

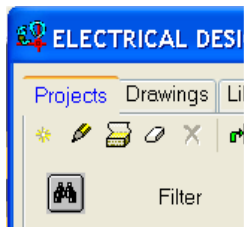
Adding Parts that are not in the Schematic

All Electrical Designer Users

Any part can be added into the BOM or indeed the panel assemblies without having to insert the part into a schematic.

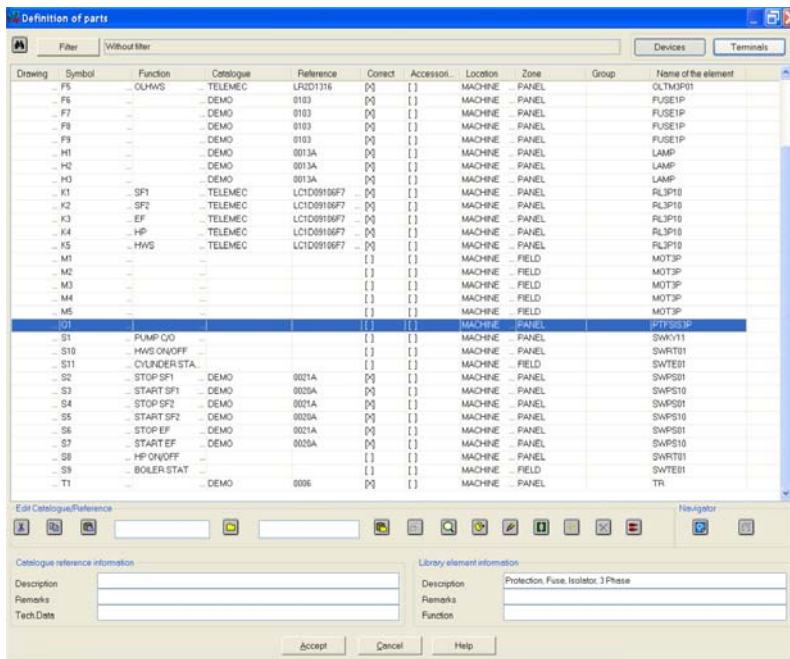
As an example, you may have inserted an isolator into the schematic but you would like to show both the isolator body and handle in the panel assembly. In this instance, the handle is not an accessory to the isolator but a primary part associated to the isolator.


In Electrical Designer select the *Projects* tab




Select the  Define Parts icon.

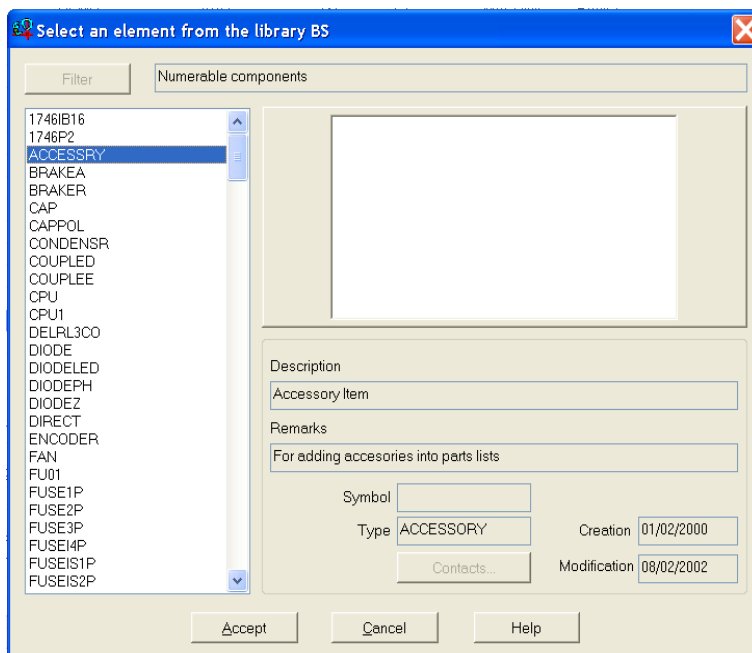
The *Definition Of Parts* dialogue will open. Highlight the part (e.g. isolator) from the list of components.



Select the  *Catalogue / Reference* icon and assign a manufacturer and part to the main body of the isolator. In this example we will use a catalogue of *Demo* and a reference of *2015*.

Still highlighted on the isolator in the list, select the icon  *Insert*.

Select any element from the library such as *Accessory* and *Accept*.



The component SNA and details are copied but you can assign a different catalogue and reference. In this example, the part *Demo : 6000*.

Inserted component data

Quantity: 1

Symbol: Q1

Location: MACHINE

Zone: PANEL

Group:

Catalogue: DEMO

Reference: 6000

Accept Cancel Help

Both of these parts will now be available for insertion in a panel assembly.

Note: The part must have valid height, width and depth information or a footprint in the Edit Reference dialogue.