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 designing the case in 3D and printing it out on a 3D printer. In the end, you will have a "Useless Box"

The following procedure follows the completion of Arduino Lab #5 – Servo Motors.

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, go to the class website and under "Useless Box Project" watch and complete the following tutorials…

   a) "Using Diptrace – Library Setup"
   b) "Making the Useless Box Schematic"

Use 'Diptrace' to create PCB Layout…

1) It's time to design the Printed Circuit Board (PCB).   Again, using DipTrace 'LAUNCHER', select/launch 'PCB LAYOUT'.

   Once the program is open, go to the class website and under "Useless Box Project" watch and complete the following tutorials…

   a) "Making a PCB Layout"
   b) "Making the PCB Layout for the Useless box"

   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.
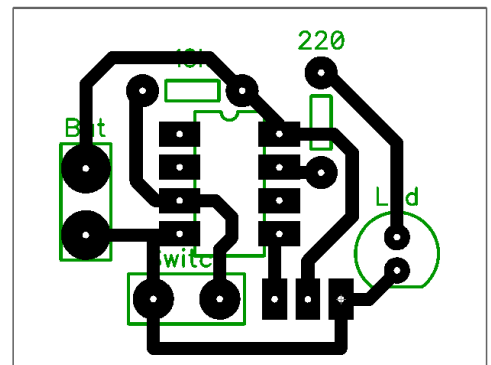
Make a Printed Circuit Board...

1) Creating the PCB. At this point you are going to print out your design and show your teacher. <u>This is the time to check for errors</u> (pad/drill hole sizes, traces width and spacing, connections between devices, etc.) Your print should be 'mirrored' and 'negative, select these settings in the print preview area of the Diptrace program. 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. Print it to 'Room 507'.

2) Once your design is approved, you will make your PCB...

   1. Open the PCB pdf file you created in the previous step (remember to do this in Adobe Reader).
   2. This time, 'print' the file to the laser choosing 'VLS 460' as the 'printer', make sure that 'Actual Size' is selected so your PCB size will not be altered.
   3. Open the Universal Laser Systems 'UCP (User Control Panel) on your computer and you should see the circuit.

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
   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. Positioning
      i. Using the 'FOCUS' option. Position the PCB so that the circuit will be lasered on it and the use of material is considered (ie, don't make your circuit in the middle of a large piece!)
   5. Turn on the shop air supply valves (red and yellow) and the fume extractor.
   6. 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.
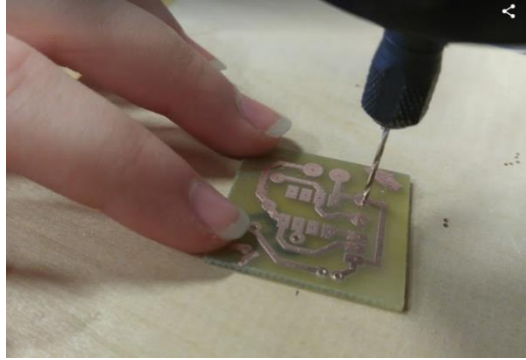
8) Install and solder in all components. Use your DipTrace pcb diagram to ensure that the parts are installed correctly. **IT IS VERY IMPORTANT** that the chip socket be installed with its 'notched' as per your diagram. Installing backwards will result in a short circuit and a broken chip!
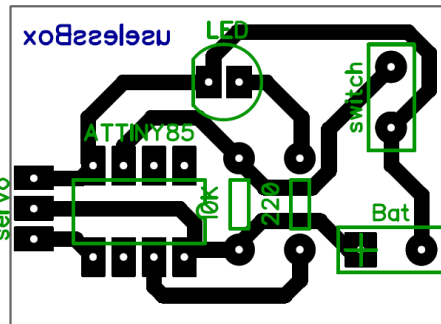
Example PCB Diagram: Component Install View (Top Side)

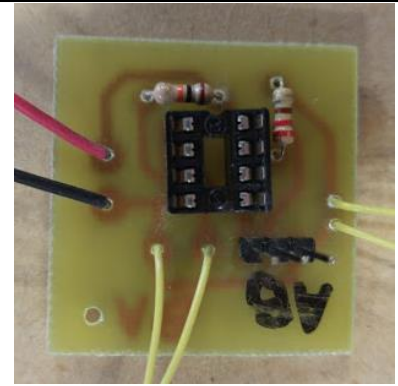If you don't have a similar diagram to the one show above, return to DipTrace and create one.

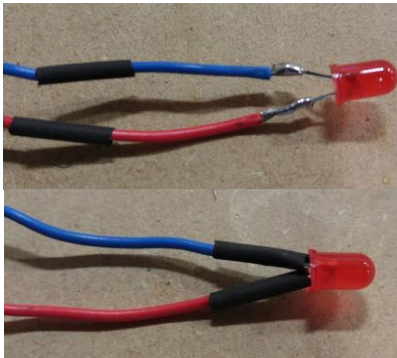**That which follows is not current. Updating is in progress. Please consult your teacher for next steps….**
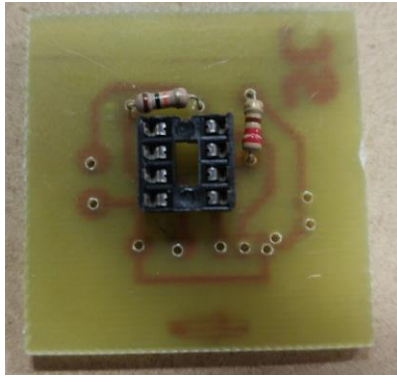
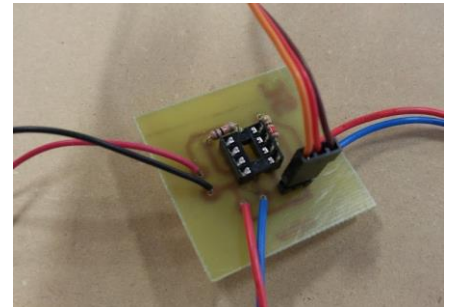|		
8 Pin DIP Socket 10K Resistor 220 Resistor	Solder all three components in place. Make sure the solders do not touch each other	Insert the AA battery pack positive and negative wires as shown. DO NOT get these backwards. Also, insert and solder in the wires for the switch and LED
Add Heatshrink to the LED leads so that the wires cannot touch each other and short out. Notice the heat shrink is on the wires BEFORE soildering ☺	Solder wires to the switch. Notice the heat shrink on the wires first ☺ Make sure you do not burn out the swith by holding the solder pencil TOO LONG on the switch legs.	Insert the servo wire as shown. Notice the colours and what direction they go.

9) The finished product should look like the following. Wires can easily be shortened…….
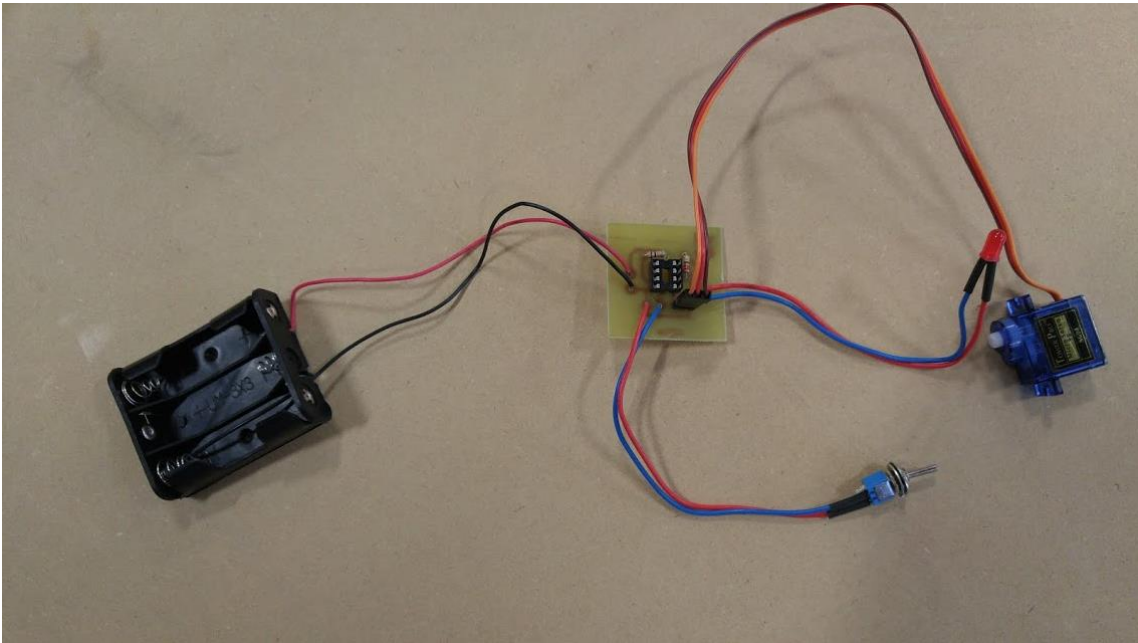
Testing time!

1. FIRST - test the board for short circuits by attaching it to a power supply WITHOUT an ATTINY85 inserted into the 8 pin socket. Connect 5 volts to the battery as shown pack (get the positive and negative correct) and make sure the board is not "shorting out" by looking at the power supply current needle. If the needle "pegs out" you have a problem. If no short, did the servo twitch? If so, that is a GOOD thing!!
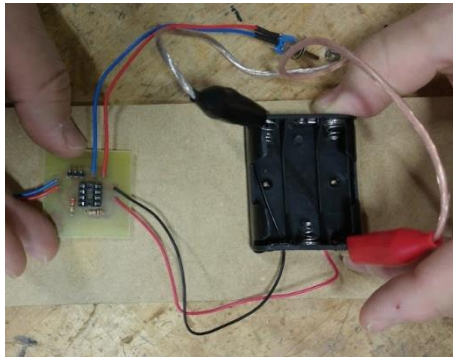
BAD!!!

2. SECOND – disconnect the power and insert the ATTINY85 as shown with the small dot on the chip (indicates pin #1) located at side of socket with notch. If you put it in the wrong way……..it will short out!!! Reconnect to the power supply that has been set to 5 volts. When you change the switch, does the LED turn on/off and does the servo move? If so, you are done and it work ☺ If not……..time to trouble shoot.
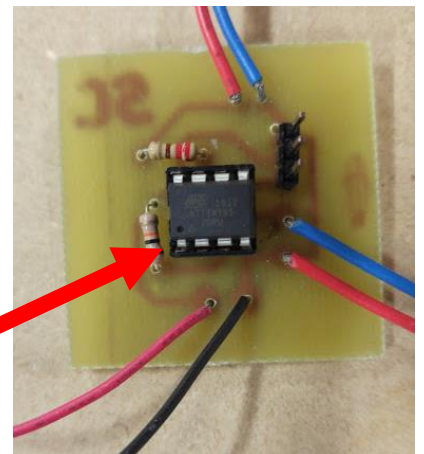

Dot

Troubleshooting Time!

STEP #1 (Establishing power)

So it doesn't work…….Start by testing for power. Using the "continuity" setting on the multi-meter, hold one of the test probes of the multi-meter on the "Positive" of the battery pack and then touch the other probe to the different spots on the board where "Positive" should be at. Does the multi-meter "BEEP"? If not, inspect the solder joints. If so, go to the next step.
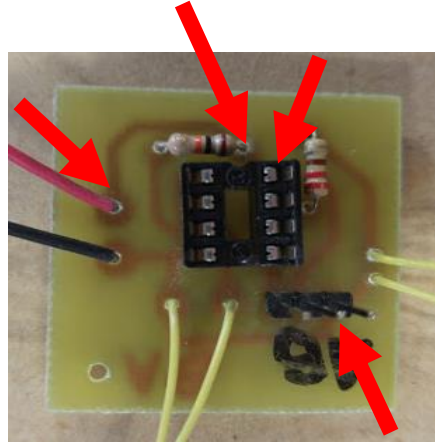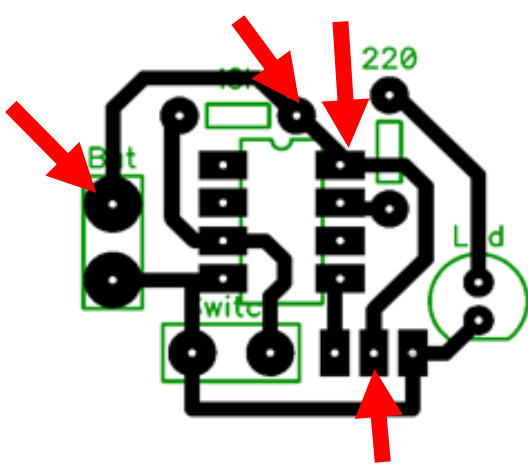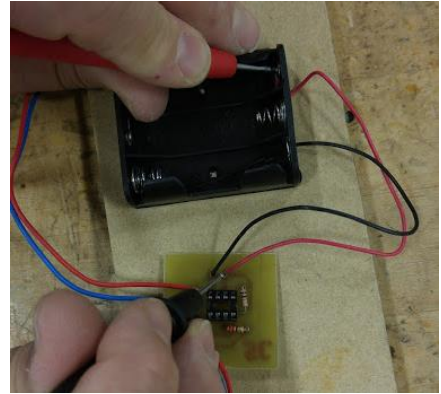
Next test for "Negative" using the multi-meter. Move the probe to the "Negative" of the battery pack. Then test the points shown below.   Does the multi-meter "BEEP"? If not, inspect the solder joints. If so, go to the next step.