Osmond PCB Reference Manual

Joseph C. Chavez

October 13, 2006

Contents

List of Figures

Chapter 1

Introduction

Corresponds to Osmond PCB Design Version 1.0.8

For the latest version, visit:

http://www.swcp.com/~jchavez/osmond.html

Osmond PCB is a flexible tool for designing printed circuit boards. Its many features include:

- Virtually unlimited board sizes
- Virtually unlimited number of board layers
- Virtually unlimited number of parts
- Spatial resolution to 10 nanometers
- Concurrent support for both Metric and English measurements
- All angle routing
- Rubber-band traces
- Variable grid
- Support for both through-hole and surface mount parts
- Circular, rectangular, and oval pads
- Easy part editing

- Multiple trace widths
- Multiple trace spacing
- Curved traces
- Silkscreen, solder-mask and auxiliary layers
- Signal and ground planes
- Copper fill areas
- Automatic clearance checks
- Easy manual and semi-automatic routing
- Built-in script language
- Gerber (RS-274X) and drill file (Excellon) output
- DXF file output
- DXF file input
- Encapsulated PostScript file output
- Net check output (IPC-356)

1.1 Document Window

Design is done in a document window. The parts of a document window are shown in Figures 1.1, 1.2, and 1.3.

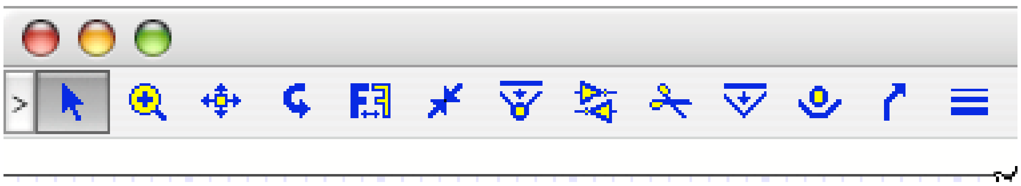


Figure 1.1: Window Top Left

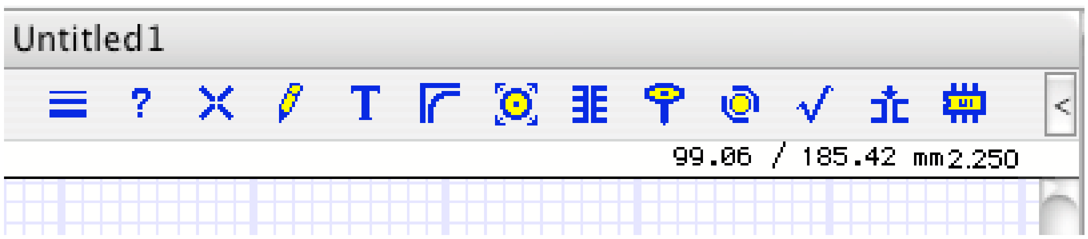


Figure 1.2: Window Top Right

The first two pictures show the top portion of the document window. The tool palette at the top of the window contains many tools that let you manipulate objects such as parts and connecting paths in the design. You select a tool by clicking on the tool button in the tool palette. Many tools also have options that you can change by double-clicking the tool button. A description of each tool can be found in the **Tools** and **More Tools** sections of this document.

The small buttons at the extreme left and right of the tool palette are scroll buttons. If your screen is not large enough to allow you to see all the tools at once, the scroll buttons can be used to move the tool palette left or right until the desired tool becomes visible.

Directly below the tool palette is a message field that Osmond uses to give you occasional bits of useful information. For example, as you move the cursor over each tool in the tool palette, the message field lists the function of the tool and tells you the keyboard shortcut (in brackets) for selecting the tool. As you move the cursor over pins, the message field lists the pin name. If the pin is part of some signal, the signal name is also listed. You can also use the message field to input commands as described in the **Command Line Interface** section of this document.

The number at the extreme right in the message field shows the scale of the view (assuming a standard 72 dot per inch monitor). The numbers to the left of the scale show the X and Y coordinates of the cursor in either mils or millimeters.

The bottom left of the design window looks like the picture shown in Figure 1.3. A pupup menu lets you move to a different layer. Next to the popup menu, a color swatch shows the color used for paths on the current layer. Clicking on the swatch brings up a dialog that lets you change the color.

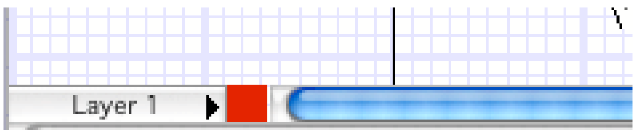


Figure 1.3: Window Bottom Left

As a shortcut, you can use the keys **1** through **0** on the keyboard to move to a different layer. If you are on Layer 1, for example, pressing the **2** key on the keyboard will take you to Layer 2. The **0** key will take you to Layer 10 (if it exists). Of course, if you have more than 10 layers you can only get to the layers above 10 by using the popup menu.

Note, however, that if you are currently on the Front Silk Layer, pressing the **2** key will take you to the Back Silk Layer and pressing the **1** key will bring you back to the Front Silk Layer. Similarly, if you are currently on the Front Mask Layer, pressing the **2** key will take you to the Back Mask Layer, and so on. The same goes for the front and back auxiliary layers.

Chapter 2

File Menu

2.1 New

The **New** command creates and displays a new untitled design. By default, the new design has 4 layers and a working area of 10 inches by 10 inches. If the default values are not appropriate for your design, you should change them immediately using the **Design Rules. . .** command from the **Edit** Menu.

Note: The working area will typically be somewhat larger than your PC board size. This will allow you to have a comfortable surrounding margin on all sides.

2.2 Open. . .

The **Open. . .** command opens an existing Osmond design file.

2.3 Open Recent

The **Open Recent** command allows you to select from a list of recently opened design files.

2.4 Import

2.4.1 Setup...

The **Import** → **Setup...** command lets you read a setup file containing general information about the design such as number of layers, design rules, size of working area, etc. This is a text file that can be produced from an existing design using the **Export** → **Setup...** command. You can also edit it with an ordinary text editor.

You can also read an existing Osmond design file with this command. If you do, only the setup information contained in the design file will be used. You can use this feature to allow a new setup to be the same as the setup used in an existing design.

2.4.2 Library...

The **Import** → **Library...** command lets you read a Library file containing part type definitions. This is a text file that can be built either with the Osmond application or with an ordinary text editor. The format of the Library file is described in the **File Formats** section of this document.

Each library that you import is added to the library of the current design, thereby increasing the size of the file that must be saved. As an alternative, you can just **Open** a library in a separate window. The **Library Parts** window, which you view via the **Window** menu, gives you access to any part defined in any open design. This allows you to use any part in any library without the need to import the entire library into your design.

You can also read an existing Osmond design file with this command. If you do, only the part type definitions contained in the design file will be used.

2.4.3 Part List...

The **Import** → **Part List...** command lets you read a file containing a list of parts for your design. Each element of the list associates a unique part name with a part type which should eventually be defined in the Library file.

You can still import a part if the part type is not defined in any library. In this case, a placeholder part type is used and you should replace this with a real part type using the **Replace Parts...** or **Replace Deferred Parts...** command before you import the Net List.

The format of the Part List file is described in the **File Formats** section of this document.

You can change part types or add parts to a design by changing a part list and importing the new part list into an existing design. If you import a part list that is equivalent to the existing list of parts in a design, nothing will change.

You can also import an existing Osmond design file with this command. If you do, only the part list information contained in the design file will be used.

2.4.4 Net List...

The **Import → Net List...** command lets you read a file containing a list of signals for your design. Each signal contains a signal name and a list of pins that should be connected to the signal. The format of the Net List file is described in the **File Formats** section of this document.

If you import a net list into an existing design, Osmond will attempt to make the design consistent with the new net list. Note that this may involve removing paths between pins if the design is changed.

You can also read an Osmond design file with this command. If you do, only the net list information contained in the design file will be used.

2.4.5 Part and Net List...

The **Import → Part and Net List...** command lets you read a file containing both a list of parts and a list of signals. This command is equivalent to doing an **Import → Part List...** followed by an **Import → Net List...** on the same file.

2.4.6 Sub Design...

The **Import** → **Sub Design...** command lets you read a file containing a complete Osmond design, which then becomes a sub-design of the current design. The intent is to allow a larger fabrication panel to be built up by combining a number of smaller designs. The smaller designs can be identical or different. Each imported sub-design is given a unique sequence number, starting with 1. Once sub-designs are imported, the **Select Sub Design...** command in the **Edit** menu lets you select a sub-design by specifying its sequence number.

When this is invoked, a dialog appears as shown in Figure 2.1 that lets you select a *Multiple* checkbox as well as X and Y offsets. If *Multiple* is not selected, the X and Y offsets are applied to a single sub-design. If *Multiple* is selected, the X and Y offsets specify the step-and-repeat spacing as multiple copies of the sub-designs fill the available working area. This is designed to ease the creation of a multi-design panel.

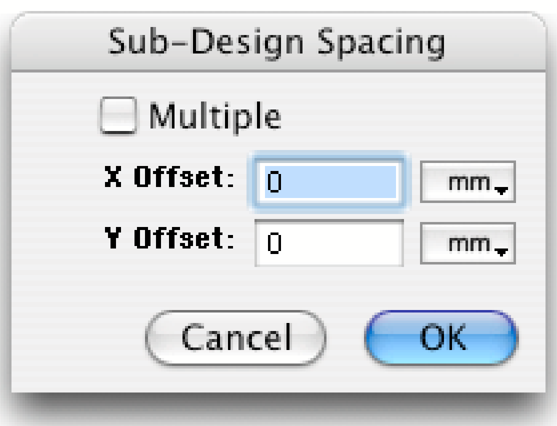


Figure 2.1: Sub-Design Dialog

2.4.7 DXF File...

The **Import** → **DXF File...** command lets you read a DXF file into the current layer of your design. At the present time, only line and text elements in the Entities segment of the DXF file are recognized. Each line element is drawn on the current layer with the current layer's default line width.

2.4.8 PDF File. . .

The **Import** → **PDF File. . .** command allows the first page of any PDF document to be displayed as a faint background image. This background image can serve as a design aid or as design verification. This feature is available in Osmond Quartz only.

2.5 Export

2.5.1 Setup. . .

2.5.2 Library. . .

2.5.3 Part List. . .

2.5.4 Net List. . .

These four commands let you export the current board setup, Library, Part List, and Net List, respectively, to output files. These files are text files and use the formats described in the **File Formats** section of this document. They can therefore be read with the corresponding **Import** commands described above.

2.5.5 Gerber Files. . .

The **Export** → **Gerber Files** command writes all necessary Gerber files into a folder named *Gerber Files* in the same folder as your design file. A new *Gerber Files* folder is created if it does not already exist. The Gerber files conform to the RS-274X format.

When the command is invoked, a dialog appears as shown in Figure 2.2. This dialog allows you to select which layers will be exported and also to specify which (if any) of the layers will be mirror imaged. The manufacturing process usually requires that some layers be mirror imaged. However, many board manufacturers have the ability to perform this mirroring themselves so you may not need to concern yourself with these options.

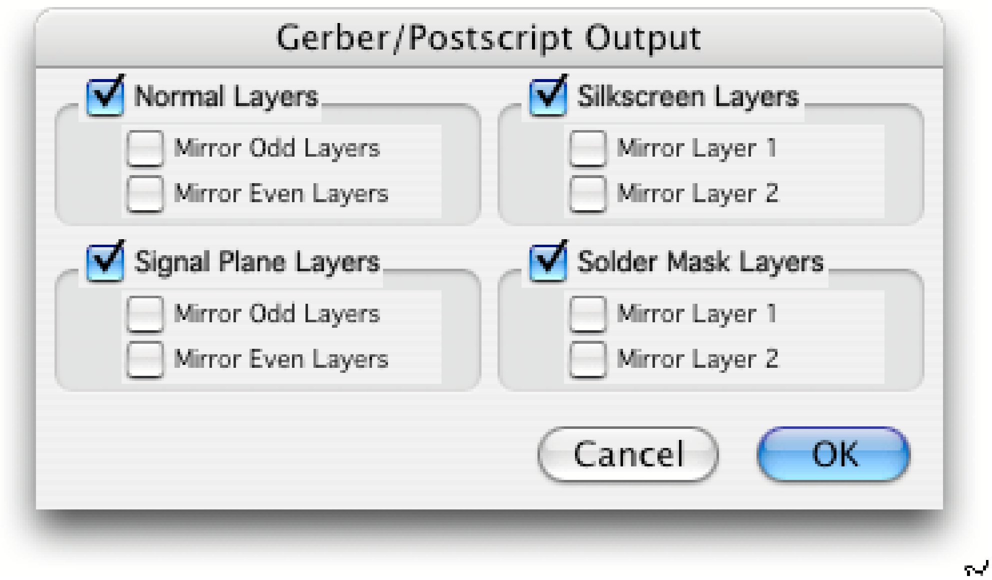


Figure 2.2: Gerber Output Dialog

Note: Make sure there are definitions for all the alphabetic characters in your current library. Osmond uses these when creating the silkscreen Gerber files which usually display part name.

The command also writes an Excellon drill file into the Gerber Files folder as well as a testing file that conforms to the IPC-356 format, and a file that provides pick-and-place data.

2.5.6 DXF Files

The **Export** → **DXF** Files command writes several DXF files into a folder named *DXF Files* in the same folder as your design file. A new *DXF Files* folder is created if it does not already exist. The DXF files show part and pad outlines as well as the position of the holes. These files can be imported into a typical CAD program to produce fabrication and assembly drawings.

This command also writes a **Hole Report** file into the *DXF Files* folder that lists the number and sizes of all the holes used in the design.

2.5.7 PostScript Files...

The **Export → PostScript Files...** command writes Encapsulated PostScript files of all board layers, including silkscreen and solder-mask layers, into a folder named *PostScript Files* in the same folder as your design file. A new *PostScript Files* folder is created if it does not already exist. These files can then be imported into other applications that accept Encapsulated PostScript.

With a printer of sufficient resolution, you may be able to produce transparancies that can be used directly to make your printed circuit boards. 300 dots per inch may be marginal but greater resolution should work well.

When the command is invoked, a dialog appears as shown in Figure 2.3. This dialog allows you to select which layers will be exported and also to specify which (if any) of the layers will be mirror imaged.

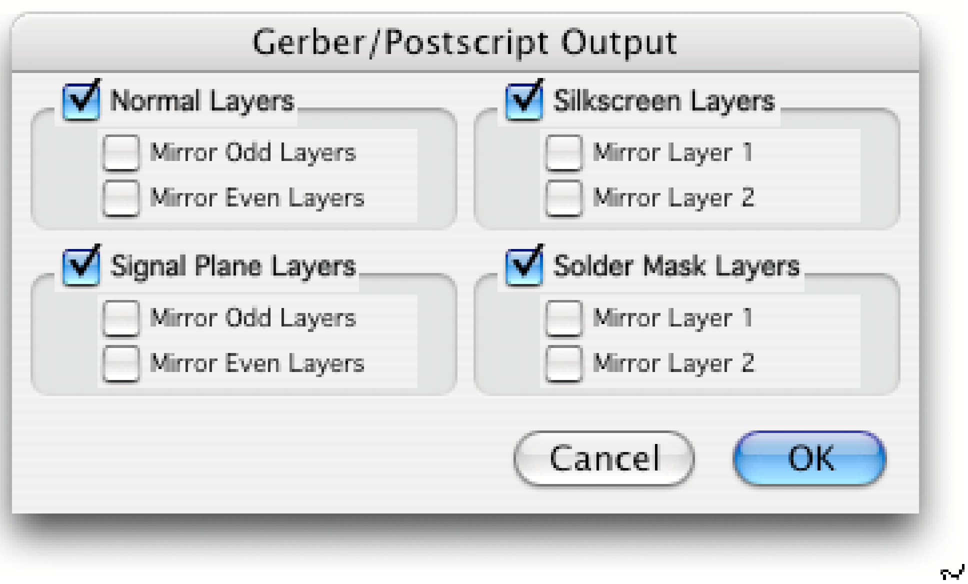


Figure 2.3: Postscript Output Dialog

Note: Make sure there are definitions for all the alphabetic characters in your current library. Osmond uses these when creating the silkscreen PostScript files which usually display part name.

2.5.8 sCAM Files

The Export CAM Files command writes files that can be used by software driving a CNC machine to produce printed circuit boards from copper clad board material by routing isolations between paths. The files are saved in a folder named *sCAM Files* in the same folder as your design file. A new *sCAM Files* folder is created if it does not already exist.

The software that actually drives the CNC machine is **sCAM**(tm), a program for the Macintosh being developed by Neil Gillies. For more information on **sCAM**, please visit Neil's web site at:

http://www.sea-gull.demon.co.uk

2.6 Compare

2.6.1 Part List...

The **Compare → Part List...** command brings up a dialog that allows you to select an external part list file for comparison with the internal part list. An alert will inform you if the internal part list is equivalent to the external part list. However, if the part lists are different, a text file will be generated showing where the two lists differ.

2.6.2 Net List...

The **Compare → Net List...** command brings up a dialog that allows you to select an external net list file for comparison with the internal net list. An alert will inform you if the internal net list is equivalent to the external net list. However, if the net lists are different, a text file will be generated showing where the two lists differ. Note that net lists can be equivalent even if the signal names are different.

2.7 Close

The **Close** command closes the current design. If the design has changed since the last time it was saved, a dialog will appear asking if the design should be saved.

2.8 Save

The **Save** command saves the current design to the same file to which it was last saved. If the design has not yet been saved to a file, a finder dialog will let you specify where the new file should be saved.

2.9 Save As...

The **Save As...** command saves the current design to a new file that you specify in a finder dialog. This file becomes the current design file.

2.10 Save Copy As...

The **Save Copy As...** command saves a copy of the current design to a new file that you specify in a finder dialog.

2.11 Revert

The **Revert** command reverts the current design to that of the last saved file.

2.12 Page Setup...

The **Page Setup...** command lets you change certain print options, the most useful being the orientation of the page.

2.13 Print...

The **Print...** command lets you print the current view of your design.

When you print, the current view is printed, including all visible layers. The current layer is always visible and other layers can be made visible or invisible through the **Layers Visible** window (invoked from the **View** menu). To print all the layers on different pages, go to each layer and print it in turn. In general, what you see on the screen is what you get on the page.

2.14 Quit

The **Quit** command quits the Osmond application. If a design has changed since the last time it was saved, a dialog will appear asking if the design should be saved.

Chapter 3

Edit Menu

3.1 Undo

The **Undo** command reverses the most recent operation including all side-effects that resulted from that operation. When the **Undo** command is invoked, this menu item changes to **Redo** to allow you to undo the **Undo** command.

3.2 Cut

The **Cut** command deletes all selected parts (if possible) and freehand paths and places them in the clipboard. Parts that have connecting paths cannot be cut; you need to remove the connecting paths first. Parts and freehand paths in the clipboard can then be pasted into this or any other design with the **Paste** command.

3.3 Copy

The **Copy** command places a copy of all selected parts and freehand paths into the clipboard. Parts and freehand paths in the clipboard can then be pasted into this or any other design with the **Paste** command.

3.4 Paste

The **Paste** command places all parts and freehand paths that are in the clipboard into the current design. The location and orientation of the parts and freehand paths are the same as they were when the part or freehand path was cut or copied originally. Freehand paths are always pasted into the current layer of the design even if they were on a different layer when they were originally cut or copied.

3.5 Clear

The **Clear** command is not implemented.

3.6 Select All

The **Select All** command selects all entities of the current design. Once selected, these parts and freehand paths can then be moved as a unit, copied to the clipboard, or otherwise manipulated.

3.7 Select Sub Design...

The **Select Sub Design...** command lets you select a particular sub-design, which is a design that has been imported into the current design with the **Import → Sub Design...** command. When sub designs are imported, they are given sequential numbers, starting with 1. This command brings up the dialog shown in Figure 3.1, allowing you to select a particular sub-design by specifying its sequence number. Once selected, the sub-design can be moved as a unit.

3.8 Move Selected...

The **Move Selected...** command lets you move all selected objects by a specified amount. This command brings up the dialog in Figure 3.2 that lets you specify the relative movement in mils or millimeters in both the X and Y direction.

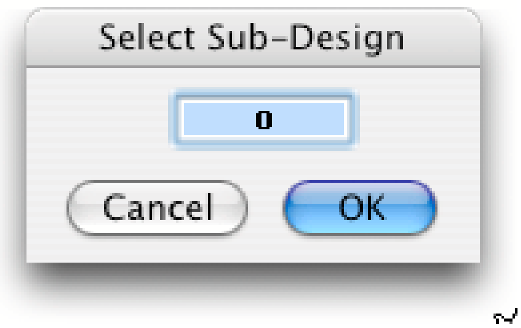


Figure 3.1: Select Sub-Design Dialog

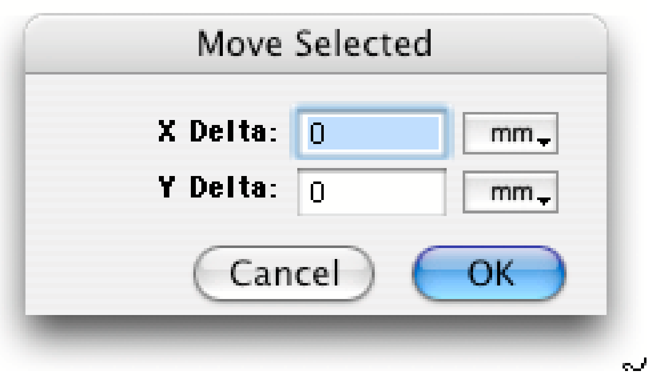


Figure 3.2: Move Selected Dialog

3.9 Design Rules. . .

The **Design Rules. . .** command brings up the dialog in Figure 3.3 to permit you to change options or characteristics of your design.

The **Board Layers** text field lets you specify the number of layers your PC board design should have. This number is the actual number of signal layers and does not include other layers such as silkscreen layers, solder-mask layers, or auxiliary layers.

The **Working Area** text fields lets you specify the horizontal and vertical size of your working area. This is not necessarily the same as the size of you PC board. You may wish to make the working area larger than your PC board to give yourself comfortable margins on all sides. Making the area too large, however, will consume additional memory and may slow the application down.

The **Default Trace Width** text fields let you independently specify the default width of signal paths for both outer and inner layers.

The **Default Spacing** text fields let you independently specify the default spacing of signal paths for both outer and inner layers.

The **Solder Mask Ring** text field lets you specify how much larger the solder mask pads are compared to the pads they enclose. A value of 10 mils, for example, will result in a solder mask pad that is 10 mils larger than the enclosed pad in every direction. Note that all the measurements can be specified in either millimeters or mils.

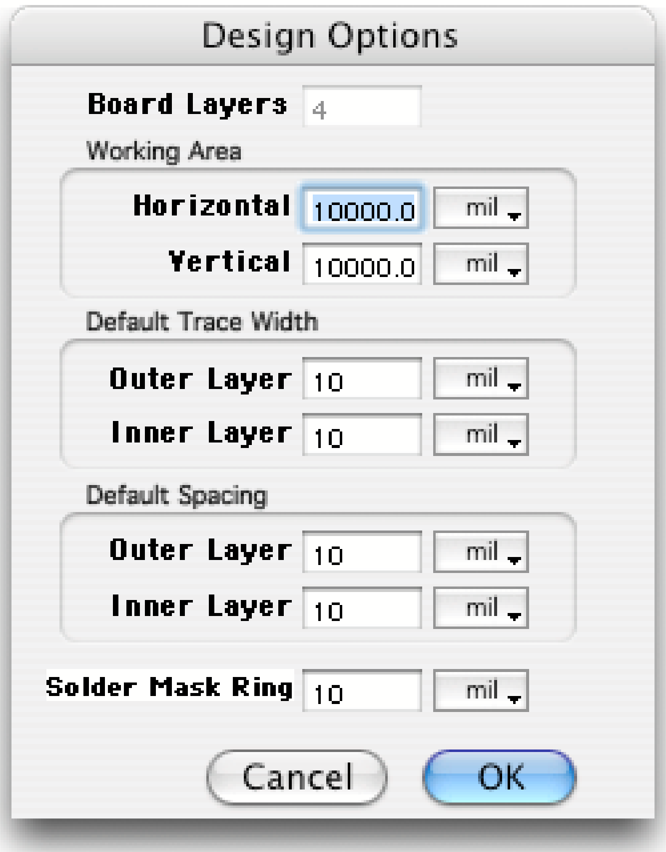


Figure 3.3: Design Options Dialog

Chapter 4

View Menu

4.1 Actual Size

The **Actual Size** command scales the view so that all items on the display are at their actual size, assuming a standard display of 72 dots per inch.

4.2 Double Size

The **Double Size** command scales the view so that all items on the display are at twice their actual size, assuming a standard display of 72 dots per inch.

4.3 Full View

The **Full View** command scales the view so that the entire design panel is visible.

4.4 Zoom In

The **Zoom In** command magnifies the view by an amount equal to the current zoom factor. The center of the view is kept at the center. To change the current zoom factor, double-click the **Zoom Tool**.

4.5 Zoom Out

The **Zoom Out** command de-magnifies the view by an amount equal to the current zoom factor. The center of the view is kept at the center. To change the current zoom factor, double-click the **Zoom Tool**.

4.6 Full Width Traces

The **Full Width Traces** command lets you toggle between displaying all paths at their full width, and displaying only the center line of all paths.

4.7 Solid Pads

The **Solid Pads** command lets you toggle between displaying all pads as an outline, and displaying pads filled in with color. The color can be changed with the **Change Color → Pads...** menu command below.

4.8 Trace Outlines Only

The **Trace Outlines Only** command lets you toggle between displaying only the outlines of paths, and displaying the paths completely filled in. This is apparent only when the paths are shown at their full width (see **Full Width Traces** command above).

4.9 Show Holes

The **Show Holes** command lets you toggle between displaying or not displaying holes in the board. The color used to show holes can be changed with the **Change Color → Hole...** menu command below.

4.10 Show Names

The **Show Names** command lets you toggle between showing or not showing part name text on the display. If part name text is shown, it is also generated when printing and when producing *Gerber* and *PostScript* files. If part name text is not shown, it is not generated when printing or when producing *Gerber* or *PostScript* files.

4.11 Show Values

The **Show Values** command lets you toggle between showing or not showing part value text on the display. If part value text is shown, it is also generated when printing and when producing *Gerber* and *PostScript* files. If part value text is not shown, it is not generated when printing or when producing *Gerber* or *PostScript* files.

4.12 Show Pin Names

The **Show Pin Names** command lets you toggle between showing or not showing pin names on the display near each pin. This is for reference only. Pin names are not generated when producing *Gerber* or *PostScript* files.

4.13 Vector Text

Historically, part names and part values are shown on screen using system fonts. However, when *Gerber* files or *Postscript* files are generated, part names and part values are made with the vector fonts that are defined in the parts library. This means that the text that you see on the screen is only an approximation of the text that appears on the board.

When **Vector Text** is selected, the vector fonts in the parts library are used to display the part names and part values on the screen. This way, what you see on the screen should match exactly what appears on the board. Of course, this works only if the vector fonts already exist in the parts library.

4.14 Show Part Origin

The **Show Part Origin** command lets you toggle between showing or not showing a small target located at the origin of each part. When this target is shown, it can be used as a handle to move, rotate, or flip the part using the appropriate tools.

4.15 Default Metric

The **Default Metric** command lets you toggle between the default Imperial and the default Metric state. In the default Imperial state, dialogs preferentially show units in mils while in the default Metric state, dialogs preferentially show units in millimeters. Also, in the default Imperial state, the cursor coordinates shown in the message line are in mils while in the default Metric state, the cursor coordinates are in millimeters.

4.16 Change Color

Background...

Path...

Pad...

Imperial Grid...

Metric Grid...

Un-plated Hole...

Plated Hole...

These commands allow you to change, respectively, the color of the background, the color of paths on the current layer, the color of filled-in pads, the color of the grid if it is in imperial units, the color of the grid if it is in metric units, and the color used to show un-plated and plated holes.

4.17 Layers Visible...

The **Layers Visible...** command brings up a dialog like that shown in Figure 4.1 that lets you specify which layers are always visible. In the Osmond system, the current layer is always visible. This dialog lets you select which layers, along with the current layer, are also always visible. By default, the front and back silkscreen layers and the front and back auxiliary layers are always visible.

The colors used for the layer names reflect the color of the paths on the given layer. These colors can be changed with the **Change Color...** command below.

The layers that are always visible are indicated by a mark at the left of each item. To change the visibility of the item, click on the layer name. You can also toggle

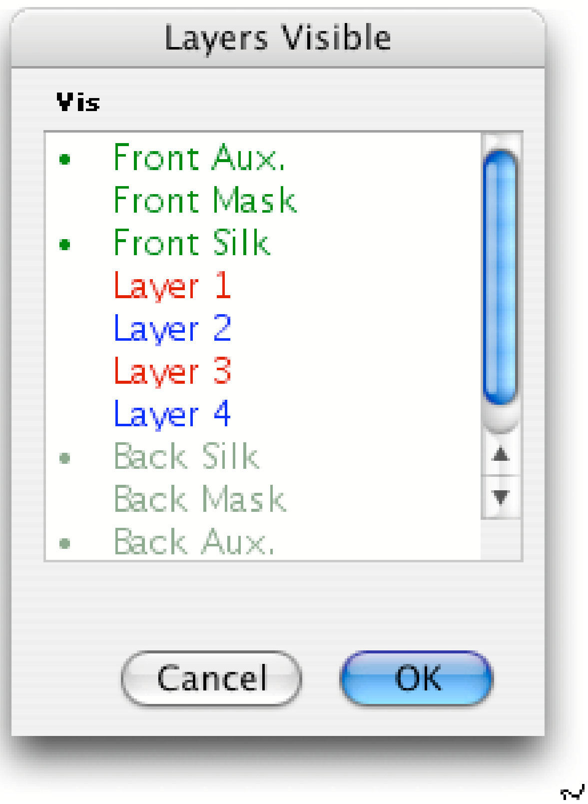


Figure 4.1: Layers Visible Window

the visibility of a signal layer by pressing the layer number on the keyboard while holding down the **option** key.

Chapter 5

Grid Menu

5.1 Imperial Units

The top portion of the menu contains common grid spacings in Imperial Units. Here you will find **5 mils**, **10 mils**, **25 mils**, **50 mils**, and **100 mils**.

5.2 Metric Units

The next portion of the menu contains common grid spacings in Metric Units. Here you will find **0.5 mm**, **1 mm**, and **2 mm**.

5.3 Snap to Grid

The **Snap to Grid** toggle command lets you turn on and turn off the **Snap to Grid** feature. When **Snap to Grid** is on, moved items always snap to the nearest grid crossing.

5.4 Custom...

The **Custom...** menu item brings up the dialog as shown in Figure 5.1 that lets you directly specify the visibility and the spacing of the grids. If you specify Equal X and Y Spacing, only one text field will be available as shown.

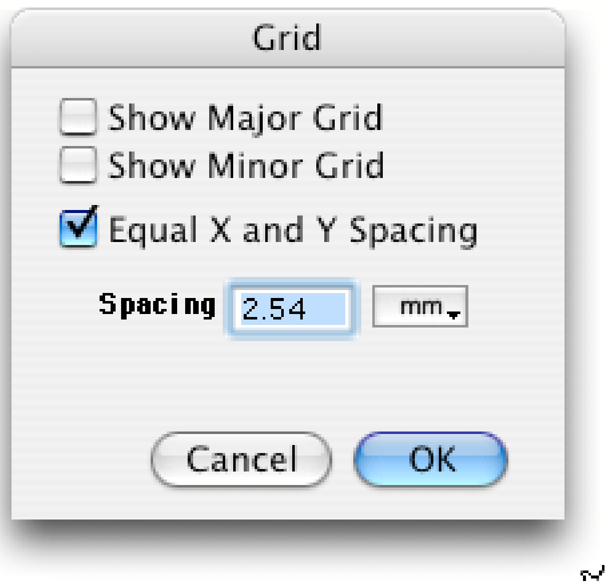


Figure 5.1: Grid Dialog with Equal X and Y Spacing

If you do not specify Equal X and Y Spacing, two text fields will be available as shown in Figure 5.2 to let you independently specify both the horizontal and vertical grid spacing.

Using the drop menu next to each edit field, you can specify the grid spacing in either millimeters or mils. See Figure 5.3

When different grid spacings are produced with the **Custom...** command, they are then added automatically to the bottom of the **Grid** menu.

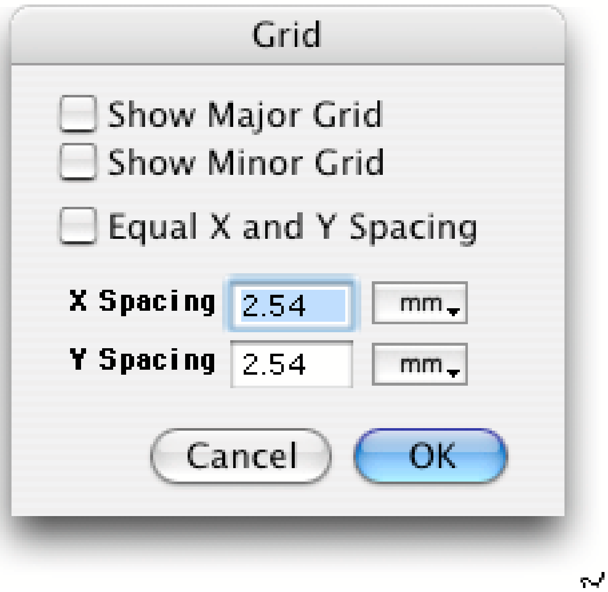


Figure 5.2: Grid Dialog with Unequal X and Y Spacing

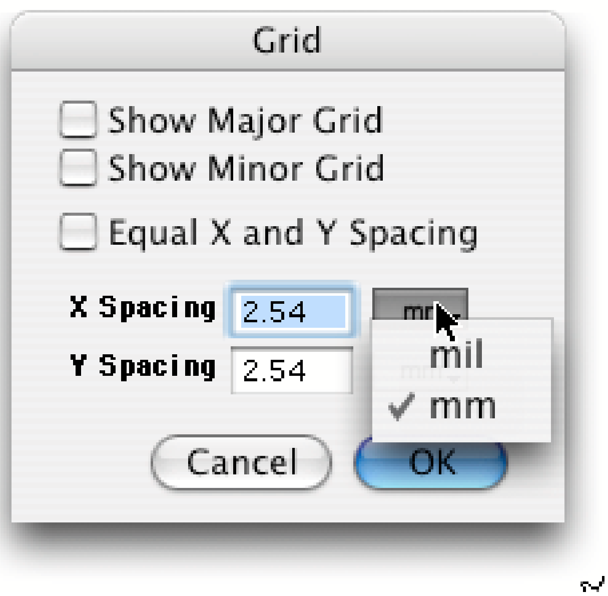


Figure 5.3: Grid Dialog - mils or mm

Chapter 6

Part Menu

6.1 Find Part...

The **Find Part...** command lets you find a part in your design. A dialog appears as shown in Figure 6.1.

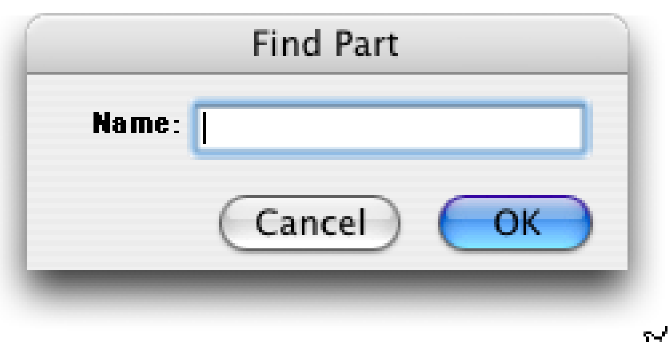


Figure 6.1: Find Part Dialog

When you enter the part name and press **OK**, the part is selected and centered in the view. If the part is in the deferred list, an alert will so indicate. You will also see an alert if the part is not found.

6.2 Group...

The **Group...** command groups all selected parts and freehand paths into a single part. A dialog is brought up as shown in Figure 6.2 that lets you give the new part a name if you wish, and to specify the height of the name that will appear on the silkscreen layer. If you do not wish to have the name of the part displayed, you can leave the Height at 0.

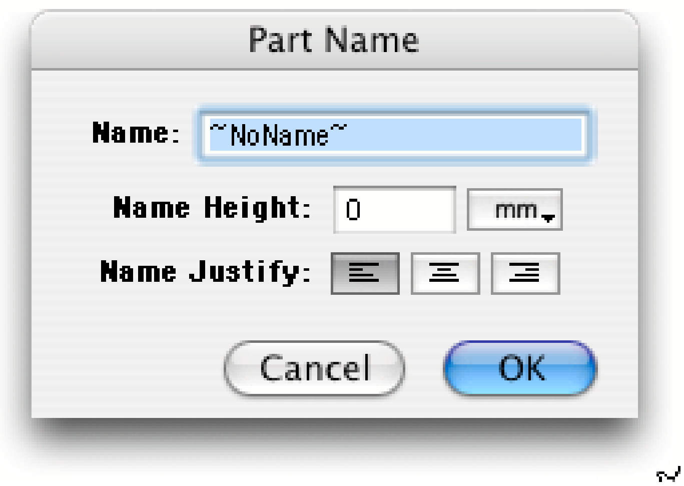


Figure 6.2: Part Name Dialog

The origin of the new part is determined by looking at the origin of the grid. If the grid origin coincides with one of the pads in the new part, or with any of the vertices of the paths in the new part, the origin of the part will be set to the grid origin. Otherwise, the origin of the part will be set to the bottom left corner of an imaginary rectangle that contains the center point of all pads and all path vertices. Therefore, if you want the origin of the part to be at the center of a certain pad, set the origin of the grid to that pad using the **Index** tool before you group.

If you group parts that each contain a single pin, these pins are simply added to the new part.

Note: The **Make New Part Type...** command in the **Parts** menu now provides a much easier method of creating new parts than using the **Group...** command. Consequently, the **Group** and **Ungroup** commands may be removed in upcoming releases and should be considered deprecated.

6.3 Ungroup

The **Ungroup** command breaks the selected part into its constituent parts and freehand paths. If a part had been previously grouped with the **Group...** command, the **Ungroup** commands reverses the process. A part containing pins will be broken up such that each pin becomes a part with a single pin. To use this command, only one part should be selected.

Note: The **Make New Part Type...** command in the **Parts** menu now provides a much easier method of creating new parts than using the **Group...** command. Consequently, the **Group** and **Ungroup** commands may be removed in upcoming releases and should be considered deprecated.

6.4 Lock

The **Lock** command locks all selected parts. Locked parts cannot be moved, flipped, or rotated.

6.5 Unlock

The **Unlock** command unlocks all selected parts. Once a part is unlocked, it can be moved, flipped, and rotated.

6.6 Defer

The **Defer** command moves all selected parts that have no connections to the deferred list. You can see a list of the deferred parts by opening the **Deferred Parts** window from the **Window** menu.

6.7 Align

Horizontally

Vertically

The **Align** → **Horizontally** and **Align** → **Vertically** commands align selected parts horizontally or vertically, respectively, with respect to their origins. If one of the parts is locked, the unlocked parts will be moved to align with the locked part. If none of the parts is locked, parts will be aligned with the part nearest the top of the screen (for **Align** → **Horizontally**) or the left most part (**for Align** → **Vertically**). If several parts are locked, the unlocked parts will be aligned with the locked part nearest the top of the screen (for **Align** → **Horizontally**) or the left most locked part (for **Align** → **Vertically**). In no case will locked parts be moved.

Left Pins

Right Pins

Top Pins

Bottom Pins

These commands align the selected parts horizontally or vertically, with respect to their left-most, right-most, top-most, or bottom-most pins, respectively. If one of the parts is locked, the unlocked parts will be moved to align with the locked part. If none of the parts is locked, parts will be aligned with the part nearest the top of the screen (for Align Left or Right) or the left most part (for Align Top or Bottom). If several parts are locked, the unlocked parts will be aligned with the locked part that is nearest the top of the screen (for Align Left or Right) or the left most locked part (for Align Top or Bottom). In no case will locked parts be moved.

6.8 Align Name

Left

Center

Right

These commands align the name field of all selected parts within the part itself. For example, the **Align Name** → **Left** command moves the name field of the part to the left edge of the part's silkscreen. The command **Align Name** → **Center**, which immediately follows, moves the name field so that it is centered horizontally with respect to the silkscreen. Similarly for the the **Align Name** → **Right** command.

Top

Center

Bottom

These three commands work in a similar manner, only moving the name vertically instead of horizontally.

Calculations are performed using the left-most, right-most, top-most, and bottom-most points of the silkscreen. If a part has no silkscreen, this command has no effect.

6.9 Update All Parts

The **Update All Parts** command scans the part list and updates those parts that do not yet have a part type defined. It also updates those parts that have had their part type modified.

If you have imported a part list that has parts whose part types are not in an open library, you can use this command to update these parts after the part type is made available. A part type can be made available either by importing a library that contains the part type, opening a file that contains the part type, or by creating the part type in the part type editor with the **Make New Part Type...** command in the **Parts** menu.

6.10 Save to Library...

This command writes a description of the currently selected part to the internal library. You will typically use this command after building a part with the **Group** command. A dialog like that shown in Figure 6.3 appears to let you specify the name of the new part type. Once a part type has been saved to the library, you can create instances of the part by opening up the **Library** window from the **Window** menu, and dragging and dropping from the Library part list, or by using the **New Part** tool. To use this command you should have only one part selected.

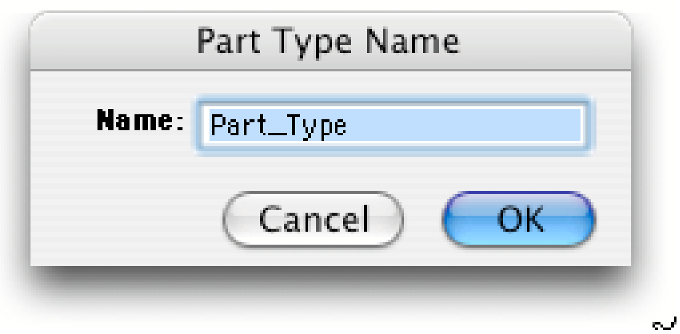


Figure 6.3: Part Type Name Dialog

6.11 Rename Parts...

This command lets you rename and/or change the name size and justification of each of the currently selected parts. For each part selected, a dialog like that shown in Figure 6.4 will appear showing the name of the part and the height of the name text, both of which you can change. The height can be specified in either mils, mm, or points.

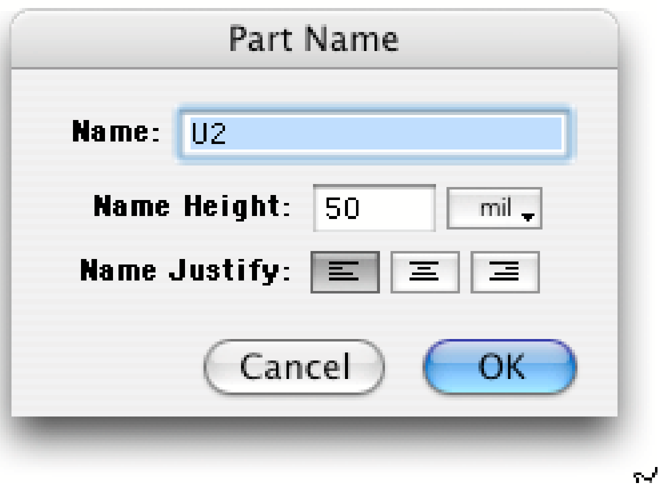


Figure 6.4: Part Name Dialog

You can also change the justification (left, center, or right) of the part name.

6.12 Revalue Parts...

This command lets you change the value of selected parts. You can also change the height and justification of the Value text. For each part selected, a dialog like that show in Figure 6.5 appears showing the name of the part, the current value, and the value text height and justification. The height can be specified in either mils, mm, or points.

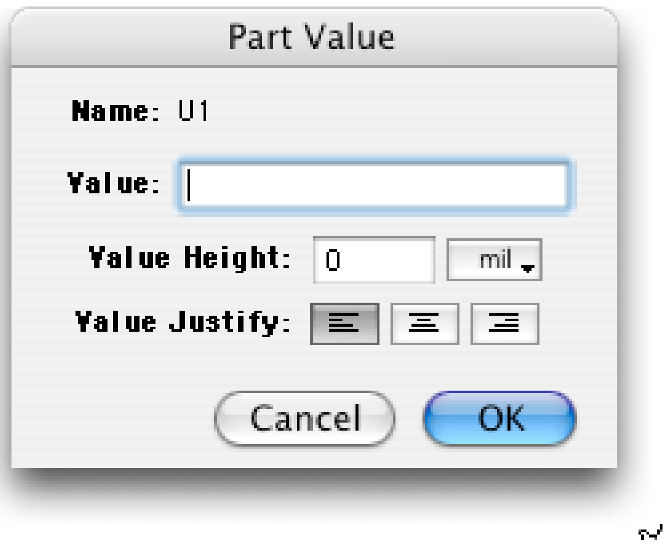


Figure 6.5: Part Value Dialog

6.13 Replace Parts. . .

This command lets you replace each of the currently selected parts with a different part type.

The dialog as shown in Figure 6.6 appears in response to this command. The dialog has two scrolling lists. The first list shows all the selected parts, including the part name, the part type, and the part value if it exists. The second list shows all the part types in the current library.

To replace a part, select some subset of parts in the first list using normal Mac selection methods, then click on the desired replacement part type in the second list, and then click on the **Replace** button. The part type field in the first list is updated and each selected entry is underlined to indicate that this part has been changed. You will notice as you click on either the part in the first list or the part type in the second list, that a small preview of the part or part type is drawn at the right.

Parts whose part types are not yet defined will be shown in **bold** in the first list.

A popup menu at the top right allows you to select the library from any open design.

Once you are satisfied with all the replacements, you can finalize the command

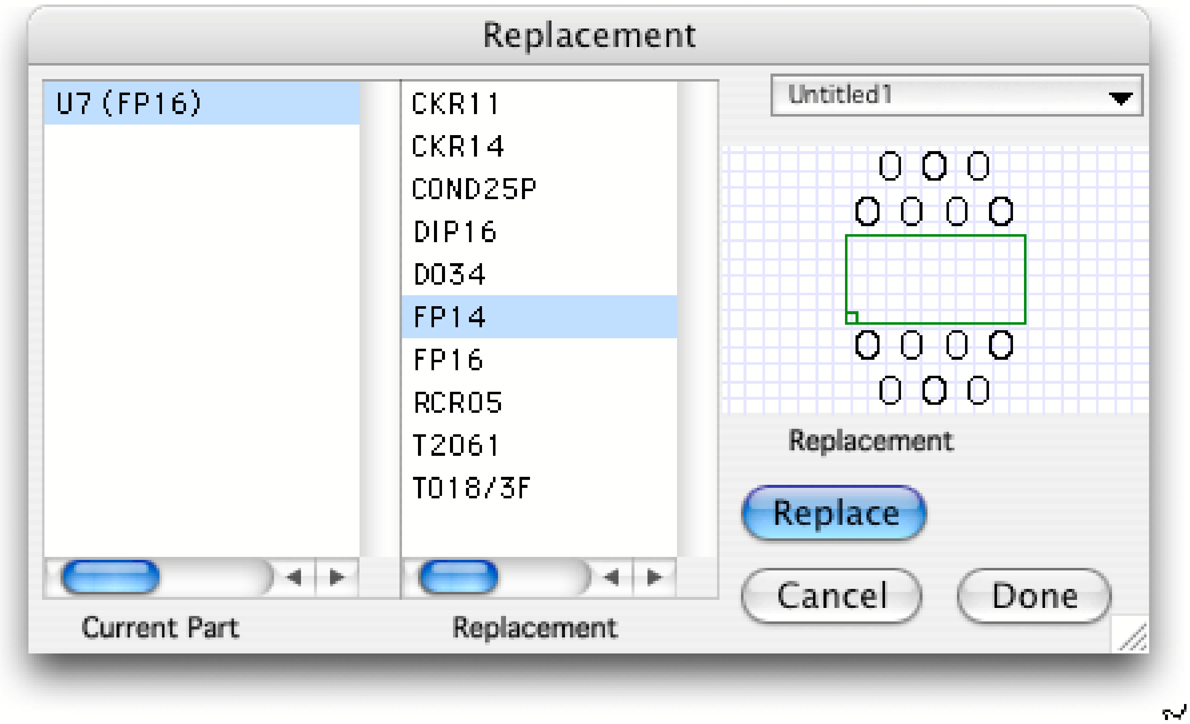


Figure 6.6: Replace Parts Dialog

by clicking the **Done** button. Clicking the **Cancel** button will leave all the parts unchanged.

6.14 Replace Deferred Parts. . .

This command lets you replace all the part types of all parts in the **Deferred List**. **Deferred List** parts are parts that have not yet been placed on the board.

When this command is invoked, a Replacement dialog appears like the one shown for the **Replace Parts. . .** command above. The only difference is that the first list is simply a list of all Deferred Parts.

You can use this command immediately after importing a new Part List to replace all the parts whose part types are not defined in the library. For example, if the part type in Part List is "generic_2pin", which is not defined in the library, you can replace this part with "RCR05" which is defined in the library.

6.15 Part Attributes...

This command brings up a dialog as shown in Figure 6.7 that lets you view and modify part attributes such as position, orientation, name, and value. A list of all selected parts is shown on the left. As you click on each item on the list, the associated attributes are shown on the right. You may modify any of the attribute for each part. When you are satisfied, hit **OK** to apply all the changes. If you hit **Cancel**, none of the changes will be made.

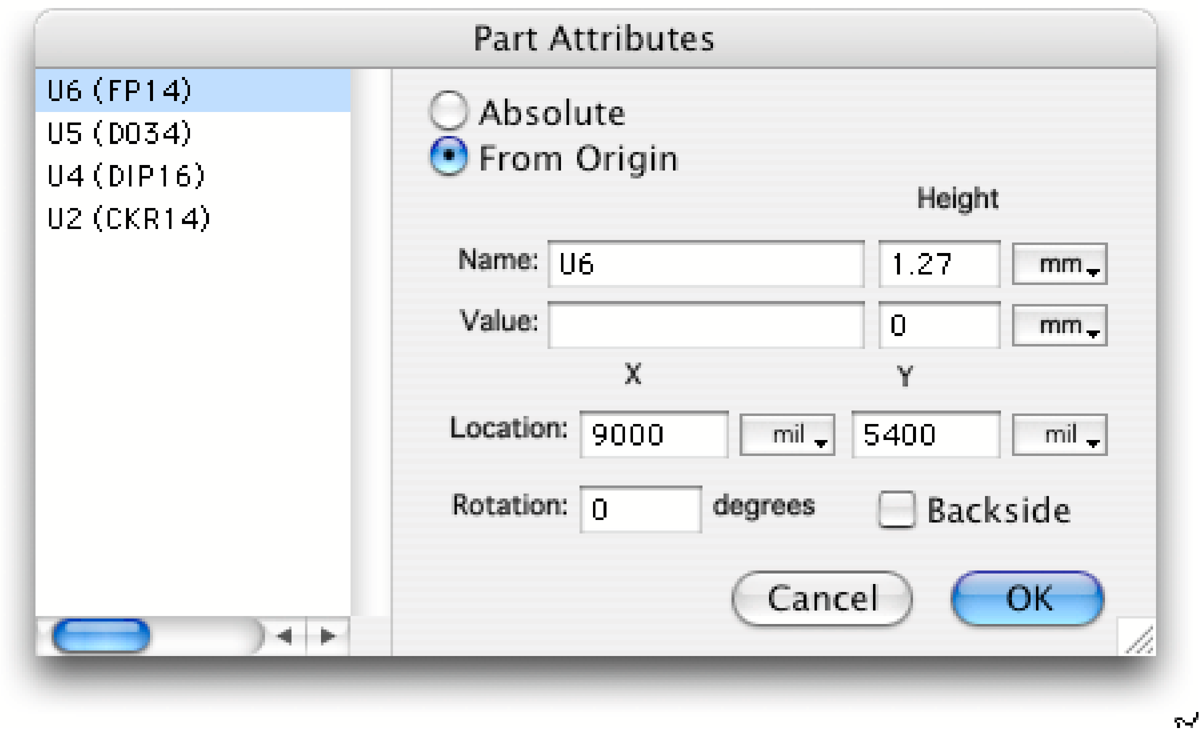


Figure 6.7: Part Attributes Dialog

6.16 Edit Parts. . .

This command lets you edit each of the currently selected parts in a new window. For each part selected, a special editing window appears as shown in Figure 6.8:

A special subset of the usual toolbox tools is at the top of the window. Tools that are not really applicable to part editing are not included. In addition, the buttons, **Save as. . .** , **Cancel** and **OK**, appear at the bottom of the window. The selected part appears centered in the window. However, this part has been dis-assembled such that each pin is now an independent part and each path is now an independent path. You can, therefore, completely change the part using many of the normal toolbox tools.


To change the position of a pin, use the **Move Parts** tool. You can also use this tool to change the position of the Part Name or the Part Value.


To change the silkscreen path, go to the Front Silk layer and use the **Drag/Remove Pegs** tool.


If you wish to add a new path to any layer, use the **Draw Freehand Path** tool. Make sure you first go to the correct layer.


To change the pin name, select the pin with the **Select** tool and then use the **Rename Parts. . .** command from the **Parts** menu.


The origin of the part is specified by the Grid Index. Therefore, if you wish to change the origin of the part, change the grid index using the **Locate Index** tool.


To change the pad-stack of a pin, use the **Change Pin/Padstack** tool.


To add pins, use the **New Pin** tool.

To delete a pin, select it with the **Select** tool and then delete it with the **delete** button on the keyboard or the **Cut** command in the **Edit** menu.

You can get more information on all the available tools in the **Tools** and **More Tools** sections of this document.

Once you are satisfied with the changes you have made, press the **OK** button. The part being edited is automatically re-assembled and the original selected part is replaced with this new part. You can also press the **Cancel** button to leave your original part unchanged.

If you press the **Save As...** button, a dialog appears (Figure 6.9) that lets you define the name of a new part type to add to the library.

6.17 Make New Part Type...

This command lets you create a new part type in a new window and add it to the current library. When you invoke this command, a dialog first appears (Figure 6.10 asking for the name of the new part type, the height and justification of the Name text, and the height and justification of the Value text. The Name and Value text normally appear on the silkscreen layer. If you do not wish the Name or Value text to appear, make the corresponding height zero.

A new part edit window then appears as shown in Figure 6.11. The grid origin, which corresponds with the origin of the part, is placed at the center of the window. The name field is also placed at the center. If a non-zero Value Height is specified, the value appears below the Name.

A subset of the normal toolbox tools is at the top of the window. Use these tools to create your new part. You will likely make extensive use of the **New Pin** tool, the **Draw Freehand Path** tool, and the **Move Parts** tool. See the **Edit Parts...** command above for a description of each of these tools.

When you have finished designing your part, click the **OK** button. This adds the part to the current library. Once the part is in the library, you can add instances of this part to your design by dragging it from the **Library Window** or by using the **New Part** tool.

6.18 Text Characters

You can use the **Make New Part Type...** command to create or modify the default text characters that are used with the **Text** tool and that are used when creating *Gerber* text in the **Silkscreen** layers.

Text characters are similar to other part types but are treated in a special way by the Osmond system. To allow Osmond to recognize a character part, it is named a special way. For example, the character A is named: 'A'. That is, a single quote followed by the letter itself followed by a single quote.

Since text can be any size, Osmond can scale the character parts by any amount as needed. To simplify the scaling, Osmond assumes (for purely historical reasons) that the height of each character, as it is defined in the library, is 14 mils tall and that the origin of the character is as described below. Newly defined characters, therefore, should remain consistent with these assumptions.

To make a character A, for example, issue the **Make New Part Type...** command and fill in the fields as shown (Figure 6.12).

When the **Part Edit** window appears, change the grid size to 1 mil and zoom in until the grid is visible. Using the **Draw Freehand** tool, draw the character as you wish and click **OK** when done. You will probably need to change the width of the **Freehand** tool to 2 mils or less. A view of the character A as it is defined in the library is shown in Figure 6.13.

Notice that the left edge of the character (including stroke width) is 1 mil to the right of the origin, the bottom of the character is 1 mil below the origin, the top of the character is 13 mils above the origin, and the width of the character is 10 mils. New characters should be consistent with these dimensions to work well with existing characters. Existing characters use a stroke width of 2 mils but you should be able to use any stroke width you choose.

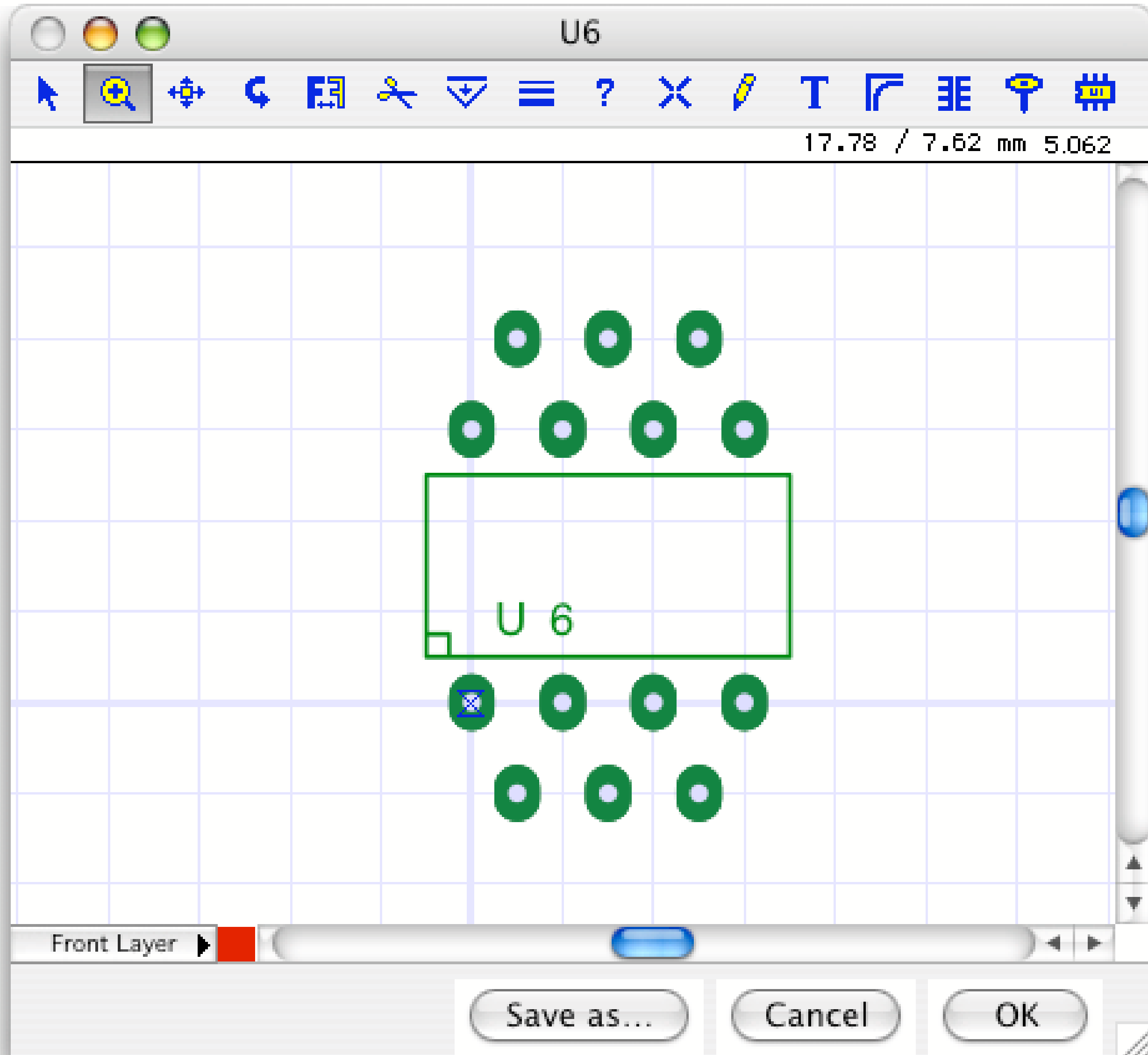


Figure 6.8: Edit Part Window

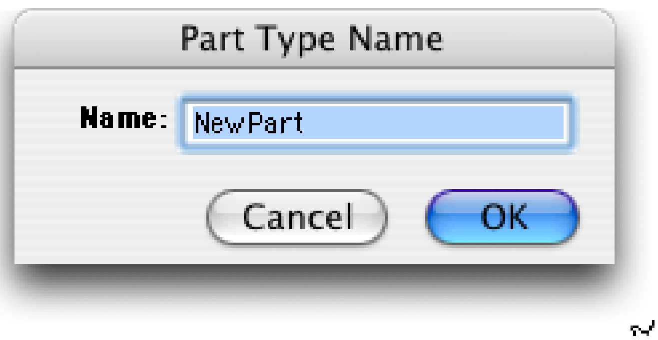


Figure 6.9: Part Type Name of New Part

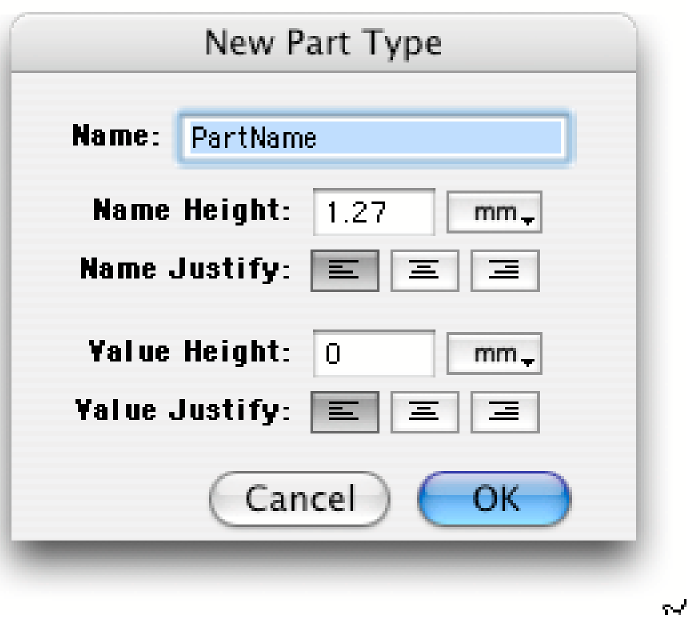


Figure 6.10: New Part Type

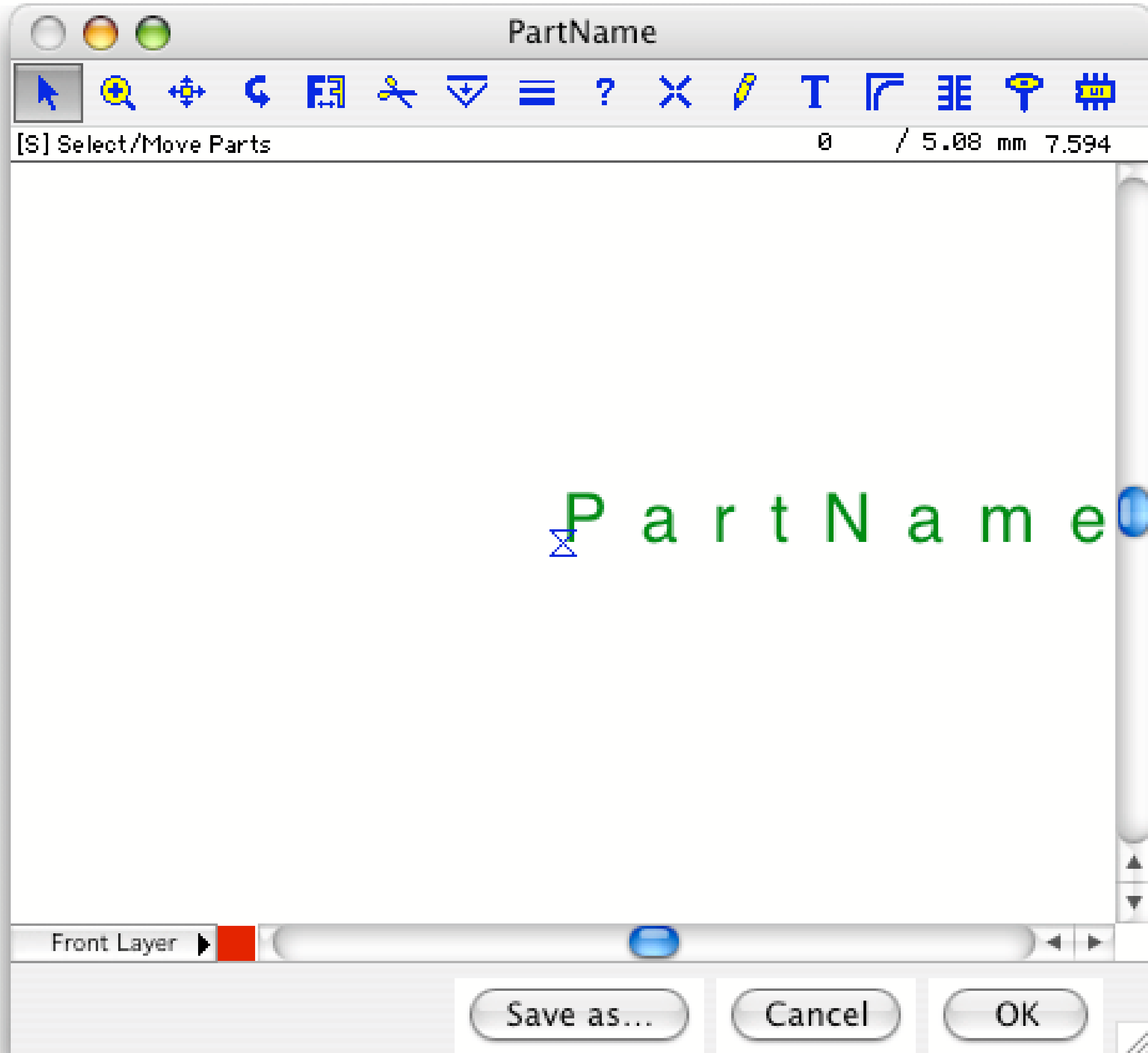


Figure 6.11: Edit Part Window

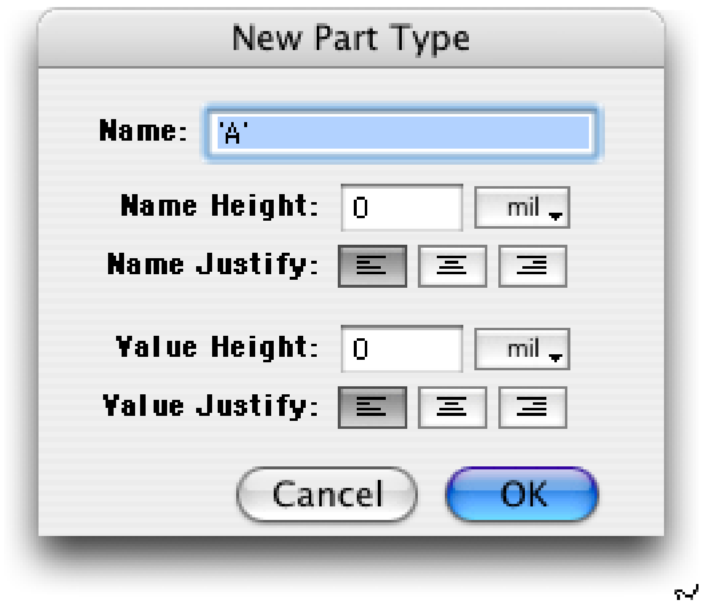


Figure 6.12: New Part Type Dialog

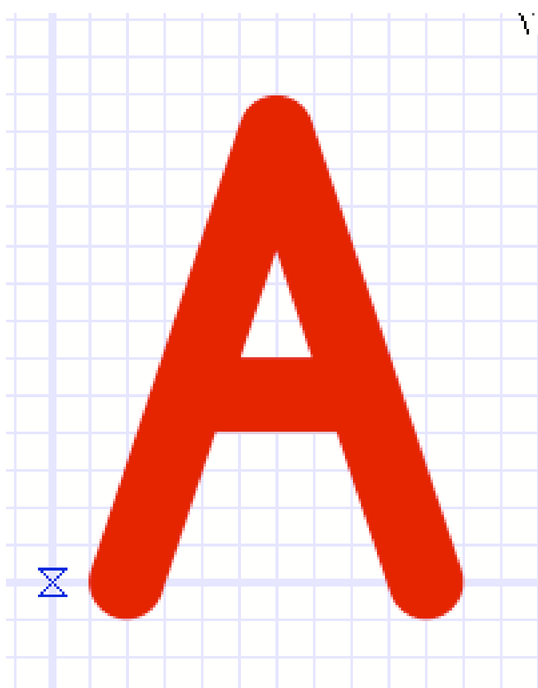


Figure 6.13: Creating a Character Part

Chapter 7

Design Menu

7.1 Check Current Layer

This command causes the Osmond system to perform a clearance check of all traces on the current layer. If any clearance conflicts are found, the affected paths and/or pads are highlighted and the dialog as shown in Figure 7.1 appears. If necessary, the view will be panned to bring the conflict into view.

Figure 7.1: Conflict Check

If you press **No**, the conflict will be skipped and the next conflict (if any) will be found and highlighted. If you press **Quit**, the check will be aborted and the dialog will disappear. If you press **Auto**, the Osmond system will attempt to resolve all conflicts automatically by moving connecting paths without further dialog. If you press **Yes**, the Osmond system will attempt to find a solution to the conflict and will display that solution highlighted while changing the dialog as shown in Figure 7.2:

Figure 7.2: Conflict Solution

If you press **Yes**, the proposed solution will be implemented and the system will move on to the next conflict (if any). If you press **No**, the proposed solution will not be implemented and the system will attempt to find and present a different solution for your approval. If you press **Auto**, the proposed solution will be implemented and the system will attempt to resolve all remaining conflicts automatically without further dialog. If you press **Quit**, the proposed solution will not be implemented, the dialog will disappear, and all remaining checks will be aborted.

You can use keyboard shortcuts instead of pressing buttons to respond to the dialog as follows:

Yes	**Y** or **return**
No	**N**
Auto	**A**
Quit	**Q** or **esc**

7.2 Check All Layers

The **Check All Layers** command is similar to the **Current Layer** command above. The **All Layers** command, however, performs a clearance check of all layers of the design.

7.3 Find Overlapping Holes

The **Find Overlapping Holes** command searches the design for two or more holes that overlap. This is usually the result of an error in the design and should normally be corrected. If a pair of overlapping holes is found, the view is moved to position the holes exactly in the center. You may need to increase the magnification to make it more obvious which holes are exactly in the center. If there are no overlapping holes, an alert will so indicate.

7.4 Rename Signal. . .

The **Rename Signal** command brings up the dialog as shown in Figure 7.3 to allow you to rename the current signal.

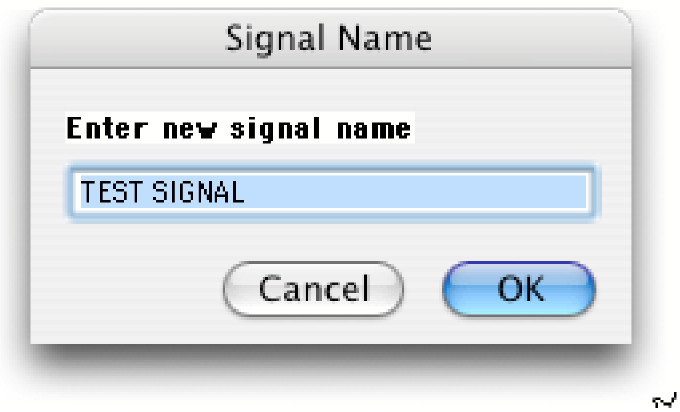


Figure 7.3: Rename Signal

7.5 Make Signal Plane. . .

The **Make Signal Plane** command brings up the dialog as shown in Figure 7.4 to allow you to specify whether or not the current layer is a (reversed image) signal plane layer, and if it is a signal plane layer, to specify various initial values.

If you wish the current layer to be a signal plane layer, check the **Signal Plane** check box. If the current layer is already a signal layer and you wish to change it back to a normal layer, uncheck the **Signal Plane** check box.

The Osmond system uses patterns of reverse image pads and paths to control whether or not a pin is connected to the ground or signal plane. In Figure 7.5, the reddish area shows where copper remains on the layer while the white area shows where copper is

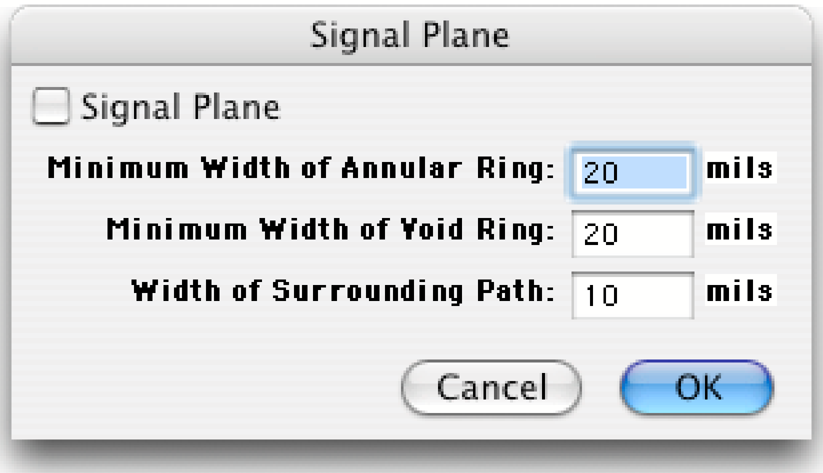


Figure 7.4: Signal Plane Dialog

removed from the layer. Gray circles show where there is a hole for a through-hole pin.

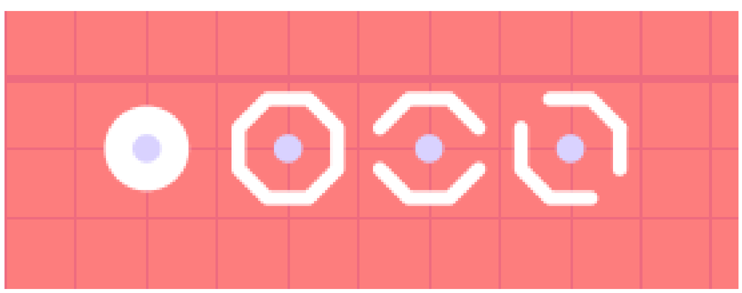


Figure 7.5: Reverse Image Pattern

In the first pattern, a reverse image pad (called a void pad) removes the copper from around the through-hole pin. This causes the pin to be completely isolated from the copper plane. The **Minimum Width of Void Ring** field allows you to specify the minimum width of the copper-free area surrounding the hole.

In the second pattern, eight reverse image paths surround the hole leaving an area of copper in an octagon shaped annular ring. The pin connects to the copper in the annular ring but is still isolated from the copper plane. The **Minimum Width of Annular Ring** field allows you to specify the minimum width of the octagon shaped copper ring surrounding the hole, while the **Width of Surrounding Path field** allows you to specify the width of the reverse image paths surrounding the ring.

The third pattern is like the second except two of the surrounding paths have been removed. Now, the pin is connected to the copper in the annular ring and also connected to the copper plane through passages on the left and right hand side of the pin. If we were to remove all the surrounding paths, the pin would still be connected to the copper plane but soldering the pin would be more difficult because of the large heat sink provided by the copper. The surrounding paths therefore act as thermal barriers to permit easier soldering.

The fourth pattern is similar to the third except two different surrounding paths have been removed to provide diagonal connecting passages to the copper plane.

When you make a signal plane on an inner layer, all pads belonging to the current signal will be converted to the second pattern while all pads that do not belong to the current signal will be converted to the first pattern. On an outer layer, however, all pads are converted to the second pattern.

For example, to make a ground plane on an inner layer, go to that layer and select the ground signal with the **Connect** tool by clicking on any pad that belongs to ground. Then make the layer a signal layer with the **Make Signal Plane...** command. Now you can connect any of the ground pads to the copper plane by using the **Thermal** tool to convert pads from the second pattern to the third pattern or fourth pattern or any other pattern that you choose.

7.6 Make Moat

The **Make Moat** command requires that one (and only one) closed path be selected. A closed path is a path in which the beginning of the path is at the same point as the end of the path.

The **Make Moat** command creates a pattern of closed freehand paths that completely surround all pads and traces within the currently selected closed path. This command is usually a prelude to the **Paint** command described below.

For example, given two pads connected by a path within a closed path as shown in Figure 7.6.

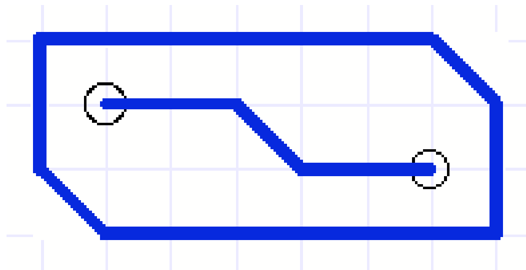


Figure 7.6: Path Surrounded by Closed Path

If we select the enclosing path we can now invoke the **Make Moat** command to produce Figure 7.7.

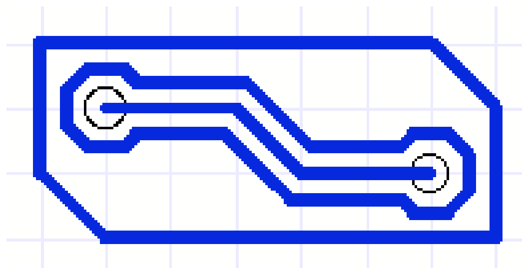


Figure 7.7: Make Moat

The widths of all new paths are the same as the width of the original selected path, and the spacings permitted by the new paths are equal to the spacing of the original selected path.

7.7 Paint

The **Paint** command allows you to fill an area with copper. The area to be filled is defined by a closed path. However, any closed areas within the selected closed area will be avoided.

Let us take, for example, the circuit pattern below consisting of a closed freehand path surrounding another closed path which in turn surrounds two pads connected by a path (Figure 7.8).

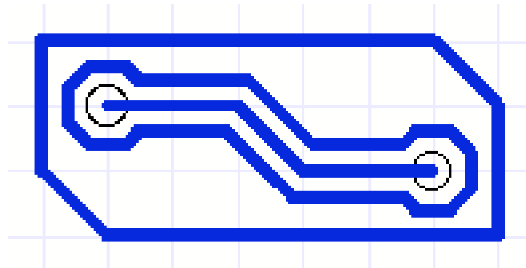


Figure 7.8: Closed Path with Moat

If we select the outer path, we can now invoke the **Paint** command to produce Figure 7.9.

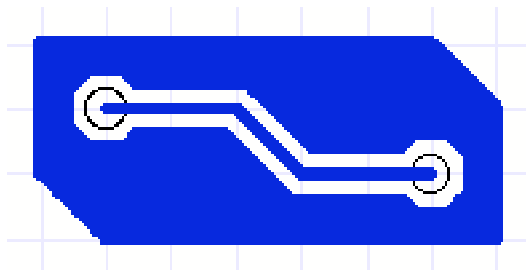


Figure 7.9: Using Paint

Paint works by drawing horizontal paths with the same width and spacing as the surrounding path. The spacing between paths is determined by the width and spacing of the surrounding path. To produce solid copper, the spacing of the surrounding path should be set to zero. If the spacing is not set to zero, you may get gaps between paths.

Make Moat and **Paint** represent historically early approaches to creating local ground plane areas. For many situations, however, you may be able to achieve superior effects by using the newer **Copper Flood** and **Selected Paths Flooded** commands described below.

7.8 Copper Flood

The **Copper Flood** command is a toggle command that turns on and off **Copper Flood** for the current layer. A layer with **Copper Flood** on is solid copper except for clear space surrounding each pad and each path. The amount of clear space surrounding each pad is determined by the default spacing of the current layer, modified by the spacing adjust of the pad itself. The amount of clear space surrounding each path is determined by the spacing of the path.

The copper is made visible on the screen whenever **Full Width Paths** is turned on in the **View** menu. However, it is drawn semi-transparently to allow features on other layers to show through.

This command can be used to create a ground or power plane on the current layer with the added ability to have embedded paths. Such paths, having non-zero spacing, are isolated from the copper plane. Use the **Thermal Tool** to create connections from pads to the copper flood area.

You can selectively remove copper in an area by drawing a path that has zero width but non-zero spacing. Because the width is zero, the path itself does not contribute any copper but the spacing causes some of the copper in the flooded region to be removed. You can also use the **Paint** command, using a zero-width and non-zero-spacing surrounding path, to remove larger areas of copper.

Copper Flood and **Make Signal Plane...** are alternate, mutually incompatible techniques for creating a signal-plane or ground-plane. Because of this, **Copper Flood** is disallowed on signal-plane layers.

7.9 Selected Paths Flooded

The **Selected Paths Flooded** command is a toggle command that turns on and off **Copper Flood** for an area of the board that is defined by a selected freehand path. This produces an area of the board similar to that created by the **Copper Flood** command above, but on a smaller and more precise scale.

Before using this command, define a region by drawing a closed path with the **Draw Freehand Path** tool. Then select this path with the **Select / Move Parts** tool and issue the **Selected Paths Flooded** command to turn on **Copper Flooding** for the region. If you select a path that already defines a **Copper Flood** region, the **Selected Paths Flooded** command will turn off **Copper Flooding** for that path.

7.10 Flip Over

The **Flip Over** command causes the entire design to flip over so that the back side of the board is now the front and the front side is now the back. Issuing the command a second time restores the design to its original state.

7.11 Dup Layer Before

The **Dup Layer Before** command creates a new layer by duplicating the current layer and placing it before the current layer. All pads are duplicated but none of the paths.

7.12 Dup Layer After

The **Dup Layer After** command creates a new layer by duplicating the current layer and placing it after the current layer. All pads are duplicated but none of the paths.

7.13 Delete Layer

The **Delete Layer** command deletes the current layer with all its pads and paths. This reduces the total number of layers by one.

7.14 Make Rats Nest

The Make Rats Nest command creates a rats nest of paths on the Rats Nest layer connecting pins belonging to every signal in the design. Paths in the rats nest can then be used by various tools, such as the **Quick Route** tool and the **Wrap Path** tool, to route paths on real board layers.

7.15 Destroy Rats Nest

The **Destroy Rats Nest** command eliminates all paths in the **Rats Nest** layer.

7.16 Make One Rats Nest

The **Make One Rats Nest** command creates a rats nest of paths on the **Rats Nest** layer connecting all pins belonging to the current signal. The current signal is usually selected using the **Choose/Connect** tool. You can also make a rats nest of one signal using the **Choose/Connect** tool by clicking on a pin while pressing the **Option** key.

7.17 Destroy One Rats Nest

The **Destroy Rats Nest** command eliminates all paths in the **Rats Nest** layer connecting pins that belong to the current signal.

Chapter 8

Window Menu

8.1 Deferred Parts

Deferred parts are parts that have not yet been placed on the board. The **Deferred Parts** command brings up a floating window showing a list of all deferred parts. Each list item consists of the name of the part followed by the part type in parenthesis. An example is shown in Figure 8.1.

To place parts on the board from the deferred parts list, click on the list item in the **Deferred Parts** window and drag it to the desired location in your design window. As you are dragging the part over the design window, a rectangular part outline will follow the cursor. As each item is placed on the board, it is removed from the **Deferred Parts** list.

8.2 Library Parts

The **Library Parts** command brings up a floating window showing a list of all the part types in an open design along with a preview window. An example list is shown in Figure 8.2. You can create a new instance of any ot these parts by clicking on the part type name in the list and dragging the cursor to your design view. You can also drag from the preview window. When you select a part from the part list, a preview of the part is shown in the preview pane. As you are dragging the part over the design window, a rectangular outline will follow the cursor. When you release

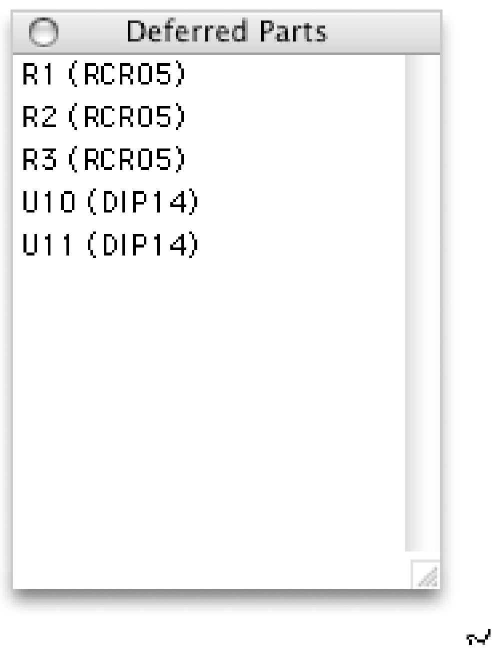


Figure 8.1: Deferred Parts Window

the drag, a new part is created and placed at the drop location. The part is given a unique name which you can specify in the Next Part ID text field. If the name is already used, the numeric part of the name is incremented until a unique name is found. The numeric part of the name is then incremented to allow you to create the next part. If you wish, you can change the name later with the **Rename Parts. . .** menu command.

This window is also used in conjunction with the **New Part** tool which provides an even easier means of creating new parts.

With the popup menu, you can examine the part type list of any open design. This can be either a true design or just a library, which is merely a collection of part types. This means, for example, that if you wish to use a part type that was used in a previous design, you can open up the previous design in a separate window, and then using the **Library** Window, drag and drop the part to your current design.

The **Remove** button can be used to remove the selected item from the list of part types in a design. However, this only works if the part type is not currently being referenced by an instance of the part in that design.

The **Rename. . .** button can be used to rename the selected part type.

The **Edit. . .** button can be used to edit the selected part type.

If the **Capture Keystroke** button is enabled, letters typed at the keyboard will scroll the list to the first part type that begins with that letter. If this button is not enabled, all keystrokes will go to the regular design window where they may be used to select various tools.

If the **Continuous Placement** button is enabled, a new part will be created and placed whenever the cursor is clicked in the design area without needing to click and drag from the **Library Window**.

8.3 Document Windows

The Osmond system allows multiple designs to be open at a time. Each open document creates a menu item in the Window menu containing the name of the design. Selecting the menu item brings the corresponding design window to the front.

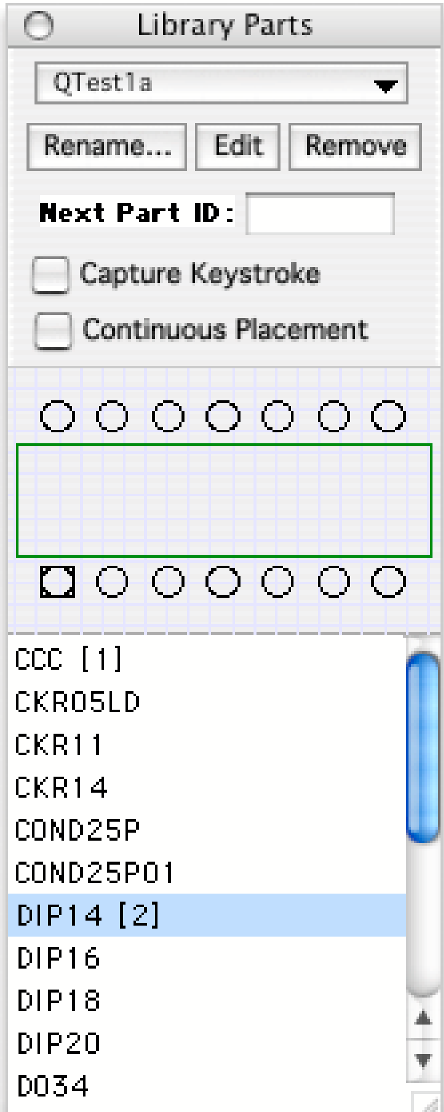


Figure 8.2: Library Window

Chapter 9

Tools

9.1 Selection Tool

Use this tool to select parts, freehand traces, and unsupported pegs.

To select this tool, click on the **Select Tool** button in the tool palette or press the **S** key on the keyboard.

To use this tool, click and drag through an area of the view. Parts that have a pad or vertex anywhere within the selection area on the current layer will be selected. Also, freehand paths that have a vertex within the selection area and unsupported pegs within the selection area will be selected. You can also select a part by clicking within one of the pads belonging to that part.

You can also use this tool to move selected parts in a manner similar to other Macintosh applications.

Parts that are selected will change their appearance: Pads belonging to selected parts will be greyed and paths will be dashed. Unsupported pegs that are selected will show a small square mark.

This tool lets you use the **shift** key, like other Macintosh applications, to add parts to the selection list (if not already selected), or to remove parts from the selection list (if they are selected).

Normally, you can select only parts, freehand paths, and unsupported pegs that have features (pads or vertices) on the current layer. If you hold down the **control** key, however, you can select parts, freehand paths, and unsupported pegs that have features on any layer.

Because moving and rotating components often go together during initial placement, this tool has been augmented to allow you to rotate items. If you hold down the **option** key, the tool can now be used to rotate parts in a counter-clockwise direction. If you simultaneously hold down the **option** and **shift** keys, the tool will rotate in the clockwise direction.

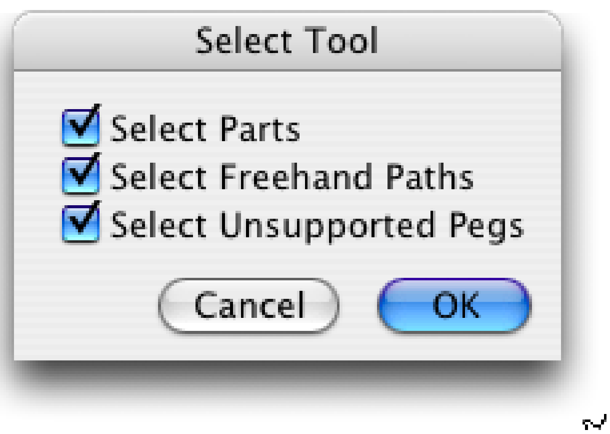


Figure 9.1: Select Tool Options

Sometime, you may wish to be able to select only parts and not freehand paths, or only freehand paths and not parts, or not unsupprted pegs, etc. If so, double-click on the **Select Tool** button in the tool palette to bring up the dialog shown in Figure 9.1.

Use the check boxes to indicate what this tool will be able to select.

If you click on a feature while holding down the **Command** key, a dialog like Figure 9.2 appears. This dialog shows the position of the feature you clicked and allows you to manually change that position. The position can be either an absolute position relative to the lower left-hand corner of the design area, or a relative position with respect to the current grid origin. The grid origin can be changed with the **Origin/Index Tool** described later. Using the pull-down menu next to the edit field, you can specify the position in either millimeters or mils.

9.2 Zoom Tool

Use this tool to increase or decrease the magnification of the view.

To select the tool, click on the zoom tool button in the tool palette or press the **Z** key on the keyboard.

 When the tool is selected, the cursor will appear like this to allow you to increase the magnification.

 If you hold down the **shift** or **option** key, the cursor will appear like this to allow you to decrease the magnification.

To use the tool, click on any point in the view. The view will be redrawn with a new magnification and centered on the point you clicked.

You can also use the tool to select a region of your design by dragging through an area in the view. When you release the drag, the view is redrawn such that the region you selected fills the view.

 If you hold down the **Command** key, the cursor appears like this.

This allows you to move the view within the window by clicking in the view and dragging.

To change the zoom factor (the amount that the magnification is increased or decreased each time the tool is used), double-click on the zoom tool button in the tool palette to bring up the dialog in Figure 9.3.

For example, a zoom factor of 2.0 will double the magnification of the view each time (or halve the magnification if the **shift** or **option** key is held down).

At the maximum magnification, each pixel on the screen represents the smallest resolution of the system (10 nanometers). For a standard 72 pixel per inch monitor, this means the maximum magnification is about 35000 times real size.

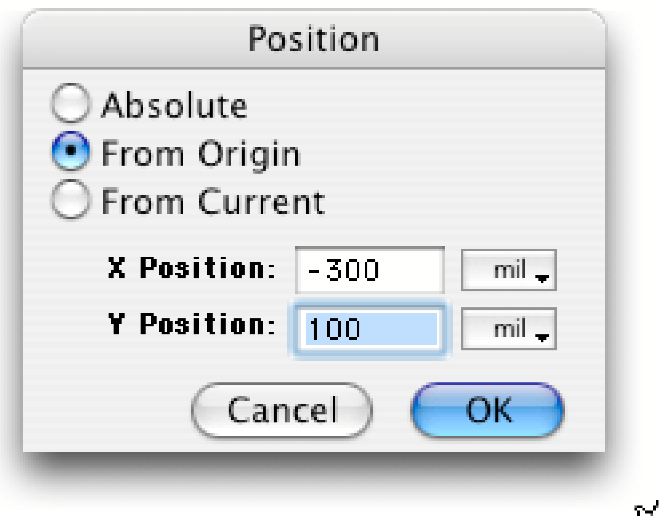


Figure 9.2: Move Dialog

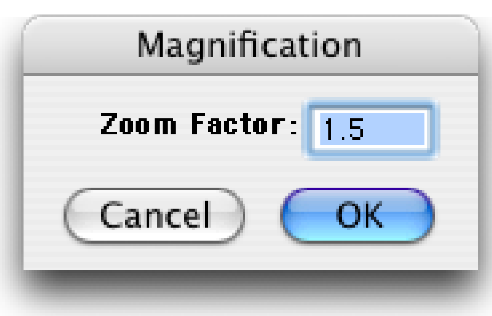


Figure 9.3: Zoom Factor

9.3 Move Tool

Use this tool to move parts, freehand paths, or silkscreen part names and part values.

To select this tool, click on the **Move Tool** button in the tool palette or press the **M** key on the keyboard.

To move a part, click near a pad or vertex that belongs to the part **on the current layer** and drag it to a new location. A gray outline of the part follows your cursor as you drag. When you release the mouse button, the part moves to the new location with the dragged pad or vertex snapping to the nearest grid location. When the part moves, all paths that connect to pads on the part stretch to maintain their connections.

Note that when you move a part with the **Move Tool**, the nearest part feature to your click point (for example, the nearest pin) will snap to the nearest grid crossing. When you move again, you can select a different feature to snap to grid. This gives you a lot of flexibility in positioning parts even if the part was not created with the same grid as the current window. If the origin of the part coincides with a feature, such as pin 1 or the corner of the silk screen box, you can move the origin of the part to a grid crossing by clicking near that feature when moving.

Similarly, you can move a freehand path by clicking near one of its vertices and dragging it to a new location. You can also move part name or part value text by clicking within the text and dragging to a new location.

If the part, freehand path, or unsupported peg that you move is a member of the group of selected items, the entire group moves together as one.

If you double-click on the **Move Tool** in the tool palette, the dialog shown in Figure 9.4 will appear. This dialog gives you fine control over what items can be moved with this tool. Each of these can be enabled or disabled individually.

If you click on a feature while holding down the **Command** key, a dialog like Figure 9.5 appears. This dialog shows the position of the feature you clicked and allows you to manually change that position. The position can be either an absolute position relative to the lower left-hand corner of the design area, or a relative position with respect to the current grid origin. The grid origin can be changed with the

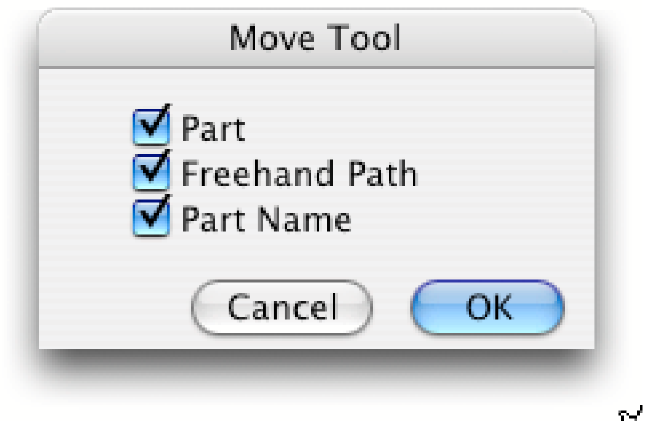


Figure 9.4: Move Tool Options

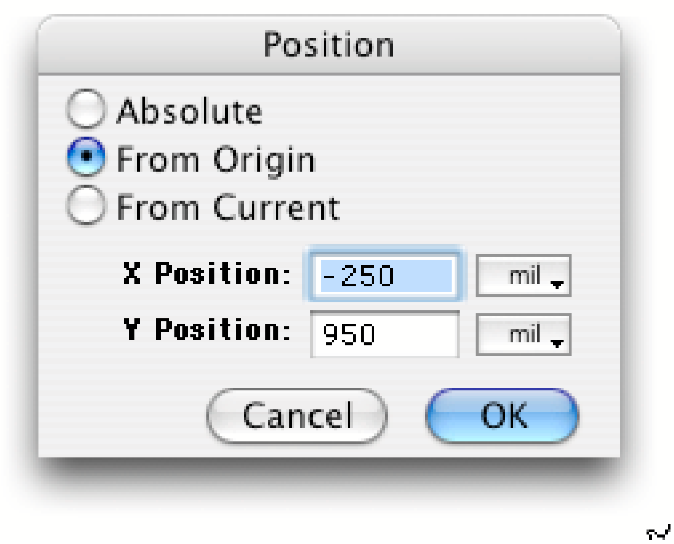


Figure 9.5: Move Dialog

Origin or **Index Tool** described later. Using the pull-down menu next to the edit field, you can specify the position in either millimeters or mils.

9.4 Turn Tool

Use this tool to rotate parts, freehand paths, or silkscreen part names.

To select this tool, click on the **Turn Tool** button in the tool palette or press the **T** key on the keyboard.

 When the tool is selected, the cursor appears like this, showing the default direction of rotation (counter-clockwise).

 If you hold down the **shift** button, the direction of rotation is reversed (to clockwise) and the cursor appears like this.

To rotate a part with this tool, click on a pad or vertex that belongs to the part **on the current layer**. The part is rotated by a certain amount (the default is 90 degrees). The center of rotation is the location of the pad or vertex you clicked. When the part rotates, all paths that connect to pads on the part stretch to maintain their connections.

Similarly, you can rotate a freehand path by clicking on one of its vertices. You can also rotate part name or part value text by clicking within the text.

If the part or freehand path you turn is a member of the group of selected parts or freehand paths, all the parts and freehand paths in the group will rotate together as a unit.

If you double-click on the **Turn Tool** in the tool palette, the dialog as shown in Figure 9.6 appears. This dialog gives you fine control over what items can be rotated with this tool. These can be enabled and disabled individually.

The dialog also gives you the ability to modify the rotation angle as desired.

Note that although parts can be rotated to any angle, the shapes of pads can only be rotated in 90 degree increments. Rotating a part by any angle will cause the pads to either rotate by 90 degrees, or not rotate at all. For parts with all circular pads, however, this does not matter.

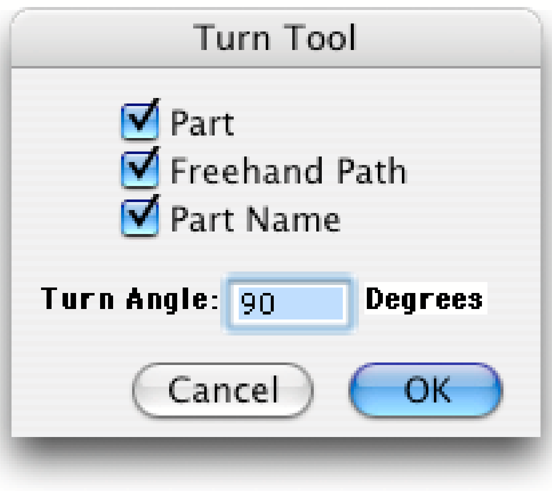


Figure 9.6: Turn Tool Options

9.5 Flip Tool

Use this tool to flip parts from one side of the board to the other.

To select this tool, click on the **Flip Tool** button on the tool palette.

To use this tool, click on a pad or vertex **on the current layer** belonging to a part. This causes the part to flip from one side of the board to the other, if possible. Specifically, the part is rotated around a line that passes through the feature that you clicked and is parallel to the part's own Y axis. When the part flips, all paths connected to pads on the part stretch to maintain their connections. If a path cannot maintain a connection (for example, because one of the pads is a surface pad that does not connect through the board), the flip is not allowed.

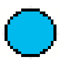
If the part you flip is a member of the group of selected parts, all members of the group will be flipped.

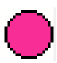
9.6 Connect Tool

Use this tool to construct paths to connect pads.

To select this tool, click on the **Connect Tool** button on the tool palette or press the **C** key on the keyboard.

To use this tool, click on either a pad or on a vertex of a connecting path. If the pad or vertex belongs to a signal net, the name of the signal will be displayed in the text window just below the tool palette. In addition, all pads that belong to the signal will be highlighted in one of two colors

 All pads that are already connected to the pad or vertex that you clicked (through any connection on any layer) will appear in blue like this.

 Pads that should be but are not yet connected to the pad or vertex that you clicked (even if some of these pads are connected to each other) will appear in magenta like this.

To construct a connecting path, click on a pad (which will turn the pad blue) and drag toward a pad that is colored magenta. As you are dragging, a gray line will form from pad to pad to indicate that a connection is possible. Other similar actions are also possible: You can click on a pad and drag to a vertex of a connecting path, or you can click on a vertex of a connecting path and drag to either a pad or another vertex. In any case, a gray line lets you know that the connection is possible.

Once you release the drag, a straight path is drawn from point to point. This path can now be manipulated with other tools to route the path around obstacles.

The system will not let you connect pads that are already connected, even if the connection is on a different layer. The only exception is that two pads that are connected to a signal plane may be connected on some other layer. The system will also not allow you to connect more than three paths from any one pad or vertex.

Normally, this tool allows you to connect only pads that belong to the same signal net as defined by the net list. Several options, however, allow you to modify what pads are connected to the signal net.

If you hold down the **Command** key on the keyboard, you can use the tool to connect pads or connection paths that are not already connected, even if the pads do not yet belong to a signal net:

- If you connect one pad that *does* belong to a signal to a second pad that *does not* belong to a signal, the second pad will be added to the signal net of the first pad.
- If you connect two pads and *neither* of the pads belong to a signal, a new signal net will be created that contain these two pads.
- If you connect two pads and *both* of the pads belong to different signals, the second pad and all of its connecting pads will be removed from its current signal and added to the first pad's signal.

If you hold down both the **Command** and **shift** keys on the keyboard, you can use this tool to remove a pad from its signal net by just clicking on the pad. Make sure that you have removed all connecting paths from the pad first.

If you hold down the **option** key on the keyboard, you can use this tool to create a rats nest on the Rats Nest layer connecting all pins that belong to the signal of the pin you clicked.

9.7 Attach Tool

Use this tool to attach an existing trace to a pad or to detach a trace that passes through a pad.

To select this tool, click on the **Attach Tool** button on the tool palette.

To use the tool, click on a trace and drag to a pad that belongs to the same signal as the trace. As you drag, ghost lines will be drawn showing the new path if the attachment is permitted. When you release the drag, the trace will be re-routed to pass through the the new pad. The illustration in Figure 9.7 shows an example circuit before and after using the **Attach Tool**.

The inverse operation, detaching a trace, can be performed using this tool by holding down the **shift** key. Starting with the illustration above right, using this tool to click on the middle pad while depressing the shift key will result in the illustration above left.

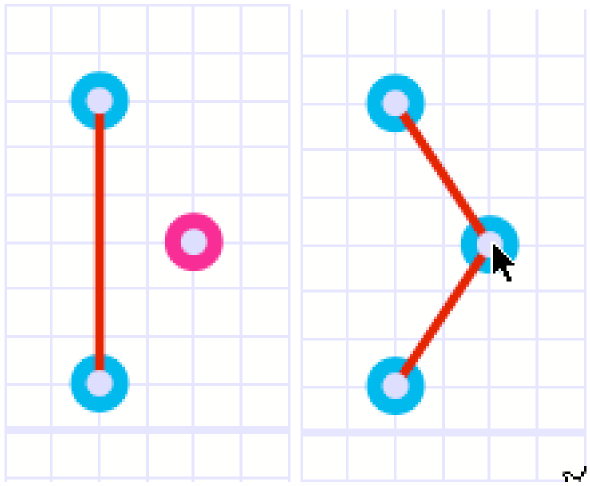
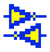


Figure 9.7: Attach Tool

You can normally attach a trace only to a pad that belongs to the same signal as the trace. However, if you hold down the **Command** key, this forces the attachment to any pad. If the pad does not already belong to a signal, it is added to the trace signal. If the pad already belongs to a signal, it is removed from its original signal (along with all connected pads) and added to the signal of the trace. Obviously, this changes the design so it should be used with caution.

9.8 Pin Swap Tool



Use this tool to swap signals and connections between two pins.

To select this tool, click on the **Pin Swap Tool** button on the tool palette.

To use the tool, click on one pin and drag to another pin. As you drag, a line is drawn from the first pin to the second pin. When you release the drag, all signals and paths attached to the first pin are transferred to the second pin and vice-versa.

Since this tool can be used between any two pins, and since this could easily change your design, this tool should be used with caution.

9.9 Cut Tool

Use this tool to remove connecting paths between pads or vertices.

To select this tool, click on the **Cut Tool** button on the tool palette or press the **K** key on the keyboard.

To use the tool, click on a path. If the path is a connecting path between two pads, the entire path is removed. If the path contains a fork where one path turns into two paths, only the path up to the fork is removed. If the path is a freehand path, only one segment of the path is removed.

If you click on a connecting path while holding down the **option** key, the paths to all pads (including forks) are removed and saved on the clipboard. You can then use the **Paste** menu command to restore the paths to the current layer or, (by first changing layers) redraw the paths on a different layer.

If you click on a connecting path while holding down the **Command** key, not only is the connecting path removed but the circuit is completely separated into two distinct signals. Since this changes the design, it should be used with caution.

9.10 Drag Peg Tool

Use this tool to modify paths by constructing, moving, or deleting path vertices or pegs.

To select this tool, click on the **Drag Peg** tool button on the tool palette or press the **D** key on the keyboard.

When this tool is selected, to serve as an aid, a small square "ghost" dot will appear over the vertex that is closest to the cursor.

To use the tool, click on a vertex and drag it to a new location. As you drag, "ghost" lines stretch like rubber bands to show the connecting paths. When you finish dragging, the vertex snaps to the nearest grid location and the connecting paths are redrawn.

If you hold down the **shift** key, clicking on a vertex will remove the vertex and the two connecting paths will become a single straight path.

If you click on a path that does not contain a vertex, or on a path segment away from a vertex, a new vertex will be created in the path which you can then drag as before.

You can use this tool with paths on the **Rats Nest** layer even if the **Rats Nest** layer is not the current layer. Clicking on a Rats Nest path first transfers the path to the current layer and then operates on it there

9.11 Wrap Tool

Use this tool to route connecting paths around obstacles.

To select this tool, click on the **Wrap Tool** button on the tool palette or press the **W** key on the keyboard.

To use this tool, click on a path and drag it beyond a pad or vertex. As you drag, "ghost" lines will stretch like rubber bands to show you the result of the drag. The path will stretch in such a way that the an adequate amount of spacing is maintained between the path and the pad or vertex around which it is "wrapped". This tool can be used repeatedly until the path is completely routed.

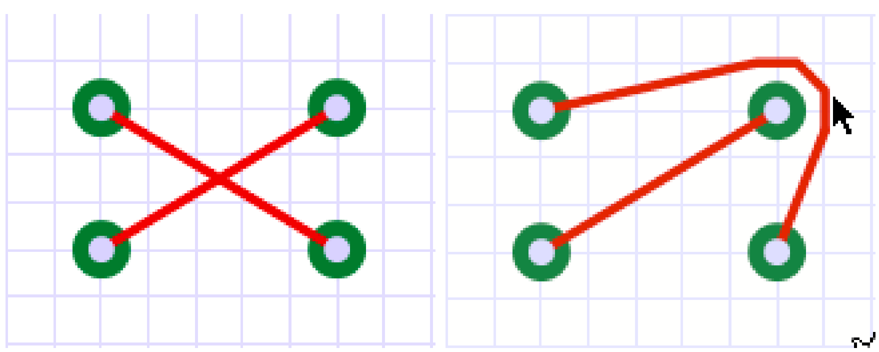


Figure 9.8: Wrap Tool

For example, the two pictures in Figure 9.8 show a simple design before and after the wrap tool is used.

You can use this tool with paths on the **Rats Nest** layer even if the **Rats Nest** layer is not the current layer. Clicking on a Rats Nest path first transfers the path to the current layer and then operates on it there

9.12 Quick Route Tool

Use this tool to route paths, especially using paths from the **Rats Nest** layer.

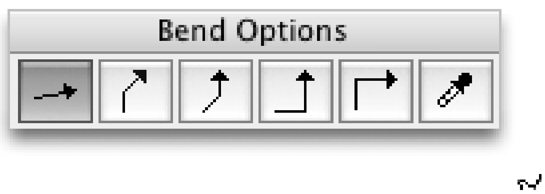


Figure 9.9: Quick Route Tool

To select this tool, click on the **Quick Route** tool button on the tool palette or press the **Q** key on the keyboard. When the tool is selected, a floating palette of bend options appears as shown in Figure 9.9.

These bend options allow you to specify how path bends are constrained as the path is routed from pin to pin.

The first option specifies no constraint. That is, paths are routed in a straight line from point to point.

The second option specifies vertical or horizontal followed by diagonal. That is, the path from point A to point B is constrained so that the first part of the path is either horizontal or vertical and the remaining portion is a diagonal path at a 45 degree angle.

The third option specifies diagonal followed by vertical or horizontal. A path from point A to point B is constrained so that the first part is a diagonal path at a 45 degree angle and the remaining portion is either vertical or horizontal.

The fourth option specifies horizontal followed by vertical.

The fifth option specifies vertical followed by horizontal.

Before describing the sixth option (which is somewhat different) let us discuss how the first five operate.

To show how the tool works, consider a simple example as shown in Figure 9.10. This shows pads connected by a path on the **Rats Nest** layer.

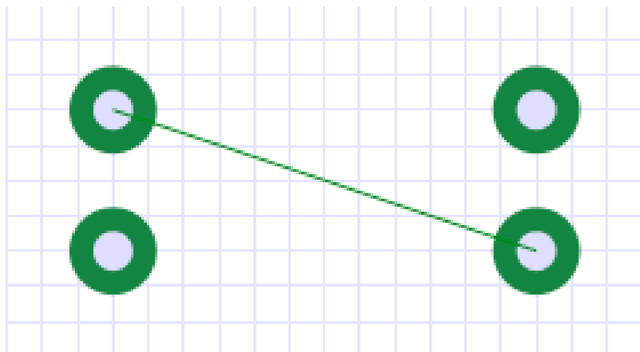


Figure 9.10: Path on Rats Nest Layer

Assuming that the second bend option is chosen (vertical or horizontal followed by diagonal), clicking on the path near the left pin (and releasing) erases the rats nest path and produces something like Figure 9.11.

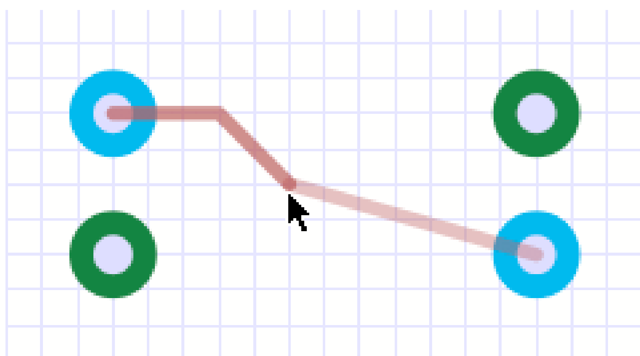


Figure 9.11: Quick Route Tool Usage 1

Because we clicked near the left pin, it is the start pin. The pin at the right is the end pin. The solid line originating at the start pin is first horizontal and then diagonal in agreement with the chosen bend option. The dotted line completes the path to the end pin.

We can move the cursor down and then click to produce Figure 9.12.

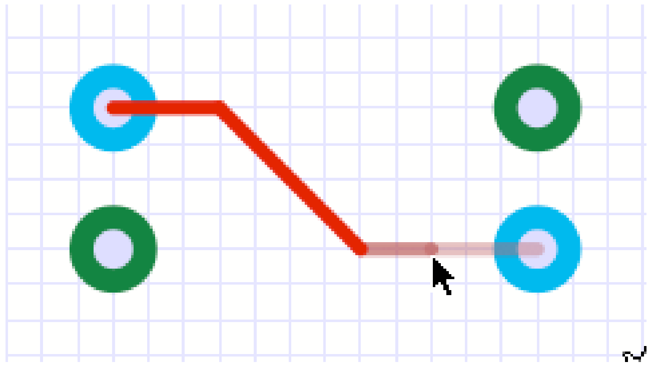


Figure 9.12: Quick Route Tool Usage 2

This shows that a portion of the final path consisting of a horizontal section followed by a diagonal section has been assigned to the current layer. At this point we could move to another layer to place a portion of the path on a different layer. Or we can simply complete the route by clicking on the end pin to produce Figure 9.13.

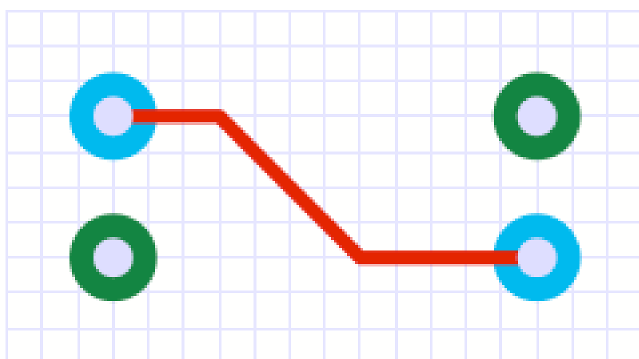


Figure 9.13: Quick Route Tool Usage 3

If we had placed different portions of the path on different layers, vias would be inserted automatically where needed.

Once you have begun to route a path, solid and dotted rubber band lines will follow your cursor until you either complete the route, suspend the route, or abort the route.

You can suspend the route by pressing the **escape** key. When you suspend the route, the route that you have done so far is kept and the path is completed temporarily with a single straight path, which you can then route later with this or other tools.

You can abort the path by either choosing a different tool from the tool palette or by clicking on the start pin.

This tool can also be used starting with paths on any layer, not just the **Rats Nest** layer. When you click on a path, normally only the single path segment clicked is involved in the route. However, if you hold down the **option** key while clicking, the path is first completely straightened. This allows you to abandon an existing route and define a new route from scratch.

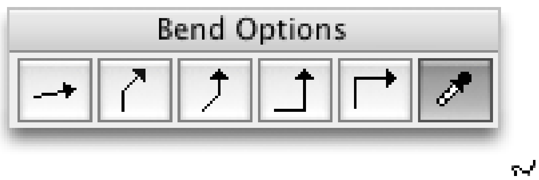


Figure 9.14: Quick Route Tool Copy

The sixth option, shown in Figure 9.14, lets you sample the pattern of bends from an existing path so you can then apply that same pattern to other paths. Of course, this works only if the spacing of end points of the new path matches the spacing of end points in the sampled path. However, one often finds this situation in practice, especially when routing bus signals.

With this option selected, you can sample an existing bend pattern by clicking on the path while holding down the **shift** key. The cursor changes to a dropper while the **shift** key is depressed. With a pattern sampled, you can now apply the pattern to another path by clicking on the other path. However, if the spacing of end points of the clicked path does not match the spacing of end points of the sampled path, nothing is changed.

9.13 Width Tool

Use this tool to change path widths and path minimum spacings.

To select this tool, click on the **Width Tool** button in the tool palette.

To use the tool, click on a path. The width and minimum spacing of the path will be changed to the current tool values and the path redrawn.

To modify the current tool values, double-click on the **Width Tool** on the tool palette to bring up the dialog below.

Normally, the tool changes the width and spacing of a path along its entire length from pad to pad. By selecting the **Apply to Single Segment Only** check box, you can force the tool to change the width and spacing of only a single path segment. This can be useful if you need to "neck down" a path to allow it to pass through a narrow passageway.

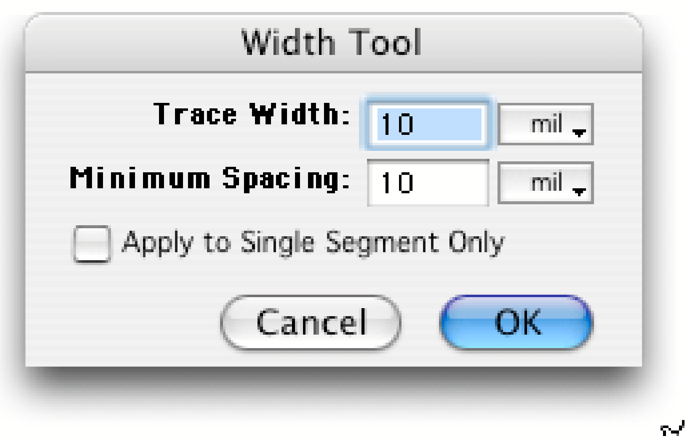
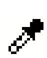


Figure 9.15: Width Tool

Using the pull-down menu next to each edit field, you can specify the width and spacing in either millimeters or mils.

If you hold down the **option** button while clicking, the dialog as shown in Figure 9.15 will appear with the current tool values changed to those of the path you clicked. Modifying the values and pressing the **OK** button will then change the width and/or minimum spacing of the path.

 If you hold down the **shift** button, the cursor will change to this icon to indicate that you are now in sample mode. If you click on any path while in sample mode, the current tool values will change to that of the clicked path. This is very useful if

you wish to make one path the same width as another path; you simply sample one path and then click on the other.

Chapter 10

More Tools

10.1 Info Tool

Use this tool to get information on items in your design.

To select this tool, click on the **Info Tool** button in the tool palette. When you do, a floating window appears as shown in Figure 10.1.

Now when you click on a feature, such as a pad or a vertex of a trace, the position information is shown in the information window like Figure 10.2.

The **Absolute** position (X and Y coordinates) is with respect to the lower left-hand corner of the design area. The **From Origin** position is relative to the grid origin. The grid origin can be changed using the **Origin tool** described elsewhere. The **From Previous** position is relative to the last item you clicked. The **Distance from Previous** field shows you the straight line distance from the last item you clicked. All numbers are in unit of mils.

If you would prefer to see the position in Metric units, click on the **mm** tab to see something like Figure 10.3.

By clicking on the **Traces** tab, you can now get information about all visible traces, including width, spacing, and layer, like Figure 10.4.

Clicking the **Pads** tab allows you to get information on pads (Figure 10.5).

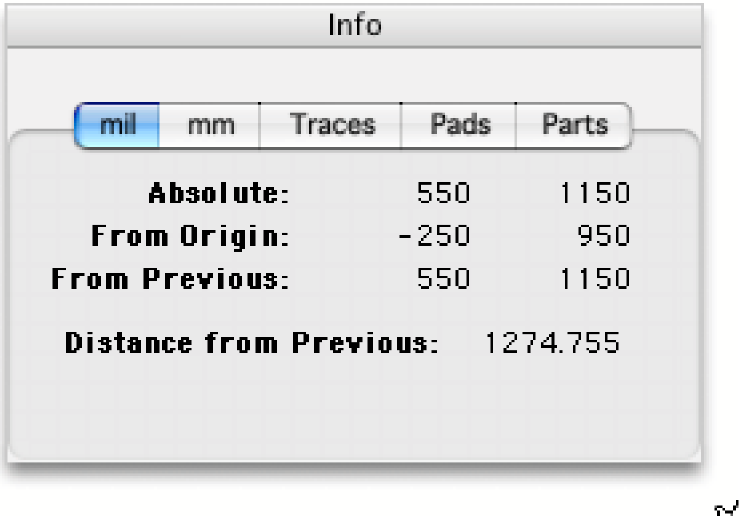


Figure 10.1: Info Tool

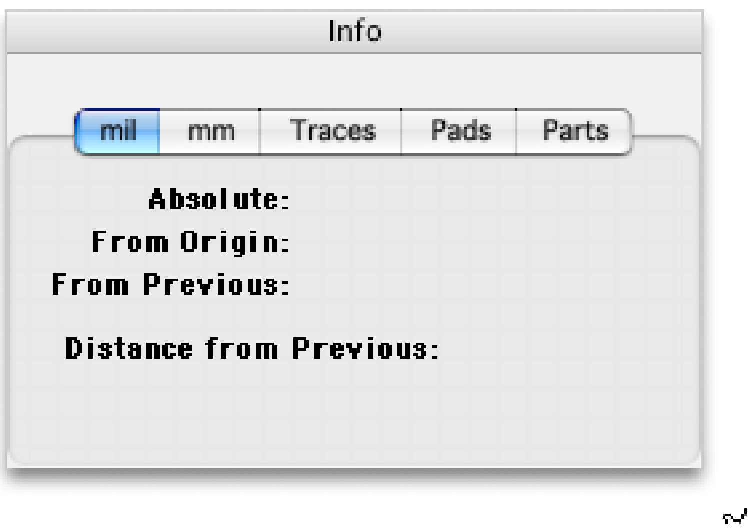


Figure 10.2: Info Tool - mil

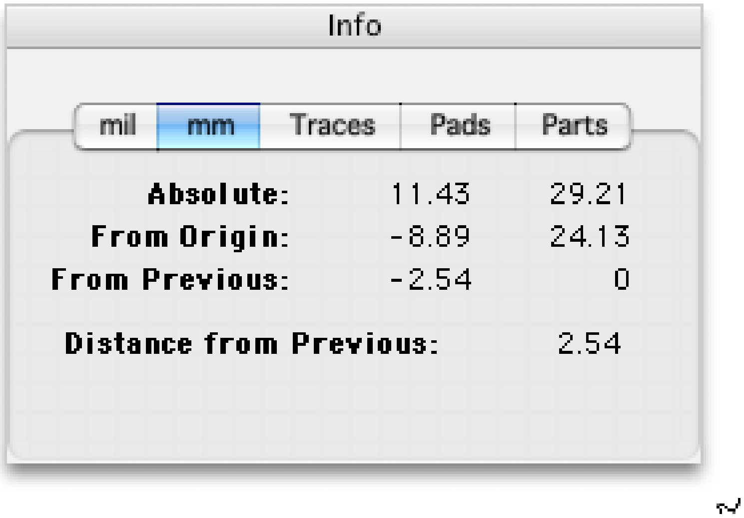


Figure 10.3: Info Tool - mm

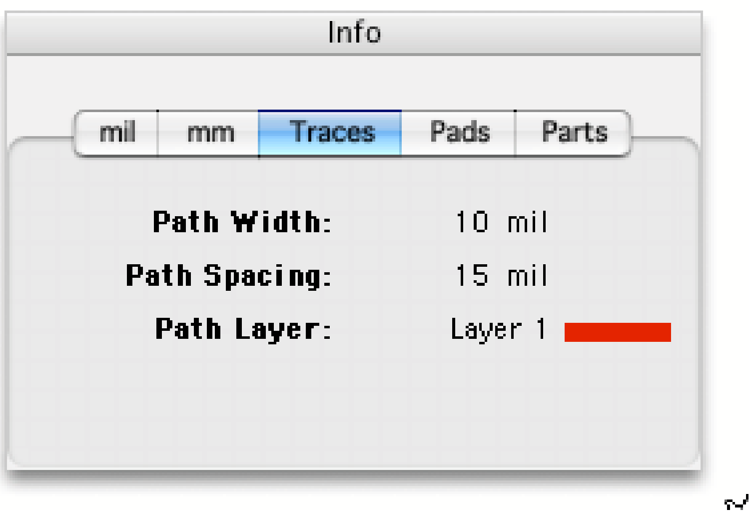


Figure 10.4: Info Tool - trace

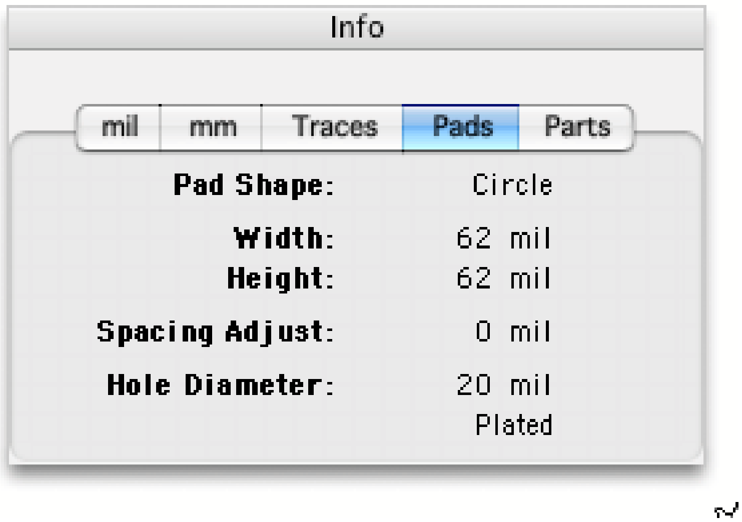


Figure 10.5: Info Tool - pads

Finally, clicking the **Parts** tab allows you to get information on parts (Figure 10.6).

10.2 Origin or Index Tool

Use this tool to move the origin or index point of the grid.

To select this tool, click on the **Origin Tool** button in the tool palette or press the **I** key on the keyboard.

To use the tool, click on a visible pad or vertex. The index point will move to the center of the pad or vertex you clicked, even if the pad or vertex is not on a grid location.

 The index indicator (which looks like this) will appear at the new index point and the grid will be redrawn so that the new index point is the origin of the grid.

If you click on a point that is not near a pad or a peg, the index point will snap to the grid location nearest the point you clicked.

If you double-click the tool in the tool palette, the dialog as shown in Figure 10.7

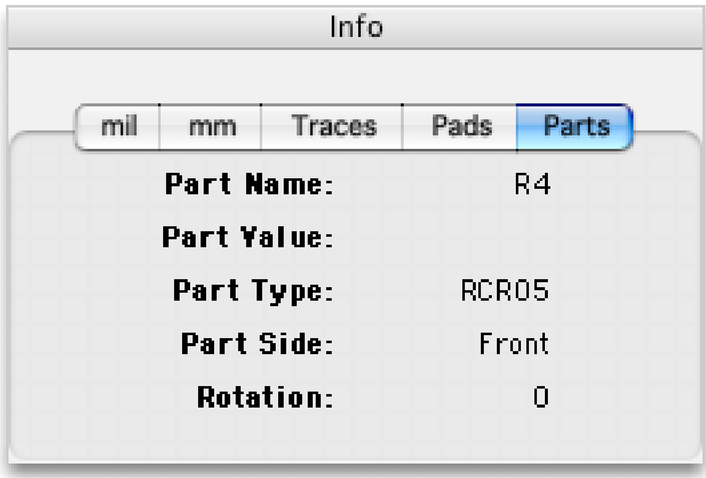


Figure 10.6: Info Tool - parts

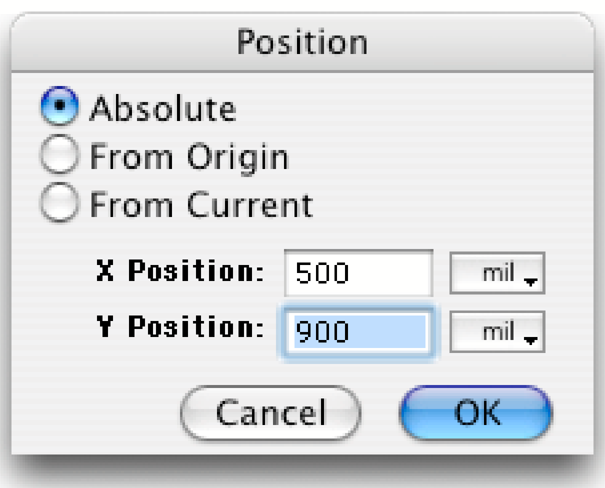
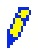


Figure 10.7: Index Tool - parts

appears. This dialog shows you the current position of the index point and lets you change it manually. Using the pull-down menu next to each edit field, you can specify the position in either mils or millimeters.

10.3 Draw Tool



Use this tool to draw freehand paths. Options allow you to draw straight lines, rectangles and ovals.

To select this tool, click on the **Draw Tool** button on the tool palette.

To use this tool, click the mouse at one point and drag to a second point. As you are dragging, a “ghost” line (or rectangle, or oval) will stretch like a rubber band to show your path. When you release the drag, a new freehand path (or rectangle, or oval) will be drawn. The path width and spacing are defined by the current tool values.

If you are drawing a straight path and the first or second point coincide with a vertex of an already existing freehand path, and that vertex has no more than two paths (i.e, it is not already a “Y”), the new path will become attached to the old path.

If you are drawing a straight path and the **Caps Lock** key is active, the angle of the line drawn by the tool will be constrained to an integral multiple of 45 degrees. If you are drawing a rectangle and the **Caps Lock** key is active, the result will be either a straight path or a square. Similarly if you are drawing an oval, the **Caps Lock** key will produce either a straight path or a circle.

To change the drawing mode or to modify the current tool values, double-click on the **Draw Tool** button on the tool palette to bring up the dialog as shown in Figure 10.8.

In **Line** mode, a straight path is drawn from the click point to the drag point. In **Rectangle** mode, a rectangle is drawn with the corners defined by the click point and the drag point. Similarly, in **Oval** mode, an oval is drawn with the corners of the imaginary enclosing rectangle defined by the click point and the drag point. In **Centered Oval** mode, the center of the oval is defined by the click point and the extent of the oval is defined by the drag point.

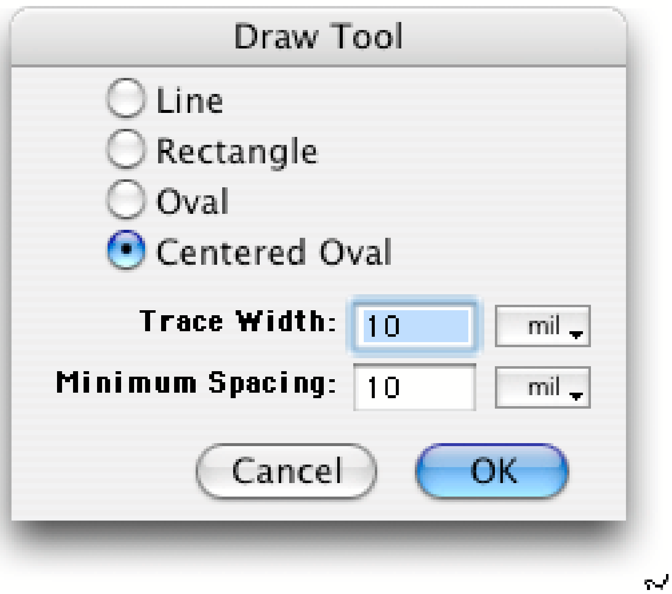
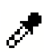


Figure 10.8: Draw Tool - parts

 If you hold down the **shift** button, the cursor will change to this icon to indicate that you are now in sample mode. If you click on any path while in sample mode, the current tool values will change to that of the clicked path. This is very useful if you wish to make the new path the same width and spacing as some other path.

10.4 Text Tool

Use this tool to create text-like parts.

To select this tool, click on the **Text Tool** button on the tool palette.

To use the tool, you should first configure it by double-clicking on the **Text Tool** button on the tool palette to bring up the dialog in Figure 10.9.

The dialog lets you select the text direction including horizontal and vertical directions, both normal and mirrored, as indicated by the orientation of the **F** character in the selection buttons. You must enter the desired text string and select the size

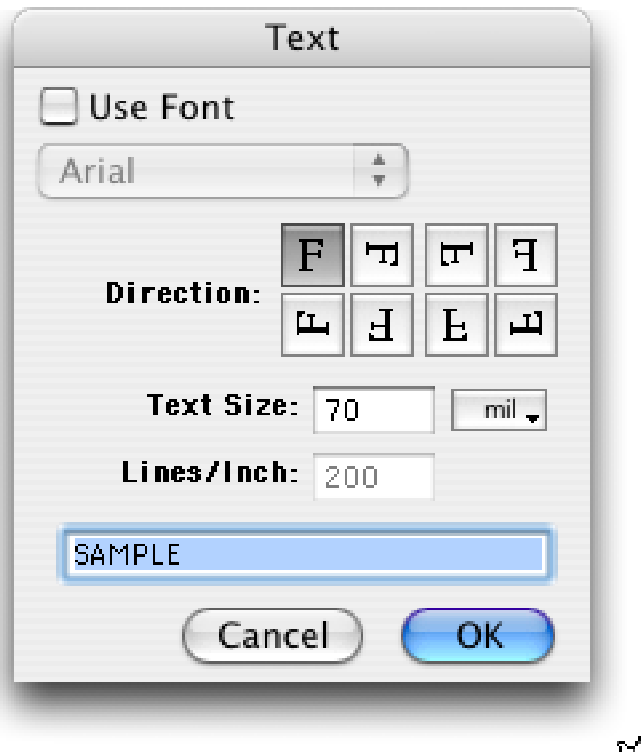


Figure 10.9: Text Tool 1

of the text in either mils, millimeters, or points.

If you do not set the **Use Font** button, the text part will be created using vector character definitions from your library. However, if you set the **Use Font** button, the dialog changes to allow you to use any font in your Apple fonts folder to create text parts (Figure 10.10).

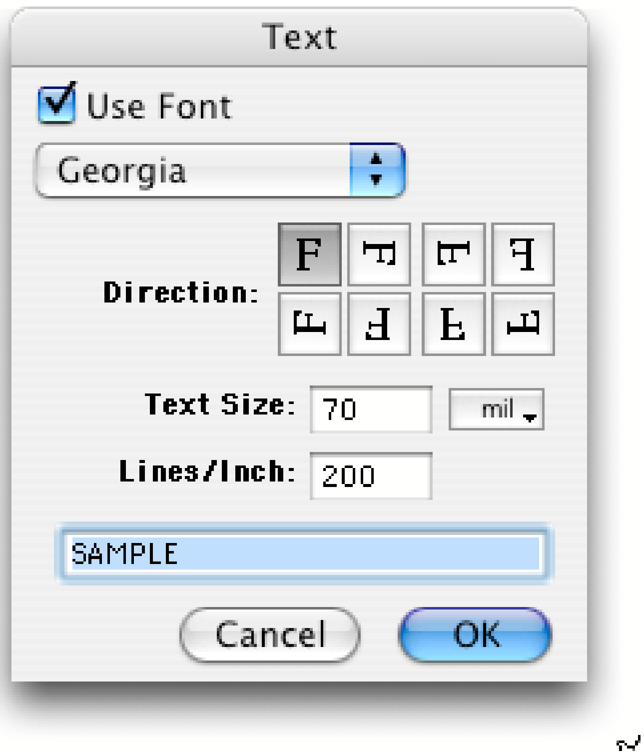


Figure 10.10: Text Tool 2

Text parts made from Apple fonts are created using a series of closely spaced horizontal, vertical, and diagonal lines in a raster pattern. Obviously, as you increase the density of lines (lines per inch), the text will more closely approximate the actual font outline. You can select the number of lines per inch used in this raster pattern by changing the number in the **Lines/Inch** field. Higher numbers will result in better looking text but also greater memory usage.

Once the tool is configured, click on any point on any layer. A text-like part consisting of the desired text string will be created and placed at the clicked location, with the origin of the part snapped to nearest grid location.

 If you hold down the **shift** button, the cursor will change to this icon to indicate that you are now in sample mode. If you click on any text part while in sample mode, the current tool values will change to that of the clicked text part. This is very useful if you wish to make a new text part similar to an already existing one.

If you hold down the **option** button while clicking on an existing text part, the above dialog appears with the current tool values changed to those of the text part you clicked. Modifying the values (including the text string itself) and pressing the **OK** button then changes the text part accordingly.

10.5 Curve Tool

Use this tool to change a sharp path corner into a rounded path as shown in the before and after pictures in Figure 10.11.

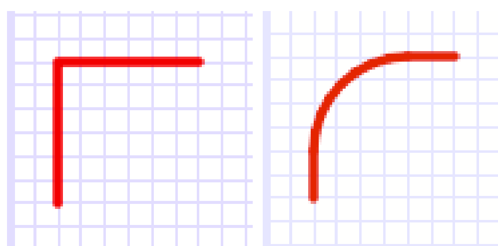


Figure 10.11: Curve Tool Usage

To select the tool, click on the **Curve Tool** button on the tool palette.

To use the tool, click on the vertex of a path where a corner exists. This vertex should not be part of a "Y". The corner will be transformed into a round path with radius equal to the tool default radius.

To change the current tool radius, double-click the **Curve Tool** button on the tool palette to bring up the dialog in Figure 10.12.

Using the pull-down menu next to the edit field, you can specify the radius in either mils or millimeters.

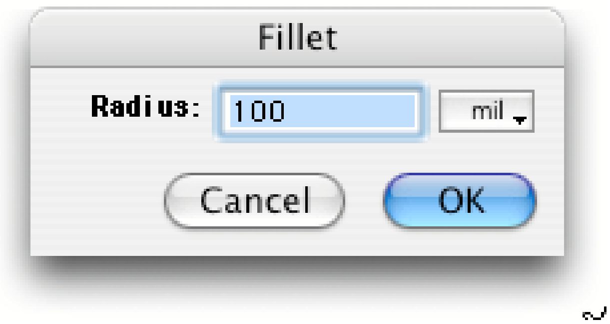


Figure 10.12: Curve Tool

10.6 Change Pad Tool

Use this tool to change the shape and size of a pad.

To select this tool, click on the **Change Pad** tool button on the tool palette.

To use this tool, click on a pad to change it to the current tool values.

 If you hold down the **Command** key, the cursor changes to this. Now if you click on a pad, it is changed to the current tool value rotated 90 degrees.

To modify the current tool values, double-click on the **Change Pad** tool button on the tool palette to bring up the dialog shown in Figure 10.13. For the moment, lets ignore the **Dual Pad** button and look at the fields that let you define a regular pad.

A pop-up menu lets you specify the shape of the pad. The values are as follows:

- Circle/Oval
- Square/Rectangle
- Hex-Oval

Whether a pad is a circle rather than an oval (or a square rather than a rectangle) depends, of course, on the width and the height.

A **Hex-Oval** shape is very similar to a regular oval shape. The only difference is in

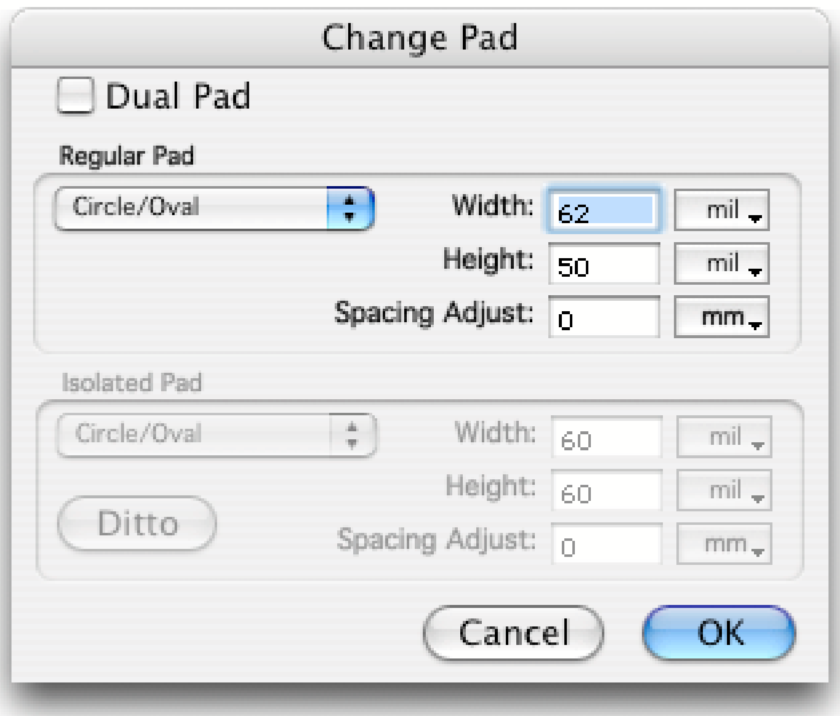


Figure 10.13: Change Pad Dialog

the way paths are routed around the pad. Paths route around a regular oval pad as if it were an octagon. This produces a routing pattern like Figure 10.14.

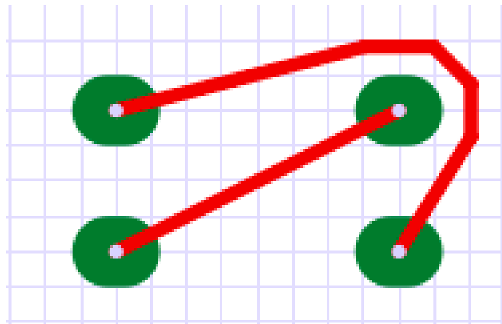


Figure 10.14: Oval Pad Routing

On the other hand, paths route around a **Hex-Oval** pad as if it were a hexagon. This produces a routing pattern as shown in Figure 10.15. This routing pattern is often preferable when we have an alternating offset or zig-zag arrangement of pads.

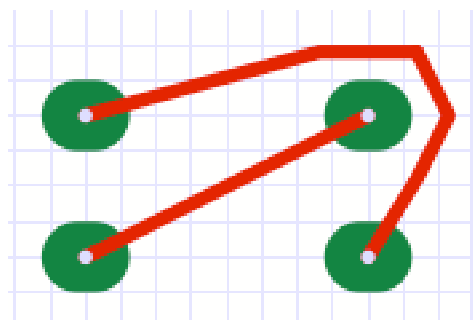


Figure 10.15: Hex Pad Routing

The **Width** and **Height** fields in the dialog allow you to determine the size of the actual pad itself.

The pull-down menu next to each edit field lets you specify the width and height of the pad in either mils or millimeters.

Paths generally route around pads based on the shape and size of the pad, and the width and minimum spacing of the path. For example, lets construct the following scenario:

- Circular pad

- Pad Width = Pad Height = 75 mils
- Path Width = 15 mils
- Path Minimum Spacing = 10 mils

In this scenario, to maintain the correct spacing, the center of the path must be at least 55 mils away from the center of the pad (75/2 + 15/2 + 10 = 55).

The **Spacing Adjust** field in the dialog lets you modify the amount of spacing that the system provides for this pad. For example, if we add the following item to the above scenario:

Spacing Adjust = 5 mils

Then, to maintain the correct spacing, the center of the path must now be at least 60 mils away from the center of the pad (75/2 + 15/2 + 10 + 5 = 60).

The **Spacing Adjust** field can be a positive number if you wish to provide additional spacing around a pad, or it can be a negative number if you wish to allow paths to approach the pad more closely than they would otherwise be allowed.

Dual Pad

In the Osmond system, a pad can have a different shape and size depending on whether it is connected or isolated. This is called a **Dual Pad**. **Connected** means that the pad has at least one connecting path, while **isolated** means that the pad has no connecting path. When a pad is changed from **isolated** to **connected** (by adding a connecting path), the shape and size of the pad changes automatically from the isolated pad shape and size to the connected pad shape and size, and vice versa.

This ability to automatically change pad shape and size depending on whether the pad is or is not connected has proven valuable in routing inner layers. A through-hole pin generally does not need a large pad on an inner layer if there is no connection on that layer. All it needs is sufficient area for a plated through hole which can be much smaller than the area needed for a pad. If a connection is needed on the inner layer, however, then a larger pad should be used. This ability can therefore increase the available area on inner layers which may make routing much easier.

Clicking on the **Dual Pad** check-box changes the dialog to look like Figure 10.16.

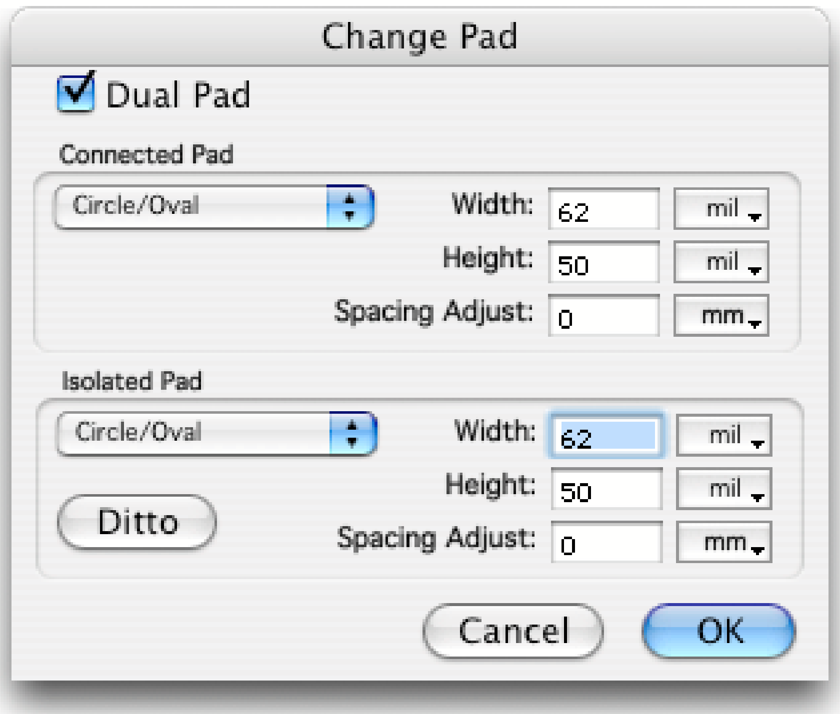


Figure 10.16: Change Pad Dialog - Dual Pad

You specify the shape and size of the **connected** pad in the upper area and specify the size and shape of the **isolated** pad in the lower area.

Of course, you may wish to have the shape and size of the pads be the same whether or not they are connected or isolated. To do this, modify the values in the **Connected Pad** area of the dialog as desired and then press the **Ditto** button in the **Isolated Pad** area.

If you hold down the **option** button while clicking, the above dialog will appear with the current tool values changed to those of the pad you clicked. Modifying the values and pressing the **OK** button will then change the pad accordingly.

 If you hold down the **shift** button, the cursor will change to this icon to indicate that you are now in sample mode. If you click on any pad while in sample mode, the current tool values will change to that of the clicked pad. This is very useful if you wish to make one pad the same as another pad; you simply sample one pad and then click on the other.

10.7 Change Pin Tool

Use this tool to change a pin, which consists of the pin hole and all the pads in the padstack.

To select this tool, click on the **Change Pin Tool** button on the tool palette.

To use this tool, click on a pad to change the associated pin to the current tool values.

 If you hold down the **Command** key, the cursor changes to this. Now if you click on a pin, it is changed to the current tool value rotated 90 degrees.

To modify the current tool values, double-click on the **Change Pin Tool** on the tool palette to bring up the dialog shown in Figure 10.17. This dialog allows you to specify the hole size, whether or not the hole is plated, and the shape and size of all the pads in the padstack.

The dialog provides three radio buttons that let you specify if the padstack defines a regular thru-hole, a surface mount pad, or a complex thru-hole. In a **Regular**

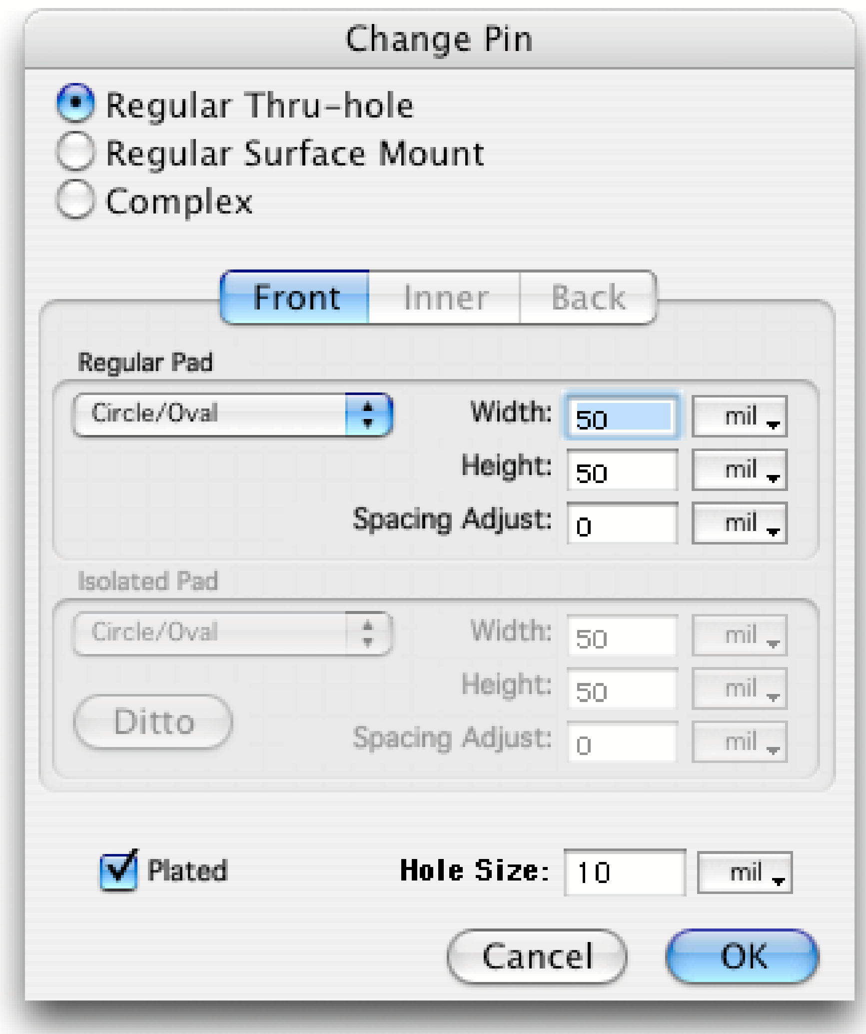


Figure 10.17: Change Pin Dialog

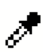
Thru-hole, all pads on all layers are identical. For **Surface Mount**, only the front pad is defined and there are no pads on the inner or back layers. In a **Complex Thru-hole**, pads on different layers may be different or even non-existent.

The dialog also presents three panels labeled **Front**, **Inner**, and **Back**. If the **Regular Thru-hole** or **Surface Mount** button is selected, only the **Front** panel is enabled since the pads on the inner and back layers are either identical to the front pad (**Regular Thru-hole**) or non existent (**Surface Mount**). For a discussion of the fields in each of the panels, see the description of the **Change Pad Tool**.

In the **Front**, **Inner**, and **Back** panels, the popup menu allows you to choose the usual pad shapes plus the additional option of having no pad at all. You may use this option when creating a surface mount pin which has no pads on the inner or far side layer.

Using the pull-down menu next to each edit field, you can specify the width, height, or spacing adjust values in either mils or millimeters.

If you hold down the option button while clicking, the above dialog will appear with the current tool values changed to those of the pin you clicked. Modifying the values and pressing the **OK** button will then change the pin accordingly.

 If you hold down the **shift** button, the cursor will change to this icon to indicate that you are now in sample mode. If you click on any pin while in sample mode, the current tool values will change to that of the clicked pin. This is very useful if you wish to make one pin the same as another pin; you simply sample one pin and then click on the other.

10.8 New Pin Tool

Use this tool to create a new pin which may later be combined with other pins and paths to create a new part.

To select this tool, click on the **New Pin Tool** button on the tool palette.

To use this tool, click anywhere in the design view. A new pin is created according to the current tool values and placed at the nearest grid location.

If you hold down the **Command** key, the cursor changes to this. Now if you click anywhere in the design view, a new pin is created from the current tool values, only rotated 90 degrees.

To modify the current tool values, double-click on the **New Pin Tool** button on the tool palette to bring up the dialog shown in Figure 10.18.

This dialog is very similar to dialog used by the **Change Pin** tool described above. The only additions are the **Name** field and the **Auto Increment** checkbox. The name field lets you specify the name of the next pin created. If the **Auto Increment** button is checked, then subsequent pins will have new names that increment numerically.

For example, if you specify **1** in the name field, the next pin created will be named: **1**. The next pin after that will be named: **2**, and the next pin after that will be named: **3**, and so on.

Pins created with this tool are in reality, individual parts containing a single pin. As such, they can be manipulated with all the tools used to manipulate parts.

Using the pull-down menu next to each edit field, you can specify the width, height, or spacing adjust values in either mils or millimeters.

If you hold down the **shift** button, the cursor will change to this icon to indicate that you are now in sample mode. If you click on any pin while in sample mode, the current tool values will change to that of the clicked pin. This is very useful if you wish to make the new pin the same as another pin; you simply sample the pin and then click at a new location to create the new pin.

10.9 Thermal Tool

Use this tool to adjust the thermal and void pads on a ground or signal plane. Within a copper flood area, this tool also allows pads to connect into the copper flood itself.

To select this tool, click on the **Thermal Tool** button on the tool palette.

A common method of forming a ground or signal plane is by etching the copper on

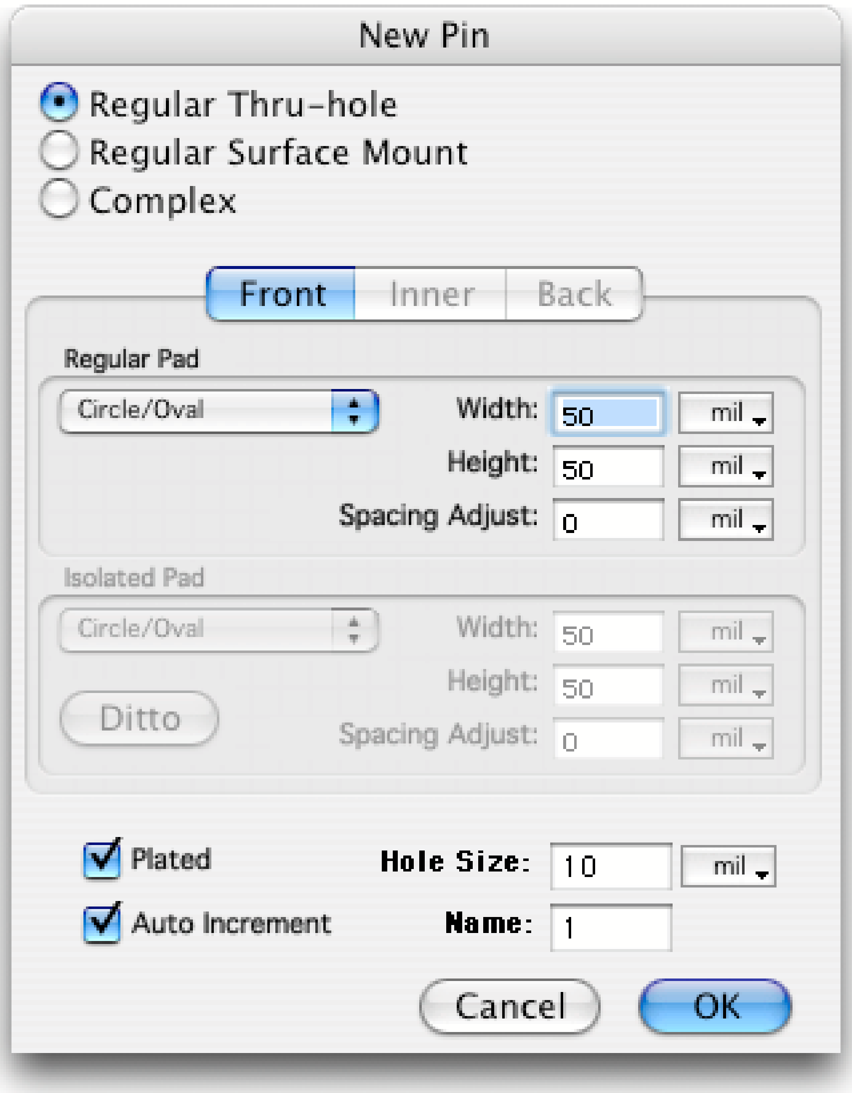


Figure 10.18: New Pin Dialog

a PC board layer with a reverse image instead of a normal image. Using a normal image, paths and pads show where the copper remains; but using a reverse image, the paths and pads show where the copper is removed.

On a signal plane layer, Osmond can use patterns of reverse image pads and paths to control whether or not a pin is connected to the ground or signal plane. In the illustration in Figure 10.19, the reddish area shows where copper remains on the layer while the white area shows where copper is removed from the layer. Black circles show where there is a hole for a through-hole pin.

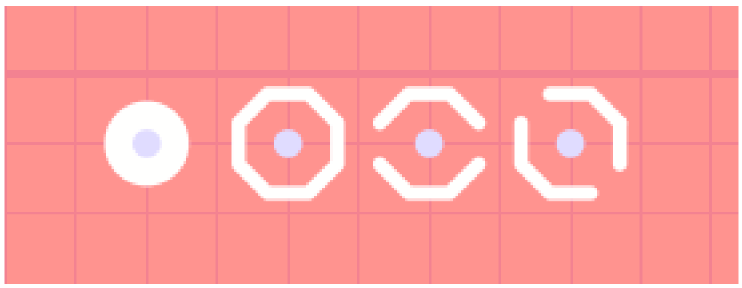


Figure 10.19: Reverse Image Pads

In the first pattern, a reverse image pad (called a void pad) removes the copper from around the through-hole pin. This causes the pin to be completely isolated from the copper plane.

In the second pattern, eight reverse image paths surround the hole leaving an area of copper in an octagon shaped annular ring. The pin connects to the copper in the annular ring but is still isolated from the copper plane.

The third pattern is like the second except two of the surrounding paths have been removed. Now, the pin is connected to the copper in the annular ring and also connected to the copper plane through passages on the left and right hand side of the pin. If we were to remove all the surrounding paths, the pin would still be connected to the copper plane but soldering the pin would be more difficult because of the large heat sink provided by the copper. The surrounding paths therefore act as thermal barriers to permit easier soldering.

The fourth pattern is similar to the third except two different surrounding paths have been removed to provide diagonal connecting passages to the copper plane.

To use this tool to change the reverse image pattern around a pin, click on a pad on a ground or signal plane layer. The pad changes according to the current tool values.

To change the current tool values, double-click on the **Thermal Tool** button on the tool palette to bring up the dialog shown below.

The dialog has two radio button that let you select if the reverse image pattern is a **Thermal Pad** (consisting of surrounding paths) or a **Void Pad**.

If the pattern is a **Thermal Pad**, the buttons to the right allow you to select which (if any) of the surrounding pads should be removed to create a connecting passage. The **Annular Ring** text field lets you specify the minimum width of the copper surrounding the hole, and the **Path Width** text field lets you specify the width of the surrounding reverse image paths.

If the pattern is a **Void Pad**, the **Void Width** text field lets you specify the minimum width of the copper-free ring surrounding the hole.

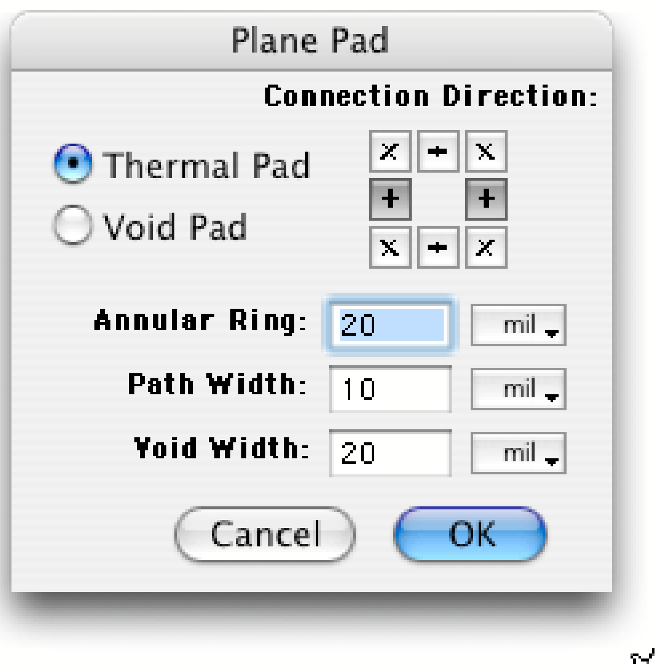


Figure 10.20: Plane Pads

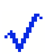
If you hold down the **option** button while clicking, the dialog in Figure 10.20 will appear with the current tool values changed to those of the pattern you clicked.

Modifying the values and pressing the **OK** button will then change the pattern accordingly.

 If you hold down the **shift** button, the cursor will change to this icon to indicate that you are now in sample mode. If you click on any pattern while in sample mode, the current tool values will change to that of the clicked pattern. This is very useful if you wish to make one pattern the same as another pattern; you simply sample the pattern at one pin and then click on the other pin.

The **Thermal Tool** can also be used to connect pads to a ground or power plane within a copper flood region. In a signal plane layer, the isolation is created by drawing surrounding reverse-image paths, and connections are made by removing one or more of these paths. In a copper flood region, on the other hand, the isolation around pads already exists and so connections are made by drawing one or more paths from the pad into the copper flood area. When you use the **Thermal Tool** on a pad that is not in a signal plane, short paths are automatically drawn from the pad out to where the edge of copper flood should be (even if there is no copper flood there).

10.10 Check Tool



Use this tool to check the clearance of a path.

To select this tool, click on the **Check Tool** button on the tool palette.

To use the tool, click on a connecting path. The path is checked for clearance conflicts and if a conflict is found, the part of the path that has the conflict will be highlighted the dialog in Figure 10.21 will appear.

If you press **No**, the conflict will be skipped and the next conflict (if any) will be found and highlighted. If you press **Quit**, the check will be aborted and the dialog will disappear. If you press **Auto**, the Osmond system will attempt to resolve the conflict automatically by moving connecting paths without further dialog. If you press **Yes**, the Osmond system will attempt to find a solution to the conflict and will display that solution highlighted while changing the dialog as shown in Figure 10.22.

If you press **Yes**, the proposed solution will be implemented and the system will move on to the next conflict (if any). If you press **No**, the proposed solution will not

Figure 10.21: Check Tool

Figure 10.22: Check Tool 2

be implemented and the system will attempt to find and present a different solution for your approval. If you press **Auto**, the proposed solution will be implemented and the system will attempt to resolve all remaining conflicts automatically without further dialog. If you press **Quit**, the proposed solution will not be implemented, the dialog will disappear, and all remaining checks will be aborted.

You can use keyboard shortcuts instead of pressing buttons to respond to the dialog as follows:

Yes	**Y** or **return**
No	**N**
Auto	**A**
Quit	**Q** or **esc**

10.11 Via Tool

Use this tool to place a via on the board. The tool works by creating a new **VIA** part (which should be defined in the current library) and placing it on the board at the specified location. The via is added to the pin list of the current signal so you can immediately connect it to any other pin with the **Connect** tool.

Note: Make sure there is a part type named VIA in your current library or this tool will not work.

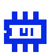
To select this tool, click on the **Via Tool** button on the tool palette.

Before using this tool a signal should be selected. (You can select a signal by clicking on any pad belonging to that signal with the **Connect** tool.) You should also make sure that there is part in the current library named: "VIA".

To use this tool, click on any location of the board. A new via is placed at the location you clicked, snapped to the nearest grid location.

A via is a part like any other part. If you wish to remove a via, therefore, you can select it with the **Select** tool and then **cut** it or **delete** it.

10.12 New Part Tool



Use this tool to create and place a new part on the board. The tool works in conjunction with the **Library Parts** window which contains lists of all part types in all open designs, either true designs or libraries. The popup menu allows you to switch among all open designs.

To select this tool, click on the **New Part Tool** button on the tool palette. The **Library Parts** floating window will become visible if it is not already so. This window looks like Figure 10.23.

Select a part type in the **Library Parts** window. A preview of the selected part appears in the preview pane. You may also specify the name of the part to be created in the **Next Part ID** text field.

Click and hold in the design area. A shadow of the new part appears where you clicked. Now you can drag this part to the desired location. Clicking again creates a new part. As each part is created, the numeric part of the **Next Part ID** is automatically incremented. However, if a part name is already being used, the numeric part of the **Next Part ID** will be incremented until the name is unique.

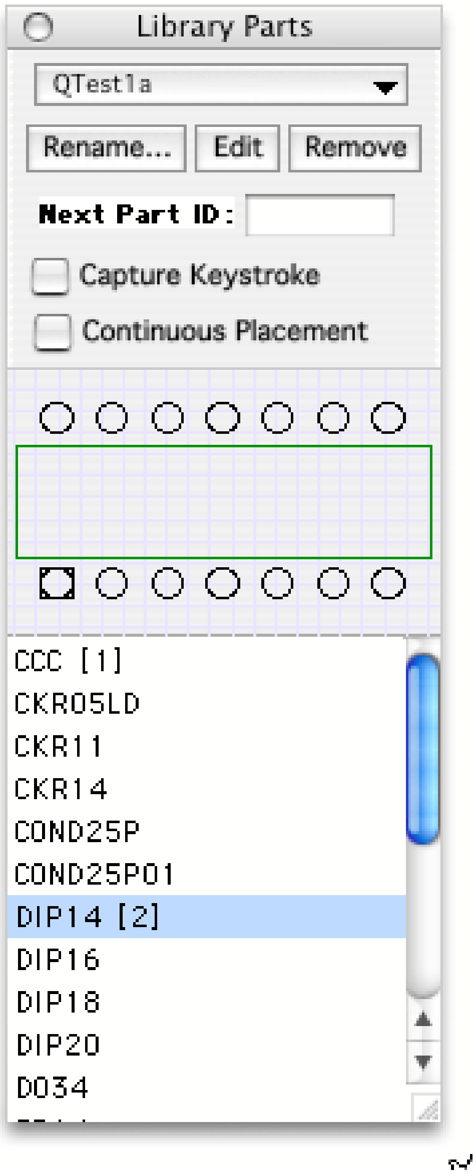


Figure 10.23: New Part Dialog

Chapter 11

Keyboard Commands

To permit rapid operations, the Osmond system supports a list of keyboard commands. Each is invoked by simply pressing a single key on the keyboard. Note that this does not mean pressing a key while holding down the **Command** key; this means just pressing the key. Keyboard commands are not active when entering text into a dialog text field or using the command line interface.

The list of keys associated with keyboard commands and their meaning is shown below.

1 .. 0	Go to layer n, where n is the number of the key pressed. Pressing 0 takes you to layer 10 if it exists.
ctrl 1 .. ctrl 0	Toggle the visibility of layer n, where n is the number of the key pressed while holding down the ctrl key.
C	Select the Connect tool from the tool palette
D	Select the Drag Peg tool from the tool palette
F	Select the Freehand Drawing tool from the tool palette
I	Select the Index tool from the tool palette
K	Select the Cut tool from the tool palette
M	Select the Move Parts tool from the tool palette
N	Show the next uncompleted net in the net list
Q	Select the Quick Route tool from the tool palette
R	Refresh the display
S	Select the Select Parts tool from the tool palette
T	Select the Turn tool from the tool palette
V	Show the previous uncompleted net in the net list
W	Select the Wrap tool from the tool palette
Z	Select the Zoom tool from the tool palette
?	Select the Info tool from the tool palette
+	Zoom in
-	Zoom out
/	Go to the command line interface
space	Select the Select Parts tool from the tool palette and refresh the display
F1	Undo/Redo the last operation
F2	Cut the currently selected objects
F3	Copy the currently selected objects
F4	Paste objects from the clipboard
home	Move view to top left hand corner of design area
end	Move view to bottom right hand corner of design area
page up	Move view up by 90% of the visible area
page down	Move view down by 90% of the visible area
left arrow	Move view left by 20% of the visible area
right arrow	Move view right by 20% of the visible area
up arrow	Move view up by 20% of the visible area
down arrow	Move view down by 20% of the visible area
Option left arrow	Move selected items left by one grid width
Option right arrow	Move selected items right by one grid width
Option up arrow	Move selected items up by one grid height
Option down arrow	Move selected items down by one grid height

Chapter 12

Command Line Interface

Some interactions with the Osmond system are best provided by a command line interface. To access the command line interface, press the / key on the keyboard. The message field at the top of the window indicates that it is ready to accept commands by printing:

Command:

Type your command at the automatically placed insertion point and press the return key. Commands can be concatenated with other commands in the same line if desired. In case of error, an error message will be displayed in the message field and all remaining commands in the command line will be aborted. A list of available commands is given in the section below. This is a preliminary list as I expect to add other commands as the need arises. Suggestions are welcome.

12.1 Command List

Command: dx dy movetn

Move all selected parts and freehand paths by dx units in the x direction and dy units in the y direction, where each unit is ten nanometers. There are 2540 of these units in a mil and 100,000 of these units in a millimeter.

Command: x y movepentn

Position the pen (used to create freehand paths) to an absolute location of (x, y) units, where each unit is ten nanometers.

Command: dx dy drawpentn

Draw a linear path starting at the current pen location to the relative location (dx, dy) units, where each unit is ten nanometers. At the end of the command, the position of the pen remains at the end of the path.

Command: x y orgtn

Place the grid origin at the absolute location (x, y) units, where each unit is ten nanometers.

Command: x xgridtn

Make the width of the horizontal grid x units, where each unit is ten nanometers.

Command: y ygridtn

Make the height of the vertical grid y units, where each unit is ten nanometers.

Command: flip

Flip all selected parts from one side of the board to the other.

Command: defer

Remove all selected parts from the board and place them in the Deferred Parts list.

Command: accept_mil f

Parse f as a floating-point representation, in units of mils, and place the equivalent integer, in units of ten nanometers, on the stack.

Command: accept_mm f

Parse f as a floating-point representation, in units of mm, and place the equivalent integer, in units of ten nanometers, on the stack.

Command: dialog_dim

Causes a dialog as shown below to be displayed. The value returned is placed on the stack.

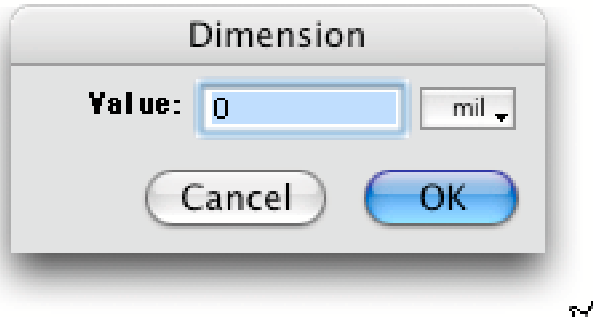


Figure 12.1: Value Dialog

12.2 Extending the Command List

The command line interpreter uses Ficl (Forth Inspired Command Language) by John Sadler. It is a fully functional Forth virtual machine. This means you can define your own commands based on combinations of the above command list and standard Forth words.

For example, I can define a command called move as follows:

: move accept_mil accept_mil movetn ;

Now, I can type the following command to move selected parts 25 mils to the right and 12.5 mils up:

move 25 12.5

To permit you to build a library of useful commands, the Osmond system on start-up will read a text file called *OsmondScript.txt* located in the same folder as the Osmond application (if is exists) using the Forth interpreter.

An example *OsmondScript.txt* file, developed by David MacDermot, is included in the distribution. This file supplies many useful commands for moving and drawing. Please read this file for more information.

For more information on Forth and Ficl, visit the following web site:

http://ficl.sourceforge.net

Chapter 13

File Formats

All data are stored in text files which can be viewed and manipulated with an ordinary text editor. Data items, such as part type definitions and path definitions, are identified by keywords which are then followed by descriptions enclosed in curly braces. This section describes the format of each of these data items.

The Osmond data file format is able to accommodate both English and Metric measurements. Whenever dimension values such as width or position are expressed in any of the formats described below, numbers without a suffix are to be interpreted as mils (0.001 inch) while numbers with an m suffix (like 12.345m) are to be interpreted as millimeters. Text height measurements with a p suffix (like 12p) are to be interpreted as points (1/72 inch).

All text starting from a # character and extending to the end of the line is ignored when the file is read. This can be useful for embedding comments in the file.

13.1 Part Types

The part type item defines the part including the number and placement of the pins, the geometry of the pins, the size and placement of the part name, and the placement of silkscreen and other paths. An example part type description is shown below for a simple CKR11 capacitor.

```
Type CKR11 {
Note "Capacitor"
```

```
NameData { 50 90 -25 0 Center }
Pad A { Circle 75 }
Pin 1 { A A A 0 0 32 P }
Pin 2 { A A A 350 0 32 P }
Path Silk
{ 80 50 W 5 270 50 270 -50 80 -50 80 50 }
}
```

The description begins with the keyword **Type** followed by the name of the part type (CKR11 in the example) followed by an open curly brace. The name can be any text string. Within the curly braces other keywords begin the descriptions of various features of the part.

The **Note** keyword indicates that a textual description of the part follows in double quotes. The description is for documentation purposes only and is not otherwise used by the Osmond system.

The **NameData** keyword begins a description of the size and placement of the name of the part that will appear on the silkscreen layer. Within the curly braces of this keyword, four numbers are found which represent in order:

The height of the name text. The X and Y coordinates of the beginning of the name text relative to the origin of the part in either mils or millimeters. The counter-clockwise rotation of the name text in degrees.

An optional justification keyword follows. This can be either **Left**, **Center**, or **Right** indicating left, center, or right justification respectively. If none of these keywords are used, left justification is assumed.

If the name of the part should not appear on the silkscreen layer, then the entire **NameData** description can be omitted.

The **Pad** keyword begins a description of a particular pad geometry that will be used later in describing the individual pins. The keyword is followed by by the pad geometry ID (A in the example above), followed by an open curly brace. Within the curly brace are shape and size descriptors. The shape of the pad is defined through the use of one of the following keywords:

- Circle
- Oval
- Rectangle

- Hexagon

The shape keyword is followed by one or two numbers which define the size of the pad. If the shape is a circle, the one number that follows defines the diameter of the pad in mils or millimeters. If the shape is not a circle, the two numbers that follow define the width and height, respectively, of the pad in mils or millimeters.

The keyword **Hexagon** defines a pad that is similar to an oval but has a somewhat different stayout shape. For more information, see the discussion in the description of the **Pad Change Tool**.

The size numbers may optionally be followed by the keyword **Spacing**, followed by a number, which roughly defines how much additional spacing, in mils or millimeters, is to be allowed this pad. For a fuller discussion, see the description of the **Pad Change Tool**.

To complicate matters further, each pad description can contain two independent geometries, one for when the pad has a connecting path, and one for when the pad is isolated. These are specified if needed by the keywords **Connected** and **Isolated** which precede the shape keyword. For example, the following is a complex but valid pad definition:

```
Pad A { Connected Rectangle 75 75 Spacing 5
        Isolated Circle 40 Spacing 20 }
```

A part type definition can have any number of pad geometry definitions although each must have a unique pad geometry ID.

The **Pin** keyword begins a description of a pin. The keyword is followed by the pin name followed by an open curly brace. The name can be any text string such as: 1, A, ANODE, DRAIN, etc., but should not contain embedded spaces or dashes. Within the curly braces are seven fields representing six items as follows:

- The ID of the pad geometry to be used on the front layer.
- The ID of the pad geometry to be used on the inner layer.
- The ID of the pad geometry to be used on the back layer.
- The X and Y coordinates of the pin, in mils or millimeters, relative to the origin of the part.
- The diameter of the hole to be used for this pin in mils or millimeters.
- The letter **P** if the hole is plated or **U** if the hole is not plated.

If a pin has no pad on a given layer (i.e., surface mount pad), use a **0** (zero) for the ID. Also, if the pin does not need a hole, use a **0** (zero) for the hole diameter.

A path description begins with the keyword **Path** followed by the name of the layer in which the path should be drawn followed by an open curly brace. The possible layer names are:

- Front
- Inner
- Back
- Silk
- BackSilk
- FrontAux
- BackAux
- FrontMask
- BackMask

Within the curly braces, pairs of numbers define the X and Y coordinates of the path, in mils or millimeters, relative to the origin of the part. A keyword **W** followed by a number (Xw) indicates that the path width changes at that point to Xw in mils or millimeters. A keyword **S** followed by a number (Xs) indicates that the path spacing changes at that point to Xs in mils or millimeters. If neither of the **W** or **S** keywords are found in a path, the default width and spacing of the path are zero.

A new open curly brace indicates the start of a separate branch in the path. The new branch path is specified by the coordinates within the new curly braces. When the close curly brace is reached, the old path resumes at the point of the branch. Each branch can have any number of other branches without limit. For example, the following defines a 20 mil wide Y shaped path on the front layer.

```
Path Front { 1000 1000 W 20 1000 1100 { 900 1200 } 1100 1200 }
```

A part type definition can have any number of path definitions.

A **Part Type** definition can recursively contain other **Part Type** definitions or references to other **Part Type** definitions. The **SubPart** keyword begins a description of a sub-part definition and is followed by either the name of a previously defined

Part Type as a reference, or the keyword **VOID** when no other part type is referenced. This is followed by curly braces that contain other information about the sub-part.

When the sub-part contains a reference to a previous part type, the only additional information needed is the relative location of the sub-part relative to the origin of the part. Otherwise, the sub-part may contain a complete **Part Type** definition.

The **Loc** keyword begins a description of the location of the sub-part relative to the orgin of the part. Within the curly braces of this keyword, up to four items are found that represent in order:

- The X and Y coordinates of the origin of the sub-part relative to the origin of the part, in mils or millimeters.
- The counter-clockwise rotation of the part about its origin in degrees. This item may be omitted, in which case the rotation is assumed to be zero.
- Either the keyword **Front**, which indicates that the sub-part is located on the front side of the board, or the keyword **Back**, which indicates that the sub-part is on the back side of the board. This item may be omitted, in which case the sub-part is assumed to be on the front side of the board.

To illustrate, the following shows a **Part Type** definition and a second **Part Type** definition that references the first.

```
Type Wye {
  Path Front
    { 0 50 0 0 { -50 -50 } 50 -50 }
}

Type TwoWye {
  SubPart Wye { Loc { 100 0 } }
  SubPart Wye { Loc { 200 0 } }
}
```

We could define an equivalent definition for part type TwoWye as follows:

```
Type TwoWye {
  SubPart void {
    Loc { 100 0 }
    Path Front
      { 0 50 0 0 { -50 -50 } 50 -50 }
```

```
  }
  SubPart void {
    Loc { 200 0 }
    Path Front
      { 0 50 0 0 { -50 -50 } 50 -50 }
  }
}
```

13.2 Part Definitions

The part definition item describes a new part instance including information such as what type of part it is and where it is located. An example of a part definition is shown below.

```
Part CKR05LD { Name CC1 Value "1000 pF" Loc { 4515 3622 -90 back } }
```

The description begins with the keyword **Part** followed by the name of the part type (CKR05LD in the example) followed by an open curly brace. The name of the part type should either match a previously defined **Part Type** definition or it should be the keyword **VOID**; anything else is an error. If it matches a previously defined **Part Type** definition, this part becomes a new instance of the referenced part type and contains all the features defined by that part type. If the name of the part type is **VOID**, this part is not an instance of a previously defined **Part Type** definition and so the only information about the part is whatever is found here in the **Part Definition**.

Within the curly braces other keywords begin descriptions of various features of the part.

The **Name** keyword is followed immediately by the name of the part (CC1 in the example above). This name can be any text string but should be unique within a design and should not contain embedded spaces or dashes.

The optional **Value** keyword is followed immediately by a string which specifies the value of the part. If the value string has embedded spaces, you should enclose it in double quotes, like this:

```
Value "10 K"
```

The **Loc** keyword begins a description of the location of the part on the printed circuit board. Within the curly braces of this keyword, up to four items are found that represent in order:

The X and Y coordinates of the origin of the part relative to the origin of the design area, in mils or millimeters. The origin of the design area is at the lower left. The counter-clockwise rotation of the part about its origin in degrees. This item may be omitted, in which case the rotation is assumed to be zero. Either the keyword **Front**, which indicates that the part is located on the front side of the board, or the keyword **Back**, which indicates that the part is on the back side of the board. This item may be omitted, in which case the part is assumed to be on the front side of the board.

A Part definition can recursively contain other Part Definitions or references to other **Part Type** definitions. The **SubPart** keyword begins a description of a sub-part definition and is followed by either the name of a previously defined **Part Type** as a reference, or the keyword **VOID** when no other part type is referenced. This is followed by curly braces that contain other information about the sub-part.

A part list file that could be input to the Osmond program would contain minimal Part Definitions like the following example:

```
Part CKR11 { Name C3 }
Part CON95 { Name J0M }
Part SIP10 { Name M91 }
Part CKR05LD { Name CC1 }
Part CKR05LD { Name CC6 }
Part DIP20 { Name M62 }
Part CKR11 { Name C1 }
Part RCR05 { Name R3 }
Part TO18/3T { Name Q1 }
```

13.3 Text Part Definitions

TBD

13.4 Signal Definitions

A **Signal Definition** provides the name of a signal and defines what pins are connected to that signal. Example **Signal Definitions** are shown below:

```
Signal "LATCHUP"
  { J0M-73 CR2-K }
Signal "+5V Power"
  { Q3-E Q1-E J0M-48 J0M-47 C1-1 }
Signal "***144053"
  { R3-1 Q3-B Q1-C }
```

The keyword **Signal** begins the Signal definition, followed by the name of the signal in quotes, followed by an open curly brace. Within the curly braces is a list of pins that are connected to this signal. Each pin is specified with the part name, followed by a dash, followed by the pin name.

A Net List file that could be input to the Osmond program would contain a series of Signal Definitions as shown above.

13.5 Path Definitions

TBD

Index