2

Lesson 2: Managing the PCB Editor Work Environment

Learning Objectives

In this lesson you will:

◆ Control the color and visibility of objects.

◆ Create and use scripts.

◆ Use the Control Panel to locate board database objects and report information about them.

In this section you will familiarize yourself with the user interface and understand how you can streamline repetitive tasks. You will also view the PCB Editor classes and subclasses, work with setting colors and visibility of objects, and learn how to use the *Display - Element* command to query design objects.

Folders and Classes and Subclasses

Folder	Classes	Subclasses
Board Geometry	Board Geometry	Outline, Plating_Bar, Assembly_Notes, Tooling_Corners, Dimension, Place_Grid_Top, Place_Grid_Bottom, Top_Room, Bottom_Room, Both_Rooms, Switch_Area_Top, Switch_Area_Bottom, Silkscreen_Top, Silkscreen_Bottom, Assembly_Detail, Soldermask_Top, Soldermask_Bottom, Off_Grid_Area
Package Geometry	Package Geometry	Assembly_Top, Assembly_Bottom, Place_Bound_Top, Place_Bound_Bottom, Pin_Number, Pad_Stack_Name, Silkscreen_Top, Silkscreen_Bottom, Body_Center, Soldermask_Top, Soldermask_Bottom, Display_Top, Display_Bottom, Modules, Dfa_Bound_Top, Dfa_Bound_Bottom, PasteMask_Top, PasteMask_Bottom
Manufacturing	Manufacturing	Photoplot_Outline, No_Gloss_All, No_Gloss_Top, No_Gloss_Bottom, No_Gloss_Internal, Ncdrill_Legend, Ncdrill_Figure, Probe_Top, Probe_Bottom, Autosilk_Top, Autosilk_Bottom, No_Probe_Top, No_Probe_Bottom
Drawing Format	Drawing Format	Outline, Title_Block, Title_Data, Revision_Block, Revision_Data

---

A design file is a composite of a number of drawing layers. The drawing elements of each of these layers can be selectively colored and turned on or off as visible or invisible layers.

The PCB Editor organizes drawing layers into a hierarchy of classes and subclasses. Each class/subclass has its own color and visibility settings. Folders are classes that have been combined together to aid you in controlling the color and visibility.

More Folders and Classes and Subclasses

Folder	Classes	Subclasses
Stackup/Conductor	Pin, Via, Etch, DRC, Anti Etch, Boundary	Top, Bottom (and all other user created board design layers)
Stackup/Non-Conductor	Pin, Via, Etch, DRC, Anti Etch, Boundary	Soldermask_Top, Soldermask_Bottom, Pastemask_Top, Pastemask_Bottom, Filmmasktop, Filmmaskbottom, Through All, Package_Top, Package_Bottom
Components	Comp Value, Dev Type, Ref Des, Tolerance, User Part	Assembly_Top, Assembly_Bottom, Silkscreen_Top, Silkscreen_Bottom, Display_Top, Display_Bottom
Areas	Route Ko, Via Ko, Package Ko, Package Ki, Route Ki, Constraints Region	Top, Bottom (and all other user created board design layers), Through All, Inner_Plane_Layers, Inner_Signal_Layers, Outer_Layers
Display	Grids, Rats top-top, Rats top-bottom, Rats bottom-bottom, Temporary Highlight, Background, Waived DRCs, Drill holes, Via label	*Subclasses not applicable*

All graphical items are stored in what is basically a two-level database scheme. The first level is referred to as a class. There are predefined classes inside the PCB Editor database. You cannot delete or add folders or classes.
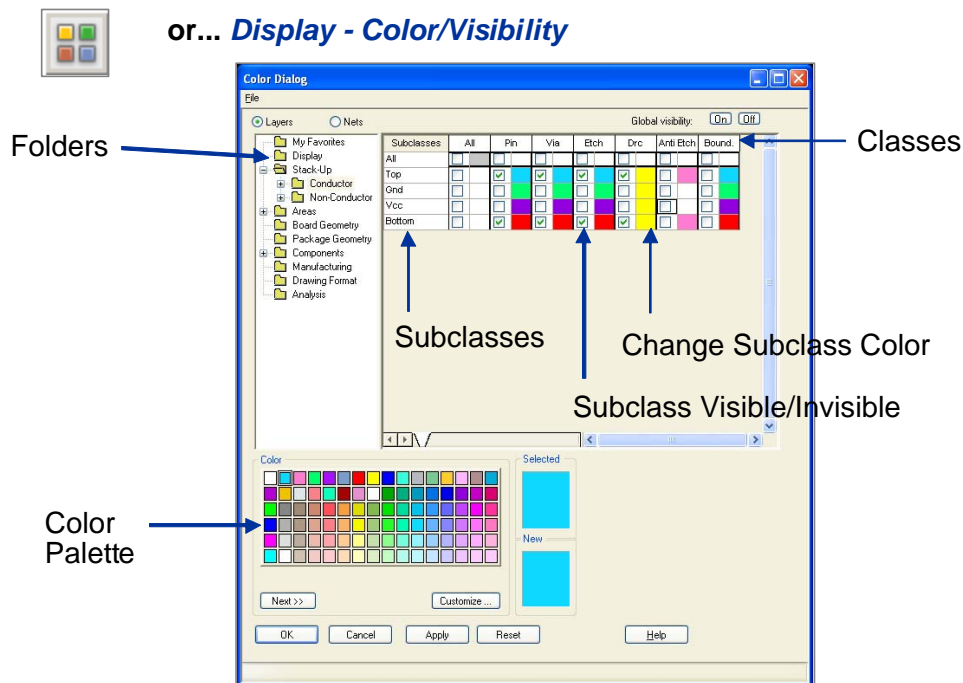
Under each class there are many subclasses. These subclasses are the second level of the database. Subclasses are often referred to as layers in the design. You cannot delete predefined subclasses, although you can add and delete your own user-defined subclasses. For example, when you want to create the outline for your printed circuit board, you draw it on a class called Board Geometry, with a subclass called Outline.

All of the board routing will appear on the subclasses under the class called Etch. This subclass has special DRC checking properties that other classes and subclasses do not have. You need to create a subclass for each layer of the printed circuit board. Thus, if you have a six-layer printed circuit board, you need to have six subclasses under the class called Etch.

Folders categorize the classes into different sections. This is used for display purposes only and only appears in the Color Dialog form.

Note that under the Etch class there is a subclass called Top and a subclass called Bottom. These names cannot be changed, nor can these two subclasses ever be deleted.

Controlling Color and Visibility

or... *Display - Color/Visibility*

Folders

Classes

Subclasses

Change Subclass Color

Subclass Visible/Invisible

Color
Palette

You display the Color Dialog form by selecting the *Display - Color/Visibility* option from the top menu or by selecting the **Color192** icon. You use this form to turn on or off the visibility for subclasses, as well as to set colors for subclasses.

To turn on or off the visibility of a specific subclass, you simply click the box to the left of the subclass. A checkmark "✓" or an "X" in this box indicates that the subclass is currently visible. If there is no checkmark "✓" or "X" in that box, that subclass is currently invisible. To change the color of a subclass, first you select the desired color from the bottom portion of the Palette dialog box, and then you select the color icon to the right of the desired subclass.

At the top right of this dialog box you see the Global Visibility buttons. With these buttons you can make all subclasses visible or invisible in the design at one time.

Remember that in order to turn a subclass off or on, you must first select the appropriate folder in the left side of the form.

Information about colors assigned to individual layers, and which layers are visible and invisible, is stored in the PCB Editor database.

The **ALL** columns and rows can be used to turn on all of the visibility for the classes or subclasses, and can also be used to set the same color for all of the classes or subclasses.

The Color Dialog form has been enhanced to support Net element based coloring. Net elements consist of pins, vias, clines, shapes, and rats. Each one can be colored uniquely across the net. Filtering is available to display just nets with color overrides.
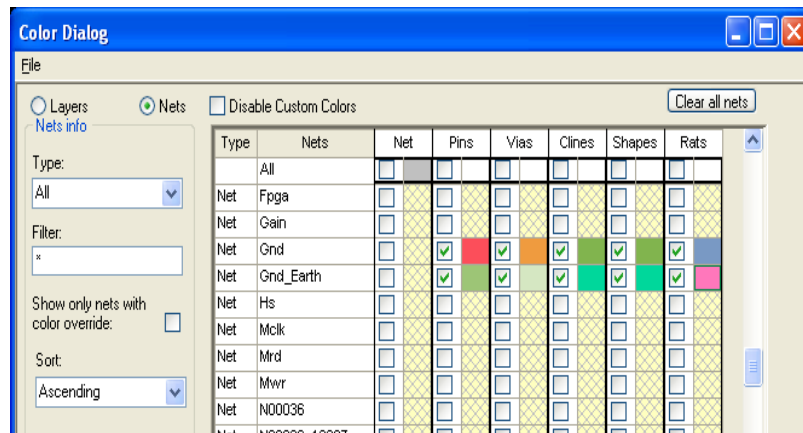
A related command is *Display - Assign Color* which assigns a color and highlights an element without requiring the use of the Color Dialog form.

Controlling Etch Visibility
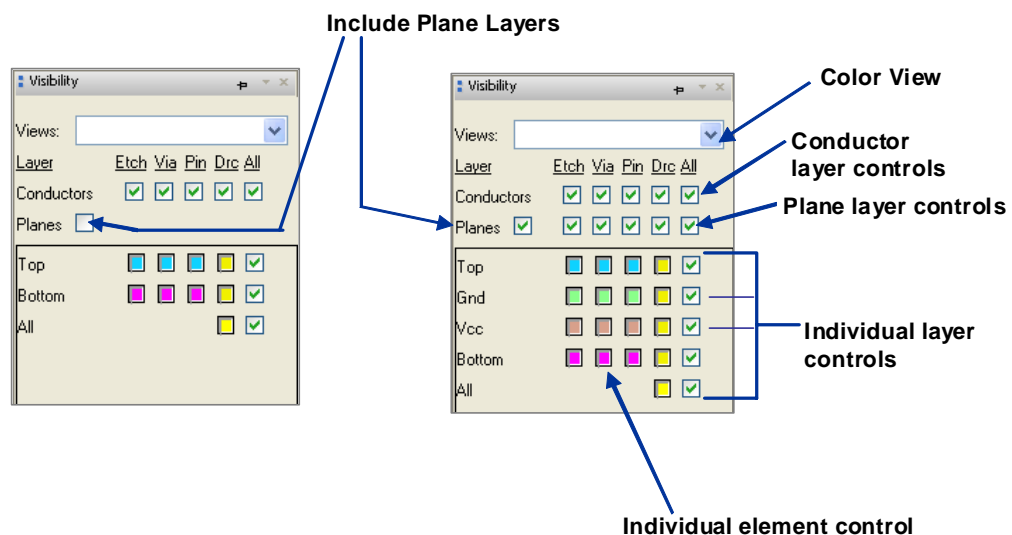
Using the Control window is a quick way to turn on or off layers or elements contained in a design. You can separately control the etch routing layers from the plane layers, as well as Etch, Pins, Vias and DRCs.

Conductor Controls

The Conductor check boxes let you individually turn on or off all etch, pin, vias or DRCs for all layers defined as conductor. By selecting the All check box, you can turn on and off all etch, pins, vias and DRCs for all conductor layers.

Plane Controls

The Planes check boxes let you individually turn on or off all etch, pin, vias or DRCs for all layers defined as plane. By selecting the All check box, you can turn on or off all etch, pins, vias or DRCs for all plane layers. If you check the Include Planes Box, you will see all the plane layers listed in this visibility form.
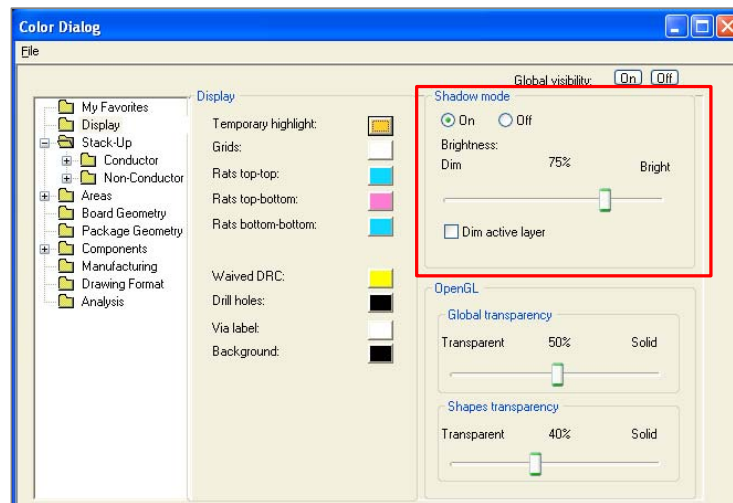
Individual Layer Control

By selecting the check box under the All column in the individual layer row, you can turn on or off all etch, pins, vias or DRCs for that layer.

Individual Element Control

You can turn on or off a single element (etch, pin, and so forth) by selecting the element.

Graphics Dimming or Shadow Mode


Toggles on/off Shadow Mode

The Graphics Dimming or Shadow Mode option gives you the ability to provide distinct levels of visibility that are based on the importance of the object. The main control for shadowing is located in the Color Dialog form under the Display folder. With Shadow Mode turned on, the brightness slide bar controls the color intensity of the non-important objects. The higher the brightness percentage, the less difference in color between the important and the non-important objects. After changing the Brightness factor using the slider, select the **Apply** button to see the changes in the PCB Editor. You use the **Shadow Toggle** icon to turn on and turn off the shadowing feature.

Objects of importance are defined as follows:

- Items that have been highlighted using the **Highlight** command

- Items that are highlighted by the current active command

- The current Active layer as defined in the Options window

The default is to have Shadow Mode disabled. When Shadow Mode is first enabled, the default brightness is 50 percent.
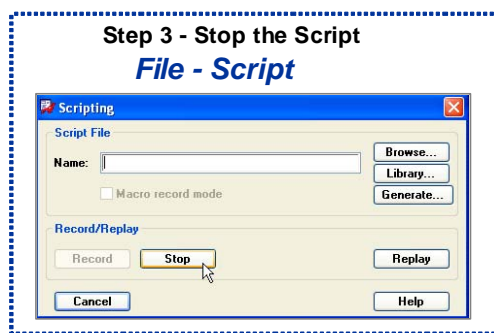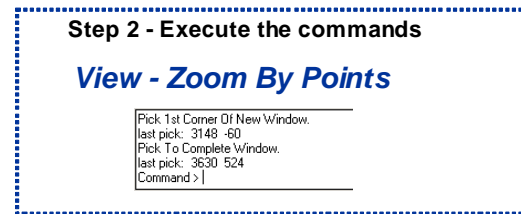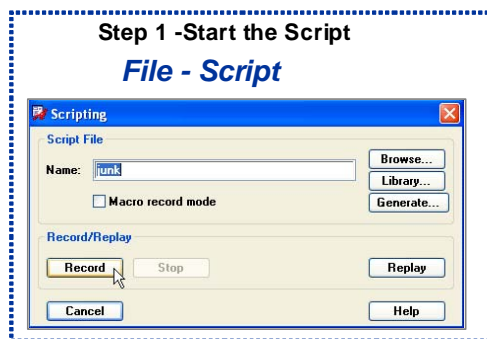
Options Window

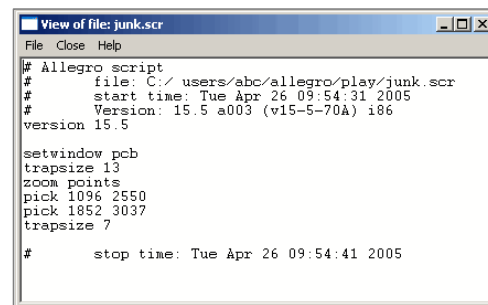| Edit - Change	Route - Connect	Add - Text
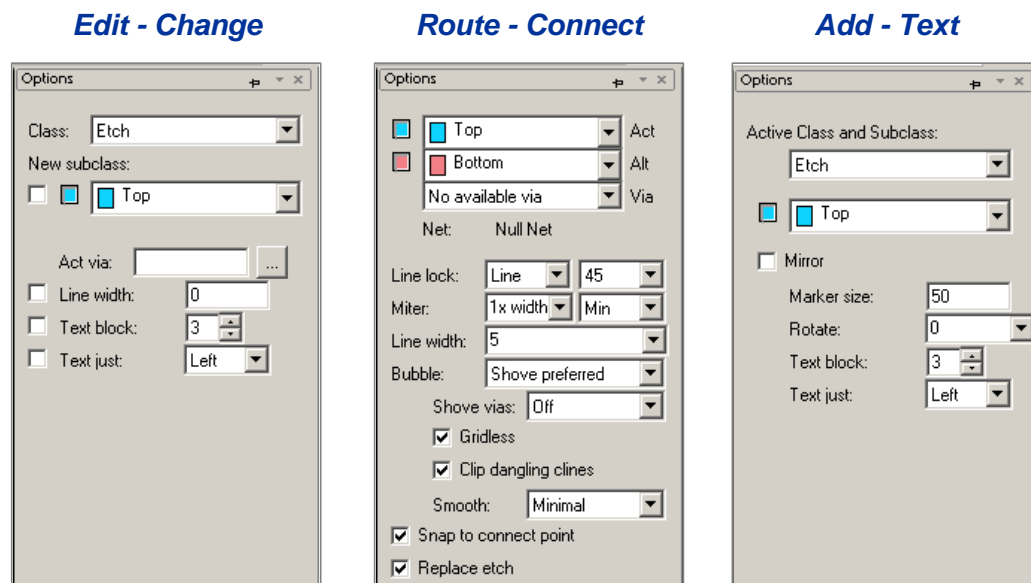
The Options window contains parameters that are used to control the current interactive command. You will notice that the parameters change from command to command.

The parameters and values you set in the Options window take effect immediately. They override definitions for the same parameters and values that may exist elsewhere in the PCB Editor software. For example, in the *Add - Text* command, the PCB Editor tool looks to the Design folder tab for the default values, which was set using the *Setup - Design Parameters* command for the rotation and text values. If you place a different value in the Options window, however, the tool uses the values you enter instead of the default values.

Scripts

Step 1 -Start the Script

File - Script

Step 2 - Execute the commands

View - Zoom By Points

```
Pick 1st Corner Of New Window.
last pick:  3148  -60
Pick To Complete Window.
last pick:  3630  524
Command > |
```

Step 3 - Stop the Script

File - Script

Resulting Script

```
# Allegro script
#       file: C:/ users/abc/allegro/play/junk.scr
#       start time: Tue Apr 26 09:54:31 2005
#       Version: 15.5 a003 (v15-5-70A) i86
version 15.5

setwindow pcb
trapsize 13
zoom points
pick 1096 2550
pick 1852 3037
trapsize 7

#       stop time: Tue Apr 26 09:54:41 2005
```

With scripts, you can have the PCB Editor record and save all your menu selections and mouse picks in a text file. You initiate such script recording by clicking **Record**. While a script is recording, the script file name appears in the Status window. All your executed commands will be recorded in a text file, until you stop the recording. You can then replay the file in the same design or a different design to quickly execute repetitive operations.

Browse displays a script file browser that lets you choose a script file to replay.

Library displays a script file browser that opens your script path location (*env* file) and lets you choose a script file to replay.

Generate displays a file browser from which you can choose a `.jrl` file to convert into a script file.

Macros are like scripts in that they let you perform repetitive actions, such as complex geometric operations, on a drawing. The difference, however, is that scripts record from absolute coordinates while macros record from relative coordinate positions in a drawing. To start recording a macro, you enable the Macro Record Mode check box.

Lab

◆ Lab: Script Files and Controlling Visibility and Color

❑ Starting a script file recorder

❑ Controlling visibility

❑ Setting colors

❑ Stopping the script file recorder

❑ Testing the script file (`colors.scr`)

❑ Using the shadow mode option

Lab 2-1:   Script Files and Controlling Visibility and Color

Objective: Set up a script file to control color and visibility of graphical elements of a design.

In this lab you will change the default visibility and color assignments on each new layer to suit your personal preferences. Changing layer visibility and assigning colors is a procedure you will want to use over and over again. You can use script files to capture repetitive procedures. From the time you enter *recording* mode until the time you stop the recorder, all your activities are captured into the script file.

Starting a Script File Recorder

1. Open OrCAD PCB Designer using the `cds_routed.brd` file in the *play* directory, if you do not already have it running.

2. Select *File - Script* from the top menu.
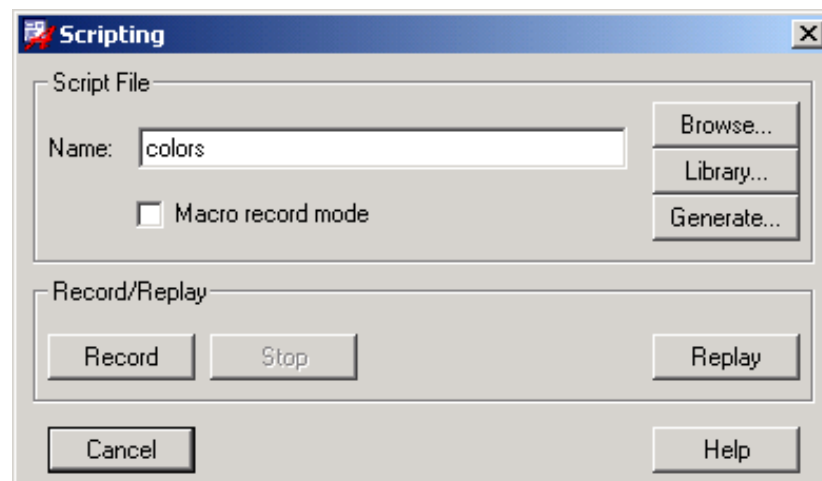
The Scripting dialog box appears.

3. Place the cursor in the File text field and type the following:

colors

Note
DO NOT press the ENTER key.

4. Click **Record**.

The Scripting dialog box disappears, and you are ready to begin recording. Each command you execute from this point forward will be entered into the script file `colors.scr`. Notice that in the Status window at the lower right corner of the Editor you will see the words *Rec colors* while you are in record mode for creating a script. In this case, *colors* is the name of the script. Later you will be instructed when to stop the recording.
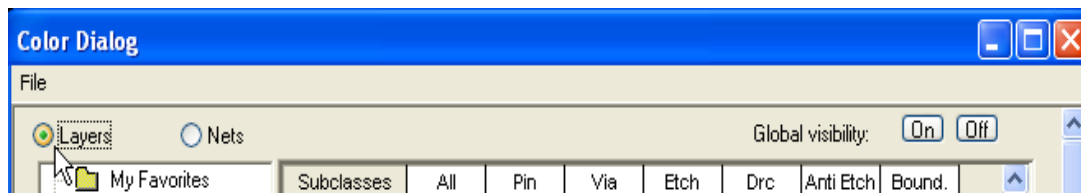
Controlling Visibility

First you can set the visibility and color assignments for the design.

**1.** Click the **Color192** icon.

The Color Dialog form appears.

2. Near the top left of the Color Dialog form, make sure that the **Layers** option is enabled. If it is not then click in the radio box to the left of **Layers** to enable the layers portion of the form.

3. Near the top right of the form, select the Global Visibility **Off** box.

4. When an alert message appears asking if you want to change all classes to invisible, click **Yes**.

This action resets all the visibilities to OFF, so you can begin turning on the layers that you wish.

5. Expand the **Components** folder, and select the Ref Des class as shown.
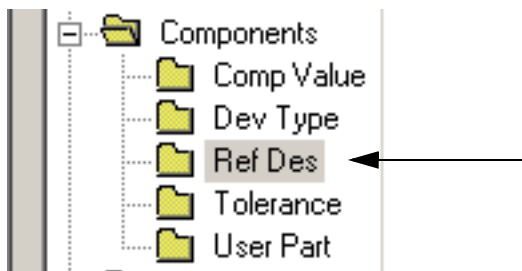
6. Under the Subclasses column, enable the visibility box for the subclass **ASSEMBLY_TOP**. A checkmark "✓" in the box indicates the subclass is turned ON.

| Subclasses	RefDes	
All	☐	
Assembly_Top	☑	
Assembly_Bottom	☐	
Silkscreen_Top	☐	
Silkscreen_Bottom	☐	
Display_Top	☐	
Display_Bottom	☐	

7. Select the **Board Geometry** folder and enable the visibility for the **OUTLINE** subclass.

8. Select the **PACKAGE GEOMETRY** folder and enable the visibility for **ASSEMBLY_TOP**.

9. Expand the **Stack-Up** folder and select the **Conductor** folder.

10. Enable visibility for subclasses in this group, as shown in the figure, then click **Apply**.

We will use the Palette section next.

Subclasses	All	Pin	Via	Etch	Drc	Anti Etch	Bound.	
All		☐	☐	☐	☐	☐	☐	
Top	☐	☑	☑	☑	☑	☐	☐	
Gnd	☐	☑	☑	☑	☑	☐	☐	
Vcc	☐	☑	☑	☑	☑	☐	☐	
Bottom	☐	☑	☑	☑	☑	☐	☐	

Controlling Colors

To change the colors of some of the subclasses in the Stack-Up folder, follow these steps.

1. Click any color button in the **Palette** section of the Color Dialog, then select the subclass color button next to **Bottom** for ETCH, PIN and VIA. (It is recommended that these subclasses all be set to the same color.)
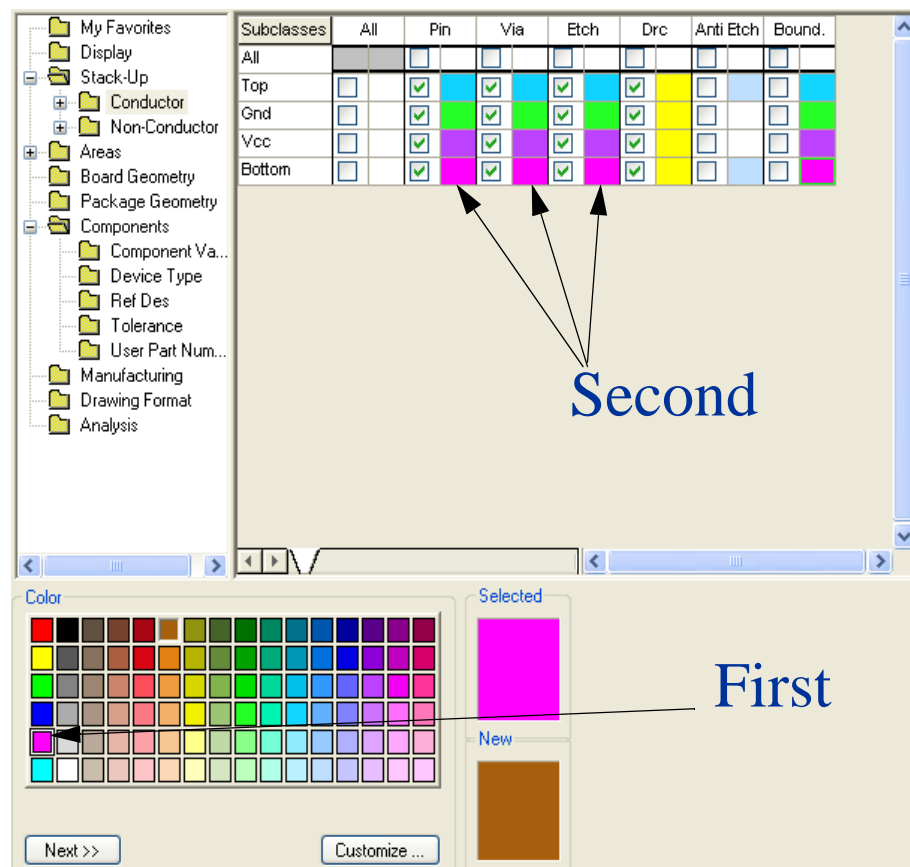
2. If you so choose, change the color of any other conductor subclass to any color you want.

3. Near the top left of the form, click in the radio box to the left of **Nets** to enable the nets portion of the form.

4. Click any color button in the **Palette**, then select the Net class color button for the subclass **GND** as shown below.
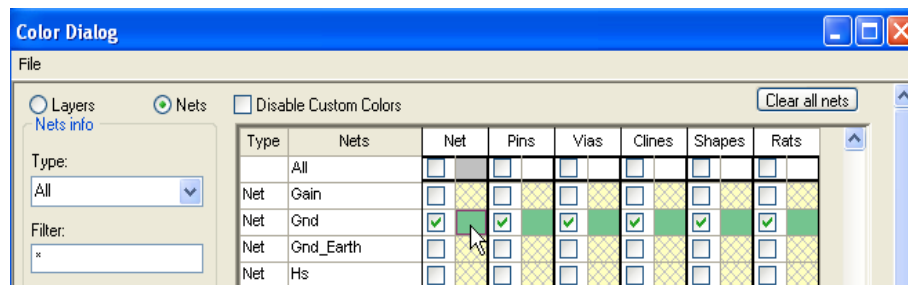
It is recommended that the classes (Net, Pin, Vias, Clines, Shapes, rats) for this net all be set to the same color. Remember that the classes appear in the top row of the selected catergory.

5. Click **OK** to apply and close the Color Dialog.

Stopping the Script File Recorder

Notice the words *Rec colors* in the Status window (lower right corner of the work area window). You are still in the script record mode, recording a script named colors.

1. Select *File - Script* from the top menu.

The Scripting dialog box appears.

2. Click **Stop** to stop the script file from recording.

All the visibility and color assignments you made have been captured in the `colors.scr` file.

3. Click **Cancel** to close the Scripting dialog box.

4. View the `colors.scr` ASCII file by selecting *File - File Viewer*. The file should be located in your *play* working directory. Be sure to change the file type in the browser menu from *(*.log)* to **All Files (*.*)** so your `colors.scr` file will display.

5. Select **colors.scr** and click **Open**. Take a look at the file.

6. Close the `colors.scr` file when you're done viewing it.

Testing the Script File (colors.scr)

1. Click the **Color192** icon.

The Color Dialog form appears.

2. Near the top right of the Color Dialog form, select the Global Visibility **Off** box.

3. When a warning appears asking if you want to change all classes to invisible, click **Yes**.

4. Click **OK** to close the Color Dialog form.

Because the visibility for all classes is turned off, nothing is displayed in the work area.

5. At the PCB Editor command line, enter the following (move your cursor over the Command tab to make the window visible):

```
replay colors
```

This command replays the script file you created, and sets the visibility and color assignments automatically.

Using the Shadow Mode Option

1. Click the **Color192** icon.

The Color Dialog form appears.

2. Select **Display** folder.

3. Select **On** for the Shadow Mode option.

4. Select and drag the Brightness slide bar. Click **Apply** to see results on the screen and stay in the form.

5. Click **OK** to apply and close the Color/Visibility menu.

Notice how the color of the current Active Class and Subclass as defined in the Options folder tab is displayed in the normal color, while all others are drawn in the dimmed color.

6. Change the Active Class in the Options window to **Board Geometry** and the Active Subclass to **Outline**.

Notice now that the board outline is drawn in the normal color and everything else except the highlighted net is displayed in the dimmed color.

7. Select the **Shadow Mode** toggle icon to turn off shadow mode.

STOP **End of Lab**

Layer Priority

Display - Layer Priority
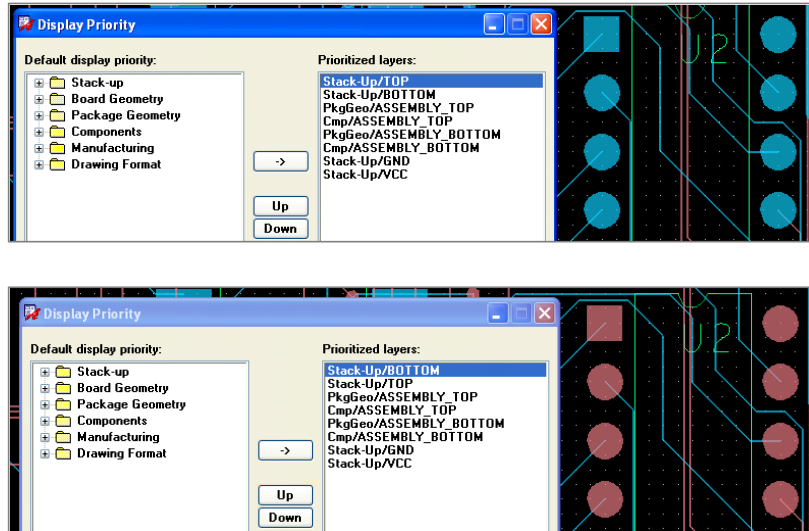
If the Active Class and Subclass in Options Window is set to a layer listed in the Prioritized layers window then the Options Window setting takes priority

For easier recognition, the Layer Priority form manages the order in which layers appear, by assigning a display priority to each layer. and overriding the default display layer. Elements are drawn based on their assigned layer priority. Your assignments are saved with the board when you click the Apply button.

Always-on-top elements include:

- Temporary or transitory objects such as DRCs or ratsnests

- Active subclasses

Example: If a Class/Subclass in the Options Window is set to one of the layers defined in the Prioritized layers window of the Display Priority form, the setting in the Options Window will take priority.

Find Window - Selectable Objects List

◆ You use the Design Object Find Filter section of the Find window when selecting database elements with the mouse.

◆ The "list" is searched for items that are checked.

◆ The first check box found when searched from top left to bottom right will be used.

Challenge: Given the settings to the right, when the Delete command is executed, what will be selected for deletion in the following two scenarios (note the cursor location)?
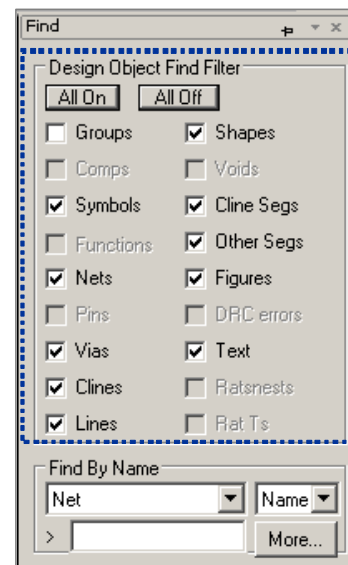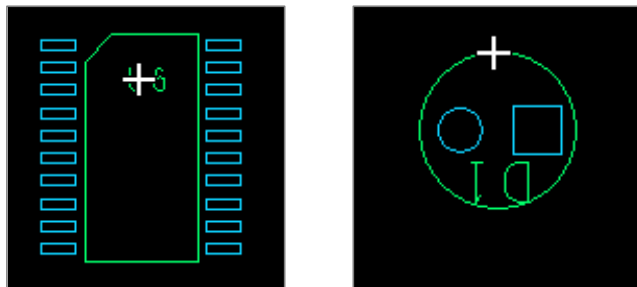
The Find window is more commonly referred to as the Find Filter. This is one of the more important forms used in the PCB Editor. It is critical that you pay attention to and understand the settings.

The top section of this form contains the Design Object Find Filter box. This section determines what types of objects in the design are to be acted upon when you select elements with the mouse.

If the pick occurs at a point where there are multiple items displayed, the system prioritizes the selection by going from the top left object in the column to the bottom right and finding the first checked item. In *both* instances of the examples shown, the entire part would be deleted. This is because the Symbols item would be the first check box found in the Find Filter.

Using the All On and All Off buttons is a quick way to turn on or off all the items in the Design Object Find Filter box. Some of the boxes will be greyed out, depending on the active command.

If you drag with the LMB and create a rectangle, all elements that match any item checked in the Find Filter will be selected.

Using the Find by Name Section

Use the Find by Name section of the Find Folder tab to:

- ◆ Highlight a net name.

- ◆ Locate a part placed in the design by its ref. des.

1. Use the pull-down field to set which type of a name you will enter.

2. Enter the name in the blank (**>**) field and use the tab key.

3. Use the **More** button to display a scrollable list of all elements matching the desired "type."

NOTE: The check boxes selected in the Design Object Find Filter have no affect when using the Find by Name section, with the exception of the **Property** pull-down field.
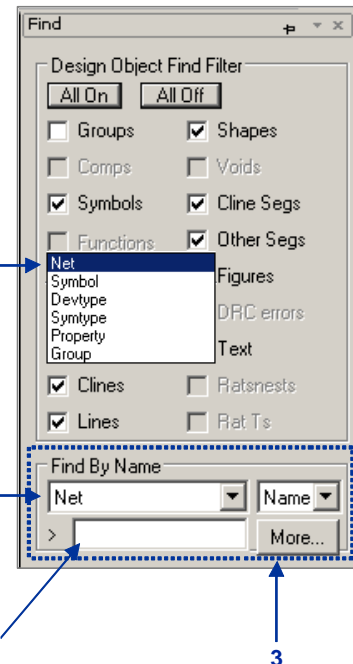
The bottom section of the Find Filter contains the Find by Name box. You use the Find by Name section to select elements by a name rather than graphically.

For example, if you wanted to highlight the net called GND, you would execute the *Display - Highlight* command, go down to the Find by Name section, click the down arrow and select Net. Then in the blank field immediately below the Net pull-down field, enter **GND**, and press **TAB**. The net named GND would then be highlighted.

The **More...** button in the lower right corner of the Find by Name section opens a scrolling window that lets you choose from a list of all available net names, component names, properties, and so forth. It should be noted that when you use the Find by Name section, the check boxes in the Design Object Find Filter section are ignored, unless the Property pull-down option is used.

Using Find by Property
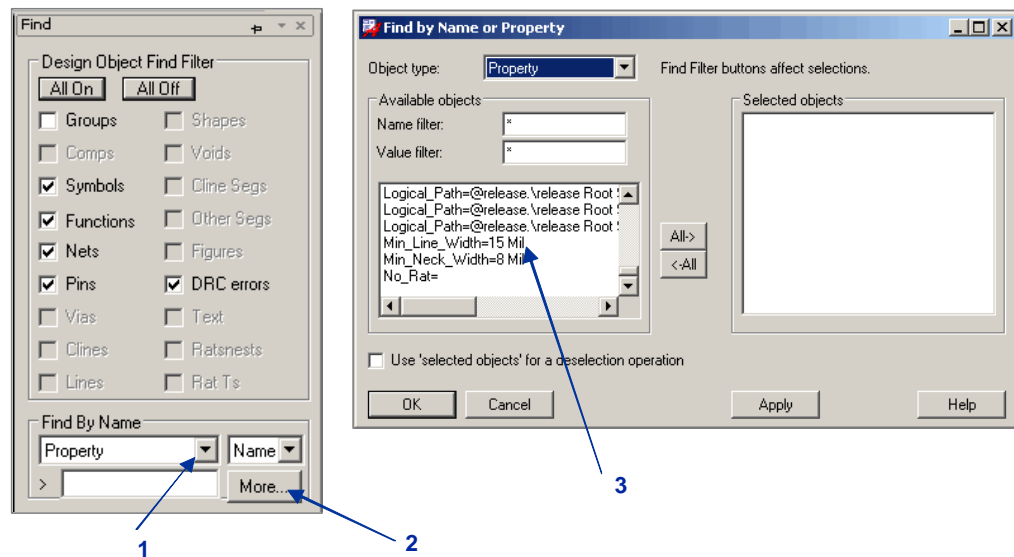
Use the Find by Property option to select database elements with a common property such as FIXED, MIN_LINE_WIDTH, and so forth.

As previously noted, the Property option under the Find by Name box uses the Design Object Find Filter section. When you select the Property option and click the More button, all properties are gathered that are attached to the checked items. A scroll list is generated specifying all the unique properties that were found.

Application Modes
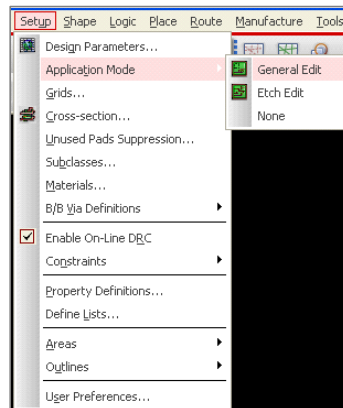
◆ Application Modes available:

- ❑ General Edit Mode
- ❑ Etch Edit Mode

◆ Switch modes using the command:

An application mode provides an intuitive environment in which commands used frequently in a particular task domain, such as etch editing, are readily accessible from RMB pop-up menus, based on a selection set of design elements you have chosen.

This customized environment maximizes productivity when you use multiple commands on the same design elements or those in close proximity in the design. Application mode configures your tool for a specific task by populating the RMB pop-up menu only with commands that operate on the current selection set.

The different application modes available are:

- ⁿ **General Edit Mode** - This is the default mode when the tool is first launched. It allows you to perform editing tasks, including place and route, as well as moving, copying, or mirroring items.

- ⁿ **Etch Edit Mode** - This application mode customizes your environment to perform etch-editing tasks. Examples are adding and sliding connections, delay tuning, and smoothing clines or cline segments.

- ⁿ **Placement Edit Mode** - This application mode is a tuned editing environment for component placement. Not available in the OrCAD PCB Editor.

Pre-selection Mode

1. Hover mouse over item.
2. Tab through hierarchical elements.
3. Right Mouse Button to perform a command.
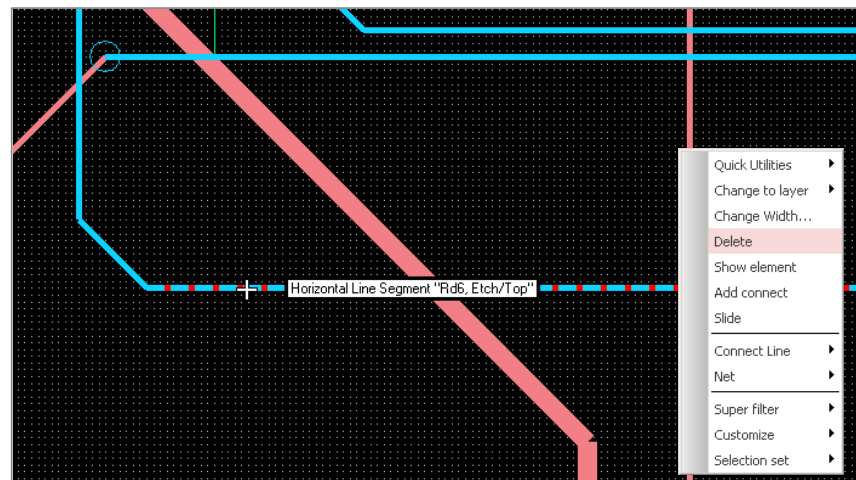
The OrCAD PCB Editor defaults to a pre-selection use model, which lets you choose a design element (noun), and then a command (verb) from the RMB pop-up menu. This pre-selection use model lets you easily access commands based on the design elements you've chosen in the design canvas, which the tool highlights and uses as a selection set, thereby eliminating extraneous mouse clicks and allowing you to remain focused on the design canvas.

While base elements such as cline segs, pins, and vias cannot be parents of other elements, they are the building blocks of which hierarchical elements such as nets, clines, and components are made. A pin is a child of a net, as well as that of a symbol and a function. Similarly, a cline could be a child of a symbol and a net. For a symbol with a shape containing a void, for example, the hierarchy may span five levels. The segment comprising the void has a hierarchy of Other Seg – Void – Shape – Symbol – Component.

If you enable more than one base or hierarchical element in the Find Filter, the base element determines the hierarchical elements you may choose. You navigate through the hierarchy by using the following or any other predefined hot key:

n Tab or Shift-Tab for all hierarchical elements

Note: The Tab key is unavailable when you select by window, which chooses only top-level hierarchical elements.

n Find Filter to disable unwanted elements

Creating a Selection Set

Mouse Action	Result
LMB pick (single select)	Clears previous selection set and adds highlighted element at the mouse location to the selection set.
Shift + LMB pick (extend select)	Adds highlighted element at the mouse location to the selection set.
Ctrl + LMB pick (toggle select)	Adds the highlighted element at the mouse location if not already in the selection set. Removes the highlighted element from the selection set if the selection set already contains it.
Selection by window	Clears previous selection set. Adds elements enabled in the Find Filter and that overlap the window to the selection set.
Shift + Select by window	Adds elements enabled in the Find Filter and that overlap the window to the selection set.
Ctrl + Select by window	Removes elements enabled in the Find Filter, overlapping the window or already in the selection set.

In application mode, the tool highlights design elements you have chosen in the design window as a selection set. Commands applicable to an application mode operate on this selection set. You modify the elements in the selection set by using any of the mouse operations described above.

Using the Super Filter
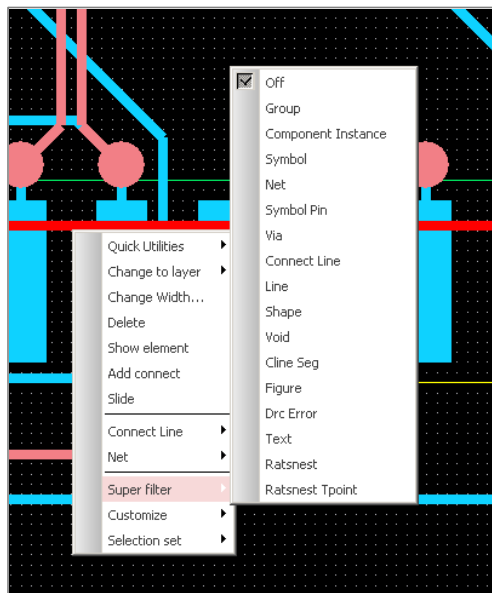
The Super filter lets you choose a particular element type to refine your selection set and temporarily disable all other elements from the RMB pop-up menu rather than the Find Filter. You can only choose one database element type using the Super filter.

For example, let's suppose you want to move many parts in your design. Without using the Super Filter, you would need to hover your mouse over a part, use the tab key to select the symbol, then move the part. By using the Super Filter and turning on only symbols, as soon as you hover your mouse over a part, the symbol is immediately selected and ready to move.

Note

The Super filter only applies when you are working in the pre-selection mode.

Context-Sensitive RMB Pop-Up Menu

The commands that populate the context-sensitive, RMB pop-up menu depend on:

◆ Current application mode

◆ Design elements already in the selection set

◆ Design elements selectable at the current mouse position

Hovering your cursor over...	...populates the pop-up menu with
an element already in the selection set	commands applicable to the selection set
an area where nothing is selectable, such as black space in the design	commands that don't use design elements as input such as the Design Parameter, Change Active Layer, Customization, Superfilter, and Options.
the cursor is not over an already selected element and the element underneath the cursor could potentially be selected	commands applicable to that element

Application-mode commands are accessible from a RMB pop-up menu based on the current selection set. The commands that populate the context-sensitive, RMB pop-up menu depend on the location of your cursor and whether you have already created a selection set.

Remember, you can further filter all elements chosen during the current editing session by right-clicking and choosing *Selection Set - Narrow Select* from the pop-up menu.

The RMB pop-up menu features several choices that appear whenever you work in an application mode:

n **Quick Utilities** allows access to frequently used functions.

n **Super filter** confines your work to a particular element type, such as all nets.

n **Customize**

– Enable Single Click Execution

– Disable Automatic Drag Operations

– Enable Shape Selection through Shape Fill

- – Enable Alternate Popup Model

- – Reset to Defaults

n **Selection Set**

- – **Clear All Selections** empties the selection set.

- – **Object Browser** lets you search for elements by name or by property.

- – **Select** appears only if selectable elements are available at the current mouse position.

- – **Narrow Select** lets you refine your selection when multiple elements have been chosen during an editing session.

- – **Toggle Select** lets you further refine the elements in the selection set after you select by window. Clicking an element with a minus sign next to it removes it from the selection set; clicking an element with a plus sign adds it to the selection set.

To work on a single element, hover your cursor over that element and then choose *Selection Set - Select - <element>* from the pop-up menu, which also clears all previous selections. You can also select the element with the LMB to select the element and clear the selection set.

If the selection set contains a mix of elements, the RMB pop-up menu displays sub-popup menus of commands applicable to those elements.

Etch Edit Default Command Execution

Element Type	Drag	Shift Drag	Ctrl Drag	Double Click
Symbol	Move	Spin	Copy	Move
Pin				Add connect
Via	Slide	Move	Copy	Add connect
Cline	Move	Move	Copy	
Shape	Move	Move	copy	
Cline Seg	Slide		Delay tune	Slide
Rat				Add connect
Rat Tee	Slide	Move		

In the pre-selection use model, you can automatically execute a default command with a click, drag, shift-drag or Ctrl-drag on an element. In the Etch Edit application mode, the default commands are as documented above.

You can set an option so that the double click column commands can be executed using a single click. In order to accomplish this, in the pre-select mode, select RMB in an open area and select *Customize - Enable Single Click Selection*.

Note

When you execute a command by dragging in any application mode, use the **Esc** key to allow the LMB to be released, yet continue dragging.

General Edit Default Command Execution

Element Type	Drag	Shift Drag	Ctrl Drag
Group	Move	Move	Copy
Symbol	Move	Spin	Copy
Via	Slide	Move	Copy
Cline	Slide	Move	Copy
Line	Move	Move	Copy
Shape	Move	Move	Copy
Cline Segment	Slide		
Figure	Move	Move	Copy
Text	Move	Move	Copy
Rat Tee	Slide	Move	

In the pre-selection use model, you can automatically execute a default command with a click, drag, shift-drag or Ctrl-drag on an element. In the General Edit application mode, the default commands are as documented above.

Note

When you execute a command by dragging in any application mode, use the **Esc** key to allow the LMB to be released, yet continue dragging.
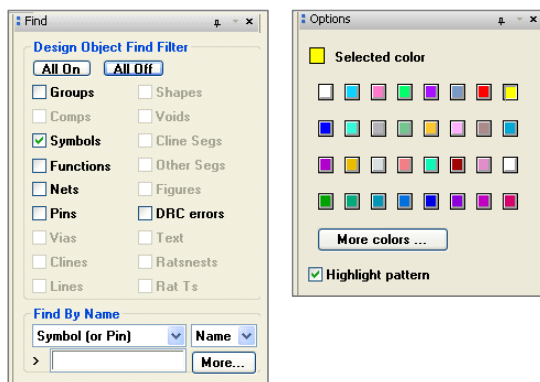
Coloring Elements

Display – Assign Color

…or

Common sequence for assigning colors to elements with the mouse is:

1 Start the command.

2 Activate the Find Folder tab.

3 Click the **All Off** button.

4 Toggle on desired elements.

5 Select desired color.

6 Select the elements in the design window to highlight.

The *Display - Assign Color* command is used to display a database element in a certain color. The type of database element highlighted is based upon the Find Filter.

You initially have a choice of 32 different colors to choose from in the Options window. Clicking on the More colors button opens a palette that contains the complete 192 colors defined in the **Color/Visibility** command.

In order to color different objects in different colors, change the Selected color in the Options window before selecting the next object. Once colored, the elements remain colored until they are dehighlighted using the *Display - Dehighlight* command.

Highlight Elements

The *Display - Highlight* command is used to highlight elements, which display in an alternating checkerboard of the element's color and the temporary highlight color defined in the Display category of the Color Dialog form. Once highlighted, the elements remain highlighted until they are dehighlighted using the *Display - Dehighlight* command.
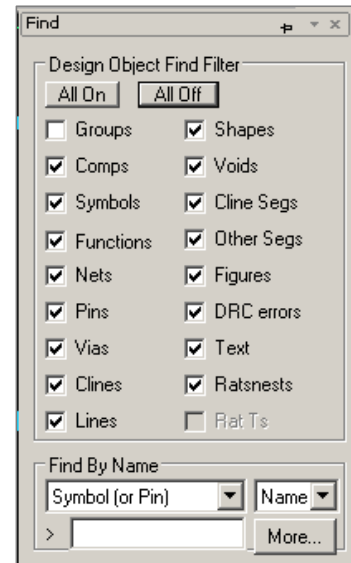
Using the Show Element Command

Display - Element

…or

Common sequence for using Show
Element with the mouse is:

1   Start the command.

2   Activate the Find folder tab.

3   Click the **All Off** button.

4   Toggle on desired elements.

5   Select the elements in the
design window.

You can use the **Show Element** command, also referred to as the **Display Element**
command, to ascertain information about an item in the design. Remember that the Find
Filter is used to determine what type of information will be displayed. Based upon the
Find Filter settings, you can determine a net name, a component's reference designator,
which padstack a pin uses, and so forth.

If you highlight an X Y coordinate in the Show Element form, the object will highlight
and be centered in your display window.

Using the Display Measure Command

Display - Measure   …or

```
Measure
File  Close  Help

        SYMBOL PIN U2.1 @ (575.0, 2075.0)
        SYMBOL PIN U2.2 @ (575.0, 1975.0)

Dist = 100.0000.  Total Dist = 100.0000.
Manhattan Dist = 100.0000.  Dx = 0.0000.  Dy = 100.0
Air Gap = 38.0000.  On Subclass:  TOP
```

☐ Nets   ☐ Figures
☑ Pins   ☐ DRC errors
☐ Vias   ☐ Text
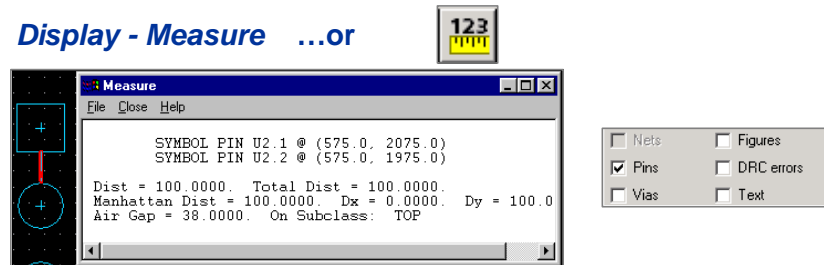
Verify the settings in both the Find folder tab and the Options folder tab

Bottom Pin

```
Measure
File  Close  Help

        SYMBOL PIN U2.1 @ (575.0, 2075.0)
        SYMBOL PIN C4.2 @ (635.0, 2200.0)

Dist = 138.6542.  Total Dist = 138.6542.
Manhattan Dist = 185.0000.  Dx = 60.0000.  Dy = 125.0000.
No air gap -- both elements are not defined on  TOP.
Common subclasses for air gap are
BOTTOM
```

☐ Nets   ☐ Figures
☑ Pins   ☐ DRC errors
☐ Vias   ☐ Text
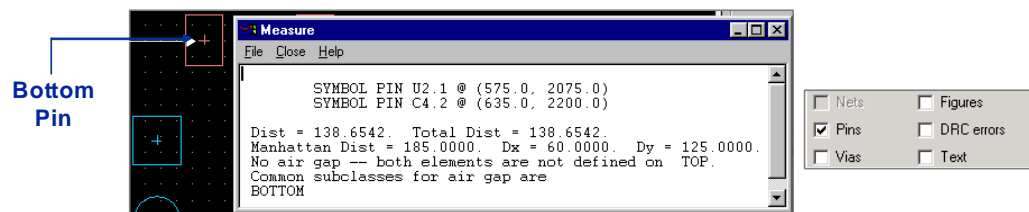
If the class and subclass settings are incorrect, the **Display Measure** command may not return the desired results.

———————

You use the **_Display - Measure_** command to determine the distance between two points. After the two points have been selected, a window is displayed detailing information about the distance between the two elements. Information displayed includes total distance, manhattan distance, the delta X and delta Y, and the air gap. The air gap will only be displayed if the two selected elements reside on the same class and subclass, and if that class and subclass are active in the Options tab. Again, it is important to remember that the Find Filter settings determine which database elements will be selected by this command. If the selection point contains no items that match the Find Filter settings, then the closest grid point will be used for determining the distance.

Exporting Design Parameters

The *File - Export - Parameters* command creates a database parameter file (`.prm`) containing customized parameter records from a design. Customized parameters are those that have a single instance in the database and include global values such as dynamic fill; grid settings; artwork format; and Xhatch style, line width, spacing, and angle.
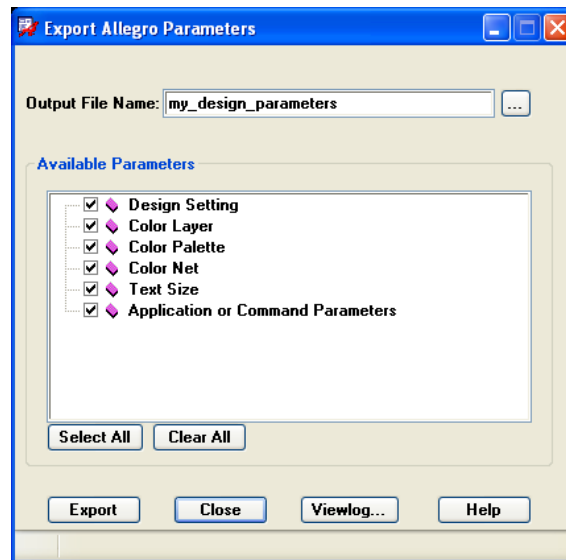
The Export Parameters are:

- n Output file name - Enter the name of a file to which to save database parameter records, or click ... to locate an existing database parameter file in the current working directory.

- n Available Parameters - Displays a list of the possible parameter records available for export. If no parameter record exists in the database, it does not display. Click to include the parameter group in the .prm file.

  - Design Setting - Global values and grid settings

  - Artwork - Artwork film definitions

  - Color Layer - Color parameters and color table

  - Color Net - Net custom color and states. When a file containing net color data is imported into any design, only the nets that exist in that design are read; the rest are ignored. Net color assignments are not overwritten, but rather incremented. To completely replace net color assignments, click Clear All Nets in the Nets section of the Color dialog box before importing a file containing net color data.

  - Text Size - Text size settings

 – Application or Command Parameters - All other supported parameters, including those for auto rename, auto assignment, auto silkscreen, global dynamic fill, autovoid, export logic, drafting, gloss line fattening, gloss dielectric generation, Options window tab settings, test prep, automatic placement, auto swap, thieving, backdrill, interactive flow planner (Allegro only), and Signoise analysis.

n Select All - Choose to include all available parameters in the .prm file.

n Clear All - Choose to remove all available parameters from the .prm file

n Export - Click to create a .prm file based on the chosen parameters.

n Close - Closes the dialog box and exits the command.

n Viewlog - Click to view error messages and other process information in the current or last generated param_write.log report.

The *File - Import - Parameters* command imports the values from .prm files centrally located in a corporate library, or in your local working directory into another design for reuse.

Labs

◆ Lab: Highlighting and Using the Find Filter
  ❑ Using the Find By Name section
  ❑ Using the Selectable Objects list
  ❑ Finding items by property
  ❑ Highlighting objects in a design

◆ Lab: Assigning Colors and Using the Find Filter
  ❑ Assign a color to the GND net in a design

◆ Lab: Using the Find Filter with the Show Element Command
  ❑ Using the Find Filter with the *Display - Element* Command
  ❑ Using the Find Filter with the *Display - Measure* Command

◆ Lab: Exporting a Design Parameters File
  ❑ Export the design parameters

The following labs will teach you how to select elements in the PCB Editor database by graphically selecting items, selecting items by their names, and selecting items by their properties. You will also learn how to use the **Assign Color, Highlight,** and **DeHighlight** commands.

The next lab will teach you how to use the *Display - Element* and the *Display - Measure* commands.

Lab 2-2:   Highlighting and Using the Find Filter

Objective: Use the Find Filter as a selection aid to highlight items.

Locating a Component Using the Find By Name Section

In this lab you will learn that the Find By Name section contains a data entry field and two field description boxes. Here is how to use these options.

1. Open OrCAD PCB Designer using the `cds_routed.brd` file in the *play* directory, if you do not already have it running.

2. Perform a *View - Zoom Fit* command to show the entire board.

3. Choose *Edit - Move* from the top menu.

4. Hover your mouse over the Find tab to display the window.

5. If needed, change the setting in the Find By Name field to **Symbol (or Pin)** as shown, and enter **U3** in the > field.
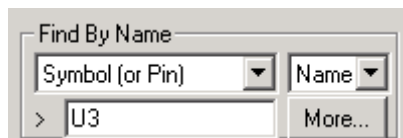
6. Press the **Tab** key.

   Part U3 snaps to your cursor and the display is redrawn to be zoomed around this part. Whatever you enter in the Find By Name field is selected for manipulation by the active command—in this case, **Move**.

7. Right-click and choose **Cancel** from the pop-up menu.

   Part U3 snaps back to its original location as you exit the active command.
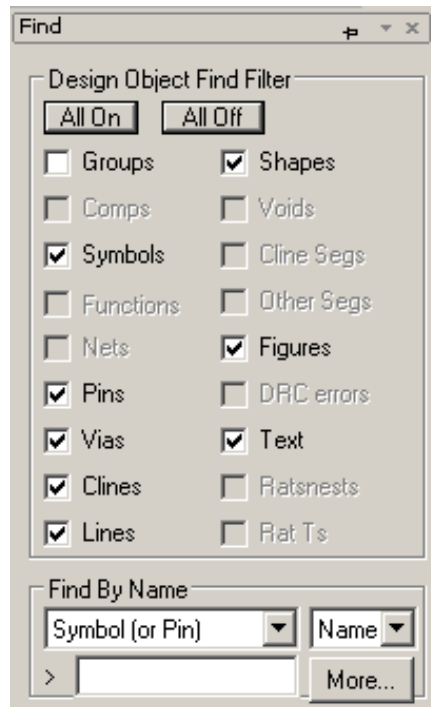
Using the Selectable Objects List

You will use the Selectable Objects list to select a particular type of object with the LMB. You can toggle different objects either ON or OFF to prevent inadvertently selecting something you don't want to edit.

1. Select *View - Zoom by Points* and zoom around U3 if it hasn't already been done.

2. Select *Edit - Move* from the top menu bar.

3. Hover your mouse over the Find tab to display the window.

4. In the Find Filter, click **All On**.

This ensures that check boxes of all appropriate objects are toggled on, as shown in the figure.

5. Click on the reference designator (text characters) **U3**.

Part U3 snaps to your cursor. In the selectable objects section of the Find window, Symbols is checked, or toggled ON. The reference designator you selected is seen as *part of* the package symbol. Because Symbols is higher in the selection hierarchy than the reference designator Text, the PCB Editor selects the item at the higher level.

6. With the cursor in the work area, right-click.

A pop-up menu appears with options for the active **Move** command.

7. Select **Oops** from the pop-up menu.

Part U3 snaps back to its original location.

8. In the Find window, click **All Off**, then enable the **Text** check box.

All items in the Find window should be unchecked except for Text.

9. Select the reference designator text for **U3** again.

This time, part U3 does not snap to the cursor. Instead, only the reference designator text snaps to the cursor.

Because of the change you made in selectable objects, the reference designator you selected is treated as a text object and the symbol is not selected.

10. Right-click and choose **Cancel** from the pop-up menu.

Text U3 snaps back to its original location.

Using the Pre-Selection Mode

As an alternative to executing the **Move** command first, you can use the pre-selection mode capability to move the text U3.

1. Hover your mouse over a pin on U3. A pop-up window should appear identifying the type of element that is currently selected. This informs you that if you use the RMB to perform a command, the selected element will be affected.

2. Press the **Tab** key several times to select different database element types.

3. Hover your mouse over the Find tab to display the window. In the Design Object Find Filter section, select the **All Off** button and then toggle on **Symbols** and **Pins**.

4. Hover your mouse over a pin on U3. A pop-up window should appear identifying the pin currently selected. Press the **Tab** key again and the popup should switch to **Symbol "U3"**. Press the **Tab** key again and it should switch back to the connect pin.

   Since you only have Symbols and Pins enabled in the Find Filter, these are the only two types of database elements that can be selected.

5. Hover your mouse over the Find tab to display the window and select **All On**.

6. Hover you mouse over the text U3. Press the **Tab** if necessary so that the popup states Text "U3". Select with the RMB and select **Move** from the popup menu. Since the text is the currently selected item, the Move command works on the text string only.

7. Right-click and choose **Cancel** from the pop-up menu.

   Text U3 snaps back to its original location.

8. Right-click and choose *Selection Set - Clear all selections*.

   Even though you cancelled the Move command for U3, it is still selected. You should always make sure that you have nothing selected (unless wanted) in the pre-selection mode as certain commands will automatically act upon pre-selected items.

Finding Items by Property

You can find objects by specifying the properties attached to them. To do so, you use the Property field under the Find By Name box in conjunction with the **Highlight** command.

When you click the More button, the Properties dialog box displays a list of properties to help you select the object you want to edit or act upon. This list of properties is affected by the button settings in the selectable objects section. To get a complete listing of available properties, you must make sure all the buttons in the selectable objects section are toggled ON.

1. Choose *Display - Highlight* from the top menu bar.

2. Hover your mouse over the Find tab to display the window.

3. Click **All On**.

   This ensures that all relevant check boxes are toggled ON, limited to Symbols, Functions, Nets, Pins, and DRC errors.

4. Under the Find by Name field, select **Property** from the drop-down list if it is not already selected.
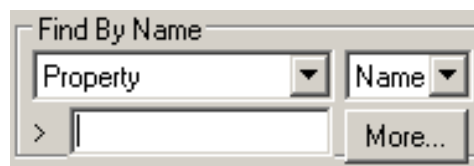
5. Click **More...** to display a browser menu of properties that exist in your design.

The Find By Name/Property dialog box appears, as shown in the figure, containing a scrollable list of available properties.
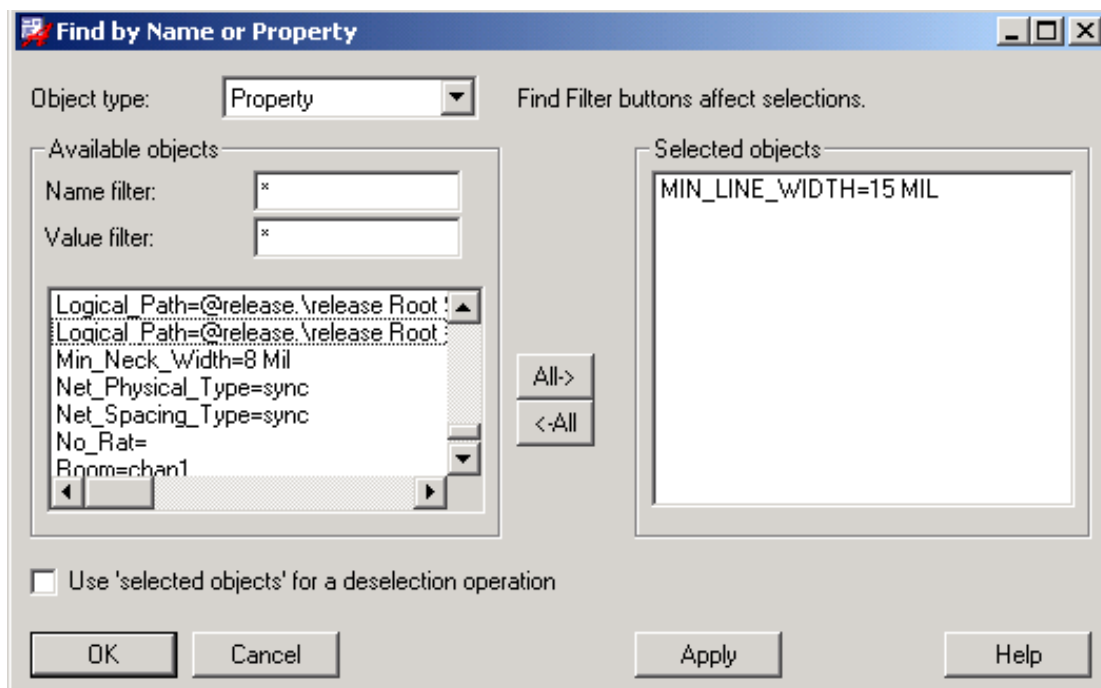
6. Scroll down and select the **MIN_LINE_WIDTH=15** property, then click **Apply**.

You have just highlighted your special voltage nets. All nets with an assigned MIN_LINE_WIDTH property of 15 are highlighted in the work area. The nets V12N, GND_EARTH, AGND and V+12 in this design have a MIN_LINE_WIDTH property attached to them.

7. Click **Cancel** to close the Find By Name/Property dialog box.

8. With the cursor in the work area, pan around to see the highlighted nets. You will also see the highlighted nets in the World View window.

9. Right-click and choose **Cancel** from the pop-up menu.

The **Highlight** command is no longer active.

Highlighting Objects in a Design

In this part of the lab, you will use the **Highlight** command to locate objects you want to highlight.

You can highlight an object whose location is unknown (so you can see where it is placed or how it has been routed). Highlighting is particularly useful on very large, densely populated designs.

1. Zoom in to the area around the **U3** part, located at the left side of the design near the center.

2. Choose *Display - Highlight* from the top menu.

3. Hover your mouse over the **Find** tab to display the window.

4. Change the setting in the Find by Name field to **Symbol (or Pin)** as shown, and type **U3** in the > field.

5. Press the **Tab** key.

Find By Name

Symbol (or Pin) ▼   Name ▼

> U3   More...

U3 becomes highlighted with the temporary highlight color set in the Color Dialog form. You can also see the highlighted part in the World View window if it is currently visible.

6. Right-click in the PCB Editor work area and choose **Done** from the pop-up menu.

7. Click the **Dehighlight** icon

or select *Display - Dehighlight* from the top menu.

8. In the Options window, click **Symbols** to remove all the permanent symbol highlights in your design.

9. Right-click in the PCB Editor work area and choose **Done** from the pop-up menu.

You can use highlights for objects other than components. Use them for critical nets, pins, properties or anything the Find Filter is capable of finding.

STOP **End of Lab**

Lab 2-3:   Assigning Colors and Using the Find Filter

Objective: Use the Find Filter as a selection aid to color items.

In this lab, you will use the **Assign Color** command to set the colors of the GND net to display its location and differentiate tit from the other nets in the design. This net receive a NO_RATS property during the NETREV netlisting process.

Assigning Colors to a Net in a Design

1. While zoomed in the area around the **U3** part, located at the left side of the design near the center.

2. Click the **Assign Color** icon or choose *Display - Assign Color* from the top menu.

3. Hover your mouse over the **Options** tab to display the window.

The Options window changes to display available colors, and the current active selected color is displayed.

4. Click on the purple color button to designate purple as the selected color.

5. Hover your mouse over the **Find** tab to display the window.

6. Change the setting in the Find by Name field to **Net** as shown, and type **GND** in the **>** field.

7. Press the **Tab** key.

The GND net becomes a the purple color. You can also see the colored pins in the World View window if it is currently visible.

8. Right-click in the PCB Editor work area and choose **Done** from the pop-up menu.

9. Click the **Dehighlight** icon

or select *Display - Dehighlight* from the top menu.

10. In the Options window, click **Nets** to remove all of the net colors in your design.

11. Right-click in the PCB Editor work area and choose **Done** from the pop-up menu.

STOP **End of Lab**

Lab 2-4: Using the Find Filter with the Show Element Command

Objective: Query information about objects in a design.

The **Show Element** command displays helpful information about selected objects. You can use this command to evaluate net names, reference designators and pin numbers, line widths, wire lengths, package types, padstack names, measured distances, assigned properties, DRC errors, and more.

Remember, the Find Filter controls what is selected, and therefore the data that is reported to you.

Using the Show Element Command

1. Zoom in to a view area around the **U2** component, which is a long DIP component located just left of the board center, and to the right of the three SOICs at the left side of the design.

2. Click the **Show Element** icon.

Note

The **Show Element** command can also be accessed from the *Display - Element* menu or by pressing the **F4** key.

3. In the Find window, select **All On**.

   This ensures that the check boxes for all objects are toggled ON.

4. Select one of the pins on the U2 component that contains etch connected to the pin.

   The Show Element report appears.

5. If your Show Element report window is covering the Find Filter, move it so you can also see the Find Filter and the U2 component.

   At the top of the Show Element form is a description of the type of object that is selected, <COMPONENT INSTANCE>. The data in this report corresponds to a description of the component instance of the Comps items in the Find Filter because the Comps category is higher in the selection hierarchy than pins or etch.

6. In the Find Filter, disable the check box next to **Comps**.

7. Select the same pin on the same component again.

This time the Show Element form refreshes to display SYMBOL information for this component package.

This report focuses on the characteristics of the physical package symbol, and corresponds to the Symbols entry in the Find Filter. Symbols is now the priority item in the Find Filter. If more than one item in the Find Filter is turned ON, then the priority goes to the highest active item in the list.

8. In the Find Filter, disable **Symbols** and select the same pin again.

The Show Element form refreshes to display FUNCTION INSTANCE information for this package. This information corresponds to the Functions entry in the Find Filter. (The pin you selected is seen as part of a function or gate within this package.)

9. In the Find Filter, disable **Functions** and select the same pin again.

The Show Element form refreshes to display NET information for this pin. This information corresponds to the Nets entry in the Find Filter.

Notice the information about etch length and any attached properties.

10. In the Find Filter, disable **Nets** and select the same pin again.

The Show Element form refreshes to display CONNECT PIN information. This information corresponds to the Pins entry in the Find Filter.

Notice the padstack information.

11. In the Find Filter, disable **Pins** and select the same pin again.

The Show Element form refreshes to display CONNECT LINE information for the connection to the pin. This information corresponds to the Clines (etch) entry in the Find Filter.

12. In the Show Element form, highlight an **(X Y)** coordinate (the numbers in parentheses) by dragging the cursor over the coordinate.

The Editor window will center the zoomed area around this coordinate. These forms are contact-sensitive. Try this a couple more times.

13. Right-click in the work area window and choose **Cancel** from the pop-up menu.

Selecting the same object generates different information, depending upon the settings in the Find Filter. It is not just *which* item you select, but also the *selection priority* in the Find Filter that matters.

When choosing the ***Display - Element*** menu item, disable all the objects in the Find Filter. Then enable only the object(s) that will generate the information you want to see.

Using the *Display - Measure* Command

**1.** In the Options window, set the Active Class to **ETCH** and the Subclass to **TOP**, as shown.
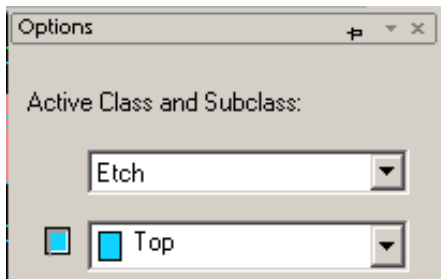
2. Choose *Display - Measure* from the top menu bar.

The PCB Editor message area prompts you to:

```
Make two picks for the distance calculator.
```

3. Select two objects that you wish to measure the distance between. Remember to check the settings in the Find Filter.

The Measure report appears, showing information about the objects (if any) selected, the manhattan distance, and air gap information. An example of the measure output is shown below. Yours will probably not match this display exactly.
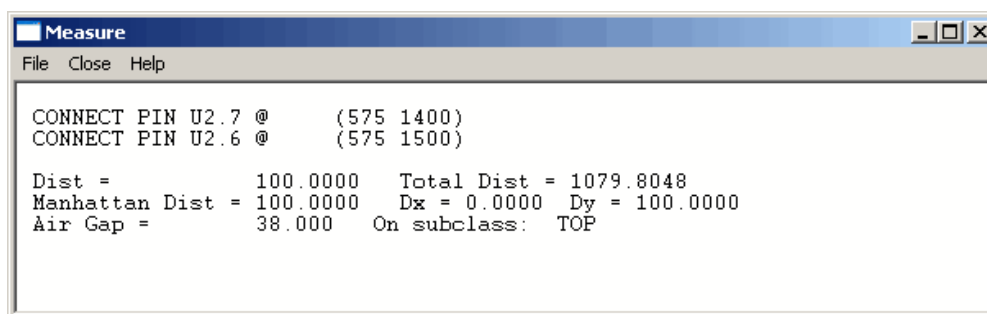
4. To exit from the *Display - Measure* command, right-click and choose **Done** from the pop-up menu.

5. Practice some more with this command if you have the time.

6. Choose the *File - Exit* menu item.

An Exit window appears, asking if you want to save any of the changes made to your current design.

7. Click **No** to log out of the PCB Editor tool without saving changes.

The PCB Editor window closes.

STOP **End of Lab**

Lab 2-5:   Exporting a Design Parameter File

Objective: Export the design parameter information for reuse in a future design.

In this lab you will export the parameters of a design to allow for importing of this parameter information into future designs. Importing the design parameters eliminates the need to manually set up the various parameters for each new design.

Exporting the Design Parameters

1. Open the OrCAD PCB Designer using the `cds_routed.brd` file in the *play* directory, if you do not already have it running.

2. Select *File - Export - Parameters* from the top menu.

3. In the **Ouput File name** field, enter `my_design_parameters`.

4. Click the *Select All* button.

5. Click the *Export* button.

6. Click the *Close* button.

7. Open the *my_design_parameters.prm* file with WordPad and view the parameters file.

8. Close the file when you are finished viewing it.

STOP **End of Lab**