PCB Design Assignment

Turn in to TA in 246 NEB during official lab hours.

Motivation

The purpose of this assignment is to introduce the student to printed circuit board design using an EDA (Electronic Design Automation) tool such as National Instrument's ***MultiSim*** & ***Ultiboard***.

Assignment

Each student was assigned a unique schematic to be drafted using MultiSim's schematic capture feature. You should create an error-free net list, import it into the PCB feature of Ultiboard, then design a proper laid-out and routing. Designs should be no bigger than 6"x6". Your should also create plots and miscellaneous design files (Gerber Files) that could be used to have the circuit boards milled locally or sent out to be fabricated professionally.

Deliverable

The student is expected to turn in the following items for grading:

1. Cover page.
2. Sample circuit original diagram that you were given in class. (1 page)
3. MultiSim schematic drawing. (1 page)
4. PCB layout drawing in 1:1 format. (1 page)

Deadline

The project deliverables are due by lab closing of the due date (as posted on the website). The TAs will gladly accept your assignment earlier. No credit for late work unless prior arrangement has been made with the instructor.

Check Class website

For current information and replacement handouts.

PCB Design Glossory

Auto-Routing

The process of letting the software place all the copper traces on the PCB design. Design rules must be properly configured first.

CAD

Computer aided/assisted design, i.e. PSPICE, Protel, Autocad etc.

DIP

Dual-in-line package. Double row footprint, used for op-amps, TTL chips etc.

DRC

Design rule check. Verifies that the design follows all the rules specified such as trace width, minimum clearance, routing layers etc. found in Protel.

EDA

Electronic design automation, computer design tools specifically for electrical engineering.

PCB Design Assignment

ERC
Electrical rule check. Found in Protel schematic. Checks for unconnected pins, and basic electrical conflicts, bus conflicts etc.

Foot Print
The physical space any component takes on a circuit board. i.e. a TTL chip with 14 legs will take up approx. ½" x 1 ½" rectangular space. Includes copper connections on the PCB.

Layers
Each PCB consists of several layers, some physical, some conceptual.

Layer-Top
Usually component side. Physically the top of the PCB board.

Layer-Bottom
Physically the bottom of the PCB board. Most soldering is done on this side on professionally manufactured PCB's.

Layer-Keep-Out
Conceptual boundary in Protel where the designer defines the outline or size of the PCB. Required before auto routing can be performed.

Layer-Overlay
Optional layer. Drawing imposed on top and bottom layers of the PCB to aid in placing components. Overlay shows the footprint and designator of each component.

Milling Machine
Computer controlled machine that can remove copper from a plain copper board and create a PCB. Accepts Gerber files from Protel.

Mils
One mil is one thousandth of an inch. 1 inch = 25.4 millimeters

Pad
Copper surrounding the drill hole on the PCB. You solder the component to the pad.

PCB
Printed circuit board, copper traces on a fiberglass board. Can be from one to thirty two layers

Silkscreen
Same as top/bottom overlay

SIP
Single-in-line-package. Footprint for a single row of connectors.

Surface mount

PCB Design Assignment

SMD/SMT
Smaller components that appear to be "glued on" to the PCB. Uses small footprints with no holes through the board, they sit on surface of the copper, hence surface mount.

Through-Hole
Old school technology where components penetrate the copper board. DIP and SIP are all through-hole footprints.

Via
Connection between different layers on a circuit board. Can be blind or buried.

Wire Wrap
Alternative prototyping method. Small wires are attached to the parts

Trace Carrying Capacity
per mil std 275

Temp Rise	10 C			20 C			30 C		
Copper	1/2 oz.	1 oz.	2 oz.	1/2 oz.	1 oz.	2 oz.	1/2 oz.	1 oz.	2 oz.
Trace Width	Maximum Current Amps								
.010	.5	1.0	1.4	0.6	1.2	1.6	.7	1.5	2.2
.015	.7	1.2	1.6	0.8	1.3	2.4	1.0	1.6	3.0
.020	.7	1.3	2.1	1.0	1.7	3.0	1.2	2.4	3.6
.025	.9	1.7	2.5	1.2	2.2	3.3	1.5	2.8	4.0
.030	1.1	1.9	3.0	1.4	2.5	4.0	1.7	3.2	5.0
.050	1.5	2.6	4.0	2.0	3.6	6.0	2.6	4.4	7.3
.075	2.0	3.5	5.7	2.8	4.5	7.8	3.5	6.0	10.0
.100	2.6	4.2	6.9	3.5	6.0	9.9	4.3	7.5	12.5
.200	4.2	7.0	11.5	6.0	10.0	11.0	7.5	13.0	20.5
.250	5.0	8.3	12.3	7.2	12.3	20.0	9.0	15.0	24.5