ADS Layout Tutorial

Note: While it is possible to create a PCB layout in ADS, other programs are available that make the layout process much easier. Eagle, for instance, provides many tools and libraries that are either more complicated to use or simply unavailable in ADS. The current PCB layout for the mini MRI transmitter and receiver were designed in ADS and thus would likely have to be edited using ADS. It is for that reason that this tutorial was written.

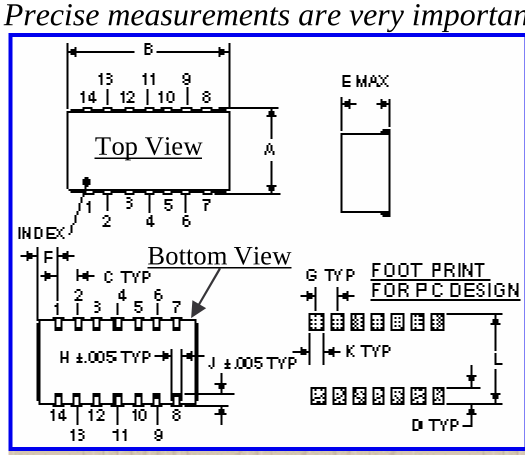
Creating a new layout project

The first step in creating an ADS PCB layout is to open ADS (Advanced Design System). At BYU this program is available on the fifth floor of the Clyde in the CAEDM Lab, or in the Biomedical Imaging Lab.

1) Select **File à New Project**, choose a name and a directory for the new project. Select one of the ADS Standards under the *Project Technology Files* option.
2) ADS will then bring up a schematic window. This window allows users to create and simulate various types of schematics using built in library components. A layout can be generated directly from this schematic window by selecting **Layoutà Generate/Update Layout**. This option is valuable for microstrip and other designs, but for component based PCB layouts it is easier to close this window and go straight to the layout window.
3) Select **File à New Design**, give the file a name, and under the *Create New Design in* option choose **New Layout Window**.

Creating the footprints

The next step in PCB layout design is to create the footprints for each component that will be used in the design. This implies that the designer knows which components will be used before the design is begun. Most companies will provide spec sheets for their products. These spec sheets often contain suggested PCB layout footprints. For example, figure 1 shows the suggested footprint and dimensions for the Minicircuits JTOS-50 Voltage Controlled Oscillator. If part dimensions are not available, the component dimensions will need to be measured. To allow for error, the footprints on the PCB are usually designed to be a bit larger than the actual measurements. *Note: Precise measurements are very important for creating a good layout.*



taken from:
www.minicircuits.com/cgi-bin/spec?cat=vco&model=JTOS-50&pix=bk276.gif&bv=4)

Case Style - BK377 (inch,mm) weight: 3 grams.								
A	B	C	D	E	F	G	H	J
.505 12.827	.789 20.041		.100 2.540	.250 6.350	.095 2.413	.100 2.540	.047 1.194	.065 1.651
K	L	M	N	P	Q	R	S	T
.065 1.651	.525 13.335							

Figure 1: Suggested footprint for the *Minicircuits JTOS-50* VCO

For this specific component (figure 1) it is important to notice that the top-left picture is a top view of the VCO while the bottom-left picture is a bottom view of the same VCO. Notice particularly that pin 1 is the lower-left pin in the top view but is the upper-left pin in the bottom view. When creating the layout it is important to use the top view for the pin definitions. If the bottom view is used, the pins will be mirrored and will not be connected to the correct pads.

1) Once the suggested footprints (or precise measurements) are obtained, the footprints can be created. The first step is to choose what layer(s) to use. Most basic PCB layouts will consist of a top plane (where the components will be soldered) and a bottom plane (which is usually a ground plane). A layer can be selected by using the pull-down menu on the upper right of the ADS layout screen as shown in figure 2.

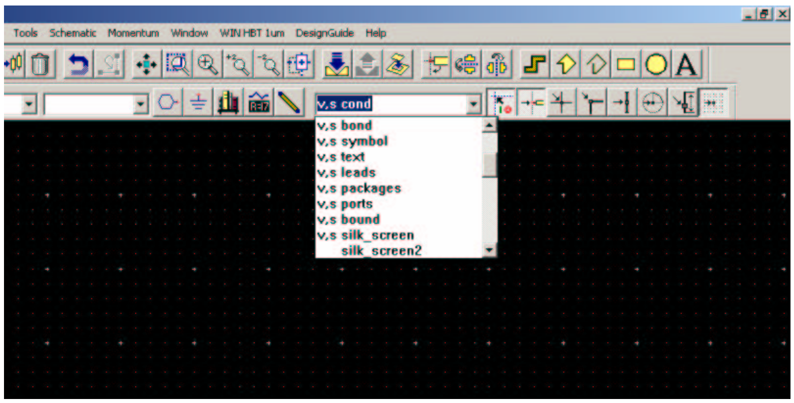


Figure 2: ADS Layout screen displaying layer pull-down menu.

The easiest layers to use are **cond** for the top plane, **hole** for any holes and vias, **cond2** for the bottom plane, and **silk_screen** for any text the designer wants to appear on the finished board. However, for the current mini-MRI layout the **pc1** layer was used for the top plane, the **pc2** layer was used for the bottom plane, and the **pcvia1** layer was used for the vias.

2) The footprint for the component in figure 1 is straightforward. As can be seen from the figure, the footprint consists of 14 pads (where the component's pins will be soldered) of equal dimension and spacing. For the VCO shown in figure 1 the pad dimensions are D ^{x} K or 100mils ^{x} 65mils (see figure 1). These pads can easily be created using the **Insert Rectangle** command. As the rectangle is placed, the dimensions will be shown as seen in figure 3. By default, ADS will use increments of 5 mils (if mils are being used) for placing and moving elements. The dimensions can be made finer by selecting **Options** à **Grid Spacing** and then selecting the desired dimension resolution. The first of the three numbers corresponds to the increment size (i.e. for <5-1-100> the increment size will be 5mils). For footprints that don't use rectangular pads, the circle or polygon tools provided by ADS can be used. If the components are leaded instead of surface mount, the pads will have to involve three layers. For example, if the layers suggested in step 1 were used for a leaded component, the **hole**, **cond**, and **cond2** layers would be required. The **hole** dimensions would need to be just slightly larger than the component's lead dimensions so that the lead can fit easily through

the hole. The **cond** and **cond2** dimensions should be at least 20% larger than the **hole** dimensions on all sides. This is important so that the metal surface area around the hole is sufficient to solder the lead to the PC board.

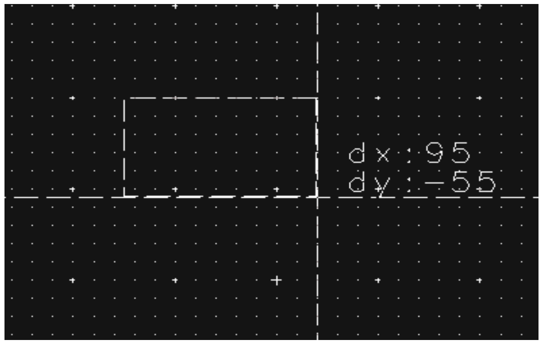


Figure 3: ADS Layout screen displaying rectangle placement

Note: It is important to know that the footprint pads are usually made larger then the actual component pins to ensure enough metal area for soldering. Most companies will use this principle when suggesting footprint dimensions, but for measured dimensions the designer will need to do this.

3) One advantage of ADS is its ability to copy, paste, rotate, and so forth. This makes the creation of footprints much easier. For the footprint shown in figure 1, the next step would be to copy the rectangle 14 times (once for each of the pads). When copying a part, ADS will prompt for the paste coordinates. Rather than inputting coordinates it is easier to select the **Default** option. The G dimension shown in figure 1 gives the pitch which is important for pad spacing. The finished footprint for the VCO is shown in figure 4. As can be seen in the figure, construction lines can be used to define spacing and other dimensions. A construction line can be placed by selecting **Insert** à **Construction Line**.

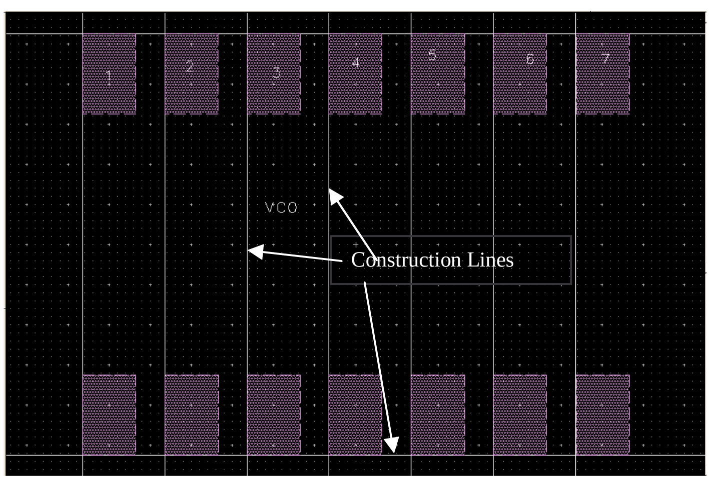


Figure 4: Finished VCO footprint with construction lines

4) Once the footprint is created, the design should be saved. This is done by selecting **File** à **Save Design As** and by entering the desired name. All designs will be saved in the *Networks* directory of the layout project.

Interconnecting the components

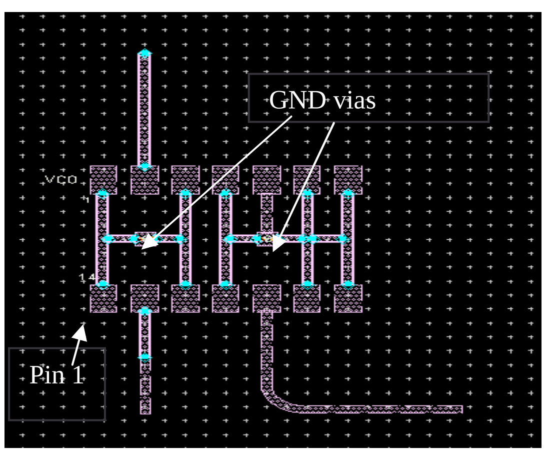
When all of the footprints are generated, the next step is to connect the components together.

1) The first step in component interconnection is to connect each component to its necessary circuitry. For example, many components require capacitors in shunt with their power supply inputs. In addition, all components will require connections to ground. A new layout design should be opened and the component footprint should be inserted by selecting **Insert** à **Component** à **Component Library**, choosing the current project under the *Libraries* window, and choosing the correct footprint (the footprint will have the same name you gave it when you saved it). *Any construction lines used for the footprint creation will not appear.* The **trace** tool should be used to create connections between components. The traces should overlap onto the components a bit to ensure that there are no gaps. The minimum trace width will be dependent on the frequency at which the circuit will be used. A minimum trace width of 20 – 25 mils is fairly safe. The method for connecting the component to ground will depend on the PC board fabrication process. If the bottom plane will be a ground plane (strongly advised), the component can be connected to ground by placing vias (holes) in the correct places. When creating ground connections, only a top (i.e. **cond**) and via (i.e. **hole**) layer will be needed. If the bottom layer (i.e. **cond2)** is used, the vias will be separated from the ground plane which will, in effect, isolate the ground vias from the ground plane. *This is a bad thing.*
   Most spec sheets will suggest biasing and other circuitry for the component. For the VCO the following information was provided for connecting the pins (see figure 5a). Based on these given pin connections, the various ground pads were connected together and traces were sent out from the pins that can be connected to other components (see figure 5b.)

Pin Connections

Port	RF OUT	V-CC	V-TUNE	GND EXT
jc	13	2	5	1,3,4,6,7,8,9,10,11,12,14

a)



b)

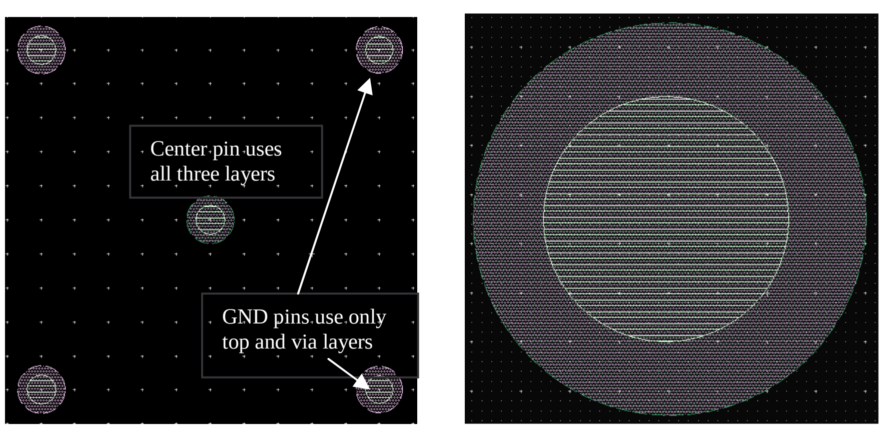
Figure 5: VCO circuitry layout (shown in 5b) based on the required pin connections (shown in 5a)

Note: For a fabrication process that can create drill plated holes, it would be better to place a ground via on every pad that needs to be connected to ground. This would eliminate the need to interconnect all of the ground pads.

Once all of the appropriate ground, capacitor, and other connections are made any desired text should be added using the silk screen layer. A minimum font size of 30 should generally be used for this text. Be sure not to silk screen on top of the footprint layer. Finally, the design should be saved (under a different name than the footprint).

2) At this point the footprint for each component should be created, connections to ground should be included, and the footprints should be connected to all circuitry suggested by the manufacturer (i.e. bias resistors, shunt capacitors, input and output capacitors, etc.). The next step is to connect the various components together. A new layout design should be opened and each component required in the design should be inserted by again selecting **Insert** à **Component** à **Component Library**. The components should then be connected according to the circuit design. This is a straightforward process and is done using the **trace** tool. *Note: Allowing two or more traces to run for long distances in parallel is poor design technique. At certain frequencies parallel traces will have coupling effects that can effect the operation of the circuit. It is also good to avoid sharp corners when possible. To do this the* ***curve*** *option under the* Corner Type *selection in the trace window can be chosen. A Curve Radius of at least 20 is suggested.*

3) The final step in component interconnection is to connect the circuit to power, input/output, and ground. For the frequencies at which the mini-MRI circuit will be operating (0 – 42.5MHz) BNC connectors are suggested for input and output. The standard CON5 design is easy to use (see figure 6a). For power and ground connections banana connectors (see figure 6b) are effective. If the bottom plane is a ground plane it will only be necessary to have one banana connection to that plane. *Again make certain to use only the top and via layers when creating this footprint.* The power source(s) will need to be connected to all appropriate components. The final MRI transmitter/receiver layout is shown in figure 7.



a) b)

Figure 6: CON5 BNC and Banana connector footprints

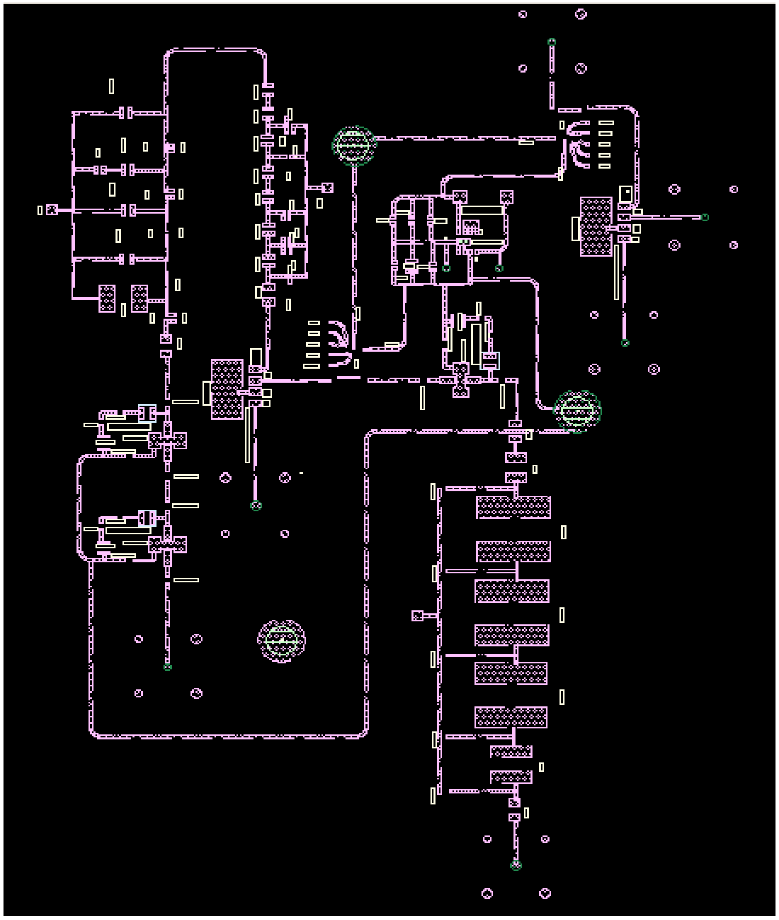


Figure 7: Final layout for mini-MRI transmitter & receiver

Note: When a footprint or other design is inserted within a higher level design (i.e. an Amp circuit is inserted into a receiver design), any modifications made to the lower level design will automatically take effect in the higher level design. For example, if the designer decided to change the Amp footprint, these changes would appear in any design that uses that Amp footprint.

Exporting the Layout for milling

The final step in PCB layout involves exporting the layout so that the PCB can actually be fabricated.

1) First select **Momentum** à **Substrate** à **Create/Modify** then hit the **Metallization Layers** tab. Go to “------Strip cond” under the *Substrate Layers* menu, choose **cond2** (or whatever layers you used for the top and bottom), and click the **Strip** button. Now go to “Alumina” under the same menu, choose **hole** (or whatever layer you used for the vias), and click the **Via** button. The menu should now look similar to figure 8.
2) To export a layout for milling select **File** à **Export** and choose **Gerber** from the *File Type* pull down menu.

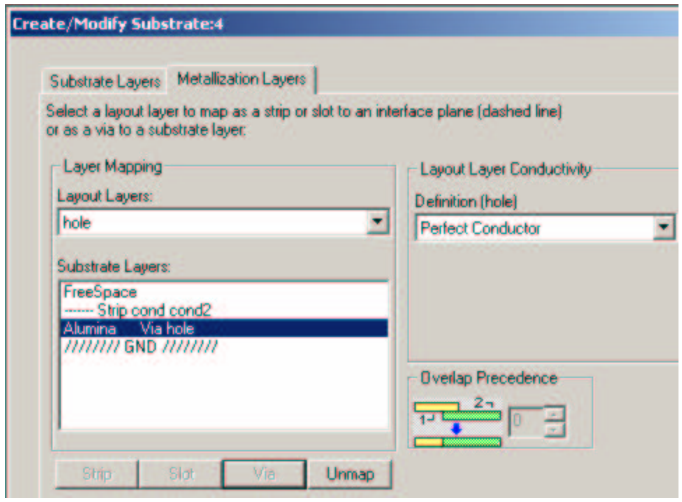


Figure 8: Modified substrate menu

3) Choose **More Options** then **Define Layers** and note the ID numbers for the layers used in the layout (i.e. for **cond** the ID number is 1 and for **silk_screen** the ID number is 14). Then close the **Define Layers** window and enter the ID numbers in the *Layer Number(s)* field (i.e. for **cond, cond2, hole, and silk_screen** the IDs would be 1 2 5 14). Make certain the **include** option is selected under *Layers* and hit **OK**.
4) Choose **Browse** then select a path and file name for the exported Gerber file. *It is easiest to use the project directory for the path.* Hit **OK**. ADS will then open two windows. The important one gives various options. The other can be closed.
5) Choose **Translation Settings** then select the **Fill** option under *Outline/Fill* and **RS274X** under *Gerber Output Format*. Hit **OK**.
6) If you want your holes to be drilled for you during the milling process, steps 6 and 7 are very important. Otherwise you can skip to step 8.
Choose **Edit Apertures**. *Note: It would be a good idea to have a pencil and paper at this point.* Click on the **Flash Circles/Rectangles/Polygons** button. Write down the number, dimensions, and block name for any vias (or holes for leaded components) desired for milling (see figure 9). Hit **OK**.

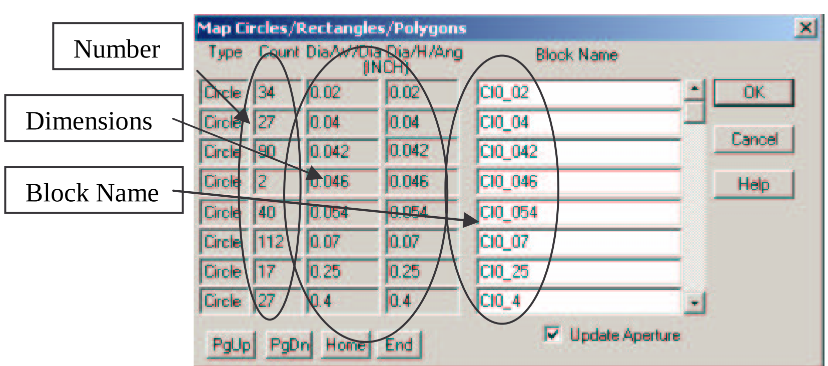


Figure 9: Example of Flash Circles/Rectangles/Polygons menu

Note: At this point it might be a good idea to look at the layout and make certain that the number of holes given in the Flash Circles/Rectangles/Polygons menu *matches the number of holes you wanted in your layout. Pay particular attention to the dimensions and make sure they are correct.*

7) Back in the **Edit Aperture** menu find each block name for which a hole is desired and note the corresponding **D-code** (see figure 10). Hit **Save**.

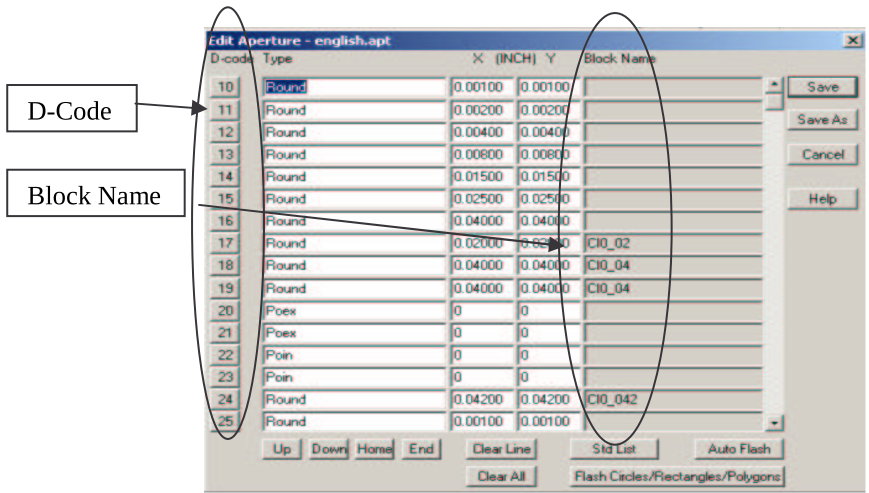


Figure 10: Example of Edit Aperture menu

8) The next step is to translate the layout mask into Gerber files. To do so choose **Translate** and select the layers for which Gerber files are desired (probably all of them).
9) Choose **View Gerber** and make sure the layout looks right. If not try repeating the export process again and do any necessary troubleshooting. *Note: If you chose to export the files to a different directory than your project directory you may get an error at this point. If this is the case, go back and select your project directory for exporting.*
10) If you want the holes to be drilled for you choose **Layer** and deselect the **Show** option for any layers that are not via layers. Choose **Edit Aperture** and locate the D-codes for the desired holes. Next give a tool number (i.e. 1 for the first tool) and drill diameter (the diameter will be based on the hole dimensions recorded in step 6). Once this is done for every hole (see figure 11), hit **OK.** Hit **OK** again.

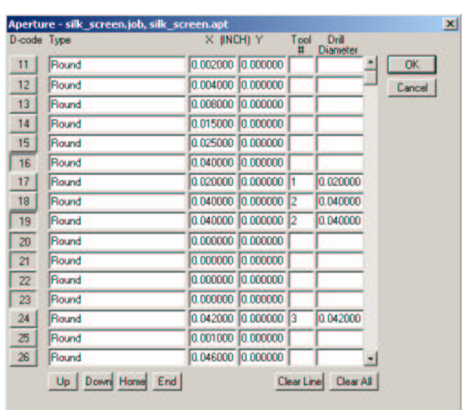


Figure 11: Tool definition example

11) Select **Tools** à **Drill** à **Excellon (Leading Zero Suppress)** then go to **Report** and make sure all the necessary drill tools are shown. Close the tool report and hit **OK**. Close the **View Gerber** window. Choose **Exit**.
12) The necessary milling files should now be in the project directory (if you chose the project directory as the path). The needed top, bottom, and silk screen files have the .gbr extension. The necessary via file has a .drl extension. If the board is to be milled here at BYU, copy the top, bottom, and via files to a disk (silk screen can't be done here). There are milling machines on the fifth floor or room 416 of the Clyde building. *For milling instructions see the milling tutorial*.

    If the layout is to be sent to a professional company for fabrication (highly recommended), contact the company to see what files will be needed. The above instructions will produce the files needed by Advanced Circuits (www.4pcb.com). Advanced Circuits will produce an 85 square inch, 2 layer, drill plated PCB with silk screen text for $33 plus shipping and handling. They also use professional software to verify that the design meets all the necessary design rules. The boards they produce are high quality. If the layout is sent to Advanced Circuits the top, bottom, via, and silk screen files will need to be compressed into a single zip file. This is done easily by selecting these files, right clicking, and choosing the Winzip option. The zip file can then be submitted at www.4pcb.com.
    Note: Make certain to specify that the bottom plane is a ground plane or they will remove all the ground plane metal.

    This tutorial was written by Ryan Robison (edited by Brandon Woodbury). If you have any questions or comments, Ryan can be emailed at rkrobi@hotmail.com. Good luck.