cadence®

Constraint Manager with OrCAD® X Capture

Product Version 23.1
September 2023

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

Contents

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.

Important

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

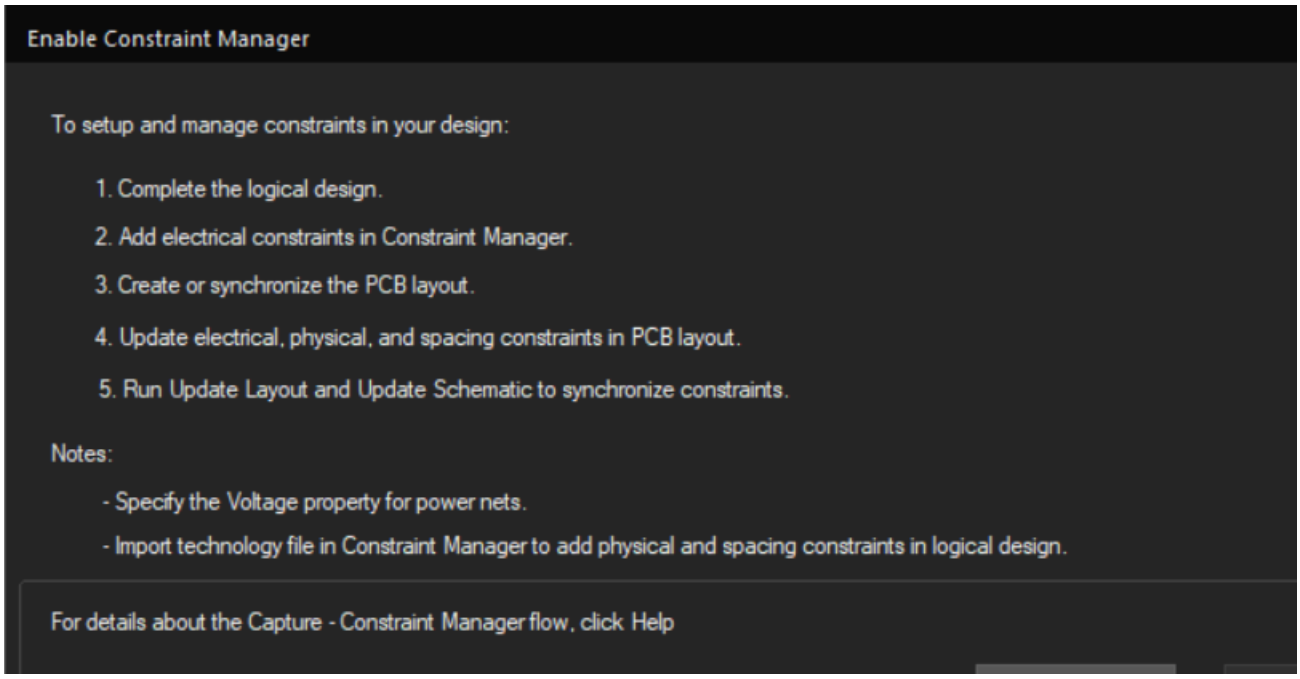
To transfer constraints from a new or existing design, do the following:

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.

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

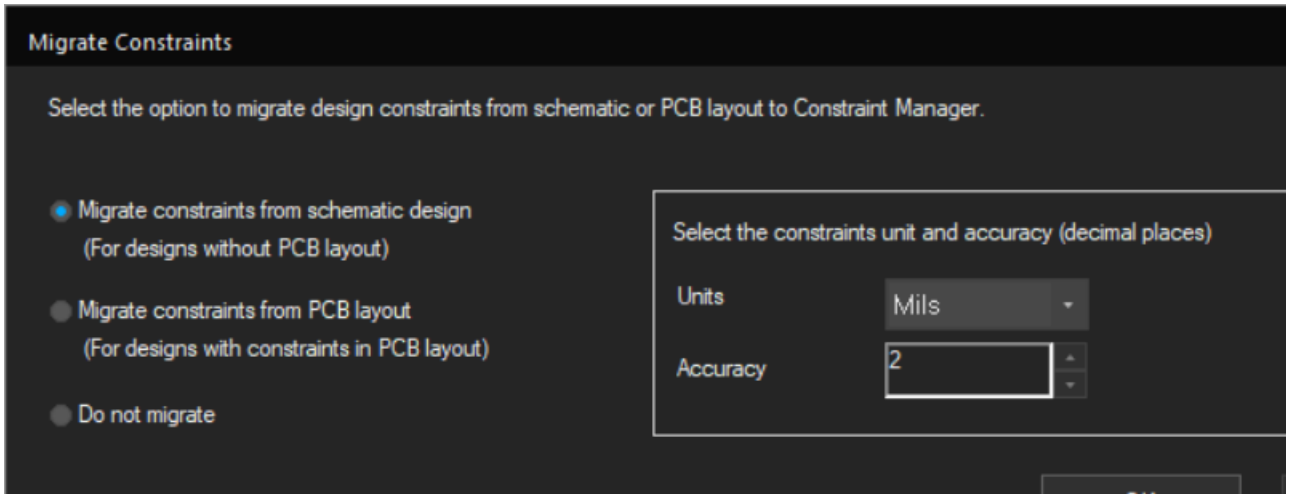


4. Click *OK*.

   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 Manger interface.

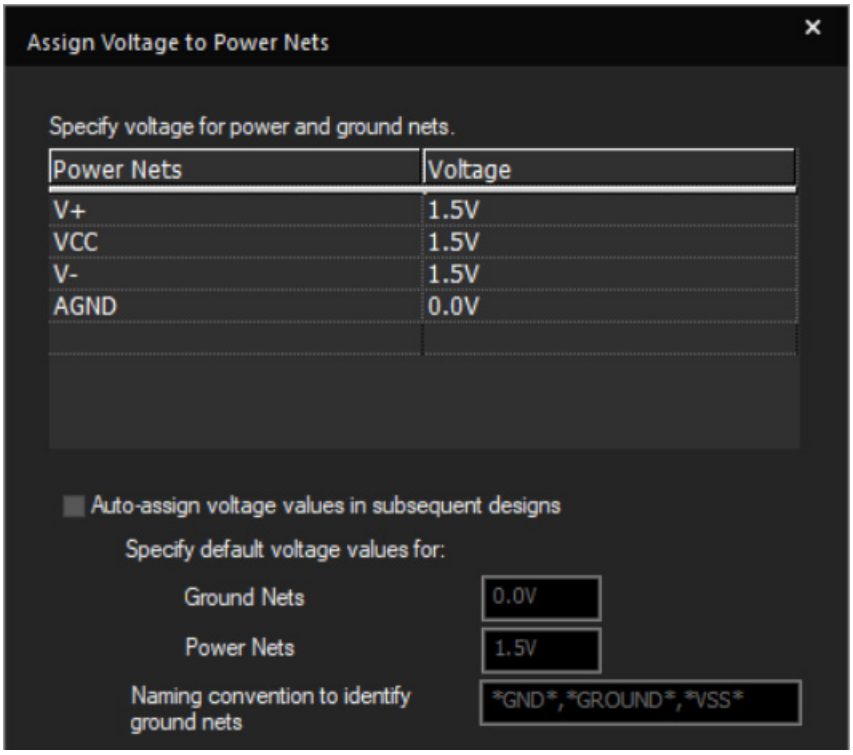


7. Select the *Do not show the message again* check box to stop this message from appearing each time Constraint Manager is invoked
8. Click *OK*.

   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.



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

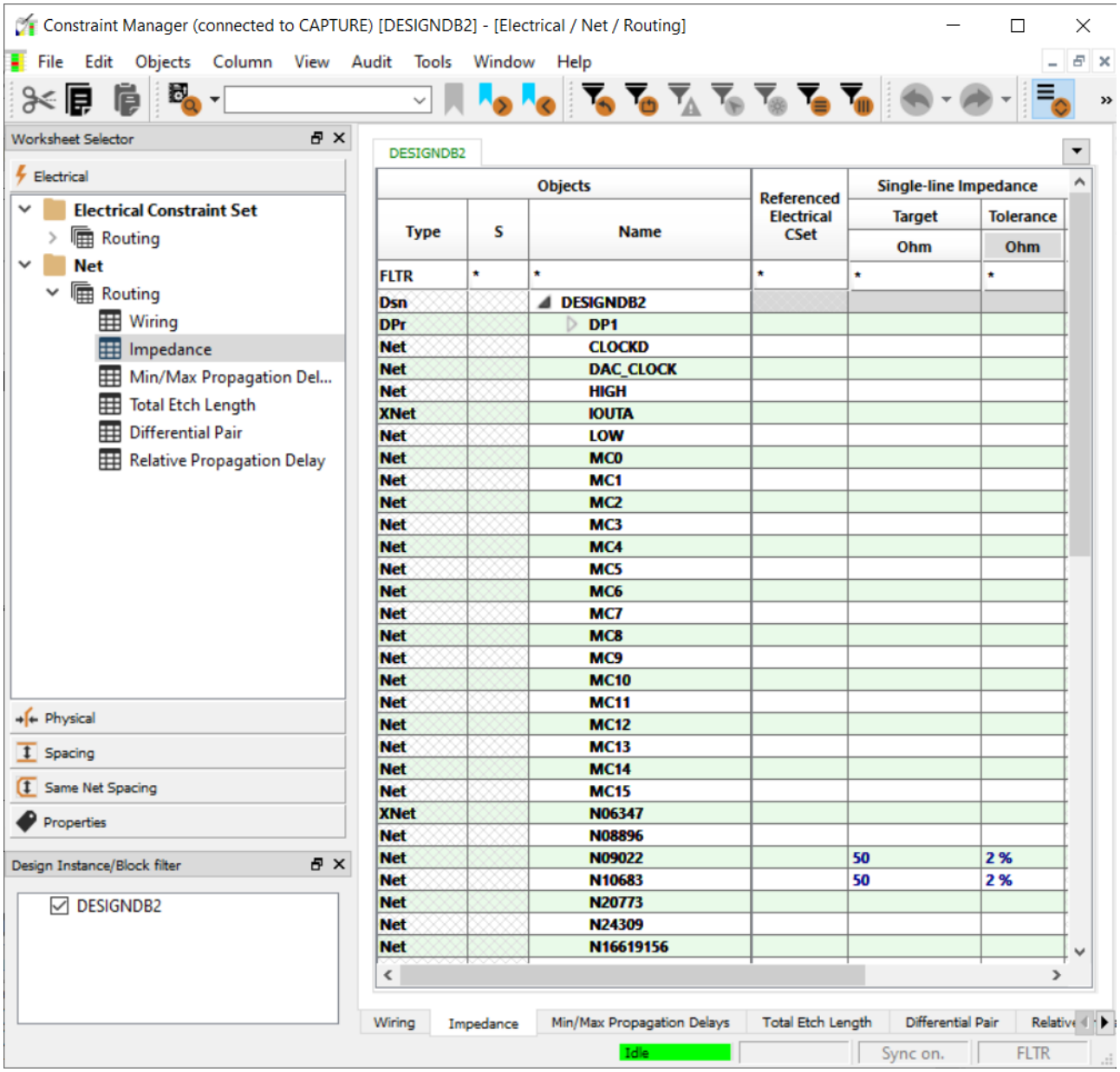
Note: After enabling Constraint Manager, you can modify voltage values for nets directly

in the Constraint Manager window. For details, see Modifying Voltage for Nets.

12. Click *OK*.

The following tasks are completed:

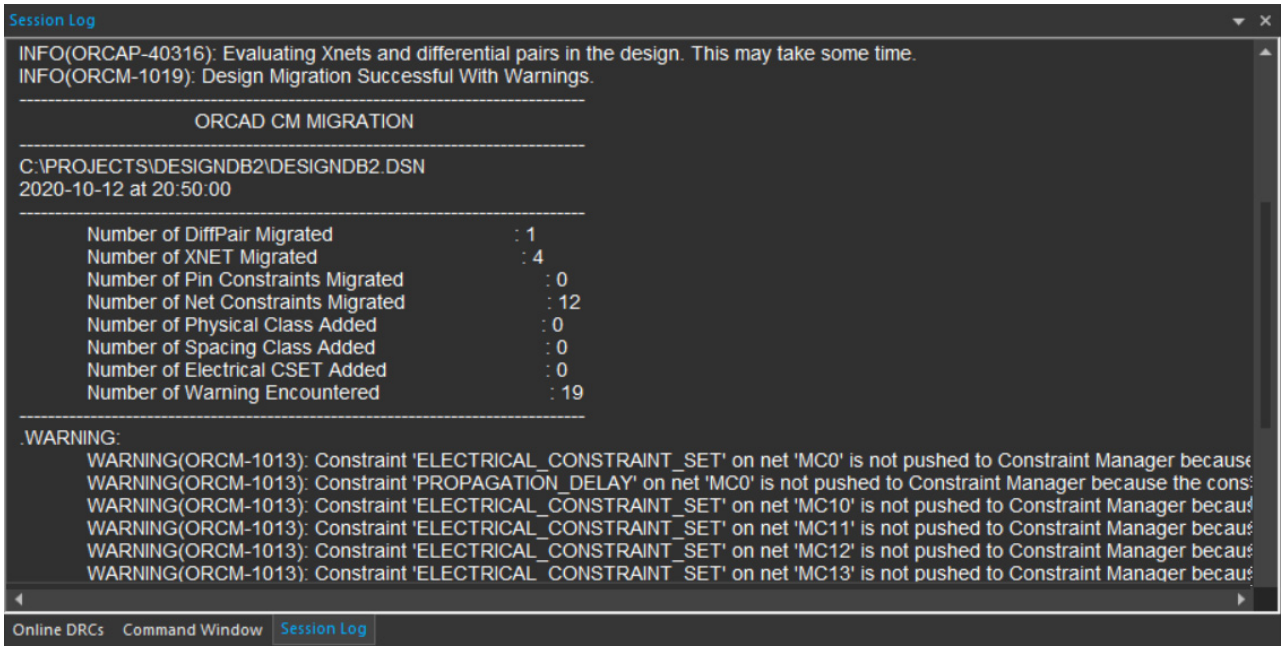
- ❑ Constraints (if any) in the schematic design are transferred to Constraint Manager.
- ❑ Constraints Manager opens. Following screen shot is for a design that has constraints.



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.



Important

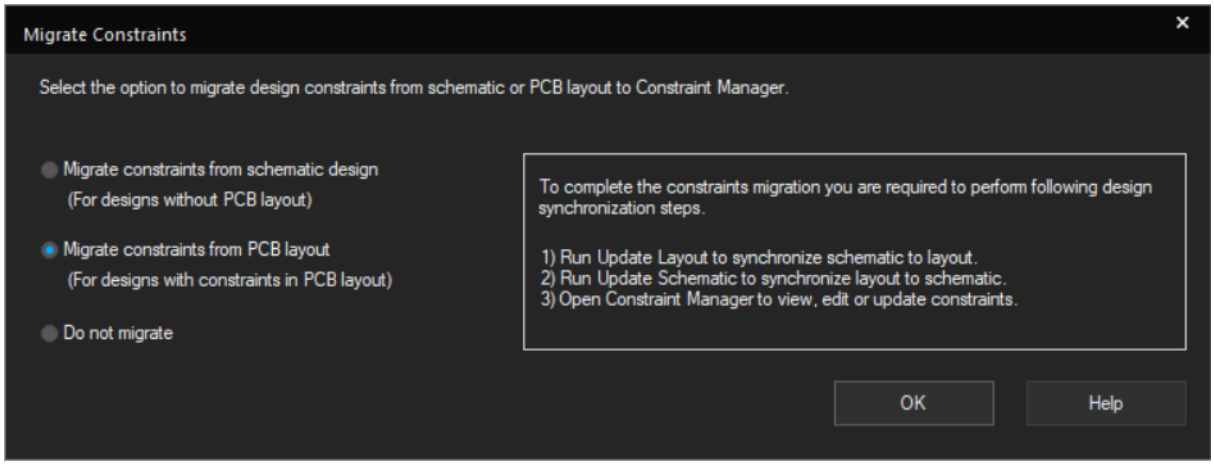
If you select the *Migrate constraints from schematic design* option for a design that has a physical layout file, the design is enabled for Constraint Manager, and the Update Layout window opens a message to indicate that the schematic design and the physical layout are not synchronized. You are also 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:

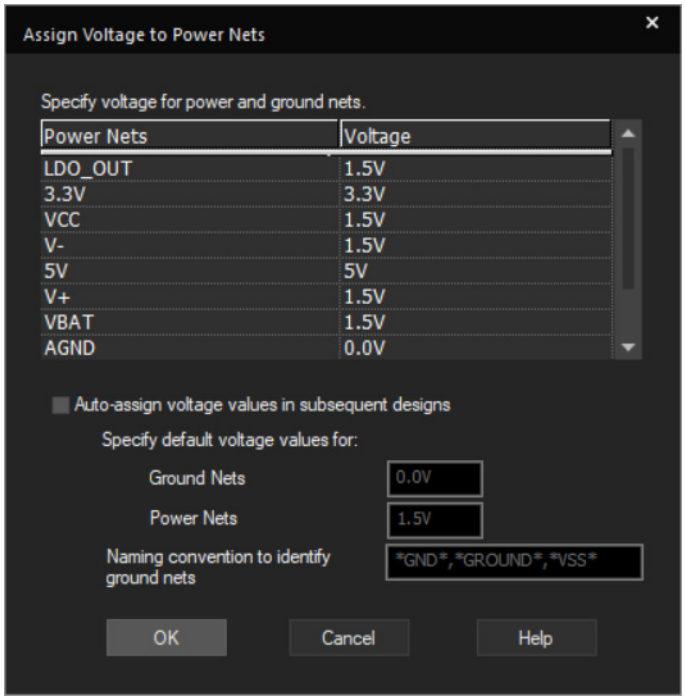
1. Before you enable Constraint Manager to import constraints from physical layout, ensure that you have already synchronized the design and board in the non-Constrain Manager-enabled mode.
2. Click the Constraint Manager icon ( )in the Capture toolbar.

   The *Migrate Constraints* dialog box appears.



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

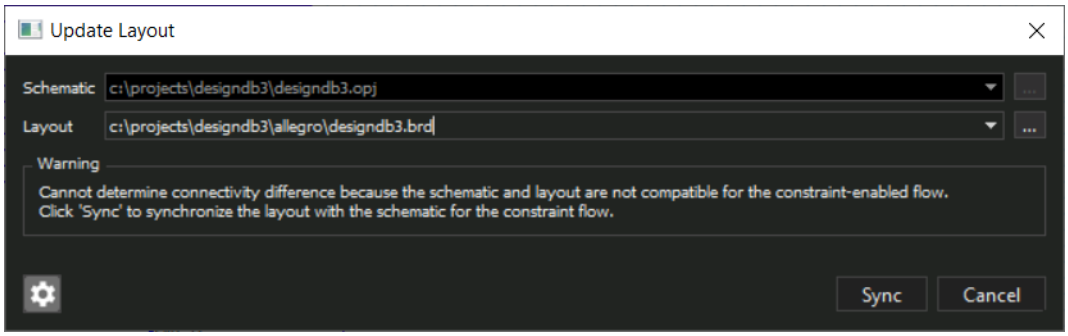
The Assign Voltage to Power Nets window opens.



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.

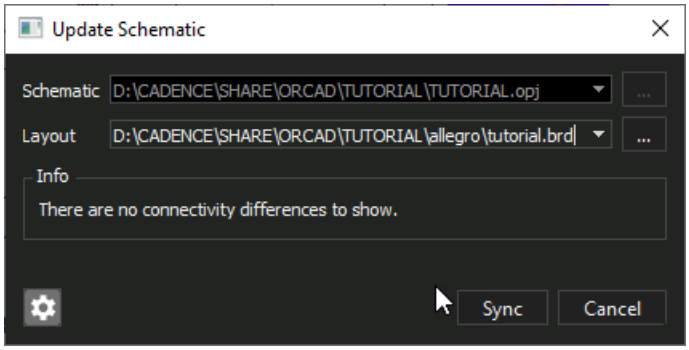


8. Click *Sync* to synchronize layout with schematic.

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

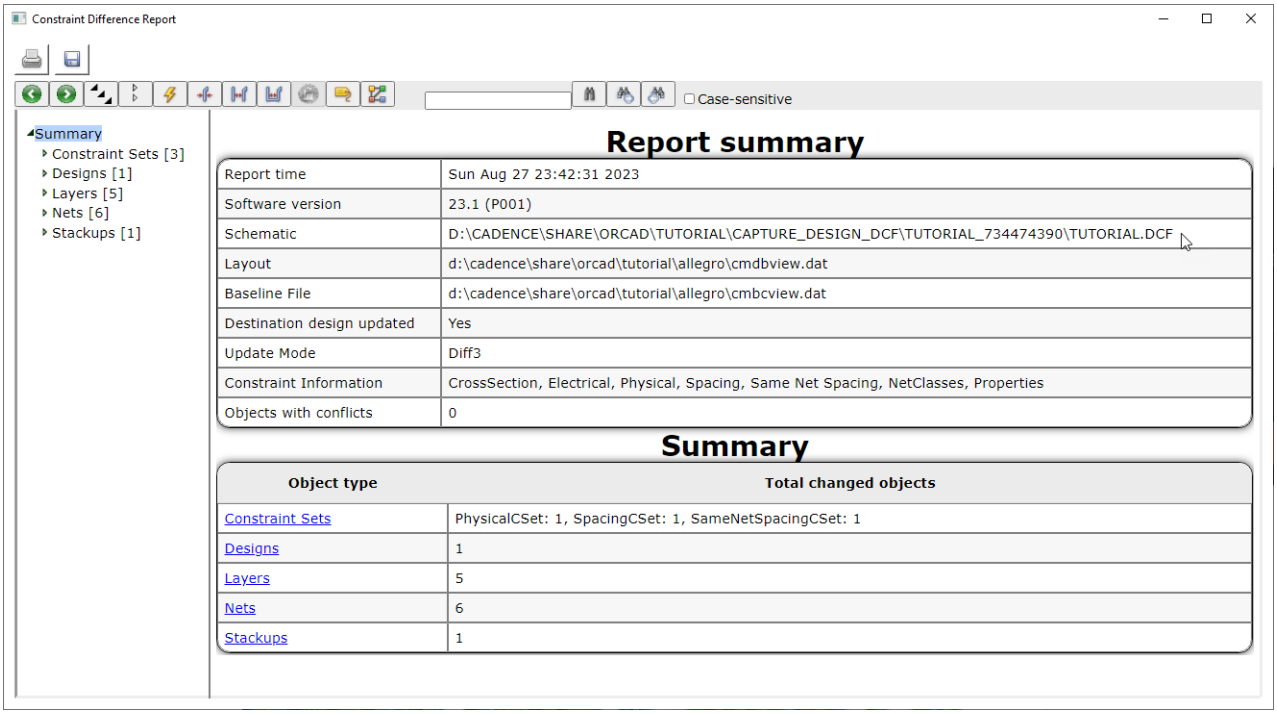
9. Select *PCB – Update Schematic* or click ().

The Update Schematic dialog box opens.



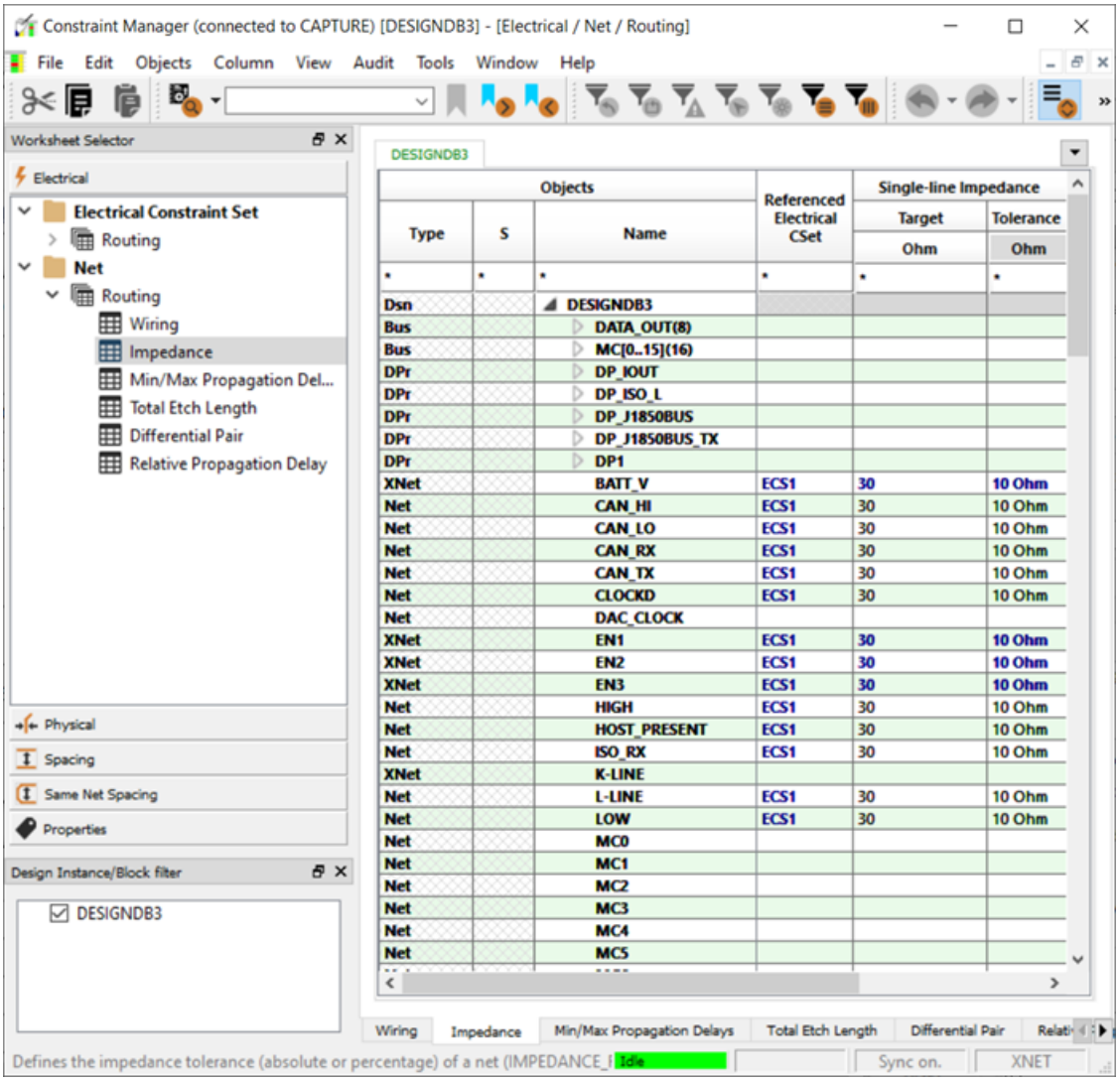
10. Click *Sync* to synchronize schematic with layout.

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



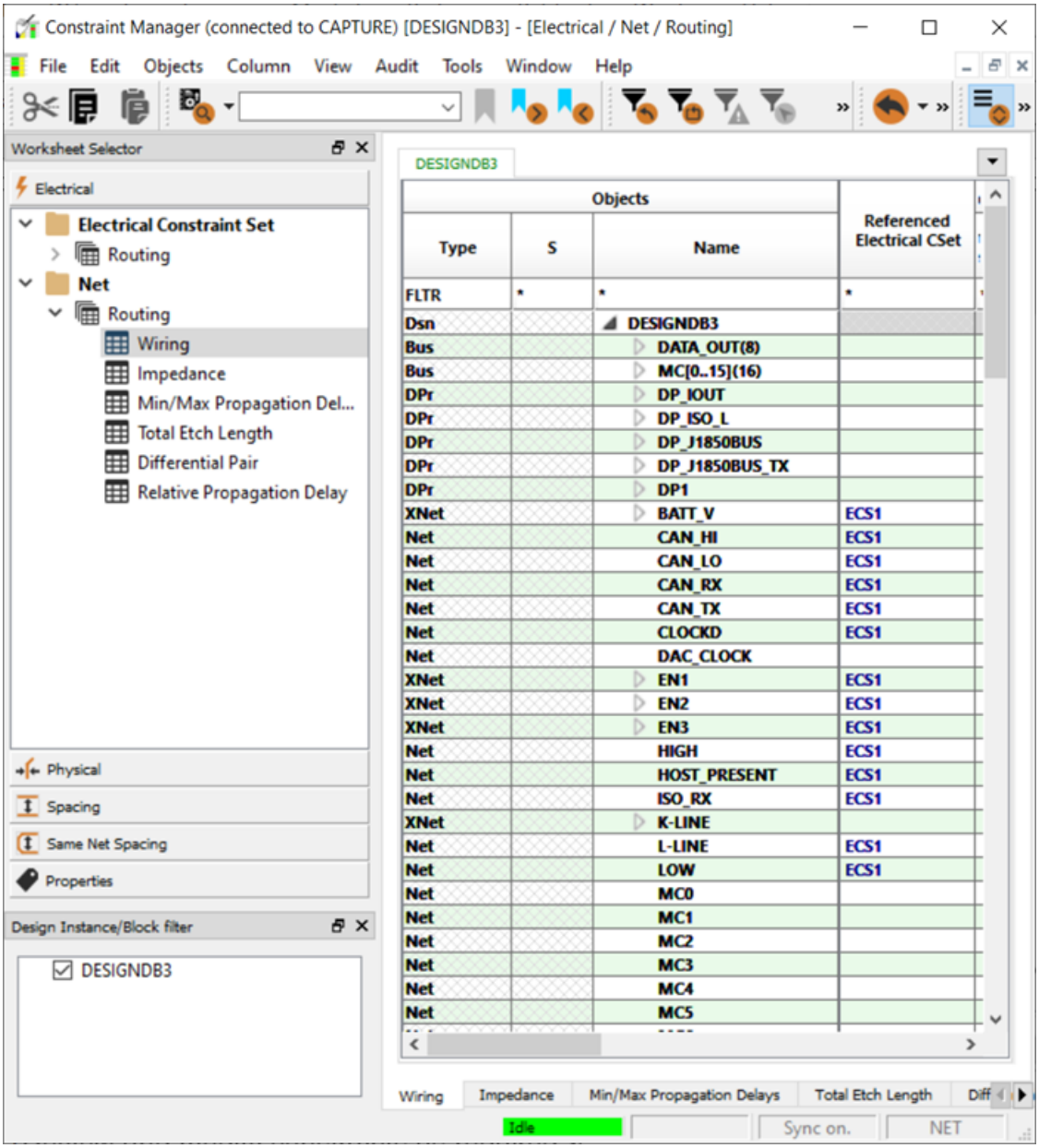
11. Click the Constraint Manager icon () to open Constraint Manager.

You can see the constraints in Constraint Manager.



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

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.

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

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.

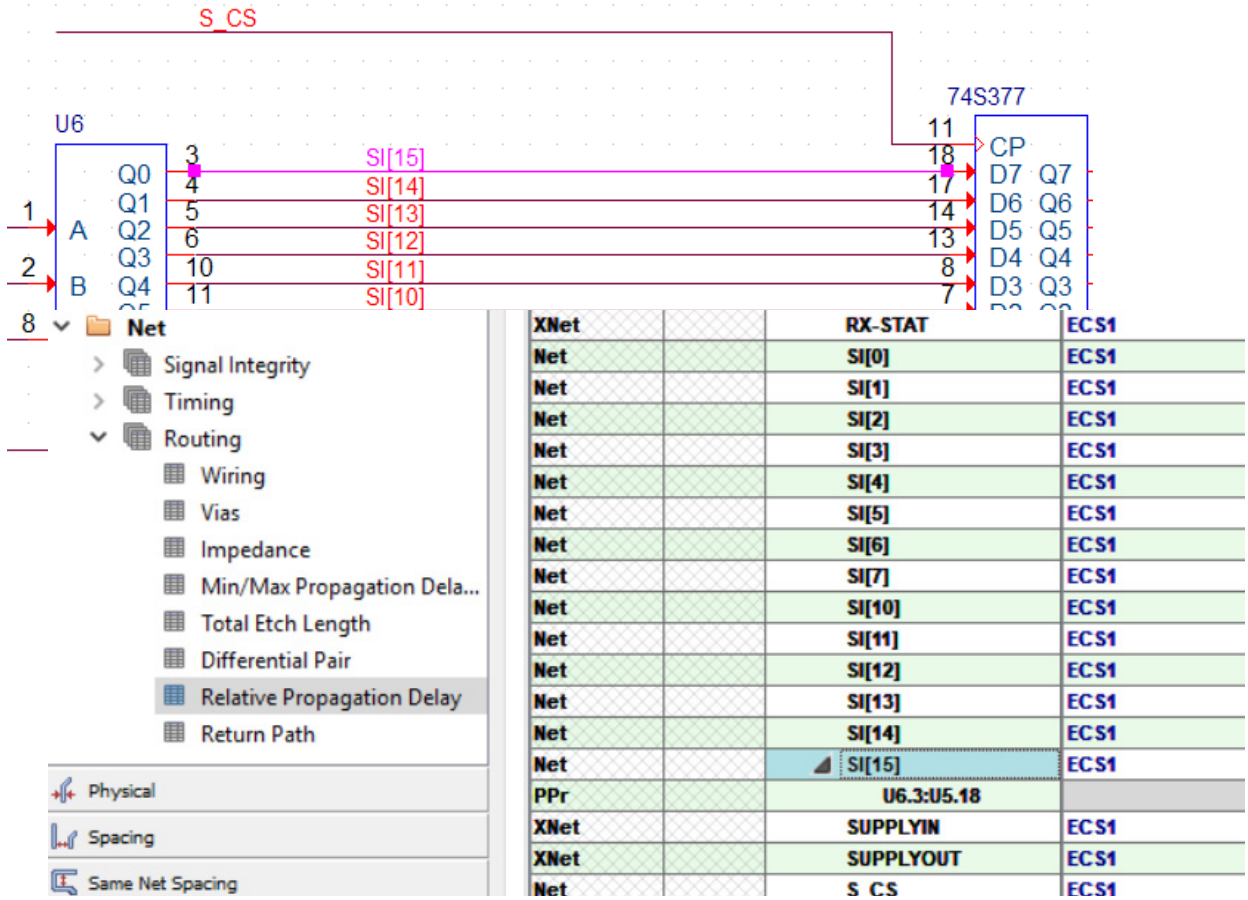
- If you copy the schematic design from a Constraint Manager-enabled project to a new design, the constraints are not copied.
- After enabling Constraint Manger in Capture, the following menu options are disabled in Capture. Their corresponding functions are now done only in Constraint Manager:
  - All 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.

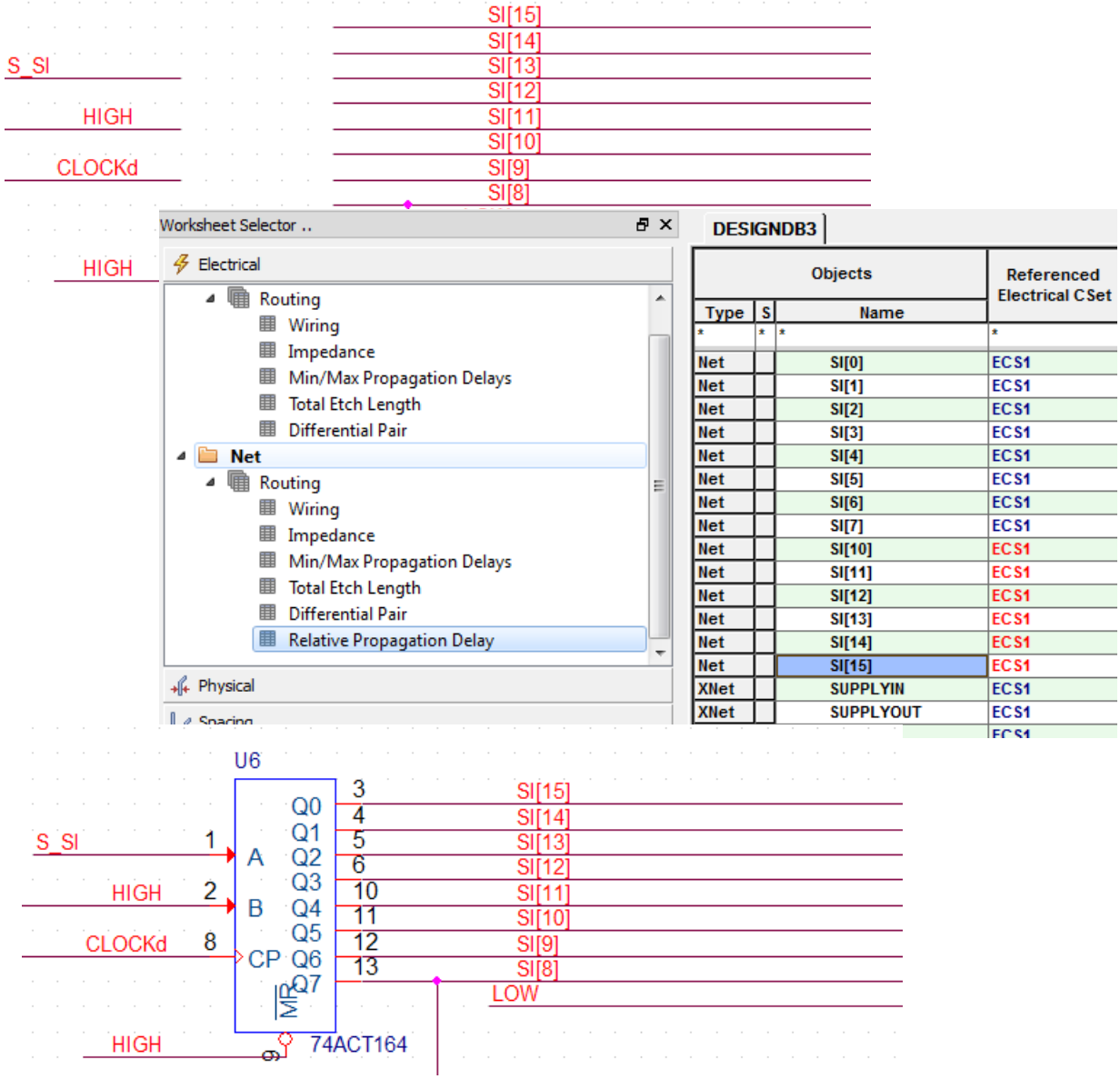
  Here is an example that suggests the changes you will observe in the undo operation.

  Pin Pairs after Replacing or Deleting Parts

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



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.



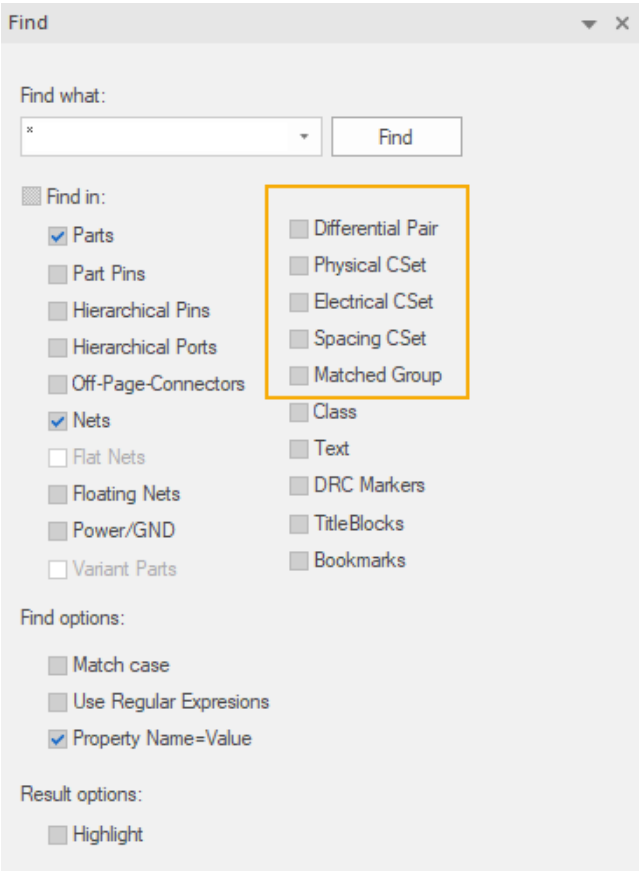
Finding Constraints

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.

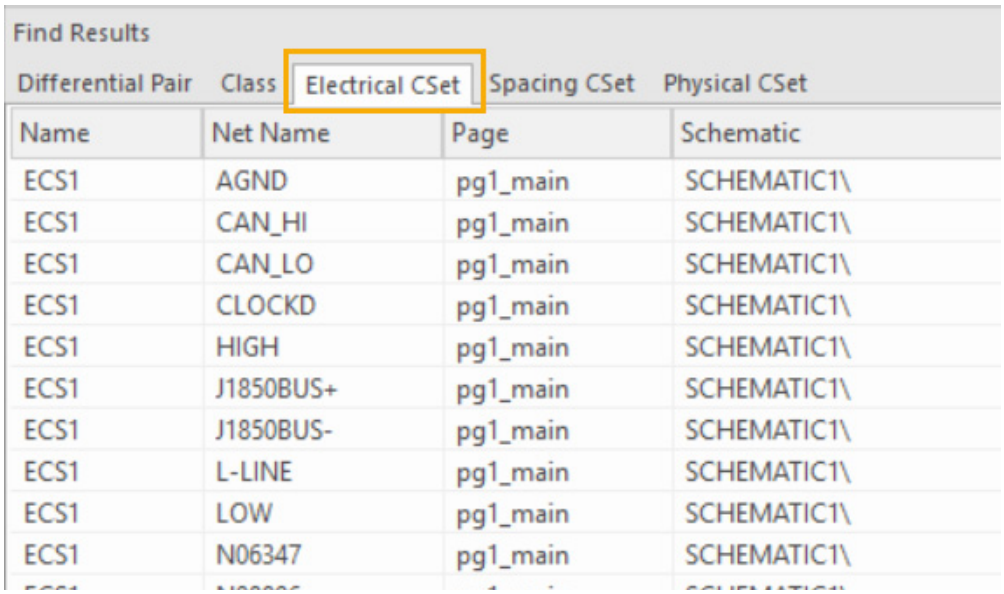


2. Select the required constraints.
3. Specify the search string in the search text box.
4. Click the *Find* button.

   The Find Results window opens displaying the search results.

5. Click the required constraint category.

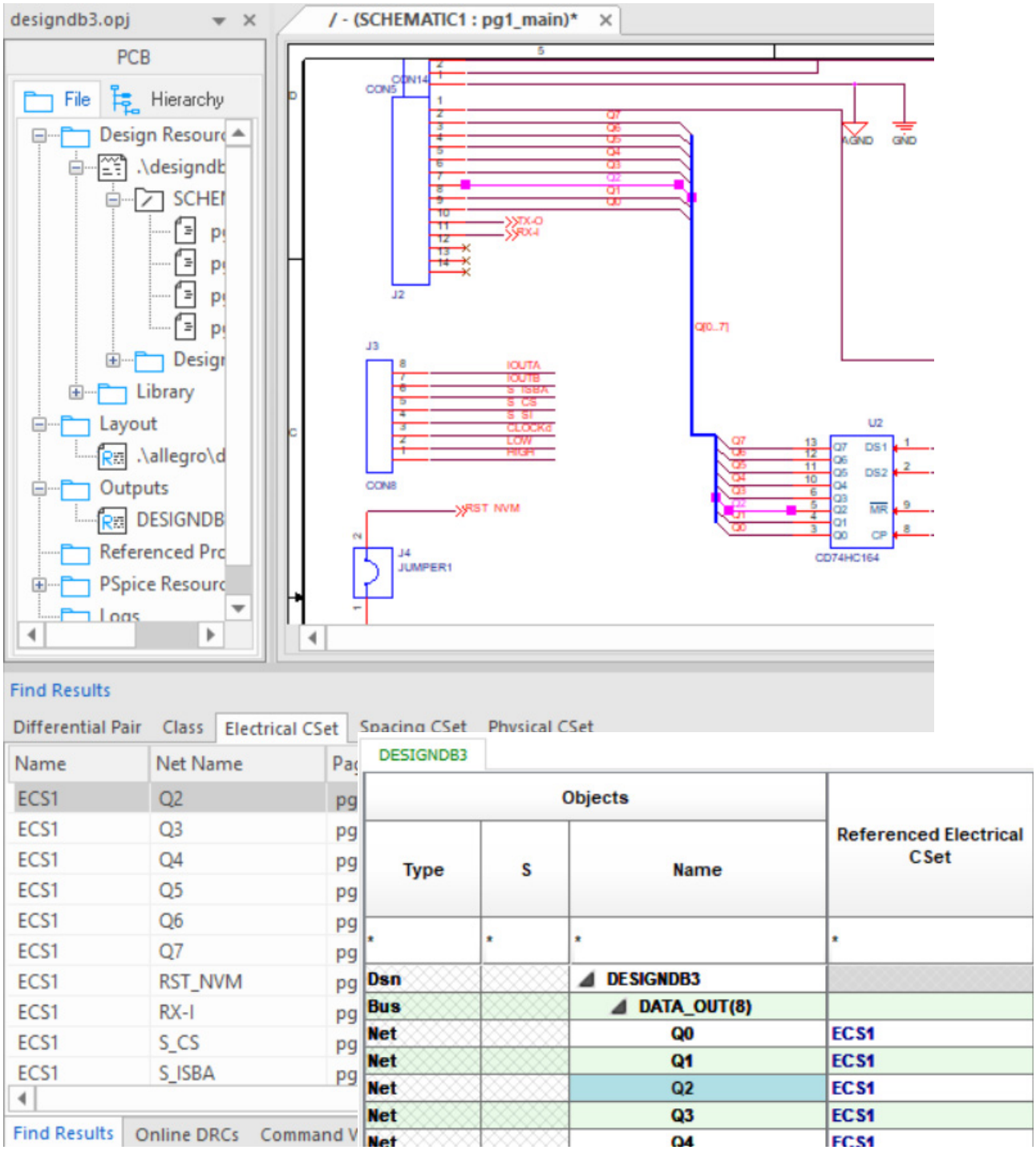
For example, click *Electrical CSet*.



6. Double-click a constraint.

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

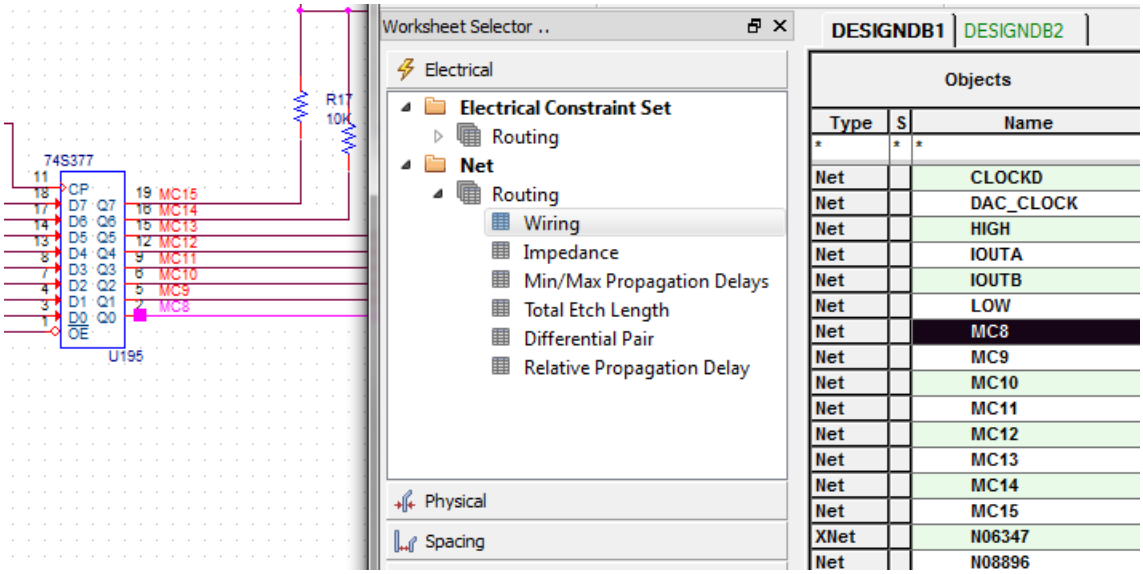
The Q2 net is selected in the schematic and in the Constraint Manager window.



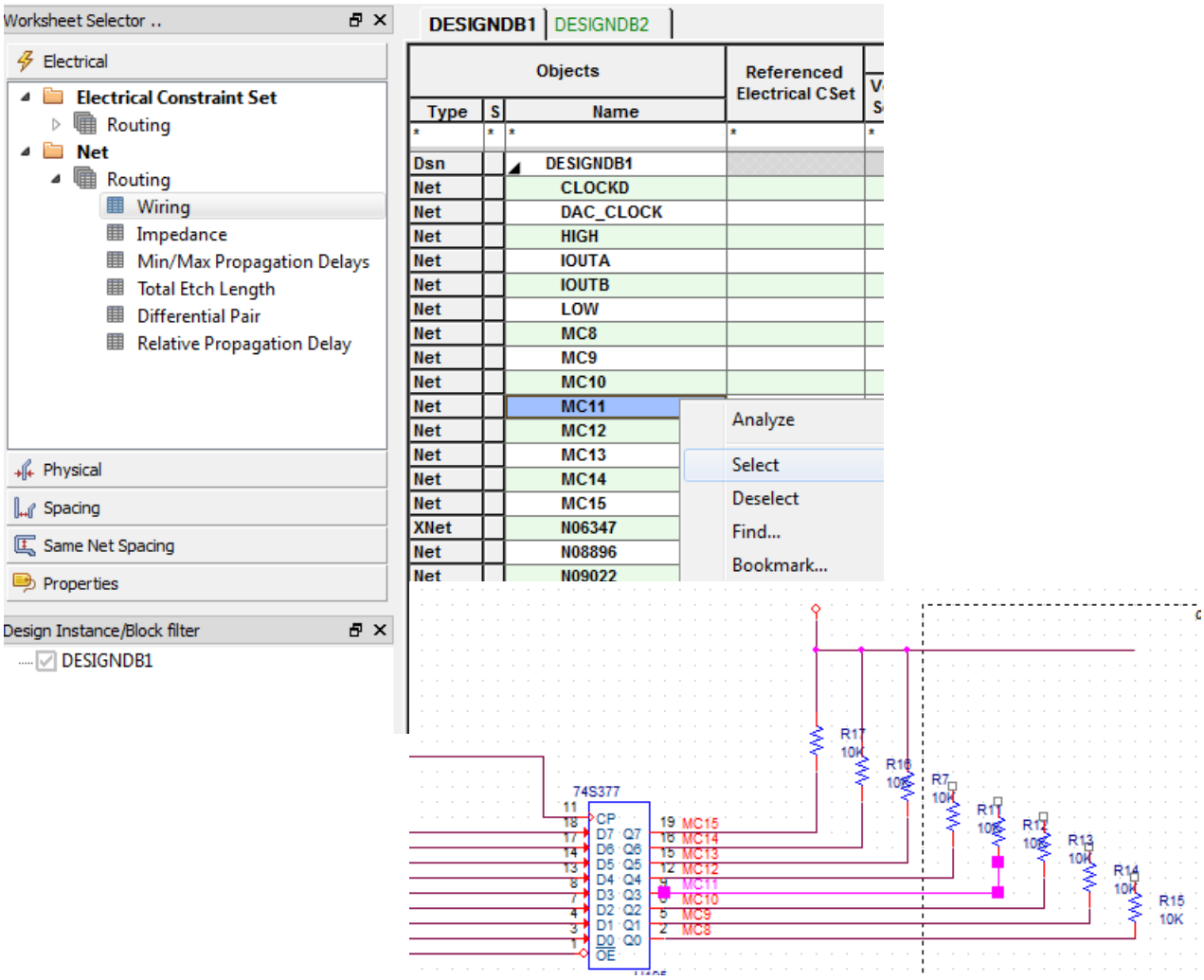
Cross-Probing of Design Objects

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.

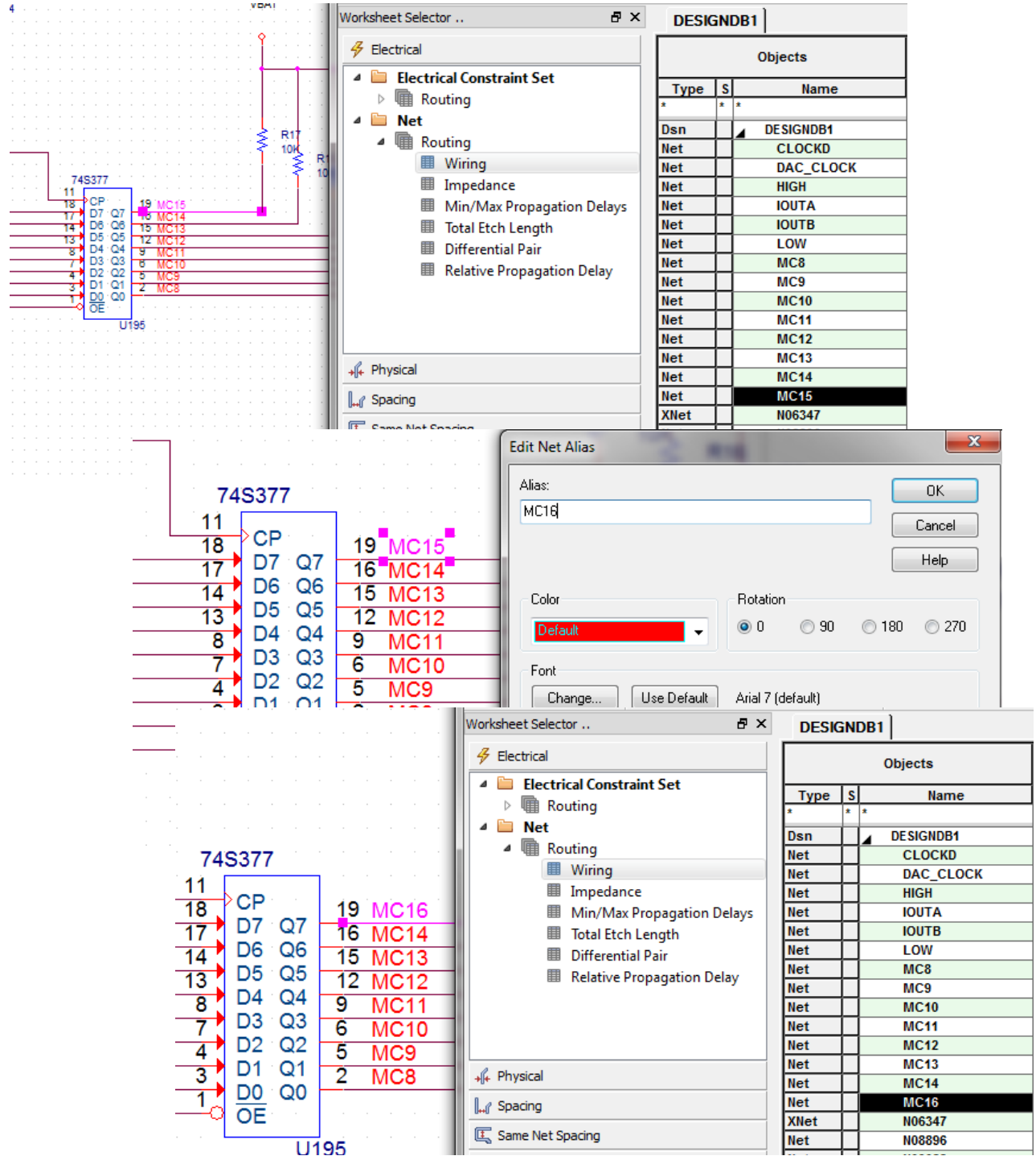


➡ Select a net name in Constraint Manager. Right-click this net name and choose *Select*. The corresponding net is highlighted in the schematic design.



Renaming Nets

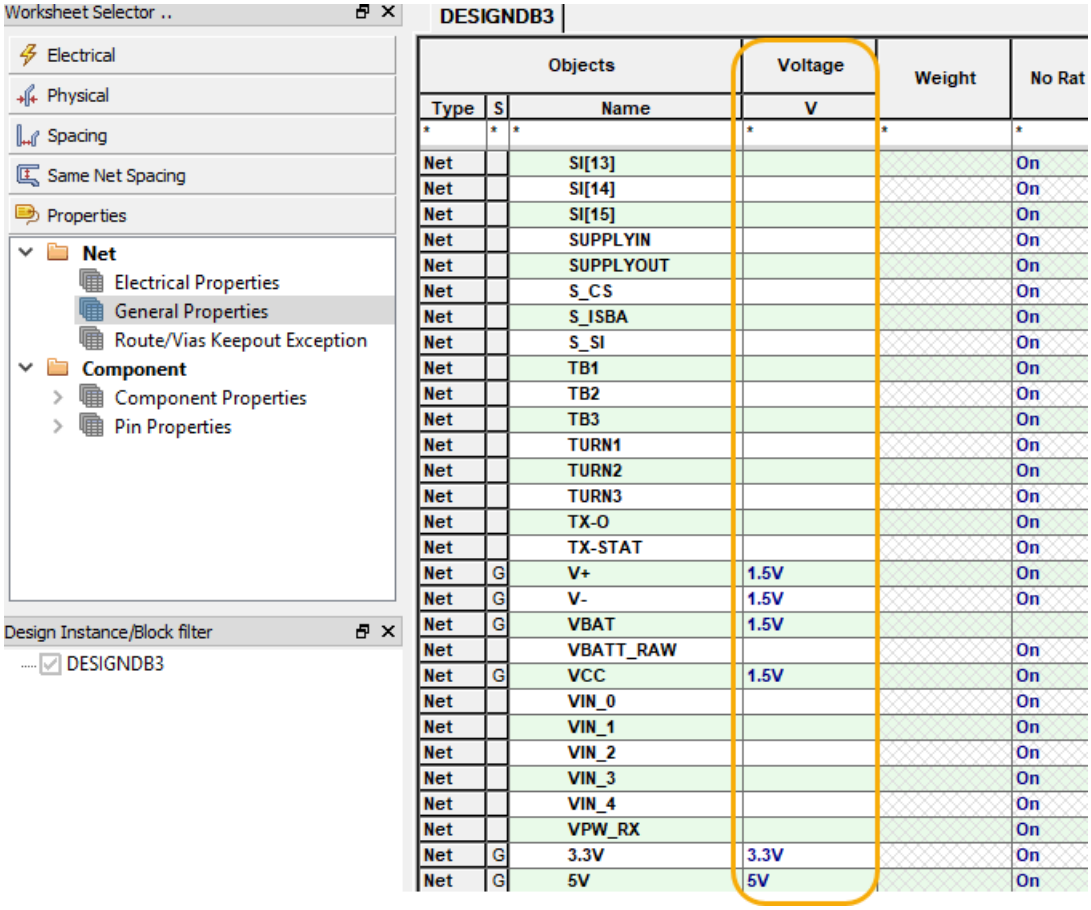
If you change the net name in the schematic, you can see the corresponding change in Constraint Manager.



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.



Objects			Voltage	Weight	No Rat
Type	S	Name	V		
*	*	*	*	*	*
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 Constrain Manager in a design in which a net traverses a passive, discrete device (resistor, inductor, or capacitor), XNets are automatically created.

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

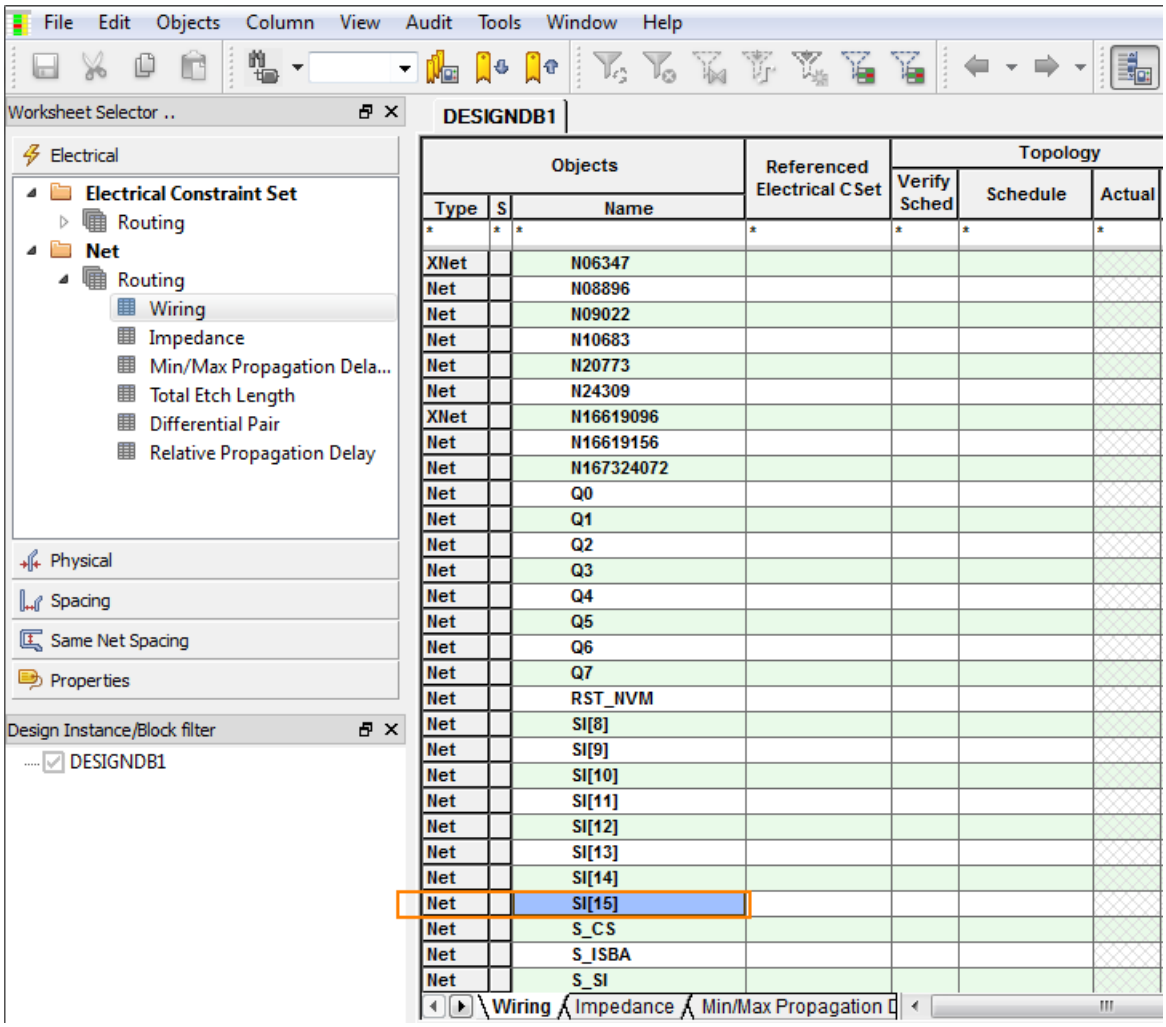
Creating XNets

To create an XNet, do the following:

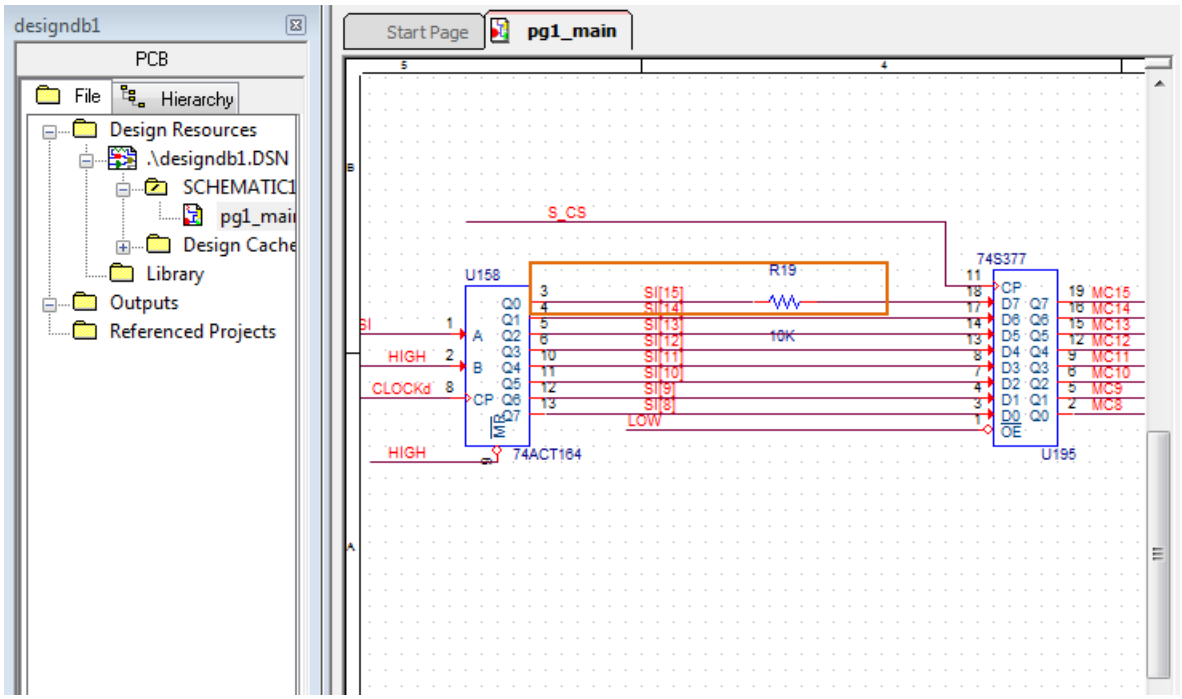
1. Open the required design in Capture.
2. Enable Constrain Manager for this design.

   The Constraint Manager window opens.
3. As an example:
   a. From the left pane, choose *Net – Routing – Wiring*.

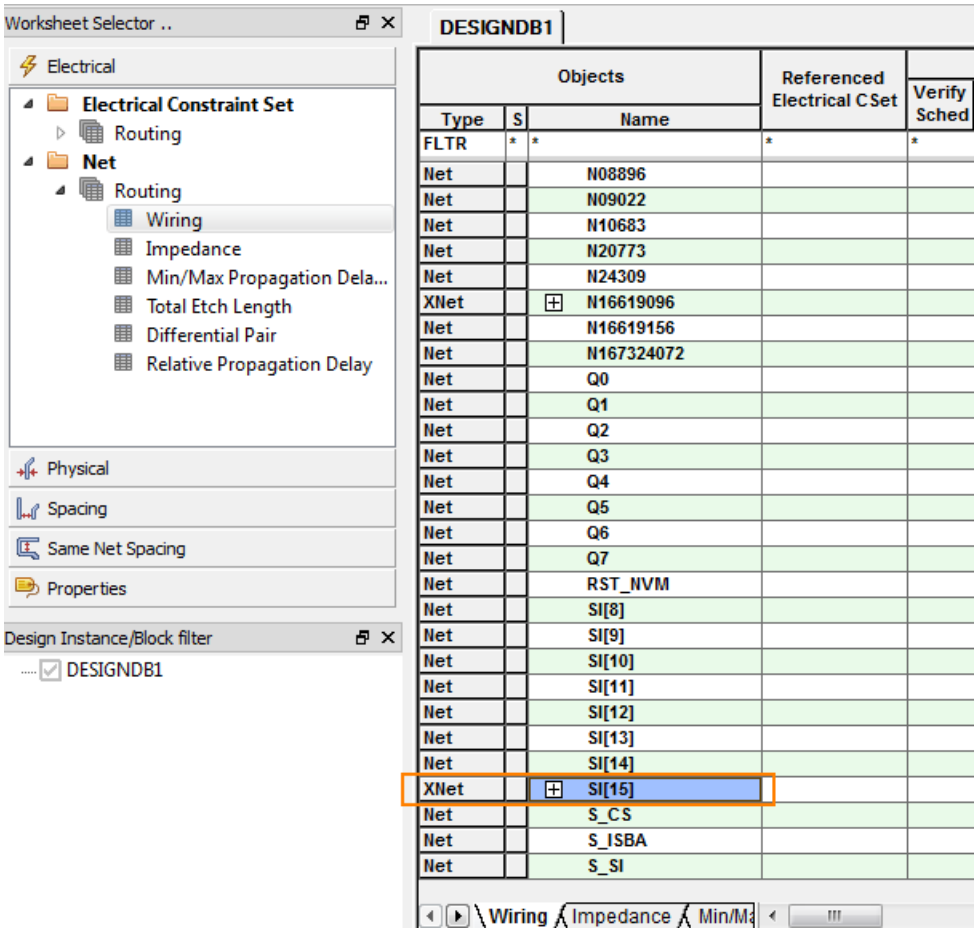
b. Check that the value corresponding to `SI[15]` in the `Type` column is `Net`.



c. Add a resistor to `SI[15]`.



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

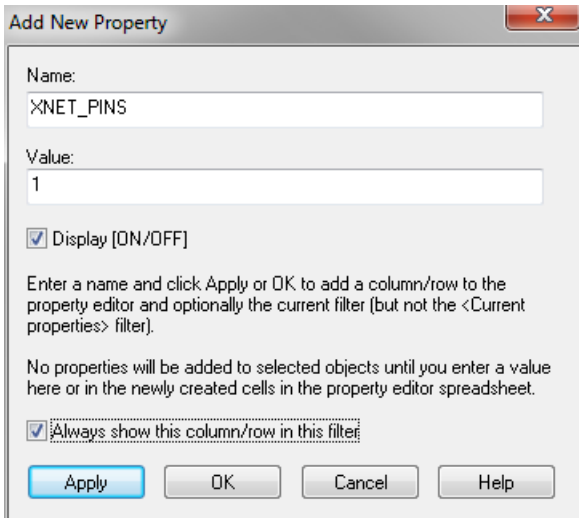


Creating an XNet using Non-Discrete Components

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*.

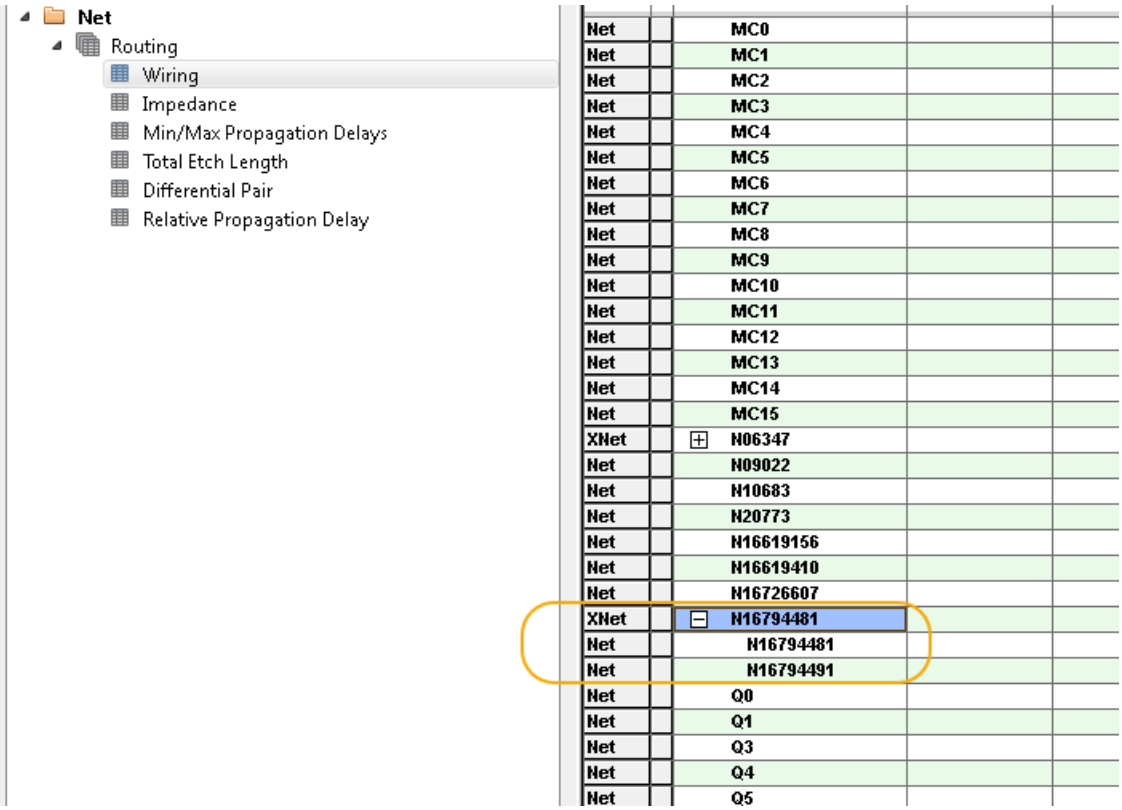
   You can see that the type has changed from `Net` to `XNet` in Constraint Manager.

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.

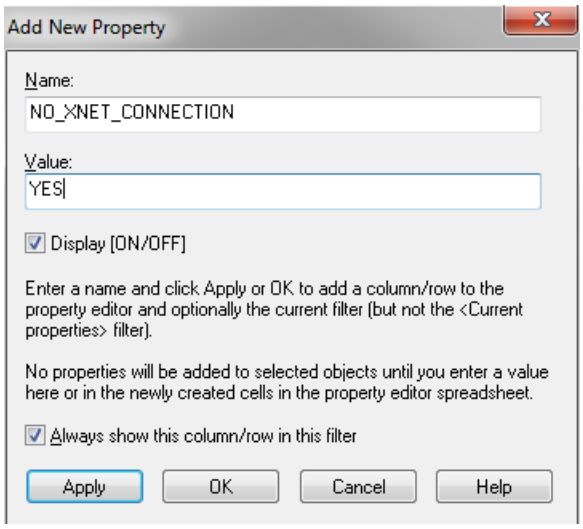
The XNet highlighted in the following example will be removed.



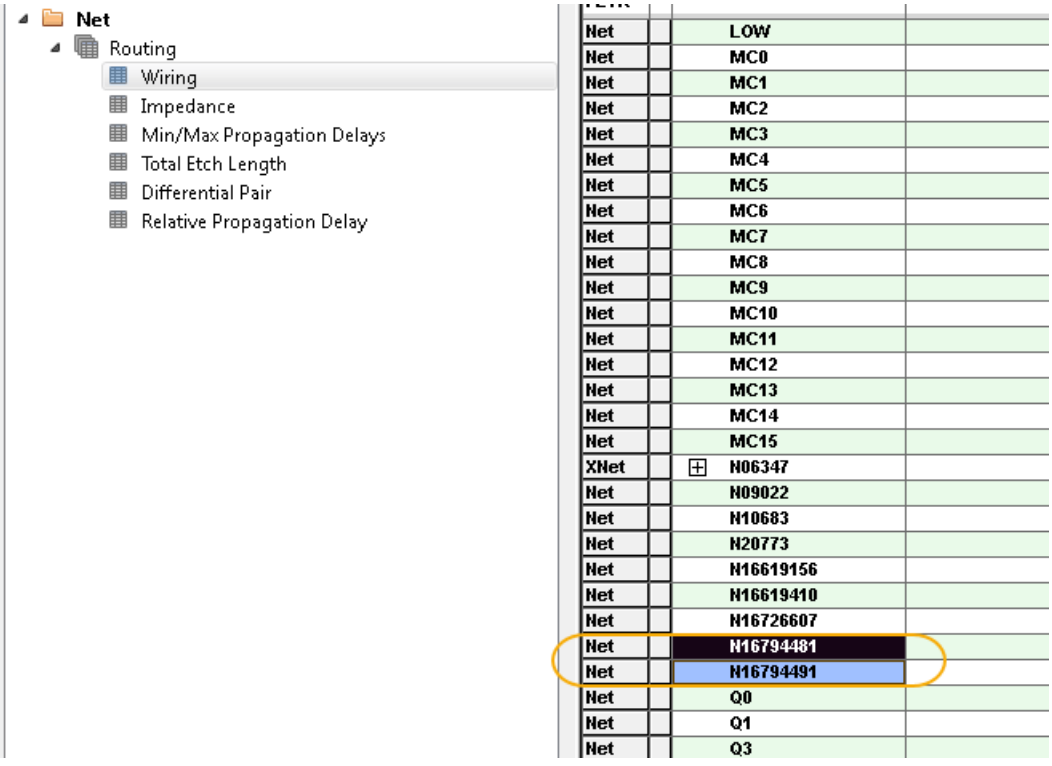
3. Right-click the discrete component and select *Edit Properties*.
4. Click *New Property*.

   The Add New Property window opens.

5. Set the `NO_XNET_CONNECTION` property to `YES`.



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



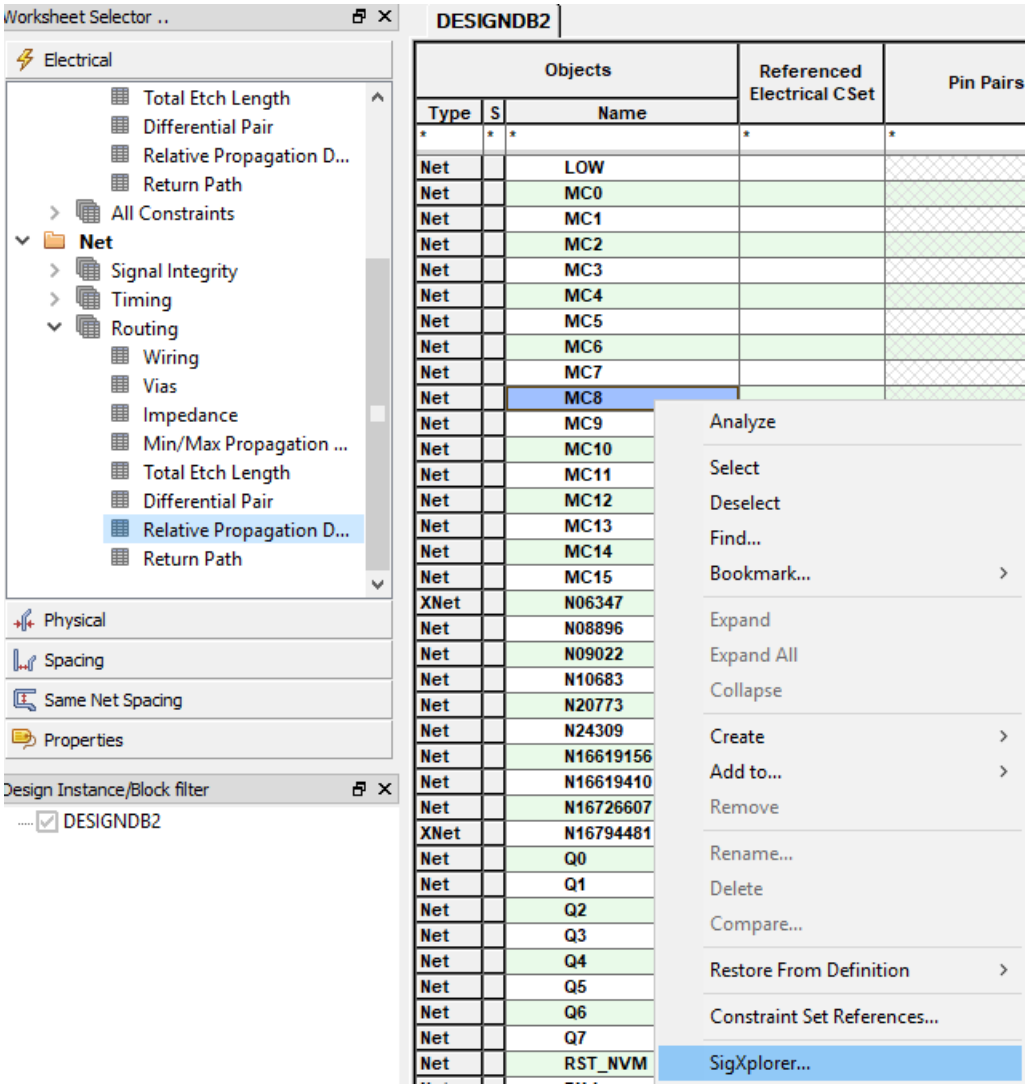
Performing Signal Analysis in Constraint Manager-Enabled Design

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 Constrain 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*.



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:

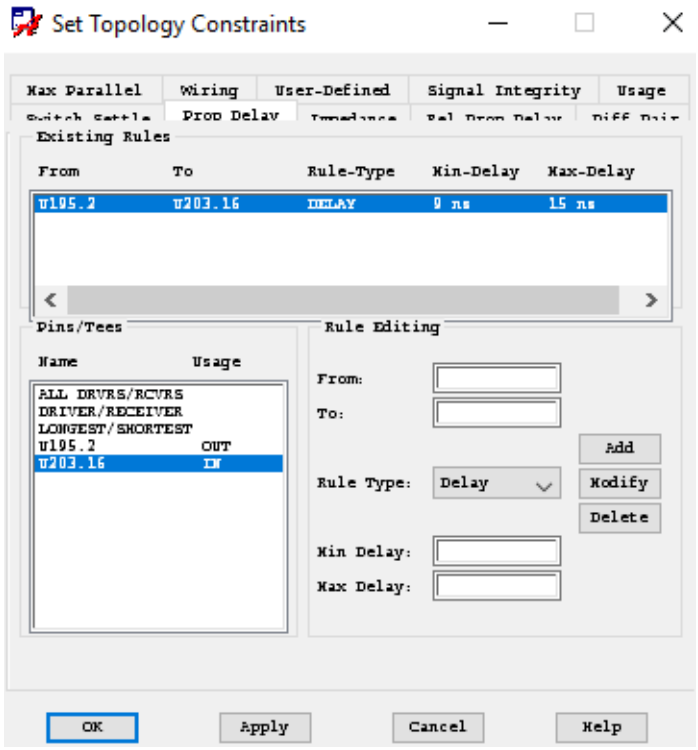
a. Click the *Prop Delay* tab.

b. Select the output and input pins.

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

d. Click *Add*.

e. A new rule is added in the *Existing Rules* section.



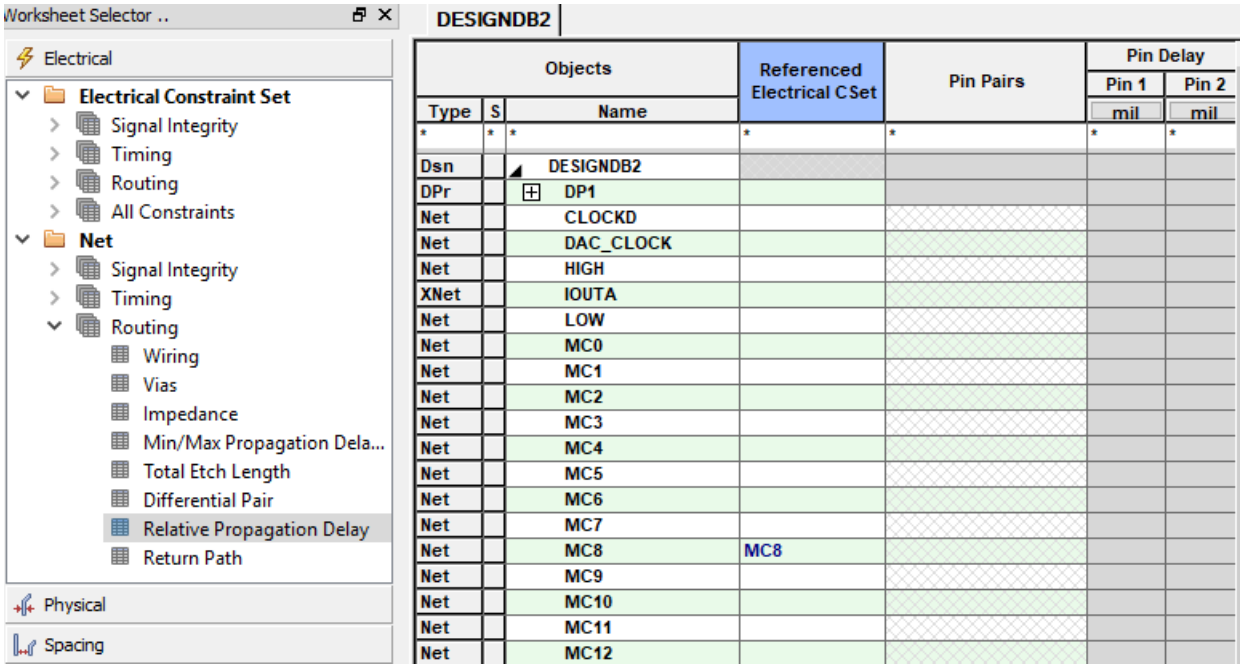
f. Click *Apply* and then *OK*.

g. Select *File – Update Constraint Manager*.

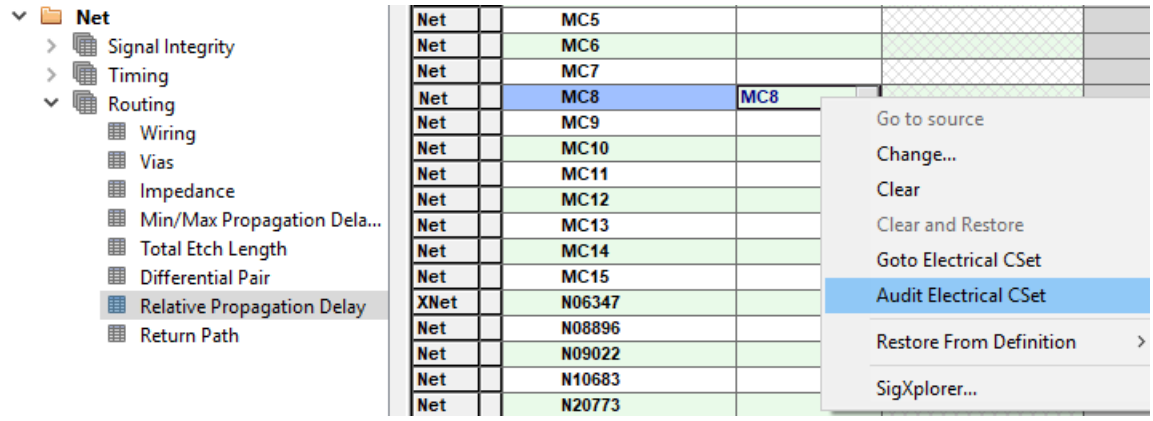
A message appears to confirm if the net can reference the Electrical Constraint Set.

h. Click *Yes*.

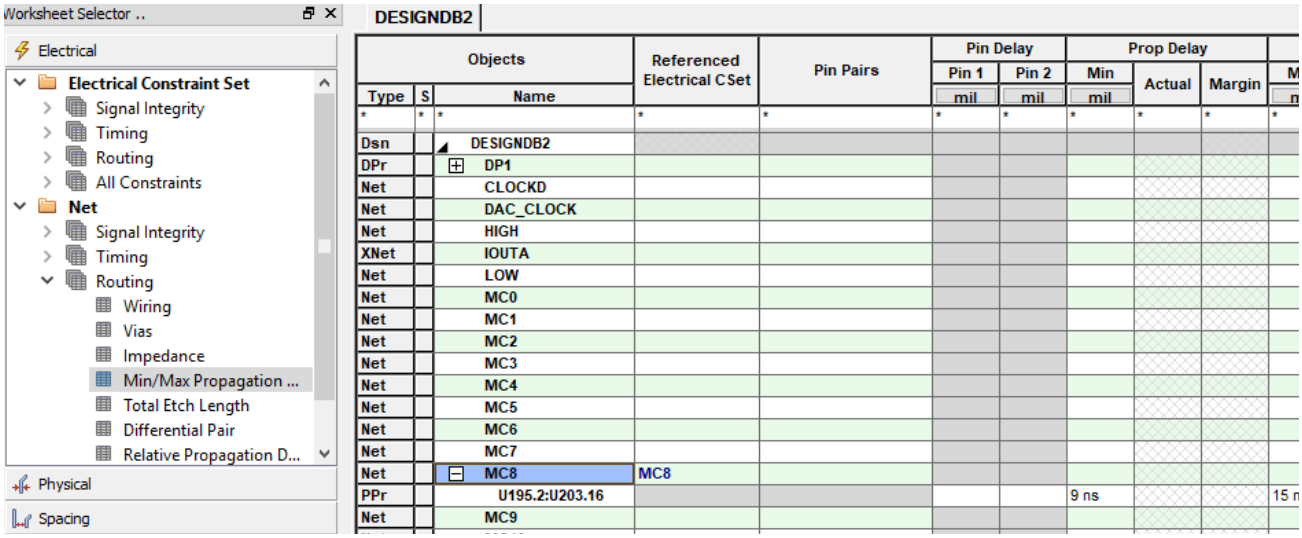
This constraint is now seen in the Constraint Manager window.



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



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



Important

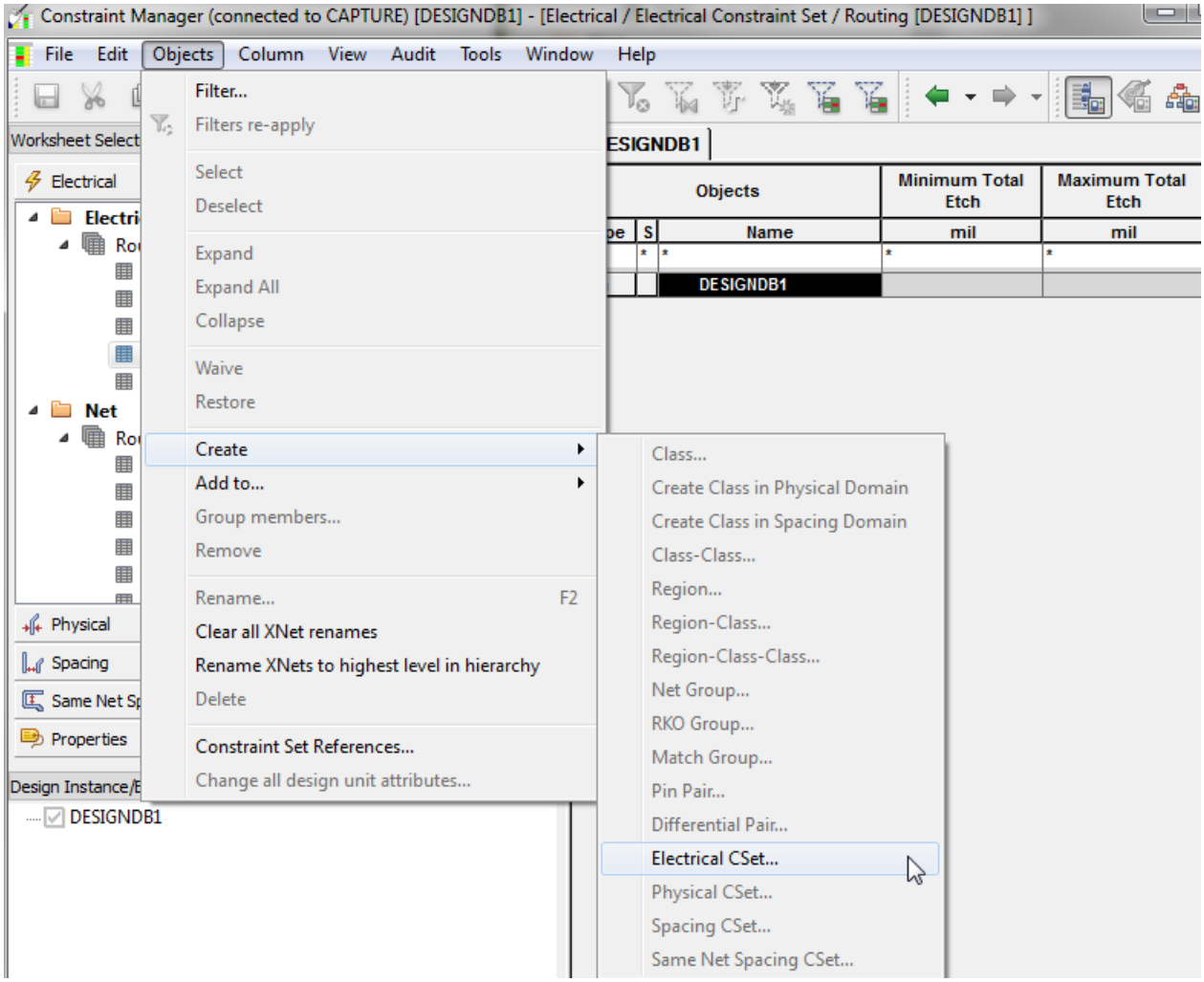
If the referenced Electrical CSet does not show the pin pair information, then *Electrical CSet Apply* feature will work with limitations. You need to upgrade to a higher license for this feature to function correctly.

Working with Electrical CSets

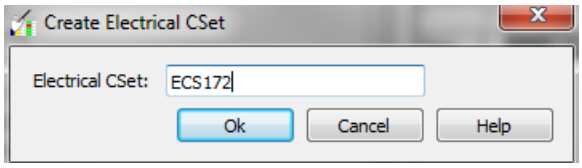
In Constraint Manager, you can define constraints on design objects using a Constraint Set (CSet). A CSet is a named and reusable collection of constraint values. CSet can then be assigned to a net in any of the *Net* worksheets.

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

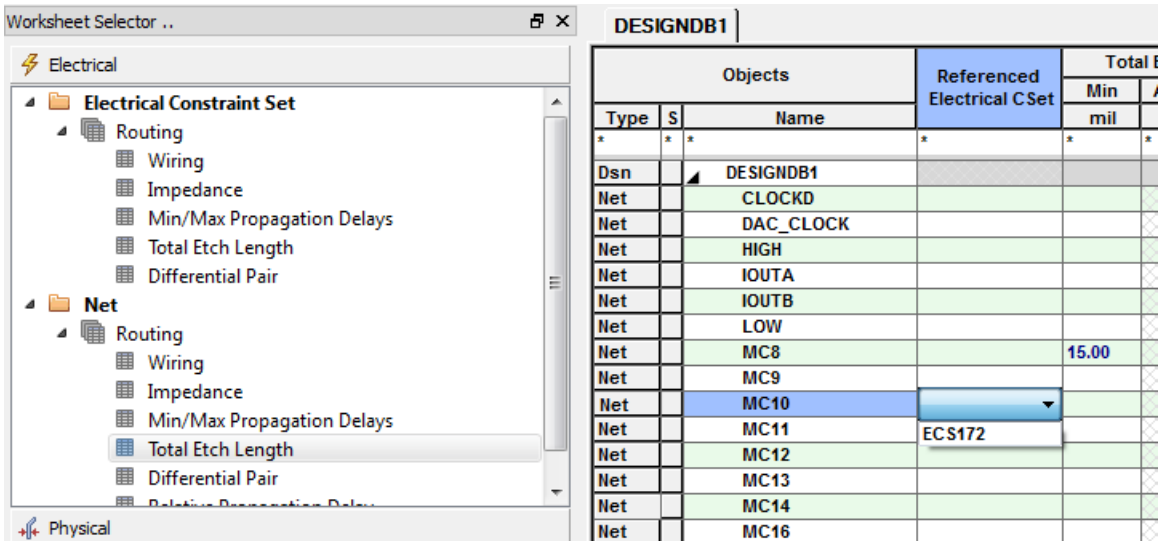
1. In a Constraint Manager-enabled design, open Constraint Manager.
2. In the left pane, select *Electrical Constraint Set – Routing – Total Etch Length*.
3. Choose *Objects – Create – Electrical CSet*.



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.



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

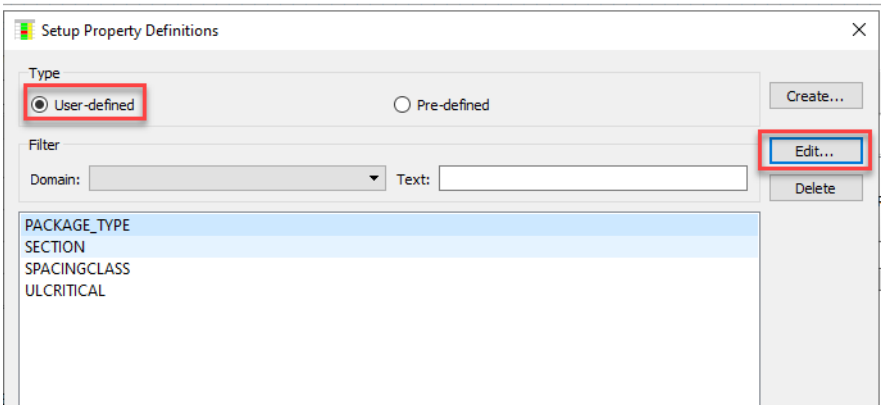
  Use the *Changes Only* mode if you need to merge concurrent constraint changes done in schematic and layout.

Updating Property Definition in Constraint Manager

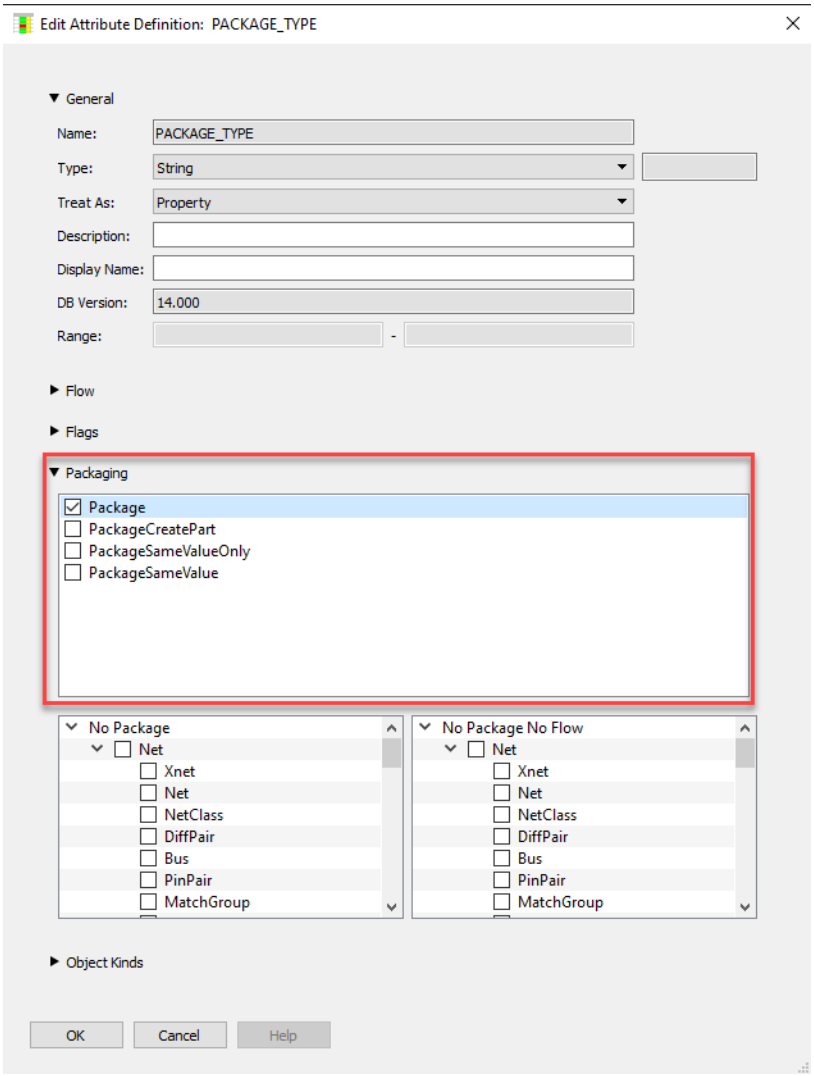
You can edit the property definition in Constraint Manager connected to Capture to prevent or enable creation of user-defined properties in Constraint Manager.

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.



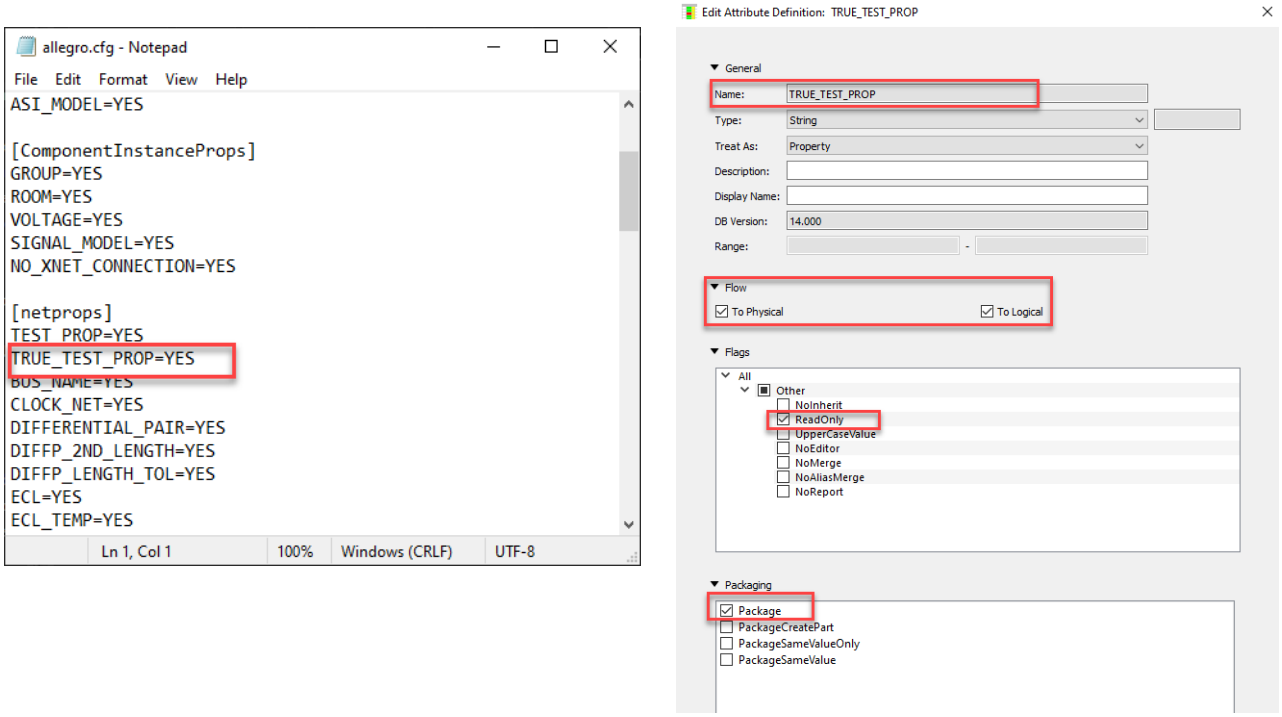
6. In the *Edit Attribute Definitions:* <property> dialog box, click *Packaging*, and select the *Package* check box.



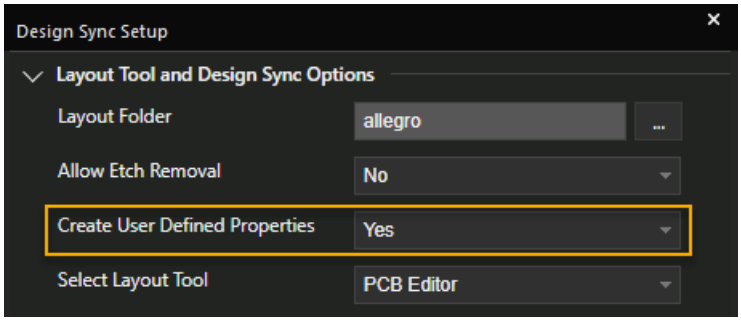
7. Click *OK* to save the changes.
8. Click *OK* to close the *Setup Properties Definitions* dialog box.

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.



- 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.