


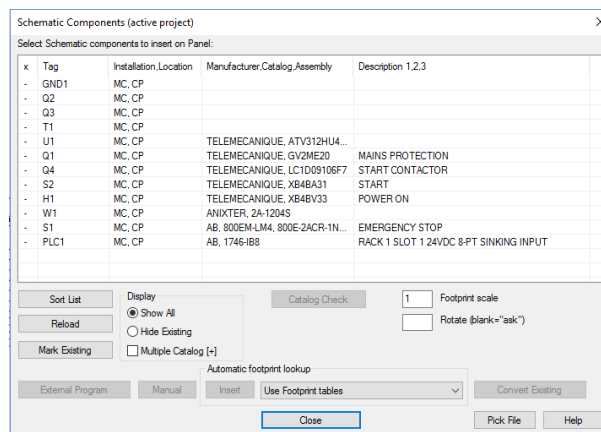
Associating a Manufacturer's Footprint

AutoCAD Electrical 2017

AutoCAD Electrical has a parts catalogue of approximately 800,000 parts (including the Cadline Localiser). Some schematic components may not carry manufacturer/catalogue information or have a part number assigned that is not listed in the footprint lookup file. In such a case, AutoCAD Electrical cannot determine which footprint block to use and will automatically prompt the user to either change catalogue assignment, select or create a footprint, or create a lookup entry on the fly.

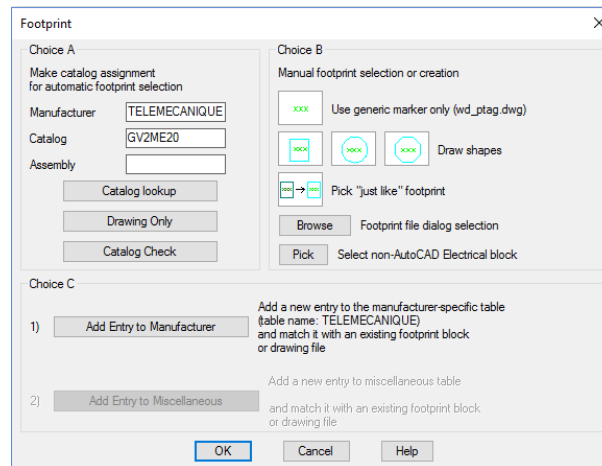
When creating a panel assembly, an example of this is when using the command  *Insert from Schematic List* (Panel ribbon > Insert Component Footprints).

This will show a list of items defined in the schematic.



When placing each component, AutoCAD Electrical will find the correct footprint details from the footprint lookup file.

If it is unable to locate the correct footprint, the following dialogue will be shown:



Choice A allows the user to redefine the manufacturer's part and to check that the part exists in the parts database.

Choice B allows the manual and one time only selection of either:

- a) The dimensions of the component
- b) The footprint file and directory location
- c) Selecting an existing "dumb" AutoCAD block (useful when converting legacy AutoCAD drawings to be intelligent)

Choice C allows the user to add an entry to the footprint lookup file so that AutoCAD Electrical remembers either

- a) The dimensions of the component
- b) The footprint file and directory location

Once a part is associated whether a one-off or permanent selection, a footprint can be placed.

Tip: If you do not want to draw complex footprints or a footprint is not available from the manufacturer, then select Geometry. Height and Width sizing information can then be simply entered as text values.