gEDA assignment

CREATE A SCHEMATIC FILE, PCB FILE, NETLIST AND A OUTPUT GERBER FILE FOR THE ATTACHED SCHEMATIC, FOLLOWING THE GUILDLINES BELOW.

1. Schematic should look as close as possible to the attached image, appropriate fields of each component should be filled in *gschem* for the same.
2. Nets in *gschem* should be labelled and the same should reflect in the netlist file.
3. The footprints are as follows:
   1. All resistors are SMD [0603].
   2. Caps are appropriate sized through holes.
   3. Transistors are through hole.
   4. Leds are SMD [0806]
   5. Attiny is a DIP8
   6. PINHD is a 6 pin through hole header.
   7. Crystal is SMD, any size
   8. Switch is Push button through hole
   9. The battery can be taken as a 2 pin header.
4. All SMD components should lie on the component side of the PCB, i.e. we have a two sided PCB

Enjoy,

Akshat