

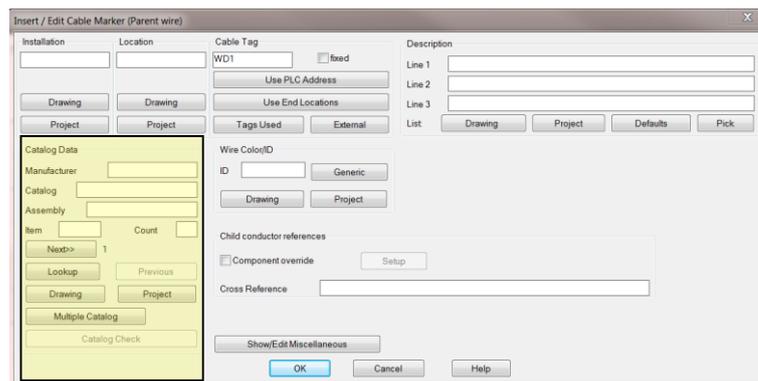
# Creating a New Cable in the Parts Database

## AutoCAD Electrical 2015 >

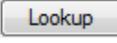
AutoCAD Electrical’s 2015 parts database looks significantly different from the previous release. In the past we have covered through a white paper the steps required to create a new cable in the parts database. We thought it prudent based on the new interface style to reproduce this document for 2015 and upwards.



Right click over an inserted schematic cable symbol and select Edit Component or select  from the Schematic ribbon tab > Edit Components panel



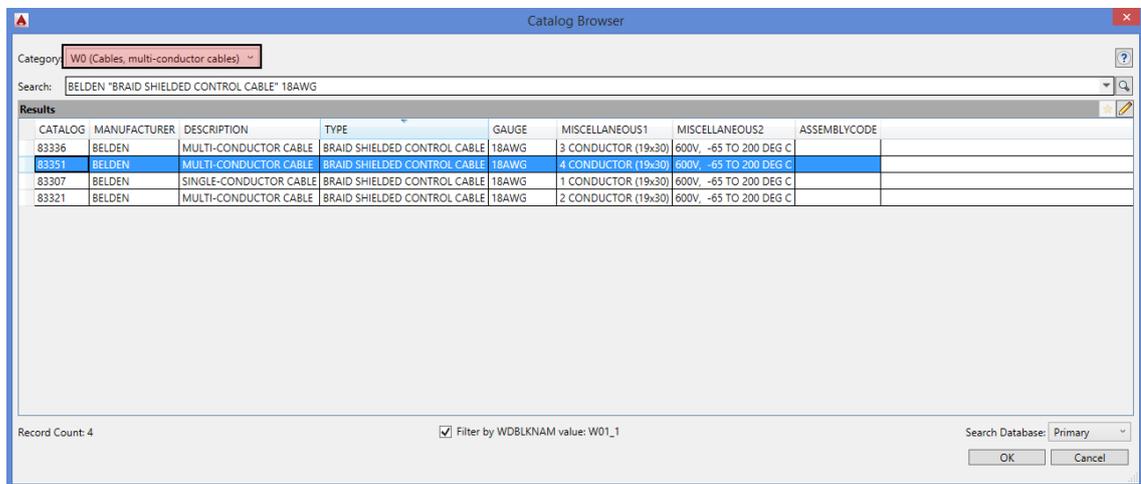
Manufacturer and part information is defined in the section highlighted.

The **primary part** is defined by selected the option 

**Optional parts** are defined by selecting the option 



AutoCAD Electrical automatically filters on the correct type of parts e.g. **W0 table = cables**



Parts can be edited or added



Parts can be filtered easily based on the search text

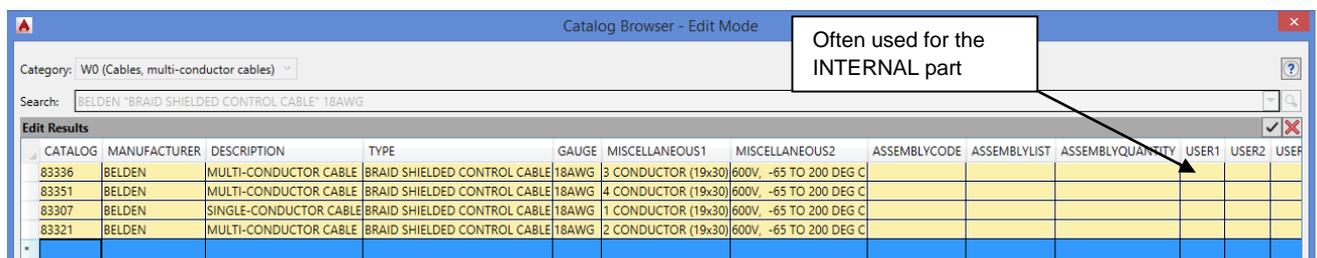


Displays the catalogue entries previously added to the favourites list for the active category

The order of the fields shown i.e. CATALOG, MANUFACTURER, and DESCRIPTION etc can be changed by dragging the **column headers**.

If you  edit the parts database to add a new part, it is easier to highlight an existing part and copy the information from the existing part rather than type completely from scratch

If you don't, the dialogue will be as follows



Catalog	Specifies the catalogue number
Count	Specifies the number of parts to add to the BOM. A blank Count indicates a quantity of 1. A valid non-blank Count is 1 through 100
Manufacturer	Specifies the manufacturer for the catalogue part
Description	Specifies the optional description text for the catalogue part
Type	Specifies the type for the catalogue part
Gauge	The type of value is based on the type of component. This value populates the fifth field in the catalogue record
Miscellaneous	The type of value can vary based on the type of component. This value populates the sixth field in the catalogue record
Assemblycode	Specifies the code to flag that this item has subassembly items. "As main-subassembly" activates the Assembly Code edit box. Use a unique name code to link this main catalogue item with other subassembly items. This code can be up to 60 characters. Spaces are allowed
Assemblylist	Specifies the code to flag as a subassembly item of a main item. To enter the ASSEMBLYLIST value, select "As subassembly" and enter the exact name of the ASSEMBLYCODE value carried by its main component. You can select the ASSEMBLYCODE list switch to speed up the process
User 1,2,3	Specifies user information. AutoCAD Electrical provides three blank user fields for your use. Each can be a maximum of 24 characters wide and are extracted into BOM reports along with all the other fields
Textvalue	Specifies optional user-defined RATING/misc attribute values. This field is used to assign text values to specific attributes on the edited component. The format for the text data encoded into this field is:  <attribute tag name1>=<text value>;<attribute tag name2>=<text value>
Weblink	Specifies the .pdf file or Web URL to associate to the component. If the Weblink entry is a Web URL, your Internet browser displays it. If it is an image file, .pdf, spreadsheet, or some other document type, then the application associated with that file extension (for example, "Open With...") displays the file.

Note: For ".pdf" display, you can include the page number to display upon document open. Add a space and the page number after the .pdf file name in the WEBLINK field value (c:\rockwell\700series.pdf 13)

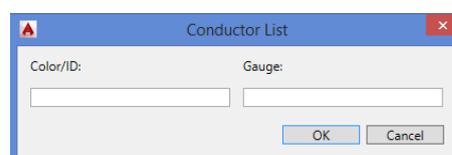
WDBLKNAM	Specifies the schematic block name (used for catalogue lookup - that is, PB11, CR) of the catalogue record. It serves as a filtration of the catalogue records based on the schematic block name. This field is used in the Filter by WDBLKNAM value when opening up the catalogue lookup dialog box. This filter provides the mechanism to remove invalid selections from the catalogue lookup window, much like the component-specific tables
	Note: Use a comma to separate multiple symbol block names.
SYMBOL2D	Default schematic symbol association
SYMBOL3D	Default Inventor (IPT) file association
COILPINS	Coil pins of a device
PINLIST	pin list of a device
PEERCOILPINS	2 <sup>nd</sup> + coil pins of a device (e.g. fwd/rev combine contactor)
PEERPINLIST	2 <sup>nd</sup> + pin list of a device (e.g. fwd/rev combine contactor)
CONDUCTORLIST	DEFINED OR UNDEFINED core details of a cable

The **USER1** value is typically used for the company's **INTERNAL part number**. This column can be moved to a more primary position in the lookup table by dragging the column header as previously defined.

The text USER1 can also be changed in the Lookup table by changing the column header in the default\_cat.mdb file. The order of the columns in Microsoft Access however **MUST NOT BE** changed.

Select CONDUCTORLIST to the right of the cell 

Select the [...] to enable the dialogue to define the core colours and gauge (size)



As an example of a screened twsited cable:

Color/ID      BK;      Gauge 0.5mm;

Color/ID      RD;      Gauge 0.5mm;

Color/ID      Scr;      Gauge 0.5mm;

Select  to save the cores of the cable

Select  to save any changes or additions

Select  to cancel any changes or additions