Wiring Blocks Creation

AutoCAD Electrical 2018

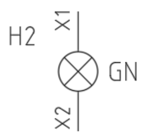
A wiring block is a symbolical representation of the actual physical device but contains wiring information that is repeated from the schematic diagrams.
There is no International standard for how wiring diagrams should be drawn so it is usually down to the specific companies' requirements as to how the image is aesthetically drawn and the information that is shown.

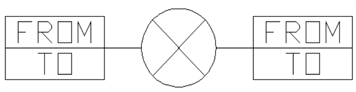
Usually the minimum information that is shown is as follows:

Wire no / from component / to component / wire type (or colour/gauge)

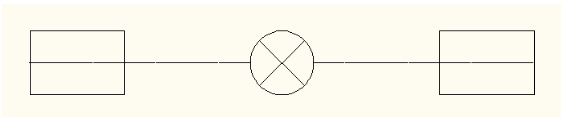
As an example, a lamp may be shown in the following representations:

SCHEMATIC **FOOTPRINT** **WIRING**



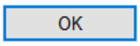


Draw the graphics for the wiring block of the component as shown below or similar:



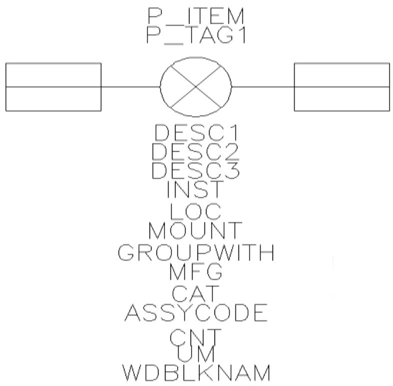
Select the command 

Select the graphics of the component just drawn and define the Insertion point of the symbol as the centre.

Define the type of symbol as a *Panel Footprint* and select 

You will then be taken into the *Block Editor*

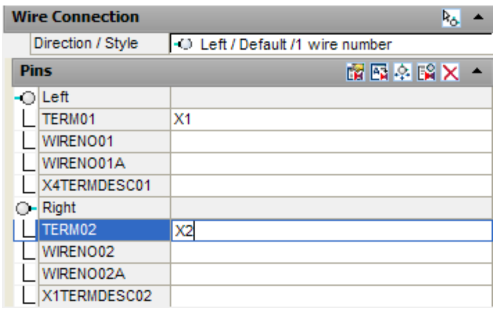
Highlight all attributes from the Symbol Builder Attribute Editor required List and insert so that your symbol looks like:



From the Wire Connection – Direction/Style dropdown, select *Others*.
Define the Terminal Style as *Terminal / 2 wire numbers* with a direction of *left*

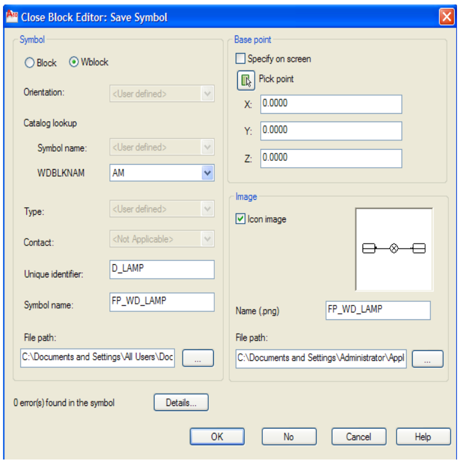
Repeat for the *right* connection

Define the terminal pins as X1 and X2 as shown



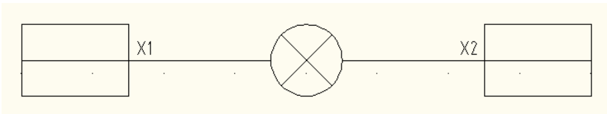
N.B. The pin numbers of the wiring diagram block <u>MUST</u> match the pin numbers of the schematic device (manufacturer's part) for wiring annotation to be placed where stipulated in the block creation.

Select *Close Block Editor* 

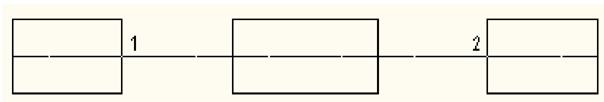


Cadline would recommend that a unique directory under the PANEL or PANEL_METRIC folder is created specifically for wiring blocks. The name of the footprint can be anything and is not governed by AutoCAD Electrical's naming convention for symbol naming. However, we would recommend that you prefix the symbol with "WD" so that you know the footprint is specifically for wiring diagram creation.

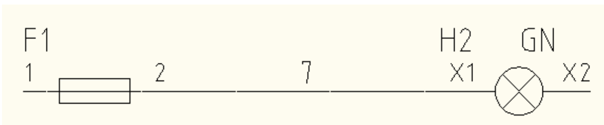
Select 



Repeat the creation of a NEW wiring block for a fuse, as an example, so that the fuse in inserted block form will look like the following:



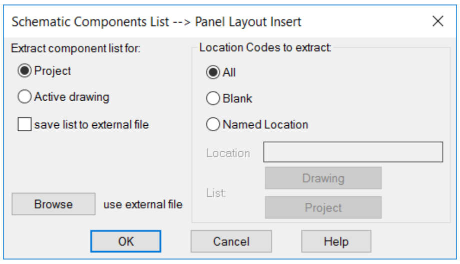
To test these wiring blocks out and how it annotates the schematic connections, you will need to draw a schematic similar to the following in a new drawing within a new project.



If you assign manufacturer and part number when placing the component, AutoCAD Electrical will remember any previously assigned wiring block to manufacturer's part associations.

From the *Panel* ribbon tab, select the command  *Schematic List*

The following dialogue will be displayed:



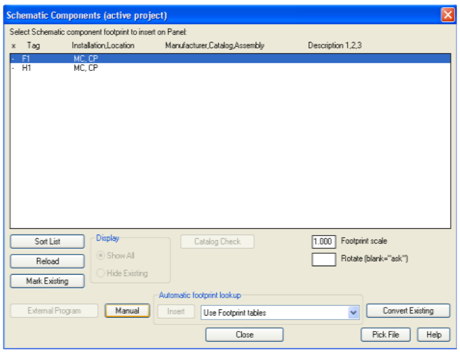
This allows you to extract only parts from a certain location in addition to filtering by drawing.

Select OK

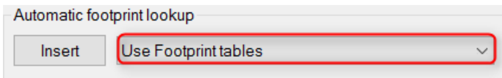
Select Do All, if the Project was selected, and then OK

The components drawn in the schematic will be shown in a dropdown list. If manufacturer, part, description has been entered this will also be shown in addition to the component TAG.

Highlight one of the components from the list as shown.

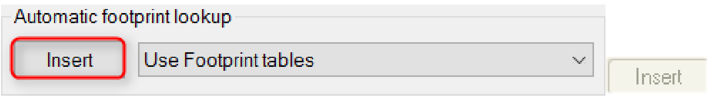


Select the dropdown to the side of *Use Footprint tables*



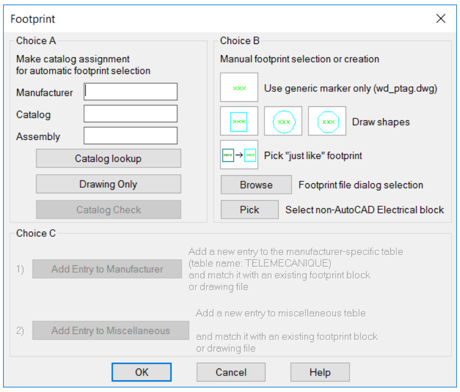
and change to *Use Wiring diagram* tables.

If a manufacturer and part has been assigned the Automatic footprint lookup > Insert option will be available



.

Select the option Manual and the following dialogue will appear:



If you did not assign a manufacturer and part, you are still able to in this dialogue. When you have assigned a part, the option Add Entry to Manufacturer will then become available meaning that you never have to associate the wiring block image with this part again.

Alternatively, you can select Browse to manually assign a wiring block to a schematic symbol.

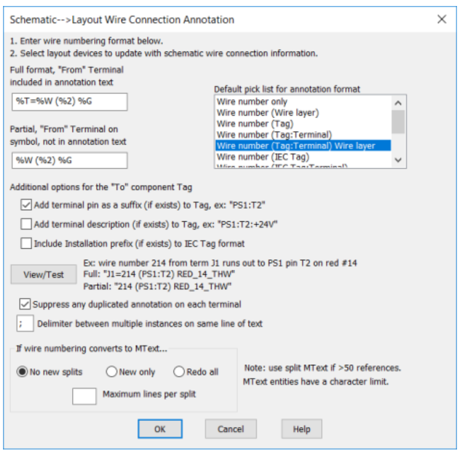
Select the correct wiring block for the fuse and pick the insertions point.
Select OK and repeat the process for the lamp.

To extract the wiring information from the schematics:

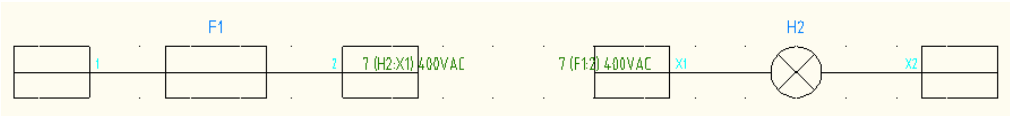
Select the option Wire Annotation

Select ⦿ *Project*, Active Drawing or select Pick and then OK

The following dialogue will be displayed:



Select the option as shown and then OK



The wire connection information will then be shown in the wiring blocks like above.

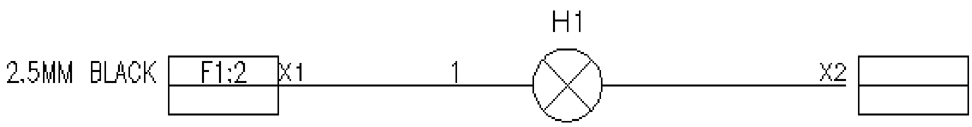
However, you can govern how information is shown by adding additional attributes to your wiring blocks. As an example, the wire from/wire type/wire number is shown as one continuous text in the above drawing.

Optional attribute names on wiring blocks/panel footprints to receive and display schematic wire connection annotation are as follows:

TERMnn	:	terminal number
TERMDESCnn	:	terminal pin description
WDEVnn	:	destination device tag
WLAYnn	:	connected wire's layer
WIRENOnn	:	wire number

Therefore by editing the original wiring blocks for both the fuse and the lamp and then adding extra attributes in this instance of WDEV01, WDEV01A, WDEV02, WDEV02A, WLAY01, WLAY01A, WLAY02, WLAY02A we can annotate the wire number, wire connecting component and colour/gauge separately.

The information that was previously on one text line will not be shown split as shown:



N.B. A Wire Annotation format of %W would be required for the above