Another Failure 20 with measurements and some test mode?

bumerang #1 March 21, 2019, 11:28pm

Hi,
My new MK-312BT is also suffering from "Failure 20".

Because all the measurements and scope screenshots I decided to create a new topic - I hope it is OK.

To be honest I am lost. I cannot say I fully understand the signal generation and the test procedure. So debugging is quite a challenge for me.

I will be grateful for any suggestions or observations.

My PCB is **1.3B.**
FW is mk312-bt/firmware/Custom Boot Message f005-MK312-BT/ElectrodesReady.bin.

I did few modifications to the MK-312BT but I don't think any of them is "Failure 20" related.

- R35 and R46 are 220k instead 200k (I guess 10% off is not the problem).
- I didn't use MAX232.
- I didn't use battery VREG – I am powering it from lab PSU directly to the battery input.
- I added a reverse protection diode (inspiration from 1.3R).
- I modified MIC input like it is in 1.3R.
- I used 22pin IDC cable and used only 2GND pins (instead of 4).
- **I used 42TU200 transformers** – some other user tested it and it is supposed to work just fine. Center tap output is not connected.
- I used two 1R resistors in parallel for R30 – I also tried 3 and 4 in parallel to fight the 42TU200 lower DC resistance.

Sometimes the ATMEGA does not startup correctly. The LCD is not correctly initialized (I guess) the top row is full of rectangles, the backlight is ON. The crystal oscillator is ticking correctly at 8MHz. Fuse bytes are correct. I tried to add 100nF cap between RST and GND to slow down the startup time – It made it even worse. The power consumption is 55mA in this case. 5V and 9V rails are OK.

Question – with a "Failure 20" on the LCD the **backlight is OFF – is this normal?** Also every 10s or so the backlight and output LEDs flashes and nothing else happens.

Measurements:

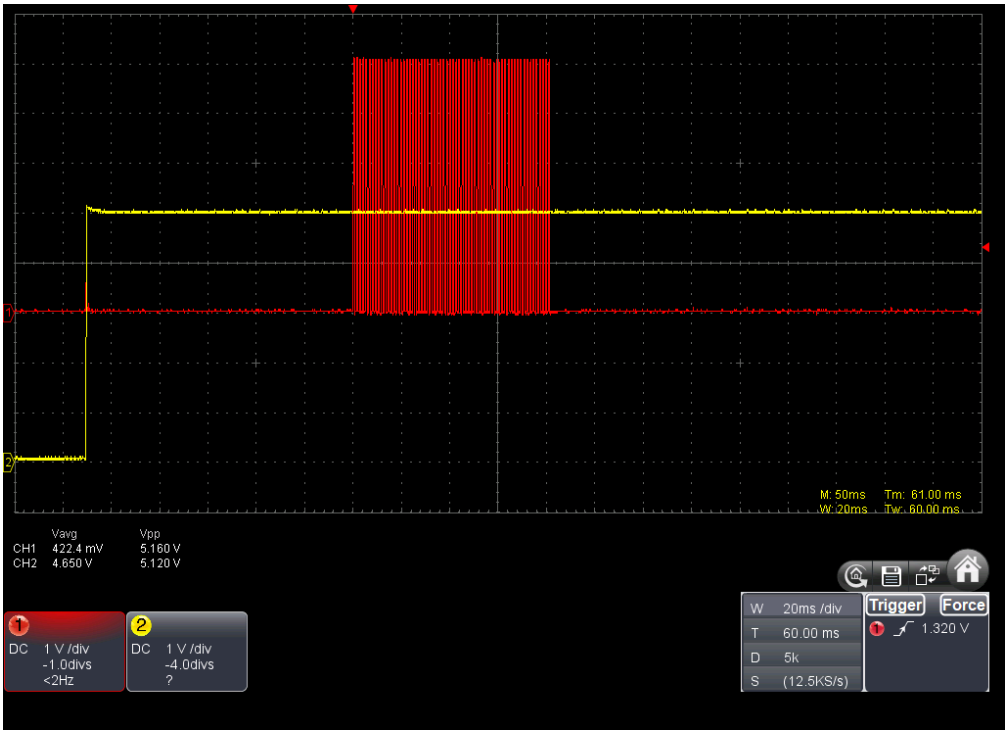
Power consumption 42mA at 12V.
5V rail: 5.10V
9V1 rail: 8.92V (I might replace 9V VREGs for more precise ones in the future)
9V2 rail: 9.20V

****Signals after power up – Yellow = 5V rail, Red = signal**

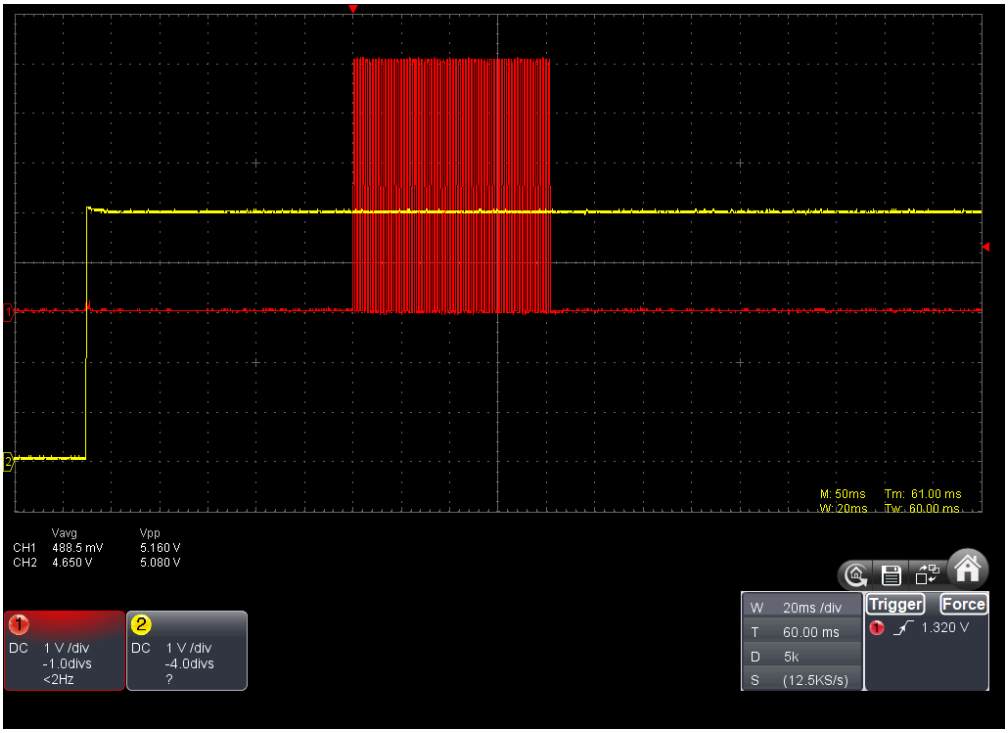
PB0: no signal

PB1: no signal

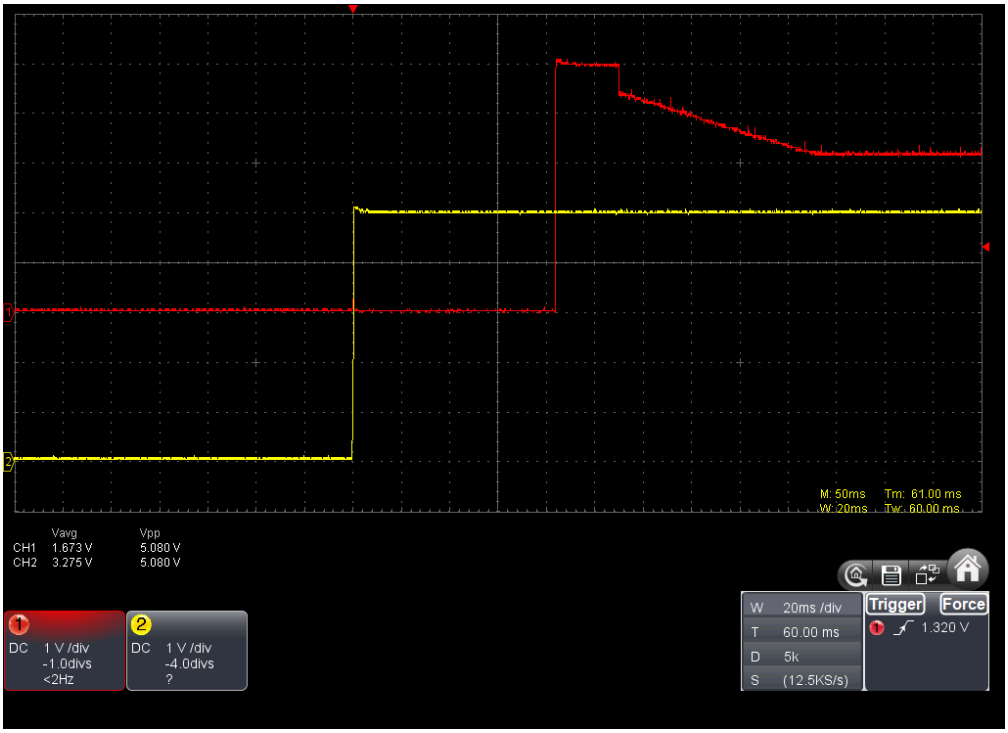
PB2:



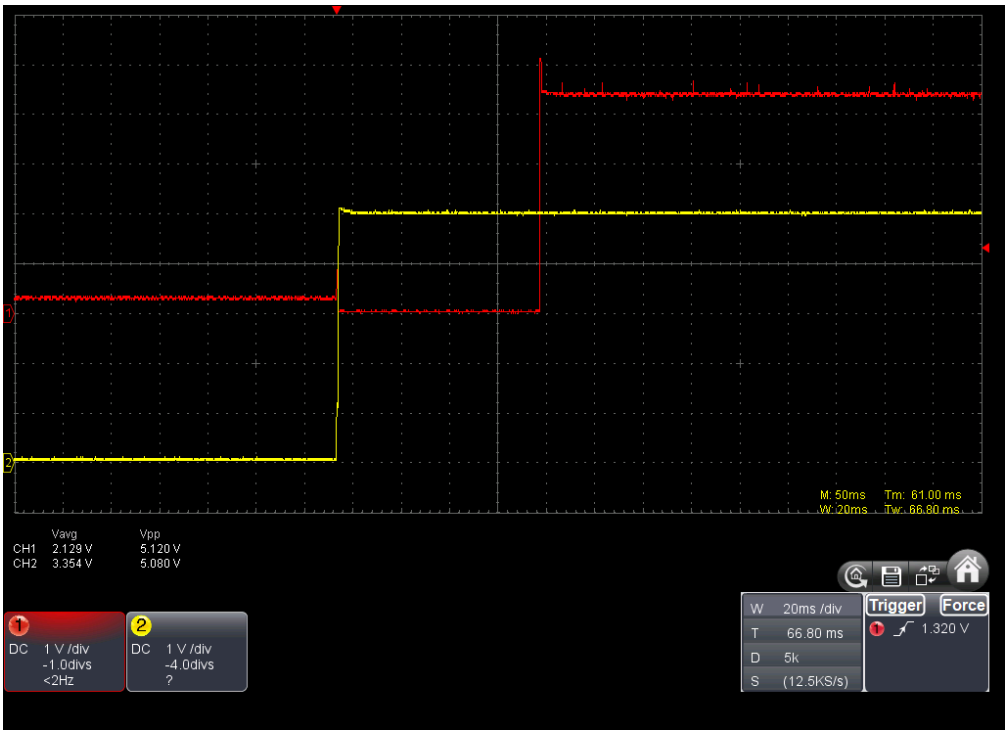
PB3:



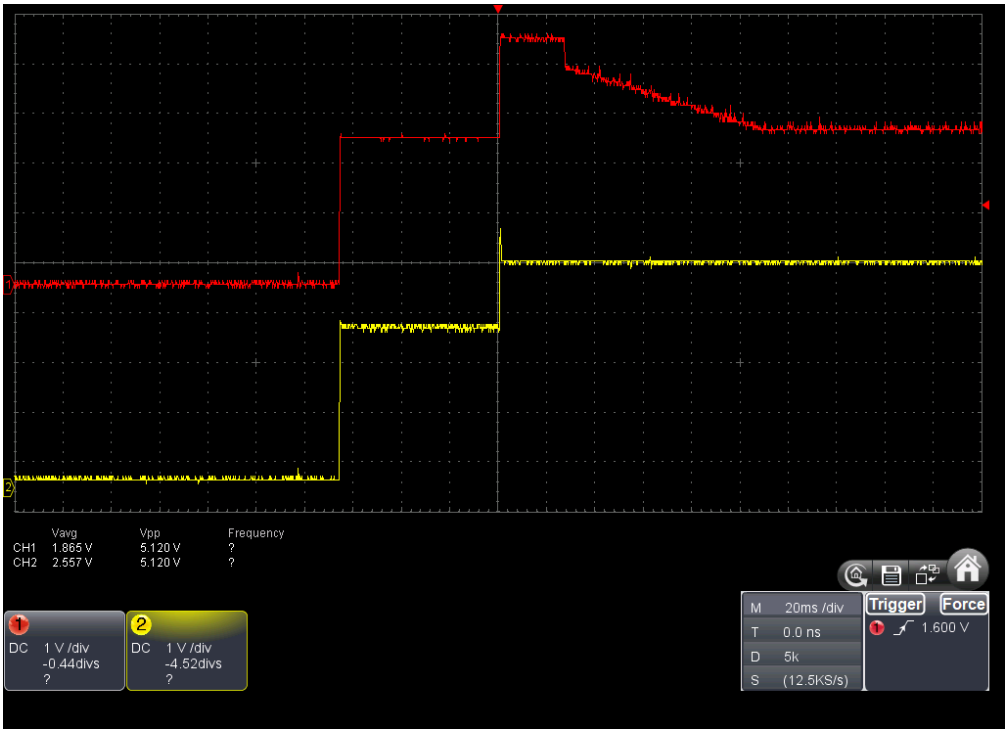
LTC1661 OUT A:



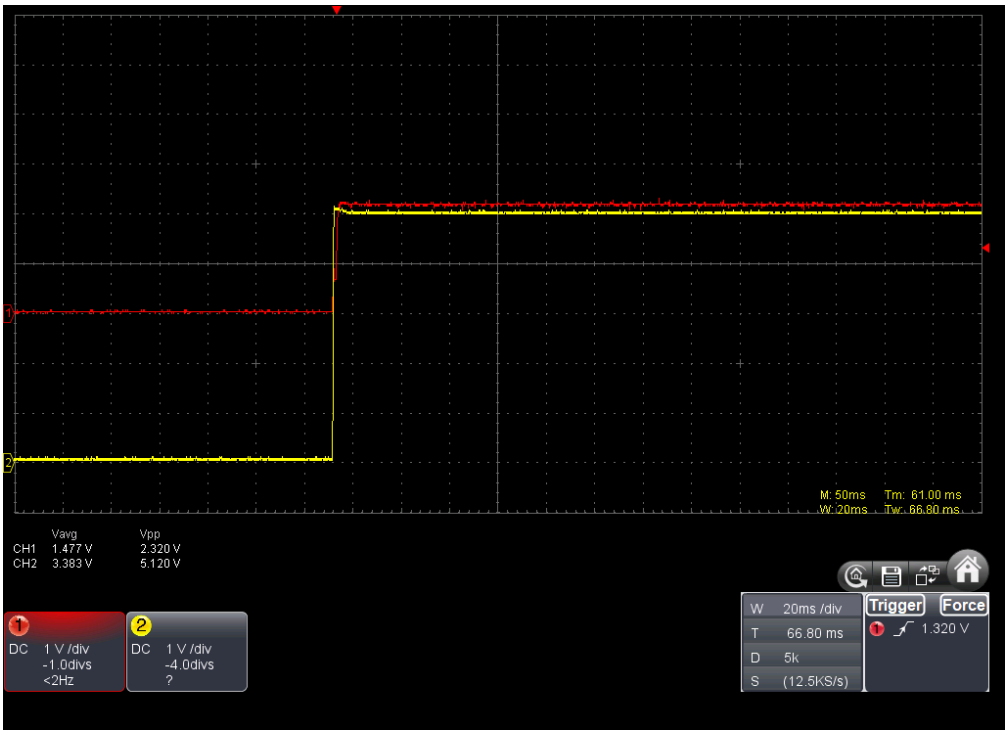
LTC1661 OUT B:



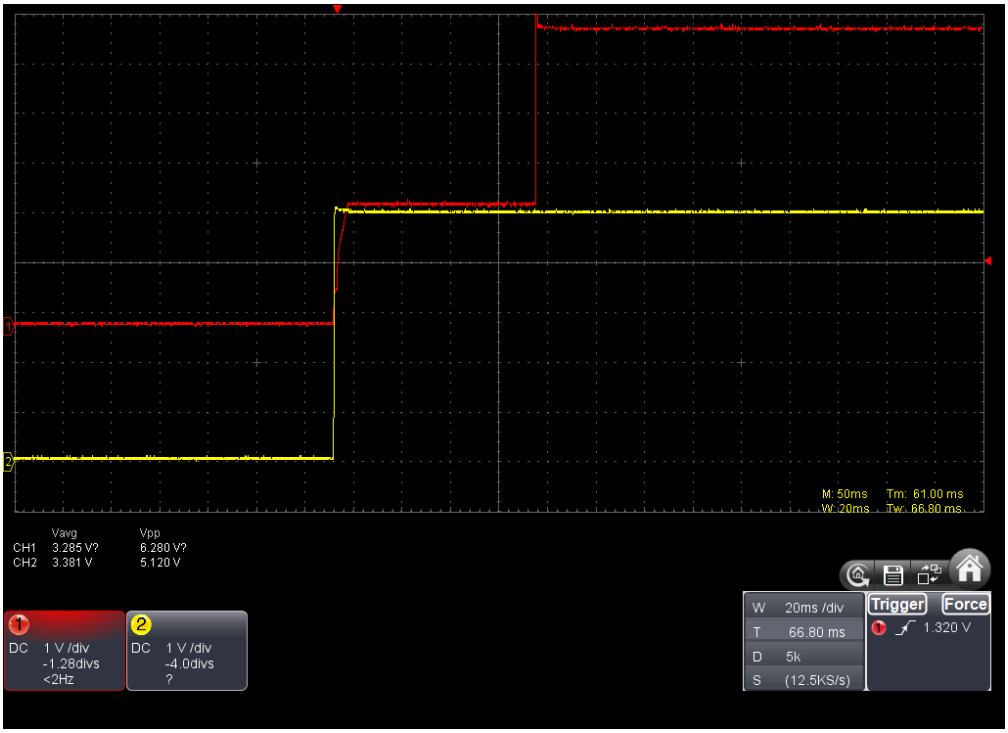
LTC1661 – Red = OUT A, Yellow = OUT B:



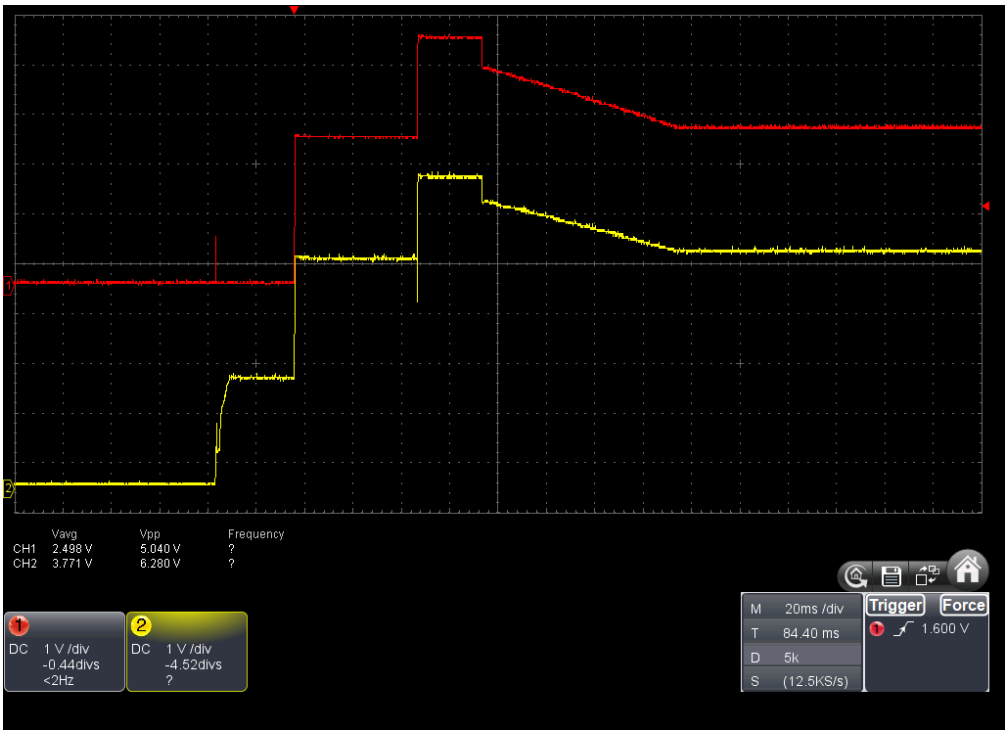
U10A output:



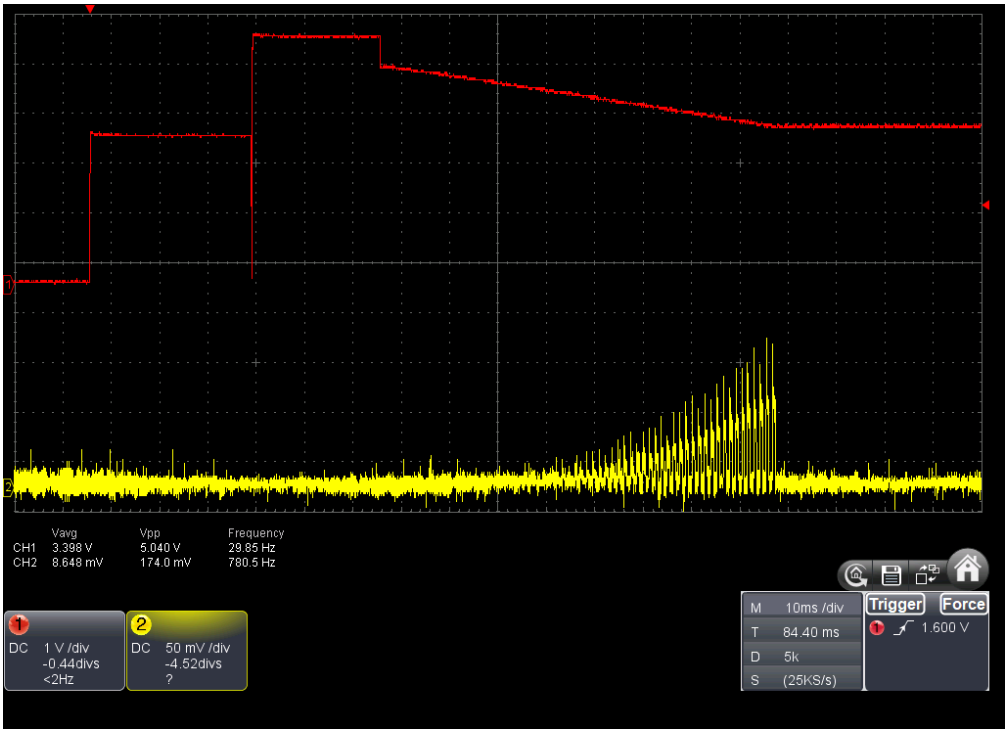
U10B output:



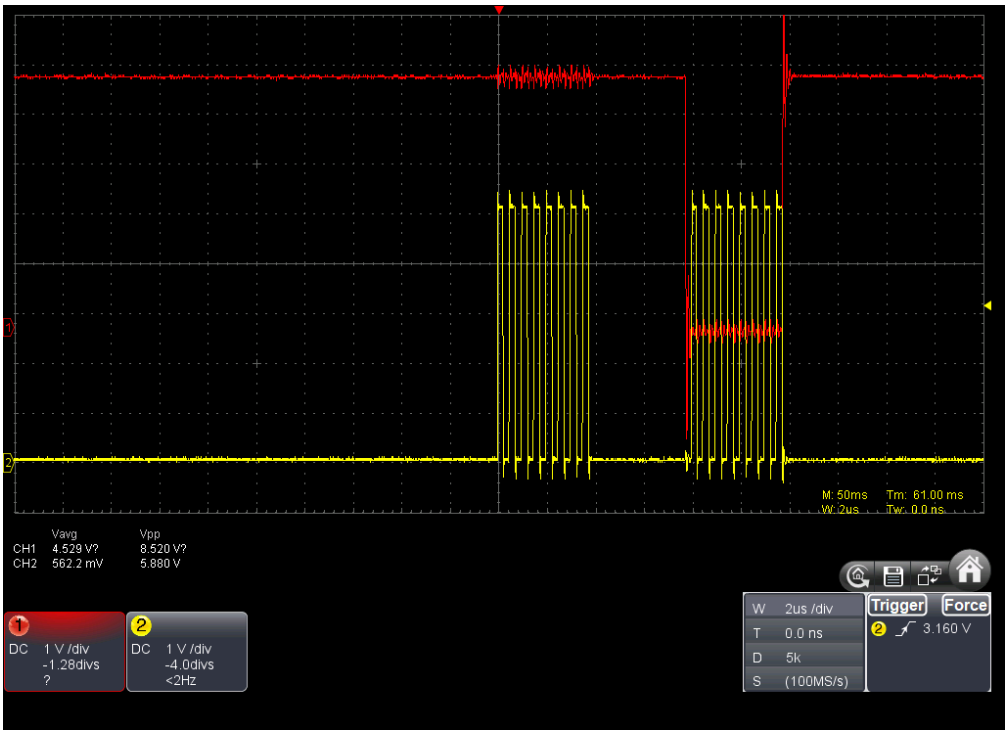
Red = OUT A, Yellow = U10A output:



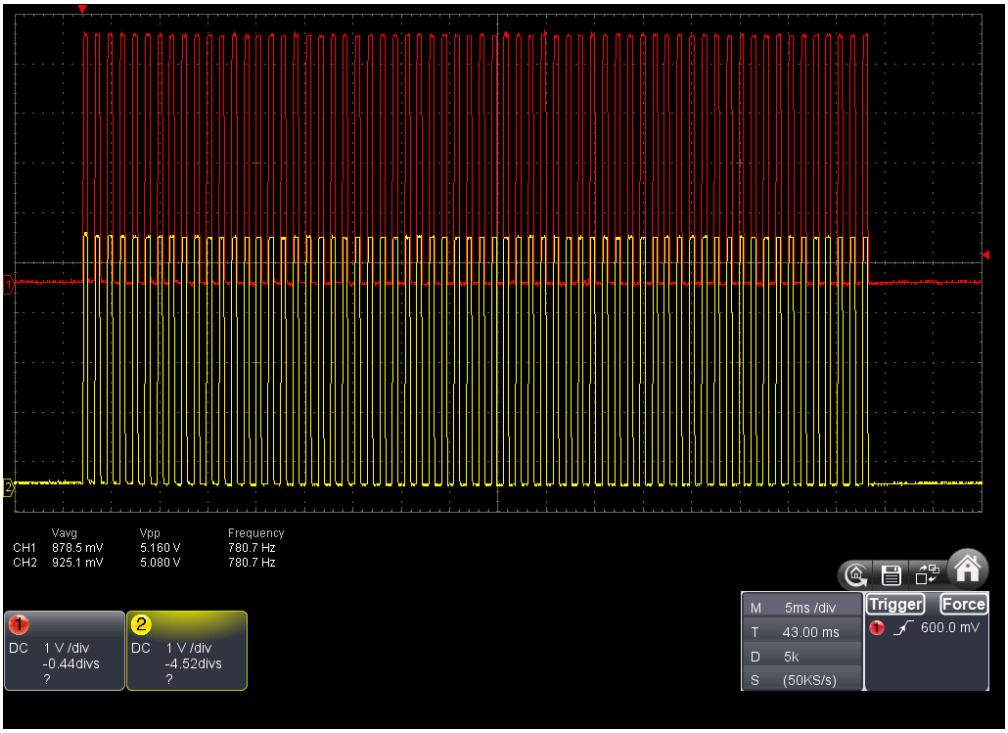
Red = OUT A, Yellow = R30 current sensing resistor:



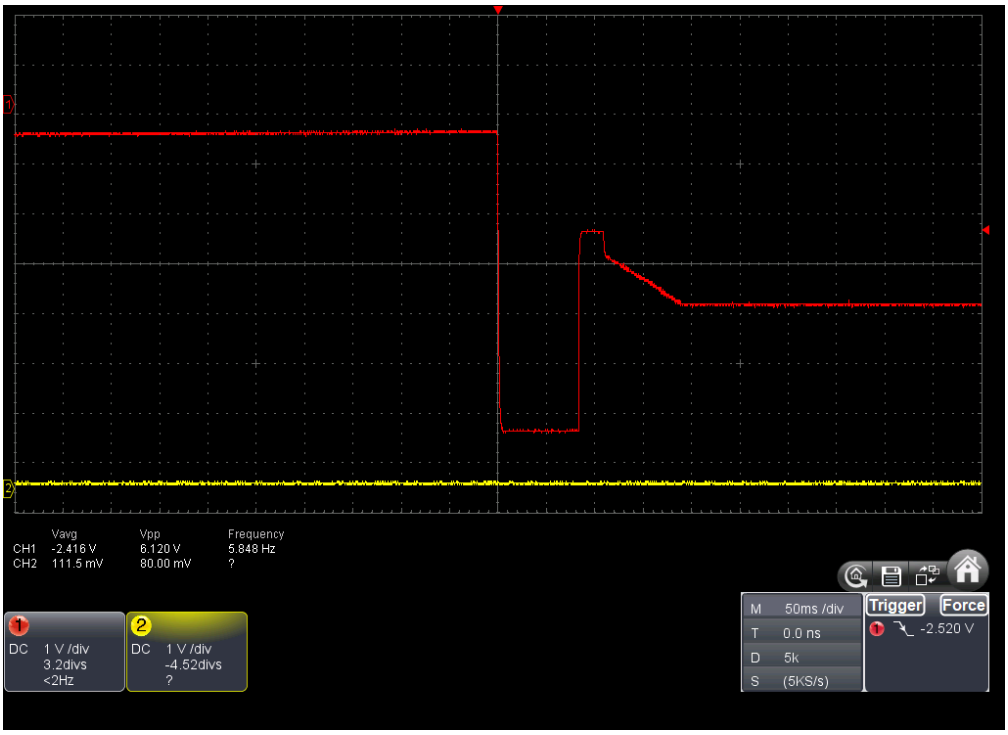
LTC1661– Red = DIN, Yellow = SCLK:



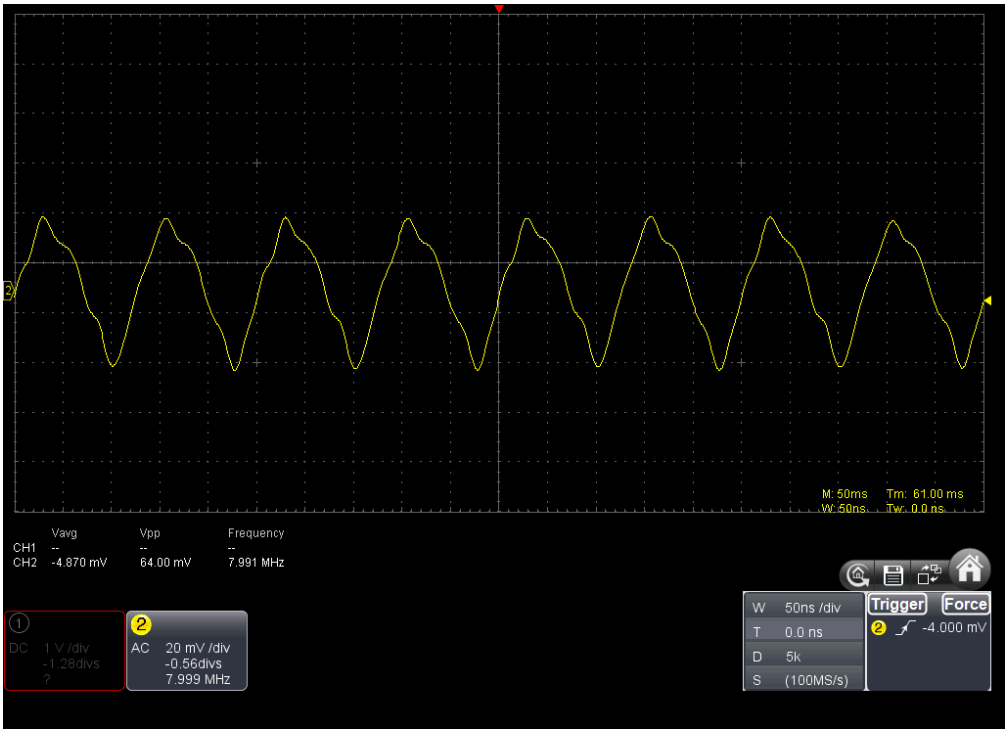
Red = Q1 gate, Yellow = Q2 gate (against ground):



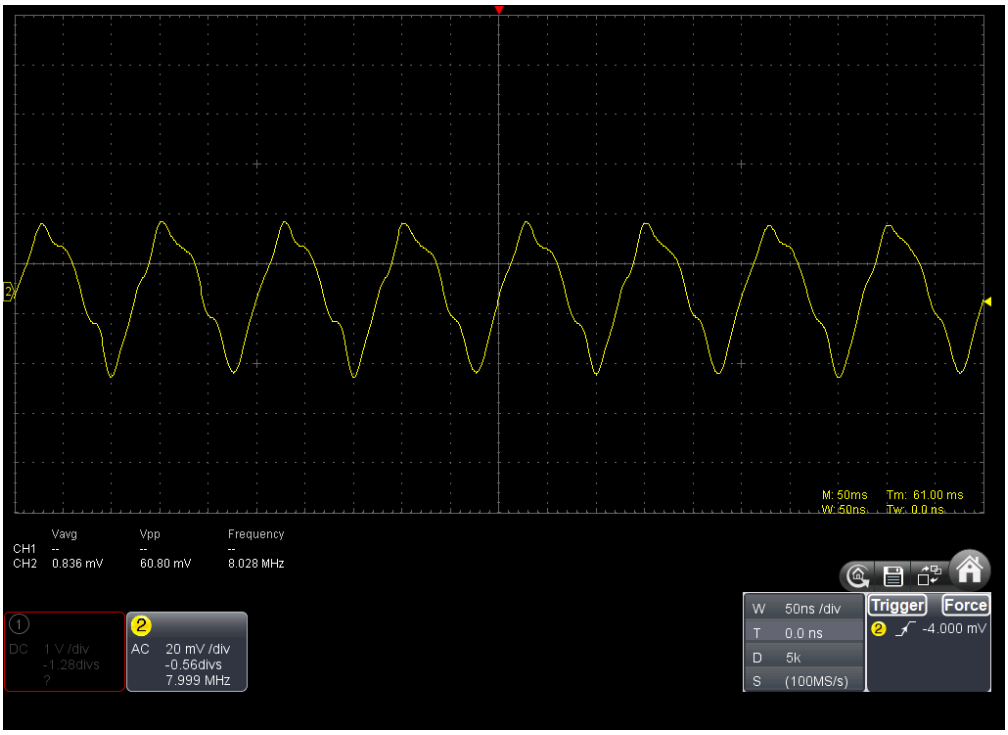
Red = Q3 gate against 9V1 rail (floating scope):



Crystal when the ATMEGA does not startup correctly:

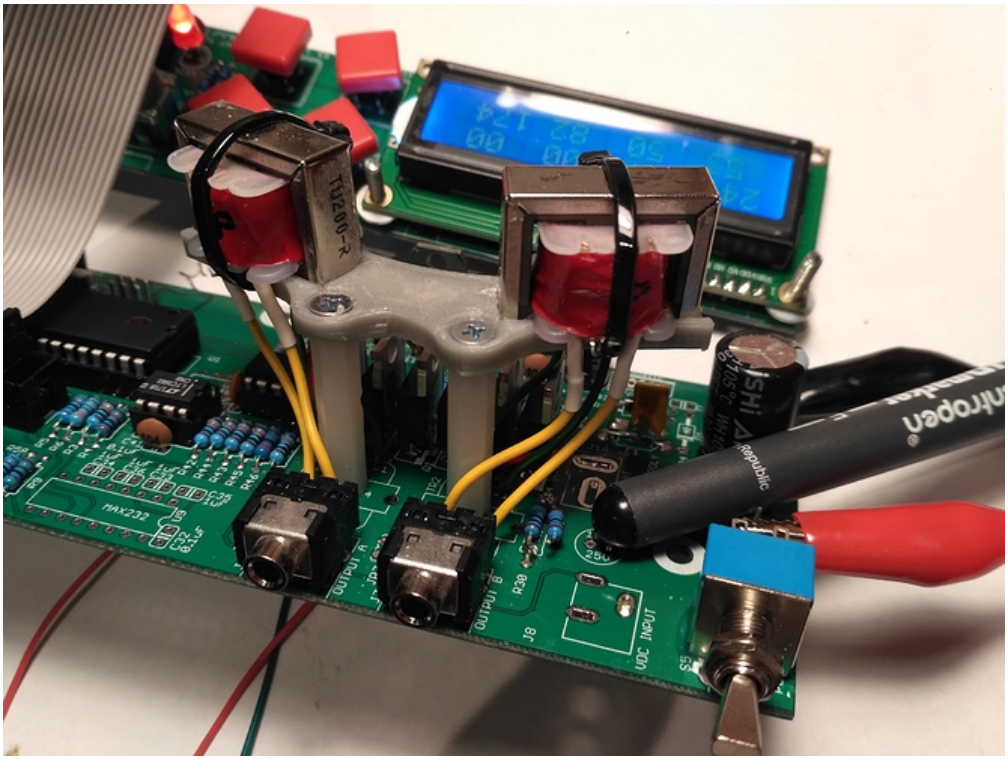


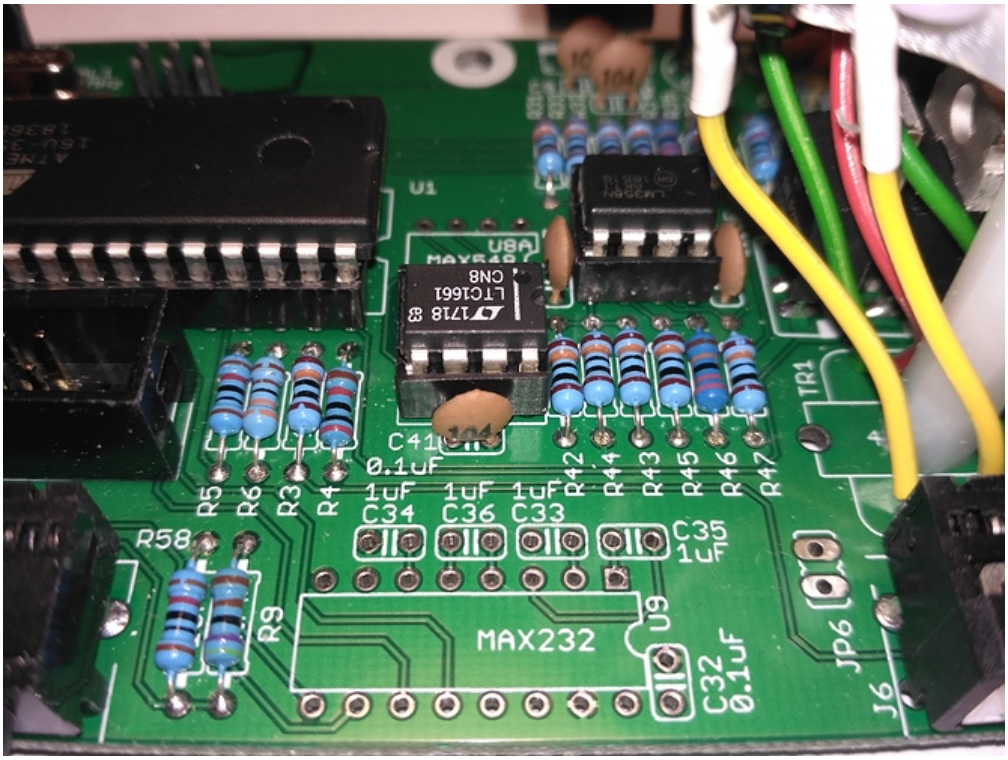
Crystal when the ATMEGA does startup correctly:

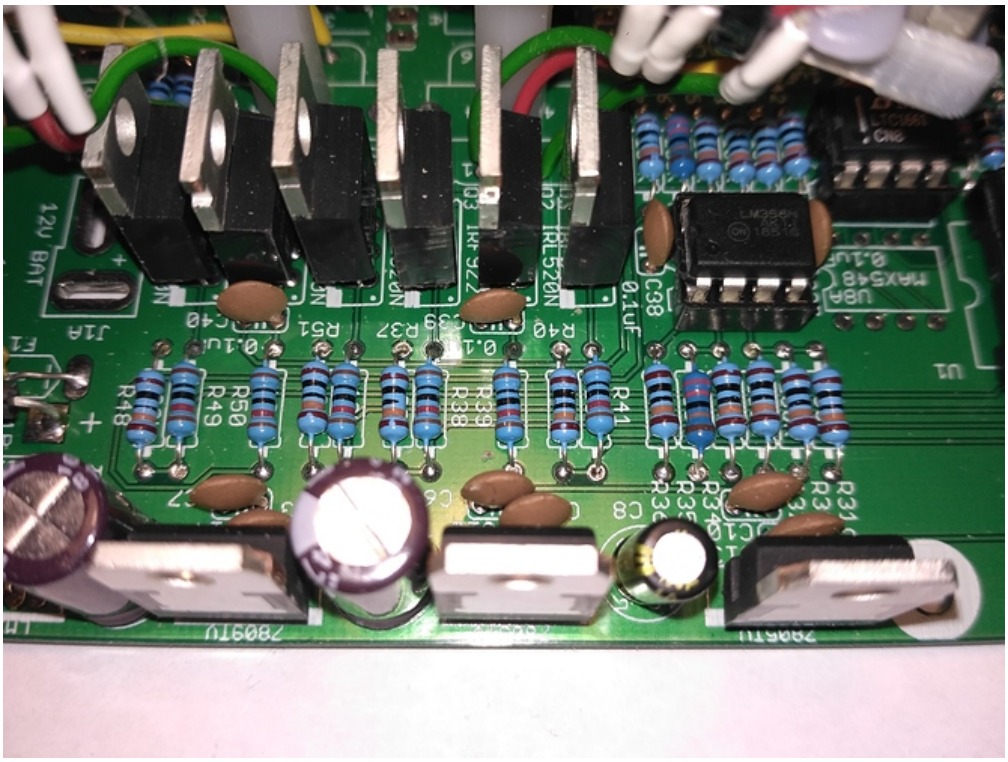


// OK, I found it is documented already
I also managed to enter some kind of a test mode. It happened when I tried to measure signal between ground and Q3 gate (floating scope). It also happens when the R30 value is too high – 1R instead of 0.5R. The MK-312BT then boots into this – see the photo. Buttons pressed change one number, Multi Adjust changes other number, and output intensity pots do nothing.

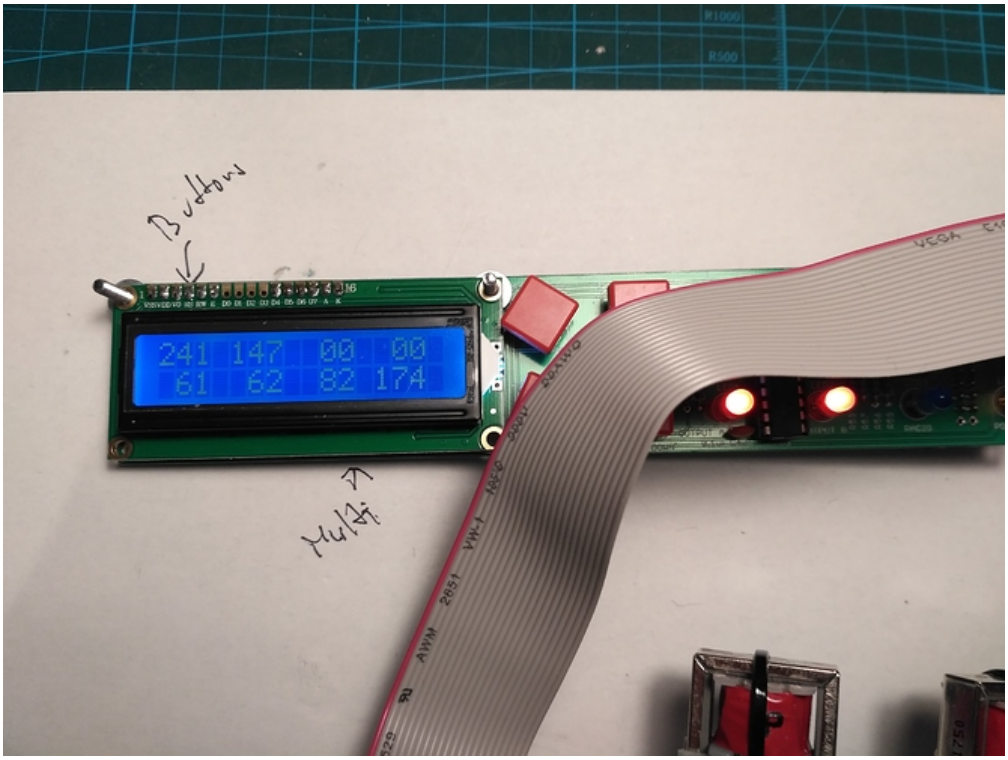
Photos:


TIP200-R
MAX232
OUTPUT A
OUTPUT B
R28
VDC INPUT





Test mode? When I disconnected one of two 1R in parallel (R30).



Can someone confirm these waveforms?

- **Is the voltage at R30 suppose to be so low - peaks are only 100mV?**
- **Is the output coming from the DAC correct?**
- **Is the output from U10A correct?**

I am unable to measure impulses across transformer coil (floating scope) - it keeps getting into “test mode” - not sure how is this possible.

I will try to desolder and measure all FETs. But I measured each of them before soldering.

Any ideas, please?

1 Like

Cerb #2 March 22, 2019, 12:20am

[quote=“bumerang, post:1, topic:629”]
Question – with a “Failure 20” on the LCD the **backlight is OFF – is this normal?** [/quote]

I don’t know the unit well enough to answer the other questions (still fighting error 20 with one of my 2 units) but on my unit which is facing error 20 the backlight is on. So i guess it’s not normal that the backlight is off. It’s also constantly flashing the LEDs.

bumerang #3 March 22, 2019, 1:16am

wow, that’s weird. Backlight is working and the ATMEGA is able to control it - it is on when in test mode.

What do you mean by “flashing the LEDs” - does it look intentional (I mean like for example 1Hz 50% duty cycle) or unintentional (like in my example ~0.1Hz and ~100ms flash)?

I will try to add few 1uF caps to 5V rail to each chip. It hurts when I see the super thin +5V copper trace. And the minimal copper pour clearance… why?!

I wonder if there is a way to identify the moment when ATMEGA restarts itself. When for example watchdog is activated - is the RST pin internally puled down so it is measurable?

My TODO list:

- Add more decoupling C
- Route +5V directly to ATMEGA with a wire
- Try the other FW - HelloFriend.bin
- Change R35 and R46 to 200k exactly
- Desolder and meassure al FETs

bumerang #4 March 22, 2019, 3:20pm

- Add more decoupling C
  - done - added 1uF ceramic + 220uF close to ATMEGA, added 4.7uF ceramic to the LCD board, added 4,7uF ceramic close to DAC
- Route +5V directly to ATMEGA with a wire
  - done
- Try the other FW - HelloFriend.bin
  - done
- Change R35 and R46 to 200k exactly
  - done - one is 201.1k the other is 201.5k
- Desolder and measure al FETs
  - done - all FETs are OK

I also measured resistance of all transformer windings - everything seems to be fine.

Nothing helped.

Any ideas what to do next?

Edit:

According to this: **https://www.falstad.com/circuit/circuitjs.html?cct=$+1+0.000005+10.20027730826997+50+5+43 r+176+112+304+112+0+100000 v+448+352+448+80+0+0+40+5+0+0+0.5 r+160+320+160+160+0+200000 a+176+176+304+176+8+15+-15+1000000+4.499937501562461+4.5+100000 w+176+112+176+160+0 w+304+112+304+176+0 w+304+176+368+176+0 w+176+160+160+160+0 w+176+192+112+192+0 g+160+320+160+352+0 r+112+192+112+320+0+100000 r+112+80+112+192+0+100000 r+160+80+160+160+0+100000 w+112+320+160+320+0 R+160+80+160+48+0+0+40+5+0+0+0.5 R+112+80+112+48+0+0+40+9+0+0+0.5**

o+10+64+0+4099+5+0.0001953125+0+2+10+3 o+4+64+0+4099+5+0.00009765625+1+2+4+3 38+0+0+1+101+Resistance

Voltages at U10A pin 2 and pin 3 **both should be 4.5V** when LTC1661 is disconnected.

Edit 2:
My voltages U10 are:

Inputs:
Pin 2 = 4.46V
Pin 3 = 4.44V
Pin 6 = 4.57V
Pin 7 = 4.60V

Outputs:
Pin 1 = 6.04V
Pin 7 = 6.39V

Outputs are suppose to be 6.25V I am few % off - I would say it is OK. Resistors around OPAMPs are OK, OPAMP is working.

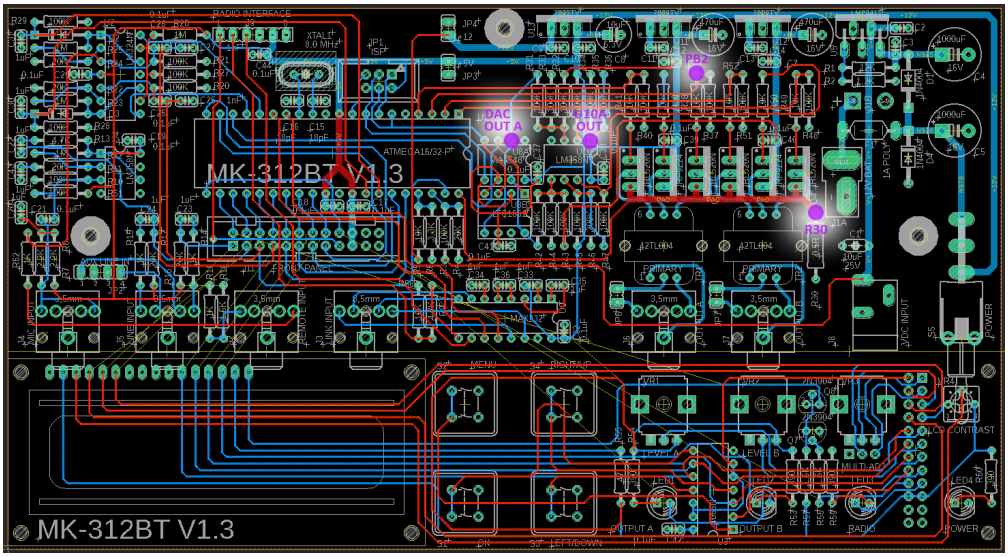
bumerang #5 March 22, 2019, 8:51pm

Looks like I'm here alone 😦

Another try - adding 1R resistor to the center tap of the transformer to more closely mimic the DC resistance of the 42TL004. No luck.
I will try to find real 42TL004 somewhere.

@Cerb : would you be so kind and measure few points on the working unit with an oscilloscope?
DAC OUT A
U10A OUT
R30
PB2
all against ground

I am interested in test sequence so please use a single shot and trigger to the PB2 (rising, ~3V).

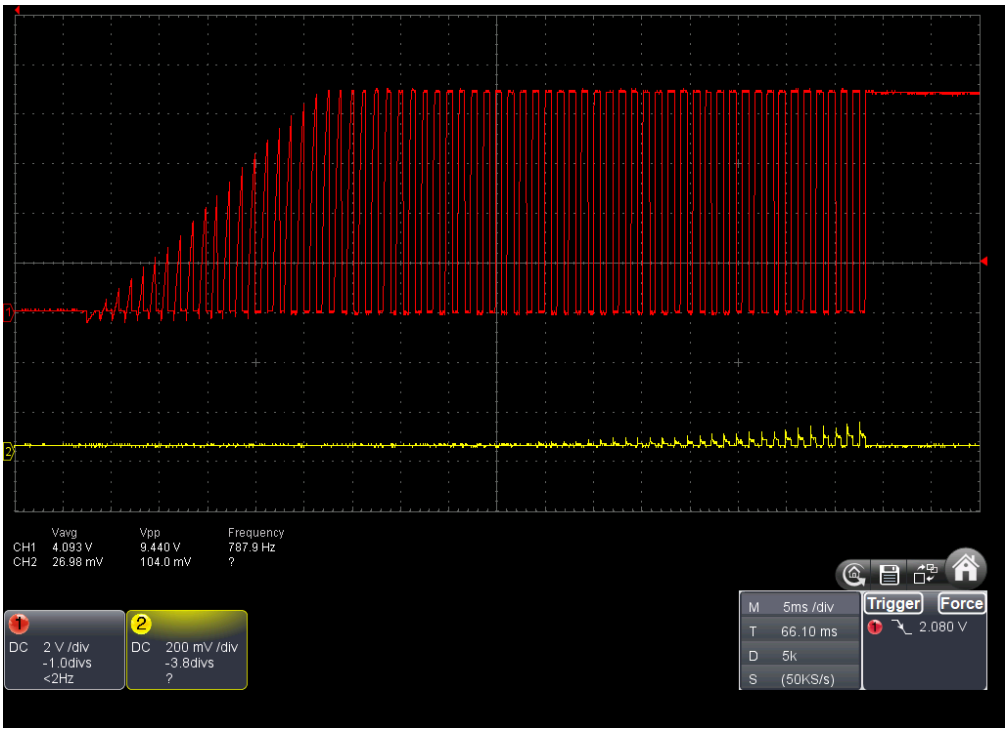


Also could you please upload the **exact binary file** you used for the working unit? Not just link to github, upload your file please.

bumerang #6 March 27, 2019, 5:28pm

How the f*ck is this possible?!

Red is Q1 drain, yellow is R30 current sensing - all against GND.



There is a voltage ramp **which means DAC + U10A + Q3 is working.**

There is a switching - the voltage is pulled down with each Q1 impulse to its gate - **this means Q1 is working.**

But there is no current flowing through the transformer - am I right? There is no significant voltage drop across R30 (check the yellow voltage scale it is 200mV/div!). I cannot explain why there

is some voltage drop at the end of the signal. I don't get it.

The only explanation is the transformer is broken somewhere inside - I measured it twice.
Or the 0.5R resistor (R30) is dead short (thus no voltage drop) which it should not be - there are two 1R in parallel - brown -. black - black - silver - brown… 1R 1%.

bumerang #8 April 2, 2019, 2:30pm

> cyberpunk:
>
> Hi bumerang,
>
> I have not built my device by now, as I am missing the firmware, but looking at your oscilloscope captures this one seems a bit weird to me: " **Red = Q1 gate, Yellow = Q2 gate (against ground):** ".
>
> This looks like Q1 and Q2 are switched simultaneously, if they are switching simultaneously you generate a current through the transformer in forward an backward direction in parallel, so the magnetic field will result to zero and therefore there will be no current on the output, but on the input.
>
> But I do not know the firmware, maybe this is just a hardware check up.

Not sure why you deleted your post but you are correct! They are suppose to alternate. I will investigate it further. Still it does not explain missing signal at R30.

Maybe (and this is big maybe based on absolutely nothing) because this is a test sequence the magnetic field canceling is desirable - you don't want to shock anyone by this test.

Anyway big thanks for your (deleted) reply. I will investigate this.

cyberpunk #9 April 4, 2019, 11:41am

I deleted the post because, just as you said it seems plausible that in a test sequence the cancelling of the magnetic field is desirable and aditionally it could not explain the low voltage drop at R30.

You could also check the current limit of Q3 (IRF9Z24). To my understanding this MOSFET is applied as current limiter (constant current source). If you can't get enough current through R30 it may be too much limited, the voltage ramp (red) shows the open-circuit voltage, to my understanding there should always be around 9V at Q1, when not switched on. It seems that you may are at the very beginning of the characteristic curve when you not even have enough current to get the 9V on Q1.

This is also exactly what you see at the very end of the test sequence, where the red curve does stay at the 9V. That the current does increase slightly at the end could have some cause in any capacitance or inductance of the circuit.

Hope this helps more.

MK312BT Firmware and function issues (Failure 20), debug suggestions?

bumerang #10 May 8, 2019, 4:01pm

FUCK YEAH! It is working!

There is a difference between IRF9Z24PBF (VISHAY) and IRF9Z24**N**PBF (INFINEON).
You can not use IRF9Z24PBF!

IRF9Z24PBF (VISHAY) has Vt=3.58V.
IRF9Z24NPBF (INFINEON) has Vt=2.98V.

Apparently because this transistor is used in the linear segment this makes enough difference for the unit to go into the Failure 20 mode.

From IRF9Z24PBF (VISHAY) datasheet:

Gate-Source Threshold Voltage	$V_{GS(th)}$	$V_{DS} = V_{GS}$, $I_D = -250\ \mu A$	- 2.0	-	- 4.0	V

From IRF9Z24NPBF (INFINEON) datasheet:

$V_{GS(th)}$	Gate Threshold Voltage	-2.0	——	-4.0	V	$V_{DS} = V_{GS}$, $I_D = -250\mu A$

They should be the same but they are not.

If there will be a demand - I will measure all important signals with a scope on a working unit and post it there. I think it will make future debugging much easier.

2 Likes

Gary #11 May 12, 2019, 11:32pm

Good finding. I want to learn more about the signals.🙏
I just finished building my MK312 and got the Failure 20 😮

bumerang #12 May 13, 2019, 12:54am

Hi Gary,
tell me which signals you need.

But first I recommend to go through this checklist:

1. double check all resistor values around the DAC and the OPAMP feeding both IRF9Z24NPBF
2. meassure Vt of your IRF9Z24NPBF (a simple component tester will do)
3. check if you have the correct transformer
4. check the transformer orientation

5. check the value of the 0R5 current sensing resistor

Also the PCB design is problematic, I am not trying to be rude, but it seems the PCB was designed by a programmer - not an electronic engineer. Grounding seems to be the main issue - and I know grounding with 2-layer PCB is always a problem.

I was facing frequent restarts of the processor - it was awful. It restarted itself after just few seconds of working. Sometimes after a button pressed. Garbage…
More decoupling caps did not helped. 5V rail is rock solid.

There are these issues I had with the board:

1. grounding - I had to route several external ground wires to critical parts of the circuit
2. ATMega reset pin should have RC circuit connected or at least external pullup (this is not the root cause of restarts)
3. **there are no pull-ups for the buttons!** There are no pull-ups in the 4066 - how does it even work? Some parasitic current? God? Add pull-up and 100nF cap for each button. It seems the self-restarts were caused by this 4066 and floating buttons. Buttons without pull-ups are sensitive to interference -> 4066 creates some bullshit signals -> ATMega freezes (or maybe processing of this bullshit signals takes too long) -> watchdog -> **restart**
4. there is a resistor in the audio input section which is not even connected on one side?!

Gary #13 May 13, 2019, 1:24pm

Hello Bumerang,

Thanks for your suggestion.

I have check the resistor values around the DAC and the OPAMP and nothing odd at all.

I had order IRF9Z24NPBF and IRL520NPBF at element14 but it turn out IRL520NPBF out of stock and I have source it again from china seller.
When I purchase IRL520NPBF, I also purchase the IRF9Z24NPBF as well but I suspect I got inferior counterfeit. I checked them both with component tester. IRF9Z24NPBF VT=2.0V C=685pf |AND| IRL520NPBF VT= 1.8V C=1.83nF

I using the 42TU200 transformers as same as you. I don’t want to have too many wires flying around and I connect 3 wire only (Have not connect the Transformer P side because I think the output side is direct connect to 3.5 output jack only and AVR will know nothing regarding the final output.

The 1/2 watt 0R5 is good. I check it value with LCR meter.

I google the IRF9Z24NPBF and IRL520NPBF image and try to located what the genuine chips exactly look like. I found tons of different image, beyond 10+ different outlook. That’s horrible, I have no idea which one is genuine now.

Please do me a favor and share the image of working IRF9Z24NPBF and IRL520NPBF. I want to know how the working one look like. Please also check them with component tester and let me know

all the measurement value you got from the tester.

Thanks!

bumerang #14 May 13, 2019, 2:39pm

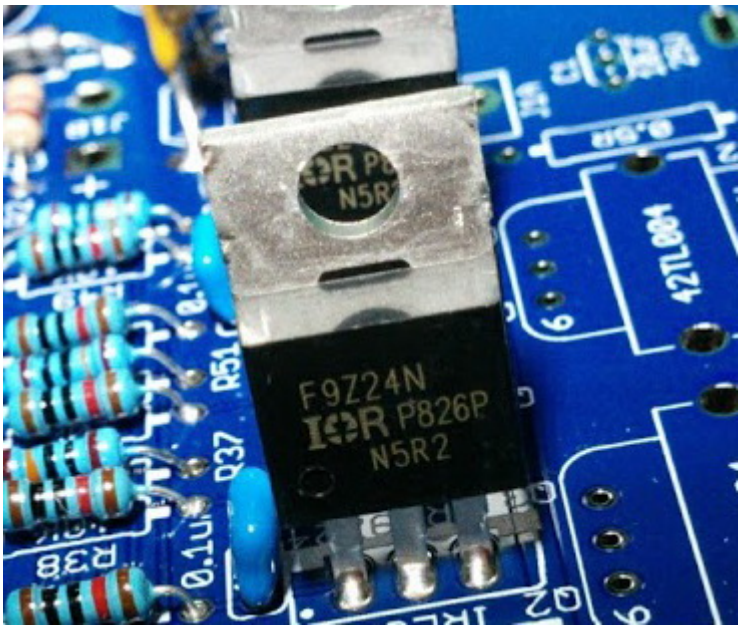
IRF9Z24NPBF (Infineon) purchased from TME has Vt=2.98V. I think this might be the problem.
There should not be a problem with the IRL520NPBF because they operate in fully saturated mode.
Mine has also Vt around 1.8V.

Note - after all the trouble I decided to buy ridiculously expensive 42TL004 transformers from Ebay and I build completely new unit.

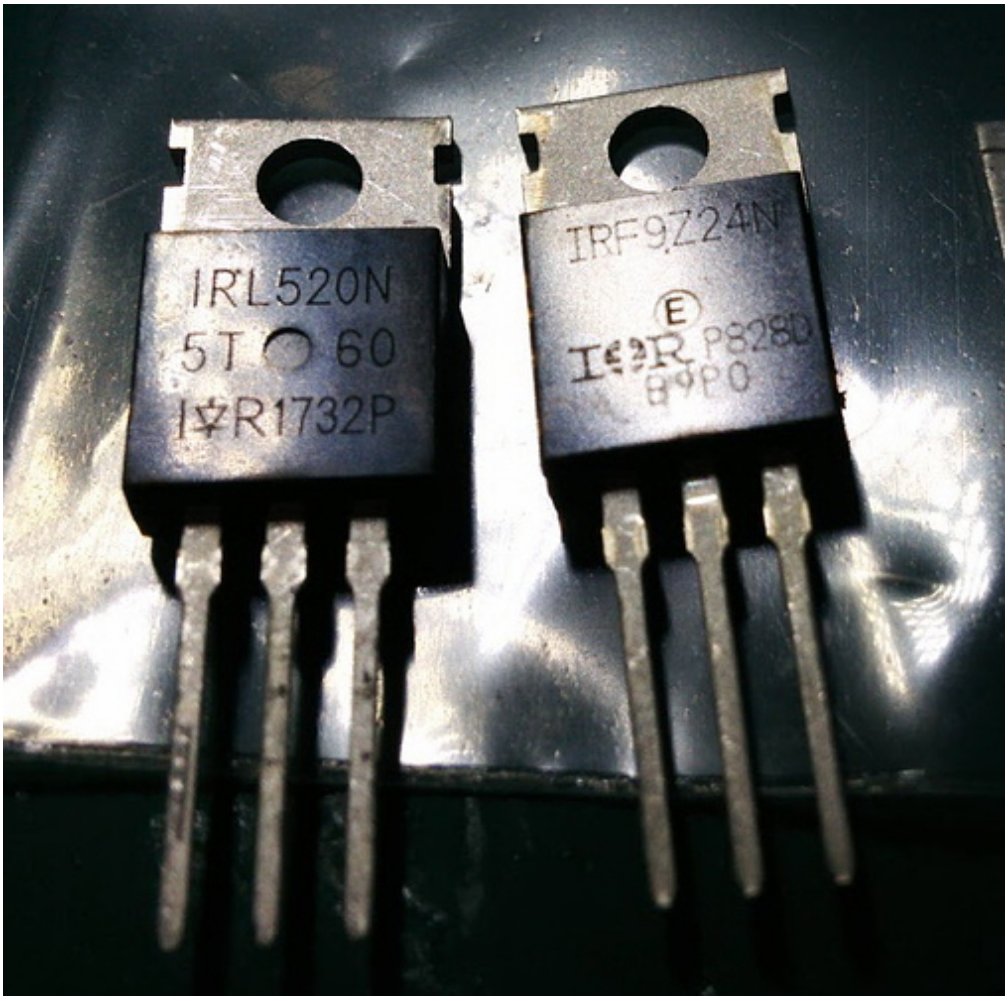
Gary #15 May 13, 2019, 3:09pm

Thanks for the picture.

I got this IRF9Z24N from element14. It have a small circle at the bottom left to indicate it’s the first PIN, which match with the PCB. Everything looks legit.



Below two are from China seller.



IRL520N = Apparently they don’t have the correct font for IR Logo 🤣
IRF9Z24N = Look legit but the small circle is in the middle. The VT did not match with Yours. 🤔

The IRF9Z24N you got have a small circle in bottom center instead of bottom left. I am really confuse about the small circle location. It should be represent PIN1, right ?
If you google about the image of IRF9Z24N, You will find tons of different small circle location 😂

bumerang #16 May 13, 2019, 3:33pm

I always thought the dimple has something to do with the plastic injection process. It is pretty hard/stupid to put TO-220 backwards. I dont think it is the “pin 1” indicator as with IC packages.
Try IRF9Z24N from a different vendor.
As I said there should not be a problem with IRL520N.

Maybe stupid idea - are those Chinese fakes even pin compatible?

Gary #17 May 13, 2019, 5:24pm

I just found this 😥

Counterfeit IR mosfet have a smaller size in die

Counterfeit IR mosfet performance issue 😰

==========

Anyway, I have order the new mosfet from mouser directly and they are on the way now. I will report back either the new mosfet solve the problem.😎

Gary #18 May 17, 2019, 7:31am

Replaced all of the counterfeit mosfet with genuine one and problem solved. No longer having failure 20.

1 Like

bumerang #19 May 23, 2019, 8:56am

Gary, do you experience any random restarts?
Do you experience problems with buttons (for example multiple presses are read by the MCU) - did you use pull-up resistors for the buttons?

Gary #20 May 24, 2019, 12:13am

I still waiting for the 42TL004. The flying wire to 42TU200 is really annoying and therefore I haven't test it completely.

I had forward the problem you mention to engineer and here is the answer.

Buttons are multiplexed with the LCD pins, the program switches these pins to the internal pull-ups when polling the input which is 40K or so. More than likely the long ribbon cable is causing these issues which are now solved with the repositioning of J10 so that it is approximately 2.5 inches long now.

You should use a shorter cable. The shorter the better.

bumerang #21 May 24, 2019, 11:21pm

Buttons are multiplexed through 4066 which is “CD4066B CMOS Quad Bilateral Switch”.

Yes, there is an internal pull-up in the AVR but how the pullup can act through the 4066? You pulling HIGH the processor side of the 4066 -> so the button side of the 4066 is also HIGH.
Then you press the button -> it pulls the button side of the 4066 LOW but at the same time the processor side of the 4066 is still pulled HIGH (with the internal pull-up)… you input the signal from both sides.

I am really confused how the 4066 acts when you feed some signal from both sides at once.

pimli #22 May 27, 2019, 10:47am

I currently have the same issues - usually it will just not boot and I get squares in the top row of the LCD with the backlight on. But sometimes I get error 20 and no backlight.
I do have the IRF9Z24NPBF and IRL520NPBF but they’re from china and look fake. I’ve ordered some real ones through mouser now but I’m just wondering if this sorted both issues or is it only the error 20, with something else causing the boot issue?

bumerang #23 May 27, 2019, 6:56pm

squares in the top row of the LCD means the LCD is not initialized properly by the MCU -> maybe the MCU is not working (try to measure the crystal oscillations) AND/OR there might be some interference issue with the ribbon cable (try to make it as short as possible) AND/OR there might be a problem with the LCD power rails (either ground or +5V… adding more ceramic capacitors certainly wont hurt).

pimli #24 May 28, 2019, 10:16am

Thanks for the quick response. I hadn’t shortened the ribbon cable yet so I cut it down as short as possible but it made no difference unfortunately
Also, I forgot to mention that when it goes to error 20, the A / B LEDs flash but they don’t when it displays the squares. That leads me to believe the problem is not with the display but I don’t have anything to check the crystal with

bumerang #25 May 28, 2019, 7:23pm

IIRC when I got “Failure 20” the LEDs also flashed once (both at the same time).

I am 90% certain there is something wrong with the 4066 (how it is connected) in combination with the program (how buttons are read - I think it can block the MCU).

Try this:

1. Add pull-up (10k-100k) to each button.
2. Add some capacitors close to the 4066 (ceramic + electrolytic combo)
3. Add some additional short grounding wire between ground planes of both PCBs.

voncosel #26 May 28, 2019, 10:47pm

> bumerang:
>
> grounding - I had to route several external ground wires to critical parts of the circuit

Which circuits did you find particularly problematic?

bumerang #27 May 28, 2019, 11:22pm

voncosel - I added several wires. The problems with booting and random restarts stopped (almost) after I added wire directly to the LCD GND pin (the other end is soldered somewhere close to the MCU).
Also worth mentioning are the pull-ups.
I needed to do both to solve this.

Because there are GND floods on both sides of the PCB it might create closed loops. This is not good when considerable pulse current is used for the transformers.
The only issue I can think of might be Low-ESR capacitors (some Nippons I had in stock) I used for the 9V rails - standard capacitors might do a better job limiting the current spikes.

voncosel #28 May 29, 2019, 11:44am

The device measures the current pulses upon boot and calibrates for it, doesn’t it?

1 Like

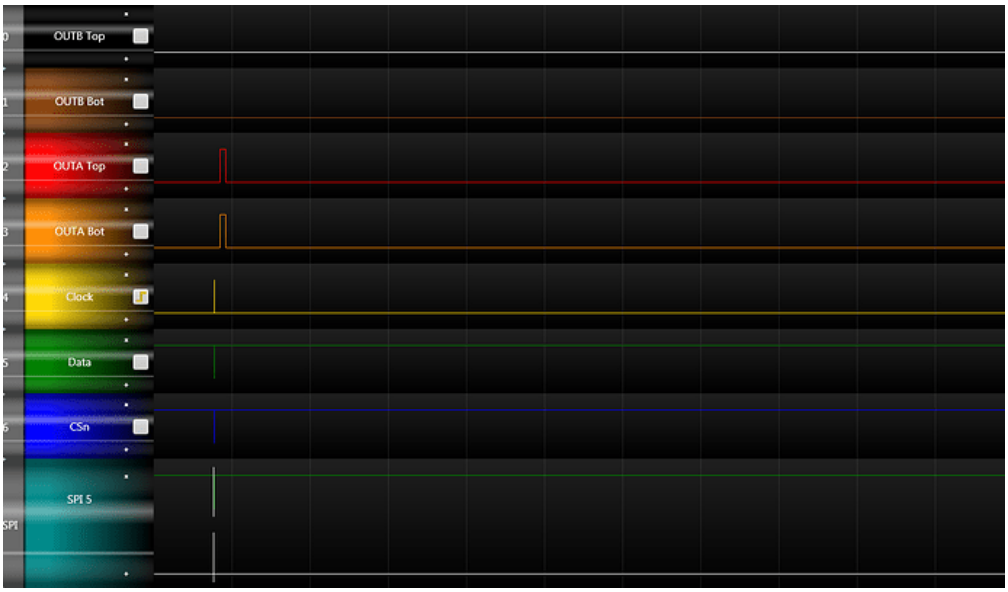
bumerang #29 May 29, 2019, 3:12pm

It calibrates for some average current. But I guess the low ESR cap will make the rising edge faster -> higher amount of harmonics signals -> more interference.

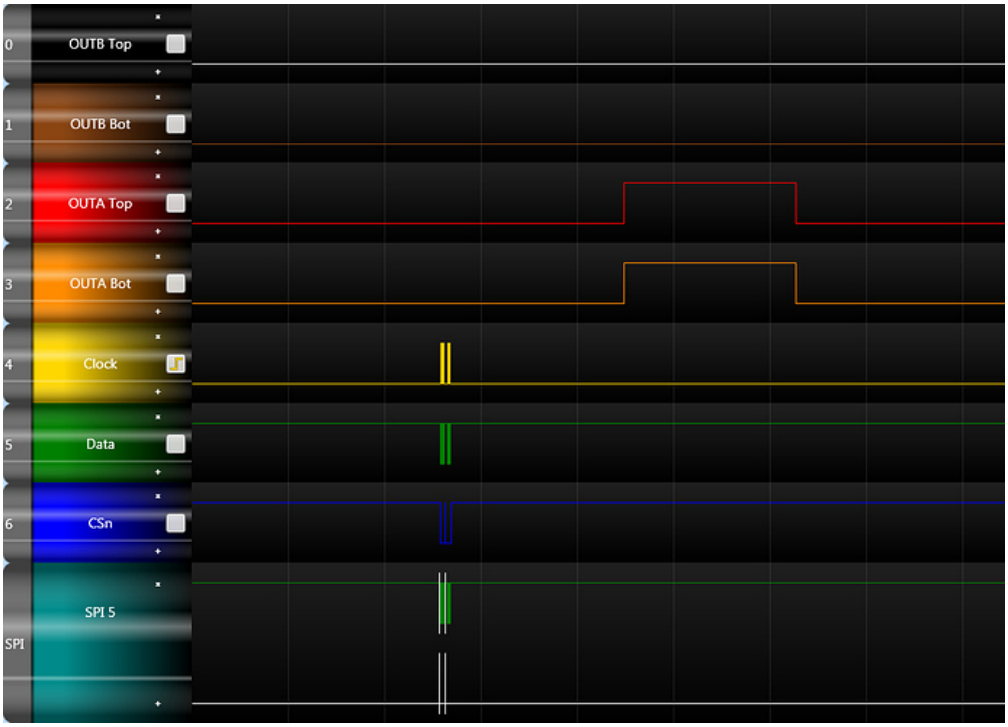
I planned to make my own PCB with majority of components being surface mount -> that way you can get GND at almost whole bottom layer. But to be honest I am disappointed by the preset programs.
IMO the ET-312 is a hype fueled by the **kink.com** production.

sparkyfart #30 November 14, 2019, 8:49pm

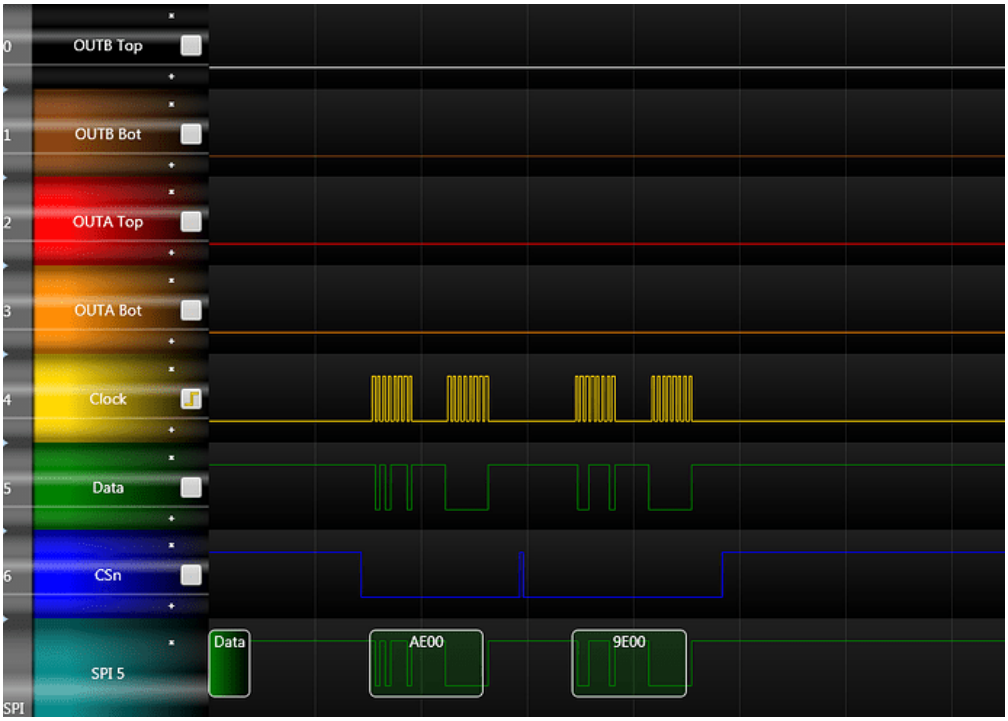
I Have the same issue, Failure 20
PCB built correctly, IRF9Z24NPBF and IRL520s from Farnell / Element14
During power up there is a Ramp on R30 (0.50 Ohm) when channel A is ramping up but there is no corresponding Ramp on R30 when Channel Bs DAC ramps up. CH A Ramps first then Channel B but no voltage is seen on R30 is this normal?
Only Channel A is examined by the ADC



Is this the case for all Failure 20s (The fault is on CHA so it just fails?)

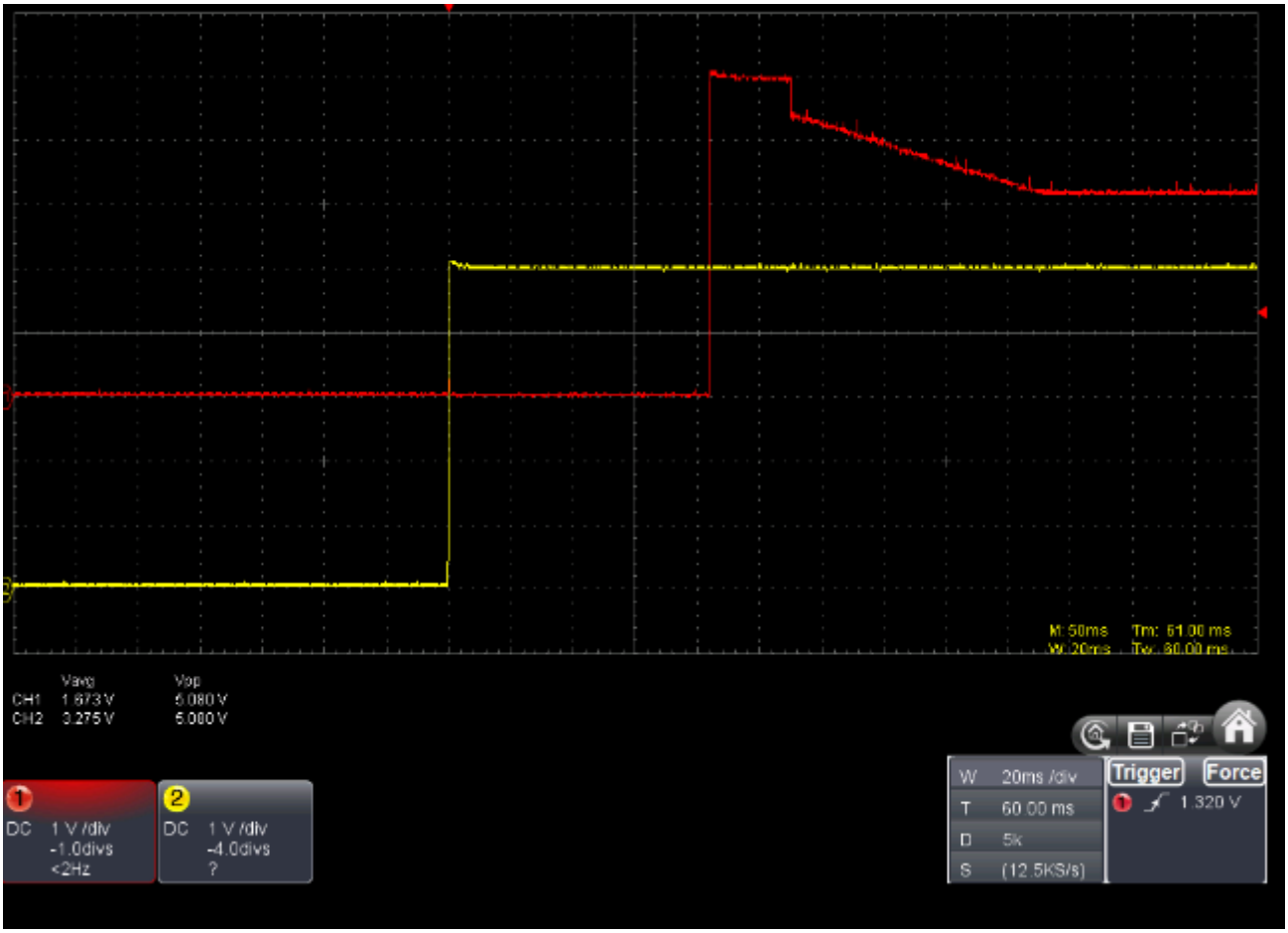


Zoomed in on DAC Data

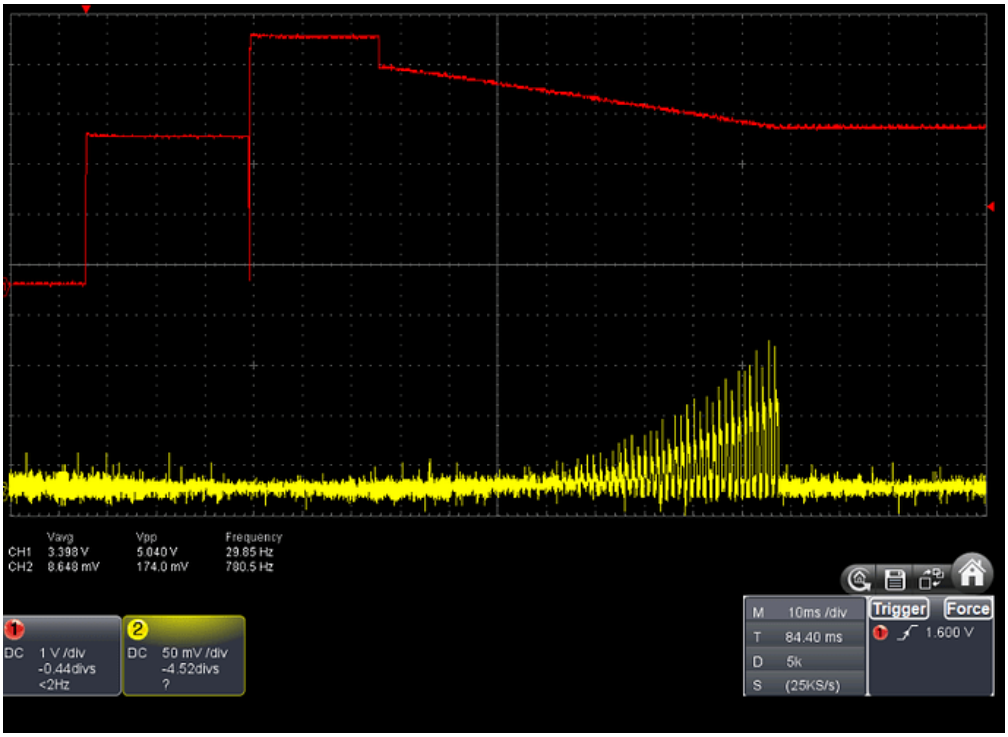


It looks like PB2 and PB3 assert (OUTA Top and OutA Bot in the logic analyzer) but PB0 and PB1 never assert (OUTB Top and OUTB Bot in the logic analyzer)

Can get scope shots of the analog side of things if needs be but then appear to match those above i.e.
This



and
This



Any Ideas Welcome.

Regards
This is the same Issue I posted here

Newest Member to the Failure 20 Club

> Hi All Been through all the board cannot see why this is occuring. Using IRF9Z24N from Farnell Transformer is in correctly (Pri to Output Jacks, or “Backwards”) R35 & R36 are 200k using LTC1661 DAC Can provide scope shots. How can I measure Vth of these IRF9Z24N devices (My Analog theory is not what it used to be) Regards SF

Newest Member to the Failure 20 Club

bumerang #31 November 14, 2019, 10:20pm

> sparkyfart:
>
> channel A is ramping up but there is no corresponding Ramp on R30 when Channel Bs DAC ramps up. CH A Ramps first then Channel B but no voltage is seen on R30 is this normal?

Hi, IIRC it will not test ChB when there is something wrong with ChA.

I would not suspect there is anything wrong with DAC. The problem probably is the IRF9Z24N - maybe different batch, maybe different manufacturer. There seems to be only tiny window to pass the test.

IRF9Z24N is a power switching mosfet and in this design it operates in the linear region - bad design I would say - that is why they need to calibrate it each time - it is not a test it is a calibration procedure.

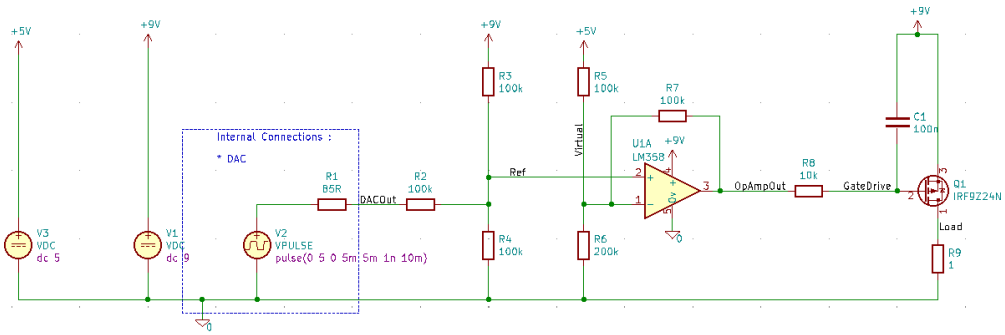
The easiest next step is to use IRF9Z24N from different vendor.

1 Like

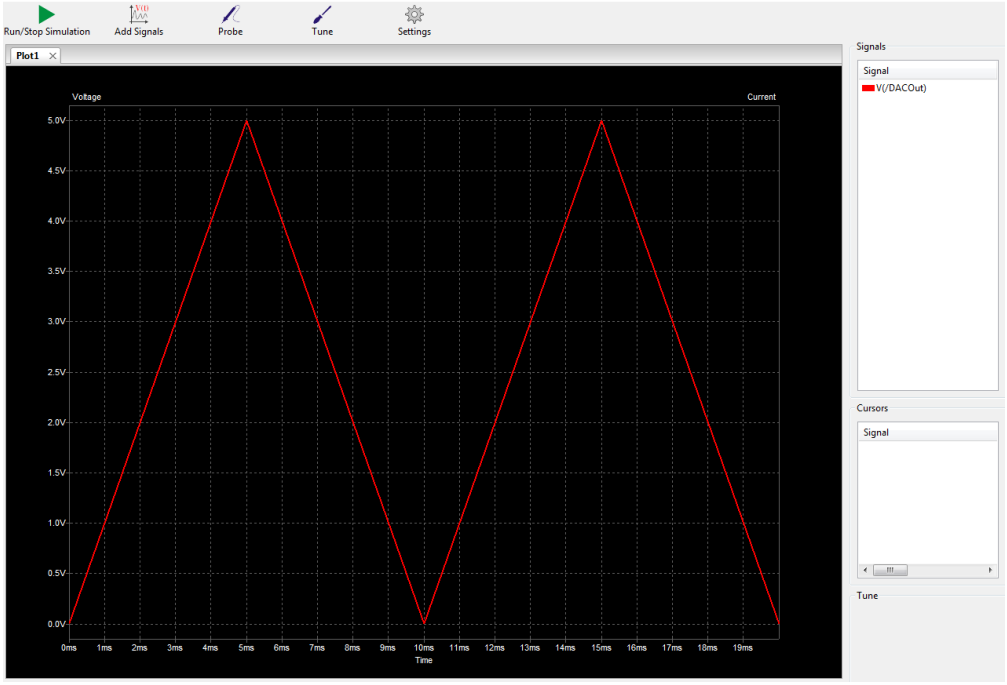
sparkyfart #32 November 15, 2019, 5:28pm

Thanks Bumerang

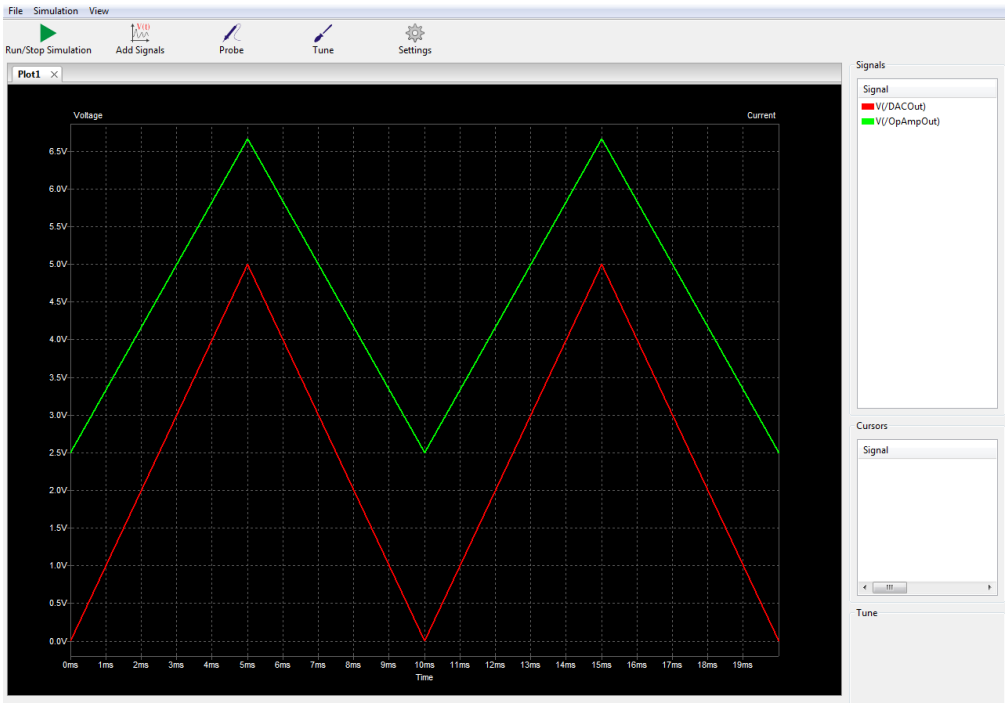
My MosFET theory very rusty, so I’ve taken to the simulator to help me out



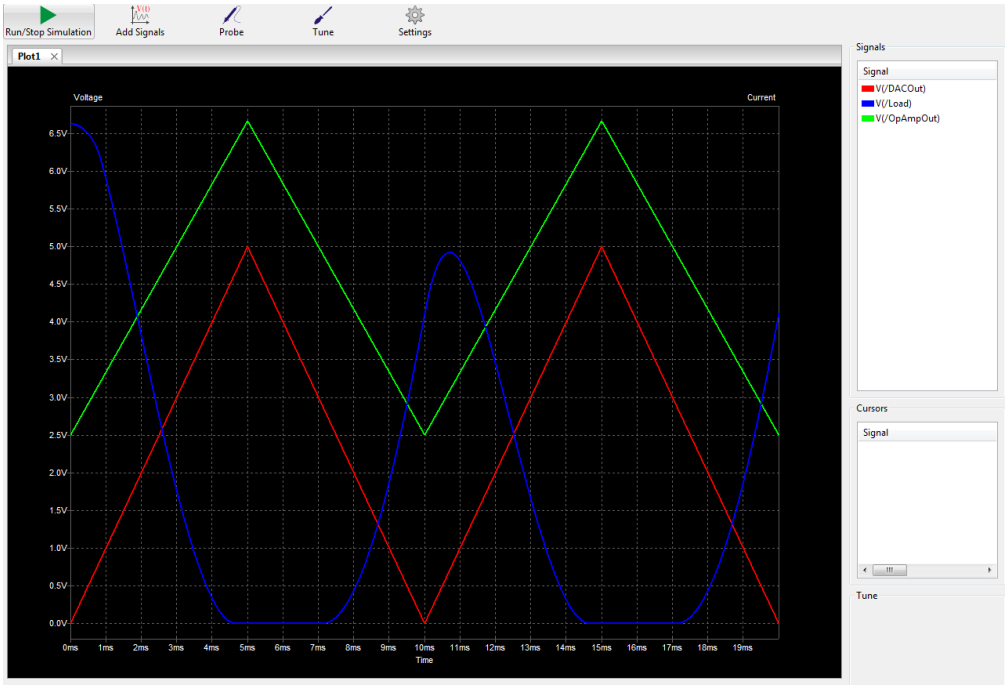
Testing the circuit with a ramp to simulate the DAC



The Opamp provides a Gain and Offset from the DAC to the FET thingy

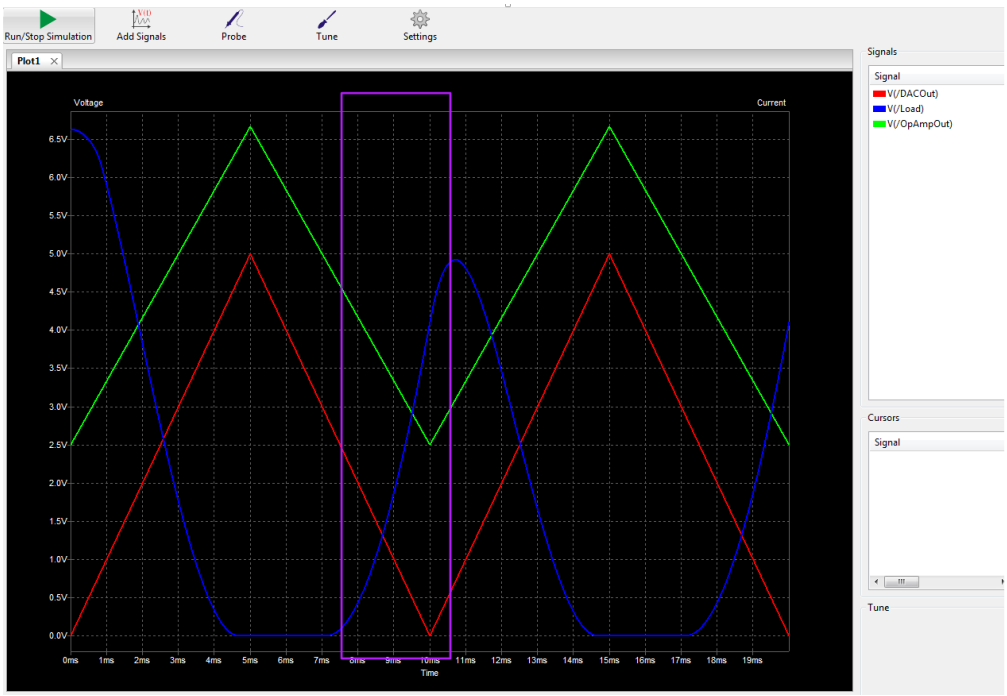


Using a 1Ohm load the MosFET



So the Blue trace is the output of the FET

When you talk about the Linear region, are you referring the the area in Purple?

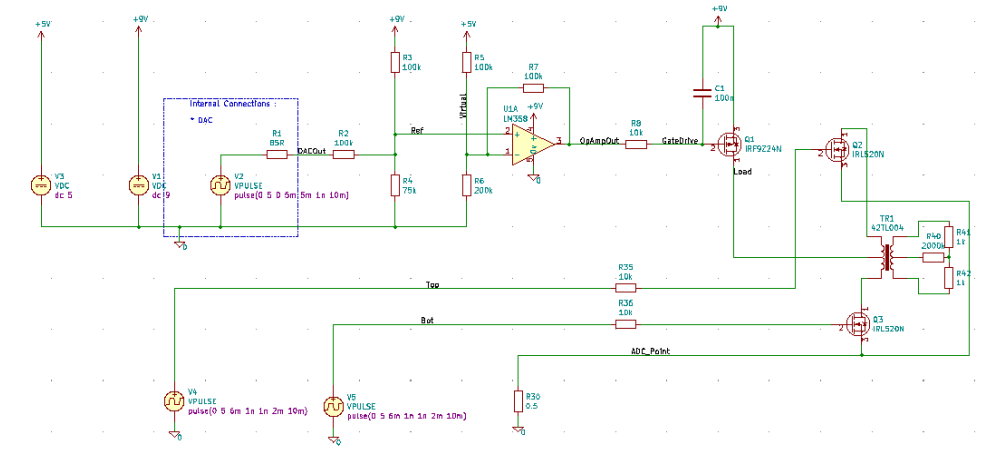


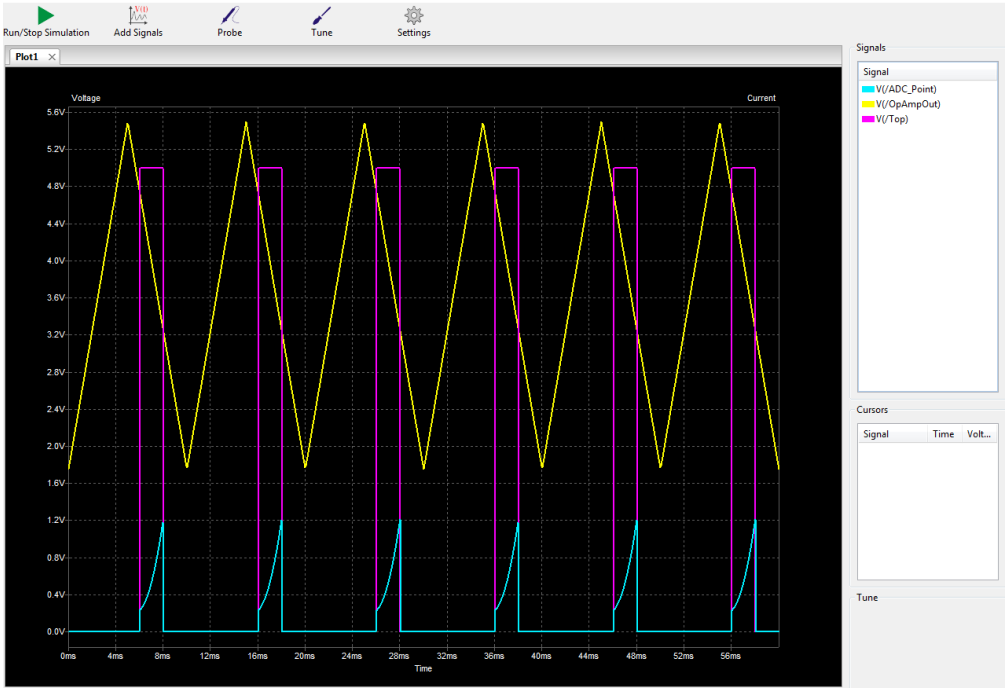
Its Quite small!

Can you prove a scope trace of the signal presented to the ADC on power up?
Its may help debug the issue.

sparkyfart #33 November 15, 2019, 5:31pm

Looking at the full system,
Behavior here is a little harder to model





bumerang #34 November 16, 2019, 2:10pm

Hi,
by linear region I mean the situation when the transistor is not fully saturated. There is no real analog feedback in the system so they modulate the output signal with these three FETs this way:
The P-channel IRF9Z24N works in the linear/ohmic region - it modulates the envelope of the signal.
Two others N-channels works in fully saturated mode and create the alternate switching.

Because of that you need some kind of lookup table (DAC output) vs (coil current) - this is the power-up calibration routine. Because each FET will be slightly different and possibly ambient temperature is also a factor.

About the simulation - I believe the IRF9Z24N has a bad model in your software because it should open much sooner.

> sparkyfart:
>
> Can you prove a scope trace of the signal presented to the ADC on power up?

You want ADC (input) or DAC (output) signal?
Because ADC is only used to measure battery voltages and to measure current sensing R.

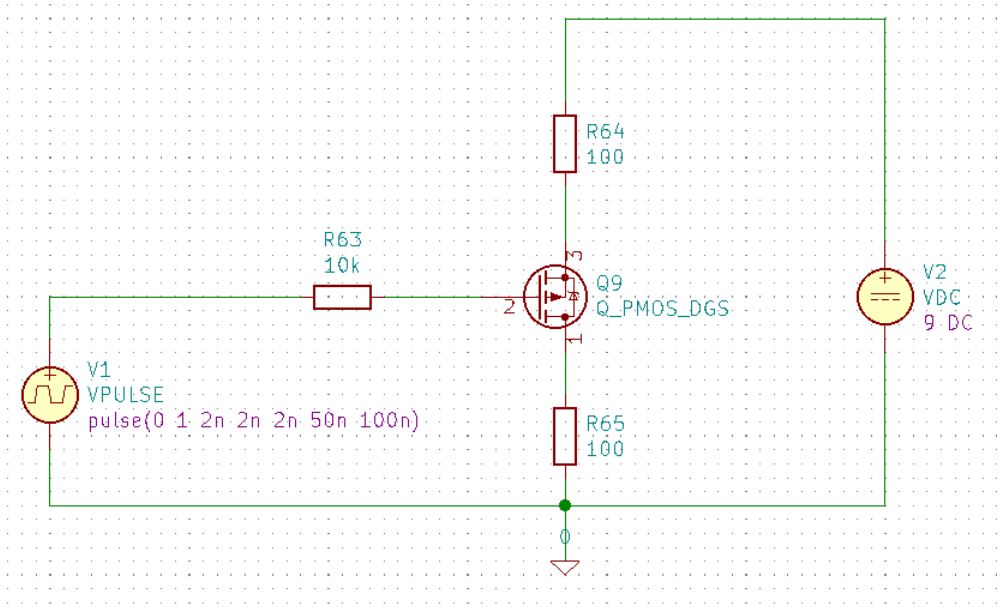
sparkyfart #35 November 16, 2019, 10:15pm

Hi Bumerang

Thanks for the explanation,

Yes can you provide a scope capture of the ADC input during calibration?
I have bench tested all the IRF9Z24Ns and they have a high Vth
I'm hoping that I can see what a passing ADC pulse looks like and
tweak the Gain, Offset or compensation to make my devices pass.

Also How do you measure Vth? (I've seen your comments above about the Vt)
My bench setup is like this



I just measure the input voltage when the devices starts conducting my Vth is about 3.15v
is there a way I can double check this?
I have a very narrow band when the IRF9Z24N works in its linear region.

Best Regards

bumerang #36 November 17, 2019, 12:54pm

> sparkyfart:
>
> very narrow band when the IRF9Z24N works in its linear region

That is the whole problem with this design 🙂

I will measure the ADC input, give me a few days.

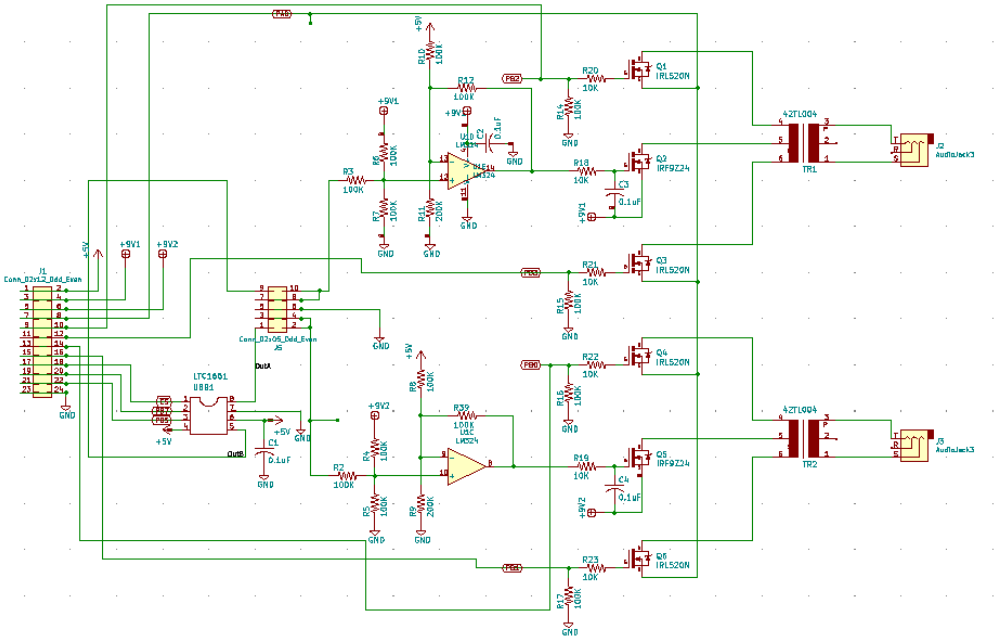
I measured Vth with Chinese component tester. It is probably not a true value but good enough when you compare two transistors.

sparkyfart #37 November 17, 2019, 2:21pm

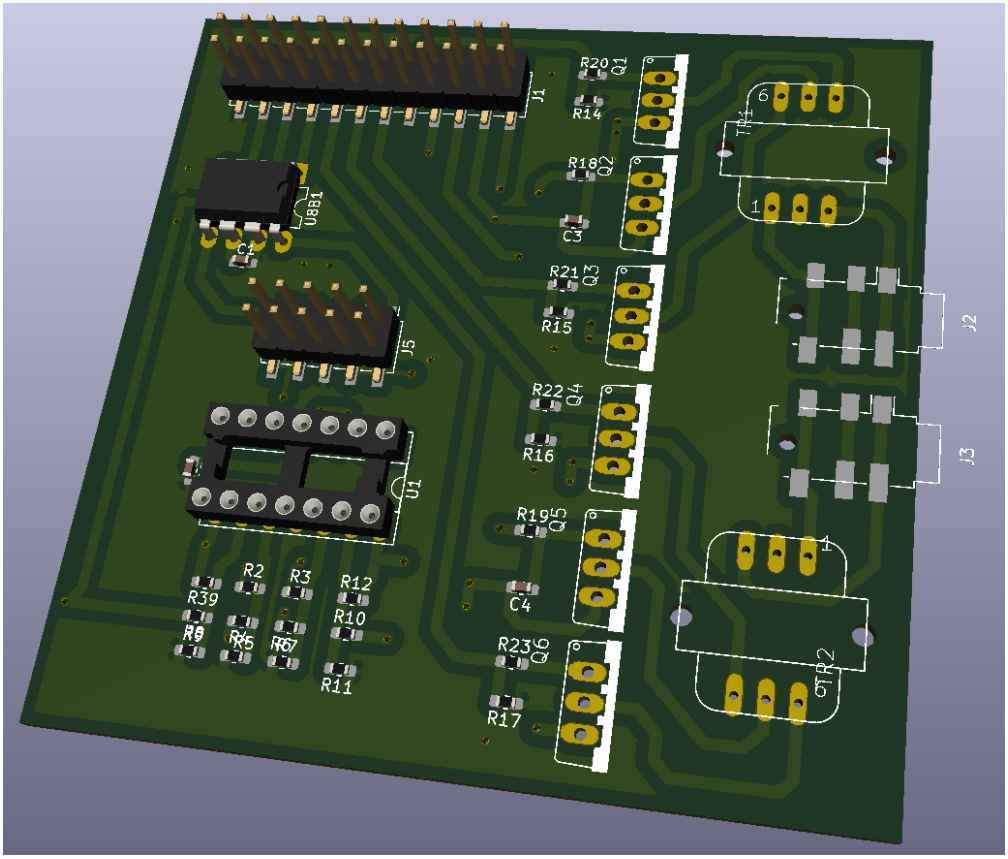
Many Thanks, Very much appreciated!

sparkyfart #38 November 21, 2019, 4:48pm

Current Plan,



is to isolate this part of the circuit and test component changes etc…
its much easier to work with a custom PCB



Will update with results if they make sense

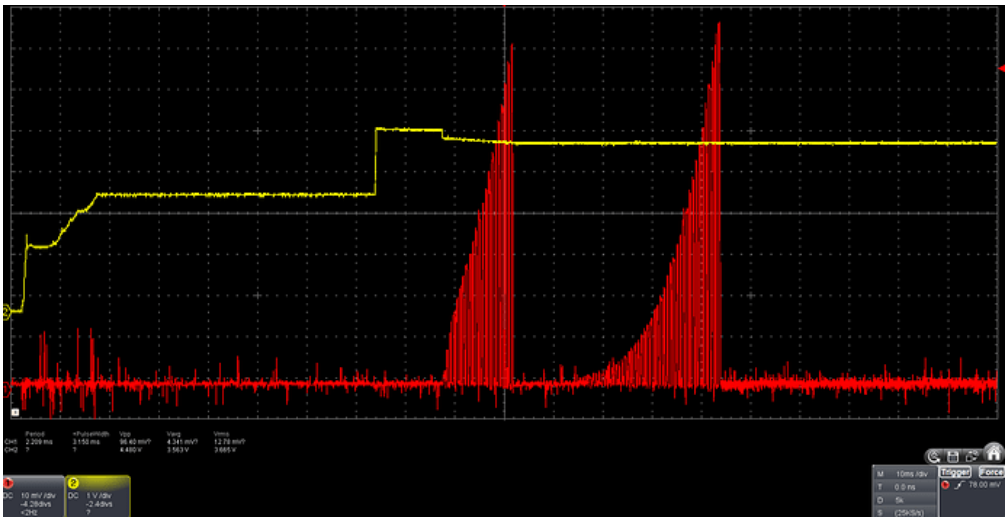
1 Like

bumerang #39 December 4, 2019, 4:07pm

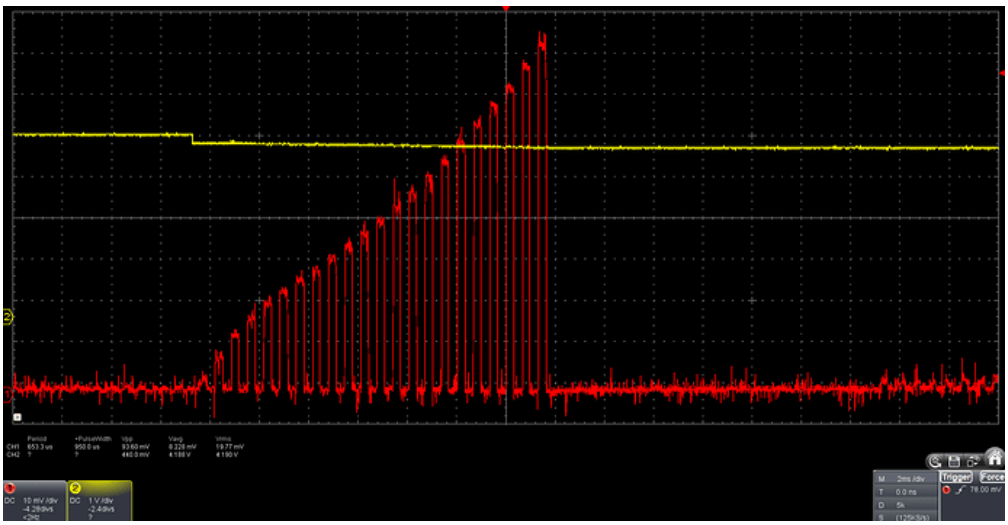
> sparkyfart:
>
> Many Thanks, Very much appreciated!

To many obligations, too little time…

There are the measurements:
R-R30 (current sensing, ADC input), Y-out_A (DAC output)



R30 closeup



And yes the R30 signal is this weak - I checked several times 1x/10x probes and setting. The testing is done at low currents.

sparkyfart #40 January 2, 2020, 7:33pm

Hi Bumerang

First off Happy New Year

Second Thank you for the plots. Much appreciated Its kind of cool to see the different calibration profiles on both channels!

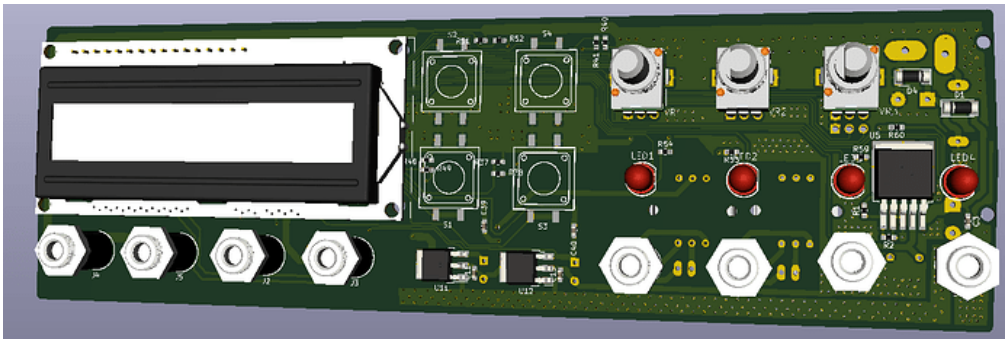
My attention has been elsewhere too.

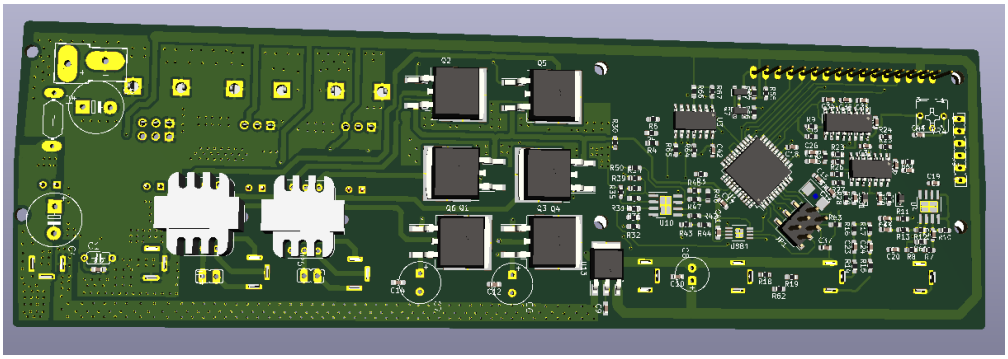
In the gaps I have been working on this, but time is scarce, I'm also suffering THPicitis Its absolute hell working with THP parts. SMD version is really needed!

SF

sparkyfart #41 January 10, 2020, 7:06pm

thinking something like this might be easier to work with





Then I can get back to debugging!

1 Like

bumerang #42 January 11, 2020, 7:18pm

That is amazing!
2 or 4 layer PCB?

sparkyfart #43 January 11, 2020, 10:47pm

Its a 2 layer PCB,
Some of the parts have been ordered from china so the plan is to wait until they arrive before committing to manufacture & assembly. Once built and tested I can post the design if its of use to anyone.

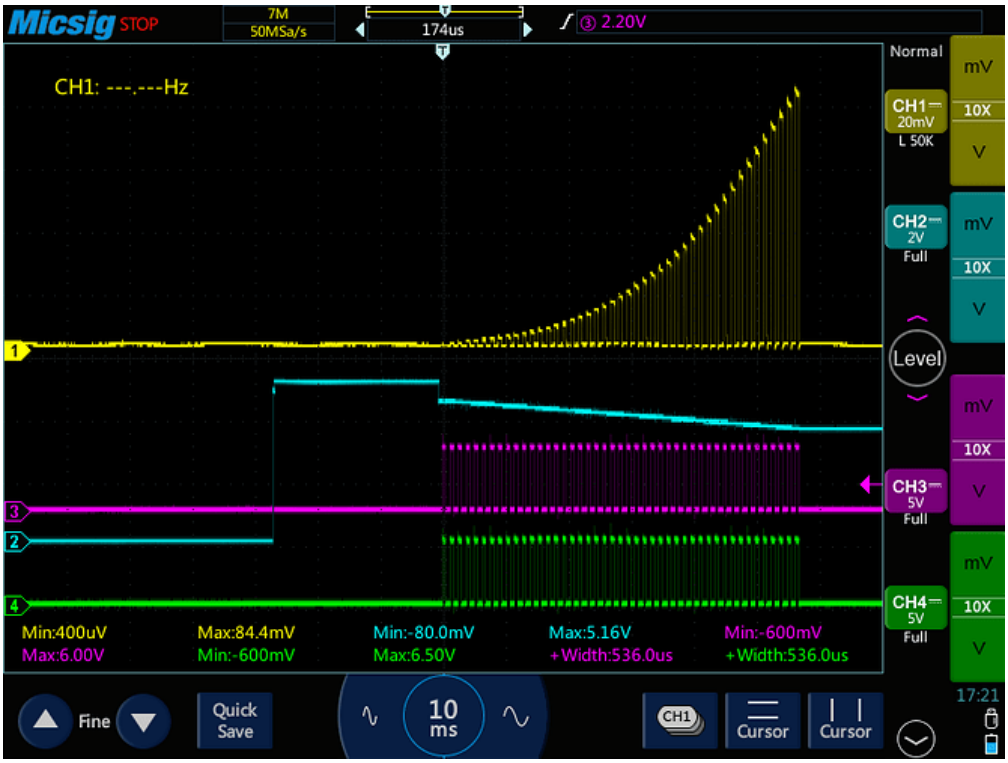
bumerang #44 January 14, 2020, 12:20am

The only thing I am afraid of is the ground plane. It seems it is significantly better than the original design but the three potentiometers share the same ground plane as the switching FETs - it might be a source of problems. You might need to ad some capacitors to the ADC inputs close to the MCU to fight the noise. The separate ground trace just for the pots would be a better approach.
OR it may works just fine 🙂

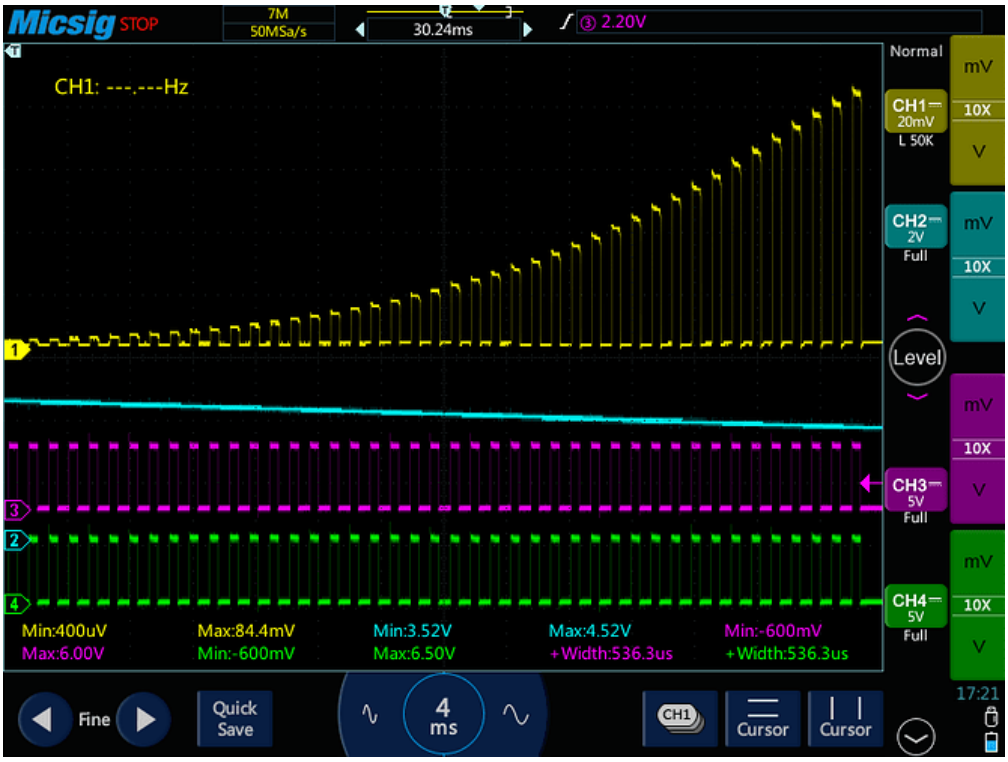
Lilly Wave vs. single ended

mer #45 February 6, 2020, 3:09pm

Hi
Maybe lets it lay here)
Init sequence for mk312bt

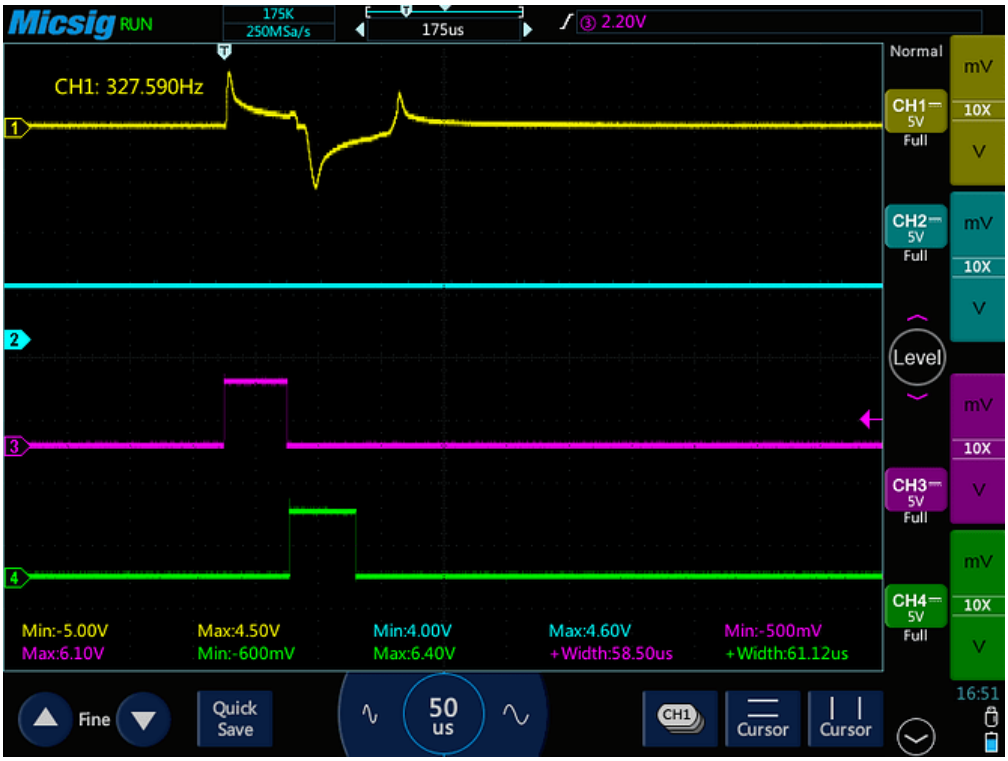


Yellow - R30 (current to transformer). 83mV max
Cyan - DAC channel A. 4.52V … 3.6V
Red - Q1 (PB2) 5V control
Blue - Q2 (PB3) 5V control



mer #47 February 6, 2020, 3:09pm

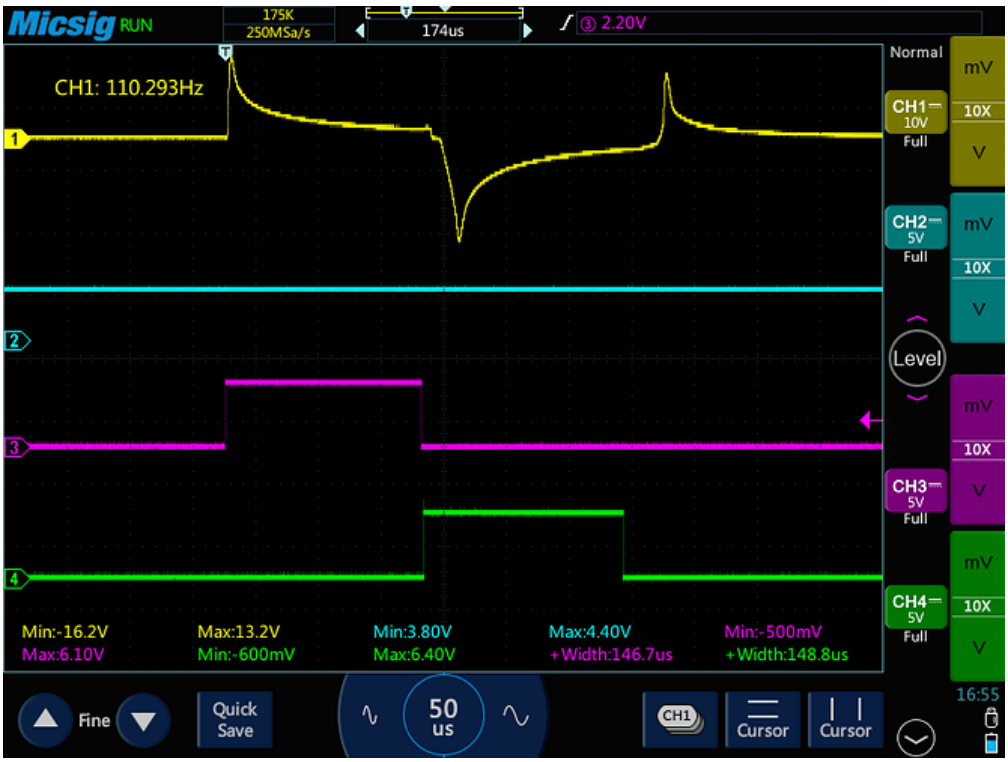
Wave mode.



Yellow - output A terminated by 100K resistor. 0%=6V, 20%=42V, 40%=160V, 50%-100%=195V
Cyan - Q3 control (power channel knob just moves this voltage. all the rest parameters is the same)
Red - Q1 (PB2) 5V control from cpu
Blue - Q2 (PB3) 5V control from cpu

Q1 and Q2 width like 60us…150us. pause between like 1.7us

Wave mode slowly changes impulse width. like the maximum here:



Sirius #48 February 8, 2020, 10:23am

It looks really good. It matches whith that, what I can see in the source code. The 83mV on the current measuring resistor are reached after 42 steps at 3.6V on the DAC. If they are not reached even after 64 steps, failure 20 occurs.

1 Like

LynxTail #49 August 11, 2020, 12:30pm

Hi!

THANK YOU very much for great job, You did! I had Failure 20 too.

All component I bougt from my local store (ru, Chip’n’Dip), even more compact cases, exclude transformers (eBay) and LTC DAC chip (Mouser). And PCBs made by PCB-Way.

So, i did two boxes for my use (cause parts was rationally buy like that, wholesale - many cheaper and PCB-Way make minimum 5; and i wish see how working couple of two). Unluck… Each from them had Failure20. I dom’t have digital oscilloscope and many experience for use it, so it was big problem. I placed all project to wardrobe and forget. I even thought, that this 400$ (one year ago exchange rate for me) was spendet in trash bin. But now… they two working! Damned IRF9Z24 _/N… Now I happy! :3

Just want say you THANK again for your great research of this problem solving, wish You health and good luck!

Treyzor #50 December 7, 2020, 10:29am

I had a failure 20. I followed the recommendations above but nothing worked.

Finally I used page 5 from the schematic pdf with my multimeter in diode test mode.

Failure 20 is a failure in the calibration code so I traced every component to every other one in the output section.

I found that R36 wasn't connected to U10 pin 2. The through hole looked fine, the pad wasn't lifted but the trace didn't have continuity. I used a resistor leg as a bodge and it worked fine.

So if you're having trouble trace everything from the mcu to ltc1661 to the dac to the mosfets and the transformers.