapes.



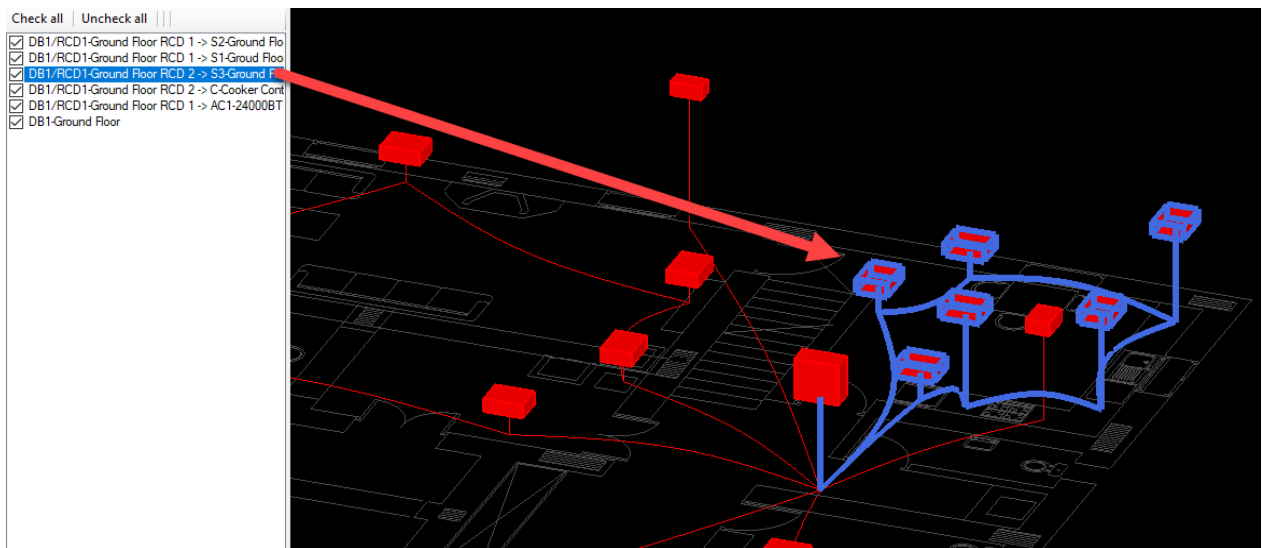
3D view window has a toolbar and a side panel.

Toolbar accommodates:

- 3D button: Switches to isometric view
- Front button: Switches to front view.
- Top button: Switches to top view.
- Xref enable button: Displays/Hides any external references on the selected layer.
- Shade button: Renders electrical components as a solid object.
- Save scene button: Saves current view as .jpg file.

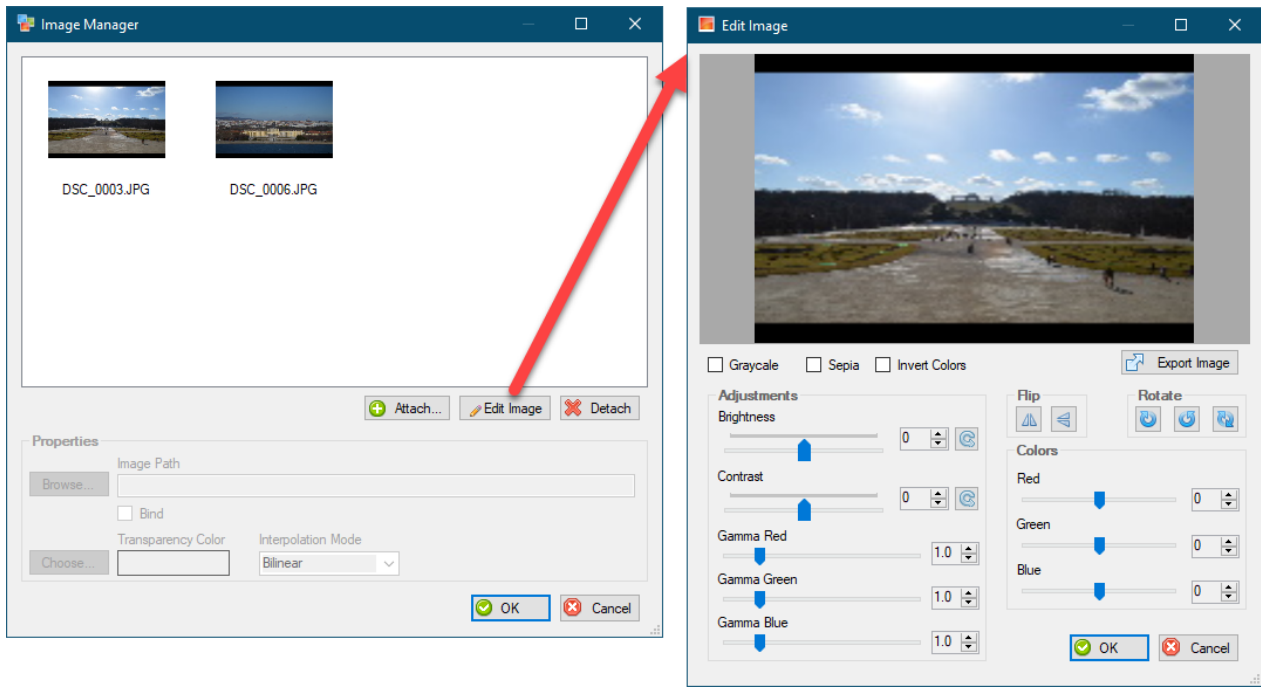


Side panel displays all the assigned circuits on the selected layer where they can be shown/hidden by means of tick boxes, and highlighted by clicking on them.

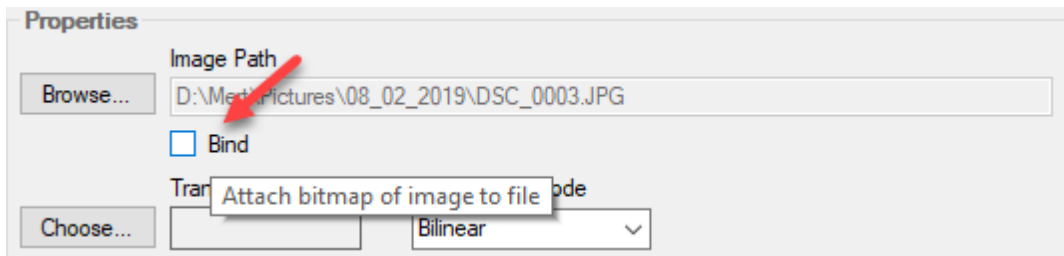


Images

This option enables the user to attach images to the ElectricalOM file. Once clicked, this button will initiate the Image Manager. All the images used on the CAD Plan Design drawing will be displayed here, and they can be edited using a built in image editor, and removed from the drawing data using the Detach button. However, Image Manager does not insert or removed any images from the actual drawing.



CAD Plan Design also lets users to attach any image to the CAD Plan Design project, and bind it to the project file. Binding an image will save the image directly to the project file eliminating requirement of additional image files. Images attached to the project do not need to be used on the drawing.



Bring Front/Send Back

When multiple external references/images are inserted on the drawing, they may be arranged overlapping. In order to arrange external references/images these two buttons become useful. Bring front will move the selected item over other items, where Send back will move the selected item under other items.

Toolbar No.4



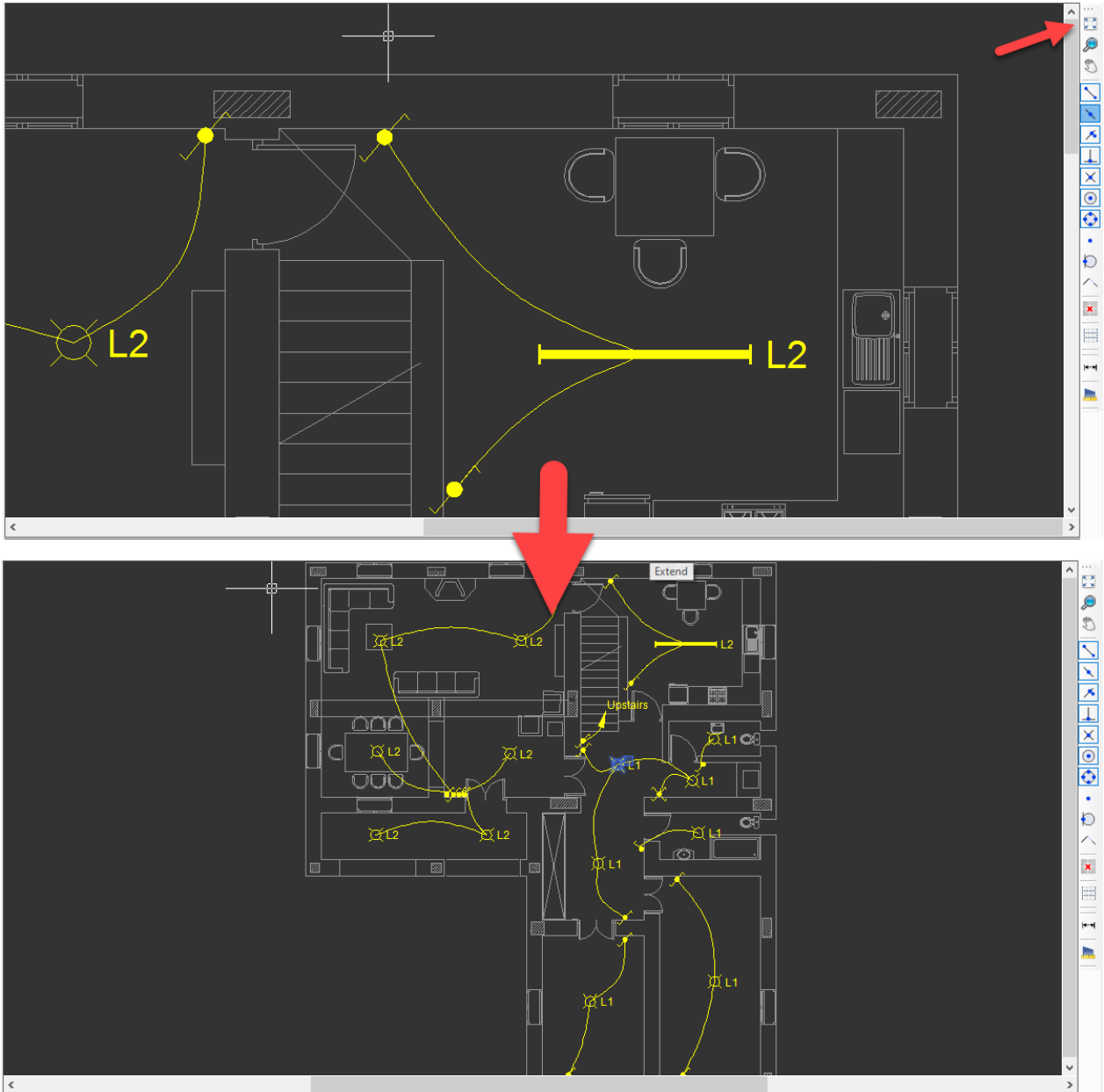
Toolbar 4 is located on the right hand side of the drawing area. It is mainly focused on zooming, object snap settings, and dimensions. For further detail refer to specific chapters.

Zooming



First three options are used to zoom in to, zoom out of, and pan the drawing area.

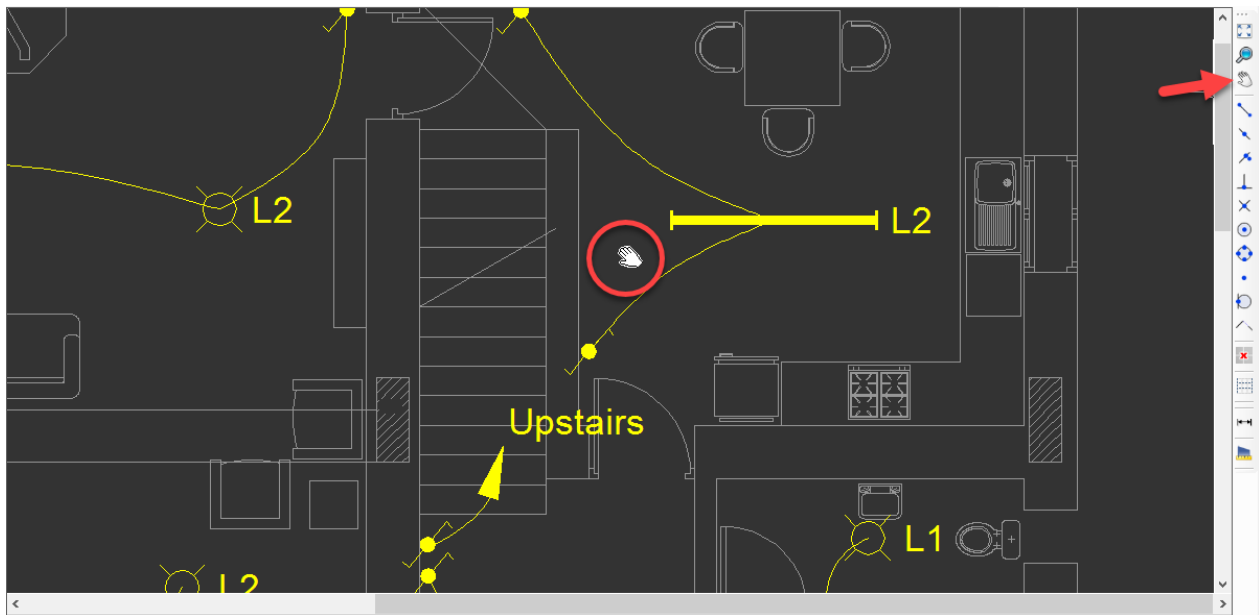
- Zoom All button will zoom in or out to fit the whole drawing to the drawing area limits.














- Zoom Window will expect the user to select a region to fill the drawing area.



- Pan will let the user to navigate the drawing by using the mouse. The mouse cursor will be change to a hand icon to indicate the pan mode. Esc key will exit the panning mode.



Object Snaps

-  Object snaps are used to select accurate points of objects while drawing. They can be
-  enabled, or disabled individually by clicking on their respective buttons on toolbar no.4.
-  Active snaps will be indicated by a blue frame on the button. Object snap function can
-  also be activated or deactivated completely by using the F3 key of the keyboard.
- 
-  The snaps will only be active while in drawing mode and will not interfere in other cases.
- While in drawing mode, active object snap points will be indicated by various shapes in green colour by default. The colour can be changed using the Palette options button,
-  see [Toolbar No.3, Other settings](#).
- 
- 
- 
- 

CAD Plan Design also indicates snap points on locked layers by default, however, this can be disabled by clicking on Disable object snaps on locked layers button, the last button on the toolbar.

Object snaps will not be indicated on external references.

Available object snap points are, in order from top to bottom:

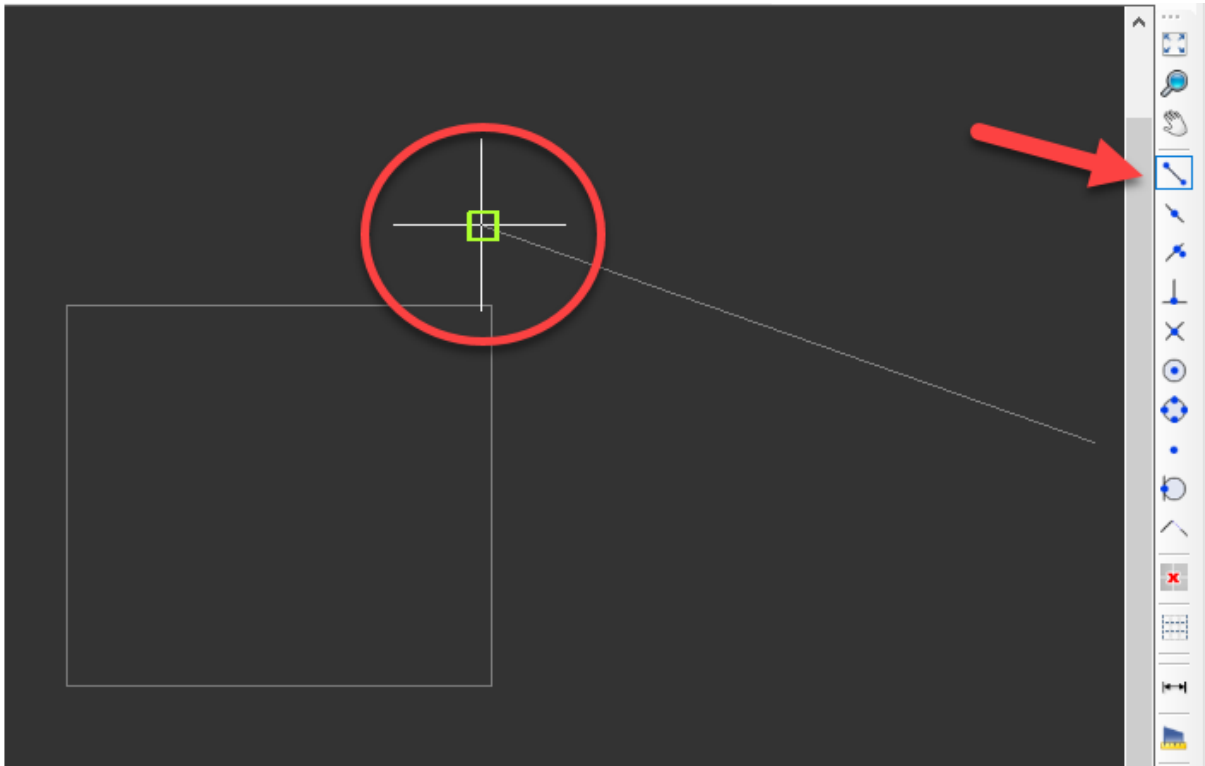
- [End Point](#)
- [Mid Point](#)
- [Nearest Point](#)
- [Perpendicular Point](#)
- [Intersection Point](#)

- [Centre Point](#)
- [Quadrant Point](#)
- [Node](#)
- [Tangent Point](#)
- [Extension Point](#)

End Point

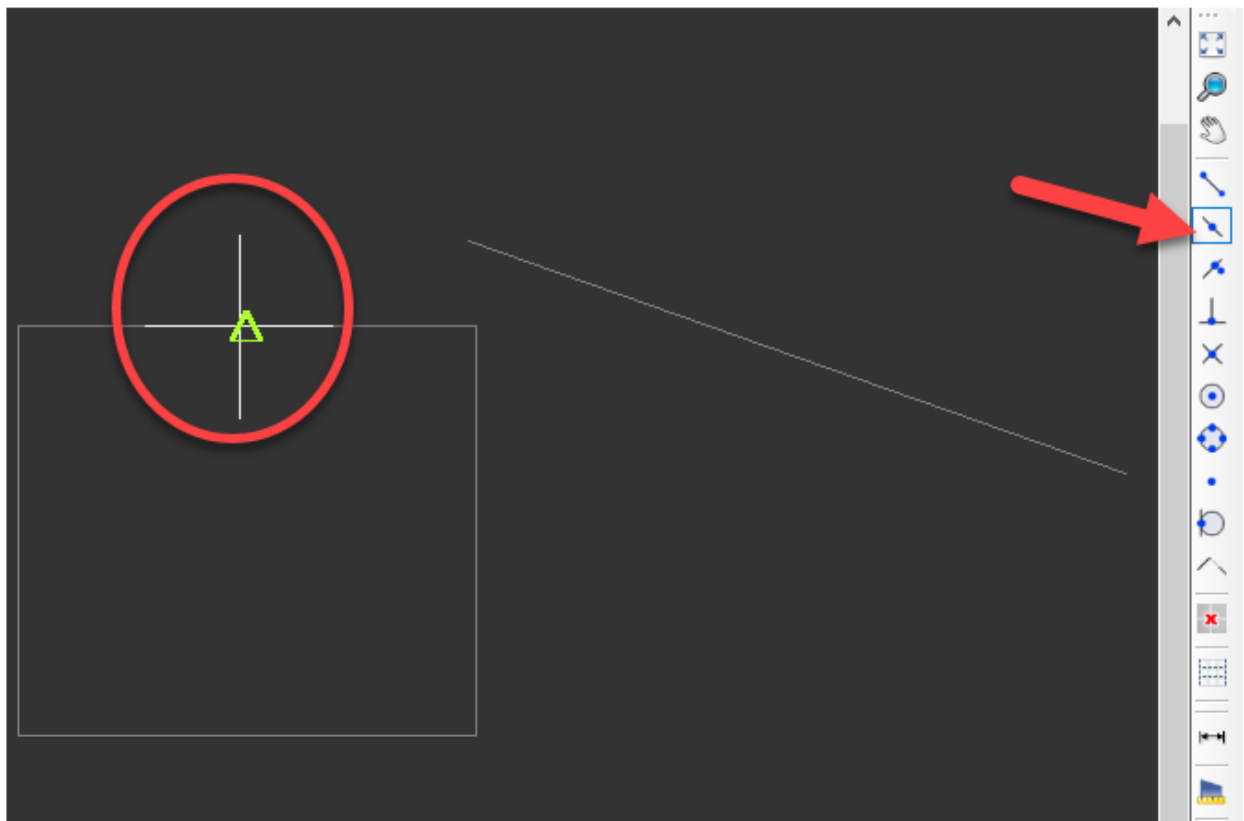
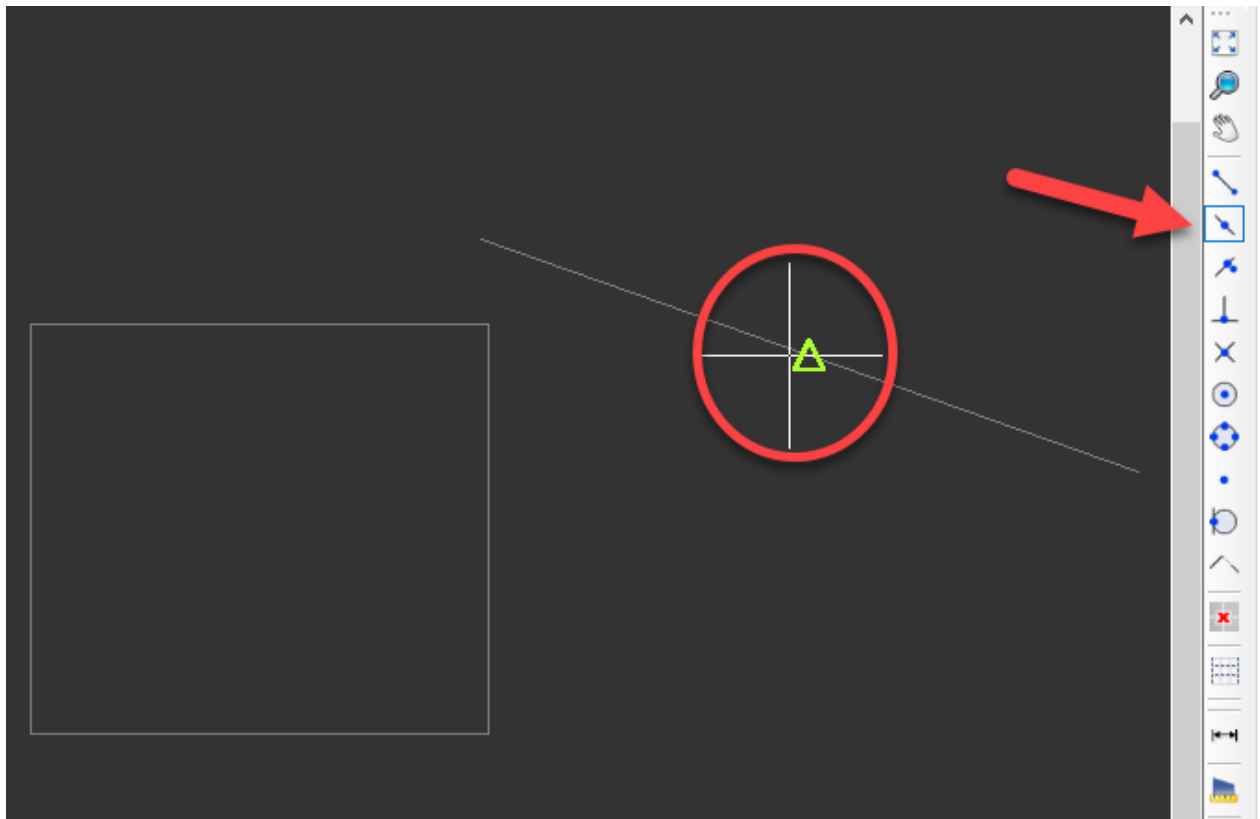
This option will indicate the end points of a line or a part of another object which is a line.

End point is indicated by a square.



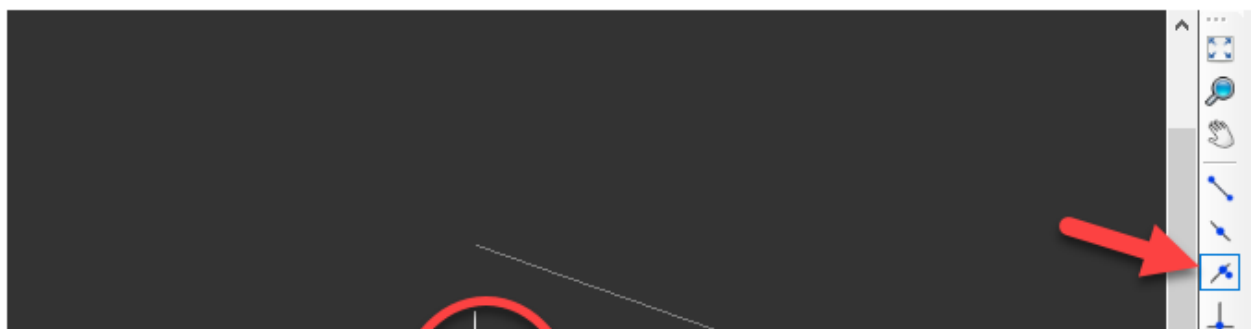
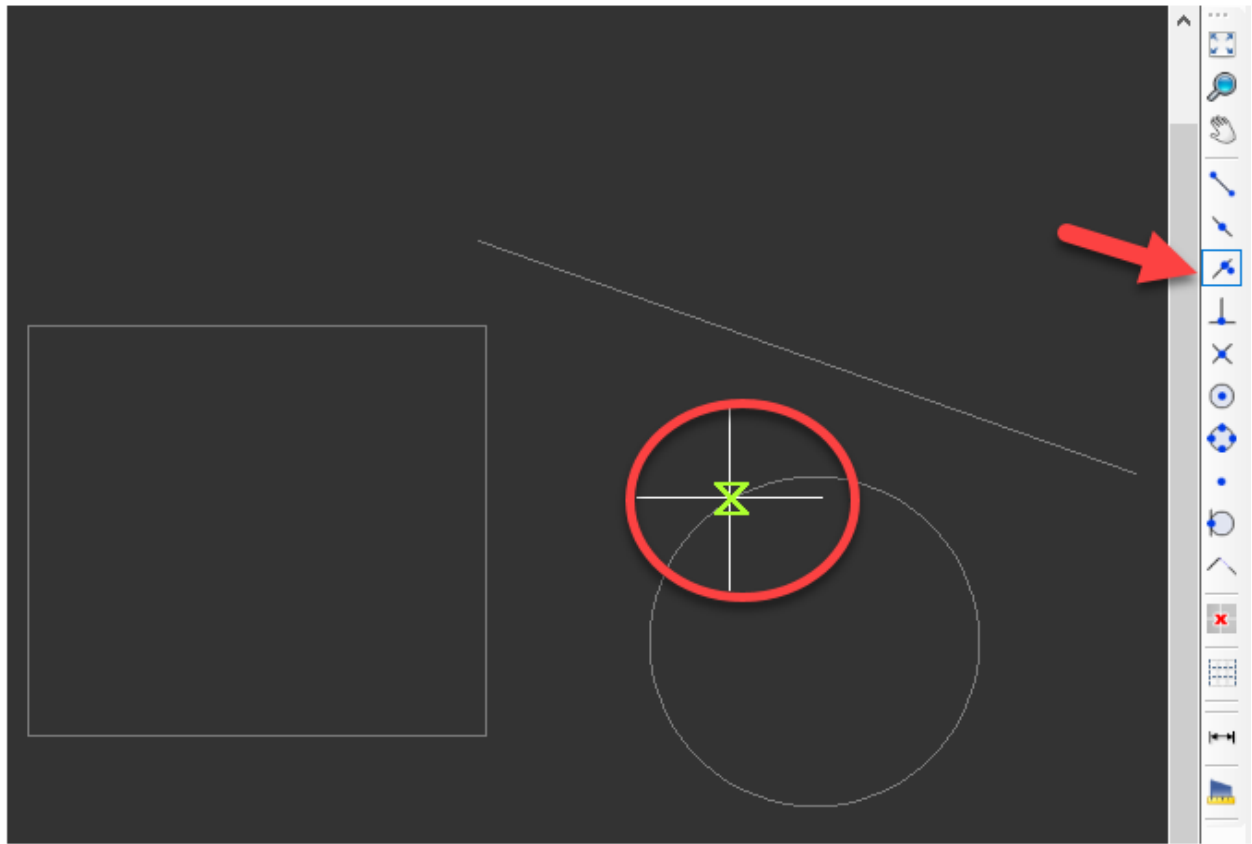
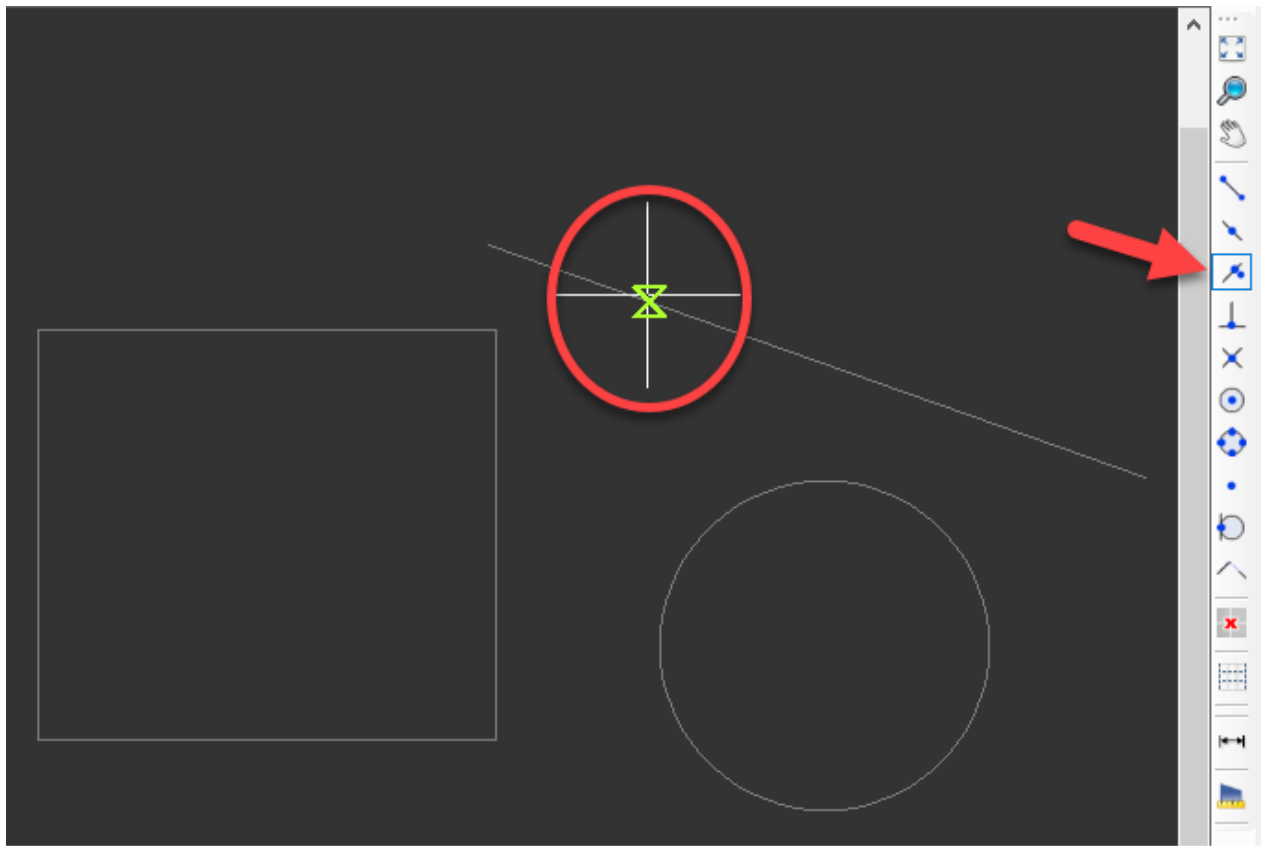
Mid Point

This option will indicate the middle point of a line or a part of another object which is a line. Mid point is indicated by a triangle.



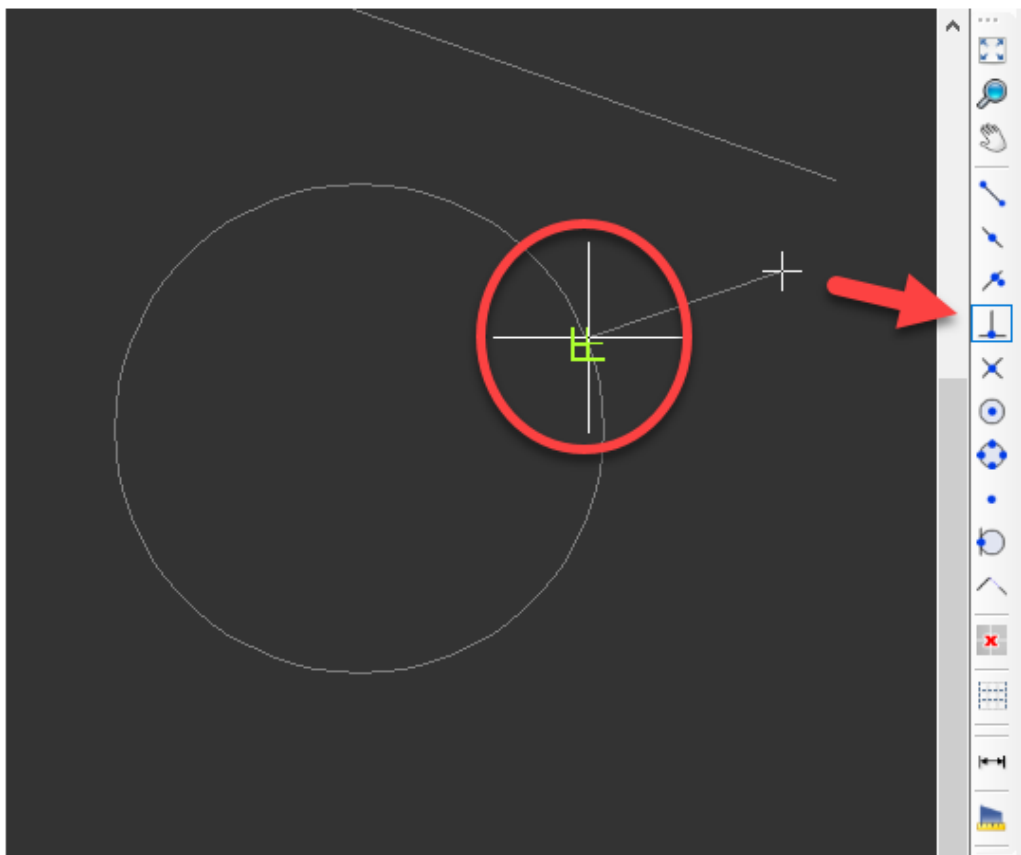
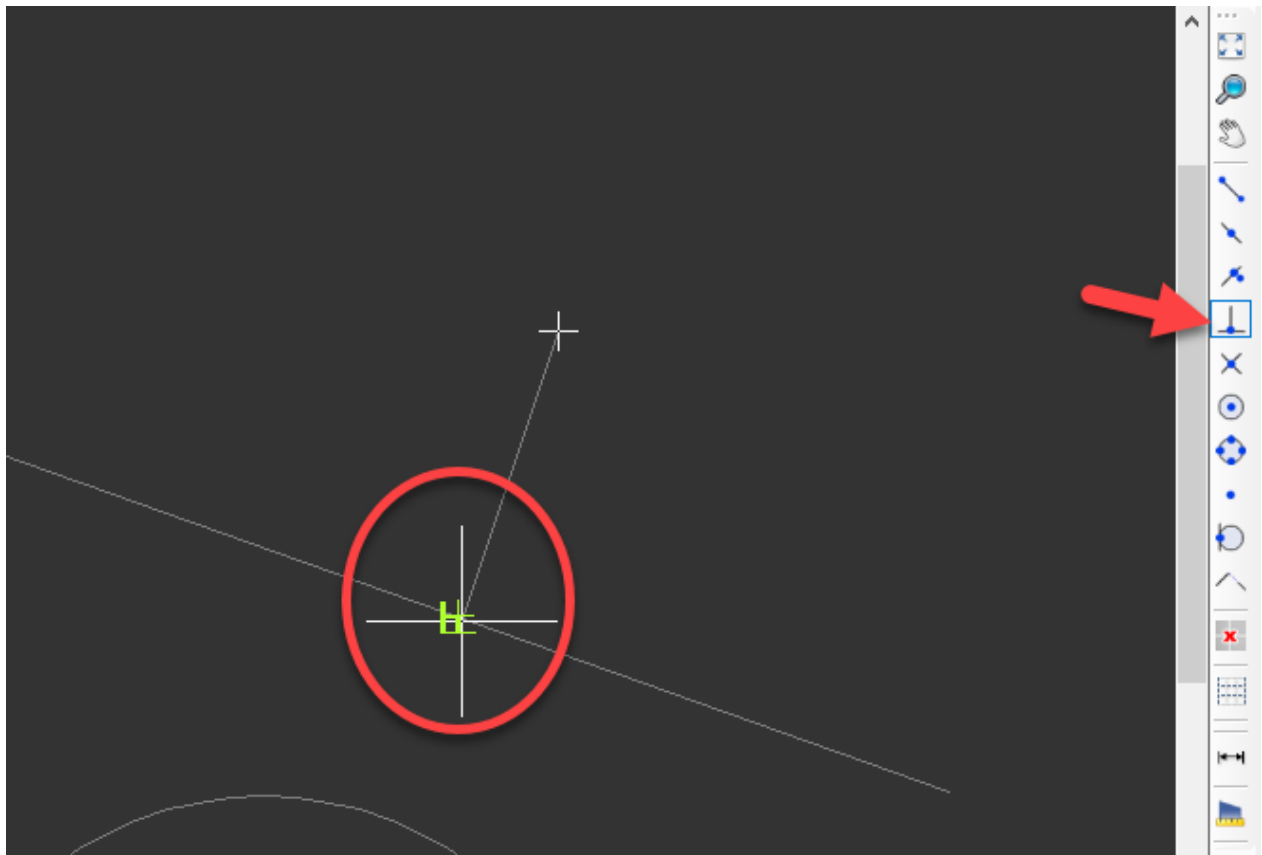
Nearest Point

This option will indicate all the points along an object which may be any shape. Nearest point is indicated by a hourglass shape.



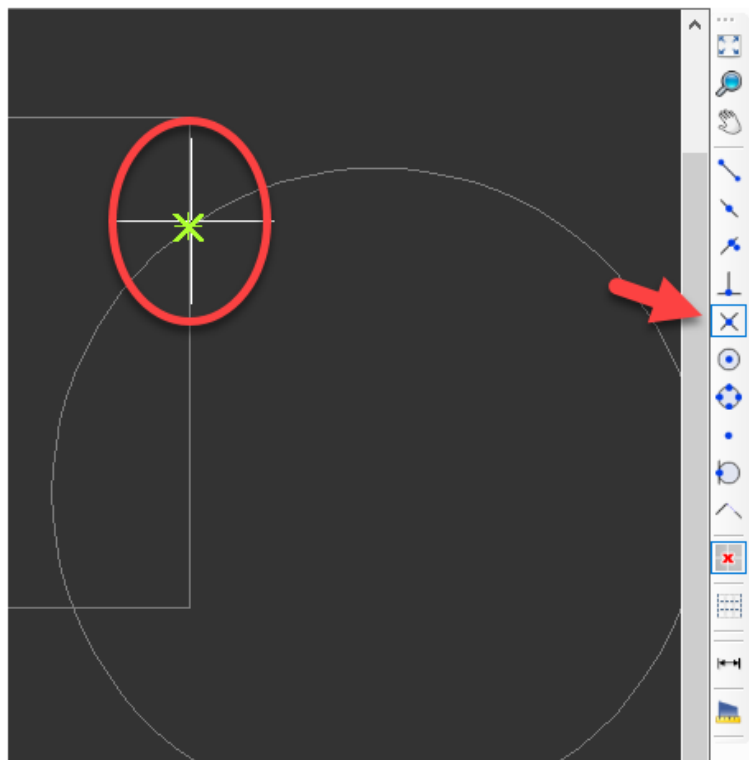
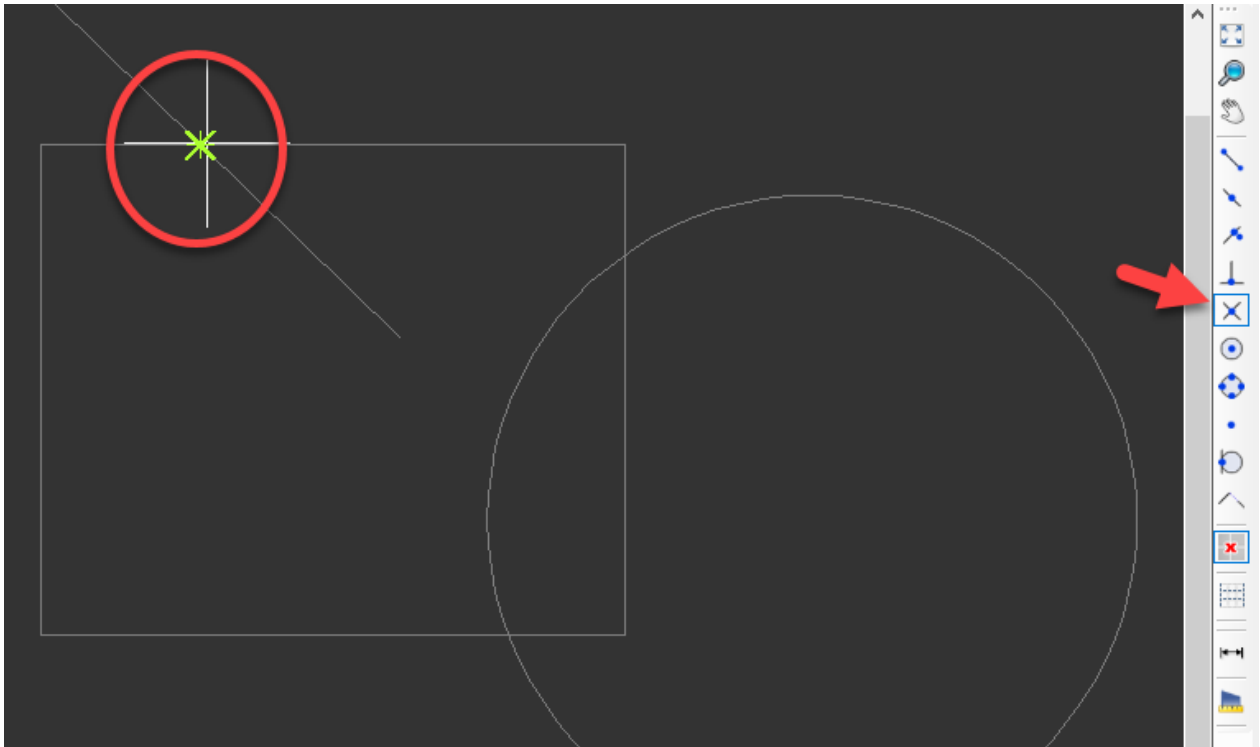
Perpendicular Point

This option will indicate the point at which the object being drawn will be perpendicular with any other object. Perpendicular point will be indicated with an L shape.



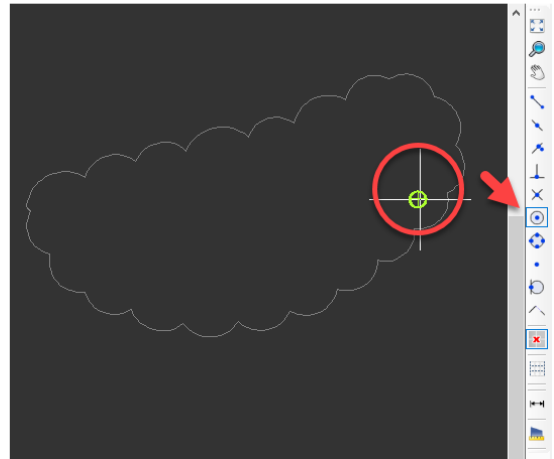
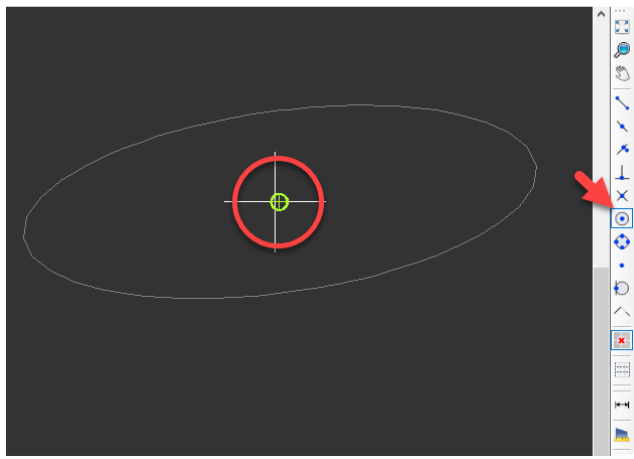
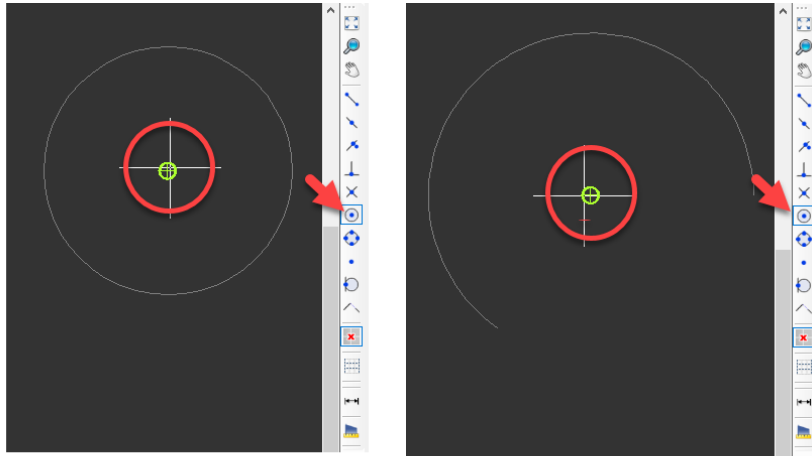
Intersection Point

This option will indicate the point at which two objects intersecting each other. Intersection point will be indicated with an X shape.



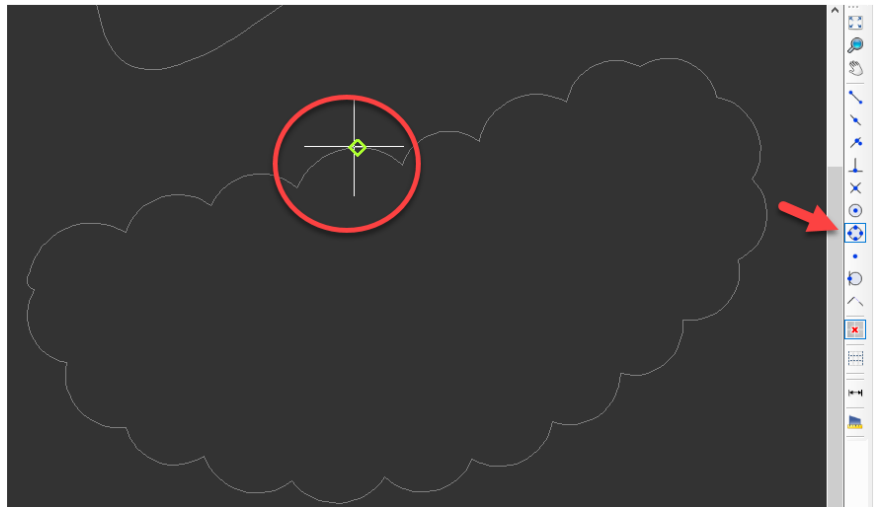
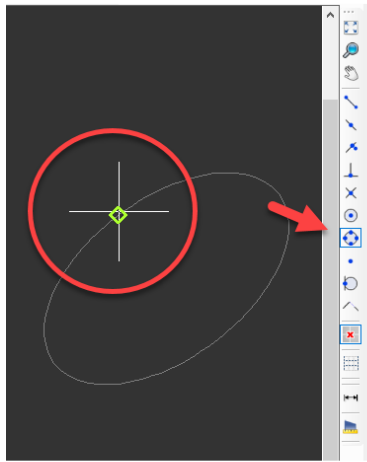
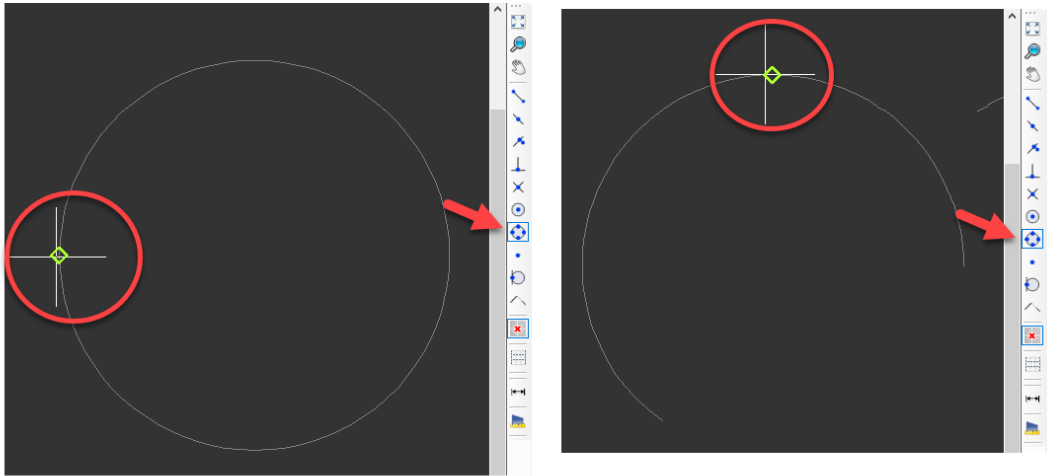
Centre Point

This option will indicate the centre point of a circular object or a circular part of an object. Centre point will be indicated with an O shape.



Quadrant Point

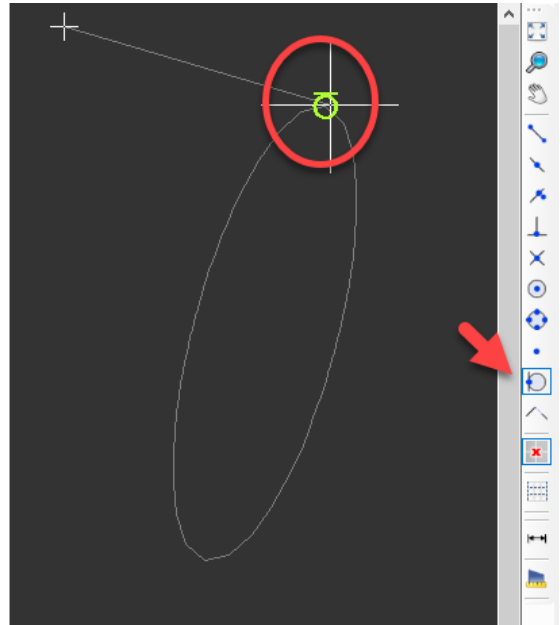
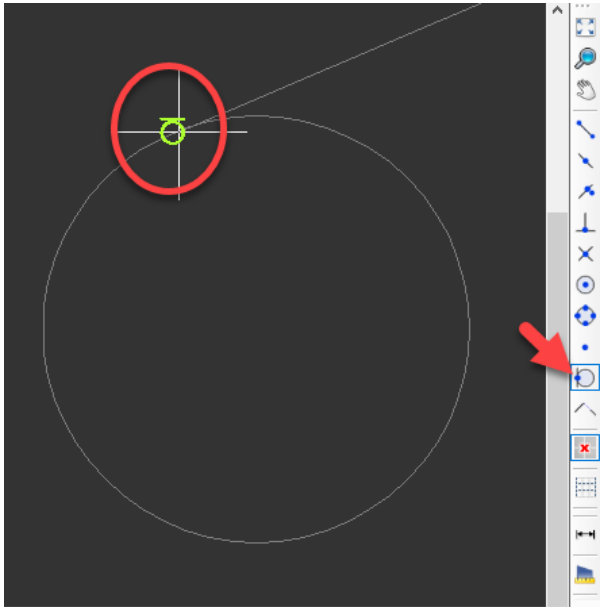
This option will indicate the quadrants of a circular object or circular part of an object. Quadrant point will be indicated with a diamond shape.



Node

Tangent Point

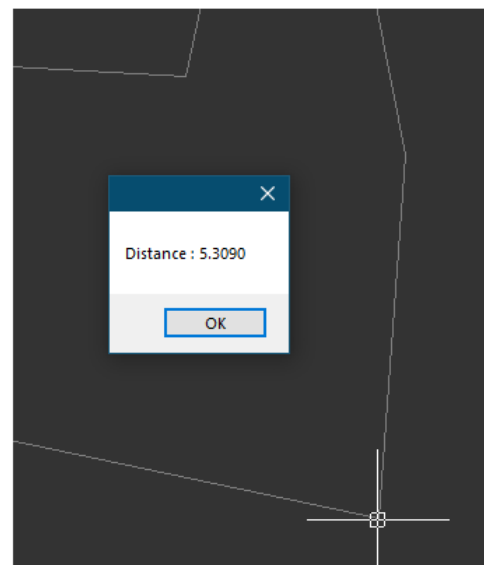
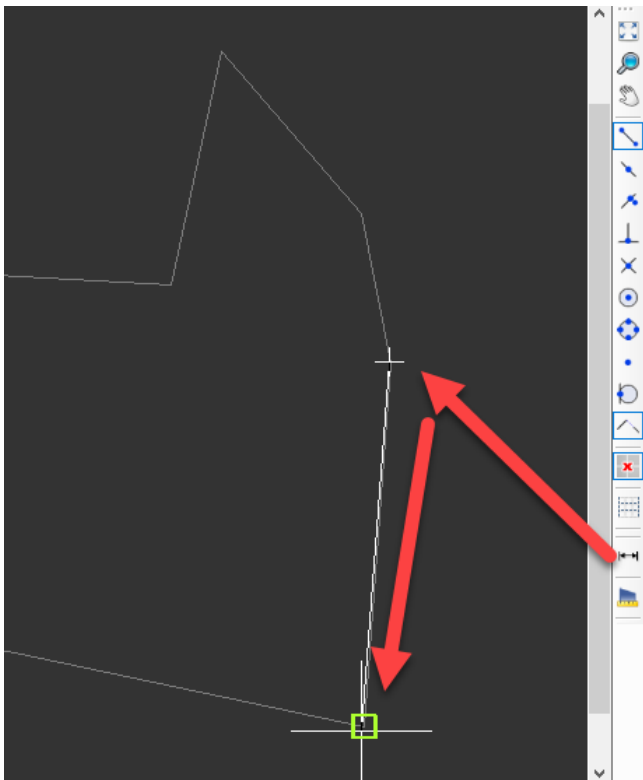
This option will indicate the point at which the object being drawn will be touching but not intersecting the other object. Tangent point will be indicated with an circle and a tangential line shape.



Extension Point

Measure length and area

- Measure distance button is used to measure a distance between two points, in metres. Once this button is clicked, then, a first and a second point must be selected on the drawing area. After selecting a second point CAD Plan Design will display the measured distance with a pop-up window.



- Measure area/length button is used to measure length and area at the same time. After this button is clicked, CAD Plan Design will expect the user to choose one of the options by typing in the corresponding letters, Points or Entity, or will execute the command as the current selection. Letter P is used for Points and E is used for Entity option. After typing in the required options, or to carry on with the current selection, user must perform a right-click or must hit the Enter button.



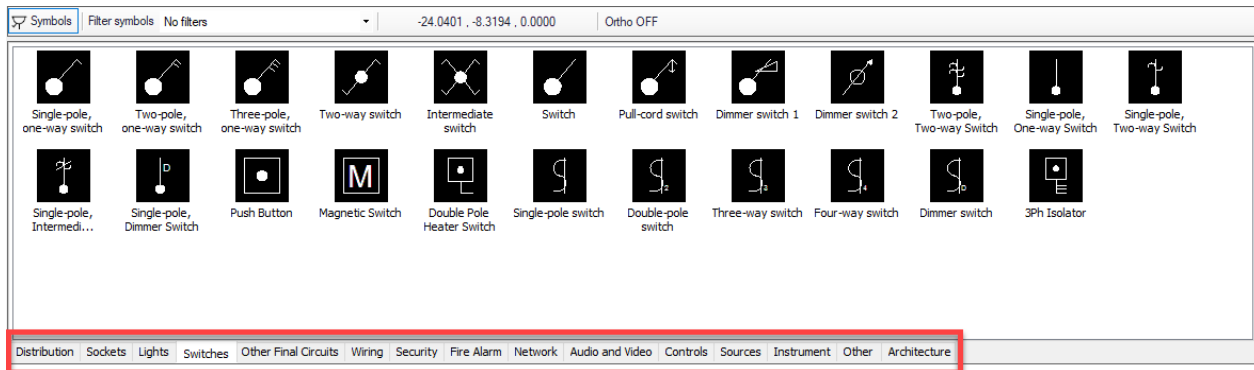
If P option is selected, CAD Plan Design will wait for points to be selected by the user. It is possible to select as many points as required. Once the last point is selected, a right-click or the Enter key will finalise the selection and the length and the area of the selection will be displayed.

If E option is selected, CAD Plan Design will wait for an object to be selected by the user. Once an object is selected, the length and area of the selected object will be displayed.

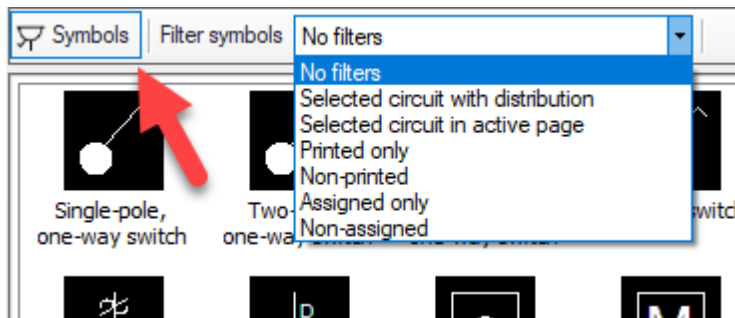
Symbols Area

Symbols area accommodates all the symbols, both built-in and user defined, categorised with tabs which are located below the symbols area. Symbols Library is a single library and all the symbols are shared between the core software and the CAD Plan Design Module.

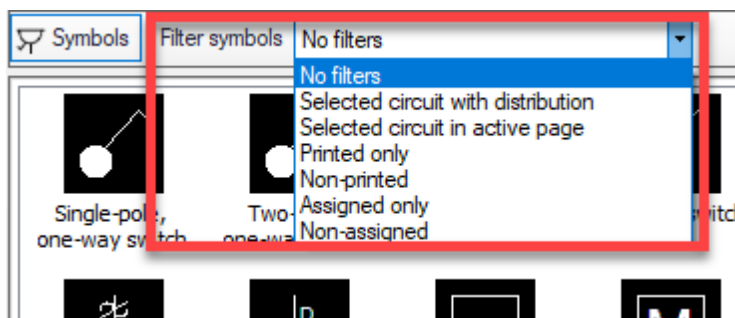
User may insert any symbol to the drawing area simply by dragging and dropping. Once a symbol is dragged to the drawing area, side panel will automatically display Draw symbol settings, see [Electrical Design Tools](#).



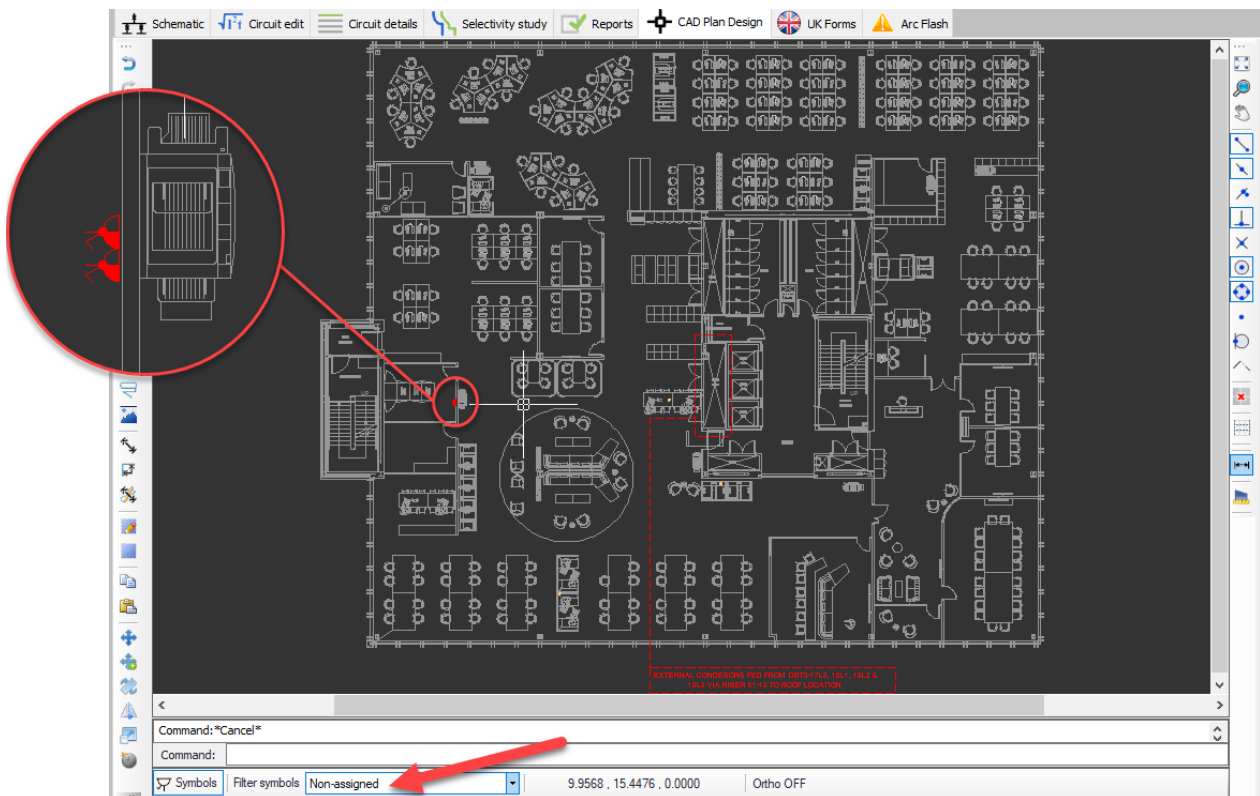
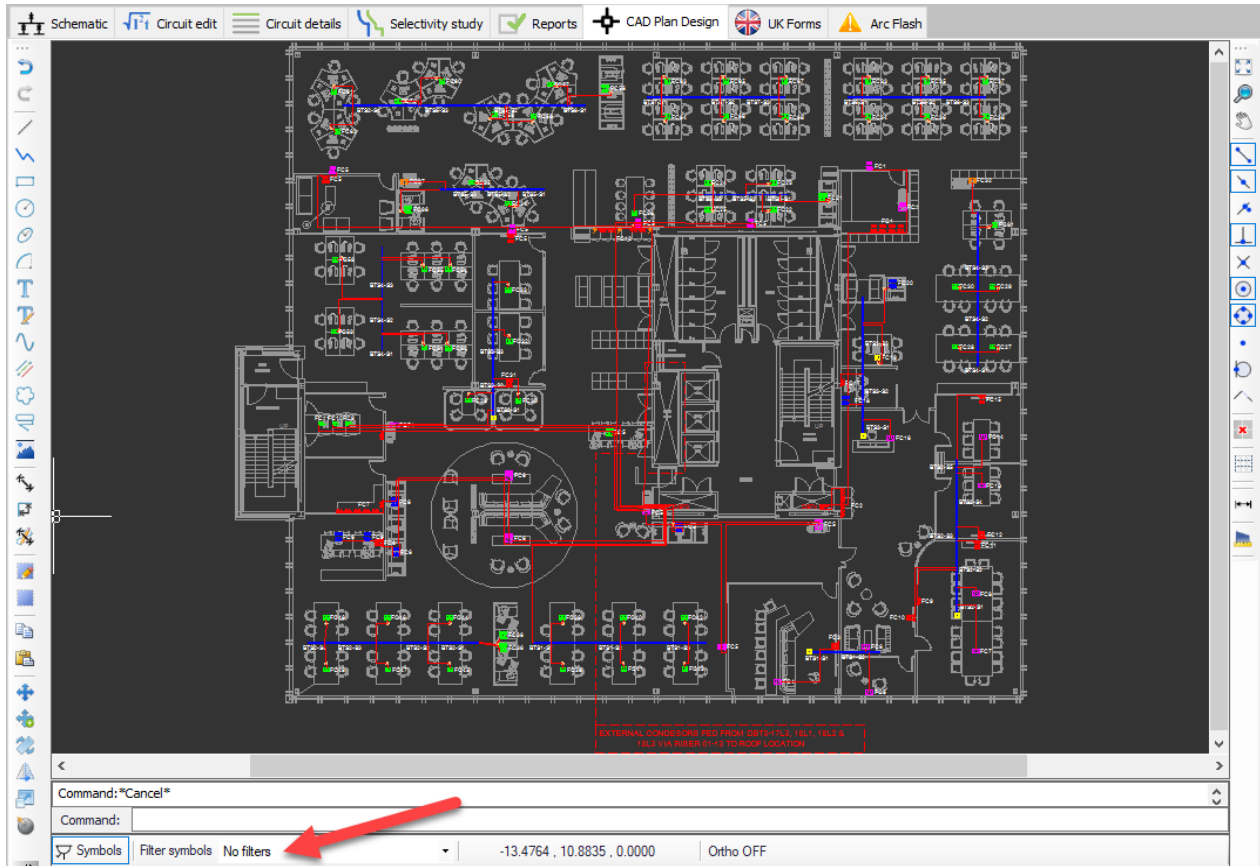
There are couple of actions that can be done using the right-click menu and these are detailed in following chapters, however, it is worth mentioning about two items just above the symbols area prior to any further information. First one is the Symbols button. This button will hide or show the symbols area.



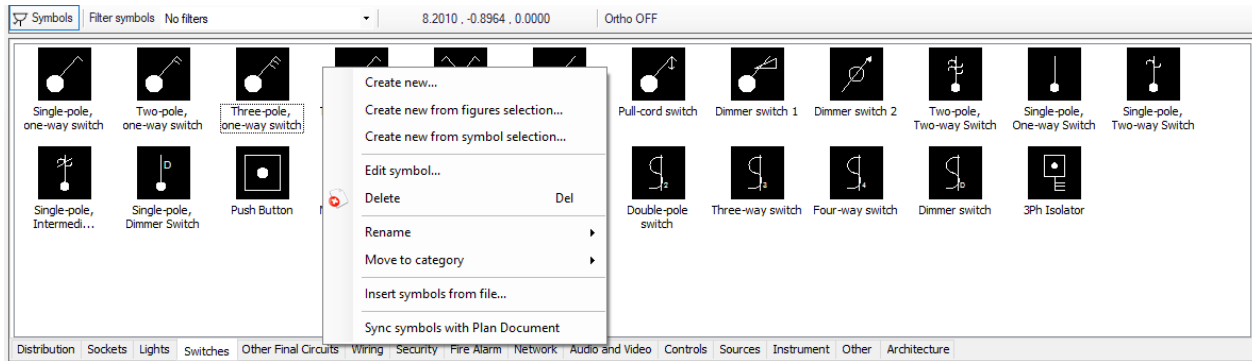
Second item is the Filter symbols drop down list which is located next to the Symbols button. User may apply various filters to symbols within the drawing area which may ease the usage.



For example, if the user selects Non-assigned option, ElectricalOM will display symbols which are not assigned to any specific circuit defined via the core software. This way, it becomes easy to identify symbols which may have been forgotten to be assigned to a circuit.



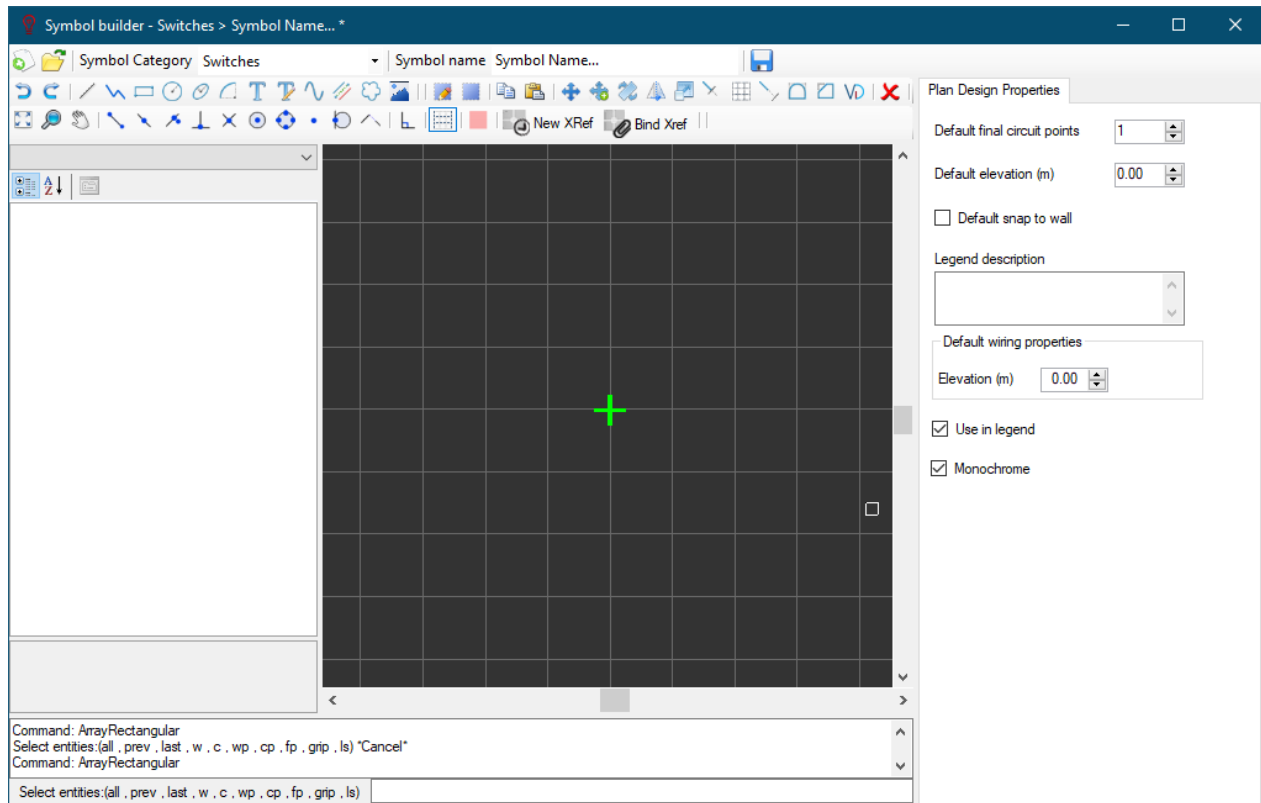
Creating a new symbol



Crating a symbol may be done in three ways. A new symbol can be created from scratch, or using an existing symbol, or via drawing area. Following three chapters will cover these three ways in detail.

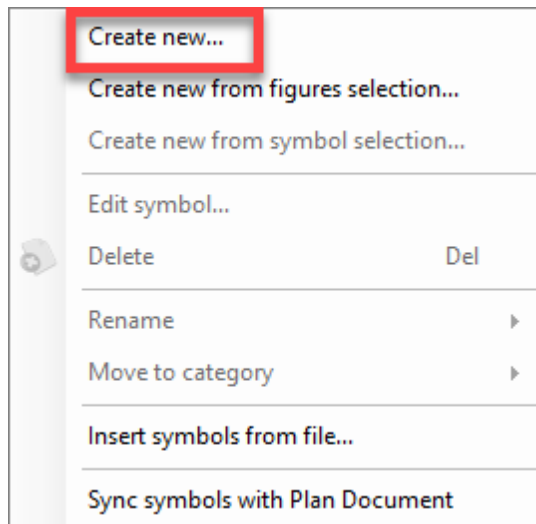
As a note, the Symbol Builder is the main tool we use to create or edit symbols throughout the ElectricalOM package and the usage is the same. Further details about this can be found in our Online Manual for ElectricalOM, [Creating and Editing a Symbol - Symbol Builder](#) section. and it will not be discussed here any further. However, there are a few additional options while using it via CAD Plan module, which are identified below.

- **Default final circuit points:** This option will tell ElectricalOM how many final circuits the created symbol will represent. This will be used while generating reports regarding quantities.
- **Default elevation (m):** This will define the default height of the created symbol referenced to Z=0. This is used in cable length calculations and 3D representation.
- **Default snap to grid:** This is set snap-to-grip feature to default, so when the created symbol is inserted this option will be enabled.
- **Monochrome:** If this box is ticked then the symbol will be inserted as a monochrome symbol even if it is set to a certain colour.



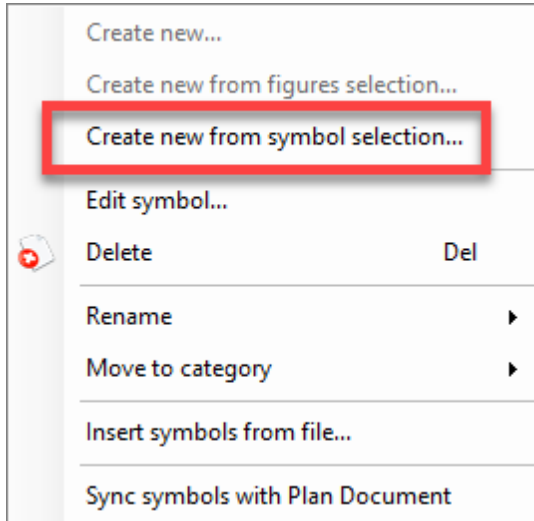
From scratch

One way is to create a new symbol from scratch. Symbol Builder can be accessed by a right-click to an empty spot within the symbols area and selecting Create new... option.

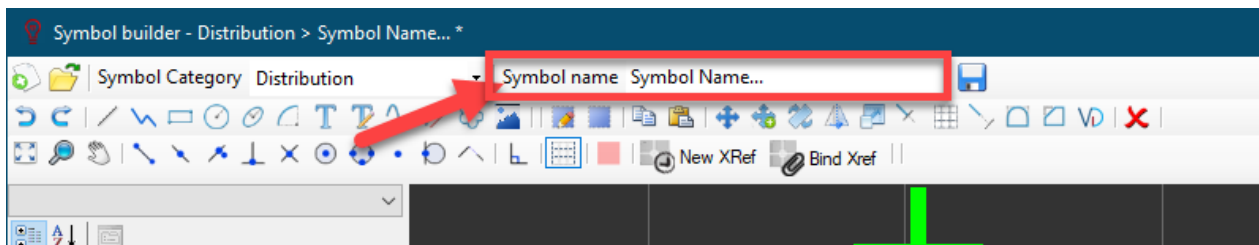


Using an existing symbol

Second way to create a symbol is to use an existing symbol. By right-clicking on an existing symbol and selecting Create new from symbol selection ... option will open the Symbol Builder for the selected symbol.

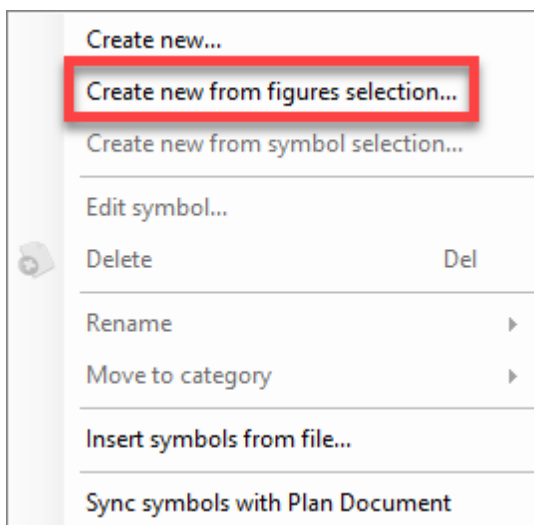


After editing the symbol, a new name can be defined by typing in a text to Symbol name field to save it as a new symbol.

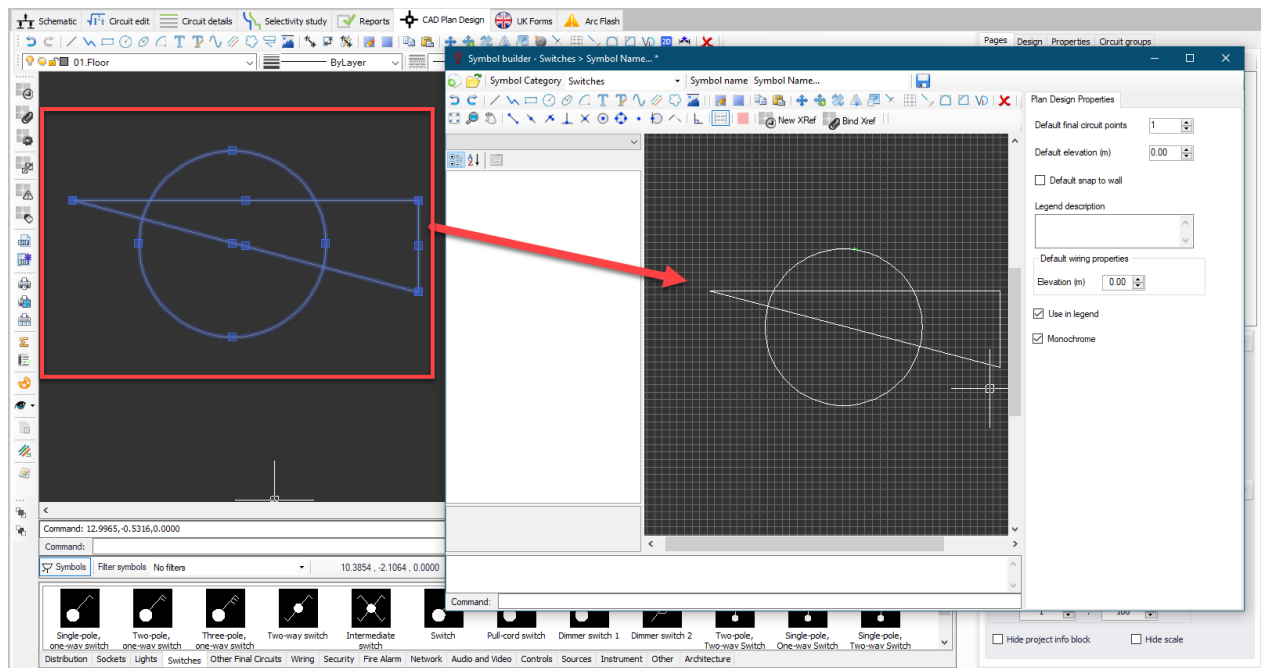


Using an existing shape from drawing area

The last way to create a symbol is via the drawing area. In order to create a symbol via the drawing area, user must select the shape first, then, perform a right-click within the symbols area and select Create new figures selection... option.

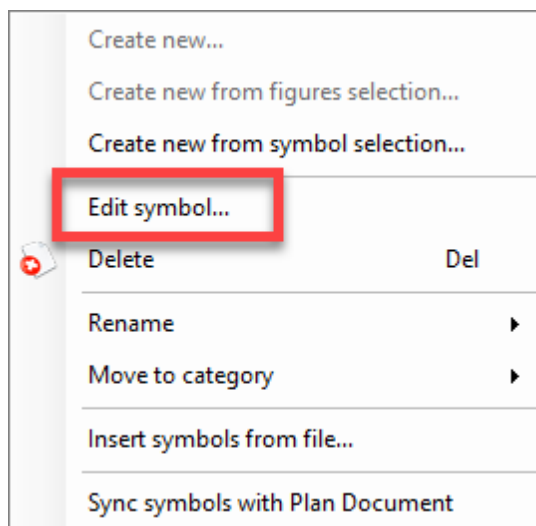


Symbol Builder will pick the selected items and display them within its own drawing area.

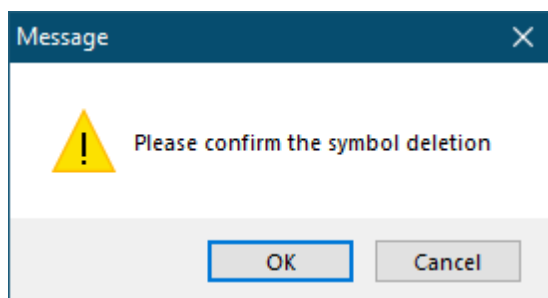
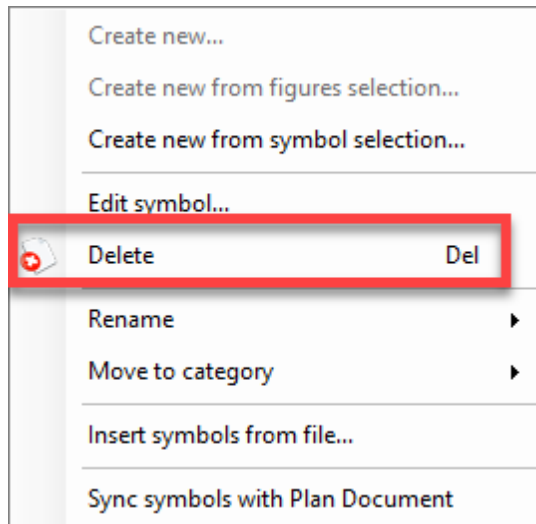


Editing and deleting an existing symbol

User may like to edit an existing symbol. In this case, the symbol requires editing can be right-clicked and the option Edit symbol... selected. This will open the Symbol Builder for the selected symbol, but this time Symbol Builder will save any changes onto the original symbol when save button is clicked.

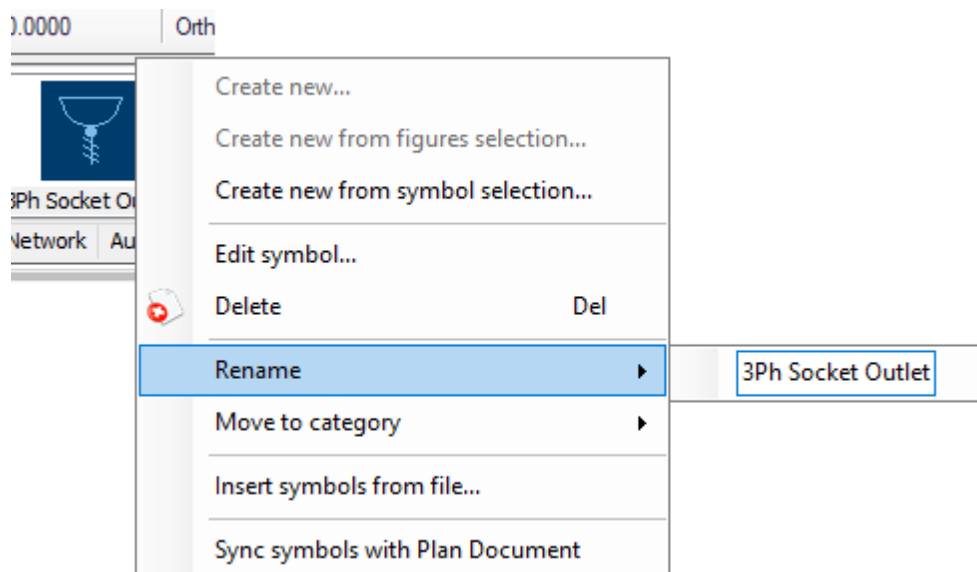


Delete option will delete a selected symbol from the library.



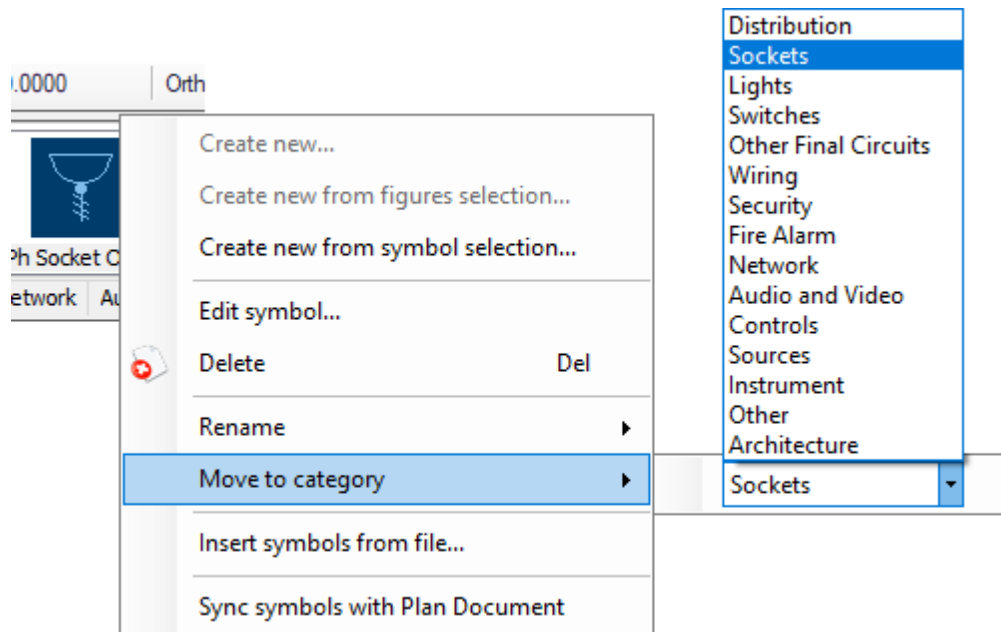
Quick rename and Quick change category

User can quickly rename a symbol without going into the Symbols Builder by right-clicking on the symbol in question. Rename option will display a text field where a new name can be typed in.



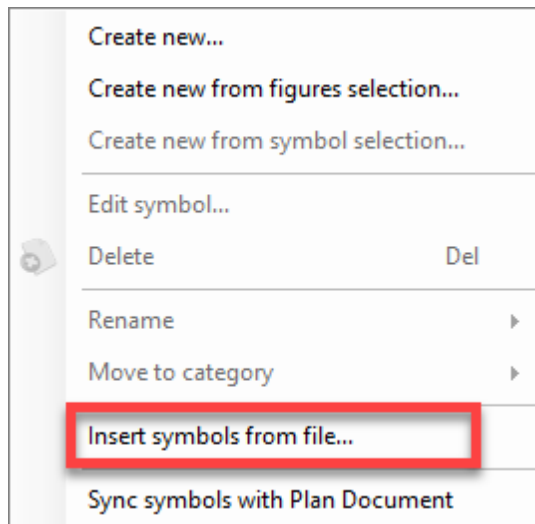
Similar to rename, a quick way of changing the category of a symbol can be done by right-clicking on an existing symbol. This time, Move to category option will display a list of

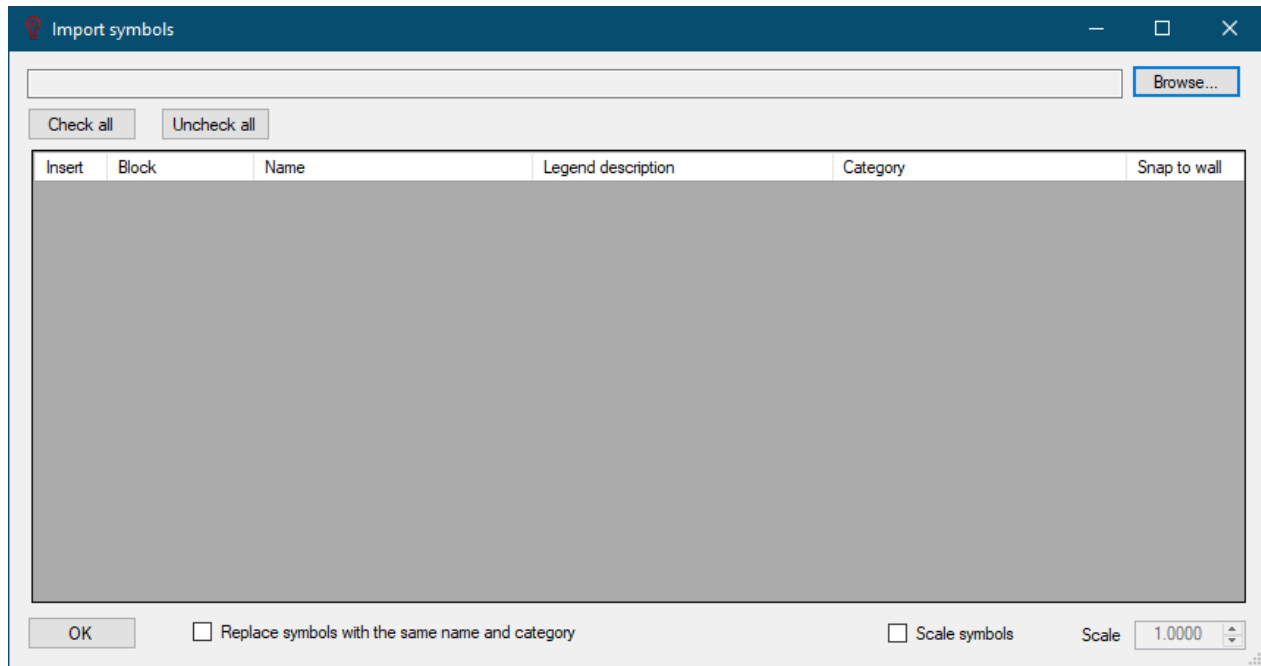
available categories so the user can select one to move the selected symbol to it.



Importing symbols

Importing symbols can be done by Insert from file... option. Import symbols windows will be displayed after a right-click on an empty spot within symbols area. Using the Browse... button, user may select a CAD or a ElectricalOM file to import symbols from it.



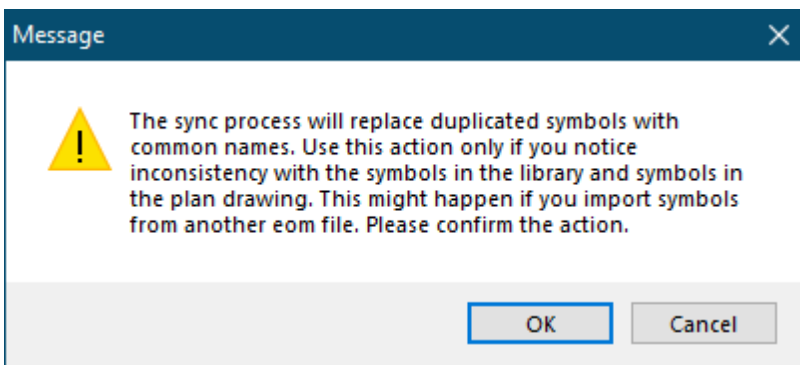
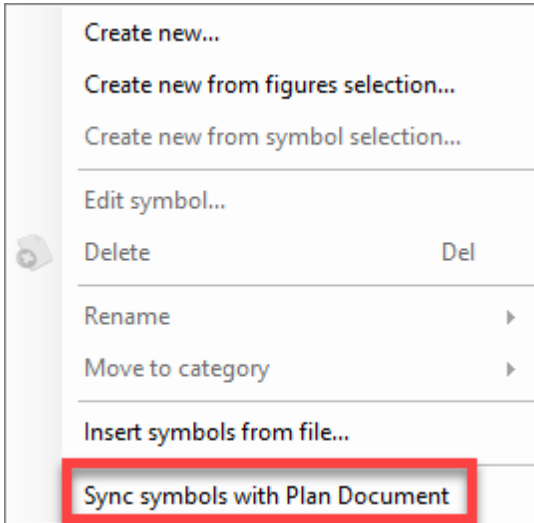


If Replace symbols with the same name and category box is ticked, then, import will replace the existing symbols with the imported ones if they have the same name and category. Scale symbols box is used to scale imported symbols to match with the built-in symbols. A scaling factor may be typed in to the Scale field.

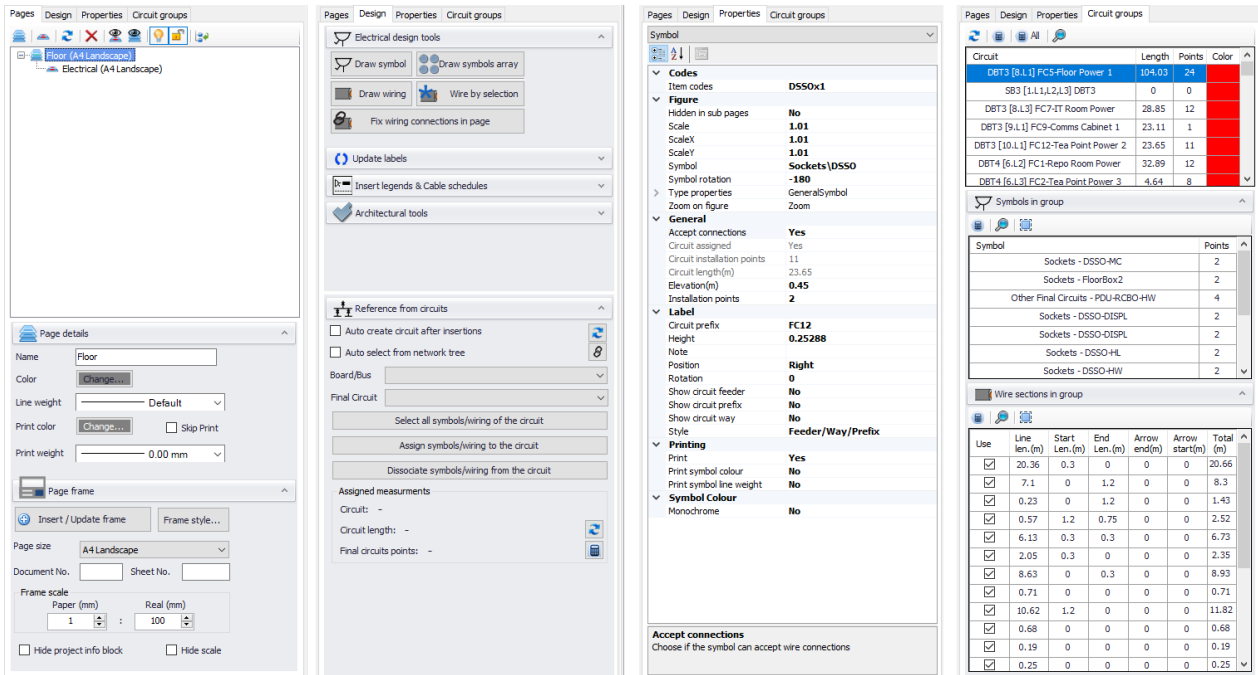
For further details about usage of the Import Symbols window please refer to our Online Manual for ElectricalOM, [Insert symbols from file...](#) section.

Synchronise symbols

Sometimes, symbols in a project may have the same names as the built-in symbols, and if these symbols are imported while the Replace symbols with the same name and category box is ticked, then, ElectricalOM will replace the built-in symbols with the imported ones. In order to reset the built-in symbols, Sync symbols with Plan Document option can be used.



Side Panel

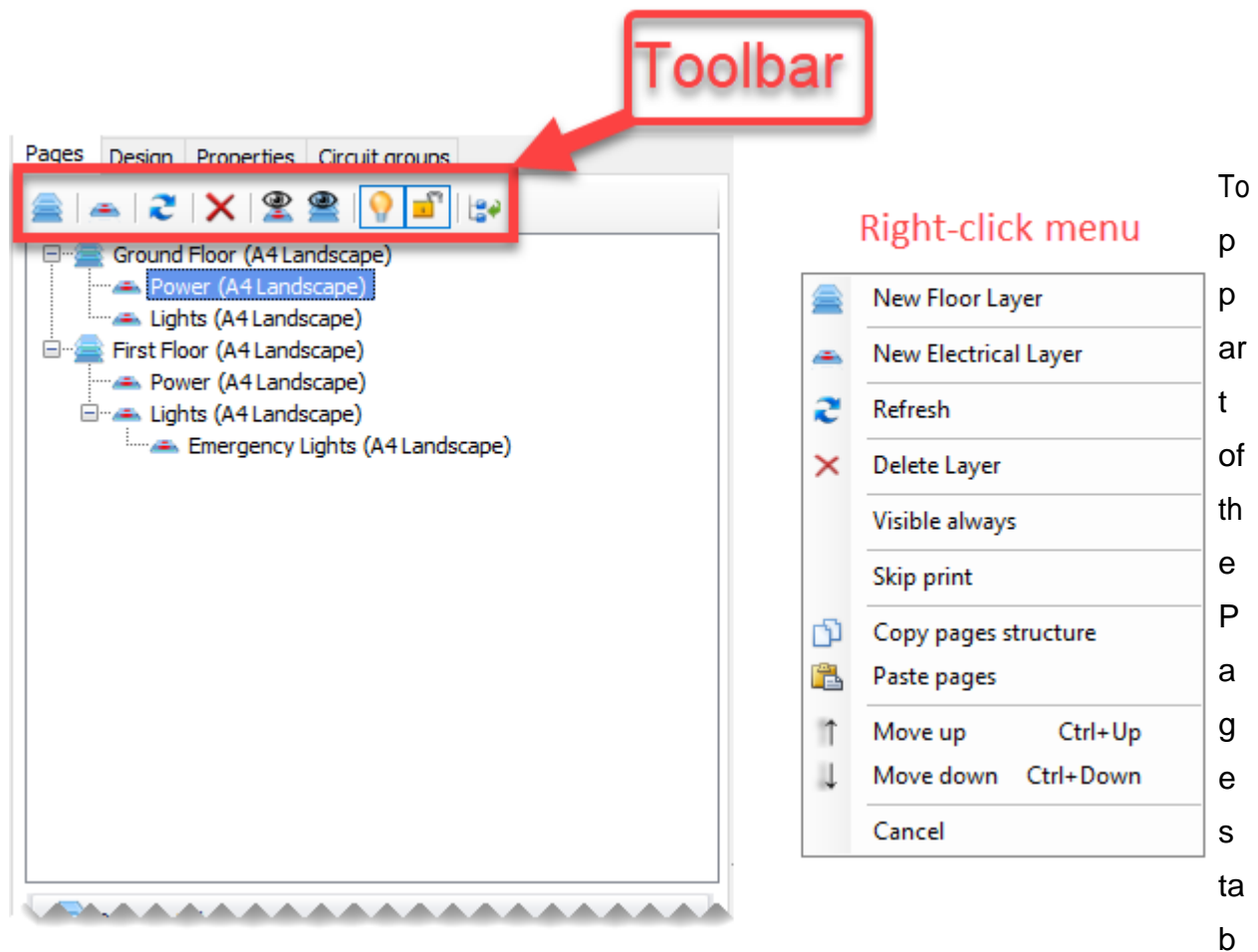


Side Panel of CAD Plan Design is where most of the settings located. It has four tabs serves different purposes. These tabs' contents have fixed functions, however, contents will

change according to the selected object at the drawing area.

- Pages tab accommodates settings related with layers and pages.
- Design tab is the main tab which is used for electrical design, where required design tools, and linking between a ElectricalOM and CAD Plan Design options are located.
- Properties tab accommodates properties of a selected item.
- Circuit Groups tab will display wiring length and installation point details of assigned circuits, is any.

Pages Tab



is reserved for layers. Toolbar located on top, and right-click menu will help users to create, edit, delete layers where related data will be shown in the white area below the toolbar.

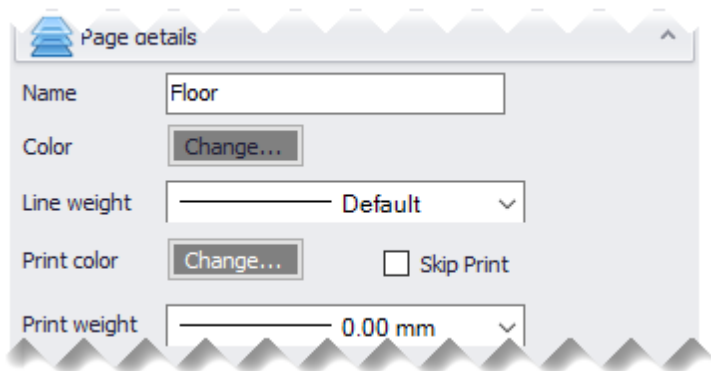
Layers have a hierarchical order within the tree representation. Layers with the same level will hide others when they are selected. A sub-layer will appear on top of the parent layer.

For example, if First Floor layer is selected then, Ground Floor layer and all of its sub layers will not be displayed. If Power layer is selected under the Ground Floor layer, then contents

of Power will be displayed together with Ground Floor layer. However, if Lights layer under the Ground Floor layer is selected, only contents of this layer will be displayed on top of Ground Floor layer contents.

Toolbar buttons and right-click menu items are , in order:

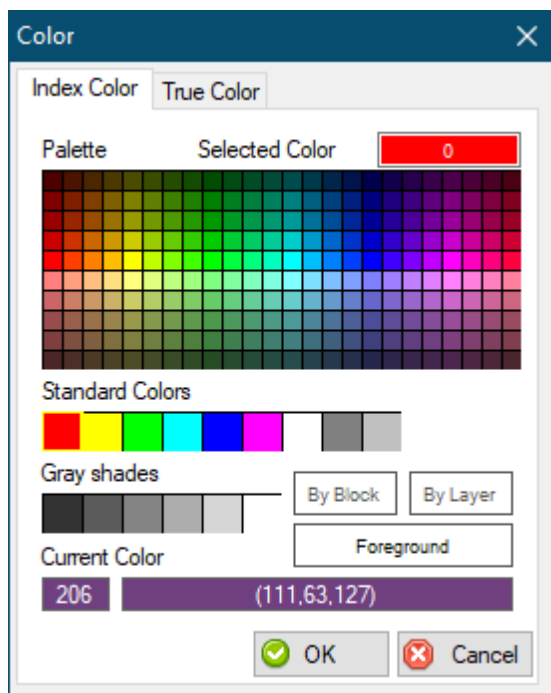
- Add background layer/New Floor Layer: This will add a layer which is not related with electrical design which may be a floor plan.
- Add electrical design layer/New Electrical Layer: This button will add a sub layer to the selected background layer where the user can draw electrical design components like wires, and insert symbols, etc.
- Refresh: This button will refresh the layers in case they do not appear correctly.
- Delete layer: This will delete the selected layer.
- Group Layers: This button will display a window where the user can select which layers to be visible when a layer is selected. The selection will override any default behaviour.
- Visible always: This option will set the selected layer to be visible all the time no matter which layer is selected.
- Skip print: This option will remove the selected layer from the print.
- Auto On/Off Layers: This button will help users to automatically turn unselected layers off. If the option is set to off, then, all the layers will be visible no matter which layer is selected.
- Auto Lock Layers: This button automatically locks unselected layers, where locked layers cannot be modified, if it is set to on mode. If this option is set to off, then, all the visible layers can be modified.
- Collapse nodes: This button will collapse nodes under selected layers only, other layers will not be affected.
- Copy pages structure/Paste pages: These will copy and paste selected layer.
- Move up/Move down: These options will move the selected layer up or down within the layers tree.
- Cancel: This will close the right-click menu.

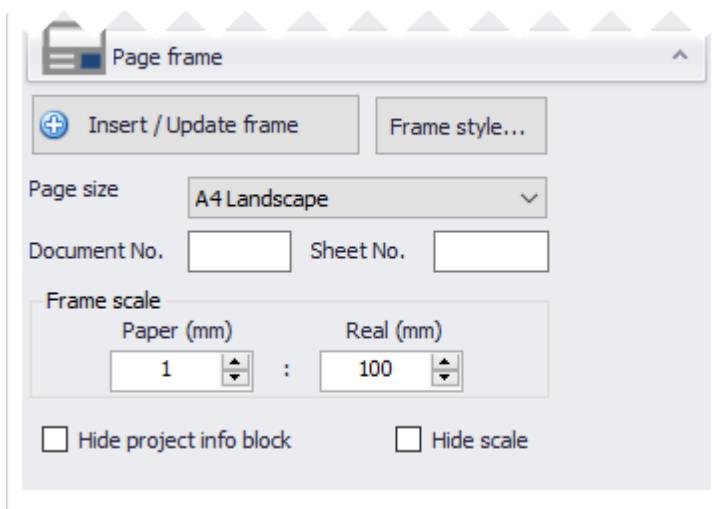
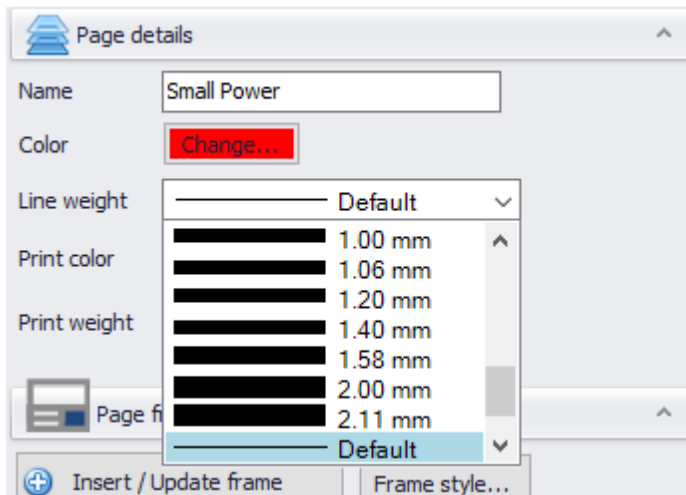


Middle section is reserved for layer properties where users may define a name, a colour, line weight, print colour and print line weight for the selected layer. Layer name can be typed into the text field next to Name, colour can be picked from the

Colour window once clicked on Colour button, Line weight can be selected from the drop down list, also applicable to Print Colour and Print Line Weight.

Colour and Line weight settings are only used for display purposes, where Print colour and Print line weight is used for printing purposes. Skip print box can be used to exclude the selected layer from print and reports.





Bottom section is reserved for frame settings. Insert/Update frame button is used to insert a new frame to the selected layer. If there is a frame already inserted, then this button will update the frame to new settings. Any changes will only be reflected if Insert/Update frame button is pressed.

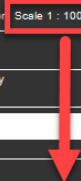
Frame style... button displays frames which can be used. ElectricalOM manual has more details regarding how to use the frame builder, please see Frame Builder [Frame Builder](#).

Page size and layout can be selected using the drop down list below these buttons, and a document number and a sheet number can be define using the related text fields. Frame scale area is used to display the drawing scale on the frame, however, this can be removed by ticking the Hide scale box.

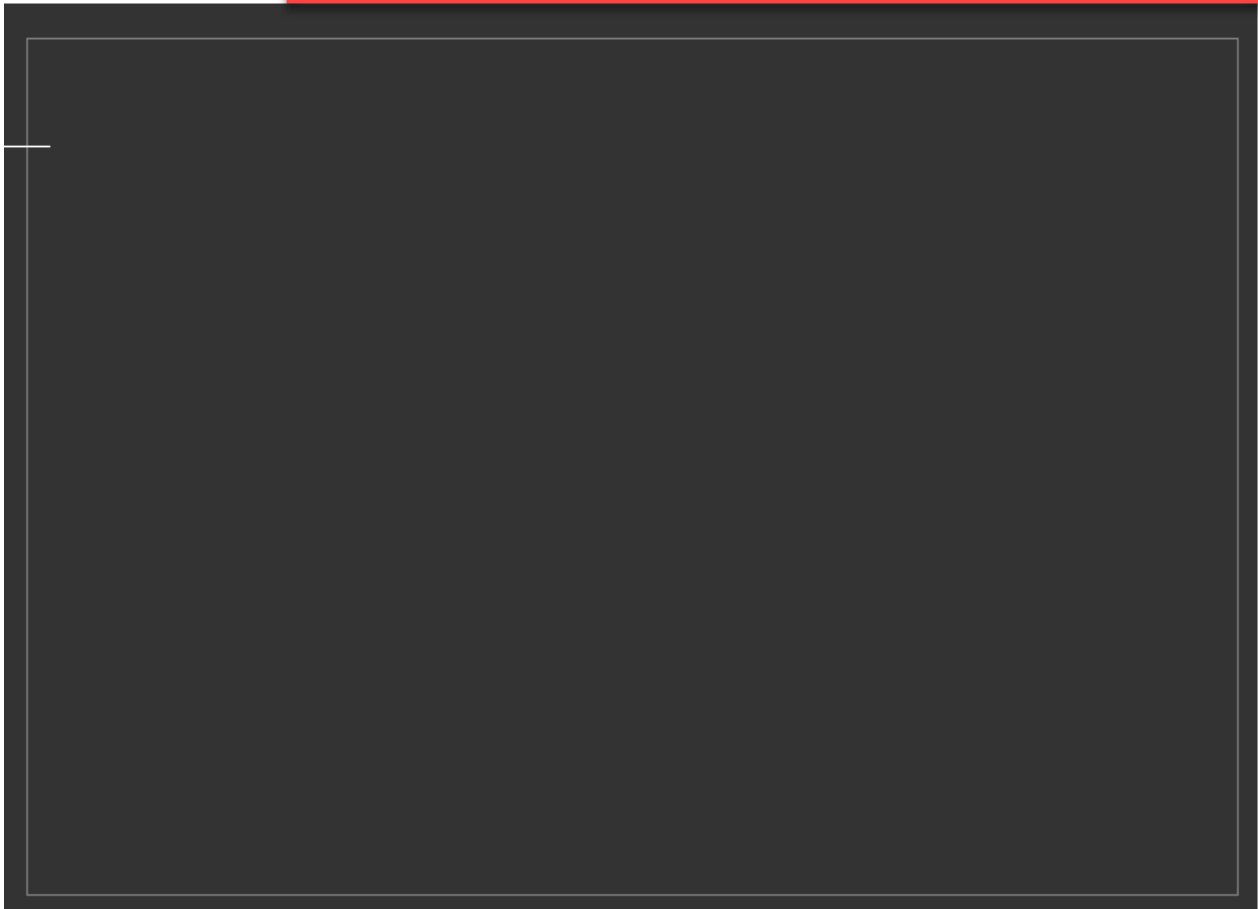
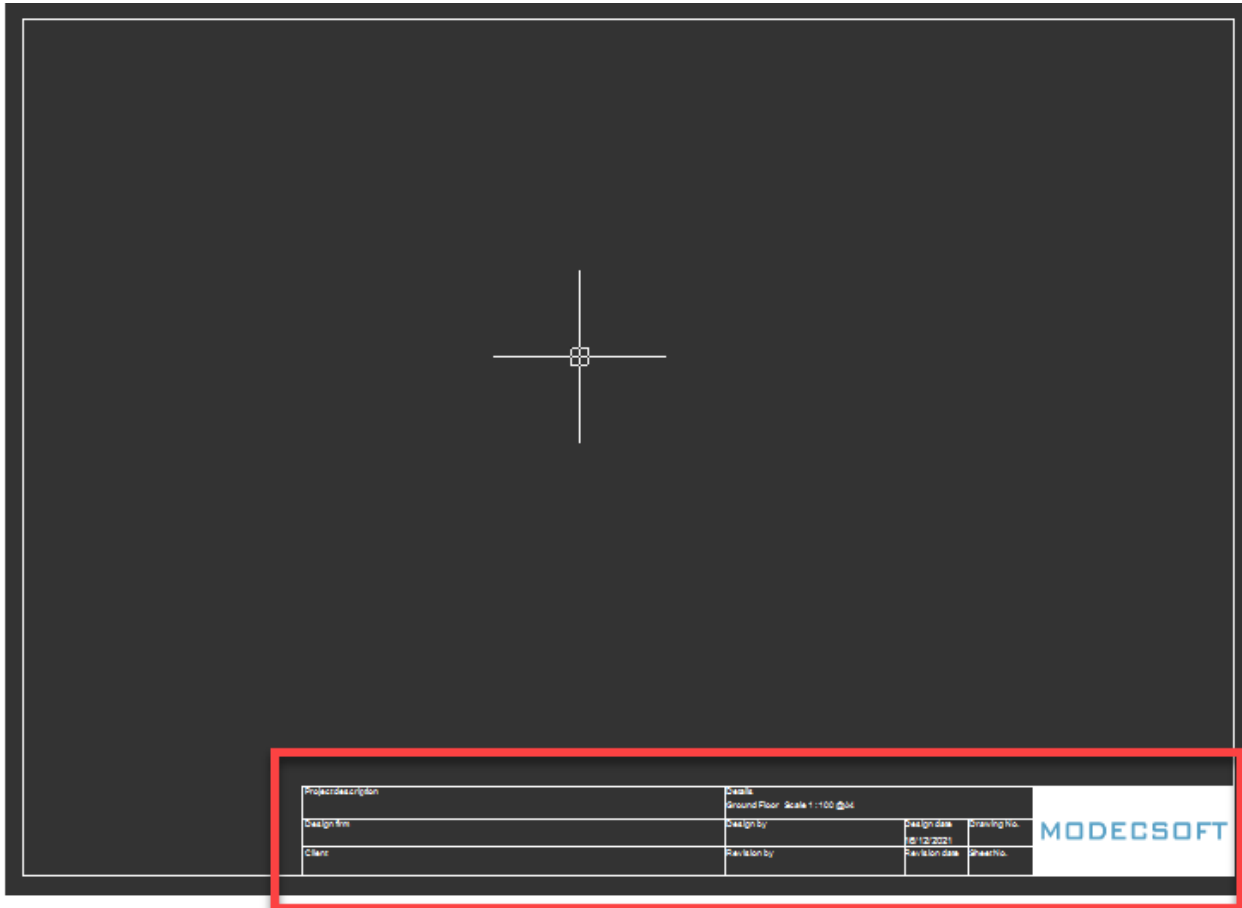
ElectricalOM CAD Plan Design Add-on Module

Project description	Details			MODECSOFT	
Design firm	Ground Floor	Scale 1 : 100 @A4	Design date		Drawing No.
Client	Design by	16/12/2021	Revision date		Sheet No.
	Revision by				

Project description	Details			MODECSOFT	
Design firm	Ground Floor		Design date		Drawing No.
Client	Design by	16/12/2021	Revision date		Sheet No.
	Revision by				



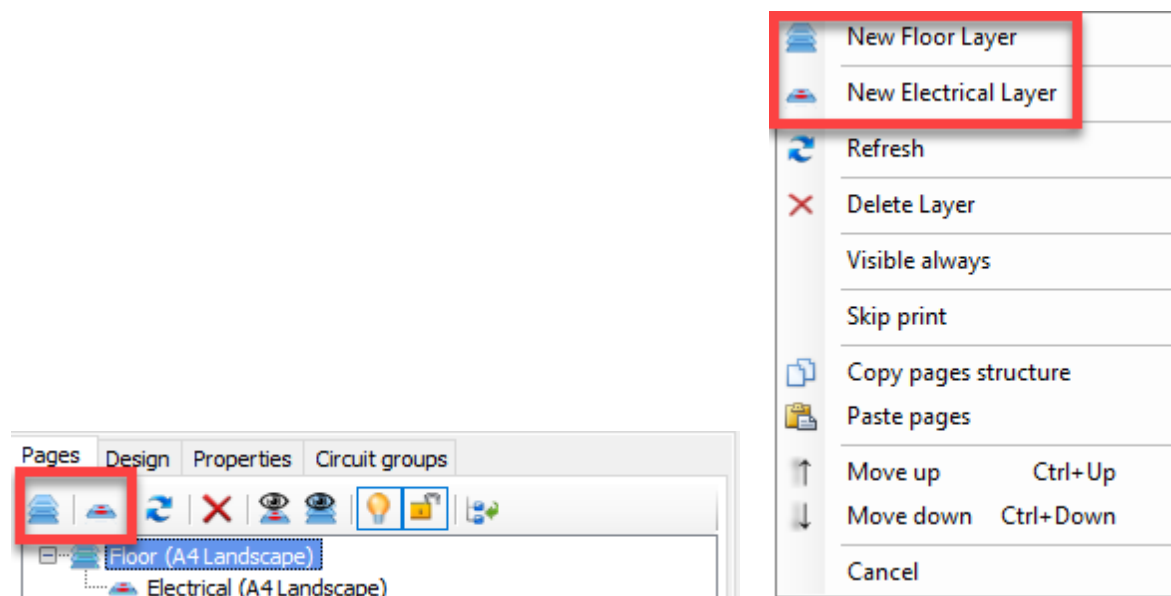
Hide project info block tick box will hide the title block completely and leave the frame only.



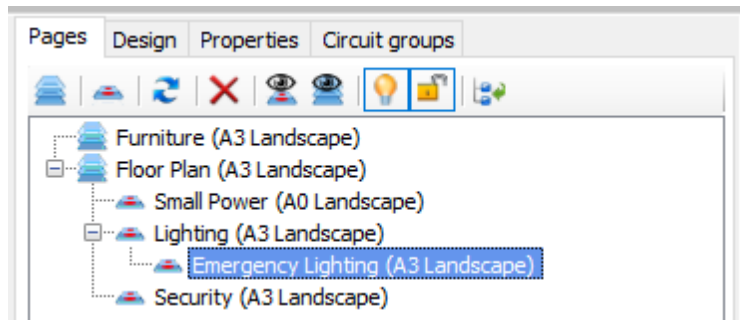
Layer Options

There are two types of layers within CAD Plan Design module. Background layers and Electrical layers. CAD Plan Design module requires at least one background layer in order to work. Each time a new background layer is created, a new category is created, meaning that a background layer cannot be created as a sub layer. For this reason, they are generally used for floor plans where electrical layers can be created as subordinate layers. On the other hand, unlike background layers, electrical layers cannot be created as top level layers and they are always a sub level layer to other layers. However, they can be created under a background layer or an electrical layer.

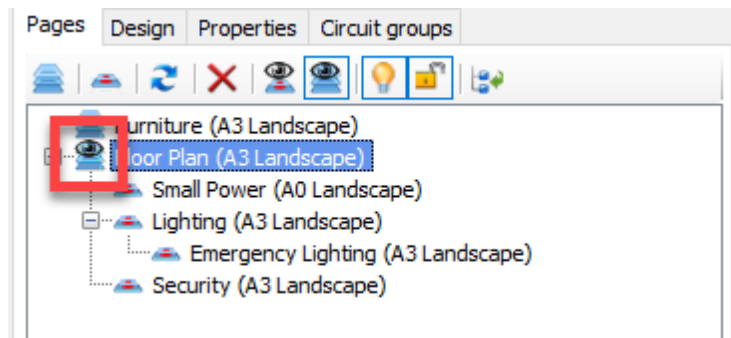
By default, CAD Plan Design module has a background layer, Floor, and an electrical layer, Electrical. In order to create a new layer, first two buttons of the toolbar or New Floor Layer and New Electrical Layer options of the right-click menu is used.



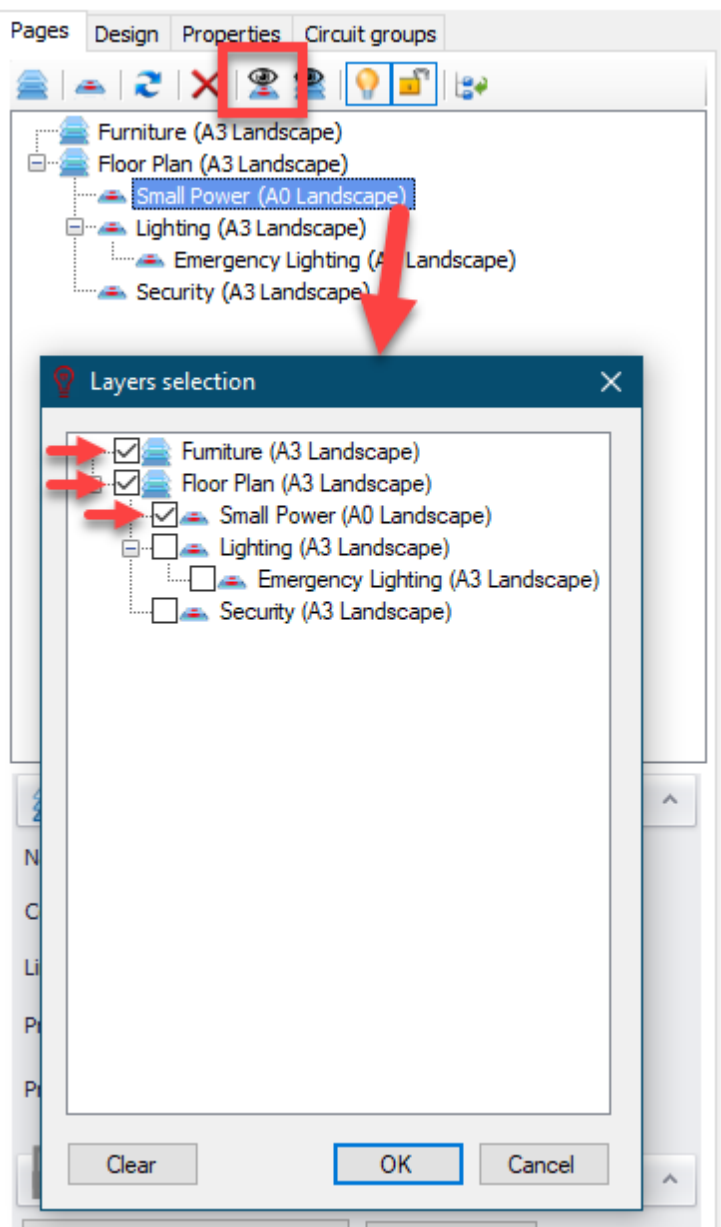
Before creating a new layer, intended location for the new layer must be selected. The new layer will be created below the selected point. The new layer will be created hierarchically obeying the rules described above.



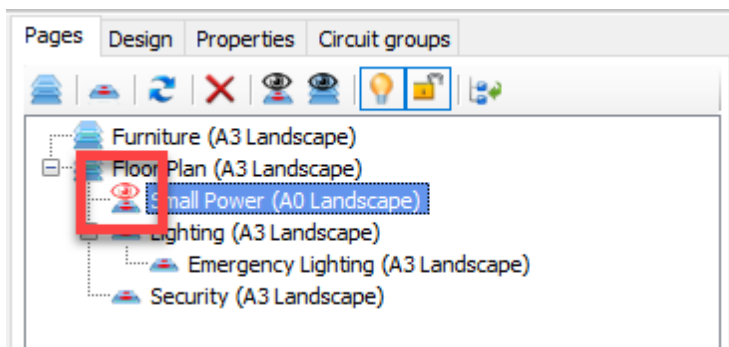
As mentioned before, there is a hierarchical relationship between layers where the one above has the priority over the one below, however, this can be overridden by the Group layers and Visible always options. If a layer is set to Visible always, then, this layer will always be displayed no matter which layer is selected. For example, if Emergency Lighting layer is set to Visible always, then, even if the Security layer is selected, which in normal circumstances will hide all other same and lower level layer contents, contents of Emergency Lighting layer will still be displayed. When a layer is set to Always visible, the icon will be changed to indicate the setting.



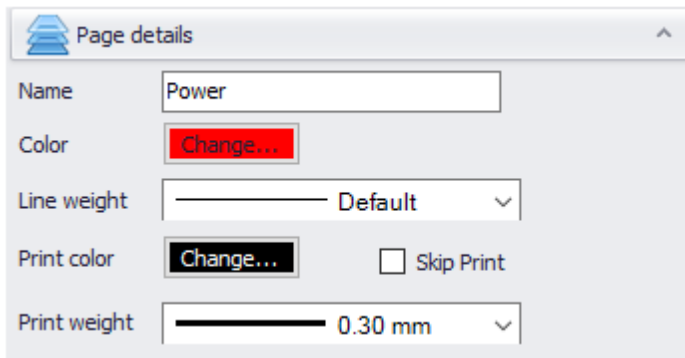
Also the user can define which layers will be displayed when a specific layer is selected using the Group layer button. Clicking on Group layers button will display Layers selection window. By ticking boxes next to layer names, the user may select layers will be displayed when selected layer is clicked, Small Power layer under Floor Plan layer, in the case below. As a side note, Group layers will override Always visible setting, so, even if a layer is set to Always visible, if it is not selected at Layers selection window, it will not be displayed.



When a layer has a group option set, the icon will be changed to reflect this.



Page Details



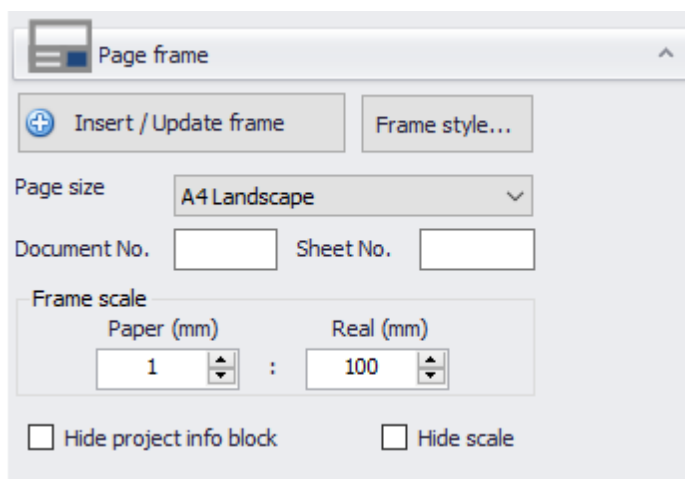
The screenshot shows the 'Page details' dialog box with the following settings:

- Name: Power
- Color: Change...
- Line weight: Default
- Print color: Change...
- Skip Print:
- Print weight: 0.30 mm

Page details section is where default layer properties are set.

- Name: Name of the layer
- Colour: Default colour associated with the layer. This can be changed by clicking on the Change... button.
- Line weight: Default line thickness associated with the layer. This can be changed by using the drop-down list.
- Print colour: Default colour will be used when printing. This can be changed by clicking on the Change... button.
- Print weight: Default line thickness will be used when printing. This can be changed by using the drop-down list.

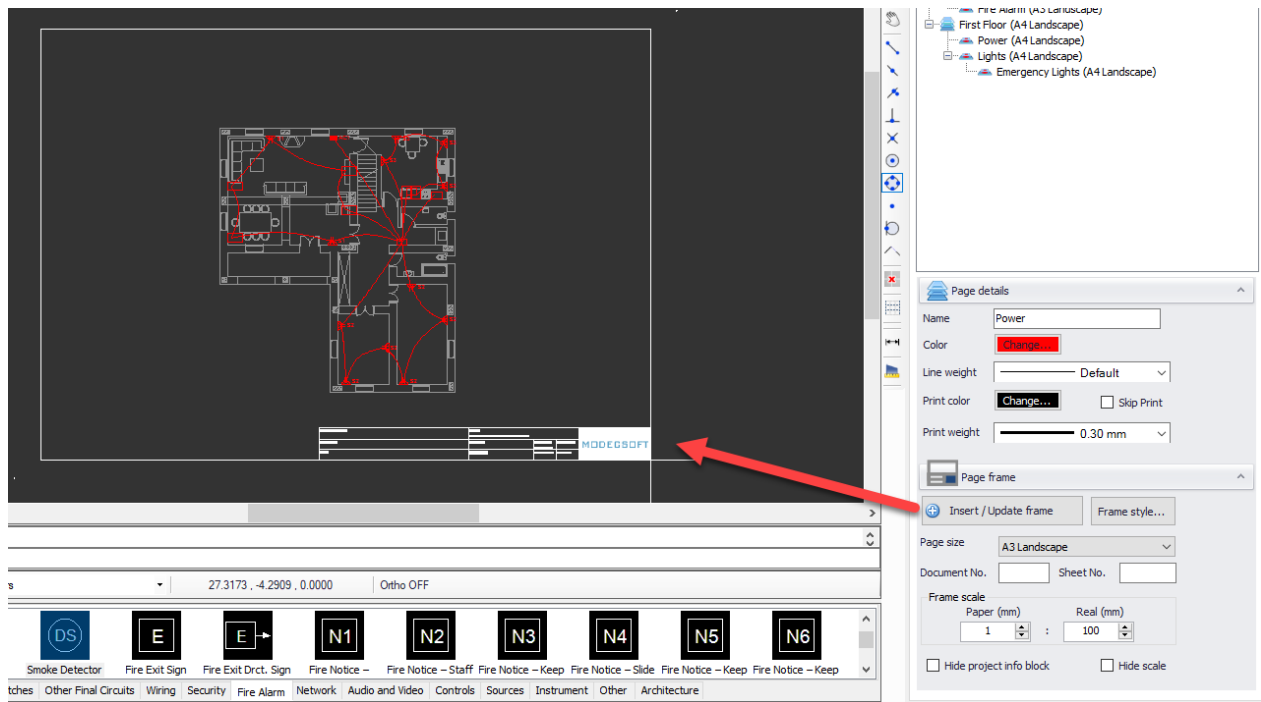
Page Frame



The screenshot shows the 'Page frame' dialog box with the following settings:

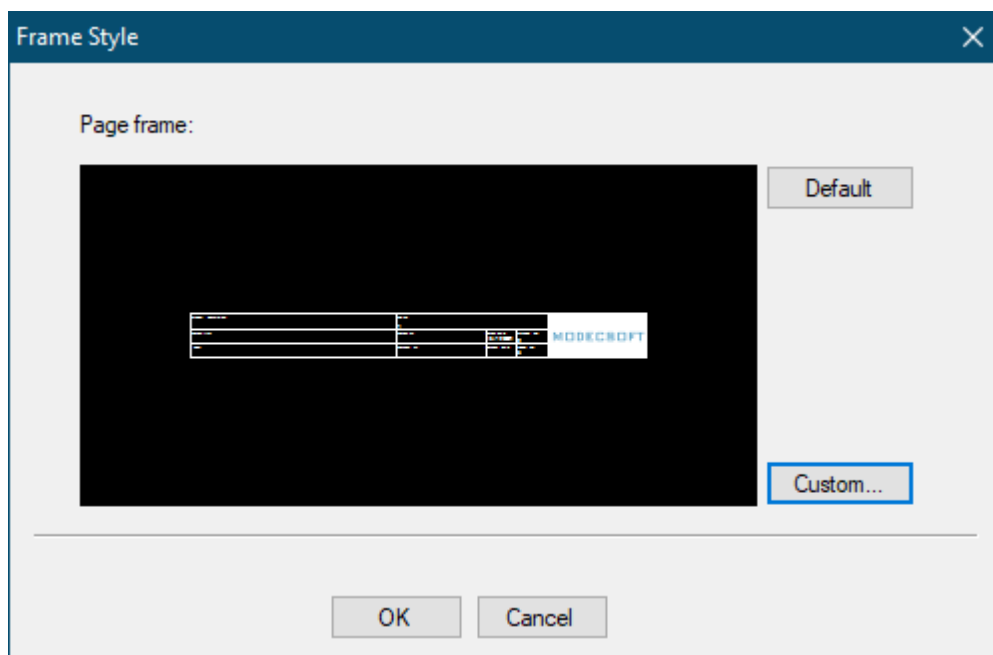
- Buttons: Insert / Update frame, Frame style...
- Page size: A4 Landscape
- Document No.: [] Sheet No.: []
- Frame scale: Paper (mm) = 1, Real (mm) = 100
- Hide project info block:
- Hide scale:

Page frame section is similar to ElectricalOM's frame feature. User can insert a frame by clicking on the Insert/Update frame button. The same button is also used to update the frame to reflect any changes.

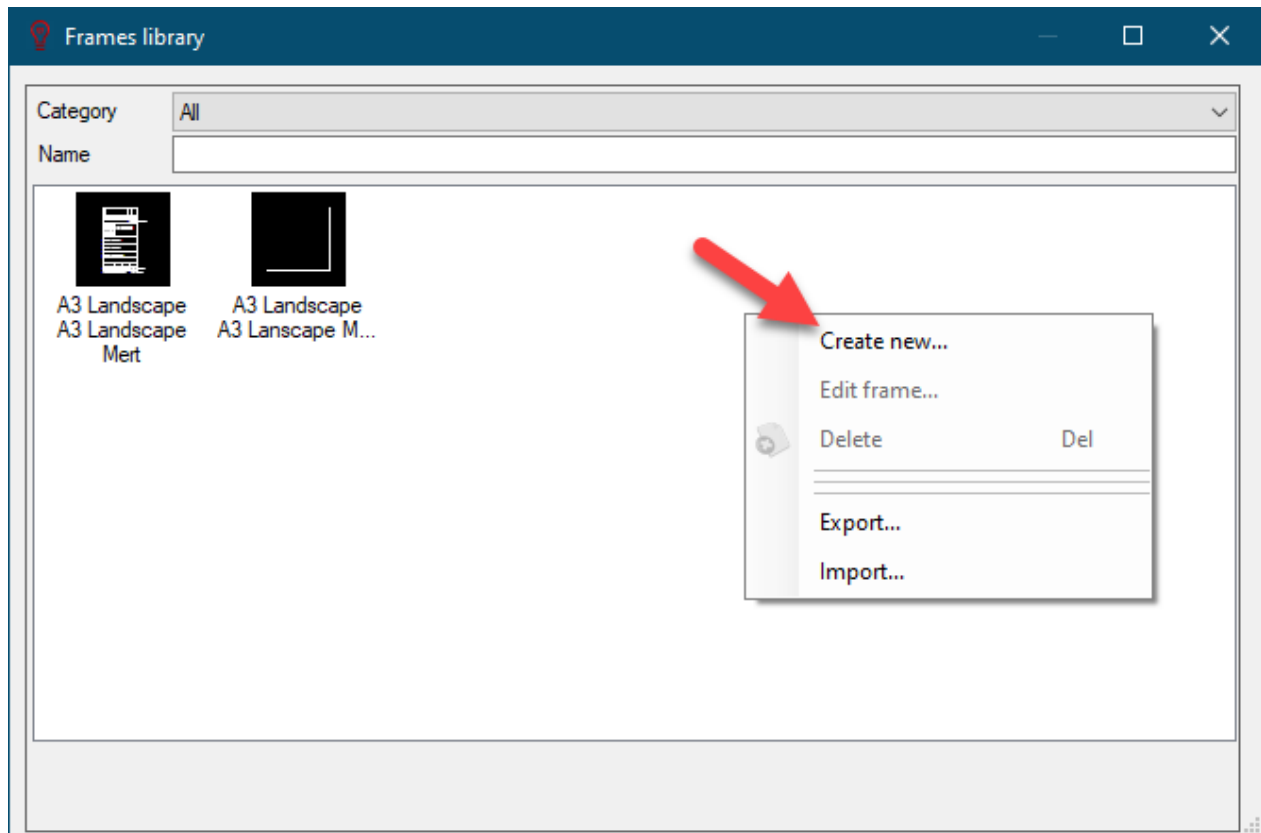


- Frame size can be set by using the Page size drop-down list and the scale can be set by using the Frame scale area.
- A document number and a sheet number can be assigned to the frame using the related text fields.
- Project info block and scale can be hidden by using the related tick boxes.

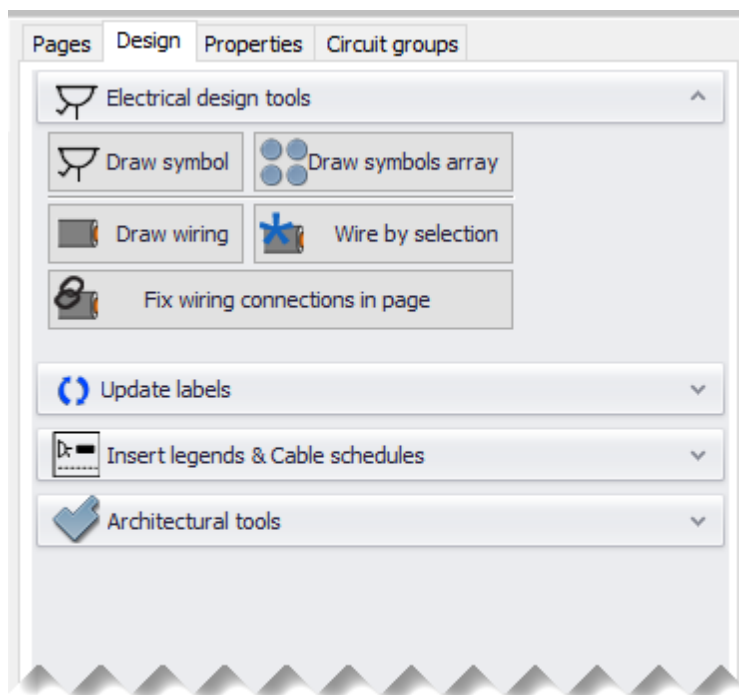
Frame style... button will initiate the Frame Style window where a custom frame can be selected by using the Custom... button.



It is also possible to create a new frame by using the Frame Builder. For further details about Frame Builder, please refer to ElectricalOM manual.



Design Tab



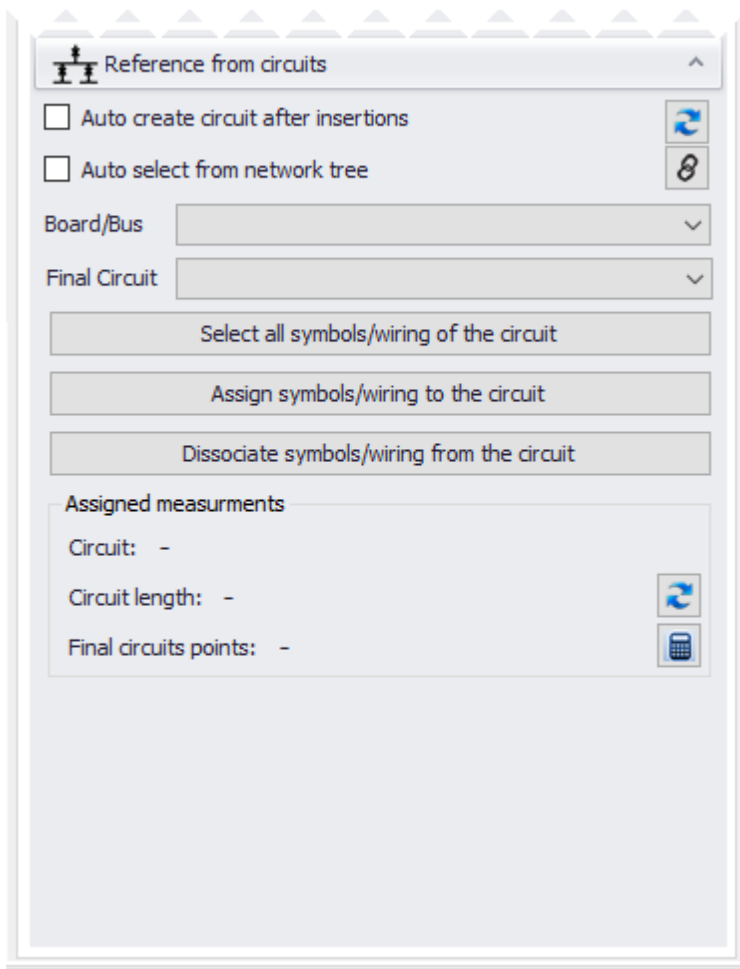
Design tab is divided into two parts.

The top part accommodates four extending section, each one having its own options once extended.

- Electrical design tools: This set of buttons are associated with electrical design tools to insert symbols and create wiring.
- Update labels: This set is used to synchronise symbol data between ElectricalOM schematic and CAD Plan Design module.
- Insert legends & Cable schedules: This set includes

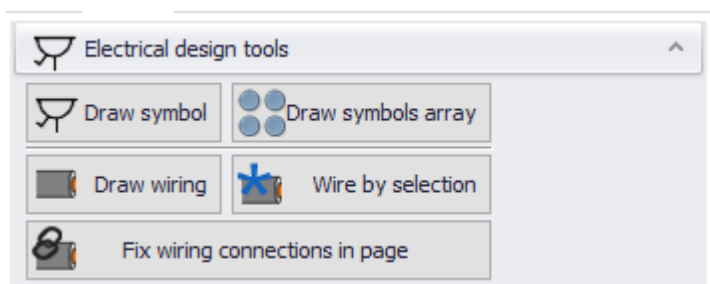
required options and setting regarding legends and schedules that can be pasted on the drawing.

- Architectural tools: This set of options are used to create architectural components like walls, columns, and areas/zones.



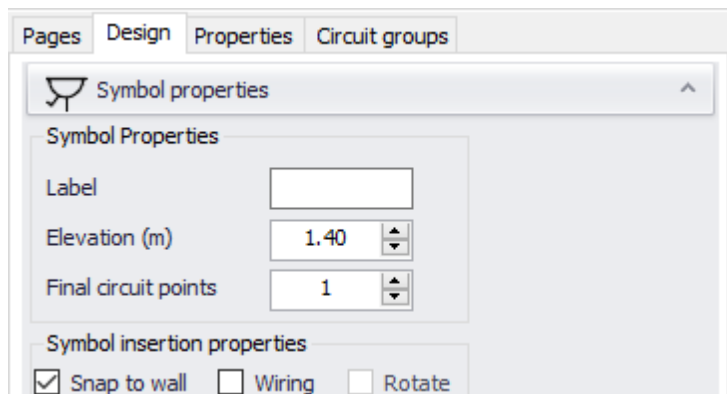
Lower part is where the user can link ElectricalOM core software components with CAD Plan Design module components, and share data in between.

Electrical Design Tools

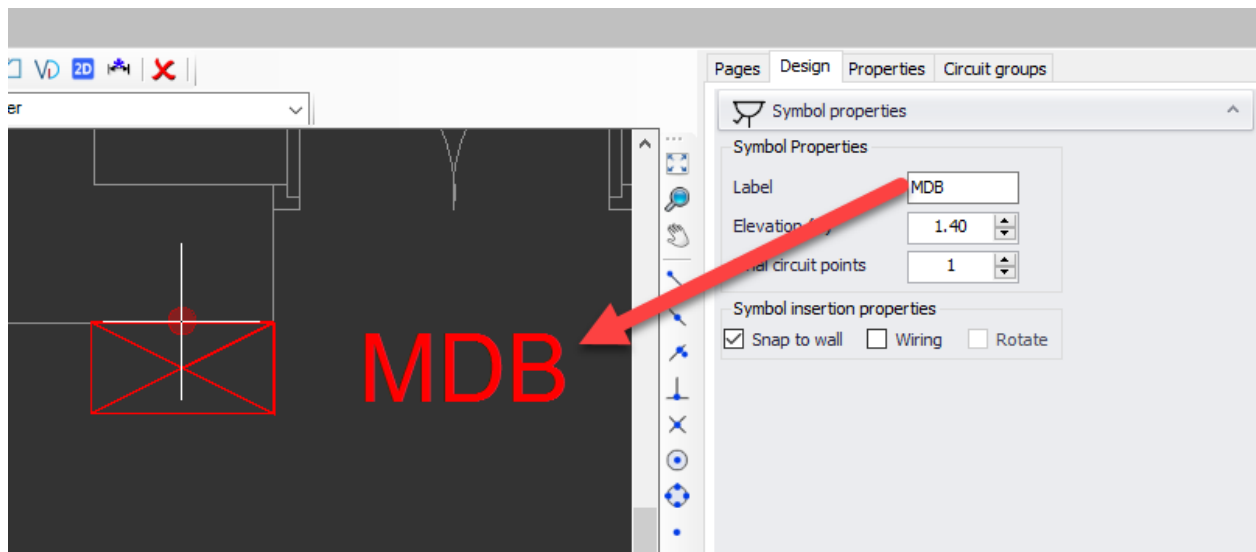


Draw Symbol

As mentioned earlier in [Symbols Area](#) section, a symbol can be inserted by drag&drop feature of Symbols Area. Once a symbol is dragged on to the drawing area, side panel will display Draw symbol settings automatically. However, it is also possible to select a symbol and then to click on Draw symbol button to switch to settings. Settings displayed here are not associated with the symbol itself but with the item the symbol will represent on the drawing.



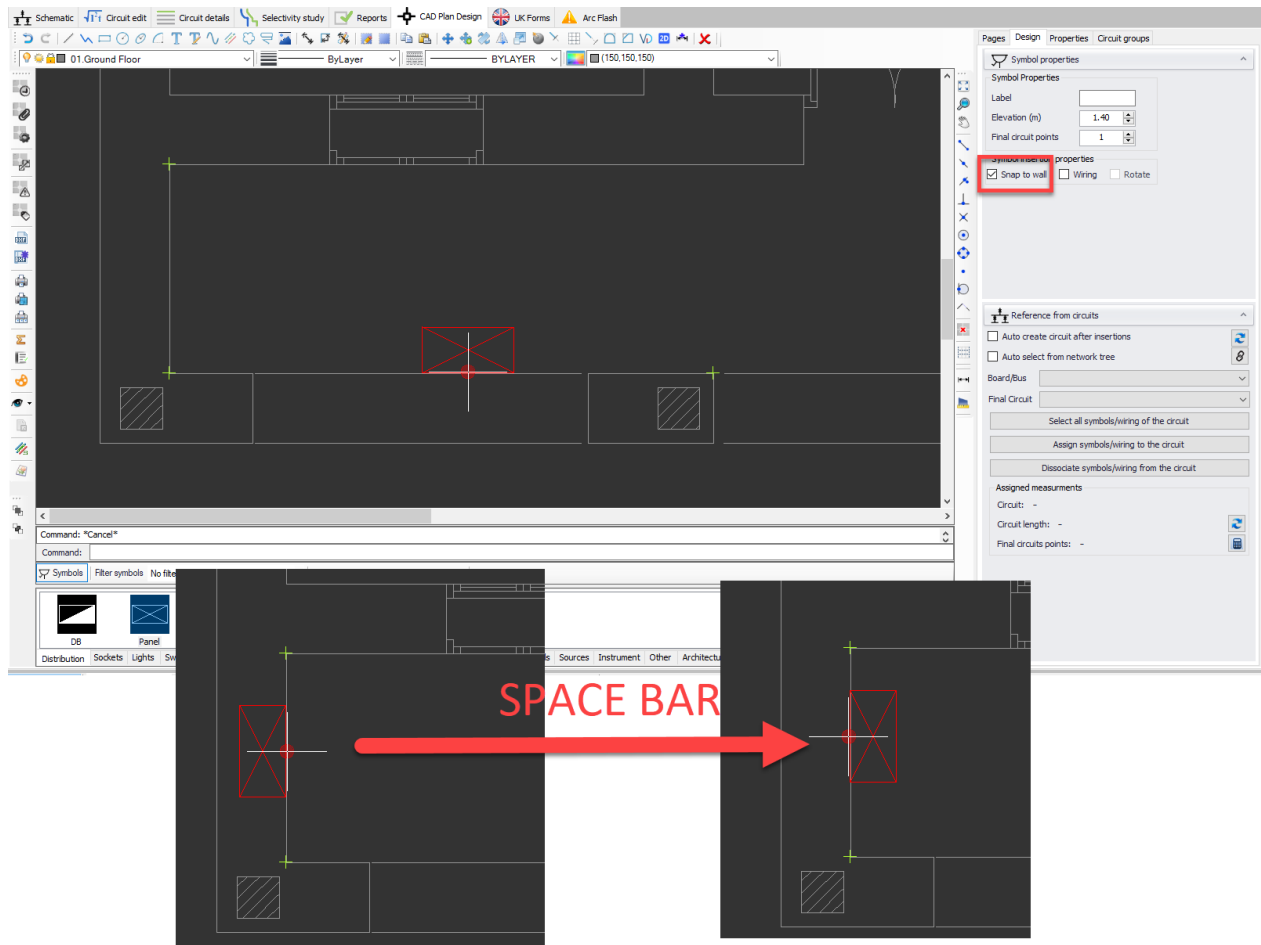
Label area is a text based input and it will be displayed next to the symbol.



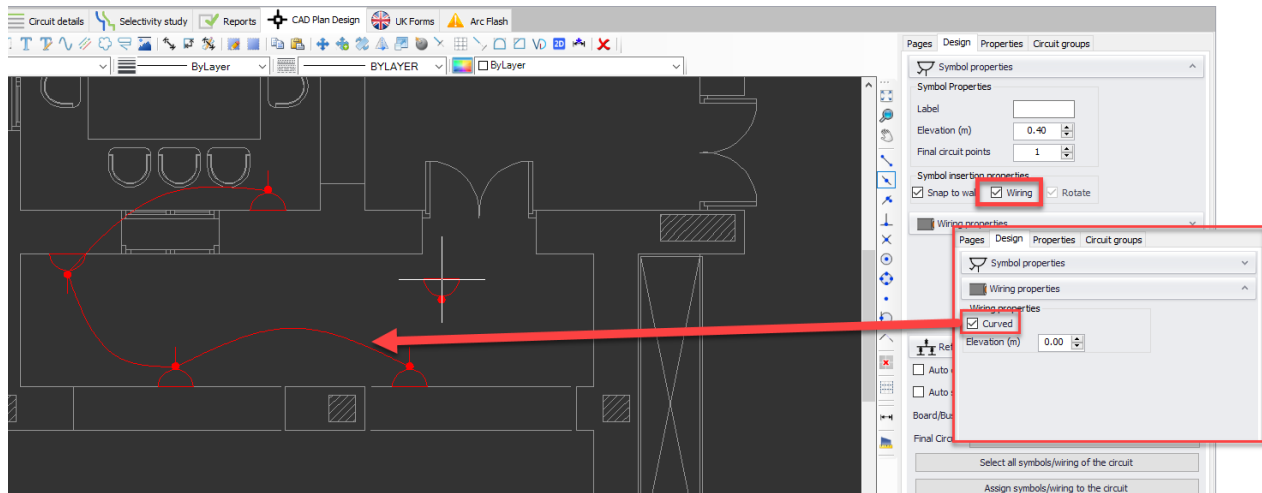
Elevation (m) represents the height of the component symbol will represent referenced to Z=0. This value will be used to estimate cable lengths and it will help CAD Plan Design module to properly display the symbol on 3D view.

Final circuit points represents the quantity of a component which will be represented by a symbol. This value will be used while populating materials list, etc.

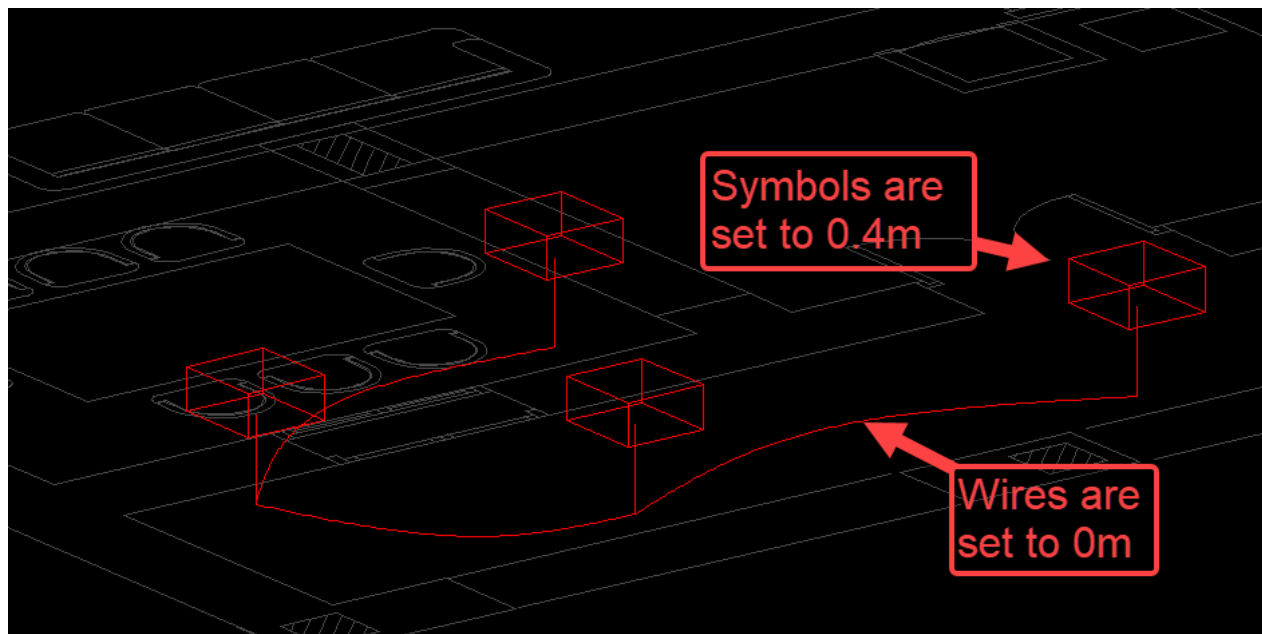
Snap to wall will rotate the symbol to match with the angle of the closest line. The symbol can be flipped by pressing the space bar.



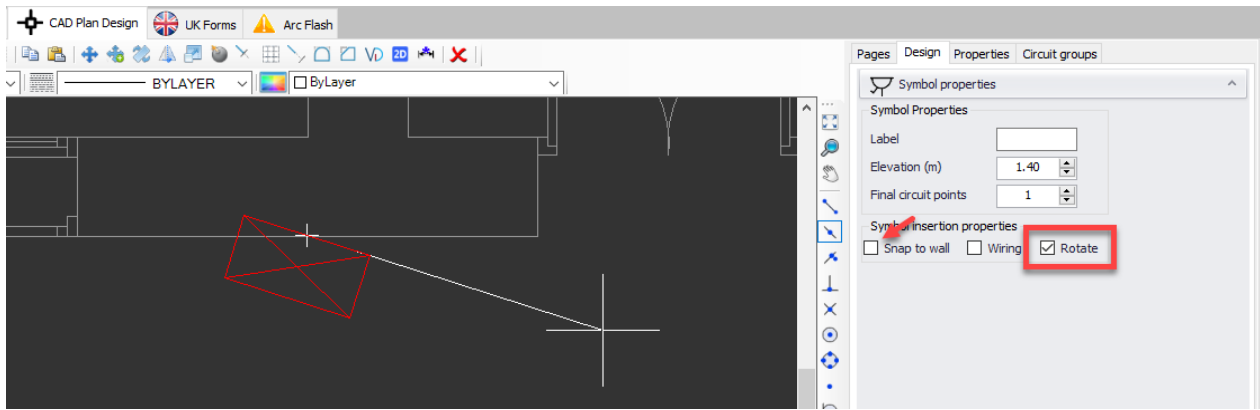
Wiring option will automatically wire inserted symbols as they are inserted. Once this box is ticked, Wiring properties will be displayed on the side panel where the user may set a drawing style, curved or straight lines, and wiring elevation in meters.



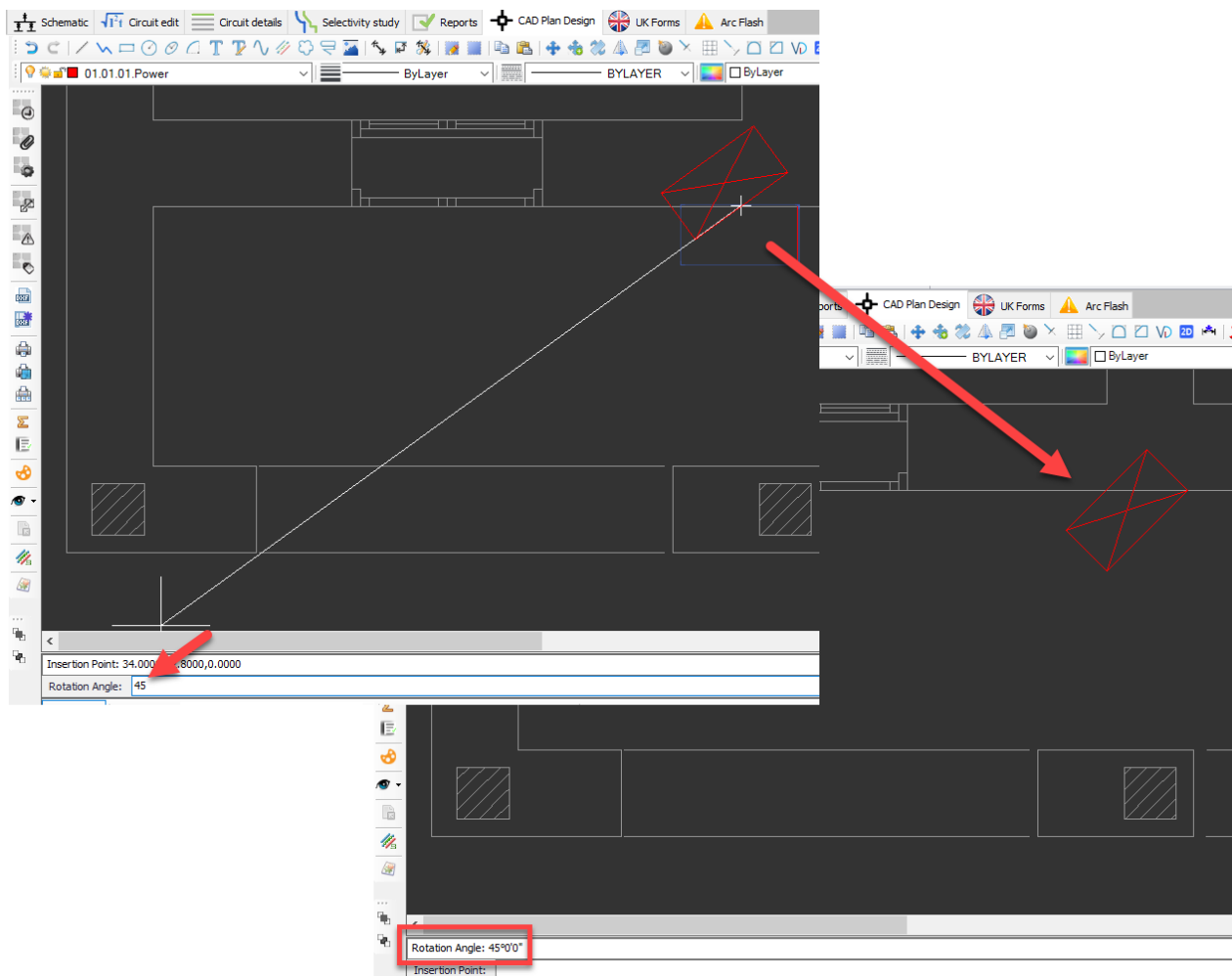
As a representation of elevation settings, please see the picture below which is an extract from 3D view of CAD Plan Design module. Here, the symbols are set to 0.4m above Z=0, which is the floor plan, and wiring is set to be at 0m.



Rotate will let the user to set a rotation after a symbol is inserted. In order to be able to use this option, Snap to wall option must be unticked.



A rotation angle can also be specified using the command line area and pressing Enter.



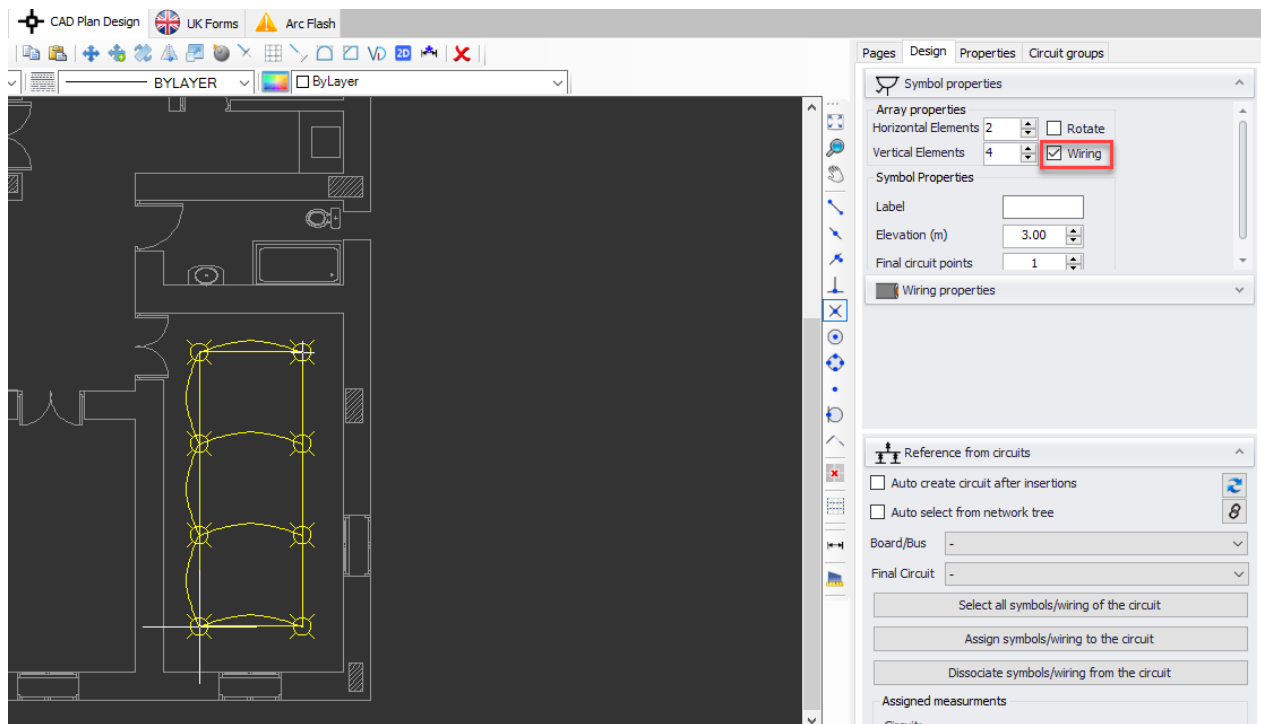
Draw Symbols Array

It is possible to create a rectangular array using this option. A symbol must be selected prior to clicking on Draw a symbol array button. Horizontal and vertical elements values can be set before selecting a start point but also be modified dynamically as long as a second point is not selected. A second point can also be specified using the Command line area, by typing in a coordinate (e.g. X,Y).

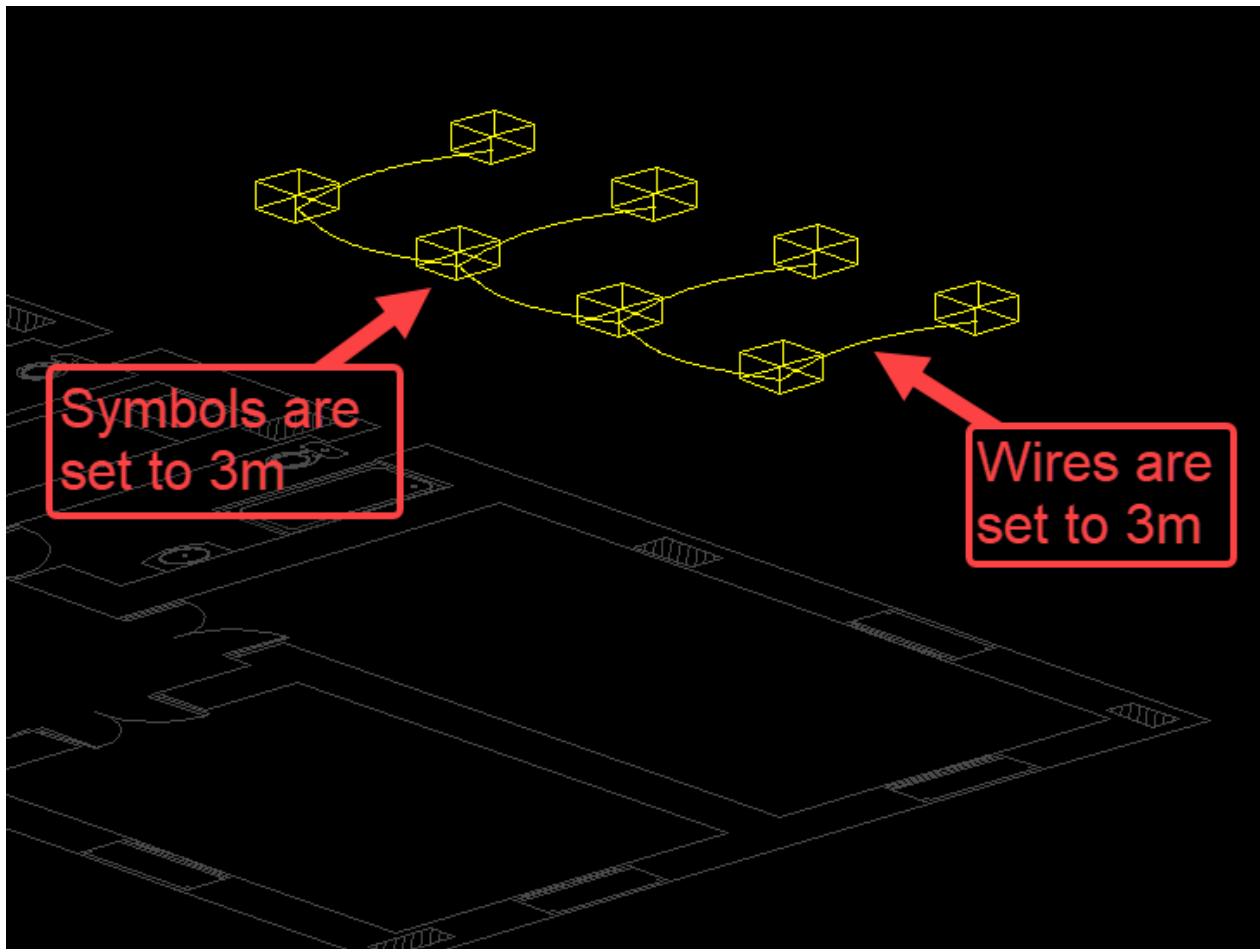
Similar to Draw symbol settings, Rotate option will let the user to rotate the array after it is finalised by defining a second point, and Wiring option will automatically wire the symbols together.

Symbol Properties will affect each element of the array. Label field is used to set a label for the array and it will be displayed next to each element of the array. Elevation will set all the array elements' height to the value entered. Final circuit points will set a quantity for each element of the array.

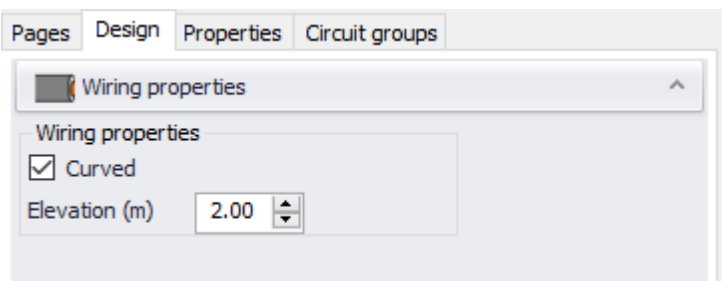
Once the array is finalised, each component will act as individual and not as a group.



As a representation of elevation settings, please see the picture below which is an extract from 3D view of CAD Plan Design module. Here, the symbols are set to 3m above the floor plan, and wiring is also set to be at 3m.



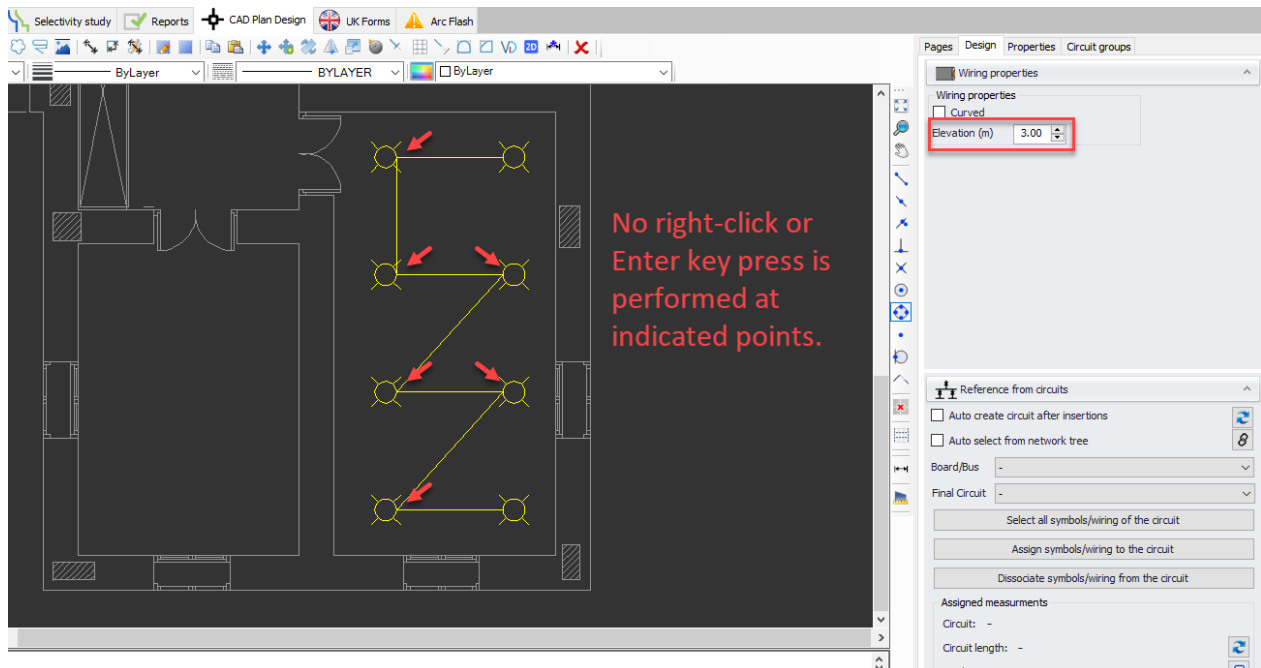
Draw Wiring



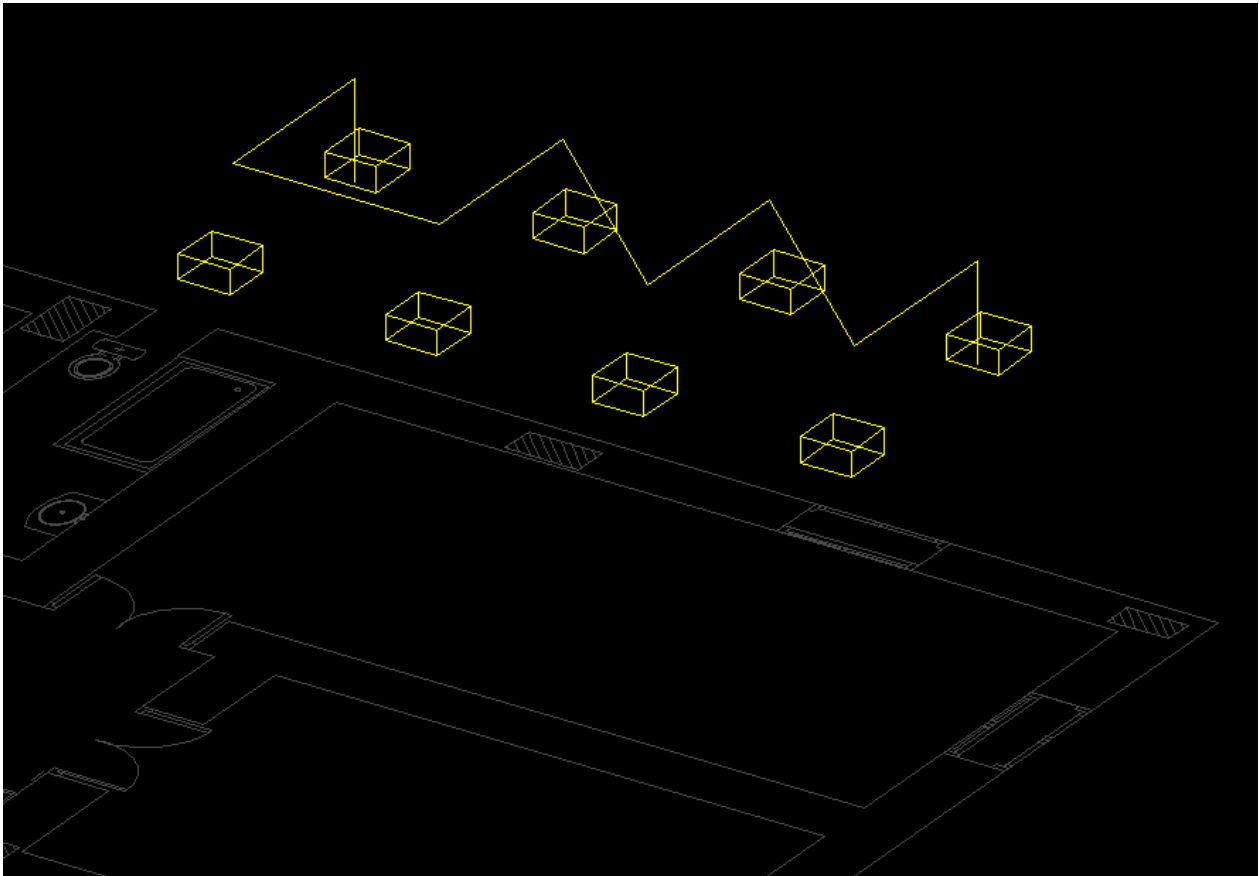
Within CAD Plan Design, cables can be drawn using this option. Once clicked on, Wiring properties window will be displayed where Curved tick box and Elevation fields can be set. Drawing a cable will start from Start point, which can be entered into the command line area, or by clicking on a point inside the drawing area, and will continue until the user perform a right-click or press Enter key. After right-clicking or pressing the Enter key, CAD Plan Design will pause drawing wiring and wait for another start point entry. A second right-click or Enter key press will finalise the drawing. The wiring from the start point up to the first right-click or Enter key will be considered as a single entity on the drawing, and when selected it will be selected as a whole, where each right-click or Enter key press will break the wiring at these points.

If it is required to connect multiple items, then, right-click/Enter key press must be performed at each connection otherwise draw wiring feature will ignore any points in between and count the wiring as a single cable from the start point to the end point.

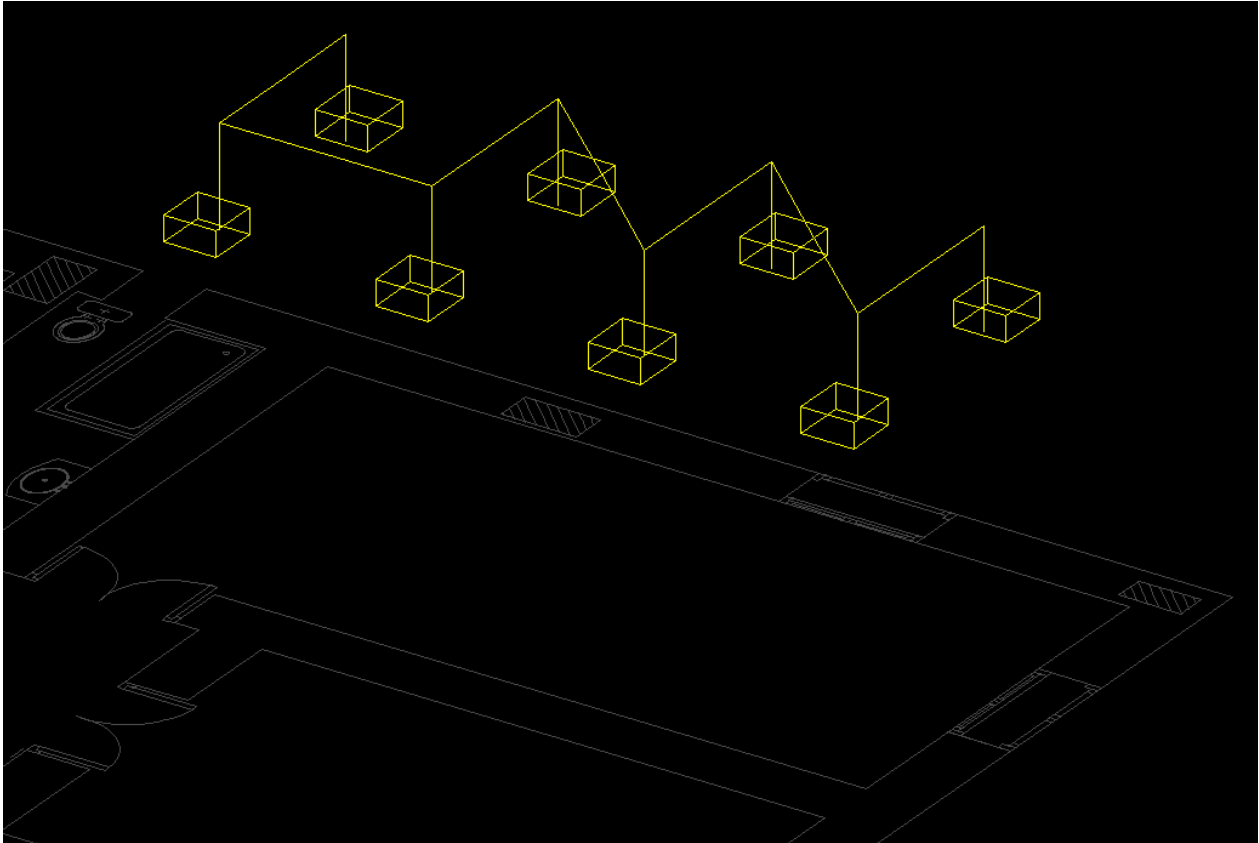
As an example, see pictures below. Here we have eight lighting points set at 2.2m, and we will draw wiring at 3m to make it easier to understand the concept. First, wiring is started from top right, then, to top left, and one level down and so on, but no right-click or Enter key press is performed at symbols.



If we look at 3D view, we will see that the wiring starts from top right and ends at bottom left symbol. There is no connection to other symbols even though we clicked on them while drawing the wiring.



Now, we will follow the same procedure, however, this time we will hit Enter key every time we click on a symbol. This will break the cable and connect it to the symbol at these points.

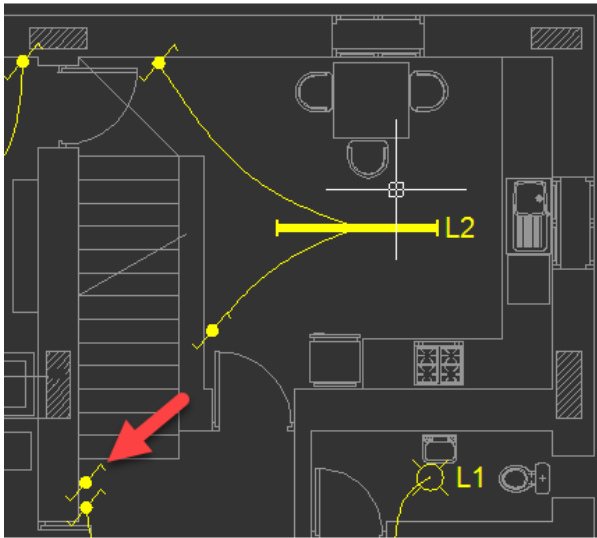


Drawing a reference wire

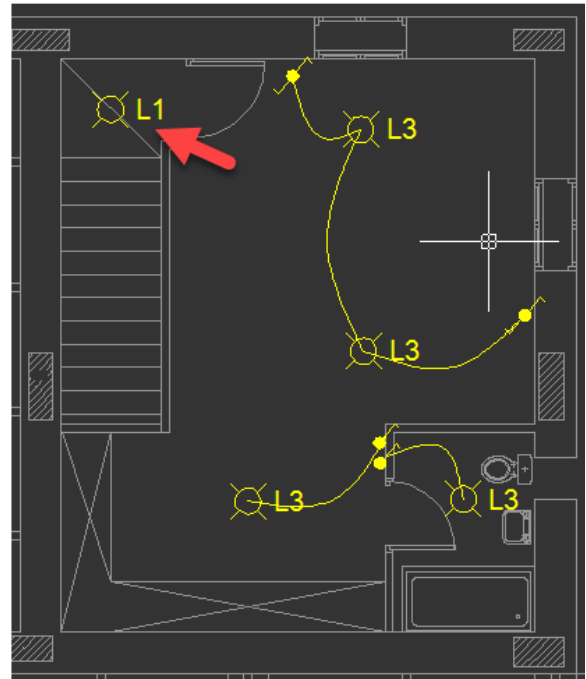
Sometimes it is not possible to draw wiring with start and end points at the same time. For example, if we want to show the wiring of a light switch which operates a light on the upper floor, or we may want to show a feeder cable for another board which is located on a different layer. In these cases, Properties Tab can be used. We will see how this is achieved here, however, other features of the Properties tab will be discussed in [Properties Tab](#) section.

Let's assume we have a switch at the ground level which will operate the light on the first floor as shown below. We will also assume the wiring length from the switch to the light on the upper level is 8 metres.

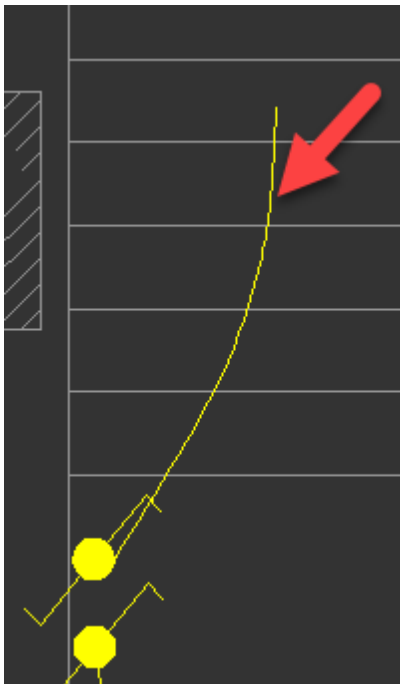
Ground Floor



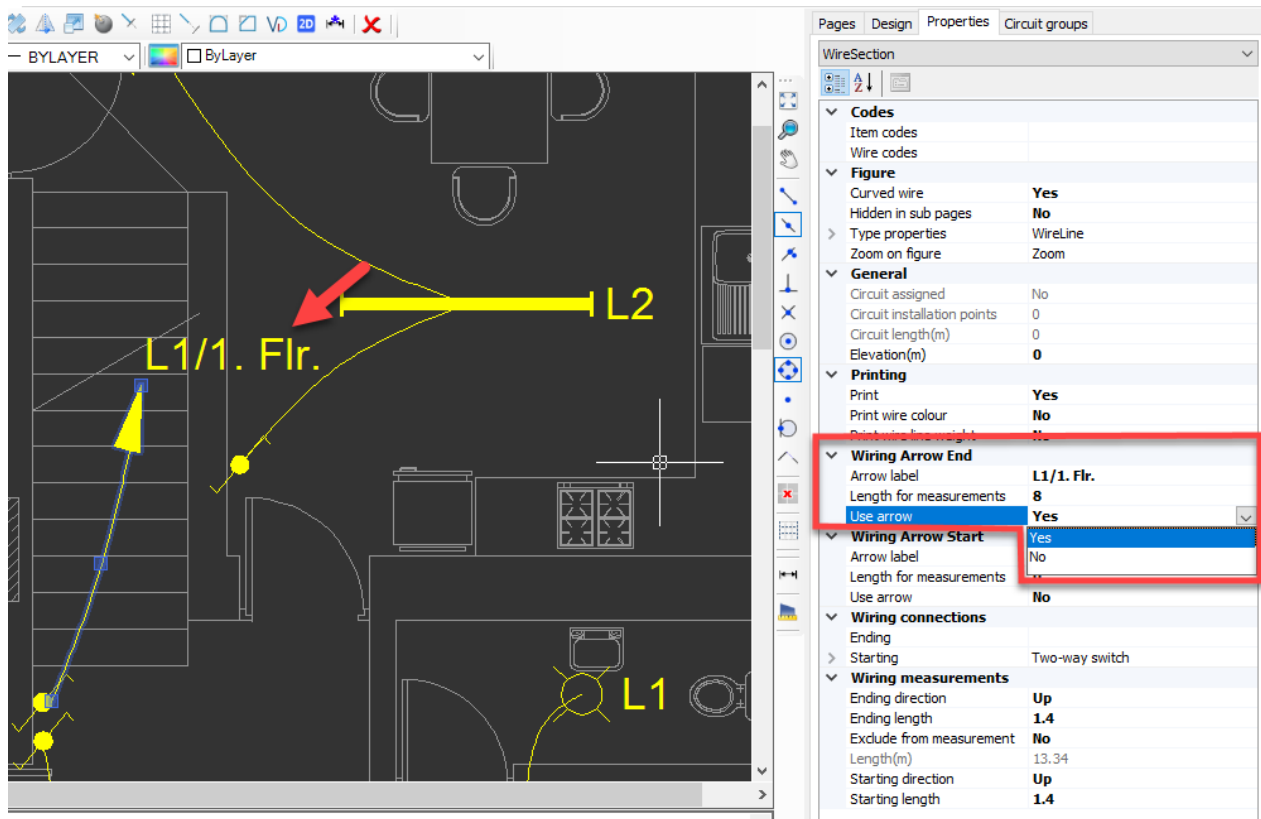
First Floor



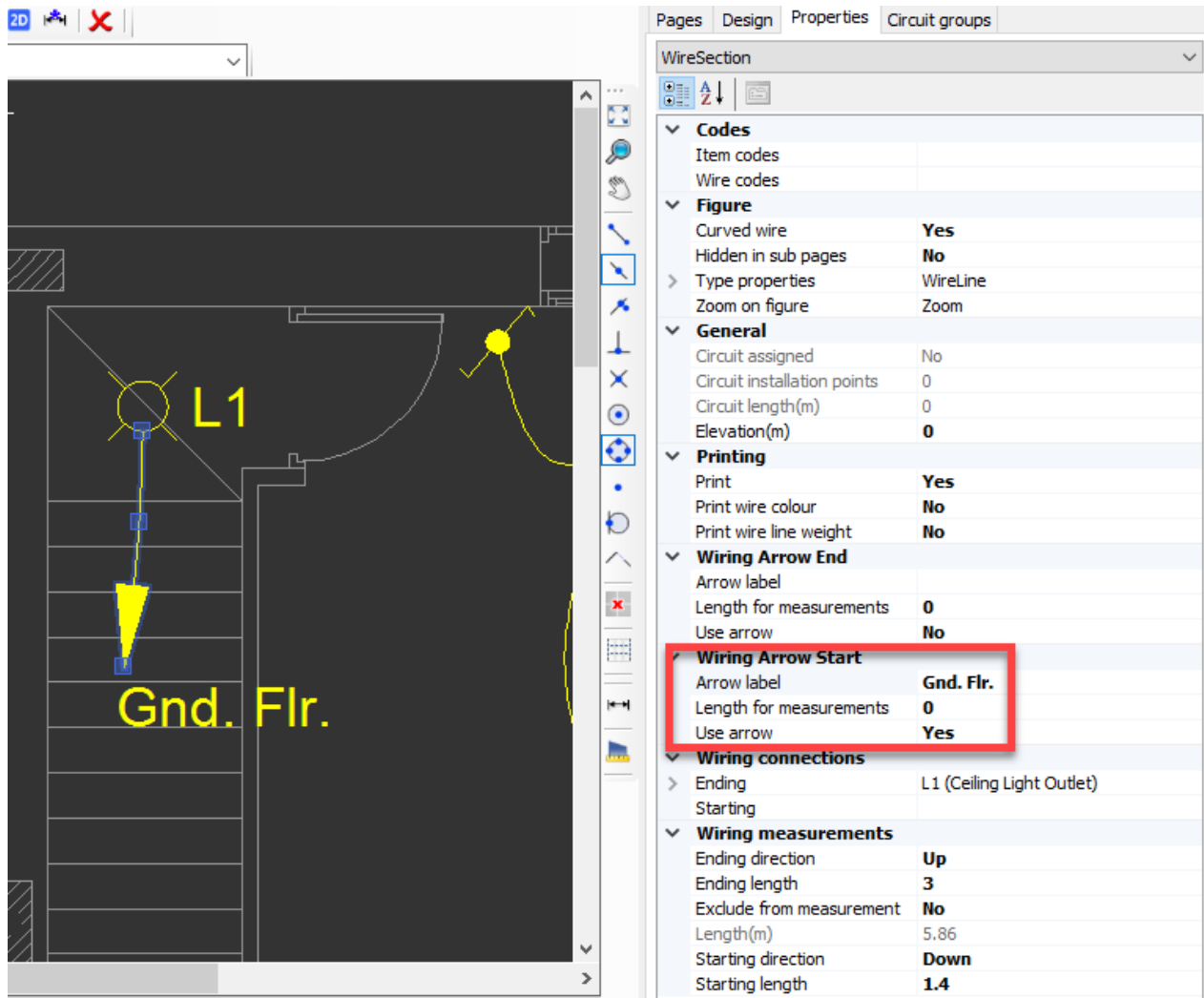
Now, using the Draw wiring button, we will draw a piece of wiring as shown below.



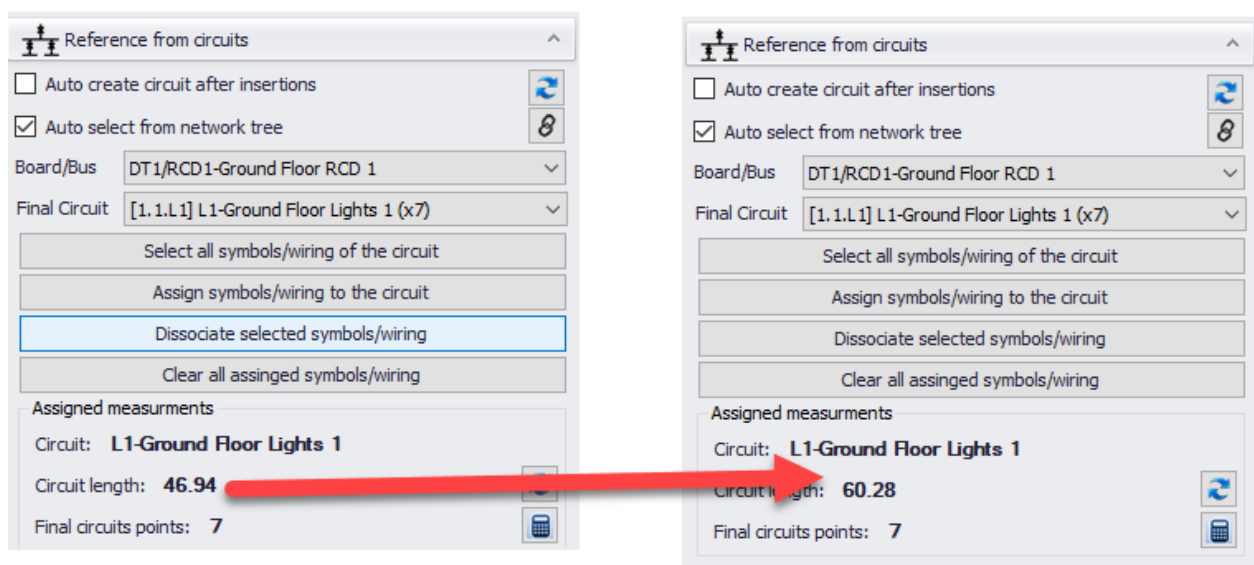
Then, we will navigate to the Properties tab, and define a label and a length for and end point arrow, see below.



Now, visually we managed to show the wiring from the switch. We can also do a similar thing for the lighting point to show the wiring to it as below. However, this time we will not define a length for this piece of wiring, because when we assign the wiring to a circuit we want this extra length to be added only once.



Once this pieces are assigned to a circuit, CAD Plan Design will add the specified length to the total length of the wiring which is 8 metres for this example, see below.



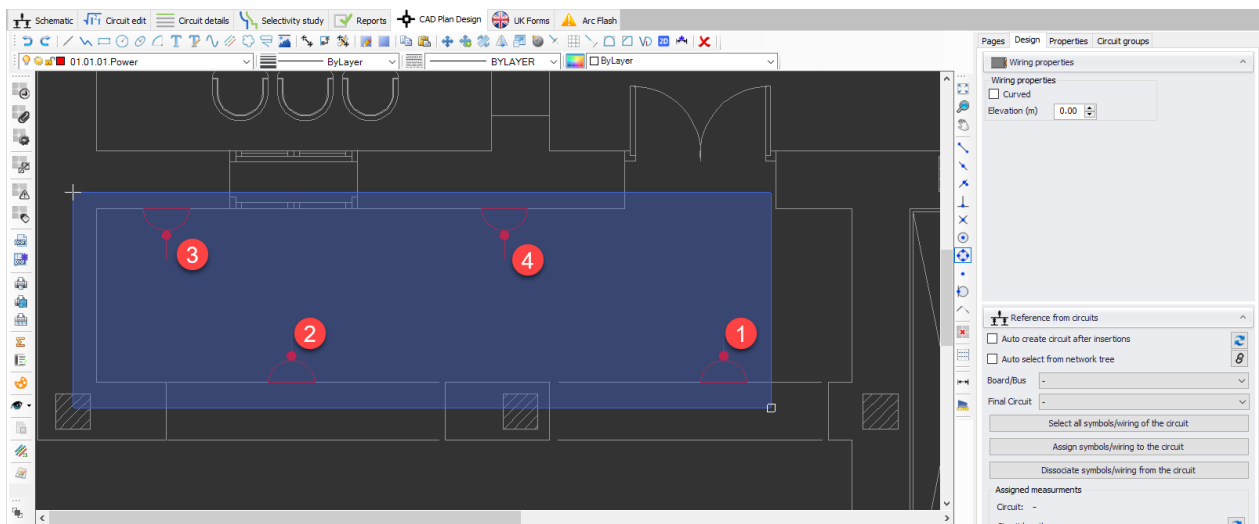
For further details about assigning symbols to circuits, please refer to [Assign CAD Plan](#)

[Design symbols to ElectricalOM circuits.](#)

Wire by Selection

It is also possible to select symbols and let CAD Plan Design module to do the wiring automatically. When the user click on Wire by selection, Wiring properties screen will be displayed on the side panel, curved line option, and elevation can be set from here. When the cursor is within the drawing area it will look like a small square while selection is ongoing. Symbols can either be selected by drawing an imaginary rectangle which will highlight the area inside to see if the symbols are within the range, or by directly clicking on the symbols.

In the example below we placed symbols in numbered order, and selected them using the rectangle feature. Selection is done by clicking at the upper left corner following another click at the lower right corner. User may continue select more symbols.

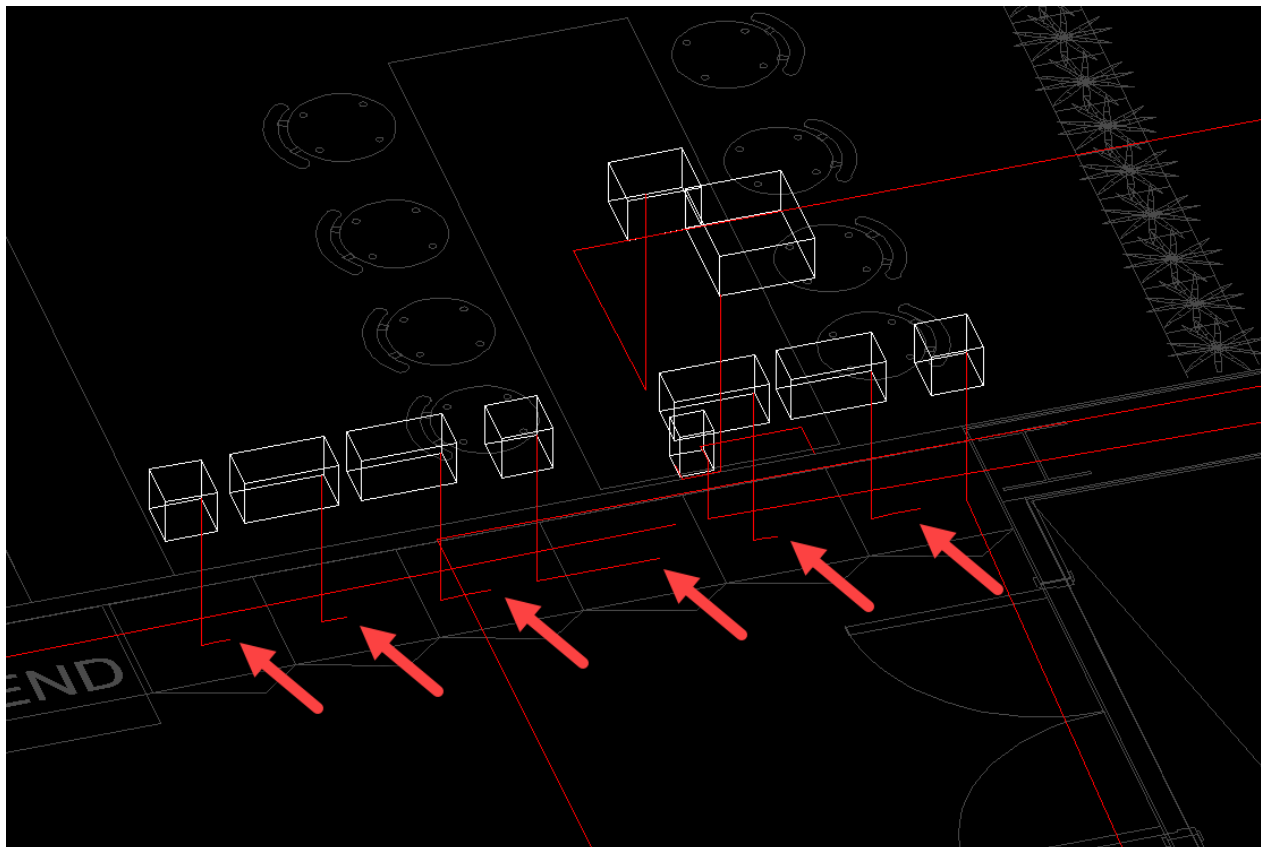
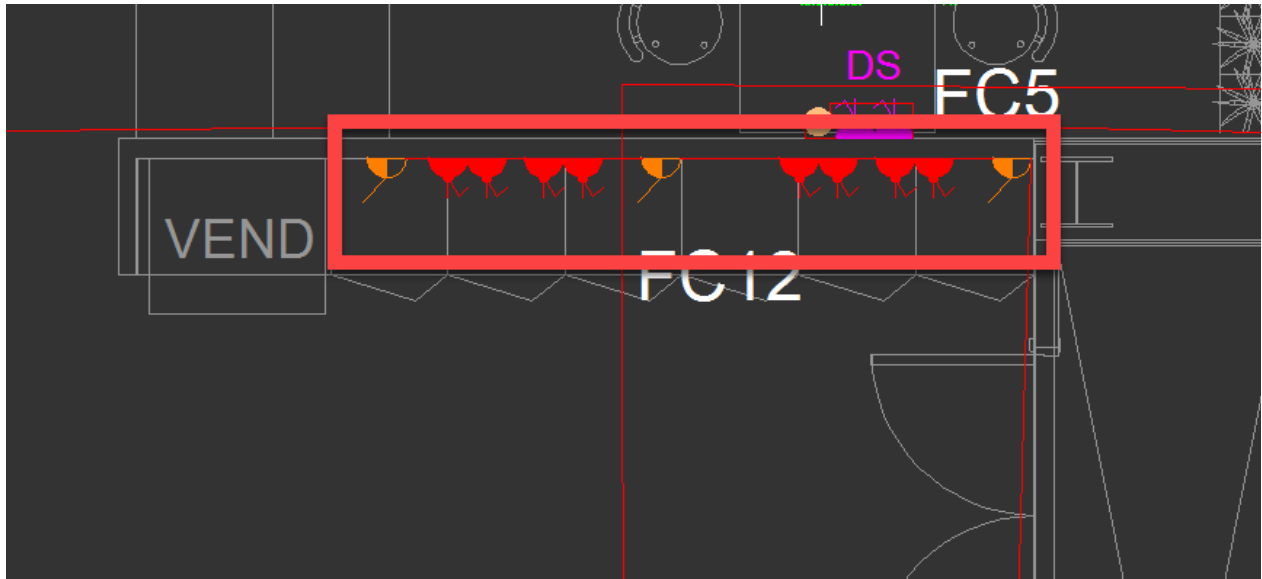


A right-click or an Enter key press will finalise selection and wire the symbols selected. As it can be seen below, CAD Plan Design wired the symbols according to their insertion order.

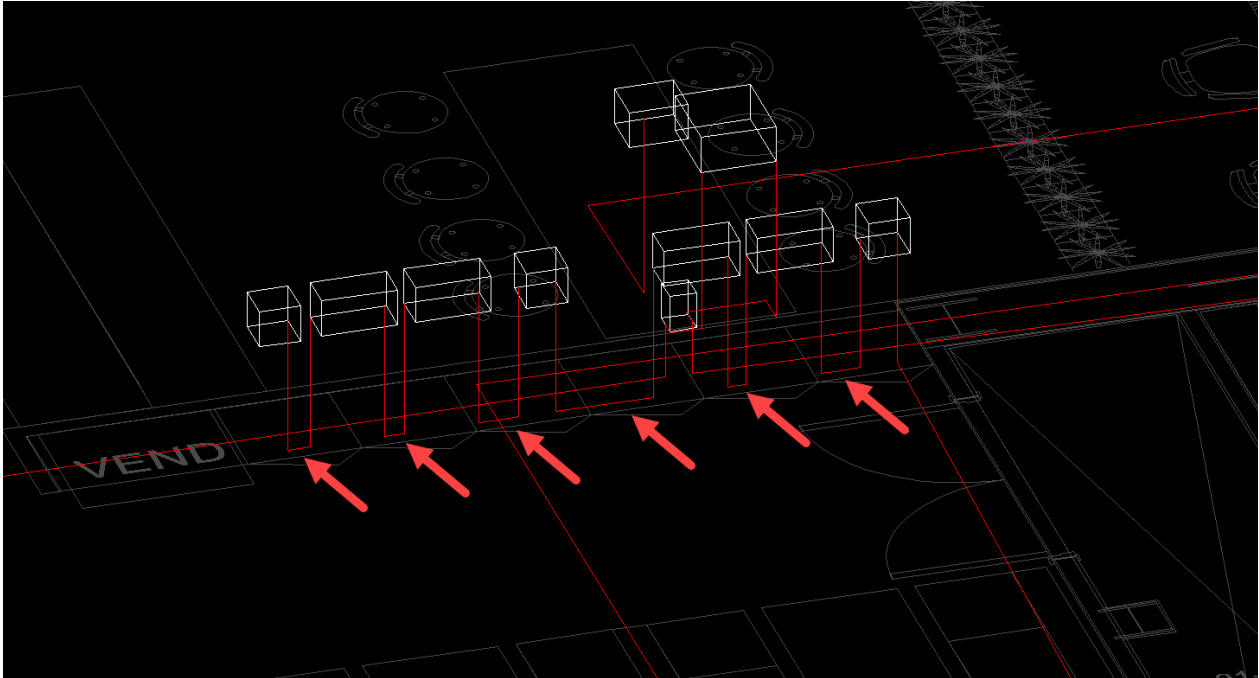
Fix Wiring Connections in Page

It is advisable to click on this button occasionally during design. This action will try to search and fix the wiring connections between circuits, and wiring sections.

If we look at the occasion below, there are couple of sockets connected together in series. However, in 3D view, there are some issues with their connections.

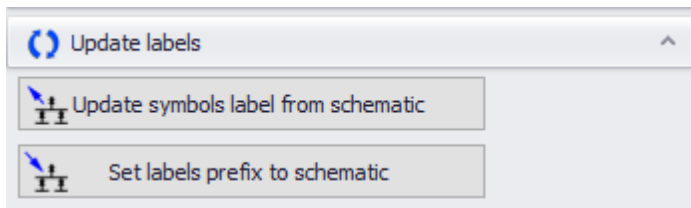


Fix Wiring Connections in Page will scan the components of the selected layer and try to fix if there are any issues, see below how the wiring is fixed.



Update Labels

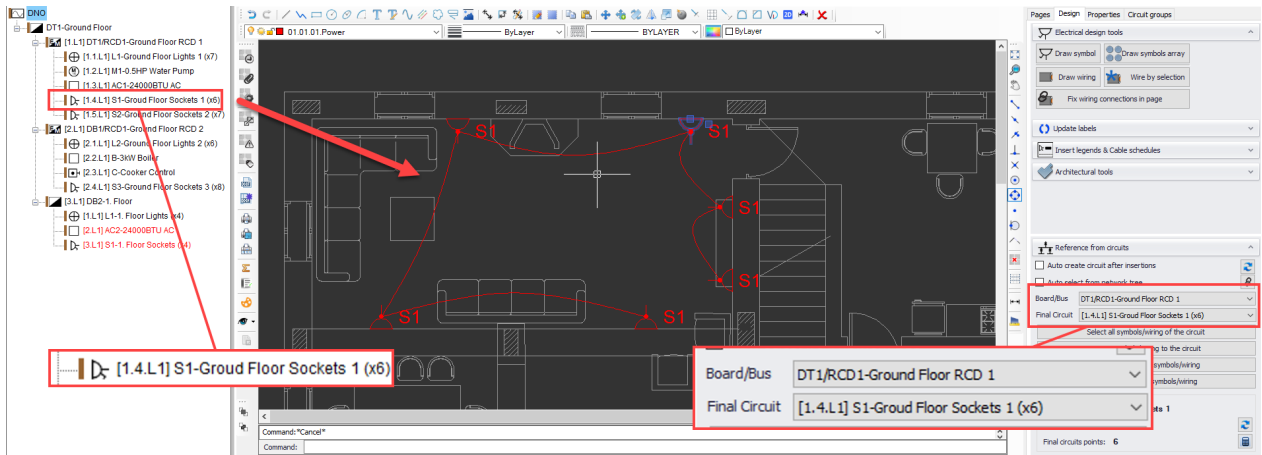
Update labels section will reflect data either from ElectricalOM schematic to CAD Plan Design drawing or vice versa. These buttons requires the symbols and wires to be associated to a circuit within the ElectricalOM network.



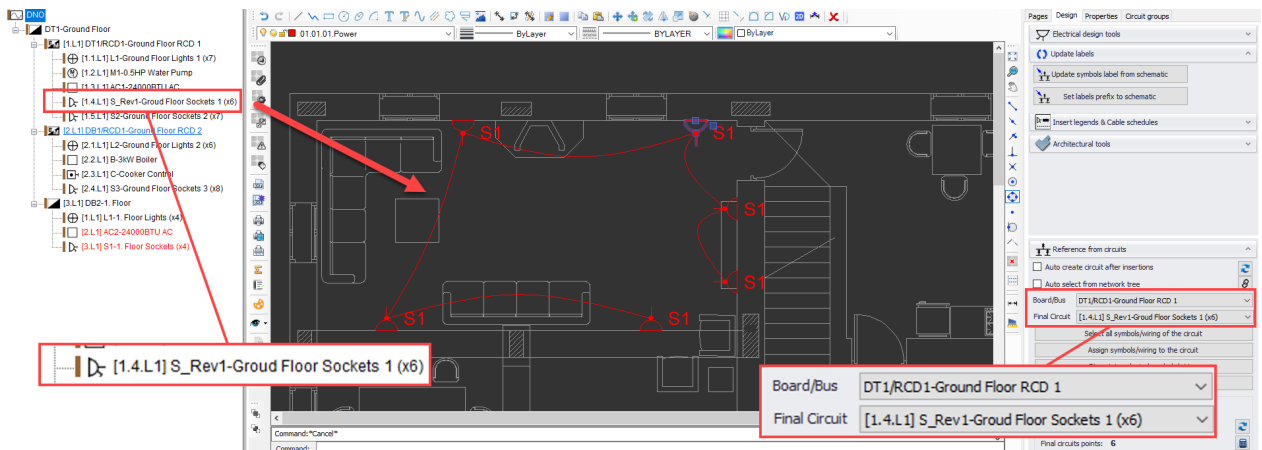
Update Symbols Label from Schematic

This option will override the circuit prefix of the symbols of CAD Plan Design module, and copy the prefixes of all associated circuits of ElectricalOM to CAD Plan Design.

Below we have a socket circuit, S1 which is already associated with a circuit of ElectricalOM. For further details about how to associate symbols with circuits, please refer to [Reference from ElectricalOM Circuits](#) section.



If we change the prefix of this circuit via ElectricalOM, the change will not be reflected to CAD Plan Design automatically. As a result, we will have a circuit with a prefix of S_Rev1 (ElectricalOM) which is associated with S1 of CAD Plan Design. Bear in mind, even if the prefixes do not match, the association still maintained.

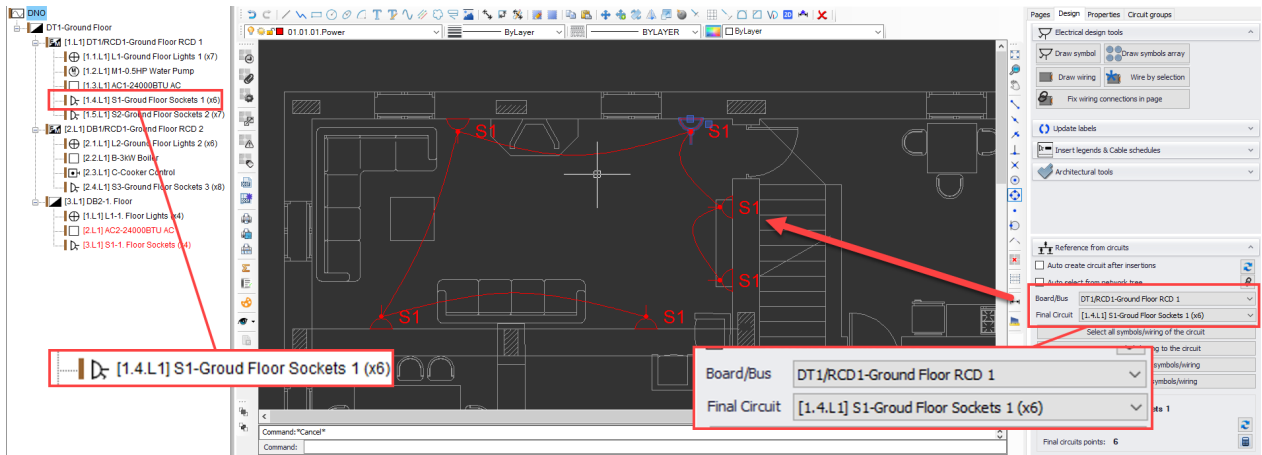


In order to copy prefixes from ElectricalOM to CAD Plan Design, Update Symbols Label From Schematic button is used. This will update all the prefixes of CAD Plan Design to ElectricalOM prefixes.

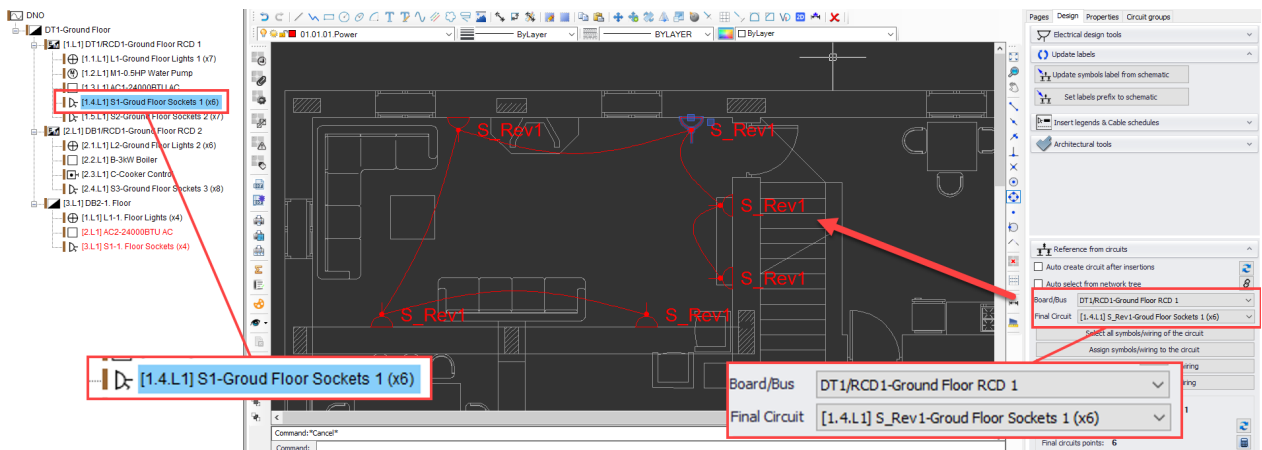
Set Labels Prefix To Schematic

This option will override the circuit prefix of the symbols of ElectricalOM, and copy the prefixes of all associated circuits of CAD Plan Design to ElectricalOM

Below we have a socket circuit, S1 which is already associated with a circuit of ElectricalOM. For further details about how to associate symbols with circuits, please refer to [Reference from ElectricalOM Circuits](#) section.



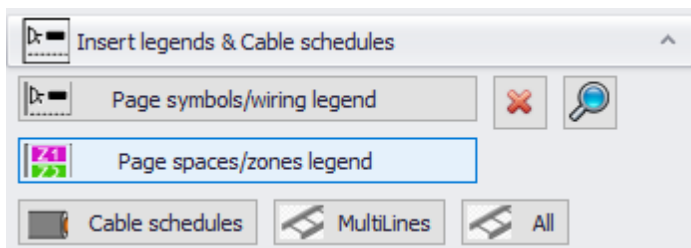
If we change the prefix of these symbols via CAD Plan Design module, see [Properties Tab](#), the change will not be reflected to ElectricalOM automatically. As a result, we will have a circuit with a prefix of S1 (ElectricalOM) which is associated with S_Rev1 of CAD Plan Design. Bear in mind, even if the prefixes do not match, the association still maintained.



In order to copy prefixes from CAD Plan Design to ElectricalOM, Set Labels Prefix To Schematic button is used. This will update all the prefixes of ElectricalOM to CAD Plan Design prefixes.

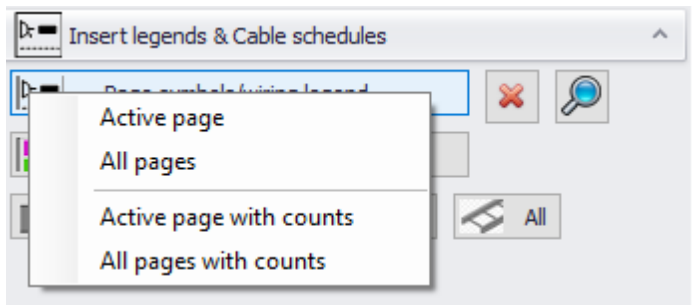
Insert Legend and Cable Schedule

There are a few legends and schedules the user may add on to the CAD Plan Design drawing. This section of side panel provides buttons to easily select required field.

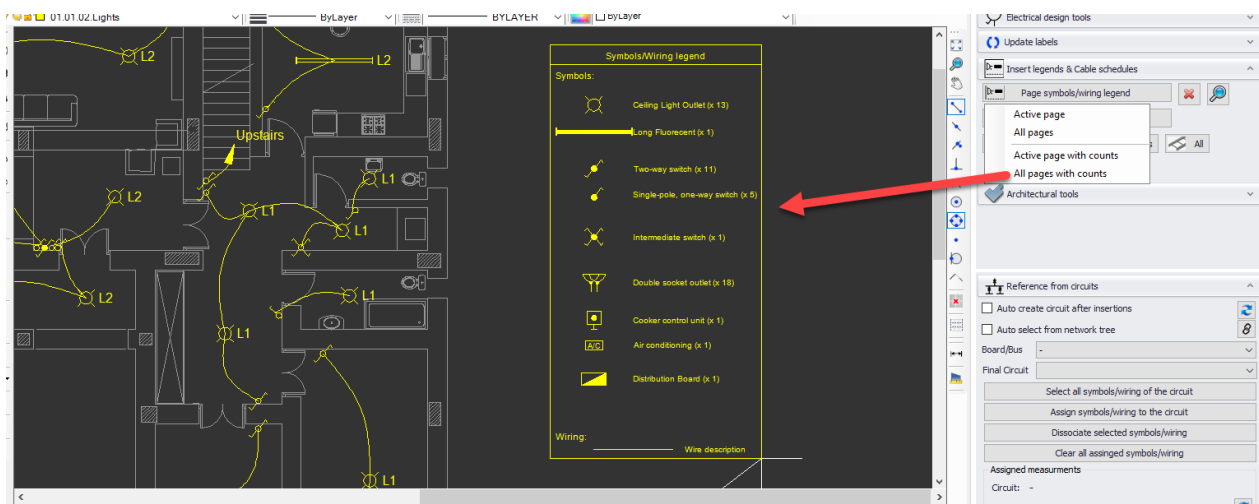


Page Symbols/Wiring Legend

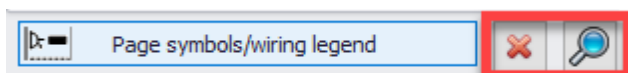
This button will display various options user may choose from.



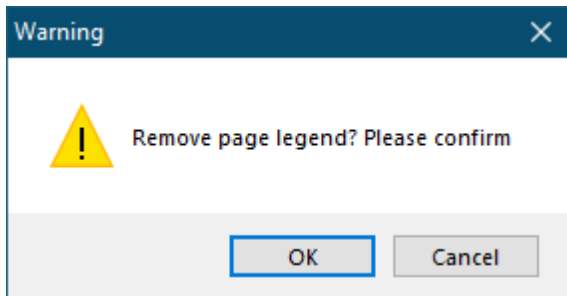
- Active page: This will create a symbols legend consists of the symbols only used on the active layer.
- All pages: This will create a symbols legend consists of all the symbol used throughout the CAD Plan Design project.
- Active page with counts: This will create the same legend as Active page option but with quantities. If a symbol is used on other layers than the active one, only the active layer quantities will be displayed
- All pages with counts: This will create the same legend as All pages option but with quantities.



Apart from above options, there are also two buttons located next to the Page Symbols/Wiring Legend button.



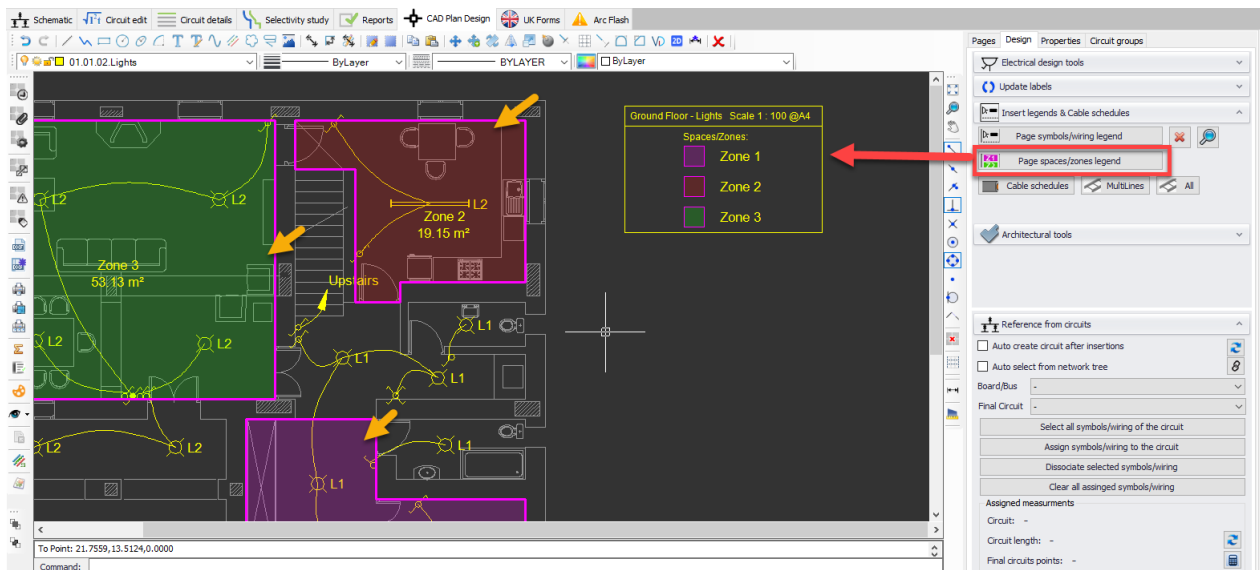
- Remove Page's Legend. Once this button is clicked ElectricalOM will confirm and delete the legend.



- Locate Legend in Page. This is used to zoom to the legend.

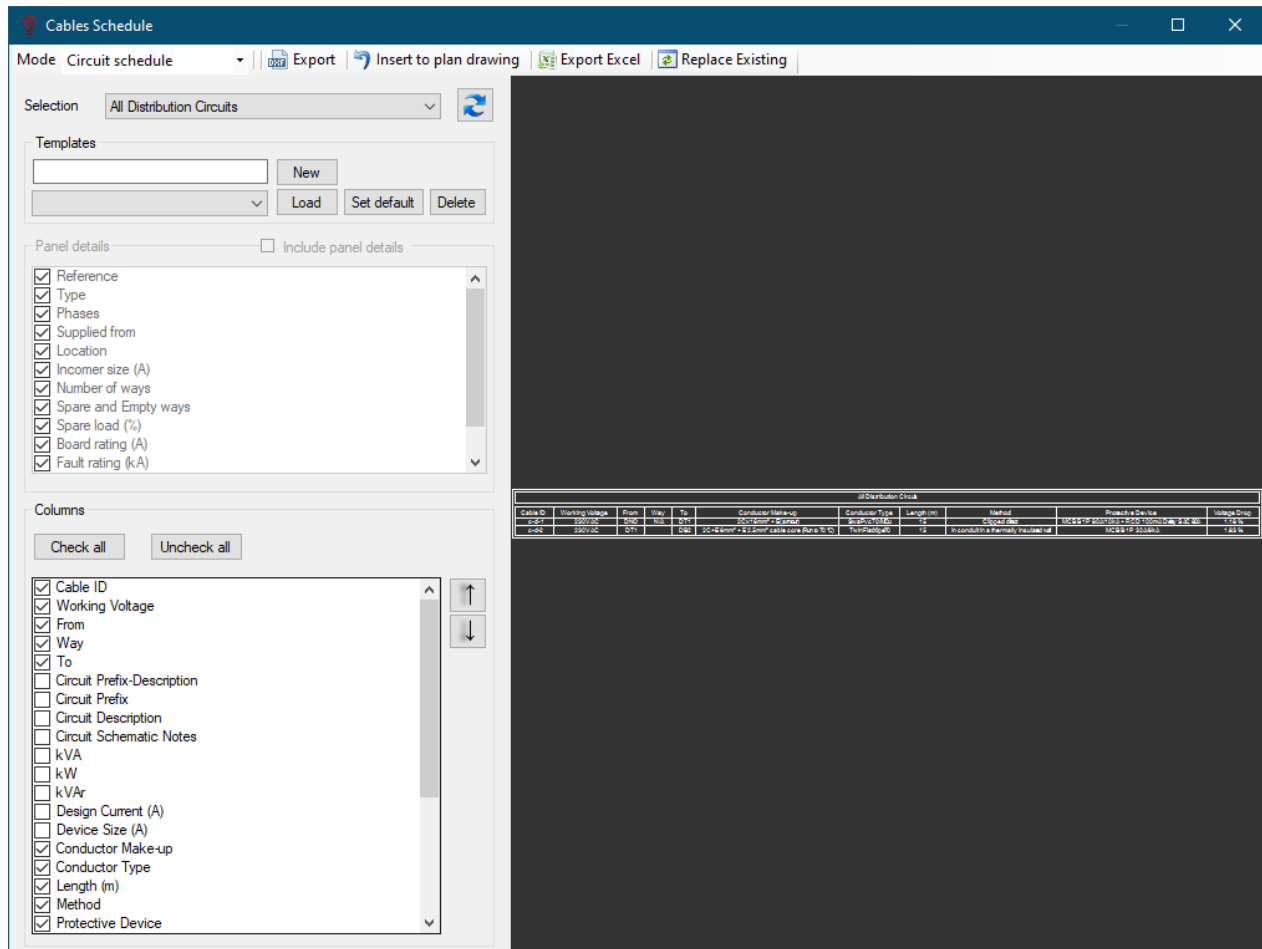
Page Spaces/Zones Legend

This button will insert a Spaces/Zones legend to the CAD Plan Design drawing. For further details about creating zones, please refer to [Architectural Tools](#). This option will insert a legend which includes zones related with the active layer only.



Cable Schedules

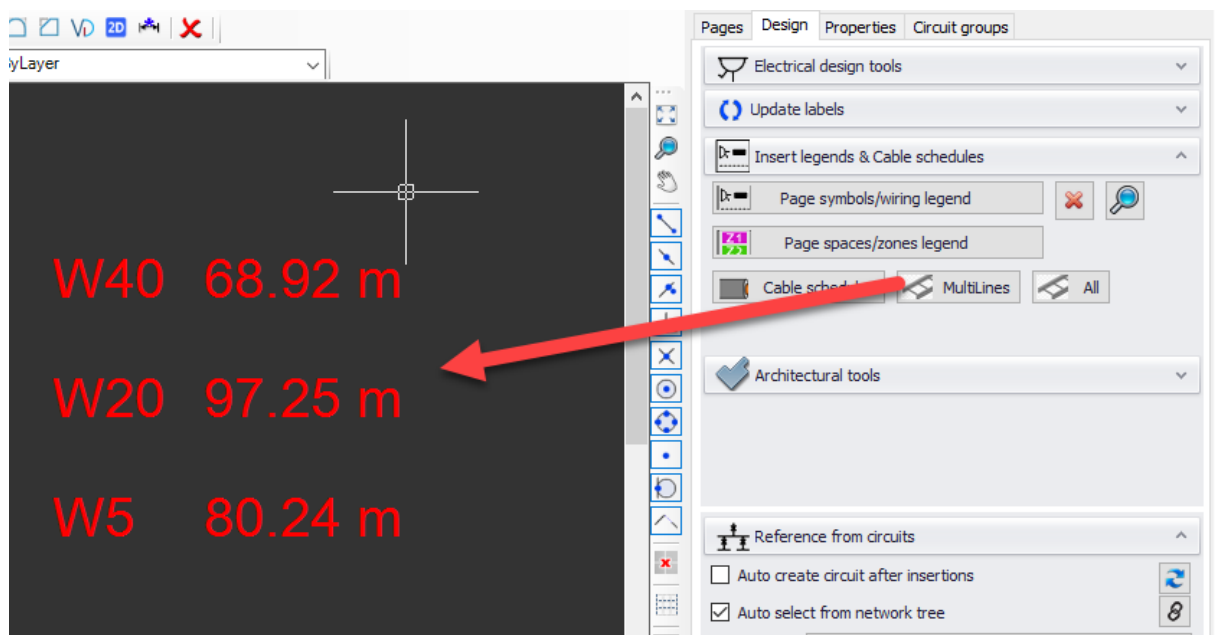
This is the same function as ElectricalOM's cable schedules. Please refer to ElectricalOM manual for details.



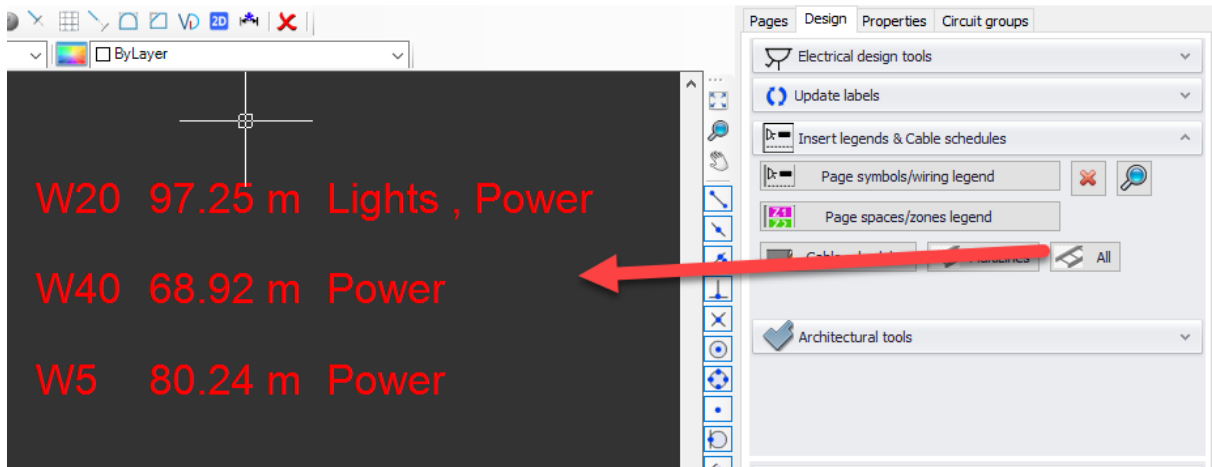
Multilines

As mentioned in previous chapters, multilines may also be used to represent cable trays. CAD Plan Design has a multiline schedule feature to enable users to create a schedule for cable trays used within their projects.

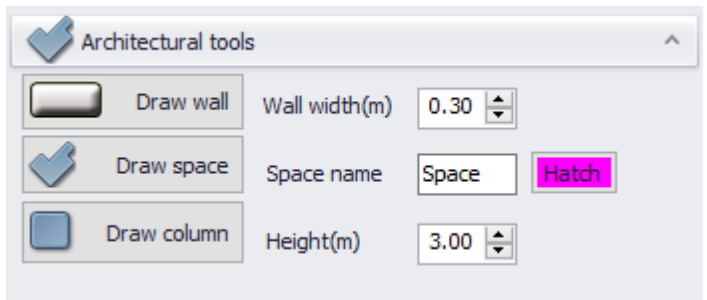
- Multilines button will list all used multilines with their total lengths.



- All button will also list all used multilines but with their respective layers.

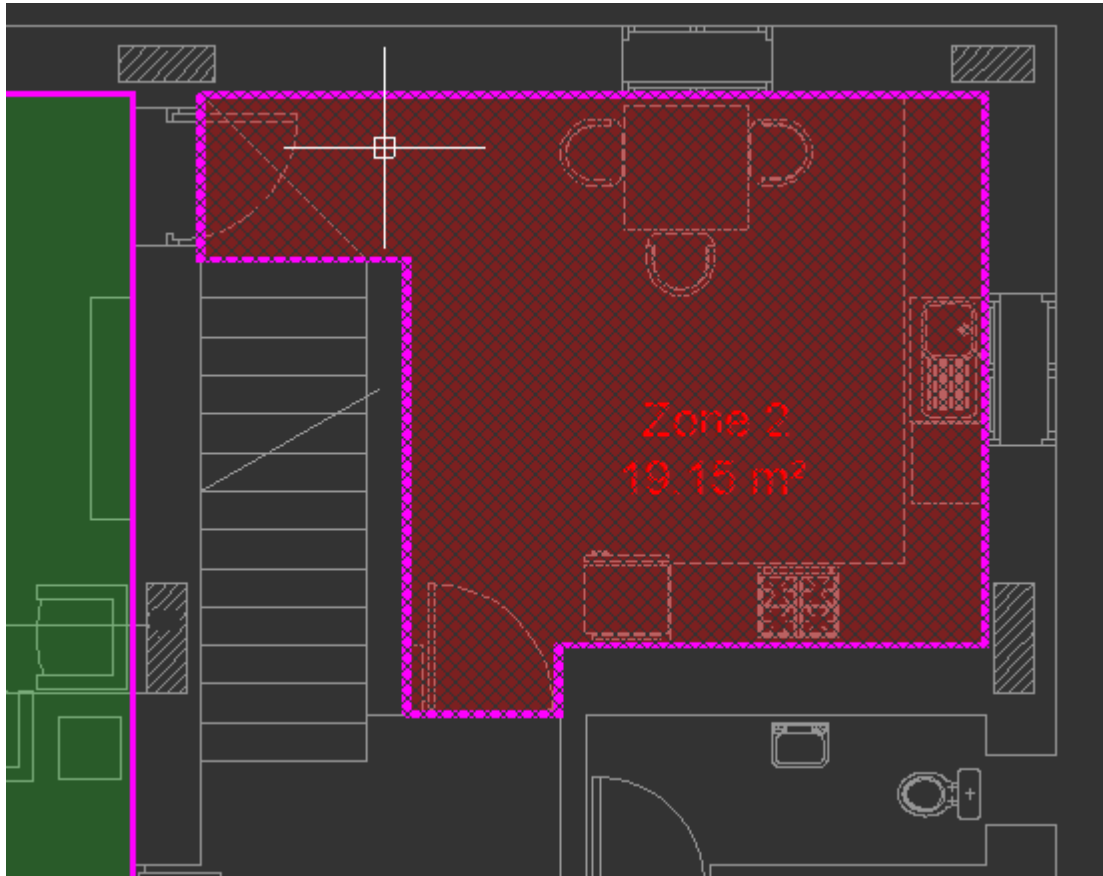


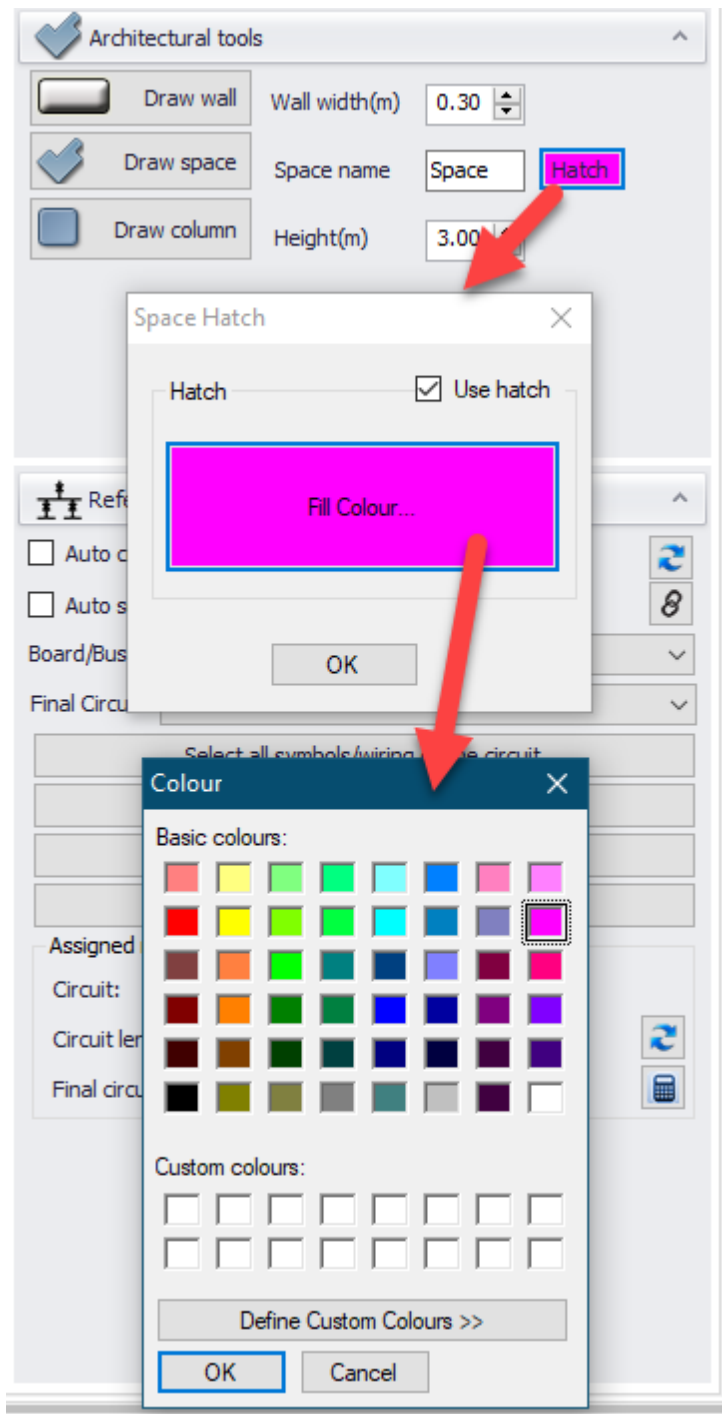
Architectural Tools



This section accommodates architectural components that may be added to the project.

- Draw wall: This option will basically draw a multiline. Spacing between two lines can be set using the Wall width(m) field. Usage is the same as drawing a line.
- Draw Space: This option creates a hatched area and assigns it as a space/zone. The name of the hatched area can be set via the Space name field, where the hatch colour can be set by means of a button. Frame colour will match with the active I0ayer's colour.





- Draw column: This option will create a rectangular prism. Height can be set by Height(m) field. The usage is the same as drawing a rectangle.

Reference from ElectricalOM Circuits

Reference from circuits

Auto create circuit after insertions

Auto select from network tree

Board/Bus: DT1/RCD1-Ground Floor RCD 1

Final Circuit: [1.4.L1] S1-Groud Floor Sockets 1 (x6)

Select all symbols/wiring of the circuit

Assign symbols/wiring to the circuit

Dissociate selected symbols/wiring

Clear all assinged symbols/wiring

Assigned measurements

Circuit: **S1-Groud Floor Sockets 1**

Circuit length: **38.01**

Final circuits points: **6**

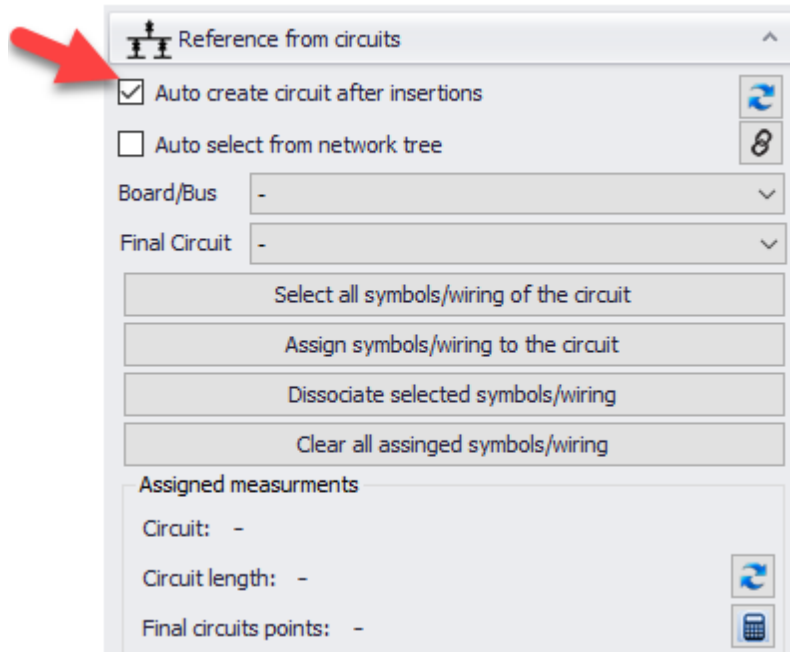
This section is the most important section as it enables the user to link ElectricalOM with CAD Plan Design module. User may like to:

- create circuits directly from CAD Plan Design module which will be added to ElectricalOM automatically, or
- insert symbols first, and then assign them to a circuit which already exists within ElectricalOM.

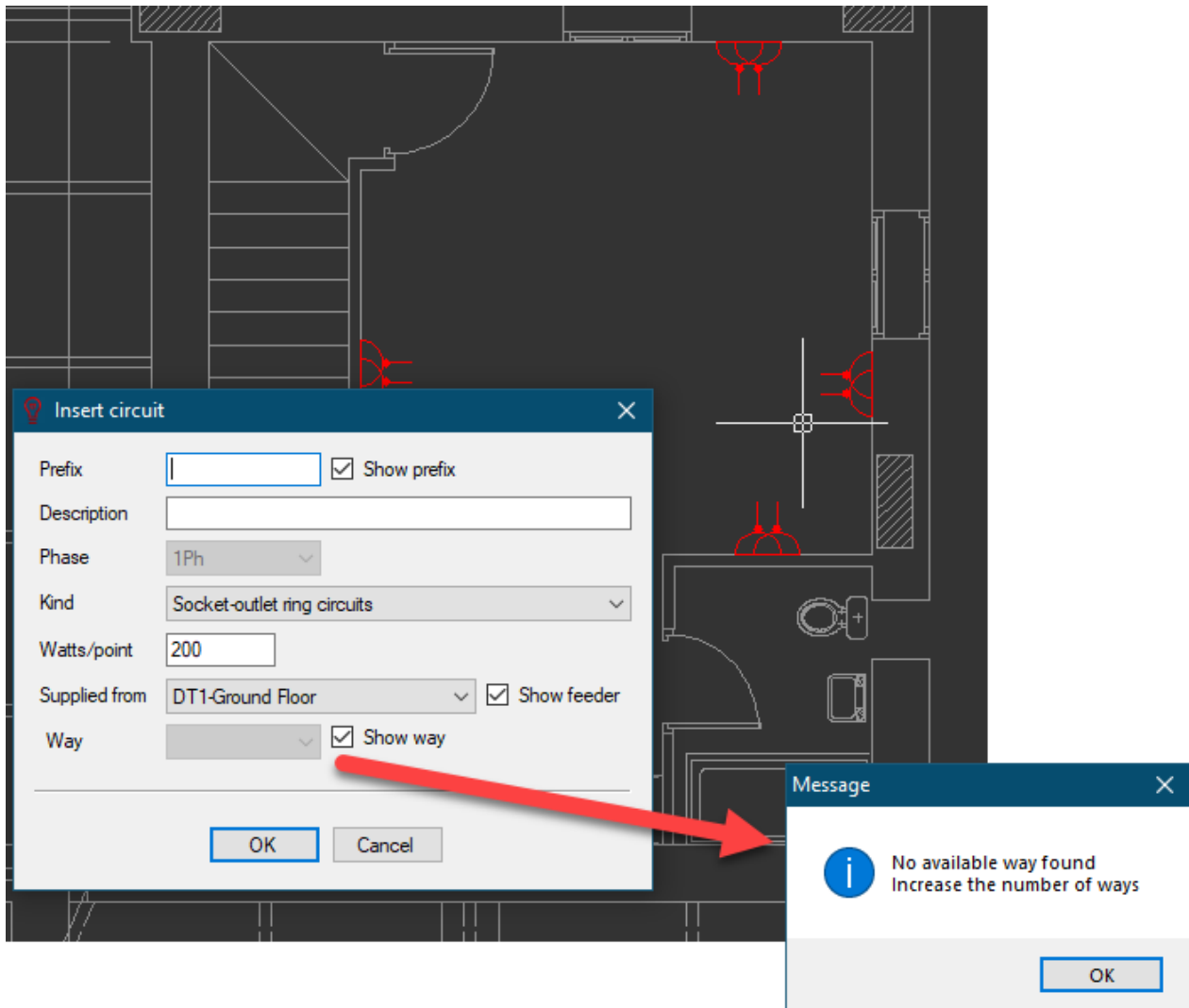
Once symbols are assigned to an ElectricalOM circuit, data related with the circuit will be displayed within Assigned measurements section.

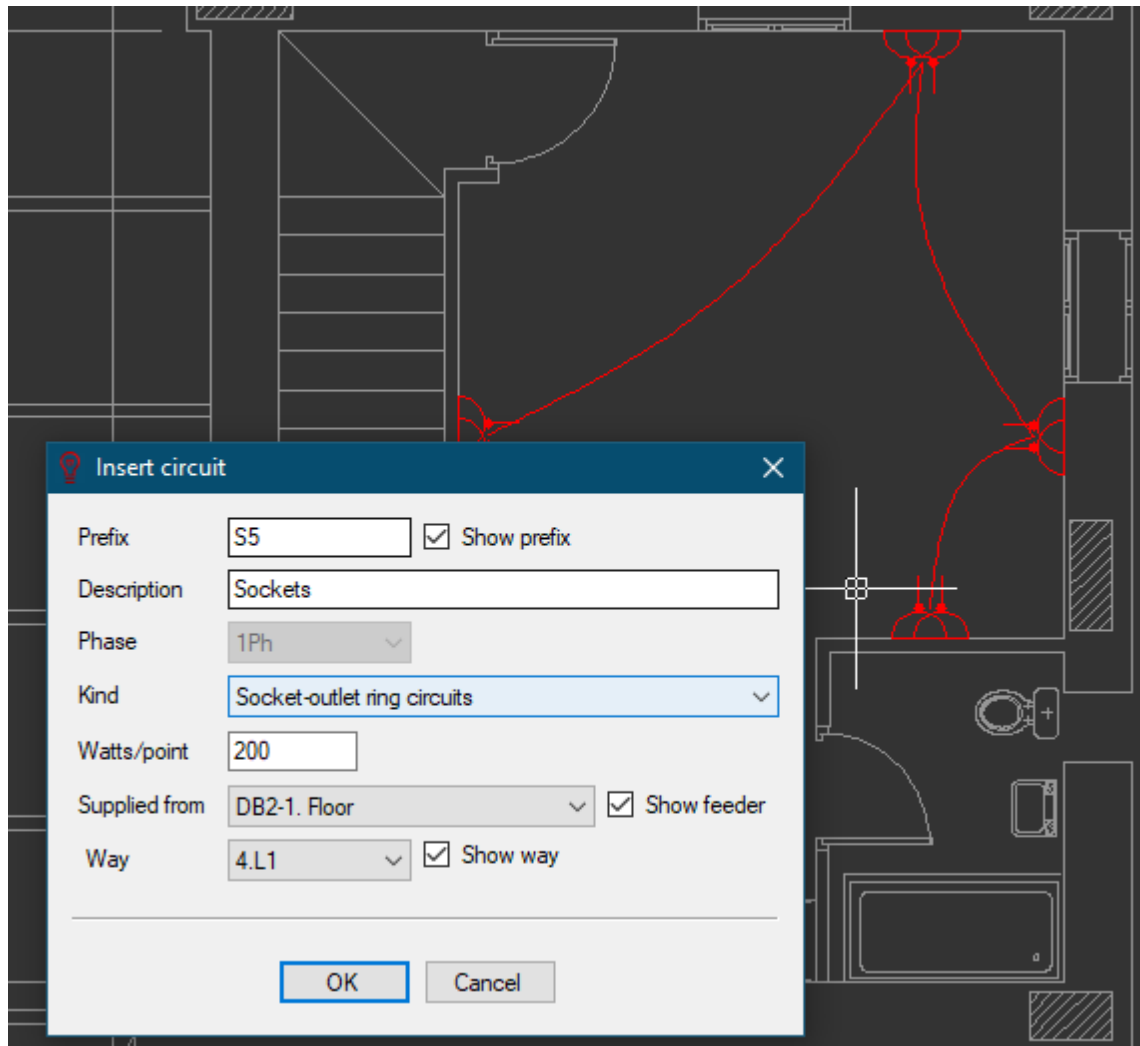
Create Circuits via CAD Plan Design

Circuits can be created via CAD Plan Design module, however, this is not as flexible as creating circuits via ElectricalOM core software. In order to create circuits, Auto create circuit after insertions box must be ticked.

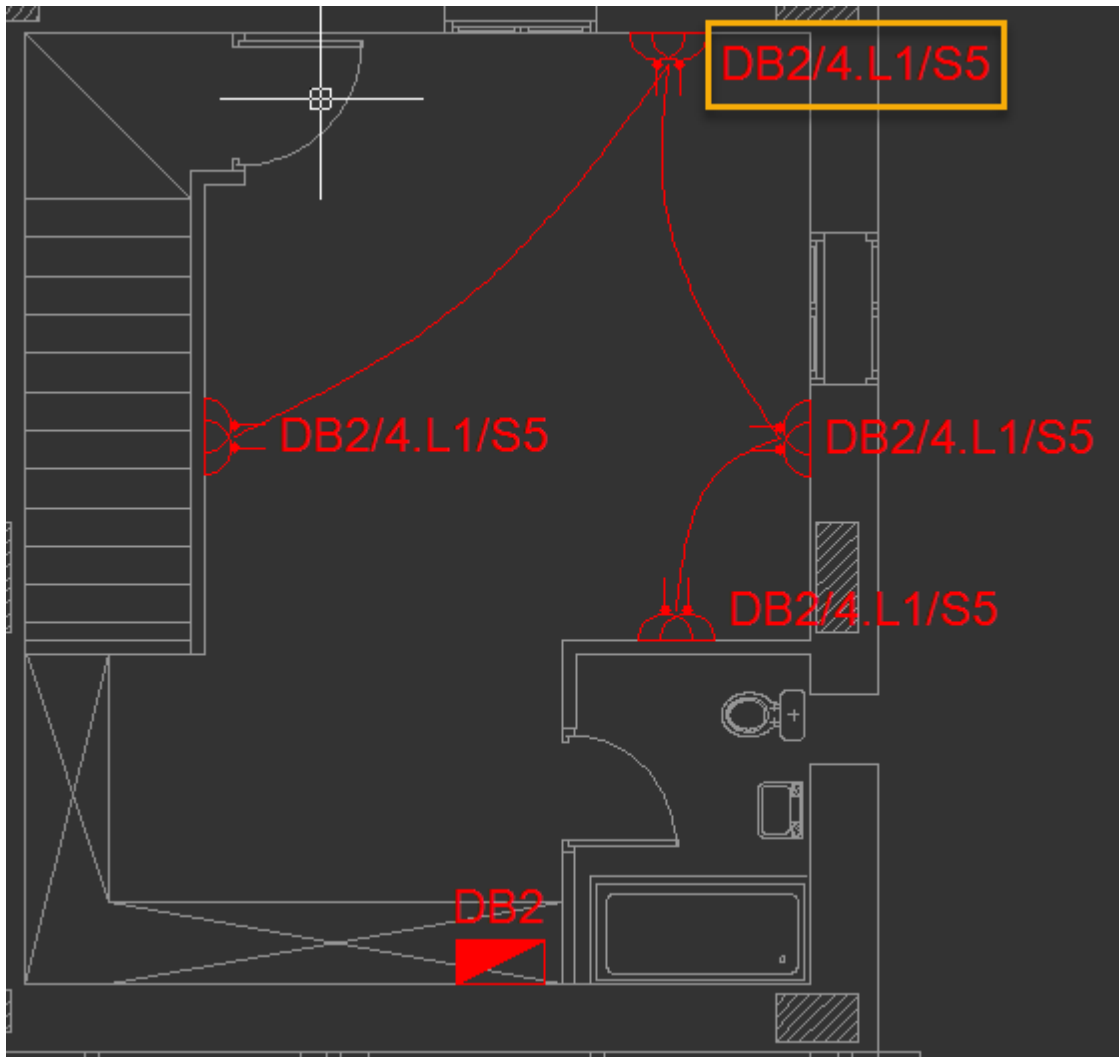


Once the symbol(s) are placed and right-click performed to finalise placement, ElectricalOM will display Insert circuit window. Using this window, user may define the circuit's Prefix, Description, Circuit kind, power per insertion point, and supply point. Way list will display available ways of the selected board, if there is no available way, ElectricalOM will display a warning and will not create the circuit.

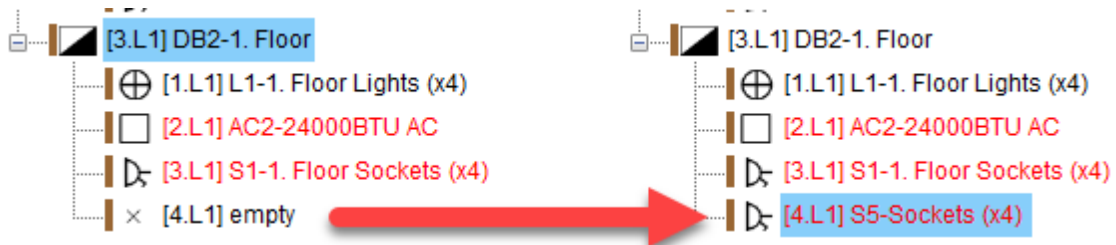


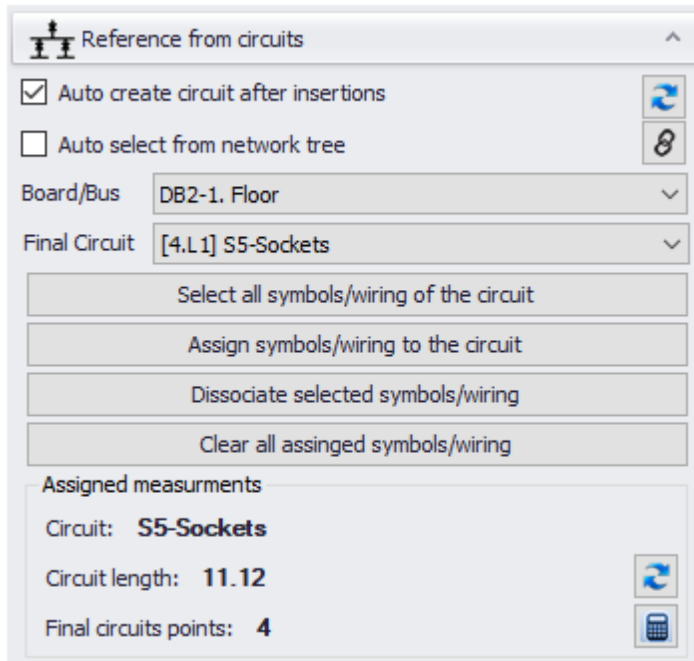


Using boxes next to the options, it is also possible to display or hide these parameters on CAD Plan Design drawing.



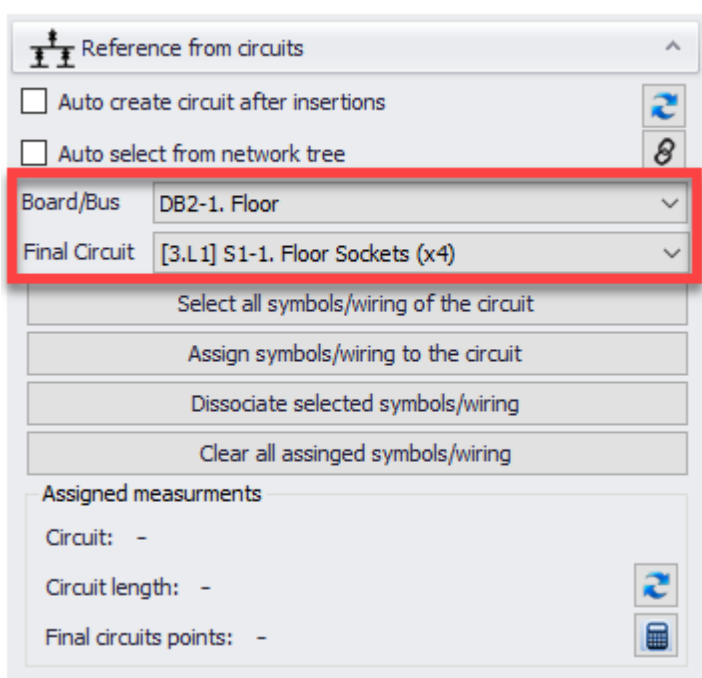
Once OK button is clicked, CAD Plan Design will create a circuit at ElectricalOM's network tree and assign the symbols to this newly created circuit also. However, user must revisit the circuit via ElectricalOM to revise circuit components like protection devices, cable cross-sections, etc.



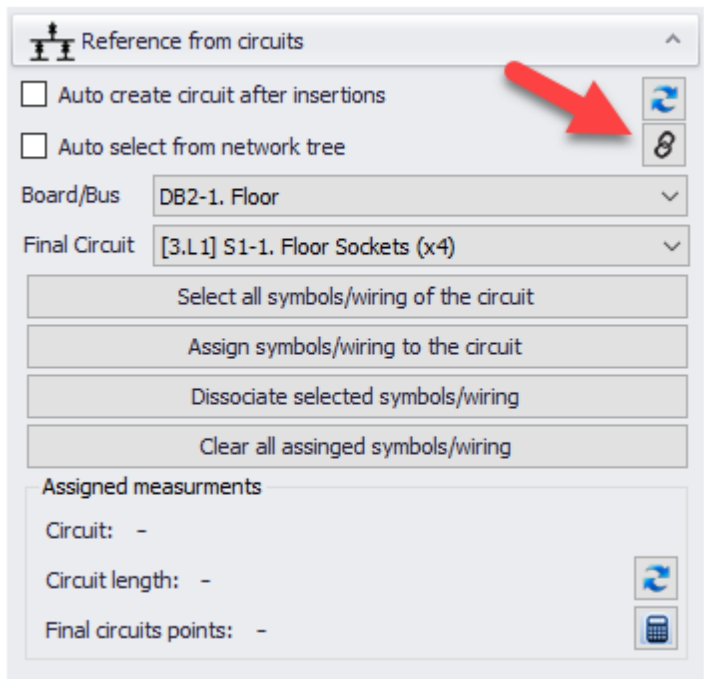


Assign CAD Plan Design symbols to ElectricalOM circuits

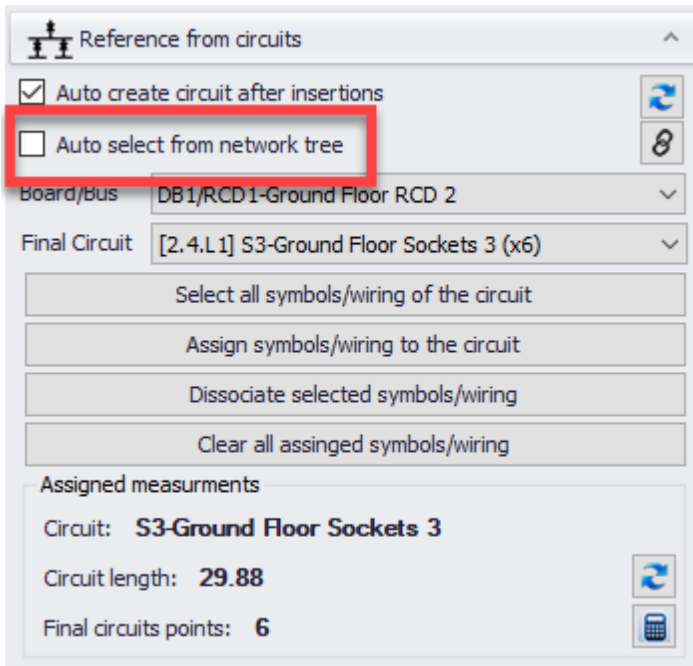
User may opt to create circuits using ElectricalOM, place symbols using CAD Plan Design, and then link them together. In this case, after placing and selecting the symbols which will be assigned, Board/Bus and Final Circuit drop-down lists can be used to select a circuit from ElectricalOM.



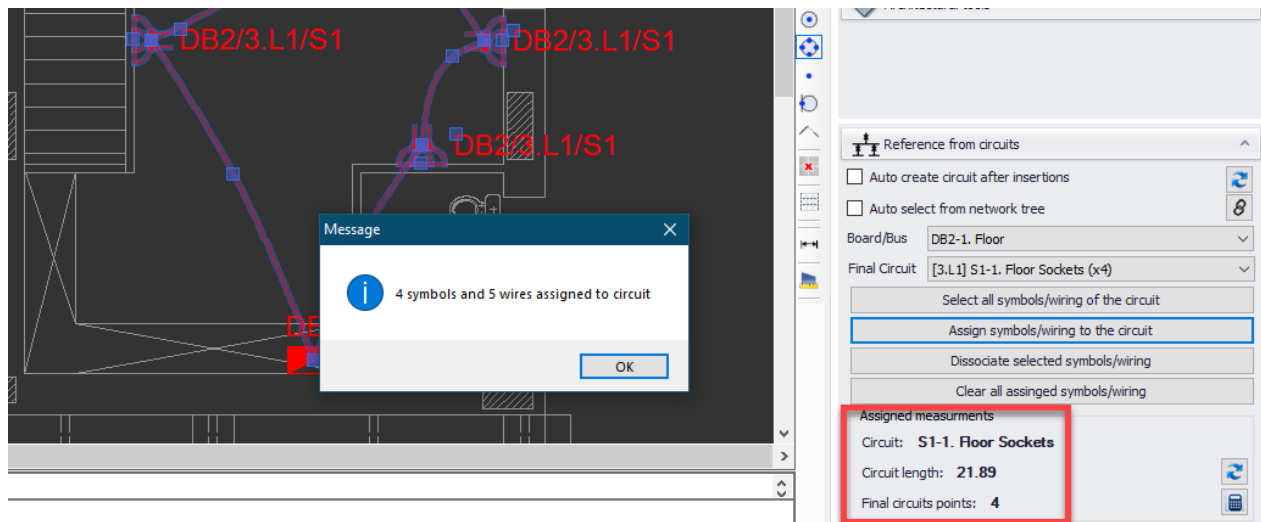
If the required circuit is already selected from the system tree, then Link circuit element from tree button will adjust the drop-down lists for the user.



If Auto select from network tree box is ticked, CAD Plan Design will automatically update the drop-down lists as the user selects a circuit from the network tree without requiring a click on Link button.

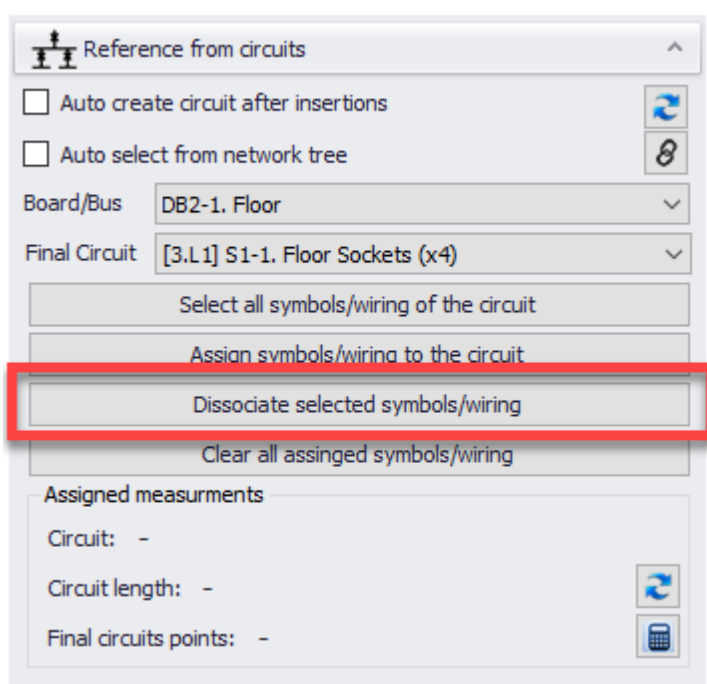


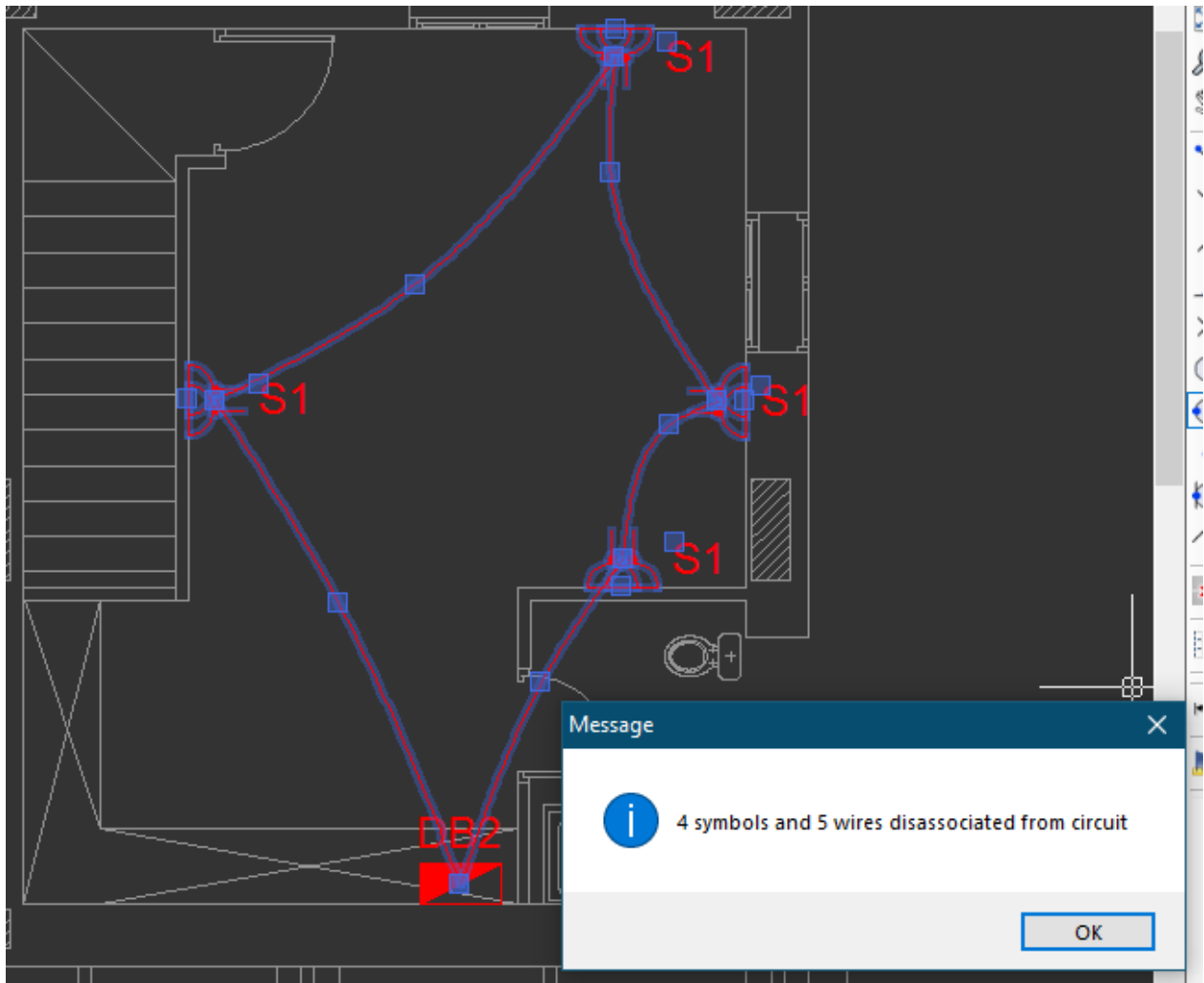
Once a circuit is selected, the last step to link ElectricalOM circuit and CAD Plan Design symbols is to click on Assign symbols/wiring to the circuit button. This will display the circuit prefix next to the symbols and will display related data at Assigned measurements section.



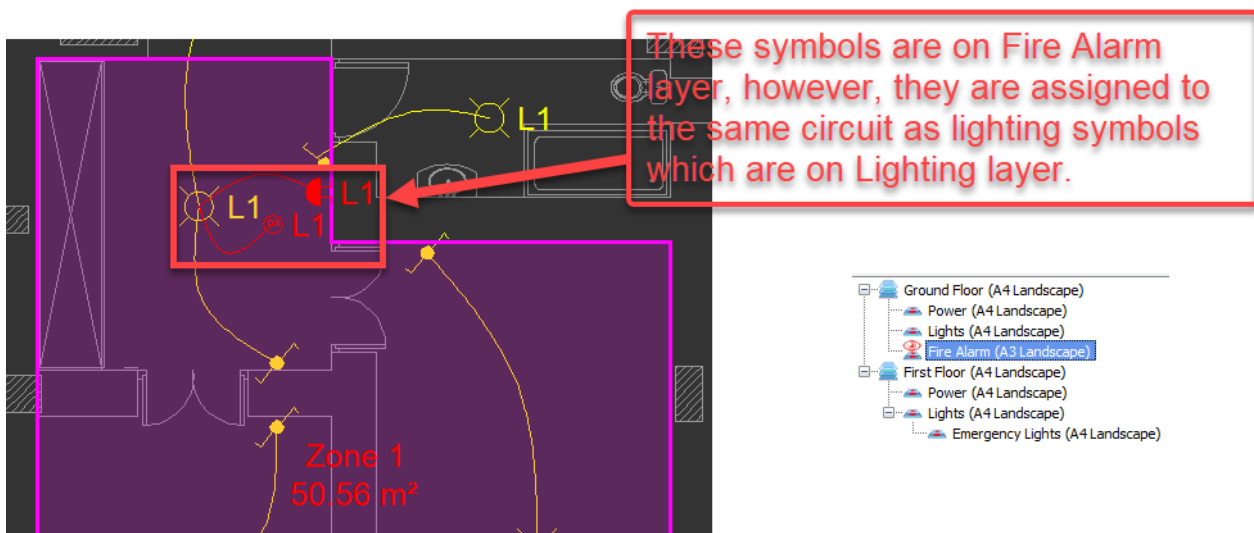
Dissociate symbols

If the user decides to remove link between symbols and circuit they are assigned to, then, aforementioned symbols must be selected first following a click on Dissociate selected symbols/wiring button.

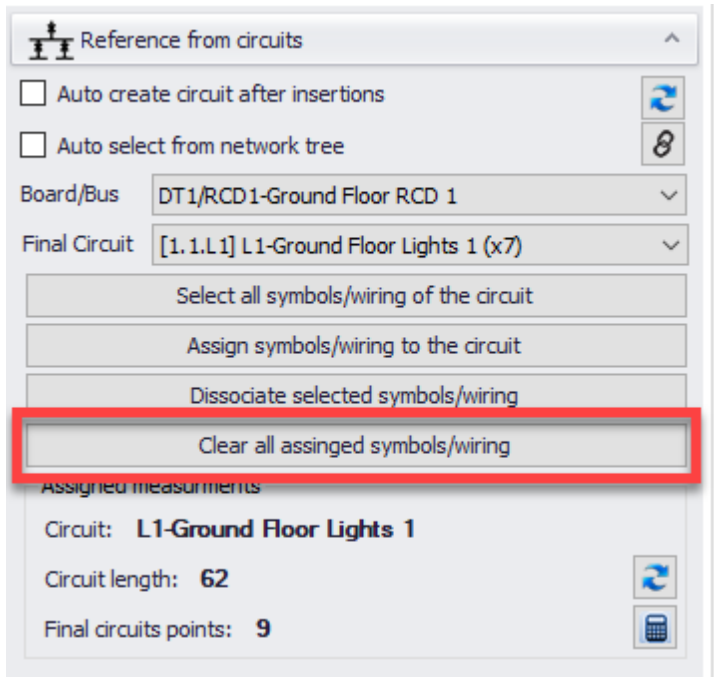




Above method works as long as all the symbols are placed on the same layer. However, if there are symbols placed on different layers which are belong to the same circuit than Clear all assigned symbols/wiring button must be used to dissociate all the symbols from a circuit.



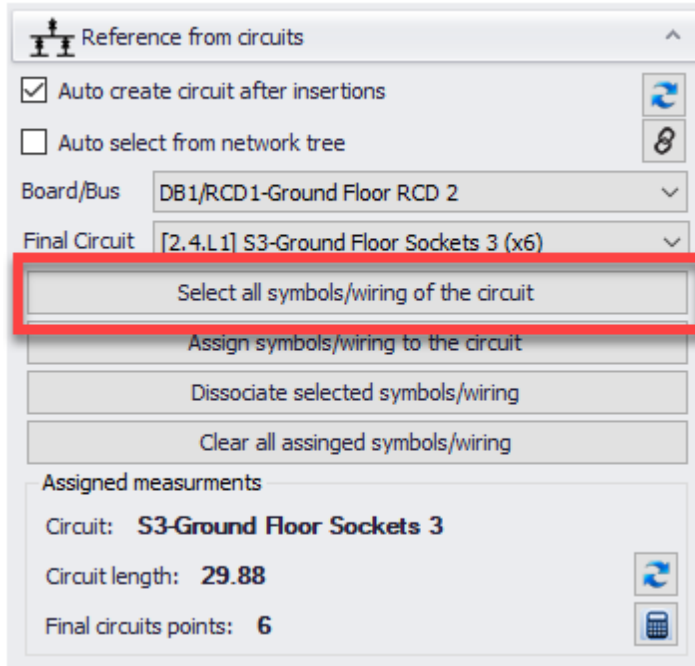
In order to dissociate all symbols from circuit L1, user must select a symbol which belongs to this circuit, no matter on which layer, and then use Clear all assigned symbols/wiring button.



It must be noted even if a circuit is dissociated, they will still retain their labels and these must be modified manually.

Select all symbols and wires of a circuit

It is possible to select a single entity of a circuit and let CAD Plan Design to select the rest of the circuit automatically. In order to do this, first, one or more symbols of the circuit must be selected. Then, Select all symbols/wiring of the circuit button can be used to select the rest of the entities.



Assigned Measurements

When an entity assigned to a circuit is elected, some information about this circuit will be displayed here.

- Circuit will display the assigned circuit prefix and description pulled from ElectricalOM. This field cannot be modified directly from CAD Plan Design module.
- Circuit length will display the estimated length of wiring for the selected circuit. This uses CAD Plan Design data like symbols elevation, cable installation elevation, for estimation.
- Final circuits points will display total installation quantity. This is not the same as quantity of the symbols used as a single symbol can be used to indicate multiple installation point.

For example, first picture shows a symbol set to 1 installation point where the second one show a symbol set to 2 installation points. For further details on how to set installation points, see [Properties Tab](#).

The screenshot shows the software interface with a wiring diagram on the left. The central toolbar contains various drawing and editing tools. The right-hand side features a 'Properties' panel for the selected symbol. Red arrows indicate the relationship between the diagram elements and the software settings.

Property	Value
Item codes	
Figure	
Hidden in sub pages	No
Scale	1
ScaleX	1
ScaleY	1
Symbol	Sockets\Double socket outlet
Symbol rotation	0
Type properties	GeneralSymbol
Zoom on figure	Zoom
General	
Accept connections	Yes
Circuit assigned	Yes
Circuit installation points	6
Circuit length(m)	34.47
Elevation(m)	0
Installation points	1
Layer	
Circuit prefix	S2
Height	0.25
Note	
Position	Right
Rotation	0
Show circuit feeder	No
Show circuit prefix	Yes
Show circuit way	No
Style	Feeder/Way/Prefix
Printing	
Print	Yes
Print symbol colour	No
Print symbol line weight	No
Symbol Colour	
Monochrome	Yes

This screenshot is similar to the one above but shows a different configuration. The wiring diagram and toolbar settings are consistent, but the 'Installation points' in the properties panel has been updated to 2, and the 'Final circuits points' in the toolbar has updated to 12.

Property	Value
Item codes	
Figure	
Hidden in sub pages	No
Scale	1
ScaleX	1
ScaleY	1
Symbol	Sockets\Double socket outlet
Symbol rotation	0
Type properties	GeneralSymbol
Zoom on figure	Zoom
General	
Accept connections	Yes
Circuit assigned	Yes
Circuit installation points	6
Circuit length(m)	34.47
Elevation(m)	0
Installation points	2
Layer	
Circuit prefix	S2
Height	0.25
Note	
Position	Right
Rotation	0
Show circuit feeder	No
Show circuit prefix	Yes
Show circuit way	No
Style	Feeder/Way/Prefix
Printing	
Print	Yes
Print symbol colour	No
Print symbol line weight	No
Symbol Colour	
Monochrome	Yes

Properties Tab

The screenshot shows the Properties Tab interface. At the top, there are tabs for 'Pages', 'Design', 'Properties', and 'Circuit groups'. Below this is a 'Symbol' dropdown menu. The main area contains a table of properties for a selected symbol. The properties are grouped into sections: Codes, Figure, General, Label, Printing, and Symbol Colour. The 'Accept connections' property is highlighted with a red box, and a red arrow points to it from the text 'Tips area'.

Codes	
Item codes	
Figure	
Hidden in sub pages	No
Scale	1
ScaleX	1
ScaleY	1
Symbol	Sockets\Double socket outlet
Symbol rotation	0
Type properties	GeneralSymbol
Zoom on figure	Zoom
General	
Accept connections	Yes
Circuit assigned	Yes
Circuit installation points	12
Circuit length(m)	34.47
Elevation(m)	0.4
Installation points	2
Label	
Circuit prefix	S2
Height	0.25
Note	
Position	Right
Rotation	0
Show circuit feeder	No
Show circuit prefix	Yes
Show circuit way	No
Style	Feeder/Way/Prefix
Printing	
Print	Yes
Print symbol colour	No
Print symbol line weight	No
Symbol Colour	
Monochrome	Yes

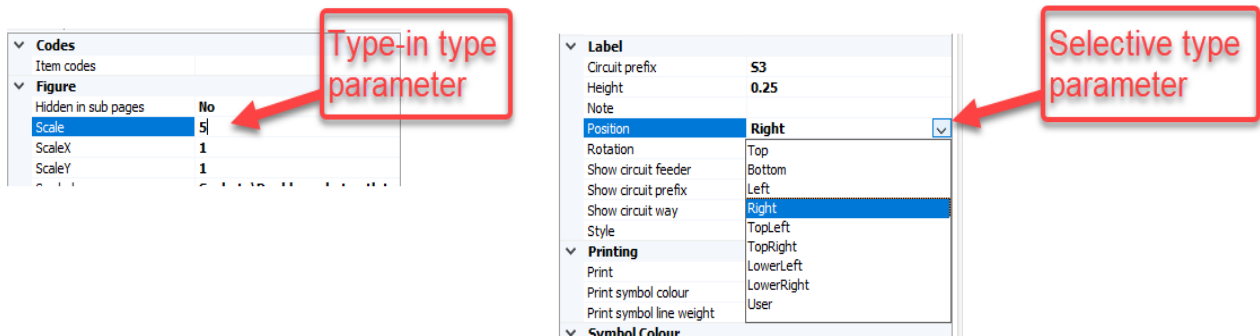
Accept connections
Choose if the symbol can accept wire connections

Properties Tab displays various properties of a selected item or selected items. This tab is dynamic, and contents will change depending on the selected items. If multiple items are selected, then only the properties common for the selected symbols will be displayed here.

Properties are grouped under titles in bold, and they can be collapsed or extended by the use of the arrows on the right hand side.

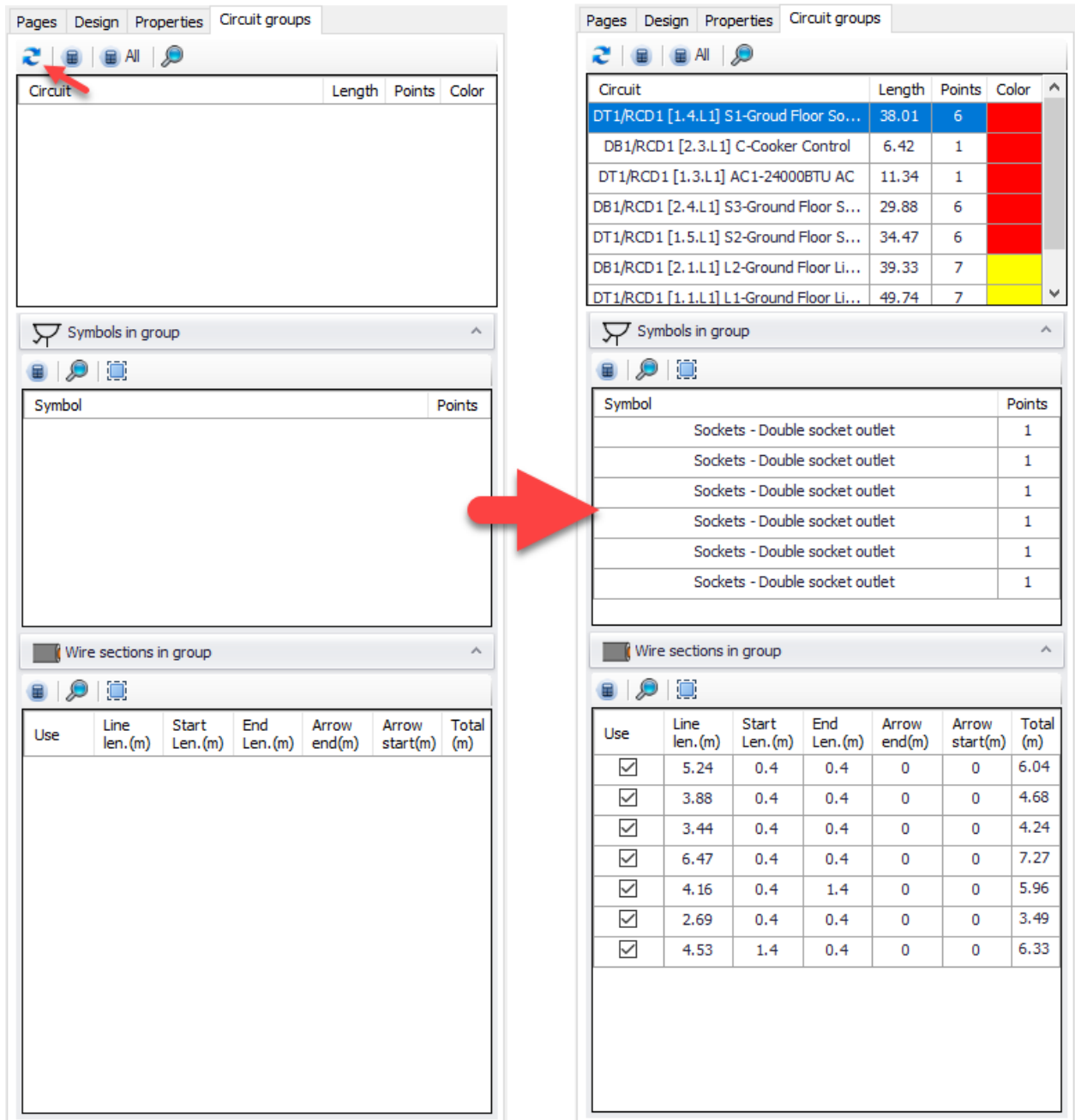
User may modify most of the parameters directly from Properties tab. Parameters which can be modified are indicated with bold fonts and can be modified by clicking on them. Once this is done, if the parameter is a type-in parameter then user may type in any value into the text area, if the parameter is a selective type then an arrow will appear on the left which can be clicked to see the options. Changes will be reflected as soon as they are done.

At the bottom, there is a tips section where a short description of each option will be displayed.



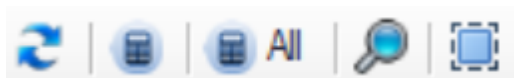
Circuit Groups Tab

Circuit Groups Tab is used to visualise the circuits on the plan drawing and make some changes. It may be empty at first but CAD Plan Design will populate the tab with related data once the user clicks on Refresh Groups button.



The tab is divided into three sections and there are some buttons at the toolbars area.

These are:

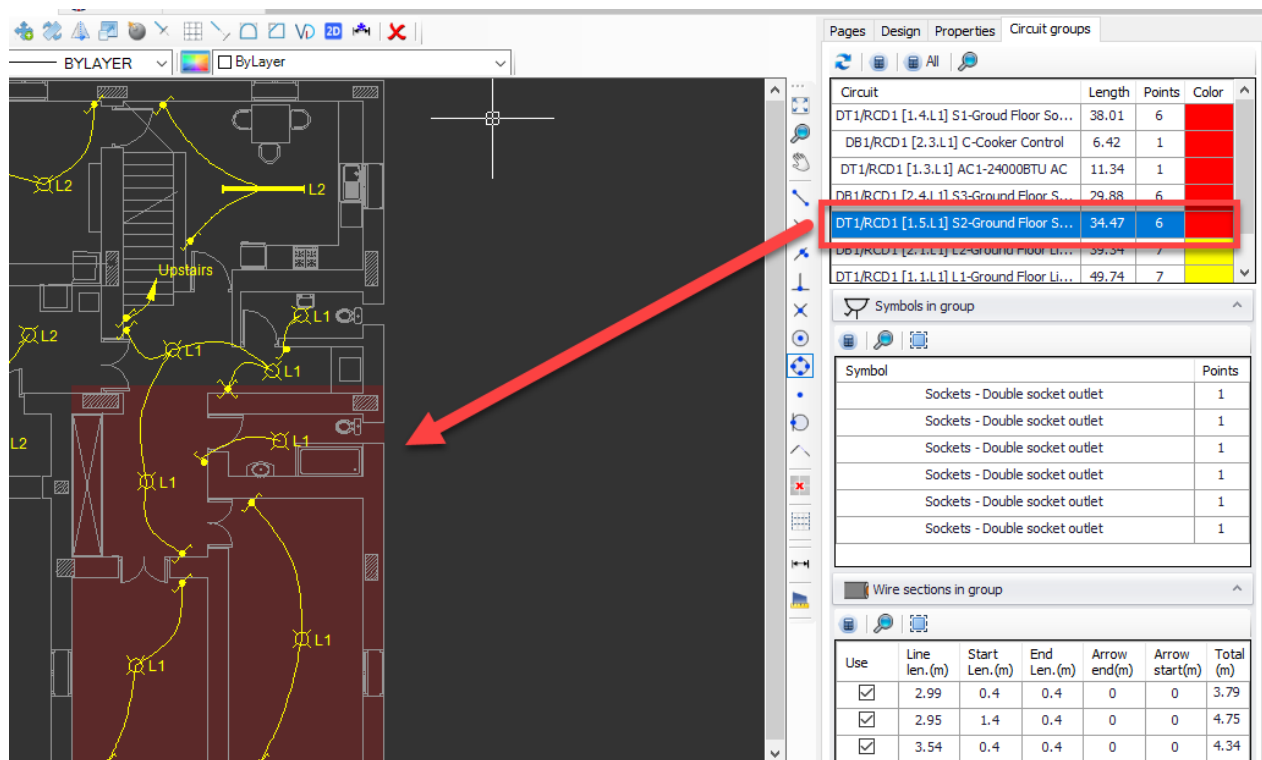


- Refresh groups button, see above.
- Apply measurement button applies any changes made within the related section only to the CAD Plan Design data, and updates ElectricalOM accordingly.
- Apply to all button applies all the changes regardless of section, and updates ElectricalOM accordingly.

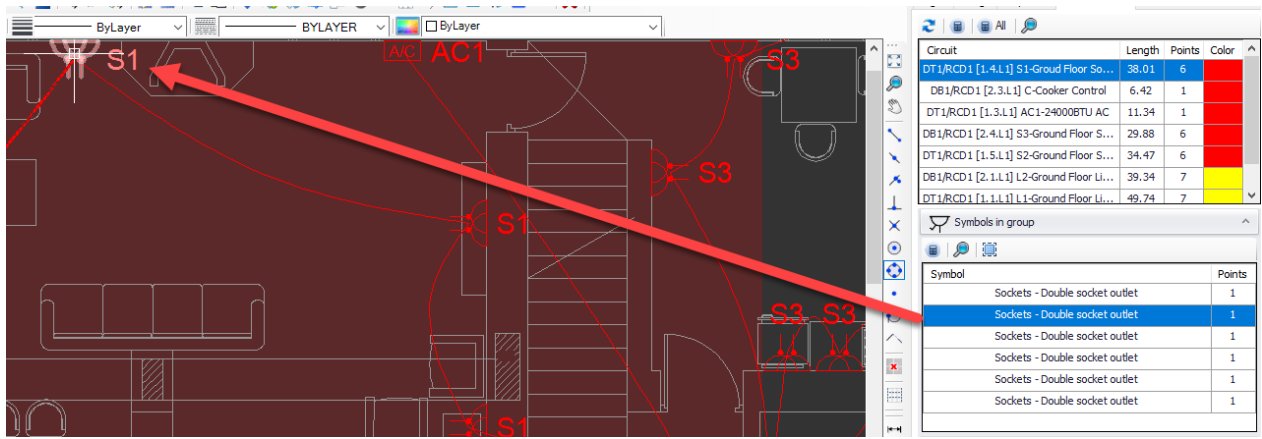
- Zoom to selected will zoom to the selected item at the drain area.
- Select all will select all items from the related list.

It is important to notice that without using Apply buttons any changes made will be limited with CAD Plan Design and nothing will be reflected to ElectricalOM schematics.

First section, located at the top, displays all the circuits which are assigned to symbols with their total wiring lengths, number of insertion points, and area colours. Area colours are used to show highlight the region of the circuit on the plan drawing. The colour is the same as the layer colour by default but it can be changed by clicking on it. When a circuit is selected, CAD Plan Design will highlight the circuit region on the plan drawing at the drawing area.



Middle section, Symbols in group, provides information about the symbols used with the selected circuit. If a symbol is selected, CAD Plan Design will highlight this symbol at the drawing area.



Last section, Wire section in group, provides wiring information for the selected circuit. The Starting Length, Ending Length, Arrow End Length, and Arrow Start Length fields can be clicked and modified by typing in any value, see [Properties tab](#) for details. Also, the use tick box can be used to include or exclude the related wiring from the length calculations, same as Exclude from measurements option at [Properties Tab](#).

Use	Line len. (m)	Start Len. (m)	End Len. (m)	Arrow end(m)	Arrow start(m)	Total (m)
<input checked="" type="checkbox"/>	5.4	1.4	0.4	0	0	7.2
<input checked="" type="checkbox"/>	1.67	0.4	0.4	0	0	2.47
<input checked="" type="checkbox"/>	3.15	0.4	0.4	0	0	3.95
<input checked="" type="checkbox"/>	2.94	1.4	0.4	0	0	4.74
<input checked="" type="checkbox"/>	2.02	1.4	0.4	0	0	3.82
<input checked="" type="checkbox"/>	1.12	1.4	0.4	0	0	2.92
<input checked="" type="checkbox"/>	2.98	0.4	1.4	0	0	4.78