



## User Defined Properties from Schematic to PCB

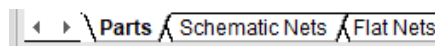
### User Defined Properties from Schematic to PCB

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.

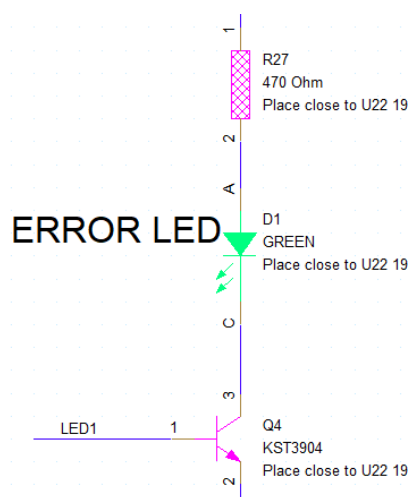
Important Note:- If you enable the Constraint Manager flow, any net based properties will be driven via the Design Sync command. For example:- Physical Constraint Set, Spacing Constraint Set, Differential Pair and min line width. Enabling the Constraint Manager flow will not affect component based properties.

### 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



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).

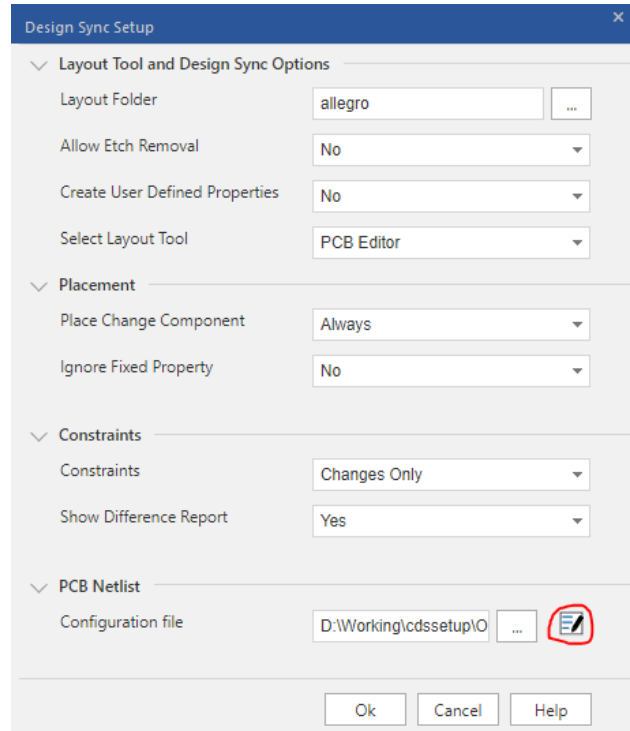


CLASS	DISCRETE	DISCRETE	DISCRETE
Color		Default	
Description	DIO, LED\Green, GREEN, --,	XSTR, BJT, NPN, KST3904,	RES, Thick Film, 470 Ohm, 1,
DESIGN_NOTE	Place close to U22 19	Place close to U22 19	Place close to U22 19

## User Defined Properties from Schematic to PCB

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
```

Note on allegro.cfg properties list. There are a few options when adding property names to the allegro.cfg file. For example, your company uses MPN instead of Part\_Number. This can still be driven through to PCB Editor. In this example you would use

```
MPN=PART_NUMBER
PART_NUMBER=YES
```

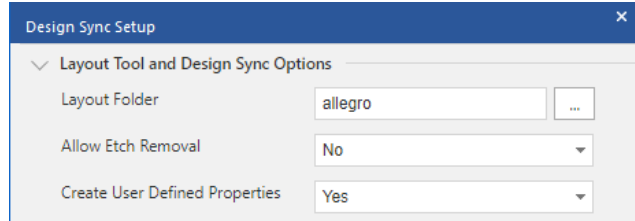
If your property name has a space in it you will need the following:-

```
PACKAGE HEIGHT=HEIGHT
HEIGHT=YES
```

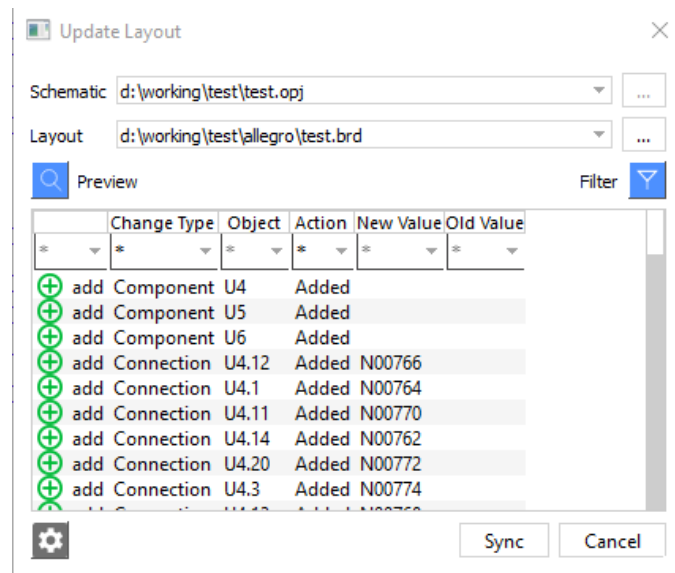
## User Defined Properties from Schematic to PCB

Save and close the allegro.cfg file. The file is now written to a new location which is usually %HOME%\cdssetup\orcad\_capture\17.4.0 so ensure you browse for this location for the Configuration file.

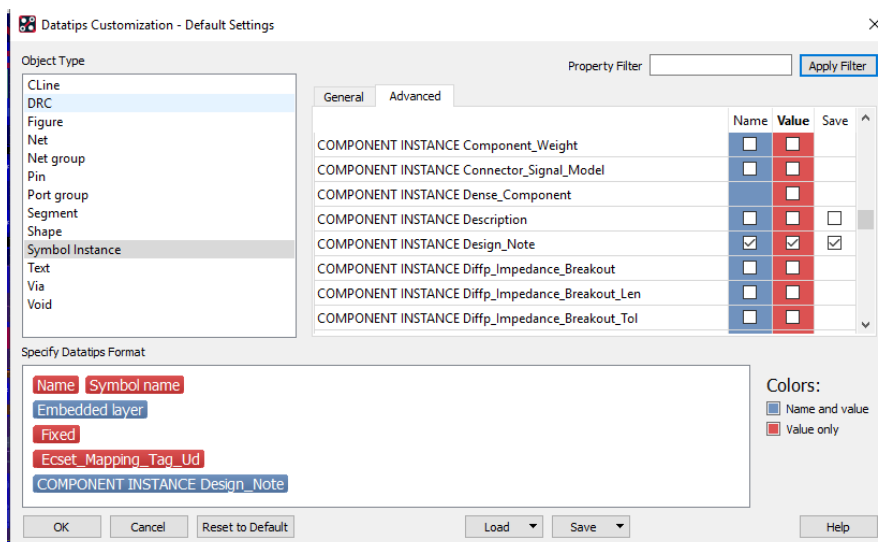
You now need to ensure that the User Defined Properties are transferred from OrCAD Capture into PCB Editor. Change the Dropdown from No to Yes then OK the Design Sync Setup form.



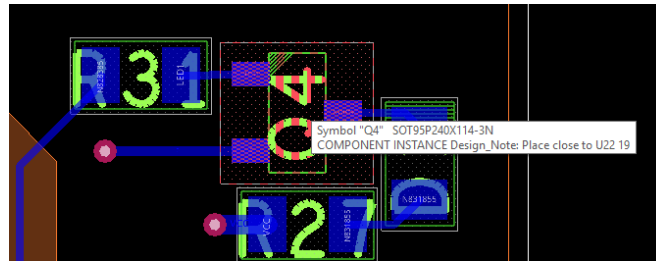
Once the setup is complete run PCB > New Layout (or Update Layout if you already have a board file) to create the board file and open PCB Editor as you would normally.



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.



The following are trademarks or registered trademarks of Cadence Design Systems, Inc. 555 River Oaks Parkway, San Jose, CA 95134 Allegro®, Cadence®, Cadence logo™, Concept®, NC-Verilog®, OrCAD®, PSpice®, SPECCTRA®, Verilog®

#### Other Trademarks

All other trademarks are the exclusive property of their prospective owners.

**NOTICE OF DISCLAIMER:** Parallel Systems is providing this design, code, or information "as is." By providing the design, code, or information as one possible implementation of this feature, application, or standard, Parallel Systems makes no representation that this implementation is free from any claims of infringement. You are responsible for obtaining any rights you may require for your implementation. Parallel Systems expressly disclaims any warranty whatsoever with respect to the adequacy of the implementation, including but not limited to any warranties or representations that this implementation is free from claims of infringement and any implied warranties of merchantability or fitness for a particular purpose.