

Drawing Your Schematics from an Estimate

AutoCAD Electrical 2017

AutoCAD Electrical has the ability to draw the schematic symbols from XLS, CSV or MDB file information. The file only has to contain Manufacturer & Part No information although a description is useful as explained in the notes below.

In the example XLS sheet below, which you can of course duplicate, it contains 3 columns of MFG, CAT and DESC along with 3 rows of information.

	A	B	C
1	MFG	CAT	DESC
2	TELEMECANIQUE	LC1D09106F7	CONTACTOR 3P 1NO 110VAC
3	TELEMECANIQUE	XB4BV33	GREEN LIGHT 110VAC
4	TELEMECANIQUE	XB4VA21	BLACK PUSHBUTTON 1NO

The order of columns does not have to be the same nor do you have to have a column header (marked in yellow).

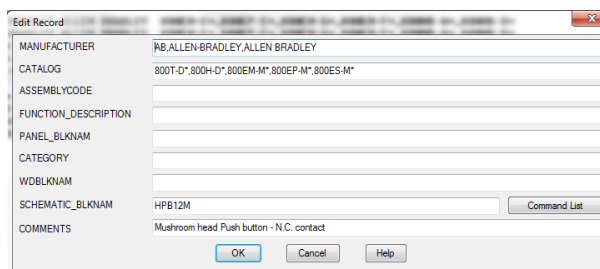
The data could come directly out of software packages such as Electrika or Estimation.

How does AutoCAD Electrical know which schematic symbols to choose from?

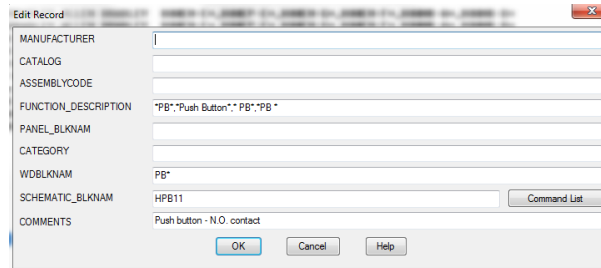
Edit the Schematic Database File Editor

This option is available from *Schematic* ribbon tab > *Other Tools* panel > *Other Tools* ▼ or you can use Microsoft Access to edit.


Wildcards (*) can be used so that if the manufacturer name in the spreadsheet is a variant of the same name that it will still select the correct symbol e.g. AB, ALLEN*, ROCKWELL*



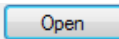
Similarly, you can also define a search on the description e.g. *PUSHBUTTON*, *PUSH BUTTON*. Even if the part defined in the XLS spreadsheet is not in AutoCAD Electrical parts database, it will still offer the correct symbol based on the description filter.

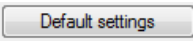


Inserting Symbols from the Spreadsheet

From the *Schematic* ribbon tab > *Icon Menu* ▼ select  Equipment List

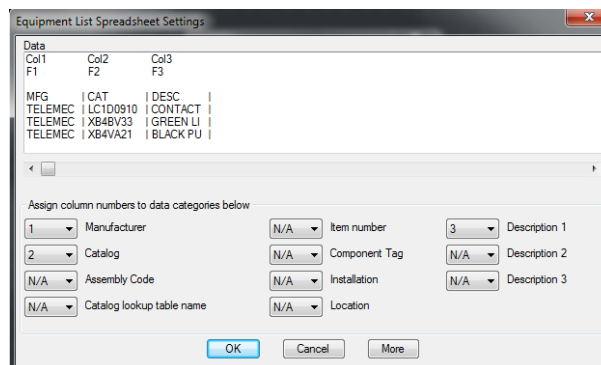
Select Files of type *.XLS

Browse to the directory of the estimate and select 

Select 

Select 

Select the mapping options as shown



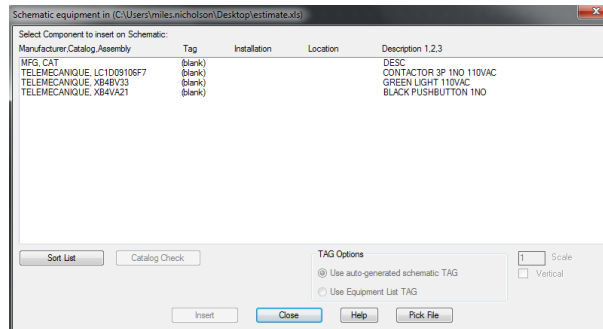
In this case Column 1 = Manufacturer, Column 2 = Catalogue, Column 3 = Description 1

Select 

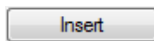
Select  to save the settings so that you don't have to do this again

Select 

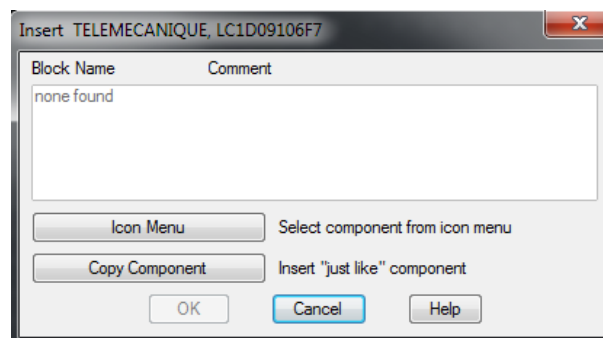
Each part will then be shown in the list



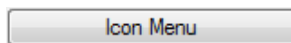
Highlight a component in the list and select



If the component **has not been defined**, then the following option will be shown

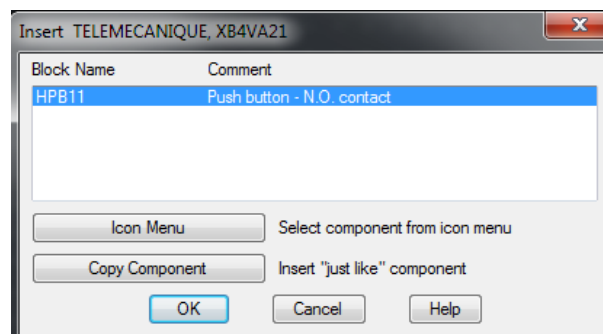


In which case, select



and choose the appropriate symbol

If the component **has been defined**, then the following option will be shown



Select

