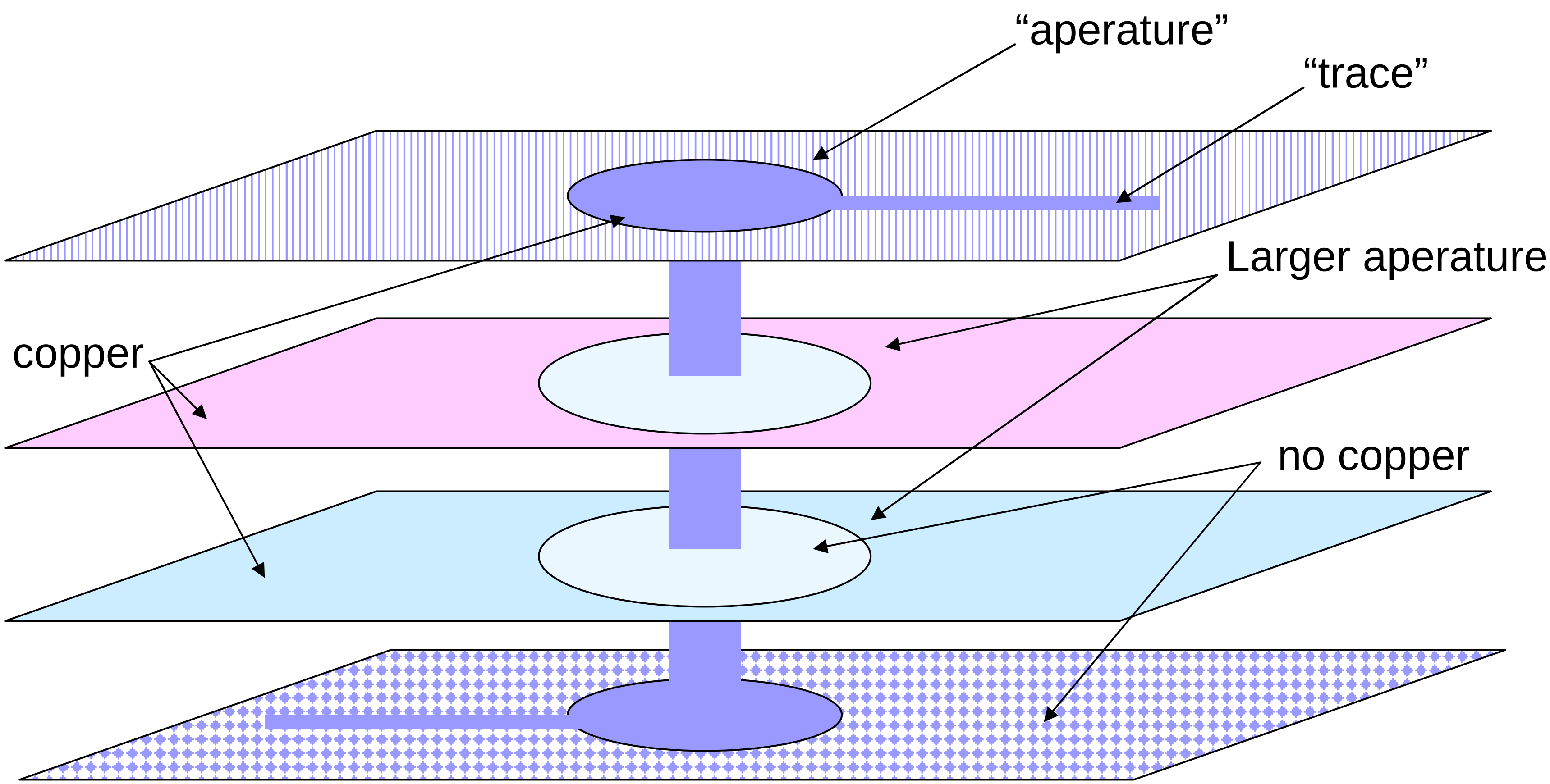
PCB Design and Assembly

ECE 189A – Oct 11, 2005

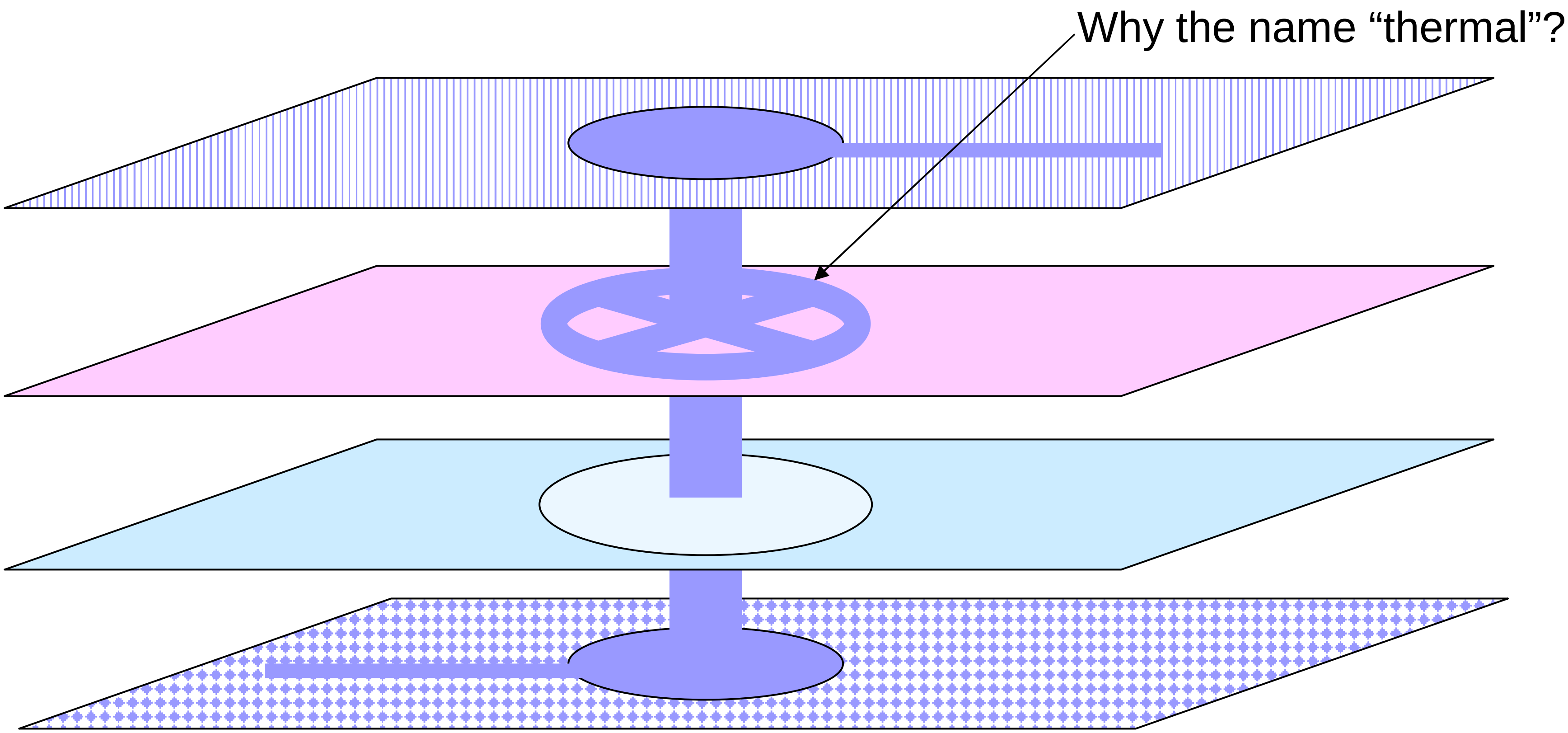
Board Details

- Four layer boards
  - Top and bottom layers for signal routing
  - Middle (or inner) layers for power and ground
    - Inner planes are solid copper
- Top side referred to as “component side”
  - Initially blank with component pads & traces added
- Bottom side referred to as “solder side”
  - Blank with pads & traces added

Changing layer e.g. from top side to bottom side requires a “via”



Special aperature called a “thermal” allows attachment to an inner plane



Thermals

- Special consideration is required for making soldered connections between the power/ground layers and pins. The term “thermal” refers to the portion of the foil layer of a power/ground plane where the connection is made to a pin. Heat that is applied to a pin which is in contact with the copper layer will be drawn away by the relatively large thermal mass of the layer. As a means to thermally (but not electrically) isolate the hole from the rest of the layer some of the copper outside the perimeter of the hole is etched away in the fabrication process leaving a few radial paths of copper to make the electrical connection from the pin to the foil layer. If pins are connected to the power plane using thermals then the thermal must be able to carry the required current without excessive local heating. Note that if heating is taking place at the connector between the pin and the board while the board is operating then there is probably an excessive voltage drop.

Electrical Design Standards for Electronics to be used in Experiment Apparatus at Fermilab, Revision 6.0 4/15/99

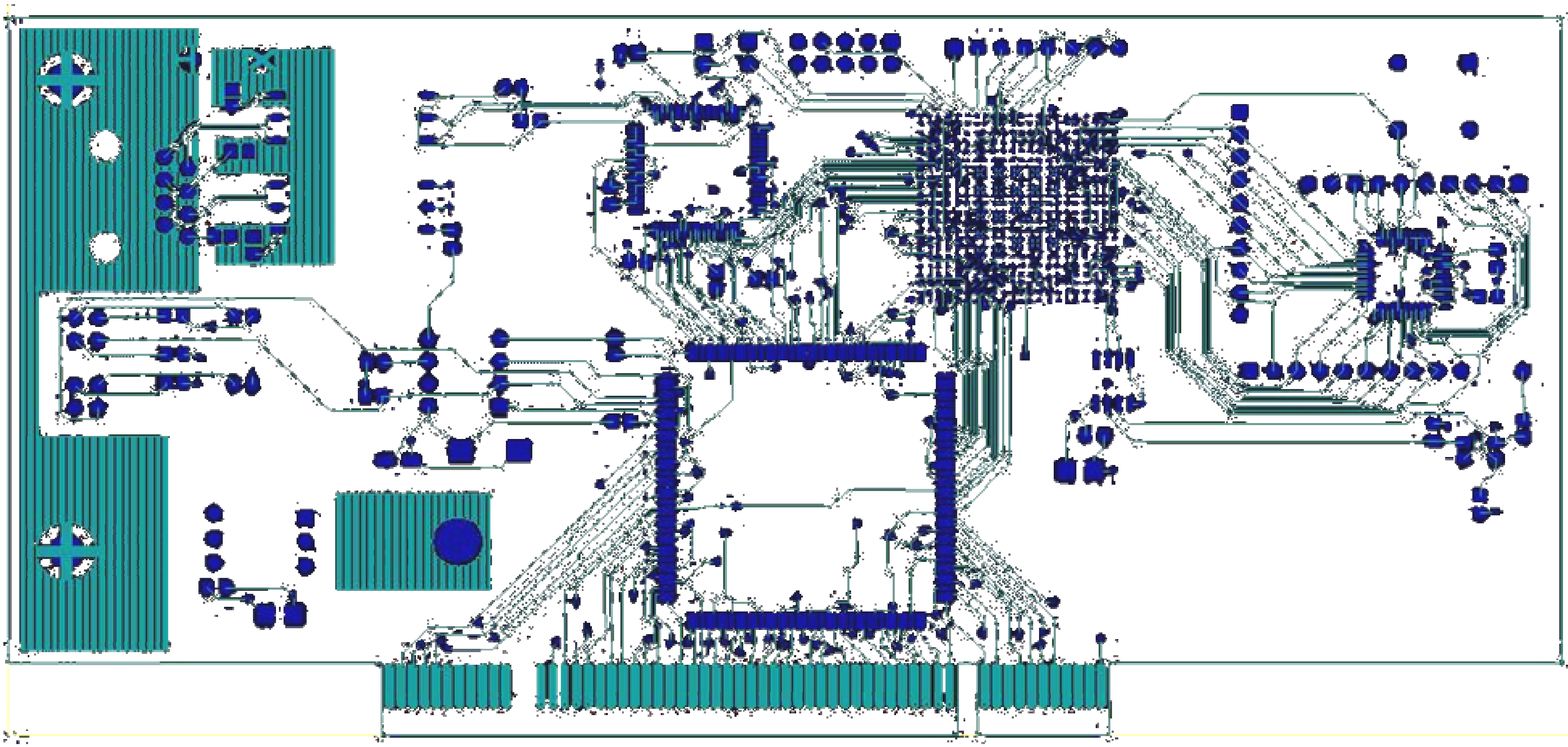
Pad Stacks

- Set of apertures implementing all flavor of via or through hole
- More complex issues in multi-layer boards
  - “Blind vias”
    - vias between inner planes … not for us
- Often we have several pad stacks defined
  - Power connectors (large diameter holes)
  - Vias (small diameter holes)
  - Mounting holes (special electrical details)

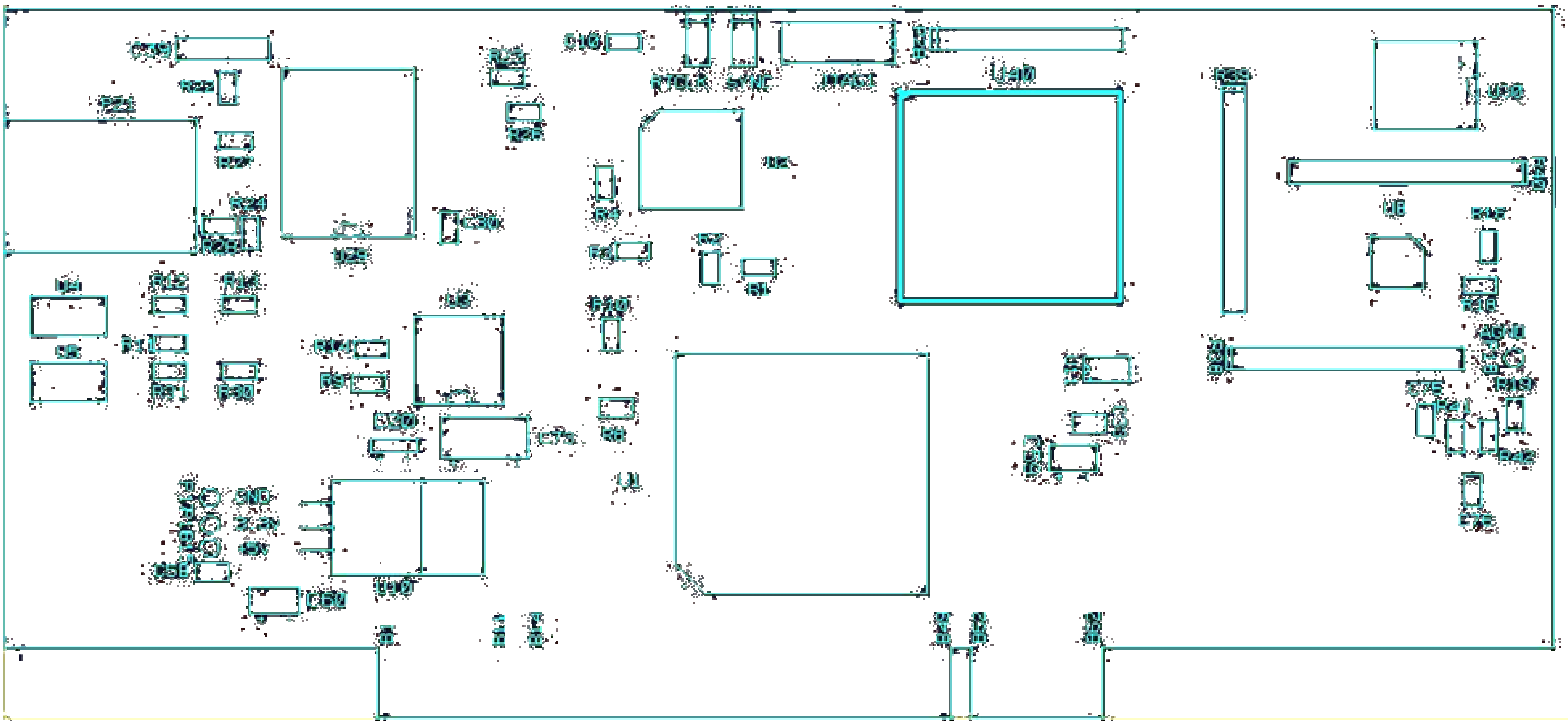
Gerber Plots --- outputs from Eagle

- Gerber plots for PCB (one per layer)
  - Component side (pattern copper interconnect)
  - Solder side (pattern copper interconnect)
  - Power plane (often negative mask)
  - Ground plane (often negative mask)
  - Component side solder mask (controls reflow)
  - Solder side solder mask (controls reflow)
  - Component side silkscreen (painted labeling)
  - Solder side silkscreen (optional, extra cost; only needed if components placed on “solder side”)
  - Solder paste masks (component and solder sides)
    - Only needed for automated assembly

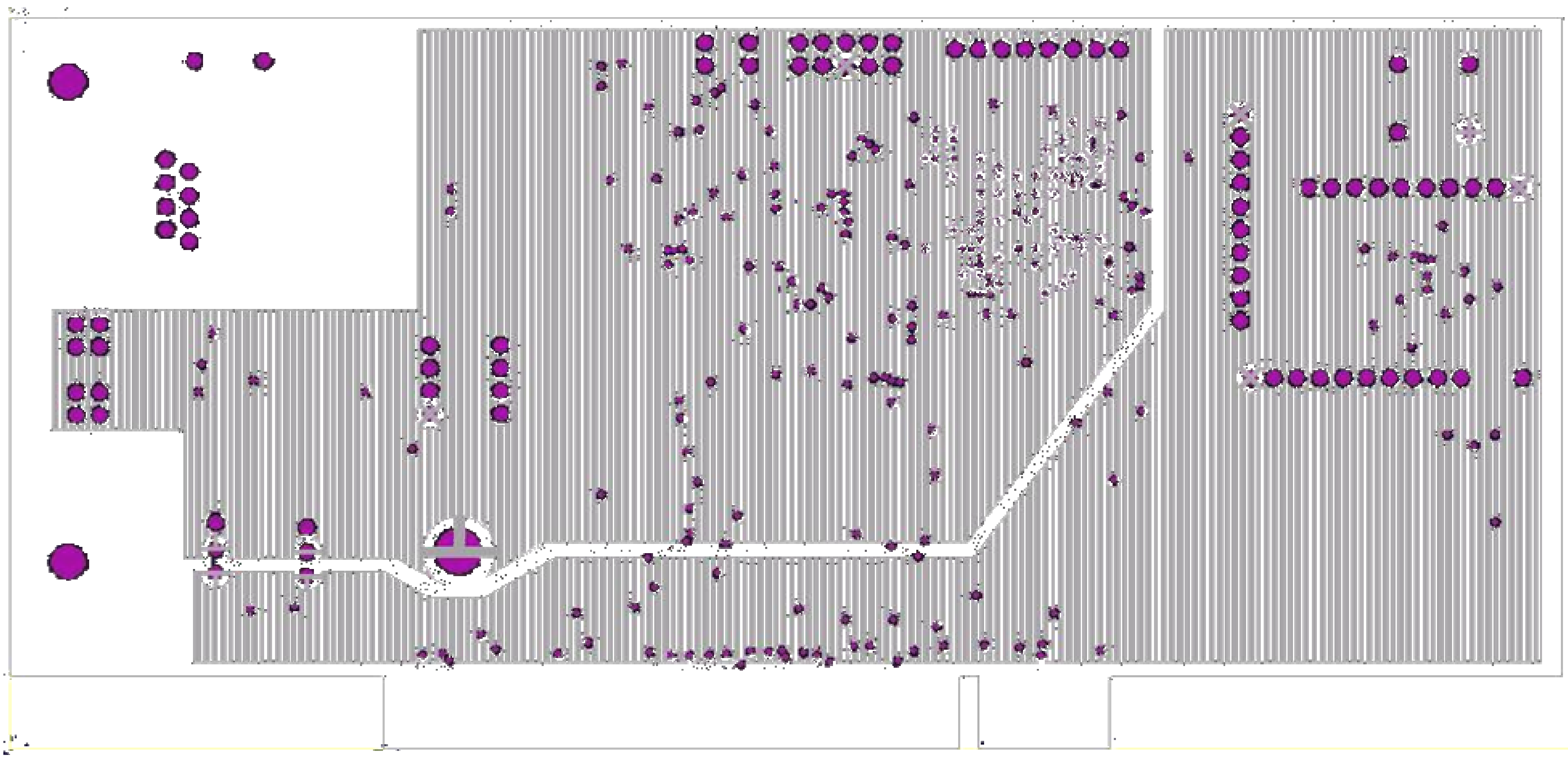
Component Side



Component Side Silkscreen



Inner Plane



AutoRouting and Beyond

- Ideally, Eagle would do all the work of generating a printed circuit board design from your schematic
  - If, by some chance, Eagle does not automatically create the perfect PCB, here are some guidelines in designing the board…

Placing Components

- Generally, it is best to place parts only on the top side of the board
  - (Rare) Exceptions: resistors and small passive components that won't be needed as test points
- When placing components, use the *snap-to-grid* feature
  - Eagle defaults to 0.050" for the snap grid

Placing Components

- First: place all the components that need to be in *specific* locations
  - e.g. connectors, switches, LEDs, mounting holes, heat sinks or any other item that needs to be mounted in a particular location.

Placing Components

- Give careful thought when placing components to minimize trace lengths.
  - Put parts next to each other that connect to each other.
    - a good job here makes routing the traces much easier.
- Arrange ICs in only one or two orientations: up or down and/or right or left.
  - if possible, align each IC so that pin #1 is in the same place for each orientation, usually on the top and/or left sides.

Placing Components

- Position <u>polarized</u> parts (i.e. diodes, electrolytic caps, etc.) with the positive leads all having the same orientation
  - Use silkscreen layer to clearly identify polarity
  - Use a square pad or silkscreeen “+” to mark the positive leads of these components

Placing Components

- Leave ample space (and then some more) between ICs for traces.
  - 0.350" - 0.500" between small-to-medium ICs
  - Allow proportionally more for larger ICs
  - Largely based on number of pins and location

Placing Components

- After placing all the components, print out a copy of the layout
- Lay each component on top of the layout
  - Check that you have allowed enough space for every part to rest without touching any others
  - Be 100% sure that the part fits its footprint !!!

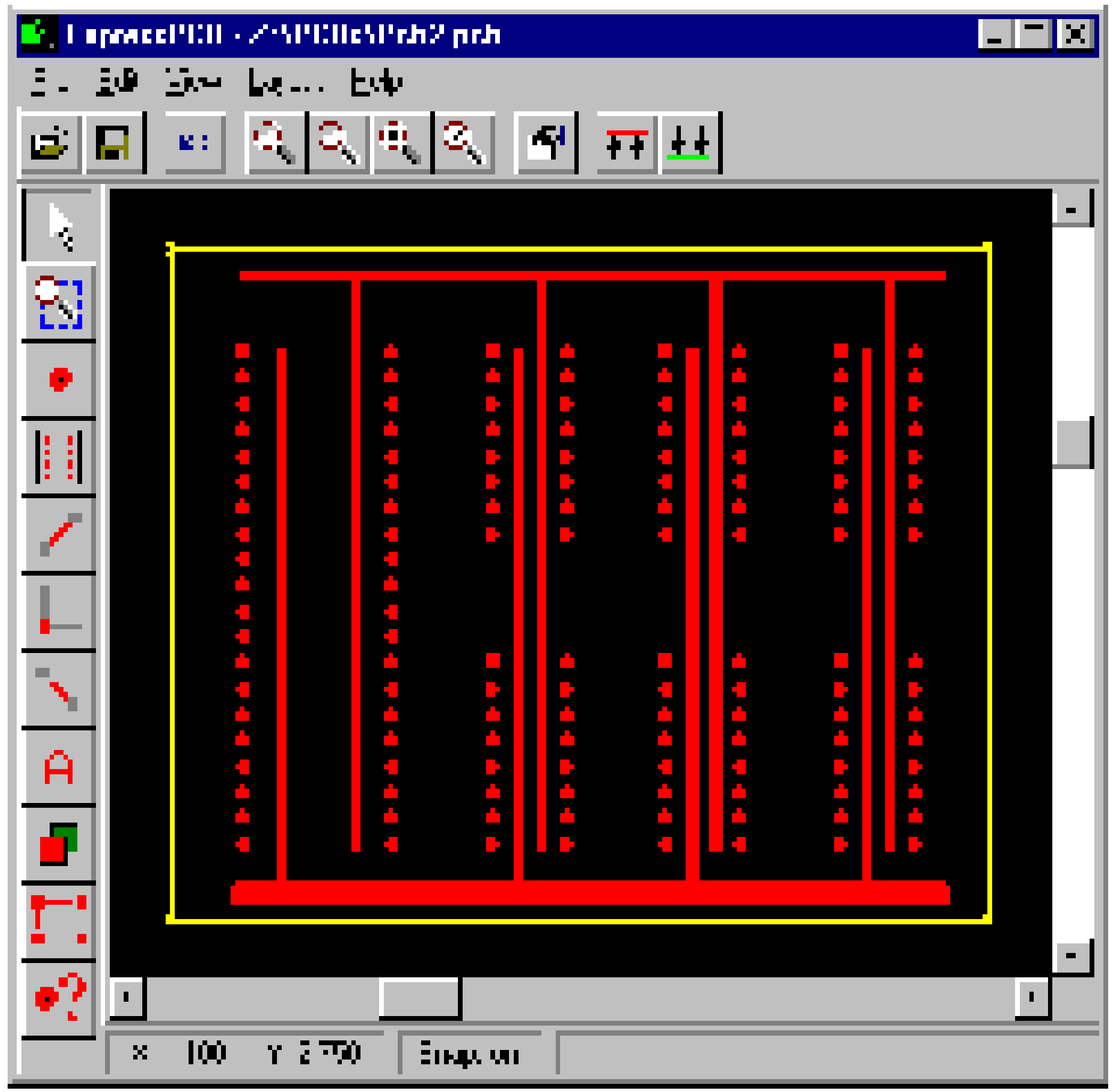
Placing Power and Ground Traces

For designs with multiple supply voltages…

- After the components are placed, layout the non-plane power and ground traces
  - essential to have solid power and ground lines, using wide traces that connect to common rails for each supply
  - very important to avoid snaking or daisy chaining the power lines from part to part

Placing Power and Ground Traces

- Routing multiple voltages



Planning Signal Traces

- Make traces as short and direct as possible
- Use vias (also called feed through holes) to move signals from one side to the other
- 2 or 3 vias per net is probably OK
  - more may be required on large nets
  - think about net shape and overall length

Planning Signal Traces

- Generally, the best strategy is to …
  - put vertical traces on one side and horizontal traces on the other
  - add vias where needed to connect a horizontal trace to a vertical trace on the opposite side.
- A good trace width for low current digital and analog signals is 0.010”
  - minimum width design rule in Eagle is 8 mils

Planning Signal Traces

- Traces that carry significant current should be wider than signal traces
- Rough guidelines of how wide to make a trace for a given amount of current

0.010" 0.3 Amps
0.015" 0.4 Amps
0.020" 0.7 Amps
0.025" 1.0 Amps
0.050" 2.0 Amps
0.100" 4.0 Amps
0.150" 6.0 Amps

Planning Signal Traces

- Space between the trace and any adjacent traces (wires), pads and vias.
  - Insure that there is a minimum gap of 0.008" between items (wire, pad, via)
    - 8 mils is the Eagle design rule; 0.010" is better.
      - Leaving less blank space runs the risk of a short circuit developing in the board manufacturing process
      - It is also necessary to leave larger gaps when working with higher voltages

Planning Signal Traces

- When routing traces (as with placing components) set the *snap-to-grid* on
  - Eagle's snap grid spacing defaults to 0.050"
    - Changing to a value of 0.025" can be helpful when trying to work as densely as possible.
  - Turning off the snap feature may be necessary when connecting to parts that have unusual pin spacing.

Planning Signal Traces

- Restrict the direction that traces run to horizontal, vertical, or 45° angles
- When placing narrow traces ($\leq$ 0.012”) avoid 90° turns
  - In the manufacturing process, the outside corner can be over-etched (more narrow)
  - Use two 45° bends with a short leg in between

Checking Your Work

- Check the routing of every signal
  - Verify that nothing is missing or incorrectly wired
  - Trace through your schematic, one wire at a time
    - Carefully follow the path of each trace on your PC layout to verify that it is the same as on your schematic.
    - After each trace is confirmed, mark that signal on the schematic with a yellow highlighter.
- Check for missing vias

Board Costs

- Board size is significant
  - The smaller the board is, the cheaper it is
    - $130 + ($0.87 * NumberOfBoards * BoardAreaInSquareInches) + ($1.25 * NumberOfBoards) + Shipping
  - Some fab houses restrict the number of drill sizes or the number of drill holes
  - Our fab house requires that boards be rectangular

Board Costs

- Surface Mount Technology (SMT) vs. Through Hole Technology (THT)
  - SMT allows for more dense, and therefore smaller, PCBs than THT
  - However, if the board becomes very dense manufacturing and assembly costs increase
    - This could increase the cost of the PCB more than what is gained by reducing its size.

Board Costs

- The cost increases with the number of layers
  - But fewer layers often increase the size of the PCB.
- It takes time to drill the holes
  - Minimize the number of vias
- Buried vias are more expensive than vias that go through all the layers … we cannot use these(!)
  - Buried vias require to drill each layer separately before they are laminated together.

Board Costs

- The size of the holes in the PCB depends on the diameter of the component legs
  - If components with different types of legs are required on the same board the machine that drills the holes cannot use one single drill to drill all the holes.
  - The more times the drill has to be changed while processing one board, the more expensive the PCB is to manufacture.
  - Our board house limits the number of different drills used and the drill sizes must be from a fixed set

Component Mounting and Soldering

- The assembly process consists of mounting and soldering the components onto the PCB
- Wave Soldering
  - THT components are most often soldered in an automated process called wave soldering
  - All components are soldered simultaneously
  - Their legs are first cut near the board and slightly bent over to keep the component in place

Component Mounting and Soldering

- Wave Soldering (cont.)
  - The PCB is then moved over a wave of liquid flux, such that the bottom side strikes the flux
    - This removes any oxide from the metal surfaces
  - After heating, the PCB is moved over a wave of melted solder
  - The solder attaches to the solder pads and component legs, and the soldering is complete

Component Mounting and Soldering

- Reflow Soldering
  - common (automated) soldering of SMT components
  - solder paste (containing both flux and solder) is applied to the solder pads
    - before the components are placed on the PCB
    - solder paste application is a silkscreen-like process
      - using Eagle's "cream" mask
  - Components placed (often automated with pick'n place machine)
  - PCB then heated in an oven
    - solder in the paste melts (i.e. it *reflows*)
  - Cooling the PCB completes this type of soldering