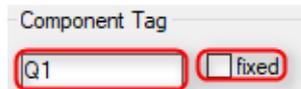# Override Family Tag Codes

AutoCAD Electrical 2018

A components tag can be manually overwritten on a per components basis, the tag imported from an external source, the family tag changed on a per project basis or the symbol permanently changed with a revised tag prefix. If changing a symbols permanent tag prefix, children should also be changed to the same family code. Furthermore, this can also be done by Microsoft Excel editing.

Manually Overriding A Component

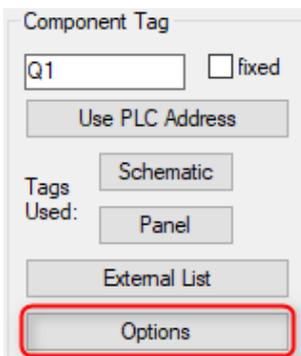*Right click* over the component Edit Component

Type in the value and select ☑ Fixed

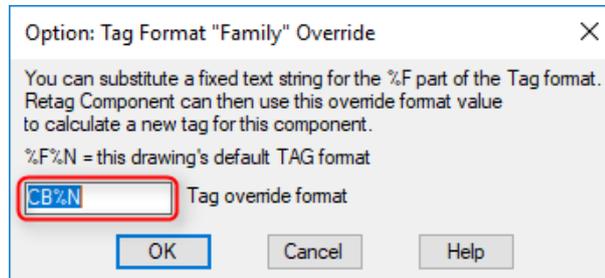All associated components such as children, footprints etc. will automatically be updated.

Changing The Automatic Prefix Of The Numbering On An Individual Component

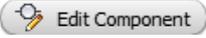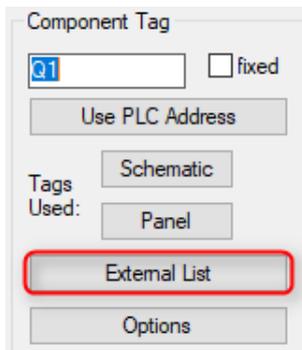*Right click* over the component Edit Component

Select

An example being "CB%N". This would number the device with a prefix of "CB" and then either the sequential or reference based automatic number.
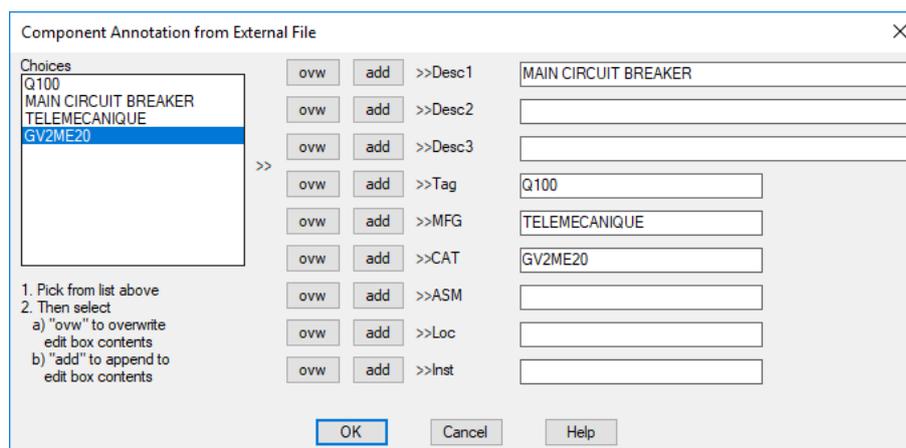
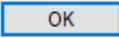### Importing The Tag On An Individual Component

Right click over the component

Select

The list must be saved as either a *.TXT or *.CSV file and should be structured. You are then able to map the text fields to what it means to AutoCAD Electrical

Select

Changing The Family Tag For All Components Of The Same Type

The wd_fam.dat file overrides the family tag code of library symbols by mapping the codes to new values. The tag code of a symbol is used in generating the tag-ID of inserted components, such as the "Q" of tag "Q100."

AutoCAD Electrical searches for this mapping file in the following order:

1. User subdirectory

    C:\Users\<<user name>>\AppData\Roaming\Autodesk\AutoCAD Electrical 2018\R22.0\enu\Support\User

2. Active <<project's>>.wdp file subdirectory
3. All paths defined under AutoCAD Options > Files tab > Support Files Search Paths

Depending on how you want to override component family names, you can move the wd_fam.dat file into the various locations mentioned above.

To always substitute a new family value for all projects you create, place the file in the User folder. (option 1)

To use AutoCAD Electrical defaults most of the time but sometimes override them with project-specific defaults, place the file in the project directory folder for the project you want to override. You can have different defaults for each project and therefore client. (option 2)

The WD_FAM.DAT file is a text file and can be created using Windows Notepad.

You can select a different name for a component family by editing the WD_FAM.DAT file. For example, if you wanted all diodes to be 'V's' instead of 'D's' you would need to add the following line to the file:

D,V

The change takes effect when you reload AutoCAD Electrical. New diode components you insert receive the "V" family code annotation instead of the library default of "D,".

Use *Retag Components* to update previously inserted components.

Changing the Library Symbol Permanently

On some components, you may wish to permanently change the TAG value but in the symbol rather than a mapping file. Hover over the symbol in the icon menu to find out the symbol name. Browse to the library where the symbol is located and open the drawing file.

Edit the TAG1 value by double clicking on the attribute or using the command DDEDIT. Change the value from the old to the new. As an example, in the symbol VSV1, you may wish to change the solenoid valve tag prefix from K to Y. Save any changes made. All new instances of the block will adopt the new numbering methodology. Existing symbols may be updated using the Update Block command.