

ECE2241b Audio Amplifier PCB Design

Circad Printed Circuit Layout Instructions

Circad is a printed circuit board layout program that is available for use in room #3108 or can be downloaded from the site www.holophase.com.

Note: The PCB layout will be checked by the Electronics Shop staff just one time and only one board will be made, so make sure all corrections are made. If any mistakes are made they must be repaired on the board.

Install Circad and run the program from the executable file. The main menu bar will appear horizontally on top. If the top menu bar disappears, click the right mouse button twice to restore the main menu bar.

- To display sub-menus move the cursor over the top menu bar and click the left mouse button.
- To select a sub-menu function, move the cursor over the desired function and click the left mouse button again. Complete the selected function and confirm.
- With the main menu bar visible click the left mouse button to “Zoom” in or click the right button to “Zoom” out. Holding the left or right mouse buttons drags the screen around.

Note: The printed circuit board (PCB) you are designing will be using a component side (top) and copper side (bottom). While viewing your board in Circad the component side is closest to you while the copper side is seen looking through the board to the other side.

Setup:

- Select “File” from the menu bar then “New” from the submenu and then “Graphic File”
- Check mark size “A” page under “PCB files” then depress <Confirm>
- In the popup menu type in the initial of your first name followed by underscore, last name and the .pcb extension (eg. m_smith.pcb) then “Save” (this allows ease of identification in PCB production).
- Select “Setup” from the menu bar then “Grid Snap”.
- Choose 0.025” in “Snap Table” window then <Confirm> (this sets component placement and line resolution snaps to a minimum of 0.025” for good PCB resolution).

Setting Required Board Size to 1.5” x 3”:

- “Zoom” in to display grid dots and move the lower left corner of the page to the lower left corner of the screen.
- Select “Place” (main menu bar) “Line” (submenu).
- Under “Layer name” choose “Bottom Copper”. Set “Line thickness” to “0.020” and <Confirm>.

- Move cursor to approx. 1 inch from the bottom, and 1 inch from the left as indicated by the coordinators. With both axis on top of the nearest dot, press < s > to select the “**S**etup” menu then < p > for the “Zero **P**oint”. The displayed cursor coordinates then become zeros to identify the reference point.
- With the cursor at the zero coordinates, press and release the left mouse button to set the beginning of the line. Move the cursor to the right until the coordinates read “3.000 0.000” then again press and release the left mouse button. This fixes the line at this corner. Move the cursor to the next coordinate at “3.000 1.500” and again press and release the left mouse button. Move the cursor to “0.000 1.500” and again press and release. Move the cursor to “0.000 0.000” and again press and release. Now press and release the right mouse button to terminate further lines.

Selecting lines and adding components, pads, and text:

- To add a line select “**P**lace”(main menu) “**L**ine”(submenu) and select bottom copper under the heading layer name. Set “line thickness” to “0.02” (this is the minimum width for good PCB resolution) and < **C**onfirm >. Power supply and Load lines should be 0.07”.
- Select “**P**lace” “**C**omponent” then click on ECE2241.lib. From the list use RES for small 0.25W resistors, 2N3904 for the 2N3904 and MPSA18 transistors, CAP1 for 47µF capacitors, CAP2 for the 100µF capacitors.
- The ECE2241.lib library will be on the computers in room 3108. If you are using a different computer, you can download the ECE2241b library file from the Electronics Shop website: <http://www.eng.uwo.ca/electrical/e-shop/default.htm>. Place the library file in the “C:\Program Files\OmniGlyph\Library” folder. After that you have to update your CIRCAD library list by selecting “**P**lace” – “**C**omponent” menu and clicking on “Modify Library File List” menu, “Browse” to ECE2241 file and press “Confirm” button.
- The currently selected component can be rotated by 90° by typing “R”.
- To add a pad select “**P**lace” “**P**ad”. Make sure “Layer Name” is entered with “0 = Pad Master” and “Pad Type Code has entry “T1 = Oval thru Pad.” In both “Pad X Size” and “Pad Y Size” set both sizes to 0.125 inches. Set the Drill Hole Size to 0.025 inches. **Note:** Use a pad of the above size for test points, inputs, outputs, and power supply pins. The pins are provided with your lab kit and must be snapped into place **and soldered**.
- To edit a pad select “**E**dit” “**P**ad/via”. The outside pad diameter can be modified by changing the Pad X and Pad Y size. The inside pad diameter is modified by changing the Drill Hole size. **Note:** Make sure that **all** pads including the library component pads have a drill hole size of 0.025 inches (these holes form a means to centre the drill bits for drilling the PC board). **Do not** place new pads on top of Library Component pads to modify the pad dimensions, use “Edit” “Pad”.
- Your name and student number must be placed on the PCB for identification, select “**P**lace” “**T**ext”. Make sure “Layer Name” is “8= Bottom Copper”, “Text Size” is a minimum of 0.072 inches and “Mirror Image” is “Normal for Layer”. The text will appear mirrored, this is correct since you are looking through the board to the other side.

Printing:

- Select “File” from the menu bar then “Printer Out” from the submenu
- At the “Source” submenu, check the “Visible Screen Area” box.
- At the “Layers” submenu highlight “Pad Master” under “Layer Name”, then right click on the color brick beside it and choose “New CMY Color” option. Choose **black** color from the palette and press “Confirm” button. This should change the color of the Pad Master brick to solid black.
- Repeat the above procedure to change the color of “Top Silk” and “Bottom Copper” to black.
- Turn off all other layers by double clicking on the “Layer Name” which will cause the “Color” box to be half filled in. Make sure the layers that were changed to black are a solid colored rectangle.
- At the “Format” submenu make sure “Print to Scale” is checked with the “Scaling Factor” set to 1.0000.

Submitting the layout:

- Use the checklist below to assist in finding errors
- When you have the PCB layout completed, submit the file on a USB memory stick along with a **1 to 1 scale printout** to the Electronics Shop (room #3105) for checking. If there are no errors the PCB will be made, but if there are errors the corrections to be made will be marked on the printout. After all corrections are made submit the final PCB layout the Electronics Shop.

Checklist: Use the check boxes and a 1 to 1 scale printout to assist in checking the PCB

	- is the board 1.5” x 3.0”
	- are all drill holes 0.025” (look at the white space at center of pads on printout)
	- are the power supply and load lines (traces) 0.07” thick
	- are all other lines a minimum of .02” thick
	- is your name and student number on the board at a min. of .072” in size and appears mirrored
	- are all connections correct to the transistors.
	- do the components have clearance from each other (placing the actual components on the 1 to 1 copy will help)