

### **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

#### 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).



		<u> </u>	<u> </u>
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

© 2020 Parallel Systems Limited

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.

Design Sync Setup	×
$ \bigtriangledown $ Layout Tool and Design Sync Optic	ons
Layout Folder	allegro
Allow Etch Removal	No 👻
Create User Defined Properties	No 👻
Select Layout Tool	PCB Editor 👻
✓ Placement	
Place Change Component	Always 👻
Ignore Fixed Property	No 👻
✓ Constraints	
Constraints	Changes Only 👻
Show Difference Report	Yes 💌
V PCB Netlist Configuration file	D:\Working\cdssetup\0
	Ok Cancel Help

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.

Design Sync Setup	×
$\checkmark$ Layout Tool and Design Sync Optic	ons
Layout Folder	allegro
Allow Etch Removal	No 👻
Create User Defined Properties	Yes 💌

Once the setup is complete run PCB > Design Sync to create the board file and open PCB Editor as you would normally.

User Defined Properties from Schematic to PCI	User	Defined	Properties	from	Schematic	to	PCB
---	------	---------	------------	------	-----------	----	-----

	Schematic	d:\demos	s\spb174\new	-demop\demop.opj	·
Layout D:\Demos\SPB174\Wew-DemoP\allegro\DEMOP-routed.brd				·	
Q Pre	view				Filter
Q Pre	view Change Type	Obje	ect Action	New Value	Filter
			ect Action Added	New Value Property: "DESIGN_NOTE" = Place close to U22 19	Filter
e add	Change Type	perty D1	Added		Filter

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.

🚼 Datatips Customization - Default Settings		×
Object Type	Property Filter	Apply Filter
CLine	General Advanced	
DRC Figure		ame Value Save ^
Net	COMPONENT INSTANCE Component Weight	
Net group	1 - 2	
Pin Port group	COMPONENT INSTANCE Dense Component	
Segment		
Shape		
Symbol Instance Text		
Via		
Void		
	COMPONENT INSTANCE Diffp_Impedance_Breakout_Tol	□ <b>□</b> <sub>→</sub>
Specify Datatips Format		
Name Symbol name		Colors:
Embedded layer		Name and value
Fixed		Value only
Ecset_Mapping_Tag_Ud		
COMPONENT INSTANCE Design_Note		
OK Cancel Reset to Default	Load 🔻 Save 💌	Help

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.