Assembly Instructions for the Mini Curve Tracer Project

Some metal working and wood working skills are needed. In addition, you will need to upload software to the microcontroller. It is expected that you have some knowledge of soldering through-hole components and surface mount components. Protecting the project from electrostatic discharge damage is important. Obtain the proper work surface for ESD prevention.

When handling chemical and materials, always use proper personal protective equipment and safety procedures. Obtain MSDS Material Safety Data Sheets for all chemicals and materials. These data sheets have information on health hazards, if the material is reactive and the flammability of the material.

When using tools and operating machinery, follow the manufacturer's instructions and safety procedures.

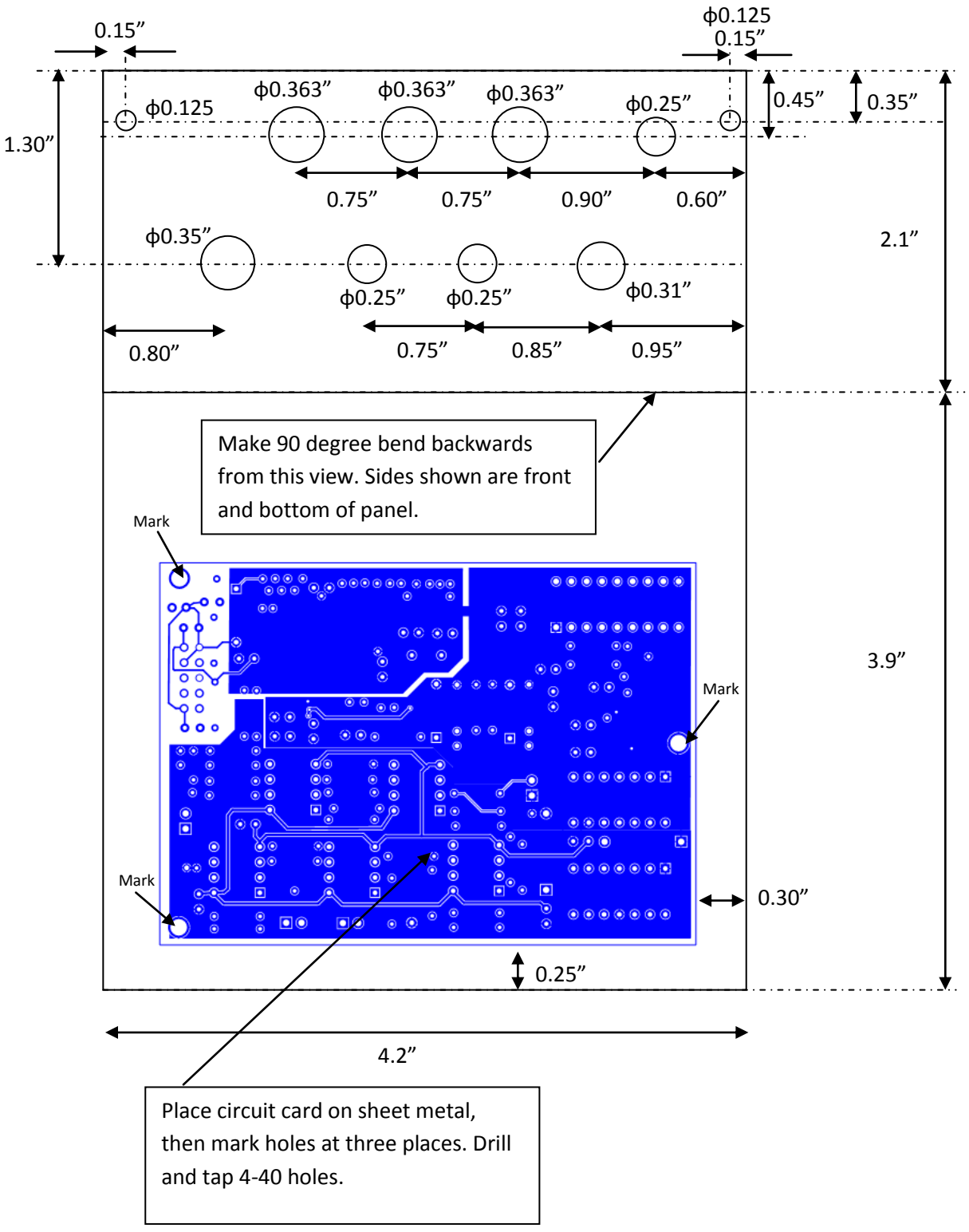
You may not have the proper tools or skills but don't give up. We all had to start somewhere. Find a neighbor, a local club, or an instructor at a local high school or community college. There are a lot of friendly people that are ready to lend a hand or some guidance.

Now let's get started.

Prior to adding components to the circuit card, the chassis panel requires marking of the hole positions for threaded fasteners that will hold the circuit card to the chassis. The panel can be made of 20 gage steel plate or a thicker gage aluminum plate. After bending and painting, the plate will look like the picture shown below. The following page shows the dimensions for the plate. All that is needed at this time is to cut the plate to 6"x4.2" and to mark the card hole positions.

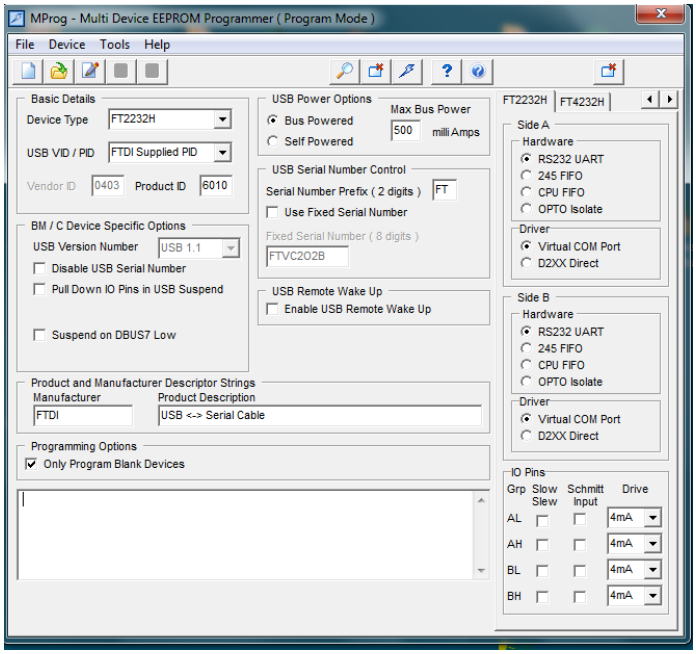
After the chassis face plate is marked, start with card assembly outlined in the step by step processed on page three.

Chassis Face Plate

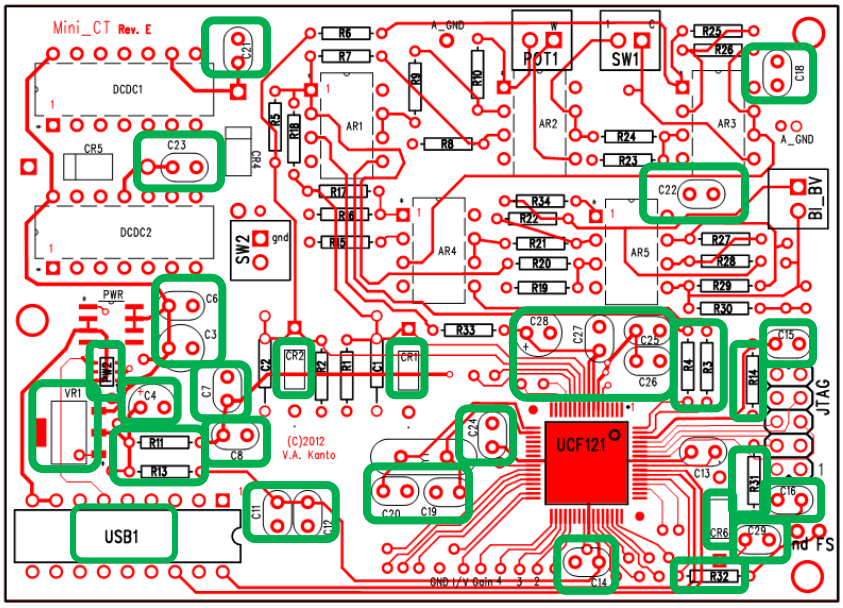


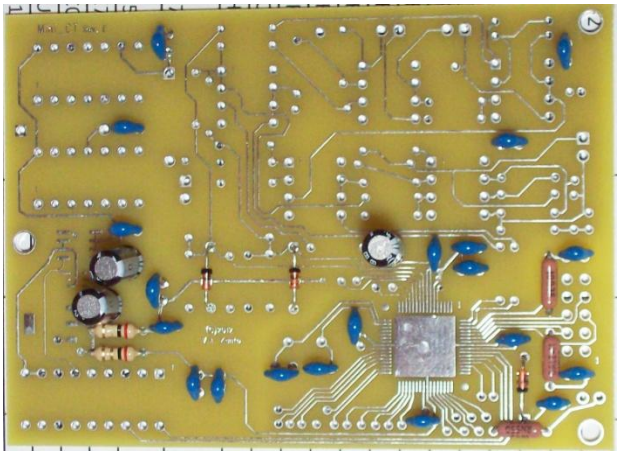
Card Assembly Instructions

1) Jumper pins 8 to 9 on the USB1232H device then connect it with a USB cable to your computer. If the drivers don't automatically install, you can go to the DLP Design web site http://www.dlpdesign.com/#Drivers to find the CDM driver down load. You will also need to download the MPROG utility.
2) When programming the USB1232H. Use the default serial number and Product ID. Program it as a serial interface and for high current 500mA operation. File minict.esp contains the programmer setup. After programming, remove the jumper between pins 8 and 9.

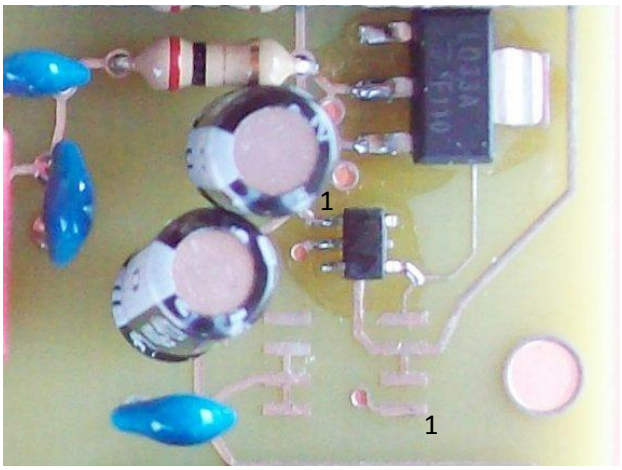


3) Install and solder the following components on the circuit board:
    a. USB1 converter DLP USB1232H
    b. VR1 3.3V regulator ST LD1117AS33TR
    c. PWR 5V switch TI TSP2041BDBVTG4
    d. 0.1uF capacitors C6, C7, C12, C16, C18, C21, C22, C23, C24, C25, and C27
    e. 1.0uF capacitors C8, C11, C13, C14, C15, and C26
    f. 0.01uF capacitor C29
    g. 39pF capacitors C19 and C20
    h. 2Ω resistors R11 and R13
    i. 4.7uF polarized capacitors C3, C4 and C28
    j. 4.7K resistors R31 and R32
    k. 100K resistor R14
    l. 1N914 Diodes CR6, CR1 and CR2
    m. 4.7K resistors R3 and R4





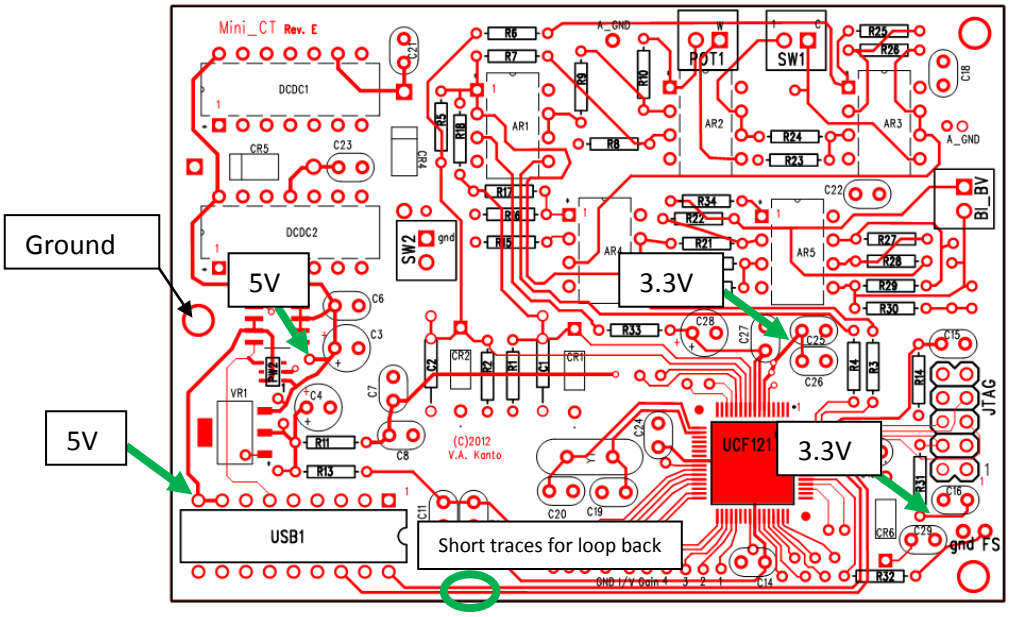
If you are like me and too cheap to pay for a silk screen print, use the diagram on the previous page to locate the component positions. This is the board after the passive components and diodes have been installed.



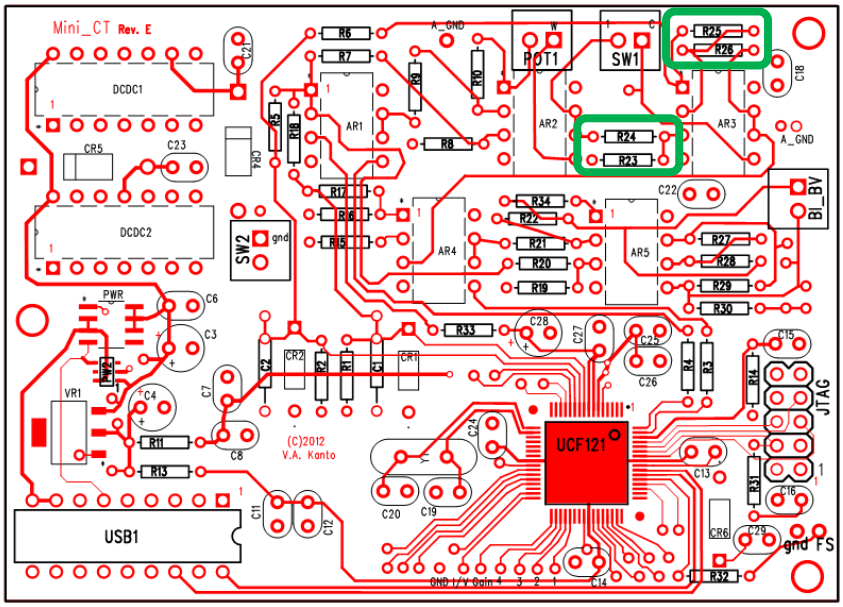
The voltage regulator and TSP2041 switch are surface mount. Pre wet the card lands with solder. Wet the areas with a rosin/alcohol liquid flux. It is ok to leave the flux on the card as long as it is rosin flux. Place the parts on the card and reflow the solder to the leads. Two sizes of the TPS2041 switch can be used. Only solder one to the board. Pin-1 is identified for both parts.

4) After installing the above parts, attach the USB1232H device to a computer. With respect to ground, use a DVM to check voltage at USB1 pin-9. It should be 5V. Five volts should also be present at the trace connected to the positive pin of C3. Positive 3.3V should be present at C25 and at C16. This test ensures the micro controller will have the correct power before it is installed.

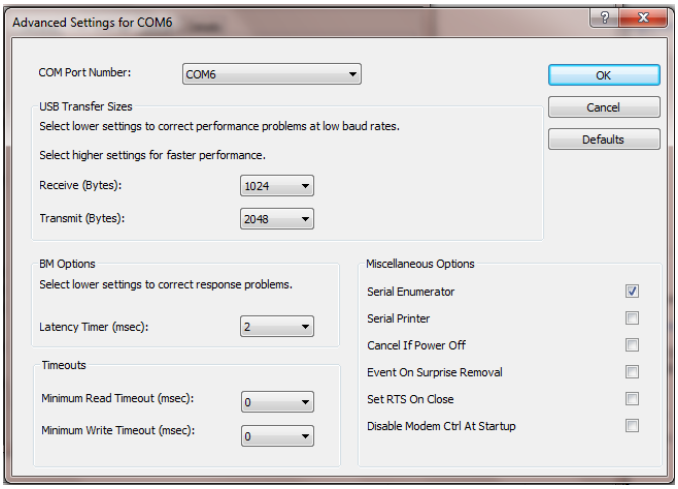
5) Testing of the serial interface can be performed using a terminal program. A program called sterm.exe is supplied if you don't have one. To do a loop back test, short together the two traces that are circled and type a few characters. Sterm.exe needs the port number. See paragraph 11) for more information on finding the port number.



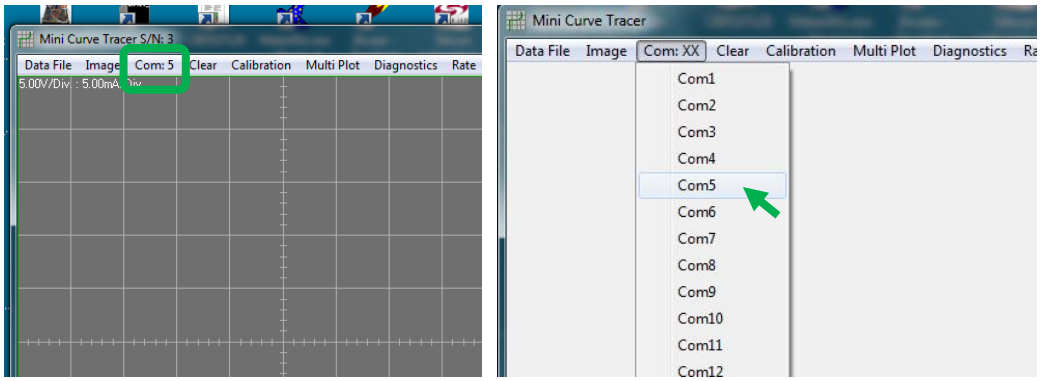
6) Take all of the 10K resistors and measure their resistance. Find the best match for two pairs of resistors. Select resistor R24 as close as possible to R23 and select resistor R25 as close as possible to R26. These resistors are used for the differential amplifier. If they are not well matched, a common mode error will result (current offset will increase with voltage). Install and solder these resistors.



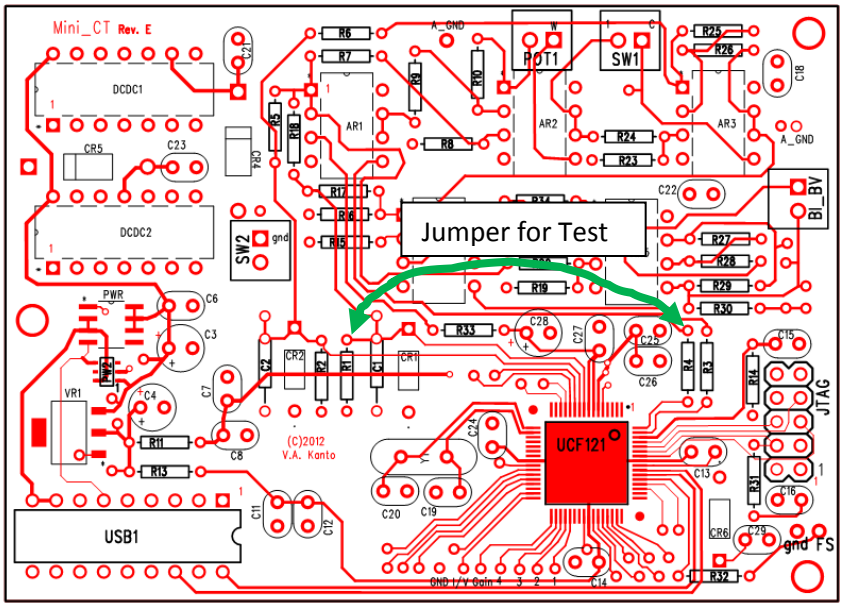
7) Install the microcontroller. Using 0.020” dia. SN60 rosin core solder and a low wattage fine tip iron 0.015”dia. tip (Weller model WM 120), wet all of the UCF121 pads with solder. Adding some liquid rosin flux will help prevent bridging solder between the pads. If a solder bridge occurs, use braided wick to remove the solder. Align the C8051F121 chip on the solder pads keeping the pin-1 orientation correct. The dot in the upper right hand corner indicates pin-1. Reflow solder on two corner pins to stake the part in place. Add liquid flux to the solder pads and reflow solder on each pad to form a fillet between the pad and lead. Slight pressure may be needed to make the pin contact the pad. Inspect for solder bridges and poor solder connections. Rework as necessary.
8) Place an insulator or tape (perforate at holes) over the Y1 solder holes then install the Y1 crystal. Solder the crystal leads to the board.
9) Install and solder the 10-pin dual row JTAG header onto the board. The board is now ready to program after a quick power check. With the exception of the loopback test, repeat test-4 to ensure the correct voltages are present.
10) Plug in the Silicon Laboratories USB debug adapter into a USB port on the computer. Also connect the 10-pin JTAG header to the JTAG connector on the Mini_CT board. The red stripe on the JTAG cable should line up with the pin-1 identifier on the board. Plug the USB1 device on the Mini_CT card into a second USB port on the computer. Start the Silicon Laboratories IDE interface and connect to the USB debug adapter. In the drop down menu, select Debug and the submenu Download Object File. Select the file mini_ct.omf. After the file is uploaded, software disconnect the USB debug adapter then remove the JTAG programming cable.
11) The Mini_CT card may need to be unplugged to initiate a reset. After the card is reset, proceed to my computer, properties, device manager and look for the serial adapters. The USB to serial converters has two ports but only the lowest number one is physically connected. Select it and set the properties, except for COM Port number, as follows:

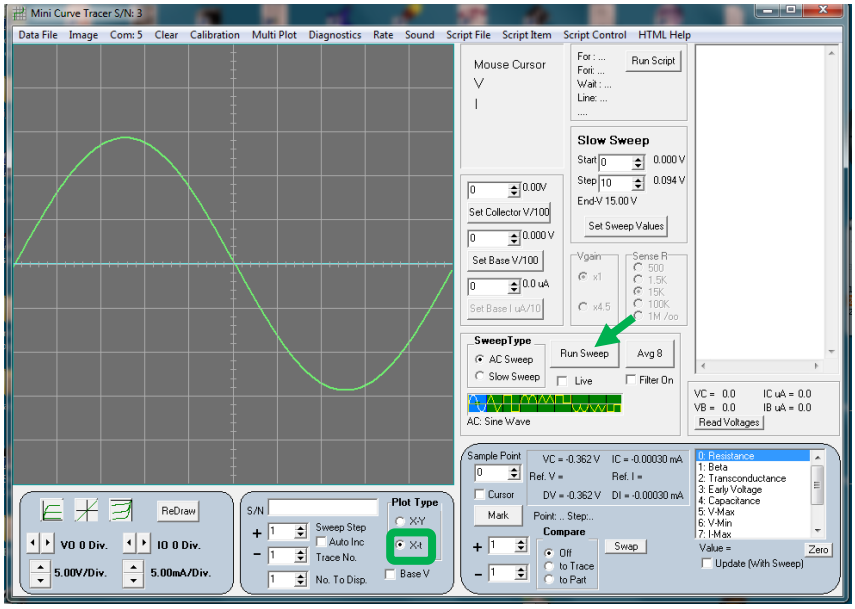


12) After the hardware is configured, take note of the COM Port Number. It may be needed if the interface software has trouble finding the serial port. Start the program mini_ct.exe. If all is well, the program will find the port and display it at the top menu. Otherwise, use the Com: menu item to drop down and select the port.



13) The hardware needed to measure current and voltage is yet to be installed. However, the DAC output can be connected to the ADC input to get a test waveform. Place a jumper clip between R1 and R4. Next, set the display for X-t mode and click the Run Sweep button. The display will show the output from the DAC waveform generator.



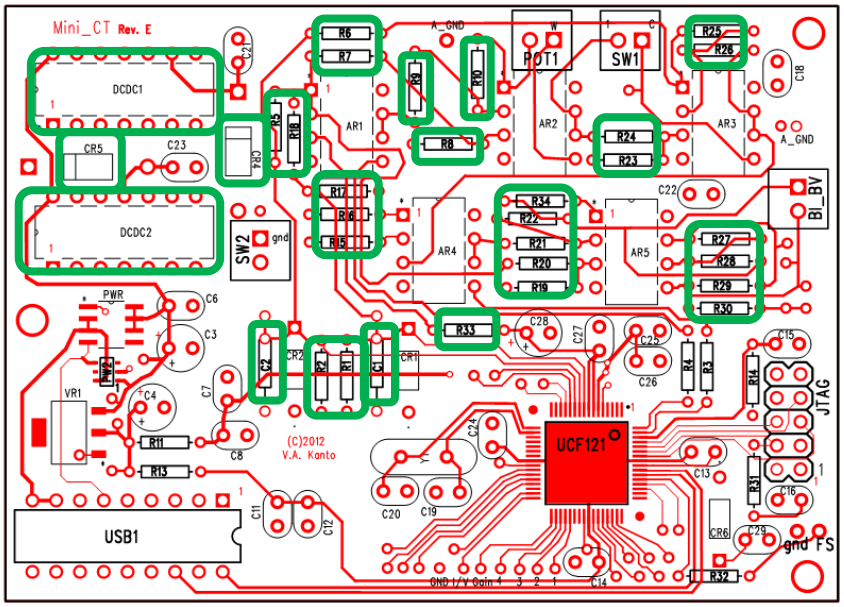


14) Disconnect the board from the computer then install and solder the following components:

a) 1N5819 Diodes CR4 and CR5
b) DCDC1 and DCDC2 converters Murata MEA1D0509DC
c) 1.0K Resistors R5, R18, R22
d) 1.2K Resistors R1, R2
e) 6.18K Resistors R6, R17
f) 15K ResistorR7
g) 13K Resistor R8
h) 4.02K Resistor R9

i) 12.5K Resistor R10
j) 2.74K Resistor R15, R20
k) 10K Resistors R16, R23, R24, R25, R26
l) 2.21K Resistor R19
m) 1.5K Resistor R21
n) 49.9K Resistors R27, R28, R29, R30
    a. Use a DMM to select as close as possible R27=R29 and R28=R30
o) 4.7K Resistor R33.

CAUTION! Small flakes of the tin plating may scrape off the leads when the resistors are installed. Be sure to remove any tin particles to prevent bridging shorts on the board.

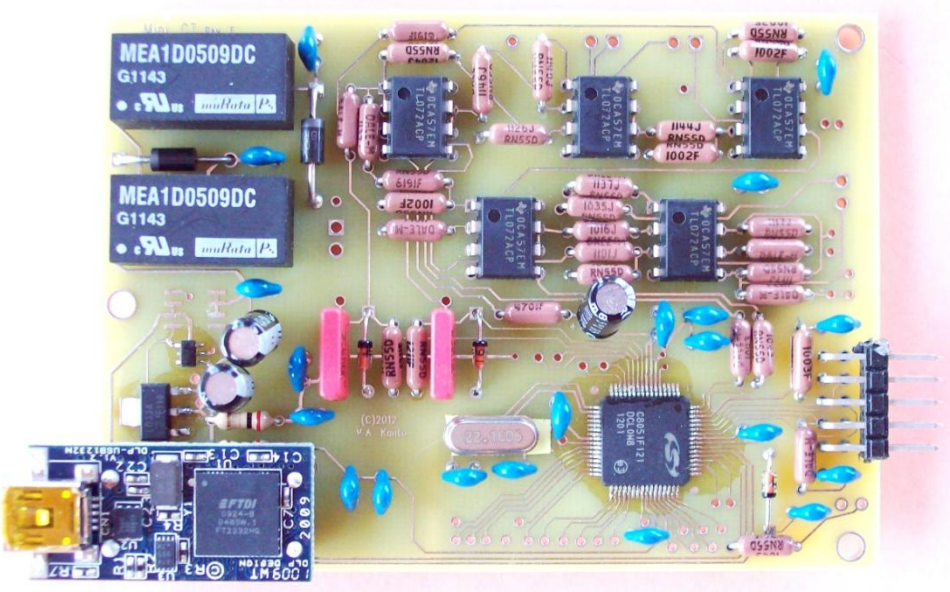


p) The analog section can be checked for incorrectly installed resistors. The following values were measured on a board. Expect close but not exact values. If values are off, look for shorts, missed solder joints and incorrect parts.

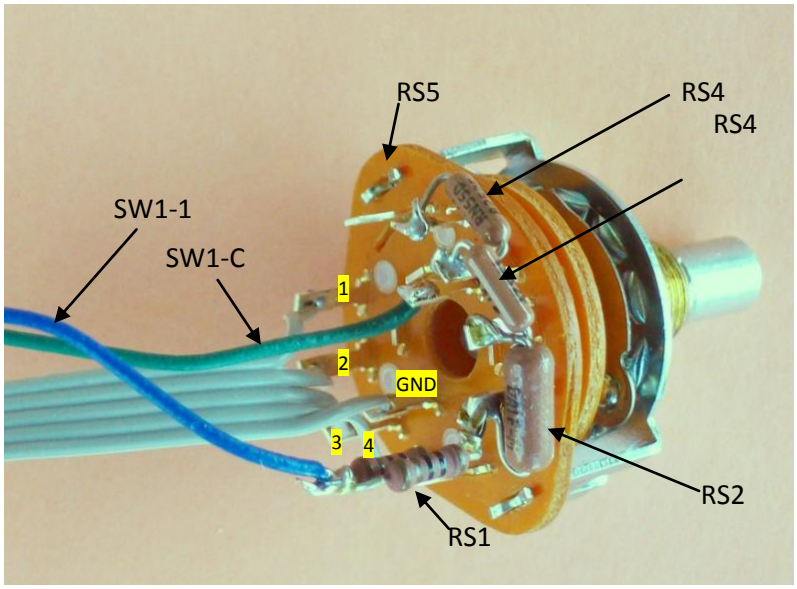
	AR1	AR2	AR3	AR4	AR5
Pin1-Gnd	873		7K	7K	200.2K
Pin2-Gnd	873		17.05K	17K	226.7K
Pin3-Gnd	308K	7.15K	10K	27.03K	1.394K
Pin4-Gnd	37.1K				
Pin5-Gnd	96.7K			96.8K	49.7K
Pin6-Gnd		20K	27.05K	230K	149.7K
Pin7-Gnd		20K	27.05K	230K	99.5K
Pin8-Gnd					

Other pins: AR1-7 to AR2-2 4.02K, AR2-6 to AR3-3 10K, AR2-1 to AR2-2 47.3K, AR4-6 to AR5-2 3.31K

15) Amplifier chips may be installed directly on the board. To make replacement easier, machine tooled solder tail sockets can be used. Prior to installing the amplifiers, plug the USB converter into a computer port. Measure pin-4 to ground on any of the amplifiers. It should be around -18V. Measure pin-8 to ground. It should be around +18V. If either voltage is not present, check for 5V to pin14 of the converters. Unplug the board and install and solder all five amplifiers.



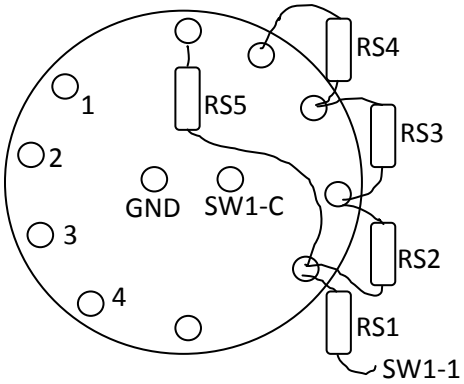
16) At this point, the board is still non-functional for display of waveforms until the potentiometer, rotary switch and toggle switches are installed.



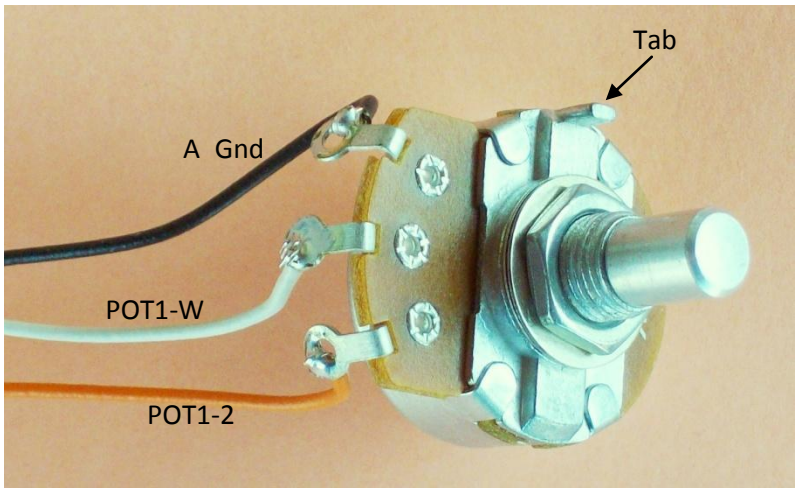
The Thevenin source (current sense) resistors RS1 through RS5 are soldered directly to the rotary switch SW1. RS5 is omitted for clarity. It is soldered at the junction of RS1/RS2 and to the 5th terminal that is open.

The flat cable connects to the computer port input to detect where the switch is set.

The flat cable is cut to 3.75” and the SW1 wires to 1.75”. Strip 1/8” of each wire end and tin with solder.

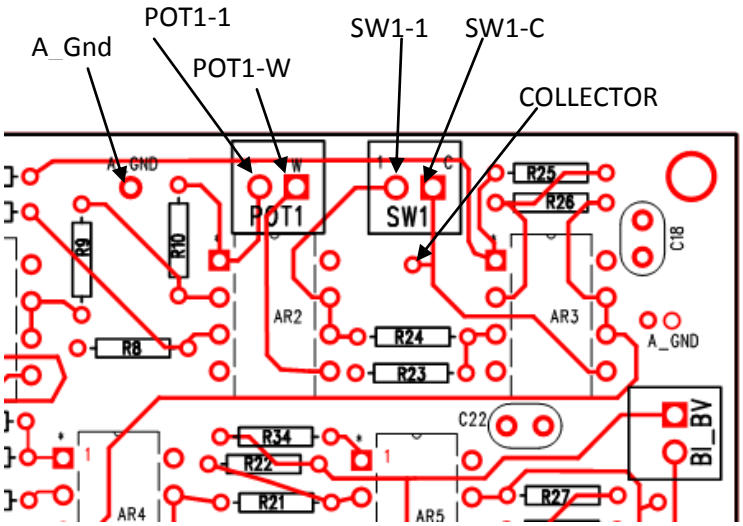


After the resistors are attached and soldered, use a digital multi meter to measure the resistance from SW1-1 to SW1-C. Measure and record the values for all five positions. These values are needed for calibration of the system.



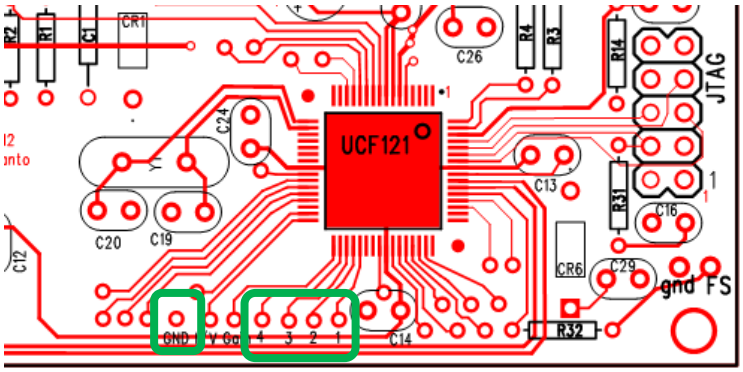
The potentiometer has three wires that can be braided for a neat look. The tab on the potentiometer should be broken off before installation.

Each wire is 3.25” in length. Again, strip back each end by 1/8” and tin with solder prior to assembly.

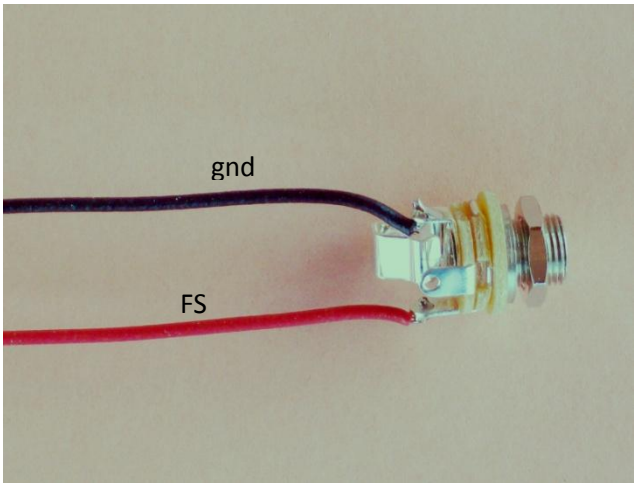


The locations for the analog wire terminations for the potentiometer and the rotary switch. Insert the stripped back wires from the card top and solder in place.

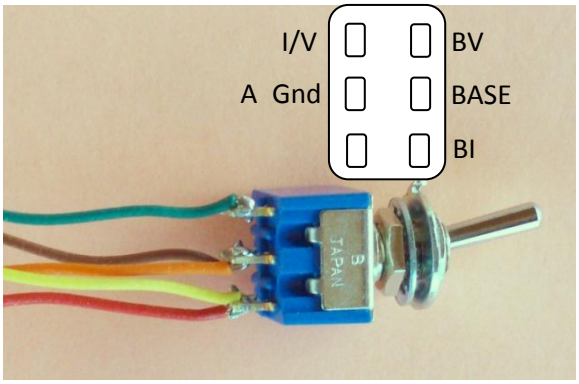
A COLLECTOR binding post wire, 3.25” in length, also needs soldering to the card.



The locations for the digital wire terminations for the rotary switch. Insert the stripped back wires from the card top and solder in place.

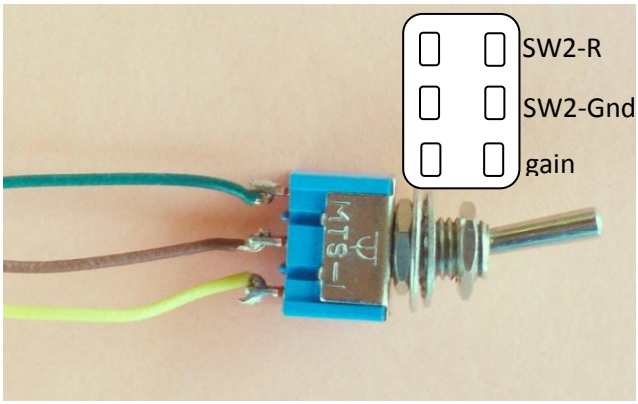


An 1/8” audio jack is used for the footswitch jack. The switch is a normally open push button pedal. Pressing the pedal down is the same as clicking on the software interface to run a trace sweep. This allows hands free operation when probing circuits. The wire lengths are 5.75”.



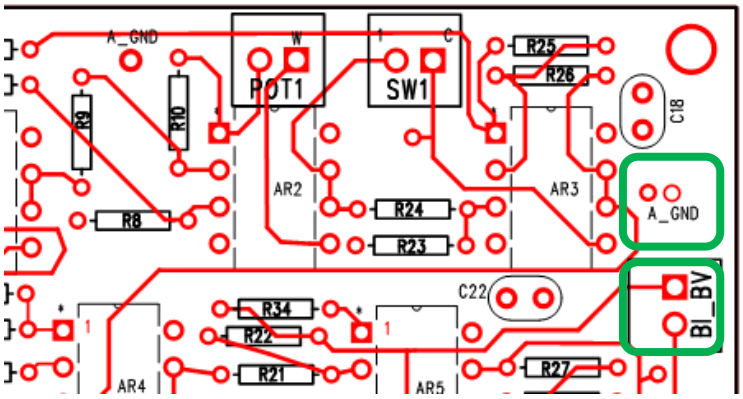
Five wires are required for switch SW3. The BASE wire connects to the front panel binding post. The switch selects between the base voltage step generator and the base current step generator. Switch position is detected using the digital I/V input.

The analog wires are 2.5” in length and the digital wire (I/V) is 3.75” in length.

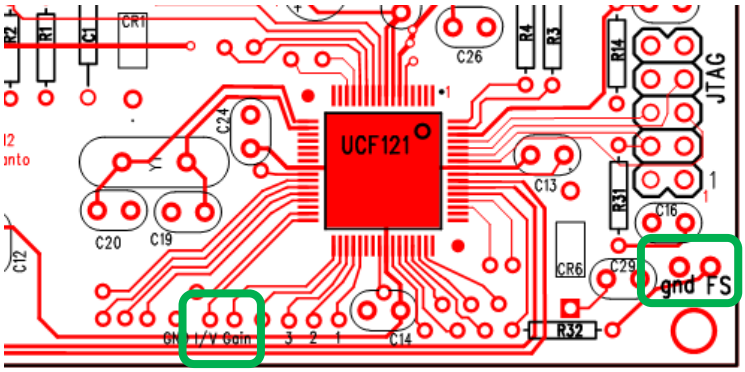


Three wires are required for switch SW2. The switch grounds a resistor on the SW2-R position. This increases the voltage measurement gain from 1X to 4.5X. Switch position is detected using the digital gain input.

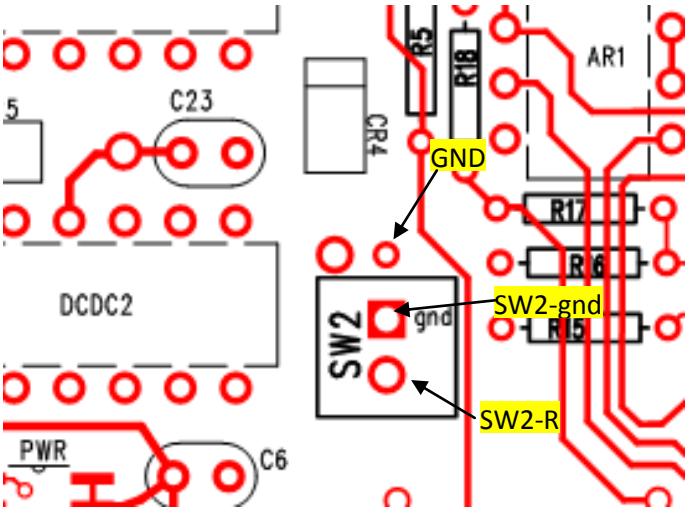
The analog wires are 2.25” in length and the digital wire (gain) is 3.75” in length.



The locations for the analog wire terminations for switch SW3. Insert the stripped back wires from the card top and solder in place.



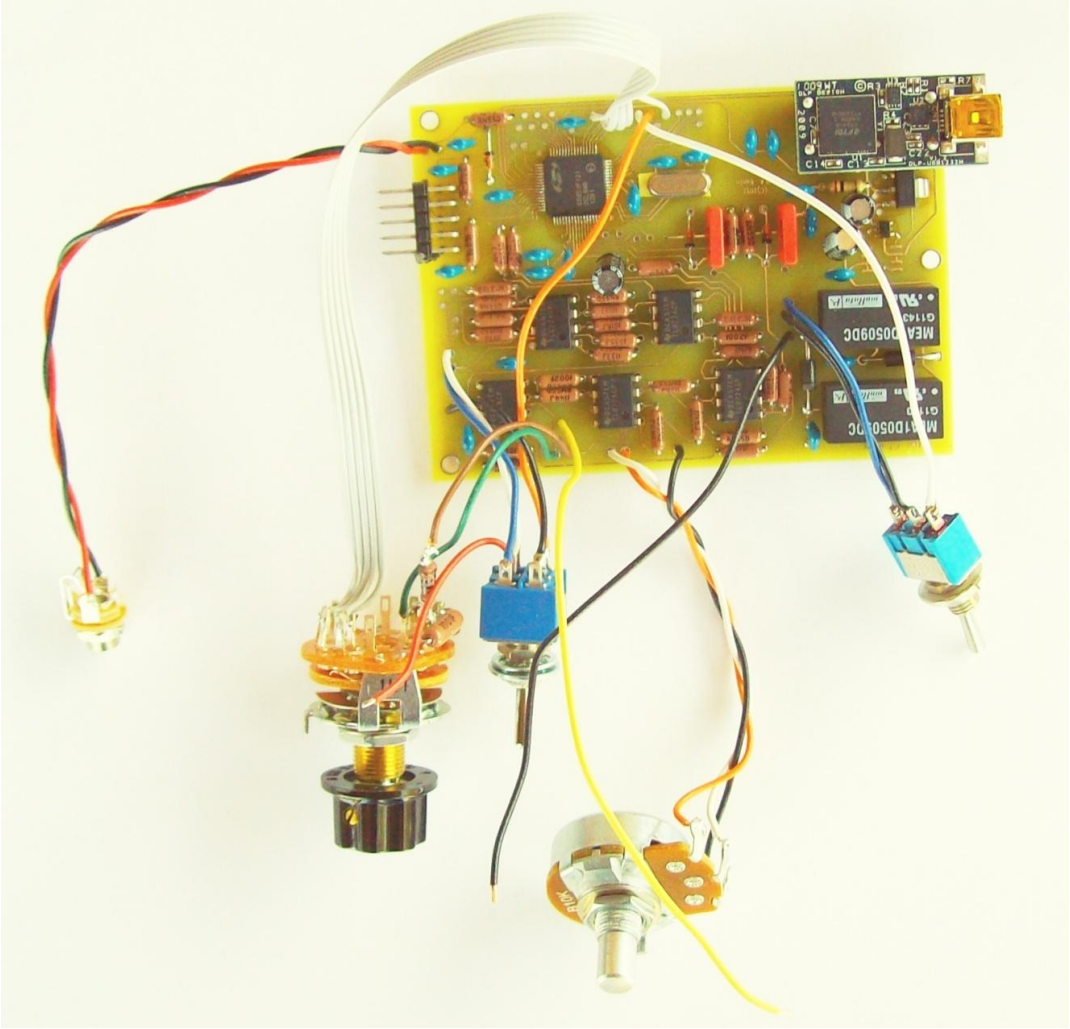
The locations for the digital wire terminations for the foot switch FS and switches SW2 and SW3. Insert the stripped back wires from the card top and solder in place.



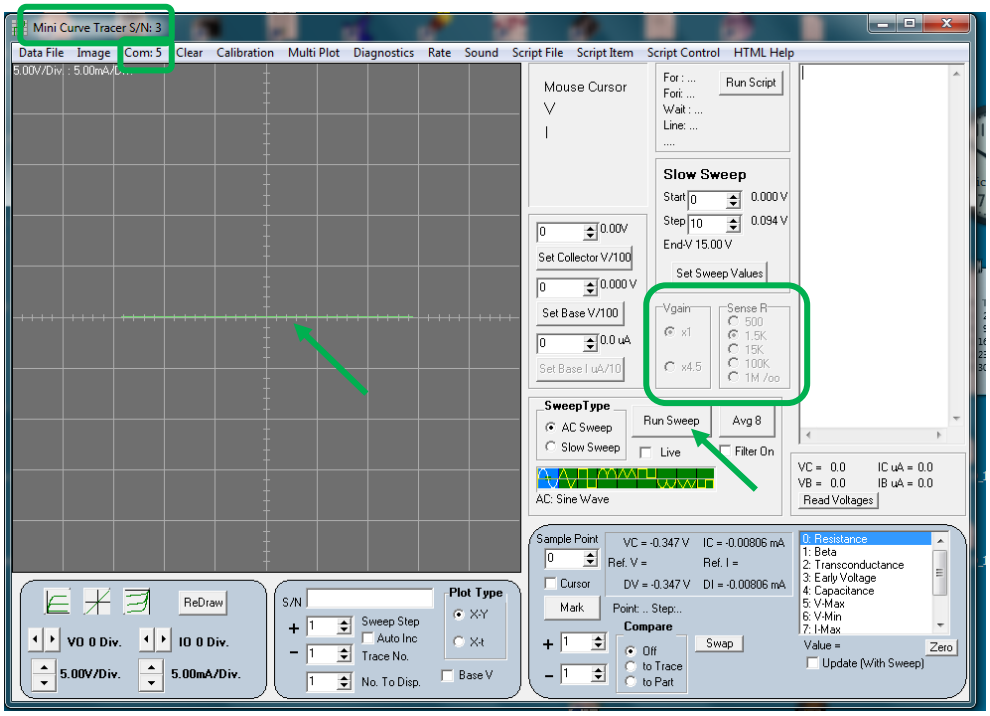
The locations for the analog wire terminations for switch SW2 and binding post GND. Insert the stripped back wires from the card top and solder in place.

The GND binding post wire is 2.75” in length.

17) After the cables and switches are installed, attach a knob to the rotary switch to aid in turning the switch.



a. Separate the cables and switches from the card to prevent shorting to the card and each other.
b. Attach the card via the USB cable to your computer. Note: Sometimes it takes a while for the operating system to recognize the serial interface. The Mini_CT.exe program will not find the card until the operating system identifies the serial interface.
c. Start the Mini_CT.exe program. If the port is present, the program will find and open it. The card serial number, normally 1, will be identified along with the switch positions and the serial port number. If the potentiometer is set to maximum clockwise rotation, then a horizontal line should be displayed. If not, turn the potentiometer clockwise and click the Run Sweep button.



The first test after switches are attached should result in identification of the switch positions and other parameters. Your card may differ depending on the switch settings.

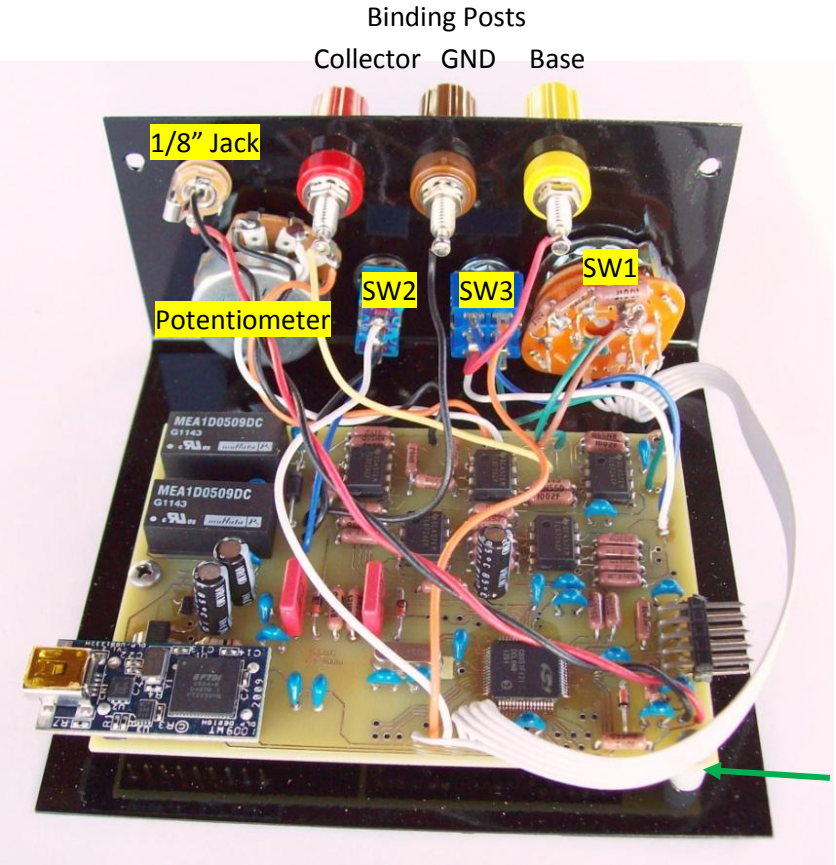
If the horizontal trace is not present, try clicking Run Sweep.

18) Face plate construction
a. Mark and drill the holes shown on the page 2 template. Drill and tap the three holes marked for the card attach points.
b. Place a 90 degree bend at the bend line marked on the drawing. Check the bend direction with image shown on page 1.
c. De burr and clean the face plate. Use a good detergent to remove all oil from the metal. If there is rust on surface, remove it with an oxalic acid based cleanser. Do not use a chlorinated cleanser.
d. Dry and paint the face plate.

19) Attach the card and switches to the faceplate assembly.
a. Three 1/4” long 0.115” I.D. 0.188” O.D. nylon spacers are needed to attach the board to the faceplate. The board is attached using 4-40 screws that are cut to length at about

0.46”. The screw thread length should include the thickness of the card 0.162” plus the length of the spacer 0.25”, the thickness of the sheet metal 0.025” and a little extra.

b. The switches, binding posts, jack and potentiometer are mounted as shown below. Wires to the binding posts are soldered to the post terminals.



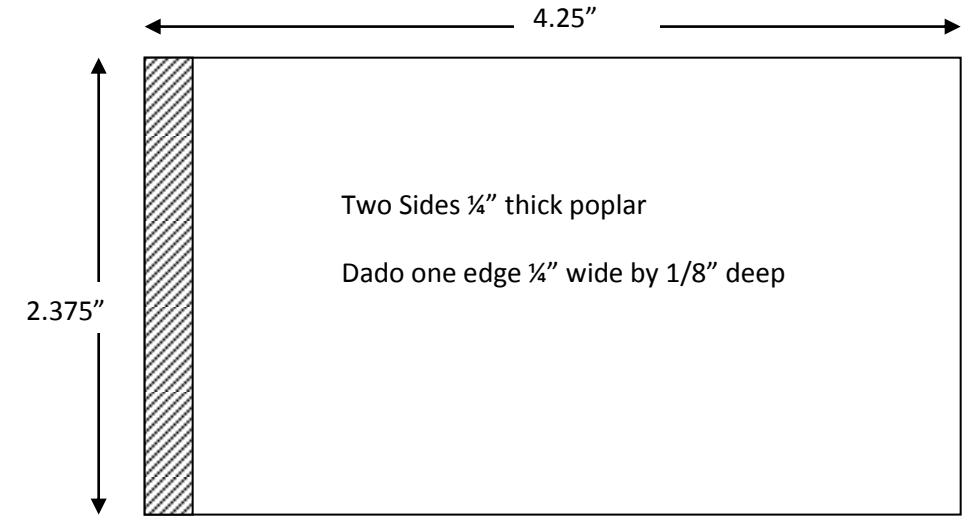
20) Construct the wooden enclosure

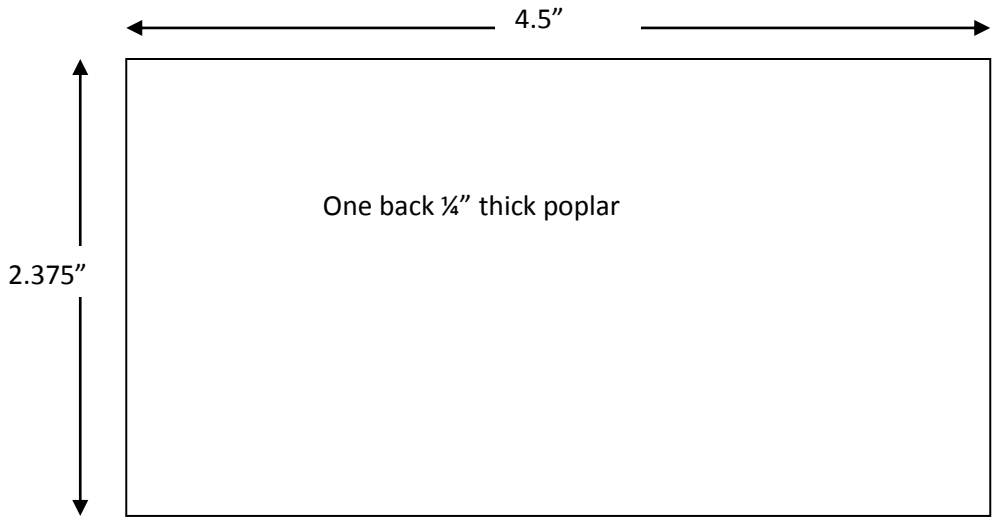
a. Use ¼” thick wood. Poplar works well as it is relatively soft and easy to work with.
b. Cut the wood using the templates shown on the following pages. There will be two identical top/bottoms, two identical sides and one back.
c. Glue the back, bottom and two sides together using wood glue. Clamp and square up the box. The top can be used for clamping but do not glue it in place.
d. Find four small pieces of scrap ¼” wood and cut to about 1”x3/4”. Split the fourth piece with a chisel and glue the split pieces to two of the others.
e. Glue the thicker pieces to the top front of the box leaving room for the top cover and the thickness of the front panel. Glue the thin piece to the back wall with a gap for the chassis to slip under the block.
f. Place the chassis in the box and mark the locations for the front panel screws and for the USB port access hole. Remove the chassis.
g. Measure the USB mark from the inside and transfer to the box outside. Drill and cut the hole to fit you USB cable. Test for fit.

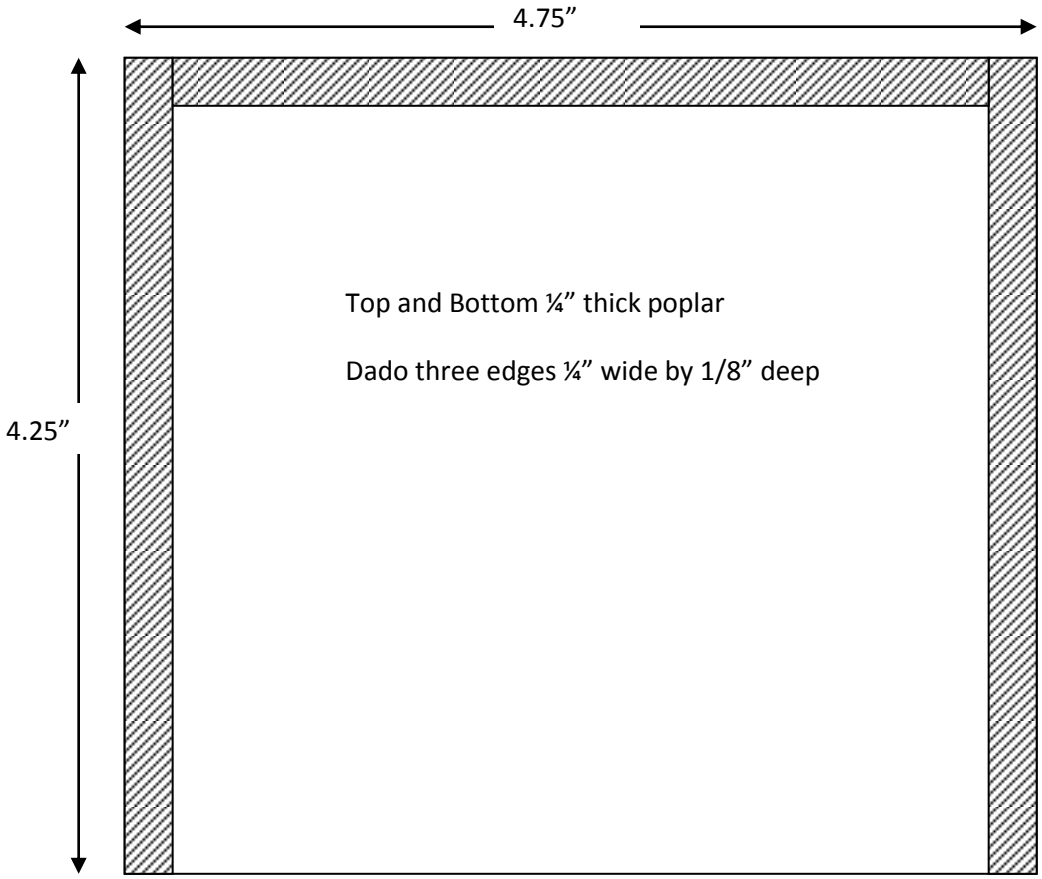
h. Glue and clamp the top onto the box. Sand after drying, stain and varnish. Drill two 0.110” pilot holes in the face plate mounting blocks.

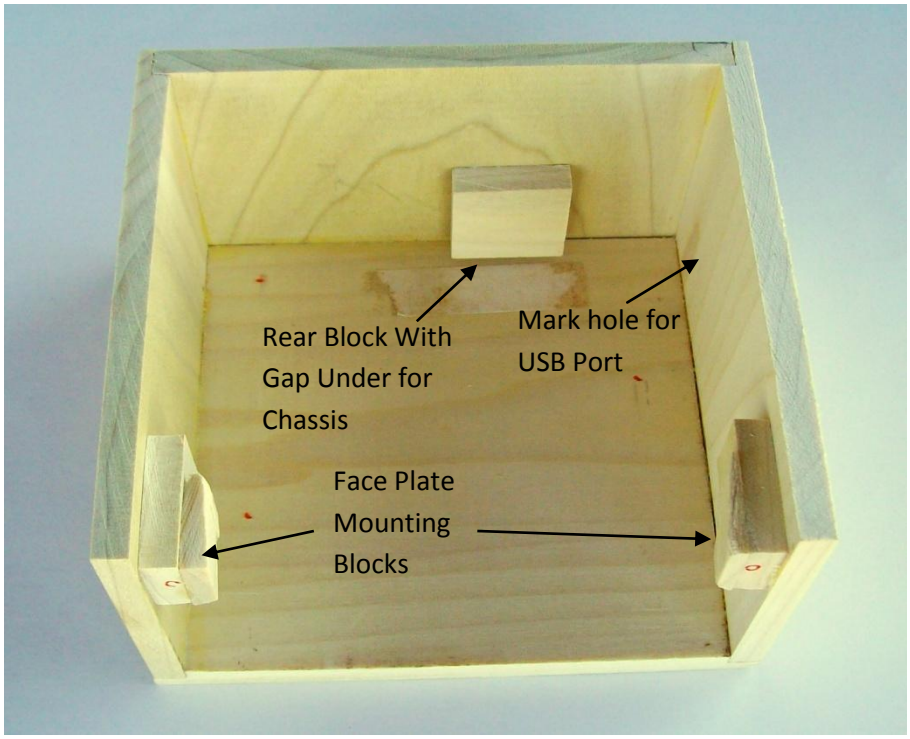
i. Install the chassis and screw the face plate to the mounting block with 3/8” long #4 Pan Head Phillips sheet metal screws.

Wood Box Templates:





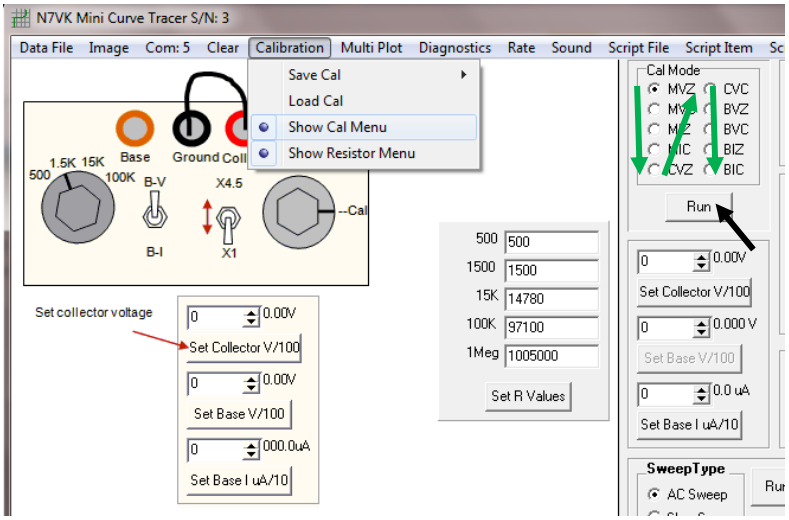

4.75”
4.25”
Top and Bottom ¼” thick poplar
Dado three edges ¼” wide by 1/8” deep



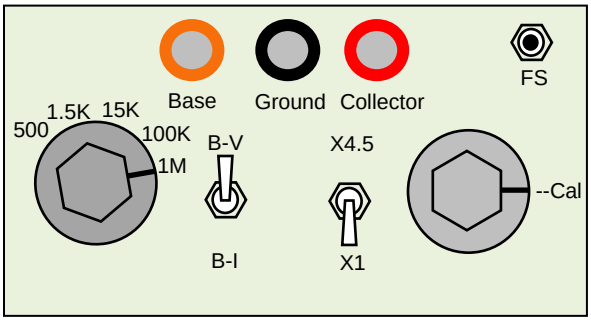
The completed Mini_CT Project

What's next?

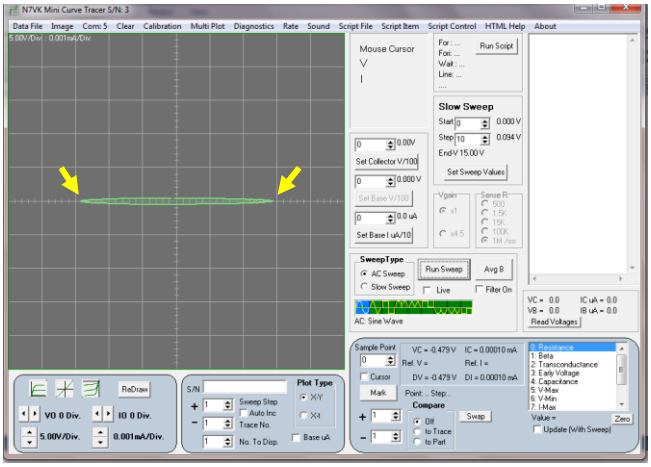
- Make labels and mark the front face plate
  - BASE
  - GND
  - COLLECTOR
  - 500
  - 1.5K
  - 15K
  - 100K
  - 1M
  - B-I
  - B-V
  - 1X
  - 4.5X
  - FS
  - –CAL
- Calibrate the system
  - Plug in Mini_CT
  - Start mini_ct.exe
  - Get your DMM ready
  - Select Menu Item Calibration/Show Resistors
    - Enter the five sense resistor values recorded during the build
  - Select Menu Item Calibration/Show Cal Menu
    - Select the Radio Button Sequence in the Cal Mode Box
      - Start at the Left Top going to the Left Bottom
      - Then, Start at the Right Top going to the Right Bottom
      - Instructions will post on the screen and in a pop up window
      - Press the associated Run button when requested



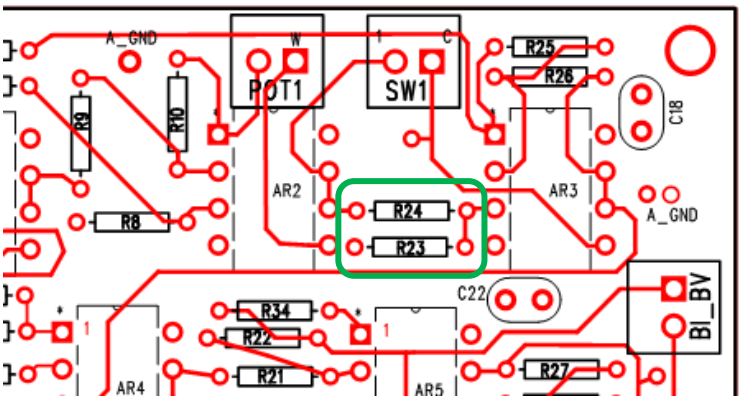
- Select Menu Item Calibration/Save Cal/Yes
- Check the Collector Current Measurement for Common Mode Rejection Ratio
  - Set up Mini_CT as shown



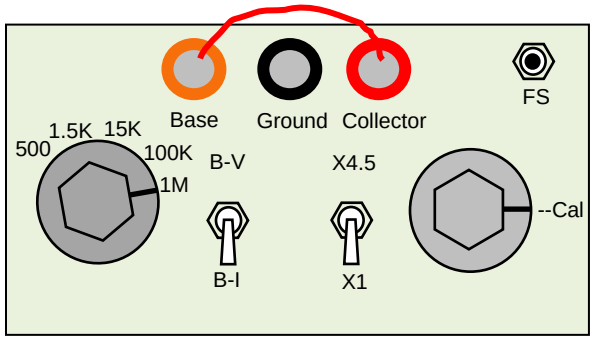
  - Run and AC sweep and set the I/Div. to 0.001mA



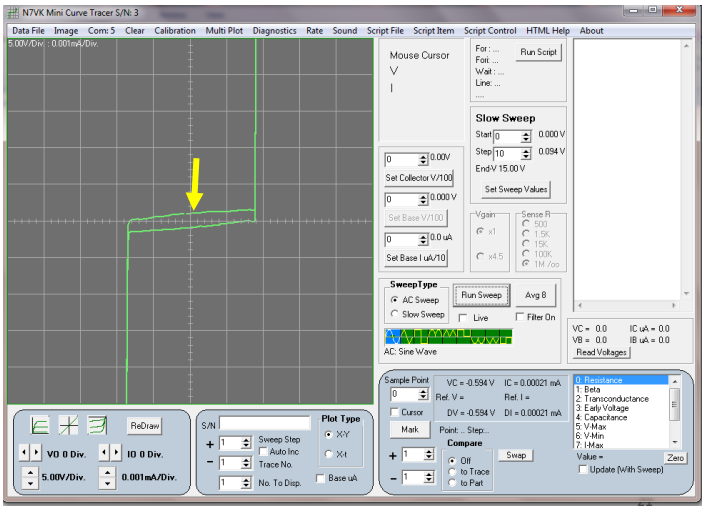
  - The ends of the sweep should be on the zero line. If there is a slope to the curve, then adjustment of CMRR is needed.
  - Adding a parallel resistor to R24 or R23 will rotate the curve to adjust the slope to zero. The value needed will be very large. For example, 10Meg ohms will change the 10K resistor by 0.1%. Use small jumper clips to find for the correct resistor value and placement.



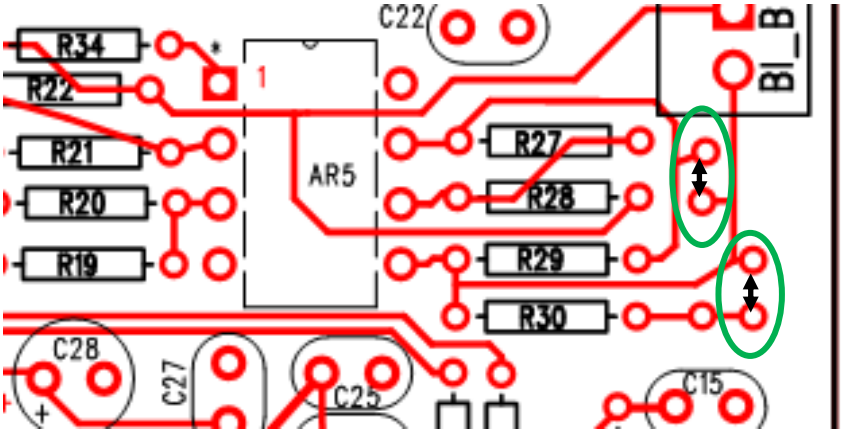
- Check the Base Current Source Voltage balance
  - Set up Mini_CT as shown



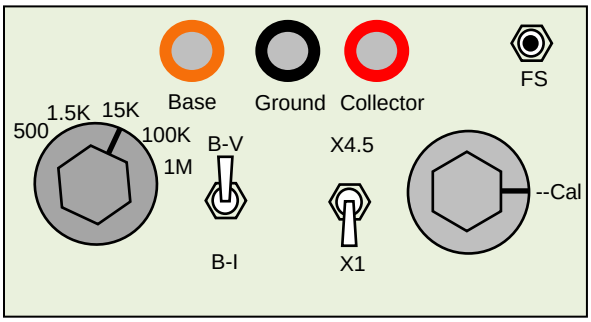
  - Set the Base Current to zero and Run an AC sweep. Set the I/Div. to 0.001mA



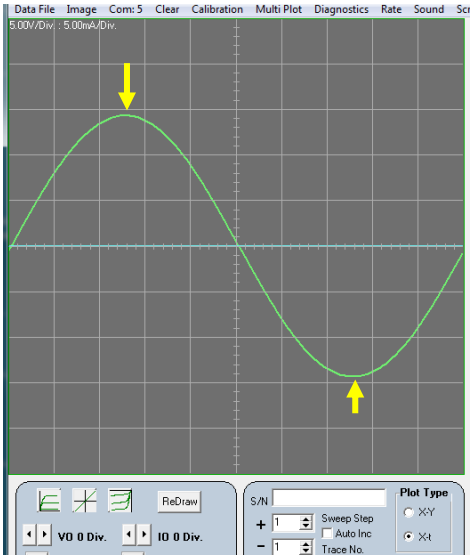
  - The Current trace slopes should be close to zero. If not, add a parallel resistor between one of the following locations. As before, the value will be in the Meg ohm range and it will require trial and error.



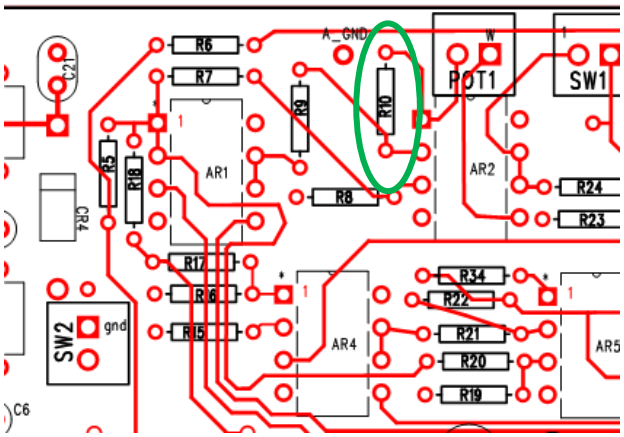
- Check the Collector Voltage Generator for Clipping
  - Set up Mini_CT as shown



  - Set the Sweep Type to AC Sweep and AC Sine Wave then run a sweep. In the X-t mode, check the waveform top and bottom for clipping.



  - If clipping is present, the amplitude can be dropped by adding a resistor in parallel with R10. Try values that are several megohms.



- Rerun the Calibration procedure after any circuit modification

- Disclaimer
    - This instruction document is a guide on how to build a Mini_CT curve tracer
    - It does not warrant you from harm, liability and damages of any kind
        - Including design, construction and use
    - It is your responsibility to work safely
    - It is your responsibility to recognize and avoid potentially lethal circuits and conditions when using Mini_CT
        - If it has more than 10V and can supply more than 10mA, then avoid connecting Mini_CT to it
    - The Mini_CT design has not been safety tested
    - Although all Mini_CT curve tracers built to date have functioned as designed, there is no guarantee of success
        - Don't give up if it doesn't work. Look for problems.
        - The causes of Mini_CT failing to function may be the result of many factors
            - Failed components
            - Incorrect component orientation
            - Incorrect part placement
            - Incorrect wiring placement
            - Environmental stress
                - Excessive soldering heat
                - Mechanical stress
                - Electrostatic Discharge Damage
            - Solder problems
                - Cracked or cold joints
                - Solder bridging or shorts
                - Lifting of traces or pads
            - Computer problems
                - Incompatible hardware
                - Incompatible operating system
                - Incorrect drivers
                - Software errors