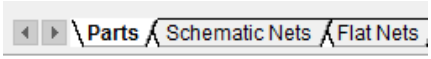
User Defined Properties from Schematic to PCB

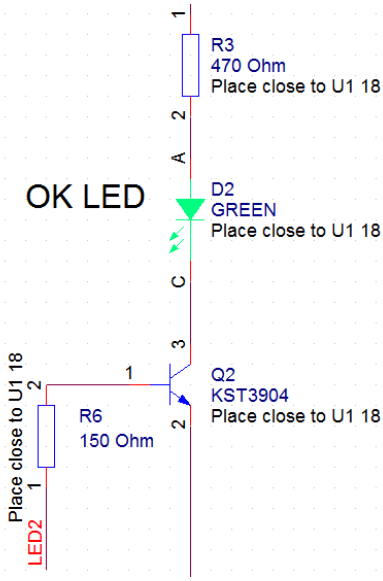
There are many instances in designs today where the person designing the electrical circuit or schematic is different from the person physically designing the printed circuit board. In these types of situations there may well be certain properties that the schematic engineer would like to pass as a property so that the PCB designer gets visibility. There are default properties like Physical Constraint Set, Spacing Constraint Set, Differential Pair as well as Constraint overrides such as MIN_LINE_WIDTH, MAX_LINE_WIDTH that are transferred automatically if set in the schematic. This app note will cover setting a user defined property such as DESIGN_NOTE which will allow the Electrical Engineer to pass on design notes or special layout instructions that can be visible on a part in PCB Editor.

Adding the Comment Property to OrCAD Capture (CIS or DE CIS)

For the specific parts that you want to add a comment property to in OrCAD Capture either double click the part or select the part and then right click – Edit Properties to bring up the Property Editor. Once the Property Editor is open, firstly ensure you are on the Parts tab at the bottom



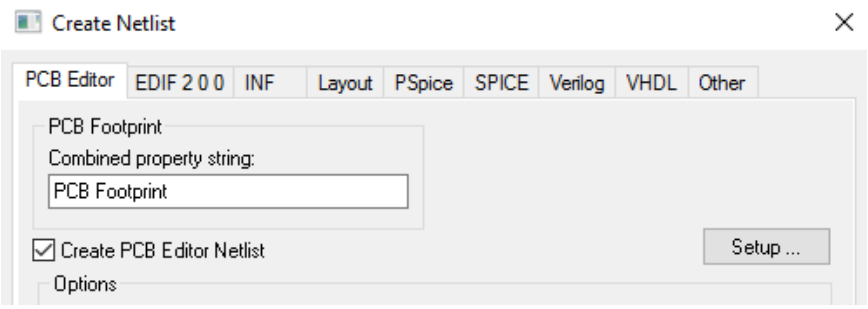
Then click on New Property and populate the Property name and value. In this example the property name is DESIGN_NOTE and the value is the actual comment so “Place close to U1 18”. If required, make the property visible so it can be seen on the schematic. This can be done inside the Property Editor, select the property name column then right click – Display, choose what to display (Value only in this example).



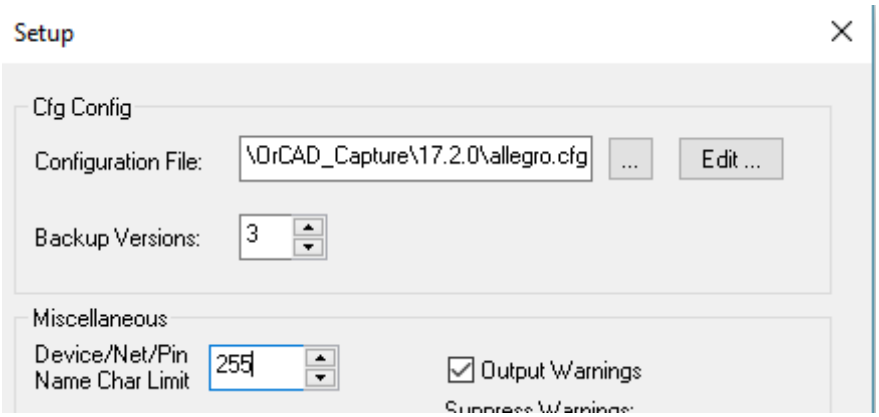
CLASS	DISCRETE	DISCRETE	DISCRETE	DISCRETE
Color		Default	Default	Default
Description	DIO, LED\Green, GREEN, --,	XSTR, BJT, NPN, KST3904,	RES, Thick Film, 470 Ohm, 1.	RES, Thick Film, 150 Ohm, 5.
DESIGN_NOTE	Place close to U1 18	Place close to U1 18	Place close to U1 18	Place close to U1 18
Designator				

Repeat as necessary for all parts in the design. Once finished save the design.

We are now ready to generate a netlist. Launch the Netlist command from Tools – Create Netlist, before we generate this we need to include the new property so it is transferred to PCB Editor. Select the Setup button:-



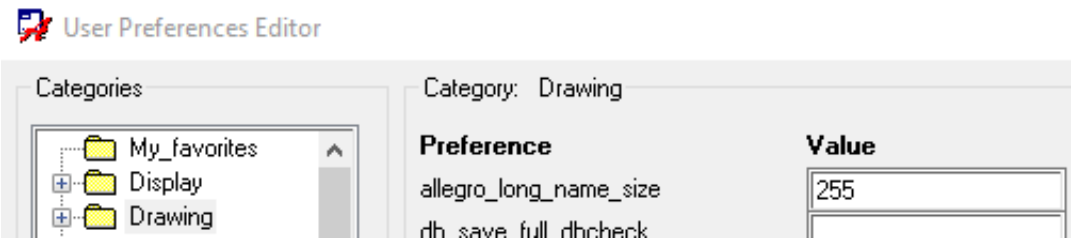
Then click on Edit to edit the allegro.cfg file to add the new property.



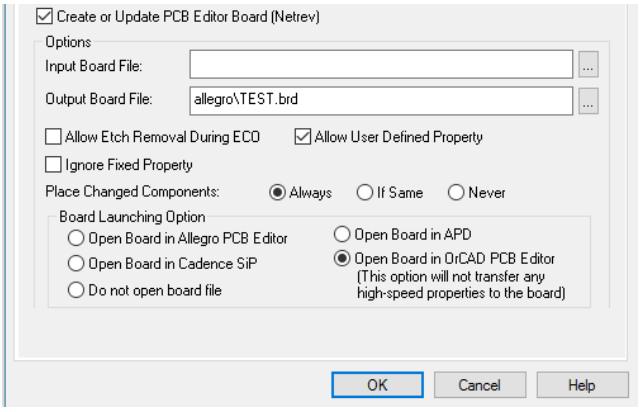
Add the following line to the section for [ComponentInstanceProps] DESIGN_NOTE=YES.

```
[ComponentInstanceProps]
GROUP=YES
ROOM=YES
VOLTAGE=YES
SIGNAL_MODEL=YES
NO_XNET_CONNECTION=YES
DESIGN_NOTE=YES
```

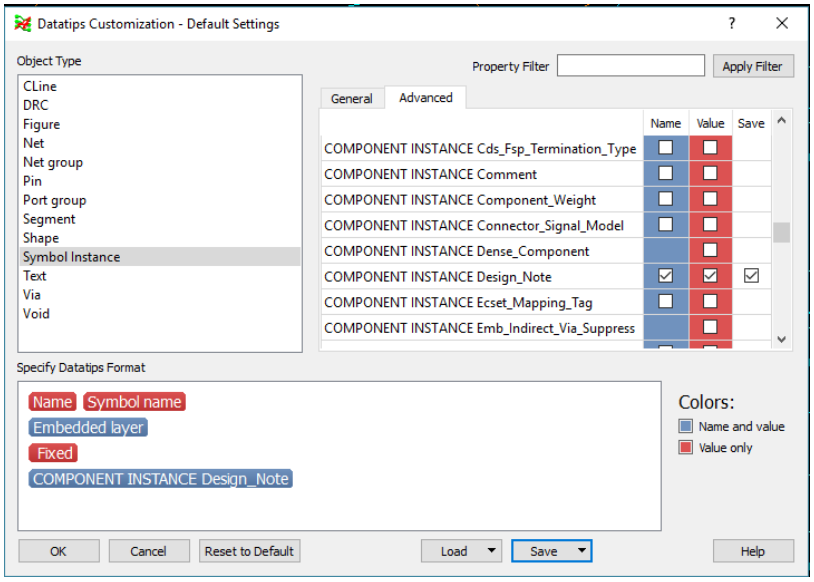
Save and close the allegro.cfg file. You may also need to increase the Device/Net/Pin/Name Char Limit depending on the length of the comments you have added. In general, this is good practice anyway and can be set to 255. You can make a similar setting in PCB Editor. Use Setup – User Preferences - Drawing and set the allegro_long_name_size to 255.



Once you make the necessary changes you can click OK to close the Setup form. You now need to ensure that you enable the checkbox to Allow User Defined Properties to transfer from OrCAD Capture into PCB Editor. Once enabled click OK to generate the netlist and open the board in PCB Editor.



Once inside PCB Editor we need to make sure that the Datatips can show this property. Go to Setup – Datatip Customization, select Symbol Instance, then the Advanced tab and locate the new property COMPONENT INSTANCE Design_Note and enable Name (Property Name), Value (Property Value) and Save if required.



If you Save this form this will write a custdatatips.cdt file to your %HOME%\pcbenv folder. You are now ready to place parts. If you hover over any part that has that property you will see the following tooltip that is driven by the settings above.