

# **Working With Net Groups and Port Groups**

**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.

Design Entry HDL 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.

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.

**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. 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.

Cadence is committed to using respectful language in our code and communications. We are also active in the removal and/or replacement of inappropriate language from existing content. This product documentation may however contain material that is no longer considered appropriate but still reflects long-standing industry terminology. Such content will be addressed at a time when the related software can be updated without end-user impact.

---

**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

---

<b>Net Groups</b>	7
<u>Creating Net Groups</u>	8
<u>Creating Net Group: Design Entry HDL</u>	9
<u>Creating Net Groups: Constraint Manager</u>	12
<u>Instantiating a Net Group</u>	13
<u>Capturing Connectivity</u>	15
<u>Locating a Net Group Instance</u>	16
<u>Adding Member Objects</u>	17
<u>Reverse Tapping</u>	19
<u>Removing Net Group Members</u>	19
<u>Adding Net Group Constraints</u>	22
<u>Deleting a Net Group</u>	22
<u>Net Groups in Hierarchical Designs</u>	24
<u>Exporting Net Group Data</u>	26
<u>Pre-QIR 9 Designs</u>	26
 <b>Net Groups in Physical Layout</b>	27
<u>Placing Components In PCB Editor</u>	27
<u>Quickplace Command</u>	27
<u>Place Manually</u>	31
<u>Net Group Visibility</u>	34
<u>NO_PCB_BUNDLE property</u>	36
<u>Locating Net Groups</u>	37
 <b>Port Groups</b>	39
<u>Creating a Port Group</u>	39
<u>Creating a Net Group</u>	39
<u>Instantiating a Net Group</u>	39
<u>Adding IOPORT</u>	40
<u>Running the Generate View Command</u>	41
<u>Using Port Groups</u>	42

## Working with NetGroups and PortGroups

---

<u>Adding Connectivity</u> .....	42
<u>Mapping Net Group Members</u> .....	43
<u>Modifying Port Group Membership</u> .....	44
<u>Deleting Port Groups</u> .....	47
<u>Generating Net Group and Port Group Mapping Report</u> .....	47
<u>Reusing Designs with Net Groups and Port Groups</u> .....	48

# Net Groups

To speed up design tasks such as capturing connectivity and routing, a new net object called Net Group has been introduced in the design tools from the Allegro platform. Net Groups are used to group net objects and provide a faster way of connecting them to components across schematic pages.

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 Groups can be of two type: single-level net groups or nested net groups. Net groups that have nets and buses as members are called single-level net groups. Net groups that have other net groups as members are nested net groups.

IGrp	S	
		DDR3_OBM_W0_LOWER (11)
Net		DDR3_OBM_W0_DQ0
Net		DDR3_OBM_W0_DQ1
Net		DDR3_OBM_W0_DQ2
Net		DDR3_OBM_W0_DQ3
Net		DDR3_OBM_W0_DQ4
Net		DDR3_OBM_W0_DQ5
Net		DDR3_OBM_W0_DQ6
Net		DDR3_OBM_W0_DQ7
Net		DDR3_OBM_W0_LDM
Net		DDR3_OBM_W0_LDQS
Net		DDR3_OBM_W0_LDQS#

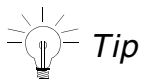
Single-Level Net Group with only Nets as members

IGrp	S	
		DDR3_OBM_W0 (2)
IGrp	S	DDR3_OBM_W0_LOWER (11)
IGrp	S	DDR3_OBM_W0_UPPER (11)

Nested Net Group with XNets, Buses, and Net Groups as members

**Note:** Nested Net Groups are only supported with higher tiered licenses.

Like all other net objects, you can apply constraints to net groups as well.



*Tip*

Net Groups must not be confused with Net Classes. Net Groups are a collection of physical net objects that supports grouping of signals as per the user requirements to ease design tasks such as connectivity, routing, and adding constraints. A Net Class on the other hand, is a collection of net objects used to define intra- and inter-class restrictions. A Net Group can be a member of a Net Class.

## Working with NetGroups and PortGroups

### Net Groups

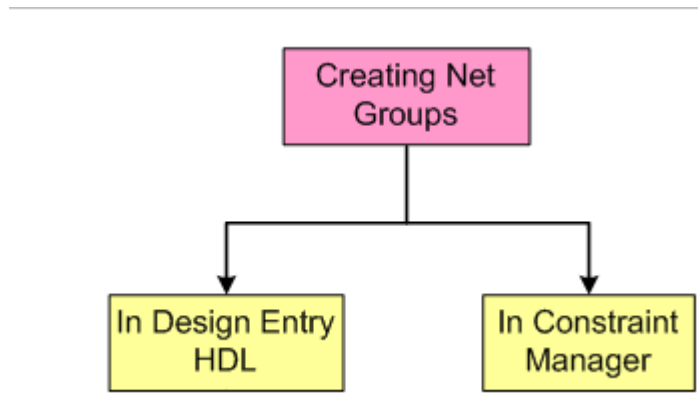
---

This document introduces you to Net Groups and the design tasks that can be performed on Net Groups. The topics covered in this chapter are:

- Creating Net Groups
- Instantiating a Net Group
- Adding Member Objects
- Removing Net Group Members
- Adding Net Group Constraints
- Exporting Net Group Data
- Net Groups in Physical Layout

## Creating Net Groups

Net group creation is supported in Design Entry HDL(DE-HDL) as well as in Constraint Manager. The following figure lists the methods used for creating net groups.



**Figure 1-1 Methods of Creating Net Group**



### Creating Net Group: Design Entry HDL

Net groups can be created in DE-HDL using the *Place* menu or the *Wire* menu. The *Place* menu is available only if the Windows mode is enabled.

To enable the Windows mode, refer to the *Windows Mode* section of *Allegro Design Entry HDL User Guide*.

**Note:** This feature is only available with the following licenses of Allegro Design Entry HDL:

- ☐ Allegro Design Authoring
- ☐ Allegro Design Entry HDL XL (Concept HDL Expert)
- ☐ Allegro Enterprise Authoring Solution
- ☐ Allegro Venture System Design Authoring
- ☐ Allegro Enterprise System Design Authoring

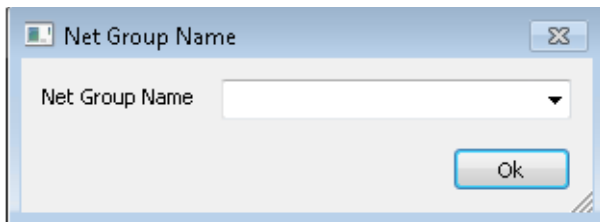
To create net groups:

1. Choose *Place – Net Group – Draw*.

Alternatively, you can also choose *Wire – Net Group – Draw*.

2. Draw (instantiate) a new net group on the schematic.

The *Net Group Name* dialog box is displayed.



3. In the *Net Group Name* field, specify the name for the net group to be created.

**Note:** Selecting a net group name from the drop-down list adds an instance of the already instantiated net group on the schematic.

4. Click OK.

A new net group is listed in the Interface Browser, under *Net Group(Schematic)*. This is an empty net group that has no members assigned to it.

5. Save the design.

## Working with NetGroups and PortGroups

### Net Groups

---

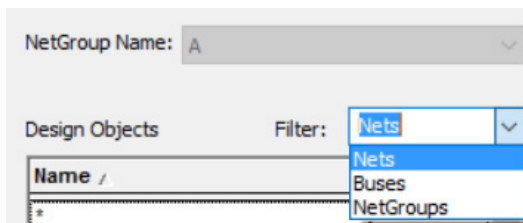
6. To add members to the new net group, right-click the net group instance and select *Edit Net Group*.

Alternatively, you can also right-click the net group name in Interface Browser and select *Edit*.

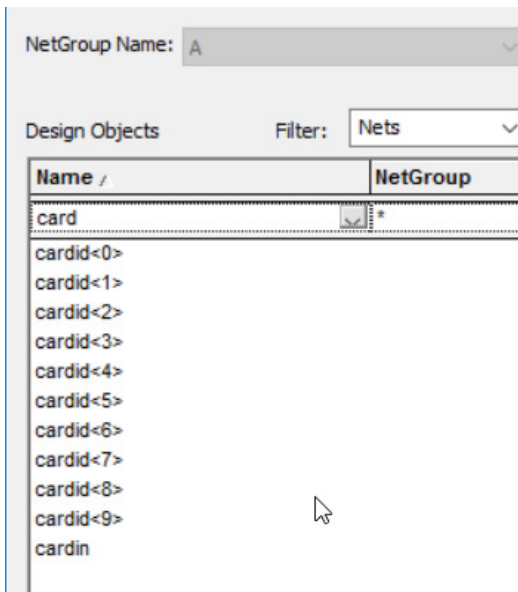
The *Edit NetGroup Membership* dialog box is displayed.

7. From the *Design Objects* list, select the net members to be added to the net group, and click the right-arrow button.

To display only one type of net objects in the *Design Objects* list, use the *Filter* drop-down list.



To filter the net objects as per their names or types use the filter combo boxes.



8. Click *OK*.

## Working with NetGroups and PortGroups

### Net Groups

---

#### *Important*

A net object can be a member of one Net Group only. If you add a net object of an existing net group to a new net group, an error message appears indicating that the net object will be removed from the original net group.

**Note:** When creating net groups in DE-HDL, differential pairs and Xnets cannot be added to the net group. Only individual nets, constituting these net objects, are added to the net group.

### Creating Net Groups: Constraint Manager

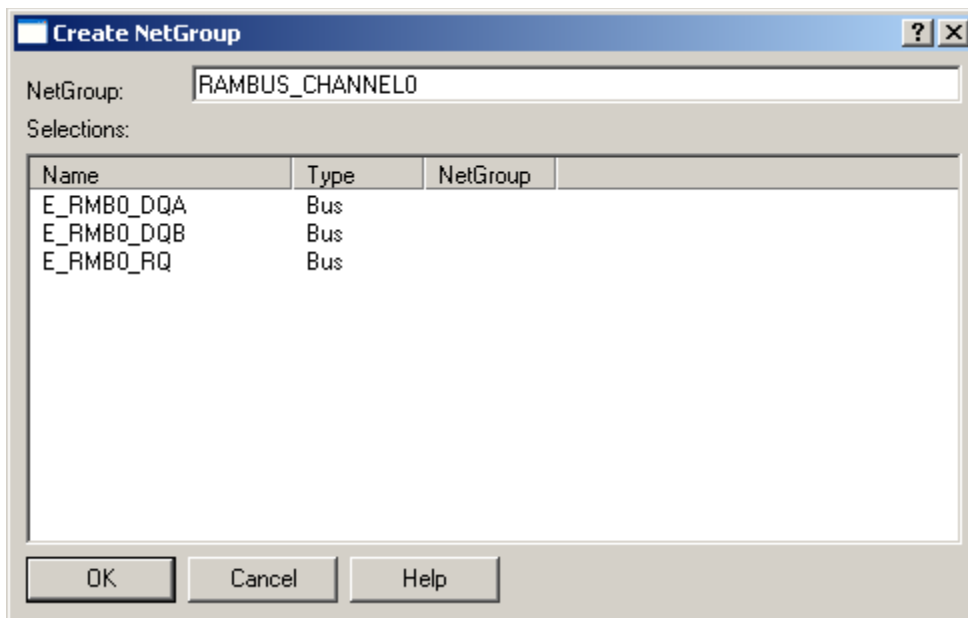
Net groups can be created in Constraint Manager launched from design capture tools as well as from PCB Editor.

1. In Constraint Manager, select the net objects to be included in a net group.
2. Right-click and choose *Create – Net Group*.

Alternatively, you can also choose *Objects – Create – NetGroup*.

The Create Net Group dialog appears with the selected net objects listed as member objects.

3. Specify the Net Group name.



4. Click OK.

The new Net Group is created and appears in the worksheet with Object Type as NGrp. The net object selected in the previous step appears as a member of the Net Group.

NGrp		RAMBUS_CHANNEL0 (3)
Bus	+	E_RMB0_DQA (9)
Bus	+	E_RMB0_DQB (9)
Bus	+	E_RMB0_RQ (8)

If you now open the Design Entry HDL window, the new Net Group name is listed in the Interface Browser as well.

#### Important

Net groups created in Constraint Manager can have differential pairs and Xnets as member objects.

#### Caution

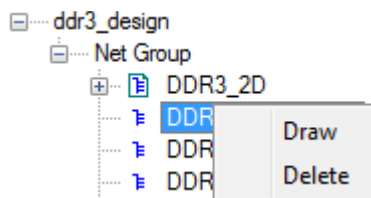
**While working on hierarchical designs, it is possible that all net groups created in Constraint Manager are not visible in the Interface Browser window. For details, see [Net Groups in Hierarchical Designs](#).**

## Instantiating a Net Group

Before you can use net groups to capture connectivity, you need to instantiate the net group on the schematic canvas. To instantiate an existing net group, you draw it on the schematic, using one of the following methods.

### Method 1

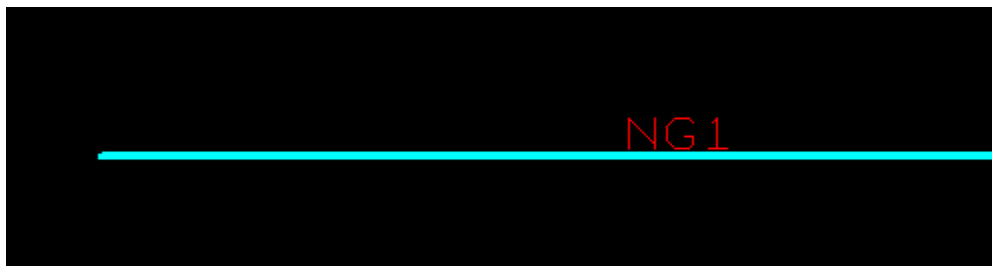
- a. In the Interface Browser window, right-click on the net group name and select *Draw*.



As you move the cursor on the schematic canvas, it changes to a crosshair cursor.

- b. Place the cursor on the schematic canvas and draw the Net Group as required.

By default, Net Groups are drawn as thick aqua colored lines as shown in the following figure.



## Working with NetGroups and PortGroups

### Net Groups

- c. Save the design.

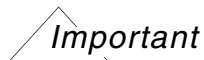


#### Caution

***It is recommended that net groups created in Constraint Manager must not be instantiated on the schematic. Only the net groups created in DE-HDL should be instantiated on the schematic canvas.***

### Method 2

1. Choose *Place – Net Group – Draw*.
2. Draw an instance of the net group on the schematic.  
The Net Group Name dialog box appears.
3. From the drop-down list, select the name of the net group to be instantiated and click OK.
4. Save the design.

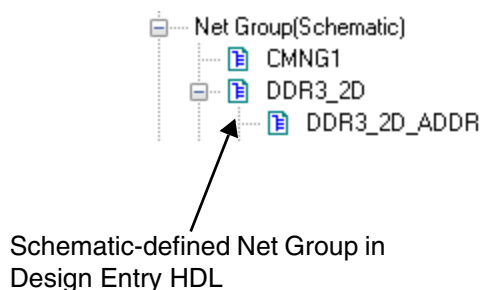


#### Important

In a design, there can only be one instance of a net group. This implies that a net group with the same name drawn on multiple schematic pages is treated as a single net group.

### Schematic-Defined Net Groups

A net group instantiated on the schematic canvas, is a schematic-defined net group. The icon next to the net group name in the Interface Browser is modified to indicate the change in the net group status. In Constraint Manager, the letter 'S' is used to indicate such schematic-defined net groups.



NGrp	S	+	CMNG1 (1)
NGrp	S	-	DDR3_S0D_ADDR (2)

Schematic-defined Net Group In Constraint Manager

## Working with NetGroups and PortGroups

### Net Groups

---

Schematic-defined net groups cannot be modified or deleted in Constraint Manager. This implies that you cannot use Constraint Manager to add or remove members from a schematic-defined net group. You can only specify constraints on them, or add these as members of other net groups, using the *Add to Net Group* command.

**Note:** On the schematic canvas, you can use reverse-tapping to add new members to a schematic-defined net group.

### Capturing Connectivity

One of the advantages of using Net Groups is that it speeds up the task of capturing design connectivity. To connect design components using net groups, you first need to draw the net group on a schematic page and then tap-out the required net objects and connect them to component pin(s). In case of large designs that span multiple schematic pages, you can draw the net group on each page and tap-out member net objects as required. Tapped-out net objects are auto-named. As you do not need to enter net names time and again, this speeds up the design capture process and also reduces the possibility of connectivity errors because of incorrect signal names.

To connect the member of a net group to a component pin, tap-out the required net and connect it to the component pin.

1. Draw the Net Group on the schematic page.
2. Right-click on the Net Group instance and choose *Tap Member*.
3. From the sub-menu, choose the net to be connected to the pin.

The wire is attached to the cursor.

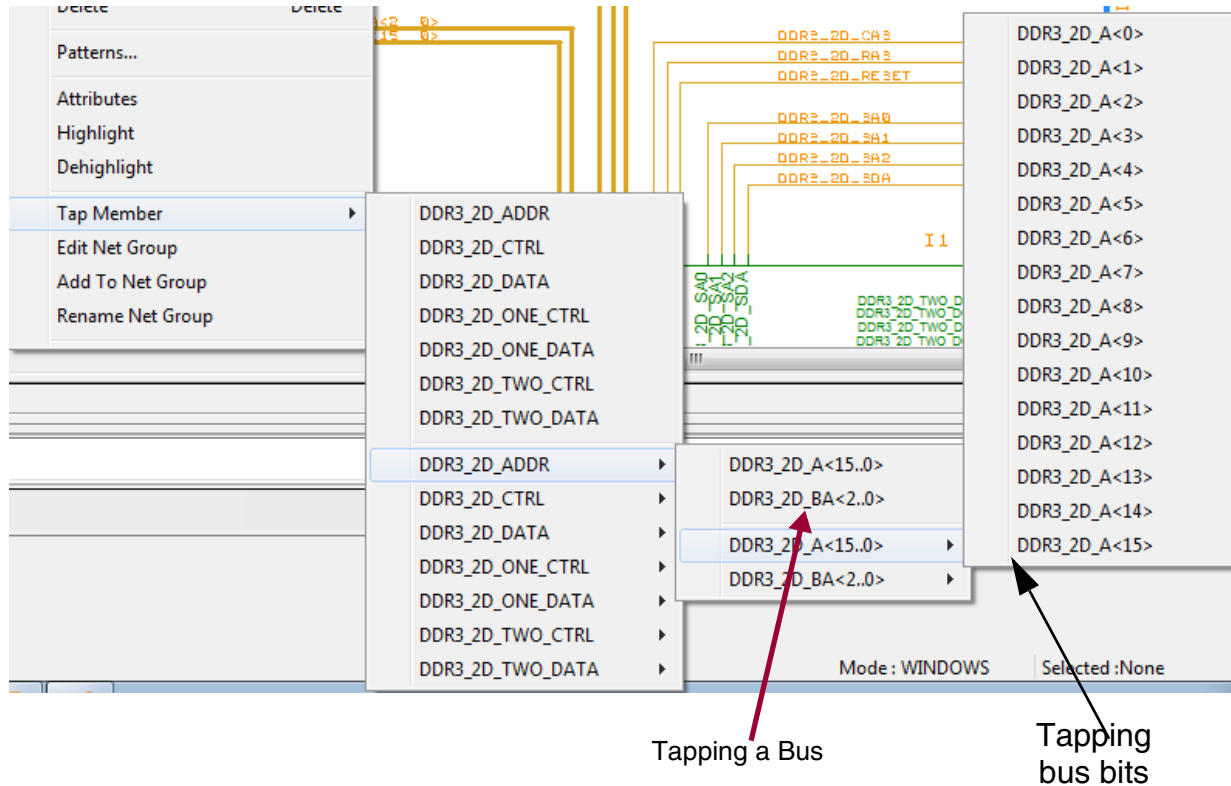
4. Click on the required component pin to establish connectivity, and save the design.

Similarly, you can tap-out other nets to capture connectivity.

## Working with NetGroups and PortGroups

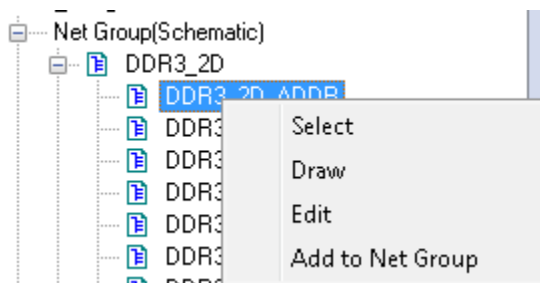
### Net Groups

If you have included a bus as a member of a net group, you have the option to tap the complete bus as well the individual nets.



## Locating a Net Group Instance

In a large multi-page schematic design, to locate the Net Group instance, right-click on the Net Group name in the Interface Browser and choose *Select*.



**Note:** The Select menu command is available only for schematic-defined net groups.



### Cross-probing Net Groups

When you select a net group only the member nets are highlighted.

## Adding Member Objects

If a net group is not instantiated in a schematic, you can add new member objects to it only using Constraint Manager. For net groups instantiated on the schematic, member nets can be added only in Design Entry HDL.

Guidelines for adding new members to an existing net group:

- A net object can be a member of one net group only.
- A net group can be a member of another net group (nested net group).



#### *Tip*

The net group that has no member object is an empty net group. While capturing a design, you can create empty net groups and add members to it as and when required.

A net object can be added to an existing net group using one of the following methods.

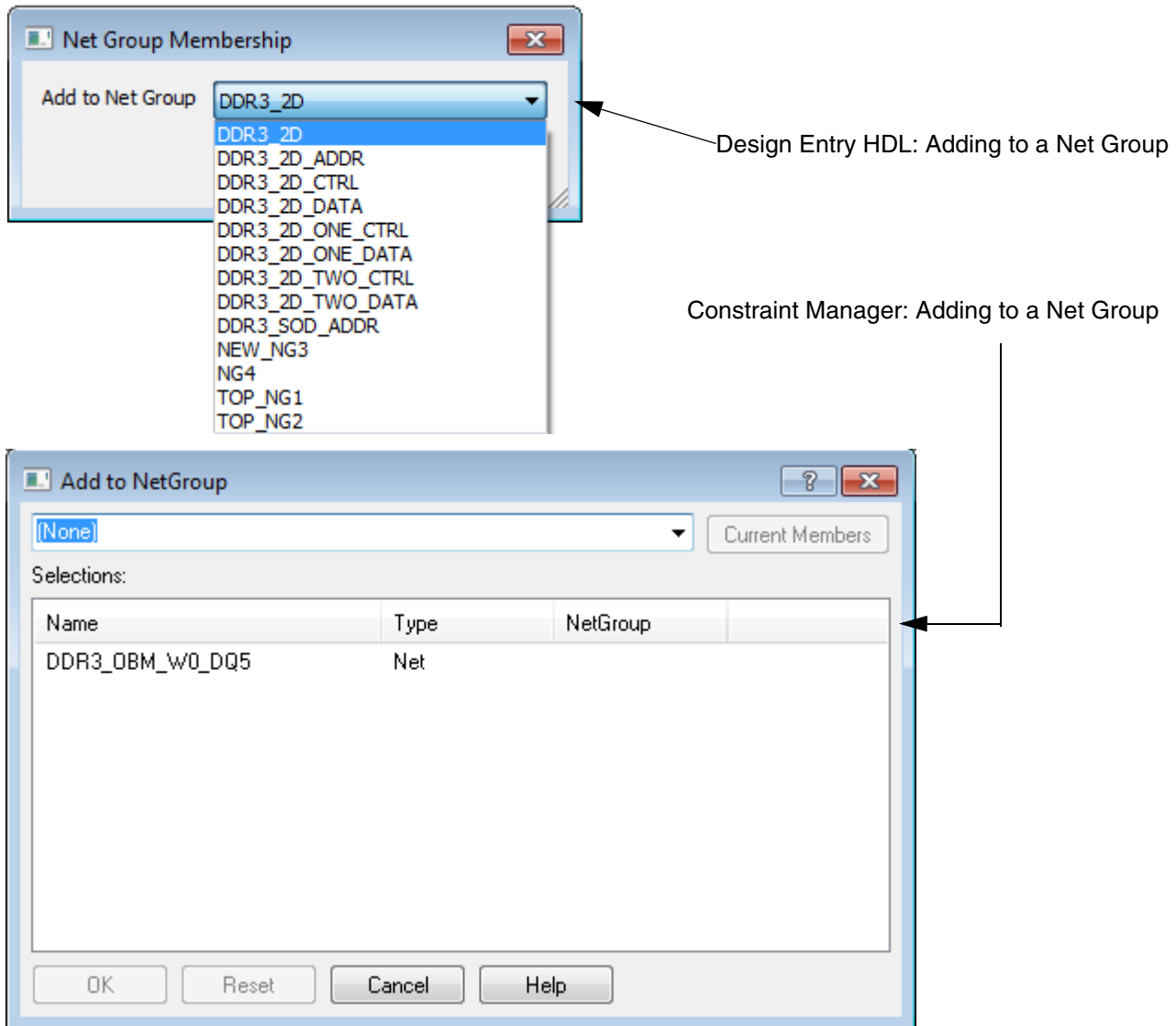
- Using the Add To Net Group command
- By Reverse Tapping (Schematic Only)

## Working with NetGroups and PortGroups

### Net Groups

#### Add To Net Group

1. Right-click on the net object and choose *Add To Net Group*.



**Figure 1-2 Adding Members To a Net Group**

2. In the displayed dialog box, click the down-arrow button to display the list of net groups available in the design.

While working in DE-HDL, only the net groups that are instantiated (schematic-defined) on the schematic are listed in the drop-down list. If there are no schematic-defined net groups, you receive a message stating that no valid net groups are available.

3. From the drop-down list, select the net group to which the selected net object is to be added and click *OK*.

The Net Group membership is modified in DE-HDL as well as in Constraint Manager.

## Reverse Tapping

While working on a schematic, you can add existing nets to a net group by dragging the net on the Net Group instance. This is called reverse tapping. In this mechanism, you add a net object to a net group by creating connectivity between the net object and the net group instance on the schematic canvas.

Steps to add a net group member using reverse tapping:

1. Select the net object to be added to the net group.
2. Drag the net object on the net group drawn on the schematic.

The net object is attached to the Net Group as shown in the following figure.



3. Save the design.

The Net Group definition is updated. To verify this, right-click on the Net Group instance on the schematic page, and select *Tap Member*. The net object appears in the sub-menu.

## Removing Net Group Members

The [Creating Net Groups](#) and [Adding Member Objects](#) sections, respectively, covered the methods using which you can create new net groups and add new member objects to existing Net Groups. This topic shows how you can modify the definition of an existing net group by deleting members from it.

To simultaneously view and modify Net Group members,

1. Launch the NetGroup Membership form.
  - ☐ To launch this dialog box from Constraint Manager, use one of the following methods.

## Working with NetGroups and PortGroups

### Net Groups

---

- Right-click on the net group name, and choose *NetGroup members*.
- Select the Net Group and choose *Object – Group members*.

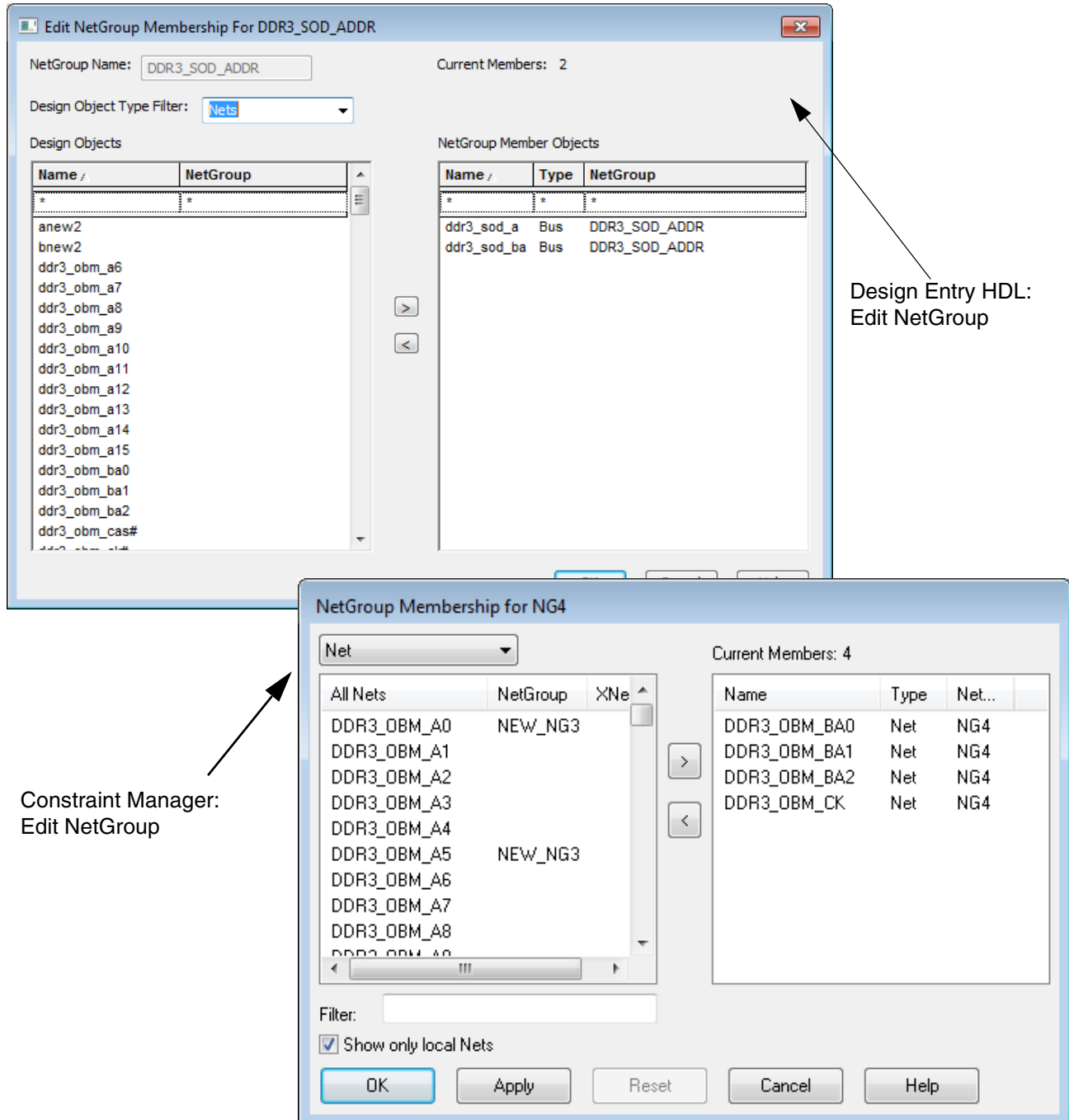
**Note:** These options are not enabled for schematic-defined net groups.

- To launch this dialog box from Design Entry HDL, use one of the following methods.
  - In Interface Browser, right-click on the net group name and choose *Edit*.
  - On the schematic canvas, right-click on the net group instance and from the pop-up menu, choose *Edit Net Group*.

## Working with NetGroups and PortGroups

### Net Groups

The Edit NetGroup Membership form displays. The existing net group members are displayed in the Current Members list.



2. To remove a net object from the net group, select the object from the Current Members list box, and click the left-arrow button.

The selected net object is removed from the net group.

3. After you have made all changes and finalized the net group members, click OK to save your changes and close the dialog box.



***To successfully remove a net object from a Net Group, ensure that the net object to be removed is not connected to the net group on the schematic. If the net object and net group connectivity is not removed, on saving the schematic, the net object is again added to the net group, as in the case of reverse-tapping.***

4. Save the design.

Modifications made to the net group are reflected in Constraint Manager as well as in the Tap Member menu in Design Entry HDL.

## Adding Net Group Constraints

After adding net groups to a schematic, you can specify net group constraints in Constraint Manager.

Like other net objects, you can specify constraints for a net group. Net groups are visible in all net-based worksheets in the Electrical, Physical, Spacing and Same Net Spacing domains. Similar to other net related objects, net groups can also be constrained in all of these domains.

For a net group member, you can override the constraint value specified at the net group level.

## Deleting a Net Group

Only net groups that are not drawn on the schematic can be deleted from the design. For schematic-defined net groups, the Delete command is not available.

To delete a net group from the design, you must first ensure that it is not drawn or instantiated on the schematic, and then remove the net group using one of the following methods:

- In Interface Browser, right-click on the net group name and from the pop-up menu, choose *Delete*.
- In Constraint Manager, right-click on the Net Group name and choose *Delete*.

**Note:** When you delete a netgroup segment which has the netgroup name, the other segment loses its netgroup name. In such cases, you can connect the other segment to the

## Working with NetGroups and PortGroups

### Net Groups

---

netgroup by using the `wire` command. Start from the segment edge that has the netgroup name and connect it to the remaining segment. You can also redraw the existing netgroup.

#### *Important*

Deleting a net group only removes the group information. The member net objects are not removed from the design.

## Net Groups in Hierarchical Designs

If you are working on a hierarchical design, the behavior of net groups created in DE-HDL and Constraint Manager might differ. This is because, in case of hierarchical designs, net groups defined in the schematic can only have net objects that are local to the current block as net group members. Conversely, a net group created in Constraint Manager can have any net object, available in the design, as a net group member.

Net groups that are defined or modified in Constraint Manager, and have member objects belonging to different blocks, are not listed in the Interface Browser window in DE-HDL.

- Interface Browser displays block-level net groups only, while Constraint Manager displays all the net groups available in the design. However, net groups created in all blocks, other than the root design, are listed as schematic-defined net groups in Constraint Manager. For these blocks, you cannot modify net group memberships in Constraint Manager.

Net group **NG\_L1** not available in '**top1**' block for instantiation.

Net groups created in the root design are available in Constraint Manager, and their membership can also be edited in Constraint Manager

Net group locked in Constraint Manager, even when it is not instantiated in the schematic.

Type	S	Name	Refer	Same	Spacin
Dsn		top1			
Dsnl		page1_i1 (block1)			
NGrp	S	NG_L1 (1)			
Bus		D (4)			
Net		net3			
NGrp		NG_T1			
Net		net1			
Net		net2			

From the root design, top1, the NetGroup Membership, Delete, and Rename commands are disabled for NG\_L1.



## Working with NetGroups and PortGroups

### Net Groups

- Net groups that are locked in Constraint Manager can be added as members to other net groups.

Locked net group NG\_L1 added as a member of net group NG\_T1

Name	Type	NetGroup
NG_L1	NetGroup	

Objects			Referenced Same Net Spacing
Type	S	Name	
Dsn		top1	
Dsnl		page1_i1 (block1)	
Bus	+	D (4)	
NGrp		NG_T1 (3)	
NGrp	S	NG_L1 (2)	
Net		net1	
Net		net2	

- If you add a net group defined in another block as a member of the net group added in the root design, the latter is not visible in the Interface Browser for instantiation.

NG\_T1 removed from Interface Browser

Objects			Referenced Same Net Spacing CSet	Line To Line mil
Type	S	Name		
Dsn		top1		
Dsnl		page1_i1 (block1)		
Bus	+	D (4)		
NGrp		NG_T1 (3)		
NGrp	S	NG_L1 (2)		
Net		net1		
Net		net2		

- Using Constraint Manager, you can create net groups that have member objects from different blocks. These net groups are listed in Interface Browser, but cannot be instantiated on the schematic. The *Draw* menu is not available for these Net Groups. Such net groups can only be used for adding constraints on net objects.

## Exporting Net Group Data

When you run the Export Physical command, along with other logical data, information about schematic net groups and related constraint data is also passed to the Allegro PCB Editor. You can launch Constraint Manager from Allegro PCB Editor to ensure that constraint data captured in Design Entry HDL is available.

## Pre-QIR 9 Designs

When you open an old schematic that has schematic-defined net groups with differential pairs and Xnets as member objects, in the current release, DE-HDL performs a sync-up task. As a result of this sync-up, instead of differential pairs and xnets objects, corresponding member nets are displayed as net group members on the schematic. Net group objects continue to display differential pairs and Xnets as net group members in Constraint Manager.

---

# Net Groups in Physical Layout

---

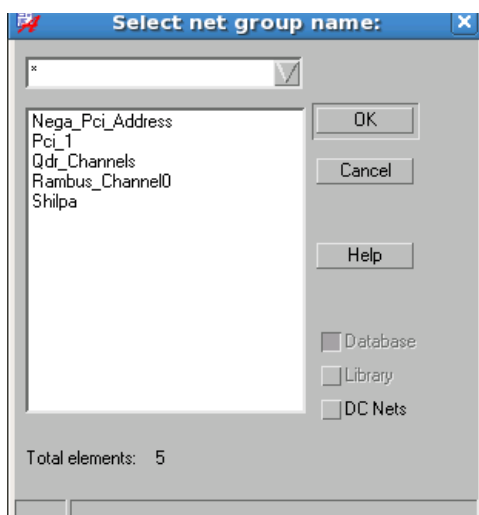
When you run the Export Physical command and generate the physical files for the schematic created in Design Entry HDL, the net group information is also passed to Allegro PCB Editor. To view the net group information, launch Constraint Manager from Allegro PCB Editor.

## Placing Components In PCB Editor

While creating the physical layout of the board in Allegro PCB Editor, you can place components based on their connectivity to a net group. For this you can use either the Quickplace Command or the Place Manually command.

### Quickplace Command

1. Choose *Place – QuickPlace*.
2. In the QuickPlace dialog box, select the *Place by net group name* option.  
The text box and the browse button next to the option are enabled.
3. To display a list of net groups available in the design, click the browse button.



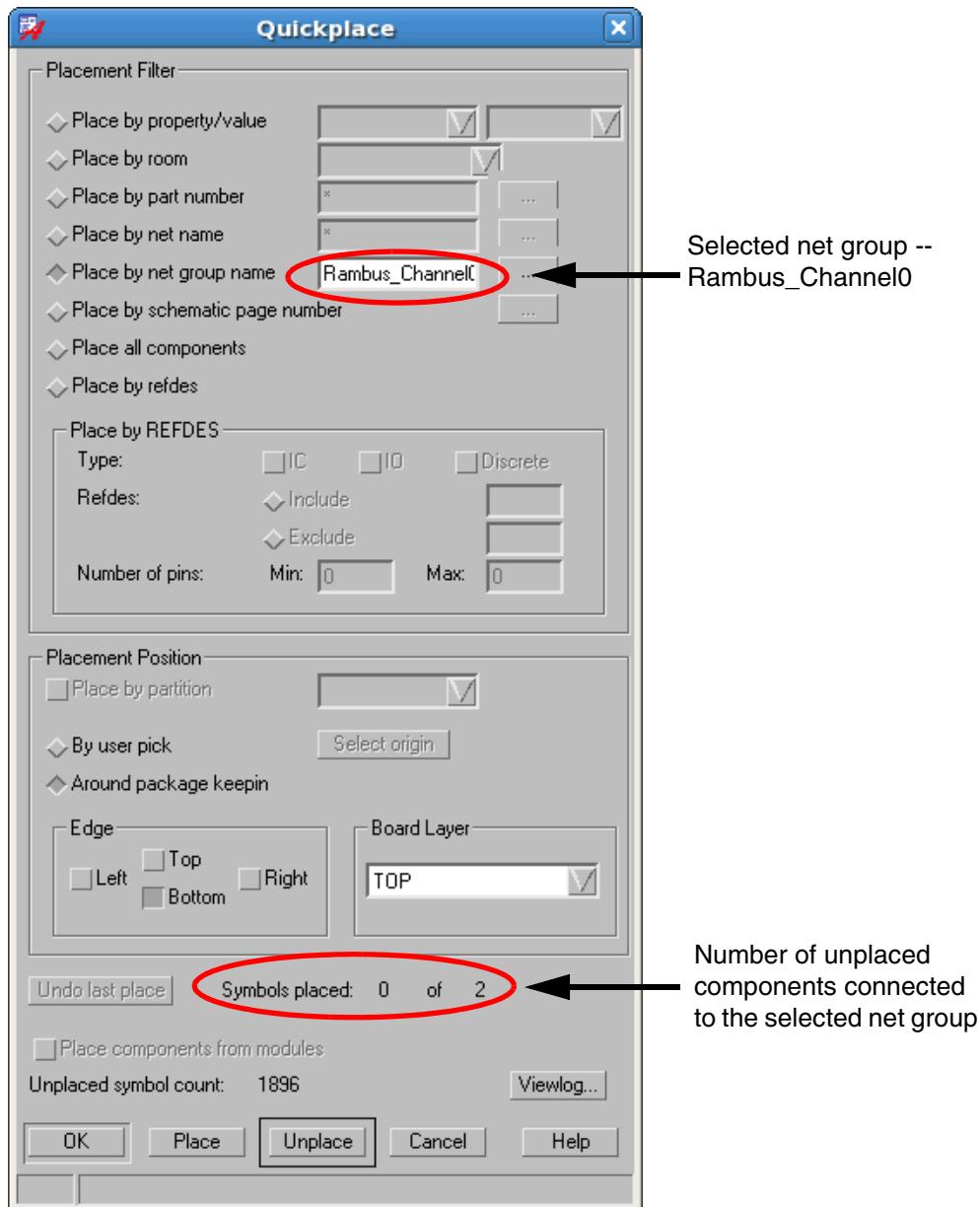
**Note:** Only the net groups that are connected to the unplaced components appear in the list.

## Working with NetGroups and PortGroups

### Net Groups in Physical Layout

4. Select the required net group and click *OK*.

The selected net group name is listed in the text box, and the number of symbols connected to the net group is listed towards the bottom of the dialog box.

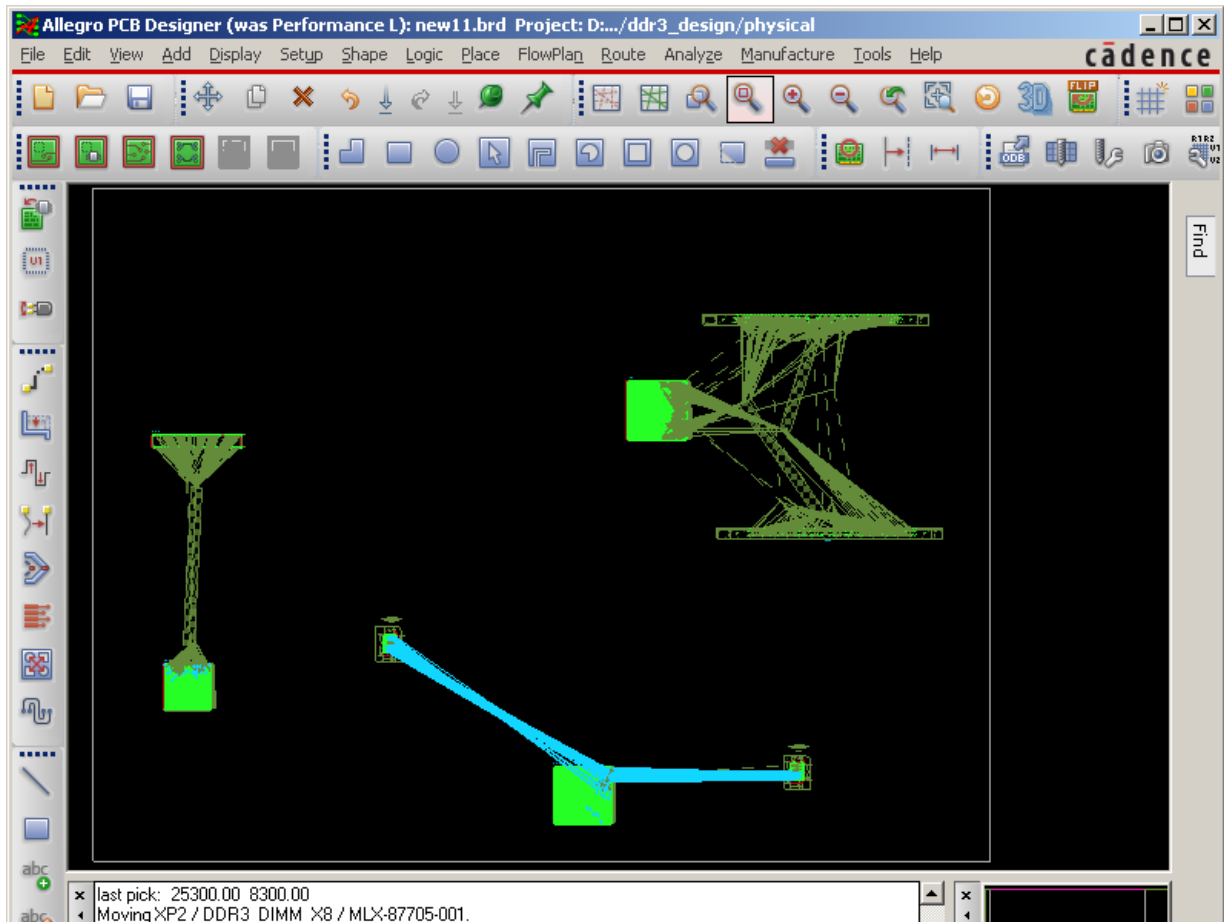


5. If required, modify the options to specify the placement positions and click *Place*.

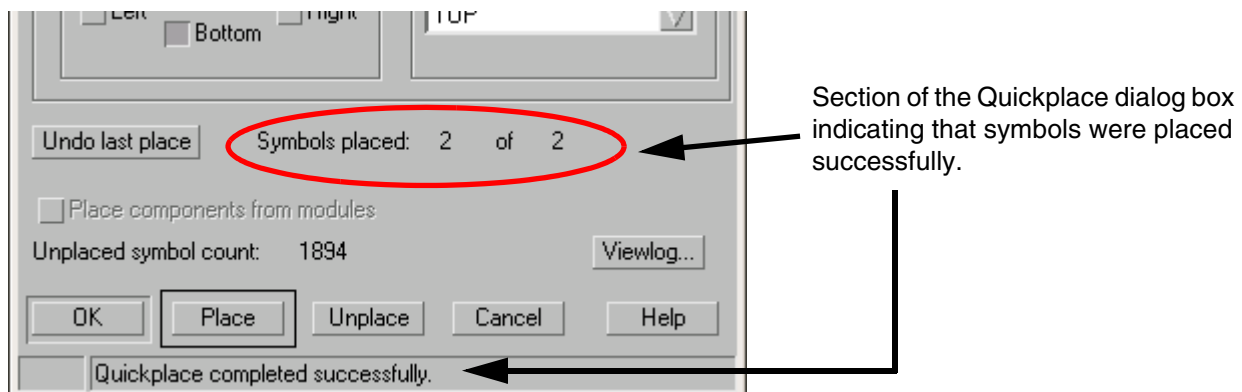
## Working with NetGroups and PortGroups

### Net Groups in Physical Layout

The components are placed as shown.



The Quickplace dialog box is also updated to indicate that the symbols have been placed successfully.



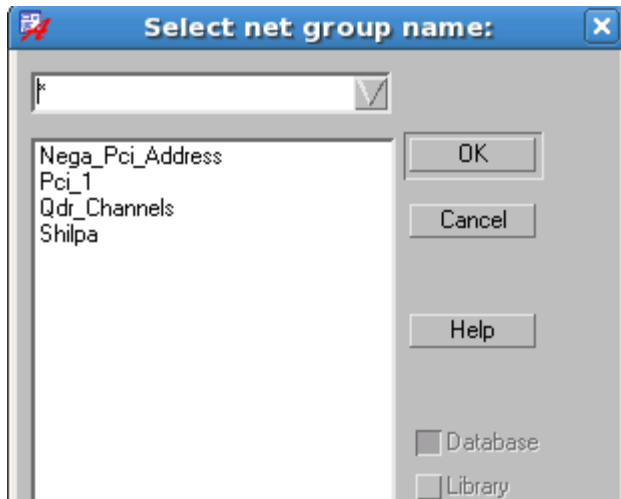
Similarly, you can add components connected to other net groups as well. If all components connected to a net group are placed, the net group name does not appear selection, in the

## Working with NetGroups and PortGroups

### Net Groups in Physical Layout

---

*Select net group name* list. For example, if you again click the browse button next to the *Place by net group name* option, the RAMBUS\_CHANNEL0 net group will not be available for selection, as shown in the following figure.



## Working with NetGroups and PortGroups

### Net Groups in Physical Layout

---

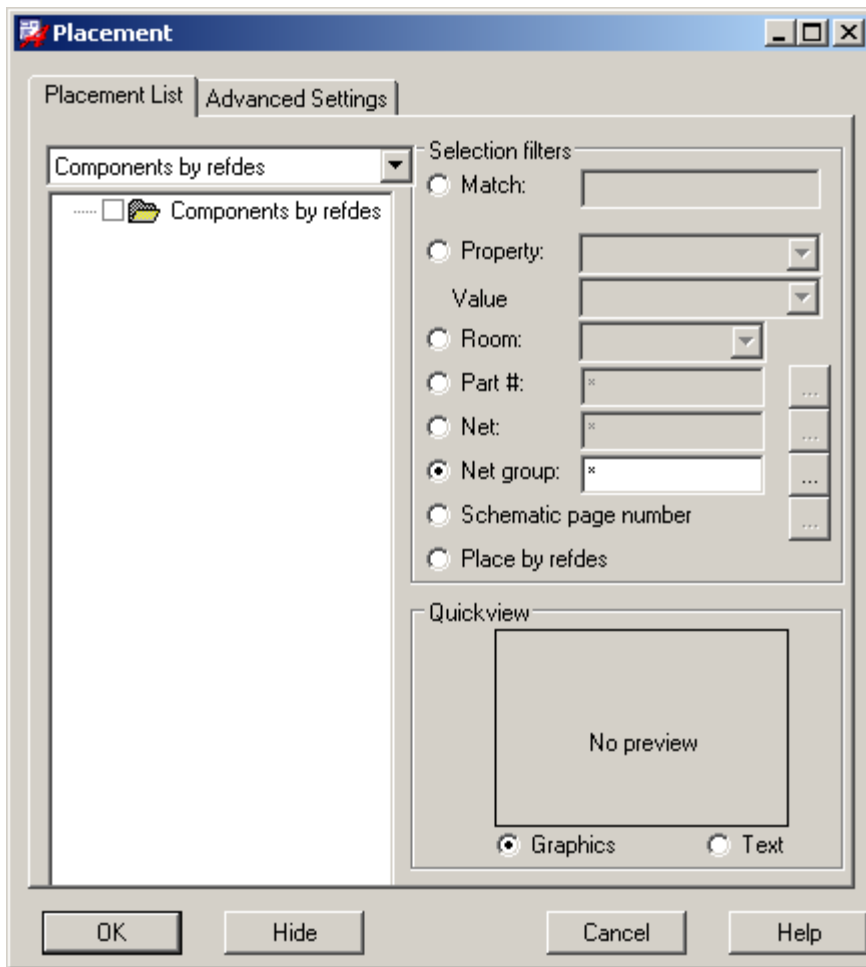
## Place Manually

To manually place the components based on their connectivity to a net group, do the following steps:

1. Choose *Place – Manually*.

Alternatively, enter the `place manual` command in the command window.

2. In the Placement dialog box, select the *Net group* option.



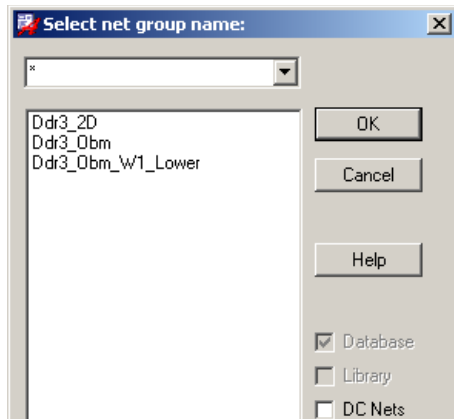
3. To display a list of net groups available in the design, click the browse button.

## Working with NetGroups and PortGroups

### Net Groups in Physical Layout

---

The Select net group name dialog box displays.



4. Select the required net group and click OK.

Components connected to the selected net group are listed in the list box. You can now place these components as required.

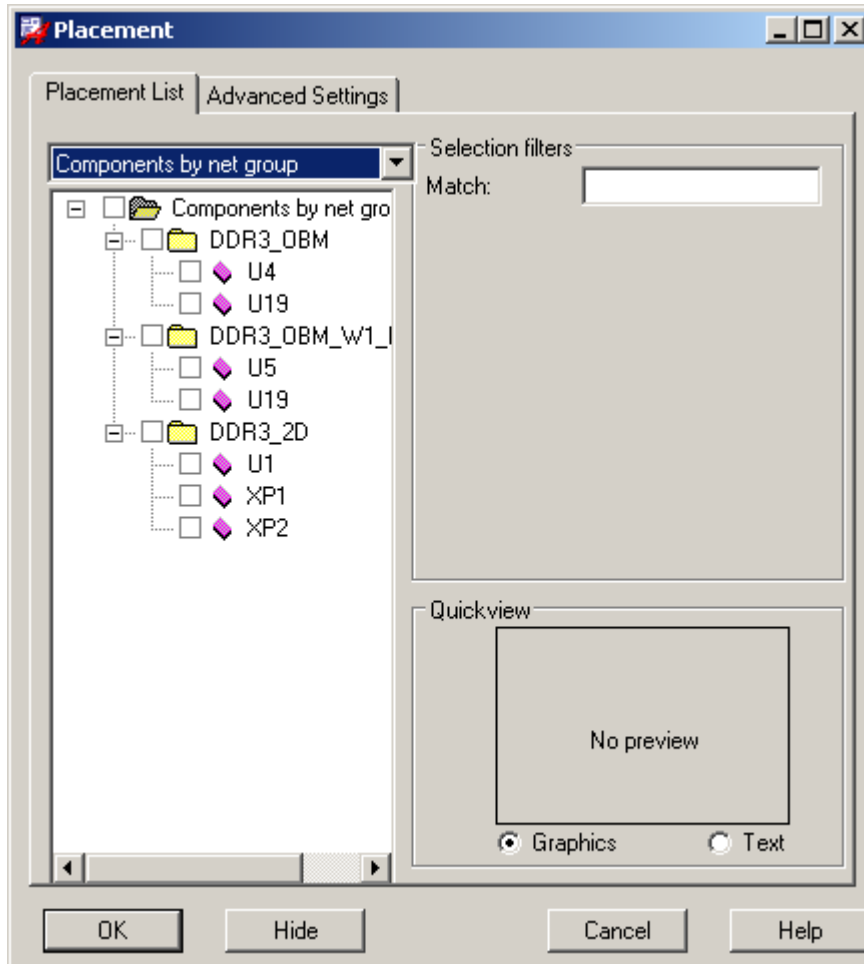


## Working with NetGroups and PortGroups

### Net Groups in Physical Layout

---

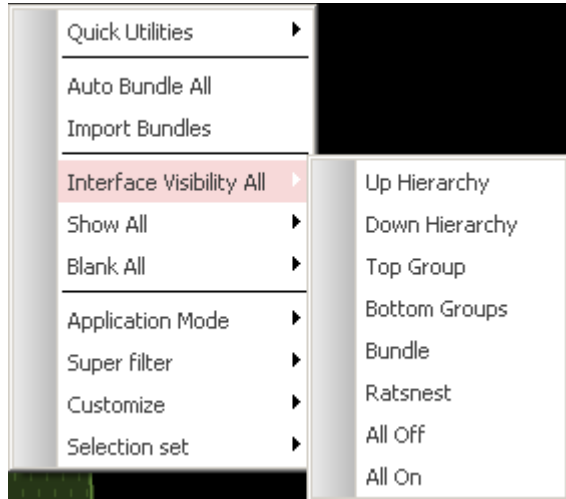
Alternatively, to view all the net groups and the connected components, from the drop-down list in the Placement List tab, select *Components by net groups*.



Now select the required components and place them as required.

## Net Group Visibility

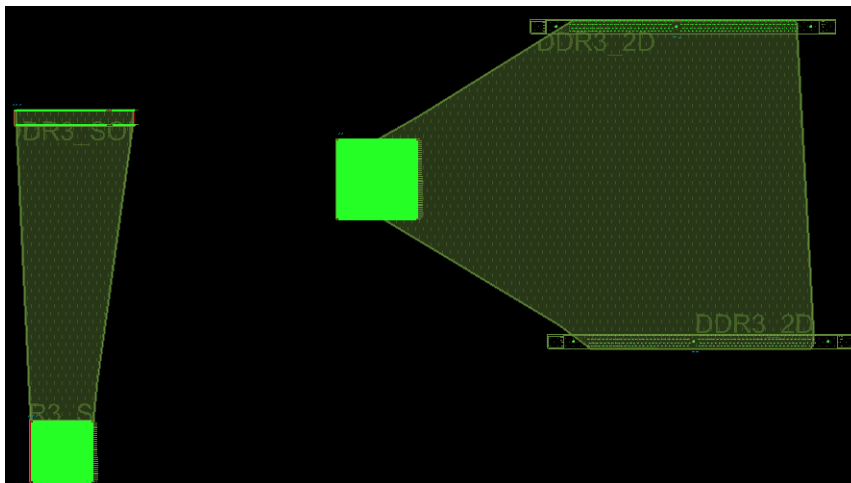
By default, in Allegro PCB Editor, net groups are represented as bundles. You can modify or change the default display by using a context-sensitive menu—*Interface Visibility All*. Using the submenu commands, you can view all the net groups at a glance or can traverse through these.



### Viewing Top-Level Net Group

1. Select the Flow Planning application mode.
2. Right-click on the canvas and from the pop-up menu, choose *Interface Visibility All – Top Group*.

The top-level net group is displayed as a polygon shape.



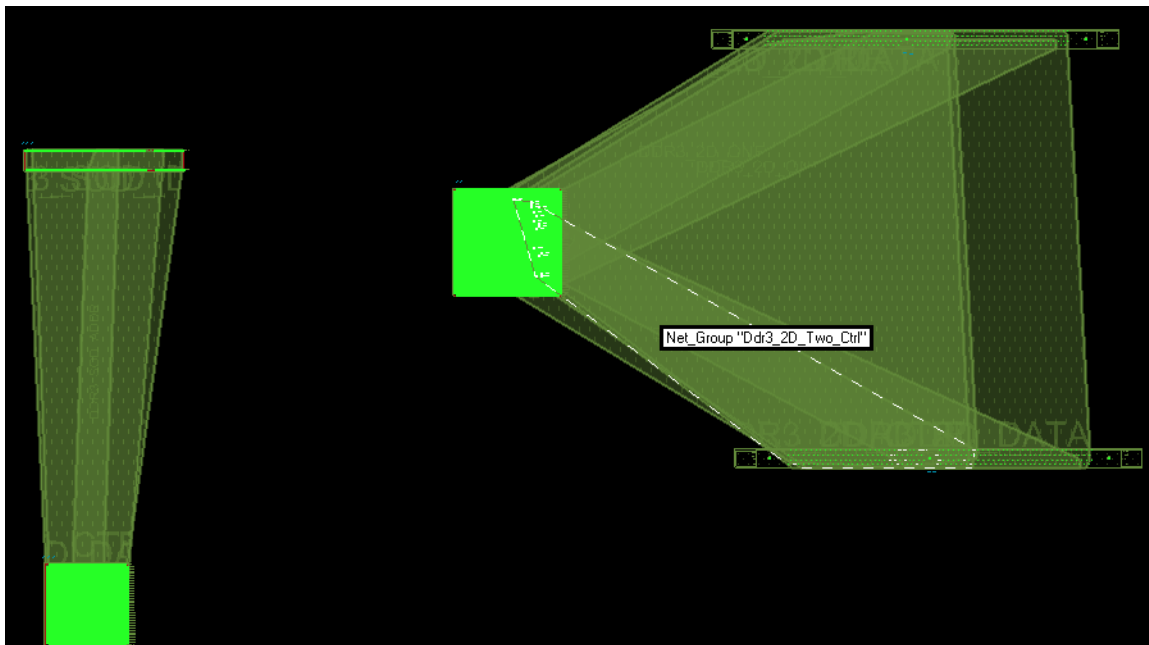
## Working with NetGroups and PortGroups

### Net Groups in Physical Layout

---

This shape indicates the area on the board that is used by the nets in the net group. Changing component placement changes the polygon shape.

3. To view the net groups that are members of the top-level net group, select *Interface Visibility All – Down Hierarchy*.



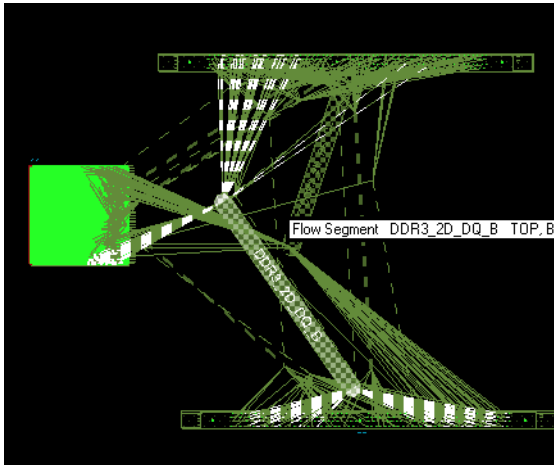
Top-level and member net groups displayed

## Working with NetGroups and PortGroups

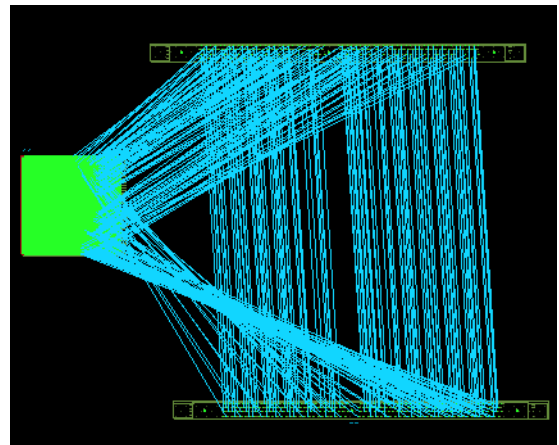
### Net Groups in Physical Layout

---

4. If you keep going down the hierarchy, bundles and finally, ratsnest are displayed.



Net Groups as Bundles



Rats

**Note:** You can enhance the display by specifying different colors in the Color Dialog box for different net groups.

### NO\_PCB\_BUNDLE property

By default, in the physical layout, net groups are represented as bundles. To disable automatic ratbundle creation for net groups, in Design Entry HDL, add the NO\_PCB\_BUNDLE property on the net group.

**Note:** In the design capture phase, this property is added by customizing worksheet in Constraint Manager, launched from Design Entry HDL.

### Usage Guidelines

- To view the net group bundles, remove this property from Constraint Manager invoked from PCB Editor.
- The property needs to be attached to all net groups for which bundle information is not to be shown.

## Working with NetGroups and PortGroups

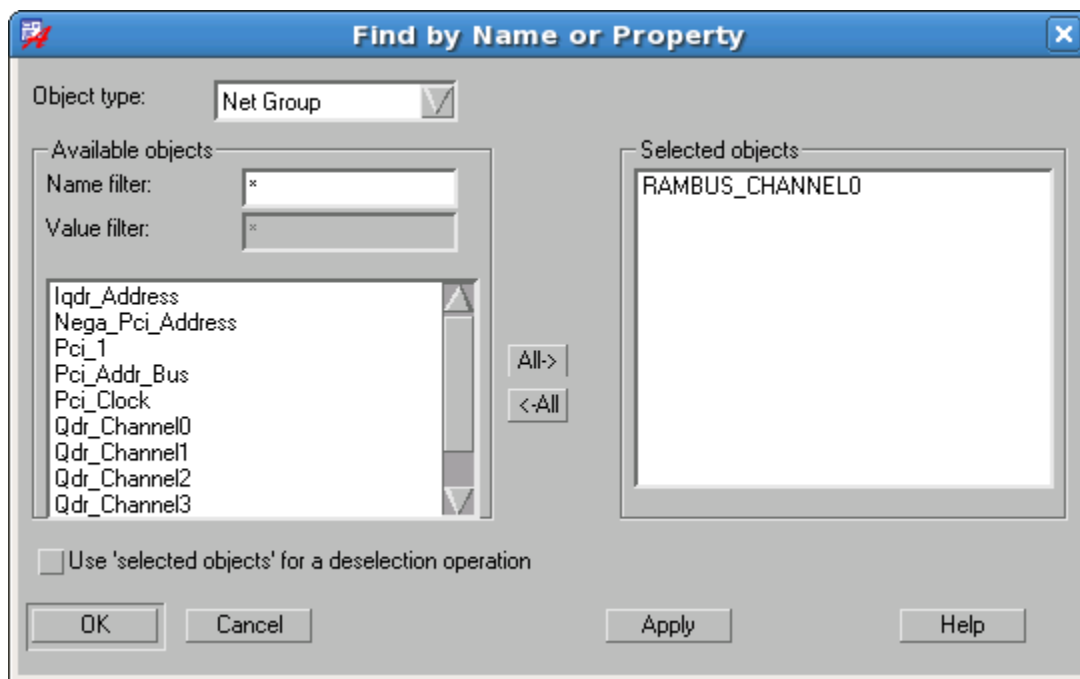
### Net Groups in Physical Layout

- While working with nested net groups, to hide all net groups, the NO\_PCB\_BUNDLE property needs to be added to the top-level net group.

## Locating Net Groups

In a dense board, you can locate a particular Net Group by using the Find filter.

1. In the Find By Name list box, select Net Group.
2. To view the list of Net Groups in the design, click More.
3. Select the Net Groups to be highlighted in the design.



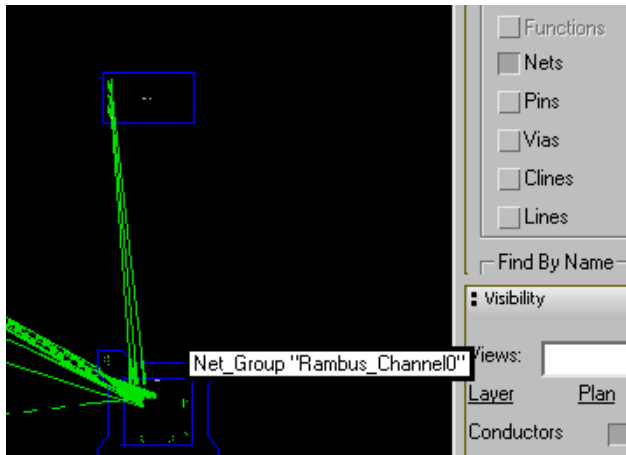
4. Click Apply.

## Working with NetGroups and PortGroups

### Net Groups in Physical Layout

---

The selected Net Group, RAMBUS\_CHANNEL0 is highlighted in the design.



Similarly, you can find all the required net groups.

---

# Port Groups

---

Port Groups are used in hierarchical designs to replicate the lower-level Net Group structure, on a higher-level block.

A Port Group is an interface port on a hierarchical symbol that corresponds to a Net Group used in the lower-level schematic. Using Port Groups, connections to Net Groups members can be created at a higher level of hierarchy, thus further speeding up the task of capturing connectivity.

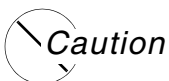
## Creating a Port Group

To create and use port groups in your hierarchical designs, complete the following sequence of tasks.

- ☐ [Creating Net Groups](#)
- ☐ [Instantiating a Net Group](#)
- ☐ Add an IOPORT to the Net Group
- ☐ Run the Generate View command.

## Creating a Net Group

For details on how to create Net Groups, see [Creating Net Groups](#) on page 8.



***Net Groups that have partial bus bits, differential pair of bus bits, and other Net Groups as members, cannot be converted to Port Groups.***

## Instantiating a Net Group

Draw an instance of the Net Group on the schematic and capture the connectivity as explained in the section, [“Instantiating a Net Group”](#) on page 13.

## Working with NetGroups and PortGroups

### Port Groups

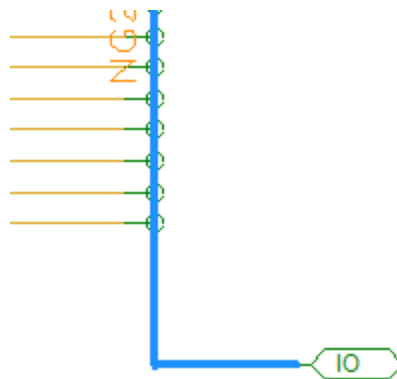
### Adding IOPORT

To mark the Net Group as an interface signal, and to make it available to the higher-level blocks for capturing connectivity, you need to change them to Port Groups.

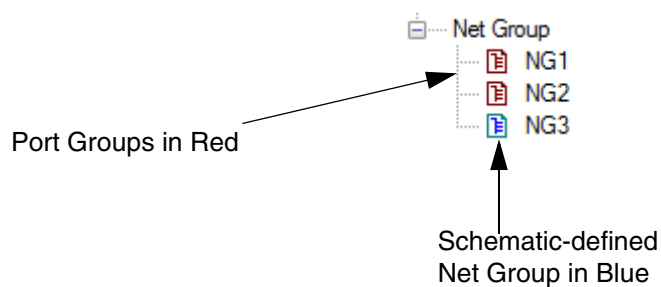
- ➔ To convert a Net Group to a Port Group, add an instance of IOPORT to the schematic-defined Net Group.

**Note:** In the SPB install hierarchy, the IOPORT symbol is available in the standard library.

Using Component Browser, add an instance of IOPORT to the Net Group instance as shown in the following figure, and save the design.



As the design is saved, the symbol in the Interface Browser changes its color from blue to red, indicating that the port group is successfully created.



### Important

Only IOPORTs can be used to convert a Net Group to a Port Group.



## Working with NetGroups and PortGroups

### Port Groups



#### Tip

If you are working on a design that had ports assigned to the member nets before creation of Net Groups, it is recommended that you run the Generate View command to refresh the port assignment in the design.

## Running the Generate View Command

To update the block symbol with the changes to the lower-level schematic, and to make the Port Group available on the block symbol, run the *Generate View (GenView)* command.

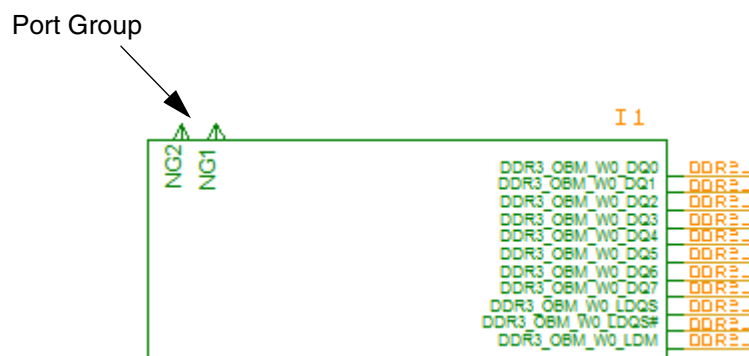
- ➔ From the *Tools* menu, select *Generate View*.
- ➔ In the Genview dialog box, verify the Lib:Cell:View and click *Generate*.



#### Caution

**Port Groups are not supported for Hierarchical Split Symbols. Therefore, ensure that the Split Symbol check box is not selected in the GenView dialog box.**

On successful run of the Genview, the block symbol is updated, and port groups are added on the upper boundary of the generated symbol.



One triangular-shaped pin is added for each Port Group.



#### Tip

If one of the members of the net group is a bus that has Xnets as bus bits, then you need to create a port group only in the master block. Make the lower-level block as the master block, add the bus to the net group, add IOPORT and run the *Generate View* command.

## Using Port Groups

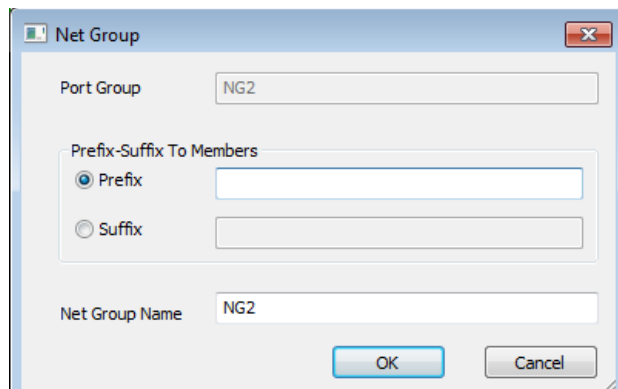
This section covers various operations that can be performed on Port Groups.

### Adding Connectivity

If you connect a signal to a Port Group, the signal changes to a Net Group with the same structure as the Net Group in the block schematic.

1. Using the Draw Wire tool button, draw a wire to connect to the Port Group.

The Net Group dialog is displayed.



In the Prefix (Suffix) field, specify the prefix (suffix) to be added to all the member nets of the new Net Group. The member nets of the new Net Group are aliased to the member nets of the Net Group in the block schematic.

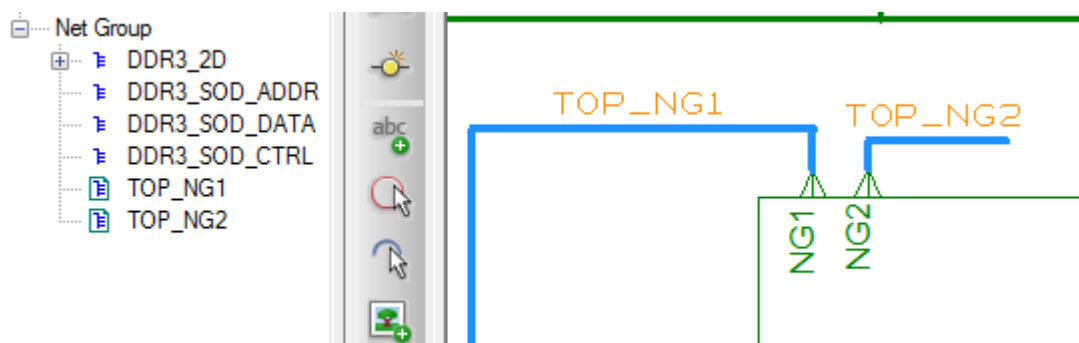
If you do not specify a value, the same member names are used.

2. In the Net Group Name field, you can either accept the default value or specify a different name for the new Net Group to be created at the top-level.
3. Click *OK*.

## Working with NetGroups and PortGroups

### Port Groups

A new Net Group is created and is visible in the Interface Browser window.

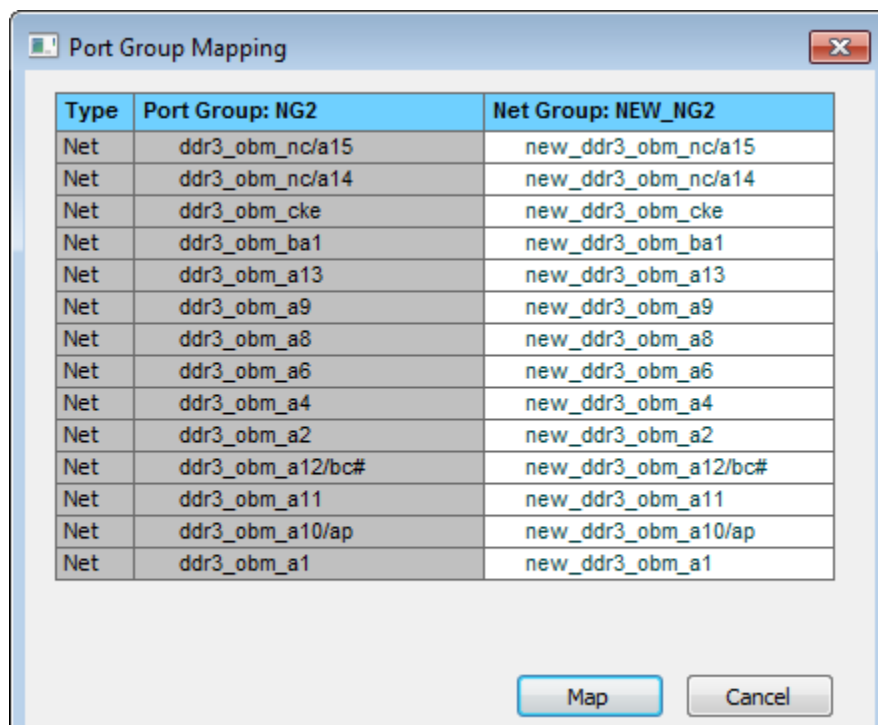


### Mapping Net Group Members

To view the aliasing of the Net Group members,

- ➔ Right-click on the Port Group and choose *Map PortGroup*.

The Port Group Mapping dialog box appears listing the member nets of the original Port Group and the member nets of the New Port Group.



## Working with NetGroups and PortGroups

### Port Groups

---

If required, you can change the connectivity between the Net Group members by modifying the mapping in this dialog box. To modify the mapping, do the following steps:

- a. Click on the member net of the new Net Group.
- b. From the drop-down list, select the Net Group member to be mapped to the corresponding member of the original Net Group.
- c. Modify the other connections as required.
- d. Click Map.

### Modifying Port Group Membership

When you use port groups to capture connectivity at the higher-level blocks, any change in the Net Group membership at the lower-level block needs to be replicated in the top-level Net Group and vice versa.

### Adding New Members

After you have captured the connectivity, you might need to modify the membership of the Net Group or the Port Group by adding new members. Adding a new member is a three step process.

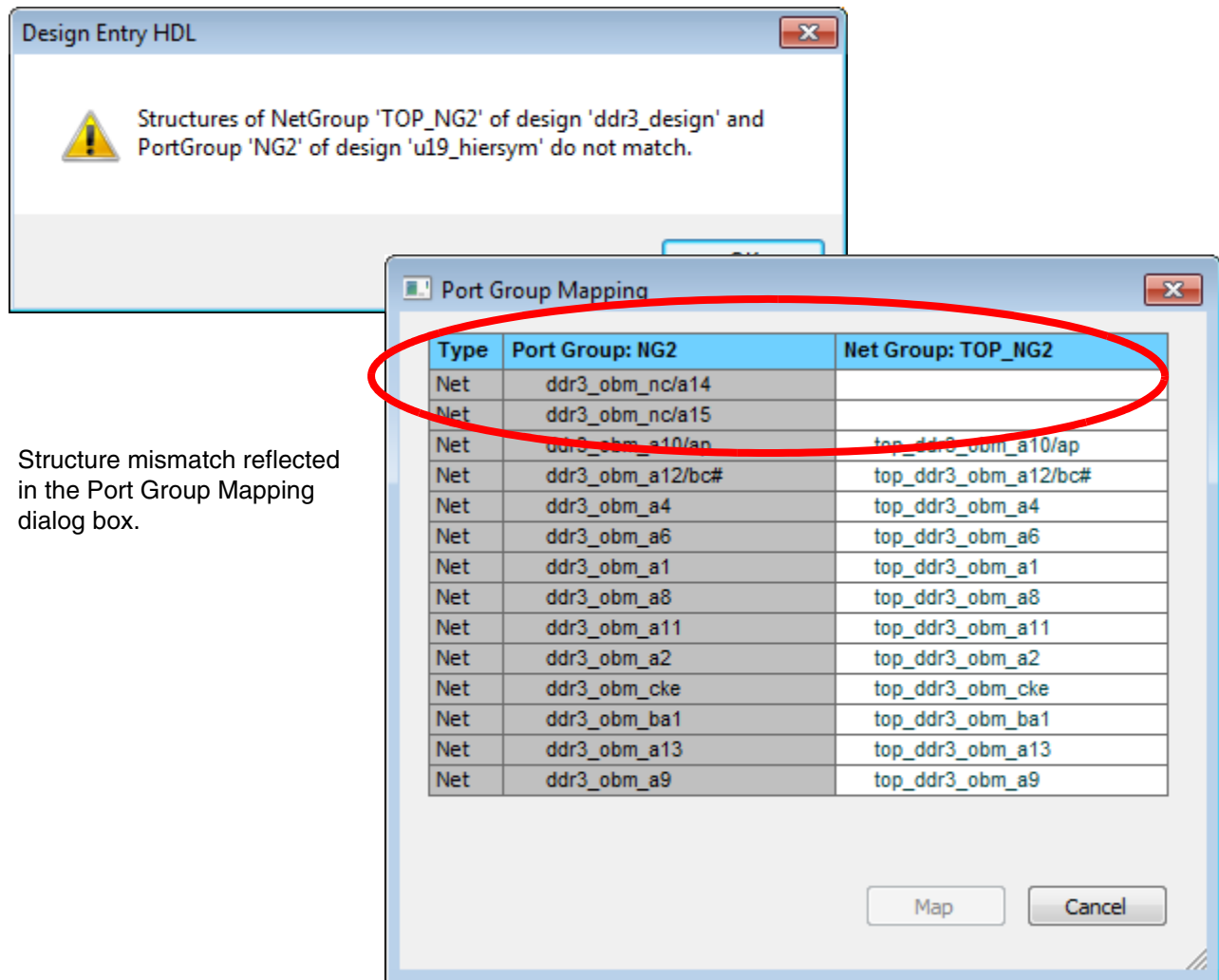
- a. Add a new member to the Net Group.
- b. Add a new member to the Port Group.
- c. Map the new Net Group and port group members using the Port Group Mapping dialog box.

## Working with NetGroups and PortGroups

### Port Groups

#### Case I: A new member added to the port group

After you add a new member to the port group at the lower-level schematic and save the design, an error message is displayed indicating that the structures of the port group and net groups do not match.



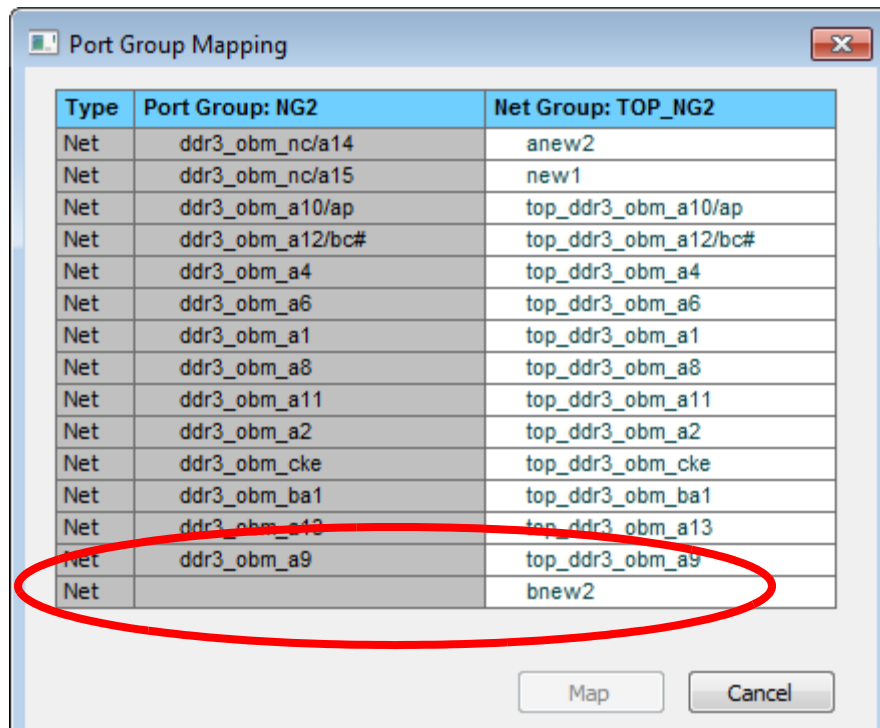
To address this error, you need to add two new member nets to TOP\_NG2 and run the *Map PortGroup* command again.

## Working with NetGroups and PortGroups

### Port Groups

#### Case II: A new member added to the net group

If you add a new member to the net group, the Port Group Mapping dialog box shows fewer members for the PortGroup, as shown in the following figure.



#### Deleting Net Group Members

Using the procedure explained in the section, [Removing Net Group Members](#), you can remove a net object from the Net Group definition. But the change needs to be done for the Net Group as well as the Port Group. To sync these changes, you need to run the *Map PortGroup* command.

#### Renaming Net Group Members

Any name changes made to the net group member objects—except Xnets—in the lower-level blocks, are reflected in the top-level port group.

For Xnets, after you rename an Xnet member, save your schematic using the Save Hierarchy command, and then remap the net group and the port group members.

## Working with NetGroups and PortGroups

### Port Groups

### Deleting Port Groups

Deleting a port group is a two-step process.

*Step 1:* Remove the IOPORT attached to the net group, and save the design.

*Step 2:* Run the Generate View command.

The Port Group symbol is updated and the port group is removed.

### Generating Net Group and Port Group Mapping Report

For design reviews, you can generate a report that contains details of Net Group and Port Group mapping. To generate the report, run the `_dumpportgroupmap` command from the Design Entry HDL command window.

The command creates a file,

`<current_design_level_name>_Netgroup_Portgroup_mapping.log`, in the *temp* folder of the project directory.

The following is a sample report generated by the `_dumpportgroupmap` command:

===== Portgroup mapping dump of 'sub_mem_top_fixed' design =====			
Type	Port Group: BYTELANE_4	Net Group: BL8	Mapping Status
Net	dq<24>	b18dq<24>	
Net	dq<25>	b18dq<25>	
Net	dq<26>	b18dq<26>	
Net	dq<27>	b18dq<27>	
Net	dq<28>	b18dq<28>	
Net	dq<29>	b18dq<29>	
Net	dq<30>	b18dq<30>	
Net	dq<31>	b18dq<31>	
Net	dqs_b14_n	b18dqs_b14_n	
Net	dqs_b14_p	b18dqs_b14_p	
Type	Port Group: BYTELANE_3	Net Group: BL7	Mapping Status
Net	dq<16>	b17dq<16>	
Net	dq<17>	b17dq<17>	
Net	dq<18>	b17dq<18>	
Net	dq<19>	b17dq<19>	
Net	dq<20>	b17dq<20>	
Net	dq<21>	b17dq<21>	
Net	dq<22>	b17dq<22>	
Net	dq<23>	b17dq<23>	
Net	dqs_b13_n	b17dqs_b13_n	
Net	dqs_b13_p	b17dqs_b13_p	
Type	Port Group: EOTEST	Net Group: PUTTE	Mapping Status
Net	a	one	
Net	b	two	
Net	c	four	
Net	d	five	
Net	six		### unMapped ###

## Reusing Designs with Net Groups and Port Groups

When you use the Import Sheet command to import a schematic page with Net Groups and Port Groups, only the design objects instantiated on the page and the connectivity on the schematic page are imported.

### *Net Group and Port Group Behavior on Importing Schematic Sheet*

1. If a net group is instantiated on the source schematic page but no members are tapped out, then, after sheet import, an empty net group is created in the destination schematic page.

**Note:** In case of port groups, after importing a schematic sheet, you need to explicitly map the net group and port group members, in the destination design.

2. If the members are tapped out in the source design, in the destination design, by virtue of the reverse tapping, only the tapped-out members are added as net group members.

**Note:** In the destination design, tapped nets must not be members of any other net group object.

3. In the source schematic, if a net object, such as differential pair, is added as a net group member, then, on importing the schematic page, the net object is not recreated, but the members are directly added to the net group in the destination design.

Net Group Definition in Source Design

NGrp	S	
		NEW_NG3 (2)
DPr		NEW_DP1
Net		DDR3_OBM_A1
Net		DDR3_OBM_A2
DPr		NEW_DP2
Net		DDR3_OBM_A3
Net		DDR3_OBM_A4

Net Group Definition in Destination Design after Importing Schematic Sheet

NGrp	S	
		NEW_NG3 (4)
Net		ddr3_obm_a1
Net		ddr3_obm_a2
Net		ddr3_obm_a3
Net		ddr3_obm_a4

Differential Pairs are lost.

For such scenarios, there is a mismatch in the net group structure in the source and destination design.