

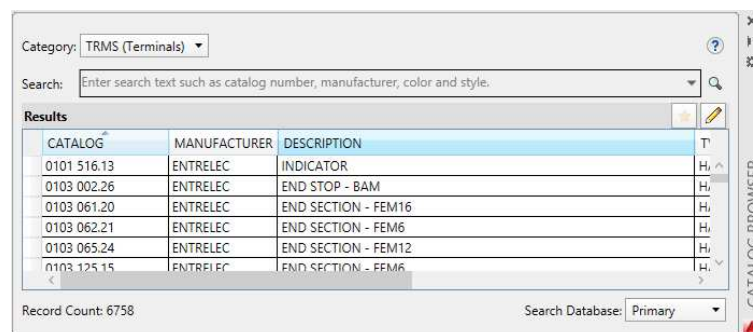
Adding a Terminal into the Parts Database & Associating a Footprint

AutoCAD Electrical 2016

Adding a part into the parts database is easy but in order for the terminal editor to draw the terminal correctly, the manufacturer's footprint must be associated with the correct part information. This month's Tips & Tricks goes through the whole process.

Adding a New Manufacturer Part

You can access the catalogue browser in the following ways:
Through the side panel (AECATALOGOPEN)




Through  Edit Component > 

Through the terminal editor  Editor >  Assign Catalog Number

Whilst AutoCAD Electrical has a very comprehensive parts catalogue, there will always be the requirement to add some parts yourself.

Ensure the Category is defined as  Category: TRMS (Terminals)

Select  to *edit* the parts database

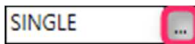
You can *highlight* an existing Row and *right click*. This will then allow you to *paste* a similar component with some minor changes.

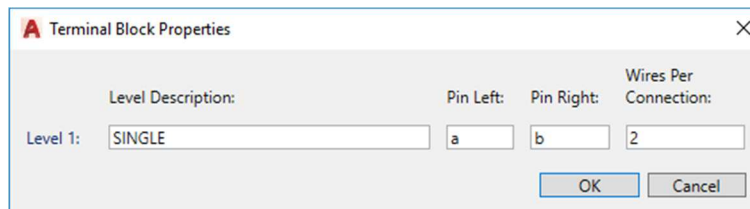
Create a new part with the following details

Catalog:	58.503.0055.6
Manufacturer:	WIELAND
Description:	Feed-through terminal WT 2,5 BL
Type:	FEED-THROUGH
Rating:	20AMPS
Miscellaneous1:	DIN rail terminal block, screw connection, TS 35 mounting
Miscellaneous2:	Nominal cross section 2,5 mm ² ; width 5 mm; colour blue
Levels:	1
LevelDescription:	SINGLE
WirePerConnection:	2

The LEVELS field relates to whether the terminal is a multi-deck terminal and is the number pf physical levels.

The LEVELDESCRIPTION typically is SINGLE for a single deck terminal. For a two-deck terminal this could be TOP, BOTTOM or UPPER, LOWER as an example.

When you enter a level description, select  and you can enter the terminal properties:



The dialog box titled "Terminal Block Properties" contains the following fields:

Level Description:	Pin Left:	Pin Right:	Wires Per Connection:
Level 1: <input type="text" value="SINGLE"/>	<input type="text" value="a"/>	<input type="text" value="b"/>	<input type="text" value="2"/>

Buttons:

Select

Select to accept changes to the parts database

Creating a Footprint for the New Part

The easiest way to create a new footprint is to copy an existing terminal footprint and stretch / manipulate the graphics to suit the new part.

If you download a footprint from the manufacturers website or via the [Cadline Community](#) then intelligence needs to be added.

If you are manipulating a copy of the ones out of the box, then there are some generic terminals in the root of the PANEL_MM library directory but make a copy and save as the manufacturer's part name.

wd_default_terminal.dwg	SINGLE DECK
wd_default_02terminal.dwg	DOUBLE DECK
wd_default_03terminal.dwg	TRIPLE DECK

You can also use these to copy the required intelligent attributes into your downloaded footprint or of course use the Symbol Builder to assign the required attributes.

Bear in mind that some manufacturers have too much detail shown and the footprint may need to be simplified.



Simplified example.




The **BASE** point of the symbol should be the midpoint of the top line in this example but it should be relative to the midpoint of the DIN rail that it will be positioned on.

The WIRENO & TERM attributes are generally positioned centrally with the WIRENOL & WIRENOR to the correct sides. The other attributes are generally made invisible in their properties.



WIPEOUT can be used to hide the DIN rail behind the components. Footprints can have a wipeout pre-defined.

Select *Home* ribbon tab > *Draw* panel ↓ > 
 Select the intersection of each corner and then select *Close*



DRAWORDER can be defined so that the Wipeout is behind the footprints but in front of the DIN rail.

Select the wipeout rectangle you have just created and then right click > **DRAWORDER** > Send to back




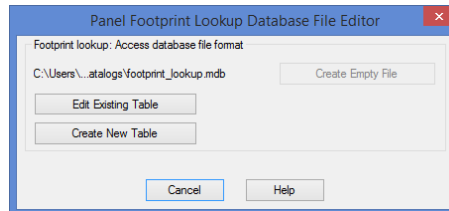
Save the drawing in a similar directory structure to other footprints within the PANEL_MM directory

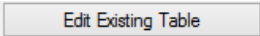
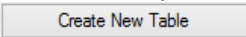
...\PANEL_MM\WIELAND\TRMS-TERMINAL BLOCKS\WT2_5.dwg


DO NOT use a “,” or a “.” In the filename

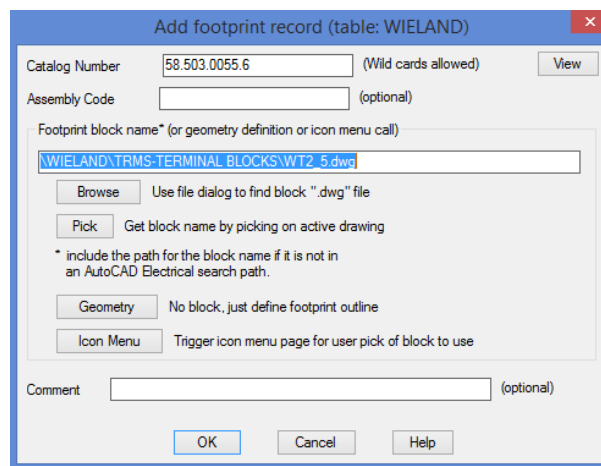
Associating the Footprint to the Part

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



If the manufacturer has already been defined in the footprint database, select  otherwise select 

Select 



Ensure you enter the part number correctly. Wildcards can be used.

The footprint location does not have to contain the full file path and can be just the file path from within the PANEL_MM directory e.g. WIELAND/TRMS-TERMINAL BLOCKS/WT2_5

Note: the use of “/” and also the “.DWG” extension is not required

Select 

Select  to accept the changes