

## Starting and Learning the ORCAD PCB Layout Tools

By Steve D. Tucker

October 1, 2003

The first step in beginning the PCB layout is to learn the tools and terminology. The Cadence PSpice (formally ORCAD) package has a great tutorial for the schematic and layout tools. The schematic tool will create the netlist needed for checking the PCB layout. Making a PCB layout without a correct schematic is asking for trouble.

To start the tool, use the follow paths from the start button.

### For Capture CIS:

Start > All Programs > MOSAIC XP > Engineering > Electrical > Cadence PSpice > Capture CIS

When the dialog box appears select "Capture CIS Studio"

### For Layout Plus:

Start > All Programs > MOSAIC XP > Engineering > Electrical > Cadence PSpice > Layout Plus

Once the tool has opened start the tutorial but selecting:

Help > Learning Capture **or**

Help > Learning Layout

A window will appear for the Layout version. I have always selected "Layout Plus." You may want to learn the Engineer's Edition but it was not available when I first learn the software.

The tutorial will have a lesson menu with exercises. I could recommend that you complete all of them except the following:

Okay to skip on Capture CIS	Okay to ship in Layout Plus
Compatibility with SDT	Autoplacement
Using Capture CIS	
FPGA design flow	

These lessons are advanced features that you will not need for your first design. I found myself looking back at these tutorials many times during the design process.

### Accessing Manuals

I have some problems accessing the manuals in the tools. Under the Help menu there is also Manuals selection. When you select it a Java GUI pops up after a few seconds. When you select a manual the web browser comes up and it loads a file (either a pdf or an html file with links to a pdf). The problem was that I always got errors. The MOSAIC people are working on it and it appears to work correctly on some accounts. Here are some ways around it:

1. When the error page comes up in the browser, erase everything before the "i:/" in the URL. This will load the document into the browser.
2. Access the documentation through the directory path:  
I:\apps\winnt\psd142\doc

By the way, this path is worth remembering. If you make your own libraries (which you should) you will sometimes have to find your way back to the included libraries. Some useful paths are:

I:\apps\winnt\psd142\tools\layout\library  
I:\apps\winnt\psd142\tools\layout\data  
I:\apps\winnt\psd142\tools\capture\library

## Tips and Warnings

1. It is not a good idea to rename the part references in the layout. This will mess up the schematic and layout link unless you know how to annotate or back annotate correctly.
2. Learn to use the spreadsheet in Layout. A lot of the settings are done with the spreadsheets.
3. Route critical stuff before autorouting. Save a version before you autoroute. Autoroute is guessing by the computer expect to have to clean it up.
4. Using the nets spreadsheet to set the sizes of net traces. This will determine what size the autorouter will use.
5. If autorouter can not fit the traces in your board, try a different technology template. Use the Layout Help command to bring up the index and search for technology templates for a description of some of them. I think I used "2BET\_SMT.TCH" for the TMS430 Stiquito project. The technology template will load new rules for trace sizes and spacing sizes. Check these you may have to change some of them back to what you want before you autoroute (i.e. power and ground sizes)
6. In Capture, the power and ground are special symbols. You can find them under: Place > Power or Place > Ground after you have open a schematic.
7. If you update a part in capture after you have already used to you will need to replace or update the design cache. Highlight the part in the design cache and select one of the two options. Read info in the help index details about replacing or updating.
8. **Important:** When you make new footprints and save it in your own library, the AutoECO (Auto Engineering Change Order) of Layout will not be about to find it when it loads the netlist the first time. This is because it searches the network drive for the footprints and you can not save your library to that location. Choose a footprint from any library that it searches with the same number of pads. When the AutoECO is finished loading the design you can change the footprint of the part. With this in mind, you should already know which footprint goes with every part before loading the design. If you fail to select even one footprint for a part the design will not load. Here is my method:
  1. Complete the schematic in Capture.
  2. Print the BOM (Bill of Materials) so you have a list of parts.
  3. Open Layout and use the Library Manager to select footprints for part.
  4. Record the footprint name on the BOM.
  5. Create any need footprints and save in your library.
  6. Select a dummy footprint for the footprints you created and record name.
  7. Return to Capture and use the properties to designate the footprint. Use the names (including dummy footprint) you record from the Layout Library Manager. Save and create netlist info.
  8. Start new design in Layout. The AutoECO will search the libraries and load the footprints it finds. If it is missing any it will prompt you to select or create one. If you don't have one by the end of the AutoECO the loading is aborted and you have to do it again.
  9. Once the footprints are loaded, you can change the footprint for the parts you created.

For the Stiquito project you may not need to do a footprint design since I made the one for the Leg Assembly. If anyone finds out how to add your library to the AutoECO search list, please let me know.