

# Developing a PCB Board

In

## PADS Layout

Micajah Worden

11-15-10

### Executive Summary

Pads is a powerful tool to develop PCBs. This document will provide a look into some of the basic functions of this software, which will allow you to develop your own designs and have them fabricated.

Key Words: Layer, Via, Net, PCB, Gerber, Netlist

## Introduction:

Printed circuit boards, or PCBs, are used in a wide variety of applications. From cell phones to microwave ovens, virtually every electrical device utilizes PCBs to expedite the development of their projects with minimal human assembly of the final product. PCBs allow a large amount of circuitry to be placed in a small area, with many different options for the number of layers on the board and thus how densely you can populate the board. They are milled, machined, and assembled by machines and can be produced in great quantities. Though the initial cost of development is high, with a large quantity for production, the cost per unit goes down as well as the likelihood of error inherent in human assembly. For this reason it is a valuable skill to be able to design your own boards. While at first tricky, with the help of this document, the help files included in the PADS Logic software package, and a little practice you will be able to develop your design, export Gerber files, and have a company fabricate your board for production.

## Objective:

At the completion of this document, the user should have the basic tools to utilize this software to develop a two layer PCB board, as well as check the project for errors, and export the necessary files to have the board fabricated. It is assumed that the user has a schematic developed for their project, and a basic understanding of electrical principles. Developing new components for the PCB board, as well as developing schematics in PADS Layout are outside of the scope of the project. For each of these operations the user can view the help files and tutorials included in the software package.

## Starting a Project:

Start a new project by going to File>new. A black screen marked with a hashing of dots should appear. To modify the layer definitions of your project go to Setup>Layer Definition. A Layer Setup dialog box will appear. The default number of layers is two. That is all you will need for this project. There are many options in this dialog box for layers. The first two layers labeled Top and Bottom are the electrical layers of the PCB. This is where all the traces and components will be located. The CM under Type specifies this as a component layer. Further down the layer list, in the nonelectrical layers, the silkscreens, drill drawings, assembly drawings, and soldermasks for the top and bottom layer of the board are located. These are necessary layers for production of the board.

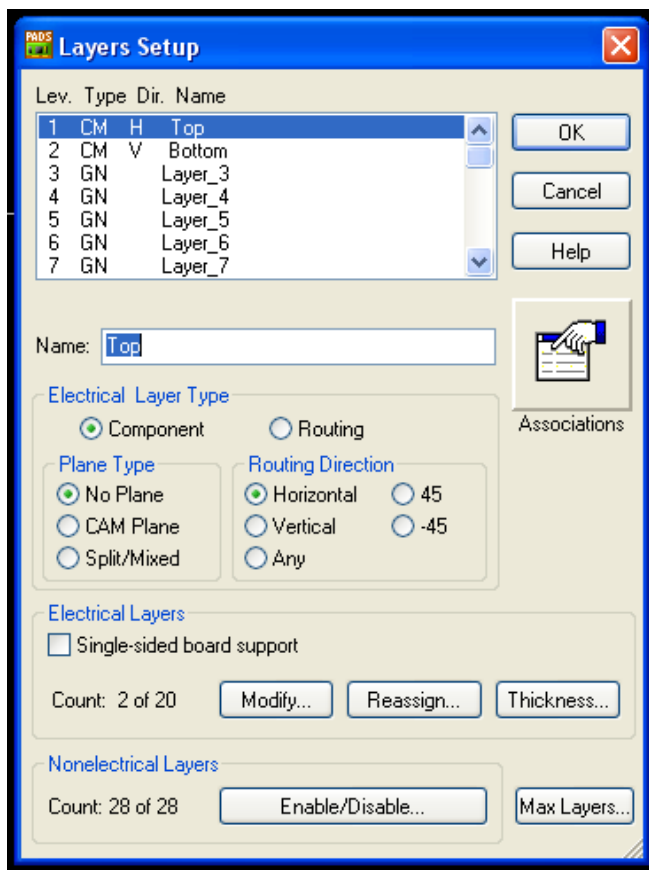
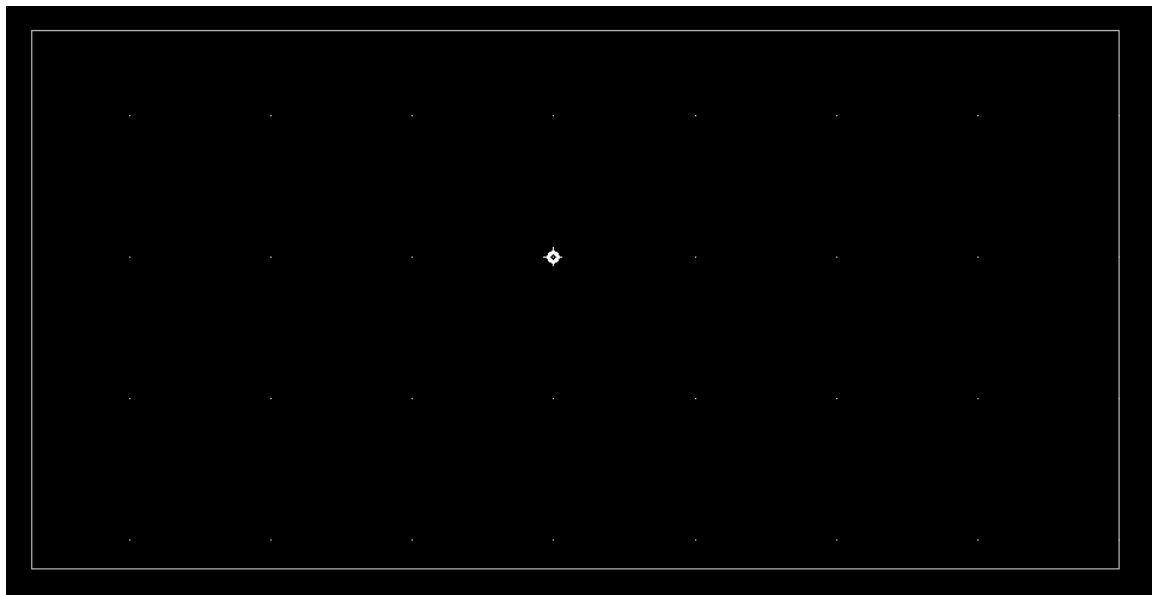


Figure 1: Layer Setup dialog box.

## Creating a Board:

Now that the layers are specified a board must be made to construct your PCB design. This can be done by selecting the Drafting Toolbar on the Toolbar. Next select Board Outline and Cutout. There are three options for general board shape; Polygon, Circle, and Rectangle. Select rectangle. Left click on the workspace and create a rectangular workspace. Now a board outline has been created.



**Figure 2:** Drafting Toolbar icon, Board Outline and Cutout icon, example of board outline in the workspace

## Design Rules:

Design Rules specify the minimum distance between traces, Vias, and pins. For this you must know the capability of the PCB fabricator you will be using. To modify the design rules for the PCB simply select the Setup Menu, and click on Design Rules. A Design Rules dialog box will appear. Click on Default rules. By selecting Clearance you will be able to adjust minimum

spacing between many different types of components. A matrix is located within this dialog box, with eight different categories for design rules. Adjusting these will help when fabricating your board if you set them greater than the minimum resolution of the PCB fabricating machine. Once you've checked the level of precision of the machines that will be fabricating your boards you can safely enter these values, to ensure that you have created a design that the machines can handle.

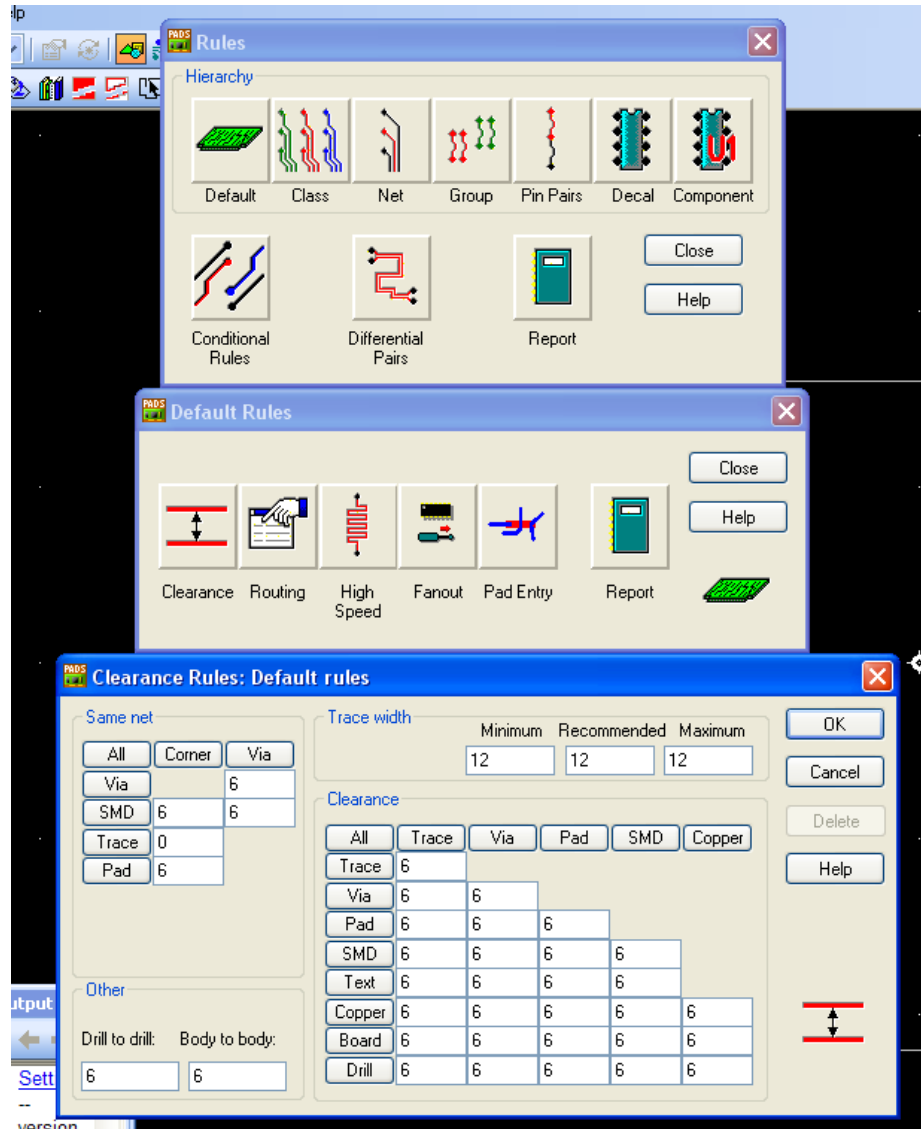


Figure 3: Clearance Rules Dialog box.

## Adding Components:

So far the board outline has been created, the layers have been specified, and design rules now govern the PCB design. Now it is time to add components to your design. Keep in mind that components can be mounted on both the top and the bottom of this two layer board, so there is no reason to crowd one side or the other. Components mounted on the Top layer will appear as red and components on the Bottom layer will appear as blue. All components will appear on the primary side of the board at the origin, and must be moved to their desired location. PADS has a large library of standard components, including resistors, inductors, capacitors, and many ICs. If your part is not found in the parts library you must create the part. This function is outside the scope of this application note, but many resources can be found in the help file of the PADS software package to aid you in this endeavor. To place a part Press the ECO button on the Toolbar. A dialog box will appear. Press the OK button. In the ECO Toolbar select ADD Component. There are many libraries to select parts from including ICs and common components, but to view the parts you must type the first letter of the part name followed by an asterix e.g. R\* to search for resistors. Select the desired component from the library and you will see an example in the Preview Area. Add the desired part to your design and place it in the desired location. The part name should appear in the component list in the Project Explorer bar. If you would like to place the component on the other side, select the component in ECO Mode, right click the component and select Flip Side. If you are importing a Netlist from a schematic parts in the library will be uploaded to the workspace automatically. You can change the name of components in ECO mode by selecting Rename Component in the ECO Toolbar. Components can also be deleted in by selecting Delete Component from the ECO Toolbar.

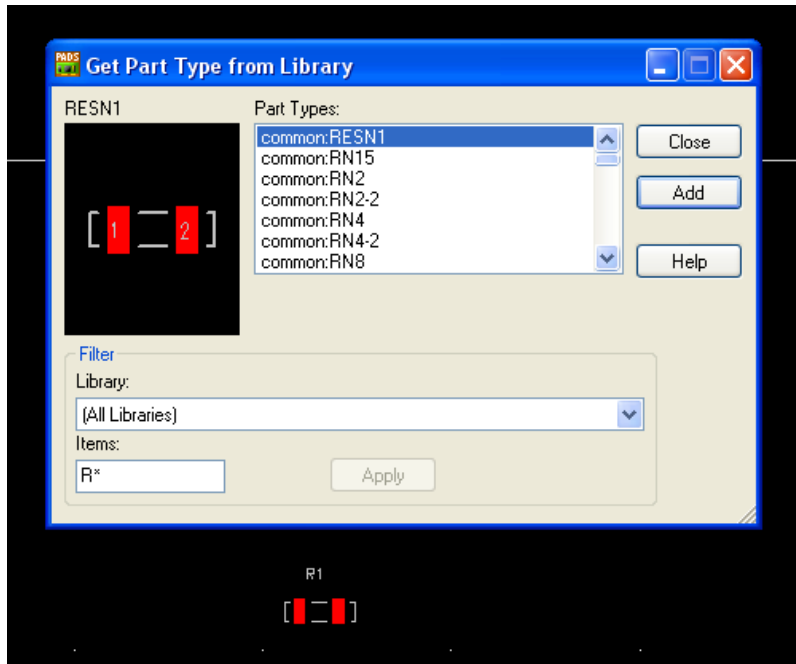


Figure 4: ECO Toolbar Icon, Add Component Icon, Rename Component icon, Get Part From Library Dialog box as well as part placed in the Board Outline.

## Creating Net Lists:

In PADS there are numerous ways to create Net Lists. For simple designs with few components this can be done in ECO mode by selecting the Add Connection icon from the Toolbar. Simply wire the desired pins together and a Net will be created, or the connection will be added to an existing Net if one of the pins is already part of a Net. Once the Net association is created, a line will appear between the pin pair. This will guide you in your wiring. If you have created a new Net, it is advisable to rename the net to something pertinent to the design such as GND, VCC or other meaningful names. This is done by selecting the Rename Net icon in the ECO Toolbar. Connections can also be deleted in ECO mode by selecting the Delete Net icon from the Toolbar and left clicking on the Net from the Nets menu in the Project Explorer window located at the

left of the screen. This is also useful for editing Nets that you have imported from a schematic and need to modify.

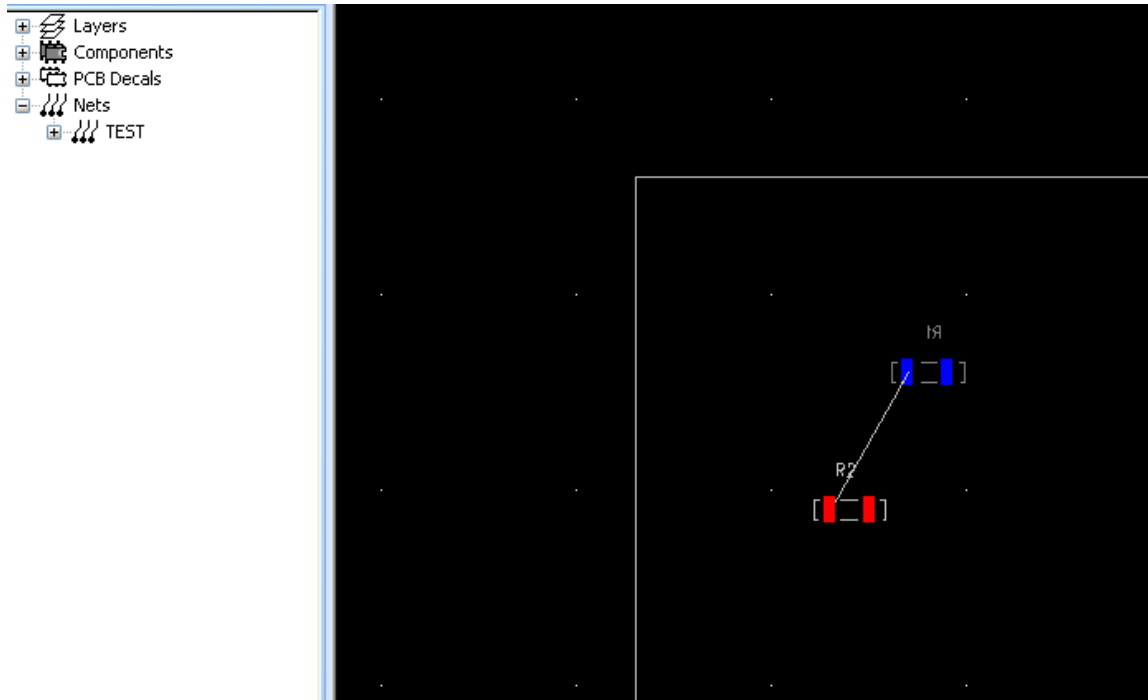


Figure 5: A Net between pin 2 of R1 and Pin 1 of R2 labeled TEST in the Project Explorer window

## Importing a Schematic Design:

If you have a completed a schematic file, this will aid you in completing your design much quicker than developing a Net List in ECO mode, and adding all components individually. To do this select file>import and search for your schematic design Netlist. Select Open, and all the parts and decals from the Netlist contained in the library will be at the origin of your board. Refer to the adding components section of this document and this will tell you how to move your components and place them in the desired location.



## Wiring the board:

Now that the parts are placed and the Nets are set, it is just a matter of wiring the board. It is important to keep in mind that red traces cannot cross red components or other red traces, because they are all located on the Top layer. Blue traces cannot overlap blue components or other blue traces because these are all on the Bottom layer. Red traces can pass through blue components and blue traces, and blue traces can pass through red components and red traces. To switch wires back and forth from top to bottom side wiring, a Via is used. Start wiring by selecting the Select Mode tab in the ECO Toolbar and double click on the desired pin of a pin pair connected by a Net connection. The Net line connecting the pin pair will follow the end of the connection you are making, and end at the closest destination pin. If the starting pin is on the Top layer a red wire will be formed. To wire a component to a pin on the other side, right click and select add Via from the menu. A circle will appear on the schematic, and you can create wires on the top or the bottom layer from this point. To end the connection on the desired pin, double click the pin. It is best to avoid right angles, because these are hard to fabricate and are inefficient with the space on the board.

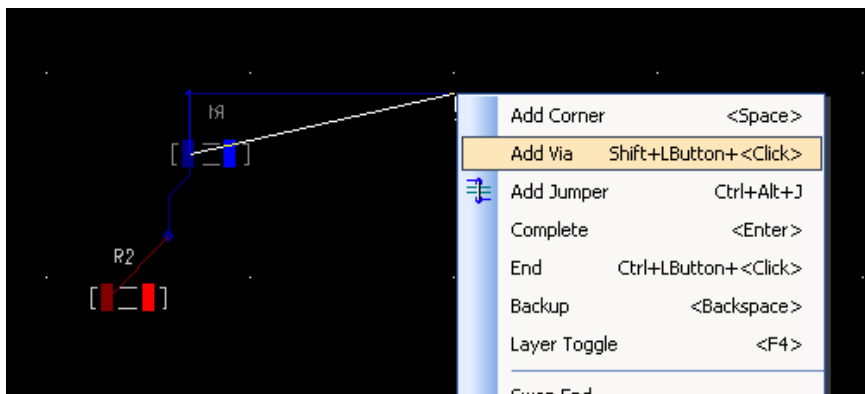


Figure 6: Wiring of Test Net from Figure 5, Add Via example

## Verify Design:

The board is wired, and now it must be tested to make sure there are no clearance errors, and missing pin pair connections. To do this select Tools, and then Verify Design located in the scroll

down menu. A Verify Design box will appear with many options. It is best to run all these tests, to ensure there are no errors on the board. When an error appears after starting the test, it will be highlighted by a circle, and will appear in the Location field of the Verify Design box. There is also an explanation of the error. Fix the error and rerun the test to verify that the problem has been resolved.

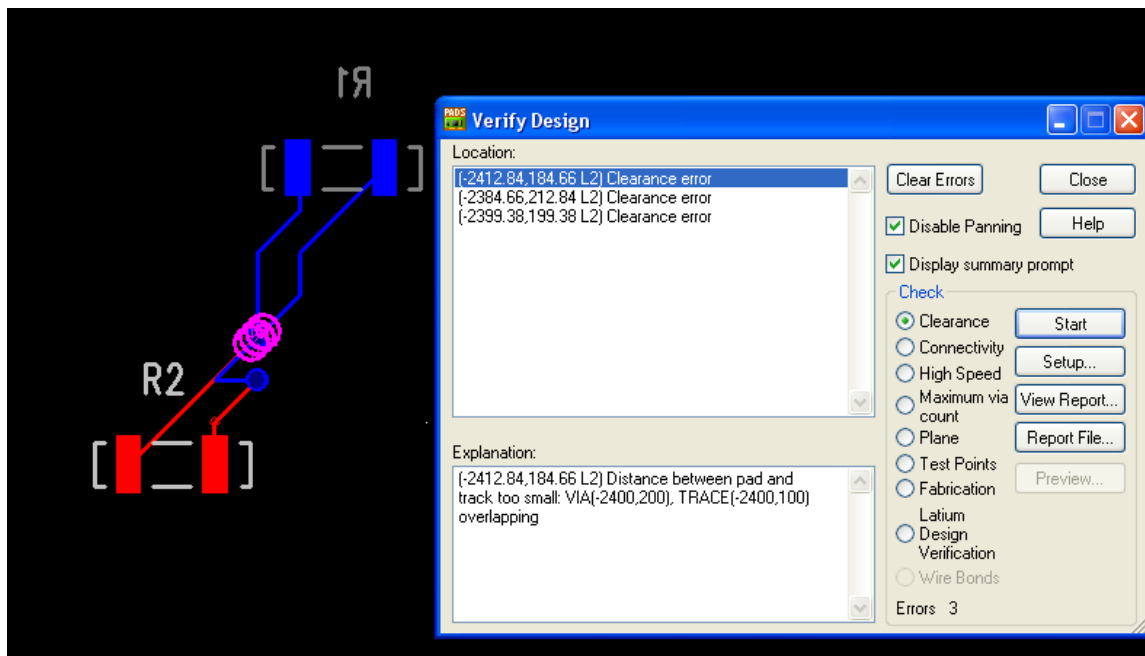


Figure 7: Verify Design dialog box as well as Clearance errors

## Gerber Files:

The board design is now completed, and has been verified by the software. To get the necessary files to produce the board, select File. Next select the Cam option in the drop down menu. The Define Cam Document dialog box will appear. Select the Add button and the Add Document dialog box should appear. Type in the name of the Document located in Document Name field of the Define CAM Documents dialog box. Select Preview Selection in the Add Documents dialog box, and an example of your Cam Document will appear on the screen. If everything appears to be in order press OK in the Add Documents dialog box. Select the desired

files required for the company that will be fabricating your board. Typically these include the drill files and the Top and Bottom silkscreens and soldermasks. Select Run and the files will be generated in the CAM folder located in the PADS folder. To export your Netlist, select Export in the File menu. In the Save as type field select the .ipc file and press save. All the necessary files have now been created to get the board fabricated, and it is just a matter of finding a company to do the fabrication, and acquiring parts.

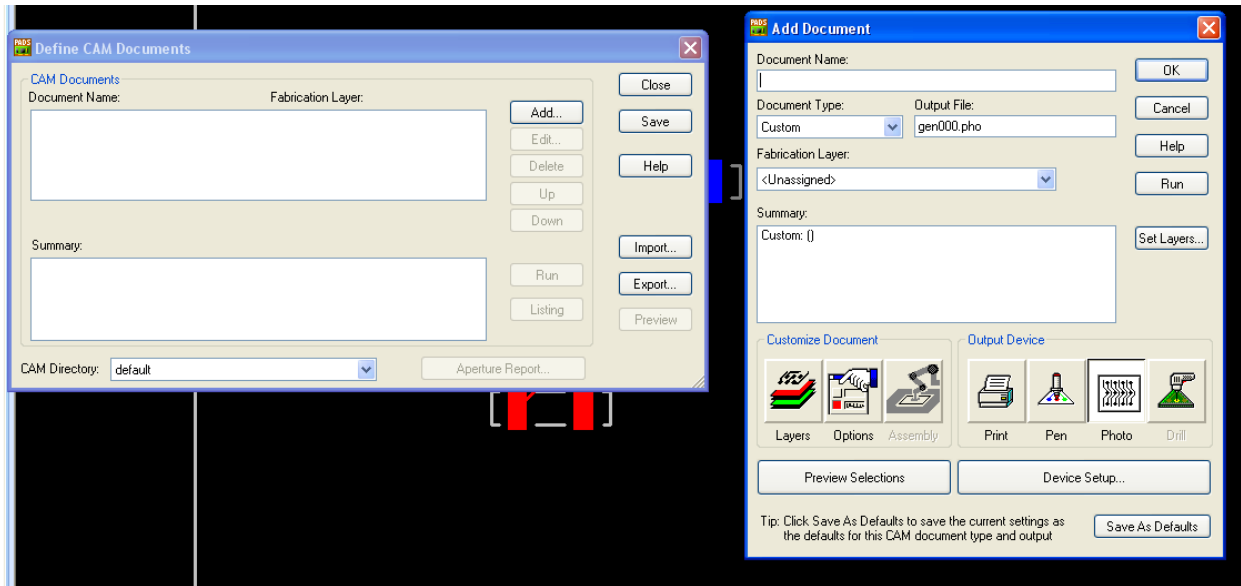


Figure 8: Define Cam and Add Document dialog boxes



Figure 9: Exporting Netlist called appnote.ipc

## References:

“PADS Layout Help Files” Mentor Graphics

Accessed 11/19/2010

“Learning the PADS Layout User Interface” Virginia Commonwealth University

[http://www.people.vcu.edu/~rhklenke/tutorials/PADS/PADS\\_Tutorial\\_User\\_Interface/UserInterface.html](http://www.people.vcu.edu/~rhklenke/tutorials/PADS/PADS_Tutorial_User_Interface/UserInterface.html)

Accessed 11/18/2010

“PADS Layout Datasheet” Mentor Graphics

[http://www.tsieda.com/data/Datasheet\\_PADS\\_Layout\\_1022760%20\\_05\\_09.pdf](http://www.tsieda.com/data/Datasheet_PADS_Layout_1022760%20_05_09.pdf)

Accessed 11/18/2010