PADS Layout Tutorial

Contractor/manufacturer is:
Mentor Graphics Corporation
8005 S.W. Boeckman Road, Wilsonville, Oregon 97070-7777.
Telephone: 503.685.7000
Toll-Free Telephone: 800.592.2210
Website: www.mentor.com
SupportNet: supportnet.mentor.com/
Send Feedback on Documentation: supportnet.mentor.com/user/feedback_form.cfm

Table of Contents

Learning the User Interface

The PADS Layout user interface is designed for ease of use and efficiency. PADS Layout is designed to meet the needs of the power user, while keeping the beginner in mind. PADS Layout's interface and interaction are similar to other Windows ™ applications. You can interact with PADS Layout using the keyboard, menus, toolbars, and shortcut menus.

In this lesson:

- Shortcuts
- Using the workspace
- Setting units of measure
- Setting grids
- Panning and zooming
- Selecting objects

Preparation

If it is not already running, start PADS Layout.

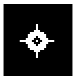
Shortcuts

You can use keyboard shortcuts to start commonly used commands or to set options. You will use some of the shortcuts throughout the tutorial.

Keyboard shortcut types:

Shortcut type	Description
Modeless Commands	Commands invoked through the keyboard. The available commands handle display options, design settings, and mouse click substitutions.
Shortcut Keys	Commands that change settings or execute commands without using the mouse. Windows standard shortcut keys, such as Alt+F to open the File Menu, are also used.

Using the workspace



The default workspace measures 56 inches by 56 inches. The origin of the workspace, or 0,0 coordinate location, is represented by a large white marker. When you start PADS Layout or open a new file, the origin is centered in the window at a medium magnification.

Move the origin

To relocate the design origin:

1. **Setup** menu > **Set Origin**.
2. Click in the workspace to indicate a new position for the origin.

Pointer position display

As you move the pointer around the workspace, its position, in absolute X,Y coordinates relative to the origin, appears on the Status Bar in the lower right corner of the screen. Throughout tutorials you will be required to refer to this coordinate data.

1. Place the pointer over the origin and note the 0,0 reading on the Status Bar.
2. Move the pointer around the workspace and note how the X,Y coordinates change as the pointer position changes.

Setting units of measure

To change the unit of measure to inches, mils (default setting), or metric units, click **Options** on the Tools menu. The Design Units settings are on the Global > General page of the Options dialog box. Leave the current setting of Mils.

Setting grids

There are two kinds of grids: working grids and display grids.

Working grids

Working grid types:

Grid	Description
Design Grid	Establishes the minimum snap distance for the pointer when editing.
Via Grid	Establishes the spacing and location of vias.

Display grid

This dot-type grid is a visual aid. You can set the display grid to match your design grid, or you can set it at larger or smaller multiples of the design grid.

1. **Tools** menu > **Options** > **Grids and Snap** > **Grids** page
2. To disable the display grid, set both coordinates of the Display Grid to a small value, such as 10. It is not really disabled, but you will not see it unless you zoom in significantly.

Grid exercise

The spacing for each grid can also be set by using modeless commands. In the following exercise, you'll use modeless commands to set the design and display grids.

1. If the Grid tab of the Option dialog box is open, click in the workspace to make it the active window. Do not close the dialog box.
2. Type **gd**, for display grid. The modeless command dialog box opens with the characters gd in the box.
3. Type **500** and press **Enter**. Note that the settings in the display grid area update in the dialog box and a corresponding message appears in the Status Bar, located in the lower left corner.
4. Close the Options dialog box.

   Requirement: You may have to zoom in or out to see the display grid; the ability to view the display grid depends on the grid value entered. See the "Using Pan and Zoom" topic.

   Alternative: You can also set the design grid in one step by typing the modeless command **gr500** and pressing **Enter**. It's best to use a space between the modeless command and the value. It's required in PADS Router.

Tips:

- Use the g modeless command for global grid settings. Entering g500 sets the design grid and via grid to 500 mils.
- To set the via grid separately, type gv and press Enter.
- Instead of specifying a single variable, you can type one of the grid modeless commands and use two variables where *xy* equals a separate rise and run of the grid spacing (gv 50 10).

Panning and zooming

Open a previously saved file

To make it easier to view changes in magnification, open the file named **preview.pcb** in the \PADS Projects\Samples folder.

Tip: Because any operation is considered part of new file creation, including the shortcut menu and grid exercises you just performed, a dialog box appears prompting you to save the old file. Click **No**.

Zoom

Several methods exist for controlling the centering and magnification of a design. In this exercise, you are going to use the mouse.

For two-button mouse operations, the Zoom button enables and disables Zoom mode. Clicking Zoom from the View menu can also enable it. While in Zoom mode, the pointer changes to a magnifying glass. For three-button mouse operations, Zoom mode is always available using the middle mouse button.

Tip: Because some manufacturers give the center mouse button a specific function, you may need to de-activate default middle mouse button functionality before you can use these features in PADS Layout. Change mouse functionality using the Control Panel.

Zoom in and out by placing the pointer at the center of the area and dragging in a specific direction. Pan and Zoom functions are also available using commands from the View menu, using the numeric keypad, and using the Windows scroll bars. See *PADS Layout Help* for more information on Pan and Zoom functions.

Practice zooming

The following procedure assumes you are using a two-button mouse except where noted.

1. Zoom button 

   Exception: If you are using a three-button mouse, skip this step.

2. Zoom in.

   a. Click and hold the left mouse button in the **center** of the object or area you want to magnify.

   Exception: If you are using a three-button mouse, click and hold the middle mouse button.

   b. Drag the pointer upward, moving the mouse away from you. A dynamic rectangle attaches to and moves with the pointer.

   c. When the rectangle encompasses the area you want to magnify, release the mouse button to complete the operation.

3. Zoom out. Repeat Step 2, but drag the pointer downward, moving the mouse towards you. A static rectangle, representing the current workspace view, appears with a dynamic rectangle representing the new view of the workspace.

4. Practice using Zoom mode to adjust the magnification.

   Tip: To re-establish the original view, click Board on the View menu.

5. If you are not using the middle mouse button, click the **Zoom** button again to end Zoom mode.

Practice panning

1. Point to the center of the new view.
2. Click the middle-mouse button.

Selecting objects

During the many phases of the PCB process, you may want to select only specific objects. For example, during component placement you may want to limit selections to components and during interactive routing you may want to limit selections to connections or routes.

To accommodate this working preference, there is a Selection Filter. With the Selection Filter, you can specify which design objects are selectable. Items turned off in the list can't be selected.

Selection Filter

Objects are organized into two categories: Design Items and Drafting Items. Layers can also be filtered.

1. With nothing selected, right-click and click Filter.

   Alternative: Press Ctrl+Alt+F to open the Selection Filter dialog box.
2. Click **Anything** to enable selection of all objects.
3. Click **Parts** to disable part selection.
4. Leave the Selection Filter open and click in the design window to make it active.
5. Place the pointer over a component outline and click to select it. You cannot select it.
6. Place the pointer over an object other than a component and select it. You can select other objects.
7. Deselect all objects by clicking in an open area.

Selection Filter shortcuts

Tip: You can quickly change Selection Filter settings using the filter presets on the right-click menu.

1. With nothing selected, and the Selection Filter dialog box open, right-click and click **Select Nets**.

   Result: The Selection Filter updates to allow selection of nets only.
2. Right-click and click **Select Anything**.

Result: The Selection Filter updates to allow selection of almost any object. Only Clusters, Unions, Pin Pair, Nets and Board Outline are still turned off.

3. Click **Close** to close the Selection Filter dialog box.

Selecting all objects of one type

You can use the Selection Filter or the selection shortcuts to select all items of one object type.

1. To bring the whole design in view, press **Ctrl+B**, the View > Board shortcut.
2. With nothing selected, right-click and click **Select Components**.
3. Right-click again and click **Select All** to select all the components in the design.
4. To deselect the selected components, press **Esc**.

Cycling selections

Selecting a particular object may take a few tries in a crowded area. To eliminate multiple selection attempts, you can cycle through all of the objects near a current selection.

To cycle selections:

1. Right-click and click **Select Anything**.
2. Type **su1.28** to search for pin 28 of component U1. The pointer positions over the center of pin 28 of U1.
3. Press **Spacebar** to select 28.
4. Press **Tab** on the keyboard repeatedly to cycle through all of the selectable objects in the vicinity of pin 28.
5. Stop cycling when you select the item you want.
6. Do not save a copy of the design.

You completed the user interface tutorial.

Creating Library Parts

You create the library elements that compose a part type in the PADS library using the Library Manager and the PCB Decal Editor.

In this lesson:

- Understanding the PADS part type and PCB decal
- Creating a PCB decal
- Adding keepouts to a PCB decal
- Assigning PCB decal attributes
- Adding attribute labels to a PCB decal
- Creating a new part type
- Creating PCB decals using the Decal wizard

Restriction

This tutorial requires the Drafting Editing, General Editing and Library Editor licensing options.

To determine whether you can proceed:

- On the Help menu, click **Installed Options**.

Preparation

If it is not already running, start PADS Layout.

Understanding the PADS part type and PCB decal

Part Type

Before you can add a part to a design, it must exist as a PADS part type in the library. A part type is composed of three elements:

- The schematic symbol, or Logic decal as it is called in the library
- A component footprint, or PCB decal as it is called in the library
- Electrical parameters, such as pin numbers and gate assignments

An example of a PADS Layout part type is a 7404.

Element	Description
Part type name	7404
Logic decal	INV (inverter)
PCB decal	DIP14 (14 pin dual in-line package)
Electrical parameters	Six logical gates (A through F) using 12 of the 14 pins with one power pin and one ground pin.

You can create a part type in PADS Logic or in PADS Layout; however, you can only create a Logic decal in PADS Logic, and a PCB decal in PADS Layout.

PCB decal editor

The PCB decal is the physical representation of the part, often called its *footprint*. The PCB decal contains a number of terminals representing component pins and a component outline. You create PCB decals in the PCB Decal Editor.

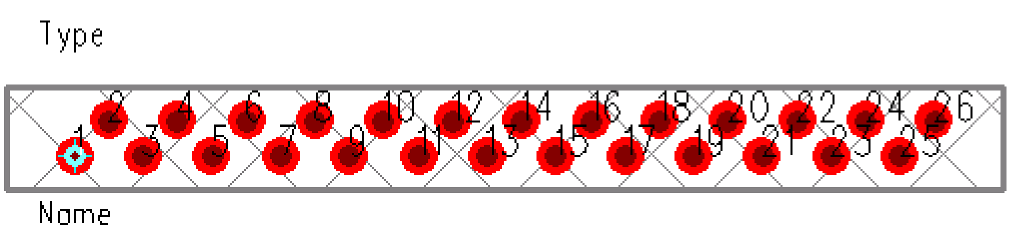
On the Tools menu, click PCB Decal Editor.

Once you enter the PCB Decal Editor, text labels and a PCB decal origin marker appear:

- The NAME text label is a placeholder for the reference designator of the part.
- The TYPE text label is a placeholder for the part type name of the part. Wherever you position these labels in the decal is where the reference designator or part type appears when a component using the PCB decal is added to the design.
- The origin marker identifies the origin of the part, which is used when moving, rotating, or otherwise positioning the part in PADS Layout. The decal origin is also referenced by automatic assembly and pick-and-place machines.

Creating a PCB Decal

In the following exercise you will create a PCB decal for the 26-pin connector, shown below, by adding new terminals, a component outline, and assigning pad stacks.



Add a terminal

The first step in defining a PCB decal is to add a terminal for each pin of the part. Each terminal is sequentially numbered as it's added to the decal. Each terminal number is its pin number, for example, terminal 1 is pin number 1, terminal 2 is pin number 2, and so on.

1. Drafting button > Terminal button .
   When the Add Terminals dialog box opens, accept the defaults and click OK. You are now in add terminal mode. Each click of the mouse button will add a new terminal at the location of the click.
2. Set the design grid to 50 by typing **g50** and pressing **Enter**.
3. Set the display grid to 50 by typing **gd50** and pressing **Enter**.
4. Move the pointer over the decal origin marker and click to add the first terminal at the decal origin.
5. Move the pointer to location **X50**,**Y50** (using the X,Y location display on the Status Bar as a guide) and click to add the second terminal.

Add multiple terminals using Step and Repeat

You can use the Step and Repeat command to quickly replicate objects, such as terminals, in a repetitive pattern.

1. Select button .
2. Right-click and click **Select Terminals**.
3. Click the first terminal.
4. Ctrl+click to add the second terminal to the selection.
5. Right-click and click **Step and Repeat**.
6. In the Step and Repeat dialog box, click the **Linear** tab.

   Tip: The Radial Placement licensing option enables Polar and Radial placement tabs.
7. Click **Right** as the Direction, set the Count to **12**, the Distance to **100**, and then click **OK**.

   Result: Twenty-four new terminals are automatically added to complete the connector pin pattern.

Assign pad shape and size

The next step in defining a decal is to assign pad shape and size. This is performed in the pad stack editor.

1. Setup menu > Pad Stacks.
2. Select **<Mounted Side>** from the Shape: Size: Layer list.

Tip: Mounted Side is the side of the printed circuit board, front or back, on which components are mounted.

3. In the Parameters area, click the **Circular** pad shape button.
4. Also in the Parameters area, change the diameter of the pad by typing **55** in the Diameter box.
5. In the Drill Size box, type **28**.
6. To change the pad shape and size for the remaining layers, select **<Inner Layers>** and **<Opposite Side>** from the Shape: Size: Layer list, and set the pad diameter for each to **55**.

   Tip: Inner Layers are design layers other than those on the top or bottom of a printed circuit board. Inner layers may be routing layers, plane layers, or a combination of both. Opposite Side is the outer layer of the printed circuit board opposite the Mounted side layer.
7. Click **OK**.

Tip: If the drill holes are not visible, type **do** and press **Enter**. The do modeless command toggles the drill outline display setting.

Create a component outline

Create part outlines with the 2D line drawing tool.

1. 2D Line button .
2. Right-click and click **Rectangle**.
3. Position the pointer at the upper left corner of the outline by typing **s -100 100** (s, negative 100, space, 100) and pressing **Enter**.
4. Without moving the pointer, press the **Spacebar** to initiate a new rectangular polygon.
5. Move the pointer to location **X1350,Y -50**.
6. Click to complete the polygon.

Adding keepouts to a PCB decal

In PADS Layout, you have the capability to define a region as a keepout for vias, traces, and other design objects. You can define keepouts on the PCB or within the PCB decal.

In this exercise you will define a via keepout for the connector.

1. Keepout button .
2. Right-click and click **Rectangle**.
3. Position the pointer at the upper left corner of the outline and click to begin drawing the keepout rectangle.

4. Move the pointer over the lower right corner of the outline and click to complete the polygon.
5. In the Add Drafting dialog box, in the Restrictions area select the **Via and Jumper** check box to assign the restrictions for the keepout.
6. In the layer list click **<All Layers>** to assign the keepout to all layers.
7. Click **OK** to complete the definition of the keepout.

Assigning PCB decal attributes

In PADS Layout you assign attributes; such as a manufacturer's part number, corporate part number, height, and other component specific information to components or part types. You can also assign attributes to PCB decals.

Add a height attribute

1. **Edit** menu > **Attribute Manager**.
2. If the Geometry.Height attribute is not already in the list:
    a) In the Decal Attributes dialog box, click **Browse Lib. Attr**.
    b) In the Browse Library Attributes dialog box, click **Refresh** to update the attribute listing to view all attributes used in the library.
    c) In the Attribute list click **Geometry.Height**, and click **OK** to add the height attribute to the decal.
3. Double-click in the Value box, and type **300** (in current-unit mils for .300”).
4. Click **Close**.

Adding attribute labels to a PCB decal

You display attributes in the design as text labels. You can also add additional labels to accommodate different size and location requirements for reference designators. The visibility and position of each label is managed within the Layout Editor.

You can predefine locations for attribute labels by adding placeholder labels in the PCB decal. The placeholders act as the defaults for new labels added or made visible in the Layout Editor.

Add a reference designator label to the decal

In this exercise, you will add an assembly drawing reference designator label to the connector decal. Labels are added to the PCB decal using the Add New Label command.

1. Add New Label button .

Result: After you click the Add New Label button, a new label appears just below the current Name label and the Add New Decal Label dialog box appears.

2. In the Attribute list, click **Ref. Des.** to assign the new label as the placeholder for an additional reference designator label.
3. In the Show list, click **Value**, enabling the display of only the value of the attribute.
4. Change the position of the label by typing **600** in the X box and **150** in the Y box. In the horizontal and vertical Justification lists click **Center** and **Center**.
5. In the Layer list, click **Assembly Drawing Top** to assign the text to the top assembly layer.
6. Click **OK** to close the dialog box and complete the addition of the label.

Tip: The new label will not appear if a color is not assigned for objects on the Assembly Drawing Top layer in the Display Colors Setup dialog box.

Exception: Reference designator colors are controlled by the Pin Num. label option instead of Attributes. Part type colors are controlled by the Type label option instead of Attributes.

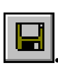
Position the other labels

Position the Name and Type labels in the decal where the text will not be obscured when visible.

1. **Select** button .
2. With nothing selected, right-click and click **Select Text/Drafting.**
3. Select each label and to move it press **Ctrl+E**.
4. Place them in the approximate location shown in the "Creating a PCB Decal" section at the beginning of the tutorial.
5. Click to complete the moves.

Save the PCB decal

You created your first PCB decal in PADS Layout. Save it to the library.

1. Save button .
2. In the Save PCB Decal to Library dialog box, click the **...\preview** library.
3. In the Name of PCB Decal box, type **26pinconn**.
4. Click **OK** and confirm messages that may appear asking if you want to overwrite the file.

Creating a new part type

Now that you've completed the PCB decals required for the tutorial, you will create a part type for the new 26-pin connector using the part type editor.

Assign general parameters

The first steps in creating connector part types are establishing the part type as a connector and assigning a PCB decal.

1. **File** menu > **Library**.
2. To create a new part type in the preview library, click **...\preview** in the Library list.

3. Click the **Parts** button.
4. Click **New**.
5. In the Part Information for Part dialog box, click the **General** tab.
6. In the Logic Family list, scroll and select the **CON** family type, which establishes the default reference designator prefix J for this part type.
7. To establish the part type as a connector, in the Options area, select the Special Purpose check box, and click **Connector**.

Assign a PCB decal

The next step is to assign a PCB decal for the part type.

1. **PCB Decals** tab.
2. In the Library list, click the **...\preview** library.
3. In the Filter box, type **26p*** and click **Apply**.
4. In the Unassigned Decals list, click the **26PINCONN** decal.
5. Click **Assign** to move the decal into the Assigned Decals list.

Add part type attributes

You can add attribute labels in the PCB decal. For part types, you add attributes and their respective values. Enter attributes directly into the attribute cell or click one from a list of all attributes in the PADS library.

For this exercise, you will add attributes to the connector part type from those attributes found in the library.

1. Attributes tab.
2. Click **Add**.
3. Click **Browse Lib. Attr**.

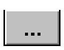
4. In the Browse Library Attributes dialog box, click the **Description** attribute and click **OK**.
5. Double-click in the Value cell.
6. Type **CONNECTOR, RIBBON, 26 PIN** in the Value cell.
7. Click **Add** again, and repeat the previous steps to add these attributes and values:

Attributes	Values
Cost	(leave blank)
Part Number	MGEG26R
Manufacturer #1	ACME
Manufacturer #2	(leave blank)

Assign CAE decals

Now assign Logic decals for the part type.

1. Connector tab.
2. Click **Add**.



3. Click the **Browse** button from the Special Symbol cell.
4. In the Browse for Special Symbols dialog box, select the **EXTIN** symbol in the …\misc library, and click **OK**.
5. Double-click in the Pin Type cell.
6. In the list click **Source** to assign input pins as sources.
7. Click **Add** again, and click the **Browse** button from the newly added Special Symbol cell.
8. In the Browse for Special Symbols dialog box, select the **EXTOUT** symbol in the …\misc library, and click **OK**.
9. Double-click in the Pin Type cell.
10. In the list, click **Load**.

Save the part type

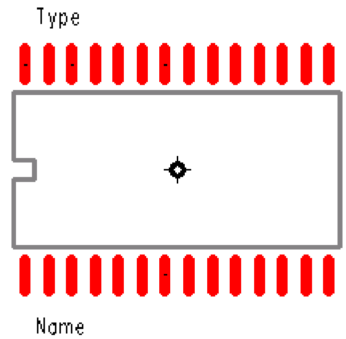
Now that the part type definition is complete, save the new part type.

1. Click **OK**.
2. In the Save Part Type to Library dialog box, navigate to the …\preview library, name the part type **26pinconn**, and click **OK**. Confirm any messages that appear asking if you want to overwrite the file.
3. In the Library Manager dialog box, click **Close**.

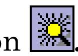
Creating PCB decals using the Decal wizard

The PCB Decal Editor provides dialog box-driven PCB decal wizards. These wizards provide automated tools for creating standard footprint patterns automatically.

Create a SOIC decal



Now that you've experienced PCB decal creation using the interactive tools in the PCB Decal Editor, you will create the SO28 PCB decal, shown at right, using the automated tools in the PCB Decal Editor.

1. On the Drafting Toolbar, Wizard button .
2. In the Decal Wizard dialog box, click the **Dual** tab.
3. In the Decal area, in the Device type section, click **SMD**.
4. In the Orientation section, click **Horizontal**.
5. In the Pins area, enter the following:
   a. Pin count box, type **28**
   b. Width box, type **24**
   c. Length box, type, **90**
   d. Pin pitch box, type **50**
7. In the Row Pitch area, click **Center to Center** and in the Value box, type **450**.
8. In **both** the Pin 1 shape and Pin shape sections, click **Oval**.
9. In the Placement outline area, in the width box type **600**, and in the Height box type **800**.
10. In the Thermal pad area, clear the **Create** check box.
11. Click **OK**. The PCB decal is automatically created.

    Result: You just created a 28-pin SOIC in a few easy steps.

To return to the Layout Editor:

- On the File menu, click **Exit Decal Editor**.

Do not save a copy of the design.

You completed the part creation tutorial.

Preparing a Design

Preparing a design involves the establishment of the layer arrangement, setting display colors, defining a board outline and a few other activities.

In this lesson:

- Establishing a layer arrangement for the PCB
- Setting display colors
- Saving a Start-up file
- Creating a board outline
- Modifying the board outline
- Adding non-electrical objects

Restriction

- This tutorial requires the Drafting Editing, ECO, and General Editing licensing options.

To determine whether you can proceed:

- On the Help menu, click **Installed Options**.

Preparation

If it is not already running, start PADS Layout. Make sure you are starting with a blank screen, or new design.

Establishing a layer arrangement for the PCB

PADS Layout enables you to define layer arrangements for the PCB including the number of layers, the nets associated with embedded plane layers, layer stackup, and layer thickness.

The tutorial design is a four-layer PCB with two internal plane layers supporting multiple plane nets. In this section you will define the layer arrangement for the tutorial design using the layer definition tools of PADS Layout.

Increase the number of layers

The default design is a two-layer PCB.

1. **Setup** menu > **Layer Definition**
2. In the Layers Setup dialog box, in the Electrical Layers area click **Modify**.

3. In the Modify Electrical Layer Count dialog box, in the Enter new number of electrical layers box type **4** to increase the number of layers from 2 to 4.
4. Click **OK**.
5. In the Reassign Electrical Layers dialog box, accept the default reassignment of layer 1 to layer 1 and layer 2 to layer 4 and click **OK**.

Set the layer arrangement and names

Once you set the correct number of layers, assign the layer types for each layer and enter layer names.

The first layer:

1. In the Layers Setup dialog box, select **Top** in the list of layers.
2. In the Name box, type **Primary Component Side**.
3. In the Electrical Layer Type area, click the **Component** layer type.
4. In the Plane Type area, click **No Plane**.
5. Set the Routing Direction to **Vertical**.

The second layer:

1. Select the second layer, **Inner Layer 2**, and name it **Ground Plane**.
2. Change the Plane Type to **CAM Plane**.
3. Set the Routing Direction to **Any**.

The third layer:

1. Select the third layer, **Inner Layer 3**, and name it **Power Plane**.
2. Change the Plane Type to **Split/Mixed** (plane).
3. Set the Routing Direction to **Any**.

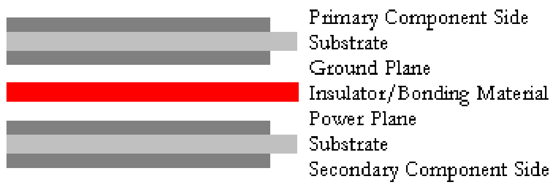
The last layer:

1. Select the fourth layer, **Bottom**, and name it **Secondary Component Side**.
2. In the Electrical Layer Type area, click a **Component** layer type.
3. In the Plane Type area, click **No Plane**.
4. Set the Routing Direction to **Horizontal**.

Set layer stackup

A typical layer stackup for a four-layer PCB is composed of a pair of copper clad fiberglass substrates, with an insulator/bonding material in between. The insulator/bonding material is typically a resin sheet that bonds the substrate pairs together when a multi-layered board is manufactured. This bonding material is also known as *Prepreg*.

Layer stackup:



Use the Layer Thickness dialog box to set the layer stackup values.

1. In the Layers Setup dialog box, click **Thickness**.
2. In the Layer Thickness dialog box, for the Copper Thickness Units click **Weight (oz)**.
3. In the Layer list, click **Primary Component Side**.
4. Double-click in the Thickness box for **Primary Component Side** and type **2** to reflect that it has a copper weight of 2 ounces

   Tip: 1 oz. of copper weight = .00135" of copper thickness.
5. In the Layer list, click **Ground Plane**.
6. Double-click in the Thickness box for **Ground Plane** and type **1** to reflect that it has a copper weight of 1 ounce.
7. In the Layer list, click **Power Plane**.
8. Double-click in the Thickness box for **Power Plane** and type **1** to reflect that it has a copper weight of 1 ounce.
9. In the Layer list, click **Secondary Component Side**.
10. Double-click in the Thickness box for **Secondary Component Side** and type **2** to reflect that it has a copper weight of 2 ounces.
11. Double-click in the **Substrate** box between **Ground Plane** and **Power Plane** and click **Prepreg** to establish this level as the insulator/bonding layer.
12. Double-click in the Thickness box for the Prepreg layer and type **35**.

    Tip: The Dielectric Constant values listed are values given for manufacturing materials such as FR-4 to describe electrical characteristics.
13. Click **OK** twice.

Setting the grids

To establish a working grid for your design, use the grid modeless commands.

1. To set all grids to 100 mils, type **g100** and press **Enter**.
2. To set the Display Grid to 200, type **gd200** and press **Enter**.

Setting display colors

The Display Colors Setup dialog box assigns or changes layer colors and makes individual items visible or invisible. You can also set the color for the screen background, board outline, and other elements here.

Assign new colors to the new layers

The increase in the number of layers requires additional color assignments.

1. **Setup** menu > **Display Colors**
2. In the Display Colors Setup dialog box, in the Layers/Object Types area, click the number **2** to select the row associated with the Ground Plane layer.
3. In the Selected Color area, click the light-brown tile. All objects on the layer assume the light-brown color.

   Tip: A square highlight indicates the active color.
4. Repeat steps 2 and 3 for the Power Plane layer, using light green for all objects.

Make objects invisible

To hide objects, assign them the same color as the design background.

1. Click the black tile (the same as the background color) in the Selected Color area.
2. In the Other area, click **Connection** to make connections invisible.

   Tip: Connections are the unrouted connection lines, often called ratsnest or airlines.
3. In the **Type** column, clear the check box to make part type names invisible on all layers.
4. For the **Ref. Des.**, **Top**, and **Bottom** columns click the tiles for both the Ground Plane and Power Plane layers since no reference designators, or component outlines appear on those layers.

Set colors for other objects

1. Click **Errors** to select the column for all layers and click the white tile to set the Errors for Primary Component Side to white.
2. Click the purple tile. In the **Ref. Des.** column click the tile for **the Secondary Component Side**.
3. With purple as the selected color, in the **Bottom** column, click the tile for the **Secondary Component Side**.
4. Click the light-gray tile. In the **Top** column, click the tile for the **Primary Component Side layer**.

5. Click the yellow tile. In the **Keepouts** column, click the tile for the **Primary Component Side** layer.
6. Click **OK**.

Saving color arrangements

You can save color arrangements to reuse them from design to design. The Save option in the Display Colors interface allows saving of color configurations. The list can then be toggled to activate and apply the color changes.

Saving a Start-up file

Start-up files are a method of reusing design options from one design to another. A start-up file contains predefined system settings such as the number of layers, display color assignments, current working grids, design rules, and many other commonly used options. You can create start-up files for a particular board type, such as a two layer or four layer PCB. You can then use the start-up file for each PCB design and avoid entering standard options with every new design.

Each time you start PADS Layout, the last start-up file used automatically opens. Each time you click New on the File menu, however, the Set Start Up File dialog box appears so you can choose a start-up file.

Saving a start-up file

Now that you established options for your design, you can save them in a start-up file for use with your next four-layer design.

1. **File** menu > **Save As Start-up File**
2. In the Save As Start-up File dialog box, type **preview** in the File name box, and then click **Save**.
2. In the Start-up File Output dialog box, type **PADS Layout Tutorial** in the Startup File Description box, leave the CAM section check box cleared, and click **OK**.

The current system settings are saved to the preview.stp file in the C:\Mentor Graphics*<latest_release>*\SDD_HOME\Settings folder.

Creating a board outline

A board outline is created using polygon creation methods similar to those for creating drafting items such as part outlines, copper areas, and copper pour areas.

1. **Drafting Toolbar** button > **Board Outline and Cut Out** button
2. Right-click and click **Rectangle**.
3. Move the pointer to location **X200**,**Y1900** and click to add the first corner of the board outline. A dynamic line attaches to the pointer.
4. Use the search modeless command to place the pointer at a specified location. Type **s 3000 300** (including spaces) and press **Enter**.
5. Press the **Spacebar** to complete the rectangle.

Modifying the board outline

The next procedure modifies a section of the board outline by changing one of the existing straight lines into an arc. Before you modify the board outline, you must adjust the Selection Filter to define the types of objects you can select.

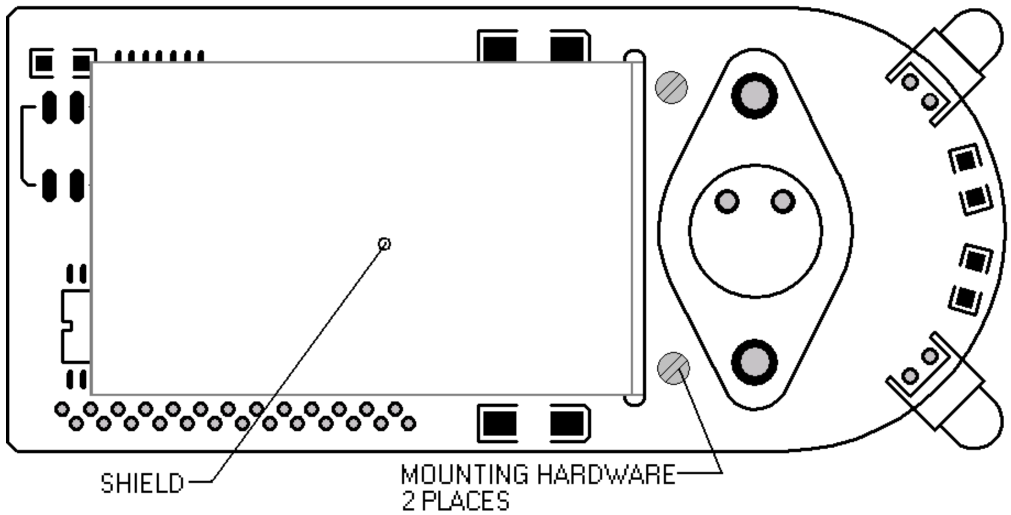
1. Right-click and click **Select Board Outline** to enable selection of board outline elements.
2. Click the vertical line on the right side of the board outline to select it.
3. Right-click and click **Pull Arc**.
4. Move the pointer to the right, pulling the arc away from the board, to location **X3800**,**Y1100**.
5. Click to place the arc.

   Tip: If you accidentally place the arc in the wrong location, repeat Steps 2 through 5.
6. With the arc still selected, right-click and click **Select Shape** to select the entire board outline.
7. Right-click and click **Add Miters**.
8. In the Add Miters dialog box, type **35** in the Radius box and click **OK**.

   Result: All 90-degree corners of the PCB are globally mitered at the specified value.
9. On the standard toolbar, click the **Board** button to fit the board outline to the screen.

Adding non-electrical objects

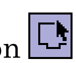
This tutorial design has a requirement for a metal shield that covers an area mostly around the center of the board. The shield is mounted on the bottom of the PCB, protrudes through a slot and covers an area on the top of the PCB. The shield is fastened to the PCB using two small screws.



Managing the shield in PADS Layout requires the creation of a slot (or board cutout) and a keepout area. In addition, it requires the addition of a component with drilled holes and a keepout to represent the shield in the design.

Creating the board cutout

You create a board cutout using the board outline and cut out command. One board outline is allowed per design. Since a board outline is already present, PADS Layout assumes the next use of the board outline and cut out command is for the creation of a board cutout.

1. **Drafting Toolbar** > **Board Outline and Cut Out** button 
2. Click **OK** to confirm creation of a cut out, and continue the operation.
3. Right-click and click **Rectangle**.
4. Move the pointer to location **X2400,Y1700** and click to begin the rectangle.
5. Move the pointer to **X2500,Y500** and click to complete the board cutout.

To complete the definition of the cutout you must convert the ends to arcs.

1. To change the working grid, type **g50** and press **Enter**.
2. Select one of the short horizontal segments of the cutout, right-click, and click **Pull Arc**.
3. Create a 180 degree arc at the end.

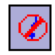
   Tip: The Status Bar contains the radius and arc degree display in the section normally reserved for the grid setting.

4. Repeat Steps 2 and 3 for the other end segment of the cutout.

Create a height keepout

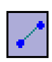
You can define a region as a keepout for vias, traces, and other design objects. You can define keepouts on the PCB or within a PCB decal.

In this exercise, you define a height keepout for the PCB under the area covered by the shield.

1. **Drafting Toolbar** > **Keepout** button 
2. Right-click and click **Rectangle**.
3. Position the pointer at the upper left corner of the board cutout (at **X2400**,**Y1700**) and click to begin the keepout rectangle.
4. Move the pointer towards the lower left corner of the board (at **X600**,**Y500**) and click to complete the keepout.
5. In the Add Drafting dialog box, in the Restrictions area select the **Placement** check box to assign the keepout as a placement keepout.
6. Select the **Component height** check box, and type **100** in the value box to keep all components with a height attribute greater than 100 out of the keepout area.
7. In the Layer list, click **Primary Component Side**, to assign the keepout to the top layer only.
8. Click **OK**.

Adding the mounting hole component

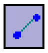
Mounting holes are added to a design as a component. For the tutorial, the mounting hole component was previously created for you. You only need to add the component to the design using the ECO commands.

1. ECO Toolbar button 
2. In the ECO Options dialog box, clear the **Write ECO File** check box in the ECO File area, and then click **OK**.

3. On the ECO toolbar, click the **Add Component** button.
4. In the Get Part Type from Library dialog box, click the **...\preview** library in the Filter area.
5. In the Items area, type **sh*** and click **Apply**. The SHIELD part type appears in the Part Types list.
6. Select the **SHIELD** part type and click **Add** to add the shield component to the design.

   Result: The part is added to the design and attached to the pointer.
7. With the part attached to the pointer, click **Close** to close the Get Part Type from Library dialog box.

8. Move the shield component to **X2600**,**Y600**, and click to place it.



9. Click the ECO Toolbar button again to exit ECO mode and adding components.

Glue the mounting hole component

Glue a component to prevent it from being moved inadvertently.

1. With nothing selected, right-click and click **Select Components**.
2. Click the SHIELD mounting holes component **X1**.
3. Right-click and click **Properties**.
4. In the Component Properties dialog box, select the **Glued** check box to assign it as a glued component.
5. Click **OK**.
6. Do not save a copy of the design.

You completed the design preparation tutorial.

Importing Design Data

You can import design data into PADS Layout from many external sources. The most common method of passing design data into PADS Layout is importing a PADS formatted ASCII file. It's the method used to import data from schematic capture products like PADS Logic or DxDesigner.

In this lesson:

- Importing a netlist from an ASCII file
- Assigning nets to the plane layers

Preparation

If it is not already running, start PADS Layout and open the file named **previewbrd.pcb** found in the \PADS Projects\Samples folder.

Importing a netlist from an ASCII file

A netlist typically contains a list of all parts used on the PCB and the nets that interconnect the parts.

Tip: These are the typical steps used to import a PADS Layout netlist exported from PADS Logic. When netlists are exported from DxDesigner, the PADS Layout netlist file (with an *.asc file extension) is located in the DxDesigner project directory.

Import the file

1. **File** menu > **Import**.
2. Navigate to the **Samples** folder and click the file named **previewnet.asc**.
3. Click **Open**. After the import process completes, components are placed at the design origin, ready for placement.
4. On the **View** menu, click **Extents** to see all of the parts placed at the design origin.

Assigning nets to the plane layers

Now that you added nets to the design, you can associate the plane layers to their respective nets. Do this using the Layers Setup dialog box.

1. **Setup** menu > **Layer Definition**.
2. In the Layers Setup dialog box, select the **Ground Plane** layer from the Layers list, and click **Assign Nets**.

3. In the Plane Layer Nets dialog box, scroll through the All Nets list, select **GND** from the All Nets list, and then click **Add** to move it to the Assigned Nets list.
4. Click **OK** to assign the GND net with the Ground Plane layer.
5. In the Layers list, select the **Power Plane** layer and then click **Assign Nets**.
6. In the Plane Layer Nets dialog box, in the All Nets list select **+5V**. Ctrl+click to also select **+12V** from the All Nets list.
7. Click **Add** to move the nets to the Assigned Nets list.
8. Click **OK** to assign the +5V and +12V nets with the Power Plane layer.
9. In the Layers Setup dialog box, click **OK**.
10. Do not save a copy of the design.

You completed the importing design data tutorial.

Importing Schematic Data

The DxDesigner Link is a multi-purpose tool primarily used for passing data between DxDesigner and PADS Layout. It also provides features for cross-selection between the products (known as *cross-probing*).

Using the DxDesigner Link in PADS Layout you will process a DxDesigner schematic and pass part, net, and constraint data directly to PADS Layout to start a design. Using the cross-probing features, you will perform schematic driven placement. You will also update a schematic with changes made in PADS Layout.

In this lesson:

- Updating a PCB design with schematic data
- Schematic driven placement
- Updating a schematic with PCB data

Requirements

- This tutorial requires that you install both DxDesigner and PADS Layout on your system.
- This tutorial requires the General Editing and ECO licensing options. To determine whether you can proceed, on the Help menu, click **Installed Options**.

Preparation

If it is not already running, start DxDesigner and open the **Preview.prj** tutorial project found in the \PADS Projects\Samples\Preview folder. Open page 1 of the Preview schematic.

If it is not already running, start PADS Layout and open the file named **previewbrd.pcb** found in the \PADS Projects\Samples folder.

Resizing the view

To get the greatest benefit of this tutorial, it is recommended you re-size the applications to each use half of your desktop.

Establish communication links

The DxDesigner link streamlines the passing of data and communication between DxDesigner and PADS Layout and is the preferred method for synchronizing schematic and PCB design databases.

An additional benefit of the DxDesigner link is inter-tool communication. Use this automation technology to *cross-probe* between linked applications. Cross-probing is having selections of nets, components, or pins made in one application result in the selection of the corresponding object in the linked application.

To enable inter-tool communication, you start the DxDesigner link and connect with a current or new session of DxDesigner.

To connect DxDesigner:

1. PADS Layout **Tools** menu > **DxDesigner**
2. In the DxDesigner Link, click the **Documents** tab.
3. In the DxDesigner Project File area, verify the project is set to preview.prj and then click **Connect** to establish a communication link.

Tips:

- When the link is established, the Connect button changes to Disconnect. Use Disconnect to break the communication link.
- Upon connection with DxDesigner, two new tabs appear in the dialog box: Selection and Placement. These tabs are discussed later in this tutorial.

Updating a PCB design with schematic data

Creating a netlist is the basic method for passing schematic data into PADS Layout to start the PCB design process. A netlist contains a list of the parts, their part types, and all of their net connections.

Use the DxDesigner Link to automatically generate and send a netlist to PADS Layout.

Send a netlist to PADS Layout

1. In the DxDesigner Link, click **Forward to PCB**.
2. In the Forward Annotation dialog box, click **Update PCB**.
3. In the Data to Pass area, select all three options to update parts, netlist, attributes, design rules, and library parts.
4. Clear the **Pause before updating design** check box and click **OK** to start the process.

   Result: The Process indicator appears and is updated at the completion of each step in the process. After the process completes, all the components are positioned at the design origin, ready for placement.
5. Click the **Show Report** button in the progress indicator to view a status report generated by each process.
6. Click **Close** to close the Process indicator dialog box.

Requirement: Do not close the DxDesigner Link dialog box.

Schematic driven placement

Now that you have a PADS Layout design with the parts and nets from the schematic, you will prepare the parts for placement. You will use cross-probing to perform schematic-driven placement. You begin by placing PADS Layout in Move mode so part selections made in PADS Logic result in movement of the selected part.

Set auto-panning mode

The DxDesigner link provides options for automatic panning and zooming to selected objects. For the purpose of this tutorial, you will disable the automatic zooming and panning.

1. In the DxDesigner Link, click the **Selection** tab.
2. In the Selection Passing area, set the PADS Layout zoom mode to **None**.

Establish a working and design grid

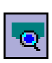
1. In PADS Layout, set the design grid to 50 mils by typing **g50** and pressing **Enter**.
2. Set the display grid to 50 mils by typing **gd50** and pressing **Enter**.

   Tip: The display grid may not be visible until you zoom in.

Disperse the components

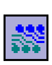
Use the Disperse command to distribute the components around the outside of the board outline.

1. In PADS Layout, on the Tools menu, click **Disperse Components** and then click Yes to confirm the dispersion.

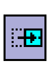


2. Click the **Board** button to fit the view to the board outline.

Enable the Move Component verb mode



1. In PADS Layout, click the **Design Toolbar** button.



2. On the Design toolbar, click the **Move** button to place PADS Layout into Move (Component) mode.

Make a selection in the schematic

1. In DxDesigner, click any pin of the **J1** connector on the left side of the schematic.

   Result: J1 is selected in PADS Layout.

2. Move your pointer to the PADS Layout window; you are now moving J1 in PADS Layout.

Place the part

Place the component on the opposite side of the board at a specific location using the part placement commands of PADS Layout.

1. In PADS Layout Right-click and click **Flip Side** to flip the part to the opposite side of the PCB.
2. Using the grid coordinate readout on the PADS Layout status bar as a guide, click to place JI at **X1650, Y400**.

Updating a schematic with PCB data

You can update a schematic with changes made in PADS Layout using the Backward from PCB button. This process compiles a netlist from the PCB database and a PADS Layout ECO file and updates the schematic.

For this tutorial, you will open a PCB design, rename all of the components, and pass the updates back to the schematic.

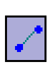
Open the PCB design and re-establish the communication link

1. In the Documents tab of the DxDesigner Link, in the PADS Layout Design area, click **Disconnect**.
2. Click **Browse**.
3. In the File Open dialog box, select the **previewplaced.pcb** file and click **Open**.
4. In the PADS Layout Design area, click **Connect** to connect PADS Layout to DxDesigner once again.

Tip: Do not save any changes in the current file if prompted to do so.

Rename the parts

Rename components using the ECO commands of PADS Layout. The ECO commands are a collection of editing tools for changing a PCB design database and include add and delete part, add and delete net, and other editing commands. Access the ECO commands from the ECO toolbar in PADS Layout.

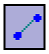


1. **ECO Toolbar** button.

   Tip: If the ECO Options dialog box does not appear, click the Options button on the ECO toolbar.
2. In the ECO Options dialog box, select the **Write ECO file** check box in the ECO file area.
3. Select the **Write ECO file after closing toolbar** check box.
4. Leave all other settings and options as they appear and click **OK**.

5. On the ECO toolbar, click the **Auto Renumber** button.

6. In the AutoRenumber dialog box, accept the default settings, and click **OK** to rename all of the parts on the design.



7. Click the ECO Toolbar button again to close the toolbar and to write the changes to the ECO file.

Update the schematic

Update the schematic using the Backward from PCB operation.

1. In the DxDesigner Link dialog box, click the **Backward from PCB** button.
2. In the Backward Annotation dialog box, click **Use Existing ECO file** in the ECO preference area.
3. In the Data to Pass to Schematic area, select all options to update parts, netlist, attributes, and design rules.
4. Clear **Pause before updating schematic** and click **OK** to start the process. The Process indicator appears and is updated at the completion of each step in the process.

   Tip: If a warning appears regarding Rule changes, click Yes.
5. When the process completes, click the **Show report** buttons in the progress indicator to view a status report generated by each process.
6. Click **Close** to close the indicator dialog box.
7. Do not save a copy of the design.

You completed the DxDesigner link tutorial.

Assigning Constraints

Assigning constraints to your design allows you to easily control and verify critical design areas. Constraint types include clearance, routing, and high-speed rules that can be assigned to nets, layers, classes (collection of nets), groups (collection of pin pairs), or individual pin pairs. You can also assign a default set of design rules that apply to all objects not having a unique rule.

In this lesson:

- Setting default clearance rules
- Setting net clearance rules
- Setting conditional rules

Restriction

This tutorial requires the Advanced Rules (for the Conditional Rules section) and General Editing licensing options.

To determine whether you can proceed:

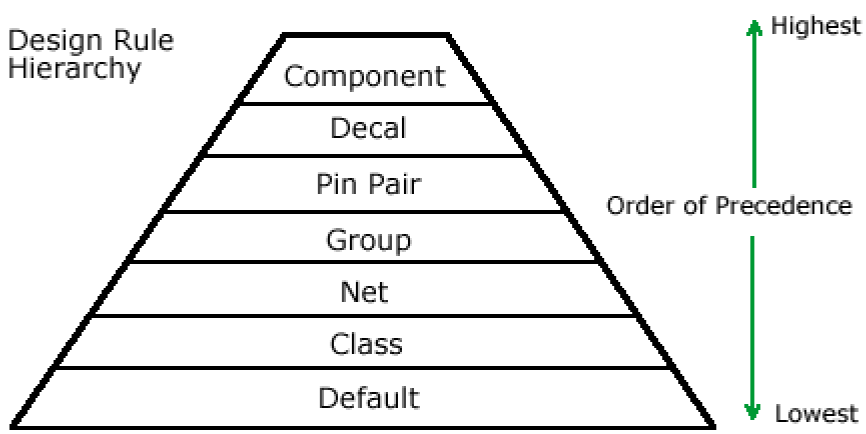
- On the Help menu, click **Installed Options**.

Preparation

If it is not already running, start PADS Layout and open the file named **previewnet.pcb** found in the \PADS Projects\Samples folder.

Setting Default Clearance Rules

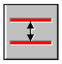
You can define clearance, routing, and high-speed rules for each level of the design rule hierarchy.



Setting default clearance rules

The Clearance area of the Clearance Rules dialog box contains a matrix of PCB design data. You can specify values for each or all data types in the matrix.

1. **Setup** menu > **Design Rules** > **Default** button.



2. Clearance button.
3. Click **All** (in the upper-left corner of the Clearance matrix) to set a global default clearance value.
4. In the Input Clearance Value dialog box, type **8** and click **OK**.
5. In the Trace Width area, type **6** in the Minimum box, type **8** in the Recommended box, and type **12** in the Maximum box.
6. Type **12** in the Same Net and Other clearance text boxes, with the exception of Trace to Crn box. Set this box to **0**.
7. Click **OK**.

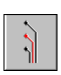
Set default routing rules

To avoid routing on the plane layers, remove them from the Selected routing layers as defined in the routing rules. The Layer Biasing area of the Routing Rules dialog box contains a list of selected routing layers. This list lets you specify which layers are permitted for routing.

1. **Routing** button.
2. In the Layer biasing area, in the Selected Layers list, select the **Ground Plane**, press and hold **Ctrl** and click **Power Plane** to add them to the selection.
3. Click **Remove** to prevent routing on the plane layers.
4. Click **OK** to close the Routing Rules dialog box.
5. Click **Close** to close the Default Rules dialog box.

Setting net clearance rules

You can assign net-specific clearances that take precedence over the default rules previously entered.



1. **Net** button.
2. Scroll through the Nets list. Ctrl+click to select **+5V**, **+12V**, and **GND**. The three selected nets appear beside Selected, under the rule type buttons.
3. Click the **Clearance** button to set the same clearance rules for all three nets.
4. In the Clearance Rules dialog box, click **All** (in the upper-left corner of the matrix) to set a global clearance value.
5. In the Input Clearance Value dialog box, type **10** as the global clearance and then click **OK**.

6. In the Trace Width area, type **10** in the Minimum box, type **12** in the Recommended box, and type **15** in the Maximum box.
7. To complete the definition, click **OK** in the Clearance Rules dialog box.
8. Click **Close** to close the Net Rules dialog box.

See also: *PADS Layout Help* for details about defining other Hierarchy rules.

Setting conditional rules

When two nets require a specific clearance between each other (to avoid adverse effects on the circuitry), you must define a conditional rule. An example of a conditional rule might be the Underwriters Laboratories (UL) requirements of segregating primary, secondary, and ground nets when alternating current is directly connected to the PCB. You can assign conditional rules between most components of the design rule hierarchy. Conditional rules can exist between nets, nets and classes, classes and pin pairs, nets on a specific layer, and so on.

To assign a net-to-net conditional rule:

1. **Conditional Rules** button.
2. In the Source Rule Object area, click **Nets.**

   Result: A list of nets appears in the Source Rule Object list.
3. Select net **+5V**.
4. In the Against Rule Object area, click **Nets**.

   Result: A list of nets appears in the Against Rule Object list.
5. Select net **+12V**.
6. Click **Create** to define the conditional rule. The new rule appears in the Existing Rule Sets area.
7. In the Current Rule Set area, type **25** in the Object to Object box.
8. Close all of the open dialog boxes.

   Result: The rule you just created will keep all objects pertaining to the +5V and +12V nets 25 mils apart.
9. Do not save a copy of the design.

You completed the design rules tutorial.

Moving and Placing Components

PADS Layout offers several methods for placing components.

In this lesson:

- Pre-placement procedures
- Placing components using Move
- Placing components using Find
- Placing components using Move Sequential
- Placing components using Properties
- Experimenting with the placement tools

Restriction

This tutorial requires the General Editing licensing option.

To determine whether you can proceed:

- On the Help menu, click **Installed Options**.

Preparation

If it is not already running, start PADS Layout and open the file named **previewrules.pcb** in the \PADS Projects\Samples folder.

Preplacement procedures

You must complete several procedures before you begin placing parts.

Setting a placement grid

1. To set the design grid to 50 mils, type **g50** and press **Enter**.
2. To set the display grid to 50 mils, type **gd50** and press **Enter**. The display grid may not be visible. If you want to see it, just zoom in.

Dispersing the components

Disperse the components around the board outline to make them all visible.

1. **Tools** menu > **Disperse Components**.
2. Click **Yes** to confirm the dispersion.
3. On the View menu, click **Extents** to fit the board and all the components in view.

Setting the net colors

To aid in the placement and correct orientation of decoupling capacitors, assign a color to the +5V net.

1. **View** menu > **Nets**.
2. In the View Nets dialog box, select **+5V** in the Net list.
3. Click **Add** to add +5V to the View list.
4. In the View list, select **+5V**.
5. Click **dark gray** to display all component pins and vias connected to +5V as dark gray.

Setting the net visibility

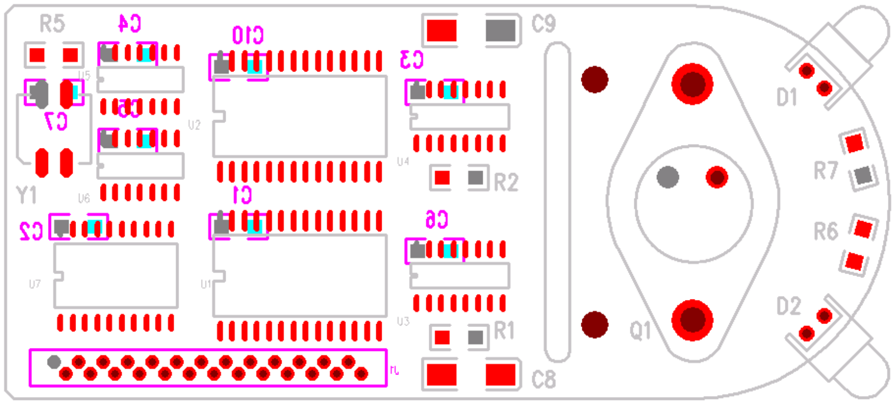
Temporarily make the plane nets invisible to help determine the best location for components.

1. With the **+5V** net still selected, clear the **Traces Plus the Following Unroutes** check box to make the net invisible.
2. In the Net list, select both the **+12V** and **GND** nets.
3. Click **Add** to move them to the View List.
4. In the View list, select both the **+12V** and **GND** nets, and clear the **Traces Plus the Following Unroutes** check box to make them invisible.
5. Click **OK** to apply the visibility settings and close the View Nets dialog box.

Placing components using Move

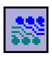
In this section you will learn how to use the Move command to place components for the tutorial design. The component positions are established to efficiently use your time with the tutorial.

Placement preview:

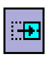


Placing the ICs

Place the ICs and their associated decoupling capacitors.



1. **Design Toolbar** button.



2. **Move** button.
3. Type the modeless command **ssu1** and press **Enter** to search and select U1. U1 attaches to the pointer and all of the nets connected to U1 appear.
4. Position U1 at X**1400**,Y**800** and click to place it.
5. Type the modeless command **ssu2** and press **Enter** to search and select U2.
6. With U2 attached to the pointer, move U2 left to right across the board horizontally and note how the nets reconnect as U2 moves. This is called *dynamic reconnect* and is useful when placing components.
7. Type the modeless command **s1400 1450** to position U2 at X**1400**,Y**1450** and press the **Spacebar** to place it.
8. Use Move to complete the placement of the following components at their respective locations:

Component	Move to
U3	2050 800
U4	2050 1450
U5	750 1600
U6	750 1250
U7	650 800

Placing the decoupling capacitors

Use the operations described in the above procedure and place the decoupling capacitors.

1. Type the modeless command **ssc1** and press **Enter** to search and select C1.

Result: C1 attaches to the pointer, but its nets do not appear because the +5V and GND nets are invisible.

2. To place C1 on the opposite side of the board, right-click and click **Flip Side**.
3. To complete the placement of C1, rotate it 180 degrees. Right-click and click **Rotate 90** twice.
4. Place it under U1 at **X1150,Y1000**.
5. Repeat the steps above to place the remaining capacitors:

Component	Move to
C2	500 1000
C3	1950 1550
C4	700 1700
C5	700 1350
C6	1950 900
C10	1150 1650

Placing with keepouts

Use this same method to locate, select, and move oscillator Y1. You will also experience the behavior of the placement commands when placement keepouts are present.

Requirement: At this point in the tutorial the DRC mode is set to off.
To enable placement keepout checking, you must change the DRC mode to Prevent.

1. To enable On-line DRC, type the modeless command **drp** and press **Enter**.
2. With Move active, type the modeless command **ssy1** and press **Enter** to search and select Y1. Y1 attaches to the pointer.
3. Position Y1 at **X2050,Y1100** (between U3 and U4), and click to place it.

   Result: You are prevented from placing a component with a height attribute that exceeds the allowable height in the keepout area.
4. To rotate Y1 90 degrees, right-click and click **Rotate 90**.
5. Place Y1 at **X400,Y1400**.
6. To disable On-line DRC, type the modeless command **dro** and press **Enter**.
7. Click **OK** to confirm the operation.

See also: On-line DRC operation and behavior is covered in the Design Verification tutorial.

Placing components using Find

You can also select and move components using the Find command. The Find command provides a mechanism to quickly search and locate design components and automatically apply commands like Select, Highlight, and Rotate 90.

Placing connector J1

You will use this method to locate, select, and move connector J1.

1. **Edit** menu > **Find**.
2. In the Find dialog box, in the Find By list, select **Ref. Designator**.
3. In the Ref. Des. Prefix list select **J** and in the Ref. Designators list **J1**.
4. Click **Apply**.

   Result: J1 attaches to the pointer. Do not close the Find dialog box at this time.
5. In the design area, right-click and click **Flip Side**.
6. Click at **X1650**,**Y400** to place J1.

Placing capacitor C7

1. In the Find dialog box, select **C** from the Ref. Des. Prefix list, and **C7** from the Ref. Designators list.
2. Click **Apply**.
3. In the design area, right-click and click **Flip Side** to flip C7 to the Secondary Component Side.
4. Place C7 at **X400**, **Y1550**.

Placing components using Move Sequential

You can select multiple components and automatically move each of the parts sequentially. You will use this method to place resistors R1, R2, and R5.

1. Design Toolbar > **Select** button.
2. In the Find dialog box, select **Move Sequential** in the Action list.
3. In the Ref. Des. Prefix list, select **R**.
4. In the Ref. Designator list, Ctrl+click to select **R1**, **R2**, and **R5**.
5. Click **Apply**.
6. Click **Yes to All** in response to the message.

   Result: The first component in the group attaches to the pointer. Once the first component is placed, the next component in the group automatically attaches to the pointer.
7. Place the resistors at the following locations:

Resistor	Move to
R1	2050, 550
R2	2050, 1200
R5	400, 1700

8. Close the Find dialog box.

Placing components using Properties

You can use the Component Properties dialog box to place parts. Use this method to place the LEDs and their resistors. These parts are positioned around the arc at the right side of the board.

1. **View** menu > **Extents**.
2. Type the modeless command **ssd1** and press **Enter**.
3. Right-click and click **Properties**.
4. In the Layout Data area, type **3500** in the X box, type **1600** in the Y box, and type **315** in the Rotation box.
5. Click **Apply** to modify the location and orientation of the part.
6. Leave the Component Properties dialog box open, and click in the workspace. Type **ssd2** and press **Enter** to select D2.
7. Place D2 and resistors R7 and R6 at the following locations:

Component	Move to

D2	3500, 600 with a rotation of 225 degrees
R6	3675, 925 with a rotation of 255 degrees
R7	3675, 1275 with a rotation of 105 degrees

8. Click **OK**.

Experiment to place the three remaining components

Place transistor Q1 and its filter capacitors C8 and C9 using any of the placement methods previously described in this tutorial.

Place the components at:

Component	Move To
Q1	3100 1200 with a rotation of 90 degrees
C8	2100 400
C9	2100 1800

Do not save a copy of the design.

You completed the moving and placing components tutorial.

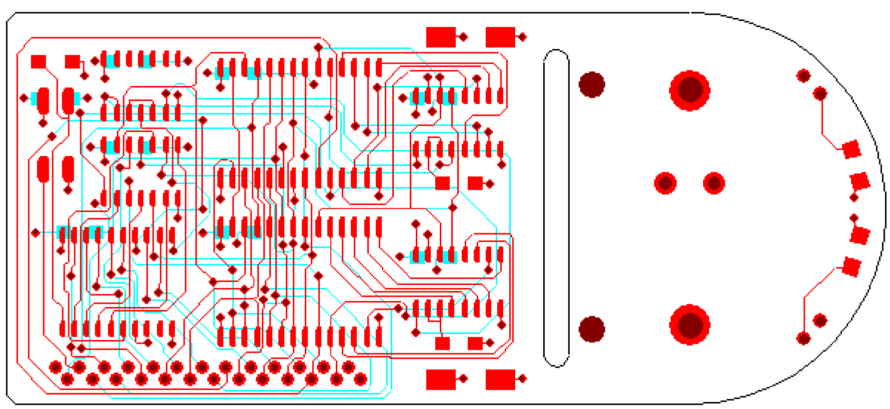
Creating and Editing Traces

PADS Layout contains several interactive and semi-automatic routing tools designed to shorten design time and increase productivity. These tools include Dynamic Route Editing (DRE), Bus Routing of two or more traces simultaneously, curved traces, mitered trace corners, T-Routing (for tying into tracks of the same net regardless of start or end points), On-line Design Rule Checking (DRC), and copy routines.

In this lesson:

- Pre-routing procedures
- Using the manual route editor
- Modifying traces
- Copying traces
- Deleting traces
- Routing plane nets
- Using on-line design rule checking (DRC)
- Routing with keepouts and cutouts

Example of the completely routed PCB



Restriction

This tutorial requires the General Editing, and Split Planes licensing options.

To determine whether you can proceed:

- On the Help menu, click **Installed Options**.

Preparation

If it is not already running, start PADS Layout and open the file named **previewplaced.pcb** in the \PADS Projects\Samples folder.

Prerouting procedures

Before you begin routing, you need to perform a few prerouting procedures. The procedures vary by individual and by design. The following preparatory steps are designed specifically for this tutorial. Follow these steps to receive the full benefit of the tutorial.

Change the color display of items

To improve visibility and reduce screen clutter while you are routing traces, disable the display of items not required for interactive routing.

1. **Setup** menu > **Display Colors**
2. In the Display Colors Setup dialog box, clear the layer check boxes for the **Ground** and **Power plane** layers.

   Result: Disables the display of these plane layers.
3. In the Selected Color area, click the background color, **black**.
4. For both component layers (1 and 4) click **Ref. Des.**, **Attributes**, **Keepouts**, **Top** Outline, and **Bottom** Outline to make these items invisible.
5. In the Selected Color area, click **light green**.
6. In the Other area, click **Connection** to display the connections as light green.
7. Click **OK** to accept the changes and close the Display Colors Setup dialog box.

Define a layer pair

Defining a routing layer pair minimizes the amount of time spent on manual layer changes during interactive routing. Pairing routing layers limits layer changes to the members of the layer pair. For this four-layer design, the obvious routing pair is the Primary and Secondary Component layers.

1. **Tools** menu > **Options** > **Routing** > **General** page.
2. In the Layer Pair area, in the First layer list, select **Primary Component Side** and in the Second layer list select **Secondary Component Side**.
3. In the Options area, clear the **Show Trace Length** check box to facilitate routing later in this tutorial.

Tip: Do not close the Options dialog box.

Define the default trace angle

The trace angle setting determines the degree of adjacent trace segments as they are introduced during interactive routing. There are three trace angle settings:

Angle Setting	Description
Diagonal	Adjacent trace segments are limited to 45 degree intervals.
Orthogonal	Adjacent trace segments are limited to 90 degree intervals.
Any Angle	Adjacent trace segments are not limited to any angle.

For the purpose of this tutorial, set the Line/Trace Angle to Orthogonal.

1. **Design** page.
2. In the Line/Trace Angle area, click **Orthogonal**.
3. Click **OK** to close the Options dialog box.

Set a routing and via grid

In this lesson, use a routing and via grid to assist in the learning process.

Set a routing grid and via grid to 8 1/3 mils to accommodate the 8 mil trace and 8 mil space requirements of the design.

To set the routing grid and via grid:

- Type **g8.33** and press **Enter**.

Enable real width display

You can choose to enable or disable true width display to see trace widths in their real width. The default is to display lines and traces at 8 mils or greater in real width. Set the real width display to 0 mils.

To set the real width display:

- To display all lines and traces equal to or greater than 8 mils in their real width, type the modeless command **r0** and press **Enter**.

Using the manual route editor

The manual route editor is the core route editing function. Many of the operations for defining traces in the route editor resemble other operations covered in the tutorials, such as those used for creating polygons and line items. This minimizes the learning curve by letting you apply the same skills to many areas.

All connections are converted to traces by selecting the connection and creating new corners and layer changes using pointer actions and keyboard combinations.

Resize the view

Zoom in for better visibility.

1. Highlight the net 24MHz using the highlight net modeless command: type **n 24mhz** and press **Enter**.
2. Zoom into the upper left corner of the design, centering the view around the short portion of the 24MHz net, connecting the oscillator to the resistor.
3. Type **n** (without the netname) and press **Enter**, to unhighlight the net 24MHz.

Start routing

Route with the manual route editor.

1. With nothing selected, right-click, and click **Select Traces/Pins/Unroutes**.
2. On the standard toolbar, in the Layer list, select **Primary Component Side** to set it as the current layer.
3. Select the short connection of the net 24MHz, between the oscillator and the resistor.
4. Right-click and click **Route**.

   Result: The beginning of a new trace segment attaches to the pointer.

   Tip: At this point, you are in DRC Off mode. New trace segments are not prohibited from shorting to other objects.
5. Experiment with the pointer movements and trace reactions. Move the pointer around and note how one end of the connection attaches to the end of a new route segment and the other end remains attached to a pad as end point of the connection. This helps you determine where you are going as you route.

On nets with more than one connection, the end point of the connection automatically snaps to the nearest pin of the net. Experiment with this reconnect feature.

6. Zoom out and move the pointer in the direction of the lower left corner of the board. Once the end point of the trace segment is closer to U7, it detaches from its current end point and snaps to U7.
7. Move the pointer toward the upper left and the trace segment snaps back. This feature lets you reconnect traces on-the-fly while routing, avoiding the need to manually reorder connections.

Tip: To exit routing commands at any time, press Esc or click the Undo button on the standard toolbar to undo any actions.

Change the trace angle mode

New segments are constrained to 90-degree increments from the origin of the trace segment because the current trace angle mode is Orthogonal.

To change the trace angle mode:

- Start routing the connection again. While the trace is attached to the pointer, right-click, point to **Angle Mode**, and click **Diagonal**.

Once you change the angle mode, move the pointer around. Notice how new segments are now constrained to 45-degree increments.

Add new corners

While in the route mode, with the end of a new segment attached to the pointer, move the pointer and click to add a new corner.

Delete new corners

To remove new corners:

- Right-click and click **Backup**.

  Alternative: Press Backspace.

Change layer

You can change layers the same way you add corners, except Shift+click. You can initiate layer changes at the current pointer location or at the last corner location while routing.

To initiate a layer change at the current pointer location:

- With a new route segment attached to the pointer Shift+click where you want to change layers.

  Result: A new via is added at the location of the click. The second layer of the routing layer pair is now the current layer.

To initiate a layer change at the last corner location:

1. Click to add a new corner.
2. Move the pointer away from the new corner, right-click and click **Layer Toggle**.

   Alternative: Press F4.

   Result: A via is added to the last corner added.

Restriction: Vias are not added when On-line DRC is activated and there are obstructions to safe placement of the vias.

Ending traces

You can partially complete or end a trace in almost the same manner as adding corners and vias:

- **Ctrl+click** to end a route.

Requirement: While routing a segment, right-click, point to End Via Mode and click End No Via. If the End Via Mode is set to End Via, Ctrl+click will end a route with a via instead.

Completing traces

You can complete a route in two ways.

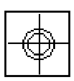
To complete a route using the Complete command:

- With a new trace segment attached to the pointer, right-click, and click **Complete**.

  Alternative: Double-click quickly to complete the route.

  Result: The trace is completed from its start to its destination.

To complete a route without using the Complete command:



1. With a trace segment attached to the pointer, define a route pattern from its start point to its end point, and click the mouse button when the complete symbol appears (at the center of its destination pin).

   Result: The trace is completed, in many instances without smoothing or cleaning up the route.

2. Practice routing the short connection of the net 24MHz to completion. Click the **Undo** button to remove previously made routing connections.

Deleting traces and trace segments

1. With nothing selected, right-click, and click **Select Anything**.
2. Click a segment of a completed trace connection and press **Delete**.

3. On the standard toolbar, click the **Undo** button to undo the deletion.
4. Shift+click to select the whole pin pair.
5. Press **Delete** to unroute the pin pair.
6. Click **Undo**. Traces are required for the next steps.

Modifying traces

Modify traces by selecting a trace segment or via and then using the commands on the shortcut menu. You have more freedom to edit routes in DRC Off mode, but you can perform route editing in any DRC mode.

Experiment with the route editing commands by selecting various route segments, vias, and corners. Use the shortcut menus or the keyboard shortcuts to initiate the move, stretch, split, add corner, add via, or other editing commands.

See also: *The PADS Layout Help*

Rerouting traces

Another feature of the route editing tools is the ability to change trace patterns using the trace creation commands. This is referred to as a reroute.

1. Select any trace segment.
2. Right-click and click **Route**.
3. To create a new trace pattern and complete it, double-click on any other point in the trace or trace segment.

   To complete it on component pins and vias, use the previously mentioned trace completion commands during reroute.

Tip: When you attempt to complete the new segment on another point of the trace, the first click of the double-click action attaches the new pattern to the existing pattern. The second click completes the reroute. For best results, minimize mouse movements while double-clicking.

Copying traces

You can duplicate traces and trace segments to speed repetitive tasks by copying and placing previously created traces.

1. Select the first segment of the trace you want to duplicate.
2. Shift+click the ending segment to select a range of traces.
2. On the Edit menu, click **Copy**. A copy of the trace attaches to the pointer.

   Alternative: Press Ctrl+C.
3. Zoom out. With a trace attached to the pointer, move the pointer over a pin to which a connection is attached.
4. Click the pin. A copy of the trace attaches to the pin and another copy remains attached to the pointer until you right-click and click **Cancel**, or you press **Esc**.

Tip: To orient the trace on the pointer during copying, right-click and use the commands on the shortcut menu.

Deleting traces

At any point in this exercise, you can delete all of the current traces by selecting all of the nets and then pressing Delete.

1. With nothing selected, right-click and click **Select Nets**.
2. Right-click and click **Select All** to select all nets.
3. Press **Delete** and click **No** when the message "Do you want to undo this action?" appears.

Routing plane nets

For a typical PCB with embedded planes and surface mounted parts, plane net routing is limited to routing a small segment out of the pad and terminating it with a via to provide contact with the internal plane.

Before you begin routing plane nets, you need to update the visibility of the plane nets.

Update net visibility

During the placement stages of this design, the display of certain plane nets was disabled to get a clear view of the components during placement.

Before you can route the plane nets, you need to re-enable the display of those nets.

1. **View** menu > **Nets**.
2. In the View Nets dialog box, select net **GND** from the View list.
3. To enable the display of the GND net, select the **Traces plus the Following Unroutes** check box.
4. To limit the display of the nets to only the routed portions of the nets, click **All Except Connected Plane Nets**.
5. In the View list, press **Ctrl** and click to select the **Default**, **+5V**, and **+12V** nets.
6. To disable the display of these nets, clear the **Traces plus the Following Unroutes** check box.
7. Click **OK**.

Perform a length minimization

Press Ctrl+B to bring the whole PCB in view. The display of the plane nets reveals that the net lengths of the plane nets were not minimized.

- On the Tools menu, click **Length Minimization**.

Route end modes

End modes help you avoid repeatedly using a layer change and end command to make every plane connection (you can set a route end mode that always ends routes with a via). The three modes are:

End Via Mode	Description

End No Via	Routes end *without* a via at the end point of the route.
End Via	Routes end *with* a via at the end point of the route.
End Test Point	Same as End Via but the via is also a test point.

To change the route end mode:

1. Right-click and click **Select Traces/Pins/Unroutes**.
2. Start routing a connection, and with a route segment attached to the pointer, right-click, point to **End Via Mode**, and click **End Via** as the new mode.
3. Find a location for the via, and Ctrl+click to create a pin escape to the plane.

Route the plane nets

Using the routing commands from the previous exercise, experiment with routing net GND using Ctrl+click to end the routes. Note the presence of the plane thermal indicator (an X on the via) each time you end a portion of the GND net. This symbol indicates eligibility for a thermal relief; it will make contact with the plane.

Using on-line design rule checking (DRC)

You can enable real-time design rule checking during placement and routing operations to ensure that design constraints are maintained throughout the design process. This interactive checking is called DRC. You set DRC modes in the Options dialog box or by using the dr modeless commands.

There are four modes of DRC operation:

DRC Mode	Description
DRC Off	Turns off design rule checking. Modeless command: dro.
DRC Ign Clr	Ignores all clearance rules but prevents the intersection of traces during routing. Modeless command: dri.
DRC Warn	Generates error messages for violations. Modeless command: drw.
DRC Prevent	Prevents you from creating violations. Modeless command: drp.

Update net visibility

Before you can experiment with on-line DRC, you need to update the display of some nets.

1. **View** menu > **Nets**.

2. In the View Nets dialog box, in the View List, select the **Default**, **+5V**, and **+12V** nets.
3. To enable the display of these nets, select the **Traces plus the Following Unroutes** check box.
4. To limit the display of the nets to the routed portions, click **All Except Connected Plane Nets**.
5. Click **OK**.

Experiment with routing in DRC Prevent mode

To enable the DRC Prevent mode:

- Type **drp** and press **Enter**.

Continue routing traces. Note the presence of the DRC violation indicator – the octagonal Guard Band – when you attempt to create spacing violations.

You can also experiment with the Complete command in DRC prevent mode by double-clicking to complete traces.

Routing with keepouts and cutouts

The components and the design itself contain keepouts. In addition, a board cutout was added to the PCB. With DRC prevent enabled, you can't violate these areas during route editing.

Set display colors for keepouts

Previously, you made keepouts invisible. For the following exercise, turn on the display of the keepouts.

1. Open the file named **previewpreroute.pcb** in the \PADS Projects\Samples folder. Don't save changes to the previous design.
2. **Setup** menu > **Display Colors**.
3. In the Display Colors Setup dialog box, click **yellow** and click **Keepouts** on the Primary Component Side.
4. Click **light blue** and click **Keepouts** on the Secondary Component Side.
5. Click **OK**.

Via keepouts

The J1 connector decal contains a via keepout. Try to add a via under J1.

1. If Prevent is not the current DRC mode, type the modeless command **drp** and press **Enter**.
2. To search for J1 pin 25, type **sj1.25** and press **Enter**.

Result: The pointer is positioned over the pin. The view may change depending upon your current zoom level and the center of your current view.

3. Click the connection to J1.25 and start routing the connection by right-clicking and clicking **Route**.
4. Move your pointer to the left and try to press Shift+click to add a via in the keepout outline of J1.

   Result: You cannot insert a via.
5. Move the trace outside and away from the keepout outline and try to insert the via again.

   Result: The via is permitted outside of the keepout.
6. Press **Esc** to end the Add Route mode.

Routing keepouts and board cutouts

The design contains keepouts and a board cutout to accommodate the shield on the bottom side of the PCB. Try to add routes across the keepout and under the keepout area on the bottom of the PCB.

1. With DRC in prevent mode, , to search for U4 pin 9 type **su4.9** and press **Enter**.
2. Click the connection to U4 pin 9 and start routing the connection by right-clicking and clicking **Route**.
3. Move your pointer to the right until it crosses the board cutout.

   Result: The DRC Guard Band appears and you are prevented from adding a trace across the cutout.
4. With the new trace attached to the pointer, create a trace path up and around the top of the cutout.
5. Add a via, continue routing, and attempt to add a trace in the keepout area to the right of the cutout.

   Result: The DRC Guard Band appears and you are prevented from adding a trace in the keepout area on the Secondary Component Side.
6. Do not save a copy of the design.

You completed the creating and editing traces tutorial.

Creating Traces with Dynamic Route

The Dynamic Route Editor (DRE) is another powerful interactive routing feature. Instead of indicating each trace corner, as with the Manual Route Editor, you simply start the trace and move the pointer in the direction in which you want the trace to flow. Trace corners are dynamically added as you move the pointer.

In addition, the Dynamic Route Editor also offers dynamic bus routing. Use bus routing to simultaneously route groups of nets such as address and data busses.

In this lesson:

- Using the Dynamic Route Editor
- Rerouting with the Dynamic Route Editor
- Autorouting with the Dynamic Route Editor
- Using the bus routing features of the Dynamic Route Editor

Restriction

- This tutorial requires the Dynamic Route Editing and General Editing licensing options.

To determine whether you can proceed:

- On the Help menu, click **Installed Options**.

Preparation

If it is not already running, start PADS Layout and open the file named **previewpreroute.pcb** in the \PADS Projects\Samples folder.

Using the Dynamic Route Editor

The core feature of the dynamic route editor is the dynamic route command. You use it to convert connections to traces quickly and without DRC errors.

Enable DRC

In order to use the dynamic route editor you must be in DRC prevent mode.

- Type the modeless command **drp** and press **Enter**.

Resize the view

1. Highlight the net 24MHz using the highlight net modeless command. Type **n 24mhz** and press **Enter**.

2. Zoom into the lower left corner of the design, centering the view around the longest connection of the net 24MHz (bottom of the net).
3. To unhighlight net 24MHz, type **n** (without a netname) and press **Enter**.

Route using the Dynamic Route Editor

1. With nothing selected, right-click and click **Select Unroutes/Pins**.
2. To set the angle mode to Orthogonal, type **ao** and press **Enter**.
3. Click net 24MHz at a point close to U7 (bottom of the view), right-click, and click **Dynamic Route**.

   Result: A trace dynamically attaches to the pointer.
4. Move the pointer vertically on the board, in the general direction of the destination of the connection. Notice how dynamic route automatically chooses a path around obstacles and creates a trace pattern for you.

   Tip: To back up a route while using Dynamic Route, slowly trace back over the newly created trace pattern.
5. Zoom out. Experiment with dynamically routing by moving the pointer, with the route attached, around obstacles, move your pointer closer to the destination of the connection and double-click. Notice how the Dynamic Route Editor automatically completes the trace and smoothes the pattern.

Tip: At any point in this exercise you can exit Dynamic Route mode by pressing Esc.

Many of the commands that apply to manual routing also apply when using dynamic routing. Backspace removes the last corner added; Shift+click inserts a via at the current pointer location causing a level change; Ctrl+click ends the route at the current location with or without a via.

Experiment with routing traces and using the commands while routing.

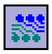
Rerouting with the Dynamic Route Editor

You can also use the Dynamic Route Editor to reroute in the same manner as the manual route editor.

1. Right-click and click **Select Anything**.
2. Click any trace segment, right-click and click **Dynamic Route**.
3. To create a new trace pattern and complete it, double-click on any other point in the trace or trace segment. Otherwise, use normal trace completion commands when completing on component pins and vias during reroute.

Autorouting with the Dynamic Route Editor

You can also autoroute connections on a single layer using features of Dynamic Autorouting. The Auto Route command completes routes that do not require a layer change.



1. Design Toolbar > **Auto Route** button.
2. Zoom in around the area of the short unrouted connections between J1 and U1 (right half of J1).
3. Click one of the short connections between J1 and U1. Note how the connection automatically completes for you.
4. Experiment with Auto Route on the other short connections and the memory bus connections between U1 and U2.

Using the bus routing features of the Dynamic Route Editor

The interactive routing capabilities let you selectively route all net members of large multi-bit data buses simultaneously by compacting the trace patterns into multi-trace bus patterns while maintaining all design rules. This results in reduced design time and optimal space usage on the PCB. Bus routing uses the DRE option, where traces and vias are moved aside to make room for new routing without creating clearance violations.

Before you can initiate a bus route on the tutorial design, you must perform a few preparatory steps.

Resize the view

Use the Find dialog box to highlight nets and resize the view around the nets to route.

1. **View** menu > **Extents**.

2. On the Design Toolbar, click the **Select** button to exit Auto Route mode.
2. Right-click and click **Find**.
3. In the Find dialog box, select **Nets** from the Find By list.
4. In the Action list, select **Highlight**.
5. Scroll through the Nets list, and Shift+click to select the **$$$5799**, **$$$5801**, and **$$$5803** nets.
6. Click **Apply** to highlight the nets. The selected nets are highlighted on the design.
7. Use the zoom commands to resize the view so the three highlighted nets fill most of the view.

8. With the $$$5799, $$$5801, and $$$5803 nets still selected in the Find dialog box, select **Unhighlight** from the Action list, and click **OK** to unhighlight the nets and close the Find dialog box.

Set a routing grid and routing angle

Use the modeless commands to set a routing grid.

1. For the purpose of this exercise, set the routing grid and via grid to 25. Type **g25** and press **Enter**.
2. To set the angle mode to diagonal, type **ad** and press **Enter**.

Start a bus route

2. **Bus Route** button.
3. Right-click and click **Select Pins/Vias/Tacks** to limit your selections, as necessary, for bus routing.
4. Area select the three pins of U2 (the large SOIC - lower left pins) connected to the nets.

   Result: The interactive bus routing mode is now activated. Initially it appears as if you are dynamic routing a single connection, except multiple nets are selected. The route currently attached to the pointer is the *guide route*. Each time you add a trace corner or via to this route, the other members of the bus will follow the guide route.

   Tip: At any point in this exercise you can exit the routing command operation by pressing Esc.
5. Moving in an upward direction, add a vertical segment a short distance from the pin, and click to add a corner to the guide route.

   Result: The other members of the bus catch up to that corner of the guide route.
6. Move the pointer to a point just below the destination pin of the guide route and add another corner.

   Result: The other members catch up to the guide route once again.
7. To complete the bus, right-click and click **Complete**.

   Result: All members of the bus are completed and smoothed.

Bus routing with via patterns

Bus routing can also automatically insert vias in typical via patterns. Whenever you add a via to the guide route, the remaining members of the bus are also routed to vias.

1. On the Standard toolbar, click the **Undo** button to undo the bus routes.
2. Area select the three pins of U2 once again.
3. Moving in an upward direction, add a vertical segment a short distance from the pin, then Shift+click to add a via to the guide route.

Result: A via is added to the guide route. Also, the other members of the bus catch up and create a repetitive via pattern.

4. Press **Ctrl+Tab** repeatedly to cycle through each of the possible via patterns.
5. When you cycle to a perpendicular via pattern (three vias arranged horizontally), continue routing the bus.
6. Place the pointer under the destination pin leaving enough room for a via, and Shift+click to add vias to the bus.

If a via for one of the bus members cannot fit into the pattern, the bus router pauses and allows you to cycle the via pattern.

1. Press **Ctrl+Tab** to cycle the via pattern.

   Result: Once all bus members have a via, you return to bus routing mode and the current route becomes the route guide.
2. Right-click and click **Complete** to complete the bus.
3. Do not save a copy of the design.

You completed the dynamic route editor tutorial.

Autorouting with PADS Router

With the PADS Router Link you can set up autorouting options from PADS Layout and then send the design to PADS Router for autorouting. When autorouting completes, PADS Router closes and a new session of PADS Layout opens containing a routed version of your file.

The link gives you the option of automatically opening the current design file in PADS Router to view routing progress or to autoroute in the background – to speed up the process.

In this lesson:

- Preparing the design for autorouting
- Transferring the design to PADS Router for autorouting

Restrictions

This tutorial requires the General Editing and Split Planes licensing options.

To determine whether you can proceed:

- On the Help menu, click **Installed Options**.

Preparation

If it is not already running, start PADS Layout and open the file named **previewpreroute.pcb** in the \PADS Projects\Samples folder.

Prepare the design for autorouting

Before you begin autorouting, you must set up your design file for proper autorouting in PADS Router.

Update net visibility

In the placement tutorial, the display of certain plane nets is disabled for a clear view of how the components are interconnected during placement. Before you can route the plane nets, you need to enable the display of those nets.

1. **View** menu > **Nets**.
2. In the View list, select **+5V**, **+12V**, **GND**.
3. In the View Details area, select the **Traces Plus the Following Unroutes** check box to enable the display of these nets.
4. Click **All Except Connected Plane Nets** to limit the display of the nets to only the routed portions of the nets.

5. Turn off the color for net **+5V** by selecting it and clicking **None** in the Colors by Net area.
6. Click **OK**.

Perform a length minimization of net connections

Displaying the plane nets reveals that the net lengths were not minimized.

- **Tools** menu > **Length Minimization**

  Alternative: Press Ctrl+M.

Transferring the design to PADS Router for autorouting

The process of autorouting PCB designs in PADS Router begins with the PADS Router Link.

The design file used in this tutorial has a Conditional Rule that requires the Advanced Rules licensing option, however you can delete the conditional rule in the design file once it is open in PADS Router and proceed with the tutorial. Instructions for this workaround are included below.

Start the PADS Router Link and assign link options

1. **Tools** menu > **PADS Router**.
2. In the PADS Router Link dialog box, click **Autoroute in Foreground** in the Action area. This allows you to watch the autorouting process.
3. In the Options area, click **Grid** and then click **Setup**.
4. On the Grids page of the Options dialog box, turn off the **Snap to Grid** check boxes for the Design, Via and Fanout grids.
5. Click **OK**.
6. When you are ready to begin autorouting in PADS Router, click **Proceed**. In a few moments, a session of PADS Router appears and it automatically starts autorouting your design.

   Exception: The process stalls at this point if you don't have the Advanced Rules licensing option. The design file opens in PADS Router, with the following message on-screen, "You are not licensed to route with the Advanced Rules in your design. You must remove the rules or obtain an Advanced Rules license." Follow these steps to remove the conditional rules and continue the process:

   a. In the message box, click **OK**.
   b. On the **View** menu, ensure the **Project Explorer** is turned on.
   c. In the Project Explorer window, expand the **Net Objects** branch.
   d. In the Net Objects branch, expand the **Conditional Rules** branch.

e. In the Conditional Rules branch, right-click on the conditional rule "**+12V: +5V (All layers)**" and click **Delete**.

f. On the **Tools** menu, point to **Autoroute** and on the submenu, click **Start**.

Tip: If you are running the PADS Router Link for the first time, the PADS Router Monitor appears after you click Proceed. See the next section for more information.

If you ran the PADS Router Link previously, the PADS Router Monitor remains on your Windows Taskbar after you click Proceed. Click the PADS Router Monitor button on the Taskbar to open the PADS Router Monitor dialog box.

The PADS Router Monitor

Use the PADS Router Monitor to track autorouting progress and set options for post-routing.

1. If PADS Router already finished your design, close PADS Router and the second PADS Layout window containing the autorouted design. Use the PADS Router Link again and immediately proceed with the following steps.
2. While PADS Router is routing your design, click the PADS Router Monitor button on the Taskbar to open the PADS Router Monitor dialog box.
3. Click **Close When Finished** and make sure **Load the Resulting File** and **Exit PADS Router on Completion** are also selected.
4. Wait until PADS Router is done autorouting.

   When PADS Router finishes autorouting, the PADS Router Monitor dialog box and the PADS Router session closes. The routed file appears in a new session of PADS Layout.

Result: After PADS Router completes autorouting, the PADS Router Link returns the file to a new PADS Layout session. At this point, you should close the original PADS Layout session to avoid possible confusion due to duplicate instances of PADS Layout.

5. Do not save a copy of the design.

You completed the PADS Router Link tutorial.

Creating Split Planes

PADS Layout provides automated tools to help you define plane separations and divisions quickly. To facilitate the definition of split planes, you can differentiate plane nets from other nets by assigning them different colors. You create separate plane areas by drawing the lines of separation.

In this lesson:

- Setting layer display and plane options
- Creating a plane area
- Connecting the plane areas

Restriction

This tutorial requires the Split Planes licensing option.

To determine whether you can proceed:

- On the Help menu, click **Installed Options**.

Preparation

If it is not already running, start PADS Layout and open the file named **previewrouted.pcb** in the \PADS Projects\Samples folder.

Setting layer display and plane options

Before you split the plane layer you must perform a few preparatory steps.

Assign Colors to the Nets

To make it easier to distinguish the two power nets, assign different net colors.

1. **View** menu > **Nets**.
2. From View List, select the net **+5V**, and then click **dark gray** from the Color by Net area.
3. Select the net **+12V** from View List, and then click **yellow** from the Color by Net area.
4. Click **OK**.

Set layer display

To facilitate the process of defining and splitting the plane, turn off all irrelevant layer data.

1. **Setup** menu > **Display Colors**.

2. Select the **Power Plane** layer check box to enable the display of the Power Plane.
3. Clear the **Primary** and **Secondary Component Side** check boxes to disable the display of both Component layers.
4. Click **OK**.

Setting split plane options

Before defining the plane area, set the split plane options.

1. **Tools** menu > **Options** > **Split/Mixed Plane** page.
2. In the Mixed Plane Display area, click **Plane Thermal Indicators**.
3. In the Automatic Actions area, select all of the check boxes except the second last option: *Use design rules for thermals and antipads*.
4. Click **OK**.

Creating a plane area

A split/mixed plane must have a plane area defined before you can divide the plane layer into separate areas for each net. There is an automated feature to create a plane area by tracing around the board outline.

Define the plane area

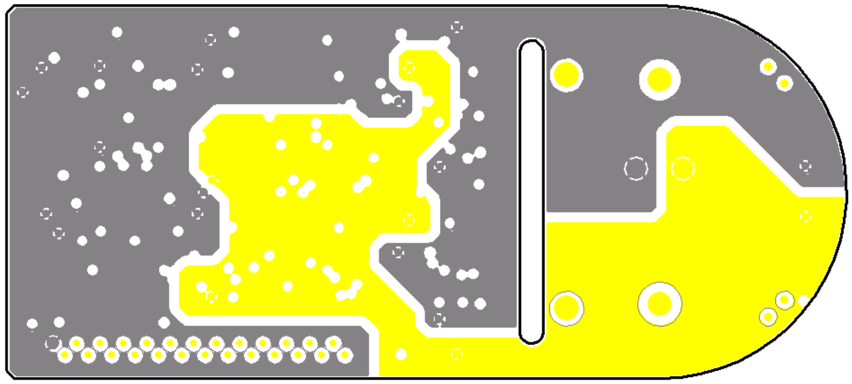
To quickly create a plane area of the entire board, reuse the shape of the board outline for the plane outline.

1. On the Standard toolbar, in the Layers list select **Power Plane**, to make it the current layer.
2. With nothing selected, right-click and click **Select Board Outline**.
3. Click any segment of the board outline, right-click, and click **Select Shape**.
4. Right-click and click **Create Plane Area**.
5. The majority of the plane area will be for the +5V net. In the Net assignment area, select net **+5V** from the list and click **OK**.

   Result: A +5V plane area outline is created just inside the board edge.
6. Press **Esc** to deselect the Board Outline.

Define the plane separation

Once you define a plane area polygon, use the Auto Separate command to define the separation between the two plane areas, similar to what is shown below.



To split the plane:

1. Click outside the board outline to ensure nothing is selected.

2. If the Drafting toolbar is not currently open, click the **Drafting** button on the Standard toolbar.

3. On the Drafting toolbar, click the **Auto Plane Separate** button.
4. To set a design grid of 25, type **g25** and press **Enter**. Select an edge of plane +5V to begin defining the plane separation.
5. Right-click and click **Diagonal**.
6. Define a path that separates all points of the net +12V from net +5V. There is no definitive shape for the +12V plane area, just make certain the area you define includes all +12V pins and no gray colored +5V pins.

   Tip: Use Backspace to delete new corners or to back up while defining the plane separation.
7. When you complete the separation, position your pointer over an edge segment of the +5V plane polygon and double-click to complete the operation. Two new plane polygons appear.

   Tip: If you get the error, “All corners must be within the plane area polygon”, try extending the line outside the plane area and double-click to complete.
8. In the Assign Net to Selected Polygon dialog box, if the upper outline is selected, select **+5V** from the Choose Existing Net list and click **OK**. If the lower outline is selected, select **+12V**.
9. When the adjacent outline is selected, select the other netname from step 8 and click **OK**.
10. Click **Esc** to exit Auto Plane Separate mode.
11. Press **Ctrl+B** to bring the whole board in view.

Connecting the plane areas

Now that you defined the planes, you must connect them to see the full effect of the split.

1. **Tools** menu > **Pour Manager** > **Plane Connect** tab.
2. Select **Power Plane** in the Layers list.
3. Click **Start** and **Yes** to proceed with connecting the planes.

   Tip: A therm.err report might open if some thermal connections can't be made based on how the separation line was drawn. Close it.
4. In the Pour Manager, click **Close**.
5. Do not save a copy of the design.

You completed the split planes tutorial.

Creating Copper and Pour Areas

Using copper pouring and area filling in PADS Layout, you can quickly produce and edit insulated copper areas for shielding, power and ground supply, and thermal dissipation.

In this lesson:

- Creating the copper pour outline
- Flooding the pour outline
- Toggling pour and hatch outlines
- Editing the copper pour hatch

Restriction

This tutorial requires the Copper Flood, Drafting Editing and General Editing licensing options.

To determine whether you can proceed:

- On the Help menu, click **Installed Options**.

Preparation

If it is not already running, start PADS Layout and open the file named **previewsplit.pcb** in the \PADS Projects\Samples folder.

Set Layer Display

To facilitate the process of defining the pour outline, turn off all irrelevant layer data.

1. **Setup** menu > **Display Colors**.
2. In the Display Colors Setup dialog box, clear the **Ground** and **Power Plane** check boxes to disable the display of both Plane layers.
3. Click **OK**.

Creating the copper pour outline

A pour outline defines the boundaries of the copper pour area. You use the Flood command to fill the area defined by the pour outline with copper hatching. After flooding a pour outline, its pour outline is temporarily invisible. The pour outline and the copper hatching are not displayed at the same time. In the following exercise, you will define and flood a pour outline.

Set the grid and the current layer

1. To set the design grid to 25, type **g25** and press **Enter**.
2. On the standard toolbar, in the Layer list, select **Primary Component Side** to set it as the current layer.

Define the pour outline

1. If the Drafting Toolbar is not open, click the **Drafting Toolbar** button.

2. **Copper Pour** button.
3. Right-click and click **Rectangle**.
4. Position the pointer at **X2500**,**Y1875** and click to start drawing the rectangle.

   Tip: If the pointer increments aren't 25, the pointer isn't snapping to grid. On the Tools menu, click Options. Under Grids and Snap, click Grids. For the Design Grid select the Snap to Grid check box. Click OK to apply the change and close the Options dialog box.
5. Move the pointer toward the lower right corner of the board and click again at **X3000**,**Y325** to complete the rectangle.

Assign a width and net

1. In the Add Drafting dialog box, in the Width box, type **12**.
4. In the Net assignment area, select **GND** from the list, and then click **OK** to apply the changes and close the dialog box.

Modify the pour outline

1. Press **Esc** to exit Copper Pour mode.
2. Press **Esc** again to deselect the pour outline.
3. Right-click and click **Select Documentation**.
4. Select the right edge of the rectangle you just created.
5. Right-click and click **Pull Arc**.
6. Pull the right edge of the pour outline into an arc that is slightly inside, and concentric with, the arc in the board outline.
7. Click to complete the arc.

Flooding the pour outline

Now you can flood the pour outline. The flooding operation creates a hatching pattern that fills the area enclosed in the pour outline.

1. Right-click and click Select Shape to select the whole pour outline.
2. Right-click and click **Flood**.

3. Click **Yes** to the "Proceed With Flood?" message.

After flooding the pour outline, you can see the area of poured copper. In addition, an error report may appear detailing the pins or vias having two or less spokes. If so, review the error report and close the window before proceeding.

Toggling pour and hatch outlines

When you flood the pour outline, the pour outline is replaced with hatch outlines and polygons. The hatch outline is the shape of the polygon, and the hatch pattern fills the shape.

The pour outline is still present in the design; you just can't see it until you toggle the display from hatch outline to pour outline.

1. Type **po** and press **Enter**. This command switches between displaying the pour outline and the poured copper hatch inside it.
2. Type **po** and press **Enter** again to revert to the display of the hatch outline.

You completed the copper pour tutorial.

Adding Dimensioning

PADS Layout provides you with tools to document the physical size and shape of your PCB. You can choose between standard or datum dimensioning, and you have complete control over the format of the dimension to help you conform to corporate or industry standards.

You access these tools within the Automatic Dimensioning toolbar.

In this lesson:

- Setting the dimensioning options
- Adding a horizontal dimension
- Adding a vertical dimension
- Adding a leader dimension
- Adding an arc dimension
- Dimensioning the cutout

Restriction

- This tutorial requires the Drafting Editing licensing option.

To determine whether you can proceed:

- On the Help menu, click **Installed Options**.

Preparation

If it is not already running, start PADS Layout and open the file named **previewpour.pcb** in the \PADS Projects\Samples folder.

Change the view and color display

Before you begin the process of setting design options and adding dimensions, you must make a few adjustments to the zoom level and color display.

1. Change the zoom level so that the board outline fills three quarters of the viewing window, leaving enough working space around the PCB to add the dimensions.
2. On the **Setup** menu, click **Display Colors**.
3. In the Display Colors Setup dialog box, clear the Primary and Secondary Component Side layer check boxes.
4. Scroll through the layers and on the Drill Drawing Layer (layer 24), assign the color **red** to both **2D Lines** and **Text**.
5. Click **OK**.

Setting the dimensioning options

Before you begin adding dimensions, you must assign a few options.

Setting the units

The current unit setting establishes the units expressed in the dimensions. If the current units are set to inches, then all dimensions are expressed in inches.

Set the current units to Inches.

1. **Tools** menu > **Options** > **Global** > **General**.
2. Click **Inches** in the Design Units area.

Assigning a layer for the dimension items

You can choose the layer assignment for the text and line item portions of the dimension.

1. **Dimensioning** > **General**.
2. In the Layers area, click the **Drill Drawing** layer for both **Text** and **Lines**.

Assigning text properties

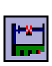
You can also assign options for how the numbered text appears in the dimension.

1. **Dimensioning** > **Text**.
2. Delete the Inches suffix by double-clicking in the Suffix box and pressing **Delete**.
4. Click **Horizontal** as the Default Orientation.
5. Click **Inside** as the Default Position.
6. In the Precision area, type **2** in the Linear box, and type **0** in the Angular box.
7. In the Displacement area, click **Centered**.
8. Click **OK**.

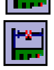
Add a horizontal dimension

Once you complete the assignment of the options, you can add dimensions to the design. First, add a dimension for the horizontal length of the PCB.

1. Right-click and click **Select Board Outline**.



2. **Dimensioning Toolbar** button.



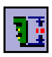
3. On the Dimensioning toolbar, click the **Horizontal** button.
4. Right-click and click **Snap to Corner**. A check mark indicates the active mode.

5. Click the left vertical segment of the board outline polygon. An alignment marker appears.
6. Right-click and click **Snap to Midpoint**.
7. Select a point along the arc on the right side of the board outline. A new dimension is created and is attached to the pointer.
8. Place the new dimension above the board outline.

Tip: Click the Redraw button on the standard toolbar to clear the alignment markers from your view.

Add a vertical dimension

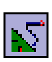
Now add a dimension for the vertical length of the PCB.



1. **Vertical** button.
2. Right-click and click **Snap to Corner**.
3. Select the top Horizontal segment of the board outline near the left side of the board. Select the bottom horizontal segment of the board outline near the left side of the board. A new dimension is created and is attached to the pointer.
4. Place the new dimension to the left of the board outline. To redraw the view press **Ctrl+D**.

Add a leader dimension

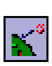
The two corners on the left edge of the board outline are mitered. Add a leader dimension to call out the details of the chamfers.



1. **Leader** button.
2. Select a point near the center of the chamfer at the bottom left of the board outline. A new leader dimension attaches to the pointer.
3. Move the pointer down and to the left, and double-click to complete the leader line.
4. In the Text Value dialog box, type **.035 x .035 2 PLACES** and click **OK**.

Add an arc dimension

To define the size of the arc on the right side of the PCB, use a radial dimension.



1. **Arc** button.
2. Select a point along the arc of the board outline. A new dimension attaches to the pointer.

3. Click to place the arc dimension.

Tip: To remove leading or trailing zeros from the dimension text, enter Select mode (press Esc). Right-click and click Select Documentation. Double-click on the dimension text and modify the text string in the Dimension Text Properties dialog box.

Dimensioning the cutout

Experiment with the dimensioning features to define the size and location of the cutout in the PCB. Use the routines in this lesson to dimension the cutout.

See also: *PADS Layout Help*

- Do not save a copy of the design.

You completed the automatic dimensioning tutorial.

Checking for Design Rule Violations

The Verify Design command lets you check your design for clearance, connectivity, high speed, and plane errors. This advanced space checking is quick and accurate to .00001".

You can check the Clearance rules and perform Connectivity checks to ensure that your design is fully routed. High-speed Checking searches for electrodynamic violations to avoid problems with critical signal and high-speed designs. Plane net checks verify that thermals are generated on plane layers.

You can modify traces to intentionally create some spacing errors, and then try the checking and reports routines.

In this lesson:

- Performing a design rule clearance check
- Viewing spacing errors
- Checking PADS Router-specific rules

Restriction

This tutorial requires the Verify Design licensing option.

To determine whether you can proceed:

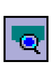
- On the Help menu, click **Installed Options**.

Preparation

If it is not already running, start PADS Layout and open the file named **previewdim.pcb** in the \PADS Projects\Samples folder.

Performing a design rule clearance check

The Verify Design tools in PADS Layout scan the design database looking for all design rule violations. If errors are detected, they are identified with a graphic error symbol and itemized in the Verify Design dialog box.



Tip: To check the entire board, you must display the entire board. On the standard toolbar, click the **Board** button.

1. **Tools** menu > **Verify Design**.
2. In the Verify Design dialog box, in the Check area, click **Clearance**.
3. Click **Setup**.

4. In the Clearance Checking Setup dialog box, make sure the **Net to All**, **Board Outline**, **Keepout**, **Drill to Drill** and **Body to body** check boxes are selected.
5. Click **OK** to accept the changes and close the Clearance Checking Setup dialog box.
6. Click **Start**.

   Result: The Status Bar indicates the layer being checked. A progress indicator displays the degree of checking. The previewdim.pcb design should contain two errors.

Viewing spacing errors

After the verify design operation has completed, a dialog box appears indicating errors found. View the marked error in the design.

1. In the notification window, click **OK** to view the errors. All violations detected are identified with a small graphic symbol. Different symbols are used to identify violations for each different check type.

   See also: The "Interpreting Error Markers" paragraph in the “Verifying the Design” topic in *PADS Layout help*
2. Leave the Verify Design dialog box open and zoom in to any area of the design.
3. Clear the **Disable Panning** check box to enable automatic panning of your view to the selected error.
4. In the Verify Design dialog box, select an error from the Location list.

   Result: The screen redraws, displaying the selected error in the center of the view. The Explanation list describes the error in detail.
5. Do not save a copy of the design.

Preparation

Open the file named **previewrouterverify.pcb** in the \PADS Projects\Samples folder.

Checking PADS Router-specific rules

You can also check PADS Router-specific advanced routing features while the design is open in PADS Layout. Set Verify Design to check the design with PADS Router by selecting the Latium Design Verification option.

1. **Tools** menu > **Verify Design**.
2. In the Check area, click **Latium Design Verification**, and click **Setup**.
3. In the Latium Checking Setup dialog box, select the items you want to check and click **OK**.

4. Click **Clear Errors** to remove existing error markers from the design.
5. Set the area to check by scaling the design in the PADS Layout workspace. Only the visible part of the design and only visible objects will be verified during Clearance checking. Other types of checking will verify objects in the whole design.
6. In the Verify Design dialog box, click **Start** to verify the design.

   Result: When you run Latium Design Verification, the following occurs:

   - The design is saved on disk under a temporary name.
   - A new instance of PADS Router starts in the background.
   - PADS Layout sets the necessary Design Verification options in PADS Router.
   - PADS Router saves the design, loads the design back into PADS Layout, and closes.
   - Verify Design reads error information from the design database, displays error markers in the workspace, and displays error descriptions in the Verify Design dialog box.
7. Review the errors and close the Verify Design dialog box when finished.
8. Do not save a copy of the design.

You completed the design verification tutorial.

Updating the Schematic

To facilitate the design engineering process, PADS Layout offers Engineering Change Order (ECO) features.

In this lesson:

- Generating ECO records
- Updating your PADS Logic schematic

Restriction

This tutorial requires the ECO licensing option.

To determine whether you can proceed:

- On the Help menu, click **Installed Options**.

Preparation

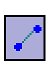
If it is not already running, start PADS Layout and open the file named **previewassy.pcb** in the \PADS Projects\Samples folder.

Generating ECO records

In the typical engineering environment, design modifications are documented by an Engineering Change Order (ECO). Implementing ECOs in a design may require pin and gate swaps, part deletions or additions, net deletions or additions, renaming components, renaming nets, or changing part types.

You can perform ECO changes and record them in an ASCII file named with a *.eco extension. This file, in its native format, can be read into PADS Logic for backward annotation of the schematic.

To enable ECO functions:



1. ECO Toolbar button.

   Tip: If the ECO Options dialog box does not appear, click the Options button on the ECO toolbar.

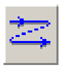
2. In the ECO Options dialog box, select the **Write ECO file** check box and clear the **Append to File** check box. Leave the file name as previewassy.eco.
3. Select the **Write ECO File after closing ECO Toolbar** check box.
4. Click **OK**.

Automatic component rename

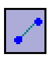
One feature of the ECO function is automated part renumbering. You do this by defining the components to renumber and specifying a numbering pattern.

1. **Auto Renumber** button.
2. In the Auto Renumber dialog box, if all the prefixes in the Prefix List are not already selected, click **Select All**.

   Tip: The Select All button changes to UnSelect All when everything is already selected.
3. Change the **Cell size:** On X **1**, On Y **1**.



4. Click the **Left to right and top to bottom** button in both **Top** and **Bottom** areas.
5. In the *Start Renumbering from* area, click **Top**.
6. Click **OK** to start automatic renumbering.



7. On the Standard toolbar, click the **ECO Toolbar** button to close the ECO toolbar and to write the changes to the ECO file.

Result: The parts are renumbered. A before and after list is created in standard PADS Layout ECO format in a file that is located in the \PADS Projects\Samples folder with the name previewassy.eco.

Updating your PADS Logic schematic

You can import the resulting ECO file with the renumbered parts directly into PADS Logic and instantly update your schematic.

Requirement: This section of the lesson requires PADS Logic. If you don't have PADS Logic, skip this section.

Preparation

If it is not already running, start PADS Logic and open the file named **preview.sch** in the \PADS Projects\Samples folder.

Import the ECO File

Examine a few of the reference designators in the schematic. Now import an .eco file.

1. PADS Logic **File** menu > **Import**.
2. Click **No** if the message "Save file before loading" appears.
3. In the File Type area of the dialog, change ASCII File (*.txt) to **ECO Files (*.eco)**.
4. Select the **previewassy.eco** file in the \PADS Projects\Samples folder, and then click **Open**.

5. In the “ECO input completed” notification window, click **OK**.

   Result: All reference designators on the schematic are updated and the schematic is redrawn.

6. Do not save a copy of the design.

You completed the reports and engineering change orders tutorial.

Creating Reports and CAM Files

Whether you want to print a quick rendering of the layout in progress or commit the final design to finished artwork, the Computer Aided Manufacturing (CAM) commands offer a complete selection of printing and plotting options.

In this lesson:

- Generating reports
- Creating a new CAM directory
- Creating CAM documents
- Plotting layer selections
- Setting up the device
- Saving and plotting CAM documents
- Creating multiple documents

Restriction

This tutorial requires the CAM, Copper Flood, Split Planes licensing options.

To determine whether you can proceed:

- On the Help menu, click **Installed Options**.

Preparation

If it is not already running, start PADS Layout and open the file named **previewassy.pcb** in the \PADS Projects\Samples folder.

Generating reports

Create a parts list report

1. **File** menu > **Reports**.
2. In the Reports dialog box, click the **Parts List 1** report and click **OK**. The new report appears in your default text editor. After reviewing the report, close the text editor.

Recommendation: More advanced preformatted reports and tools exist in the Basic Scripts interface. On the Tools menu, point to Basic Scripts and then click Basic Scripts on the submenu.

Creating a new CAM directory

All of the processing performed that results in files or CAM data is stored in a single project folder for each design. You can save and reuse it for the current design or copy it to other designs.

Flooding your design

The pour areas of your design, including the plane layers, appear unfilled because the fill data is not stored with the file. You can recreate it with a few clicks.

1. **Tools** menu > **Pour Manager** > **Flood** tab.
2. Click **Flood All**.
3. Click **Start** to connect all pour outlines and click **Yes** to confirm the connect operation.
4. Click the **Plane Connect** tab.
5. Click **Start** to flood all plane outlines and click **Yes** to confirm the flood operation.
6. Click **Close**.

Creating the CAM folder

In the CAM portion of the program, you define the type of plots, the output devices, and various data to create artwork or check plots for your design.

1. **File** menu > **CAM**.
2. In the Define CAM Documents dialog box, in the CAM Directory list, click **<Create>**.
3. In the CAM Question dialog box, type **preview** and click **OK**.

Result: The list reflects the subfolder name. You created a project folder in which to store all your CAM files. The new folder is located inside the CAM subfolder.

Creating CAM documents

Each output of a pen plot, photoplot, or NC Drill is considered a CAM document. Each CAM document definition contains all the data for the plot type, selected items, and other parameters.

1. Click **Add**.
2. In the Add Document dialog box, in the Document Name box, type **Photo - Primary Component Side**.
3. In the Document Type list, click **Routing/Split Plane**.
4. In the Layer Association dialog box, click **OK** to accept the Primary Component Layer.

   The Summary area displays the layer name and default selections. The next step is to configure the layers selections.
5. In the Output Device area, click **Photo** to assign a Gerber photoplot file as the output of this document.

Plotting layer selections

Add additional objects to the plot using the Layers feature.

1. In the Customize Document area, click **Layers**.
2. In the Select Items dialog box, in the Other area, select the **Board Outline** check box. A black box appears next to the Board Outline item.

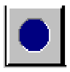
   Tip: Primary Component Side appears in the Selected area. The Items on Primary area is highlighted in the color black for selected pads, traces, lines, vias, copper, and text. These entries match the default output when processing routing layers.
3. Click **Preview** to preview your selections and see how they might appear on the photoplot.
4. In the Zoom area of the Preview window, click **Board** to fit the layer image to the view.

   Tip: Zooming in the Preview window works the same as zooming in a design; drag up to zoom in and drag down to zoom out. You must, however, use the left mouse button to zoom instead of using the middle mouse button. By using the Zoom buttons, you can also zoom to the board limits or the extents of the database to include items outside the board outline.
5. Click **Close**.
6. In the Select Items dialog box, click **OK**.

Setting up the device

For the final setup, determine the proper output device and the settings for that device.

1. In the Add Document dialog box, click **Device Setup**.
2. In the Photo Plotter Setup dialog box, click **Add**.
3. Type **10** (or some other number if 10 is used) when prompted for the Aperture Number.
4. Click **OK**.



5. In the Flashes area, click the **Round** button for a round aperture.
6. Type **.02** in the Width box.
7. Click **Augment** and answer **Yes** to automatically fill in the rest of the apertures required to process a photoplot of the Primary Component Side.

   Tip: If the Augment on-the-fly check box is selected, the required apertures are calculated at plot time and then added to the list automatically.
8. Click **OK**.

Saving and plotting CAM documents

Now that you selected all the items to include in the photoplot and defined the apertures to augment on-the-fly, you can process the photoplot.

1. Click **OK** to close the Add Document dialog box and to save the document settings.
2. In the Define CAM Documents dialog box, click **Run** to create the photoplot file.
3. In the prompt, answer **Yes**.

Result: The file art001.pho, which is the default name, is written to the \PADS Projects\CAM\preview folder that you created at the beginning of this section. An aperture report file called art001.rep, listing apertures used in the design, is also created and placed in that folder.

Creating multiple documents and the Aperture Report

The Document Name box displays a list of all created document names. It currently contains the document name of the file you created above, Photo - Primary Component Side. If additional documents exist, they appear here also. You can select multiple documents in the Document Name box and click Run to plot multiple documents.

1. Click **Aperture Report**. You are prompted to create a file called preview.rep.

2. Click **Save** to create a report containing a summary of photoplot apertures contained within the specified subfolder.

   Result: The report appears in the default text editor. Review the report and close the text editor before proceeding.

3. Click **Close** in the Define CAM Documents dialog box to exit CAM, and click **No** to decline saving the created file.
4. Do not save a copy of the design.

You completed the computer aided manufacturing (CAM) tutorial.