



# The Capture Constraint System

## The Capture Constraint System

The Capture Constraint System, provides access to the Constraint Manager to allow management of rules for a PCB Editor design from OrCAD Capture.

The purpose of this note is to provide an overview of the Capture Constraint System and not provide a detailed discussion of the application of the constraints. A detailed discussion is covered in the Constraint Manager Reference Guide available from the Cadence Help application in HTML form, or available for download from the Cadence Online Support site.

Since the schematic “knows nothing” about the physical PCB being designed, access to rules which rely on knowledge of the board cross section will not be available unless a template board, or a technology file, is loaded. Users may create their own templates and technology files, with any of the OrCAD, or Allegro, PCB Editor tools. Users of standalone OrCAD Capture can create a default board to use as a template, this is discussed in more detail later in this note.

## What rules are available

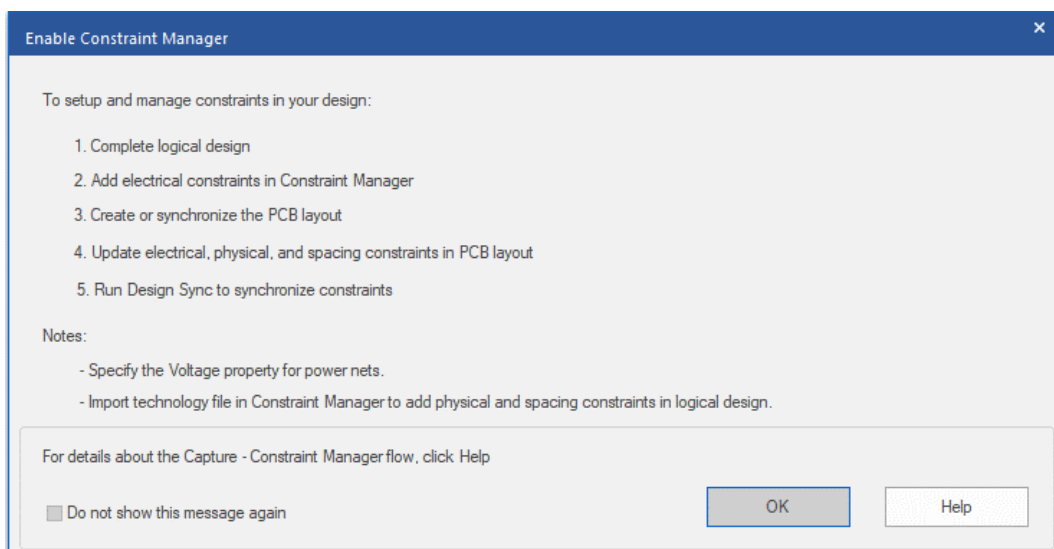
Rules shown in the Capture Constraint System reflect the license that OrCAD Capture was started from. With an OrCAD Capture, or an OrCAD PCB Designer Standard license, only rules applicable to the OrCAD PCB Designer Standard will be available. Rules applicable to OrCAD PCB Designer Professional will be available if an OrCAD Capture CIS, or OrCAD PCB Designer Professional license is used.

## Enabling the Constraint Manager flow

When OrCAD Capture, or OrCAD Capture CIS starts, the Constraint Manager icon will be displayed in the toolbar.



To enable the Constraint Manager flow, left-click the Constraint Manager icon.



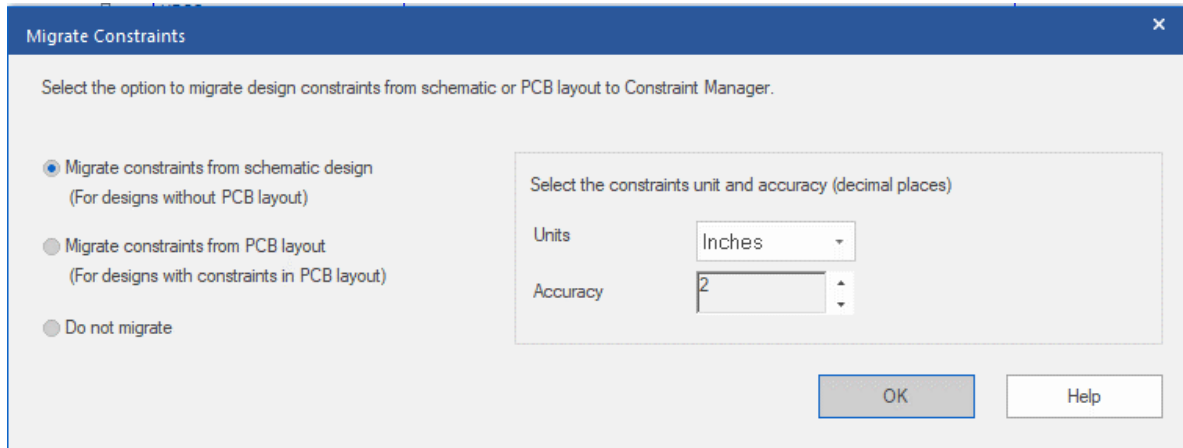
## The Capture Constraint System

Left-click>OK

The Migrate Constraints form will be shown.

### For designs without PCB layout

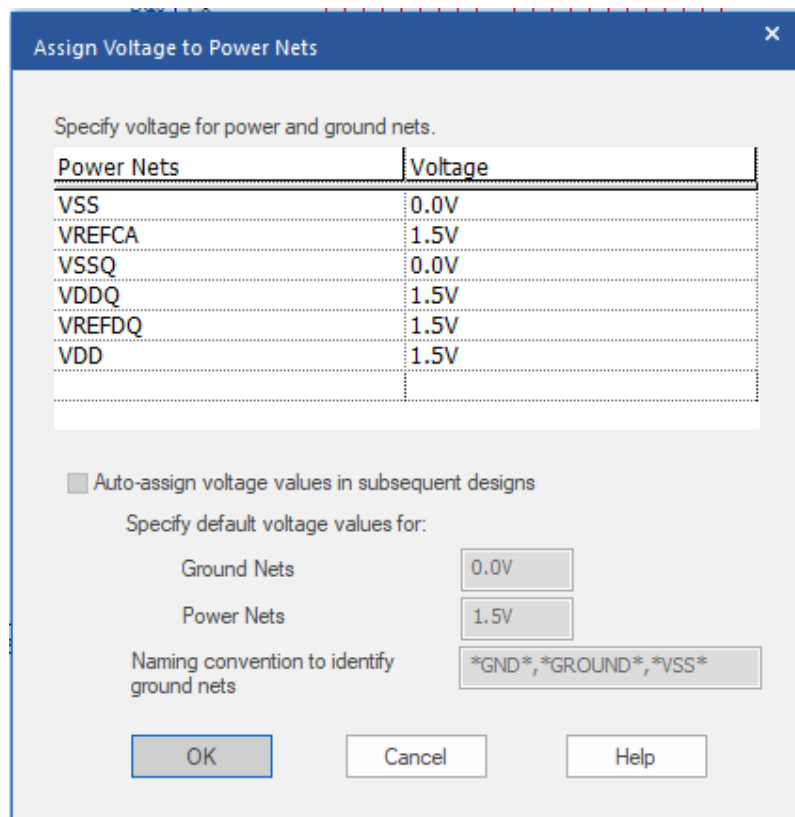
If there is no PCB, leave the “Migrate Constraints from Schematic Design” set and specify the Units and accuracy, “mm” and 2 decimal places would be typical.



The Migrate Constraints dialog box is shown. It has a title bar with a close button. The main text says "Select the option to migrate design constraints from schematic or PCB layout to Constraint Manager." There are three radio buttons: "Migrate constraints from schematic design (For designs without PCB layout)" which is selected, "Migrate constraints from PCB layout (For designs with constraints in PCB layout)", and "Do not migrate". To the right, there is a section titled "Select the constraints unit and accuracy (decimal places)" with a "Units" dropdown menu set to "Inches" and an "Accuracy" spinner box set to "2". At the bottom right are "OK" and "Help" buttons.

Left-click>OK to convert the project to “Constraint Manager Enabled”. (Refer to Note 1 page 7).

Then the Assign Voltage Nets to Power Nets form will open.



The Assign Voltage Nets to Power Nets dialog box is shown. It has a title bar with a close button. The main text says "Specify voltage for power and ground nets." Below this is a table with two columns: "Power Nets" and "Voltage".

Power Nets	Voltage
VSS	0.0V
VREFCA	1.5V
VSSQ	0.0V
VDDQ	1.5V
VREFDQ	1.5V
VDD	1.5V

Below the table is a checkbox "Auto-assign voltage values in subsequent designs" which is unchecked. Underneath is the text "Specify default voltage values for:" followed by two rows: "Ground Nets" with a text box containing "0.0V" and "Power Nets" with a text box containing "1.5V". Below that is the text "Naming convention to identify ground nets" with a text box containing "\*GND\*, \*GROUND\*, \*VSS\*". At the bottom are "OK", "Cancel", and "Help" buttons.

Assign appropriate voltages to the Voltage Nets. (Use the Auto Assign settings to automatically assign Voltages to Power Nets in other designs opened and Constraint Manager enabled in the same Capture session)

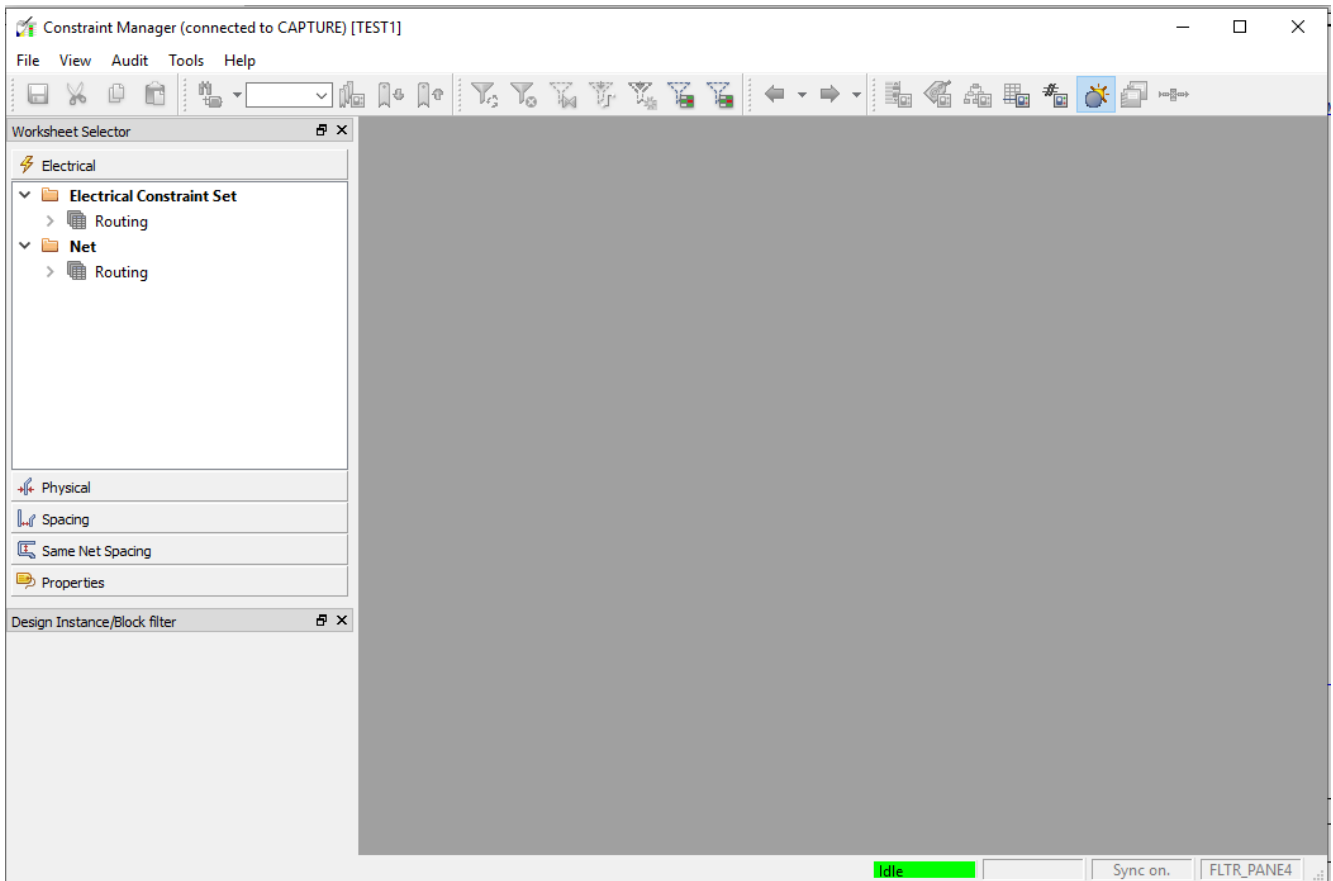
(The Assign Voltage Nets to Power Nets form can also be opened from the SI Analysis>Identify DC Nets menu)

## The Capture Constraint System

Left-click>OK

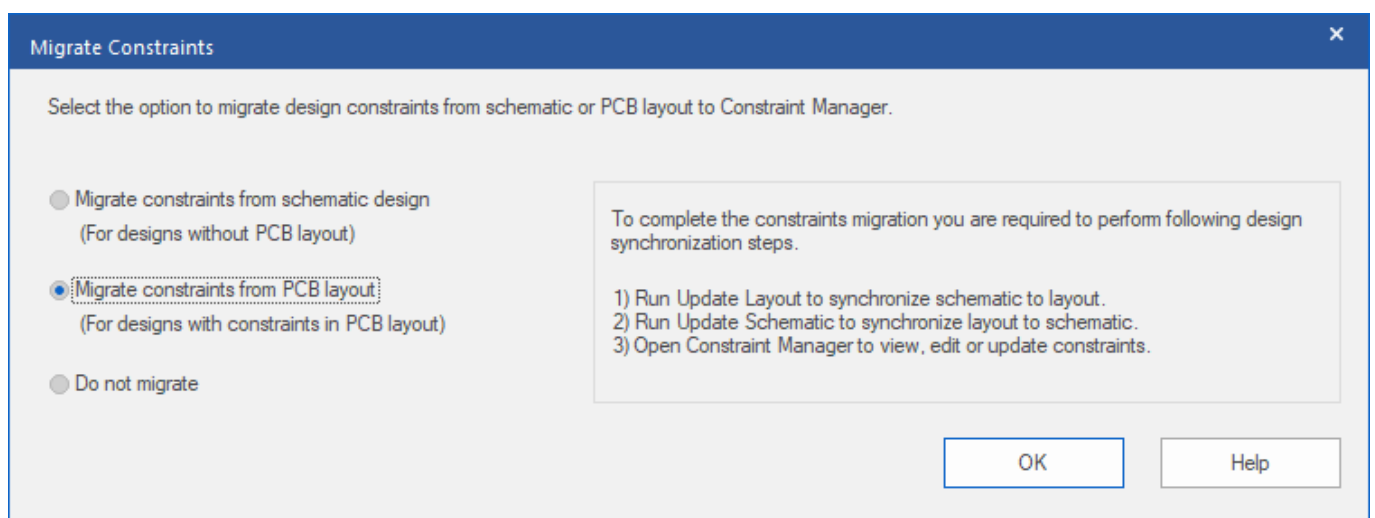
(A report is generated that shows the list of transferred constraints, constraint properties (if any), and warnings (if any). To review the report summary, use the Window>Session Log menu)

The Constraint Manager window opens.



### For designs with constraints in PCB layout

Select the “Migrate Constraints from PCB layout” option.



Left-click>OK and the steps from the form MUST be then performed, in order, to complete the Constraint Manager enabled flow migration:

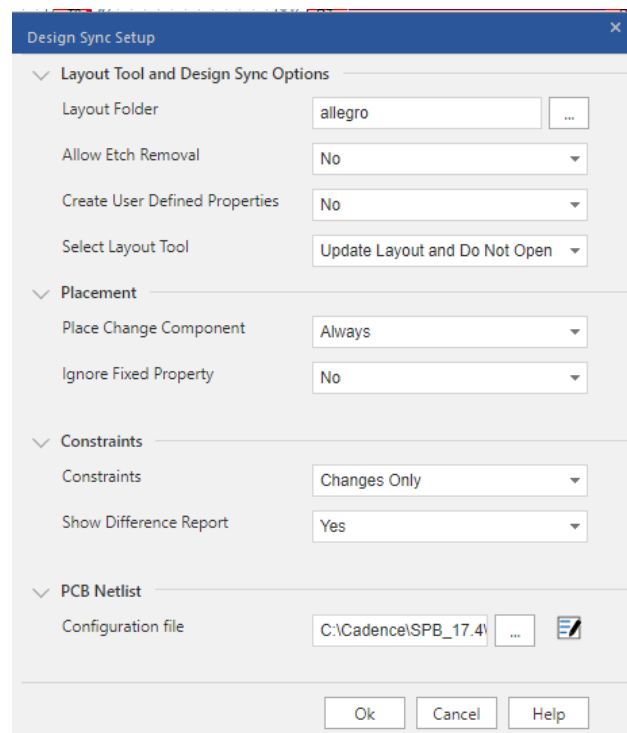
- 1) Use PCB>Update Layout and synchronise the current schematic onto the board file. (the board does not need to be opened)
- 2) Use PCB>Update Schematic and synchronise the board file back to the schematic (including Constraints).
- 3) Use the Constraint Manager to View, Edit or Update the Constraints.

(Note that only changes are transferred by this processing, so any existing board assigned Constraints will not be changed, since the Schematic has none until the migration processing has been completed)

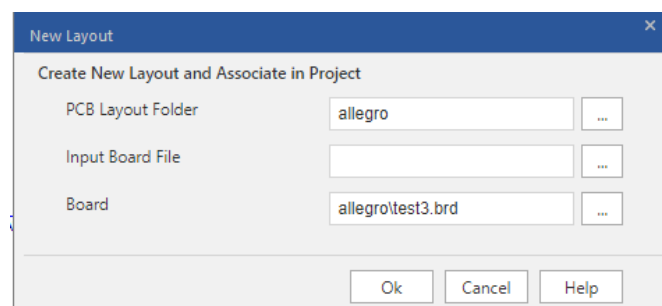
### Getting Physical and Spacing Constraints without a PCB

#### If you have a Capture license and no technology, file

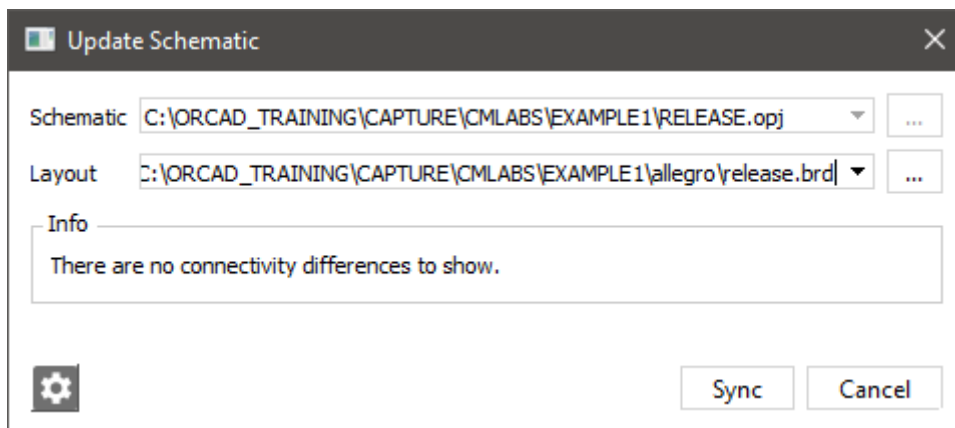
You can create a BRD file by running the Design Sync command into a “default” board file without opening the board file, then running Design Sync again from that BRD file back to the schematic. To do this run PCB>Design Sync Setup and change the Select Layout Tool from PCB Editor to Update Layout and Do Not Open.



Choose OK and then run PCB>New Layout



Leave the defaults and click OK. You will notice the board file is created and shown in the Project window under the Layout section. Now choose PCB>Update Schematic and left-click>Sync.



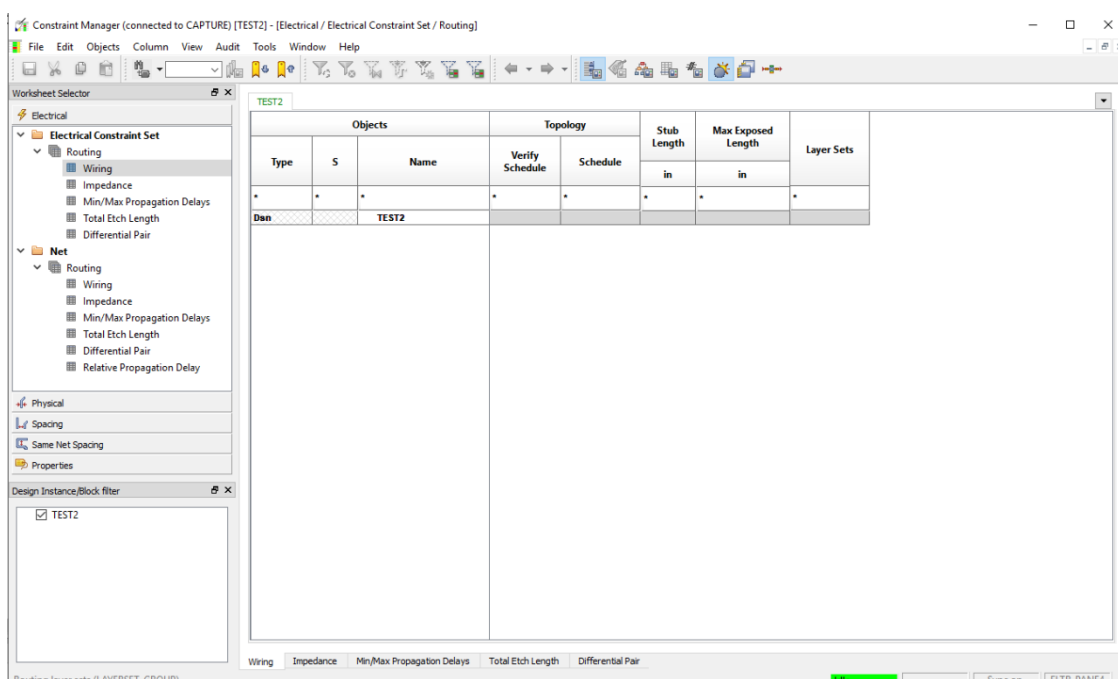
You can then use the Constraint Manager to View, Edit or Update the Constraints. (Note: PCB Footprints MUST be available for this to work successfully).

You can load a technology file by launching the Constraint Manager from within Capture and using the File>Import>Technology file entry, browse to the technology file and use the Overwrite option to load all the constraints from the technology file.

### The Constraint Manager window

The Constraint Manager has the Worksheet Selector at the left-hand side to allow the Worksheet to be selected from the Domain of interest. Domains are: Electrical, Physical, Spacing, Same Net Spacing and Properties. Note that, at this point, the Physical, Spacing and Same Net Spacing are not enabled, unless the design has been synchronised with a Board file.

Within the Domains are Workbooks and Worksheets which is where the values for the rules are entered. The following screenshot shows the Electrical>Electrical Constraint Set>Routing Workbook selected, and the Wiring Worksheet opened. Note that other Worksheets can be selected from within this Workbook by selecting the tabs at the bottom. The Electrical Constraint Set values are set in this Worksheet and then applied to Nets and Net Classes in the Electrical>Net>Routing>Wiring Worksheet.



## The Capture Constraint System

There are more columns in the Electrical>Net>Routing>Wiring Worksheet because the Net Worksheets will show the Actual values for the routing and Margins between the routed Nets and the Constraints set for them.

Objects			Referenced Electrical CSet	Topology			Stub Length			Layer Sets				
Type	S	Name		Verify Schedule	Schedule	Actual	Margin	Max in	Actual in	Margin in	Name	Actual	Margin	Length Ignored in
Net		TEST2												
Net		N01076												
Net		N01078												
Net		N01080												
Net		N01082												
Net		N01084												
Net		N01086												
Net		N01088												
Net		N01090												
Net		N01092												
Net		N01094												
Net		N01096												
Net		N01098												
Net		N01100												
Net		N01102												
Net		N01104												
Net		N01106												
Net		N01108												
Net		N01110												
Net		N01112												
Net		N01114												

## The Constraint Domains

The Physical domain sets the rules for the routing for the design. The routed width, via selection, differential pair coupling parameters, blind and buried via stagger, “T”s and routing on layers.

The Spacing domain sets the rules for spacing between net objects in the design where nets are different. Line routed lines, vias, pins and so on. There are many settings since each object to object combination must be catered for.

The Same Net Spacing sets the rules for spacing between net objects where nets are the same.

The Electrical domain sets the rules for high speed designs where the routing needs to be controlled, or matched to ensure integrity of signal timing, for example.

The Properties domain will list properties for nets and components. The Voltage property that is set at the constraint manager launch is one example.

## Creating Constraint Sets

Constraint Sets can be created in the Electrical, Spacing or Same Net Spacing Domains. Left-click in the Domain, then the Constraint Set type. For the Electrical Constraint Set, left-click in the Design Name entry and right-click>Create Electrical CSet. For the Physical, Spacing and Same Net Spacing Constraint Sets, left-click in an existing Constraint Set, Default for example, and right-click>Create Physical CSet, Spacing CSet, or Same Net Spacing CSet, depending upon which domain is active. Once the new Constraint Set is added, set the values.

## Applying constraints

Typically, rules are set within a Constraint Set for the domain, Nets that require the same Constraint Set are grouped into Net Classes and then the required Constraint Set is applied to the Net Class. If required, Constraint Sets can be applied to individual Nets and values can also be applied to individual Nets using the cells for the values.

Constraints can be applied through the Constraint Manager in Capture, the design synced through to PCB Editor, modified in PCB Editor and then design synced back to Capture, keeping the schematic and the board tightly coupled.

### XNet automated creation

Once the Constraint Manager flow is enabled, XNets, eXtended Nets where a “terminator” combines two Nets, will be automatically created for any two pin components with a CLASS Property Value of DISCRETE, effectively assuming that they are “coupling” individual Nets into an eXtended Net. To avoid the creation of eXtended Nets where not intended, two pin components with a CLASS Property Value of DISCRETE will require the “NO\_XNET\_CONNECTION” Property with a Value of YES to be applied to the Part.

Note that only numerical values are recognised for the Value property to correctly create XNets. If the Value property isn't numerical, for example the Value is “22R 1%”, the XNET\_PINS property will be required on the Pins of the part to create the XNet, use the same value of XNET\_PINS assigned to the pair of Pins to create the XNet. Using a two-pin resistor as an example, the XNET\_PINS property would be assigned to both Pins with the same value on each of them.

### Notes:

- 1) Back up your design before enabling Constraint Manager because once enabled, you cannot change it to a non-Constraint Manager-enabled design. (A copy of the DSN file is saved as <design\_name>.dsn.3sav when the Capture Constraint Manager is invoked)
- 2) As far as practicable, modify the constraints after completing the design.
- 3) Specify the Voltage property for power nets.
- 4) Import the technology file in Constraint Manager to add physical and spacing constraints in the logical design.
- 5) For distributed design environment, manage constraints in a single environment. It is recommended that constraints are added either in Capture or in PCB layout before running the schematic to PCB and PCB to schematic flows.
- 6) If a board is associated with a design, to migrate constraints, ensure that you use the Import from physical layout option in the Migrate Constraints dialog box.
- 7) Do not rename a Constraint Manager-enabled non-root-schematic design, else the all the constraints will be lost.
- 8) If the Constraint Manager-enabled design is opened in a lower release version of Capture and edited, the constraints will be lost.

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.