

Amplifone HV and Deflection FAQ and Guide  
by Michael Kelley  
Version 1.2  
March 28, 2002

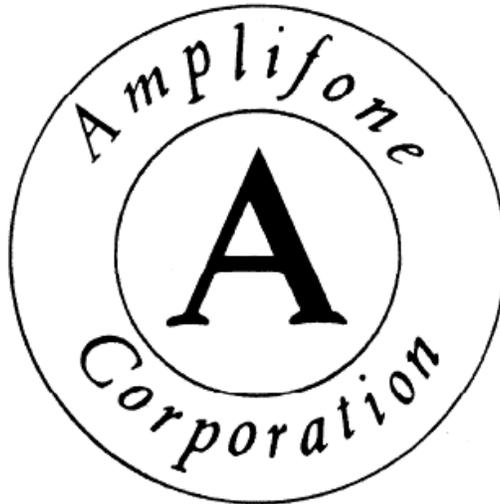


Table of Contents

INTRODUCTION.....2  
AMPLIFONE VECTOR MONITORS .....3  
AMPLIFONE HIGH VOLTAGE (HV) BOARDS .....4  
AMPLIFONE DEFLECTION BOARDS .....32  
AMPLIFONE TROUBLESHOOTING WAVEFORMS .....38  
AMPLIFONE DEFLECTION PROBLEMS AND SOLUTIONS .....40  
AMPLIFONE PICTURE TUBES AND DEFLECTION YOKES .....41  
MONITOR REPAIR LOGS .....42  
OFFICIAL ATARI AMPLIFONE DOCUMENTS .....46  
Appendix A: Common Ground Connections .....52  
Appendix B: Testing Transistors.....53

---

Copyright (c) 1999, 2002 by Michael Kelley. You may freely distribute this document, but it may not be sold for profit or personal gain. I make no warranty about any of the information in this document, and assume no liability for damage incurred with its use to person or property. No affiliation with the Amplifone Corporation is expressed or implied.

---

## **INTRODUCTION**

---

I wrote this with the help of several people, and hopefully this will be of some assistance to those who decide to restore Amplifone monitors for their Atari Color Vector games. Gregg Woodcock's Vector Faq is excellent; I learned a great deal about Wells-Gardner Color X/Y monitors. However, it goes into little depth in covering Amplifone HV and Deflection boards. That is why you are reading this document. If you have any corrections or comments, please let me know. I'll do my best to help you if you're having troubles as my schedule permits. Please try to understand, though, I work 3 jobs. (Why do you think it took me so long to write this thing?) My e-mail address is mkdud@aol.com.

Please try to read through the ENTIRE faq to determine your problem. I think I have covered every problem I've seen with the Amplifone boards in the faq. If you have anything to add, please let me know!! I'm always up for comments and suggestions, and would always like to hear more info on the historical side of things as far as Atari goes. Some day, I would like to write a book about the Atari coin-op division.

### **DISCLAIMER**

**Working on monitors can be dangerous. You are dealing with high voltage equipment, as well as a potential risk of implosion of a picture tube. I take no responsibility for damage to yourself, or your equipment. If you are unfamiliar with high voltage systems, DO NOT put yourself in danger. Get someone qualified to work on it, and watch and learn from them. I suggest Randy Fromm's videotapes on Monitor Repair to learn the basics and proper safety steps for working on monitors. If you choose to jump in and work on your monitor without any knowledge of what you are doing, I can assure you, you will damage your equipment, and quite possibly yourself.**

### **Acknowledgements**

I would like to thank these people: Joe Welser (who wrote the first known Amplifone HV guide, and helped me several times with great advice), Gregg Woodcock (who gave me a greater understanding of Vector monitors through his faq), John Robertson (who developed the WG6100>Amplifone HV mod), Randy Fromm (for being the greatest tech in the world, as well as a great teacher - "Grasshopper, you crazy to mess with the Amplifones!"), Jack Hammond (who worked at Amplifone), Bret Pehrson (who actually used this guide), Scott Evans for the Amplifone waveforms, Joe Bachmann, Zonn Moore and last, but certainly not least, Ed Henciak, who was always there to help me fix everything when it blew up.

---

## AMPLIFONE VECTOR MONITORS

---

I've had many people (especially operators!) look at me like I was crazy for wanting to restore Amplifone Monitors. Their look gets even more wide eyed when they find that I am replacing a perfectly good Wells Garner color X/Y with an Amplifone. Why replace a nice working WG?? Without getting technical, simply look at the picture of a tuned up WG and compare it to an Amplifone in similar condition. I think you'll be amazed at the difference. The Amplifone has a higher refresh & slew rate, and the Amplifone tube is medium resolution. The result is a sharper picture that simply looks better than any WG. You are seeing the picture the way it was supposed to be, when games like Quantum, Star Wars, and Major Havoc appeared when they were new around 1982-1985.

The problems with the Amplifone HV Transformer and HV boards are infamous, and many have called it the "trashifone", as well as several other names that won't be mentioned here. However, consider the reliability of the Amplifone deflection board. Also, (with the help of this faq) the HV board can be made to be very reliable. The picture on a properly tuned Amplifone will blow away your best Wells-Gardner, mark my words. Restoring Amplifones can be VERY worthwhile. When you see the picture of your favorite Atari color Vector game on an Amplifone (for the first time to many of you) I think you will agree, "it's worth it".

### Schematics

When using this guide, you will be VERY confused unless you have a nicely printed schematic of the HV and Deflection boards. The schematics are available on GameArchive (Thanks Jess!) (<http://www.gamearchive.com>). While the original HV schematics are helpful, they do not show the 5w 50-ohm resistors for some reason. It does show the 1N4001 diodes at the +/- 24v regulators. However, it lists capacitors C3 and C4 as 100uf @35v. These are rated BELOW what I recommend - see the section on capacitors later in this guide. If you have any different schematics for the Amplifone HV board, please let me know.

---

## AMPLIFONE HIGH VOLTAGE (HV) BOARDS

---

### HV BOARD REVISIONS

I am aware of 3 HV board versions. Let me know if you find any others, I would like to document them here. The earliest feature no 50-ohm 5w resistors and no 1N4001 diodes at the +/- 24v regulators. As far as I know, this early revision was always blue, though there are later factory modified blue boards with the 5w resistors and 1N4001's at the regulators. Other colors of Amplifone HV boards are beige and green. They are all labeled A201012, and this is sometimes followed by -01, -02, -03. (Note: the -01, 02, and 03 do not always indicate the revision.)

The early blue boards (with no 5w resistors and no 1N4001's at the regulators) are pretty scarce; I understand that they only appeared in Quantum. The blue boards that have been factory modified with the 1N4001 diodes and the 5w 50-ohm resistors at the regulators may have appeared in early Star Wars machines, as well as the Tollian Web (Major Havoc) prototype and other vector prototypes.

Quantum was the first arcade machine to ever use the Amplifone. Though there are references to Gravitator and Space Duel using the Amplifone, I have yet to see a Gravitator or SD cabinet with Amplifone racks inside. If you see one of these, please let me know. The beige and green HV boards are pretty common; these were found in both Star Wars and dedicated Major Havoc machines. Another point worth mentioning is that the same Amplifone HV board (and deflection board) is used in both the 19" and 25" monitor (25" found in Star Wars cockpit) and both monitors run their HV at 19.5 kV.

### **Blue HV Boards (or rev.1, with no 5w 50-ohm resistors or 1N4001's at the regulators)**

Blue HV boards without the 5w resistors or diodes at the regulators are really rare, but before anybody gets any E-bay ideas (joke), they might only be worth something to the person who needs that particular # board to match all their #'s in their Quantum. Unmodified, these are for use ONLY in a Quantum machine. Throw a stock blue HV in a Star Wars or Major Havoc, and you run the risk of blowing your 24v regulators, killing your MPSU07, your BU406D, and your precious HV transformer.

Blue Amplifone HV boards have some obvious differences from the other HV boards that are very easy to spot. The easiest thing to notice is that on the Quantum series blue boards, there are no 50-ohm 5w resistors across the 7824 and 7924 regulators, and there are no 1N4001 diodes at the regulators. These were put on the later boards to reduce transient voltages to the regulators during stressful times for the monitor in games like Star Wars (Death Star blowout) and Major Havoc (space sequences, wide explosions, etc.) The 5w 50-ohm resistors are not mandatory in Quantum, though it won't hurt to add them. In my opinion, the 1N4001 diodes at the regulators are a "must have" in ANY machine. If you plan on using the blue HV board for anything other than Quantum, the resistors are a "must have". Quantum does not "stress" the monitor nearly as much as Star Wars or Major Havoc, so adding the 5w 50-ohm

resistors is optional if the board is just for Quantum. However, you should add the diodes REGARDLESS of what the blue board is used for.

Some big differences on the blue HV board appear in the HV overvoltage section. This can get REALLY confusing because these differences do NOT appear on any schematic that I have seen. You will notice there is no component in the spot marked R18. There is also a 1k resistor piggybacked to the 0.01uf 50v glass capacitor C9. This resistor has its own spot and is labeled R19 on the beige and green board. It also appears as R19 on the schematic. R16's spot on the blue HV board has a 1N754A zener diode in it. This zener diode is actually CR4 on the schematics and on the beige and green boards. R16 in the schematic is shown as a 470-ohm resistor, and on the beige and green board R16 is a 470-ohm resistor. R19 on the blue HV board is 68k, this goes from the base of Q5 (2N3904) to C10 (0.1uf @50v) and the anode side (line side) of the Zener diode (1N754A) in location R16. There is no R19 listed on the schematic, and no R19 appears on the beige or green boards. Also, when comparing the blue HV board to the newer beige or green HV boards, the over voltage LED is oriented in reverse. (The anode [line side] faces away from the focus assembly instead of towards it on the beige and green board.)

Amidst all this confusion, the key thing to remember is that the overvoltage section (where all the above differences are found) is essentially the same as the overvoltage section in the schematic, and on the green and beige HV boards. These differences can be VERY confusing when trying to follow a schematic. I have yet to see a schematic that accurately covers the blue board, if you find one, please let me know. It would be a great help. Perhaps someone could make a picture illustrating the above differences? I really am no good at text pictures...

As mentioned before, there are some factory-modified blues with the diodes mounted to the back (solder side) of the board across the regulators, and the 5w 50-ohm resistors on the parts side. If these components test OK, you can skip the next section and feel very lucky.

### **Adding the 5w 50-ohm resistors and Diodes to your early Blue HV board**

These 5w 50-ohm resistors go from pin 1 of the 7824 (VR1) to pin 3 of the 7824, and across pins 2 and 3 of the 7924 (VR2). You will have to drill 4 holes in your HV board for the resistors.

For the 7824, I drill 2 holes near the middle of the board. Look at the board, solder side up, and orient it so that the focus assembly is on the right, and the long black heatsink bar is closest to you. Measure 3" in from the right edge of the board, and 2 & 1/2" up from the edge that is closest to you. This is the location of the first hole. The 2nd hole should be 3 & 15/16" in from the right side, and 2 & 1/4" up from the edge closest to you. You should use heat RESISTANT tubing (not heat shrink tubing). This is available from Mouser, TechAm, and Housefelt. Use this tubing to insulate the leads on these newly installed resistors. These resistors can get HOT. You want to make sure there is sufficient distance between the 5w 50-ohm resistors and any critical components on the board. I have seen the heat from these resistors damage components they are close to, as well as the board itself, so mount them in a safe manner.

For the 7924, I drill a small hole (only big enough for the resistor lead) in the middle of the trace that runs to pin 2 of VR2 and goes to the negative side of C2 (470uf @ 50v), and another hole right next to the triangular trace that leads out from pin 3 of the 7924. (This is the trace that combines the negative end of C4 (100uf @ 50v), and one end of C13 (0.1uf @50v). If you follow this trace all the way up the board, it leads to the emitter of the BU406D, labeled E on the parts side of the board.

The very early Blue HV board is also missing 1N4001 diodes on each regulator. These diodes should be installed on the early Amplifone HV board, NO MATTER WHAT game it goes in. You can drill holes to surface mount these, or mount them on the solder side. If you choose the solder side, you will want to insulate the leads with heat shrink or clear plastic tubing so they do not cross any traces. The 1N4001 diodes go at the outputs (pin 3) of each regulator to ground. On the 7824, the cathode side goes to the +24 output line coming from pin 3 of the regulator. On the 7924, the cathode side (line side on the diode) goes to ground. Again, PUT THESE DIODES ON THE BOARD no matter what game you are running. Keep in mind that many of the later blue HV boards had the diodes on the solder side of the board, so look before you go drilling.

### **Beige and Green HV Boards**

The beige and green Amplifone HV boards are revised with the 5w 50-ohm resistors and 1N4001 diodes at the regulators. If you see one without this "standard equipment" please let me know. As far as I know, the beige boards appeared after the early blues, and the greens were found in later Star Wars and most all dedicated Major Havoc machines. Here are the differences that I have seen:

- The values of C3 and C4 are often 220uf @ 35v instead of 100uf @ 35v found on early blue boards.
- C8 (0.033uf, poly on blue board) is now a rectangular plastic case on most beige and green boards, but is rated the same.
- All beige and green boards have the over voltage LED mounted so the anode (flat edge side) faces the focus assembly.
- The orientation of the 20k HV overvoltage pot R17 on some green boards is sometimes sideways when comparing its orientation to R7 (the frequency adjustment pot to the 555 ic) on beige boards.

Note: I have seen a few beige HV boards with a layout just like a blue (C9 jumpered with 1k resistor, nothing in the R18 spot, no surface mount diodes at the regulators, no R19 on the board) but all of these had the 5w 50-ohm resistors on the parts side, and diodes at the +/-24v regulators mounted to the back (solder side) of the board. These beige boards look a bit smaller, but in every other way are identical to the newer modified blue board. The overvoltage LED faces anode away (line side) from the focus assembly, like any other blue HV board. Again, make SURE the leads to your 1N4001's are well insulated – especially on boards with the diodes on the solder side. The older hard plastic insulation used on these

diodes at the factory can deteriorate over time, and if a lead shorts traces on the board, you are going to have a blowout.

---

## **UPGRADES THAT SHOULD BE PERFORMED ON ALL AMPLIFONE HV BOARDS**

### **1. 1N4001 diodes at the regulators**

As mentioned before, you should add these diodes no matter what game your HV board is used in. These diodes are on all boards except for the very early blues. Remember to check the back of the board; many later blues and some small beige boards have them mounted on the solder side.

**IMPORTANT.** Make sure your leads to your 1N4001's are well insulated – especially on boards with the diodes on the solder side. The older hard plastic insulation used on these diodes at the factory can deteriorate over time, and if a lead shorts traces on the board, you're going to have a blowout.

### **2. 5w 50-ohm Resistors at the +/- 24v regulators**

As mentioned extensively before, these can be added to any board, and are required on all boards not used exclusively in Quantum.

### **3. Common Electrolytic Capacitor Changes**

These capacitor changes apply to ALL Amplifone HV boards. First and foremost, use 105 degree low ESR capacitors. I believe Bob Roberts (<http://www.therealbobroberts.com/>) has these at good prices, and he might be stocking Amplifone cap kits now to cover all your needs. Zannen Electronics (806-793-6337) also sells Amplifone cap kits, and I have heard Aubrey is stocking 105 degree low ESR's now. However, I do not know if any of these kits are at the values that I use below. Some of my suggestions below are OVERKILL, but after I work on one of these boards, I usually do not have to work on it ever again. If you want to use the values that I do, here they are:

- C1 and C2: 470uf @ 100v axial capacitor (105 degree low ESR)
- C3 and C4: 220uf @ 50v axial capacitor (105 degree low ESR)
- C5: 4.7uf @ 100v radial capacitor (105 degree low ESR)

#### 4. +/- 24 Volt Regulators

Go with the 1.5 amp rated 7824CV and 7924CV, instead of the 1 amp rated originals. These are available through Mouser (<http://www.mouser.com>). On most boards, you will see that the 7824 (VR1) is **NOT** insulated to the heatsink bar using a case insulator and heat sink compound. Only the 7924 (VR2) should be insulated. You should replace the "rubber" style insulator on the 7924 with a mica insulator and a good, but thin coating of heat sink compound. The older style rubber insulators can deteriorate, and it is best to replace it while your working on the 7924. You can get TO220 insulators and heat sink compound from Mouser as well. Use a light coating of heat sink compound, and apply an even coat to the back of the regulator. Then, put a light coating to the back of the mica insulator where it will contact the heat sink bar. Tighten the screws so they are pretty tight, but do not overdo them. I put the screws in first to hold the regulator in place, and then solder it to the board. Make sure your solder joints are very good at the board. This is a spot that will make a cold joint if your initial joint is not good.

#### 5. Solder Joints

I suggest reflowing all solder joints where the harness wires go to the HV board. I usually heat the joint up, and use a soldapult to remove all of the old solder. Then, I resolder each wire. After you do this, check for good continuity between each joint at the board, and it's endpoint on the harness. You will have to open your neck connector to check the joints at the pins. You can usually do this using 2 small screwdrivers. Be careful not to crack your neck connector when opening it. While you are checking continuity, make sure the DAG ground connector is good, and make sure its pins are in tight. I check DAG ground continuity from the board to the DAG spring on the CRT.

Some other solder joints that are common problems are those at all the electrolytic capacitors, the BU406D (Q2), the MPSU07 (Q3), the +/- 24v regulators (VR1 and VR2 as mentioned above), joints on the HV transformer, joints on the MCI (red mystery can), and ALL wires to the focus assembly. You will also want to make certain your wires to the focus assembly and anode cap are well insulated and don't have any cracks.

#### 6. BU406D (Q2 on schematic)

This transistor is a critical part on the Amplifone HV board. You **MUST** use a BU406D, a regular BU406 does not have the internal dampening diode or internal resistor, and using one of the BU406's can kill your HV transformer. I have heard of using an external diode on a regular BU406, and heard that this can work, but I prefer using a tested good BU406D. Just be sure you are getting BU406D's and **not** BU406's. I would order at least 3 of these at a time.

BU406D (produced by SGS) sources:

**Computer Component Source**

135 Eileen Way  
Syosset, NY 11791  
Tel: (516) 496-8727; 800-356-1227  
Fax: (800) 926-2062  
E-mail: orders@ccs-sales.com

**Richardson Electronics**

40W267 Keslinger Road  
PO Box 393  
LaFox IL, 60147-0393  
Tel: (630) 208-2200  
Fax: (630) 208-2450  
www.rell.com

**Banzai Musical Products**

Adolf-Martens-Str. 8  
12205 Berlin Germany  
Tel: +49-30-84108010  
www.banzaieffects.com

Thanks to David Shoemaker, Hans Osvoll and Jess Askey for sourcing the BU406D's, I ordered these a while back, and was not sure where I got them.

A good BU406D should test as follows using diode test on your DMM:

Looking at the BU406D from the front, pin one is the base, pin 2 is the collector, pin 3 is the emitter (from left to right). Test using the diode test (you CANNOT test a BU406D on HFE). In the chart listed below, Red (positive on your meter) goes to the first pin, black (negative on your meter) goes to the second. The heatsink tab on the transistor is also the collector, so if your second pin is already clipped, just use the tab.

- Base to collector = Junction drop (between 0.470 and 0.690v)
- Base to emitter = reads a short, or about 0.020v (YES, A SHORT. This is correct, remember the internal dampening diode and resistor??)
- Collector to base = Infinity or open circuit
- Collector to emitter = Infinity
- Emitter to collector = Junction drop
- Emitter to base = Short

While we are working on the BU406D, you will want to make sure the collector heatsink is securely soldered to the board, and that you are getting good continuity from where the collector of BU406D connects from the heatsink to the board. **THE BU406D DOES NOT GET INSULATED** to the heatsink. The heatsink is actually the collector connection. When tightening the screw for the BU406D, use the same tension that you did on the +/-24v regulators. It has to be tight enough to make good contact. You will have to clip the collector pin (pin 2) to install the BU406D to the board if it hasn't been done already.

**7. MPSU07 (Q3 on schematic)**

These transistors seem to be pretty reliable according to the HV boards that I have seen. However, you will want to pull and test it while your working on the board. If it's bad, it can damage other components. It should test like any normal NPN, you can use HFE on this one, or diode test. If you need a replacement, an original MPSU07 can be hard to find. I use SK3199 (available from Mouser) as it is rated over spec to the original MPSU07.

**8. MCI "the red mystery can"**

Well, thanks to Zonn Moore, the "mystery can" is no longer a mystery. I am putting Zonn's explanation here:

-----  
It seems fairly apparent that the HV section of the Amplifone is regulated much like that of the old (and maybe some new) constant voltage AC supplies that use ferro-resonant transformers.

Basically the idea is that a transformer can only be driven to a maximum voltage, after which it saturates and an increase in input voltage does not correspond (nearly as much) to an increase in output voltage. This maximum voltage, before saturation, is highly dependent upon the resonance frequency of the transformer.

The constant voltage AC regulators use a capacitor to tune a winding of an isolation transformer to 60hz, causing the transforming to go into saturation. After which voltages of around 70v to 140v input has little effect on the output, and changes in current loading also have a much smaller effect. Pretty cool.

If you look at the schematic of the Amplifone HV you will see no feedback from the output voltage, back into any sort of regulator. In fact the Amplifone contains no HV regulation circuit of any kind! (With the exception of the over voltage regulator, see the following note.)

The 555 is simply set to a frequency, which drives (unregulated) the Q1/Q2 amplifiers, which drive the primary of the current boost transformer T1, which drives Q3, also with no regulation!

I have no way of knowing exactly what's inside MCI, but the only way for this circuit to regulate it's output voltage, given changing output currents, would be for some type of ferro-resonant regulation to be taking place. MCI must be a type of tuning coil, possibly a coil and capacitor, that along with the inductance of T2 is used to set the resonant frequency of a ferro-resonant regulator. (Which is probably why T2 is so expensive and hard for other companies re-engineer.)

These companies are probably winding the HV to give the proper increase in voltage, but not taking into account it's inductance and Q ratings, which in most HVTs are not nearly as critical. As

a result, when it comes time to test the transformer in the circuit, they fail at regulating the output voltage. Just a guess mind you, I wonder if anyone could verify this?) MCI combined with T2 must form a resonant circuit. As the 555 is brought closer/farther from the resonant frequency of MCI/T2 the voltage coming out is higher/lower.

-----

Thanks, Zonn! Now that damn red can is starting to make some sense.

I know that these MCIs can fail, and when you have a bad one, it can blow +/- 24v regulators like crazy, as well as precious BU406D's. I have seen two boards that were blowing +/- 24 regulators as well as BU406D's, and discovered that the 5w 50-ohm resistor near the +24 regulator had burned the side of the MCI can. For the life of me I could not figure out why I was killing regulators and BU406D's. I replaced the MCI with one I got from another HV board (this one was not burnt), and everything worked. I know of no place to source the MCI. If you do, let me know. Watch the placement of your 5w 50-ohm resistors so they are not too close to the MCI.

## 9. **Brown Jumpers (zero-ohm jumpers)**

These damn things should be taken off of the HV board, and replaced with a small piece of heavy solid wire. There is only one jumper on the HV board, it is labeled W2, and is located right next to C2. You may want to stomp on this jumper after you pull it out. These jumpers have caused countless headaches. They often go dead, and can cause you to waste a lot of time trying to find your problem. There are a bunch of them on the Amplifone deflection board, but that comes later in the Amplifone Deflection guide. The white and tan jumpers are usually good and do not have to be replaced.

Note that these jumpers cannot be reliably tested. There have been recorded cases where the jumper tests ok (in and out of circuit), but fails under load.

## 10. **Fuse Protection**

Do yourself a favor and put inline fuses in the harness that feed the +V and -V to the HV board. It will keep you from roasting perfectly good parts. The +V will run with a 1.5 Amp fuse and the -V will run fine with a 1.0 Amp fuse. If you look at the schematic you will see that there is no protection for the HV board because the closest help is the A/C fuses on the deflection board, which moves major power to the deflection transistors. If you want to see how much power just feel the deflection transistors heatsink on an amplifone that has been running a while.

## 11. Mounting a Fan

In my opinion, a fan is a necessity in the cabinet when using an Amplifone monitor. I use a 120v AC fan. I tap the 120v AC line that goes to the marquee light, and mount the fan so it blows directly on the HV board. I try to aim the fan directly at the HV transformer. By using your 120v AC to the marquee light, you are not straining your audio regulator by tapping into any DC current. This will do wonders for the life of many components on your HV board. I use a fan that is no bigger than 100mm x 80mm (approximately), and it should produce between 22-50 CFM. Make sure you attach the ground wire to the fan (green in your 3 wire harness to the marquee light assembly). I attach the ground to the case of the fan. Try to use a fan that runs smoothly and quietly, you do not want a ton of vibration in the cabinet. I recommend the one listed here:  
<http://www.web-tronics.com/webtronics/csa8038ac.html>

Radio Shack also carries several different types of suitable 120 VAC fans for around \$20 each.

---

## AMPLIFONE HV TRANSFORMER

### History

Now for the part you've all been waiting for. The famous saying, "Red is Dead, Black is Bulletproof." The part of the Amplifone monitor that can partially be blamed for the downfall of all Vector arcade machines. The part that causes many people to wonder why you would bother restoring these Amplifone monitors in the first place. I am going to put a bit (well, a bunch) of history here. I feel a component as infamous as the red Amplifone HV transformer warrants it. I am always looking for more information on the historical side of things, so if you know any more info, or, if anything needs corrected, please let me know.

If you have no interest in history, you can skip this section and move to the "Red is Dead" section that follows.

Much of this information comes from second hand sources, so I can not say that it is 100% accurate, but it seems to make a lot of sense. I was lucky enough to correspond with Carl Newman, a friend of Frank Ballouz (former Atari VP in the coin-op division) and Jack Hammond, a former employee at Amplifone in Brownsville, Texas.

Amplifone was a company that made various electrical and electronic components, and was located in Brownsville, Texas. This is about the southernmost part of Texas, right near the Mexican border. Before making the Amplifone Vector monitor, they made things as simple as coils for solenoids in electric bells, and complex as radar systems used in the U.S. military.

Herein lies the connection to Atari. Rumor has it that while Ed Rotberg was working on the Army Battlezone, he was forced to work with engineers from Amplifone that designed radar assemblies for the military.

Amplifone apparently knew a lot about radar, but nothing at all about arcade machines. Much of Rotberg's displeasure with the Army Battlezone project had to deal with the Amplifone/military engineers. Ed was used to working with brilliant and creative minds at Atari like Lyle Rains, Dave Theurer, and Ed Logg. He hated being around the stuffy ex-military Amplifone guys who were 30 years older than he was. They had no interests similar to his. Ed was stuck, with no way out, and almost quit over this. Not to mention Ed's inherent dislike of making a 'game' that would be used in war.

However, Ray Kassar (Atari Chairman) saw the Army Battlezone project as a great way to give Atari a ton of credibility, and free publicity. During Ed Rotberg's time at Amplifone, Ed suggested to Frank Ballouz (VP in the coin-op division) that Amplifone could be a possible supplier of X/Y monitors for Atari arcade machines. Atari was having big trouble meeting demand for the Wells-Gardner 6100 color Vector monitors in 1981.

Tempest had a big production run going at this time; it was the ONLY arcade machine in history that completely sold out the first 20,000 units before they were produced. All orders were prepaid. The only thing holding up future sales was the fact that Atari could not receive monitors fast enough from Wells-Gardner to meet the demand. Thousands of Tempest cabinets were built and were sitting in the warehouse without monitors. The situation was critical. When you consider that Atari had another pretty hot color Vector game known as Space Duel ready to roll, the monitor situation was even more intense.

Frank Ballouz checked out the Amplifone building and saw that they would be very capable of supplying monitors to meet the demand that Wells-Gardner could not. He suggested to Ray Kassar that Atari contract with Amplifone to produce color Vector monitors. Instead of giving Amplifone a contract, Ray decided to buy them out. Steve Ross (owner of Warner Communications) gave the nod, and just like that, in the spirit of true vertical integration, Atari owned Amplifone. According to Jack Hammond the entire process took less than one week. "Frank was here on a Thursday, and Atari bought the company by Tuesday the following week." Atari had a TON of muscle at this time. This was about 2 years before the crash, and you might say Atari was at its peak power in both the coin-op and home video market. For Atari, buying Amplifone was like your average arcade phreak dropping a quarter into a machine.

Here is where the story on the Amplifone Color X/Y monitor becomes clouded. Apparently, the design for the Amplifone Color X/Y came from Atari, and I think it would have been patented, but I have not found any of these patents yet. If anybody has more info on this, please let me know. The design was a fantastic one for its time. The deflection board is much more reliable than those found in the Wells-Gardner 6100, and the HV board was also designed fairly well. As you are probably aware, the HV board had one problem. I feel that this problem was a major factor in the eventual downfall of all Vector games in the arcades.

The "evil seed" in the production of the monitor came from the manufacturing of the HV transformer. A HV transformer must be manufactured under ideal conditions - it must be dry, with very low to no humidity. The temperature must be consistent (preferably around 70-90 degrees Fahrenheit) to prevent condensation. A clean room is not a necessity, but the air should be relatively free of any foreign particles (smoke, dust, etc..)

Many of the "know nothing" Atari exec VP's at this time enjoyed traveling to different Atari locations, looking at things they knew little or nothing about, while throwing their weight around. Rumor has it that a group of Atari big wheels came into the Amplifone plant in late 1982 for a "visit and inspection" before taking a much needed (ha!) retreat to Corpus Christi, Texas.

With this group was a person who would eventually become president of the entire consumer electronics division. I am refraining from using this individual's name. Though my case is strong enough to win a possible defamation lawsuit, I do not have time for the hassle. This individual (in my opinion, as well as many others) is one of the people who can get a great deal of the blame for the entire downfall of Atari. If I write a book someday, I will elaborate further than in this faq. Beware. He was hired by Ray Kassar from Jovan, a perfume company. He later became president of the consumer electronics division. He was immediately fired after Warner stock dropped 40 points in one day, but he did plenty of damage before this. I will refer to him as Mr. P.O. from here on.

Apparently, while touring the plant in 1982, Mr. P.O. could not understand why he was told not to smoke in the building, and even had a technician fired for telling him to put out his cigarette and to "please smoke outside the building." He also could not understand why it was so uncomfortable in the building. He was amused by the winding room, and liked to watch the machine wind the wire on the HV transformer spools. "He thought it looked like a big sewing machine..."

Somehow(?!), during Mr. P.O.'s visit, a cigarette ends up in a garbage can at the Amplifone warehouse. This was where the big spools of wire were kept until they were needed to wind the HV transformers in the winding room. The garbage can catches fire, and the buildings' sprinkler system comes on. The spools get soaked. Wanna guess what happened next??

The Amplifone employees are irate. Several are ready to kill Mr. P.O. Mr. P.O. orders them to "clean this mess up, and get the machines running, unless you want to loose your job! We've got deadlines to meet!..."

I'm willing to bet that these spools that got soaked were used in the production of MANY Amplifone HV transformers. Though the wire is insulated, the moisture that the wire was exposed to became trapped in the coil in the HV Transformer. After the HV transformers were used for a length of time, the moisture caused the insulation on the HV wire to break down. When this happens, the HV transformer no longer works.

## Followup

This information was posted by Jed Margolin (jmargolin.com), former Atari employee, and hardware designer of many Atari games.

-----

By now everyone has heard the story of the Universally-Hated Atari Executive who went down to Brownsville to visit Amplifone, ignored requests not to smoke inside the building, and triggered the fire sprinklers, ruining every Amplifone HV transformer ever made. While there was never any shortage of Universally-Hated Atari Executives, and while an incident of this type could have happened, I have my doubts about it. For one, I didn't hear about it until I saw it on the Web about two years ago, long after I left Atari Games.

The story I heard, while I was still at Atari, was that the method used to make the HV transformers turned out to be sensitive to humidity, and the first HV transformers to burn up were the ones made during hurricane season.

Very likely, the real story is simply that the potting material used in the transformers was not up to the task.

Regardless of the cause, by the time the problem appeared the Operators had already made a good return on their investment. That was good news for the Operators but bad news for those of us trying to keep their 17 year-old Star Wars games alive.

I've had two Amplifone HV transformers burn up in my TomCat game. I finally replaced the HV assembly with one from a Wells-Gardner monitor. The only difference is that the Wells-Gardner supply does not provide 6.3VAC for the CRT heater. (I put in a 6.3VAC filament transformer.)

-----

I would like to use this paragraph to give Thanks to Jack Hammond, former Amplifone employee, who provided me with a TON of journals, technical notebooks, and materials from the Amplifone labs, Hopefully when this material is sorted and scanned (Thanks Tom!) it will be of help to all who work with Amplifone monitors. I can only hope my mind works as well as Jacks at age 86, even though the "history" section seems like Atari "lore" today...

---

## **RED IS DEAD**

Most of the Amplifone HV transformers that die are red. Breaks in the insulation can occur in either the primary or secondary winding, and there is no "common" failure amongst the red HVT's. When the spools got wet from the fire at the Amplifone warehouse, there was no way to predict when or exactly where the insulation in the red HVT's would break.

Some people (myself included) have reds that work great to this day. I have been observing serial numbers of these working red flybacks. It seems that some of the very early ones can be fine (serial numbers under 800 or so, you will see the number on the heatsink bar "XY No. \_\_\_\_\_"). Some of the very late ones can be fine (above 17,000) as well. Perhaps this indicates the ones roughly before 800 and after 17,000 were not made from the soaked spools? I have also seen HVTs within the "Danger Zone" of serial numbers that work, but I think the odds are against these lasting very long.

Jess Askey has been kind enough to put a serial number registry for HV boards with red HVT's on the GameArchive. Please register your board's serial numbers here, and please test your HVT's before registering them. <http://www.gamearchive.com/>

Perhaps we can narrow down the serial numbers to determine which reds might be good or bad.

In any case, you CAN NOT count on a red Amplifone HVT, regardless of the serial number. If you find one that seems OK (see "Testing your HVT") you may choose to use it until it quits. Be aware that when the HVT does fail, you WILL also lose your BU406D, and you have a good chance of losing your +/- 24v regulators as well as some of your diodes. One diode that has a chance of blowing when the HVT goes is CR3, the 1,500v diode right behind the HVT. This diode is not very easy to find, and it is not cheap. If you find a good source, please let me know. I plan on putting a "Sourcing" section to this guide later that will show the best places to get parts for Amplifone HV boards.

Another risk you run with a red HVT is that of fire (ask anybody who runs a G08!). You do not want to have one of these burn in your prized Quantum, Star Wars or Major Havoc cabinet. I have only seen one go out flaming, but have heard of several other people who have as well. The one I saw was in a Star Wars UR, around early 1984. The inside of the cabinet was seriously burned, and the heat was great enough to warp the plastic housing around the monitor. If your cabinet is in great cosmetic shape, DO NOT risk using a red HVT.

---

## **BLACK IS BULLETPROOF**

After the red HVT fiasco, Atari commissioned Penn Trans of Bellefonte, PA to make a replacement. Some of these may be rarely found as NOS through some Atari parts distributors. (You have VERY slim odds on this, but it may be worth trying in your area). These look a lot different than the original reds. They are black, a bit smaller than the reds, and do not use connections from pins on the base of the HVT to the board. The Penn Trans HVT has 6 wires that come out near the base facing away from the focus assembly. They go through the 2nd hole in the HV board, right next to the BU406D.

The Penn Trans HVT's are labeled Penn Trans, and have an address listed as Wingate, PA. These are the first black replacement HVTs, and were built way overspec to the original reds. If you find a HV board with one of these HVT's, you have good odds that the HVT is fine. As far as I know, these Penn Trans HVTs were all installed to replace reds and no blacks were manufactured stock. I have only heard of one failing over the past 10 years. There are rumors of people finding these from some distributors (Betson,

Atlas, etc..) for low prices. I know of one friend who bought 4 for \$100 about 9 years ago. The part number is 926862802, replacement for Atari part #A201005. If you find these cheaper than \$100 each, you may want to buy them.

WinTron Amplifone HVT Replacement

<http://wintrontech.com>

phone: (814) 625-2720

Part Number: 926862802

The common replacement for the red HVT today is the WinTron transformer / multiplier. WinTron is actually Penn Trans, they just changed their name after the company changed hands. Their part number is still 926862802. These things are not cheap. Currently they are \$190 each, and there are no quantity discount that I am aware of. While they are expensive, the replacement HVTs are put together VERY well, and are supposed to be even superior to the PennTrans HVT's. They are pretty similar to the Penn Trans HVT's in appearance and carry a one-year warranty.

You will have to make some minor changes to your Amplifone HV board to accommodate the WinTron. After removing your red, you will want to over drill the mounting holes to 3/8". The original holes are a bit too small to fit the replacement Wintron mounting bolts and the 6 wires on the WinTron. When you remove your red HVT, you will want to make note of the gray wire coming from the base of the red HVT. On the beige/green board, this gray wire goes to a location labeled "WHT", and on the blue, it is labeled "HTR WHT". The mounting nuts that went to the bolts on the red are too small to work on the WinTron (or a Penn Trans) mounting bolt. You will have to find one that works, as the WinTron is not shipped with one. (You would figure for \$190 they could put one on....)

If you are mounting the WinTron HVT on an early Blue or Beige board (empty slot at R18, to make things easy) you will want to move the 3 black ground wires to a different location to allow the WinTron HVT to sit comfortably in it's socket. I usually mount these ground wires at the lower right corner of the board (oriented with the focus assembly on top and the heatsink bar at the left. I simply drill 3 very small holes in the ground trace on the lower right of the board, and use a small jewelers file to scrape the insulation off of the solder side of the board where the holes are. Now, you can solder the 3 black ground wires to the board at their new location (this is actually where they appear on the newer beige and green boards) and the WinTron HVT can sit in it's spot without being on top of the ground wires.

Once you get the WinTron HVT secured to the board, you will want to solder it's wires into place. I am going to explain this so that you can visualize their locations in relation to a clock. The WinTron installation instructions can get a bit confusing, so this may help out. Hold the board solder side up, with the focus assembly at the top and the heatsink bar at the left. We are going to look at the solder spots from left to right, reading them clockwise. There are spots at 7:00, 8:00, 11:00, 1:00 or 2:00, 4:00, and 5:00. The final spot used for the gray wire is labeled "WHT" on the newer beige & green boards, and "HTR WHT" on the early blue and beige (empty spot at location R18). This is the spot where the gray wire was from the old HVT.

You will want to run the wires through the second hole in the board just behind the HVT, and solder them as follows:

7:00 = orange  
8:00 = no connection  
11:00 = green  
1:00 or 2:00 = black  
4:00 = yellow  
5:00 = red  
"HTR WHT" or "WHT" = gray

And for the more technically minded, some additional information from David Fish:

ORG wire from XFMR to PCB hole T2 pin 1. Goes to COLLECTOR of Q3 (BU406D).

GRN wire from XFMR to PCB hole T2 pin 4. Video B+, connects to anode of CR3.

BLK to PCB hole T2 pin 6 or 7. This is GROUND.

YEL wire from XFMR to PCB hole T2 pin 8. This is +24VDC.

RED wire from XFMR to PCB hole T2 pin 9. Low side of MCI.

GRY wire from XFMR to PCB hole T2 pin 10. This is the filament's hot side.

---

## TESTING YOUR HVT

I am going to start this section with a suggestion: Get a Bob Parker/Dick Smith Line OutPut Transformer ("LOPT") tester. They are available from John's Jukes. Here is the address:

<http://www.flippers.com/fbt-main.html>

They are defiantly worth the money, (I think about US\$40 shipped) and if you work on monitors fairly often, you will love having one. John also stocks a nice ESR meter for testing capacitors in circuit in monitors, both Vector and raster. I have one of the ESR meters and it is great. You may want to order one when getting your LOPT tester. Using the LOPT, I put the red lead to the 7:00 pin (orientation as mentioned above) and the black lead to the 5:00 pin. A good ring is anywhere in the green on the meter, though I have seen plenty of good HVT's ring in the yellow range. If you are seeing a red light on your LOPT tester, it is time to call WinTron, or try John Robertson's Amplifone HV hack using a Wells Garner HVT. The hack is posted on John's website: <http://www.flippers.com/vid-tips.html>, and is included at the end of this document. John's hack works, though you are going to have to find a way to fit another chassis in the cabinet.

You will want to ring your HVT using the LOPT meter as follows:

---

red lead @ 7:00, black lead @ 5:00  
red lead @ 9:00, black lead @ 5:00  
red lead @ 7:00, black lead @ 4:00  
red lead @ 9:00, black lead @ 4:00

---

If you are testing a Penn Trans or WinTron, you are likely to see yellow rings because of the dampening diodes in the HVT. At best, these will ring on the low green level so do not be alarmed. Note: a working transformer may even test lower than yellow (red or even black). If you experience this, I recommend visually validating the rings of the transformer with an oscilloscope. This can be done by connecting up the scope to the transformer in parallel to the LOPT tester.





I reconnected the HV overvoltage protection circuitry and the LED was off (yay!). Resoldering the CR1 diode brought the kV down a little (for some reason), but I was able to adjust it back up with R7 (frequency adjust). I was then able to set the HV over-voltage (R17) correctly.

So that was it. Maybe there are a few tips in here that Gregg Woodcock can put in his FAQ (Pull up a leg of CR1 to disable HV overvoltage circuitry, change R6 to increase/decrease HV, whatever...)

---

Thanks Mark! Keep in mind; he is referring to a Penn Trans HVT. Odds are if you are using the WinTron, you might not see this problem, but it is important to keep in mind if you feel everything is fine on your board and you are still getting no HV. Again, DO NOT disable your HV overvoltage protection unless you really know what you are doing. You might want to try using a Variac to gradually power up the monitor (Thanks John Robertson!) to prevent possible X-ray emissions.

One last piece of HV adjustment advice from Joel Rosenzweig. Make sure the brightness control is put somewhere in the 0 - 50% range before powering up the board. It would be terrible to have the brightness too high and burn a spot in your monitor. The other point is that if you have the brightness too low, when you power it all up, you may see \*nothing\*! If you have the brightness somewhere shy of the middle position, at least you will see something on the display if all goes well.

---

## REPAIR YOUR AMPLIFONE HV BOARD

I guess this section should have appeared first, but I felt it was more important to discuss the components, boards, and their differences before the repair guide. Joe Welsler deserves a lot of credit here; he was the first to create a guide of any kind. Some may choose to just read this section and ignore the previous ramblings.

First, remembering my Randy Fromm lessons, GARBAGE IN = GARBAGE OUT. Before doing any work to the HV board, you will want to verify that your deflection board is producing the +/- 30v (this can vary from +30 to +39vdc, and from -30 to -39vdc). Check this at J103, and you will be looking at pins 6, 7, and 8 of the J103 connector. Pin 6 is ground, pin 7 is -30 to -39vdc, and pin 8 is +30 to +39vdc. Remember that this is an unregulated voltage; it may swing a bit as you test it. You will also want to check this voltage on both the + and - side for excessive AC ripple. If you are seeing ripple over 0.2v, you may want to replace the big caps (4,700uf @50v, c17 and c18) on the deflection board. If you see no +/-30v at J103, check the fuses f3 and f4 (5A 250v SB) on the deflection board, followed by the rectifier diodes (CR 14, 15, 16, 17). These diodes can often develop cold solder joints, so make sure they are soldered properly. I have covered more on the Amplifone deflection board in the Deflection section found below.

Once you have checked and are receiving proper voltages from the Amplifone Deflection board, complete the following steps. All of the steps below are to be done with the HV board removed from the machine, and with no power applied.

**Step 1: Check your HV transformer**

I use a Dick Smith LOPT tester (as mentioned before in "Testing your HVT") and test the HVT in circuit. (NOT with the power connected to the board, but with the HVT still soldered to the board - I got worried somebody might actually try ringing their HVT with the power on! Remember what I said about damaging yourself as well as your equipment?!) If your HVT is ringing properly you can feel lucky and move to step 2. If you are getting "marginal" rings (low yellows), you may want to pull the HVT off of the board and test it.

**NOTE!** Keep in mind that it is normal for a red HVT to ring a lot higher than a PenTrans or WinTron, whether it is in or out of circuit.

Pulling the HVT from the board is best done by using a soldapult or desoldering gun to get the solder out of the pins, then gradually heat and pull the HVT off, applying heat to the stuck pins. Be careful removing the HVT, I have seen a few HV boards cracked from trying to pull the HVT out before all the pins are loose. Then, use the LOPT to test again, and see if your marginal ratings are better.

If your HVT is ringing bad, save your quarters, and call WinTron, or get a WG chassis and use John Robertson's HV hack.

**Step 2: Pull and test your BU406D (Q2 on schematic)**

Visually inspect the component at Q2 for obvious damage and to ensure that it is the correct part. Oftentimes, previous owners/operators incorrectly replaced the component with the more-readily available BU406. Running the HV board with the incorrect component can cause irreparable damage to the HVT.

A good BU406D should test according to the chart listed in the BU406D section of this faq. Remember, when the HVT goes, it WILL take the BU406D with it 99% of the time. Leave the BU406D out before going to step 3, you will not want it in circuit until you are almost done with these steps. If your BU406D is testing bad, you will have to replace it.

**Step 3: Pull and test your MPSU07 (Q3), both 2N3904's (Q1 and Q5), and the 0.8amp 100v SCR (Q4)**

The 2N3904 transistors are easy enough to get, and testing them is fairly easy using either Diode test or HFE on your meter. If your BU406D or MPSU07 is bad, they can often take the 2N3904 transistors with them. If you need to replace the MPSU07, you can use an SK3199 as shown in the MPSU07 section of this faq. Testing the SCR (Q4) is easy enough, just pull it and test on diode test. If you are reading an open circuit or short in any direction, you'll have to replace it. The SCRs are relatively inexpensive and not very hard to find. I did not include a section on Q4 because I do not think I have seen one go bad yet.

**Step 4: Test all resistors and diodes**

Testing all the resistors is simple enough. They can be tested in circuit and should be within 10% of their specified values. To test the diodes, I simply pull one lead out of circuit and use diode test. Most all of the diodes and resistors should be easy to replace if you find any that are bad. The only tricky diode is the large one (1,500v) right near the HVT; it is labeled CR3. This can be hard to find, let me know if you have a good source.

**Step 5: Recap the board**

I recap the HV board using 105-degree low ESR caps as listed in the "Common Electrolytic Capacitor Changes" section. See the table for the values that I recommend. Be sure C3 and C4 are not too close to the 5w 50-ohm resistors at the regulators (if your board has them). The heat from the resistors can damage the caps, as well as the +/- 24v regulators and the red MCI "mystery can".

**Step 6: Check for good +/- 24v**

You will want to test the 7824 (VR1) and 7924 (VR2) to see that you are getting a good +/- 24v. Remember that you should check the insulator (and replace it if it is the "rubber" type) on the 7924 before applying the power. I use a mica T0220 insulator and silicone heat sink compound.

As long as the BU406D is out of circuit, it is relatively safe to apply power to the HV board. With the BU406D removed there is no chance of any HV being generated.

This test is done with the HVT wired to the deflection board. Plug in your harness to J103 on the deflection board (right next to the rectifier diodes and just in front of the big caps). Connect your DAG ground to the deflection board (from the tube spring). You do not have to connect the yoke plug (j105), the neck connector (j101), or the HV anode. I use microclips to the leads of the

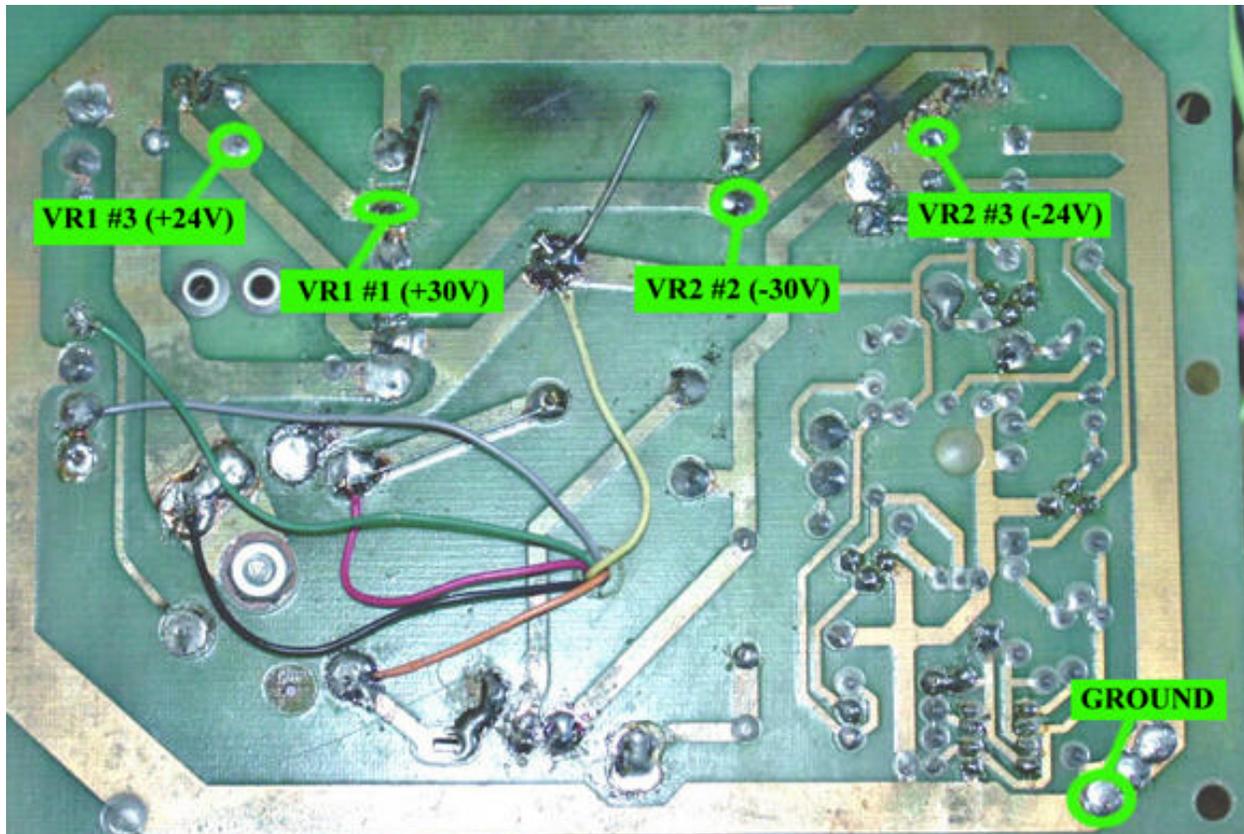
regulators in order to test the voltage. If you are testing the board inside a cabinet, the lower back ledge (where your back door rests when it is on of your machines) makes a decent test area. You can put the heatsink on top of the panel where the power switch is mounted, so the HV board stands up with the solder side facing you. Your harness should have no trouble reaching the HV board when it sits here.

These regulators take in +30V and -30V in order to output +24V and -24V. The input voltages come pre-filtered from the deflection board. Although spec is +30/-30, I typically see +33/-33.

To test the input voltages on the HV, check pin 1 of the 7824 (VR1) and pin 2 of the 7924 (VR2). Spec is +30/-30. If your 7924 tests very low or 0, check/replace the wire jumper W1 on the deflection board.

Put the red lead of your meter to the output of the 7824 (+24v) (pin 3) and the black lead to ground (pin 2). The 7924 (-24v) uses pin 1 as ground and pin 3 as the output. Your +24 output should be anywhere from +24 to +27.5vdc, and your -24 output can be as low as -36v if your board has the 5w 50-ohm resistors at the regulators. If you are missing either +/- 24, I suggest using the 7824cv & 7924cv as a replacement (see the 24v regulator section of the faq). Do not panic if your readings are not exactly +/- 24. The 5w 50-ohm resistors can make your readings a bit wacky when there is no load on the regulators.

It can often be difficult to place your DMM lead on the voltage regulator without shorting to another leg. You can choose test points that are on the same solder trace as the VR leg, yet is more convenient to test at. For example:



(photo courtesy of Brett Pehrson)

**Step 7: Check for a waveform at the base of the BU406D**

You will need an oscilloscope to see this waveform. Put the red lead to where the base of Q2 goes (on the solder side), and put the black lead to ground. The parts side shows what pin would be for the base. AGAIN, we are doing this with the BU406D out of circuit. You should see a nice 20kHz waveform. If you do not, you will want to start checking all components in the oscillator circuit, including the 555 timer (U1). A bad 2k pot (R7) or 20k pot (R17) can cause weird things in the oscillator circuit. You may want to clean or replace these pots if everything in the oscillator circuit is OK and you are still having problems seeing a good waveform at the base of the BU406D. The 555 should produce a small waveform at pin 3 of the chip. You can use your scope to trace through the circuit and see where the waveform is being lost. You may want to recheck Q1 and Q3 if your oscillator circuit seems OK, but you still get no waveform at the base of your BU406D.

**Step 8: Add 5w 50-ohm resistors and 1N4001 diodes at +/-24v regulators**

As mentioned in the previous section, you will always want to add the diodes to your HV board if they are not present. Make sure the leads are well insulated if your 1N4001's are on the solder side, as stated earlier in the faq. Unless you are only using the board in a Quantum, you will also add the 5w 50-ohm resistors. Be careful placing the resistors, they do get hot and can damage nearby components.

**Step 9: Put the BU406D back in circuit**

Aside from securely soldering the BU406D, you want to remember that you **DO NOT** insulate the transistor to the heatsink. You may also want to take this time to verify your wiring harness, and make sure your continuity is good all around. Do not forget to check the DAG ground, as well.

**Step 10: Adjust the High Voltage**

Now you are ready to try out your HV board. Follow the section labeled "Adjusting your HV" and be sure to read Mark's post. Remember, if you are using a WinTron, the internal protection in the HVT may prevent it from going to 25 kV.

**Step 11: Install a fan**

You will want to install a fan after you get your HV board working properly and have it adjusted. This will make life a lot easier for your HVT, the BU406D, and your +/- 24v regulators.

**A Summary of Replacement Parts For The Amplifone HV Board**

<u>Quantity</u>	<u>Description</u>	<u>Location</u>
1	WinTron HVT Replacement Part Number: 926862802	
1	BU406D	Q2
1	MPSU07 (replacement is SK3199)	Q3
2	2N3904	Q1 and Q5
1	0.8amp 100v SCR	Q4
2	5w 50-ohm resistors	
2	1N4001 diodes	
1	2k pot	R7
1	20k pot	R17
1	7824 (replacement is 1.5 amp rated 7824CV)	VR1
1	7924 (replacement is 1.5 amp rated 7924CV)	VR2
2	470uf @ 100v axial capacitor (105 degree low ESR)	C1 and C2
2	220uf @ 50v axial capacitor (105 degree low ESR)	C3 and C4
1	4.7uf @ 100v radial capacitor (105 degree low ESR)	C5
1	Replacement jumper wire	
1	Cooling fan	

## AMPLIFONE HV PROBLEMS AND SOLUTIONS

**PROBLEM**: Why am I not getting +/- 24v from the regulators? The board is dead! I replaced the regulators, but I'm getting nothing...

**SOLUTION**: Check the stupid, asinine, and idiotic brown jumper located right next to C2. GET RID OF IT. See section on brown jumpers in the faq. I have seen ones that test fine and then decide to quit just as power is applied to the board for the first time in 15 years. You are probably thinking, "I tested that jumper, right before replacing the +/-24v regulators, and it was fine...It can't be that..."

Also, check your +/- input voltage from J103 on the deflection board to be sure you are getting the +/- 30v. You may also want to double-check your harness.

Note: Spec is +30/-30 VDC, however it is within acceptable range to see as much as +33/-33.

**PROBLEM**: The 24-volt regulator has failed.

**SOLUTION**: Check for cracking around the regulator leads on the PCB. Also check for cracking around the leads of C3 and C4.

**PROBLEM**: I am continuously losing BU406D's. My HVT is ringing fine on the tester. My +/- 24v is OK. My MPSU07 is fine. I get no HV, and the BU406D is dying after power is applied to the board.

**SOLUTION**: Make sure you have good continuity between the collector heat sink, and the collector of the BU406D transistor itself. Check to see that the heatsink is securely soldered in place, and that it is conducting. Make sure you still have a waveform at the base of the BU406D. If you are missing your waveform (or seeing a line that is almost flat), recheck your 555 oscillator circuit. A "flat line" on the scope can easily kill your BU406D. If your oscillator circuit is fine, and you have a waveform at the base of the BU406D, you may have a marginal failure in the HVT. It might still ring OK, but if you see "pulsing" in the rings, this can be killing your BU406D. Time to either call WinTron, or perform John's 6100 WG>HV hack.

I have never seen a modern WinTron fail, but there is always a first. Remember to save your WinTron papers as their HVT carries a 1-year warranty, and they are very good about taking care of their customers.

**PROBLEM**: I am losing +24 and -24v regulators. Sometimes they even blow apart! Both regulators are insulated, my HVT rings fine.

**SOLUTION**: Make sure that your red MCI can is clear of the 5w 50-ohm resistor near the MCI case. Inspect the MCI case for cracks and burns. Try replacing the MCI (if you have a spare). Check the Zener Diode 1N754A in the overvoltage circuit (CR4) and replace it if necessary. Also, check the 20k overvoltage pot (R17), and replace as necessary.

**PROBLEM:** I have absolutely no HV.

**SOLUTION:** First and foremost, check the +30/-30 input from the deflection board to the HV board. Of particular note is the -30 source, which passes through W1 (notorious zero-ohm jumper). There should be +30 VDC between pins 6 (GND) and 8 (+) of P103, and -30 VDC between pins 6 (GND) and 7 (-). Note that spec. is +30/-30, however you will typically see up to +33/-33 volts which is acceptable.

If your Amplifone HV is dead then it is probably either resistor R12 near the HV transformer or the HV transformer itself. **CHECK THE RESISTOR FIRST** since it is much easier and cheaper to replace. Simply unsolder it and see if it has the proper resistance (2.2K Ohms). You will also want to check to see if the over-voltage protection circuit is misadjusted causing it to activate even though the HV is in a tolerable range. If LED (CR2) is lighted, then the over-voltage protection circuit is activated. If it is on then you either have a misadjusted potentiometer or a damaged HV circuit. Mark the initial position of R19 and tweak it a tiny bit to see if the LED turns off. Do not move it too much or you may damage other parts of the circuit or even your tube if the HV is indeed too high.

**PROBLEM:** Any problem with deflection or HV.

**SOLUTION:** Replace all zero-ohm jumpers (brown resistor-looking components, labeled W?) on the deflection and HV board. As noted elsewhere in this document, these notoriously defective jumpers are prone to strange failures that cause even stranger failure conditions.

**PROBLEM:** The voltage is low on the supply lines on the HV PCB.

**SOLUTION:** There are jumpers on the supply lines. The jumpers are marked with a "W" on the schematic. They look like resistors on the board and have a single black band on them. The jumpers should have no resistance. However, the brown jumpers may have created some resistance to them. They should be replaced with a piece of wire. The white and tan jumper should have no problems, although you should replace them anyway.

**PROBLEM:** No filament voltage from the high voltage (HV) PCB.

**SOLUTION:** Check for a loose connection on the lugs that hold the HV transformer to the PCB.

**PROBLEM:** What should the output of the 555 IC on the HV PCB be?

**SOLUTION:** The output at pin 3 of the IC should be a 20-kHz square wave. This is adjusted with potentiometer R7.

**PROBLEM:** There is no high voltage. The positive and negative 24 volts are present.

**SOLUTION:** Check transistor Q3. The problem may be that the transistor tab (collector) may not be making good contact with the heat sink.

**PROBLEM:** The schematics for the Amplifone HV cutoff circuit do not match the board assembly.

**SOLUTION:** There are three board revisions. You can tell which one you have from the table here. The schematics for all three boards are shown.

**DISPLAY REVISION INDICATORS:**

<b><u>PCB Revision</u></b>	<b><u>R16</u></b>	<b><u>R18</u></b>
Original	470 Ohms	27K Ohms
First rev.	Zener	Missing or 27K Ohms
Second rev.	470 Ohms	33K Ohms

**PROBLEM:** On the HV PCB, the output voltage of the regulator is good when the pin is lifted, but there is no voltage when the pin is connected back to the board.

**SOLUTION:** Check for bad 0.1uF glass capacitor across the voltage regulator. The decoupling capacitor may be shorted.

## WELLS-GARDNER 6100 HV HACK

### Replacing blown Flyback/LOPT (High Voltage) transformers with my 'RasterHack'

Subject: A workable substitute for the Flyback for Ampliphone XY monitors (and other XY games)  
From: John Robertson  
Date: 1997/02/28 ("\*\*" Updated 1998/05/20)  
Newsgroups: rec.games.video.arcade.collecting

Ok, Here goes...

I found that the Wells Gardner K46XX series of monitors uses the SAME picture tube as the Ampliphone monitor, and realizing that 19Kv HV is 19Kv HV, I took a spare WG monitor chassis, and isolation transformer (mandatory, if you don't know why DO NOT TRY THIS!) and did the following:

- (1) Removed the video card (not required), unsoldered the screen and focus wires from the Ampliphone (Am) tube socket, and soldered in the WG equivalents.
- (2) Next I hooked up the HV lead to the Am tube, and hung the yoke (you **\*\*DO\*\*** need it) of the WG chassis, on the side of the cabinet wall, and plugged in the WG to the Isolation transformer.
- (3) Made sure that there was a 1-ohm resistor in series with the filament of the tube.
- (4) Made sure that the new monitor chassis was connected to the electrical ground in the game!
- (5) When it lit the tube up, I could then turn on the Star Wars (actually an ESB s/d but who cares ;-) ) and again could play the game!

...

**\*\*There is now a schematic available for download, it's about 850K in size...\*\***  
(<http://www.flippers.com/images/ampli-wg.jpg>)

This shows the wiring changes that are needed for the WG Chassis to work. Actually you can still use the WG neckboard, I just thought it would be simpler to eliminate it (into spare parts) and have a single "Screen" control to adjust instead of the possible confusion of the drives, cutoffs etc that are no longer used... (\*\*added May 20, 1998\*\*)

:-#)#

P.S., if this really works for you why not treat this idea as shareware and tip my shop \$10US (postal money order please) for the use of the idea? This is an experiment in cost recovery for a small struggling shop that loves to tackle the tough problems. If this works I'll post some more ideas, including test equipment for that is based on the IBM PC for doing board repair and others.

--

(Pinballs, Jukes, Video Games) | John's Jukes Ltd. 2343 Main St.  
VOICE (604)872-5757 FAX 872-2010 | Vancouver, BC, Canada V5T 3C9  
e-mail to -> [jrr@flippers.com](mailto:jrr@flippers.com) | <http://www.flippers.com>  
#(#-: "Old pinballers never die, they just flip out." :-#)#

---

## AMPLIFONE DEFLECTION BOARDS

---

### Introduction

This section is not near the size or scope of the Amplifone HV portion of this FAQ. Amplifone deflection boards were designed VERY well. In my opinion, the Amplifone deflection boards are much better than the Wells-Gardner 6100 deflection boards. Deflection transistor failures and failures in the low voltage section are rare. They become almost non-existent when the boards are modified with the upgrades in this guide. I know of two friends who have installed revised versions of the LV2000 to replace the low voltage section of the Amplifone deflection board. However, I do not feel that a LV2000 is necessary on the Amplifone. (Thanks to Anders and Jeff for making the LV2000 for the WG 6100! It is a "must have" for any WG6100.)

I have only seen the LV section of an Amplifone deflection board fail once in the past 17 years, and the failure was not the fault of the design.

There is no history section here on the Amplifone deflection board. I really know nothing of its history; any info you can provide would be great. I have even tried searching for the patents on the Amplifone monitor with no luck. Please contribute anything you might have. It would be great to add a History section to this guide like the one in the Amplifone HV guide.

### Amplifone Deflection Board Revisions, Schematics

I have only seen 2 variations in the deflection board design. It seems that the very early ones did not include the 10k "X" and "Y" size pots located at the junction of R1 and R2 (y size) and R24 and R25 (x size). These early boards were used in Amplifone equipped Gravitar and Space Duel (VERY rare), as well as all Quantum machines. I'm guessing Atari just decided to use the game board pots to adjust the x and y size.

There are some early deflection boards (serial numbers under 1,000) with the RGB drive and bias pots arranged differently than the later boards. Most of these were boards without the x or y size controls. If you have a deflection board without the size pots, you can add them if you want. The pots are 10k, and they can be mounted on top of resistors R1 and R2, and R24 and R25. You will want to see the Amplifone Schematic (Thanks Jess!) at Gamearchive:

<http://www.gamearchive.com/>

I have seen deflection boards in three colors: blue (first revision), beige (second revision), and green (third revision). The only differences between the deflection boards were those mentioned above.

## AMPLIFONE DEFLECTION BOARD UPGRADE

The following steps should be followed to upgrade and improve all Amplifone deflection boards. Think of it as a 'super' cap kit.

### **Step 1: Remove and Replace all Jumpers**

The very first thing I do when repairing any Amplifone deflection board is to remove all of the "zero-ohm" brown jumpers. They are as evil and malicious as the red HV transformer on the Amplifone HV boards. However, they are much less costly and are easier to replace. These jumpers are located at any spot on the board where you notice a "W" designation. They are brown, and almost look like resistors. You will find W1 through W8 on the deflection board. I suggest using #4 to #6 gauge wire to replace them. I usually save clipped leads from components such as resistors or capacitors for doing this. Notice how many of these \*@%\$# jumpers crumble and break when you heat them to take them out?

I HATE these damn jumpers! MANY times I have seen them cause deflection and HV board problems. You might test them, and they will ring out just fine at 0-1.4 ohms. However, they often gain resistance or open when power is applied to them for the first time in over 10 years. You are thinking, "oh, I tested the jumper(s), its not that..." and you go through everything until you finally realize the jumper(s) is (are) the problem.

Some of the zero ohm jumpers may have already been replaced with wire at the factory, but chances are you will have to replace all of them.

### **Step 2: Add the Diode Modifications**

You will need the following parts to make this modification:

- (4) 1N5342B 7.5v 5 watt Zener diodes (Mouser part #625-1N5342B)
- (4) 1N4002 diodes (Mouser part # 583-1N4002)
- About 15 inches of 5mm heat shrink tubing (4mm will work as well)

The first thing you have to do is connect the 1N5342B Zener diodes anode-to-anode in pairs. (The line side of each diode will face away from each other). Take two Zener diodes, and twist the leads of the anode side of each diode together. Leave about 1 inch of lead distance between each anode, and solder the pair together. Repeat this procedure for the other 2 Zeners. You should now have two sets of 2 Zener diodes, connected anode to anode with about 1" of lead space in the middle. Leave the end of the leads long; we can always trim these later.

Next, you will want to desolder the orange (x input) and yellow (y input) wires from the deflection board. Clear the solder holes for each with a soldapult or your desolder station. This is a good time to check the orange and yellow wires for crimping; they may have become strained from the weight of the harness. If needed, cut and restrip them.

Solder one cathode side of [the above] 1N5342B diode pair to the parts side of R6 (1.5-ohm, 10w) closest to Q22. This is the end of R6 that faces away from the heatsink. Solder a cathode side of the OTHER diode pair to the parts side of R29 (1.5-ohm, 10w) that faces away from the heatsink. You should see a black wire on the board that links R6 to R29. You are attaching each diode pair to the same end of the resistors as this black wire. Check for cold joints on the solder side of R6 and R29 after you have soldered each diode pair to the parts side.

Now, use your heat shrink tubing to insulate each diode pair. Slide on enough of the tubing to leave about 3/4 inch exposed at the unattached end of each diode pair. Take the unattached end of the diode pair attached to R6 and feed it through the yellow hole (y input), and take the unattached end of the diode pair attached to R29 and feed it through the orange hole (x input). Leave enough exposed lead on the parts side to resolder the yellow and orange wires to their respective diode pairs. Solder in each diode pair, and then solder on the orange and yellow wire to each.

Next, we are going to put the 1N4002's on the solder side of the board. If you look at the solder side of the board you will notice there are 2 rivets that attach the heatsink to the board. Rotate the board so that the heatsink is closest to you, solder side up. You want to scrape a 1/2" area of the green insulation from the ground area about 1/2" NE of the left rivet, and 1/2" NW of the right rivet. Locate the two nuts and bolts that are closest to the center of the heatsink. Scrape two 1/2" squares again, about 3/4" to the right of the left bolt, and about 3/4" to the left of the right bolt.

Find the emitter trace for each of the 3792 transistors (Q7, Q17). This trace is connected to the collector of each 3716 transistor (Q6, Q16). The emitter trace for Q16 and Q17 runs from the collector of Q16 through the emitter of Q17, to the fuse socket F2. The emitter trace for Q6 and Q7 runs from the collector of Q6 through the emitter of Q7, to the fuse socket F1. You will want to scrape about 1" of the green insulation off of this trace between Q16 and Q17, as well as 1" off of the trace between Q6 and Q7. We need a clean surface to solder to.

Now, we are ready to attach our 1N4002 diodes. Solder the cathode (line side) of the 1N4002 diode to the trace that goes to the emitter of Q17. DO NOT solder the diode directly to the emitter of the transistor, but to the trace about 1/2" away from the emitter. Soldering directly to the transistor will just make things difficult should you ever have to replace the transistor. Now, insulate the diode with your heatshrink tubing, leaving enough open area so you can solder the anode end to the collector (this is the 1/2" square we scraped from the insulation closest to the right-center bolt). Now you can solder the anode end to the collector of Q17. You will want to use this same technique to solder the remaining 1N4002's, following the designations below:

- For 3792's (Q7, Q17) 1N4002 cathode is at the emitter, anode is at the collector of each transistor.
- For 3716's (Q6, Q16) 1N4002 cathode is at the collector, anode is at the emitter of each transistor.

I hope to have a picture illustrating this soon. Hopefully everyone can understand the above text description for now.

### **Step 3: Recap the Deflection Board**

Here is my replacement cap list. All caps should be 105 degree low ESR's. As in the HV section, many might consider these replacements OVERKILL, but I have not replaced caps for a second time yet.

- C7, C8, C9, C10 1uf @ 100v 105 degree low ESR (axial)
- C17, C18 4700uf @ 100v 105 degree low ESR (radial)
- C19, C20 100uf @ 50v 105 degree low ESR (radial)
- C24 33uf @ 250V 105 degree low ESR (radial)

**Be sure to install all caps with correct polarity.**

### **Step 4: Replace X and Y size pots**

X and Y size adjustment on the deflection board is done through the two thumb pots R76 and R79. I usually install these on early boards that do not have them. Even if the pots are on the deflection board, I replace them. Original ones can often cause jitters in the monitor. Tuner cleaner is only a temporary fix. Simply replace these 10k pots, as they cost less than \$1 each.

### **Step 5: Resolder and Repair Harness Wires**

Check all harness wires where they connect to the board. These wires often become crimped, and you will often find some attached only by one strand. Cut and restrip the wires as needed, and resolder all the harness joints whether they look like they need it or not. After you are done, check the continuity of each wire with your meter from the board to each male harness plug in the molex connector. Clean the molex connector if needed.

### **Step 6: Check for Shorts, Opens**

I use the diode test on my meter to check each diode in circuit for a good one-way junction drop of 0.4v to 0.8v. If I see any readings that look unusual, I will pull one lead and accurately check the diode. I have seen a few CR3, CR4, CR12, and CR13 (all 1N914's) go bad, so you may just want to pull a lead and test these diodes. Do not hesitate to replace any questionable diode. If one of the diodes fails, it can, and usually will take other components with it.

I also use the diode test on all transistors, pulling and testing any that show unusual readings. It can be common to find a 2N3904 or a 2N3906 that has been pushed over on the board, causing its leads to short together. Make sure the transistor leads are situated properly and are not shorting themselves or a nearby component. Shorts can also happen with the leads for Q4 and Q14 (MPSU57) and Q5 and Q15 (MPSU07). Amplifone deflection boards have the RGB color transistors (Q23-25, D40P5, NTE228A) mounted near the bias and drive pots. The base of these transistors can short to nearby components if they are bent away from their plastic "snaps" and lean toward the groups of resistors above them.

I test all resistors in circuit first. If I see any unusual readings (15% out of spec), I pull a lead to test the resistor out of circuit. You may want to double check R14 (1.2k, 1/2 watt) R72 (1.6K, 1/2 watt). I usually upgrade them to 1 watt.

**Step 7:** Replace all Deflection Transistors, Test MPSU07's, MPSU57's

When looking at a board for the first time, I always test the fuses F1 and F2. You can bet if one is blown, you probably have a bad 3716 or 3792 (or both) on that side. Before removing old deflection transistors, I test them on diode test in circuit. If you find a short in a deflection transistor you will want to pull and test the MPSU57 and MPSU07 next to it as well. Always test your old 3716's and 3792's before you install new ones. This can lead you to find a problem before the new transistor is installed so you do not risk damaging a new one.

I always remove and replace all deflection transistors when rebuilding a deflection board. I also replace the mica insulators. Remember to apply a thin, even coat of heat sink compound, and always check on continuity for shorting to the heatsink by the base, collector, or emitter.

**Step 8:** Check, and if Necessary, Replace Fuses

F1 and F2 should be 3A 250v Fast Blow  
F3 and F4 should be 5A 250v Slow Blow

**Step 9:** Find and Resolder Cold Joints

The most common places I find cold joints are at the harness wires, the thermistor (R74), the fuse sockets (F1-F4), HV jack (J103), and yoke jack (J105). R6 and R29 (1.5-ohm 10 watt power resistors) can also get cold joints, as well as the jack for degaussing coil (J104).

**Step 10:** Clean jacks and connectors to main harness and J103, replace bad pins

These connectors can often get pitted. I've seen some so bad that they will break when you put any pressure on them. I first check these for continuity. If they are showing some resistance (over 3

ohms), I replace the pins. These are standard 0.093" molex pins. I get them from Hosfelt Electronics (800-524-6464)

### **A Summary of Replacement Parts For The Amplifone Deflection Board**

<b><u>Quantity</u></b>	<b><u>Description</u></b>	<b><u>Location</u></b>
2	3A 250v Fast Blow	F1, F2
2	5A 250v Slow Blow	F3, F4
2	MPSU57	Q4, Q14
2	MPSU07	Q5, Q15
2	3716	Q6, Q16
2	3792	Q7, Q17
1	1.2k, 1/2 watt resistor	R14
1	1.6K, 1/2 watt resistor	R72
2	10K pot	R76, R79
4	1N914	CR3, CR4, CR12, CR13
4	1uf @ 100v 105 degree low ESR axial capacitor	C7, C8, C9, C10
2	4700uf @ 100v 105 degree low ESR radial capacitor	C17, C18
2	100uf @ 50v 105 degree low ESR radial capacitor	C19, C20
1	33uf @ 250V 105 degree low ESR radial capacitor	C24
4	1N5342B 7.5v 5 watt Zener diodes	
4	1N4002 diodes	
15 inches	4mm or 5mm heat shrink tubing	
	Replacement jumper wires	

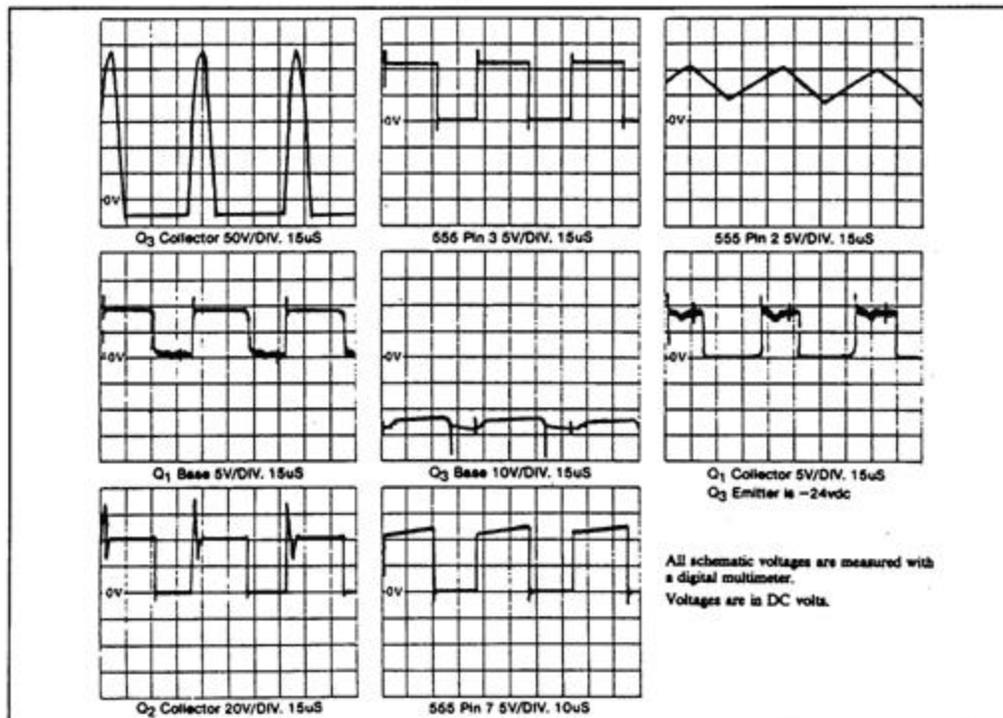
You may notice that according to the deflection schematic, there should be ferrite beads on the base (middle leg) of the MPS-U57's (Q4, Q14). If your beads or missing, or you otherwise need to replace them, I've found that Mouser part # 542-FB73-110 is a suitable replacement.

## AMPLIFONE TROUBLESHOOTING WAVEFORMS

Have an Oscilloscope handy? If so, these scope references should help you to diagnose your Amplifone deflection and HV boards. NOTE: You will need a working set of Star Wars game boards in test mode to make sense of these waveforms. If you do not have a Star Wars boardset, you are out of luck, although I suspect a crosshatch test pattern from just about any Atari color vector game board will work. Simply apply power to your Amplifone monitor, apply the proper Star Wars crosshatch test pattern, and scope the appropriate points with your probes.

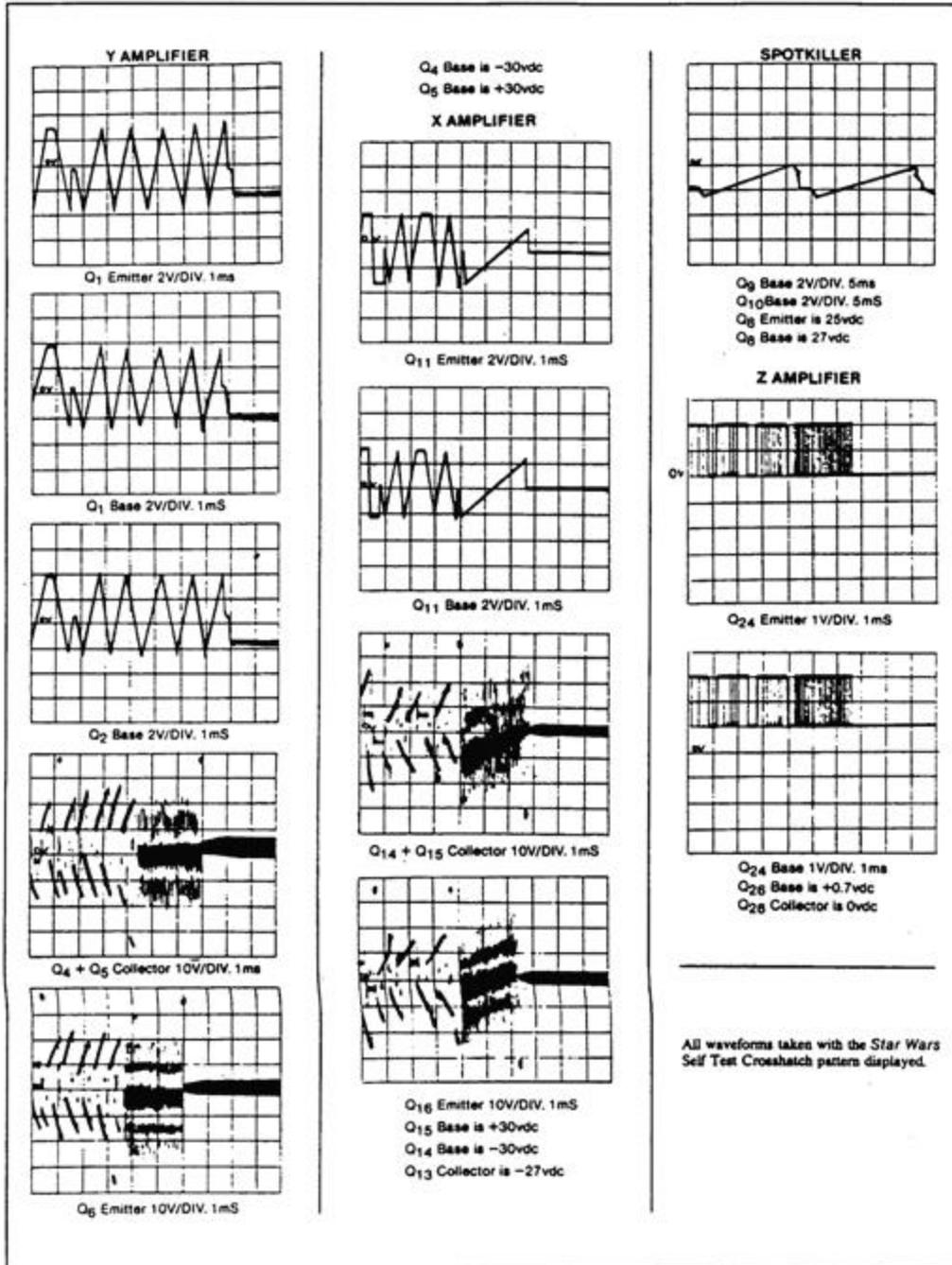
### Atari Star Wars Amplifone X-Y Monitor T'shooting Waveforms

To help troubleshoot the Atari Amplifone 19-inch Color X-Y Display Unit, the following waveforms are listed.



[anyone have a cleaner copy?]

AMPLIFONE TROUBLESHOOTING WAVEFORMS (cont.)



[anyone have a cleaner copy?]

---

## AMPLIFONE DEFLECTION PROBLEMS AND SOLUTIONS

---

**PROBLEM:** Can any other power transistors be used on the Deflection board?

**SOLUTION:** Yes, MJ15003 (NPN) replaces 2N3716, and MJ15004 (PNP) replaces 2N3792.

**PROBLEM:** The schematic for the deflection board shows CR5 as a 1N714 diode.

**SOLUTION:** The schematic number is wrong. It should be labeled a 1N751A zener diode. The rating of 5.1 volts is correct, however.

**PROBLEM:** The picture shrinks in from the negative X and Y sides of the screen.

**SOLUTION:** On some deflection boards, R35 and R12 may have a 15-Ohm resistor and a 30-Ohm resistor in parallel to get 10 Ohms. If these come loose, then some picture shrinkage may occur.

**PROBLEM:** The voltage is low on the supply lines on the Deflection PCB.

**SOLUTION:** There are jumpers on the supply lines. The jumpers are marked with a "W" on the schematic. They look like resistors on the board and have a single black band on them. The jumpers should have no resistance. However, the brown jumpers may have created some resistance to them. They should be replaced with a piece of wire. The white and tan jumper should have no problems, although you should replace them anyway.

---

## AMPLIFONE PICTURE TUBES AND DEFLECTION YOKES

---

From The Amplifone Manual:

19-Inch 90-Degree Medium-Resolution Cathode-Ray Tube Assembly

19" Color Cathode-Ray Tube: 134004-001

Deflection Yoke Wedge: 178058-001

Yoke and Harness Assembly: A201108-001

CRT Convergence Ring: 171010-001

25-Inch 90-Degree Color Cathode-Ray Tube Assembly

25" Color Cathode-Ray Tube: 134005-001

Deflection Yoke Wedge: 178058-001

Yoke and Harness Assembly: A201113-001

CRT Convergence Ring: 171010-001

The Amplifone 25" yoke assembly is the same as a Wells-Gardner 19K6100 and can be interchanged as required.

---

### **CRT (Neck) Pinout**

Here is the complete pinout of the Amplifone neck/tube.

1. G3 (focus grid)
2. not used
3. not used
4. not used
5. G1 (control grid)
6. G (green cathode)
7. G2 (screen grid; brightness)
8. R (red cathode)
9. H (heater)
10. H (heater)
11. B (blue cathode)

---

## MONITOR REPAIR LOGS

---

-----  
From: Bret Pehrson <bret@classicade.com>  
Newsgroups: rec.games.video.arcade.collecting  
Subject: Service Log: Amplifone Deflection #1  
Date: Sun, 18 Nov 2001 22:58:35 -0800

Recent service on a non-working Amplifone Deflection board:

Symptoms:

- Purchased non-working and untested

Visual Inspection:

- Checked/replaced incorrect fuses

Bench test:

- Hear deflection chatter, no HV
- No -30v output to HV board
- Check 0-ohm jumper W1 (original still installed), reads open

Repair:

- Replace W1 w/ jumper wire

Bench test:

- Operational, adjust X & Y size

Result:

- Unit working

Parts Used:

- 1" piece of jumper wire
- Approximate parts cost: \$0.01

Time:

- 15 minutes

-----  
From: Bret Pehrson (bret@classicade.com)  
Subject: Service Log: Amplifone Deflection #2  
Newsgroups: rec.games.video.arcade.collecting  
Date: 2001-11-18 22:59:07 PST

Recent service on a non-working Amplifone Deflection board:

Symptoms:

- Purchased non-working and untested

Visual Inspection:

- Checked/replaced incorrect fuses
- Replaced R74 (posistor for degaussing coil) w/ used part
- Replace Q14 (unknown replacement installed) w/ new equivalent SK3200
- Replace Q6 and Q7 (unknown or unidentifiable parts) w/ new parts
- Replace visibly worn silpads on Q4, Q5, Q14, Q15 w/ new mica insulators
- Install missing ferrite bead on base of Q14

Bench test:

- Powers up, have vertical deflection, horizontal deflection is unstable and bounces around 1" wide

Repair:

- Resolder worn/broken power/harness wires
- Inspect for cold-solder joints (should have done more thorough job during visual inspection), find 4: 1 at F4, 2 at J103, and 1 at J105

Bench test:

- Operational, adjust X & Y size

Result:

- Unit working

Parts Used:

- 2N3716 (1)
- 2N3792 (1)
- SK3200 (1)
- Posistor (used) (1)
- Ferrite bead (1)
- Mica insulators (6)
- Fuses (4)
- Approximate parts cost: \$10

Time: - 60 minutes

-----  
From: Bret Pehrson <bret@classicade.com>  
Newsgroups: rec.games.video.arcade.collecting  
Subject: INFO: Amplifone HV Diagnostics  
Date: Wed, 24 Oct 2001 22:03:01 -0700

I had to get my Star Wars up and running, so I had to do some Amplifone diagnostics. I'll share the experiences for anyone that needs it.

Problem: No high voltage.

I knew the game was working as I could see the game when connected up to my scope and it would play blind.

I capped and checked the HV and deflection boards using a kit from Bob Roberts. I noticed that previous owner had installed a BU406 on the HV board (big no-no), so replaced it with a BU406D. Still no HV. I could hear deflection, though.

Checked the BU406D for output w/ my scope -- had appropriate square-wave output.

Checked the two voltage regulators on the HV board (VR1 7824 which regulates the +24 volts, and VR2 7924 which regulates the -24 volts):

VR1

- Pin 1 (input): 32.9 volts (spec is +30 volts)
- Pin 2: ground
- Pin 3 (output): 23.75 volts (spec is +24 volts)

VR2

- Pin 1: ground
- Pin 2 (input): 0.65 volts (spec is -30 volts)
- Pin 3 (output): 0.74 volts (spec is -24 volts)

The 33 volts on pin 1 of VR1 is acceptable input. VR2 has no input voltage, hence no output. Power for the HV is through the deflection board, so I traced back the -30 source and the first component I hit after the connector is W1 (0-ohm jumper wire -- looks like a brown resistor). Tested w/ ohm meter (should read 0-3 ohms), read open. Replaced w/ thick-gauge wire. Retested and had power to VR2 and appropriate -24 volt output. (Note: it is a well-known fact that the 0-ohm jumper wires on the deflection and HV boards are extremely flaky and should be replaced w/ thick gauge wire.)

Result: High voltage, game plays.

-----  
From: Bret Pehrson <bret@classicade.com>  
Newsgroups: rec.games.video.arcade.collecting  
Subject: Service Log: Amplifone HV #1  
Date: Sun, 18 Nov 2001 22:58:29 -0800

Recent service on a non-working Amplifone HV board:

Symptoms:

- Purchased non-working and untested

Visual Inspection:

- Replaced Q3 (previous owner had installed incorrect BU406) w/ new BU406D
- Replaced VR2 (visible damage/exploded component) w/ new higher-power 7924CV
- Fixed several cold-solder joints
- Resoldered ground wires

Bench test:

- Powered up fine, adjusted brightness (SCREEN).

Result:

- Unit working

Parts Used:

- BU406D (1)
- 7924CV (1)
- Approximate parts cost: \$5

Time:

- 45 minutes
-

---

## OFFICIAL ATARI AMPLIFONE DOCUMENTS

---

### Amplifone Deflection Board Upgrade

(From the Major Havoc Installation Instructions (TM-268))

Perform the following procedure to modify the Amplifone Deflection PCB (see Figure 8).

1. Set the Deflection PCB on a clean work surface.
2. Connect two type-1N754A Zener diodes together, anode to anode, as shown in Figure 6. Use a soldering iron to solder the two anode leads together.
3. Connect two type-1N756A Zener diodes together and solder as described in step 2.
4. On the component side of the Deflection PCB, locate the yellow wire connected to resistor R1 (left center of the PCB).
5. Solder one cathode lead of the two type-1N754A Zener diodes (soldered together in step 2) to the yellow wire on the soldered side of the Deflection PCB as shown in Figure 8.
6. Solder the other cathode lead of the two type-1N754A Zener diodes to the nearest ground on the PCB as shown in Figure 8.
7. On the component side of the Deflection PCB, locate the orange wire connected to Resistor R24 (right center of the PCB). [Figure 8, entitled "Modifying the Amplifone Deflection PCB", showing a full scale picture of the solder side of the Deflection PCB, has been omitted for obvious reasons. It does not show anything that cannot be derived from the included text; it was merely a "visual aid".]
8. Solder one cathode lead of the two type-1N756A Zener diodes (soldered together in step 3) to the orange wire on the soldered side of the PCB as shown in Figure 8.
9. Solder the other cathode lead of the two type-1N756A Zener diodes to the nearest ground on the PCB as shown in Figure 8.
10. On the soldered side of the Deflection PCB, solder the cathode lead of a type-1N4002 diode to the emitter of transistor Q17 as shown in Figure 8.
11. Scrape away the green protective coating at a convenient point on the PCB, and solder the anode lead of the type-1N4002 diode to the collector of transistor Q17 as shown in Figure 8.
12. On the soldered side of the Deflection PCB, solder the anode lead of a type-1N4002 diode to the emitter of transistor Q16 as shown in Figure 8. **[NOTE: This is the opposite of what you did in step 10.]**
13. Scrape away the green protective coating at a convenient point on the PCB, and solder the cathode lead of the type-1N4002 diode to the collector of transistor Q16 as shown in Figure 8.
14. On the soldered side of the Deflection PCB, solder the cathode lead of a type-1N4002 diode to the emitter of transistor Q7 as shown in Figure 8. **[NOTE: This is the opposite of what you did in step 12.]**
15. Scrape away the green protective coating at a convenient point on the PCB, and solder the anode lead of the type-1N4002 diode to the collector of transistor Q7 as shown in Figure 8.

16. On the soldered side of the Deflection PCB, solder the anode lead of a type-1N4002 diode to the emitter of transistor Q6 as shown in Figure 8. [NOTE: This is the opposite of what you did in step 14.]
17. Scrape away the green protective coating at a convenient point on the PCB, and solder the cathode lead of the type-1N4002 diode to the collector of transistor Q6 as shown in Figure 8.

---

### Atari Field Service Bulletin

(courtesy of Al Kossow)

TECH TIP from the Atari FIELD SERVICE DEPARTMENT  
Atari Color X-Y Display Deflection PCB

You should do the following modifications to help prevent the Amplifone Deflection PCB from failing.

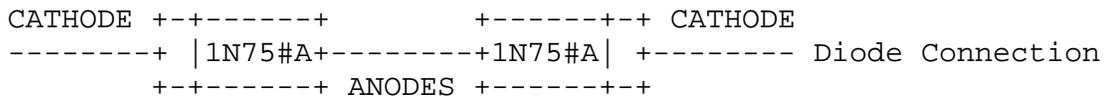
#### Parts List

<u>Quantity</u>	<u>Description</u>	<u>Part No.</u>
6	Type-1N4002 Diode	31-1N4002
2	Type-1N754A 6.8V Zener Diode	131002-001
2	Type-1N756A 8.2V Zener Diode	32-1N756A
2	12 Ohm 5% 1/4 W resistor	110000-120

1. Connect the two 1N754A Zener diodes together as shown below in Figure 1. The connection is made as follows: bend the anode ends of both diodes into a "fish-hook" pattern. Hook the two fish-hooked leads together, and solder them. Remember that too much heat will destroy the semiconductor material.
2. Connect the two 1N756A Zener diodes together as shown in Figure 1. Use the same technique as described in Step 1 above.
3. Remove diode CR2 and solder in a type-1N4002 diode in its place. [NOTE: On older monitors, CR2 is labeled D602.]
4. Remove diode CR11 and solder in a type-1N4002 diode in its place. [NOTE: On older monitors, CR11 is labeled D702.]
5. Remove resistor R12 and solder in a 12-Ohm, 1/4W resistor in its place. [NOTE: On older monitors, R12 is labeled R609.]
6. Remove resistor R35 and solder in a 12-Ohm, 1/4W resistor in its place. [NOTE: On older monitors, R35 is labeled R709.]

7. Find the Y-Deflection Circuit (upper left area of the schematic). Resistor R1 has two leads to it. Find the lead that goes to the yellow wire. Connect this lead to the cathode of one of the type-1N754A diodes. Connect the cathode of the other type-1N754A diode to ground.
8. Find resistor R24. It has two leads: one runs to an orange wire. Connect this lead to the cathode of one of the type-1N756A diodes. Connect the cathode of the other type-1N756A diode to ground.
9. Find the type-2N3792 transistor Q17. You will be installing a type-1N4002 diode across the transistor's emitter and collector. Solder the cathode lead of the type-1N4002 diode to the emitter, and solder the anode to the collector of this transistor.
10. Find the type-2N3617 transistor designated Q16. You will be installing a type-1N4002 diode across the transistor's emitter and collector. Solder the cathode lead of the type-1N4002 diode to the collector, and solder the anode to the emitter of this transistor. [NOTE: this is the opposite of what you did in Step 9.]
11. Find the type-2N3792 transistor Q7. You will be installing a type-1N4002 diode across the transistor's emitter and collector. Solder the cathode lead of the type-1N4002 diode to the emitter, and solder the anode to the collector of this transistor.
12. Find the type-2N3716 transistor Q6. You will be installing a type-1N4002 diode across this transistor's emitter and collector. Solder the cathode lead of the type-1N4002 diode to the collector and solder the anode to the emitter of this transistor. [NOTE: this is the opposite of what you did in Step 11.]

FIGURE 1.  
Making Fishhook Connections

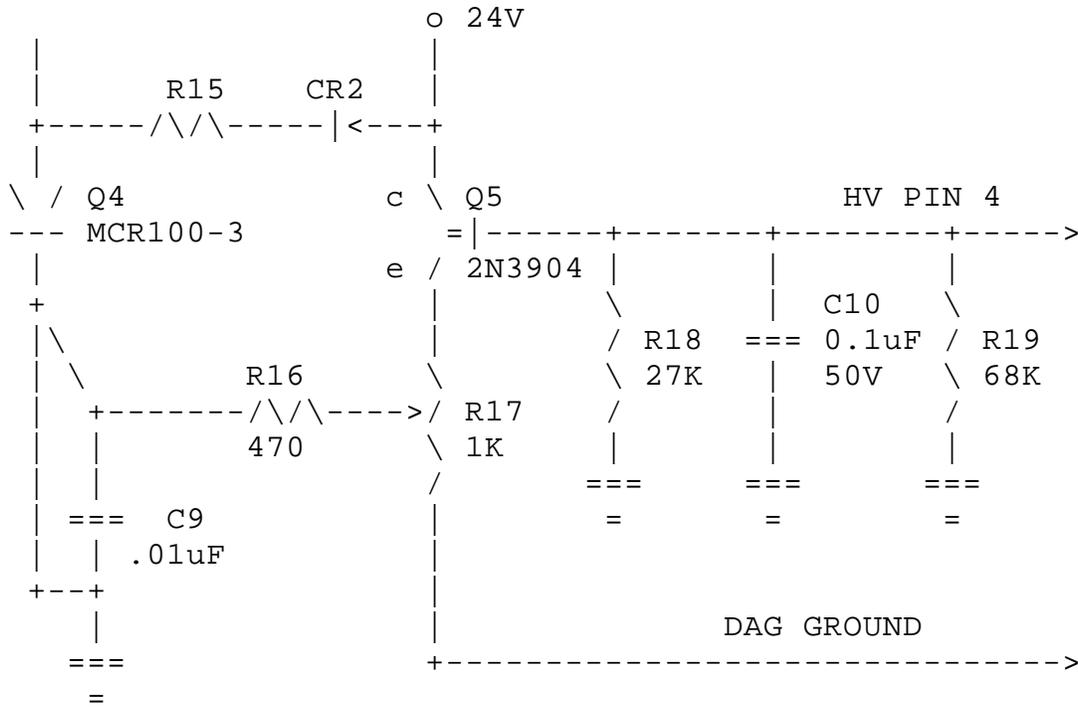


[NOTE: You get the idea; snub diodes across the deflection amps, back-to-back Zeners on the input to ground. Since I made these modifications, I have not had a deflection amp go out (but I am running the game with the back off now, too.)]

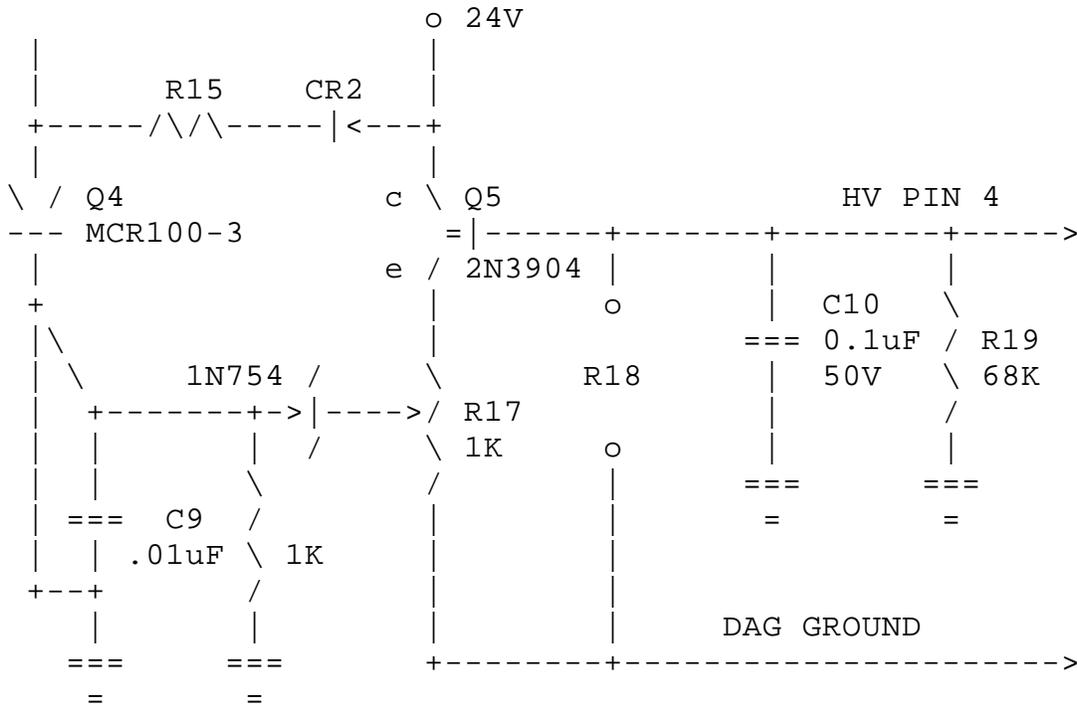
**Amplifone HV Revision Schematics**

(ASCII art by Gregg Woodcock)

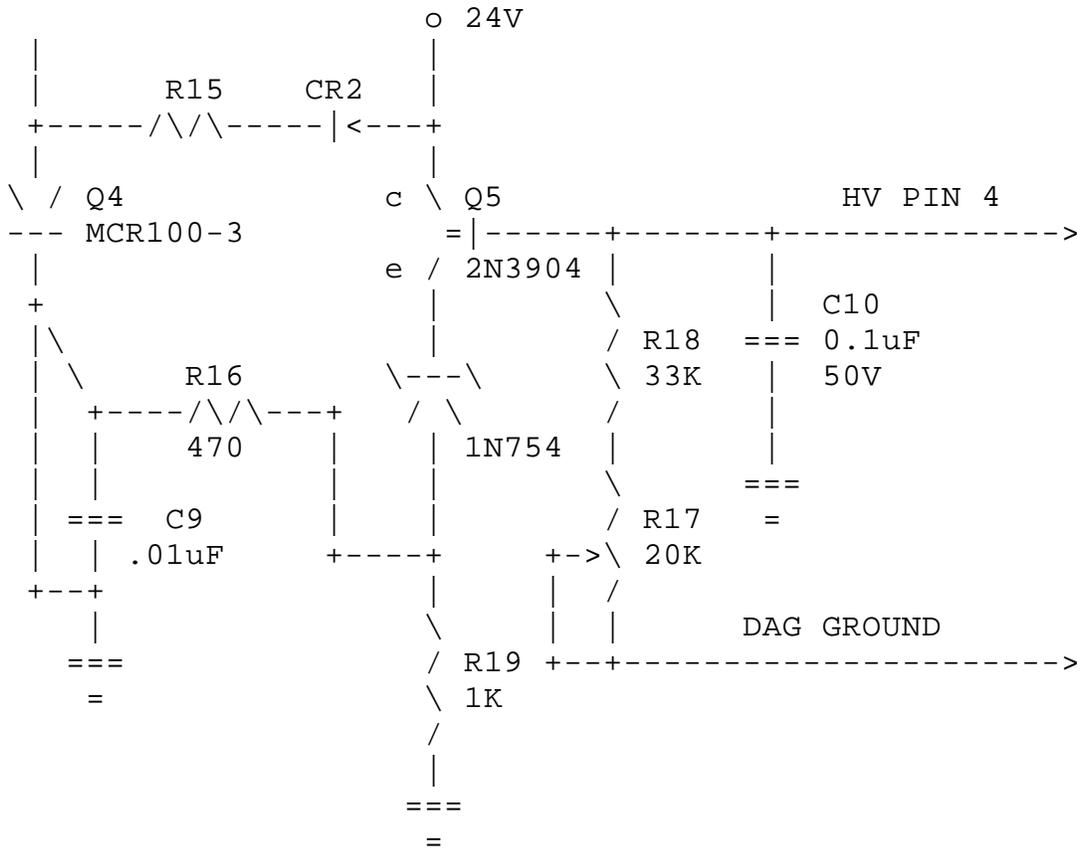
**ORIGINAL AMPLIFONE HV PCB**



**FIRST REVISION AMPLIFONE HV PCB**



**SECOND REVISION AMPLIFONE HV PCB**



---

## Appendix A: Common Ground Connections

---

From: John Robertson <jrr@flippers.com>  
Newsgroups: rec.games.video.arcade.collecting  
Subject: TechTIP: How to make VECTOR MONITORS very RELIABLE!  
Date: 22 Oct 2001

It's been a little while since my last Tech Tip, but this is something that's been on my mind for a while now, and a posting in the Vector mail-list got the following response from me...:

Vector monitors blow up because the ground reference for the monitor drifts relative to the logic boards (MPU and video) when the power supply connections overheat. This will then bias the input signals offset enough to overdrive the outputs. Hence my argument for chucking the original power supply and putting in a switching supply. I started doing that about ten years ago and have not lost a single Electrohome/Sega monitor since. I assume this also kills Tempest/Star Wars/Major Havoc/... monitors etc. Those pesky grounds get a few ohms resistance and all sorts of nasty things happen.

I first discovered this on Gottlieb pinballs over ten years ago-the ground for the regulator would overheat the pin/wiper contact which would become a small resistor and thus the ground of the MPU would drift up relative to the cabinet ground, which also happened to be the ground path for the driver transistors. When the MPU ground would change to about 0.5 to 0.7VDC above cabinet ground the base of the transistors would then start to conduct as the MPU would be trying to turn off the transistors, but the Emitters are tied to the cabinet ground. Hence the transistors would start to conduct... You will recall that transistors generate far more heat when they are used at the beginning of their working range rather when they are switched completely on and off as in regular vector monitors (or solenoid drivers, etc.). So in a little while, it croaks. No obvious cause...replace the transistors and everything works. For now...

So get **VERY GOOD GROUND (COMMON) CONNECTIONS BETWEEN THE MONITOR, MPU AND POWER SUPPLY** for reliability!!!!!!!!!!!!!!!!!!!!!!!!!!!! Solder fat conductors with nasty heavy gauge connectors between each component in the system. Put in healthy **SWITCHING SUPPLIES!**

Happy vectors will result.

John :-#)#

---

## Appendix B: Testing Transistors

---

Most of the failures in the Amplifone color vector monitor (as is the case with most electronic devices) are semiconductor failures, specifically, the transistors. All transistors discussed in this document can be tested in the same way; it does not matter if they are the large chassis-mounted transistors or the tiny PCB-mounted transistors. With the transistors out of circuit, set your multi-meter on Rx1K scale and use the following procedures.

**NOTE: ANALOG AND DIGITAL MULTI-METERS REQUIRE DIFFERENT TESTING PROCEDURES FOR TRANSISTORS!** Digital meters always show infinite resistance for all 6 combinations (if you accidentally get your skin involved it will show something around 2M Ohms). The best way to test transistors with a DMM is to make use of the "diode test" function, which will be described after the analog test. For both methods, if you read a short circuit (0 Ohms or voltage drop of 0) or the transistor fails any of the readings, it is bad and must be replaced.

Why do Digital Voltmeters read open circuits on diodes and transistors?

Because of the ability to use amplifiers, DVM can use much smaller voltages to check resistance. For the most part this is a good thing. It allows you to check resistors in circuit, without turning on things, like transistors.

Diode junctions (which there are two of in a transistor) do not "turn on" until they reach somewhere around 0.4 ~ 0.7 volts, depending upon what they are made of, and a lot of other stuff. In a way, diode junctions are similar to neon light bulbs, they act like open circuits until the right voltage is reached, and then they act like shorts, until the voltage drops below the critical threshold. Without proper current limiting, the diode junctions explode. The thing about diodes is that they only do this in one direction, if you switch the test leads, they do not conduct at all. (Well, until the voltage gets **much** higher, and then it is a bad thing. ;^)

Sometimes you want to be able to "turn on" the diode junctions (to test them), so DVMs have a "Diode" test mode. This places enough voltage on the test leads to turn on the diode junction. The number you read on most meters is the actual turn on voltage threshold across the diode.

---

### TESTING TRANSISTORS WITH AN ANALOG OHMMETER

For type NPN transistors, lead "A" is black and lead "B" is red; for type PNP transistors, lead "A" is red and lead "B" is black (**NOTE:** this is the standard polarity for resistance but many multi-meters have the colors reversed; if the readings do not jive this way, switch the leads and try it again). Start with lead "A" of your multi-meter on the base and lead "B" on the emitter. You should get a reading of 2.5K Ohms. Now move lead "B" to the collector. You should get the same reading. Now try the other 4 combinations and

you should get a reading of infinite Ohms (open circuit). If any of these resistances is wrong, replace the transistor. Only 2 of the 6 possible combinations should show a resistance and that value should be 2.5K Ohms; none of the resistances should be 0 Ohms (shorted).

---

## **TESTING TRANSISTORS WITH A DIGITAL MULTI-METER**

Set your meter to the diode test. Connect the red meter lead to the base of the transistor. Connect the black meter lead to the emitter. A good NPN transistor will read a JUNCTION DROP voltage of between 0.45v and 0.9v. A good PNP transistor will read OPEN. Leave the red meter lead on the base and move the black lead to the collector. The reading should be the same as the previous test. Reverse the meter leads in your hands and repeat the test. This time, connect the black meter lead to the base of the transistor. Connect the red meter lead to the emitter. A good PNP transistor will read a JUNCTION DROP voltage of between 0.45v and 0.9v. A good NPN transistor will read OPEN. Leave the black meter lead on the base and move the red lead to the collector. The reading should be the same as the previous test. Place one meter lead on the collector, the other on the emitter. The meter should read OPEN. Reverse your meter leads. The meter should read OPEN. This is the same for both NPN and PNP transistors. Thanks to Randy Fromm <randy@randyfromm.com> for this excellent summary of the diode test method.

---

## **END**

---