Procedure for PCBoard Layout

Introduction

The following 6 pages of instructions will take you step by step through the creation of your PCB using Orcad Layout. If you are planning to manually lay out your PCB, you should begin with **Section I**. **Section II** will guide you through the creation of your PCB from a preexisting schematic created using Orcad Capture. **Section III** will tell you how to create the data files that you will submit to the technicians who will make your board. **Section IV** will tell you how to print the layers of your design.

Section I: Manual Approach

System Settings

1. In the Orcad Layout Plus Demo go to the **File** drop menu and select New. You must then select a template (Try Default.tch for now) and click Open.
2. A window will appear asking for a Netlist Source. Click Cancel.
3. A black screen will appear containing a red information box. Hit the "O" key to zoom out (Use the "I" key to zoom in).
4. Go to the **Options** drop menu and select System Settings. In the system settings window make sure Display Units are mils (m). You may define the other setting to whatever values you prefer (Play around with it and decide what you like best). Press O.K.
5. From the **Options** drop menu select Gerber Settings. In the Gerber Preferences window change the output resolution to 2.3 Format and make sure the Incremental and CR After Each Block are **not** selected. Press O.K.

Create board outline

1. Click on the Obstacle Tool button in the toolbar.
2. Place the crosshair of cursor on the black field of the design window and double-click. The Edit Obstacle window should appear.
   - Make sure the Obstacle Type is Board Outline.
   - Change the Width to 62 mils.
   - Make sure the Obstacle Layer is Global layer.
   - Press O.K.
3. Begin drawing the enclosed border of the design. Don't worry about being exact because it can be edited later.

Place Components

1. Click on the Component Tool in the tool bar.
2. Place the cursor in the design window and right-click the mouse. Then select New from the pop-up window. An Add Component window will appear.
   - Make sure the Route Enabled box is checked.
   - Click on Footprint.
   - Select the appropriate footprint of the component from the Select Footprint window.

a) Click on Add

b) Select all the libraries (Select the first library, hold down shift, got to the last library in the last column and click on it. All the libraries should be selected.)
c) Click O.K. to any and all messages that pop-up.
d) Now select a library that contains the appropriate footprint of the component used.
   - For your information: Capacitor footprints, resister footprints, and diode footprints are in the libraries TM_CAP_P, TM_AXIAL, and TM_DIODE respectively. Footprints for Op-amps, opto-isolators, or any other multi-pin chip are in the DIP100T library. Transistor (and Triac) footprints are in the TO library.
e) Select the appropriate component footprint and click O.K.

- Place the footprint where you like.

Routing (Connecting pins)

Two methods of routing can be used. The first method (Method 1) is to connect the nodes with the Connect Tool and then allow the software to route automatically. The second method (Method 2) is to draw the routes manually.

- Method 1: Using the Connect tool
  1. Click on the Connect Tool in the tool bar.
  2. Clicking the right mouse button and select Add from the drop menu.
  3. Connect the pins: place the crosshair over a pin and press the left mouse button. Then place the crosshair over the pin you want connected and click the left button. When you have finished connecting the pins in that net (node), click the right mouse button and select End Command. Continue this process until all the nodes are connected properly.
  4. Now you must specify the track widths – we suggest track widths greater or equal to 50 [mils]. Click on the View Spreadsheet button in the toolbar. A pop-up menu will appear.
     - Select Nets and the Nets spreadsheet will appear.
     - Click on the cell containing "Width". All the cells in that column should be highlighted.
     - Right click the mouse and select Properties from the pop-up menu. The Edit Nets window will appear.
     - Change Min Width to a value greater or equal to 50. Change Conn Width to the track width >= 50mils. Change Max Width to a value greater than >50. Click O.K.
     - All the widths should change to the appropriate values in each row.
     - Close the spreadsheet.
  5. In the **Auto** menu select Autoroute, then Board. The software will perform the proper routing. When it is done, a pop-up window will announce that the task has been completed. Press O.K.
  6. If you need to edit the widths of any tracks created by the autoroute:
     - In the **Tool** menu select Track, and then select Tool.
     - While holding down the shift key, place the crosshair on a track (route) and press the left mouse button. Then right-click the mouse to open the pop-up window.
     - Select Change Width. The Track Width window will appear.

    - Enter the new width. We suggest >= 50 (mil). And mark the Net box (this changes all the tracks connected to the same node). Click O.K.
    - Repeat until all the tracks have been modified.

- Method 2: Using the Obstacle Tool
  1. Select the Obstacle Tool button from the toolbar.
  2. To initialize the drawing mode, place the crosshair over a node and double-click. The Edit obstacle window will appear.
     - Make sure the Obstacle Type is Free Track.
     - Set Width to the desired value (50mil)
     - Select the layer the track will apply to: Top (component) or Bottom (solder).
  3. You may now draw the tracks. Make sure none of the routes intersect.
  4. You may switch between layers by pressing "1" for Top, "2" for Bottom, or by selecting the desired layer from the toolbar.

- Go to **Section III. Creating Files**

Section II: Using Capture Schematic for PCB Layout

The schematic should have already been created and simulated to ensure that it works.

In Capture

1. While in Orcad Capture, open the Project containing the schematic that will be utilized in Layout.
2. Go to Project Manager and select the design or schematic (either will work).
3. From the **Tool** menu select Create Netlist and the Create Netlist window will appear.
   - Select the Layout tab
   - In Netlist File, enter the directory and file name desired. Make sure that it has an MNL extension.
   - Click O.K.

In Layout

1. Now open Orcad Layout Plus Demo.
2. Go to the **File** menu and select New.
   - The load Template File window will appear. Choose one of the technology or template files listed (we recommend Default.tch) or one that you have previously created. Click on Open and the Load Netlist Source window will automatically appear.
   - Select the .MNL file that you created and Open it. The Save File window will pop-up.
   - Save the file to a directory (A:\, D:\, or C:\TEMP). It will begin to import the Schematic and the *Automatic ECO Utility* will come up.
   - Another window may pop-up, called Link Footprint to Component. Click on Link Existing Footprint to Component. A pop-up window will request a new footprint that can replace the component's current footprint.
     * For your information: Capacitor footprints, resister footprints, and diode footprints are in the libraries TM_CAP_P, TM_AXIAL, and TM_DIODE respectively. Footprints for Op-amps,

opto-isolators, or any other multi-pin chip are in the DIP100T library. Transistor (and Triac) footprints are in the TO library.
Select a library and choose the appropriate footprint for each component it asks for.

- A black screen will appear containing the layout. Hit the "O" key to zoom out (Use the "I" key to zoom in).
- Go to the **Options** drop menu and select System Settings. In the system settings window make sure Display Units are mils (m). You may define the other setting to whatever values you prefer (Play around with it and decide what you like best). Press O.K.
- From the **Options** drop menu select Gerber Settings. In the Gerber Preferences window change the output resolution to 2.3 Format and make sure the Incremental and CR After Each Block are **not** selected. Press O.K.

3. Click on the View Spreadsheet button in the toolbar. A pop-up menu will appear.
   - Select Layers and the Layers spreadsheet will appear.
   - Click on the cell containing "Layer Type". All the cells in that column should be highlighted.
   - Right-click the mouse and select Properties from the pop-up menu. The Edit Layer window will appear.
   - Make sure Unused Routing is the only box selected and click O.K. All layer types should change to "Unused".
   - Now select the Top and Bottom layers. Right click. Select Properties from the pop-up menu and the Edit Layer window will appear.
   - Make sure Routing Layer is the only box selected and click O.K.
4. Now you must specify the track widths – we suggest track widths between 25 and 50 [mils]. Click on the View Spreadsheet button in the toolbar. A pop-up menu will appear.
   - Select Nets and the Nets spreadsheet will appear.
   - Click on the cell labeled Width. All of the cells in that column should be highlighted.
   - Right-click the mouse and select Properties from the pop-up menu. The Edit Nets window will appear.
   - Change Min Width to a value of 50. Change Conn Width to the track width to a the value of 55. Change Max Width to a value of 60. Click O.K. All the widths should change to the appropriate values in each column.
   - Close the spreadsheet.
   - View spreadsheet, open pad stacks, select pad width , right click and select properties, set pad width to 80, click O.K. , close spread
   -
5. Now click the Component Tool button in the tool bar.
6. Click on the components and move them to the desired locations.
7. Create Board Outline:
   - Click on the Obstacle Tool button in the toolbar.
   - Place the crosshair (cursor) on the black field of the design window and double-click. The Edit Obstacle window will appear.
   - Make sure the Obstacle Type is Board Outline. Change the width to 62 mils. Make sure the Obstacle Layer is Global Layer. Then press O.K.
   - Begin drawing the enclosed border of the design. Don't worry about being exact because it can be edited later. Right-click mouse and select End Command when finished.

8. In the **Auto** menu select Autoroute, then Board. The software will perform the proper routing. When it is done, a pop-up window will announce that the task has been completed. Press O.K.
9. In the **Auto** menu select Design Rule Check… and press O.K. (NOTE: Select All tests)
10. In the **Auto** menu select Cleanup Design. This will make the tracks "cleaner" (autoroute usually results in jagged tracks and this will straighten them).
11. If you need to edit the widths of any tracks created by the autoroute:
    - Go to the **Tool** menu. Select Track, then select Tool.
    - While holding down the shift key, place the crosshair on a track (route) and press the left mouse button. Then right-click the mouse to open the pop-up window.
    - Select Change Width. The Track width window will appear.
    - Enter the new width. We suggest between 50 and 60 (mil). And mark the Net box (this changes all the tracks connected to the same node). Click O.K.
    - Repeat until all the tracks have been modified.

Section III: Creating Files

Creating the Gerber files.

1. In the **Options** menu select Post Process Settings. A Post Process spreadsheet will appear.
2. In the **Window** drop menu select Tile. A Design window will appear that contains the your design.
    - Click on the cell, in the Post Process spreadsheet, containing *.TOP.
    - Right-click the mouse and select Preview from the pop-up menu.
    - The Top layer will appear in the Design window. Make sure it is correct.
    - Click on the cell, in the Post Process spreadsheet, containing *.BOT.
    - Right-click the mouse and select Preview from the pop-up menu.
    - The Bottom layer will appear in the Design window. Make sure it is correct.
    - Click on the cell, in the Post Process spreadsheet, containing *.GND.
    - Right-click the mouse and select Preview from the pop-up menu.
    - The Ground Layer should appear containing your board outline.
    - Click on Options in the menu bar and select Colors.
    - The Color window should appear. Click on and then delete all the items under the Data column except for Background, Board outline, and DRC Errors.
    - Make sure that DRC Errors is the only Data item left that is invisible (i.e. ///// should appear under the Color heading for DRC Errors). Close the Color window and the board outline should be all that appears in the Design window.
    - Click on the cell, in the Post Process spreadsheet, containing *.GND.
    - Right-click the mouse and select Save Colors from the pop-up menu.
    - Click on the cell, in the post process spreadsheet, containing *.PWR
    - Right-click the mouse and select Preview from the pop-up menu.
    - The Ground layer should appear containing your board outline.
    - Click on options in the menu bar and select Colors.
    - The color window should appear. Delete all items under the Data column except for Default power and Free text.
    - Right-click the mouse and select Restore Original Colors from the pop-up menu.
    - The entire design will be restored.

3. Click on the cell containing "Plot Output File Name". The other columns should now be highlighted.
4. Right-click the mouse and select Properties from the pop-up menu.
5. In the Post Processing Settings window the Format should be Extended Gerber, and make sure the Enable for Post Processing is **not** selected (Option block). Click O.K.
   All entries in the Batch Enabled column should be "No" and all the entries in the Device column should be "EXTENDED GERBER".
6. Now highlight the *.TOP, *.BOT, *.GND and .PWR and *.DRD rows.
7. Right-click the mouse and select Properties from the pop-up menu.
8. In the Options block you now select Create Drill Files, Overwrite Existing Files and Enable for Post Processing and click O.K.
   The top three cells and the bottom cell of the Batch Enabled column should now say "Yes".
9. Right click the mouse and select Run Batch from the pop-up menu.
10. Click O.K. to any messages that appear.
11. A notepad window will appear that tells you the files created. Make note of the file names and where they are located (should be "filename".bot, "filename".top, "filename".gnd, "filename".drd and "filename.pwr).
12. In Windows NT Explorer, go to the directory where the files have been saved. There are five files that must be transferred to floppy or zip disk and submitted to the technicians: "FILENAME".BOT, " FILENAME ".TOP, "FILENAME".GND, "FILENAME".DRD, "FILENAME".GTD and "FILENAME".PWR.

Section IV: Printing Layout Designs

Creating a Printout of Your Design

1. Perform steps 1-2 under the above section **Creating the Gerber Files**, if you have not already done so. Otherwise, proceed to step 2.
2. Click on the cell containing "Plot Output File Name". The other columns should now be highlighted.
3. Right-click the mouse and select Properties from the pop-up menu.
4. In the Post Processing Settings window the Format should be Print Manager, and make sure the Enable for Post Processing is **not** selected (Option block). Click O.K.
   All entries in the Batch Enabled column should be "No" and all the entries in the Device column should be "PRINT MANAGER".
5. Now highlight the *.TOP, *.BOT, *.GND, *.DRD and *.PWR.
6. Right click the mouse and select Plot to Print Manager… from the pop-up menu.
7. Make sure that the printouts show the correct layers (Top, Bottom, Ground (board outline), Drill layer and Drill File (with chart). Submit the printouts along with your disk containing the four Gerber files mentioned in the preceding section to the technicians.

Note: If you want your board to be fabricated and avoid any delays:
Track's width has to be minimum of 50 [mils]
Pad's width has to be minimum of 75 [mils]