HOW TO MAKE DOUBLE-SIDED CIRCUIT BOARDS I
San Francisco State University
School of Engineering
by Larry Klingenberg
April 2006

1. **Download and Install EAGLE**, available free from http://www.cadsoftusa.com/, which makes circuit boards
    a. The following files should be downloaded
        i. [eagle-4.xx.exe] Circuit Board Program
        ii. [manual-eng.pdf] Documentation
        iii. [tutorial-eng.pdf] Documentation
    b. The following restrictions apply to the free Light Edition of Eagle
        i. Board area is restricted to 100 x 80 mm (about 3.9 x 3.2 inches)
        ii. Only two signal layers can be used
        iii. A schematic can consist of only one single sheet
2. **Download and Install ViewMate**, available free from http://www.pentalogix.com/Download/download.html, which allows viewing of circuit board Gerber files. Gerber files are files that are sent to the Board house to manufacture the circuit boards. The board displayed on the computer screen by Eagle is not necessarily what you get. The Gerber file viewer will show the manufactured board.
3. **Open New Project**
    a. Start Eagle software. In Control Panel window, go to *File* => *New* => *Project*
    b. The name of the project will default to New_Project_x, where x is the number of the project.
4. **Create a Schematic**
    a. In Control Panel, highlight New_Project_x and go to *File* => *New* => *Schematic*
    b. Save the schematic, *File* => *Save*, for the first time and give it a name
    c. Select the Control Panel window and notice that the new schematic is now under the project folder
5. **Set the Grid**
    a. In the schematic window, go to View => Grid
    b. Select
        i. *Display* => *On*; *Style* => *Dots*;
        ii. *Size* => 0.05 in.; *Multiple* => 1; *Alt* => .025 in.
    c. Parts will now snap to the grid of .05 inch. However, in the event that a smaller movement is needed for part placement, depressing the Alt key will allow movements of .025 inch.
6. **Make Available the Libraries**
    a. In command line at top of schematic page, type: *use* * <enter>
    b. This command makes available for use all part libraries

7. Change Text Font Always to Vector Type

a. Always use Vector type font for text. The Vector type font is the only one that reproduces on the manufactured board according to what you see on the computer screen. The Gerber viewer program called ViewMate is designed to view the manufactured board according to the files generated by Eagle.

b. In the schematic page, go to *Options => User interface =>* check *Always vector font*.

8. **Adding a Frame in the Schematic Window; Grouping and Moving**

a. A Frame is a page size outline with a document information box in the corner.

b. In the schematic window, *Edit => Add*; in the search window at the bottom, type *frame*; select *Letter_L*; then OK.

c. After placing Frame into schematic, press <Esc> <Esc> to exit the Add tool.

d. To move the frame, select the Move tool; *Edit => Move*. Then select the Frame and move cursor. The Document Information box in the Frame will need to be moved also. Move frame so that lower left hand corner is placed on the large cross, which is the coordinate 0,0.

e. Use *View => Zoom to Fit* to center the view on the screen.

f. As an alternative to moving the frame and document information box separately, they can be "grouped" together and moved as a unit. To move the Frame and the Document Information box together, group them.

i. *Edit => Group* to select the grouping tool.

ii. Using cursor, drag a box around the Frame and box to select them as a group

iii. Select the Move tool. Notice the instruction in the lower left-hand corner of the window. *"Left-click to select object to move (right-click to move group"*. Right-click to move the group.

9. Drawing the Schematic -- Adding Parts

a. Assuming that we are designing a 555 timer in astable mode to blink an LED. The circuit requires 1-555 timer, 3 resistors, 2 capacitors, and one LED.

b. Place a 555 timer, resistors, and capacitors on the sheet

i. *Edit => Add*

1. In the search box at the bottom, type in **555* to search for any combination of terms with 555 in it.
2. Select TIMER, then OK. Click OK to place the part in the schematic and use <Esc> <Esc> to quit.

ii. To add a resistors, capacitors use the *rcl.lib*; for the LED, the *led.lib*.

1. *Edit => Add* to bring up the libraries
2. The libraries for the previous search of the 555 timer will be shown. To restore the view of all the libraries, clear the search box and press <Enter>.
3. Click the "+" on *rcl.lib* to drop down the menu.

4. Select the *RESUS-10 (name my be different).* This is a resistor with U.S. symbols and holes that are 10 mm apart. Select OK and place, with each click of the mouse, as many as you need. Use right-click of the mouse to rotate the part. <Esc> to quit.
5. In the *rcl.lib*, look for ELC-2.5L. This is an electrolytic capacitor with leads that are 2.5 mm apart. Use this capacitor for the 2.2 uF. Use a CAP-7.5 for the 0.01 uF capacitor.
6. In the *led.lib*, look for LED5MM and select *Add*. LED stands for Light Emitting Diode.

c. Delete, Rotate, Mirror, Copy, Move, etc. as needed using the drop down Edit menu.

d. When using a multiple package IC, such as a TL072 op amp, there are two op amps in a package. They will be placed, for example, as IC1A and IC1B. The power supply pins can be added using the "Invoke" command from the Edit menu

10. Draw the Power Supply Sources

a. In the Control Window, Library, select *battery.lbr*. Choose *AB9V* and add to schematic.

b. To connect the power supply and grounds, use a symbol instead of "wiring" them together.

i. In the Control Panel, go to *supply2.lbr* and select *+9V*. This is a symbol that, when attached to pins, will automatically be common to all without the need for wiring.

ii. Attach the +9V symbol to the positive side of the battery and to all connections that are common to that. Use right-click of the mouse to rotate part as needed. Place the +9V symbol at the <u>tip</u> of the battery terminal marked +. An error will result later if it overlaps.

iii. In supply2.lib, select GND and place on the negative side of the battery and to all connections that are grounded (ground symbol).

11. Wiring the Schematic Diagram

a. Use *Draw => Net* to make the wiring connections. (Do not use the wire command for wiring.) <u>Be sure not to overlap the lines when connecting to parts.</u>

12. Assign Values to the Parts and Change Name if Needed

a. Use *Edit => Value* and select the part. Fill in the window with desired value

b. Use *Edit => Name* and select the part. Fill in the window with desired name change.

13. Check for Errors When Schematic is Finished

a. Use *Tools => Erc*. (One warning will usually show regarding the power supply.)

b. Check the Control Panel window and notice that an *.erc file has been created which denotes errors and warnings, if any, in the schematic design.

14. Fill in your name in the document box

a. Use *Draw => Text* and place text in document box.

b. Use *Edit => Change Size => .1* and select text to resize.

HOW TO MAKE DOUBLE-SIDED CIRCUIT BOARDS II

15. Generate the Board From the Schematic

a. To generate the board, go to *File => Switch to board*; answer *Yes* to message.

b. In newly created Board window, group components and drag into rectangle. The rectangle is the new circuit board size.

c. Move and rotate components as desired.

   i. Notice that each part has a cross somewhere. This is used for selecting that part for moving, deleting, etc.

   ii. The parts value and name move with the component. To separate the name from the component in order to move it separately, use the *Edit => Smash* tool. The name can be moved independent of the part. To undo, use the <shift> key while "smashing".

   iii. Leave at least two grid spaces (100mil) from edge board to component or text.

   iv. Leave at least two grid spaces (100mil) between pads when placing components.

d. Resize the circuit board using Move command by selecting the sides of the rectangle. Keep 100mil from any pads or text.

e. While moving the components, notice that the "rubber-banding" also moves. Usually a better rubber-band route exists. Use *Tools => Ratsnest* to rearrange the rubber-banding to the shortest distances.

16. Measure Between Two Points

a. Use this to measure the size of the board or distance between parts.

b. The cross on the screen show the absolute coordinates 0,0. Use *View => Mark* to place a relative 0,0 coordinate as point 1 anywhere. Then move cursor to anywhere point 2 and notice the relative coordinates in command line. The "R" values are the x,y distance from point 1 and the "P" values are the polar coordinates from point 1.

17. Define Signal Classes

a. Traces should be sized according to manufacturers specs and Ampacities. Create different signal classes and define sizes and spacing using Net Classes.

   i. **Minimum trace size and Ampacity**: (300mA, 10mils); (400mA, 15mils); (700mA, 20mils); (1 Amp, 25mils); (2 Amp, 50mils); (4 Amp, 100mils).

   ii. Preferred minimum trace width, 15 mils. Use this as the default size. Some Board houses can manufacture traces down to 8 mils, if needed.

   iii. Preferred minimum spacing between traces and parts, 15 mils. Use this as the default. Spacing can be as small as 8 mils for some Board houses.

b. In the Schematic window, go to *Edit => Net classes* to create the classes of signals.

   i. For Number 0, always leave name as Default and set Width = 15mil; Clearance = 15mil. These specs will then apply to all traces that are undefined in subsequent Net classes.

   ii. Number 1, we can arbitrarily name this Power, Width = 20 mil, Clearance = 15mil.

   iii. Number 2, we can arbitrarily name this Ground, Width = 25 mil, Clearance = 15 mil.

18. Assign Signal Class to Traces

a. Select the tool, *Edit => Change => Class*; select number 1, Power.

b. Click the cursor on the rubber-band that is connected to the plus side of the battery, which we will define as Power. After clicking, nothing seems to happen, but it and all instances of the rubber-banding connected to this are now defined as Power and will have traces and clearances according to the definitions created earlier in the Net classes.

c. To verify the class properties of the rubber-band, select *View => Info*. Now select the rubber-band that was selected as Power. A window will show the properties indicating the net class = Power. The width is still 0 because the traces have not been laid yet. Click other traces other than Power while in *Info* mode and note that they are Class = 0 default.

d. Select the class for Ground (same as (a) above) and select one rubber-band that is connected to the minus side as the battery which we will define as Ground. All instances of Ground now inherit the width and clearance defined in Net classes. Use *View => Info* to verify that the class Ground has been assigned.

19. Create Board Traces

a. To create traces from the rubber-banding (ratsnest) use *Tools => Auto => OK*.

b. To undo traces and restore the ratsnest, select all the traces using the Group tool. Then use the *Edit => Ripup* tool. (Remember to right-click to ripup the group – see instructions in command line in lower left of screen)

c. After creating traces, verify trace widths using *View => Info*.

d. Check trace destinations. Use *View => Show* which will highlight an individual trace in its entirety. The trace connection points can be verified for correctness.

e. As a final check, run a Design Rule Check. Instead of default settings, use supplied file PCB-POOL.dru. Change location in Control Panel *(Options => Directories)*. From the circuit board screen activate (Use *Tools => Drc => OK*. In the lower left-hand corner of the screen it should read "DRC: No errors".

20. Place Text on Board – the Silkscreen Layer

a. Before writing text on the board, the silkscreen layer must be chosen as the destination layer for the text.

b. *Edit => Change => Layer*; select *tPlace* for the top of the board silkscreen (recommended) or choose *bPlace* for the bottom of the board silkscreen. Click OK.

c. Very important: Change the font type to Vector. This is the only font that displays correctly between the computer image and the output file. *Edit => Change => Font => Vector*.

d. The text must not cover the pads, since these are solder points. Covering traces is OK.

e. Create the text. *Draw => Text*. Right-click to rotate if needed when placing text.

f. To change size of text, *Edit => Change => Size*, select size and click cursor on the cross of the text.

g. To check the size, font and layer of text, use *View => Info* and click cursor on the cross of the text.

h. To delete text, use *Edit => Delete*.

21. Create Gerber Files and Drilling Data

a. The Gerber and Drilling data files are the files necessary for the Board house to manufacture the boards.

b. To create Gerber files, go to the Control Panel. Expand and select *CAM Jobs => gerb274x.cam*.

c. When the CAM window appears, in that window go to *File => Open => Board*, select the current board being worked on, and *Open*.

d. In the CAM window, select the *Silk screen CMP* tab. The silkscreen will print the layers that are enabled in the chart on the right. Enable by clicking the number layer for *Dimension, tPlace, tName, tValue*. This will print the board outline (Dimension), top of board text (tPlace), top of board component names (tName), and top of board component values (tValue). If (silkscreenJ) text was placed on bottom of board, enable the bPlace layer, otherwise do not enable it.

e. Create files. Select *Process Job*. Close CAM window. The Gerber files have been created and are listed in the Control Panel under the Project.

f. Create preliminary Drilling Data file. In the Control Panel, expand the *User Language Programs*. Right-click on *drillcfg.ulp*, select *Run in Board => Inch => OK => OK => Save*. This is the preliminary process that is required before actually generating the drilling data. A *.drl file has now been created in the Project menu.

g. Create Drilling Data file. In the Control Panel, expand and double-click *CAM Jobs => excellon.cam*. When the excellon window appears, select in that window *File => Open => Board*, select the board, then *Open*. Next, select *Process Job* and close. All the drilling data files have now been created and are listed in the Control Panel under the Project.

22. Check Output Files With Gerber Viewer (ViewMate)

a. Start the program called ViewMate.exe. Viemate is a "gerber file viewer". A black slate appears.

b. Open Eagle and place Eagle and Viewmate windows side by side. Display the Eagle Project files. Drag and drop the *.cmp, *.plc, *.sol, *.stc, and *.sts files one at a time onto the ViewMate black slate. The resulting view in each case is what the Board house will be manufacturing.

   i. *.cmp is the component side.
   ii. *.plc is the silkscreen for component side
   iii. *.sol is the solder side
   iv. *.stc is the soldering mask for the component side
   v. *.sts is the soldering mask for the solder side

c. Check for obvious irregularities, like traces too close, text overrun outside the board border, etc. The most common error is text overrun due to not selecting the Vector type font. Make changes by starting up Eagle and going through the board procedure again.

d. When done, close ViewMate.

23. Prepare Files To Be Sent to the Board House

a. Create a README.txt file with the following information for the Board house.
   i. Your name, address, phone number, email
   ii. Name of Project
   iii. Number of boards desired
   iv. Thickness of board: 62mils (the Gerber files do not specify a board thickness and this value is typical)

b. If the board house accepts *.brd files, simply attach that file and include the readme.txt information. Upon your request, the board house will send you an estimate to produce the boards.

c. If the board house does not accept *.brd files directly, zip all the Eagle project files and the README.txt file into one zip file. Send the zip file to the Board house of your choice for an estimate. The Board house does not need all the Eagle files in the Project, but to be on the safe side, send them all (*.brd, *.cmp, *.drd, *.dri, *.drl, *.erc, *.gpi, *.plc, *.pro, *.sch, *.sol, *.stc, *.stc, *.sts, README.txt) as one zipped file.

24. **Board House**

a. PCB-Pool www.pcb-pool.com
   i. The only manufacturer that will take multiple board designs and only charge for one set up. Excellent prices

b. Advanced Circuits, www.4pcb.com http://www.4pcb.com/
   i. Good rates for students: $33 per board, no minimum. Look under Specials -> Student Discount. Boards must be sent to a University.

c. Sierra Proto Express www.protoexpress.com
   i. I've used the following Board house with very good results. Fast turnaround time, excellent quality boards, reasonably good prices.

d. Custom PCB http://www.custompcb.com
   1. Lowest cost I've seen.