# 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.

Important Note:- If you enable the Constraint Manager flow, any net based properties will be driven via the Design Sync command. For example:- Physical Constraint Set, Spacing Constraint Set, Differential Pair and min line width. Enabling the Constraint Manager flow will not affect component based properties.

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

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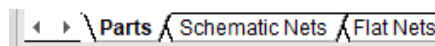
Then click on New Property and populate the Property name and value. In this example the property name is DESIGN_NOTE and the value is the actual comment so "Place close to U1 18". If required, make the property visible so it can be seen on the schematic. This can be done inside the Property Editor,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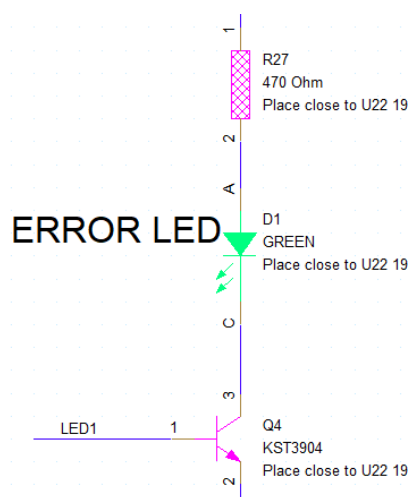

CLASS	DISCRETE	DISCRETE	DISCRETE
Color		Default	
Description	DIO, LED\Green, GREEN, --,	XSTR, BJT, NPN, KST3904,	RES, Thick Film, 470 Ohm, 1.
DESIGN_NOTE	Place close to U22 19	Place close to U22 19	Place close to U22 19

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

We are now ready to run Design Sync. Before we run this command, we need to configure the tools to include the new property, so it is transferred to PCB Editor. Go to PCB > Design Sync Setup, then click the Edit icon to edit the allegro.cfg file to add the new property.
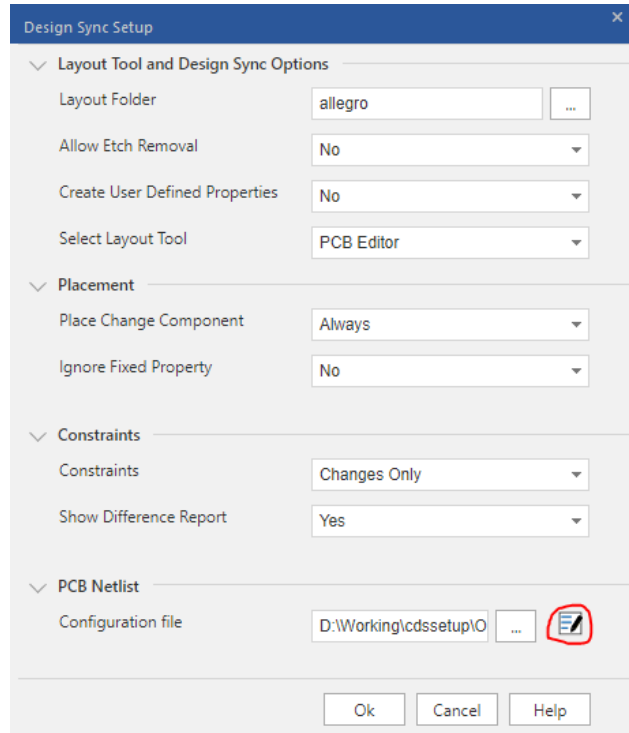

Add the following line to the section for [ComponentInstanceProps] DESIGN_NOTE=YES.

```
[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 now need to ensure that the User Defined Properties are transferred from OrCAD Capture into PCB Editor. Change the Dropdown from No to Yes then OK the Design Sync Setup form.
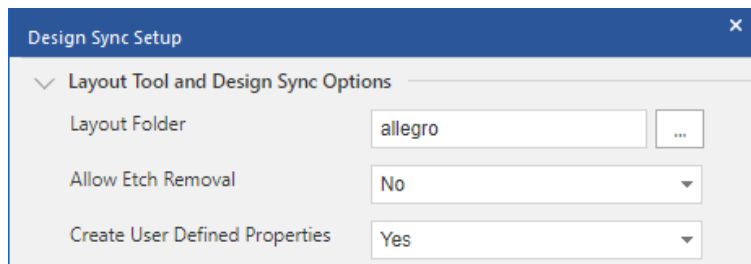

Once the setup is complete run PCB > Design Sync to create the board file and open PCB Editor as you would normally.
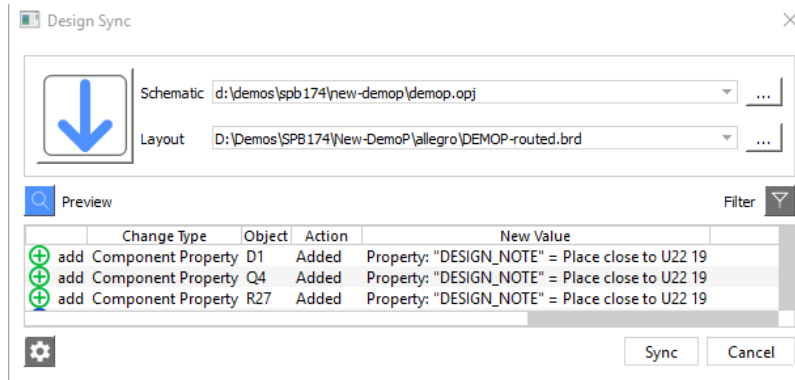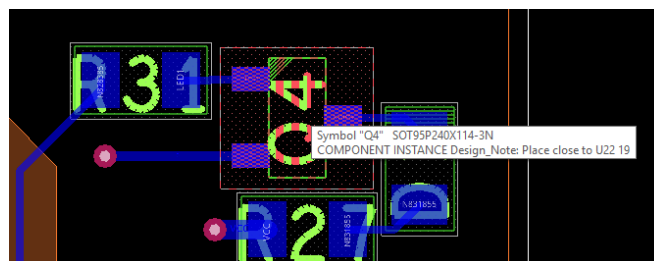
Once inside PCB Editor we need to make sure that the Datatips can show this property. Go to Setup > Datatip Customization, select Symbol Instance, then the Advanced tab and locate the new property COMPONENT INSTANCE Design_Note and enable Name (Property Name), Value (Property Value) and Save if required.
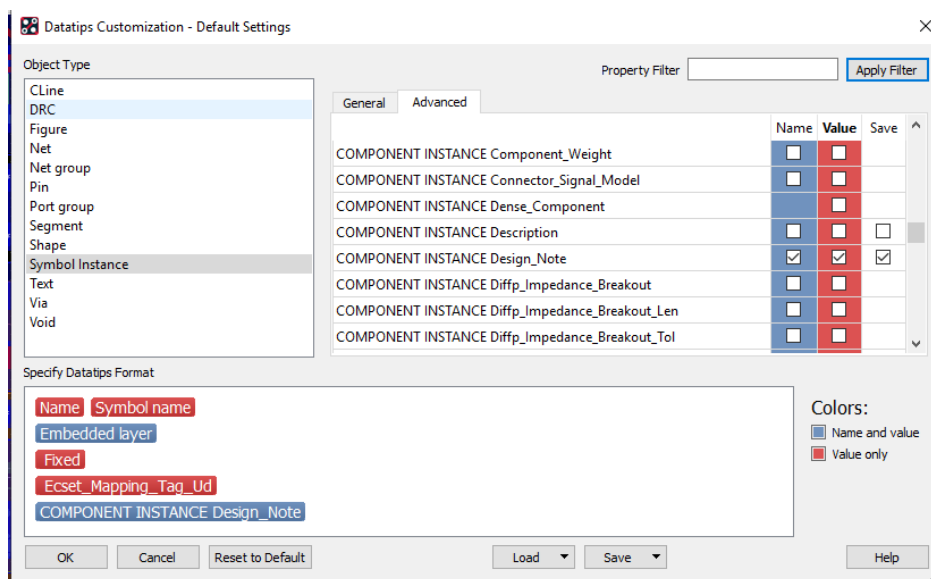
If you Save this form this will write a custdatatips.cdt file to your %HOME%\pcbenv folder. You are now ready to place parts. If you hover over any part that has that property you will see the following tooltip that is driven by the settings above.