Lesson 7: Setting Design Constraints

Learning Objectives

In this lesson you will:

• Explore the design rule system and apply design rules for physical and spacing dimensions

• Add, change, and delete properties of components and nets

In this section you will set up your design rules. Design rules are known as Constraints in the OrCAD and Allegro PCB Editor and are the rules that must be followed while routing your design. Typical constraints are the line width to be used during routing, line-to-line spacing, line-to-pad spacing, and so on.
Design Layout Process

This design flow is used throughout the entire course. Each box in this flow represents a common step in the design of a printed circuit board. As indicated in the design flow, the Set/check CBD (Correct By Design) rules and constraints step will now be discussed.
Introduction to Design Rules

- There are five types of design rules:
  - **Electrical Constraints**: Performance characteristics
  - **Physical Constraints**: Line width, via selection, and layer restrictions
  - **Spacing Constraints**: Clearances between lines, pads, vias, and copper areas (shapes)
  - **Same Net Spacing Constraints**: Clearances between lines, pads, vias, and copper areas (shapes) on the same net. These are differentiated from Net to Net rules
  - **Design Constraints**: Package checks, soldermask checks and negative plane island checks

- For Physical and Spacing, there are two categories of design rules:
  - Default Rules: Used to specify the rules to be applied to nets having no special routing requirements
  - Special Rules: Used for any net requiring different rules applied to them

The OrCAD and Allegro PCB Editor tool has a set of predefined rules, such as Line-to-Pin Spacing, or Minimum Line Width. You can define values for each rule within the context of a constraint set. A constraint set is a group of rules that have been bundled together to make value assignments easier for the user.

This rule ‘bundling’ is based upon the type of constraint set.

  - **Electrical Constraints** - constraints governing electrical behavior and performance of an entire net (for example, Max Length and differential pairs).
  - **Physical Constraints** - constraints govern physical construction of a net (for example, minimum line width, via selection, and allowed etch layers).
  - **Spacing Constraints** - constraints govern the spacing between objects on different nets (for example, line-to-thru-pin spacing).
  - **Same net Spacing Constraints** - constraints govern the spacing between objects on the same net.
  - **Design Constraints** - setting or unsetting of package DRC checking, Negative Plane Islands constraints, Soldermask constraints.

There are two different categories of Physical and Spacing design rules that can be applied to your design. The **Default** category is used to specify the rules to be applied to nets that have no special routing requirements. Any nets that need different rules applied to them fall into the **Special** category. For these nets, you must identify the nets requiring the special rules, and also create/set the special rule values.
The Constraint Manager

To set your design rules, use the Setup - Constraints - Constraint Manager command from the top menu. The Allegro Constraint Manager form is opened. You can access and create all of the required physical, spacing, same net spacing and design constraints from this form.

You can also directly access either the electrical constraints, the physical constraints, the spacing constraints, or the same net spacing constraints by using the commands Setup - Constraints - Electrical, Setup - Constraints - Physical, Setup - Constraints - Spacing, or Setup - Constraints - Same Net Spacing. These separate commands provide a direct path to the worksheet you want to work on in the quickest manner possible.

The Constraint Manager contains several different sections. The standard menu row and icon row are available for use. The Left Pane contains different sections where you select which type of design rules you wish to set or view. The Work Area is the section where you will set the rules for your design, or view the current rule values.

The Status Window should always be checked for Warning or Error messages. If you attempt to set a value in a cell that cannot be modified, a message in the Status Window will identify this fact.
The Left Pane of the Constraint Manager is divided into six different domains. Each domain has several Folders available, each of which has several Workbooks/Worksheets available. The four different domains available in the left pane are:

- **Electrical** - These are the Electrical Constraints where you define the electrical behavior and performance of an entire net. Not covered in this class, covered in the Advanced PCB Editing Class.

- **Physical** - These are the Physical Constraints where you define the characteristics of the routing. Rules contained in this domain include line width, trace necking values, allowable vias, and so on. When you create a new rule set, it is referred to as a Physical CSet (Constraint Set).

- **Spacing** - These are the Spacing Constraints where you define the clearance between objects. Rules contained in this domain include line-to-line spacing, line-to-thru-pin spacing, via-to-thru-pin spacing, and so on. When you create a new rule set, it is referred to as a Spacing CSet (Constraint Set).

- **Same Net Spacing** - These are the Same Net Spacing Constraints where you define the clearance between objects that are independent of Net to Net Spacing DRCs. Rules contained in this domain include line-to-line spacing, line-to-thru-pin spacing, via-to-thru-pin spacing, and so on. Net Class-Class is not supported. When you create a new rule set, it is referred to as a Same Net Spacing CSet (Constraint Set).

- **Properties** - This domain allows you to assign properties to nets and components. This is an alternative method to the *Edit - Properties* command, which will be discussed later in this section.

- **DRC** - This domain lists all DRCs in your design separated into the Physical Worksheet, Spacing Worksheet, Design Worksheet, and External Worksheet.
Constraint Manager Work Area

The Constraint Manager work area is where you set or view all the design rules for your design. The work area will have the appropriate values that match the worksheet you have selected. All the values available will be covered later in this section.

The Objects and Type column will vary based upon the worksheets you have open. The objects and types can be such items as Electrical CSets, Physical CSets, Spacing CSets, Buses, Nets, Pin Pairs, and so on. Hovering your cursor over an object or type will display a tool tip window identifying the object or type currently being hovered over. The Status Window will also display the tool tip information.

You can turn on and off the column numbers by using the View - Options command and selecting or deselecting the Row Number option in the Workbooks section. You can also use the row number icon to toggle on/off the row numbers.
Setting Default Physical Values

The first step in creating your physical rules for your design is to set the default rules. These rules will be used for the nets that have no special routing requirements. You can set the same rules for all routing layers in your design by setting the values in the DEFAULT row of the Constraint Manager. If you need to set different rules for different layers in your design, you can expand the DEFAULT row by selecting the “+” character. You will see a row for each layer you have created in your board stack-up. You can now set different values as required on any layer in your design. The values you can set are:

- **Line Width, Min** - This is the minimum line width at which a connection can be made. When you manually route a connection, this value will be used by default. If you route at a width less than this value, a DRC error will be created.

- **Line Width, Max** - This is the maximum line width at which a connection can be made. If you use a line width greater than this value, a DRC error will be created.

- **Neck, Min Width** - This is the minimum line width at which a connection can be made when using the neck mode. The neck mode option is available when routing by using the Right-Mouse-Button pop-up menu item Neck.

- **Neck, Max Length** - When in the necking mode, this is the maximum allowable length at which a connection can be routed before returning to the minimum line width. Note that this value is cumulative length of the necked sections across the entire net.
• **Vias** - This is the list of via padstacks (.pad files) that are allowed to be used with your default nets.

• **BB Via Stagger, Min** - This rule specifies the minimum center-to-center distance between the connect point of one pin or via (the X, Y location of the pin or via) and the connect point of the other, where the two pins or vias are on the same net and have a single cline connecting them.

• **BB Via Stagger, Max** - This rule specifies the maximum center-to-center distance between the connect point of one pin or via (the X, Y location of the pin or via) and the connect point of the other, where the two pins or vias are on the same net and have a single cline connecting them.

• **Allow Etch** - If set to True, routing is allowed on the subclass/layer. If set to false, routing is NOT allowed on the subclass/layer.

• **Allow Ts** - This specifies when and where T junctions (points where there are 3 or more segments of etch) are allowed. Values are:
  – **Not Allowed** - Prohibits T junctions.
  – **Anywhere** - Specifies that T junctions can form at a pin, via, or on a connect line (cline). This is the default.
  – **Pins Only** - Allows T junctions to only form at a pin.
  – **Pins Vias Only** - Allows T junctions only at a pin or via.

• **Allow Pad-Pad Connect** - Specifies whether a pin/via whose “connect point” lies within the extents of another pin/via forms a direct connection without the presence of an intermediate cline. For example, to allow symbol surface-mount device pads to have associated fanouts embedded without the need to draw a connect line. The choices are:
  – **All Allowed** - Specifies that direct connections can form anywhere. This is the default.
  – **Via/Pin Allowed** - Specifies that direct connections only can form between via and pin.
  – **Via/Via Allowed** - Specifies that only direct connections between via and via can form.
  – **Not Allowed** - Prohibits direct connections everywhere.
Adding a Via Selection to a Net or Constraint Set

When there is a requirement to include a selection of multiple vias, the multiple selection listing is added as a physical rule value in the Physical Constraints. This list of multiple vias can be applied to a Physical Constraint Set (CSet) or to an individual net.

Adding to a CSET

Adding to a Net

Clicking on the Vias column in either the CSet or the individual net opens an Edit Via List that provides a list of the available vias. From this list any padstack can be used as a via.

Once the selection of vias have been made they will appear in the Via category for either the CSET or net that you applied them to.
Creating a New Physical CSet

You will probably have nets that require different physical rules than the default rules. These are your special nets. You need to create a new Physical CSet for these nets. You can create as many physical CSets as required in your design. To create a new CSet, perform the following steps:

1. Select the **All Layers** worksheet under the Physical Constraint Set folder.

2. Select the **DEFAULT** (or any other existing Physical CSet) cell in the Objects column.

3. Select **Objects - Create - Physical CSet** or **RMB - Create - Physical CSet** from the Constraint Manager menu bar or from the **Right-Mouse-Button - Create - Physical CSet** popup menu.

4. Enter in a new Physical CSet name in the Create Physical CSet form. You can use the Copy Constraints From: option to copy existing constraints if you wish.

5. Enter in the new values to match your new physical routing rules. You can select the “+” character next to the physical CSet you just created and set different values on different layers in your design.
Identify the Special Physical Nets

1. Select Net > All Layers Worksheet

Enabling here will create the NET CLASS for both the Physical and Spacing CSETS

2. Browse to and select the SPECIAL NETS.

3. From the RMB – select Create > Net Class

4. Assign Net Class name

After you have created the physical rules for your special nets, the next step is to identify these nets. You perform this task by assigning nets into Net Classes. You can create as many Net Classes as are required within your design. To create a Net Class, perform the following steps:

1. Select the All Layers worksheet under the Net folder.

2. Scroll through the nets and select the nets you want to assign into the new net class you are about to create.

3. Select with the Right-Mouse-Button and choose Create - Class from the pop-up menu.

4. Enter in a new Class name in the Create Net Class form. You can create this new class in both the physical section as well as the spacing section. Select OK and the nets you have selected will be placed in the net class.
Assign the Net Class to a Constraint Set

Now that you have created a net class, you need to assign the net class to a previously defined constraint set. Select the All Layers worksheet under the Net workbook. There are two different methods to assign the net class.

The first method is to select in the Net class row (in this case, the Special net class) in the cell under the Referenced Physical CSet column. When you select this cell (with the Left-Mouse-Button), a pull-down menu will appear with all of the defined Physical CSets listed. Select the appropriate CSet from the pull-down to make the assignment.

An alternate method is to select the Net Class cell with the Right-Mouse-Button and select the Constraint Set Reference option in the pop-up menu. In the Physical CSet References form, select the pull-down menu to list the already defined Physical CSets. Select the required CSet and select the OK button.
Assign Rules Directly to a Net

As an alternative to creating a net class, adding the nets to the net class, and assign a Physical CSet to the net class, you can assign rules directly to nets. In the Net Folder section, you can select on a net(s) and assign a Physical CSet directly, as shown in the top picture above.

You can also set values directly on a net(s) without assigning the net to a Physical CSet. Select on the cell in the net row and enter a new value, as shown in the bottom picture above.

In either case, note that when you change a value from the default value, the color changes to blue. This indicates that the rule in that cell does not match the default value assigned. You can control the color used for these overrides by using the menu sequence View - Options, setting the Color Palette option to Custom, and setting the Directly Set color.
Lab

Lab 7-1: Setting Physical Rules

– Setting the Default Physical Rules
– Define the Special Physical Rules
– Identify the Special Physical Nets
– Add the Net Class

The following lab will allow you to familiarize yourself with the process required to set physical rules and create special physical design rules. You will learn how to create new design rules, identify the special nets, and apply the new design rules to the special nets.
Lab 7-1: Setting Physical Rules

Objective: Define physical routing rules for special nets.

Setting the Default Physical Rules

New designs use a 5-mil line width as the default trace width. This design requires a 6-mil line width for all non-critical/non-special nets.

1. If you don’t already have the OrCAD and Allegro PCB Editor tool running, start the OrCAD and Allegro PCB Editor.

2. Choose File - Open and open the unplaced.brd design file you saved previously, if it is not currently open.

3. Select Setup - Constraints - Physical from the OrCAD and Allegro PCB Editor main menu.

   The Constraint Manager form opens and the Physical section is displayed.

4. Select the All Layers Worksheet under the Physical Constraint Set folder.

   This should be the worksheet already open, but it is good practice to make sure you have the correct worksheet open.

5. In the Default row, change the Min Line Width, Min Neck Width and the Min BBVia Stagger values to 6.

   Your form should look like below:

   ![Worksheet](image)

Defining the Special Physical Rules

Assume the nets VCLKA and VCLKC require a larger line width (8 mils) than the default values. First, create the new rules by creating a new Physical CSet.
1. Select the Default cell.

2. Select Objects - Create - Physical CSet from the Constraint Manager menu or the Right-Mouse-Button - Create - Physical CSet popup menu command.

3. In the Create Physical CSet form, enter 8_mil_line and select the OK button.

4. In the 8_MIL_LINE row, change the Min Line Width, Min Neck Width and the Min BBVia Stagger values to 8.
   Your form should look like below:

Identifying the Special Physical Nets

Now that you have created the physical routing rules for the special nets, you need to assign the VLKCA and VCLKC nets to a Net Class and assign that class to use the 8_mil_line Physical CSet.

1. Select the All Layers worksheet under the Net folder as shown below.

2. Scroll through the nets section so both the VCLKA and VCLKC nets are visible.
3. Select the net VCLKA, and shift-select the net VCLKC so that both nets are selected.

4. Select with the Right-Mouse-Button and choose *Create - Net Class* from the pop-up menu.

5. Enter the name SYNC in the *Net Class* field. Verify that the “Create for both physical and spacing” option is checked, and select **OK**.

---

**Assign the Net Class**

You just created the new net class SYNC and identified the special nets VCLKA and VCLKC belonging to this net class. Now you must assign the SYNC net class to use the 8_MIL_LINE rule set.

1. Select the *Referenced Physical CSet cell* in the SYNC row and select 8_MIL_LINE from the pull-down menu as shown below. You will need to scroll to the top of the spreadsheet in order to locate the SYNC Net class.

Now the SYNC net class, which contains the two special nets VCLKA and VCLKC, will use the 8_MIL_LINE rule such that when either of these two nets is routed, the line width used will be 8 mils instead of the default 6-mil-wide line.
2. Select File - Close from the Constraint Manager window.

3. Continue by choosing File - Save As.
   A browser form appears.

4. Rename this drawing by entering the following in the File Name field:
   constraints

5. Click Save to save the constraints.brd file.
   The constraints.brd file is saved to disk.

Note
Do not exit from the OrCAD and Allegro PCB Editor. The next lab will continue from this point.

End of Lab
Setting Default Spacing Values

The first step in creating your spacing rules for your design is to set the default rules. These rules will be used for the nets that have no special routing requirements. You can set the same rules for all routing layers in your design by setting the values in the DEFAULT row of the Constraint Manager. If you need to set different rules for different layers in your design, you can expand the DEFAULT row by selecting the “+” character. You will see a row for each layer you have created in your board stack-up. You can now set different values as required on any layer in your design.

The spacing values you set are for edge-to-edge clearance, or the air gap between the two elements. You can specify different values for lines, pins, vias, shapes and holes. For pins, you can specify different values for thru pins, surface-mount pins, and test pins. For vias, you can specify different values for thru vias, blind/buried vias and test vias.
Creating a New Spacing CSet

1. Select one of the worksheets under the All Layers workbook under the Spacing Constraint Set folder.

2. Select the DEFAULT (or any other existing Spacing CSet) cell in the Objects column.

3. Select Objects - Create - Spacing CSet or RMB - Create - Spacing CSet from the Constraint Manager menu bar or from the Right-Mouse-Button - Create - Physical CSet popup menu.

4. Enter in a new Spacing CSet name in the Create Spacing CSet form. You can use the Copy Constraints from: option to copy existing constraints if you wish. The constraints used for the copy will be based upon which CSet you selected when executing the Create command.

5. Enter in the new values to match your new spacing routing rules. You can select the “+” character next to the spacing CSet and set different values on different layers in your design.

You will probably have nets that require different spacing rules than the default rules. These are your special nets. You need to create a new Spacing CSet for these nets. You can create as many Spacing CSets as required in your design. To create a new CSet, perform the following steps:
Identify the Special Spacing Nets

(Method 2)

1. Select any worksheet under the All Layers workbook under the Net folder.

2. Select the <design name> cell (this is the name of the current database opened in the OrCAD and Allegro PCB Editor) under the Objects column.

3. Select Objects - Create - Net Class from the Constraint Manager menu bar.
   a. An alternate method is to use the Right-Mouse-Button Create - Net Class pop-up menu option.

4. Enter in a new Net Class name in the Create Net Class form. You can create this new class in both the physical section as well as the spacing section.

5. Select the newly created net class cell, and select Objects - Add to - Net Class from the Constraint Manager menu bar or from the Right-Mouse-Button - Net Class members... popup menu command. A Net Class Membership form will be displayed. Select the nets from the left side that you want to be identified in this Net Class. Select the “>” button to add the selected nets to the net class.
Assign the Net Class to a Constraint Set

Now that you have created a net class, you need to assign the net class to a previously defined constraint set. Select one of the worksheets under the All Layers workbook under the Net folder. There are two different methods to assign the net class.

The first method is to select in the Net class row (in this case, the VDNETS net class) in the cell under the Referenced Physical CSet column. When you select this cell (with the Left-Mouse-Button), a pull-down menu will appear with all of the defined Spacing CSets listed. Select the appropriate CSet from the pull-down menu to make the assignment.

An alternate method is to select the Net Class cell with the Right-Mouse-Button and select the Constraint Set Reference option in the pop-up menu. In the Spacing CSet References form, select the pull-down menu to list the already defined Spacing CSets. Select the required CSet and select the OK button.

Note that you can also assign rules directly to nets in the same manner as was shown in the Physical section under the topic “Assign Rules Directly on a Net”.

Note that you can also assign rules directly to nets in the same manner as was shown in the Physical section under the topic “Assign Rules Directly on a Net”.
Lesson 7

Setting Design Constraints

Net Class to Net Class Spacing

Previously, you assigned a spacing rule directly to the Net Class. This means all etch running next to any of the nets in the net class will use the assigned CSet applied to the net class. There may be cases where you require different spacing between nets in net classes. To accomplish this, you create Net Class to Net Class spacing rules.

First, you must work under the Net Class-Class folder. Select the appropriate worksheet. Select one of the Net Classes with the Right-Mouse-Button and select Create - Class-Class. A Create Class-Classes form will be displayed with all of the available net classes (except Default) available. Select the Net Class combination you wish to create and select Apply. A new Type CCls (Class-Class) row will be created under the selected Net Class. You can now specify the required spacing rules for this combination either by directly entering the values in the appropriate cells, or by assigning a Spacing CSet. Make sure you work in the CCls (Class-Class) row of the worksheet.
Labs

• Lab 7-2: Setting Spacing Rules
  – Set the Default Spacing Rules
  – Define the Special Spacing Rules
  – Assign the Net Class

• Lab 7-3: Setting Class-Class Rules
  – Define a New Special Spacing rule
  – Assign Nets to a New Net Class
  – Create the Class-Class Rule

The following labs will allow you to familiarize yourself with the process and steps required to set spacing design rules in your design. You will learn how to identify the special nets, create new design rules, and apply the new design rules to the special nets. You will also proceed through the steps required to create a net class-class rule.
Lesson 7

Lesson 7 Setting Design Constraints

Lab 7-2: Setting Spacing Rules

Objective: Define spacing routing rules for special nets.

Setting the Default Spacing Rules

New designs use a 5-mil space as the default clearance. This design requires a 6-mil space for all non-critical/non-special nets.

1. If you don’t already have the OrCAD and Allegro PCB Editor tool running, start the OrCAD and Allegro PCB Editor.

2. Choose File - Open and open the constraints.brd design file you saved previously if it is not currently open.

3. Select Setup - Constraints - Spacing from the OrCAD and Allegro PCB Editor main menu.

   The Constraint Manager form opens and the Spacing section is displayed.

4. Select the Line Worksheet under the All Layers Workbook in the Spacing Constraint Set folder.

   This should be the worksheet already open, but it is good practice to make sure you have the correct worksheet open.

5. In the Default row, change all values to 6.

   a. Select the “Line to Line” cell once.
   b. Perform a shift-select in the “Line to Hole” cell to select the entire cell.
   c. If the 5.00 mil value in the “Line to Hole” cell does not become selected, double click in the same cell while holding the shift button.
   d. Enter the new value of 6 and hit the tab key.

   The entire row selected should change to 6.00.

   Make sure to perform these same procedures for the Pins, Vias, Shapes, and Hole Worksheets. Make sure to check for horizontal scroll bars in the Pins and Vias worksheets to ensure you are selecting all possible cells.

Defining the Special Spacing Rules

Assume the nets VCLKA and VCLKC require a larger clearance (8 mils) than the default values. First, create the new rules by creating a new Spacing CSet.
1. Select the **Default** cell in any worksheet under the All Layer Workbook under the Spacing Constraint Set folder.

![Image](image1.png)

2. Select **Objects - Create - Spacing CSet** from the Constraint Manager menu or the **Right-Mouse-Button - Create - Spacing CSet** popup menu command.

3. In the Create Spacing CSet form, enter **8_mil_space** and select the **OK** button.

4. In the **8_MIL_SPACE** row, change all values to **8**. Make sure to do this for the **Lines**, **Pins**, **Vias**, **Shapes**, and **Hole** Worksheets. Your Constraint Manager should look similar to below:

![Image](image2.png)

### Assigning the Net Class

When you created the SYNC net class in the Physical Rule section, you created that class in both the physical domain and in the spacing domain. All that is left to do now is to assign the SYNC net class to use the **8_MIL_SPACE** rule set you just created.

1. Select any worksheet under the **All Layers** workbook under the **Nets** folder.

2. Select the **Referenced Spacing CSet cell** in the **SYNC** row and select **8_MIL_SPACE** from the pull-down menu as shown below.

![Image](image3.png)

Now the SYNC net class, which contains the two special nets VCLKA and VCLKC, will use the **8_MIL_SPACE** rule such that when either of these two nets is routed, all etch will remain 8 mils apart.
Note

The SYNC Net class was created in the Spacing Domain when you checked the option “Create for both physical and spacing” option when you created the SYNC Net class in the Physical labs.

3. Save the drawing and continue by clicking File - Save from the OrCAD and Allegro PCB Editor main menu.

4. Click Yes to confirm the overwrite.

The constraints.brd file is once more saved to disk.

End of Lab
Lab 7-3: Setting Class-Class Rules

Objective: Create a new class of nets and set a class-class rule.

Assume the nets VD0...VD7 are critical and they must NOT interfere with the SYNC nets (VCLKA and VCLKC). The spacing required between these new nets and the SYNC nets must be 15 mils at a minimum. You will create a new 15-mil space rule, assign the VD* nets into a new net class, and create a Class-Class rule between the VD* nets and the SYNC nets using the new 15-mil space rule. First, you must create the new spacing rule.

Defining a New Spacing Rule

1. Select the Line Worksheet under the All Layers Workbook in the Spacing Constraint Set folder.

2. Select the Default cell.

3. Select Objects - Create - Spacing CSet from the Constraint Manager menu or the Right-Mouse-Button - Create - Spacing CSet popup menu command.

4. In the Create Spacing CSet form, enter 15_mil_space and select the OK button.

5. In the 15_MIL_SPACE row, change all values to 15. Make sure to do this for the Lines, Pins, Vias, Shapes, and Hole Worksheets. Your Constraint Manager should look similar to below:

<table>
<thead>
<tr>
<th>Type</th>
<th>Objects</th>
<th>Line</th>
<th>Thru Pin</th>
<th>SMD Pin</th>
<th>Test Pin</th>
<th>Thru Via</th>
<th>BI Via</th>
<th>Test Via</th>
<th>Microvia</th>
<th>Shape</th>
<th>Bond Finger</th>
<th>Hole</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>constraints</td>
<td>8.00</td>
<td>8.00</td>
<td>8.00</td>
<td>8.00</td>
<td>8.00</td>
<td>6.00</td>
<td>6.00</td>
<td>6.00</td>
<td>6.00</td>
<td>6.00</td>
<td>6.00</td>
</tr>
<tr>
<td>SCS</td>
<td>DEFAULT</td>
<td>6.00</td>
<td>6.00</td>
<td>6.00</td>
<td>6.00</td>
<td>6.00</td>
<td>6.00</td>
<td>6.00</td>
<td>6.00</td>
<td>6.00</td>
<td>6.00</td>
<td>6.00</td>
</tr>
<tr>
<td>SCS</td>
<td>8_MIL_SPACE</td>
<td>8.00</td>
<td>8.00</td>
<td>8.00</td>
<td>8.00</td>
<td>8.00</td>
<td>8.00</td>
<td>8.00</td>
<td>8.00</td>
<td>8.00</td>
<td>8.00</td>
<td>8.00</td>
</tr>
<tr>
<td>SCS</td>
<td>15_MIL_SPACE</td>
<td>15.00</td>
<td>15.00</td>
<td>15.00</td>
<td>15.00</td>
<td>15.00</td>
<td>15.00</td>
<td>15.00</td>
<td>15.00</td>
<td>15.00</td>
<td>15.00</td>
<td>15.00</td>
</tr>
</tbody>
</table>

Assigning Nets to a New Net Class

Now that you have created the spacing routing rules for the special nets, you need to assign the VD* nets to a Net Class.
1. Select the **Line Worksheet** under the **All Layers Workbook** in the **Net** folder as shown below (you may need to expand the **All Layers** workbook by selecting the “+” character to the left of All Layers):

2. Select the **constraints** cell under the **Objects** column with the Left-Mouse-Button.

3. Select the **constraints** cell with the Right-Mouse-Button, and from the Right-Mouse-Button pop-up menu select **Create - Net Class**.

4. In the **Create Net Class** form, enter **VDNETS** for the name, **unselect** the option to create the net class for both physical and spacing, and select the **OK** button.

5. Select the **VDNETS** cell with the Left-Mouse-Button.

6. Select **Objects - Add to - Net Class** from the Constraint Manager menu or the **Right-Mouse-Button - Net Class members...** popup menu command.

   If the nets are not listed in the **All Nets** window, perform step 7. If the nets are listed in the **All Nets** window, proceed to step 8.

7. Select the scroll arrow next to Net in the upper left corner and select Net.

8. Scroll through the nets section to make sure that all of the VD* nets are visible.

9. Select the net **VD0**, and shift-select the net **VD7** so that all nets are selected.
10. Select the “>” button to assign the selected nets to the net class VDNETS.

![Net Class Membership for VDNETS]

11. Select **OK** to assign the nets and close the form.

**Note**
The previous steps to assign the VD* nets to the VDNETS net class is an alternate method than you used when assigning the VCLKA and VCLKC nets to the SYNC net class. You can use whichever method you prefer.

**Creating the Class-Class Rule**

The final step is to create a class-class rule between the SYNC net class and the VDNETS net class to use the 15 mil space rule you created.

1. Select the **Line Worksheet** under the **All Layers Workbook** in the **Net Class-Class** folder as shown below (you may need to expand the **All Layers workbook** by selecting the “+” character to the left of All Layers):
2. Select the **VDNETS** cell.

3. Now select the **VDNETS** cell with the Right-Mouse-Button and select *Create - Class-Class* as shown below:

4. Select **VDNETS** on the left side pane and **SYNC** on the right side pain as shown below:

5. Select the **OK** button.
6. Select in the **Referenced Spacing CSet cell** for the **Sync Net Class-Class cell**. Select the **15_MIL_SPACE CSet** as shown below.

![Referenced Spacing CSet](image)

You have now set a spacing rule such that any time a SYNC net is routed next to a VDNETS net, the air gap between the two nets will be 15 mils.

7. Select **File - Close** from the Constraint Manager window.

8. Save the drawing and continue by clicking **File - Save** from the OrCAD and Allegro PCB Editor main menu.

9. Click **Yes** to confirm the overwrite.

   The `constraints.brd` file is once more saved to disk.

STOP **End of Lab**
Lesson 7  Setting Design Constraints

**Analysis Modes**

The Analysis Modes dialog box contains the modes and options for the Design, Electrical, Physical, Spacing, Same Net Spacing, and SMA Pin checks. These modes are accessible in the Constraint Manager from the Analyze - Analysis Modes menu command. In the OrCAD and Allegro PCB Editor they are also available from the Setup - Constraints - Modes menu command.

**Design Options and Modes**

The Design Options window specifies plane, testpoint, and mechanical hole parameters for the enabled Design Modes checks.

The Design Modes window controls whether specific design rules checks for planes, testpoint, and mechanical holes are triggered based on changes to the board layout.

- **Negative plane islands** checks for isolations when using a negative plane.

- **Negative plane islands** oversize scales up the pad geometry before the negative plane islands
Negative plane sliver flags “slivers” or small undesired webs of copper between two objects, usually formed by placing antipads or thermal pads of pins or vias spaced too close to other padstack items or the negative plane boundary.

Testpoint to component specifies spacing checks between edges of testpoint pads and components.

Testpoint location to component specifies spacing checks between testpoint locations (center) and components.

Testpoint under component flags testpoints under components.

Mechanical pin to mechanical pin checks for a minimum spacing between mechanical holes.

Mechanical pin to conductor checks for a minimum spacing between mechanical holes and conductors.

Minimum metal to metal checks to ensure minimum metal to metal clearance is met.

Duplicate drill holes detects duplicate drill holes spanning the same layers.

Design Options and Modes (Soldermask)

Analyze - Analysis Modes

Design Options (Soldermask)
Lesson 7 Setting Design Constraints

The Design Options (Soldermask) window specifies soldermask and pastemask parameters for the enabled Design Mode checks.

**Soldermask alignment** specifies and checks the alignment tolerance required for the proximity of package soldermask to placebound and the pad soldermask to pad geometry.

**Soldermask to soldermask** specifies spacing checks for pad soldermask to pad soldermask, symbol soldermask to symbol soldermask, and symbol soldermask to pad soldermask.

**Soldermask to pad and cline** specifies spacing checks between the soldermask and pads and connect lines.

**Soldermask to shape** specifies spacing checks between the soldermask and shapes.

**Pastemask to pastemask** specifies spacing checks for pad pastemask to pad pastemask and pad pastemask to package based pastemask.

**Design Modes (Package)**

![Design Modes (Package)](image)

**Analyze - Analysis Modes**

*Design Modes [Packages]*
The Design Modes (Package) window specifies whether the Package to Package, Package to Place Keepin, and Package to Place Keepout checks will be generated if there is any overlap between the appropriate type of shapes.

**Package to package** flags packages that overlap one another.

**Package to place** keepin flags packages that extend beyond a place keepin.

**Package to place** keepout flags packages that extend inside a place keepout.

**Package to room** flags packages that are located in rooms to which they are not assigned.

**Embedded DRCs - Package height to layer** spec

---

**Electrical Modes**

![Electrical Modes](image)

The Electrical Modes control whether a specific design rule checks for electrical modes are triggered based on changes to the layout.

**Total etch length** checks for the minimum and maximum etch requirements for Nets.
All differential pair checks for uncoupled length, and minimum line spacing between the two nets.

Physical Modes

The Physical Modes window specifies whether or not the respective physical constraints should be checked. Checks can be made for the following conditions:

- Min line width, Min neck width, and Max neck length
- Max line width
- Allow etch on subclass
- Allow T junctions on subclass
- Min and Max blind/buried via stagger
- Pad-pad direct connect, and Vialist DRC.
Spacing and Same Net Spacing Modes

The Spacing Modes and Same Net Spacing Modes windows specify whether or not the respective spacing constraints should be checked. Both of these modes enable or disable the testing of the exact same spacing conditions between lines, pins, via, shapes, bond fingers, holes, and blind/via gap.

The Spacing Modes cover these spacing conditions when they are applied to different nets. Example: an ENABLE via connect line near a RESET pin - spacing requirement of 8 mils.

The Same Net Spacing Modes cover these spacing conditions when they are applied to the exact same net so that you can place the elements closer to each other. Example: an ENABLE via close to an ENABLE pin - spacing requirement can be 4 mils.

Note
If any of the spacing categories are blank then they are not being checked for. If any spacing category box has a gray square in it then not all selections in that category is enabled for checking.
SMD Pin Modes

The SMD Pin Mode ensures that the placement of vias is properly contained within SMD pads.

- The Via at SMD fit
  - **On** denotes that the via pad must be contained within the SMD pad.
  - **Off** denotes that the center of the via cannot extend beyond the edge pad.

- The Via at SMD thru
  - **On** denotes that the thru vias are allowed in SMD pads.
  - **Off** denotes that the thru vias are not allowed in SMD pads.

- The Etch turn under SMD pin checks to detect etch compensation buried within the pad.
Property Assignments and Changes

It is important to understand that there is overlap between properties and constraints. Properties override constraint values. For example, a design contains a special net class with an assigned physical rule set. This rule set calls out a Minimum Line Width of 8 mils. If the property min_line_width is set to 10 and is assigned to one or more nets from that group, those nets will obey the property value rather than the physical rule set value. Therefore, in this case, the net will be routed at a 10-mil line rather than an 8-mil line.

When you select the Edit - Property command, you must first identify the elements for property assignment. Use the Find Filter form to select elements either by pick or by element type plus name or list. Use the Find By Name/Prop section of the Find Filter to identify elements with existing properties. The OrCAD and Allegro PCB Editor tool then displays the properties available for that element type. Two examples of element types and their properties are:

- Components and component properties
- Nets and net properties

Once an element is identified, the Edit Property form appears. The Edit Property form lets you assign properties to design elements, or delete or modify the current values of an assigned property.
Select the properties you want to attach from the scroll list and click on the **Apply** button. Some properties require values (for example, min_line_width) while others do not. To modify existing property values, follow the same process. To remove an existing property, click the **Delete** button next to the selected property before applying.

The icons **Fix** and **Unfix** have been added as an aid in quickly adding and deleting the fixed property to any object.

**Constraint Overrides**

![Step 1: Edit Property](image)

<table>
<thead>
<tr>
<th><strong>Step 1</strong></th>
<th>Find foldable window</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Setup – Constraints – Physical Overrides</strong></td>
<td><strong>Setup – Constraints – Spacing Overrides</strong></td>
</tr>
</tbody>
</table>

The two commands **Setup - Constraints - Physical Net Overrides** and **Setup - Constraints - Spacing Net Overrides** are commands that you can use to change the routing of a net. This is accomplished by adding predefined properties to nets. Any values that you assign via these commands will override the constraints as defined in the Constraint Manager. When a property is assigned to a net, it will override the Constraint Manager value.

For example, suppose net ABC uses the Default Physical rules in the Constraint Manager, and the Default Physical rules specify a minimum line width of 5 mils. If you use the **Setup - Constraints - Physical Net Overrides** command and assign the property Min_Line_Width to net ABC with a value of 10 mils, when you route the net it will be routed at 10 mils. All other rules for the net (such as Max_Line_Width, Min_Neck_width, and so on) will be determined by the Default Physical rule set.
Physical Constraint Resolution

While routing your design, you may wonder why the routing of a net has certain physical characteristics, like why is it routed at the width displayed. You can use the Display - Constraint command to generate a report of the constraint information.

To show the physical attributes, execute a single select on an element. The information displayed will be in two sections. The top section will include information about the element picked, such as the x/y location, net name, and so on. The bottom section will display the Constraint rules used for the selected item such as the Constraint set name, constraint set rules, and the constraint values.
Spacing Constraint Resolution

While routing your design, you may wonder why the routing of a net has certain spacing characteristics, like why is the air gap between the route and a pin a certain value. You can use the Display - Constraint command to generate a report of the constraint information.

To show the spacing attributes, drag a window around the two elements. The information displayed will be in two sections. The top section will include information about the elements picked, such as the x/y locations, net names, and so on. The bottom section will display the Constraint rules used for the selected item such as the Constraint set name, constraint set rules, and the constraint values.
**DRC Marker Display**

DRC markers store the following information about a design rule violation:

- DRC class, subclass, and location
- Type of constraint set (physical or spacing)
- Name of constraint set
- Constraint type being violated (for example, *Line to Thru Pin Spacing*)
- Data concerning first element in violation (type of element, location, refdes, if a package/part, and so on)
- Data concerning second element (if there is one) in violation (type of element, location, refdes, if a package/part, and so on)
- To display a filled DRC marker, enable `display_drcfill` in the Display category of the User Preferences

DRC markers have two characters, one in each side of the ‘bow-tie’, that identify the type of constraints violation being marked. Each character is a key as to what type of violation exists. In the example shown, the “L” represents a “Line.” The “K” represents a “Keepout” (such as a routing keepout). So therefore, in this case, this is a line to routing keepout violation. In other words, a piece of etch exists in an area that has been identified as a routing keepout area.

To display the DRC filled, as shown in the example, enter on the OrCAD and Allegro PCB Editor command line “set display_drcfill” or use the User Preferences Editor. The `display_drcfill` option can be found under the Display category.
Lab

- Lab 7-4: Working with Properties
  - Learn how to use the Edit Properties form to add, delete, and change property value assignments
    - Attach properties to components
    - Add a ROOM property
    - Add properties to nets
    - Show existing properties on design elements
    - Delete properties

The following lab will let you familiarize yourself with the process required to work with the design constraints and add, modify and delete properties. You will learn how to modify the design constraints, attach properties to nets, components and areas, show existing properties, and delete properties from database elements.
Lab 7-4: Working with Properties

Objective: Attach, display, and delete properties in a design.

Attaching Properties to Components

1. Start the OrCAD and Allegro PCB Editor tool and open the constraints.brd file in your working directory if it is not already the open design.

2. Choose **Edit - Properties** from the top menu.

3. In the **Find By Name** section of the **Find Filter** (Find foldable window), click the scroll button to set the field description box to **Comp (or Pin)**.

4. Click in the text entry field (>), and enter:
   
   j1

   When you press the Tab key, the Edit Property and the Show Properties forms appear. Notice that the J1 connector has no properties attached to it.

5. In the **Edit Property** form, select the **Hard_Location** and **Fixed** properties from the scroll list.

   These properties appear on the right.
6. Toggle the Property Values to **TRUE** if required.

![Edit Property Window](image)

7. Click **Apply**.

   In the *Show Properties* window, the properties **HARD_LOCATION** and **FIXED** are added to component J1.

![Show Properties Window](image)

**Note**

The FIXED property prevents the component from being moved. The HARD_LOCATION property prevents the component reference designator from being changed during the automatic rename process.

8. Click **Close** to close the *Show Properties* window.

9. Click **OK** to close the *Edit Property* form.
Attaching Fixed Properties to Symbols Using Icons

In the last lab you added properties by using a form. An easier way to add a Fixed property to an object is explained in this lab. You will add a Fixed property to the board outline symbol and the two BNC connectors. This step can be done while you are defining the template for the master board design.

1. Select the **Fix** icon.

2. In the Find Filter, select the All off button, then turn on only Symbols.

3. Click on the two **BNC connectors** on the right side of the board and the **board outline**.

   This adds a Fixed property to these three objects so they won’t be inadvertently moved while placing other components.

**Note**

There is also an **Unfix** icon available to delete the Fixed property from symbols. If you select the RMB while in the command, you will see a menu selection that will **Unfix All**. We will not be using this command on our design at this time.

Adding the ROOM Property to Components

1. Choose **Edit - Properties** from the top menu.

2. In the **Find By Name** section of the **Find Filter**, click the scroll button to set the field description box to **Comp (or Pin)**, if this is not already set.

3. Click **More**.

   The **Find by Name or Property** form appears.

4. Scroll through the list of component names and select **D1**, **D2**, **D3**, and **D4**. (Or you could set the **Name Filter** to d* and just these reference designators will appear.)
When you select each name, it disappears from the list on the left and is added to the list of Selected Objects on the right of the form, as shown:

5. Click **Apply** in the *Find By Name or Property* form.

Four components are now selected for editing.

6. In the *Edit Property* form, select **Room** from the list of *Available Properties* in the scroll list.

7. In the blank *Value* field next to the *Room* property, enter the room name:

   **LED**

   You want to add this in uppercase letters since property names are case sensitive.

8. Click **Apply**.
In the Show Properties window, the ROOM property is added to all four components.

9. Click OK to close the Edit Property form.

10. Click OK to close the Find By Name or Property form.

**Attaching Properties to Nets**

In this section of the lab you will attach a property to several nets in the design.

1. In the Find By Name section of the Find Filter, click the scroll button to set the field description box to Net.

2. Click in the blank field under Net, and enter:
   
   vcc

When you press the Tab key, the Edit Property and the Show Properties forms appear.

**Note**

Pre-existing properties in this net were displayed.

3. Scroll the list in the Edit Property form and click on Min_Line_Width.

This property now appears in the right side of the table.
4. In the blank field next to **Min_Line_Width** property, enter the value of the line width:

   15

5. Click **Apply**.

   In the **Show Properties** window, the **MIN_LINE_WIDTH** property is added to the net **VCC**.

6. Follow the same steps (2 through 5) to attach the **MIN_LINE_WIDTH** property to net **GND**, and set the value to **15 MIL**.

7. Click **OK** to close the Edit Property form.

8. Right-click and choose **Done** to exit from the **Edit - Property** command.

**Showing Existing Properties on Elements in the Design**

There are several ways to display properties attached to elements in the design. The **Edit - Property** command lets you identify the parts or nets in which you are interested. The Show Properties window lets you identify the properties in which you are interested.

1. From the top menu, choose **Edit - Properties**.

2. Click in the blank field under **Net** in the **Find By Name** section, and enter:

   *
When you hit the Tab key, the Edit Property form displays a list of any property that has been attached to the nets in the current database.

The Show Properties window displays all of the nets in the design, and the properties attached to each net.

3. In the Show Properties window, click the Save icon to open the File - Save As form.
   A browser form appears.

4. Enter the following name to save the file:
   netprops

5. Click Save in the browser form.
   The file netprops.txt is written to the current working directory. This file contains the same information as the Show Properties window, and can be used to check property assignments for the design.

6. Click the X icon to close the Show Properties form.

7. Click OK to close the Edit Property form.

8. Choose Display - Property from the top menu.
   The Show Property form appears. It contains a scrollable list of properties.

9. Select the Room property from the list of Available Properties.

10. Click the Show Val button.
    The Show window displays a list of functions and components with the ROOM property attached.

   **Note**
   You assigned the ROOM property to D1, D2, D3 and D4 with the value of LED. All other assignments were made in the schematic/net list.

11. Click the X icon to close the Show form.

12. Click OK in the Show Property form.

**Deleting Properties**

1. Choose Edit - Properties from the top menu.

2. In the Find By Name section of the Find Filter, click the scroll button to set the field description box to Comp (or Pin).
3. Click in the blank field under **Comp (or Pin)**, and enter:

   \[ \text{j1} \]

   When you hit the Tab key, the *Edit Property* dialog box appears and the Show Properties window displays all of the properties attached to J1.

4. Enable the box on the left side of the property named **HARD_LOCATION**, as shown, then click **Apply**.

   ![Edit Property Window](image)

   The property disappears from the *Show Properties* window. These steps can be used whenever you need to delete a property from an element.

5. Click **OK** to close the *Edit Property* form.

6. Right-click and choose **Done** to exit the *Edit - Properties* command.

7. Choose **File - Save** from the top menu.

   A Save window appears, prompting you to decide whether you want to overwrite the existing `constraints.brd` file.

8. Click **Yes**.

   The `constraints.brd` file is saved to disk.

9. Choose **File - Exit** from the top menu of the OrCAD and Allegro PCB Editor to exit the OrCAD and Allegro PCB Editor software.

   **End of Lab**