

Creating Printed Circuit Board

Application Note
By Takahisa Nakahama
Team 12
RMS G-meter
March 28, 2007

Executive Summary:

In many electronic devices, PCBs (Printed Circuit Board) are used. These PCBs can be created using variety of different programs and tools. This application note will explain the process of creating PCBs using the software and tools provided by the ECE 480 Lab and the ECE shop.

Keywords: PCB
Cadence

Introduction:

PCBs are used in many electric devices because they are faster and easier than hand wiring when manufacturers mass produce a product. The creation of PCBs is done in few steps. First, you must have a final schematic for your design. Once you know that the schematic is correct and everything will function correctly, we are ready to create a PCB.

Objective:

Goal of this application note is to teach the reader how to create a PCB using Capture CIS and Layout Plus. After reading the application note, the reader should be able to create a PCB for their design using the resources at Michigan State University.

Creating a library using Capture CIS:

Once you have finalized your schematic design, you are ready to create a PCB. First program you will use is called Capture CIS. This program allows users to draw out a schematic so it can be blue printed later using a different program. Capture CIS can be found at: Start => All Programs => Cadence PSD 15.1 => Capture CIS. After opening the program, select PCB Design Studio with Capture CIS then create a new project (New => Project). When you start a new project, you will be asked to create a name for the project (in this application note, project name will be referred to as filename) then check the box next to schematic and make a directory to save your project then click OK. This will set up the project so user can draw out the schematic for the project. First step in making a schematic is to find or create a library. Creating a library is recommended since it is a simple task and you will have all the parts you need in one library.

To create a new library, select: New => Library and add the library to the project. To create parts, right click on the folder called \library1.olb in the filename.opj window and select New Part. Give a name and a prefix (example: Name = Resistor, Prefix = R) for the part and click OK. Now you should see a window containing a dotted box and the prefix of the part. Select place rectangle (forth from the bottom) from the vertical tool bar on the right side and create a box (Seems to be best to make the box size the size of the dotted box). Now click on place pin (third from the top) and new window will pop up. In the place pin menu, name and number the pin (example: pin, GND, Vdd) then select bidirectional in the type box as shown in Figure 1. Place the pins around the boxes.

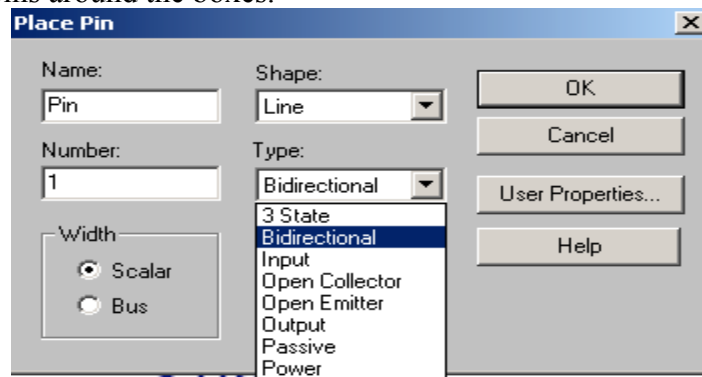


Figure 1.

Number of pins will depend on the number of pins used for that piece (if there are 28 pins on the PIC, create 28 pins). Lastly, save the part into the library. Repeat this process for all the parts used in the project.

Creating schematic using Capture CIS:

After the library is completed, go back to the schematic window to draw out the schematic of the project. Select Place Part (second from the top) from the vertical tool bar on the right hand side. In the Place Part window, find the library which was just created. To find the library, you may have to browse and add the library. After selecting the library, find the part to add to the schematic as shown in Figure 2. After placing all the required pieces, wire all the pieces together by selecting Place Wire (third from the top) and connecting each piece.

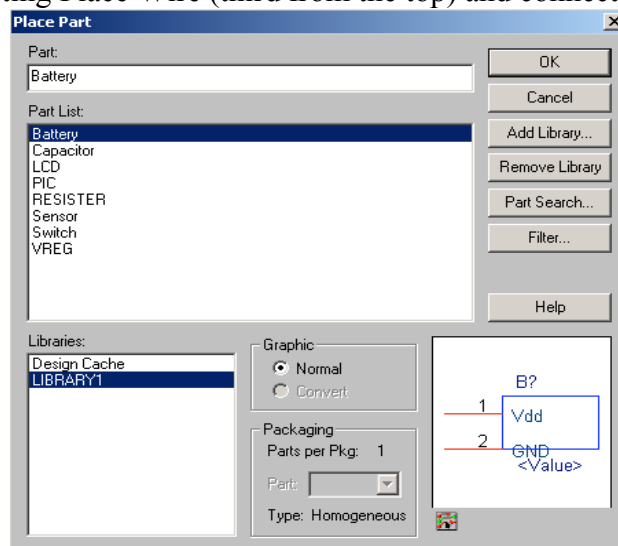


Figure 2

When wiring all the pieces together, be sure to wire them correctly and make sure there is no mistake. If there are mistakes, the circuit on the PCB will be wrong and the project will not function correctly.

Creating library in Layout Plus:

After drawing out the schematic, open Layout Plus (Start => All Programs => Cadence PSD 15.1 => Layout Plus). Similar to Capture CIS, you will need a library to work with. Once again, you can either create your own library or find the library within the software. Creating the library would be easier since you will have all the pieces required in the same library. To create a library, go to: Tools => Library Manager and new window should pop up. Click on Create New Footprint in the Library Manager window and give the part a name (example: PIC, Resistor). To create pins, select Pin Tool from the top tool bar then right click and select New and place the pins where they are necessary. For the spacing between pins, two dots on the grid equal the distance between two holes on the proto-board provided for the ECE 480 labs. After placing all the pins, create a border for the part by selecting Obstacle Tool from the tool bar and creating a

box around the pins (Example of a part is shown in Figure 3). Now save the part by selecting save in the Library Manager window.

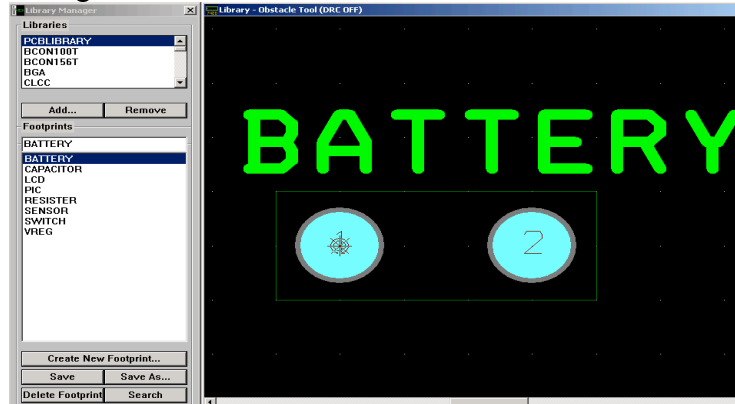


Figure 3

When saving the footprint for the first time, create a library and store all the parts you create into that library. Repeat the process for all the parts required in the design. If you want to modify the library later on, you must add the library to the library list to access your library.

Creating a Netlist:

Once the library is created, go back to Capture CIS program. Double click on a part from the library. Once the window pops up, select: Options => Package Properties. After opening Package Properties, put the name of the footprint created in Layout Plus library (be sure to copy the name exactly, it is case sensitive) into the PCB Footprint box. Repeat the process for all the parts in the design. After adding PCB Footprint on all the parts, replace all the parts in the schematic with the parts you just modified. When replacing the parts, make sure all the parts are connected to the correct wire. Next step is to cleanup cache of the library and the design. This can be done by right clicking on the library cache => cleanup cache for the library cache and design cache => cleanup cache for the design cache. After clearing cache, select (highlight) filename.dsn then go to Tools => Create Netlist. In the Create Netlist window, click on the Layout tab then save the MNL file to a directory. Once MNL file is created, go back to the Layout Plus program.

Blueprinting the PCB:

In Layout Plus, select File => New. AutoECO window will pop up. In the AutoECO window, click browse next to the first box and select _Default.tch (the file on the top left of the browse window) and click Open. Now select browse next to the second box and find the MNL file created in Capture CIS. Then check the box saying Use design library only which is found at the bottom of the AutoECO window. After checking the box, browse and find the library which was created earlier. Click on Apply ECO once all the set up is completed. A window will pop up saying Accept ECO. Click on Accept ECO. If the netlist do not match the parts created in the library of Layout Plus, the program will give you a different window. You cannot go on without matching the netlist with the parts in the library. When this window pops up, there are two things you can do. One of the steps you can take is to re-make all the parts in Capture CIS. When you

re-make the parts, make sure the name is exactly the same as the parts you created in Layout Plus. When you re-make your part, you must exchange the new parts with the old parts in the schematic. Another way is to manually search for the part in Layout Plus library that matches the parts in the netlist. First way seemed to work better for me. So I would recommend re-making all the parts first to try and match all the pieces. Once you have all the pieces matched, you should get a window with all of your parts and wiring on it. Now, line up all the parts by selecting the part and moving them. After you are satisfied with the placement of the parts, place a border around them by selecting Obstacle Tool. ECE Shop cannot make a PCB with inner layers, so you can only use top and bottom layers on your PCB. To eliminate these layers, go to View Spreadsheet => layers and double click on the inner layers and check the box saying Unused Routing. Now go to Auto => Autoroute => Board. By doing this, program should route your board for you. If you want to change the placing of the parts, you can undo routing by Auto => Unroute => Board. Once the board is created, save all the files as a gerber file. This can be done by going to Options => Post Process Settings. Make sure the device is set to EXTENDED GERBER for all plot output file name. Then right click anywhere on the table and select Run Batch. Now zip this up and e-mail or send it to the ECE shop and they will create your PCB for you.

Conclusion:

There are many ways to create a PCB. This is one of the easiest ways to create a PCB or printed circuit board since all the software and equipment is provided for you. Also, there are many sources you can use when you run into a problem. These sources include the help menu in the programs, manual for the program which is posted on the ECE shop website, and most importantly the staff in the ECE shop can help you as well.

References:

User's guide for Layout Plus: <http://www.egr.msu.edu/eceshop/pdf/layug.pdf>

User's guide for Capture CIS: <http://www.egr.msu.edu/eceshop/pdf/capug.pdf>

Help menu in Layout Plus

Help menu in Capture CIS