

PCB Design utilizing Cadence Software

Application Note

Kyle Schultz

11-9-11

ECE 480

Design Team 5

Keywords: *Schematic, PCB, Fabrication, Cadence, Design Entry CIS, Allegro*

Table of Contents

Abstract	<u>1</u>
Introduction	<u>1</u>
Objective	<u>1</u>
Getting Started	<u>2</u>
Schematic Placement	<u>3</u>
PCB Creation	<u>4</u>
Fabrication Preparation	<u>10</u>
Conclusion	<u>12</u>
Recommendations	<u>12</u>

Abstract

The purpose of this application note is to educate the user on the basics of PCB design using Cadence software, provided to the students at Michigan State University as well as any individual who has access to the complete Professional Edition of the software. The Cadence software package is an extremely useful tool utilized for development of Printed Circuit Boards (PCBs). At first, this package may seem difficult to learn, but once the user begins to use the software, the learning curve begins to change.

Introduction

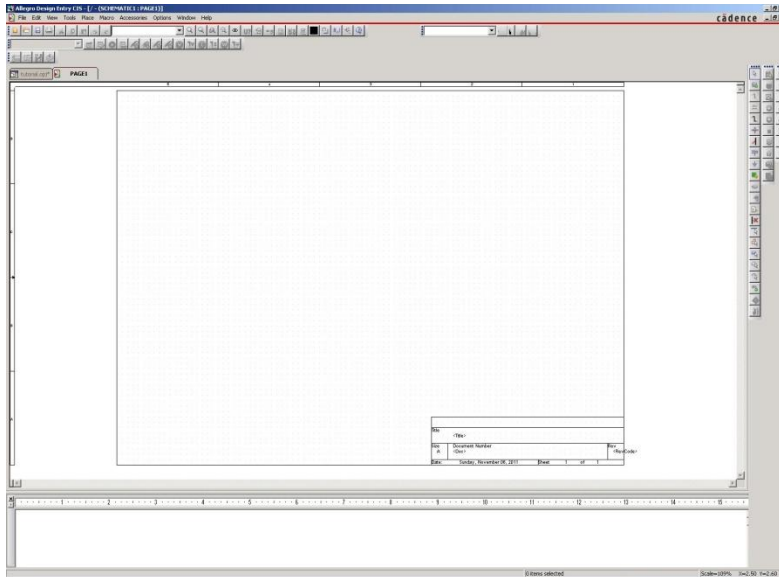
Having the skills necessary to develop individual PCBs is advantageous to any student planning on starting a career in Electrical Engineering. The reason is because Printed Circuit Boards can be found in any commercially distributed item that includes electronics. Looking at PCB manufactures and their prices, one may notice that the cost to produce a single board is high. This is true, however one should note the number of boards is inverse proportional to cost. Meaning that as the quantity of boards increase, the cost per board will decrease. It is of great importance to also note that the complexity of one's board greatly determines the cost of fabrication. If the users PCB is only single layered, the cost to fabricate will be much less than one who has a 4 layer board.

Objective

The objective of this application note is to allow the user to begin using the Cadence software in order to create a simple PCB. One will also be able to check their design against the rules that have been established within the software. This application note was written under the assumption that the user has a basic understanding of the design process. One should understand the fundamentals of component placement, board routing, and proper layering of the board.

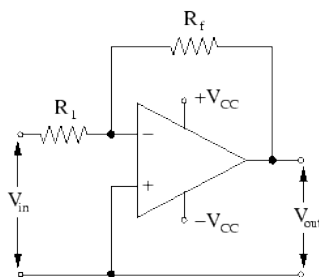
Getting Started

To begin the design process, the user will be using Cadence Design Entry CIS. Once the program has been opened, go to File>New>Project. In the small window that pops up, assign a name for the project that is easy to remember. Under Create a New Project Using, make sure to mark Schematic. At the bottom of the screen is a dropdown box that is labeled Location. Click the Browse button and assign the project to a directory that you will be able to locate later, then press OK. Again, go to File>New>Design. This will associate the schematic file to this project. A blank sheet will be displayed named PAGE1 and is linked to the project.

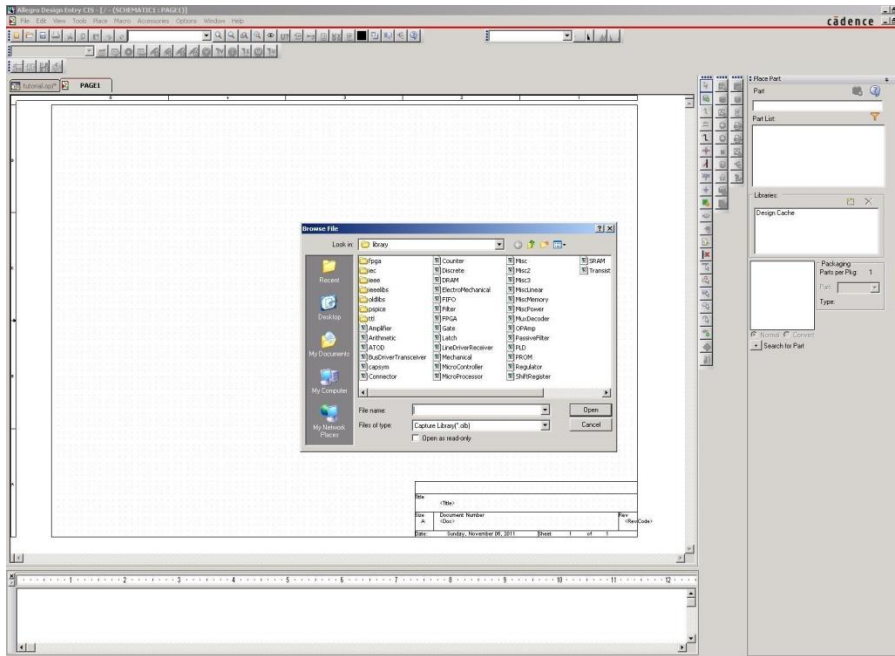


Schematic Placement

If the above steps have been followed correctly, the project should be ready for the schematic design process. For the purpose of this application note, the user will be creating the circuit below:



The first task that needs to be completed is placing the components onto the screen. Design Entry CIS has a wide selection of components already installed in the default library. Please note, that if this library does not include a particular part being used in your circuit, it is very easy to locate these components on manufacturer websites and import them into the library. At the top of the screen, go to Place>Part and a sidebar will appear including the libraries already embedded to the design. When creating a new drawing, there will be no Libraries in this area. To add a new library, click on the square box above Design Cache and a list of all the libraries will pop up.

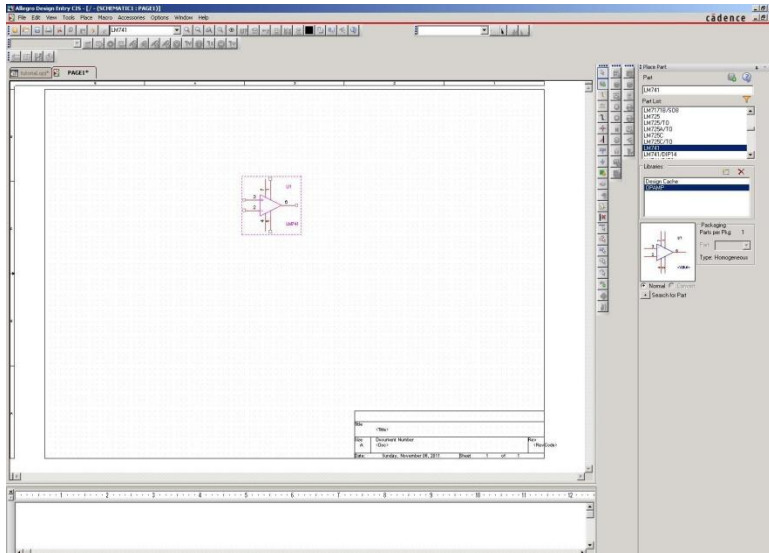


For the circuit that will be created, we will need:

LM741 OP-AMP

Two resistors

Click on OPAMP then open. The Op Amp Library should now be listed under Design Catch. Now Go above to the Part List and scroll down until you see LM741. Double Click on this component and place it onto the screen. If one was to need multiples of the same objects, they could place them all on the screen now because once a component is selected; it will be available for placement until the escape but is pressed.



Now the resistors need to be added to the library. Go to the add library button again, find the Discrete file in the previous library and add it to the list. There are a lot more components in this library so to make it easier to find one, click inside the bar below Part and type “resistor”. There will be many options to choose from but for the purpose of the application note just select resistor. Two of these components are needed, so add them to the screen and press Esc. To edit the properties of each resistor, right click on the symbol, and select edit properties.

After the elements have been modified for the design, the schematic can now be wired up. To move an element around on the screen, simply click in the middle of the element and move to destination. If any device needs to be rotated, simply right click that device and select rotate.

The power and ground symbols can be found on the icon bar located to the right of the schematic.



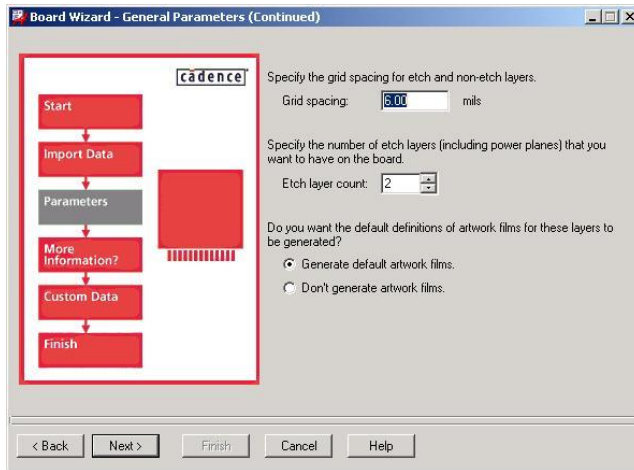
To add a wire to the schematic, you can find the icon in the same location listed above.

PCB Creation

Now that your schematic is complete, the next step is to create the PCB file. Go to File>Export Design. Make sure you are in the EDIF tab, in the Save As path save the file as the same name as the project.EDF. Click Browse for the Configuration file, and in the capture folder select allegro.cfg then press ok.

Open PCB editor, select File>New and in the second option space select Board (Wizard). A box will open explaining what this program will do. Select Next. Since we do not have a template, click on "No" then "Next". Select Next two times since we do not already have the

information they are requesting. Now the user should be at the General Parameters. For Units, select Mils, for drawing size select A. It is up to the user where they would like to have the origin of the drawing specified. We will be using “at the lower left corner”. The next screen will ask for Grid spacing, select 6.0 mils because that is common for most fabricators. Make the Etch layer count 2 for this application note. Since we do not have artwork, select Generate art forms.

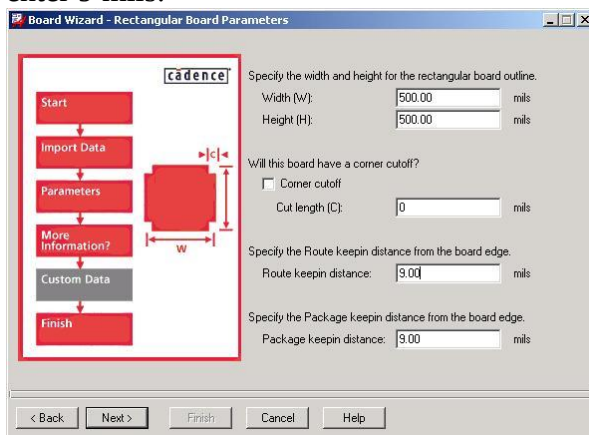


The next screen is to specify the Etch layer properties. For both top and bottom select Routing layer.

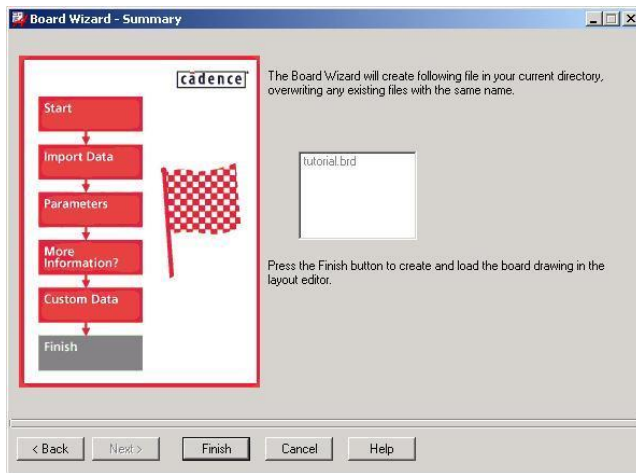
The following screen is for Spacing constraints. This is where the user will specify Minimum Line width, Line to Line spacing, Line to Pad spacing, and Pad to Pad spacing. To be safe, enter “7” in each space. At the bottom of the screen, you need to specify the Default via padstack. Browse until Smd110rec23 is found, select and press ok.

The next page will prompt you to define the board outline. We will be using a rectangular board for this design.

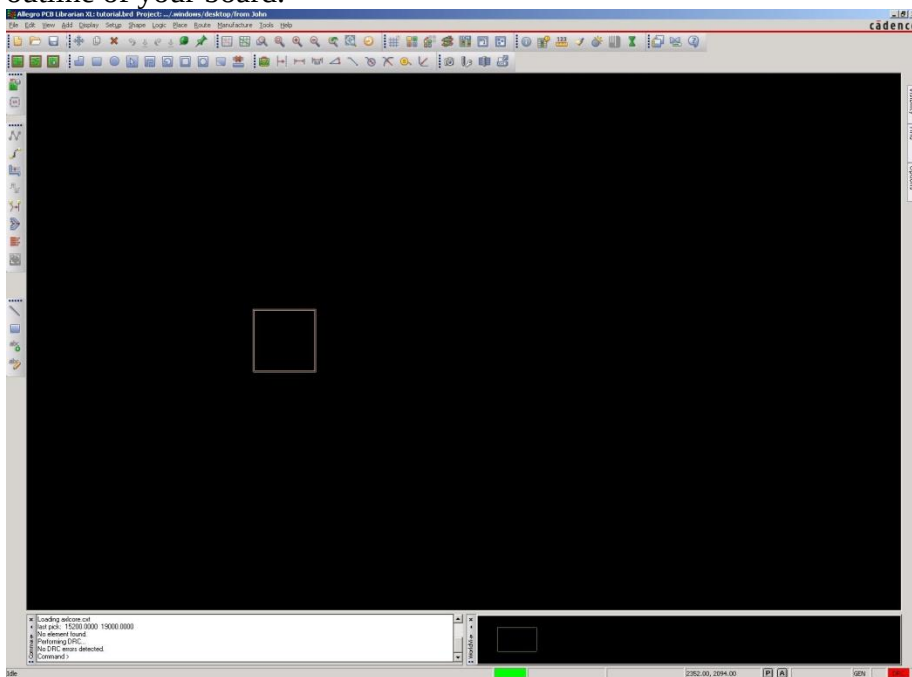
The next screen is intended to customize this data. For Width of the board select 500 mils, height will also be 500 mils. Ignore the next option asking for a corner cutoff, we will not be utilizing this option. For Route keepin the user can enter 9 mils and for Package keepin also enter 9 mils.



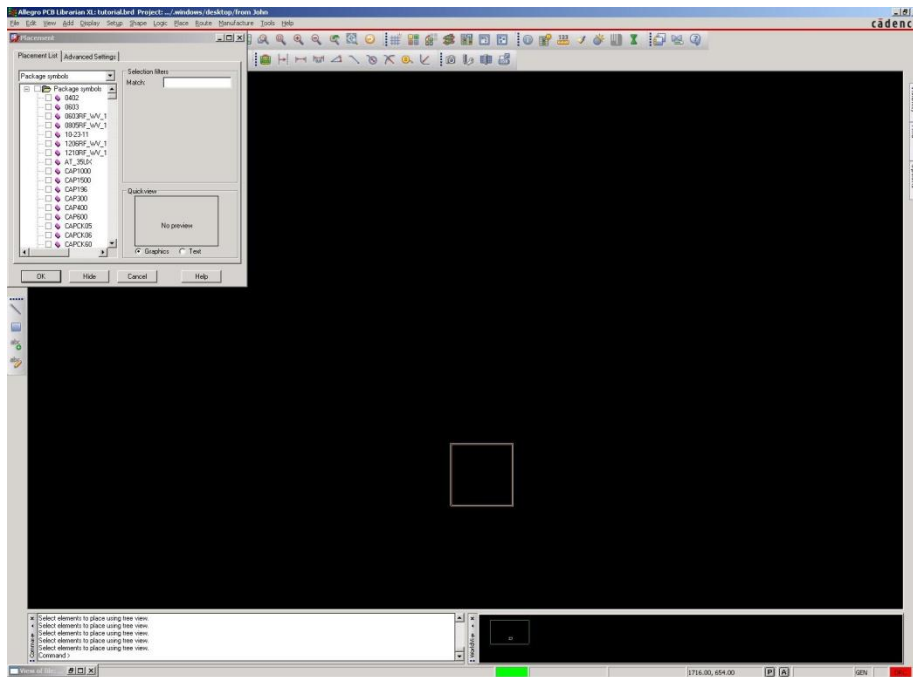
The following screen should tell the user that a .brd file will be created in their current directory. This is critical to the success of this application note. If no such file has been created, please press the back button and check your previous steps. Once everything is correct click finish.



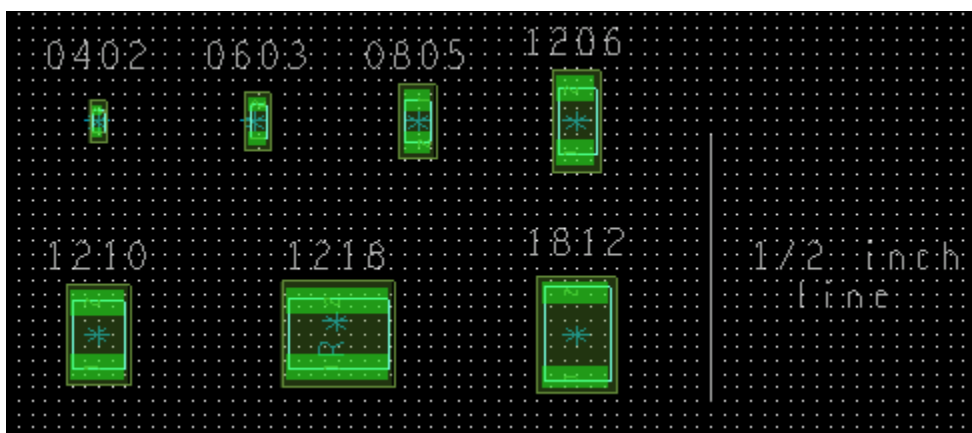
The screen will close and there should be a square placed on your screen now. This is the outline of your board.



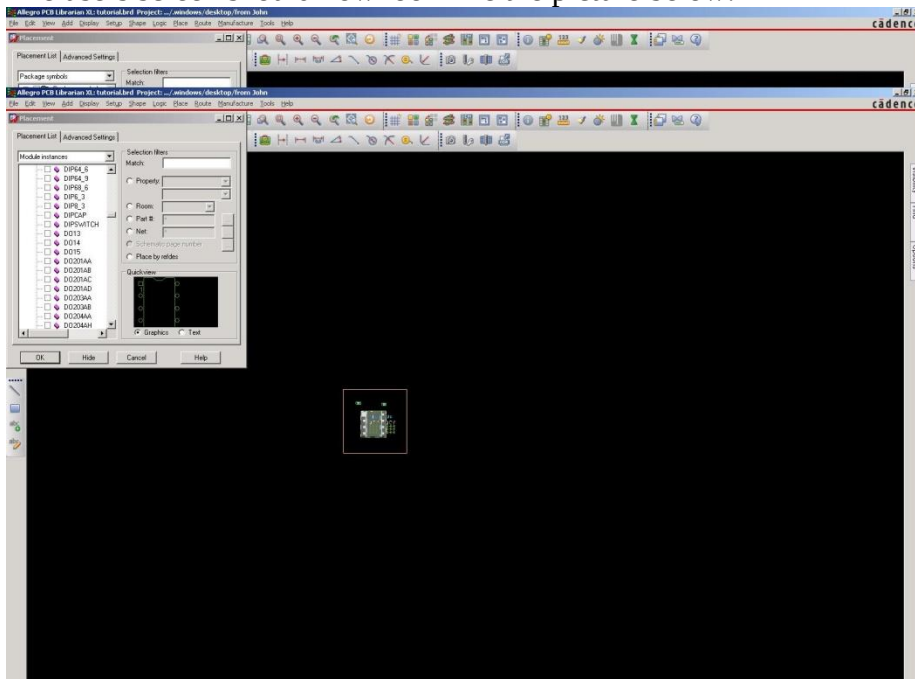
The user is now ready to start placing components on the board. At the top of the page select Place>Manually and under the Placement List tab, select Package symbols in the dropdown box.



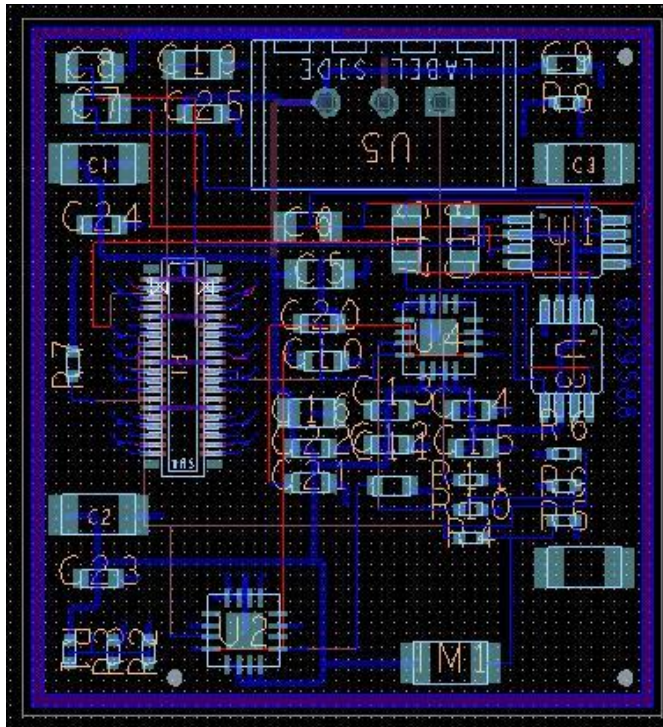
The LM741 package that will be used in this application note is an 8-DIP and the resistors will be surface mount packages. Scroll through the selection of packages and select DIP8_3. The resistor package that will be used is the 0402, which is at the top of the list. For future reference, one should take in to account the package sizing for resistors, depending on the complexity of design. When designing a PCB for the first time, one might have only used through hole resistors in prototyping a circuit, but when spacing is a major concern, one could reduce the area of the schematic greatly by using smaller size resistor packages. Below is a physical representation of this criterion.



The users screen should now look like the picture below:



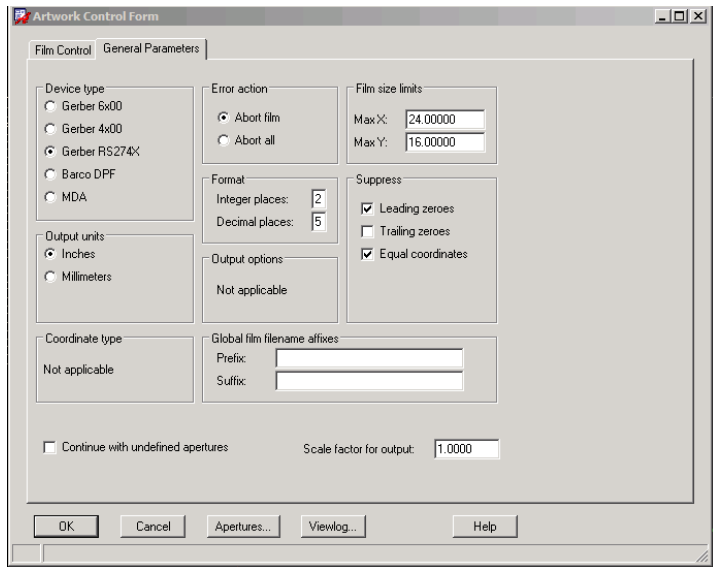
After this step, one should locate where they would be putting the power source and ground grid and place them into the PCB. The user is now complete with the PCB design process. After using the software and learning all of the capabilities, one could go from designing a simple circuit like the one done in this application note, to a complex design that has real world applications. An example of such design is shown below.



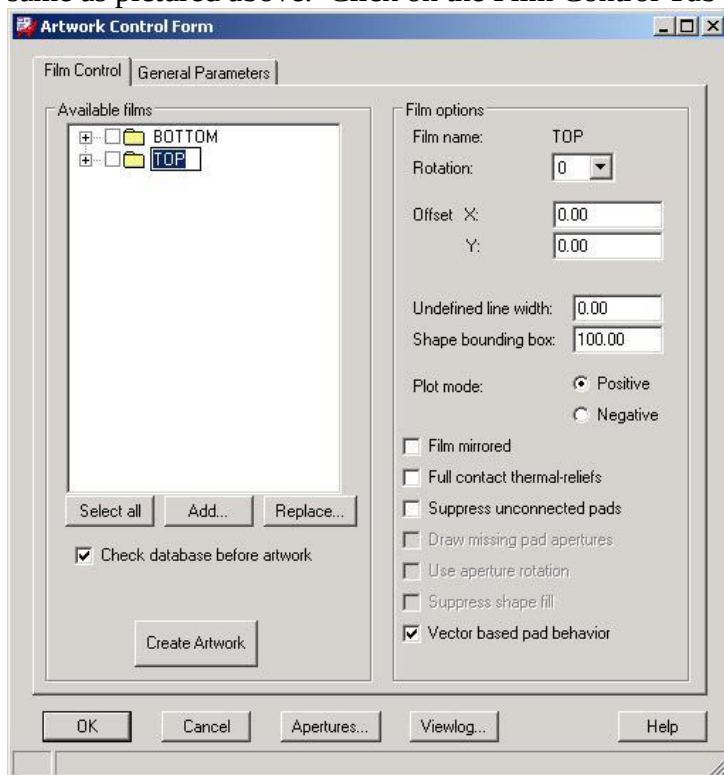
Fabrication Preparation

Once the PCB design is complete, the user will need to prepare the files in order to fabricate the board. There are two possible ways in which to extract the board files information that will be needed for PCB fabrication manufacturers. The first method is Gerber files and the second is ODB++. Set up and preparation are the same for both. For this application notes, we will be using the ODB++ method. Using the Gerber files will complete the same objective, but includes extra steps.

For the first step of generating the necessary files the user will need to setup for Artwork files. Go to the top of the page and select Manufacturer>Artwork and you should be in the General Parameters tab.



For the purpose of this application note, be sure that all general parameters are entered in the same as pictured above. Click on the Film Control Tab to the right of your current tab.



This board is very basic. When working on complex designs in the future, the list of available films will increase dramatically. When this is the case, you can edit the names of each layer so that you know what they are for. To edit the name, click on the name repeatedly until it allows you to edit. Once finished, go ahead and press enter then OK in the lower left hand corner. When naming the different files, it helps to keep in mind that these files belong together so the names should be similar.

Next, the user is ready to extract the information. The two file types we will be concerned with are the .brd and .dsn files.

The .brd is the main file. It has everything inside that the PCB manufacturer will need. The .dsn file is where the schematic is located. We can retrieve the logic symbols for all the parts here.

To extract the necessary information, the user will need the ODB++ add on program. One place to download this from is <http://www.mentor.com/products/pcb-system-design/downloads/odb-inside-trader>.

Once you have downloaded and installed the above program, it will take you through step by step on how to formulate the files you are interested in.

Conclusion

Once the user has reached the end of this Application Note, they should now be able to create a simple schematic using Cadence Design Entry CIS. The user will then be able to create a PCB design using this schematic and prepare the files for fabrication.

Recommendations

Please note, that there are many PCB design software packages available. In general, they all deliver the same results to the user, which are custom files needed by a PCB manufacturer to fabricate the users design. For the beginning user who has never used these types of software packages before, it is recommended to stay away from Allegro. This is the personal opinion of the Author of this application note. This was decided after many weeks of trying to learn the software, in order to complete one part of a design process. After weeks of lost time, the design schedule was behind because of this software. In order to not run in to the same problem, one could use a different software package with a less steep learning curve.

