PCB LAYOUT GUIDELINES

<u>**LAYING OUT YOUR BOARD:**</u>

1. The specifications below are based upon the capabilities of the LPKF machine used here on campus to fabricate your Printed Circuit Boards (PCB's). Because of this, there will be some design limitations you must take into consideration before you begin your PCB design. That being said, you will be glad to know that you will find a lot more leeway once you're out there working in industry; so when you get out there, design away!

2. **Limitation #1** – Before you begin laying out your PCB, you need to understand that the PCB's fabricated on the LPKF machine here on campus **ARE NOT** plated through from the TOP to BOTTOM or vice-versa. Typically, the holes you drill are copper plated through from one side of the PCB to the other – **we skip this process –** This **IS NOT** the case in the outside world!… You can still make these transitions but **YOU MUST** <u>physically complete</u> any and all of these signal, power, and ground connections after you receive the PCB in your hands. There are ways to accomplish this so you may want to take this into account as you begin to layout your PCB.

    a. Utilize the lead of a through-hole component that has the same signal in common on both top and bottom layer of the PCB. To complete the connection you simply top and bottom solder the component lead to the surrounding pad.

    b. Create a via where you need to make a transition from the one layer to the other in the absence of a through-hole component lead. You will have to install a small wire jumper at this location at time of assembly. Vias are simply a pad and hole that are placed on the PCB and specifically used to complete an electrical path for either a signal, power or ground from one side of the PCB to the other. Components are not placed in these vias.

3. **Limitation #2** – Since we skip the copper plating process, all of the pads associated with drilled holes on your PCB's **DO NOT** have the same integrity or mechanical strength regular PCB's fabricated by a board house who does. Because of this, mechanical stress transmitted to the solder joint connection becomes a concern and issue. An inferior mechanical connection is created anywhere you install a toggle switch, a high profile component, a header strip, connector, or hookup wire. There is a chance that either the solder joint connection will crack or more than likely, the entire pad will break away from the PCB, isolating from its intended signal path. As result, either way, you will end up either an intermittent connection or open circuit at the locations where these parts are soldered to the board. This is preventable to a certain degree if you simply beef-up the pads and traces attaching to these areas. Again, out in industry this is usually not the case unless the component is poorly designed in for mechanical integrity.

4. **Limitation #3 –** No trace shall be LESS THAN 0.010 inches (0,25mm) – The LPKF machine has difficulty maintaining this tolerance so wider traces are better. Real estate permitting (room permitting), try to use trace widths of 0.02 – 0.040 inches (0,6mm – 1,0mm) for signals and 0.040 inch (1,0mm) or higher for power and ground connections.

   If in the event you must place a narrow trace smaller than 0.010 inches (0,25mm) because the part you are designing in pads are such they must, reduce the traces down to the same width as the pad you are connecting to but as you exit that area, begin to increase the width but then increase its trace width back to a wider level as you route away from the pad and into wider open areas.

5. **Limitation #4 –** Spacing between traces, pads, and vias shall be NO LESS THAN .010 inches (0,25mm) for the same reason as Limitation #3. Set global preferences 0.010 inches (0,25mm).

<u>GENERAL DO's AND DON'T's</u>

1. All PCB designs shall be double sided.

2. Try to place all of your circuitry connections for through-hole components on the BOTTOM side of the PCB. In other words, the through-holes components will be on the TOP side but the trace connections (circuitry) to these parts will be on the BOTTOM side. This practice will help you make it easier for you to solder your circuit assembly.

   If it is not possible to route a given signal path due to congestion or simply a blocked path, then use vias to make transitions between the Top and Bottom of the PCB. But take caution in doing so as not to place these vias in areas of your PCB where a component will cover it up once it is installed, making it either difficult to install a jumper or that the presence of a via might affect the installation of parts being placed over it. (In the outside world, routing circuitry and the placement of vias will be less restrictive but the information mentioned above will still come into play.)

3. When placing Integrated Circuits (IC's), Connectors, or Header/Pin Strips, try to select component footprints from the library whose pads indicate Pin-1 with a single square pad amongst the round ones if available. This will make it easier to identify Pin-1 while you are working on your board.

4. Most of the through-hole resistors you will be using in your design are likely to have a ¼ Watt power rating. When laying out your PCB, the lead spacing for these parts should be either 0.4 inches (10 mm) or 0.5 inches (12mm). You will want to select components from the component library with either of these footprints designed into them, unless you are using a larger component. In which case, pay close attention to the Lead Spacing perimeter properties as you select components.

5. All PCB's shall contain a ground plane (i.e. Polygon copper pour or flood) on both the Top and Bottom layers of your design unless your board will be used for High Voltage applications. i.e. 110/220 Volts. When doing so, make sure the Polygon in placed on the inside of the board outline/dimension.

6. The Ground plane MUST be connected to a least one ground pad of your circuit must be connected to ground plane **ON BOTH** the Top and Bottom layers. To ensure this happens, you will have to allocate or assign the polygon copper pour to the signal name “Ground” or equivalent.

7. When adding Surface Mount Technology (SMT) resistors, capacitators, or diodes into your board layout – for ease of assembly in the lab, try to select to use 1206 (imperial) or 3216 (metric). Out in industry, you will likely use much smaller parts, tiny.

GERBER DATA FILES:

1) If you have not noticed by now, you will see that your PCB is built in layers. Even though a finished PCB might look as if it is only two layers, in reality from a PCB CAD software perspective, there can be anywhere from 5 to 30+ layers of CAD layer information! Every layer has a different function to serve which, when all layered together, create a circuit board. On campus, we only use a basic portion of those CAD layers available. We typically only use:

   a) Dimension Layer (aka Board Outline)
   b) Top Signal Layer
   c) Pads Layer
   d) Vias Layer
   e) Holes Layer
   f) Drills Layer
   g) Top Silkscreen Layer
   h) Bottom Signal Layer
   i) Bottom Silkscreen Layer

   Please note that again, that out in industry you will use many more layers that those listed above, especially if you are designing a multi-layer board.

2) In order to fabricate your newly completed PCB design, you will need to provide Gerber Data Files to anybody you expect to fabricate your PCB. These files must be generated by you using various combinations of the many different CAD layers you created while designing your PCB. You will now have to pick and choose which of these CAD layers to use in order to process and generate TOP, BOTTOM, DIMENSION, and DRILL Gerber Data files. This Gerber data is necessary in order for other machines to read and interpret your design, such as with the LPKF machine. Please note that once you generate these files, they cannot be changed or altered. If you do need to make changes to your layout, you can always go back to your CAD design layout and make them there; but you MUST generate a new set of Gerber files in order for any of your changes to take effect!

3) As mentioned earlier, depending on which CAD package you are using, you will need to select the proper combination of CAD layers above to create the desired Gerber Layer. EAGLE uses a CAM Processor to create Gerber data files; the table below indicates which CAD layers are required to create the corresponding Gerber layers:

Gerber Top Layer = Top Signal Layer + Pad Layer + Via Layer (+ Dimension).*
Gerber Bottom Layer = Bottom Signal Layer + Pad Layer + Hole Layer + Via Layer.*
Gerber Dimension Layer = Dimension Layer (if not included in Top Layer)*
Echelon Drill Layer = Holes Layer + Drills Layer*
*****Use Gerber RS274X for Gerber output format AND Echelon_26 for Echelon output format.
The four Gerber Layers mentioned above are Mandatory to fabricate your PCB!

4) While Gerber data files themselves are pretty the much the same across every PCB CAD layout package, the method in which they are generated varies from one PCB CAD package to the other. I suggest you "YouTube" this process for the CAD Layout software you are using to become familiarized with the process. There is always the HELP section usually found within the ribbon along the top of the design display.

5) Please be advised that ONLY ONE FIRST ARTICLE PCB WILL BE FABRICATED UNTIL YOUR DESIGN IS FULLY PROVEN. NO EXCEPTIONS!

6) Given this strong advisement, you should go to www.pentalogix.com and download their free Gerber Viewer named **Viewmate**. With this viewer, you can import the Gerber date files you generated into Viewmate and view your PCB as the LPFK prototype machine or any other PCB manufacturer will view and fabricate it – The PCB layout you see on your monitor using PCB CAD software is NOT the same layout as the one you will see on Viewmate, even though they may look the same. The image on Viewmate is a WYSIWYG whereas the CAD software view is not! Using Viewmate will allow you to see and correct your mistakes prior to submitting them for fabrication.

SENDING GERBER DATA FILES:

1. All Gerber files shall be emailed to Mark Bruno – No thumb drive file exchanges will be accepted. This method provides him with your contact info should he need to contact you. His email address is mbruno@sdsu.edu.

2. All filename shall be in the following meaningful format indicated below.
   [Your Team Name or Name]_[A meaningful Description of the PCB]_ [Revision]
   For example: BSmith_CPU_v1.00; TeamA_CPU_v1.00

3. **PCB Cut-off times:** For your PCB to be considered for fabrication if the aforementioned criterion has been meet, is 10:00 for same-day service, work-load permitting. Otherwise please plan for 24 – 36 Hour turn around.