

Editing the Footprint Database

AutoCAD Electrical

When you add new parts into the catalogue database, you should also edit the footprint database to associate either dimensions or a physical drawing to the new part(s). In addition to existing parts, you may want to change the footprint to a more detailed form rather than the generic 2D image. The quickest and easiest way is to wait until you need to place the new part as a footprint for the first time within an assembly drawing.

[Associating a Footprint](#)

However, you may wish to add several component associations with their counterpart footprints at the same time. This can be done within the *Footprint Database Editor*.

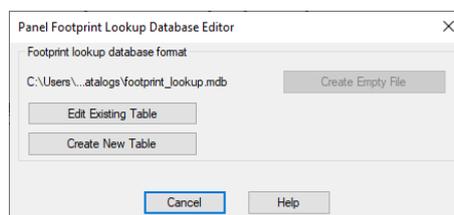
The DWG symbol will either need to be created or supplied by the manufacturer and it can either be a “dumb” DWG, or a DWG with associated intelligence (AutoCAD® Electrical will add electrical intelligence on the fly when a footprint is inserted).

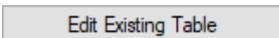
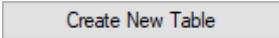
[Importing a DWG Footprint & Assigning Intelligence](#)

Manufacturers footprints can be downloaded from a variety of 3rd party websites such as [Traceparts](#), [Cadenas Part Solutions](#) or even the manufacturers web sites themselves.

Once the catalogue data has been added into the *default_cat.mdb*, we can then associate the footprint within the *footprint_lookup.mdb* or wait until the panel assembly insertion.

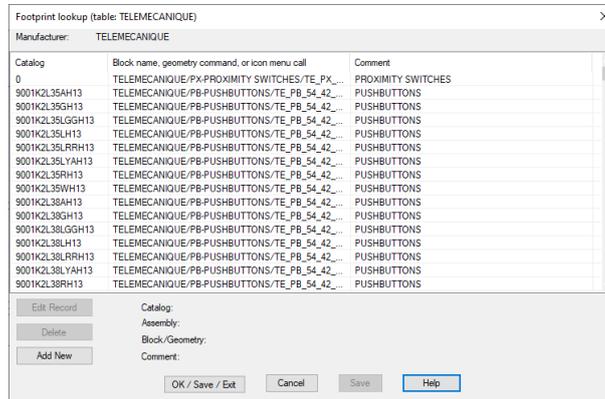
Select the *Panel* ribbon tab > *Other Tools* panel  >  *Footprint Database File Editor*



Select  if the manufacturer already exists
 Select  if the manufacturer is new

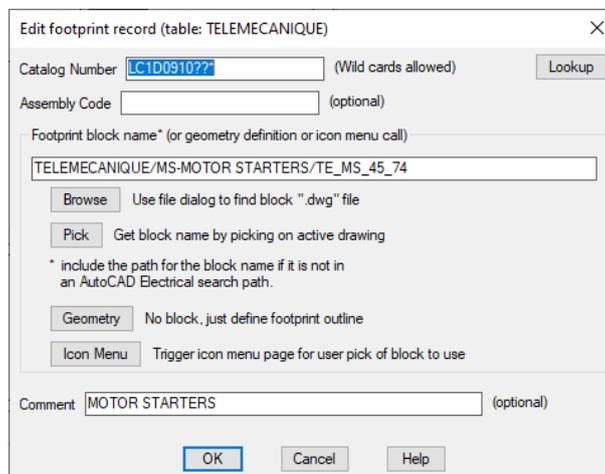
In this example, we will  and edit an existing part to alter its dimension information to an actual 2D/3D footprint DWG.

Select **TELEMECANIQUE** and **OK**



Locate the footprint entry of **LC1D090#??***

Select **Edit Record**



Wild cards are allowed in the search meaning that you can have *one* footprint catalogue number associated with *several* parts in the *default_cat.mdb*.

- * = match any characters
- ? = match any single character
- # = match any single numeric digit
- @ = match any single alphabetic character

When you create a footprint or download a footprint, you should save the AutoCAD DWG within the PANEL_MM library preferably within the same style of structure that already exists:

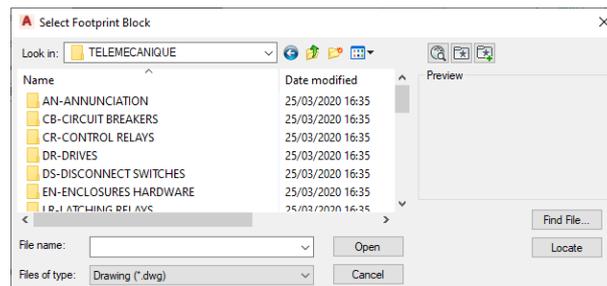
...\PANEL_MM\<<MANUFACTURER>>\C<,COMPONENT CATEGORY>>\

If you decide to choose an alternative location for the footprint, then you will need to define the additional new path within the {Project name} > *Properties* > *Project Settings* > *Library and Icon Menu Paths*.

As an example, if you have saved the footprint in the location of:

...\PANEL_MM\TELEMECANIQUE\MS-MOTOR STARTERS

Then select



Highlight the correct DWG and select

Select

If the *PANEL_MM* directory path has been defined in the project properties, you only need the sub-directory path of TELEMECANIQUE\MS-MOTOR STARTERS\lc1d09f7.dwg defining rather than the full path.

Select

Select to accept the changes

With this methodology, the correct footprint will always be inserted for the part. If the part is changed to another manufacturer and/or catalogue, then AutoCAD® Electrical will prompt you to change the footprint.