Milestones and Input Checklist for
Printed Circuit Board (PCB) Layout

This document is intended to facilitate PCB layout at Redgarden Engineering on time and within budget. The document includes the *milestones* in a typical PCB layout along with a *PCB layout input checklist*.

The checklist is limited in scope so that it can apply to many different companies. A more complete list can be created to conform to a company's internal process and requirements. Supplying some of the information in the checklist ahead of time can dramatically decrease the overall time to layout the PCB. In particular, supplying footprint information in advance allows finding or creating the footprints before the clock starts for the actual layout.

PCB Layout Process Milestones

A. **Advance Information & Scheduling**: The electronics design engineer lets the PCB layout person know that the layout is coming. If the board is on a very tight schedule, he may supply some of the information from the checklist (such as the footprints) in advance along with a purchase order to allow working with that information.

B. **Transfer of Full Package**: The electronics design engineer supplies all or most of the information listed below to the layout person and supplies a purchase order so that the layout can begin.

C. **Review of Parts Placement**: The layout person supplies the electronics design engineer with an initial parts placement for the top and bottom. The layout person waits for the electronics design engineer to review this placement before proceeding. If changes are necessary in the initial parts placement, the electronics design engineer approves these before the layout person proceeds further.

D. **Engineering Change Orders**: Any changes that the electronics design engineer makes in the schematics after the layout has commenced are imported into the layout. The cost and schedule may increase depending on the scope of these changes.

E. **Review of Routing**: The layout person supplies the electronics design engineer with a PCB layout including routed traces. The layout person waits for the electronics design engineer to review this routing before proceeding. If changes are necessary in the routing, the electronics design engineer approves these before the layout person proceeds further.

F. **Final Outputs**: The layout person re-sequences the reference designators and generates the final outputs for the PCB including Gerber files and assembly and

solder mask views. The electronics design engineer reviews this information and requests any final changes in the layout.

G. **Invoice**: The customer receives an invoice and promptly pays for the layout work.

PCB Input Checklist

1. ☐ **Board Outline**: This shows the dimensions and shape of the PCB. The board outline can be in the form of a DXF file for direct importation into the layout software. It can also be a dimensioned drawing that the layout person uses to create the outline in the PCB layout software. Typically the board outline includes dimensions for any components that require an exact placement such as connectors going directly to another circuit board. The board outline also typically includes mounting holes and any additional desired cut-outs.

2. ☐ **Board Stack-up**: This is a sketch or a table that shows a cross-section of the board layers. If desired, Redgarden can create or recommend a stack-up for you. Considerations include PCB fabrication price and manufacturability, electro-magnetic emissions and susceptibility, ability to route, noise, amperage requirements, controlled impedances, cross-sectional symmetry, and high-voltage creepage and clearance. An example is shown below for a four-layer board with internal power and ground planes:

Layer #	Description	Thickness (mils)
	Top, ½-oz plated copper	0.7
1	Top, ½-oz copper foil	0.7
	FR4 #1	24.0
2	Power Plane	0.7
	Pre-Preg	10.4
3	Ground Plane	0.7
	FR4 #2	24.0
4	Bottom, ½-oz copper foil	0.7
	Bottom, ½-oz plated copper	0.7
	Total Thickness:	**62.6**

3. ☐ **Rough Parts Placement**: This is a sketch and/or description that shows where the most important parts will be placed on both the top and bottom of the board such as a pre-amplifier, microprocessor, programmable logic part (e.g., an FPGA), and connectors. The rough parts placement may also include a description of how to split up the use of the power and ground planes for different sections of the board.

4. ☐ **Bypass Capacitor Connection**: Typically, bypass capacitors are placed on the bottom and connect directly to power and ground planes if they exist. Please state any preferences concerning the bypass capacitors.

5. ☐ **Orientation of Parts**: State if you have a preference for how parts such as diodes and integrated circuits should be oriented. Typically, pin one of most parts is closest to a particular corner of the board unless more important considerations intervene. Redgarden can make recommendations for individual part orientation during the layout.

6. ☐ **Reference Designator Re-Sequencing**: State if you want reference designator re-sequencing and if you have a preference as to the corner where the re-sequencing would start. Typically, the reference designators for the parts (e.g., R1 and R2) will be re-sequenced so that the reference designators for each prefix (e.g., R, C, U) go from left to right in approximate rows with the first row being at the top.

7. ☐ **Special Labels**: Provide a list of the special labels you would like and your preference for their location. For example, certain test points or LEDs are often labeled with their function rather than their reference designator (e.g., GND, 5V, 10MHz). Circuit boards may include a label with the country of origin, the company's internal part number for the PCB, and a white label where the serial number and/or date for a printed circuit assembly (PCA or stuffed PCB) can be written in.

8. ☐ **Part Groupings**: Provide groupings of parts that may not be obvious. For example, the resistors and capacitors associated with an op amp circuit may form a group. This is an area where Redgarden can speed up the process since the layout is being done by an analog engineer, not by a draftsperson. Therefore, it is not necessary to specify all of the part groupings. However, this can be useful if the schematic does not make it obvious how parts should be grouped.

9. ☐ **Special Considerations**: Provide a list of any special considerations such as for controlled trace impedances, traces carrying significant amounts of current, high voltage creepage and clearance specifications, special return current traces, separation distances for high-speed traces, and differential pairs. Again, these are areas where Redgarden can speed up the process. However, you must supply certain critical information that may not be obvious from looking at the schematic. These considerations may also include any sample layouts from manufacturers' application notes for critical portions of the circuit such as DC-DC switching converter circuits.

10. ☐ **Footprint (Decal) Information**: These are the desired PCB part outlines that will be printed onto the top and/or bottom of the board. Generally, this is the most frequent cause of critical mistakes in PCB layout. Although Redgarden can figure out or infer what the footprints should be, it is usually preferable if the customer can supply and double-check the footprint information. Most schematic packages (such as Orcad Capture) allow the footprints assigned in the schematic part symbols to be listed (and checked) in a spreadsheet. In addition, it is much faster if actual drawings of the desired footprints are provided—or at least the actual part outline

drawings. A useful way to supply (and check) this information is to open the PDF datasheet file for each part. You can usually use the graphical selection tool in Adobe Reader to copy the part outline and any suggested actual part footprint drawing. These can then be pasted into a Microsoft Word document showing (and labeling) the information for all of the types of parts in the schematic. Going through this process and double-checking it against the bill of materials (BOM) often highlights incorrectly-specified part packaging. If desired, Redgarden can put together this information for you as part of its engineering (but not layout) services.

11. ☐ **Schematics**: This includes the electronic design schematics (usually in PDF format) and the netlist file generated from the schematic. It is helpful if you supply the BOM including the assigned part footprints.

12. ☐ **Desired PCB Outputs**: These generally include the following. State whether other outputs are required.

    a. Full set of Gerber files to allow fabricating the PCB
    b. Assembly views of the top and bottom to allow the electronics design engineer to create of annotated assembly drawings
    c. Information for the creation of solder masks
    d. A back-annotation listing to allow the original schematics to be updated.