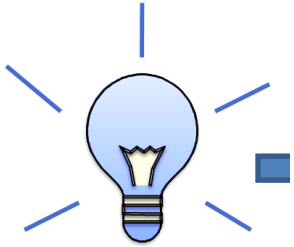
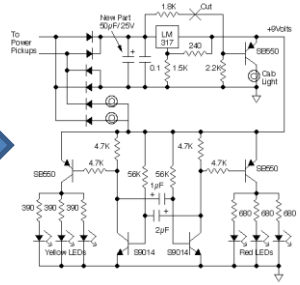
From Idea to PCB



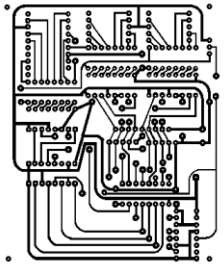
Altium



Schematic

- xxx.SchDoc

Altium



Layout

- xxx.PcbDoc

Altium

Gerber Files

- xxx.gtl
- xxx.gbl
- xxx.gm1
- xxx.txt

LPKF Laser & Electronics

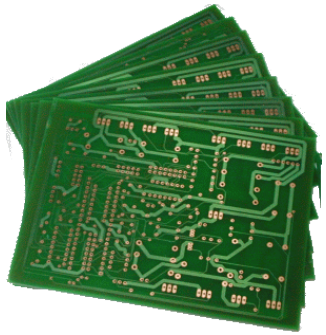
CircuitCAM

Output Files

- xxx.cam
- xxx.lmd

LPKF Laser & Electronics

BoardMaster



Recommended Strategy

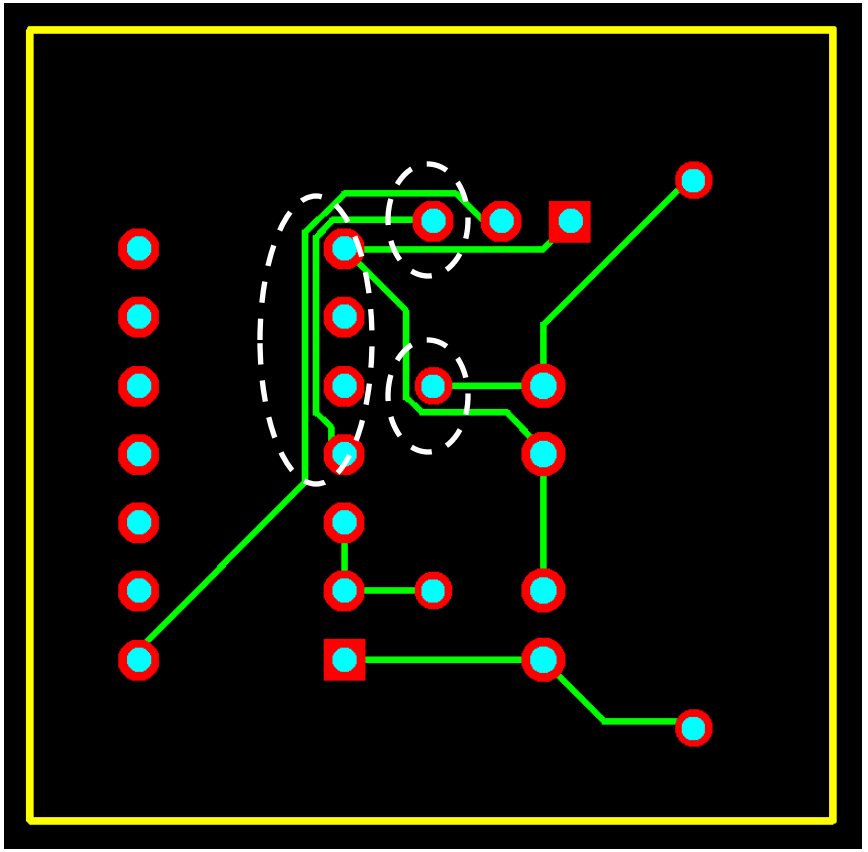
- The PCB prototyping facility can produce a board in a couple of hours but it has limitations.
  - Resolution limitations resulting in ...
    - larger than normal traces
    - smaller than normal solder pads
  - No green insulation layer
- Your main goal is a board that works. Leave ample space between components to avoid unwanted solder bridges. If you need a board that is tightly spaced, send it out for production.

Rapid Prototype Design Rules

- Plated holes are time consuming to make and are not 100% reliable. So ...
  - Make single-sided board if possible.
  - Delete any traces on the top layer that do not cross traces on the bottom layer, and redraw them on the bottom layer.
- Pads are small and difficult to solder. So ...
  - Add teardrops to pads (Tools / Teardrops ...)
  - Drag traces on bottom layer away from pads.
  - Remember to create rubout areas when you lay out your board in CircuitCam (but just around pads).

Bottom Layer

- Maximize clearance between traces and solder pads
  - The circled traces are too close to pads.
  - It is likely that you will create a short between the pad and the trace when you solder this circuit together.



How to Create a Gnd Plane

(automatically avoids most clearance problems)

- Increase the distance between nets. This needs to be at least 1mm or it will be very hard to solder your board.
  - Design / Rules (Electrical / Clearance)
  - Many of your pads will turn green since they violate this rule (don't worry about it)
- Layout your board normally
  - Auto Route / All
- Erase your existing Gnd Net
  - Tools / Un-Route / Net (Select Gnd net)
- It may be possible to re-route traces from the Top to the Bottom Layer.
- Add Teardrops
  - Tools / Teardrops ...
- Create a Gnd plane
  - Place / Polygon Pour
  - Connect to Net (Gnd)
  - Define area you want (entire Bottom Layer)
- Add any missing Gnd connections on the Top Layer
- When you use CircuitCAM, do not define any rubout areas. You can't rubout any of the GND plane anyway and they are not needed on the Top Layer.

Example Result