

Homebrewing PC Boards

Designing and Fabricating
Circuit Boards
Using Eagle CAD Software

Homebrewing PC Boards

- CAD Fundamentals
- Obtaining Eagle freeware
- Drawing schematics
- Board layout
- PC board fabrication

CAD Fundamentals

- Draw (capture) the schematic first
- Create new symbols & footprints as needed, store them in a library
- Transfer design to board layout
- Place components on board
- Route circuit traces
- Add mechanical features

CAD Fundamentals

- Why Schematic Capture?
 - It documents your project, even if copying it from another source
 - Footprints accompany schematic symbols in the design files
 - You modify or create your own footprints here
 - A netlist is created from the schematic, important for board layout and error checking

CAD Fundamentals

- Creating new symbols & footprints
 - Build a library of your custom parts
 - Symbol and footprint editor is built-in
 - User groups provide support
 - Calipers and datasheets are your best friends

CAD Fundamentals

- Board Layout
 - Eagle places footprints at random on the screen along with a blank board outline
 - Components are connected by thin traces, forming a rat's nest
 - Move each component to the board so that a signal flow develops and rat's nest is minimized
 - Draw traces to connect the components
 - Fill in ground plane by hand

Eagle CAD

- Advantages
 - Available in a Freeware version
 - Able to make double sided boards
 - Generates Gerber files for professional PCB fabrication
- Disadvantages
 - Limited board size: 3.2 x 4 inches
 - Limited to single & double sided boards
 - Significant learning curve

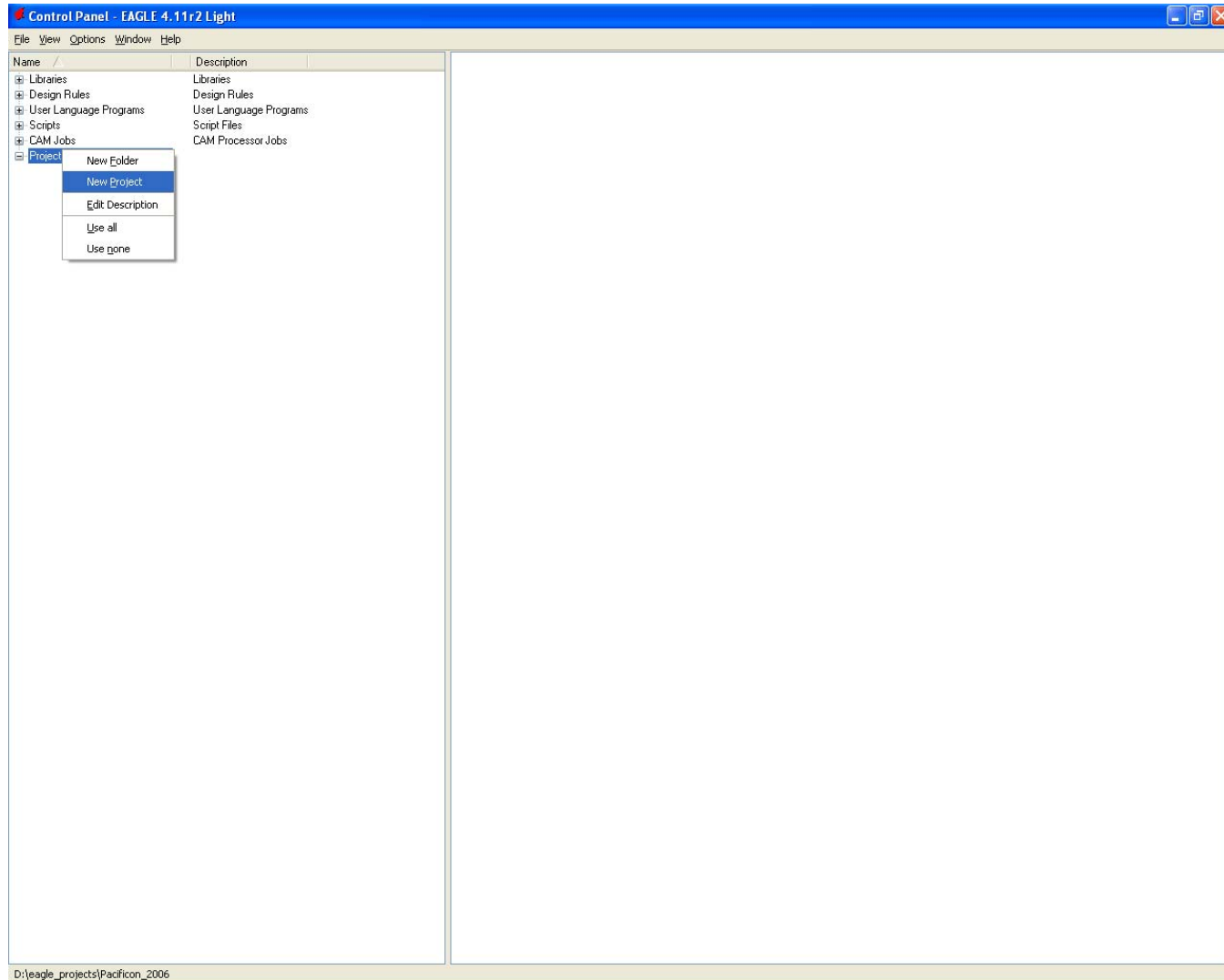
Obtaining Eagle Freeware

- Homepage
 - www.cadsoft.de
- Guided Tour
 - www.cadsoft.de/Tour/tour00.htm
- Freeware Download
 - www.cadsoft.de/freeware.htm

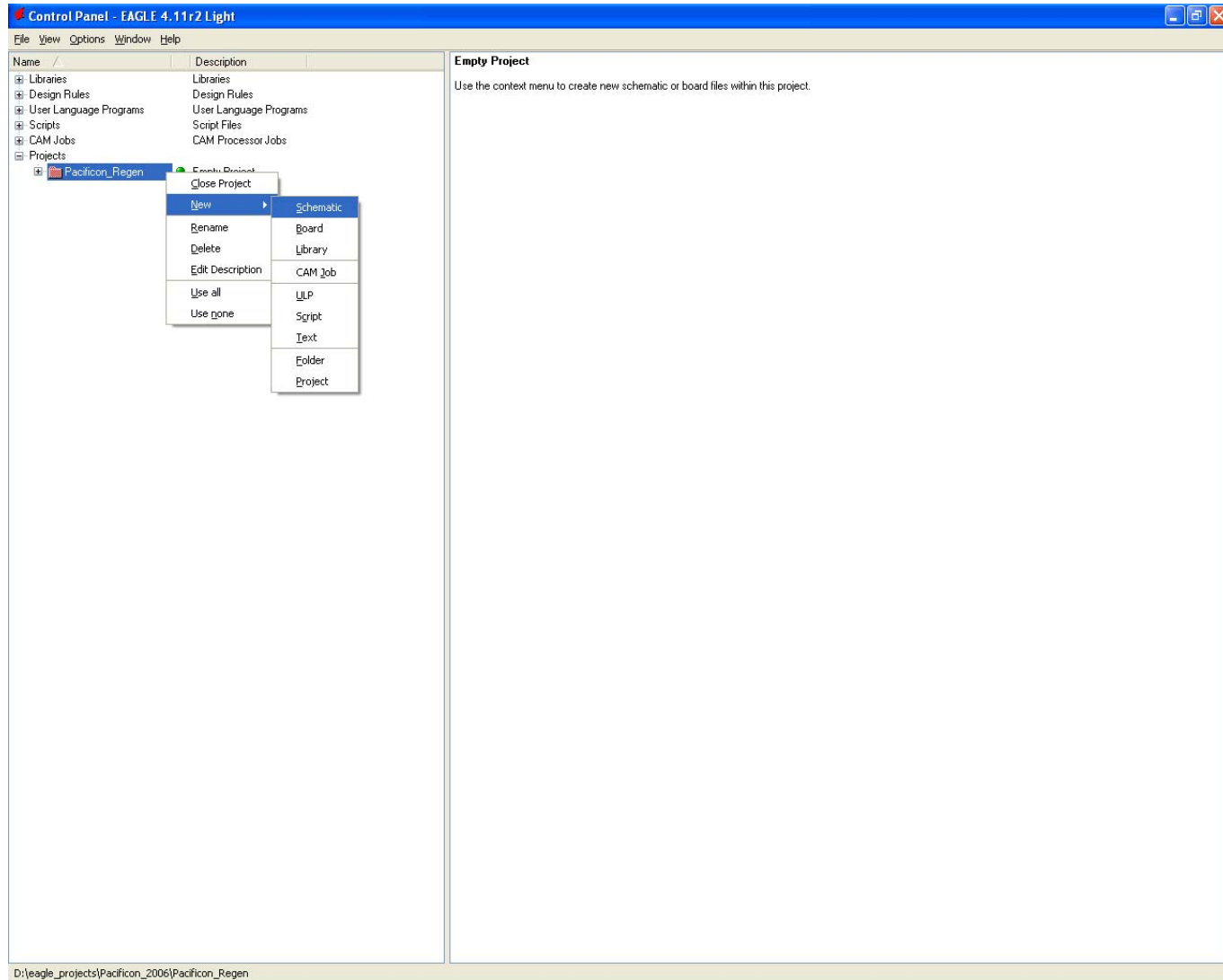
Eagle in a Nutshell

- Start the software
 - Create a project
 - Add a new schematic

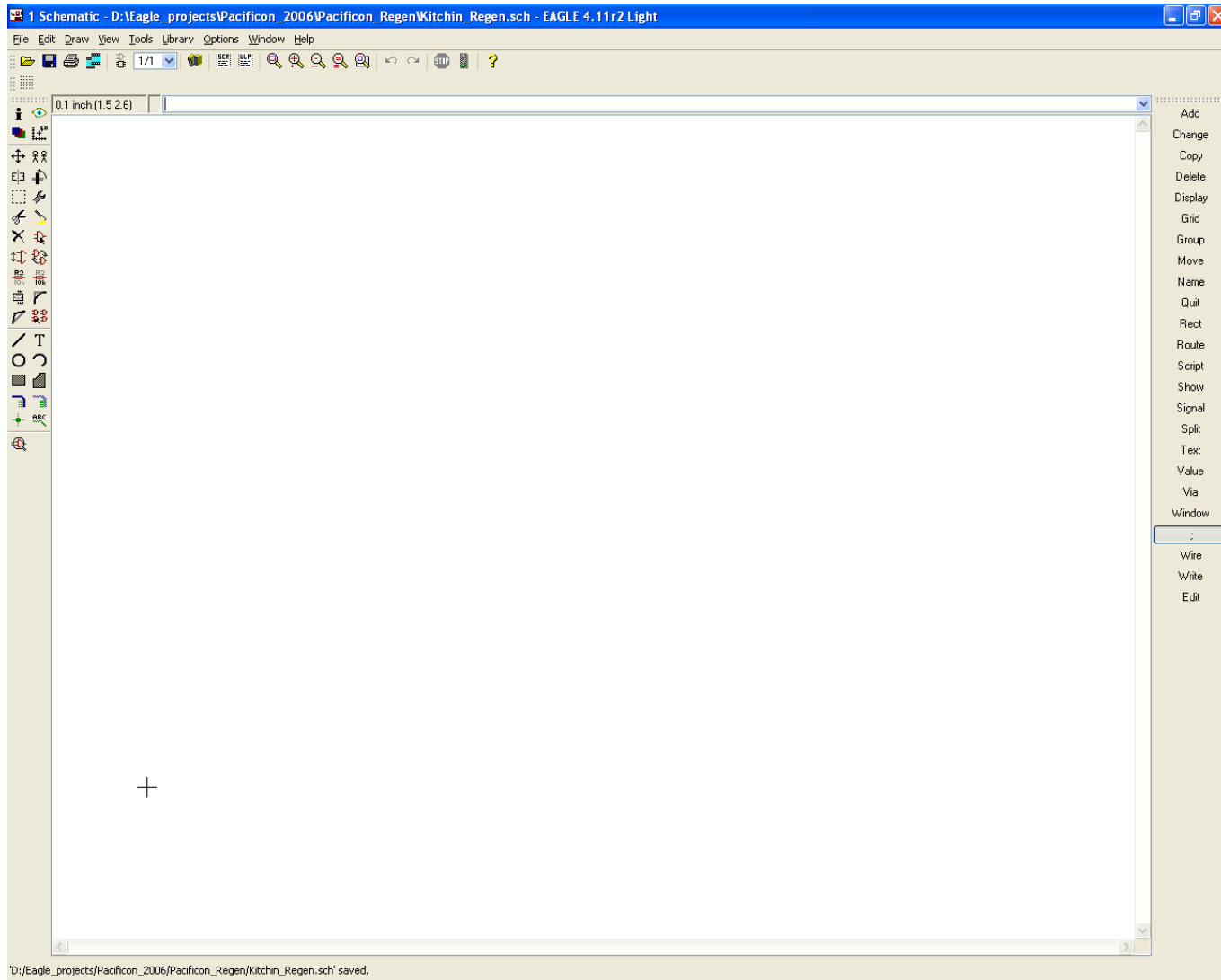
Eagle in a Nutshell



Eagle in a Nutshell

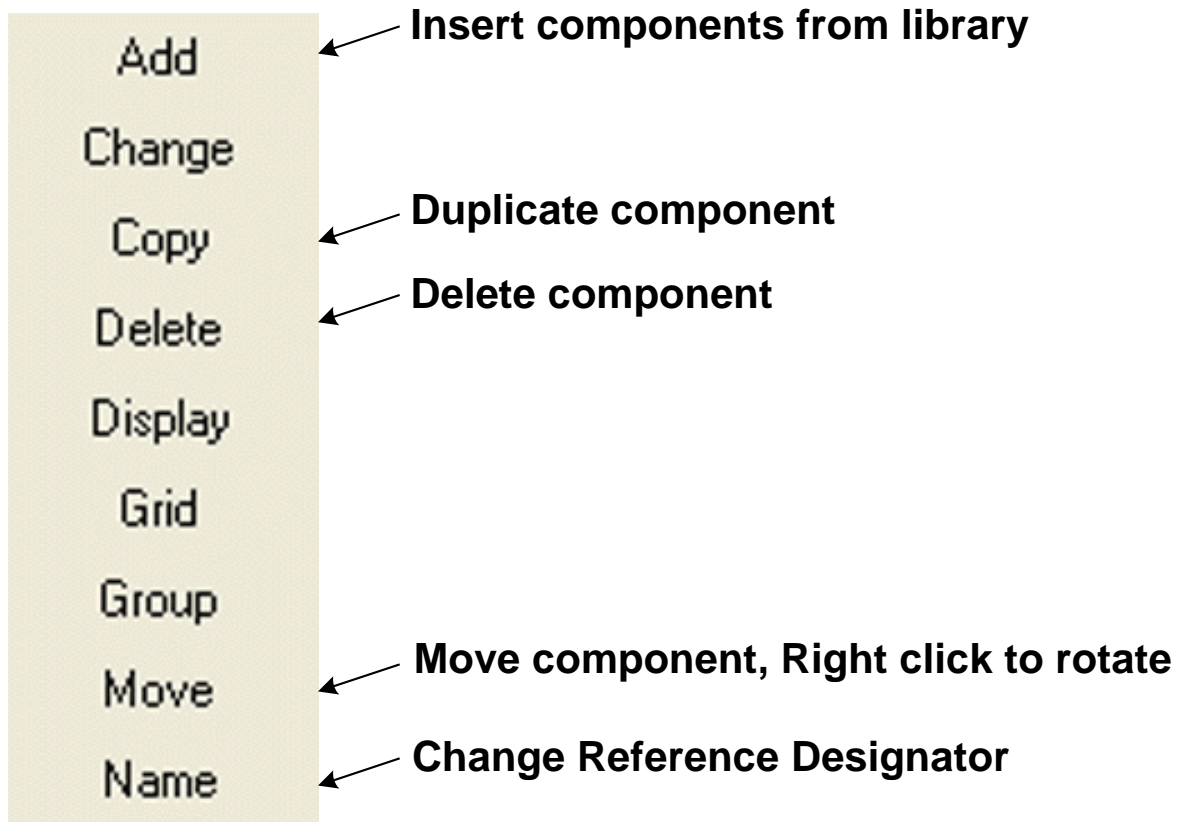


Eagle in a Nutshell



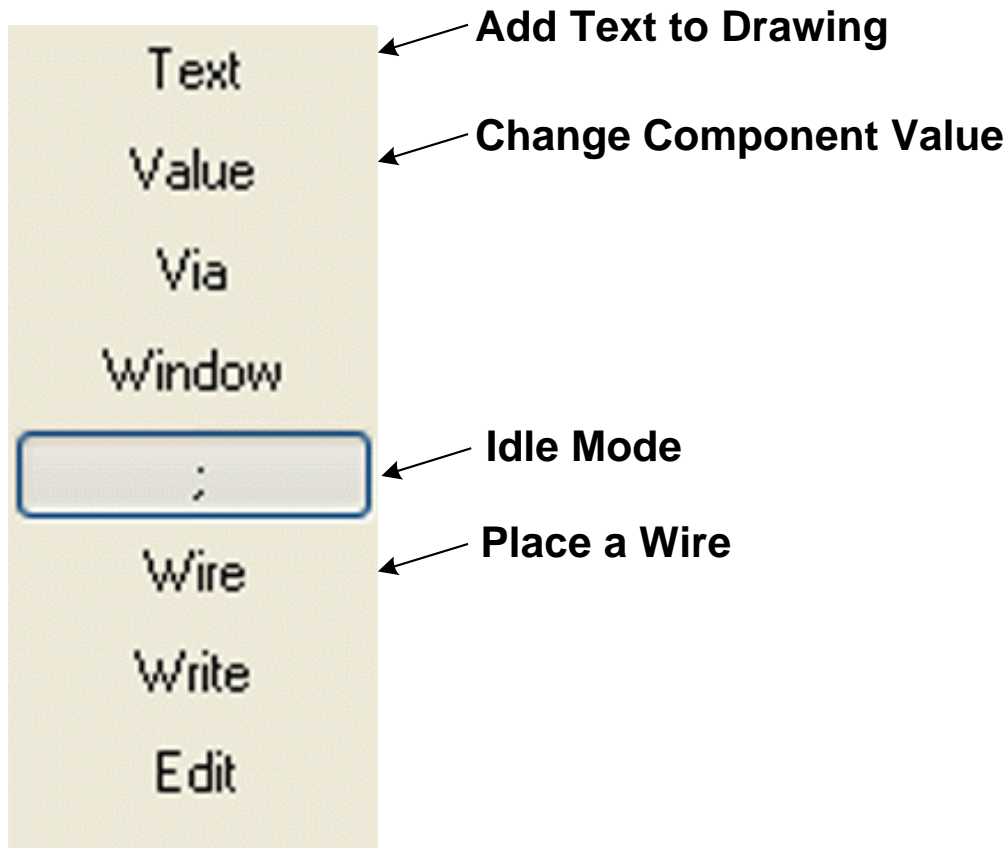
Eagle in a Nutshell

Right Toolbar Tour – Part 1



Eagle in a Nutshell

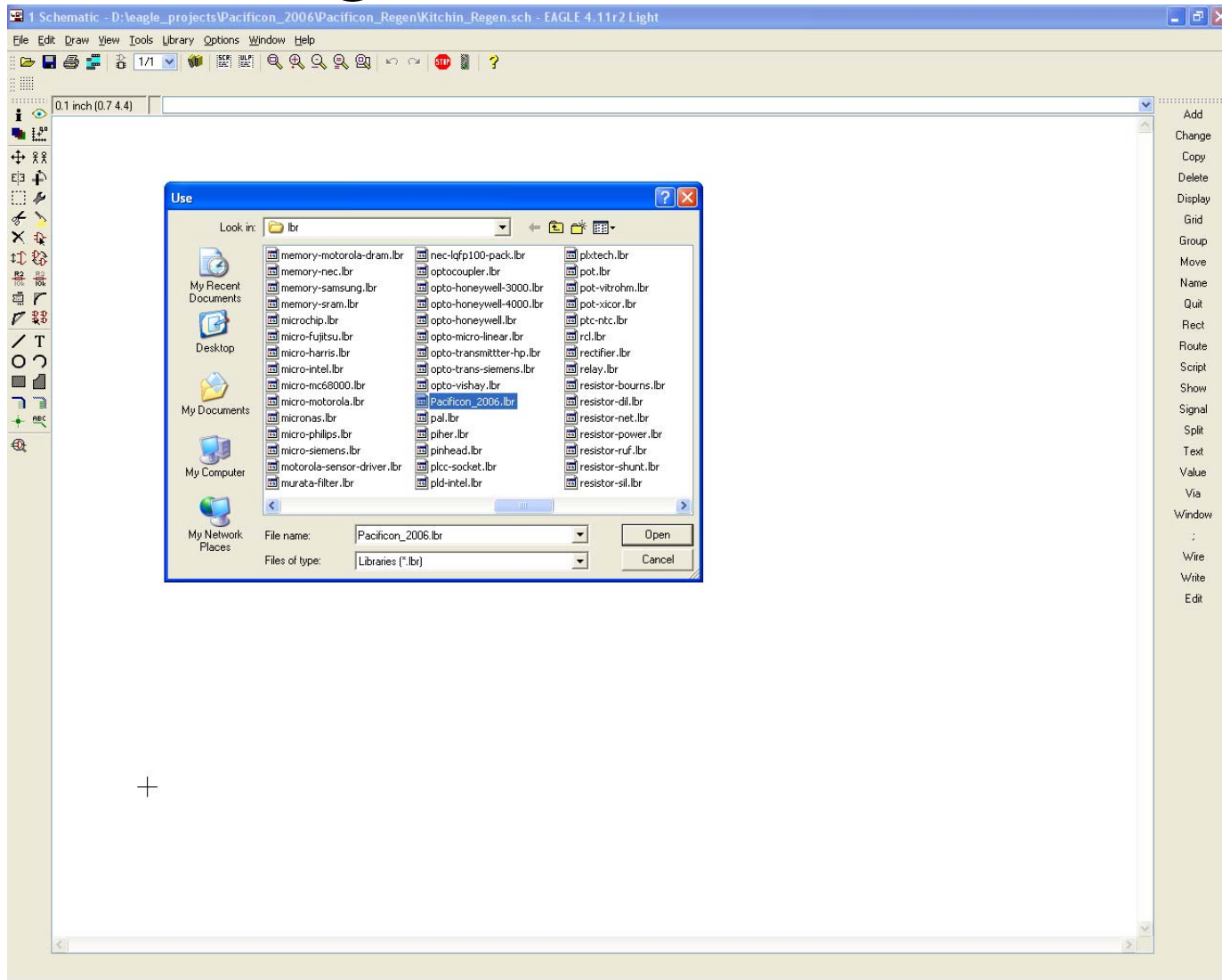
Right Toolbar Tour – Part 2



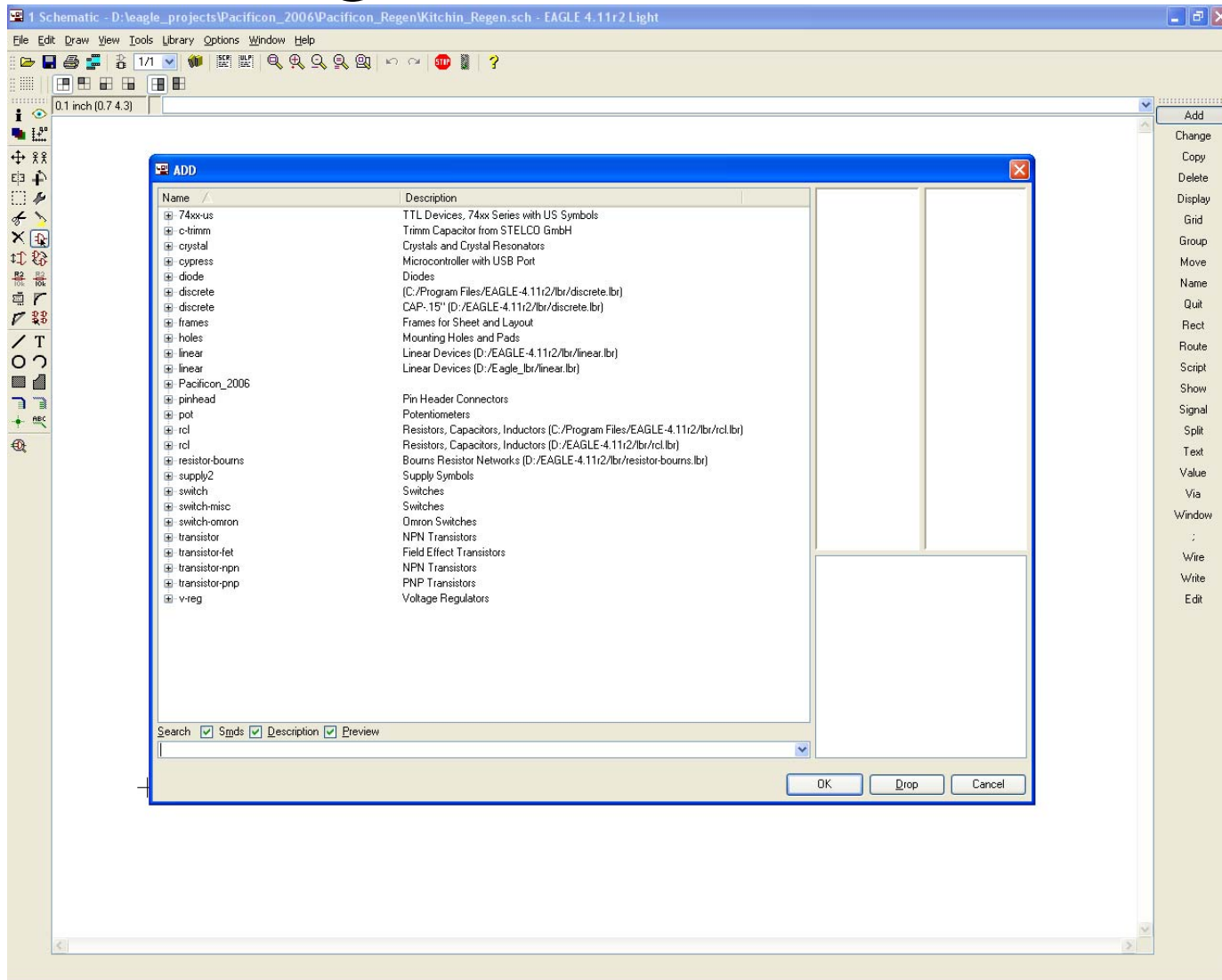
Eagle in a Nutshell

- Drawing the schematic
 - Copy Pacificon_2006 to:
 - C:\Program Files\EAGLE-4.11r2\lbr
 - Tell Eagle to use the Pacificon_2006
 - Add components
 - Connect them
 - Add mounting holes and text

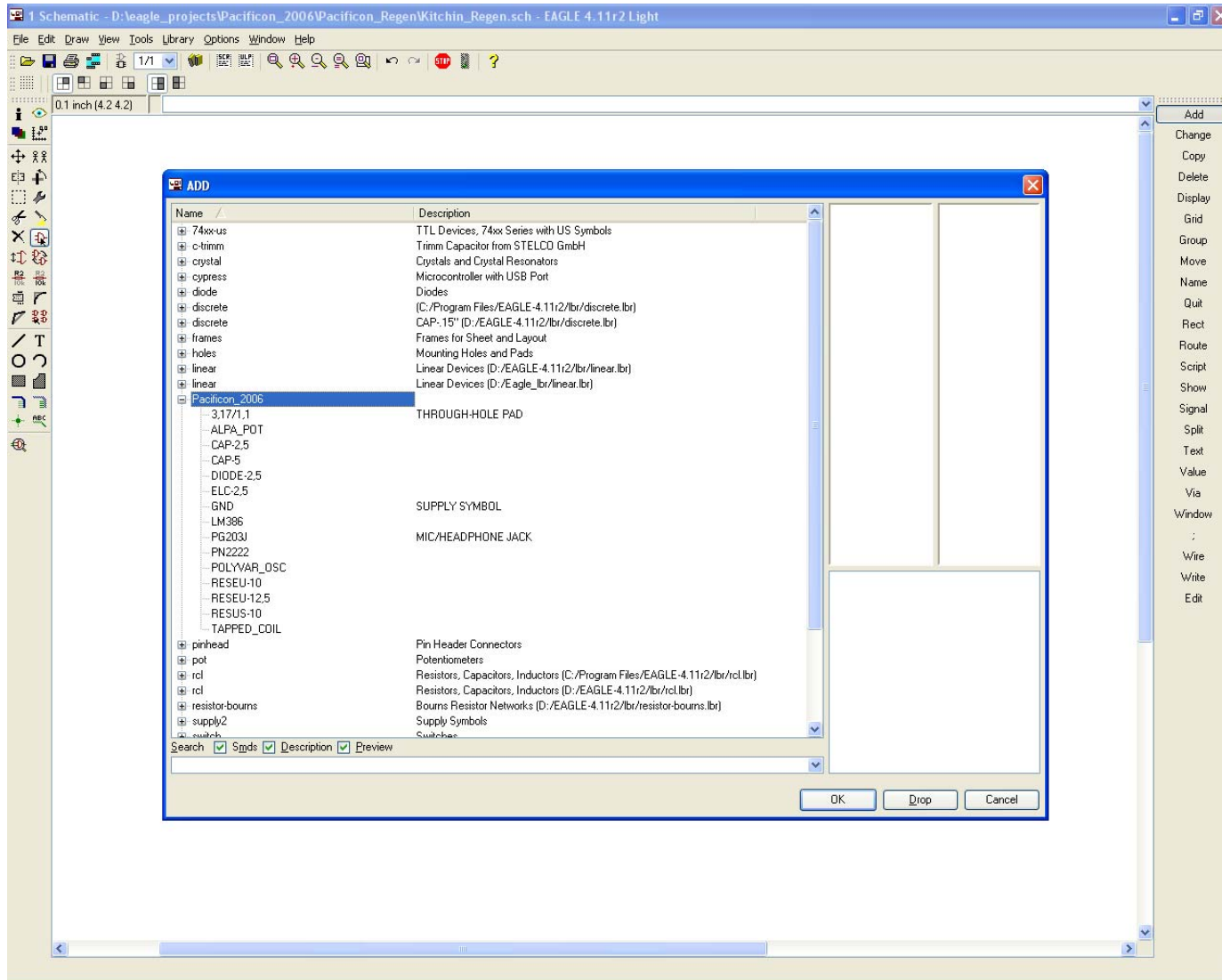
Eagle in a Nutshell



Eagle in a Nutshell



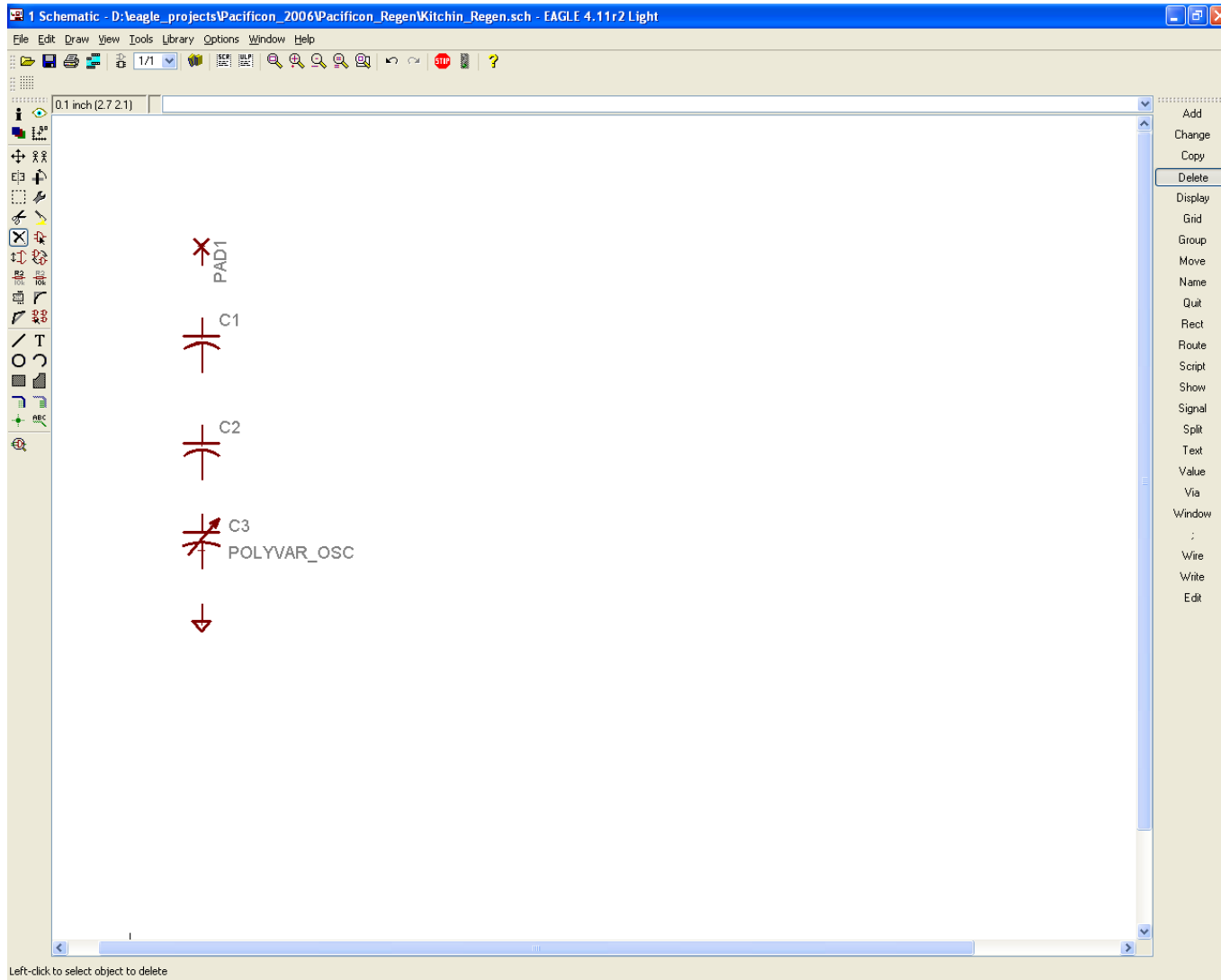
Eagle in a Nutshell



Eagle in a Nutshell

- Add the following components. Right click to rotate. Left click to place. ESC to end repeat placement.
 - One 3,17/1,1
 - Two CAP-5
 - One POLYVAR_OSC
 - One GND

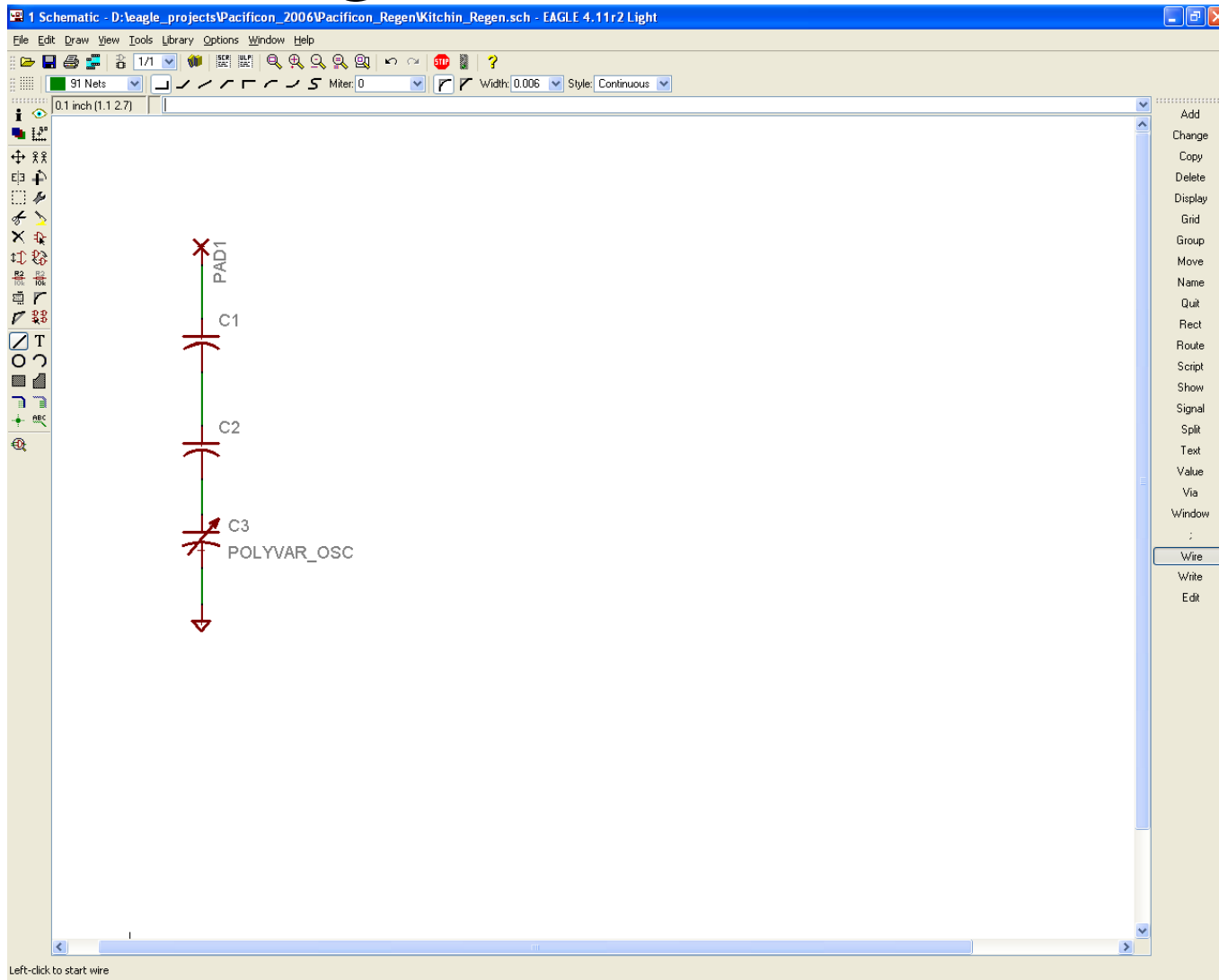
Eagle in a Nutshell



Eagle in a Nutshell

- Click on the Wire Tool to connect the components
 - Move cursor to starting node and left click once to start wiring
 - Move to destination node and left click twice to stop wiring

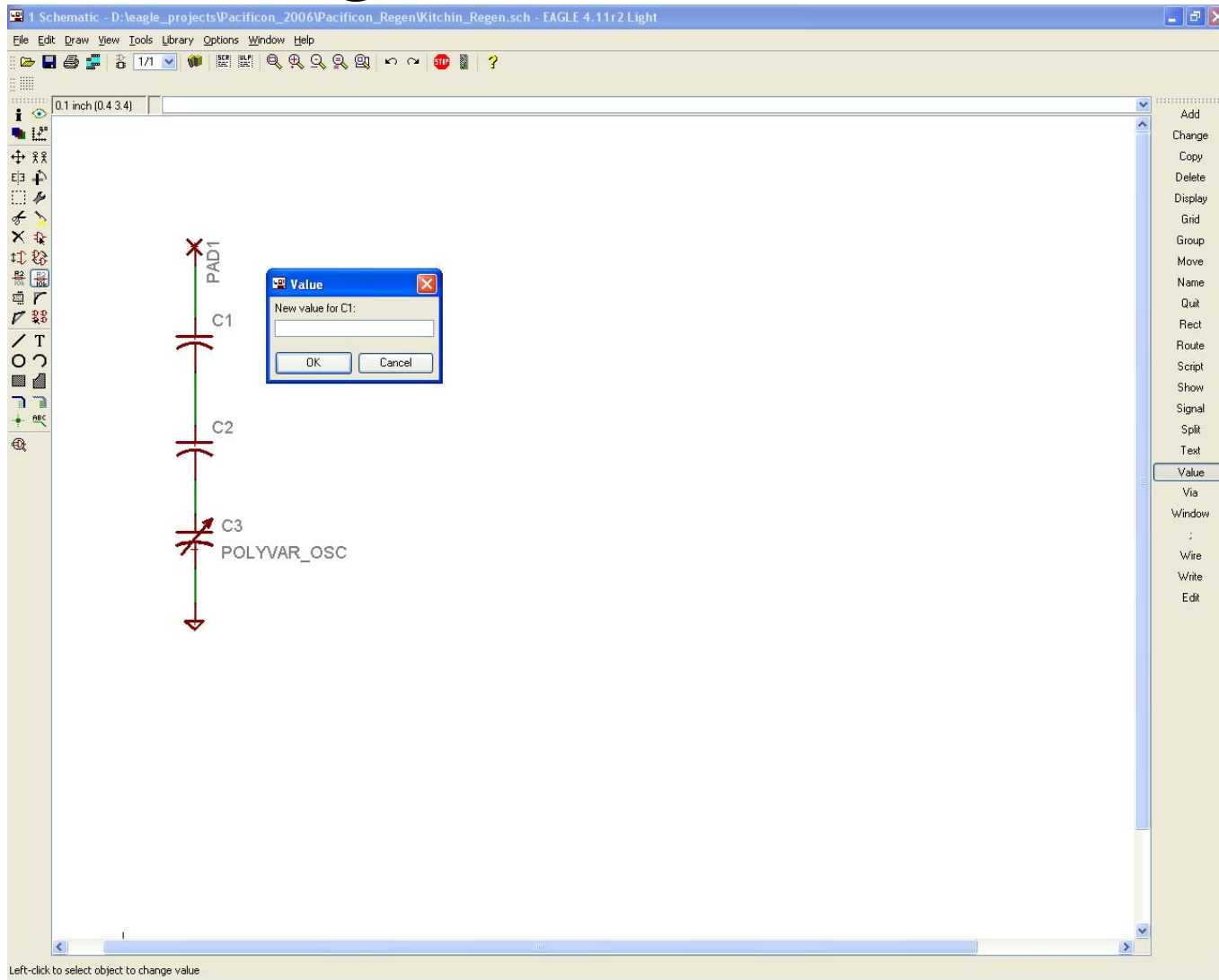
Eagle in a Nutshell



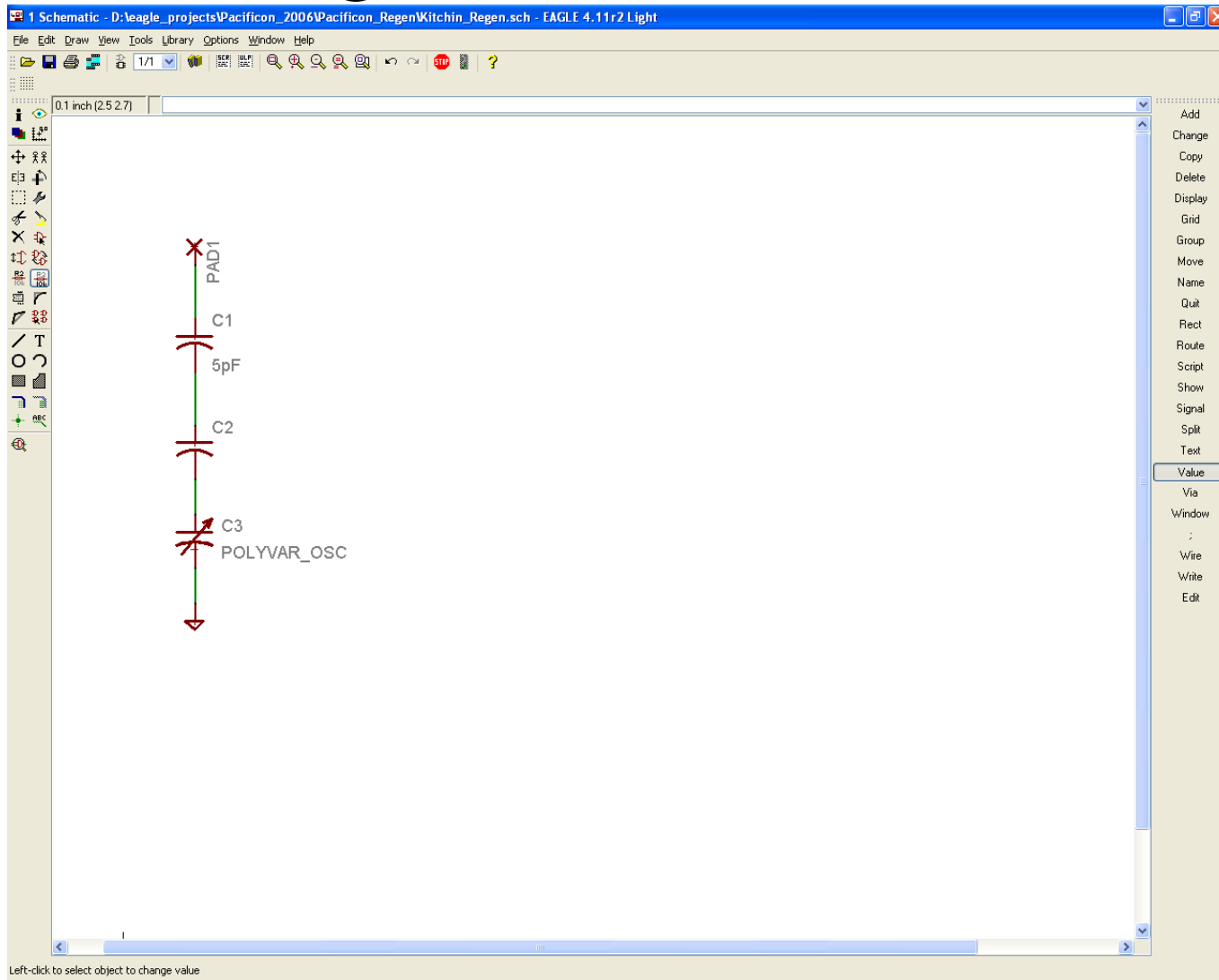
Eagle in a Nutshell

- Use the Value Tool to add component values.
 - Left click on the component
 - Type the component value & hit Return or left click the OK button

Eagle in a Nutshell



Eagle in a Nutshell



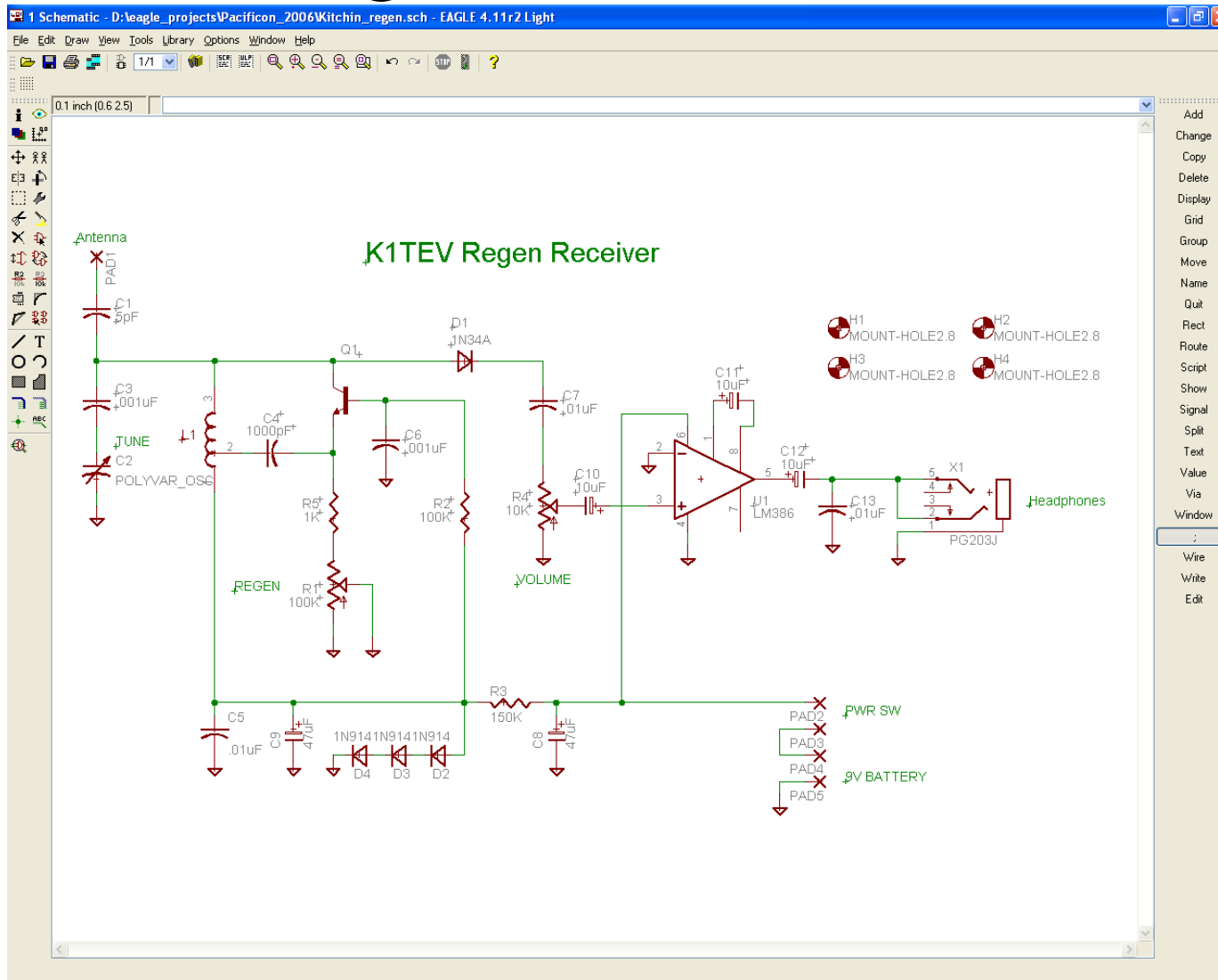
Eagle in a Nutshell

- Continue placing components, assigning values and wiring them up.
- Use the Junction Dot on the left tool bar to connect wires.


Eagle in a Nutshell

- Use the Text tool to add descriptions
- Use the Change tool to change the text size
- Finally, add four 2,8 mounting holes from the library

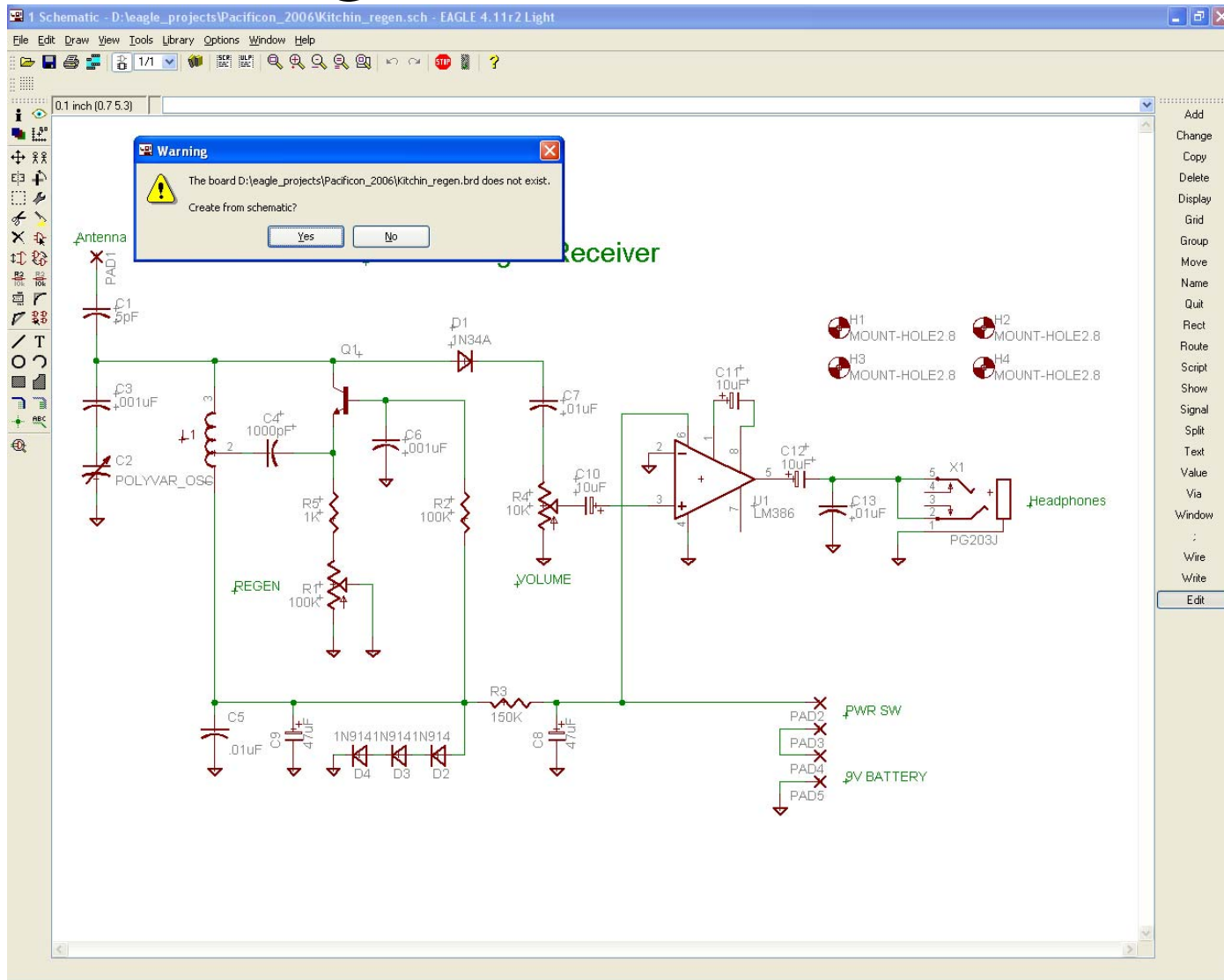
Eagle in a Nutshell



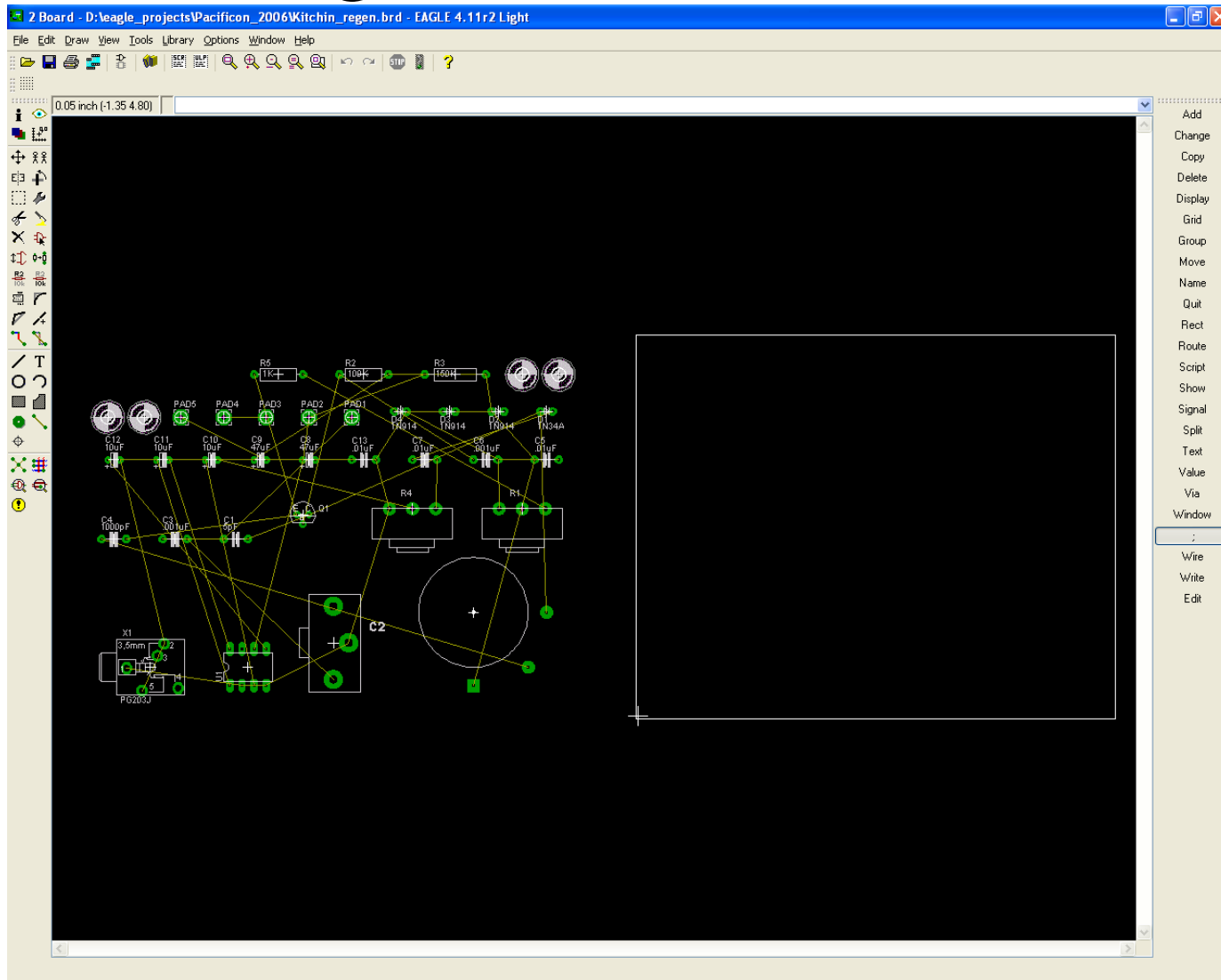
Eagle in a Nutshell

- Schematic is complete, it's time to transfer the design to the board
- Click on the  icon on the top toolbar. This creates a board layout
- You can also open the File menu and select the “Switch to Board” item

Eagle in a Nutshell

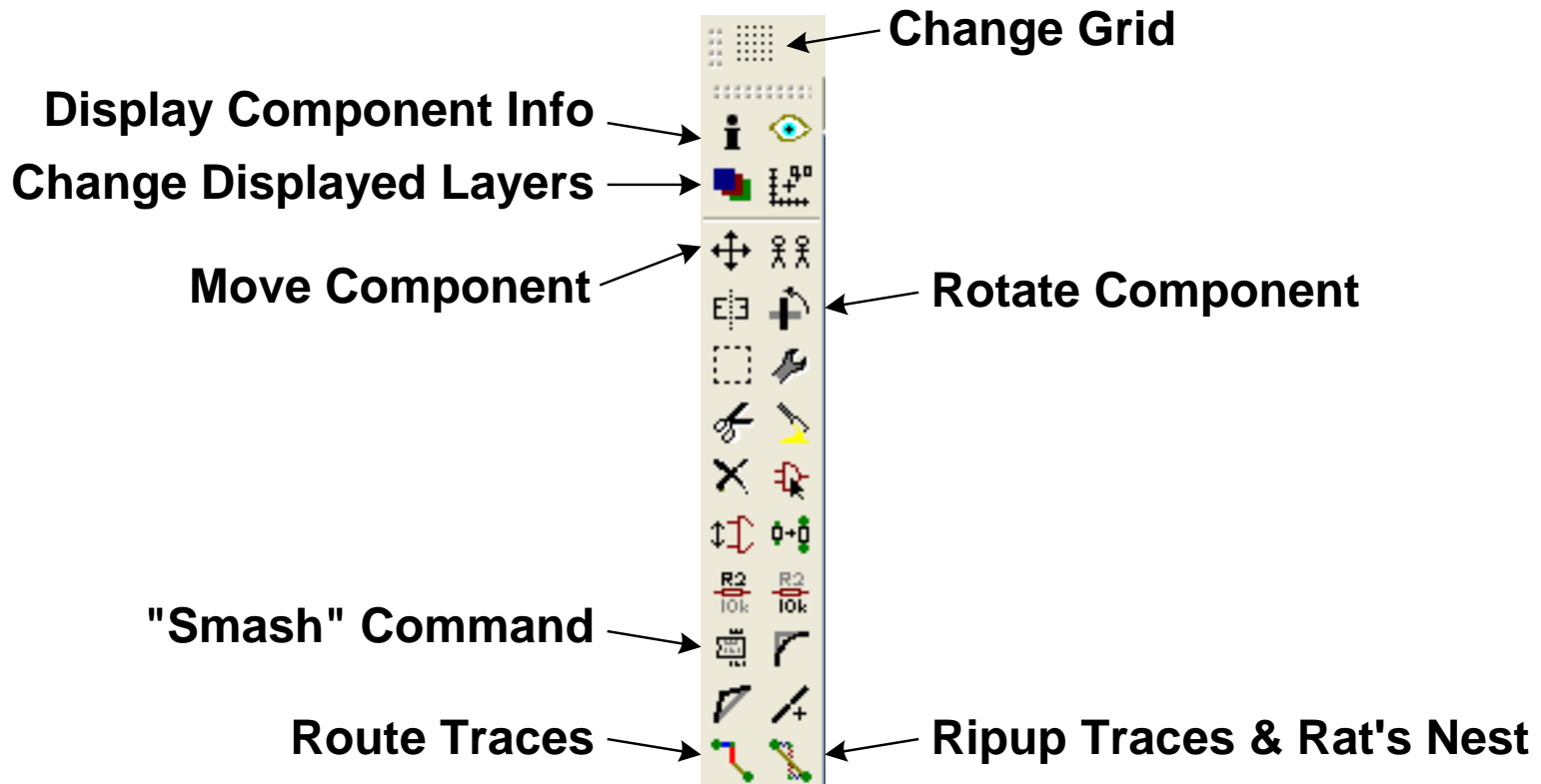


Eagle in a Nutshell



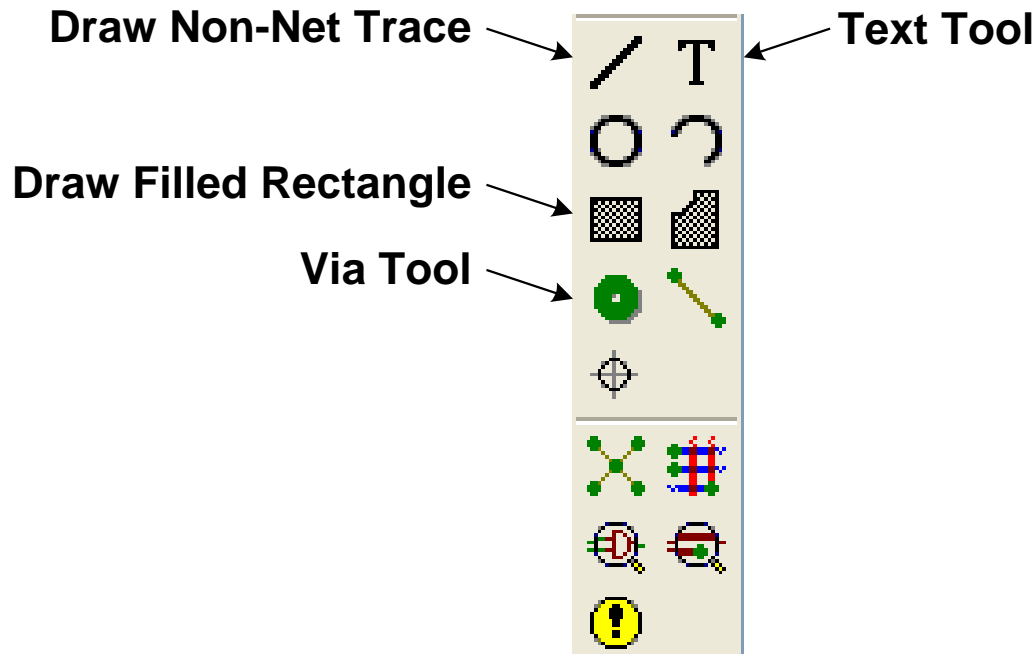
Eagle in a Nutshell

Left Toolbar Tour – Part 1

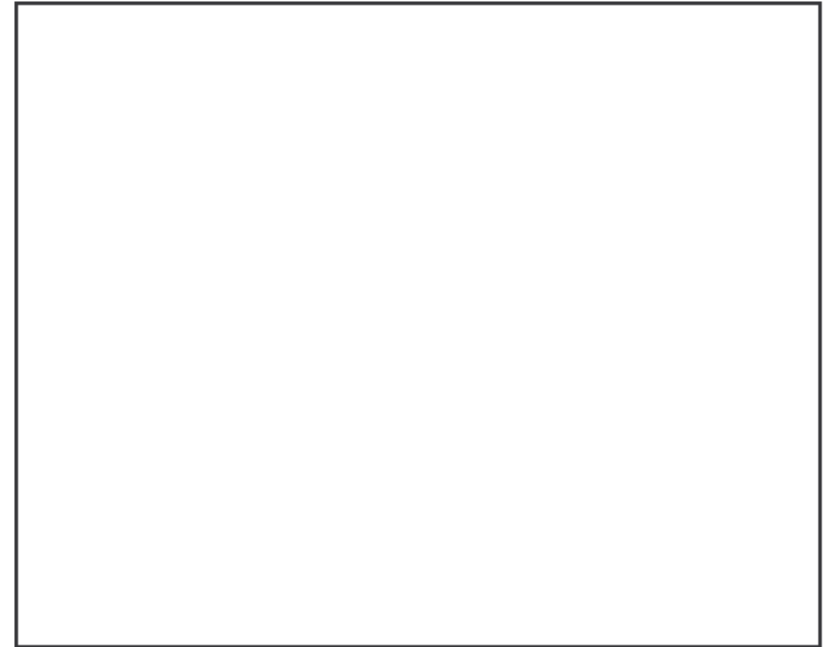
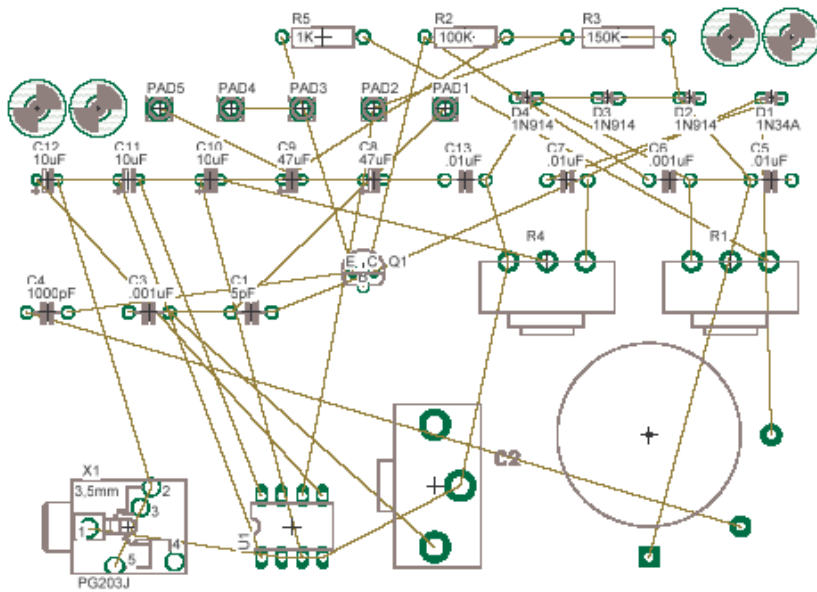


Eagle in a Nutshell

Left Toolbar Tour – Part 2



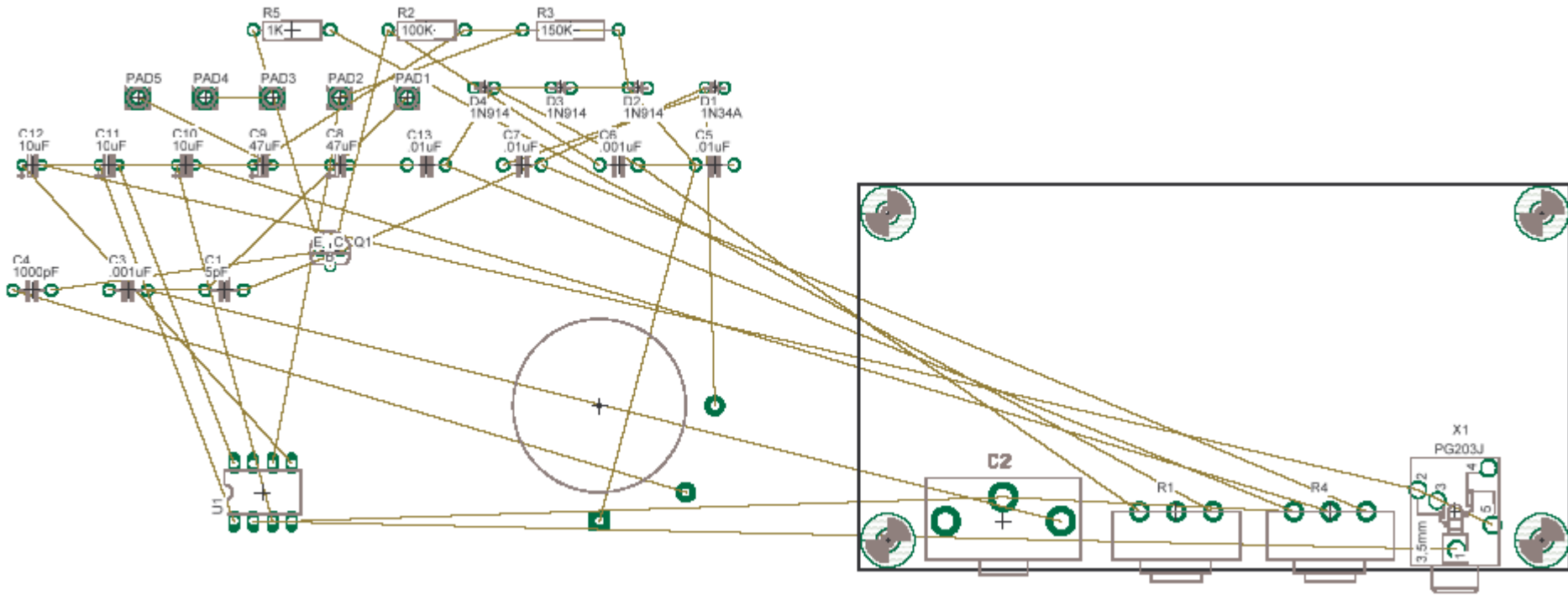
PC Board Layout



PC Board Layout

- Place the components on the PC Board
 - Use the Move tool to resize the board outline to 3.70” x 2.00”
 - Place the mounting holes at the corners
 - Move the controls to the front edge

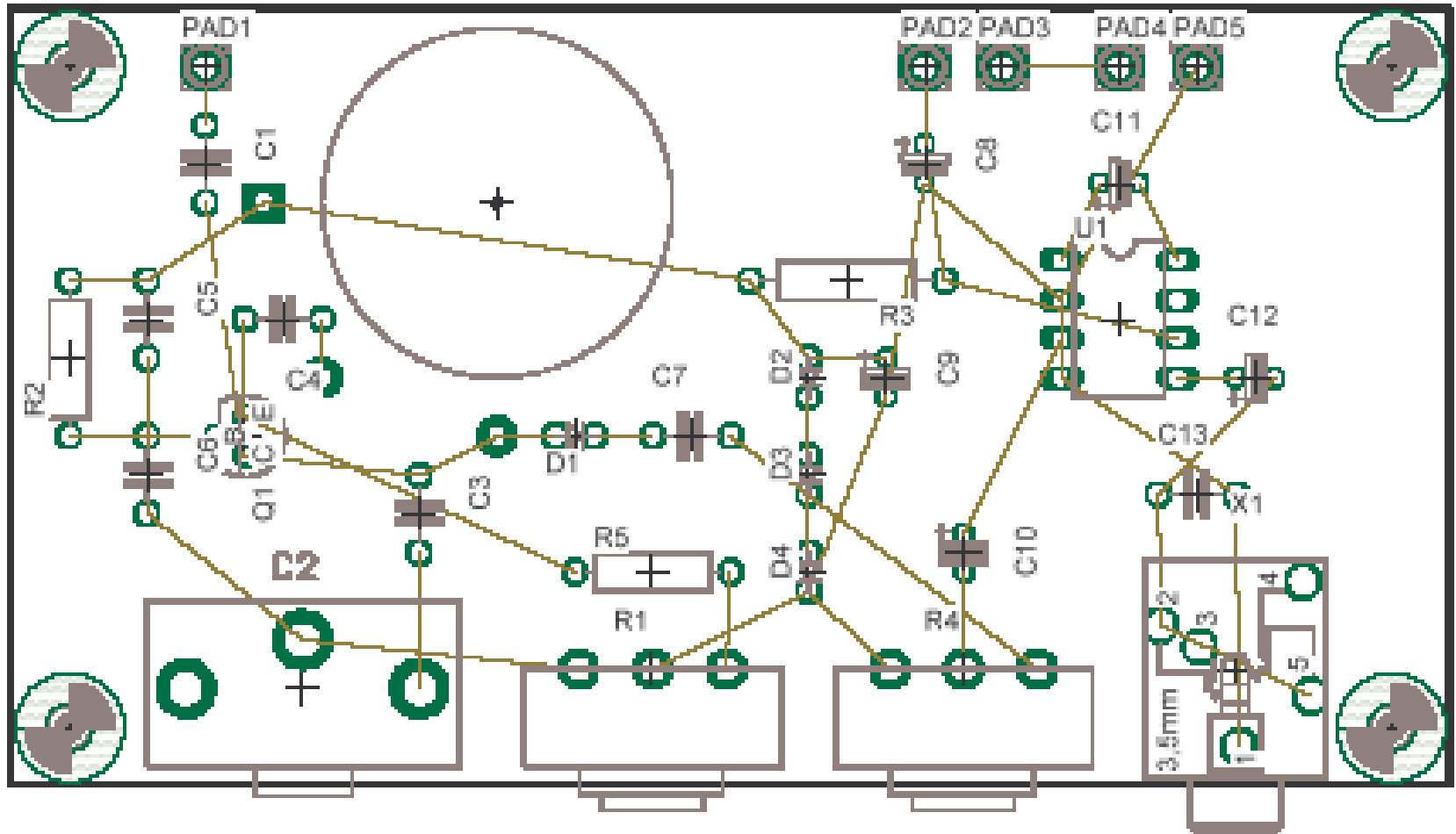
PC Board Layout



PC Board Layout

- Continue to place components with an eye toward minimizing trace lengths
- Use the Ripup tool to clean up the rat's nest.
- “Smash” a component to allow movement of the reference designator and value independent of the component placement. This works in schematic capture too
- Use the Layer Display command to turn off component values (Layer 27) for legibility

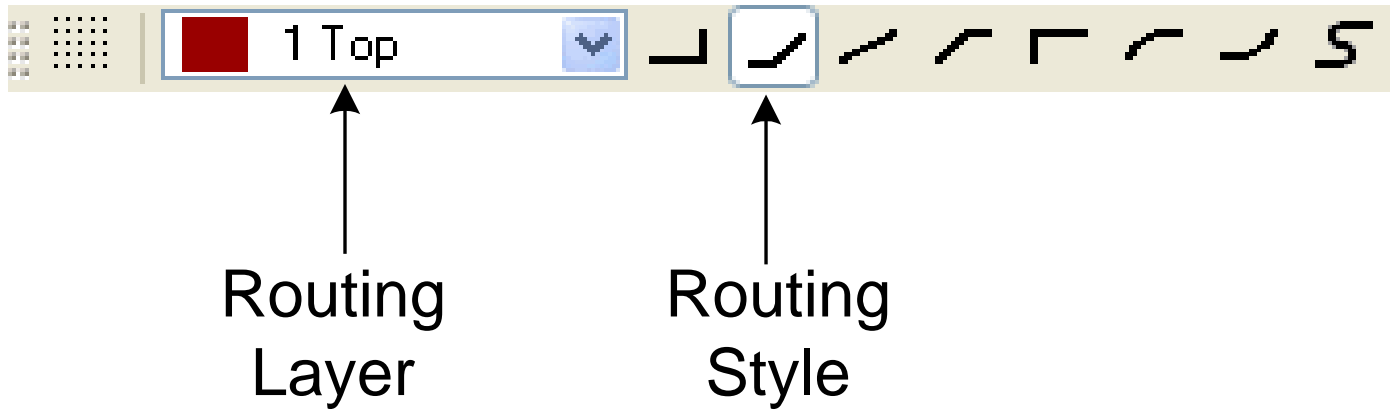
PC Board Layout



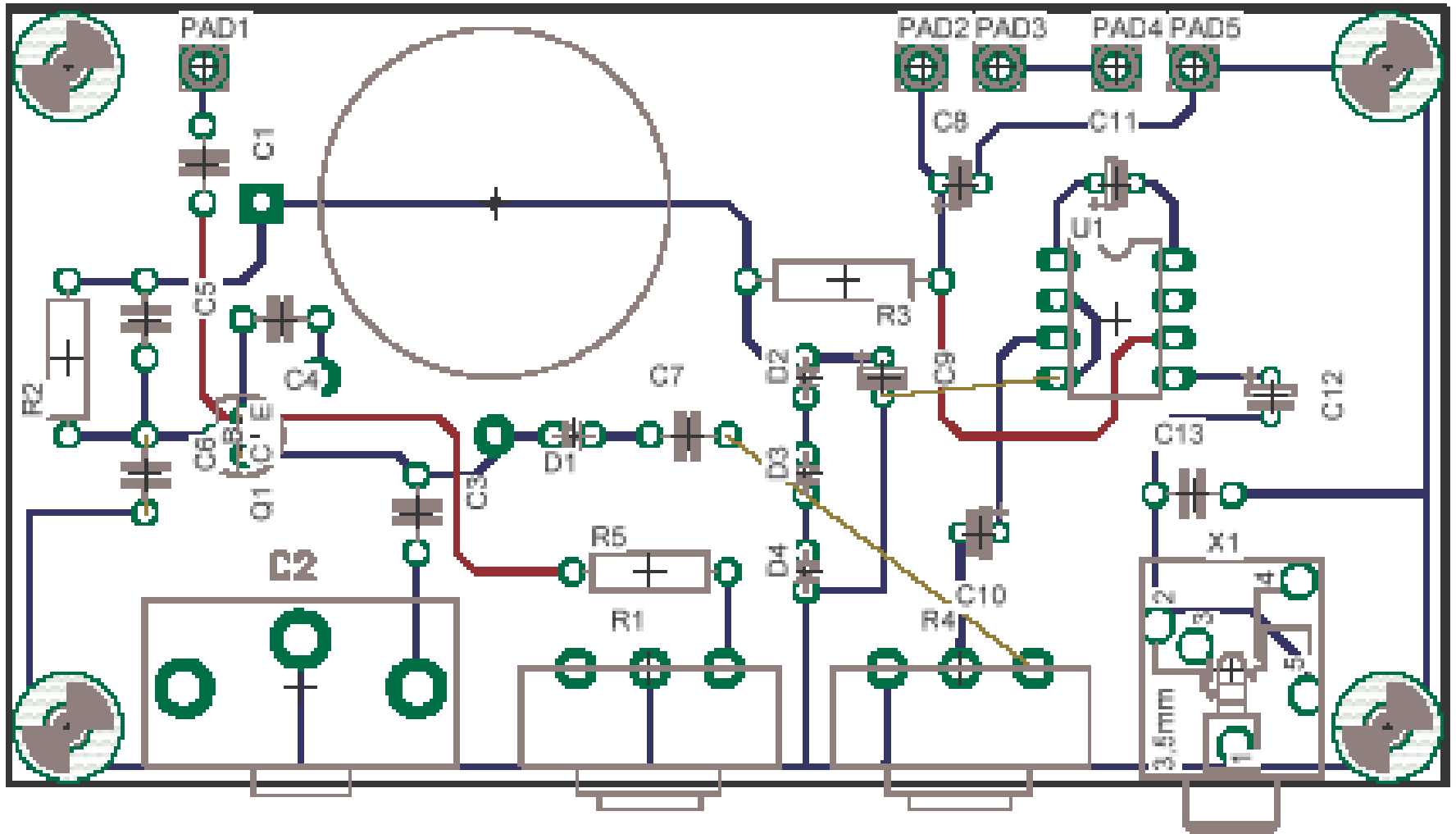
PC Board Layout

- Now start routing the traces
- Click on the route tool
- Select the Routing Layer, top or bottom
- Select the Routing Style
- Click on a component pad and move cursor to draw a trace.
- Routing hint: Start with the grounds!
- Computer emits a “ding” when a connection is completed

PC Board Layout



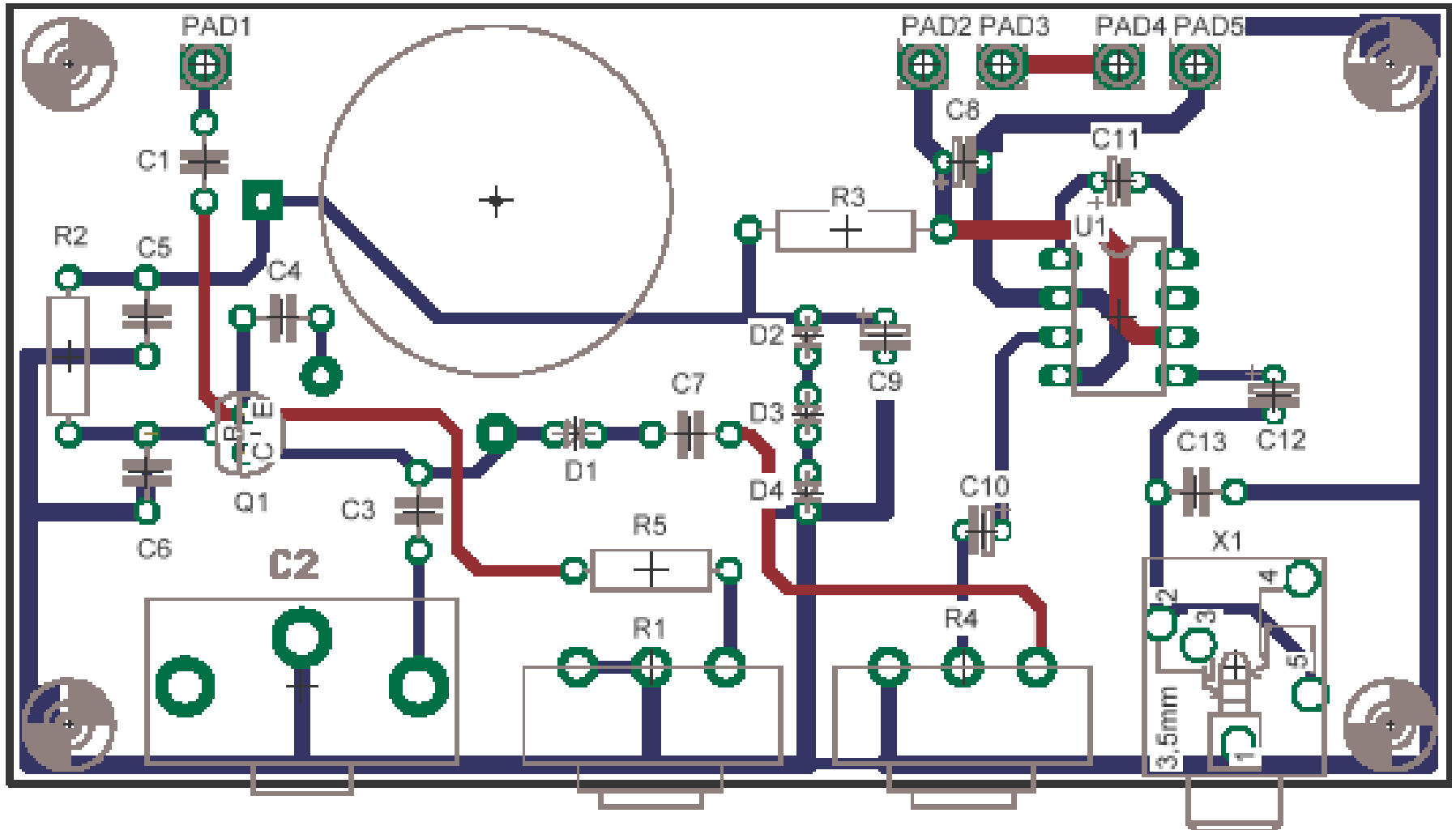
PC Board Layout



PC Board Layout

- Review unrouted traces
- Adjust trace positions, move components and increase trace width.
- Use the Change Tool to select .024 trace widths and click on the signal traces
- Select the .040 trace width for power and ground and change the widths

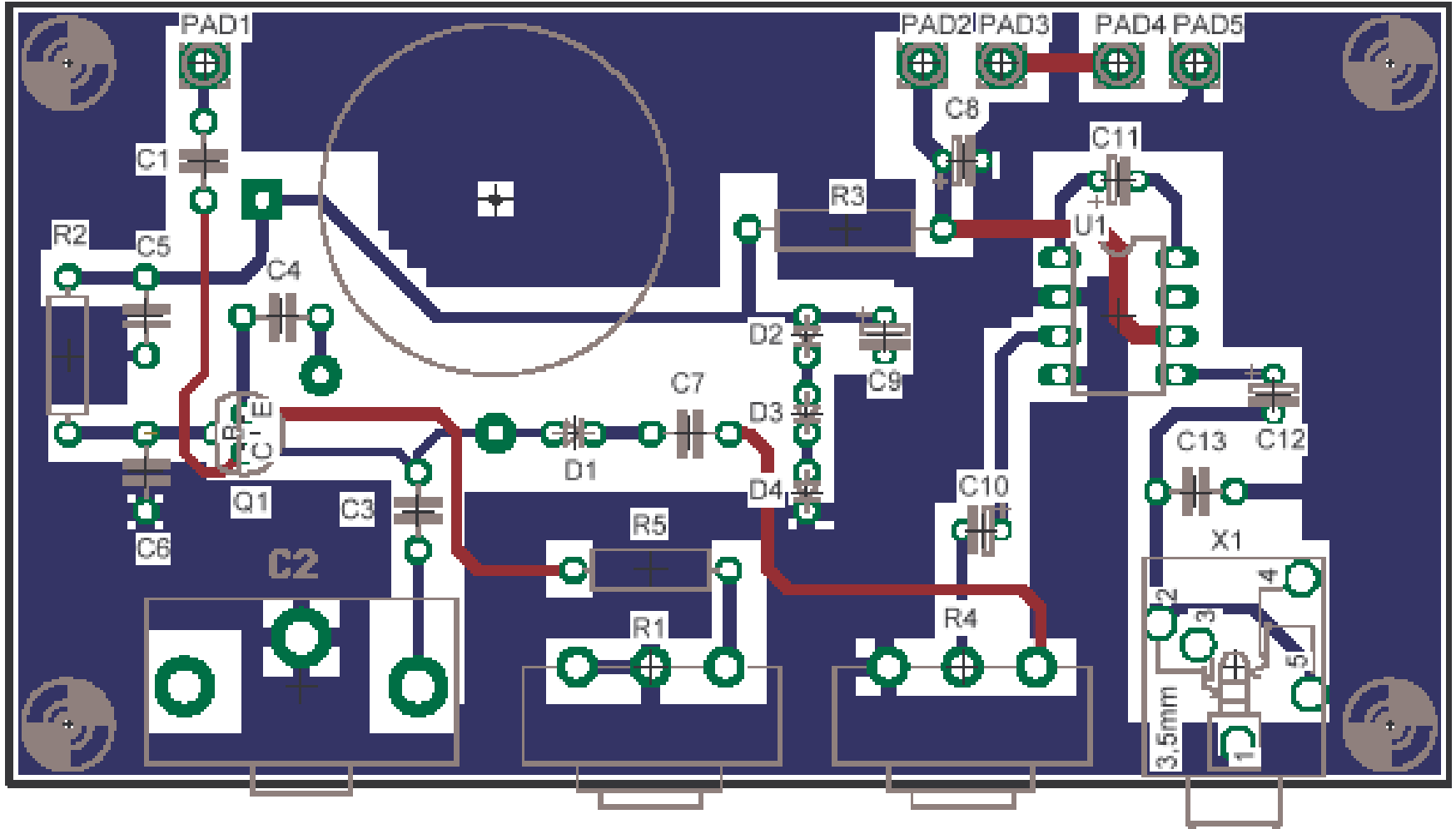
PC Board Layout



PC Board Layout

- The board will probably work as-is, but a solid ground plane reduces trace inductance.
- Let's paint in a ground plane on the bottom of the board using the Rectangle tool
- Connect pads to the ground plane using the “/” tool

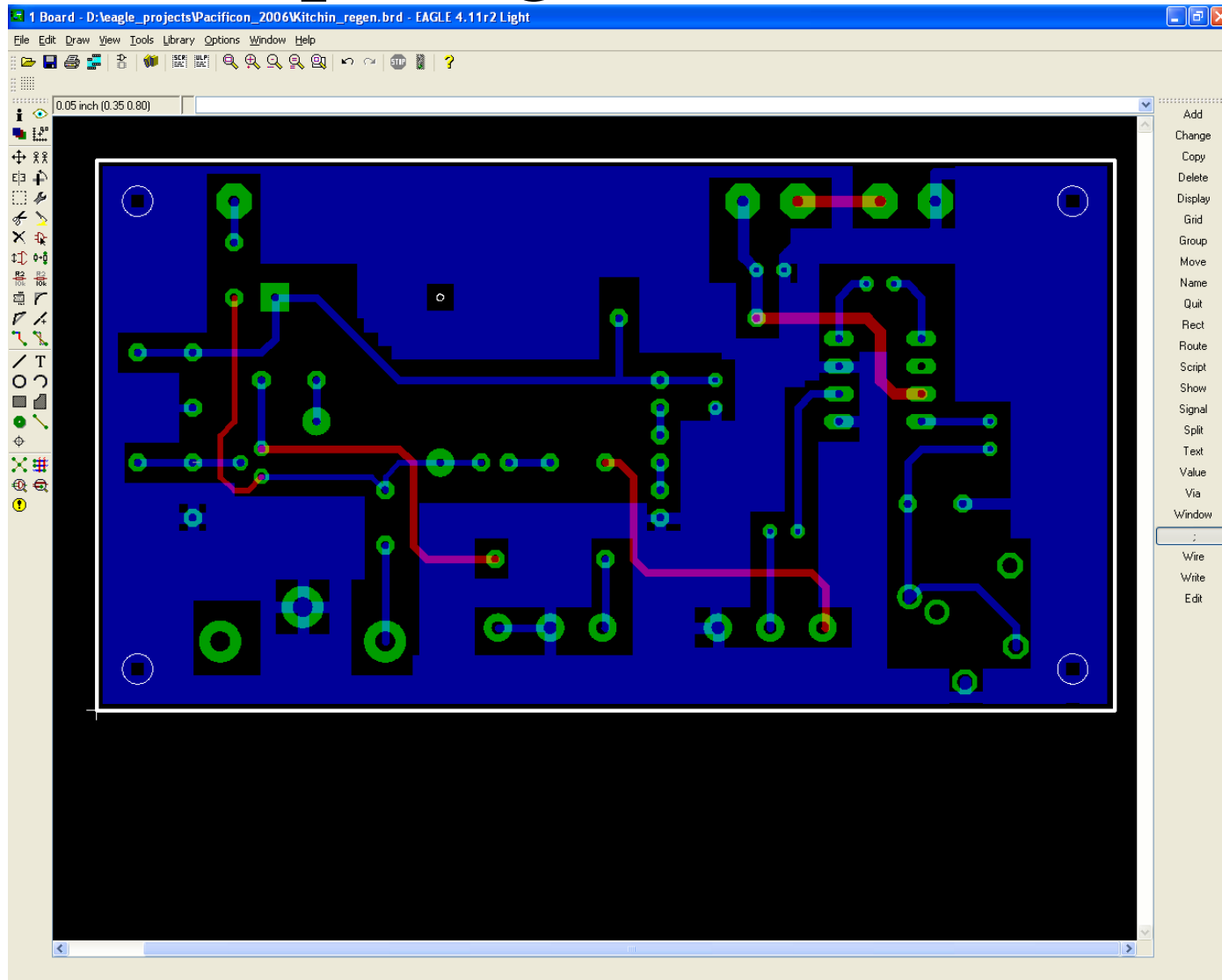
PC Board Layout



Preparing the Artwork

- Once the layout is complete, it's time to print the transparencies
- The top (component) side is printed normal
- The bottom (solder) side is printed as a mirror image
- Using the Layer Display tool, disable Layers 21, 23, 25, 48 & 51

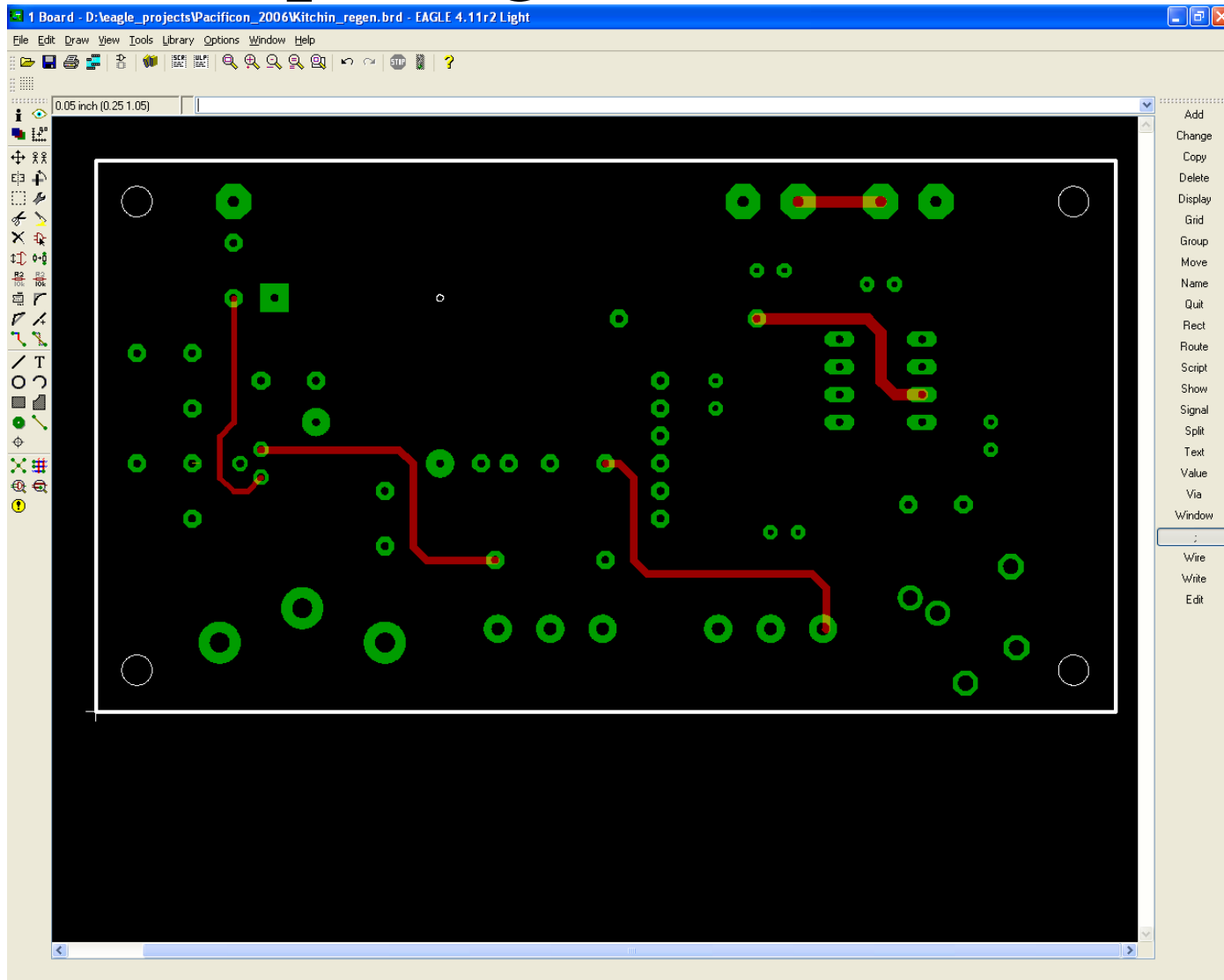
Preparing the Artwork



Preparing the Artwork

- Now turn off display of Layer 16, the bottom copper layer

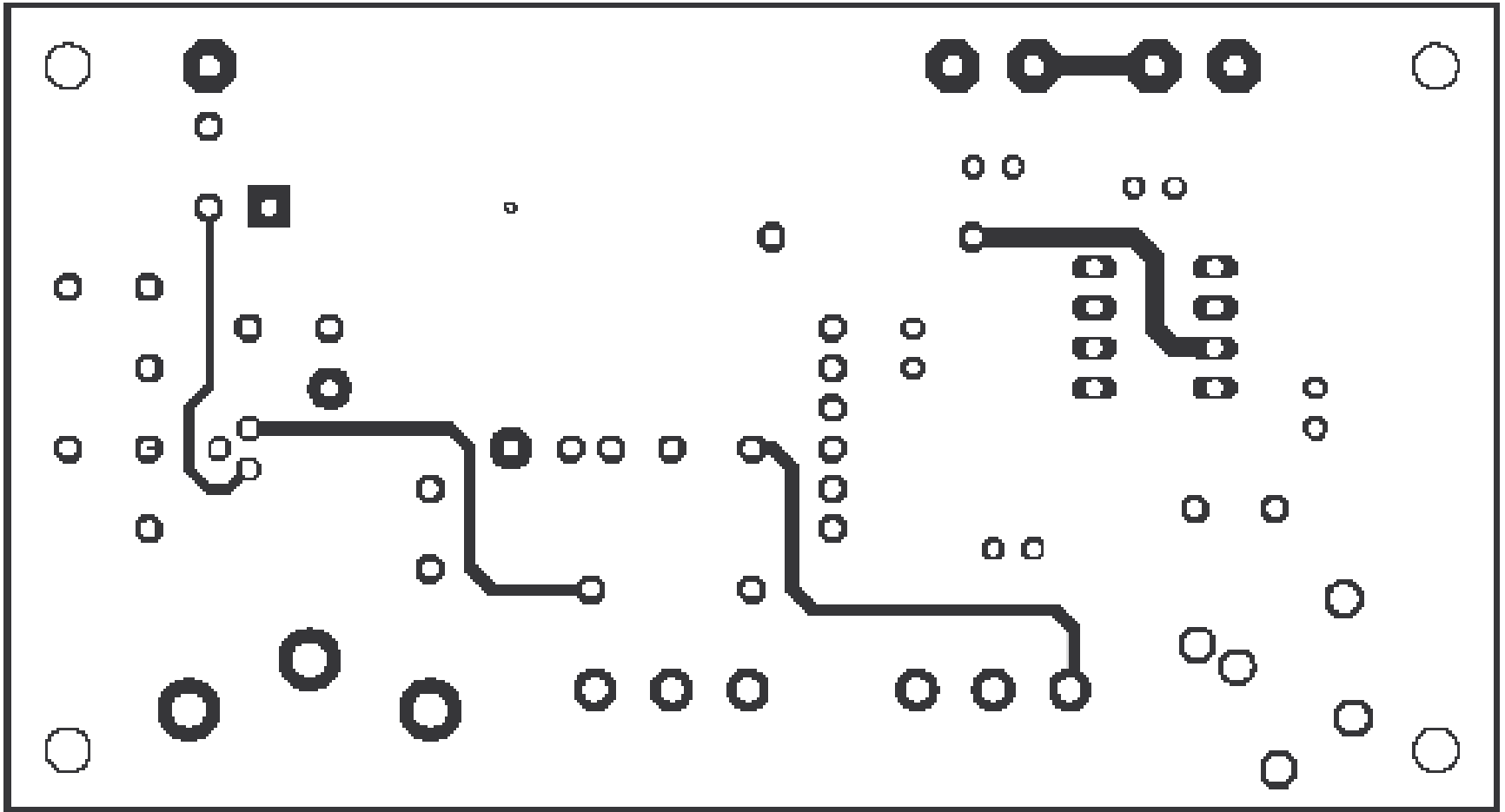
Preparing the Artwork



Preparing the Artwork

- Print the top copper layer
 - Use 1:1 print ratio
 - Select Black color
 - Select the highest resolution on your printer – use “Best Photo” if available
 - Print on the appropriate transparency

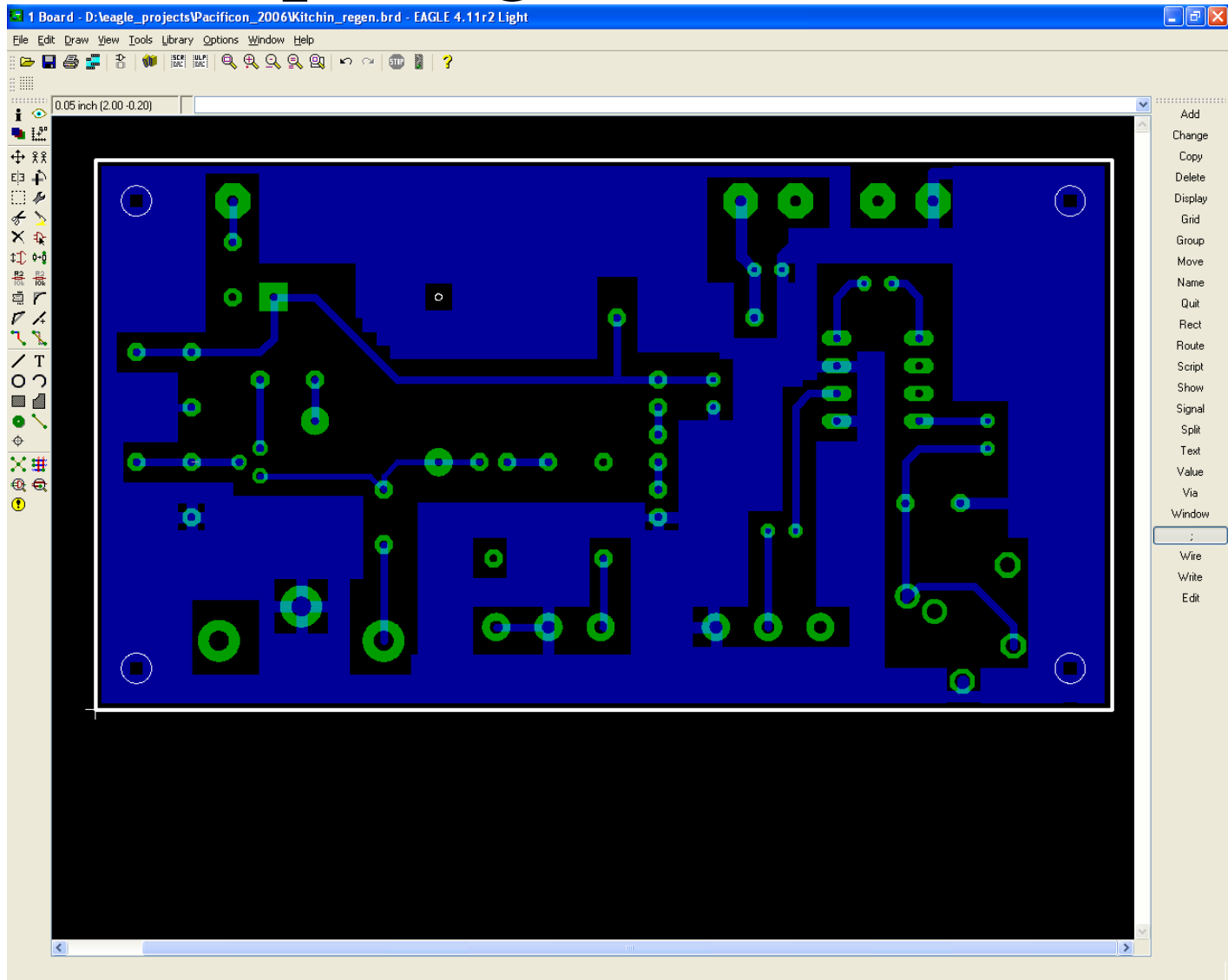
Preparing the Artwork



Preparing the Artwork

- Turn off the top layer
- Turn on the bottom layer

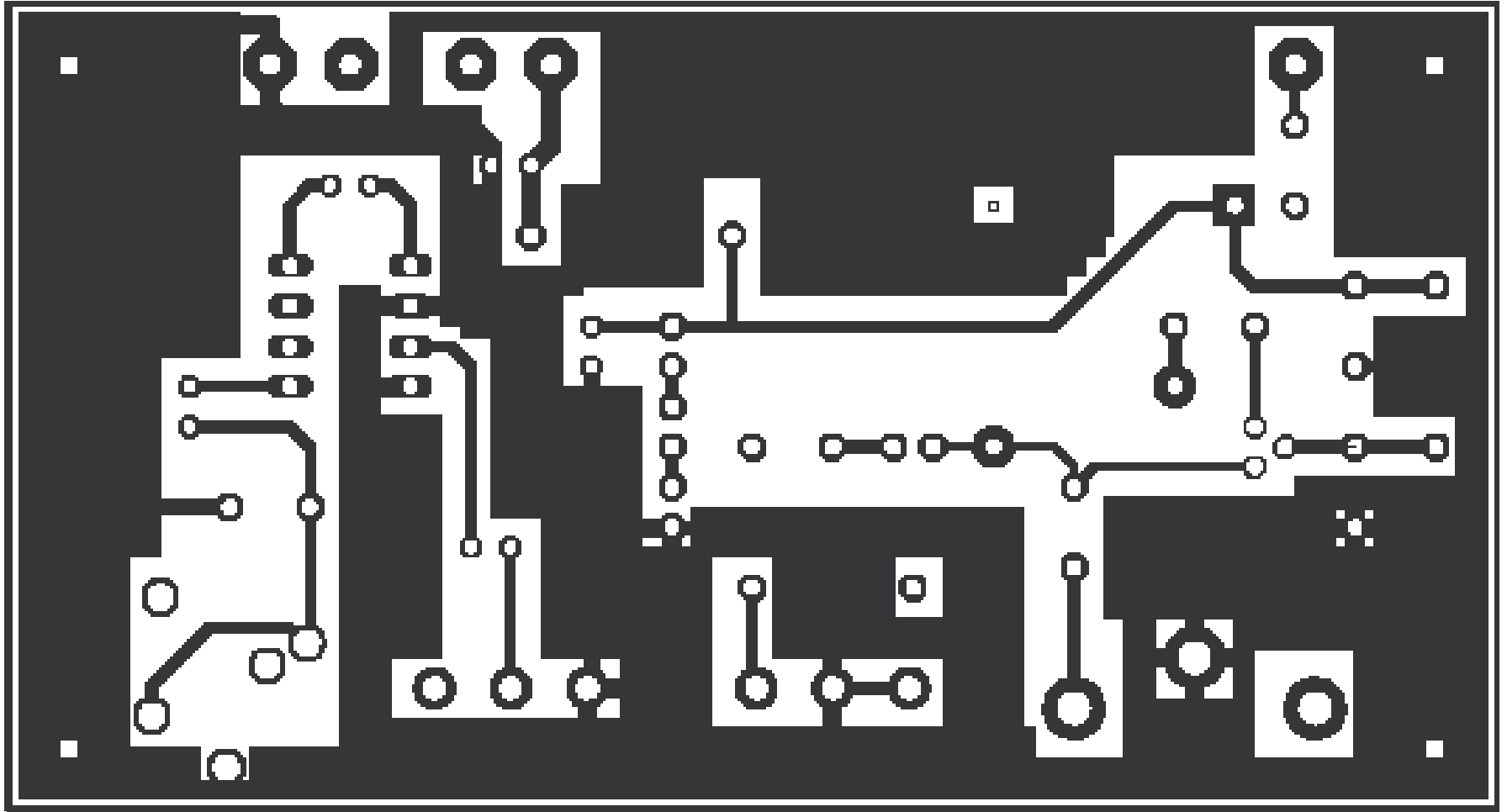
Preparing the Artwork



Preparing the Artwork

- Print the bottom copper layer
 - Use 1:1 print ratio
 - Select Black color
 - Select Mirror Image

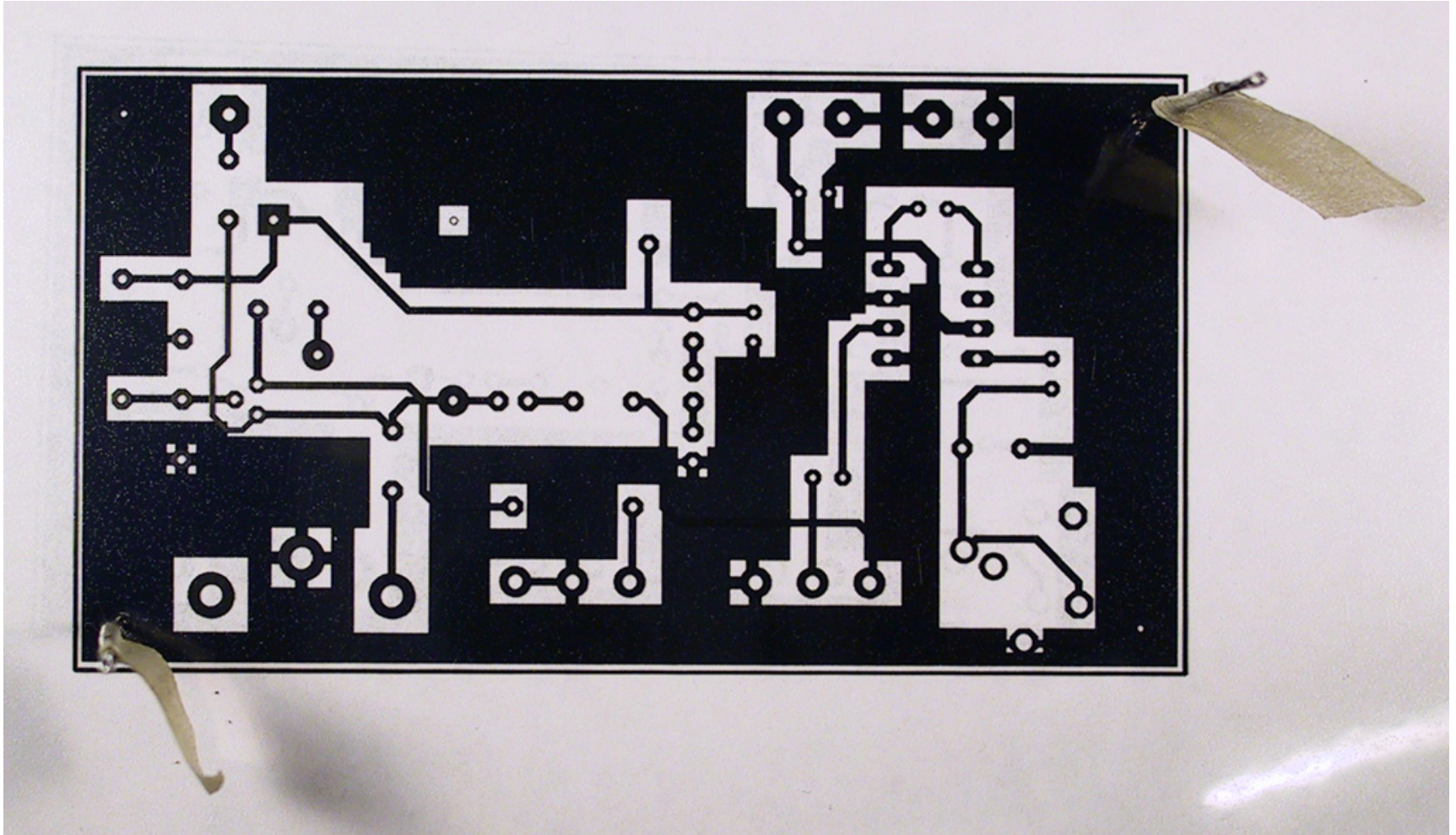
Preparing the Artwork



Preparing the Artwork

- Once the transparencies are printed, invert the bottom layer and place the top transparency on top
- Align the layers and lock the layout by pressing sewing needles into diagonally opposite mounting holes. This establishes layer-layer registration

Preparing the Artwork



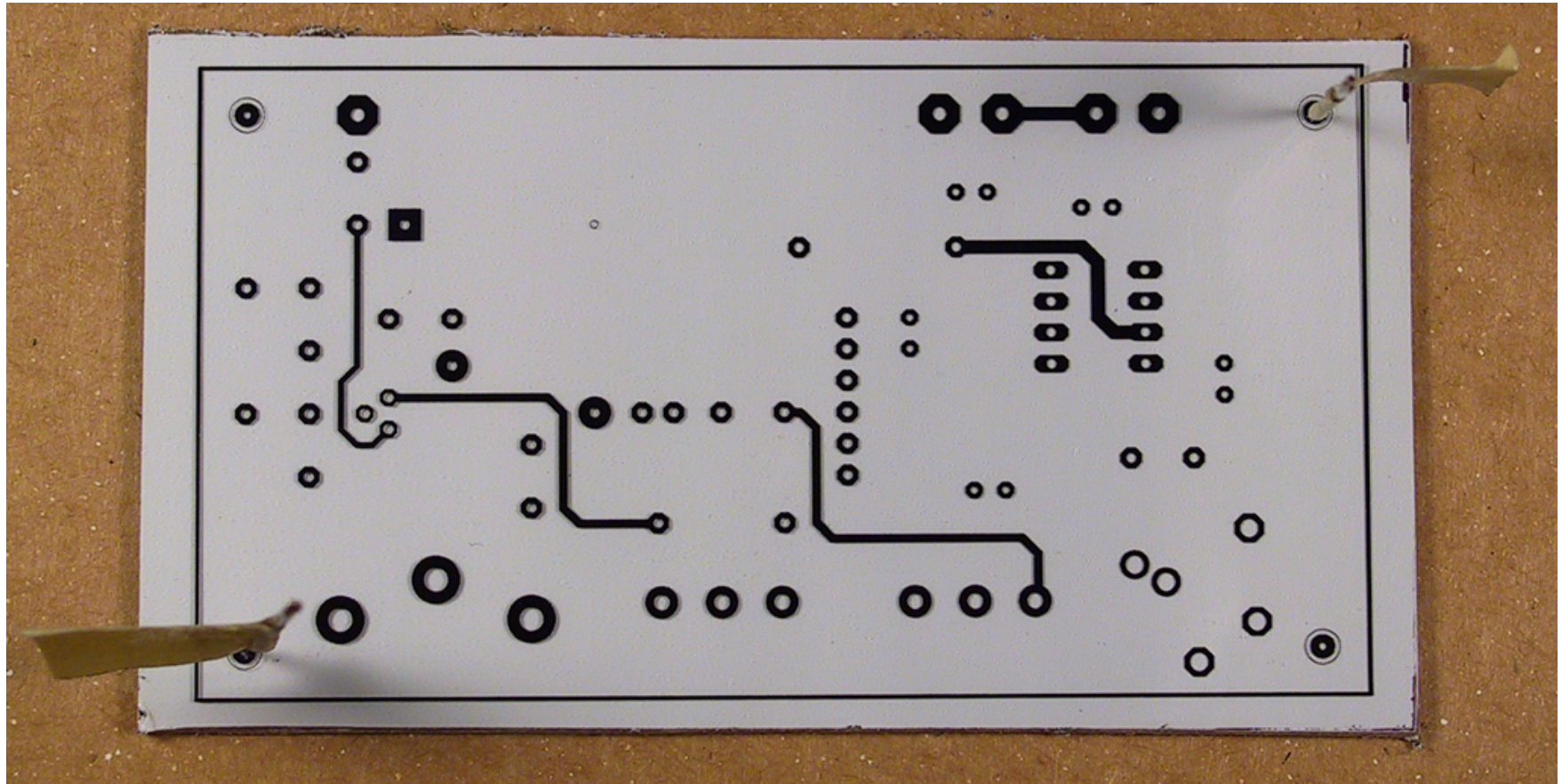
PC Board Fabrication

- Materials and Tools
 - MG Chemicals 600-series presensitized double-sided board
 - MG Chemicals 418-500ml Developer
 - Radio Shack Cat No. 276-1535 etchant
 - Masking tape
 - 15W Blacklight
 - 6 inch ruler
 - Glass plates & weights
 - Timer
 - Plastic/glass etching tank or zip-lock plastic bags
 - Rubber gloves
 - Hacksaw, files & dust mask
 - Carbide PC drill bits & drill press

PC Board Fabrication

- Cut board to rough dimensions & deburr the edges with a file. Leave approx. 1/8" around all edges
- Mark the registration holes on the board using sewing needles. Tap the pins with a small hammer

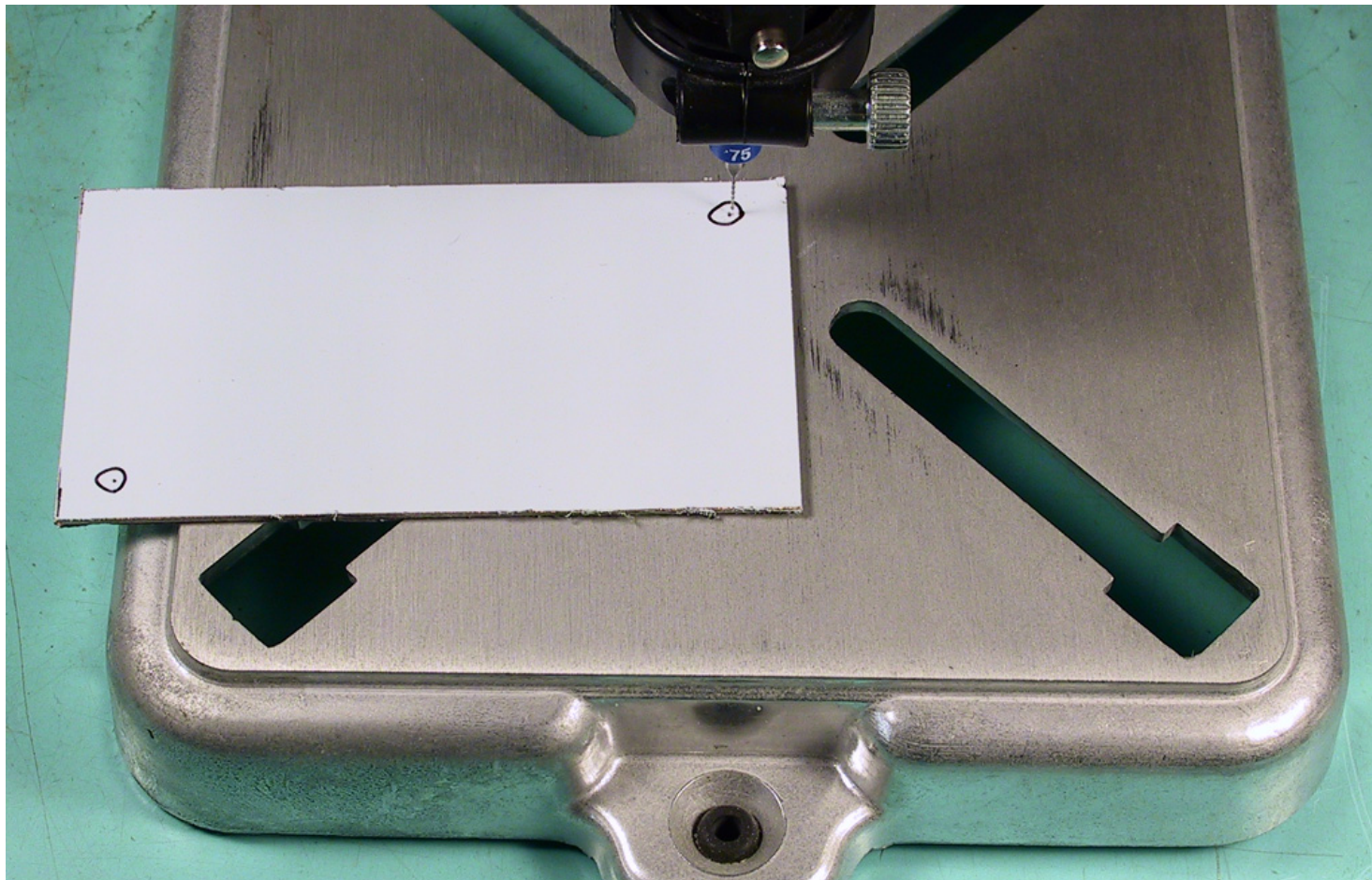
PC Board Fabrication



PC Board Fabrication

- Next, drill the registration holes using a small bit, approx. .020”

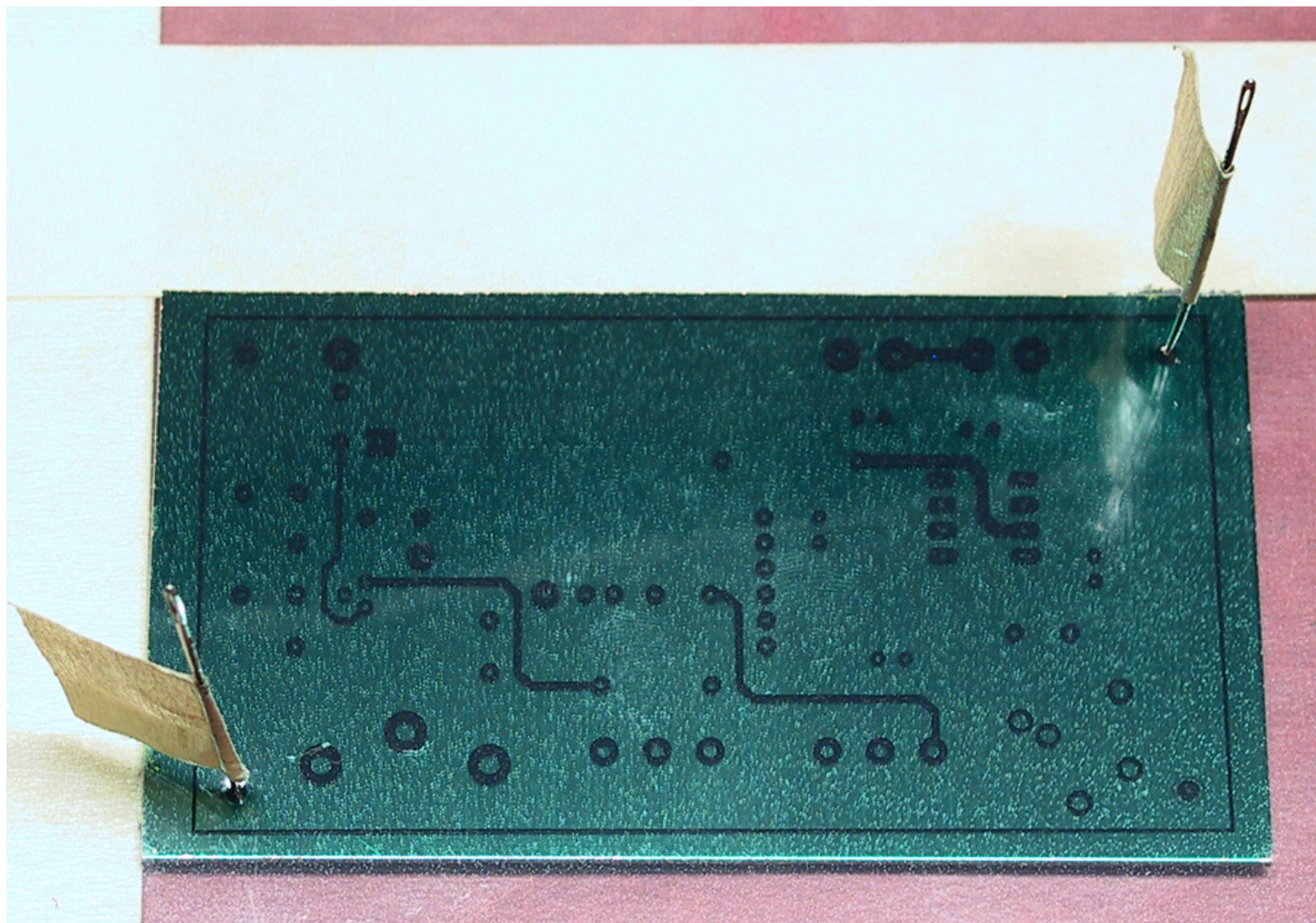
PC Board Fabrication



PC Board Fabrication

- Apply masking tape at right angles on the bottom glass to provide stop edges
- Remove the protective film and save it.
- Place board against the masking tape corner
- Lay top transparency on the board and lock its position using sewing needles in the drilled holes
- Tape the transparency to the bottom glass

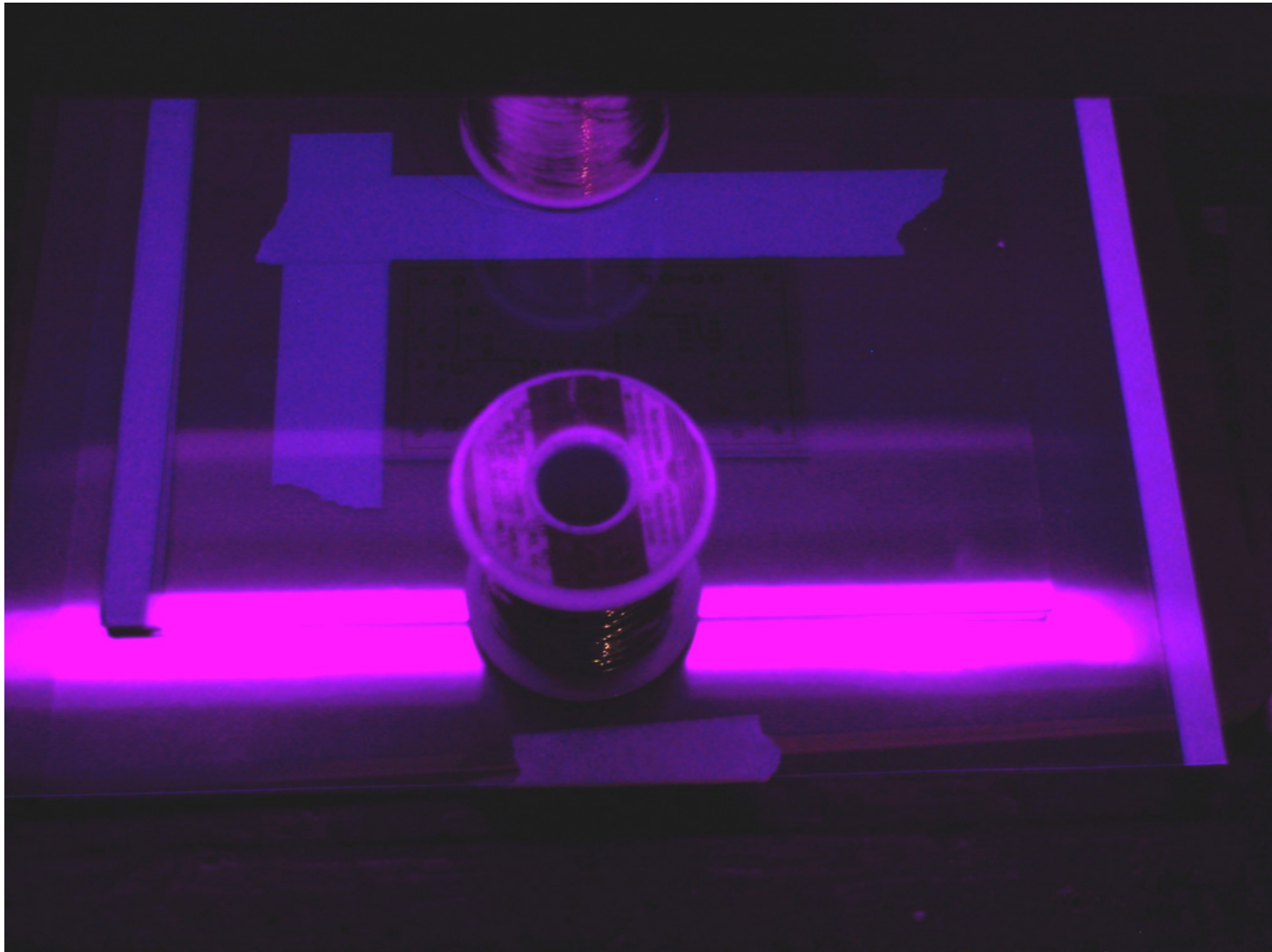
PC Board Fabrication



PC Board Fabrication

- Remove the needles
- Put top glass over transparency and weigh it down
- Set the blacklight 6 inches above the top glass and parallel to the board
- Turn on the blacklight and set the timer for 8 minutes, 30 seconds +/- . This is where you fine-tune your process

PC Board Fabrication



October 14, 2006

W3CD

67

PC Board Fabrication

- After the top side is exposed, cover it with the saved film. Mark the bottom edge to avoid flipped board mistakes
- Turn the board over, remove the protective film and place against the masking tape.
- Repeat transparency registration using sewing needles
- Tape down the bottom transparency, apply top glass and weights
- Expose for the same period
- After exposing both sides, it's time to develop the board

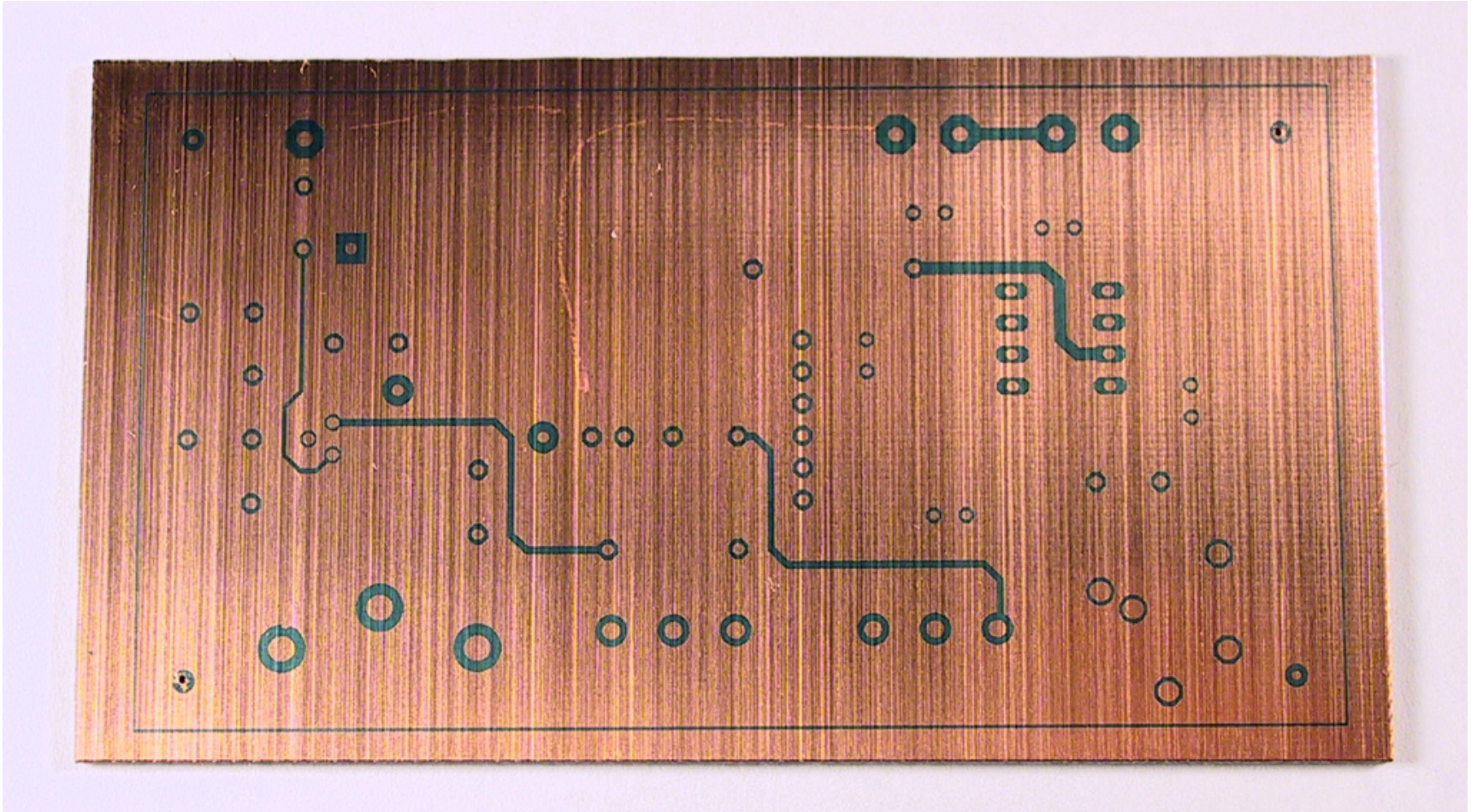
PC Board Fabrication

- Dilute the developer according to instructions, typically 10:1
- Warm solution to about 100F to speed the process
- Develop board in a plastic bag
- Rinse in cold water
- Dispose the developer according to local ordinances

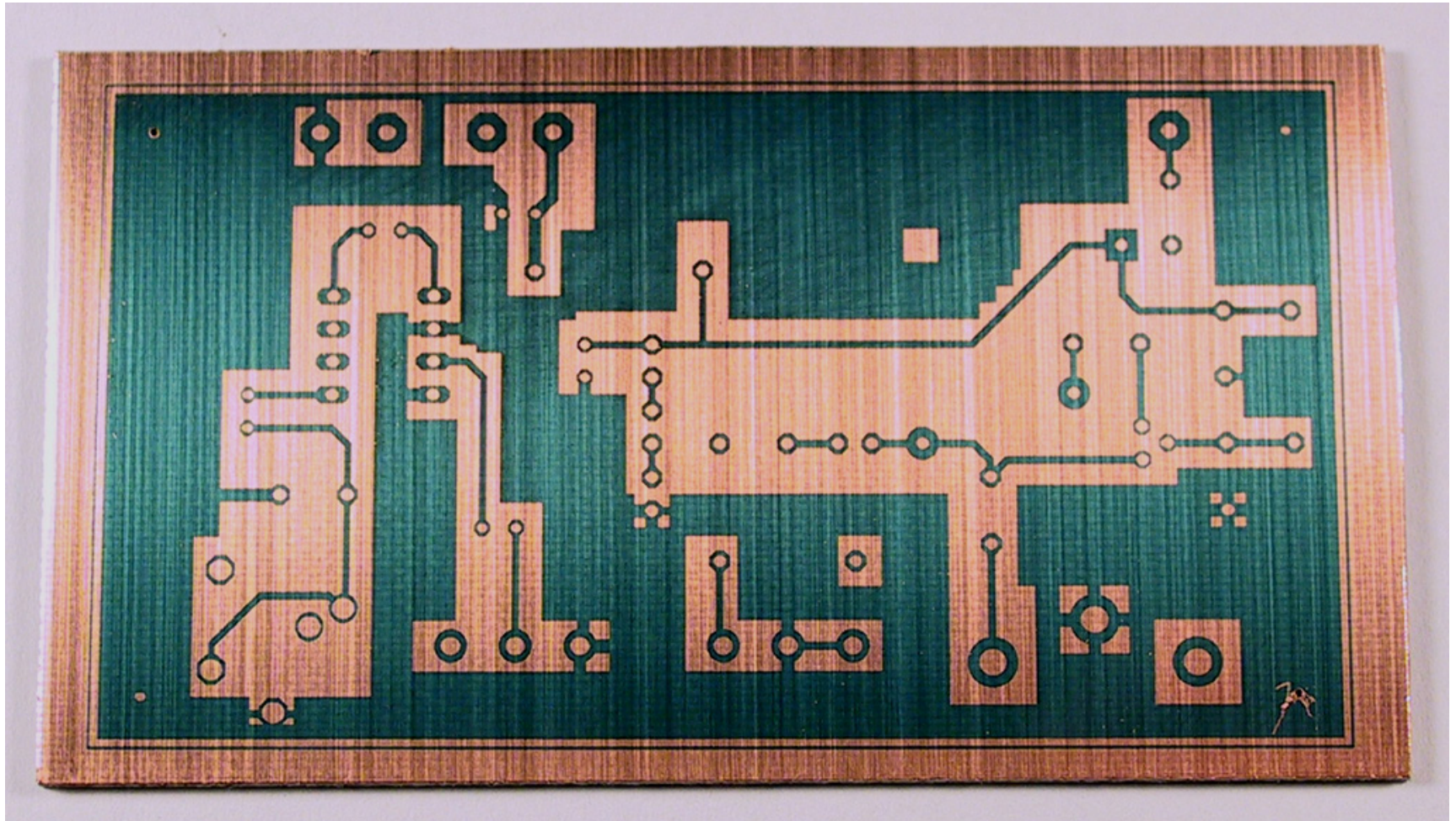
PC Board Fabrication



PC Board Fabrication



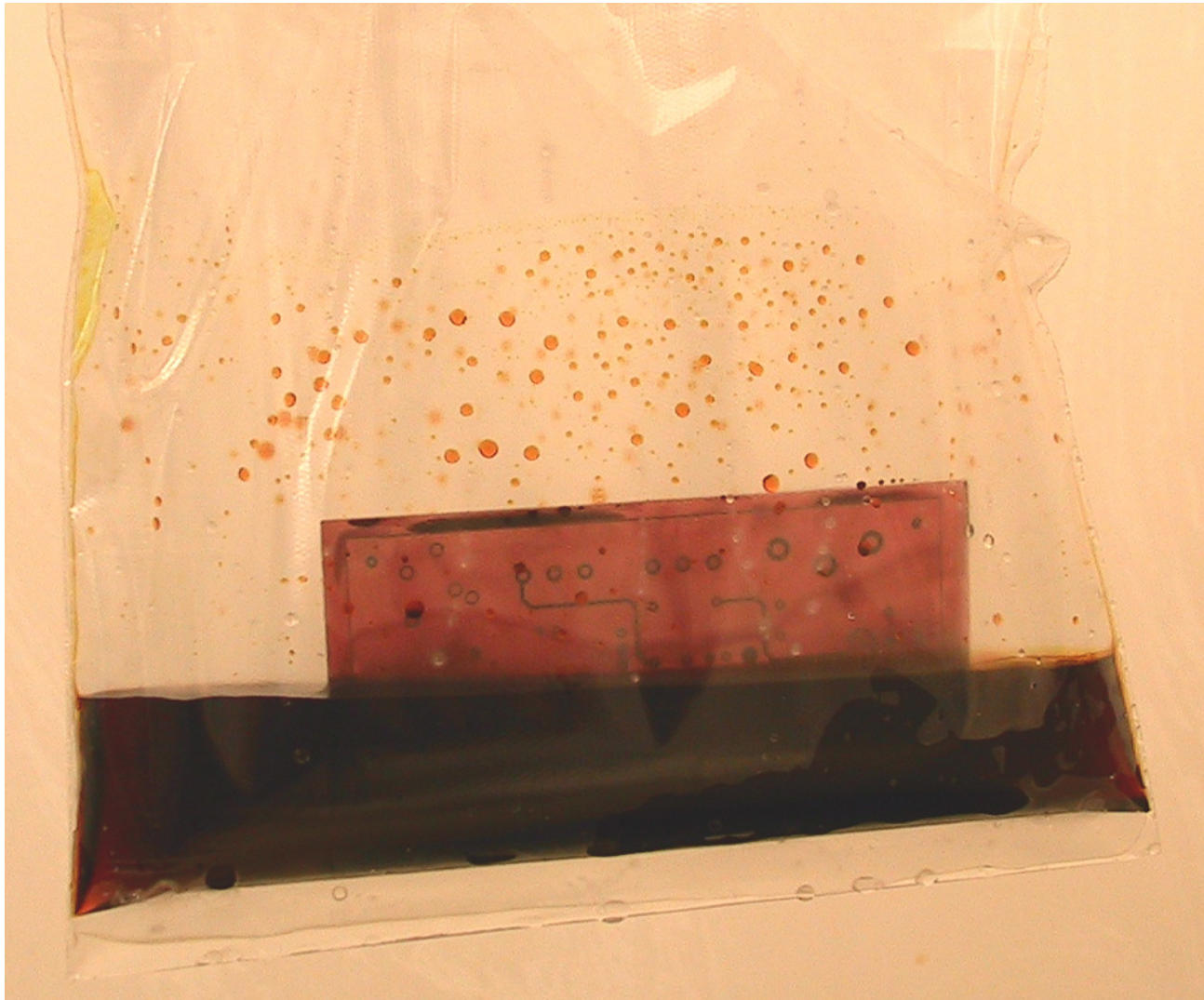
PC Board Fabrication



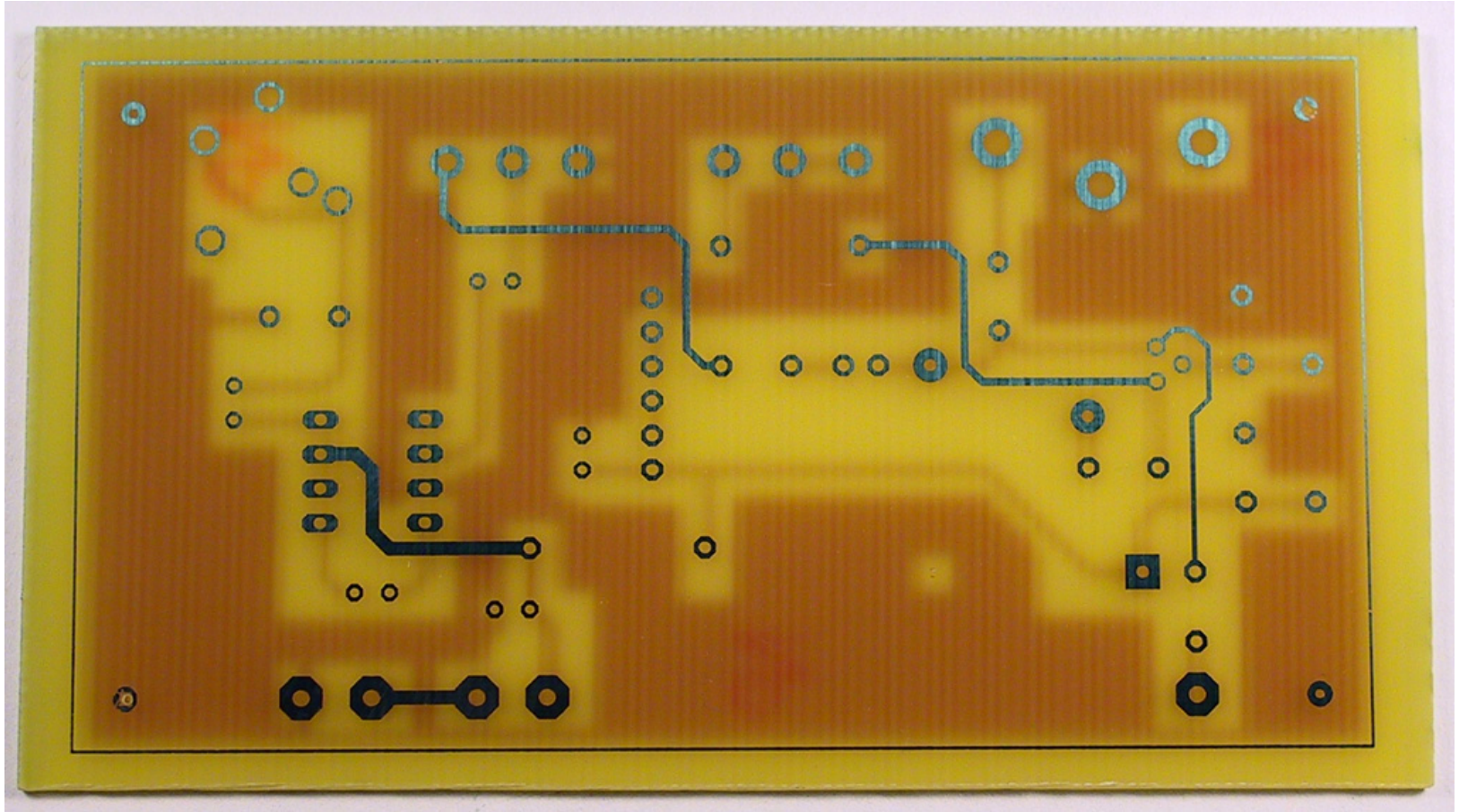
PC Board Fabrication

- It's time to etch the board
- Nest two ziplock bags
- Place exposed board gently in bag
- Add enough etchant to cover half the board
- Close bags and place in 100-110F water bath
- Agitate by hand until copper is etched
- Remove board and rinse well
- Dispose of etchant according to local ordinances

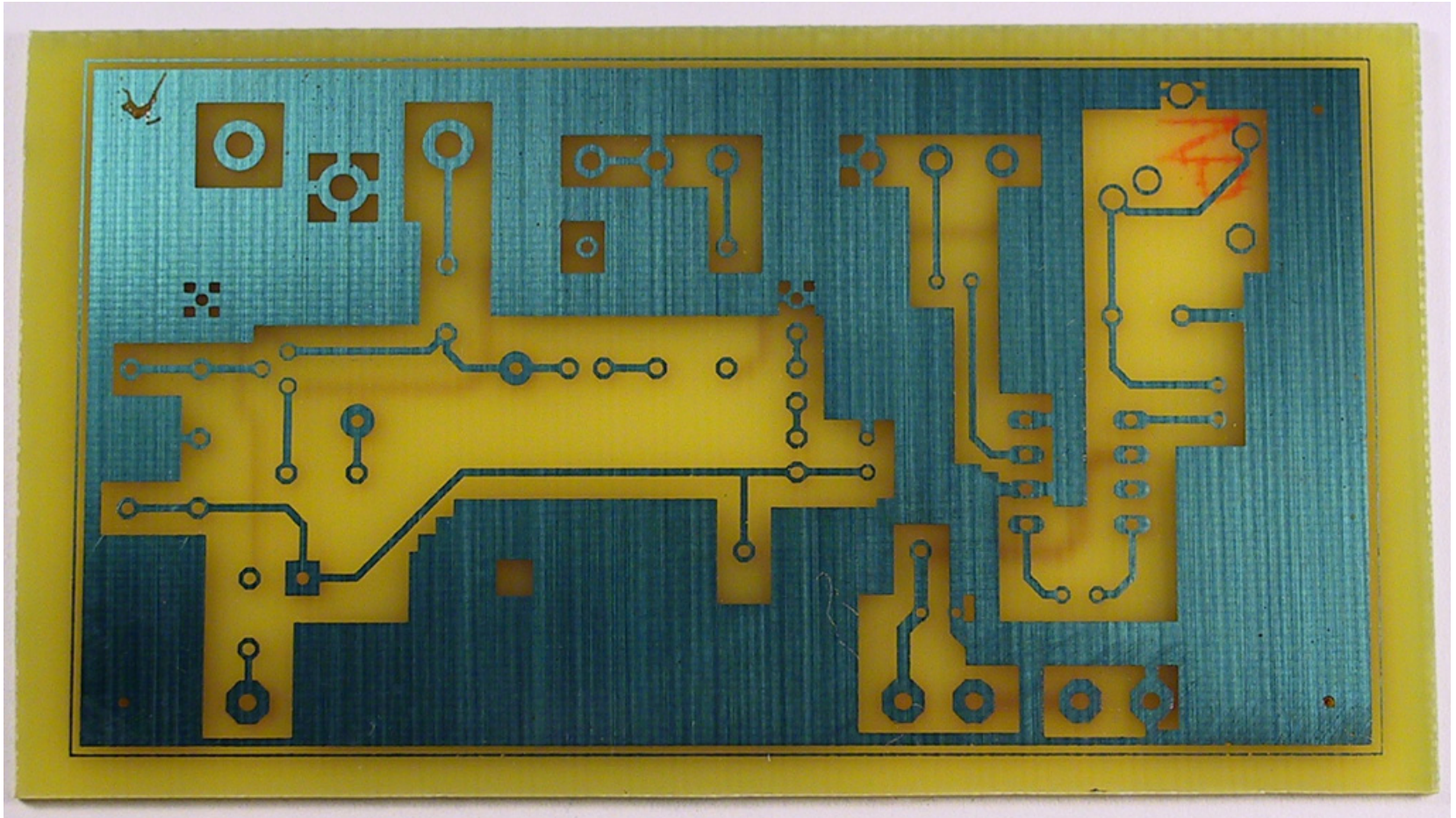
PC Board Fabrication



PC Board Fabrication



PC Board Fabrication



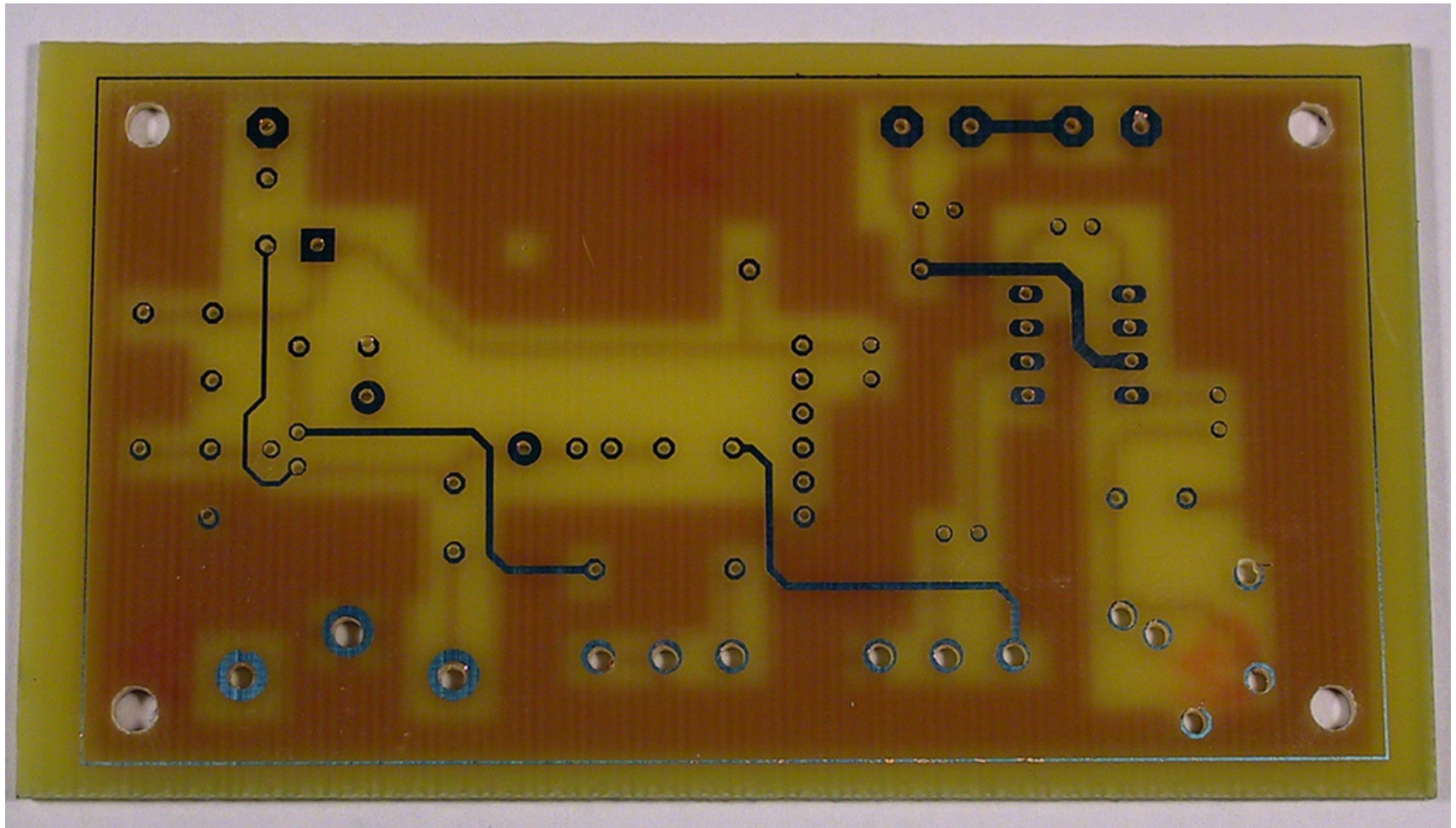
PC Board Fabrication

- After etching, drill and trim the board
- Wear a dust mask!!
- Drill sizes:
 - Mounting holes: .125
 - Headphone jack: .059 - .063
 - Pots: .050 - .055
 - Coil pads: .065
 - Polyvaricon center lead: .070
 - Polyvaricon side leads: .063
 - All others: .031

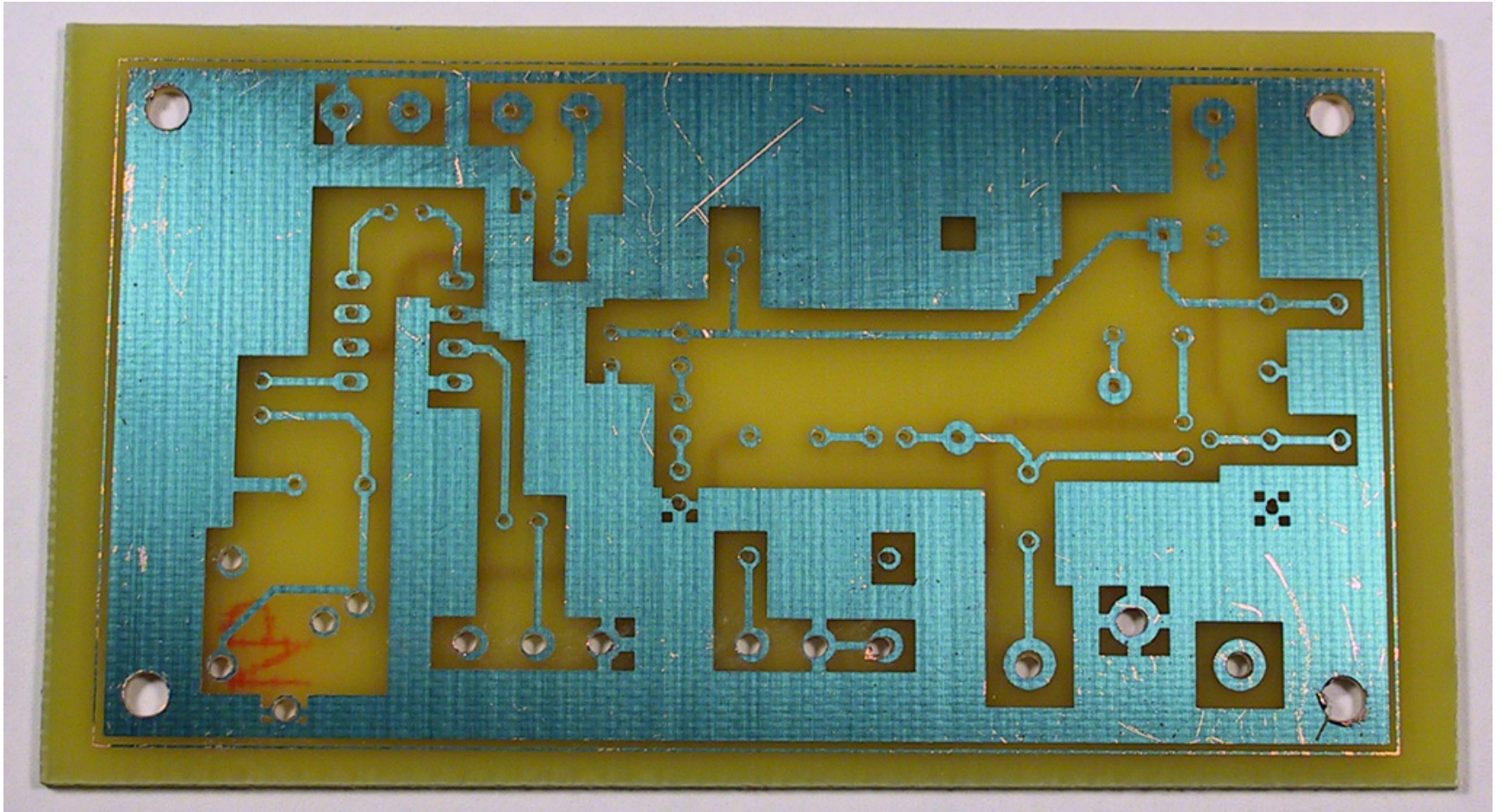
PC Board Fabrication



PC Board Fabrication



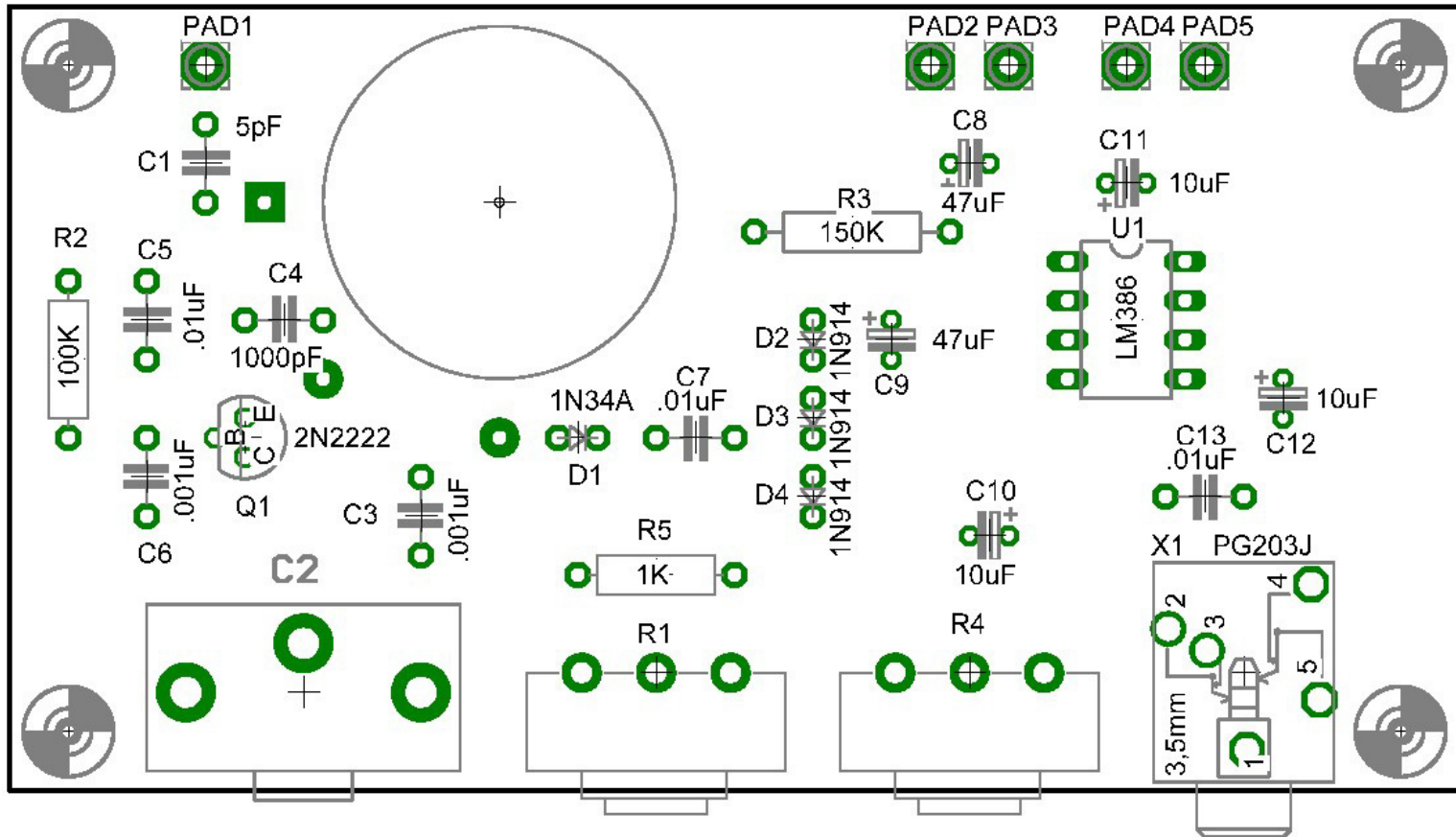
PC Board Fabrication



Assembling the Board

- It's helpful to print a layout of the board
 - Turn off the top and bottom copper layers (1 & 16)
 - Turn on layers 21, 23, 25 & 27
 - The component outlines, reference designators and values will appear

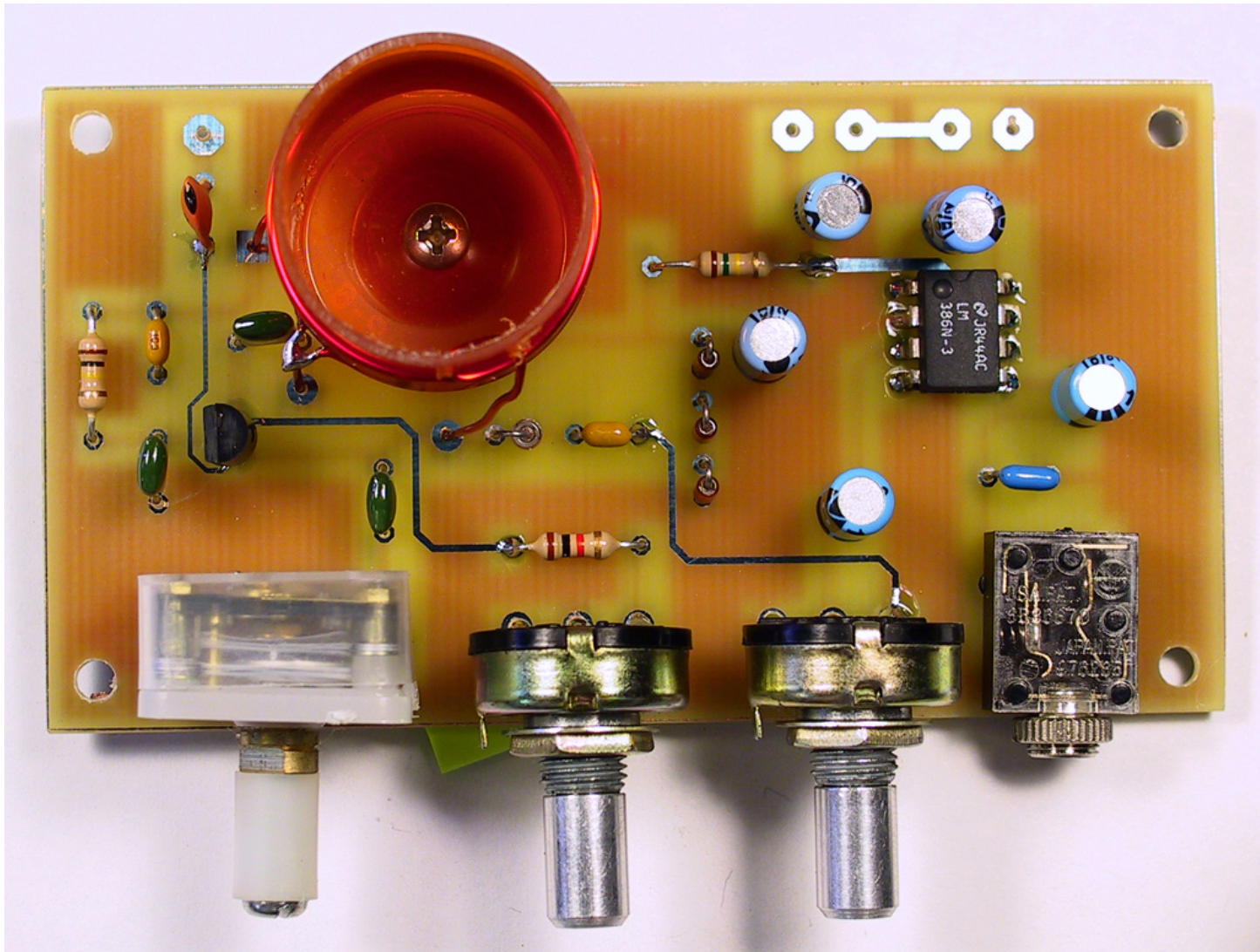
Assembling the Board



Assembling the Board

- Remember the board doesn't have plated holes. Solder both sides of the board where there are traces on the component side

Finished Project!!



October 14, 2006

W3CD

84