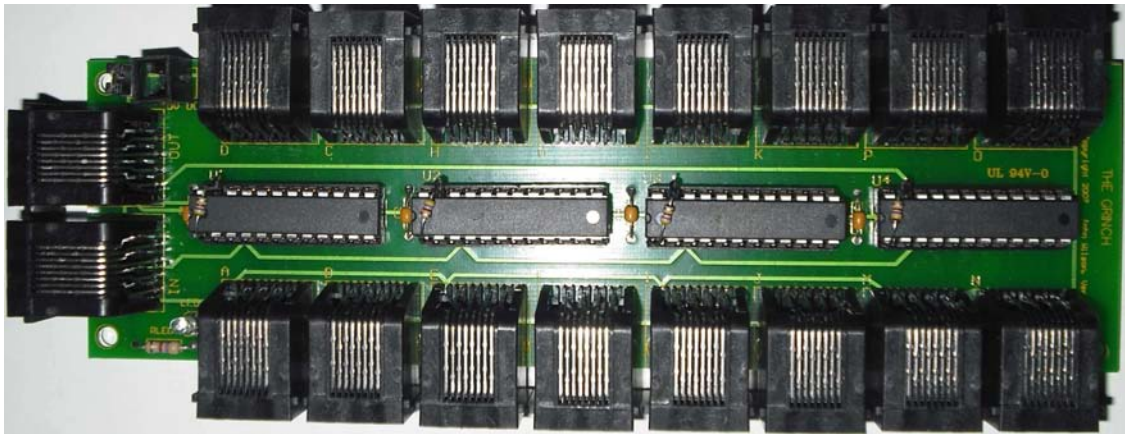


Adding current limiting resistors to your Grinch controller



By Robert P. Martin

In developing my Grinch DMX Dimmer interface I noticed that some people were having problems with erratic operation and unwanted channels turning on.

I have seen some Grinch controllers that would cause unwanted channels to come on when certain combinations of channels were turned on and I have also seen some controllers react strangely to the input signal, such as erratic operation.

This is most likely due to the fact that to do dimming the data is sent at a very fast rate to the controller and if anything interrupts this, you will see it as erratic or unwanted operation of channels on the controller.

I have heard from users of the Grinch DMX dimmer that they need to have the cable to the Grinch controller very short to work properly; sometimes only a couple of inches! In my testing I have used up to a 6 foot CAT5 patch cable with no problems, so this caused me to believe that there was something else going on.

After some investigating, it was found that the problem most likely stemmed from the fact that the Grinch design does not include current limiting resistors on the IC's. What this does is it causes excessive heat and current to be drawn by the IC's on the Grinch controller, which in turn appears to affect the channel data integrity.

When running a Grinch as an on/off controller this does not appear to happen, but when trying to do dimming it can really cause havoc!

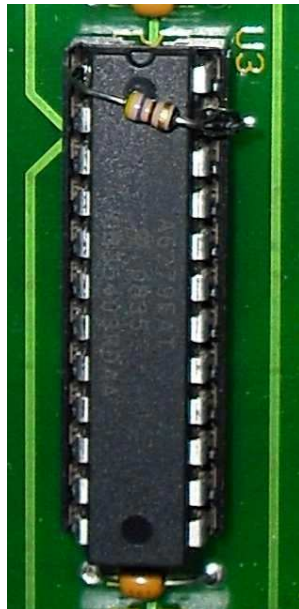
In testing several Grinch controllers both with and without current limiting resistors, it was found that without the resistors the Grinch IC's ran much hotter, had much higher (almost 90% more) idle current draw and many times had problems with erratic operation. The controllers with resistors installed run much cooler, draw significantly less current and rarely had any problems.

My suggestion? Add current limiting resistors to your Grinch controller.

How you ask? It's very easy.

There are several ways you can do this, but there are two methods I personally use.

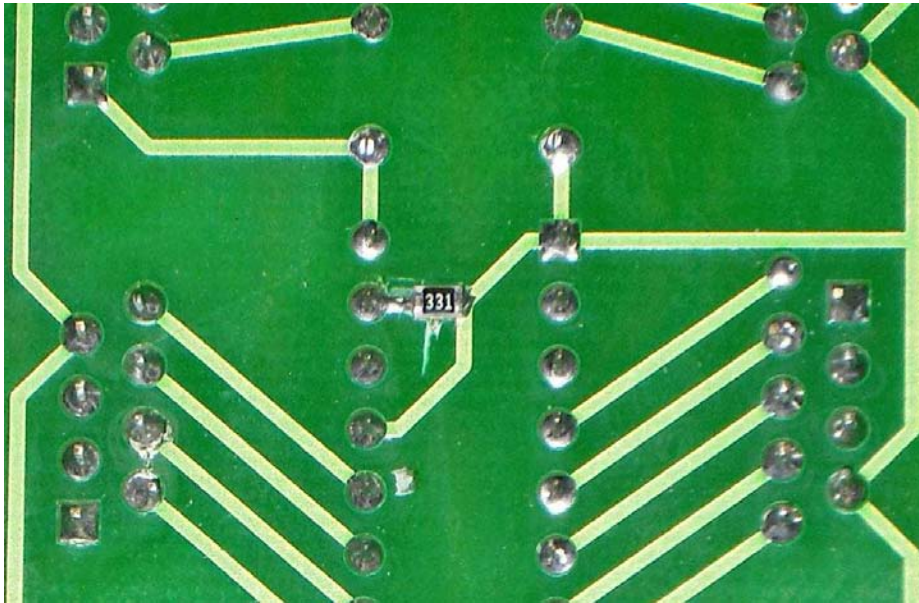
One is to add the resistors directly to the Grinch IC's like this:



In this picture you will see I have soldered a 1/8 Watt resistor directly to the IC between pin#1 and pin#23. Pin#1 (on the upper left) is connected to ground, and pin#23 is connected to what is called the "REXT" pin, which sets the output current limit for all of the IC's outputs. Pin#23 has been bent up and cut short and the resistor lies directly across the top of the IC. You can also use a 1/4 Watt resistor, but it is a tighter fit between the pins.

The problem with this method? While it's an easy fix to do, if you ever have to replace the IC you need to replace it with one that has been modified such as this one, not to mention there's the possibility of breaking the pin off when bending it up. The advantage of this modification is that if you already have your controller mounted and it is difficult to remove, then as long as you can remove the IC's this is a much easier modification to do.

Another method is to modify the PC board, which is more involved since you will need to cut traces, like this:



Note that this modification is done on the **BOTTOM** (solder side) of the PC board.

In the center of this picture you will see I have cut the trace that runs between pin#1 (the square pad on the upper right) and pin#23 under IC U2 and soldered in a surface mount resistor, although you can use a 1/8 or 1/4 Watt resistor as well. I chose to use surface mount resistors since I'm comfortable with soldering them, and I like the clean look it produces.

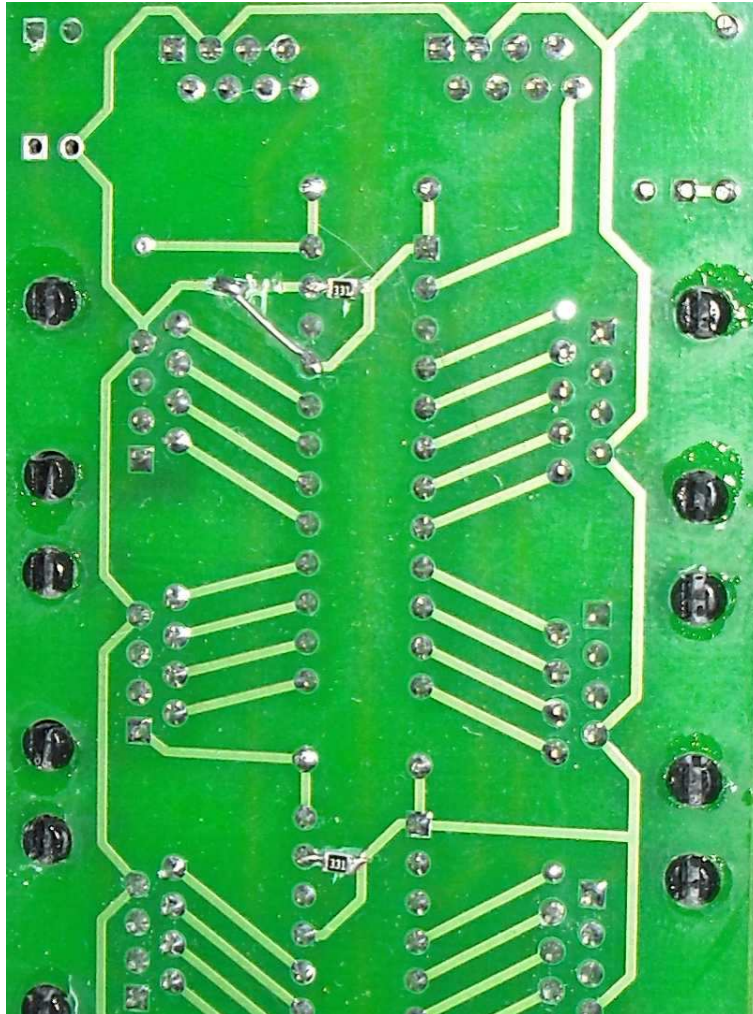
The problem with this method? It's more difficult than the first method and requires cutting of small traces on the PC board, something not everyone is comfortable with.

This is my preferred method since once you have done it you can change the IC if needed and not have to worry about soldering a resistor on the replacement IC.

NOTE: These pictures are of the V1.0 PC boards supplied by John Wilson.

There is one caveat if you should decide to modify your PC board; the traces under U1 are slightly different than U2, U3 and U4.

Here's a picture of this difference

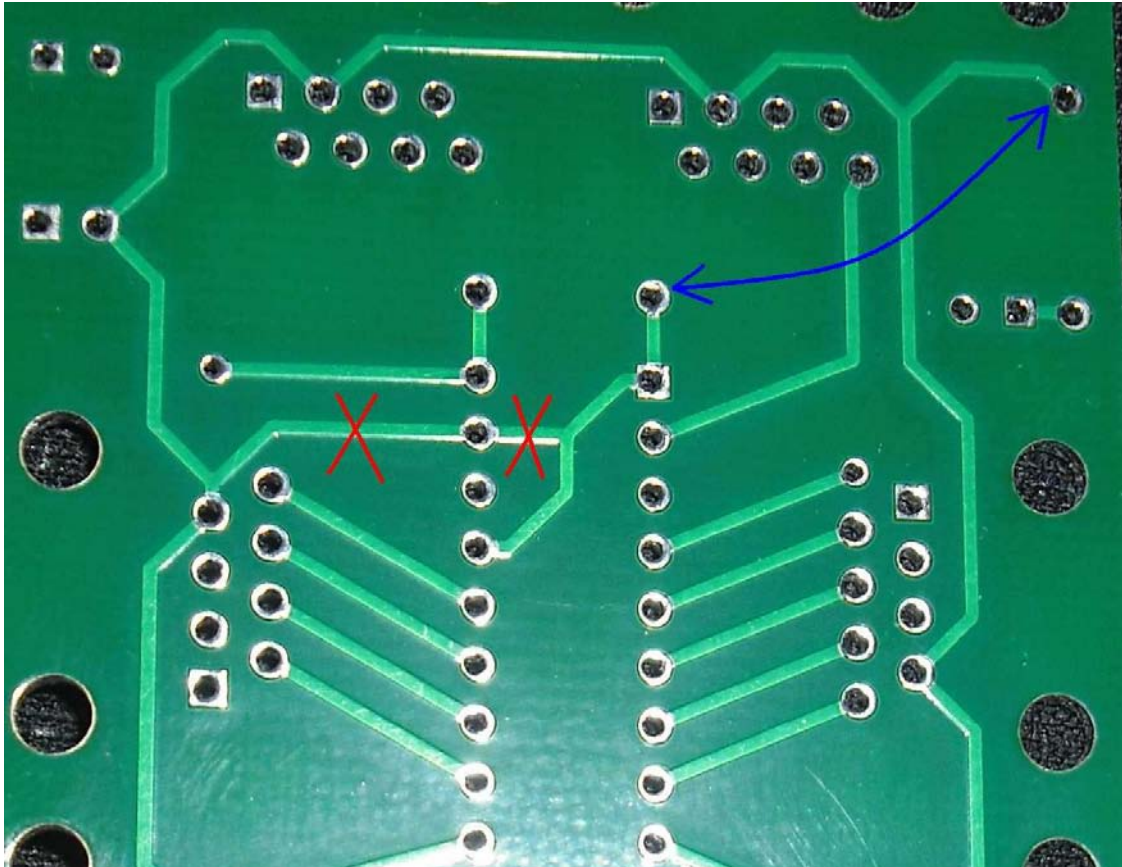


At the top is U1, which has an extra trace just to the left of pin#23. Compare it to U2 below it and you will see the difference.

This trace must be cut and then re-routed to the trace that runs between pin#1 and pin#21 as it is a trace that supplies ground for this IC. The other three IC's get their ground supply from a trace on the right, connected through pin#1.

Once this trace has been rerouted then the procedure is the same as the other three IC's.

Here is a more detailed picture of where to cut the traces under U1

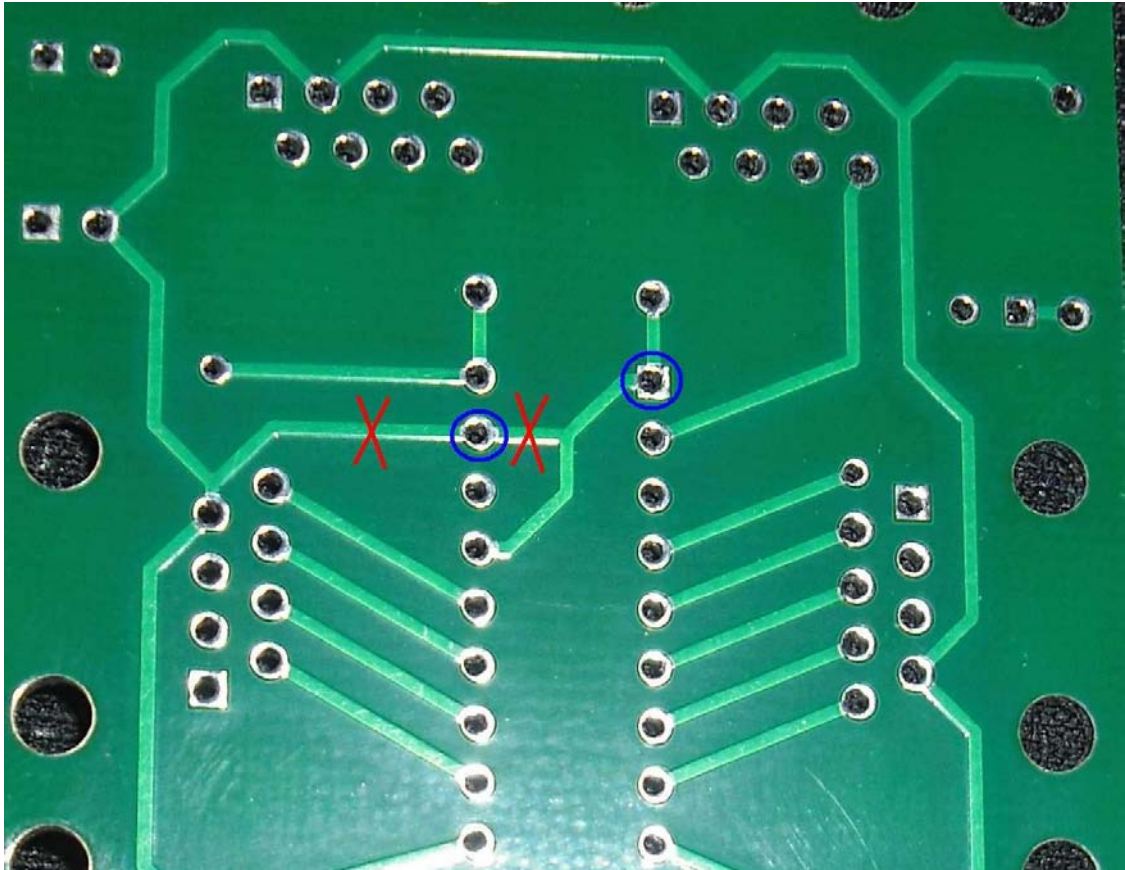


The **RED** X's are the traces that you would need to cut, and the **BLUE** line shows where you would solder the jumper to ground pin#1.

Note that this ground trace circles the bottom perimeter of the board, so there are other possible places you could place this jumper, but the one I show is fairly easy to do and avoids the possibility of shorting out any other pins or traces.

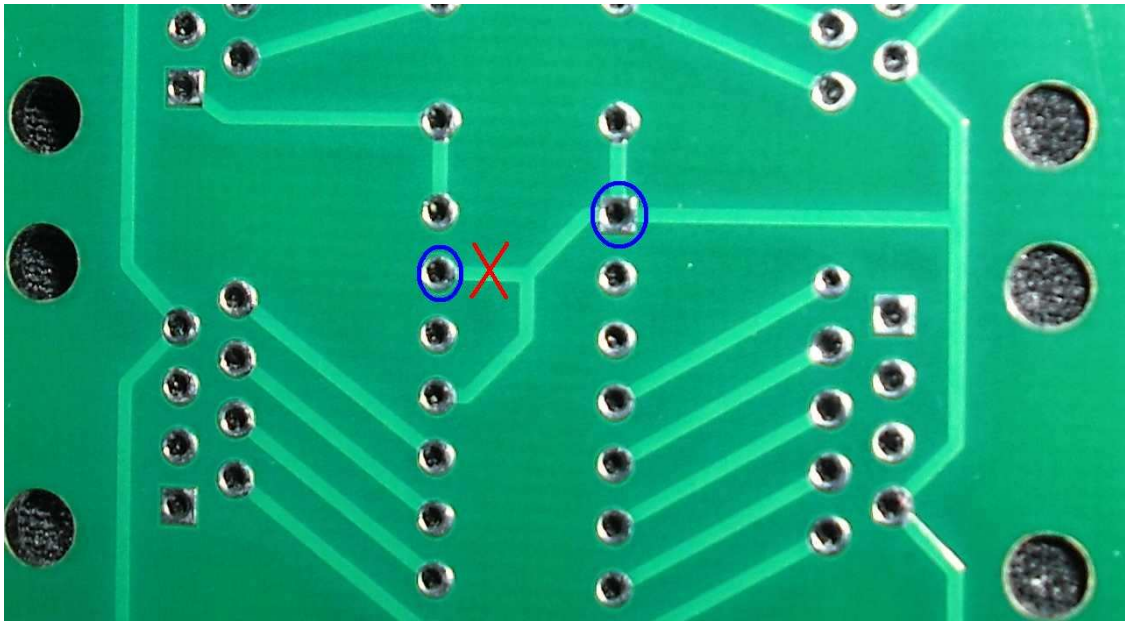
Once you have cut the traces and rerouted the ground you will need to solder on the resistor.

Here's a picture of where this would go under U1



The **RED** X's are the traces that you cut and **BLUE** circles show the pins that you would need to solder the resistor to.

Now that the modification to U1 has been done you can do the other IC's. As mentioned before, the traces under U2, U3 and U4 are identical. Here's a picture that shows where you would cut under U2, U3, and U4



You would cut where the **RED** X is and you would solder the resistor between the pins with the **BLUE** circles.

OK, by now you are probably asking “so what value of resistors do I use?”

The answer to that is it depends... If you were paying attention to this in the previous pictures, you will see that I used both 330 ohm and 470 ohm resistors. The reason for this is that I run SSR's that have 680 ohm resistors in them which limits the current to about 5ma per channel, so as long as you are capable of sending at least that much current then it really doesn't matter. For reference, a 470 ohm resistor limits the output current to about 40ma and a 330ohm would be about 55ma. .

I also have some of my own designs that have the opto isolators directly connected to the IC outputs and in that case I use a 3.9K resistor which limits the current output to around 5ma.

My thinking on this is if you have resistors in your SSR's, then all you need is the minimum resistance, which for the Allegro A6279 is about 225 ohms which will limit the output current to about 75ma.

Use this chart to determine what resistance for whatever current output you need. If you are using IC's other then the Allegro 6279, consult the manufacture's data sheets for their suggested resistor values.

