

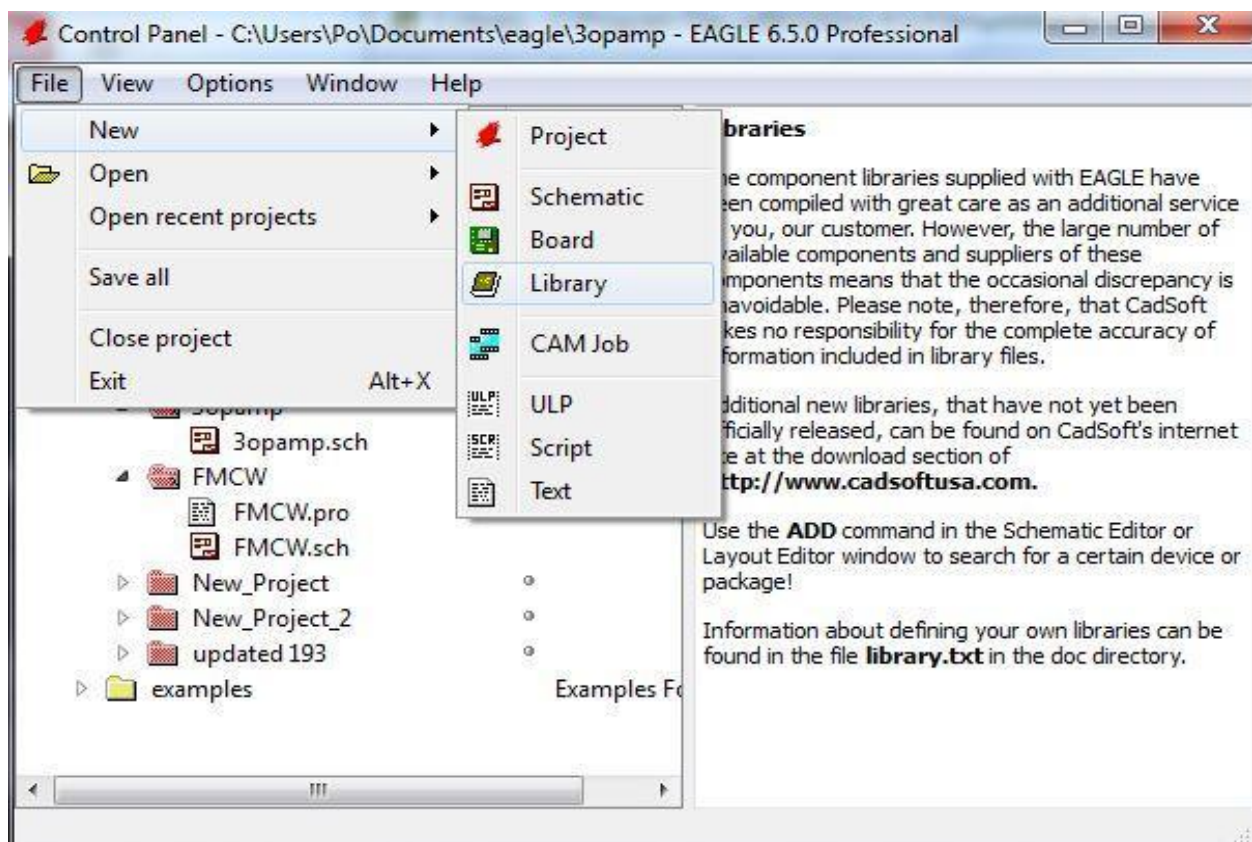
Application Note

Schematic

Schematic is the first step to start in doing PCB. All you need is to draw the design onto the schematic and make sure every pin of each component is connecting properly. It's crucial to double check the pins are all connected. We had experience with having one pin left out. It looks like it was connected on the schematic but when you look closer, it wasn't. We found out that problem after we finished the whole layout and placed the components. We ended up redoing the layout part since there will be an extra line need to connect and we have to adjust the adjust the place of components to make some space out.

Creating Component Library

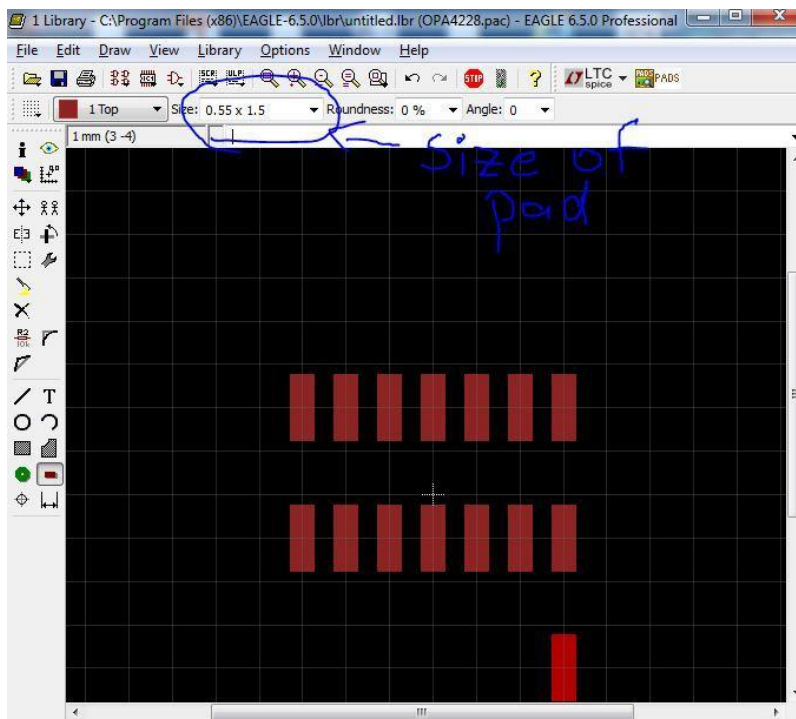
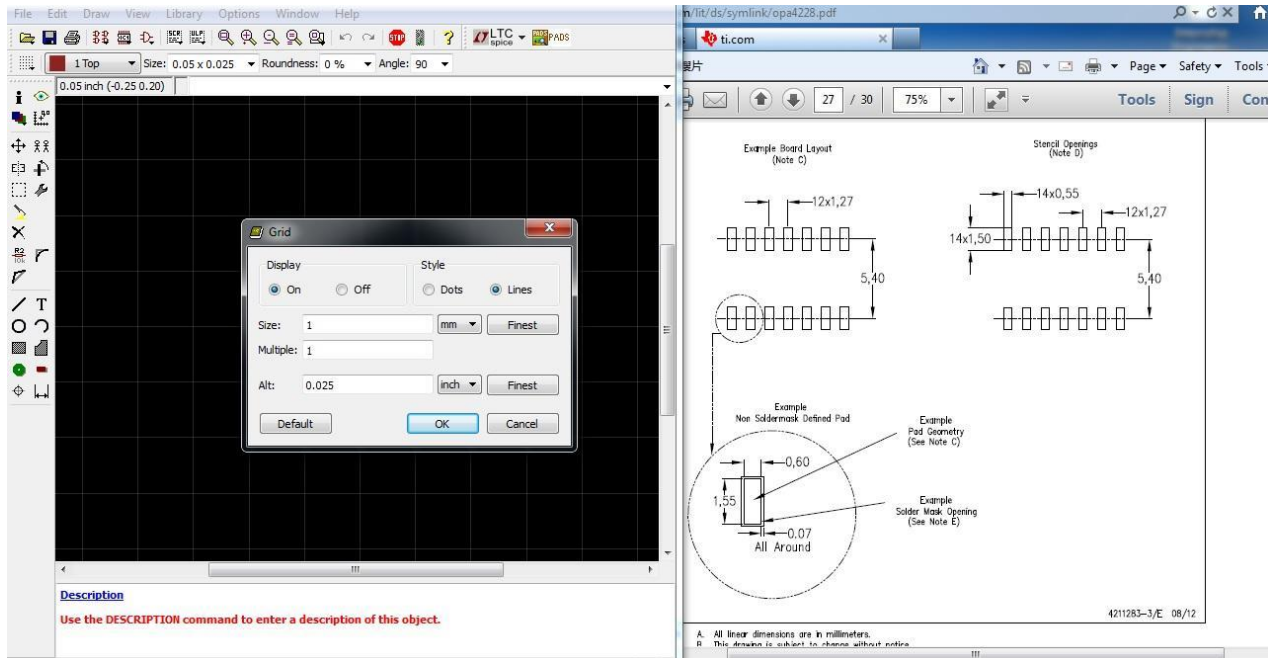
The software we used to create our PCB is Eagle-6.5.0. First, we need to open a blank page to start creating a new component library job. File > New > Library.



Second, we usually start with the editing the package first. Click on package and name it as the component's number or id to start editing. The package name should help you look up the component quicker in the library. I suggest use the name on the datasheet.

In my example, we will use OPA4228. It is an amplifier from TI. The datasheet can be found at here. (<http://www.ti.com/product/OPA4228/technicaldocuments>)

We will need to look for the PCB land pattern information on the datasheet. It will give you information to show you how to place the pad for the package. Then you will need



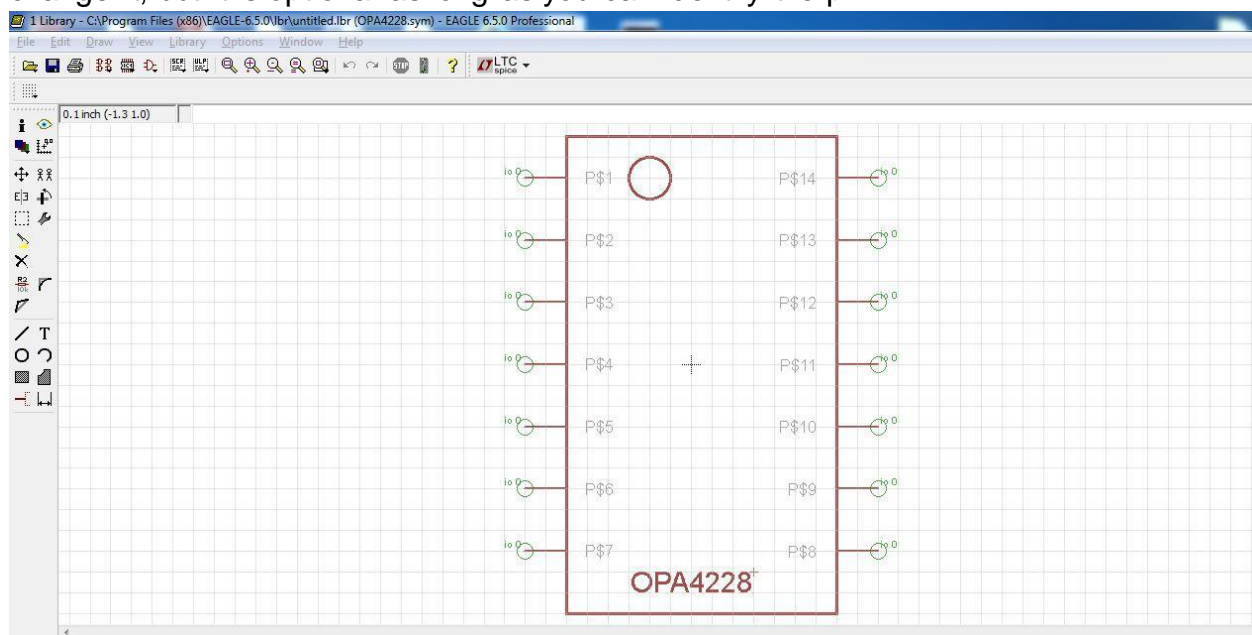
to check the unit of the information they provided. It is stated at the bottom of the page. All the dimension is measure at millimeters. Next, check the unit of the Eagle. If is default to inch, you should change it back to mm before you start.

Second, we can start working on place the pad for the package. Look up the size of the pad in the land pattern information page, and adjust the size of the pad in Eagle. For this case, the size is 0.55mm x 1.50mm and we need 14 pads

for this component. I will just place 14 pads on the blank space first and adjust the

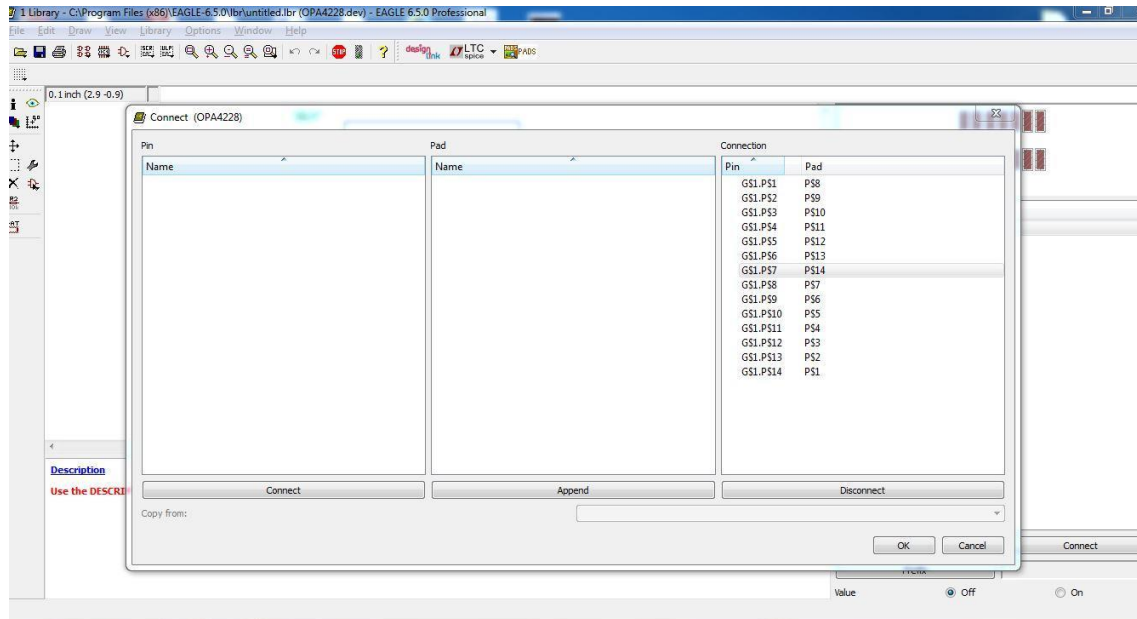
distance between the pads later. Next, we can click on the Info button locate on the left toolbar to do some calculation and adjust the distance between the pads. Every components in the package part should add a mark to indicate which one is pad one so the people who do the solder can know how should them place the components. After this, we can save and move on to the symbol part.

Click on the symbol and name it. I suggest you put the same name as you enter for the package. This will be shown on your schematic, so the dimension will not affect the layout. I will just draw a box and put 14 pins on and add a text on the bottom to indicate the chip name. Some people like to change the name of the pin to help them draw the connection on schematic. You can right click on the pin and choose properties to change it, but it is optional as long as you can identify the pin.



The last part will be device, the function of device is to connect the pins on the symbol and the pads in package so when you do layout, you will have an idea about where will this pad connect and it can help you to think of routing the lines.

Click device and name it the same as symbol and package. On the left hand side toolbar, there is an Add button which is next to the delete. Click add to choose the symbol and place it on the white space. On the right bottom side, click the New button to add the package. Click connect to connect the symbol and package. This is the most important part. When you do connect, you have to check to see which one is your pad one. You can check it by going back to the package part and check the properties of it. Since I put the first pad on the upper side so my pad 1 should connect to pin 14 and my pad 8 should connect to pin 1.



Then you can press OK to connect them and save it as the name of the components. After saving it, go to the schematic page. Click library > use > select all and ok. Then the components will be available in the Add now.

One more thing we need to keep in mind is the components need to be completely same as the datasheet described. Sometimes the datasheet might be hard to understand. If anything unclear or confusing, you should discuss with professor or TA to try to make it clear before move on to the next step. The reason is those minor unclear might ruin the whole design and waste the funding.

PCB Layout

When doing the layout for the RF components, we should pay extra attention on matching the transmission line. It is very important to have the transmission line match at 50 ohm. You should research on the stack-ups of the board, the material of the board, and the dielectric constant of the material that the fabrication house will use. Then calculate the width of your transmission to prevent loss on your design.

PCB Soldering

Good soldering is a must for a good design. Good soldering can ensure each component is well placed and make good contact with the pad on the board. Too much solder might lead to short circuit and the design cannot work as expected. Always have a DMM to test after soldering each components can save time and reduce the chance of having error in the design. Desoldering skill might be needed to fix errors and misplace components. The UCDart Forum has some useful resource on showing how to solder some small components. This class gave us a good opportunity to utilize our soldering skills.