Lesson 2: Placing Components

What You Will Learn

This lesson teaches you how to use PCB Router's interactive and automatic placement tools.

In this lesson you will learn:

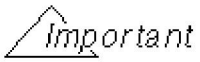
- the basic steps for placing components.
- how to set placement options.
- how to set placement rules.
- how to preplace connectors and other critical components.
- how to place large components.
- how to edit the placement.
- how to place small components.

This lesson takes about 60 minutes to complete.

What to do Before You Begin

Before you begin this lesson, do the following:

- Complete *Lesson 1: Learning Basic Concepts*.



A PlaceBase license is required to complete the tasks in this lesson. For details on PCB Router licensing, products and features, refer to Chapter 1 of the *Allegro PCB Router User Guide*.

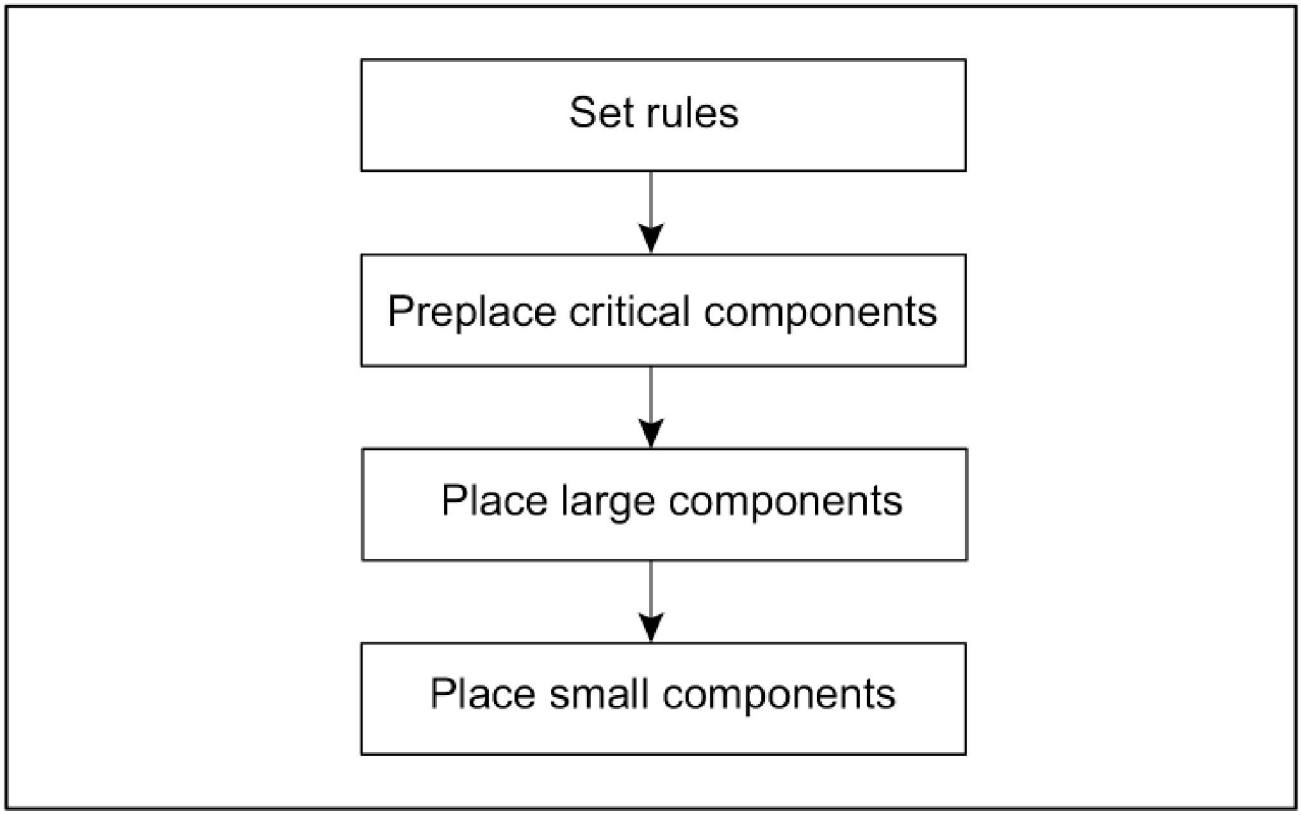
Understanding the Basic Steps for Placing Components

Placing components consists of four basic steps.

- Setting placement rules

- Preplacing critical components
- Placing large components
- Placing small components

Figure 2-1 Basic Steps of Component Placement



In the first step, you set placement rules. During both automatic and interactive placement, the design is checked for rule violations.

In the second step, connectors and other position-critical components are preplaced and locked in position using interactive placement tools.

In the third and fourth steps, components are placed and locations are optimized to reduce manhattan lengths and guide crossings. Large components are placed first and then small components.

Setting Placement Rules

PCB Router provides a comprehensive set of placement rules. Basic placement rules control the spacing between components, the orientation of components, and the sides on which the autorouter places components. More advanced rules control floor planning based on power dissipation, power supply, and component height. In this lesson, you will set basic rules. For more information on setting advanced placement rules, see the Placement online help.

Understanding the Rules Hierarchy

Placement rules can be specified at different levels and, consequently form a hierarchy. Rules at higher levels in the hierarchy override rules at lower levels that are set for the same physical objects. For

example, consider what happens if you set a global (PCB) spacing rule of .25 inches for all components and a spacing rule of .8 inches for a specific connector. The autorouter follows the .8 spacing rule only in the area surrounding the connector. The autorouter follows the .25 spacing rule in the areas surrounding other components. The component-level spacing rule overrides the PCB-level spacing rule.

The following table shows the levels of hierarchy in which you can set rules and the order of precedence for all placement rule levels. Global placement rules (PCB rules) have the lowest precedence, and image_image rules have the highest precedence.

Table 2-1 Rules Hierarchy

This rule level . . .	Does this . . .
image_image	Sets rules between images. An image is the footprint definition of a component. This is the highest precedence rule.
family_family	Sets rules between families. A family is a group of images.
room_image_set	Sets rules for an image_set assigned to a room.
room	Sets rules for a room, which is an area of the design in which the autorouter places specified components.
super cluster	Sets rules for a super cluster, which is a group of components that is treated as a single component.
component	Sets rules for a component, which is an instance of an image.
image	Sets rules for an image, which defines a component footprint. An image is the footprint definition of a component.
image_set	Sets rules for images with the same type property. The type properties are large, small, capacitor, or discrete.
pcb	Sets global rules for all components in the design. This is the lowest precedence rule.

Note: The order of precedence is fixed and cannot be changed.

Setting a Placement Grid and Spacing Rule

Using the Placement Setup dialog box, you can set a placement grid and a PCB spacing rule.

PCB Router does not require that you define a grid for automatic placement, but if design or manufacturing rules dictate a placement grid, you can set the grid for all components (SMD and through-pin) in the Placement Setup dialog box.

Note: You can set separate placement grids for SMD and through-pin components by choosing *Rules - PCB - Placement Grids*.

In addition to setting the placement grid, you can use the Placement Setup dialog box to set a PCB spacing rule. A PCB spacing rule sets the minimum spacing allowed between all components in the design. Rules are set at other levels through the *Rules* menu. In the following procedure, you will set a PCB spacing rule of .05 inches.

Task: Set a placement grid and spacing rule

Procedure

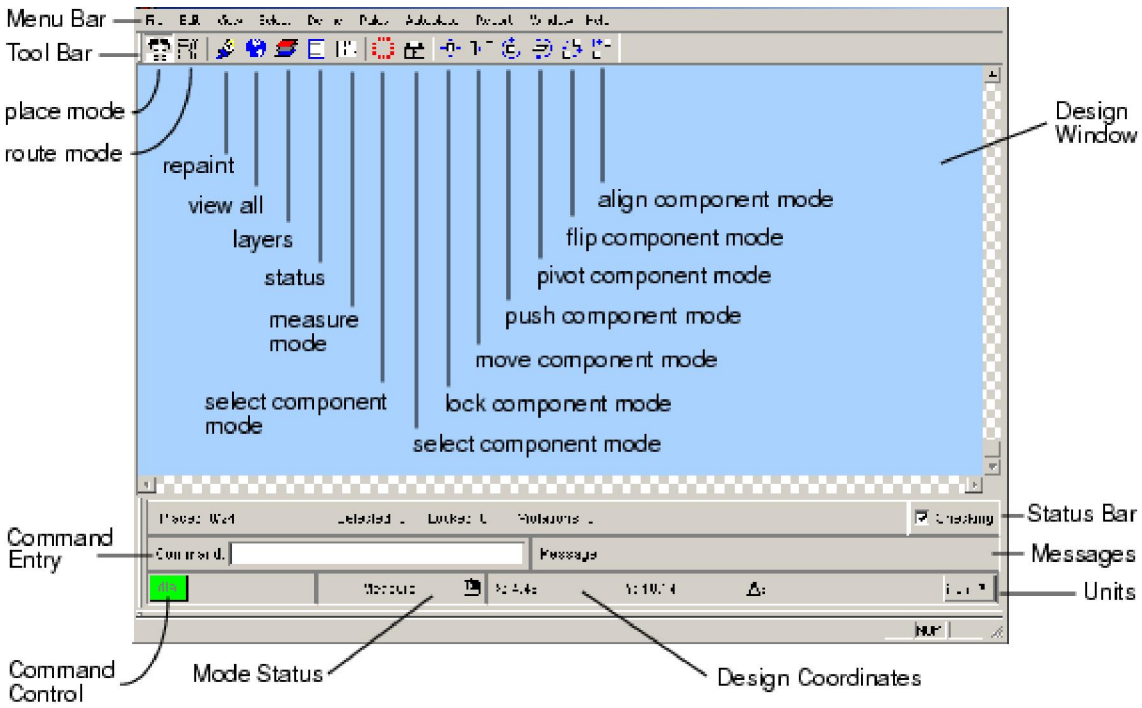
1. Start PCB Router and load `lesson2.dsn` from the `tutorial` directory. See Where to find the Accompanying Lesson Files for the location of this directory.

   When you first start PCB Router, you are in Route mode. The menu bar and status bar are specific to autorouting. You need to change to Place mode.

2. Click the *Place* button on the tool bar.

   The tool bar changes.

Figure 2-2 Placement GUI (Windows)



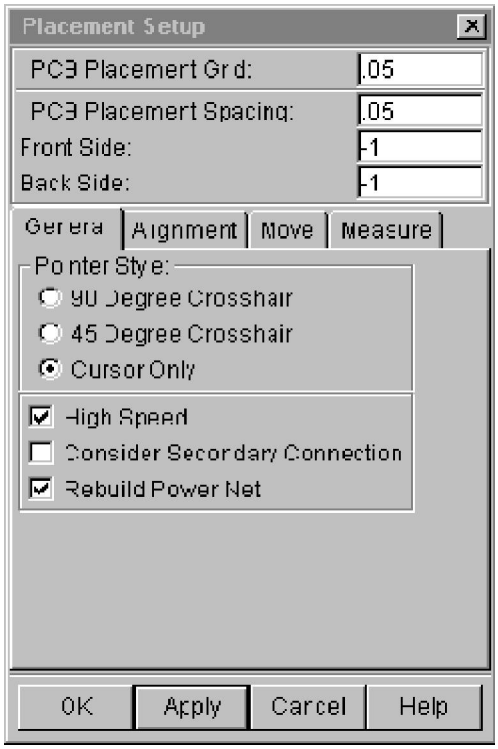
3. Choose *Autoplace - Setup*.

   The Placement Setup dialog box opens.

4. Enter **.05** in the *PCB Placement Grid* data entry box.

5. Enter **.05** in the *PCB Placement Spacing* data entry box.

   The Placement Setup dialog box looks like the following.



6. Click *OK*.

You defined a grid of .05 inches and a spacing rule of .05 inches. Notice that the *Checking* option in the right side of the status area is selected. This option checks for placement rule violations while you interactively place components and prevents placing or moving a component if it violates a rule. If you want to place a component in a location that violates a rule, turn off *Checking*.

Note: You can set separate spacing rules for SMD components, through-pin components, and between SMD and through-pin components by choosing *Rules - PCB - Spacing*. You also can use this command to see the current PCB spacing rule.

Preplacing Connectors and Critical Components

After you set placement rules, you are ready to preplace connectors and other critical components. In this stage of placing components, you will:

- display the component reference designators.
- place components by specifying XY locations.
- place components from a defined list.
- lock the preplaced components.

Displaying Component Reference Designators

In the following procedure, you turn on the reference designator labels so that you can identify components.

Task: Display component reference designators

Procedure

1. Choose *View - Labels*.

   The View Labels dialog box opens.

2. Make sure *Ref Des* is selected.

3. Make sure the *Side* is set to *Both*.

4. Click *OK*.

   A reference designator label appears in the center of each component.

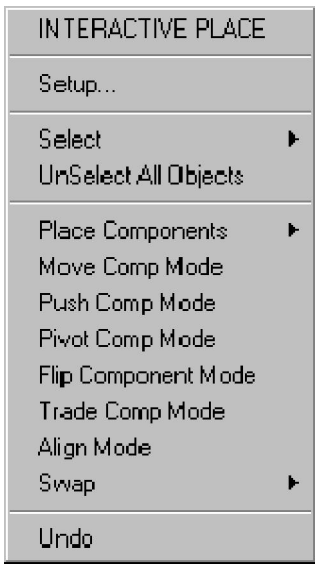
   Note: The router scales the work area. If you do not see all the designator labels, you need to zoom in to adjust the magnification of the work area.

Placing Components by Specifying X,Y Locations

Through the Interactive Place menu, the autorouter offers a variety of ways to place components interactively. You can place a single component, a list of components, and multiple components. You can place components by specifying X,Y locations, by clicking a point in the work area, or based on connectivity.

The Interactive Place menu appears when you press the right mouse button in the work area as shown in the following figure.

Figure 2-3 Interactive Place Menu



To select a command from the Interactive Place menu, you will follow these steps:

1. With Place mode active and your pointer in the Design window, press the right mouse button.

   The Interactive Place menu appears.

2. Slide your pointer over the menu to highlight the desired command.

3. Select the command by pressing the left mouse button.

In this section, you will place the origins of components J1, J2, and U9 at exact X,Y locations.

Task: Place components by specifying X,Y locations

Click on the following link to see how this is done before attempting the task on your own.

Show me

This demonstration runs in your browser for approximately 2 minutes and 12 seconds.

Procedure

1. Press the right mouse button and choose *Place Components - XY Location*.

   The Place Component dialog box opens, as shown in the following figure.

2. Scroll the *Components* list to locate J1.

3. Select J1 in the *Components* list.

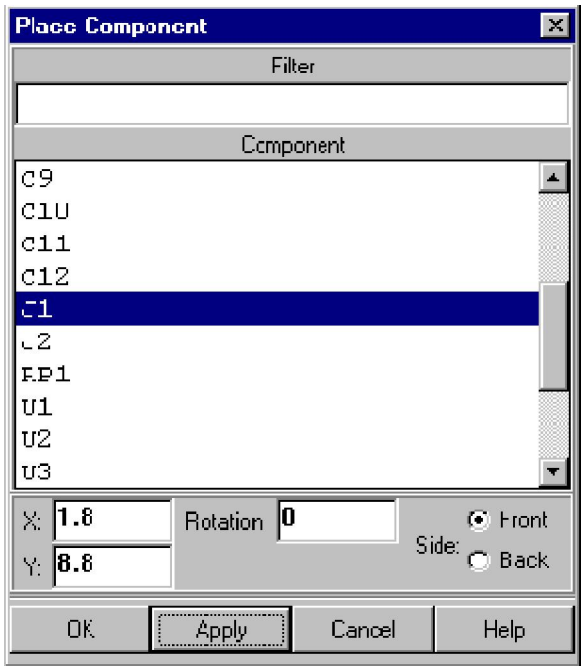
   J1 is highlighted.

4. Enter **1.8** in the *X* data entry box.

5. Enter **8.8** in the *Y* data entry box.

6. Make sure the *Rotation* is *0*.

7. Make sure the *Side* is *Front*.



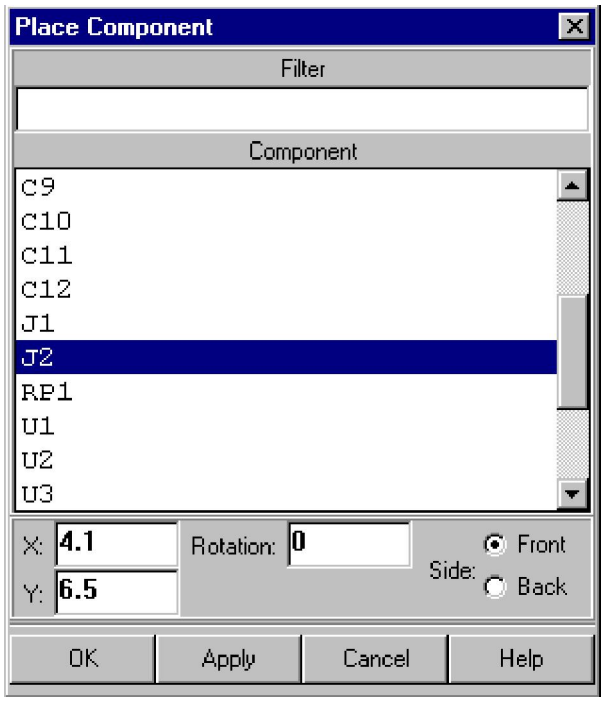
8. Click *Apply* to place the origin of J1 at 1.8, 8.8.

Next you place J2.

9. Select J2 in the *Components* list.
10. Type **4.1** in the *X* data entry box.
11. Type **6.5** in the *Y* data entry box.

    Keep the *Rotation* at *0* and *Side* set to *Front*.

    The dialog box looks like the following.



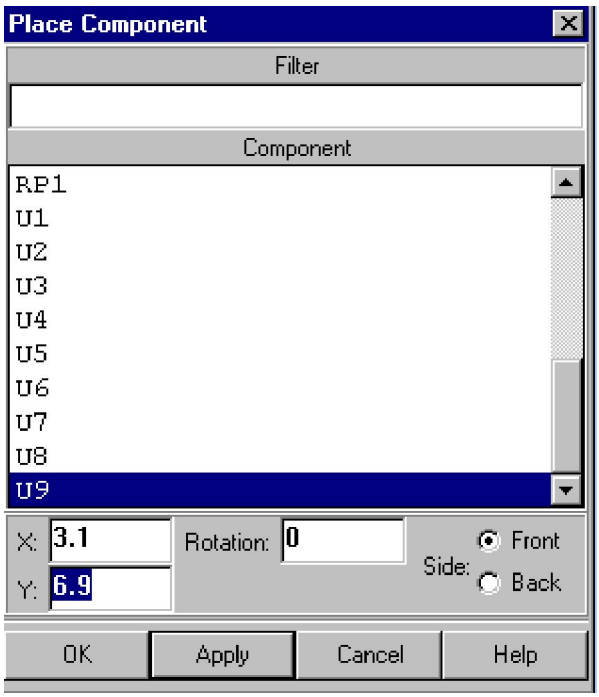
12. Click *Apply* to place the origin of J2 at 4.1, 6.5.

    Next you place the PGA component labeled U9. Like the connectors, U9 has a large number of interconnections with other components in the design, which is why you interactively place it.

13. Click U9 in the *Components* list.
14. Enter **3.1** in the *X* data entry box.
15. Enter **6.9** in the *Y* data entry box.

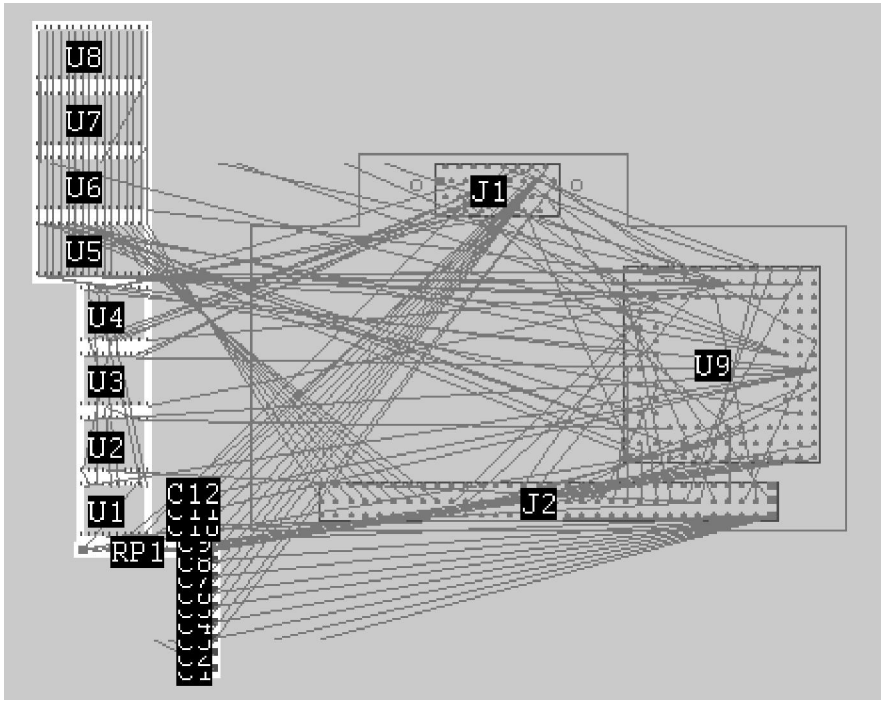
    Keep the *Rotation* at *0* and *Side* set to *Front*.

    The dialog box looks like the following.



16. Click *OK*.

The design is shown in the following figure.



You just learned how to place components by specifying X,Y coordinates. Next you will learn how to place a list of components.

Placing Components From a List

Using Place List mode, you can specify a list of components and place them in the order you specify. The first component in the list attaches to the pointer. Drag the component to the location you want, and click the left mouse button to place it. Immediately, the next component in the list attaches to the pointer so you

can drag the component to the location you want and click the left mouse button to place it. You repeat this process until all components in the list are placed.

Task: Place components from a list

Click on the following link to see how this is done before attempting the task on your own.

Show me

This demonstration runs in your browser for approximately 3 minutes and 50 seconds.

Procedure

1. Press the right mouse button and select *Place Components - Place List Mode*.

   The Place Component List dialog box opens.

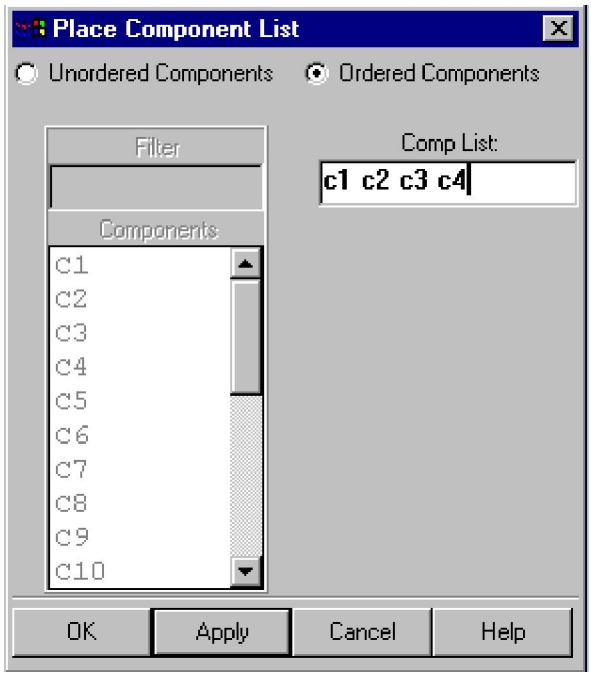
2. Select *Ordered Components*, which tells the autorouter to place the components in the order that you specify in the *Comp List*.

3. Type the following in the *Comp List* data entry box:

   c1 c2 c3 c4

   Note: Make sure you leave a space between each reference designator.

   The Place Component List dialog box looks like the following.



4. Click *OK*.

   C1 attaches to the pointer. You will need to rotate it 90 degrees counterclockwise before you place it.

5. Press the right mouse button and select *Pivot Mode - 90*.

   A pivot arm attaches to the center of the component. This pivot arm looks like a string that pivots the component in the direction you move the pointer.

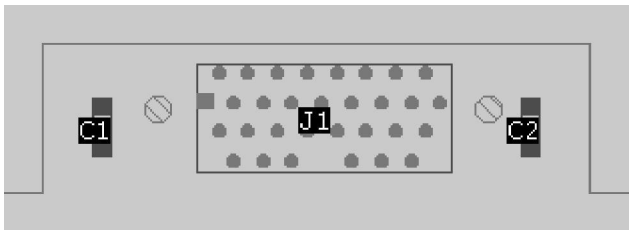
6. Move the pointer away from the center of the component in a counterclockwise direction.

   The component rotates 90 degrees counter clockwise. increments as you move the pointer.

   DR:90 appears near the right side of the status bar when the component is 90 degrees from its original position. DR stands for Delta Rotation.

7. Click the left mouse button when C1 is 90 degrees counterclockwise from its original position.

   You are ready to place C1. The following figure shows where to place C1 and C2.



8. Drag C1 to the left of J1 and click to place it.

   You can use the middle mouse button for the following operations to pan or zoom to a new location.

   C2 attaches to the pointer after you place C1.

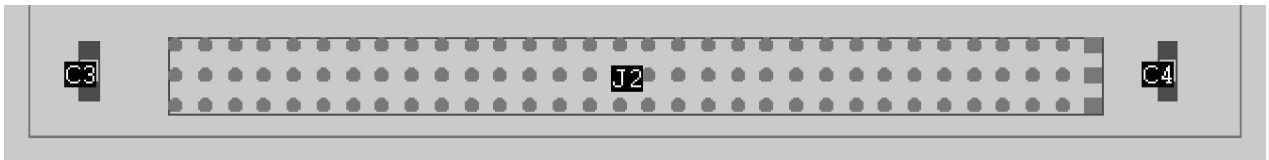
9. Repeat steps 5 through 7 to rotate C2 90 degrees counterclockwise.

10. Drag C2 to the right of J1 and click to place it.

    C3 attaches to the pointer.

11. Repeat steps 5 through 7 to rotate C3 90 degrees counterclockwise.

    You are ready to place C3. The following figure shows where to place C3 and C4.



12. Drag C3 to the left of J2 and click to place it.

    C4 attaches to the pointer.

13. Repeat steps 5 through 7 to rotate C4 90 degrees counterclockwise.

14. Drag C4 to the right of J2 and click to place it.

Note: If you do not like where you placed a component and you want to move it, press the right mouse button and select *Move Component*. Click the component, drag it to the new location, and click to place it.

Next you will lock the preplaced components so the autorouter cannot move them during automatic and interactive placement.

Locking Components

After you place critical components, you lock them in position so PCB Router does not move them during automatic or interactive placement. If you need to move a component at a later time, you can unlock it, move it, and lock it again.

Task: Lock critical components

Procedure

1. Click the *Lock* button on the tool bar.

   Lock Position appears in the mode status area.

2. Click a point to the left of C1.

3. Drag the pointer diagonally around C1, J1, and C2, and release the left mouse button when the bounding box encloses the components.

   The locked components display a magenta border. Magenta is the default *locked* color.

4. Repeat steps 2 and 3 to lock components C3, J2, and C4.

Note: You can unlock the components while you are in Lock Position mode. To unlock the components, click on each locked component or draw a bounding box around the components. You also can use *Edit - [Un]Lock Components* and *Edit - [Un]Lock Components Mode* to lock and unlock components.

Placing Large Components

PCB Router assigns the large and small properties to components based on the number of pins. A large component contains four or more pins. A small component contains three or less pins. PCB Router usually places large components with the highest connectivity first.

In this section you will:

- define areas where you do not want components placed.
- automatically place large components.
- interchange placed components to reduce manhattan lengths, minimize crossovers, and reduce congestion.

Defining Areas where PCB Router Cannot Place Components

Before you automatically place the large components, you need to provide routing space around the connectors. To do this, you specify an area around each connector where the PCB Routercannot place components. These areas are called placement keepout areas.

Task: Define placement keepout areas

Procedure

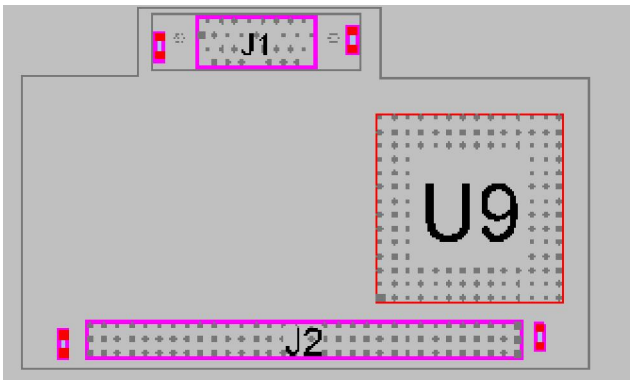
1. Choose *Define* - *Keepout* - *Draw Mode*.

   Draw Keepout appears in the mode status area. You will draw a bounding box as close as possible to C1, J1, and C2.

2. Click a point to the lower left of C1.

3. Drag the pointer diagonally around C1, J1, and C2, and release the left mouse button when the bounding box encloses the components. If you do not like how you drew the box, choose *Define* - *Keepout* - *Draw Mode* to remove the box and start with step 2 to draw a new bounding box.

   In the following figure, the ratsnest is not displayed. On your screen, the area around C1 and C2 might be difficult to see because the ratsnest is displayed.



   The ratsnest lines are called unroutes. You will turn off the display of unroutes, so that you can see the keepout area more easily.

4. Choose *View* - *Guides* - *Off*.

   The keepout area is now easier to see.

5. Press the right mouse button and select *Define Polygon as Keepout*.

   The Add Polygon as Keepout dialog box appears, as shown in the following figure.

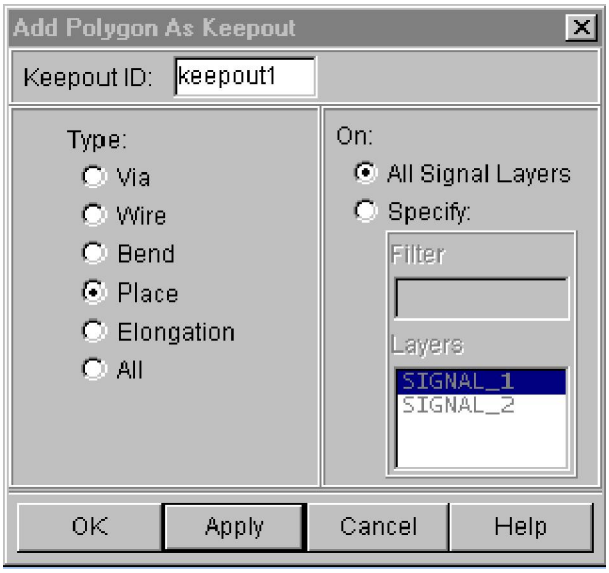
6. Make sure the Keepout ID is *keepout1*.

   The *Keepout ID* is the name the autorouter assigns to the keepout area.

7. Select *Place* as the *Type*.

   This sets the area as a placement keepout, in which PCB Router cannot place components.

The Add Polygon As Keepout dialog box looks like the following.



8. Click *OK*.

   The keepout area appears as a bounding box filled with a crosshatched pattern.

   Note: Information about keepouts will display in the measure box and output window if the object selection button is turned on in the layers panel next to the keepout button.

   Next you create a keepout area around C3, J2, and C4.

9. Make sure Draw Keepout mode is still set by checking the mode status area.

10. Click a point to the lower left of C3.

11. Drag the pointer diagonally around C3, J2, and C4, and release the left mouse button when the bounding box encloses the components. Remember to draw the boundary box as close as possible to the components.

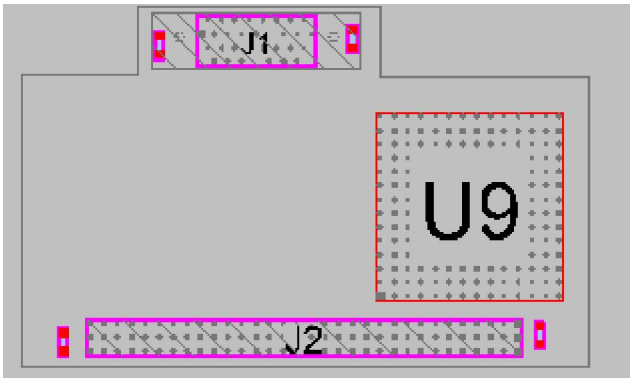
12. Press the right mouse button and select *Define Polygon as Keepout*.

    The Add Polygon as Keepout dialog box opens.

13. Make sure the Keepout ID is *keepout2* and *Place* is selected.

14. Click *OK*.

    The keepout areas appear as crosshatched rectangles, as shown in the following figure.



Next you will automatically place the large components. First, display the unroutes so you can see the connectivity while the autorouter places the components.

15. Choose *View - Guides - All*.

    The guides display.

    Note: To create a keepout area by entering coordinates, choose *Define - Keepout - By Coordinates*.

Automatically Placing Large Components

You initially place ICs and other large components by using the InitPlace Large Components dialog box. You use the dialog box to control the preferred component spacing, placement side, and component orientations.

These preferences are not rules. Preferences set in the InitPlace Large Components dialog box are followed if they do not violate placement rules.

You will place the large SMD components on the front side with a vertical orientation and the PTH component (RP1) on the front side with a horizontal orientation.

Task: Automatically place large components

Procedure

1. Choose *Autoplace - InitPlace Large Components*.

   The InitPlace Large Components dialog box opens, as shown in the following figure.

2. Make sure *All* is selected on the components panel.

   This option places all large, unplaced components.

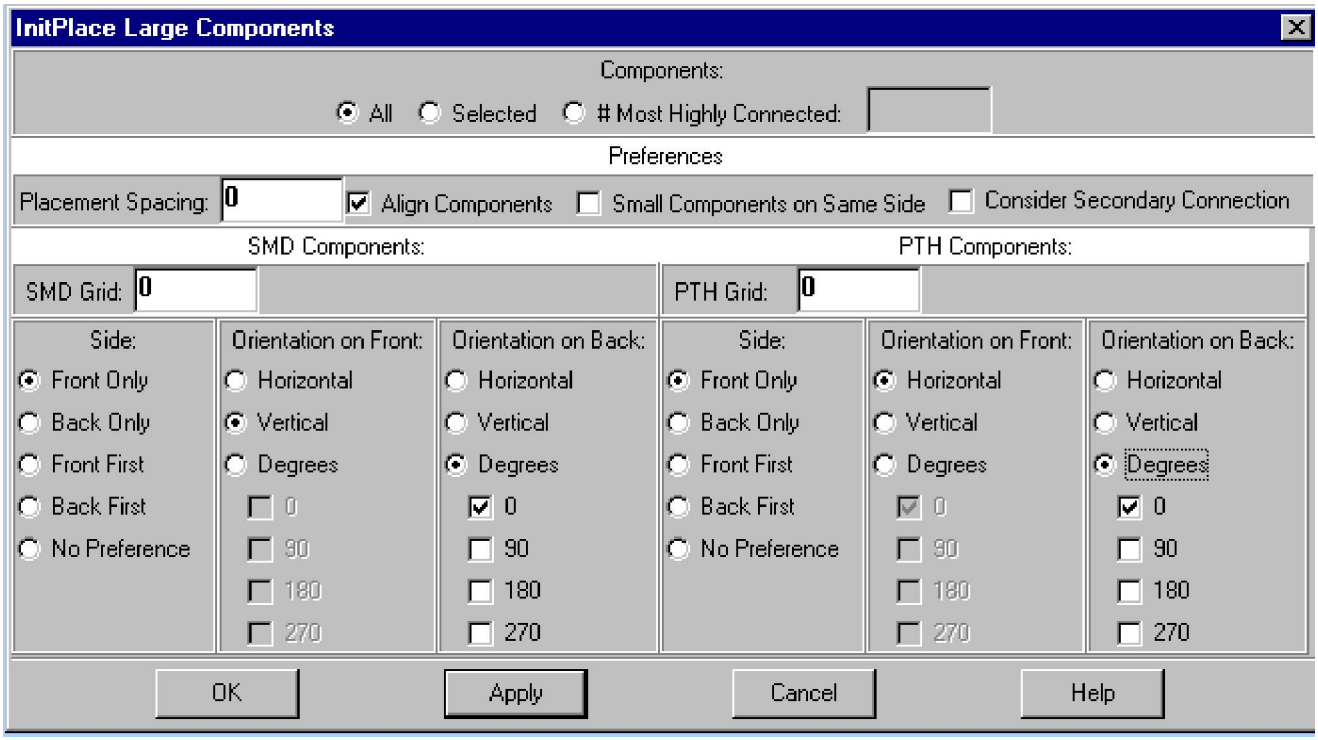
3. Select *Front Only* for the *Side* on the *SMD Components* panel.

   This option is on the left side of the dialog box.

4. Select *Vertical* for *Orientation on Front* on the *SMD Components* panel.

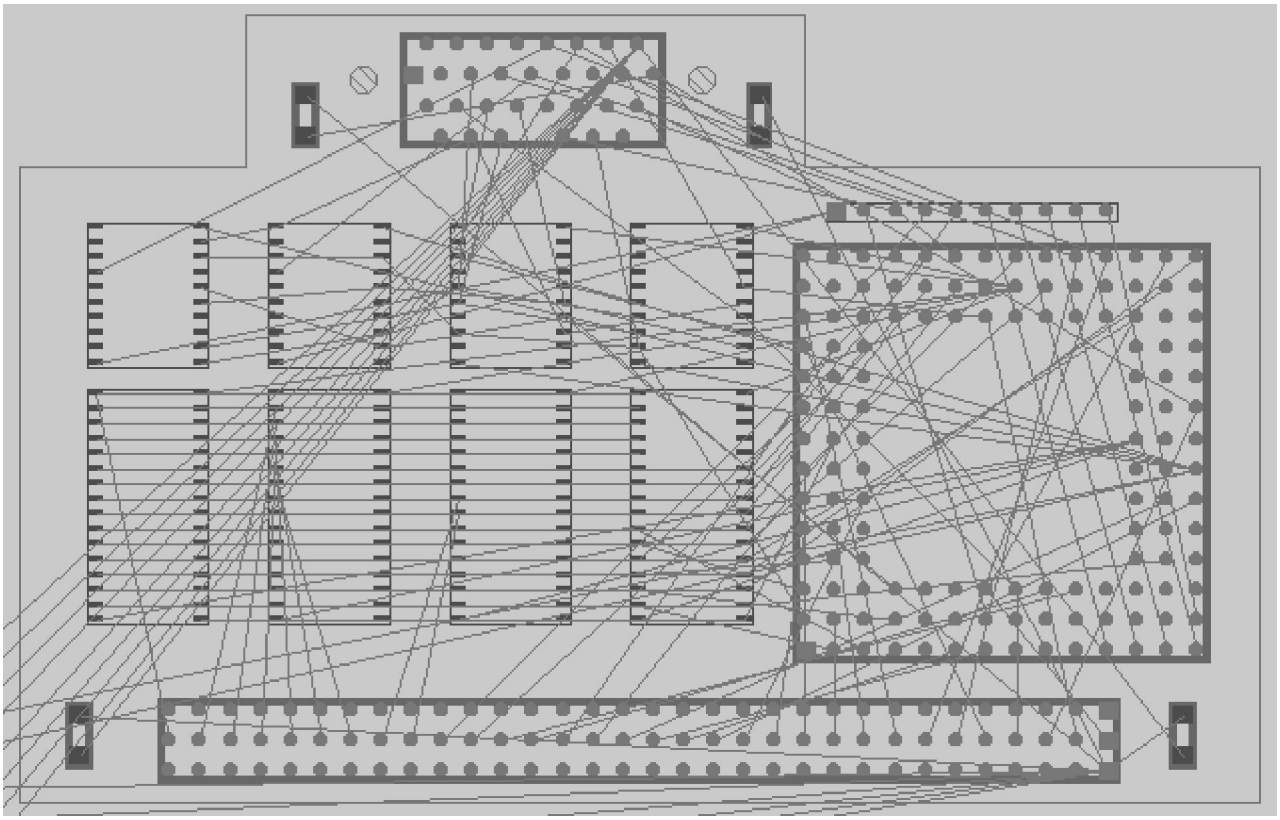
5. Select *Front Only* for the *Side* on the *PTH Components* panel.

6. Click *Horizontal* for *Orientation on Front* on the *PTH Components* panel.



7. Click *OK*.

   The autorouter places the SMD and PTH components, as shown in the following figure.



Interchanging Components

After the initial placement, you interchange component positions to reduce manhattan lengths, minimize crossovers, and reduce congestion.

The interchange operation is sometimes referred to as a pairwise interchange because the operation applies to a single pair of components at a time. The goal is to place interconnected components in close proximity to reduce wire lengths.

Multiple interchange passes usually produce the best results. Use eight or more interchange passes. If an interchange pass does not improve weighted manhattan lengths compared to the previous pass, the interchange operation stops and the remaining passes are skipped. You can use the placement status report to compare manhattan lengths before and after interchange passes.

Task: Interchange large components

Procedure

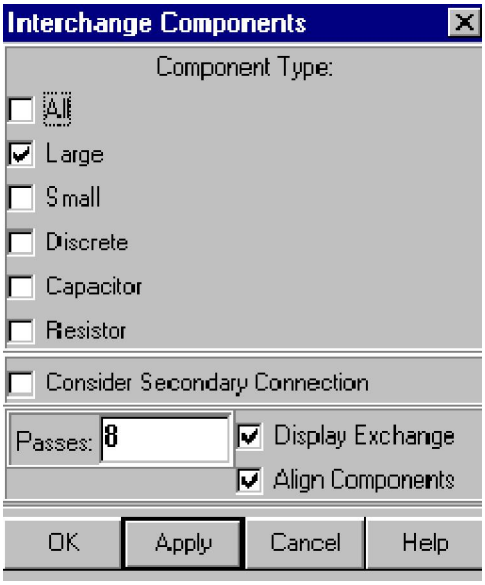
1. Choose *Autoplace - Interchange Components*.

   The Interchange Components dialog box appears.

2. Make sure *Large* is selected.

3. Make sure **8** is in the value for *Passes*.

   The Interchange Components dialog box looks like the following.



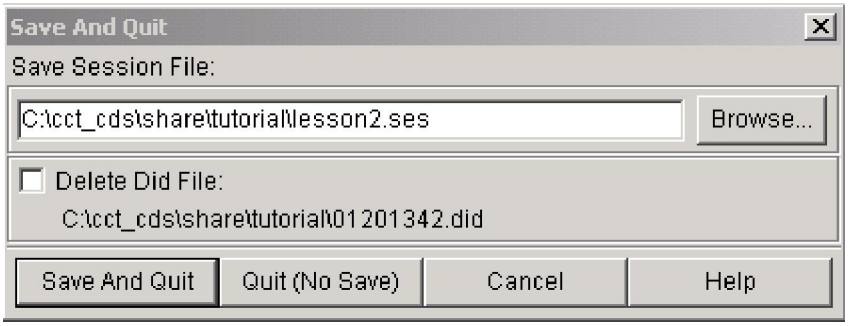
4. Click *OK*.

   After each component interchange, the work area is repainted because *Display Exchange* in the dialog box was selected. This button controls whether graphics are updated with each component interchange. To improve performance with larger designs, you unselect *Display Exchange*.

5. Choose *File - Quit*.

   The Save and Quit dialog box opens, as shown in the following figure.

6. Click *Delete Did File* to remove the did file.



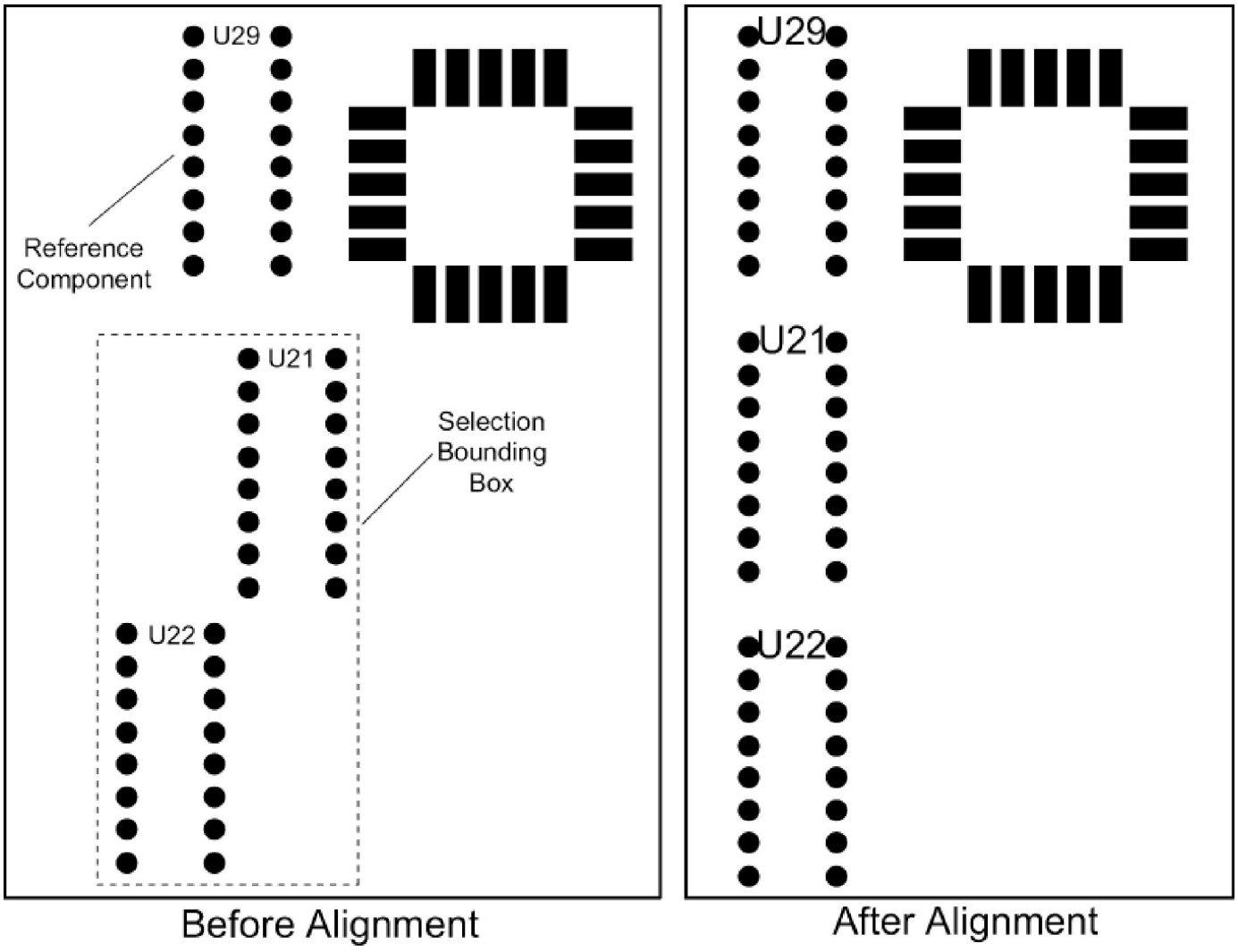
7. Click *Quit (No Save)* to exit without saving a session file and Did file.

Next you will learn to interactively align components.

Aligning Components

You can align components by using Align Comp mode. In this mode, you select the components you want to move, then you select a reference component to which these components align.

Figure 2-4 Component Alignment



If *Checking* is on and the alignment causes a placement violation, PCB Router does not align components. If you want to ignore rule violations, turn off *Checking* in the status bar. You'll leave *Checking* on in the following procedure.

Task: Align components

Procedure

1. Start PCB Router and load `lesson2.dsn` from the `tutorial` directory.

   You need to change to Place mode.

2. Click the *Placement* button on the tool bar.

   You will load a placement file.

3. Choose *File - Read - Placement*.

   The Read Placement dialog box opens.

4. Click the *Browse* button.

   The *Open* dialog box appears for Windows platforms.

   The *Select File* dialog box appears for UNIX platforms.

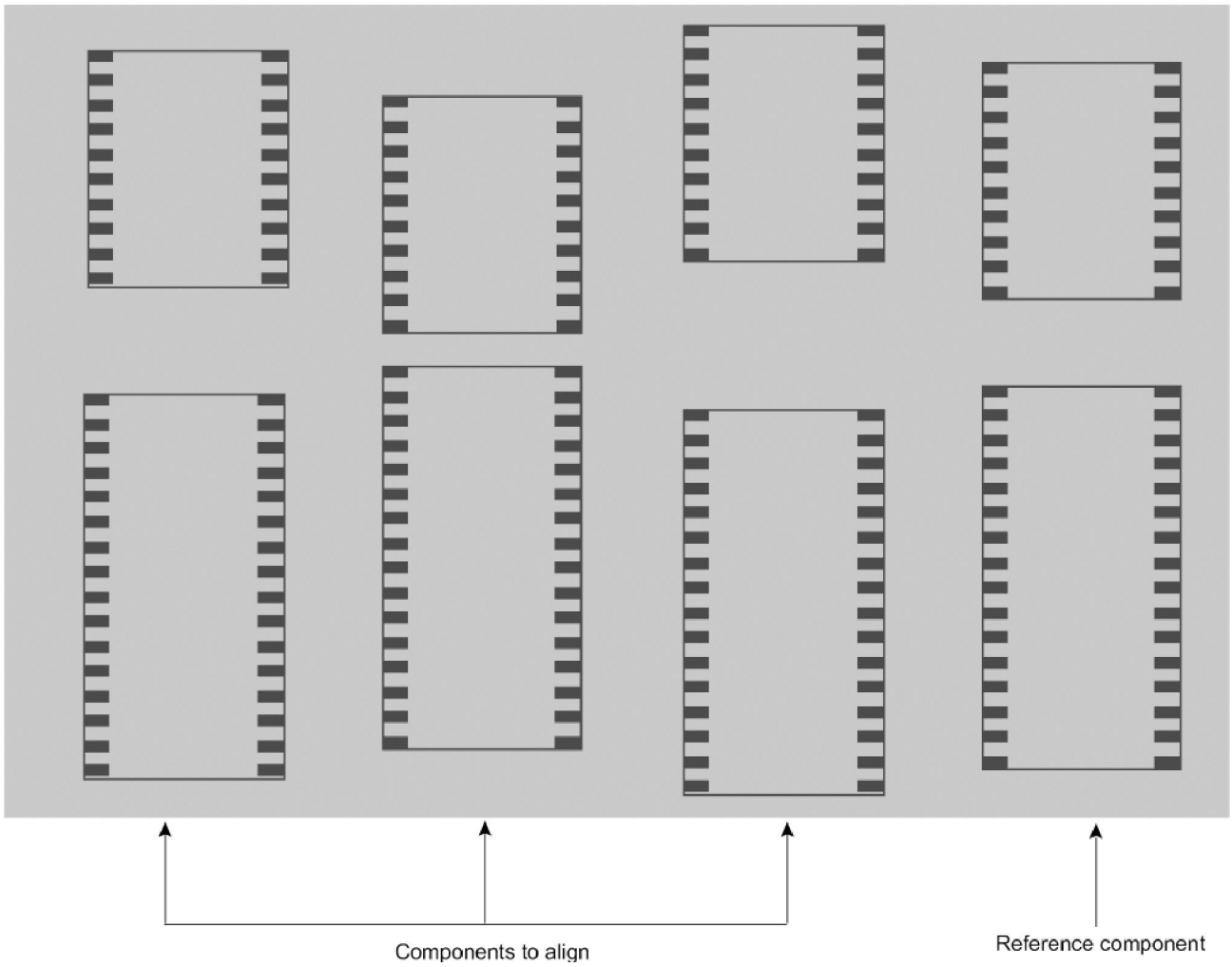
5. Change to the tutorial directory for Windows platforms and open `lesson2.plc`.

   Make sure `lesson2.plc` is selected and click *OK* for UNIX platforms.

   The filename is added in the Read Placement dialog box.

6. Click *OK*.

   The Placement file loads. The Small Outline Integrated Circuits (SOICs) are not aligned, as shown in the following figure.



7. Choose *View* - *Guides* - *Off* to turn off the guides.

   You will align the SOICs.

8. Press the right mouse button and select *Align Mode*.

   Align Comp appears in the mode status area.

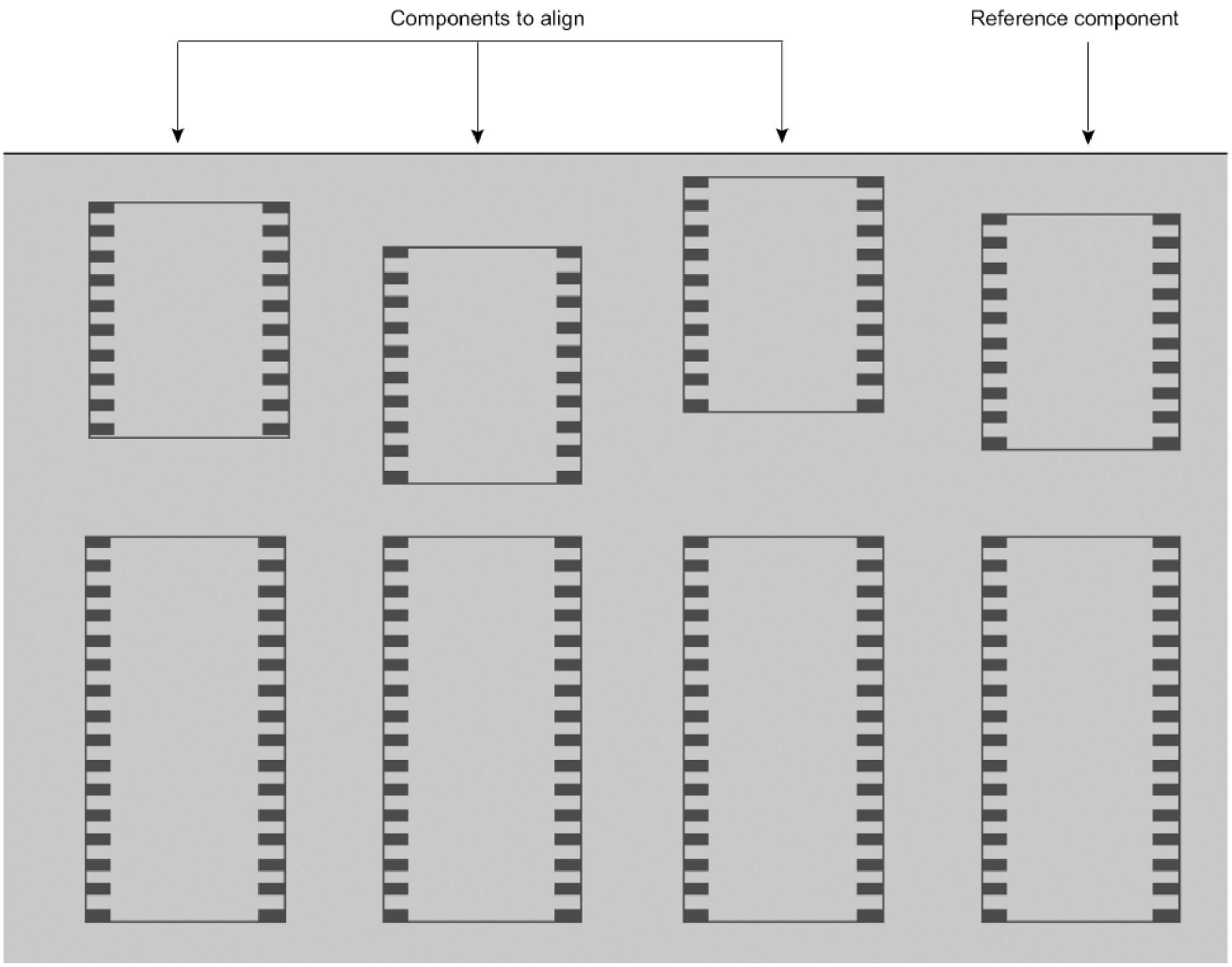
   You will use the right SOIC as the reference component to align the three SOICs.

9. Drag the pointer to enclose the three 32-pin SOICs that are located on the left as shown in the previous figure.

   The three SOICs are selected. You will align these SOICs to the 32-pin SOIC that is located on the right.

10. Click in the 32-pin SOIC that is located on the right.

    The components align, as shown in the following figure.



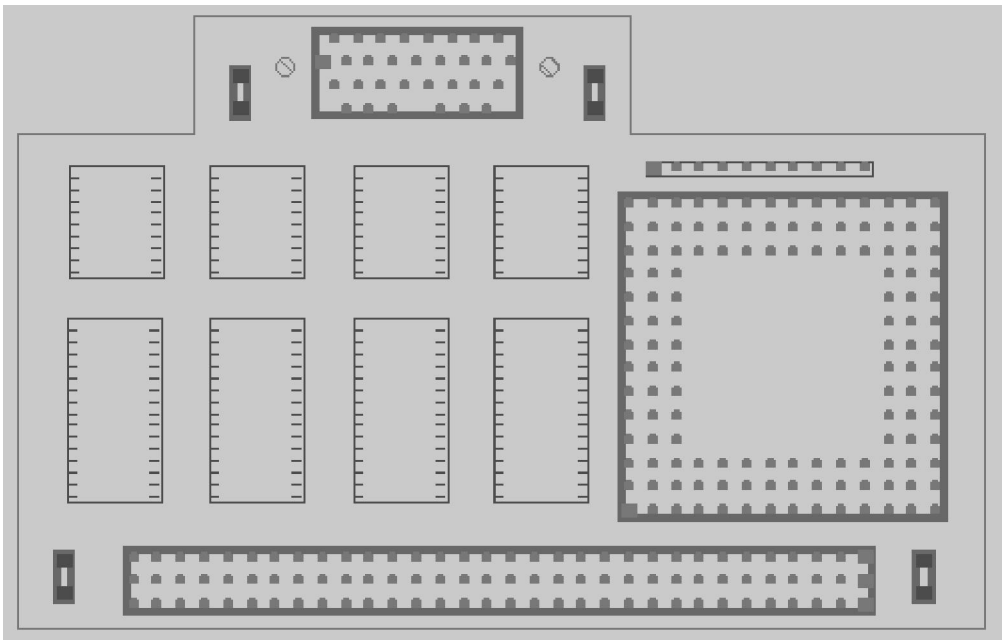
Next you will align the three 20-pin SOICs that are located above the 32-pin SOICs as shown in the previous figure. Align Comp mode is still active.

11. Drag the pointer to enclose the three 20-pin SOICs that are located on the left.

    The three SOICs are selected. You will align these SOICs to the reference 20-pin SOIC component that is located on the right.

12. Click in the 20-pin SOIC that is located on the right.

    The components align, as shown in the following figure.



Note: PCB Router uses a reference point on each component to make the alignment. In the previous procedure, the alignment was relative to the upper left pin of the reference component, which is the default. You can change this reference point by using the Placement Setup dialog box.

Placing Small Components

You can place small signal components by choosing *Autoplace - InitPlace Small Components*. To place decoupling capacitors, you will use another method.

PCB Router can *learn* a component pattern, which is the side, location, and orientation of a small component relative to a large component. This component pattern is applied by placing other instances of the small component image in the same pattern to other instances of the large component image.

In this section, you will place the capacitors by having PCB Router learn and apply component patterns.

You will complete the following tasks

- Display power pin labels
- Move, flip, and pivot a capacitor
- Learn the component pattern
- Apply the component pattern to other instances of the large component image

Displaying Power Pin Labels

You will need to display the power pin labels to orient the components. To display the power pins, turn on the power pins layer.

Task: Display power pins labels

Procedure

1. Click the *Layer* button on the tool bar.

The Layers panel appears. The Layers panel controls layer visibility, layer routing direction, and layer selection.

2. Click *Power pins* on the Layers panel. Power pins is located near the bottom of the Layer panel.

   The power pins display.

3. Click *Close* on the Layers.

Now that you can see the power pins, you will move C5 (on the PCB bottom side) so that the power pin of C5 is under the power pin of U1 (on the PCB top side).

Moving, Flipping, and Pivoting a Component

An easy way to place a component is to use Move Comp mode. When you click on a component in this mode, the component's reference designator displays. As you move the pointer, a ghost image of the component follows. The next click places the component at the location under the pointer.

You will need to display the reference designators, which appear at component centers.

Task: Move, flip, and pivot a component

Procedure

1. Choose *View - Labels*.

   The View Labels dialog box opens.

2. Make sure *Ref Des* is selected.

3. Click *OK*.

4. Press the right mouse button and select *Move Comp Mode*.

   Move Comp appears in the mode status area.

5. Click on C5, which is the bottom component in the column of unplaced components. You can use the middle mouse button to zoom in if necessary.

   The component attaches to the pointer. You need to flip the component to the back side by using the Move Comp menu.

6. Press the right mouse button and select *Flip* from the Move Comp menu.

   This menu contains commands that manipulate the component before you place it.

   Next you need to pivot the component so that the power pin is at the bottom of the component, as shown in the following figure.

7. Press the right mouse button and select *Pivot - 90*.

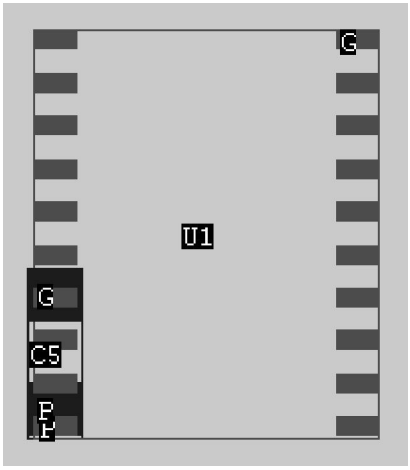
This command pivots C5 in 90 degree increments.

C5 attaches to the pointer.

8. Move C5 to the lower left of U1 (so that the power pin of C5 is under the power pin on U1). You can use the middle mouse button to zoom in if necessary.

9. Click to place the power pin of C5 under the power pin of U1.

   The following figure shows the relationship of C5 to U1.



Note: After you attach a component to the pointer and move the pointer, you can return the component to the previous location by pressing the right mouse button and selecting *Cancel*.

Next PCB Router will learn this component pattern and apply it to the components with the same image ID.

Learning and Applying the Component Pattern

Now that you placed C1, the autorouter can learn the component pattern of the small component (C5) in relationship to the large component (U1). PCB Router learns the orientation, location, and side of the small component with respect to the large component. It then places other instances of the capacitor image in the same pattern to other instances of the SOIC image.

Task: Learn and apply the component pattern

Procedure

1. Click the *Select Comp* button on the tool bar.

   Sel Comp appears in the mode status area.

2. Drag the pointer to enclose U1 and C5.

The two components are selected and *Selected:2* appears in the status area.

3. Choose *Autoplace - Small Comp Pattern - Learn*.

   PCB Router stores orientation, location, and side information about the relationship between C5 and U1.

4. Unselect C5 and U1 by dragging the pointer over them.

5. Choose *Select - Images - Sel Image Mode*.

   Sel Image appears in the mode status area. Using this mode, you can select all instances of an image by clicking on one instance.

6. Click on U1.

   U1, U2, U3, and U4 are selected.

7. Choose *Autoplace - Small Comp Pattern - Apply to Selected*.

   The learned pattern is applied, and the power pins of C6, C7, and C8 are placed under the power pins on U2, U3, and U4.

8. Choose *Select - UnSelect All Placement Objects*.

   Next you place the power pins of C9, C10, C11, and C12 under the power pins of U5, U6, U7, and U8 by repeating the steps from the previous section.

9. Press the right mouse button and select *Move Comp Mode*.

   You will move the power pin of C9 under the power pin of U8.

10. Click on C9.

    You need to flip C9 to the back side.

11. Press the right mouse button and select *Flip*.

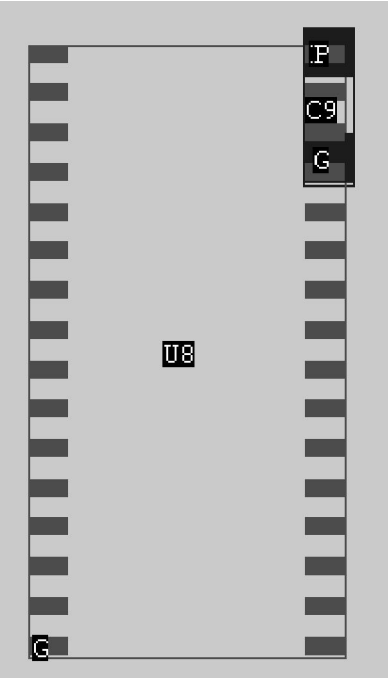
    Next you pivot C9 so that the power pin is at the top.

12. Press the right mouse button and select *Pivot - 90*.

13. Move C9 to the upper right of U8 (so that the power pin of C9 is under the power pin on U8).

14. Click to place the power pin of C9 under the power pin of U8.

    The following figure shows the relationship of C9 to U8.



Next you will learn and apply this component pattern to the unplaced capacitors.

15. Click the *Select Comp* button on the tool bar.

    Sel Comp appears in the mode status area.

16. Select *U8* and *C9* by dragging the pointer over them and releasing the left mouse button.

    The two components are selected, and *Selected:2* appears in the status area.

17. Choose *Autoplace - Small Comp Pattern - Learn*.

    PCB Router stores orientation, location, and side information about the relationship between C9 and U8.

18. Unselect *C9* and *U8* by dragging the pointer over them.

19. Choose *Select - Images - Sel Image Mode*.

20. Click on *U8*.

    U5, U6, U7, and U8 are selected.

21. Choose *Autoplace - Small Comp Pattern - Apply to Selected*.

    The learned pattern is applied, and the power pins of C10, C11, and C12 are placed under the power pins of U5, U6, and U7.

22. Choose *Select - UnSelect All Placement Objects*.

    Congratulations! All components are placed.

Quitting the Autorouter and Saving Placement Results

When you are satisfied with your placement results, save your work. You can save your work in a Placement file or a Session file. If you save a Placement file, you can reload the file at the start of a session or anytime during the session. The Placement file is useful if you want to perform multiple placement trials and compare the files to choose the best result. You create a placement file by choosing *File - Write - Placement*.

If you create a Session file, you load the Session file only when you start PCB Router. A session file contains a reference to the original design filename as well as detailed placement, floorplan, swap, netlist, and route data. You will exit PCB Router and create a Session file in the next procedure.

Task: Quit the Autorouter and save a Session file

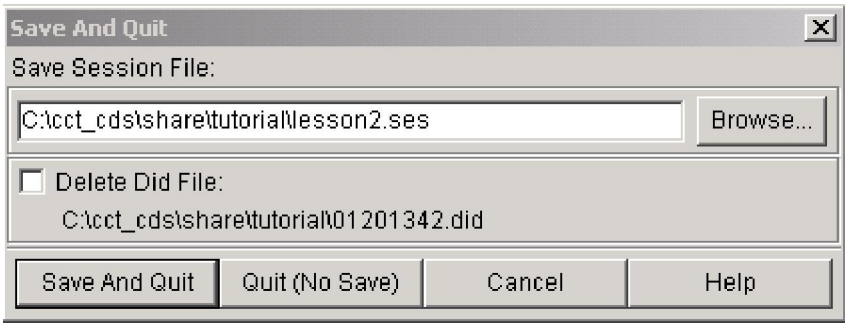
Procedure

1. Choose *File - Quit*.

   The *Save And Quit* dialog box appears with *lesson2.ses* in the *Save Session File* data entry box, as shown in the following figure.

2. Click *Delete Did File* to remove the Did file.

   In Lesson 1, you learned that a did file contains the command history from a the autorouter session. You do not need this file for the sesssion, so you will delete it.



3. Click *Save and Quit*.

   PCB Router exits and saves the placement information in the Session file.

What You Learned

In this lesson, you learned how to place components interactively and automatically.

You learned :

- the basic steps used in placing components.
- how to set placement options.
- how to set placement rules.
- how to preplace connectors and critical components.

- how to place large components.
- how to edit the placement.
- how to place small components.

In the next lesson, you will learn how to autoroute a PCB design.

Return to top of page

Library Contents Index < Previous Next > View/Print PDF Search Feedback Help Exit