

The “USELESS BOX” Project

The useless box is exactly what it implies.... A project that is pretty much “Useless” and is made for pure entertainment. You are going to go through the process of building this project from breadboarding and programming it using Arduino, to making a circuit board and finally installing all components into a case to create a finished product. And, In the end, you will have a “Useless Box” of your very own!



PROCEDURE:

Use ‘DipTrace’ to create a Schematic...

- 1) Locate the application **DipTrace** on your computer and run the ‘**LAUNCHER**’. The Launcher presents you with options to run several sub-programs, select/launch the ‘**SCHEMATIC CAPTURE**’ app.

Once the program is open watch and complete the following tutorials...

- a) Schematic Tutorial 1 – [Library Setup](#)

Location of DipTrace Libraries:

Use ‘File Explorer’ to navigate to:

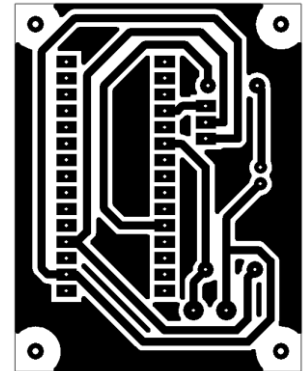
‘This PC’ > ‘SchoolShare’ > ‘Maker Wood Studentz’ > ‘Electronics 10’ > ‘Useless Box’ > ‘diptrace libraries’

- b) Schematic Tutorial 2 - [Making the Useless Box Schematic](#)
- 2) By now you are aware, the above tutorials used the ATTiny microcontroller. The project has been updated to now use the Arduino Nano microcontroller. Revisit your schematic design and make the necessary changes using the following document as reference: [Nano Schematic Update Document](#)
- 3) Print out your final schematic and submit it for approval to proceed.

Use 'DipTrace' to create PCB Layout...

- 1) It's time to design the Printed Circuit Board (PCB). To complete this task follow the video tutorials, a & b, below:
 - a) [PCB Tutorial 1 – Board Setup](#)
 - a. [Reference for Board Dimensions \(pdf\)](#)
 - b) [PCB Tutorial 2 – Component Placement and Trace Routing](#)

- 2) Once both Videos are complete, on your computer screen you should have a PCB Layout that looks pretty close to the one to the right.
- 3) Use 'print preview' to first view your design as it will be printed. Make any adjustments necessary, then print to the printer in Room #507. Have the teacher get your print, then check it over looking for: pad/drill hole sizes, traces width and spacing, connections between devices, etc. Then show your teacher for final approval.



Make a Printed Circuit Board...

- 1) Now that your PCB layout has been approved by your teacher, it is time to get things ready for the Laser to make your board. In 'print preview' select 'mirror' and 'negative'. When selecting a printer, choose 'Microsoft to PDF' and save the PDF file to your project's folder where you are keeping files for your project. Navigate to this file and RIGHT Click on it to select 'Open With'. Choose the app '**Adobe Reader**' to view your file.
- 2) Lasering your PCB
 1. With your file open in Adobe Reader, choose 'VLS 460' as the 'printer', make sure that 'Actual Size' is selected so your PCB size will not be altered.
 2. Open the Universal Laser Systems 'UCP (User Control Panel) on your computer and you should see the PCB layout on the screen.
- 3) Obtain a piece of copper clad board that has been painted with etchant mask/resist (flat black spray paint)
- 4) Place the board in the laser and complete the steps below:
 1. Plug the USB for the laser into your computer and once the option in the UCP is available, use the red power button to turn it on. Let the laser 'Initialize'.
 2. Focus

- i. Using the 'FOCUS Option' position the spotting laser (red laser) over the circuit board.
- ii. Use the plastic focus tool adjust the distance between the board's surface and the carriage
- iii. Adjust the board position so that the image will fit exactly on the board. This is fiddly and will take care and attention to detail. DO NOT proceed until this has been achieved.

3. Settings

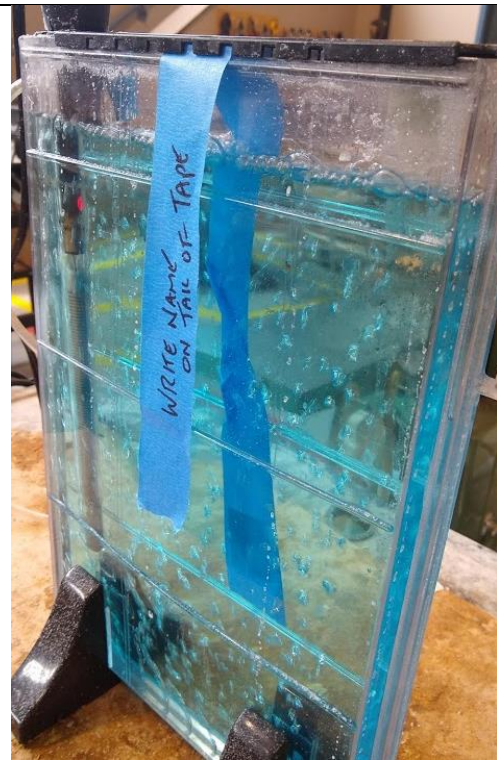
- i. You will be making 3 laser runs to completely burn the paint from the board. Select 'Settings' then the 'Manual' tab at the top. Select the Black 'Raster' settings and set these to the following.
 1. Run 1 – Power 40%, Speed 75%, PPI 500%
 2. Runs 2 & 3 – Power 100%, Speed 20%, 800%
4. Turn on the shop air supply valves (red and yellow) and the fume extractor.
 5. Run the job (big green 'play' button in the UCP). When completed DO NOT MOVE THE BOARD. After making the settings adjustments noted above, the 2nd and 3rd runs.

5) Etching your board. You are going to use the “Bubbler tank”. It has a chemical in it called “**Ammonium Persulfate**” that dissolves unprotected copper. The black paint (“mask”) protects your copper circuits from the Ammonium Persulfate. **HOWEVER**, leaving your board in the tank too long will result in ALL the copper being removed!! It should only be in the tank until the unwanted copper is removed.

- Use a piece of blue masking tape to create a way to suspend your board in the etchant tank
- Write your name on the end of the tape that will stick to the tank
- Twist the tape 180 just before the board so the board will face away from the side of the tank

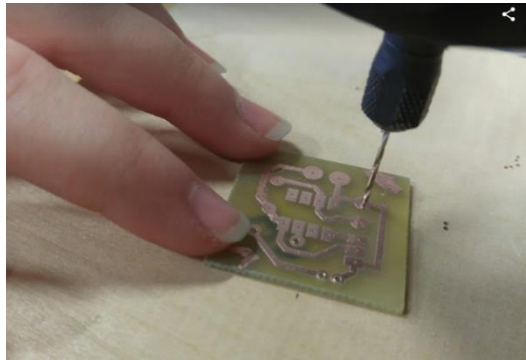


- **Make sure you have safety glasses on!!**
- Suspend the circuit board in the etching tank with the surface of the copper to be etched exposed to the bubbles
- The tank should be bubbling, and the heater should be on. The warmer the chemical, the faster it will etch
- The lighter colour of the chemical, the faster it will etch.
- Regularly monitor the process and remove the board from the tank when done. Place it in the container provided to take it to the sink and rinse it off

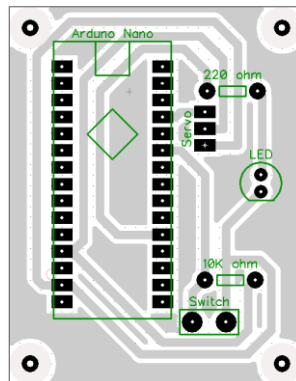


6) Remove the black paint from your circuit using Acetone and paper towel. This should only be done in a well ventilated area wearing gloves and safety glasses. Dispose of the paper towel in the ‘oily waste can’. Perform a final polish of the copper traces using fine steel wool.

- 7) Drill holes using the Dremel drill.
in the centre of each pad.



- 8) Use steel wool to shine the copper prior to soldering
- 9) Install and solder in all components. Use your DipTrace pcb diagram and the classroom sample to ensure that the parts are installed correctly.



Example PCB Diagram: Component Install View (Top Side)

If you don't have a similar diagram to the one show above, return to DipTrace, create one and print it.