

# **Constraint Manager with OrCAD® X Capture**

**Product Version 23.1  
September 2023**

© 2023 Cadence Design Systems, Inc. All rights reserved.

Portions © Apache Software Foundation, Sun Microsystems, Free Software Foundation, Inc., Regents of the University of California, Massachusetts Institute of Technology, University of Florida. Used by permission. Printed in the United States of America.

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Product PSpice contains technology licensed from, and copyrighted by: Apache Software Foundation, 1901 Munsey Drive Forest Hill, MD 21050, USA © 2000-2005, Apache Software Foundation. Sun Microsystems, 4150 Network Circle, Santa Clara, CA 95054 USA © 1994-2007, Sun Microsystems, Inc. Free Software Foundation, 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA © 1989, 1991, Free Software Foundation, Inc. Regents of the University of California, Sun Microsystems, Inc., Scriptics Corporation, © 2001, Regents of the University of California. Daniel Stenberg, © 1996 - 2006, Daniel Stenberg. UMFPACK © 2005, Timothy A. Davis, University of Florida, (davis@cise.ulf.edu). Ken Martin, Will Schroeder, Bill Lorensen © 1993-2002, Ken Martin, Will Schroeder, Bill Lorensen. Massachusetts Institute of Technology, 77 Massachusetts Avenue, Cambridge, Massachusetts, USA © 2003, the Board of Trustees of Massachusetts Institute of Technology. All rights reserved.

**Trademarks:** Trademarks and service marks of Cadence Design Systems, Inc. contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 800.862.4522.

Open SystemC, Open SystemC Initiative, OSCI, SystemC, and SystemC Initiative are trademarks or registered trademarks of Open SystemC Initiative, Inc. in the United States and other countries and are used with permission.

All other trademarks are the property of their respective holders.

**Restricted Permission:** This publication is protected by copyright law and international treaties and contains trade secrets and proprietary information owned by Cadence. Unauthorized reproduction or distribution of this publication, or any portion of it, may result in civil and criminal penalties. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. Unless otherwise agreed to by Cadence in writing, this statement grants Cadence customers permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used only in accordance with a written agreement between Cadence and its customer.
2. The publication may not be modified in any way.
3. Any authorized copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement.
4. The information contained in this document cannot be used in the development of like products or software, whether for internal or external use, and shall not be used for the benefit of any other party, whether or not for consideration.

**Disclaimer:** Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information.

**Restricted Rights:** Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR252.227-7013 et seq. or its successor.

---

# Contents

---

<u>Constraint Manager with OrCAD X Capture</u>	5
<u>Managing Constraints in Capture</u>	5
<u>Constraint Manager-Enabled Mode in Capture</u>	6
<u>Migrating Constraints</u>	7
<u>Migrating Constraints from Schematic Design</u>	7
<u>Importing Constraints from Physical Layout</u>	12
<u>Constraint Objects</u>	18
<u>Managing Design Objects</u>	20
<u>Working with Schematic Objects and Operations</u>	20
<u>Finding Constraints</u>	24
<u>Cross-Probing of Design Objects</u>	27
<u>Renaming Nets</u>	29
<u>Modifying Voltage for Nets</u>	30
<u>Working with Constraint Objects</u>	31
<u>Managing XNets</u>	31
<u>Performing Signal Analysis in Constraint Manager-Enabled Design</u>	38
<u>Working with Electrical CSets</u>	43
<u>Modes for Processing Constraints: Overwrite and Changes Only</u>	45
<u>Updating Property Definition in Constraint Manager</u>	46
<u>Handling User-Defined Properties</u>	48
<u>Known Good Practices</u>	50

## Constraint Manager with OrCAD X Capture

---

---

# Constraint Manager with OrCAD X Capture

---

A constraint is a user-defined requirement applied to a net or pin-pair in a design. Electrical constraints (ECS) govern the electrical behavior of a net or a pin-pair in a design. For example, you can capture a constraint to define the propagation delay and relative propagation delay for a driver-receiver pin-pair in your design.

To capture constraints, Cadence provides a tool named Constraint Manager. You can use Constraint Manager with OrCAD X Capture (referred to as Capture elsewhere) to define and manage electrical constraints as you implement the logic.

Depending upon license used for Capture, Constraint manager provides different level of functionality for managing constraints.

For more information on Constraint Manager, see:

- *Allegro Constraint Manager User Guide*
- *Allegro Constraint Manager Reference*
- *Allegro Platform Constraints Reference*

## Managing Constraints in Capture

To specify constraints, you can use the existing mode to manage a subset of constraints using the property editor.

The Constraint Manager-enabled mode is optional. You can enable this mode at any phase of the PCB design flow and on any of the following:

- ☐ New schematic design
- ☐ Existing schematic design
- ☐ Existing schematic design with PCB layout

## Constraint Manager-Enabled Mode in Capture

Before you use Constraint Manager to manage constraints in your design, you need to understand the following:

- What is Capture-Constraint Manager Flow?
- How to enable Constraint Manager in Capture

### What is Capture-Constraint Manager Flow?

In the Capture-Constraint Manager flow, Constraint Manager is used to define, manage, and assign constraints on the Capture schematic.

The recommended sequence of tasks to manage design constraints using Constraint Manager is:

1. Complete the logical design.
2. Add electrical constraints in Constraint Manager.
3. Create or update the PCB layout.
4. Update electrical, physical, and spacing constraints in the PCB layout.
5. Run the *PCB – Update Layout* and *PCB – Update Schematic* commands to synchronize constraints.

You can now specify the physical and spacing constraints in Constraint Manager in Capture.

**Note:** You should avoid simultaneous editing of design objects, such as nets with while assigning and modifying constraints.

### How to enable Constraint Manager in Capture

To use the Capture-Constraint Manager flow, you need to enable Constraint Manager. To do so:

1. Back up your design.



Ensure that the schematic design and PCB layout are synchronized.

2. Select *PCB – Constraint Manager* or click the *Constraint Manager* icon in the PCB toolbar.
3. Use the options in the *Migrate Constraints* dialog box to transfer constraints as required.

### **Important**

After you enable a design for Constraint Manager, it cannot be changed back to a non-Constraint Manager-enabled design.

## Migrating Constraints

You can enable Constraint Manager on a Capture design with migration of constraints using any one of the following two options:

- Migrate constraints from schematic design.

This can be a schematic:

- ☐ with constraints
- ☐ without any constraints

- Import constraints from physical layout.

### Migrating Constraints from Schematic Design

To transfer constraints from a new or existing design, do the following:

1. Open Capture.
2. Create a new design.

Or

Open an existing design.

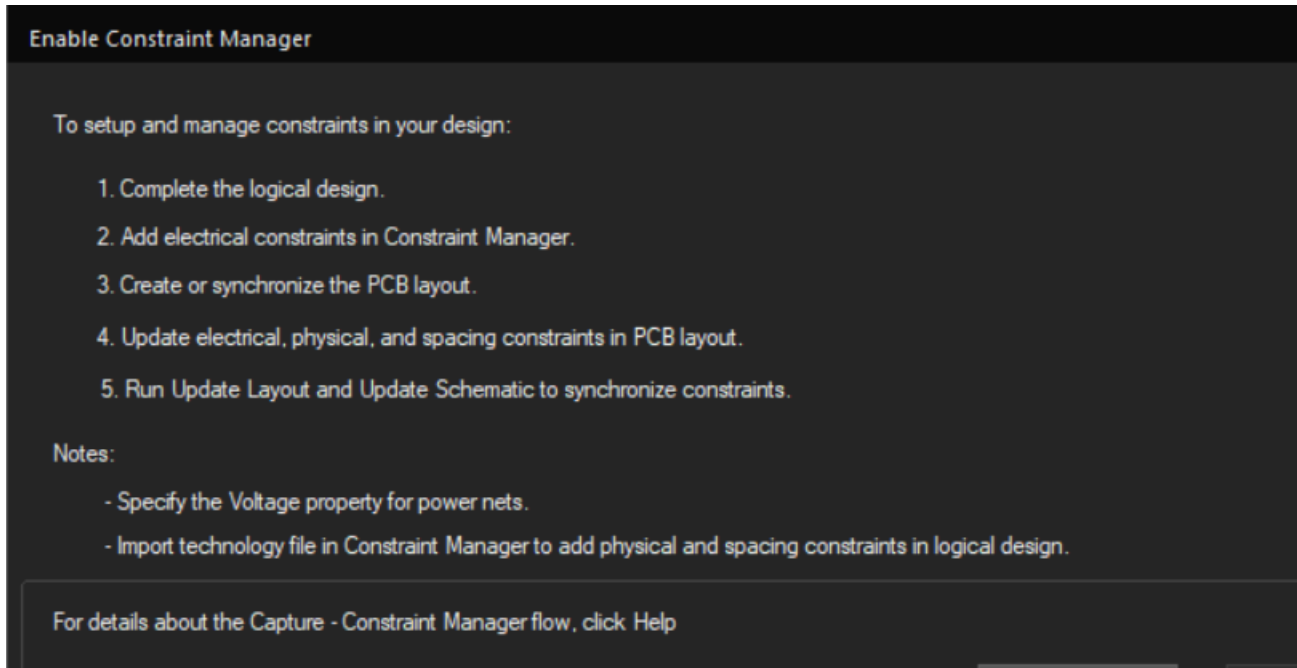
3. Select *PCB – Constraint Manager* or click the *Constraint Manager* icon in the Capture toolbar.

## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

---

An information window appears to explain the Capture-Constraint Manager flow.

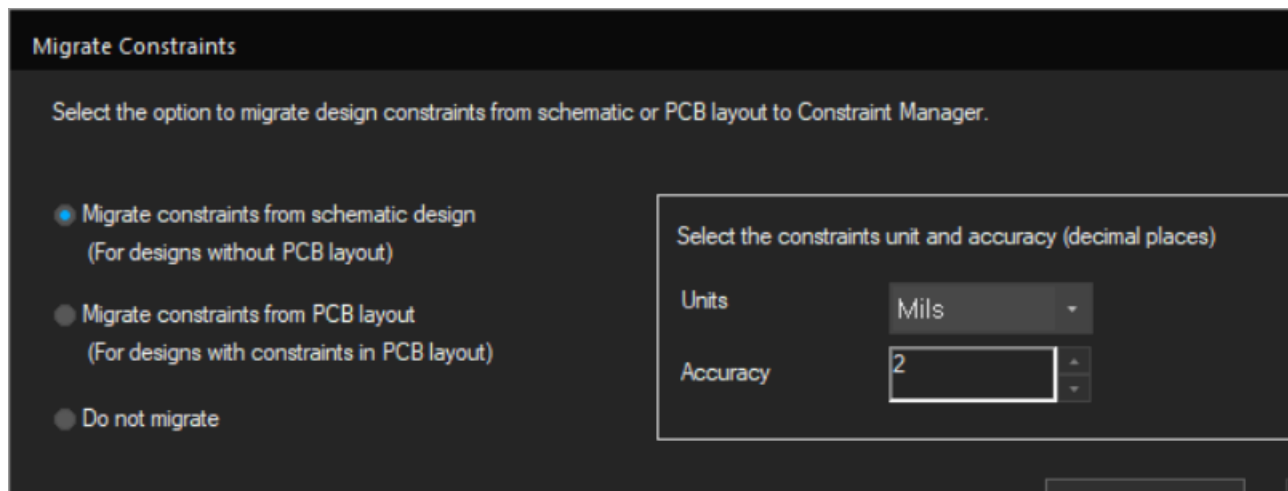


4. Click *OK*.

The *Migrate Constraints* dialog box appears.

5. Select *Migrate constraints from schematic design*.

6. Specify the unit to be used for physical and spacing constraints in the Constraint Manager interface.





## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

---

7. Select the *Do not show the message again* check box to stop this message from appearing each time Constraint Manager is invoked

8. Click OK.

The *Assign Voltage to Power Nets* dialog box opens.

9. To identify the power and ground nets, specify the voltage values for them.

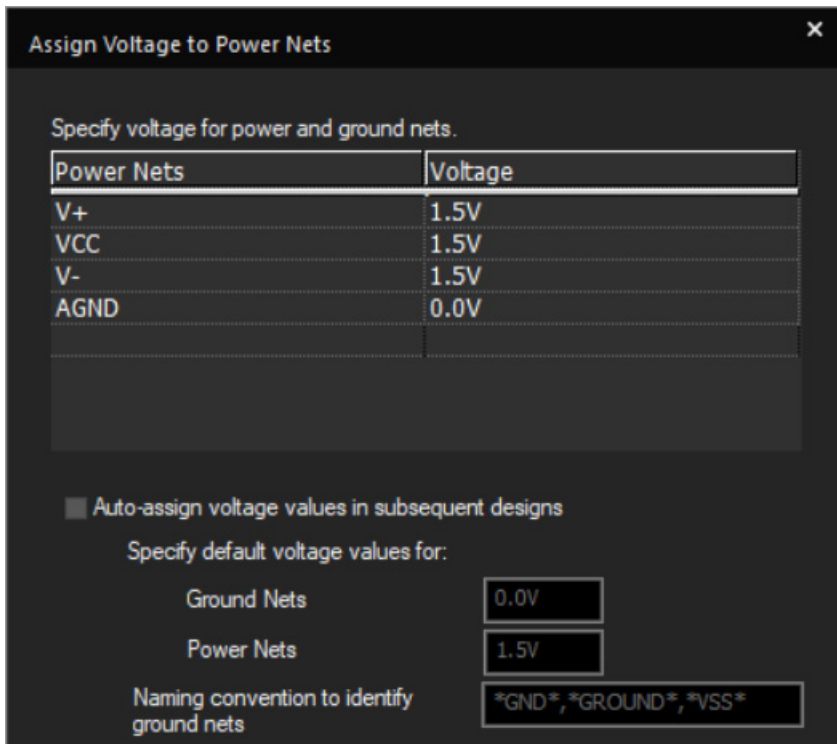
10. Select the *Auto-assign voltage values in subsequent designs* check box to automatically assign voltages to power nets in other designs opened and Constraint Manager-enabled in the same Capture session.

11. You can also specify the:

- ☐ default voltage values for ground and power nets
- ☐ naming convention to identify ground nets

### Important

The *Power Nets* shown in this window are the nets connected to power pins.



Power Nets	Voltage
V+	1.5V
VCC	1.5V
V-	1.5V
AGND	0.0V

☐ Auto-assign voltage values in subsequent designs

Specify default voltage values for:

Ground Nets: 0.0V

Power Nets: 1.5V

Naming convention to identify ground nets: \*GND\*, \*GROUND\*, \*VSS\*

You can open this dialog box from *SI Analysis – Identify DC Nets*.

**Note:** After enabling Constraint Manager, you can modify voltage values for nets directly

## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

in the Constraint Manager window. For details, see [Modifying Voltage for Nets](#).

#### 12. Click OK.

The following tasks are completed:

- ❑ Constraints (if any) in the schematic design are transferred to Constraint Manager.
- ❑ Constraints Manager opens. Following screen shot is for a design that has constraints.

Type	S	Name	Referenced Electrical CSet	Single-line Impedance	
				Target Ohm	Tolerance Ohm
FLTR	*	*	*	*	*
Dsn		DESIGNDB2			
DPr		DP1			
Net		CLOCKD			
Net		DAC_CLOCK			
Net		HIGH			
XNet		IOUTA			
Net		LOW			
Net		MC0			
Net		MC1			
Net		MC2			
Net		MC3			
Net		MC4			
Net		MC5			
Net		MC6			
Net		MC7			
Net		MC8			
Net		MC9			
Net		MC10			
Net		MC11			
Net		MC12			
Net		MC13			
Net		MC14			
Net		MC15			
XNet		N06347			
Net		N08896			
Net		N09022		50	2 %
Net		N10683		50	2 %
Net		N20773			
Net		N24309			
Net		N16619156			

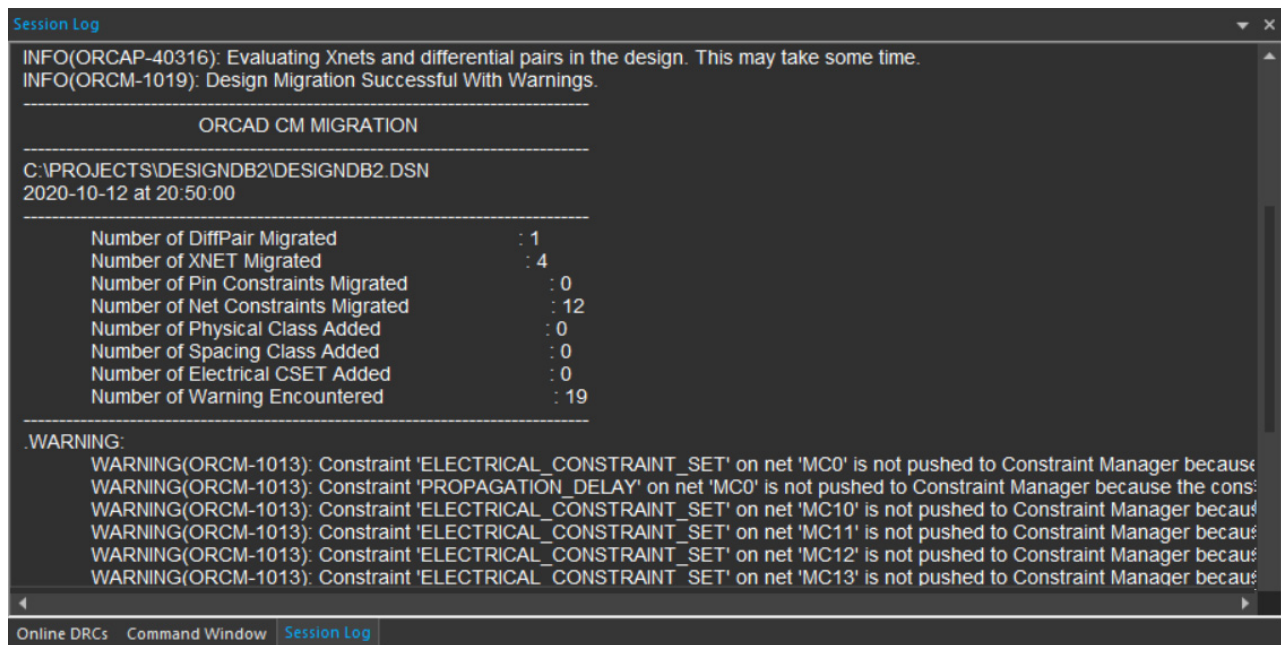
## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

**Note:** It is recommended to run annotation in Capture to see the correct data in Constraint Manager.

- ❑ A report is generated that shows the list of transferred constraints, constraint properties (if any), and warnings (if any).

To review the report summary, see the session log file from the *Windows – Session Log* menu command or the *View – Session Log* menu command.



```
Session Log
INFO(ORCAP-40316): Evaluating Xnets and differential pairs in the design. This may take some time.
INFO(ORCM-1019): Design Migration Successful With Warnings.

-----
ORCAD CM MIGRATION
-----
C:\PROJECTS\DESIGNDB2\DESIGNDB2.DSN
2020-10-12 at 20:50:00

-----
Number of DiffPair Migrated      : 1
Number of XNET Migrated         : 4
Number of Pin Constraints Migrated : 0
Number of Net Constraints Migrated : 12
Number of Physical Class Added   : 0
Number of Spacing Class Added    : 0
Number of Electrical CSET Added  : 0
Number of Warning Encountered    : 19
-----

WARNING:
WARNING(ORCM-1013): Constraint 'ELECTRICAL_CONSTRAINT_SET' on net 'MC0' is not pushed to Constraint Manager because
WARNING(ORCM-1013): Constraint 'PROPAGATION_DELAY' on net 'MC0' is not pushed to Constraint Manager because the cons
WARNING(ORCM-1013): Constraint 'ELECTRICAL_CONSTRAINT_SET' on net 'MC10' is not pushed to Constraint Manager becau
WARNING(ORCM-1013): Constraint 'ELECTRICAL_CONSTRAINT_SET' on net 'MC11' is not pushed to Constraint Manager becau
WARNING(ORCM-1013): Constraint 'ELECTRICAL_CONSTRAINT_SET' on net 'MC12' is not pushed to Constraint Manager becau
WARNING(ORCM-1013): Constraint 'ELECTRICAL_CONSTRAINT_SET' on net 'MC13' is not pushed to Constraint Manager becau

Online DRCs  Command Window  Session Log
```


### Important

If you select the *Migrate constraints from schematic design* option for a design that has a physical layout file, the design is enabled for Constraint Manager, and the Update Layout window opens a message to indicate that the schematic design and the physical layout are not synchronized. You are also prompted to click the *Sync* button to synchronize the layout with the schematic. However, this step is not mandatory.

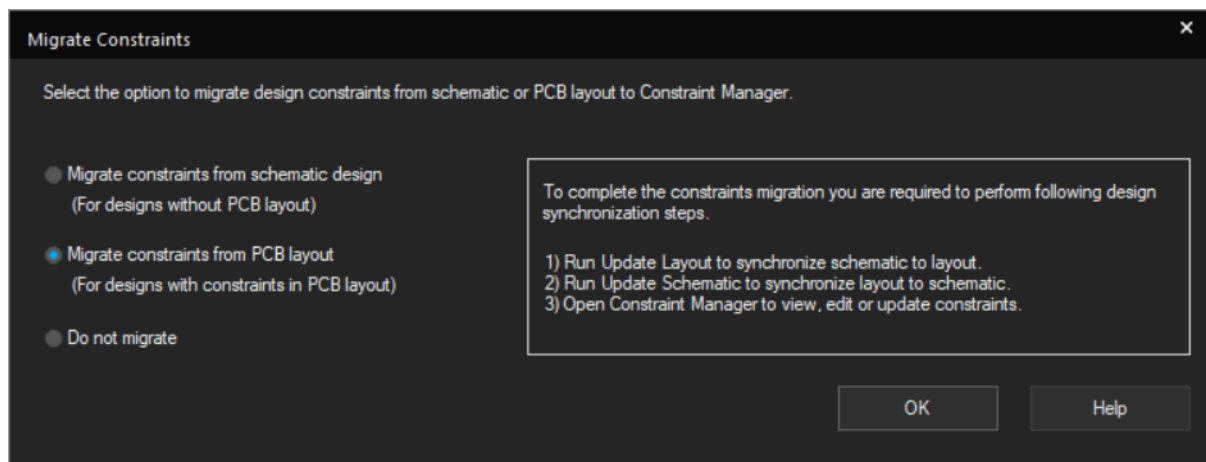
## Importing Constraints from Physical Layout

To transfer constraints from a physical layout, do the following:

1. Before you enable Constraint Manager to import constraints from physical layout, ensure that you have already synchronized the design and board in the non-Constraint Manager-enabled mode.

2. Click the Constraint Manager icon (  ) in the Capture toolbar.

The *Migrate Constraints* dialog box appears.



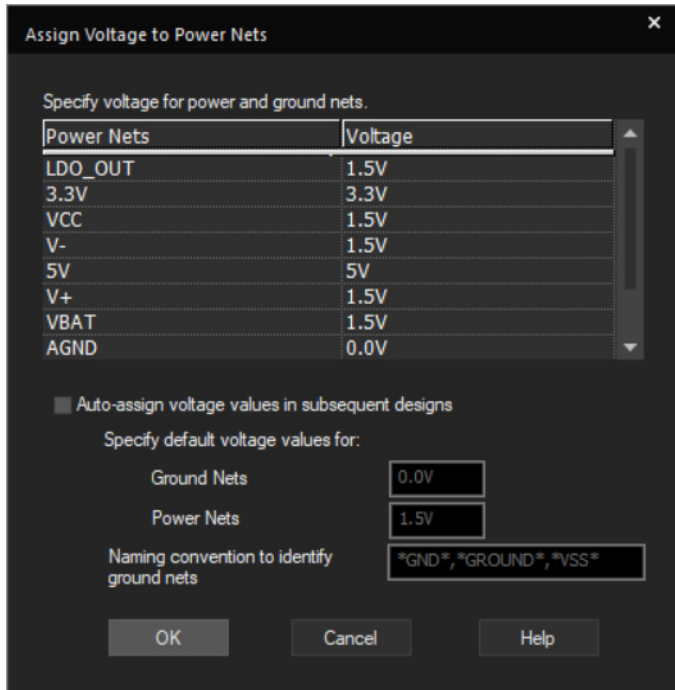
3. Select *Migrate constraints from PCB layout*.
4. Click *OK*.


## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

---

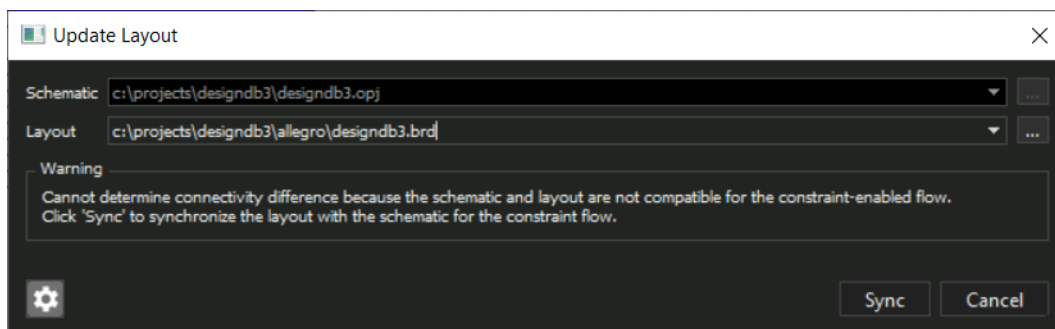
The Assign Voltage to Power Nets window opens.



5. Modify the voltage for ground and power nets.
6. Click *OK*.
7. Choose *PCB – Update Layout* or click (  ).

The *Update Layout* dialog box opens.

The following message appears if the layout file is not yet Constraint Manager-enabled.



8. Click *Sync* to synchronize layout with schematic.

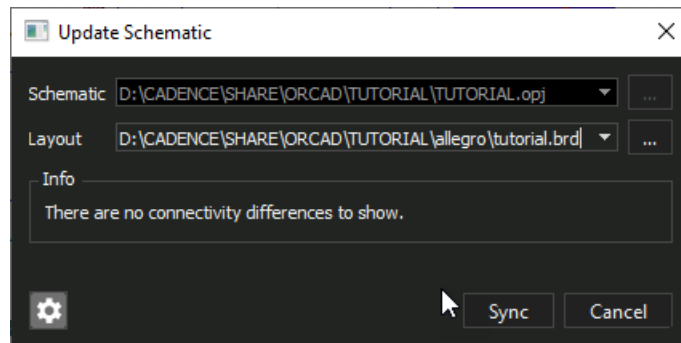
## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

After successful update of the layout file, the Constraint Difference Report window appears indicating the status of the transferred constraints.

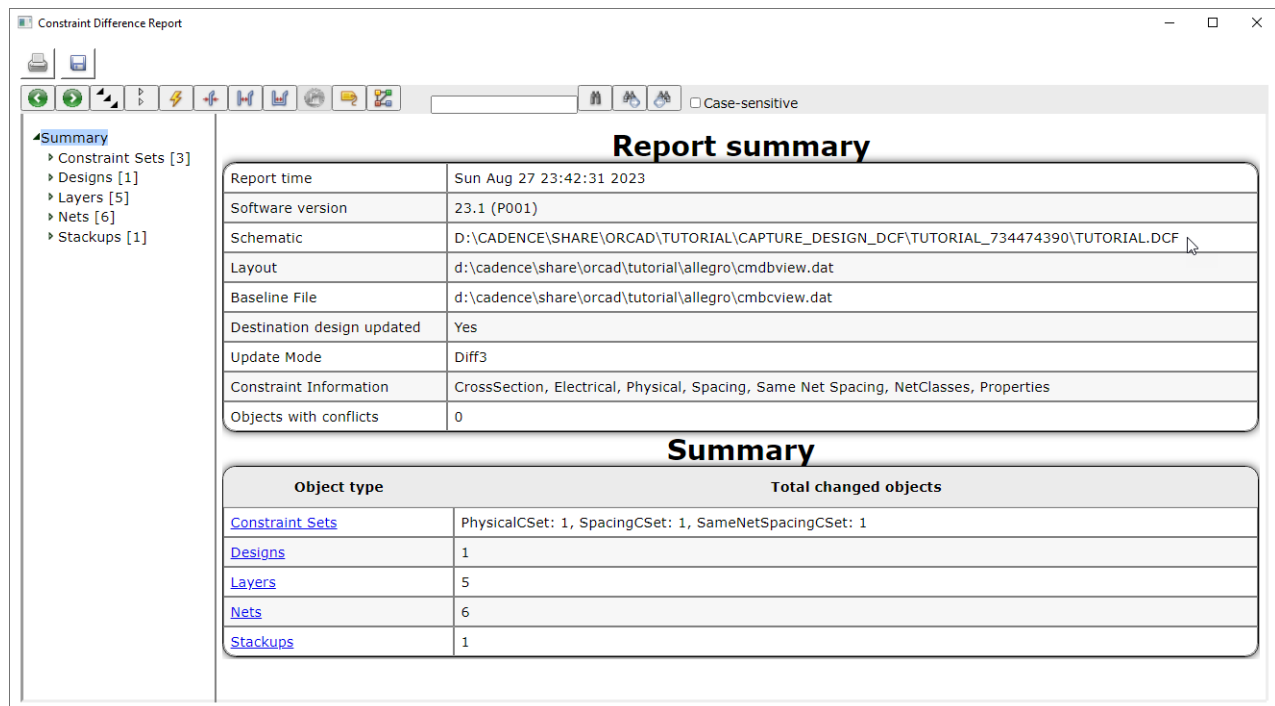
9. Select *PCB – Update Schematic* or click (  ).


The Update Schematic dialog box opens.



10. Click *Sync* to synchronize schematic with layout.

The Report Summary window appears indicating the status of the transferred constraints.



11. Click the Constraint Manager icon (  ) to open Constraint Manager.

## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

You can see the constraints in Constraint Manager.

Constraint Manager (connected to CAPTURE) [DESIGNDB3] - [Electrical / Net / Routing]

File Edit Objects Column View Audit Tools Window Help

Worksheet Selector

DESIGNDB3

Type	S	Name	Referenced Electrical CSet	Single-line Impedance	
				Target	Tolerance
				Ohm	Ohm
Dsn	*	DESIGNDB3	*	*	*
Bus		DATA_OUT(8)			
Bus		MC[0..15](16)			
DPr		DP_IOUT			
DPr		DP_ISO_L			
DPr		DP_J1850BUS			
DPr		DP_J1850BUS_TX			
DPr		DP1			
XNet		BATT_V	ECS1	30	10 Ohm
Net		CAN_HI	ECS1	30	10 Ohm
Net		CAN_LO	ECS1	30	10 Ohm
Net		CAN_RX	ECS1	30	10 Ohm
Net		CAN_TX	ECS1	30	10 Ohm
Net		CLOCKD	ECS1	30	10 Ohm
Net		DAC_CLOCK			
XNet		EN1	ECS1	30	10 Ohm
XNet		EN2	ECS1	30	10 Ohm
XNet		EN3	ECS1	30	10 Ohm
Net		HIGH	ECS1	30	10 Ohm
Net		HOST_PRESENT	ECS1	30	10 Ohm
Net		ISO_RX	ECS1	30	10 Ohm
XNet		K-LINE			
Net		L-LINE	ECS1	30	10 Ohm
Net		LOW	ECS1	30	10 Ohm
Net		MC0			
Net		MC1			
Net		MC2			
Net		MC3			
Net		MC4			
Net		MC5			

Wiring Impedance Min/Max Propagation Delays Total Etch Length Differential Pair Relat

Defines the impedance tolerance (absolute or percentage) of a net (IMPEDANCE\_1: 10 Ohm) Sync on XNET

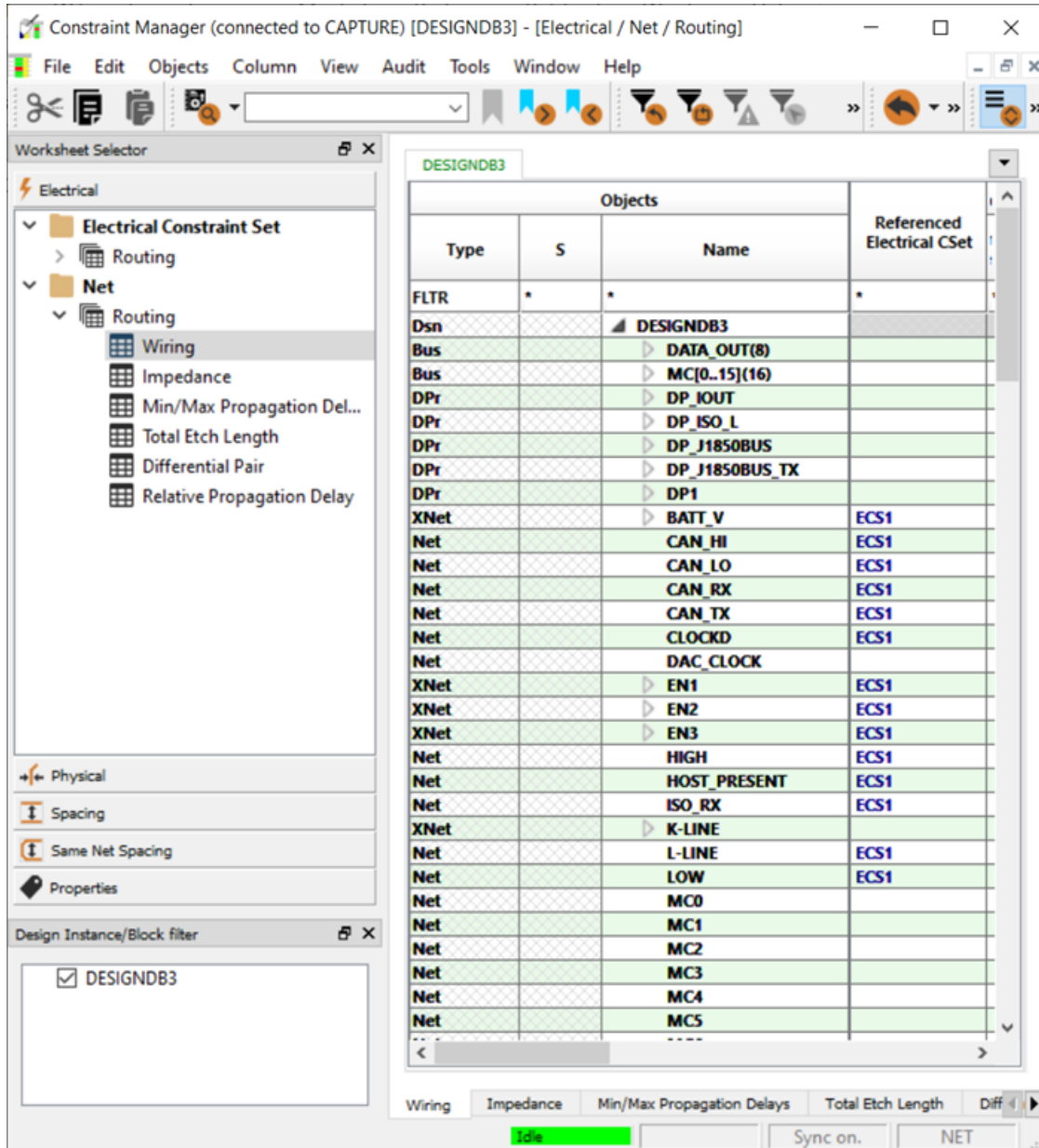
**Note:** After a successful run of the *Update Schematic* operation, the constraint migration process completes. Note that the nets with discrete components (even without



## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

the SIGNAL\_MODEL property assigned) are converted into XNets.



You might see differences depending on the file you used to capture constraints.

*Technology* files (TCFX) are design independent, which means they do not contain connectivity data, such as nets, differential pairs, or buses. These files are used to initialize designs to baseline their technology. This will not backannotate any design specific data such as net groups created in PCB Editor.



## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

---

The *Dictionary and Constraints* files (DCFX) contain all the constraint data for a specific design. These files are used to create a backup of the current constraint data for a design.

12. Review and modify constraints as required.
13. Save the design.

## Constraint Objects

Along with constraints, you can also manage the following constraint objects in Constraint Manager:

- **Extended Nets or XNets**

A net represents an electrical connection from one pin to another pin (or pins) on the same device or on a different device.

If the path of a net traverses a passive, discrete device (resistor, inductor, or capacitor), then each net segment is represented by an individual net entity in the board database. The constraint system, however, interprets these net segments as a contiguous extended net, or an XNet. An XNet can also traverse connectors and cables in a multi-board configuration. Capture automatically creates XNet for nets associated with 2-pin passive components.

- **Pin Pairs**

A pin pair represents a pair of logically connected pins, often a driver-receiver connection. Pin Pairs may not be directly connected but they must exist on the same net or XNet. You use pin pairs to capture specific pin-to-pin constraints for a net or an XNet. You can also use pin pairs to capture generic pin-to-pin constraints for CSets. Generic pin pairs are used to automatically define net- or XNet-specific pin pairs when the CSet is referenced.

- **Differential Pairs**

A differential pair represents a pair of nets or XNets that have to be routed in a way that the signals passing through them are opposite in sign with respect to the same reference. This ensures that any electromagnetic noise in the circuit is canceled out.

You can create a differential pair in Constraint Manager. In Constraint manager, you can create differential pair manually or can specify setup option for auto-creation of differential pairs.

- **Match Groups**

A Match Group is a collection of nets, XNets, or pin pairs which must all match (in delay or length) or be relative to a specific target within the group.

- **Net Group**

A Net Group is a collection of various net (signal) objects. Different types of net objects, such as nets, buses, and net groups, can be members of a Net Group.

- **Net Class**

## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

---

A Net Class constraint object lets you group net objects that share common characteristics and require a similar constraint requirement.

#### ■ Constraint Set (CSet)

A CSet is a named, reusable collection of constraint values.

Constraint Manager organizes constraints and CSets into the following domains:

- ☐ Electrical
- ☐ Physical
- ☐ Spacing
- ☐ Same Net Spacing

You can assign the appropriate constraint set to objects in your design, changing references (or re-defining the currently assigned constraint set) as your design requirements change. A constraint set can be referenced by any number of objects in your design. CSets are not supported in the design domain.

## Managing Design Objects

This section explains the changes that you will observe while using schematic design objects and related operations.

- [Working with Schematic Objects and Operations](#)
- [Finding Constraints](#)
- [Cross-Probing of Design Objects](#)
- [Renaming Nets](#)
- [Modifying Voltage for Nets](#)

## Working with Schematic Objects and Operations

Following is the list of changes you will observe when you enable Constraint Manager and use the Capture-Constraint Manager flow:

- **Property Editing**

All constraints and constraint-related properties for net objects are managed in Constraint Manager only.

The component properties for parts and pins are to be managed only from Property Editor in Capture.

- **Assigning Voltage**

You need to assign voltage to power nets. You can also set this function on auto-assign mode for all subsequent designs being opened and enabled for Constraint Manager in the same session.

- **Handling Buses and Net Groups**

Buses and net groups if created in the Capture schematic are not recognized as bus or net groups in the Constraint Manager user interface. These appear as individual nets.

Buses and net groups (if any) available in the layout editor and in Constraint Manager invoked from the layout editor are available in the Constraint Manager user interface for a Constraint Manager-enabled design.

- In a non-Constraint Manager-enabled design, the PCB netlist files are created as `pst*.dat` files. In a Constraint Manager-enabled design, these files appear as zipped files (`*.cdsz`) in the project folder.

## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

---

- If you copy the schematic design from a Constraint Manager-enabled project to a new design, the constraints are not copied.
- After enabling Constraint Manager in Capture, the following menu options are disabled in Capture. Their corresponding functions are now done only in Constraint Manager:
  - ☐ All menu options related to Electrical CSets
  - ☐ View XNet Signals
  - ☐ Create Differential Pair

**Note:** In the legacy design mode, the above-listed operations work in Capture.

- The undo operation is limited to schematic operations only. The undo operation will not undo any constraint-related operation.

Here is an example that suggests the changes you will observe in the undo operation.

#### **Pin Pairs after Replacing or Deleting Parts**

## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

If a pin-pair is associated with a net and you delete or replace the component attached to it, the pin-pair definition disappears.

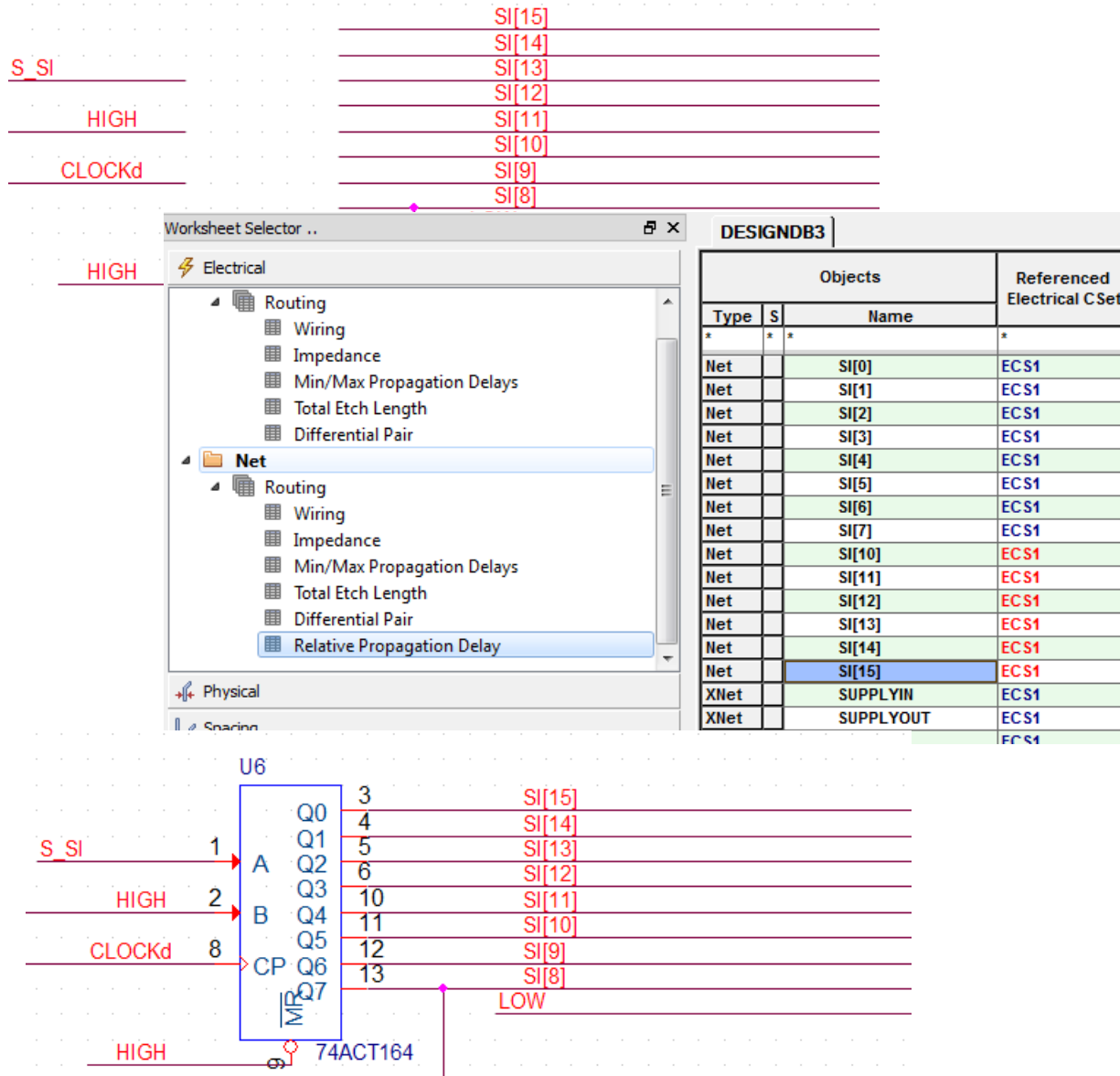
The screenshot displays a schematic diagram at the top and the Constraint Manager table below it. The schematic shows a component U6 (74S377) with pins 1, 2, and 8 connected to a net labeled S\_CS. The pins are connected to Q0, Q1, Q2, Q3, Q4, and Q5. The net S\_CS is connected to the CP pin of the 74S377 component. The Constraint Manager table lists the nets and their associated pin-pairs.

XNet		RX-STAT	ECS1
Net		SI[0]	ECS1
Net		SI[1]	ECS1
Net		SI[2]	ECS1
Net		SI[3]	ECS1
Net		SI[4]	ECS1
Net		SI[5]	ECS1
Net		SI[6]	ECS1
Net		SI[7]	ECS1
Net		SI[10]	ECS1
Net		SI[11]	ECS1
Net		SI[12]	ECS1
Net		SI[13]	ECS1
Net		SI[14]	ECS1
Net		SI[15]	ECS1
PPr		U6.3:U5.18	
XNet		SUPPLYIN	ECS1
XNet		SUPPLYOUT	ECS1
Net		S_CS	ECS1

## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

If you delete the component and perform the undo operation, the component is added to the design but the associated pin-pairs are removed from Constraint Manager and need to be added again.



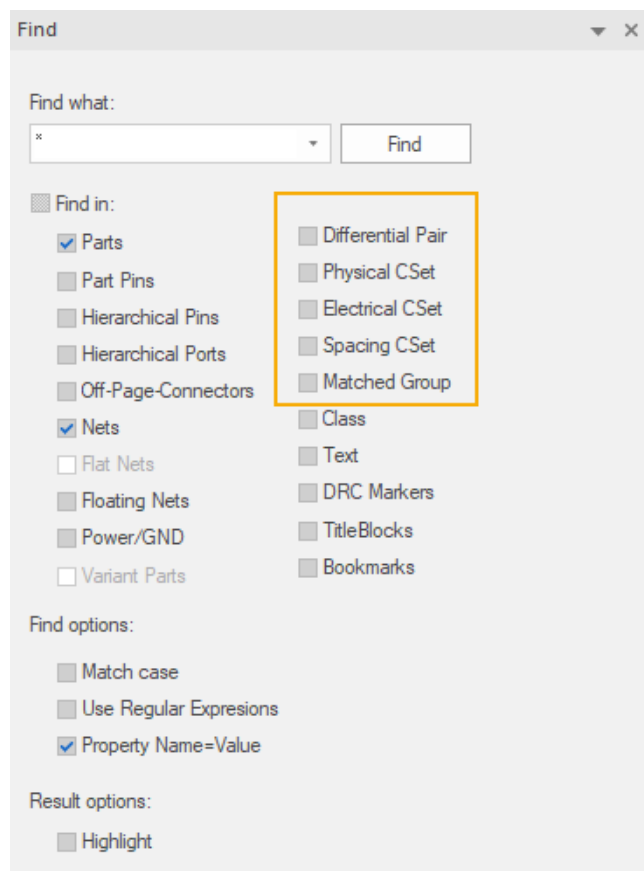
## Finding Constraints

You can use the Search functionality to find transferred constraints. To do so:

1. Select *Edit – Find* or press **CTRL+F**.

The Find pane opens.

Observe the various parameters such as constraint categories using which you can filter the search results.



2. Select the required constraints.
3. Specify the search string in the search text box.
4. Click the *Find* button.

The Find Results window opens displaying the search results.

5. Click the required constraint category.



## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

---

For example, click *Electrical CSet*.

Find Results			
Differential Pair	Class	Electrical CSet	Spacing CSet    Physical CSet
Name	Net Name	Page	Schematic
ECS1	AGND	pg1_main	SCHEMATIC1\
ECS1	CAN_HI	pg1_main	SCHEMATIC1\
ECS1	CAN_LO	pg1_main	SCHEMATIC1\
ECS1	CLOCKD	pg1_main	SCHEMATIC1\
ECS1	HIGH	pg1_main	SCHEMATIC1\
ECS1	J1850BUS+	pg1_main	SCHEMATIC1\
ECS1	J1850BUS-	pg1_main	SCHEMATIC1\
ECS1	L-LINE	pg1_main	SCHEMATIC1\
ECS1	LOW	pg1_main	SCHEMATIC1\
ECS1	N06347	pg1_main	SCHEMATIC1\

6. Double-click a constraint.

For example, Double-click the net, Q2 in the *Electrical CSet* tab.

## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

The Q2 net is selected in the schematic and in the Constraint Manager window.

The screenshot displays the OrCAD X Capture interface. The top window shows a schematic diagram of a circuit. The bottom window shows the 'Find Results' tab of the Constraint Manager, which lists the objects and their referenced electrical constraints.

Name	Net Name	Page
ECS1	Q2	pg
ECS1	Q3	pg
ECS1	Q4	pg
ECS1	Q5	pg
ECS1	Q6	pg
ECS1	Q7	pg
ECS1	RST_NVM	pg
ECS1	RX-I	pg
ECS1	S_CS	pg
ECS1	S_ISBA	pg

Objects			Referenced Electrical CSet
Type	S	Name	
*	*	*	*
Dsn		DESIGNDB3	
Bus		DATA_OUT(8)	
Net		Q0	ECS1
Net		Q1	ECS1
Net		Q2	ECS1
Net		Q3	ECS1
Net		Q4	ECS1

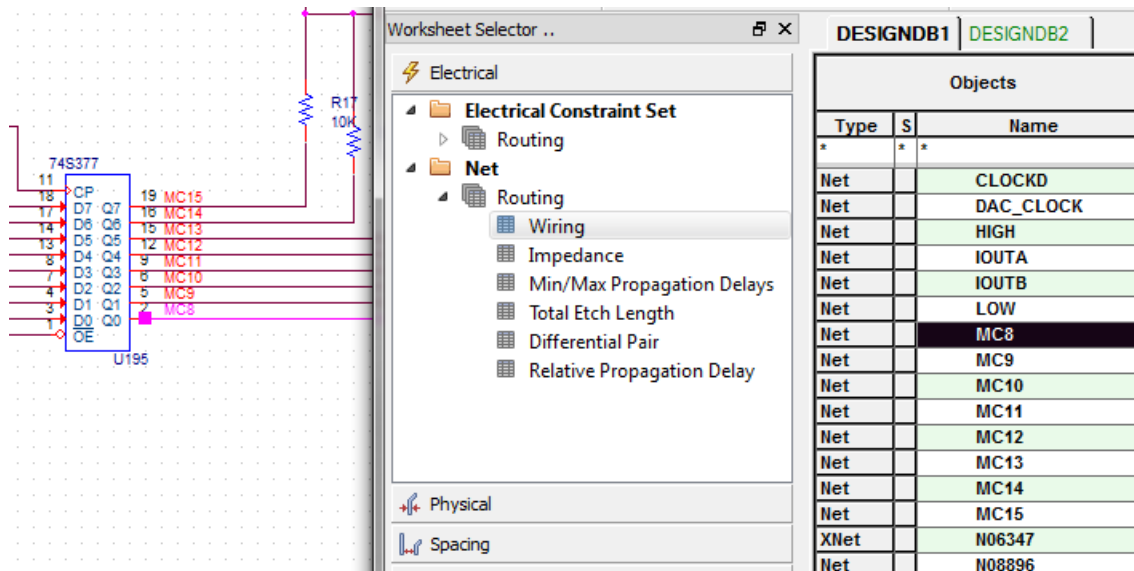
## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

## Cross-Probing of Design Objects

You can use the cross-probing feature to navigate between the schematic nets and the net entries in Constraint Manager.

- ➔ Select a net in the schematic design, the corresponding net name is highlighted in Constraint Manager.



## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

- ➔ Select a net name in Constraint Manager. Right-click this net name and choose *Select*. The corresponding net is highlighted in the schematic design.

The screenshot displays the OrCAD X Capture interface. On the left, the 'Worksheet Selector' window shows the 'Electrical' tab selected, with the 'Net' folder expanded. The 'Routing' folder is also expanded, showing various constraint types like 'Impedance', 'Min/Max Propagation Delays', etc. The 'DESIGNDB1' window is open, showing a list of objects. The 'MC11' net is highlighted in blue. A right-click context menu is open over the 'MC11' net, with the 'Select' option chosen. The schematic diagram on the right shows a circuit with a 74S377 decoder, several resistors (R1, R2, R3, R4, R5, R6, R7, R8, R9, R10, R11, R12, R13, R14, R15), and multiple multiplexers (MC8, MC9, MC10, MC11, MC12, MC13, MC14, MC15). The 'MC11' net is highlighted in blue, corresponding to the selection in the Constraint Manager.

Type	S	Name	Referenced Electrical CSet	V	S
Dsn		DESIGNDB1			
Net		CLOCKD			
Net		DAC_CLOCK			
Net		HIGH			
Net		IOUTA			
Net		IOUTB			
Net		LOW			
Net		MC8			
Net		MC9			
Net		MC10			
Net		MC11			
Net		MC12			
Net		MC13			
Net		MC14			
Net		MC15			
XNet		N06347			
Net		N08896			
Net		N09022			

# Constraint Manager with OrCAD X Capture

## Constraint Manager with OrCAD X Capture

### Renaming Nets

If you change the net name in the schematic, you can see the corresponding change in Constraint Manager.

The image illustrates the process of renaming a net in OrCAD X Capture. It shows a schematic diagram of a 74S377 chip connected to a resistor network. The net names are listed in the schematic: CP, D7, Q7, D6, Q6, D5, Q5, D4, Q4, D3, Q3, D2, Q2, D1, Q1, D0, Q0, OE, MC15, MC14, MC13, MC12, MC11, MC10, MC9, MC8.

The Constraint Manager interface is shown in three states:

- Initial State:** The 'DESIGNDB1' database is open, showing a list of nets. The net 'MC15' is highlighted.
- Editing State:** The 'Edit Net Alias' dialog box is open, showing the net 'MC15' and the new alias 'MC16'.
- Final State:** The 'DESIGNDB1' database is open, showing the updated list of nets. The net 'MC16' is now listed, and 'MC15' is no longer present.

## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

## Modifying Voltage for Nets

After you enable Constraint Manager, to modify voltage values for nets, do the following:

1. In the Constraint Manager window, select the *Properties* selector bar.
2. Select *Net – General Properties*.
3. View or modify the values in the *Voltage* column in the worksheet.

The screenshot shows the OrCAD X Capture Constraint Manager window. On the left, the 'Worksheet Selector ..' panel is open, showing a tree view of properties. The 'Net' folder is expanded, and 'General Properties' is selected. Below this, the 'Design Instance/Block filter' panel shows 'DESIGNDB3' selected. The main area displays a worksheet for 'DESIGNDB3' with columns: Objects, Voltage, Weight, and No Rat. The 'Voltage' column is highlighted with a yellow box. The worksheet contains a list of nets and their associated voltage values.

Type	S	Name	V	Weight	No Rat
Net		SI[13]			On
Net		SI[14]			On
Net		SI[15]			On
Net		SUPPLYIN			On
Net		SUPPLYOUT			On
Net		S_CS			On
Net		S_ISBA			On
Net		S_SI			On
Net		TB1			On
Net		TB2			On
Net		TB3			On
Net		TURN1			On
Net		TURN2			On
Net		TURN3			On
Net		TX-O			On
Net		TX-STAT			On
Net	G	V+	1.5V		On
Net	G	V-	1.5V		On
Net	G	VBAT	1.5V		
Net		VBATT_RAW			On
Net	G	VCC	1.5V		On
Net		VIN_0			On
Net		VIN_1			On
Net		VIN_2			On
Net		VIN_3			On
Net		VIN_4			On
Net		VPW_RX			On
Net	G	3.3V	3.3V		On
Net	G	5V	5V		On

## Working with Constraint Objects

This section explains some functions related to constraint objects in Constraint Manager:

- [Managing XNets](#)
- [Performing Signal Analysis in Constraint Manager-Enabled Design](#)
- [Working with Electrical CSets](#)

### Managing XNets

When you enable Constraint Manager in a design in which a net traverses a passive, discrete device (resistor, inductor, or capacitor), XNets are automatically created.

In the Constraint Manager interface, such objects are specified as `XNet` in the `Type` column.

### Creating XNets

To create an XNet, do the following:

1. Open the required design in Capture.
2. Enable Constraint Manager for this design.  
The Constraint Manager window opens.
3. As an example:
  - a. From the left pane, choose *Net – Routing – Wiring*.

## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

- b. Check that the value corresponding to SI [15] in the Type column is Net.

The screenshot shows the OrCAD X Capture Constraint Manager interface. The left pane displays the 'Electrical' tree structure, with 'Net' selected under 'Routing'. The main pane shows a table of constraints for 'DESIGNDB1'. The table has columns: Objects (Type, S, Name), Referenced Electrical C Set, and Topology (Verify Sched, Schedule, Actual). The row for 'SI[15]' is highlighted with a blue selection bar and an orange border.

Objects			Referenced Electrical C Set	Topology		
Type	S	Name		Verify Sched	Schedule	Actual
XNet		N06347				
Net		N08896				
Net		N09022				
Net		N10683				
Net		N20773				
Net		N24309				
XNet		N16619096				
Net		N16619156				
Net		N167324072				
Net		Q0				
Net		Q1				
Net		Q2				
Net		Q3				
Net		Q4				
Net		Q5				
Net		Q6				
Net		Q7				
Net		RST_NVM				
Net		SI[8]				
Net		SI[9]				
Net		SI[10]				
Net		SI[11]				
Net		SI[12]				
Net		SI[13]				
Net		SI[14]				
Net		SI[15]				
Net		S_CS				
Net		S_JSBA				
Net		S_SI				

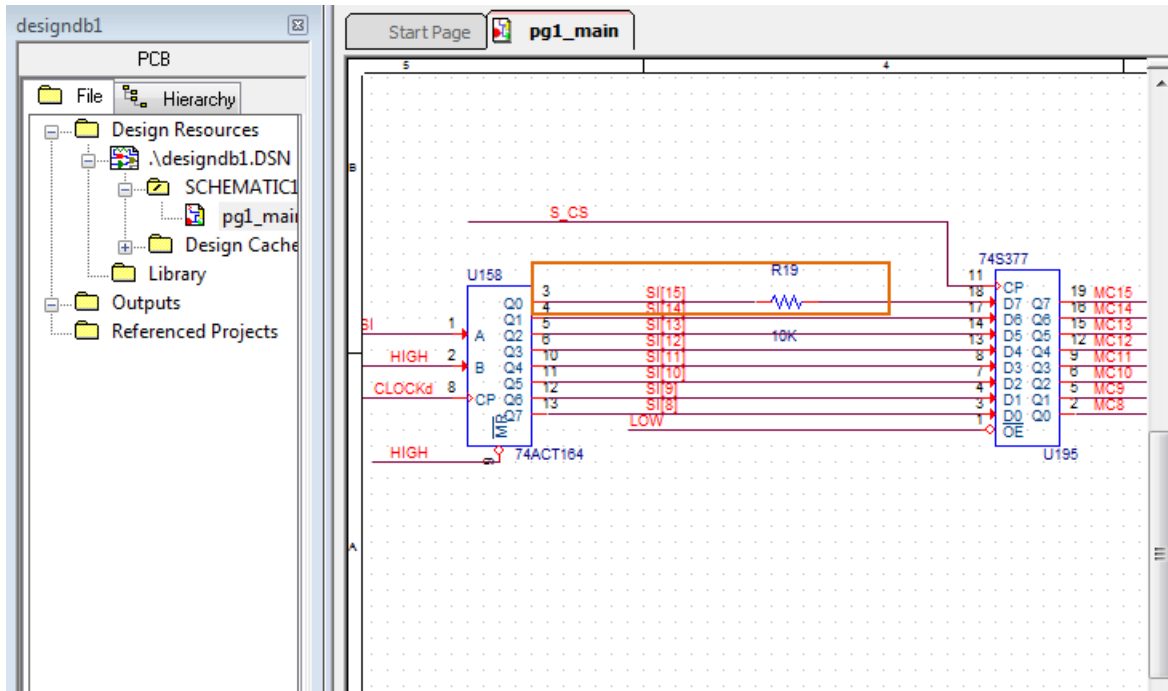


## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

---

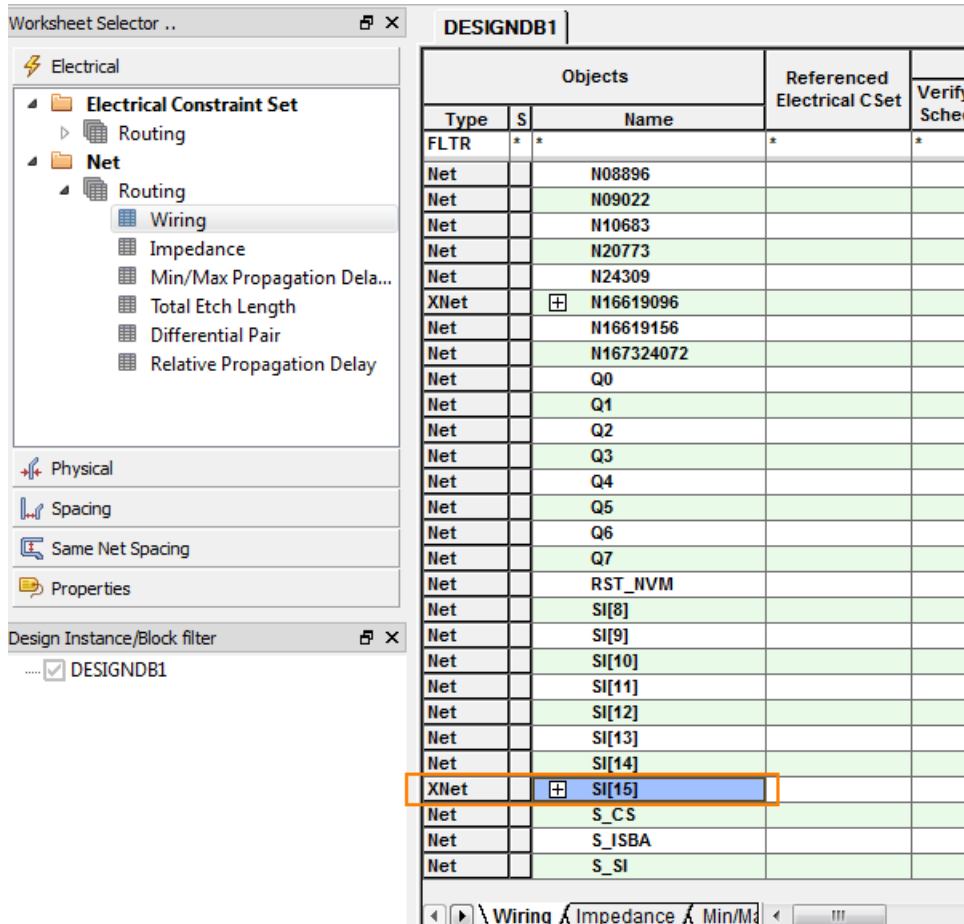
c. Add a resistor to SI [15].



## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

- d. Check that when you add a resistor, Net changes to XNet in the Type column as highlighted in the following screen shot.

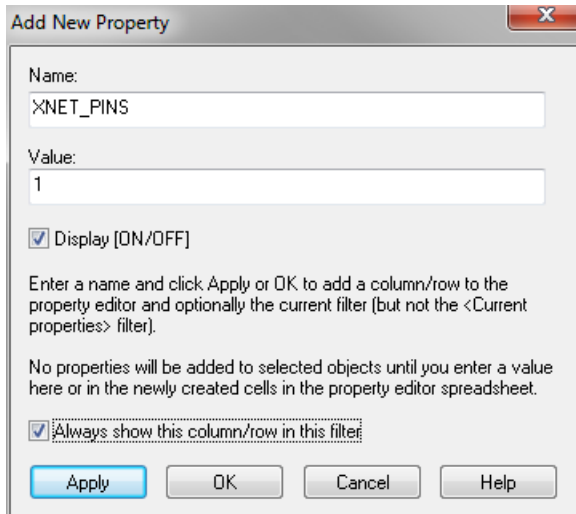


### Creating an XNet using Non-Discrete Components

In case of non-discrete components, to convert a Net into XNet, do the following:

1. Open the required design in Capture.
2. Enable Constrain Manager for this design.
3. Select the pins through which an XNet connection needs to be created.
4. Right-click the pin set and select *Edit Properties*.
5. Click *New Property*.

6. Specify the XNET\_PINS property to the selected pin set.



The applied property will be attached to one of the pin pairs.

You must apply the XNET\_PINS property for each net that needs to be converted.

7. Click *Apply*.

You can see that the type has changed from Net to XNet in Constraint Manager.

## Removing an XNet

To remove an XNet, do the following:

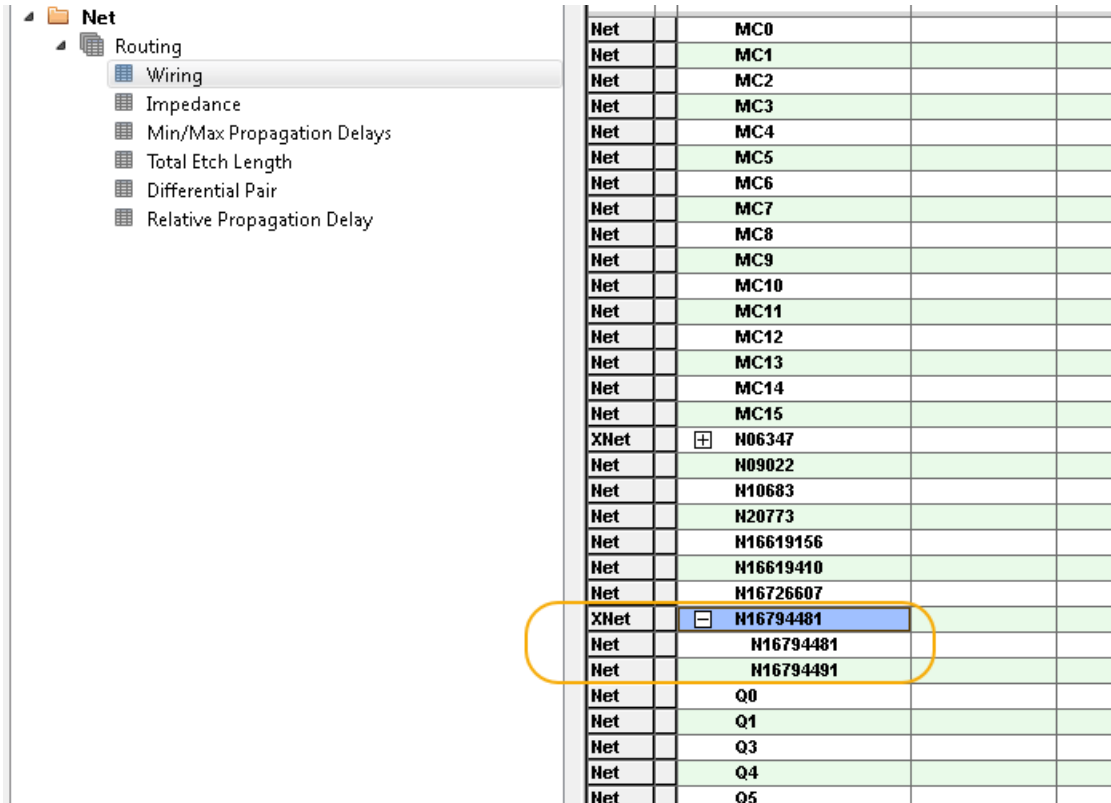
1. Enable the required design for Constraint Manager.
2. Select a discrete component in an XNet.

## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

---

The XNet highlighted in the following example will be removed.



The screenshot shows the 'Net' tree on the left with 'Routing' expanded, showing 'Wiring' and various constraints. The main table lists nets and their properties. The 'XNet' N16794481 is highlighted with a yellow circle.

Net	MC0		
Net	MC1		
Net	MC2		
Net	MC3		
Net	MC4		
Net	MC5		
Net	MC6		
Net	MC7		
Net	MC8		
Net	MC9		
Net	MC10		
Net	MC11		
Net	MC12		
Net	MC13		
Net	MC14		
Net	MC15		
XNet	N06347		
Net	N09022		
Net	N10683		
Net	N20773		
Net	N16619156		
Net	N16619410		
Net	N16726607		
XNet	N16794481		
Net	N16794481		
Net	N16794491		
Net	Q0		
Net	Q1		
Net	Q3		
Net	Q4		
Net	Q5		

3. Right-click the discrete component and select *Edit Properties*.
4. Click *New Property*.

The Add New Property window opens.

## Constraint Manager with OrCAD X Capture

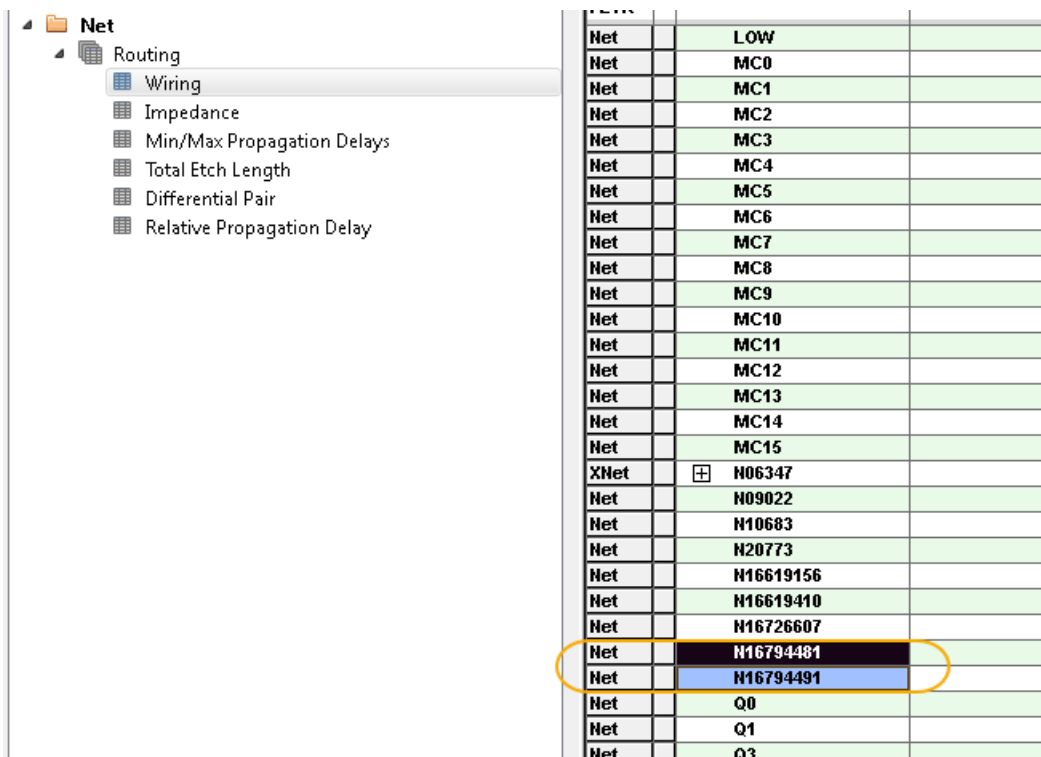
### Constraint Manager with OrCAD X Capture

5. Set the NO\_XNET\_CONNECTION property to YES.



The 'Add New Property' dialog box is shown. The 'Name' field contains 'NO\_XNET\_CONNECTION' and the 'Value' field contains 'YES'. The 'Display [ON/OFF]' checkbox is checked. Below the fields, there is explanatory text: 'Enter a name and click Apply or OK to add a column/row to the property editor and optionally the current filter (but not the <Current properties> filter). No properties will be added to selected objects until you enter a value here or in the newly created cells in the property editor spreadsheet.' At the bottom, there are four buttons: 'Apply', 'OK', 'Cancel', and 'Help'.

You can see that the *Type* has changed from XNet to Net in the Constraint Manager UI.



The screenshot shows the Constraint Manager UI. On the left, a tree view under 'Net' shows 'Routing' expanded, with 'Wiring' selected. The main area displays a table of constraints. The table has columns for 'Type', 'Name', and 'Value'. The 'Type' column shows 'Net' for most entries, with 'XNet' for 'N06347'. The 'Name' column lists various net names, and the 'Value' column shows 'LOW' or 'MC' followed by a number. The row for 'N16794481' is highlighted in blue, and the row for 'N16794491' is highlighted in orange.

Type	Name	Value
Net		LOW
Net		MC0
Net		MC1
Net		MC2
Net		MC3
Net		MC4
Net		MC5
Net		MC6
Net		MC7
Net		MC8
Net		MC9
Net		MC10
Net		MC11
Net		MC12
Net		MC13
Net		MC14
Net		MC15
XNet	N06347	
Net	N09022	
Net	N10683	
Net	N20773	
Net	N16619156	
Net	N16619410	
Net	N16726607	
Net	N16794481	
Net	N16794491	
Net	Q0	
Net	Q1	
Net	Q3	

## Performing Signal Analysis in Constraint Manager-Enabled Design

You can view the topology of the signal flow and also assign constraints, such as propagation delay and relative propagation delay in the schematic and manage them using the Constraint Manager UI.

To view the topology and assign constraints, do the following:

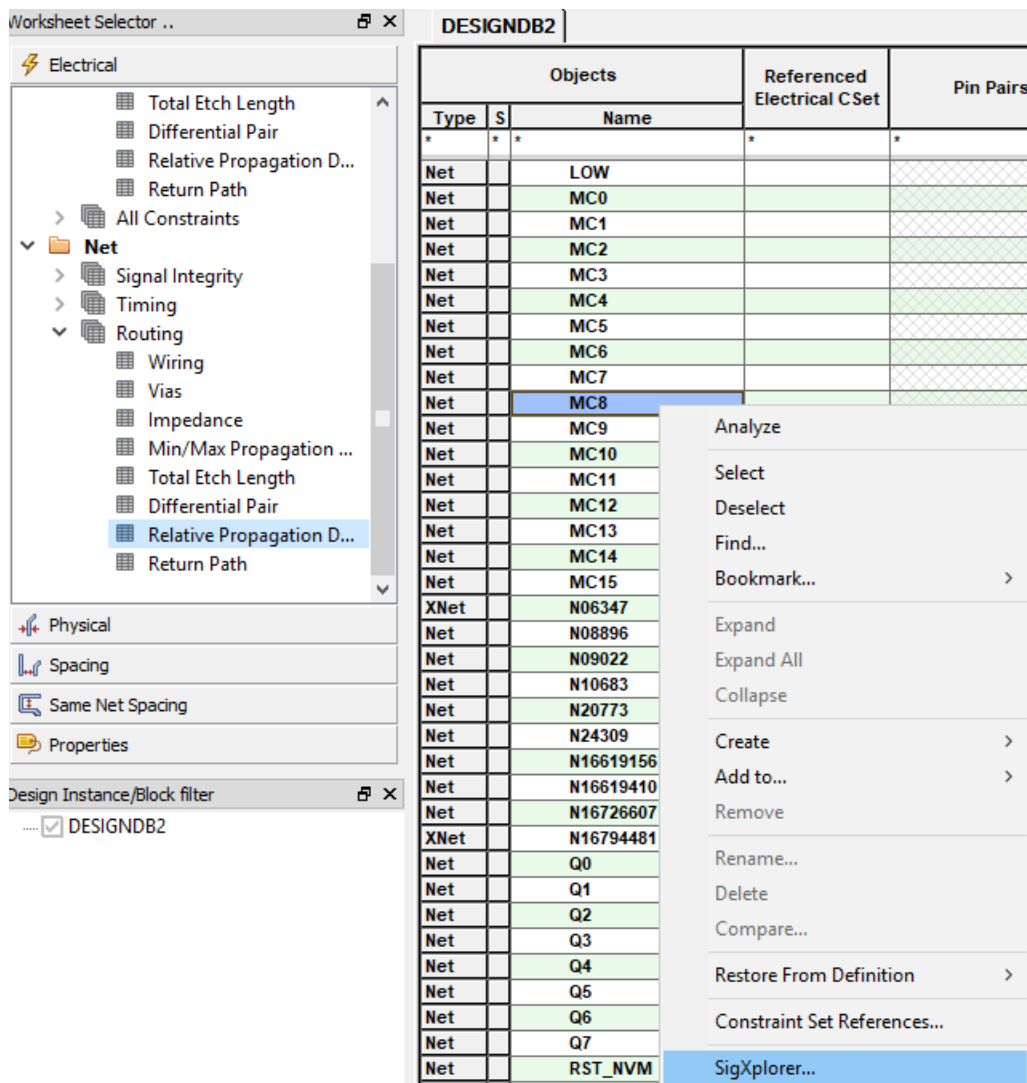
1. Open the required design in Capture.
2. Enable Constraint Manager on this design.

The Constraint Manager window opens.

3. As an example:
  - a. From the left pane, choose *Net – Routing – Relative Propagation Delay*.
  - b. Select a net, MC8.
  - c. Right-click MC8 and select *SigXplorer*.

## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture



d. Select a license with SI capabilities.

The SigXplorer OrCAD PCB SI window opens.

e. Choose *Setup – Constraints*.

The Set Topology Constraints window opens.

4. As an example, in the Set Topology Constraints window:

a. Click the *Prop Delay* tab.

b. Select the output and input pins.

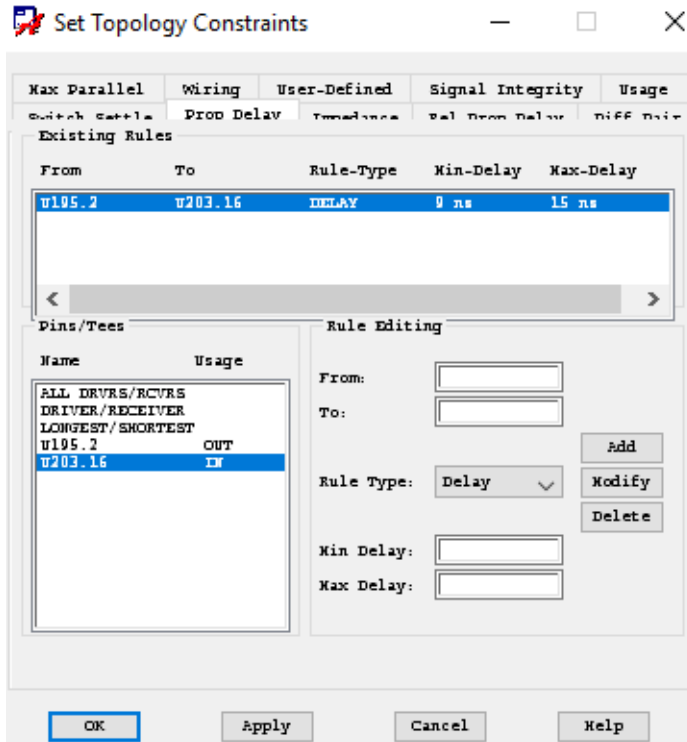
c. Specify the min delay as 9 ns and max delay as 15 ns.

## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

---

- d. Click *Add*.
- e. A new rule is added in the *Existing Rules* section.



- f. Click *Apply* and then *OK*.
- g. Select *File – Update Constraint Manager*.

A message appears to confirm if the net can reference the Electrical Constraint Set.

- h. Click *Yes*.



## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

This constraint is now seen in the Constraint Manager window.

Worksheet Selector ..

DESIGNDB2

Objects			Referenced Electrical CSet	Pin Pairs	Pin Delay	
Type	S	Name			Pin 1 mil	Pin 2 mil
Dsn	*	DESIGNDB2	*	*	*	*
DPr	+	DP1				
Net		CLOCKD				
Net		DAC_CLOCK				
Net		HIGH				
XNet		IOUTA				
Net		LOW				
Net		MC0				
Net		MC1				
Net		MC2				
Net		MC3				
Net		MC4				
Net		MC5				
Net		MC6				
Net		MC7				
Net		MC8	MC8			
Net		MC9				
Net		MC10				
Net		MC11				
Net		MC12				

Electrical

Electrical Constraint Set

- Signal Integrity
- Timing
- Routing
- All Constraints
- Net
  - Signal Integrity
  - Timing
  - Routing
    - Wiring
    - Vias
    - Impedance
    - Min/Max Propagation Dela...
    - Total Etch Length
    - Differential Pair
    - Relative Propagation Delay
    - Return Path

Physical

Spacing

- Right-click this Electrical Constraint Set (Electrical CSet) to view the operations you can perform on it.

Net

- Signal Integrity
- Timing
- Routing
  - Wiring
  - Vias
  - Impedance
  - Min/Max Propagation Dela...
  - Total Etch Length
  - Differential Pair
  - Relative Propagation Delay
  - Return Path

Net	MC5		
Net	MC6		
Net	MC7		
Net	MC8	MC8	
Net	MC9		
Net	MC10		
Net	MC11		
Net	MC12		
Net	MC13		
Net	MC14		
Net	MC15		
XNet	N06347		
Net	N08896		
Net	N09022		
Net	N10683		
Net	N20773		

Go to source

Change...

Clear

Clear and Restore

Goto Electrical CSet

Audit Electrical CSet

Restore From Definition >

SigXplorer...

## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

6. Select *Net – Routing – Min/Max Propagation Delay*, you can view the pin pair that is created.

Worksheet Selector ... DESIGNDB2

Objects		Referenced Electrical CSet	Pin Pairs	Pin Delay		Prop Delay			M
Type	Name			Pin 1 mil	Pin 2 mil	Min mil	Actual	Margin	
Dsn	DESIGNDB2								
DPr	DP1								
Net	CLOCKD								
Net	DAC_CLOCK								
Net	HIGH								
XNet	IOUTA								
Net	LOW								
Net	MC0								
Net	MC1								
Net	MC2								
Net	MC3								
Net	MC4								
Net	MC5								
Net	MC6								
Net	MC7								
Net	MC8	MC8							
PPr	U195.2:U203.16					9 ns			15 n
Net	MC9								

### Important

If the referenced Electrical CSet does not show the pin pair information, then *Electrical CSet Apply* feature will work with limitations. You need to upgrade to a higher license for this feature to function correctly.

# Constraint Manager with OrCAD X Capture

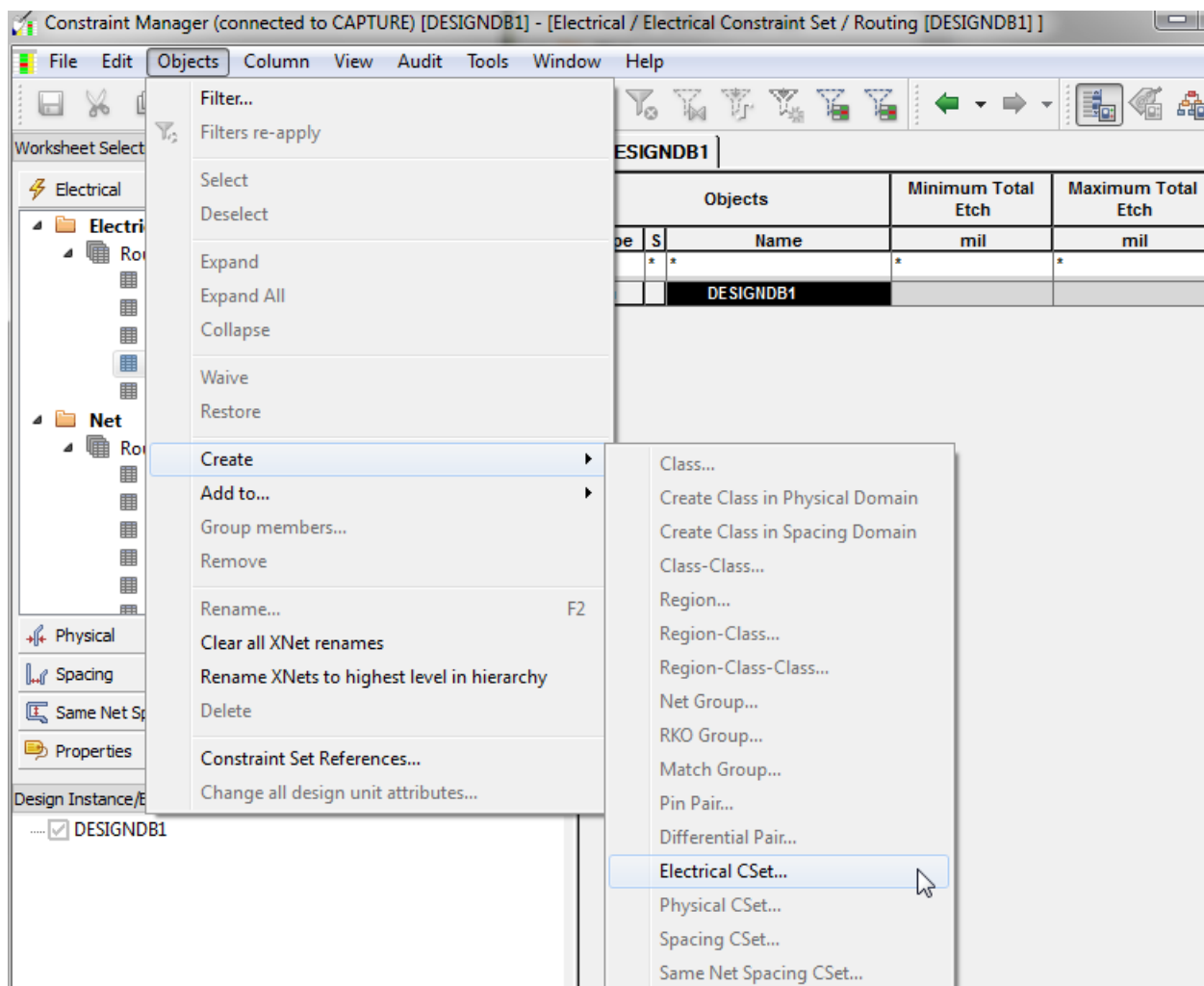
## Constraint Manager with OrCAD X Capture

### Working with Electrical CSets

In Constraint Manager, you can define constraints on design objects using a Constraint Set (CSet). A CSet is a named and reusable collection of constraint values. CSet can then be assigned to a net in any of the *Net* worksheets.

To assign an Electrical CSet to a net, do the following:

1. In a Constraint Manager-enabled design, open Constraint Manager.
2. In the left pane, select *Electrical Constraint Set – Routing – Total Etch Length*.
3. Choose *Objects – Create – Electrical CSet*.

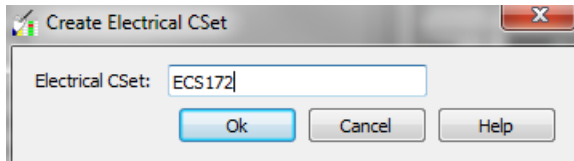


## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

---

4. Specify the name of the Electrical CSet.



5. Specify the minimum and maximum total etch length for this ECSet.
6. Select *Net – Routing – Total Etch Length*.
7. Select the nets corresponding to which you want to assign a ECSet under the *Referenced Electrical CSet* column.

Worksheet Selector ..

**Electrical**

- Electrical Constraint Set
  - Routing
    - Wiring
    - Impedance
    - Min/Max Propagation Delays
    - Total Etch Length
    - Differential Pair
- Net
  - Routing
    - Wiring
    - Impedance
    - Min/Max Propagation Delays
    - Total Etch Length
    - Differential Pair

**Physical**

**DESIGNDB1**

Objects		Referenced Electrical CSet	Total Etch Length	
Type	S		Min	Max
Dsn	DESIGNDB1			
Net	CLOCKD			
Net	DAC_CLOCK			
Net	HIGH			
Net	IOUTA			
Net	IOUTB			
Net	LOW			
Net	MC8		15.00	
Net	MC9			
Net	MC10	ECS172		
Net	MC11			
Net	MC12			
Net	MC13			
Net	MC14			
Net	MC16			

## Modes for Processing Constraints: Overwrite and Changes Only

All constraints captured in schematic get transferred to PCB using the schematic to PCB create process.

You can transfer constraints using one of the following two options using the Update Layout and Update Schematic dialog boxes:

- Overwrite

Use the *Overwrite* mode when you need to overwrite the constraints in schematic or in board file.

- Changes Only

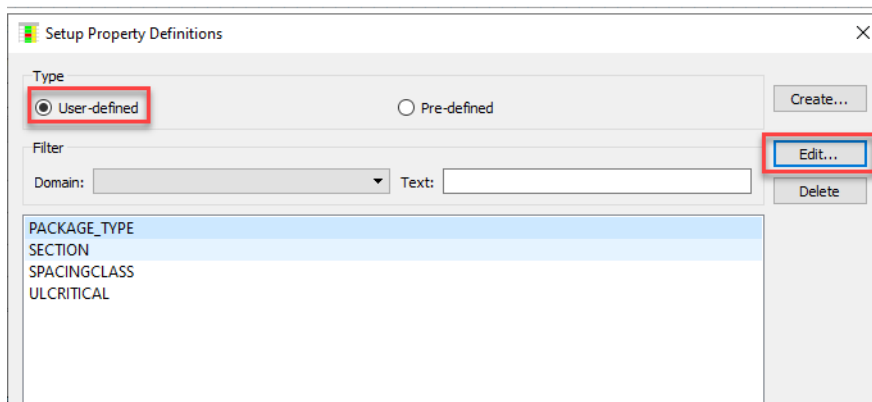
Use the *Changes Only* mode if you need to merge concurrent constraint changes done in schematic and layout.

## Updating Property Definition in Constraint Manager

You can edit the property definition in Constraint Manager connected to Capture to prevent or enable creation of user-defined properties in Constraint Manager.

To edit property definition, do the following:

1. Launch Capture.
2. From Capture, click the *Constraint Manager* icon to launch Constraint Manager.
3. In Constraint Manager, choose *Tools – Setup property definitions*.
4. In the *Setup Properties Definitions* dialog box, ensure that *user-defined* is selected for the *Type* field.
5. Select the property and click the *Edit* button.



## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

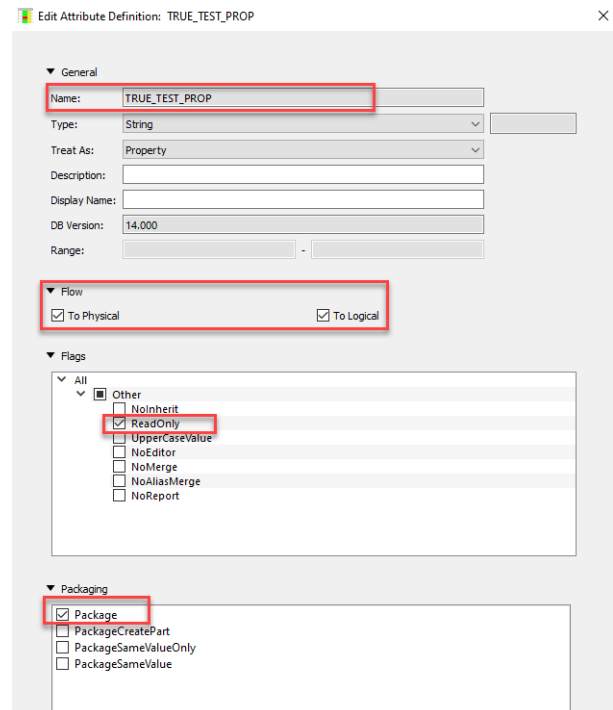
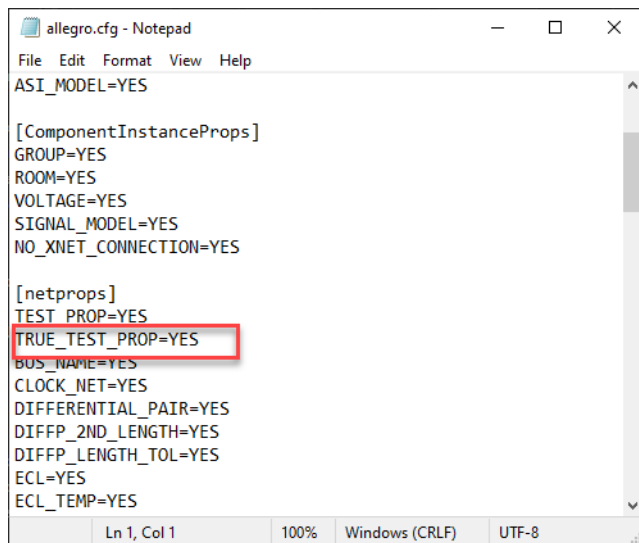
6. In the *Edit Attribute Definitions: <property>* dialog box, click *Packaging*, and select the *Package* check box.

The screenshot shows the 'Edit Attribute Definition: PACKAGE\_TYPE' dialog box. The 'General' section is expanded, showing fields for Name (PACKAGE\_TYPE), Type (String), Treat As (Property), Description, Display Name, DB Version (14.000), and Range. Below this are sections for Flow, Flags, and Packaging. The Packaging section is expanded and highlighted with a red rectangle, showing a list of checkboxes: Package (checked), PackageCreatePart, PackageSameValueOnly, and PackageSameValue. Below the Packaging section are two sections: 'No Package' and 'No Package No Flow', each with a list of checkboxes. At the bottom are buttons for OK, Cancel, and Help.

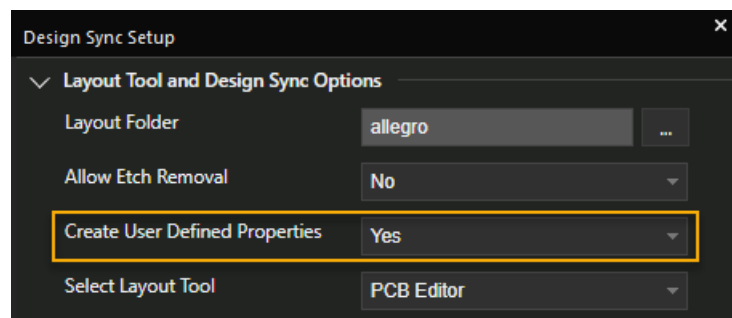
7. Click *OK* to save the changes.
8. Click *OK* to close the *Setup Properties Definitions* dialog box.

## Handling User-Defined Properties

On running the front-to-back flow, a Package type definition is created for each user-defined property defined in `<installation_directory>\tools\capture\allegro.cfg`, and added in the constraint dictionary file (`<project_name>.dcf`). After the Package type definition is added into the `.dcf` file, the property is treated as a non-Constraint Manager property in both Capture and PCB Editor.



For this process to work, you need to set the value of the *Create User Defined Properties* field to *Yes* in the *Design Sync Setup* dialog box accessible from the *PCB – Design Sync Setup* menu command.

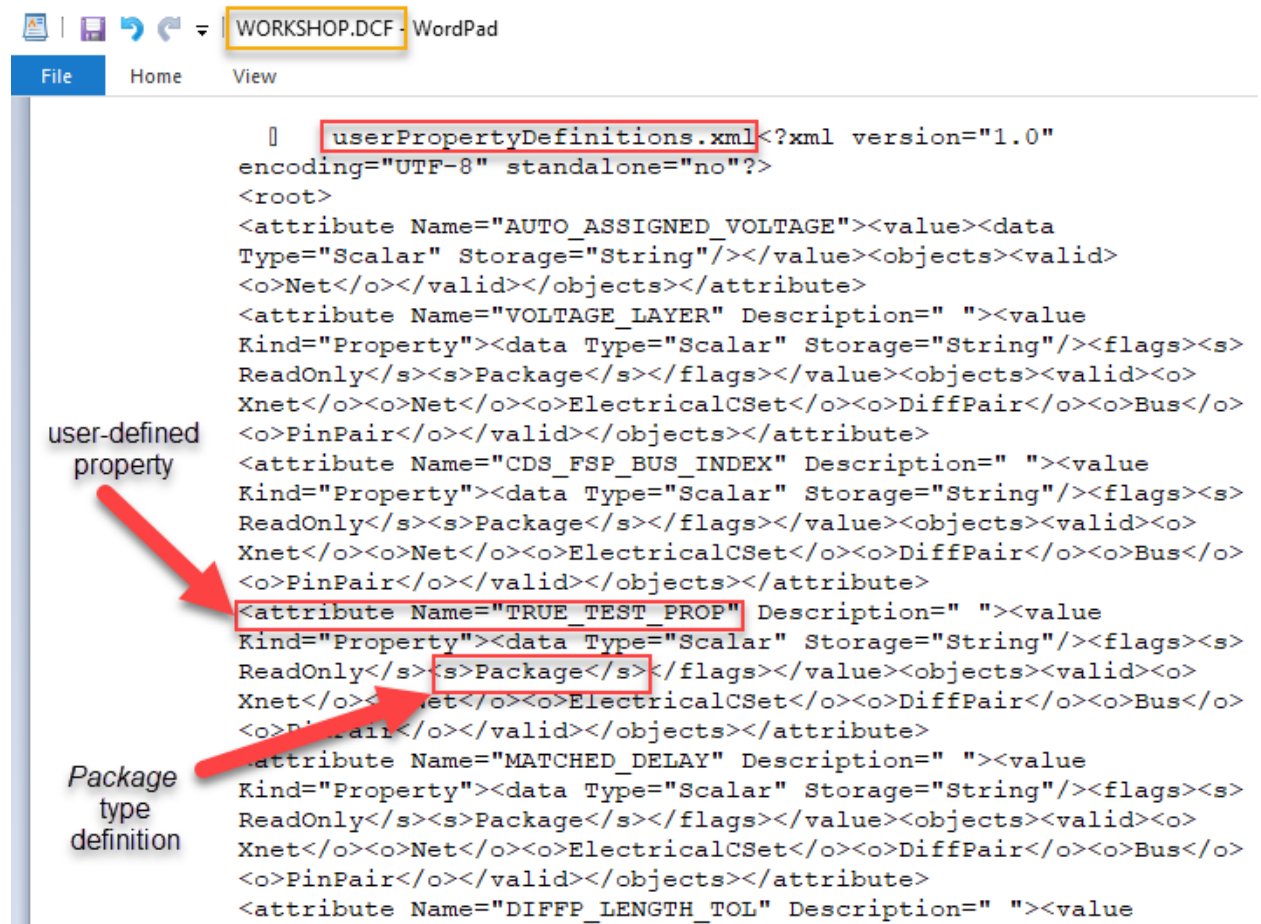




## Constraint Manager with OrCAD X Capture

### Constraint Manager with OrCAD X Capture

- If a user-defined property in the .dcf file exists in Constraint Manager (connected to Capture), it is deleted from there and its value is added or updated in Property Editor in Capture.
- For a new user-defined property defined in allegro.cfg, but not present in Constraint Manager (connected to Capture), on running the front-to-back flow, the Package type definition is added in the .dcf file.



- If a user-defined property is already pushed into Constraint Manager (connected to Capture) regardless of whether the user-defined property was initially added in schematic editor or in Allegro Constraint Manager, it is handled in the following manner:
  - ❑ On running the front-to-back flow, the Package type definition for each property defined in allegro.cfg is added in the .dcf file.
  - ❑ The property with Package type defined in Constraint Manager (connected to Capture) is synchronized with the Capture schematic.
  - ❑ The property is deleted from Constraint Manager (connected to Capture).

## Known Good Practices

This section lists some of the good practices when using the Capture-Constrain Manager flow.

- Back up your design before enabling Constraint Manager. If enabled, you cannot change it to a non-Constraint Manager-enabled design.
- Modify the constraints after completing the design.
- Specify the `Voltage` property for power nets.
- Import the technology file in Constraint Manager to add physical and spacing constraints in the logical design.
- 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.

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.

- Do not rename a Constraint Manager-enabled non-root-schematic design, else all the constraints will be lost.
- If the Constraint Manager-enabled design is opened in a lower release version of Capture, the constraints will be preserved only if the design has not been modified.