

Insert Symbols With Part Number

AutoCAD Electrical has a utility that isn't talked about very often but is becoming more popular and users are starting to take advantage of. The utility/command I am referring to is the Catalog List command that resides on the Schematic tab on the Insert Component Ribbon and on the Panel tab on the Insert Component Footprint ribbon. This utility/command allows you to insert both schematic and panel symbols into your drawings by selecting your part numbers from a list instead of the more traditional way which would be using the Icon Menu. If you use the same part numbers on your designs this can be a very quick and easy way to insert your symbols. Even if you don't use the same part numbers from design to design there are many users out there that simply prefer this method of inserting symbols.

Schematic Component or Panel Footprint

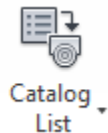
This utility/command inserts schematic or panel symbols by choosing a catalog number or a component description from a user-defined pick list. The data displayed in this pick list is stored in a database in generic Access format. The file name is wd_picklist.mdb and can be edited with Access or from Add/Edit/Delete along the bottom of the dialog box for the pick list. The AutoCAD Electrical normal search path sequence is used to locate this file.

Both schematic and panel layout symbols can be included in the pick list database. Only schematic or panel entries are displayed at a time depending on whether the routine is called from the AutoCAD Electrical Schematic or Panel tabs.

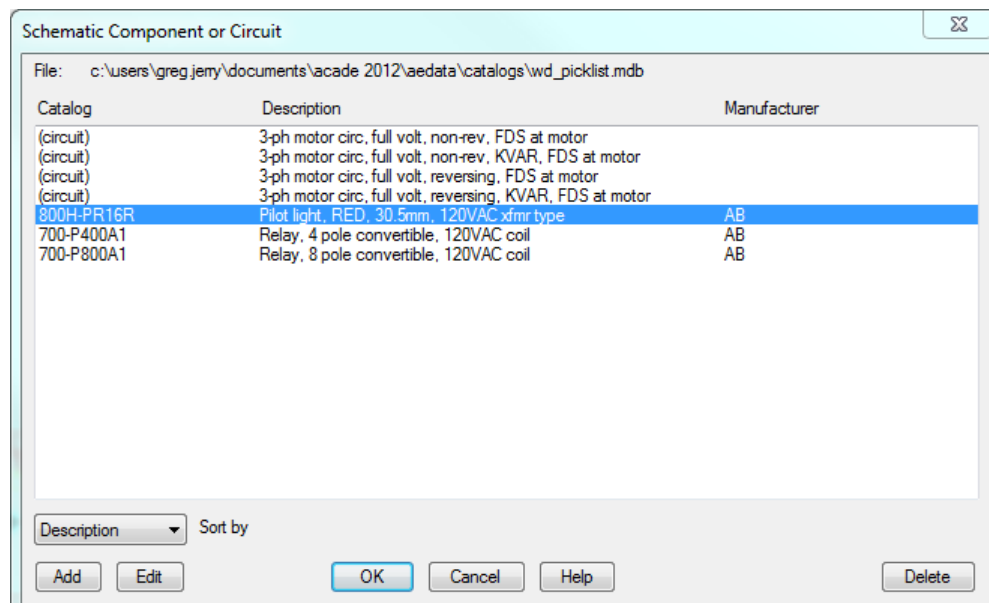
Procedure for Inserting Components from Catalog Lists

Note: This example displays the procedure for inserting schematic symbols. The same procedure can be used for panel symbols by going to the panel tab.

1. Click the Schematic Tab
 - ⇒ Insert Component Ribbon
 - ⇒ Insert Component Drop-Down Menu



- ⇒ Catalog List



2. Sort the component list by catalog, description, or manufacturer.
 3. Select the component to insert.
 4. (Optional) Click Edit to make any changes to the catalog record. Modify the record in the Edit Record dialog box and click OK.
 5. (Optional) Click Add to create a new record. If the new record is similar to an existing record, highlight the existing record before you click Add. Modify the record in the Add Record dialog box and click OK.
 6. Click OK.
 7. Specify an insertion point in the active drawing.
- Make any changes in the Insert/Edit Component dialog box and click OK.