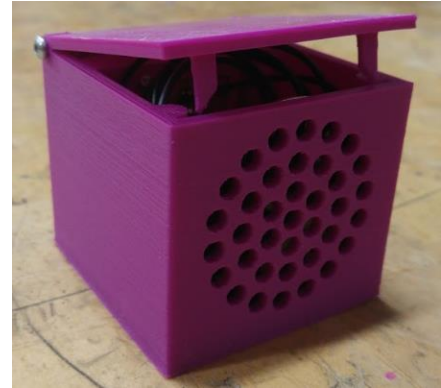# The Water Piano

Complete Project Instructions

This circuit uses capacitors to store and discharge energy at a rate which sounds similar to the frequencies our ears can hear as sound waves. It uses the human body and a trail of water as a variable resistor to vary the pitch, which lets us play the water like a piano with our fingers.
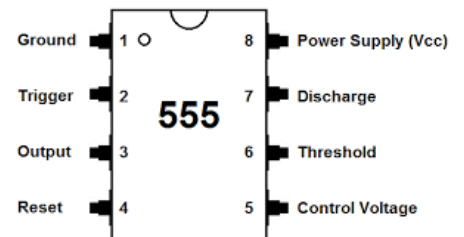
Instructions:

1. **Breadboard the schematic below.**

    Instead of a battery and switch, test the circuit with a power supply set to 9V. Once complete and functional, show the instructor.

WGSS Water Piano Circuit
Source: R Moslinger
Design: J Brett
April 2015
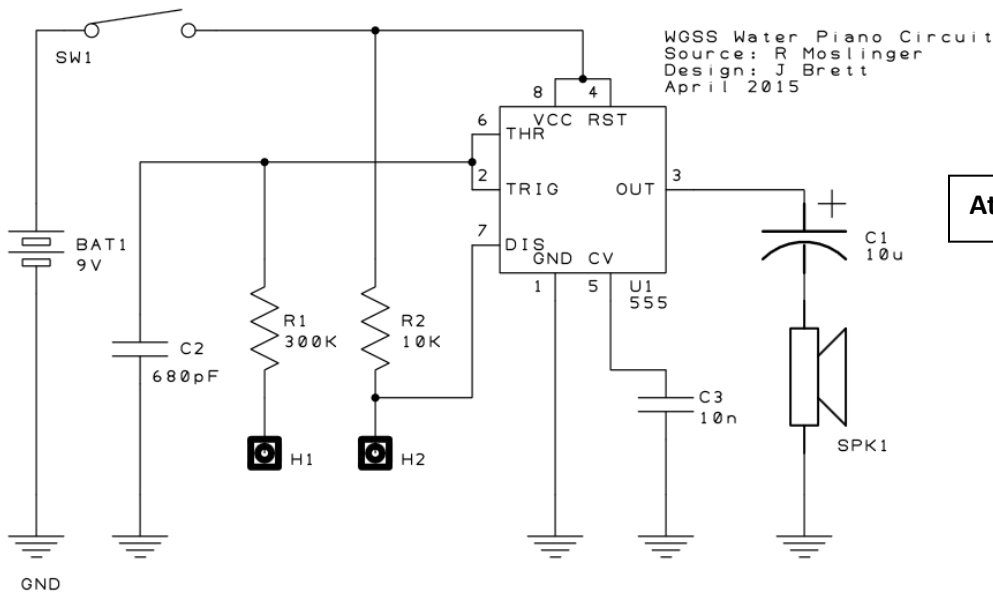
Attention: The 555-Timer Pin Locations

H1 to a washer. H2 to user's hand.
Place washer in a thin layer of water
on a desk. Touch water with other
hand. Pitch changes with distance.

Put the "H1" terminal in a puddle of water on the desk and hold "H2" with your hand. Then touch the water with your other hand at varying lengths from the H1 terminal to create different frequencies.

Switch the Capacitor #2 (680 pf) for a different capacitor. Try a lower value capacitor (470 pf) and a higher value one (820pf, 1nF or 1.5nF). What does this do to the sound frequency?

Save this breadboard – we'll make a printed circuit board from the breadboard design.

2. Open the program "**Diptrace**":   Start with a **Schematic Capture.**

   There are **Video Tutorials** on http://makerspace.isfeldschool.com/electronics-robotics-9/ to help you through all the Diptrace instructions.

3. We need to first set up a new library to use the parts we have in stock at this school. Make a new library by clicking on library tab (top toolbar)

   ➔ Library setup
   ➔ Add
   ➔ Select the new "library" that pops up and rename it Isfeld Parts
   ➔ Add library
   ➔ Navigate to :H -> MAKER/WOODWORK -> Diptrace -> select (highlight) all the files
   ➔ OK
   ➔ On the top left toolbar you'll see "Components": click there and then select the "Isfeld Parts" library that you just created

4. Make the schematic by clicking on components on the left toolbar, then placing them on the circuit. Start with the 555 timer, because that's what the whole project is built around.

   o Use the 1/4w resistors for both resistors

   o Use the 10uF capacitor for C1

   o Use the .1uF capacitor for C2 and C3

   o Use a Test Point for H1 and H2

5. Then place wires between the components to complete the schematic. Make sure all the components are connected as shown in the schematic or the picture to the right, this is important for the next step. Note that a node with a dot means the traces are connected, while a node without the dot means they don't connect.
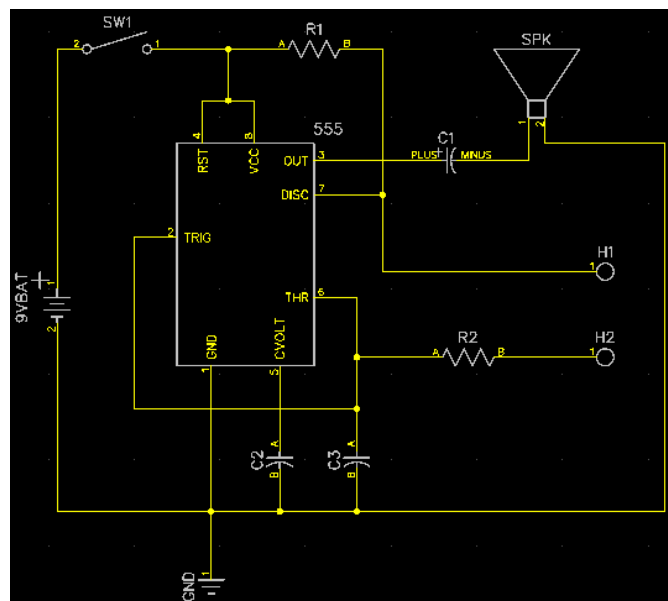
6. Make a **Printed Circuit Board** from the schematic by clicking file -> convert to PCB

7. What will show up will look like this, with the blue lines signifying a connected copper trace. Rearrange the components so that they are closer to the appropriate pins
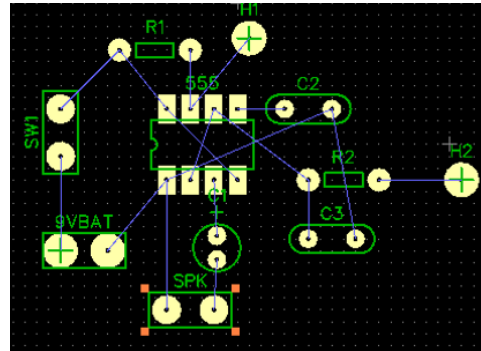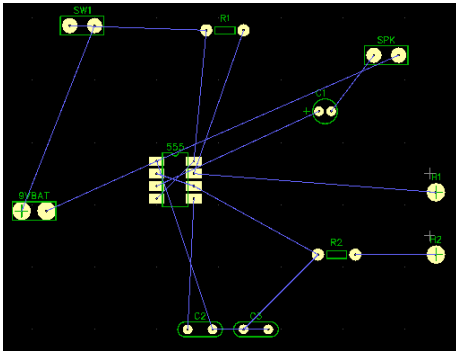

8. Now we'll change a few settings to the autoroute function

    o Go to **Route** -> **Current Autorouter** -> **Check Grid Router**

    o **Route** -> **Autorouter Setup** -> Uncheck **Use All Layers** and set **Number of Layers** to **1**

    o **Route** -> **Route Setup** -> **Trace Width** = **0.04** in. and **Trace Clearance** = **0.02** in.

9. Run **Autoroute** ▶. Most traces will complete themselves, but some will still be symbolized by the blue line and need to be completed by hand ✎. You will need to be creative with placing the traces, because some need to cross over each other. You can run a copper trace between the terminals of wide components like the 555 timer, the speaker terminals and the capacitors.
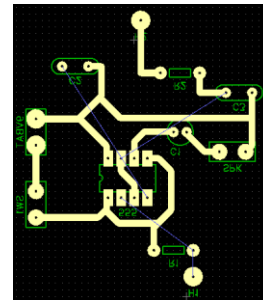
10. To alter the trace routes:

    o Go to **View** -> **Layer Display** -> Check **All Layers**

    o Change the **Current Signal/Plane Layer** to bottom (2)

    o Now left-click and drag traces to move them or right-click -> **Unroute Trace** to delete it and redraw it

    o Keep rearranging the components until they are as close as possible, but not too close that the traces touch each other, or the component outlines (in green) touch each other
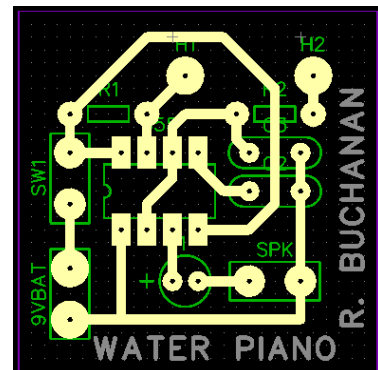
11. Again, all the steps from 2-10 are explained in a series of videos on http://makerspace.isfeldschool.com/electronics-robotics-9/

12. Print the PCB on Toner Transfer Paper (ask the teacher)

13. Cut a piece of Copper Clad Board to the same size as your PCB. The case is made for a board that's 1 ½" x 1 ½". Clean the copper with a piece of steel wool to remove grease and dirt, and then don't touch the copper with your fingers.


14. Transfer the PCB from your printed Toner Transfer Paper onto the Copper Board
    - Turn on the T-shirt Press and let it heat up to 350 degrees. Set the timer to 20 seconds.
    - Once the T-shirt press reaches temperature, place the **copper side up** and the PCB paper black **(ink) side facing down** on top of the copper
    - Close the lid until the lever clicks in place, wait until the timer beeps and pop the lever back up
    - The T-shirt press and your copper board are HOT! Use pliers to take your project off. Run it under water for 15 seconds to loosen the paper and pull it off.
    - Carefully check for breaks in the black traces. These black traces will be the copper connections between components, so they need to be unbroken.
    - Ask for a special "circuit board pen" to write your initials on the board and to complete some traces in case some of the toner didn't transfer properly
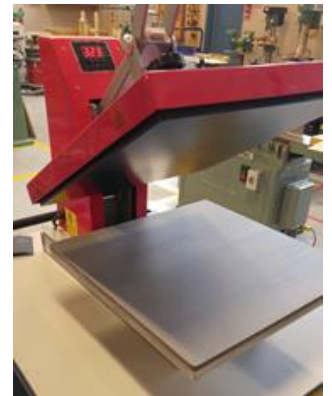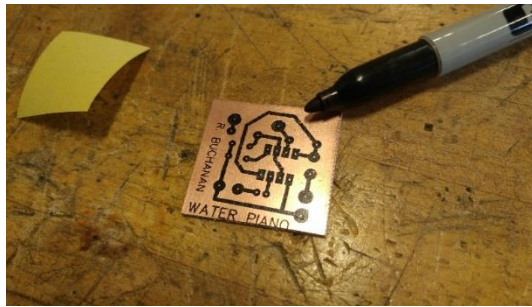

15. Etch the PCB in the Etching Tank. It uses Ammonium Persulfate to eat the copper away where there is no black toner.
   - Drill a hole in one of the corners and use a piece of wire to hang the PCB in the tank. **Don't drop the board in the tank**
    - It should take about 10 minutes, however if you leave it in the tank too long it will also eat through the toner, so watch it carefully and take it out as soon as you see all the copper has disappeared.
    - Once it's done, take it out and close the lid. Run the board under water for a few seconds, then keep the taps on for a minute (to wash out the ammonium persulfate so it doesn't eat the copper pipes). Then wipe the board with paper towel.
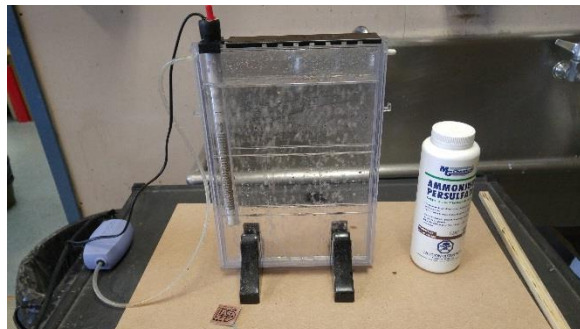
16. Clean off the black toner my lightly scrubbing it with steel wool.
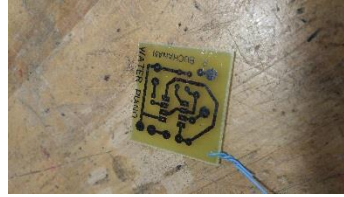
17. Drill out the holes with the Dremel drill in the centre of each pad. **Use Safety Glasses.**
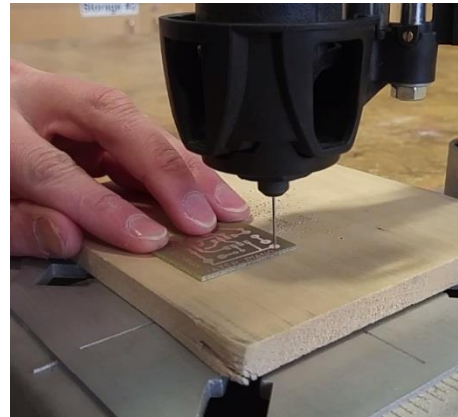
18. Place the components in the appropriate holes by sticking them through the top, so that the metal pins and copper are both on the bottom. Reference your PCB design on Diptrace to make sure you get the parts in the right place.

    o Make sure the 10nF capacitor is installed the right way with the cathode facing the speaker to ground

    o Cut 2 wires about 4" long and solder them to the terminals of the speaker

    o Cut 2 wires about 4" long and solder them to either end of the switch

    o Cut 2 wires about 12" long, solder them to H1 and H2 pads and solder a small washer or nut to the other end. These will be the terminals that you "play" the piano with.
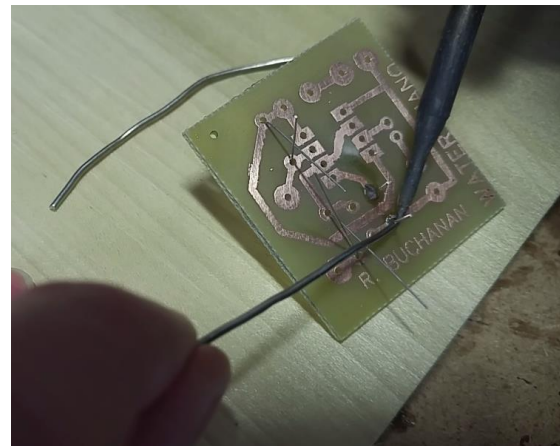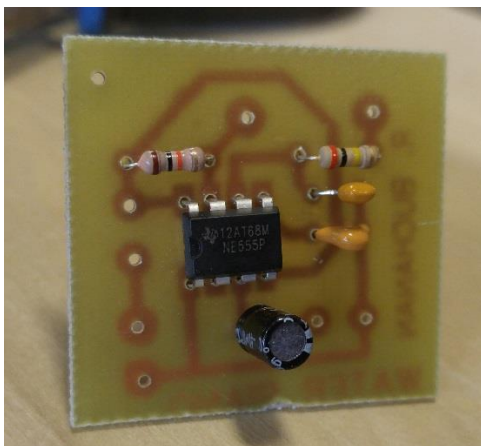
19. Solder the components to the board. Use enough solder to cover the pad and pin completely, but not too much that it's "globby" or touching another pad or trace.

20. Follow the video tutorials for how to make the 3D printed case.