



## PCB Titleblocks using Schematic Data

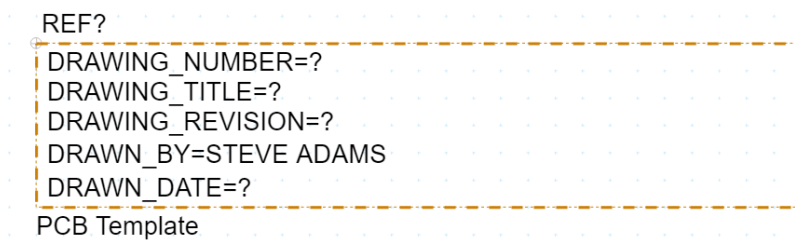
### How to drive PCB Titleblocks from data in the Schematic (OrCAD Capture).

Many users use Titleblock properties in the schematic for design name, revision, document number etc. In most cases this are the same properties required in a PCB design where users will supply their data with a Titleblock. Currently this is a manual process. This app note will describe the process of creating a symbol for the schematic which has various properties (user defined) that can be transferred and displayed inside PCB Editor.

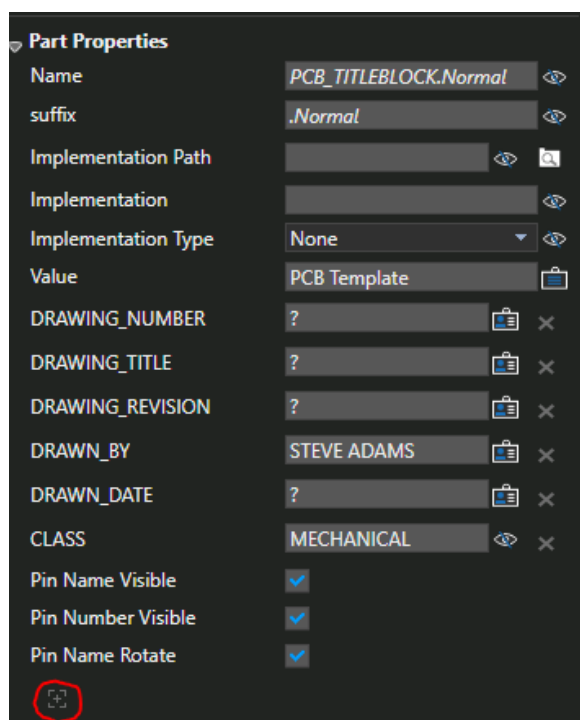
#### Create the schematic symbol placeholder

Start OrCAD Capture and either open an existing library or create a new library (File – New – Library). Create a new Part (Select Library Name and right click – New Part) then specify a symbol name, PCB Footprint (this will link to a mechanical symbol titleblock) and specify a RefDes placeholder (in this example REF is used).

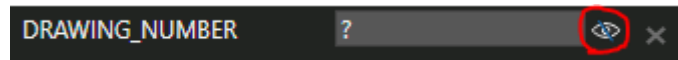
You now need to create the symbol graphics. No pins are required for this symbol (because we have no connectivity) so draw the outline / lines etc as required. In this example I have simply drawn an outline to contain the properties.



To add the required properties, click on the + sign at the bottom of the Part Properties window and type a name and value. In this example I have used a ? for the actual values since this would be different between designs and can be edited in the actual schematic.



To make the properties visible, click on the Eye icon and choose Name and Value. Once set the properties are displayed and can be positioned as required.



A key property required is CLASS=MECHANICAL which ensures that symbol can be transferred to the PCB via a Netlist / Design Sync and can be placed as a mechanical part (so no connectivity). This property does not need to be visible in the symbol but MUST exist.

Once complete save the symbol and start/open your Capture design. Place the Symbol and double click to edit the properties of the symbol to set the required values.

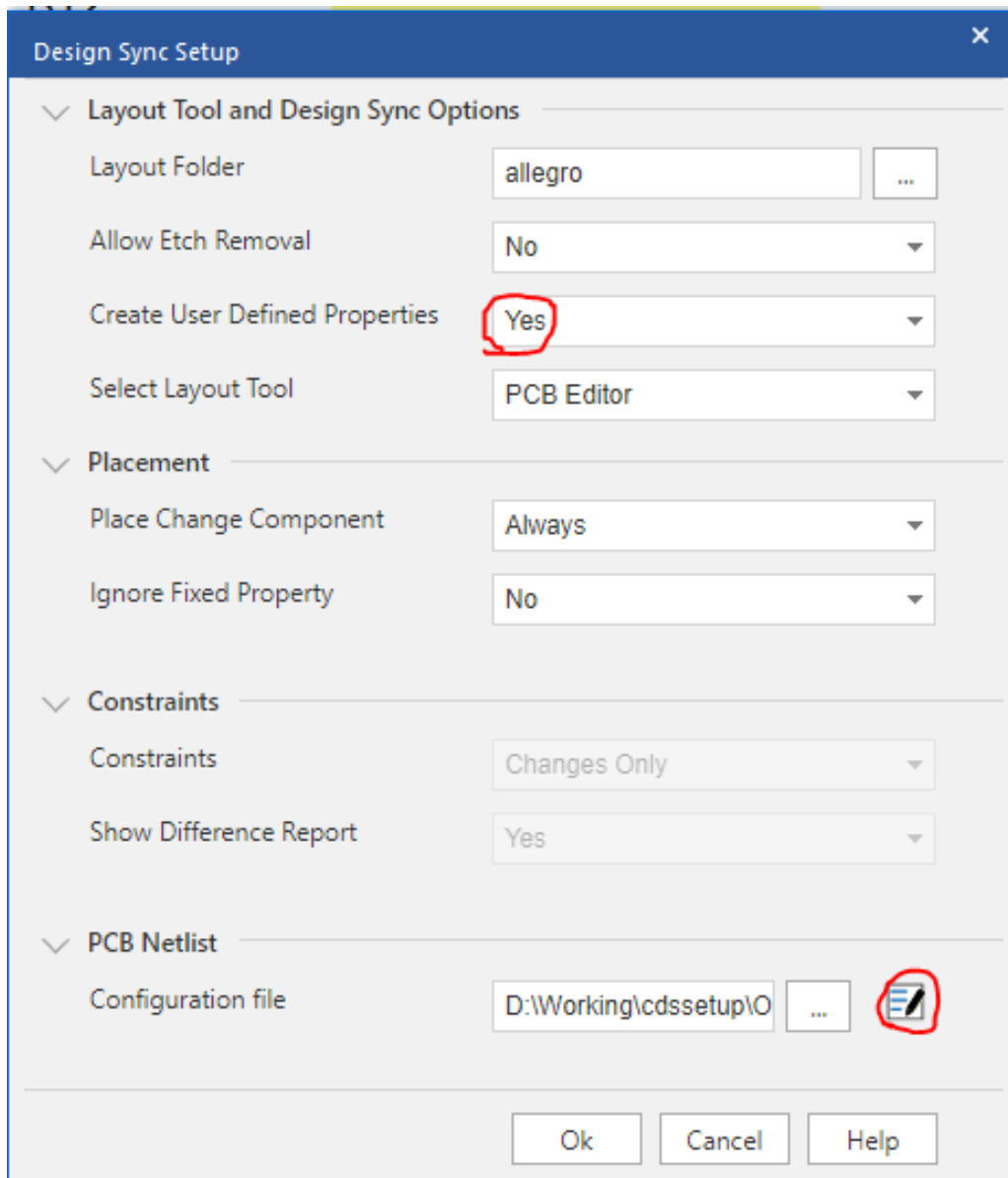
	A
	+ SCHEMATIC1 : PAGE1
CLASS	MECHANICAL
Color	Default
Designator	
DRAWING_NUMBER	ABC1234567890
DRAWING_REVISION	A
DRAWING_TITLE	MY DESIGN
DRAWN_BY	STEVE ADAMS
DRAWN_DATE	22/01/2021
Graphic	PCB_TITLEBLOCK.Normal
ID	
Implementation	
Implementation Path	...
Implementation Type	<none>
Location X-Coordinate	830
Location Y-Coordinate	1040
Name	INS1939095
Part Reference	REF1
PCB Footprint	PCB_TITLEBLOCK
Power Pins Visible	<input type="checkbox"/>
Primitive	DEFAULT
Reference	REF1
Source Library	D:\LIBRARY\SCHEMATI...
Source Package	PCB_TITLEBLOCK
Source Part	PCB_TITLEBLOCK.Normal
Value	PCB Template

REF1

DRAWING\_NUMBER = ABC1234567890  
 DRAWING\_TITLE = MY DESIGN  
 DRAWING\_REVISION = A  
 DRAWN\_BY = STEVE ADAMS  
 DRAWN\_DATE = 22/01/2021

PCB Template

We are now ready to send this data to PCB via the Design Sync command. We need to configure the allegro.cfg file to send these properties to PCB Editor via the netlist files. Go to PCB – Design Sync Setup then click the Edit button next to the Configuration File. The allegro.cfg file is opened in a text editor.



Add the property names you wish to transfer to the [ComponentDefinitionProps] section. You can name the properties anything you want to except DESIGN\_NAME which is a reserved property. You also need to ensure that there is no “white space” between values. Use an underscore (\_) or hyphen (-) to separate words. In this example I have added the following:-

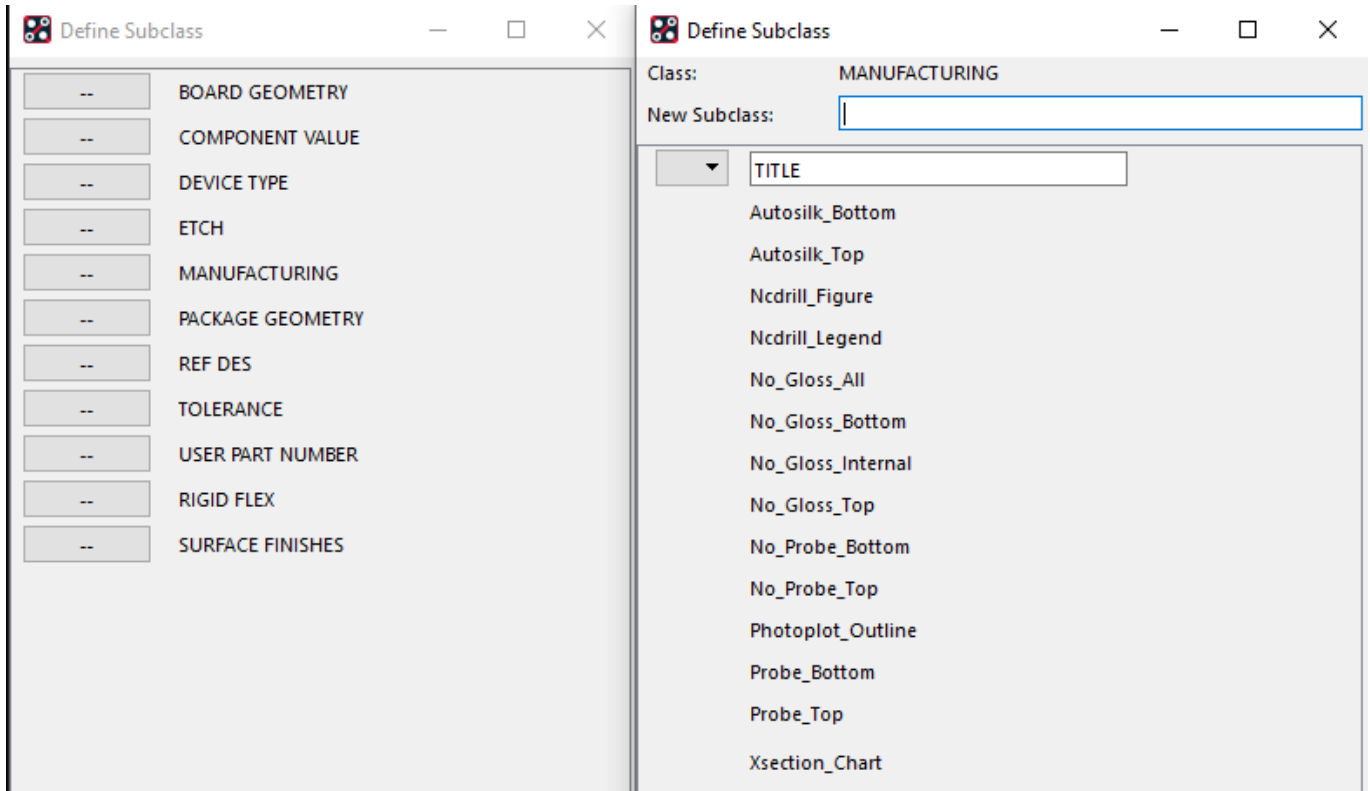
```
DRAWING_NUMBER=YES  
DRAWING_TITLE=YES  
DRAWING_REVISION=YES  
DRAWN_BY=YES  
DRAWN_DATE=YES
```

Save the allegro.cfg file. You also need to ensure that “Create User Defined Properties” is set to Yes.

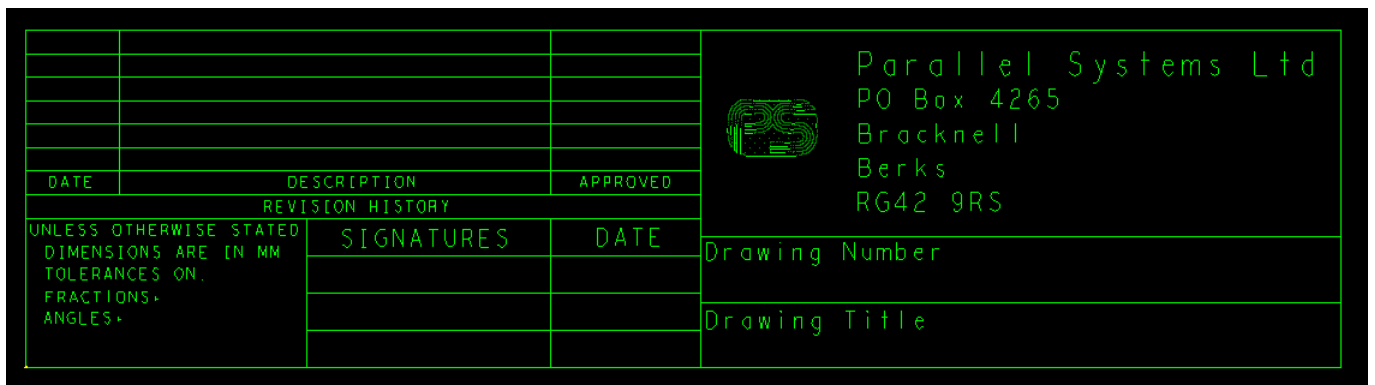
### Create a Mechanical Symbol for the Titleblock

Launch PCB Editor and choose File – New – Mechanical Symbol. Specify a name and location (controlled by your psmppath location). If you haven't set this up refer to :- [Defining padpath, psmppath and steppath \(parallel-systems.co.uk\)](http://www.parallel-systems.co.uk)

In this example I decided to create a user defined subclass to draw my titleblock on. Go to Setup - Subclasses, click on Manufacturing then in the New subclass form type TITLE followed by a return which adds the new subclass. OK to close the forms.

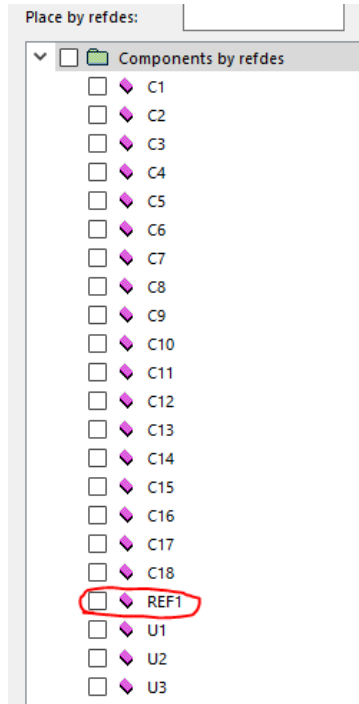


Design your Titleblock symbol as required. You only need to have text objects to reference each cell. The example below just uses text and lines to create the titleblock. You can also include company logos etc if required. Once complete save your symbol.



### Transferring Properties and displaying them

Run PCB – New Layout (or PCB – Update Layout if you have an existing design) to transfer the updated schematic netlist into PCB Editor. Once PCB Editor opens invoke the Placement Edit Application mode (Setup – Application Mode – Placement Edit) and in the Options pane you’ll see a list of parts ready to place. You should also see a RefDes for the Titleblock symbol REF1 (if that’s the RefDes placeholder you used).



Place the Titleblock in the PCB. A quick check to make sure the properties have been transferred invoke the command Check – Elements (Display – Element in Allegro), ensure that Comps is enabled on the Find pane then left click the Titleblock symbol. You should see a list of your properties:-

Show Element

LISTING: 1 element(s)

< COMPONENT INSTANCE >

Reference Designator: REF1  
 Package Symbol: PCB\_TITLEBLOCK

Component Class: MECHANICAL  
 Device Type: PCB\_TITLEBLOCK\_22/01/21  
 Value: PCB Template

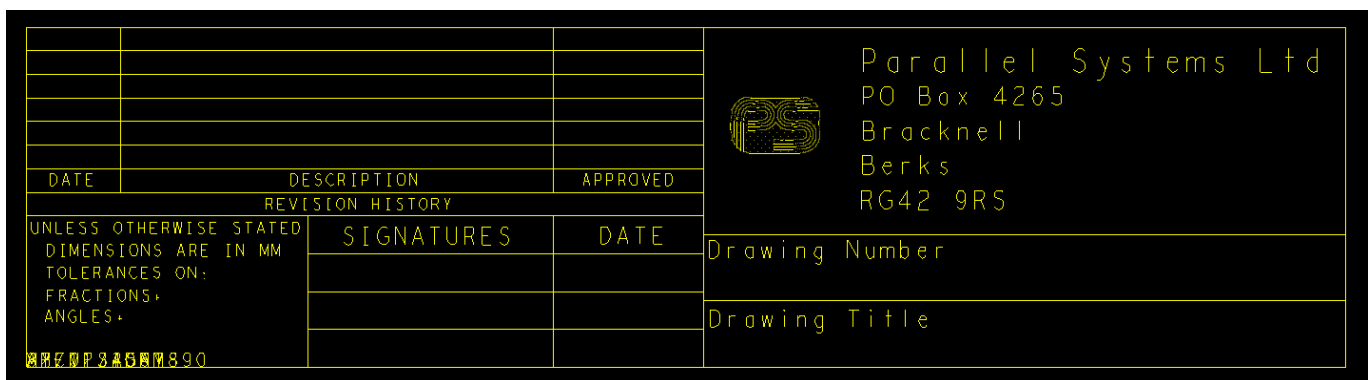
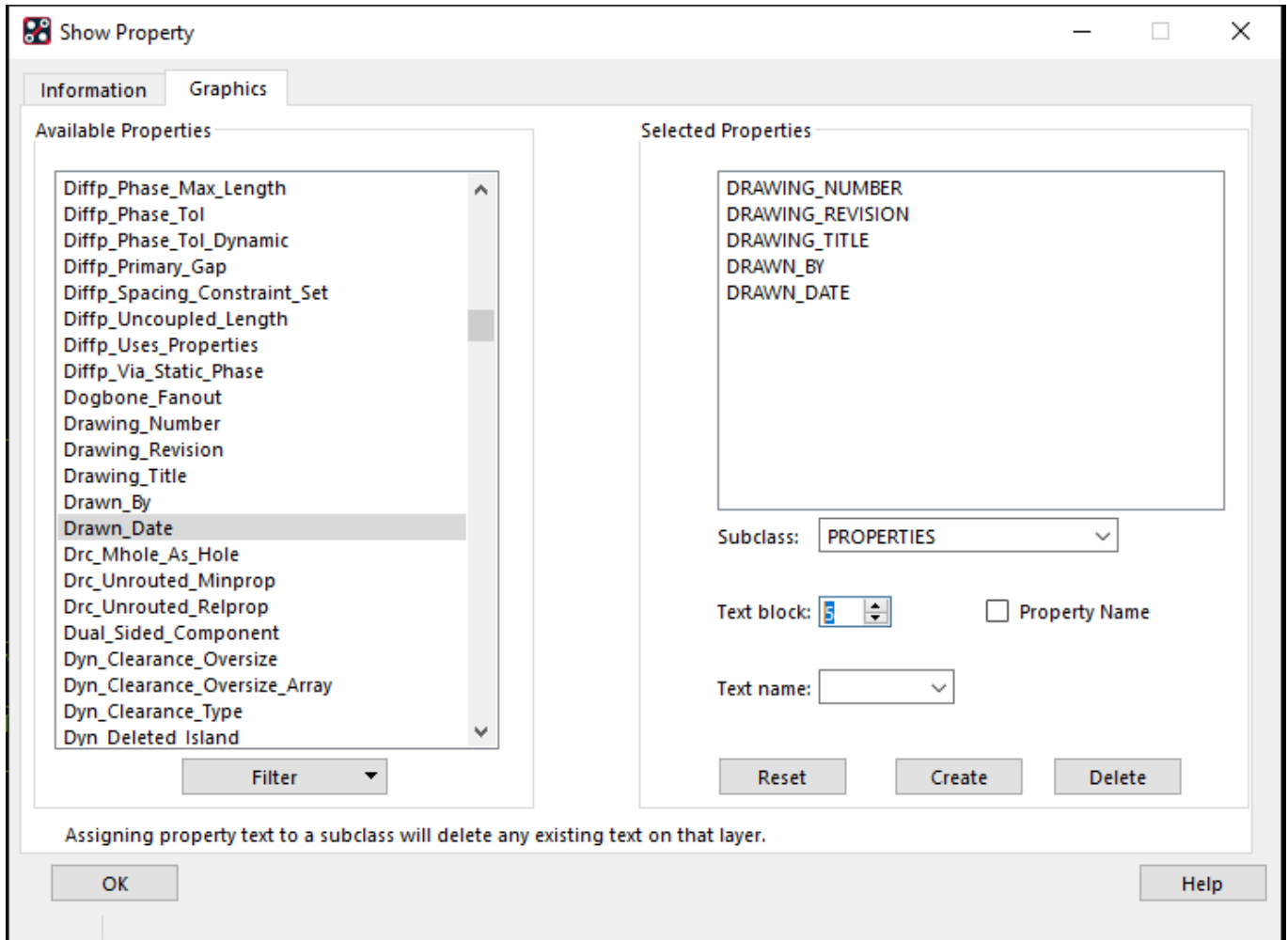
Placement Status: PLACED  
 origin-xy: (10.1600 7.6200)  
 rotation: 0.000 degrees  
 not\_mirrored

Function(s):  
 Designator: TF-11  
 Type: F-1

Properties attached to component definition  
 DRAWING\_NUMBER = ABC1234567890  
 DRAWING\_REVISION = A  
 DRAWING\_TITLE = MY DESIGN  
 DRAWN\_BY = STEVE ADAMS  
 DRAWN\_DATE = 22/01/2021  
 VALUE = PCB Template


## Making the Properties Visible in PCB Editor

We now want to make the properties visible so they can be positioned correctly. Use the command Check – Properties (Display – Property in Allegro) and in the Graphics tab scroll the list of available properties to locate your user defined ones, left click each property name and it is moved to the right hand “selected properties” pane. Set an appropriate Text block or use a Text name if you have one, then click on Create to generate the text labels. They should all be placed on the bottom left hand corner of the titleblock.



Using the Edit – Move command (or hover right click – Move) re-position each property into the appropriate cell on your Titleblock. By default, the properties are added to the Manufacturing / Properties subclass but once they are available in PCB editor they can be copied to any subclass you require.

Remember to include the new Titleblock subclasses in your artwork control form so that they can be included in any artworks / odb++ / IPC2581 data that you need to export.

			 Parallel Systems Ltd PO Box 4265 Bracknell Berks RG42 9RS
DATE	A	APPROVED	
REVISION HISTORY			
UNLESS OTHERWISE STATED DIMENSIONS ARE IN MM TOLERANCES ON: FRACTIONS + ANGLES +	SIGNATURES	DATE	Drawing Number ABC1234567890
	STEVE ADAMS	22/01/2021	Drawing Title MY DESIGN

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.