

# How to Create Wiring Diagrams

All Electrical Designer Users

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 also 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



In Electrical Designer there are two libraries specifically for wiring diagrams in WD and WD\_IN. There contain a limited amount of symbols in the libraries as these types of symbols are generally bespoke to each client.

Select  *New Element* and fill in the details as follows

Create element <45>

Name of the element  
LAMP-TEST

Category: Auxiliaries

Dates:  
Creation: 12/09/2011  
Modification: 12/09/2011  
Verification: 12/09/2011

Description: WIRING DIAGRAM LAMP

Symbol  
Control  
Type  
Use

Remarks

Family: Auxiliary graphic

Draughtsman 1  
Draughtsman 2  
Draughtsman 3  
Draughtsman 4

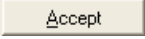
Phase ...  
Sizes ...  
Define connections ...  
Graph Termin Drawing

Definitions: Auxiliary graphic

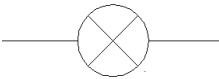
Prototype library  
Prototype

Change

Accept Cancel Help

Select 

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



Select  *Definitions Wizard*

Enter the values as shown and select 

Element measurements

Element measurements

Rectangular

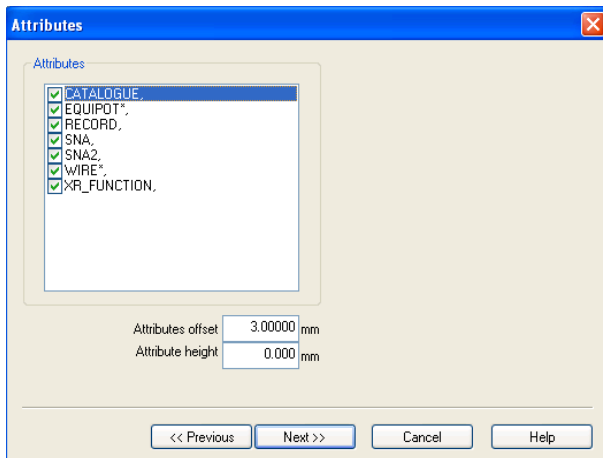
Height: 0.00  
Width: 0.00  
Depth: 0.00

Cylindrical

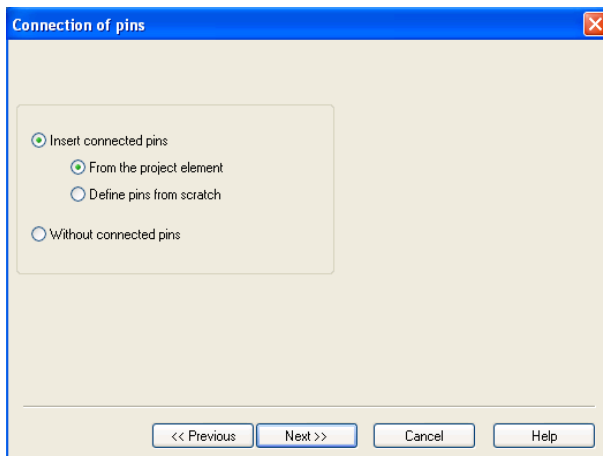
Diameter: 1.00  
Depth: 1.00

<< Previous Next >> Cancel Help

Select all attributes as shown



Select

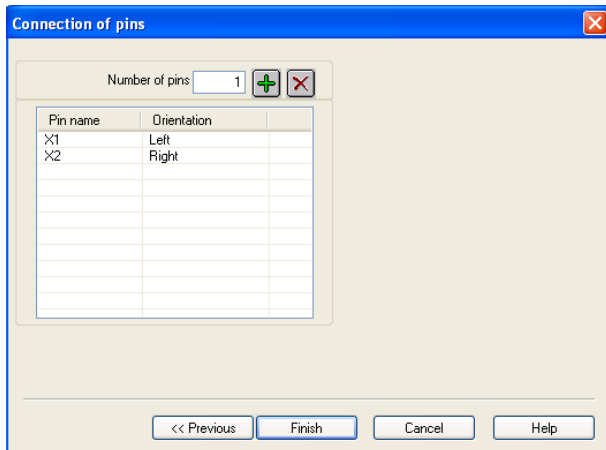


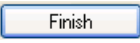
Electrical designer can automatically pick up the required number of connections from the Library Element or you can define the pins from scratch.

Select  *Define pins from scratch*

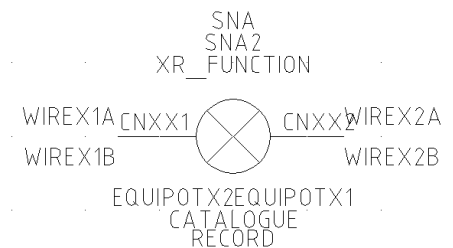
Select

Define the correct number of pins and their orientation



Select 

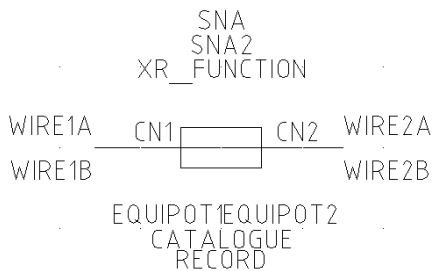
Define the Base point as being the centre of the lamp and all pin attributes will be placed. Electrical Designer will then prompt for each remaining attribute position. Position accordingly.



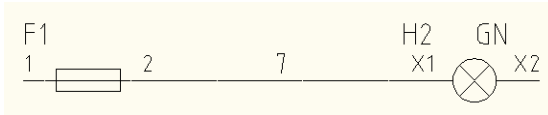
**N.B. The pin numbers of the wiring diagram block MUST match the pin numbers of the schematic device (manufacturer's part) for wiring annotation to be placed.**

Select  *End Drawing / Element*

Repeat the creation of a NEW wiring block for a fuse, as an example, so that the fuse will look similar to the following:

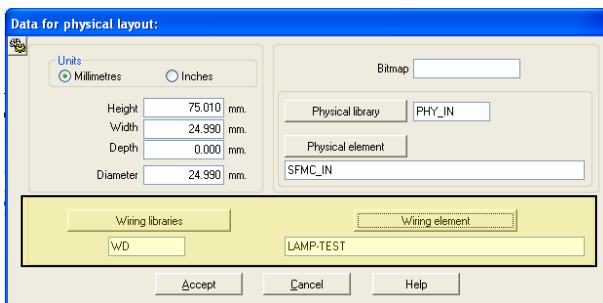


To test these wiring blocks out and how it annotates the schematic connections, we need to draw a schematic similar to the following in one of a new projects drawings:



Define CATALOGUE and RECORD information to both items. The parts you assign must be associated to our new wiring diagrams symbols in the record card > Dimensions

As an example you will note the wiring diagram information defined in the ACERISOFT part 0013A.



Ensure both items have parts that have the new WD symbols associated

The Connections and Cables editor allows the user to define the order of connections required. This aids the user in deciding how a 0V connection might loop round several connections. You have to enter into the Connections and Cables Editor to be able to produce wiring diagrams

From the Project Browser, select  *Edit Connections and Cables*


If your project only has Devices, select  *Devices*; if your project has Devices and Terminals, select  *Both*


Select

Select  *Connect Up*

If you wish to accept the connections and their order, select

Create a  *New Drawing* of type

Select  *Insert Component* from the Physical Layout toolbar

Highlight the component in the list and then select 

Repeat so that both components have been placed

The wire annotation will now appear in your drawings

