# BCRA SOUND CARD INTERFACE INSTRUCTION MANUAL



By Edwin T. Scallon, NN1ES

### ((Q)) WAIDGW WIACT

#### INTRODUCTION

The first sound card was introduced in 1981 by IBM for their PC. The IBM Music Feature card was a significant advancement in computing technology, allowing users to add sound capabilities to their computers for the first time.

This innovation laid the foundation for the development of modern sound cards, which have since become standard components in most computers. Since the invention of sound cards, significant technological advancements have taken place. Early sound cards primarily focused on basic audio playback capabilities, while modern sound cards now offer high-fidelity audio output, advanced noise cancellation, surround sound processing, and support for virtual reality audio technologies.

The integration of digital signal processing (DSP) and dedicated audio processors has also improved sound card performance and enabled the implementation of advanced audio effects and enhancements. Additionally, the introduction of external USB sound cards has provided flexibility and portability for users seeking high-quality audio solutions on the go.

PSK-31, historically was the first Computer => Radio // Radio=>Computer program used by Amateur Radio digital enthusiasts.

PSK-31 was developed and named by English amateur radio operator Peter Martinez (call sign G3PLX) and introduced to the wider amateur radio community in December 1998.

PSK31 or "Phase Shift Keying, 31 Baud", also BPSK31 and QPSK31, is a popular computer-sound card-generated radio-teletype mode, used primarily by amateur radio operators to conduct real-time keyboard-to-key board chat, most often using frequencies in the high frequency amateur radio bands (shortwave). PSK31 is distinguished from other digital modes in that it is specifically tuned to have a data rate close to typing speed, and has an extremely narrow bandwidth, allowing many conversations in the same bandwidth as a single voice channel. This narrow bandwidth makes better use of the RF energy in a very narrow space thus allowing relatively low-power equipment (5 watts) to communicate globally using the same skywave propagation used by shortwave radio stations.

Since the introduction of PSK-31 our sound card interface has morphed into an "all digital mode" item capable of not only PSK-31 but also, PSK-63, RTTY, SSTV, NBEMS, CW plus many other modes on your HF or VHF/UHF rig.



# **ACKNOWLEDGMENTS**

The Bristol County Repeater Association
Takes Pride in Acknowledging the Following
Individuals for Their Contribution to the BCAR
Sound Card Interface Project:

- Wheat Kelley, N1YCQ
- Mike Joseph, KC1LIK
- Skip Denault, KB1CN
- Kevin Spratt, N1KJS



## INSTALLATION OF COMPONENTS ON THE DIGITAL SOUND CARD INTERFACE

#### = CHAPTER ONE THE INSTALLATION =

#### COMPONENTS INCLUDED IN THIS KIT:

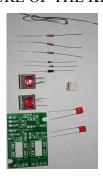
#### BELOW ARE PICTURE OF THE KIT

- 1 1N4001 DIODE
- 2 1N4148 DIODES
- 2 2.2K Ohm RESISTORS
- 1 220 Ohm RESISTOR
- 1 4N25 OPTO -COUPLER
- 2 .1 uF (MFD) CAPACITORS
- 2 600:600 AUDIO TRANSFORMERS

12" ROSEN CORE SOLDER

1- PRINTED SOUND CARD PRINTED CIRCUIT BOARD - Hereafter (PCB)





### TOOLS REQUIRED FOR COMPONENT INSTALLATION

#### LIST:

- Soldering Iron Rated at 25- 45 watts
- Helping Hands To Hold Board During Soldering
- Wire sniper, wire striper
- Solder Supplied In Kit

The solder we are using is identified as follows:

Flux: RA300 Rosin flux. Classification: ROM1

Flux core content 2.2%

Gauge: 0.81mm (0.032') Std wire gauge: 21 Alloy: Lead 93.5% Tin 5.0% Silver 1.5%

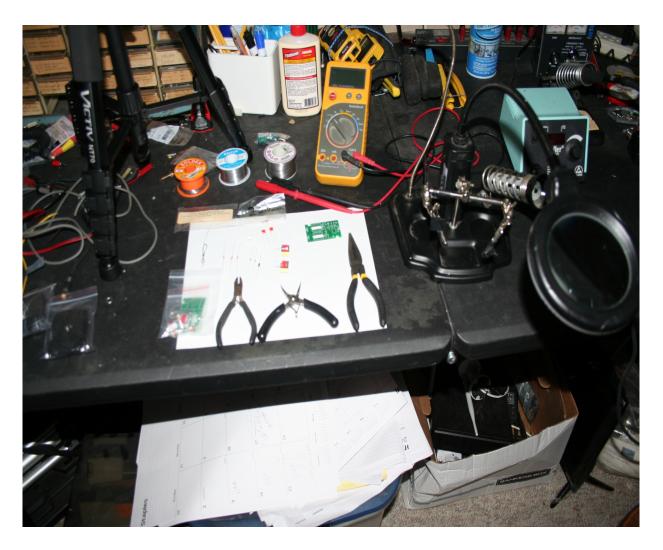
Melting Range: 296 - 301°C Residue removal: Not required.

The Kit will be supplied FREE to Boy Scouts During PCB "Build Out" AND FREE to all NEW MEMBERS who join the Bristol County Repeater association (BCRA)<sup>1</sup> All tools will be supplied to the Scouts and new members for use during the "Build Out Sessions

If you Purchased a Kit from BCRA and DID NOT join the Kit will cost \$ 45.00 and you will need to purchase a soldering iron sufficient for most printer circuit board construction. They can be found either on Amazon soldering tips and a stand for \$9.95 or at Harbor Freight (Chicago Elec 30 watt soldering Iron Item #2527 for \$5.99) Helping hand at Harbor Freight (HFT Helping Hands Item # 1763 for \$4.99)

<sup>&</sup>lt;sup>1</sup> Bristol County Repeater Association https://bcra.club/bcrawprel/bcra-membership-form/

### THE TOOLS THAT YOU MAY WANT TO EMPLOY WHEN INSTALLING COMPONENTS IN THE PCB



There are 2 pair of needle nose plyers, one "regular" sized and the other smaller one is a spring loaded mini needle nose plyers. There is also a mini diagonal wire cutter commonalty refereed to as a diag wire snip. The closer you can trim the excess leads after soldering the several components the better your board will look and it will look professionally built.

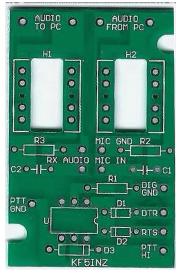
Next is a pair of helping hands and a VOM which stands for Voltage Ohm Meter. This is the device you will see used during the installation of the components and also pictures in this booklet.

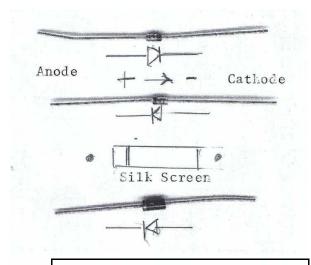
### STEP BY STEP ASSEMBLY INSTRUCTIONS FOLLOWING THE VIDEO

#### STEP 1 - INSTALL DIODES IN D1 and D2

The first installation will be the DIODES (D1 and D2). Now look at the video and the below picture of the Interface Circuit Board the Diodes next to the Board:

The two Diodes Below are D1 & D2 1N4148





This Is D3 1N4001 to Be Used Later

Note: THE TWO TOP DIODES HAVE STRIPES THE TOP DIODE'S STIPE IS ON THE

RIGHT SIDE AND THE DIODE DIRECTLY BELOW IT HAS ITS STIPE IS ON THE LEFT. THESE DIODES ARE IDENTIFIED AS 1N418A STANDARD P-N JUNCTION DIODES USED FOR SWITCHING HIGH SPEED RECTIFICATION AND PROTECTION OF TELECOM CIRCUITS.

WHY DIODES? THEY PERMIT CURRENT TO RUN IN ONLY ONE DIRECTION. IF YOU INSTALL THE 1N418A DIODES WITH THE STRIPE NOT MATCHING THE SCREEN PRINT STIPES ON D1 AND D2 THEY WILL NOT PERMIT CURRENT TO FLOW IN THE CORRECT DIRECTION. HERE WE WANT THE CURRENT TO RUN FROM THE DTR AND RTS TO PIN 1 OF THE OPTO-COUPLER WHICH IS THE BASE OF THE INTERNAL LIGHT EMITTING DIODE (LED).

THE BOTTOM DIODE IS A 1N4001 DIODE AND WE WILL BE INSTALLING IT LATER ON IN THE INSTALLATION.

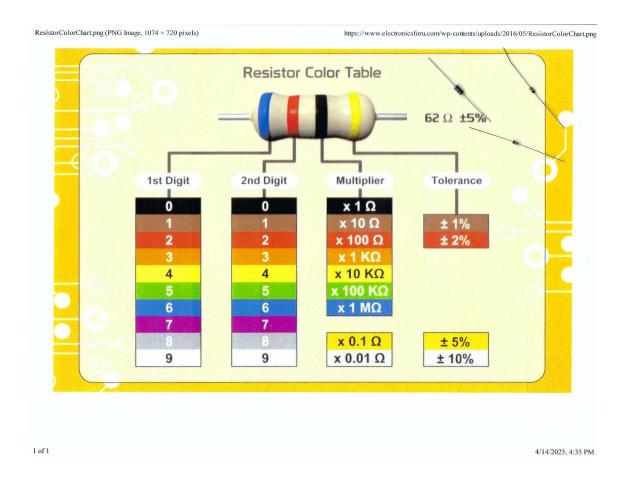
NOW Install the 1N418A diodes both in the D1 position and the D2 position making sure that you install the diodes paying particular attention to the stripes on the diodes with the stipes silk screened on the circuit board itself.

Watch the video for the correct installation of Diodes D1 and D2.

#### STEP 2 - INSTALL TWO 2.2K Ohm RESISTORS

WE WILL INSTALL THE TWO 2.2k Ohm RESISTORS IN THE CIRCUIT BOARD DESIGNATED AS R2 AND R3. They are located just below the H1 and H2 SCREEN PRINTS for th Audio Transformers. THEY DO NOT HAVE POLARITY AS THE DIODES SO INSTALLING IN EITHER DIRECTION IS ACCEPTABLE.

Note: In the package of parts, the original 2.2k Ohm resistors as well as the 220 Ohm resistors were not color coded but just had stipes. Building circuit boards one must use the correct value resistor. Below is a Color Code for the Resistor Values. We will use this method first to identify the 2.2K Ohm resistors and then use an Ohm Meter to confirm the selection of resistors for installation in R2 and R3.



#### USING THE RESISTOR COLOR CODE TABLE DETERMINE THE VALUE OF THIS RESISTOR

RESISTOR VALUE DETERMINED BY ITS COLORED STRIPES . . . .

THIS RESISTOR HAS:

A RED STRIPE IN THE FIRST DIGIT

A RED STRIPE IN THE SECOND DIGIT POSITION

AN ORANGE STRIPE IN THE THIRD DIGIT POSITION

THE FOURTH BAND IS THE TOLERANCE OF THE RESISTOR WHICH IS +/- 1%

2.2k Ohm 1% will have a value:

Between 2.178 k 2.2K -1% And 2.222 k 2.2K +1%

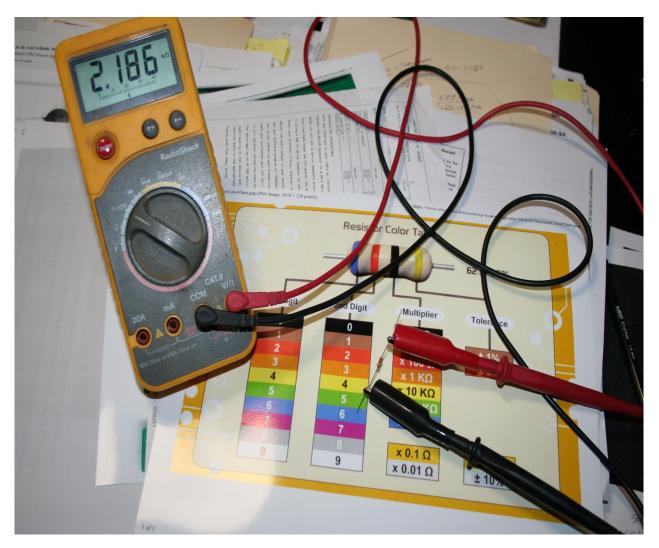
THEREFORE, THE ABOVE RESISTOR IS A 2.2K Ohm Resistor

SCHEMATIC OF RESISTOR:

PCB SILK SCREEN SYMBOL

The most sure way to determine the value of a particular resistor is to use an Ohm meter to check the value. We will now verify what we have just determined based on the Resistor Chart on the previous page.

Now we will use an Ohm meter connected to the same Resistors to make sure they are in fact 2.2k Ohm.



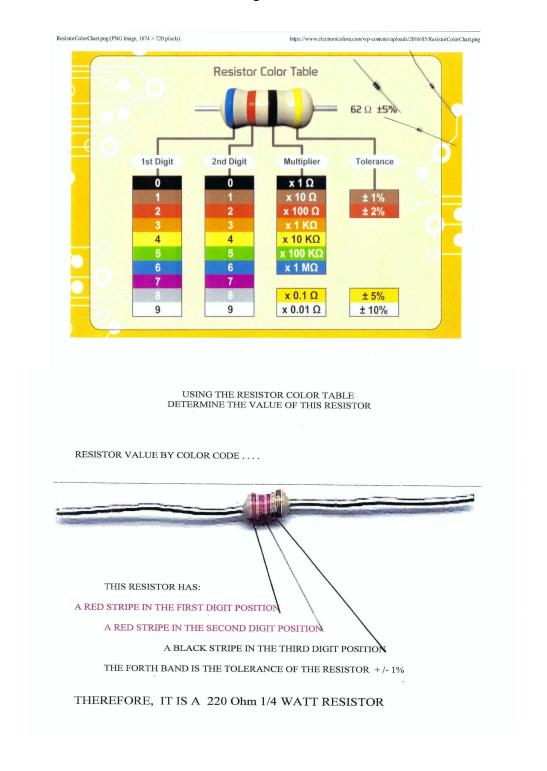
NOTE: IN THE UPPER RIGHT CORDER OF THE OHM METER, YOU WILL SEE THAT THE VALUE IS 1K FOR THE 2.2K Ohm RESISTOR. THE ACTUAL RESISTOR IS A 2.186k Ohm RESISTOR, WHICH IS WITHIN CORRECT TOLERANCE.

So now we can install both 2.2k Ohm resistors in R2 and R3, what the video installing the resistors.

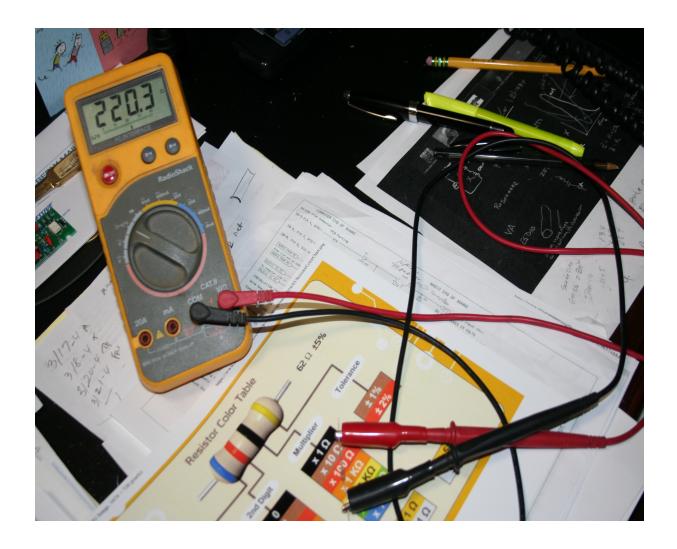
Watch the video for installation of the 2.2k Ohm Resistors

#### STEP 3 - INSTALL THE 220 Ohm Resistor

We now turn our attention to the 220 Ohm resistor. Again we will use the Resistor Value Color Code chart to make sure we are installing a 220 Ohm resistor in R1.



Again lets verify this resistor is in fact a 220 Ohm resistor using an Ohm Meter:



We can clearly see the decimal point after the Zero (0) and in the top right hand corner the Ohm symbol 'I' present without the use of the K Ohm symbol, therefore this resistor is a 220 Ohm resistor.

We then can now install the 220 Ohm resistor in R1 which connects the DIG GRD to PIN #2 OF THE OPTO-COUPLER WHICH IS THE CATHODE OF THE INTERNAL LIGHT EMITTING DIODE.

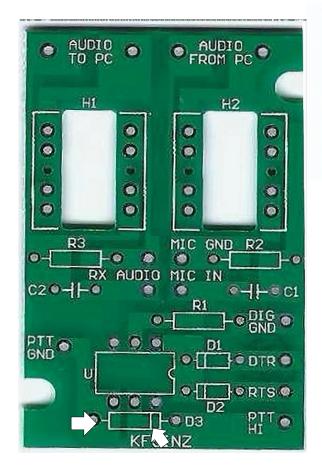
#### View the video of the installation of R1 on the Circuit Board.

220 Ohm Resistor w/ -1% tolerance >217.8 Ohms 220 Ohm Resistor w/+1% tolerance < 222.2 Ohms

#### STEP 4 - INSTALL THE 1N4001 DIODE IN D3

NEXT we will install the 1N4001 DIODE IN THE D3 POSITION on the Circuit Board.

#### LOOK AT THE CIRCUIT BOARD:





D3 1N4001 Diode

THE OPTOCOUPLER IS
Identified on the Silk Screen as a
'U' on the PCB

THE D3's POSITION IS JUST BELOW THE 6<sup>th</sup> PIN OF THE OPOTO-COUPLER AND CONNECTS TO PIN #5 OF THE OPTO-COUPLER'S BASE. THE OPTO-COUPLER PIN #5 ALSO CONNECTS TO THE CATHODE of D3 and PTT GND. D3'S Anode is also connected to the OPTO-COUPLER's Pin #4.

WHY USE THE DIODE IN THAT POSITION. Look at the stipe on the Diode and the stipe on the silk screen diagram on the circuit board, it is opposite that of D1 and D2, so as we know, it prevents current flowing from the PTT HI but allows the PTT GND and to Pin 1 of the Opto-Coupler

Watch the video of the installation of the 1N4001 DIODE.

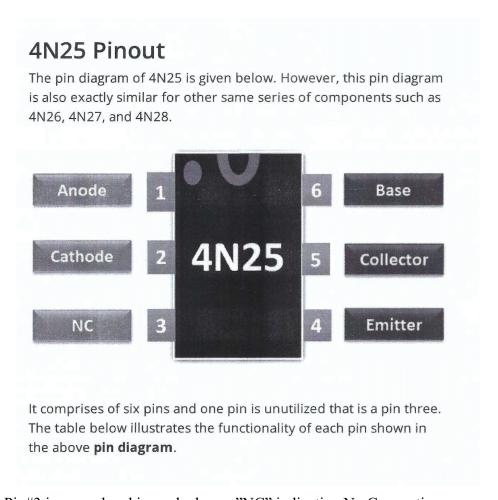
WHY USE THIS DIODE? The 1N4001 Diode is a silicon rectifier which blocks current or voltage it is usually used in AC Circuits to convert AC to DC. It actually is a free wheeling diode to protect the circuit. Again allowing current to flow in only one direction.

#### STEP 5 - INSTALL THE OPTO-COUPLER

NEXT is the installation of the 4N25 OPTO-COUPLER, 'U'. This device needs some explaining. There are SIX PINS ON THE OPTO-COUPLER. EACH PIN CORRESPONDS TO A POSITION OF EITHER A LIGHT ADMITTING DIODE OF THE ANODE, BASE, COLLECTOR OR CATHODE OF THE INTERNAL EMITTER TRANSISTOR.

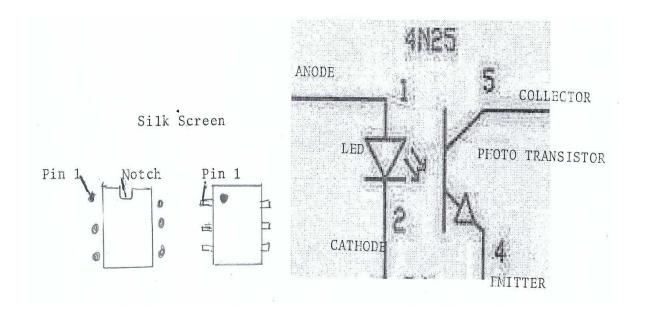
#### LETS LOOK AT THE OPTO-COUPLER:

An Optocoupler is an integrated Circuit (IC) that combines an LED and a Photo Transistor



Pin#3 is unused and is marked as an"NC" indicating No Connection.

#### THE SCHEMATIC DIAGRAM OF THE OPTO-COUPLER IS SHOWN BELOW:



Although the OPTO-Coupler we are installing has 6 "legs" or "Pins" we will be soldering all 6 of the leads (legs) to the circuit board, however, we will be only utilizing 4 legs as shown in the above schematic diagram.

The 4N25 OPTO-COUPLER IS A WIDELY USED DEVICE THAT ALLOWS ONE TO CONNECT AND CONTROL CIRCUITS WHILE KEEPING THEM ELECTRICALLY ISOLATED. THE ISOLATION IS ACHIEVED USING AN INFRARED LED AND A PHOTO TRANSISTOR.

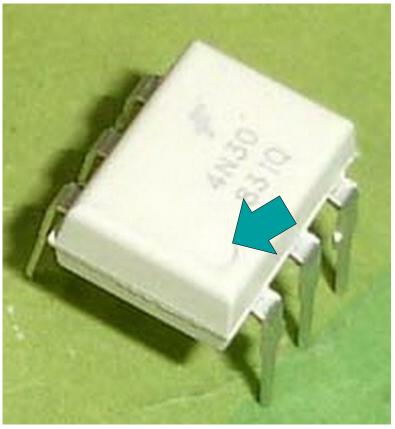
When current flows from the anode to the cathode through the LED, it emits light that activated the PHOTO TRANSISTOR, enabling current to pass in the second circuit. This setup ensures safe communication between circuits that may have different voltage levels, protecting sensitive components from damage.

In our application we do have two circuits, the first from the computer RS-232 and the second from the Radio. Eventually, we will connect the RS-232 connector to the DC-9 serial port of the PC, we most likely will use a USB to serial adapter. This interface needs the RTS line to go logic High at least 5 volts in order to activate the OPTO-COUPLER.

We will address the communication ports and their installation in the second video and Second Chapter, that being the installation of the leads to the radio and computer.

NOW the installation of the OPTO-COUPLER is not TOO complex, however, it does take some particular attention to the top of the device.

ON THE TOP OF THE 4N25 OPTO-COUPLER THERE IS AN INDENTATION, IT LOOKS LIKE A FAINT DOT. IT IS CRITICAL YOU FIND THE DOT AND ALIGN IT WITH THE SILK SCREEN CUTOUT ON THE CIRCUIT BOARD OPPOSITE END OF THE SCREEN PRINT OUTLINE FROM THE 'U'. YOU WANT TO INSTALL THE OCTO-COUPLER WITH THE DOT ON THE SAME SIDE AND IN THE SAME POSITION AS THE INDENTATION ON THE CIRCUIT BOARD. THE DOT SIDE SHOULD BE ON SAME OPPOSITE SIDE OF THE CIRCUIT BOARD AWAY FROM THE 'U' SYMBOL ON THE CIRCUIT BOARD.



I fully realize that this is not a 4N25 OPTO-COUPLER BUT IT IS DEMONSTRATIVE OF A DOT ON THE BOTTOM RIGHT CORNER OPTOCOUPLER.

Lets install the Optocoupler (I have PREVIOUSLY hyphenated the spelling of the word Optocoupler simply to reinforce the point that this device has a photo emitting diode internally.

Watch the video for the installation of the Optocoupler.

12

#### STEP 6 - INSTALL THE TWO 0.1 uF CAPACITORS

NEXT ARE THE 2 CAPACITORS. The installation of these components is rather simple and there are no specific directions as to their installation. But before we install, what is the reason we use a pair of capacitors.

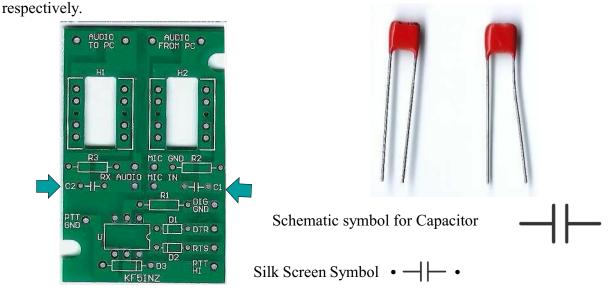
WHY USE CAPACITORS? In the world of electronics; a tiny component often plays a pivotal role: the 0.1 uF capacitor, lie the silent guardians of our devices, these capacitors ensure stable operation by filtering out unwanted noise. From smart phones, to complex industrial machinery, understanding the 0.1 uF capacitor is essential for anyone diving into the fascinating realm of circuits and electrical engineering.

When a digital circuit switches states, it draws current in rapid pulses, causing voltage fluctuations on the power supply line. Decouple these fluctuations that can cause unstable behavior of the IC and neighboring components. By placing a 0.1 uF capacitor close to the power pin of an IC, it acts like local charge storage, smoothing out the voltage spikes and providing a stabile voltage for the IC to operation.

In addition to decoupling, 0.1uF capacitors are also used for by passing high-frequency signals to ground. These capacitors present a low impedance path for high frequency noise signals to go to ground, reducing the impact of this noise on the desire signals.

Here, we are using the capacitors to eliminate any noise on our signal since the eventual digital picture will show any and all noise instead of good solid waterfall highlights of incoming signals.

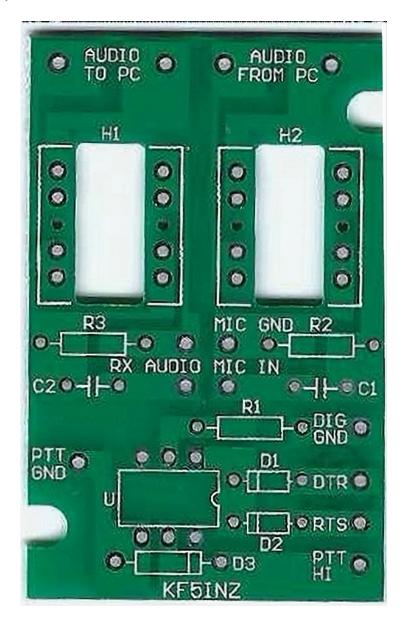
Install the capacitors in C1 and C2 which are DIRECTLY BELOW the R2 and R3 resistors



Watch the video for the installation of the two 0.1 MFD capacitors.

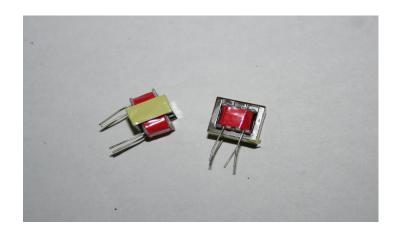
#### STEP 7 - INSTALL BOTH AUDIO TRANSFORMERS

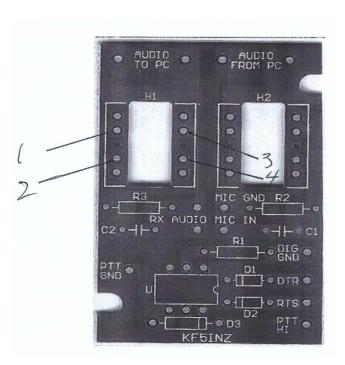
LASTLY, we will talk about the audio transformers and their installation of the circuit board.



You will note that the Audio Transformers only have 4 Pins, leads or "Legs". However, in H1 and H2 there 8 locations for leads. We will be only using the middle two pads.

Below is a visual placement page where the audio transformers will be installed on the Printed Circuit Board.







Install Transformer in this position Vertical



DO NOT INSTALL SIDEWAYS HORIZONTAL

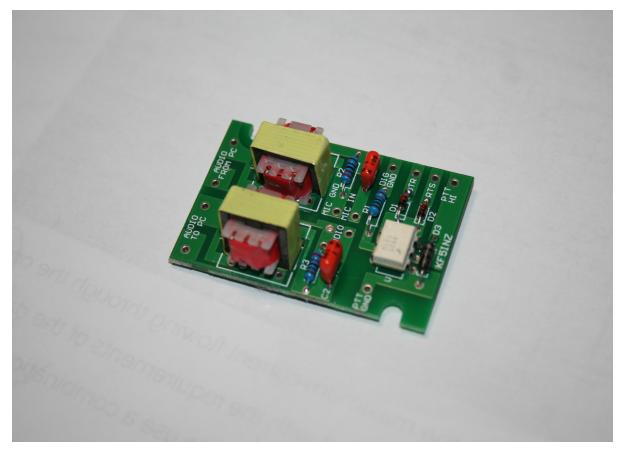
These are the audio transformers which we will install in the Printed Circuit Board paying particular attention to the below pin positions for the 4 leads.

Use only the two center pads for the audio transformers seen above for H1. Likewise install THE OTHER audio transformer ON THE RIGHT SIDE IF THE CIRCUIT BOARD PICTURED, in H2 using the four companion holes as we did installing the transformer in the four center holes of H1.

#### NOW INSTALL THE AUDIO TRANSFORMERS - WATCH THE VIDEO

We are done. The finished Sound Card Interface Circuit Board should look like the one below:

CONGRATULATIONS ON SUCCESSFULLY INSTALLING ALL THE COMPONENTS ON YOUR SOUND CARD INTERFACE BOARD !!!



YOU NOW HAVE COMPLETED = CHAPTER ONE =

- = CHAPTER TWO = WILL BE COMPLETING THE CONSTRUCTION AND INSTALLATION OF THE LEADS FOR BOTH THE COMPUTER DB-9 AND THOSE OF THE RADIO.
- = CHAPTER THREE = WILL BE THE INSTALLATION OF THE SOFTWARE TO INTERPRET THE SIGNALS FROM THE RADIO TO THE COMPUTER AND VICEVERSA SENDING THE CHARACTERS TO THE RADIO FROM THE COMPUTER.