QRP SWL HOMEBUILDER

RF — Test and Measurement Homepage

FAQ | Site Update Log | About... | Design Apps

Topics 1998 - 2005

- SWL Page
- RF Preamps
- QRPHB Software
- Popcorn DC Receiver Mainframe
- 40 M Popcorn Superhet Receiver
- Broadband Transformer Topics
- Diplexer Topics
- Ugly Construction
- Wee Willy 75 M DSB Transceiver
- W7ZOI JFET Biasing Tutorial
- EMRFD Review
- Tapped Cap Impedance Transformation
- More Active Antenna Experiments
- Fun with LEDs
- SWL Receiving Antenna Experiments

Topics 2006 - 2009

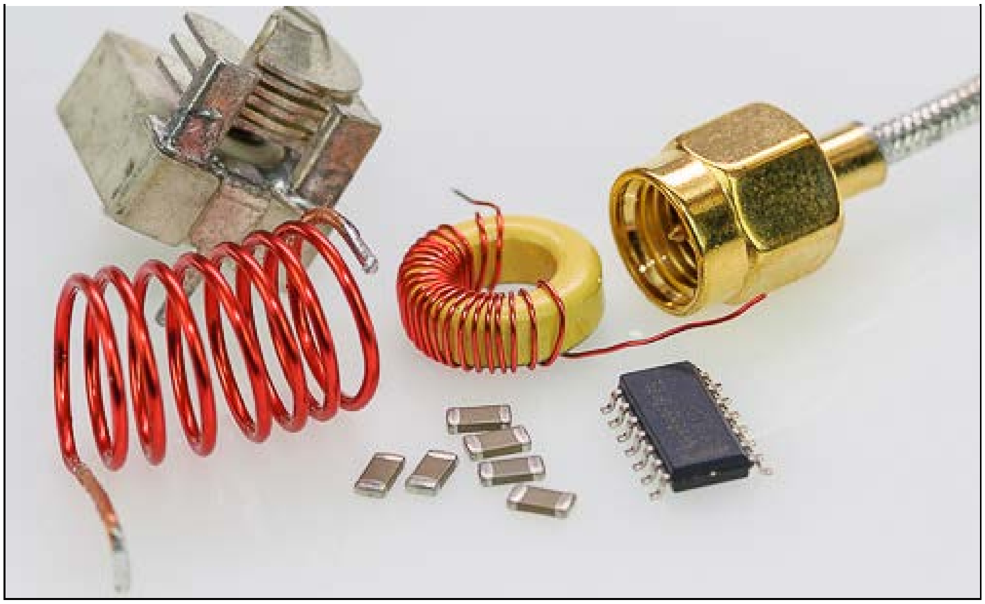
- Experiments with JFET Biasing
- TRF Receiver for WWV
- MF and HF SWL Receiving Antenna
- MF and HF Receive Antenna Splitter
- MF TRF Receiver
- Junk Box Low Pass NDB Filter
- More NDB Ideas and Circuits
- Low Pass Audio Filter Experiments
- TRF WWV Receiver for 5 MHz
- Complementary-Symmetry Amplifier Biasing Basics
- Low Power Audio Amp Experiments
- Two Bravo Receiver Experiments
- ICOM IC-7200 Review
- Electronic Hobbyist Circuits
- Output of a Diode Ring Mixer in an Oscilloscope

Topics 2010 - 2011

- Low Noise Crystal Oscillators
- Crystal Parameter Checker Supplement
- Hobbyist Circuits 2010
- RF Workbench Page 1
- RF Workbench Page 2
- AF Transistor Input Z Experiments
- RF Workbench Page 3
- Misc RF Experiments 2011
- Hobby and Fun 2011
- QRP Modules 2011
- Receiver Band-pass Filters
- RF Workbench Page 4
- VFO-2011
- VHF Butler Oscillator

Topics 2012 - 2014

- VHF to the Max !
- HF Ragbag
- RF Workbench Page 5
- VHF-FM
- Sundry Experiments 2012-13
- VHF - Veronica
- RF Workbench Page 6
- HF Embarcadero
- Caitlyn 310 : UHF Beginnings



Make it — then measure it!

RF — Test and Measurement

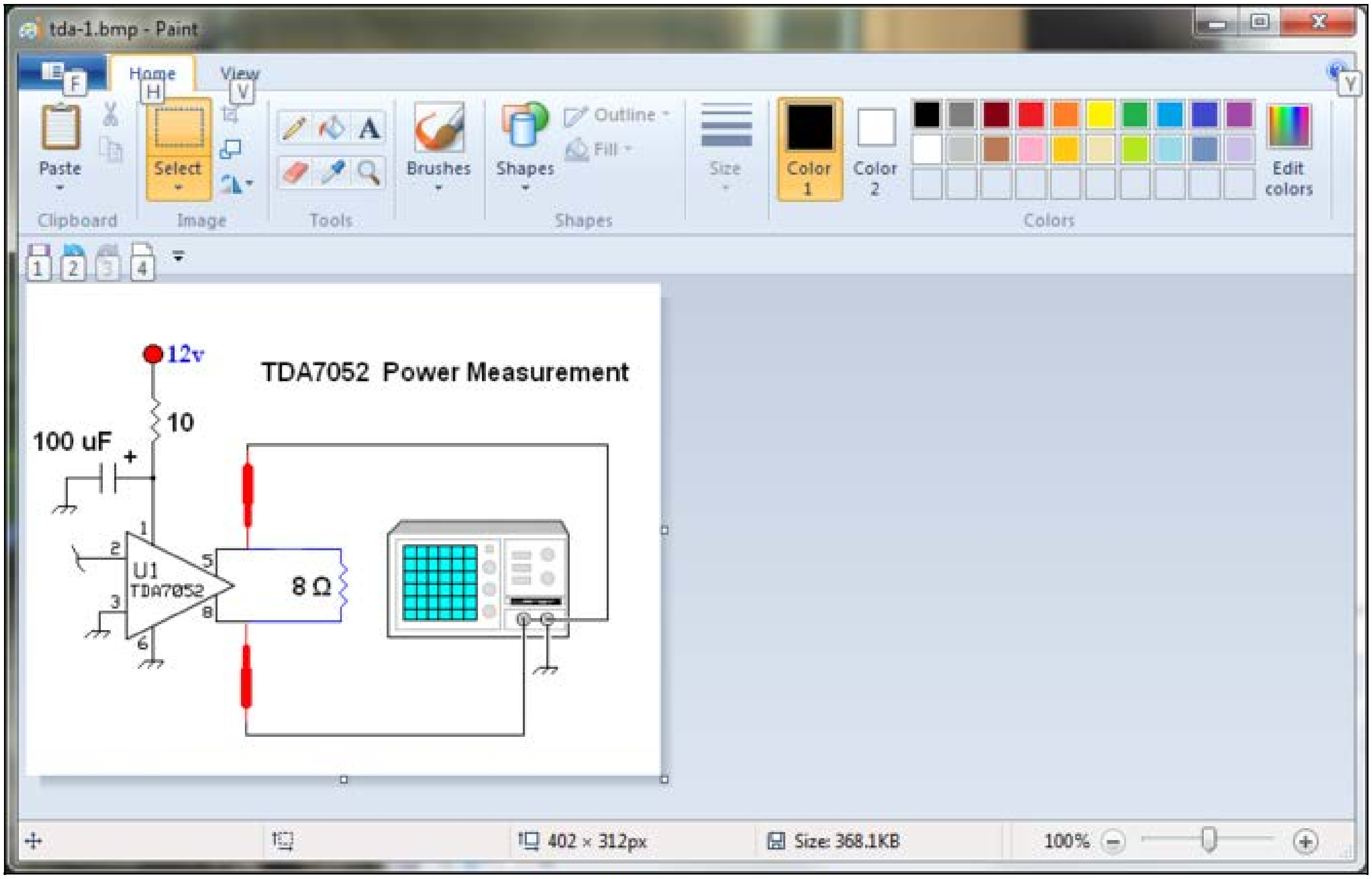
Frequently Asked Questions

1. How to you make your schematics?

I hand draw my schematics using the Paint Program that ships with all Microsoft Windows(tm) operating systems.

I strive to make schematics as clear and as small in file size as possible.The Windows Clipboard is used extensively to copy and paste the desired components from previous schematics to new schematics — very few new components are ever drawn; rather they are recycled from schematic to schematic. My system allows me to paste color 3-D labels and small photographic bitmaps into schematics.

The raw 24-bit bitmap drawings are compressed to 16-bit png files. Prior to May 17, 2010, bitmaps were compressed as 8-bit gif files. An example drafted circuit using the Windows 7 Paint program follows.



At least 25 people have sent or recommended software for making schematics. Thank you for this kind gesture, but I prefer my current method and after 15 years — make schematics quickly.

2. How come you don't supply parts lists? Other people do.

The answer is simple; lack of time. It takes considerable time and effort to put up a new web page and also to maintain a large web site. I save time by leaving the parts list up to the builder. In addition, this site is about experimentation and using what parts you have on hand.

3. Why didn't you answer my email?

I answer all legitimate emails as soon as possible. Our POP3 server gets an average of 2016 spam emails per month (December 2013 data), however, our software removes 99.27% of these and I never see them. Occasionally, legitimate emails are filtered in error and I apologize. I've received as many as 83 legitimate emails in a single day so can get behind. Please keep emailing — when readers stop emailing, I'll know the site has fallen totally obsolete [it may already be obsolete] and delete it.

Our mail server software logs and analyzes the network information of all spammers and may automatically filter and/or block their addresses or even their entire ISPs at the router level. Analysis indicates that 90% of our SPAM comes from just 3 countries and if you happen to live in one of these countries, the filtering will be especially sensitive.

This sounds dogmatic and unfriendly, however, until you've set up a domain and must handle ++ spam emails, endure and then develop router control software and other strategies to handle DDoS attacks and so forth, your completely naive about the 'dark side' of running a homebrew web server. Massive amounts of bandwidth might be otherwise wasted by allowing unwanted server use unless we actively counter these activities to keep the site running well for legitimate hobbyists. Not to mention all the wasted time. Further:

The 3 host web servers, the power, the server software, the security apps, the internet bandwidth etc.are owned or purchased by my family, and as you know, nothing is free. Despite many offers by companies to place ads on my pages, I've kept the site advertisment free and running pops.net costs us a few thousand dollars each year. We ask you to please respect our site for the sake of the experimenters who visit.

Pops.net server rack in our warehouse.

While I appreciate that some people might want to email invite me onto their social networks, I do not have time to participate. All email traffic from or involving social networks see this page for a list is deleted automatically by our POP3 server control software.

I **never** buy or sell parts via email, nor exchange hyperlinks. Never. I do give free parts to those in need though.

All email with the .info domain is blocked.

The number of people selling kits has jumped up by ~4 dB in the past 5 years. Increasingly, builders who need help with kits were emailing me for support. I rarely build kits and my knowledge regarding kit building is nearly 0. Please contact your kit seller for help.

You may wish to enquire with the kit seller about their online support polices and promptness prior to purchase. Additionally you might try the "support" email address provided and see if and how promptly they reply. Most of the popular kit sellers (AADE, Kits and Parts, etc.) provide excellent support to their customers. Like anything else online; buyer beware.

4. How come you didn't link to my web site - I linked to yours?

A big thanks to the folks who link to this web site! The QRP/SWL HomeBuilder site focus is content, not web links. Making a lot of links means spending time testing for and tracking down dead links - the so called "link rot". Time spent on the web site is time away from the electronics work bench. In addition, it is not logistically possible to reciprocate in kind, as hundreds of web sites and blogs have linked this site.

5. I see the word "popcorn" used a lot on this site- what's this all about?

Popcorn connotes the essential theme of the web site; simple, frugal, without fuss and over use of technical jargon, or complex math and engineering techniques. The QRP/SWL HomeBuilder web site is referred to as the popcorn site by many. The site targets hobbyists. The emphasis is fun. The hope is that it will attract new people to electronic design, measurement and experimentation. Hopefully, this site stimulates interest in QRP homebrew electronics.

Soon after I began building electronic circuits, my teachers and the popular electronic-related media of the day pushed me towards etched, printed circuit boards. I complied and this killed my passion for electronics. For me, habitually stuffing circuit boards lacks creativity and freedom.

Later, I discovered people were building guitar and bass amps using point to point wiring techniques with terminal strips and partial circuit boards. I became interested in building and repairing guitar amps and this passion continues today. In 1992, the discovery of 2 QST articles changed everything *for me (complete reference provided):* The Ugly Weekender: parts 1 and 2 by Roger Hayward, KA7EXM and Wes Hayward, W7ZOI; published in QST for August 1981 and June 1992. This was my first exposure to Ugly Construction and it was immediately adopted as the defacto standard bread boarding method in my electronics work shop. In fairness, etched circuit boards are a great tool, but not essential for the experimenter.
After working with Ugly Construction over time, considerable progress was made in understanding RF circuits and one output was the launch of this web site in 1998.

Currently, little has changed, I continue to prefer scratch-homebrew rather than kit-homebrew electronics. My interest in Short Wave radio and analog electronics has grown considerably. For me, electronic circuits hold a certain mystique which arouses my curiosity to learn, enjoy and share. As a lay person, this web site has facilitated meeting some awesome people through email from all continents and it has been a privilege to learn from them, my mentors, book and web authors and often enough; from my mistakes.

5. What do you mean by a 5K1 or 3K3 resistor value?

For E24 or 5% tolerance resistors 5K1 = 5.1K, 3K3 = 3.3K and so on. For E96 or 1% resistors 31.6K is written as 31.6K. All resistors are 1/4 watt unless otherwise specified.

6. How do you measure audio amp output power?

Please see Figure 4 on this web page. Any amp when cranked, outputs much greater power than when it is providing a clean sine wave. The quoted power for any audio power amp on this web site is the maximum average power it will give before the pure sine wave becomes distorted.

7. I noticed a new web page appears and then it is edited for 1-2 weeks. When is the web page completed?

When a new web page is added, it takes a week or so to find and change some of the grammar and spelling errors. Sometimes new ideas or feedback will cause me to further edit a web page at any point in time. This whole web site is a work in progress. The last date any given web page was edited is posted on the bottom of the web page.

8. Do you buy or sell stuff?

No and no. I receive numerous emails from people asking me to sell them stuff. I do not sell anything - no parts, books, coffee cups, ball caps, tee-shirts, ad space — nothing. I do not buy parts in commercial-quantity volumes and have no need to make contracts for obtaining any electronic components. Every week, Asian companies email to ask about buying their parts — please note, my answer is *always* the same: no thank you.

9. Questions and concerns about printing and printability

Each year, a few readers email to complain how poorly the web pages print. This is true and I apologize.

Some people prefer pdf files for easy printing. I have resisted going to pdf format for 3 main reasons:

1. The web site audience is international and many are using web translators. PDF files are 8-bit graphic image files and do not translate.
2. More and more readers are using mobile computer devices and pdf files are a pain for them.
3. We should all print less often to save resources

As an experimenter, I dislike crammed, small-size schematics and feel they should be drawn for *maximum* clarity. Therefore, my schematics tend to have a lot of white space and color contrast. I try to make them no wider than 700 pixels, although sometimes it's impossible to do this. Big schematics are *not* printer friendly. The only practical solution is to click on and open them in a separate browser window for easier sizing and printing.

I also feature big photos which burn up a lot of printer paper. Project photos are important to me; they provide a more intimate glimpse into the bench work and promote the real purpose of the site — building stuff.

A potential printing solution for Microsoft Explorer 8 users; Click

10. I have noticed in your CMOS logic photographs, you don't always ground unused input gates. Isn't this bad?

Proper CMOS logic practice mandates the grounding of all unused input gates. In prototypes and experiments, I don't always do this as I generally want to re-use the IC in other experiments. This is a cost saving measure. When you build a lot of stuff, it can get expensive and recycling parts makes sense. In keeper circuits or critical prototypes, unused input gates are directly soldered to the copper clad board. This also anchors the IC very well.

11. What is the proper URL of the home page?

http://www.qrp.pops.net

The following pre-2006 URL was decommissioned August 6, 2010: http://www.qrp.pops.net/default.htm

12. What are the QRPHB Design Centers and Professor Ivanenko character about?

This web site is about design and not just providing circuits to copy; I'm hopeful that the QRPHB Design Center concept initiated in 2011 will invigorate the site. Design Centers are the presentation of simple, but useful algorithms for amateur builders to advance their skills on the bench.

Professor Vasily Ivanenko ( ), a *fictitious* retired Russian physics professor wants to share his knowledge and give back to society. He signifies each Design Center. Professor Ivanenko was drawn for me by Rod Adams in 1996 using the Paint program that ships with Windows (the same app I make my schematics with). Rod did all of the other original bitmap art for this website including the coil guy and junk box pictures. This character was inspired by one of my favorite photographers: Irving Penn — this photograph, which is all over the web.

A new character; Dr. Natasha Petrovna appeared in late Summer 2012.

The professors are just a good bit of fun — add intrigue, characters on whom to focus and a means of identifying Design Centers. Electronics with just math and physics bores us all. Adding splash, color, clear photographs and characters such as the coil guy or the Professors boosts the site's appeal and provides a creative outlet for me.

13. Why did you kill your blog?

Time mostly. *My* blog wasted *yours* and my time. I carefully analyzed my personal yield from blogs in 2012 to 2013 — for the most part, blogs just entertain + share trivia, or rehash someone elses idea(s), or 'innocently' attract you in hope to sell stuff — and sometimes, just fulfill the author's need for attention. I don't seek, nor have time for entertainment or spectacle within my RF hobby and I certainly don't wish to waste *your* time. Each to his own, I suppose.

My analysis showed that unfortunately, blogs rarely boosted my understanding of electronic design or measurement practices. My ardent focus is to learn + improve and then pay some of this knowledge forward on a web site. Of course, everything in context — many exceptional people blog. For example, Dave AA7EE, or Jason, NT7S.

Most of the RF design and measurement people I follow keep old fashion web sites and provide generous email support. Design and measurement web sites, plus reputable and/or peer-reviewed industry and hobbist books, journals and multimedia work best for me.

Further, great elmers don't just publicly hang-out on blogs, or web server groups, or publish Utube videos — some just check their emails and when asked — give wonderful support without fanfare. Hats off to these humble folks. Thank you!

14. What oscilloscope should I buy?

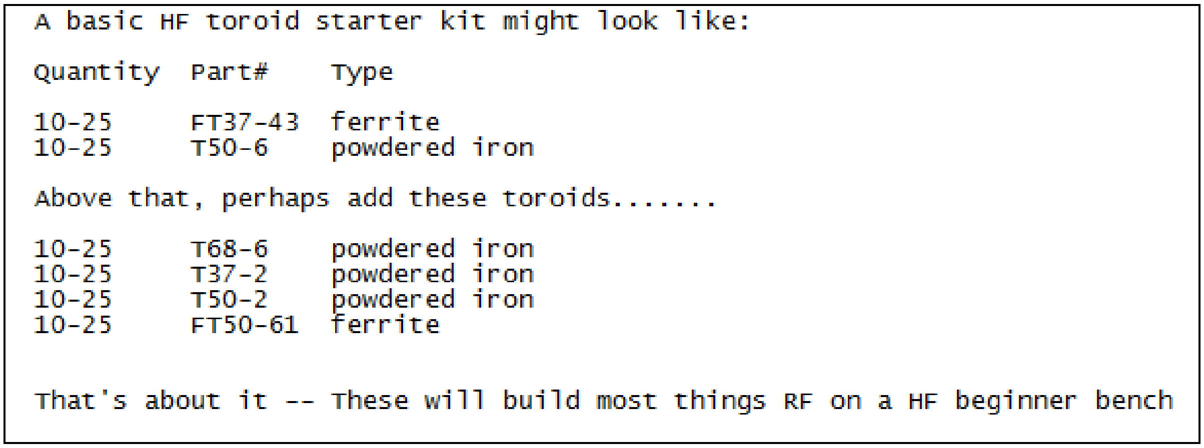
Yikes — a tough question I get nearly every month. Please do your research. My best answer is buy the best 'scope you can afford. Are you a casual experimenter, or sit in your lab a lot? I prefer DSO's , however, made due with a old boat- anchor CRT for my first 10 years. The Rigol 1052/1152 seem popular entry-level choices due to their cost versus performance ratio. On the other hand, view this video to see how much Rigol DSO technology has changed.

I owned and sold my 1052 to a builder in Michigan — a worthy choice like many other 'scopes. IMHO, the FFT and math functions on the 1052 and 1152 suffer due to low memory depth and clock jitter.

Even if you only work at HF, a bandwidth 5X higher than your main frequencies of interest works better for showing harmonics. Again, I advise people to simply buy the best 'scope they can afford since it will form the heart of your test bench.

15. I'm a beginner — what toroids should I buy?

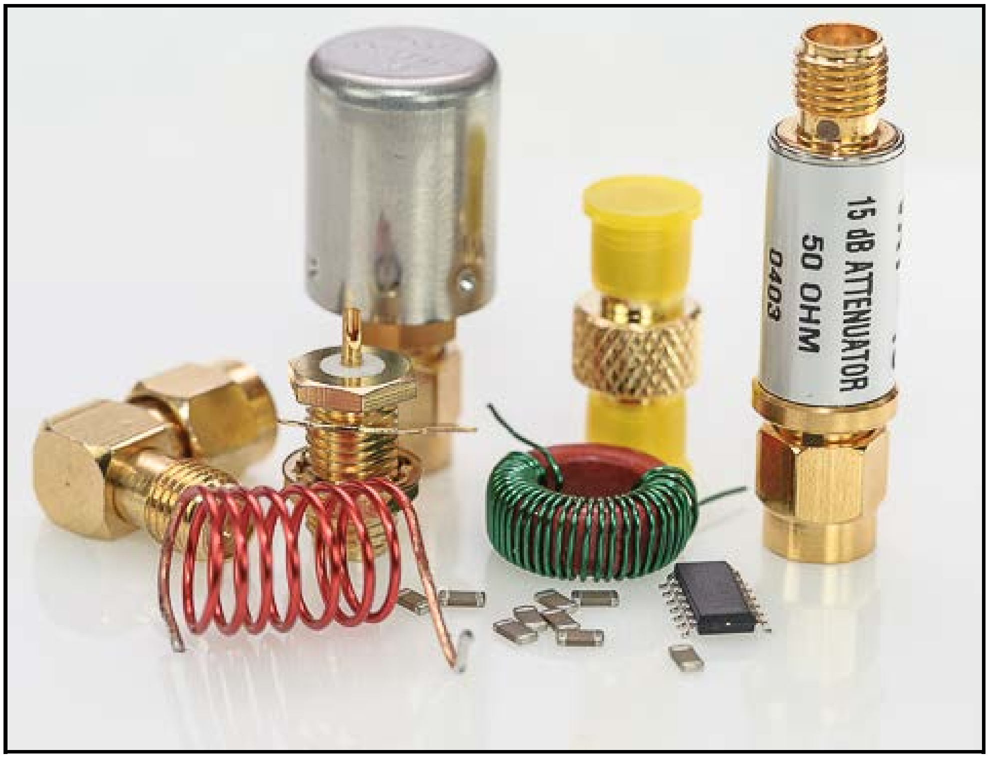
Opinions will vary, but here's what I recommend. Buy any quantity you wish, but sometimes minimum quantities apply and shipping to some countries costs dearly, so I tend to order enough parts to last me for awhile.



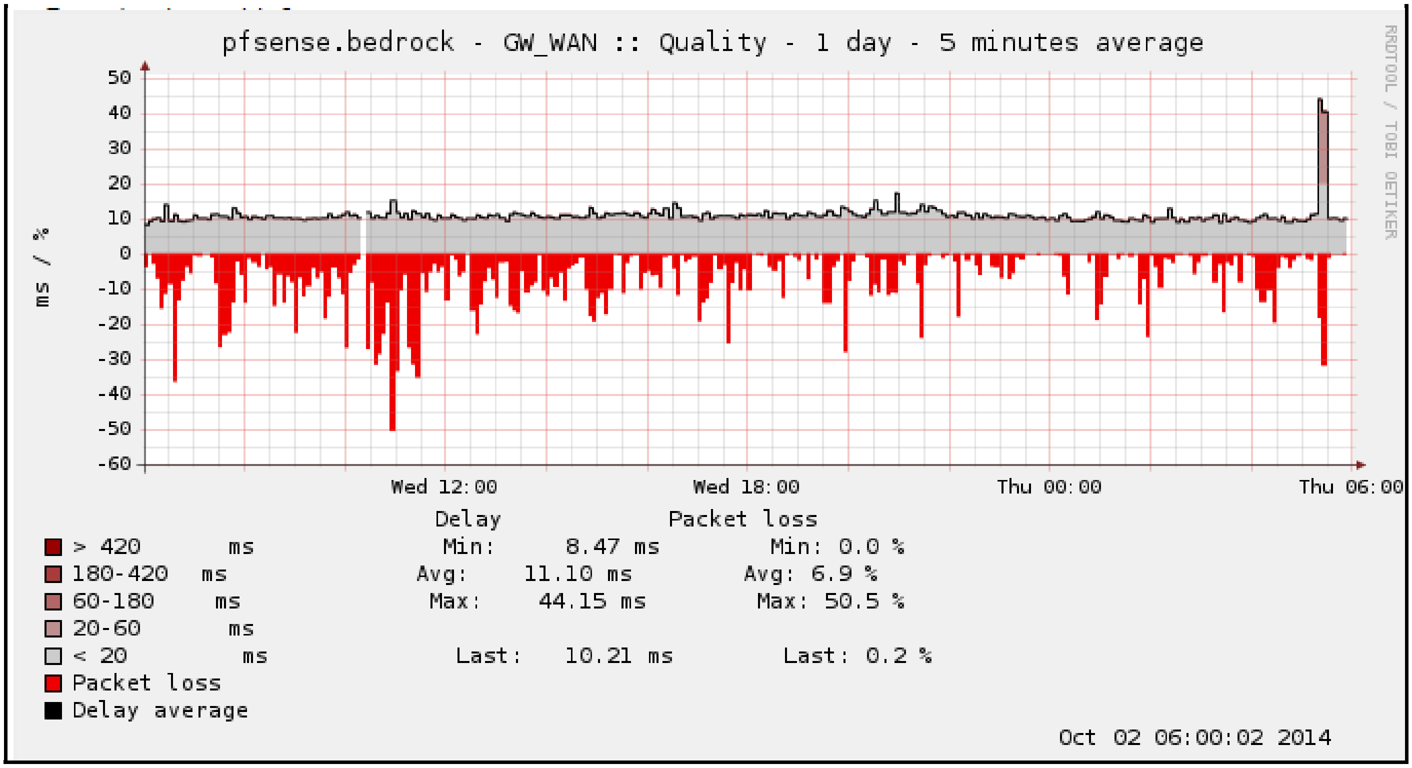
```
A basic HF toroid starter kit might look like:

Quantity  Part#    Type

10-25     FT37-43  ferrite
10-25     T50-6    powdered iron

Above that, perhaps add these toroids.......

10-25     T68-6    powdered iron
10-25     T37-2    powdered iron
10-25     T50-2    powdered iron
10-25     FT50-61  ferrite


That's about it -- These will build most things RF on a HF beginner bench
```

RF — Test and Measurement

QRP — Log: Updates To Permanent Content



Oct 5, 2014 --- The end of QRPHB? Expect the Qrp site to go down each day for 1-4 hours. As I wrote before, we're losing an average of 10% of the outgoing packets and this peaks as high as 40% when many people are accessing the site. See an example graph below.



On Oct 2, 1 of our ISP's techs came and tested our WAN -- "it's better than most" he said. Hopefully they can do something to boost the performance on their side. Wave bought out all the local competition and we're stuck with them [this is happening everywhere]. Click for 1 of the many links writing about the emerging internet cable company monopoly.

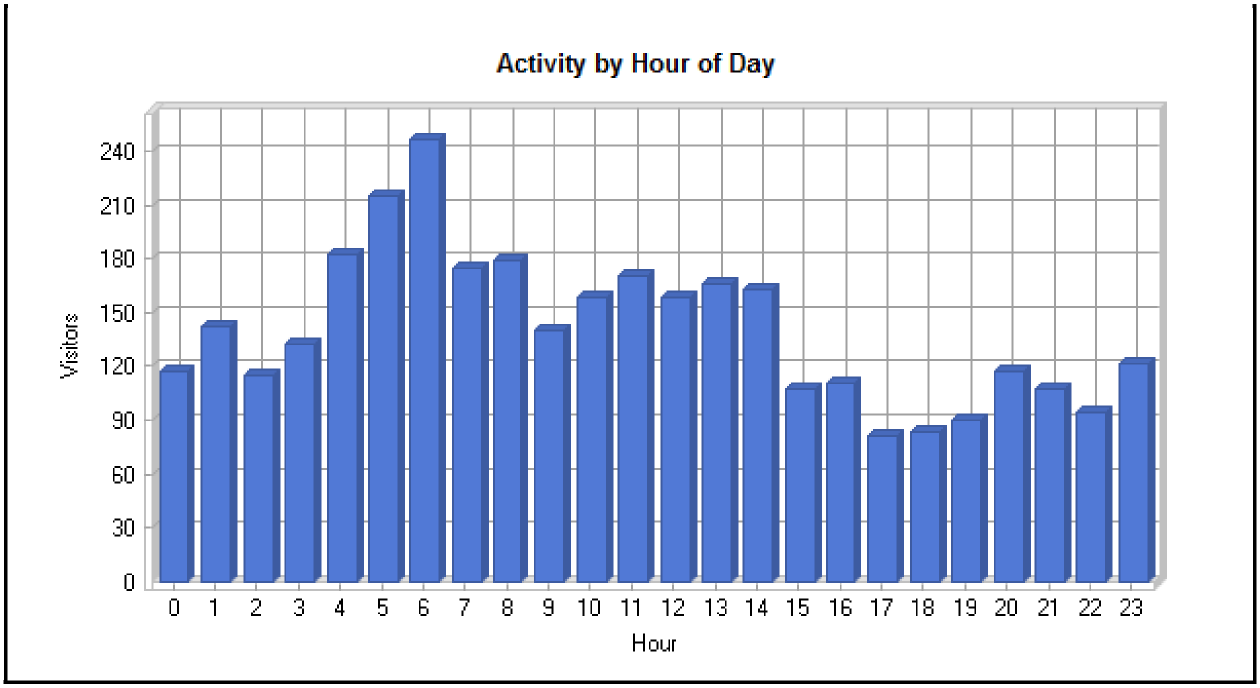
You'd think that in 2014 we could properly host a web site like we've done since 1998? To boot, Stuart is paying for a high bandwidth business account and getting less than 1% of the promised bandwidth. We chatted today and will wait patiently for Wave to improve our outflow - but only for so long.

If Wave can't or won't boost performance, I am leaning towards taking this web site down as opposed to moving it -- it's been a good run. QRPHB creates a lot of work and I could get way more done without the web site hassles. So, 2014 might be it for me. If so, thanks for coming here all these years and best of luck with your experiments!

--- Sep. 19 --- Wave Broadband, our Internet provider suffered major problems recently and I shut the site down for 4 days rather than have it limp along pathetically. Well the Wave saga *continues*: the site loads slowly and pictures are missing etc.

Our email server remains working even when the main site is down. You can always email me if the site is not working for you -- or if you have a comment or question.

Vistor volume rose this Summer so we added a better router that provides bit by bit data performance collection and new features to help throughput. At minimum user bandwidth, a unique visitor hits the site every ~0.7 minutes. I'll start formal bench experiments on October 3 --- to kick off the sites' 16th season.



August 31, 2014 — Kit to upgrade my HP frequency counter to 3 GHz added as Section 5 on Caitlyn 310 — UHF Beginnings.

August 18, 2014 — QRP-POS Data on the Sundry Web Page. Look at the end of Section 8: Popcorn AF Amps For Receivers — Reprise. This new, all-discrete AF stage will go into the Funster Line receiver and ranks as 1 of my best in terms of power and headroom.

August 12, 2014 — Funster Line: a QRP 40M band CW trans-receiver added to HF Embarcadero web page as Menu item 4. Click here for Funster. Only the transmitter is presented for now.

June 18, 2014 — I tweaked the page now called About... on the top level menu. Also, on this web page, I added a new essay for 2014 just under the essay for 2010. I'm off the bench until Fall, but hope to add a little content on rainy days or such. I'm about 2 years behind in presenting some of my experiments.

May 22 and 26, 2014 — New Supplemental Web Page for VHF-FM launched. This page and another supplement will house some new receivers over time. The new supplement is linked on the original VHF-FM web page in section number 4.

May 5, 2014 — Section 7: NE612 Mixer Diddy added to VHF Veronica

April 14, 2014 — Completion of Return Loss Bridge Experiments [added Bridge #4] on Caitlyn 310.

April 7, 2014 — Return loss and VCO experiments added to Caitlyn 310.

March 7, 2014 — Section 3 added to VHF-FM. A DC Converter for VCOs. **March 25** QRP-POSDATA for March 2014 Poor Hams Scalar Network Analyzer (PHSNA) added to Section 1 of Sundry.

February 14, 2014 — Caitlyn 310 — New repository web page for my venture into UHF. Click. Surprisingly, the site averages ~ 3000 unique visitors every 24 hours. Click for the Feb 25 tally

January 15, 2014 — The FAQ was often missed and therefore moved to the top level menu + editted/augmented.

January 4, 2014 — A follow-on version of the K3NHI QEX power meter added to RF Workbench 5 as section 6.

December 15, 2013 — I added Section 3 to HF Embarcadero. VXO and VCXO Notes. I significantly updated the VFO-2011 web page on Dec 17, 2013.

November 7, 2013 — Section 6 added to VHF — Véronique. 1 photo added to the end of the Ugly Construction page. Severe Fall weather conditions took down the server for ~1 hour today.

November 1, 2013 — I added Section 2 to HF Embarcadero. A vestigal set of notes regarding my attempts to update the Popcorn Superhet receiver.

October 31, 2013 — Section 5 added to VHF — Véronique.

October 27, 2013 — VHF-FM re-formatted and Section 2 added. I imagine this page will disinterest many.

October 18, 2013 — 3 rarely accessed web pages removed from drop down menu, but not deleted from the server. New QRP-Posdata added to HF-Ragbag (near the end). Section 3 edited and Section 4 added to VHF — Véronique.

October 11, 2013 — Build Season 15 begins [I bench experiment Oct to May]. I added HF Embarcadero to hold all my HF and perhaps AF experiments this season. I'll also add content to VHF FM, VHF — Véronique and RF Workbench 6 over the next 12 months. Thanks.

September 9, 2013 — Blog deleted.

August 8, 2013 — Section 2 of RF Workbench 6 added. Measuring PA collector V and I to calculate efficiency. On August 16 — I added a new essay on the Ugly Construction page called Is Ugly Construction Less Reproducible than Manhattan?

June 25, 2013 — I started RF Workbench 6. It will take 1 year to complete

April 14, 2013 — I added Section 3: 50 Ω MMIC Bench Amplifer to the VHF 2013 Veronica web page.

March 31, 2013 — I started a new content page called Pin Outs

March 25, 2013 — Section 9: an essay about the 1981 Progressive Receiver, plus the final section — 10: Miscellaneous Pictures and Figures added to Sundry 2012-13

March 18, 2013 — Section 8. Popcorn AF Amplifier for Receivers — Reprise added to Sundry 2012-13

March 10, 2013 — I added a new VHF content page for 2013: VHF-2013 - Veronica

February 16, 2012 — I added Section **7** -- A Journey Above HF -- to the Sundry Experiments 2012-13 web page.

February 14, 2012 — I added Section **6** -- Non-Mechanical Iambic Paddle -- to the Sundry 2012-13 web page.

February 1, 2013 — A seperate QRP—POSDATA added to RF Workbench 4. Now QRP— POSDATA 1, 2 and 3.

December 15, 2012 — I added Section 4 to Sundry Experiments 2012 - 2013 . A PLL circuit from EMRFD.

January 5, 2013 — I added Section 5 to Sundry Experiments 2012 - 2013 . A simple AF feedback amp. An essay concerning L-C meters was also added on RF Workbench 5. Section **5.**

December 22, 2012 — I deleted the web page ***Tuning VFOs with a PN Junction*** since some of the experiments were poor quality and performed back in 1998 when I was more ignorant than now. I've learned much since then and my new VCOs from the past 1-2 years reflect this knowledge. The ***Selected QRP Reading list*** and ***Cascode 7 Receiver*** web pages were also wiped.

December 15, 2012 — I added Section 4 to Sundry Experiments 2012 - 2013 . A PLL circuit from EMRFD.

December 3, 2012 — I added Section 3 to Sundry Experiments 2012 - 2013 . Interview with Jason from Etherkit

November 13, 2012 — QRP — Posdata added to the VHF to the Max web page. Section 5: Z-Comm VCO.

November 1, 2012 — Sundry Experiments 2012 - 2013 web page added.
QRP — PosData added to Section 2 of **Power Meter Calibrators** on RF Workbench 5 and updated again on Nov 22, 2012.

October 16, 2012 — VHF FM web page added. Already, it has spawned a first supplemental web page

Sept 24, 2012 — QRP — Posdata added to the end of the HF Ragbag web page. I added a bypass and decouple network for HF to lower VHF.

August 17, 2012 — QRP — Posdata #2 added to bottom of the Receiver Band-pass Filters web page.

August 6, 2012 — Section 5: **Some Experiments with RF Bypass Capacitors** added to the HF Ragbag web page. Also a new QRP — Posdata added to the bottom of the Crystal Parameter Checker web page.

August 1, 2012 — QRP — Posdata added to the Receiver Band-pass Filters web page.

July 12, 2012 — I added a corrected schematic on the Wee Willy page: Wayne. M0WAY — 14 MHz PA under August 25, 2011. Also, added a new essay on **Microphonics in DC Receivers**. See Section 4 on the HF Ragbag Page

June 23, 2012 — RF Workbench 5 added. Click here.

May 31, 2012 — Galina discovered that I neglected to publish the proper version of the Hobby and Fun 2011 page and corrected my error. A fine-tuneable Wien Bridge Oscillator idea from Ken Kuhn now appears at the page bottom.

April 21, 2012 — I heavily edited RF Workbench 1 and 2.

April 13, 2012 — Web site purge. I removed RF Filters, VFO 1998, QRP Workshop Ideas, Miscellaneous Schematics and Photos, Base-biased VFO, Funster Transceiver, Miscellaneous Circuits and Ideas 2005 and Crystal Oscillator Offsets. **Reason:** substandard.

April 6, 2012 — 50 MHz Receiver Pre-amp and Filter added to VHF to the Max web page .Section 4.

March 26, 2012 — HF Ragbag web page added to top-level menu. Non-VHF experiment repository for 2012.

March 19, 2012 — VHF to the Max web page added to top-level menu. 50 MHz VCO experiments added to this page in Section 3.

March 14, 2012 — EMRFD review edited.

February 29, 2012 — Minor edits to the Audio Transistor Input Impedance Experiments web page. Also, I updated the calculation of the common emiiter amplifier base input resistance using the better formula: **Rin = (B+1)*(re + RE')** [while ignoring REB]. This is my favorite web page on the site.

February 26, 2012 — QRP— Posdata added to the bottom of the 2nd NDB web page

February 17, 2012 — I introduced a new miscellaneous VHF page: VHF to the Max — I'll slowly add stuff over 2012. Major editing done to the Broadband Transformer web page.

February 4, 2012 — RF Workbench 1 and 2 significantly edited. 2 new photos added.

January 28, 2012 — QRP — Posdata added to Crystal Parameter Checker web page and to QRP Modules 2011 under 7 MHz VCO Experiments on this web page. I re-wrote the temperature compensation section of the VFO-2011 web page and added 3 photographs.

January 3, 2012 — Web site change: I update web essays with an "epilogue section". In 2012 and on, they will be called QRP — Posdata (Spanish for post-script or epilogue). Posdata #2 added to the RF Workbench 4 web page.

December 17, 2011 — The Butler Did It ! - First VHF Experiments 2011 web page added. A 50 MHz frequency doubler added Dec 26, 2011.

Nov 12, 2011 — Our server went down for 16 hours. Both AC power and cable Internet to the warehouse failed after a rain storm and wind gusts knocked down some trees that severed the hydro and cable wires. Expect more weather-related down time as Winter approaches.

Oct 15, 2011 — VFO-2011 added.

Oct 2, 2011 — RF Workbench Page 4 added. On **Oct 17, 2011** I added an epilogue.

Sept 20, 2011 — Double Stacked Toroid VFO 2008 web page pulled off. It was substandard and some of the material will re-emerge on a VFO 2011 page this Winter.

Sept 19, 2011 — I updated the SWL essay since it was 6 years old and much has changed with respect to Internet radio. Over time I have received ++ emails expressing different views. I am more a SWL than a Ham and offer just 1 opinion and live by a "each to his own" mantra. Context is everything - this is a radio electronic experimenters site that recognizes SWL'ers are important members of the radio community. Click. Wee Willy web page updated again!

Sept 12, 2011 — Design Center concerning popcorn receiver band-pass filters added. Click.

Aug 26, 2011 — Minor update at the end of Wee Willy DSB transceiver. ***New Junk Box Blog format***. Change is good.

July 30, 2011 — New web page QRP Modules listed on the main menu. Currently under construction.

March 31, 2011 — 2 photos added to the Ugly Construction web page

March 19, 2011 — New content; Hobby and Fun 2011 . I'll slowly add more stuff over the year.

Feb 12, 2011 — New content. Miscellaneous RF Experiments 2011

Dec 29, 2010 — New content. RF Workbench Page 3

Dec 12, 2010 — Final additions to the the 2010 Hobbyist Page added — these concern matching FETs, BJTS and diodes.

Nov 10, 2010 — Some editing and 2 photos added to RF Workbench page 2.

Oct 9, 2010 — This October marks the 12th season of experiments for the site. I have 4 partially completed web pages on the go — pure craziness. I decided to finish them 1 at a time and then add them sequentially. Today, a new web page was added and is 1 of 2 supplements to a future main QRP audio page: Audio Transistor Input Impedance Experiments. The first new "permanent" content since March 2010.

Oct 7, 2010 —The Junk box page lay out was simplified: Accommodating the various modern Web devices plus screen resolutions proved difficult with the old html code. Although less impressive, the new format updates quickly and looks the same on every computer.

Aug 15, 2010 — The cable supplying the Internet connection failed. The site went down for 25 hours. Expect more shut downs over the next couple of days as we resolve any remaining problems.

August 6, 2010 — I decommissioned the historic home page URL http://www.qrp.pops.net/default.htm. This page was a html hard-coded parallel version of the the *correct* home page URL http://www.qrp.pops.net. It was just too much work to continue to update the old (pre-2006) home page in addition to the proper home page.

June 28, 2010 — Web subtitle change : **Amateur Radio Electronic Design** to shorten the name and reflect the site's main purpose.

Effective May 17, 2010 — Schematics now use the 16-bit png format. I have abandoned the 8-bit gif format

May and June 2010: The pops.net net control crew rest up for the next big wind storm. The servers lost power for many hours on May 4 and June 12 due to bad weather. Severe storms arising from the Pacific Ocean threaten our AC power lines each Spring.

March 2010

Wide Range L-C Oscillator added to Hobbyist 2010 page. Link **March 7**

JavaScript Applet **K** added to the QRP Tools page **March 11**

RF Workbench Page 2 added. Link **March 15**

February 2010

JavaScript Applet **J** added to the QRP Tools page **Feb 6**

Editing of the Low Noise Crystal Oscillator web page **Feb 22**

RF Workbench Page 1 added. Link **Feb 18**

January 2010

Experiment #6 added to the Hobbyist page **Jan 2**

New content Low Noise Crystal Oscillator **Jan 10**

2 photos added to the bottom of the Ugly Construction web page. **Jan 12**

1 photo added to the Broadband Transformers page **Jan 12**

New content Hobbyist Page 2010 **Jan 13**

New content: Supplement to JavaScript Applet G which is located on the QRP Tools page **Jan 11**

Additions to the Low Noise Crystal Osc page: the DSO versus CRO essay, plus 5 MHz crystal oscillator added **Jan 31**

JavaScript Applets H and I added to the QRP Tools web page **Jan 31**

December 2009

It has been brought to my attention that the email replies I am sending are not compatible with some of the latest email software such as Thunderbird etc. The reason was that I was using a homebrew email program written over 10 years ago. It is now obsolete. A new email platform is now in place along with a completely **new email address**. Consult the email web page for more information. **Dec 30**

Experiment #5 added to the Hobbyist page FAQ updated and edited. **Dec 30**

Experiment #4 added to the Hobbyist page **Dec 26**

Experiment #3 added to the Hobbyist page **Dec 23**

What does the output of a diode ring mixer look like in your oscilloscope? This has become a FAQ. The question is answered in a contribution by Wes, W7ZOI. Big thanks to Wes for this content. **Dec 22**

I am very pleased to present the **Mike, KL7R Memorial Receiver Experiments**. Click here **Dec 19**

New bulleted list format added to this page to improve readability **Dec 19**

A minor addition was added to the bottom of the Ugly Construction web page. Flux pen photo and text. **Dec 19**

Minor updates to the VFO 2008 "Stacked Toroids" web page under **Epilogue - December 19, 2009**

Update to JavaScript applet **Item E: Calculate Cut off Frequency for an RC Low Pass Filter**. Now has a capacitor range from 0.1 nF to 1500 nF. QRP tools page. **Dec 19**

Editing plus a photo added to the broadband transformer page. This page was improved to support an upcoming project. **Dec 17**

Drafting errors on this schematic corrected (fuse position+ negative rail LED polarity). From this web page. Thanks to Paul, K0EET and Tom for the good eyes and their emails. **Dec 16**

VFO Experiments 2009 updated again; 3 images added. Supplemental web page added and updated **Dec 13**

New JavaScript applet added to QRP Tools page; **Item F: dBm calculator. Dec 12**

Nov 22-27, 2009

1 Hertz Precision Time Base added to the Hobbyist Page.
VFO Experiments 2009 added
The top level menu item "Java Tools" was renamed **QRP Tools.** This menu provides a link to my Webmaster's page and some basic JavaScript applets. Some new material will appear on the Junk Box page during this time.

Nov 20, 2009

Finally; Fall-Winter experiments begin. A small update on the VFO 2008 web page under A 3.5 MHz VFO for Diode Ring Mixers was added.
The RF Preamp page was updated with some content that was first presented on the Junk Box web page. I have received a lot of email about these amps and decided to permanently add them to the site.
The W7ZOI file linked on this page is now in pdf format.

Nov 1, 2009

The Ugly Construction web page was augmented and re-written. 2 new photos were added. This term actually came from Wes and Roger Hayward.
A new Electronic Hobbyist page was added to the drop down 2009 menu.

Oct 25, 2009

Some small updates to the Junk Box page were made. I don't ever think I have been so excited about upcoming Fall and Winter experiments as there are a number of cool, new ideas in my notebook. Extra work and travelling have kept me off the bench, but this will cease in mid November. After that, it's back to the work bench. The Fall-Winter experiments will include some HAM, SWL, general electronics and tube guitar amp experiments. Thanks for your feedback and ideas!

Oct 19, 2009

RSS feed. Click on the **orange RSS icon** above to establish a feed. I will only show 1 item; the latest addition of major new content to the site. Additions to the Junk Box page will occasionally be counted as "new content" and will be included on the feed.

Can anyone guess what brand of beer is in the tool box ? Yes.... its

Oct 12, 2009

Fall weather has come to Western North America! Thanks to Cor, PA3COR for debugging the JavaScript code on this page. Apps number D and E now work in Firefox. Rediscover the fun and learning of scratch homebrew electronics!

RF — Test and Measurement

About My Web Site

Welcome Friends!

Introduction

Welcome to the QRP and SWL Homebuilder web site. I write about my experiments with relatively simple and primitive electronic circuits.

Avoiding excessive algebra and obscure parts, I emphasize and show fundamental bench practices.

Through real experiments I examine topics to challenge and intrigue amateur designers — providing examples and describing ways to plan, problem solve, breadboard and measure your circuits.

As amateur experimenters we ought to advance in our hobby; not just perform cookbook electronics. Designing and improving your circuits requires considerable knowledge and effort. Fortunately, others selflessly share their ideas to teach us.

In time, you may recognize your electronics workbench as your greatest teacher. Bench experiments involve us thinking about and measuring our circuits so we know what's happening instead of relying too much on folklore, guessing and copying others. Designing and/or simulating circuits with software can enhance your learning but does not obviate the need to spend time in the trenches with meters, wires and solder.

People often learn skills by modeling others. We need sound examples of how other builders work and think to inform our own designs — inspired, creative and active learning driven by experience and reflection.

At some level, our bench experiences are stories of growth and realization sparked by going and doing. For example, why did the designer choose a particular resistor value? You try different resistor values while measuring the results and increase your knowledge. Collecting schematics, kit building or just thinking do not provide as intimate a learning experience as soldering your ideas on a bare copper board. Talking, tweeting or day dreaming about design is not the same as *doing it*.

Russian novelist Fyodor Dostoyevsky describes the contrast between real life and passivism; "love in action is a harsh and dreadful thing compared to love in dreams", The Brothers Karamazov, Братья Карамазовы. Dare to dream, but better yet, dare to innovate — to design and build *your own* circuits. You may start by just modifying a favorite circuit or scaling a stage to another frequency. We need more innovators and less imitators to grow and sustain our great hobby.

Electronic design produces more than a completed circuit. On the bench, even joy is experiential – a moment of discovery (or several discrete moments) yields more pleasure than stuffing a circuit board or operating a piece of gear. Creativity trumps process every time!

I hope this site demonstrates my passion for building basic, "popcorn" circuits and sharing ideas. Please remember I am just a lay person experimenter and not an electrical engineer.

Regards, Todd, VE7BPO

Essay for 2010

Building or buying test equipment and acquiring a good reference library are important to your experiments. Spurred by the realization that sound bench measurement practices are at the heart of good design, test equipment receives greater focus in 2010 and on.

A reference library is vital to our electronic experiments; good examples lead to better experiments. Poor circuits are everywhere and some builders can't tell a good design from a bad one. Minimalism and simplicity aren't excuses for sloppy design when your goal is to learn. Collecting and sharing well designed circuits helps us avoid wasting time and experiencing frustration. Circuits with attributes like well defined input or output impedances, low noise or harmonic distortion are desirable to fuel experimentation. Look for better quality circuit examples in 2010 and on.

The Internet is changing how we read and write. The prevalence of small portable web devices such as iPhones, ever increasing numbers of web sites and blogs, and the use of search engines create fierce competition among sites. Modern sites attract your attention with varied visual, aural and textural media and unfortunately, hype and pseudo-journalism. Narrative writing is more skimmed than read. Brief is in — bullets, subtitles, lists and graphics replace long lines of narrative prose which no one seems to have time for anymore.

Have you noticed the changes on this website? New content still contains lots of narrative writing, but assumes an active voice, with emphasis on brevity, clarity and speaking directly to you, the reader. Sharing mostly obsolete, analog 1970's-style circuits, QRP/SWL HomeBuilder attracts a tiny, niche audience. I believe the success of this website depends on providing good and diverse content — not Tags, RS feeds, adopting net-speak, or self promotion. You be the judge.

Essay for 2014

The Internet of Everything?

Bucking the trend, my contribution to amateur RF homebrew remains informational and not social. Why?

Social media information represents a Pandora's Box of good science and opinion, mediocre thought, or trash potentially created and/or disseminated by anyone who's connected. We accept that much of our social media content doesn't come from the best or brightest — some people are just plain interesting, or express themselves vigorously, seem like-minded, or touch our hearts.

Some builders, like me, seek objectivity and not just "likes" and "follows" based on sentiment and spectacle. While a few radio builders may prefer to join hands and sing Kumbaya, or pat themselves and others on the back simply because their breadboard actually works, a trifling of us care more about how and why our circuits work. We *like* measures and measurement tools and *follow* science, experiments and the works of those who shine brightly.

What's wrong with plain information, unfettered discovery, experimental rigor, objectivity and rational, kind thought?

It's not that these characteristics don't attribute social web clients — they do, but the negative impacts of social media worry me a little. A brief list of concerns: loss of privacy, the threat of wasting time while really just isolating ourselves from our real friends + family. The numbing exposure to the Internet of Ads and Spam. Still too, bubbling up like purulent sores come the charlatans, the misinformation peddlers, the opinion spammers, and those who anonymously leave stinging sarcasm, or outright hatred [hostile online comments that attack people, or divert a healthy flow of ideas].

Running a low-tech web site with nearly 0 commercials suits me better.

SEO — Search Engine Optimization

I've read that Google analyses your web site content, the number and quality of the sites that link to your pages, their search engine clicks and so forth. In part, Google seems to rank a site based on how relevant and authoritative they believe it is. Some people specifically employ SEO techniques to gather in more traffic.
To my surprise, each year, tens of thousands come to this site via search engines like Google. I don't think my material seems too relevant or authoritative. I've made no effort at SEO, so I conclude that you, my readers have more to do with the site's success than anything I've ever done.

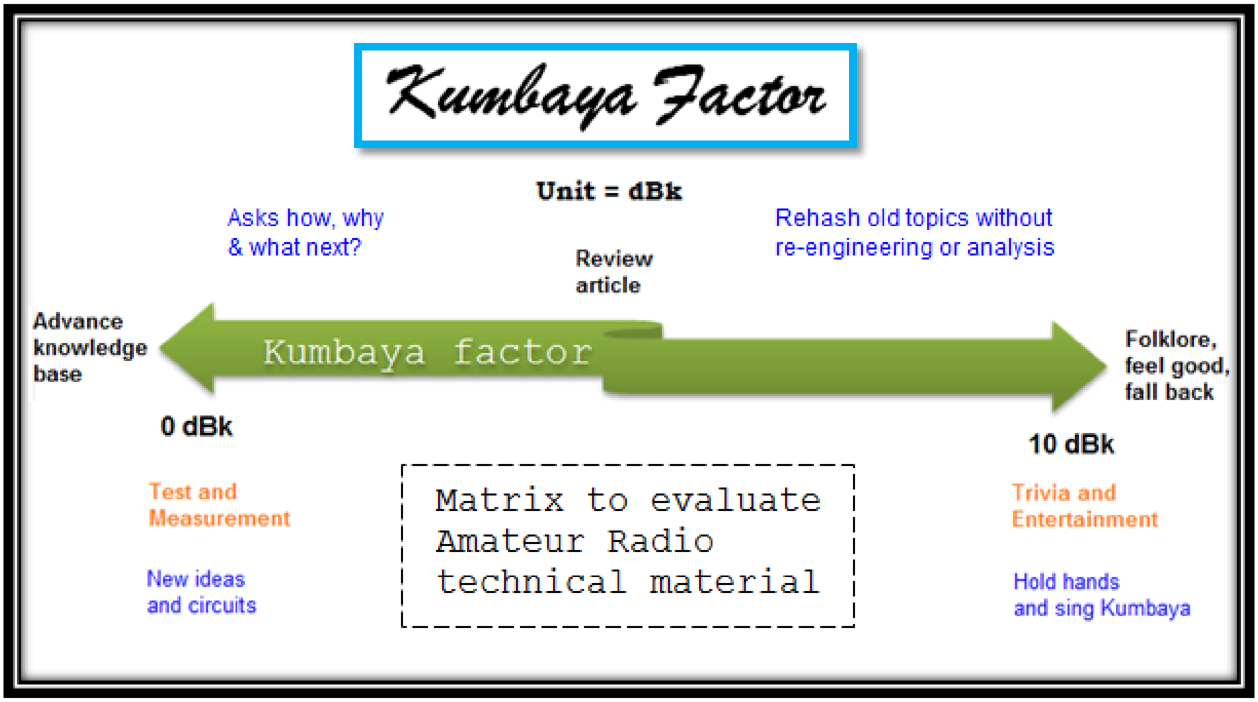
Thank you.

Hope Invigorates

Invigorated by the excellent work either emailed to me by experimenters such as Michel F6FEO, or Dick, N4HAY; or posted on blogs or Community sites like Yahoo, I feel hopeful about the future of our hobby in 2014. The PHSNA Yahoo group leaders, the recent work of Jason, NT7S, Steve VE7SL and many others show that *amateur design* experiments still have a pulse + respirations.
The aforementioned get my vote for their MOF like behavior: a strong blend of creativity, tradition and quality.

Looking Ahead — Future Site Content

Most of my new receiver work involves quadrature and in-phase mixers fed with (2) local oscillators; 1 output shifted 90 degrees from the other — essentially, EMRFD Chapter 9. Even my Funster [a personal, lowbrow trans-receiver I drag onto hill and dale] now contains phasing receiver circuitry to reduce the opposite sideband by 20 dB along with further low-pass filtering. I hope to add some Funster content to HF Embarcadero in Winter 2014. Like many of you, the 1 resource I lack the most is time.
While I'm thrilled with the notion of a receiver appliance that contains just an antenna, LNA, ADC and some sort of "wonderfall" display or speaker, I'm still smitten by analog design with hardware. Still, the I-Q mixer will offer a nice transition into SDR should I ever wish to spend my free time writing C# and not melting solder.



Best to you!

Miscellaneous

My special thanks to Wes Hayward, W7ZOI for his generous support and elmering over many years.

EMRFD is the main reference of my site

All permanant content circuits were built and tested. Schematics are drafted as carefully as possible. Please accept that bench and/or drafting errors may occur. No liability arising from the application, use, or misuse of these projects that results in direct or indirect damage or loss is assumed.

Full price is paid for all parts used and no monies are or were received for promoting any products or companies on this web site. Any ads, hyperlinks or mention of commercial products or companies is out of courtesy only.

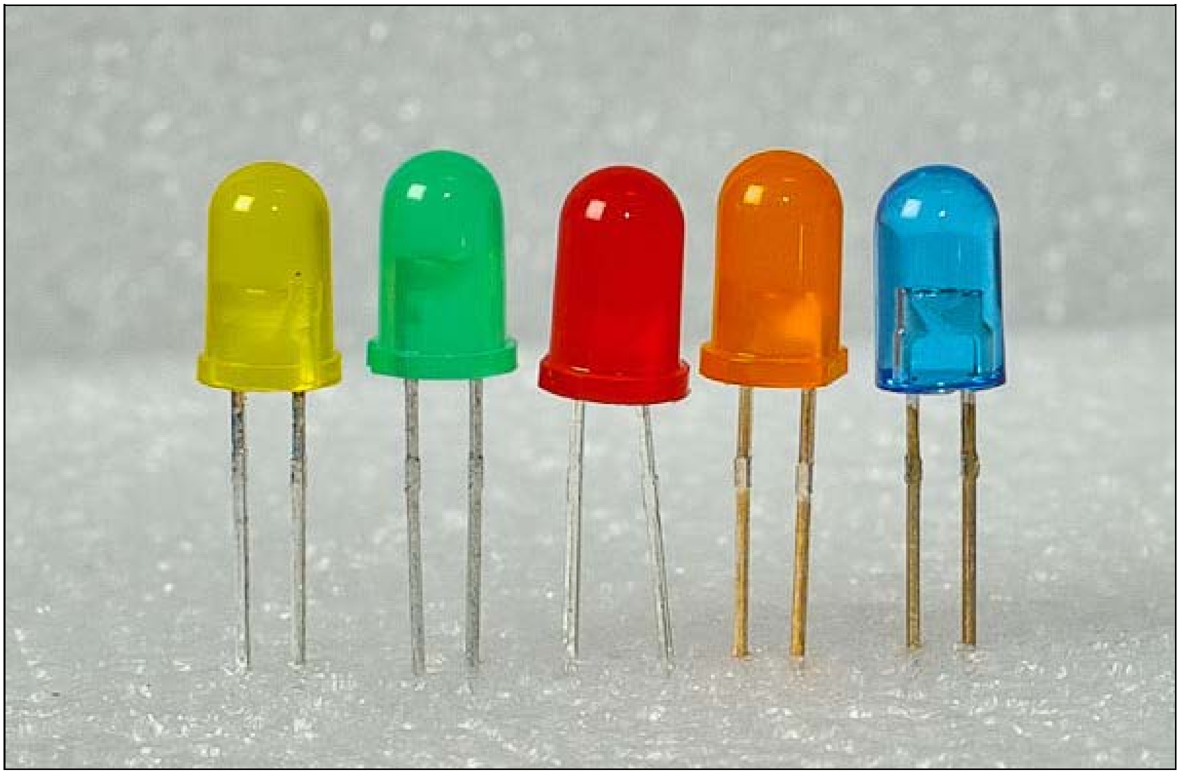
"Until you build and measure it, you don't know what you don't know";
Rick Campbell, KK7B; VHF Open Sources — Design of Low Power High-Stability Low Phase Noise Single Frequency VHF Sources with High Spectral Purity; 2008

Information Regarding the Compression of Schematics

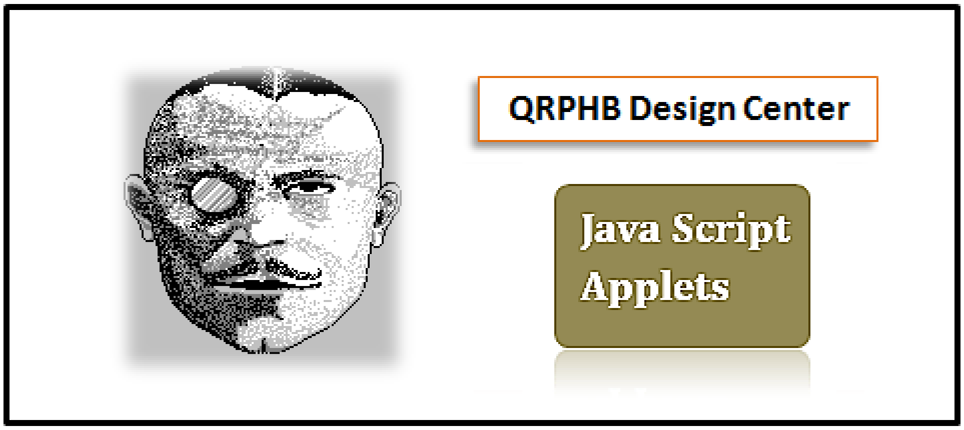
I see many electronics web authors compressing black and white schematics as jpg files. This results in distortion of the schematic. Schematics are best compressed using the 8-bit, lossless LZW algorithm which means converting the file to a png, gif, or even pdf format. The files sizes will typically be smaller than .jpg compression, have no distortion and can be edited easily.

- RAC is the National Amateur Radio Society of Canada
- For my web page concerning support of the **Radio Amateurs of Canada**, please click here

---



The hand drawn image bitmaps on this web site (logo etc.) are by Rod Adams. All website photographs were taken by VE7BPO except as indicated.

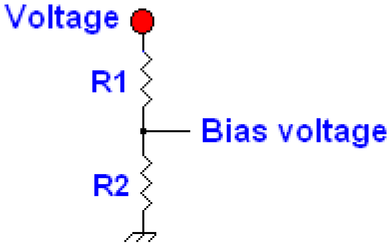


1. Click for the old QRP HomeBuilder Graphics page. Click for my Pin Outs page. Click for my Homepage

---

2. Some simple tools written in JavaScript for the QRP/SWL HomeBuilder:
`Minimal input error checking`

A. Calculate DC Voltage Divider Bias

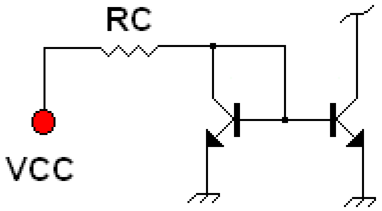


Enter Voltage: Enter R1: Enter R2: Calculate Bias voltage =

B. Calculate Inductive Reactance

Enter Inductance in uH: Enter Freq in MHz: Calculate XL (ohms) =

C. Calculate DC Current for a Current Mirror



Enter VCC: Enter RC in ohms Calculate Current (mA) =

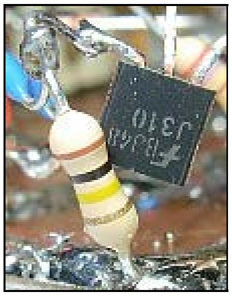
D. Calculate # of Turns To Obtain a Desired Inductance on a Ferrite Torroid

Enter Inductance in millihenries: Select Core: FT37-43 Calculate Turns =
The *AL for this ferrite core is* =

E. Calculate Cut off Frequency for an RC Low Pass Filter

Enter resistor value in ohms: Select capacitor values in uF: 0.01 Calculate 3 dB down frequency (Hertz) =

F. Calculate Power in dBm and mW from Peak to Peak Voltage



Enter measured peak-to-peak voltage into a 50 ohm load: Calculate dBm, mW

Application Note: This web site follows the EMRFD standard for dBm power measurement. dBm = the power delivered into a 50 Ohm resistive load which is temporarily substituted at that point in the signal chain.

G. Calculate Lm and Cm For a Crystal using the G3UUR Method

Enter frequency in **MHz** written on crystal (series resonant frequency):

Enter measured frequency in **MHz** with switch open: Enter measured frequency in **MHz** with switch thrown:

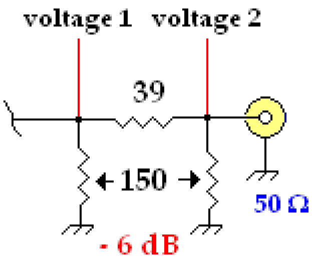
Enter crystal capacitance in **pF**: Enter open switch circuit capacitance in **pF**:

Calculate Xtal Parameters **Cm** = femto Farads , **Lm** = Henries

Supplemental web page for this applet: Crystal filter measurement and adjustment Link

---

H. Calculate Decibel Power Gain or Loss from 2 Peak-peak Voltages



Enter voltage 1: Enter voltage 2 Calculate dB gain or loss =

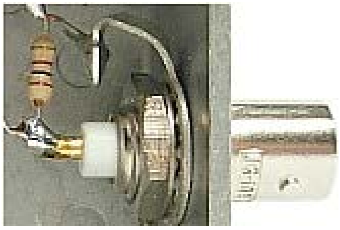
I. Calculate Decibel Power Gain or Loss from Input and Output power

Enter input power in watts: Enter output power in watts Calculate dB gain or loss =

J. Calculate dBm and mW from RMS Voltage (50 ohms)

Enter RMS voltage Calculate dBm mW

K. Calculate Return Loss and VSWR (50 Ohms Detector)



Enter the detector signal in pk-pk volts when the unknown port is terminated in an open circuit: Enter the detector signal in pk-pk volts when the unknown port is terminated in the unknown impedance:

Calculate Return loss = dB, VSWR=1:

Measurements per Figure 7.41 EMRFD. Schematic here

L. Calculate Power from the DC Output of an AD8307 Meter

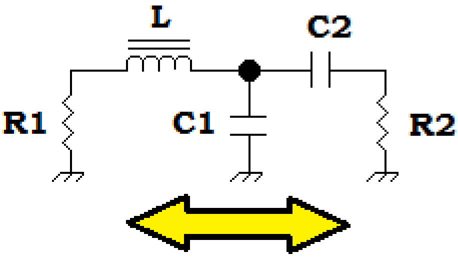
1. Linear calibration steps:

Enter measured DVM voltage at -10 dBm: Enter measured DVM voltage at -20 dBm:

2. Calculate power in dBm from DVM voltage:

Enter measured DVM voltage: Calculate Power =

M. L-C-C Tee Network



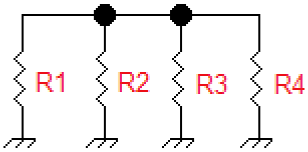
Enter **frequency** in MHz:

Enter **R1** in Ω: Enter **R2** in Ω: *R1 must be < R2, but the network is bi-directional*

Enter Q: *Perhaps start with 2-5*

Calculate L-C-C Values C1 = pF, C2 = pF, L = uH

N. Parallel Resistor Values (2-4 resistors)

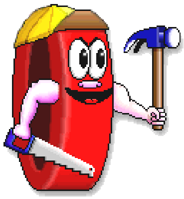


Enter R1: Enter R2 Enter R3: Enter R4:

Calculate R Parallel Reset R = ohms

O. LCR -- a Reactance Calculator In beta -- do not use !

Mode: L C to Reactance Inductance: uH Enter frequency in MHz Calculate LCR



This page last updated: September 21, 2013

RF — Test and Measurement

Short Wave Listening коротковолновое радио

Introduction to Short and Medium Wave Radio Listening

Short wave radio listening was a childhood passion and I enjoy being an SWL just as much today and log at least 800 hours of SWL per year. There seems to be many web pages devoted to construction of radio equipment for the amateur radio experimenter but relatively few for the shortwave radio devotee. I decided to expand this web site to include projects for the SWL Homebuilder in 2005.

My favorite bands are 49 meters (5.9 - 6.2 MHz) at night-time and 19 meters (15.1-15.8 MHz) during the daylight hours. I also listen to medium wave DX around 1400 - 1600 KHz.

Why Listen to Analog Short Wave Radio?

Is analog short wave radio dead? I think not.

No doubt, short wave radio has passed its prime and is slowly dying, however, it's still fun and/or relevant to some.

World band radio: Almost 1/2 of the world's population lives on $2.00 *USD* or less per day. The Internet (the main alternative to shortwave radio) poses a luxury to many poor people living in lower-income countries — experienced travelers or those who support people in developing countries will understand this statement. In some countries now, ranking in the middle class just means you have a full-time job. In addition, oppressive governmental regimes may limit foreign media and Internet access: LW, MW and SW radio can break through obstacles such as natural or man-made disasters, borders, poverty and censorship.

For SWL hobbyists, analog shortwave radio entertains, informs and best of all, provides opportunities to analyze propagation and experiment with real radio topics including static, solar flares, QRN, antennas, grounding, baluns, coax, and wire. SWLing poses an adventure — it's unpredictable, challenging and increasingly difficult as stations decrease and QRN increases. I've built many antennas and even some noise cancelling circuits just to pull in a few Dx stations. The sport of SWL lies in making DX contacts: a theme shared with Ham radio.

What About Internet Radio?

Radio by definition is the transmission and reception of electromagnetic waves of **radio frequency**; but perhaps blue-tooth or Wi-Fi reception from a hot spot qualifies as *radio* in the modern era? Just as peanuts aren't nuts, Internet server or webcasted radio is not RF broadcasted radio. I think Internet radio is great, but fundamentally a very different medium from that enjoyed by SWL fans

Internet radio involves a radio player decoding a stream of compressed bits fed from a Internet radio station or virtual receiver. In some cases, the material originates from a real radio station that also broadcasts an AM or FM signal. For example, you can tune FM station Rooskie Radio "Русское Радио" in much of Slavik Europe or play them on a computer device anywhere you can get an Internet connection.

For lovers of foreign content, listening to Internet radio makes sense; providing convenience, a good signal when bandwidth is high and 24 hour per day listening on 1 IP address. Internet radio offers a much cheaper way for content providers to beam their news and music services around the globe — we've seen numerous large broadcast radio services such as the BBC World Service reduce or drop analog SW and add Internet radio, satellite and digital SW transmissions for their customers.

The exciting growth of independent and niche Internet radio stations increases personal freedom of choice and provides opportunities for unique interest providers and consumers to find each. Media streaming companies and manufacturers of Internet radio *players* and their worldwide distributors benefit too.

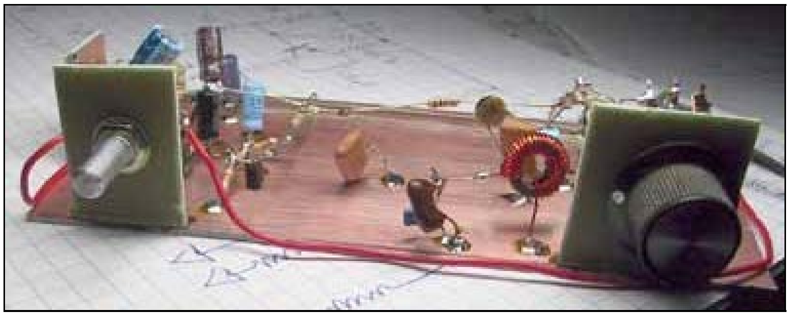
This technology is a far cry from tuning the SW bands with a homebrew or commercial radio frequency *receiver* and a length of wire slung in a tree. Perhaps, the greatest advantages of Internet radio are that you don't have to get up early, or stay up late to pull in some rare Dx, nor do you need any radio skills or special equipment — perfect for the majority of listeners. But we're SWL radio hobbyists: people who listen for both content and because we love radio propagation and gear.

There is nothing wrong with Internet radio, or any of the modern data streaming techniques however, SWL aficionados driven by skill, the thrill of Dx and love of their experimental hobby share a special bond that Internet radio doesn't give them.

Assembling a station The most important component in your radio shack is your antenna. Don't hesitate to safely experiment with the many antenna designs available on the world wide web. Your sure to find a commercial unit or home brew antenna design that suits your real estate and budget.

Your next task is to find a receiver. It is difficult to recommend any one receiver because there are so many excellent commercial receivers to choose from. If you are thinking about purchasing a used receiver, you might consider checking eBay to find a receiver or to learn the going price for used gear. The ultimate SWL experience in my opinion is to build and operate a receiver on at least 1 band.

Favorite SWL and SWL-related Web Sites



Wikipedia-Shortwave Bands A good description of the bands and their general propagation.

Canada's SWL-DXer website 🍁 Hard core Canadian web site dedicated to SWL. Thanks gentlemen!

http://www.bobsamerica.com/swl

http://www.dxing.info/

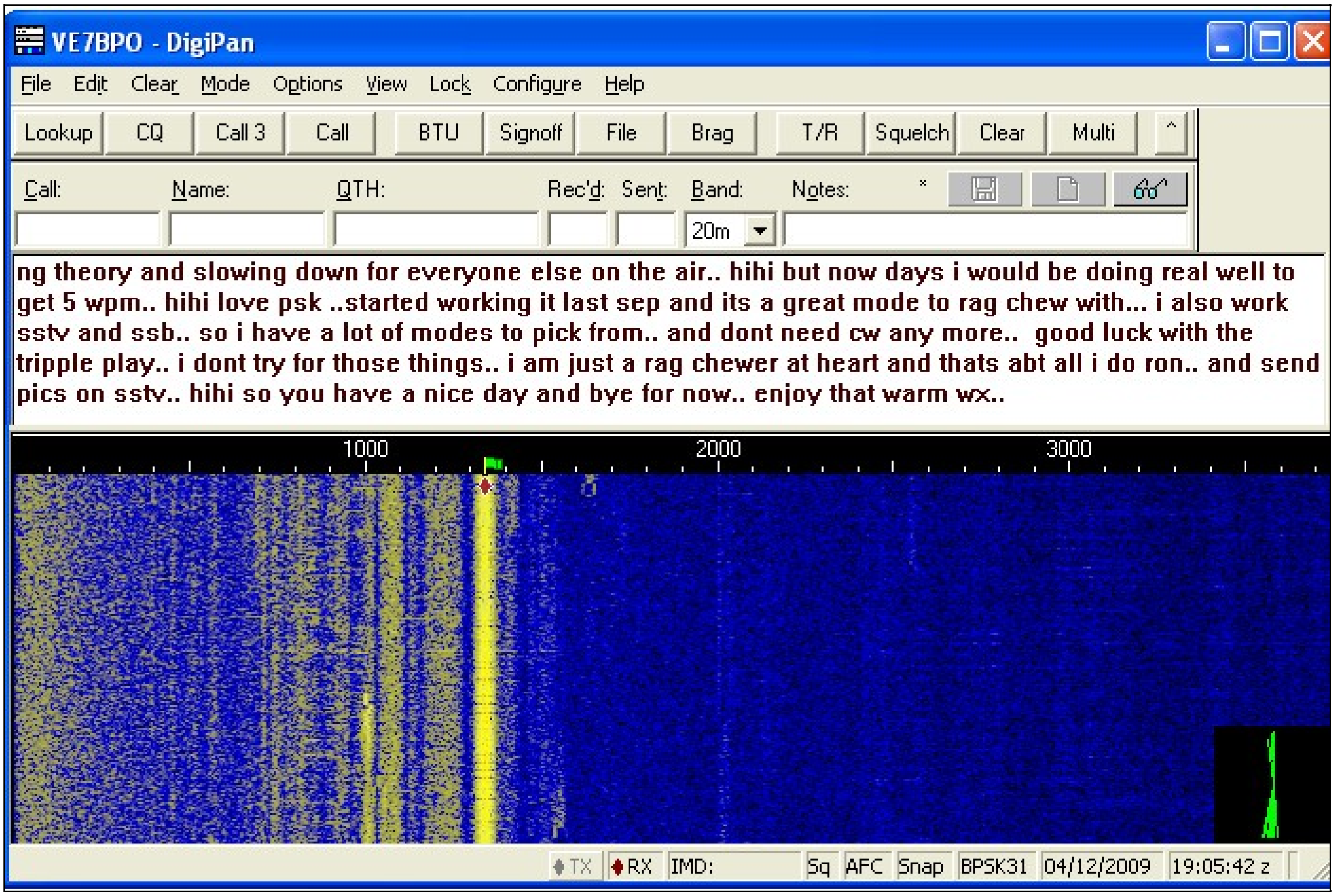
Doug's Shortwave Radio Page

AA6V's SWL Links

Method for soldering a PL-259 to RG-213 or RG-8

Digital Modes For SWL Fans

PSK31



Band: 20m

ng theory and slowing down for everyone else on the air.. hihi but now days i would be doing real well to get 5 wpm.. hihi love psk ..started working it last sep and its a great mode to rag chew with... i also work sstv and ssb.. so i have a lot of modes to pick from.. and dont need cw any more.. good luck with the tripple play.. i dont try for those things.. i am just a rag chewer at heart and thats abt all i do ron.. and send pics on sstv.. hihi so you have a nice day and bye for now.. enjoy that warm wx..

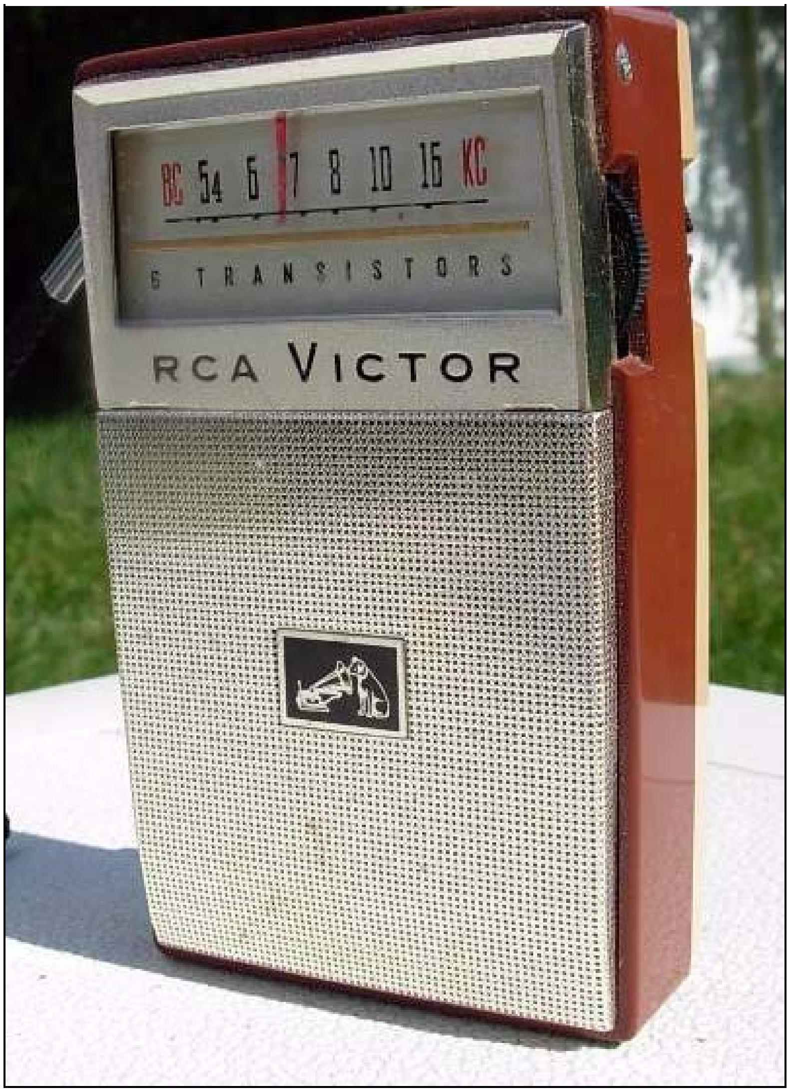
BPSK31 04/12/2009 19:05:42 z

There are a number of good sites about this relatively new HAM mode. All that you minimally need is a receiver dialed in at 14.070 USB (or another

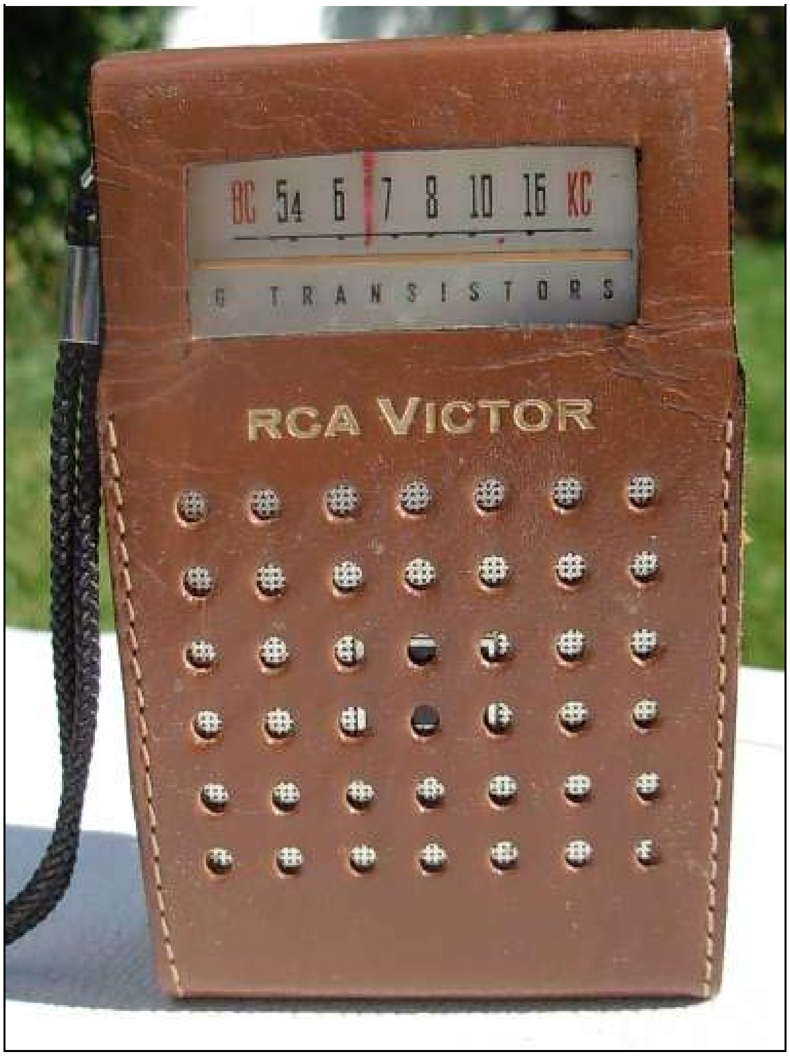
PSK31 frequency), a microphone hooked to your computer sound card and some free software. The software (DigiPan 2.0) is available at http://www.digipan.net/ .

I use a USB interfaced microphone and place it about 2 cm from my receiver speaker. If HAMs are operating; you should hear some warbles and see some waterfalls on your screen. Click on one of the waterfalls to begin receiving the text. I knew nothing about this mode, but was up and running in 15 minutes.

Additional Short Wave and Medium Wave Receiver Photos


BC 54 6 7 8 10 16 KC
6 TRANSISTORS
RCA VICTOR

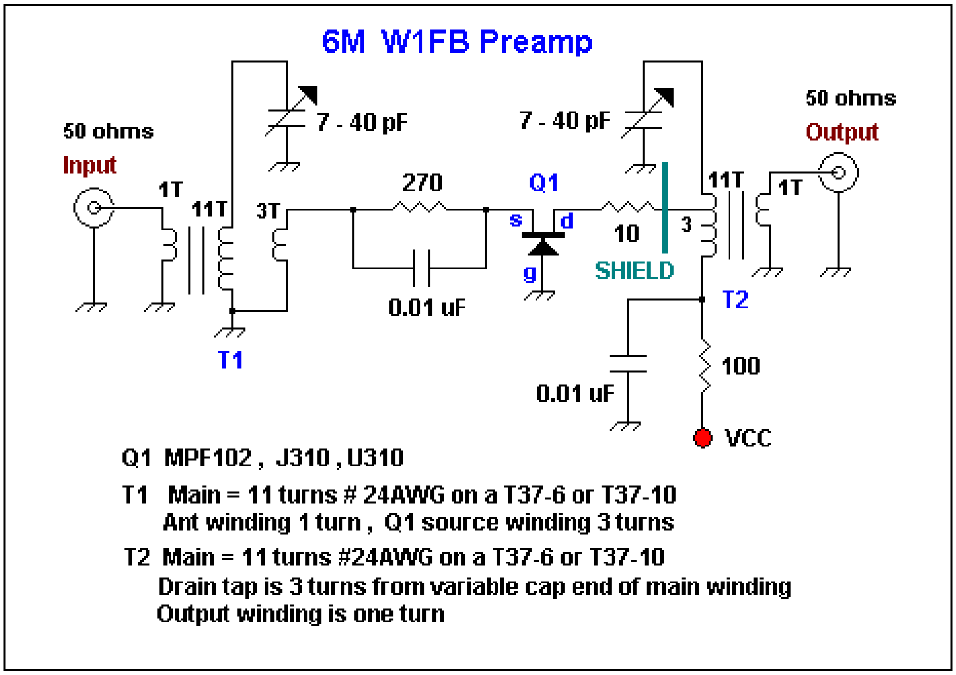

RCA VICTOR
SISTOR
DIO
MODEL GP-526
RCA VICTOR
RCA VICTOR


BC 54 6 7 8 10 16 KC
6 TRANSISTORS
RCA VICTOR

RF Preamps

W1FB 6M RF Preamp

Discussion:



Q1 MPF102 , J310 , U310

T1 Main = 11 turns # 24AWG on a T37-6 or T37-10
Ant winding 1 turn , Q1 source winding 3 turns

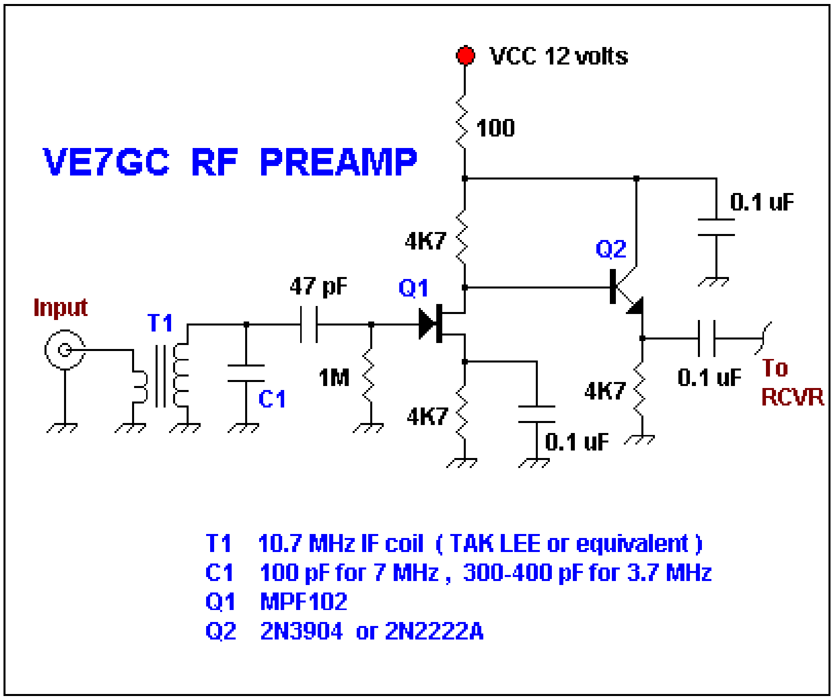
T2 Main = 11 turns #24AWG on a T37-6 or T37-10
Drain tap is 3 turns from variable cap end of main winding
Output winding is one turn

Here is a schematic sent to me by W1FB many years ago. It is very similar to a 6M two-stage preamp that he published in QST in the mid eighties. Doug really favored the grounded gate FET for narrow band preamps. His published work is replete with examples of them on just about every band. I built that amp and remember getting about 10 dB gain, which is all that I wanted for the 6M direct conversion receiver using a diode ring detector that I was building. The great feature of the amp is that it combines a band pass filter and preamp in one. I lost the original schematic that Doug sent me but was delighted to see that I made a bitmapped drawing of it on a floppy disk that was recently re-discovered when we were moving an old desk. The shield shown in the schematic was a small piece of grounded ,double sided PC board in which, I made a small chamfered hole in to pass the lead going to the T2 tap. The shield, along with very short component leads will help minimize parasitic oscillations. The T2 tap is 3 turns down from the end of the T2 main winding that connects to the variable capacitor. Doug specified T37-10 cores for the inductors, but I substituted T37-6 cores and used the same number of windings as specified for the T37-10 core inductors. It worked fine.

VE7GC Popcorn RF Preamp

Discussion:

Here is an easy RF preamp by Dick Pattinson, VE7GC. It uses a single tuned circuit at the front end and can connect directly to a mixer or product detector in a simple receiver project. Note how Dick provided adjustable RF gain control for this circuit in his Wee Willy project on this website. If you can not find Tak Lee green 10.7 MHz IF coils, probably any other brand of 10.7 MHz slug tuned IF transformer would work. The Mouser catalog number is 421F123 . If your 10.7 MHz IF coil has a built in capacitor at the base , remove it. A fixed inductor may also be wound using a powdered iron torroid core and then all or a portion of the C1 capacity would be made variable. The input impedance is 50 ohms and the output impedance is low due to the Q2 emitter follower stage.



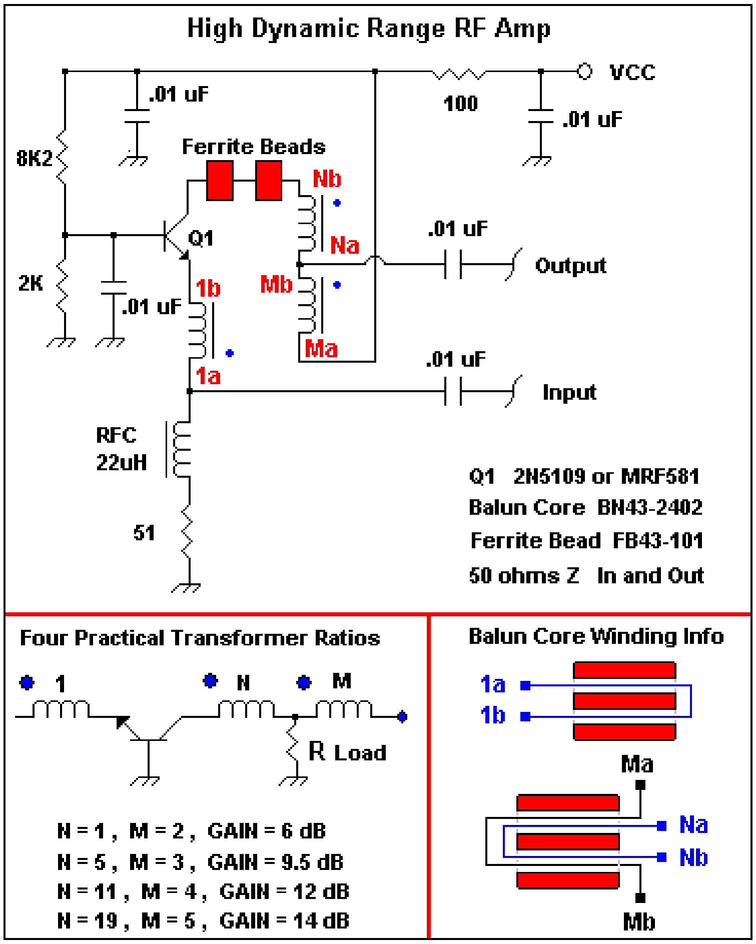
A Low Noise, High Dynamic Range Broadband RF Amp

Discussion:

This schematic is a version of a circuit developed and patented by David Norton and Allen Podell in June 1974. This variation was described by Joe Reisert, W1JR in the now defunct Ham Radio Magazine. The Norton design uses transformer coupling to achieve "noiseless negative feedback" and is really outstanding.

A great article utilizing and augmenting on this technique receivers is by Jacob Makhinson, N6NWP in QST magazine for Feb 1993 with "A High Dynamic Range MF/HF Receiver Front End". Makhinson arranged 2 in push-pull to obtain excellent results. Obtain a back-issue of QST for closer study. **Note** that the fore mentioned Feb QST article has the coil phasing wrong and the correct phasing can be seen at this web site from QST for July 1996. There is also information about Norton feedback RF amplifiers in EMFRD.

If you are building a contest-grade receiver and need a good RF preamp and/or post mixer amplifier, the Norton type is quite suitable. An amp built using a 2N5109 can have a noise figure in the 2.5 - 3dB range. I have also built them with 2N3866, MRF517, MRF581 and a 2N5179 although the last transistor would be a somewhat poorer choice. This schematic with a 2N5109 is good from 1.8 to 150 MHz with a 1.2:1 VSWR or less according to Joe Reisert. I have even put one in a friends CB radio and he was delighted.



Winding and Construction Hints

Making the Norton amps requires some planning to keep all component leads as short as possible. The transistor leads and any connecting components should be trimmed as short as practical to promote stability. Sketch the component layout on a piece of paper and modify it until you are satisfied you have designed a good layout. The ferrite beads on the transistor collector aid in stability and should be used to preserve the noise figure by squashing any oscillations should they develop. The 22 uH choke can be the little epoxy coated units that are color coded and look somewhat like resistors. Do not use a choke less than 22 uH.

Before winding, the builder must first decide how much gain is needed from the amp. For an RF preamp, the stage should have gain equal to or greater than the passive stages after it. Also there will be losses in the transformer, so the theoretical gain of the Norton amp maybe 1 dB off and will need to be factored in. For the purposes of discussion, a 9.5 dB amp is desired , so N = 5 and M = 3. The first step is to mark one side of the core with a dab of liquid paper, paint or a small piece of tape. This will allow you to keep track of the transformer later. To mark, hold the core so that both channels are parallel to the floor, one on top of the other. Apply your dab of paint to the top of the core and use the marked top to denote the A windings. 1a, Ma and Na will all start from the top channel in the balun core. Using 32 AWG wire for all three windings, start with winding 1 and wind the single turn from point 1a to 1b. Cut off the leads so they are shorter than 5 centimeters (2 inches). Next, wind Ma to Mb three complete turns through the binocular core and trim the leads if needed. Tie a small knot in the wire at both ends. This will clearly mark this M winding. Both windings should look like the diagram under the schematic. 1a to 1b are on the left of the balun core and winding Ma to Mb are on the right side of the core. Mb has a distinguishing knot at the tip of both wire ends. Ma starts from the top of the core which you have marked with a dab of paint or something. Finally, wind Na to Nb five complete turns through the core in the same direction as the previous winding M. Strip wires Na and Mb (Mb has the knot), twist together and solder. Scrape the enamel off the leads **very** gently with a sharp hobbyist knife.

Insert the transformer in your circuit and cut the leads to their proper length and then solder away. It maybe preferable to pre-strip the leads on winding 1 as it is hard to strip the enamel off a fine wire that has only one turn and it may accidentally pull out of the core. If it does, just re-insert it into the balun core on the correct side. Once you have soldered Na and Mb you can always identify the windings later because you have marked the top of the balun core which denotes the A windings. Try and make your windings gently tight as if there is too much slack you may have difficulty getting the last few windings thru the core channels. A 14 dB gain amp maybe **impossible** to wind with 32 AWG wire, it may best to use 34 AWG for that amplifier. I have never built one for greater than 12 dB. The transformers are a bit tedious to wind, however persevere and the results will be well worth it. For HF, you can substitute 0.1 uF caps for the 0.01 caps shown if you like.

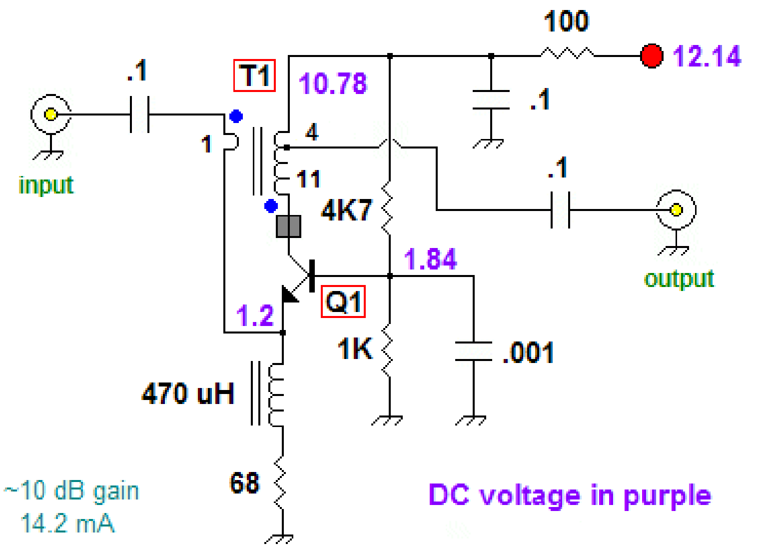
Toroidal Inductor Norton Amp Experiments

Discussion:

The amp shown in the schematic to the right uses a ferrite torroid for the transformer and has ~10 dB gain. Winding 1 turn of wire over the cold end as shown in the schematic is tricky. Try to keep this link as short as possible. A ferrite bead or a 22-51 ohm resistor on the transistor collector is desirable. You can try increasing the turns (1:21:5 etc) to experimentally obtain more gain from this amp. The torroid version is a valid option for builders who do not have balun (binocular) core ferrites in their junk box. Toroidal inductors are certainly easier to wind then binocular core versions.

In 2007, I built several Norton "noiseless feedback" RF amps using FT50-43 and FT37-43 ferrite torroids. These are outstanding and I recommend using them in projects. The input and output Z is 50 ohms. The overall BJT topology is reminiscent of a common base amp. I have some basic information concerning this amp on this web page . They are straight forward to build. The biggest problem is the phasing of the single turn link. Get it wrong and your amp can turn into an oscillator.

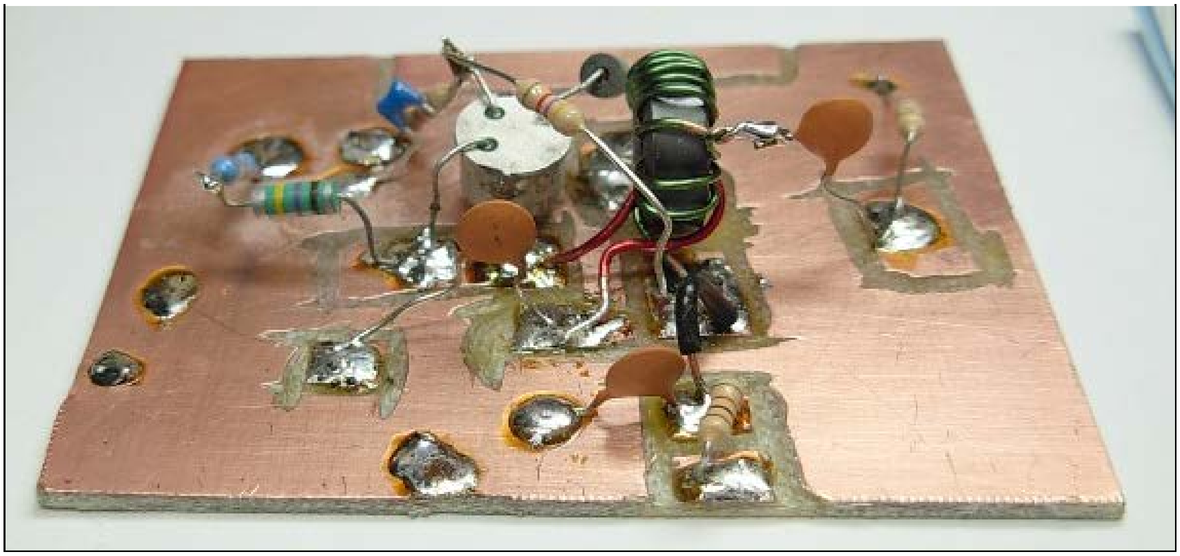
Figure 1



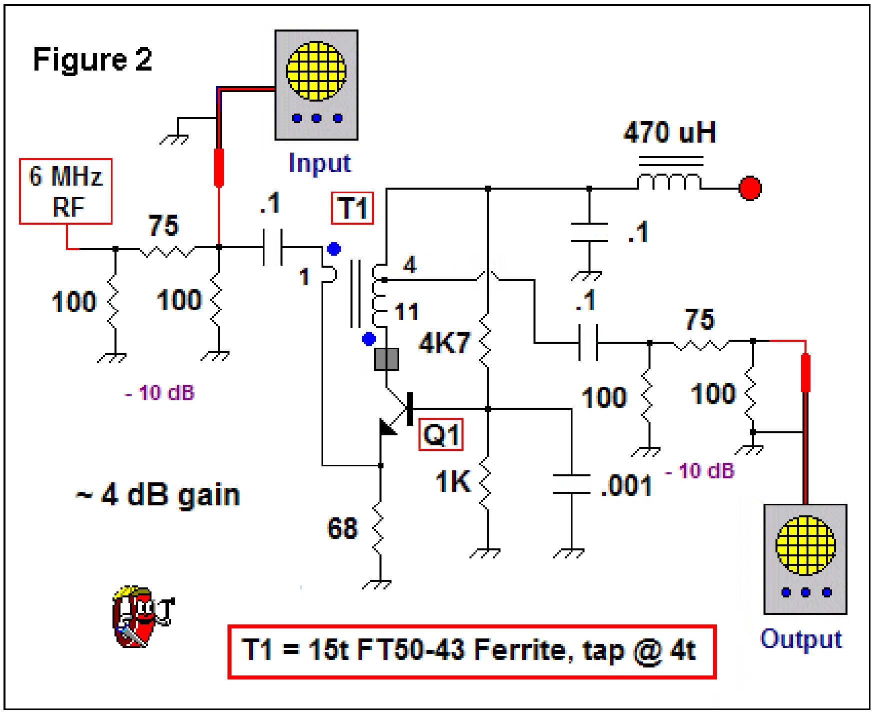
~10 dB gain
14.2 mA

DC voltage in purple

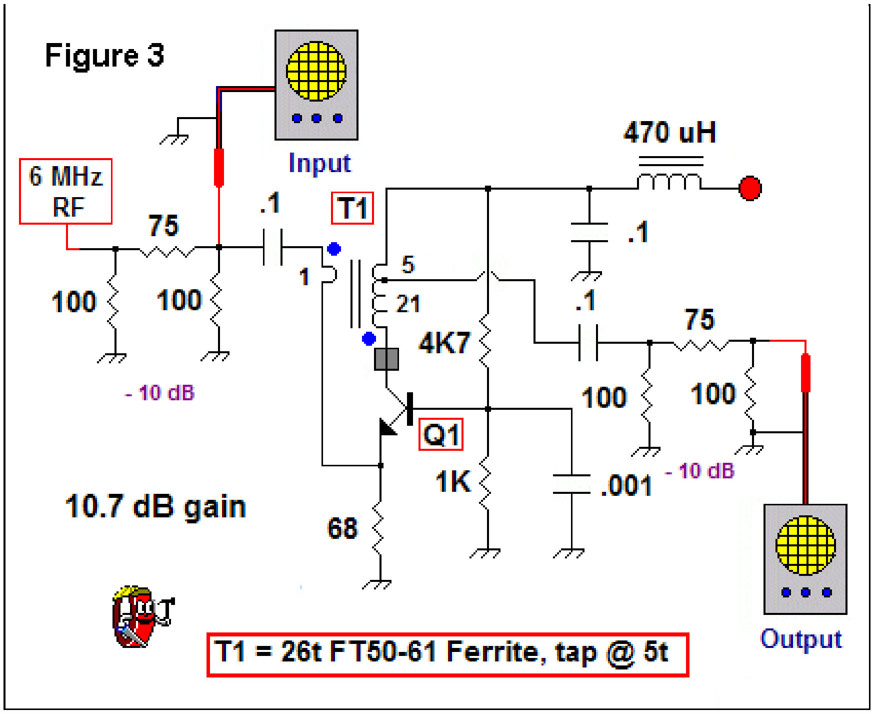
Q1 = 2N5109 or NTE 123 etc
T1 = Secondary: 15 turns with a tap a 4 turns from the cold end.
Primary: 1 link over cold end
T1: use FT50-43 or FT37-43 ferrite



Shown above. The breadboard of Figure 1.



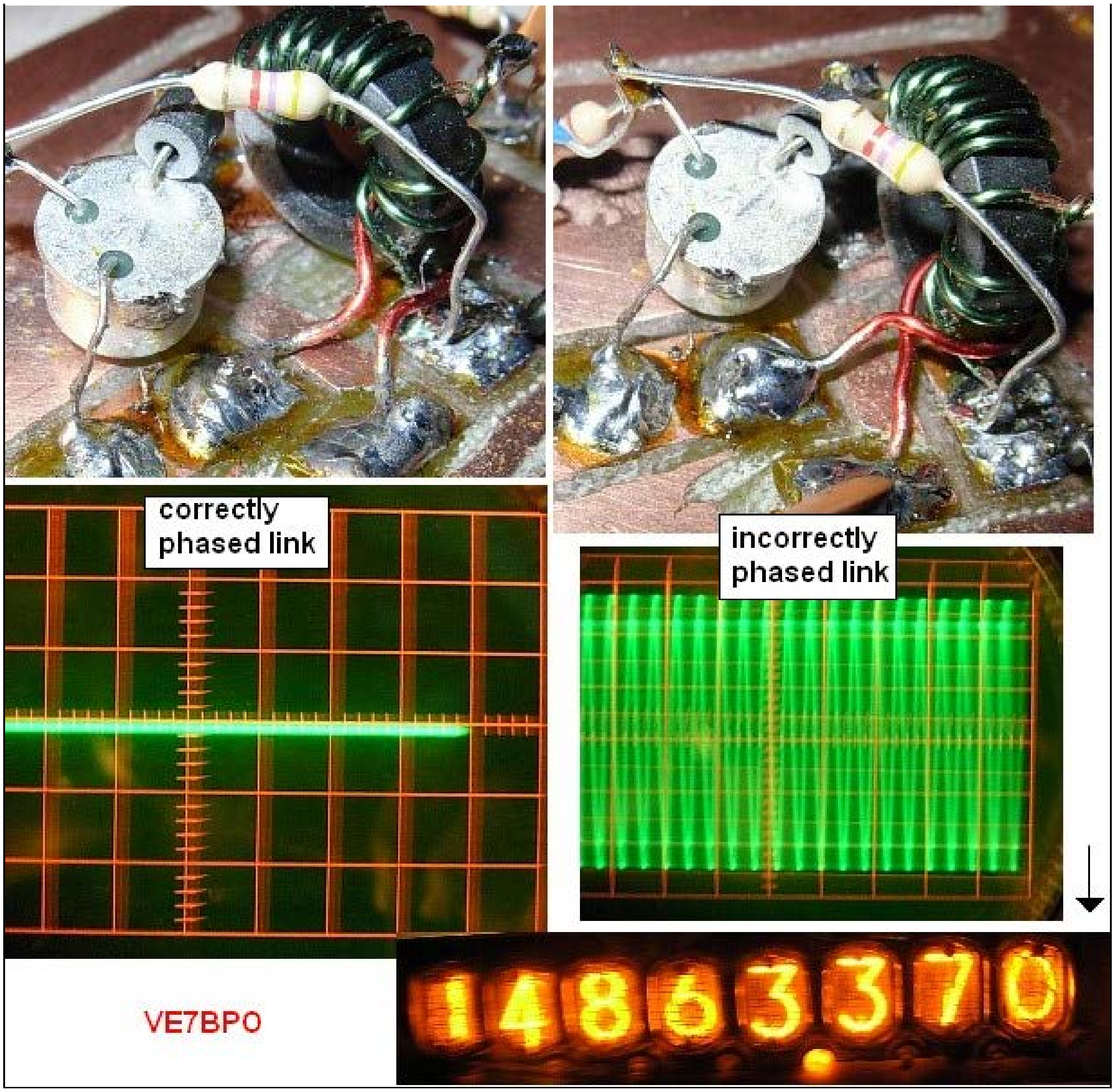
Shown above is the Figure 1 amp above (labeled Figure 2) with a 50 ohm -10 dB pad on the input and output, so gain is low. I used these pads to evaluate the amp in a number of experiments. I never got around to writing up these experiments on the web site and likely never will. I wish I had more time as my notebooks are full of unpublished experiments that would be great content for this web site.



10.7 dB gain

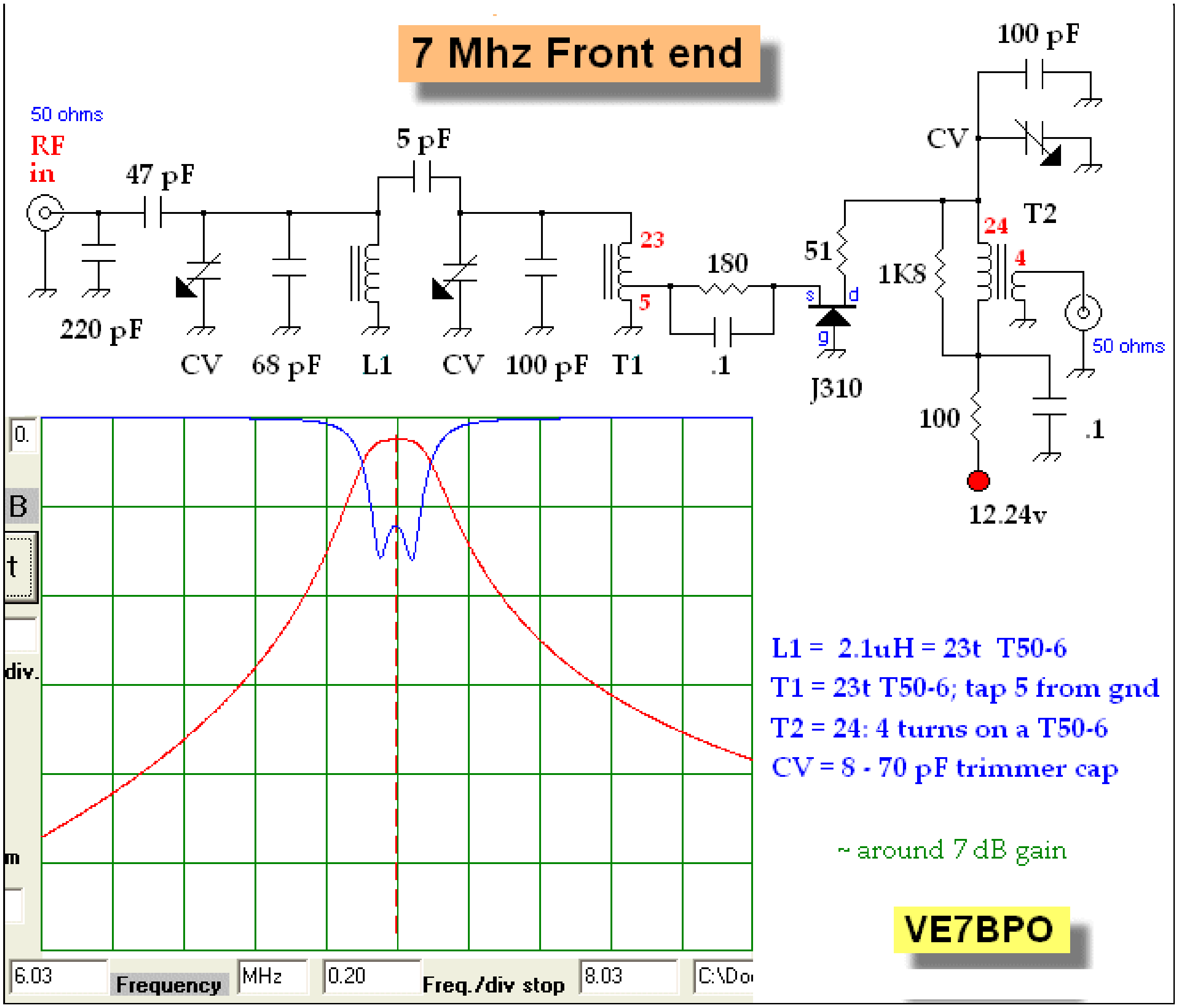
T1 = 26t FT50-61 Ferrite, tap @ 5t

The amp above (labeled Figure 3) is a hot one; 10.7 dB gain even with 10 dB of attenuation. You can leave off the input pad and decrease the output pad to -6dB if you want or require a wide band, low noise RF amp with lots of guts. Most builders use binocular ferrite cores for the inductor, but torroids work fine for many applications.



Shown above is a photograph of 1 of the experiments from 2007. The one turn link from the Norton amplifiers just above is shown phased correctly and then phased incorrectly. Note the oscillation in the "badly wired" amp at 14.86 MHz. I routinely check all of my noiseless RF amps using the oscilloscope. Occasionally, I will put a shunt coil and cap (from input to ground) on the input to "exaggerate" any oscillations. This has proven to be a useful technique for testing if the phasing of the one turn link was done correctly.

RF preamp for the 40 Meter band with 3 tuned filters



L1 = 2.1uH = 23t T50-6

T1 = 23t T50-6; tap 5 from gnd

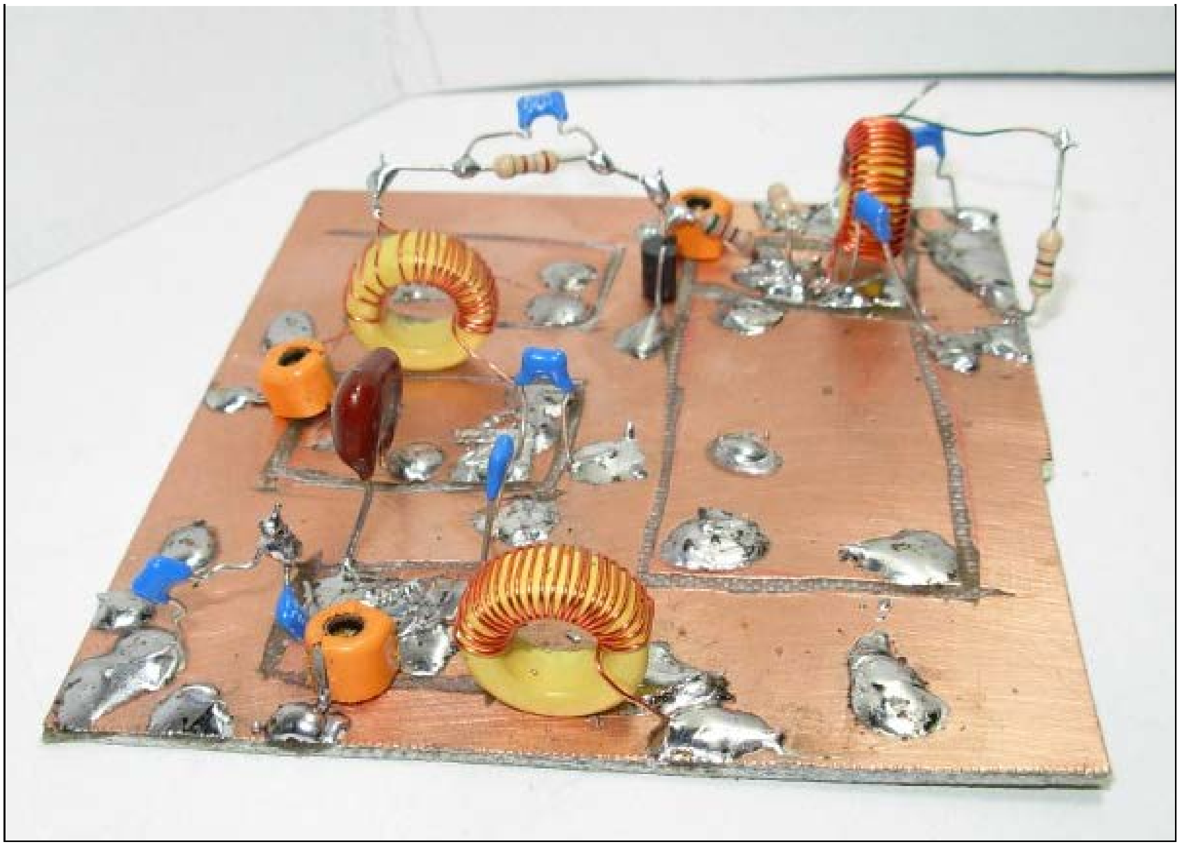
T2 = 24: 4 turns on a T50-6

CV = 8 - 70 pF trimmer cap

~ around 7 dB gain

VE7BPO

An experimenter's 40 Meter band front end for CW. This has a double-tuned filter and a low gain, lower noise RF amp. Great circuit for isolation of a product detector or mixer in a popcorn receiver.



A photograph of the above 40 meter band front end, double-tuned filter plus tuned common gate RF amplifier. Input and output Z is 50 ohms.

QRP SWL HOMEBUILDER

Amateur and Short Wave Radio Electronics Experimenter's Web Site

QRP HomeBuilder Software

Disclaimer:

THE QRP HOMEBUILDER SOFTWARE OFFERED HEREIN ("THE SOFTWARE") DOES NOT COME WITH ANY WARRANTY, EXPRESS OR IMPLIED. IF YOU MAKE USE OF THE SOFTWARE, PLEASE BE AWARE THAT YOU DO SO AT YOUR OWN RISK. NEITHER THE AUTHORS OF THE SOFTWARE AT JENNA DESIGN NOR ANY OTHER PARTY WILL ACCEPT RESPONSIBILITY FOR ANY OCURRING OR UNFORESEEN CONSEQUENCES OR DAMAGES THAT ARISE AS A RESULT OF THE USE OR MISUSE OF THE SOFTWARE.

Technical Info and Distribution

The QRP HomeBuilder applications are written in C++ for speed and compactness. Apps will specified as GUI (graphical user interface) or 32-bit console based (DOS look). Anyone may display or distribute these applications via website or diskette providing that they do not charge for the program(s). I know longer have a C++ compiler and no future work on these applications is anticipated.

QRP - RELATED APPLICATIONS FOR DOWNLOADING

CoilBuilder_99

CoilBuilder_99 is a powdered iron inductor winding application. Enter desired inductance, select core size and mix and press the Calculate button to determine the correct number of windings for your inductor. Data is also given showing, core color, permeability, frequency range, AL value and maximum number of turns versus wire guage for the chosen core size. Encompasses 12 different core sizes and 8 different mixes of powdered iron. Calculated results can be stored on a disk file or printed out.

Style: GUI, File size: 90K, zipped, 44K.

Bug Fixes: Some missing AL values for # 7 material added April 24/99. K6WHP's superior version is linked below.

Current Version is: 4 / 24 / 1999

Download the CB99.zip file

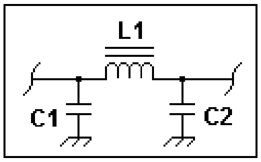
PI Filter Designer

PI Filter Designer is a simple 3 element 50 ohm input and output impedance pi filter designing application. This program allows the user to design simple lowpass filters by selecting from a variety of standard capacitor values either empirically or to suit what you have on hand. The filter 3 dB cutoff frequency and required L1 inductance are automatically calculated and displayed. In addition, the user may select an additional capacitor value to put in parallel with both caps C1 and C2. In this app XL = XC = 50 ohms impedance. No other impedances can be calculated with this program.

Style: GUI, File size: 47K, zipped, 22K.

Current Version is: 1 / 14 / 1999

Download the pifilter.zip file



CapCoder

CapCoder gives the capacitance in microfarads, nanofarads and picofarads and tolerance of any capacitor code entered into its input section. Example : 104J. This app uses numeric spin-buttons and a combo box so that no typing is required for data entry.

Style: GUI, File size: 48K, zipped, 22K.

Bug Fixes : A nanofarad conversion error was corrected July 2, 1999.

Current Version is: 07 / 02 / 1999

Download the capcoder.zip file

Resistor Coder

Resistor Coder gives the resistance in ohms of any resistor color code entered into its input section. Four or five band resistors can be accommodated by this program. This app uses drop-down combo boxes so that no typing is required for data entry. Results may be saved to a disk file or directly printed.

Style: GUI, File size: 58K, zipped, 27K.

Current Version is: 1 / 16 / 1999

Download the resistor coder.zip file

Ferrite

Ferrite is used to calculate the number of turns required on toroidal ferrite cores to achieve the desired millihenry-value inductance. 15 different ferrite toroids are included in this application. This program will calculate the winding data for an inductance range of 0.001 to 27 millihenries.

Style: Console, File size: 64K, zipped, 31K.

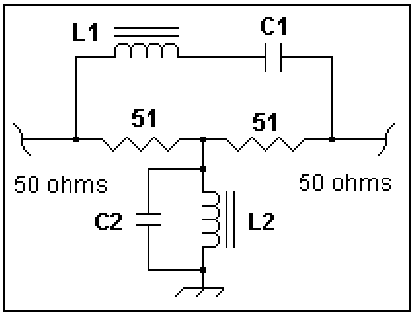
Bug Fixes: Thanks to PA3CKR for the bug report; fixed Jan 19/99.

Current Version is: 1 / 19 / 1999

Download the ferrite.zip file

Universal Diplexer

Universal Diplexer calculates the inductance and capacitance values for a Bridge-Tee diplexer based upon a chosen superhet receiver intermediate frequency. The diplexer is the Joe Reisert, W1JR popularized design discussed under Diplexer Topics on this web site. The user inputs an IF and presses the Calculate button to have the capacitor and inductor values given in pF and uH respectively. The diplexer schematic is included in the application. Note that the this is for the Q = 1 version of the Bridge-Tee Diplexer.



Style: GUI, File size: 49K, zipped, 22K.

Current Version is: 1 / 19 / 1999

Download the diplexer.zip file

HF Dipole

A very basic program for calculating the length of each leg of a 1/2 wave wire dipole antenna. Program good for 1 - 500 MHz, although intended for MF - HF useage. This app does nothing more than the standard 468/freq (MHz) type calculations. It was written for DOS many years ago and ported to Windows. The output shows the 1/2 wavelength and 1/4 wavelength design wire length in feet and meters. This app is probably of no help to experienced antenna designers.

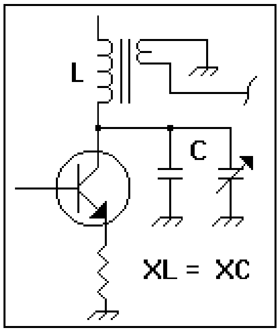
Style: GUI, File size: 46K, zipped, 22K.

Update : Minor improvements made Feb 9, 1999

Current Version is: 2 / 9 / 1999

Download the hf_dipole.zip file

Resonator



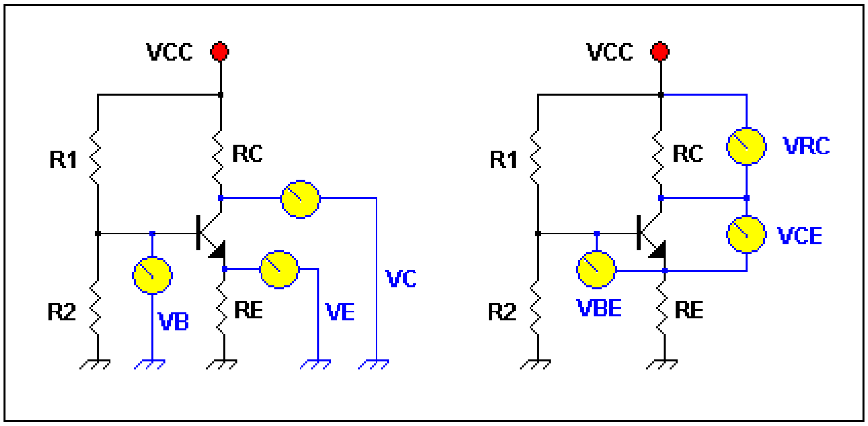
This application calculates the inductor and capacitor values for the tank circuit of a simple bipolar transistor RF amp. The basic schematic is shown above. Enter the center frequency plus the inductive/capacitive reactance you desire and press the Calculate button to calculate the necessary inductance and capacitance for L and C respectively.

Style: GUI, File size: 50K, zipped, 21K.

Current Version is: 1 / 23 / 1999

Download the resonator.zip file

NPN DC-BIAS



This application calculates the various voltages and currents of a simple voltage divider bias NPN bipolar transistor amp. The following is calculated: IB, IC, IE, VE, VB, VC, VCE and detection of Saturation or Cutoff. The user can alter the VCC, VBE, transistor beta and any of four resistor values R1, R2, RC and RE by picking the transistor value from a standard-value resistor table or manually entering the value. The schematic illustrates some of the voltage measuring points on the transistor schematic. *This app is in final BETA.*

Style: GUI, File size: 73K, zipped, 32K.

Current Version is: 16 / 04 / 1999

Download the nbias.zip file

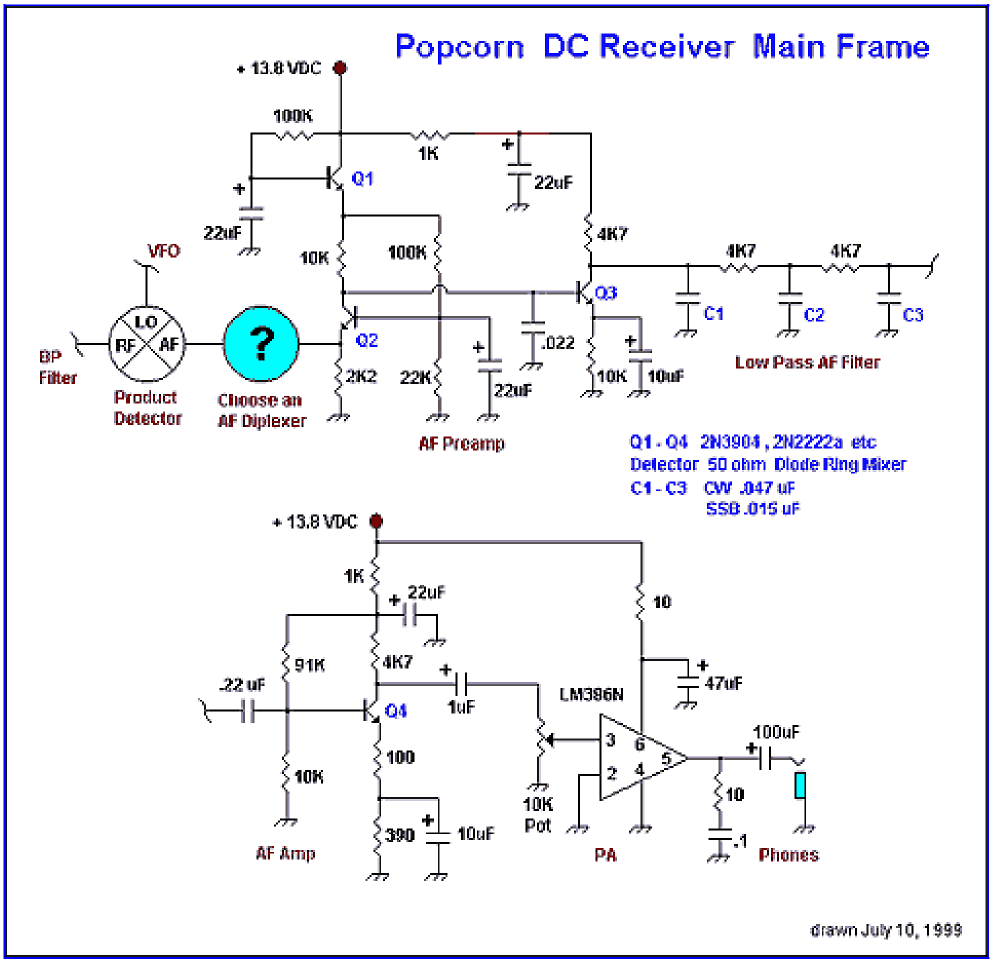
Amateur and Short Wave Radio Electronics Experimenter's Web Site

Popcorn Direct Conversion Main Frame

Discussion:

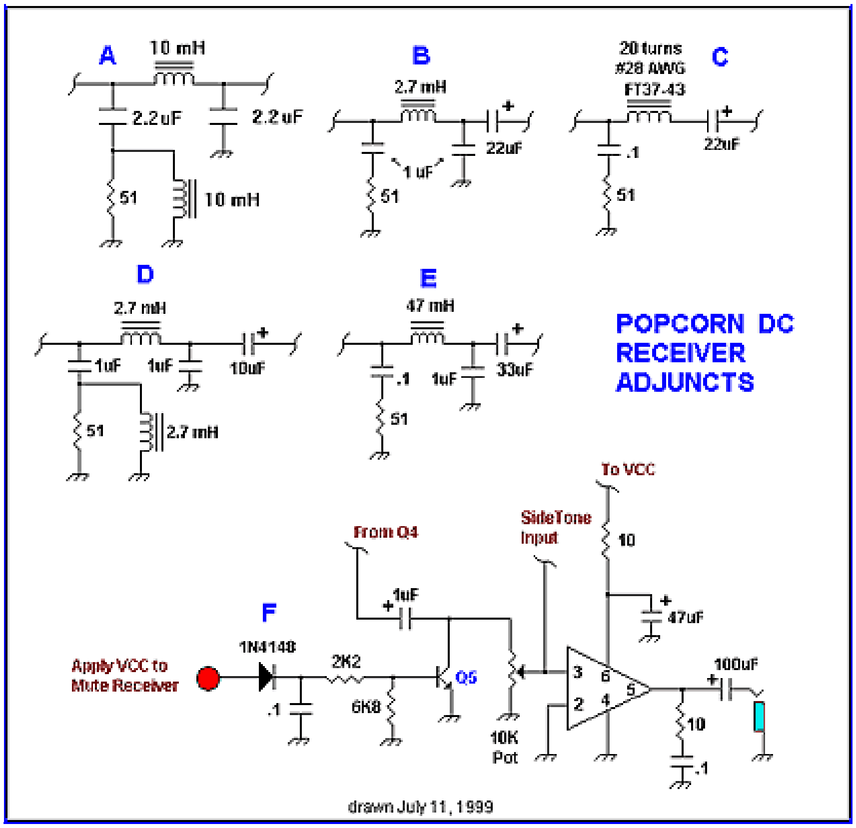
Note: if you click on a schematic, a larger version will appear in a new web browser window.

Shown to the right is the schematic to a low cost popcorn direct conversion receiver main frame. To complete the receiver, a front end band pass filter and a VFO with an output power of 7 dBm is required. This is indeed a frugal project using 4 cheap transistors, an RC low pass filter and an LM386N for output power to a pair of low-impedance headphones. The builder also has a choice of 5 diplexers and an optional mute circuit. This receiver is easily built using Ugly Construction and can be built in 3-4 hours with a bit of luck.



Product Detector and Diplexers

The 50 ohm diode ring product detector can be commercial units such as the Mini-Circuits SBL1 or TUF-1 or homebrewed 50 ohm impedance units. Five simple "diplexers" are shown in the lower "Adjuncts" schematic for you to choose from. The one you choose will depend on available parts, cost and your requirements in a popcorn receiver such as this. These diplexers are mostly of the low pass filter variety and provide a ~50 ohm termination to the diode ring mixer and some matching to preserve the product detector dynamic range. I realize that except for (A) and (D) these audio frequency filters are not truly diplexers and will not provide DC to daylight matching. The intent of this web site is not high performance-high cost



design and please do not confuse it as such. Note that electrolytic capacitors that bypass to ground such as the 1 uF caps must be non-polarized or bipolar for best results.

The (A) diplexer is by W7ZOI and is described on the Diplexer Web Page on this site.

The (B) and (D) diplexers are my designs and the (D) diplexer is the (B) diplexer with out the high pass component.

The (B) diplexer shown has a 3000 hertz 2 pole high pass/2 pole low pass design. This 2nd order filter provides reasonable overall matching Capacitors are standard-value, non-polar electrolytic types.

The (C) diplexer is a very basic, but very practical choice for this receiver.

The (E) diplexer is one that I used in one of my first DC receivers and the 47 millihenry inductor is a standard value unit sold by Mouser Electronics and others.

Another diplexer choice for this receiver might be the unit described by Rick Campbell, KK7B in his Binaural I-Q receiver project published in the March 1999 issue of QST.

Update May 15, 2009

There was confusion regarding the 2.7 to 47 mH inductors mentioned on this web page. I originally wound just the 2.7 mH inductor on a ferrite, but not the others. This is not a great idea as losses are high. For millihenries-value inductors, commercial parts should be purchased. A good brand to consider might be Epcos. Sorry for causing confusion.

10 milliHenry inductor
* Not wound at home

AF Preamps

The AF preamp section follows that of the Ugly Weekender Receiver designed by Wes Hayward, W7ZOI. I tried many other configurations and came to the conclusion that these two simple but elegant stages give a winning combination of low noise, good gain, low parts count, low hum and good AM broadcast band rejection. The Q1 transistor decouples the receiver preamp very well and no hum was detected in the headphones providing a well filtered DC power supply was used. The Q2 grounded base amp provides a low impedance termination of the product detector and diplexer stages. Q2 and Q3 are direct coupled and provide lots of gain to drive the succeeding low pass filter without it adding a huge abundance of noise to the signal. The bypass capacitor (0.022 uF) is essential to bypass any broadcast AM detected in the Q1 stage to ground. Other values of capacitors maybe tried, but do not omit this critical part.

Low Pass Filter

I cannot handle listening to a DC receiver on a crowded band without some low pass filtering. The high pitch heterodynes effect my concentration and give me a headache. Nevertheless, it is neat to temporarily listen to an unfiltered DC receiver; to hear the pure and wonderful signals possible by beating RF directly into audio. I prefer low pass to band pass filters at audio and have used many combinations of active filters using discrete components and op-amps, as well as passive designs using AF inductors to build wave filters. This receiver uses none of these devices, however they could be easily substituted for the filter shown. Connected to Q3 is a simple, cheap RC low pass filter based upon the design criteria given on the Discrete Component RC Audio Filters web page on this web site. The cutoff values you calculate will be ballpark and values of 0.047 uF for CW and 0.015 uF for SSB were chosen, but other values could just as easily been used and please do not hesitate to experiment with the caps and/or the resistors to suit the parts you have on hand. For the capacitors in the low pass filters, avoid using ceramic disk type caps if you want the best possible performance. Polyester, polypropylene, polystyrene or polyester film type are all suitable, however, ceramic caps will work if you are really going junk box/low cost.

I attempted to make a wave file to demonstrate the low pass filter. I came right off the headphone jack into the input of the of my 16-bit PC sound card via a step up audio transformer and the results were a little disappointing. Sixty-cycle hum and distortion of loud stations were added by the sound card. The sound file is big (636 KB) and is a digital recording of me tuning through a 30 meter pile-up using the lowest sample rate and frequency possible on my computer. The low sample rate/frequency also degraded the sound somewhat as well, but I decided to put it on the page, warts and all. The DX station was a VK2 and sure did cause a lot of excitement on 30 meters that night around sunset on the left coast. Actually the wave file demonstrates how good the receiver sensitivity and AM radio immunity is. In addition, the low receiver background noise is also very apparent underneath the constant 60 cycle hum. The 60 cycle hum and clipping of loud CW signals is not heard in the headphones and is a soundcard manifestation. Perhaps the best method would be to come of Q4 and go right into the sound card with a smaller line-in signal voltage. Download the popdc wave file

AF Driver and Final Amp

Connected to the input and output of the Q4 stage are small value capacitors to provide some high pass filtering for the receiver amplifier chain. Some emitter degeneration is used on Q4 to provide a better termination of the preceding RC low pass filter. The receiver amplifier chain has a lot of gain and when the 10K pot is turned to minimal resistance (cranked) , the LM386N can be driven into distortion. You may want to limit the maximum gain with a series resistor connected to the 10K pot after building and testing this receiver.

The final AF amp is the perennial LM386N, a low cost, easy to use AF amp. Turn it upside down and solder pins 2 and 4 right to your copper ground plane to anchor this part. It can easily be configured to drive a small speaker.

An optional mute circuit is shown in the "Adjuncts" schematic and is labeled (F). This circuit is a simple transistor switch which grounds the output from Q4 and mutes the receiver audio. This circuit switches rapidly and there are no annoying pops or clicks to be heard in the headphones when it is switched. Apply the VCC to the diode as shown to mute the receiver during transmit if the receiver is used in conjunction with a transmitter. Q5 in the mute circuit can be a 2N3904 or 2N2222a or substitute.

In addition, a suggested side tone input to the LM386N is shown. I have started to use simple one section RC filters on the output of my side tone oscillators to smooth the waveform into a more pleasing audio tone.

Conclusion:

This popcorn receiver can be made very inexpensively and has good sensitivity and a reasonable noise level and selectivity. I tested this main frame on 30 and 40 meters and really enjoyed it. This receiver main frame could be combined with an inexpensive VFO using tuning diodes to keep cost down and the popcorn factor up. Although it does not use tuning diodes, a 40 Meter band VFO schematic has been placed on the VFO page.

Here is a YouTube Link using the receiver with a different front end filter and VFO. This is not my radio or video.

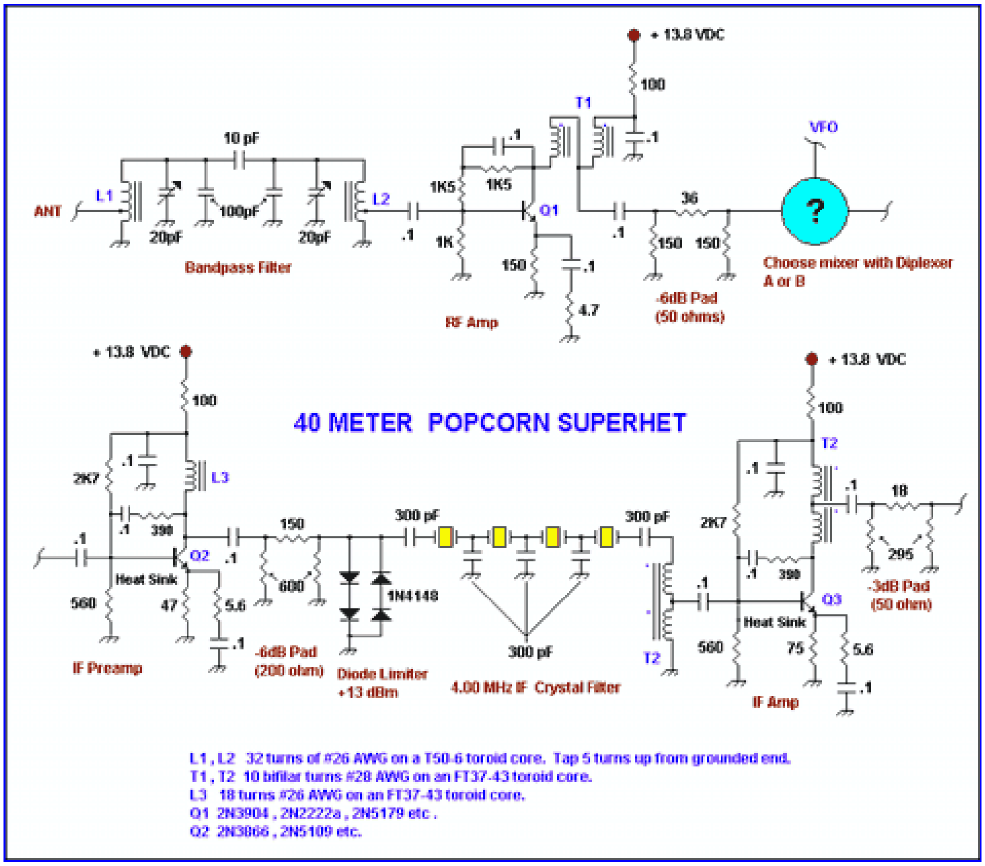
A blog post from Peter AK6L --- it's good to see builders moving beyond kits.

40 Meter Popcorn Superhet Receiver

Discussion:

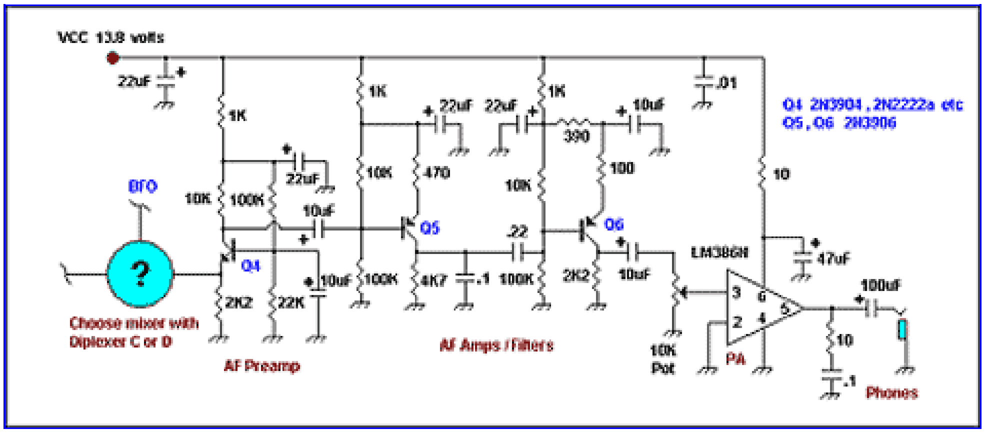
Note: if you click on a schematic, a larger version will appear in a new web browser window.

To the right is the schematic for a no-frills, relatively low-cost CW superhet receiver with a 4.00 MHz Intermediate frequency. There is no AGC or RF gain control, however this receiver has good large signal handling capability. This receiver uses just 6 bipolar transistors and an op amp for reasonable volume into headphones. Much of the ideas/design of the various stages must be credited to Wes Hayward as I borrowed heavily from his previous work and through ideas obtained by discussion. If one were to homebrew the diode ring mixers, indeed this would be a very low cost receiver giving reasonable performance which outperforms any NE602 based superhet receivers that I have built or listened to. Below the main schematic is a diplexer diagram that allows the builder to choose from one of two RF and AF diplexers used to terminate the diode ring mixers.

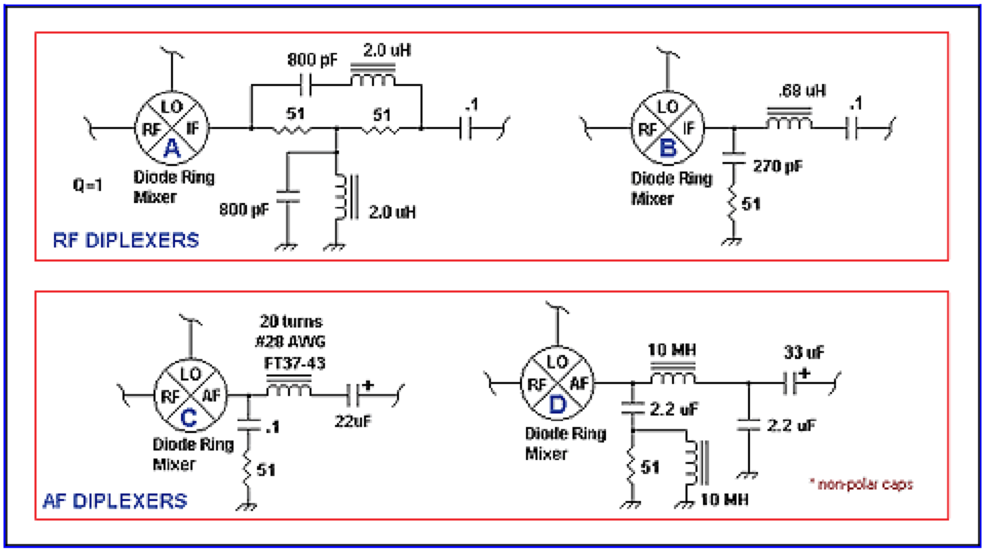


Band pass Filter and RF Preamp

From the 50 ohm receiver antenna jack, first off is a double-tuned band pass filter which was designed by Rick Campbell, KK7B and works very well. The trimmer caps can be the 5 - 20 pF units sold by Digi-key and Mouser. The fixed-value caps in my prototype were inexpensive monolithic ceramic capacitors purchased from Digi-key. Rick used an NP0 ceramic for the 10 pF coupling cap plus silver-mica type for the 100 pF caps in his original design. For possible lower insertion loss, the probable best/cheapest way would be to use all NPO ceramics for the fixed value caps in this filter.



The RF amp is my favorite popcorn RF amp ; a 50 ohm feedback amp. A grounded-gate JFET amp was tried in its place and was also found to be quite suitable and does not require the -6dB pi attenuator that follows the feedback amp as shown in the schematic. The feedback amp's 50 ohm input impedance properly terminates the band pass filter. The -6 dB attenuator pad following the amp to helps provide a 50 ohm input impedance for the mixer and to reduce stage gain which aids in preserving the signal to noise ratio of the receiver. If a builder wants a little more sensitivity, the pi attenuation pad could be reduced to -3 dB however this may effect the receiver dynamic range. The transformer T1 is one of 2 broadband transmission line transformers in this receiver. It transforms the 200 ohm collector impedance to 50 ohms for the succeeding stage.



Mixer and Diplexer

A 50 ohm diode ring mixer (7dBm) such as the Mini Circuits SBL-1 or TUF-1 or homebrew are all suitable. Following the mixer is an RF diplexer of your choice. The more complex Brifge-Tee (Q = 1)diplexer (A) is an excellent design, however maybe overkill in a popcorn superhet such as this. For the (A) diplexer, to get the necessary 800 pF for the capacitors, simply parallel a 470 with a 330 pF or a 120 pF with a 680 pf capacitor. The inductors at 2.0 uH are wound on powdered-iron torroids. You can use # 26 AWG wire and it requires 22 turns on a T37-2 core or 20 turns on a T50-2 core. In addition, you can use a #6 material torroid to wind the inductors. This diplexer is described elsewhere on this web site.

The simpler (B) diplexer uses a ~3 times the IF frequency that I have seen this basic design in many textbooks and articles and provides reasonable matching with a 50 ohm inductive and capacitive reactance. The cutoff frequency chosen was 11.78 MHz as this allows the use of a standard value capacitor (270pF). To wind the 0.68 uH inductor use 13 turns on a T37-2 torroid or 12 turns on a T50-2 powdered iron torroid core. You can easily use 24 - 26 AWG wire for the inductor.

IF Preamp , Crystal Filter and IF Amplifier

Except for the inductors, the IF preamp and IF amp are identical and both warrant a small clip-on heat sink as they draw reasonable current. The standing current maybe increased or reduced by changing the 47 and 75 ohm resistors connected to the Q2 and Q3 emitter respectively. Factors such as available power supply current versus dynamic range requirements may come into play. One may want to stand more current in the IF preamp and less current in the IF amp. For example, the 75 ohm resistor on the Q3 emitter could be increased considerably and/or the 5.6 ohm degeneration resistor could be increased as well if less stage current draw is wanted. The 2N3866 transistor is usually a cheaper way to go for these amps than the 2N5109, but the choice is up to you as you may have something available in your junk box. The 200 ohm -6dB pad following the IF preamp should not be omitted as it helps prevents the stage from seeing reactance's created ahead by the crystal filter. The four diodes form a 13dB limiter to protect the crystal filter should a catastrophically large signal be present in the receiver's front end. They maybe omitted. A -3dB 50 ohm resistive pad terminates the IF amp and helps establish a 50 ohm input impedance for the product detector ahead. **Click here** for more on the IF preamp.

This receiver has a narrow IF Cohn Crystal filter. Bandwidth is ~ 405 hertz, which unfortunately makes tuning quite sharp however this filter is very nice for crowded band conditions. The IF filter crystals should be closely matched in frequency to prevent unwanted ripple in the pass band. Generally, you have to buy 10 and then if you have a frequency counter, use the receiver BFO stage to test your crystals for matching. Pick the closest 4 crystals and use them in your filter. It does not matter if the crystals have series or 20 pF load capacitance, but it does matter that they are matched in frequency within 40 hertz of one another or better for this receiver. For my prototype receiver, I purchased ten 20 pF load capacitance 4 MHz crystals and luckily found 4 that matched each other within 9 hertz! For those builders who do not have a frequency counter, some QRP parts retailers sell matched sets of crystals. It is important to note that the BFO should be set on the high side of the IF frequency as simple crystal ladder filters have a steeper upper passband than lower pass band.

The crystal filter is terminated by the 4:1 transmission line transformer and then 50 ohm impedance of the IF amplifier. The -3dB pad following the IF feedback amplifier helps to terminate the crystal filter by helping ensure a 50 ohm IF amp input impedance and should not be omitted. Place a 75 and a 220 ohm resistor in series to get the required 295 ohm resistance on each leg of the pi attenuator.

Many may balk at just one stage of IF amplification, but since there is no AGC and this is a CW receiver, it works well. A feedback amp is once again used to provide correct input and output impedances for stages connected to the IF amp. Following the IF amp is another attenuator set for -3dB and then a 50 ohm diode ring mixer.

Product Detector, AF Diplexer and Audio Amplifiers

The mixer/detector can be SBL-1 or TUF-1 types or homebrew if you want to reduce costs further as the mixers are the single most expensive components in this receiver.

Again a choice of diplexers is required. The (C) AF diplexer is very simplistic but very practical if you are trying to keep costs low. The (D) diplexer is designed by W7ZOI and is from the Diplexer Web Page on this site.

Following the diplexer, a grounded base audio amp provides a 50 ohm termination to the product detector. AF gain and some AF filtering are provided by Q5 and Q6 which together attenuate frequencies less than 72 hertz and greater than 638 hertz. This amplifier pair are described on the discrete AF filters web page on this web site. Keep your leads short on all the AF transistors.

The final AF amp is the perennial LM386N, a low cost, easy to use AF amp. Turn it upside down and solder pins 2 and 4 right to your copper ground plane to anchor this part. There are a number of low-noise alternatives to the LM386 available which are generally more expensive but would be quite suitable. Discrete component AF amps can also be used, but a popcorn part such as the LM386N maybe cheaper and easier.

VFO and BFO

VFO schematic

BFO schematic

Alternate version of this receiver

Construction Ideas

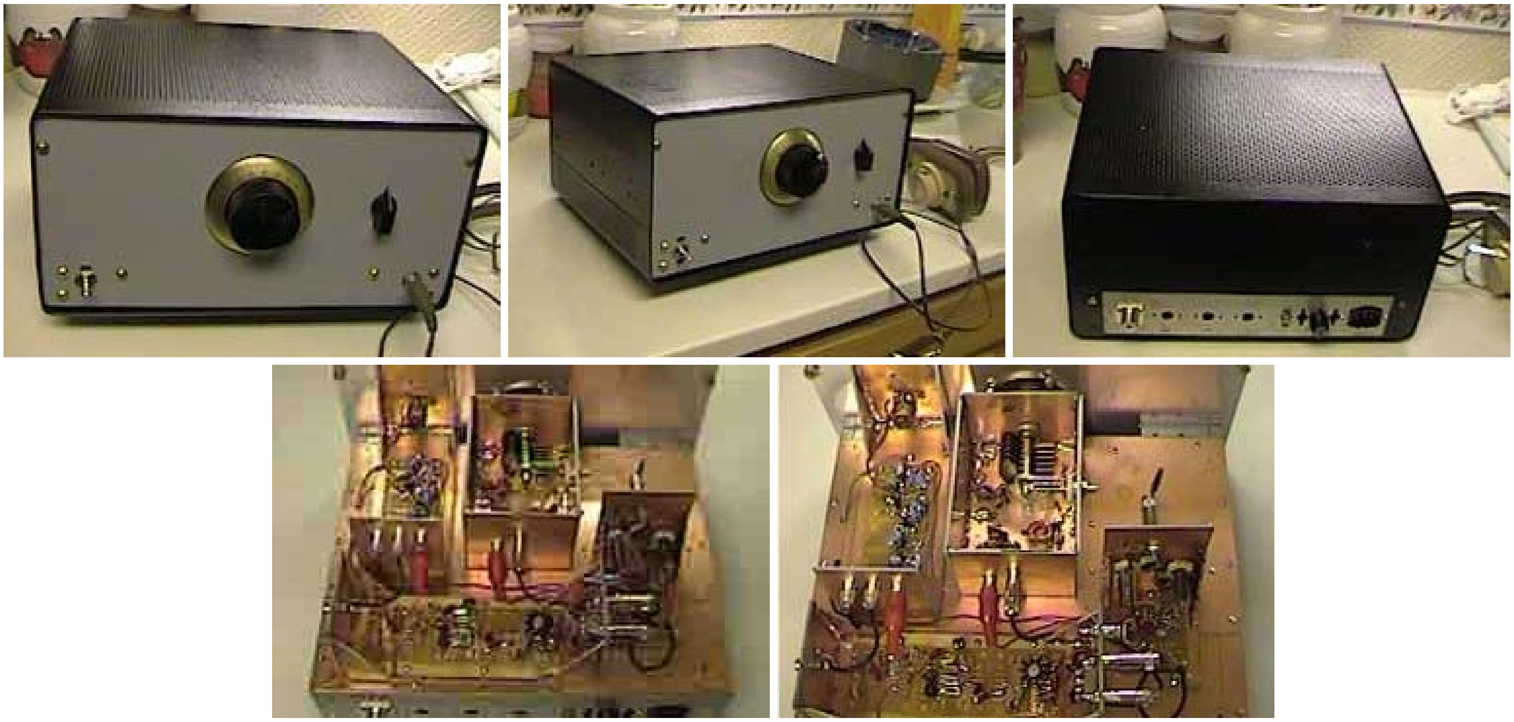
When constructing any project, build in small modules and test each one separately. For instance, the AF amp should be built first and then tested by injecting a very low-level audio frequency tone into that stage and listening for output in your headphones. Every QRP workbench should have a simple AF tone oscillator from a schematic similar to the ones used for keying side tones in CW transmitters. The encased oscillator should have to a 100K or so potentiometer connected to the output to vary the output signal amplitude. Generally use maximum resistance on the 100K pot to start with and reduce this resistance slowly as the in-test amplified oscillator output could be very loud!. After testing the AF amp, build the 3rd AF preamp stage including the 10K panel mounted pot so you can vary the gain going into the AF amp. Now inject the AF oscillator output into the input on the pot and vary the 10K pot to ensure that the stage you built is working. It should be a lot louder now and should go up and down in volume with the 10K pot. Finally build the remaining preamp stage and once again test the circuit with your AF oscillator. The output into the phones should be painfully loud now when cranked up! The next stage to build would be the BFO. If you do not have a scope, peak the tuned circuit by watching the S meter on a radio receiver located nearby. Ensure that you put a load on the output winding of the BFO such as 47 ohm resistor to ground. A small piece of wire can be used as an antenna if the BFO signal is too weak to activate the S meter on your receiver. Once peaked, you can now use the BFO to match your IF crystals. To use the BFO to match your crystals, use a small wire to bypass or disconnect the 60 pF variable capacitor that is used to connect the crystal to ground. In other words, the bottom lead of the crystal is connected to ground with a short piece of wire. This makes testing your crystals a little more scientific as the variable capacitor cannot influence the crystal frequency during testing. You can also use the BFO in conjunction with a scope or voltage probe to test the various RF amps in the receiver. I do this all the time with my scope. Proceed with this build a stage, test a stage method and you should be rewarded with a functional end product.

KK4RF's version of the 40 Meter Band Popcorn Superhet

Marty, KK4RF emailed me and described his version of the popcorn superhet from this web page and contributed some great info and photos of his receiver. Of note is Marty's use of Radio Shack IC boards for mounting the components for each stage other than the VFO circuit. This is yet another variation from ugly construction that I have also used which works extremely well. Marty built the VFO using pure ugly construction and I was glad to hear that he is enjoying good frequency stability even with the lid off the VFO enclosure.

He built the receiver into an old Heathkit HW-12 single-bander case from the 1960's and it is a very attractive receiver to say the least. He found an old National Velvet Vernier Drive at the Virginia Beach Hamfest this year and used it to tune the VFO. Don't you love Hamfests! For the BFO he used an BFO circuit with a 4 mHz crystal from a different receiver project (a project that never quite worked.) He built a small power supply and located it along with the BFO under the chassis.

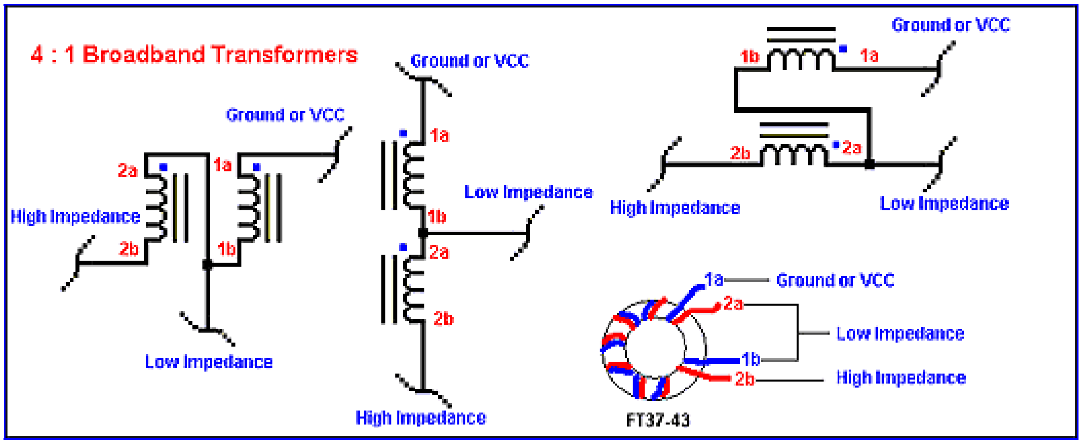
I like Marty's generous use of ground plane and neat stage layout. He reports good selectivity with his IF filter and apparently built four superhets that did not work before building this popcorn version. This is more a testimony to Marty's perservance to home building than to this receiver design in my opinion. I won't tell you how many rig failures I have personally incurred, as it would take a long time! Many thanks to Marty, KK4RF for the feedback and great pictures.



Amateur and Short Wave Radio Electronics Experimenter's Web Site

Broadband Transformers plus Diode Ring Mixers

Discussion



There are 2 basic types of broadband transformers used in most QRP work — conventional and transmission line style. Both types may be wound on ferrite toroids, pot cores or rods, however, I only discuss the toroidal transformers we employ to give a 4:1 impedance transformation.

I use these transmission line transformers on many projects on the QRP / SWL HomeBuilder web site. For MF and HF uses, a ferrite core permeability of 850-900 is generally required and the FT37-43 ferrite core proves suitable. Shown above are 3 equivalent schematics of the 4:1 transmission line transformer. You'll probably find that the center drawing easiest to conceptualize, however, with closer examination, all 3 schematics are the same and transform signals from unbalanced 50 Ω impedance up to 200 Ω unbalanced impedance or visa-versa.

The high impedance is 200 Ω and the low impedance is 50 Ω in all cases. It is important to know that these transformers are symmetrical and the points labeled Ground or VCC can be switched with the point labeled High Impedance. *Click on the schematic to enlarge it.*

Winding the 4:1 Transformers

Wire Twisting

Transmission line transformers are wound with bifilar (2 wires — generally twisted together). Winding these transformers is very easy. All you need are two ~18 cm (7 inch) pieces of #28 AWG enamel coated wire and an FT37-43 ferrite toroidal core.

A shop vise, a ruler, plus a brace and bit hand drill may aid your construction — I bought my brace and bit drill at a garage sale for 2 dollars. You need to twist the 2 pieces of wire together to get ~3-4 twists per cm (8-10 twists per inch) in the wire. To do this, loosely twist the wires at one end and place these twisted ends in a bench vise. Next, place the free wire ends together in your brace and bit drill chuck (no drill bit) and tighten up the chuck so that the wires are held securely.

Try to keep wire length and tension equal. Start turning your drill to twist the wires together and every once and a while measure how many twists per cm with a ruler. When you get to 3-4 twists per cm (8-10 twists per inch) you're done and then may trim the excess leads with a wire cutter in preparation for final winding and soldering.

At VHF, I often use just 3 or 4 total turns on an FT23-43 toroid with a piece of wire just a few cm long. I place the 2 wires in the vise and twist them using a pair of pliers held parallel to the wire.

Transmission line transformers will also work if the wires are untwisted. 3-4 twists per cm only serves as a non-criticial guide. Never wind your simple 4:1 transformers with bare wire.

A brace and bit hand drill plus a vise provides a good way to twist your wires.

Final Wiring and Soldering

Leaving a 2.5 cm (1 inch) lead, wind ten complete loops through the toroidal core leaving a small gap between the start and finish leads.

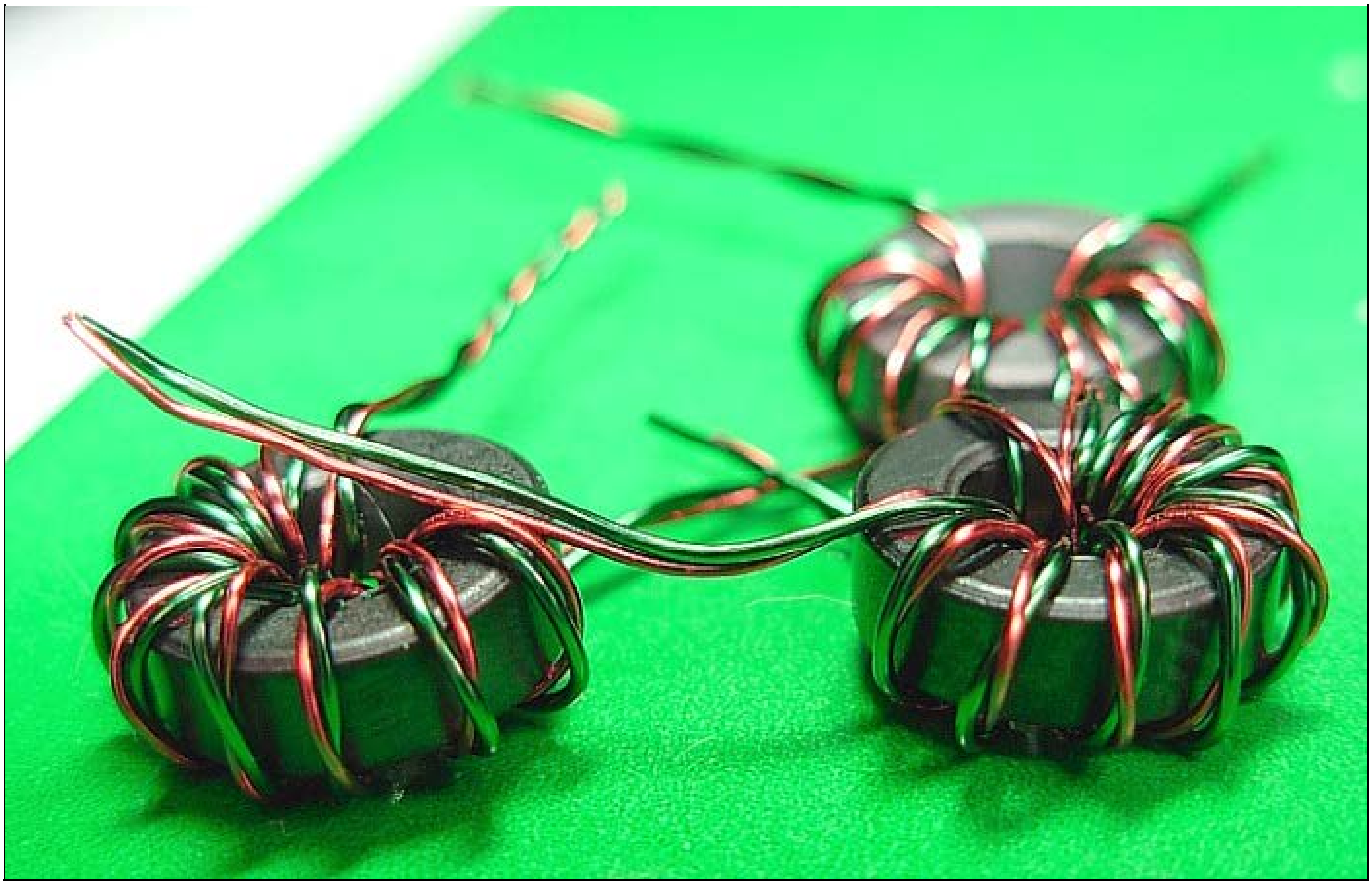
Untwist the leads a little so that you have 4 separate wires. One set of these wires wires will be called winding #1 and the other winding #2. You need to identify them and further break them into 1a, 1b and 2a and 2b. Generally I regard the the top two windings as (A) and the the bottom two wires (B), however, use whatever system works best for you. Strip off the enamel at the tips of all four leads and then get your ohmmeter or better yet, a beeping continuity tester.

Start on one of the top (A) wires by connecting the ohmmeter or continuity beeper to it and then touch one of the bottom wires and then the other bottom wire. Whatever bottom wire (B) shows continuity with your top wire should be marked along with the source top (A) wire with paint, liquid paper, tape, or whatever you like.

I prefer to wind 2 different colored wires if possible.

Designate the marked wire pair winding number 1. You may also want to test for a short circuit — there should be no connection between wire set 1 and wire set 2 at all. So now you have 2 wires sets, winding set 1 is marked and winding set 2 is unmarked. The top two wires are arbitrarily labeled A and the bottom two wires are labeled B . Refer to the schematic above for clarification. Connect 1b to 2a and twist them together and then solder. Your done!

It's really easy to make these things don't you think?.



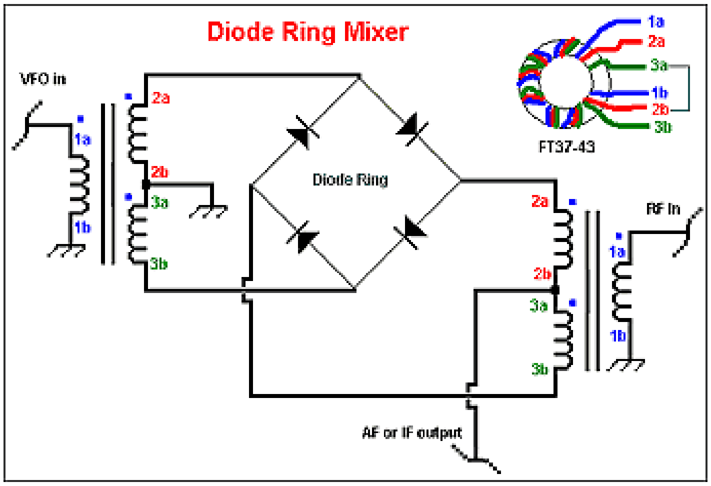
A trio of bifilar transformers wound on FT37-43 ferrite toroids. 2 colors of wire reduces errors and speeds up construction. Consider making up 5 at a time, so you have them on hand and do not have to interrupt your experiments.

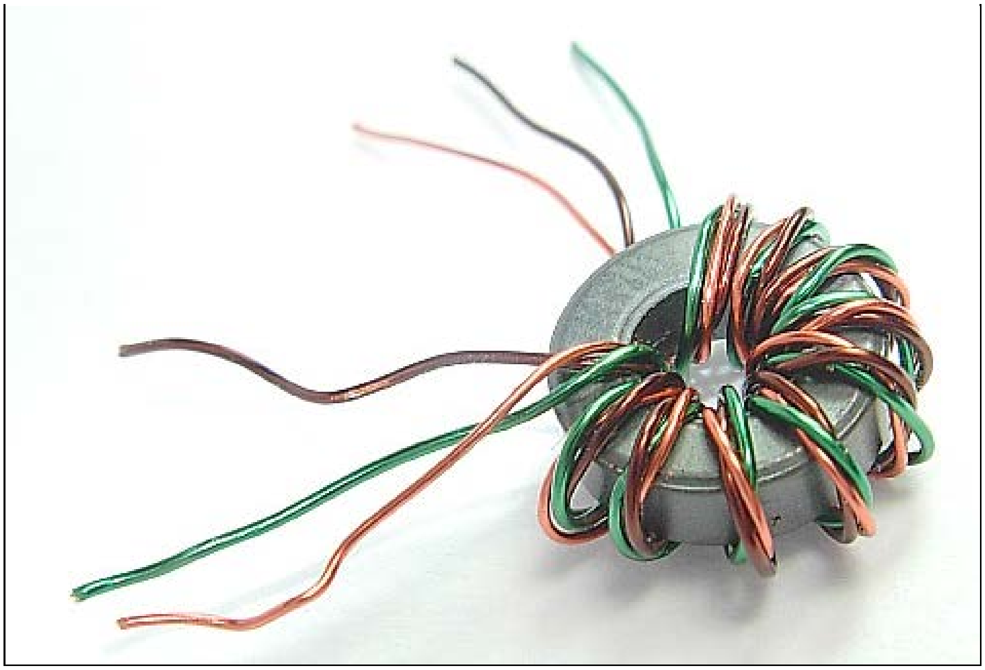
Homebuilding Diode Ring Mixers

Discussion:

Easy to make, homebuilt diode ring mixers give a low-cost alternative to commercial diode ring mixers. A double-balanced diode ring mixer has 2 unbalanced to balanced transformers and a diode ring. The impedances at the three ports is 50 Ω. The transformers are wound with #28 AWG enamel coated wire on a FT37-43 ferrite toroidal core using a trifilar (3 wire) technique.

The wire twisting and winding technique is done as described above for the bifilar transformers. The connections 2b and 3a are twisted together and soldered. Again you will have to develop a technique to help you distinguish the wires from 1 another. *Click on the schematic to enlarge*

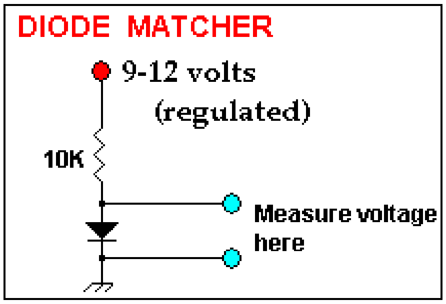




A trifilar transformer wound using 3 colors of wire on an FT37-43 toroid

A sample of the enamel coated copper wire collection I wind inductors and transformers with. In stock are wire gauges from 18 to 34. Like everything else, this collection started small and grew over time. Be vigilant for bargains and when you find a good price, purchase a whole bunch as it does not go bad. The Belden wire (orange spool ends) is over 40 years old and the enamel insulation remains perfect.

Diode Matching for Mixers



Discussion

For optimal results Schottky or Hot-Carrier diodes should be used. However, common diodes such as the 1N914, 1N4148 or 1N4454 are all quite suitable and are much cheaper. The four ring diodes should be matched to help mixer balance and thus carrier suppression. At MF and HF the most critical matching required is the forward voltage drop across the diode and this is easily performed with a sensitive voltmeter.

Set your voltmeter on the 2 volt scale to give you 3 decimal places for matching the voltage drops. Try and find 4 diodes close to one another. In addition, best results maybe obtained if all the diodes are the same type (i.e. all 1N4148) and if they are all from the same manufacturer. Look above for easy schematic to match your diodes with a voltmeter. Give the diode under test at least 20 seconds to warm up and stabilize before taking your voltage measurement.

RF — Test and Measurement

Diplexers Topics

Introduction

My original web page on diplexers was rather incomplete and received some criticism from electronic engineers albeit the focus of this web site is "popcorn" designs. Wes Hayward, W7ZOI provided me some excellent schematics, analysis and simulations for diplexers which terminate doubly balanced mixers and these are presented below. After this section, the W1JR Bridge-Tee RF Diplexer from the original QRP HomeBuilder diplexer web page is presented along with new commentary and simulations by W7ZOI.

The final section presents a practical diplexer for terminating a product detector. All graphical images labeled as Figures 1-24 are copyright and property of W7ZOI and may not be presented elsewhere. Updated September 23, 2000.

W7ZOI Diplexer Notes

The usual amplifier is a two-port circuit. That is, it has an input port consisting of two terminals and an output consisting of two more. One terminal (ground) can be shared between the ports. Many filters are also two-port networks, including most of the ladders networks we use so often. Many other networks have three or even more ports. A common example is a mixer, which has three ports. Another example of a three port network is a diplexer. This linear network is usually designed around two port filters where one end of two different filters are paralleled to form an input port. This is illustrated as Figure 1. The purpose of a diplexer is usually to force a frequency constant impedance to occur at the input port, even though we usually only use one of the two output ports for signals. The simplest form of diplexer uses a pair of 1 element filters, a low pass and a high pass. This is shown in Figure 2.

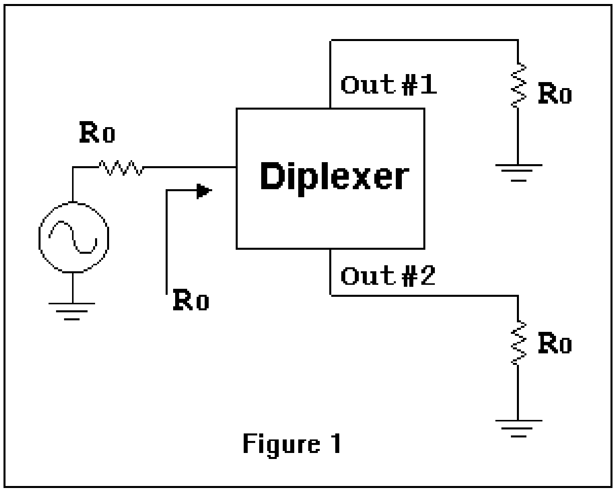
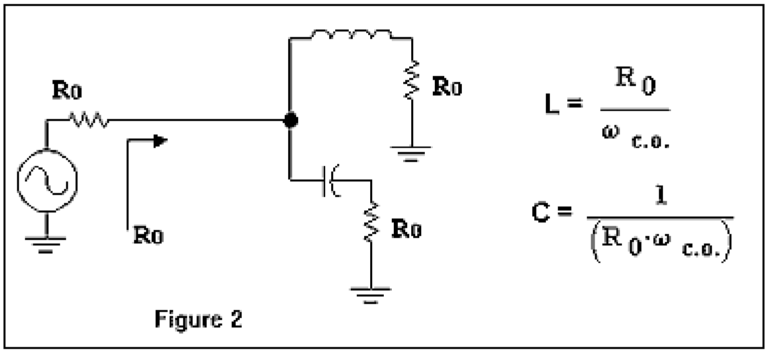


Figure 1



$$L = \frac{R_0}{\omega_{c.o.}}$$

$$C = \frac{1}{(R_0 \cdot \omega_{c.o.})}$$

Figure 2

Where the equations give the L and C that provide a perfect match. The angular frequency is called the cross-over. A familiar example is the cross over used in audio systems. The network that splits signals is a diplexer. Here is an example where both outputs are used. Another form of diplexer is the band pass/band-stop combination. This is shown in Figure 3:

Let's now consider further some examples, some that work and some that don't work as well. First, let's look at an audio diplexer that follows a product detector in a DC receiver. The load of interest is the first audio stage, which has a 50 Ohm input resistance. The diplexer offered is Figure 4. Note that this is not the combination of filters. It just looks like a low pass with an extra resistor. The response of this circuit is shown in Figure 5. The transmitted signal never gets

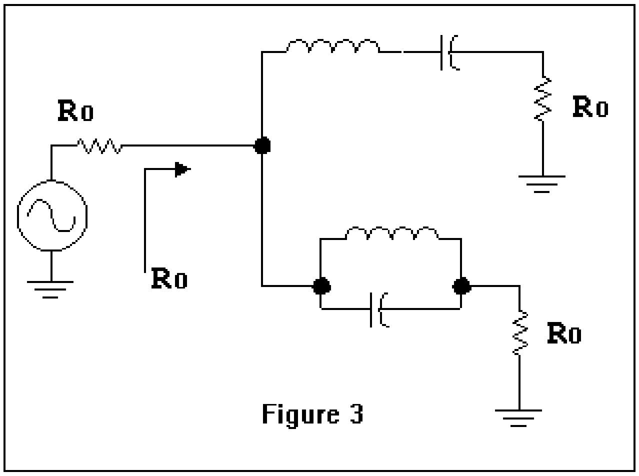
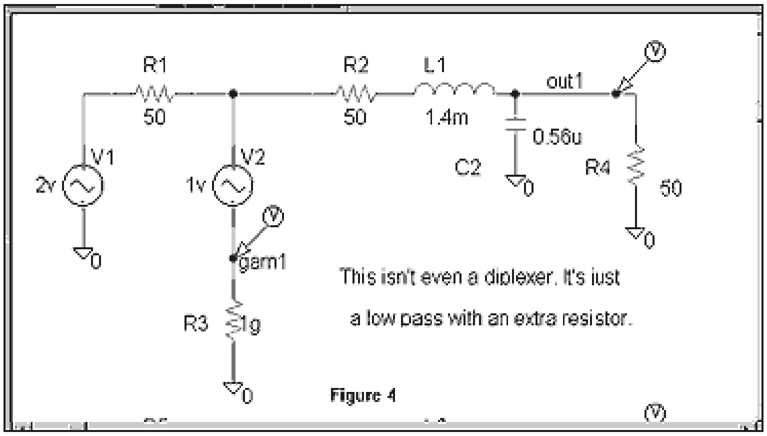


Figure 3

up to the desired 1 volt in the low pass passband while the impedance match, represented by reflection coefficient, never gets down to the desired zero. The response is just that of a lossy low pass filter.

In 2012; Click on many of the diplexer images to see the original sized version



This isn't even a diplexer. It's just a low pass with an extra resistor.

Figure 4

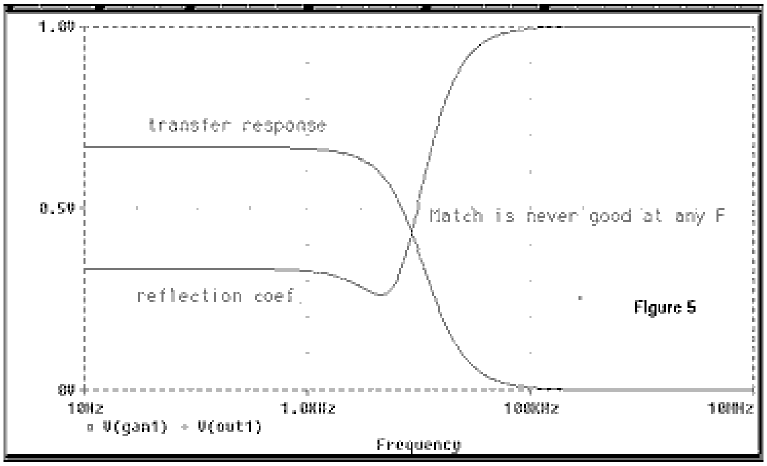
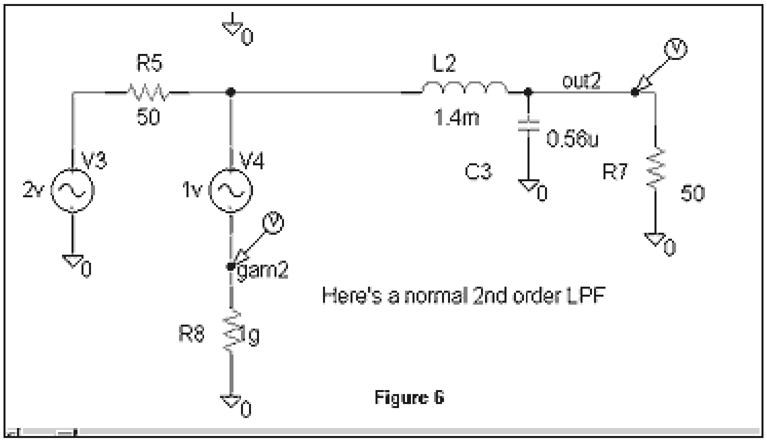


Figure 5

The normal filter circuit without the extra resistor is Figure 6. The corresponding output response is shown in Figure 7. Note that the transmitted signal is now up at 1 while the reflection is down to zero, both within the passband. Transmission goes to zero while reflection is 1 in the stopband.



Here's a normal 2nd order LPF

Figure 6

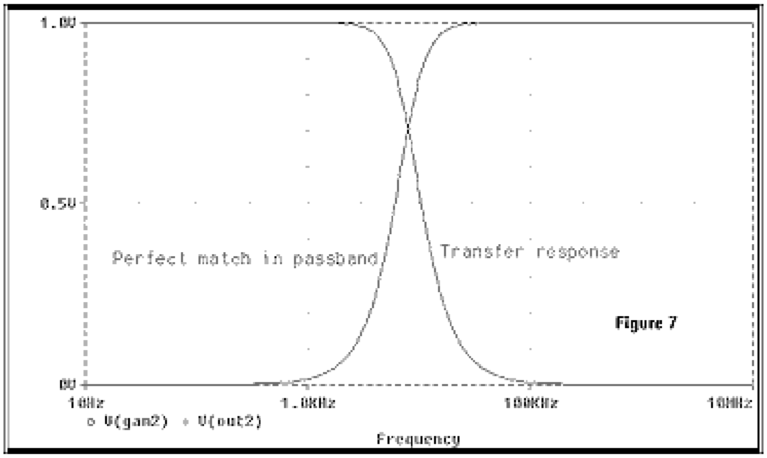
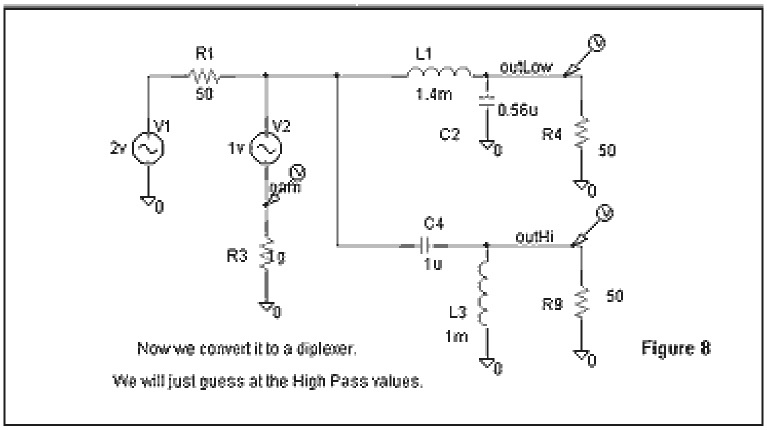


Figure 7

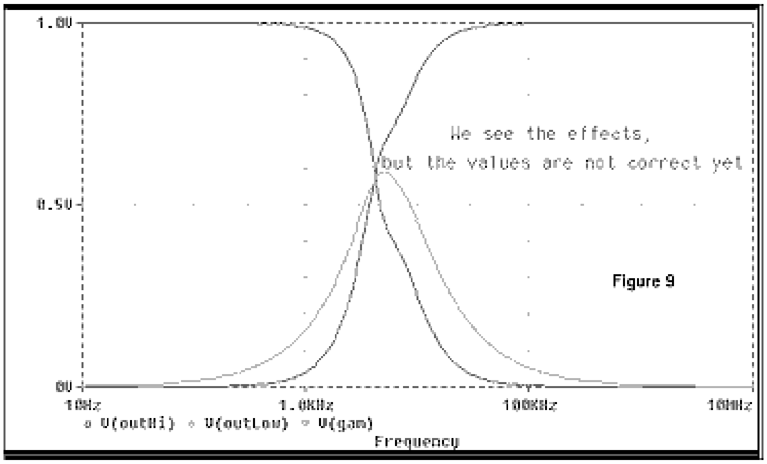
Now let's use the low pass and put a high pass with it to try to form a diplexer. This is shown in Figure 8 where we now have just guessed at component values. The response, shown in Figure 9, has high pass and low pass outputs that we might expect. The match is good at the frequency extremes, but is only so-so in the transition band.



Now we convert it to a diplexer.

We will just guess at the High Pass values.

Figure 8



We see the effects,
but the values are not correct yet

Figure 9

Let's now look at a carefully designed pair of two element filters. The circuit is Figure 10 an is a final example. The corresponding response is Figure 11. It is hard to see, for the response merges in with the baseline. However, the reflection is zero and it is zero everywhere. This filter was designed for a 1 kHz crossover, so it can be scaled to other frequencies with ease.

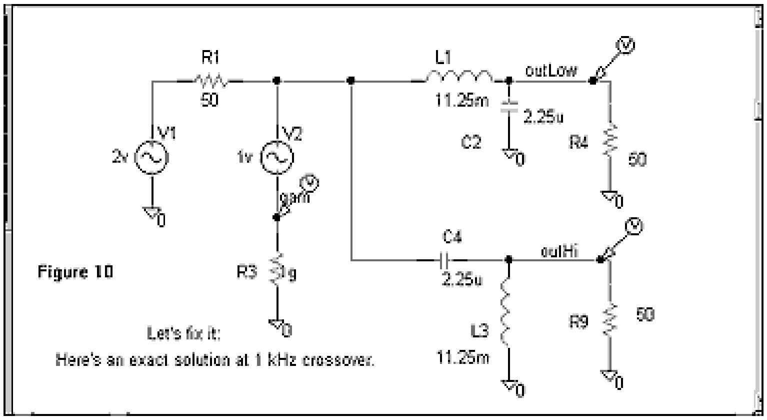


Figure 10

Let's fix it:
Here's an exact solution at 1 kHz crossover.

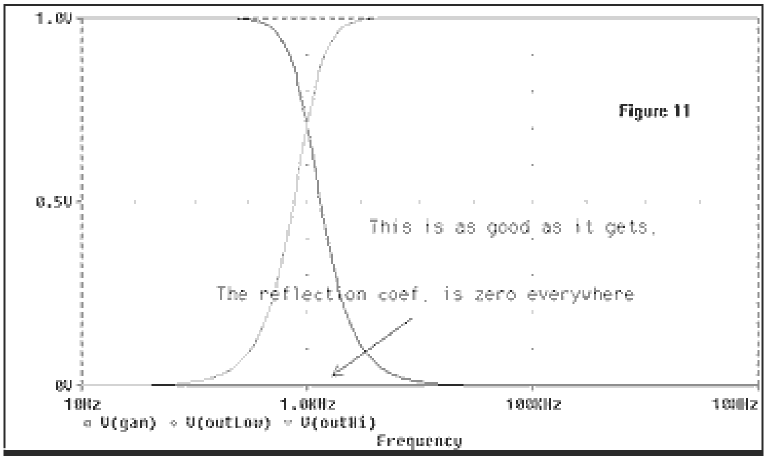
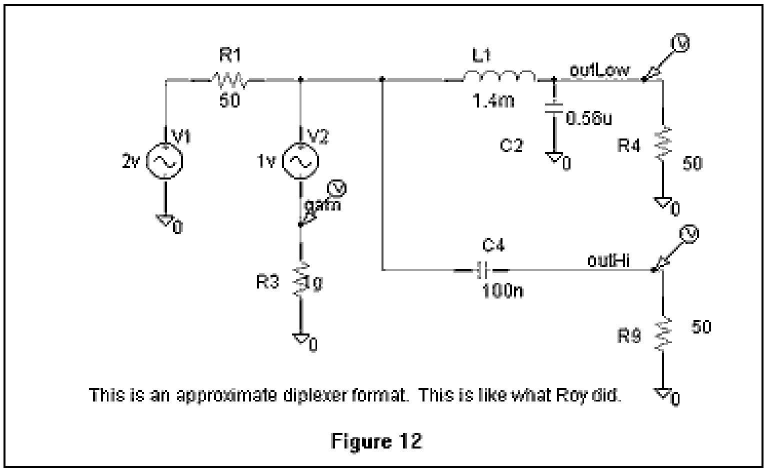


Figure 11

This is as good as it gets.

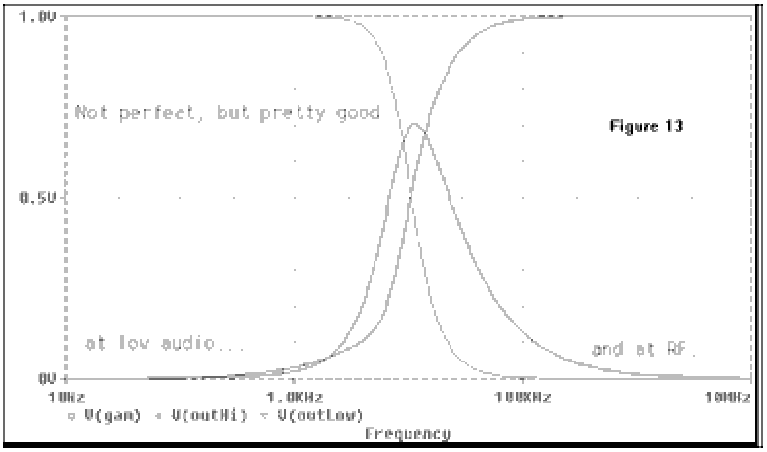
The reflection coef. is zero everywhere

Figure 12 is another final audio example. This circuit is very similar to the one used in the past by Roy Lewallen, W7EL, although the inductor was smaller at 100 uH in his Optimized rig. The response of this diplexer is shown in Figure 13. This is not perfect, but it is probably quite a good performer in typical receiver situations.



This is an approximate diplexer format. This is like what Roy did.

Figure 12



Not perfect, but pretty good

Figure 13

at low audio...

and at RF.

Finally, here's a higher frequency example. 5th order low pass and high pass filters are combined. The filters have a cross over at about 150 MHz. Note that there is a slight reflection in the transition band. This is probably just the result of our having rounded some values in the design process. Figures 14 and 15. An outstanding reference on this is Nic Hamilton, G4TXG, "Improving Direct Conversion Receiver Design," Radio

Communications, April 1991.

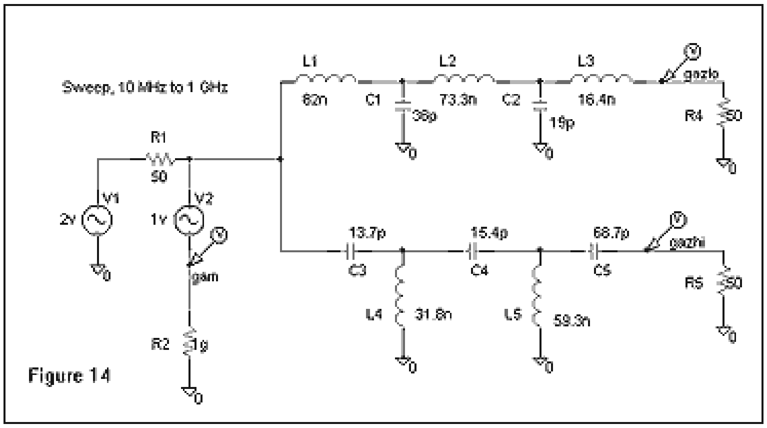


Figure 14

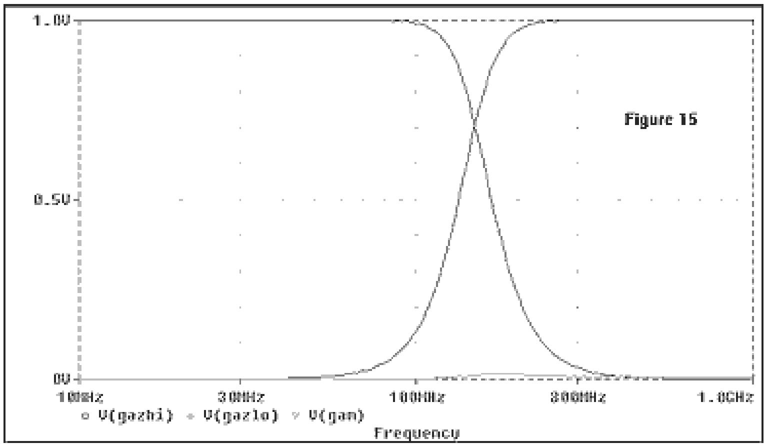
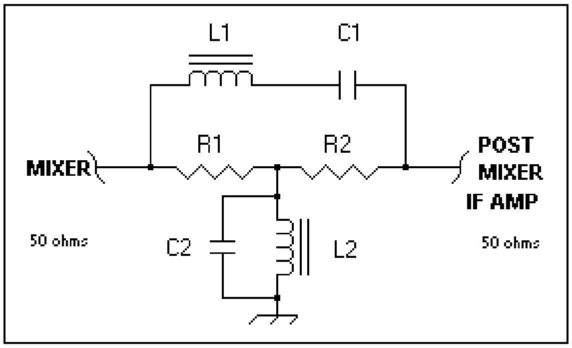


Figure 15

Bridge-Tee RF Diplexer



This is an excellent bandstop/bandpass diplexer popularized by Joe Reisert W1JR. This easy to build diplexer has a low parts count and is easily built using Ugly Construction. Resistors R1 and R2 present a 50 ohm impedance to the mixer output and a 50 ohm impedance to the input of the post mixer amplifier. The IF frequency is passed through the diplexer while out of passband RF is given a low impedance path to ground. The capacitance for C1 is generally built up by substituting the nearest standard value capacitor or by placing 2 or more capacitors in parallel with each other to achieve the desired value. The same procedure is then repeated for the C2 capacitance. For more strenuous purposes, a portion of C1 and C2 or the inductors L1 and L2 can be variable and adjusted on the bench. The inductors can easily wound on powdered-iron toroid cores. I have used T50-2 or T50-6 type toroids with good results. The Q of the inductors is 1.

It is possible to design a more generalized form of this diplexer with a higher loaded Q in the resonators. The diplexer shown and used in the program has a Q of 1. This was used by W1JR in his VHF/UHF World Column in the now defunct HAM Radio Magazine for March and November 1984. It was also more recently used by Jacob Makhinson, N6NWP in his A High-Dynamic Range MF/HF Receiver Front End in QST for February 1993. The actual formulae for this diplexer is far more complex than the simplified formula shown below or used in the program, but both provide a very good approximation for the Q = 1 version as used by W1JR and N6NWP. If you wanted Q=10, the series tuned circuit would use L that is 10 times as high with C to resonate. The parallel tuned circuit would then use C that was 10 times higher with L to resonate.

A supplemental web page with some hard-core mathematics for this diplexer can be found on the Diplexer Supplemental Page.

Simplified Formulae (Q = 1):
R1 and R2 are always 51 ohm resistors.
Inductors L1 and L2 -> 50 / (6.283 * frequency in Megahertz)
Capacitors C1 and C2 -> 1 / (6.283 * 50 * frequency in Hertz)

Example 1: For a 9 Mhz IF , L1 and L2 = 0.88 microhenrys and C1 and C2 = 350 picofarads
Example 2: For a 4.92 MHz IF , L1 and L2 = 1.62 microhenries and C1 and C2 = 647 picofarads

I wrote a simple program to do the math for the Q = 1 version. Download the Bridge-Tee RF Diplexer Diplexer Program

---

Comments and analysis by W7ZOI

This is a double ended version of the first order bandpass/bandstop design presented earlier. But it's a good one, within the constraints of what it can do. The first is the simulation schematic for the diplexer, which is better termed a Bridge-Tee Diplexer. (There are bridge Tee filters and attenuators too.) That figure is entitled Figure 16.

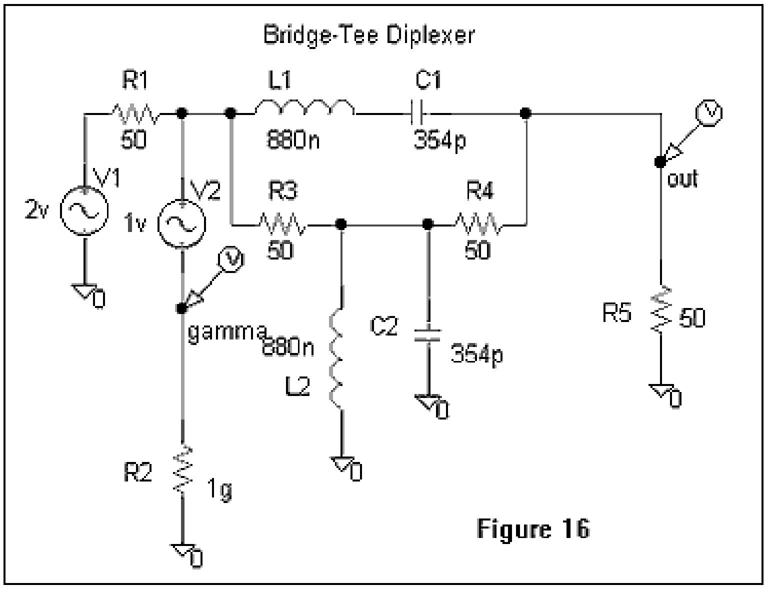


Figure 16

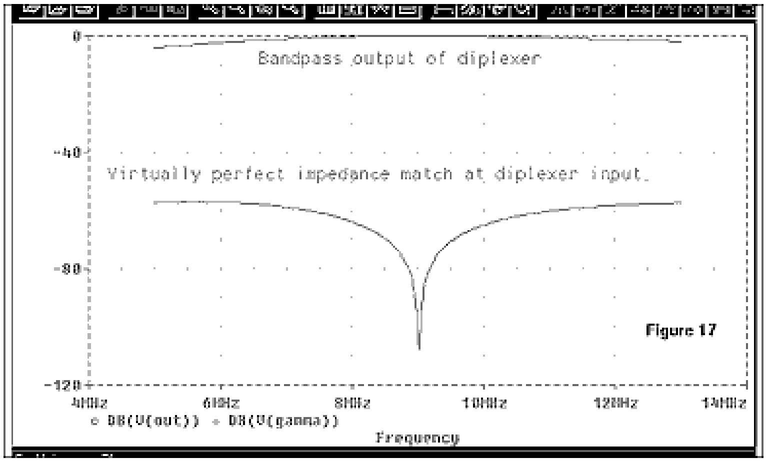
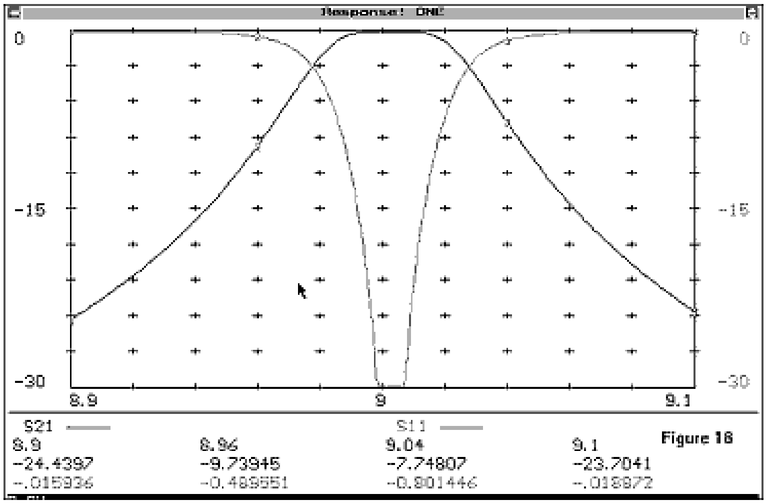


Figure 17

The response for this circuit is shown in Figure 17. This is extremely good. The through response is very flat owing to the low Q of the series tuned circuit. But even better is the match. It is very good. Indeed, it would have been perfect except for slight roundoff errors that occurred as we designed the networks.



```
S21 ——              S11 ——
8.9        8.96       9.04       9.1
-24.4397   -9.73945   -7.74807   -23.7041
-.015936   -0.489551  -0.801446  -.018872
```

Figure 18

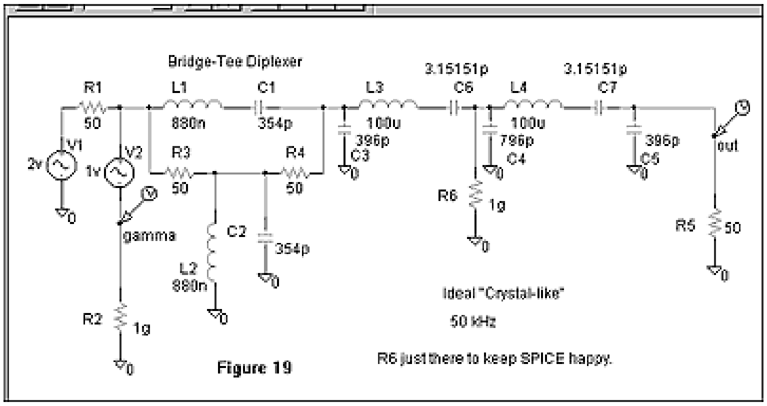


Figure 19

This kind of thing works fine if you really have a perfect match following the diplexer. But what if you don't. There are some places where they do not do the job that some folks think they will do. For example, a diplexer WILL NOT cause the impedance to be flat if it is followed by a filter. The diplexer must still be properly terminated at both output ports. In Todd's usual applications, he is worried about providing a good mixer termination for a product detector. The audio amp that he uses will usually have a common base first stage and that will present a good wideband load to the diplexer, so he is okay. But other folks have placed a diplexer after a switching mode mixer that then drives a narrow filter. The diplexer then does little good. To illustrate this situation, I designed a "crystal like" two pole LC bandpass filter with a 50 kHz bandwidth. This represents the general case where we try to put a diplexer between a mixer and a filter. The filter response by itself is shown in Figure 18. The schematic for the diplexer and following filter is in Figure 19.

The response for the combination is in Figure 20. Here we see a passband response that is fine; it's just the repeat of the filter response we already saw. However, the input impedance looking into the diplexer, the impedance that would be seen by a mixer, is terrible. The return loss is 0 dB at all frequencies except where we get within the passband of the filter.

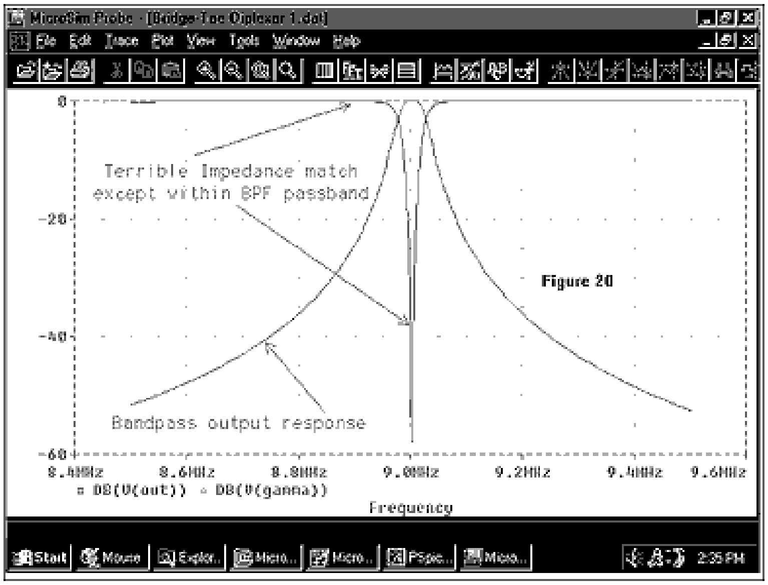
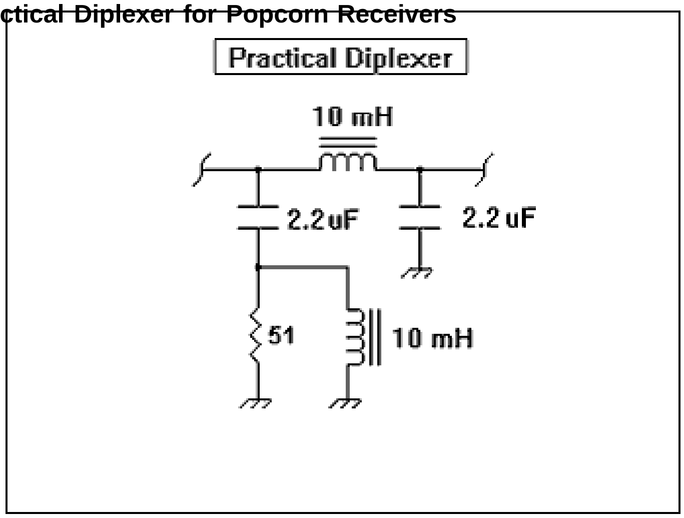


Figure 20

Practical Diplexer for Popcorn Receivers



Building a diplexer to follow a product detector is not a cheap endeavor. Audio inductors and capacitors such as metalized polyester film types are not common in many builder's junk boxes. I really like the design shown in Figure 10 and wished to use it because it uses just 2 inductors and capacitors which is in keeping with the popcorn nature of this website. The main difficulty is that the inductors and capacitors are not standard value types and series connecting components to achieve the desired values would add to both the cost and size of the finished product.

Obviously, it will not likely match from DC to daylight. That is not the intention of this simple design or this web site in general. I asked Wes to place just 2 standard value capacitors and inductors in the Figure 10 diplexer design and see what happens.

Here was his response to my request:
OK, here are some "practical values." Note that things don't really change that much. We start with 11.x mH and 2.25 uF. Change the inductor to 10 mH and get Figure 21. Then change the cap to 2.2 uF and see almost no change in Figure 22.

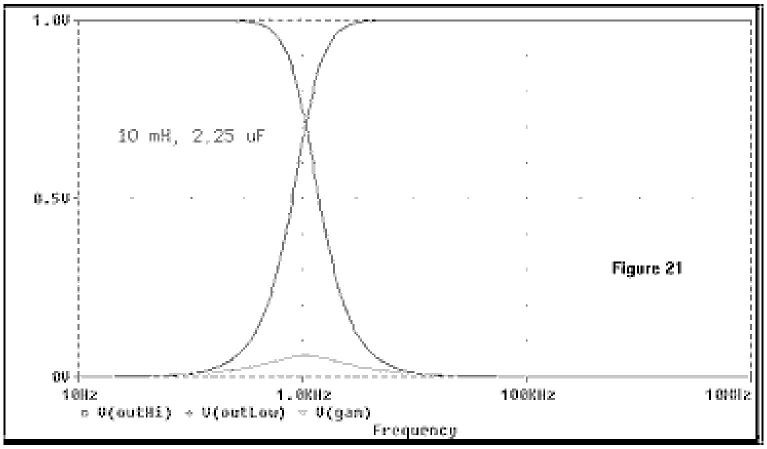


Figure 21

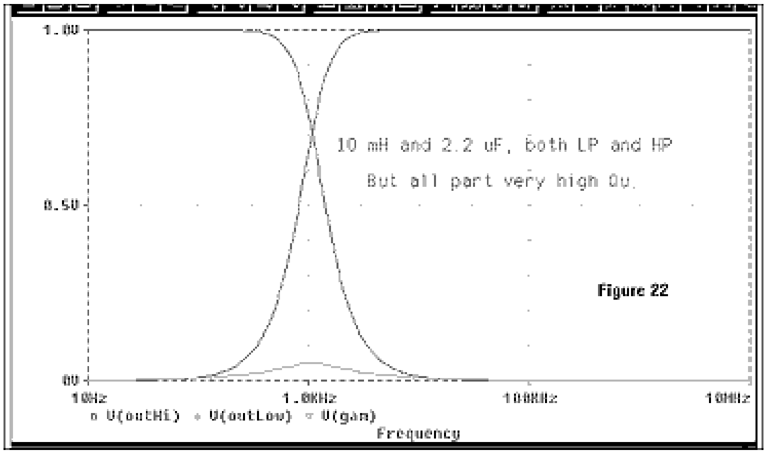


Figure 22

But now move into the world of even greater reality and acknowledge that many of the inductors we use at audio are very low Q. Change Qu of L to 10 at 1 kHz, so I put 6.3 Ohms in series with each L to get Figure 23. And do the same thing, but with a dB scale, for Figure 24. Note that we can see the difficulties, but things are still pretty good. We see some loss (about 1 dB) in the low pass path and less than perfect match. But the match is still very good. 20 dB is about 1.1:1 vswr, much better than 99.9% of the hams can really measure. (A 10 dB match is about 2:1.) Hope this is what you were after....Wes

It was and I will use this "practical" diplexer in my next popcorn DC receiver project. Note that the practical diplexer input and output impedance is 50 ohms and the 2.2 uF caps should not be polarized capacitors such as regular electrolytic types which have a positive and negative polarity.

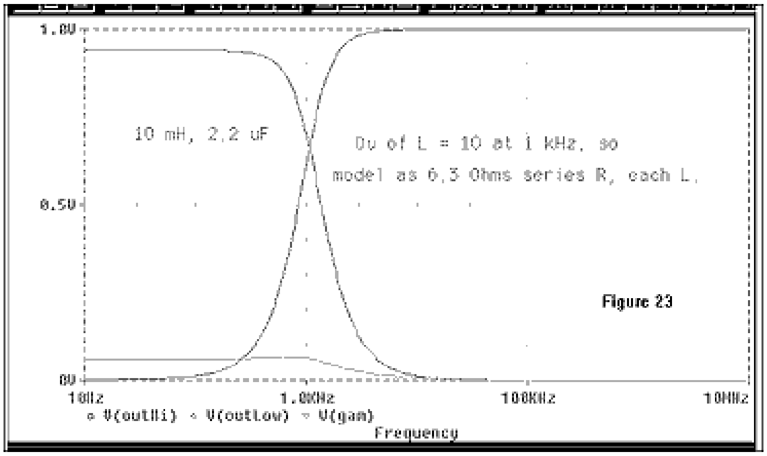


Figure 23

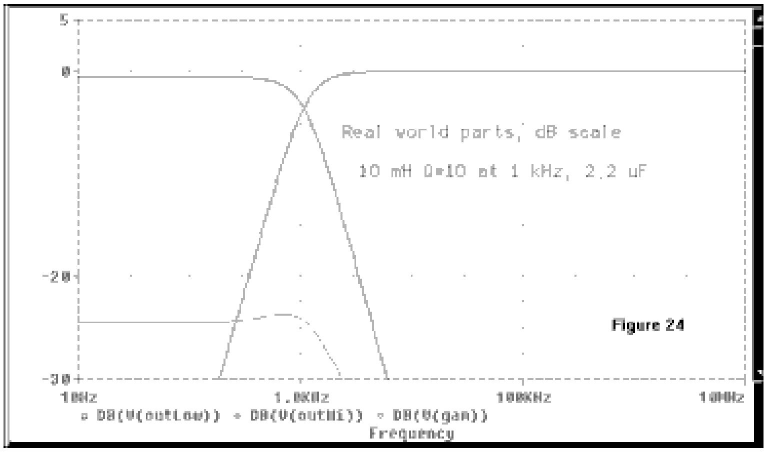


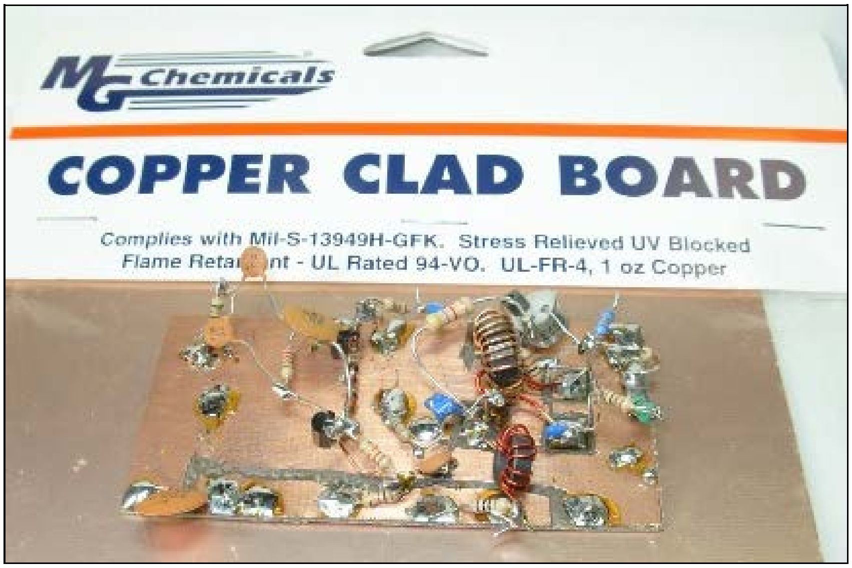
Figure 24

Many thanks to Wes Hayward, W7ZOI for his work on this page.
A version of this web page in Russian Cyrillic

RF — Test and Measurement

Ugly Construction

Discussion

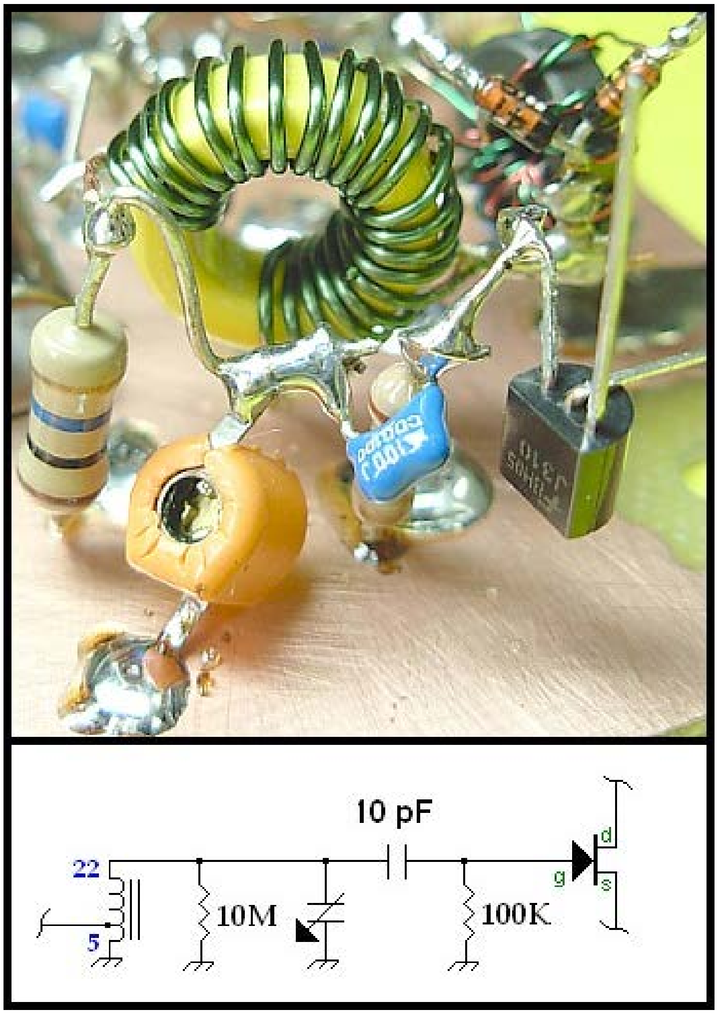


We enjoy many ways to build electronic circuits. For example, you might breadboard on a perforated circuit board, an etched PC board, a sheet of copper clad board, or even a piece of copper wire. In the hay day of tube electronics, builders used terminal strips and point-to-point wiring within the project chassis. I mostly breadboard using Ugly Construction.

Ugly Construction, "dead bug", or "ground-plane construction" involves building circuits on top of a double or single-sided copper clad board (copper side up for single-sided board). The copper ground-plane provides a low impedance ground and mechanically supports the parts soldered to it. Component leads requiring grounding are soldered *directly* to the copper surface, while the ungrounded leads of these parts anchor any ungrounded parts connected to them. Isolated sections called stand-offs hold other ungrounded or remotely located parts.

Example stand-offs include high value resistors (10 Megohm or greater), terminal strips, or small copper islands glued onto or cut into the copper ground-plane. Parts such as transistors, IC's or commercial diode ring mixers are generally flipped upside down and anchored by their grounded lead(s). Metal encased parts such as crystals can be grounded by a short wire or directly soldered upside down to the copper board.

DC voltage wires, or decoupling resistors may be supported by soldering 1 lead of a bypass capacitor to the ground plane while the other lead holds the DC voltage carrying part up off the copper board a short distance. I mostly use grounded caps for stand-offs and not 10M resistors.



Shown above — a 10 Megohm, half-watt, stand-off resistor anchors the "hot" inductor terminal plus supports the ungrounded trimmer capacitor terminal. The coil's 24 gauge wire provides additional mechanical stability. The signal loss from adding the 10M resistor was about 0.1 volts peak to peak in 1 experiment.

Ugly Construction allows the experimenter total control over the design of a project and in my opinion, its greatest strength is *speed*. Ugly Construction yields rapid and flexible bread boarding — very appealing for prodigious home builders.

The Origin of the Term "Ugly Construction"

Roger Hayward, KA7EXM and Wes Hayward, W7ZOI coined the term "Ugly Construction" while writing the "Ugly Weekender" published in the August 1981 issue of QST. I asked Wes about this in 2009. The term was a takeoff from the 1958 book entitled The Ugly American by William Lederer and Eugene Burdick.

A big part of the learning of the QST article was Ugly Construction! The term and the bread boarding technique emphasized the fact that there is no correlation between the "prettiness" of a construction project and the way it works. According to Wes, the goal had a couple of corollaries. "First, people like myself who do NOT have the knack for doing pretty projects can still build successful radios. Second, is that we all need to look at our projects after the fact to discover what it is that really makes them work well. In the case of the Ugly Weekender, the thing that makes it fly is that there is a wonderful ground plane with that PC board material".

Indeed, this transceiver functions very well; especially after you temperature compensate the VFO. Wes also built versions for the 30 and 80 meter bands. I have versions on 15, 40 and 80 meters. The transmitter portion is a true QRP classic; both as a Ham radio transmitter and because it promoted "ugly" ground-plane or dead bug bread boarding techniques to the scratch homebrew community.

Classic Ugly Construction

This term emerged in Spring 2010 and describes the archetype popularized by Roger and Wes Hayward. All ungrounded leads not anchored to other parts are attached to the copper ground-plane via high ohm resistors — *no* glued pads or islands cut in the copper board.

In-situ comparisons of a 10 megohm resistor versus islands cut into the ground plane and glued-on Manhattan-style pads demonstrated that the resistor had the lowest capacitance; around 1 pF versus 4 pF or greater for the pads or islands. Click for a high resolution transmitter chain built with 100% Classic Ugly Construction. Click for a crystal oscillator.

Above — *Classic* Ugly Construction using a high ohm stand-off resistor. The top of the 10 megohm resistor is the VCC connection point. It feeds a 150 ohm / 47 uF decoupling network connected to a transistor collector resistor. Bypass capacitors also serve as stand-offs and I prefer thick lead (100 volt or greater) caps for stand-off duty.

Above — The original Ugly Weekender. Photograph used by permission of Roger, KA7EXM.

Above — The original Ugly Weekender. Now this is *Classic* Ugly Construction. Please refrain from building this transceiver and adding modern notions such as a PIC microcontroller keyer — that's just wrong!

Photograph used by permission of Roger, KA7EXM.

Ugly Construction Variants

The most popular Ugly Construction variant is called Manhattan style. Manhattan or "paddy board" construction uses small square or round pads cut or stamped from PC board that are glued copper side up onto a large copper clad board also placed copper side up. The small pads or "islands" serve to anchor ungrounded components. Components soldered to the pads such as transistors or ICs are generally not positioned upside down like in Classic Ugly Construction. Many Manhattan style builders use IC sockets as well. These hobbyists sometimes build beautiful looking layouts — Manhattan is a wonderful bread boarding technique. Google for more information. The best Manhattan construction and copper board chassis bashing I've seen comes from Dave, AA7EE. Click for a blog describing his version of the WBR regenerative receiver. *Linked with the permission of Dave, AA7EE.*

Another interesting variant is used by Dick Pattinson, VE7GC. The circuit board is placed copper side up and holes are drilled and countersunk so that the holes are isolated from the ground plane. Ungrounded components are connected underneath the main board through the countersunk holes. There are many such variations. Each Ugly Construction variant has advantages and disadvantages.

On this website, stand-offs are created by cutting a few lines into the copper board with a small, motorized hobbyist cutting tool; with high megohm value resistors, and occasionally by a small Manhattan style pad or 2. Manhattan pads are great for supporting components needing solid anchoring such as a trimmer capacitor or potentiometer.

Classic Ugly Construction dominates circuits breadboarded after May 2010.

The motor tool may also be used to grind off the copper underneath where VFO toroidal inductors will lie, so that the inductor Q is not effected by the being glued onto a copper surface. In audio projects, I may also grind off the copper around the copper board mounting bolts so that they are isolated from the chassis and do not provide multiple grounds and create the potential for ground loops.

Is Ugly Construction Less Reproducible than Manhattan?

I've received well over 1000 emails about Ugly Construction since launching the site in 1998. Some feel that circuits made with Manhattan pads are more reproducible than Classic Ugly Construction builds. This might be true, but to my knowledge nobody has performed a comparison trial.

The important question is why would this happen? I've read/heard opinions that the stray L and C from the long component leads associated with Ugly Construction might wreck circuit reproducibility, but respectfully disagree from DC to HF. In microwave breadboards, we fabricate lumped element inductors and capacitors (*i.e. precisely dimensioned Manhattan pads*) right into the PC board — Manhattan pads glued all around a breadboard may potentially exhibit much more stray L and C than a few component leads in an Ugly build. Also, wise Ugly builders keep their lead lengths short where it counts: for example, RF bypass + ground and at the input/output of a BJT or FET that offers gain into UHF.

I conjecture human error probably inflicts more problems for Ugly Construction builders — Manhattan building, with its slower pace might trigger less mistakes by newbies. Still, too, Manhattan builders tend to make prettier, squared and aligned circuits and it's easier to spot trouble — plus they look nicer in photographs and some builders carefully document and photograph their builds for others to admire and strictly copy. I've see Manhattan build photos where every resistor tolerance band pointed in the same direction — wow! I think it might be difficult to put such a 'work of art' into an RF-tight metal box for much-sought isolation.

Further, in Ugly Construction — upside down parts might wreak havoc on the "spatially challenged" builder. Who knows? I'm comforted knowing that kit sellers who provide a screened printed circuit board with explicit instructions, still must provide major email support to mitigate build errors. To err in an ugly fashion is human?

Whatever variant of construction you choose, it's sure to be a winner!

Further Discussion

Wire

Non-stranded (solid core) copper wire such as the 22 AWG 3-color package sold by Radio Shack seems a good choice for hook up wire. With non-stranded wire, you do not have to worry about little stay wire hairs causing shorts and it's easier to wrap around components leads. I use red for wires that carry positive voltage, green for grounding and black for wires that carry AC signals short distances. In addition, RG-174 or shielded wire is used to carry AC signals for distances greater than 10 cm, and for connecting stages requiring 50 ohm input or output impedances such as diode ring mixers or low-pass filters.

Your Health

Please consider the following safety comments:

For regular soldering, ensure ventilation of your room — flux fumes can be harmful. Open your shop window and/or use a small fan to improve fresh air intake;

Whenever possible, perform high wattage soldering outdoors;

When grinding paths on copper clad boards, wear a small particulate respirator, gloves, plus ear and eye protection and most importantly; **do it outside** for yours and your family's health;

Wash your hands after soldering and handling freshly cut, fiberglass dust laden copper clad boards.

Soldering Irons

For soldering copper clad boards together, AC grounds on tube guitar amp chassis and performing antenna work, I currently use a Weller SP 80L (80 watt) soldering iron. It is heavy and unwieldy, so you have to be very careful when its plugged in. These high wattage soldering irons produce lots of smoke.

Shown above is an 80 watt "heat torch". My main soldering irons are typically in the 30-35 watt range. Consider keeping at least 1 back up soldering iron, as you never know when a soldering iron is going to burn up. My current 35 watt iron is shown below. These Weller irons have a built in lamp which lights when they are plugged in; a very nice feature. I also keep a small stock of new soldering iron tips.

Copper Clad Board

Some builders ask about sources for copper clad board. I personally use boards made by MG Chemicals as they have dealers in my city and are reasonably priced and good quality. Try the search words **copper clad board** plus **your country name** in your favorite web search engine. A few links follow, but as I have only used boards sold locally, I can't comment about the online companies.

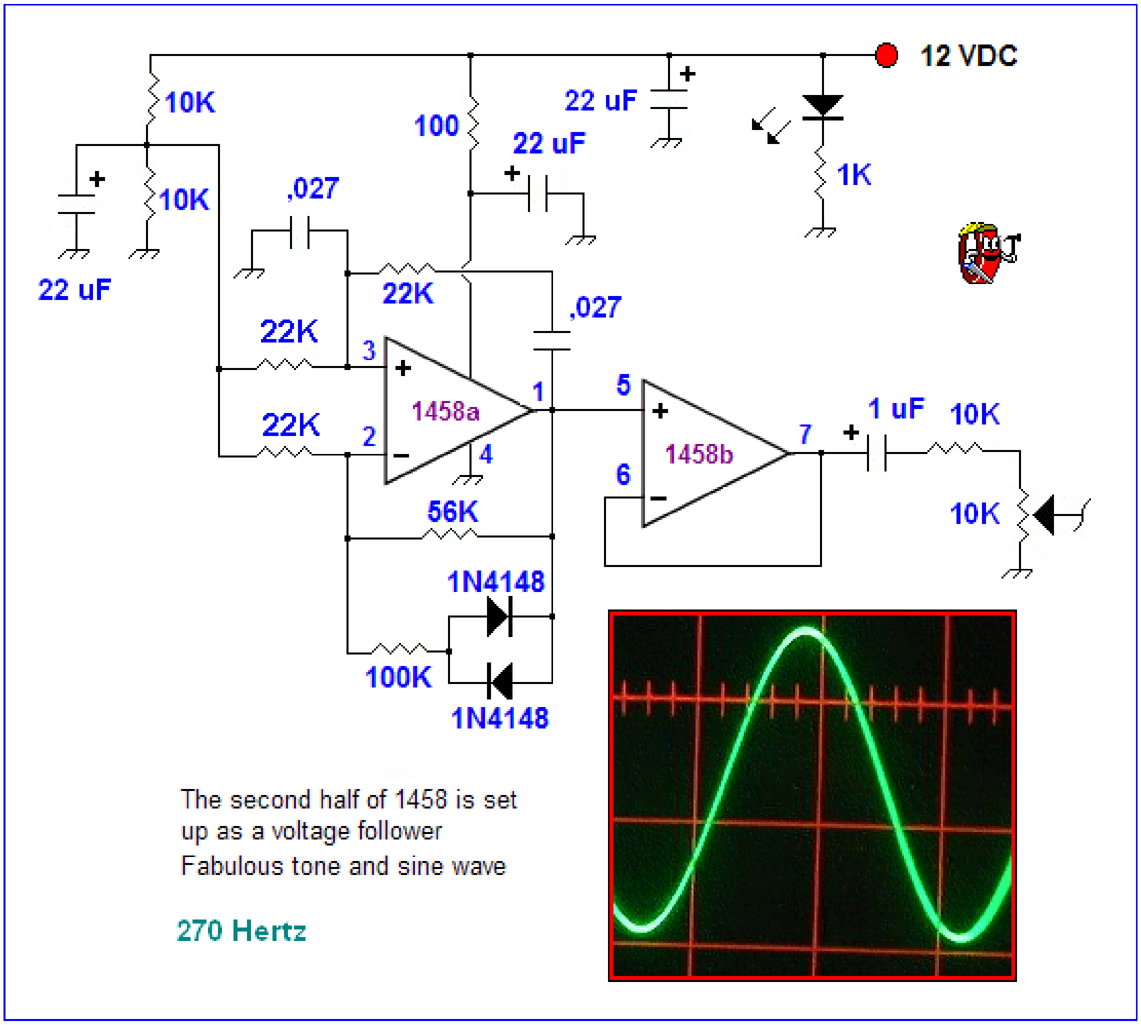
MG Chemicals Worldwide distributer index

Electronic Goldmine Online store

Circuit Specialists Online store

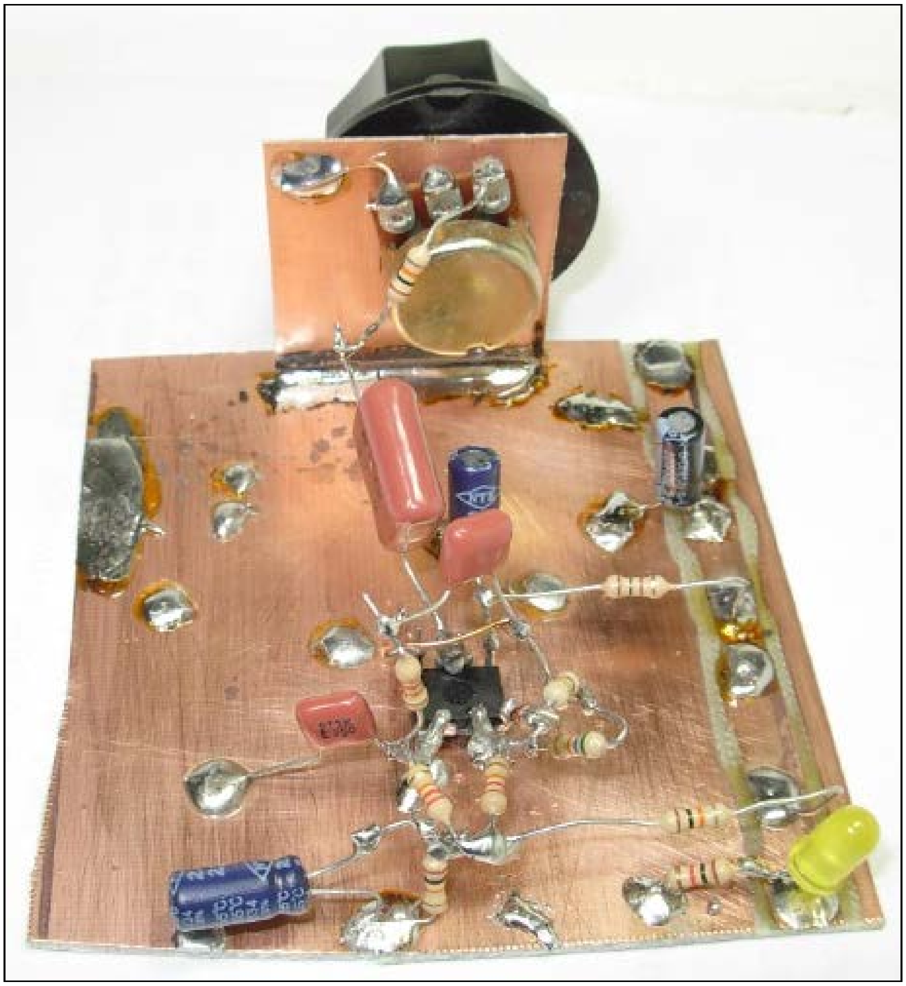
Miscellaneous

Shown below is a schematic and the Ugly Constructed version of it.

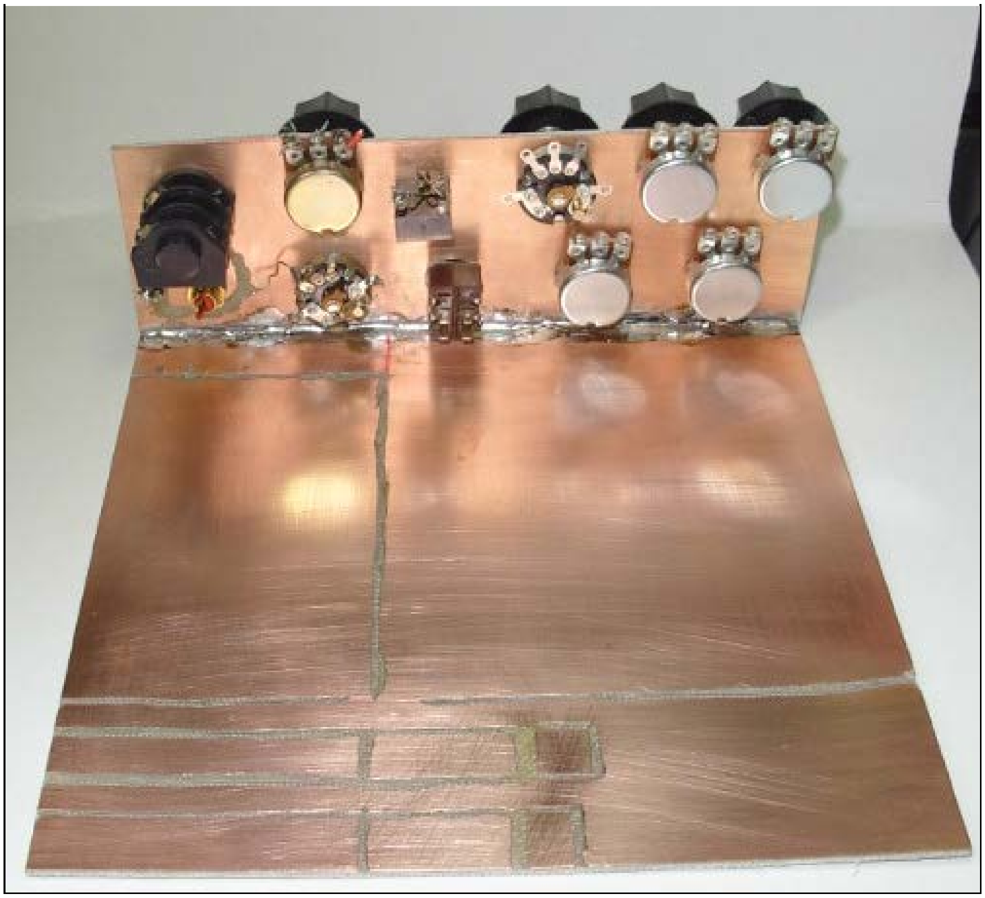


Above. The schematic of an adapted sine wave audio frequency oscillator taken with permission from EMRFD , Figure 12.4. EMRFD is the main reference for this web site. The original schematic author was Wes, W7ZOI

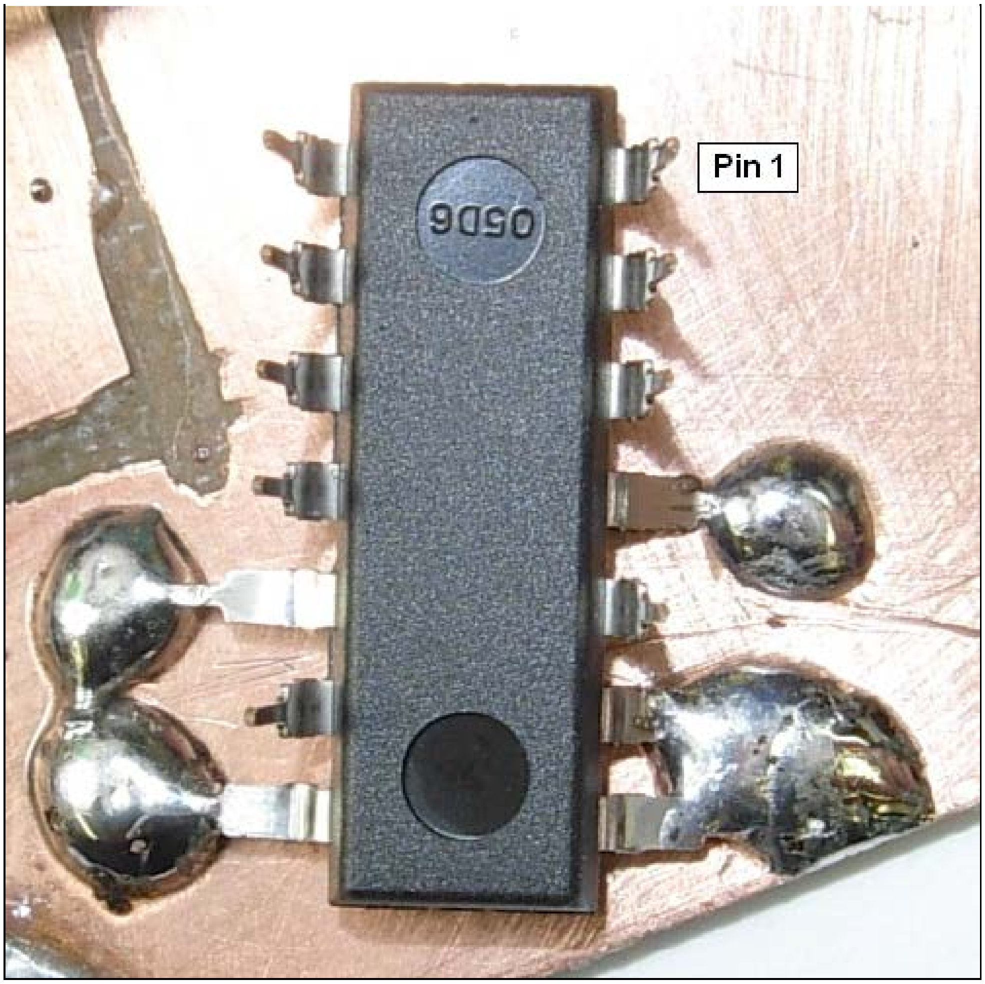
Above. I built this circuit from start to scope in about 25 minutes. This was a scrap, pre-used board with a positive voltage path and a potentiometer holder already on it When miniaturization is not your goal, construction is much easier and faster. I re-use parts and boards to save money. You may remove entire stages from 1 board and solder them onto another.



Above. Note how the 10K output potentiometer holder is soldered to the main copper clad board. The grounded 10K resistor is used to anchor the 22K resistors connected to IC pins 2 and 3 and can be seen in the foreground.

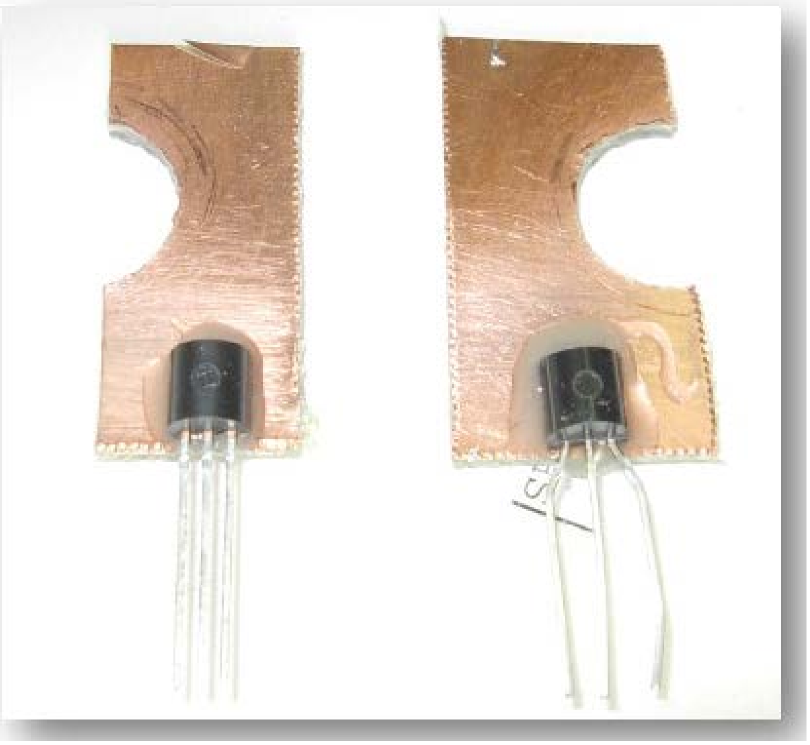


Shown above is another project. Entire control panels can be built from copper clad board for prototype circuits. In this board are numerous cut paths, 7 potentiometers, 3 jacks and a switch. Do not build an LC VFO over double sided copper clad board; lest it become "a capacitor" and affect your frequency stability.



Shown above is a CD4013B D Flip-Flop soldered "dead bug" style. Pins 4, 6,7, 8 and 10 are grounded to the copper surface; well anchoring this part. Using proper static precautions, I have never had a CMOS device failure using Ugly Construction and also save the price of an IC socket.

Above — a method to anchor op-amps using a split (negative and positive) power supply. Pins 4 and 8 are soldered to the copper board via a 10 megohm resistor. A 0.1 uF ceramic bypass capacitor is also connected to these pins. The resistor leads were left a little long to allow easy connection of the power supply wires. I write each pin's polarity on the board to avoid wiring mistakes.



Shown above is another use for copper clad board; heat sinks. In this case, 2 scraps are epoxy glued to 2N3904 and 2N3906 transistors.

A flux pen like this one from MG chemicals is a handy item for the QRP workshop. They are great for applying liquid flux to allow easy and precise soldering of SMT components. Also sometimes when adding components to ground in already built up circuit boards, it can be difficult to get your soldering iron down at a low angle for proper heat transfer. Some flux can help solder a part to the copper ground plane in these situations.

Shown above are the basic tools used to cut copper clad board. A felt pen marker, small square and a set of straight aviation shears. If you press one end of the copper board into the side of your bench and keep pressure on the handle of the aviation shears with your thigh, it is possible to make long, straight cuts. The board will flex and move out of the way as you cut. Your leg provides the force to advance the aviation shears.

The motorized grinding/cutting tool used to carve out small pathways in copper clad board.

Steel wool used to clean up copper clad board before construction. A box like this will last for years.

For 15 years, I've used this 9 mm cutter to scrape the enamel insulation off the magnet wire wound on toroidal inductors.

QRP SWL HOMEBUILDER

Amateur and Short Wave Radio Electronics Experimenter's Web Site

VE7GC Wee Willy 75 Meter DSB Transceiver Project

Introduction

I meet a lot of interesting amateur radio enthusiasts online and on the air. Among them is Dick Pattinson, VE7GC who was first licensed in 1934. When Dick isn't sailing he builds and operates his homebrew QRP gear to hams around the west coast of Canada and the U.S.

Presented is a double sideband transceiver that Dick calls "Wee Willy". This rig has a low parts count and is easily built using non-etched PC board techniques. Dick even makes his own radio case using copper clad PC board for the front and back panels and cardboard for the rest of the case. The set itself is in a case 1 1/4 by 2 1/4 by 5 1/2 inches. There is a separate container which holds a 6 volt rechargeable battery and speaker. The speaker/battery case is about 2 1/2 inches cubed and is not shown.

The circuits are built in three sections on the circuit board, namely TX, RX, and VFO. An electret condenser microphone is on front panel along with T/R switch, volume and frequency adjust. The back panel has the antenna jack, power input cord and the speaker jack. The battery pack has a 100 uF capacitor to across the power leads for additional filtering and is shown in the VFO schematic.

The text that follows is clipboard pastings from email that Dick sent me with some additional comments and expansions by me. Dick's project exemplifies practicality and innovation and with that is a major contribution for the QRPHB site.

Construction Methods

Dick's electronic construction method is quite fascinating and represents yet another derivation of ugly construction. The electronic wiring is done on single sided PC board, copper side up. Small holes are drilled through the PC board material to allow component leads to pass through them. Then the holes carrying active leads are chamfered (countersunk) with a larger drill bit which is not run all the way through the PC board. This leaves an ground-insulated side to the hole and prevents a component lead short circuit. The copper being topside allows both convenient and short component grounding. The Wee Willy parts layout is extremely neat and compact. Dick, presumably through practice has great skill with this technique and I plan to try it in the future.

The project case is constructed from 1/16th inch cardboard which is cut and bent to fit the electronic PC board. Once cut, the outer surface and edges at the front are covered with tissue paper or Kleenex (tm) type tissues soaked in white glue. The applied tissue paper and glue is allowed to dry and then additional coats are added to build up a body. Alternately, the cardboard case can be coated with lots of glue and the covering material imbedded in the glue. The air bubbles are pressed out and extra glue is added where necessary. When enough material has been added to cover up and strengthen the case joints and the glue is perfectly dry, the case is painted with Rust Coat Enamel available at hardware stores. The end result is a glossy, durable finish which looks very sharp.

Transmitter

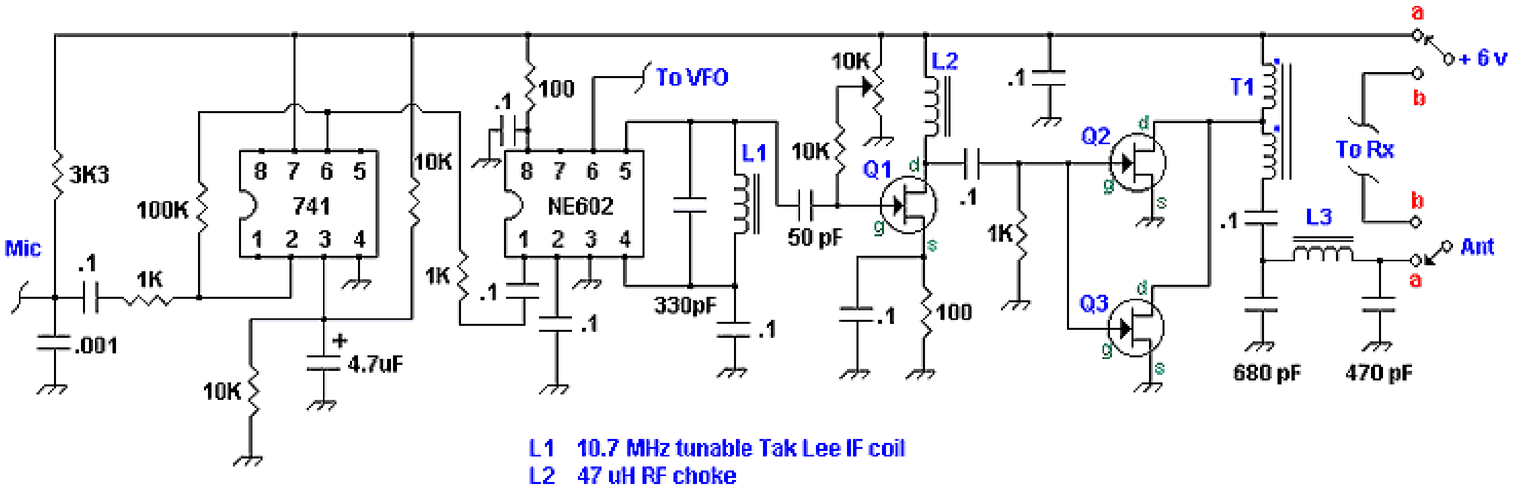
This transmitter uses an electret condenser microphone (Dick used an Archer 270-90). The mic is built right into the front panel of the chassis and this of course guarantees short mic leads. A 741 op amp is used as a speech amplifier which in turn drives the balanced modulator a Signetics NE602 doubly balanced mixer. The input and output impedance of the NE602 mixer is around 1500 ohms.

To adjust the transmitter, set the bias control on the VN10 stage to ground and tweak L1 to resonance using an RF probe or scope on the VN10 input. The input signal must be audio, spoken into the front panel microphone to get the DSB. Once L1 is tuned, connect a 50 ohm load to the antenna with some sort of RF indicator (such as a RF power meter) and advance the bias control to give a watt or so output. Then speaking into the microphone should result in a DSB signal suitable for communicating on QRP! No audio input should result in no RF output. The supplied voltage should be kept at 6 volts, remembering that NE602's cannot stand voltage greater than 9 volts. With suitable voltage control such as a 6.8 volt zener diode on these chips, one could use higher input voltage with a corresponding RF output.

There is another way of setting the bias on the VN10. After aligning L1, with a ammeter in the six volt supply line, advance the bias control until the input current increases about 10 mA (with no modulation). If you do not have an FT37-77 ferrite core, substitute 10 bifilar turns on a FT37-43 ferrite core for the T1 transformer

.

Wee Willy Transmitter



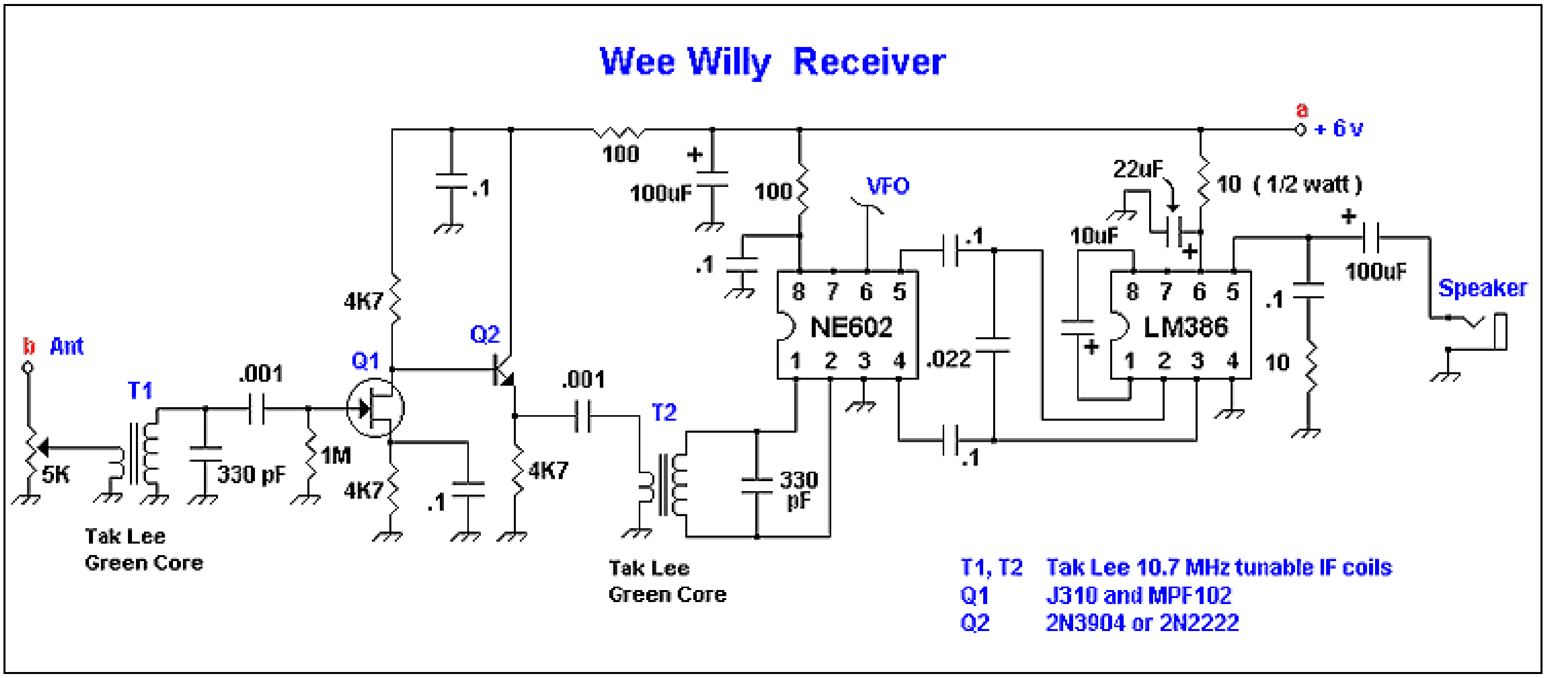
L1 10.7 MHz tunable Tak Lee IF coil
L2 47 uH RF choke
L3 22 turns # 24 AWG on a T50-2 Powdered Iron Core
T1 T1 is 6 turns Bifilar wound #24 wire on FT37-77 core
Q1 - Q3 VN10 nJFETs

Receiver

The receiver is a direct conversion type with a manual RF gain control in the form of a 5K potentiometer. Listening to a weak signal on the desired frequency the RF stage and the mixer core (T1 and T2) adjustments are made until you hear the loudest possible signal, keeping the input test signal as low as possible. When the receiver is connected to a doublet antenna there is no lack of incoming signal, which can be controlled by the front panel RF gain control.

The antenna and 6 volt supply is switched manually from TX to RX mode and back by a front panel mounted switch. If you can not find Tak Lee green 10.7 MHz IF coils, probably any other brand of 10.7 MHz slug tuned IF transformer would work. The Mouser catalog number 421F123 would work well and in another 80 meter project I used it with a 470 pF capacitor instead of the 330 pF cap shown. I would start with Dick's 330 pf cap and if it will not tune to resonance sharply, slightly increase the cap value up to see if a bit more capacity is required to resonate it on the desired 75 Meter frequency. Note that the secondary coil on the L1 transformer in the transmitter schematic is unused. If your 10.7 MHz IF coil has a built in capacitor at the base, remove it.

During receive, the standby drain current at 6.0 volts was 24 mA and on loud signals it rose to 100 mA. If this is too much, probably the easiest thing to do would be to put in a series resistor from positive to the LM386 to limit the drain current. To get output on the speaker it is a matter of how loud you want it for the drain you draw. If earphone only reception is okay, then the drain could be reduced considerably.



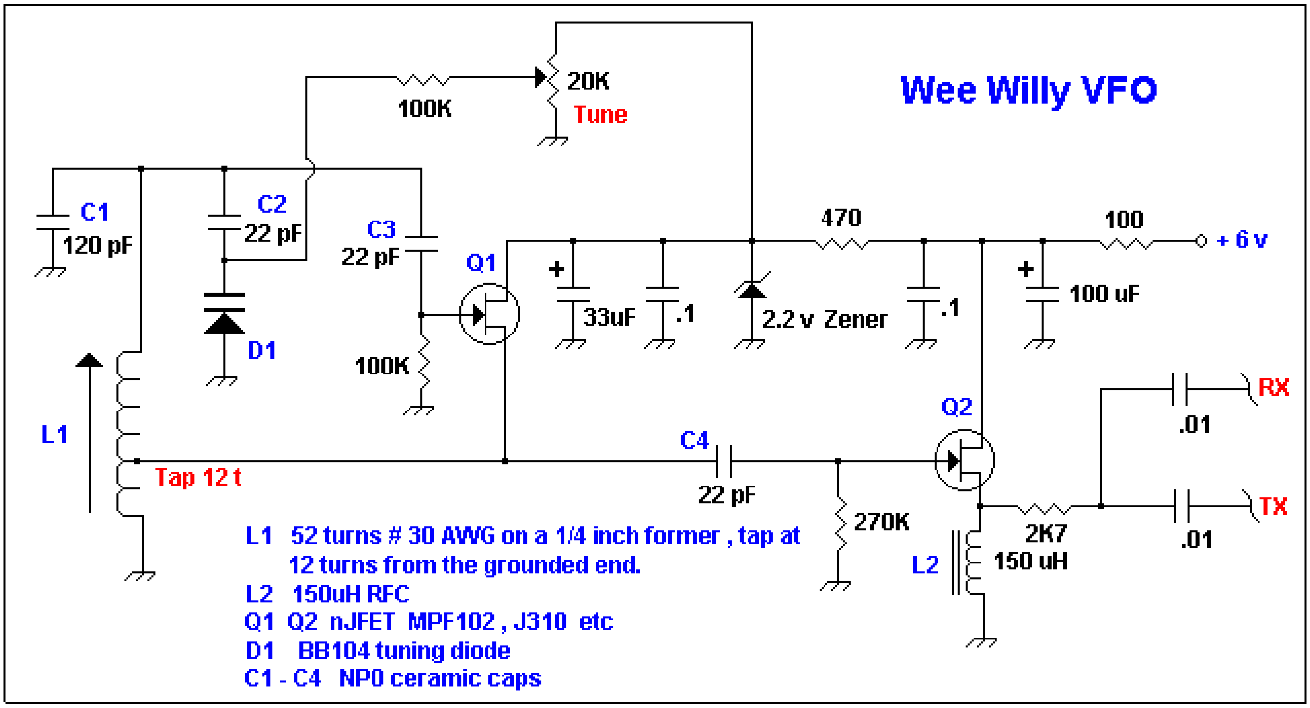
VFO

Dick's diagram indicate that this VFO was based upon a design presented in SPRAT for summer 1995. The main inductor L1 is wound with #32 AWG wire on a 1/4 inch slug-tuned coil former. This coil would have an XL somewhere between 250 - 310 ohms, so if you cannot find a coil former as described , you could easily wind one on a powdered iron toroid and make a portion of the C1 capacity variable for adjustment. A suggested alternate inductor is 53 turns of #26 AWG on a T68-6 core powdered iron core.

Dick suggests checking an old television to find suitable coil formers such as the one he used. It would probably be best to distribute the 120 pF C1 capacity among 3-4 capacitors to enhance stability. These caps should be NP0 ceramic for best results with frequency stability.

Dick's oscillator uses the slug tuned core to put VFO frequency close in frequency to where you want to operate and the variable resistor tuner on the front panel allows adjustment around the incoming signal to get the correct pitch. The desired band-edge is easily set by adjusting the slug while listening to the VFO frequency as audio on another receiver that has a frequency readout or directly with a frequency counter.

The L2 150 uH RF choke can be a simple epoxy unit which resembles a resistor. The D1 variable capacitance diode is a BB104 which has ~ 35 pF capacitance on each side. These are available at Dan's Small Parts and Kits whose URL is in the Links section of the site info web page. Experimentation with other tuning diodes could produce a practical alternative to the specified D1 part.



Operating Wee Willy

Dick sent me Willy in the mail and the first available moment I fired the little rig up and spanned the VFO which tuned from ~ 3721 to 3738 KHz. I then proceeded to tune 3729, the frequency of BCEN, our SSB provincial public service net and on my first break was able to check in with one call sign repeat to the net control station. The band conditions were noisy and most signals were S8 or lower, however Wee Willy's 1.5 watts P.E.P. were able to check me in with my folded Marconi antenna. I later changed the L1 slug and worked some stations higher up the band. My audio reports were favorible and no one knew I was running DSB. This is a fun radio and it looks cute to boot!

When transmitting, I had to be careful to keep my hand away from the VFO compartment to prevent pulling the VFO frequency with the capacitance change from my hand. The VFO has reasonable long term frequency stability and copying CW stations with the receiver was possible without frequent tuning readjustments.

The following are some digital photos of Wee Willy taken by VE7ZAC.

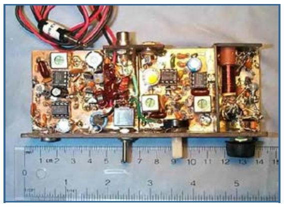
Move your mouse over the images for a larger version.

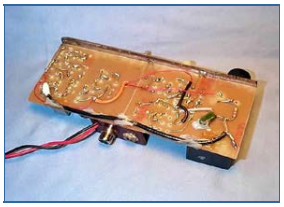
From left to right the electret mic, TX/RX switch, RF gain control and VFO tuning control. The number 375 is Dick's project identification number. He has given me many schematics of his projects and each has a unique number and case color. For the non-Canadians, the large coin on top of Willy is our "Loonie" a 1 dollar coin. The large 2 color coin in front of the rig is our "Toonie" a 2 dollar coin.

The hole on the left of this rear panel shot is for adjusting the L1 slug for the VFO. From wherever on the band the VFO is set a front panel tuning range of about 17 KHz was possible. Also shown are the speaker jack , BNC antenna connection and DC power leads.

Wee Willy with the cover off.

These top view shots clearly shows the 3 distinct compartments. From left to right the TX , RX and VFO sections. The VFO has a PC board seperator for shielding. Dick's construction method is well illustrated with this photo. The IC's are in sockets. The 3 VN10s have a small tab on the top. There are no heat sinks on the VN10 finals and they do require any for a 6 volt supply voltage. The rear panel jacks and VFO inductor are all mounted in PC board material.





Here is a bottom shot showing the connecting wiring through the chamfered PC board holes. Many thanks to Dick Pattinson, VE7GC for allowing me to present one of his projects.

Wee Wee Willy by NM8T

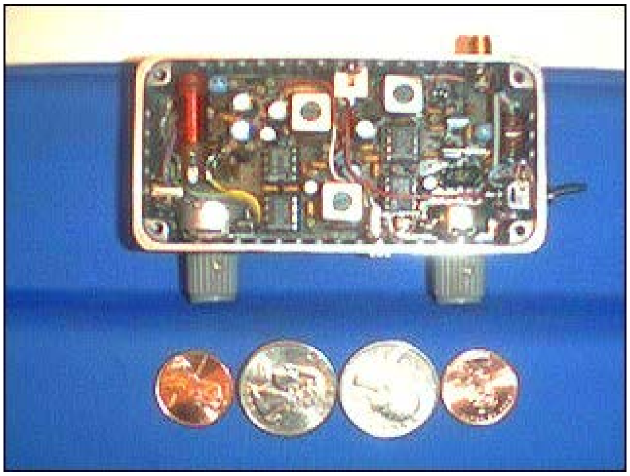
NM8T Builds Wee Wee Willy

Steve White, NM8T emailed me a few weeks ago that he was building Wee Willy and was not getting the expected output power. I forwarded his email to Dick and they problem solved the issue. It turns out that the V10KM's were the culprit.

Here are their final two emails:

Steve: I think you have found a solution to your problem. I did not know that VN10's are different. I bought mine in 1994 from a radio parts supplier and they are marked VN10KM F324A1. Have fun with your new rig and let us know how you are doing with it. 73, Dick

Dick, I have completed my little rig and have had a blast with it so far. My first CQ yielded me 3 contacts in Pittsburg, Pa. My friend across town gave me a 20 over S9 report. The three hams in Pa. were astounded that I had only 1.5 watt PEP. I had to use 12 volts and used small 5 volt regulators to feed the NE602's. This works very well. I even used a little homebrew tuner with it and got my swr flat. I am going to take the little rig to Dayton Hamvention next year. I go every year and take a little project each year to show everyone. I hope it is ok to call my little rig Wee Wee Willy, since it is a little smaller than your version. I want to thank you for your help in getting this rig completed and also thank Todd for his help in getting me in contact with you. Your little rig is very neat and right up my alley for a QRP rig. I look forward to working you someday on QRP. I'm 47yrs. old on the 24th of this month. I have a whole room full of QRP rigs such as HW-7, HW-8, HW-9, Argonaut 509, 515, Powermite PM-3a. Have built a Cascade, LCK, Spider, MXM, Keyers, Tuners, and all kind of other projects. I love QRP and hope to build many more soon. The Wee Willy is one of my favorite rigs. Here are some pictures for you and Todd. Thanks and Hpe CU Agn. 73's NM8T (No more 8 Tracks) Steve White - Fayetteville, WV



My special thanks to Steve White and Dick Pattinson.

Countless builders have constructed the Wee Willy on several bands. Here are 2 more examples. Paul, KE7HR Link1 and Link 2. The finals even made it into the the BITX17 transceiver. Allan Yates published some good work on the final amp.

I've also tweaked Dick's design, but decided to leave his little rig and this web page alone out of respect to him. It's always easier to improve rather than actually *design* something. I'll never better the experiences I had operating and documenting his transceiver.

Update February 15, 2011

I lost touch with Dick and decided to call him on Feb 10, 2011. He's 95 years old and no longer building. Sadly, his wife Christina died 2 years ago. She was a beautiful person and a gifted painter. Dick remains astonished that Wee Willy - Project 375 QRP Radiophone has been built by at least 50 people (that I know about) world wide, the subject of blogs and the basis/inspiration for numerous radio projects. I met Dick on 80 meters and my wife and I once visited his residence and spent 2 glorious, days on Saltspring Island with he and his wife.

Dick sent me the radio and documentation on Aug 13, 1998. He was a hardcore builder and a professional electronics technician who, for a time, flew his float plane to service equipment in the Gulf Islands of British Columbia. Dick keeps many binders full of his carefully numbered and well documented projects. He gave me a few sheets for my reference library as a gift. Dick's work inspired me early on and I cherish the time I had Wee Willy in my radio shack - blown away by DSB, QRP and such a novel little radio.

Dick asked me to send his greetings to all QRP Homebuilders everywhere.

More Updates

Aug 25, 2011

Hi. Been viewing your web site for a while, lot of great information for builders. I thought i would let you know that I built the "Wee Willy", it works great, so good that I decided to try building one for 20m. I built the 20m version with a 14.318 crystal instead of a VFO ideal for my local net, also changed the PA circuit using IRF510 giving me 2.5W output, the transistor can run 5W, but it does get rather hot, I must say that the quality of the transmitted audio is amazing, everyone comments how good it is and some do not believe it is home-brew!!! I have put a few CQ calls out with a simple dipole and made contacts into Belgium, Netherlands, Finland and Saudi Arabia. Many thanks again for putting the Wee Willy on your website, it has opened my eyes to how well you get get out with QRP power.

Regards Wayne. M0WAY. Click for Wayne's fabulous Wee Willy bread board.

Sept 19, 2011

Wayne, M0WAY wrote again,

I built another Wee Willy and put it into a box, but with a few extra bits:

A new VFO was added that runs 14.190 – 14.350 MHz (an 11 MHz crystal oscillator + a 3.2 MHz VFO + a band-pass filter) I got the power up to 4W, however the audio was a little over powered from the extra RF, so I had to reduce the audio gain and added an antenna switching circuit, after I burnt out my receive circuit with the 4W output.

Here's Wayne's PA (from the NE602 on ** updated July 12, 2012 **) Click for a photo of his latest incarnation. Thanks again!

Oct 19, 2011

Mark, WA4JAT built a "0 cost" Wee Willy from 100% junk box parts including a CB radio transistor for the final. Click and click. Mark tunes the VFO using an air variable capacitor mounted in the former Heath Kit signal tracer project chassis. The cable running out of the front is the microphone coax with an electret mic at the end. Employing large size VFO, Rx and Tx boards facilitate easy future modifications. Awesome work Mark!

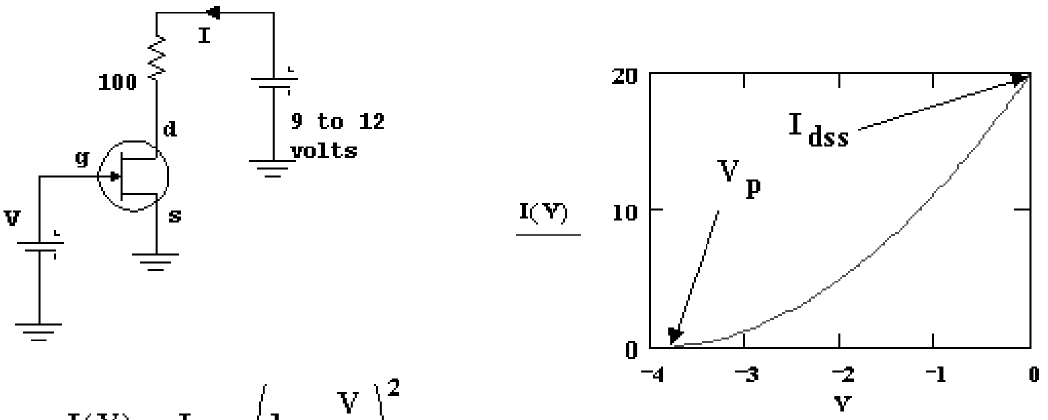
RF — Test and Measurement

JFET BIASING TUTORIAL BY W7ZOI

JFET Biasing and an Amplifier Design   W7ZOI, 30Dec00.

Junction Field Effect Transistors (JFETs) are generally no more difficult to bias than a bipolar transistor. In many ways they behave essentially like a classic triode vacuum tube. The current flowing between FET drain and source is controlled by the gate-to-source voltage, V_{GS}. (This is the voltage you would measure with a voltmeter connected between the gate and the source.)

The Fig. 1 schematic defines the parameters we use to describe the DC behavior of a JFET. The 100 Ohm resistor is added to preserve UHF stability during measurements. The gate voltage, V, is then varied while measuring drain current. The results are then plotted, as shown in Fig. 1. The FET is characterized by the simple quadratic equation below the schematic with the two FET related parameters of I_{DSS} and V_P. I_{DSS} is the drain current when the gate is short circuited to the source. The pinchoff voltage, V_P, is V_{GS} when the drain current is essentially zero. The example in Fig. 1 has I_{DSS}=20 mA and V_P=-4 volts. Source and drain current are identical in the JFET.



$$I(V) := I_{dss} \cdot \left(1 - \frac{V}{V_p}\right)^2$$

$$I_{dss} := 20 \text{ mA} \qquad V_p := -4 \text{ volts}$$

Figure 1

Basic behavior of an N-Channel depletion mode JFET.
The numbers shown basically illustrate the ideas.

The experimenter wishing to use a JFET in a circuit will most often use the method called "self biasing" where a resistor is placed in the source lead. The gate is grounded. The FET current also flows in the resistor. The current generates a voltage drop that elevates the source voltage above that of the gate, which is the same as biasing the gate negative with respect to the source. This establishes a drain current less than I_{DSS}.

Drain voltage is not critical so long as it is a few volts greater than the source.

The resistor value for a desired bias is calculated with Equation 1 if I_{DSS} and V_P are known. But these parameters are rarely well specified. For example, the popular J310 is characterized only by ranges: I_{DSS} is 24 to 60 mA and V_P is -2 to -6.5 volts.

$$R_s := \frac{V_p}{I_D} \cdot \left(\sqrt{\frac{I_D}{I_{dss}}} - 1\right) \qquad \text{Eq.1}$$

I_{DSS} and V_P are easily measured for a specific FET with the circuit shown in Fig. 2. The circuit is built with a variety of source resistor values and the source voltage is then measured for each with a high impedance voltmeter. The pinchoff voltage will then be the negative of the source voltage when R_S is a large value such as 100K. Grounding the source through a current meter generates a direct reading of I_{DSS}. This brief measurement will provide enough data to allow biasing.

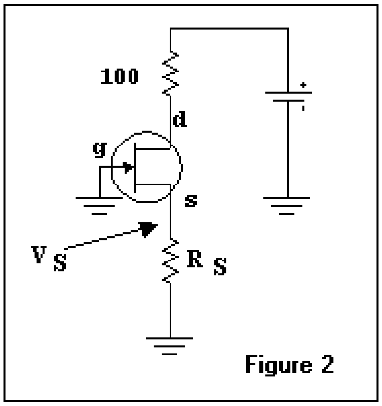


Figure 2

Circuit used to determine FET DC parameters.

This experiment was performed with a junk box 2N5454, producing the parameters I_{DSS}=11 mA and V_P=-2.8 volts. The current equation from Fig. 1 is plotted in Fig. 3 with these parameters. Additional points were measured producing points close to the curve.

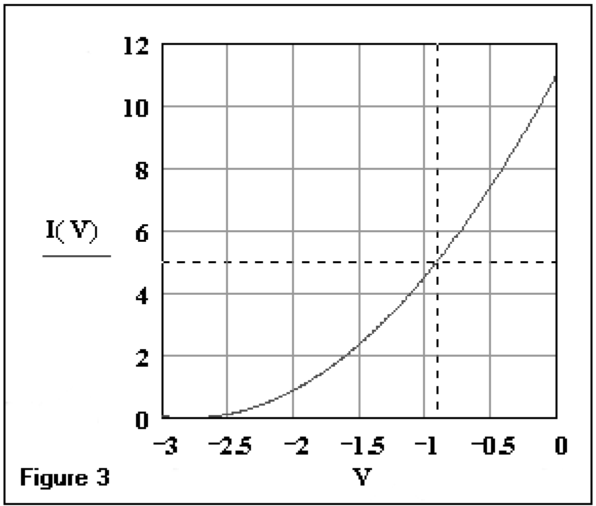


Figure 3

Data and smooth curve for a 2N5454 we measured.
This FET has a pinch off of -2.8 volts and a IDSS of 11 mA.

This exercise SHOULD be done with virtually every FET that the experimenter uses. It is not enough to obtain data for a given FET type, although if a group of parts are available from one manufacturing batch, moderate uniformity might be expected.

We mentioned the J310 earlier with an I_{DSS} spread from 24 to 60 mA. That 2.5:1 span range is one of the better of commonly available parts. The historically popular MPF-102 has a 10:1 spread in I_{DSS} (2 to 20 mA) with V_P extending from -0.5 to -7.5 volts! Clearly, this part should be used only with extreme care. It's a good RF part, but one MPF-102 may not resemble the next.

Assume the FET we summarized in Fig. 3 was to be used in an amplifier and that we wished to bias the FET for 5 mA. The 5 mA point can be picked on the graph and matches V_{GS} of approximately -0.9 volt. 0.9 volt drop from 5 mA matches a resistor of 180 Ohms. Application of Eq. 1 produces 182 Ohms. Crosshairs indicating the 5 mA point are included in Fig. 3.

Once a FET has been biased, it can be used in a small-signal amplifier. The significant gain parameter is the transconductance, given by Eq. 2. Transconductance is the drain signal current flowing for each volt of signal applied to the gate. Note that g_M depends upon the bias current. The g_M value given in data sheets corresponds to operation at I_{dss}.

$$g_m = 2\cdot\frac{I_{dss}}{V_p}\cdot\left(1-\frac{V_{GS}}{V_p}\right) \qquad \text{Eq.2}$$

$$I_{dss} := .011 \quad V_p := -2.8 \quad V_{GS} := -0.9$$

$$g_m := 2\cdot\frac{I_{dss}}{V_p}\cdot\left(1-\frac{V_{GS}}{V_p}\right) \qquad g_m = -5.33163\cdot 10^{-3}$$

Figure 4

Data used to produce transconductance for the FET used in our sample amplifier.

Evaluation of g_M from Eq. 2 for our 5 mA bias produces the data in Fig. 4.

Consider the amplifier shown in Fig. 5. We will use this as an output buffer for an oscillator. Assume that we have tapped a signal from the oscillator tank where a voltage of 1 volt peak-to-peak is available. When this is applied to the FET gate, a current of 5.33 mA pk-pk will flow. The transformer uses a 15 turn drain winding and a 3 turn output link on a FT37-43 ferrite toroid. This turns ratio transforms the 50 Ohm load to appear as 1250 Ohms in the drain line. A current of 5.3 mA pk-pk flowing here produces a voltage of 6.25 V pk-pk. The no-signal drain voltage will be about 11 volts, so positive and negative 3 volt excursions will not cause voltage limiting. The current will vary from the 5 mA DC value down to nearly 3 mA (still well above zero) and up to nearly 8 mA, still below I_{dss}. The signal at the 50 Ohm load is reduced by the transformer to 1.33 V pk-pk. This is about +6 dBm, enough power to drive a diode ring mixer. This simple circuit is a low noise, efficient, and very useful amplifier.

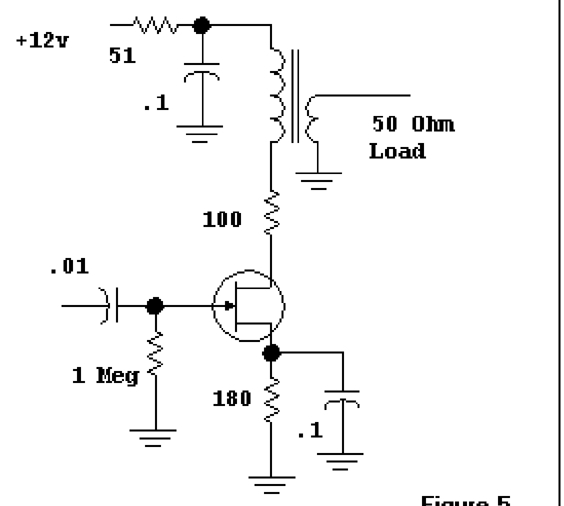


Figure 5

Common source amplifier biased for 5 mA drain current.

Figure 6 shows a FET amplifier that uses "long tail" biasing. A negative power supply is now used, forcing the biasing resistor to behave like a constant current source. This method can accommodate a wide range of FETs of a given type if the bias is set up for $I_d<I_{dss}$-min.

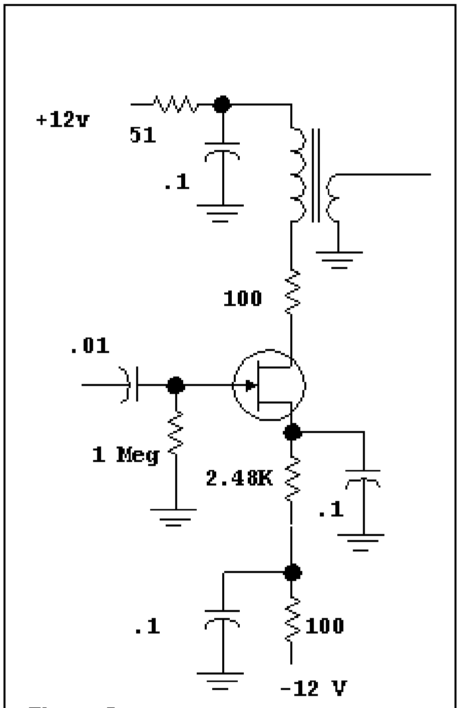


Figure 6

Amplifier with "long tail" biasing. This amplifier is biased for 5 mA and is identical in performance to that of Figure 5, but does not require the careful device characterization.

EMRFD Review

Experimental Methods in RF Design

First published by the ARRL in 2003, EMRFD serves as the main QRP/SWL Homebuilder site reference.

Written by Wes Hayward, W7ZOI, Rick Campbell, KK7B and Bob Larkin, W7PUA, EMRFD is the follow-on to the 1977 ARRL publication, Solid State Design For The Radio Amateur (SSD).

A treasure trove of narrative and tabled information, schematics, references and photographs adorn this lengthy (512 page) book that comes with a data CD filled with software and key reference papers. With the included CD, EMRFD is about twice the size of the original SSD.

Wes and crew emphasize fundamental radio equipment design and bench testing rather than providing the usual catalog of circuits to just casually copy and not carefully examine.

Since we amateur builders own varied skills, abilities and test equipment, some sections may intimidate readers, while others may inspire and drive your bench practices to a higher level of competence and joy. The heart of RF design lies in measurement and reason: EMRFD emphasizes this and in doing so alienates some readers.

For some, homebrew radio electronics and kit building are synonymous — plenty of kits are sold to builders who chose to build someone else's design, rather than capture their own ideas on a scrap of copper board, or at least, to modify their equipment to suit their needs. EMRFD may appeal to builders who enjoy learning about RF *design*, or want more innovative and creative bench experiences — 1 stage at a time.

Although stressing that *build and measure* = a proven way to improve in this hobby, Wes and team share other pearls including bread boarding techniques, parts lay out, hot parts and pitfalls to avoid.

The first chapter is simply called Getting Started. On page 8, Wes shares his first simple receiver design — this page starts your breadboarding in haste! EMRFD features information and designs for all levels of experimenters, although basic electronics knowledge and some experience are needed to get the most out of its content.

Replete with sidebar examples about measuring or calculating data for common circuits, EMRFD also offers general purpose stages including a universal monoband superhet receiver front end or AF chain instead of less versatile, single- application circuits. The design information feels vibrant and flexible and some of the material is original, or presented in a way that adds to the existing amateur radio knowledge base. For example, new front-end mute circuits, or the cascode JFET mixer and RF amplifier.

Wes also shares some new ideas for RF and IF amplifiers ranging from simple to state-of-the-art. This book has something for everyone — I prefer to describe EMRFD's influence on our hobby in simple action verbs such as: improve, innovate, inspire, explore and transform.

Software:

The Microsoft Windows programs run on everything up to and including Windows 7. While a few programs are new, others represent updates of the historic W7ZOI Ladpac applications. Upon mastery of the Ladpac program suite, you'll generate the needed stages to boost *your* designs.

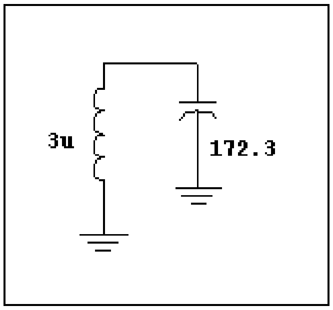
For example you can design complex double or triple tuned band-pass filters for your projects without any math — the software gives you the series capacitor values so you need not worry about putting links on your inductors to match input or output impedances.

EMRFD = the **must have** reference book for your homebrew workshop.

ARRL EMRFD web link

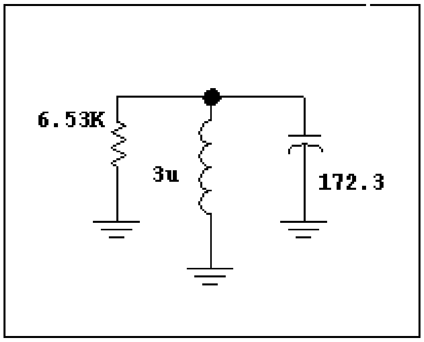
Link to the W7ZOI errata page for EMRFD

Tapped Capacitor Impedance Transformation in LC Bandpass Filters



We often use a pair of capacitors to match impedances at the termination ends of LC bandpass filters. The circuit consists of a shunt capacitor at the termination followed by a series capacitor connecting to the high Z end of a parallel tuned circuit. Some readers have asked about how the capacitors are picked. Although there is considerable flexibility in some of the choices, it is not empirical as some have guessed. A simplified double tuned circuit design sequence is presented in the sidebar on page 3.14 of EMRFD and this is the beginning of the analysis used in the programs DTC.exe and TTC.exe contained on the EMRFD CD. I'll not go into too much detail here, for it's in EMRFD, beginning on page 3.8, and in chapter 3 of IRFD. Here is what happens in the programs:

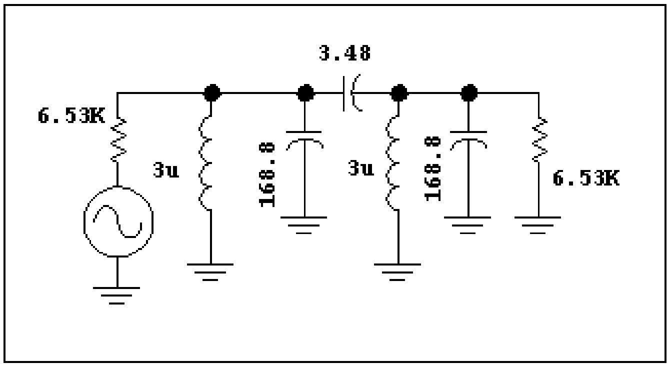
Bandpass filter design begins with an almost arbitrary choice of inductor. We pick 3 uH for a 7 MHz double tuned bandpass filter that we will use to illustrate the ideas. This L resonates at 7 MHz with 172.3 pF.



A bandwidth and center frequency are picked for the filter. This establishes a filter Q. The Q of an end section is then determined by the desired filter shape (Butterworth, etc)Let's say we want to do a 7 MHz center frequency Butterworth filter with BW=0.2 MHz. Filter Q is then 35=7/0.2. QE will then be 35x(root(2))= 49.5. The QE value in the sidebar (p3.14) includes the effect of finite inductor Q. Assume a lossless inductor for this example.

So what does this mean? It means that the end tuned circuit, when not coupled to the rest of the filter, needs to be set up to have a Q of 49.5. This is experimentally significant. (ref: QST, Dec, 1991). The reactance at 7 MHz of our 3 uH inductor is X=131.9 Ohms, so we need to load each end with a resistance R=QX=6.53K-Ohm.

What this means is that we would realize our double tuned bandpass filter with center frequency of 7 MHz and bandwidth of 200 kHz if we terminated a simple double tuned circuit in 6.5K at each end. The filter is shown below where we have used additional equations (EMRFD p 3.14) to calculate the coupling and tuning capacitors.



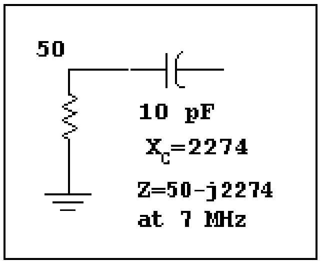
This is a useable filter design, for it will generate the desired shape and bandwidth. But, it is not very practical; it does not fit in our low impedance world.

The filter can be redesigned. One classic, but usually impractical solution is to scale the filter to lower impedance levels. For example, if we dropped L from 3 uH to 23 nH, we could directly load our filter with 50 Ohms and get the required end section Q. But this is not at all practical. First, it's difficult to build inductors with L this small and still have reasonable Q at 7 MHz. Second, the parasitic inductance of the rather large (high C) capacitors that we would attach to this inductor would begin to compare to 23 nH. A better re-design would use transformation circuits, schemes that will let us use a 50 Ohm termination (or whatever we need) and make it function as a 6.5K resistance when seen by the inductor.

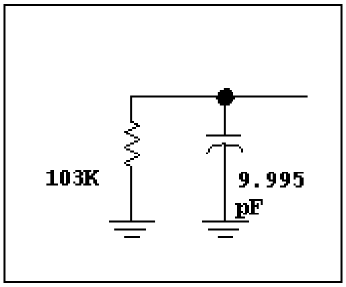
One such scheme is a transformer. This could be realized with ferrite cores or with links inductors wound on the existing 3 microHenry parts. But link coupling with design precision is a challenge of it's own. Conventional two and three element transforming networks (L, pi, Tee) are also suitable.

The simplest transforming circuit uses a series capacitor. Let’s do some analysis to see how this works:

We have arbitrarily picked a 10 pF capacitor to illustrate the idea. At 7 MHz, the reactance of a 10 pF capacitor is 2274 Ohms. Hence, the complex impedance of the 50 Ohm resistance and the series capacitor is Z=50-j2274. The impedance transformation behavior of this circuit is studied by transforming the series impedance to a parallel admittance. Recall that Y=1/Z. So Y=1/(50-j2274). The result is Y=(G+jB)=9.67E-6 + j4.4E-4. Of special importance is the resistive real part of the admittance.



The 10 pF series capacitor in our example makes the 50 Ohm resistor look like a 103 K-Ohm resistor in parallel with a capacitor that is nearly 10 pF. The general case causes a R0 value resistor to look like a value of Rp with a series capacitor with reactance given by

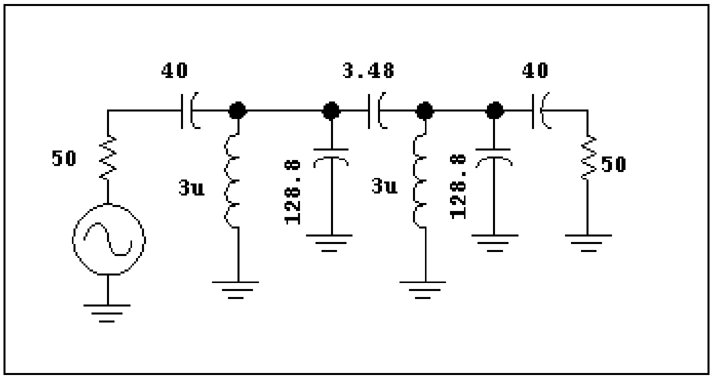


This equation is #3.1-5 from IRFD where it is derived.

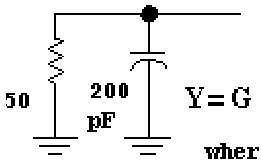
$$X_C = \sqrt{R_p \cdot R_0 - R_0^2} \qquad \text{Eq. AA}$$

Our original example filter needed a parallel resistance, Rp, of 6.5K-Ohms, which is produced by a series capacitor with a reactance of 568 Ohms. This is a capacitor of 40 pF. Another version of our double tuned bandpass filter is then

Notice that the tuning capacitors, the elements across the inductor, have dropped as C is added at the ends.



A capacitor in series with a resistance produces higher equivalent resistance. Consider a parallel combination. Here we pick a value of 200 pF as the parallel C and calculate the admittance. This is then converted to an impedance with Z=1/Y and the individual components are evaluated. This result is shown to the right.

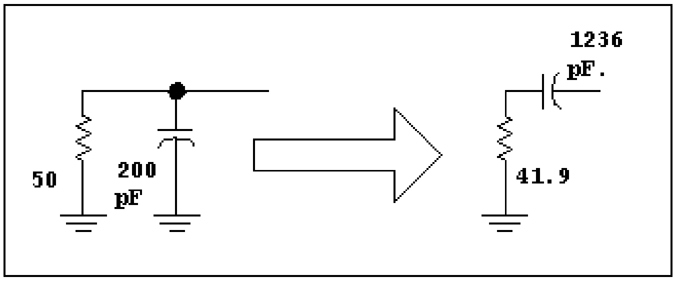


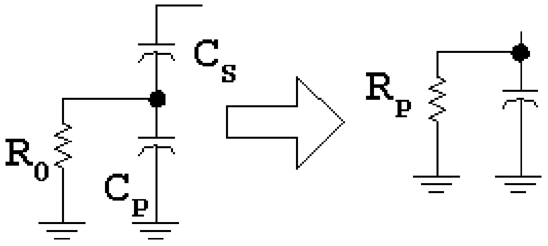
$$Y = G + j \cdot B$$

where

$$G = \frac{1}{R} \quad \text{and} \quad B = \omega \cdot C$$

The parallel capacitor transforms the 50 Ohms to behave like a lower value.
In this case, we obtain about 42 Ohms in series with a 1236 pF capacitor.





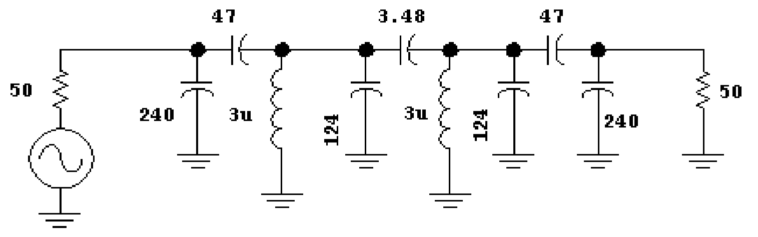
$$C_p = \sqrt{\frac{R_p \cdot C_s^2}{R_0} - \frac{1}{R_0^2 \cdot \omega^2}} - C_s$$

Eq. BB

A series capacitor transforms a termination to “look like” a higher resistance while a parallel capacitor “transforms to” a lower R. Clearly, the combination of the two can generate about any result we need, realizing that they will also produce reactance that must be absorbed into the existing tuned circuits. The mathematics (now symbolic and not just number manipulation) is messy, but not difficult. A resistive termination R0 (50 Ohms or whatever) is paralleled by a capacitor Cp. The admittance is calculated and is converted to an impedance. The impedance of a series capacitor, Cs, is then added. The result is converted back to an admittance. The resistive real part is extracted and inverted to yield Rp. This expression is then solved to yield a design equation:

This is the equation used in the programs. A related expression provides the equivalent capacitance, needed to calculate the tuning capacitor for each resonator.

If we use the program with the center frequency and bandwidth presented earlier, we find that the minimum allowed series capacitance is 40 pF. If we then insert a value of 47 pF, we see that parallel 240 pF capacitors are needed to properly load the resonators. This variation is shown to the right. Circuits using the capacitor tap are practical, for they allow existing junk box parts to be used. The topology has little other utility, offering virtually the same response as a filter using only series capacitors for loading.



Useful link shared by Tim, KE7VYD on Feb 4, 2014. Click

Amateur and Short Wave Radio Electronics Experimenter's Web Site

More Active Antenna Experiments

Introduction

Many builders emailed me requesting a simple, broadband VPA (voltage probe antenna) design with more power gain than the common gate versions I have presented elsewhere on this web site. Connecting a whip antenna to a cascode JFET stage described by W7ZOI in Experimental Methods in RF Design is 1 method I considered.

I built the version shown in Figure 1 almost 2 years ago. This VPA. although more powerful, overloaded the front end of my test receiver with multiple RF signals. Clearly some tuning on the input was needed.

Experimental Broadband Cascode JFET VPA

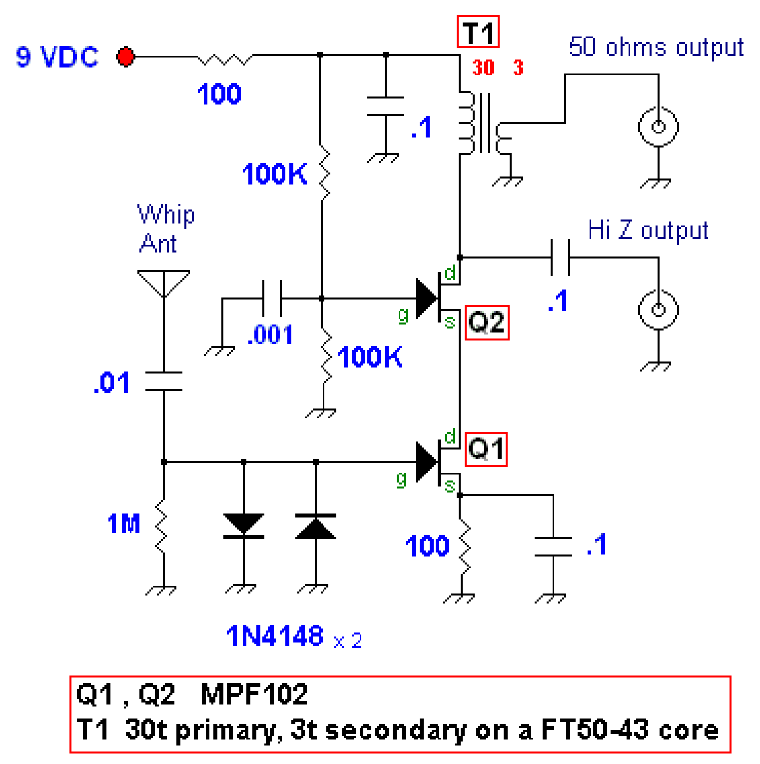
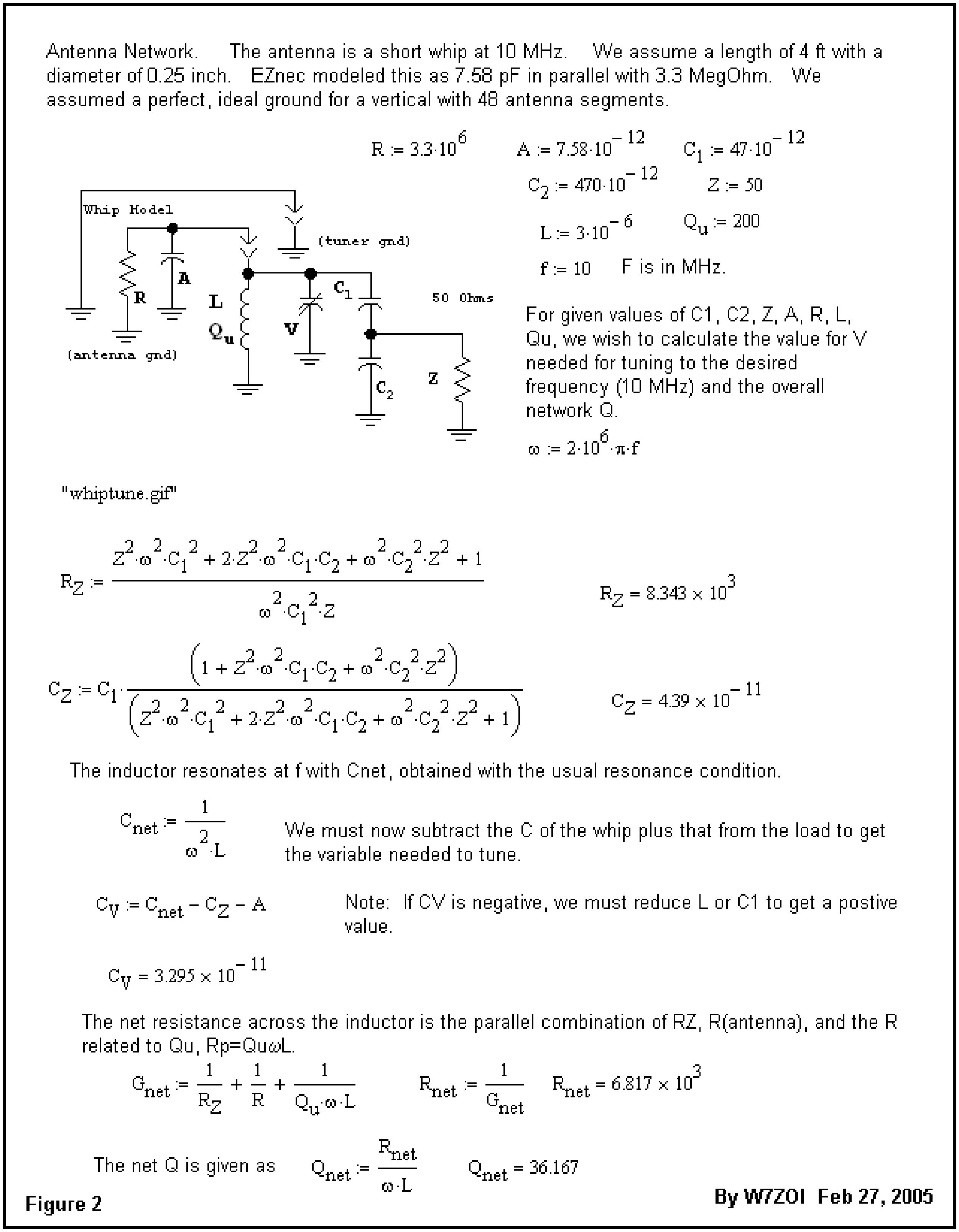


Figure 1

Feb 2005 by VE7BPO

The Tuned Whip

Previous experimentation confirmed that it is easy to tune a short whip antenna by connecting it to the hot end of an L C (inductor and capacitor) tank circuit. The high impedance whip antenna was "matched" to a JFET RF amplifier by placing a high value (1 megohm or greater) on the JFET gate to ground. Although this method is practical, I desired a network to transform the output impedance of the tuned whip tank tank circuit to a known impedance. I do not possess the knowledge or mathematical skill to design such a network and asked Wes Hayward if he might consider doing this for me. My desired parameters for the network were 10.0 MHz, a 50 ohm output impedance and a 4 foot (122 cm) whip. Please refer to Wes' calculations and schematic in Figure 2 below. This math is difficult, however, a practical design for experimentation is provided.



Antenna Network. The antenna is a short whip at 10 MHz. We assume a length of 4 ft with a diameter of 0.25 inch. EZnec modeled this as 7.58 pF in parallel with 3.3 MegOhm. We assumed a perfect, ideal ground for a vertical with 48 antenna segments.

$R := 3.3\cdot10^{6}$ $A := 7.58\cdot10^{-12}$ $C_1 := 47\cdot10^{-12}$

$C_2 := 470\cdot10^{-12}$ $Z := 50$

$L := 3\cdot10^{-6}$ $Q_u := 200$

$f := 10$ F is in MHz.

For given values of C1, C2, Z, A, R, L, Qu, we wish to calculate the value for V needed for tuning to the desired frequency (10 MHz) and the overall network Q.

$\omega := 2\cdot10^{6}\cdot\pi\cdot f$

"whiptune.gif"

$$R_Z := \frac{Z^2\cdot\omega^2\cdot C_1{}^2 + 2\cdot Z^2\cdot\omega^2\cdot C_1\cdot C_2 + \omega^2\cdot C_2{}^2\cdot Z^2 + 1}{\omega^2\cdot C_1{}^2\cdot Z} \qquad R_Z = 8.343\times10^{3}$$

$$C_Z := C_1\cdot\frac{\left(1 + Z^2\cdot\omega^2\cdot C_1\cdot C_2 + \omega^2\cdot C_2{}^2\cdot Z^2\right)}{\left(Z^2\cdot\omega^2\cdot C_1{}^2 + 2\cdot Z^2\cdot\omega^2\cdot C_1\cdot C_2 + \omega^2\cdot C_2{}^2\cdot Z^2 + 1\right)} \qquad C_Z = 4.39\times10^{-11}$$

The inductor resonates at f with Cnet, obtained with the usual resonance condition.

$C_{net} := \dfrac{1}{\omega^2\cdot L}$ We must now subtract the C of the whip plus that from the load to get the variable needed to tune.

$C_V := C_{net} - C_Z - A$ Note: If CV is negative, we must reduce L or C1 to get a postive value.

$C_V = 3.295\times10^{-11}$

The net resistance across the inductor is the parallel combination of RZ, R(antenna), and the R related to Qu, Rp=Quω L.

$G_{net} := \dfrac{1}{R_Z} + \dfrac{1}{R} + \dfrac{1}{Q_u\cdot\omega\cdot L}$ $R_{net} := \dfrac{1}{G_{net}}$ $R_{net} = 6.817\times10^{3}$

The net Q is given as $Q_{net} := \dfrac{R_{net}}{\omega\cdot L}$ $Q_{net} = 36.167$

Figure 2

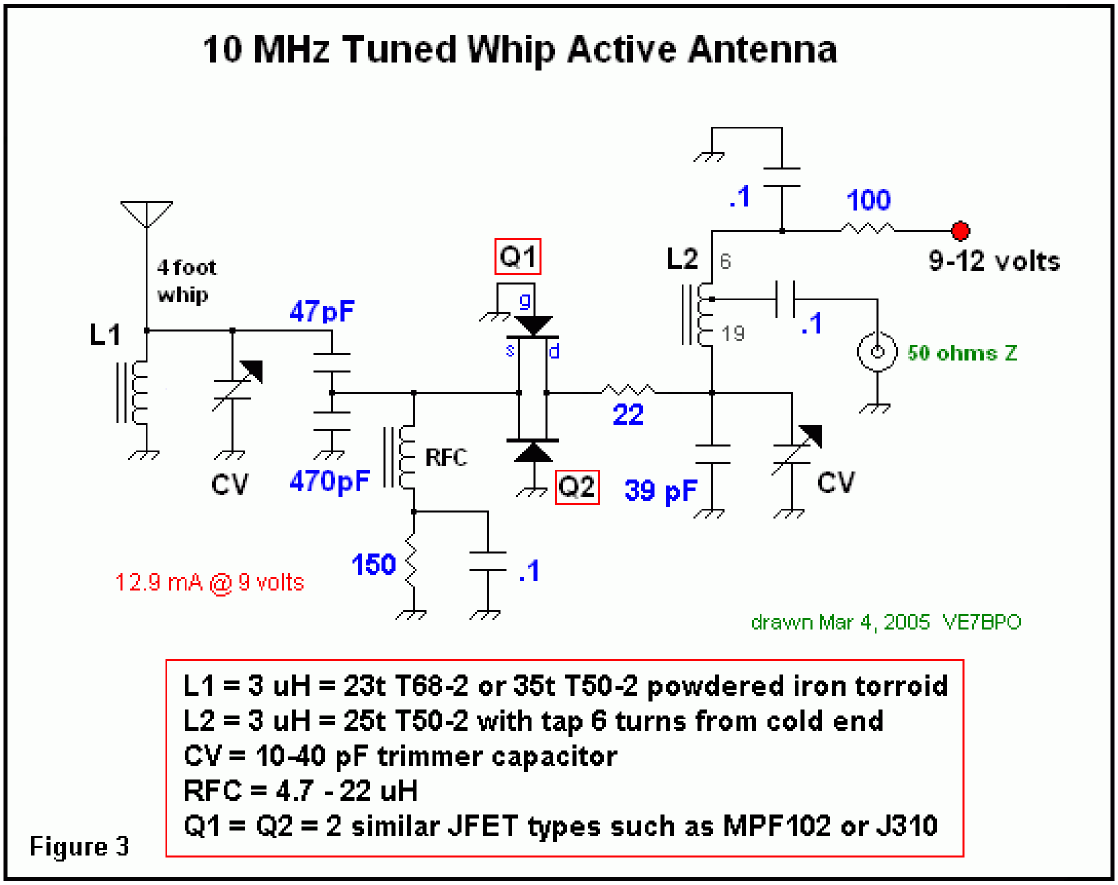
By W7ZOI Feb 27, 2005

Common Gate Amplifier Version

I built the circuit shown in Figure 3 and Figure 4 and tested it on a medium grade SWL receiver (Realistic DX 300), rather than an expensive Amateur Radio receiver. I required this active antenna for experimenting with a 10 MHz WWV superheterodyne receiver I am designing. For practical analysis, VE7TW and I did listening tests with a commercially made 4 foot telescopic whip antenna that is fitted to a standard PL-259 connector (Figure 5) and his deluxe multi-band commercial SWL antenna up a 25 foot tower.

The Figure 3 VPA was very quiet
and pulled in WWV much better

than the plain 4 foot whip of Figure 5, however, received station signal strength was quite weak when compared to the outside antenna. Our conclusion was that considering the significant losses of the 4 foot whip antenna it was connected to, the common gate RF amp does not likely have enough voltage gain to please most builders. This amp did present a low impedance to the whip network and no spurious oscillation were measured on the bench. Do not omit the 22 ohm or similar value resistor in the drain of the FET. It is used to to push the UHF parasitic oscillation tendency into the ground. Such oscillations will trash your receiver mixer intermodulation performance.



L1 = 3 uH = 23t T68-2 or 35t T50-2 powdered iron torroid
L2 = 3 uH = 25t T50-2 with tap 6 turns from cold end
CV = 10-40 pF trimmer capacitor
RFC = 4.7 - 22 uH
Q1 = Q2 = 2 similar JFET types such as MPF102 or J310

Figure 3

The 150 ohm source resistor can be increased in value and/or 1 of the JFETs removed if you wish to reduce the current draw on a 9 volt battery. It might be better to substitute 1 better JFET such as the J310 rather than use the "popcorn" MPF102 as shown. This VPA may be practical for a receiver that has an existing broadband RF preamplifier. The tap on L2 was found experimentally and the output impedance is probably higher than 50 ohms, but is likely a reasonable low impedance match to most receiver front ends. A broadband transformer for L2 might also be a good choice.

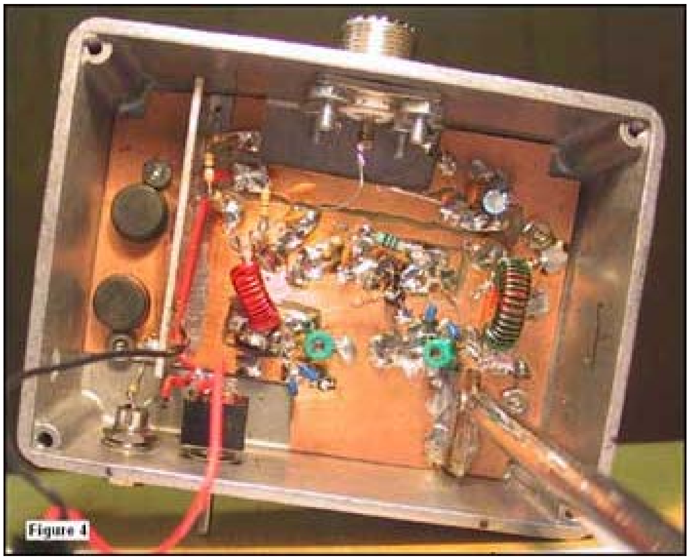


Figure 4


Figure 5

Figure 4 above: For the whip network, it is critical that you use a inductor that has an unloaded Q of 200 or above. Practically speaking, this means you cannot use a fixed value inductor such as an epoxy coated or molded RF choke. Use a powdered iron torroid instead. In my test VPA designs, for L1, I used a T68-2 core wound with (the green) 22 gauge enamel coated wire to get as high an unloaded Q as possible. If you use the T50-2 core, use 24 gauge wire if possible. Higher Q = lower losses.

Figure 5 above: This is a 4 foot whip antenna factory connected to a PL-259 that came with the Realistic DX300. It presents a very high impedance to the test receiver front end and probably wasn't a good choice to compare the VPA designs to.

Cascode JFET Amplifier Version

It was decided to use a cascode JFET amplifier to obtain more power gain. The whip network was changed to try to match the 10K input of the JFET amplifier shown in Figure 6. The whip network capacitor values (150 and 33 pF) were calculated to the best of my ability. This amplifier was tested in the same manner as the Figure 3 design. It worked very well. The WWV signal that morning was not very strong and could not even be heard with the plain 4 foot whip. The signal strength of the tuned whip was just below that of the outside antenna. The outside antenna was much quieter however and had less fading. The tuned whip antenna was quite noisy in comparison. The high gain RF amp brought up the strength of the environmental noise sources in the house. The RF gain of the receiver was reduced to compensate for the added noise.

Another problem was noted with this and other tuned drain versions of the cascode JFET amplifier; instability.

Recently, I connected a tuned drain version of the Figure 1 VPA to a receiver that contained a tuned input stage and was able to measure oscillations in the VPA with my scope. The FET drain tank and the tuned input amp seemed to be interacting.

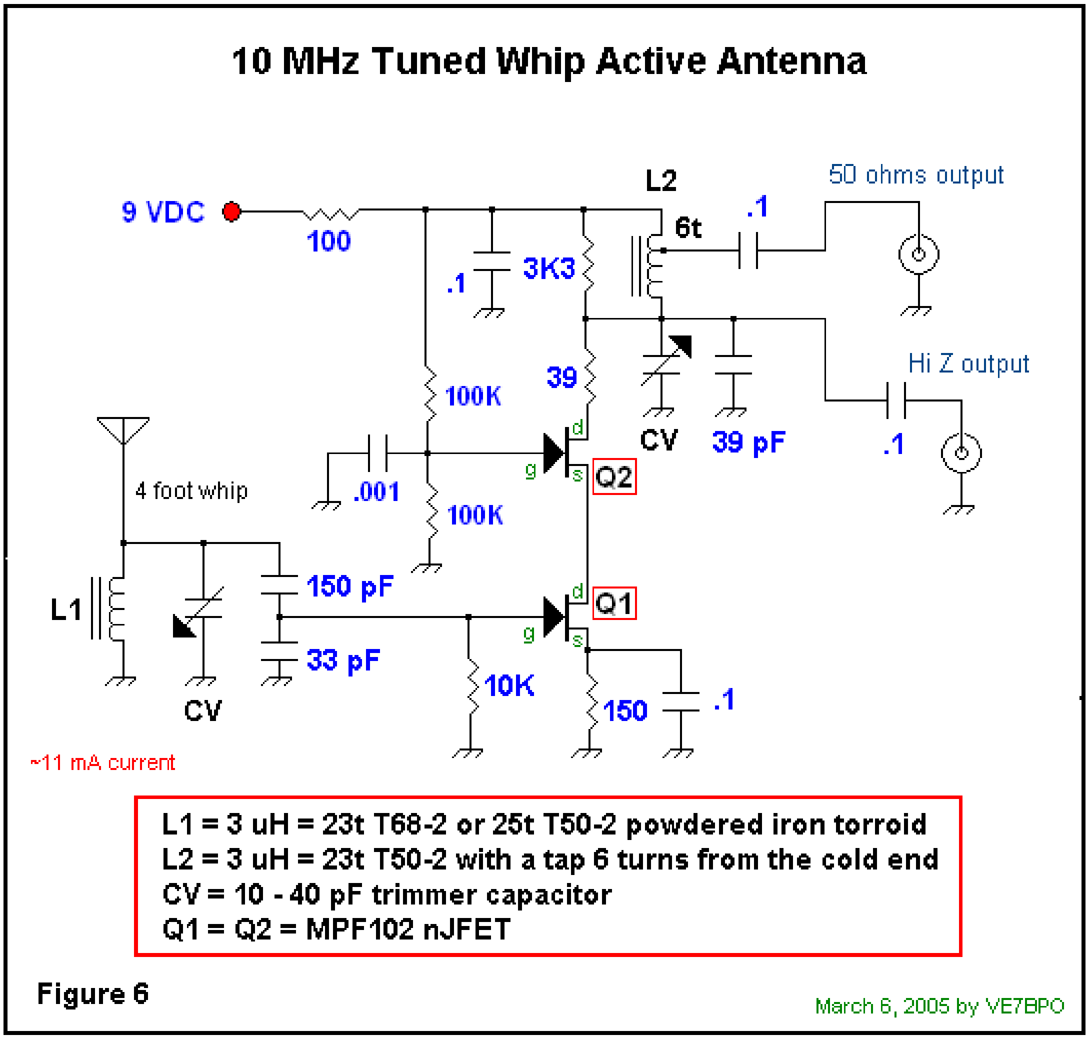


Figure 6

A "swamping" resistor was placed across the VPA drain tank circuit. I had to use a resistor value of less than 1200 ohms to eliminate this instability. This greatly reduced the gain and selectivity advantage of a tuned output and I realized that output tuning may be impractical for many reasons. Some SWL builders use regenerative receivers and such a problem would be disastrous. I sent the Figure 6 schematic to Wes Hayward and he suggested using a broad or wide band amplifier as shown in Figure 8.

Instability can also occur in broadband output versions and a swamping resistor is still necessary but is used mostly to force an output impedance so that a transformer can be designed.

All of the cascode JFET amplifiers shown have fixed bias on Q2. Variable gain is possible by changing the bias voltage on Q2 with a voltage divider and/or modifying the amplifier circuit to give a greater range of bias controlled voltage gain. Please refer to EMRFD page 6.17 for information regarding this. A switchable resistor attenuator might also be practical for some builders.

Figure 6 above: The tuned whip network is connected to a cascode JFET amplifier. A dual gate MOSFET would also be a great choice. I have many on hand, but chose the cascode JFET topology because many builders no longer have access to these devices or prefer not to use the more available surface mount types. They are also more expensive.

Figure 7 to the right: Detail of the tapped L2 inductor should you decide to experiment with a tuned drain version or need one for another project. Wind your coil and leave an extra long loop for your wire tap. Cut the tapped loop at the midpoint and use a small piece of folded ~150 grit sandpaper to remove the enamel from each of the 2 wires. Twist the now bare wires together and lightly solder them. Cut the end wires to the required length and use the sandpaper to remove the enamel. A method to strip enamel off wire is a frequently asked question for me and sandpaper works well.

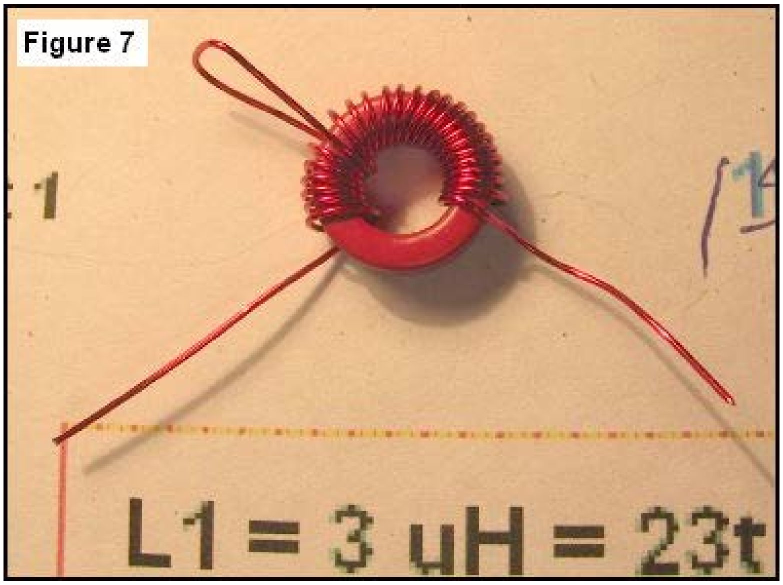


Figure 7

Broadband Output Version

Figure 8: The broadband version W7ZOI suggested to try building. I modified the output transformer in the Figure 6 project and tested it with Tom, VE7TW. We really liked it. By adjusting the network trimmer capacitor, I was also able to tune the 30 meter Amateur radio band as well. For 30 meter band use, I peaked the tank circuit at 10.125 MHz by listening to receiver noise with a home brew direct conversion receiver and was suitably impressed.

This is the active antenna design I wll use for my future projects where strong voltage gain is required. If your receiver has a higher impedance such as 500 ohms, you might try using a couple more links on the output transformer secondary winding.

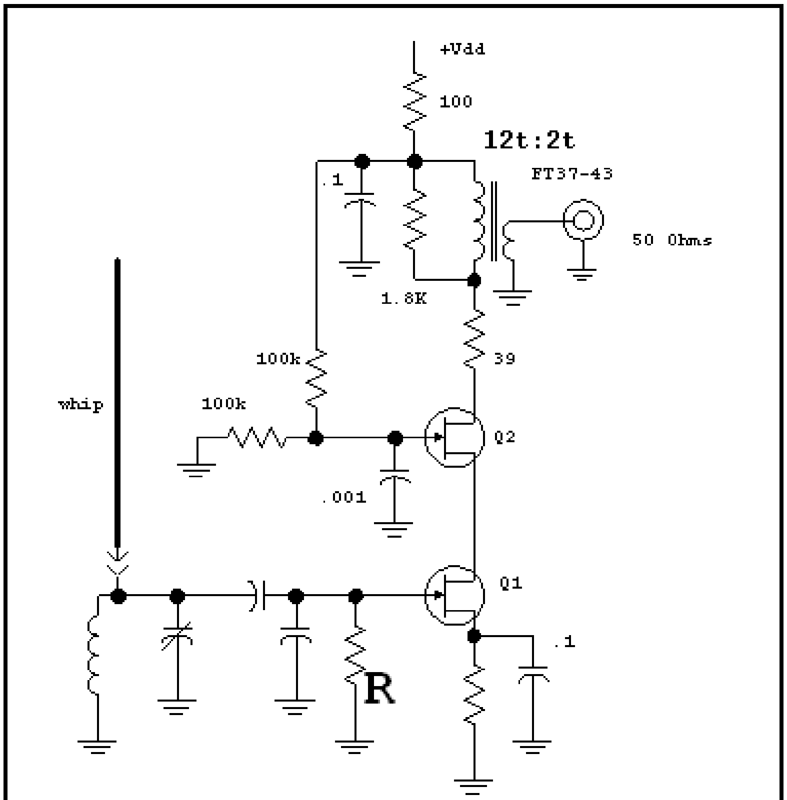


Figure 8

W7ZOI March 6, 2005

Tuning a Whip To Other Frequencies

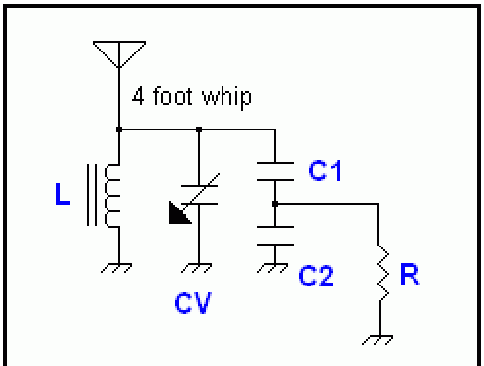


Figure 9

The ability to calculate the network values for different frequencies may prove difficult for those who lack software and/or math skills. To that end, a table follows which has some radio frequency bands and some suggested starting values for the Figure 9 parameters. Please note these are calculated and are suggested starting points based upon my limited understanding of radio electronics. Experimentation is the best method to find what component values will work for you.

Emails regarding the component values used in actual experiments is greatly welcomed.

The **R** value is the input impedance of your RF amplifier. In the case of the cascode JFET amp, it is the Q1 gate resistor.

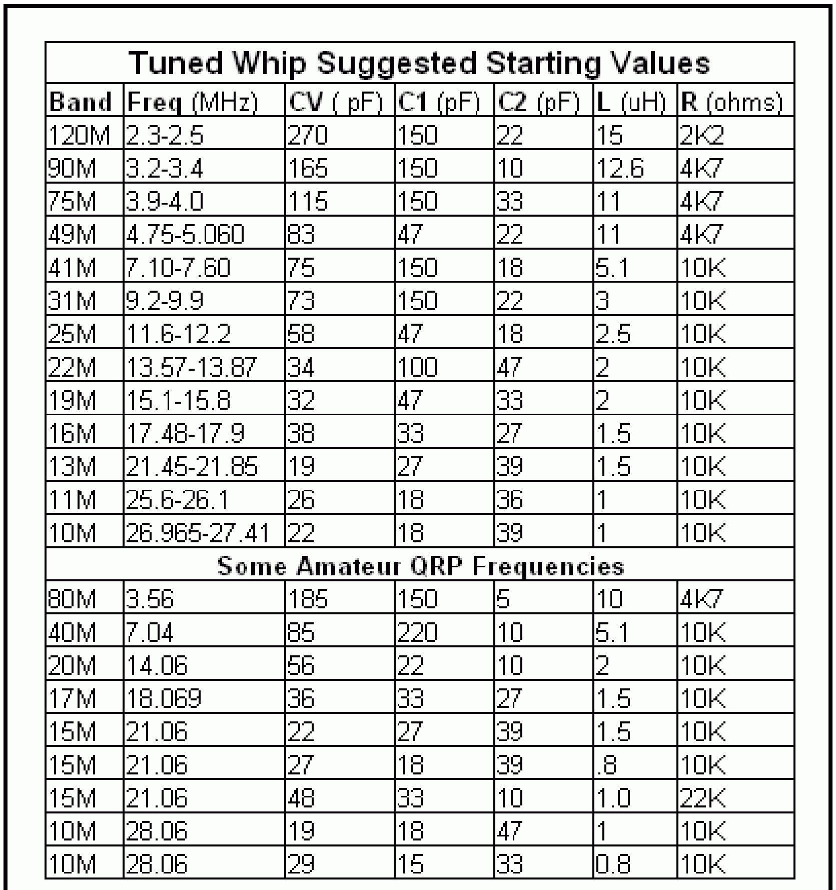
Note that the actual circuit CV value is typically much lower than the suggested (calculated) CV value from the chart. CV is used to **resonate** the tank. CV is dependent on several factors including the capacitance of the whip antenna, your RF amplifier input capacitance , your circuit layout, component lead lengths and variations in the powdered iron core and C1 and C2 capacitors values. Expect that your whip antenna will exhibit between 8 and 15 pF of capacitance. You need to subtract this from the suggested (calculated) CV value from the Figure 10 table.

Wes, W7ZOI told me that the whip antenna capacitance will remain constant as you change frequency providing you are below 1/4 wavelength for a given frequency. Here is a good web site applet to calculate wire or whip 1/4 or 1/2 wave lengths per frequency: http://www.csgnetwork.com/antennagenericfreqlencalc.html.

The actual circuit CV might include a trimmer capacitor plus a parallel fixed value capacitor.

How to find the correct trimmer capacitor for any tuned circuit you wish to resonate

I suggest you chose a circuit CV value by placing a variable trimmer capacitor in your circuit that when set to minimum will be below half or more than the calculated CV value. Then peak the whip tank circuit using a test oscillator and scope or RF probe or by just using receiver noise. Now temporarily add a 5-10 pF capacitor in parallel with trimmer capacitor. Just barely solder it in place or even just hold it in place without touching the leads. If the output increases, you were under the correct circuit CV value. Add more capacitance and check again. Repeat until you are satisfied with your chosen circuit CV value.



Tuned Whip Suggested Starting Values						
Band	**Freq** (MHz)	**CV** (pF)	**C1** (pF)	**C2** (pF)	**L** (uH)	**R** (ohms)
120M	2.3-2.5	270	150	22	15	2K2
90M	3.2-3.4	165	150	10	12.6	4K7
75M	3.9-4.0	115	150	33	11	4K7
49M	4.75-5.060	83	47	22	11	4K7
41M	7.10-7.60	75	150	18	5.1	10K
31M	9.2-9.9	73	150	22	3	10K
25M	11.6-12.2	58	47	18	2.5	10K
22M	13.57-13.87	34	100	47	2	10K
19M	15.1-15.8	32	47	33	2	10K
16M	17.48-17.9	38	33	27	1.5	10K
13M	21.45-21.85	19	27	39	1.5	10K
11M	25.6-26.1	26	18	36	1	10K
10M	26.965-27.41	22	18	39	1	10K
Some Amateur QRP Frequencies						
80M	3.56	185	150	5	10	4K7
40M	7.04	85	220	10	5.1	10K
20M	14.06	56	22	10	2	10K
17M	18.069	36	33	27	1.5	10K
15M	21.06	22	27	39	1.5	10K
15M	21.06	27	18	39	.8	10K
15M	21.06	48	33	10	1.0	22K
10M	28.06	19	18	47	1	10K
10M	28.06	29	15	33	0.8	10K

Figure 10

If after adding the initial 5-10 pF capacitor, the output decreases, try peaking the tank again to see if you can restore the signal strength you had before you added the temporary capacitor. If after peaking, the signal strength is down, you now have too much capacitance and can remove the temporary capacitor.

You just might also have too much capacitance. You might try a smaller variable cap or reduce the value of any fixed capacitors in parallel with your trimmer to make sure your minimum capacitance is not too high to properly resonate the input tank circuit.

The point is you need to be able to tell if you have too little or too much capacitance for CV and by going under and over you can tell if you are truly resonating the tank when you adjust the trimmer capacitor. Experimentation will tell you.

Another option is to put in a front panel adjusted variable capacitor. Front panel switchable inductors might also allow other bands to be tuned with 1 tuned whip network.

Moving your body as you adjust the trimmer capacitor can change the tuning, so please keep this in mind.

Figure 10 above. Picking an inductor value for the whip network can be tricky and sometimes trial and error is required. This table may be used to find starting values for the Figure 9 network. Below 41 meters, I suggest trying a lowered RF amp input impedance as shown to allow practical component values. Most of these calculations have not been tested.

I think an indoor active antenna for 74 meters or below might just be a noise generator.

I chose a frequency mid band for any given SWL band on the chart. The bandwidth of the input network is wide enough so this should be suitable to cover a good portion of the band.

Building An Active Antenna

To build this active antenna, chose the input tank network values from the Figure 10 chart or from your own calculations and then use them in the Figure 6 circuit. The Q1 source resistor can practically be from 100 to 390 ohms depending on how long you need your 9 volt battery to last. Increasing this resistor value will reduce the amplifier power gain. Try different values and see for yourself!

Some Practical Examples: 40 and 41 Meter Band

An active antenna that would provide coverage of the entire 40 meter Amateur Radio and 41 meter Shortwave band was designed. A varactor diode was used as the tuning element. The tuning voltage to the varactor was controlled by a 10K potentiometer which also had an integral switch. The finished VPA is shown in Figures 11 and 12.


Figure 11


Figure 12

Figure 13 below: A hotter JFET, the J310 was used in this VPA. In the test receiver, I was able to peak a signal from ~ 6.90 to 7.60 MHz. Tuning is very sharp but peaking is easily performed by turning the potentiometer gently back and forth while listening to receiver noise or a station. It might have been better to use a smaller value zener diode as when the 9 volt battery fades below 7.5 VDC, the zener diode will not conduct and the voltage regulation will fail. Having said that, this "hotter" VPA is intended for use with an external power supply as current draw at 9 and 12 volts is 19 mA and 28.9 mA respectively.

I tried using this VPA as the antenna for the Cascode 7 receiver shown elsewhere on this web site. When the VPA was peaked at the receiver tuning frequency, loud oscillations occurred. The receiver and the VPA were about 1 meter apart. I had to turn on the -10 dB attenuator and detune the VPA for the oscillations to stop. Moving my hand near the whip antenna varied the oscillations. The high gain, tuned circuits of the Cascode 7 receiver are not a good choice for an active antenna.

Future receivers projects will have a integral VPA and clearly the front end of these receivers will have to be designed carefully. A low cost Grundig receiver was also overloaded with this VPA.

This VPA worked well with other receivers which did not have a tuned, high gain preamp.

Figure 14: From the chart, the MV209 exhibits about 44 pF (guessing) when 0 volts are

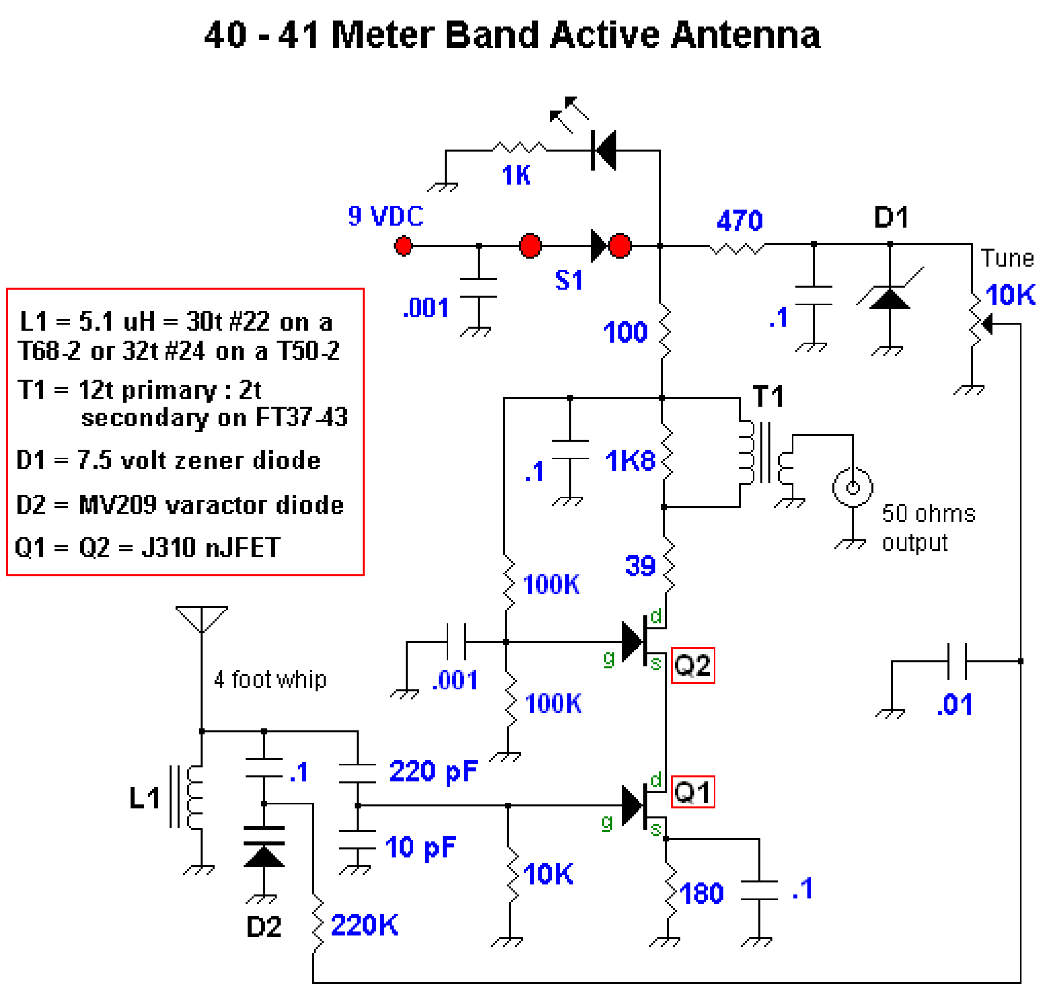


Figure 13

Mar 12, 2005 VE7BPO

applied to it.

The VPA was built and tested before the tuning diode components were added. A 7.039 MHz crystal oscillator with a piece of wire for an antenna was used as an RF source. The T2 secondary was connected to ground via a 47 ohm load resistor.

A 47 pF fixed capacitor was lightly soldered in parallel with L1 and the voltage was measured with an oscilloscope. A 10 pF capacitor was then carefully held across L1 and the voltage increased by 0.25 volts. A 22 pF capacitor was then tried and the voltage decreased much below that of just the 47 pF capacitor. The nearest standard value I had on hand above 47 pF was 56 pF. The 47 pF capacitor was removed and replaced with the 56 pF one. I tried holding a 5 pF capacitor in parallel with the 56 pF and the measured voltage decreased. I had experimentally determined that to resonate L1 at 7.039 MHz I needed between 47 and 56 pF for the CV value. This range should be close enough to resonate the tank at 7.0 MHz as well.

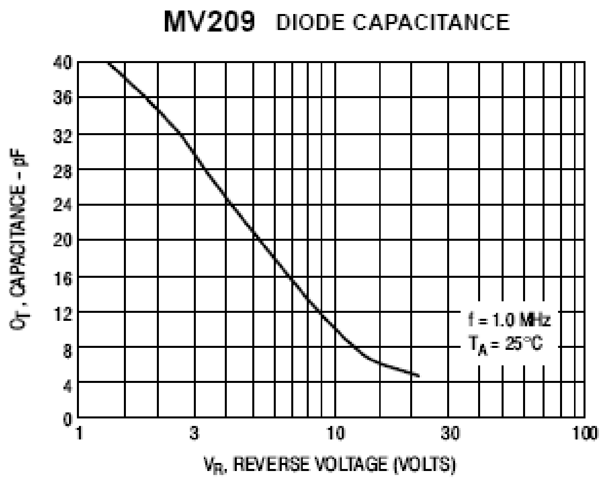


Figure 14

I then chose a varactor diode. The MV209 would be perfect for my project based upon the Figure 14 chart. I anticipated that I might have to place a small trimmer capacitor in parallel with the varactor to resonate the tank at the my lowest design frequency which was 7.0 MHz. As it turned out, in addition to the varactor capacitance, the voltage control circuit added additional capacitance and I actually needed 0.30 volts (measured between the 10K pot and the 220K resistor) to resonate the whip at 7.039 MHz. This was perfect; I did not need a trimmer capacitor! At 0 volts to the varactor diode, my whip resonated ~ 6.90 MHz.

---

5 MHz WWV Cascode Bipolar Amplifier

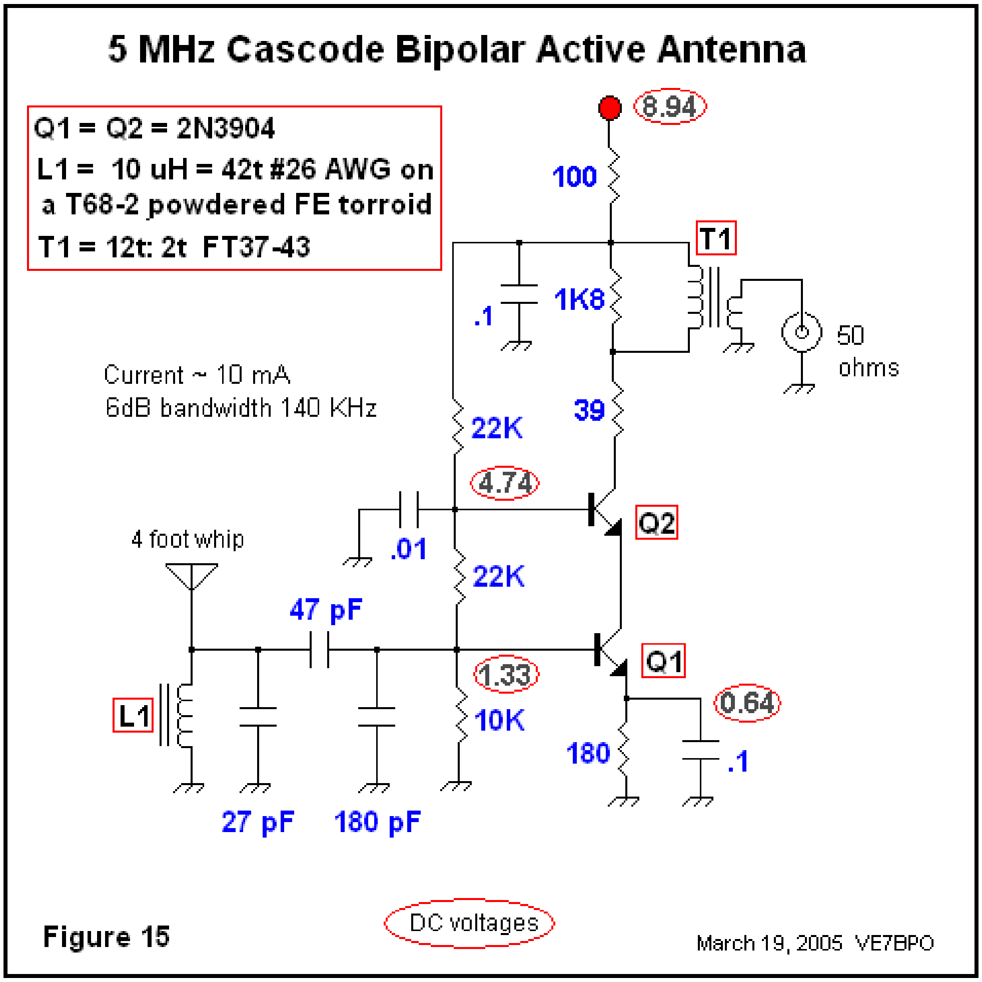


Figure 15

I wanted to build a non-FET version as shown in Figure 15. The tuned whip tank was originally resonated with a 5 - 40 pF trimmer capacitor. I unsoldered this trimmer capacitor and measured it with a meter; it was 27 pF. A 27 pF fixed value capacitor was soldered in and tested. The circuit was resonant at 4.98 MHz. This was close enough for me and also the 3 high Q fixed value capacitors provided a very narrow 6 dB bandwidth along with the inductor. The output impedance value of the tuned whip is around 200 ohms to match the Q1 bipolar amp input impedance.

Listening tests indicated that this circuit probably had too much gain at 5 MHz. It might be favorable to lower the Q1 emitter current to 7 mA or so by raising the Q1 emitter resistor or decreasing the Q1 bias voltage. Also, a series feedback, degenerative resistor on the Q1 emitter might be considered. This active antenna was comparable to the outdoor reference aerial for signal strength, however, predictably was much noisier.

Final Comments

I found that using lower Q trimmer and fixed value capacitors undesirably increased the tuned whip bandwidth presumably by lowering the resonant circuit Q. The inductor unloaded Q was the dominant factor however. The worst case scenario was a tuned whip built with junk-quality parts which had a -6 dB bandwidth of ~390 KHz.

I also learned that you should expect high gain amplifiers to oscillate and specifically design to reduce or suppress this tendency.

The 2005 Active Antenna experiments were fun and provided many learning opportunities. An active antenna is a perfect weekend project. There is no substitute to learning by building and testing electronic circuits with your own hands.

My sincere thanks to all of the friends who helped me with these experiments.

Experiments by Other Builders

What follows are some VPA experiments by others that were sent to me by email. I seek your feedback and photographs to help improve this web site and also to gain motivation to add more new content.

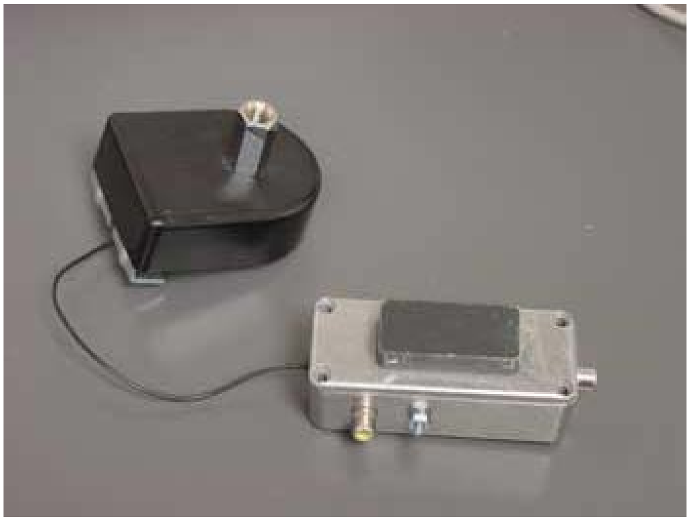
Joe, K9LY
Hi, Todd :

Attached are some photos of a voltage probe amplifier that I built using ideas from your excellent website. I use a TenTec 1254 receiver in the car and listen to some shortwave broadcasts during the daily commute to and from work. The antenna is a 4-foot whip that screws into the trunk-lip mount shown. The amplifier is held to the bottom of the trunk lid by a magnet and has survived for several weeks without falling off.

The amplifier is tuned by a varicap diode and covers approx. 9-14 MHz. The tuning voltage comes from a potentiometer that I added to the front panel of the receiver.

I decided to use the 2N3904 cascode amplifier because I liked the idea of using the most common transistor possible. The LC-tuned input is nice because the antenna whip is held at chassis ground potential, which should help prevent damage to the amp caused by static buildup.

The TenTec 1254 Receiver. http://radio.tentec.com/kits/Receiver



Above left: Inside Joe's trunk lid SWL Active antenna. Great ugly construction in a sturdy Hammond chassis.
Above right: Joe's remotely tuned SWL active (or voltage probe) antenna amp and whip holder. Thanks Joe!

Amateur and Short Wave Radio Electronics Experimenter's Web Site

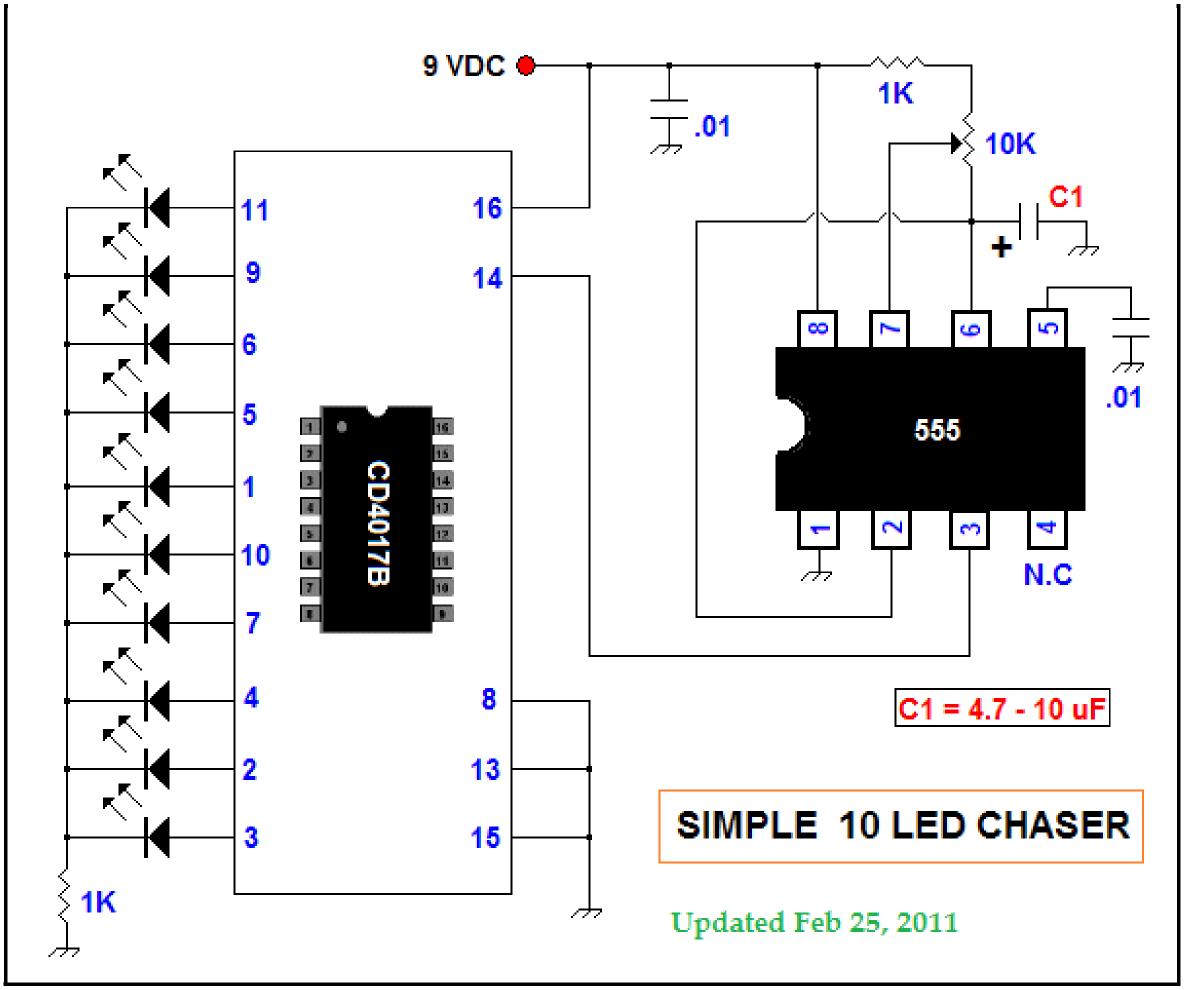
Fun with LEDs

Introduction

This summer I built several LED projects including sequential LED chasers (sequentially left to right) and also "Nite-Rider" style which go (sequentially left-right-left-right-etc). Many more LED schematics can be found on the World Wide Web via a Google search. LED projects are great fun for both HAM's and SWL's alike. They are also a lot of fun for children to experiment with. Currently, I am experimenting with PIC microcontrollers to perform LED "tricks".

I also built several very bright LED flashlights which run on a single 1.5 volt battery. For ultra-bright LED flashlight schematics, check out Dick Cappel's excellent and very informative web site. He has a number of LED driver circuits and other great schematics and theory. To wind the inductor for these LED flashlights, I had good success using an FT-37-43 ferrite torroid core. I used at least 40 turns of wire which is generally center tapped.

LED Chasers



Above. This is the schematic for a ***very*** basic 10 LED chaser I built. I prefer my "chasers" to run slower than most and chose a 10 uF capacitor for C1. The 10K pot can reduce the flash speed from not moving to whatever minimum time constant is possible with the C1 value you choose. Don't bother with ultra-bright LEDs for these "chaser" projects as cheaper, lower millicandela (mcd) LEDS work fine. I favor blue and green LEDS. The 4017 decade counter is a fabulous part and can be driven to flash a row of LEDs with a 555 timer chip or a discrete BJT multivibrator.

Update Feb 25, 2011: Many "experts" have emailed, or flamed me on web forums to say this circuit can't possibly work. I really hope these unhappy men cultivate enough humility to study and understand the 555 and more importantly; to reap some of the happiness and joy that comes from being positive and helpful to others. Since 2005, greater than 300 builders have emailed to say this simple circuit works and they want to learn more about electronics.

My intent was to have the least number of parts to flash some LEDs. Some new builders become overwhelmed when the parts count is high — I once shared this fear and relate. You'll see a number of different bias circuits for the 555. Many builders run the reset pin; **Pin 4** high (connected to the 9 volt battery) and as a rule, this is a good thing to do, but it's not necessary for the circuit to work. Pin 7 is an open collector output to ground — its primary purpose is to discharge the capacitor.

It's important for the DC voltage in the pot wiper to not become too close to the + 9 volt rail or VCC (This happens when when the pot is rotated so that maximal DC voltage appears on pin 7), as pin 7 would draw excessive current. In my original schematic I left out a series resistor from VCC to the pot to eliminate this problem. After some thought, I added a 1K resistor on Feb 25, 2011, although this limits the rate somewhat. I run the pot on my circuit about mid-range and it hasn't been re-adjusted (or turned off) since 2005.

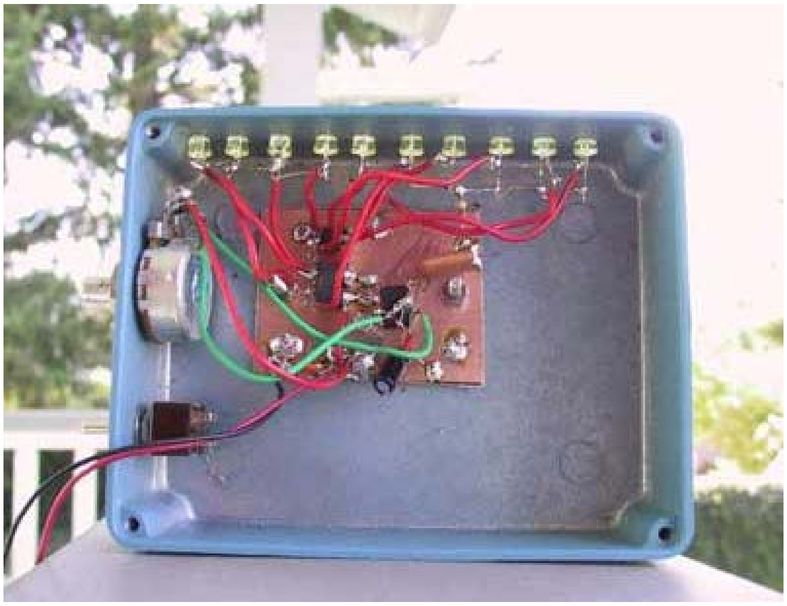
Also, the rotated pot wiper shouldn't get within a couple of hundred ohms of the capacitor as that too would cause excessive current spikes into pin 7. Generally, I prefer not to have much current on the wiper of a pot or, at least, try to keep the current small. Often, you can use a pot to set the desired timer speed and then remove, measure and substitute 2 standard value fixed resistors. A better way is to use math and calculate the resistor values, but this involves math and some people want nothing to do with equations.

This circuit is meant to provide a minimalist working circuit, but doesn't provide a great example of 555 design. Happily, for those wanting to learn more, countless great 555 tutorials may be found on the web. One of my 555 favorite sites is that managed by fellow Canadian Rob Paisley. Increasingly, I am exchanging electronics-related emails with model railroad enthusiasts across the globe and many of them know of Rob's wonderful web site.

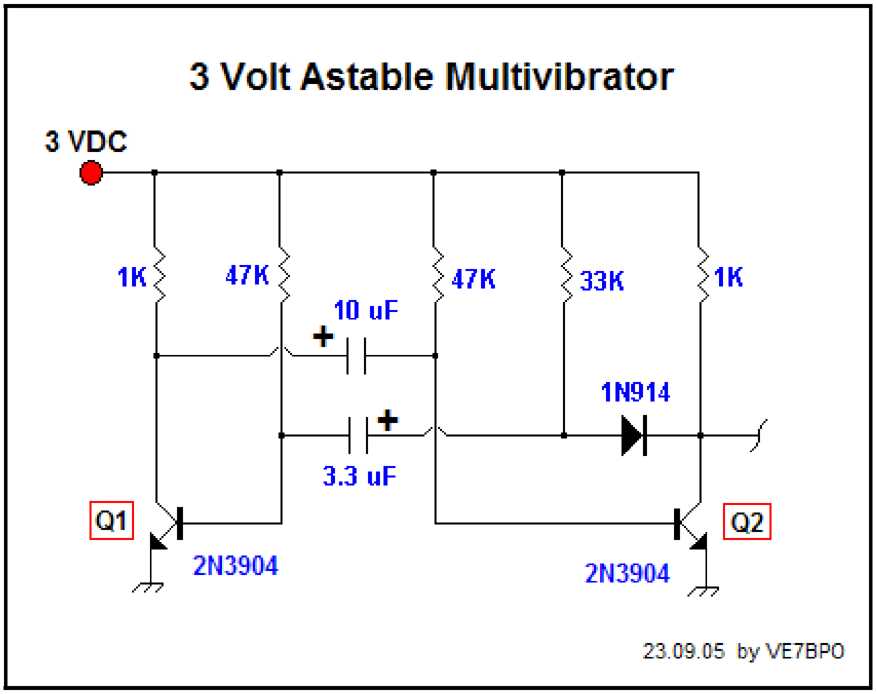
Matthew Ritchie built and posted a nice version of the LED Chaser on YouTube. A reader sent in this breadboard photo.

By far, the coolest device incorporating the LED chaser lies within a sculpture called Cyanic by Seattle-area artist Allet. Click for his web site. Cyanic may be found on the New Sculpture Build section on his web site. You have to start the Quick Time video manually with a mouse click. I love Allett's work and his lastest light sculpture exemplifies how the Internet can unite creative people with a positive attitude.

A 10 LED sequential flasher in a blue Hammond chassis. The schematic is shown above.

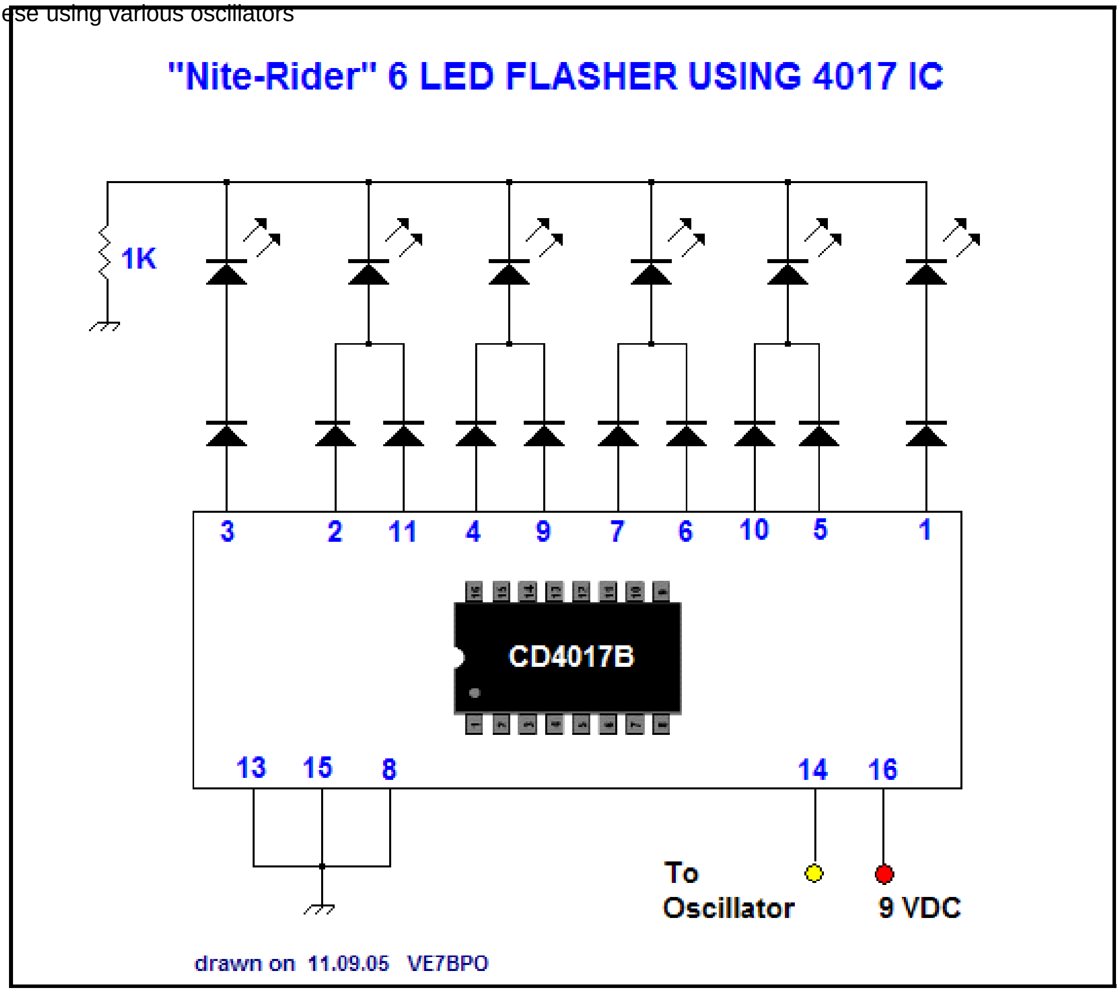


An RC oscillator designed for a 3 volt LED chaser. It oscillates quite slowly so the LED chaser it triggers will not be overly distracting. Some RC oscillator design details are discussed later. This oscillator triggered a 4017 decade counter instead of the 555 timer chip shown in the "Simple 10 LED Chaser" schematic. There are many links describing the theory of the 2 transistor astable multivibrator on the World Wide Web. I also have some information on this web page.



Above . This is a tiny 3 volt chaser which uses an LED bar instead of discrete LEDs. It draws 3.8 mA peak current on pulses. It uses the optimized BJT astable multivibrator shown directly above which fires at ~120 cycles per minute (slowly). The 3 volt battery pack is hidden behind it and should last several months. Soldering the LED bar was not an easy task. The plastic Hammond case measures 2.46 by 1.38 inches (6.25 by 3.5 cm).

A schematic to allow the 4017 decade counter to sequentially flash 6 LEDS left-right-left-right-etc. Connect your favorite square wave oscillator to pin 14. I built 4 of these using various oscillators and LED colors. You might consider using lower DC voltages and if so, may adjust the 1K current limiting resistor by using ohm's law. The 10 small signal diodes may be any appropriate type including the 1N914 or 1N4148. None of my 4 projects exceeded 6 mA peak current draw, so battery life is excellent. I increased the 1K resistor to 1K5 in my 4th project as I found the LED's that I used too bright.



The prototype "nite-rider" project with messy wiring. The holes for the LEDs were bored with a hand drill and it shows! The discrete transistor multivibrator can be seen behind the 4017 IC.

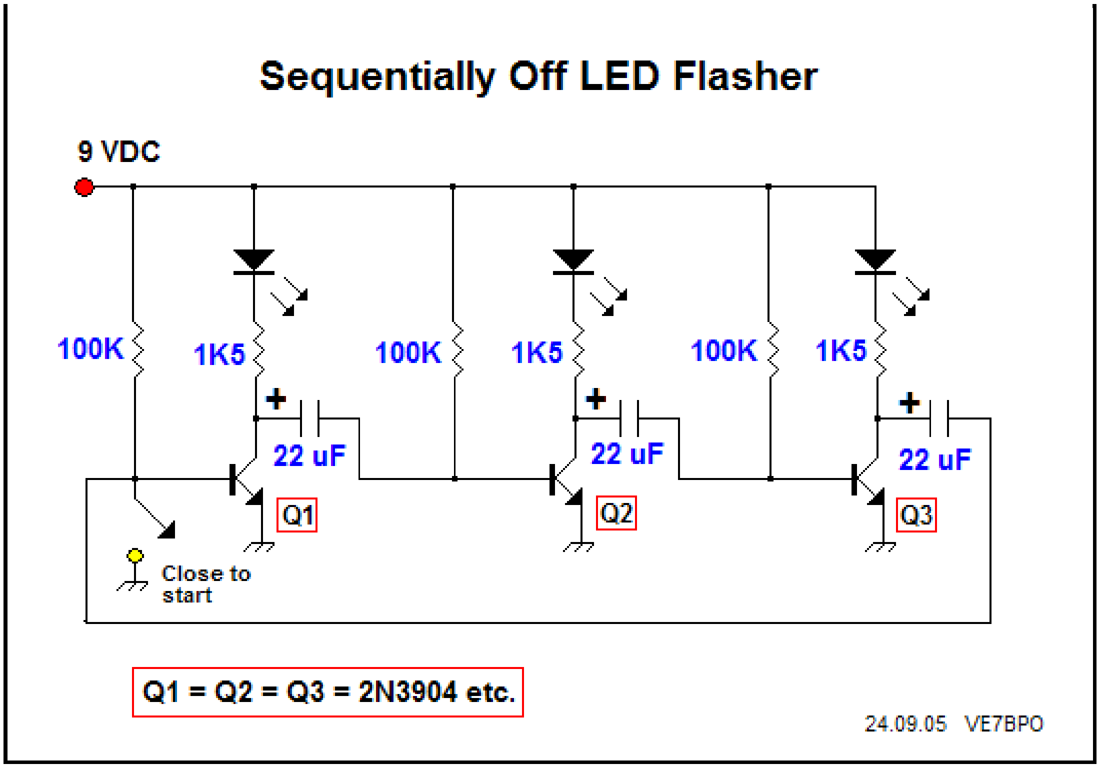
One of the four "nite-rider"

project chassis I built. After completion, this one was given to the son of VE7KPB. When drilling in a plastic chassis, I learned it is best to use a drill press set to a lower speed.

Sequentially Off LED Pulser

This circuit uses a series of transistors with an RC pair to pulse a string of LEDs.

This the favorite LED experiment I performed this summer. This flasher circuit is different in that it turns **off** alternate LEDs for about 1 second in sequence. When you connect this circuit to the 9 volt battery, all of the transistors are usually placed in saturation and therefore all the LEDs are **on**. Closing the switch on the base terminal of Q1 for a moment initiates the correct pulse sequence. The pulse initiates in Q1 which turns off the LED connected to the Q1 collector for about 1

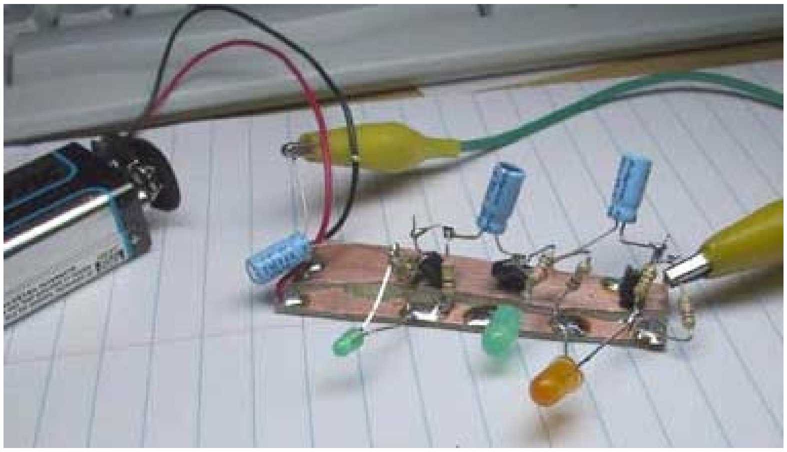


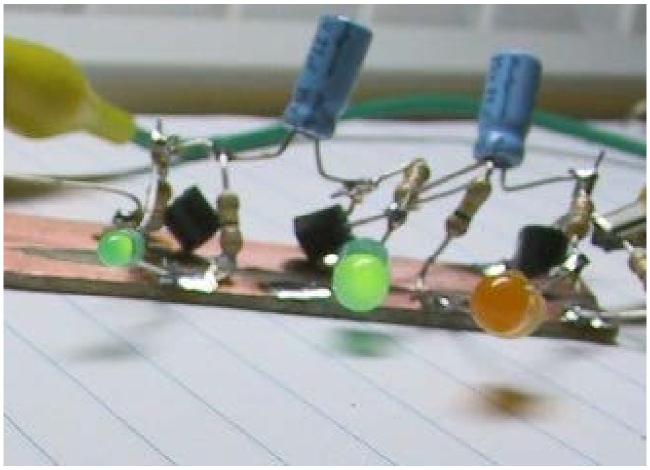
second. When Q1 turns back on (goes into saturation), Q2 turns off. When Q2 turns back on then Q3 turns off and so on. The circuit is a closed loop and many more stages may be added.

You can experiment with different base resistor and coupling capacitor values to vary the speed of the LED string or to create a sense of randomness by varying each transistor's RC stage separately. This is a fun circuit!

Youtube link (not mine).

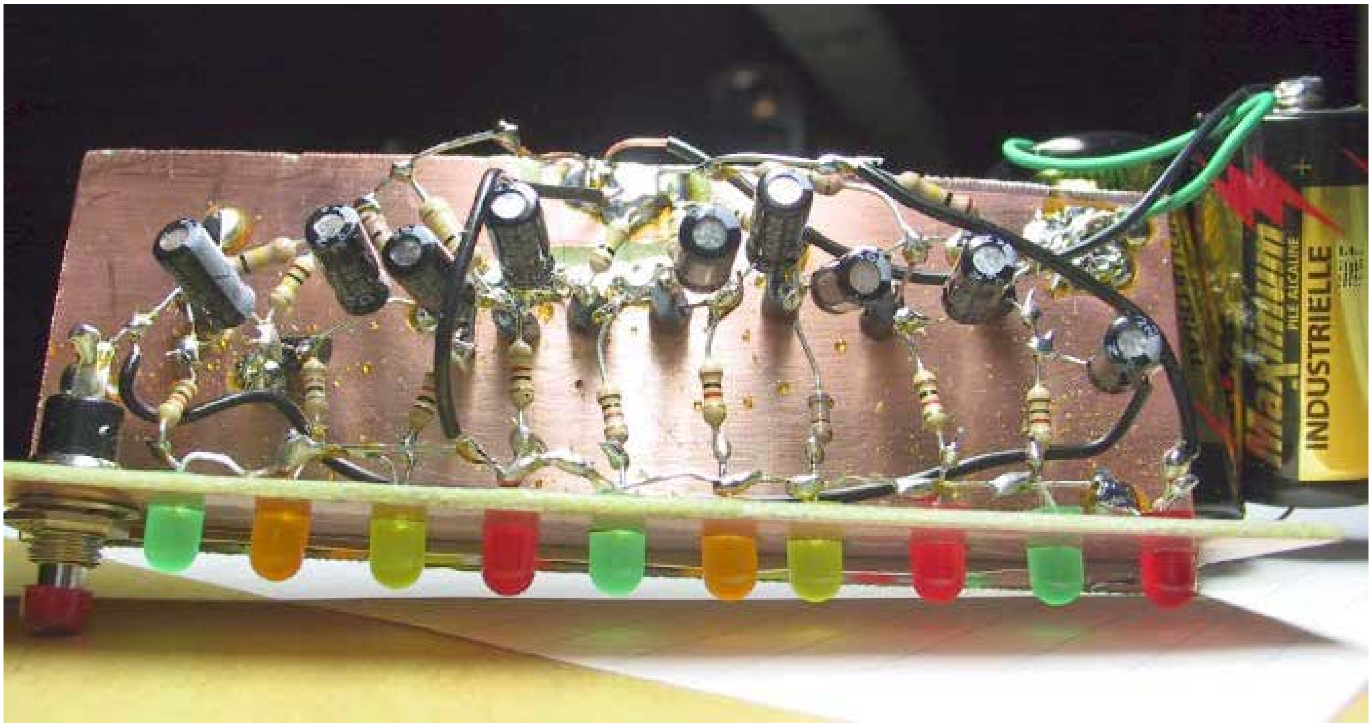
The prototype 3 transistor version. I just used a piece of wire to ground the Q1 base terminal and establish the correct pulse sequence after powering it up. For the LEDs, transistors, resistors and capacitors you can use whatever appropriate parts that you happen to have on hand. Current draw is less than 10 mA with a fresh 9 volt battery. Decrease the 1K5 current limiting resistor to 1K or so if you want brighter LEDs at the expense of more current draw. Do not operate this circuit above 9 volts unless you connect diodes from the transistor emitters to ground to prevent emitter-base breakdown.

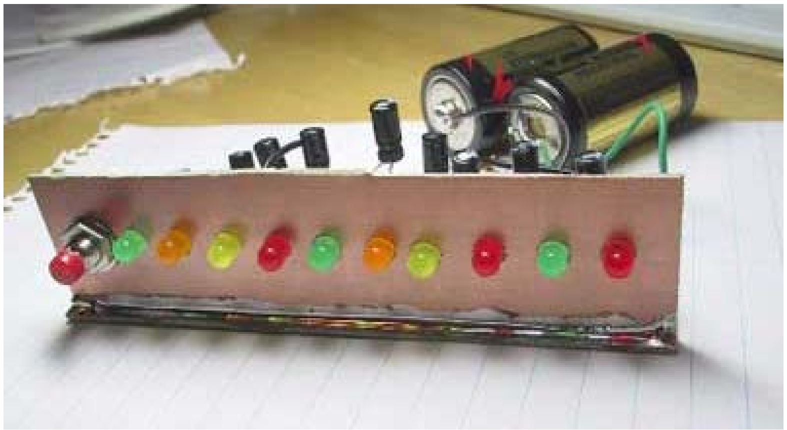




LED 1 and 2 are on and LED 3 is off at this moment in time.

Above and below photographs. This low current version has 9 LEDs connected in a chain and is powered by 3 volts. The 10th LED (extreme right hand side) is a flashing LED which is directly connected to the 3 volt supply and also uses a 1K current limiting resistor. Total peak current draw is only ~ 7 mA, yet it is still bright enough to see at night-time. The power supply is 2 D-cell batteries connected in series and then to the circuit by soldering wires directly onto the batteries with a 100 watt soldering iron.





NoNot counting the 10th flashing LED, 5 of the 9 LEDs are on at any given moment. A sequential flash effect is noted (the state of each LED flip-flops and shifts over 1 position each flash). If you build this project with an even number of LEDs, the sequential effect is not seen. Half of the LEDS (spaced every other LED) are on and the other half are off at any moment. The same LEDs are lit or unlit each pulse. Thus the effect is more like a typical multivibrator LED flasher. This variable, even versus odd number of stages property makes the circuit quite versatile.

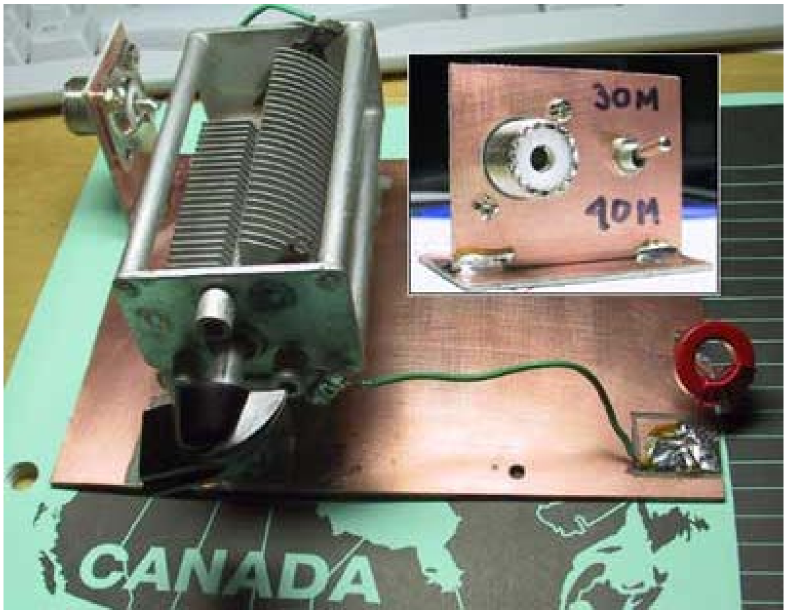
Conclusion

I hope that you have some fun experimenting with these and other circuits.

Amateur Radio Electronics Designer's Web Site

SWL Receiving Antenna Experiments

Introduction



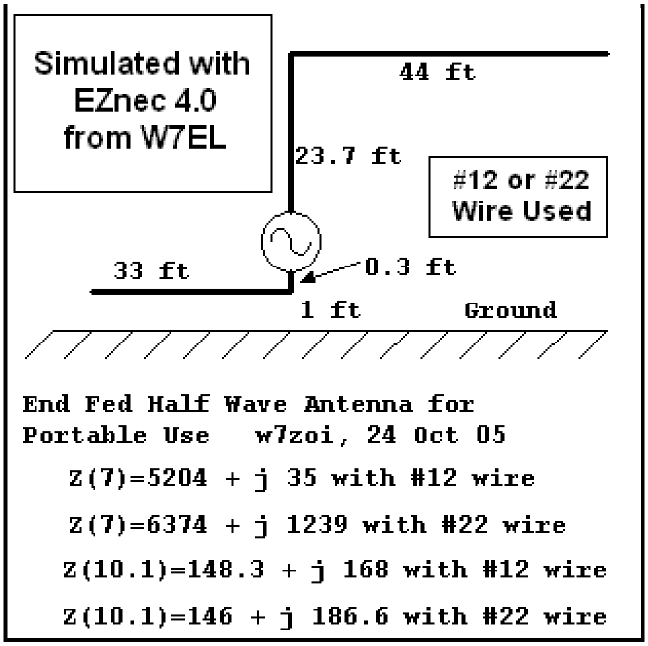
I have a lot to learn about SWL antennas. What follows are some brief experiments I performed in late October 2005. I have been experimenting with a half wavelength end-fed wire for use as a portable 40M band HAM transceiver (receive and transmit) antenna. This wire antenna is 67 feet long. End fed wires are very popular with those who pack a small portable transceiver when backpacking and camping. No feed line is required and the far end of the wire can be strung up using objects such as nearby trees or collapsible, portable poles. An elaborate ground system is not required. The return for the RF energy to ground might be grounding rod(s), short or long radial(s), or even just capacitively coupling to the local environment (including the operator!). Simple tuners are easily built to transform the high (thousands of ohms) wire impedance to the 50 ohms or so required by the transmitter.

I wanted to know if I could use this antenna as a tunable receive antenna for the the 30 and 31 Meter bands in addition to a tunable transmit/receiver antenna for my HAM radio work on 40 Meters. **What I verified is that tuning a multiband receive only antenna is not very practical.** When you tune a receive antenna you increase received noise and desired signals proportionally and therefore do not improve the signal to noise ratio in a meaningful way. Sometimes until you perform some experimentation, you don't really believe even good advice.

To the right: A computer simulation of a 40 Meter band end-fed Wire performed by W7ZOI on W7EL's EZnec program. The simulation was for 10.1 and 7.0 MHz with a 22 and 12 gauge wire. One 33 ft radial was used from the base of the 23.7 ft piece up 0.3 feet from the ground in this simulation. Z is impedance. Z and j are complex numbers used to represent the multi-dimensional quantities of the AC analysis of this antenna. In actual fact, j is an imaginary number. I suggest you might just ignore j unless your are well informed about impedance arithmetic.

Antenna Matching

A tuner can help match the impedance of the wire antenna and feed line (if used) to the input impedance of the receiver at a given frequency. This will result in more received signal and noise voltage to the receiver's input. HAM radio enthusiasts use antenna tuners to transform the impedance between the radio and the antenna tuner to 50 ohms to allow maximal output power from their transmitter. Non- amateur radio operators, can not use transmitters to



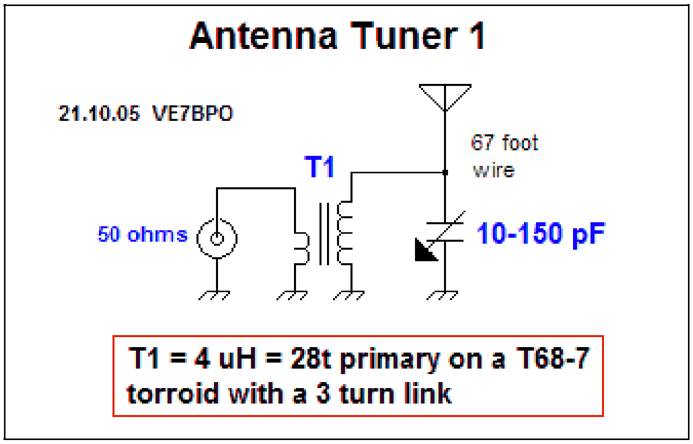
match their antennas. This leaves either using receiver noise, S-Meter or an antenna analyzer such as the MFJ259. I just used my ears and S-meter. All of the tuners presented work as transmitter tuners as well. Any network used for transmitter work must be able to handle the output power of the transmitter final amplifier.

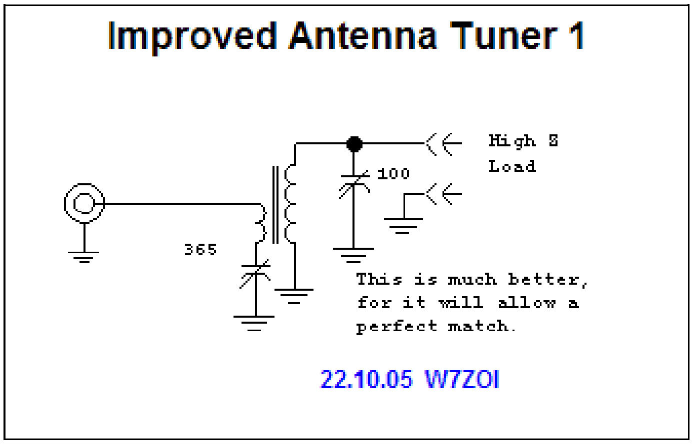
The Wire Antenna Experiments

I tried 3 different antenna tuners to see if I could tune an 18 gauge wire on 40-41 and 30-31 meters. My wire went from my computer room in the basement out a hole in the wall and sloped at ~ 50 degrees up to a rope tied to a tree in my backyard. The tip of the antenna is about 50 feet (15 meters) high. A 10 gauge insulated ground wire also passes from the computer room outside under the back lawn. It is "earthed" to two, 2 meter copper grounding rods hammered into the ground. No direct connection to the house ground system and the outside antenna grounding system should be made as this may result in increased receiver noise.

Antenna Tuner 1

The schematic on the left below, is very popular with HAMS who use it to to tune monoband end-fed wires. It is very simple and works reasonably well. Although the capacitor was able to resonate the 35 turns inductor, the T1 turns ratio was wrong and reducing it to 28:3 was required to get the maximum receiver signal in my experiments on the 40 and 41 meter bands. The alternative was to shorten or lengthen the wire antenna which is not very practical as it meant repeatedly climbing a tree.

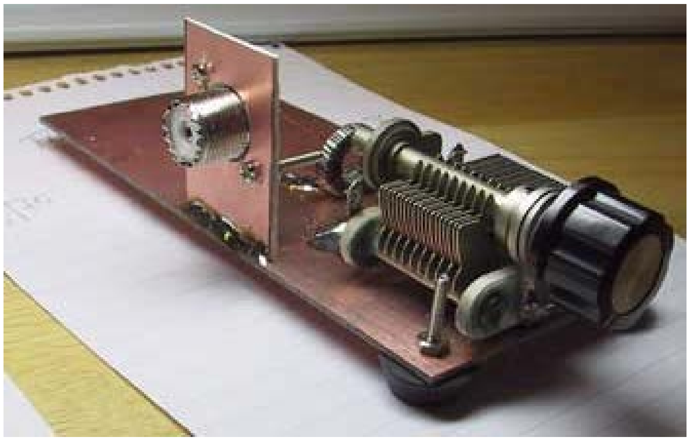




Above right: An improved version of the Antenna Tuner 1 schematic from W7ZOI. This tuner has 2 user "tweaking" adjustments much like most modern commercial antenna tuners (which typically also have a band changing adjustment).
I did not build this version.

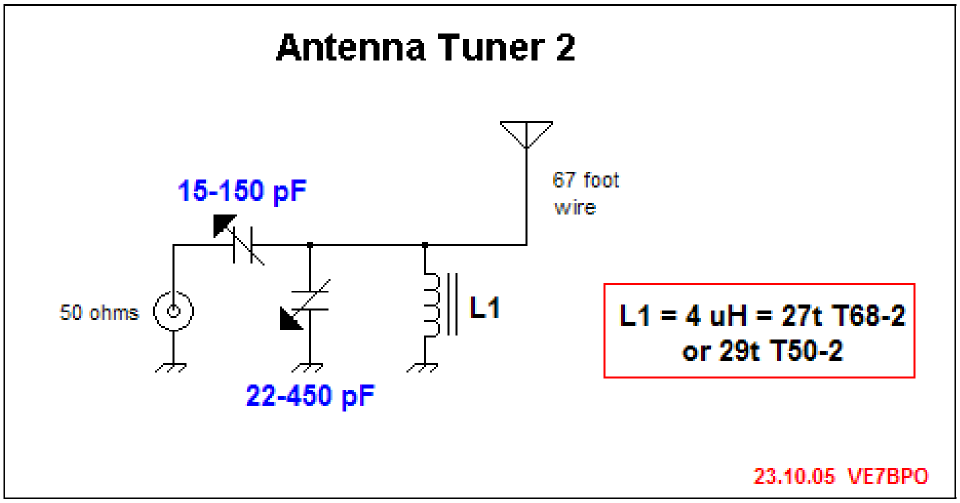
In addition, this tuner would not tune the 67 foot wire on the 30 or 31 meters band. This was no surprise. When I turned the variable capacitor, I noticed some change in received signal, but not much. The signal strength was very poor and I could not hear much of anything.

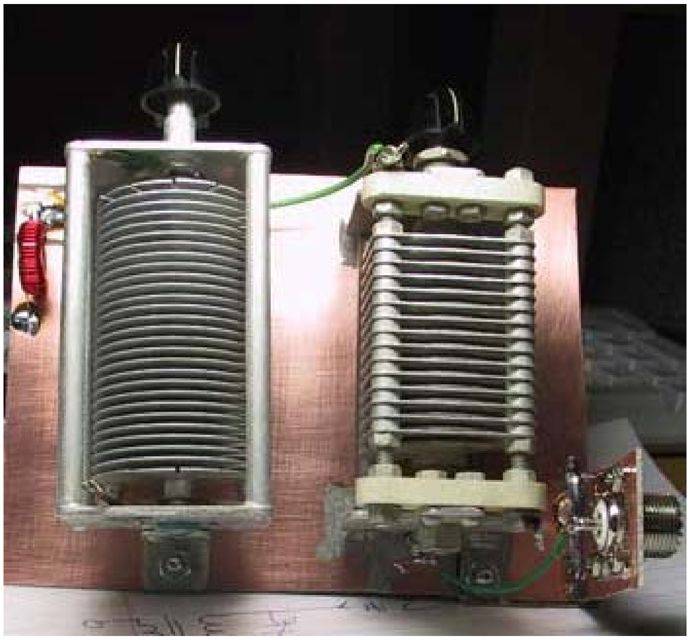
Below. Two built up views of the Antenna Tuner 1 schematic. I soldered the antenna wire to the circuit in the isolated area connecting the inductor and capacitor. The outside antenna ground wire was soldered to the large copper ground plane.

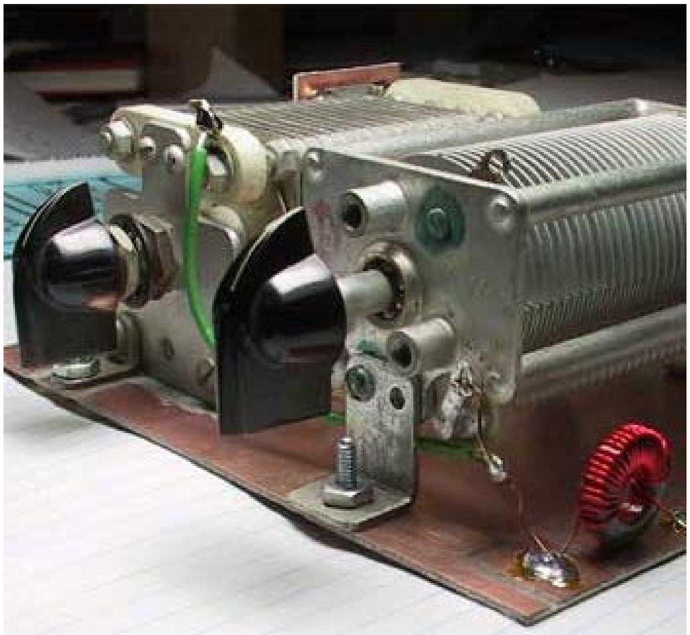


Antenna Tuner 2

This was the next tuner I built. Antenna tuner 2 tuned very sharply on the 40 and 41 Meter bands. It is designed to match a high impedance antenna, so it could not match the medium impedance (~150 ohms) wire to the 30 and 31 meter bands very well at all. Since ~ 150 ohms is fairly close to my receivers 50 ohm input impedance, I just connected the wire antenna directly to my receiver. The received noise and signals were then much stronger than those with the Antenna Tuner 2 network in the circuit on 30-31 meters.

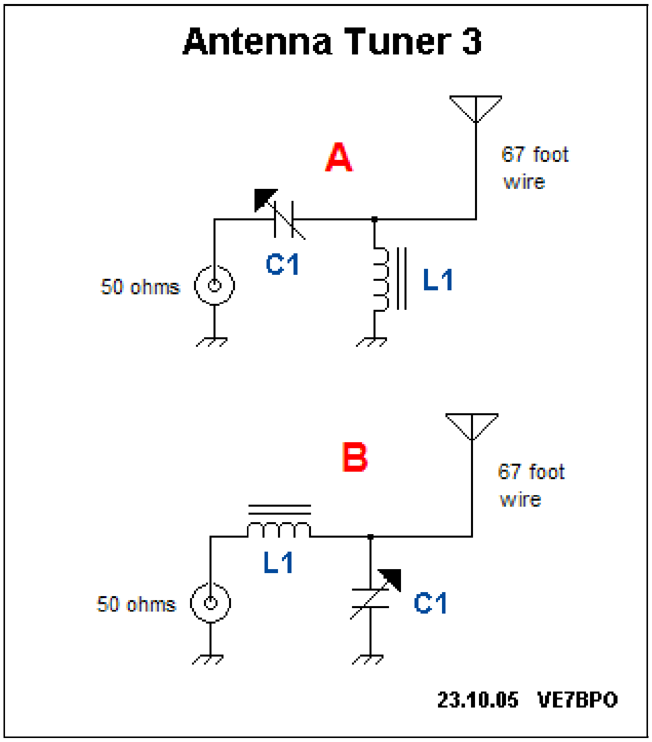




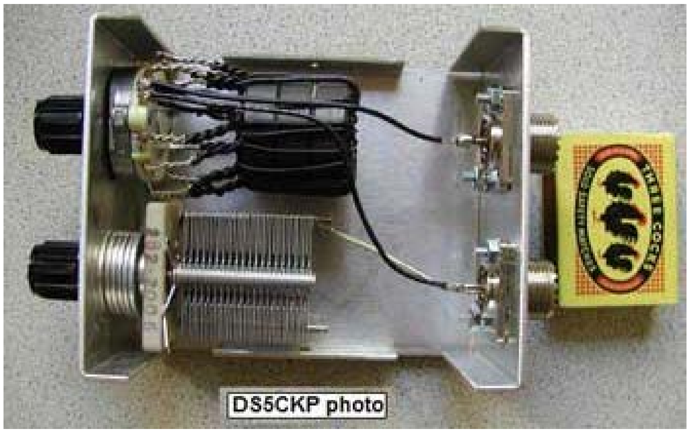


Above: Two constructed views of the Antenna Tuner 2 schematic. The variable capacitors were bought at a HAM festival in 1992 for 2 dollars each.

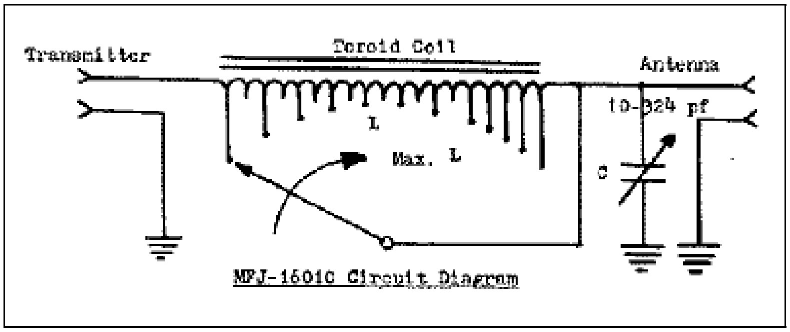
Antenna Tuner 3



The next antenna tuner topology I tried was the familiar L network. Circuit **A** is configured in a shunt L (inductor to ground) and series C (capacitor) and is a high pass L network. Circuit **B** is configured in a series L and shunt C and is a low pass L network.



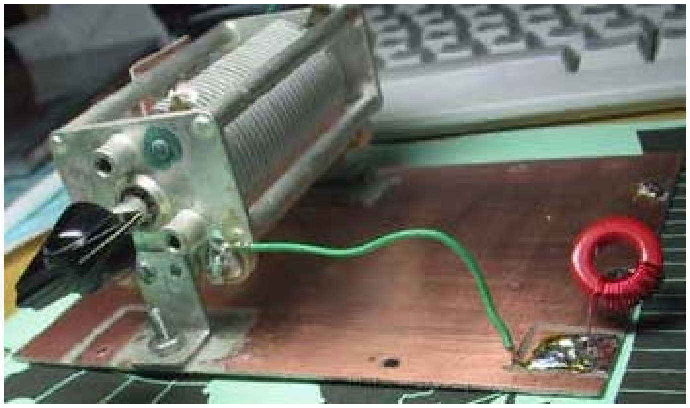
L networks (especially the low pass form) are very popular as random wire tuners. MFJ sells an excellent version as the model MFJ-16010. A photograph of this tuner is shown below right. The above left photograph was taken by **DS5CKP** who also sells a random wire antenna tuner at: http://user.chollian.net/~cyberline/ckptuner.htm

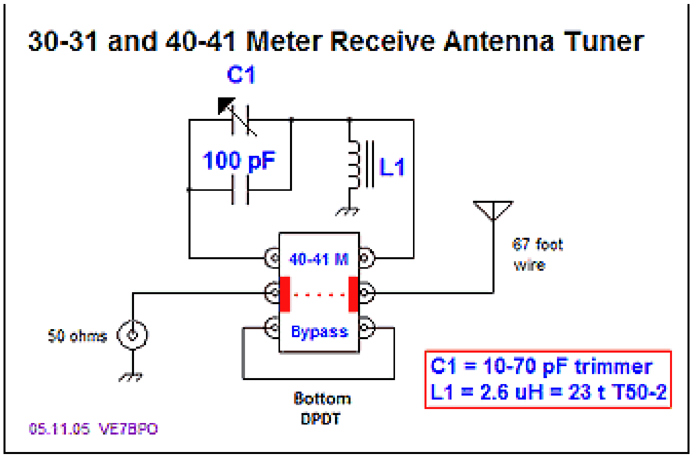


Above. MFJ web publishes their manual including the schematic for the MFJ-16010 random wire antenna tuner. The inductor is actually wound on 3 stacked (probably ferrite) cores which are tapped. The taps are connected to a front panel mounted 12 position switch. This allows coverage from 2-30 MHz. As a simple experiment, I tried stacking 2 and then 3 FT-50-61 ferrite cores and was able to get a wind range of inductances from the 6 taps I made on my test inductor.

Numerous examples of the L network antenna tuner can be found on the web and in print including the 2006 A.R.R.L. handbook. I decided to try the high pass L network topology to experiment with.

I built part **A** of the Antenna Tuner 3 schematic with shunt L and series C. On the 40-41 meter bands I tried 3 different inductors; 4 uH, 2.1 uH and 1.3 uH. I tuned the network to get the greatest receive noise and S meter reading and measured the variable capacitor value. At 7.30 MHz, the capacitance values were 86 pF, 141 pF and 182 pF respectively. Although non-critical, I settled on a 2.6 uH inductor (23 turns on a T50-2 powdered iron torroid) so I could use a junk-box 10-150 pF variable capacitor to resonate it. Later I decided just a trimmer capacitor might do. You need around 125 pF at 7.30 MHz to tune the network with a 2.6 uH inductor (to give you a ballpark C value to start with).

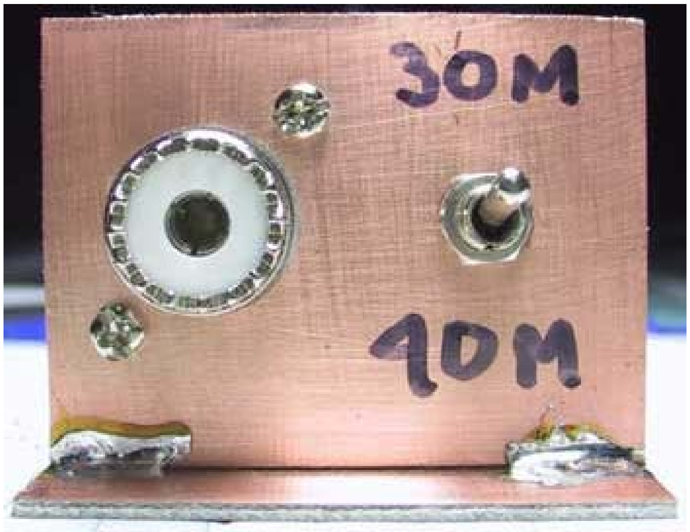


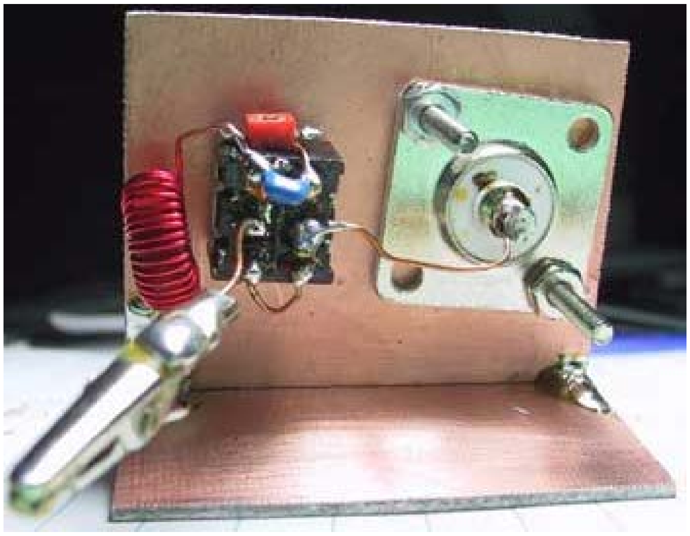


Schematic to the right: A very small receiver tuner that allowed the L network to be switched in and out of the antenna path was constructed. The L network is tuned on 40-41 meters via a small trimmer cap on top of the double pole, double throw switch. A small plastic alignment screw driver is used for signal peaking. An air variable capacitor would be much easier to tune as you move up and down these bands.

Below left: Front view. Technically, the switch label should indicate **40M** and **bypass** as the bypassed antenna could be used on any band.

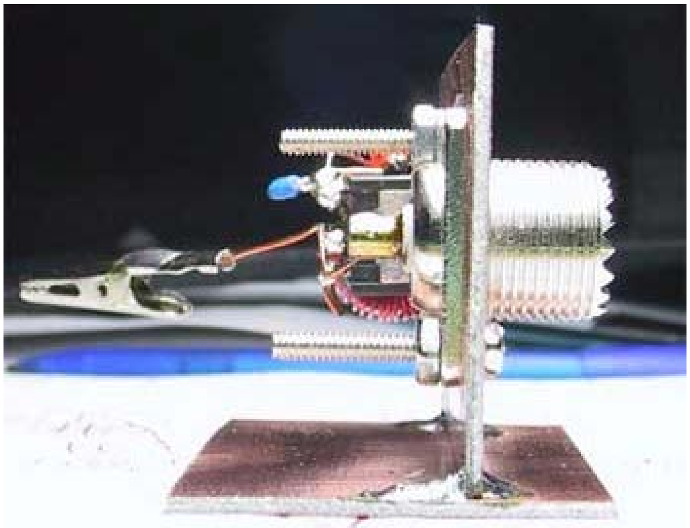
Below right: Rear view The trimmer cap can be seen on top of the switch.





Below left: Side view, the antenna wire is held via an alligator clip. This allows me to unclip and ground the antenna when it is not in use. The antenna ground wire is soldered to the copper ground plane.

Below right: A T6-8 core wound with with 24 gauge wire. I used larger powered iron torroids to wind my inductors, however, they can also be wound on ferrite cores or be air-wound. The core size, wire gauge and type of inductor used can affect the selectivity and insertion loss of the tuner network, however for practical receiving purposes it is not a major concern.



Conclusion

The final experiment allow me to easily switch between a matched and direct 67 foot wire antenna on 40 and 41 meters. I performed several listening tests and generally agree with those who say receive antenna tuners offer little to no improvement in signal readability. The only advantages I can think of for matching a receive antenna to a reasonably quiet and sensitive receiver are:

1. There may be some improvement in the receiver front end filter function as these filters are designed to have a specific input impedance.
2. Certain balanced mixers may function better with the correct impedance on their RF port.
3. The tuner itself (depending on design) may marginally improve the front end selectivity of the receiver it is connected to.

I do like the noise roar and louder signals with the matched antenna on 40 and 41 meters, although this is totally subjective. I think the reason for this is that as a HAM radio operator who always matches the antenna for any band I am on, I am used to louder signals and noise levels. I also spent most of my first 10 years of HAM radio operation on 80 and 160 meters which are relatively noisy bands and have been conditioned so that noise is "normal".

On 30-31 meters the bypassed antenna worked quite well and the L network can be switched in as an attenuator. Perhaps you might build up a tuner and try for yourself!

Although they are simple, low cost and easy to put up, it is likely unwise to use an end-fed wire without coaxial feed line as a receiving antenna. The ground wire is part of the antenna system and easily picks up household generated noise which will present to your receivers input. The time honored and easiest methods to reduce receiver noise are to get your antenna away from the house and other noise sources, use buried coaxial cable feed line to the house and directly earth ground the shield of the feed line with stakes. Antenna/ feed line "link coupling" by a transformer may also reduce noise (especially if the antenna system is balanced) and this topic begs further study.

Suggested Links

There are some fabulous web sites on the topic of SWL antennas and reducing receiver noise. I suggest these 5. Try a Google search for more.

http://www.dxing.info/equipment/

http://www.hard-core-dx.com/nordicdx/antenna/feed/feed1.html

http://www.aa5tb.com/efha.html

http://www.nyx.net/~dgrunber/

As usual, I learned a great deal from the process of experimenting. I look forward to spring when some more antenna experiments can be performed.

Amateur and Short Wave Radio Electronics Experimenter's Web Site

Experiments with JFET Biasing

The most common way of biasing a Junction Field Effect Transistor (JFET) is with a source resistor. This method, shown in Fig 1 below, has the advantage of offering negative feedback to stabilize the bias conditions. This is the same thing that happens when a bipolar transistor uses an emitter resistor. Self bias can be used as a method to evaluate a JFET to determine the critical parameters that describe it: Idss and Vp. These are discussed in Chapter 2 of **Experimental Methods in RF Design** and many other places.

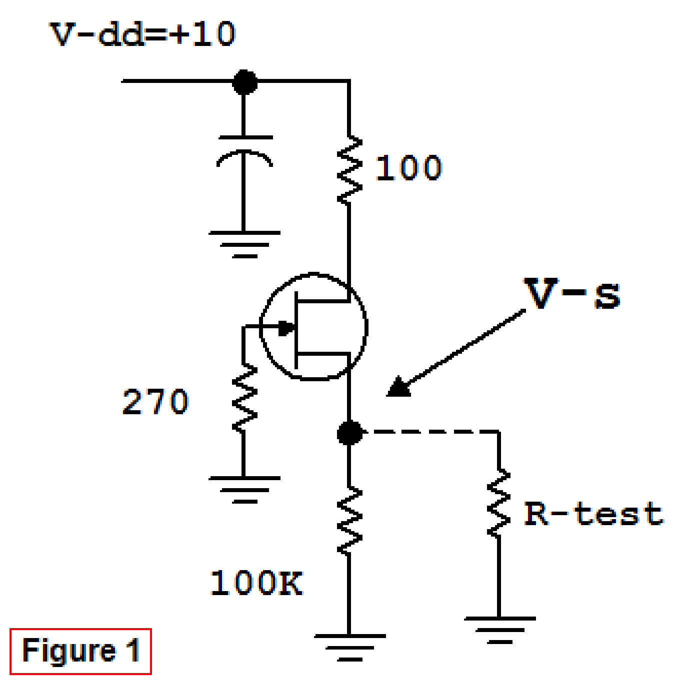


Figure 1

The method used in our experiment is to set up the FET of interest in a test fixture with a power supply, bypass capacitor, resistors in the drain and gate to suppress parasitic oscillations, and a handful of extra resistors, R-test, that can be paralleled with an existing 100K source resistor. A digital volt meter (DVM) is the basis for the measurements. We begin by using the DVM to measure the resistance of our test resistors, for the values will be used in calculations. The DVM is then attached to the FET source to measure the DC voltage. The first value we measure is with no attached R-test. The measured value will be very close to the FET pinchoff voltage.

The measurements we will perform infer drain current as a function of gate to source voltage. The physics of the FET support the model that there is no gate current so long as the gate is not forward biased with regard to the source. Hence, the drain current equals that in the source. We will measure the source current by measuring the voltage drop across the source resistor. The gate is at ground potential, for there is no gate current, so the gate to source voltage is just the negative of the source to ground voltage.

The resistors that I pulled from my stock for some measurements were marked as 22, 39, 68, 100, 150, 300, 510, 680, 1K, 2K, 3.3K, 6.8K, and 10K Ohm. The measured values are shown in attached figures. A systematic pattern was noticed with all of the measured resistances under the marked value, suggesting an error in the calibration of my DVM, a Fluke Model 73. All resistors were 2% carbon film 0.25 Watt. However, when I measured a 499 Ohm, 1% metal film resistor, it came up exactly at 500 Ohms. The differences between the measured values and those marked on the part were small enough that I neglected the details and used measured values for calculations.

The first FET I examined was a 2N5454, a common JFET that I had in my junk box. The source voltage was 3.26 with nothing but the 100K for source bias. I started my measurements with the largest resistor, 10K. The voltage dropped to 2.90 and was stable. I merely held the resistor in place rather than soldering it. The resistor was kept in place long enough to get a stable reading that I could record in my lab notebook. All results were of the same character until I got to the 300 Ohm resistor. At that point I started to notice a slight heating effect. The source voltage was 1.523, but slowly dropped to 1.518 volts. This behavior continued through the lower value resistors. The 22 Ohms produced 264 mV on the source that then dropped to 256 mV.

Later I examined a J310 JFET. This is a much larger area part than the 2N5454 with an Idss that is about three times larger. With the 22 Ohms in the test fixture, V-source went to 701 mV, but settled at 652 mV. The drain current was then 32 mA. With a 10 volt power supply, there was nearly 200 mW dissipated in the FET. This is within ratings, but high enough to produce heating. Operation at higher voltages and at Idss would further tax the part. One must take care when doing these measurements to be sure that the source voltage is observed quickly.

Attached are the MathCad documents that I used to examine the data. A spread sheet such as Excel could be used, but I prefer the graphics of MathCad. The second page for the 2N5454 shows a graph for the observed data as well as a calculated one. The two FET parameters for the 2N5454 were varied to obtain a good correspondence between the two. The part had Idss=15 mA with Vp=-3.5 volts. This is similar to the popular MPF-102, but close to the high Idss extreme for that part.

The data presented for the J310 is more abbreviated with only two points shown. I picked the 22 Ohm and 1K source resistors. This still produced data that is very close to that obtained with many more data points.

My initial analysis suggested that we could characterize the FET by measuring the source voltage with 100K in place to approximately determine Vp, and to then short circuit the source through the mA scale on the DVM to obtain Idss. This is a reasonable start. However, the pinchoff will usually be a few percent more negative (for an N-channel depletion mode part). The long leads in the source also make me feel uncomfortable with regard to parasitic oscillations.

After the DC measurements were done, I thought it wise to look at the potential for oscillation. The J310 was in the test circuit at this time. The TO-92 J310s are parts that are well known for their propensity for oscillation, so I guessed that it would not be difficult to coax this one into such a mode. But this was not what I found. I eliminated the 100 Ohm drain resistor, but moved the FET close to the 0.1 uF bypass. This bypass is not a very good one for VHF and upward. No oscillation was seen. I then eliminated the gate resistor, replacing it with the normal gate lead. Still no oscillation. I eventually added a gate inductor and a parallel tuned circuit in the source. The source bias resistor had a RF choke in series with it. I finally saw a robust VHF oscillation, but nothing else up through 1.5 GHz.

Experiments with JFET Biasing, after correspondence with VE7BPO on 14Feb06. (See earlier Mathcad file "Jfet_bias1_ve7bpo.mcd) Measurements on 2N5454.

$n := 0..12$ **index for measurements and calculations. Vsn=measured source voltages.**

$$Rx_n := \begin{pmatrix} 21.6 \\ 38.6 \\ 66.9 \\ 98.3 \\ 149.4 \\ 298.3 \\ 508 \\ 671 \\ 998 \\ 1997 \\ 3270 \\ 6720 \\ 9810 \end{pmatrix}$$

<== Measured Rx value

R values paralleled with 100K ======>

$$Rp_n := \frac{1}{\left(\frac{1}{Rx_n} + \frac{1}{100400}\right)}$$

$Rp_n =$

	0
0	21.595
1	38.585
2	66.855
3	98.204
4	149.178
5	297.416
6	505.443
7	666.545
8	988.177
9	$1.958 \cdot 10^3$
10	$3.167 \cdot 10^3$
11	$6.298 \cdot 10^3$
12	$8.937 \cdot 10^3$

$$Vs_n := \begin{pmatrix} 0.264 \\ 0.429 \\ 0.654 \\ 0.85 \\ 1.09 \\ 1.523 \\ 1.845 \\ 2.001 \\ 2.203 \\ 2.492 \\ 2.652 \\ 2.82 \\ 2.90 \end{pmatrix}$$

$$Vg_n := 0 - Vs_n$$

$Vg_n =$

	0
0	-0.264
1	-0.429
2	-0.654
3	-0.85
4	-1.09
5	-1.523
6	-1.845
7	-2.001
8	-2.203
9	-2.492
10	-2.652
11	-2.82
12	-2.9

Use Mathcad "vectorize" operator.

$$Id_n := \overrightarrow{\frac{Vs_n}{Rp_n}} 1000$$

$Id_n =$

	0
0	12.225
1	11.118
2	9.782
3	8.655
4	7.307
5	5.121
6	3.65
7	3.002
8	2.229
9	1.273
10	0.837
11	0.448
12	0.325

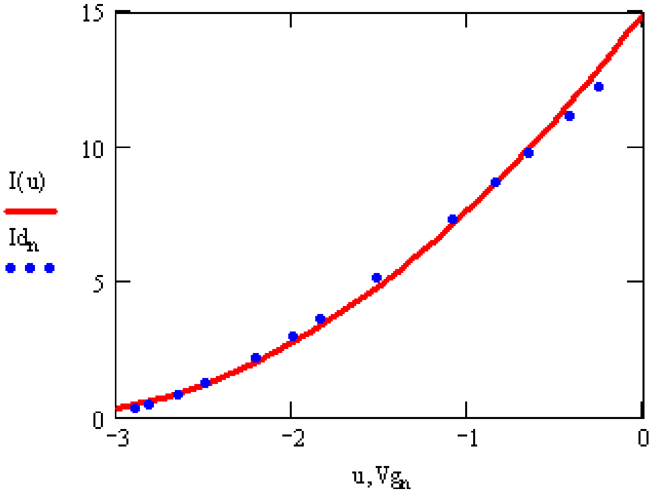
Let pinchoff be $u_p := -3.5$ **and** $I_{dss} := 15$

$$u := -3, -2.9 .. 0$$

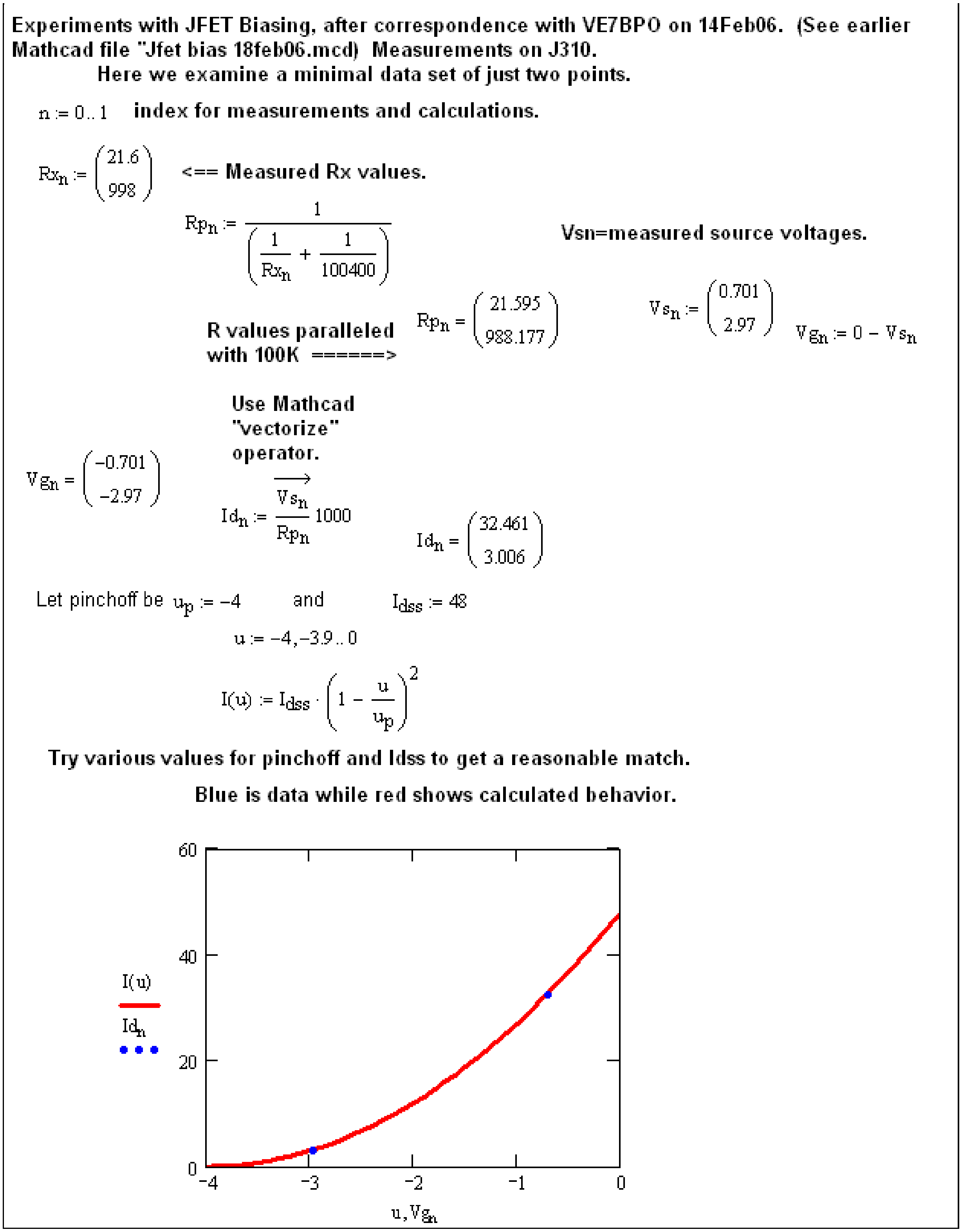
$$I(u) := I_{dss} \cdot \left(1 - \frac{u}{u_p}\right)^2$$

Try various values for pinchoff and Idss to get a reasonable match.

Blue is measured data while red shows calculated behavior.



Experimental Note: During the measurements, I noticed a definite thermal change in Idss when dissipation was high. This occurred at higher currents, with low source bias resistors. The Vg values I used in my data were the first ones recorded with the DVM. The most extreme case was with the 22 Ohm source R. The source V started at 264 mV and went down to 256 mV. The effect was more dramatic with a larger part, a J310. That part, when operated with a 22 Ohm R, started with 701 mV and dropped to 652 mV.



Experiments with JFET Biasing, after correspondence with VE7BPO on 14Feb06. (See earlier Mathcad file "Jfet bias 18feb06.mcd) Measurements on J310.

Here we examine a minimal data set of just two points.

$n := 0..1$ **index for measurements and calculations.**

$Rx_n := \begin{pmatrix} 21.6 \\ 998 \end{pmatrix}$ **<== Measured Rx values.**

$$Rp_n := \frac{1}{\left(\frac{1}{Rx_n} + \frac{1}{100400}\right)}$$

Vsn=measured source voltages.

R values paralleled with 100K ======> $Rp_n = \begin{pmatrix} 21.595 \\ 988.177 \end{pmatrix}$

$Vs_n := \begin{pmatrix} 0.701 \\ 2.97 \end{pmatrix}$ $Vg_n := 0 - Vs_n$

$Vg_n = \begin{pmatrix} -0.701 \\ -2.97 \end{pmatrix}$

Use Mathcad "vectorize" operator.

$$Id_n := \overrightarrow{\frac{Vs_n}{Rp_n}\,1000}$$

$Id_n = \begin{pmatrix} 32.461 \\ 3.006 \end{pmatrix}$

Let pinchoff be $u_p := -4$ and $I_{dss} := 48$

$u := -4, -3.9..0$

$$I(u) := I_{dss}\cdot\left(1 - \frac{u}{u_p}\right)^2$$

Try various values for pinchoff and Idss to get a reasonable match.

Blue is data while red shows calculated behavior.

18 Feb, 2006 by W7ZOI

Many thanks to Wes, W7ZOI for this contribution.

QRP SWL HOMEBUILDER

Amateur and Short Wave Radio Electronics Experimenter's Web Site

---

10 MHz WWV Receiver Experiments

New feature: Click on a schematic to load it into a separate browser page for printing

Introduction

For nearly a year, I have been trying to develop a tuned radio frequency (TRF) 10 MHz, WWV, AM receiver. My initial RF stages were common emitter or common source stages with tuned input and output. Despite careful layout, parasitic oscillations plagued these designs and they were discarded. Later, I discovered that only tuning the input of RF stages reduced this tendency towards instability and still provided reasonable selectivity. Different detectors were also tried and evaluated.

A simple receiver that sounds great and is fun to build and experiment with follows. My special thanks to Wes, W7ZOI for performing many of the simulations and providing suggestions which kept me going.

WWV Audio Files from Sept 26, 2006

WWV File 1

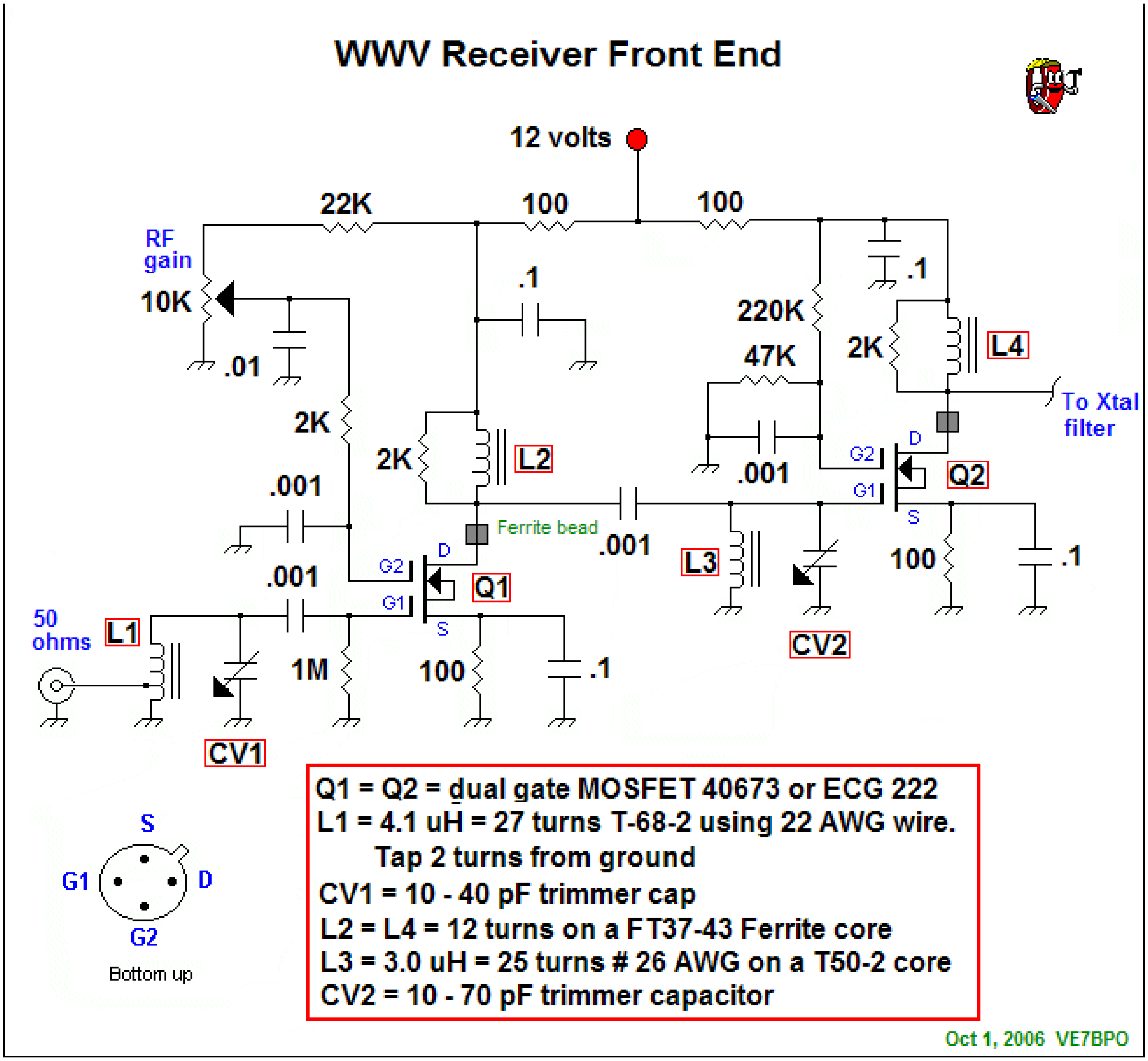
WWV File 2

WWV File 3

The audio was digitally recorded using an electret condenser microphone held 3 cm away from the receiver speaker. The files were compressed using the WMA format.

Supplemental Web Page added June 29, 2007

Receiver Front End



Q1 = Q2 = dual gate MOSFET 40673 or ECG 222
L1 = 4.1 uH = 27 turns T-68-2 using 22 AWG wire.
Tap 2 turns from ground
CV1 = 10 - 40 pF trimmer cap
L2 = L4 = 12 turns on a FT37-43 Ferrite core
L3 = 3.0 uH = 25 turns # 26 AWG on a T50-2 core
CV2 = 10 - 70 pF trimmer capacitor

Oct 1, 2006 VE7BPO

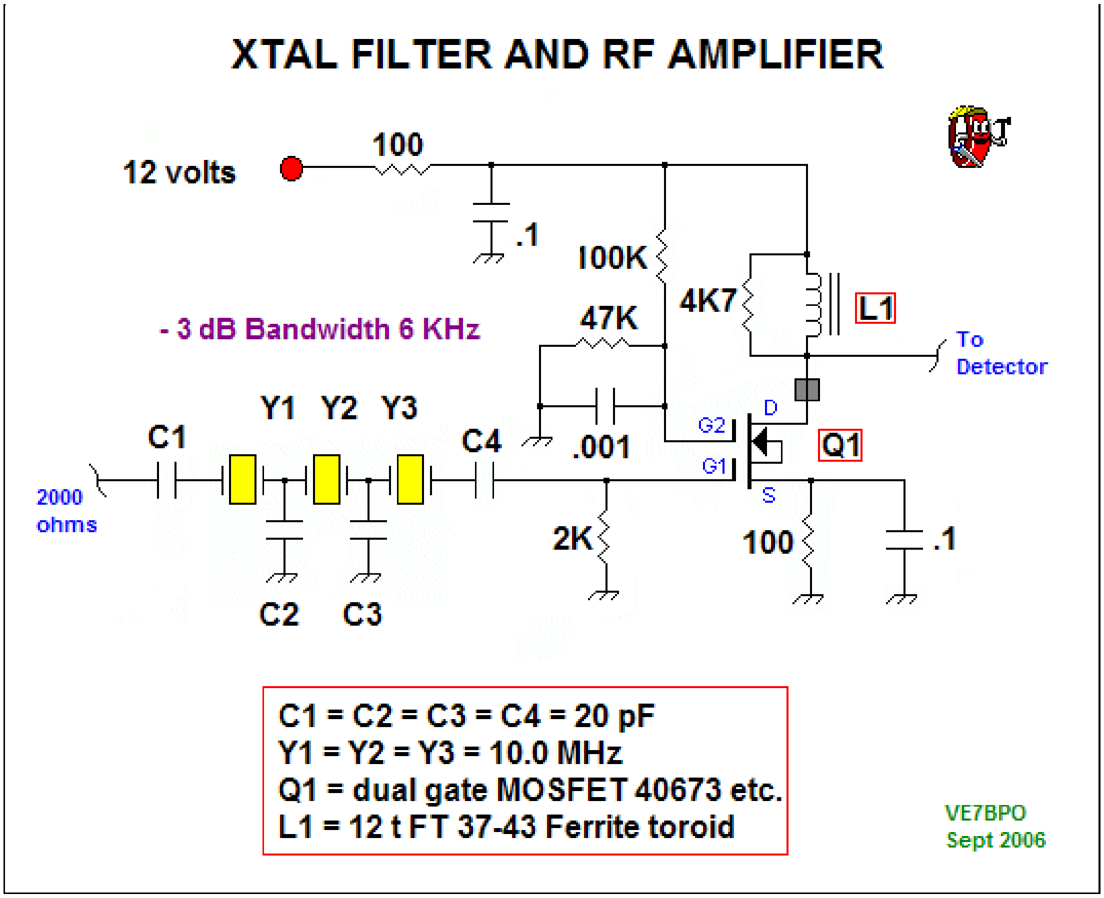
Above schematic. The receiver front end has just 1 single-pole filter. For even greater selectivity (but greater insertion loss), consider moving the L1 tap to 1 turn from ground. My receiver was connected to a 80 meter dipole via an antenna tuner. The antenna tuner provided additional selectivity. No local broadcast band (BCB) signals were heard when the chassis lid was tightened on. You may require additional RF high pass or band pass filtering in your location.

The RF gain control is very basic and only the first 1/3 of the 10K pot is used to go from minimal to maximal gain. Modifications to allow more precise variation in RF gain for dual gate MOSFETs or cascode JFETS are shown in EMRFD. The method shown works fine. For the most part, I keep it set to minimal gain. Using higher gain than necessary, increases receiver noise and may overdrive the detector.

TRF receivers require careful layout. A piece of wire greater than 2-3 cm between the stages may be enough to plague your receiver with local BCB interference depending on your layout and chassis integrity. For interstage connections that had greater than a 2 cm gap, shielded 50 ohm cable was used to prevent BCB interference.

Dual gate MOSFETs provide adequate gain and low noise. you might consider cascoding 2 JFETS for each RF stage if you cannot obtain them. titleernatively, bipolar feedback amplifiers may be used and examples are provided later on this web page.

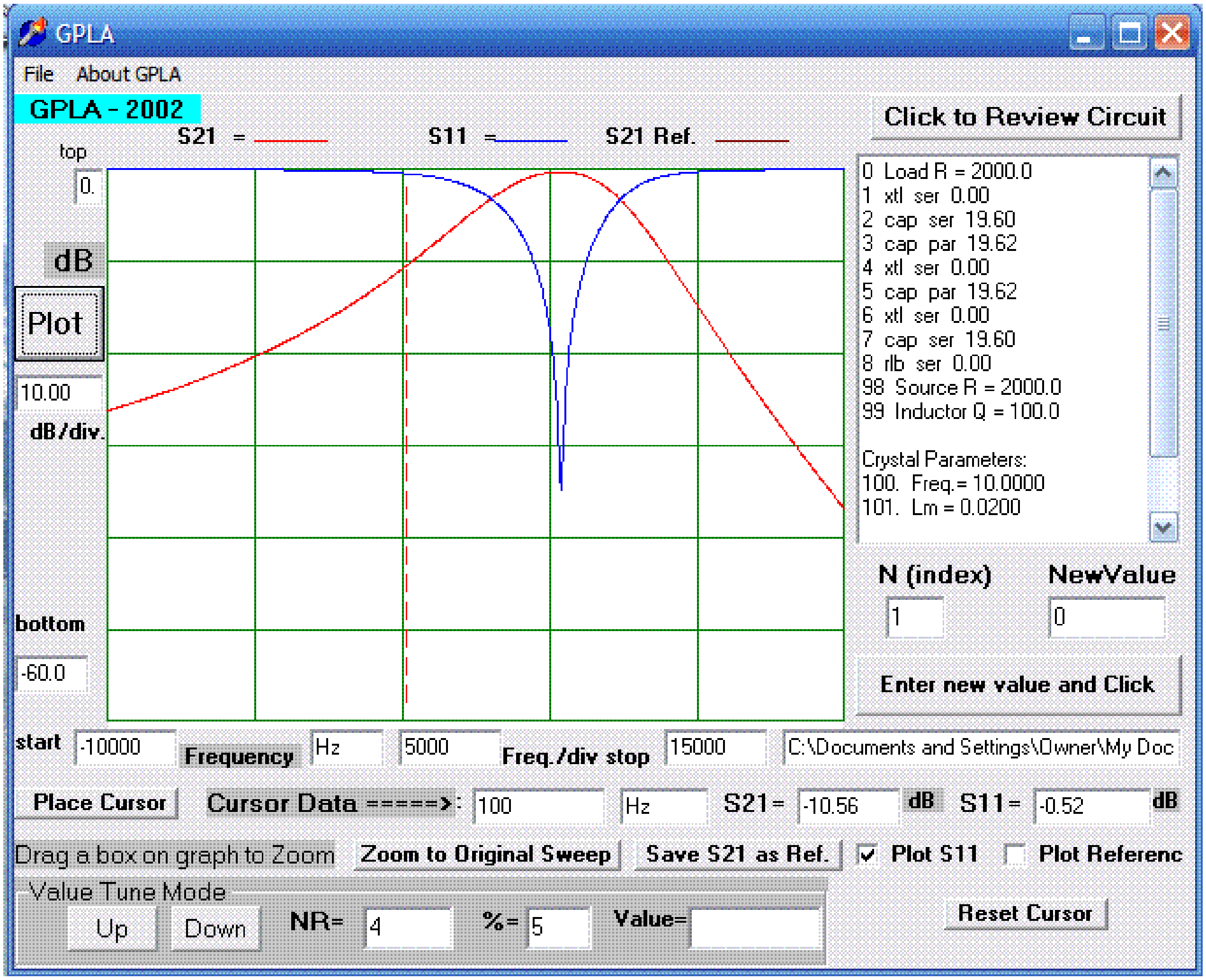
XTal Filter and RF Amplifier



C1 = C2 = C3 = C4 = 20 pF
Y1 = Y2 = Y3 = 10.0 MHz
Q1 = dual gate MOSFET 40673 etc.
L1 = 12 t FT 37-43 Ferrite toroid

Schematic: The Q2 output impedance is 2000 ohms to match the input impedance of the Cohn crystal filter. This filter was designed by Wes, W7ZOI. Matched, computer grade, 10 MHz crystals were used. Choose 10 MHz crystals that are marked for a 20 pF or 32 pF load capacitance if possible. Using a 10 MHz crystal oscillator, find 3 that are closest to one another in frequency.

You may substitute 2K2 resistors instead of the specified 2K with a slight penalty in pass band shape.



```
0  Load R = 2000.0
1  xtl  ser  0.00
2  cap  ser  19.60
3  cap  par  19.62
4  xtl  ser  0.00
5  cap  par  19.62
6  xtl  ser  0.00
7  cap  ser  19.60
8  rlb  ser  0.00
98  Source R = 2000.0
99  Inductor Q = 100.0

Crystal Parameters:
100.  Freq.= 10.0000
101.  Lm = 0.0200
```

top 0.

10.00 dB/div.

bottom -60.0

N (index) 1

NewValue 0

start -10000 Hz

Freq./div 5000

stop 15000

C:\Documents and Settings\Owner\My Doc

Cursor Data =====>: 100 Hz

S21= -10.56 dB

S11= -0.52 dB

☑ Plot S11

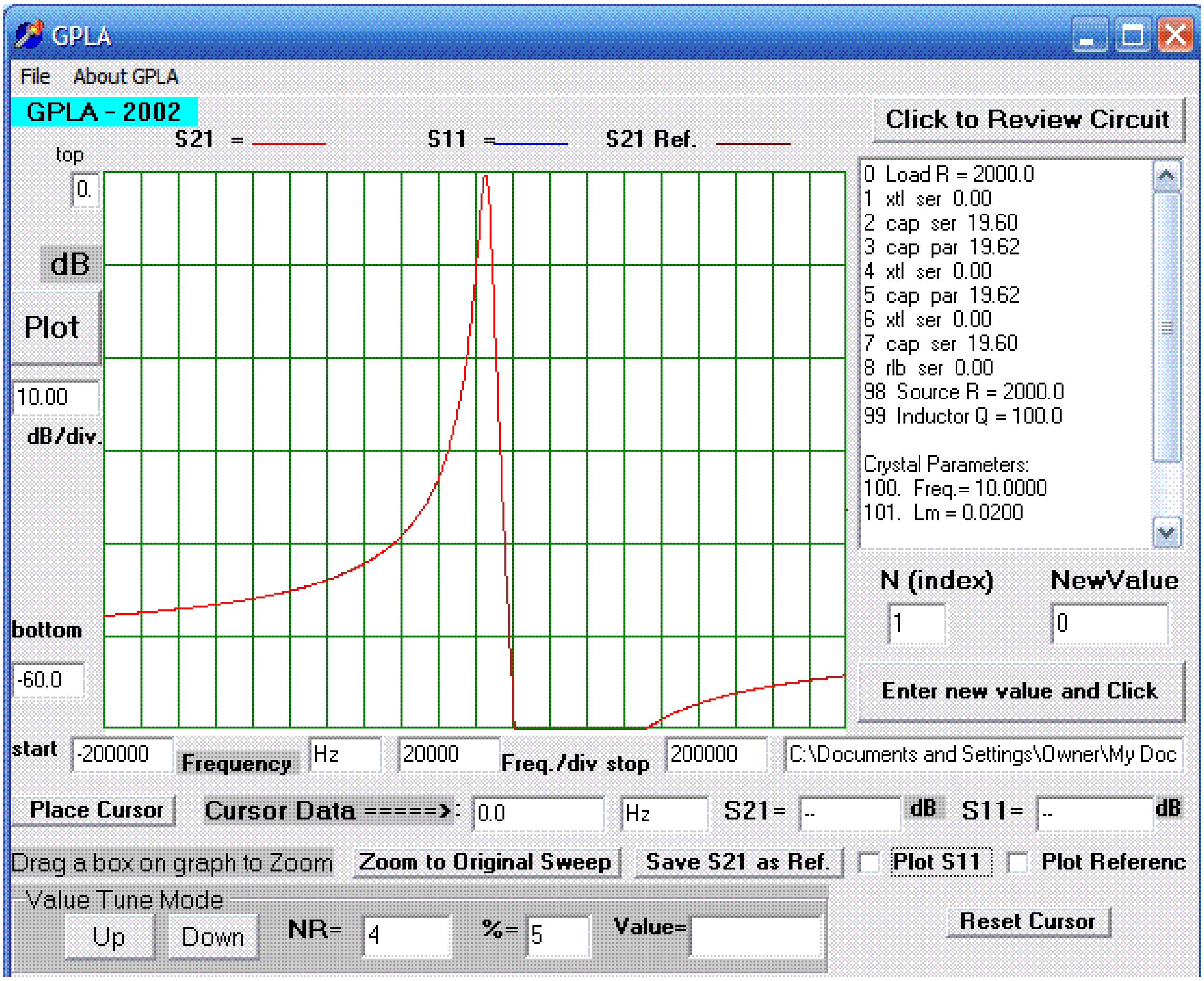
☐ Plot Referenc

NR= 4

%= 5

Above graphic. A simulation of the receiver crystal filter using GPLA, a program written by Wes, W7ZOI that comes with EMRFD. EMRFD is the major reference for this web site and I recommend that you add this book and companion software to your home library. The pass band is not symmetrical. It is mistuned for the lower pass band frequencies and would serve better as an upper sideband filter. Nevertheless, it works reasonably well and is simple to build and tolerant to component variation and match. At certain times, a very strong shortwave station at 9.985 MHz can be heard along with WWV. This usually occurs in the early evening when the WWV signal is not that strong at my location. For most of the day and night, whether WWV is present or not, very little interference has been detected.

Bypassing the crystal filter is an interesting experiment. As many as 5 stations were heard simultaneously and these varied as time passed. I heard Radio Vatican, Radio Habana and many other broadcasts during 1 evening. At one point I heard a station at 9.75 MHZ, WWV and a strong CW carrier at 10 .110 MHz!

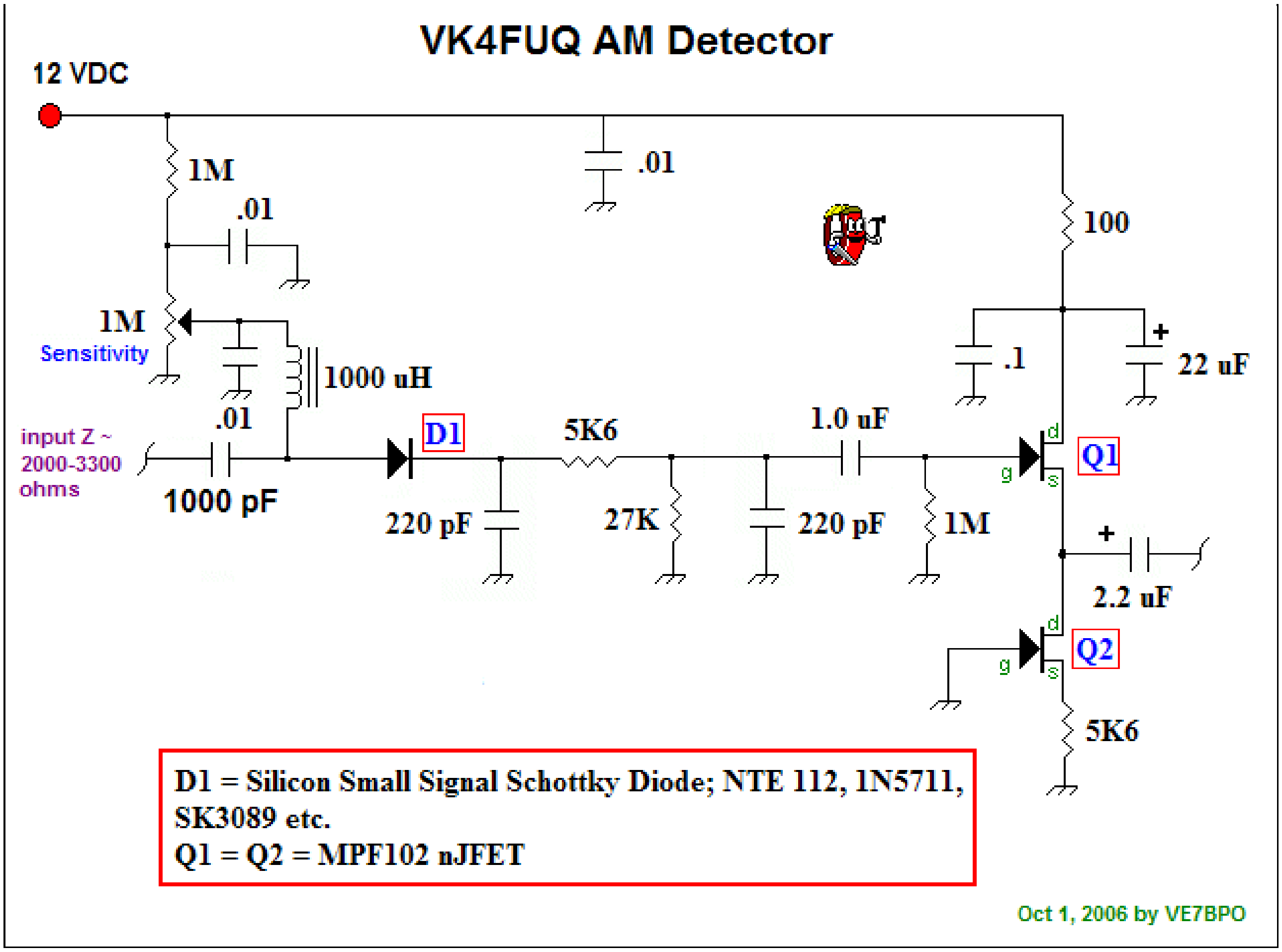


0 Load R = 2000.0
1 xtl ser 0.00
2 cap ser 19.60
3 cap par 19.62
4 xtl ser 0.00
5 cap par 19.62
6 xtl ser 0.00
7 cap ser 19.60
8 rlb ser 0.00
98 Source R = 2000.0
99 Inductor Q = 100.0

Crystal Parameters:
100. Freq.= 10.0000
101. Lm = 0.0200

top: 0.
dB/div.: 10.00
bottom: -60.0
N (index): 1
NewValue: 0
start: -200000 Frequency: Hz
Freq./div: 20000
stop: 200000
C:\Documents and Settings\Owner\My Doc
Cursor Data =====>: 0.0 Hz
S21= -- dB
S11= -- dB
NR= 4
%= 5
Value=

Above graphic. This is a sweep that goes from 200 kHz below to 200 kHz above 10 MHz to show the stop band response of this filter. This filter has a pretty decent response considering the low cost and effort involved.



VK4FUQ AM Detector

D1 = Silicon Small Signal Schottky Diode; NTE 112, 1N5711, SK3089 etc.
Q1 = Q2 = MPF102 nJFET

Oct 1, 2006 by VE7BPO

Above schematic. This detector is fabulous. It was designed by Felix Scerri, VK4FUQ. He has a web page explaining his high fidelity detectors at the Elliot Sound Products (ESP) site: http://sound.westhost.com/articles/am-radio.htm

The ESP web site is a personal favorite. Rod Elliot has one of the best do-it-yourself electronic web sites available. The main URL for his site is http://sound.westhost.com/index2.html

My sincere thanks to Rod and Felix for permission to present Felix's detector on this web page.

His improved AM detector has 3 positive advantages; it has high bandwidth, low distortion and incredible (and variable) sensitivity. I cannot get over how nice this detector sounds compared to others I have built and analyzed during weak and strong signal testing. The variable bias control allows the listener to adjust the bias to maintain detected audio fidelity even when the RF signal is weak.

This detector uses a UHF mixer diode often found in older television sets. Increasing the diode bias from O volts towards maximum causes three things to happen:

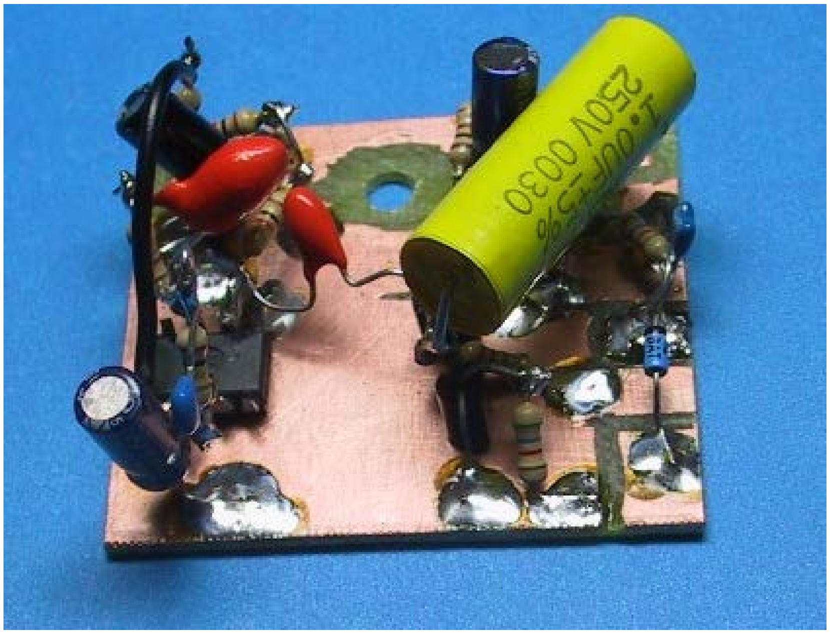
1. Increased sensitivity.
2. Increased audio high frequency response.
3. Slight increase in receiver noise.

When the WWV RF signal is weak, turning the bias off may result in the detected WWV signal disappearing. Increasing the bias will bring WWV back in. I generally run the bias control pot about 1/2 way and of course, higher as WWV fades out. I like the fidelity that the bias adds even when the WWV signal is strong. Note how the WWV audio quality continues to be high in fidelity as WWV fades out in this sound file.

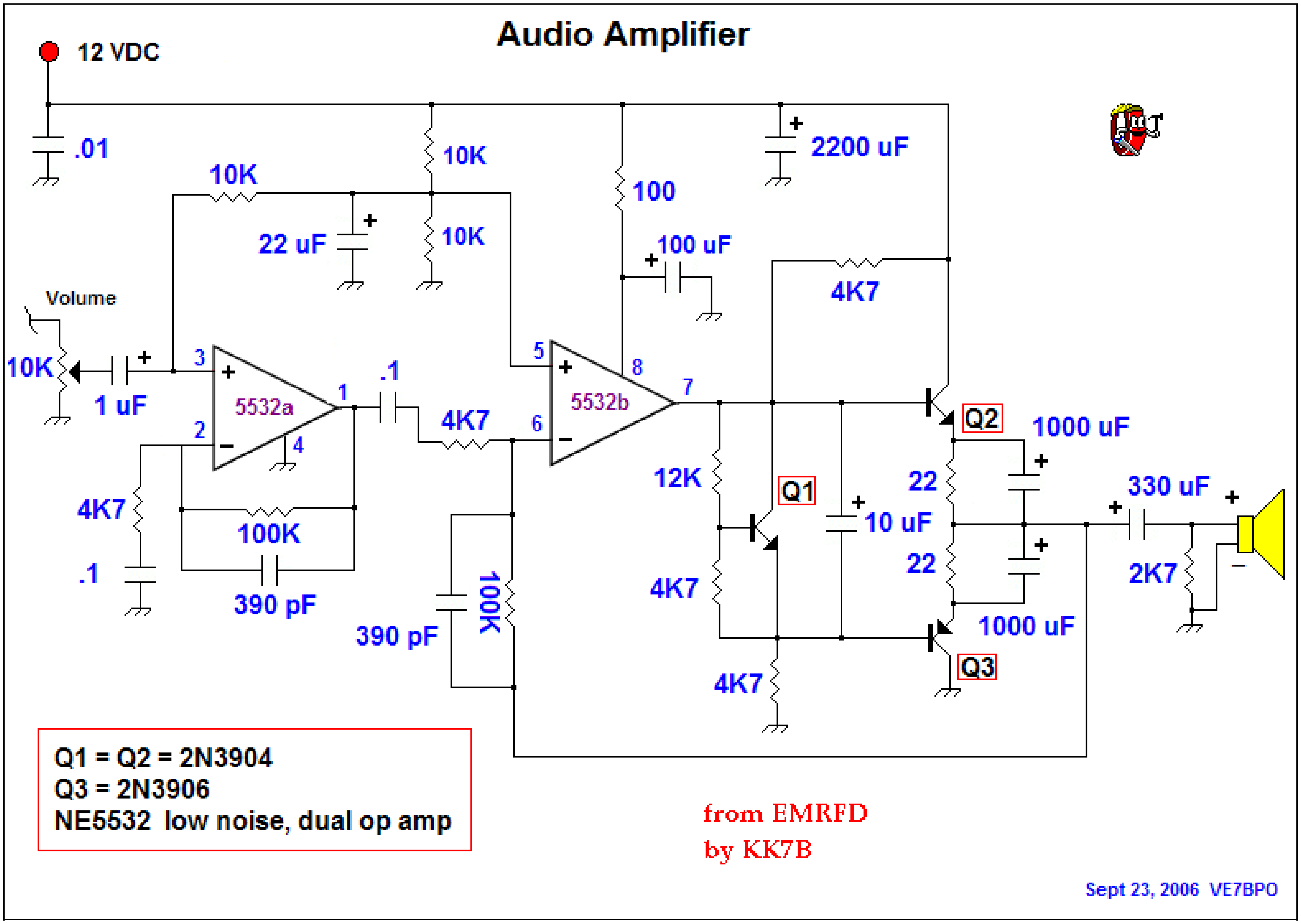
Felix called for a 1 mH radio frequency choke. The largest I had in stock was a 1000 uH choke. I had to decouple it as shown to prevent oscillations from occurring in my receiver. For the 1 uF and 2.2 uF capacitors, I used polyester film types which sounded better than electrolytic capacitors.

Oct 13, 2006: Note. The 1000 pf input cap to the detector was omitted in error in the original schematic which is now correct.

To the right: The detector board. On the left is an op amp preamp stage that was later disconnected as it was not needed. Note the copper is removed where the chassis mounting nut contacts the copper board. Both audio boards were isolated from chassis ground and star grounded to a single point. The speaker negative terminal was also directly connected to this point. There is no hum.



Audio Amplifier



Above schematic. Rick, KK7B designed this low noise audio amplifier. It is from EMRFD. This superb AF amp greatly compliments the VK4FUQ detector. This is the best speaker audio amp under 1 watt I have ever used. Distortion is very low as long as it is not over-driven. I increased some capacitor values compared to the original schematic. Please refer to EMRFD for details on this stage. The chassis of this receiver greatly increases the low frequency response. On the 1 second pulses of WWV, the receiver "knocks" like a metronome. This does not occur when the chassis lid is off.

WWV web site: http://tf.nist.gov/stations/wwv.html All the often subtle pulses and tones transmitted at various times during the hour can be heard with this receiver.

To the right: A bread board of the AF amp. This is the audio amp I shall use in future projects which contain a speaker. Kudos to KK7B.

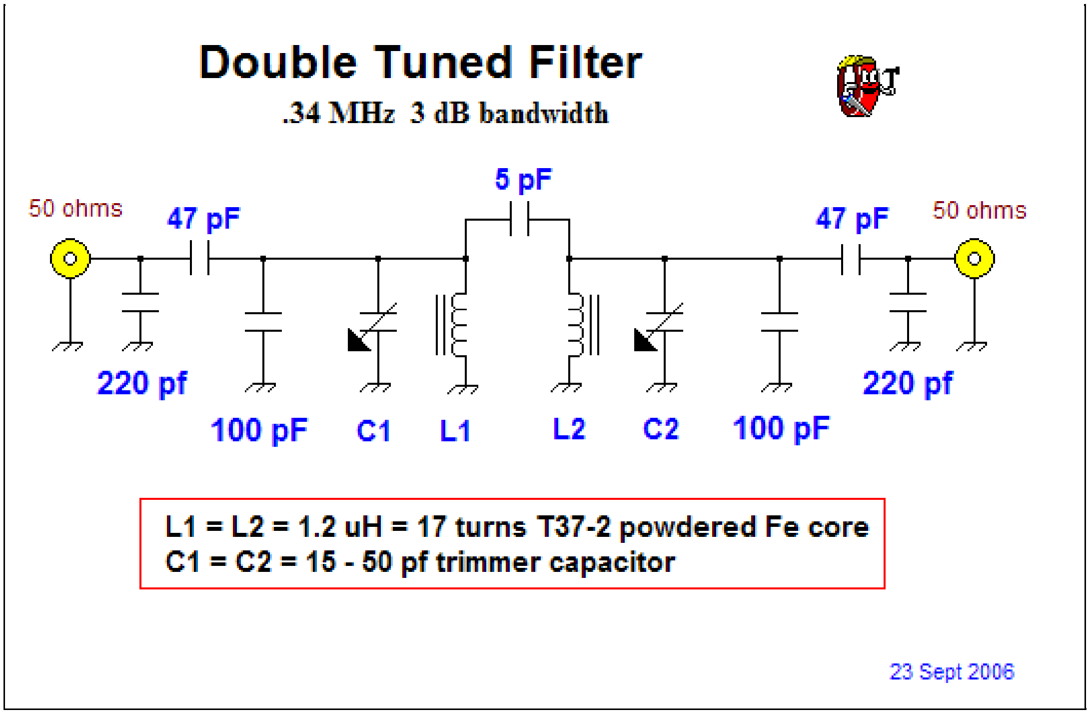
Below 3 images: Different views of the TRF receiver. On the front from left to right are the bias "sensitivity" control, volume control with integral power switch and blue LED "power on" indicator.

On the rear from left to right are the 12 VDC input jack, an unused switch (was an -10 dB attenuator at 1 point), the RF gain control and a coaxial SO239 connection.

Further Experiments

What follows are some of the ideas and circuits tried over the past year.

Schematic to the right: A 10 MHz, double tuned RF band pass filter that may be used ahead of the receiver. Insertion loss is ~ 3 dB and this filter uses a 5 pF coupling capacitor which are not too difficult to find. Filters with bandwidths of 150 - 180 KHz were also tested.



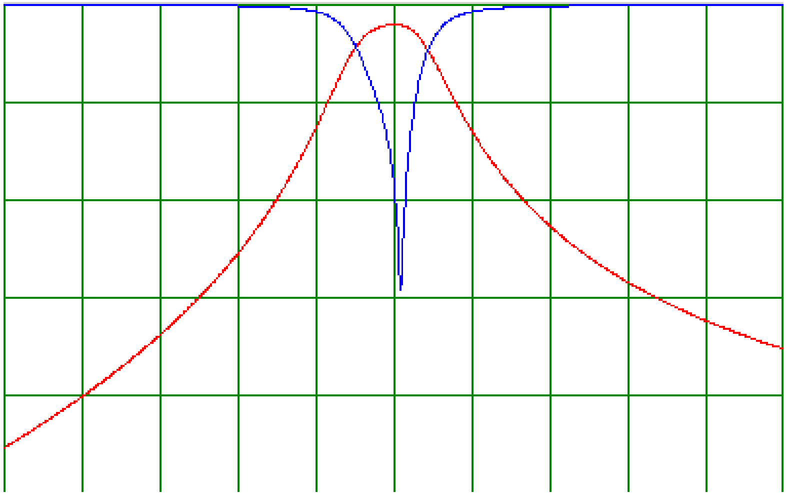
Double Tuned Filter

.34 MHz 3 dB bandwidth

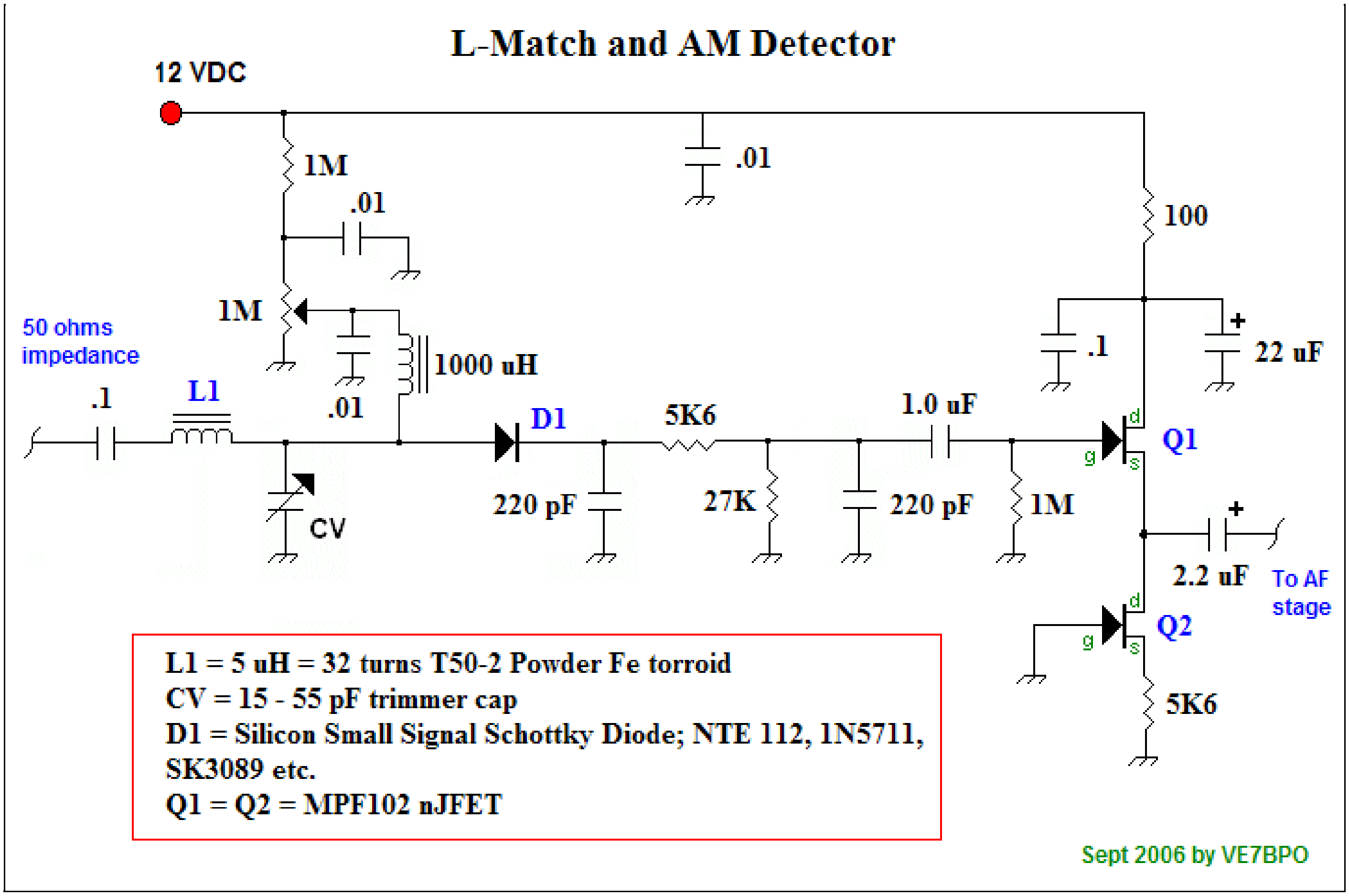
L1 = L2 = 1.2 uH = 17 turns T37-2 powdered Fe core
C1 = C2 = 15 - 50 pf trimmer capacitor

23 Sept 2006

To the right: A GPLA simulation of the popcorn DTC shown above. titlehough a little mistuned, it is reasonable for a filter that uses common junk box values and has low insertion loss.



L-Match AM Detector

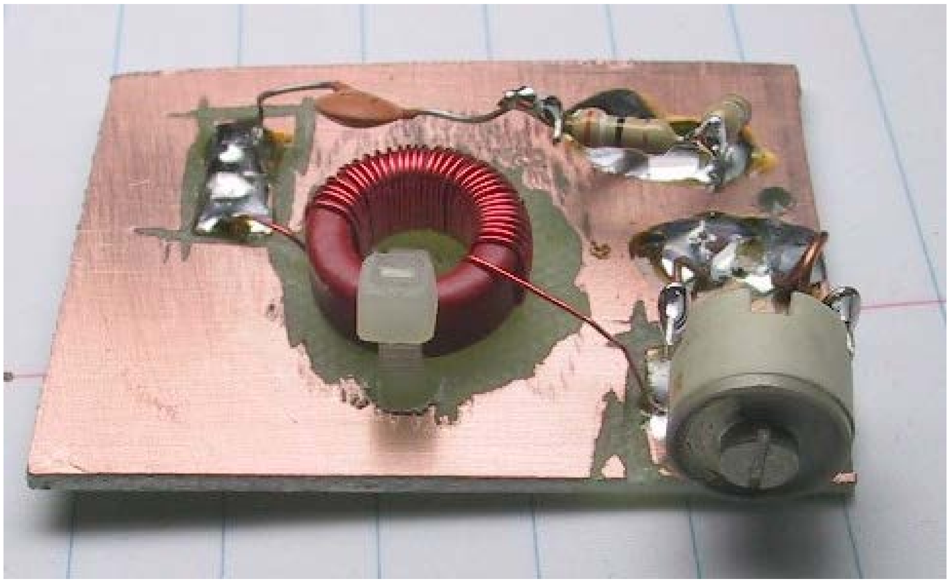


Above schematic. The VK4FUQ detector can also be used to follow a 50 ohm output impedance stage by using an L-match as shown. The L match tunes very sharply. I peaked the L-match with the bias at 0 volts.

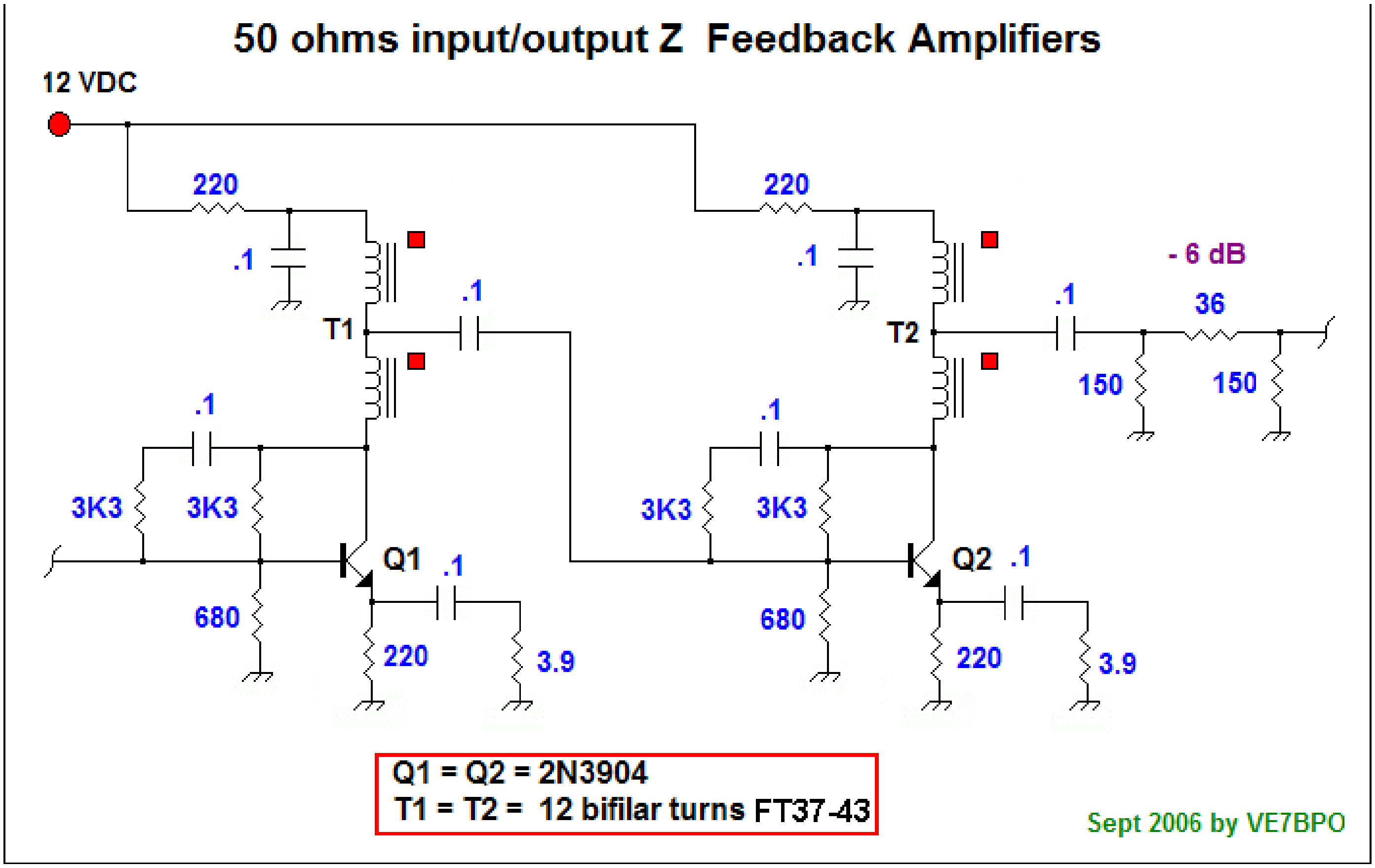
The input impedance of the detector is related to the DC current flowing in the diode. This is established by the adjustable bias current or "sensitivity control". The input resistance will be 26/I, where I is the current in mA.

For example, if the current in the diode is 10 microamps (0.01 mA) the input Z is 2600 Ohms. I have found that any input Z value from 2000 to 5100 ohms worked well with this detector.

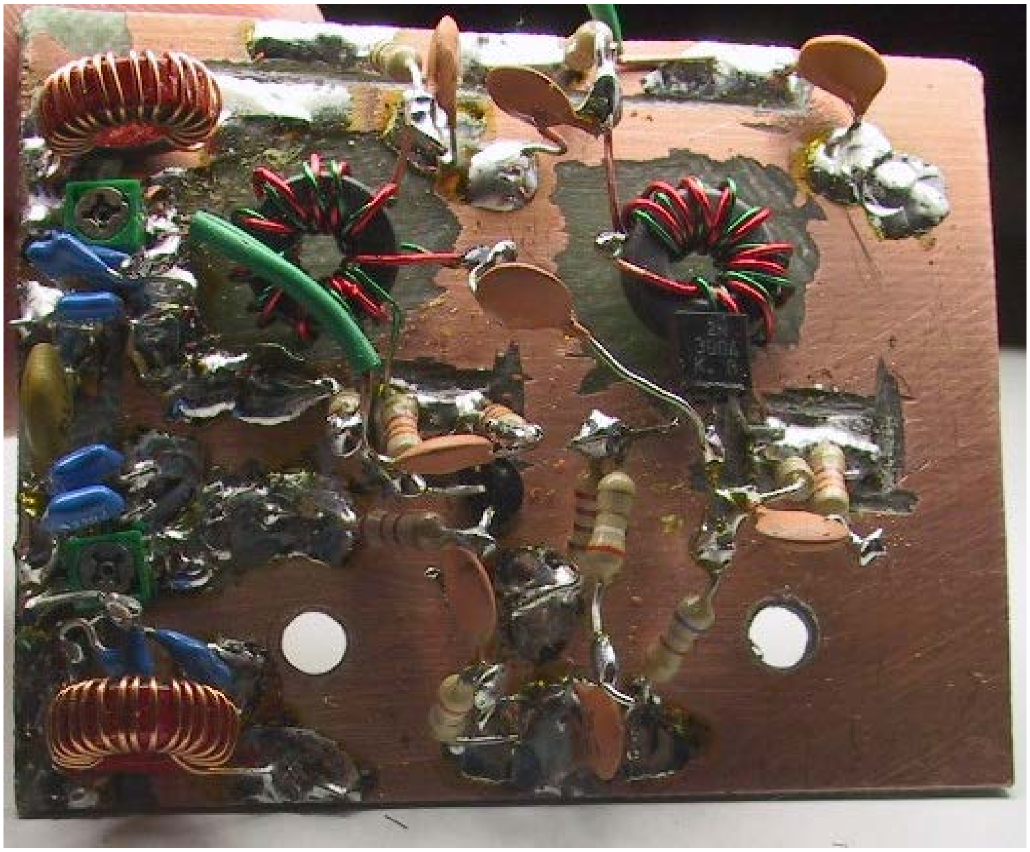
Image to the right: A photograph of the L-match connected to a - 6dB 50 ohm pad which terminated the 50 ohm feedback amplifier that drove the L-match.



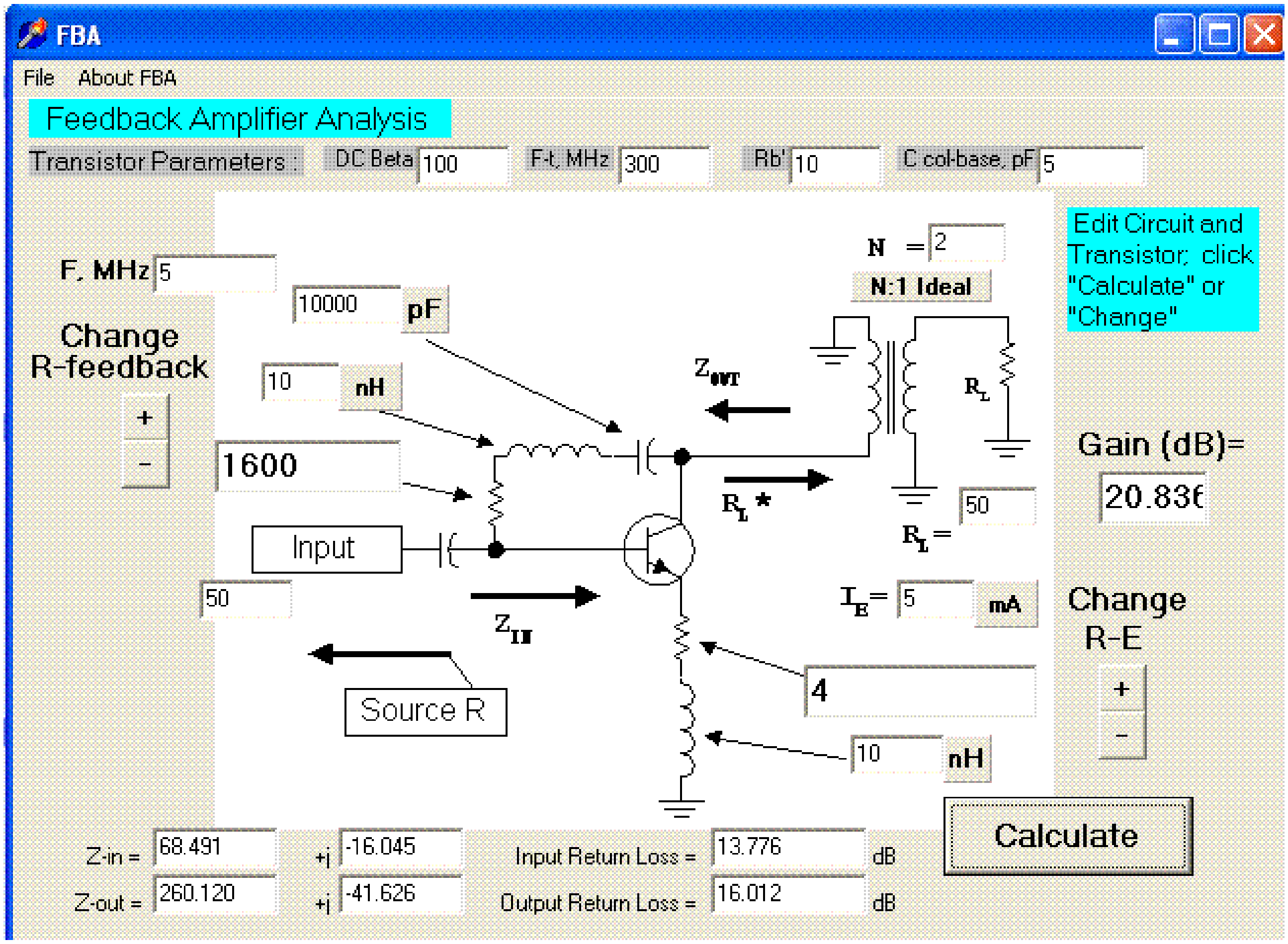
Feedback amplifiers



Above schematic. Feedback amplifiers may be used as RF amplifiers for a TRF receiver. This stage followed the crystal filter in one version of my TRF receiver. Stability was excellent. This feedback amplifier was designed by Wes, W7ZOI. It has ~ 20 dB gain and draws a little over 5 mA current.



Above graphic. 2 feedback amplifiers are shown on this breadboard. The double tuned filter (DTC) shown earlier is also built on this board. In this version, the crystal filter was omitted and replaced with the DTC.



Feedback Amplifier Analysis

Transistor Parameters : DC Beta 100 F-t, MHz 300 Rb' 10 C col-base, pF 5

F, MHz 5

10000 pF

10 nH

1600

50

N = 2

R_L = 50

I_E = 5 mA

4

10 nH

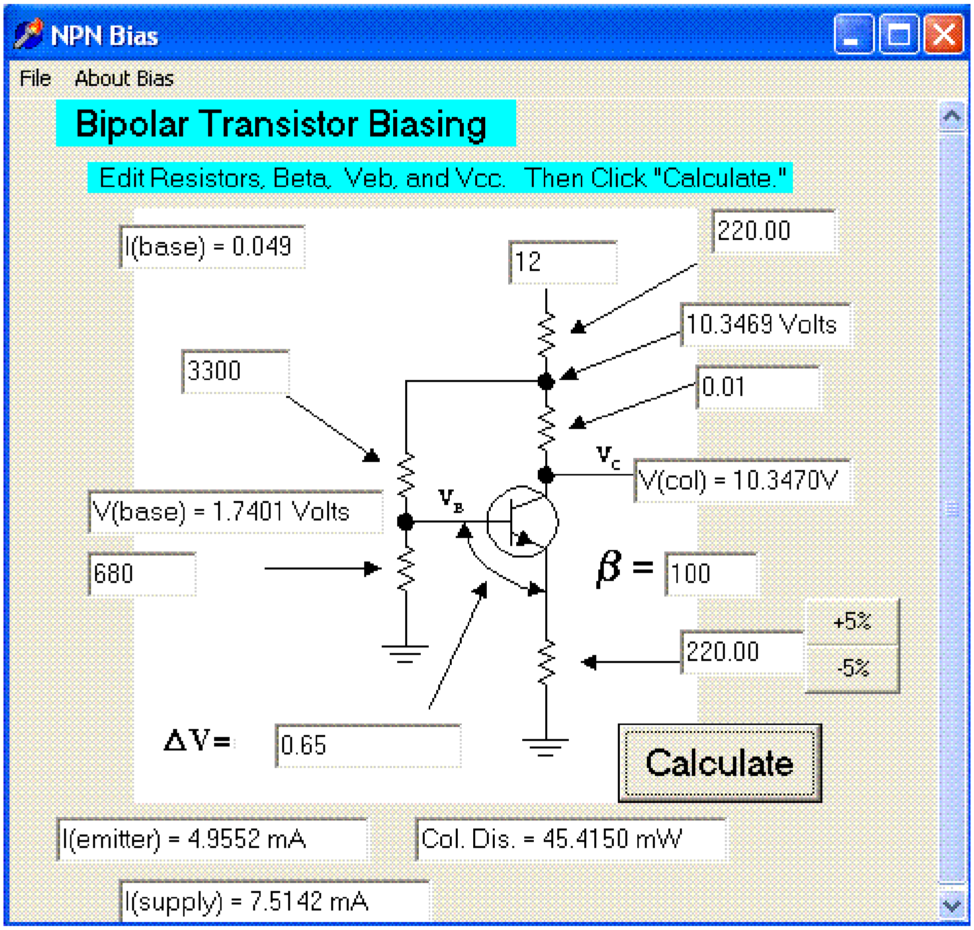
Edit Circuit and Transistor; click "Calculate" or "Change"

Gain (dB)= 20.83

Z-in = 68.491 +j -16.045 Input Return Loss = 13.776 dB

Z-out = 260.120 +j -41.626 Output Return Loss = 16.012 dB

Above graphic. Using software that ships with EMRFD, W7ZOI designed the feedback amplifier used in this version of the receiver.



Bipolar Transistor Biasing

Edit Resistors, Beta, Veb, and Vcc. Then Click "Calculate."

I(base) = 0.049

12

220.00

10.3469 Volts

0.01

3300

V(col) = 10.3470V

V(base) = 1.7401 Volts

680

β = 100

220.00

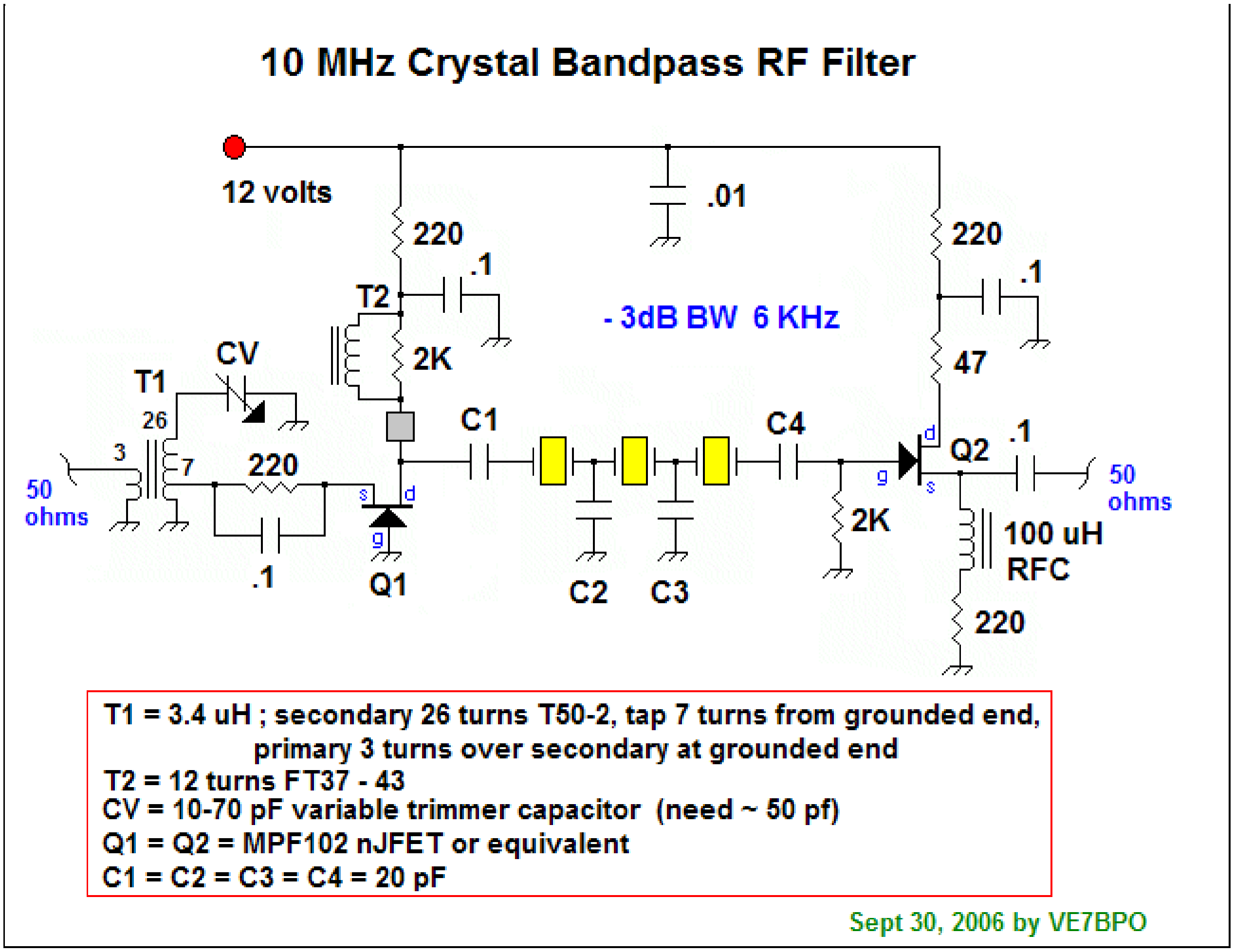
ΔV= 0.65

I(emitter) = 4.9552 mA

Col. Dis. = 45.4150 mW

I(supply) = 7.5142 mA

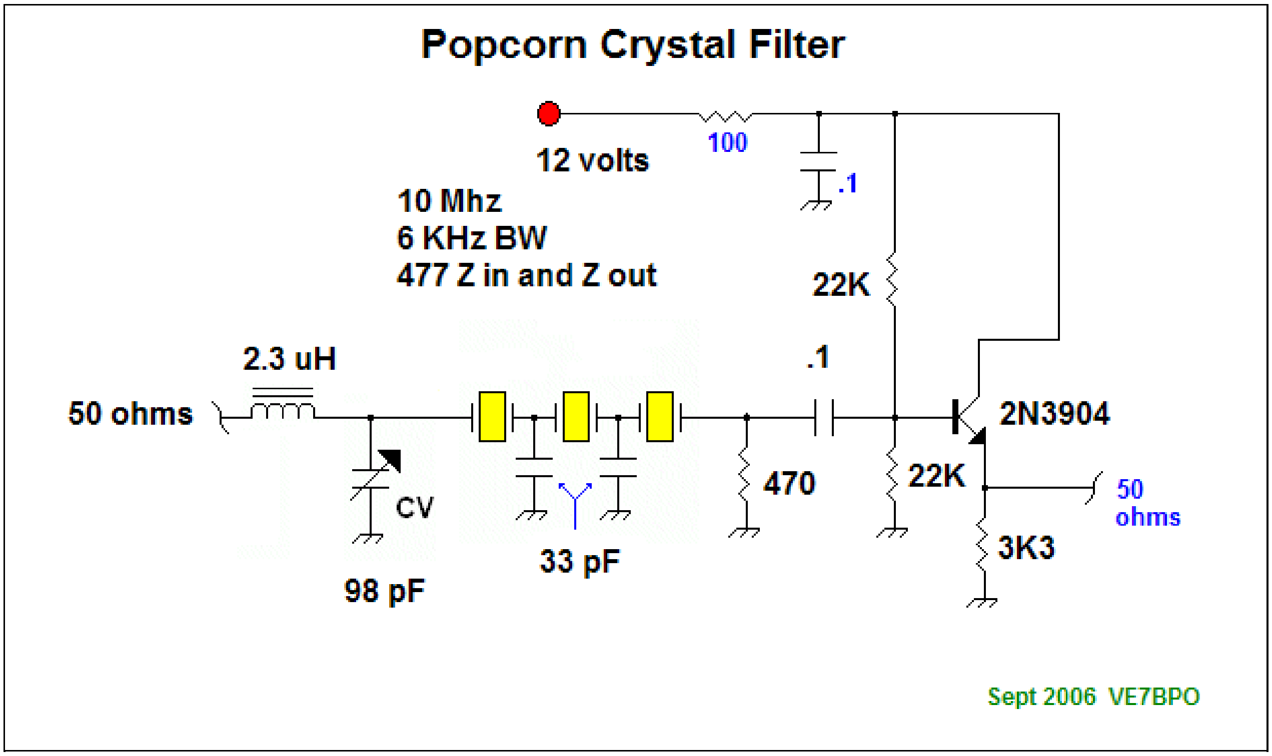
Above graphic. The feedback amplifier bias resistor values were also calculated using software written by W7ZOI and included with EMRFD.



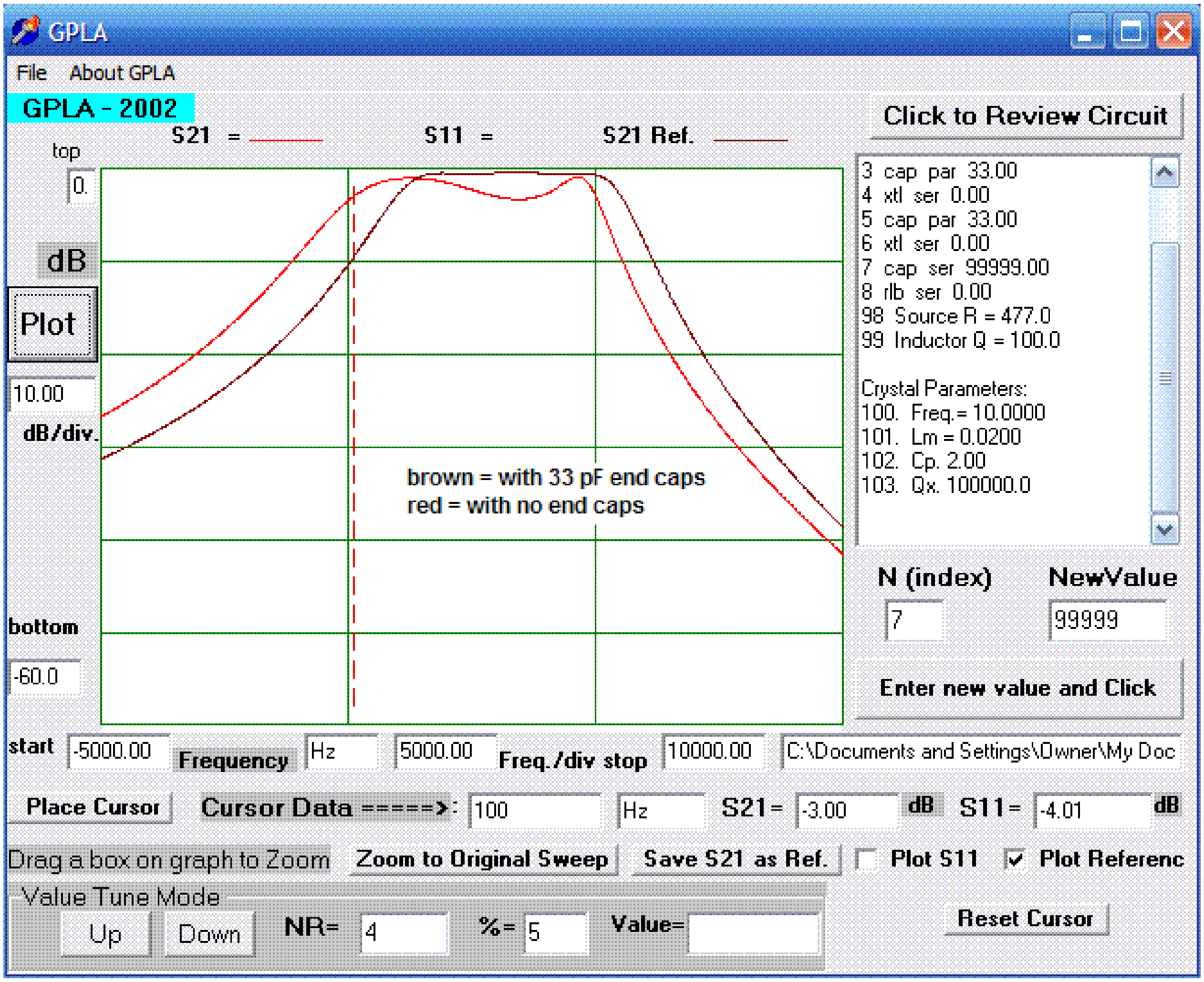
T1 = 3.4 uH ; secondary 26 turns T50-2, tap 7 turns from grounded end, primary 3 turns over secondary at grounded end
T2 = 12 turns FT37 - 43
CV = 10-70 pF variable trimmer capacitor (need ~ 50 pf)
Q1 = Q2 = MPF102 nJFET or equivalent
C1 = C2 = C3 = C4 = 20 pF

Sept 30, 2006 by VE7BPO

Above schematic. This circuit has the crystal filter matched to 50 ohms input and output stages using JFETs. The JFETs also serve to provide a little more gain. Careful layout is required to reduce BCB interference for all stages in a TRF receiver.



Above schematic. This is the original crystal filter that I designed. The input and output impedance is 477 ohms. The input was matched to the preceding 50 ohm stage using an L-network. A emitter follower is used to match the output to the 50 ohm stage which followed. Later, the emitter follower was replaced with the source follower (with a 470 ohm gate resistor) that is shown in the schematic directly above. The source follower had greater immunity to BCB interference and provided a better termination for this filter. This popcorn filter worked well, titlehough occasionally there was another station in addition to WWV, in the pass band. This also happened with the filter used in the final version of this receiver.

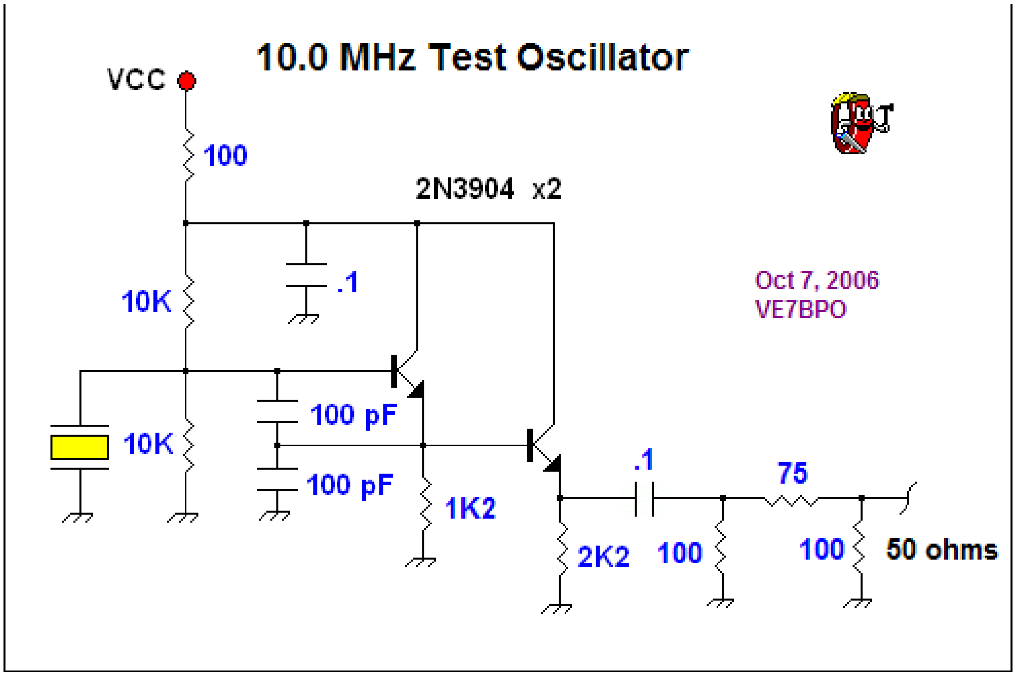


```
3  cap  par  33.00
4  xtl  ser  0.00
5  cap  par  33.00
6  xtl  ser  0.00
7  cap  ser  99999.00
8  rlb  ser  0.00
98  Source R = 477.0
99  Inductor Q = 100.0

Crystal Parameters:
100.  Freq.= 10.0000
101.  Lm = 0.0200
102.  Cp. 2.00
103.  Qx. 100000.0
```

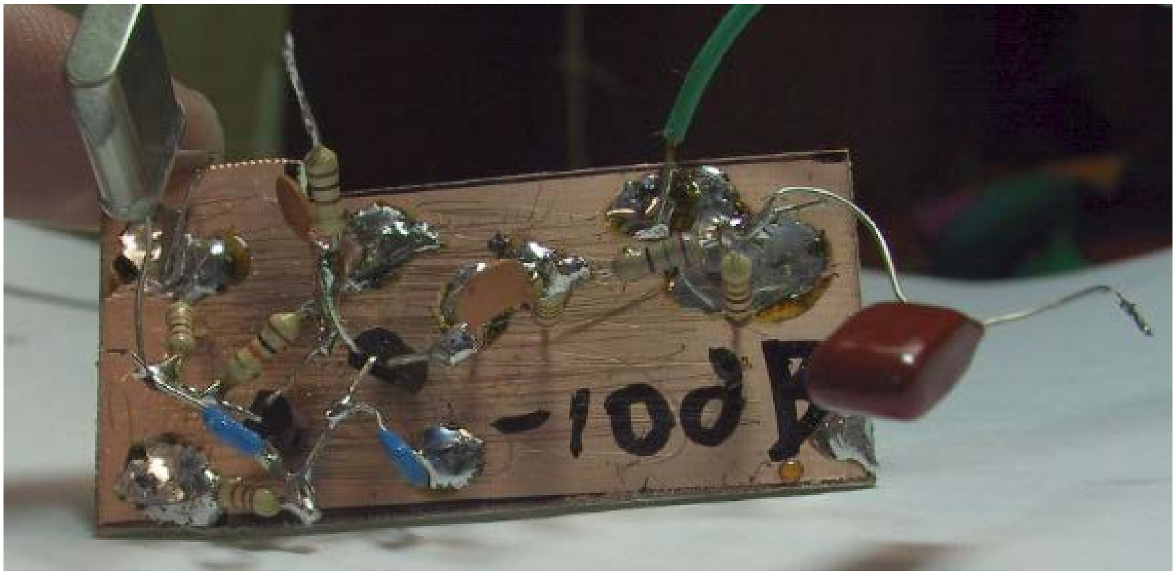
top 0.
10.00 dB/div.
bottom -60.0
N (index) 7
NewValue 99999
start -5000.00
Hz
5000.00 Freq./div
stop 10000.00
C:\Documents and Settings\Owner\My Doc
Cursor Data =====>: 100 Hz
S21= -3.00 dB
S11= -4.01 dB
☐ Plot S11
☑ Plot Referenc
NR= 4
%= 5
Value=

Above graphic. Here are GPLA simulations of the popcorn filter with and with out 33 pF series end capacitors which serve to tune the filter. The brown tracing illustrates that it is better to include a series 33 pF cap at each end of the "popcorn" crystal filter. I did not use this filter because the dual gate MOSFET RF amps used in the final version, have better gain driving or following the 2000 ohm filter designed by W7ZOI.

GPLA is a "must-have" program. You can "tune" filters with different or asymmetric input/output impedances.

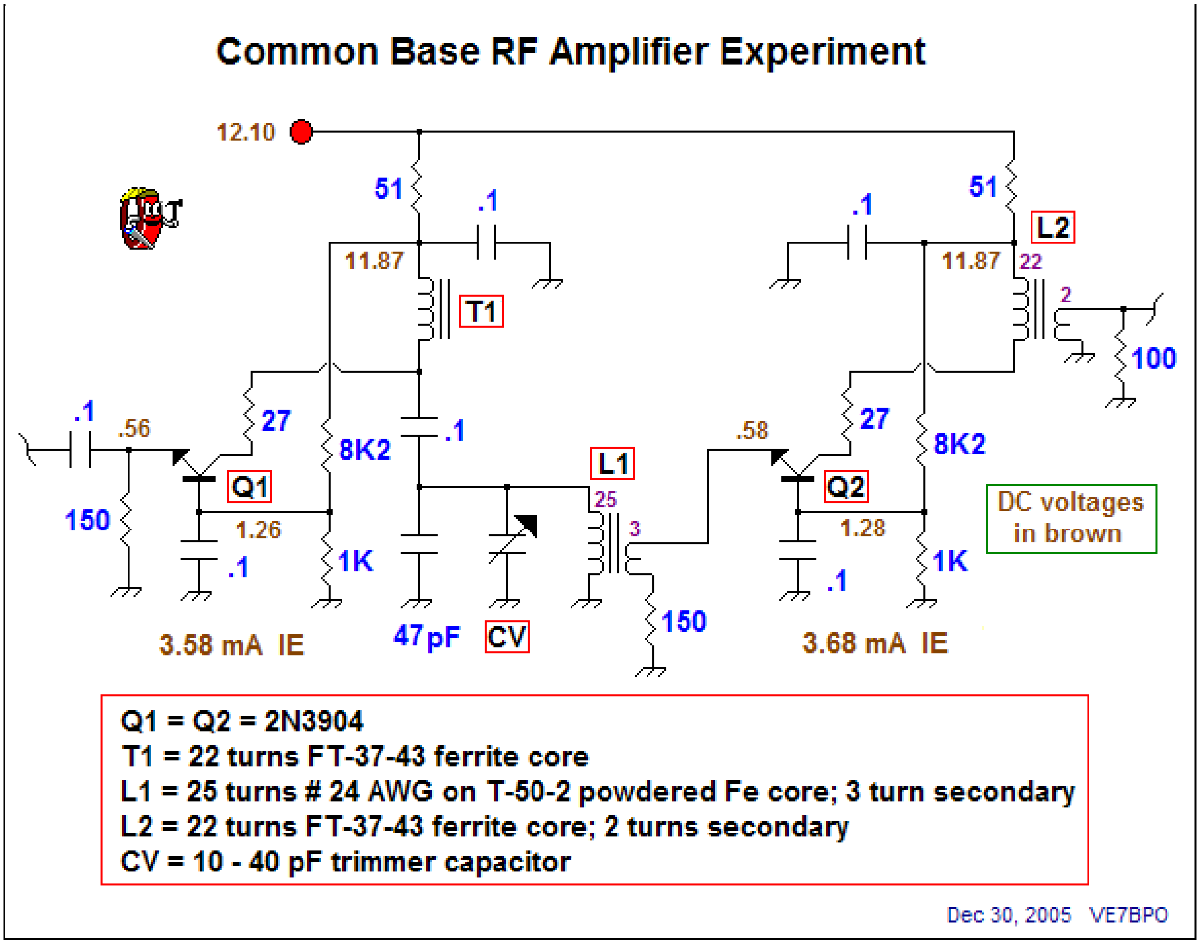


Above schematic. This 10.0 MHz crystal oscillator has a - 10 dB, 50 ohm pad on the output and was used to match the crystals, test the RF amps and align the filters used in these experiments.



Above graphic. A breadboard of the test oscillator shown above.

Above graphic. Some of the bread boards developed during experimentation. My final receiver layout (and potentiometer positioning) is not optimal, however this is a prototype and I had no idea what the finished version would look like.



Common Base RF Amplifier Experiment

Q1 = Q2 = 2N3904
T1 = 22 turns FT-37-43 ferrite core
L1 = 25 turns # 24 AWG on T-50-2 powdered Fe core; 3 turn secondary
L2 = 22 turns FT-37-43 ferrite core; 2 turns secondary
CV = 10 - 40 pF trimmer capacitor

Dec 30, 2005 VE7BPO

Above schematic. This is one RF amp that was built for this receiver. The turns ratios on L1 is too drastic to afford much gain.

Above graphic. I have been told many times that my breadboards are very ugly looking. This breadboard of the schematic directly above, shows that occasionally, I can build a nice looking circuit!

Conclusion

The highlights of these experiments were VK4FUQ's detector and KK7B's AF amplifier.

When constructing such a receiver, *build backwards*. Install the speaker and then build and test the AF amp. Test it by touching your finger to the input and listening for noise or BCB radio. Turn the 10K pot and verify that the noise increases or decreases appropriately. Perhaps test it using an AF oscillator.

If it works, you get immediate positive feedback and motivation to continue. If it does not work, you only have 1 stage to trouble shoot.

Next, build the detector. To test it, connect a piece of wire about 25 cm long between the RFC and the anode of the diode. You should then hear local BCB radio. Slowly turn the bias potentiometer from 0 to fully on. Notice how increasing the bias may bring in 2 or more stations compared to when it was at 0. Also notice how it changes the tone and sensitivity of the detector. Try shortening the "test antenna" and observing how sensitive this detector is with the bias increased.

If all went well, you now have an AM radio!

Next add in the Q3 RF amp and again test it using a short piece of wire. Then continue on until you arrive at the antenna connection for your receiver.

Best regards, VE7BPO

Amateur and Short Wave Radio Electronics Experimenter's Web Site

MF and HF Receiving Antenna

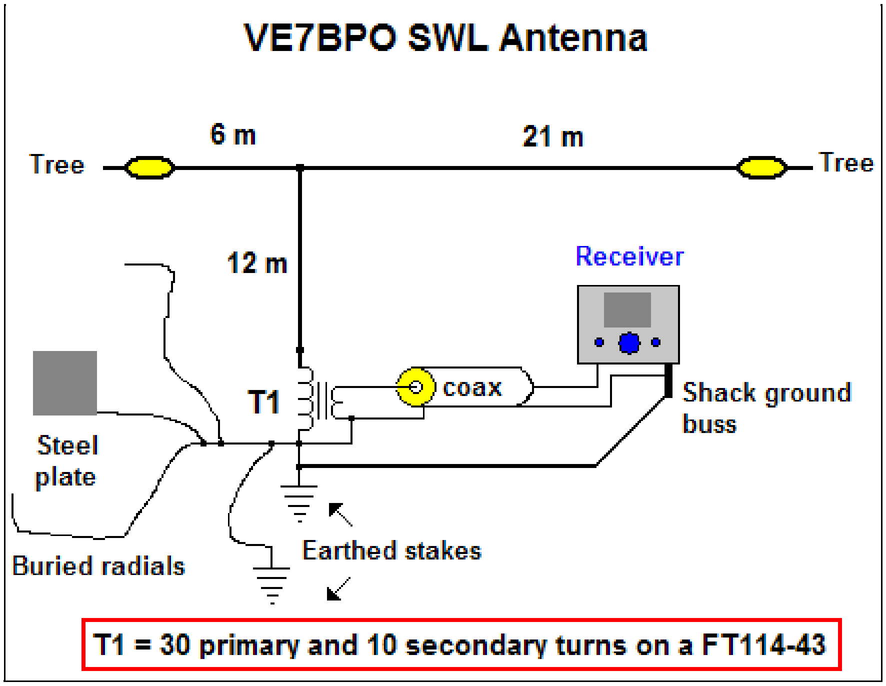
Introduction

My first shortwave antenna was a simple end-fed wire which started at my bedroom window and extended out horizontally to a tree which was 25 feet away from our house. The antenna feed line was a short piece of wire that connected to the near end of the antenna and entered the house through a small hole I made in my wooden window sill. This feed line was directly connected to my receiver's high impedance antenna input. My station ground was long piece of wire that was connected to a copper pipe located in the bathroom next door. While this antenna brought in "the world" to my bedroom, it was extremely noisy. Directly connecting your antenna feed line and house ground system to your receiver are not good RFI reduction practices. This web page will explore some experiments in trying to minimize the Radio Frequency Interference (RFI) arising from my local environment.

Indoor RFI sources are usually plentiful. Electrical appliances such as washing machines, televisions, DVD players, computers and electrical wiring may all emit RFI which your antenna, or directly connected house ground system may pick up and feed to your receiver. Certain indoor devices may be really strong RFI sources and will have to be eliminated or decoupled. Outside of your house are also potential sources of RFI. These may include such things as power transformers, electric fence and garage door openers. RFI location and reduction is out of scope for this web page, however a good place to learn more is the ARRL RFI book (out-of-print: search on Amazon). To find RFI sources in your home and neighborhood, try using a battery powered AM radio. At my QTH, I located a noisy VCR inside the house my Grundig S350. We rarely use this VCR and now just leave it unplugged until we actually need to operate it. I tuned the receiver to an empty frequency and found this VCR by trial and error. **Please note this web page is concerned with feeding a shortwave listening antenna and does not describe providing protection against lightning.** For web sites which covers lightening plus RF ground please refer to this offering from W8JI or eHam.net.
Protect your home and family from lightening !!

Outdoor MF and HF Antenna

The schematic to the left summarizes the outdoor VE7BPO MF and HF receiving antenna system for summer 2007. Although modest for a big city lot, this antenna seems to pull in the DX and is relatively free of RFI. This antenna was just a case of "putting as much wire in the sky as possible" and the dimensions are indicated for interest sake only. The 27 meter long horizontal section is supported between 2 trees at a height of about 14 meters high. The weight of the



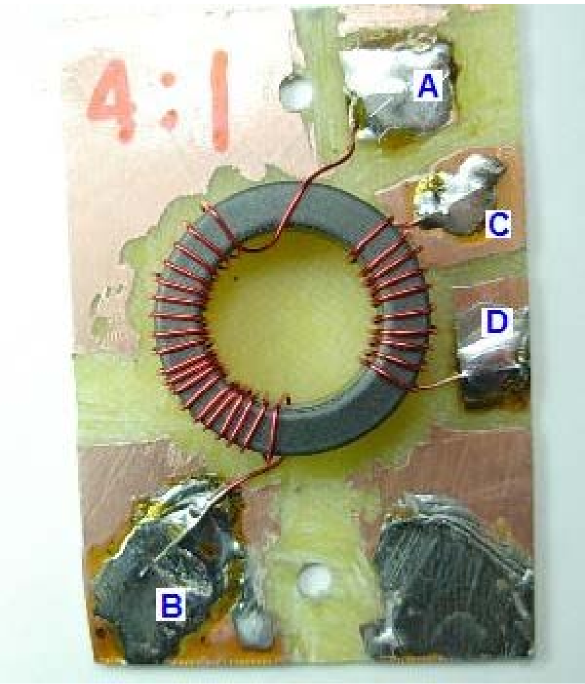
vertical element wire plus slack in the horizontal wire droop it to about 13 meters high in the center. The vertical section is soldered to the horizontal wire 6 meters from the nearest anchoring tree and runs straight down to the antenna feed point which is about 1 meter off the ground. The feed point is a piece of copper-clad PC board (with isolated sections created with a hobbyist motor tool) and is bolted to a long copper pipe which serves as the first station earth-grounding stake. A transformer (T1) configured as a UNUN **(unbalanced-to-unbalanced)** is used to interface the antenna with 50 ohm coax that runs through the house and into the radio shack. Some rudimentary experiments with the UNUN and the earth-grounding system were undertaken.

The methods I used to potentially lower unwanted RFI to my antenna system are as follows:

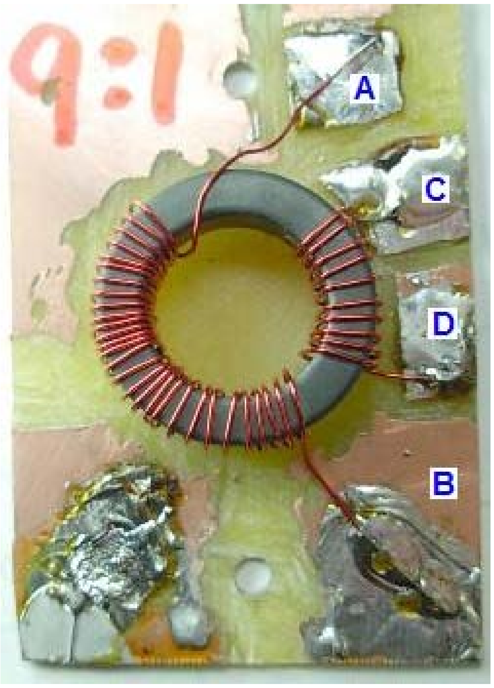
1. The receiver and power supply are independently connected to a single, central ground point (ground buss) in the radio shack.
2. 6-10 gauge wire is used for my ground system (not including the radials which are bare 12 gauge wire).
3. The ground wire connecting to my first earth stake to the station ground buss is just outside the shack window and is short as possible to provide a low impedance and low inductance path for MF and HF frequencies.
4. There is a second ground stake located 1 meter from the primary ground stake (I will add 2-4 more in time).
5. I have a large piece of steel buried underneath the soil tied in to my system as well as 3 bare copper radials. The radials are 3 - 7 meters in length.
6. New RG58/U coax was used as the feed line.
7. All wire splices in the grounding system are soldered and taped up. I used conductive grease (to prevent oxidation at the wire-stake interface) on any clamps connected to ground stakes. My ground stakes are ~ 2 meters long.
8. The earth grounding area soil is moist and peat-laden and is watered regularly.
9. I plan to maintain this ground system every 2 years.

4:1 UNUN

My antenna is almost an end-fed wire with both a vertical and horizontal section. I do not have the gear to measure the impedance versus frequency in the MF and HF bands. I do know that on some bands it may present an impedance of several thousand ohms and a transformer can smooth out the variation in impedance versus frequency so my receiver sees a relatively low impedance on most bands. The transformer also serves to help reduce RFI from my antenna system by eliminating unwanted common mode currents flowing on the outside of the coax braid. Grounding the antenna via the UNUN will also prevent static electricity from building up on the antenna. My first UNUN had a 4:1 impedance ratio. It is shown to the right. The antenna connects to point **A**. The ground stake connects to point **B.** Point **C** connects to the inner wire of the coax and point **D** is connected to the braid and also the grounding stake. I used 24 AWG wire and A FT114-43 ferrite core. You can clearly see there are 20 primary windings and 10 secondary windings loosely coupled. I chose the FT114-43 core because I had it on hand and the 24 gauge wire provides good mechanical support for the coil. I could have used an FT50-43 ferrite torroid as well with a smaller wire gauge. You can also use a bifilar transmission line type transformer. I was very happy with this UNUN and however it did not have as much signal strength as I expected on the 160 meter amateur band and below.



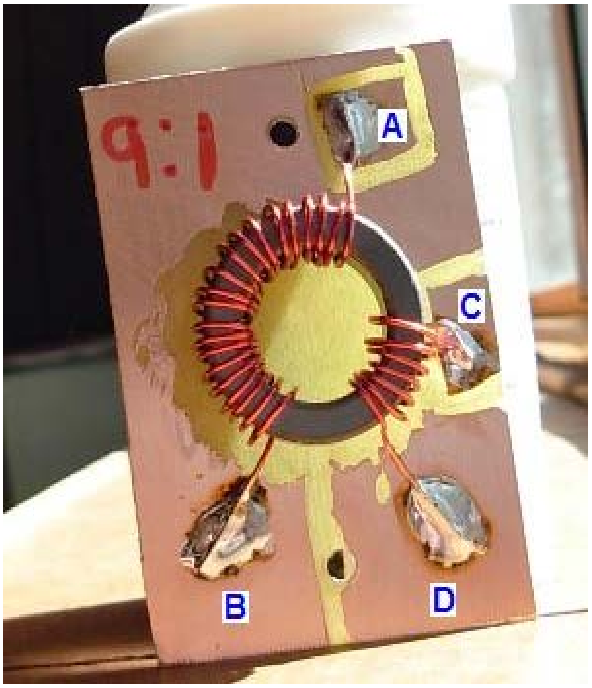
9:1 UNUN



Next I tried a 9:1 impedance ratio UNUN. This an extremely popular impedance transformation ratio for end-fed or random wire SWL antennas. I wound 30 primary and 10 secondary turns on a FT114-43 ferrite torroid. The connection points are identical to those described in the above 4:1 UNUN. Remember that the impedance transformation ratio is the square of the actual turns ratio on your transformer; thus my 3:1 turns ratio is a 9:1 impedance ratio. Electrical engineers commonly use a rule when winding broadband transformers such as these. The inductive reactance (XL) of the smaller winding must be at least 4 times the load impedance at the lowest frequency that the transformer "looks" into. So for 50 ohm coax, the XL should be at least 200 ohms at 500 KHz which is the lowest frequency I intend to receive. The formula for XL is **XL = 6.28 X F X L**. Frequency (F) is in Hertz and L is the inductance in Henries.

At 500 KHz my inductor has an XL of 189 ohms which is almost perfect. I should have used 11 turns which is an XL of 229 ohms and strictly observes the 4X rule. Therefore my UNUN ideally should have used 33:11 turns on the FT114-43 torroid. If you use a FT50-43 torroid, use the same 33:11 turns ratio; this will provide 198 ohms XL at 500 KHz. For practical purposes, my 30:10 UNUN should work fine as I rarely tune frequencies less than 1000 KHz. I found this UNUN to have strong signals all the way down to MF and decided to use a 9:1 impedance ratio for my antenna system. Many experimenters and a few commercial UNUNs recommend the 9:1 impedance ratio for multiband end-fed or random wire antennas. Eventually I will encase it in a water and UV proof enclosure.

Conclusion



My experiments while constructing a reasonable quality MF and HF receiving antenna confirmed that using a UNUN, coax and a good RF ground system can reduce common mode RFI in my receivers.
I also tried temporarily connecting my ground system to a copper water pipe located in my shack while listening to WWV at 5 MHz and immediately the noise level rose 2 S-units on my receiver.
This pipe was clearly not grounded in my house where there is a mixture of plastic and copper water pipes. Additionally, my antenna wire and feed point is away from the house in a quiet area according to listening tests using a Grundig S350.

It is relatively easy to construct a UNUN on your bench using a ferrite torroid. Many builders have emailed me to say they do not feel comfortable winding torroids. Torroids are easy to use and by winding a couple and experiencing some success, your confidence working with them will surely improve. If you live in a part of the world where you can not easily obtain a suitable ferrite core, just email me and I may send you an FT50-43. You can also choose a ferrite with a different core permeability. Some builders use number 75 material. I used the FT114-43 because I get all my torroids from W8DIZ and just use what he has in stock for my projects. If you really do not want to construct a UNUN, commercial products are available on the web on sites such as http://www.arraysolutions.com/Products/baluns.htm.

I wish you good luck with your own antenna experiments and please be safe!

Some SWL Antenna Related Links

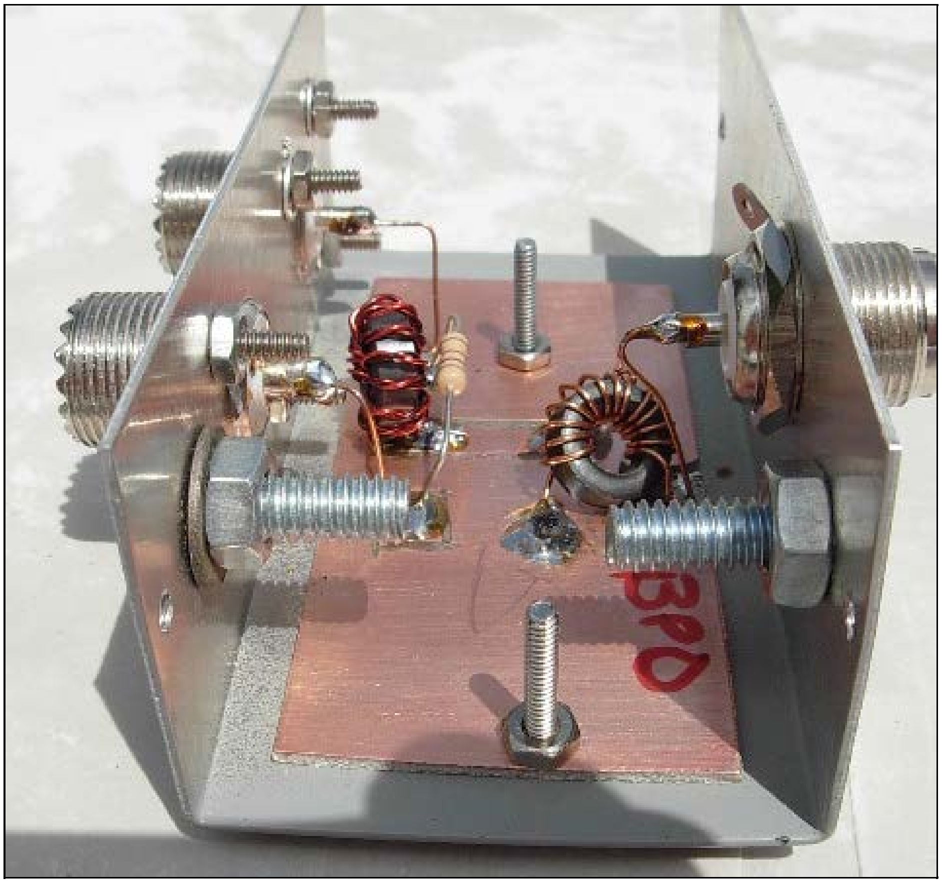
L.B. Cebik, W4RNL was a respected antenna expert. There is great information on his web pages

Build a Shortwave Antenna. A good overview of home brew multi-band antennas by N4UJW

Amateur and Short Wave Radio Electronics Experimenter's Web Site

MF and HF Receive Antenna Splitter

Introduction



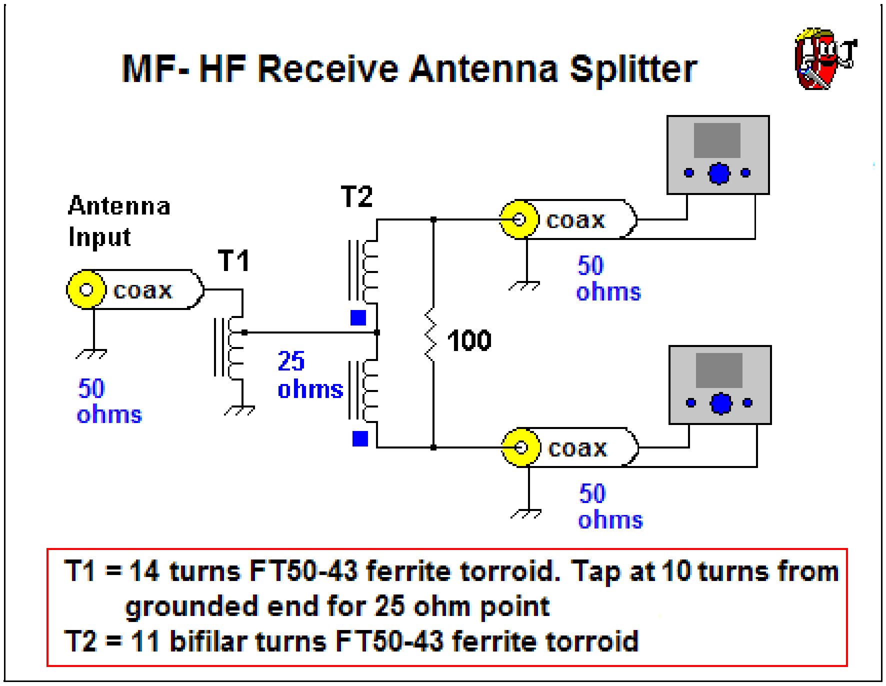
As a radio experimenter, I have numerous MF and HF receivers to listen to but usually only 1 main outdoor antenna. Typically, this means that only my main radio receiver is connected to the outdoor antenna and my other receivers must use small indoor antennas with or without RF preamplifiers. I wanted to to permanently connect my main radio shack receiver and the receiver in the room directly above the shack to my main MF and HF antenna at the same time. The solution was to build a simple antenna splitter which allows the 2 radio receivers to connect to the single coaxial antenna feed line while preserving the correct impedance at all connection points. This project is based upon the splitter presented in EMRFD labeled Figure 3.81.

Each receiver and the antenna feed line have a 50 ohm characteristic impedance. This in-phase splitter is passive and has a loss of just over 3 dB. It is designed to operate from 500 KHz up to 30 MHz.

Please do not transmit through this device.

Project Schematic

The schematic to the left illustrates the entire splitter network from the antenna input to the input of the 2 receivers. T1 and T2 are broadband ferrite transformers with enough inductive reactance to tune down to the bottom of the broadcast band. If you only require a splitter for HF, then wind T1 with 10 total turns and a tap at 7 turns from the grounded end and T2 with just 10 bifilar windings. I used FT50-43 cores to allow the use of thicker gauge wire which provides reasonable securement of the coils without external anchors, and because bigger inductors are easier to photograph. The FT37-43 ferrite core would also be a good choice, especially if miniaturization is a design goal.



T1 = 14 turns FT50-43 ferrite torroid. Tap at 10 turns from grounded end for 25 ohm point
T2 = 11 bifilar turns FT50-43 ferrite torroid

Circuit Building Details

The antenna splitter breadboard is shown to the right. I used 3 colors of 22 gauge enamel covered wire to make my inductors. T2 is the actual splitter network coil and is the lower transformer in the photograph. The 100 ohm resistor serves to isolate the 2 ports connected to the receivers and absorbs impedance mismatches which may present when one terminal is not properly terminated. Note that the characteristic impedance at the input of this 3 port network is only 25 ohms. You can choose to ignore this or use an additional network such as a broadband transformer or an L- match between your antenna coax and the splitter to match this 25 ohm impedance.

I chose to use T1 which is an autotransformer with a tap at approximately the 25 ohm point. The splitter network worked well during my tests. Having only 1 receiver versus 2 receivers connected made no difference to the signal strength due to the excellent output port isolation.

Note that Wes, W7ZOI uses this 3 port network network several times in EMRFD. One example is the Lichen transceiver while another is the 6M superhet receiver presented in Chapter 6. Consult EMRFD for further discussion of this and other multiple port networks.

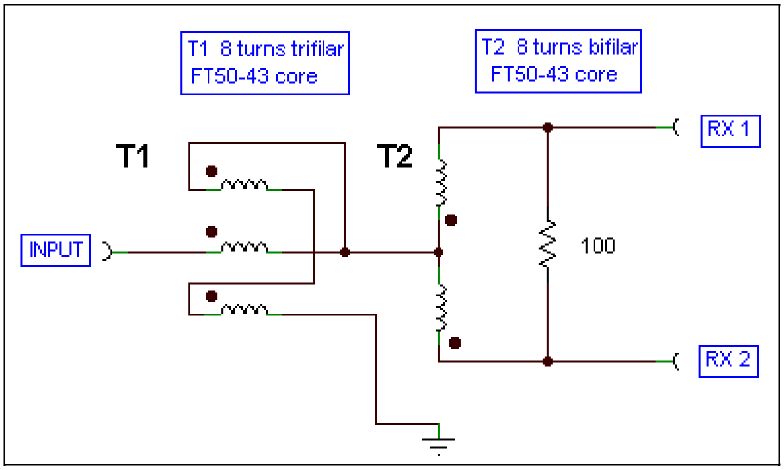
T2 is a bifilar transformer. The 2 wires were twisted together by securing one end of the 2 wires in a vise and the other end of these 2 wires in the chuck of a brace and bit (manual) hand drill. I twisted the hand drill until I had 8 twists per inch on the 2 wires . I used 2 color wire for ease of construction, however, it is almost as easy to tell the windings from one another by using an ohm meter or audible continuity tester.

Chassis

To the right is the completed project showing the SO-239 connectors which are wired to the antenna splitter output ports. A chassis from an old project was recycled for this new project. The large bolts seen in some of the of the photos were used to fill in holes which had been drilled for the old project. This was done to provide improved RF shielding. The bolts also increased the weight of the chassis and help keep it from tipping over. Although it does not look as attractive as if I had used a brand new project chassis, considerable cost savings were realized. These little Hammond project boxes are getting very expensive. Also the splitter is kept an the back of my main radio desk where it is out of sight anyway.

Update August 10, 2008 - Contribution by Dave, G4AON

This original network was designed for use in the MF to HF spectrum. Limiting this network is the input matching transformer T1 which negatively effects the T2 output port isolation; especially at 41 meters and higher. Testing by Dave, G4AON confirmed this. Dave designed, built and tested a trifilar wound, UNUN input matching transformer which provides a much flatter response for T2 port isolation from 0.1 to 52 MHz.

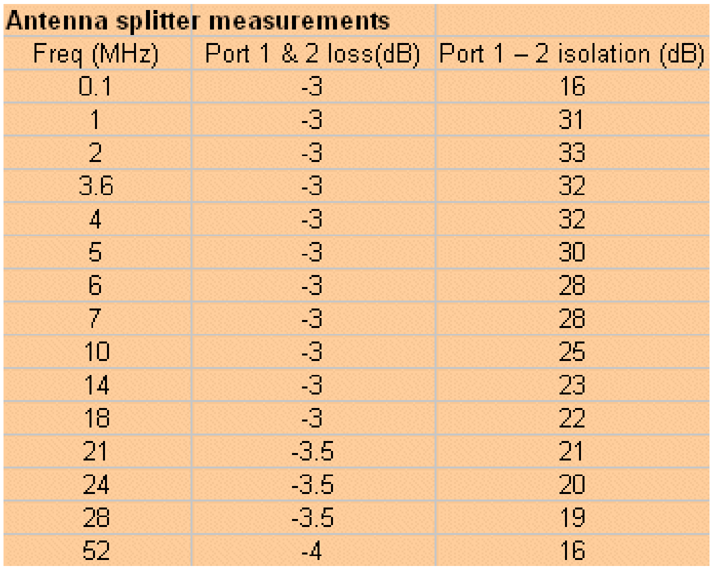


To the left you can see the G4AON input circuit for T1. In keeping with a design optimized for higher frequencies, less total turns are used on the transformers. His trifilar wound input transformer version is going to generate an impedance of (16/24)^2 x 50 = 22 Ohms at mid-band. My variation will generate (10/14)^2 * 50 = 25.5 ohms at mid-band. Using his version of T1 as opposed to my simple auto-transformer, Dave was able to provide better isolation of the output ports than the original design across a wider range of frequencies.

In the two popular, commercially sold RF splitters we have examined, the company did not even bother to match the input to the T2 transformer and some builders have written me to say they just omitted T1 and for their typical SWL listening this worked out fine for them. Increasing isolation across a wider frequency band and also matching the T2 input are issues that you the builder will have to consider.   Certainly the lossy and often non-predictable #43 ferrite material is a factor which might affect your transformer performance. While a trifilar transformer is a little more difficult build for a novice as compared to an auto-transformer, this improved design might work very well at your QTH. Testing like Dave did is certainly the way to go and I greatly appreciate his contribution.

To the right are Dave's excellent bench measurements. He used a Marconi 2018 signal generator, a Racal 9301 RF millivoltmeter and a Bird load on the other port. Kudos to Dave for performing this experiment and contributing to the receive antenna splitter knowledge base.

Dave's web site.



Antenna splitter measurements

Freq (MHz)	Port 1 & 2 loss(dB)	Port 1 – 2 isolation (dB)
0.1	-3	16
1	-3	31
2	-3	33
3.6	-3	32
4	-3	32
5	-3	30
6	-3	28
7	-3	28
10	-3	25
14	-3	23
18	-3	22
21	-3.5	21
24	-3.5	20
28	-3.5	19
52	-4	16

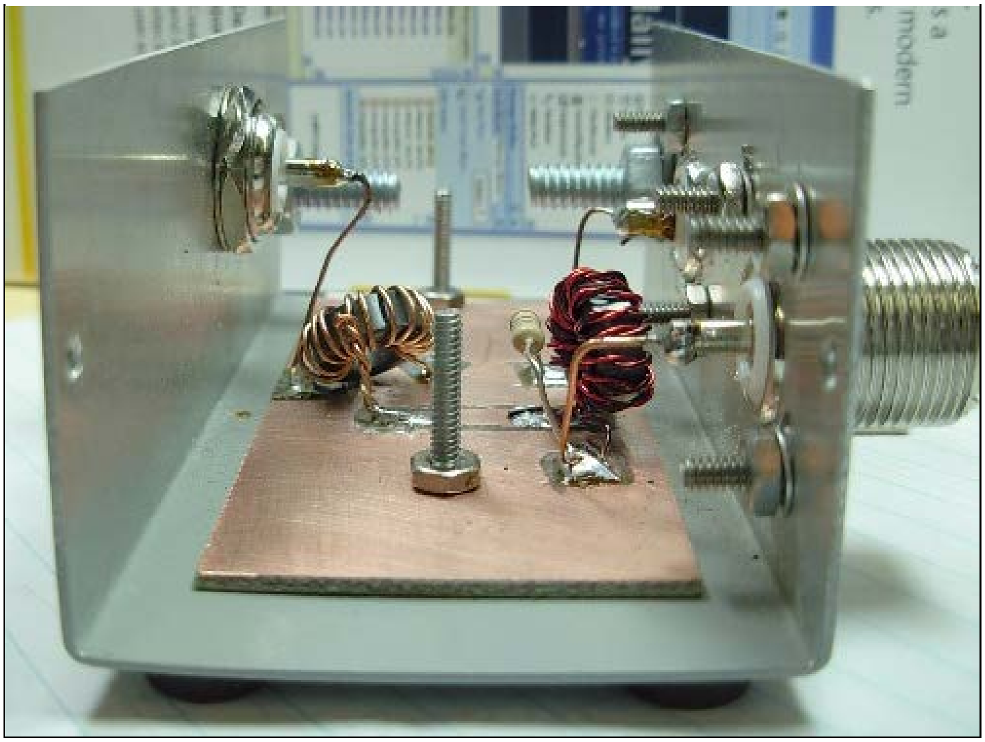
Conclusion

It is really awesome to be able to connect 2 receivers to the same outdoor antenna. The 3 port network and cabling to the additional receiver does not seem to increase receive noise levels from RFI in the house. Most likely this is due to the fact I am using shielded coax, a shielded project box and have a good RF ground system.

This is a simple project you can build in one evening.
I hope you receive some good DX! 73 es CUL, VE7BPO

Here is a link from F6AOJ

Additional Photos



SWL HOMEBUILDER

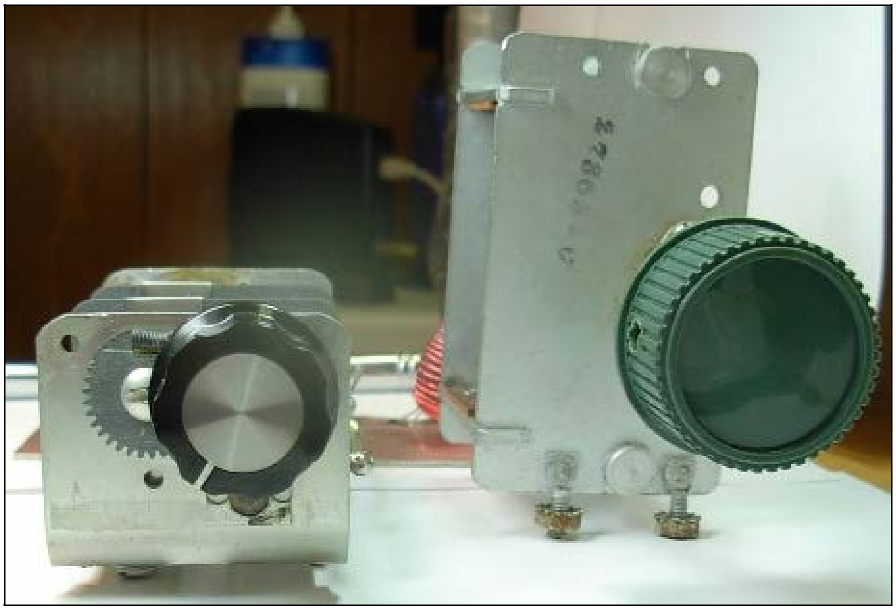
Amateur and Short Wave Radio Electronics Experimenter's Web Site

Medium Frequency TRF Receiver

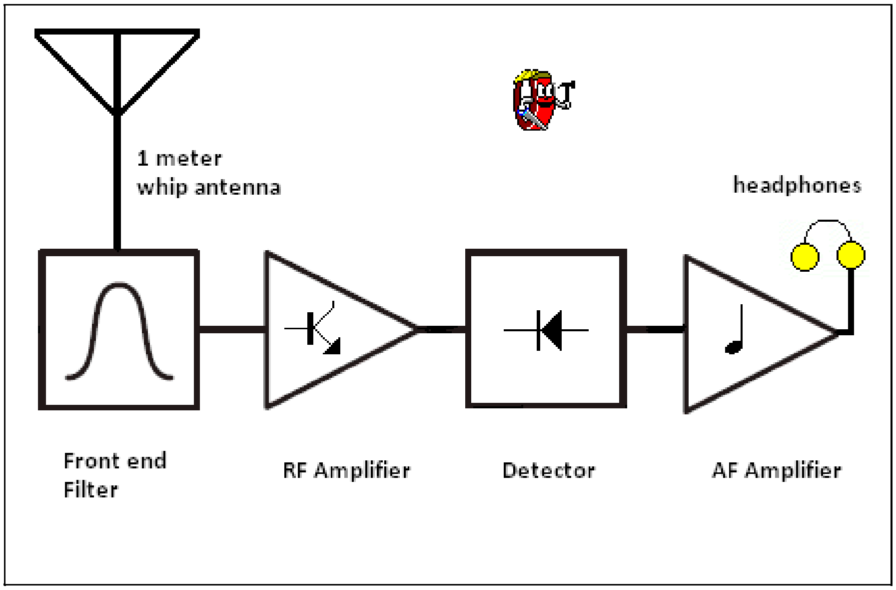
Introduction

This series of experiments was initiated in 2006, stalled, and was finally completed 16 months later with the inspiration provided by work regarding zero power receivers web-published by Wes, W7ZOI in late summer 2007.

Described is a complete receiver, built and presented backwards from the audio stage to the antenna. The design goals were to build a Tuned Radio Frequency broadcast band receiver with one RF amplifier, a high performance detector and a simple, headphone-level audio stage.



Receiver Block Diagram

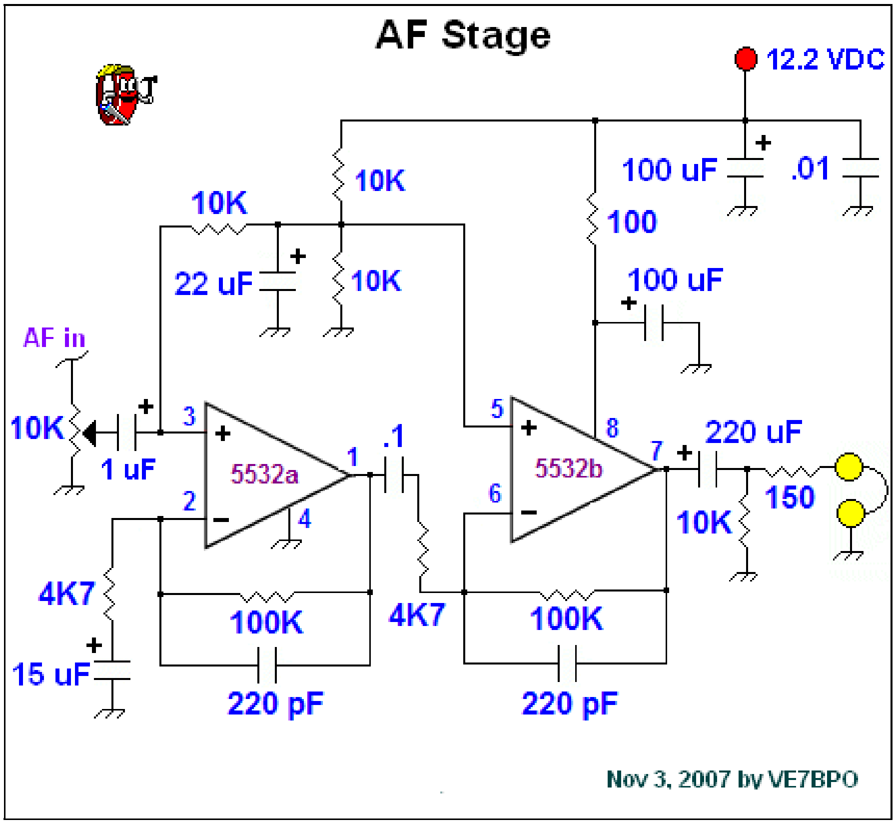


The receiver block diagram is shown to the left. The antenna is a ~ 1 meter long whip purchased from Radio Shack in the USA. A single cascode bipolar junction transistor amplifier boosts the RF voltage and drives an envelope detector which is terminated by a JFET source follower. The source follower connects to a 10K volume potentiometer which controls the AF signal voltage into a headphone-level audio amplifier. Like most of the projects on this site, the intention is to present some circuits and ideas for experimentation. This receiver is designed for local broadcast band AM radio reception, however, the various circuits could be used in or titleered for DX receivers as well.

[A Supplemental Page can be found here]

Audio Stage

The AF amplifier is a superb design by Rick, KK7B and is featured in many projects in EMRFD. This audio amplifier uses one 5532 op amp and has low noise and high gain. The 220 pF feedback capacitors can be increased to boost the low frequency response. I have built 6 or 7 versions of this stage and have used feedback capacitors up to 560 pF for this purpose. In the audio path, polyester film capacitors were used to try to improve the audio quality. Additionally, the value of the 15 uF capacitor connected to pin 2 is flexible. The quiescent current draw of this stage at 12.2 VDC is 12.3 mA. Some builders may have to increase the 100 uF filter capacitor on the main 12.2 volt line to overcome motorboat oscillation. None occurred in my breadboard version. I suggest using this audio stage instead of the LM386 or discrete component final audio amplifiers in all projects which call for a headphone-level audio power amplifier on this web site.

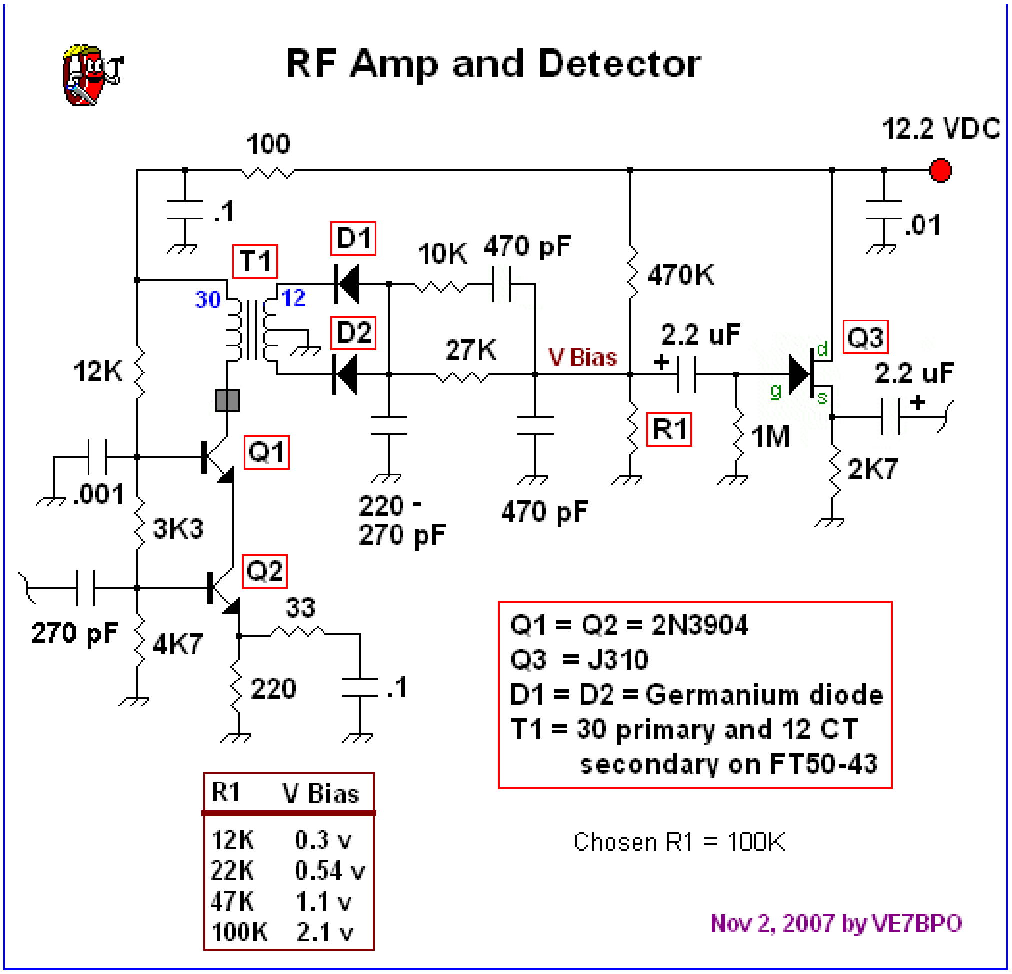


Nov 3, 2007 by VE7BPO

Close up of the KK7B audio amplifier breadboard from the 10K potentiometer to headphone jack

---

Cascode BJT RF Amplifier and High Performance Detector



Q1 = Q2 = 2N3904
Q3 = J310
D1 = D2 = Germanium diode
T1 = 30 primary and 12 CT
secondary on FT50-43

R1	V Bias
12K	0.3 v
22K	0.54 v
47K	1.1 v
100K	2.1 v

Chosen R1 = 100K

Nov 2, 2007 by VE7BPO

Above is the combined RF amplifier, detector and JFET source follower schematic.

Cascode BJT RF Amplifier

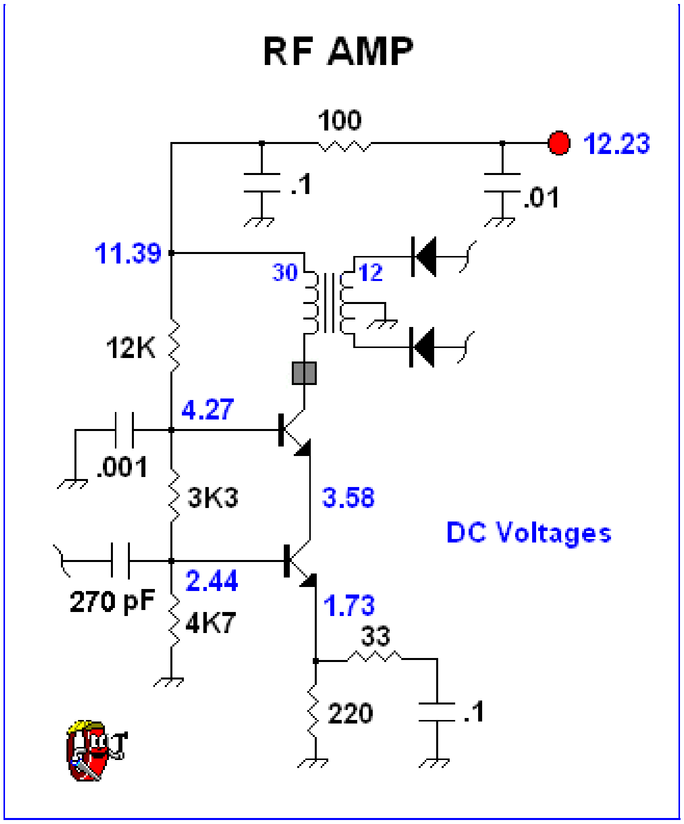
To the left is a simplified RF amp diagram taken from the schematic above indicating the measured DC voltages for reference purposes.

T1, the output transformer was wound on an FT-50-43 ferrite toroid. An FT37-43 would also be suitable. Number 28 gauge enamel coated magnet wire was used for the 30 turn primary and 26 gauge wire was wound over top to make the secondary 12 turn, center-tapped winding. The 26 gauge wire was used for the secondary winding because it provided good anchoring of the transformer by the center tapped ground connection.

You may consider substituting a 22 to 100 ohm resistor for the ferrite bead on Q1. It suppresses VHF oscillations.

Detector

To the right is a photograph of the detector from the Q1 transformer through to the JFET source follower. Schottky/hot carrier diodes or germanium diodes such as the 1N34A with a low forward voltage drop are strongly recommended. I have found there to be significant variation in



sensitivity between different types of these diodes. The 2 germanium diodes I used were matched as described on this web page. A number of detectors were built and tested for this receiver, however, the design shown had the best audio quality when compared to the others. The virtues of this detector include low noise, high bandwidth, high sensitivity and low distortion. although a little complex, this is a detector worthy of consideration in your AM receiver projects. The center-tapped Q2 transformer secondary and the 2 diodes provide full wave detection. This serves to reduce distortion somewhat and cancel even-order harmonics in the carrier signal. You may eliminate one of the diodes and convert the Q2 transformer secondary to a conventional, single link.

A 470K ohm resistor and R1 form a voltage divider that sets the detector bias voltage (V Bias). Some measured R1 values and corresponding bias voltages are shown in the schematic. I chose an R1 value of 100K for my final version. You may have to increase or decrease the R1 value to suit your local detector sensitivity requirements. You could also substitute a bias potentiometer for front panel adjustment of the receiver sensitivity. In this detector, changing the R1 value also changed the detector frequency response. I built a separate voltage divider with roughly the same V bias consisting of a 68K and a 15K resistor and swapped it for the 470K and 100K pair. Interestingly, the 470 K and 100K pair had better low frequency response and slightly higher sensitivity than the 68K and 15K voltage divider.
Diode detectors are best driven with a high impedance source and followed by a high impedance load. Q3, a simple JFET source follower provides a high impedance load. You might want to substitute a "popcorn" MPF102 for the high Idss J310 indicated in the schematic as a J310 is not really required here. If you substitute a MPF102, please increase the source resistor from 2K7 to 4K7 ohms.

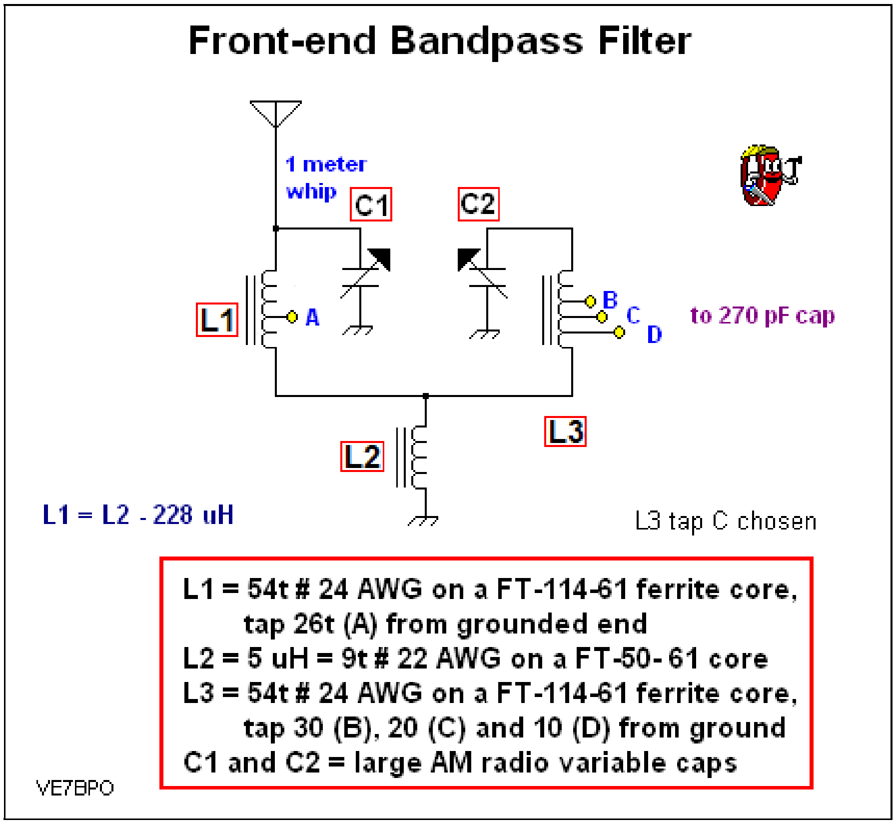
Front-end Band Pass Filter and Antenna

In late summer 2007, Wes, W7ZOI conducted experiments with zero power receivers (crystal sets and such). He wound some inductors using ferrites with an unloaded Q of over 270 at MF! Please check out Wes' web site. His work with high Q ferrite inductors illustrates the importance of quantitative measurement and also provided the following revelation; we really do not have to resort to large, air core, Litz wire coils to build high-performance inductors at MF! The early prototype front end for this project was built using FT50-61 ferrite cores, however after Wes emailed me his work on zero power

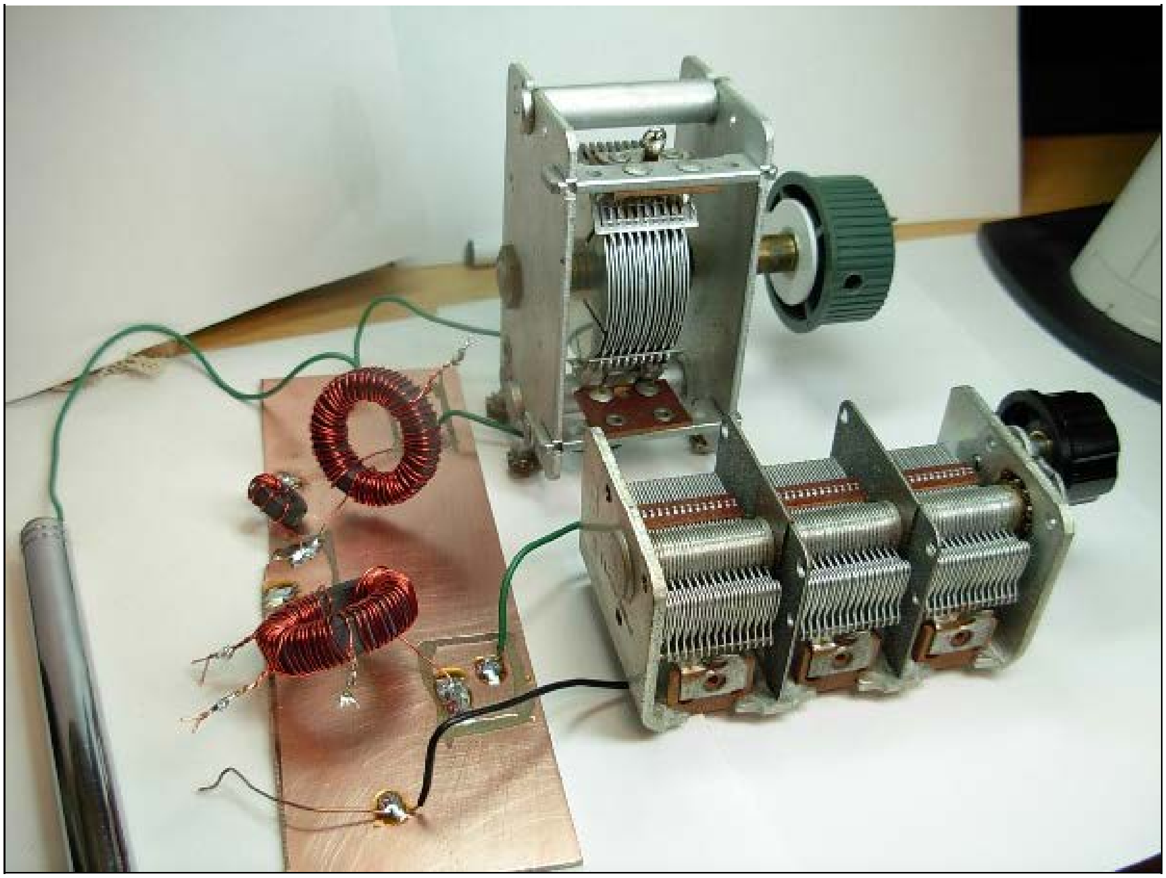
receivers, I had to get some FT-114-61 ferrites for the front end of this receiver. The next day, I emailed Mark Laurain from Amidon Associates Inc and ordered some FT-114-61 ferrite toroids. The arrival of these ferrites prompted me to finish this project and put it up on the web.

The schematic on the right is the final band pass filter used for the front end. I initially tried using just L3 for the front end, but I was unable to just tune a single station. In my city, there are 2 powerful AM radio stations at 630 and 1150 KHz. With a single inductor, I could peak one of the stations, but the other could be heard in the background. Thus, the double-tuned band pass filter presented was designed and built. Now only one station can be detected with this circuit and tuning is sharp. Most builders would use a dual-ganged variable tuning capacitor, however, I elected to use 2 separate variable capacitors. Considerable flexibility with this circuit is possible. You will have to experiment to best determine your local sensitivity versus selectivity needs and to suit the variable capacitors you have available. Large AM receiver capacitors are getting hard to find. I obtained the 2 variable capacitors shown in the photographs below from 2 old receivers found in a second hand store. One of the receivers was a Marconi tube radio that was in poor condition. I paid $5.00 for both radios and harvested the 2 beautiful variable capacitors as well as some other parts such as knobs, switches and terminal strips. Never pass up on an old, derelict radio as a potential variable capacitor source!

Front-end Bandpass Filter



L1 = 54t # 24 AWG on a FT-114-61 ferrite core,
tap 26t (A) from grounded end
L2 = 5 uH = 9t # 22 AWG on a FT-50-61 core
L3 = 54t # 24 AWG on a FT-114-61 ferrite core,
tap 30 (B), 20 (C) and 10 (D) from ground
C1 and C2 = large AM radio variable caps

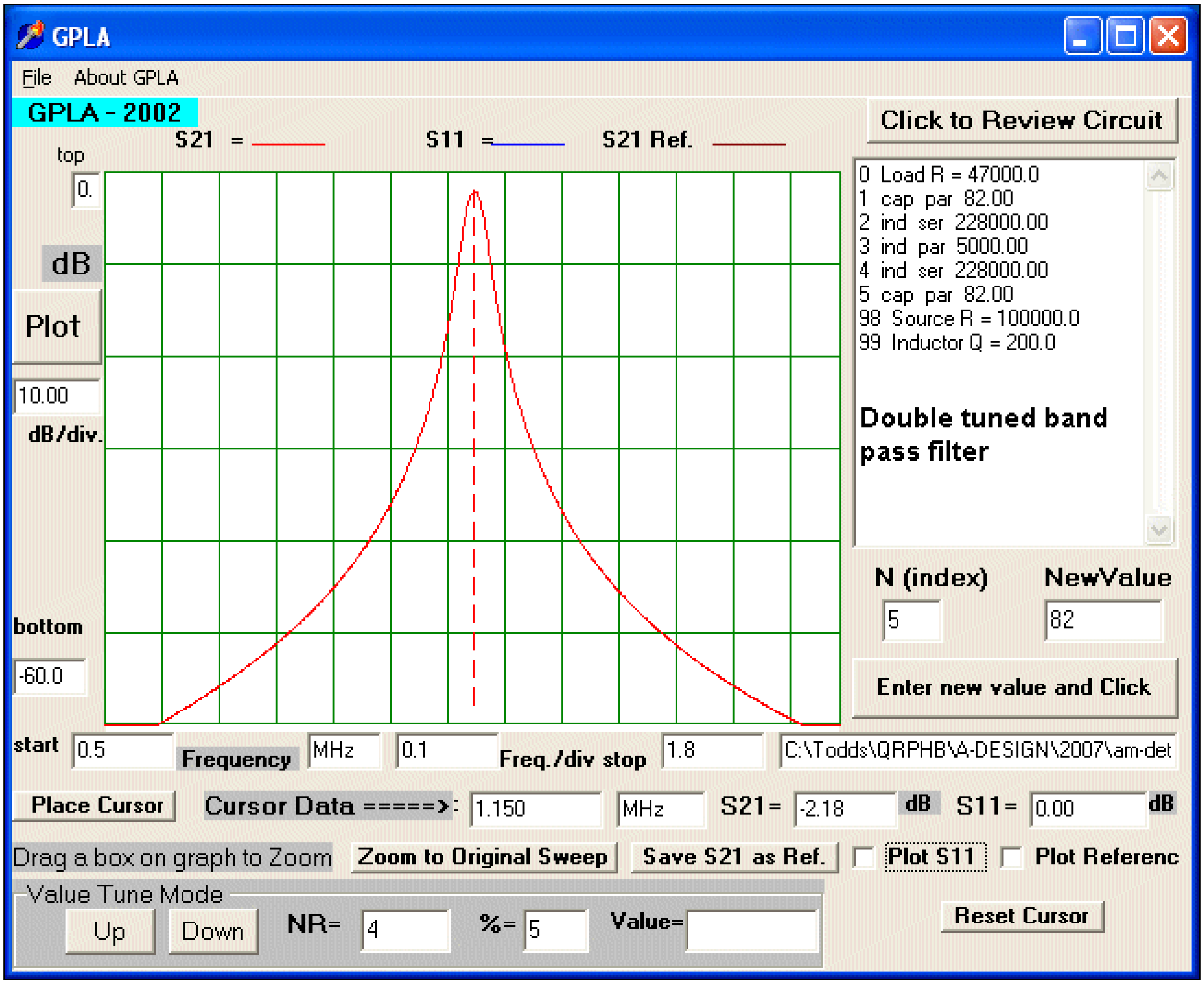


Shown above are 2 photographs of the band pass filter breadboards. The 2 variable capacitors had a variation of ~ 24 to 500 pF. There is considerable interplay between the 2 capacitors. For my QTH, it was better to peak C2 first and afterward to peak C1. Consider that L1 and L2 have a hot end and a grounded end. The antenna is connected to the the hot end of L1. Predictably, when substituting the L1 center tap as the antenna connection, the selectivity of the L1-C1 tank is increased and the sensitivity or received signal strength is reduced. This also occurs when testing the various tap points on L3 to feed the RF amplifier-detector stages. In the final circuit, I settled on Point C, 20 turns from ground. Using Point D, reduces sensitivity and increased selectivity. The opposite is true when using Point B. You the builder, have to determine which L1 and L3 connection points to use based on your own experiments and local factors. You may also change the receiver sensitivity by making changes such as increasing or decreasing the 270 pF coupling capacitor value, the emitter degeneration on Q2, or the detector bias.

Band pass Filter Analysis

It is impractical to sweep a BCB band filter using variable capacitors, so some analysis using GPLA, a program that ships with EMRFD was used to plot and better understand the double-tuned band pass filter response. A worst case inductor unloaded Q of 200 was used, but I imagine that the actual Q of L1 and L3 is much higher. For the source impedance, 100K ohm was used conjecturing that a short whip antenna at 1150 KHz would have a very high input impedance and not load down the L1 inductor. In reality, it is likely the antenna input impedance might be closer to 1 Megohm, however, I am using the worst case scenario. If the filter performs better than simulated - all is great! Higher source and load impedances and higher unloaded Q inductors would decrease the bandwidth of this filter which is desirable.
Note that I am concerned that L2 at 5 uH may may overcouple the 2 tank circuits. I did not see a double humped response on GPLA analysis, however, experimentation with L2 may be in order for the more astute homebuilder. You might consider lowering the L2 value to 3 or 4 uH and performing some testing. The load impedance for L3 was rather arbitrarily chosen. Considering that various taps on L3 may be used, the XC of the 270 pF coupling capacitor and the input impedance of the RF amplifier, I just chose 47K as the L3 load impedance. Below are 2 screen captures of GPLA plots. The top graph is the double tuned band pass filter and below it is the single tuned band pass filter consisting just of L3 and C3.
These graphs lead to 2 main conclusions:

1. The final band pass filter design appears to be reasonably sound.
2. We can understand why I could not tune in a single radio station with just L3 and C3 as the band pass filter; the filter skirts are not very steep and the second unwanted station was also amplified and detected.



top: 0.

10.00 dB/div.

bottom: -60.0

```
0  Load R = 47000.0
1  cap  par  82.00
2  ind  ser  228000.00
3  ind  par  5000.00
4  ind  ser  228000.00
5  cap  par  82.00
98  Source R = 100000.0
99  Inductor Q = 200.0
```

Double tuned band pass filter

N (index): 5

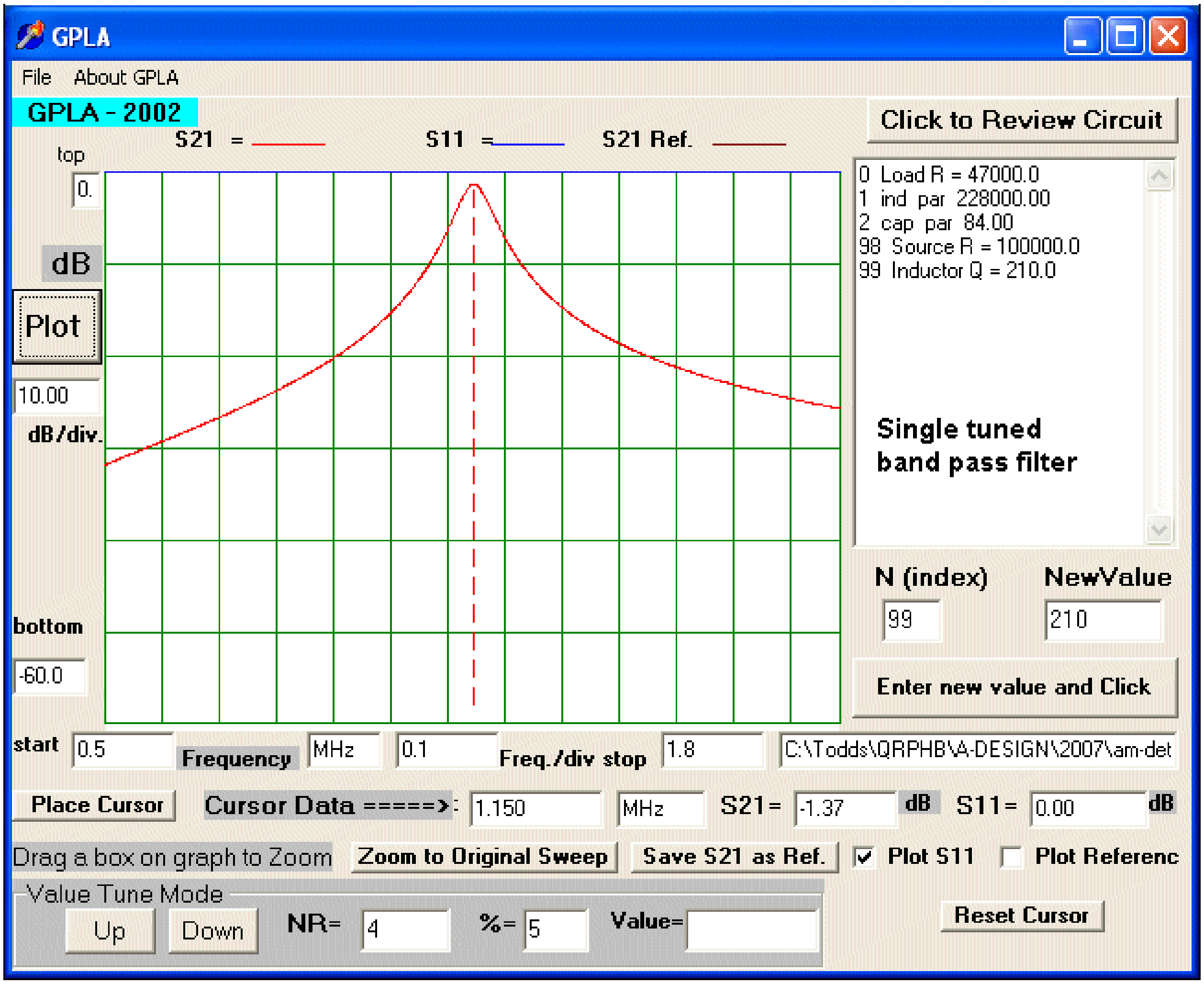
NewValue: 82

start: 0.5 Frequency: MHz 0.1 Freq./div stop: 1.8 C:\Todds\QRPHB\A-DESIGN\2007\am-det

Cursor Data =====>: 1.150 MHz S21= -2.18 dB S11= 0.00 dB

☐ Plot S11 ☐ Plot Referenc

NR= 4 %= 5 Value=



```
0  Load R = 47000.0
1  ind  par  228000.00
2  cap  par  84.00
98  Source R = 100000.0
99  Inductor Q = 210.0

Single tuned
band pass filter
```

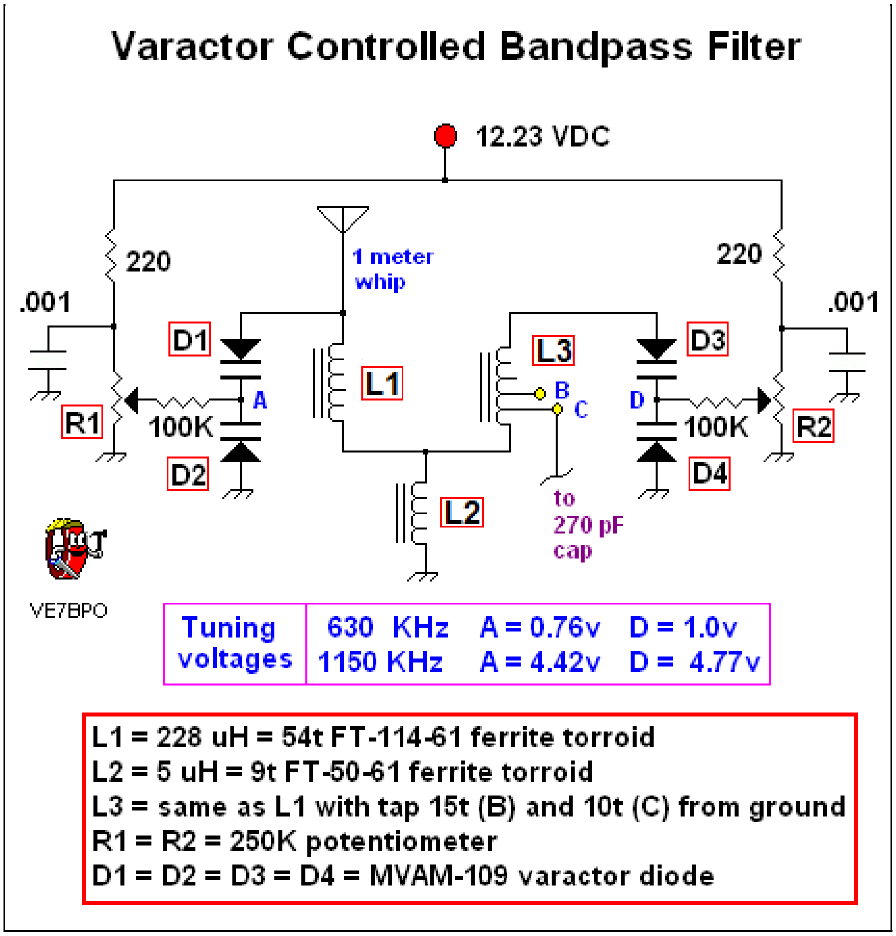
top: 0.
10.00 dB/div.
bottom: -60.0
start: 0.5 MHz
Freq./div: 0.1
stop: 1.8
C:\Todds\QRPHB\A-DESIGN\2007\am-det
Cursor Data: 1.150 MHz
S21= -1.37 dB
S11= 0.00 dB
N (index): 99
NewValue: 210
NR= 4
%= 5
Value=

Varactor Tuned Front-end Filter

On November 11, 2007, I decided to investigate whether or not variable capacitance or varactor diodes could effectively replace the air variable capacitors in the band pass filter. In my parts cabinet were several MVAM -109 which is an obsolete but still readily available part. Another varactor, especially designed for tuning AM receivers is the 1SV149. This varactor is manufactured from Toshiba and is also appropriate.

While not comparable to the Q of 300 or greater of a good quality air variable capacitor, varactors are smaller, cheaper and can be easily ganged together so that only 1 potentiometer is required to tune the front-end filter. To the right is a photo of the varactor tuned front-end filter breadboard.

To the left is the schematic of the varactor tuned front-end band pass filter. The air variable capacitors were unsoldered from the original filter breadboard and a small board drilled and fitted with two 250K potentiometers was soldered to it.

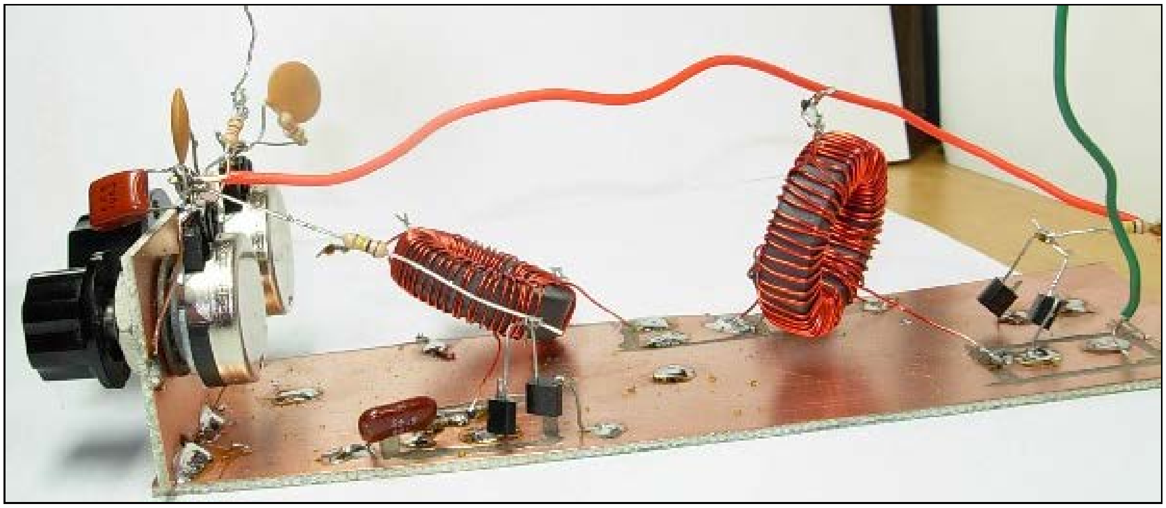


Tuning voltages			
	630 KHz	A = 0.76v	D = 1.0v
	1150 KHz	A = 4.42v	D = 4.77v

L1 = 228 uH = 54t FT-114-61 ferrite torroid
L2 = 5 uH = 9t FT-50-61 ferrite torroid
L3 = same as L1 with tap 15t (B) and 10t (C) from ground
R1 = R2 = 250K potentiometer
D1 = D2 = D3 = D4 = MVAM-109 varactor diode

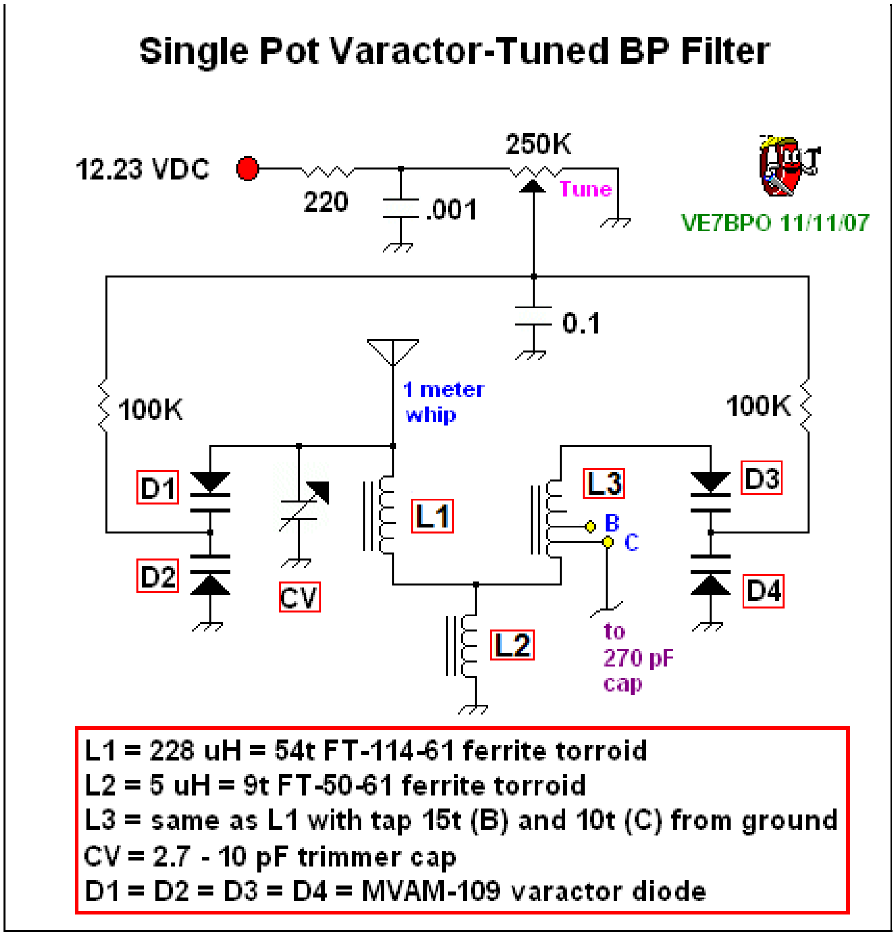
L3 was also modified to have taps at 10 and 15 turns from ground. I conjectured that since the varactor diodes have less Q than their air variable cousins, it would be wise to tap down on L3 to try and increase the selectivity of the L3 tank circuit. In the end, I used the tap at 10 turns from ground for my receiver as signal strength was still acceptably strong. You may choose to use the tap at 15 or some other point to suit your local selectivity/sensitivity requirements.

I was able to tune in single stations as I previously did with the air variable capacitors. Tuning is "touchy". Ten-turn pots would be a better choice, however, are not very frugal for such a project. You get used to tuning with conventional potentiometers after a few minutes or so. I measured the reverse voltages required to tune the 2 main local AM radio stations and they are tabled in the schematic. The L1 tank requires slightly more capacitance to resonate than the L3 tank. Thus it takes a little less applied reverse voltage to the varactor pair resonating the L1 tank compared to the varactor pair resonating the L3 tank .A side view photograph of the varactor breadboard is shown directly below. The component leads have been kept long so that I can recycle parts from experimental project to project as possible. This helps contain costs. Shorter lead lengths and proper lay out should be pursued in any final projects you build.



Single Varactor Tuned Front-end Filter

Tuning with a single potentiometer ganged to both varactors is easy to do after learning from the experiment above. All that is required is a method to compensate for the differences in capacitance between the the 2 LC tanks. I placed a high-Q (Q=300) variable trimmer capacitor in parallel with L1. By listening to the receive signal strength and tuning in one radio station using the potentiometer, I was able to peak CV for the strongest signal. I did this for both 650 and 1150 KHz and actually unsoldered CV and measured its value with a capacitance meter. The CV value was ~ 6 pF for both frequencies. I decided to replace CV with a fixed 5 pF silver mica capacitor and left it there in my final filter version. Your results will probably be different. I suggest just leaving CV


Single Pot Varactor-Tuned BP Filter

L1 = 228 uH = 54t FT-114-61 ferrite torroid
L2 = 5 uH = 9t FT-50-61 ferrite torroid
L3 = same as L1 with tap 15t (B) and 10t (C) from ground
CV = 2.7 - 10 pF trimmer cap
D1 = D2 = D3 = D4 = MVAM-109 varactor diode

and using this trimmer cap to peak the signal once you have tuned a desired radio station with the main tuning potentiometer. An alternative to using CV is to vary L1. You could try compressing the number of L1 windings to allow tracking of the 2 LC tank circuits.

For the varactors, I used back-to-back VVC diodes as opposed to just a single varactor to resonate each tank. This was done in an effort preserve the highest varactor Q possible. The RF voltage of the AM RF signal may be high enough to forward bias a single varactor during a portion of the AC signal and degrade Q. This does not happen when back-to-back diodes are used. Almost all high-grade FM tuner schematics I have seen use back-to-back varactor diodes in their various ganged, tunable band pass filters. The major drawback of back-to-back diodes is your tuning range is reduced because you now have 2 capacitors in series. Experimentation may be required to achieve the BCB band-spread that you desire. You can add another pair of varactors in parallel or add some parallel fixed capacitance or even change the L1 and L3 inductance values for example.

This receiver tunes nicely and sounds fabulous. Last evening I was able to tune in 5 different AM stations, however, other than the local 2 radio stations, the others were quite faint. This is not bad considering this receiver has only 1 RF amp and a 1 meter long antenna. This band pass filter could be adapted as a pre-selector for AM radio reception. To match 50 ohms, lower L1 and L3 tap points could be chosen or a few links of wire may be wound around the inductors.

In the photograph below, you can see the 5 pF capacitor soldered in parallel with the MVAM-109 pair associated with L1. The antenna also connects to the ungrounded end of the 5 pF capacitor. Below in the last photograph; since only one potentiometer is used for tuning, a large knob was screwed on to the pot control shaft to make tuning a little easier. The solder-laden 220 ohm resistor is the connection point for the regulated 12.2 VDC. The 220 ohm resistor on the left has been cut from the 12.2 VDC connection point so 0 voltage goes to the left potentiometer.

Final Thoughts

I emailed Wes, W7ZOI and asked him why it is better to inductively couple a tuned circuit which use air variable tuning capacitors. Wes wrote his answer in the form of a complete web article entitled Coupling Methods in the Double Tuned Circuit. Big thanks Wes! From his summary, when the inductors used to resonate each tuned circuit are constant, and inductive coupling is used, the coupling of the resonators will remain constant as the variable capacitors are tuned across the band. Please download and study his web article for it not only discusses coupling in the double tuned circuit, but provides some insight into using his LadBuild and GPLA software from EMRFD.

Amateur and Short Wave Radio Electronics Experimenter's Web Site

Junk Box NDB Low Pass Filter

Introduction

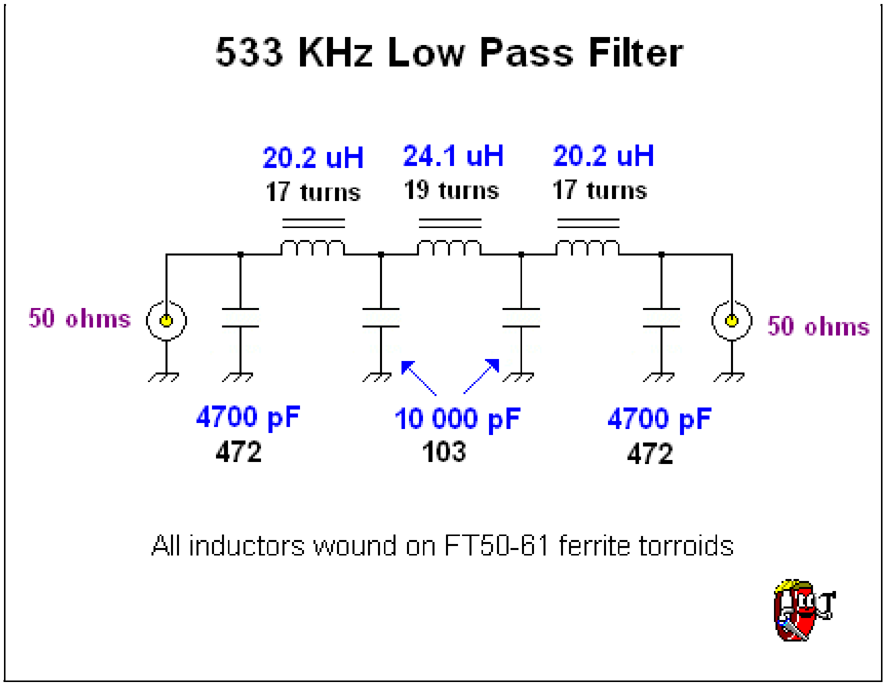
With winter approaching, many HAM and SWL hobbyists find intrigue in tuning in NDB or **N**on-**D**irectional **B**eacons. Although the tunable NDB band depends on your location, in Canada it may be found in a band ranging from about 190 to 535 KHz. Canadian beacons either have just a carrier (no offset)or are tuned using the USB with about a 400 Hertz offset, however, different offset frequencies and certainly LSB are used when receiving DX from other countries.

Less than 10 Km away from my QTH is a 10 KW AM radio station at 1150 KHz. On my test receiver, the S-meter reads off the scale (> 60 dB over S-9) when tuned to this frequency. This local radio station causes spurious, second-order intermodulation products (direct mixing) that all but wipes out some weaker NDB stations that I am trying to tune in. Certainly, having a 500 Hertz crystal IF filter is useful, but attenuating this local QRM is also desirable and is the topic of this web page. Many general coverage receivers offer limited or in some cases no filtering of the NDB band, however an outboard filter is an easy project to build in one afternoon.

Update Oct 11, 2010: Here is a link to a version of this project built by Robert, K5TD

Project Schematic

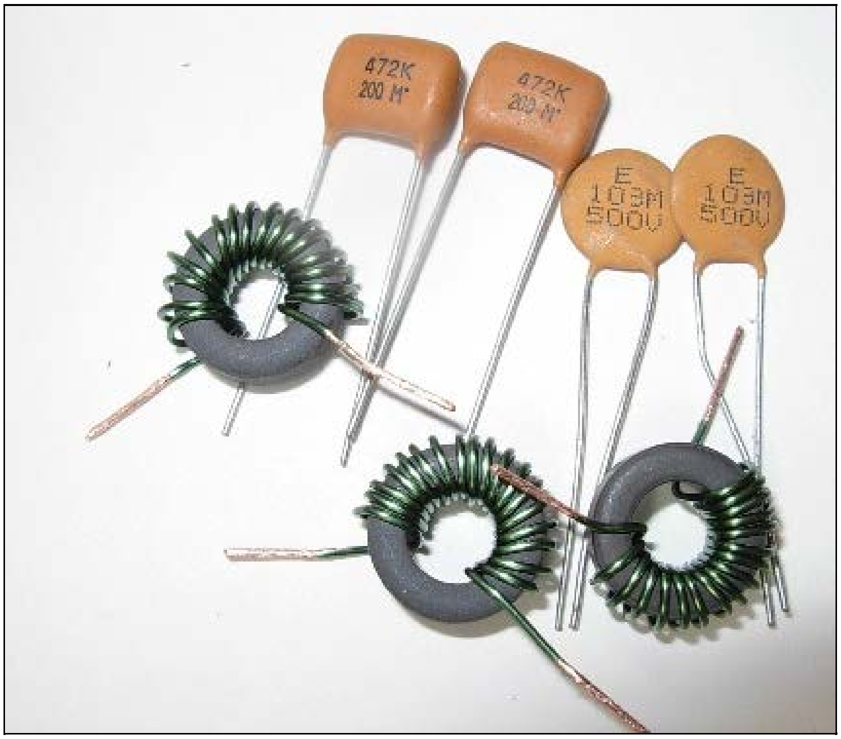
To the left is the project schematic. It seems odd to build a low pass filter to reduce BCB interference (as usually a high pass filter is required for this purpose at HF) however for NDB, an aggressive low pass filter is required. For simplicity sake, a 7 element Chebychev low pass filter was chosen. Since it is easy to wind reasonably high-Q inductors for 10 uH and greater inductance using number 61 material on a ferrite torroid, the FT50-61 core was chosen for all of the inductors. Number 22 gauge wire was used for the coils to keep the unloaded Q as high as possible. The FT37-61 ferrite is also suitable, but will have less Q and require smaller gauge wire. Use 19 turns instead of 17 for the 20.2 uH and 21 turns instead of 19 for the 24.1 uH coil. Do not use number 43 material ferrite cores.



Components

I do not stock RF capacitors greater than 2200 pF, so junk box ceramic capacitors were used to build this filter. In fact, this design specifically uses more common, standard value capacitors to reduce cost and to not have to order in parts. Certainly, the astute builder could use higher quality capacitors or even large powdered iron torroids instead of the ferrite cores for inductors if higher performance is desired. Try to use high Q capacitors if you can find or are purchasing them. Poly or silver mica caps would be great choices. You can substitute a 5000 pF capacitor for the 4700 pF called for in the schematic.

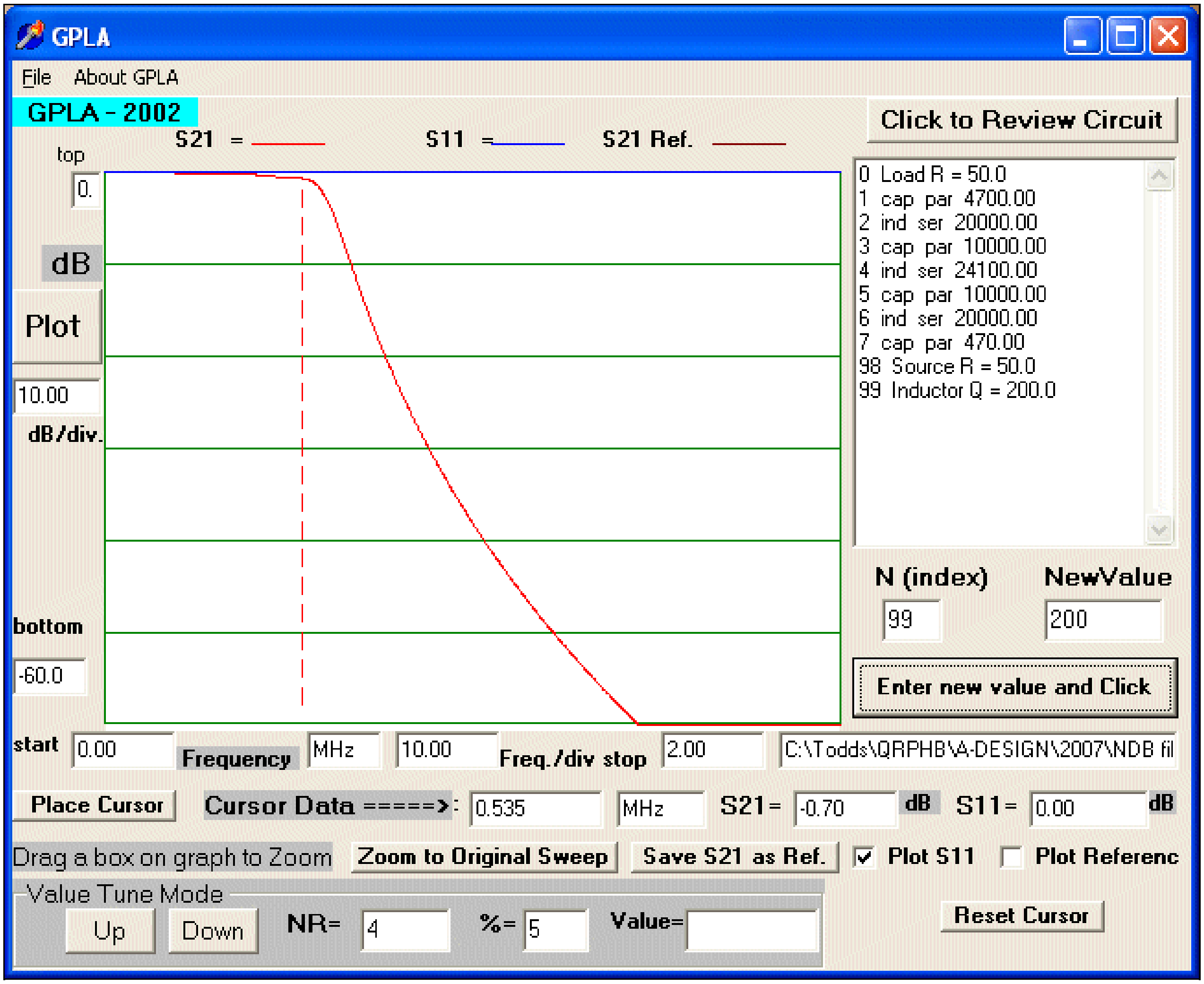
To the right is a photograph of the components I used in the project breadboard.



Breadboard

To the left is the completed project. Ugly construction as always, was used. The inductors were spaced at least 2.5 cm (1 inch) apart at right angles to try to minimize unwanted coupling.

GPLA Simulation



0 Load R = 50.0
1 cap par 4700.00
2 ind ser 20000.00
3 cap par 10000.00
4 ind ser 24100.00
5 cap par 10000.00
6 ind ser 20000.00
7 cap par 470.00
98 Source R = 50.0
99 Inductor Q = 200.0

top: 0.
10.00 dB/div.
bottom: -60.0
start: 0.00 MHz
Freq./div: 10.00
stop: 2.00
C:\Todds\QRPHB\A-DESIGN\2007\NDB fil
N (index): 99
NewValue: 200
Cursor Data =====>: 0.535 MHz
S21= -0.70 dB
S11= 0.00 dB
☑ Plot S11
☐ Plot Referenc
NR= 4
%= 5

Above is the plot of the filter during simulation with GPLA. The simulation calculated an attenuation of ~46dB at 1150 KHz. At 1000 KHz the signal was 40 dB down, at 800 KHz it was ~24 dB down and at 630 KHz, the attenuation was only ~5.7 dB! Clearly this filter is not suitable if the offending BCB interference is from a station significantly less than 0.8 MHz. For my situation, this filter is acceptable. A 5 element Chebychev filter was also designed and plotted but was discarded as there was only 32 dB attenuation at 1150 KHz.

Since I wanted to tune as high as 535 KHz, the 533 KHz cutoff frequency was chosen. Additional work to help those with strong BCB interference at the lower BCB will be attempted in the future and presented on another web page.

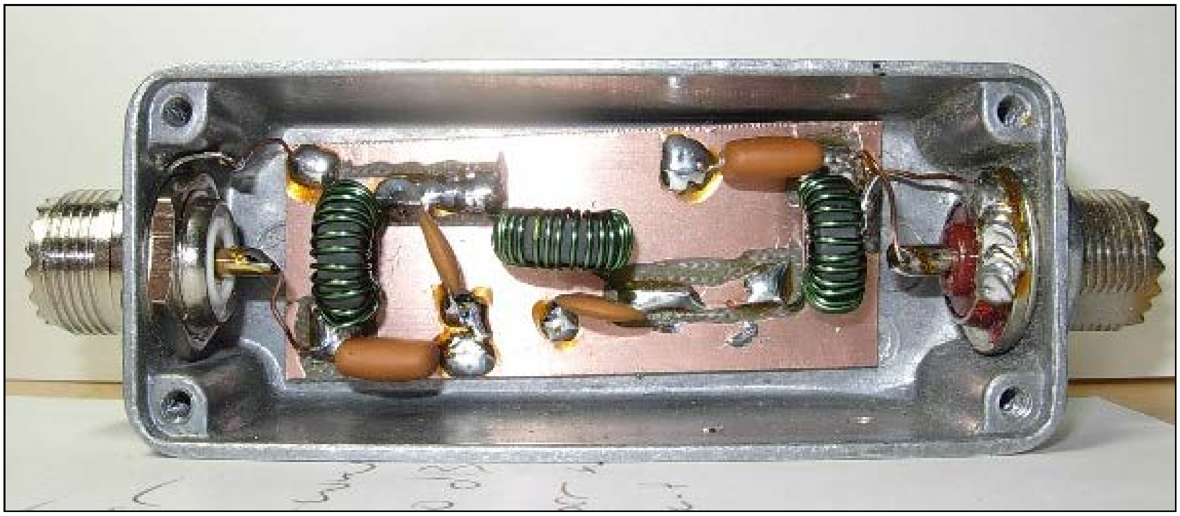
Receiver Testing and Comments

Click on the picture to the left to hear the beacon YWB at 389 KHz with a 500 crystal hertz IF filter engaged on a borrowed Icom R-75 receiver. W7ZOI did some measurements on his R75 receiver S-meter using a signal generator and step attenuator. From S9 on up to 60 over, the steps were very accurate. However, below S9, correlation was poor. The built in attenuator is -20 dB when engaged. On my test receiver, I did some A/B testing with the filter in or out. For 1150 KHz (without the low pass filter) I had to engage the receiver's attenuator as without it, the S-meter reads off scale. With the attenuator engaged, the S-meter reads 50 dB over S-9 when tuned to 1150 KHz.

With the filter connected between the receiver and the feed line, (and the attenuator engaged) the S-meter read S-9.This is a drop of about 50 dB at 1150 KHz which means that this filter pretty much works as designed. I love the Icom R75 receiver; it is good value with it's many features in a compact package. Further testing was undertaken on other frequencies.

When listening to **WWV**; At 2.5 MHz, without the filter, the S-meter read S-9. With the filter inline, I could not hear WWV or see any S-meter reading. At 5 MHz without the filter, the S-meter read was at 20 dB over S-9. With the filter inline, I could still hear WWV very faintly, but the S-meter did not register. There was little noticeable attenuation at less than 700 KHz when using this filter.

Additional Photos and information



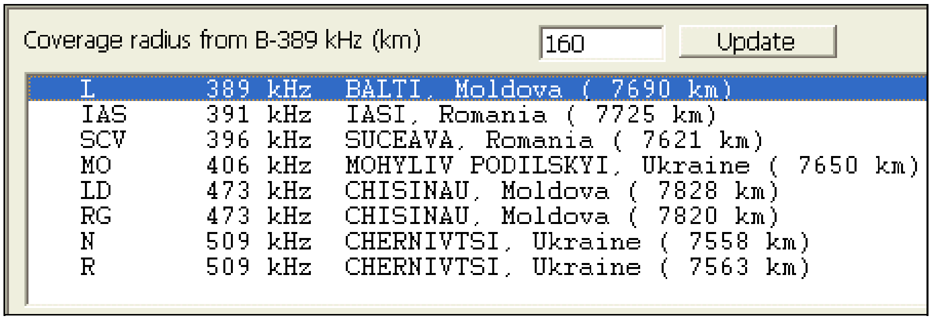
Shown above is the completed project in a Hammond die-cast case with SO-239 connectors at each end.

Shown above a photo of the Skookum beacon SX. It is on 389 KHz. This NDB is located in Skookumchuck BC, Canada

Links

My friend and fellow NDB enthusiast, Ken, VE7KPB has a posting on his web site showing some of the beacons he has logged from his QTH. Consider trying some of these frequencies from your own QTH to get used to finding beacons. Note you must temporarily allow pop ups to see his excellent log.

We recommend this non-directional beacon search and log utility program called WWSU from VE3GOP It must be registered and is a wonderful low cost tool.
I was near beacon L in Balti, Moldova (Балти, Молдова) in 2006. Below is a snippet from the VE3GOP program showing beacon L and also some nearby beacons. (Я изучаю русский язык). Приветствую Вас дорогие друзья!



Coverage radius from B-389 kHz (km) 160

```
L       389 kHz  BALTI, Moldova ( 7690 km)
IAS     391 kHz  IASI, Romania ( 7725 km)
SCV     396 kHz  SUCEAVA, Romania ( 7621 km)
MO      406 kHz  MOHYLIV PODILSKYI, Ukraine ( 7650 km)
LD      473 kHz  CHISINAU, Moldova ( 7828 km)
RG      473 kHz  CHISINAU, Moldova ( 7820 km)
N       509 kHz  CHERNIVTSI, Ukraine ( 7558 km)
R       509 kHz  CHERNIVTSI, Ukraine ( 7563 km)
```

Martin Francis has an excellent NDB web site including the free program called NDB WEBLOG for a number of platforms

Some beacons may be located using this NavAid web site

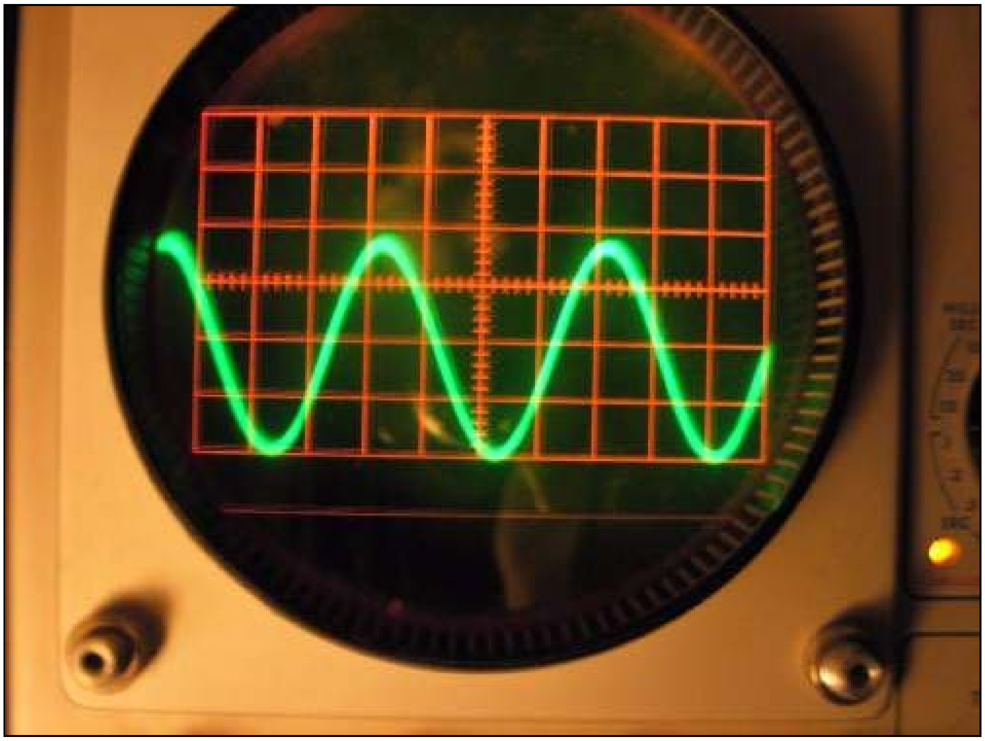
Clint, KA7OEI has an informative web site regarding NDB listening including using digital computer processing to dig out weak signals. This is also a great overview site for newcomers to NDB.

Conclusion

To the right is the outcome when I connect my frequency counter directly to my antenna coax cable feed line. 1150 KHz is my nemesis frequency! Happily it can be tamed with a little filtering to allow NDB listening and logging.

To the right is the outcome with my SWL antenna coaxial feed line connected directly to the scope. The scope was on the 0.5 volts per cm scale, so the peak to peak voltage is 0.2 volts. In just about any high-gain audio amplifier I build, if I touch my finger to the input, I can hear AM 1150 loud and clear - no wonder!

Good luck with your own NDB adventures!



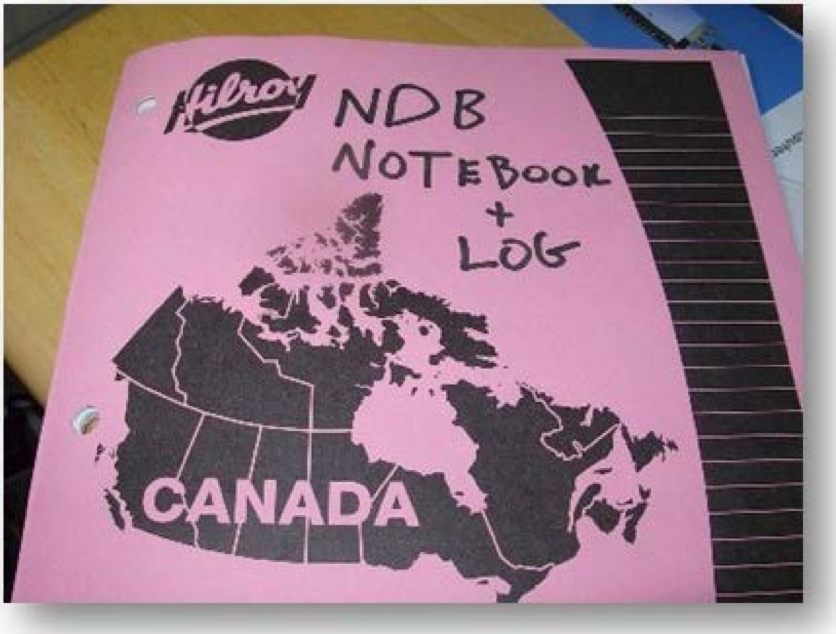
More NDB Information and Circuits

Introduction

Latest Update: December 3, 2012

This web page holds a collection of NDB-related ideas, experiences and hopefully will include some feedback from fellow NDB enthusiasts.

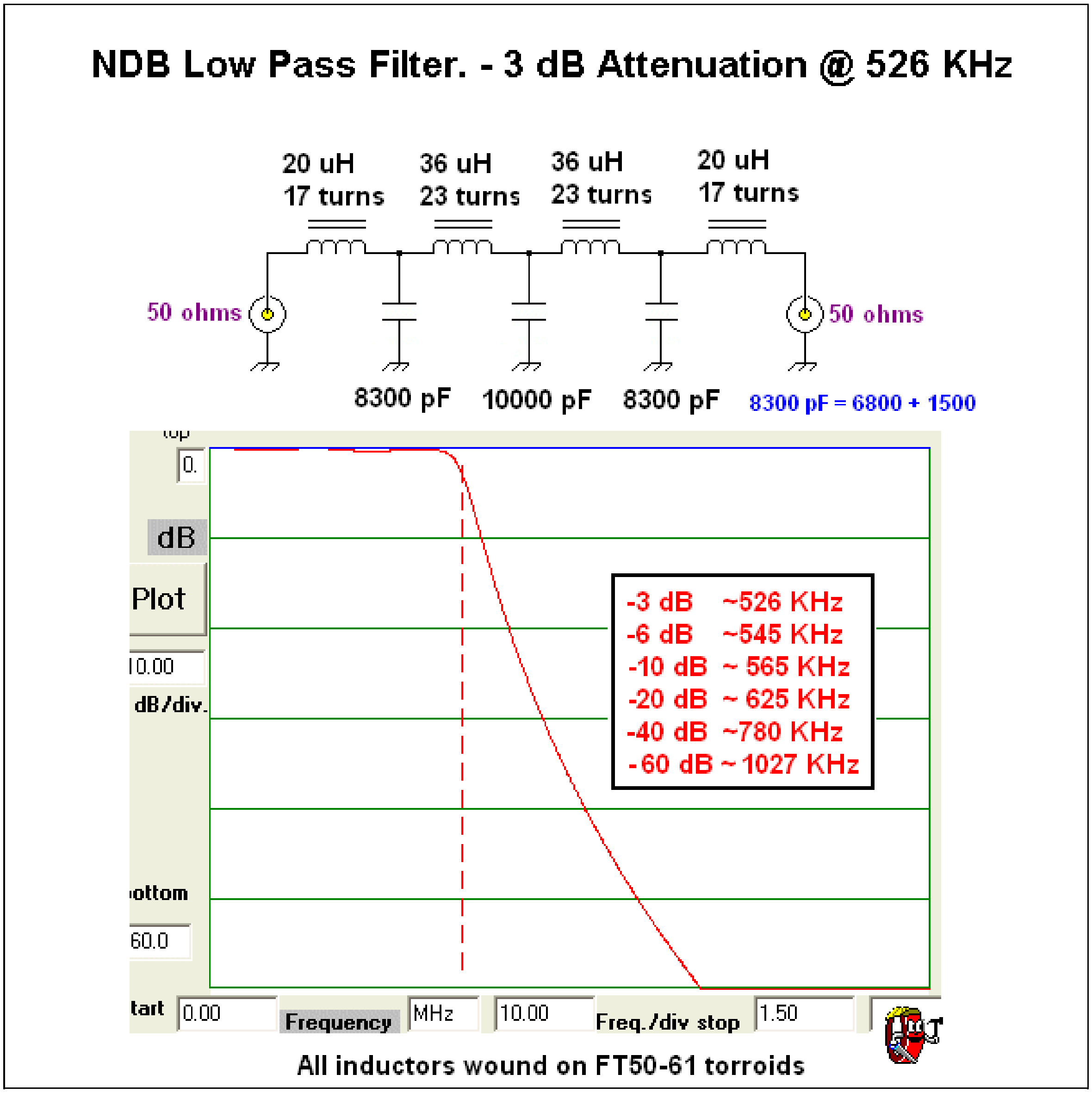
I devoted a new notebook to this topic and hopefully with inspiration from band listening and communicating/learning from others, I will fill it over time. New content will be added to the bottom of the existing material as QRP-Postadata



Improved NDB Chebyshev Low Pass Filter

A popcorn or "junk box" low pass filter was designed and presented on this web page. After discussion with VE7TW and testing a Realistic DX-300 and other receivers, it became apparent that even more attenuation of a strong local BCB station at 1150 KHz was desirable. In addition, there are other moderately strong AM radio stations from 630 to 800 KHz (especially at night time) which maybe causing mixer intermodulation distortion products. A fault of the junk box low pass filter is poor attenuation below 800 KHz and a better design was a prudent goal. Building on the learning obtained from the junk box filter experiments, an improved 7 element Chebyshev low pass filter was designed and is presented directly below. The 3 dB cut off of this filter is calculated to be 526 KHz. This is the filter that I now use for my home radio station. At my

nemesis frequency of 1150 KHz (where a powerful local radio station broadcasts), the attenuation is calculated to be 68 dB. It takes careful layout and a conductive chassis to realize this level of attenuation, but the effort is worth it. In very strong AM BCB locations, you might consider placing 2 such filters in series between your antenna and receiver if required.



NDB Low Pass Filter. - 3 dB Attenuation @ 526 KHz

-3 dB ~526 KHz
-6 dB ~545 KHz
-10 dB ~ 565 KHz
-20 dB ~ 625 KHz
-40 dB ~780 KHz
-60 dB ~ 1027 KHz

All inductors wound on FT50-61 torroids

The schematic and simulation of the improved NDB low pass filter is shown above.

Non-directional Beacon Identification

It is interesting to visit nearby beacons. In the photograph to the right is XC which broadcasts at 242 KHz.

I have learned that it is very important to confirm the NDB stations your hear via a database or list. What you hear on the air should match the database/list for **both** call sign and frequency, else suspect that you may have copied it incorrectly. **RNA**, the definitive signal list for North and Central America plus Hawaii may be found here.

Three Questions

Steve Ratzlaff, AA7U is an experienced NDB DXer and has been listening to beacons since the mid-1980's.
I asked him the following 3 questions:

1. LF beacons do little more than send their station identification in Morse code, are mostly low power and generally might be perceived by some people as boring and low tech. Yet, on the World Wide Web, one finds numerous web sites, software, projects and commercial equipment all passionately dedicated to NDB listening. What's all this fuss about listening to beacons?

Steve: It's a hobby that requires quite a bit of skill and technical accomplishment to get the most from the equipment. Most folks have AC noise to deal with, which can be particularly bad at LF. Finding an antenna that works at LF and that can be used at your own location can be a major task; finding a radio that has decent LF sensitivity, or an LF converter to use with an existing radio--all these must be detitle with just to begin hearing anything at LF. I find it to be quite a challenge. If it were easy to receive LF beacons then I probably would have lost interest years ago! It's true that in recent years several software programs have become available that allow finding beacons somewhat easier--one simply looks for them on the computer screen and decodes the dots and dashes of the beacon being received. This is quite popular among beginners and veterans alike. But the traditional method of aurally listening for the morse code idents of beacons is probably used more often, though many are combining both aural and software techniques now.

2. Let's say I live in a small city lot or even an apartment. I have modest equipment and/or not a huge amount of cash to spend on gear for NDB listening. From the antenna through to the headphones, what are some basic recommendations you might give to a newcomer wanting to get started in NDB listening?

Steve: The radio must have decent sensitivity at LF, or else an LF converter must be used. Due to high local AC noise, any type of LF antenna used indoors will be a poor substitute to one that can be placed outdoors. A few portable radios cover the LF NDB frequency range that will work for hearing local beacons, though the radio may need to be used outside to get away from AC noise. The discontinued Sony 2010 was the standard for portable radios for reasonable LF performance. Newer radios like the Degen DE1103 have been found to work reasonably well at LF and can be bought for well under $100 by mail order from eBay sellers; or the more expensive Kaito 1103 version, which has a warranty, can be obtained from several distributors like Universal Shortwave. The much more expensive semi-portable Eton E1 works well at LF, but is more in the price range of a tabletop radio. The Icom R75 is currently the best bargain in a tabletop radio that has very good LF sensitivity as well as 1 Hz tuning, which is an asset if a narrow external audio filter is used. I'm not too optimistic about what someone living in an apartment or high rise building might do to successfully receive LF beacons indoors. Often the AC noise level is too high to be able to use an indoor antenna. But some have been able to use loop antennas indoors for the stronger signals. An example of a top of the line commercial loop would be the Wellbrook ALA1530 or LFL1010. Unlike at shortwave frequencies, where simply tossing a wire out the window to a nearby tree or other support, or even running the wire around the room inside, will usually work fairly well, at LF a wire less than several hundred feet generally doesn't perform very well. It can be argued that an active whip antenna makes a very good LF antenna, and doesn't take up much room, but it must be used outdoors. And if there are strong AMBCB signals, then the active antenna, either loop or whip, must have very good overload resistance otherwise it can generate distortion of its own from the strong BCB signals.

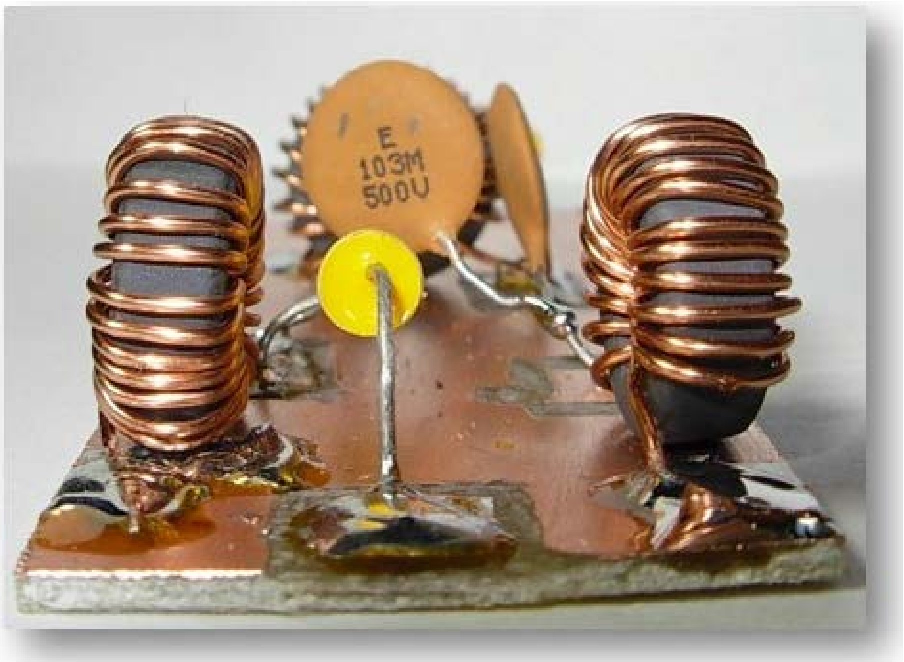
3. What kind of distances are considered DX for NDB?

Steve: NDB DX is pretty much a relative thing. One just starting out might be thrilled to hear a beacon from the next town, or from the other side of his own state or province. As one improves his listening setup and gains experience, then usually DX goals also expand to try to hear beacons farther and farther away. NDB DXing generally is not a competitive hobby, unlike amateur radio with its various competitive "contests". Each person's listening setup, local noise level, etc. is usually very different from someone else's, even someone in the same town or general area. One person might live in the suburbs and have a lower noise level than his friend who lives right in town and has a much higher noise level. One might have room to put an antenna in a quiet spot; the other might be limited to much less. People who live near an ocean generally have a much better chance at hearing something exotic offshore than folks living far inland. Folks living in the central or eastern part of North America have many more beacons available to be heard than folks in western North America. But there are always a few beacons that are much stronger than most, and can be heard from long distances of 1000 miles or more, pretty much anywhere in North America at night. One example would be 206 GLS in Galveston, Texas, which runs around 2000 watts, has a large antenna, and is generally readily heard anywhere in North America at night--that beacon might be 1500 miles or more away, and might be considered real DX. However another 25 watt beacon from the same general area in Texas might be hard to hear only several hundred miles from that beacon. So "DX" is pretty much a relative term. Ndblist, an international email list devoted to beacons, is open to anyone with an interest in beacons--members post their loggings there. What might be a local beacon to someone might be DX to someone in a different part of the country. All levels of experience are welcome.

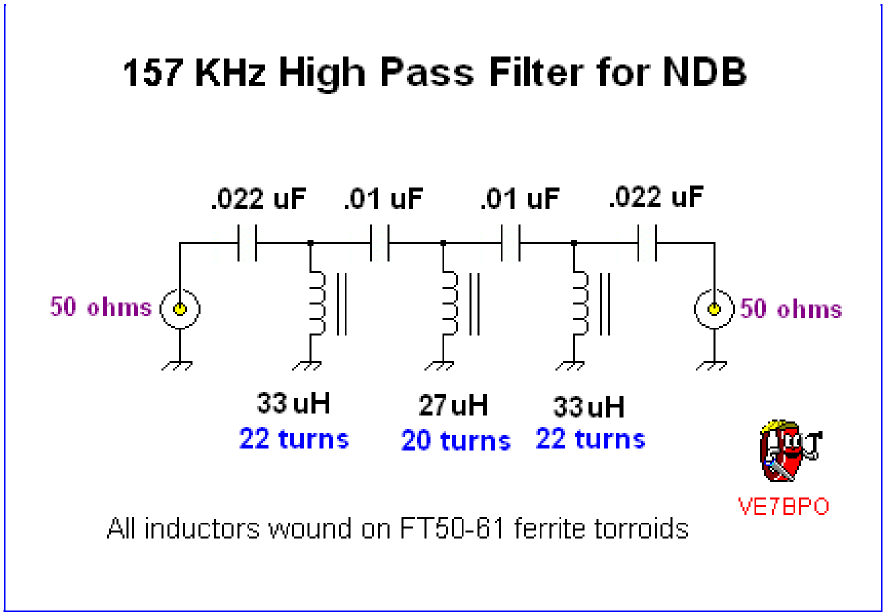
Thanks Steve.

NDB High Pass Filter

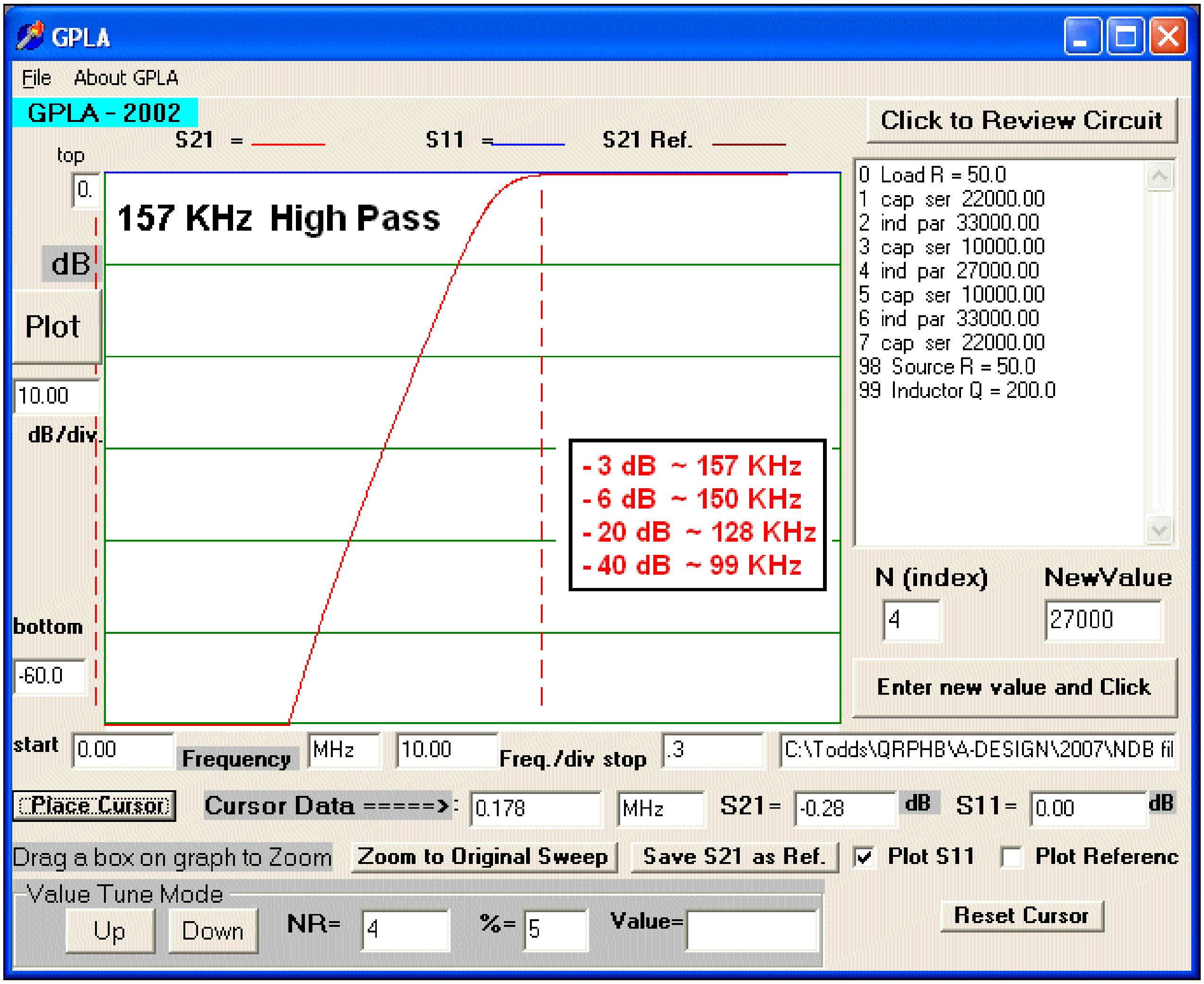
A high pass filter using standard value capacitors was designed using GPLA. although, such a filter would not help AC line noise and RFI generated in the house, I suspected my antenna was picking up local noise from below the NDB band. This filter was mounted inside a die-cast Hammond box with a SO-239 at each end. I used 22 gauge enamel covered wire for the inductors. A photo of the filter is shown to the right.
For the 0.01 uF caps, I used junk box ceramic capacitors with a 20% tolerance, however, I did measure a bunch and found 2 within 5% tolerance for my filter bread board.



To the left is the filter schematic. This is an N = 7 Chebyshev high pass filter with a 3 dB cut off of 157 KHz. This cutoff frequency allowed the use of common, standard value capacitors and also even turns numbers to reach the desired inductance for the inductors when wound with FT50-61 ferrite cores.



Use 5% tolerance, high Q caps such as polystyrene or NP0 ceramic and **not** junk box bypass-grade ceramic capacitors as possible. I used trashy ceramic caps for the 0.01 uF parts due to lack of better parts at the time of building and testing.



- 3 dB ~ 157 KHz
- 6 dB ~ 150 KHz
- 20 dB ~ 128 KHz
- 40 dB ~ 99 KHz

0 Load R = 50.0
1 cap ser 22000.00
2 ind par 33000.00
3 cap ser 10000.00
4 ind par 27000.00
5 cap ser 10000.00
6 ind par 33000.00
7 cap ser 22000.00
98 Source R = 50.0
99 Inductor Q = 200.0

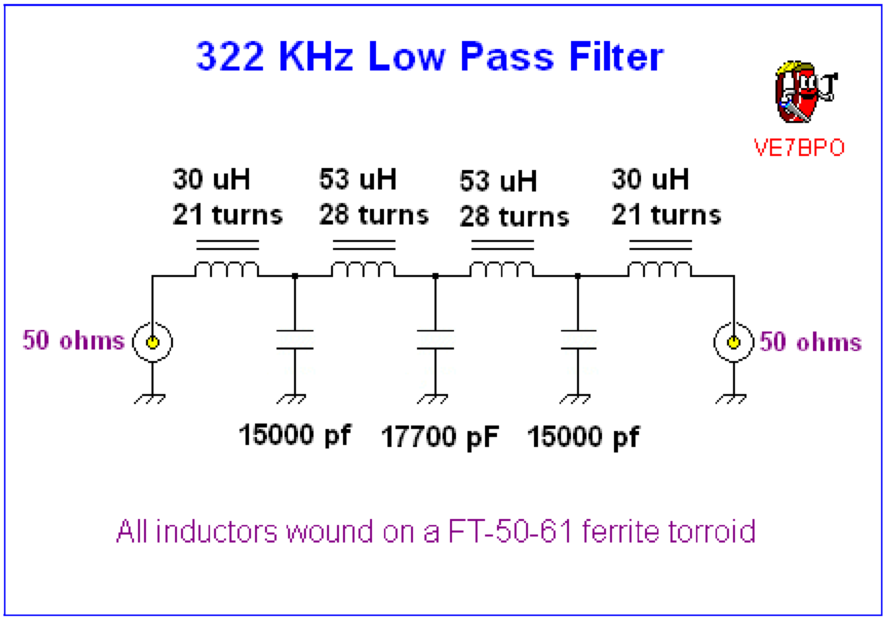
top: 0.
dB/div: 10.00
bottom: -60.0
N (index): 4
NewValue: 27000
start: 0.00 MHz
Freq./div: 10.00
stop: .3
C:\Todds\QRPHB\A-DESIGN\2007\NDB fil
Cursor Data =====>: 0.178 MHz
S21= -0.28 dB
S11= 0.00 dB
☑ Plot S11
☐ Plot Referenc
NR= 4
%= 5
Value=

Above is the filter GPLA simulation. In particular, I have harsh noise from about 110 KHz on down. At 78 KHz, where this filter has a calculated attenuation of ~ 56 dB, I made an audio file of the band noise. This is in AM mode with the filter out for a few seconds and then in line. With the high pass filter in line, there is pronounced attenuation of the noise and my local 10 KW BCB station at 1150 KHz suddenly appears. Prior to this it was hidden by the harsh noise. At frequencies less than ~200 KHz (without my low pass filter) I can hear this BCB station intermittently as I tune around. I suspect that the R75 filtering down at 200 KHz and down is insufficient to stop this monster station.

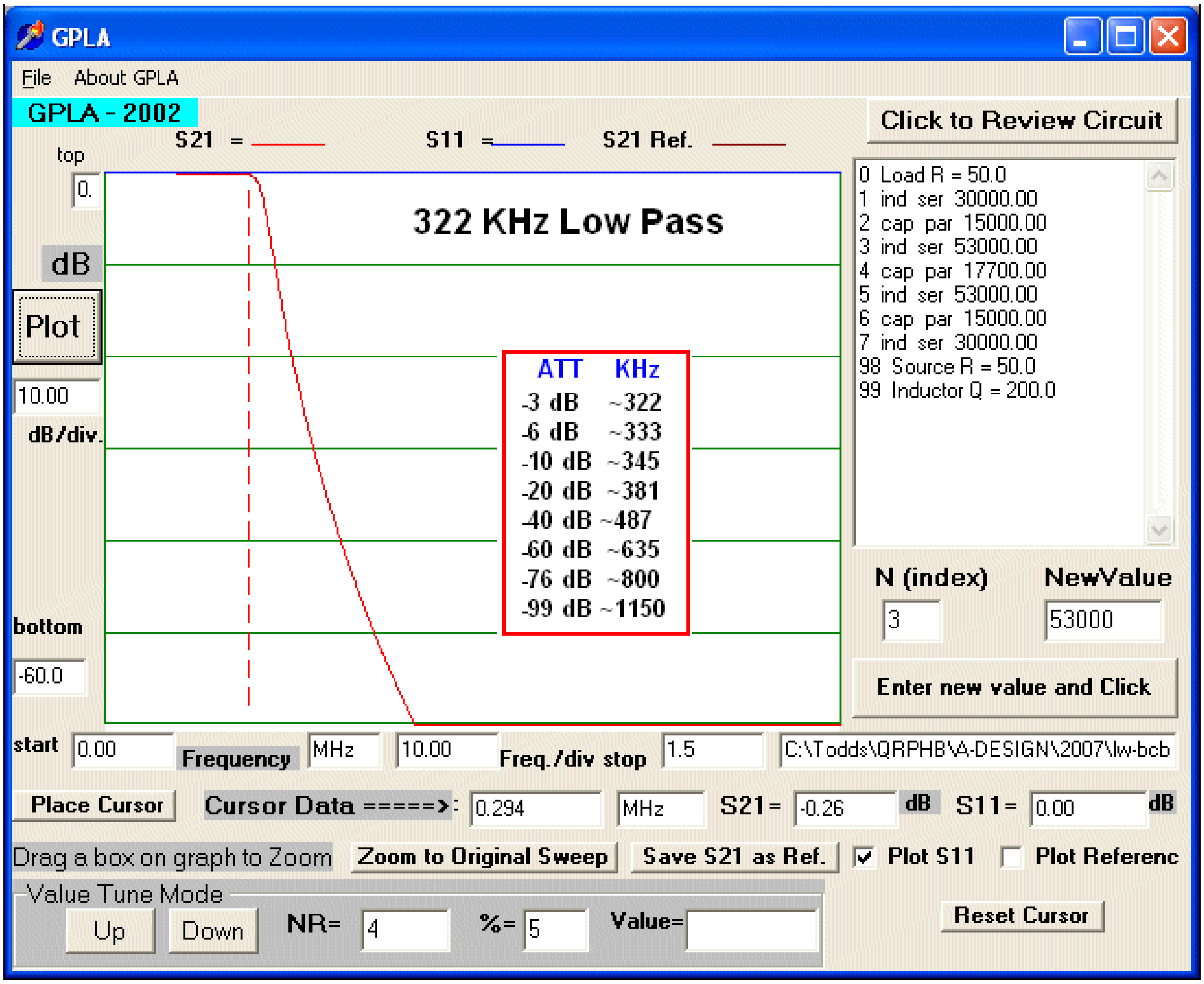
At my QTH, using a high pass filter reduces some of the noise on the NDB band. At my location, a high pass plus a low pass filter in cascade between my antenna and my receiver results in less QRN and easier weak signal copying.

Long Wave Broadcast Radio Filter

I learned about LW Broadcast radio from Steve Ratzlaff. In particular, радио россий "Rah-deo RaSEE" (make sure you roll the R!) can occasionally be heard on the west coast and broadcasts at night-time using 500-1000 KW power. The frequencies he recommended to try were 153, 180, 189, and perhaps 171, 234 and 279 KHz. I have terrible problems with a local BCB radio station at 1150 KHZ that causes intermodulation distortion and/or blow-by detection at and below 200 KHz in addition to a terrible noise source at 78-120 KHz. Therefore, I built another cascade low pass/high pass filter and placed it in the same chassis as my regular NDB low pass/high pass combination filter for use when tuning LWBC and perhaps for when listening to frequencies less than 200 KHz.



Above is. the schematic of the 322 KHz low pass filter. In the photograph above, you can see a 50 ohm pad at the input that was used only during testing. This filter offers a calculated attenuation of ~ 98 dB to my 1150 KHz interfering station. In reality it is not possible to achieve this level of attenuation, however, there is no detectable 1150 KHz signal interference with the filter in line which makes me happy. Click here to listen to the dramatic difference with regard to interference this filter makes at my QTH with my receiver tuned to 199 KHz. The receiver is set for wide band AM detection; first without the LWBC filter and then with the filter switched in. When the filter is switched in, the BCB interference disappears and a Canadian NDB (UAB @ 200 KHz can faintly be heard along with our cat meowing in the background. It is not possible to listen to LF without aggressive low pass/high pass filtering at my QTH.



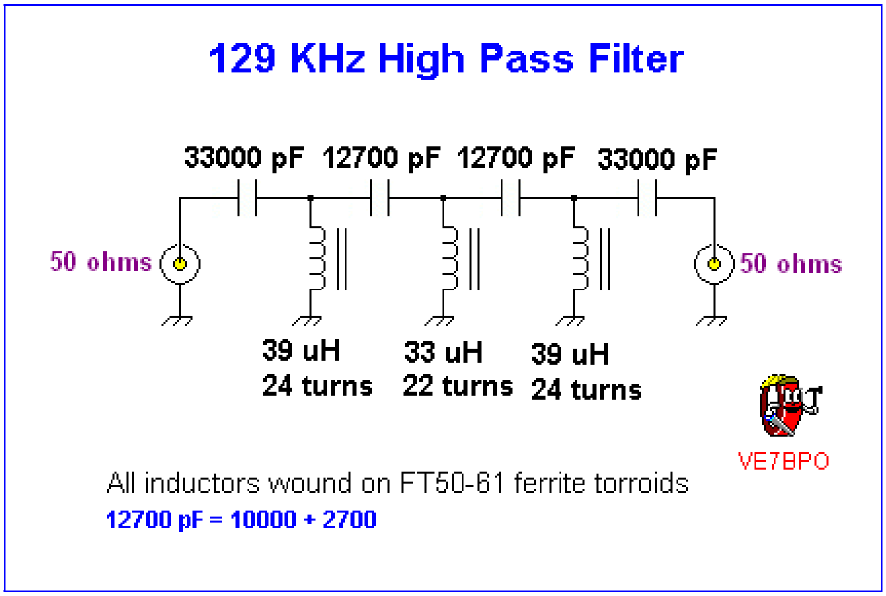
322 KHz Low Pass

ATT	KHz
-3 dB	~322
-6 dB	~333
-10 dB	~345
-20 dB	~381
-40 dB	~487
-60 dB	~635
-76 dB	~800
-99 dB	~1150

0 Load R = 50.0
1 ind ser 30000.00
2 cap par 15000.00
3 ind ser 53000.00
4 cap par 17700.00
5 ind ser 53000.00
6 cap par 15000.00
7 ind ser 30000.00
98 Source R = 50.0
99 Inductor Q = 200.0

top: 0.
dB/div.: 10.00
bottom: -60.0
start: 0.00 MHz
Freq./div: 10.00
stop: 1.5
C:\Todds\QRPHB\A-DESIGN\2007\lw-bcb
N (index): 3
NewValue: 53000
Cursor Data =====>: 0.294 MHz
S21= -0.26 dB
S11= 0.00 dB
☑ Plot S11
☐ Plot Referenc
NR= 4
%= 5
Value=

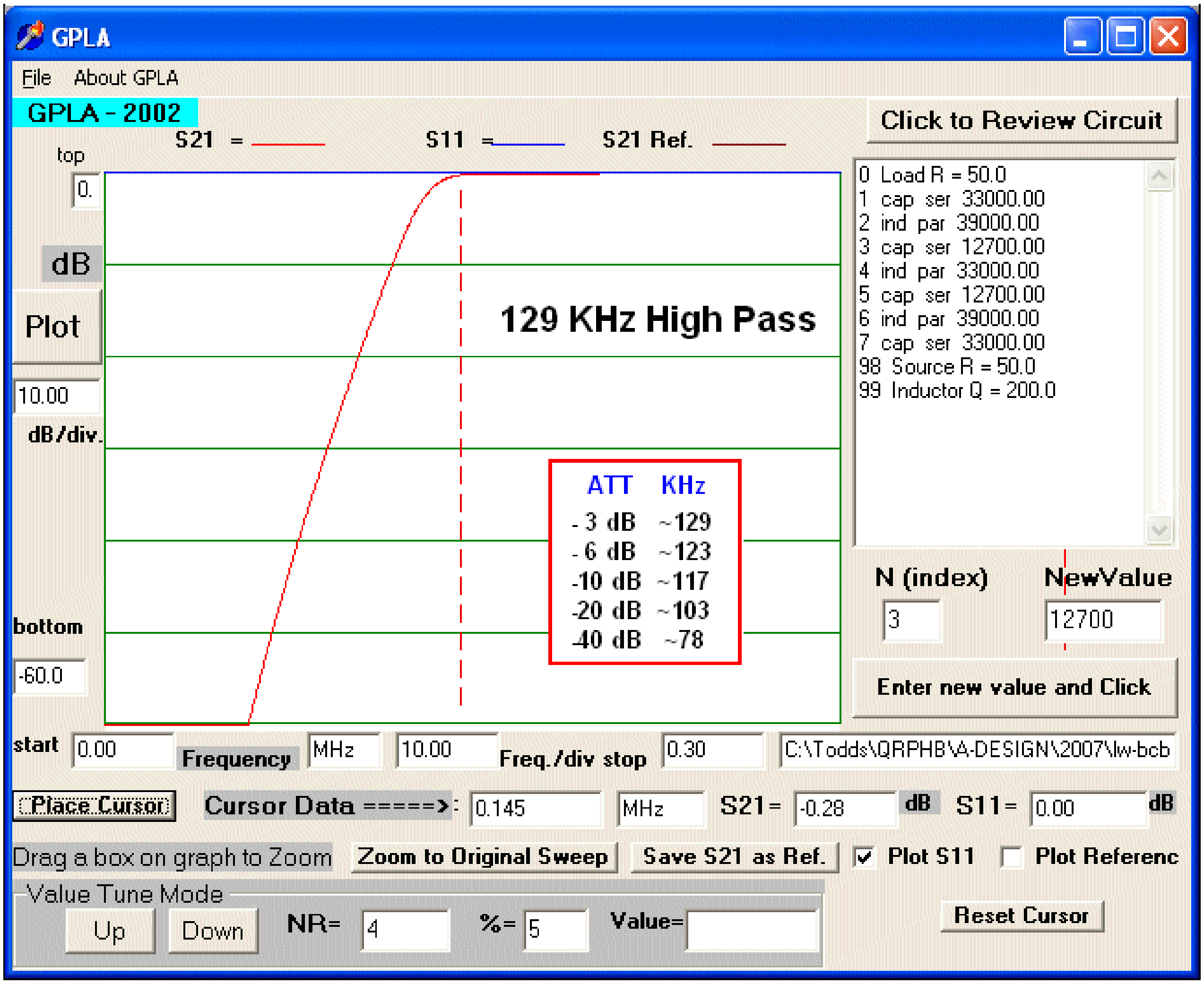
Above is the GPLA simulation of the LWBC low pass filter.



All inductors wound on FT50-61 ferrite torroids

12700 pF = 10000 + 2700

Above is the schematic of the 129 KHz LWBC high pass filter. In either of the 2 filters, capacitor values can be obtained using 1 or 2 standard value capacitors in parallel. The cutoff frequencies of both filters were chosen to allow using practical component values.



129 KHz High Pass

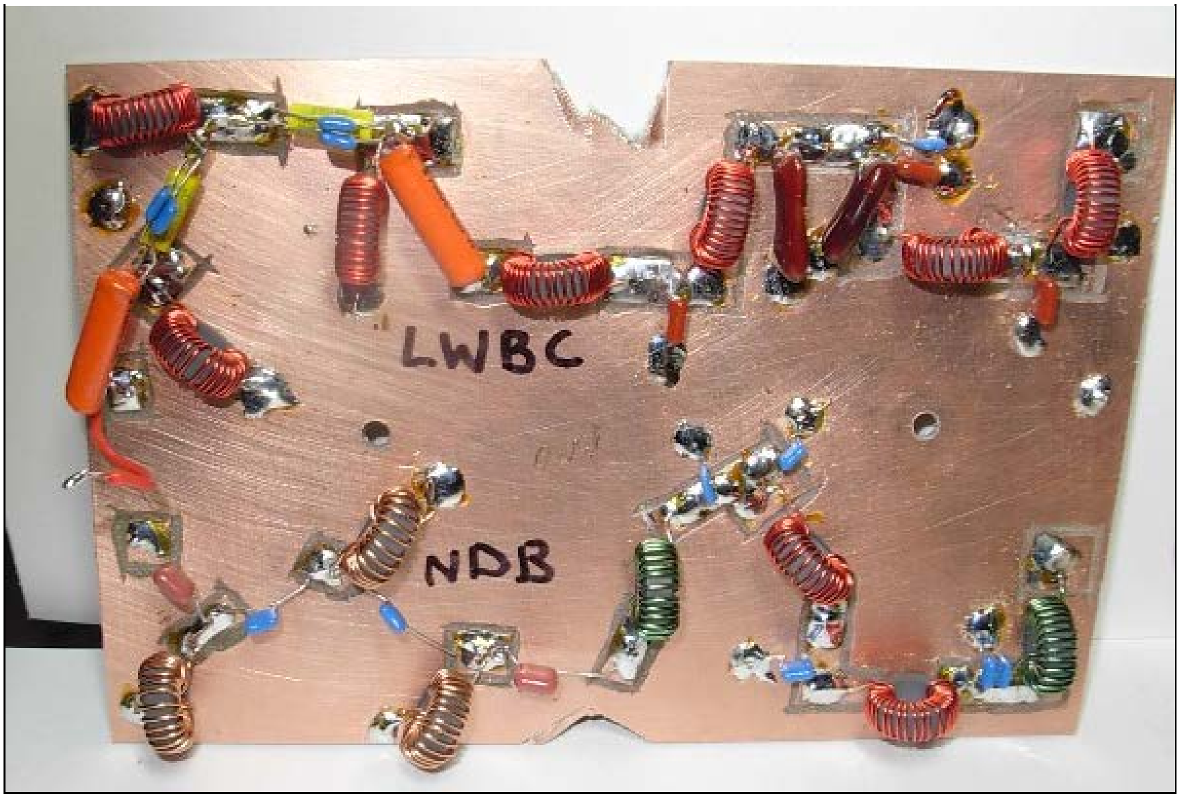
ATT	KHz
- 3 dB	~129
- 6 dB	~123
-10 dB	~117
-20 dB	~103
-40 dB	~78

0 Load R = 50.0
1 cap ser 33000.00
2 ind par 39000.00
3 cap ser 12700.00
4 ind par 33000.00
5 cap ser 12700.00
6 ind par 39000.00
7 cap ser 33000.00
98 Source R = 50.0
99 Inductor Q = 200.0

top: 0.
dB/div.: 10.00
bottom: -60.0
start: 0.00 MHz
Freq./div: 10.00
stop: 0.30
C:\Todds\QRPHB\A-DESIGN\2007\lw-bcb
N (index): 3
NewValue: 12700
Cursor Data =====>: 0.145 MHz
S21= -0.28 dB
S11= 0.00 dB
☑ Plot S11
☐ Plot Referenc
NR= 4
%= 5
Value=

Above is the GPLA simulation of the LWBC high pass filter. The high pass filter might not be needed at your QTH. My LWBC filter has the low pass filter before the high pass filter. I.e. they are in series or cascaded.

Dual NDB and LWBC Filters



For use in my radio shack, I built LWBC and NDB filters inside 1 chassis with separate inputs and outputs. Some photos of this project are shown directly above and below. The NDB filter is the 526 KHz low pass filter in series with the 157 KHz high pass filter. The LWBC filter is the 322 KHz low pass filter in series with the 129 KHz high pass filter. High Q caps were used and the inductors were wound with either 22 or 24 gauge wire to obtain a relatively high unloaded Q. The large Hammond project case allowed reasonable spacing of the inductors and a nice long input to output layout.

Beacon XJ @326 KHz

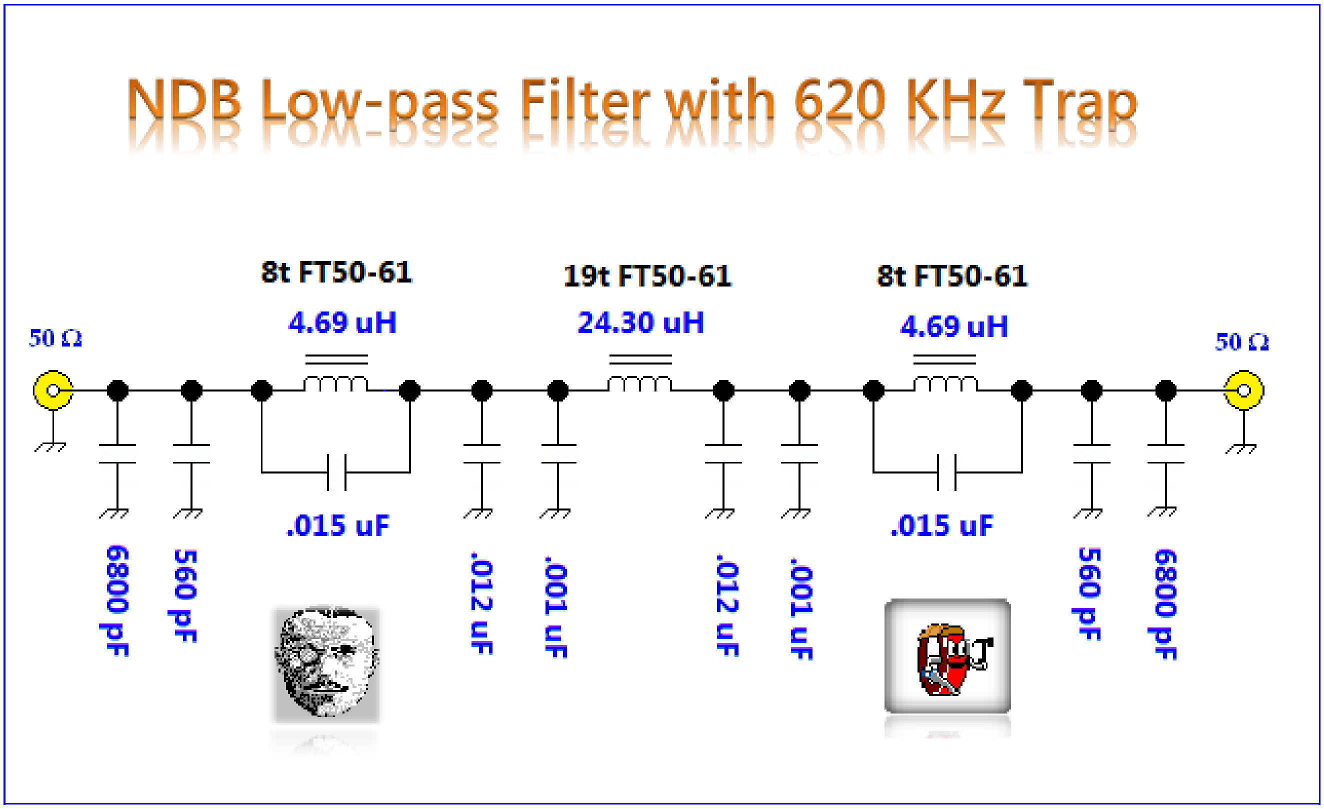
Above is NDB XJ in Fort St. John, BC. Photo by VE7KPB in August 2008.

QRP — Posdata: NDB Low-pass Filter with Trap

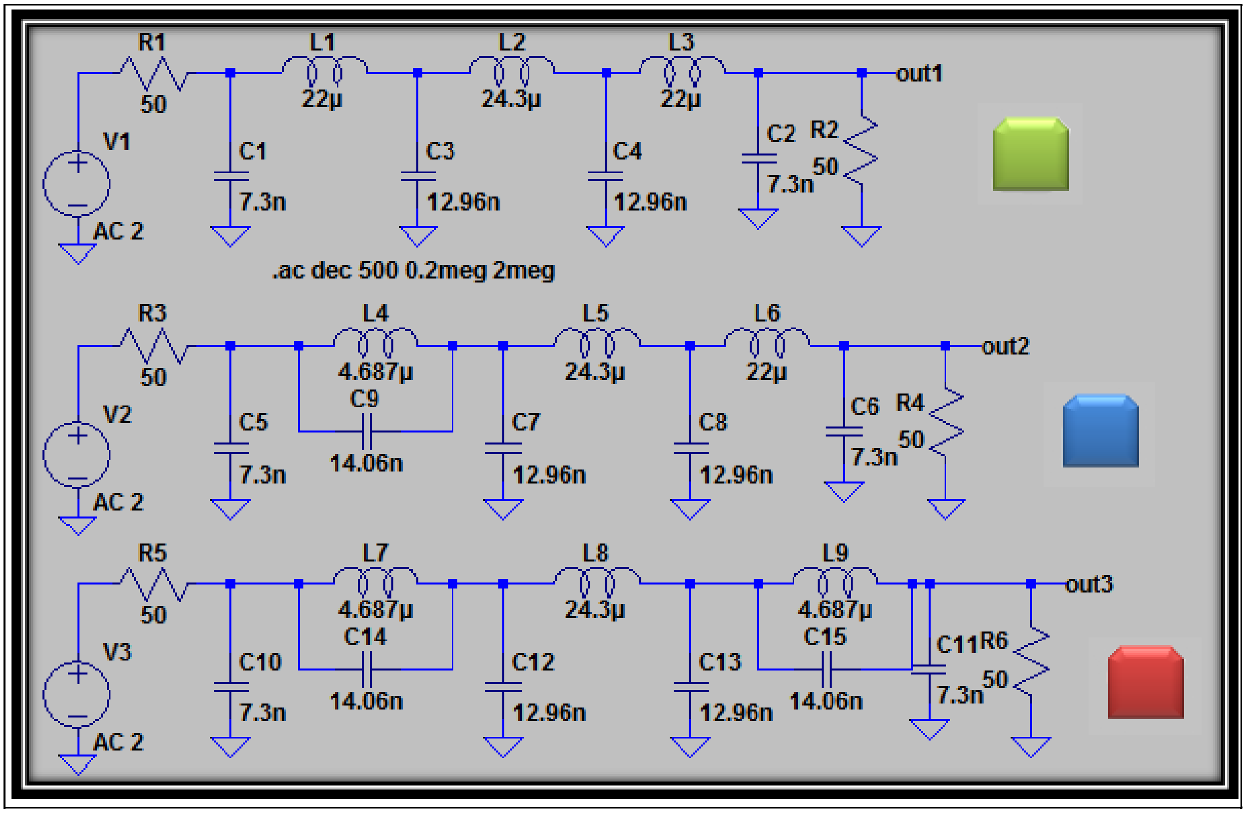
An email from Rick, NU7Z spawned this 2012 addition.

Depending on their design, typical NDB low-pass filters provide less than 20 dB attenuation at 620 - 630 KHz, and if you hear a strong station on this frequency — good luck!

Rick sought a filter with a trap at ~ 620 KHz — after mulling around, we encountered design problems with a trap frequency so close to the low-pass cut-off frequency and later asked Wes, W7ZOI if he might help design our filter.



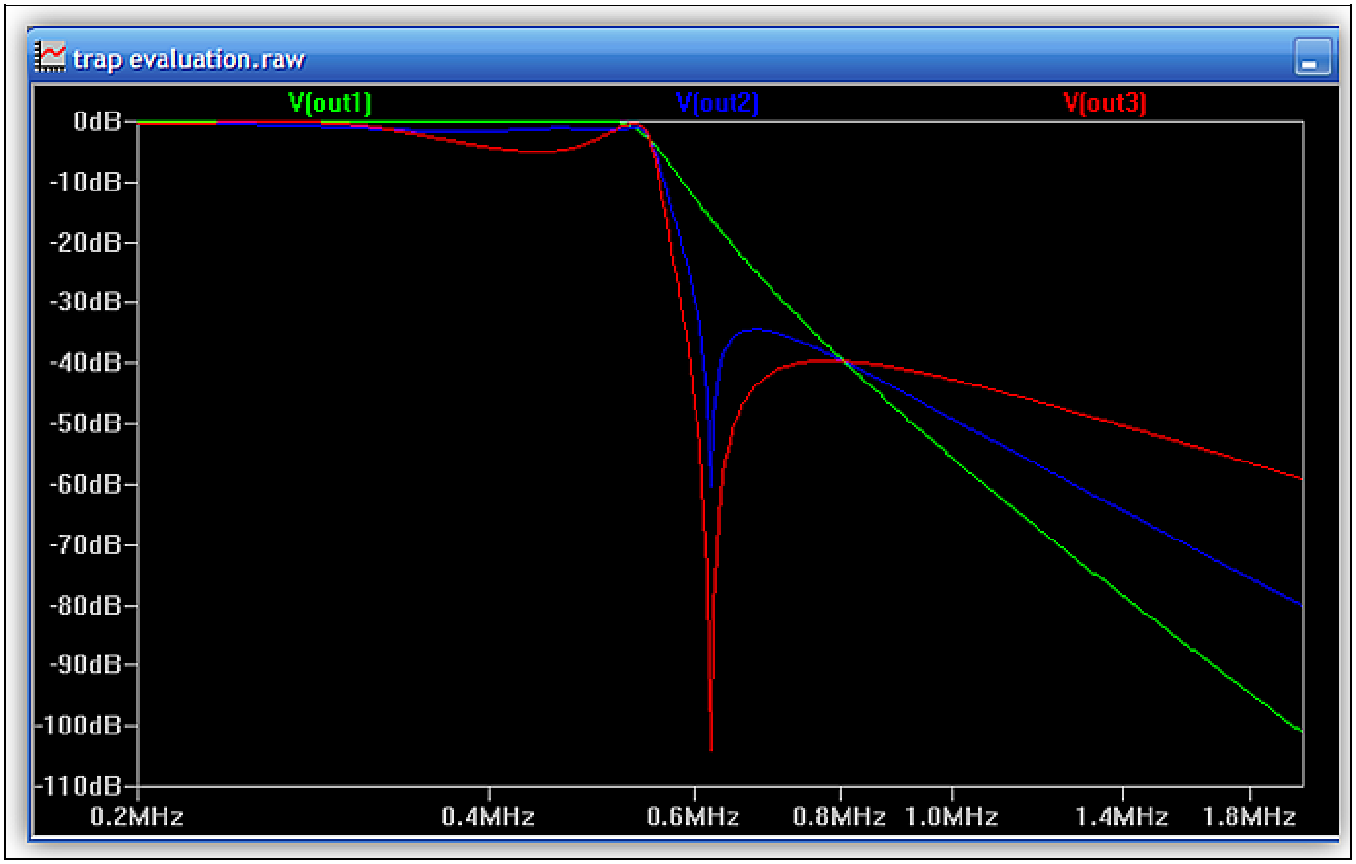
Above — My version of the Wes, W7ZOI designed NDB low-pass filter with a trap. **Red** filter below



Above — A 7th order, 0.1 dB ripple **Chebyshev low-pass filter** with a 550 KHz cut-off filter evolved to include **1 trap**, and then **2 traps** at 620 KHz.

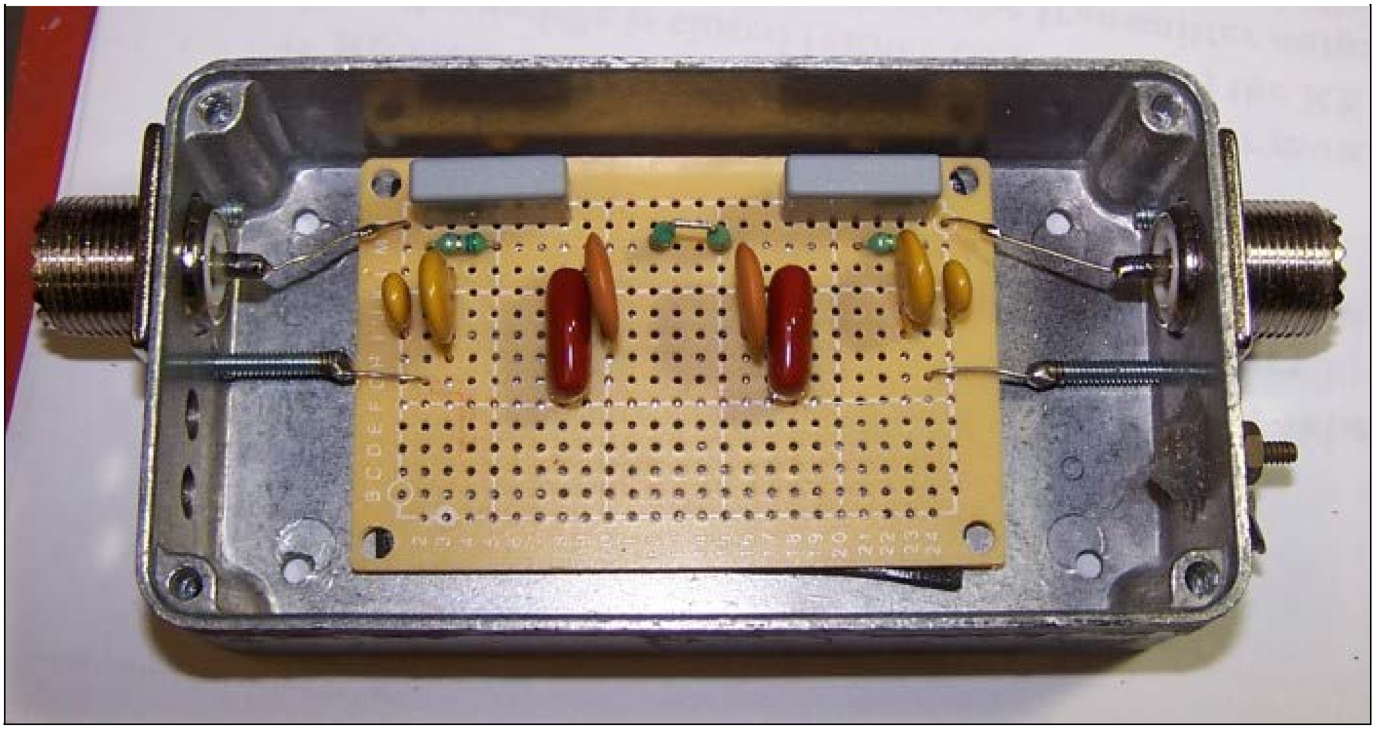
We learned that in simple situations, you may modify the elements of a low-pass filter so that the usual inductor is replaced by a parallel trap. See Wes' work in EMRFD Chapter 3; in particular, Figure 3.10.

Wes wrote he's employed this technique successfully before — for example, to add harmonic suppression to a simple output network for a QRP transmitter, although he hadn't added traps to higher order filters like the 1 we wanted. Click for a file containing the math contributed by Wes, W7ZOI.



Above — The SPICE analysis of the 3 color-coded filters above. This design excludes the impact of finite L and C and the unloaded Q that could significantly affect function since the trap frequency is close to the low-pass cutoff frequency. These factors usually worsen the insertion loss near cutoff, but since we're using this filter in a noisy RF environment, filter misperformance should be tolerable.

In the future, Wes recommended designing an elliptical low-pass filter with software such as that distributed by AADE.



Above — A version of the filter built by Rick, NU7Z using epoxy-coated inductors for the L's. The insertion loss with these inductors = ~ 5 dB, although he runs a 40 dB receive preamp and can accommodate such losses.

Despite employing a loop receiving antenna, he could not listen around 500 KHz due to a loud, local broadcast station at 630 KHz. Inserting this filter reduced this 630 KHz signal from 40 dB over S-9 down to S-1 on his receiver S-meter.

Fantastic! Big thanks (большое спасибо) to Wes, W7ZOI.

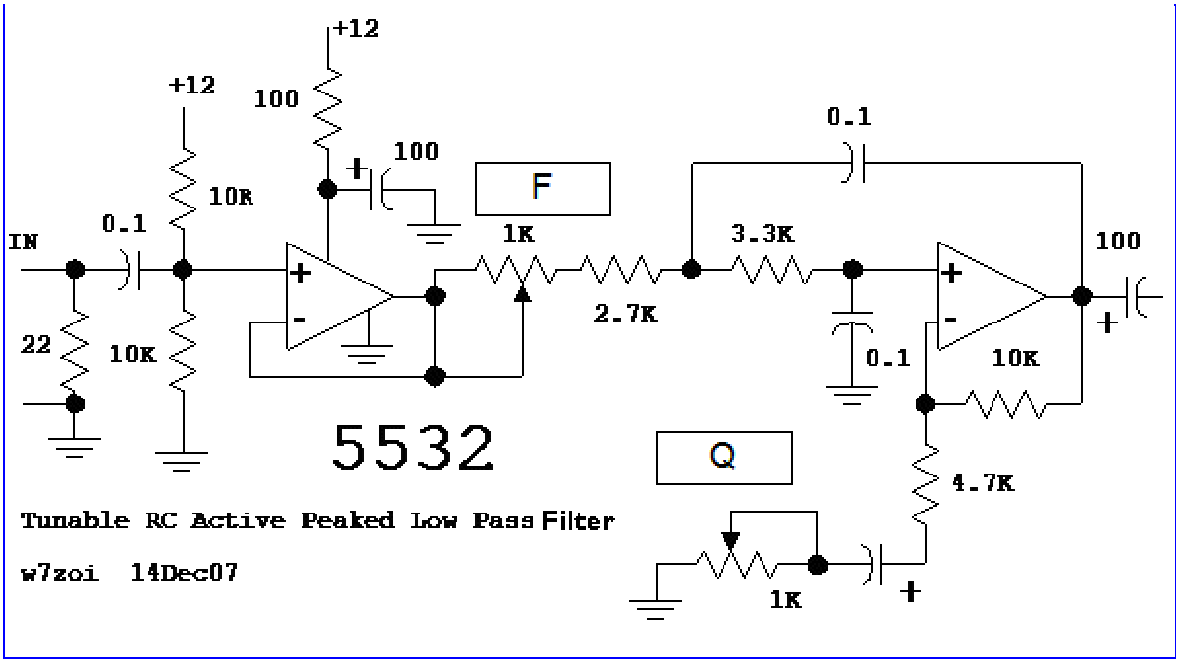
Amateur and Short Wave Radio Electronics Experimenter's Web Site

440 Hertz Peaked Low Pass Audio Filter Experiments

Introduction

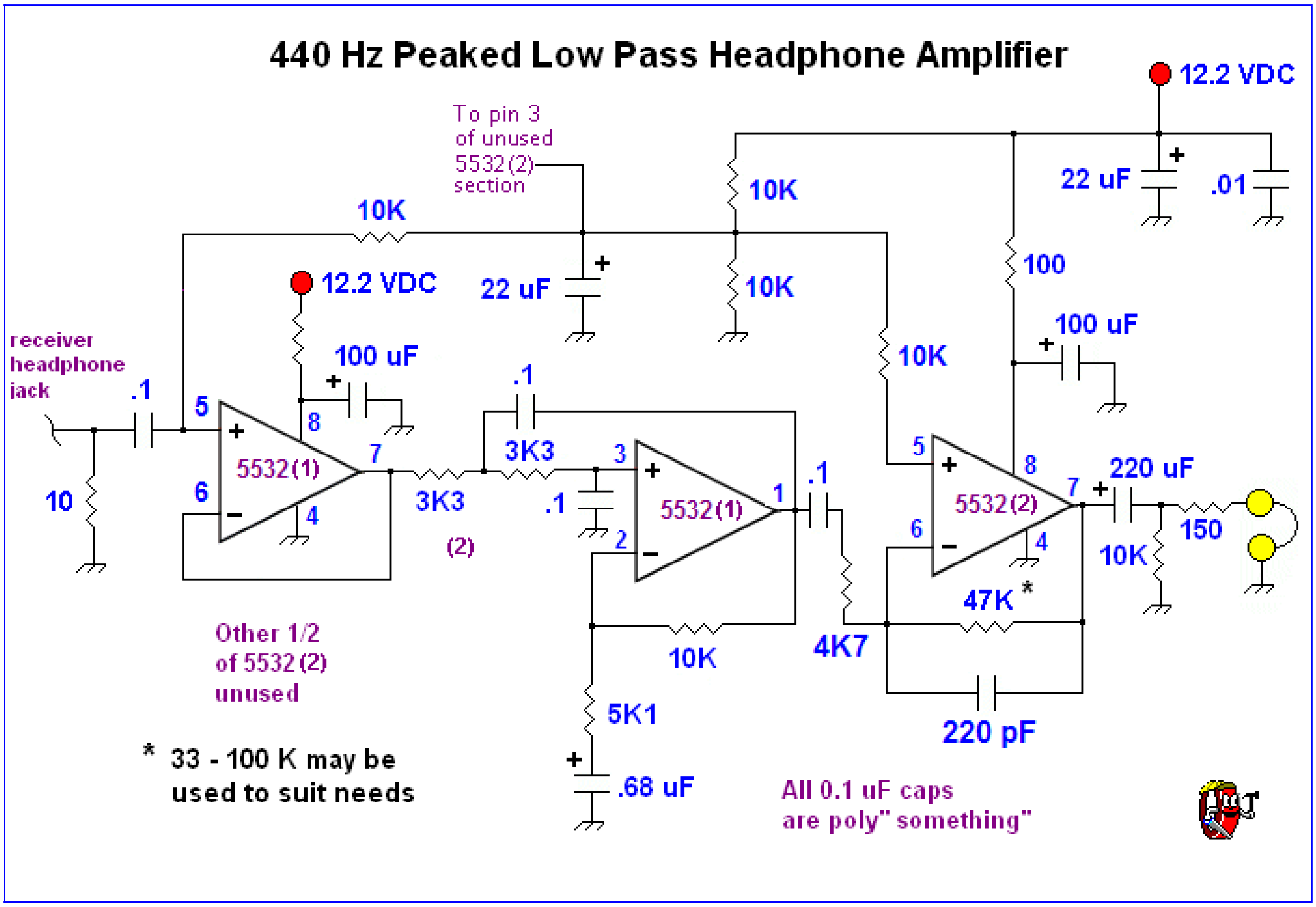
At audio frequencies, low pass filtering can go a long way to improving CW and beacon reception. It has a been a long time since I built one and therefore decided to experiment with some designs using the 5532 op amp. EMRFD has a great section on RC active audio filters starting at Chapter 3.5 and this is where I began. After some experimentation, I remembered a peaked low pass filter designed and published by Wes, W7ZOI in the 1970's. This filter became very popular in Russia after publication in a 1971 Russian Amateur Radio Journal. I asked Wes if he might design another low pass filter peaked for 440 Hertz, which is my favorite CW beat note frequency. This filter is intended for use as an out board headphone jack device for the Icom R75 or other receiver.

Base Project Schematic



Above is the filter designed by Wes, W7ZOI. It has two 1K pots you can tweak; a subtle frequency control and a Q adjustment. It is theoretically possible to adjust the Q too strongly as to cause oscillation, although this did not happen with my bread board.

Schematic and Circuit Building Details

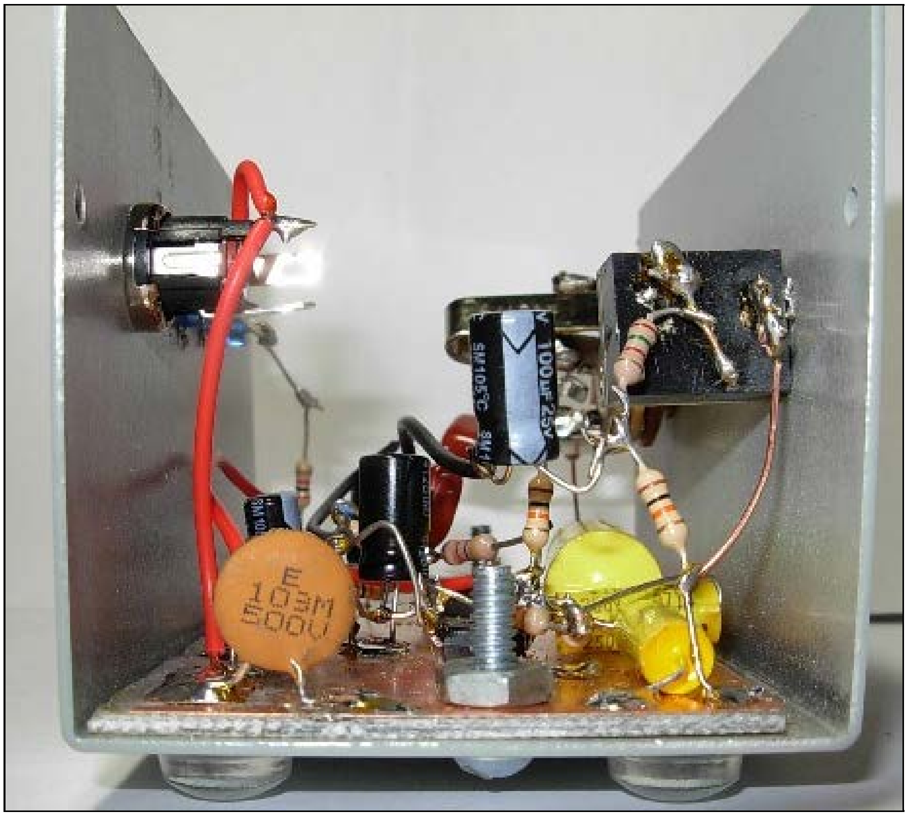


Above is the final filter design. My prototype filter had 2 variable pots and I disconnected and measured them after discovering my favorite setting and then constructed the version shown above using fixed-value resistors instead of the potentiometers. If you like to tweak knobs, you might leave 1 or both of the pots in. The 0.68 uF cap can be raised as high as 4.7 uF if you have poly caps this large in capacitance value. although, it seems wasteful, I did not use one half of the second 5532 op amp. Feel free to add another pole of low pass filtering or something else to utilize this stage if you like. Pin 3 of the unused, second op amp 1/2 is connected to the 10K/10K voltage divider bias with a wire as indicated in the text.

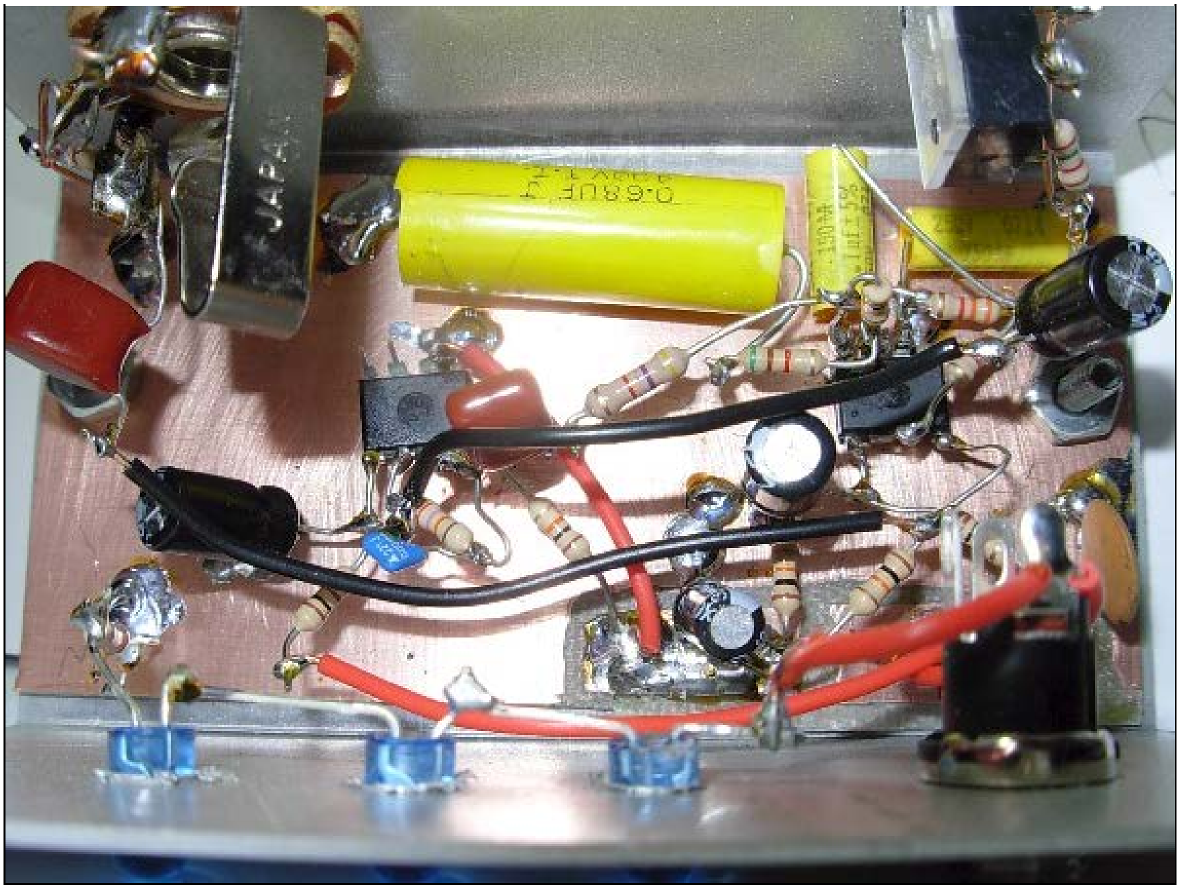
This filter sounds the best when the R75 volume control is minimally turned on as I suspect some of the wide band noise heard is from the ICOM AF chain. More importantly, If the R75 audio gain control is turned too high, the filter will be overdriven and sound distorted This is especially true when using the 2.4 KHz wide SSB filter on the R75. The 10 ohm filter input resistor attenuates the receiver output and makes it more difficult to overdrive the audio filter. The 500 Hz filter at the 9 MHz Receiver IF has quite a bit of of loss and with this filter switched in, it is difficult to overdrive the audio filter. For best results, an audio filter should be placed just after the first AF preamp stage, however, using the headphone jack is the only option available for adding AF filtering in most commercial receivers I have used.
The second op amp stage is used to increase the headphone volume and the 47K feedback resistor can be adjusted to suit your needs. It is really important to experiment with the component values which will match your receiver and the IF filters and antenna you have. For example the input shunt resistor may be increased from 10 to 18-22 ohms if you always use a narrow IF filter during CW and beacon listening or received signals are low in volume. This is an experimenter's circuit, not a finished project. Overall, this circuit has low output volume and is really gentle on the ears in terms of noise and amplitude.

Project Breadboard and Samples

Shown to the right is a side photo of the experimental project. The big yellow Mallory polyester caps were used as I did not have any other desirable AF filtering caps in my parts stock. The day after, I built this filter, my parts order (including a big selection of polyester film audio caps) from Digi-Key arrived, however this is Murphy's law. These Mallorys are good quality capacitors- just a little large!



Low pass filters can really help reduce noise during reception. Two example audio files follow. These were heard in the NDB band using a 2.4 KHz SSB IF filter.
290 YYF
312 UNT
The occasional scratchy noises are me moving as I held the headphones around the microphone.



Above is the filter photographed from above. The 3 blue LEDs are used to light the lower row of buttons on the R75 as it is difficult to see them with a low level of light in the radio shack. No hum is heard with this filter. When you build AF amps or filters with the 5532, after soldering pin 4 to the copper ground plane, start out by connecting the components associated with pins 6 and 7 and then 1 and 2. I suggest this as placing components between adjacent pins is often the most difficult part of building when using ugly construction with op amps.

If by accident you make an unintentional solder bridge between 2 adjacent pins on an op amp, simply heat the 2 bridged pins up and gently drive a small screw driver between the pins. This should remove your unwanted solder bridge. The 5532 op amp is quiet and relatively inexpensive. In EMRFD, there are countless examples of how one can use them in a variety of applications.

Additional Information and Photographs

Above is a photo of the audio filter in action on the NDB band. The blue LED reflection can be seen on the receiver. For serious NDB and CW pile up work, narrow band pass audio filters are generally required. This simple audio filter experiment might be useful as a spring board for your own AF filter experiments and to learn the filter requirements of your own particular receiver.

Above is the filter photographed from the rear. The DC power cord has a built in RFI filter. For homebuilt projects, DC power cords can be obtain by cutting the power cord off old unwanted or broken "wall wart" power supplies. This provides you with a nice cord with a built-in plug. I collect old AC "wall wart" transformers for this purpose. This filter is powered by the main 12 volt DC supply on my radio bench.

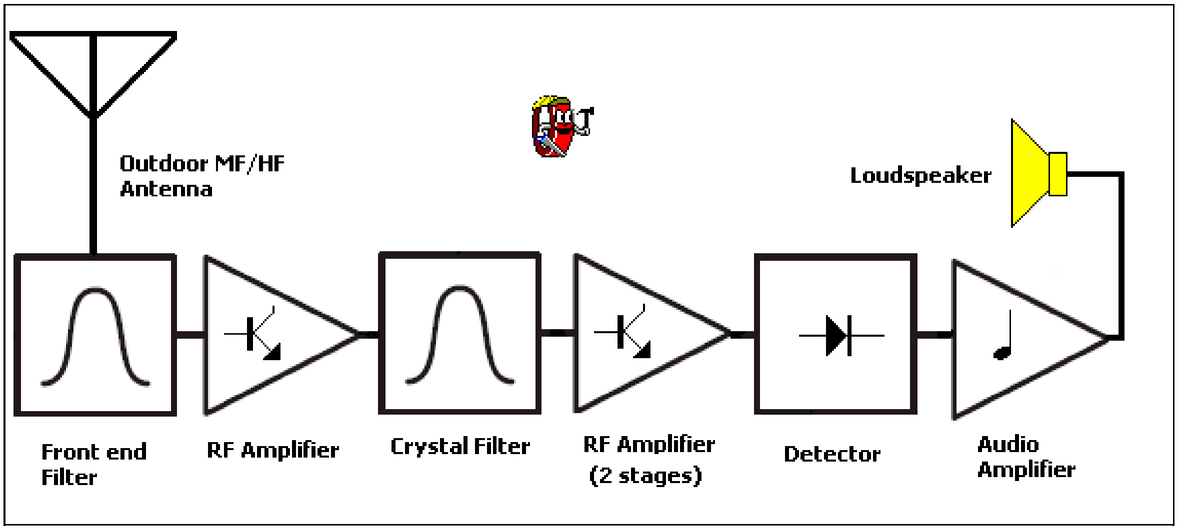
Cascode Hybrid-Based WWV Receiver for 5 MHz

Introduction

This was my favorite project of 2007. When I web published the original TRF WWV receiver for 10 MHz in 2006, there were many complaints that I used hard-to-find dual-gate MOSFETs and also that the AF stage lacked the popcorn factor that this web site has become strongly associated with. In this experimental project, these 2 concerns are addressed.

The cascode JFET and BJT amplifier stage used in this receiver is based upon the amplifer described in the Hybrid Cascode IF Amplifier article which was published in QST for December 2007 and designed by W7ZOI and WA7MLH. This amplifier topology has many advantages including high gain + low noise, that it can function well at DC voltages less than 12 VDC and that the noise figure does not degrade when the BJT bias (and stage gain) is lowered during AGC action. Please read the QST article and also refer to the W7ZOI web site for more details on the IF amplifier and the cascode hybrid topology.

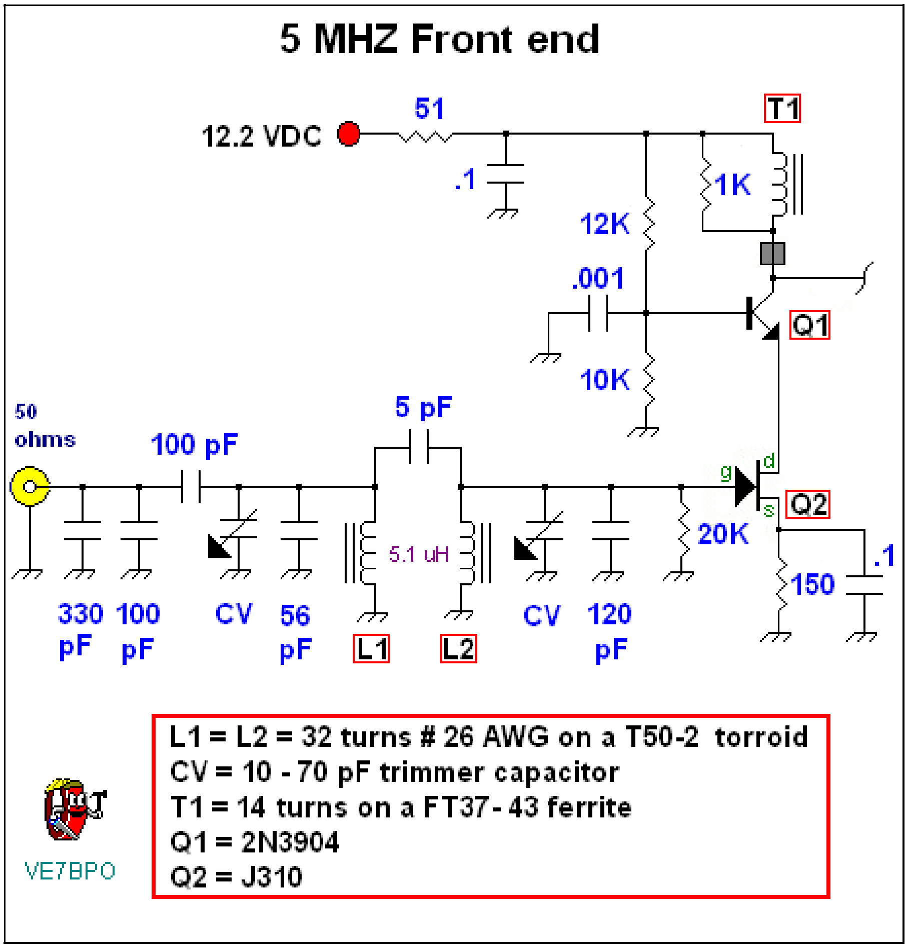
Receiver Block Diagram



The receiver block diagram is shown above. To hear digitally recorded examples of this receiver, click here , here or here. No attempt was made to make these files sound better than they really are- there is signal fading, room noise etc. A electret microphone was placed near the loud speaker to record these audio samples. *Note I am now compressing audio files in the mp3 format to allow listening by those who use Linux as their operating system.* **A supplemental web page** to this main web page is linked here

---

Receiver Front End: Band Pass Filter and First RF Amplifier



This receiver is meant to interface with a standard 50 ohm feed line. Testing was performed using my MF/HF antenna.

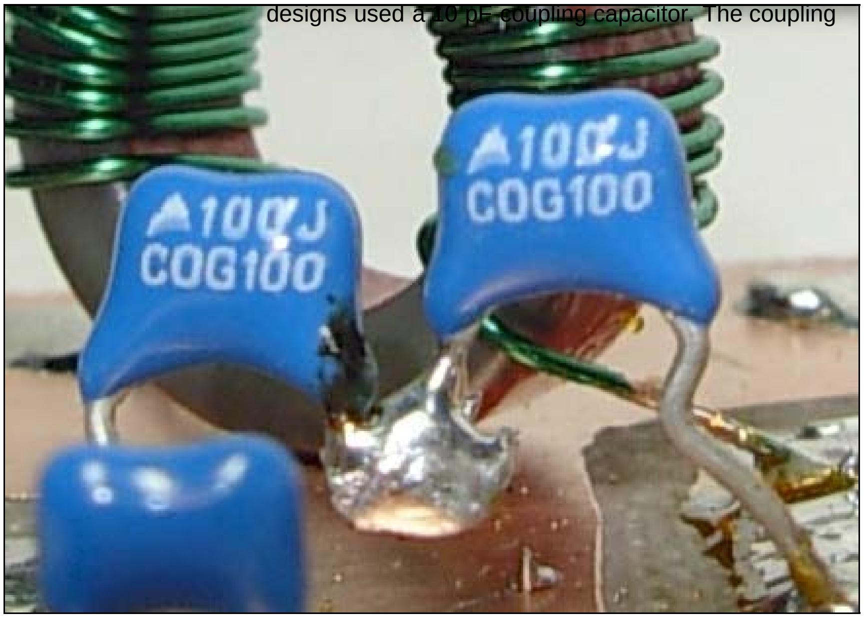
I built 3 separate bread boards of this receiver and tried varying the number of RF amps, using different detectors (as well as different detector followers) and eventually built and tested this basic receiver design for 5, 6 and 10 MHz. With respect to using the cascode hybrid amp (and probably any other amplifier type) **in a TRF receiver**, I learned 3 things:

1. Do not operate the RF amps at maximum gain. I built some very powerful amps with a Q2 source resistor of 47 ohms and over 6 volts bias on Q1. While powerful, this amp broke into oscillation and also consumed much current (nearly 20 ma).

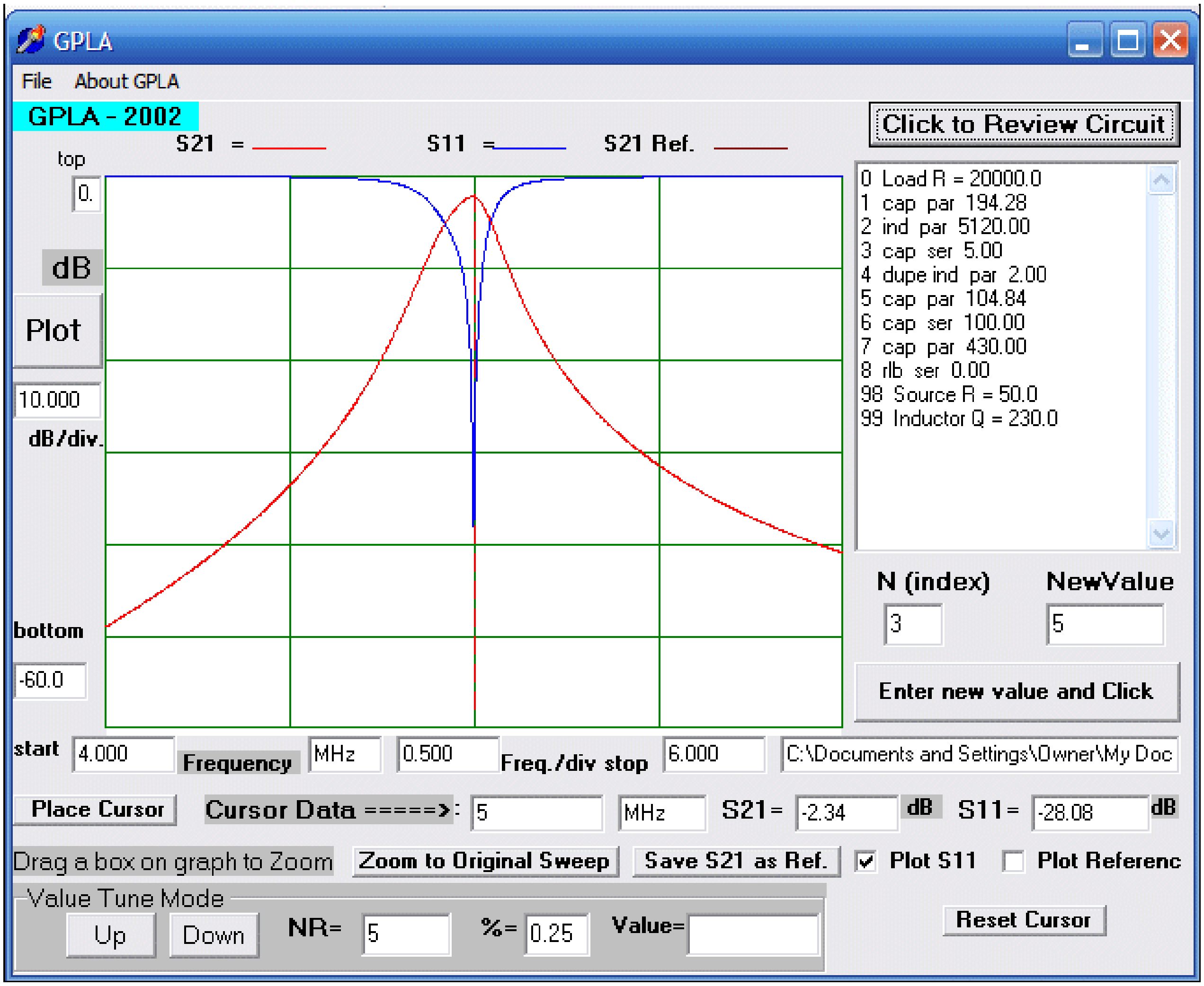
2. Keep the RF stages at least 2-3 cm apart to reduce the chance of parasitic oscillations.

3. Keep the input band pass filter at least 2 cm from the Q1/Q2 amp or you might encounter some unwanted oscillations.

For the front end band pass filter, a reasonably narrow bandwidth was desired. When sweeping early filter designs using a signal generator and oscilloscope, a double humped response was noted. These filter designs used a 10 pF coupling capacitor. The coupling capacitor was then decreased to 5 pF. To obtain the required 5 pF, two 10 pF capacitors were placed in series as shown in the photograph directly to the right.



I struggled with this filter design because one end is terminated in the gate resistance of Q2 of the hybrid cascode amplifier and was not the standard 50 ohm impedance termination. My early filter designs suffered severe insertion loss or poor selectivity. I asked Wes, W7ZOI, for some instruction on solving my filter problems. I learned that this filter topology is referred to as a singly terminated, double tuned band pass filter. Wes designed the front end band pass filter for the 5 MHz receiver for us all to learn from and for this I am very grateful to him.



```
0  Load R = 20000.0
1  cap  par  194.28
2  ind  par  5120.00
3  cap  ser  5.00
4  dupe ind  par  2.00
5  cap  par  104.84
6  cap  ser  100.00
7  cap  par  430.00
8  rlb  ser  0.00
98  Source R = 50.0
99  Inductor Q = 230.0
```

top: 0.

10.000 dB/div.

bottom: -60.0

N (index): 3

NewValue: 5

start: 4.000 Frequency MHz

0.500 Freq./div

stop: 6.000

C:\Documents and Settings\Owner\My Doc

Cursor Data =====>: 5 MHz

S21= -2.34 dB

S11= -28.08 dB

☑ Plot S11

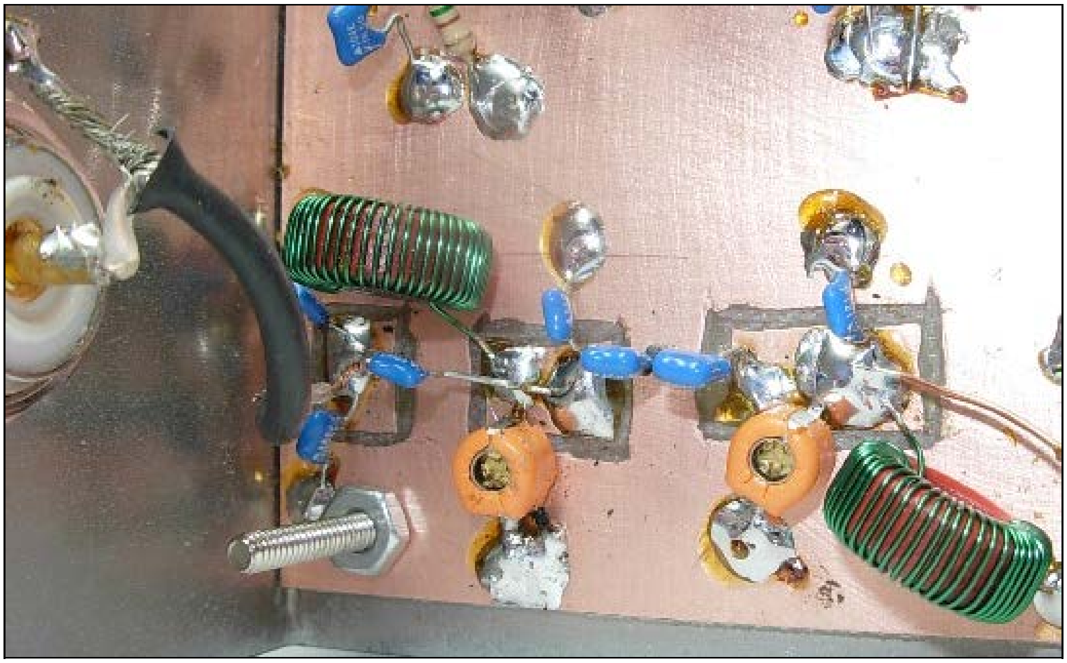
☐ Plot Referenc

NR= 5

%= 0.25

Value=

Above. A GPLA simulation of the singly terminated, double tuned filter designed by W7ZOI. A double tuned circuit is mandatory ahead of the WWV receiver as local BCB and other RF energy will be amplified by the first RF amp and may distort the WWV signal in the crystal filter or even might blow-by the crystal filter and be detected and heard in the speaker.

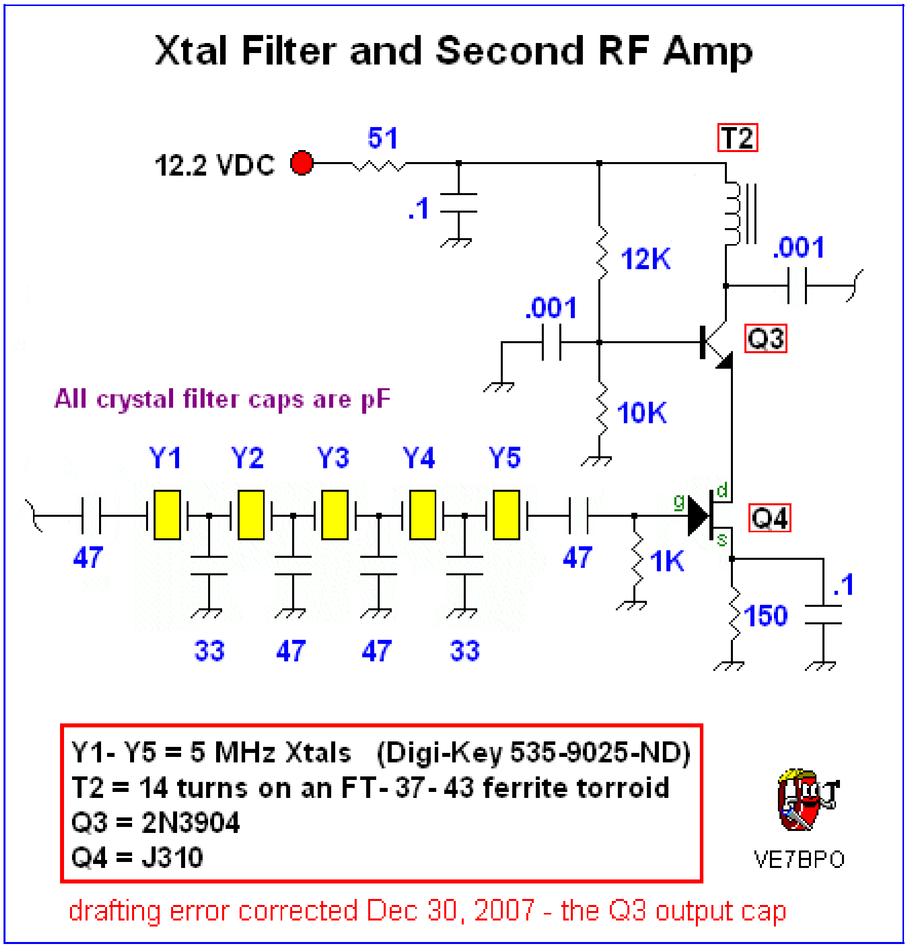


Directly above is a close up photograph of the input filter bread board. Filter tuning was done by ear (and screwdriver!) Simply tune the trimmer capacitors for the loudest audible WWV pulses in the speaker and you are set. If you can't locate a 20K gate resistor for Q2, a 22K resistor will work okay.

Crystal Filter and Second RF Amplifier Stage

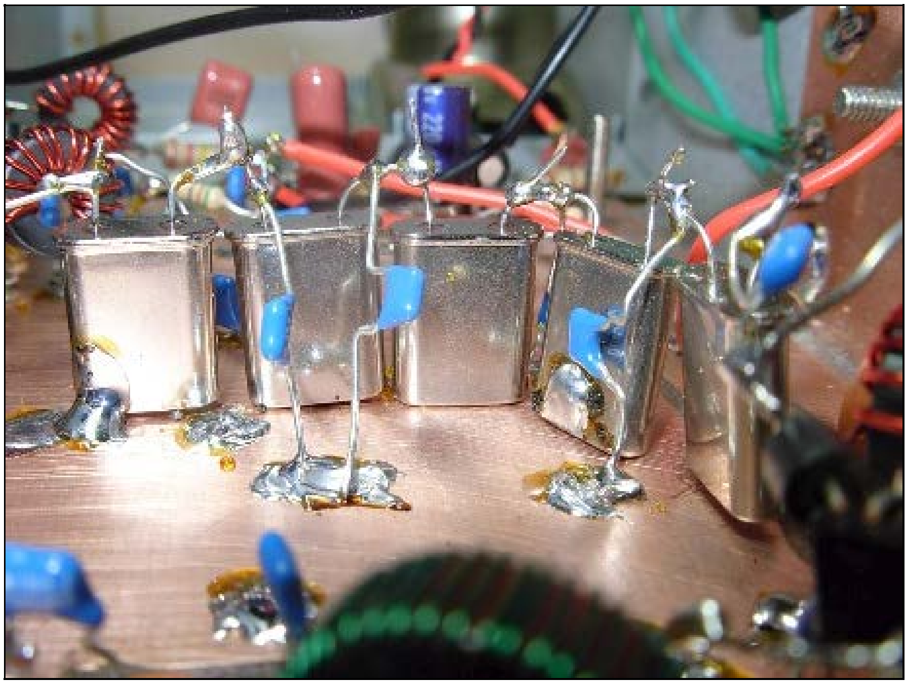
In the schematic to the right is the crystal filter and second RF amplifier. The input impedance of the crystal filter is established by the 1K shunt resistor across the output transformer on Q1. The output impedance of the crystal filter is set by the 1K gate resistor of Q4. A filter input/output Z of 1000 ohms gave the best overall shape and bandwidth during my testing.

Developing this filter was difficult. My first batch of junk box crystals had a low motional inductance and with the filter I built I could hear stations ~400 KHz below and/or above the filter center frequency in addition to WWV. After giving up in frustration for nearly 2 months, a batch of 10 crystals were ordered from Digi-Key. These were microprocessor crystals; ones with 18 pF load capacitance in a HC49/U holder. The new filter was tweaked and tested and now provides single signal reception of WWV. Your own results may vary according to your crystal parameters. The Digi-Key part number is provided for reference purposes only.

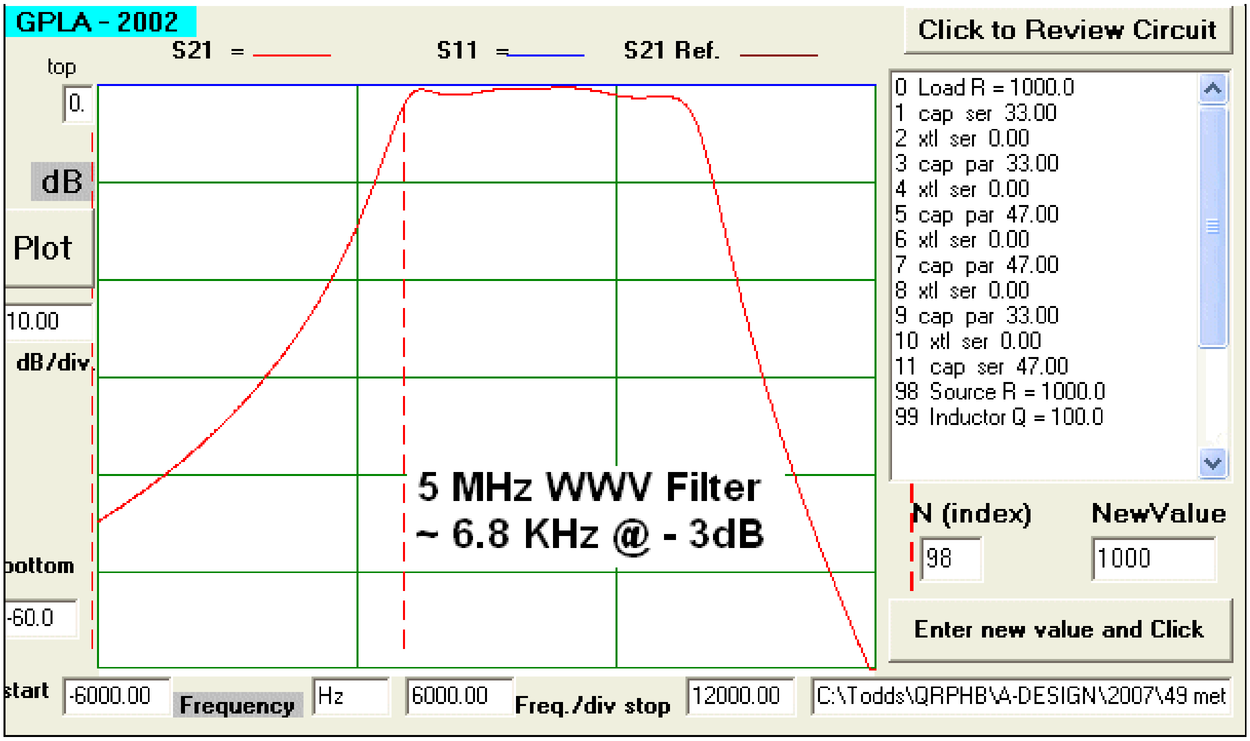

Xtal Filter and Second RF Amp

Y1- Y5 = 5 MHz Xtals (Digi-Key 535-9025-ND)
T2 = 14 turns on an FT- 37- 43 ferrite torroid
Q3 = 2N3904
Q4 = J310

drafting error corrected Dec 30, 2007 - the Q3 output cap



To the left is a close up photograph of the 5 MHz crystal filter. The crystals were turned upside down and the outer cases were directly soldered to the copper ground plane as you can easily see in the crystal to the left of the others. The rest of the crystals as well as one of the 47 pF tuning capacitors were soldered on the other side and solder points are hidden from view. The crystals were positioned to keep the output of Q1 away from the input of Q4. ***Stage lay out is very important in TRF receivers.*** I found stage layout to be far more important than keeping lead lengths short from my experimentation.



```
0  Load R = 1000.0
1  cap  ser  33.00
2  xtl  ser  0.00
3  cap  par  33.00
4  xtl  ser  0.00
5  cap  par  47.00
6  xtl  ser  0.00
7  cap  par  47.00
8  xtl  ser  0.00
9  cap  par  33.00
10  xtl  ser  0.00
11  cap  ser  47.00
98  Source R = 1000.0
99  Inductor Q = 100.0
```

top 0.

10.00 dB/div

bottom -60.0

tart -6000.00 Frequency Hz 6000.00 Freq./div stop 12000.00 C:\Todds\QRPHB\A-DESIGN\2007\49 met

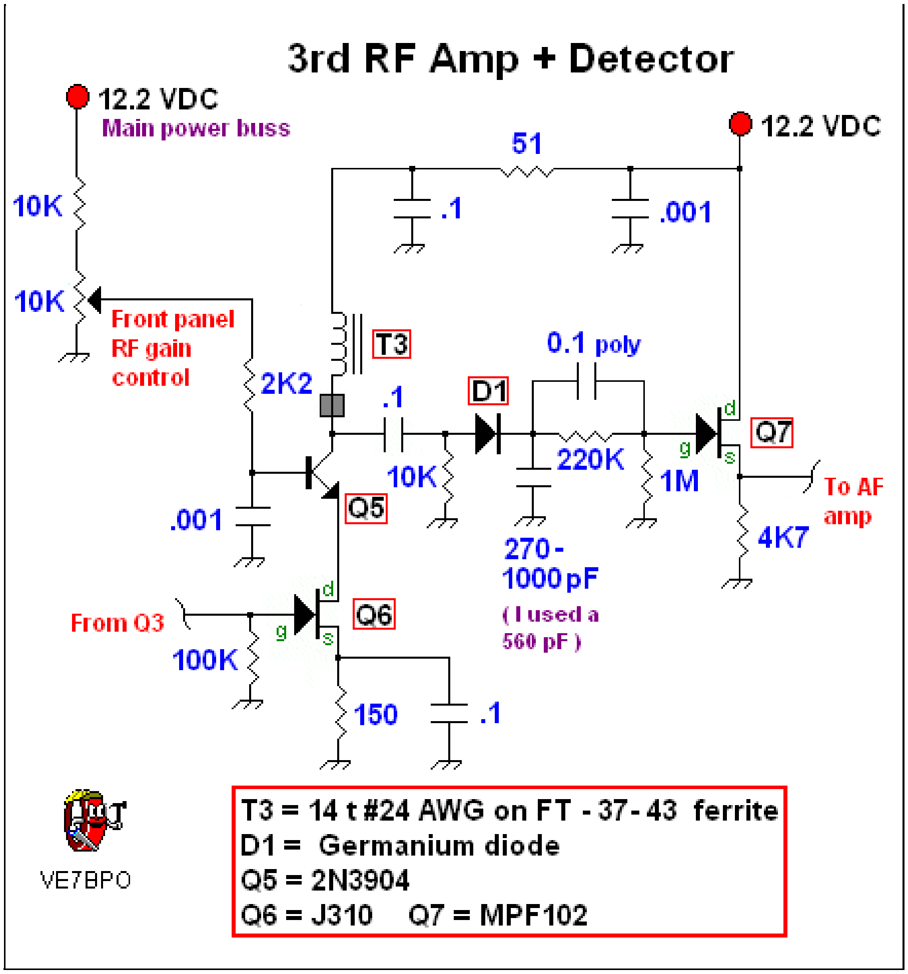
N (index) 98

NewValue 1000

Directly above is the GPLA simulation of my crystal filter. The 5 MHz point is not centered exactly in the middle of the pass band, but a reasonable AM filter was built nonetheless. Crystal parameters, especially motional inductance and capacitance can make or break your filter. Motional inductance and capacitance describe the L and C values that make up the crystal's electrical LC model. Very large inductive and capacitive reactance values at the specified operating frequency give the crystal its extraordinarily high "quality factor" or "Q". For example, If the motional L is too low, your filter may not work as expected; providing single signal reception of WWV. The Lm was 0.02 and the Cp was 5 in the crystals which I used for my filter. In general, low Q crystals will give poor results. Oppositely, crystals with very high Q may give a lower then expected bandwidth and this may reduce AM receive fidelity. Experimentation is necessary.

Third RF Amplifier Stage and Detector

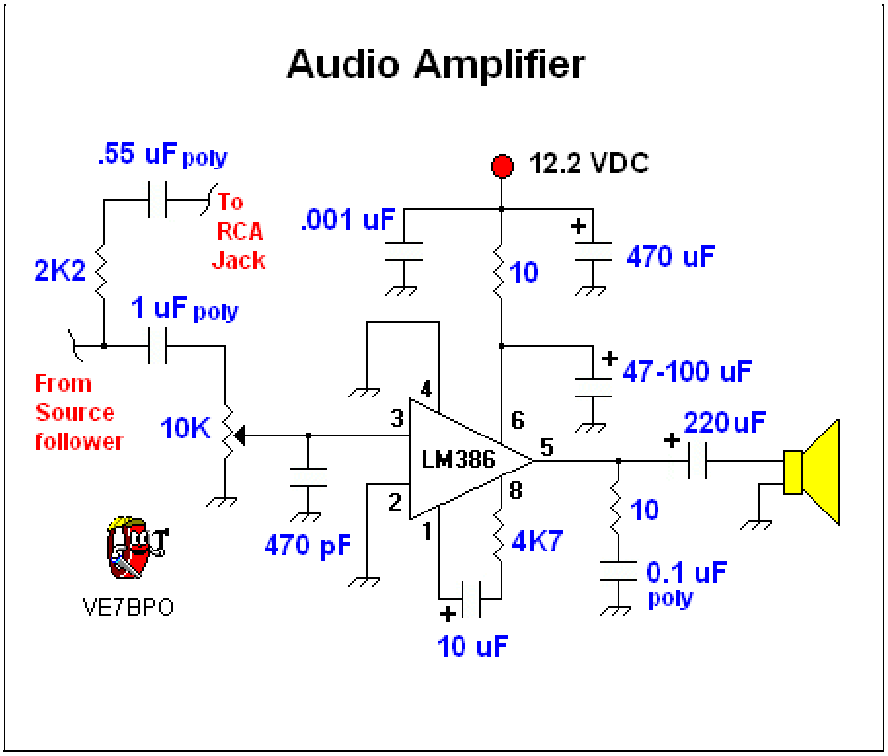
To the left is schematic of the final RF stage and the envelope detector. This RF stage has variable gain by means of a front-panel mounted 10K potentiometer which is used to vary the bias on Q5. The input Z of this stage is 100 K and is set by the Q6 gate resistor. The output of Q5 is AC coupled to a detector designed by Wes, W7ZOI. I performed considerable experimentation with basic diode detectors as well as detector source followers; some of which I sent to Wes for his consideration. He designed and emailed me back this simple, good sounding detector design which uses the gate voltage of Q7 to bias the germanium diode. Other types of diodes such as as hot carrier diodes will likely not have the output voltage of the Germanium type. Germanium diodes, when biased, had more noise and high frequency response in addition to higher output when compared to others I tried during my experiments. Diode detector guru, Felix, VK4FUQ advised me of an excellent diode he is now using called the BAT46. The audio samples of a local AM radio station using this diode and his other hi-fi lab equipment that he sent me are beyond fantastic.



The photograph on the right is a close up of the enveloped detector designed by W7ZOI. The germanium diode was purchased from The Source in Canada (Radio Shack in the USA). The blue, partially hidden shunt capacitor is a multi-layer ceramic 560 pF cap. The other capacitors are metalized, polyester film types. Ensure correct diode polarity.

Audio Stage

To the left is the schematic of the audio stage. The very "popcorn" LM386 AF chip is used to please the audience who complained about my AF stages not having enough popcorn factor. A 4K7 resistor was inserted between pins 1 and 8 to reduce the gain somewhat. Thus, the LM386 is still being operated in the high gain mode but won't hurt your ears with loud noise and distortion. The 470 pF cap on pin 3 may be changed or eliminated. It is a simple low pass filter.



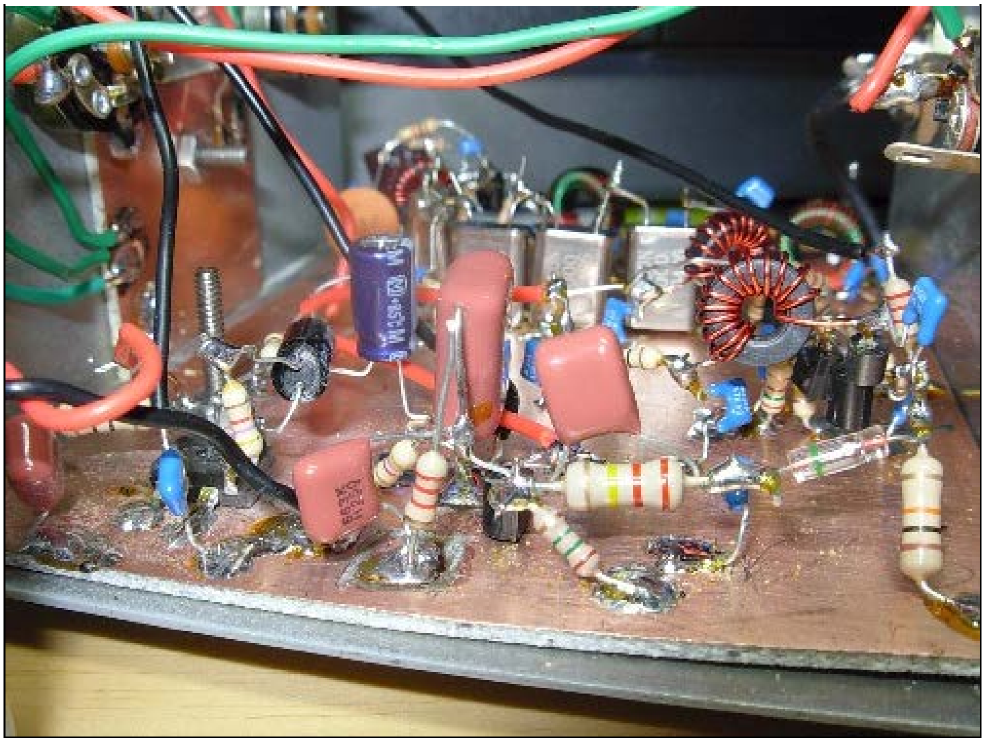
A secondary, audio output connects to a front panel mounted RCA phono jack. This allows me to use my lab grade (KK7B AF amp) and turn the audio off on the normal receiver AF amp.

The photograph to the right is a close up of the LM386-based audio stage. This is where I started. After drilling the chassis, wiring the speaker, installing the chassis potentiometers, making the main power buss and LED indicator, the AF stage was built on the main board. The main board was then temporarily soldered in and tested. (Some connections were made via alligator clips such as the speaker wires). When the AF amp worked as expected, the main board was removed from the receiver chassis and the net stage was built. Up next were the detector and source follower. After bread boarding these, again the main board was laid in the chassis, wired, tested and then removed when all was functioning well. To test the detector I touched my finger to the input and heard local BCB radio. Following this, RF Amp #3 was added to the main board and again the main board was temporarily wired up and tested by touching the input of Q6 with my finger and observing that a local broadcast radio station increased/decreased in amplitude when the RF gain control was turned up and down. DC voltages were also measured and considered from project start to finish.

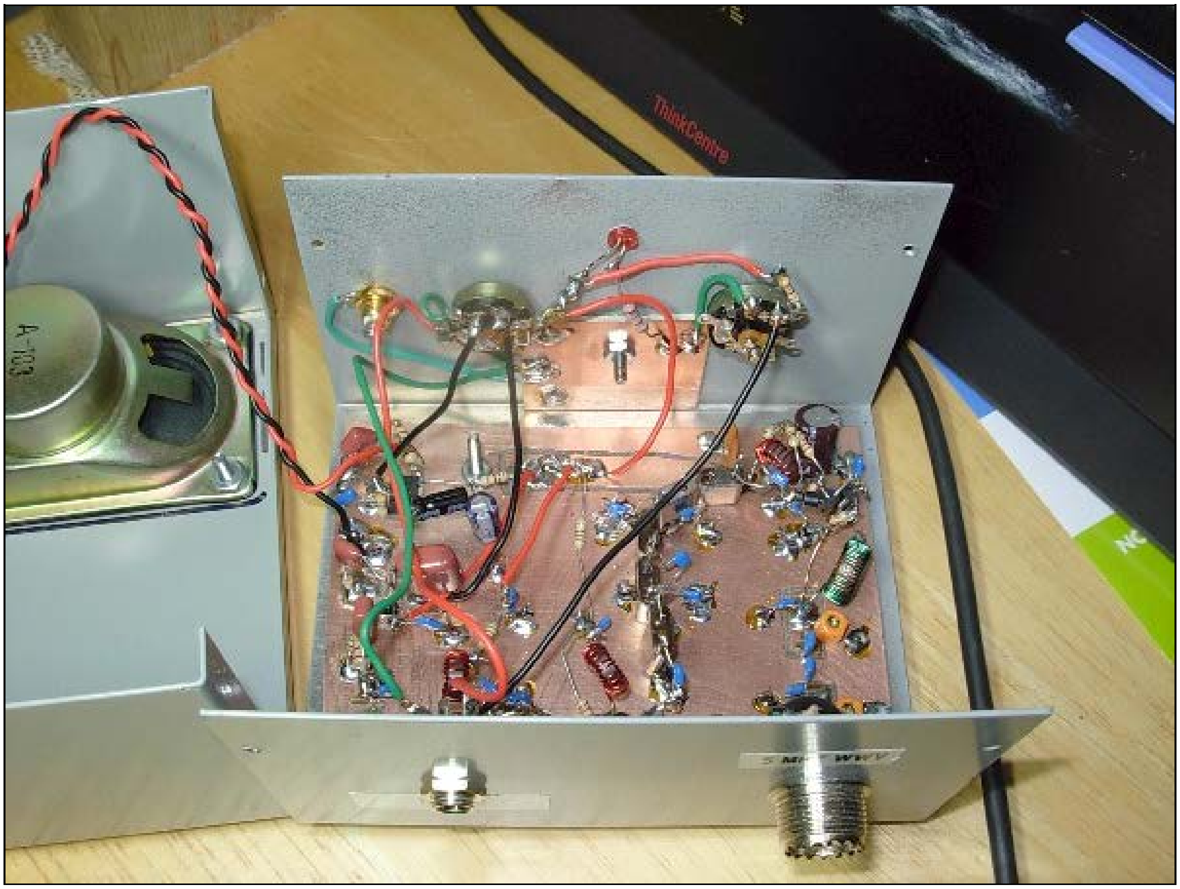
Actually, all you need to do is connect a band pass filter such as this to Q6 and the components after and you will have a nice TRF BCB AM radio. Each successive stage was built and tested, so when the receiver was finished, I already knew that it worked. I cannot emphasize enough how important it is to build your receiver backwards and test each stage as you go. There is strong temptation to start at the antenna connection and work until you get to the speaker, but please consider doing the opposite.

The bare copper wire in the photograph is the positive connection point for the speaker wire. It was trimmed somewhat during final assembly to reduce the possibility of it shorting.

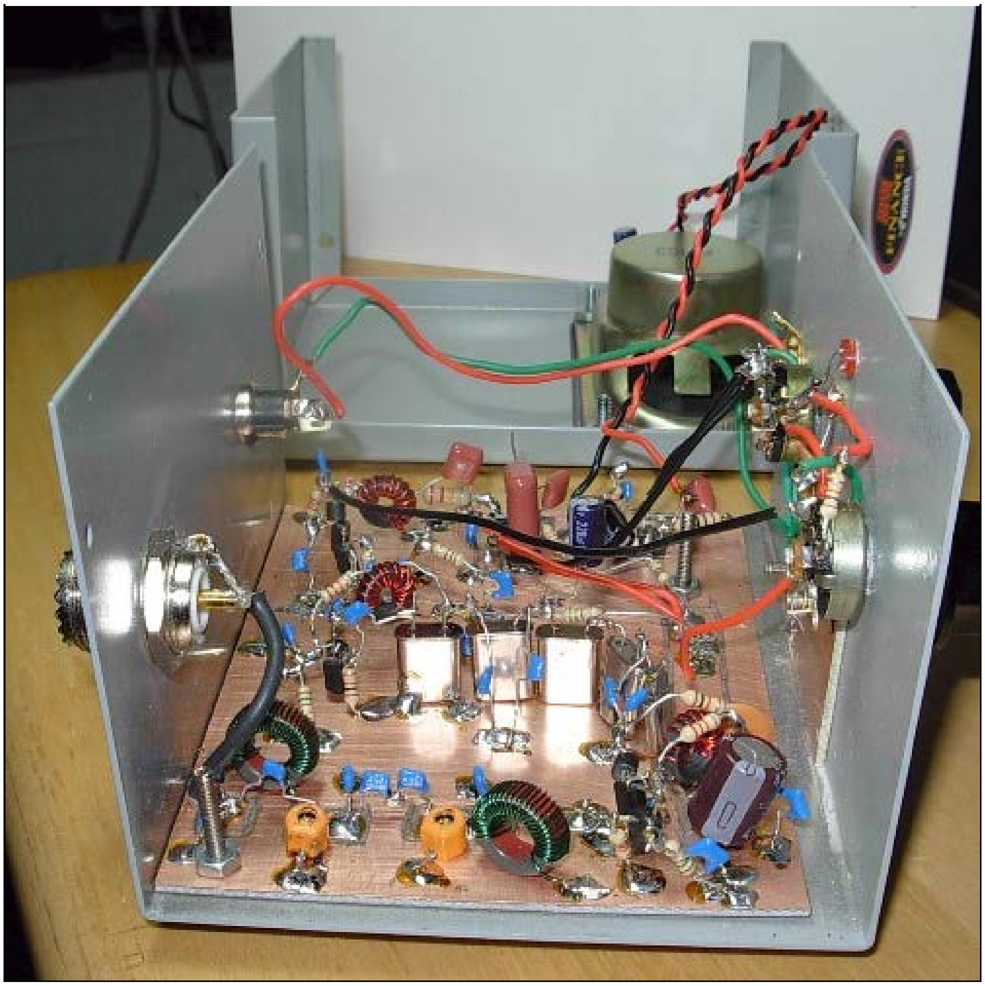
Miscellaneous Photographs



The photograph above shows some of the detail of the receiver main board from the right hand side which contains the detector, source follower and audio amp stages from right to left.



The photograph above shows a top view of the main chassis and also the chassis cover with the speaker bolted on and wired up.



This wider angle photograph shows the main board from the left side. From left to right in the nearground are the SO-239 antenna connector, LC band pass filter and first RF amplifier.

The photograph above shows the speaker attached to the Hammond chassis top. Holes were drilled in the chassis lid with a drill press to allow the sound to pass through.

The photograph above shows the reverse view of the receiver chassis.

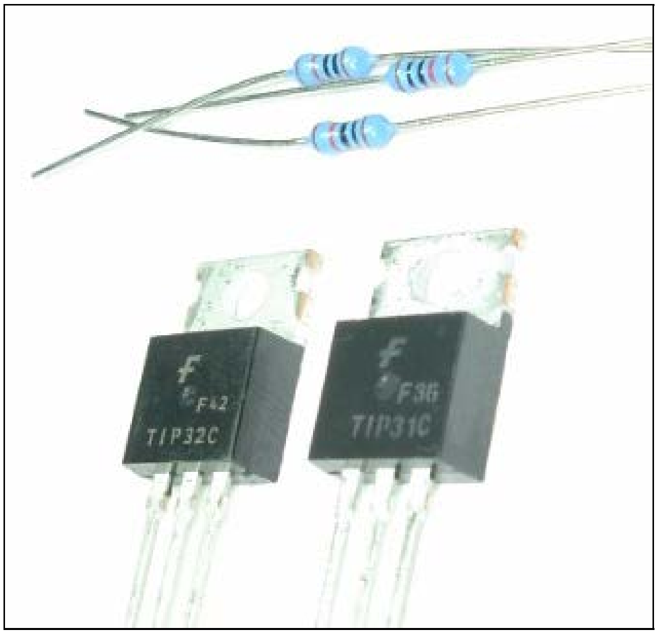
RF — Test and Measurement

Complementary-Symmetry Amplifier Biasing Basics

Introduction

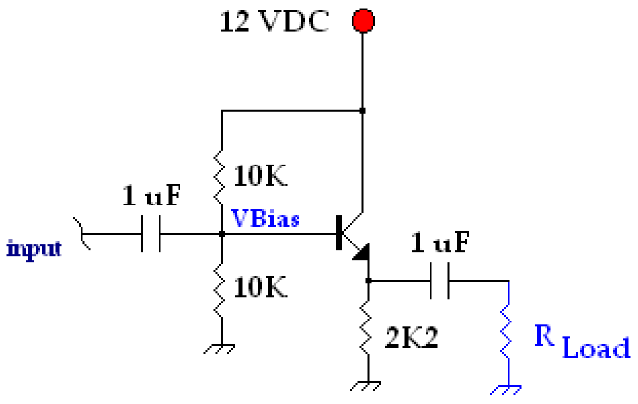
This page provides information concerning the biasing of Class-AB, complementary-symmetry audio amplifiers. These schematics should be considered theoretical, as design considerations such as thermal stability, negative feedback and component power ratings are minimized or excluded for sake of clarity.

The basic 2 transistor complementary-symmetry amplifier may be used as a simple low power AF amp or as a building block for a high powered stage such as a 50 watt guitar amplifier. It is important to understand how to properly bias your AF power amps to reduce distortion and to promote easy troubleshooting when problems arise. This web page describes the hows and whys of biasing in a progressive manner with minimal math.



Discussion

Figure 1

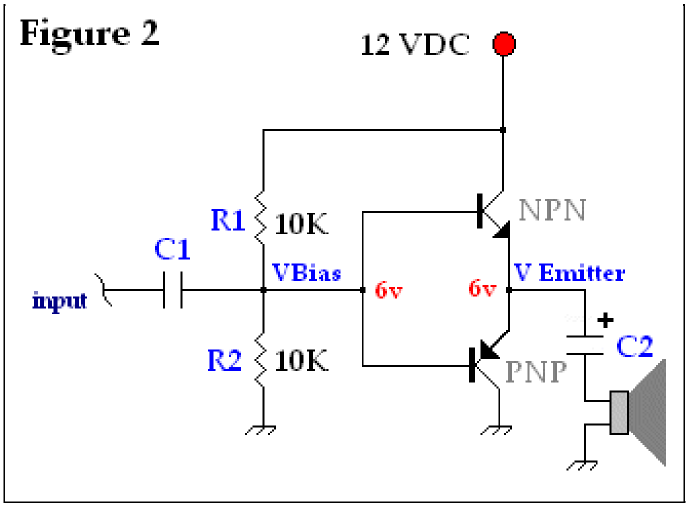


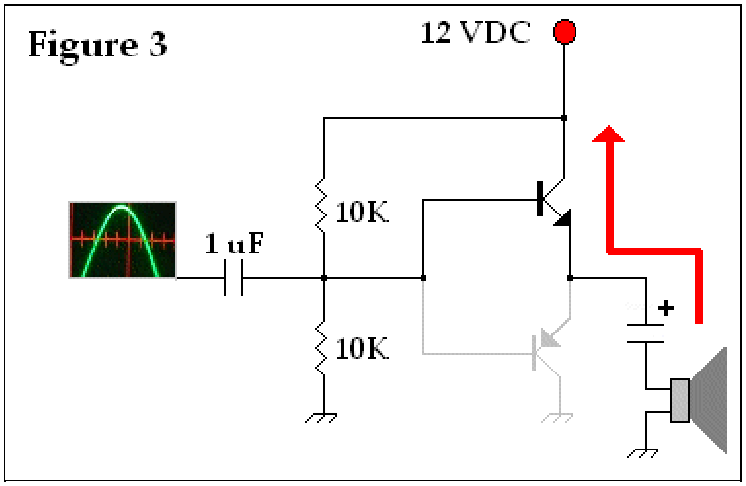
A review of the common collector amplifier (which is more commonly called the emitter follower) is a good place to start. We may refer to the complementary-symmetry transistor pair as complementary emitter followers since they are an NPN and PNP emitter follower connected in series. An emitter follower amp is shown in Figure 1. Its properties include:

- ✓ input on the base - output on the emitter
- ✓ high input impedance and low output impedance
- ✓ a voltage gain of 1
- ✓ good current and power gain

In an appropriate configuration, these qualities are perfect for driving a low impedance load such as an 8 ohm speaker with large output currents that are not provided by our typical transistor or op amp voltage amplifier stages. In many cases, we bias the emitter follower with a voltage divider network comprised of 2 identical value resistors. In Figure 1, the voltage divider consists of a series pair of 10K resistors and thus VBias = 6 volts. These 10K bias resistors will be used throughout this web page as the circuits evolve.

In Figure 2 is a pair of complementary emitter followers which have their bases biased with our now familiar series connected 10K bias network for a VBias of 6 volts. When the power is turned on, output capacitor C2 charges through the NPN transistor until it reaches about 6 volts *(theoretical value used to keep things simple)*. When the voltage at point V Emitter reaches 6 volts, the NPN transistor goes into cutoff because VBias voltage now equals the V emitter voltage. Recall that the NPN transistor base must be positive with respect to the emitter for current to flow. Both the NPN and the PNP transistor are in cutoff. This is the amplifier's quiescent state *(assuming no signal is applied to the input via C1)* and is called Class B bias.

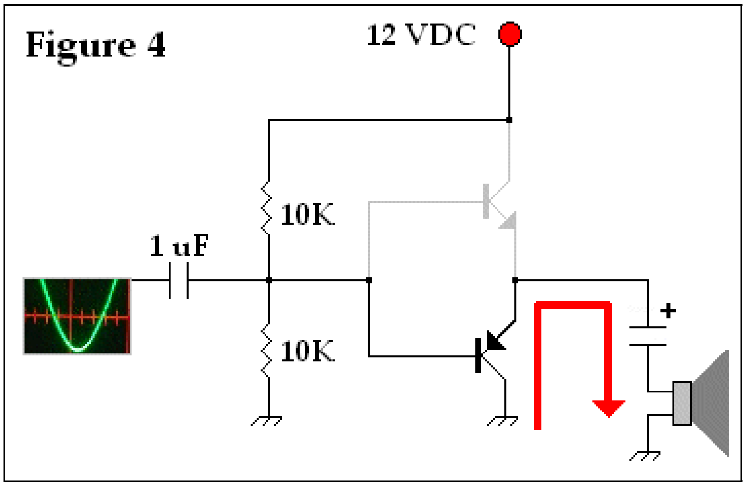


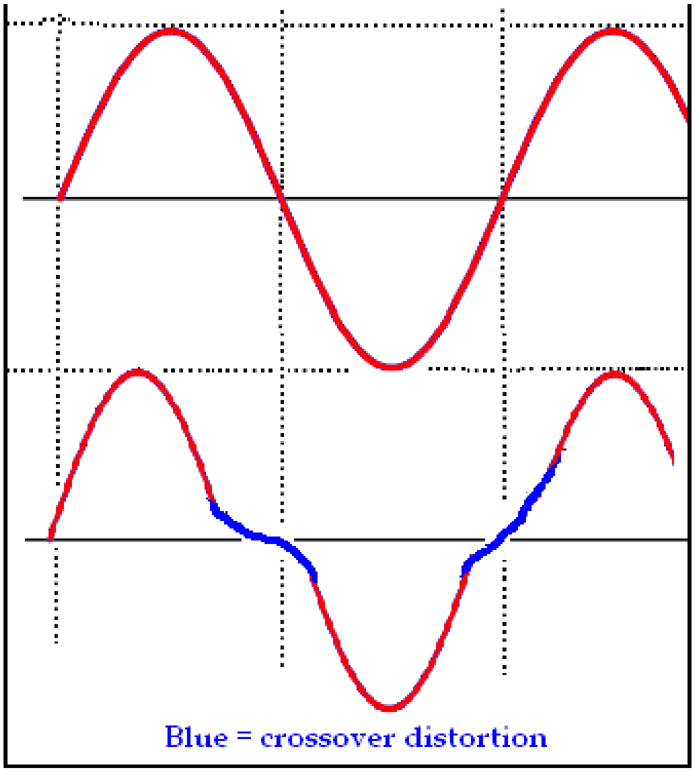


In Figure 3, a positive going signal is applied to the input capacitor. The NPN transistor becomes forward biased and turns ON. Current flows **through the NPN transistor** and charges capacitor C2 to a higher potential. The PNP transistor stays in cut off. The NPN transistor is an emitter follower connected to the speaker.

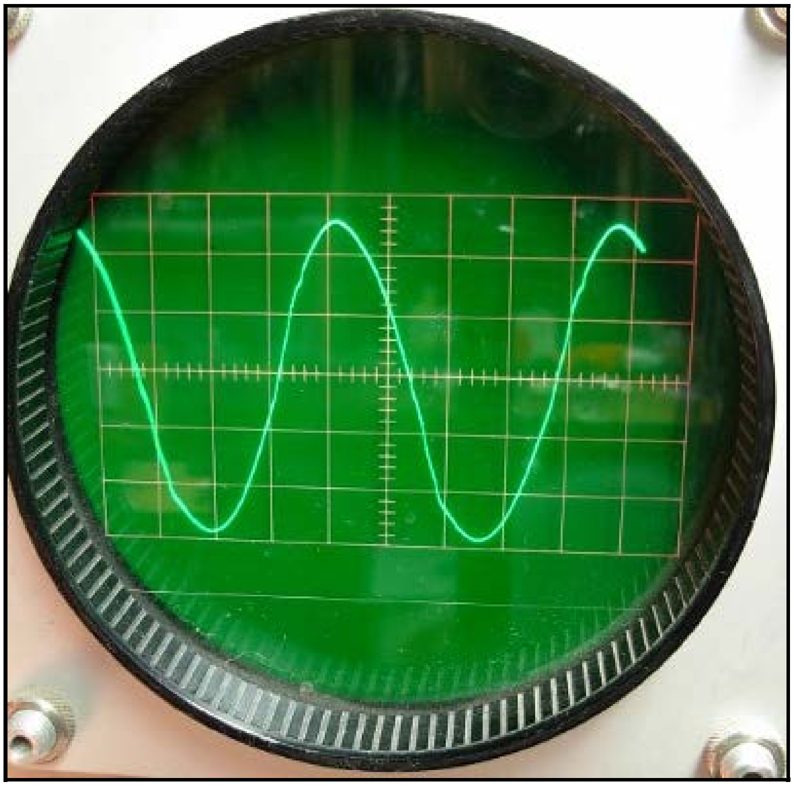
In Figure 4 a negative going signal the (negative half-cycle) is applied to the input. Q3 turns ON and discharges the output capacitor through the speaker as shown in **red.** The PNP transistor is an emitter follower connected to the speaker.

Thus the NPN and the PNP transistor conduct on alternate half cycles which causes AC current to flow through the speaker. The complementary emitter followers are said to be in push-pull operation.





The circuit of Figure 2 has a significant problem; output signal distortion. Silicon transistors such as the 2N3904 and 2N3906 will not conduct until their bases are forward biased by somewhere around 0.7 volts. For the NPN transistor, this means that it will not conduct until the input signal has gone positive by about 0.7 volts. Oppositely, the PNP transistor will remain in cut off until the input signal goes negative by approximately 0.7 volts. As a result, there is a dead zone during the point in time when one transistor cuts OFF and the other turns ON. Shown above is a normal sinusoidal AC waveform in red and another with the distorted waveform of Figure 2 in red and blue. This distortion is called crossover distortion because it occurs at the zero crossing point of the AC waveform. This introduces odd-order harmonics into the output signal. Such is the drawback of the Class B amplifier.



The above photograph shows crossover distortion in an under-biased power amp.

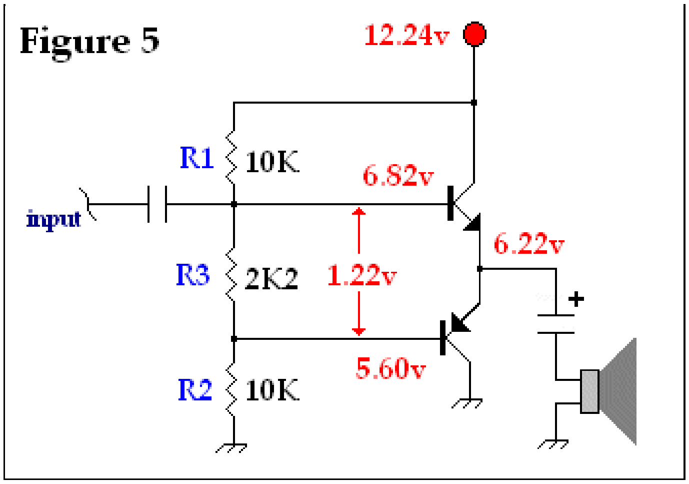


Figure 5

Figure 5 shows the principle technique used to reduce crossover distortion; both transistors are (slightly) forward biased almost to conduction in their quiescent state. As a result, any amplitude of positive or negative going signal will bias the appropriate transistor into conduction. An easy way to achieve this biasing is by adding 1 resistor to our 10K voltage divider network. In Figure 5 is a circuit I built, measured and listened to. R3, a 2K2 ohm resistor was placed in between R1 and R2, our usual 10K bias resistors. As a result , both transistors are forward biased.

That is: the base of the PNP transistor is negative with respect to its emitter and the the base of the NPN transistor is positive with respect to its emitter. As a rule of thumb, you need to drop at least 1 volt across R3. I chose a 2K2 resistor and it worked fine in my particular amp. The Figure 5 biasing topology is rarely used as it puts a series resistance on the PNP input among other problems; however, it exemplifies the basic principles of biasing our complimentary pair. With the forward bias on the transistor pair we now are in Class AB. The output capacitor serves to block the quiescent DC current from flowing through the speaker.

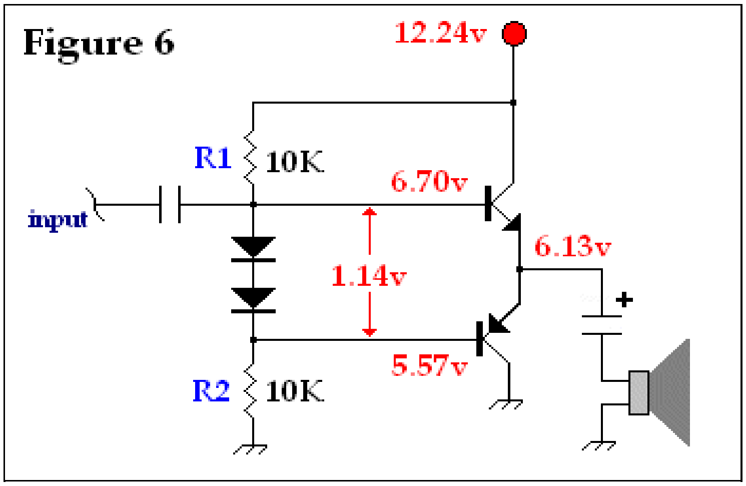


Figure 6

Figure 6 illustrates an improved biasing method over that of Figure 5 by using a pair of silicon diodes. You see this circuit used a lot by hobbyists. The voltage divider consists of 2 resistors and the 2 diodes. The 2 series connected diodes are connected in parallel to the NPN and PNP transistor base-emitter junctions which serves to keep the transistors turned on slightly. The net effect of the diode pair is the same as R3 in Figure 5. The voltage drop per diode was measured at 0.57 volts. The AC resistance of these 2 forward biased diodes is non-significant. There is major problem with the diode/resistor voltage divider; no way to adjust the diodes forward voltage drop. If each diode's forward threshold voltage is unequal to the base-emitter junction voltage of each transistor, either not enough forward bias is applied, or the 2 transistors may be turned on too much reducing efficiency and possibly cause excessive heating. Additionally, the pair of diodes lack the ability to provide temperature compensation when the transistors get hot.

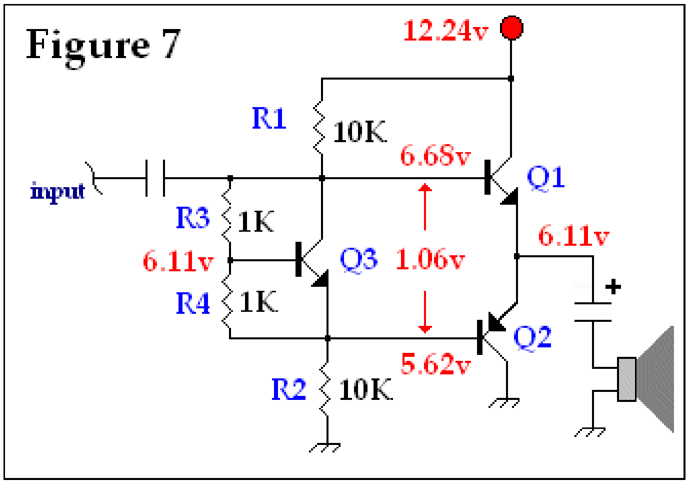
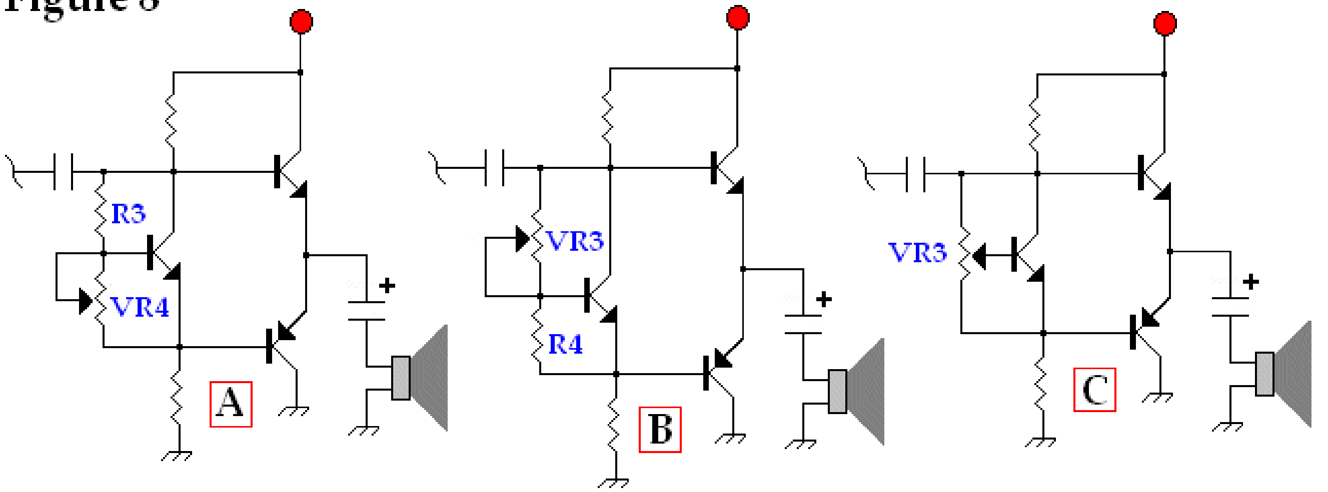


Figure 7

Figure 7 shows the best way to bias our complimentary pair. Our familiar 10K-10K voltage divider is kept, but a transistor Q3 with its own biasing resistors R3 and R4 are added. You might think of R3 and R4 as a voltage divider within a voltage divider. Q3 is referred to as an amplified diode or DC level shifter. It often receives local thermal feedback from the power follower output transistors. This usually involves mounting Q3 on the same heat sink as the finals. If the output transistors heat up, so does Q3 and this results in a a smaller voltage drop across Q3 which translates into less forward bias to Q1 and Q2. Within limits, Q3 with its own base-emitter junction provides variable forward bias for the output transistors.

Shown above is the breadboard of the Figure 6 circuit built on scrap of copper clad board. Transistors were 2N3904 and 2N3906 types, diodes were 1N4148. The capacitor and resistor to the right were a low pass filter (10 ohm and 0.1 uF) to stabilize the output. The unseen speaker was connected to the red and green wires.

Figure 8

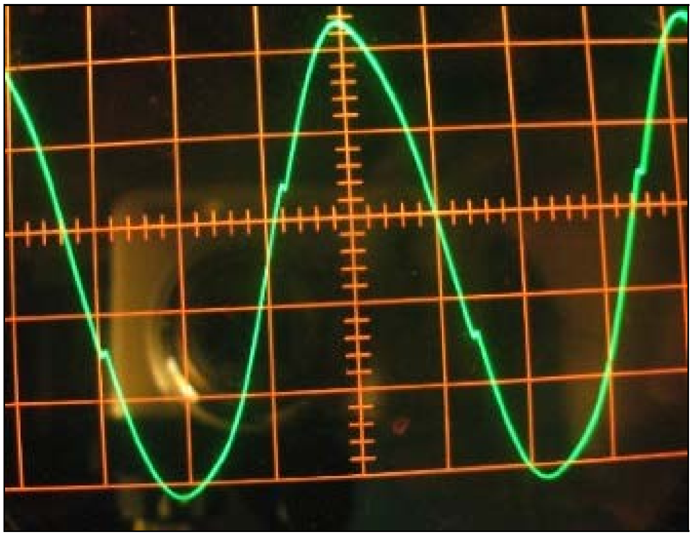


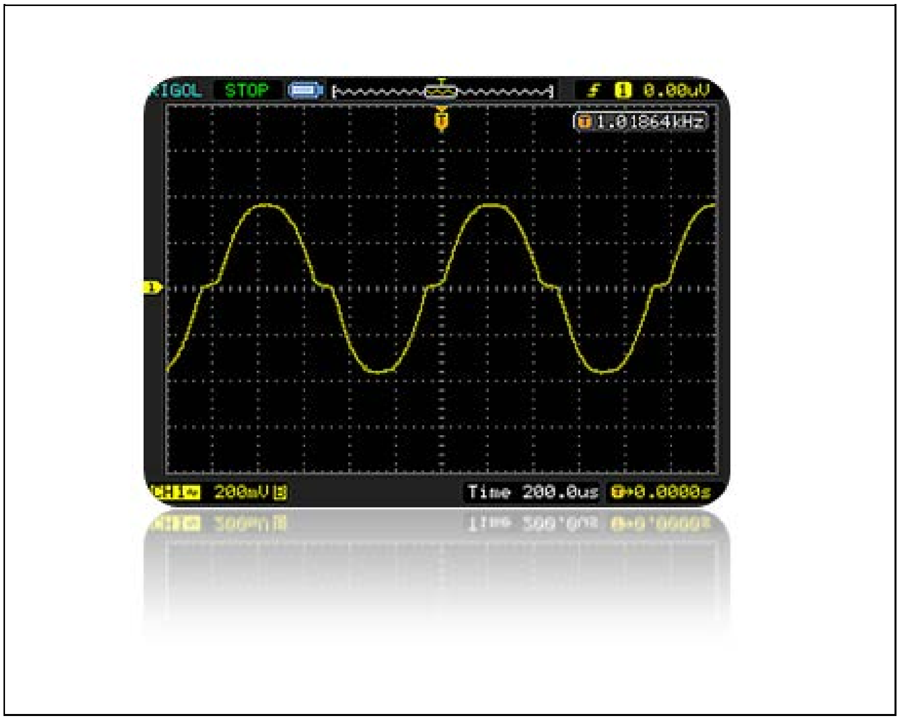
In practice, either R3 or R4 is often replaced with a trimmer potentiometer or a trimmer potentiometer is used instead of R3 and R4 and sometimes R3 and R4 are not of equal value. Shown above in Figure 8 are 3 variable bias topologies for Q3 that I have used. In some cases you will notice that the builder places a fixed value resistor or even a diode in series with the potentiometer in circuits like A or B. Using a potentiometer allows precise adjustment of the quiescent bias current and the ability to dial in the lowest crossover distortion possible. You can set the bias current using any combination of an oscilloscope and signal generator, a voltmeter, an ammeter or possibly try do it by ear when listening for and removing crossover distortion at low volume levels. The procedure I have read to adjust the bias by listening is as follows: Allow some low level signal through the amp so you can just hear it in the speaker. Turn the potentiometer from 1 extreme to another until crossover distortion is heard. Move the pot in the opposite direction until the crossover distortion disappears.
From my limited experience; in some amplifiers under 2 watts or so, you may not hear much of an audible change in crossover distortion when adjusting the bias control potentiometer, so the listening method is not useful in certain cases. It is worth mentioning, that crossover distortion sounds awful and you can usually hear it in amplifiers that are under biased.

Many builders just have a multimeter. In this case, measure the voltage drop across Q3 (the amplified diode) and ensure that is a least 1.1 volts and then slowly adjust the bias up or down from that point. Ultimately, you may have to just make the final bias setting by deciding what voltage drop across Q3 and/or what complementary pair quiescent current you want to establish. It is really not that difficult. *Whatever method you use,*

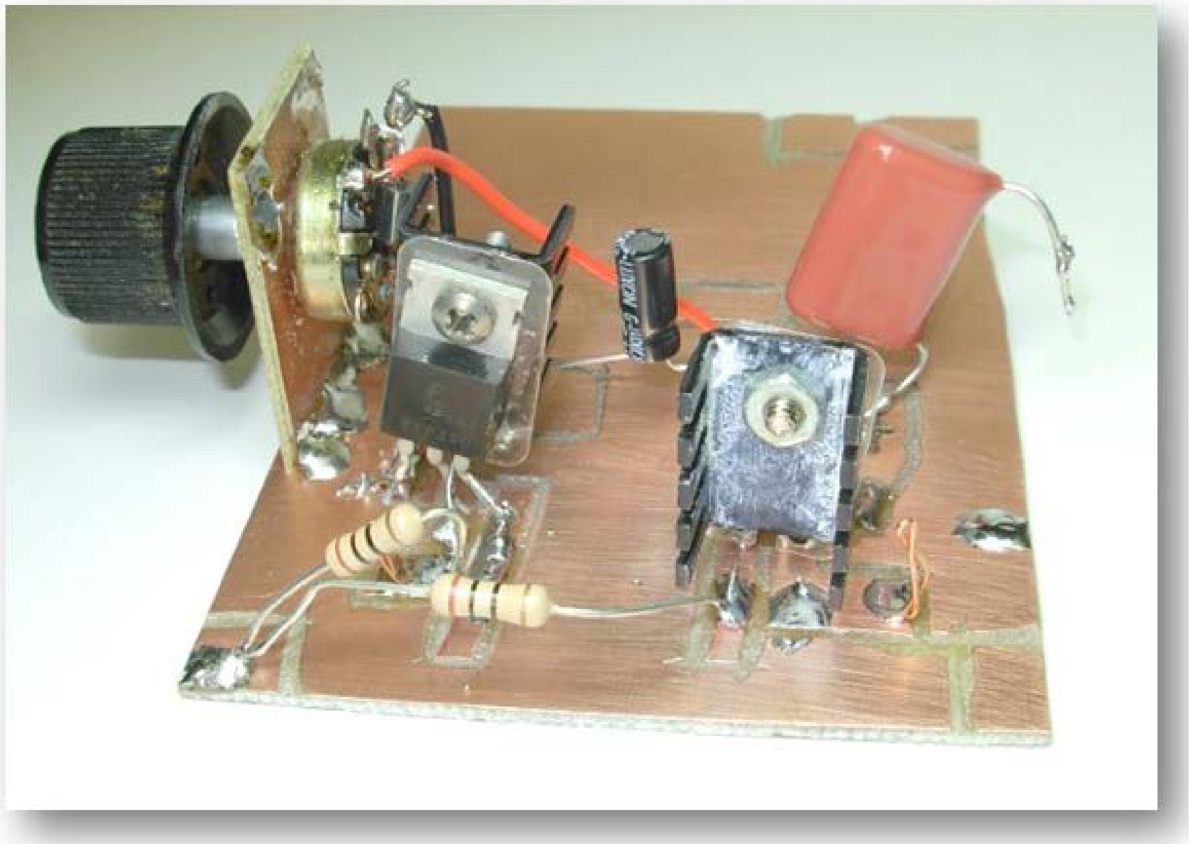
always re-check the Q3 voltage drop and amplifier bias current with no input signal to inform yourself of what is happening.
I am uncertain of the best method to measure the amplifier quiescent current, however I normally measure it using an ammeter connected in series with the emitter of the NPN transistor of the complementary pair.

Shown to the right and below are more images of what crossover distortion can look like in an under-biased power follower stage. The *right* output waveform also contains a little harmonic distortion, however that's a separate issue. I listened to this amp when connected to a speaker and music audio source; the audio had a noticeable "grungy" distorted sound. As mentioned, crossover distortion sounds terrible. The bias to the power followers (a complimentary pair of two 2N3906 and two 2N3904 transistors set up as Darlington emitter followers) was increased and the crossover distortion disappeared. A post bias adjustment audio listening test confirmed that the crossover distortion was gone.



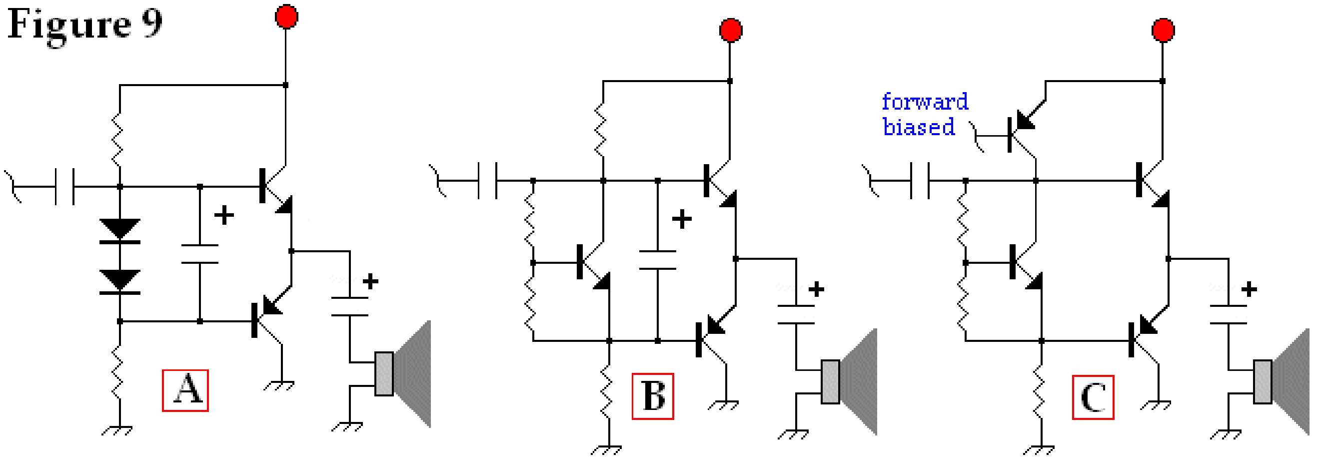


Click for a Russian language mp3 audio file. Vladimir (Volodya), a fellow builder in Ukraine, wanted an A - B comparison of Class AB versus Class B (cross-over distortion). In this audio file, I tweak a potentiometer biasing a pair of power followers to give contrast between the 2 amplifier classes. Under biased AF amps sound terrible in any language! The audio source was a cassette player. The speaker output was recorded, digitized and stuffed into the mp3 file.



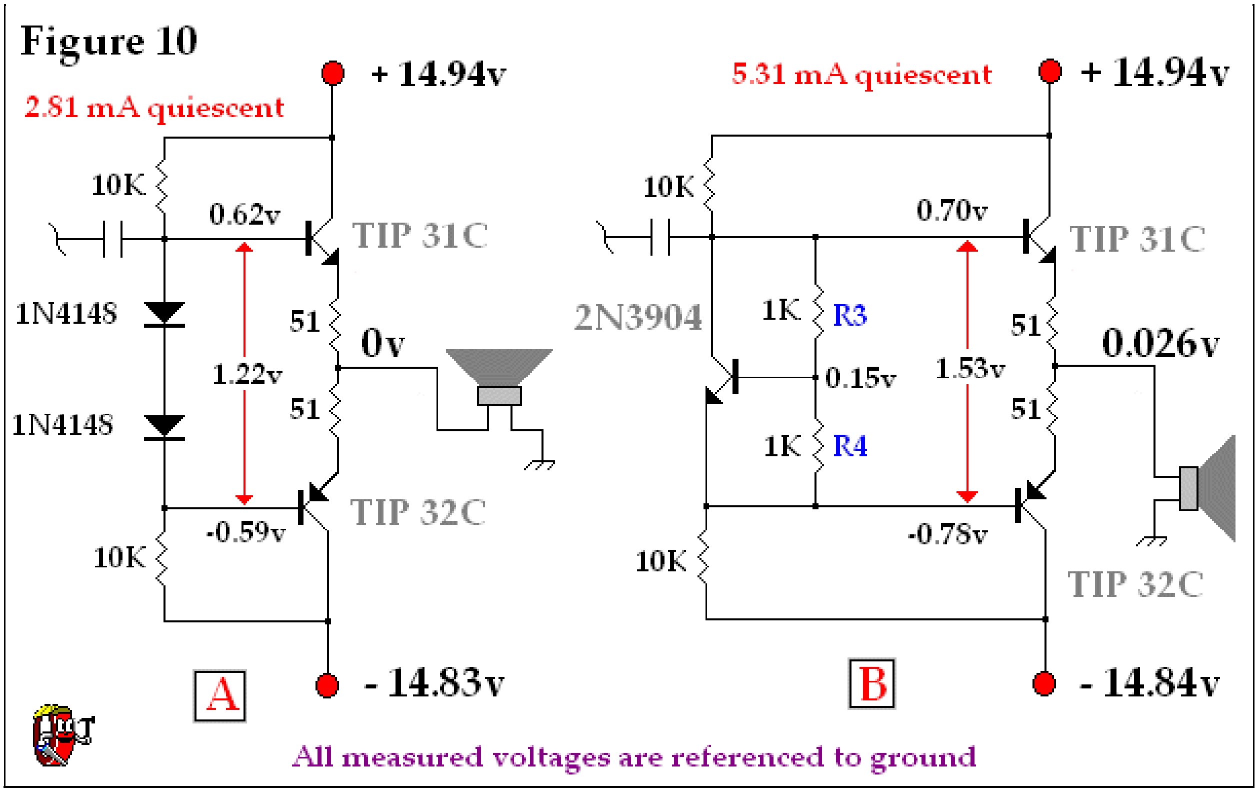
Shown above is a bread board of a complete amplifier utilizing a 10K pot to vary the bias on the amplified diode *(see Figure 8 c)*. With a 12.22 volt power supply, turning the potentiometer from one extreme to the other varied the current draw of the amplifier from about 0 to 95 mA. The average quiescent current draw of a properly biased **single complimentary pair** was somewhere between 5 and 10 mA in my bread boards.

Figure 9

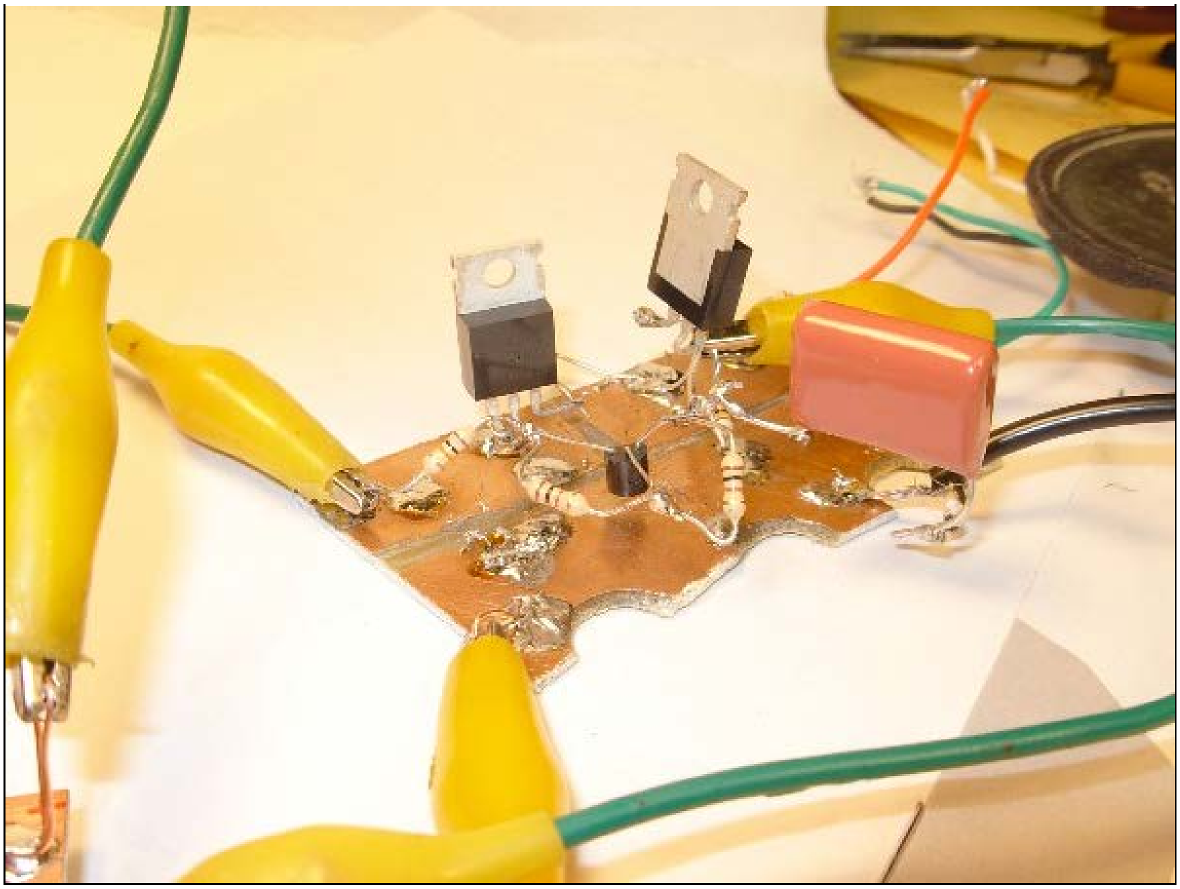


You may see a capacitor inserted between the output transistor bases as shown in Figure 9A and 9B. I have seen capacitor values from 4.7 uF to 100 uF used and the value is not critical, however, from my experiments, I have learned it is mandatory. This capacitor serves to keep the bias voltage constant as the AC signal swings up and down. Some engineers refer to the amplified diode an **NPN shifter bias amplifier** or a **level shifter**. Its function is to charge up the capacitor between the bases of the power follower NPN and PNP pair to a voltage difference that establishes the quiescent current.

In 9C, R1 has been replaced with a PNP transistor which is usually forward biased by another transistor. You may observe any number of variations of the basic biasing circuit presented in Figures 6 and 7, including 3 or more small signal diodes, 2 amplified diodes, current sources, feedback loops and more. although the techniques vary, the authors are still just biasing the complementary emitter followers to achieve low crossover distortion, stability and/or thermal tracking.

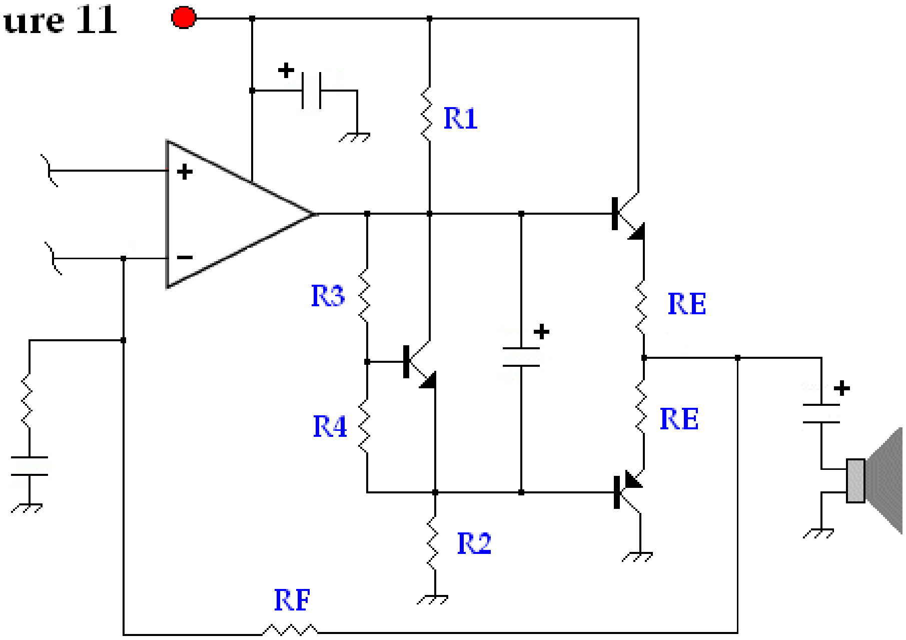


Shown above in Figure 10 are 2 amplifiers using a split power supply. The split power supply offers increased headroom due to a greater AC voltage swing as well as increases the available RMS output power without using super high AC power transformer secondary voltages. In addition, the split supply works well with op amps and if desired, enables you to reduce the number of coupling capacitors by allowing direct coupling of the preamp and speaker to the power amplifier. Coupling capacitors alter frequency response and perhaps may present phase shift issues. In some cases, we as builders use coupling capacitors to provide effects such as high pass filtering, however in Hi-Fi amps, enhanced low frequency response is usually desired; which necessitates the use of high value coupling caps in single power supply amplifiers. In split supply amps, the choice of using a coupling capacitor or not is available to you. In the Figure 10 a and b circuits above, the speaker is directly coupled to the complementary emitter followers output. Note that the voltage at this point is 0 or nearly 0 volts. For any given power supply voltage you chose (split or not), please ensure the amplifier components can handle the current and subsequent heat when a signal voltage is applied. This topic is out of scope. Build and measure...build and measure...



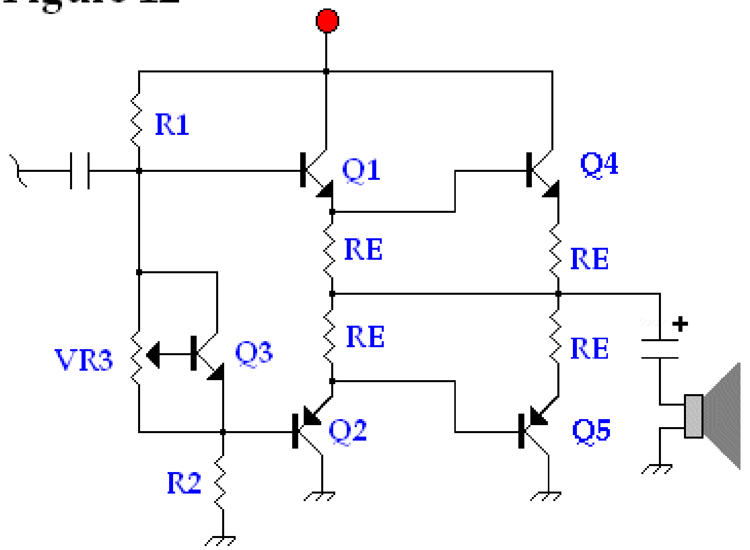
Shown above is a breadboard of the Figure 10b circuit. Additional experiments using even higher voltages were also performed, hence the moderate power TIP transistors were utilized. I burnt up four 2N3904/6 transistors performing many experiments with biasing over 3 nights. Some of the outputs of these experiments will be presented in future projects.

Figure 11



Shown above in Figure 11 is a complimentary emitter follower pair directly coupled to an op amp. The amplified diode and its biasing network is inside the op amp feedback loop. There are examples of this circuit in EMRFD and also on this web site. In single supply powered op amps, it is possible to omit R1. An example of this may be found in Figure 12.30 in EMRFD. Using a low noise op amp such as the NE5532 to drive your power followers can give outstanding results.

Figure 12



Shown above in Figure 12 is another theoretical power amp which illustrates the building block aspects of the simple stage we have been discussing. Q4 and Q5 are cascaded with Q1 and Q2 to build up the current (Darlington emitter followers). Such an amp could have several watts of output power depending on the supply voltage. The emitter resistors on Q4 and Q5 are often 0.47 to 1 ohm power resistors.

References

EMRFD Although Rick Campbell and Bob Larsen contributed chapters and circuits, the principal author is Wes Hayward. It amazes me that any human being could know so much about electronics and is so willing to share his knowledge. Respect.

Henderson, John. Electronic Devices Published in 1991 by Prentice Hall

Oleksy, Jerome E. Practical Solid-state Circuit Design Published 1974 by Howard W. Sams and Co.,Inc

Slone, Randy G. Understanding Electricity and Electronics Published 1996 by TAB Books-McGraw Hill

Amateur and Short Wave Radio Electronics Experimenter's Web Site

Low Power Audio Amp Experiments

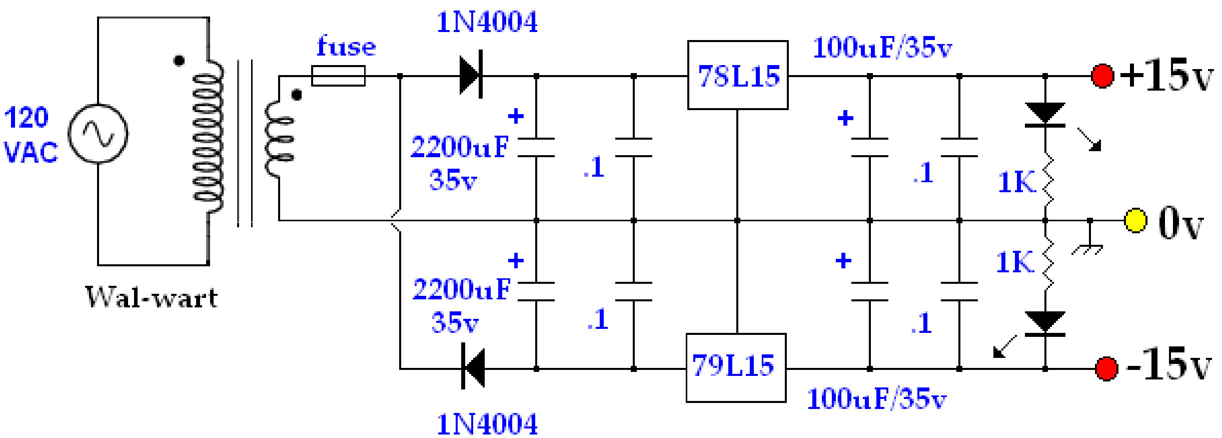
Introduction

This web page contains some experiments on simple, low power, speaker output audio amplifiers. Presented are ideas, some measurements and examples of audio amplifiers which will likely sound better than the IC audio chips commonly seen in many receiver projects. This web page is a follow-on to this one and is a completely new area of experimentation for me. Audio amps were built using both split and positive power supplies. In all cases the complimentary power followers were driven by an op-amp. I tried building some power amps using discrete transistor differential amplifier stages with current sources as the driver, but the noise performance and simplicity of the NE5532 or NE5534 op-amp was superior.

Split or Bi-polar Power Supply Audio Amplifiers

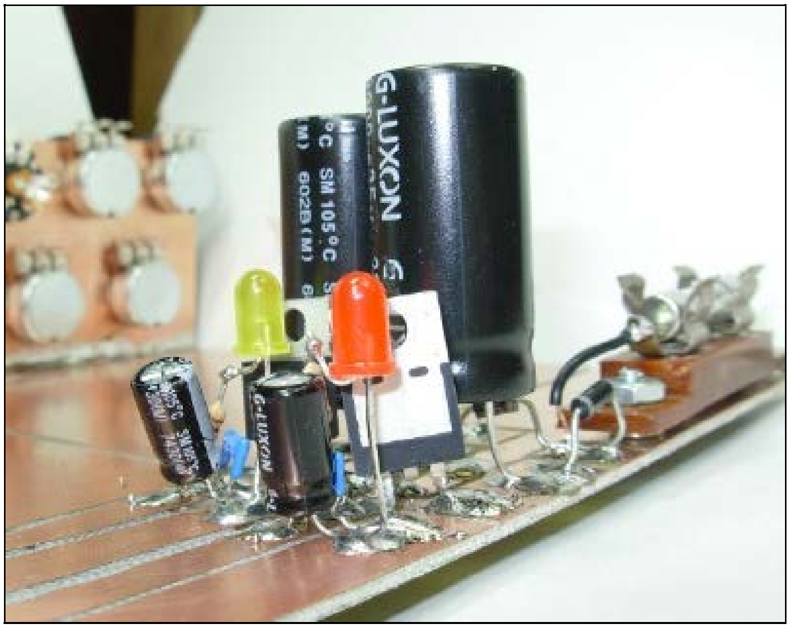
Figure 1

Split Power Supply



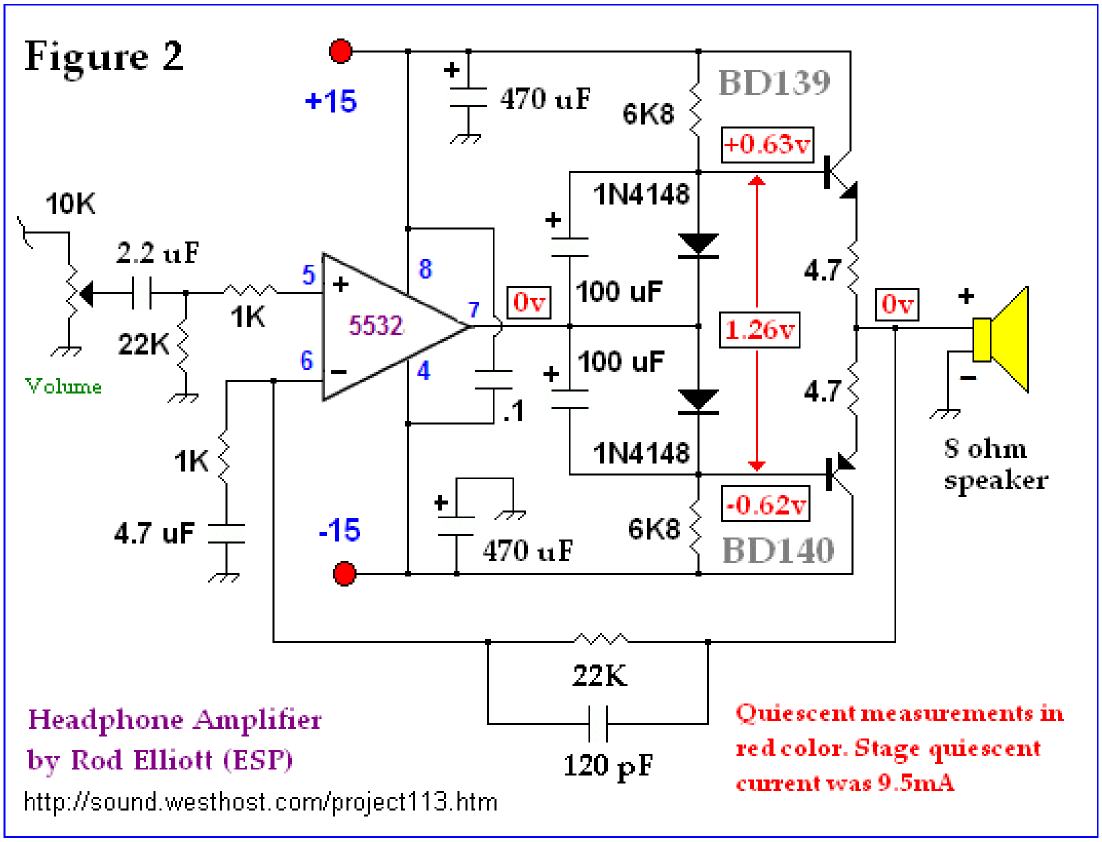
corrected Dec 16, 2009

In order to build up some split power supply amplifiers, a basic power supply was constructed and the schematic is shown in Figure 1 above. I found it was essential to regulate the voltage or hum would appear on the output. Choose a standard value fuse that is rated somewhere just above the maximum current you measure. I used 2 different AC output power transformers which were in the 18-24 volt, 375 mA to 1 amp range. The LEDS are strongly suggested. They inform you when there is power applied and their relative brightness will also often fall when higher current is being drawn on one side or the other. This alerted me to an accidental solder bridge to ground on more than 1 occasion.



The split power supply is shown in the photograph to the left. The retro Bakelite fuse holder is from an old tube audio amplifier. If you are wondering why the copper clad board is so large, the power supply is part of a future project. Some builders would use even greater value filter capacitors than those shown. Heat sinks on the voltage regulators are required for supplying DC to higher power amplifiers.

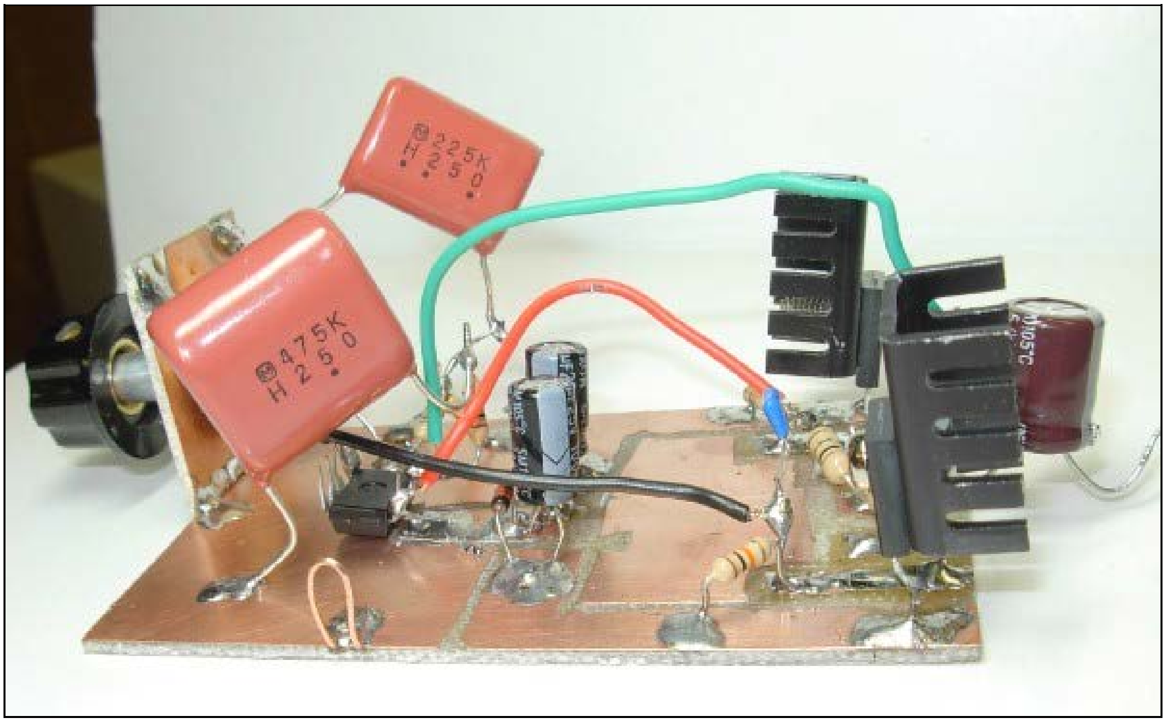
Rod Elliot Headphone Amp



The first amp I built is shown in the Figure 2 schematic above. This amplifier was designed by Rod Elliot and is used with his permission. Rod's ESP web site is a virtual treasury of audio design information. If you are into understanding audio design, visiting his web site is strongly recommended. Rod sells printed circuit boards for all of his circuits if you prefer this building method . Note I have made some minor modifications to some part values. although primarily designed as a low distortion, high power, headphone amplifier, it drives an 8 ohm speaker very well. I was able to drive this amplifier as high as 0.68 watts average power with a pure sine wave output during analysis. Power measurement is discussed in the next section. Note that on this web page, I quote the entire stage quiescent current. Since the op-amp and the

2 (or more) power followers are a "package", it is a lot easier to just measure the current at the power supply lead(s) of the stage than unsolder and lift up a transistor lead. In this case, with no input single, the stage current was about 9.5 mA. I did check and about half the current is going to the op-amp with the other half to the transistor pair. This is a wonderful sounding amplifier and Rod has an entire web page devoted to it, so I will not comment further.

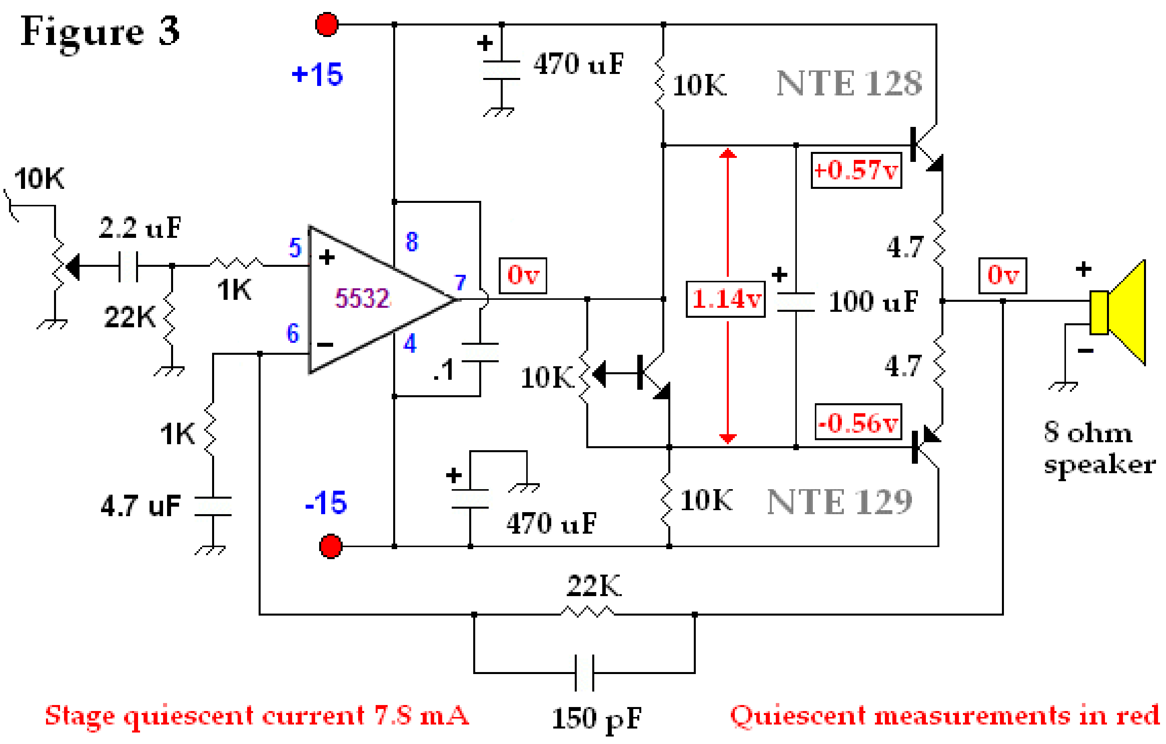
There are a variety of suitable transistor pairs for audio power amplifiers depending on the power output you are choosing. I stock just a few; BD139-140, TIP 41C-42C, NTE 128-129. The higher beta 2N3904-3906 or 2N4401-4403 pairs worked well in the low power, single power supply amplifiers shown on this web page. I also performed some higher output power experiments which required the TIP and BD transistors and these are not shown.



Above photograph. A breadboard of the Rod Elliot headphone amplifier. This early version had a temporary output capacitor. When first testing a new circuit that has a direct speaker output, it might be a good idea to temporarily use an output capacitor until you measure your voltages and current and feel your transistor temperatures. This will save your speaker if you made a big mistake and/or blow up the transistors when you first power it up.

Amplified Diode Biased Audio Amp

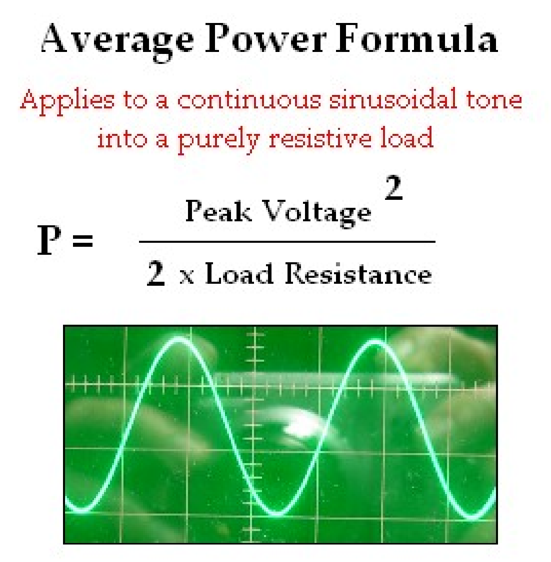
Figure 3



Above in Figure 3 is a split power supply audio amp using an "amplified diode" to control the bias. The bias transistor was wedged between one of the output transistors and a piece of copper clad board to allow thermal tracking. The 10K bias control resistor was a trimmer type suspended over the copper clad board in most of my bread boards. Usually, you just need to set and forget about this resistor after initial set up. I adjusted the bias by watching in my oscilloscope with a low level, 1 KHz sine wave connected to the input. I measured the various voltages and stage current at quiescent and have indicated these values in red for learning purposes. The bias current range was 7.2 to 154 mA when turning the 10K trimmer pot from 1 extreme to the other. The maximal clean output average power of this amp was 0.78 watts. I used press on heat sinks for the NTE128-129 pair and they ran quite warm to touch. These TO39 type packaged transistors are somewhat difficult to heat sink compared to the TIP/BD transistor packages where you can just bolt on a heat sink of any size that is required. Please remember that the metal tab on the TIP and BD transistors is connected to the collector terminal.

In the above photograph is my first bread board of the Figure 3 amplifier. This particular version had TIP transistors, a 4 ohm speaker and an 470 uF output capacitor. Note the full size 10K bias control potentiometer on the left hand side. This was purely a experimenter's bread board, but it sounded amazing when listening to music through it.

Harmonic Distortion and Measuring Output Power



Average Power Formula

Applies to a continuous sinusoidal tone into a purely resistive load

$$P = \frac{\text{Peak Voltage}^2}{2 \times \text{Load Resistance}}$$

Figure 4

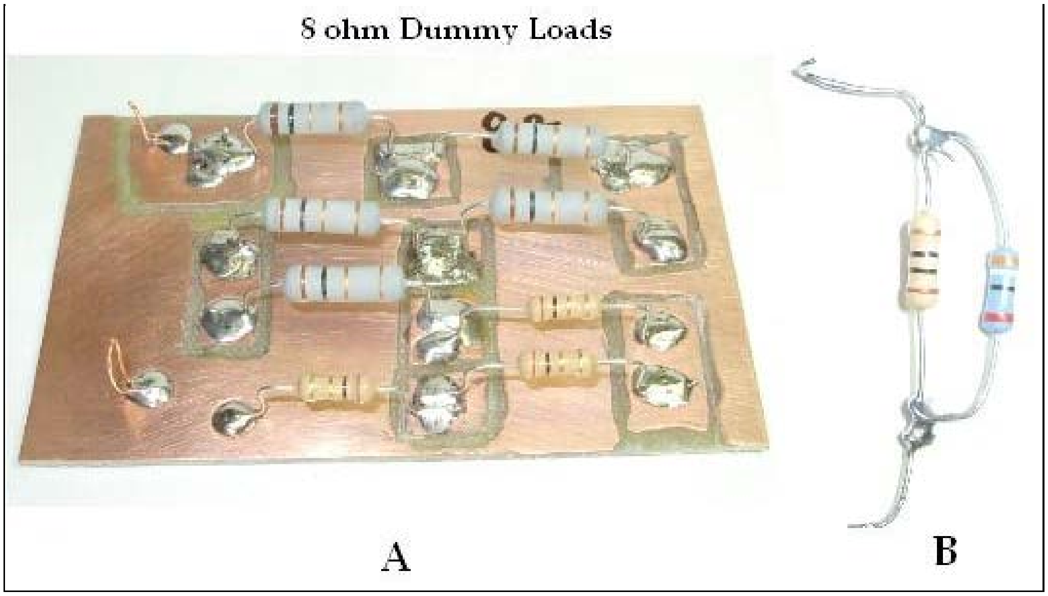
In Figure 4 is the formula used to calculate the average power of the circuits on this web page. For example if you measure 6 volts peak to peak on the oscilloscope, (3 volts peak voltage) and your resistive load is 8 ohms, the average power is 560 milliwatts. At any point in an AC waveform there is power and it may be reported using a variety of ways. Was it clean? distorted? a peak value? an RMS value? - often it is unclear.

To be clear, I measured the peak voltage on a pure, undistorted sine wave into an 8 ohm resistor. Stated power values are the mean sine wave power calculated with the formula shown. See this somewhat controversial link for details. You may not agree with my methodology, however, it allows you to compare the circuits on this web page. If you really must know the peak power, multiply the stated average sine wave power by 2. I will leave the power measurement and calculation debate up to scholars; as a lay-person, I need something simple.

The bench voltage measurement was as follows: The amplifier was connected to a 1 KHz pure sine wave generator and the 10K volume control pot was advanced just until any sign of distortion of the amplifier output sine wave appeared. Voltage measurement was taken at the point just before distortion occurred.

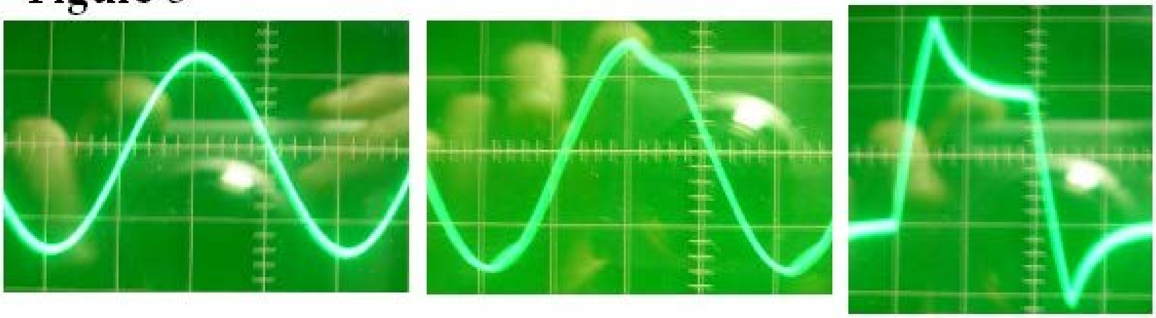
It is difficult to photograph a sine wave without a tripod. Motion, the angle, light reflection and jpeg graphic compression all wreck the perfect sine wave. In the Figure 4 graphic above is a typical 1 KHz output waveform from my power amplifiers (squeaky clean) at the amplifiers maximum average power level.

To the right is a photograph of my AF signal generator. This is an old, tube device but the output sine wave is beautiful. I did not perform spectrum analysis with a computer audio sound card program and will leave this up to audiophiles. These audio amps sound great; especially when compared to the IC audio power amps that many of us tend to use in our receivers.



In the above photograph is **A**, an 8 x 1 ohm resistor load and **B**, an 8 ohm load made from parallel 1/2 watt 10 and 39 ohm resistors. In dummy load **A**, I used 5 two watt metal film resistors plus 3 half watt resistors. In the future, I will obtain 3 more 2 watt resistors and replace the 1/2 watt resistors for a 16 watt rating. For a quick resistive load, **B** is the way to go for most of the circuits on this web page.. You can make a 4 ohm load from parallel 4.7 and 27 ohm resistors. In truth, a single resistor or any combination of resistors adding up to the desired load R value will work.

Figure 5



In Figure 5 above are some scope waveforms ranging from mildly distorted to full-on dirty.

Power Amplifier Concerns

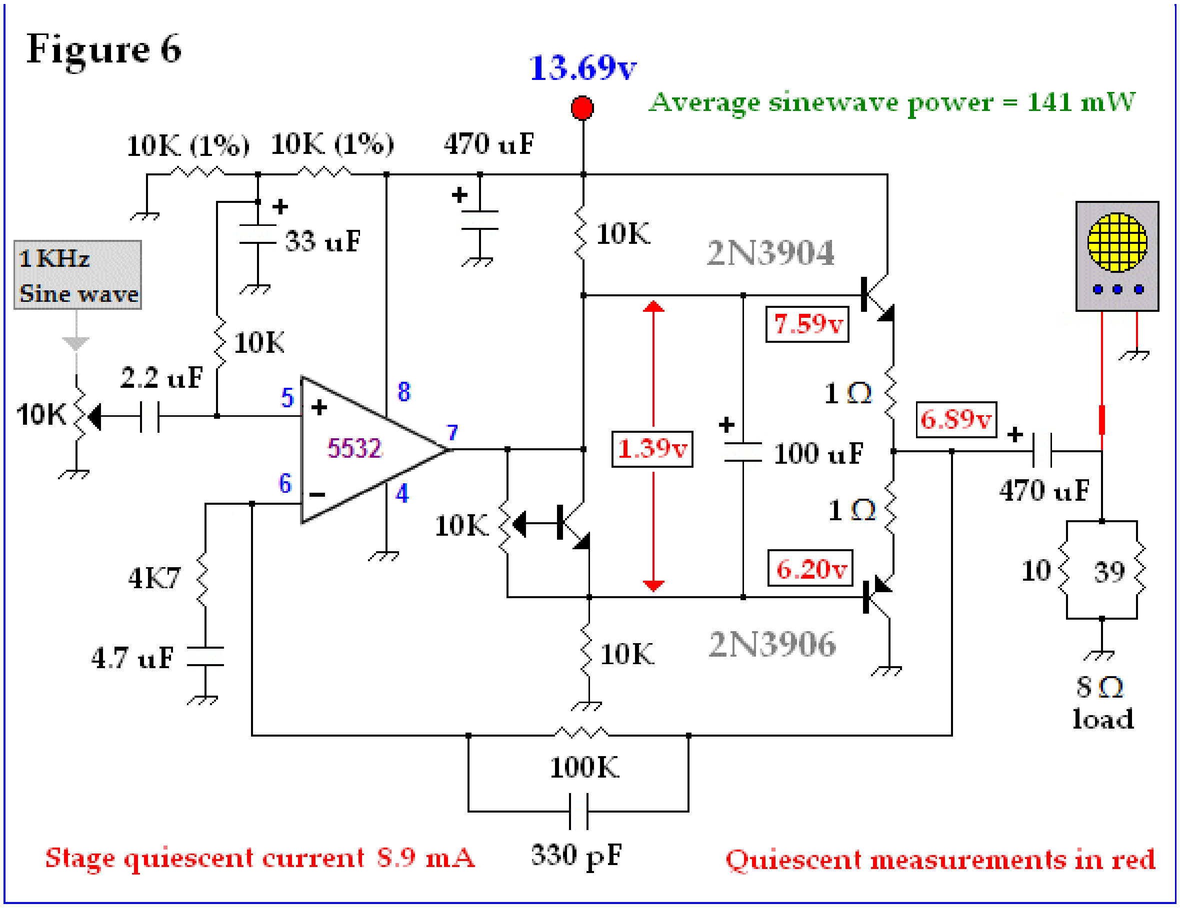
Although the amplifiers on this page are 0.15-0.8 watts or so, they can consume relatively large current compared to the usual voltage amplifier circuits we build. Some potentially helpful tips to help keep away ground loops, oscillations and thermal run away are suggested as follows:

✓ Connect your negative speaker terminal directly to the AF power amp (do not use a common ground for the negative speaker terminal).
✓ Use big power supply line bypass capacitors (no 10 uF caps here)
✓ Keep your audio amplifier copper clad board separate from the rest of your circuit boards and star ground it to your main power supply ground point.
✓ Use heat sinks on your final transistors and voltage regulator(s) when you go for bigger power
✓ Watch your layout - keep the output away from the input etc.
✓ Watch your emitter resistor power ratings in "higher wattage" amplifiers. Burning resistors stink.

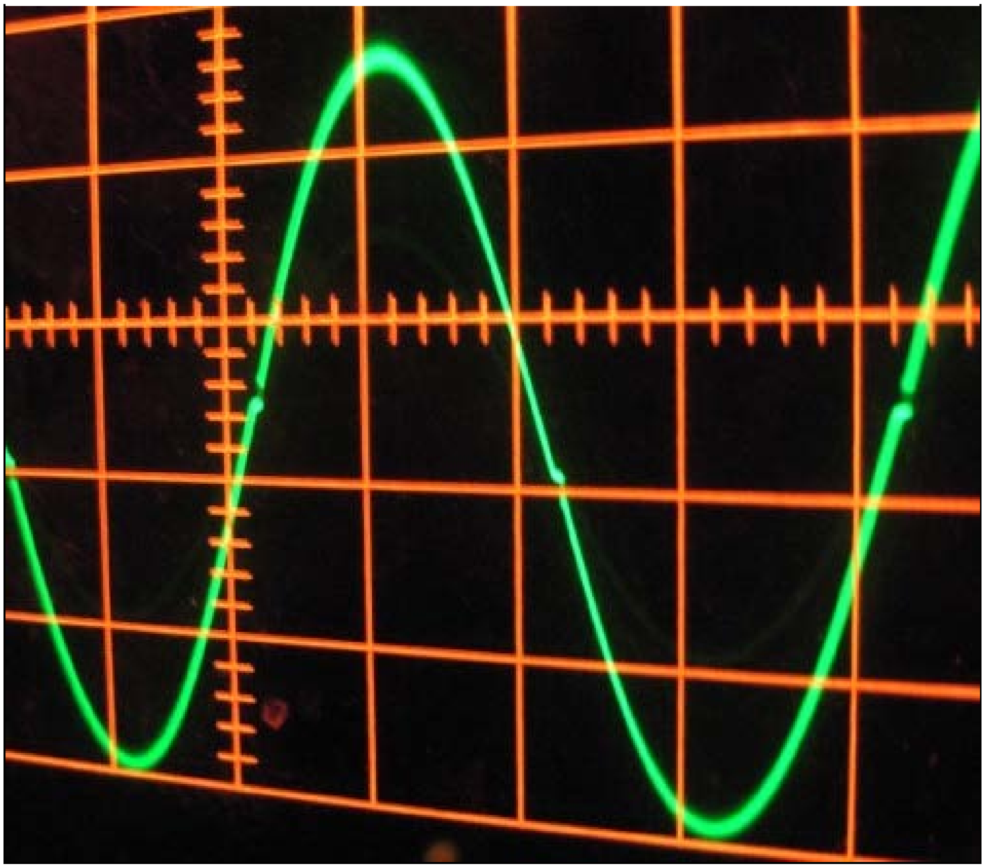
Single Power Supply Audio Power Amps

Since most 12 volt power supplies are actually closer to 14 volts; these experiments were performed with a typical radio bench DC power supply at 13.69 volts.. Figure 6, 7a and 7b represent evolving experiments aimed at obtaining greater output power.

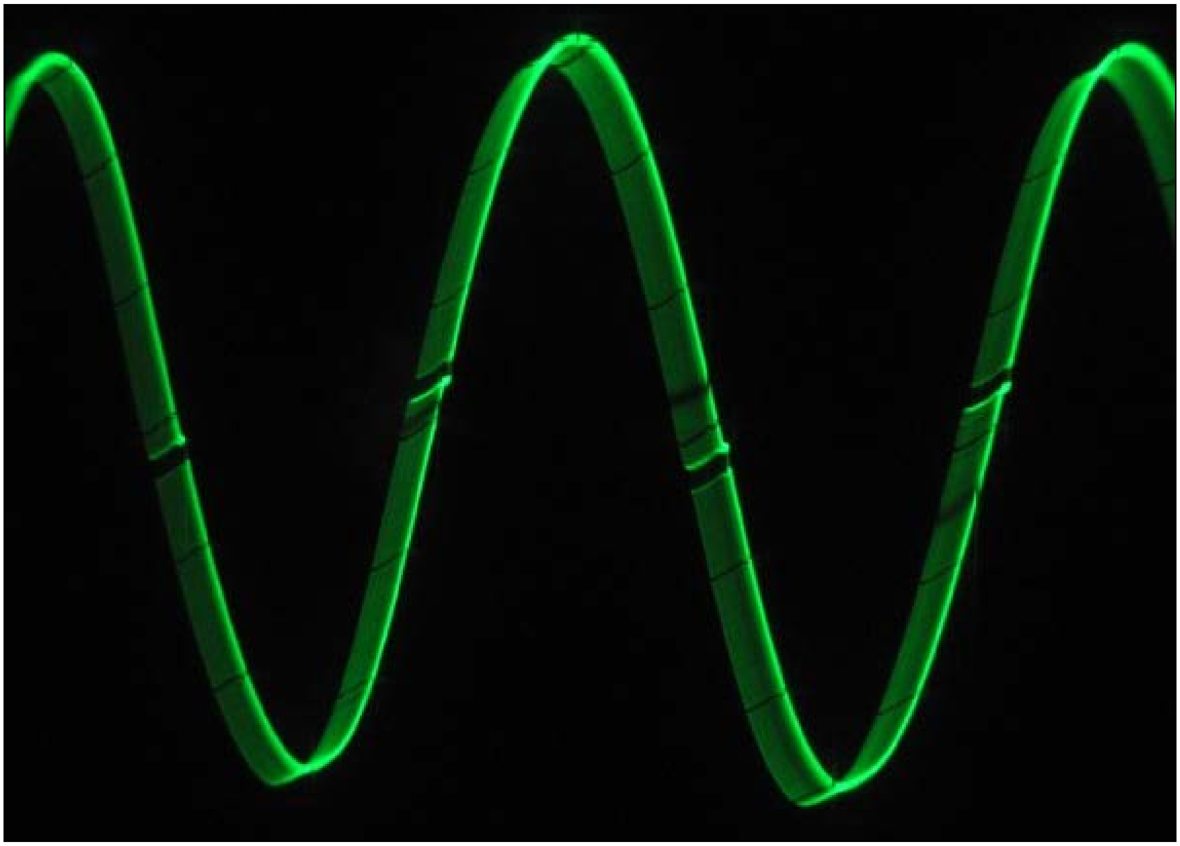
Figure 6



Shown above in Figure 6 is the fundamental design using one op-amp and 2 power followers. It is shown in an AC output power measurement configuration. The bias current range was about 5 to 100 mA when turning the 10K trimmer bias control pot from one extreme to the other. Maximal sine wave average output power was only 141 mW. Nevertheless, it might be loud enough for some receiver applications.
I connected this amplifier to a VCC of 15 volts. The maximal sine wave average output power was then 220 mW. In all of the single supply audio amps presented , increasing the VCC increased the maximal power output. Driving these amplifiers beyond a pure sine wave output power resulted in predictable harmonic distortion plus the re-emergence of crossover distortion in the output. This was an incredible learning; how could there be crossover distortion re-emerging in a amp that was properly biased to begin with? Increasing the bias current to the maximum level did not remove this crossover distortion. After emailing this question to Rick, KK7B and Wes, W7ZOI, and reading their replies, my best guess was that at some power level, the 5532 op-amp can not provide enough current to properly drive the complementary symmetry pair. The AC current in the output transistors may be limited by the base drive of the op-amp and they were no longer forward biased at the crossover point.



Above photograph. This is the Figure 6 amp driven past the point where the sine wave is pure. Note the crossover distortion blips on the sine wave. The base drive current for the power follower pair all comes from the op-amp. At this point there is likely not enough base drive to keep the base emitter junctions forward biased.



In the above photograph I blacked out the room and photographed the same scope waveform as above while shaking the camera from side to side. This adds some horizontal spreading of the signal and provided more information about what was happening as compared to a single,

clear oscilloscope trace.
I should mention that this crossover distortion blip occurred in all of the Figure 2 to 6 amps when they were driven past the point where a pure sine wave was seen. It is clear that maximal available power from a simple audio amplifier like this **(one NE5532 op-amp plus 2 power followers)** is constrained and thus its application is limited. Greater output power is possible using a split supply per Figures 2 and 3, however, a typical radio project has a 12 volt, single power supply. These basic amplifiers with a single power supply, may be very appropriate for projects such as a compact radio receivers or a code practice oscillator project, but not for applications where you require louder audio.

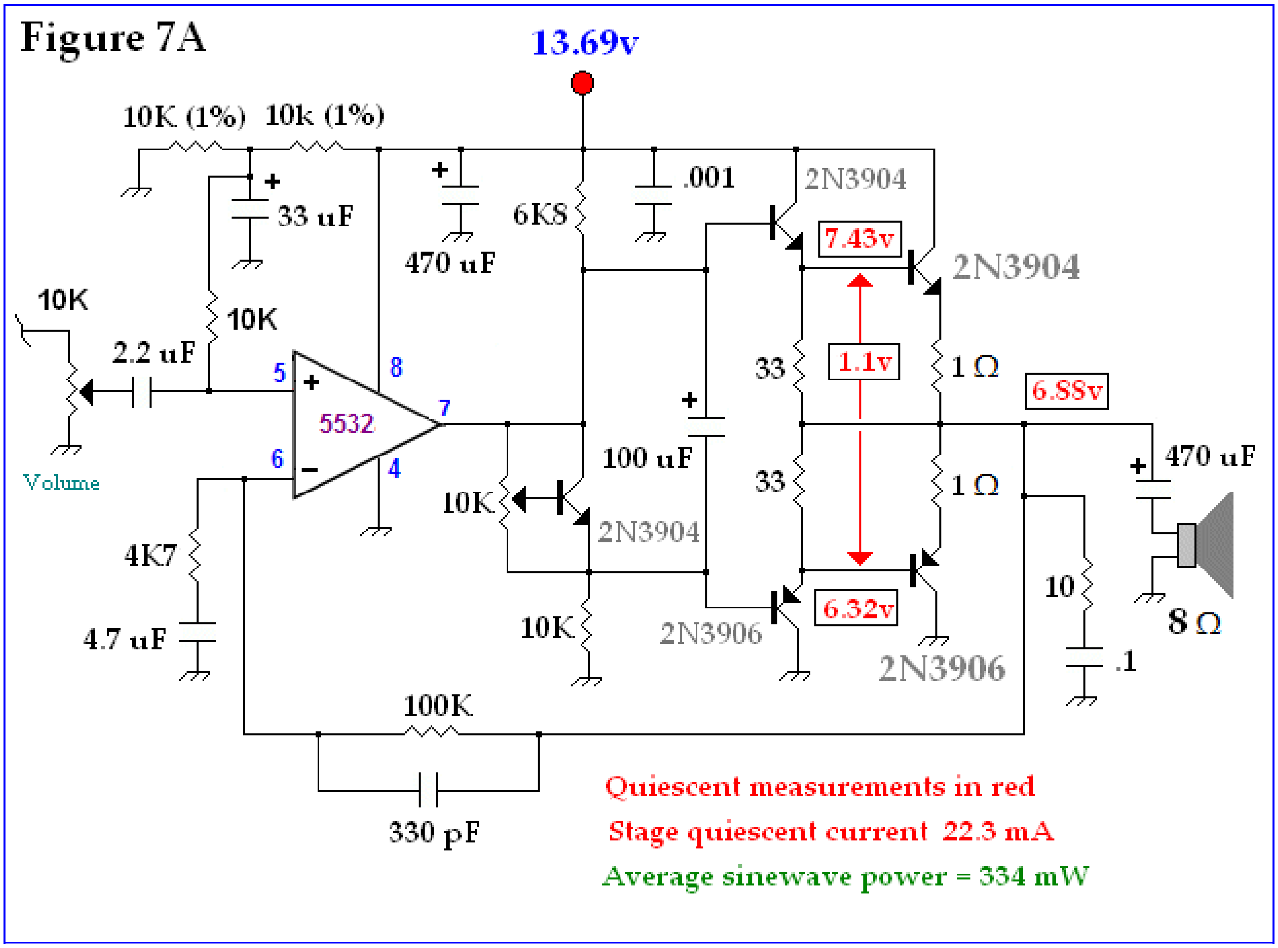


Figure 7A

Quiescent measurements in red
Stage quiescent current 22.3 mA
Average sinewave power = 334 mW

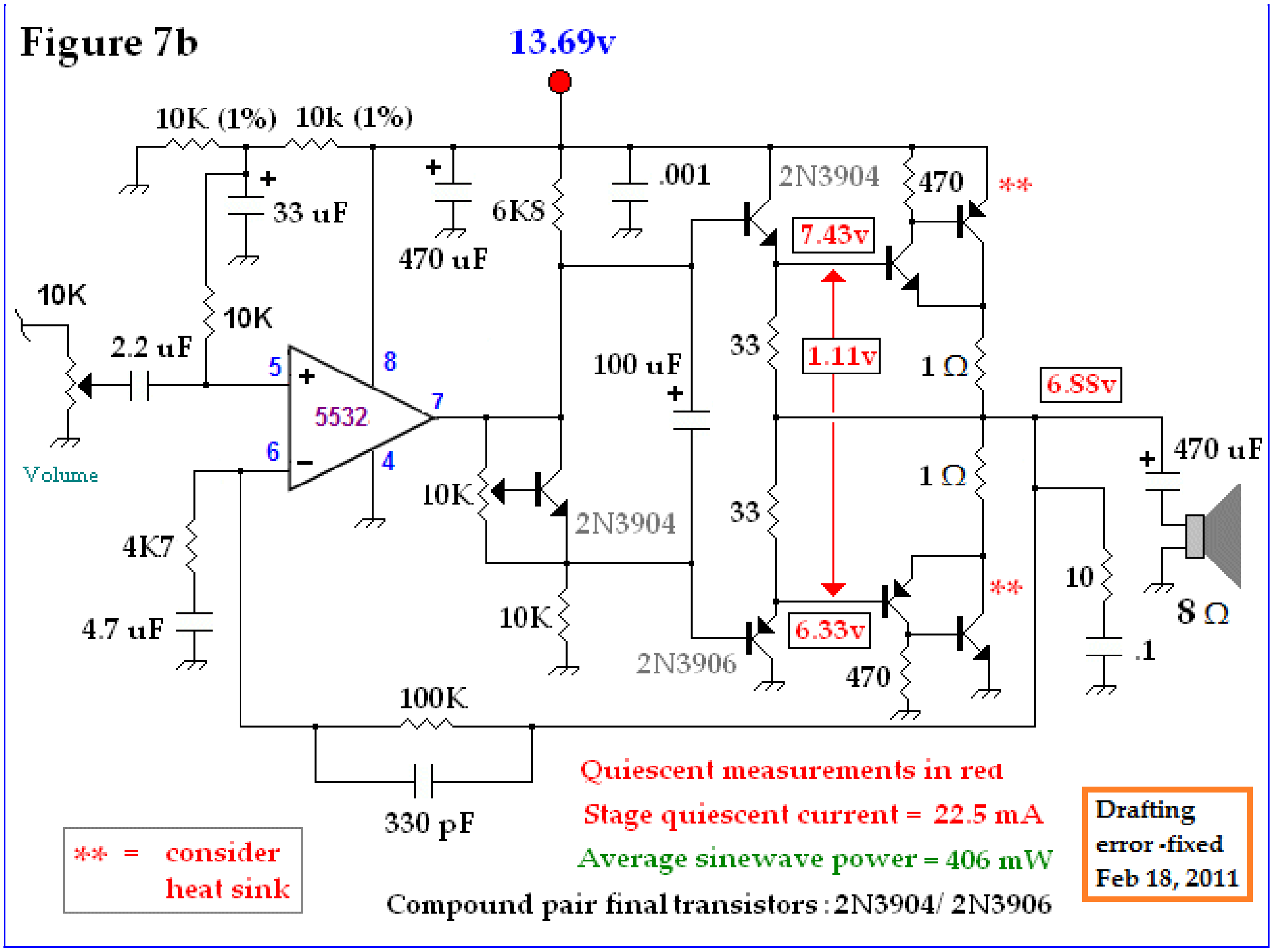
Shown above in Figure 7a is an easy method to get more output power from the Figure 6 amplifier; add another set of complimentary pair current amplifiers. I found 33-39 ohms to be a good emitter resistor value during my experiments. Many hi-fi amp builders will use greater emitter resistor values, however, a design goal was to get more output power from our 12 volt supply. Series emitter resistors are used to improve linearity and operating-point stability. I kept the final power follower pair emitter resistor values at 1 ohm to get maximal output power. An output 10 ohm + 0.1 uF low pass filter was used to help prevent oscillations in view of the low emitter resistor values on the finals.

Biasing

The top 10K bias resistor was lowered to 6K8 to facilitate "more linear" setting of the output transistor bias with the 10K trimmer potentiometer. It did not help much. Setting the bias is very delicate procedure and you must turn the screw driver very slowly. In my bread board, the optimal stage bias current was 22.3 mA but anything around 20 mA should be fine. If you do not have an oscilloscope, after ensuring that there is no input signal, connect an ammeter in series with the positive power supply lead. Turn the bias potentiometer with with the screwdriver until you get close to 22 mA. If you only have a voltmeter, the rule of thumb of 1.1 seems to work... Measure the voltage across the final 2N3904-2N3906 bases and ensure the difference is at least 1.1 volts while adjusting the 10K trimmer pot. Personally I do my biasing with an oscilloscope at at least 2 different frequencies on the signal generator, however the for mentioned methods will work okay. This is a popcorn stage and a popcorn web site after all!

For the lowest potential noise, consider using metal film type resistors in your audio amps and "polysomething" capacitors wherever AC signal is coupled to another component or ground, excluding the output capacitor.

Figure 7b



Quiescent measurements in red

Stage quiescent current = 22.5 mA

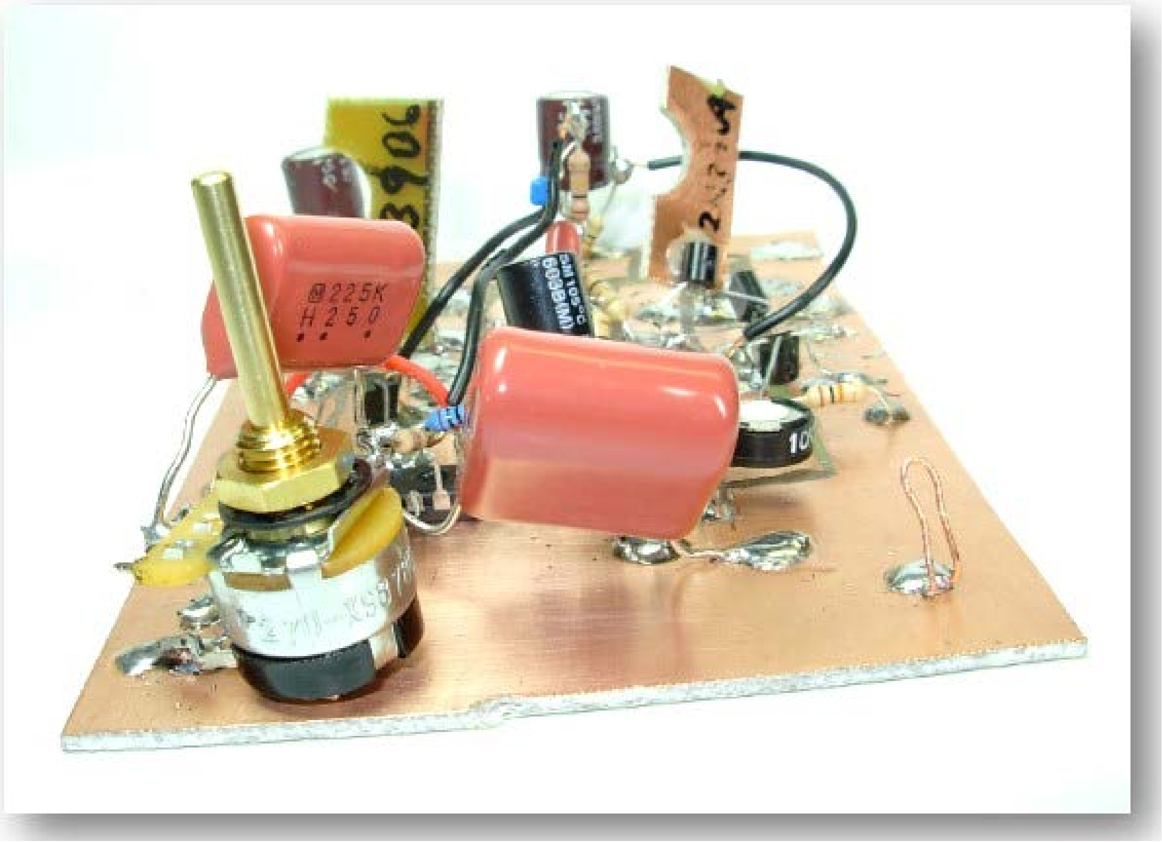
Average sinewave power = 406 mW

Compound pair final transistors : 2N3904/ 2N3906

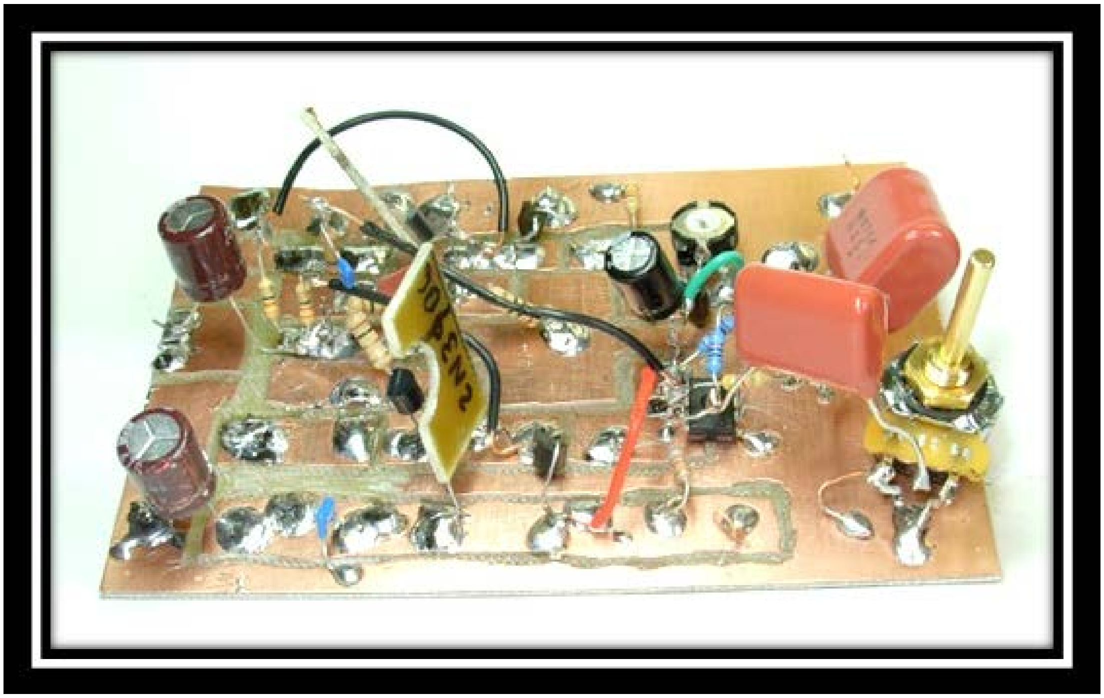
** = consider heat sink

Drafting error -fixed Feb 18, 2011

Further experiments to increase output power were frustrating. Finally a compound or Sziklai pair was trialed and increased the average power to over 400 mW as shown in Figure 7b . I used a small piece of copper clad board on the finals for a heat sink, although they really didn't get that warm. Ideally, the amplified diode should also be glued onto one of the heat sink boards for thermal tracking. This amplifier is now in a chassis as a bench reference audio amp for receiver testing.



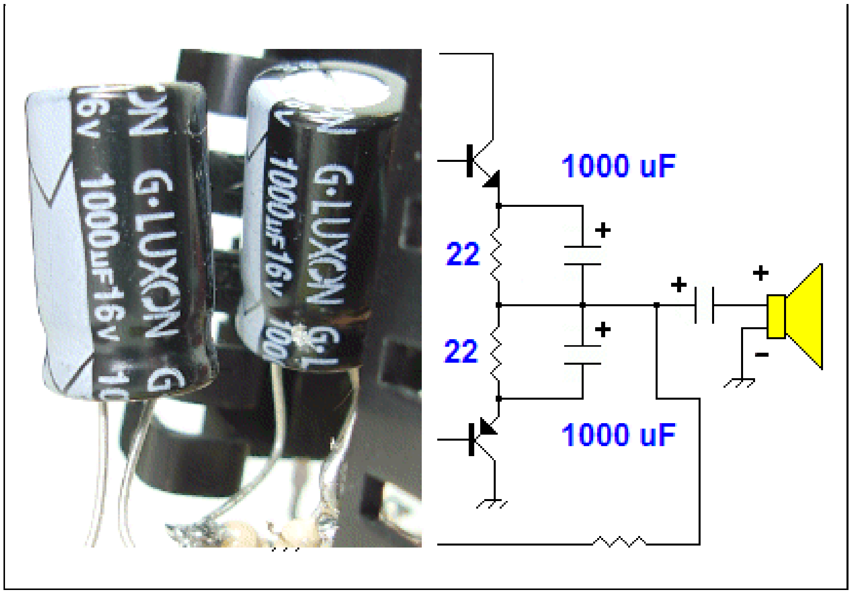
In the above photograph is the Figure 7b prototype. I am using a new miniature potentiometer for my experiments that I bought from Digi-Key. The base has 2 leads which can be soldered right on the copper clad board for easy anchoring and removal after testing.



Another view of the bread board on which the Figure 6, 7a and 7b experiments were conducted.

KK7B Headroom Boosting Emitter Capacitors

We first learned about using large value emitter caps in audio amp complimentary pairs from EMRFD. Experimentation revealed that these capacitors do 2 things: **1.** Can increase the amplifier sine wave headroom and **2.**



Add some low pass filtering. I learned from Rick, KK7B, that he designed his EMRFD amp to achieve low output power, low distortion and lower DC current drain. He desired a clean output audio amplifier for his R2 series of receivers without needing a lot of quiescent current or heat sinks on the 2N3904-2N3906 pair. The caps were added to make the amplifier think it had much lower emitter resistors at AC than the 22 ohm resistors he used in the EMRFD projects. When Rick made measurements and simulations of the amplifier, it was very stable, had low distortion and provided a very nice clean sound at all signal levels, from very weak signals in a few milliwatts of noise, to music driving the speaker. The result is outstanding and Rick's design was the catalyst for my own interest in audio amplifier experimentation.

I performed experiments with these capacitors and found that they increased my amplifier power and head room in some cases, and that the boost is indirectly proportional to the emitter resistor value. With 1 ohm output transistor emitter resistors the boost is generally not that significant. With 4.7 ohm or greater emitter resistors, they can make a big difference and you might consider trialing them for more power and headroom as appropriate. They can also add a nice, warm sound to your audio amp. Refer to EMRFD for numerous examples of this technique.

Popcorn Audio Amplifier

What follows is a popcorn or "poor man's" audio power amp using the 2N3904-2N3906 pair. To meet true popcorn criteria, all of the capacitors used in my breadboard were electrolytic and you can substitute different values from your own junk box. It would be better to use "polysomething" capacitors for the NE5532 pin 5 and 6 signal capacitors if you have them. I normally use a 1 uF to 4.7 uF poly-type capacitor in series with the 4K7 resistor on pin 6. The 270 pF feedback capacitor could be omitted or substituted with a higher or lower value to suit whatever high frequency roll off you desire.

The transistor glue-on heat sinks seen in the 1 bread board photograph are completely unnecessary. This BJT pair were used in other higher power experiments as well. The 22 uF capacitor between the transistor bases is essential from my experimentation. Without this capacitor, the amplifier headroom decreases and crossover distortion occurs. You can use the other half of the 5532 for a preamplifier or use a NE5534 instead. A 741 op-amp would be a horrible substitution. The NE5532 performance is breathtaking considering its low cost. 181 mW is surprisingly loud. All resistors are quarter watt rated. What a fun little amplifier! The schematic is Figure 8.

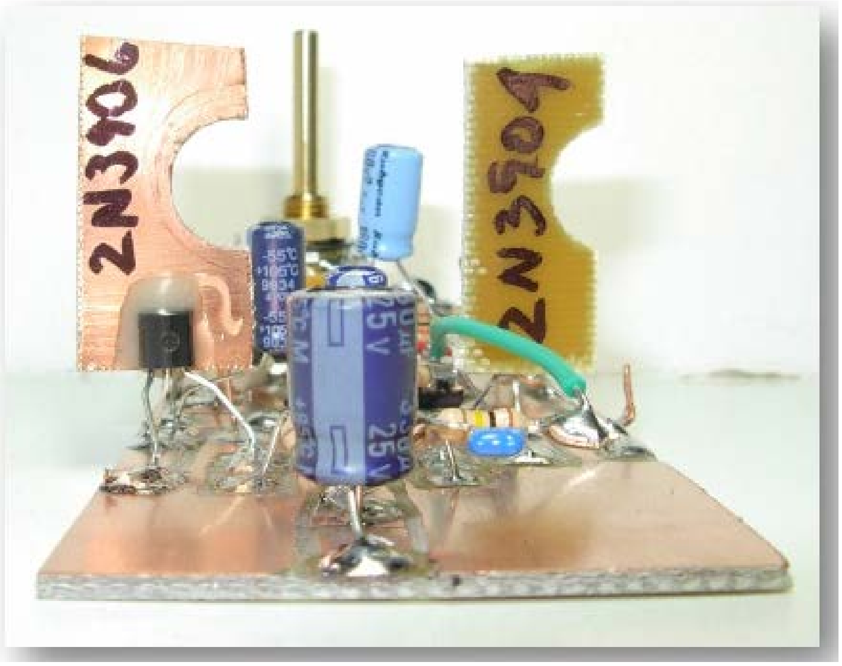
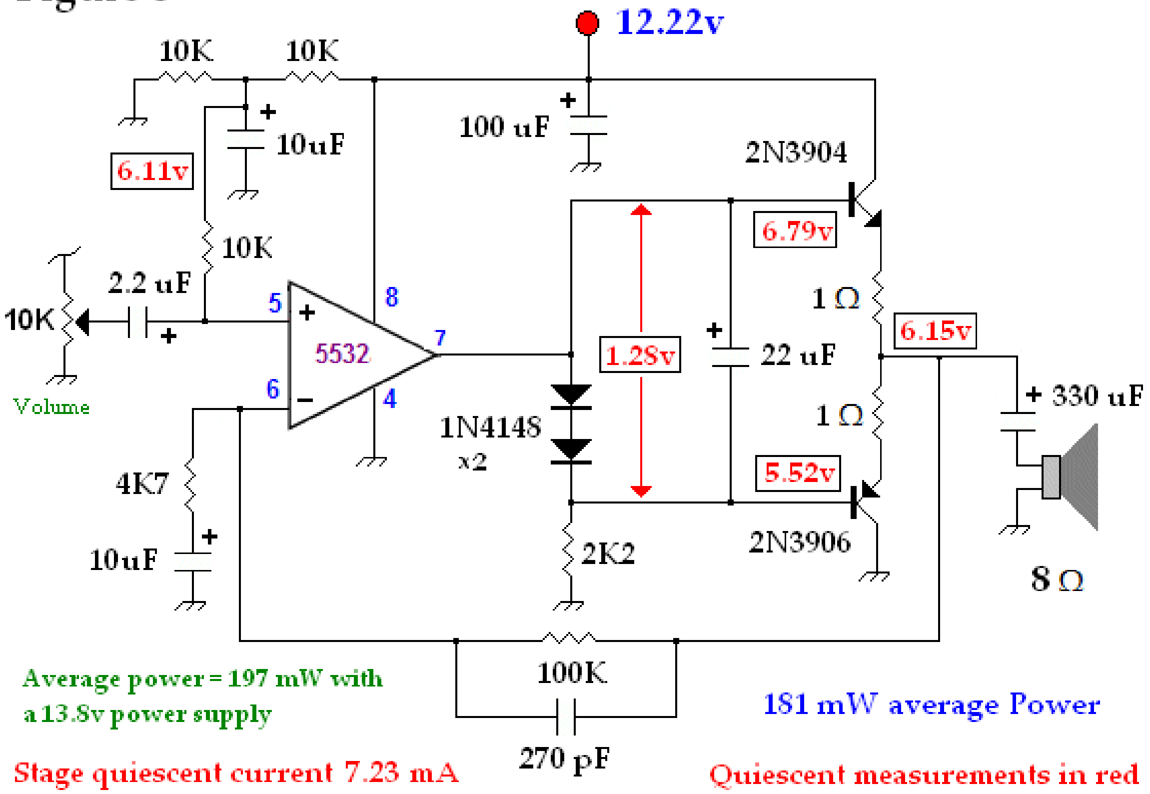


Figure 8



Average power = 197 mW with a 13.8v power supply

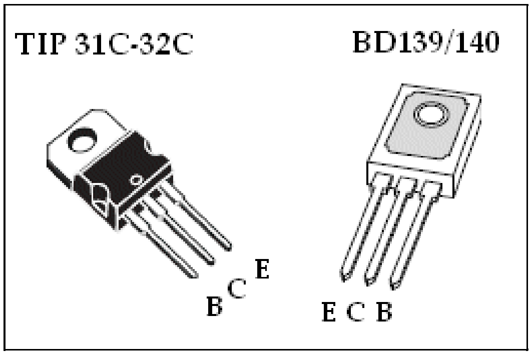
181 mW average Power

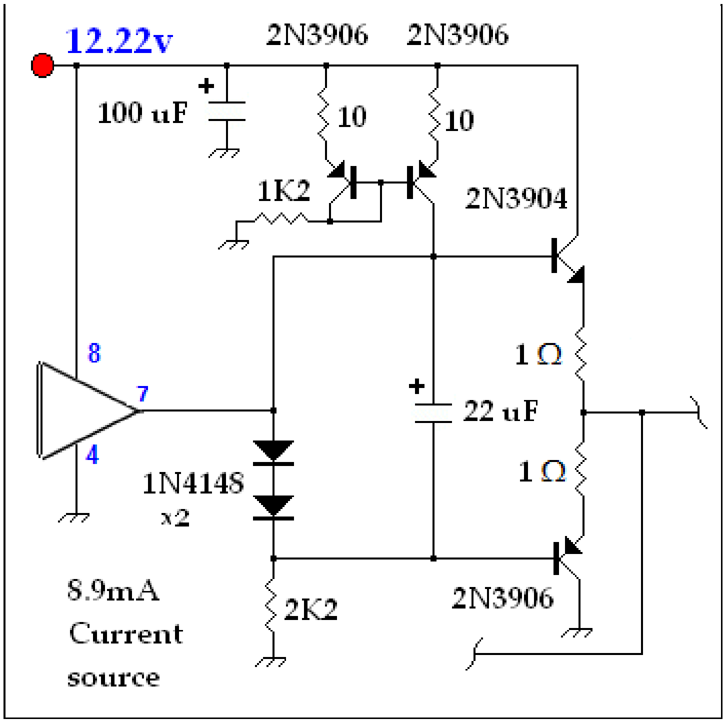
Stage quiescent current 7.23 mA

Quiescent measurements in red

The bread board of the Figure 8 popcorn amplifier using transistors without heat sinks.

Additional Outputs





I tried putting a current source on the Figure 8 bias and it made no difference to the amplifier characteristics according to my simple oscilloscope, listening and DC analysis.

One of the full wave rectifier, voltage regulator and filter bread boards used in these experiments. I went as high as 24 VCC on some single supply amps I tested and was getting over 5 watts average output power


NPN

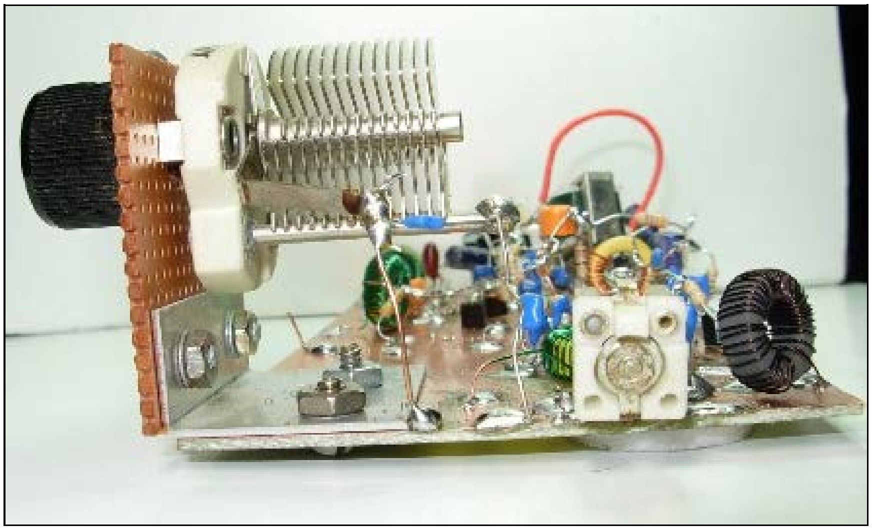
QRP SWL HOMEBUILDER

Amateur and Short Wave Radio Electronics Experimenter's Web Site

Two Bravo Receiver Experiments

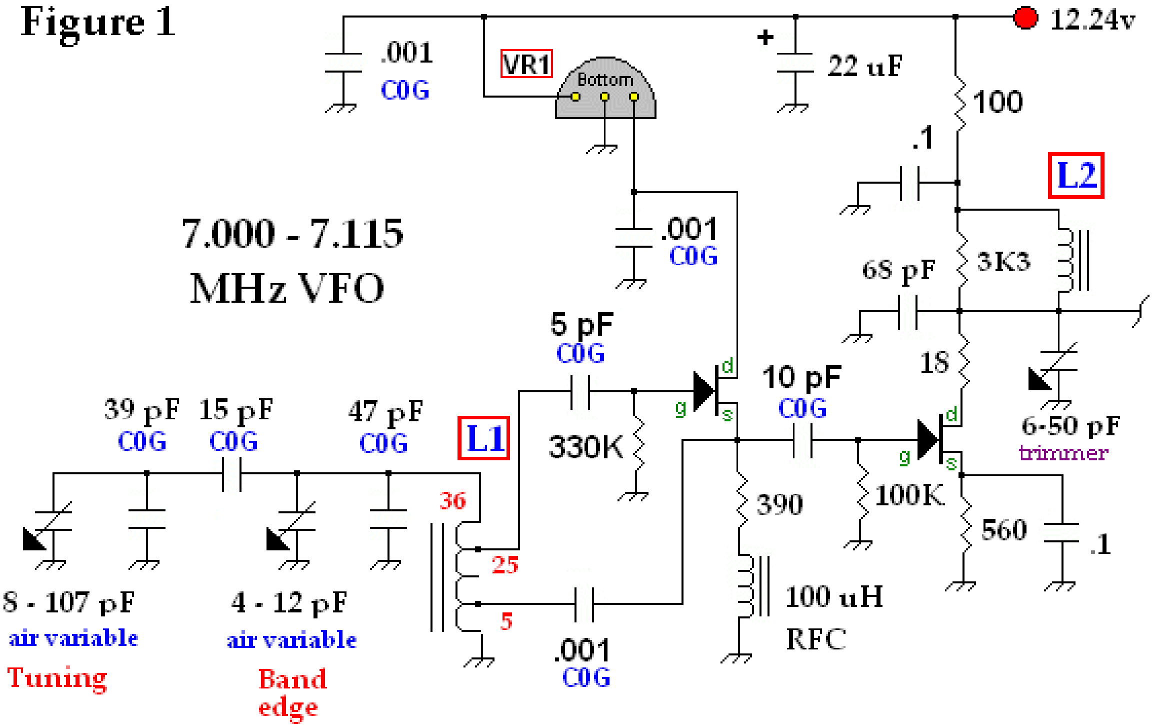
Introduction

An experimental direct conversion receiver is presented. This 1990's style receiver was built to re-familiarize with DC receivers and try out a few new ideas. Design-on-the-bench bread boarding was used exclusively and was a pleasant way to both learn and pass time. Feedback has been received stating that that certain stages of previous receiver experiments were either too basic or too complex and thus a particular receiver was not built. This web site is as much a cookbook as anything. Kludge together whatever receiver stages you want; no project is meant to be set in stone. This receiver has a high popcorn factor with MPF102 and 2N3904s as the main semiconductors.



Variable Frequency Oscillator

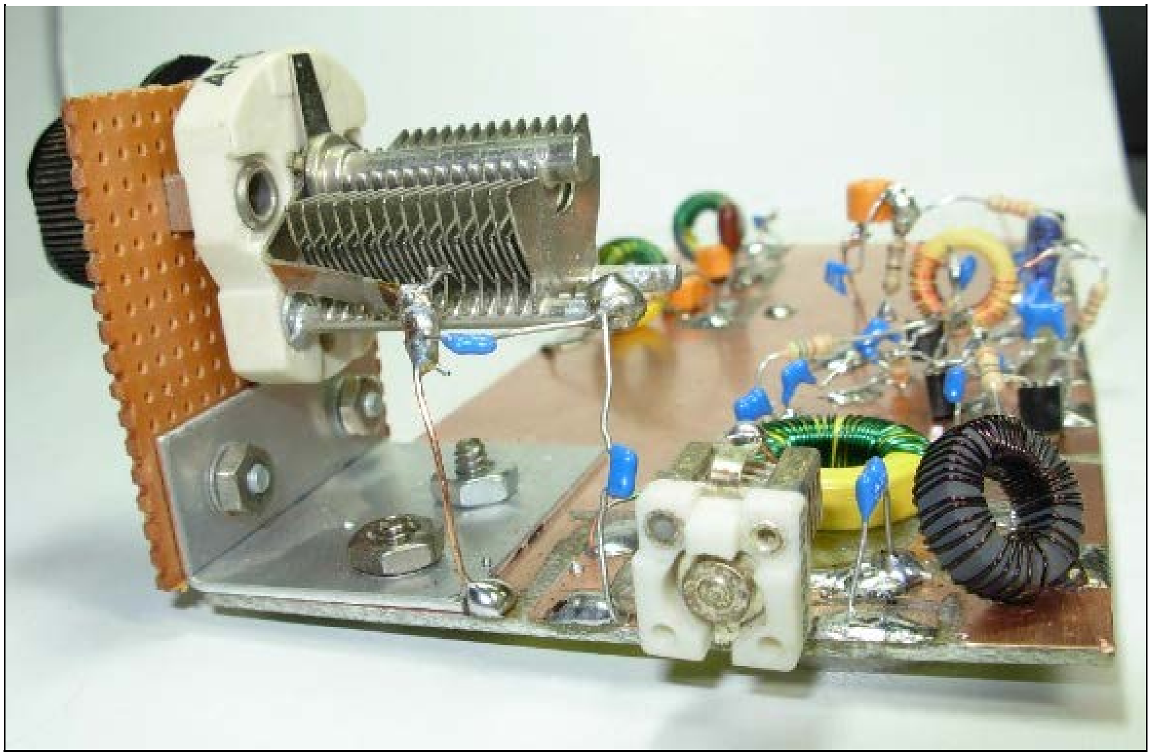
Figure 1



L1 = 6.1 uH = 36t on a T68-6 torroid; tap at 5 and 25 turns (used 26 AWG)
L2 = 3.6 uH = 30t on a T50-6 torroid or 35t on a T37-6
N-channel JFETs : MPF102
100 uH RFC used was wound; 38t on a FT50-61 ferrite torroid

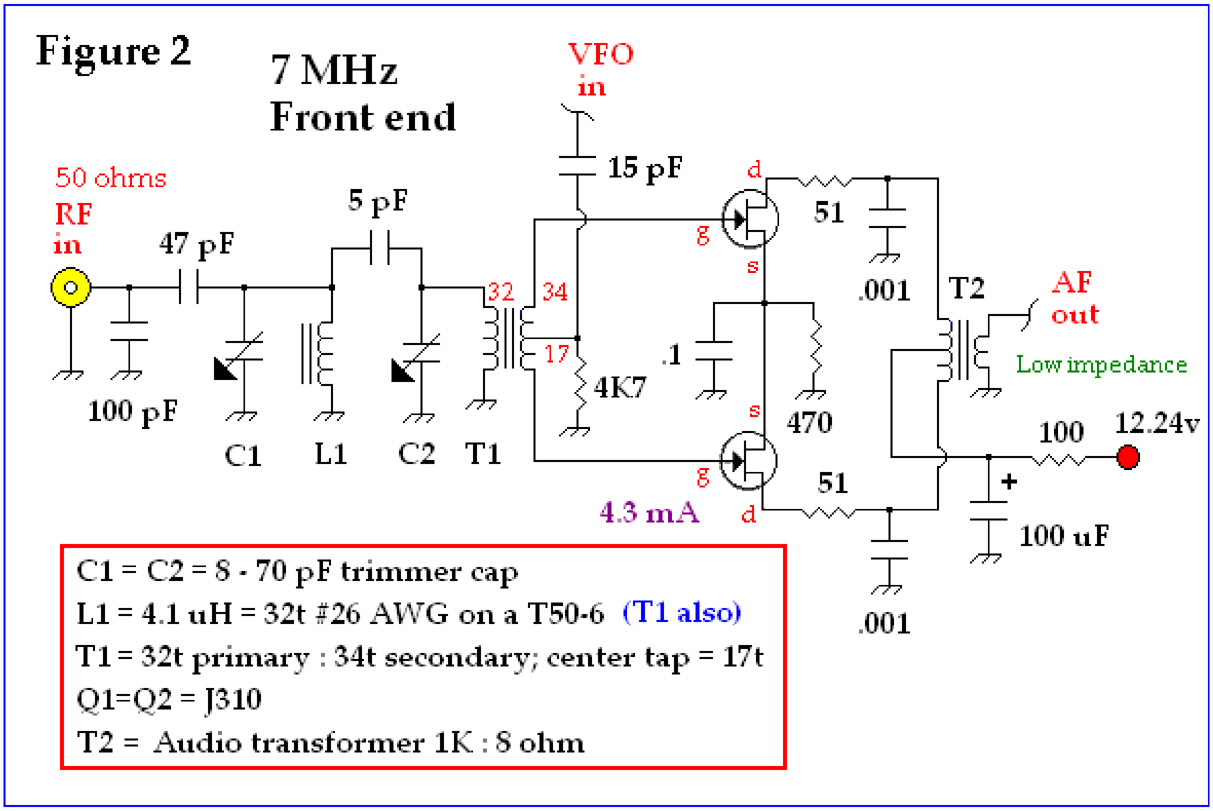
The first stage built was the VFO shown above in Figure 1. The oscillator portion is based upon Figure 4.15 from EMRFD. The VFO resonator tank is isolated from the JFET by tapping down as shown. This is an outstanding VFO topology. See this web page for a few more details and a coil tap calculator. I favor high L to C ratios in my RF tanks, although this does not affect the VFO function. The tapped inductor in this oscillator allows you to use a high RF voltage (low C + high L) while still keeping the FET gate AC voltage at a reasonable level. The buffer amp was designed for high output power and supplies nearly 5 volts peak to peak to the product detector local oscillator port. You can vary the output voltage by increasing or decreasing the 15 pF coupling capacitor for use in other projects.
To peak the L2 tank trimmer capacitor, use a scope, RF voltmeter, or temporarily connect a 10K (or greater value) resistor load to ground via a 10 - 47 pF output capacitor and adjust this capacitor while listening with a nearby CW receiver. (Use a short piece of wire as an antenna.) Additionally, you could also peak this trimmer cap while listening to a CW signal with the completed 2 Bravo receiver. It takes around 100 pF to resonate the L2 tank at 7.040 MHz in case you are wondering.

Since air variable capacitors were used for tuning and to set the band edge, Q is high and frequency stability is excellent. My 1 hour frequency drift was 50 Hertz uncovered. The high RF energy in the tank circuit results in low noise. The L1 taps also allow the use of a 5 pF gate coupling capacitor rather than the hard to locate 3.3 pF cap used in many example VFO schematics. With different buffer/amps as required, this is now my number 1 VFO topology and it is nothing short of stellar. Note that the 100 uH RFC can be wound with 15 turns on an FT37-43 ferrite torroid, or replaced with a fixed value choke.



In the above photograph is the VFO bread board. I used 26 gauge wire for the inductor and took my time to make sure the wire was laying flat on the T68-6 torroidal core. You can pull the wire tighter if you wash your hands before winding.

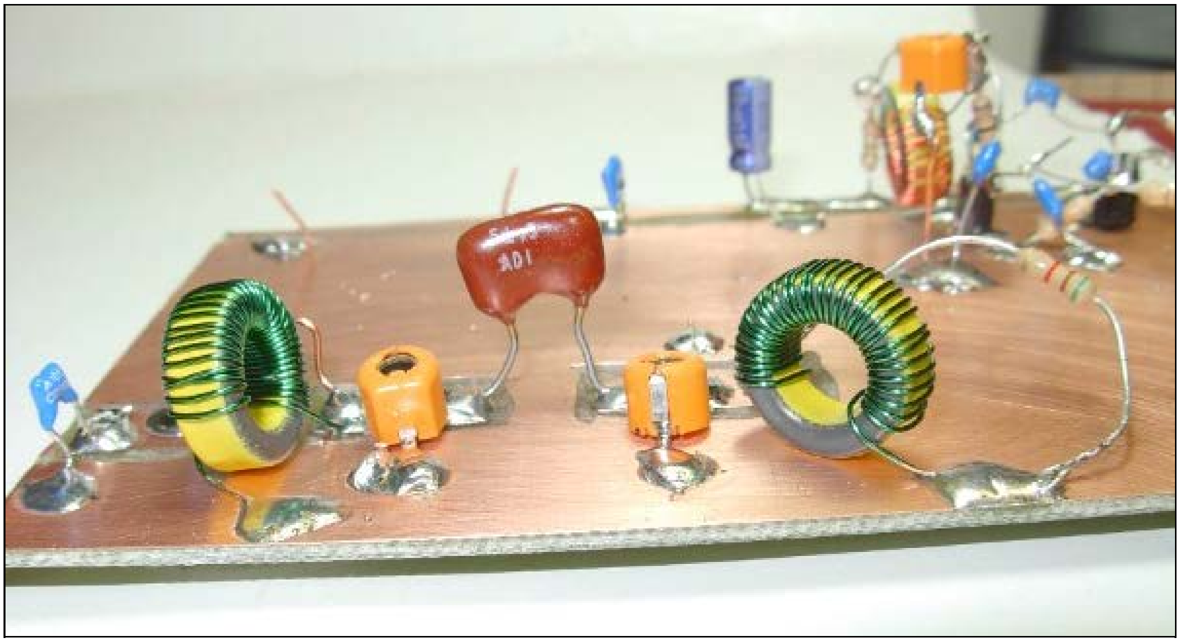
Band pass Filter and Product Detector



C1 = C2 = 8 - 70 pF trimmer cap
L1 = 4.1 uH = 32t #26 AWG on a T50-6 (T1 also)
T1 = 32t primary : 34t secondary; center tap = 17t
Q1=Q2 = J310
T2 = Audio transformer 1K : 8 ohm

Please refer to Figure 2. The second stage constructed was the double tuned band pass filter. You will need about 50-54 pF to resonate L1 and T1 at 7.040 MHz. The C1 and C2 values chosen are thus perfect for tuning the 40 Meter CW band. I peaked my particular front end filter at a center frequency of 7.025 MHz using a 50 ohm output impedance RF generator and then did some fine tuning with an antenna connected after the receiver was constructed. You may also just tune C1 and C2 for maximum signal strength when listening to band noise and QSOs. Filter bandwidth is sufficient to cover the whole CW sub-band. No AM broadcast band radio was heard during several nights of testing.

The product detector is single balanced for improved port isolation and BCB rejection. Lay out your circuit to try to achieve symmetry. The schematic calls for J310s. I built the first prototype with MPF102 that were matched for Idss. To find two with the same Idss, I had to measure 16 transistors! This is too painful, and I recommend just using a pair of J310s. The words "matched" and "MPF102" should not be used in the same sentence! Ideally, your J310s should be matched, however, the process should not take as long as for MPF102 JFETS. The choice is yours to make. T1 is a little tricky to wind, however, your best effort should be good enough.
Some builders will be unhappy with using a audio transformer (T2), however, they are still in catalogs and online stores, or can be harvested from an old transistor radio. CB radio modulation transformers are also a possible source. A higher impedance audio transformer, will likely give even more conversion gain. Without the 51 ohm drain resistors, oscillations occurred in my bread board.



Above. A temporary 5K1 (5.1K) resistor was soldered across the second tank for testing when the front end filter was designed on my work bench.

Audio Pre-amplifier

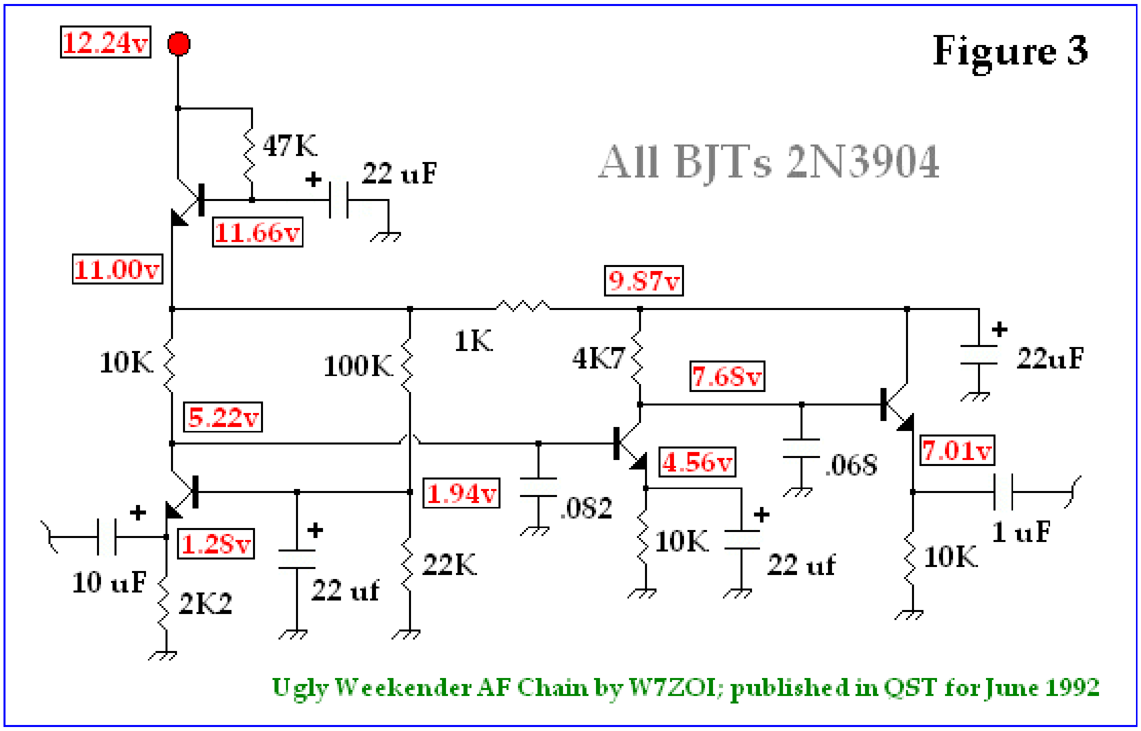


Figure 3

All BJTs 2N3904

Ugly Weekender AF Chain by W7ZOI; published in QST for June 1992

To match the low impedance winding of the audio transformer, a common base amp topology was chosen. I decided to use a favorite circuit; the

audio chain from the first amateur band receiver that I ever built -The Ugly Weekender. The final common emitter feedback amplifier from the original schematic was omitted as the 3 stages above provided enough voltage gain. The Figure 3 amplifier is worth studying. It is difficult to DC couple audio amplifier stages and not end up with your second and/or third stage in saturation. This example of good design by W7ZOI illustrates how to do it. The third stage, a common emitter amp is a level shifter and drops the DC voltage back down, although this stage is AC coupled to the volume potentiometer. For lower noise, you could AC couple the first common base amp to a 5532 op-amp, although, this would reduce the popcorn factor a bit. Do not expect ear blasting voltage gain from this humble circuit. It provides reasonable drive to the power amp stage.
The 0.82 and 0.68 uF capacitors shunt any detected RF energy to ground and also provide some low pass filtering. The original schematic called for 0.1 uF capacitors and which values work the best is yours to decide. Polyester film type capacitors were used in the bread board.

Photographed above is the Figure 3 bread board. You can also see the VFO buffer/amp and the audio transformer. I tried several AF preamplifiers, but preferred Figure 3 to all others.

Audio Power Amplifier

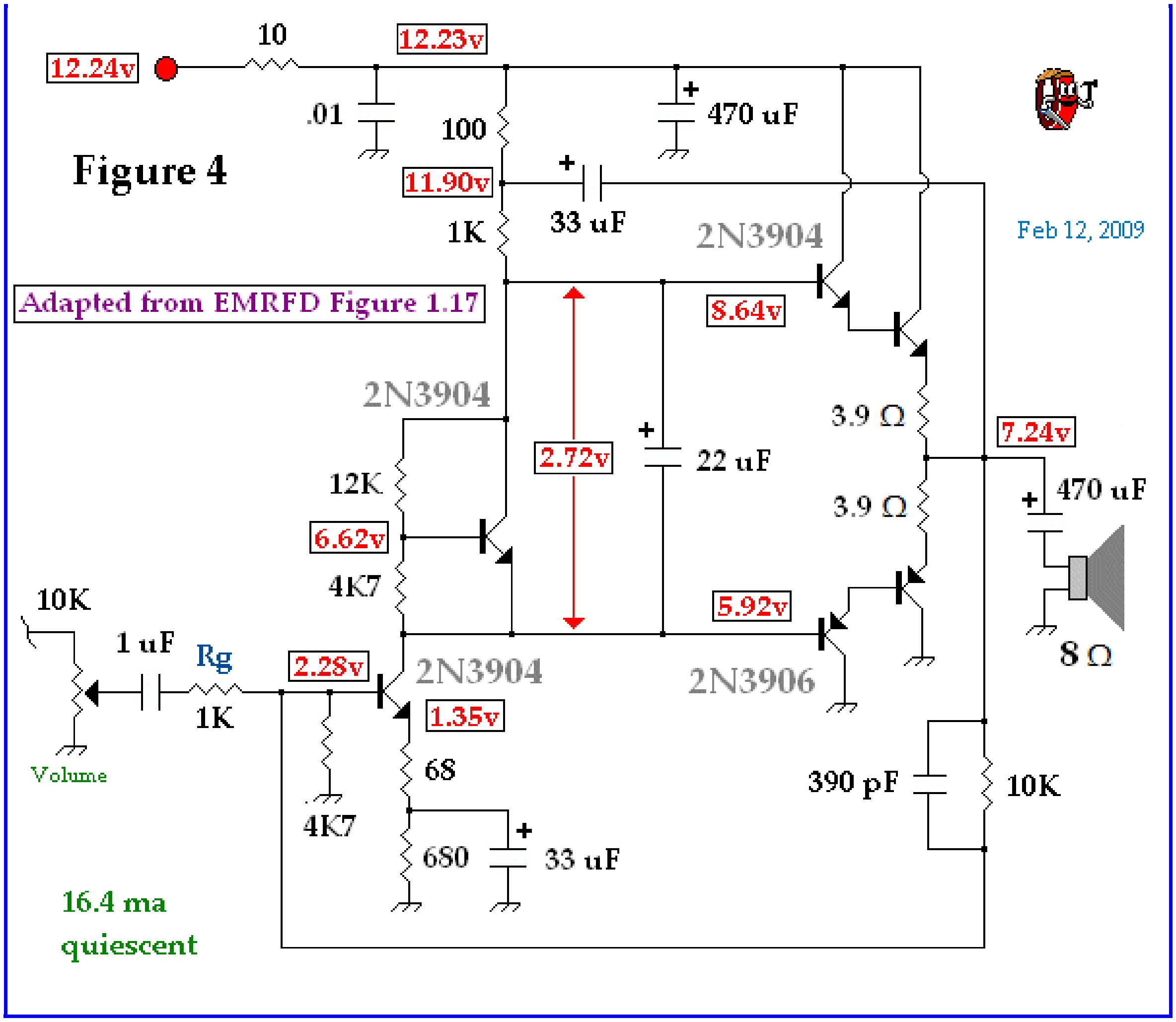
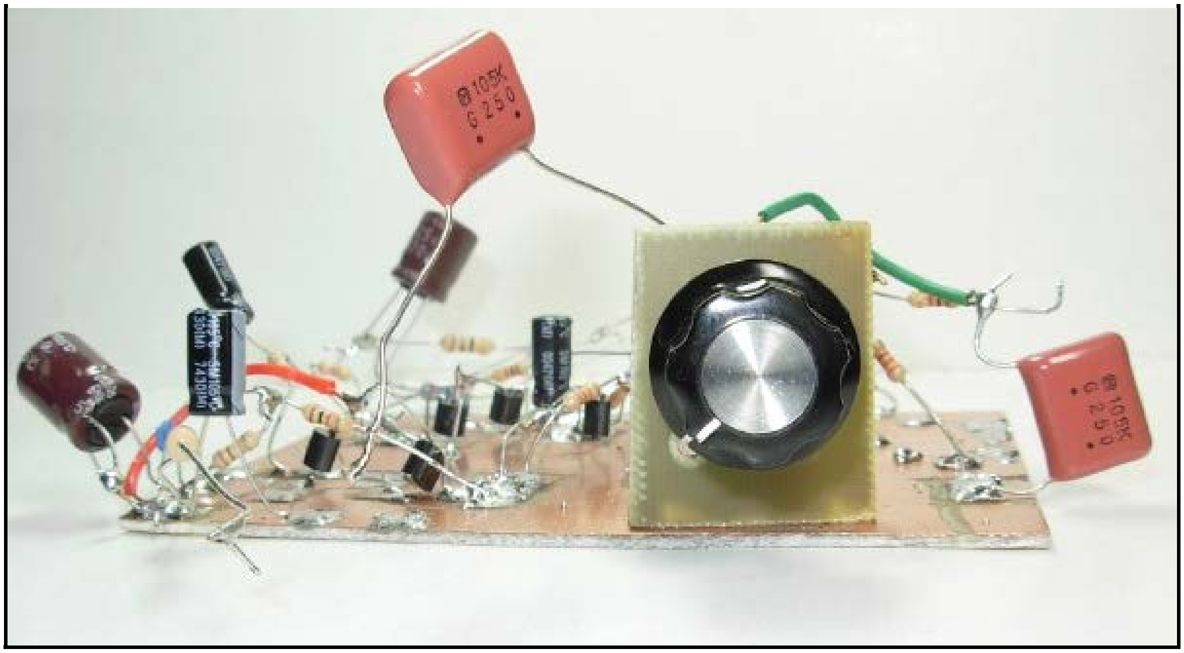
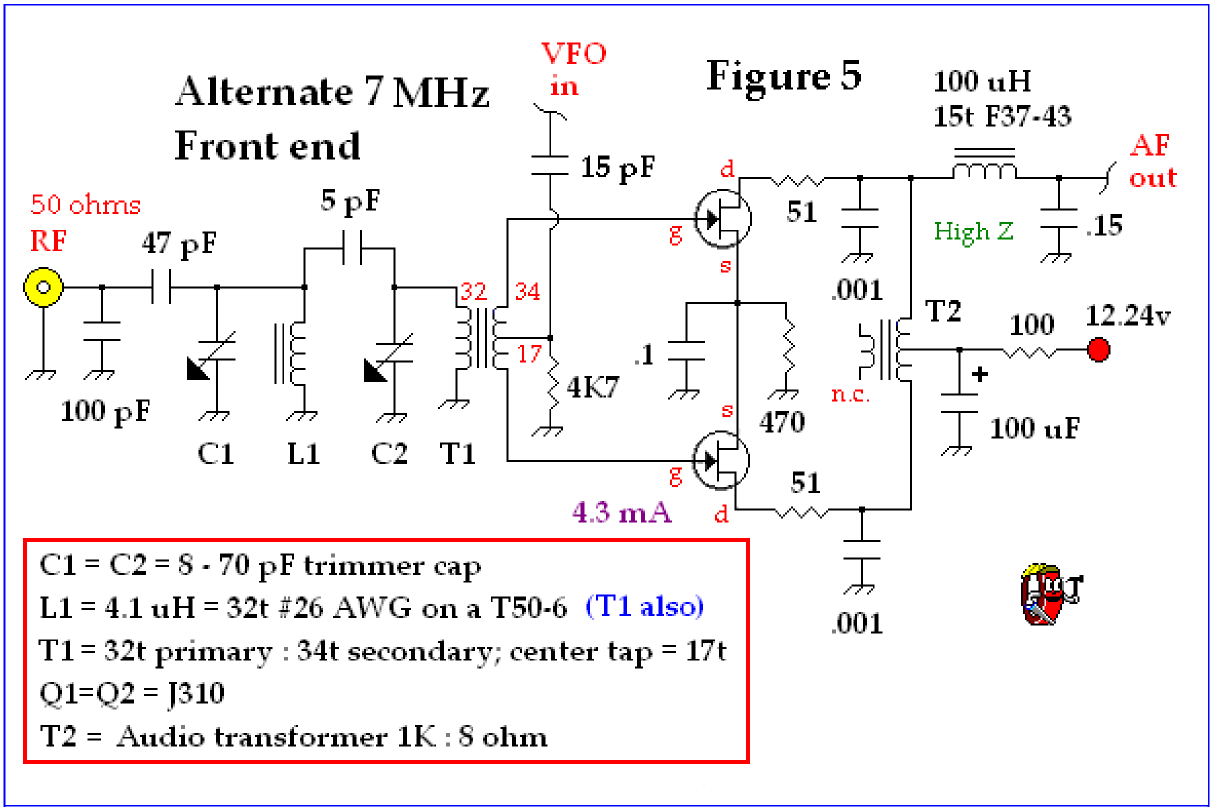


Figure 4 provided 3 nights of experimentation. The base circuit for this amplifier was Figure 1.17 from EMRFD. To increase power gain and reduce harmonic distortion, Darlington configured emitter follower pairs were employed. This worked, except the power followers were under biased and had serious crossover distortion. To remedy this, the amplified diode (level shifter) bias was increased until a sine wave was seen on the oscilloscope. This was achieved by replacing the 10K resistor (R6 in the original schematic) with a 4K7 ohm resistor. The next task was to try to increase the voltage gain. Rg in EMRFD Figure 1.17 called for a 3K3 resistor. Rg was dropped to 1K; this worked. The degenerative feedback on the common emitter amplifier of EMRFD Fig 1.17 was also dropped somewhat. Next some bootstrapping feedback was AC coupled to the collector of the main common emitter amplifier. Each of the 2 collector resistor values was changed around and the outcome was recorded. Ultimately the 100 ohm plus 1K ohm resistor series pair was chosen and provided a boost of 0.85 volts peak-to-peak clean voltage gain to the output waveform. The power follower emitter degeneration resistors were also decreased from 22 to 3.9 ohms. The result is a low distortion power amp with about 150 mW of clean average power output. This receiver is not super loud, but it is reasonably loud and the audio is bell clear. If you use this amplifier stage in other projects that have a higher gain pre-amplifier, I recommend keeping Rg at 3K3 ohms as this amplifier will likely exhibit lower distortion characteristics.



Shown above is the Figure 4 bread board. A 10 ohm resistor was used to decouple this stage. Without the resistor, audio oscillations at around 850 Hz manifested when the volume was greater than about half way up. The voltage drop across the 10 ohm resistor is trivial. You may have to increase this resistor value if you experience instability. Expect all amplifiers to oscillate and decouple them accordingly. The 390 pF feedback capacitor is required. The Figure 4 amplifier exhibits greater gain as frequency increases and in a direct conversion filter with no low pass filtering, this would be very harsh indeed. Feel free to experiment with the value of this feedback capacitor. Kudos to W7ZOI for the EMRFD Figure 1.17 schematic which serves as a great specimen to inform and challenge us experimenters. The original common emitter amplifier (Q1 in EMRFD Figure 1 .17) bias current is perfect and although I tried increasing and decreasing it, I returned to the originally specified bias resistor values.

Additional Outputs



C1 = C2 = 8 - 70 pF trimmer cap
L1 = 4.1 uH = 32t #26 AWG on a T50-6 (T1 also)
T1 = 32t primary : 34t secondary; center tap = 17t
Q1=Q2 = J310
T2 = Audio transformer 1K : 8 ohm

Figure 5 depicts the product detector with a high impedance output. From my experiments at least, it was better to use the low Z coil for improved product detector balance and audio voltage gain.

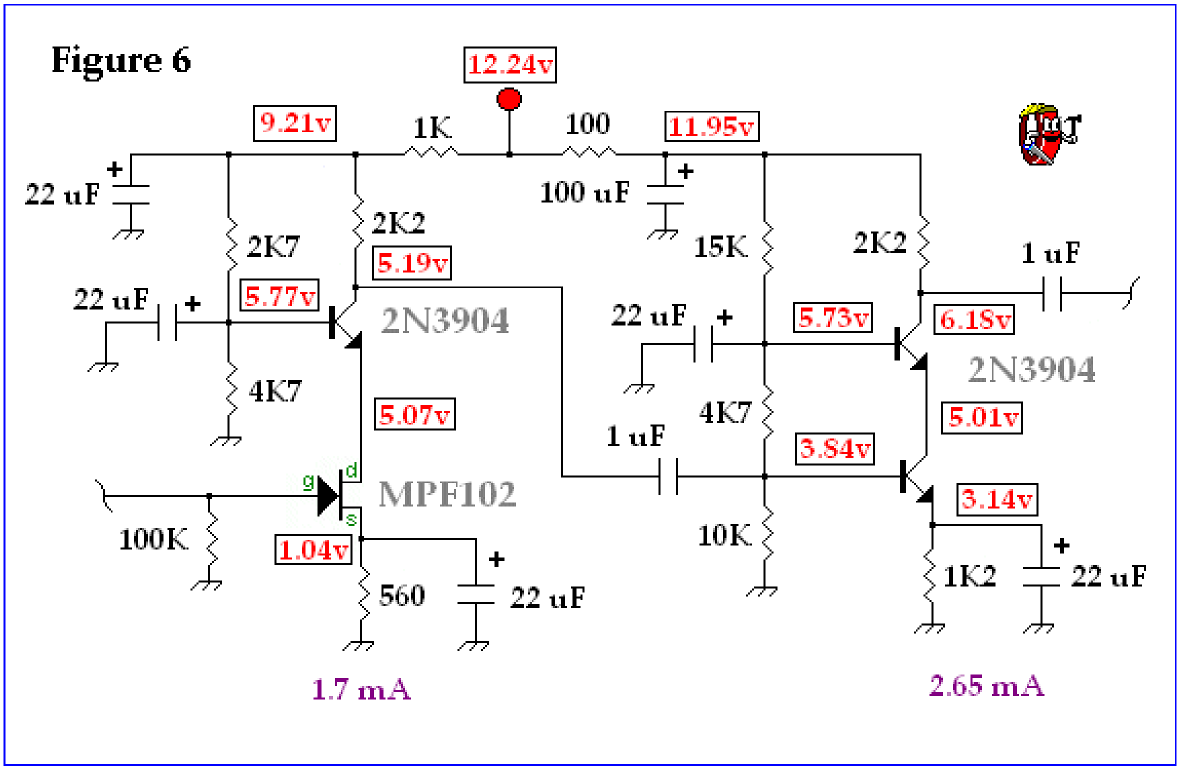
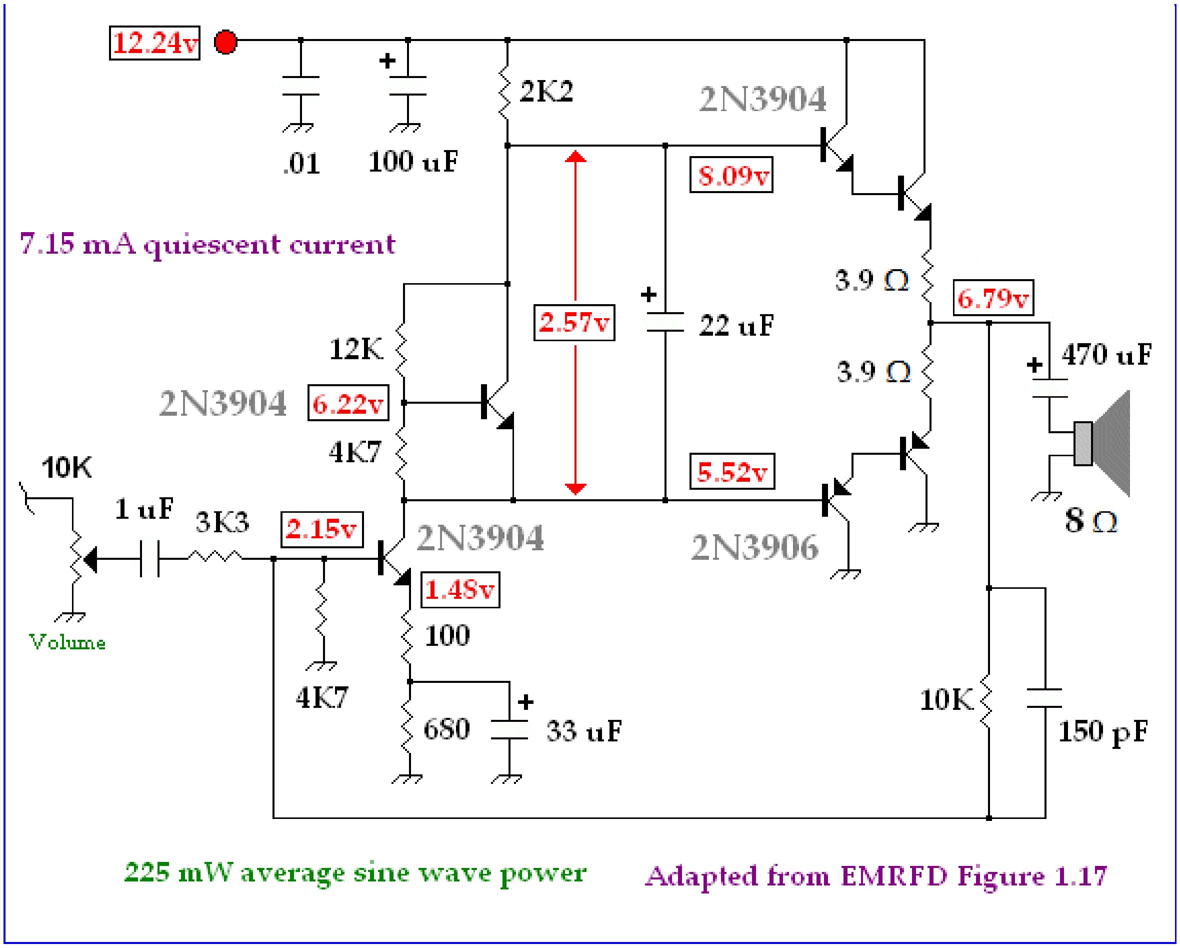


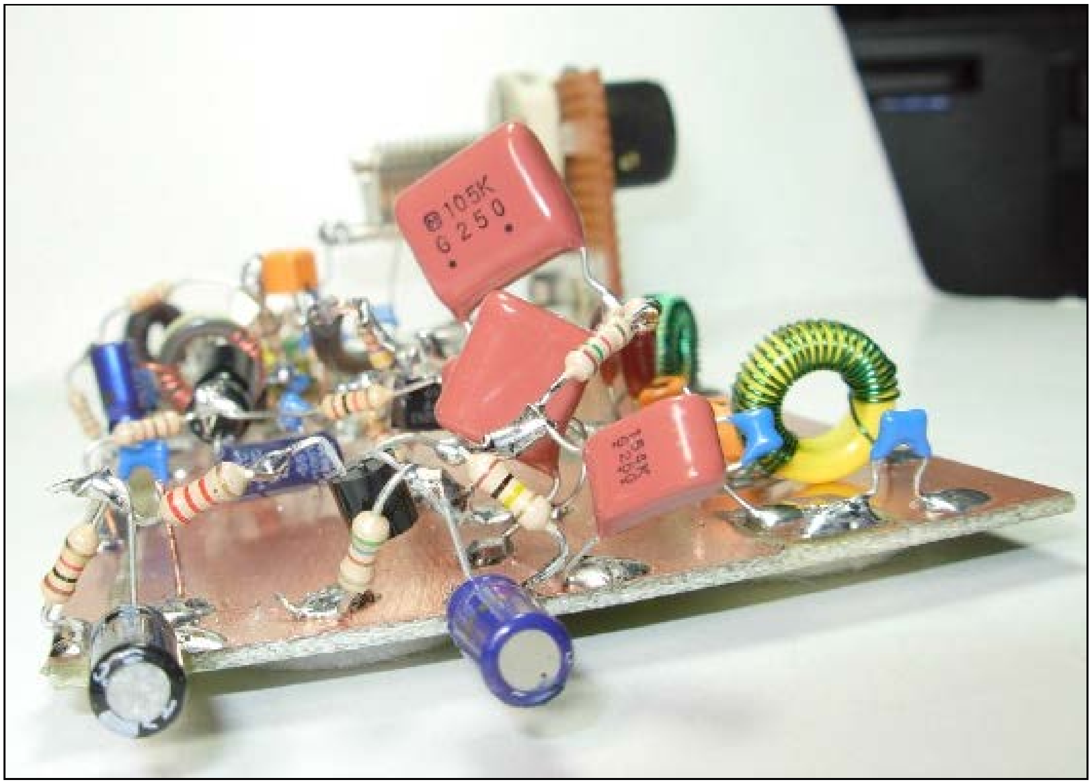
Figure 6: A high impedance input audio stage. The first stage is a hybrid cascode. The second stage is common emitter, common base cascode. Care was taken with transistor biasing to try to optimize distortion characteristics. Certainly, I am a total novice with such amplifiers and more time on the bench and also with computer simulation is required to better understand these amplifiers. At any rate, the schematics with DC voltages are posted for others to study and hopefully improve.



225 mW average sine wave power

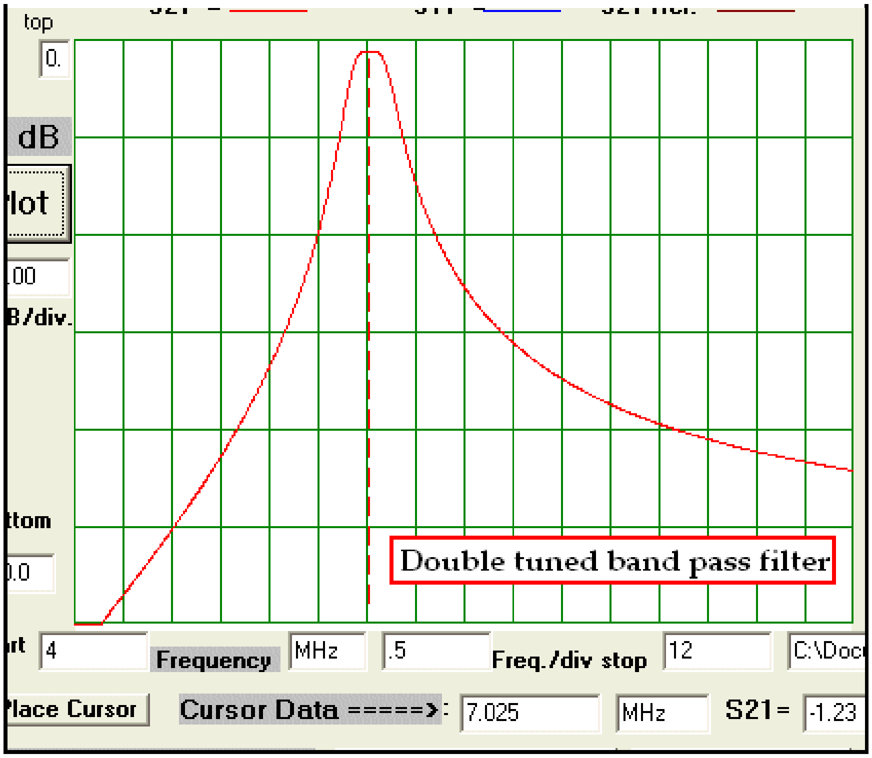
Adapted from EMRFD Figure 1.17

I have 3 adapted versions of the EMRFD Figure 1.17 amplifier in my note book. This is my favorite and has the greatest clean maximum average power output of all of the 3 versions. This power amplifier somewhat lacks sufficient voltage gain for the 2 Bravo receiver (with its relatively low gain audio pre-amplifier) and thus Figure 4 was chosen as the more suitable power amplifier. You could lower the 3K3 resistor to increase the voltage gain. This circuit begs experimentation.



A couple of low pass RC audio filters were tried but later abandoned. One filter with its 1 uF AC coupling capacitors is shown in the photo above. It is quite an experience to hear an unfiltered direct conversion receiver. I love the purity. This is okay for an experimental or casual receiver, but not for a contest rig. Low pass filtering is definitely required in that context.

A different angle photo of the VFO bread board. My "build most of the project on 1 copper clad board" construction technique is not really suitable for "a keeper" receiver. VFOs should ideally be in a shielded box. Proper construction techniques and grounding ideas for DC receivers can be found in EMRFD, so they are not covered on this web page.



rt 4 Frequency MHz .5 Freq./div stop 12 C:\Doc

Cursor Data =====>: 7.025 MHz S21= -1.23

A GPLA simulation of the front end band pass filter centered at 7.025 MHz.

Amateur and Short Wave Radio Electronics Experimenter's Web Site

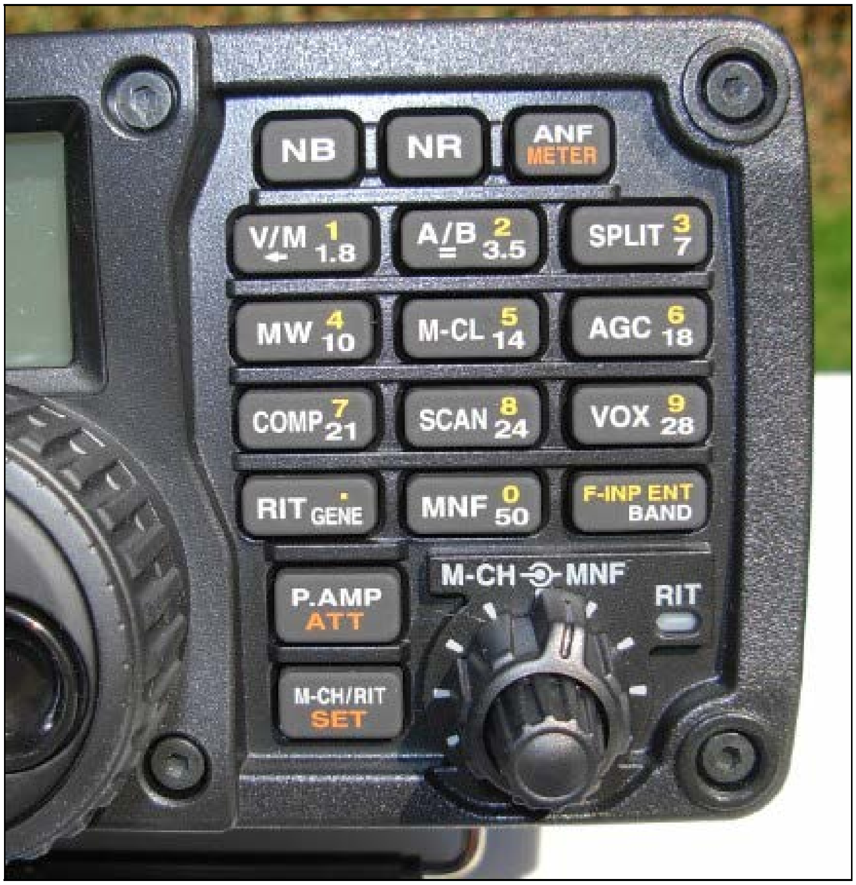
ICOM IC-7200 Listening Tests and Observations

Introduction

In Spring 2009, I evaluated and photographed the ICOM IC-7200 transceiver. For this web page, only its receiver was evaluated for HAM and SWL purposes by using my eyes and ears. For technical evaluation, please click on these assessments written by Adam, VA7OJ/AB4OJ or Peter, G3SJX . Additionally, eHam.net has a review web page to consult and the ARRL publication, QST for June 2009 has a review. With my modest camera and lighting equipment, it was difficult to well photograph this very dark colored transceiver indoors.

I like ICOM radio equipment and am therefore biased in my review. Please consider trying this and any other radio out before you purchase it. Our needs, expectations and budgets tend to be uniquely different. Like others, I would rather own the new ICOM IC-7600, Yaesu FT2000D or Elecraft K3, however, my budget does not allow this.

It appears that the target audience for this transceiver is as follows: portable or field/emergency communication usage and/or it is oriented towards entry level HAMs or perhaps those wishing a modern DSP-filtered back up rig. My review is from a SWL/HAM perspective.



General Thoughts

Although modern and stylish, this radio is very easy to use. The owners manual is well written with clear examples of how to perform the various setting changes. After reading the manual and and trying the radio buttons and knobs out, I pretty much mastered receive operation on my first night. The LCD display is small, however is reasonably adequate considering that this whole radio is compact. I believe some operators will have difficulty with this diminutive display. The S-meter is a little difficult to see. You can push a front panel button which announces the S-Meter reading, set frequency and mode in English or Japanese language if you can't quite see the display at an odd viewing angle.

The VFO knob has adjustable resolution and works well, however, it lacks that silky smooth/weighted feeling of many other radios including the R75 receiver. The ICOM engineers had there work cut out for them; include all the modern interference fighting features; place them in a small chassis; make it easy to use and come in on budget. They did it! This radio is full of useful interference management tools. One example is the (DSP) manual notch filter; it works superbly. There is no FM mode. I imagine by dropping FM mode capability , the designers were able to use the specified 6 KHz roofing filter at the second IF which has the potential to improve dynamic range at close-in signal spacing for some modes. When I started in Amateur Radio many years ago, roofing filters were never mentioned, but have become a huge marketing lever and seemingly a topic of much confusion.

DSP IF Filtering

The DSP IF filtering works very well. Does it function better than more traditional crystal IF filtering? Yes and no. It is a question of compromise for me. The greatest DSP attribute is that there are no expensive crystal filters to purchase. Additionally, you can customize the desired IF filter bandwidth (wide, medium and narrow) for each mode and also set a "hard" versus "soft" filter shape. The soft shape equates less ringing and potentially less listener fatigue than the hard setting. I have never liked listening to CW signals through stiff, 6-8 pole 250-500 Hz crystal filters on any receiver, so for me, adjustable DSP IF filtering is preferable. DSP IF filtering is not perfect as authors like Rob Sherwood, NC0B have presented, however, ICOM have a pretty good DSP platform out now and I am pretty sure their latest technology from the IC-7600/PRO 3 on down have also been used in this receiver. After all, you can always try to find an old Drake and order some crystal filters for it, or spend significantly more cash on a high-end transceiver if you need better performance.

Shown to the the left is a few front panel controls and the front firing speaker. This speaker is reasonably nice sounding. There seems to be less harmonic distortion and audio gain than the R75 receiver. The radio front panel controls are well thought out and seemed intuitive to use after reading the manual and/or just trying them out. The various hardware components such as potentiometers, rotary switches, microphone jack, PL-259 jack etc seem not to be of the highest quality. When you compare such components to that used in their older designs, there is

evidence of modern cost containment. I have seen many reviews describing this transceiver as rugged. Certainly the diecast frame is solid, and I hope the forementioned hardware is as well. There is only one antenna connection. I believe that any radio offering the 6 meter band should offer two SO239 antenna connections, however, this likely would have crowded the back panel layout and increased cost for the RF in/out switching.

The VFO knob (shown to the right) looks and feels a little cheap. Functionally, however, the knob spins very well and precise tuning is easy to perform. One of my first tasks was to input various the frequencies I use for both amateur and short wave listening into the (201 maximum) memory slots. This was very easy to do and each memory channel also stores the mode and filter setting.

AM Reception (R75 versus IC-7200)

The IC-7200 is a good short wave receiver. With a high quality external speaker connected, pleasant and warm sounding AM audio may be heard; however it is not Hi-Fi. That is; the IC-7200 is not an audiophile AM receiver. There is no synchronous AM detection and no 15 KHz wide IF filter for example. For AM reception, the user may choose from an 8000 to 800 Hz IF bandwidth in the 3 switchable filter settings. During testing, I set my wide filter setting to 8000 Hz, my medium setting to 6000 Hz and my narrow setting to 3000 Hz for AM reception. Of course these bandwidths can be further adjusted at any time. Local MW broadcast radio sounded great with an 8000 Hz IF bandwidth during testing. The various AM reception appropriate interference controls such as pass band tuning (PBT), automatic notch filter (ANF) and digital noise reduction are all configurable or adjustable and I found the ANF and digital noise reduction helpful when listening on the crowded 49 meter band during a rain storm.

I performed A/B comparisons with the IC-7200 and the R75 simultaneously hooked to the same antenna. For AM reception, they are comparable with the R75 having marginally better sounding (more Hi-Fi) audio. I had the R75 IF filter bandwidth set at 15 KHz and the IC-7200 IF bandwidth set at 8 KHz (both at their maximum IF bandwidth).

CW/SSB Reception (R75 versus IC-7200)

In CW mode, both were set to have a 500 Hertz IF bandwidth.(The R75 had the FL-100 model 500 Hertz crystal filter in its 9 MHz IF slot). The IC-7200 was quieter, clearer and less overloaded by adjacent stations during pile ups. There were occasionally weak signals I was able to copy on the IC-7200 that I could not even hear on the R75 due to noise. The IC-7200 is a joy to use on CW; really fantastic.

On SSB, I found both receivers fairly comparable, but the IC-7200 was better for pulling weak signals out of the noise as the noise floor was a little quieter and the audio a little more crisp. My wife also agreed with me in her "independent tests" of the CW and SSB reception. I am certain to catch flak because my subjective comments are based upon listening tests and not measurements. It is interesting to note that in almost every aspect of our lives, preferences are made using 1 or more of our 5 senses. From listening, observing and also reading the reviews of others, it seems ICOM has a hit with the IC-7200.

Mouse over the images below left to view a full size photograph

The IC-7200 in the sun.

Rear view of the transceiver.

Top angle view. There are ventilation holes at the top right hand side. You can just see the ventilation screen just next to the right top panel screw.

Reverse angle top view. The military look is attractive.

The IC-7200 and R75 side by side.

Another photograph of the 2 radios from the front angle for comparison. The LCD display size difference is quite obvious in this shot.

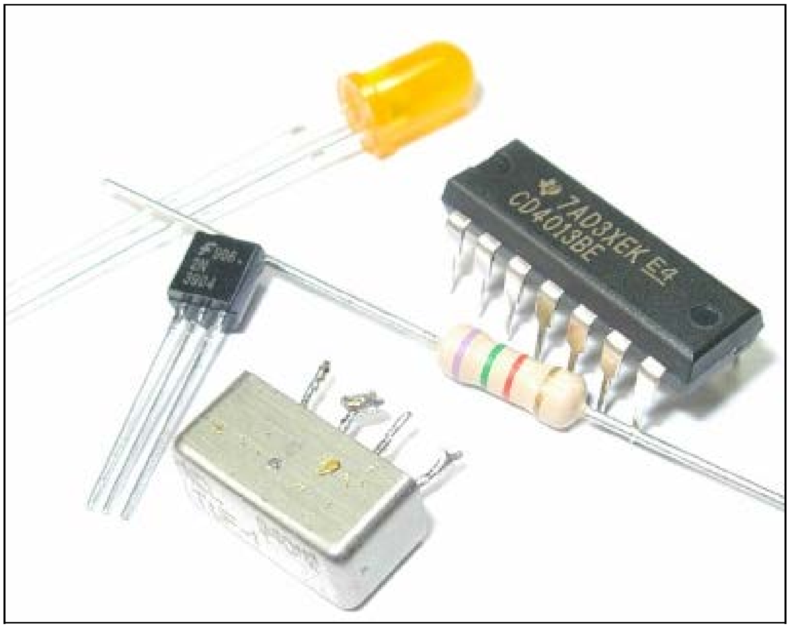
The receivers at a right angle. The IC-7200 is a glad update, although, I would likely not upgrade from the R75 to the IC-7200 just for receiving purposes. If I was considering choosing the IC-718 (transceiver version of the R75), the IC-7200 is worth the extra money. It is also an excellent transceiver consideration for an entry-level rig, for back up purposes. or for the budget-minded operator.

Conclusion

The IC-7200 is a serious choice for amateurs seeking a good, modern HF plus 6 Meter transceiver. It is more than adequate as an SWL receiver if you are an amateur radio operator plus SW listener like myself. Please try one out for yourself if you are considering this transceiver.

Amateur Radio Electronic Design

Electronic Hobbyist Circuits



This page will house a collection of brief hobbyist experiments.

1. Pseudo-Random Number Generator

This circuit describes a simple, 6-bit random number pseudo-generator used to study binary counters and in particular, shift registers. Some very basic background information about binary counters and shift registers is provided. In reality there are dozens of different shift register topologies available and it can get quite complex. If you wish to find a good logic tutorial website, I strongly recommend Ken Bigelow's site as it has interactive diagrams. Flip-flops are also covered well on wikipedia and many other web sites.

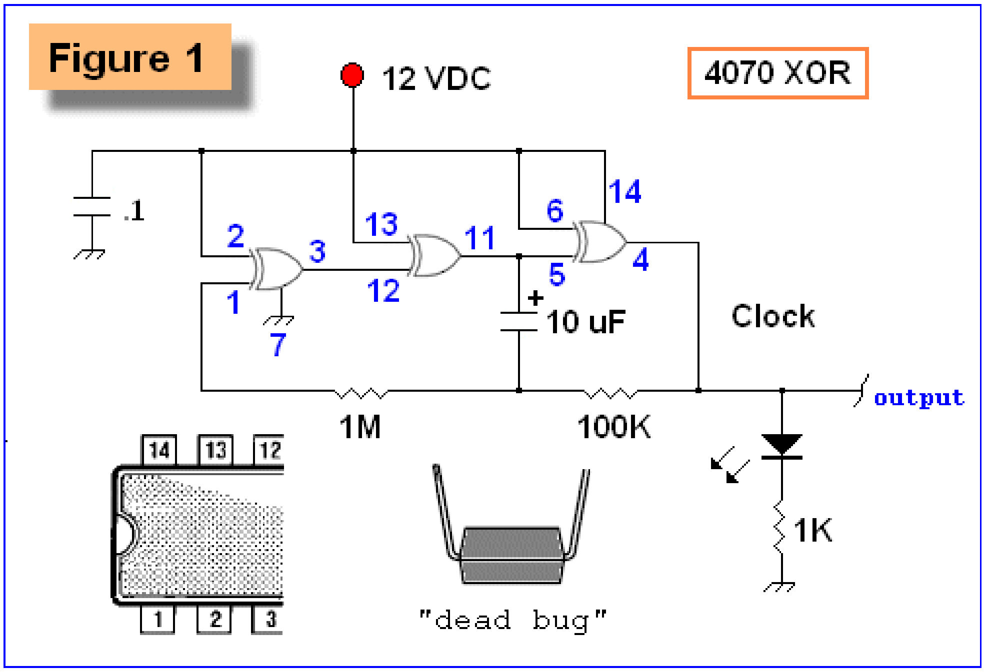
Binary Counter: The circuit most often used as a counter is called a binary flip-flop. The basic flip-flop can be viewed as a toggle switch having either an ON or OFF position. This is the binary state 1 (HIGH) or 0 (LOW). Like the toggle switch, the binary flip-flop has 2 binary states 1 or 0. A binary flip flop counter counts in a sequence such as 0, 1, 0 ,1 etc. A straight binary counter can be built by using 1 or more flip-flops connected in a manner that the binary number stored in these flip-flops will represent the total number of trigger pulses received at the counter input.

Ring Counter: A ring counter has 2 or more flip-flops cascaded so that the output from one flip-flop becomes the input of the next flip-flop. The flip-flops are connected so that all of their outputs are at the binary state 0 except for one flip-flop. By pulsing the input of the ring counter, it will sequentially change the binary state of the succeeding flip-flop from binary 0 to binary 1. The flip-flop that contains the binary 1 indicates the count of this binary counter. The maximum number of pulses that can be counted by N flip-flops is N pulses.

Shift Register: A serial entry shift register is similar to a ring counter, except that the output flip-flop is not connected to the input flip-flop. Like the ring counter, the flip-flops are cascaded so that the output from one flip-flop becomes the input of the next flip-flop. All the set trigger and reset trigger inputs are tied together to form what is called a shift bus. Clock pulses are applied to the shift bus to cause the stored binary information to shift from left to right; one bit position per each received clock pulse. In Figure 2, this serial input/output + parallel output register has its 5th and 6th bits exclusive ORed to the serial input to form a pseudo-random sequencer, which is called a pseudo-random number generator by some.

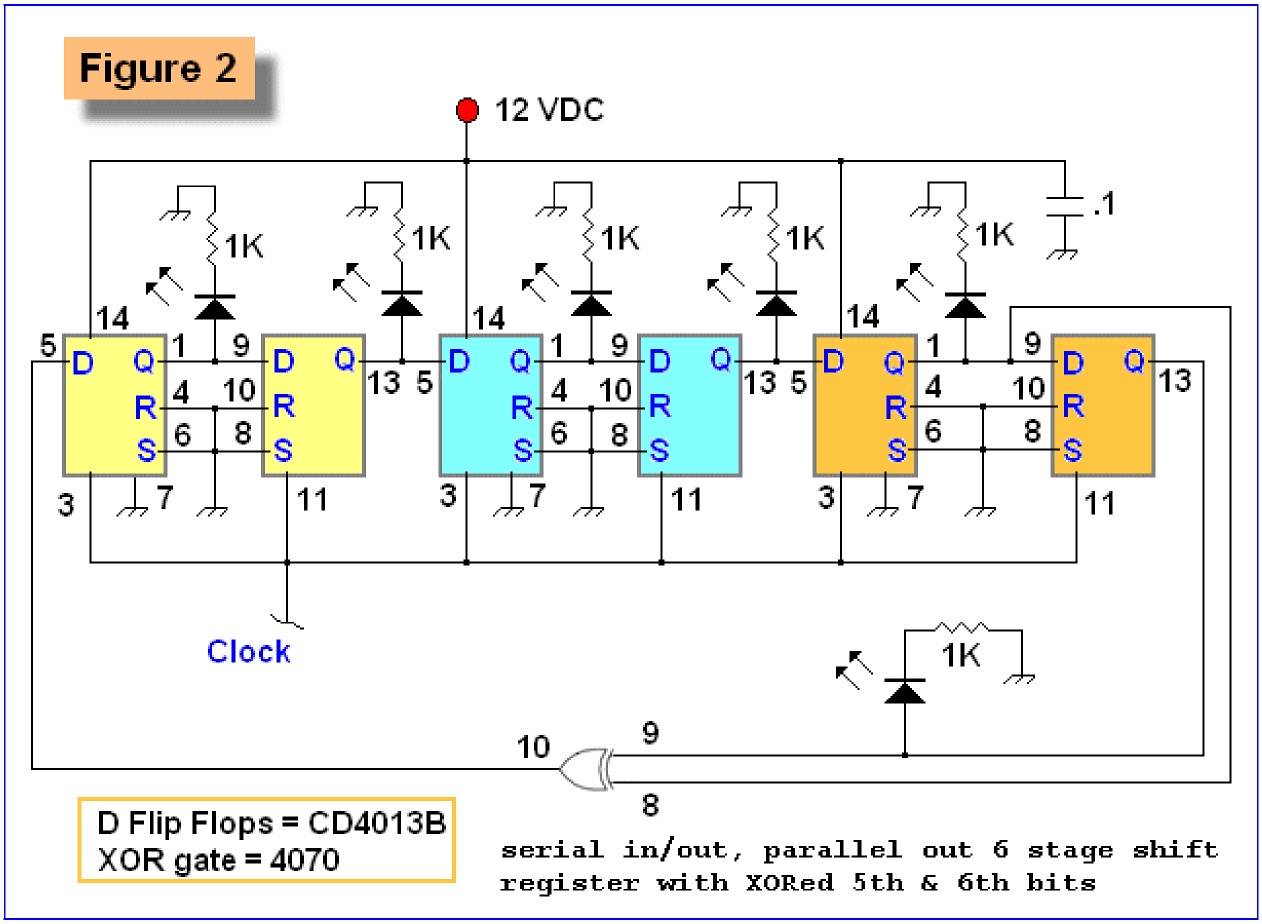
The CMOS logic ICs used were one 4070 XOR (Exclusive OR) and three CD4013B D flip-flops. Junk box LEDs were used to observe the binary state of the clock and each of the 6 bits of the shift register.

Figure 1



Since only one XOR gate is needed for the shift register, the remaining gates were configured to make the clock. These gates are essentially wired up as inverters to form an astable multivibrator with a frequency of about 0.45 Hertz or 27 pulses per minute. Shown above in Figure 1 is the clock schematic and the pin 1 marking for all of the digital ICs on this web page. The output LED is not mandatory, but will instantly tell you whether or not your clock is working. I built this whole circuit using Ugly Construction with the ICs flipped upside down in a "dead bug" fashion. You can increase the clock speed by decreasing the 100K resistor or the capacitor values. F Hertz = 1/ (2.2 * R * C) with R in ohms and C in farads. The slow clock speed was chosen to better observe the digital output of the shift register.

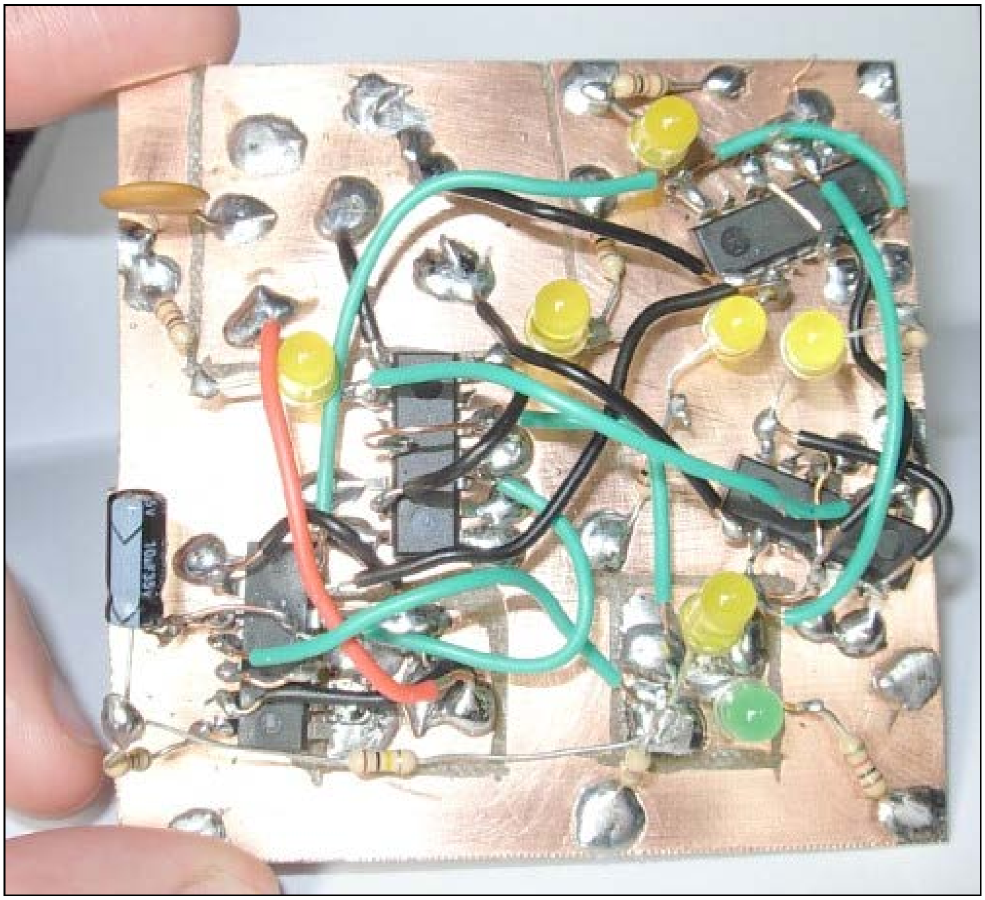
Figure 2



serial in/out, parallel out 6 stage shift register with XORed 5th & 6th bits

In Figure 2 is the shift register. Each 4013 was wired up as 2 cascaded flip-flops and connected to the clock. Power was applied and then a test lead was used to bring pin 5 of the first flip-flop HIGH (connected to 12 volts for 1-2 seconds) . Both flip-flop state monitor LEDS turned ON in sequence with subsequent clock pulses. Afterwards, pin 5 was set LOW (shorted to ground with a test lead for a couple of seconds) and each LED turned OFF in sequence with subsequent clock pulses. The remaining two 4013s were wired up and tested the the same way and then finally the last XOR gate was wired up.

To avoid error, frequent pin counting and a systematic approach is recommended. For example, for each 4013, I soldered the ground pins, wired the pin 14 VDD, connected the clock to pins 3 and 11, then wired up the pin 1 and pin 13 LEDs. Systematic construction techniques are something that you the experimenter can develop and perfect over time. This approach saves time and grief. On some projects, when you have a lot of pins wired up, tracing and repairing an early mistake can be difficult.

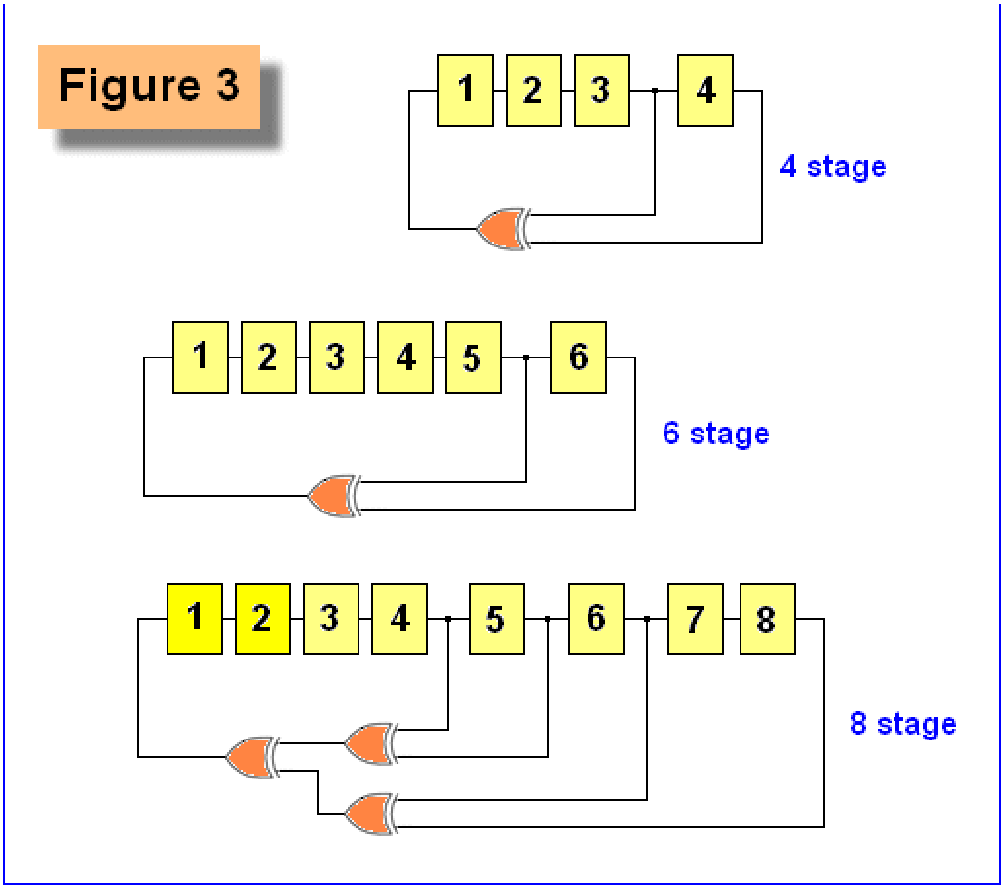


Shown above is a bread board of the entire pseudo-random number generator. I just built in on a scrap of board and did not lay it out so the LEDs were in a row, as I am not going to keep this project. The clock state monitor is the green colored LED. There are 63 possible states or combinations of the 6 bits (111000, 100110, 100101, 000101, 000001 etc.) State 000000 is disallowed and will hang up the shift register. If your clock LED is flashing and no shift register LEDs are lit, then "reset" by momentarily setting pin 5 of the first flip-flop HIGH (momentarily apply 12 volts). Long live the reset switch!

Pseudo-random numbers are now mostly generated by computer microprocessors controlled by software and have applications in cryptography, electronic music, security and many other applications. This "hardware" pseudo-random number generator experiment was really cool and if you want to randomly flash some LEDs, this could be the project to use!

If you are new to digital electronics; (like me) Welcome! Starting small with projects like this one will hopefully lead to increased confidence and problem solving skills for even bigger projects. You can also build the shift registers with J-K flip-flops, but it is more difficult and 4013s or other series D flip-flops are cheap as Борщ (borscht).

Figure 3

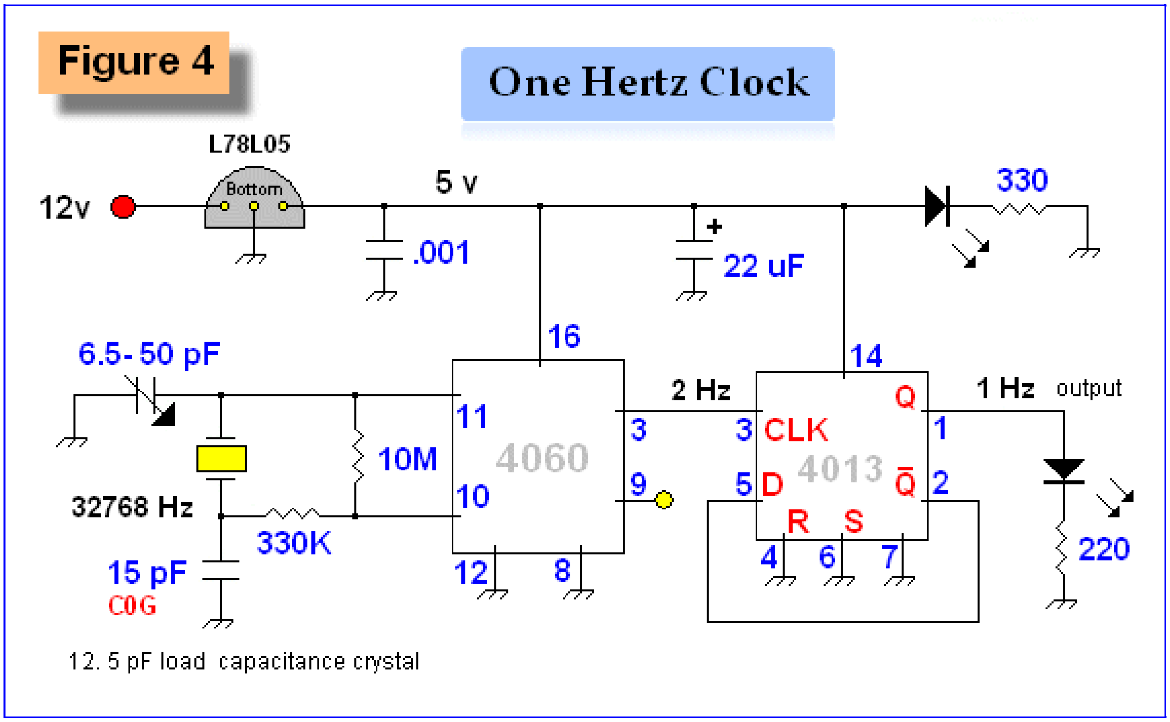


Shown above in Figure 3 is how to hook up the XOR gate(s) for 4, 6 and 8 stage pseudo-random number generators. The 6 stage shift-register is of course, Figure 2 above and is presented for reference purposes. The 8 stage version = 1 byte.

2. One Hertz Precision Time Base

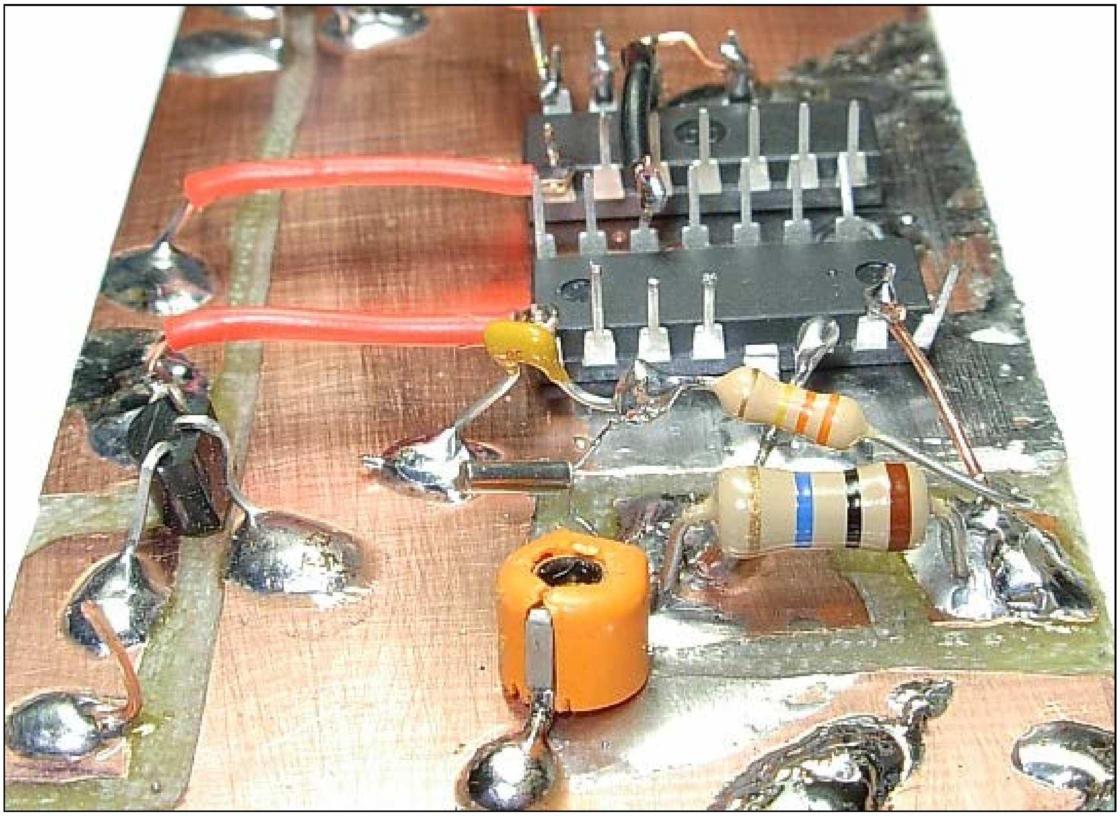
Digital clocks are very interesting. In the past 6 months, 10 -15 RC clocks have been constructed and tested. RC oscillators in the KHz to Hertz range are surprisingly frequency stable. For many projects, a plain RC clock is adequate, however, like in radio design, a crystal controlled time reference is sometimes required. Two examples of projects requiring precision clocks are time of day clocks and frequency counters. Presented is a 1 Hertz clock built from two 4000 series CMOS Logic ICs. Here is a great 4000 series tutorial with pin outs and more.

Figure 4



Shown above in Figure 4 is the complete schematic with an output LED for testing. In the past, the MM5369 17 Stage Oscillator/Divider was popular for hobbyist precision time bases, however, it has gone obsolete. The 4060 ripple counter is a good "modern" replacement, although a different crystal is required. A 32768 Hertz crystal was used and is divided 16384 times to provide a 2 Hz output. The 4060 then drives a 4013 D Flip-flop configured as a divide by 2 to provide a 1 Hz output frequency. Key parts references may be found on the Webmaster's page.

Shown above is the frequency and output waveform when a frequency counter and oscilloscope are (respectively) connected to pin 9 of the 4060 in Figure 4. The 6.5-50 pF trimmer pot is used for calibration. Originally, I used a trimmer cap instead of the fixed 15 pF capacitor shown between the 330K resistor and the crystal. After adjusting this trimmer capacitor for the best looking waveform, I removed the trimmer cap and measured it at 13 pF. I substituted the nearest standard value I had in my parts collection; 15 pF. It was interesting to measure the 4013 output frequency at 1 Hz.



A close up photo of the 4060 oscillator/divider breadboard. The 10M resistor used was a 1/2 watt rated R as I have dozens of these in my parts collection. You can see the tiny cylindrical crystal just above and left of the orange Murata trimmer capacitor. It is oriented horizontally. This is a useful time base for the QRP workshop.

3. 10000 and 5000 Hz Multivibrator Clock

It is fun to occasionally build circuits using discrete semiconductors rather than with ICs. A 5000 Hz digital clock was needed for an experiment. It was decided to use multivibrators for the basic oscillator and a divide by 2.

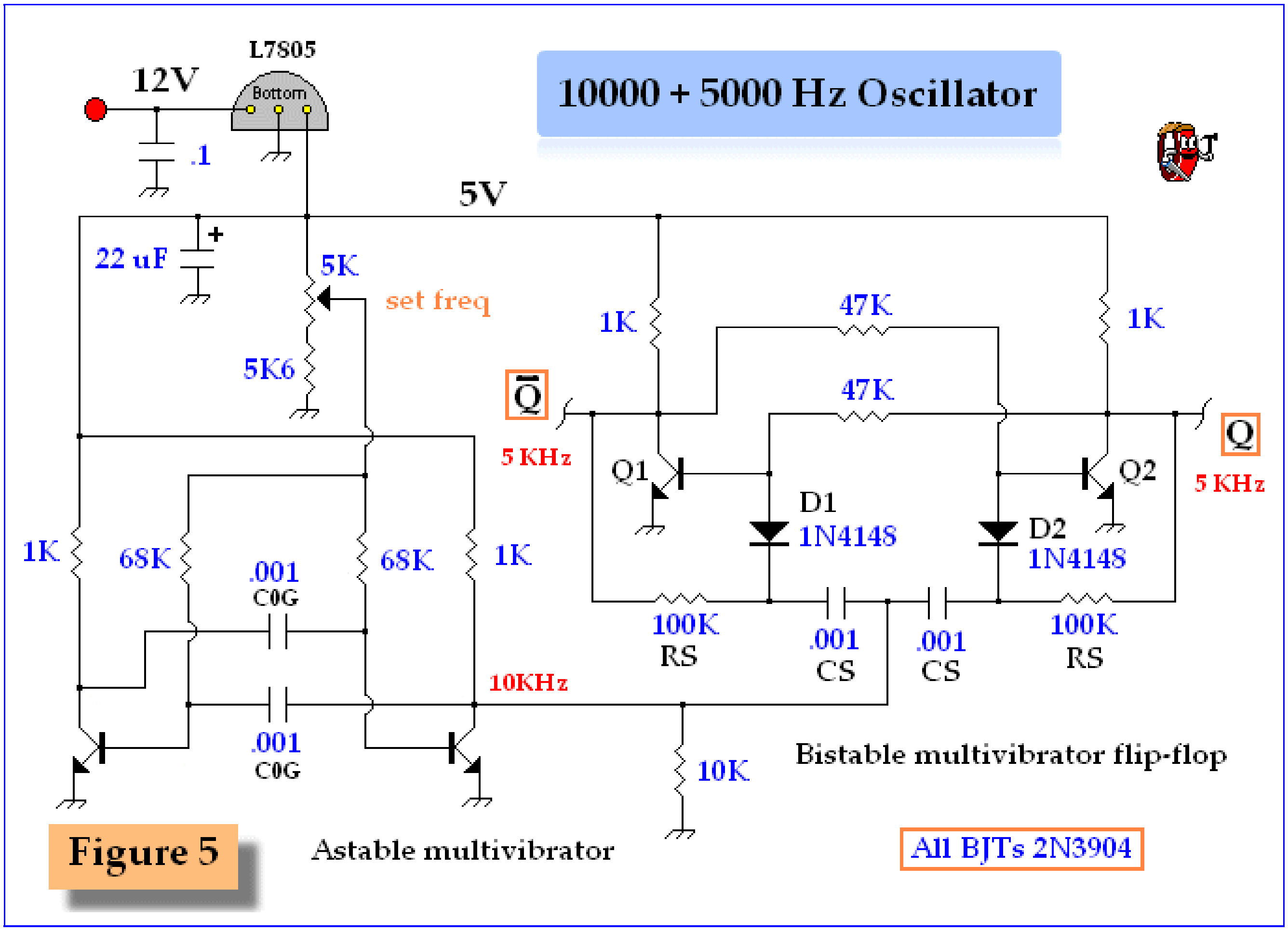


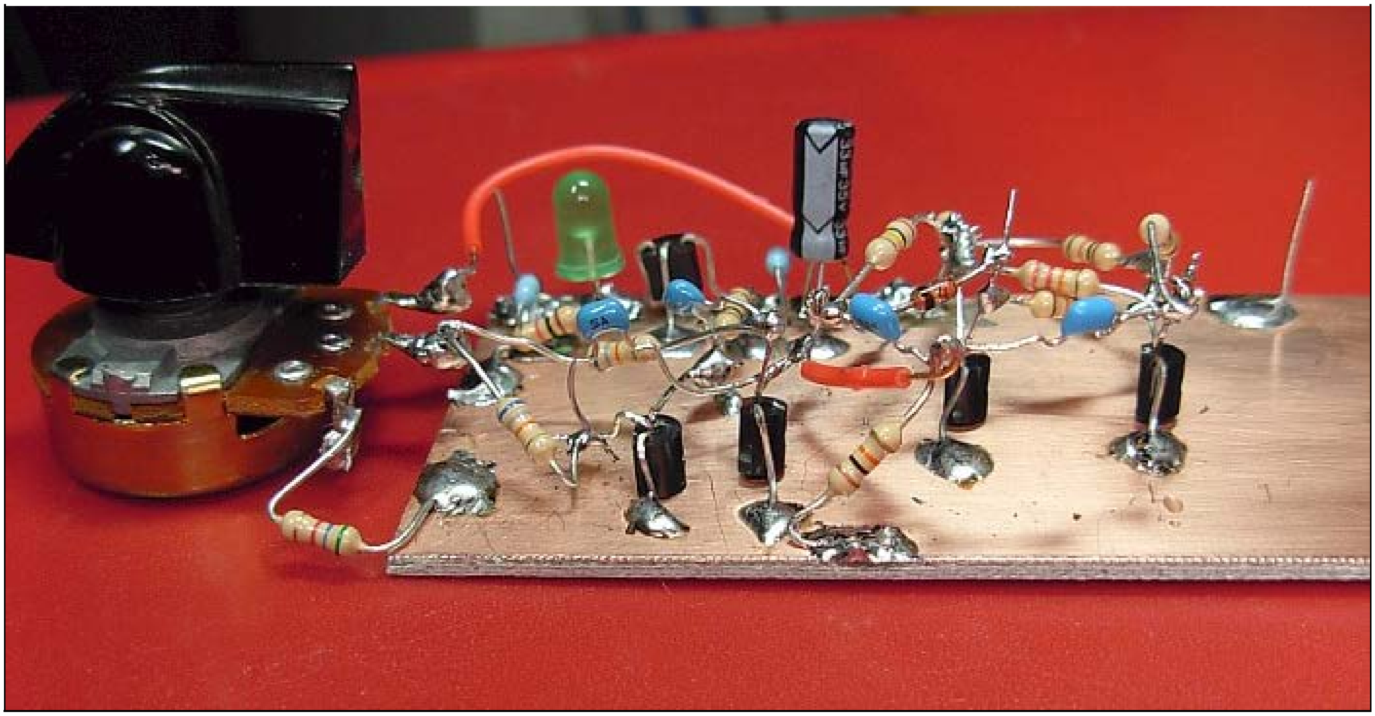
Figure 5

Figure 5 is the entire circuit. The tuning range of the astable multivibrator was about 7060-10650 Hz. The 5K pot was slowly adjusted until 10000 Hz was measured in a frequency counter. Following testing of the astable multivibrator, the flip flop was built and examined. Astable multivibrator function has been discussed previously on this web site.

Please refer to the bistable multivibrator. It is a one input circuit set up for toggle or flip-flop operation. Negative edge pulses applied between the two 0.001 capacitors will cause the binary state of Q1 and Q2 to change to the opposite state. The multivibrator circuit is made up of Q1, Q2 and the 47K and 1K base and collector resistors respectively. The other components D1, D2, the RS resistors and CS capacitors comprise a steering circuit to generate the proper response to the negative edge pulses. When a negative input pulse arrives, it is guided to the base terminal of the ON transistor, but prevented from reaching the base terminal of the OFF transistor.

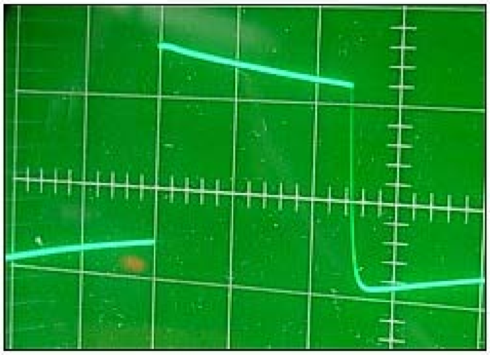
In order to study this circuit at DC, I temporarily exchanged the 0.001 timing capacitors in the astable multivibrator with some 22 uF electrolytic caps to slow it down. Referring back to the bistable multivibrator, let us assume that Q1 is OFF and Q2 is ON. The collector voltage of Q1 is high (cut off). The collector voltage of Q2 is low (saturation). The Q1 collector is connected to the cathode of D1 by the 100K RS resistor. The cathode of D1 is reverse biased by the high Q1 collector voltage and also because its anode is held close to 0 volts by the 47K resistor connected to the collector terminal of Q2. It would take a very strong negative input pulse to forward bias D1 enough to reach the Q1 base terminal. The Q2 collector voltage is nearly 0 volts and therefore the D2 cathode has little to no reverse bias voltage via its RS. Thus, any small amplitude negative input pulse will cause D2 to become forward biased, reach the base of Q2 and drive Q2 OFF. Once Q2 switches off, in turn Q1 is toggled ON and its collector voltage goes low. The large reverse bias on D1 disappears. However, Q2 is now OFF and D2 will now be strongly reverse biased which will steer the next negative input pulse to the base of Q1. This is the basis of the circuit's negative edge flip-flop operation.

In another experiment, I changed the .001 C0G capacitors of the astable multivibrator to 470 pF. This gave a usable range of 22968 to 14832 Hertz (11484-7416 Hz at the Q1 and Q2 output) . Looking at the output of the flip-flop in the oscilloscope; at the higher frequency range, the flip-flop could not keep up and failed to divide by 2. I found experimentally that the time constant of each of the CS and RS components seemed to be the problem. When the CS capacitors were also decreased to 470 pF, the flip-flop worked properly.
As you increase the flip-flop operation frequency, speed up bypass capacitors might also be required across the 47K base resistors of Q1 and Q2 . A suggested starting value to try is 220 pF. Some builders also bypass the resistors in the RS steering circuit at higher frequencies, however, this is getting a little crazy. It is really important to look at the output waveform in the oscilloscope to ensure reasonable performance.



Shown above is the Figure 5 breadboard prototype.

Figure 6



5 KHz output waveform of Q2

4. One KHz Digital and Analog Oscillator

A 1 KHz oscillator with 5 volt digital outputs 180 degrees apart and an analog output was sought. The frequency had to be near to, but not exactly 1000 Hertz. A major question to answer was how much low pass filtering is needed to remove the odd harmonics from digital circuits?

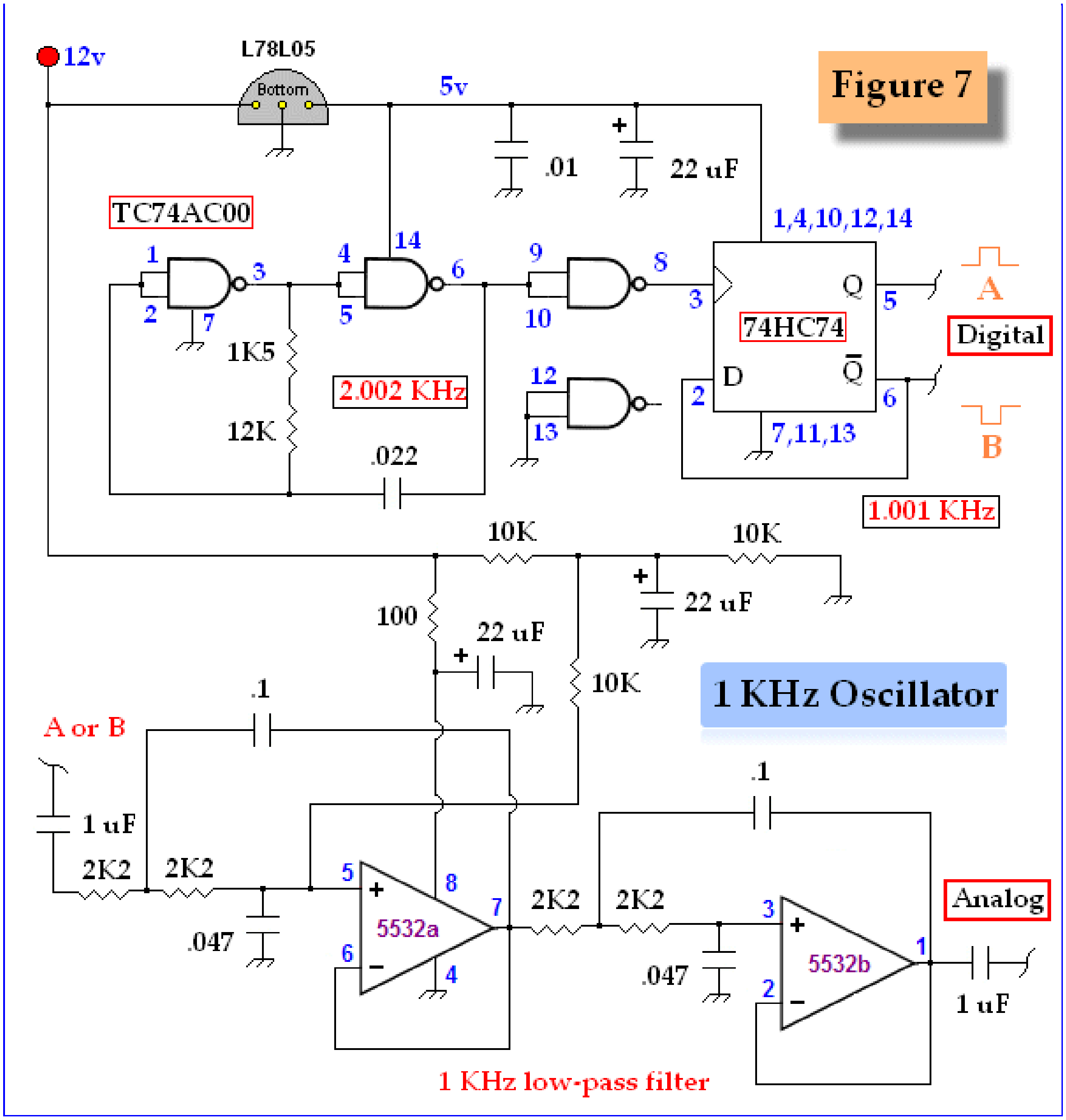


Figure 7 shows the complete schematic. NAND gates from a 74AC00 were wired as inverters and with the 13.5 K resistance and a 0.022 uF polyester capacitor, the frequency was 2002 Hertz. To improve the digital waveform and get the desired 2 outputs, a D flip-flop was used. The output frequency was 1001 Hertz. The digital part was completed!

For the analog filtering, active low-pass filters were tried, and in total 4 poles with a 1 KHz cuff off worked reasonably well. The filter uses the 5532 op-amp with common vales capacitors and resistors. Poly"something" caps were utilized.

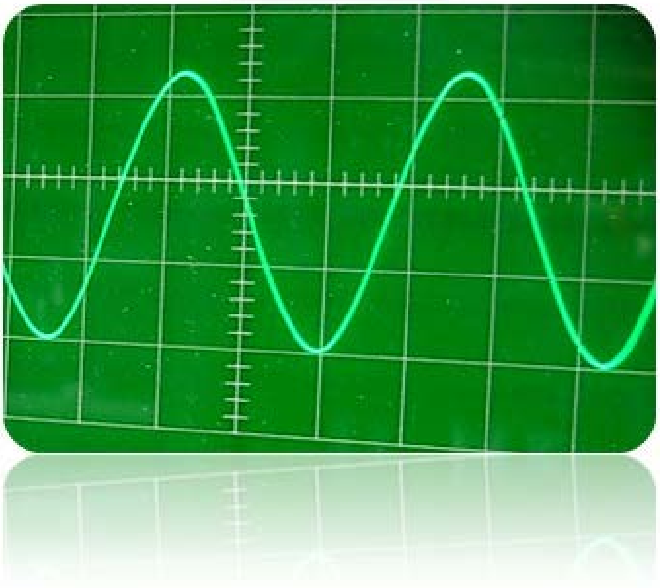


Figure 8

In Figure 8 is the output waveform of the low pass filter stages. A pretty nice sine wave was achieved and this oscillator could see duty for testing audio amplifiers. The scope was photographed at an angle to avoid the camera reflection and this distorts the sine waves a little.

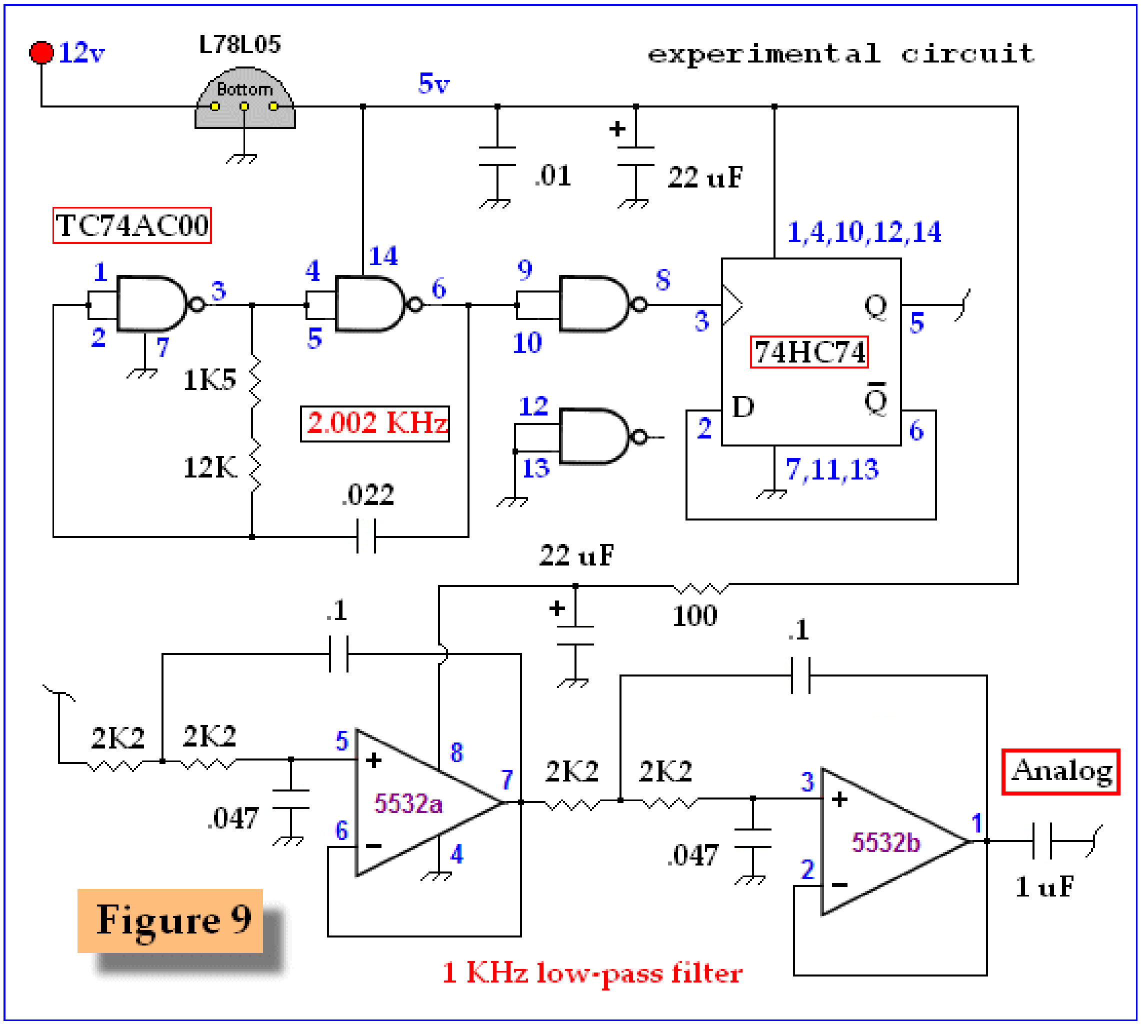


Figure 9

Figure 9 depicts an experiment with the op-amp biasing. If the op-amp is run at 5 volts VCC, the bias requirement is 1/2 VCC or 2.5 volts. The DC voltage at the output of the D flip-flop was 2.55 volts. The 2K2 resistor was connected directly to this output and this eliminated the VCC/2 resistor bias network and a coupling capacitor.

Figure 10

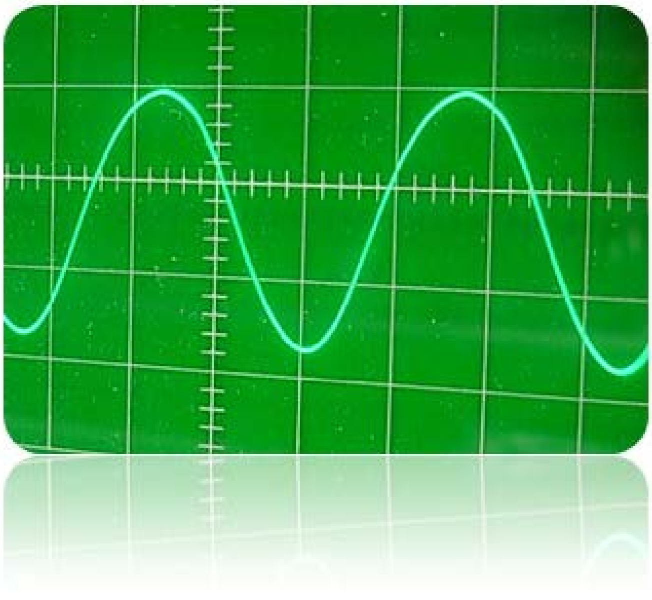
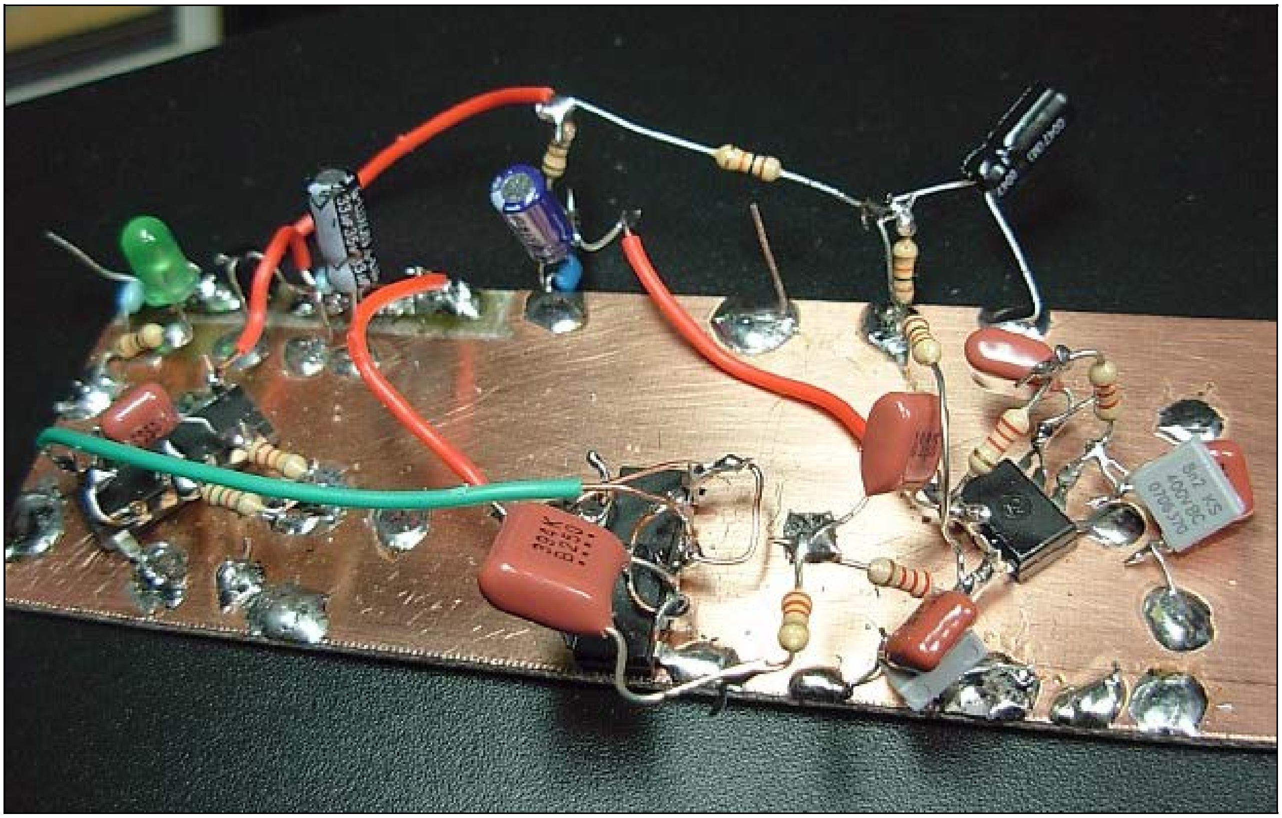


Figure 10 shows the output waveform of Figure 9. The AC waveform has harmonic distortion and thus the Figure 9 circuit will not be kept nor utilized.



The Figure 7 breadboard. A 0.39 coupling capacitor (not 1 uF) was used between the D flip-flop and 5532a in this particular version. Unfortunately, no 0.047 uF caps were available for the low-pass filters and therefore a 0.039 plus a 0.0082 were placed in parallel for each of the .047 uF caps.

5. One KHz Low Distortion Signal Generator

Although I own a variable frequency wein bridge oscillator, it has been been set to 1 KHz for 2-3 years and is large and temperamental. It was decided to make a low distortion sine wave oscillator for just this one frequency. The circuit will be placed in a box along with another signal generator.

There are a number of ways to build signal generators using op amps. Countless example circuits may be found on the World Wide Web and some of them are really fantastic. Chose whichever method works best for you. Some might find my circuit to be overkill, but to each his own.

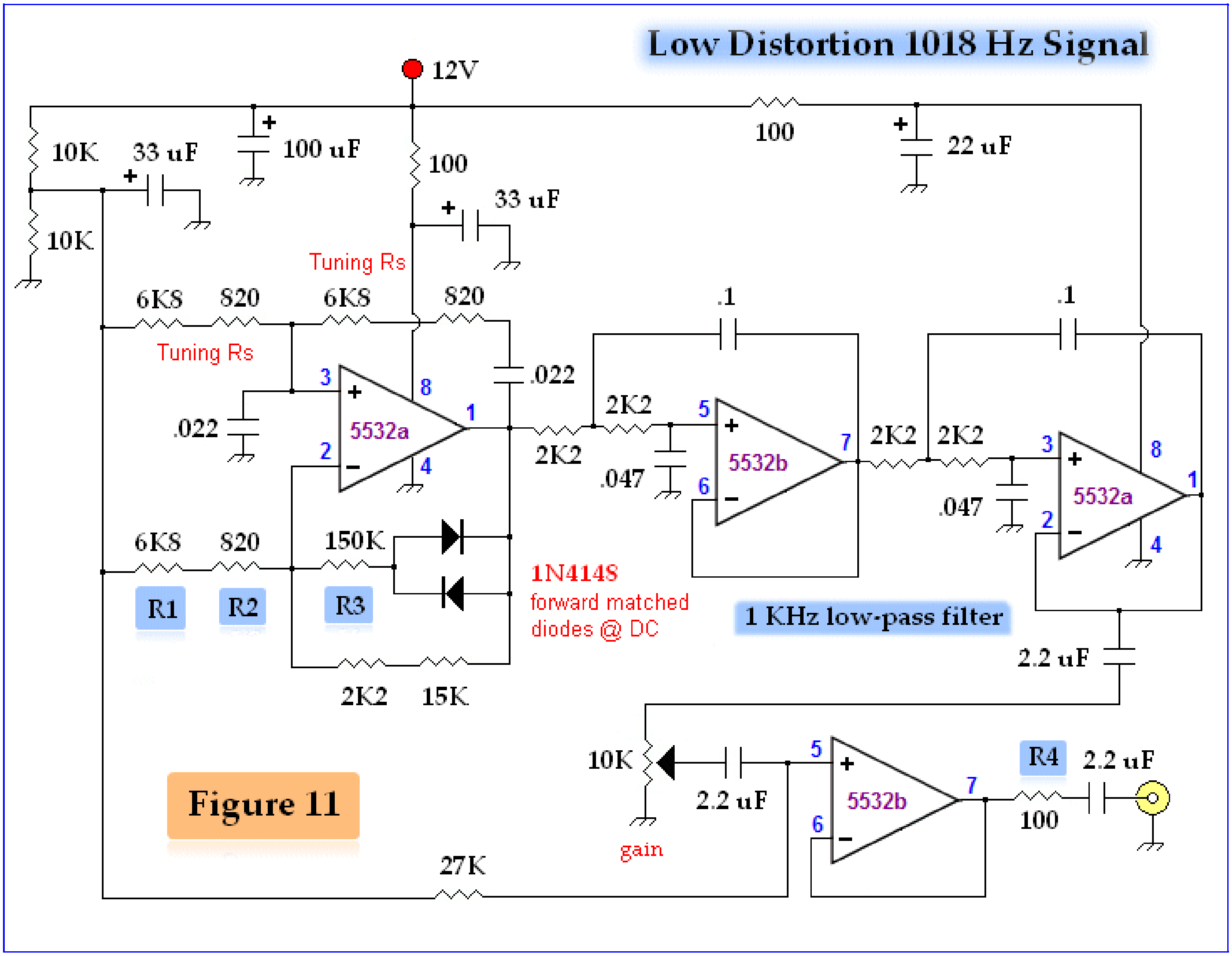


Figure 11 shows the entire circuit. The Wein bridge oscillator is from EMRFD and was designed by Wes, W7ZOI.

A 0.22 uF capacitor was chosen for the tuned circuits. Using resistors from my parts collection, 1018 Hz was the closest I could get to 1 KHz. This is the 6K8 + 820 ohms resistors labeled "tuning Rs". Other values were tried. For example, a single 6K8 gave 1142 Hz and a single 8K2 gave 939 Hz.

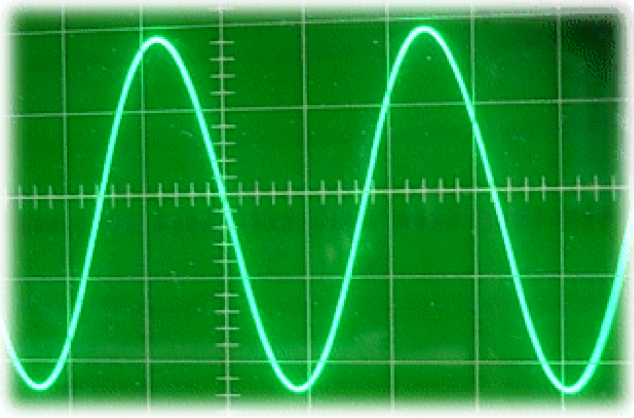
When the circuit was first built, I used a 10K on the VCC/2 bias point to pin 2 and a 22K feedback R from pin 1 back to pin 2. The sine wave had mild distortion. By experimentation, it was learned that the resistance from the VCC/2 bias point to pin 2 significantly affected the waveform purity. The 2K2+15K plus the R1 + R2 resistance values shown were determined by using a potentiometers rather than fixed resistors. Care was taken to adjust the feedback resistance from pin 1 back to pin 2 to keep away any overdrive distortion. I do not understand this, but even changing the 820 ohm R2 to 570 ohms, altered the sine wave purity.

The best looking sine wave came when the resistance from the VCC/2 bias point to pin 2 was the same as the tuning resistance; 6K8 plus 820 ohms. Later, the pin 1 to pin 2 feedback resistance was chosen for an unclipped; waveform with a reasonable output voltage using a potentiometer. The potentiometer was removed and measured at 17.1K, thus the 15K + 2K2 were soldered in. It was also discovered that by increasing R3 from 56K or 100K to 150K slightly improved the waveform.

The Figure 7 low-pass filter was connected to the main oscillator as shown. The final op-amp stage was used as a buffer between the low-pass filter and the gain control. R4 is used to set whatever output impedance you choose. Practically speaking, it could be any value between 47 and 620 ohms. Many AF oscillators have an output impedance of 600 ohms and 620 is the nearest E24 standard value. For my project, a 100 ohm

R4 was chosen. Output peak to peak voltage is 0.0 to 4.84v continuously.

Figure 12

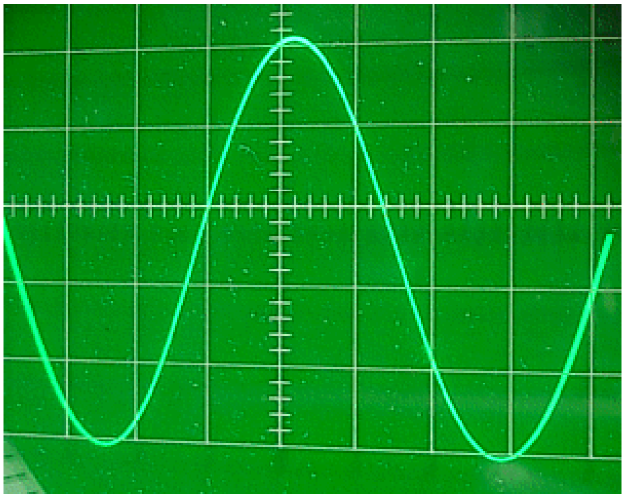


Here is the raw output of the basic Wien bridge oscillator. It is hard to photograph well, but it is stellar to say the least.

The Figure 11 breadboard mounted in a plastic Hammond chassis. The voltage regulator seen to the left of the bottom polyester cap is a 7812. This project has its own regulated power supply. Other view of Fig 11. Two 10 Megohm standoff resistors were used to help support all the resistors soldered to pins 1, 2 and 3 of the main oscillator.

A front view photo of the AF oscillator. A separate 7 plus 14 MHz oscillator circuit and controls will be placed on the right hand side of this box. The orange power ON indicator LED was epoxy glued into the chassis hole. Putting circuits in cabinets is one of the most expensive aspects of homebrew construction. One must be ever vigilant for bargain chassis boxes and hardware to keep costs down. Techniques such as gluing in the LED rather than purchasing a separate holder and recycling knobs and switches are also practiced for cost containment.

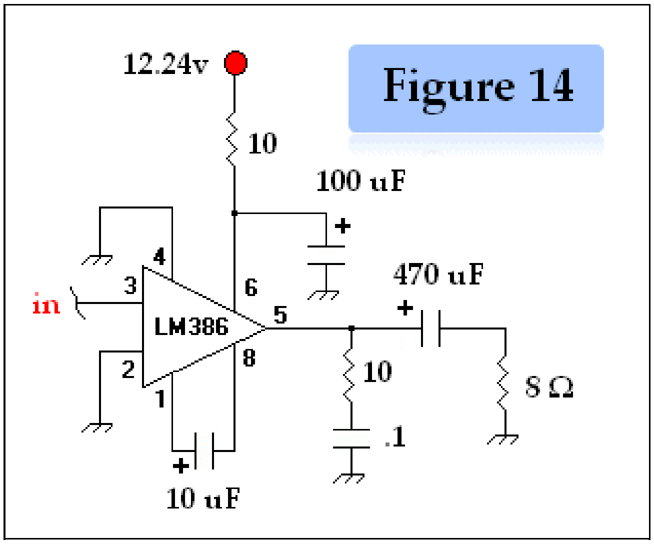
Figure 13



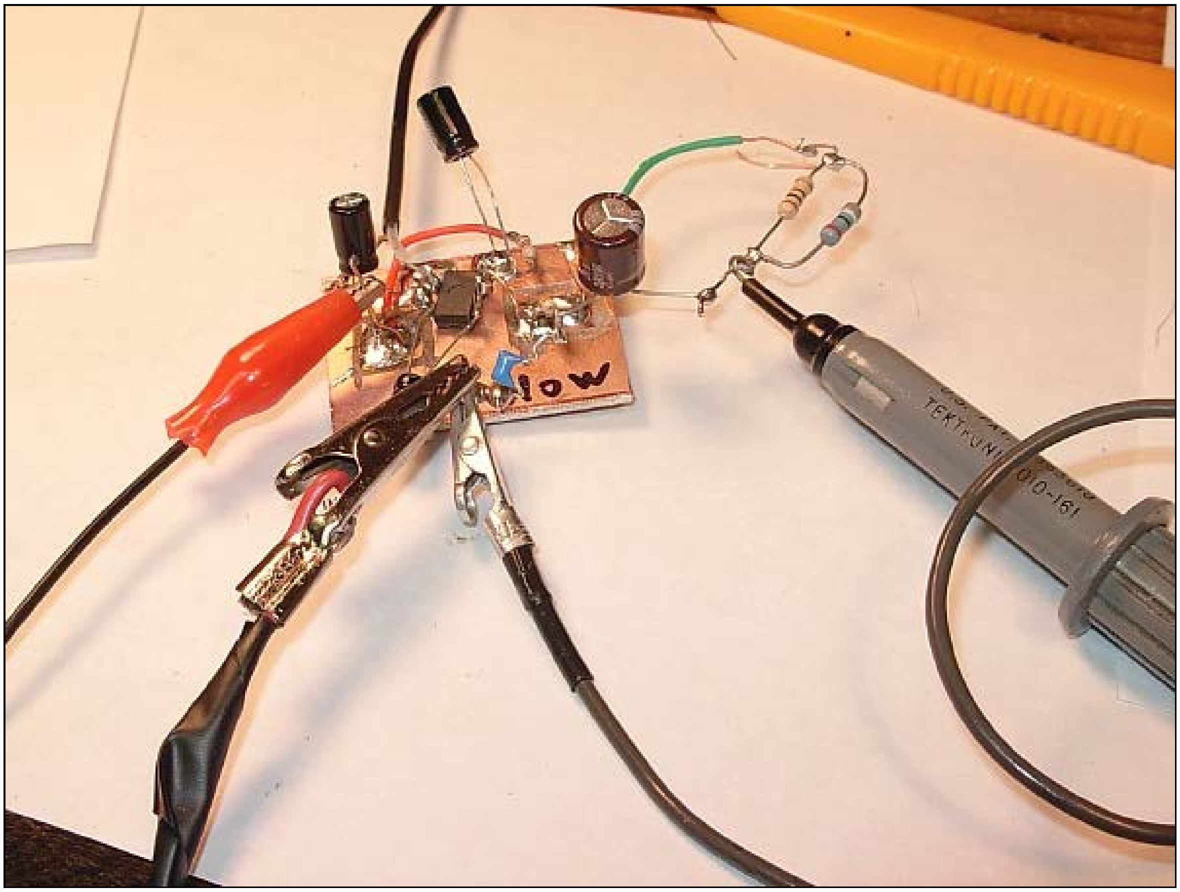
The output of Figure 11 is shown in Figure 13. This is the best sine wave seen ever on my scope. I looked at it closely and there is no change in line thickness or symmetry anywhere. It inverts with no change on the scope. Testing audio amplifiers will now be much more fun.

6. LM386 Power Experiments

The LM386 is an IC audio amp that has been used in thousands of hobbyist projects over the past 2 decades. By adding a capacitor +/- a resistor between pins 1 and 8, this device's internal gain can be changed from x20 to up to x200.



Test circuit schematic in Figure 14.



The experiment breadboard is shown above. A very standard configuration. The amplifier drove an 8 ohm, 1 watt resistive load.

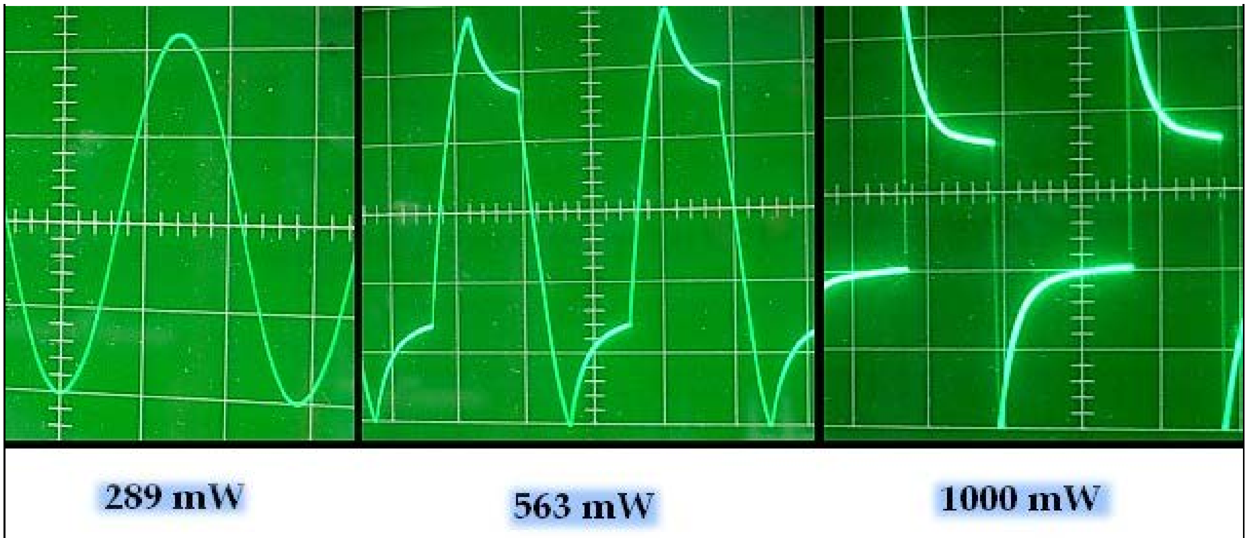


Figure 15

Over the years, I have noticed some kit sellers and project authors claiming that their LM386 based AF stages gave 1 or occasionally even 2 watts of output power into an 8 ohm speaker. This was confirmed on the bench. This device will output 1 watt into 8 ohms at 1018 Hertz with little problem. However, this is clearly 1 watt of square wave distortion.

The quiescent current of the LM386 was around 7 mA. The signal generator gain was increased until the first signs of distortion appeared. The gain was then backed off a little so a pure sine wave was observed in the oscilloscope. The current was ~ 155 mA and the measured power was 289 mW. Please refer to Figure 15 for the 289 mW sine wave. This was the clean signal power of the LM386 on my bench. The output waveforms at 563 mW and 1 watt are also shown. Extreme harmonic distortion occurred above 300 mW. This device will draw 240 mA or more when driven and clipping hard. It is not my intention to malign the LM386. It is a useful part, albeit a little dated. Its AF gain capability versus size is something to behold. Many builders have moved to the TDA7052 audio amplifier IC, or like myself, build their own low noise audio power amps.

The 12.24 volt DC supply and the 1018 Hz AF audio oscillator used in these experiments.

Time Domain Output from a Diode Ring Mixer

22 Dec 2009, w7zoi

Figure 1. Some available experimental modules.

Some folks wonder about the output that they should see on their oscilloscope when looking at the output from a diode ring mixer. There is no set, pat answer. The output can change dramatically as levels, frequencies, and even terminations are changed. This complication is illustrated here with a few screen shots, taken with a Rigol DS1052E 50 MHz bandwidth digital storage oscilloscope. The experiments started with the following pile of modules. Your collection will probably differ.

Figure 2. The inside of the module containing a Mini-Circuits SBL-1 diode ring mixer. This is a standard part that is essentially generic.

The first experiment was to set up a pair of 10 MHz signal sources. One was from a homebrew generator, shown below.

Figure 3

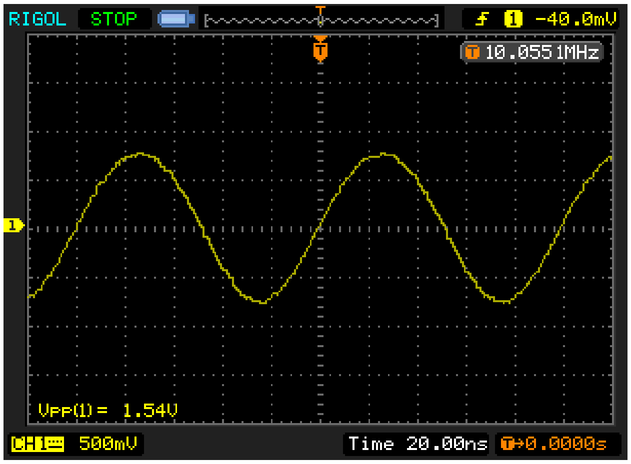


Figure 3. The 10 MHz signal from a homebrew signal source. The 1.54 volt peak to peak signal is applied to a 50 Ohm terminator at the oscilloscope. The delivered power is then +7.7 dBm.

This signal was filtered with a 14 MHz low pass circuit. This caused the amplitude to drop by 0.2 dB. The source was then attached to the LO (local oscillator) port of the SBL-1 mixer.

Figure 4

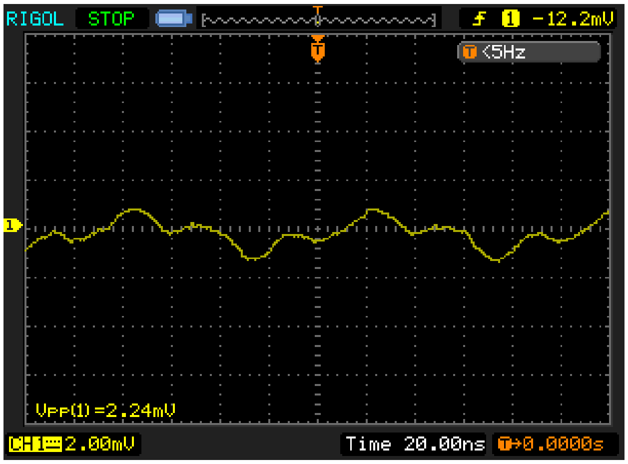


Figure 4. The IF output from the mixer when there is nothing attached to the RF port. Note the scope sensitivity of 2 mV/div.

Next, we attached a 50 Ohm terminator to the R mixer port.

Figure 5

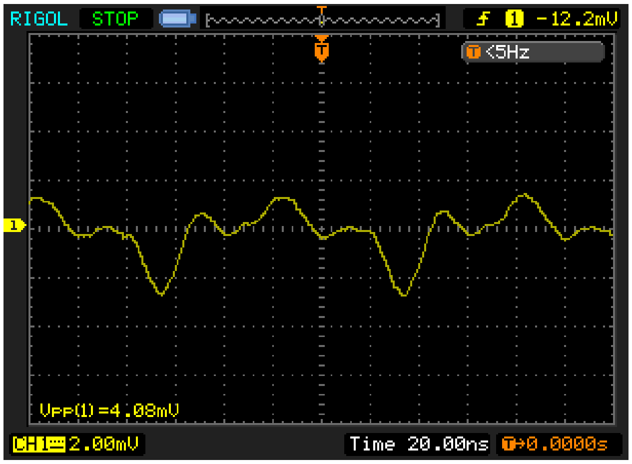


Figure 5. The IF output with LO drive, but without an R signal. But the R port is now terminated. This waveform, when compared with Fig 4, shows just how sensitive the mixer can be to termination.

In the next experiment, a -20 dBm signal was applied to the R port. The frequency was very close to the 10 MHz LO that is still present.

Figure 6

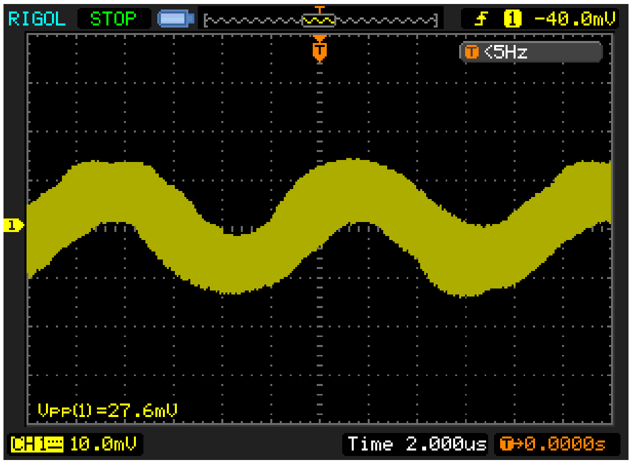


Fig 6. There are two dominant signals from the mixer. One is a low frequency at 100 kHz. But this is accompanied by a high frequency of about 20 MHz. These two outputs, a sum and difference frequency, are expected from any mixer.

A filter can isolate the two dominant outputs. This is shown below where a 500 kHz low pass filter is inserted in the line between the mixer and the oscilloscope. A 6 dB pad is between the mixer and the filter, for direct insertion would upset the termination of the mixer.

Figure 7

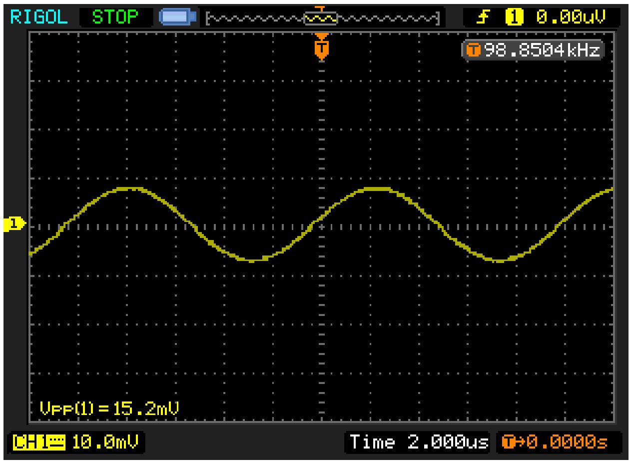


Figure 7. The output of about 100 kHz after a low pass filter is inserted in the mixer output.

The next experiments emulate a SSB transmitter. We start with a signal at 11.06 MHz with strength -20 dBm. (This is a common IF used in homebrew SSB transceivers such as the BITX-20.) This is applied to the mixer R port. The L port is driven with a +7 dBm signal at 3.19 MHz. The LO signal is low pass filtered to attenuate harmonics, a measure that is probably not necessary, but the filters were there. The IF output is shown below.

Figure 8

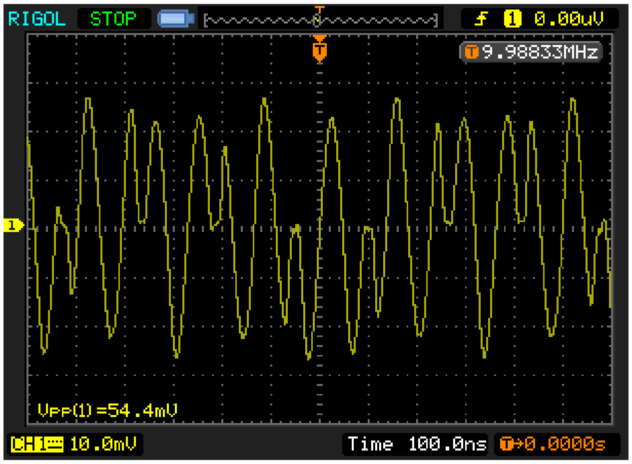


Figure 8. Time domain output of a SBL-1 set up as the “transmit” mixer in a SSB rig. This is far from the “perfect sine wave” that some folks tell us we should observe. This waveform contains many different frequency components. The counter output should not be interpreted to have any meaning. (I should have turned it off.)

The signal of Fig 8 can also be viewed with a spectrum analyzer. This is shown below. This measurement was taken with the August 1998 QST Spectrum Analyzer and not the FFT routine in the DSO. The Rigol scope has a nice display for an analyzer.

Figure 9

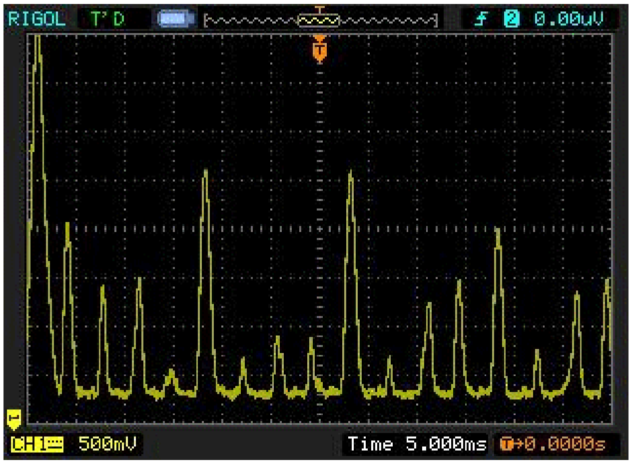


Figure 9. Spectrum of the signal shown in Fig 8. The largest signal on screen is that at the left, which is the spectrum analyzer zero spur. This is a spurious output that is typical of most SA systems. The desired signal at 11.06+3.19=14.25 MHz is just to the right of center. But the image is also present at the different frequency of 7.87 MHz at about 3 major divisions from the left edge.

Alas, I didn’t find a 14 MHz bandpass filter in the junk box. Such a filter would have allowed selection of the dominant 14 MHz component while attenuating all the rest of the junk shown. The many other signals are the result of harmonic mixing. That is, we observe IF outputs at N x FLO +/- M x FRF where N and M are integers. Some of these spurious outputs can be quite strong with diode ring mixers. They are best avoided with high frequency LO signals. In this case, a LO at 14.25+11.06=25.31 MHz would produce a much cleaner output spectrum. It is much easier to obtain LO stability with an oscillator built at 3.19 MHz.

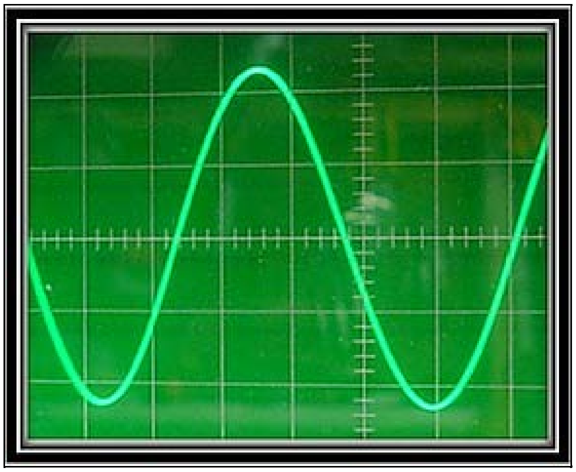
Bottom Line

It is not reasonable to have a well defined, predictable time domain (i.e., normal oscilloscope) output from a mixer. The exact results depend upon too many variables. A spectrum analyzer can be used to garner much more information.

RF — Test and Measurement

Low Noise Crystal Oscillators

Introduction

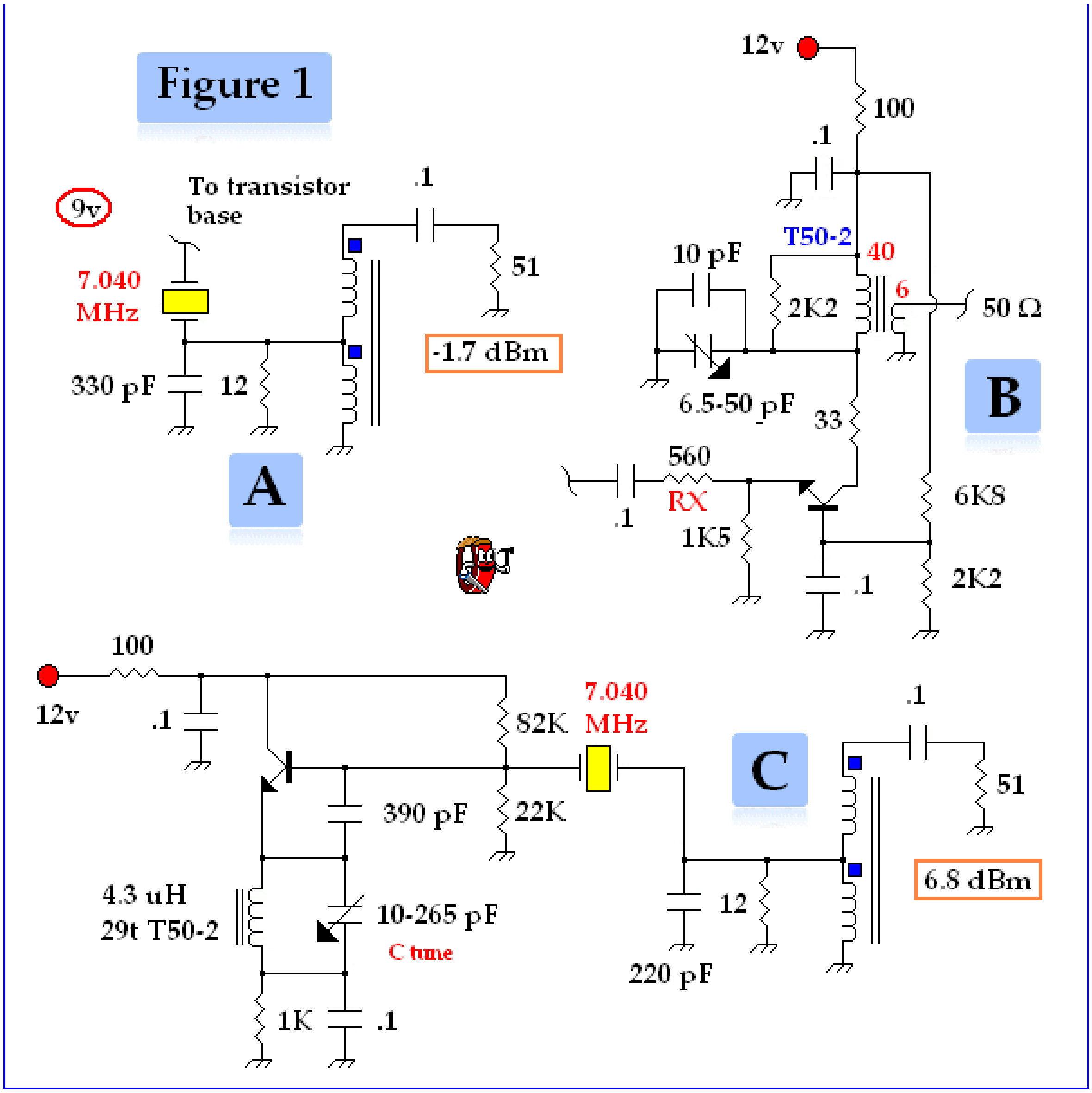


Some experiments were conducted to build a low noise crystal oscillator with 50 ohm output at 7 and 14 MHz for the test bench.

Some readers might wonder why build such an oscillator? Although a variable RF signal generator is an important bench tool, it is also nice to have a fixed frequency signal source on your favorite HAM band. This RF source can be connected to other 50 ohm modules such as band-pass filters, diode ring mixers or feedback amplifiers to conduct experiments at a whim. The project goal was a low noise oscillator with 2 outputs so it could drive a divide by 2 flip-flop for 40 meter band digital mixer work, or fundamental frequency use on the 20 and 40 meter bands.

Presented are some experiments carried out to realize this goal. Only some of the better experiments and circuits are shown. An additional circuit was added Jan 31, 2010.

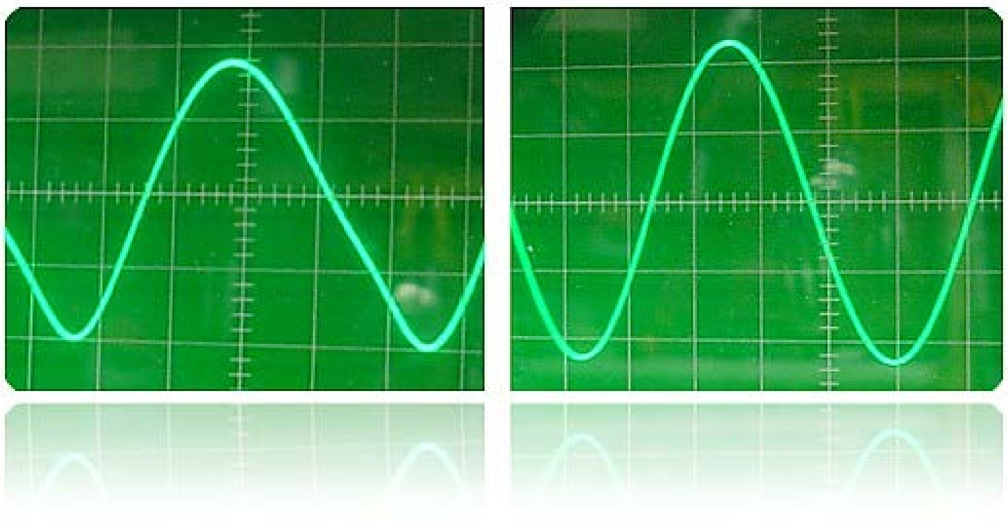
First Steps



The 7.040 MHz crystal used is an AT cut, HC6-U holder part made locally by West Crystal. I reviewed the information concerning crystal oscillators in Chapter 4.5 of EMRFD and then started melting solder. The oscillator output was extracted as described in EMRFD Figure 4.24. The output is low distortion, low impedance and low gain. Like in most experiments, I built it, measured the DC voltages and then looked at the AC voltages in the oscilloscope.

In order to measure the output, a 51 ohm load was transformer coupled as shown in Figure 1a. I am uncertain if this was a good method for power measurement, however, it allowed comparison of the experimental circuits. As shown, the output with a 9 volt regulated power supply is low; -1.7 dBm. Still, a circuit like EMRFD Figure 4.24 can be used with a variety of crystal frequencies and has great utility.

Figure 2



Signal from BJT emitter

Signal from between cap and crystal

Figure 2 shows two output waveforms taken from my crystal oscillator. When a signal is taken from the emitter of the main oscillator transistor (what we typically do) harmonic distortion occurs as shown in the above left. Actually, the distorted waveform photo above left looks better than most do. Typically, they look like this. Many builders will just place a low pass filter on such an oscillator's output and be very satisfied with the harmonic content in their signal. Certainly this is a good, common and practical way to go. However, for some builders, the experimenter's journey is what counts. That is, the fun and learning occurs during designing/building/testing and not just operating home built gear.

When the output is taken from between the shunt capacitor and the crystal per Figure 1A, a much cleaner sine wave is available. The photo above right tells this story. The focus of all of the experiments on this web page is boosting this lower distortion, lower phase noise signal into something useful. To increase the base oscillator output voltage, the VCC was raised to 12 VDC and the BJT emitter circuit was tuned per Figure 1C. Resident on my work bench are a few potentiometers and a 10-254 pF air variable capacitor with short attached leads. These parts are inserted into test circuits to allow tweaking of R or C as desired. Once the desired tweaking is performed, the potentiometer or variable capacitor is removed and measured. The closest fixed value R or C is then substituted as appropriate. In this experiment, the highest output voltage occurred when the variable capacitor measured 181.7 pF. Thus in my version, 33 pF and 150 pF capacitors were placed in parallel and are shown later in Figure 3.

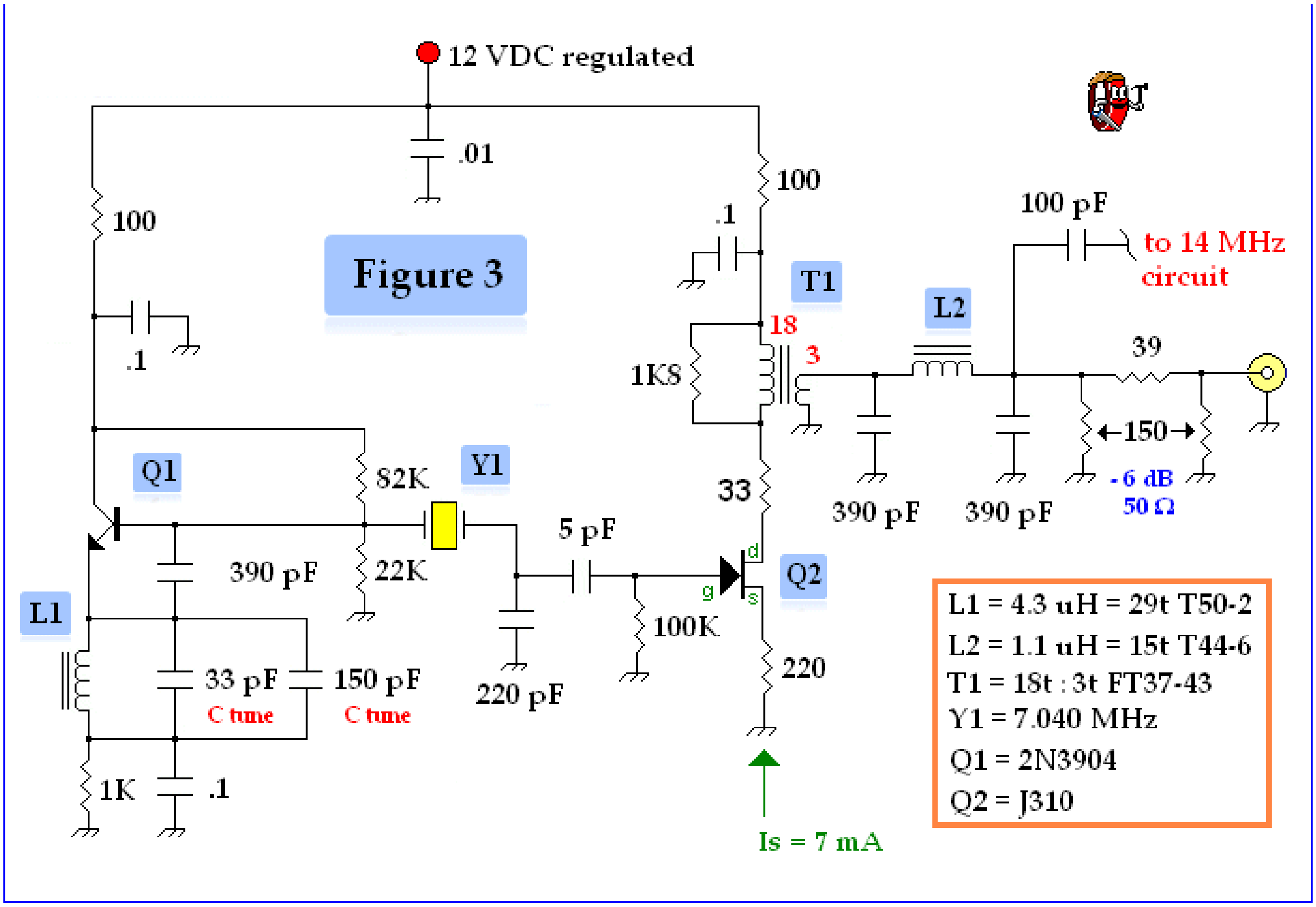
The Figure 1A output transformer circuit was again used and the power output was 6.8 dBm. Being tuned to 1 crystal frequency is the biggest drawback of the 1C circuit. Tuning a crystal oscillator as described earlier is easy to do however.

The next task was to design and build a buffer/amplifier. To match the low impedance of the Figure 1C crystal oscillator, a common base amp was built. The circuit was morphed over time, however, the initial design is shown as Figure 1B. It was interesting to note that a series resistor (RX) is required to keep the waveform pure. Any RX value less than 470 ohms compromised the sine wave purity. The 560 ohm R shown was perfect, however, as expected, attenuated the oscillator signal. In order to get a decent output voltage, the common base amp had to be run at 5 mA or greater current and ultimately collector tuning was added to try and realize an output voltage greater than 4 dBm.

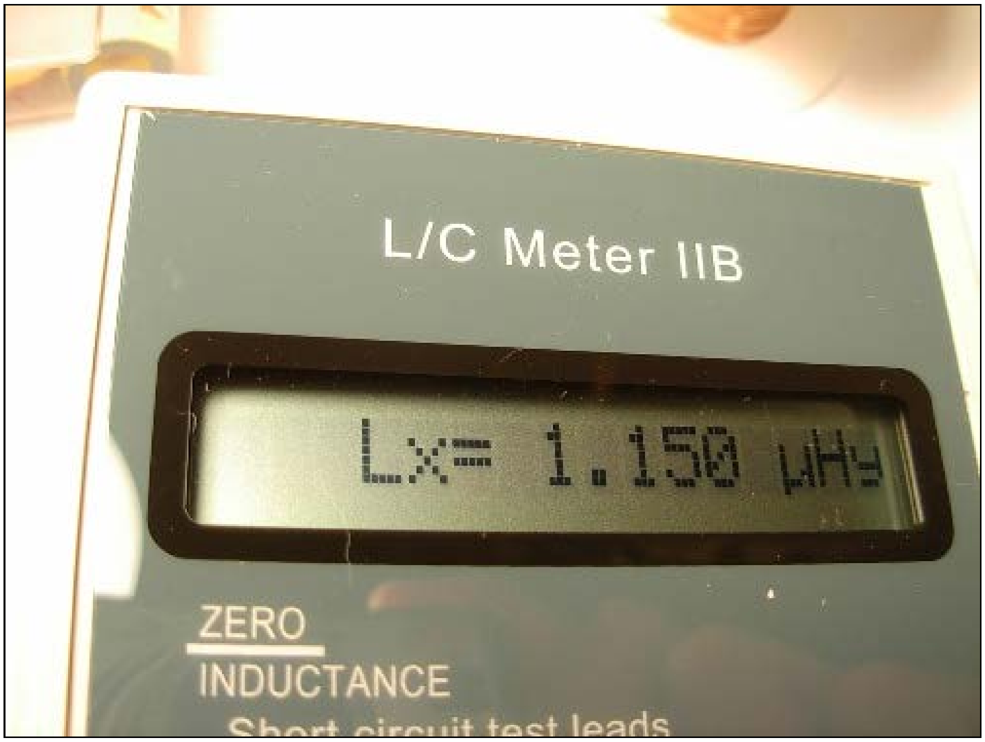
Through experimentation I learned that adjusting Fig 1B's tuning, emitter current and RX value all could distort the output of the main oscillator at certain values or settings. Running high current also invites parasitic oscillations and soon it was realized that common base was perhaps not the best choice (at least for me) as another separate amplifier stage would be required to get a decent output voltage with or without an attenuator pad.. After trying a number of different buffer-amplifiers including a 50 ohm feedback amp, I chose a favorite circuit which I know has excellent gain plus back to front isolation and would not distort the oscillator waveform; a lightly coupled JFET amp.

---

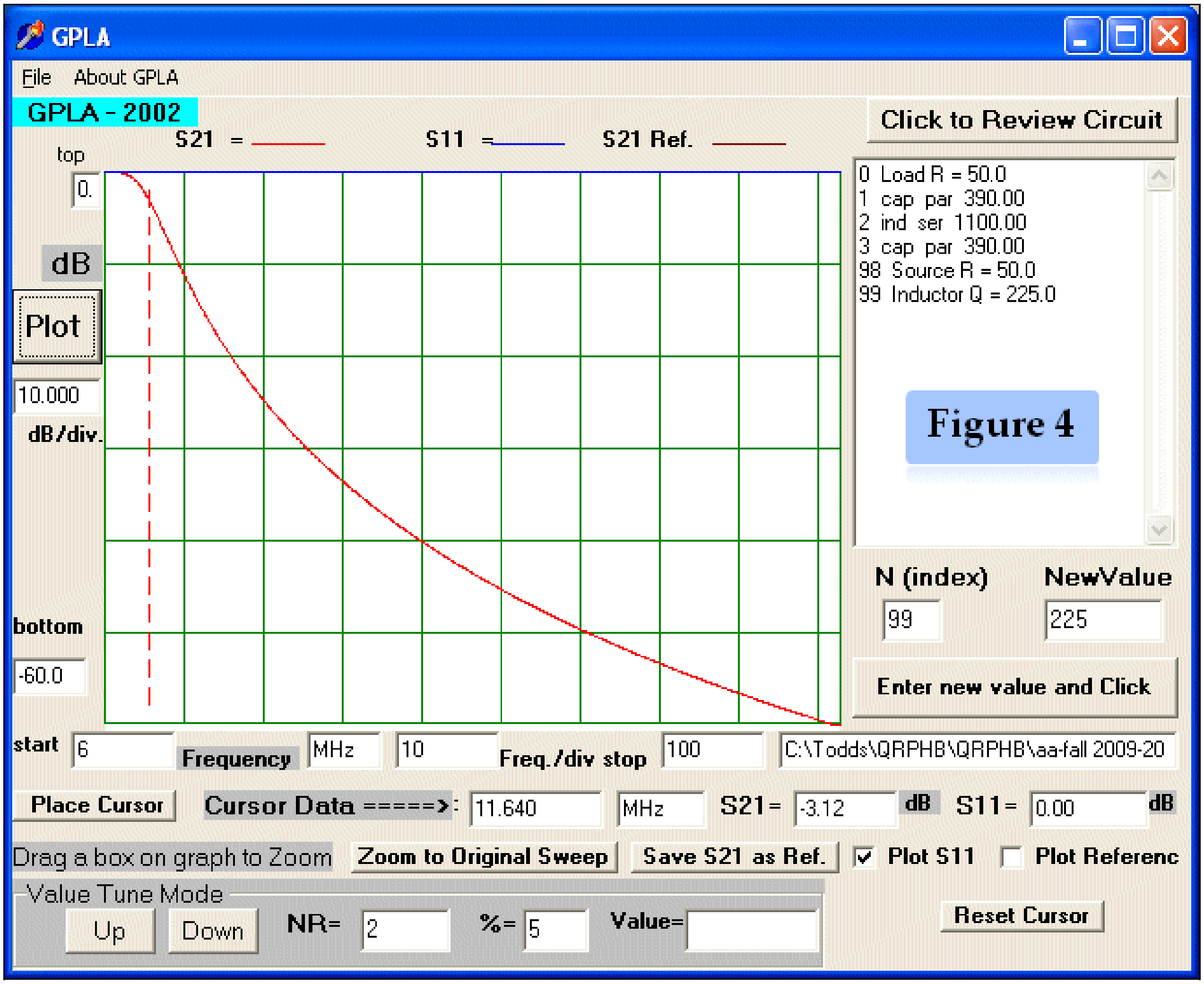
7 MHz Circuit



In Figure 3 is the complete schematic of the 7 MHz portion of the low distortion crystal oscillator. The output was 0.52 dBm. A 6 dB 50 ohm pad ensures a well-buffered 50 ohm termination. This aids in calculating gain or loss in circuits it drives. You could easily decrease this to a -3 or -4 dB pad. This reflects the wisdom handed down by our mentors who encourage building RF stages in 50 ohm impedance blocks. A Q2 source bypass capacitor was not placed as it increased harmonic distortion in the output signal. The output is filtered with a simple pi filter. The 100 pF coupling capacitor connect this circuit to Figure 6; the 14 MHz circuit.



L2 was wound and measured. It is desirable (but not absolutely necessary) to perform measurement on powdered iron toroids to compensate for variations in wire spacing and toroid permeability. I used a T44-6 core; use whatever appropriate powdered iron toroid you want.

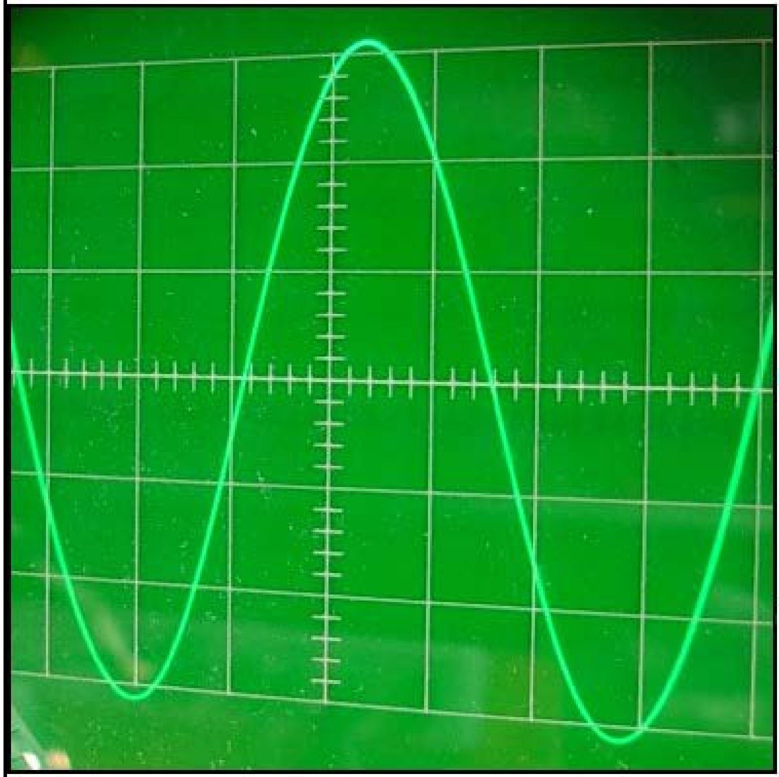


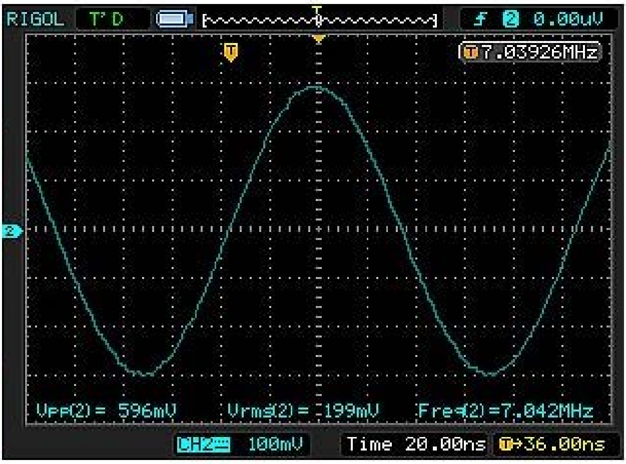
```
0  Load R = 50.0
1  cap  par  390.00
2  ind  ser  1100.00
3  cap  par  390.00
98  Source R = 50.0
99  Inductor Q = 225.0
```

top: 0.
10.000 dB/div.
bottom: -60.0
start: 6 Frequency MHz
10 Freq./div
stop: 100
C:\Todds\QRPHB\QRPHB\aa-fall 2009-20
N (index): 99
NewValue: 225
Cursor Data =====>: 11.640 MHz
S21= -3.12 dB
S11= 0.00 dB
☑ Plot S11
☐ Plot Referenc
NR= 2
%= 5
Value=

A GPLA analysis of the Pi low-pass filter is shown in Figure 4. The basic circuit was designed with PI Filter Designer on this page and tweaked in GPLA. You may wish to omit this filter or perhaps design a better one yourself.

Figure 5





-0.52 dBm, 0.88804 mW

The 7 MHz output waveform on the Tektronix (left) and the Rigol oscilloscopes. On the Tek scope, the output power was 0.50 dBm and on the Rigol scope it was 0.52 dBm. The Rigol is an amazing oscilloscope, but only has 256 horizontal lines of resolution, therefore cannot replicate the stellar and beautiful waveform tracings of the ancient Tek scope. In reality, probably, no other modern scope can. I have received many positive comments about the old Tektronix oscilloscope waveforms. It is important to mention, that Rigol waveform viewing is not bad, just very different. The visual display is incredibly accurate and its triggering options, bandwidth, sampling rate and waveform display tools are fantastic. фантастический !

During these experiments, the 40 year old (plus) Tektronix scope was distorting frequencies greater than 10 MHz and breaking into oscillations. After 3 major repairs in 2009, the scope replaced with a Rigol DS1052E. Signal viewing will certainly different; that is for sure. The decision to move from a cathode ray oscilloscope (CRO) to a digital storage scope (DSO) was not taken lightly.

DSO versus CRO - some comments from the workbench

Choosing a CRO versus a DSO is an individualized process. It is your decision alone. Questions to ask yourself may include: What are my needs? What is my budget? Do I have weight and/or space constraints? Carefully weigh the advantages and disadvantages of each.

Proponents of CROs state that these scopes cannot generate artifacts, nor distort the signal. This of course is true as long as the scope bandwidth is adequate. Further, some people feel that aliasing or artifact generated in DSOs due to undersampling (taking too few samples of a waveform) is unacceptable. They may even feel that DSOs are not precision measurement instruments as a result. Limited horizontal screen resolution in DSOs is also a bugaboo for some experimenters and provides further evidence of DSO inferiority in the minds of these folks. These concerns are indeed valid; however, black and white thinking is a little out of fashion in a world more containing shades-of-gray.

The DSO takes a series of samples and stores them in memory. When sufficient samples are present, they are assembled and displayed. The sampling rate of a DSO is variable and depends on the time base setting used. Modern DSOs, like the Rigol, Tektronix Oscilloscopes and Agilent Oscilloscopes have better sampling rates and larger memories than their predecessors and hence aliasing is less of a problem than before; although in some measurement situations, undersampling can occur. One must always well consider and interpret whatever you are measuring and if you use a DSO, always keep undersampling in mind. When first using a DSO, you are on the bottom of a learning curve, however, with attentiveness and practice, one can learn to look for and possibly mitigate undersampling should it occur. In certain cases, a CRO will be superior to a DSO. In my discussions with others about Rigol signal viewing , only 1 significant "aliasing" problem has been noted by a builder when he tested a balanced modulator. The display did not give the expected result (was not filled in as expected) and a CRO was pulled out and the problem was verified. The builder knew there was a problem and could understand why it occurred. This builder also wrote that this was not so much a problem, as a reality of using a DSO.

Some techniques I have gleaned from the Internet about detecting aliasing may include the following:

Vary the time base over several ranges. Events occurring near the time base should be reproducible and if they are not, undersampling might be occurring.

As possible, use a single sweep and dot display. The 'dots' will indicate just where the scope took each sample. If the dots are far apart relative to the waveform timing, aliasing is a possibility.

Some techniques to minimize aliasing:

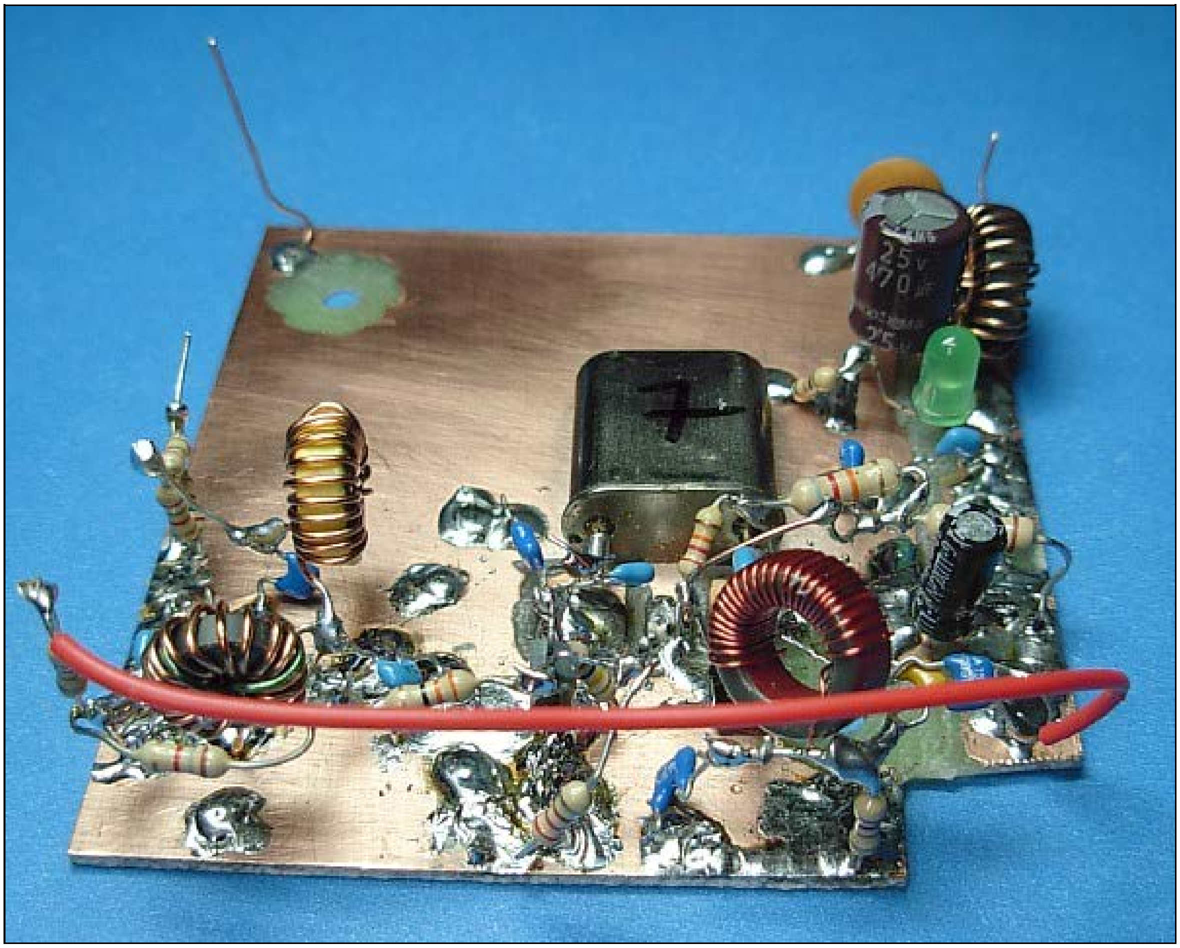
Choose linear interpolation when using math functions.

Use bandwidth limiting in low level measurements (The Rigol seems to automatically use B/W limiting in these situations).

Use trace averaging for low level measurements as possible.

The Rigol weighs 50 times less than my old scope and fits on a small shelf in my small workspace. As a hobbyist, it meets my needs and budget plus has some very cool features. Undersampling is considered and in some cases, such as low level measurement, an analog scope might be a better choice. Happily, a CRO is available to me if I really stress out over it. Signal viewing was taken for granted with my old scope. In some ways, this DSO has prompted me to dig deeper; to become more vigilant and thoughtful about signal measurement and display. If you have a spectrum analyzer, and use a DSO for signal viewing, the ability to perform slow sweeps while maintaining a perfect display is quite enjoyable. Again, please decide the CRO versus DSO issue for yourself (or maybe get a hybrid). The DSO is not a perfect solution to every signal viewing situation, but their constraints are quite livable considering their numerous modern features.

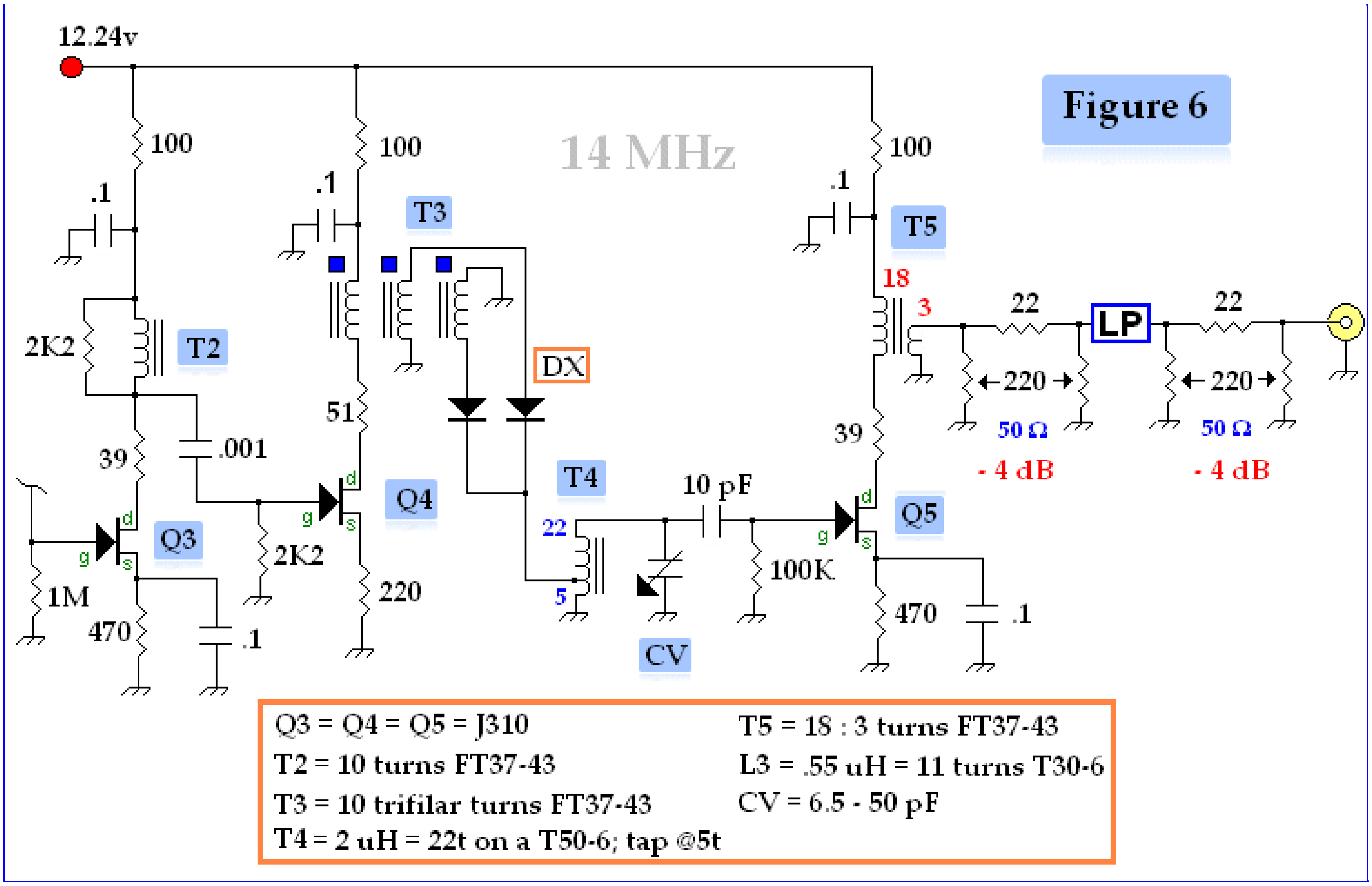
DSO's: “They are not your father’s oscilloscope”; that is certain!



The Figure 3 breadboard. AWG 24 to 26 gauge wire was used in the various inductors and transformers to better secure or anchor these parts.

---

14 MHz Circuit



Q3 = Q4 = Q5 = J310
T2 = 10 turns FT37-43
T3 = 10 trifilar turns FT37-43
T4 = 2 uH = 22t on a T50-6; tap @5t
T5 = 18 : 3 turns FT37-43
L3 = .55 uH = 11 turns T30-6
CV = 6.5 - 50 pF

Figure 6 is the final schematic for the 14 MHz circuit as developed on the workbench. Your design might look very different than mine. In the first version of this circuit, there were only 2 JFET amplifiers and the resultant output voltage was too low (even without a 50 ohm attenuation pad). To compensate, I ran the source current of the JFETs above 15 mA, placed a source bypass capacitor on Q4 and also used a 1000 pF capacitor to couple the input to Figure 2. Some fairly bad harmonic distortion was measured at the output and it seemed crazy to run so much current. Therefore it was decided to run a third JFET amplifier and use only modest current in the trio of JFET amplifiers.

CV tunes very sharply and required some care when peaking the output voltage.

Figure 7

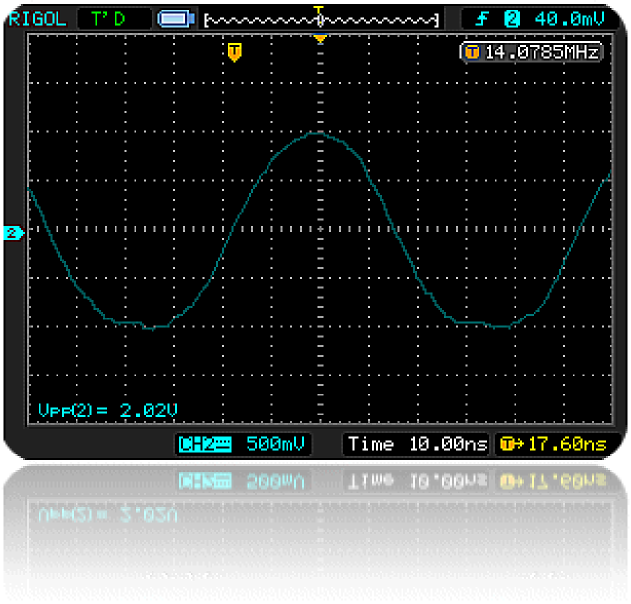
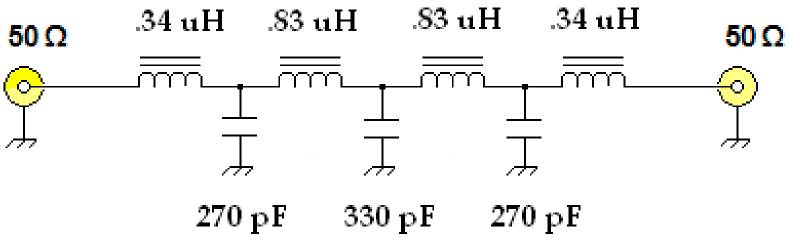


Figure 7 is a screen capture of the 14 MHz circuit output measured using a sensitive 50 ohm terminated oscilloscope. This was with no low-pass filtering. The output is distorted. Presumably this happened in the diode frequency doubler. This is not a low-distortion oscillator.

15.3 MHz Low-pass Filter



A N = 7 Chebyshev low-pass filter was inserted in Figure 6 at point LP. I checked with a spectrum analyzer and the 2nd harmonic was down 38 dB. There were no other measurable harmonics after that.

Figure 7b

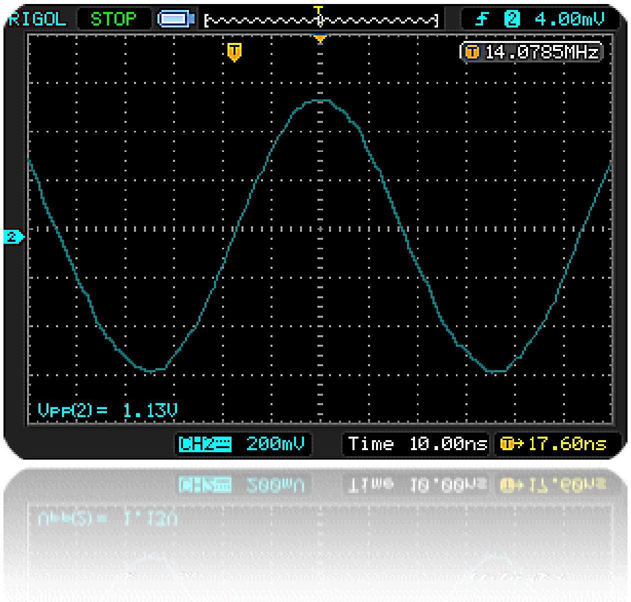


Figure 7b is a screen capture of the 14 MHz circuit output after the low-pass filter and final attenuator pad. Vpp on this graphic = peak to peak voltage = 1.13 volts. The output power is 5.04 dBm, or 3.19 mW . The Q4 and Q5 source resistors and the output attenuator pads are 2 areas of the circuit where you might easily change the output power. In the end, the circuit labeled Figure 6 was chosen. Your output voltage will probably vary, but can be easily adjusted as described.

Figure 8

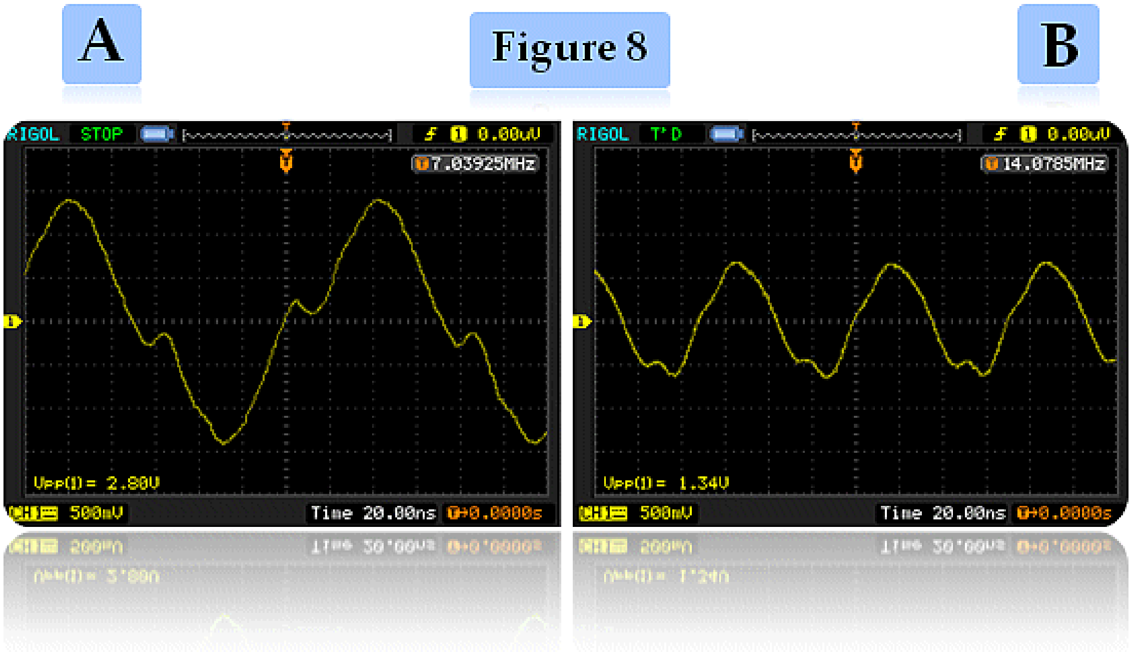
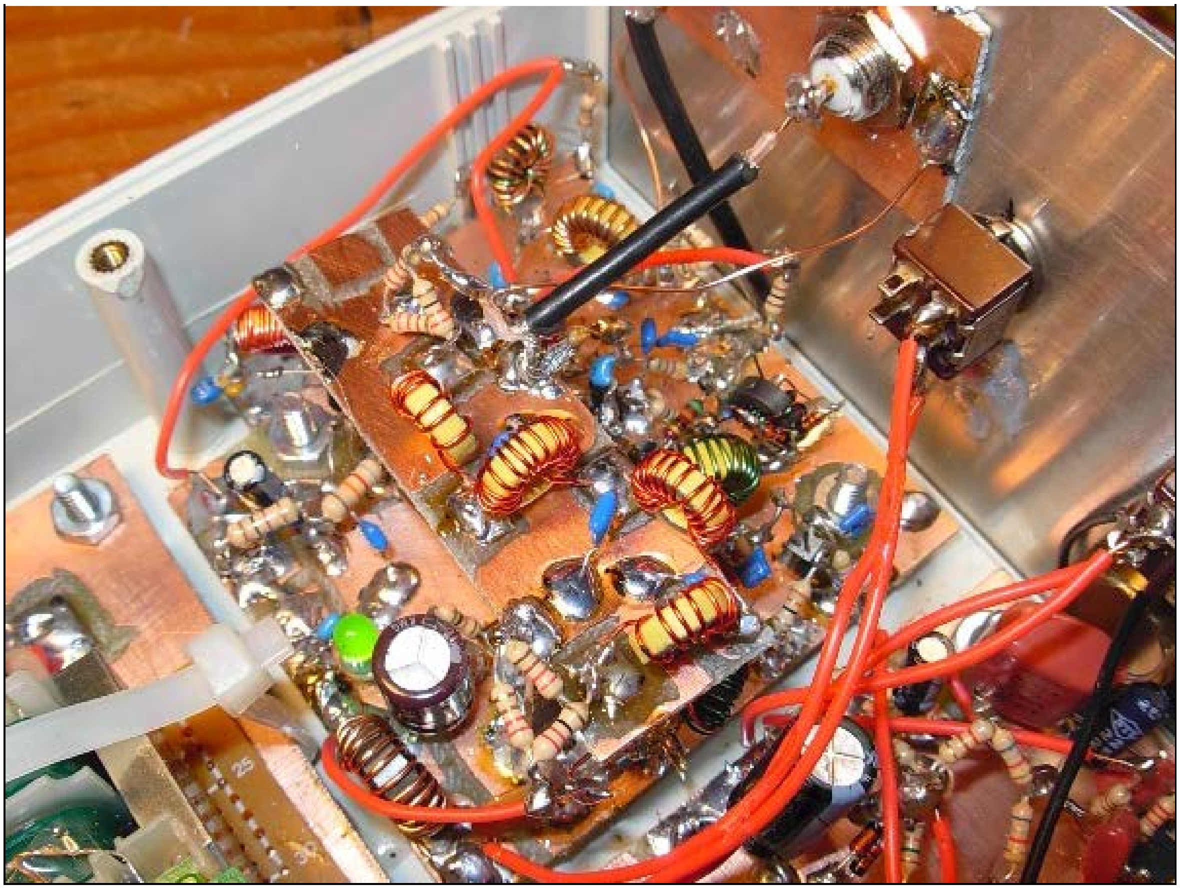
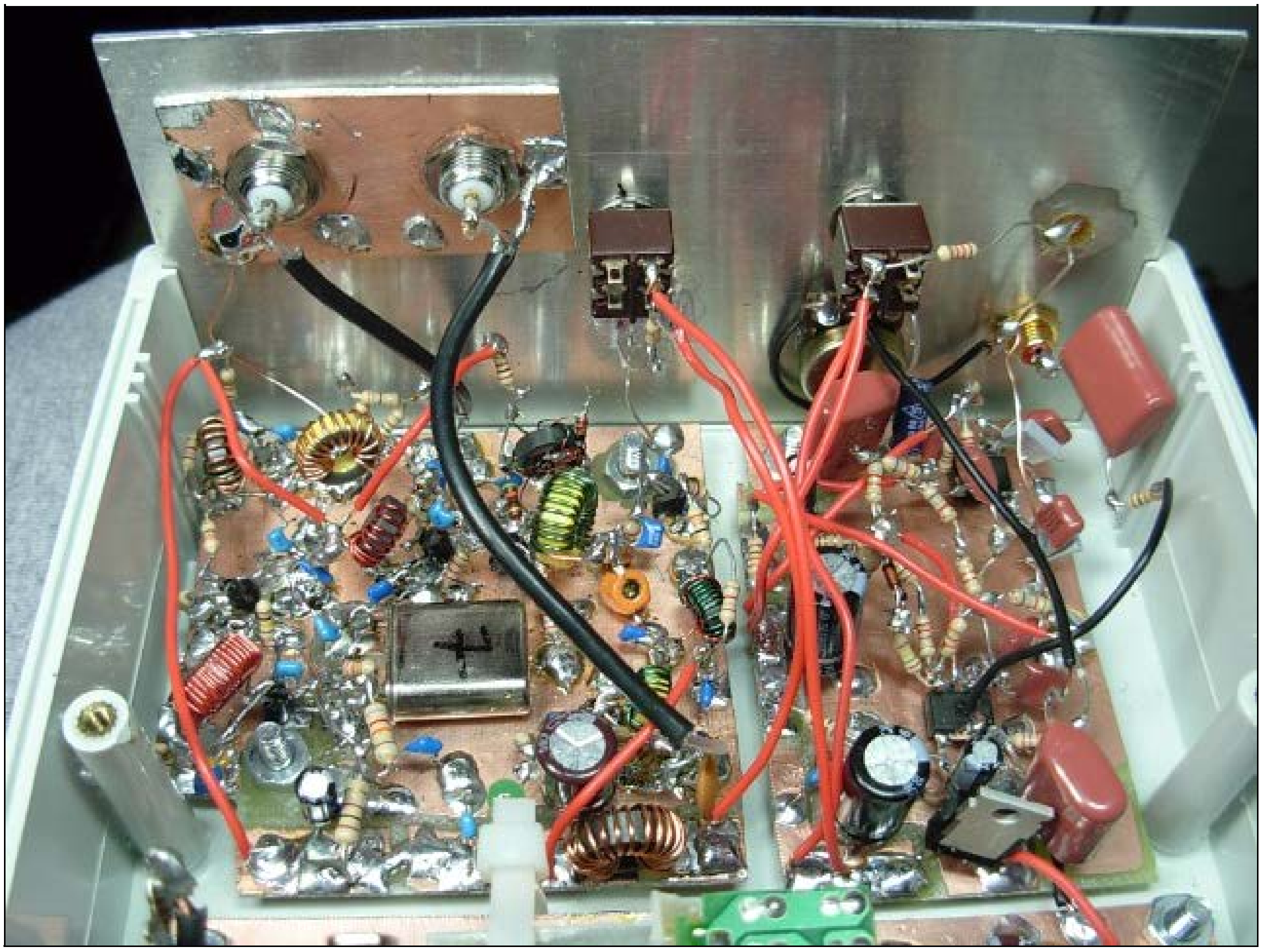


Figure 8 are Rigol digital screen captures. Figure 8A is a measurement taken from the anode at point DX from Figure 6. Figure 8B is a measurement taken at the cathode of DX and shows distortion caused by the diodes, reduced AC voltage and of course, frequency doubling.



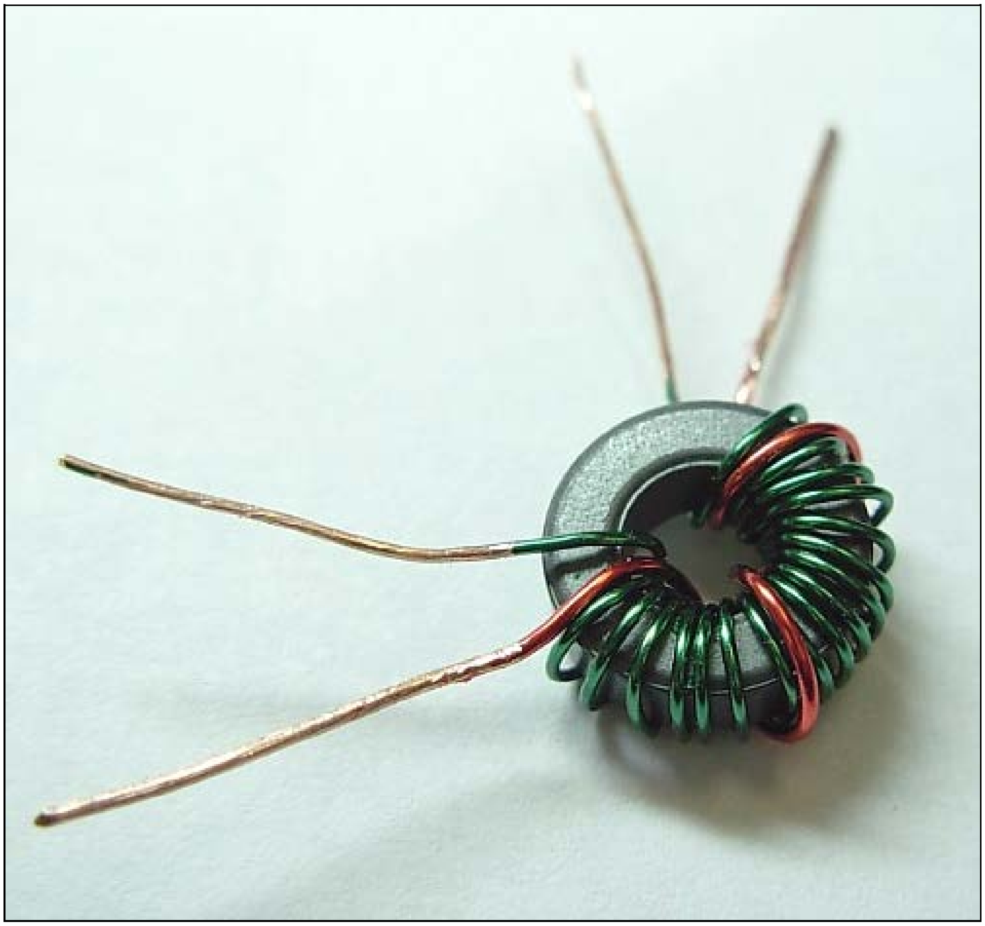
The 7 and 14 MHz circuits bread boarded and mounted in a chassis. The 7.039 MHz and 14.079 MHz outputs are connected to BNC jacks via RG-174 cable. The 15.3 MHz Chebyshev low-pass filter and -4 dB attenuator pads are on a raised Ugly Construction board. Some VCC decoupling parts are also on the bottom RF board.



The 7 and14 MHz crystal oscillator board mounted in a project chassis along with a 1 KHz low noise oscillator This photo was a prototype version that did not have a Chebyshev low-pass filter after the 14 MHz stage.

Front view of the .001, 7 & 14 MHz oscillator. It is really fun to build your own test equipment.

Transformer Notes and Conclusion



A photo of T5 from Figure 6. Some builders have emailed and stated they do not like to wind inductors/ transformers. I always ask them why? Often these builders were concerned with little details such as wire gauge and spacing, choosing the core size and which magnetic material to use. The Radio Amateur literature is replete with great tutorials on winding coils using toroids. Truly; the more toroids you wind, the easier it gets. Here are some simple points for beginners:

Powdered iron toroids are generally for tuned circuits. I.e. A capacitor and the inductor are tuned to a center frequency. Powdered irons containing the #2 and #6 material tend to tune sharply and have fairly high Q

Ferrites toroids are generally for use in broadband or wideband (untuned) applications. #43 material is relatively low Q and lossy as compared to the number #2 and #6 powdered iron toroids

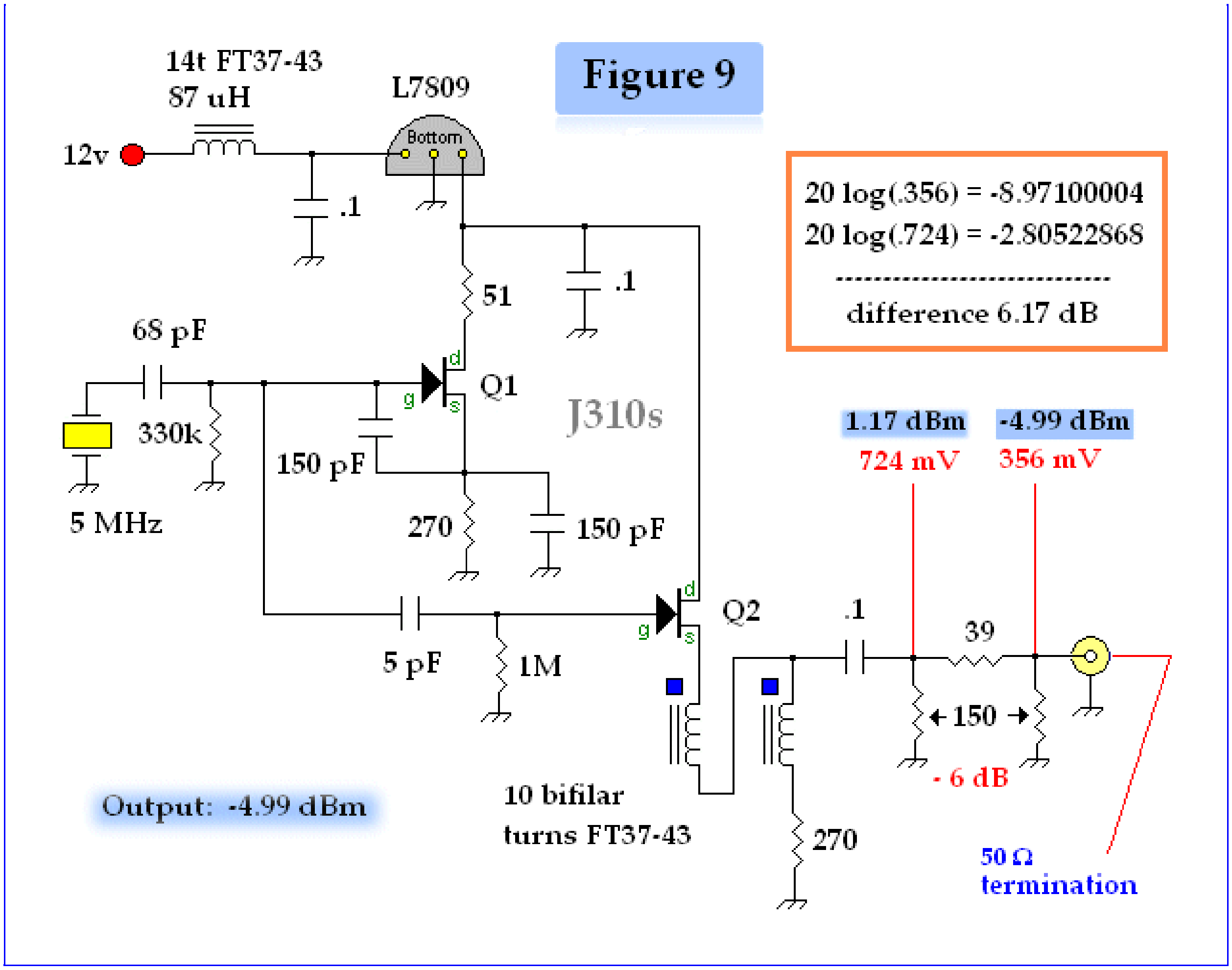
Wind your inductors with enamel coated magnet wire. Popular gauges include 28, 26 and 24, but this is quite variable.

Minimally, you could get by with just #43 ferrite and #6 powdered iron toroids. For example, FT37-43 ferrites and T50-6 plus T68-6 powdered iron toroids could build a lot of inductors/transformers. In the photo above, I used #24 AWG wire for the 18 turns and #22 AWG wire for the 3-turn link. The 3 turn link is grounded on one end and well anchors the transformer. Thicker wire was chosen because Ugly Construction was used and the part is really anchored with the #22 AWG wire.

Conclusion

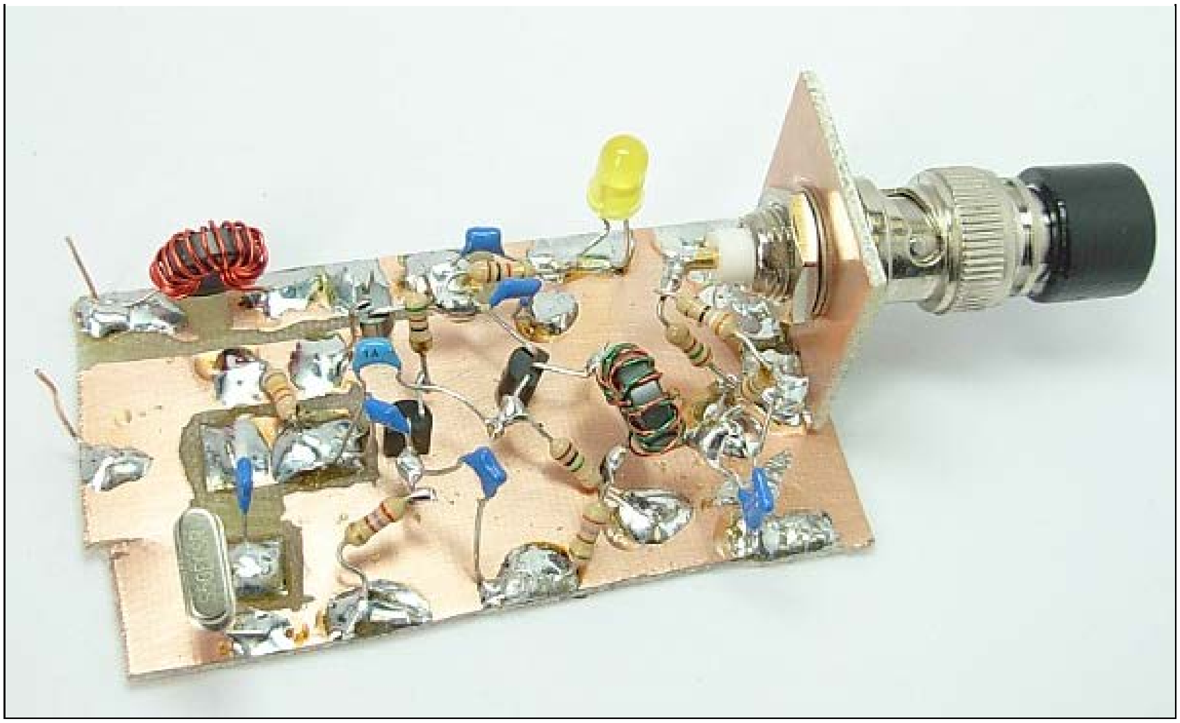
It took quite a lot of experimentation to reach the project goal. The experience was pleasant and comparing the old and new oscilloscopes was an added bonus. Perhaps, the circuits are overly complicated, however, they are critical signal generators for my workbench. Well designed signal generators have extensive RF-proof shielding to allow serious attenuation. These will have to do.

Addition - Jan 31, 2010 5 MHz JFET Low Noise Oscillator



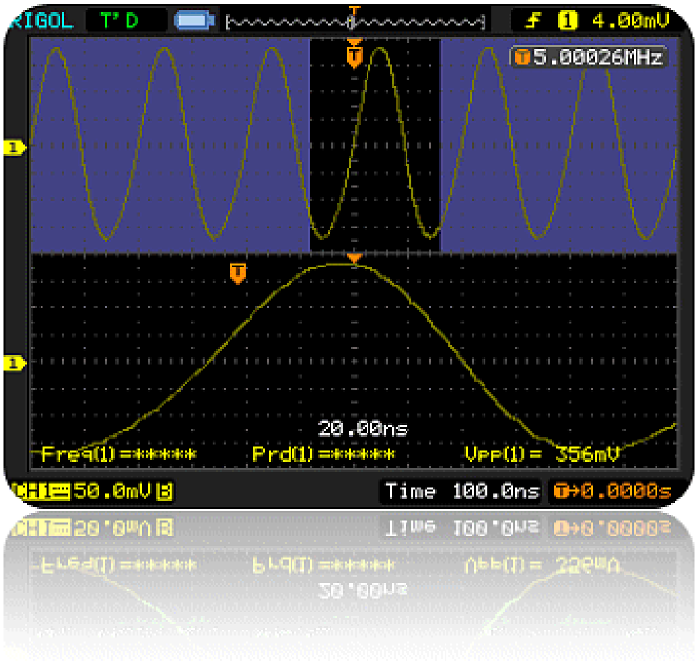
Another experimental low noise oscillator was built for 5 MHz. The desired output power was -5 dBm. The schematic and peak-peak output voltages and powers are shown in Figure 9. The 68 pF capacitor in series with the crystal was found experimentally by using a variable capacitor and measuring the output voltage and observing the scope waveform. When these appeared to be optimal, the trimmer capacitor was removed and measured at 67.17 pF. This circuit can easily be adjusted to give higher output power such as 7 dBm. To increase power, you may consider increasing the VCC to 12 volts, add a 0.1 uF source bypass capacitor in parallel with the 270 Ohm resistor of Q2 and/or adjust the pad attenuation. There are other circuit alterations to increase power, but the aforementioned are a good start.

It is important to measure your output voltages with a 50 ohm load connected to the device. Standard value resistors were used which throw off the value somewhat, but the actual attenuation of the pad is very close to the 6 dB attenuation expected. The math can be done with software such as Applets H and I on the QRP tools web page.



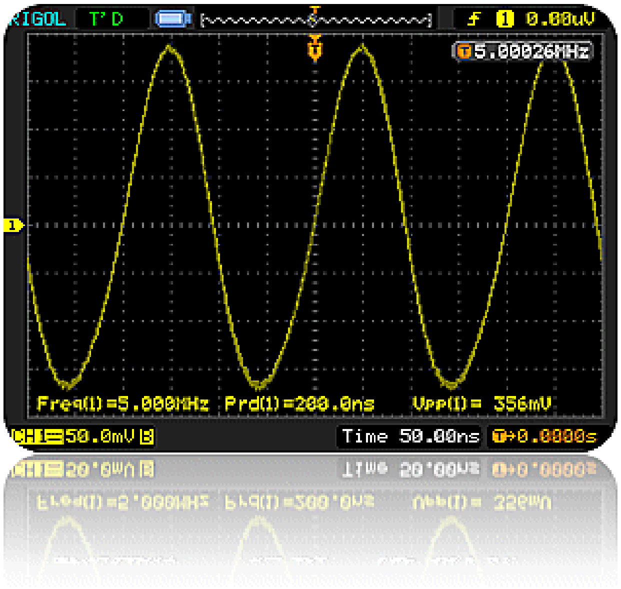
The breadboard of Figure 9 is shown above. The - 5 dBm output was required for some upcoming experiments and to study log and power measurements. The black BNC appliance is a 50 ohm load. These are very handy, but not a necessity. In most cases, a simple 51 ohm resistor to ground is the load.

Figure 10



A screen capture from Figure 9 signal viewing maneuvers.

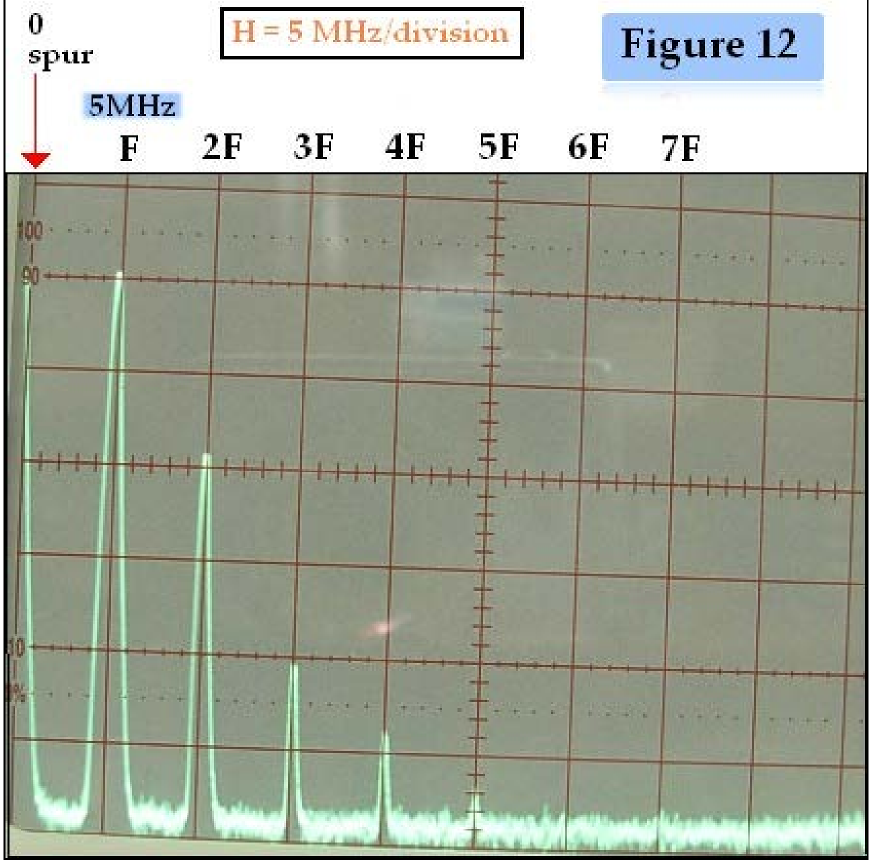
Figure 11



The output waveform looks good - even on a DSO!

Spectrum Analysis

Figure 12



The output of the Figure 9 oscillator was further attenuated -19 dB for analysis in a spectrum analyzer. The -19 dB was chosen as this 50 ohm pad uses near standard resistor values and would well ensure a very safe signal level for the spectrum analyzer. The pad is shown in the Figure 14 schematic. The second harmonic (2F) is down about 20 dB from the fundamental. Each horizontal division is 5 MHz and each vertical division is 10 dB. Compared to other more conventional oscillators that were checked in the spectrum analyzer, this oscillator has low harmonic content.

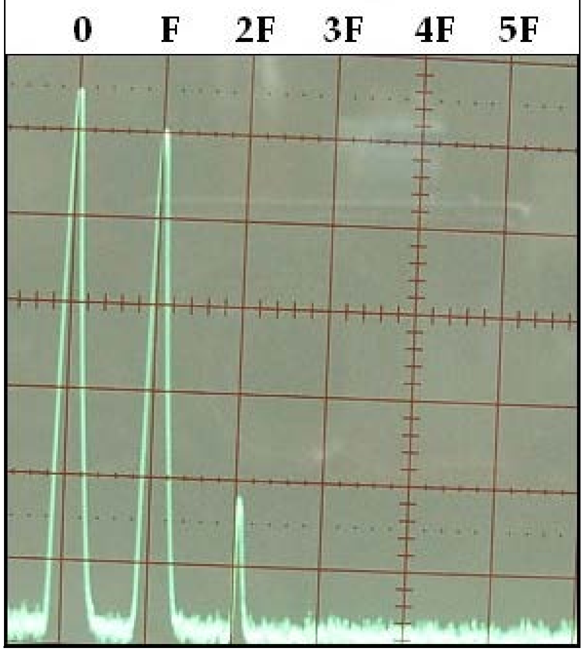
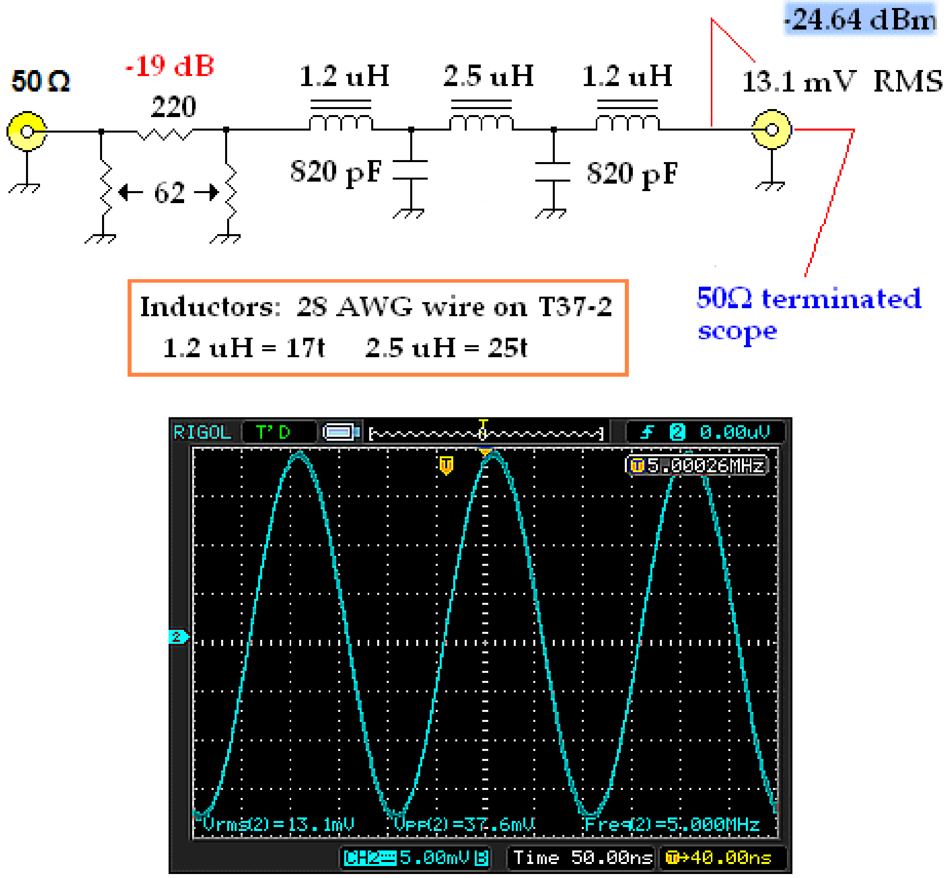


Figure 13

A N=5 or 5 element Chebyshev low-pass filter was placed after the Figure 9 oscillator and connected to the spectrum analyzer to see what happens. The 3 dB down frequency of this low-pass filter was 6.53 MHz. The second harmonic is now ~42 dB down. There are no measureable harmonics after 2F. It is really cool to "see" what a filter does to a signal.

Figure 14



The schematic of the -19 dB attenuator pad and the low-pass filter is shown. To measure the output, the 10X probe was disconnected. The circuit was connected to the oscilloscope via 50 ohm coaxial cable and the scope input was terminated with 2 paralleled 100 ohm resistors. The 50 ohm scope termination technique will be discussed in a future web page addition. The RMS voltage values were inputted into Applet J on the QRP Tools page to calculate the output power. The RMS output was 0.131 volts which calculated to an output power of -24.6 dBm.

Crystal Parameter Checker

Introduction

This web page is a supplement to JavaScript Applet **G** on this web page. This software does the math using a simpified version of the method to determine motional inductance and capacitance developed by David, G3UUR. This is a very basic tutorial meant as an introductory guide for novice builders.

Shown in the photo above is 1 of my crystal parameter checkers. The schematic may be found in many places including EMRFD Figure 3.35 (See Errata) and on this pdf by Nick, WA5BDU. A power indicator LED has been added, but the circuit is the standard design. In this breadboard, the crystal being measured is tack soldered in.

Many builders just copy other builder's I.F. filter schematics, however, your crystal filters will perform better if your design is based upon the exact parameters of the crystals you have. For the simple design or optimization of a crystal filter, it is necessary to measure crystal parallel capacitance plus take other measurements to calculate motional inductance and capacitance. Determining your crystal parameters is not difficult if you have a capacitance meter, a frequency counter and some math skills. It is easiest to use a program to crunch the math; hence I wrote a stupid-simple JavaScript applet.

Designing filters is another story; it takes knowledge, practice and good software for this. Filter design theory has been extensively covered by Anatoly Zverev, Wes Hayward and others. The work of Nick, WA5BDU is also greatly appreciated. His presentations and references are excellent for those keen on learning more about filter design.

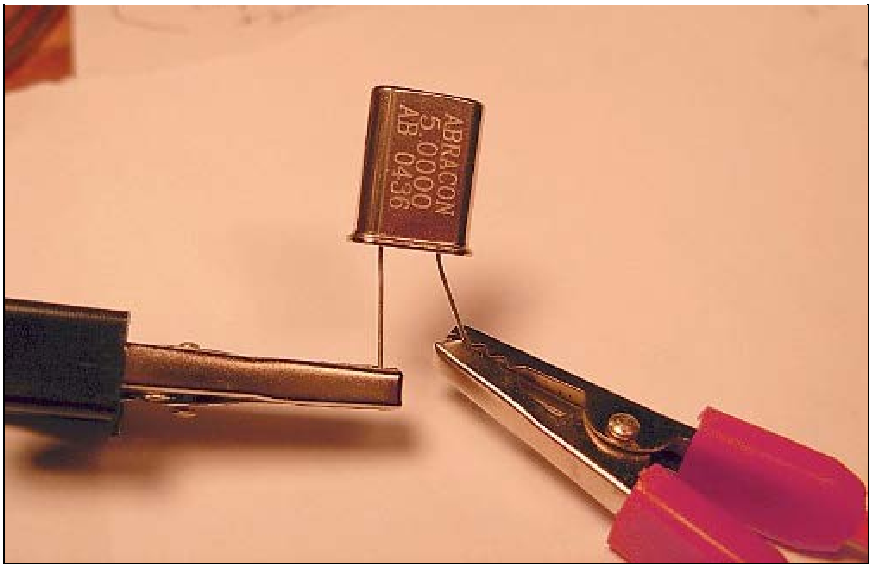
A four crystal 5.00 MHz SSB I.F. filter was desired. 20 crystals were on hand — they were from the same batch. The crystals were all placed in the above oscillator and their frequency was measured. The 4 crystals closest matching in frequency were set aside. The crystal parameters of these 4 were then determined. Typically these values are averaged and this average is used to design or tweak the filter using software.

---

1. Measure Capacitance

The procedure for determining the parameters of 1 crystal is described.

The first step is to measure the crystal capacitance (called parallel capacitance) using a capacitance meter.



Measuring the crystal parallel capacitance

Parallel capacitance of the above 5.0 MHz crystal in an [AADE LC meter]()

Next, measure the capacitance of the open switch plus the 33 pF fixed value cap wired in-situ. This will give you the total circuit capacitance of the open switch, the 33 pF fixed value capacitor, and any stray capacitance from your crystal holder, wires, etc. The switch itself plus stray wiring will be a few pF so the total should be around 36 to 40 pF or so. In my test oscillator, the result = 41.19 pF as shown. On my other crystal checker with a better switch, it's 36.9 pF.

In the calculation of crystal Lm and Cm, the parallel capacitance and the switch circuit capacitance will be summed.

---

2. Measure Frequencies

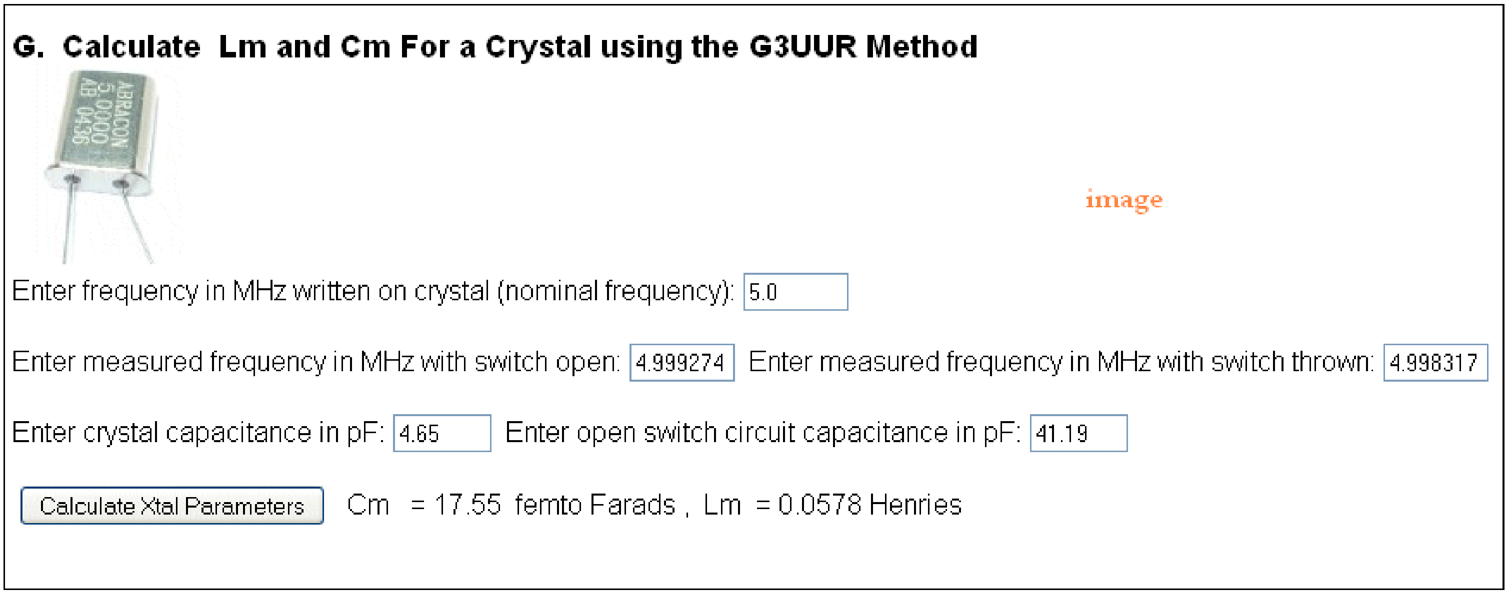
A crystal is put in the oscillator with the switch open. Record the frequency. Your counter must have resolution down to 1 Hertz. After recording this value, throw the switch and measure and record this frequency. You now have all the measured values required to calculate motional parameters and adjust or design a filter. Motional parameters are calculated in [Applet G]().

Frequency measured with the switch open = 4.999274 MHz

Frequency measured with the switch thrown = 4.998317 MHz

---

3. Do Math by Hand or with Software



G. Calculate Lm and Cm For a Crystal using the G3UUR Method

Enter frequency in MHz written on crystal (nominal frequency): 5.0

Enter measured frequency in MHz with switch open: 4.999274 Enter measured frequency in MHz with switch thrown: 4.998317

Enter crystal capacitance in pF: 4.65 Enter open switch circuit capacitance in pF: 41.19

Cm = 17.55 femto Farads , Lm = 0.0578 Henries

The applet **G** calculation of the crystal parameters using the above measured values

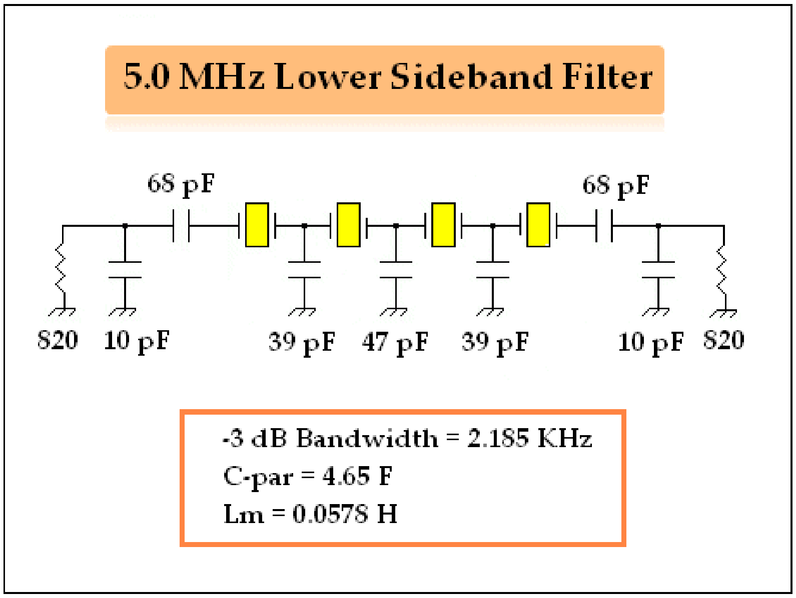
---

4. Example Filter Adjustment

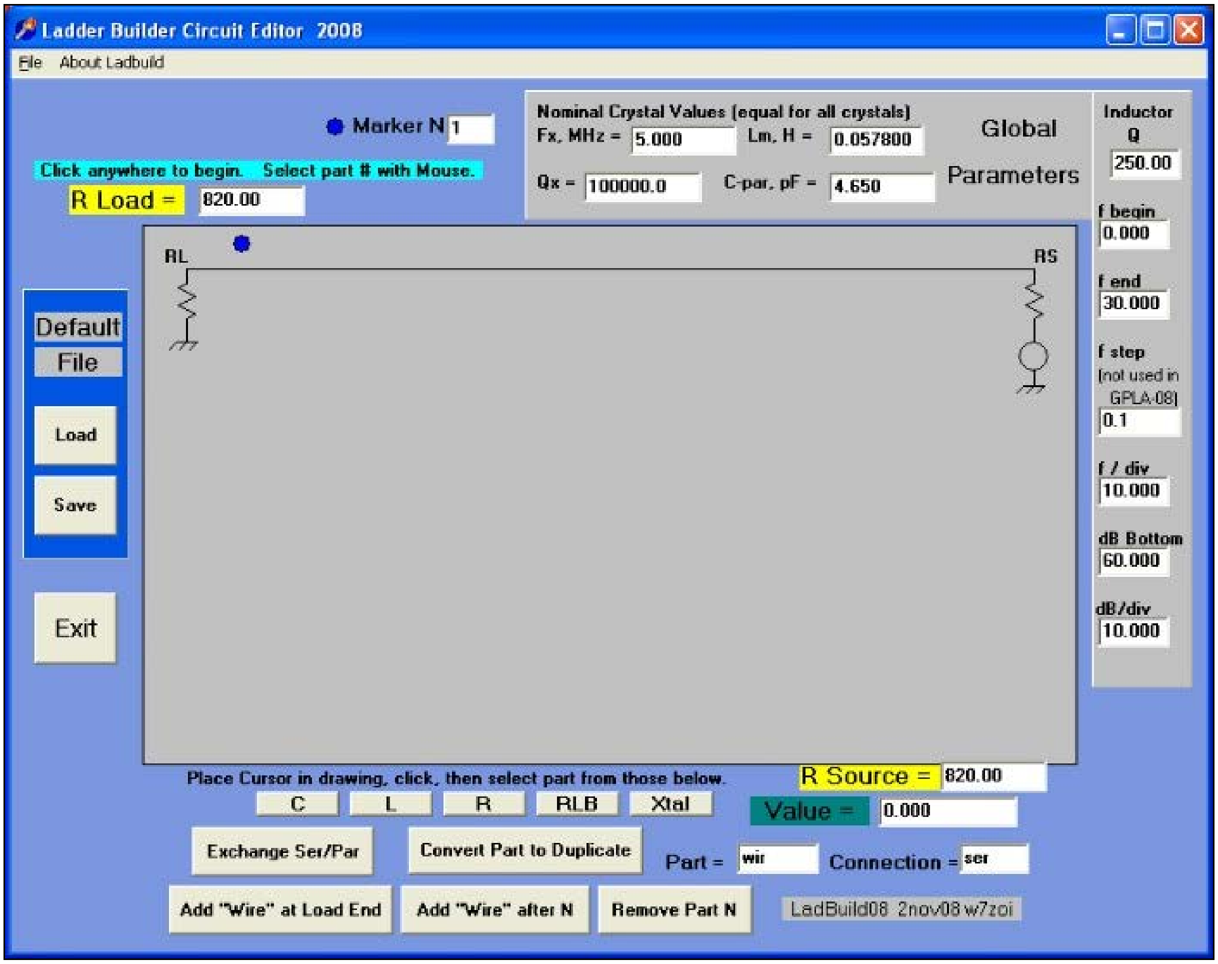
It is assumed that most builders will use software to design or tweak their crystal filters. The only 2 programs tried *(to date)* include AADE Filter Design and the Ladpac software collection that supplements EMRFD. I am more familiar with the Ladpac programs written by Wes, W7ZOI. Only these programs are demonstrated. Please read the instructional file Ladpac2008 Manual.pdf to understand these programs. *The Ladpac software bundle includes GPLA.*

The purpose of this tutorial is not to teach crystal filter design, but to describe a relatively simple method to tweak an existing design using your measured crystal parallel capacitance and its calculated motional inductance.

The first step is to digitally format your filter into a file that can be analyzed in GPLA. In my opinion, the easiest way to do this is to use the ladder circuit editor ladbuild02.exe or better yet, its update - ladbuild08.exe. The model filter follows:



The model 5 MHz SSB filter



Marker N 1

R Load = 820.00

Fx, MHz = 5.000

Lm, H = 0.057800

Qx = 100000.0

C-par, pF = 4.650

Inductor Q 250.00

f begin 0.000

f end 30.000

f step (not used in GPLA-08) 0.1

f / div 10.000

dB Bottom 60.000

dB/div 10.000

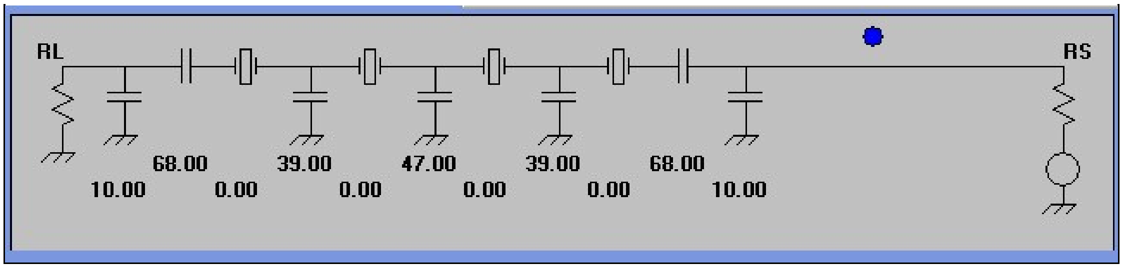
R Source = 820.00

Value = 0.000

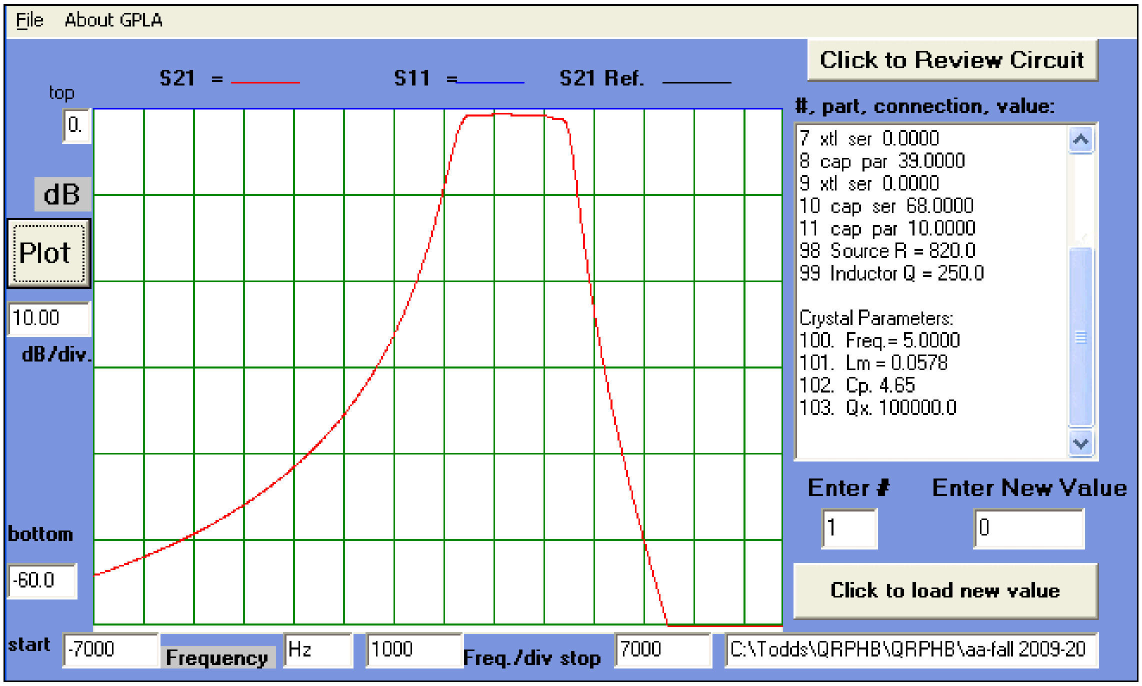
Part = wir

Connection = ser

Clear any existing components and enter the termination R, C-par and Lm values. Qx is set at 100000



Build your filter within the editor. Save your work. Start up GPLA and load your newly saved filter.



top: 0.
10.00 dB/div.
bottom: -60.0
start: -7000
Hz: 1000
stop: 7000
C:\Todds\QRPHB\QRPHB\aa-fall 2009-20

#, part, connection, value:

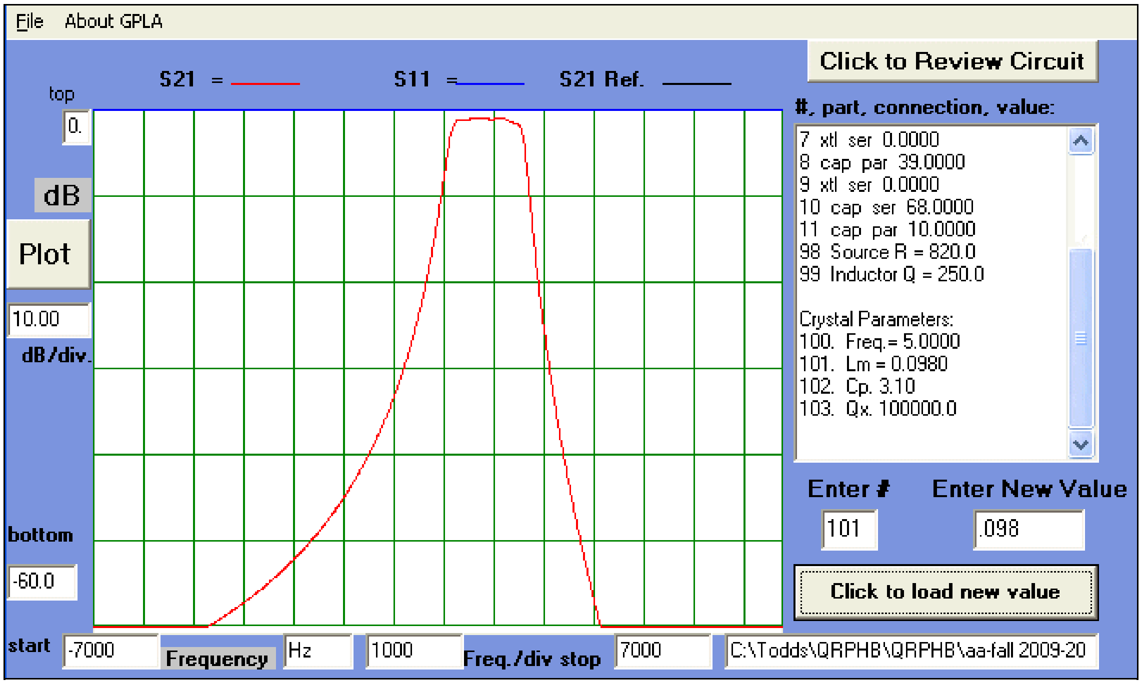
7 xtl ser 0.0000
8 cap par 39.0000
9 xtl ser 0.0000
10 cap ser 68.0000
11 cap par 10.0000
98 Source R = 820.0
99 Inductor Q = 250.0

Crystal Parameters:
100. Freq.= 5.0000
101. Lm = 0.0578
102. Cp. 4.65
103. Qx. 100000.0

Enter #: 1
Enter New Value: 0

Set a sweep and **x** axis increment (-7000, 1000 and 7000 in this example). Push the Plot button

Let's say you wanted to use this filter design and have determined the average parameters of the 5.00 MHz crystals in your parts collection. Let's assume that for your crystals, C-par = 3.1 pF, and Lm = 0.098H. Input these values in GPLA.



top: 0.
10.00 dB/div.
bottom: -60.0
start: -7000
Hz: 1000
stop: 7000
C:\Todds\QRPHB\QRPHB\aa-fall 2009-20

#, part, connection, value:

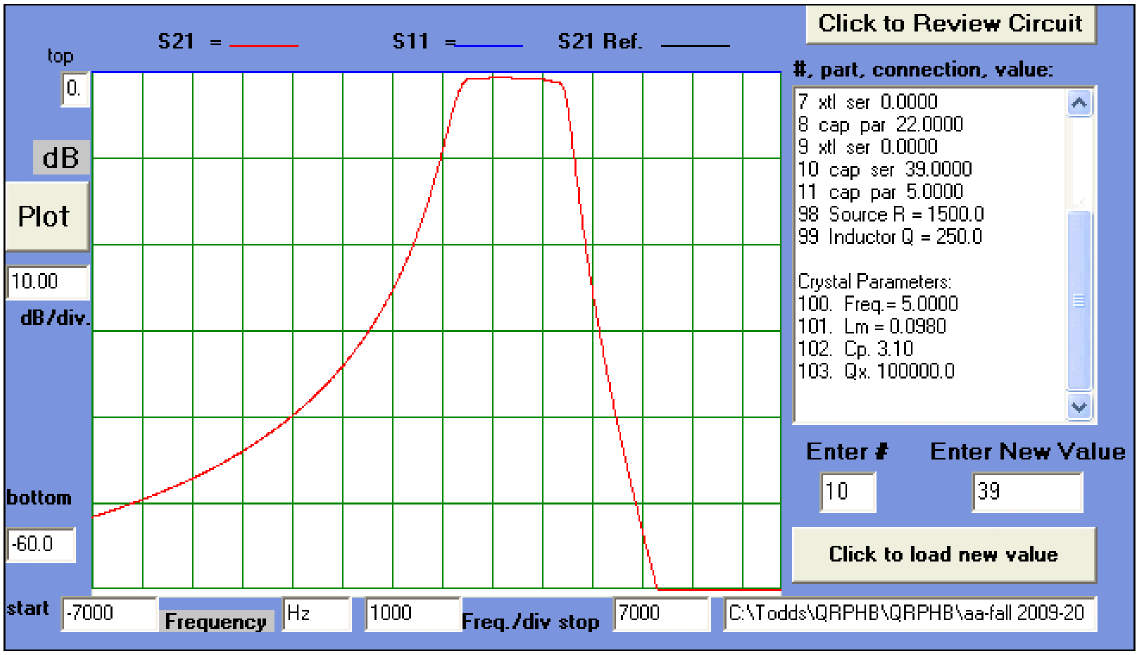
7 xtl ser 0.0000
8 cap par 39.0000
9 xtl ser 0.0000
10 cap ser 68.0000
11 cap par 10.0000
98 Source R = 820.0
99 Inductor Q = 250.0

Crystal Parameters:
100. Freq.= 5.0000
101. Lm = 0.0980
102. Cp. 3.10
103. Qx. 100000.0

Enter #: 101
Enter New Value: .098

Look what happened to the crystal filter's bandwidth. Our - 3dB bandwidth is now somewhere around 1464 Hertz. This simple experiment illustrates how important it is to use the parameters of <u>your</u> crystals to obtain a desired filter response.

Experiment with the various functions in GPLA to learn how to use it. Set whatever reasonable sweep you want. This program is best learned by using it repeatedly.



top: 0.

10.00 dB/div.

bottom: -60.0

start: -7000 Frequency: Hz Freq./div: 1000 stop: 7000

C:\Todds\QRPHB\QRPHB\aa-fall 2009-20

#, part, connection, value:

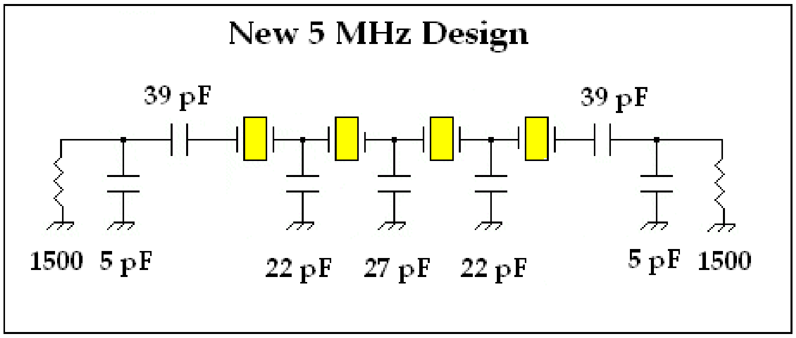
```
7 xtl ser 0.0000
8 cap par 22.0000
9 xtl ser 0.0000
10 cap ser 39.0000
11 cap par 5.0000
98 Source R = 1500.0
99 Inductor Q = 250.0

Crystal Parameters:
100. Freq.= 5.0000
101. Lm = 0.0980
102. Cp. 3.10
103. Qx. 100000.0
```

Enter #: 10

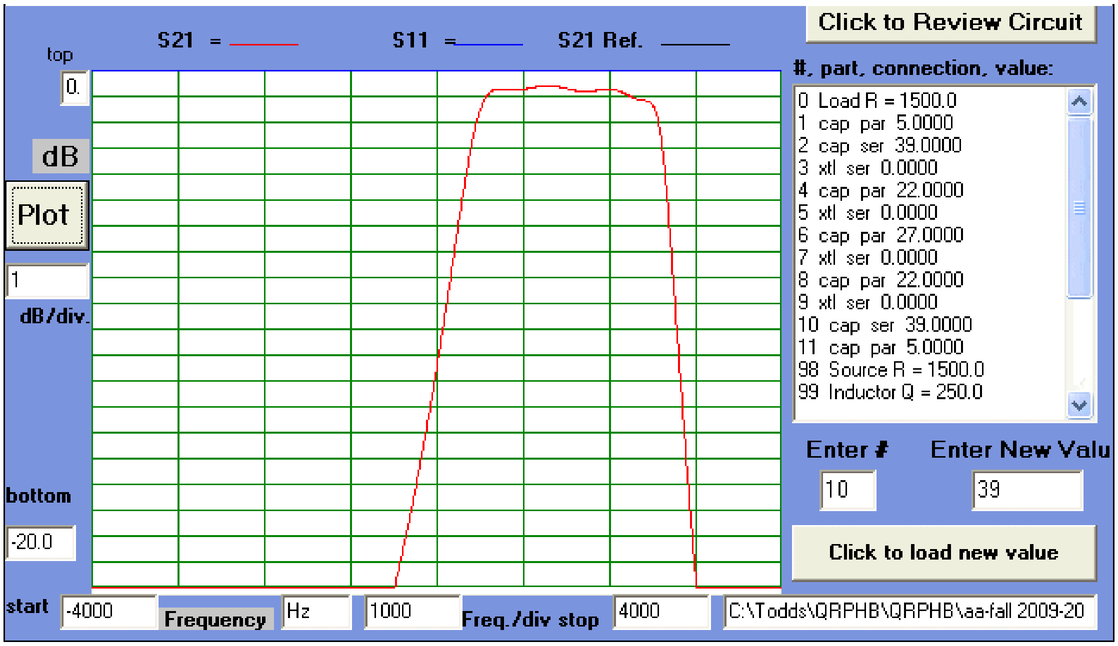
Enter New Value: 39

In the above screen capture, the above filter was tweaked to "re-establish" a -3dB bandwidth of ~2.172 KHz. All adjustment was performed entirely in GPLA by swapping capacitor values and observing the resultant waveform. When you get an overall pleasing bandwidth plus shape, but there is too much ripple at the top, generally you must increase the terminating R values. This is the brute-force, manual way to tune filters. For this method, you need not understand terminology such as as series resistance, MESH, K or Q values, Butterworth response, or Chebyshev with 0.1 dB of ripple.

Admittedly, at first, this method can be quite time consuming and tedious, however, with practice, you may be able to tweak a filter in only a few minutes. Clearly, the more you dig into understanding crystal filter design, the better your filters can be, however, getting overly complex can scare off builders who are new to this hobby. Note these filters use standard value capacitors and resistors; perfect for popcorn I.F. filters.



The original 5 MHz Model filter with updated C and R values using Lm = 0.098 and C-par = 3.1



top: 0.
dB/div.: 1
bottom: -20.0
start: -4000 Hz 1000 4000
C:\Todds\QRPHB\QRPHB\aa-fall 2009-20

#, part, connection, value:

```
0  Load R = 1500.0
1  cap  par  5.0000
2  cap  ser  39.0000
3  xtl  ser  0.0000
4  cap  par  22.0000
5  xtl  ser  0.0000
6  cap  par  27.0000
7  xtl  ser  0.0000
8  cap  par  22.0000
9  xtl  ser  0.0000
10  cap  ser  39.0000
11  cap  par  5.0000
98  Source R = 1500.0
99  Inductor Q = 250.0
```

Enter #: 10
Enter New Valu: 39

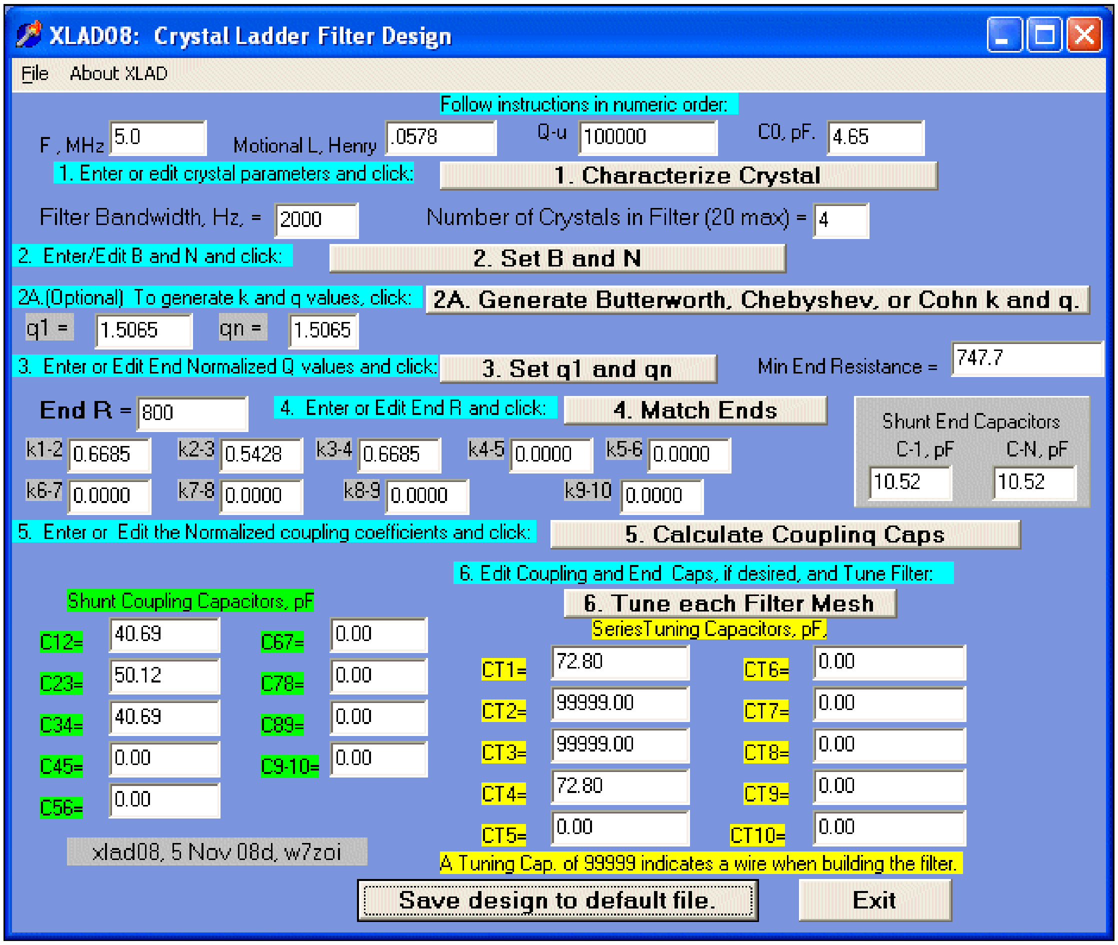
GPLA zoom of the **Y** axis showing the first 20 dB of attenuation.

---

5. The Model 5 MHz SSB filter Design

Although this page is not about crystal filter design, an example follows for reference purposes.

For designing filters, the application xlad08.exe is a good choice. The following 3 screen captures show the raw design process and GPLA analysis of the model 5 MHz Lower Sideband filter shown earlier with C-par = 4.65 pF and Lm = .0578. There are some great articles in print and on the Web to study if you want to learn about filter design. The Ladpac software from EMRFD is excellent. My special thanks to Wes, W7ZOI for answering my questions about his software. From this information, I was able to make this web page.



Follow instructions in numeric order:

F , MHz 5.0   Motional L, Henry .0578   Q-u 100000   C0, pF. 4.65

1. Enter or edit crystal parameters and click:

Filter Bandwidth, Hz, = 2000   Number of Crystals in Filter (20 max) = 4

2. Enter/Edit B and N and click:

2A.(Optional) To generate k and q values, click:

q1 = 1.5065   qn = 1.5065

3. Enter or Edit End Normalized Q values and click:

Min End Resistance = 747.7

End R = 800

4. Enter or Edit End R and click:

Shunt End Capacitors
C-1, pF 10.52
C-N, pF 10.52

k1-2 0.6685   k2-3 0.5428   k3-4 0.6685   k4-5 0.0000   k5-6 0.0000
k6-7 0.0000   k7-8 0.0000   k8-9 0.0000   k9-10 0.0000

5. Enter or Edit the Normalized coupling coefficients and click:

6. Edit Coupling and End Caps, if desired, and Tune Filter:

Shunt Coupling Capacitors, pF

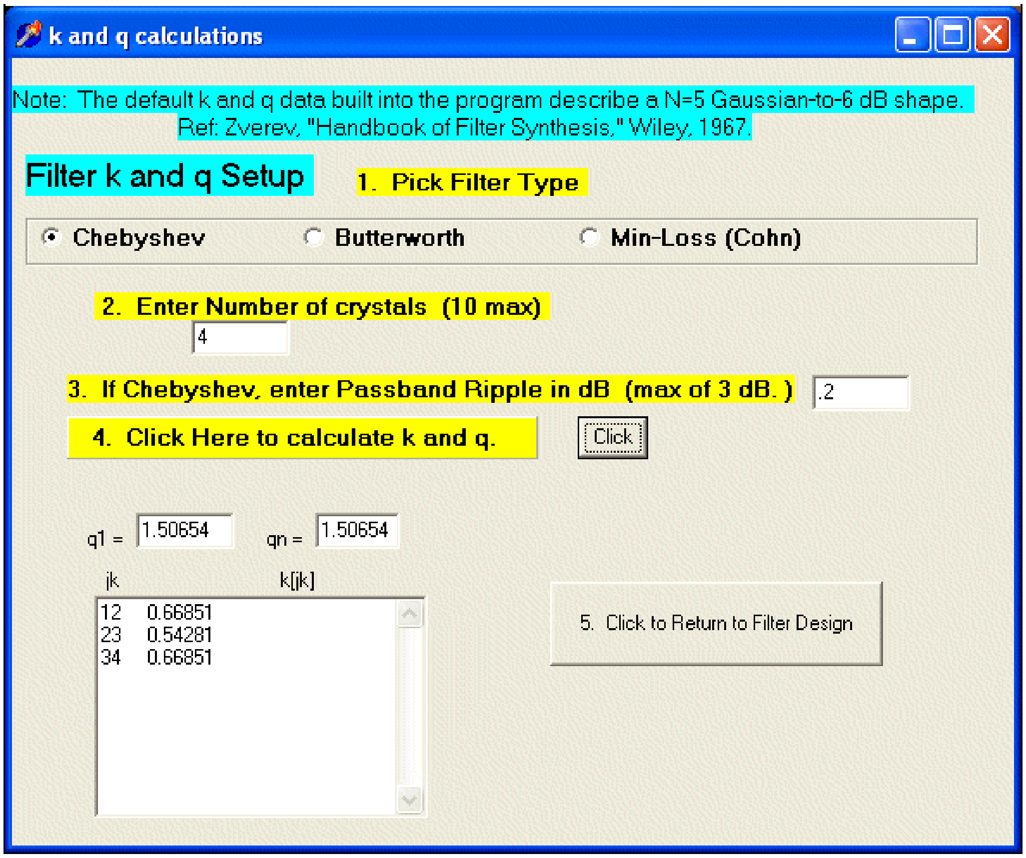
C12= 40.69   C67= 0.00
C23= 50.12   C78= 0.00
C34= 40.69   C89= 0.00
C45= 0.00   C9-10= 0.00
C56= 0.00

SeriesTuning Capacitors, pF,

CT1= 72.80   CT6= 0.00
CT2= 99999.00   CT7= 0.00
CT3= 99999.00   CT8= 0.00
CT4= 72.80   CT9= 0.00
CT5= 0.00   CT10= 0.00

xlad08, 5 Nov 08d, w7zoi

A Tuning Cap. of 99999 indicates a wire when building the filter.



k and q calculations

Note: The default k and q data built into the program describe a N=5 Gaussian-to-6 dB shape.
Ref: Zverev, "Handbook of Filter Synthesis," Wiley, 1967.

Filter k and q Setup

1. Pick Filter Type

● Chebyshev ○ Butterworth ○ Min-Loss (Cohn)

2. Enter Number of crystals (10 max) 4

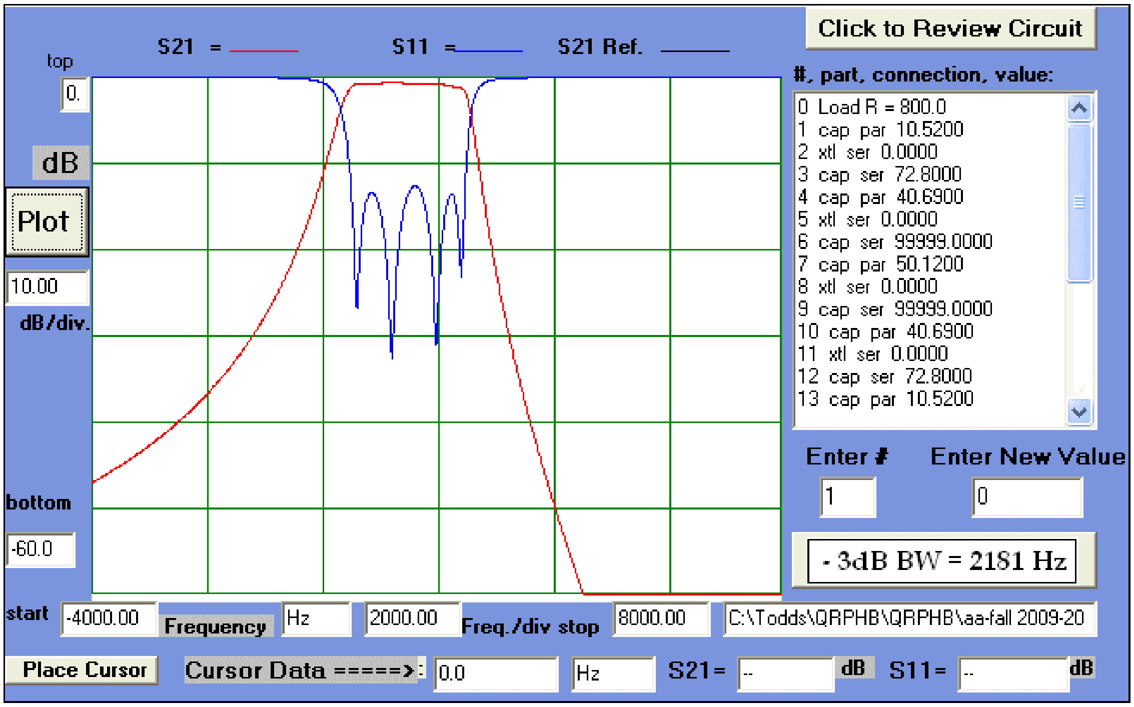
3. If Chebyshev, enter Passband Ripple in dB (max of 3 dB.) .2

4. Click Here to calculate k and q. Click

q1 = 1.50654 qn = 1.50654

jk	k[jk]
12	0.66851
23	0.54281
34	0.66851

5. Click to Return to Filter Design



top 0.

10.00 dB/div.

bottom -60.0

start -4000.00 Frequency Hz 2000.00 Freq./div stop 8000.00 C:\Todds\QRPHB\QRPHB\aa-fall 2009-20

Cursor Data 0.0 Hz S21= -- dB S11= -- dB

#, part, connection, value:

0 Load R = 800.0
1 cap par 10.5200
2 xtl ser 0.0000
3 cap ser 72.8000
4 cap par 40.6900
5 xtl ser 0.0000
6 cap ser 99999.0000
7 cap par 50.1200
8 xtl ser 0.0000
9 cap ser 99999.0000
10 cap par 40.6900
11 xtl ser 0.0000
12 cap ser 72.8000
13 cap par 10.5200

Enter # 1 Enter New Value 0

- 3dB BW = 2181 Hz

---

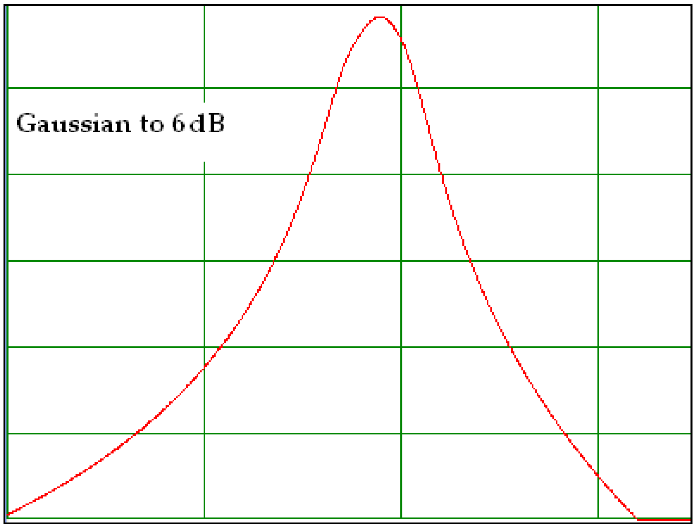
6. Conclusion

This web page presents a brief method to calculate crystal motional parameters and as required, to adjust crystal filter circuits to function optimally. This approach like my Java-script applet are simplistic to avoid the fear factor associated with crystal ladder design.

Listening tests are also valuable for assessing crystal filter function. Is the bandwidth as you expected? Does the filter ring excessively? Does it sound tinny? In the recent past, the crystal filters in 2 kit receivers/transceivers were tweaked as a favor to friends. *Please note, I have total respect for people who sell kits and appreciate the contribution they make to our hobby.*

The crystals of these Cohn type filters were removed and analyzed and the bandwidth was not as specified. In 1 case, the filter shape looked terrible in GPLA. Clearly, the kit sellers provided crystals which had markedly different parameters from those used by the original circuit designer. The I.F. filter capacitors were replaced with appropriate values and the R values were adjusted via either resistors and/or transformer ratios and the improved filters sounded pleasant.

It is a real treat to listen to a receiver with a well designed crystal filter — sadly, I don't enjoy this experience that often.



In my opinion, the best sounding CW crystal filter design is the Gaussian to 6 dB. Some operators would never use such a filter in a contest-grade receiver as the filter skirts are not steep enough for them. There are tricks to make the stop band better (more like a Chebyshev response, but without the ringing), however, this topic is out of scope.

I sincerely ask for your feedback on the G. JavaScript Applet. Does it work correctly? How could it be improved? Can you contribute better code? Thanks and 73.

QRP — Posdata for January 2012 — More on Crystal Ladder Design

Important to both superheterodyne receivers and single sideband transmitters, crystal ladder filter design lies juxtaposed as both a favorite and feared RF design topic. Newer builders may lack math skills, and/or become paralyzed by the terminology — or lack the ambition to learn or apply good bench practices. Even a cursory Internet search returns many fabulous files to read — witnessing a crystal ladder filter design article explosion.

The difficulty characterizing and building filters has progressively declined since the advent of the first handheld computers — improvements in personal computers and filter design software allows astute builders to pursue even complex xtal filter response shapes in 2012.

In QST for July 1987. Wes, W7ZOI wrote **Designing and Building Simple Crystal Filters**. This article promoted Cohn or Min-loss filters and its intent was to transcend the math and measurement associated with xtal ladder filters of the day and allow builders to just frequency match some crystals, and then go experiment.

In my estimation, this article proved revolutionary — soon after, builders around the globe, Elecraft, and other kit companies embraced this technique/topology and the rest is history. *(I call it the paper that launched a thousand kits).* If you're a new builder and feel overwhelmed by the material on or referenced by this web page, please consider first obtaining this article.

Learning about crystal ladder filter design is time well spent.

In 2011/2012 I explored the works of 3 builders who share their work via the web and/or journals.

Horst Steder, DJ6EV and Jack Hardcastle, G3JIR

The Steder and Hardcastle works emphasize that we need to measure/calculate crystal L and C parameters, plus the coupling and tuning capacitances (not just frequency). Through emails with Horst, DJ6EV, I learned many things, but 3 stand out:

1. It's better to design a good filter than fix a bad one.
2. Careful measurement of your crystal parameters plus software design and simulation = the best chance for getting your desired performance.
3. Deriving motional parameters from a 3 dB bandwidth measurement remains a great way to characterize multiple xtals. *In my opinion, the G3UUR method is the easiest way to characterize a small batch of xtals.*

Some of the earliest references to modern crystal ladder design I've found were written by Jack Hardcastle and published in RadCom and QST — I later confirmed this by reading work by Wes, W7ZOI and others. Hardcastle's and Haywards' work proved foundational for the experimenters that followed including David Gordon-Smith, Chris Trask, Jim Kortge and many others.

Steder and Hardcastle's combined experience assessing and/or documenting crystal ladder design spans decades.

Their QEX article **Crystal Ladder Filters for All** may be legally downloaded from the ARRL website here.

Program download URL: http://www.arrl.org/qexfiles (select 2010, "11x09_Steder_Hardcastle.zip")

The QEX article describes Steder's Microsoft Windows™ program, methods to derive motional parameters, plus cites many important references. The main program calculates practical lower-sideband crystal ladder filters based on the exact equations published by M. Dishal in 1965. Hardcastle transformed these equations into a computer useable form in 1983 and Steder incorporated these equations into a modern, easy-to-use and interactive application with nice graphing and table displays.

For simplicity, the program assumes lossless crystals, however, the calculated values can easily be transferred into another simulation program such as GPLA to add or refine parameters such as loss resistance.

The main "Dishal" application calculates filters with Butterworth, Chebychev and constant-k (Cohn) properties and the so-called "QER" filter type by G3UUR (a low ripple version of the Cohn filter). Further; sub-programs in the top-level menu calculate xtal parameters (by both the G3UUR and 3 dB method), plus L-C impedance matching and ladder termination networks. An extensive help file well explains the program.

Iacopo Giangrandi — Introduction to Crystal Ladder Filters

Link: http://www.giangrandi.ch/electronics/crystalfilters/xtalintro.html

Iacopo (Jack) uses a transmission measurement to infer the motional parameters — inserting a series capacitance and measuring the series resonant frequency shift was also described in 1998 by Rolf-Dieter Mergner, DJ9FG.

His web article/applications provides what is likely needed by most builders — simple filter synthesis while avoiding expensive test gear. His filters plots/figures are spectrum analyzer measurements that I really like. Although his program can generate aymmetrical filters that some builders might not be used to, the frequency domain plots indicate proper function.
Giangrandi's filter design programs appear to be based on Jack Hardcastle's work and possibly content published in a paper by Patrick Magnin, F6HYE and Bernard Borcard, F3BB in Radio REF for April 1990.

I encourage you to try all the methods and applications mentioned to discover what works best for you. Don't lose heart, for characterizing crystals with a vector network analyzer is also a time-consuming endeavor and simple often = best on the QRP Workbench.

QRP — Posdata for August 2012 — Measuring Crystal Q

Prior to July 26, 2012, I could not measure Xtal Q. Why? I tried to measure crystal Q with the shunt-series tuned method (essentially the crystal acts as a trap and a step attenuator is used to calculate the insertion loss the xtal exhibits when centered in the notch) but failed because I could not precisely set the frequency with my homebrew L-C oscillator.

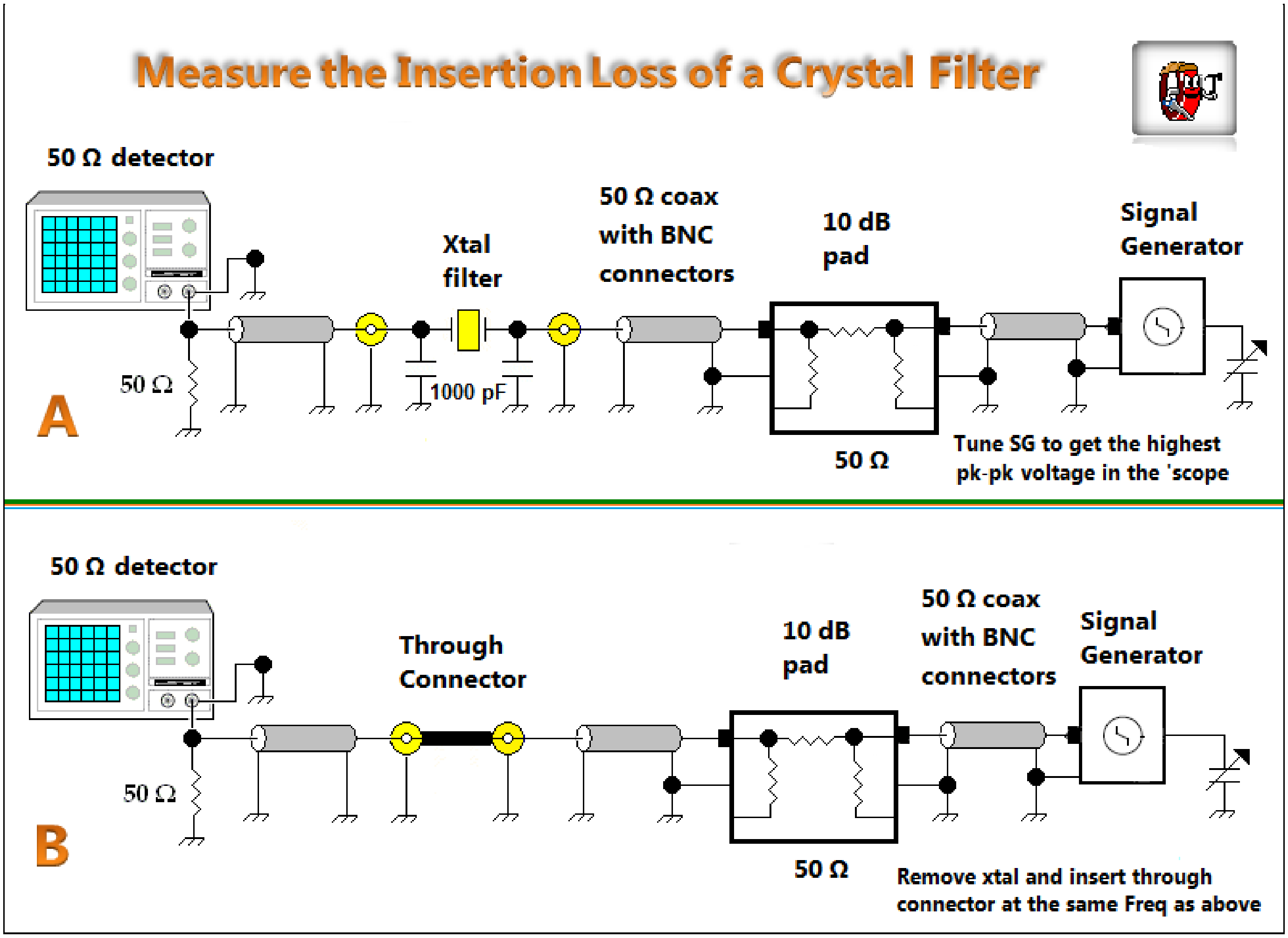
You really need a DDS or a Si570 based signal generator and preferably a spectrum analyzer to exact the measurement with the "trap method". The DDS is critical, while the SA only preferable — a power meter, or a 50 ohm terminated 'scope can work as the detector if a low-pass filter is placed just after the xtal.

On July 26, 2012 Wes, W7ZOI developed a simplified method and wrote a pdf file called **Simplified Measurement of Crystal Q** after feedback from myself and John Larkin about Q measurement difficulties without a digital-based signal generator. Unfortunately, this pdf file is no longer offered for download by W7ZOI. His method works well and I'm glad that as of now, I can *completely* characterize any crystal I own.

I present an experiment showing how I measured the Q of a 10 MHz crystal applying the new method developed by Wes, W7ZOI.

The crystal is evaluated as a N=1 low-pass filter resonated by a shunt capacitor at each end. I stuck with Wes' suggestion to try 1000 pF at 9 to 10 MHz. For lower frequency xtals he recommended trying larger value shunt capacitors. Just experiment with the shunt capacitance — if you use too high a C for a given crystal, your xtal low-pass filter bandwidth might get too narrow to measure with an L-C based signal generator.

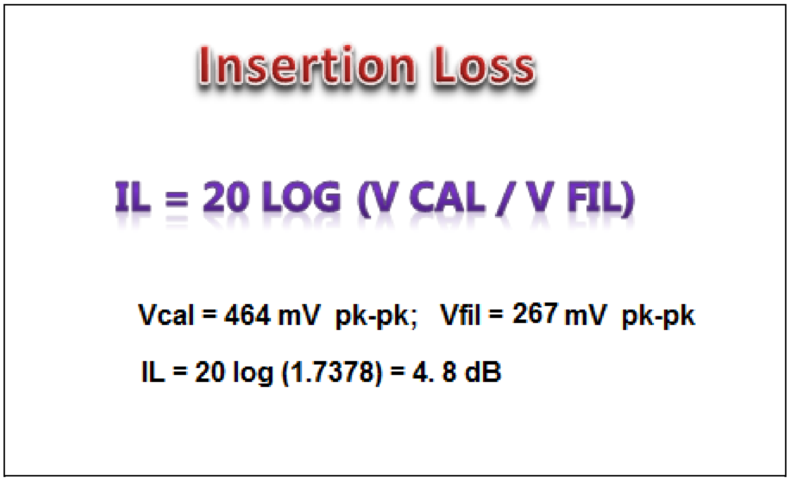
The following diagrams employ 2 programs from the Ladpac software that ship with EMRFD



Above — Measurement of crystal filter insertion loss. In Part A, I carefully tuned my signal generator to get the highest peak-peak voltage in my 50 Ω terminated oscilloscope. I recorded this AC voltage as V Fil. In Part B, I removed the crystal filter board and replaced it with a BNC clad RF through-connector. I recorded this AC voltage as V Cal.

I discuss this standard method to measure the insertion loss or gain of a *device under test* in RF Workbench 2.

Even with the simplified method, you'll need a signal generator with good tuning resolution. My generator is shown on VFO 2011 as the 2.8-10.5 MHz Signal Generator. This is my version of the EMRFD Figure 7.27 generator.



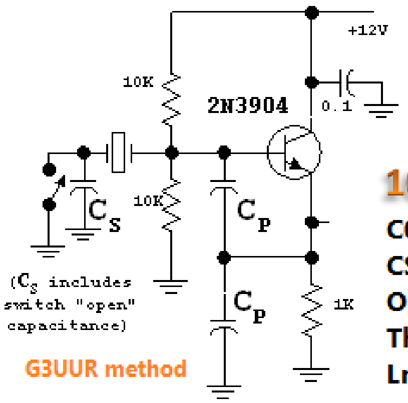
Insertion Loss

IL = 20 LOG (V CAL / V FIL)

Vcal = 464 mV pk-pk; Vfil = 267 mV pk-pk

IL = 20 log (1.7378) = 4. 8 dB

Above — The formula for insertion loss using peak-peak voltages. With my 10.0 MHz crystal, V Cal = 464 mV pk-pk and V Fil = 267 mV pk-pk. The insertion loss of my crystal = 4.8 dB.



$$C_M = 2 \cdot C_S \cdot \frac{\Delta F}{F}$$ Old Equation, p 3.19.

$$C_M = 2 \cdot (C_S + C_0) \cdot \frac{\Delta F}{F}$$ Updated Equation

10 MHz Xtal

C0 = 4.87 pF

CS = open switch + cap = 37.4 pF

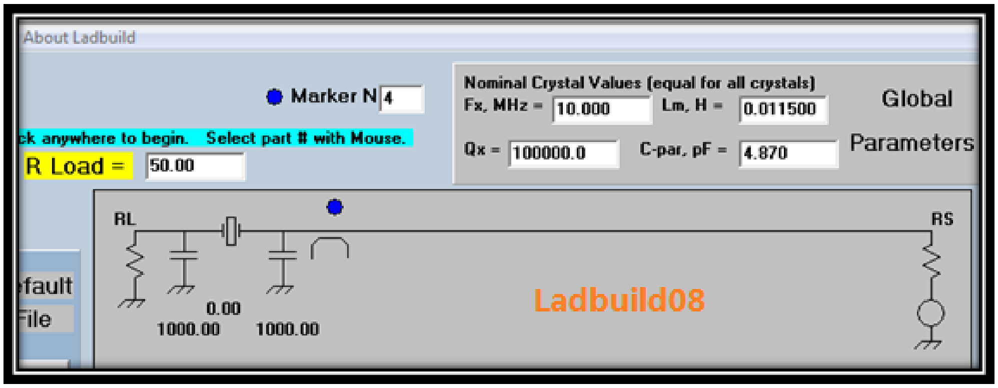
Open switch F = 9.998710 MHz

Thrown switch F = 9.996104 MHz

Lm = 0.0115 H

Measure C0

Calculate Lm



Marker N 4

R Load = 50.00

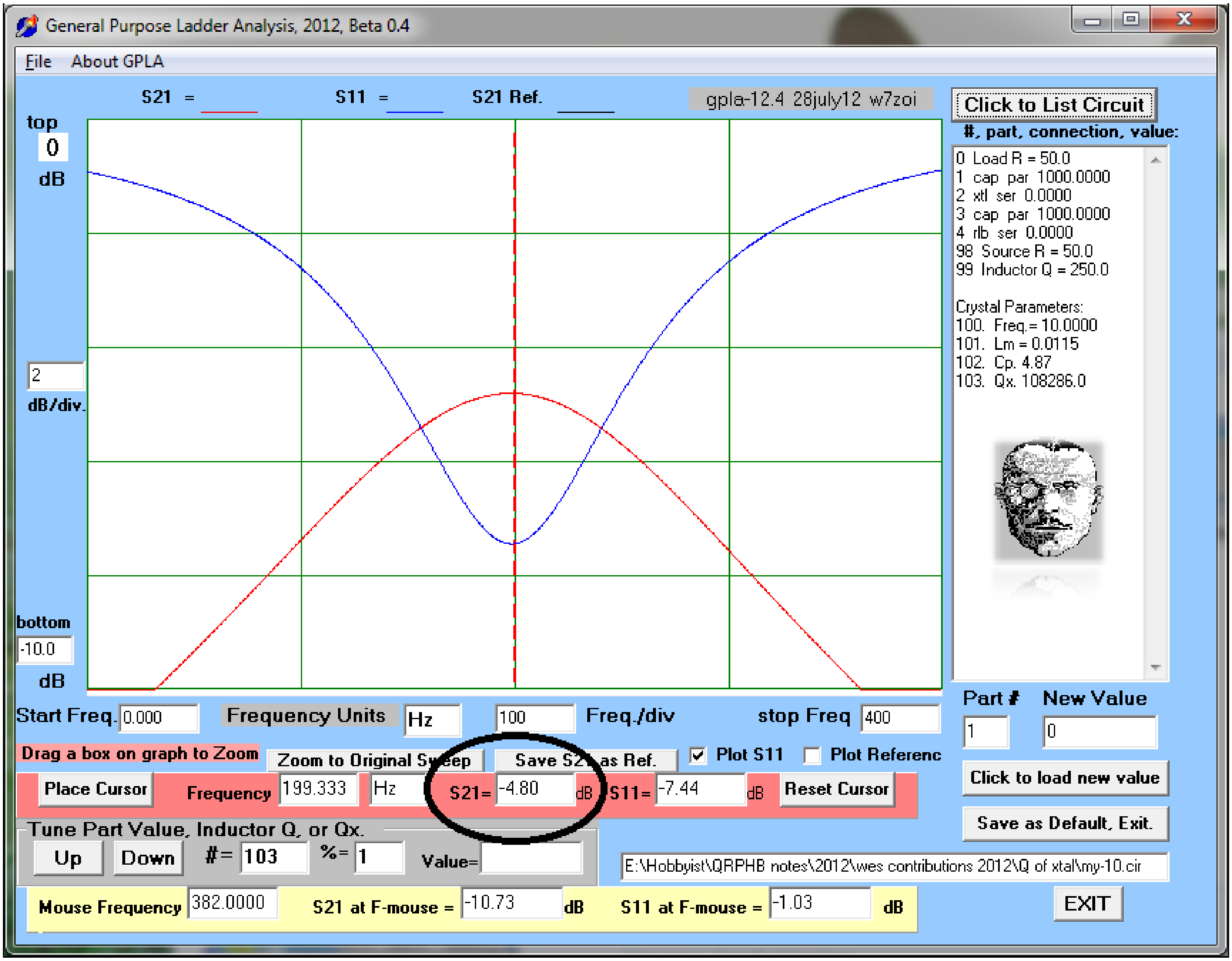
Nominal Crystal Values (equal for all crystals)

Fx, MHz = 10.000   Lm, H = 0.011500

Qx = 100000.0   C-par, pF = 4.870

0.00
1000.00   1000.00

Above — First I measured C0 (C-par) and then with the G3UUR method, calculated the motional inductance of my xtal. Finally, I entered all the needed variables into Ladbuild08 to make a filter to analyze in GPLA.



0 Load R = 50.0
1 cap par 1000.0000
2 xtl ser 0.0000
3 cap par 1000.0000
4 rlb ser 0.0000
98 Source R = 50.0
99 Inductor Q = 250.0

Crystal Parameters:
100. Freq.= 10.0000
101. Lm = 0.0115
102. Cp. 4.87
103. Qx. 108286.0

top 0 dB

2 dB/div.

bottom -10.0 dB

Start Freq. 0.000

Frequency Units Hz

100 Freq./div

stop Freq 400

☑ Plot S11

☐ Plot Referenc

Frequency 199.333 Hz

S21= -4.80 dB

S11= -7.44 dB

#= 103

%= 1

Value=

Part # 1

New Value 0

E:\Hobbyist\QRPHB notes\2012\wes contributions 2012\Q of xtal\my-10.cir

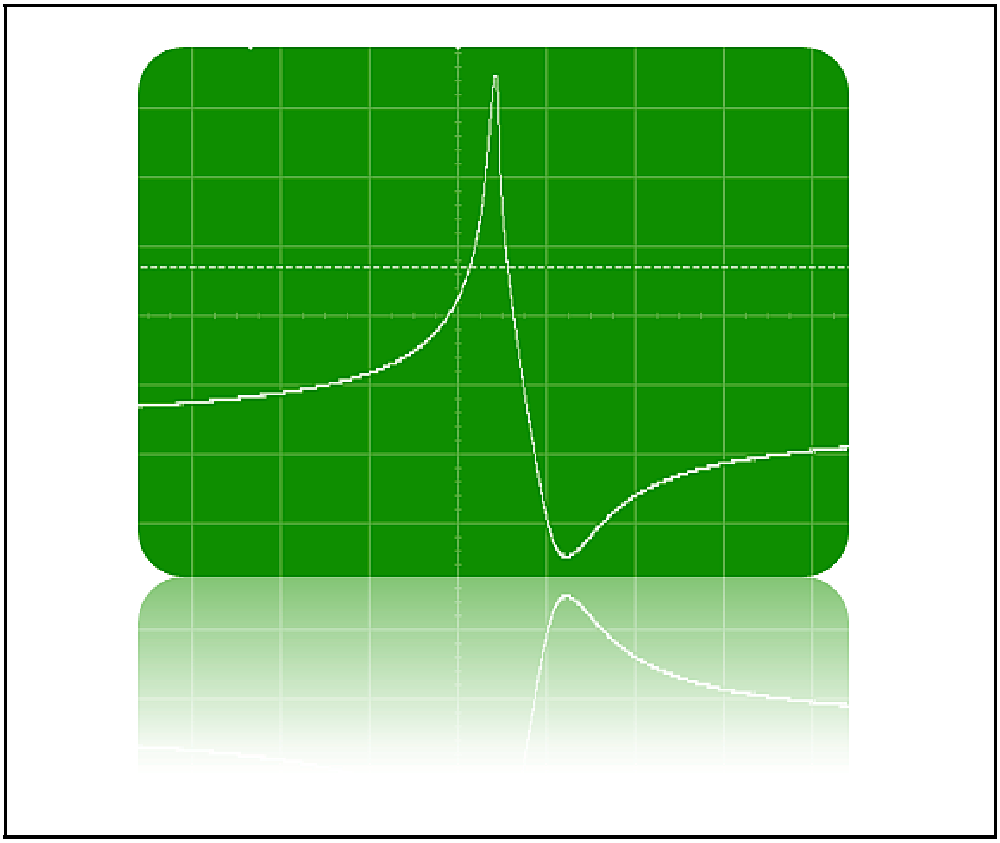
Mouse Frequency 382.0000

S21 at F-mouse = -10.73 dB

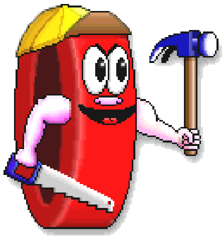
S11 at F-mouse = -1.03 dB

Above — With GPLA, I adjusted the value for *Qx* up or down until my *centered* S21 value indicated -4.8 dB (the insertion loss of my crystal determined earlier). My crystal Q = Qx = 108286.

Above — 10 MHz Crystal filter breadboard.

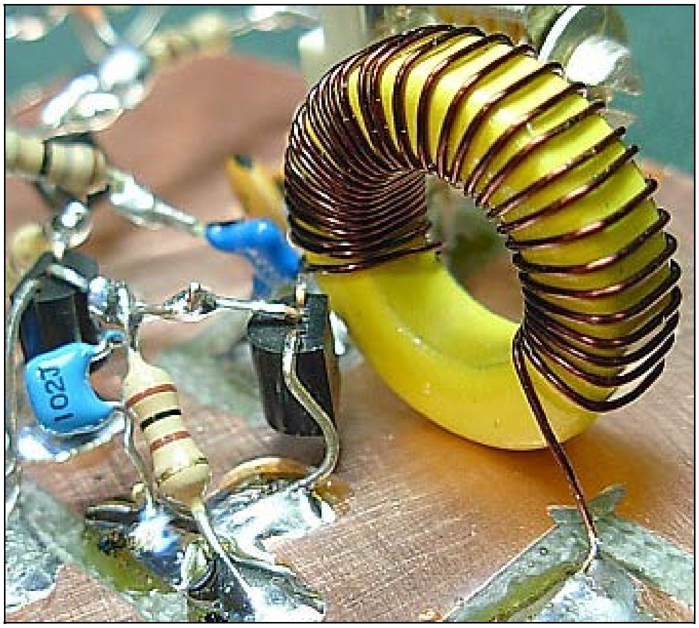


Above — A sweep of the 10 MHz Crystal filter used to determine the Crystal Q



Amateur Radio Electronic Design

Electronic Hobbyist Circuits 2010



This page houses a collection of brief hobbyist experiments.

1. LM380 Power Examination

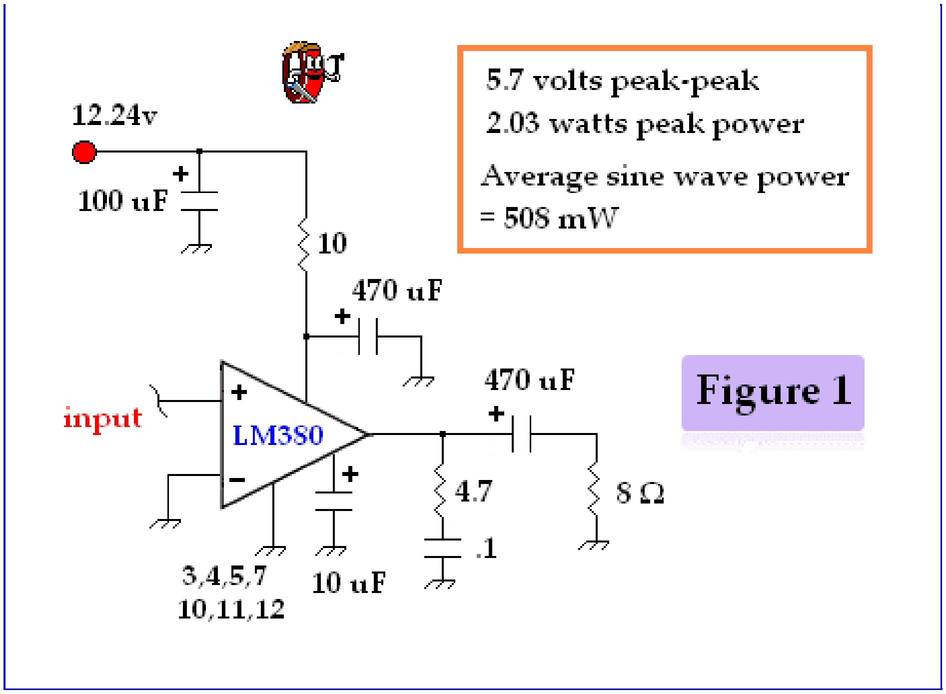
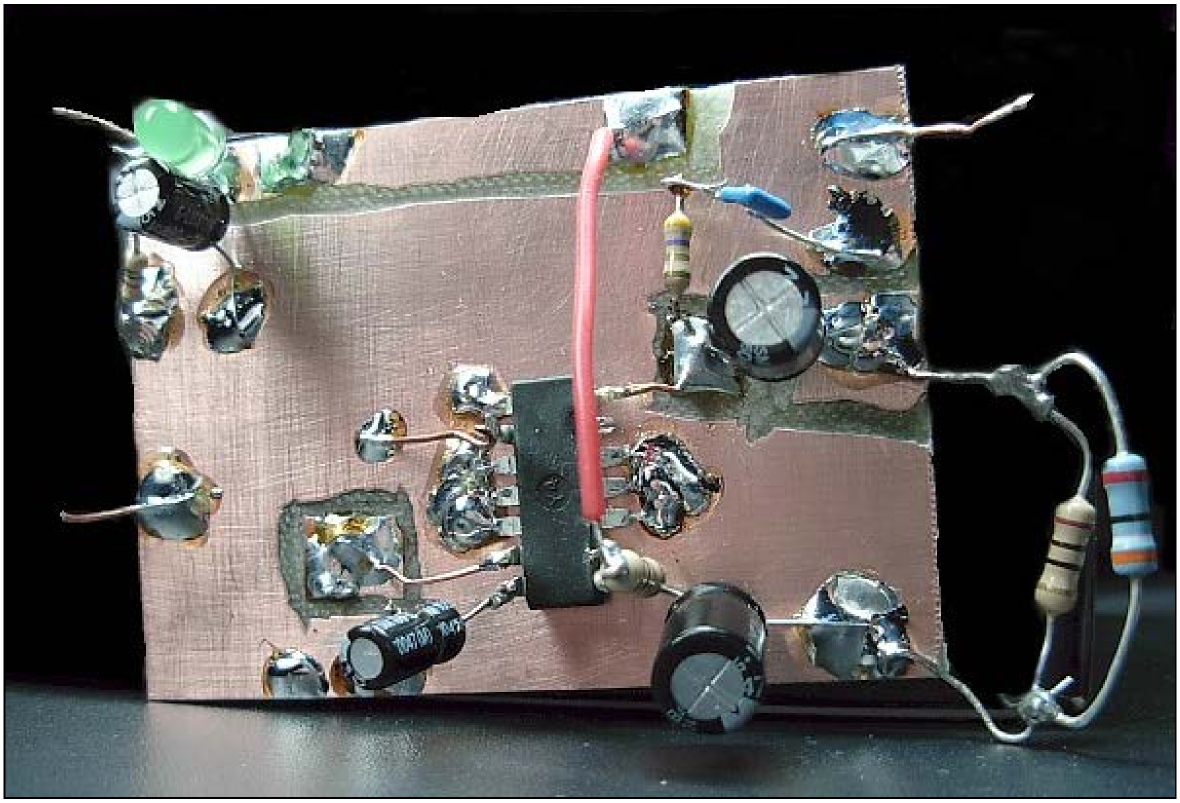
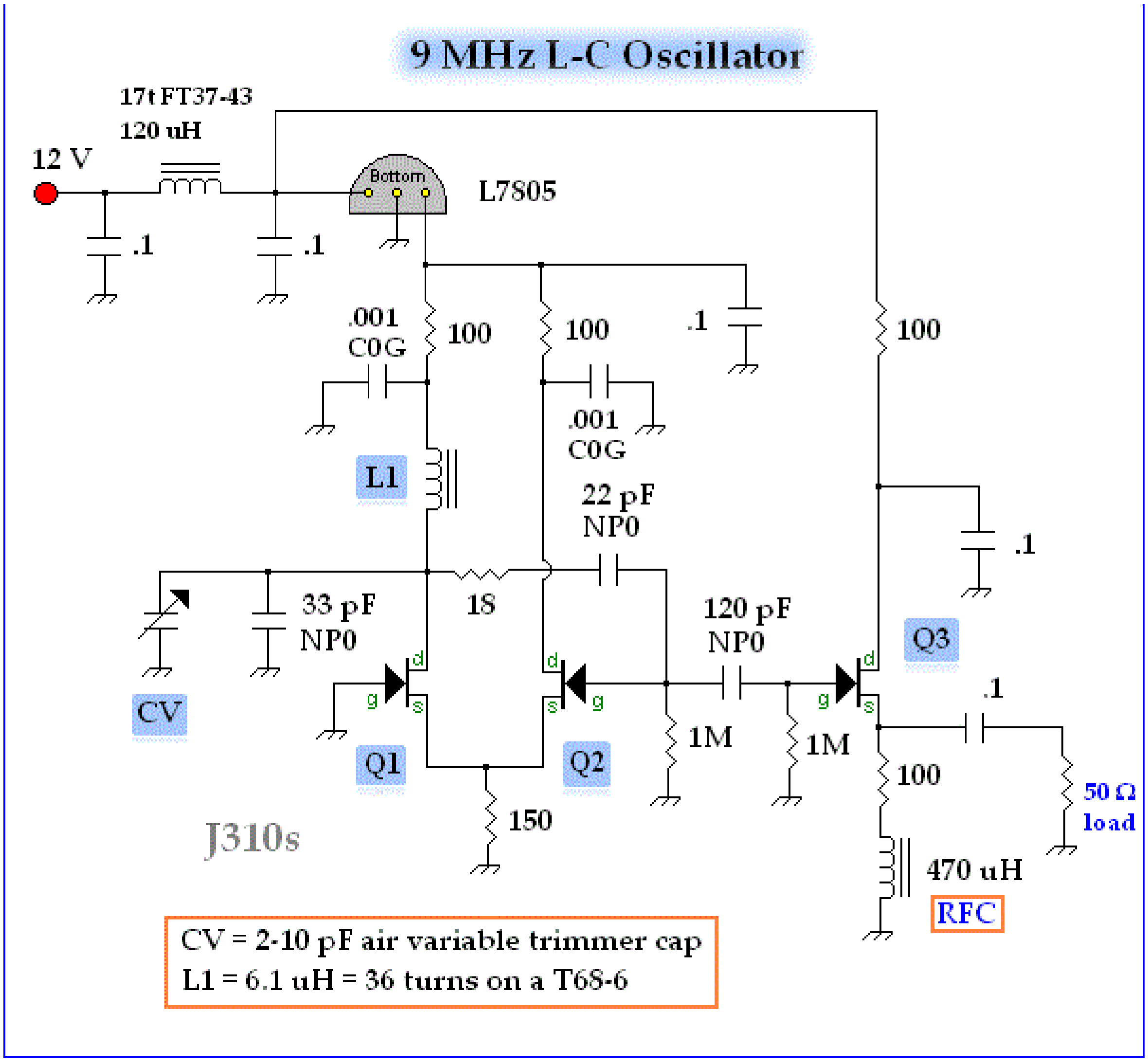


Figure 1 shows the test set up. This is a good part with an input impedance of 150kΩ. The gain is internally fixed at 34 dB. The average clean power was 508 mW. The test input frequency was 1018 Hertz.



The breadboard of Figure 1 is shown above.

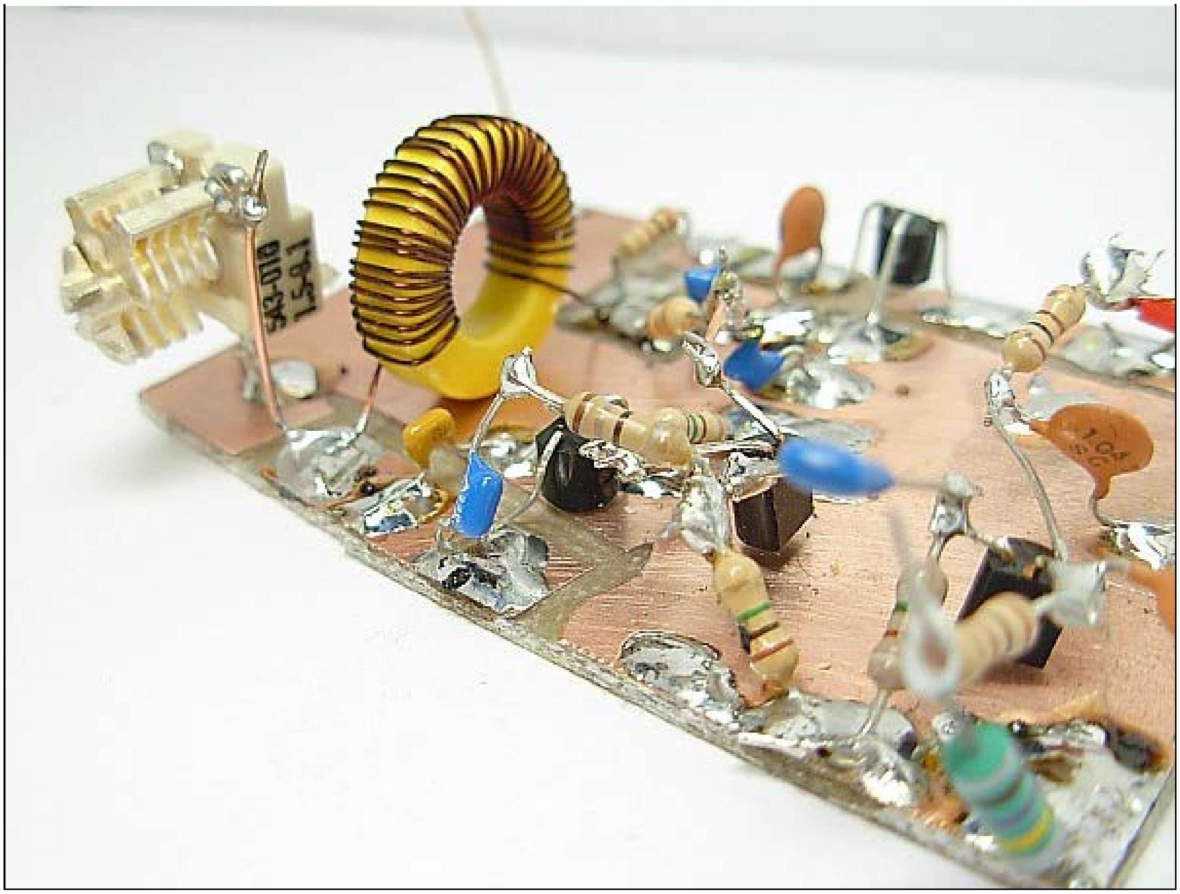
2. Wide Range L- C Oscillator



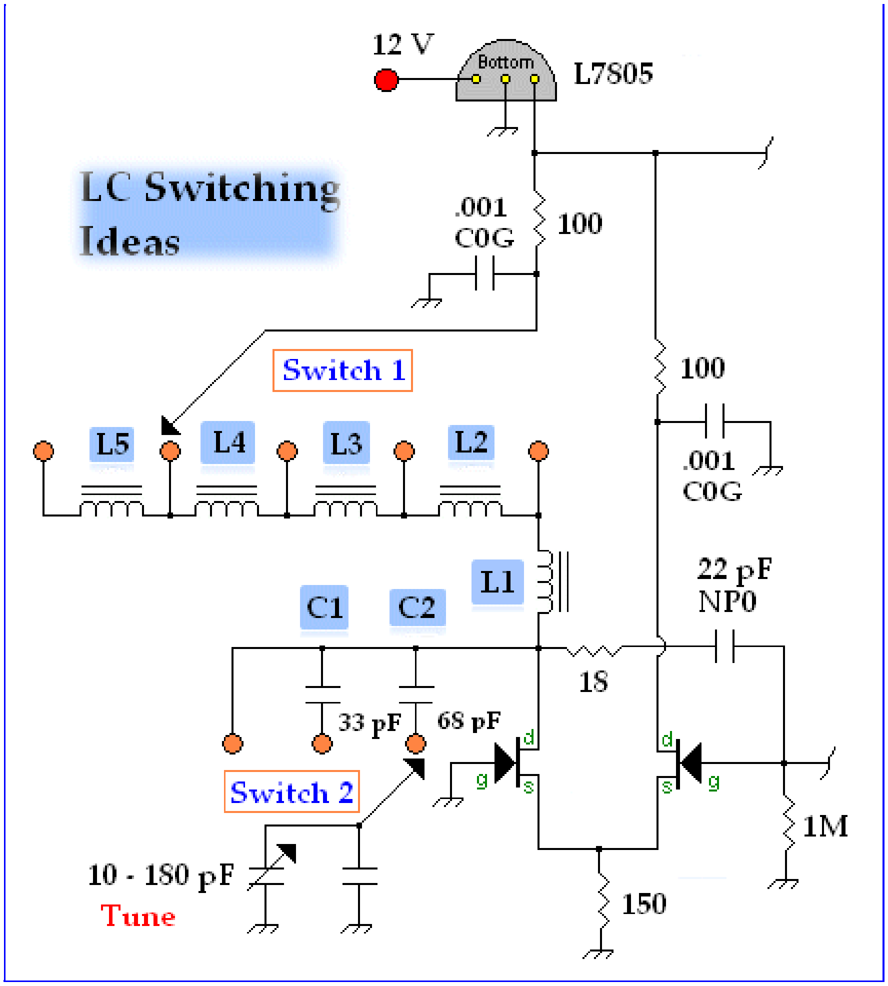
Shown above is a single frequency version of a VFO topology which allows a wide frequency range when additional switched inductors and/or capacitors plus a tuning variable capacitor are used. One good usage example would be a to use such a VFO to drive a bridge to make a wide range antenna analyzer. Q1 is essentially a common gate amplifier. The source is driven and the output is taken off the drain. This FET exhibits no signal phase shift. Q2 is a source follower that is AC coupled through that 22 pF capacitor The 18 ohm resistor is used to kill UHF parasitic oscillations. The Q2 follower also has no phase shift. Connecting the output of Q2 back to stage Q1 gives zero phase shift. The L-C tank will select the frequency where 0 phase shift is obtained. The tank will show phases other than 0 away from its resonance.

Q3 is an AC-coupled source follower to further buffer the VFO from its load. The RFC can be anything from your junk box, although it should likely be low Q. The low-pass decoupling filter on the the 12 volt supply path can also be anything reasonable. I wound mine using 17 turns on an FT37- 43 ferrite toroid. Its purpose is to keep RF from traveling down the 12 volts DC voltage wire to other parts of your circuit.

Any component connected to the L-C tank (at the Q1 drain, or the cold end of L1) can affect VFO tuning and drift. Temperature compensation will be necessary to achieve perfect stability. I use NP0 and C0G caps interchangeably. In the design shown, stability was good and the output had low measured distortion. This VFO will pretty much oscillate with any reasonable L and C values in the tank circuit. I found frequency stability was a little better with a higher L to C ratio. This is a great experimenter's circuit. One version built oscillated at 150 MHz.

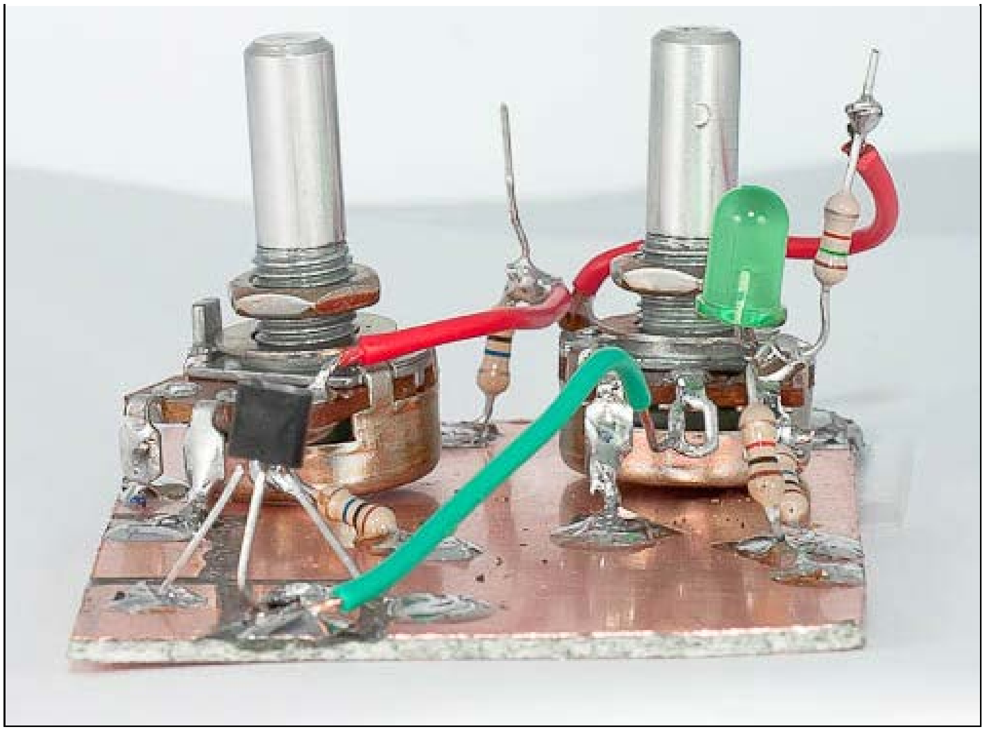


The breadboard of the above schematic. Pull the wire on your #6 powdered iron toroids tight to prevent air gaps between the toroid and the wire. Number 26 gauge wire was used on L1 as shown. High Q tank parts will garner the best results.

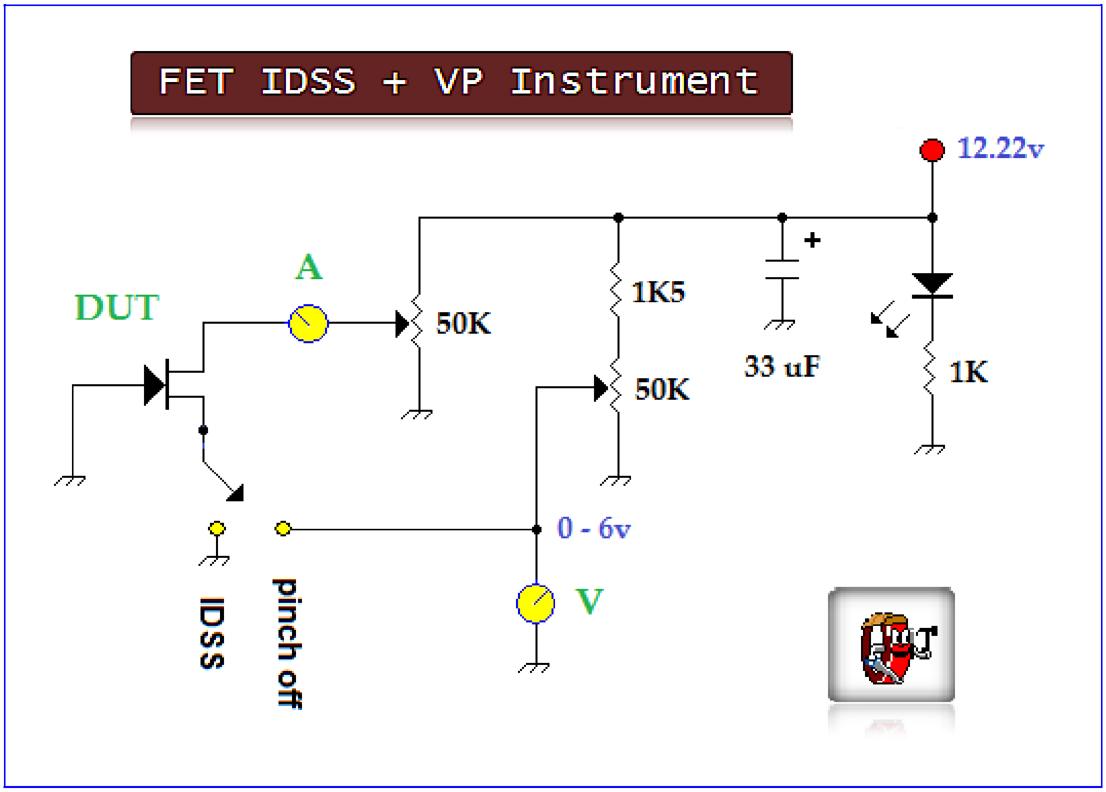


Some potential switching ideas are presented above. The builder is in total control of the tuning range and must calibrate the L and C values according to needs and the parts on hand. Output power will vary according to the L-C ratios and some designs include automatic signal amplitude leveling and/or RF gain controls.

3. FET Matching



I find matching high IDSS FETs like the J310 to be a pain. I generally matched them for IDSS and occasionally for IDSS and VP. Observations that when the IDSS of 2 or more FETs match, their pinch-off voltage (VP) also matches, led me to not measure VP. In addition, the variability of VP measurements causes me distress. Click here for a tutorial if you don't understand the terms IDSS and VP.



Above — The device I use to measure IDSS and VP. From Ken Kuhn's web site.

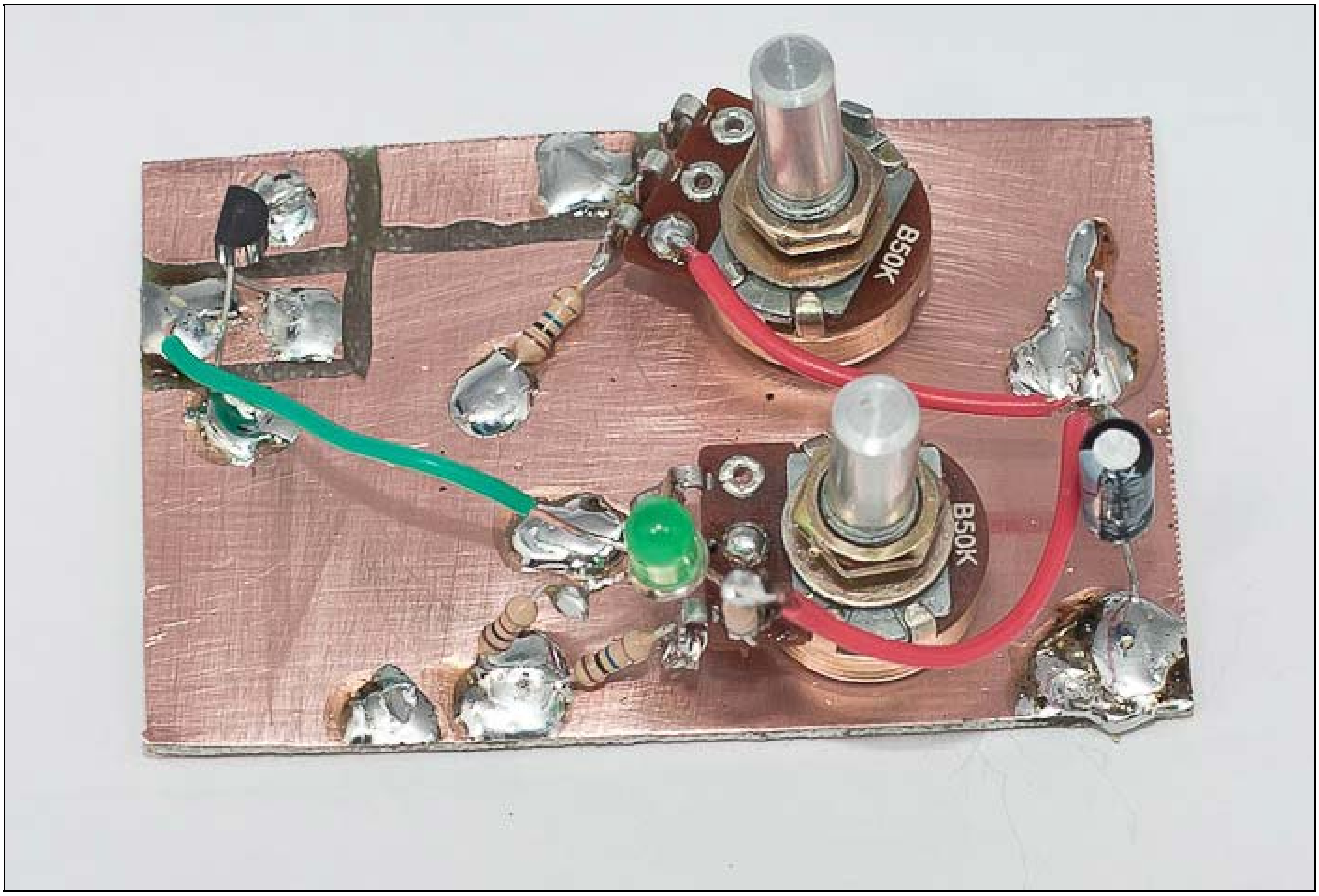
Conceptually IDSS and VP aren't difficult to understand — measuring them is another story. With the above device, first IDSS is measured; the final drain voltage potentiometer setting is left and then I measure VP. While measuring IDSS in high IDSS FETs, heating can occur and you may actually see current start to drop as you increase the drain voltage (negative temperature coefficient). On J310 specification sheets, the manufacturers state they pulse the current during measurement to prevent heating. While performing IDSS measurements, I am fearful of destroying the FET I am trying to characterize! Measuring VP is also problematic.

I have tried 3 methods to quantify VP:

Adjust the 0 - 6 volt supply until I think the current goes to 0. Serial measurements 1 day apart can vary by a variation of as much as 0.5 volts; it's quite subjective.

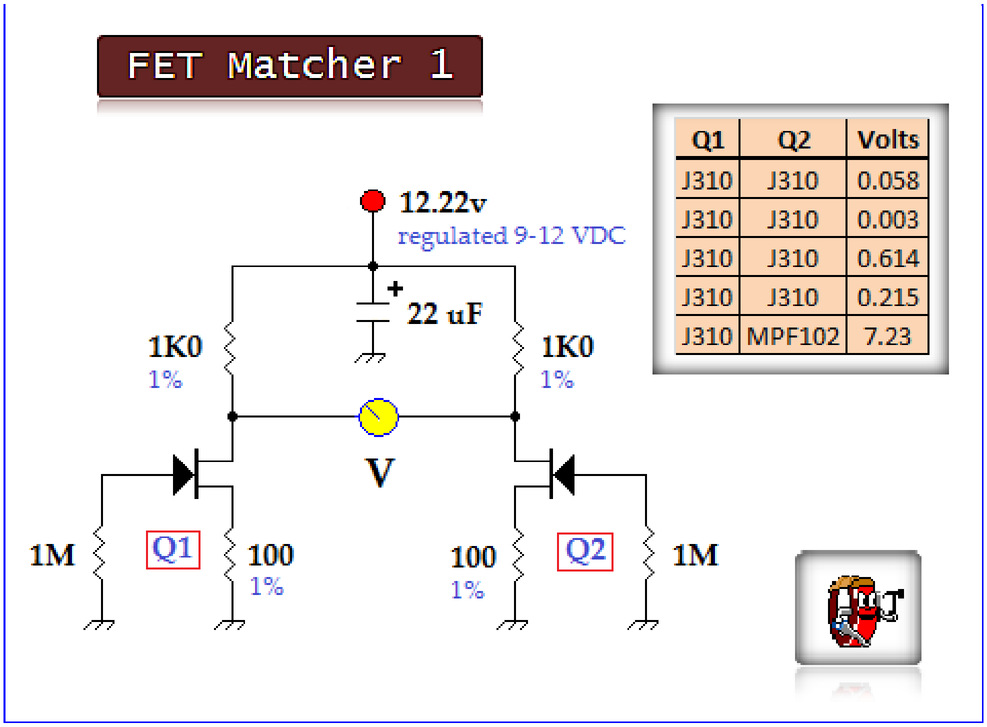
Adjust the 0 - 6 volt supply and measure the gate voltage that produces a drain current somewhere between 0.1 and 1 percent of IDSS and declare that to be VP.

Adjust the 0 - 6 volt supply so the ammeter reads ¼ IDSS and multiply this voltage by - 2.0. Refer to Ken Kuhn's site for details. Although reasonably accurate, the second order math is only a rough approximation — the real math is impossible to do by hand as it involves fractional exponents and these exponents and other factors vary as a function of the physical JFET geometry.



Above — The breadboard of the device I use to measure IDSS and VP. There is no actual switch, I either ground the green wire to the copper board, or tack solder it to the 0 - 6 volt potentiometer wiper. 10 megohm resistors plus the pot ground wire anchor each pot to the copper clad board.

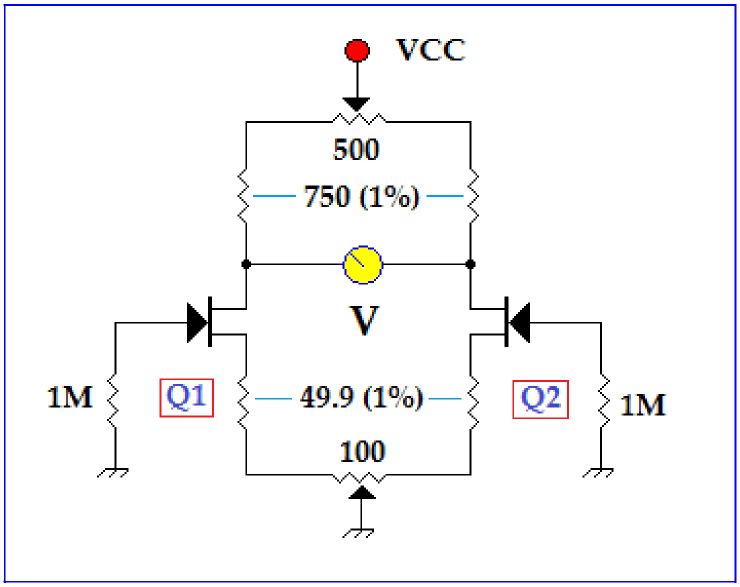
All 3 methods to quantify VP frustrate me. There must be a way to match J310s or other FETs without characterizing them. I frequently collaborate with readers to problem solve and learn. A potential solution contributed by a supportive reader follows:



Q1	Q2	Volts
J310	J310	0.058
J310	J310	0.003
J310	J310	0.614
J310	J310	0.215
J310	MPF102	7.23

Above — A bridge is used to match a pair of JFETS. It's often best to match devices in a circuit that closely resembles the one that you intend to use them in. The differential output of each drain is measured by placing a DVM lead on each drain and recording the voltage. Generally, I stick a FET in the Q2 slot and put FETs from my parts bin in the Q1 slot to match it. The results of 5 different FET pairs are tabled above. A match <= 50 mV is probably acceptable and in 1 case, I found a match of 3 mV! You can match 1 FET with many using this device.

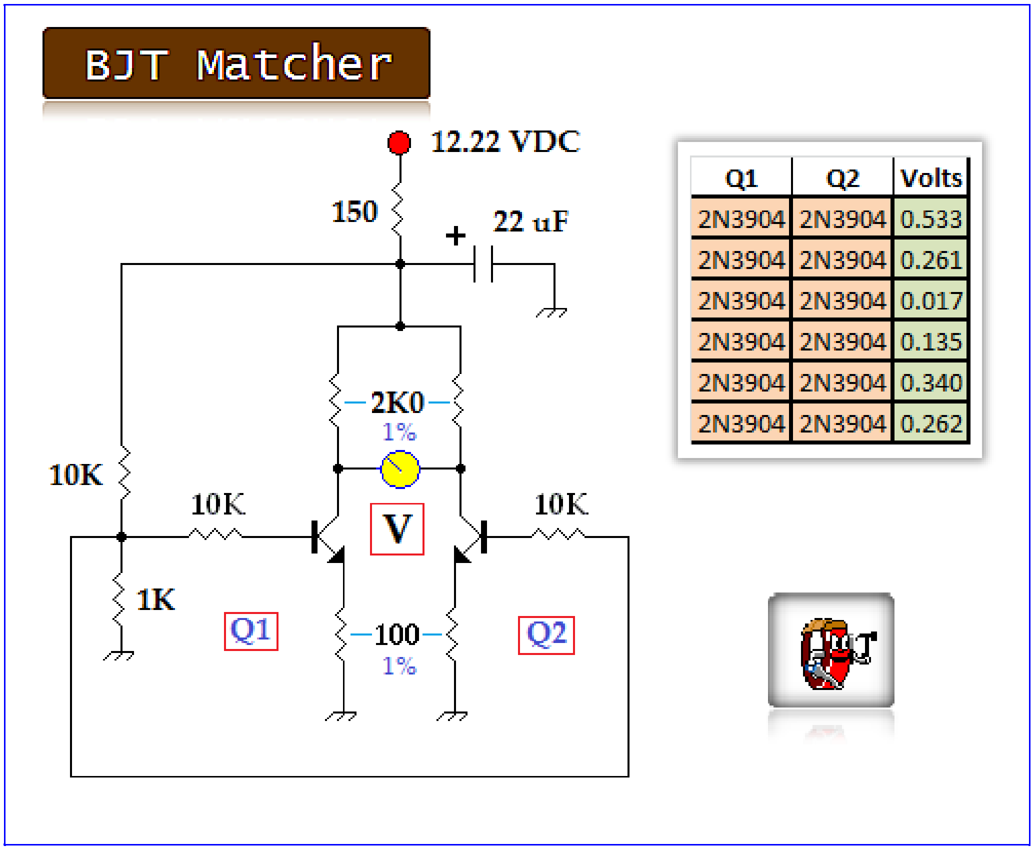
Note the poor match when an MPF102 and a J310 were tested. 1% tolerance resistors are recommended for the bridge.



Above — A set and forget precision bridge using trimmers to establish a perfect DC match on both halves. If you don't have 1% parts, the trimmer resistors offer a solution. You can place a trimmer at either the drain or source end as shown and just use 5% resistors. Calibrate each half of the bridge with your ohm meter. I cover bridges on this web page if you need more information on them.

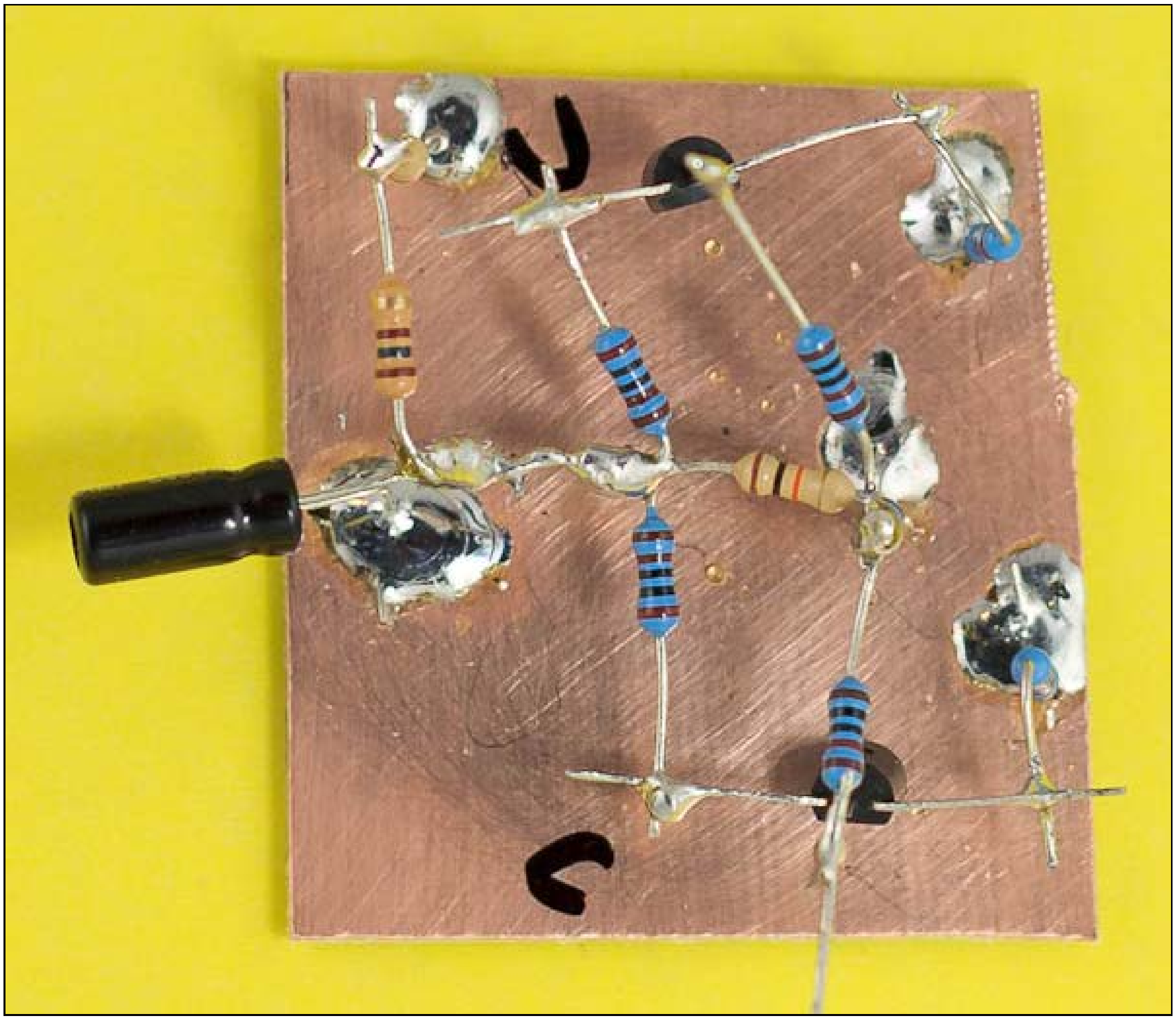
Above — A differential FET matcher breadboard. This version had a 9.1M gate resistor on Q1 by accident, although it made no difference to the experiment, as no DC gate current flows.

4. BJT and Diode Matching

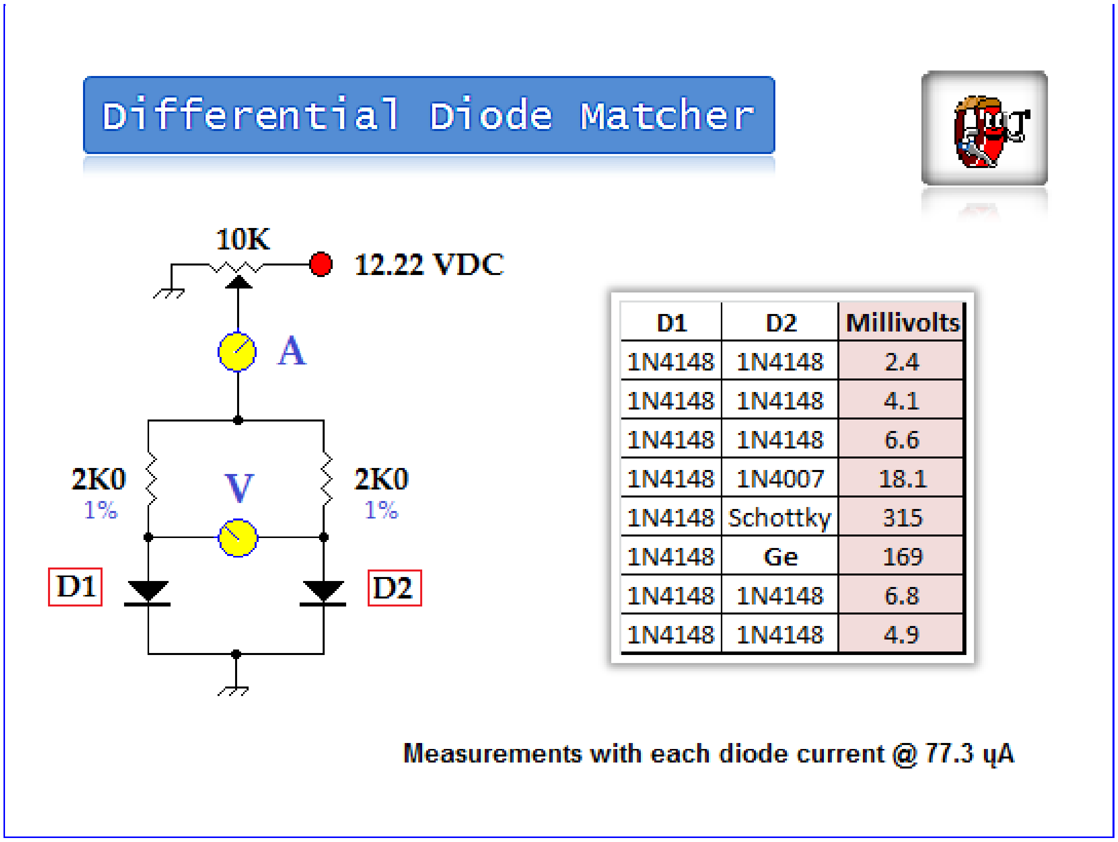


Q1	Q2	Volts
2N3904	2N3904	0.533
2N3904	2N3904	0.261
2N3904	2N3904	0.017
2N3904	2N3904	0.135
2N3904	2N3904	0.340
2N3904	2N3904	0.262

Above — Differential BFT matcher. Differential voltage matching works for bipolar junction transistors and diodes also. PN junctions are thermally sensitive — let them stabilize before testing. I measured 6 2N3904 transistors and the tabled results remind us why this transistor isn't the best choice for matched BJT circuits.



Above — The breadboard of the differential transistor matcher. Any reasonable resistor values will work fine, but don't run the current too high — my circuit has a nominal emitter current of 2.25 mA.



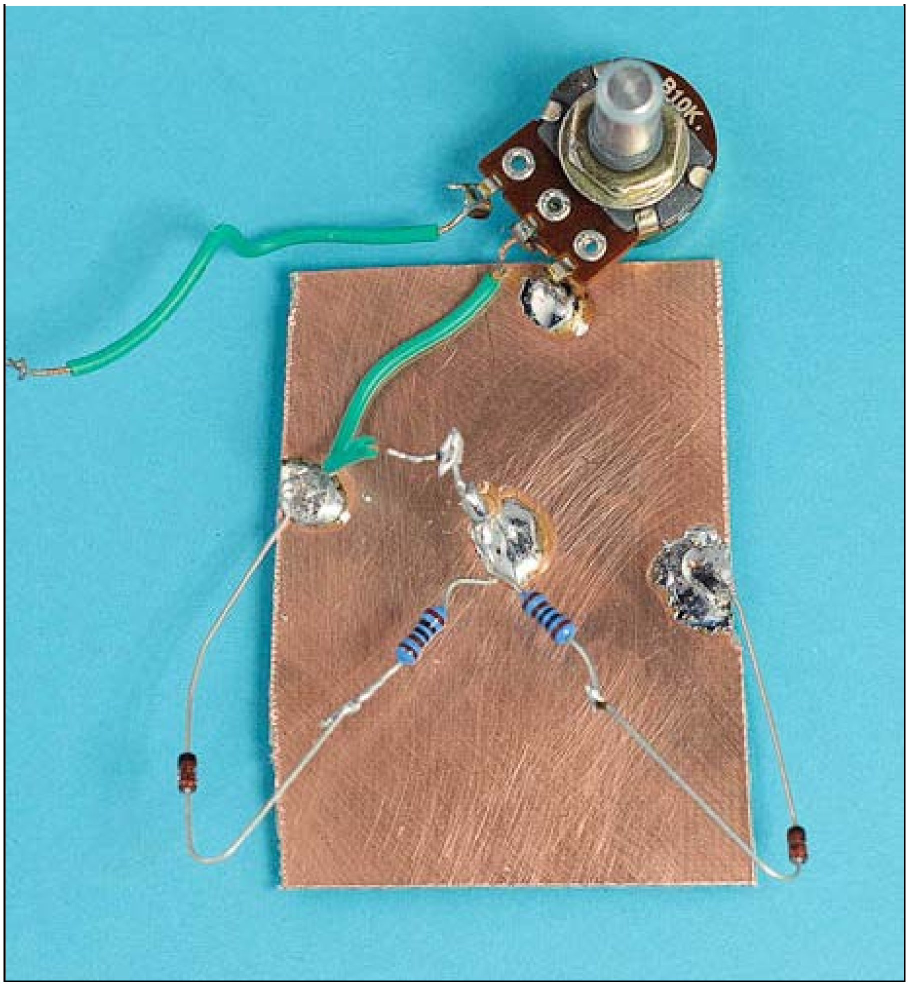
Differential Diode Matcher

D1	D2	Millivolts
1N4148	1N4148	2.4
1N4148	1N4148	4.1
1N4148	1N4148	6.6
1N4148	1N4007	18.1
1N4148	Schottky	315
1N4148	Ge	169
1N4148	1N4148	6.8
1N4148	1N4148	4.9

Measurements with each diode current @ 77.3 ųA

Above — The differential diode matcher. The 10K pot allows you to vary the current to suit your needs. The ammeter reads double the diode current. Considering only 1 diode; the current ranges from 0.01 to 5.82 mA. You can drop the 1% metal film 2K0 resistors down as low as 470 ohms or so if you need serious current. Tabled are some measurements performed with the 10K pot dialed to give 0.636v (77.3 uA per diode). This provided excellent sensitivity. Measurement was performed on my DVMs 200 mV scale. Builders might experiment with diode current to assess measurement sensitivity and linearity or even to match the diodes across a range of current values.

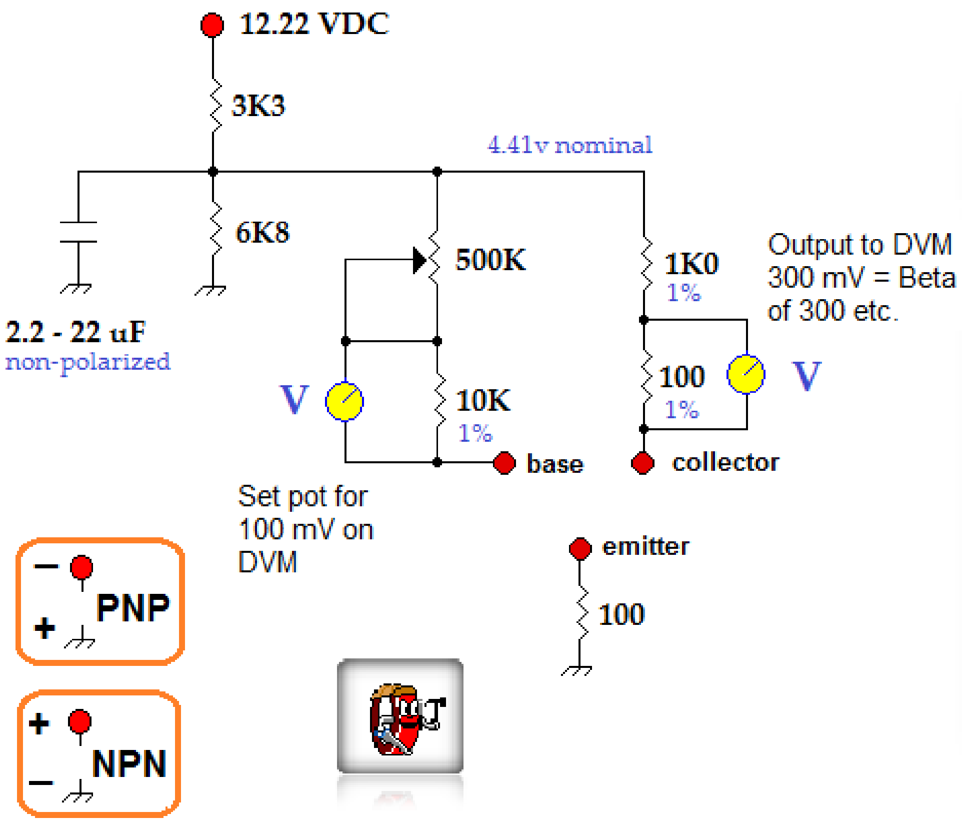
Clear glass passivated diodes can be affected by light — photons will pass through the glass and knock electrons through the barrier. Ensure each D.U.T is exposed to the same amount of ambient light. Some microwave detector diodes can be damaged with as little as 1 mA of current. Replace the 2K0 resistors with 10-15K resistors, or lower VCC to prevent damage .

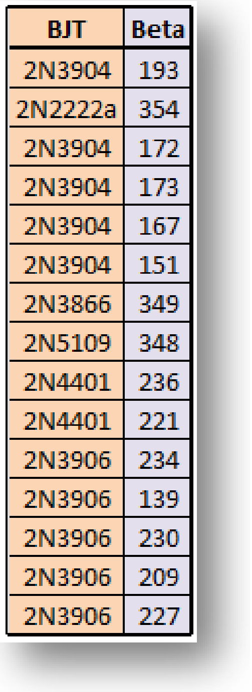


Above — Breadboard of the differential diode matcher. The 2K0 resistors are suspended by a vertically mounted 10M stand-off resistor.

5. Bipolar Junction Transistor Beta Tester

Bipolar Transistor Beta Tester





BJT	Beta
2N3904	193
2N2222a	354
2N3904	172
2N3904	173
2N3904	167
2N3904	151
2N3866	349
2N5109	348
2N4401	236
2N4401	221
2N3906	234
2N3906	139
2N3906	230
2N3906	209
2N3906	227

Above — Disappointed with the transistor beta testers in our common, low-cost digital multimeters, we did the logical thing; designed and built our own. This collaborative project was more an experiment with BJTs than anything else. It's about as simple a beta measurement device as you can make and still get good results. Preventing damage to our parts inventory underpins this design — the 100 Ω emitter resistor plus ~ 10 microamps of base bias keeps the IC low to help avoid smoke since most new small signal transistors have a beta of 100-400.

Ensure the correct polarity for PNP versus NPN transistors. The voltage divider targets 5 volts using a standard ~12 volt supply; I just used whatever resistors were handy and ended up with the 6K8 — 3K3 pair. VCC should be regulated. Perform the measurements with a single multimeter allowing time for stabilization.

To use: Set the potentiometer so that the voltage drop across the 10K resistor is 100 mV. Then move your DMM leads to the 100 Ω resistor and measure the beta. This device measures beta, the static gain at DC.

Measuring beta is a bit inexact since beta is affected by so many variables as follows:

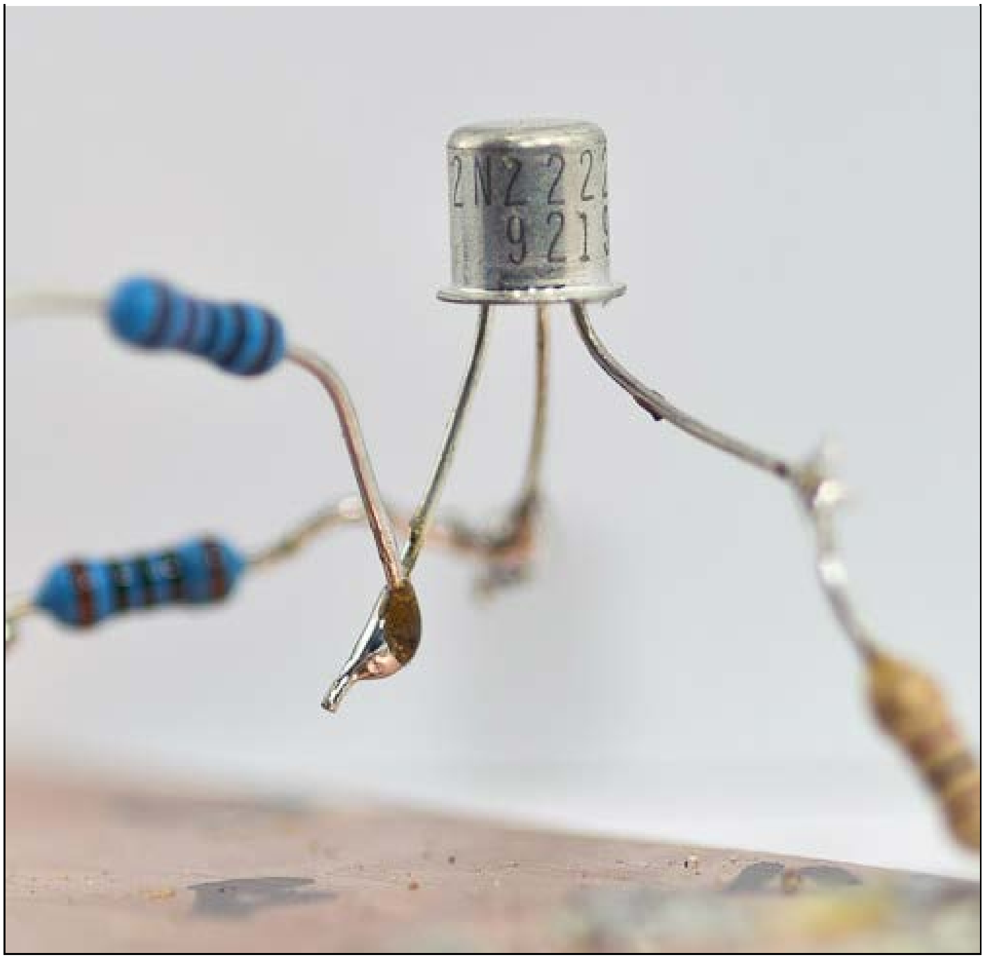
Beta tends to be low at low operating currents and rises and plateaus for medium currents and then falls at higher currents.

Beta tends to increase with temperature.

Beta is affected by the voltage between the collector and emitter -- this is a weak effect except when the voltage is very small.

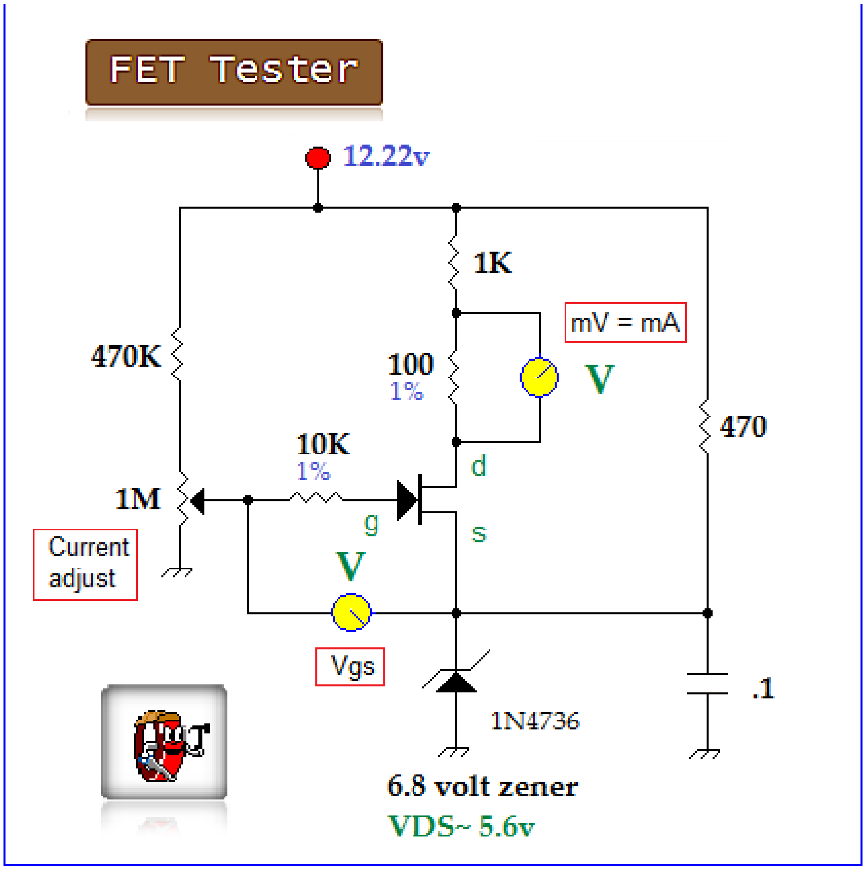
The beta can vary as the battery depletes in DMM beta testers.

Above — Breadboard of the QRP beta tester. We hope you have as much fun with this circuit as we did.



Above — Testing a 2N2222a. We found measuring transistors to be very instructive; comparing our results to specification sheets, handling the different BJTs, gaining experience — all part of the exciting world of bench electronics. Such circuits, although simple, are great for both learning and design.

6. FET Tester - VGS Measurement Tool

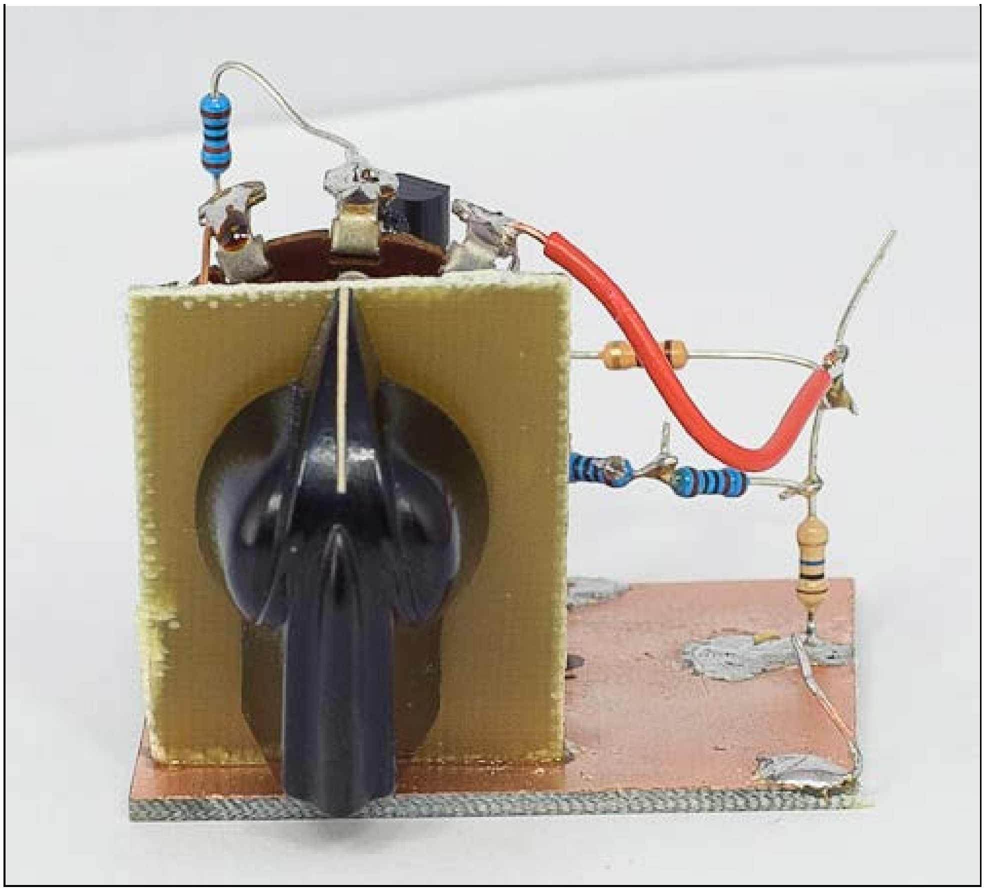


Above — A FET VGS Tester. The final collaborative output from late December 2010: a device to examine VGS. A zener diode value from 6 -7.5 volts or so should work okay. VDS should be greater than the nominal VP for greatest accuracy. The VDS using 7.5 volt zener diode was 4.85 volts. The range of VGS on my breadboard was roughly 6 to -6 VDC without the 470K range limiting resistor; it's up to you if you want to limit the VGS range.

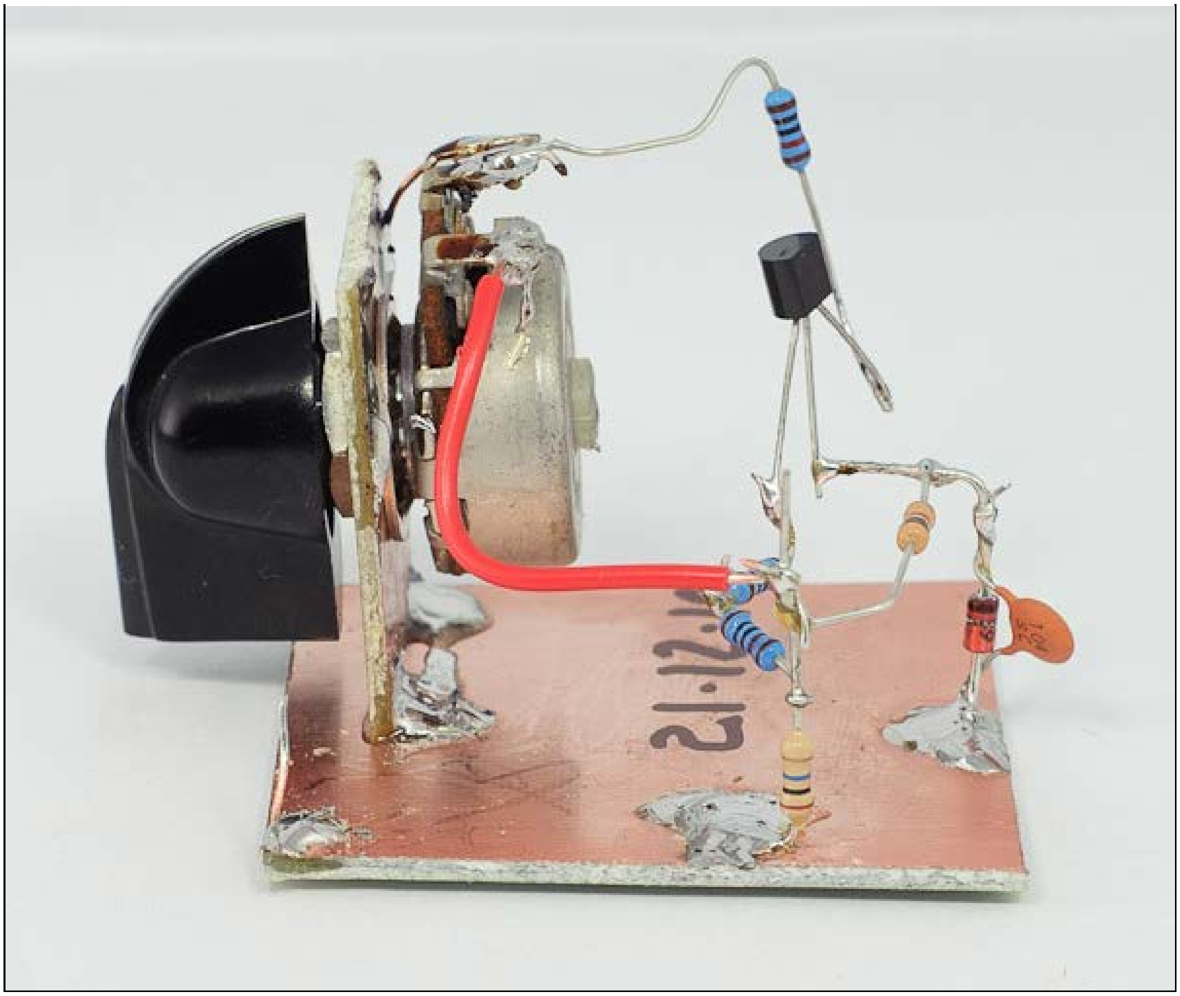
Using the mV scale on your DVM, potentiometer adjust the current until the meter just reads 0. Then move your leads to the VGS test points to read the pinch-off voltage.

The 0 current point will only ever be close. The interesting issue with the 0 point as defined is it never goes to true zero — just zero enough — that point may be hidden if your meter scale truncates lower current measurements to zero. If you're able to repeatedly get consistent VP measurements with whatever method your using, then it's likely accurate.

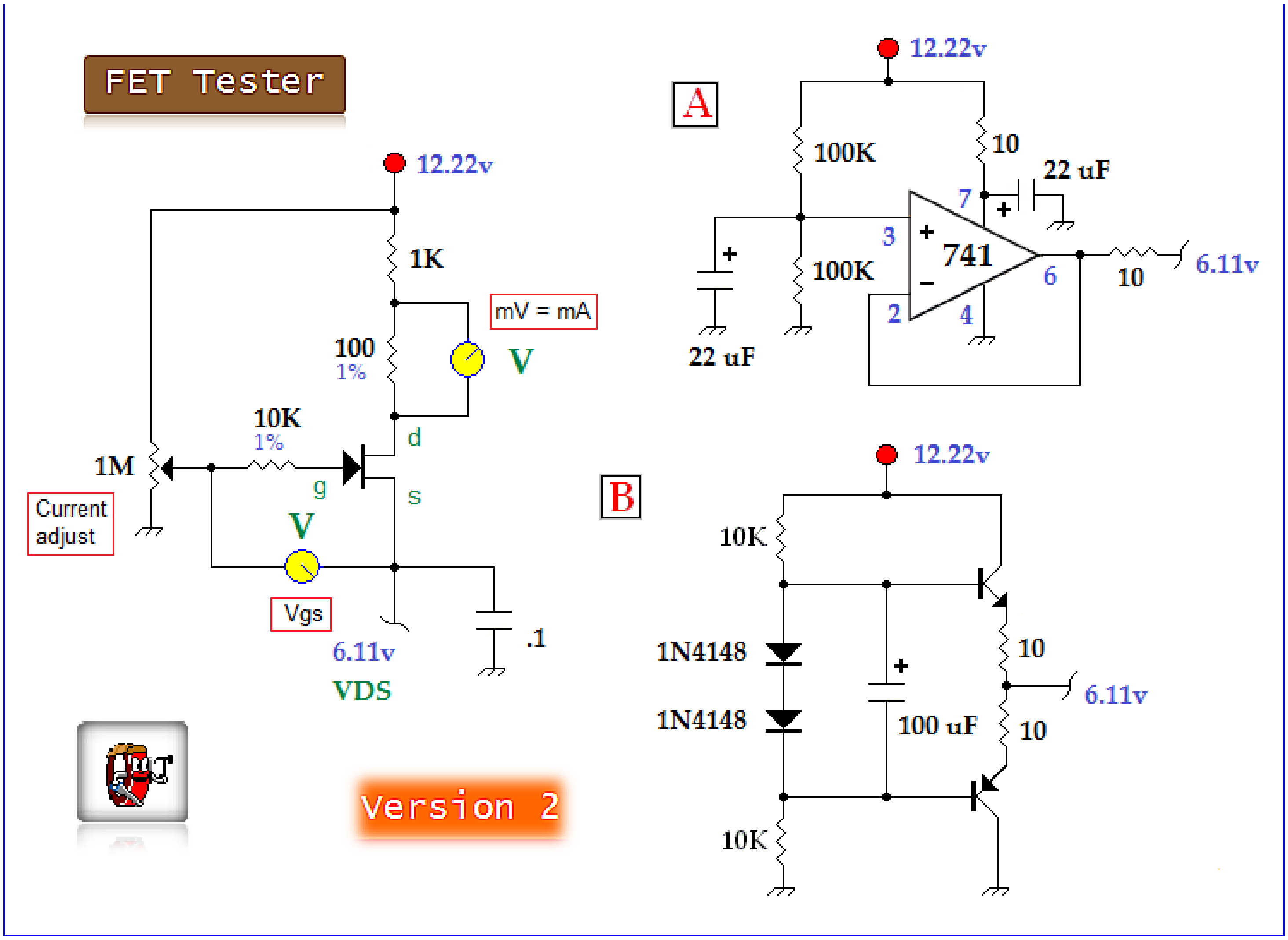
We thought about measuring gate leakage current across the 10K gate resistor, but accurate measurement is impossible with this device. The gate leakage current is going to be in the low nanoampere region at the very highest unless the JFET is bad. The expected voltage across the 10K resistor would be a few 10s of microvolts. A better way to measure gate leakage is to use the 10M resistance of a DVM instead of this resistor. The DVM then acts as a current meter by measuring the voltage across the 10M resistance. A reading of 200 mV would mean a current of 20 nA.



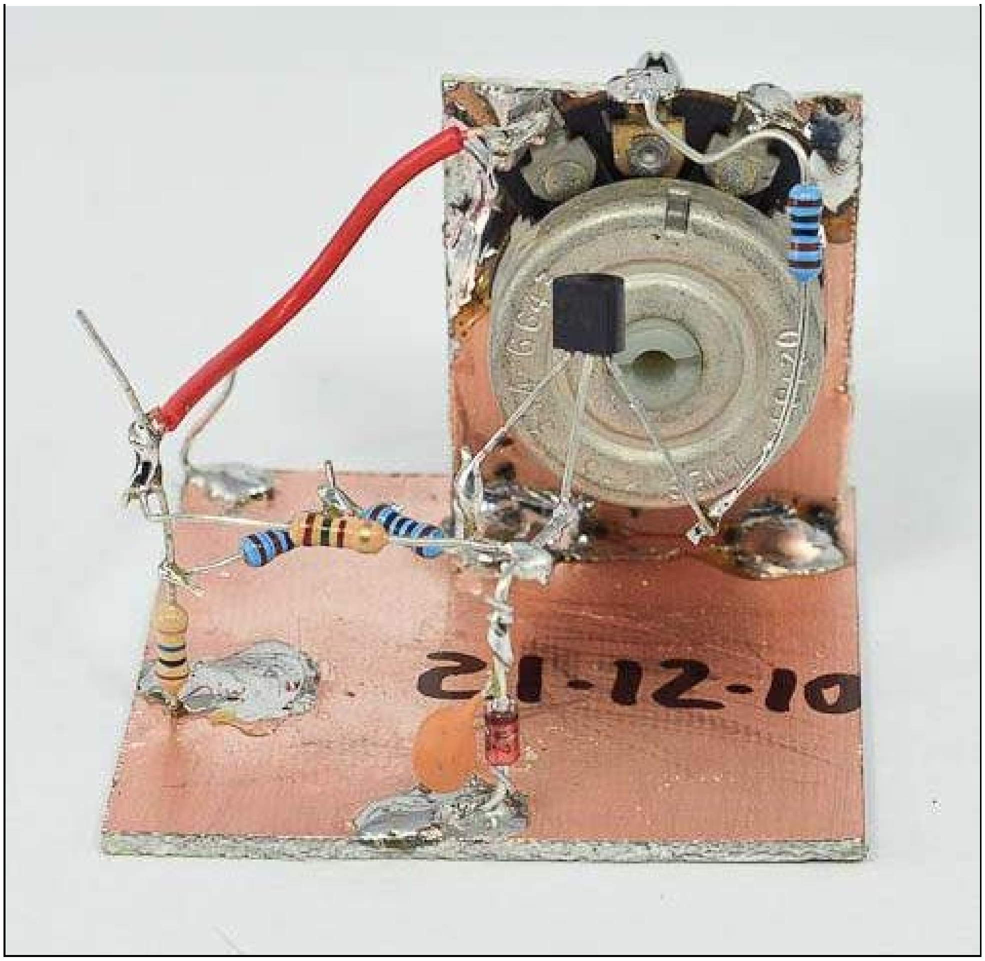
Above — Front view of the FET tester with a "chicken head" knob on the potentiometer.



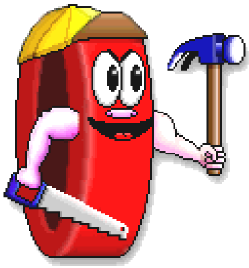
Above — Measuring a J310 with the FET tester prototype breadboard.



Above — Instead of a zener diode, low impedance VCC/2 splitters are shown as A and B; the zener diode is easier.



Above — Reverse view. I didn't bother with the 470K VGS range limiting resistor in my breadboard.



RF — Test and Measurement

RF Workbench Page 1

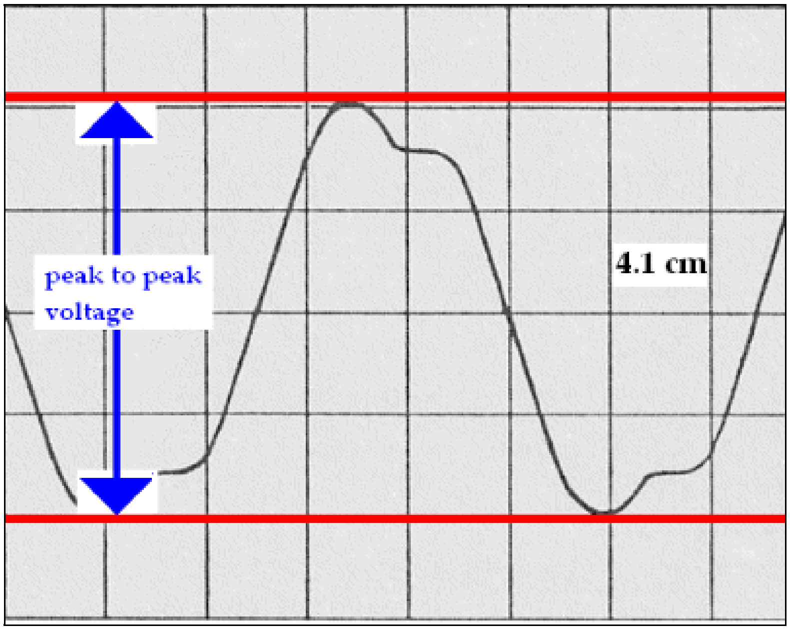
The first installment of a multi-part series exploring basic 50 Ω RF circuit measurement. This web series borrows heavily from the work of Wes, W7ZOI.

RF Workbench 1 shows a way to measure AC signals and quantify power. As circuit builders, knowing the power gain *or loss* in dB of stages like filters or amplifiers; or the absolute power in dBm of our RF signal sources dominates our bench work.

Ignite your bench measurement — Better measurement fidelity inspires confidence, creativity and fun!

Calculate Power Gain from your Oscilloscope Peak-Peak Voltage

An oscilloscope peak-to-peak voltage provides a popular way to determine power gain or absolute power.



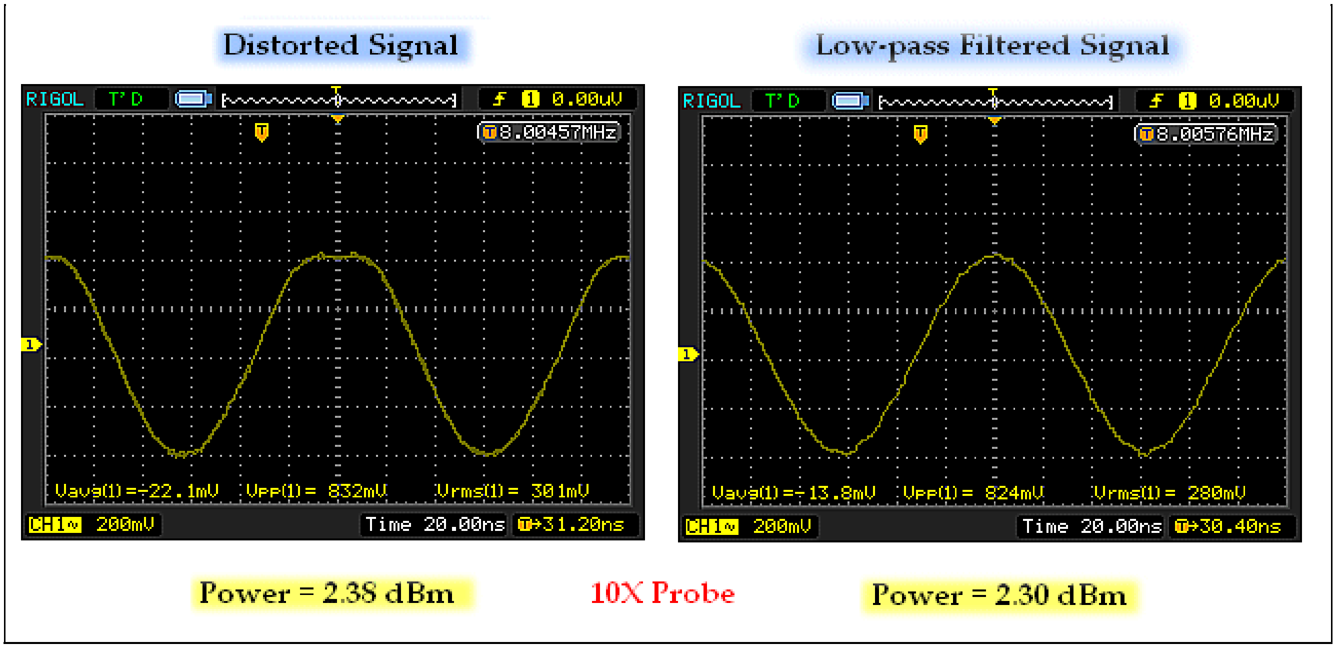
To calculate the power of a sinusoidal waveform, you measure its AC voltage. Assuming your 'scope is calibrated, the first step is to measure the amount of vertical deflection on the screen.

In the figure shown left, the vertical deflection = 4.1 cm. Multiply this measurement by the volts/cm setting of your scope. Lets assume your scope was set to 0.1 volts/cm. Thus the result = 0.41 volts. One final multiplication is required; you must multiply the resultant voltage by the attenuation ratio of the probe. In most cases, a 10X probe is used. Therefore, the measured peak-to-peak voltage is 0.41 x 10 = 4.1 volts. DSOs output numeric voltage readings in addition to that shown — a nice feature.

This signal exhibits a major problem; it's distorted. To calculate the power from peak-to-peak voltage, you require a sine wave. To obtain a sine wave, this distorted signal must be low-pass filtered. Let's examine this topic with some real experiments.

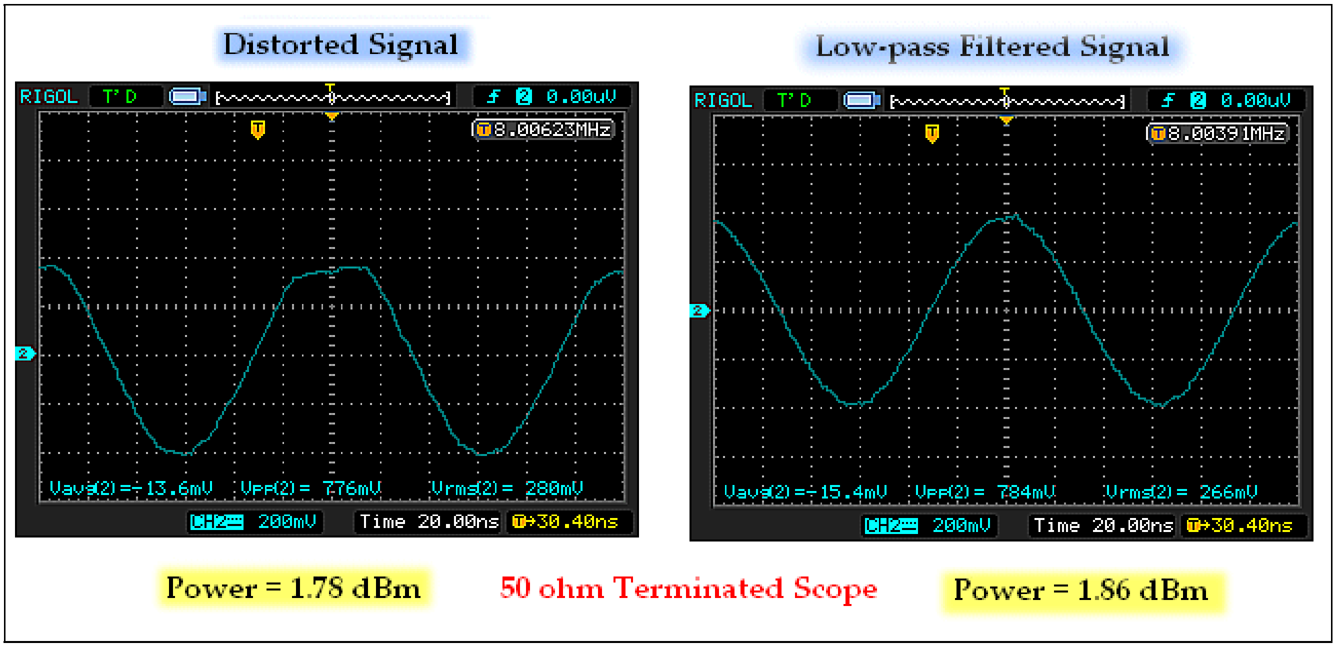
Signal Viewing versus Power Experiments

Measuring 'scope signals takes some skill to get accurate, reproducible results. Consider the following signals taken from an 8 MHz VFO.



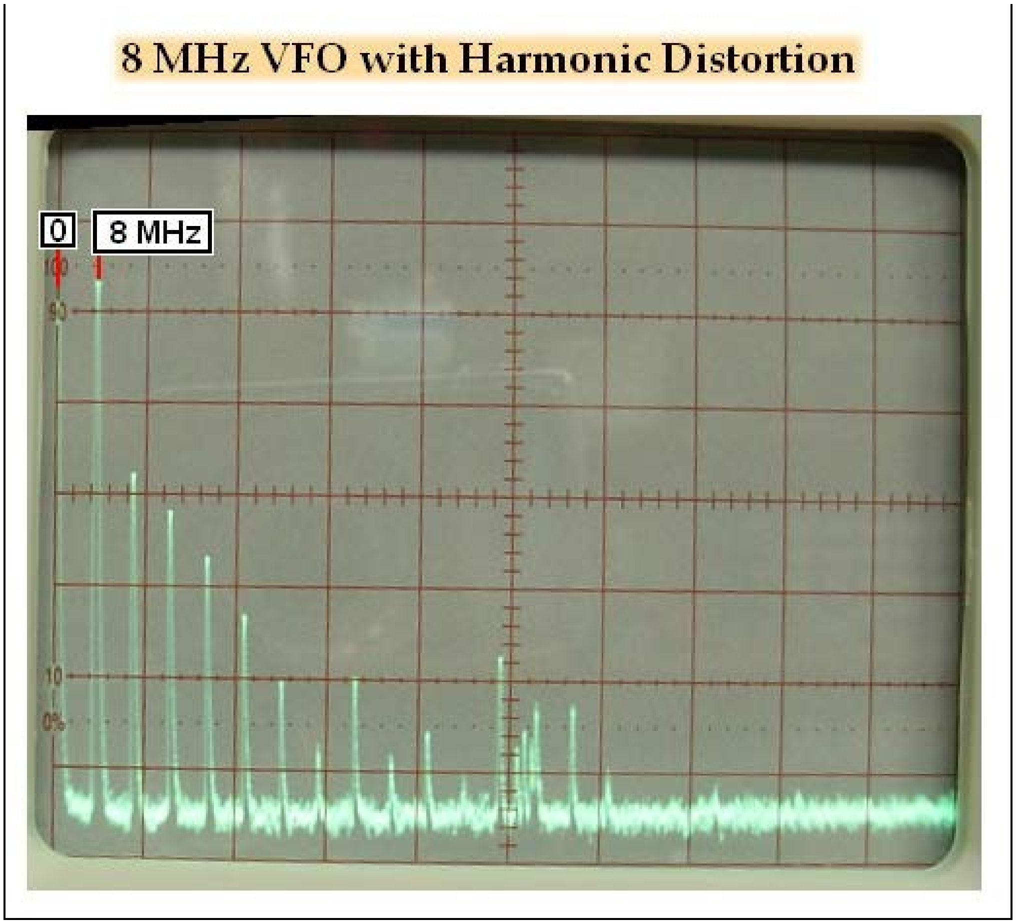
Examine the distorted signal on the left and compare it to same signal after low-pass filtering. I calculated the power in dBm from the peak-peak voltages (Vpp) shown as 832 mV and 824 mV: left to right respectively.

Only the sinusoidal power proves accurate. I'll discuss the formula to assess power soon.

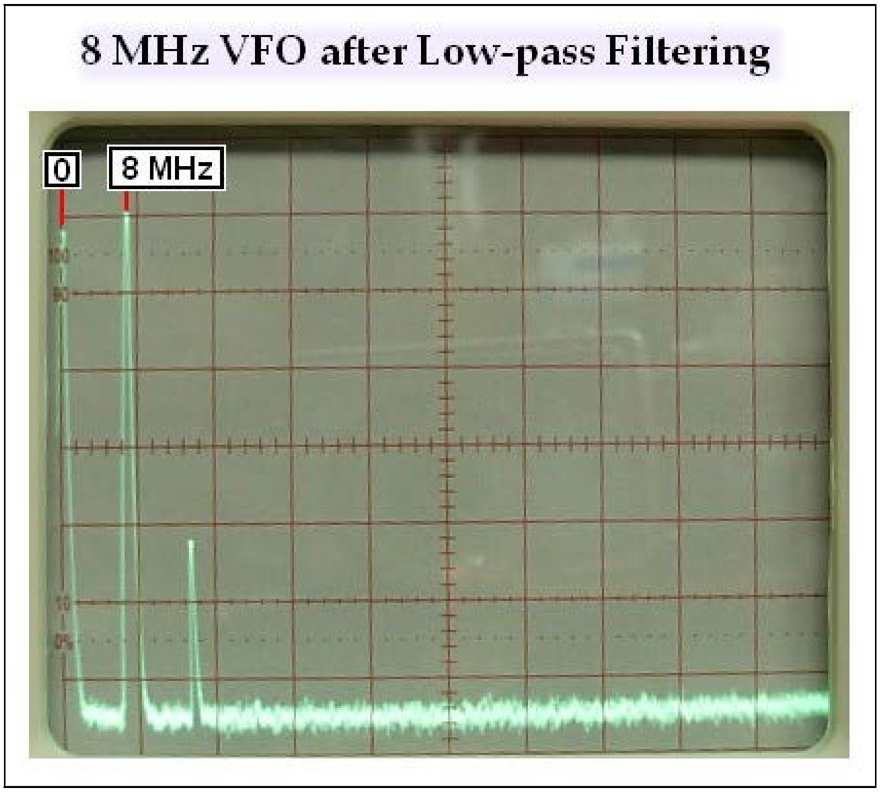


The same 8 MHz VFO examined with a 50 Ω terminated oscilloscope; a superb measurement technique that offers greater sensitivity. In all 4 cases shown, the vertical scale = 0.2 volts/cm.

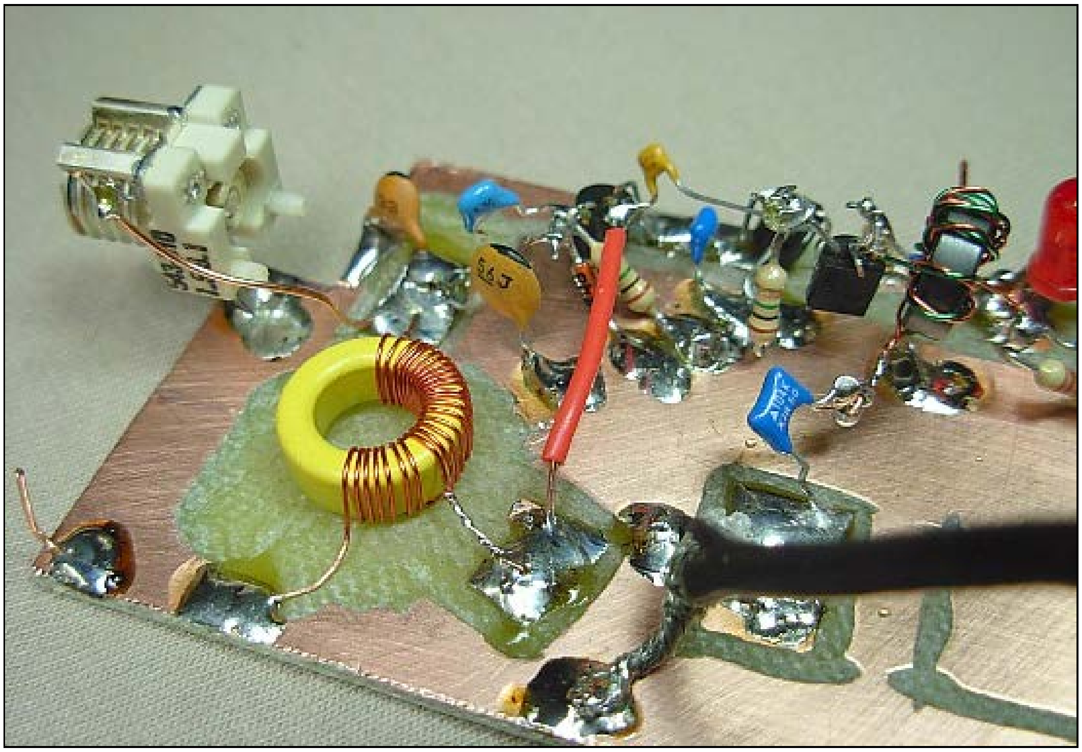
To better compare the distorted and filtered signals, I attenuated the output of the VFO to allow safe examination with a spectrum analyzer. A spectrum analyzer graphs the power (in dBm) of all measured frequencies on its Y axis against a user defined frequency range on the X axis.


8 MHz VFO with Harmonic Distortion

Spectral analysis of the distorted 8 MHz signal. The second harmonic (16 MHz) is about 22 dB down from the fundamental. The signal is rich in harmonics that causes error in the calculation of the output power. Each vertical square is 10 dB. Each horizontal square = 20 MHz. The harmonics go 2x fundamental, 3x fundamental, 4x fundamental and so on.


8 MHz VFO after Low-pass Filtering

Spectral analysis of the 8 MHz VFO after passing through an N = 5 Chebyshev low-pass filter. The second harmonic now lies about 40 dB down from the 8 MHz carrier (-40 dBc) and the 3rd harmonic is almost down in the noise. Each horizontal square = 10 MHz.

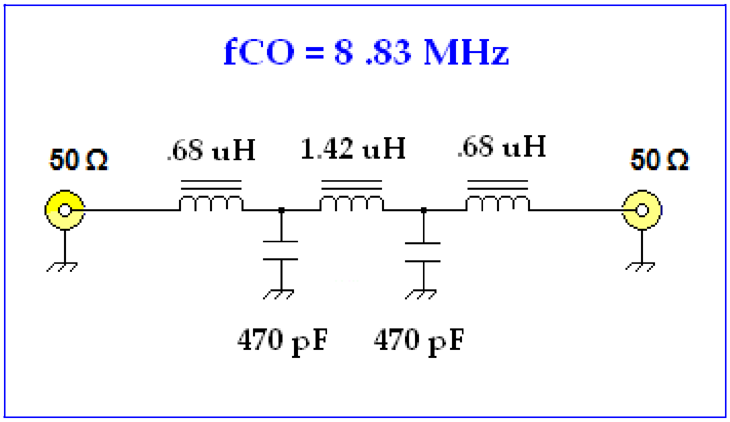


The breadboard of the 8 MHz oscillator oscillator from the above experiments. The output drove a BJT amp biased to give distortion and a 50 Ω output. I adjusted the frequency with the high Q air-variable trimmer capacitor seen to the left.

Low-pass Filter

Some builders wonder why I only employ sine wave signal generators on this web site. To calculate power, they require no low-pass filtering — now you know why. If you're calculating power from a distorted signal, a stiff low pass filter helps ensure measurement fidelity. All of my signal sources feature a 2nd harmonic response of at least -30 dBc, but -50 dBc is typical. To filter receiver front ends, signal generators, or mixer outputs, I keep several 7 element low-pass filter bench modules on hand that cover several 3 dB cut-off frequencies between 3 and 60 MHz.

Although, any old low-pass filters might work fine, Wes, W7ZOI suggests an N = 5 Chebyshev with 0.2 dB of ripple at about 1.2x the signal frequency as a starting point for designing a test-bench low-pass filter. If you don't know how to design low-pass filters choose a pre-designed filter from a filter table. For the experiments above, I selected a filter from an ARRL Handbook. See the schematic below:



Calculating Power (dBm and mW) from Peak-to-Peak Voltage

To calculate the power from peak-peak voltage, the load impedance (Z) must be known. In RF design, the standardized impedance value = 50 Ω. For CATV and video, 75 Ω is common, and in audio and telecommunication design, a 600 Ω impedance dominates. Although we can technically employ any Z, this web site conforms to the 50 Ω RF impedance standard.

The SI unit of power is the watt. In radio, we might see the term dB used, however, dB is a decibel comparison between 2 signals and not an absolute value like the watt. On the bench, dBm serves as the most common and useful term — dBm is the measured power ratio in decibels referenced to 1 milliwatt.

dBm represents an absolute power — so useful because both large and small signals are quantified with 1 number. Some important bullets follow:

- 0 dBm = 1 mW
- 3 dBm = ~2mW; so doubling the power from 0 dBm equals a 3 dB increase in power
- Increasing the power from 0 dBm to 10 dBm boosts power by 10 dB. The power is now 10X baseline or 10 mW
- 20 dBm = 100 mW
- -27 dBm means that the output has ~500 times less power than 1 milliwatt. -27 dBm = 0.002 mW or 2 microwatts

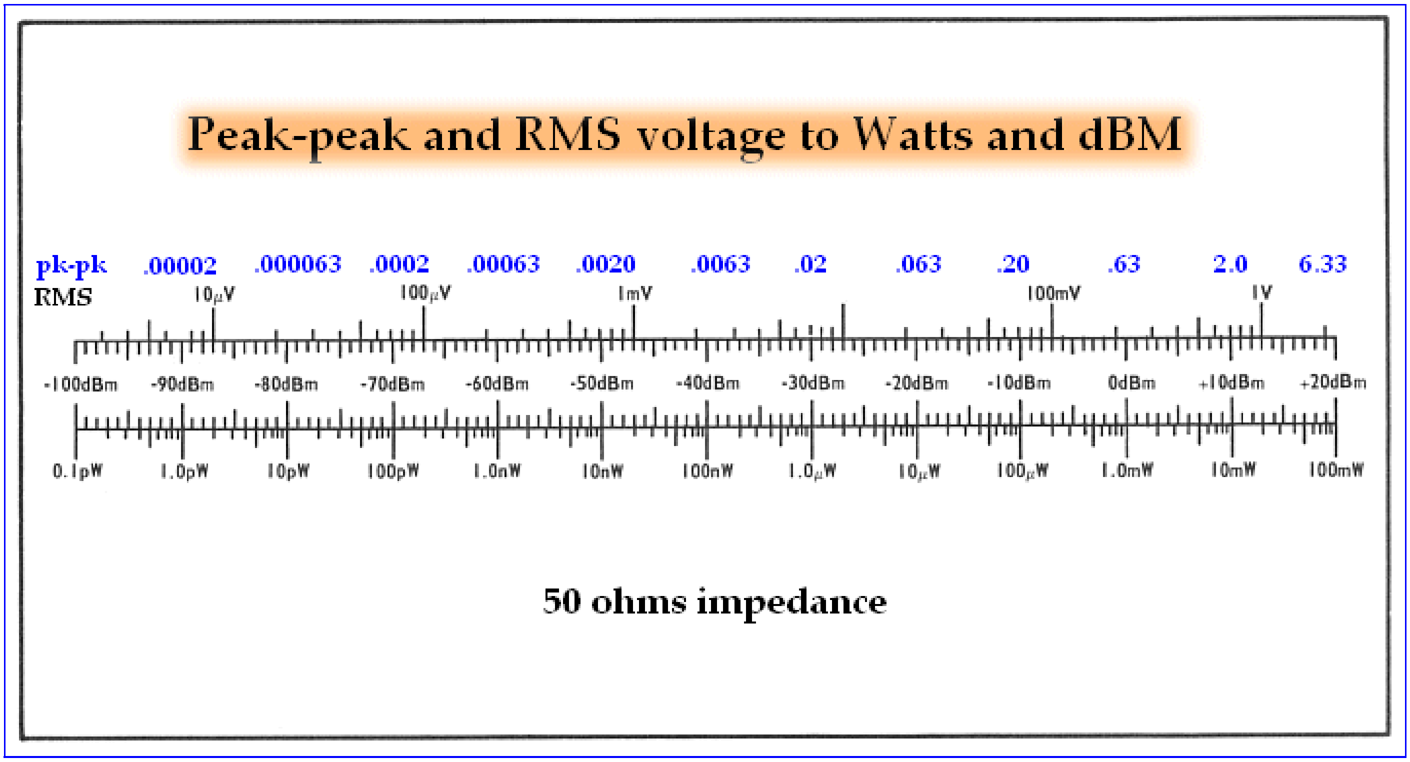
Hopefully over time, you' ll ingrain the concept of logarithmic power gain or loss (in dB) and power referenced to 1 mW (in dBm). This is bread and butter radio design information you must know.

50 Ω Measurement Virtues

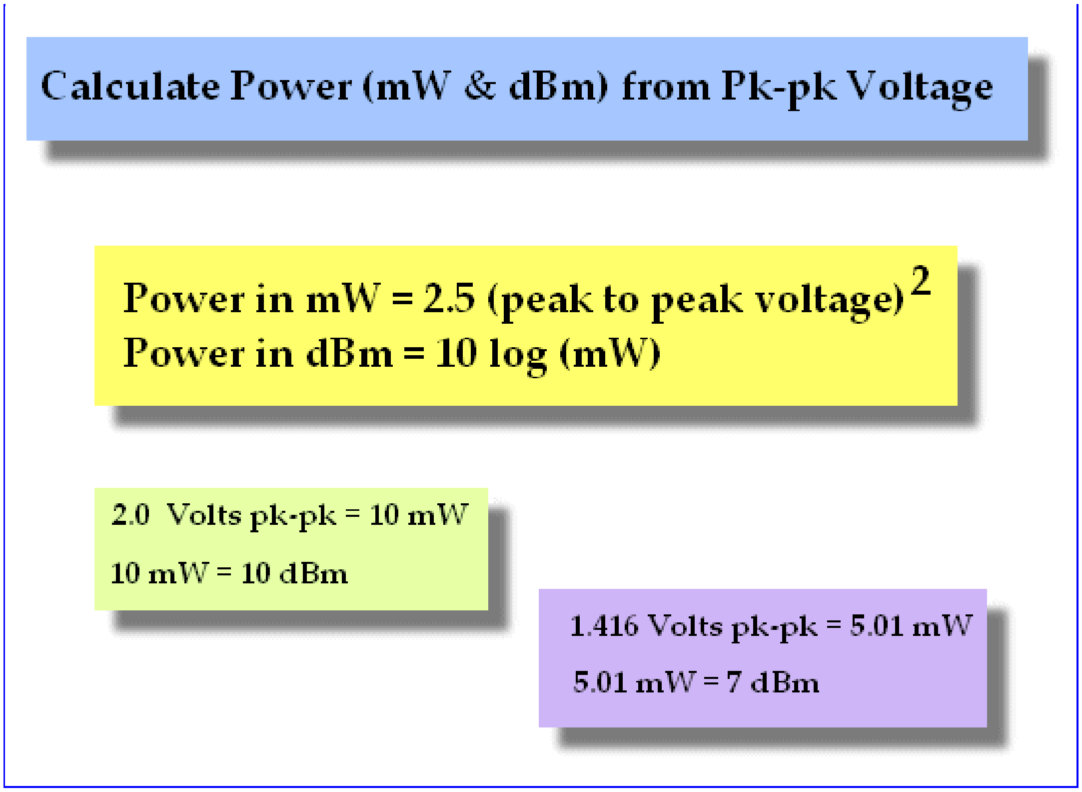
You build a VFO, measure it with your 'scope; calculate the output power into a 49.9 Ω resistor and then record this power in dBm. Let's say it's 6 dBm. VFO output power = 6 dBm.

Next, you place a 6 dB attenuator pad on the VFO output. VFO output power now = 0 dBm.

Finally you connect a 10 dB gain RF amplifer to your VFO. Your VFO output now = 10 dBm. What a beautiful system ! ..... it really gets fun when we measure down at - 30 dBm and so forth.



The chart above really helps you visualize the relationships of mW, AC voltages and dBm



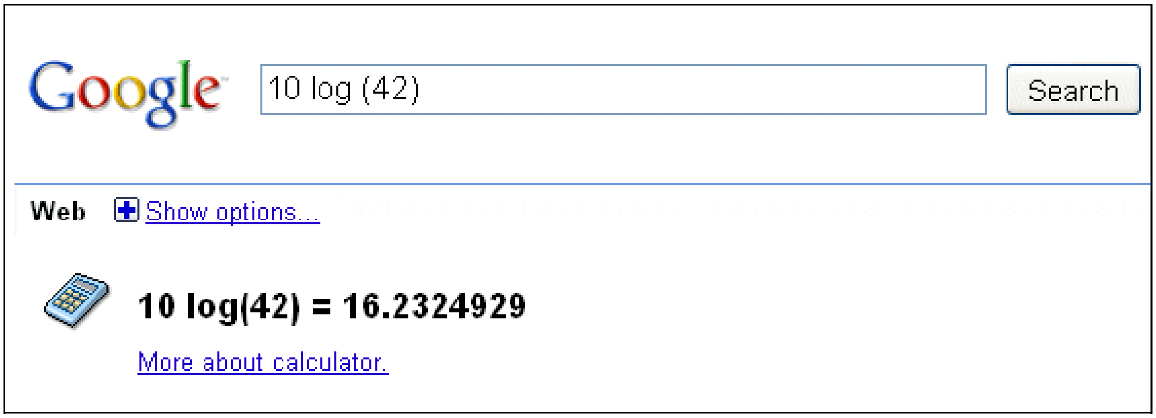
Calculate Power (mW & dBm) from Pk-pk Voltage

Power in mW = 2.5 (peak to peak voltage)^2
Power in dBm = 10 log (mW)

2.0 Volts pk-pk = 10 mW
10 mW = 10 dBm

1.416 Volts pk-pk = 5.01 mW
5.01 mW = 7 dBm

In order to get from peak-to-peak voltage to power, math is required. I show the formula above. You may elect to skip the math and calculate dBm or mW from peak-to-peak voltage with software. A number of programs are available; I wrote 1 here as Applet **F**.



10 log (42)

10 log(42) = 16.2324929

If you lack a scientific calculator, Google has math functions. Shown above = a logarithm calculation

In Search of 50 Ohms

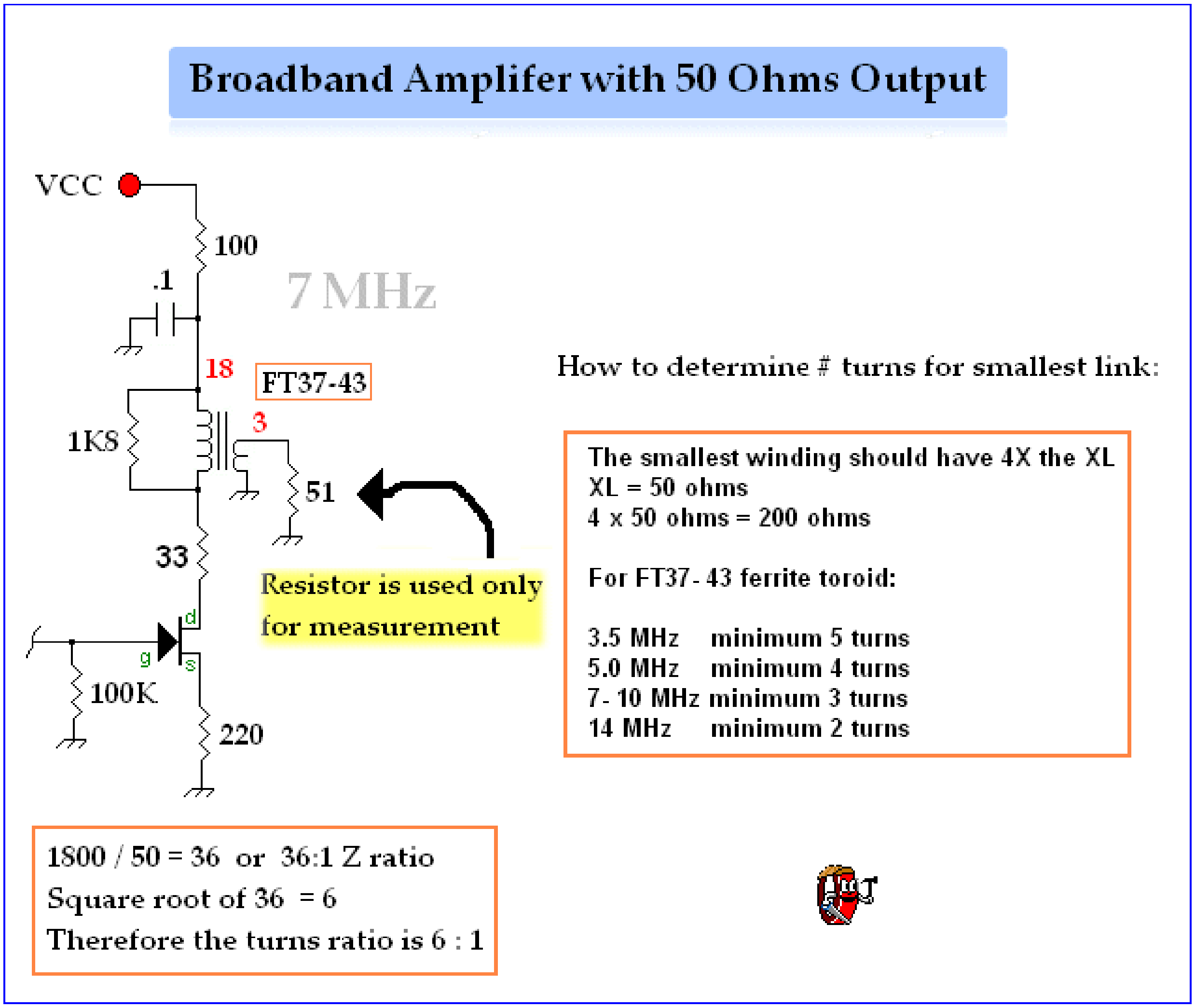
How do I establish a 50 Ω output impedance in my RF amplifier? I get this question often.

From my experience, in **simple** amplifiers lacking negative feedback, a 50 Ω output impedance must be created by inserting a resistor somewhere in the circuit that forces a 50 Ω output impedance.

For example, we might use a 50 Ω collector resistor in a BJT amplifier (or a 50 Ω drain resistor in a JFET amplifier), or place a fixed resistor in parallel with the collector/drain transformer and use a secondary winding to establish the 50 Ω output impedance. Sometimes we'll place a series resistor (say from 22 to 51 ohms) on the output of an emitter or source follower to bring the low output impedance up to 50 Ω. These form basic explanations and usage examples may be found on many schematics on the QRP / SWL website.

We experimenters also employ negative **feedback** with or without output transformers to establish a 50 Ω input and/or output Z and I show many examples on this site.

The following diagram explores 1 method to get a 50 Ω output impedance in a simple amplifier. It doesn't matter if the transistor is a JFET or a BJT, the principle is the same. This diagram and tutorial are simplistic and meant to help novice builders learn to design their own amplifier stages. You may connect any resistor value across the output of a transformer to calculate power, however, this web site only considers 50 ohms.



Broadband Amplifer with 50 Ohms Output

How to determine # turns for smallest link:

The smallest winding should have 4X the XL
XL = 50 ohms
4 x 50 ohms = 200 ohms

For FT37-43 ferrite toroid:

3.5 MHz	minimum 5 turns
5.0 MHz	minimum 4 turns
7-10 MHz	minimum 3 turns
14 MHz	minimum 2 turns

1800 / 50 = 36 or 36:1 Z ratio
Square root of 36 = 6
Therefore the turns ratio is 6 : 1

The above diagram describes a broadband (untuned) amplifier. I employed a FT37-43 ferrite toroid: a common part. Other ferrite toroids may be substituted, however the table depicting the minimum number of turns won't apply.

Consider the BJT amp shown. The transformer primary winding is shunted with a parallel 1800 Ω resistor. The 1K8 resistor "forces" a 1K8 ohm collector output resistance in the primary winding.

To transform the 1800 Ω primary impedance to 50 ohms; use a 3 turn secondary link. Calculate the primary to secondary turns ratio as follows:

1800 ohms divided by 50 ohms = 36. The impedance ratio = 36:1.
The turns ratio is the square root of the impedance ratio; thus the turns ratio is 6:1. The primary winding must have 6X the number of turns of the secondary winding. In the 3rd RF Workbench web page, you'll see that the above explanation pertains to the "ideal transformer", however, the concept is useful — especially to the target audience of this website.

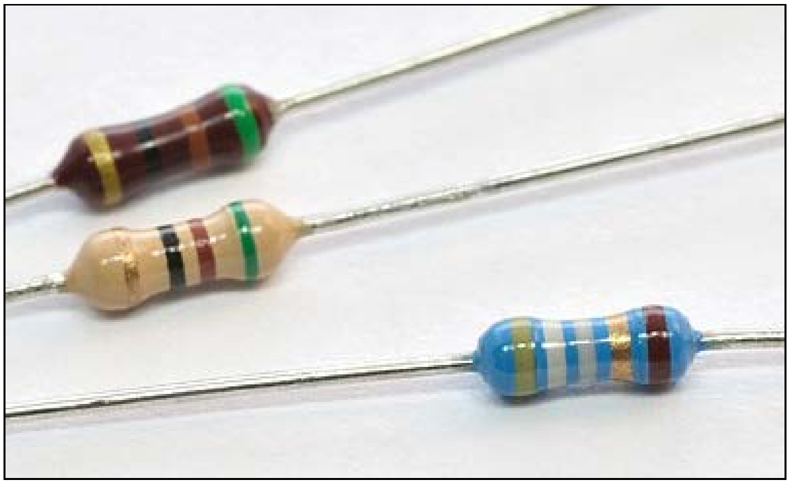
New builders might ask — why not wind 6 turns for the primary winding and 1 turn for the secondary winding? We avoid this because the smaller or secondary winding should have have a minimum inductive reactance (XL) of 4X the impedance it is connected to. Thus for a 50 ohm circuit, the minimum XL = 200 Ω at the design frequency.

This design rule serves only as a rough guide. We employ the minimum 4X rule because employing an XL less than 4X may create unwanted signal losses and affect the smaller winding's impedance. The table to the right of the amplifier shows the minimum numbers of secondary turns for a few common frequencies with the FT37-43 ferrite toroid.

Thus for our 7 MHz amplifier, we need at least 3 secondary turns and multiply this number by the turns ratio to give a 18:3 turns ratio. You might also choose 24:4.

For ferrites other than the FT37-43, calculate the minimum number of turns with the XL= 2 pi * F*L formula and detemine L from the turns versus AL toroid data, or measure L with an inductance meter.

For AC measurement a 50 Ω purely resistive load should be temporarily connected between the output link and ground. This might be a 51 ohm resistor, a 49.9 ohm 1% metal film resistor, 2 parallel 100 ohm resistors, or some other "50 Ω" load. We measure peak-to-peak voltage across the load and then calculate the power in dBm or mW. After measurement, the temporary 50 Ω load is removed and the circuit connected to the succeeding stage.

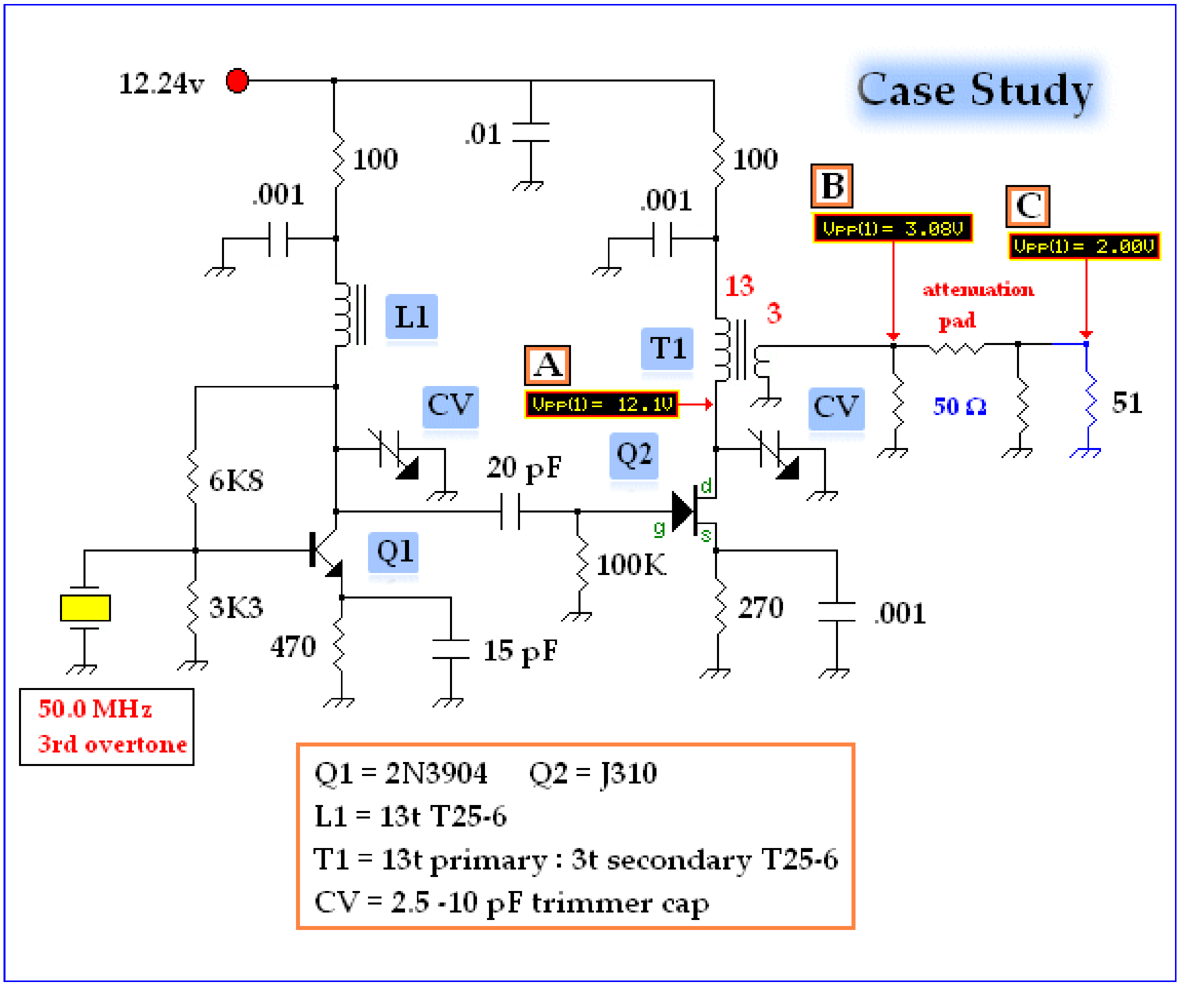


51 and 49.9 Ω resistors

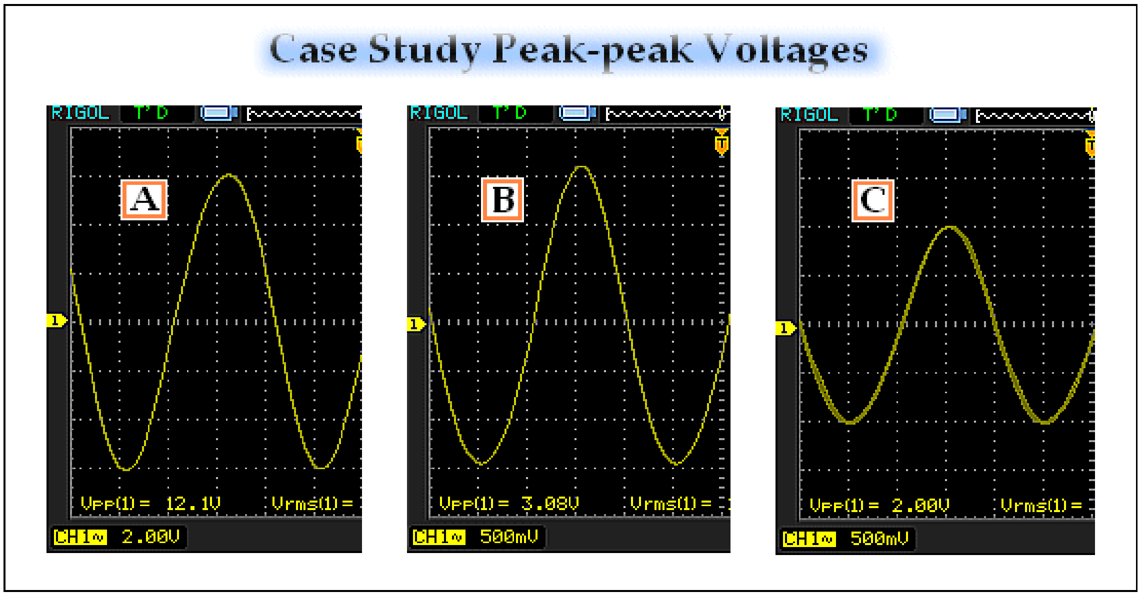
We measure the peak-peak voltage across this 50 ohm resistor with a 10X 'scope probe; or alternately may connect the device output to a 50 Ω terminated scope to measure peak-peak voltage to calculate power.

I normally measure in a 50 Ω measurement environment and temporarily solder a BNC connector onto my breadboard and connect this port to a 50 Ω terminated scope with coax. After testing and voltage measurement, I remove the RF connector and then build and test the next stage.

Case Study



Pretend that you breadboarded the above circuit entitled "Case Study". This is a 50 MHz crystal oscillator and buffer. The crystal fundamental frequency = 16.7 MHz, but the L1 tank is tuned to its 3rd overtone; 50.0 MHz. You measure and record the peak-to-peak voltages at the points labeled A, B and C.



The peak-to-peak voltages are shown as Vpp. The vertical scale (volts/cm) is shown on the bottom of each figure.

Examine Point A. The AC voltage = 12.1 volts peak-peak. Compare this to the peak-peak voltage at Point B. Note the difference. Some builders emailed me after they measured similar differences on the primary and secondary transformer windings of their circuits with a 10X probe. These builders felt something must be wrong. All normal; you can expect the peak to peak voltage to roughly decrease (or increase) by the transformer turns ratio.

The 12.1 volts peak-peak decreased by a factor of 4.3 (13 / 3 turns ratio) which is 2.8 volts peak-peak. In our case, the measured secondary peak-peak voltage was 3.08 volts — in the ballpark. Please remember this serves as a **coarse** guide only. It helps you to know what to reasonably expect during signal viewing.

Peak-to-peak voltage changing in accordance with the transformer turns ratio represents a simplistic explanation describing the "ideal transformer". To understand real world transformer function, you must contemplate factors such as Ohm's law for AC, conservation of energy (this is what causes the voltage to drop while preserving power) and basic transformer behavior. These principles are explained in publications such as The ARRL Handbook for Radio Communications, or the RSGB Radio Communication Handbook. An old high school physics text book might prove a better reference.

Here are the case questions:

1. Calculate the power in dBm at point B

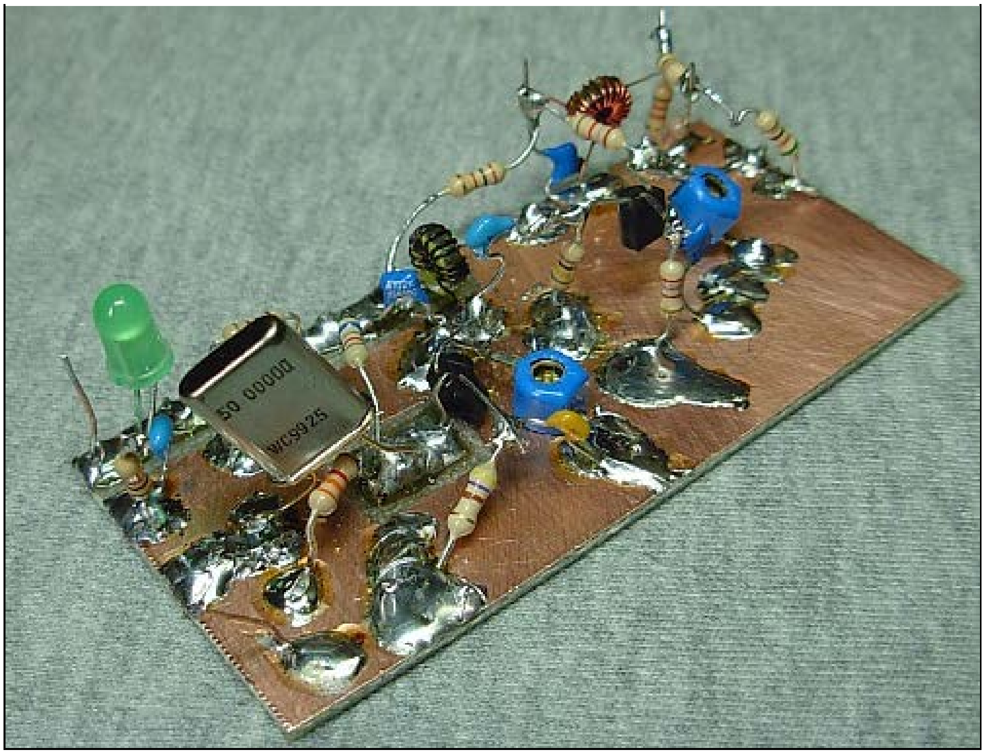
2. Calculate the power in dBm at point C

3.What is the attenuation in dB of the 50 ohms attenuation pad?

4. What is the output power in mW of this stage?

Click on this link for the answers and to see the actual resistor values of the attenuator pad.

Finally, placing a 10X probe at Point **A** will de-tune the L-C tank circuit somewhat and thus alter the AC voltage. In real-world building; to tune Q2, tweak the variable capacitor (CV) with your 10X probe connected to Point C.



The breadboard of the 50 MHz oscillator prototype.

Oscilloscope Probing

10X Oscilloscope Probe

Please refer to EMRFD Chapter 7 for great information about measuring power in RF circuits. The 10X oscilloscope probe is one of the most important measurement tools to have on your bench. There are countless web articles concerning the 10X probe, so I don't have much to add.

Take care of your 10X probe: don't solder components you've clipped your probe to; avoid setting heavy objects on the cable; store it carefully and inspect it frequently.

When do you use a 10X probe ? Measure with a 10X probe for in-situ ("in place", or "in circuit") voltage measurement and in situations where you can afford a 10X reduction in sensitivity. In low level measurements such as millivolt level measurements, the reduced sensitivity of a 10X probe may reduce or disallow accurate measurement. Additionally, the 20 pF or so capacitance of a 10X probe can detune resonant circuits; especially at VHF on up.

Close up of the Rigol oscilloscope probe 10X and 1X switch.

50 ohm Terminated Oscilloscope

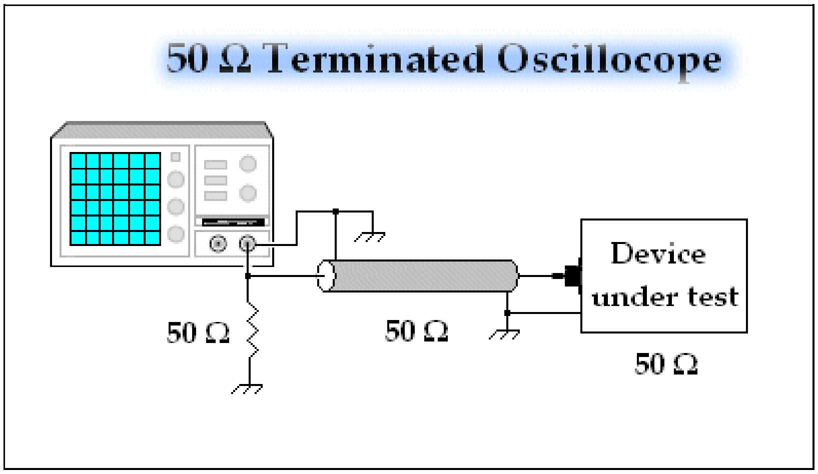
At RF, we generally work with (or try to work with) circuits with 50 Ω impedances. If possible consider performing your measurements in a purely 50 ohm environment.

That is — instead of using a 10X probe, shunt the oscilloscope input port to ground through a 50 Ω resistor and connect your test circuit to the 'scope with 50 ohm impedance coaxial cable. On my 'scopes, I have Channel 1 set up for the 10X probe work and Channel 2 set up for a 50 ohm environment.

I asked Wes about the benefits of performing measurement in a 50 Ω environment. I learned the main advantage of a 50 ohm approach is a well defined port impedance. The second virtue; a 10X greater voltage sensitivity — the increased sensitivity for low level measurement amazes me. In some cases, small signals that I couldn't accurately measure with a 10X probe, gave an excellent scope tracing with more consistent voltage readings in a 50 Ω environment.

You also may enjoy improved signal viewing. For example, in a few cases I have observed harmonic distortion with a 50 ohm terminated scope unseen with a 10X probe I confirmed this distortion with a spectrum analyzer.

If you have never performed measurement in a 50 Ω environment, consider trying it out — you'll enjoy it. You may buy commercial 50 Ω feed-through devices that connect to your oscilloscope input, or homebrew your own, but try to keep the 50 ohm termination as close to the oscilloscope input as possible.



Establishing a 50 Ω impedance measuring environment. The oscilloscope input is terminated with a 50 Ω resistance and connected to a device with a 50 Ω output impedance via 50 Ω coaxial cable.

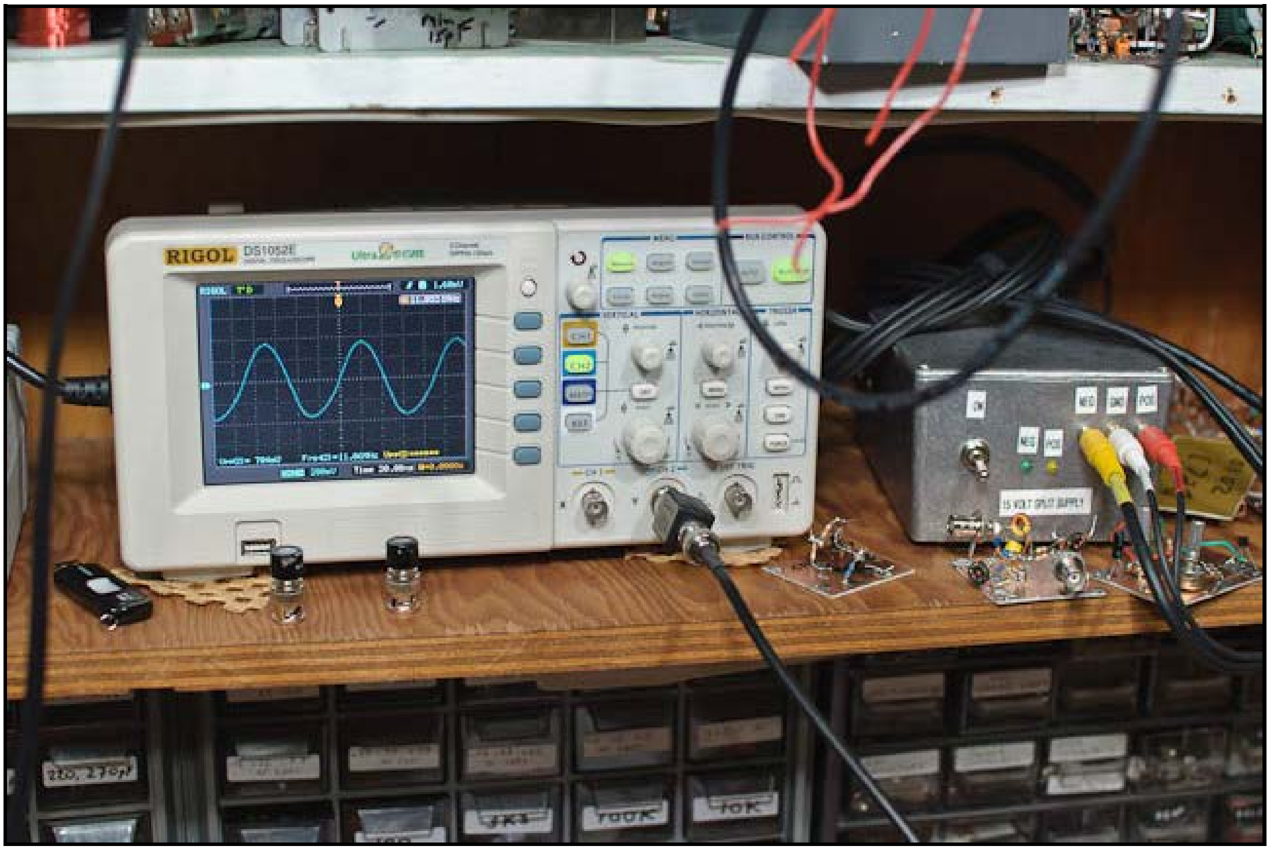
Try not to routinely connect a feed-through attenuator pad to your feedthrough 50 Ω 'scope terminator — error may arise.

My very first homebrew 50 Ω scope terminator module with 2 female connectors. I connected this module to my oscilloscope input via a commerical 9 cm long 50 Ω coaxial cable with a male connector on each end.

Two parallel 100 ohm resistors formed the 50 Ω load. Ideally, the 50 Ω shunt resistor should be right at the 'scope's female BNC jack — so this homebrew module shown fell short as a stalwart 50 Ω terminator. Inspired to move to a 50 Ω environment and lacking a male BNC connector, it did the job until my commercial version came by mail. You might find oscilloscope feedthrough terminators for sale at Ham festivals.

An ideal homebrew solution — place a male and female BNC connector in a small metal box very close together to allow a very short interconnecting wire. The box would hang off of the oscilloscope. Better still are commercial, shielded 50 Ω feed through terminators which thread right onto the oscilloscope's female BNC input jack.

Above — a commercial 50 Ω feed-through BNC terminator on my oscilloscope input.



Above — Measuring in a 50 Ω environment. Bliss! Я люблю это.

RF Current Sampler

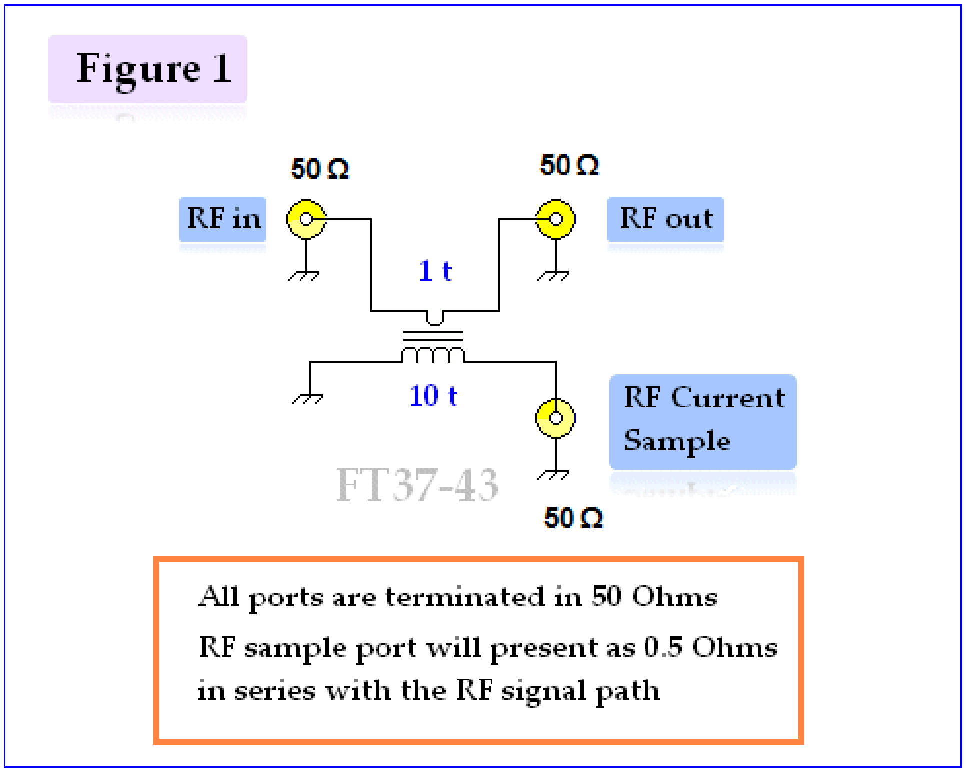


Figure 1

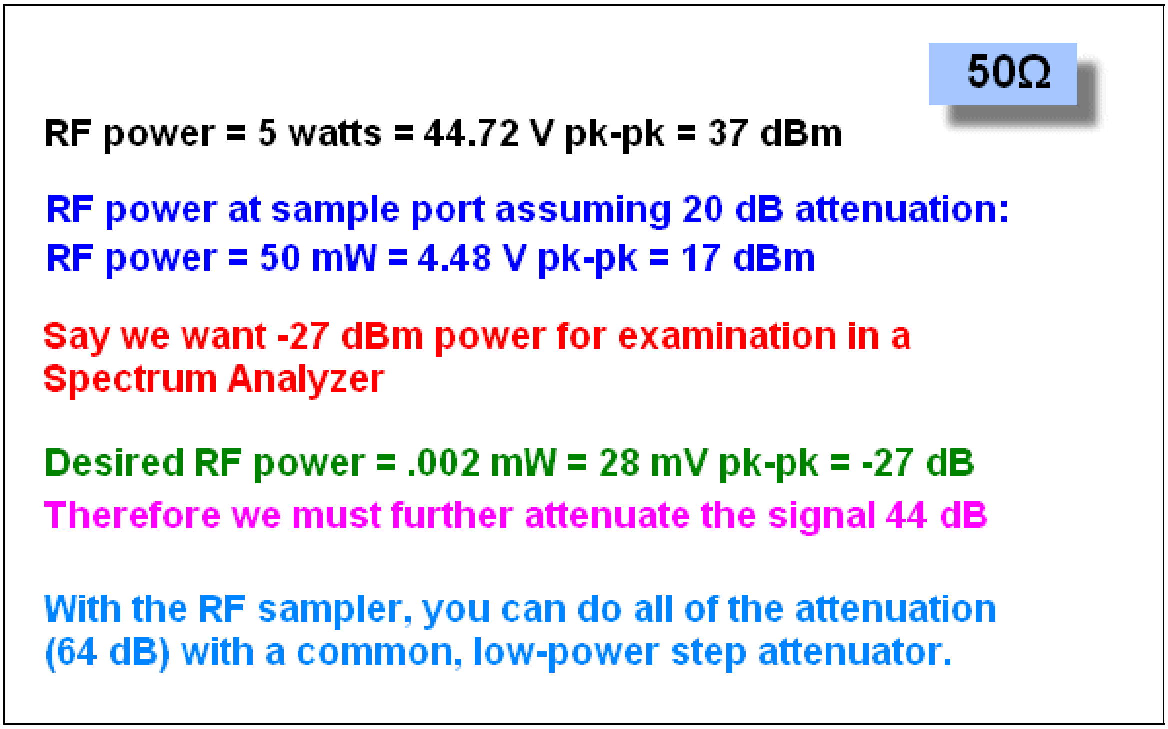
All ports are terminated in 50 Ohms

RF sample port will present as 0.5 Ohms in series with the RF signal path

Figure 1 shows a basic circuit to sample RF current from a power stage such as a QRP transmitter. Many experimenters lack 50 Ω step attenuators rated to handle transmitter-level power. One basic solution is to sample the RF current of the power amplifier using a wideband step-

down transformer. Terminate the RF current sample port with a 50 Ω impedance device. This may include a spectrum analyzer, power meter, receiver with an attenuator, or a 50 Ω terminated oscilloscope.
A usage example = examining a transmitter's spectral purity with a spectrum analyzer. The output power at the sample port will drop by 20-22 dB. A 50 Ω impedance step attenuator can be used to further reduce this power level to whatever you want. For this chore, a typical experimenter's 1-2 watt step attenuator works, since it never "sees" the higher wattage transmitter power.

For example, a 5 watt amplifier 20 dB down is 0.05 watts or 50 mW at the RF current sampler port. 50 mW = 17 dBm. To examine this signal with a spectrum analyzer you may wish to decrease the power down to -27 dBm. The following chart shows the basic process.



50Ω

RF power = 5 watts = 44.72 V pk-pk = 37 dBm

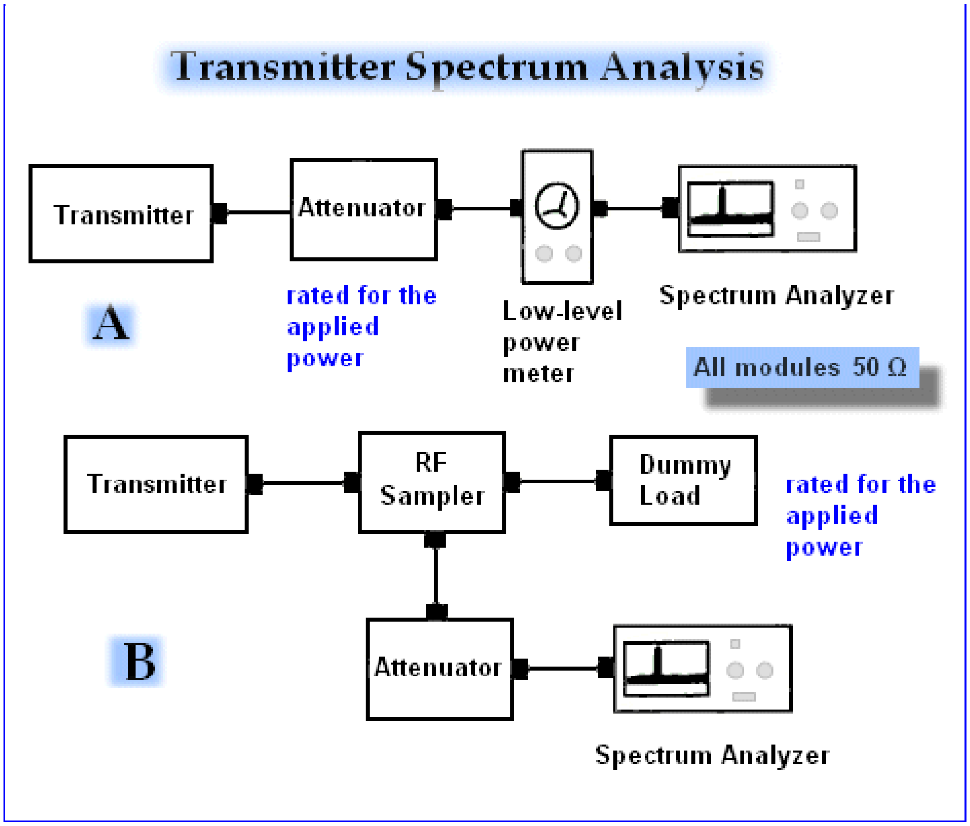
RF power at sample port assuming 20 dB attenuation:
RF power = 50 mW = 4.48 V pk-pk = 17 dBm

Say we want -27 dBm power for examination in a Spectrum Analyzer

Desired RF power = .002 mW = 28 mV pk-pk = -27 dB
Therefore we must further attenuate the signal 44 dB

With the RF sampler, you can do all of the attenuation (64 dB) with a common, low-power step attenuator.

A Hammond chassis shields the RF current sampler used on my bench.



The above graphic illustrates 2 methods to examine the output of a transmitter in a spectrum analyzer. Method B is described above. The dummy load must handle the transmitter output power, however a 5 or 10 watt dummy load is easy to make. Method A requires a step-attenuator which can handle the transmitter output power. The low-level power meter promotes the need to quantify the output power before you connect anything to the output of the attenuator. This is also true for Method B.

Always measure the output power at the RF sample port with your oscilloscope or low-level power meter before hooking up any expensive low-level measurement device such as a spectrum analyzer! осторожно!

Miscellaneous Photos

Above — a 50 Ω BNC terminator. These are essential QRP work bench items and may be found on eBay for cheap.

Click for a photograph of 4 of my BNC RF port terminators: 27, 50, 75 and 100 ohms from left to right — useful to calibrate and test RF circuitry.

RF — Test and Measurement

RF Workbench Page 2

Welcome to part 2 of a web series exploring basic RF measurement and bench practices. This installment builds on the information from the RF Workbench Page 1

I share introductory and practical content on attenuation, the return loss bridge, insertion loss or gain and spectrum analyzers. Consult EMRFD and use your favorite web search engine for more information.

The Attenuator Network

Like onions in your kitchen, the importance of attenuators can't be overstated. On the bench, attenuator pads go in nearly every test circuit to deliver correct power and/or help match port impedances.

Think of attenuators as passive networks that intentionally insert power loss between 2 components independent of frequency. For example: you might follow a 50 Ω signal source with an attenuator to decrease its power, increase its return loss, and/or buffer it from downstream impedance changes.

Most attenuator networks have fixed input and output impedances.

The input and output impedances may be the same, or different.

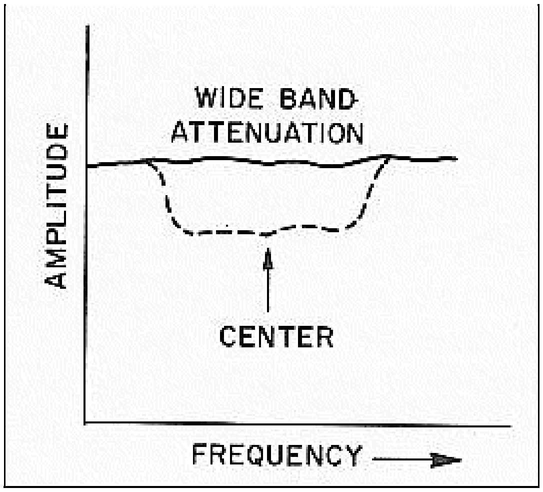
Attenuation may be fixed or variable. Most often, we use simple, fixed resistive pads that function as voltage dividers.

Express attenuation in dB.

Attenuators increase return loss and reduce VSWR.

Attenuators may function as buffers to isolate stages.

All attenuators on this web site feature 50 Ω input and output ports.



The desired response of an attenuator network. Practically speaking, device construction techniques including shielding will limit how high in frequency your attenuator will properly operate.

A pair of commercial 50 Ω input/output impedance step attenuators from the past. Occasionally, you'll find them for sale at Ham festivals or estate sales. Most work well for HF and perhaps even VHF work depending on their design and condition. Visually examine and test the attenuator before use.

How to Design and Build Attenuator Networks

To design attenuators with a 50 Ω input and output impedance, I recommend viewing a table. Click for a table.

After choosing the degree of precision; solder up 2 or 3 resistors and you're done. Nearest value 5% tolerance resistors offer reasonable precision for our popcorn circuits, but combining 1% and 5% tolerance resistors works too.

Choose low inductance resistor such as carbon film types and strive for short lead lengths. Well consider resistor power dissipation — for example, an 8 dB attenuator pad will dissipate 84% of the RF passed through it. I have seen attenuator pads that were exposed to high power and some or all of the resistors were burnt and turned to charcoal. Clearly the operator did not regard the power rating of the attenuator resistors. Refer to EMRFD Section 7.4 for practical information concerning attenuator design and power dissipation.

Three of the attenuator pads from my bench attenuator drawer. When using Ugly Construction or its variants, you can solder in, change, or remove attenuator pads at a whim. A small stock of these pads speeds up your work flow.

A 10 dB attenuator pad from my collection. This box uses two 100 Ω (5%) resistors and a 68 ohm (1%) resistor for the 96.2 and 71.2 ohm resistances called for. I used the 1% tolerance part because all my 68 Ω resistors are metal film 1% tolerance 1/4 watt types. Perhaps, I'll pursue a closer match to the calculated resistor values 1 day, but this module works okay. You may also stick 2 seperate attenuators in the little Hammond box shown.

Two commercial BNC feed-through (in-line) attenuators. I use these *every* day and prefer them over homebrew R networks since they don't require a coaxial cable. I own many: two 6 dB, two 10 dB, one 3 dB and a 20 dB: all were purchased on eBay.

Two commercial SMA in-line attenuators for my VHF and UHF experiments: 6 and 15 dB. Click for a marker table with 4 data points derived from sweeping these 2 filters.

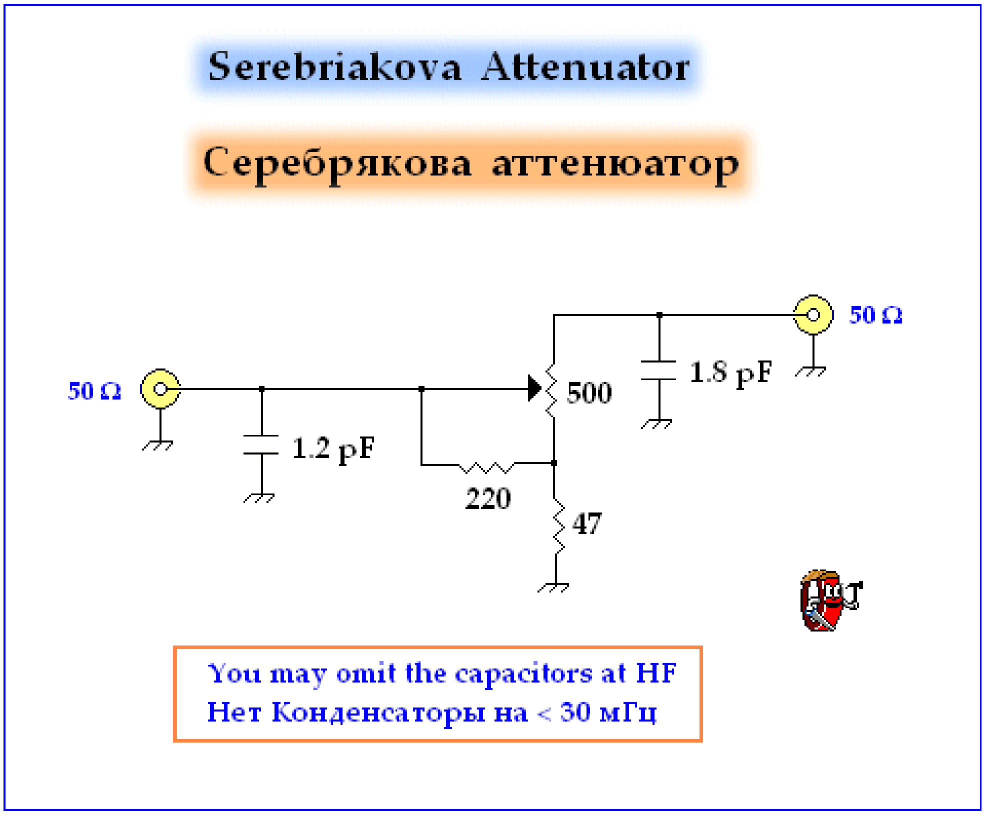
Step Attenuators

A step attenuator belongs on every serious RF workbench. They allow in-situ attenuation adjustment with a degree of precision as low as 1 dB. Step attenuators are nothing more than switched calibrated resistances and the switches can be SPDT, relays, rotary or digitally-controlled types. The quality and price of commercial attenuators varies widely. Experimenter concerns include the minimum attenuator insertion loss, power rating, return loss, noise from switch contacts and noise from the resistors themselves.

A homebrew step attenuator makes a great weekend project and almost every radio handbook contains 1. Web linked projects plus commercial kits may be found online — use your favorite search engine to find them.

Some homebuilders prefer 1% metal film resistors to keep resistor noise down. Stick your step attenuators in a metal, RF-proof box and insert quality interfaces such as BNC, N or SMA connectors. Your needs, budget and parts collection determine the outcome when you home build one.

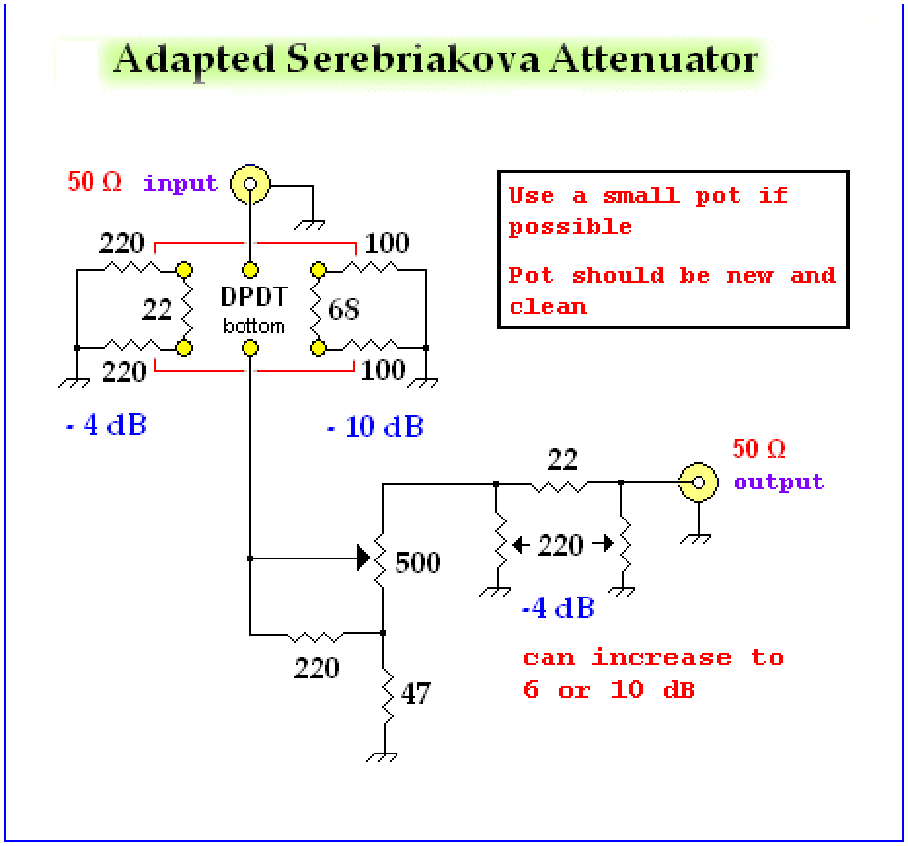
Serebriakova Attenuator - Серебрякова аттенюатор (50 Ом)



You may omit the capacitors at HF
Нет Конденсаторы на < 30 мГц

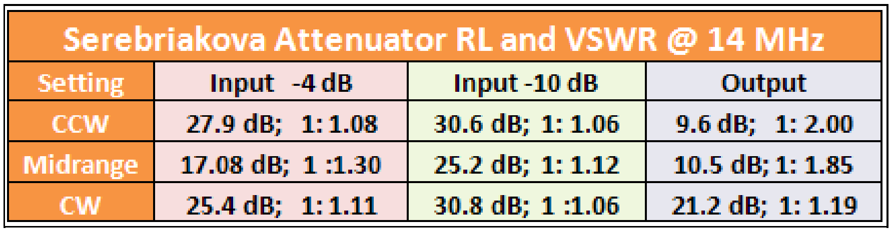
The Serebriakova; a simple, variable attenuator well suited for QRP homebuilding. Filled with gratitude to its Russian designer's family, I share this contribution with my readers. This attenuator network makes signals smaller or larger in a 50 ohm environment via a potentiometer. My analysis indicates acceptable performance considering its simplicity. The input match is close to 50 Ω across the range of the potentiometer. The output match across the potentiometer range is mediocre. Click here for a DC match analysis from Wes, W7ZOI. As shown, you wouldn't place this device on your main bench signal generator output as the output impedance diverges widely during amplitude adjustment.

Add fixed attenuator pads on the input and/or output to improve matching into 50 ohms. This circuit could serve in multiple applications including an RF gain control on a receiver front end, for bench measurement (when adapted) and for a low-level transmitter gain control. The Serebriakova attenuator may function up to 500 MHz in a carefully constructed, shielded box. The input and output capacitors may be omitted below 30 MHz. The attenuation varies a minimum of 20 dB when turning the potentiometer from CCW to CW. Click for a build by he yl.



A variant of the Serebriakova attenuator is shown above. Input and output matching are enhanced by fixed attenuator pads. The input match into 50 ohms is fine. After testing, I learned that the fixed 4 dB output attenuator pad is likely too low to ensure a wide range output match into 50 ohms. A 6 or 10 dB output pad is preferable, however, if this is your only variable attenuator, the device would then only be usable for very low-level work. You can decide what value of input or output pads to use.

A new, clean and small size 500 ohm pot works best. Store your potentiometer collection in sealed plastic bags to keep out workshop and house dust.



Serebriakova Attenuator RL and VSWR @ 14 MHz

Setting	Input -4 dB	Input -10 dB	Output
CCW	27.9 dB; 1: 1.08	30.6 dB; 1: 1.06	9.6 dB; 1: 2.00
Midrange	17.08 dB; 1 :1.30	25.2 dB; 1: 1.12	10.5 dB; 1: 1.85
CW	25.4 dB; 1: 1.11	30.8 dB; 1 :1.06	21.2 dB; 1: 1.19

Shown above are return loss (RL) and VSWR measurements performed on the adapted Serebriakova attenuator shown above. Clearly the input match is better than the output match. The output match did not significantly change when the attenuation switch was moved from 4 to 10 dB attenuation or back.

Based upon these values, it might be a better compromise to put a 3 to 4 dB pad on the input and a 10 dB attenuator pad on the output to ensure an output RL of at least 20 dB. Some might argue that the output RL should be higher. Perhaps, but the match is pretty good for such a simple circuit. Let's put it in perspective; a commercial signal generator that sells on the Internet for $450.00 U.S dollars was measured by a builder I know in the UK and he found a best case RL of 10 dB ! Jim later sold it and built a homebrew signal generator with a 35 dB return loss at all frequencies.

Fixed attenuator pads provide a good remedy for mismatched ports and I discuss why and how in the next section.

The shielded, adapted, Serebriakova attenuator. When home building your personal version, strive to make the AC connections as short as possible. The above device has nearly 23 dB of variable attenuation at 14 MHz. If you can't build, find, nor afford a precision step attenuator for your QRP workbench, this device may work okay for you.

Impedance Matching, Return Loss and VSWR

We radio folks build, buy and apply lots of gear with a stated *nominal 50 Ω input or output impedance*. In truth, a *pure* 50 Ω impedance occurs rarely and components in an RF system are frequently mismatched. Almost every Ham radio operator matches their antenna impedance to their feed line + radio to maximize transmitter output power — but radio and antenna system matching is often the only case where these Hams match their gear.

In contrast, we experimenters, tirelessly match our 50 ohm RF system components — this work flow avails our **modus operandi** on the bench**.** And so, we builders match the input and/or output ports of *all* our RF stages: signal generators, filters, splitters, antennas and so on. You can easily measure the impedance match of your RF components with a basic device based upon a Wheatstone bridge; the return loss bridge. First, let's discuss matching a little more:

On the RF Workbench, we talk about return loss, reflection coefficient and VSWR to quantify impedance matching. I only consider return loss and VSWR on this web site.

When 2 system components are impedance matched, maximal power transfers from 1 device to the other. If the impedances are different, RF power is reflected back to the signal source. This reduces the amount of power delivered to the load. Transmitted and reflected waves moving along a transmission line superimpose and cause standing or stationary waves. The greater the impedance mismatch between the 2 components, the larger the amplitude of the standing waves. Mathematical formulas compute how much power is lost due to mismatch. Consider reading a great tutorial on SWR, Return Loss, and Reflection Coefficient linked here by Wenzel Associates.

Return Loss

Return Loss = the difference between the outgoing incident power and the reflected power as a result of the mismatch between the the signal source and its load. Return loss is expressed in dB as a positive number on this web page. The higher the return loss, the better the impedance match. An ideal prefect match would have a RL of infinity; that is, no power is reflected back to the signal source and all of the incident power is delivered to the load. If a circuit has no load (open circuit), the RL is 0 dB — all of the power is reflected back to the signal source.

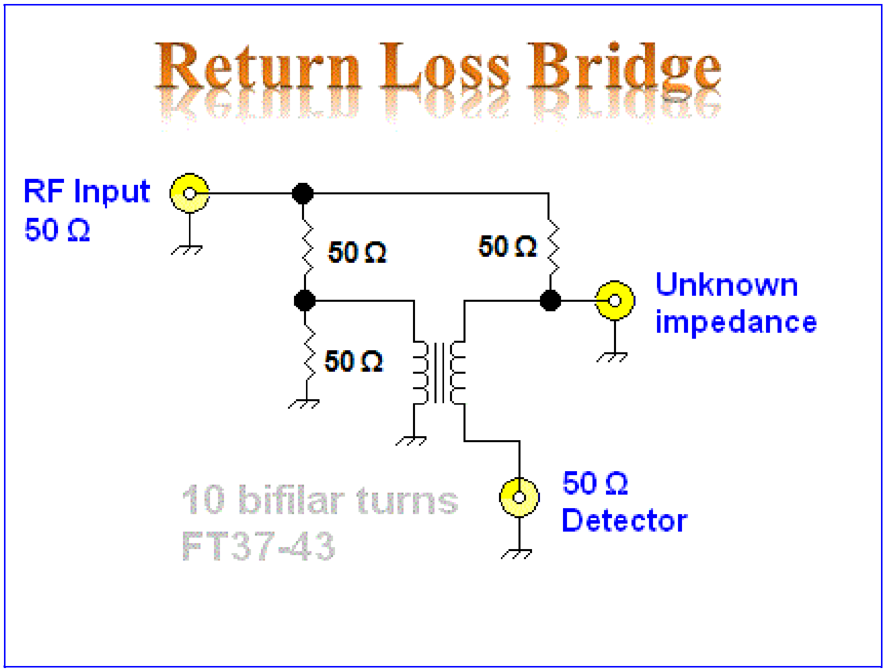
Other terms quantifying return loss are S11 and S22, however S11/S22 are the negative of return loss: RL = 20 dB or S11/S22 = -20 dB. We say S11 as S — one — one and S22 as S — two — two. I discuss these S-numbers, or Scattering Parameters elsewhere.

VSWR

Voltage standing wave ratio is another measure of how well the components in an RF network are impedance matched. Increasing the return loss lowers the VSWR and vice-versa. Most amateur radio enthusiasts are familiar with VSWR and often refer to it as "match" or "SWR". RL and VSWR can be derived mathematically from one other. VSWR = [10^(RL/20) + 1] / [10^(RL/20) - 1]. **Note X ^ Y means X raised to the power of Y therefore 2^3 = 2x2X2 = 8.**

Thus a RL of **10 dB** = 1: 1.92 VSWR and **20 dB** = 1:1.2 VSWR and **30 dB** = 1:1.07 VSWR

In EMRFD, Wes presents a return loss bridge as Figure 7.41. This circuit, shown below is easy to build and use.



The 50 Ω impedance detector may include a spectrum analyzer, power meter, receiver with an attenuator, or a 50 Ω terminated oscilloscope. On my bench, a 50 ohm terminated scope is favored.

Let's measure the return loss of a 27 Ω resistor to learn how. The procedure with a 50 Ω terminated 'scope follows:

Connect a 50 Ω output impedance signal generator to the bridge RF input port with 50 ohm coax;

Connect a 50 ohm terminated oscilloscope to the detector port via 50 ohm coaxial cable;

Record the peak-to-peak (open circuit) voltage with no load on the end of a short coax cable connected to the unknown Impedance port;

Record the peak-to-peak voltage with "unknown" coaxial cable terminated with the 27 Ω resistor

Calculate the power difference in dBm between these 2 peak-to-peak voltages.

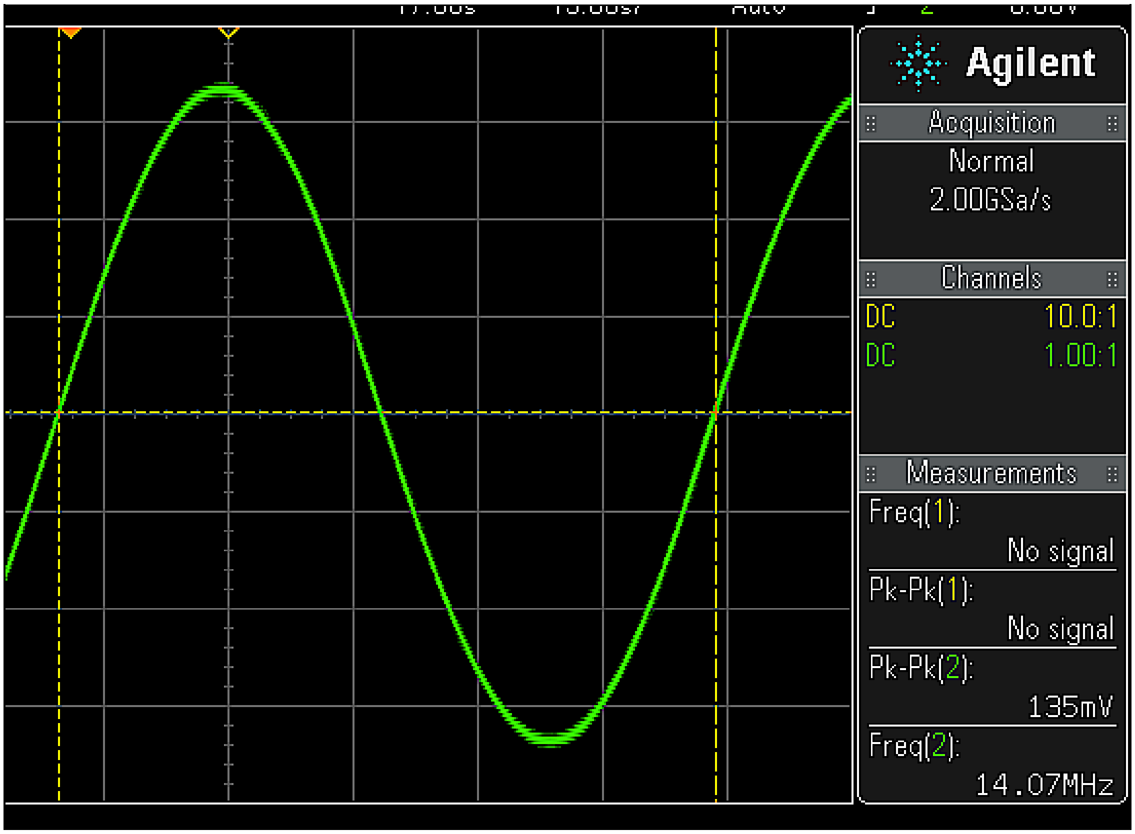
Return loss = the difference in dB between these 2 values calculated by hand or with software. Please refer to the RF Workbench Page 1 for information how to calculate power. I wrote a JavaScript Applet that take these 2 peak-to-peak voltages and calculates RL and VSWR; its labeled **K** on this web page.

Before measuring the unknown RL of a circuit, we usually connect a 50 Ω terminator to the unknown impedance port and calculate the best possible return loss: we refer to this value as bridge directivity — the best possible match for that return loss bridge at that test frequency. I keep a permanent 50 Ω terminator + a barrel connector on my bench for this purpose.

Click for a photo of the gear I use for all RL measurement. *Best viewed at full resolution*

Lets run through the procedure to measure the return loss of a 27 Ω resistor again, but with added photographs and 'scope captures. I tested at 14.070 MHz.

Above — We'll measure the return loss of this device; a 27 Ω resistor soldered on a BNC connector. We call this a resistive terminator and I keep a small collection of 27 - 100 Ω terminators on-hand for calibration purposes.



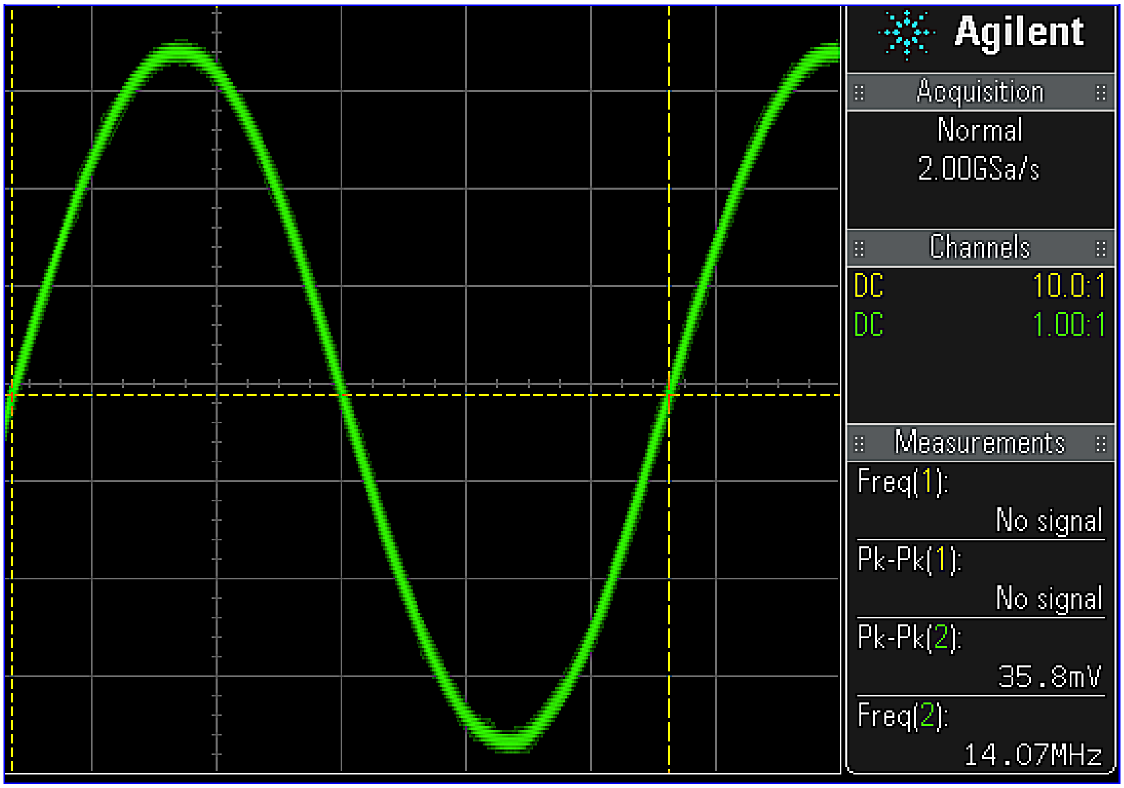
Normal
2.00GSa/s

DC 10.0:1
DC 1.00:1

Freq(1): No signal
Pk-Pk(1): No signal
Pk-Pk(2): 135mV
Freq(2): 14.07MHz

Shown above — The peak-peak voltage with a 14.07 MHz oscillator connected to the RF port; a 50 Ω terminated scope connected to the detector port; and a 20 cm — unterminated — 50 Ω cable connected to the unknown impedance port. **The open circuit measurement.**

Above — Next, I connected the 27 Ω terminator to the unknown impedance coaxial cable with a through-connector interface.

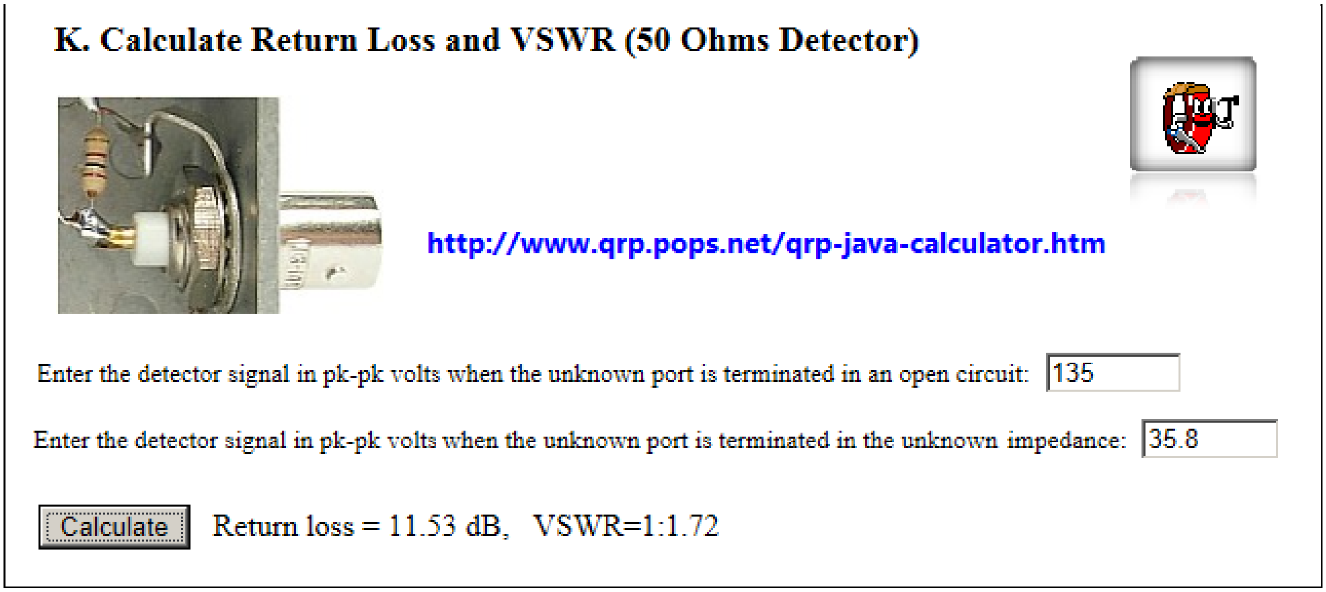


Acquisition
Normal
2.00GSa/s

Channels
DC 10.0:1
DC 1.00:1

Measurements
Freq(1): No signal
Pk-Pk(1): No signal
Pk-Pk(2): 35.8mV
Freq(2): 14.07MHz

Shown above — The peak-peak voltage with a 14.07 MHz oscillator connected to the RF port; a 50 Ω terminated scope connected to the detector port; and a 27 Ω resistive terminator across the end of the unknown impedance cable**. The reflected signal from a 27 Ω resistor.**



K. Calculate Return Loss and VSWR (50 Ohms Detector)

http://www.qrp.pops.net/qrp-java-calculator.htm

Enter the detector signal in pk-pk volts when the unknown port is terminated in an open circuit: 135

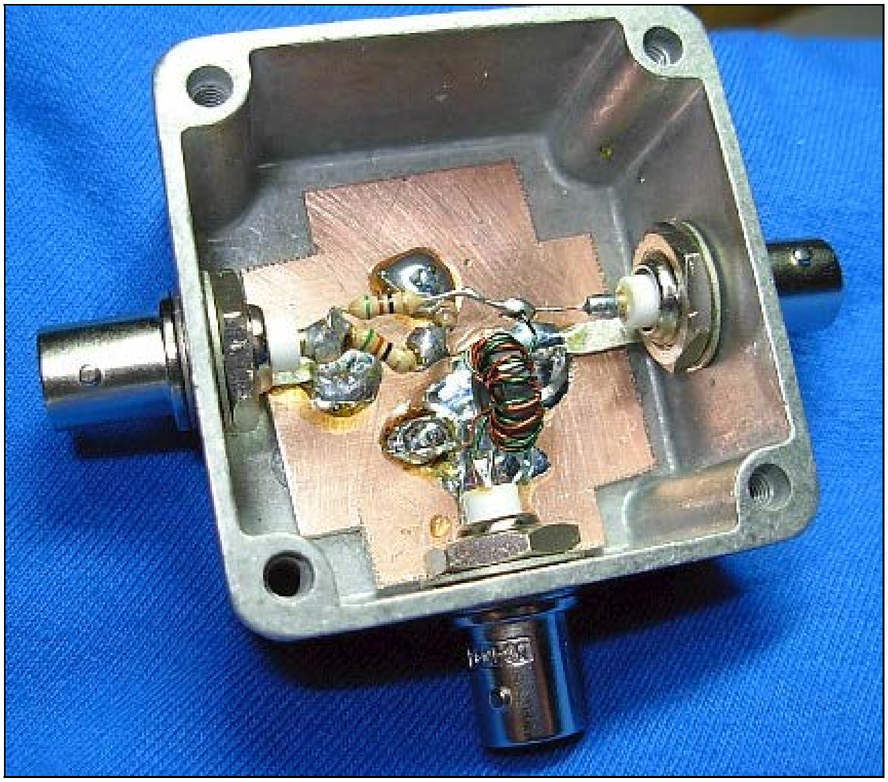
Enter the detector signal in pk-pk volts when the unknown port is terminated in the unknown impedance: 35.8

Return loss = 11.53 dB, VSWR=1:1.72

Shown above — Calculating the return loss and VSWR of the 27 Ω resistor with my JavaScript Applet **K**. mV versus volts peak-peak does not matter since we calculate a ratio of power.

Use the return loss measurement procedure depicted above to measure the return loss of any device you choose. If your D.U.T. has 2 ports, terminate the unmeasured port in 50 Ω. I show further RL measurment examples on RF Workbench 3.

I measured the return loss of some commercial gear in my shack and yard and will share 2 examples: 1) An expensive commercial transceiver I borrowed had an input port return loss of 15 dB (a 1.4:1 VSWR) The return loss of 15 dB indicates that the reflected wave power is 15 dB lower in power than the incident wave. 2) With a borrowed commercial bridge, my tuner-matched antenna revealed a return loss of ~60 dB.



A RL bridge from my bench built with 51 Ω 5% tolerance resistors. I show a better RL bridge and some other experiments on RF Workbench 3.

Return Loss and the Attenuator Network (How Do Attenuator Pads Improve Component Matching?)

We routinely employ attenuator pads to increase return loss in a 50 Ω RF environment. For example, let's say you're testing a signal generator and measure a return loss of 6 dB. If you place a 10 dB attenuator pad after the signal generator, the return loss increases to 26 dB. If we used a 6 dB pad instead, the return loss would now = 18 dB. In both cases the return loss is increased by 2x the attenuator pad value. The doubling of return loss occurs because both the incident wave and reflected signals pass through the attenuator pad — that's how attenuator pads improve matching.

Attenuator pads reduce power, but that is why somebody invented the RF amplifier.

What is the minimally acceptable return loss for a device such as a signal generator? No single answer exists. The minimum return loss depends on the context: are you making precision circuits or just tuning an antenna?

Precision Circuits:

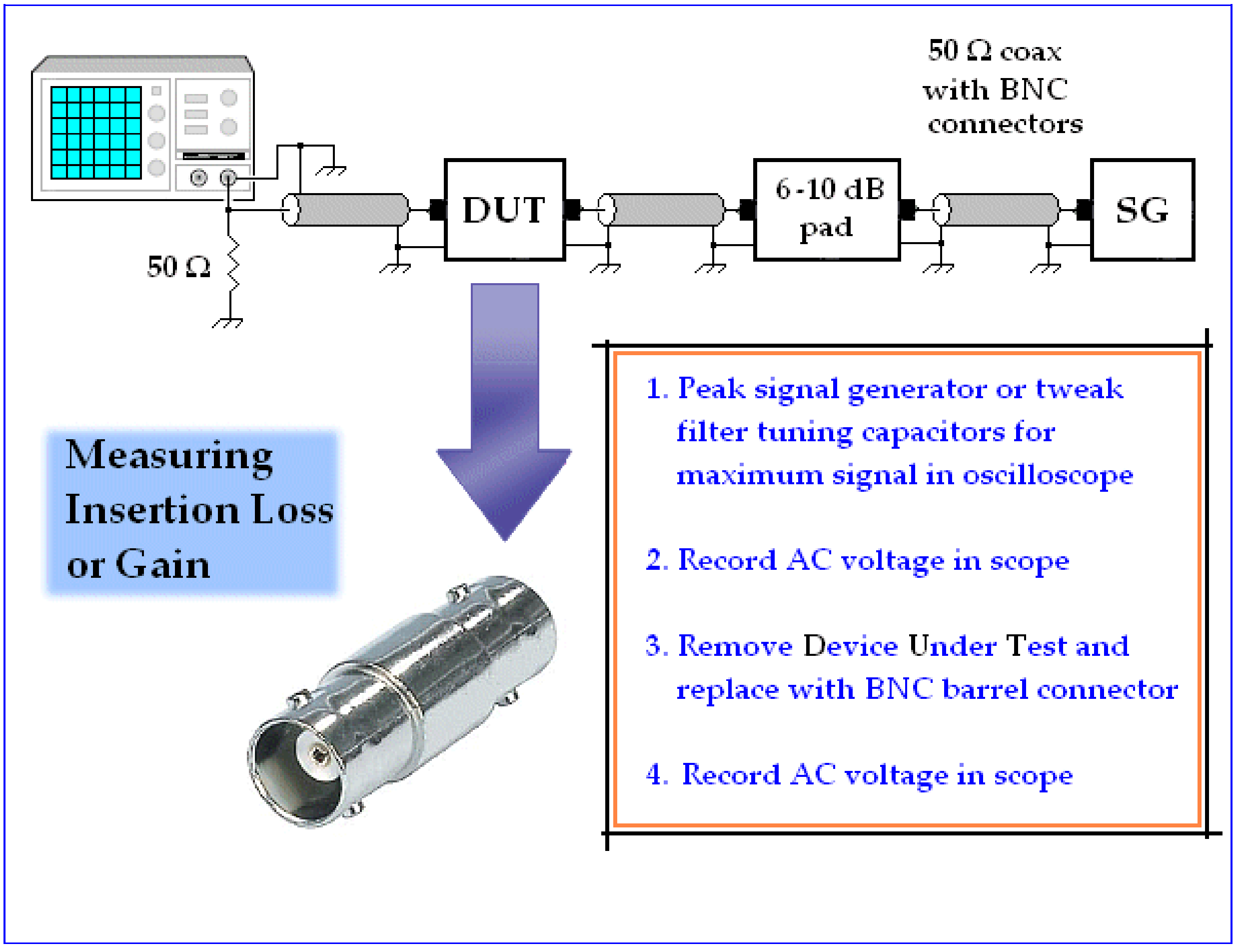
For amateur experimenter bench circuits, aim for a return loss of at least 20 dB. This often means adding an attenuator pad to the ports of your signal generator, amplifier, or other device to get a minimum 20 dB return loss. For an electronic engineer, the minimal return loss is probably higher; maybe 30 dB or so. I have read conflicting opinions about this and for some people — me included — design overkill is normal.

Antenna tuning:

When tuning an antenna for full transmitter output power, the minimal return loss is around 14 dB (a VSWR of 1:1.5). If you measure an antenna system return loss of 14 dB or better, the match is fine. Many Hams will protest a 1:1.5 VSWR and ardently chase a 1:1 VSWR on every frequency with their antenna tuner.

A Method to Measure Insertion Loss or Gain

Often, we want to measure the gain of an amplifier, or the insertion loss of a filter, or attenuator pad. I show how to do this with a 50 Ω terminated scope:



1. Peak signal generator or tweak filter tuning capacitors for maximum signal in oscilloscope

2. Record AC voltage in scope

3. Remove Device Under Test and replace with BNC barrel connector

4. Record AC voltage in scope

The circuit starts with a signal generator set to the frequency of interest. I show an attenuator pad in this diagram to stress that the signal generator output port must have a return loss >= 20 dB.

Connect the input of the 50 Ω **D**evice **U**nder **T**est to the generator output via 50 Ω coax

Connect the 50 Ω output of the D.U.T. to your 50 Ω terminated oscilloscope

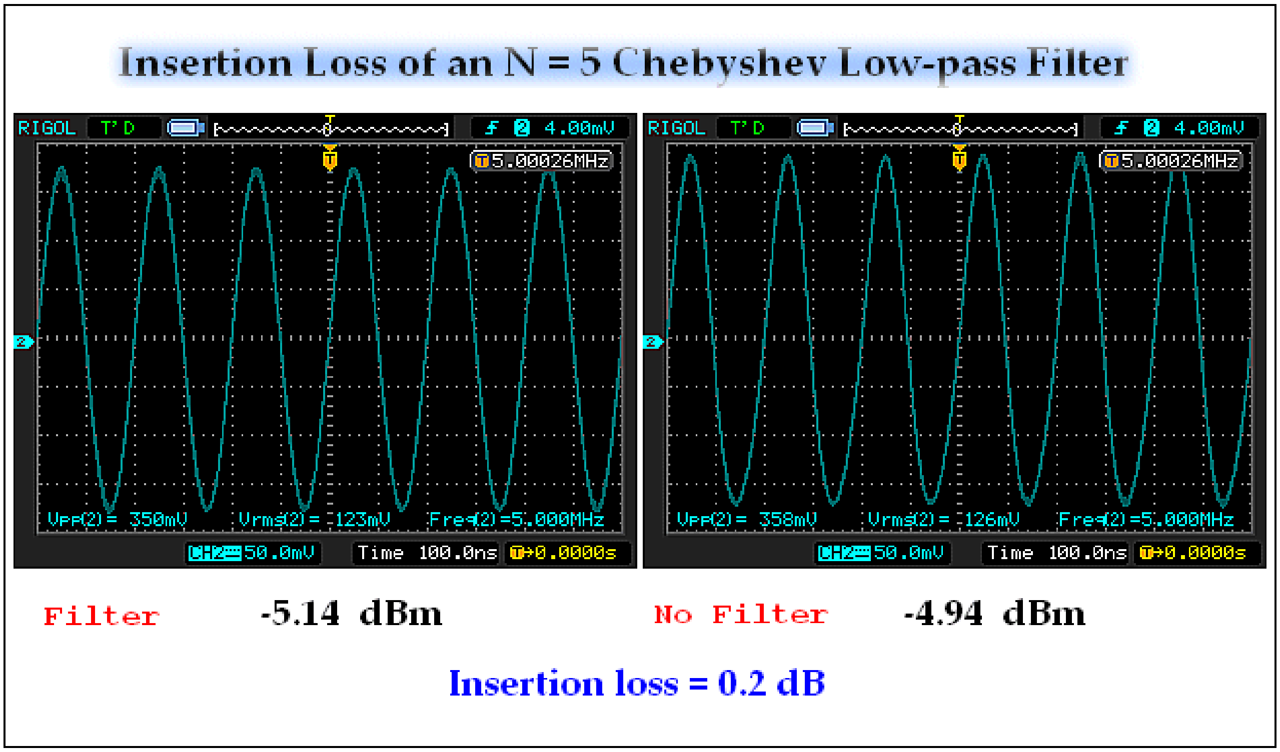
Turn on the signal generator and if needed, peak the signal; In the case of a low-pass filter, the signal generator frequency control is tweaked to give a maximum pk-pk voltage in your 'scope. When evaluating a band-pass filter, tweak the filter trimmer capacitors for the maximum signal at the desired center frequency. Signal peaking ensures that losses caused by the filters are not caused by the filter mistuning, or in the case of the low-pass filter, to allow for cutoff frequency deviation caused by component value variations. It may be necessary to increase the signal generator amplitude to view a good quality signal in your 'scope.

Record the peak-to-peak voltage.

Remove the DUT and replace it with a BNC through-connector and record this peak-to-peak voltage.

Calculate the power in dBm of the 2 recorded voltages — their difference equals the insertion loss or gain in dB. I wrote a JavaScript Applet to do this. Click and scroll to **H**

This awesome measurement technique controls the input and output impedance and uses the same coaxial cables with and without the D.U.T. for accuracy. Some builders might choose to terminate the D.U.T. with a 50 Ω resistor and measure with a 10X scope. The capacitance of the probe may alter measurement in some cases. As always, choose your measurement technique based upon whatever gear you own and how exacting your standards are.


Insertion Loss of an N = 5 Chebyshev Low-pass Filter

Filter -5.14 dBm

No Filter -4.94 dBm

Insertion loss = 0.2 dB

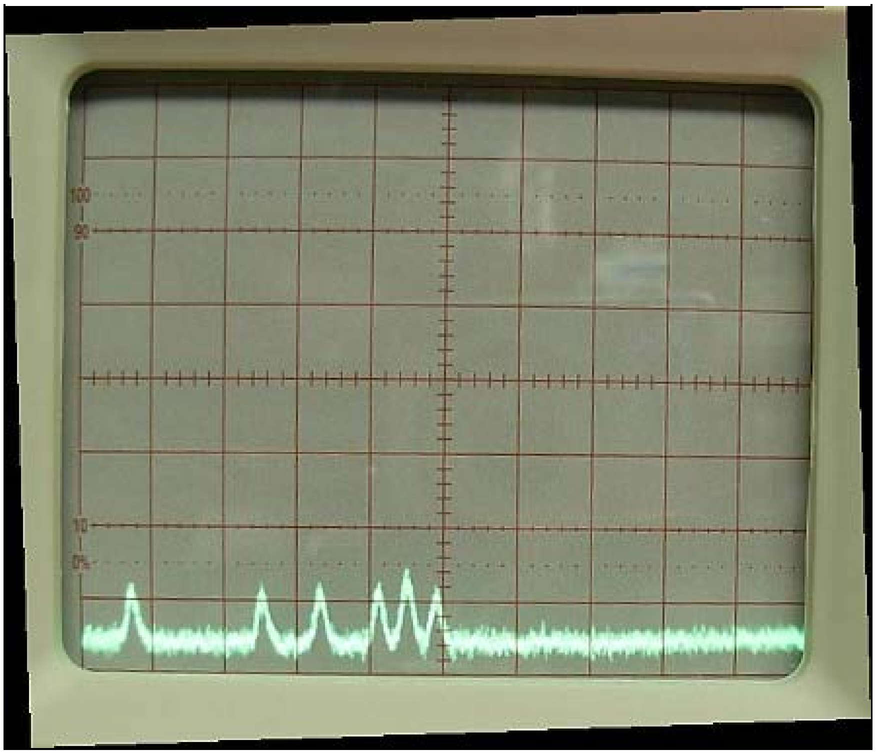
Spectrum Analyzers - Comments from the Workbench

Electronics professionals ruminate that spectrum analyzers are uncommon because experimenters perceive them as esoteric and difficult. My own opinion differs. Spectrum analyzers are relatively uncommon because of one reason - cost. I have watched prices on sites like eBay with amazement. The ads go something like this: 1.5 GHz spectrum analyzer for sale. Built in 1982. Ships in 2 pieces weighing over 22 kilograms. Minimum bid $1850.00. *And...sorry, I live in Florida, U.S.A. and in all likelihood, shipping these 2 heavy pieces is going to cost you a fortune. In the attached ad photos you can see lots of wear and tear, plus some screen burn-in on the display....* Guaranteed to turn on however!

Perhaps I exaggerate or even lampoon the perceived value of old boat anchor spectrum analyzers, but I have bought and sold cars for less money. Be prepared - spectrum analyzers are not cheap. They are however, very cool and open the door into a truly fascinating world. Frequency domain circuit measurement (spectrum analysis) addicts and intrigues. Homebuilding a spectrum analyzer is a serious option, but requires advanced building skills. Click and click for the W7ZOI/K7TAU project.

In recent times, the Rigol DSA-815 spectrum analyzer with tracking generator proved a game-changer to the bloated price of heavy, old and tired gear. Click for a Rigol datasheet. Signalhound also sells spectrum analyzers and tracking generators . A tracking generators *plus* spectrum analyzer allows you to sweep your device under test over a range of frequencies.

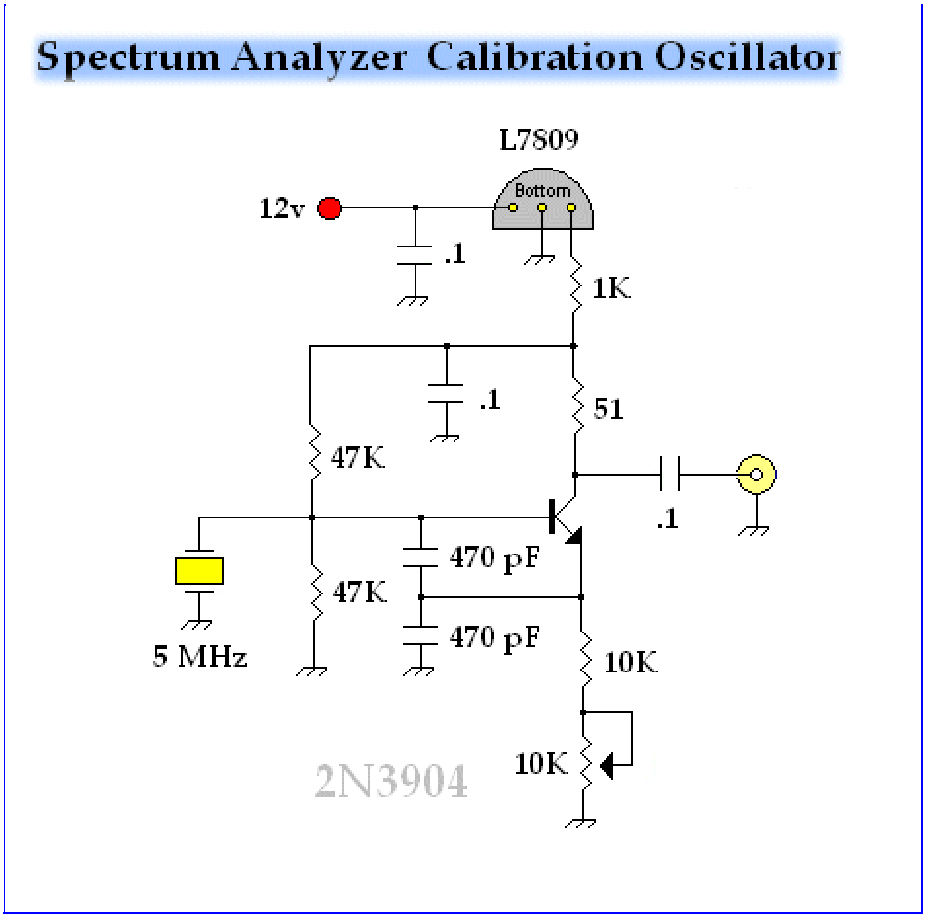
Prior to using a spectrum analyzer, I casually considered shielding stages or placing critical pieces in RF-proof boxes. Quickly I learned that RF in our home and community can and does get into your projects. p>



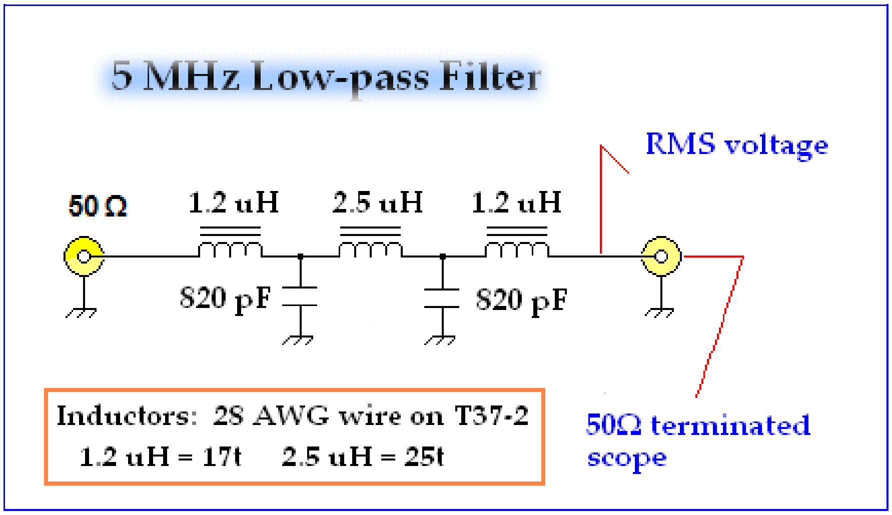
The center frequency of the display = ~150 MHz. The signal spikes appeared and disappeared after 4-9 seconds or so — after a little detective work with my scanner, I learned they were local police and ambulance FM radio conversations. I noticed this interference when I took the lid off a RF-tight band-pass filter — these signals arose in a 28 MHz superhet receiver !! While low in amplitude, experiences like this inform us to watch for lurking RFI.

I found numerous sources of RF in our home with a spectrum analyzer — the clothes washing machine during its spin cycle proved to be the worse RFI generator. RF-tight shielding with SMA, or BNC connectors and DC feed-through capacitors and aggressively decoupling and bypassing DC lines eliminated many of RFI problems during my experiments. I now better appreciate these anti-RFI techniques.

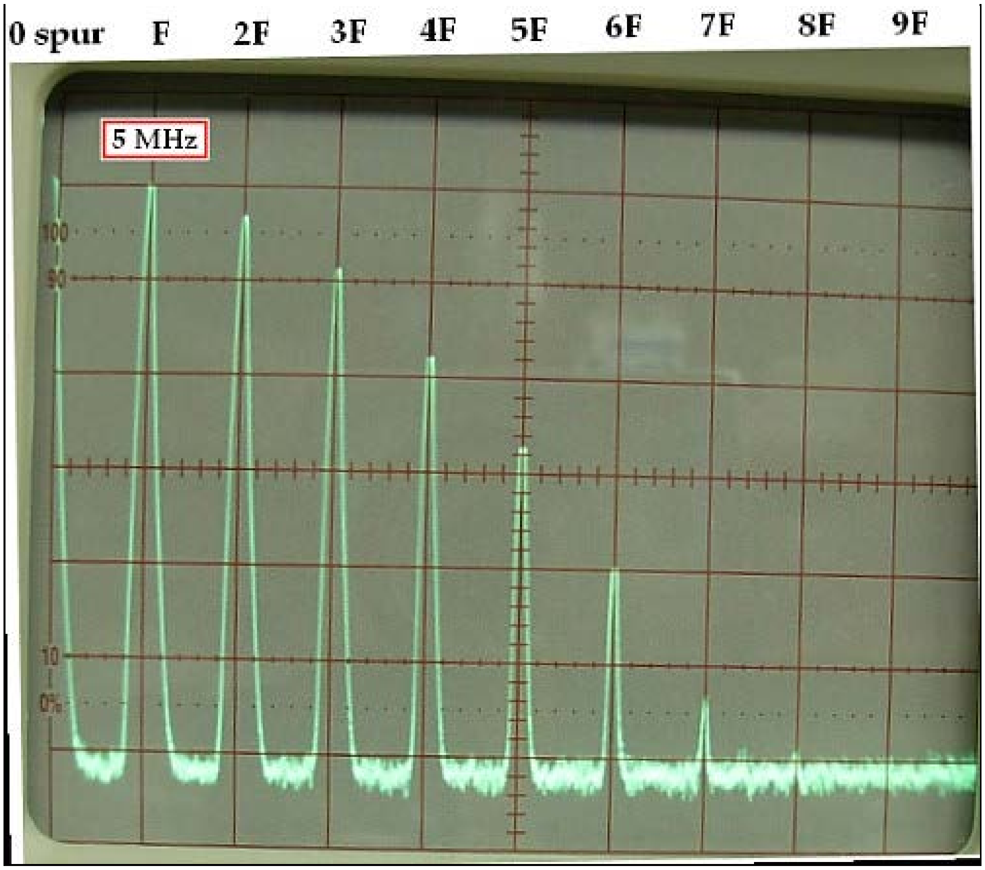
Spectrum Analyzer Calibrator



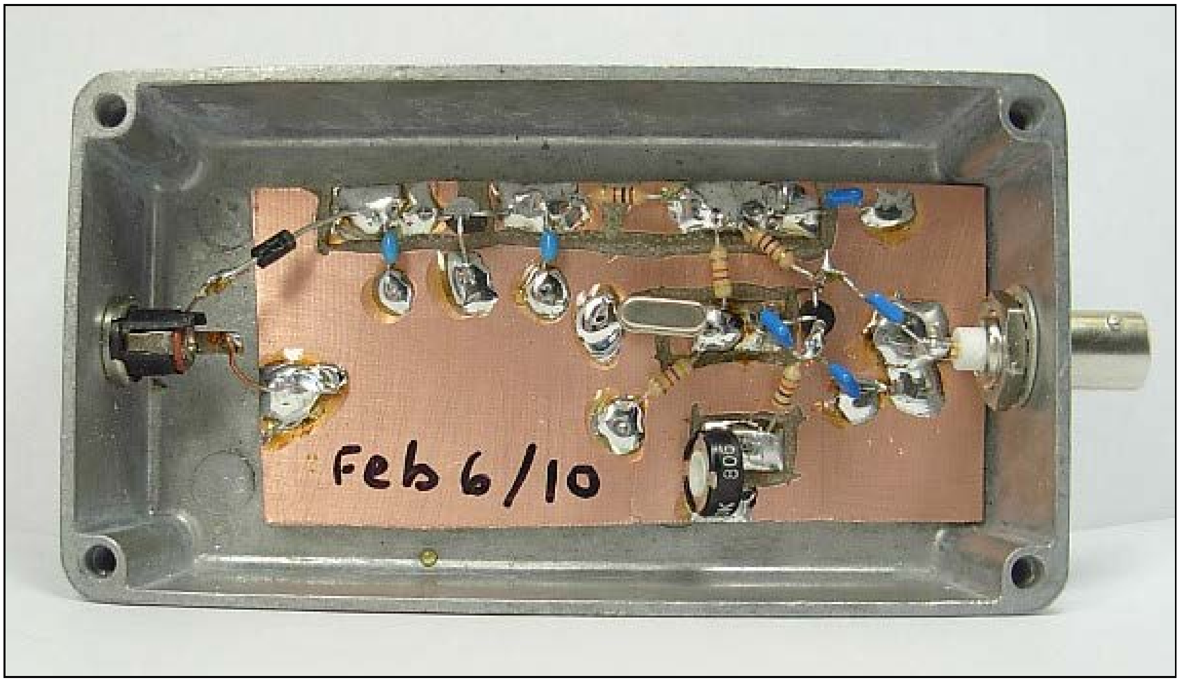
A harmonic rich, spectrum analyzer calibrator designed by Wes, W7ZOI and displayed with his permission. Adjust the 10K potentiometer to provide the output power needed to calibrate your spectrum analyzer. I set mine to -27 dBm. Be careful when connecting signal generators to your spectrum analyzer, since a higher than rated input power may destroy the mixer/front-end of your spectrum analyzer and cost you dearly.



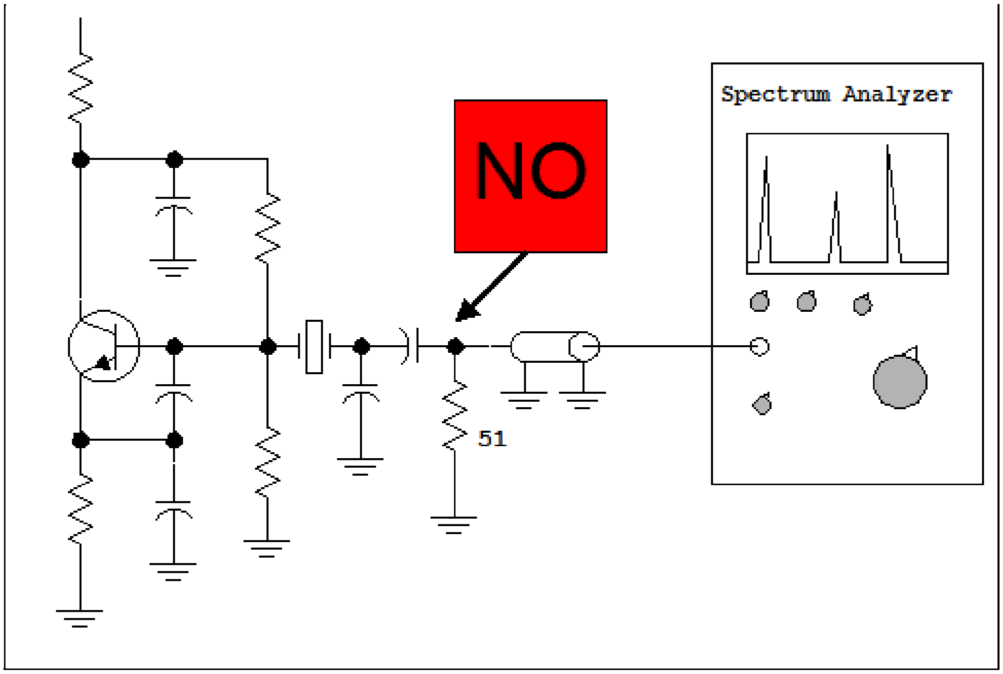
I used this filter to set the -27 dBm power needed to calibrate my spectrum analyzer.



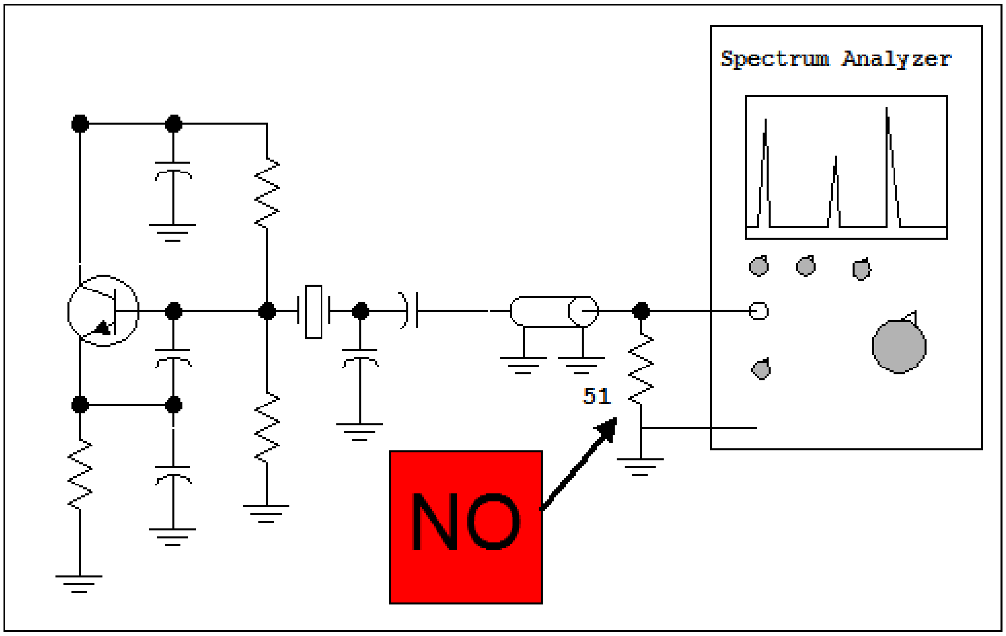
Spectrum analysis of the 5 MHz spectrum analyzer calibrator.



Breadboard of the 5 MHz spectrum analyzer calibrator.



Don't use a "50 ohm" termination when measuring with a 50 Ω impedance spectrum analyzer.



No resistor is required, as the the input impedance of the SA is 50 Ω.

Miscellaneous Photos and Notes

Some of the 50 Ω modules built during the RF Workbench page 1 and 2 experiments


1 3 6 9 30 dB
10 WATT LOAD


15542
HAT-6+
6 dB ATTENUATOR
50 OHM
15542
VAT – 6
6 dB ATTENUATOR

Amateur Radio Electronic Design

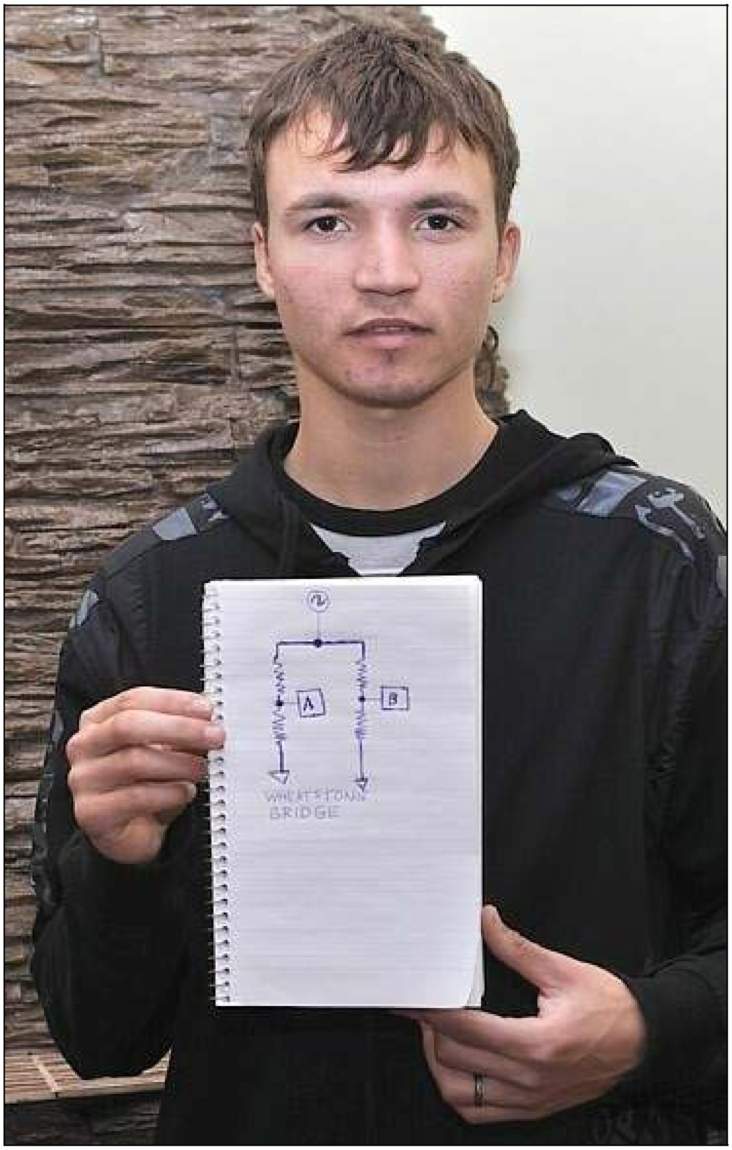
Audio Transistor Input Impedance Experiments

Introduction

I examined audio transistor amplifier input impedance during Spring — Summer 2010 and generated enough content for a web page.

On this web page, I explore determining AF amplifier input impedance by using network theory and calculation, plus direct measurement with instruments containing a Wheatstone bridge. This content emphasizes learning through performing bench experiments and I hope it sparks your own experiments and research into impedance measurement test equipment and theory.

Many RF circuits require termination with stages containing a well defined input impedance. Consider, for example, amplifiers that follow L-C low-pass filters, diode ring mixers or crystal filters — a known impedance (usually 50 ohms) must terminate these stages to optimize return loss. A special case is the diode ring product detector; which must be followed by a 50 ohm input impedance audio amplifier. How do we design or assess a small-signal audio amplifier that has a 50 ohm input impedance? This question spawned every experiment on this web page — the content grew and evolved along with my understanding of this topic.



Audio transistor input impedance may be calculated with equations or software, however, doing the math or using or affording these programs might be problematic for some amateur builders. Additionally, component variances such as transistor Beta and different power supply voltages can causes significant differences between the theoretical and the actual input impedance realized. Further, amplifiers, such as a feedback pair that involve combinations of series or shunt feedback can be difficult to analyze accurately using equations during small or large signal analysis. It may be easier for experimenters to just measure and tweak amplifier components on the bench — the focus of this web page!

As an rank amateur, I have much to learn, and by no means am an expert in electronic design. If you see an error on this web site, disagree with my analysis, or have suggestions for improvement, please email them — I am an amateur hobbyist, who earns no money from this site, and who relies on the assistance of others to keep the content as accurate and vibrant as possible.

The topics:

Part 1: Some basic transistor network theory and how to calculate input Z

- Part 2: 50 ohm input impedance Wheatstone bridge measurement
- Part 3: Measuring unknown impedances
- Part 4: Miscellaneous circuits, scans and photographs.

Notes:

Small signal analysis refers to modeling or examining an amplifier at a single operating point (its bias point) and applying linear equations which assess the amplifier with no signal applied. In small signal analysis, we assume that the signal is so small that transistor gain, capacitance and other factors are static.

Part 1: Some Basic Transistor Network Theory and How to Calculate Input Z

Some basic network theory plus methods to calculate the input impedance of common emitter and common base audio amplifiers.

Three basic small-signal transistor parameters include *beta,* emitter resistance *Re*, and bulk resistance *REB*.

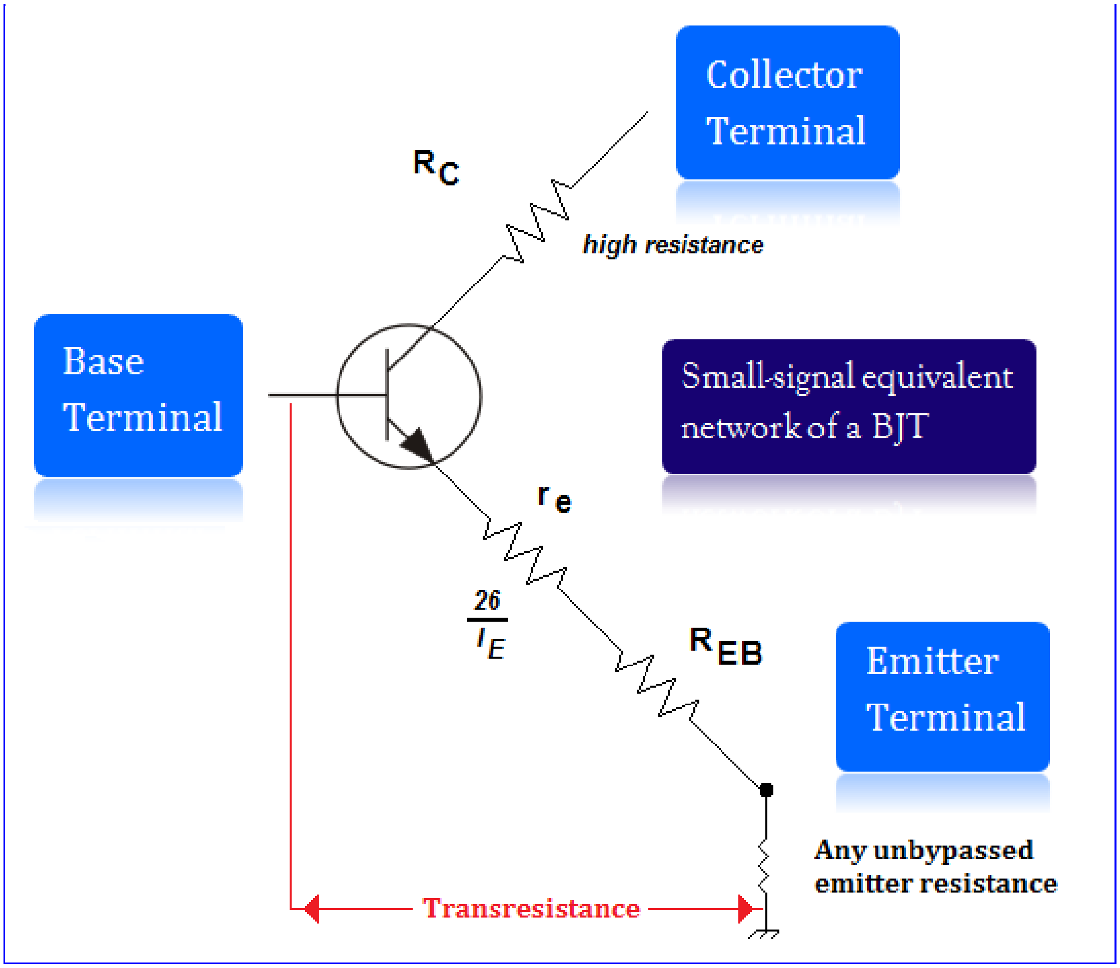
Beta is the term used to designate the current gain of a common emitter circuit — it's the ratio of collector (output) current to base (input) current.

Small-signal emitter resistance; Re = 26 / IE ,or, 26 divided by the emitter current in millamperes. For example, an emitter bias current of 0.52 mA gives a small-signal emitter resistance of 50 ohms, or visa versa . Re is the resistance seen looking into the emitter whether the stage input is the transistor base or emitter terminal. Re is the dynamic resistance of the input junction due to carrier action.

REB represents the bulk resistance of the semiconductor not arising from contact resistance; in other words, it's the DC resistance of the base and emitter leads plus the pn junction. Typically REB = 2 to 6 ohms and is often ignored (your choice) when the current is low — say, for example, < 9 mA for a typical common-emitter voltage amplifier. In large power transistors or for switching operations, the typical REB value may vary. REB, in part, limits the maximal gain of a transistor.

The constant 26 used when calculating the dynamic resistance of a forward-biased PN junction is derived via calculus. Professor Kuhn's website link containing the math.

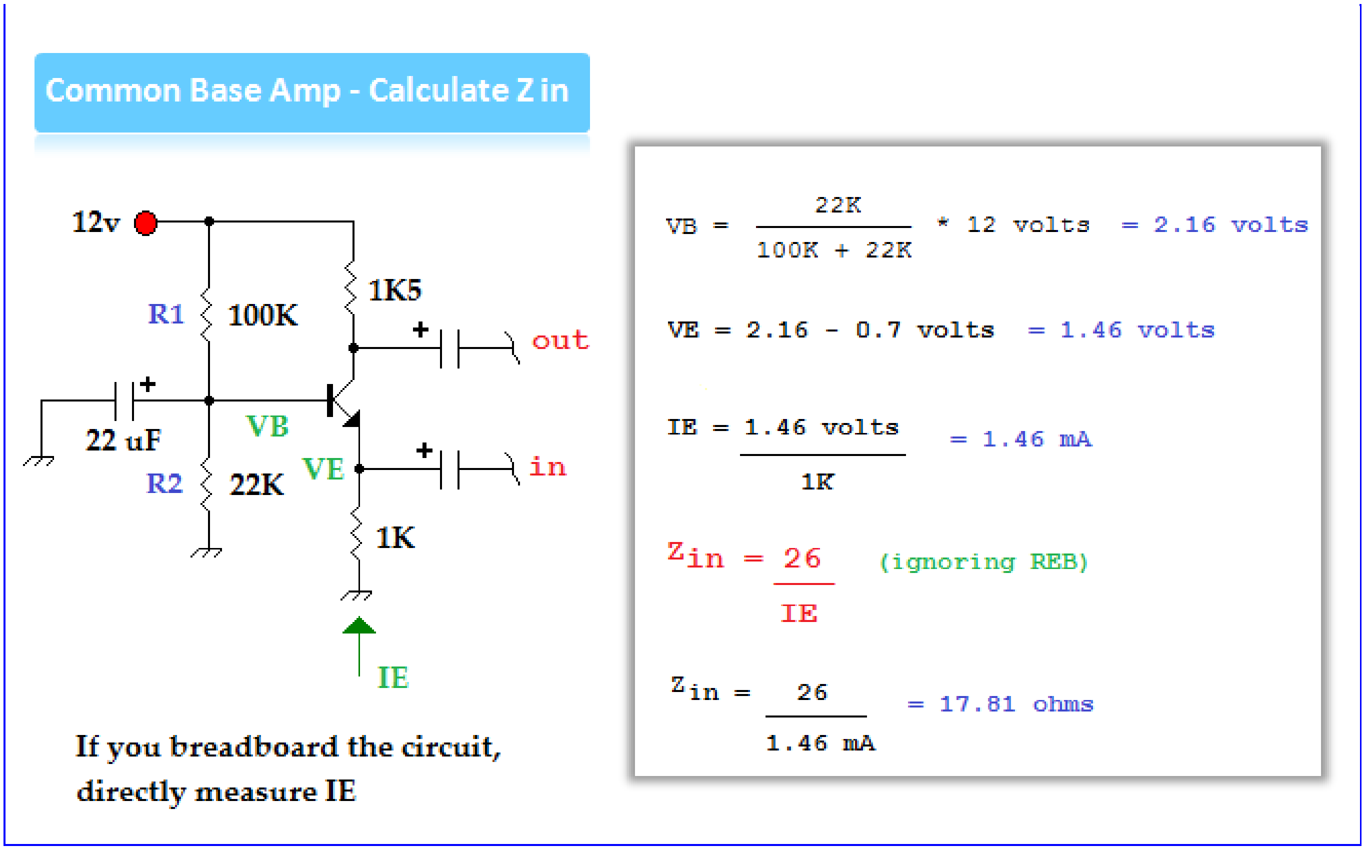
There is also a base spreading resistance generally known as 'rbb' that, in effect exists laterally across the transistor. A simple model puts rbb at about 100 ohms in series with the base and it's one of the causes of finite transistor frequency response. While interesting, rbb isn't discussed further.



Above — the small-signal equivalent network of any transistor. re = 26/IE. Also, re + REB + any unbypassed external resistor may be termed the *Transresistance*, a DC ohmic value representing the total resistance of the emitter. The collector resistance RC is high because of its reverse bias. Collector resistance is not considered when calculating input impedance of simple AF transistor stages.

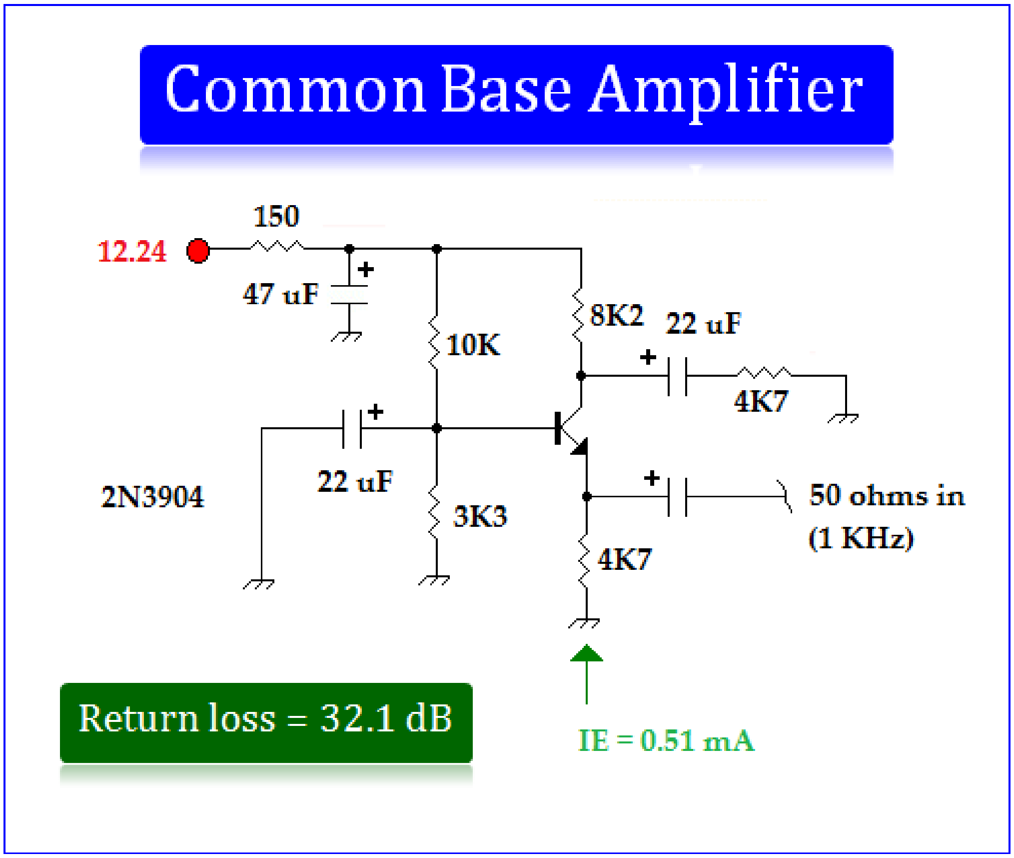
Calculating the input resistance of a common base stage

Calculating the input impedance of a common base amplifier is easy. Input impedance = 26/ emitter current (IE). You can either bench measure or calculate the emitter current using DC analysis. Click for the formula to calculate emitter current . A complete example follows:



Above — An example common base amplifier and its input impedance calculation. In this example, emitter current is calculated using DC analysis. On the bench, it's better to un-ground the 1K emitter resistor and connect your ammeter between this resistor and ground to directly measure IE. REB was ignored and = 0.

Consider the 50 ohm input Z common base amplifier we often use after a diode ring product detector plus diplexer:



Above — A common base amplifier built for a direct conversion receiver in Spring 2010. This amplifier is shown in test setup for bench analysis

— with a DC decoupling network and an AC coupled 4K7 resistive load. The emitter current established using 5% tolerance resistors was 0.51 mA. Therefore, the calculated input Z is 26/0.51 = 51 ohms. The return loss of this amplifier as measured with the *active* 50 ohm Wheatstone bridge device described later on this web page was a spectacular 32.1 dB! If a different power supply voltage or biasing/emitter resistors were used, the IE would change and along with IE, the input impedance and return loss.

This amp illustrates that testing and tweaking AF amplifiers on the bench will garner the best results. If I just copy someone else's design; perhaps with a different DC voltage, or decoupling network and don't adjust the emitter current by tweaking the base biasing or emitter resistor resistors, the input impedance could differ significantly. Whenever you build a common base amplifier, measure its DC current and as necessary, tweak resistors to get the current needed for a perfect impedance match. It is good practice to measure all the DC voltages and emitter current on any amplifier you build — you will learn what is normal, what to expect and perhaps detect errors or parts failure(s).

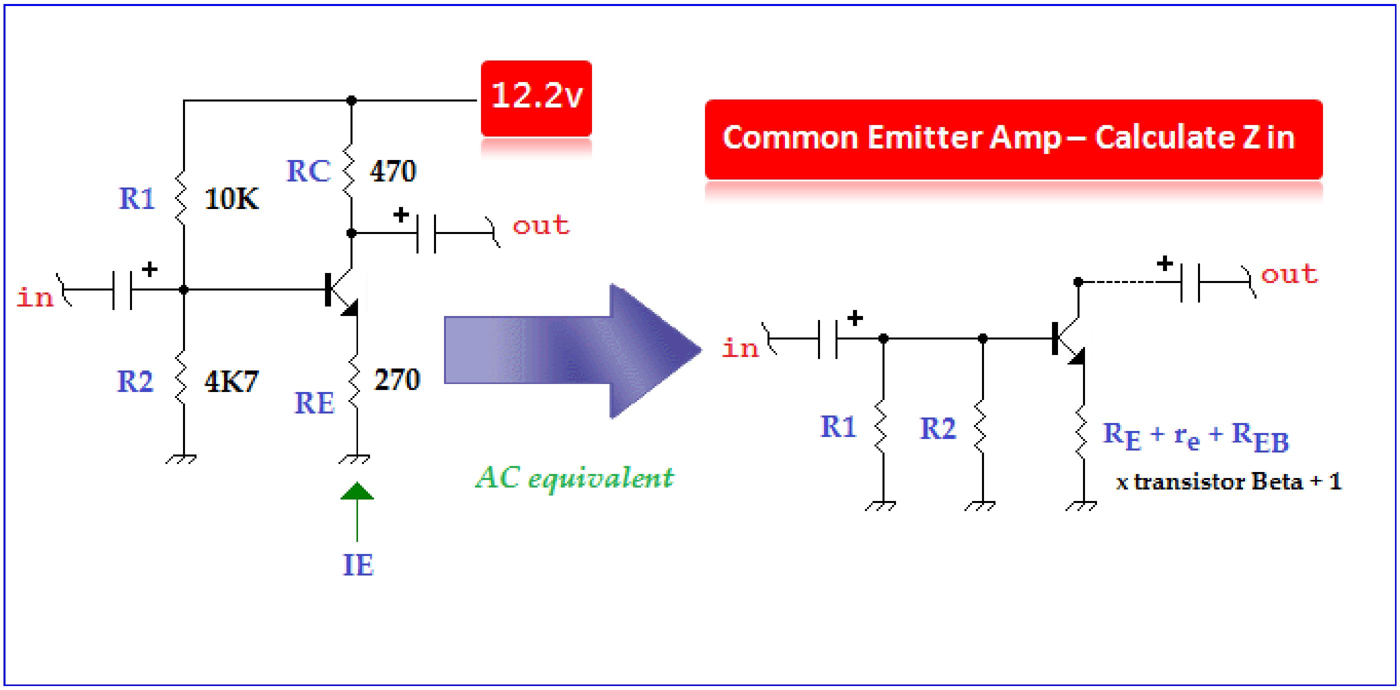
Performing return loss measurement is also a fantastic way to ensure good matching to the 50 ohm impedance diode ring product detector that feeds this amplifier.

Calculating the input resistance of a common emitter stage

Calculating the input impedance of a common emitter amplifier is also straight forward, but not as easy as the common base amplifier.

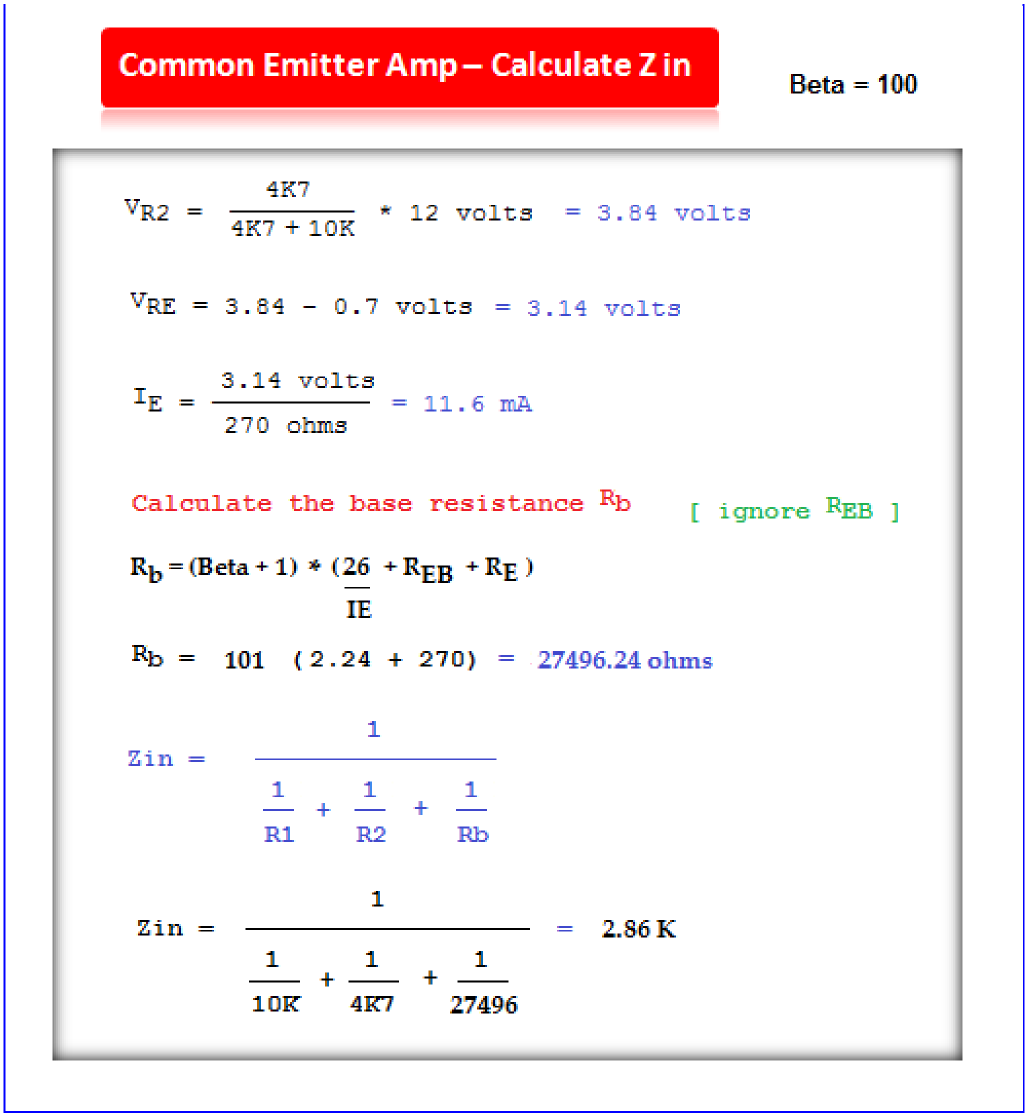
In the **common base amplifier**, the emitter is the input element, therefore the input signal resistance is 26 / IE + REB. Often we ignore REB. If current of the common base amplifier is for example, 2 mA or so, then the 2-6 ohms of REB may be significant as 26 / 2 mA = 13 ohms. REB may be a factor because 2-6 ohms is a significant percentage of the total input resistance.

For a **common emitter amplifier**, the input resistance looking into the base is Beta (26/IE + REB + RE). Again REB is often ignored. We need to include any transistor DC biasing resistors which are also seen by the input signal as it moves through the transistor base. An example follows:



Above — An example common emitter amplifier re-drawn to illustrate how the input resistances combine to provide the AC input impedance. In this case, the 270 ohm emitter resistor RE is un-bypassed. R1, R2 and the components RE, re and REB are in parallel as the DC supply acts as a short to ground for the AC input signal. The components RE, re and REB (if used) must be multiplied by the transistor Beta value (+ 1) since the resistance looking into the base is Beta times that looking into the emitter.

Therefore: Rin = (B+1)*(re + REB + RE'). Normally we ignore REB so practically speaking Rin = (B+1)*(re + RE')



Common Emitter Amp – Calculate Z in

Beta = 100

$$V_{R2} = \frac{4K7}{4K7 + 10K} * 12 \text{ volts} = 3.84 \text{ volts}$$

$$V_{RE} = 3.84 - 0.7 \text{ volts} = 3.14 \text{ volts}$$

$$I_E = \frac{3.14 \text{ volts}}{270 \text{ ohms}} = 11.6 \text{ mA}$$

Calculate the base resistance R_b [ignore R_{EB}]

$$R_b = (\text{Beta} + 1) * \left(\frac{26}{IE} + R_{EB} + R_E\right)$$

$$R_b = 101 \; (2.24 + 270) = 27496.24 \text{ ohms}$$

$$Zin = \frac{1}{\frac{1}{R1} + \frac{1}{R2} + \frac{1}{Rb}}$$

$$Zin = \frac{1}{\frac{1}{10K} + \frac{1}{4K7} + \frac{1}{27496}} = 2.86 \text{ K}$$

Above — The math for the common emitter circuit shown directly above using DC analysis to calculate the current. On the bench, we just measure the emitter current (no need to calculate it). We assume IE = IC for a common emitter amplifier. REB = 0 (when ignored). If the 270 ohm resistor RE was bypassed with an electrolytic capacitor, the 270 ohm resistance would also = 0; and then Rb = Beta + 1 * (26/IE).

Conclusion

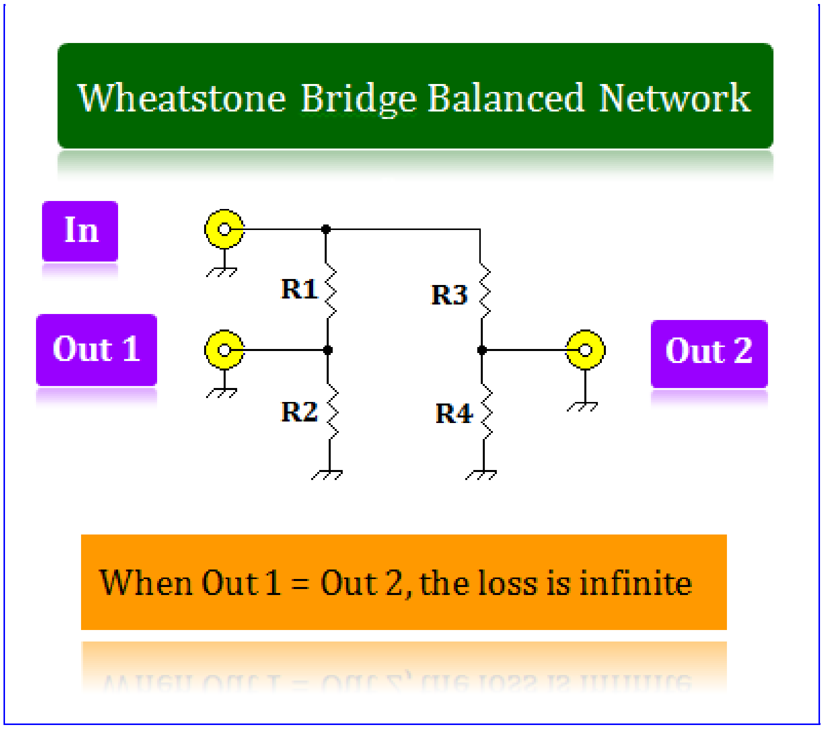
This theory explains how to calculate input impedance in 2 basic transistor AF amplifiers. Consult an electronics text for further explanation. Although the arithmetic is simple, quite frankly, it's a little boring. Let's go to the bench and have some fun. I was quite naive about measuring AF amplifier input impedance; however, my experiments yielded some knowledge and a strong appreciation for the Wheatstone bridge network. Onward...

Part 2: 50 ohm Input Impedance Wheatstone Bridge Measurement

Testing for a NULL or measuring the return loss of AF amplifiers with a 50 ohm input impedance.

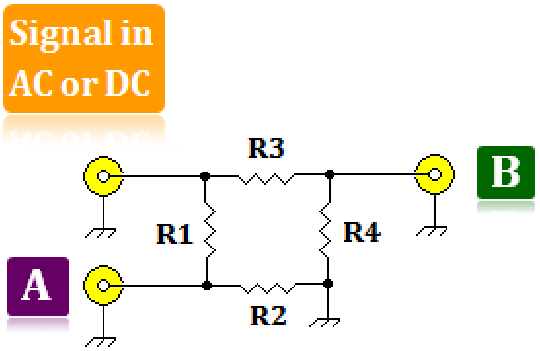
Refer to the diagram on the right. Redrawn in a way more familiar to builders, the Wheatstone bridge network is just a pair of voltage dividers in parallel. We measure the difference in AC voltage between the ports labeled Out 1 and Out 2. The bridge is said to be balanced and produce NULL or 0 output when Out 1 and Out 2 are equal in voltage. Another description — when in perfect balance, the signal loss due to mismatch between the output ports is infinite. However, if this balance is disturbed by a mismatch between ports Out 1 and Out 2, an AC voltage appears and the return loss decreases in proportion to the mismatch (within limits and providing your instrument can measure accurately).

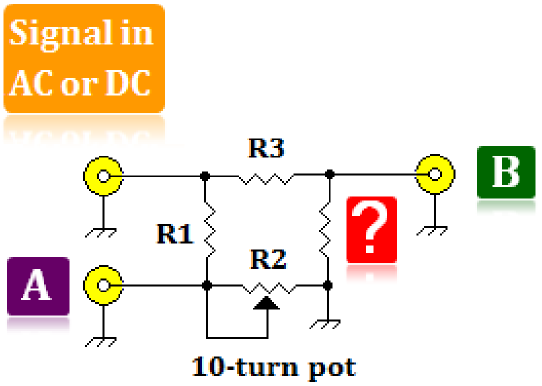
Let's focus on some practical bench applications in the new millennia: On your bench, you might employ a Wheatstone bridge network to measure return loss (or VSWR) or to simply to detect a NULL indicating a close impedance match between 2 stages. Specific examples include tuning your feed line and antenna, checking the match between a signal generator and a filter, or measuring an audio amplifier input impedance. In my estimation, the Wheatstone bridge lies among the most important test circuits in the amateur designer's arsenal; worthy of study and experimentation.

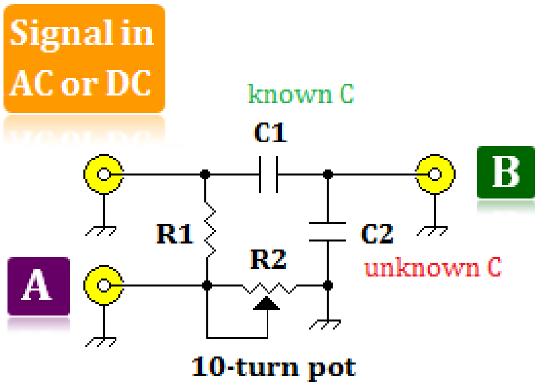


Notes:

- The input signal can be AC or DC, but all discussion is confined to an AC signal source
- E96 (1%) metal film resistors were used in all Wheatstone bridges
- All bridges were tested at 1 KHz
- Ensure you do not overdrive your bridge; lest distortion occur! When a bridge is overdriven, you might cancel the fundamental frequency when balancing the bridge, but not the harmonics! Therefore, parasitic harmonics appear in the output that skew the the NULL or return loss values. I learned low pass filtering the amplified bridge output is really important.
- Any distortion in the bridge output means you must reduce your input signal drive level; however, this may reduce the accuracy of the RL measurements. There is no free lunch! You generally want just enough input signal to accurately measure the signal with the bridge at NULL.







Wheatstone Bridge

Comments from the Workbench

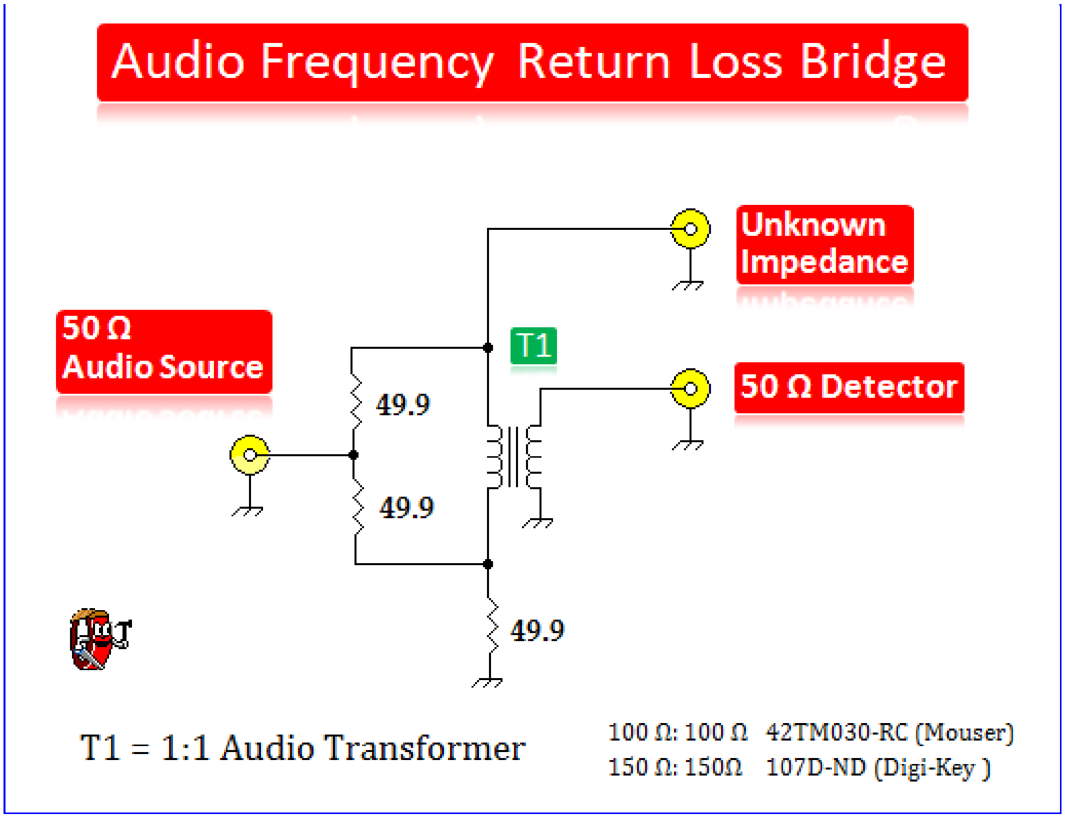
Above — an evolution of the 4 resistor bridge into a device to measure impedance and capacitance. In its classic form, each bridge leg is a resistor voltage divider with a detector connected across ports A and B. If ports A and B have equal voltages and R1 = R2, then R3 and R4 must also be equal; the bridge lies balanced or in a NULL state. If you remove R4 and measure an unknown resistance, the bridge will return to balance after adjusting pot R2 to equal the unknown resistance. In most cases, R2 is calibrated and the impedance is read directly off the potentiometer dial. Bridges can be arranged to measure unknown capacitance, inductance , frequency and other parameters by using precision 1% fixed components, calibrating the 10 turn pot to indicate the desired parameter, or for deriving the unknown value via equations.

Builders of lore used bridges to quantify many values on the bench. Although we have better ways to measure inductance and capacitance today, the Wheatstone bridge is still the *king* when it comes to simple measurement of network impedances; for example, QRP antenna tuners. Some builders use an LED to indicate bridge imbalance.

Building a passive instrument to measure the return loss of a 50 ohm input Z audio preamplifier.

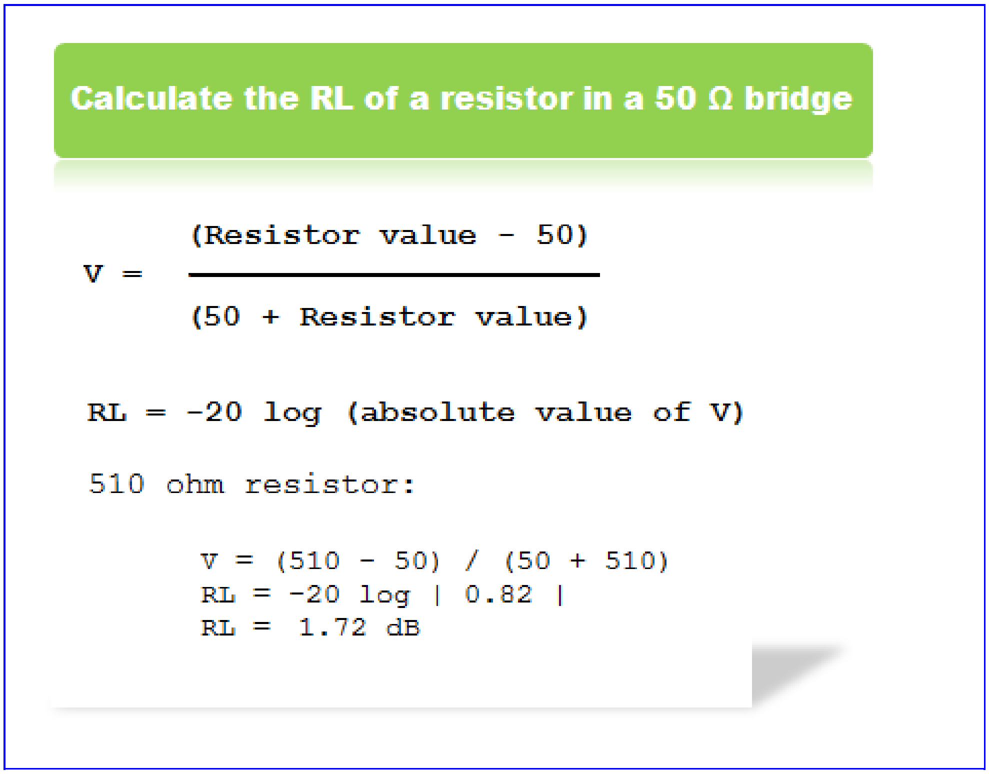
Non-radio folks don't generally understand this — to properly terminate a diode ring mixer, 50 Ω impedance stages are needed. The inspiration driving *all* the experiments on this web page was to design a 50 ohm input impedance common emitter audio preamplifier to follow a diode ring mixer. I could have just used the familiar common-base amplifier popularized by Roy, W7EL, but of course, wouldn't learn anything. Somehow, I became drawn in by curiosity and generated enough content to fill a whole web page.

I decided to try and build some return loss bridges and test them by using known, fixed-value resistors as the unknown impedance. My first bridge, was an AF version of this RF return loss bridge using a junk box 600 ohm, 1:1 audio isolation transformer. It didn't work until I rearranged it as shown in the schematic below.



Above — A simple return loss bridge using an AF transformer and 50 ohm detector. Suitable detectors are described here in the section covering return loss bridges — I used a 50 ohm terminated scope. Using 20 log (peak-peak voltage) to crunch the 50 Ω AC voltage into dB, the bridge was measured at open circuit, plus with various fixed 5% tolerance resistors terminating the Unknown Z port. Using a junk box 600 ohm 1:1 AF transformer, my results initially seemed good, but upon analysis were fraught with error. Note the suggested transformers in the schematic.

Above — The very first AF return loss bridge built. Anchored to the ground plane with resistors, the transformer was a 600 ohm *junk box* special. Although I was able to achieve a deep NULL using a 49.9 ohm resistor, the return loss was 86 dB; not possible. Additionally, other fixed resistors gave return loss values more than 4-5 dB away from the proper value. Likely, my junk box transformer lacked sufficient inductance for 1 KHz. For testing your bridge, use a formula to inform you of what RL value to expect for a given fixed resistor.



Calculate the RL of a resistor in a 50 Ω bridge

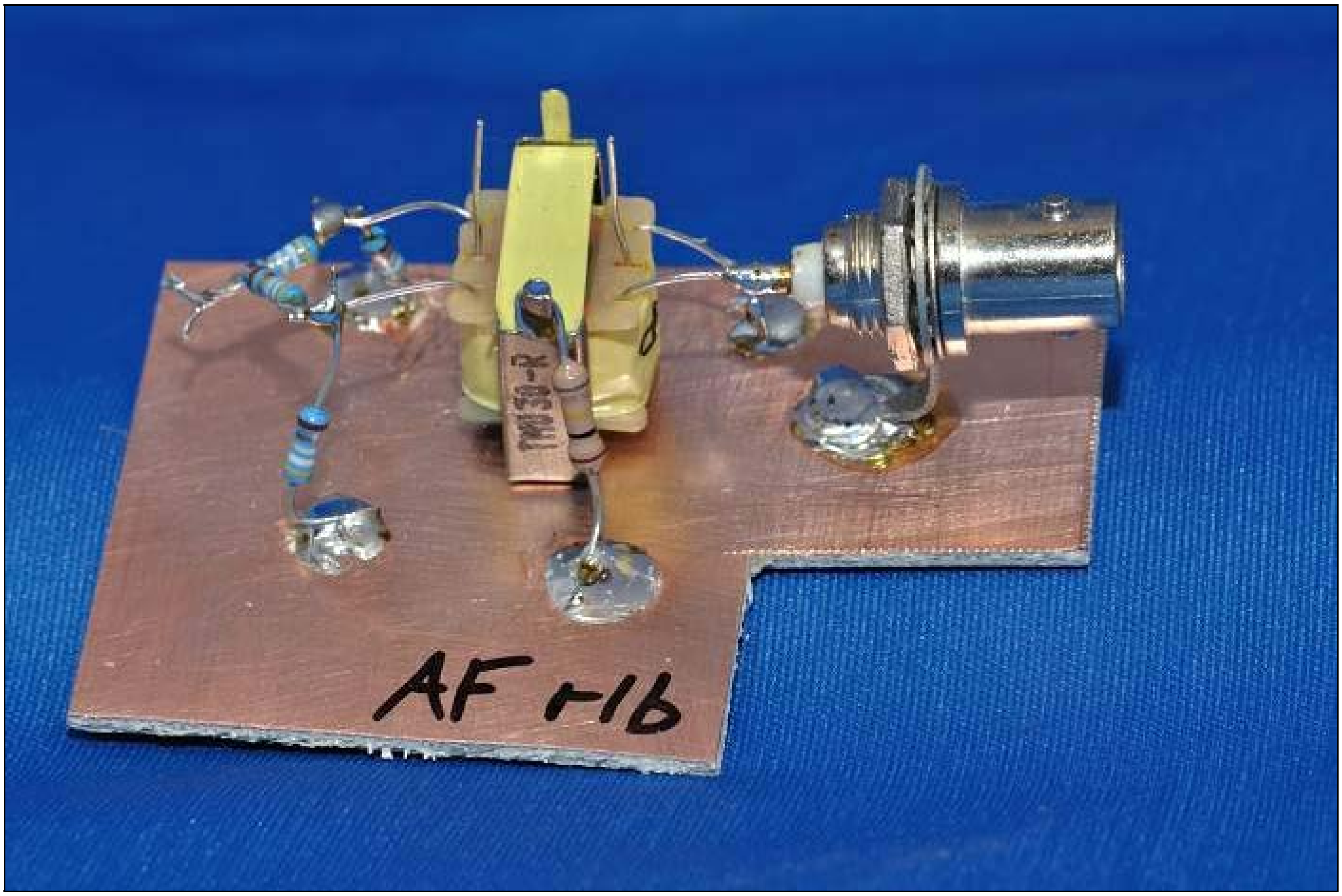
$$V = \frac{(\text{Resistor value} - 50)}{(50 + \text{Resistor value})}$$

RL = -20 log (absolute value of V)

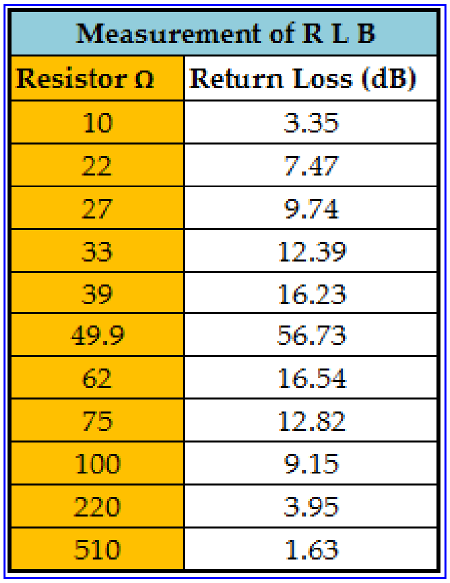
510 ohm resistor:

```
V = (510 - 50) / (50 + 510)
RL = -20 log | 0.82 |
RL =  1.72 dB
```

Above — The formula to calculate the expected return loss for a fixed resistor placed in the Unknown Impedance port on a Wheatstone bridge. Click for a table of Return Loss values for some non 50 ohm resistors. Your RL values, will rarely be exact, but should be close to the predicted value. A well functioning bridge should yield a return loss of > 40 dB using a 51 to 47 ohm resistor as the Unknown Impedance.



Above — A second bridge was built after obtaining a 100 Ω : 100 Ω AF transformer from Mouser Electronics. This transformer was ideal (each coil has ~ 1H in inductance!). Bench testing indicated good function. My results are tabled below:



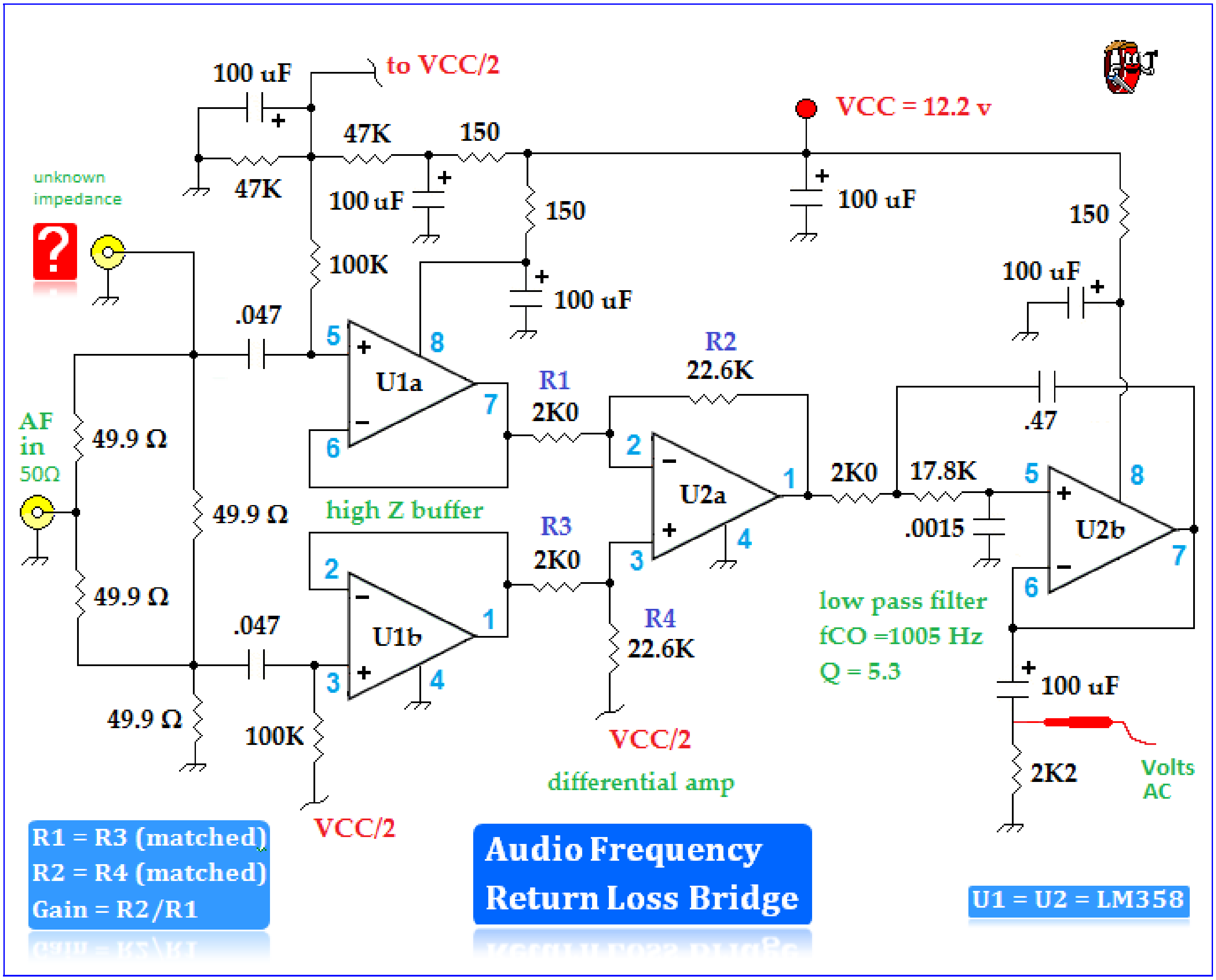
Measurement of R L B	
Resistor Ω	Return Loss (dB)
10	3.35
22	7.47
27	9.74
33	12.39
39	16.23
49.9	56.73
62	16.54
75	12.82
100	9.15
220	3.95
510	1.63

Above — A table of the above 50 ohm Wheatstone bridge return loss measurements. These results are acceptable. The NULL with a 49.9 ohm resistor was incredibly sharp and garnered a RL of 56.73 dB. My AF source was a low noise 1 KHz, 50 ohm output impedance signal generator.

If you do not need return loss, and only require a NULL to indicate a match, a common 600 ohm transformer may work okay for you.

Building an active instrument to measure the return loss of a 50 ohm input Z audio preamplifier.

The results of my early experiments with a passive bridge were encouraging. Noting that most builders would have difficulty obtaining a 100 Ω : 100 Ω AF transformer, a version using op-amps was sought. My first 3 designs did not work properly and I became discouraged. Some guidance from Wes, W7ZOI allowed me to problem solve and experience success.



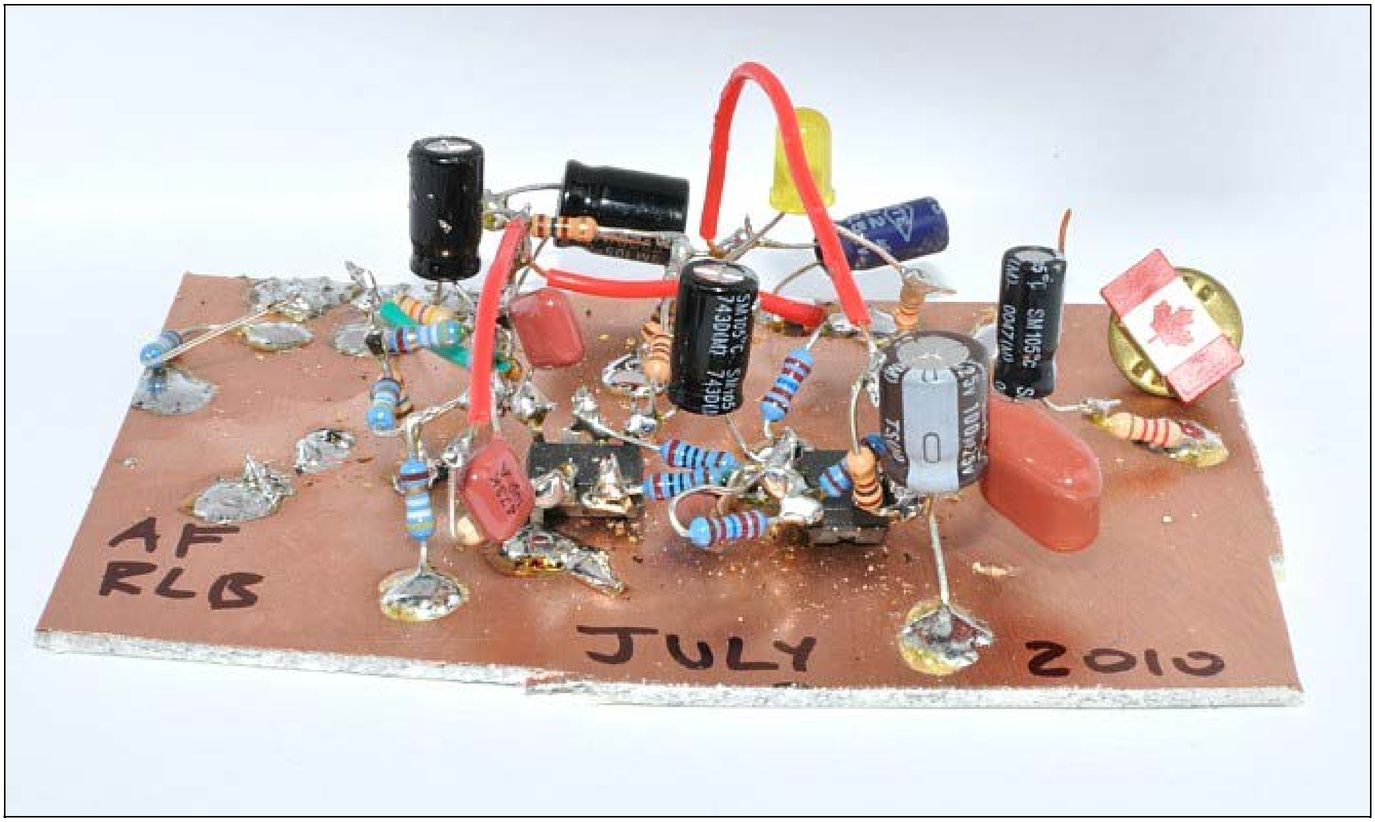
Above — Schematic of my active Wheatstone bridge, amplifier and low-pass filter for measuring the Return Loss of 50 ohm input impedance AF amplifiers. I built 3 copies of the above device; best results occurred when careful layout and planning were employed. Optimal performance occurred when encased in a metal box.

The bridge was built from 1% metal film resistors. 0.047 polyester film capacitors lightly couple the bridge to high impedance op-amp buffers labeled U1a + U1b. My experiments informed me that to minimize loading on the bridge is important. The LM358 is an excellent op-amp choice, but almost any other op-amp could be employed successfully. U2a is the differential amplifier and matching R1 + R3 and R2 + R4 with 1% tolerance resistors is critical; 5% resistors did not work well. The gain is non-critical — feel free to choose reasonable resistor ratios based upon the resistors you have in stock. The differential amp promotes the unfortunate side effect of amplifying both the desired AF source plus any common mode signals. Although common mode suppression is an important consideration when designing instrumentation amps, fortunately, performance is fine. A amplifier topology using a differential amp across the bridge was trialed, but functioned identically to the simpler differential amp shown. Consult a textbook for more information on Instrumentation amps. Much information was gleaned from Professor Ken Kuhn's web site.

The output is low-pass filtered by a single stage Sallen-Key low-pass filter with a peak frequency of 1 kHz and a Q of 5. This filter gain at 5 at 1 kHz is 0.328 at 2 kHz and 0.123 at 3 kHz. Thus, the second harmonic is reduced by a factor of (5/0.328) = 15.2 and the third harmonic is reduced by a factor of 40.6. **Do not omit a low pass filter**. I chose a 1 KHz cutoff, but experimentation indicated a low-pass cut-off frequency as high as 10 KHz may work okay if you plan to use the bridge at frequencies other than 1 KHz.

Power supply decoupling proved important. When less DC low-pass filtering (less than the 150 Ω plus the 100 uF capacitors shown) was employed, some low frequency audio noise appeared in the output.

I measured using a X1 oscilloscope probe on the output of U2b.

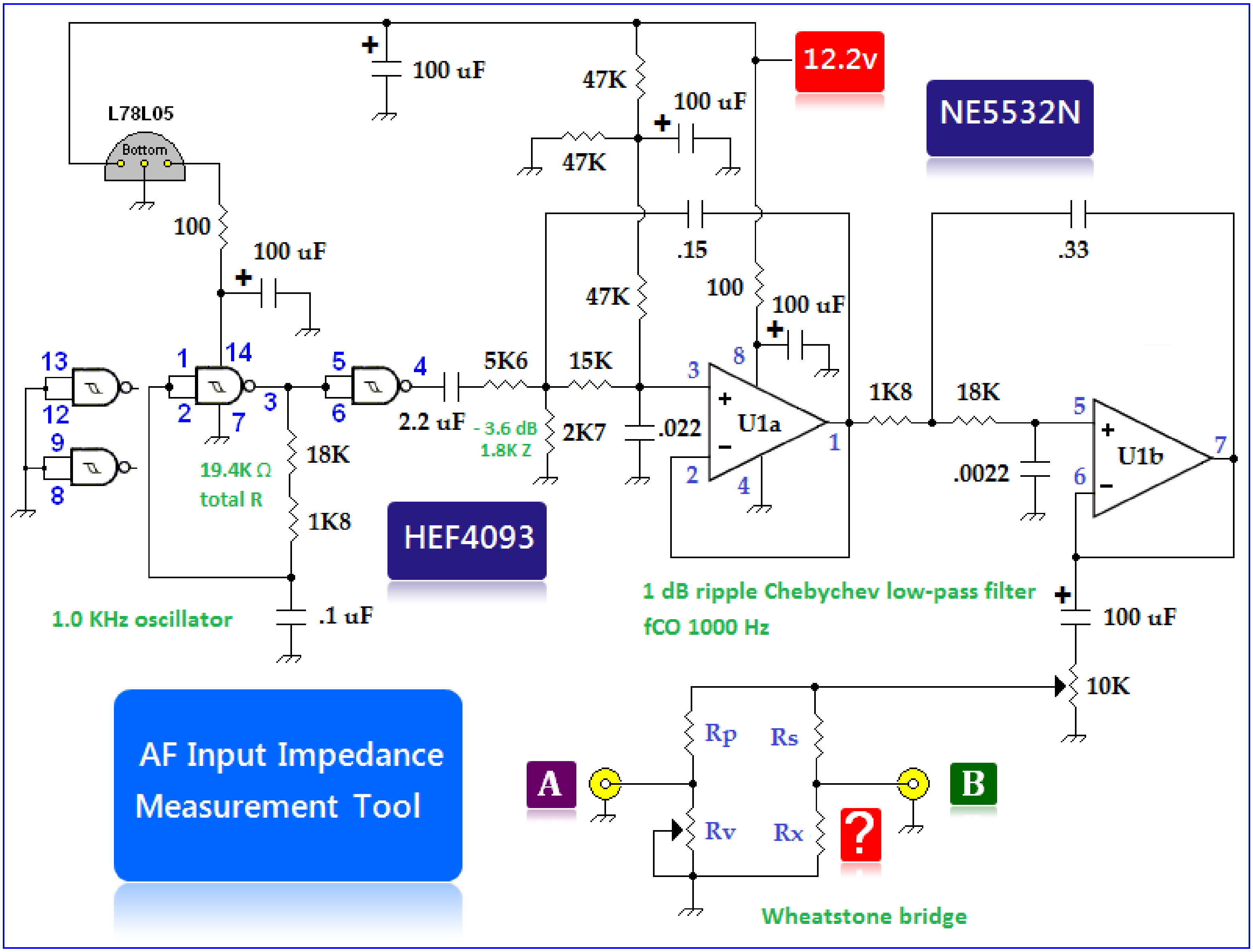


Above — A breadboard of the active Wheatstone bridge schematic located above. When tested with fixed value resistors, the RL @ 49.9 ohms was 55.4 dB and close to predicted value with other test resistors. This instrument will be put in a metal case and become a permanent part of my test equipment arsenal.

Part 3: Measuring Unknown Impedances

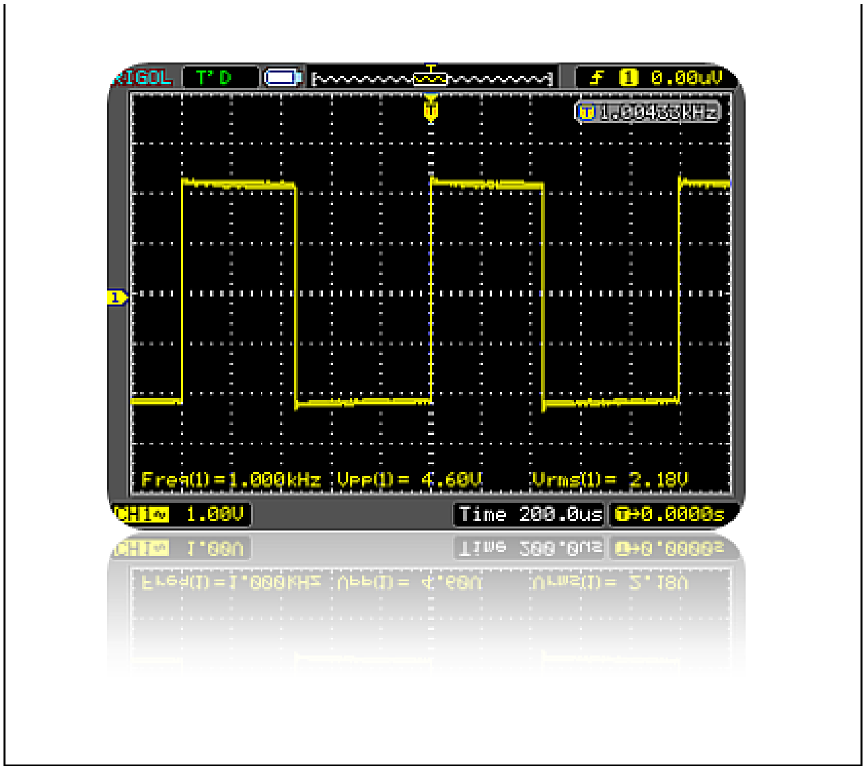
Building an instrument to measure the input impedance of an audio preamplifier using a NULL.

Above — One of several Wheatstone bridge circuits built in the Spring-Summer of 2010. In these bridges, the potentiometer was calibrated and the panel labeled using fixed resistances for calibration. One big challenge is range or resolution; dependent on the bridge resistor values and what impedance you are trying to measure. Greatest accuracy is associated with 5 or 10 turn potentiometers, but these are expensive. Often, I used standard, linear taper pots to save money during my experiments. Over 7 different bridges were built and tested. To save time, I didn't photograph many of my projects from the summer.

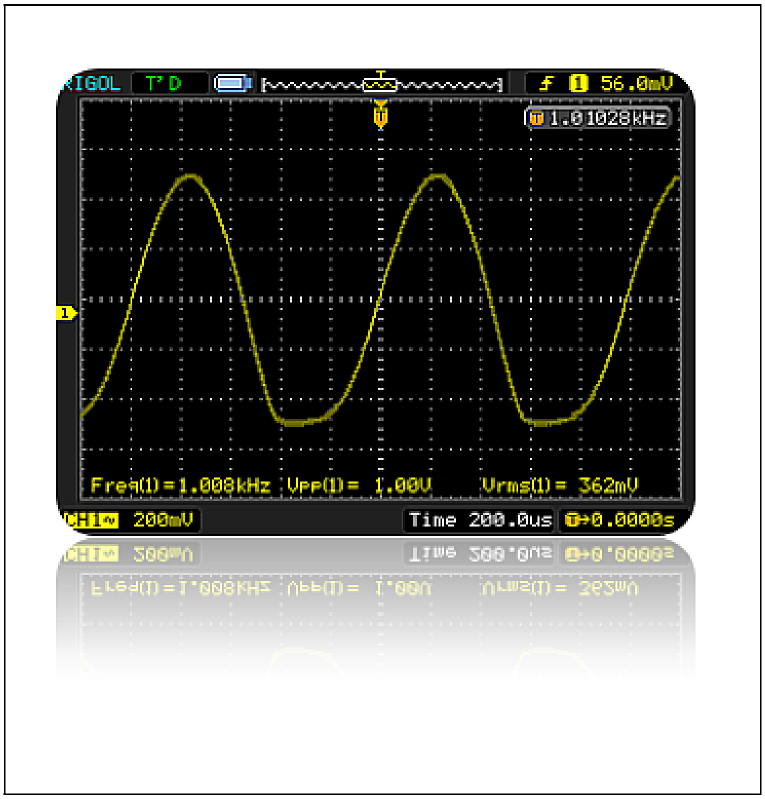


Above — A complete 1 KHz signal generator, low-pass filter and bridge circuit which became the prototype for most of my experiments in this section. Click for a high resolution photo of 1 of the breadboards during construction.

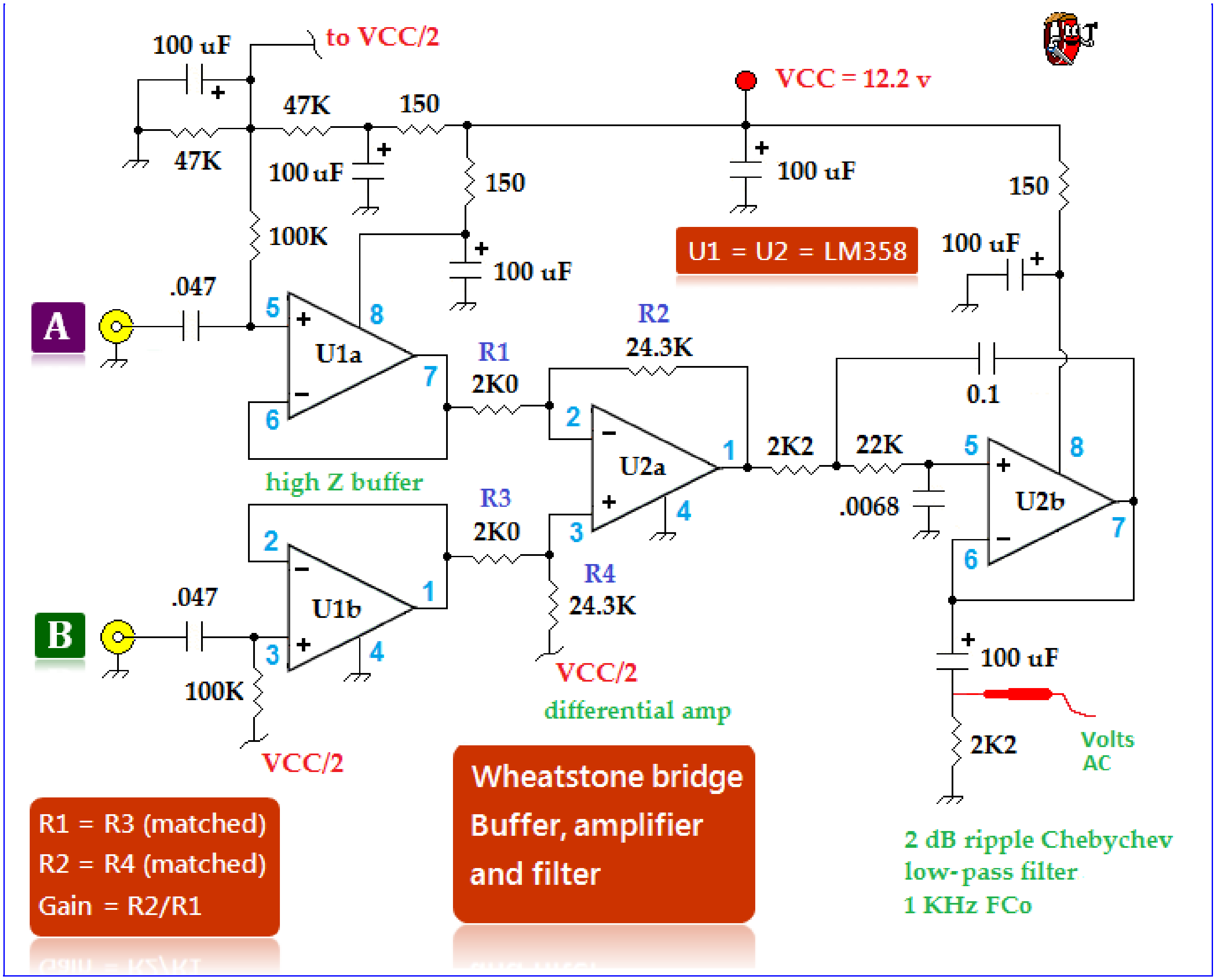
The 1 KHz signal generator is a digital oscillator built with 2 gates from a 4093. This excellent oscillator uses a single R + C network for tuning and requires a voltage regulator for frequency stability. The output signal is attenuated 3.6 dB and low-pass filtered by 4 poles of active filtering. A 10K pot controls the drive into the bridge circuit. The bridge outputs are labeled **A** and **B** and require buffering, amplification and low-pass filtering similar to the active bridge shown earlier. These functions and some comments on the bridge resistors come later.



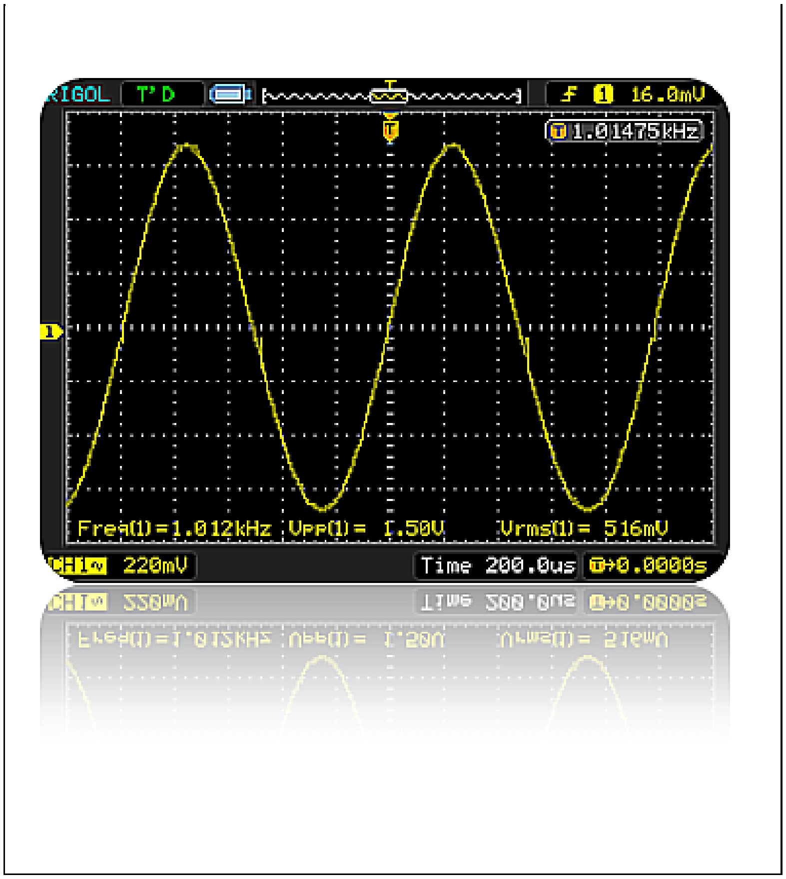
Above — The oscilloscope waveform from the digital 1 KHz oscillator. Digital clocks fascinate me and this was an untried design. Initially a CMOS 555 timer was considered, however, I own many 4000 series NAND Schmitt triggers and pressed 1 into service. Another good choice might be the 74HC132. The first NAND gate (inverter) contributes 180 degrees of phase shift, while the RC low-pass filter tank circuit digitally shifts the AC the other 180 degrees. Output noise is filtered by both the low-pass filter and the input Schmitt trigger dead band or hysteresis. The result is a fairly crisp square wave that may rival the 555.



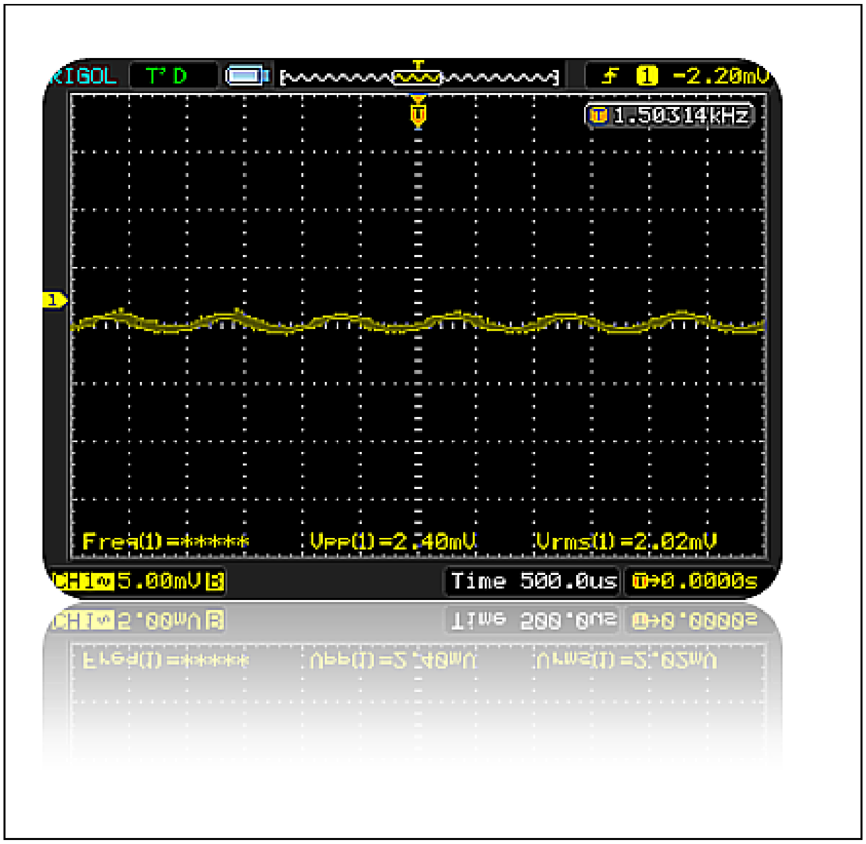
Above — The output of the 1 KHz low-pass filter. A sine wave is desirable, but not critical; suppression of energy in the range of 5-10 KHz, informed the filter design goal. At 8000 Hz, the attenuation is > 80 dB. All good.



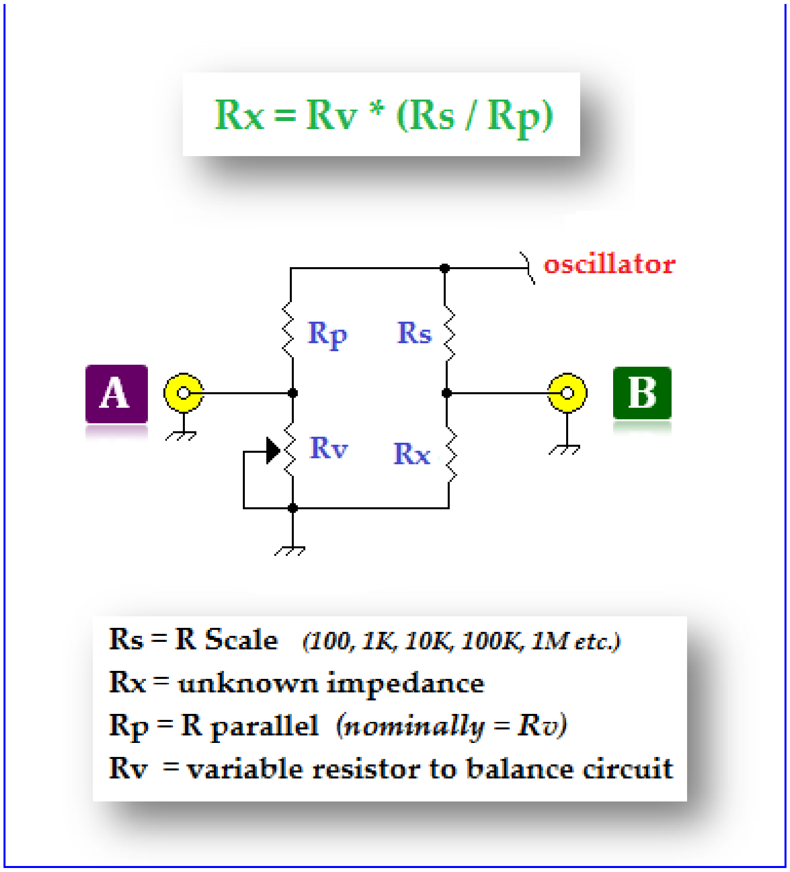
Above — The buffer, differential amplifier and low-pass filter employed during this series of experiments. Function is identical to the similar stage described earlier.



Above — The output of amplified and filtered Wheatstone bridge at open circuit (no resistance at the Unknown Impedance port).



Above — the output of the amplified and filtered Wheatstone bridge at NULL (potentiometer setting balanced to match the resistance at the Unknown Impedance port. Below 2 mV, accuracy is lost.



$$R_x = R_v * (R_s / R_p)$$

R_s = R Scale *(100, 1K, 10K, 100K, 1M etc.)*

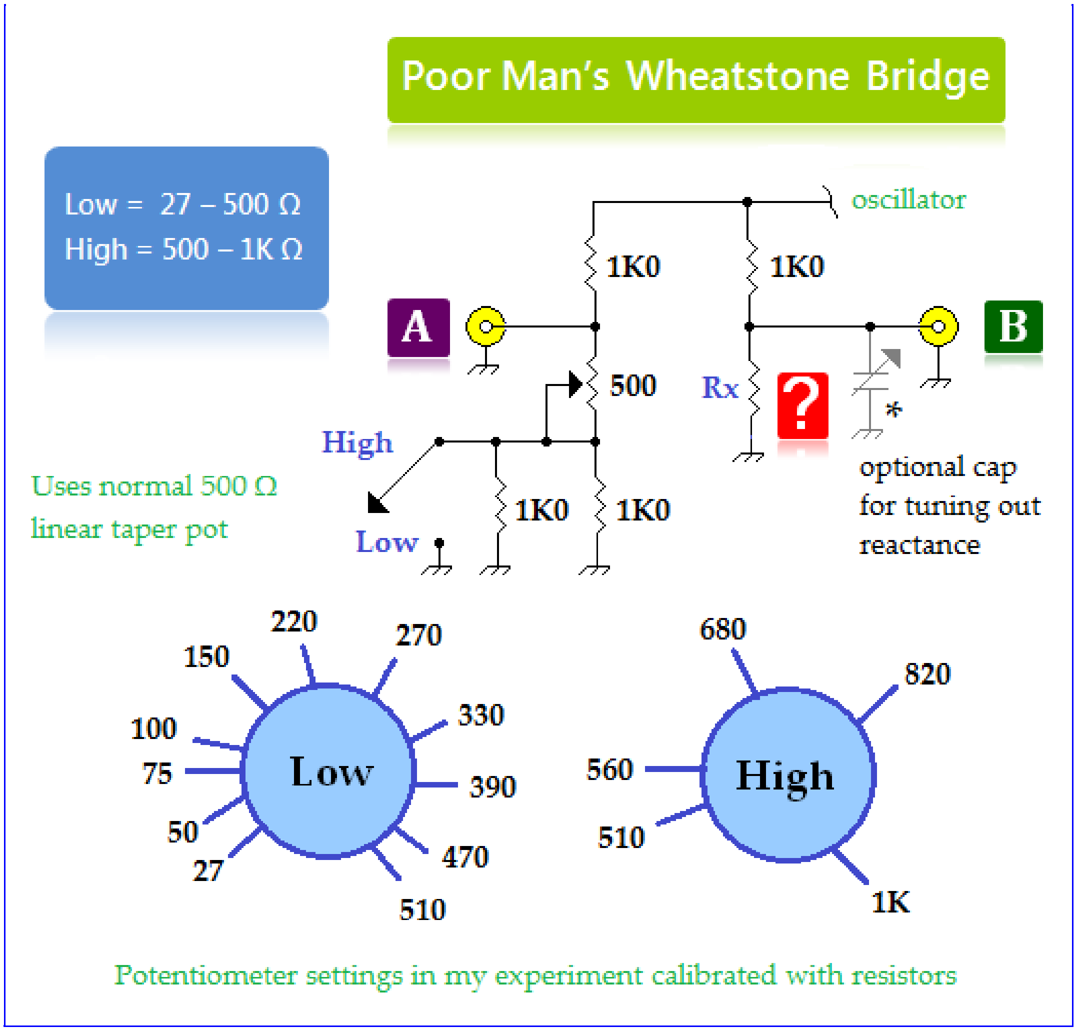
R_x = unknown impedance

R_p = R parallel *(nominally = R_v)*

R_v = variable resistor to balance circuit

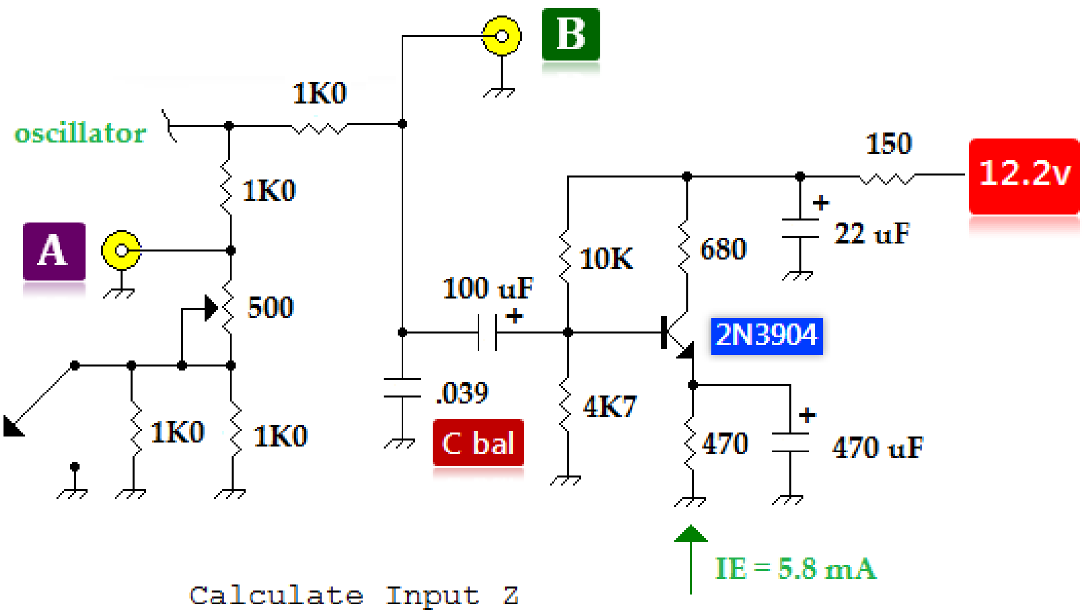
Above — The math behind the bridge. I found when R Variable (Rv) was the same resistance as the fixed resistor R parallel (Rp), reasonable resolution was possible. "Reasonable resolution" means your pot has a good range of rotation as you go from the lowest to highest measureable impedance. Generally, Rv has to be less than the maximal impedance you are trying to measure.

R Scale (Rs) can be switched in decades via a panel mount switch to cover a wide range of resistance with good resolution, or just be 1 or 2 values. It's your design call.



Above — My poor man's impedance measurement device that uses a common 500 ohm linear taper pot as the balancing resistor. In order to get good pot resolution, the desired range is switched. This bridge measures impedances at the Unknown Port from about 27 ohms up to 1K with decent resolution. The blue circles depict how I calibrated the front panel of my device using 2 colors. This device had an average return loss of 32.5 dB when a NULL was obtained.

Measuring resistors to calibrate a bridge is quite different from real-world measurement of reactive AF amplifier loads — if the unknown resistance has a large inductive or capacitive reactance, obtaining good bridge balance might prove difficult. Your bridge can *only* null the in-phase signal. An extension to the standard bridge involves adding a series or shunt capacitance (depending on the phase of the reactance) to the A or B port. This may allow you to null the reactive part and also provide the reactive impedance value as well. An outstanding reference may be located with your favorite search engine: Look for the manual for the **General Radio GR1650 Impedance Bridge.** I found a copy and the download was very slow, but worth it. This manual may be the greatest reference every published on the Wheatstone bridge and comprehensively covers tuning out the reactance of complex impedances amid a myriad of other topics.



Calculate Input Z

$$\text{Input } Z = \frac{1}{1/10K + 1/4K7 + ((\text{Beta}+1) * (26 / 5.8))}$$

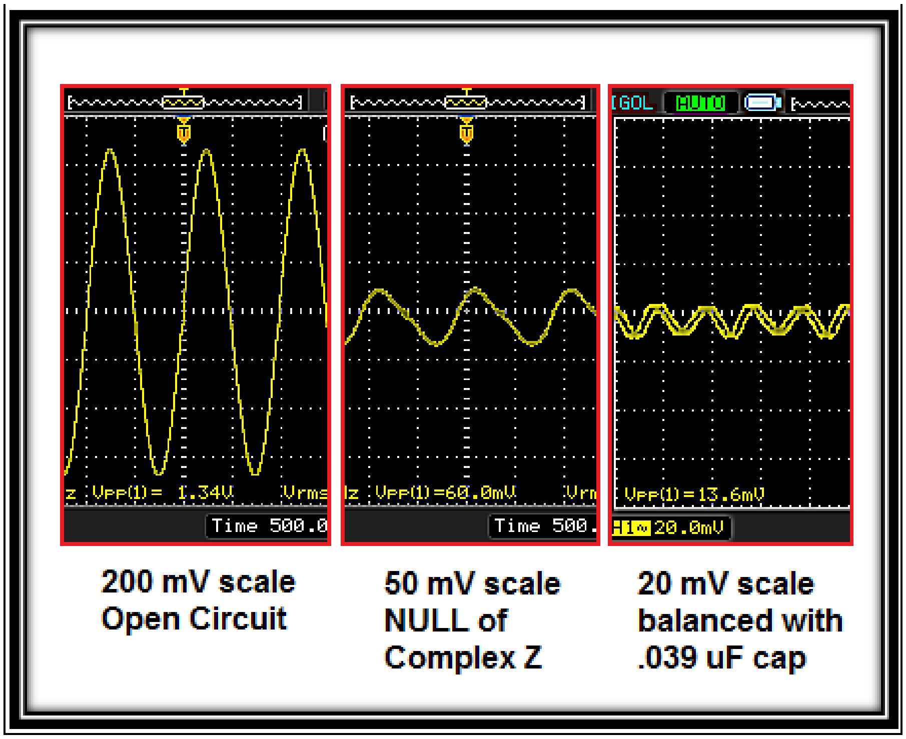
Assuming a Beta of 100 : Z_{in} = 397 Ω

Assuming a Beta of 200 : Z_{in} = 703 Ω

Z_{in} measured with above bridge = 595 Ω

Above — The poor man's bridge measurement of a test AF amplifier on my bench. The reactive component of the amplifier input impedance was minimized using a 0.039 uF capacitor found experimentally. Of particular interest, is the difference between the *calculated* input Z and the *actual* input Z measured with the bridge.

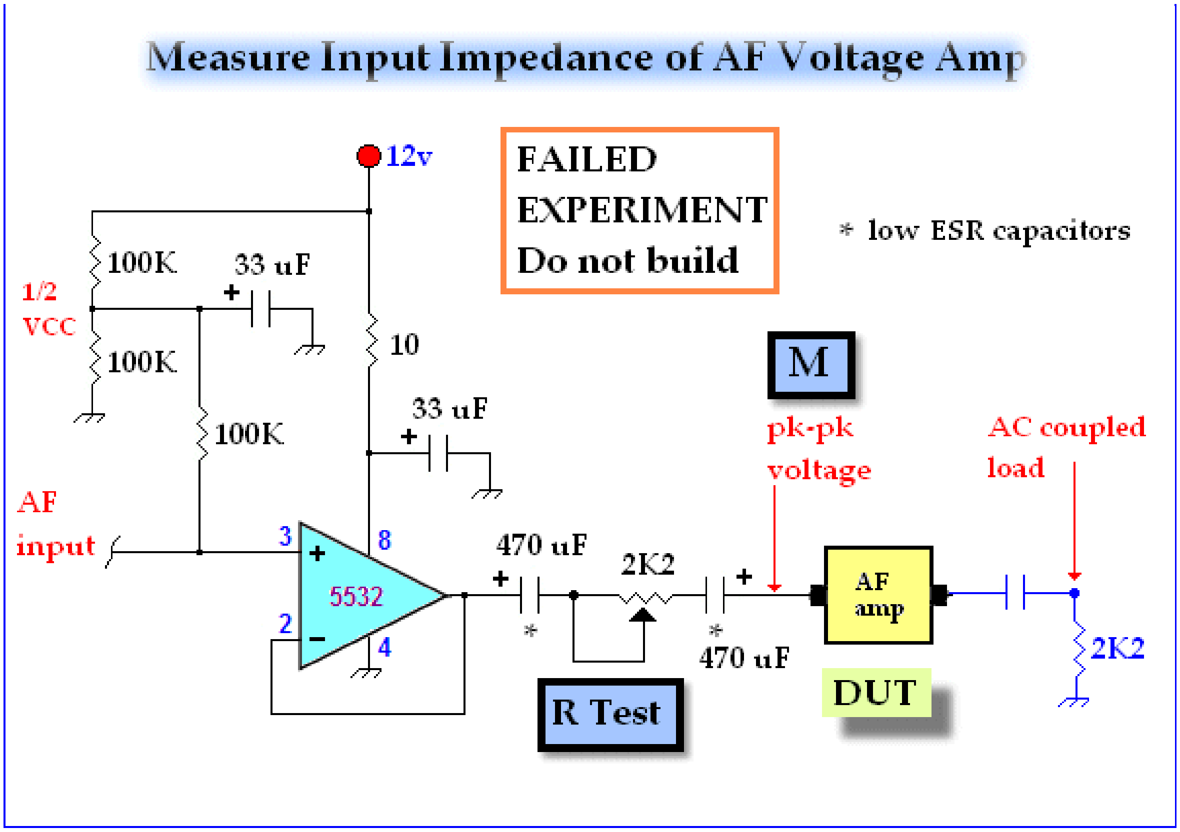
The Beta of the 2N3904's in my collection ranges from about 100 to 225. Calculations with 2 different Beta values are shown (RE is *well* bypassed with a 470 uF capacitor, so, re = (Beta * 26/5.8 mA) . The measured input Z was 595 ohms. I confirmed this by removing the 0.039uF tuning capacitor, plus connecting a fixed 595 ohm resistance to the Unknown Impedance port. I then turned the potentiometer fully clockwise and adjusted it for a NULL. When the bridge was nulled, the potentiometer knob pointed at the same mark as when the amplifier was connected to this Unknown Impedance port.



Above — The oscilloscope output waveform of the amplifier circuit shown above: open circuit, with the potentiometer balanced as well as possible and finally, the potentiometer balanced with the addition of a 0.039 uF capacitor attached to Port B. Although, I was able to get a NULL without the capactitor by just tweaking the potentiometer, slightly better precision was obtained after adding the 0.039 uF balancing capacitor.

Part 4: Miscellaneous Circuits, Scans and Photographs

Above — More accurate results will be obtained with a calibrated 10 turn potentiometer to balance your bridge. A local store sold me this precision 10-turn, 10K pot for about 11 dollars (still expensive for me). They normally sell for twice this price in Canada.



Above — My first input impedance measurement device that didn't work. It turns out my experiments were performed *incorrectly*, however, I'm glad because this failure spurred me to investigate bridge networks. The series resistance method *is* worth understanding and happily, Jeff, AD6MX described the correct procedure in a private email received December 2010. I quote him below:

"The series resistance method for input impedance should start with the variable resistor disconnected from the node to be measured. The open circuit voltage at the end of that resistor is measured (the resistor value doesn't affect the open circuit voltage since there's no current into an open circuit.)

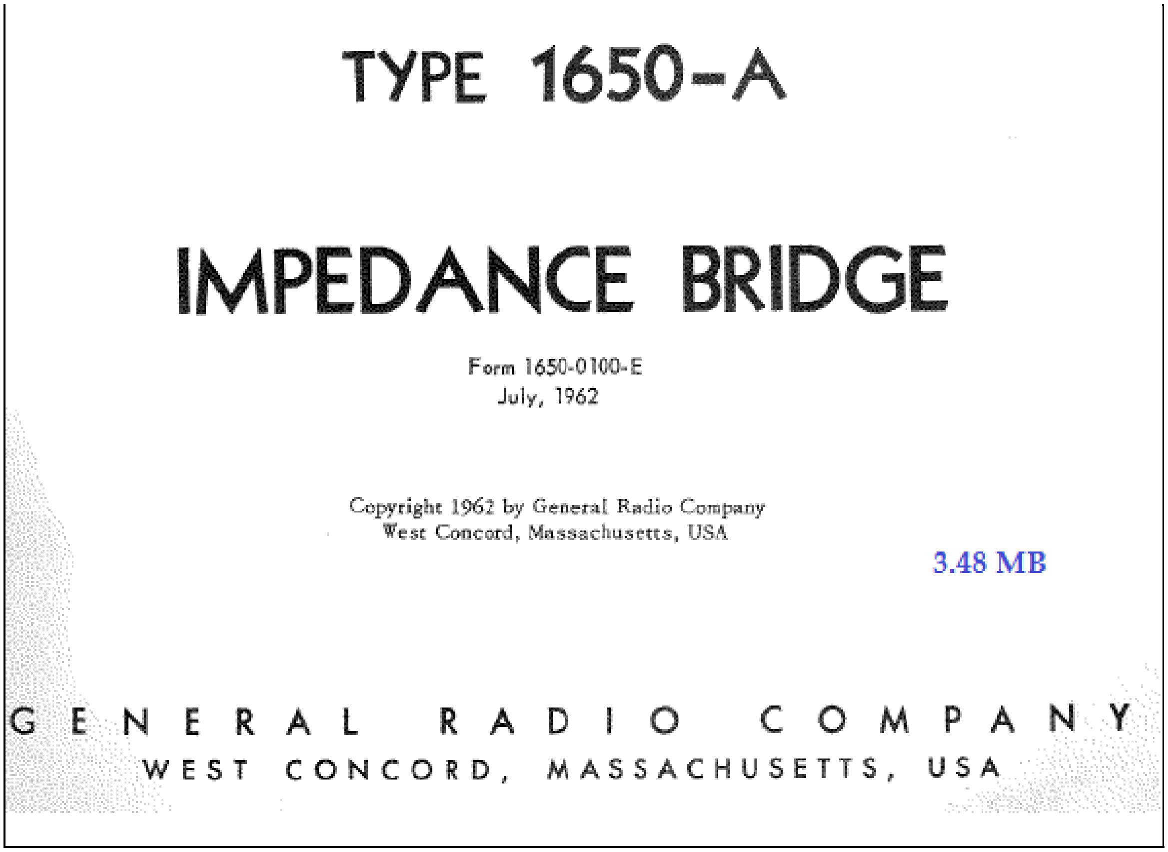
Next the free end of the variable resistor is then connected to the input node and the resistor is finally adjusted for half the open circuit voltage at the same end of the resistor, at the input node being measured. What happens is the variable resistor and the node input impedance form a voltage divider, with equal arms or branches.

The value of the resistor when measured out of the circuit is the same as the input impedance at the measured input node. This scheme has some assumptions: the driving amplifier has negligible output impedance compared to the measured impedance, and the input impedance is purely resistive, with no reactance or V-I phase shift.

The phase shift condition may be checked by taking these 3 voltage measurements: across each branch of the divider separately, and also the driving source voltage (across both branches in series.) The sum of the separate branch voltages should match the source voltage when there is little phase shift.

This 3 voltage scheme is used in some antenna analyzers in order to measure phase shift. For checking for the resistive condition, it's not important the 3 voltage method has a sign ambiguity which needs an additional step to resolve. Your description seemed to suggest starting with zero series resistance, but you see that is not the same as the procedure above. The applied voltage needs to be small enough that the amplifier remains operating in its linear range during the measurement".

Thanks for this info Jeff!



TYPE 1650-A

IMPEDANCE BRIDGE

Form 1650-0100-E
July, 1962

West Concord, Massachusetts, USA

3.48 MB

GENERAL RADIO COMPANY
WEST CONCORD, MASSACHUSETTS, USA

Above — A modified scan of General Radio's *über awesome* manual for the Type 1650-A Impedance Bridge.

RF — Test and Measurement

RF Workbench Page 3

This web page is the third installment of a 6 part series that explores basic measurement of RF circuits.

Part 3, further examines Return Loss Bridges from a bench-practice viewpoint.

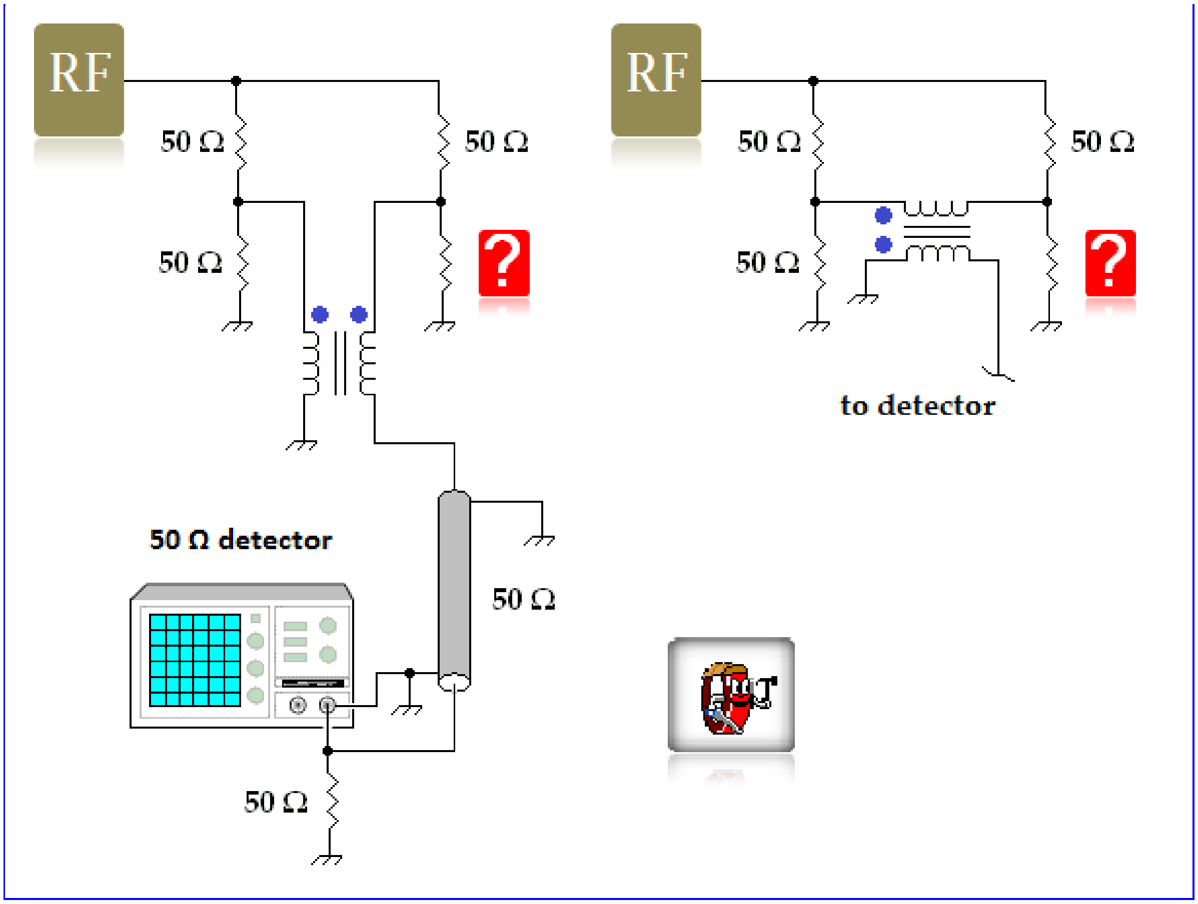
I borrow heavily from the work of Wes, W7ZOI per correspondence, direct contributions and from EMRFD.

I focus on measuring low-level, HF circuits with a return loss bridge — topics such as using the bridge for antenna matching are omitted and readily found on the web. This web page contains minimal text and just relies on simple diagrams and photographs to transfer ideas and knowledge. Information regarding wideband bridge network function may be found elsewhere on this and other web sites and in EMRFD.

More on Return Loss

Gear

Equipping for a 50 Ω measurement environment in 2010 greatly improved my design capacity. The 50 Ω terminated oscilloscope makes a sensitive and accurate detector for return loss measurement. Discussion about using a 50 Ω oscilloscope termination is on the RF Workbench 1 web page.



Above left — The RLB and measurement set up from EMRFD. Occasionally, you may see bridges using a different balun transformer wiring as shown to the right of this figure.

Above — All the needed parts to home build a return loss bridge. For some, the parts investment might seem substantial, but what hobby isn't expensive? If you consider the cost of commercially manufactured bridges, a homebrew solution seems a bargain. Recycled parts and a home-built chassis are inviting cost-containment techniques. See the web site of Jim, K8IQY for an example of a homebrew RLB chassis. Jim, a Manhattan style construction wizard, builds the nicest looking gear — he puts me to shame.

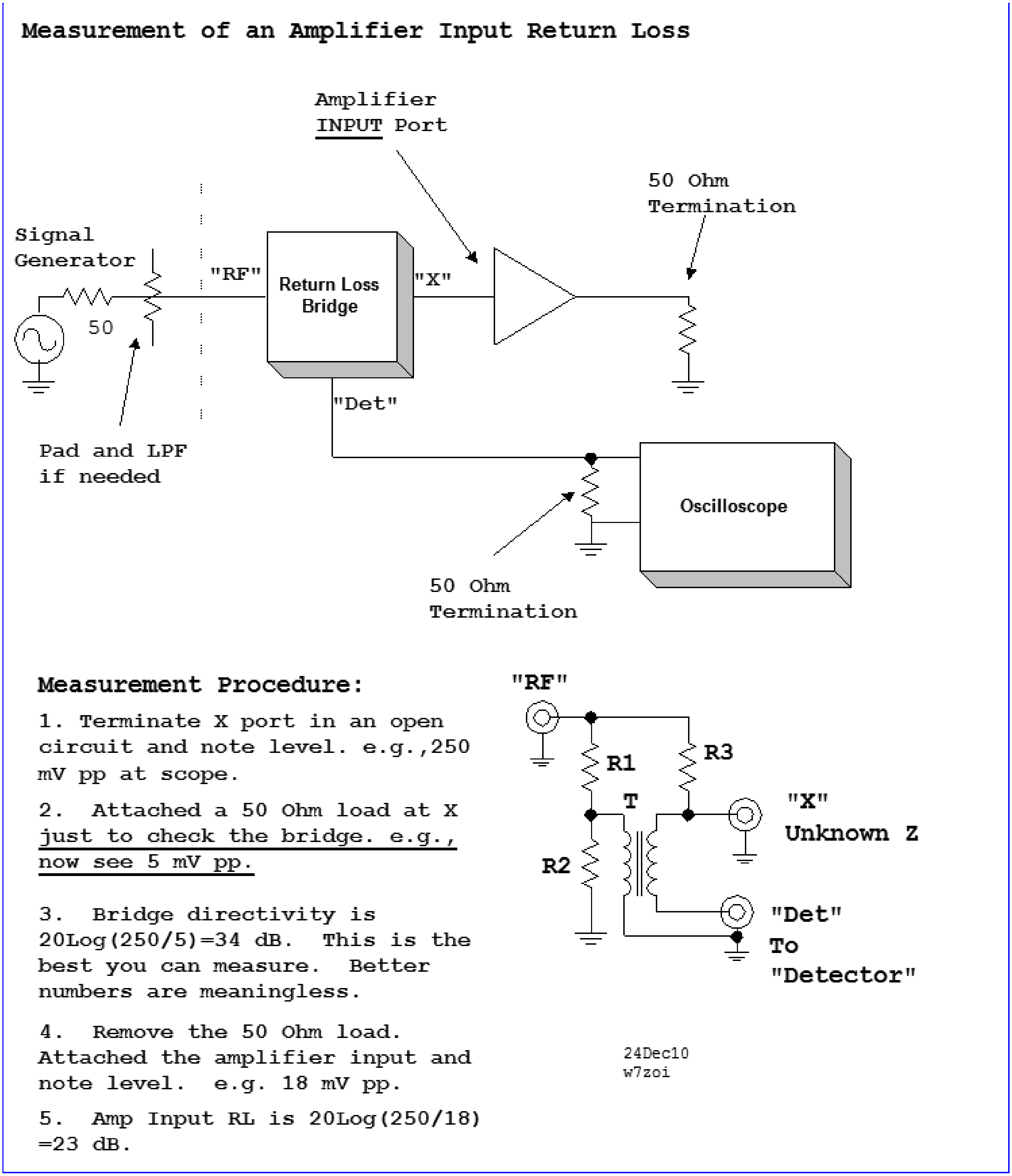
Above — A completed bridge. I used 1% tolerance 49.1 ohm resistors and an FT50-43 ferrite toroid for the bifilar wound transformer. Inductance = 38.4 uH. Many builders use the FT37-43 ferrite core. I prefer using 2 colors of enamel coated wire to avoid confusion when building stuff with transmission line style transformers and all I had in 2 colors was 24 gauge wire The bigger size ferrite toroid better accommodates the 24 gauge wire, plus photographs better.

Bridge directivity of the above RLB was 30 dB at 7 MHz, 34 dB at 14 MHz, 35.6 dB at 21 MHz, 42.1 dB at 50 MHz and 43.4 dB at 100 MHz.

If you build a circuit with a return loss close to 30 dB, it's a good day.

Above — The completed RF-tight bridge. Don't forget to label your network ports.

Measurement Technique



Measurement of an Amplifier Input Return Loss

Measurement Procedure:

1. Terminate X port in an open circuit and note level. e.g.,250 mV pp at scope.

2. Attached a 50 Ohm load at X just to check the bridge. e.g., now see 5 mV pp.

3. Bridge directivity is 20Log(250/5)=34 dB. This is the best you can measure. Better numbers are meaningless.

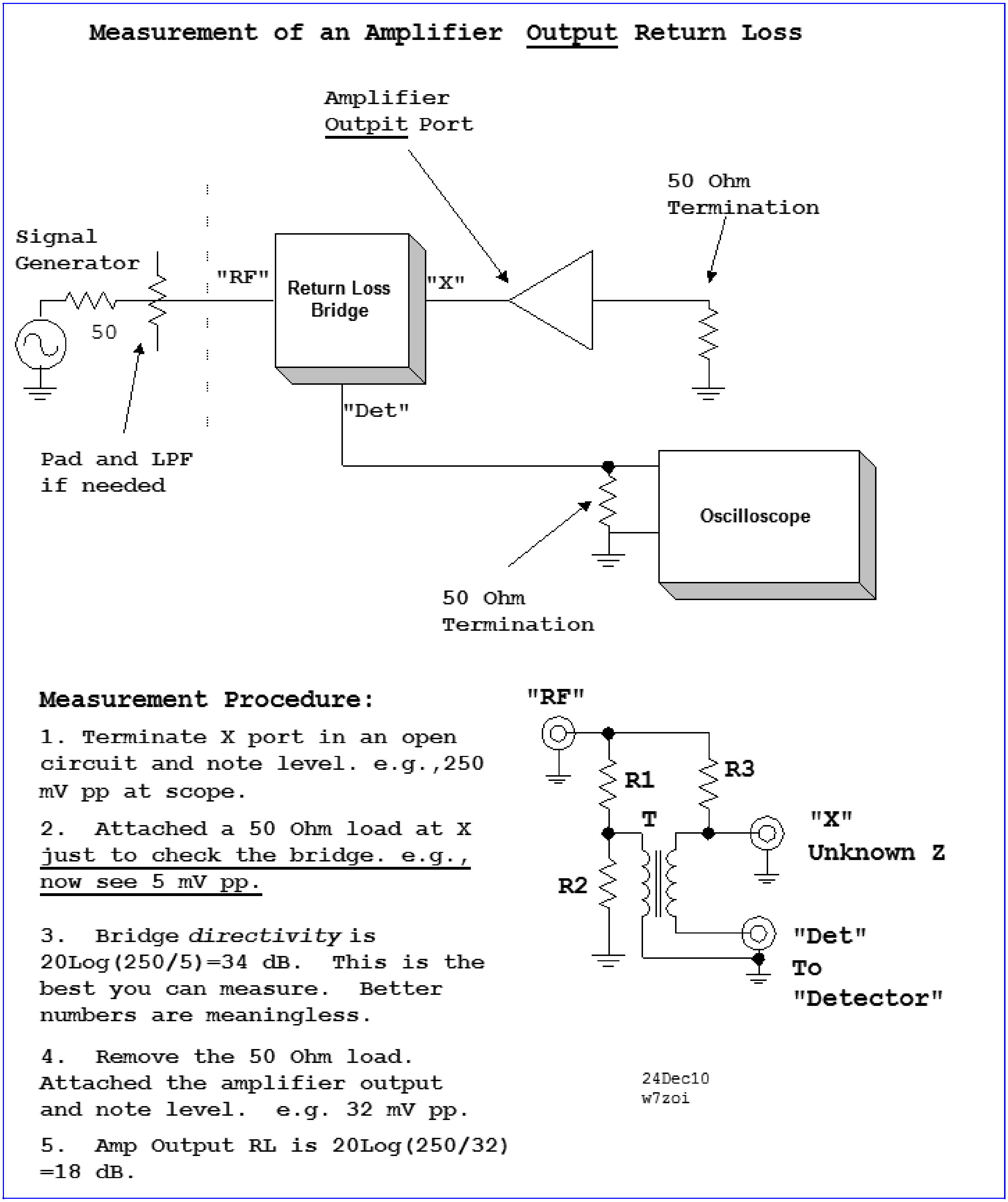
4. Remove the 50 Ohm load. Attached the amplifier input and note level. e.g. 18 mV pp.

5. Amp Input RL is 20Log(250/18) =23 dB.

Above — Measure the return loss input of an amplifier. You'll need at least one 50 ohm BNC feed-through terminator on your bench to test amplifiers with wired-in BNC connectors (such as on this amp); else just solder a 47, 49.9 or 51 ohm resistor from the amplifier output to ground. The BNC connectors allow you to quickly and solderlessly interface components such as filters, attenuators, oscilloscopes or **50 Ω** signal generators.

A typical amp measurement work flow may go something like this: Measure gain using a signal generator and the 50 ohm terminated scope; add the bridge and measure input return loss; finally, flip the amplifier around and measure output return loss. All 3 functions can be performed in 2

— 5 minutes including time to drink coffee + perform calculations by computer or with a HP scientific calculator.



Measurement of an Amplifier Output Return Loss

Measurement Procedure:

1. Terminate X port in an open circuit and note level. e.g.,250 mV pp at scope.

2. Attached a 50 Ohm load at X just to check the bridge. e.g., now see 5 mV pp.

3. Bridge *directivity* is 20Log(250/5)=34 dB. This is the best you can measure. Better numbers are meaningless.

4. Remove the 50 Ohm load. Attached the amplifier output and note level. e.g. 32 mV pp.

5. Amp Output RL is 20Log(250/32) =18 dB.

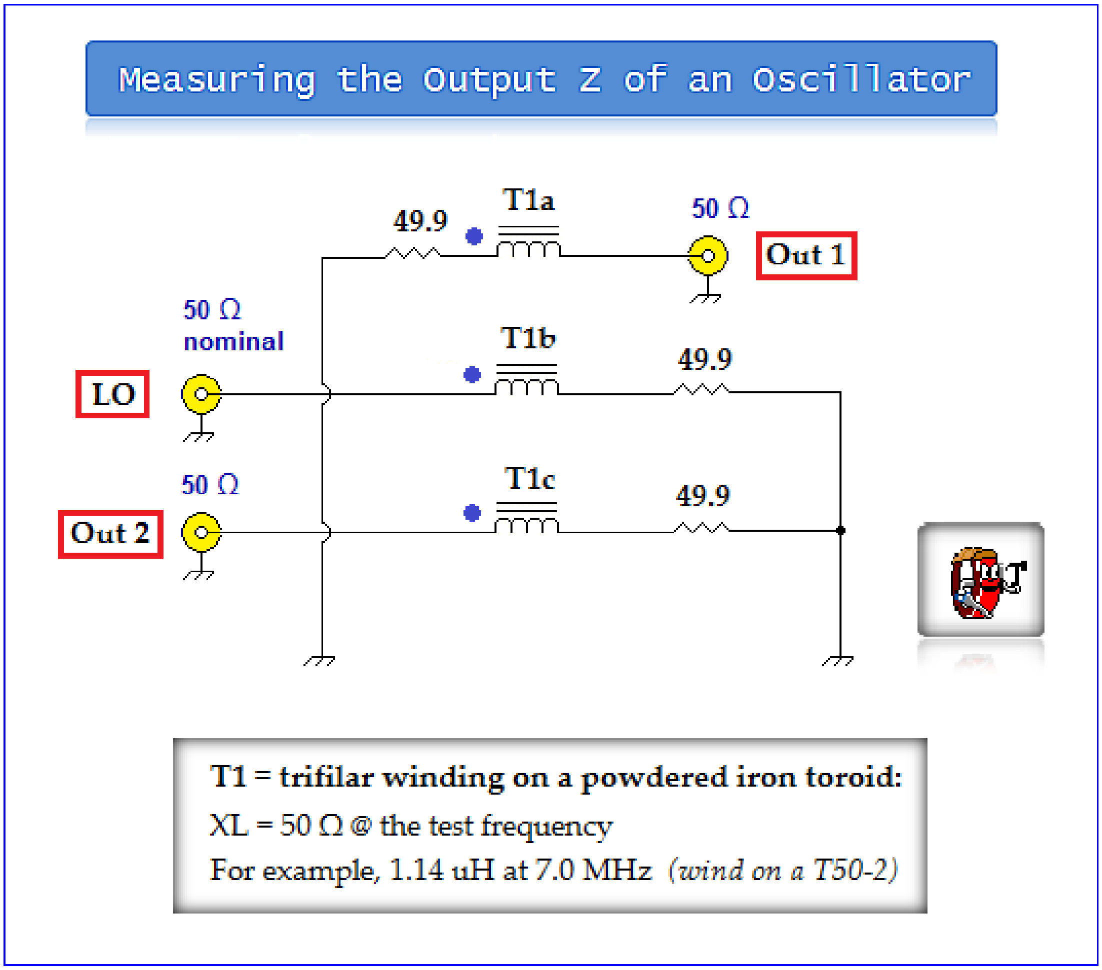
Above — Measure the return loss output of an amplifier. The above 2 procedural diagrams provided by Wes, W7ZOI. Many thanks to Wes. These figures are copyrighted © by Wes Hayward, 2010.

Your signal generator should have a return loss of at least 20 dB for greatest accuracy — all of my bench test generators have at least 30 dB of return loss. If you have a signal generator with a low impedance output and place a 10 dB attenuation pad on the output, you'll have at least 20 dB of return loss.

In the above figures, Wes gives an open circuit return loss of 250 mV; I set my signal generator output so the open return loss is somewhere between 170 and 250 mV; this allows you to accurately measure a really good 50 ohm return loss at >= 5 mV or so. Some people may have trouble going any lower than 5 mV due to scope accuracy. This is just something to consider.

Bench Exploration

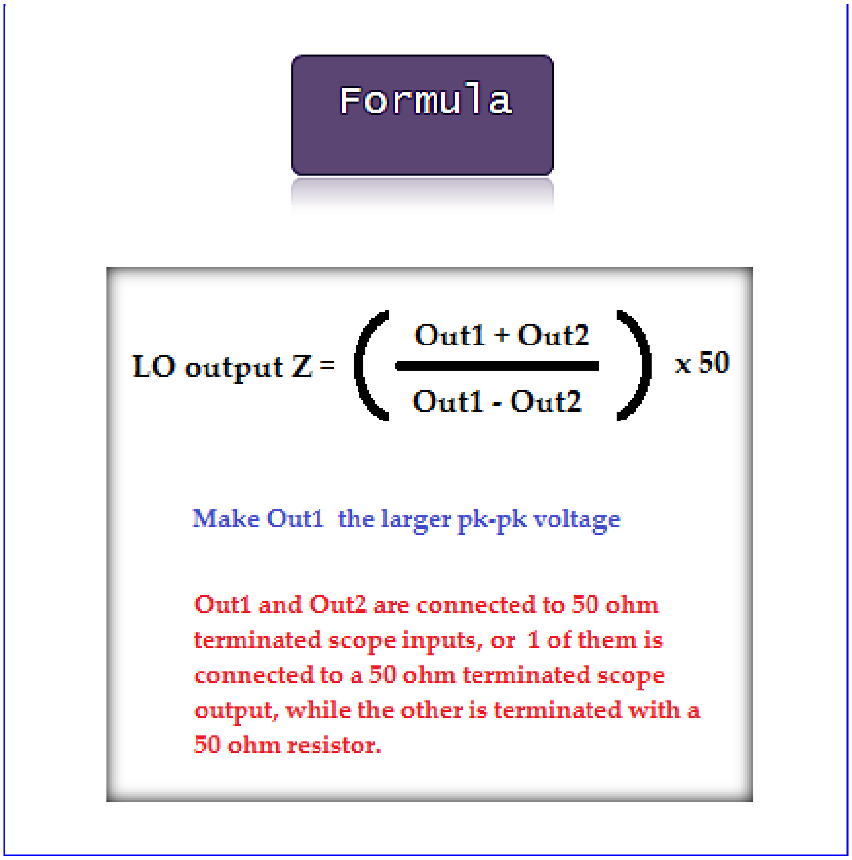
For me at least, a special case of return loss measurement exists; measuring the return loss of a local oscillator. Since the oscillator under test must be *on* during measurement, it's emitting a signal at the same frequency as the bridge signal generator and interferes with measurement. If some 50 ohm attenuation is added to reduce the local oscillator under test output signal amplitude, this increases the return loss of the local oscillator under test. This is normally a good thing, however, we seek the **raw** output return loss or output impedance of the local oscillator under test.


Measuring the Output Z of an Oscillator

T1 = trifilar winding on a powdered iron toroid:
XL = 50 Ω @ the test frequency
For example, 1.14 uH at 7.0 MHz *(wind on a T50-2)*

Above — An initial experiment that a builder from Michigan, USA and I first used to measure the output impedance of a local oscillator consistent with the breadth and scope of this web site. We wanted something simple and wished to avoid building a vector network analyzer or performing ugly algebra. I built a simple crystal oscillator for 7.0 MHz using an output transformer wound to give a low impedance output. The circuit was measured and calculated using the instrument above and the formula and procedure below.

The calculated output Z was 33.2 ohms. I build a standard value resistor 6 dB attenuator pad from this table. After fitting the pad, I re-measured and re-calculated the output impedance at 46.8 ohms. This seemed okay. I built a couple of oscillators for other frequencies and the output impedances were hundreds of ohms! — disappointing. Still, we were on the bench in a solution-focused mode and needed to try something else.

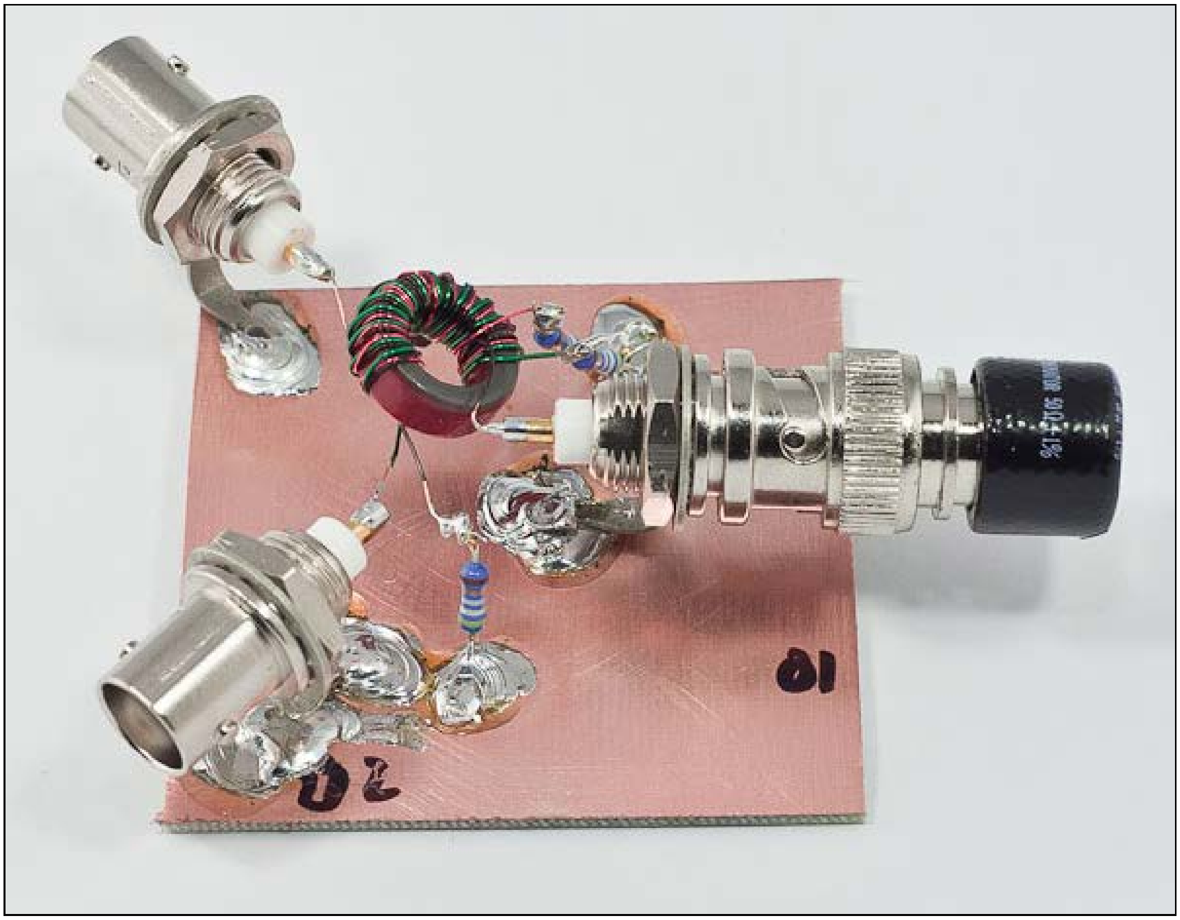

Formula

$$\text{LO output Z} = \left(\frac{\text{Out1} + \text{Out2}}{\text{Out1} - \text{Out2}}\right) \times 50$$

Make Out1 the larger pk-pk voltage

Out1 and Out2 are connected to 50 ohm terminated scope inputs, or 1 of them is connected to a 50 ohm terminated scope output, while the other is terminated with a 50 ohm resistor.

Above — The formula for calculating the output impedance with the *experimental* local oscillator output Z device.



Above — Breadboard of the experimental L.O. output impedance bridge with a 50 ohm feed-through terminator on the Output 1 port. It failed to work as expected. Skillful adult problem solving goes something like this:

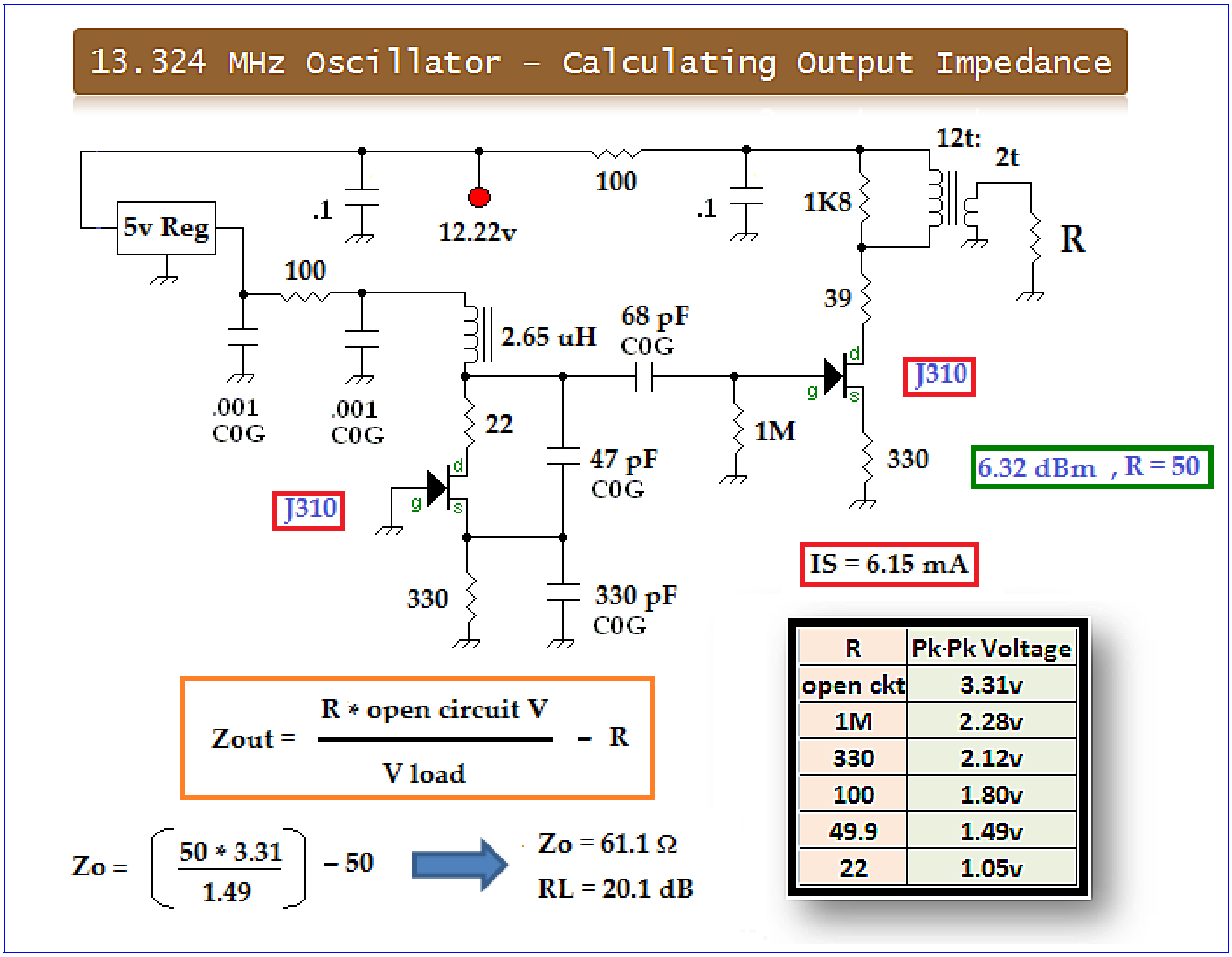
1. Identify the problem.
2. Brainstorm to generate some potential solutions.
3. Try out one of your ideas.
4. If that doesn't work, try another idea.
5. If none of your ideas work, wait a while, or ask an expert.

Well, we ran out of ideas and decided to ask experts for some more ideas; Professor Kuhn and Wes, W7ZOI.

I'll share their key messages. First, accurately measuring the output impedance of an RF oscillator can be difficult — measuring the return loss of a buffer amplifier is much easier. For this, some builders run the bridge signal generator on a slightly different frequency than the oscillator under test while using a spectrum analyzer as the 50 ohm RLB detector.

Another way is to short circuit the tank on the oscillator and measure the buffer output in the normal way — a popcorn solution indeed!

We tried calculating oscillator output impedance using different equations and 1 example is shown below. Failing to account for inductive and/or capacitive reactance plus resistance in the output circuit (including the transformer), plus upsetting the circuit during AC voltage measurement adds uncertainty to calculations — measurement seems more reliable.



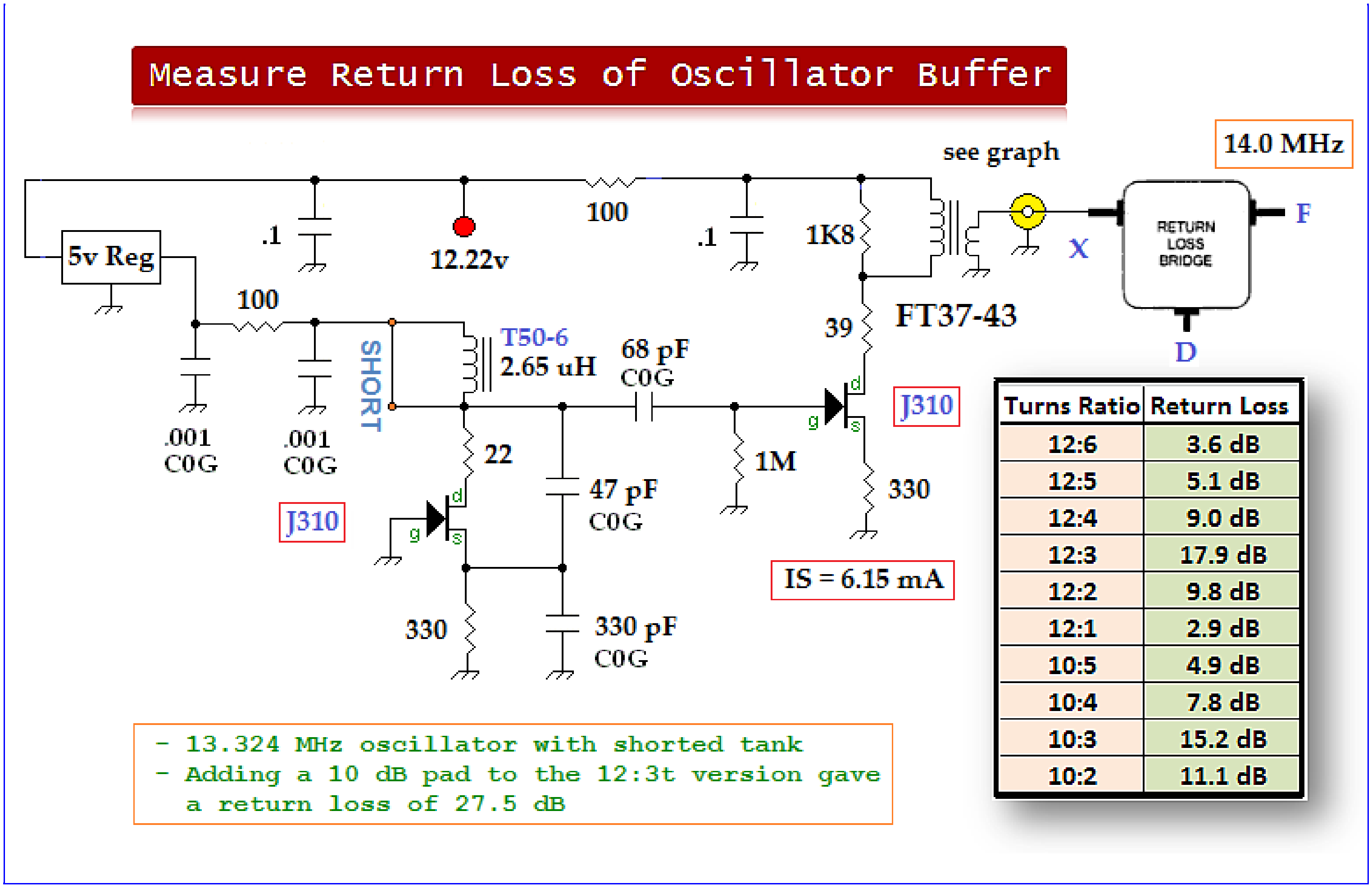
13.324 MHz Oscillator – Calculating Output Impedance

$$Z_{out} = \frac{R * \text{open circuit V}}{\text{V load}} - R$$

$$Z_o = \left(\frac{50 * 3.31}{1.49}\right) - 50 \Rightarrow Z_o = 61.1\ \Omega \quad RL = 20.1\ \text{dB}$$

R	Pk-Pk Voltage
open ckt	3.31v
1M	2.28v
330	2.12v
100	1.80v
49.9	1.49v
22	1.05v

Above — One method of calculating output impedance. Running the output at open-circuit likely effects the oscillator by changing its load despite having the JFET buffer. Some builders use this equation for calculating the output impedance in their audio amplifiers. This amplifier **should** have a 50 ohm output impedance based upon the transformer turns ratio and the 1K8 resistor across transformer: 1800:50 ohms = a 36:1 impedance ratio and a 6:1 turns ratio.



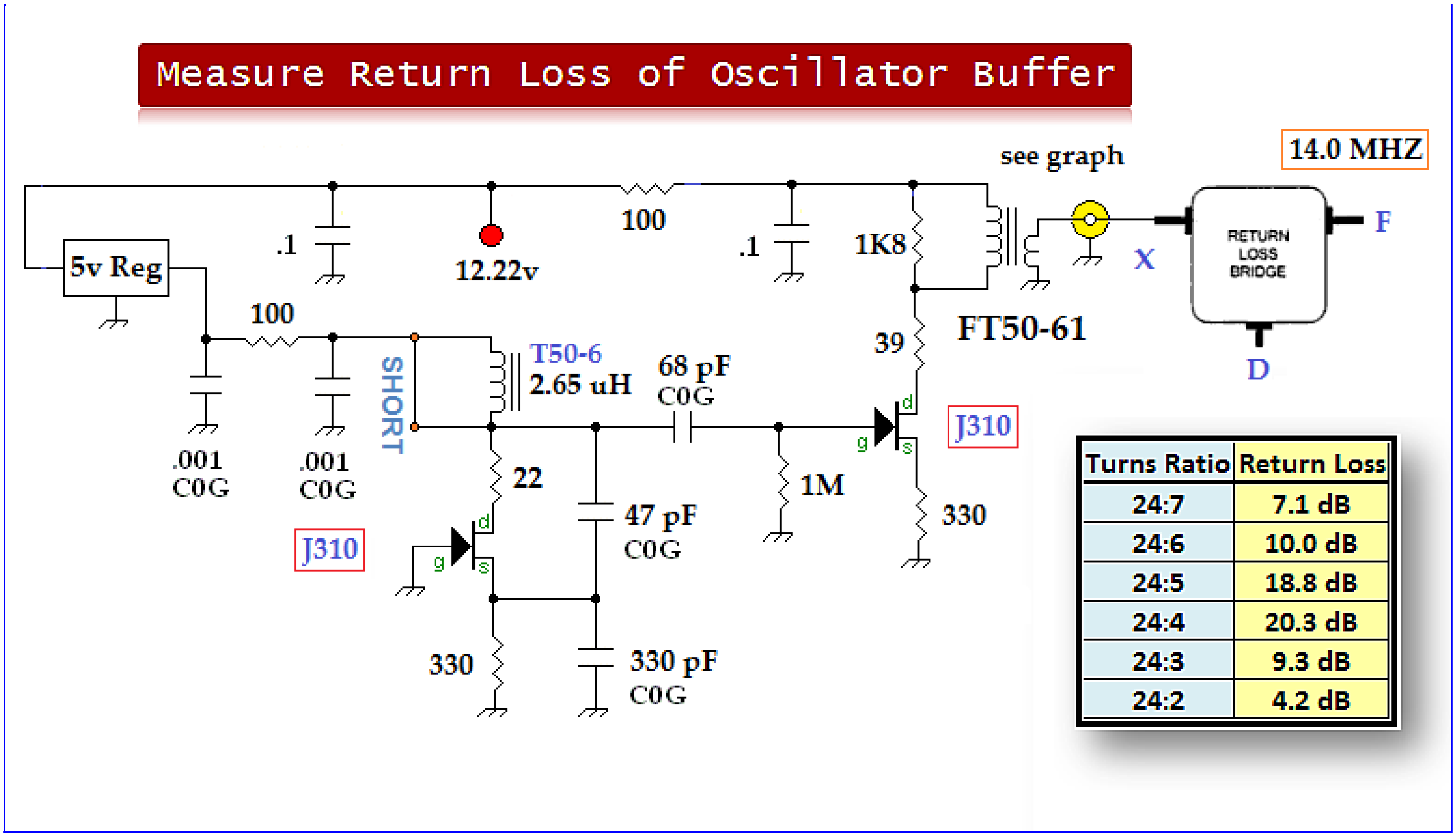
Measure Return Loss of Oscillator Buffer

- 13.324 MHz oscillator with shorted tank
- Adding a 10 dB pad to the 12:3t version gave a return loss of 27.5 dB

Turns Ratio	Return Loss
12:6	3.6 dB
12:5	5.1 dB
12:4	9.0 dB
12:3	17.9 dB
12:2	9.8 dB
12:1	2.9 dB
10:5	4.9 dB
10:4	7.8 dB
10:3	15.2 dB
10:2	11.1 dB

Above — A simple 13.3 MHz L-C oscillator was built and evaluated. After shorting the tank coil, return loss versus turns ratio was measured and tabled as shown. To my surprise, I observed the best match with a 4:1 turns ratio. This suggested that the transformer, wound on a ferrite FT37-43 toroid was exhibiting high resistance and far from the "ideal transformer". The inductance of my 12 turn transformer was 38.3 uH.

The initial secondary winding had 6 turns and then was reduced sequentially by 1 turn. After removing each turn, the 1/2 cm of increased wire length was cut off and the enamel scraped off of the new wire ending to ensure a short connection to the output jack. During measurement, unless the secondary transformer wires were kept tidy, a ~40 MHz oscillation occurred when the 1K8 resistor was disconnected. The 1K8 resistor prevented such oscillations and improved the return loss by 1-4 dB at the various turns ratios. Testing frequency was 14 MHz.



Turns Ratio	Return Loss
24:7	7.1 dB
24:6	10.0 dB
24:5	18.8 dB
24:4	20.3 dB
24:3	9.3 dB
24:2	4.2 dB

Above — The same circuit with a lower-loss, FT50-61 ferrite transformer. I could have used a FT37-61, but prefer the 50-61 as the bigger core allows the use of heavier wire which provides some robustness when performing intensive experiments. The inductance of 24 turns on a FT50-61 measured 36.2 uH. Although lower permeability ferrite toroids require more windings, this transformer is closer to the "ideal transformer" than that wound on a FT37-43 ferrite core — a 6:1 turns ratio gave the best return loss; the output Z is pretty close to 50 ohms.

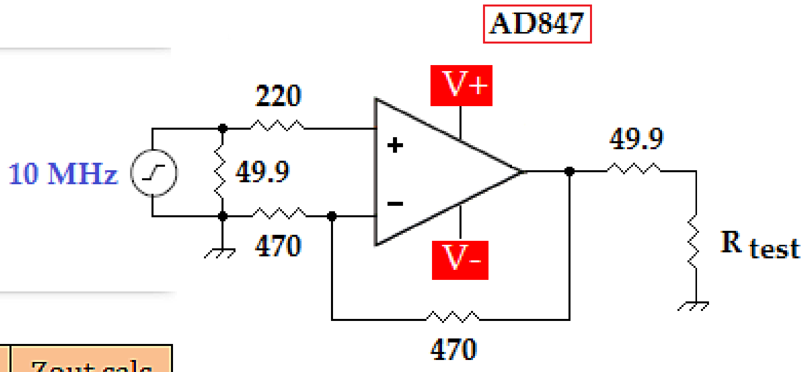
The information garnered during these tests proved enlightening and reinforces why bench measurement provides the greatest way to learn about and optimize your circuits. I hope this simple web page on return loss measurement fuels your own experiments — the most important experiments will be those *you* do on *your* bench.

Miscellaneous Figures and Photos

Calculate Output Z of an Oscillator – Experiment 1

Measure the open circuit output amplitude of the oscillator and then add a load so that the loaded voltage dropped to something in the range of 50 to 75 percent of the open circuit voltage. Then use voltage divider theory to calculate the source resistance of the oscillator.

$$V_{loaded} = \frac{V_{unloaded} * R_{load}}{(R_{oscillator} + R_{load}}$$

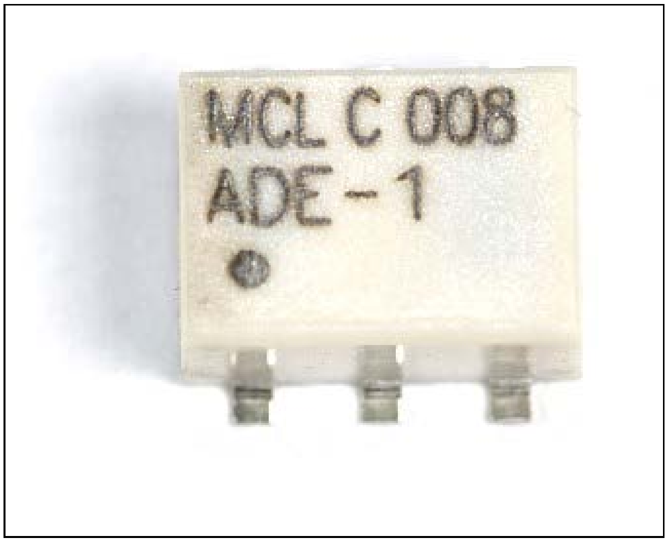


Rtest	Vout	Icalc	Pcalc	Zout calc
1000000	0.9999	9.999E-07	9.998E-07	
220	0.8148	0.0037	0.0030	49.9779
50	0.5003	0.0100	0.0050	49.9300
22	0.3056	0.0139	0.0042	49.9823

RF — Test and Measurement

Miscellaneous RF Topics 2011

Introduction



My basic goal for Fall and Winter 2010 was to fearlessly advance my RF design ability — pushing just beyond my comfort zone to impose the psychological stress that promotes focused learning. You may have experienced this in University or when working against a deadline. Cramming, "burning the midnight oil", or locked room brainstorming exemplify this approach.

These circuits feature carefully measured input and output impedances (nominal impedance = 50 Ω), plus voltage gain and DC current.

This web site spawns email with builders worldwide. Our interactions are varied; getting help, giving help — or just chat. On occasion, I design, build and/or test circuits to help struggling builders. After spending considerable time, I email them my work hoping it will help. Often enough, I never receive any acknowledgement from these readers — did the circuit work or did they appreciate I spent 1-2 hours researching their concern?. This is actually normal — we must constantly strive to overcome our innate, self-centered nature; lest it dominate our behavior. все нормальные.

To that point, I wish to gratefully acknowledge the people who support me in this hobby: Wes, Ken, Scott, Peter, Tom and the many others whose email advice and published and private work informs and inspires me.

Topics:

1. Transmit Mixer Experiments
2. Bipolar Transistor Feedback Amplifier Experiments
3. JFET Common Gate Transistor Amplifier Experiments

Navigation and Preamble:

This web page grew into a large monster — and includes a supplemental web page with numbered topics referenced in the text. I apologize for the navigation difficulties this web page poses. Equal time was spent experimenting with the circuit designs and circuit photography. I strive to provide a variety of bitmap and photographic image styles on this web site.

---

1. Transmit Mixer Experiments

Since I've never experimented with transmit mixers, I didn't appreciate how much time goes into their design. Consider, for example, the LO system from the project entitled A Monoband SSB/CW Transceiver in Chapter 6 of EMRFD. The mega low (about -20 dBm) output from a diode ring mixing a VFO and crystal oscillator is triple tuned band-pass filtered and then amplified to +8 dBm. Continuing on, the transmitter chain features more mixing, band-pass filtering and voltage amplification by a feedback amplifier chain boosting the signal to around 300 mW. The circuits needed to mix, filter and amplify this RF chain would challenge most amateur designers — me included.

Contrast this with a typical first transmitter built by a new builder. Likely your first scratch homebrew transmitter consisted of a crystal oscillator, a keyed Class A buffer/amplifier and perhaps a Class C final amplifier. No mixer was needed for we obtained a crystal cut on the frequency of choice. Our focus was **power**— getting 0.25 to 1 watt into our antenna system! A good example was the Tuna Tin 2 transmitter by the late Doug DeMaw, W1FB that only used 2 stages. Although Doug wrote his 1976 article for Hams to build a transmitter from parts found at home, kitted versions are sold today.

Returning to transmit mixers — as amateur designers, we likely need to start on a small portion of the transmit chain and then after developing some competency, slowly extend our experiments all the way to the antenna port. In Fall 2010, I just examined some basic transmit mixing to get a feel for what's involved and what to expect. Mixing signals is a complex affair encompassing topics such as intercept point, conversion gain or loss, image noise suppression, noise figure, spurious/intermodulation products and port isolation. To keep things simple, only mixer port isolation and reducing spurious mixer products were examined.

Before beginning, I express the following concern: We experimenters, as stewards of the airwaves, must build exemplary transmitters with very low spurious outputs. I follow the example of Wes, W7ZOI and others — my transmit chains have spurious frequencies at least 50-60 dB down from the carrier (dBc). As a web author and radio amateur, I never want to directly or indirectly contribute to RFI and hope you agree.

Why Use a Transmit Mixer?

If you plan to design a superheterodyne based transceiver, you'll probably need to use a transmit mixer. Also mixing 2 frequencies permits using cheap microprocessors crystals to target a desired transmit frequency; separate crystal oscillators drive the RF and LO ports of the mixer. For added flexibility, the LO can be converted to a VFO once you have the basic design working well.

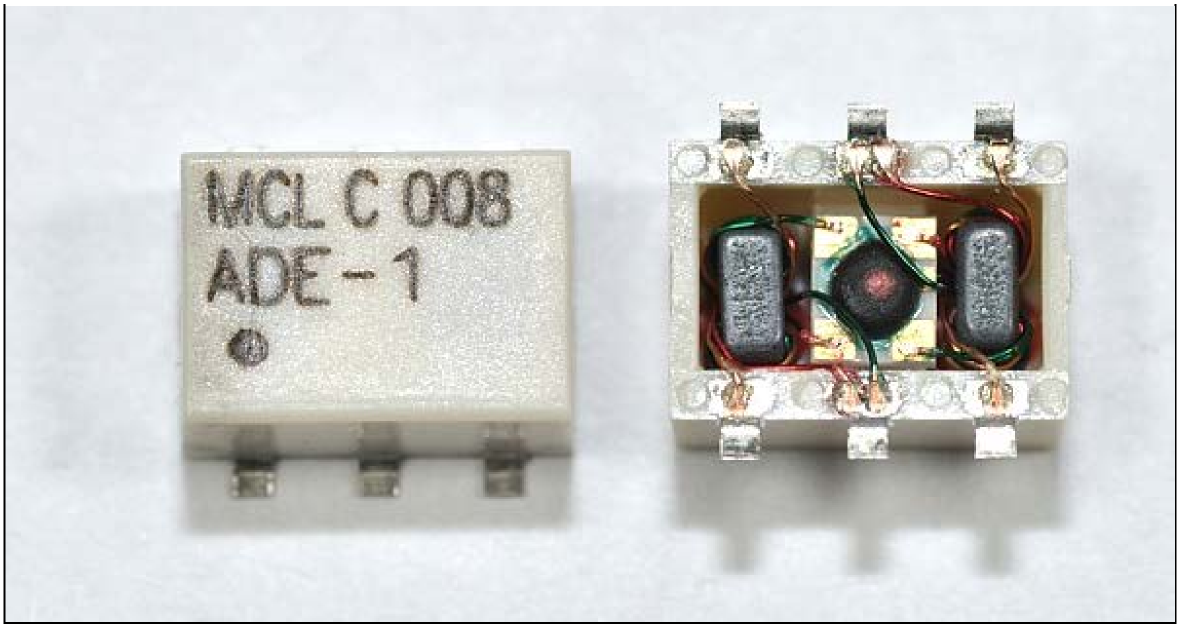
I purchased a bag of low cost crystals. By mixing 2 appropriate crystals, output on a Ham band is possible. For example, crystals at 2.048 + 5.0688 MHz = 7.117 MHz; 4.194 + 11.228 MHz = 7.034 MHz; and 3.932 + 11.046 MHz = 7.114 MHz. I frequently operate QRP on 40 Meters in the USA Novice band, so 7.114, or 7.117 MHz is okay. This helps CW operators avoid all the RTTY and QRM down in the traditional 40M band QRP frequency window.

Some Mixer Bullets

- Mixers have 3 conventionally named ports; RF, LO (local oscillator) and IF (output).
- The diode ring mixers presented are Level 7 mixers. Maximal LO power is 7 dBm.
- Many builders limit the maximum RF power into a Level 7 diode ring transmit mixer RF port to between 0 to -3 dBm.
- The term *isolation* refers to the amount of LO power that leaks into the RF or the IF ports.
- Low-pass filtering the LO *can* significantly reduce harmonic products in a mixer
- The top of the spectrum analyzer screen (always the top, and never the bottom) is called the reference level. That is the power at the top. If you have a signal generator with the output adjusted to be -27 dBm and pass this signal into the spectrum analyzer and adjust the attenuation in the analyzer to put that signal at the top of the screen, you then the reference level is -27 dBm. *(Pertains to examining a mixer output in a SA)*

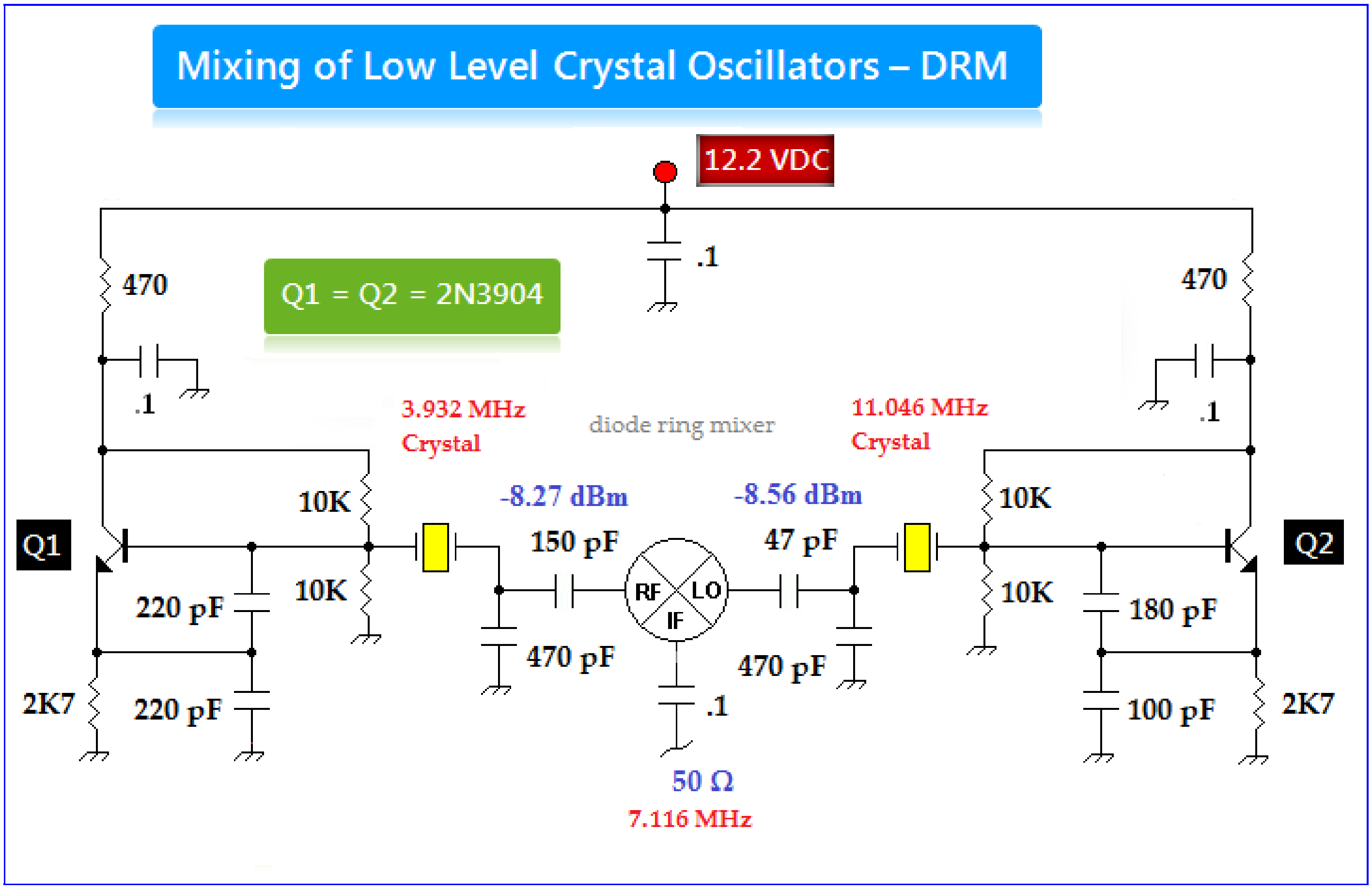
Choosing a Mixer

A number of mixers were considered; passive, active, unbalanced, single-balanced and finally, double-balanced. The diode ring mixer is an obvious good choice commensurate with my goals of reducing spurs, LO feed through and achieving high port-to-port isolation. In future web pages, other mixers may be presented, however this page is focused on the diode ring mixer. Click for a file with a few scanned pages concerning mixers from my "ideas only" notebook from ~2002. I own over 30 notebooks now.



Above — ADE-1 diode ring mixers. We're using these now as they're cheaper than the SBL-1, TUFF-1 etc. hole-though versions. Although SMT parts, they can be flipped over and wired "normally" with a little effort, steady hands and good vision. Mini-Circuits will sell them in small quantities to Hams; email them and enquire. I feel the diode ring mixer has been misunderstood by some amateur builders — lore and misperceptions that the 7 dBm LO port drive, the need for 50 ohm port terminations, a ~ 5 dB insertion loss and cost make them undesirable. Their excellent performance and design challenges are reasons why we use them; "the journey — not the destination" stuff.

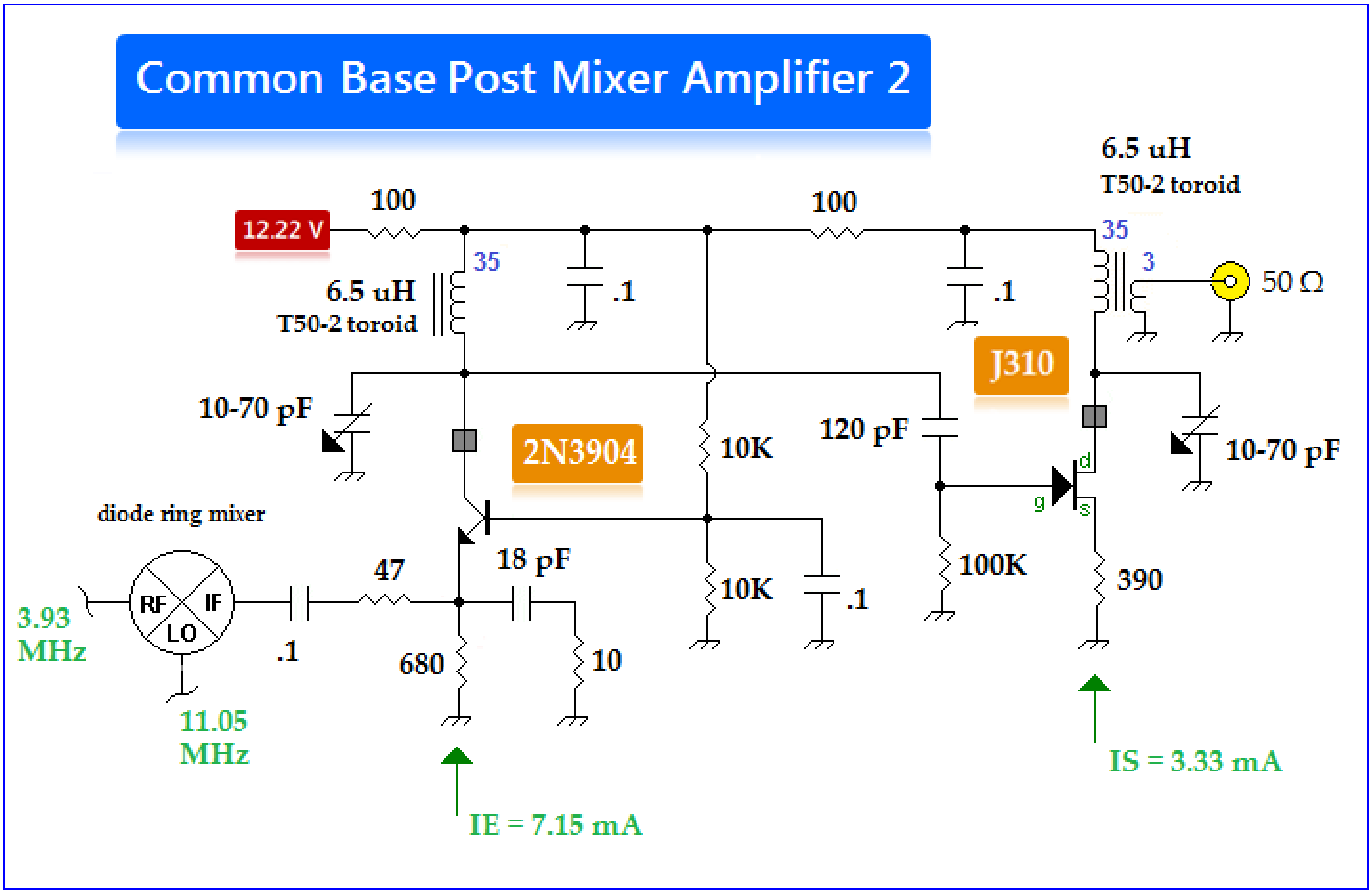
In receiver applications, some builders and kit sellers seem more focused on features such low-battery indicators, digital displays, miniaturization and cost containment than basic receiver performance. Certainly keeping cost down down deserves consideration, however, good mixer performance is king. You'll have to decide what's affordable and important and build accordingly.



Above — My very first transmit mixer experiment. My hope was to build a transmit mixer possessing low spurious output to alleviate the need for stiff, post mixer band-pass filtering such as a triple tuned band-pass filter. Thus, low-level, low distortion output was taken from between each crystal oscillators' shunt capacitor and crystal. The desired output frequency is ~7.114 MHz to build a transmitter for the 40 Meter Ham band.

The mixer output to 50 MHz looked like this in a Spectrum Analyzer. The dominant frequencies are the **sum** and the **difference**: 11.046 **+** 3.932 MHz = **14.98 MHz**; 11.046 **-** 3.932 MHz = **7.114 MHz**. The frequencies realized are *slightly* different since the oscillator output is shifted by crystal variances and from circuit capacitance.

In the experiments that follow, I built some circuits to filter and/or amplify the output of the *above* mixer circuit.



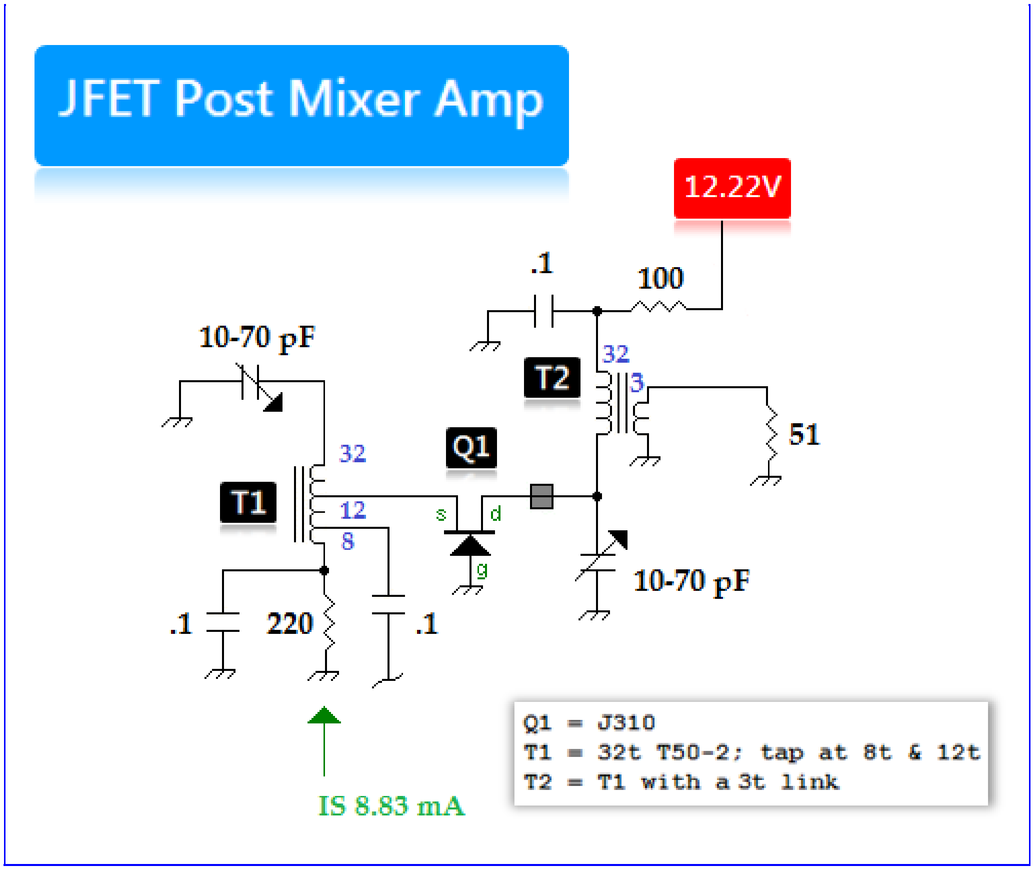
Above — The first post mixer amp; a common base input amplifier that's AC coupled to a common drain FET amp. I hoped that 2 tuned L-C tank circuits could substitute for a passive double or triple-tuned band-pass filter, plus provide some gain.

A broad-band, common base input amp was chosen to properly terminate the diode ring mixer and alleviate the need for a diplexer. A ~50 Ω input impedance is established by a 47 Ω series resistor since the 2N3904s input impedance is quite low due to the moderately high emitter current employed to boost gain and IMD performance.

This amplifier failed to reduce spurious output 50 dB down or greater — my design goal. Here are its scope and spectrum analyzer outputs; please observe that the unwanted 14.98 MHz signal is only 32 dB down from the desire IF of 7.114 MHz. An RC network consisting of a shunt 10 ohm resistor + an 18 pF cap provides additional low-pass filtering above 20 MHz. I attribute this simple filter to Dr. Ulrich Rohde as I have seen it in some of his post mixer, common base RF amplifier designs. Click for a brief supplement regarding his low-pass network ***(#2 RC Low-pass Network on the Supplemental Page)***

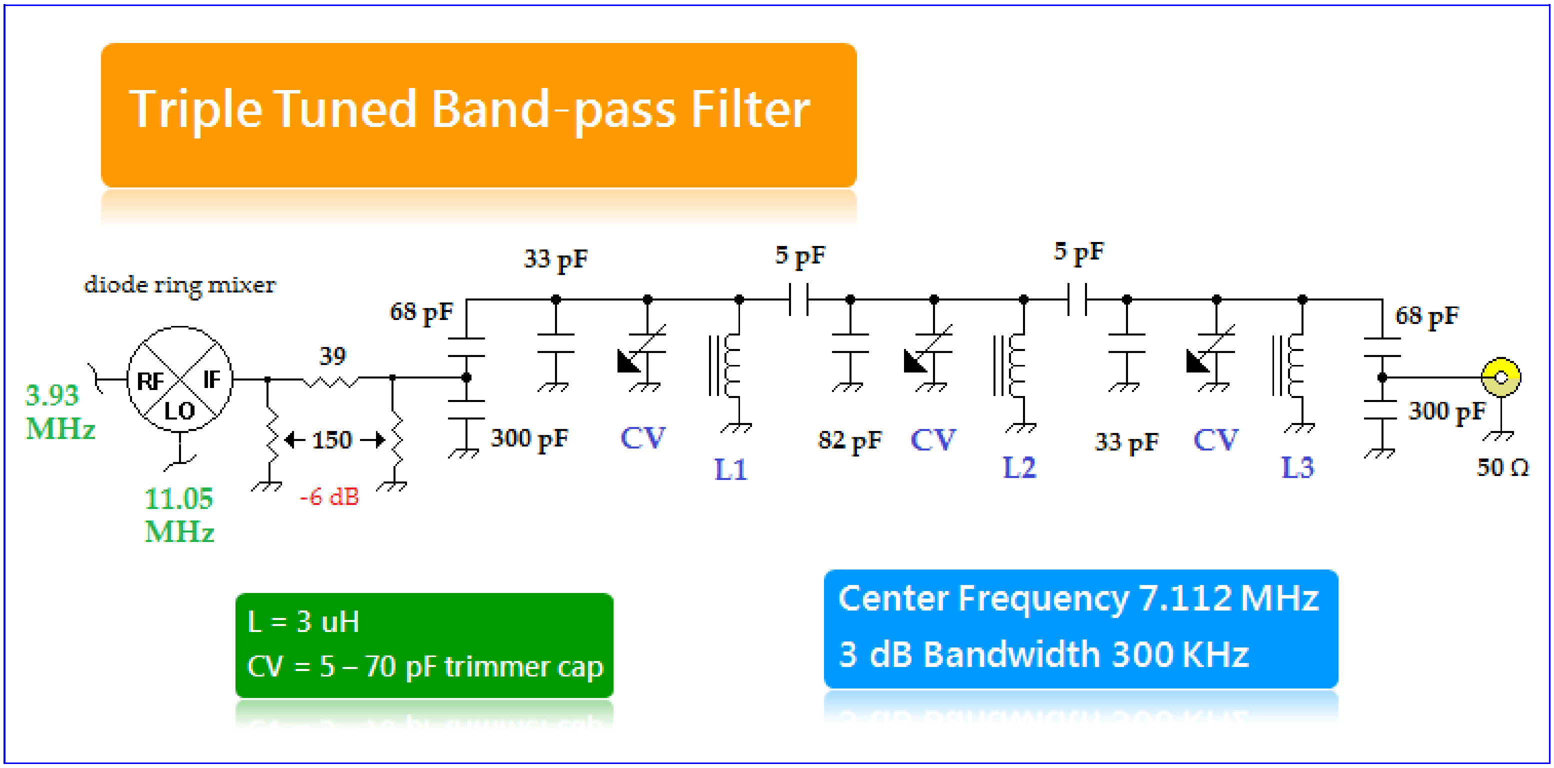
The amp design shown above was actually an improved version of this prototype. In the prototype amp, the mixer power at 14.98 MHz was only 23 dB down from the desired intermediate frequency of 7.114 MHz. You can't expect a single L-C tank to well filter a mixer output. Unfortunately, the 1K2 -12K resistor providing DC bias for the emitter follower lowers the Q of the common-base collector tank circuit. 12K is better than 1K2 in this regard. Poor performance sparked the design of the second common base amp shown above.

I use ferrite beads and 51 Ω resistors interchangeably on the collector/drains of amplifiers to snuff out UHF oscillations. According to my experiments, the resistors may work better. I purchased the ferrite beads from Diz. After the common base amplifiers shown above, I decided to try a tuned input + output common gate JFET amp:



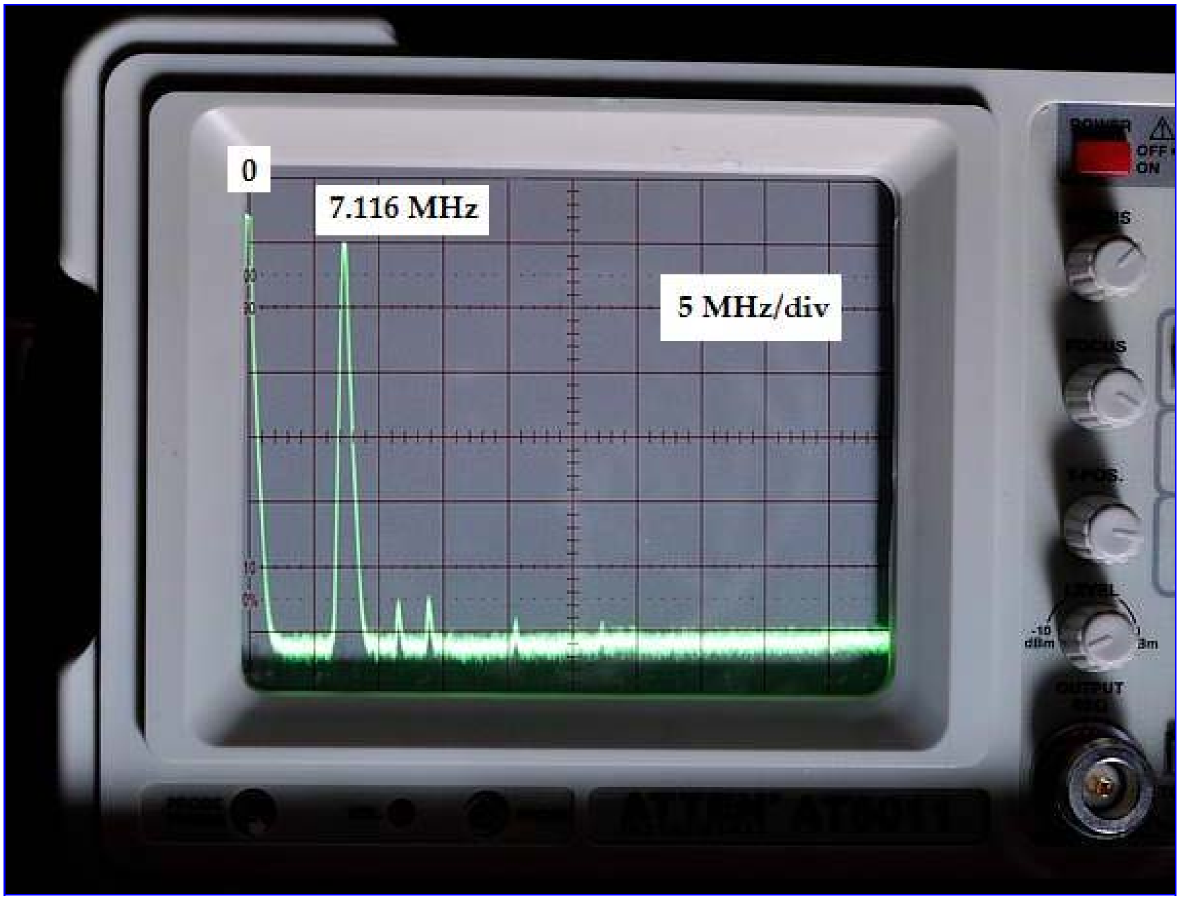
Above — A JFET common gate amp with tuned input and output built about 3 years ago. For spectrum analysis. I padded the amplifier output to provide a -28 dBm 7.116 MHz signal. The vertical resolution on the SA is the standard 10 dB/division. As shown, the 14.98 MHz signal is ~ 39 dB down; an improvement over the amplifiers shown previously. This narrow-band amplifier requires a diplexer. I wanted *better* filtering than that offered by amplified circuits with 2 tuned L-C circuits, so I halted this experiment and decided to try a triple tuned band-pass filter.

This JFET circuit experimentation spawned over 8 weeks of experiments concerning common-gate RF amps — some of them appear later.



Above — A triple tuned band-pass filter designed with software from EMRFD called TTC-08. Click for the breadboard photo. The diode ring was connected to the filter input via a 6 dB attenuator pad using short leads. Click for analysis in GPLA. All inductors are 3.0 uH — 23 or 24 turns on a T50-2 powdered iron toroid. I measured all the inductors and obtain the exact desired inductance by expanding or squishing the wire turns, or if necessary, adding (this means rewinding the entire coil) or removing a turn. If you lack an inductance meter, just winding the formula calculated number of turns will be close enough for most applications — I only got a good L- C meter in 2009 and somehow managed.

I learned that the ultimate way to peak a triple tuned filter is by tweaking the tuning capacitors while it's connected to a spectrum analyzer — what a thrill!



Above — While a little tedious to build and align, the triple tuned filter worked magnificently; the strongest spur is 54-55 dB down and the 14.98 MHz signal is gone. Insertion loss = 2.5 dB. This experiment provided a benchmark of what great post mixer filtering looks like. Post mixer filtering is an important topic worth studying further:

Why do we need filtering on a mixer output?

Let's examine mixer ports more closely. A port is just a pair of wires where signals are applied or removed. There are 2 kinds of mixer *outputs*: 1) the sum plus difference frequencies; 2) spurs.

Further, 2 kinds of spurs occur: One type is straight feed through where 1 signal from the 2 input ports makes it out to a 3rd port. Examples include LO feed through to the RF port, or LO feed through to the IF port.

The other type of spur is a mixing product such as a harmonic.

In general, the mixer output frequencies are numerically described by an equation:
IF (output) = N x L +/- M x R
N and M are both integers, 1, 2, 3, ... L = local oscillator frequency, R = radio frequency

A mixer is said to be balanced when you duplicate some of its functions and then combine them — usually with transformers. Consider, for example, the single diode mixer — they work, but the output contains ++ feed through and spurs. A mixer with 2 diodes or 2 FETs etc. can be much easier to use because the transformer combines the signals in a way that cancels some of the spurs and feed through. The double-balanced diode ring mixer uses four diodes and 2 transformers — producing even less feed through and harmonic output.

In a double-balanced diode ring mixer, the LO and RF ports are balanced and all ports of the mixer are isolated from each other. The double-balanced mixer greatly reduces, but does not stop all LO feed through at the RF and IF ports. A wideband match at 50 ohms is required to maintain mixer balance; hence you will often see attenuator pads on the LO, RF and *especially* the IF ports.

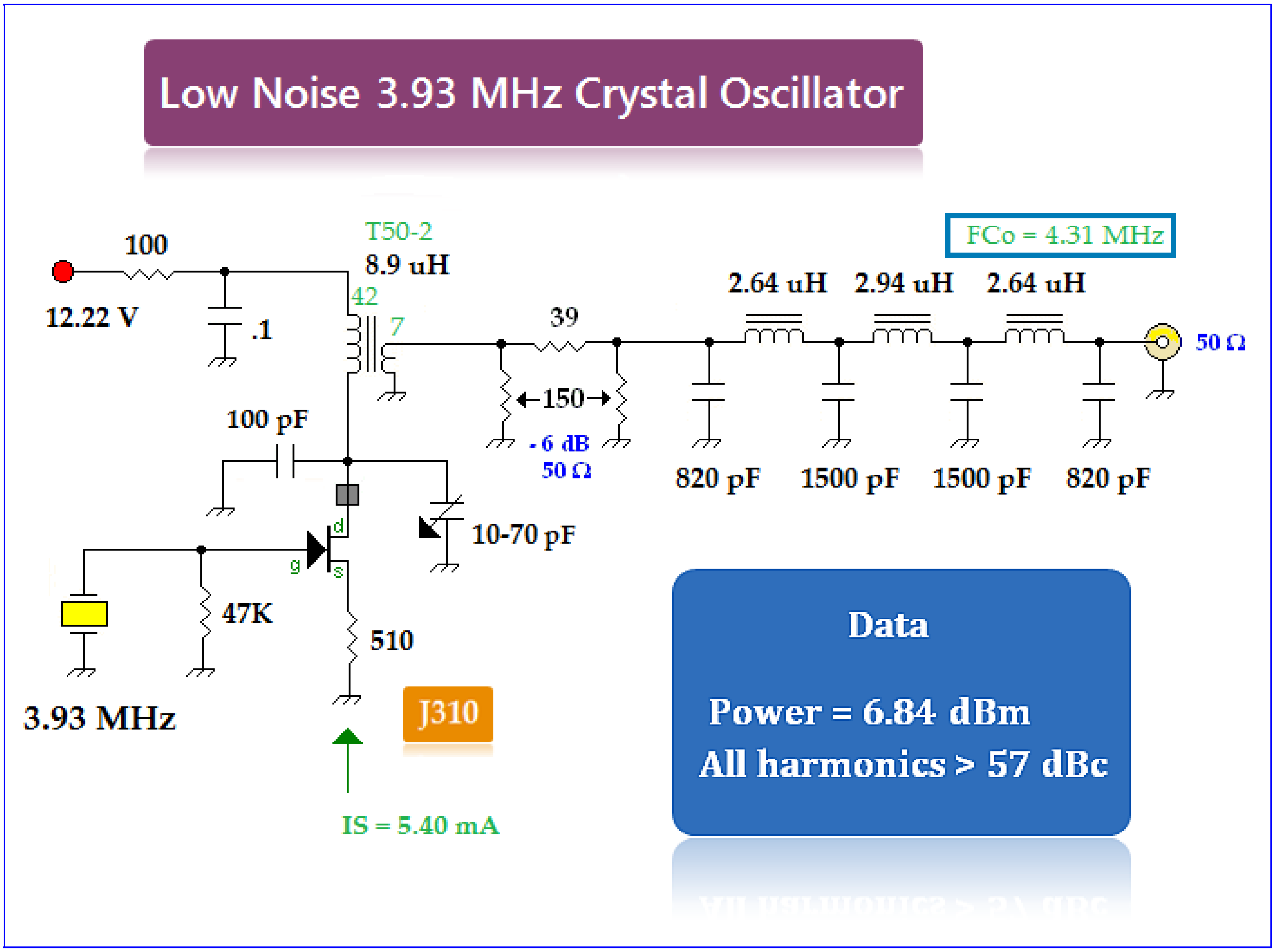
Let's focus on the IF port. Attenuator pads absorb any reflected mixer products and signals coming back into the IF port, thus increasing the match to the IF port. You may have noticed some builders use a diplexer on the IF port. The diplexer presents a wideband match to all IF port frequencies — passing the desired sum or difference frequency and absorbing the unwanted mixer products reflected back into the IF port by subsequent stages.

Since the IF output contains the sum and difference of the LO + RF, LO feed though, and other spurious energy, band-pass filtering is required to launder the IF signal into something useful. Following a transmit mixer, we filter with an L-C band-pass filter — after a receive mixer, crystal band-pass filters dominate. If you choose an unbalanced mixer or single-balanced mixer, filtering becomes more difficult than with a double balanced mixer. Unbalanced mixers are usually reserved for situations where high performance is being sacrificed for cost containment and/or want of a low parts count. There is no free lunch — you either alleviate as many mixer products as you can at a low-level with good practices, or have to deal with them down your signal chain while sacrificing optimal mixer performance.

Double balanced mixers are sensitive to non-resistive IF port terminations. When improperly terminated, the 2 transmission-line transformers work poorly — any reflected power generates high voltage across the diodes and degrades mixer performance. According to Dr. Ulrich Rohde, some proper ways to terminate the mixer include using a diplexer followed by a wideband 50 Ω feedback amp, or a common-gate JFET amplifier. *(Reference 1)*

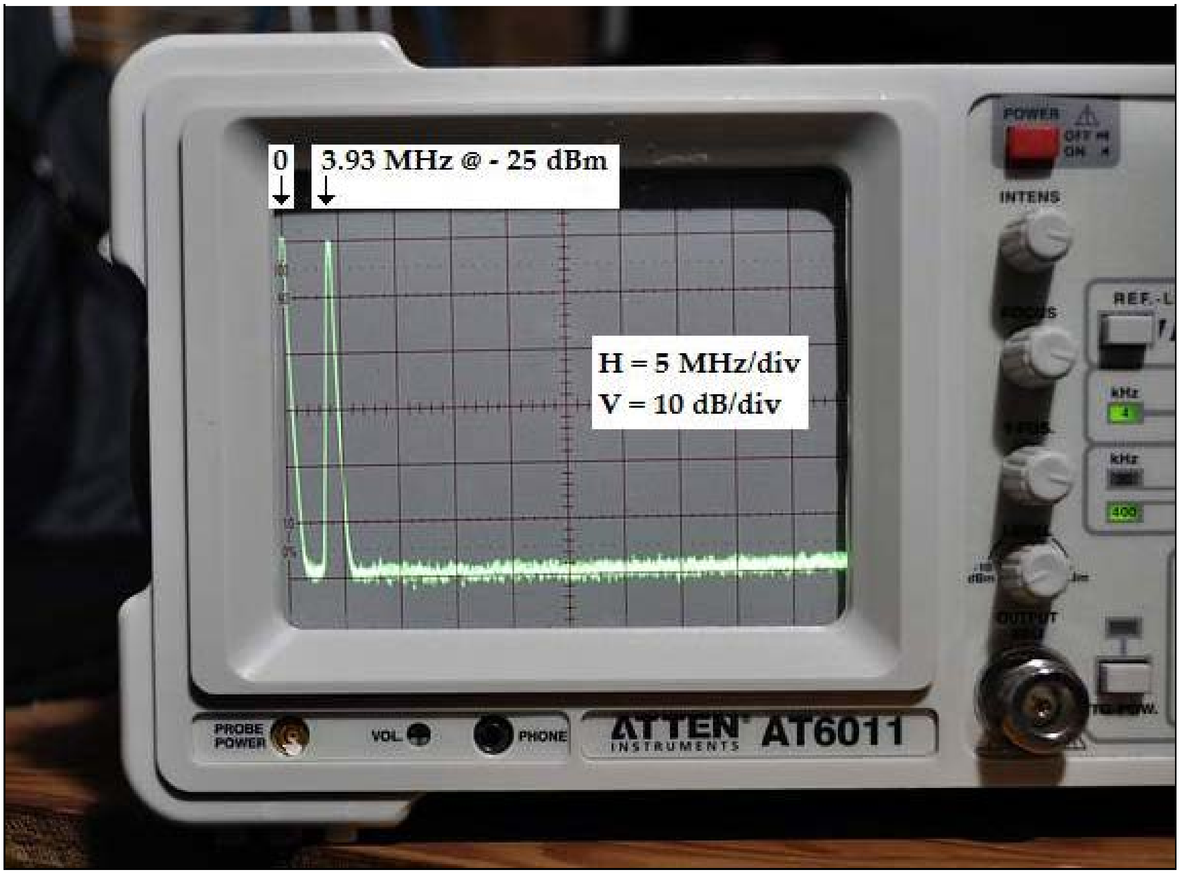
Improved Local and RF Oscillators

In the earlier experiment, I really should have the run the LO port at 7 dBm. In order to improve my experiments, new crystal oscillators were designed with emphasis on correct LO output power and low harmonic energy.

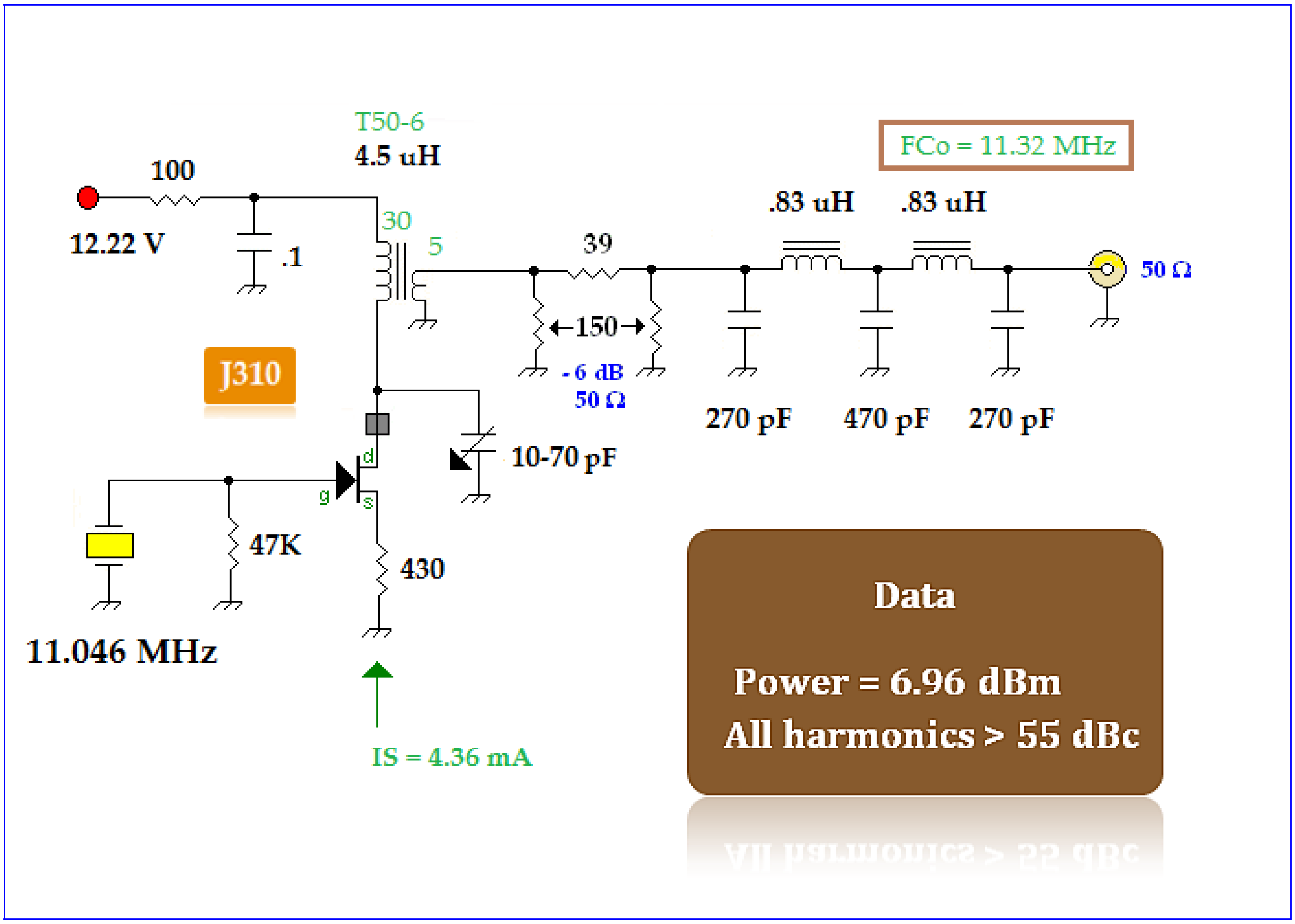


Above — The new LO; a 3.93 MHz crystal oscillator with stiff low-pass filtering. Admittedly, this 7 element Chebychev low-pass filter is overkill, however, I wanted to examine filtering and learn how much is required. On the bench — do whatever you like; even chasing crazy personal goals can be instructive and help you relate to information from texts and articles, or satisfy a whim. I can read something 100 times, but may not understand it well until I actually do it.

Above —The LO breadboard in close-up using a long focal length lens. Click for a wide angle photograph. The unsoldered end of the 100 Ω resistor in the close up photograph is where I connected the VCC.



Above — Spectrum analysis of the well-filtered 3.93 MHz LO. Not surprising, no harmonic energy is seen.

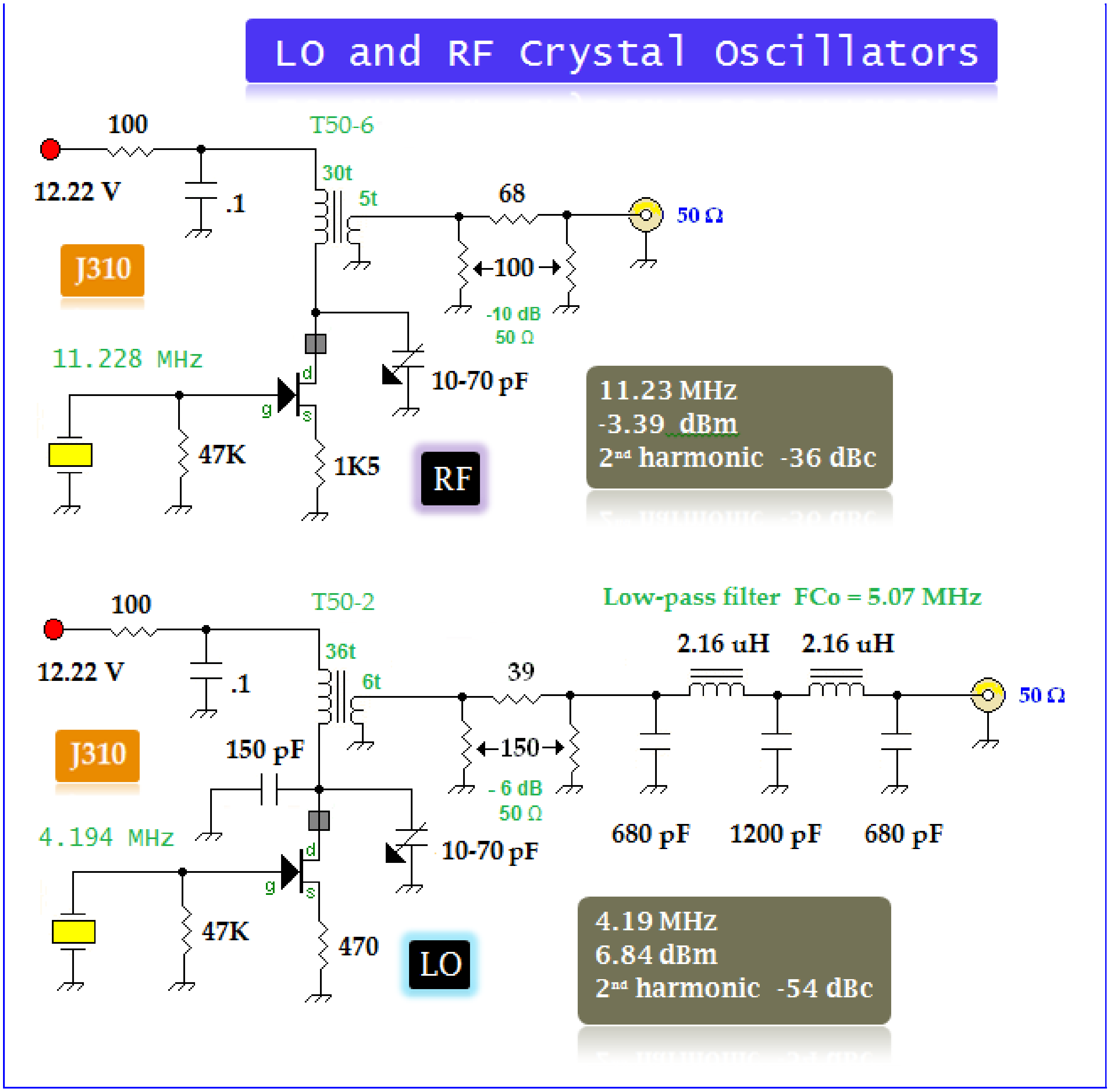


Above — The redesigned RF port oscillator. Clearly, the 7 element Chebychev low-pass filter isn't needed, so an N = 5 version was tried. Click for the spectrum analysis — again no harmonic energy was seen. In both the RF and LO signal generators, I tried to get as close to 7 dBm output power as possible.

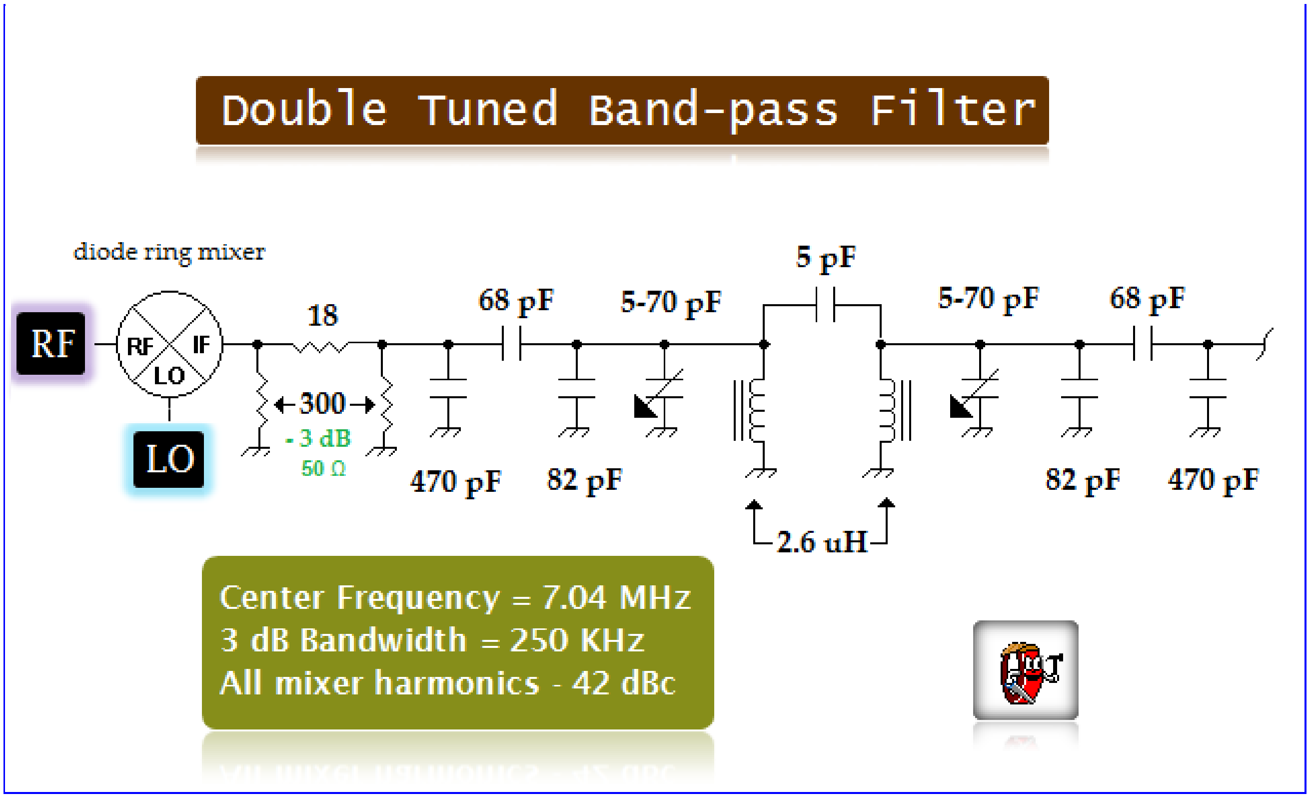
To operate this oscillator at the desired RF port signal level; for example, between 0 and -10 dBm, you might just attenuate the output with a fixed pad or step attenuator. My conclusions echo the work of others experimenters; lowering the RF port down from 7 dBm to as low as - 10 dBm, lowered the amplitude of the spurious mixer products seen in the spectrum analyzer. Click for a sample. Refer to the QST Technical Correspondence citation in the references section for more information.

Above — The 2 re-designed oscillators wired up and connected to an SBL-1 mixer. I actually connected the attenuator pads after the low-pass filters as explained later.

Two different crystal oscillators were then built:



Above — Two different crystal oscillators targeting ~ 7.034 MHz were built. Click for the breadboard photo. You can see the crystal frequencies in this photograph. The RF port oscillator power was set to -3.39 dBm by choosing a low value JFET source resistor and attaching a 10 dB attenuator pad. Relatively low harmonic distortion prompted the exclusion of a low-pass filter on the RF oscillator. The LO output power was ~ 7 dBm.



Above — The final experiment; placing a double tuned band-pass filter after the TUFF-1 diode ring mixer with the 2 latest crystal oscillators attached . This filter was in my junk box and I peaked it for the 7.034 MHz IF with a spectrum analyzer. The strongest spur was 42 dB down from the carrier — falling well short of the triple tuned band-pass filter presented before.

Clearly from all these experiments, a strong case for placing a triple tuned band-pass filter after a transmit mixer exists. If you use an unbalanced or single-balance mixer, a double balanced mixer might sufficiently not block feed through and spurious RF to keep your signal chain tidy. I enjoy studying the transmit chains of others to see how they filtered spurious and feed through RF. At the end of the day, as long as the output carrier spurs are low enough to meet your country's regulatory requirements, you're okay. Designing for low spurious emissions is an exciting challenge — one you'll miss if you don't try your hand with RF design.

A realization emerged following these experiments — I couldn't measure the return loss of the local oscillators! It technically could be done, but not by me. After 2 weeks of struggling, I engaged an American colleague with whom I occasionally build experimental circuits. After making some progress, we became stalled again. This time I asked Professor Ken Kuhn and Wes, W7ZOI for some ideas. Eventually a method to measure the RL of local oscillators came together along with enough material for another web page — RF Workbench 3.

When you do experiments, knowledge evolves as you go — for me, I learn mostly from making mistakes. I often think I should repeat most of my experiments over before presenting them, but this would consume too much time. However, footnotes can serve to steer readers for minor issues. If I had to re-build the crystal oscillators from Part 1, I'd build each crystal oscillator with a separate JFET buffer — then the return loss of the oscillator buffers could be measured as shown on RF Workbench 3 (with a shorted oscillator tank). Also, the pi attenuator pads on the crystal oscillators should follow the low-pass filters to garner the best output return loss. The good news is past experiments inform future experiments.

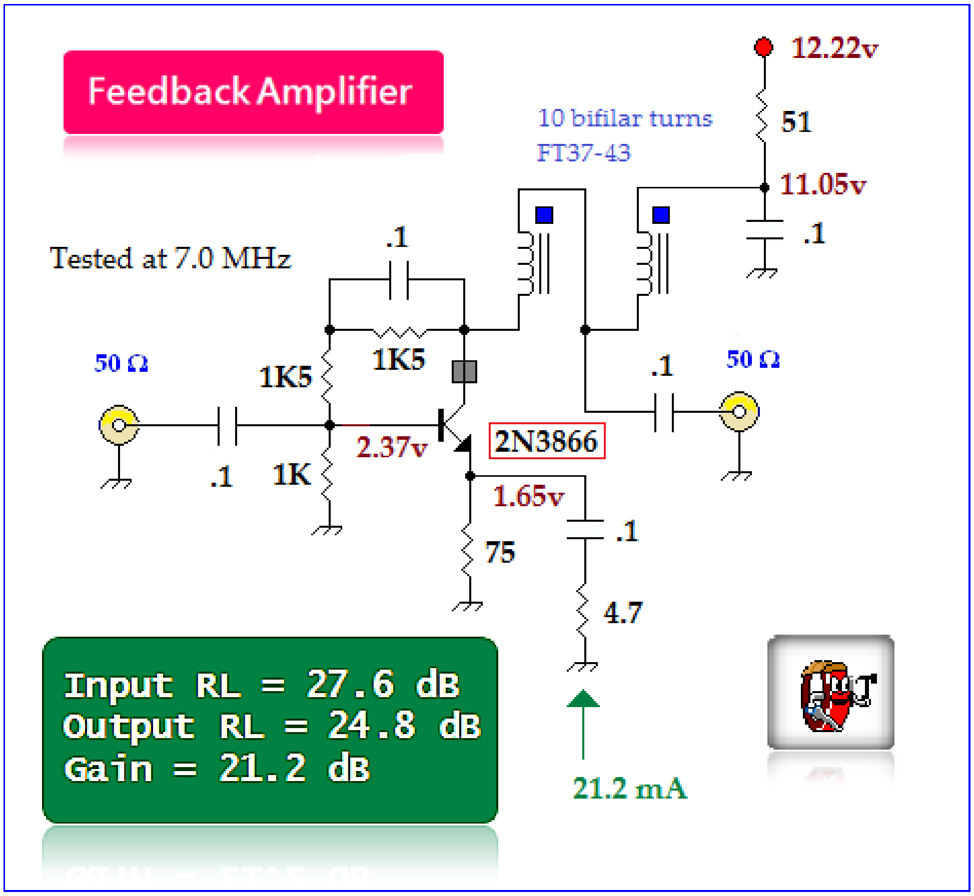
---

2. Bipolar Transistor Feedback Amplifier Experiments

I love making signals bigger — especially while preserving fidelity. It would be nice to become a reasonably competent amplifier designer — hopefully by studying sound schematics, applying software, building circuits and measuring evermore parameters this might occur. The mathematical equations of RF amplifier design seem quite daunting; they're the fodder of electrical engineers with their Hewlett Packard scientific calculators, SPICE software and GHz F-t transistors. With most things technical, as you try to advance, more questions than answers cross your mind; however, somehow this is normal and may actually signal progress.

Abundant amplifier references exist; for example, EMRDF Chapter 2: Feedback Amplifiers. This is essential reading and I won't repeat this information. Rather, I'll just share some ideas developed or reinforced on my bench. In the past, I've preferred amplifiers with narrow-band (tuned circuits) in an attempt to reduce distortion and maximize gain. Now after critically examining these tuned amps with scope and spectrum analyzer, I better appreciate the significant intrinsic feedback of RF transistors ***(the tendency to oscillate)*** and broadband designs are sought. Often you'll spend more time taming a tuned amplifier than building one.

This section focuses on return loss, bias techniques and achieving linear amplification — for example; finding ways to apply negative feedback, match the input, or how to set the collector voltage. All my experiments and thoughts about RF amplifiers are from an amateur designers' perspective and I welcome your feedback. The first amplifier shown is a classic W7ZOI topology that I call the "Beaverton Special".



Above — A classic feedback amplifier popularized by Wes, W7ZOI in books like Solid State Design for the Radio Amateur and EMRFD. My respect for this humble design increased after building and testing 4 different versions to get a feel for amps with both shunt and series feedback. Of the 4 built, this particular version became my favorite — providing excellent input and output matching without crazy high emitter current. Employing a low noise / high F-t 2N3866 transistor is icing on the cake — an attempt to maximize impedance matching and performance using this standard, fits most transistors bias/feedback circuit. The humble 2N3904 also worked well in this slot. You don't need the ferrite collector bead with a 2N3904.

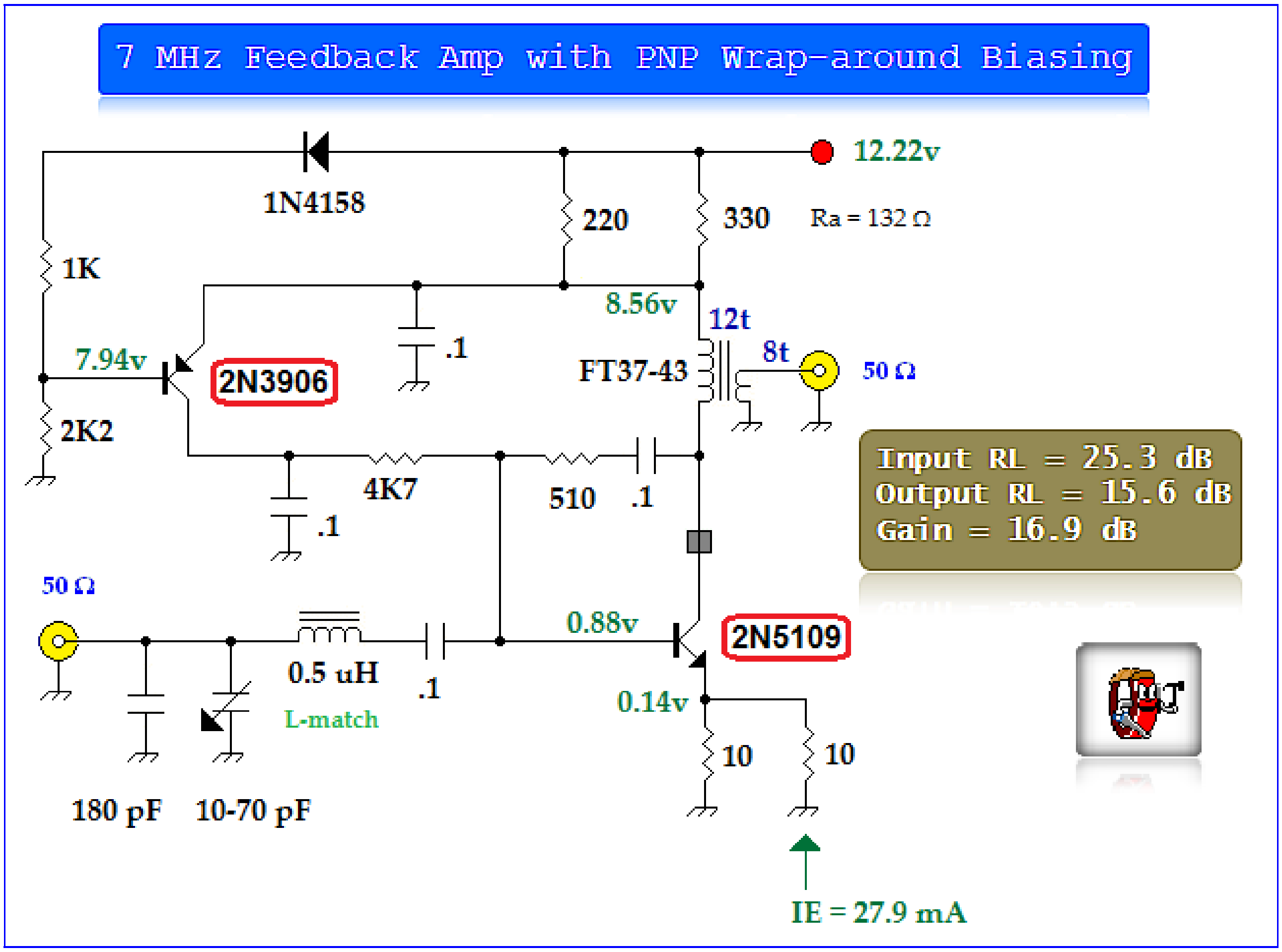
Other good experiments include trying different transistors and/or increasing the emitter current while being careful not to exceed the BJT's current rating (plus add heat sinking as required). You might also try the stage at different frequencies or perhaps sweep it to see at what frequency the gain starts to fall off.

Of the 4 BJT feedback amps shown in part 2, only this amp has a true broad-band input and output. What bothers me about broad-band linear amplifiers is that when you chain up 2 or more of them, signal fidelity generally degrades as it passes through each successive amp stage. Solutions include mopping things up with some low-pass filtering after the last stage, leaving it alone, or tuning the amplifiers (i.e. not using broadband stages).

The biggest caveat for feedback amps are variations in input and output impedance caused by source and load mismatches. For example, a 75 Ω resistor was connected to the output of the FBA above. The input return loss degraded to 16.8 dB. Further, the same 75 Ω load was removed and then connected to the input during output return loss measurement— this degraded the output RL to 23.4 dB. Clearly *load* mismatches upset return loss more than *source* mismatching. A 50 Ω attenuation pad should likely follow a feedback amp in situations where high input return loss are desired; for example, after a diode ring mixer.

Noticing a variation of the classis feedback topology in EMRFD Figure 6.140, I asked Wes, W7ZOI about it. It turns out there's another way to "skin the shunt feedback cat". The above RF amp uses a series connection of 2 feedback resistors (1K5 and 1K5 with a bypass cap across one 1K5). The result is a resistance at DC of 3K, but a resistance at RF of just 1K5. You could also use a 3K resistor directly from collector to base that is paralleled by a series connection of a 1K5 resistor plus a 0.1 uF capacitor. That network has the same impedance as my amp shown above. That is; the resistance would be 3K at DC, but 1K5 at RF.

This explanation fueled the next experiment — transistor amplifiers have 2 operating conditions; 1 at DC, the other at AC. Like a carpenter framing a house, you begin design by setting the DC bias — no small design task since bias concerns more than just establishing the base voltage and emitter current. For example, biasing may effect voltage gain, maximum signal handling capability, noise figure, impedance matching, class of operation, the operating point (sometimes called quiescent point or q-point), feedback and temperature stability. Biasing provides much to think about, however, a practical way to explore any topic is to chunk it into small, understandable pieces that become a stepping stone to advancement. Let's focus on biasing for temperature stability. The next amp uses the wrap-around PNP bias — an awesome technique.



Above — A 7 MHz FBA using PNP wrap-around biasing. I learned about wrap-around biasing from Wes, W7ZOI and share a simple way for new builders to also learn this technique as the **#1 Design Center** on the supplemental web page. Click for a prototype breadboard photograph. This amplifier employs heavy shunt feedback from collector to base. Degenerative (series) feedback from the 2 parallel 10 ohm resistors also enhances temperature stability.
Expanded bias circuit temperature stability discussion follows amplifier number 4.

The wrap-around or feedback bias scheme is good because it's self stabilizing. The diode in the PNP bias network further ensures that the PNP bias remains constant with temperature changes. It really should be to glued to the NPN transistor (or its heat sink) to allow tracking of the NPN's temperature variations. This bias circuit doesn't load the NPN base input impedance. Another great virtue is that the emitter of the amplifying transistor can be directly connected to ground allowing better performance at VHF and UHF.

Noise from the PNP will be amplified by the NPN, so the low-pass network formed by the 0.1 uF capacitor and 4K7 resistor is essential. In some related circuits, you may see an RF choke used instead of a resistor.

The actual 2N5109 input impedance is probably around 40 ohms — easy matching with an L-match network.

Amplifiers with an L-match tuned input shouldn't follow a diode ring mixer unless preceded with a diplexer since the narrow-band L-match tunes only 1 frequency. L-match networks can make an impedance bigger or smaller depending how they're oriented and also provide some low or high-pass filtering depending on the configuration. I design my L-networks on the bench using experience plus trial and error — a better way is to use software. I recommend the program called Zmat08.exe that is included on the CD that accompanies EMRFD. The software will get you close, however, bench tweaking is required since you're often matching a complex impedance, comprised in part, of stray reactance.

Setting up an L-Network for an Input Match

A suggested bench method for optimizing input Return Loss (RL) using an L-Match network.

Your task is find the "perfect" L and C values to get a RL of 20 dB or higher. Start by soldering in an inductor calculated from Zmat08.exe or according to your wisdom. Set up the amplifier for input return loss measurement. The first chore is to find the nominal target capacitance that provides the best match at the design frequency. I use a big range, air variable capacitor for this — with the input circuit connected to a return loss bridge, connect up and tune the big variable capacitor to give the greatest RL. Remove the variable capacitor, measure it, and then solder in an equivalent trimmer capacitor, and as required, fixed capacitor(s) so you can tune at least 25 pF above and below the target capacitance. Often, the target C will be close to whatever the software recommends. In amplifier 2, my C values are the 180 pF + a 10-70 pF trimmer.

Next, determine the optimal inductor. On my bench, I keep a variety of pre-wound #6; and #2 material powdered-iron toroid inductors and choose one close to the calculated or a self-chosen L value. I start with an inductor wound with 4-5 more turns than needed. After soldering it in, the RL is checked. Remove 1 or 2 turns, tweak the trimmer capacitor and again check the RL. If after removing 1 or 2 turns, the RL is going up, you've determined there was enough inductance to get the best RL. *(If the RL goes down, you probably didn't start with enough L to get the best possible RL).*

You can also also squeeze together or spread apart the toroid windings to vary inductance — the maximal inductance variation varies due to factors including wire gauge and total turns. Compressing the windings with thumb and forefinger *increases* the inductance and widening the gaps between windings *reduces* inductance on a toroid.

Assuming the RL increased after removing 1 or 2 turns, remove another turn, tweak the trimmer capacitor and check the return loss, and so on. Repeat until your return loss starts to decrease. Then add back a turn or 2 to find the absolute best match. *This procedure allows you to find the optimum inductance in-situ.* Once, you've figured out the best inductance, cut the inductor leads short, solder it in, tweak the trimmer capacitor, and then consider further tweaking the coil by expanding or squishing the windings on the toroid while looking at the RL in a bridge detector.

In summary, to get the best possible RL — design a prospective L-match with software, and then bench test to determine the optimal in-situ L and C by using values above and below the calculated L and C values while observing the results in a return loss bridge. This method seems tedious, but emphasizes that repeated bench practice and patience pays off. You can always just use the calculated L-network values and/or develop your own method to set them up.

Consider mitigating the stray inductance caused by the long lead that occurs after removing wire turns by cutting the lead and scraping off the enamel insulation every couple of turns or so. This is a gamble — If you cut the lead and need to add back a turn, you'll have to rewind the coil from scratch, or add in and solder another turn (messy). I'm often able to get a L-network RL of 22-26 dB using my method and feel it's worth the the time and effort.

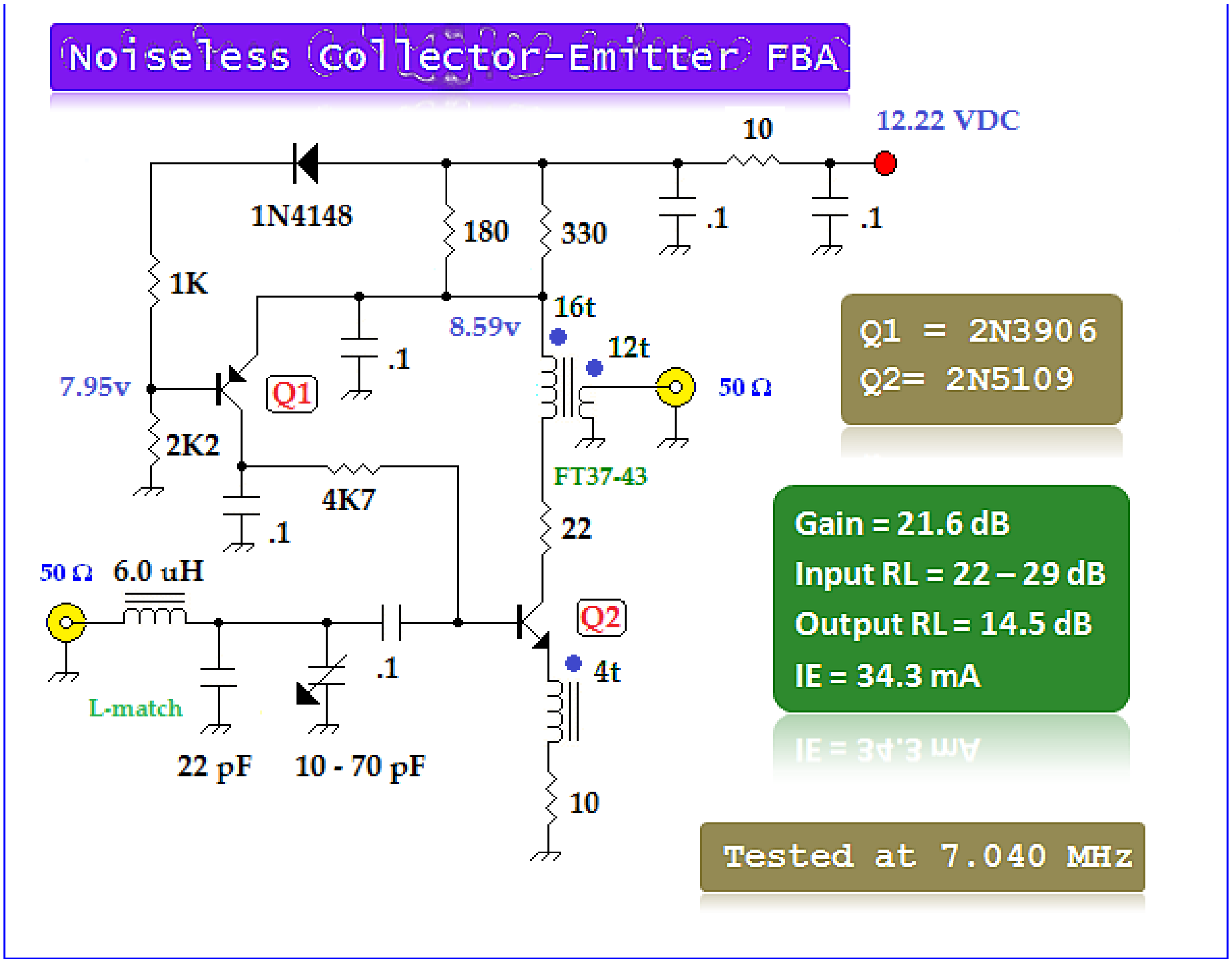
When bench tweaking the L and C values, your actually looking at the peak-to-peak AC voltage with the amp input connected to the unknown port of the RL bridge. Tune for the lowest, stable peak-to-peak voltage. Test it against the open circuit peak-to-peak voltage to calculate the RL. Since the open circuit doesn't change, you know the return loss is improving when the peak-to-peak voltage of the amplifier under test is going *down*. I store the open circuit voltage in my scientific calculator and calculate the RL from time to time as I'm tweaking the L and C values. After awhile, the whole procedure becomes automatic and quick. Once you 're done and everything's tidy, measure the open circuit and connected amp peak-to-peak voltages and calculate RL a final time. This is your reportable return loss.

You can scale matching networks from other builder's schematics by calculating the XL and XL and then applying these reactances to your desired frequency. Bench tweaking is still required. I also hope the person whom I'm copying didn't make a bench or drafting error. Be discerning about whatever your find on the Internet "Misinformation Highway" — this site included. Although I'm no philosopher, I know at least 3 things about people. They: 1. are often biased; 2. can lie; and 3. can make errors.

Above —Some toroids and the *air variable capacitor* I sometimes use to **coarsely** bench tune L-C circuits to determine the "ballpark" tuning capacitance. This capacitor features built-in reduction drive and varies from 15pF to 428 pF. When using an external capacitor connected to your circuit with short copper wires, expect some signal distortion and watch out for hand and body caused capacitance variations. The connecting wires also have reactance which won't be there when you swap in a small trimmer cap plus any fixed value capacitors.

Next up is a common emitter amp using "noiseless" feedback - this means the AC feedback is achieved with transformers instead of "noisy" resistors.

3



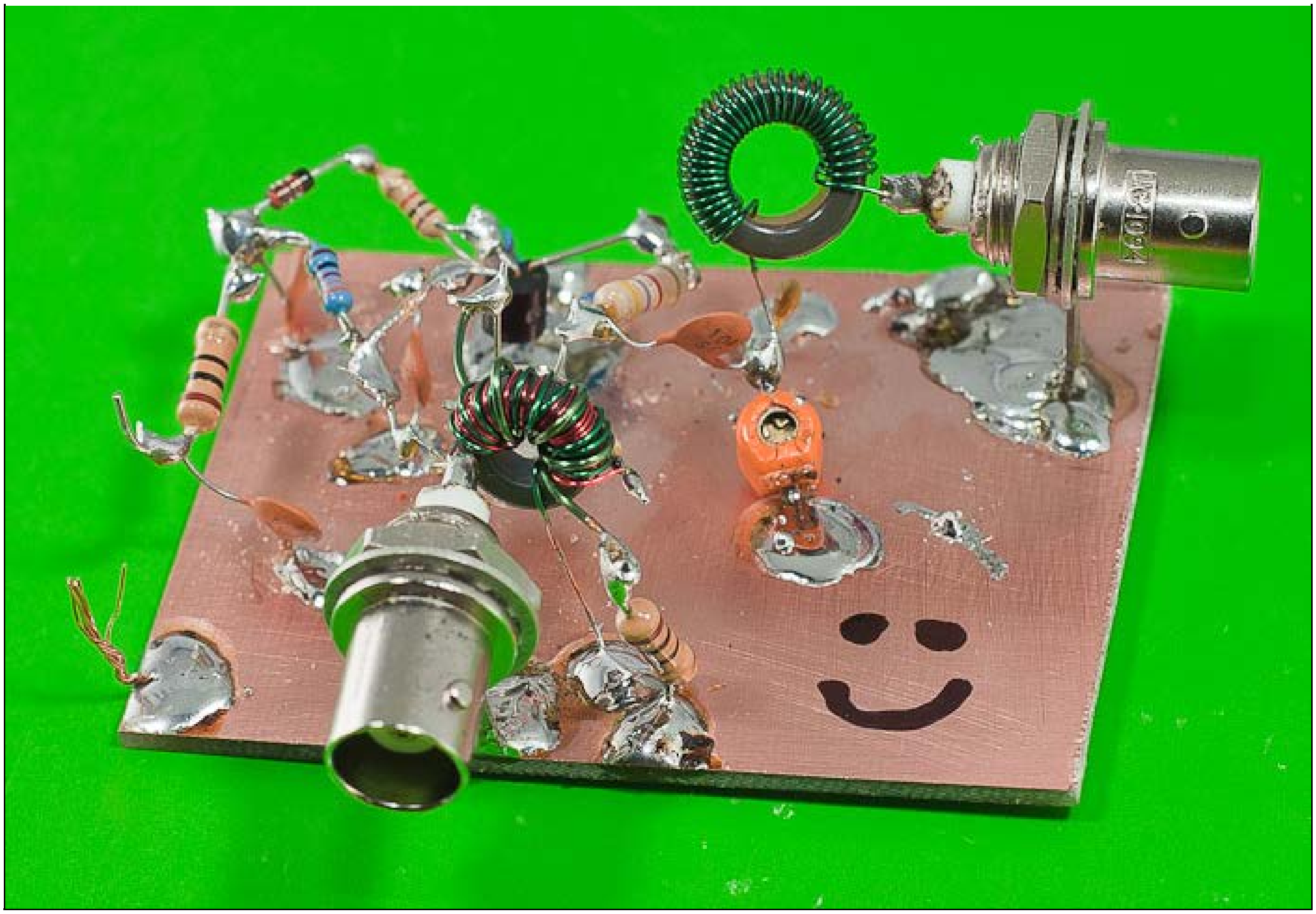
Above — Schematic of a 7 MHz collector-emitter "Griffiths" feedback amp. I ran substantial emitter current through this NPN. RC = 116.5 ohms — I paralleled 2 resistors for RC because I lack resistors between 100 and 150 ohms. The basic design is by Bruce Griffiths, who has a great web site. I thought I put up *big* schematics!

The input L-network was designed on the bench and provides a good input match peaked at 7.040 MHz — this pumped up the gain 3-4 dB. In my amp, a T50-2 powdered iron toroid inductor forms the L-match coil. Matching for the best possible input return loss is touchy and best done on the bench. For example, if the 6 uH inductor is decreased to 5.8 uH, the match could fall by 2-4 dB. With patience and careful tweaking return losses approaching 29 dB are possible, but likely too time consuming for most builders. The procedure as described earlier is pragmatic: connect a RL bridge to the input and adjust the L and C values until the lowest return loss is discovered. Even squishing or expanding the toroidal inductor windings can squeeze out a final dB or so of input matching.

Output matching proved interesting. Although I tried, the best output RL I could muster was 14.5 dB. Lowering the 10 Ω degeneration resistor or increasing the current could increase the output return loss. An output attenuator pad might be considered — a 3 - 6 dB pad would increase the output RL to over 20 dB.

All 3 output transformer windings were wound on a FT37-43 with care to keep the phasing correct. Amplifier gain is not dependent on collector current. For example, substituting an Ra of 180 ohms (clipping out the 330 Ω resistor) yielded a gain of 19.5 dB, an emitter current of ~ 20 mA and an output return loss of 12.6 dB, while the input match changed very little.

The oscillation snuffer 22 Ω collector resistor was 15 ohms in another version, however, parasitic oscillations were discovered at ~175 MHz and snuffed out by raising this resistor from 15 to 22 ohms. I sometimes go as high as 51 Ω; especially in JFET circuits.

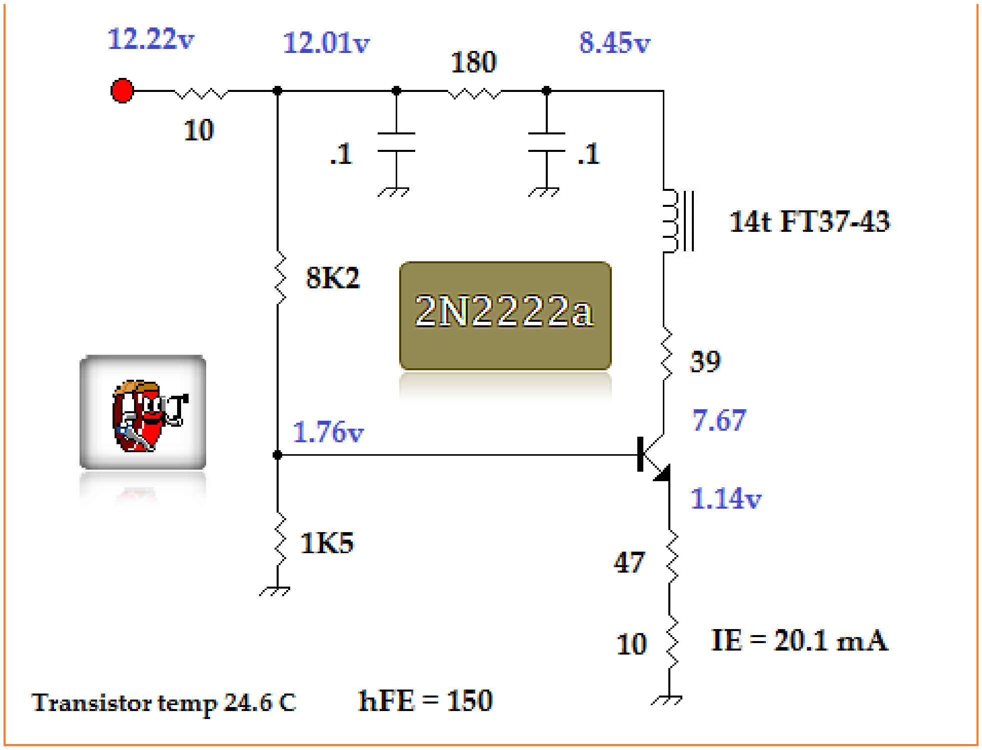


Above — The breadboard of the noiseless collector-emitter 7 MHz feedback amp. Click for a photo of another version. The hot "modern" replacement for the 2N5109 is this SMT part. I also like the BFG135 T/R BJT.

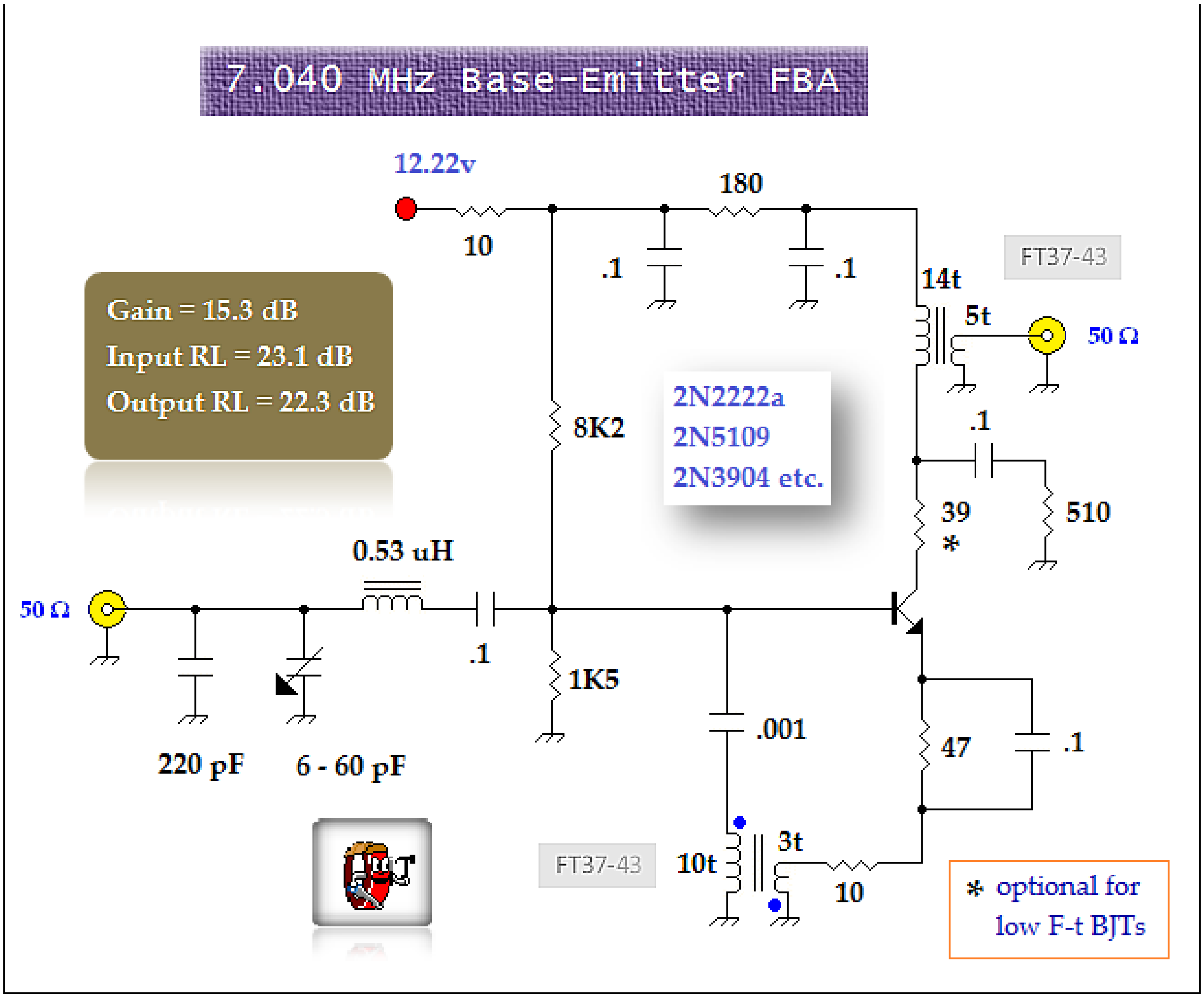
The final FBA experiments *below* use a standard voltage divider bias, tweaked for temperature compensation. The AC feedback is base to emitter — a rarely used topology in North America; although I'm not sure why.

4

Base-Emitter FBA -- DC Bias Using a 2N2222a



Above — The DC bias resistor values for a 2N2222a with a DC Beta or hFE of 150 and a emitter current of 20.1 mA. Almost every text author writes about voltage divider bias temperature stability, but some builders get bogged down in the details. Since the bipolar junction transistor is a voltage controlled device *(see section 4: QRP-POSDATA for an explanation)*, you must set up some DC voltages — I created a design center presenting an easy approach to design reasonably temperature stable BJT amps. See **#5 Design Center** on the supplemental web page. After getting the bias, the AC parts were added, and the completed schematic is shown below.



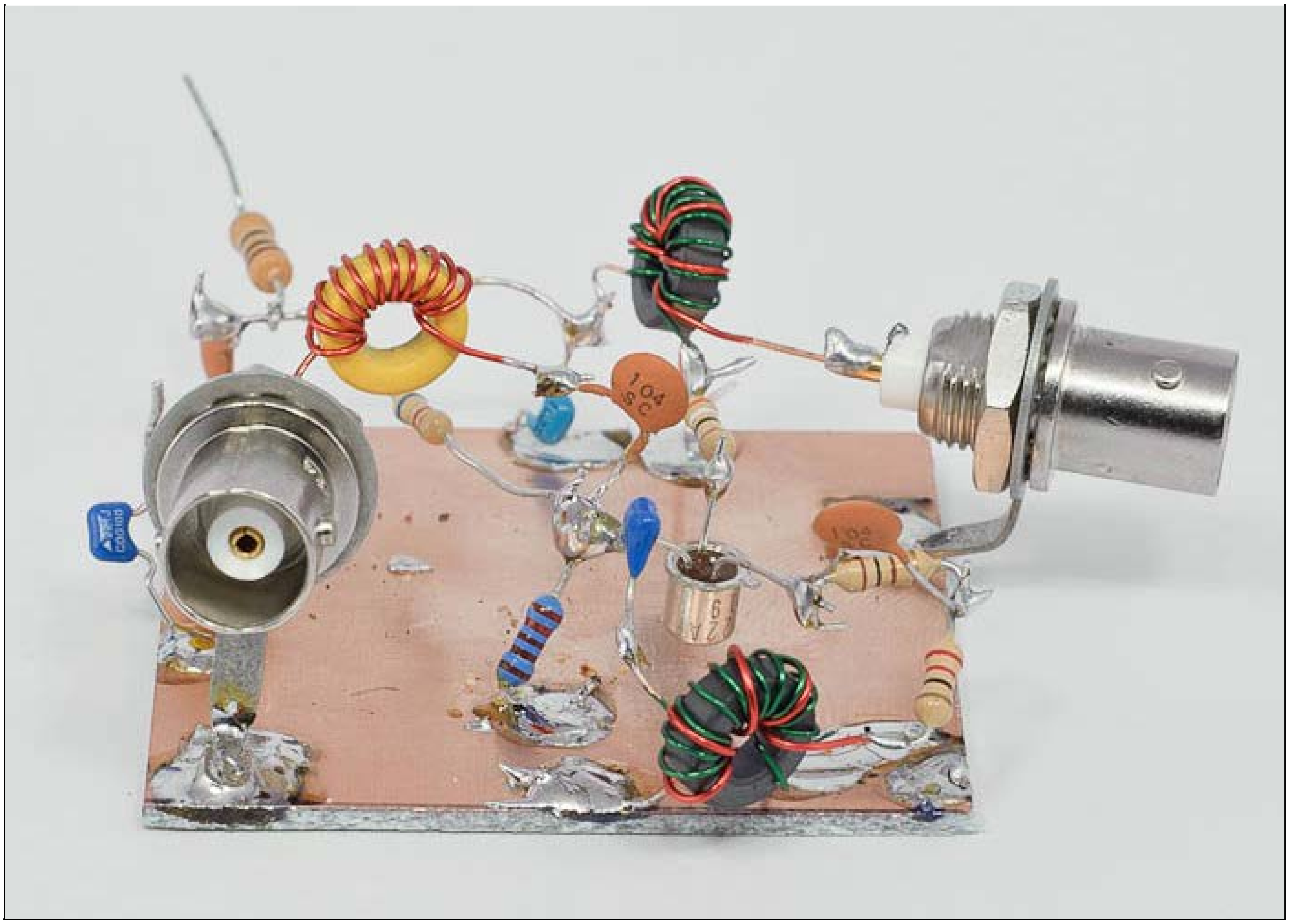
Above — A base-emitter feedback amp built Dec 21, 2010. I read about base-emitter feedback in Dr. Rohde's book *(Reference 1)*. He had some discussion, a small signal model and lots of difficult math, but no circuit examples. After searching on the web I found 1 example in the HBR-2000 transceiver; a project designed and built my respected Canadian colleague Marcus, VE7CA. Click for his web site. I decided to build my own design using a L-match to tune the input to 50 Ω.

The above amp was built around around a 2N2222a. The 39 ohm resistor is not really required with the 2N2222a. For high F-t transistors like the 2N3866, 2N5109 or microwave transistors, ferrite bead(s) or the resistor are *not* an option. Low F-t transistors like the 2N2222a or 2N3904 don't need the UHF oscillation snuffer resistor since they lack real gain at these frequencies. With the design center, you should be able to bias your own amp according to the emitter current you want — choose a BJT, measure or choose its hFE and then choose IE.

Missing from this web page is how to choose an operating point + discussion about DC load lines and related topics. I may tackle these topics on a future web page. I'm not sure anyone cares about this anymore.

The most difficult part was the output transformer. Lacking a base to collector connection, the collector impedance runs quite high and finding a good match into 50 Ω proved impossible — even with a shunt resistor across the primary coil. I saw a strategy in Marcus' amp; AC couple the collector to ground via a 510 Ω resistor. I did this. From then on, it was just trial and error to identify the optimum turns ratio for the collector transformer. An interesting experiment might be to figure out the turns ratio using a lower loss output transformer such as a FT-37-61.

The turns ratio of the various collector and drain transformers on most of these amplifier designs were determined by placing the amp in an output RL measurement setup and adding or removing secondary turns to get the highest possible RL. See the procedures for RL measurement on the RF Workbench pages.



Above — The breadboard of the first version of the base-emitter FBA.

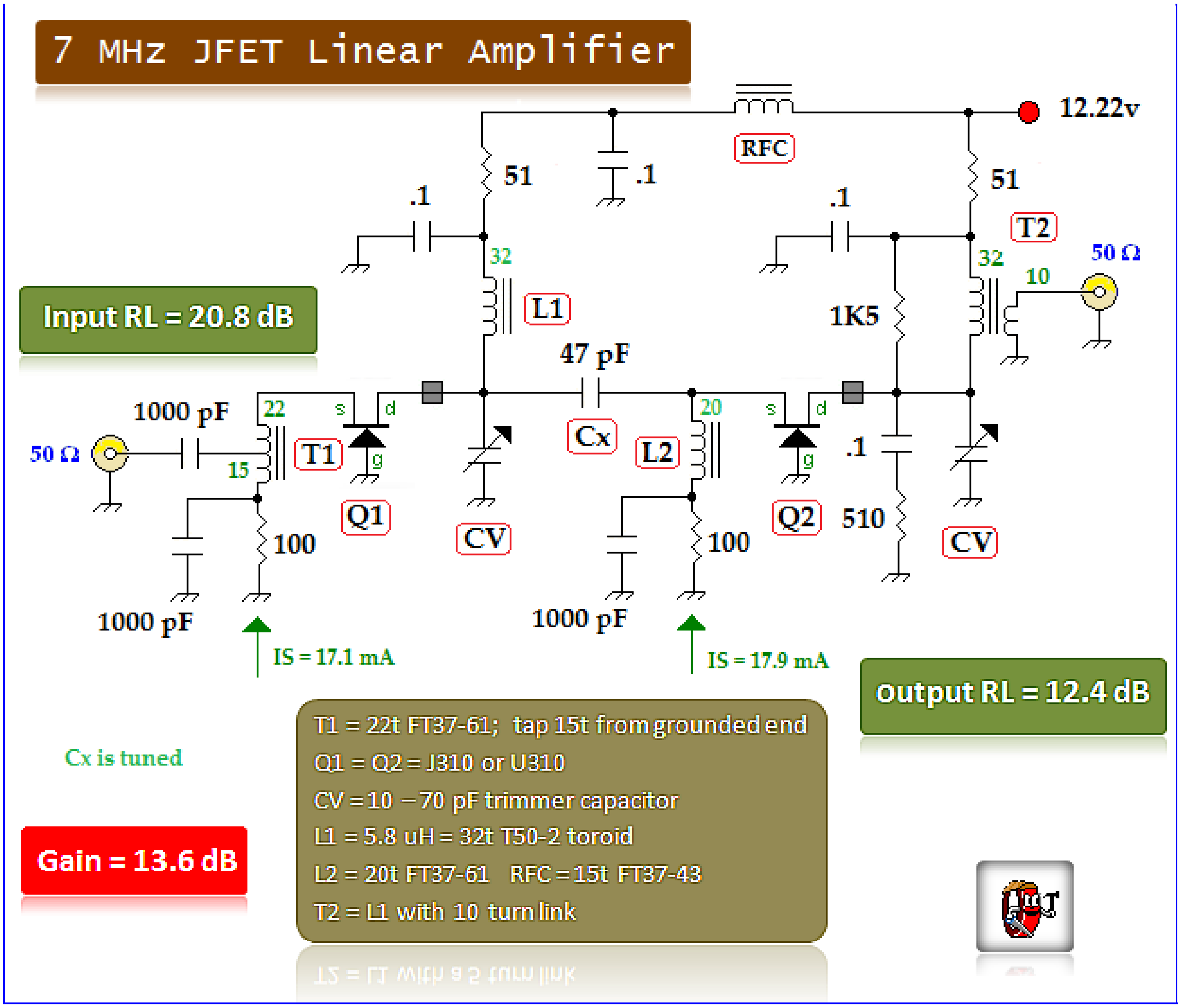
---

3. JFET Common Gate (CG)Transistor Amplifier Experiments

These experiments focus on setting up a desired input return loss and getting a reasonable output return loss in the CG amplifier. My expectation of an easy set of experiments proved wrong — assumptions never substitute for actually building and measuring.

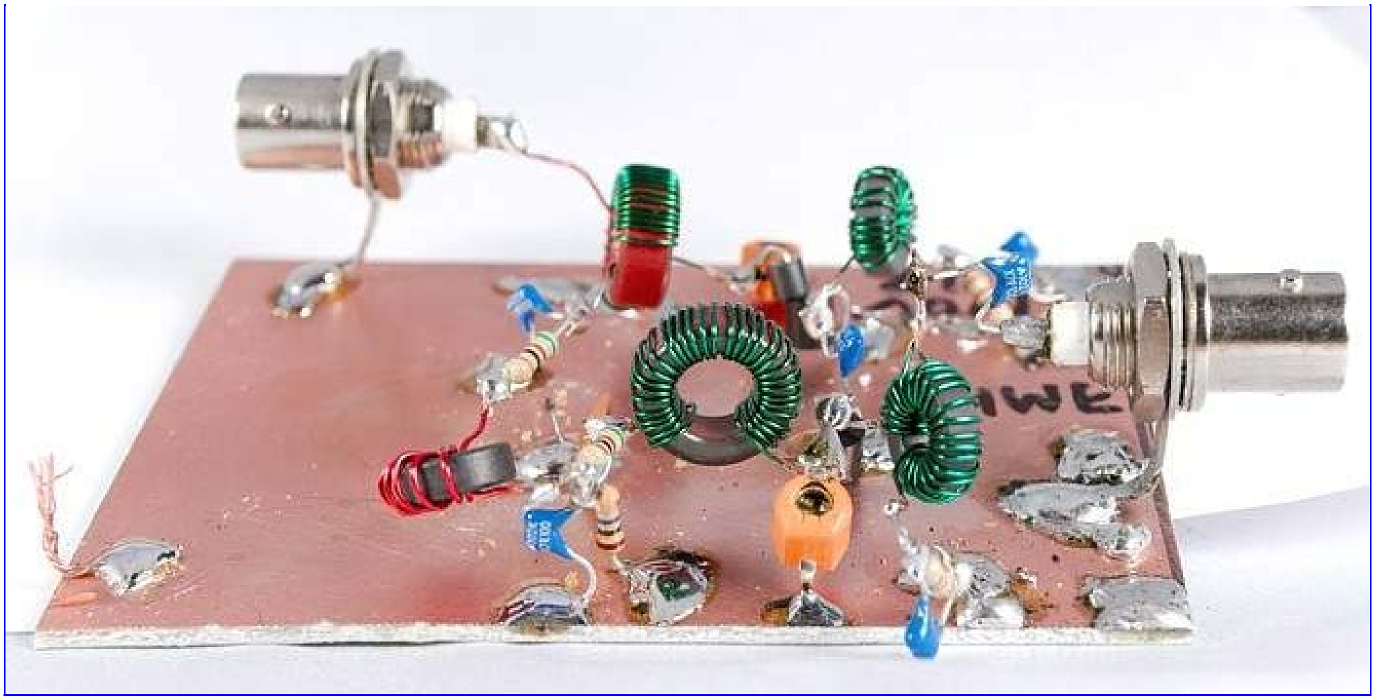
I like motorcycles. The difference between riding a motorbike versus *driving* a car parallels learning on the bench versus learning by just simulating or calculating component "ideal" values on paper or computer. In the car you're isolated from wind, smells, temperature changes and subtle road traction and camber differences that you fully sense on the motorcycle. Bench experiments prove equally visceral and experiential — the sensory input from learning as you build and test circuits imprints deeply in your mind.

1



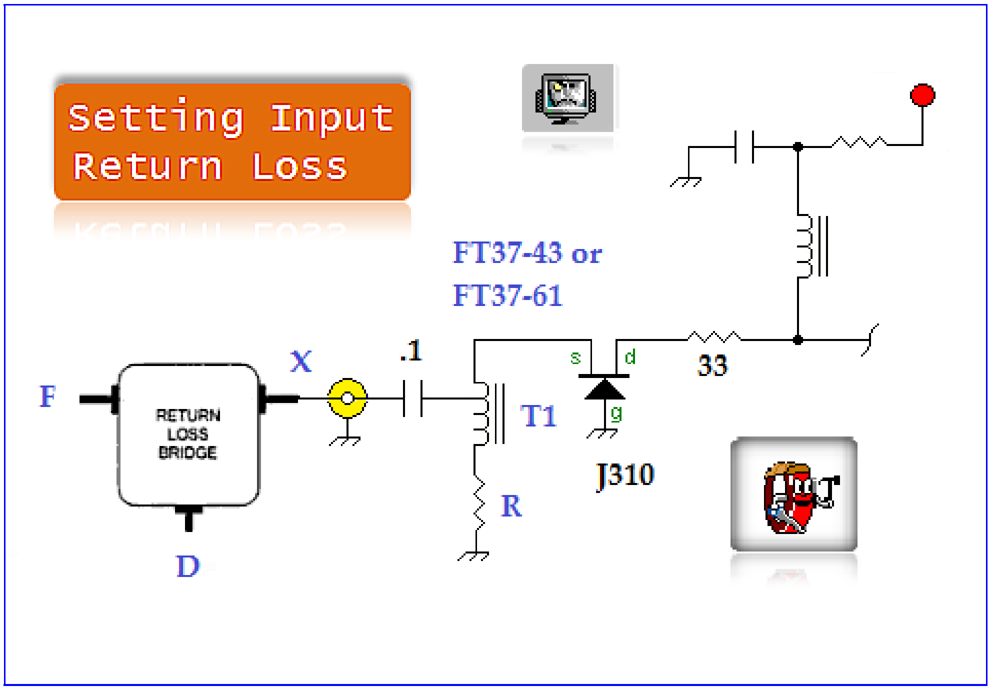
Above — A 7 MHz JFET "linear" amplifier built only for testing ideas — *do not build*. It went through several incarnations and prompted many experiments. The input return loss was deliberately set to 20.8 dB, although I set a RL from 10.0 - 28.6 dB during my experiments. Bypassing the JFET source resistor increases gain, but of course changes input RL.

The output transformer represents a terrible design, but shows the length I went to to try an obtain a decent output return loss. Working with this circuit, led me to abandon tuning the output transformer in situations where a high return loss was desired since the low value resistors required kill the tank Q significantly.



Above — Breadboard of 1 version of the prototype low-level JFET "linear" amplifiers for 7 MHz. Click Click . Cx is tuned with a variable cap and a nearest standard value substituted; in my case 46 pF was the measured value of the variable cap at point Cx.

Setting Input Impedance

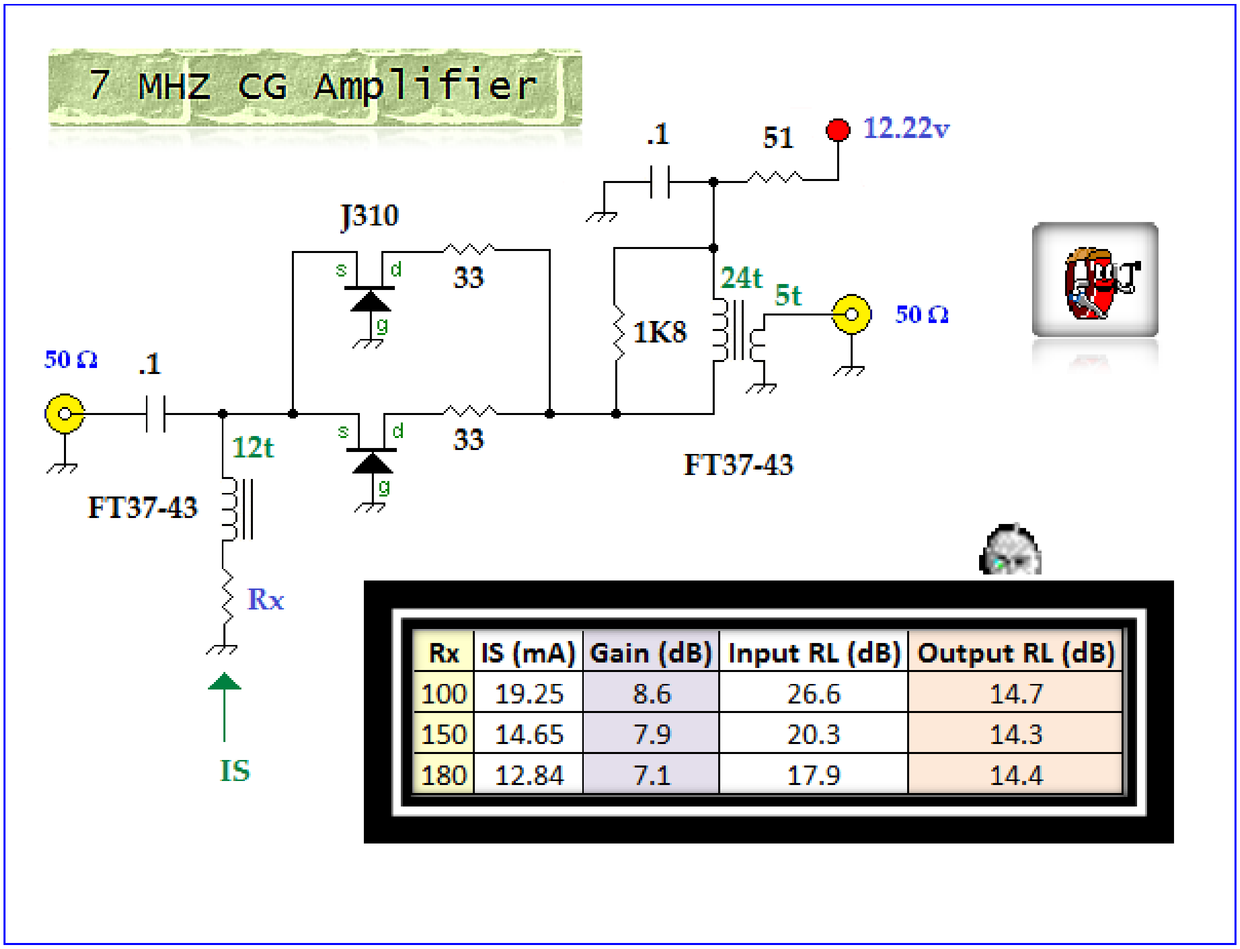


Above — The procedure used to set a desired CG amplifier input Return Loss. Numerous factors influence the input impedance and I discuss them in **#4 Some Factors Affecting Common-Gate Amplifier Input Impedance** on the supplemental web page. I keep some tapped inductors on my workbench such as this FT50-43 or these FT50-61 core inductors. To find the best return loss using such a coil, you can change tap points, remove windings and even wind more turns and solder the 1 end of your new windings to 1 end of the existing wire. Some builders omit inductor taps and manipulate the input return loss other ways as described in the supplemental article.

Normally we set the input match after establishing the output match since the output impedance dramatically affects the input impedance.

Further, you might notice that the tap point may vary between different JFETs. Most of my "real world" coils have at least 2 tap points and I choose the tap that gives the best return loss. More often than not, I bias for 14 - 18 mA and leave off the source bypass capacitor; it's your call.

The input return loss that gives the lowest noise figure is often chosen by engineers.



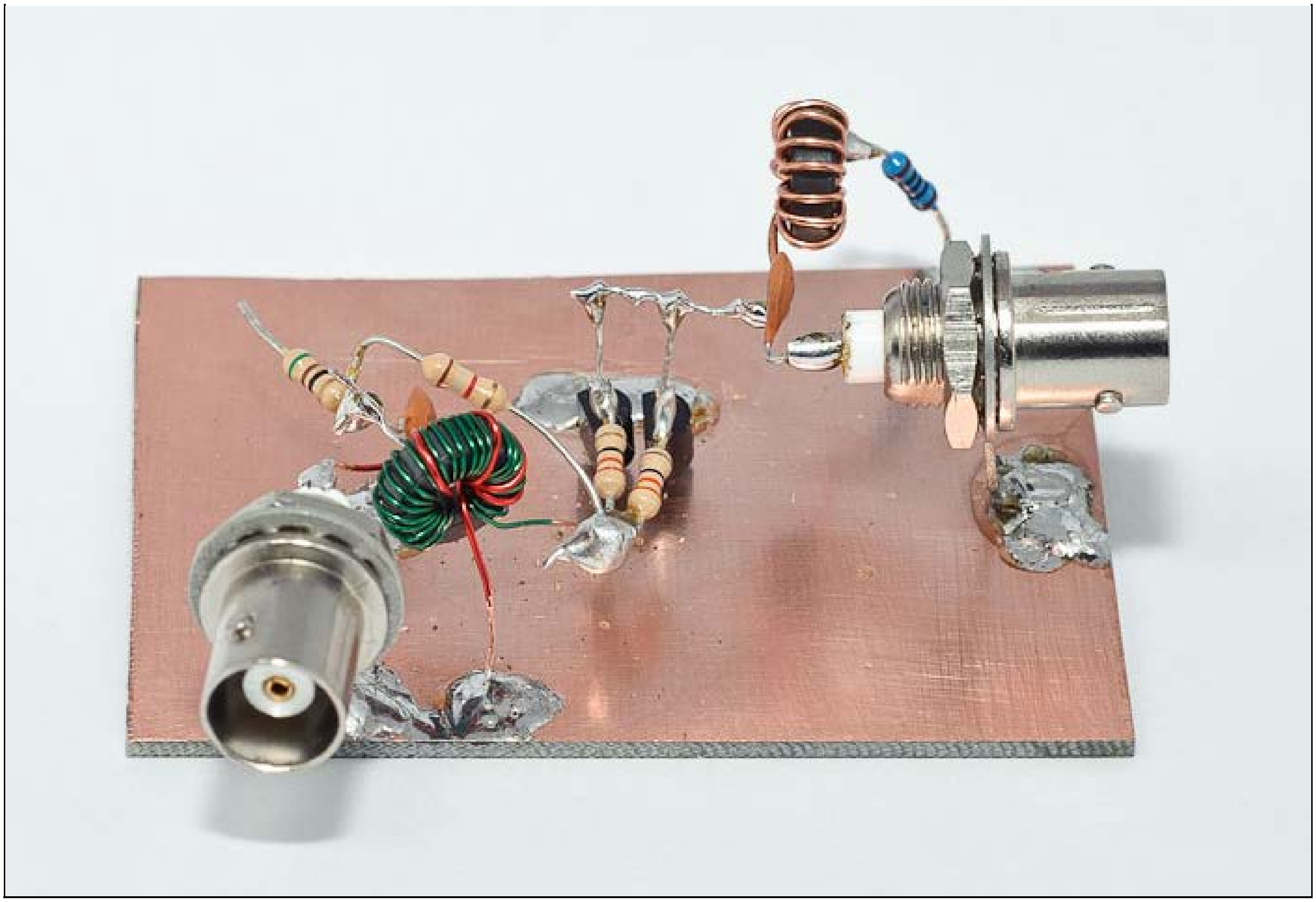
Rx	IS (mA)	Gain (dB)	Input RL (dB)	Output RL (dB)
100	19.25	8.6	26.6	14.7
150	14.65	7.9	20.3	14.3
180	12.84	7.1	17.9	14.4

Above — An experimental 7 MHz common gate amp designed to terminate a diode ring mixer. The best thing about using 2 JFETs is that you don't have to determine the tap point in the decoupling inductor (12 turns on a FT37-43 in this amplifier). I put up to 4 in parallel during my various experiments. It's faster to match just 2 JFETs, so 2 were favored.

The output RL wasn't great at ~ 14 dB, however is probably normal or better than most published amateur projects. I set the output match by adding a shunt 1K8 resistor across the primary winding and then finding the turns ratio to give the best output return loss. Without the resistor, the best output RL will be ~5 dB or worse. The resistor reduces power.

I learned that putting JFETs in parallel in a common gate amplifier reduced the output return loss in circuits using an output transformer like in the schematic above; this is unfortunate.

I wanted an output RL of 20 dB or greater — this is no small request; over a week was spent investigating transformer behavior and finding ways to improve output return loss when you really *want* to.



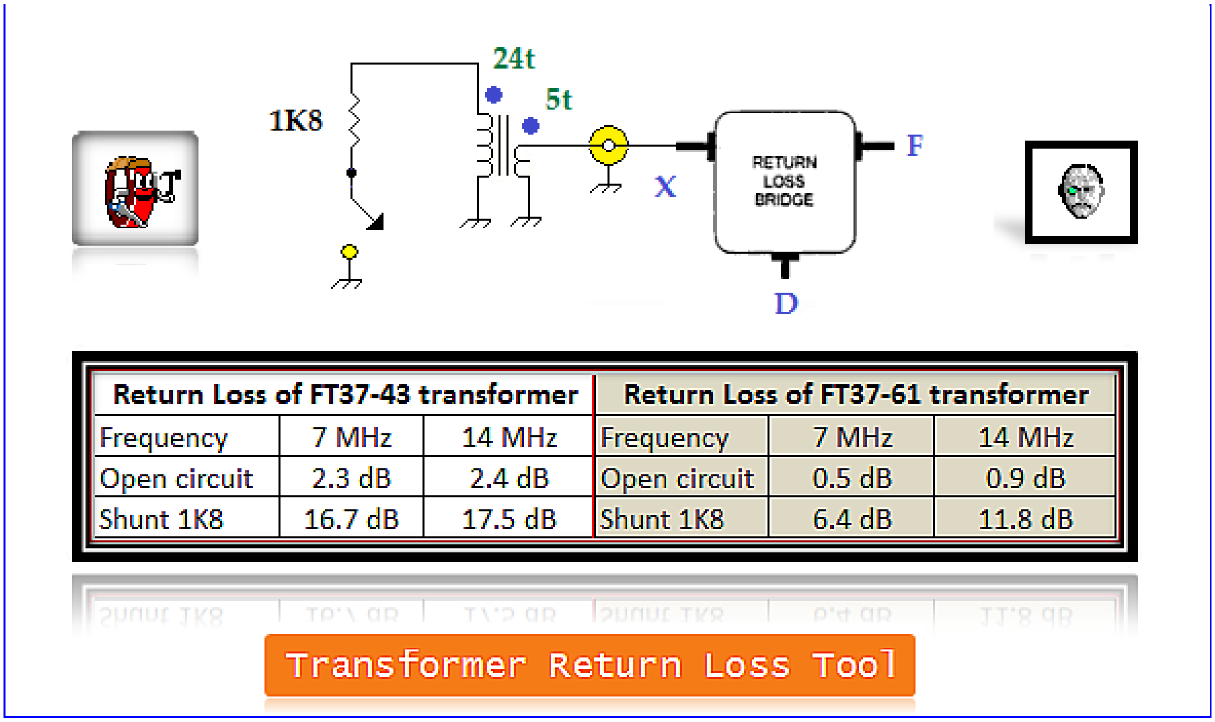
Above — The breadboard of the above 7 MHz CG amplifier.

3

Output Impedance Experiments

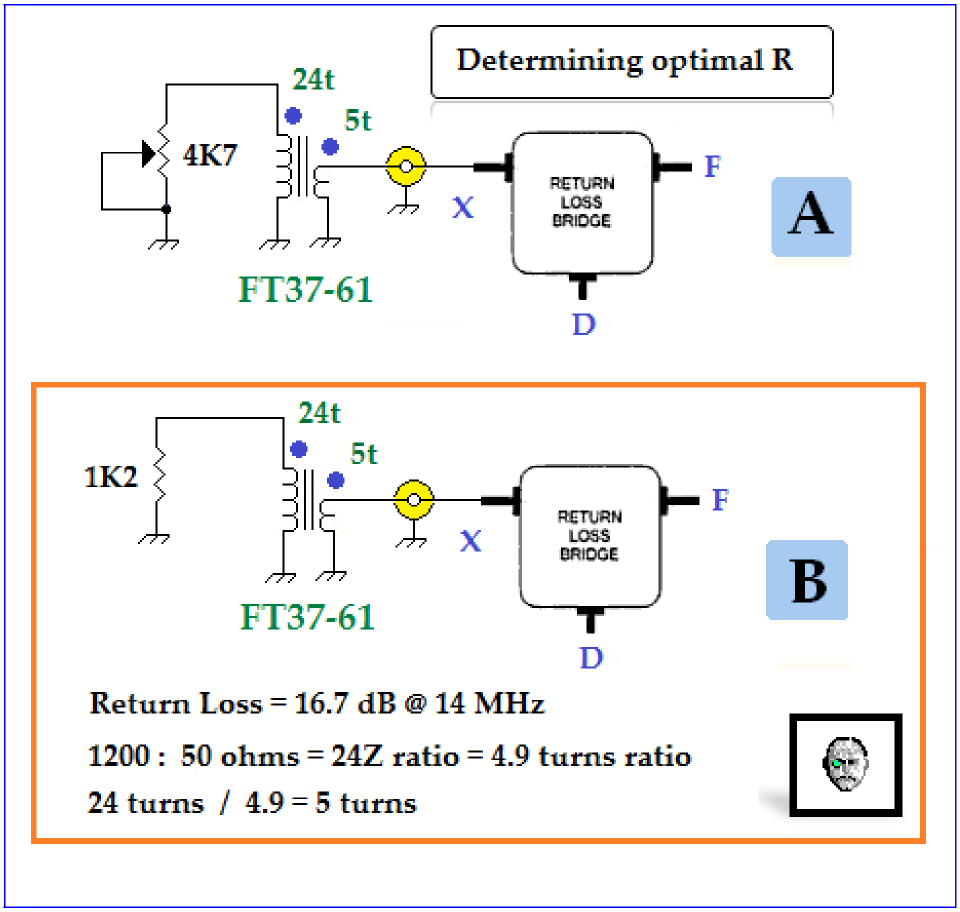
For some reason, I assumed that when using an arithmetically correct turns ratio, the output transformer will end up at 50 Ωs. For example, if I wish to transform 450 ohms to 50 ohms, I'd use a 9:1 impedance ratio (3:1 turns ratio) and get 50 ohms. Sadly, it isn't this simple — impedance transformation is complicated and whole books have been written about it. I'll share some of my experiments that might inform yours.

The first task was to built a simple jig to evaluate primary and secondary coupling, turns ratios and return loss.



Return Loss of FT37-43 transformer			Return Loss of FT37-61 transformer		
Frequency	7 MHz	14 MHz	Frequency	7 MHz	14 MHz
Open circuit	2.3 dB	2.4 dB	Open circuit	0.5 dB	0.9 dB
Shunt 1K8	16.7 dB	17.5 dB	Shunt 1K8	6.4 dB	11.8 dB

Above — The simple tool built to evaluate the return loss of a transformer out-of-circuit. In this case, I examined the 24t : 5t transformer of the 7 MHz CG amplifier shown earlier. The table shows the best possible return loss when the 1K8 resistor is across the primary coil. Additional experiments were completed and follow below.

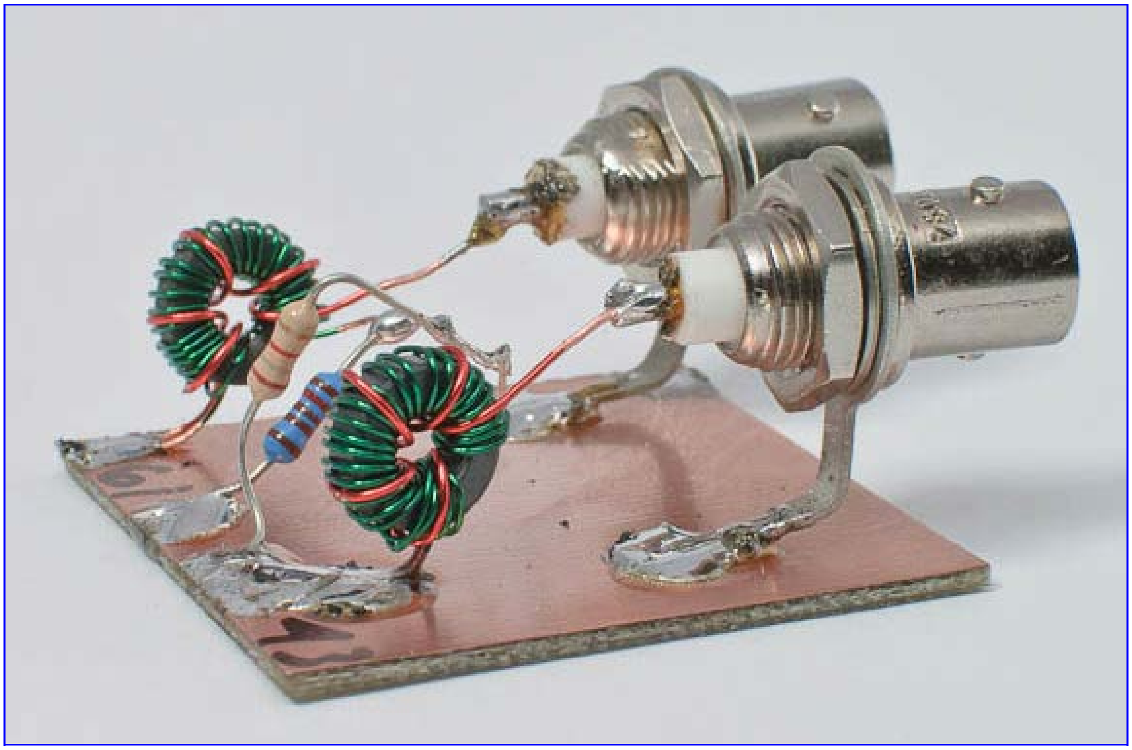


Return Loss = 16.7 dB @ 14 MHz

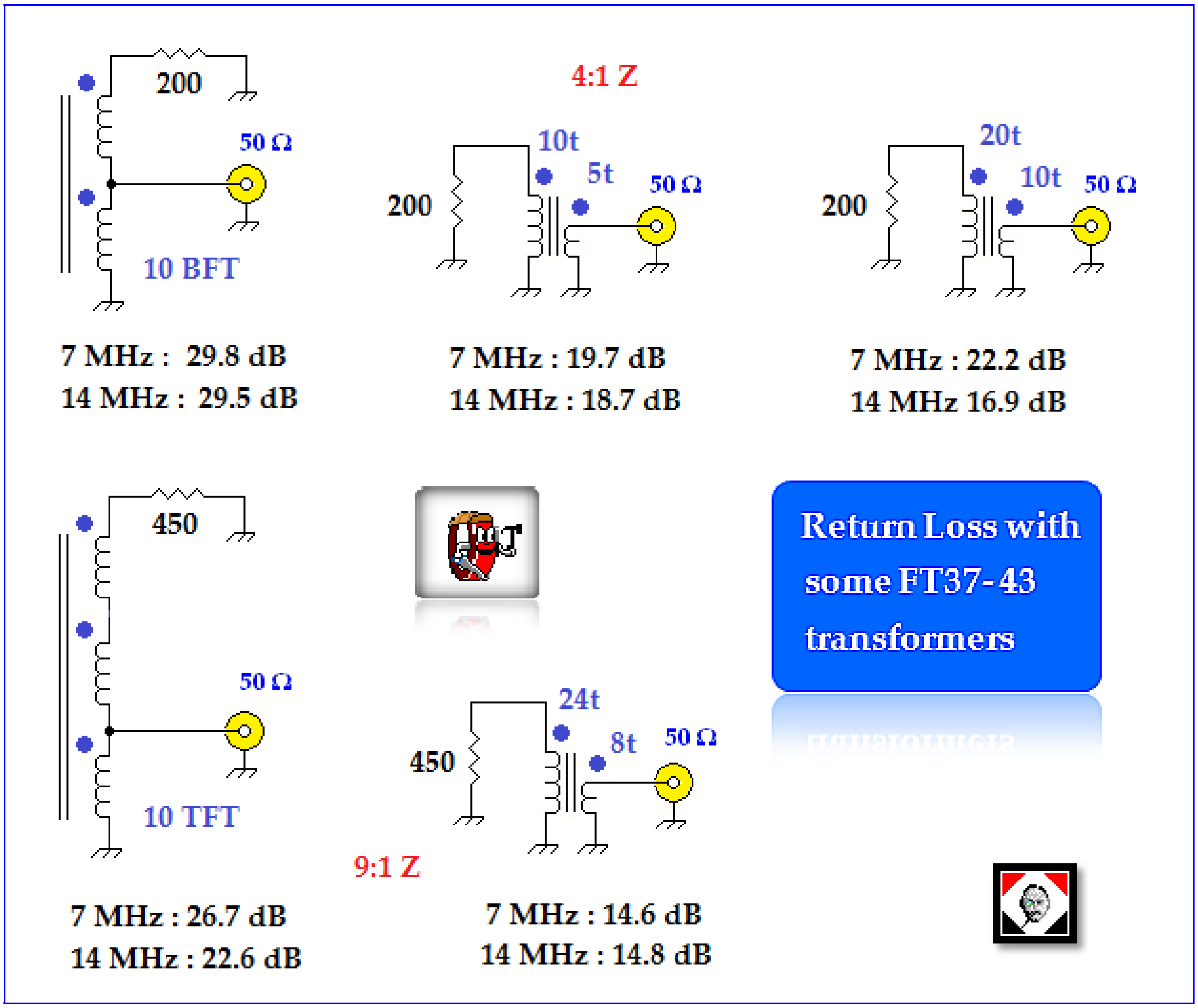
1200 : 50 ohms = 24Z ratio = 4.9 turns ratio

24 turns / 4.9 = 5 turns

Above — An experiment to see if changing the shunt resistor can improve return loss; yes it can. The shunt resistor was a 4K7 potentiometer — Using the potentiometer, I was able to determine the optimal resistance needed to increase the return loss @ 14 MHz of the **FT37-61** ferrite-base 24t : 5t transformer. The pot was removed, measured and replaced with the nearest standard value; a 1K2 resistor. The best possible RL was 16.7 dB using a 1K2 shunt resistor. At 7 MHz, the FT37-61 didn't work well. Five turns on a FT37-61 based transformer doesn't have enough inductive reactance to get a good return loss.

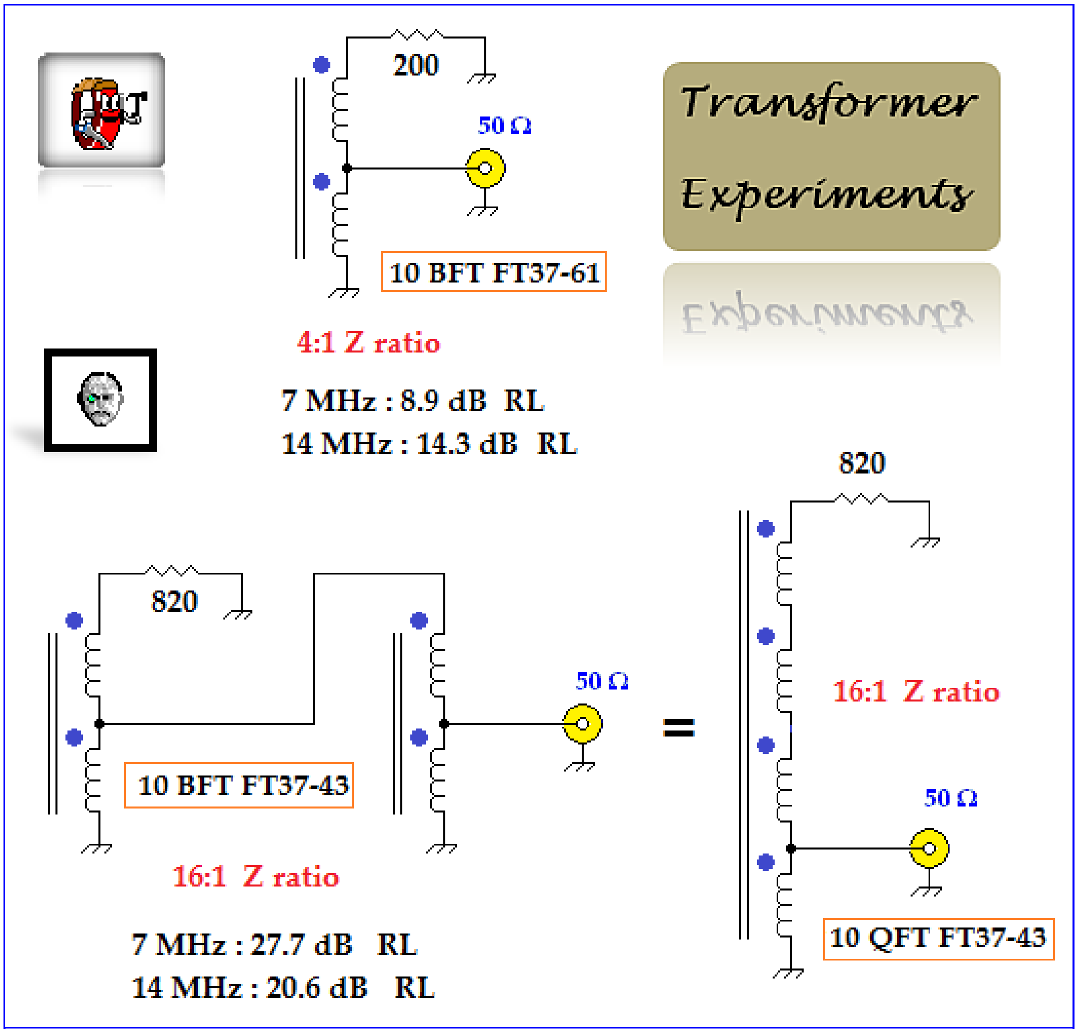


Above — The transformer testing jig. I omitted the switch shown in the schematic above and just soldered the shunt resistor across the primary winding.



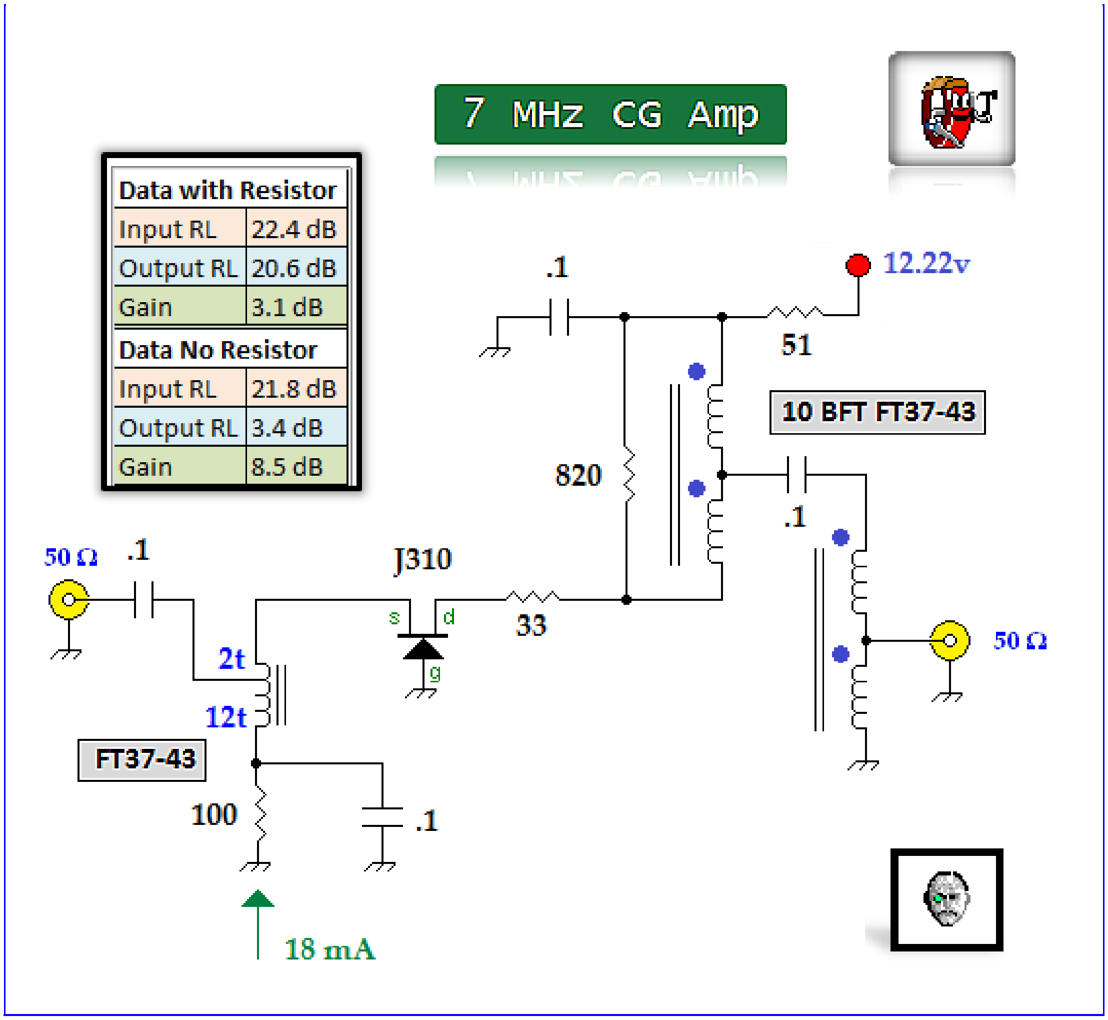
Above — Some of the outcomes using the transformer jig pictured above. While I basically understood that transformer efficiency tends to fall as the turns ratio increases, I never thought this would also happen with return loss. By *no* means do these crude experiments constitute science, but the following themes emerged:

1. The better coupling of transmission line transformers (bifilar, trifilar etc.) translates into improved RL over conventionally wound transformers
2. Limiting the turns ratio to 3:1 or less generally improved the return loss. As the turns ratio moves above 3:1, the best possible return loss tends to decrease.
3. The smaller or secondary winding should have 4-10 times the inductive reactance of the impedance it's connected to. For a 50 ohms impedance this means a minimal XL of 200 - 500 ohms. I noticed a weak trend towards better return loss with higher XLs. This means that to use a FT37-61 at 7 MHz, the secondary winding should be 9 -14 turns or so.



Above — Further transformer experiments. For a 4:1 impedance transformation at 7 and 14 MHz, a FT37-43 ferrite toroid gave a better out-of-circuit RL than the FT37-61 *The comparison transformer with a FT37-43 ferrite core was shown earlier*. It's possible to transform a big impedance such as 16:1 by cascading 2 bifilar transformers, or by using a quadrifilar transformer. I didn't build the quadrifilar transmission line transformer, but show it for completeness sake.

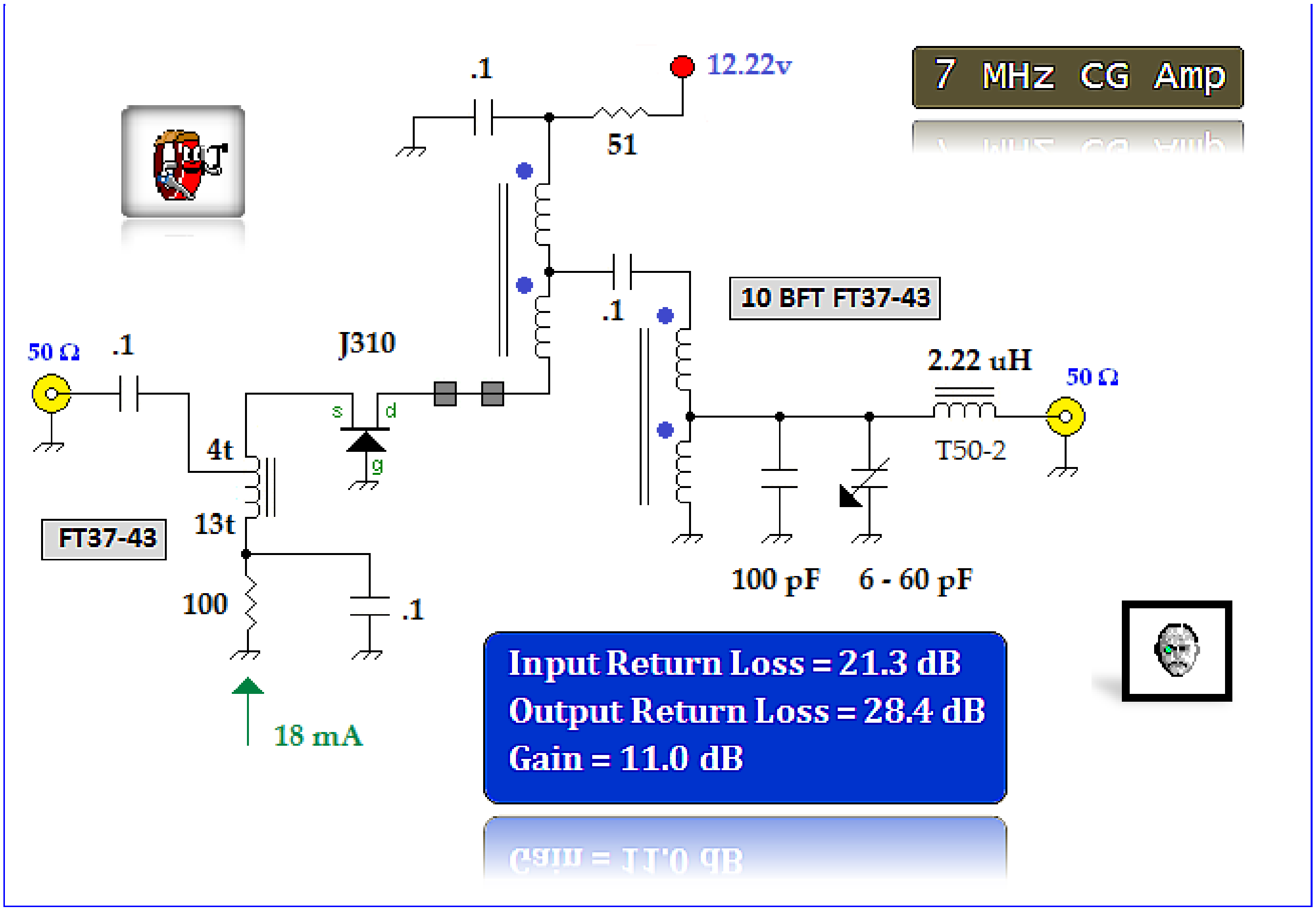
Of course, once you connect the transformers to a real circuit, things will change — still it's great to be able to examine transformer return loss in a controlled environment.



Data with Resistor	
Input RL	22.4 dB
Output RL	20.6 dB
Gain	3.1 dB
Data No Resistor	
Input RL	21.8 dB
Output RL	3.4 dB
Gain	8.5 dB

Above — a common gate amplifier experiment using 2 cascaded 4:1 Z transmission line transformers. Data with and without the 820 Ω resistor shows that while the resistor gives a great output RL; it eats a lot of power. In cases where I've seen cascaded transmission line transformers used, the resistor was omitted. The Ugly Weekender transmitter by Wes, W7ZOI provides a good example.

In many cases, it's prudent to sacrifice gain for return loss, however, when you see a builder (like the former me), put a 32:3 turns ratio on a 5 MHz amplifier output transformer and label the secondary windings **"50 Ω",** we'll know better.



Above — An evolution of the amplifier above to get the best possible output RL. I omitted the 820 ohm resistor and matched the output with an L-network. The return loss on the output of the second transmission line transformer (measured before the L-match was added) was 3.4 dB.

The L-match values were *roughly* determined by using this chart (you can also do the math). According to the chart an *(output)* RL of 3.4 dB, is either 10 ohms or ~250 ohms or so. Ten ohms is unlikely, so I designed my L-match to match 250 to 50 ohms. This provided some starting values for the L and C parts and the rest was done on the bench using trial and error with an RL bridge. At the time, this was the highest output RL I'd ever achieved.

RF engineers use math to calculate impedance (they always do). I sent the schematic to Wes, W7ZOI for his analysis and summarize his return email comments as follows: At 7 MHz, the XL of the 2.22uH inductor is 97.6 Ω, therefore the impedance looking into that with 50 Ohms as the load is 50+j97.6. A complex inversion of this value gives a complex admittance that has a real part: 0.0041. Flipping that gives 240 Ω. The equivalent reactance is inductive with a value that would be tuned by a 184 pF capacitor; a bit more than you have there — so there is some reactance presented by the center tap of the second transformer. Neglecting these details, the L net generates about 240 Ohms. The two transformers then kick the Z up by 16 to 3856 Ohms.

I was pleased that my simple chart gave a value close to his calculation. Test it out — the chart may work okay for you.

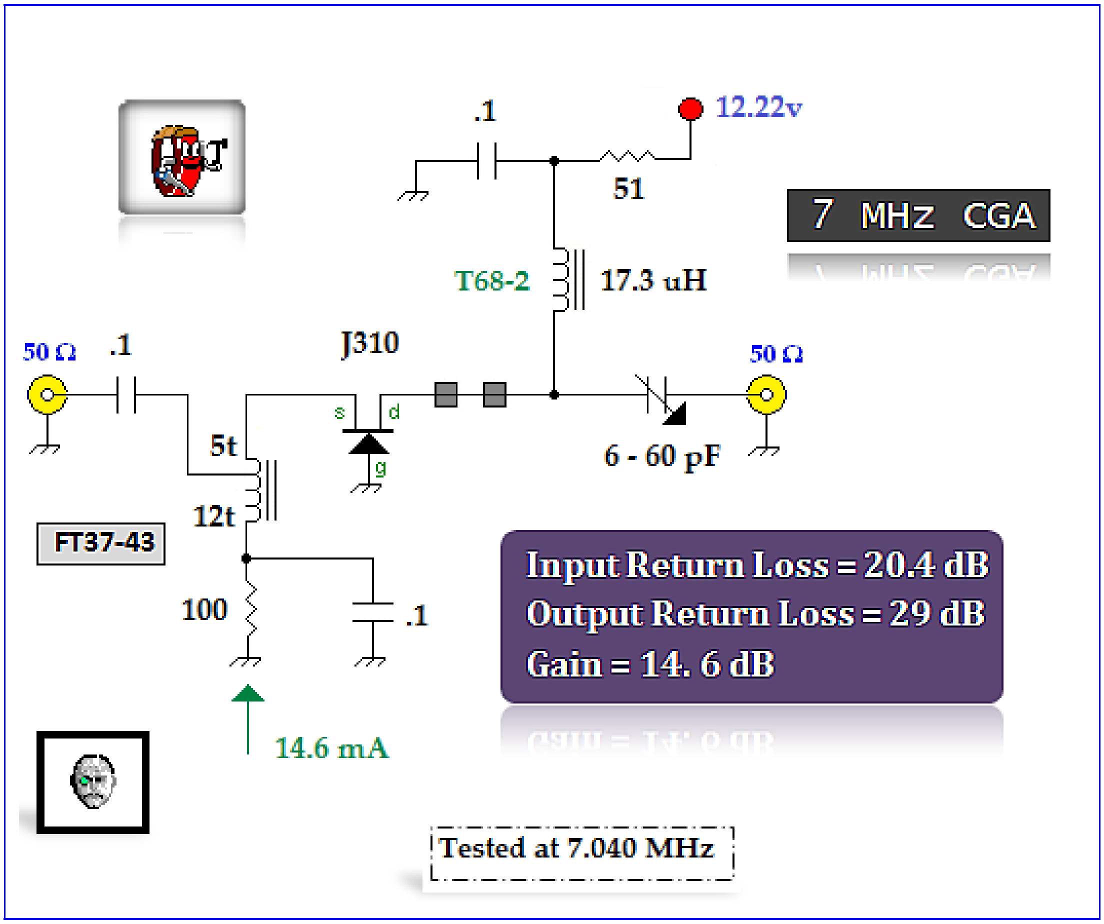
The input match is over 20 dB and reasonable. More time could have been spent on the input autotransformer by tapping and such to increase the input RL, however, time is the 1 resource we all seem to lack.

4

The final amplifier experiments employ an L-match to set output return loss. When reading electrical engineering books you'll often see all sorts of matching networks on both the input and output of FETs and BJT amplifiers. The networks look simple, but in practice, aren't. They tune sharply, have a low bandwidth and in the case of the CG amp, harbor a big problem — tuning the output for the best output return loss, **dramatically** affects the *input* return loss and potentially, your return loss measurement by the reactance affecting the RF signal in your bridge.

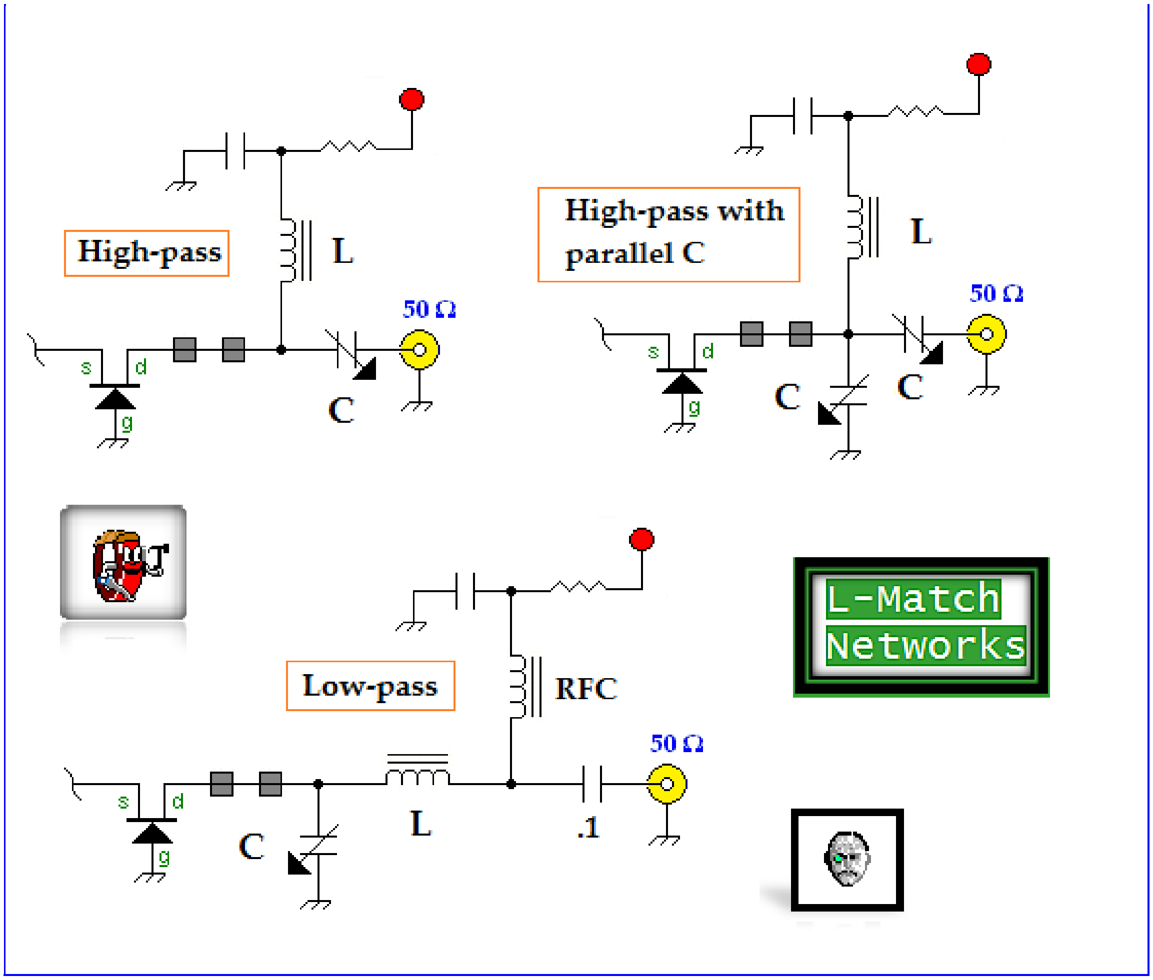
When tuning the output, you're actually changing 2 complex impedances — this is not trivial.

Also if you're off by a few pF or tens of uH in your network C and L values respectively, you can wreak havoc with the measurements. At this point, I don't possess all the skills needed to tune both the output and the input network to a RL of 20 dB or greater; especially with a broadband input.

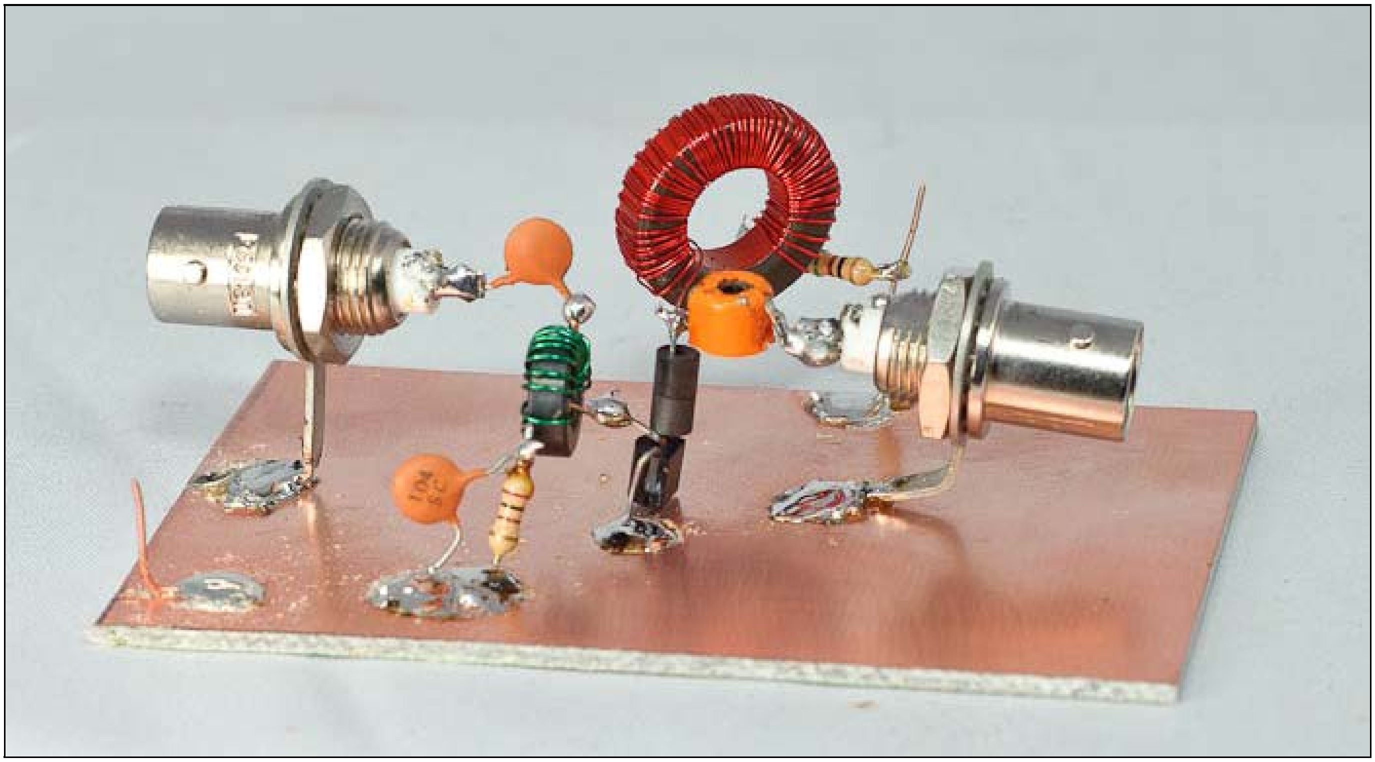


Above — A common gate amp employing a high-pass L-network to match the output. Miraculously after 2 hours of tweaking, I obtained a good input and output match; however this amp isn't reproducible. The inductor was wound on a T68-2 using 28 gauge wire — always a pain. Through trial and error, I learned that the output impedance of the drain was around 11800 ohms. Starting with 18.4 uH on theT68-2, I removed 2-3 turns at a time until a reasonably low return loss was obtained; then I removed 1 turn at a time. I went too far and had to add back a turn. I clipped the excess lead every second turn which made it tedious, but exacting. It seems that the L value is very critical – it would be nice to use a variable inductor to figure these things out. Compressing and expanding the windings also provided a simple way to vary inductance.

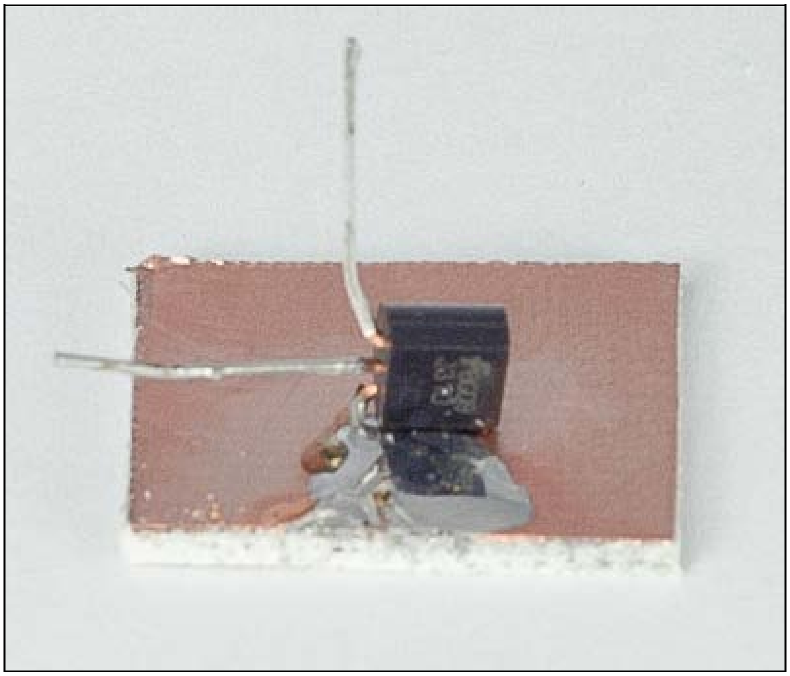
In several other circuits, the best possible input return loss was only 14 dB. Mistuning also caused oscillations to occur in one 14 MHz amp with an output network inductor of 7.4 uH. I also tried a 14 MHz amp with an L-match on both the input and output, however, was unable to match both the input and output due to the interplay between them.



Above — Here are 3 possible L-network configurations for tuning a CG amplifier output. They can be used in other circuits and are worth studying. The L-match with 2 variable capacitors generally requires lower inductance than the others.



Above — A breadboard of 1 of the high-pass tuned CG amps. The gate lead on this transistor is too long — the inductance will likely cause UHF oscillations. 2 ferrite beads were placed on the drain to mitigate these, but a better construction technique is recommended and shown below.



Above — the preferred way to ground the gate with the JFET on its side. The hole-through version of the U310 JFET has a metal case that is connected to the gate that makes it ideal for grounded gate amplifiers. Some suppliers only sell SMT versions of the U310 now.

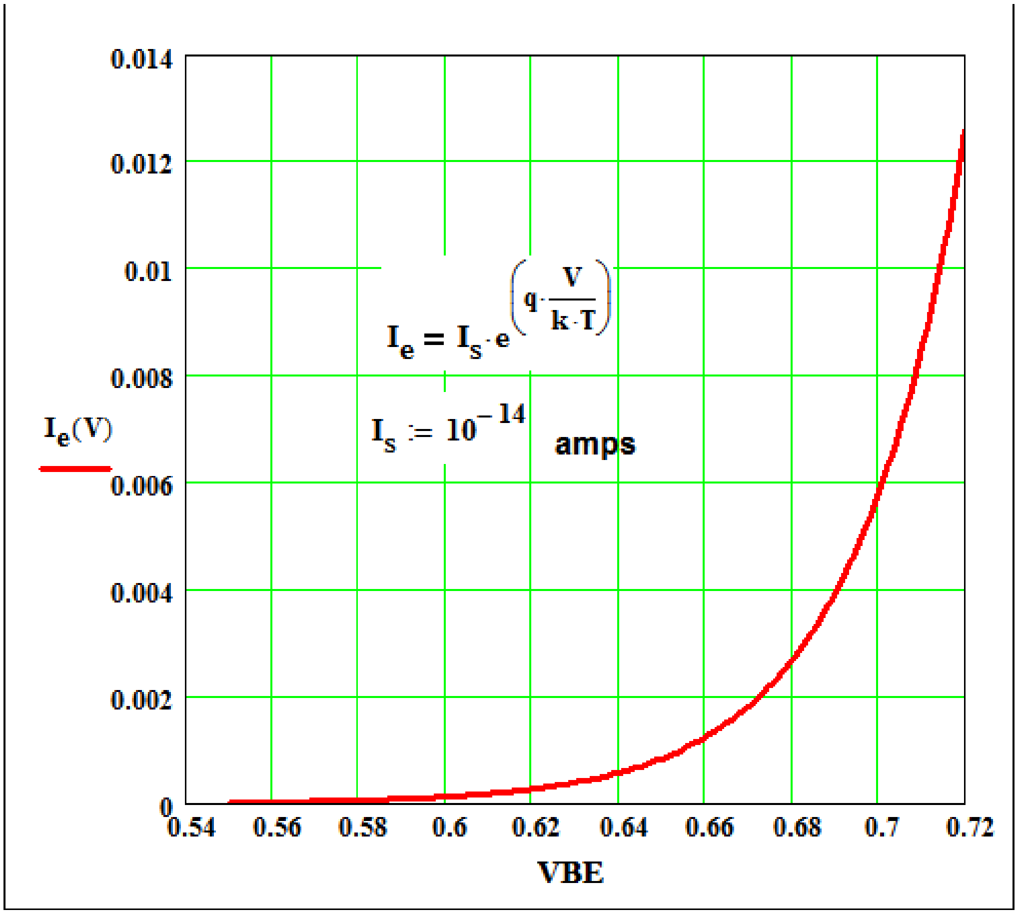
4. QRP — Posdata for January 2013: Transistor Bias Model

This discussion concerns setting up the DC bias point for linear BJT operation.

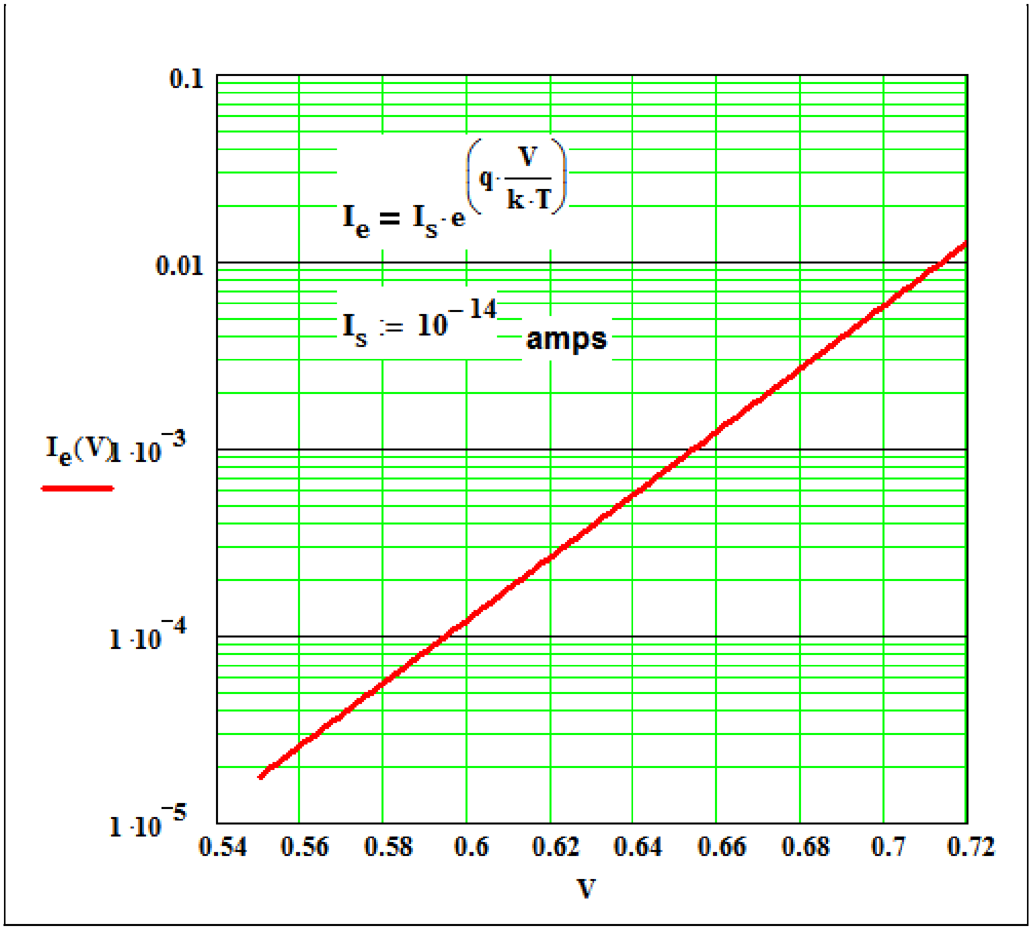
Earlier I stated that a bipolar transistor is a voltage controlled device. A few readers thought I made a typo: something I frequently do, but not in this case, since I purposely made that statement. In reality, the argument could go either way since collector–emitter current is controlled by the base-emitter current (~a current controlled device) and by the base–emitter voltage (~a voltage controlled device).

Stated using the correct physical model, a transistor is a **current controlled current source**. With external circuitry we can manipulate this physical model into a voltage controlled current source, or a voltage controlled voltage source, or even a current controlled voltage source. Whether you model the transistor with current or voltage, the math tells the truth when properly examined.

Please view the following two 2N3904 SPICE models generated by Wes, W7ZOI for me many years ago when I began to learn small signal analysis using impedance and hybrid parameters, plus set out to learn ways to establish DC bias and temperature stability in BJTs.



Above — the Y axis shows how changing base-emitter voltage or current changes the VBE. We tend to assume a VBE of 0.7, however, the math shows the truth. Whether we plot voltage or current for the Y axis data, the graph slope remains similar. The greater the applied DC voltage placed on the base-emitter port, the more current will flow.

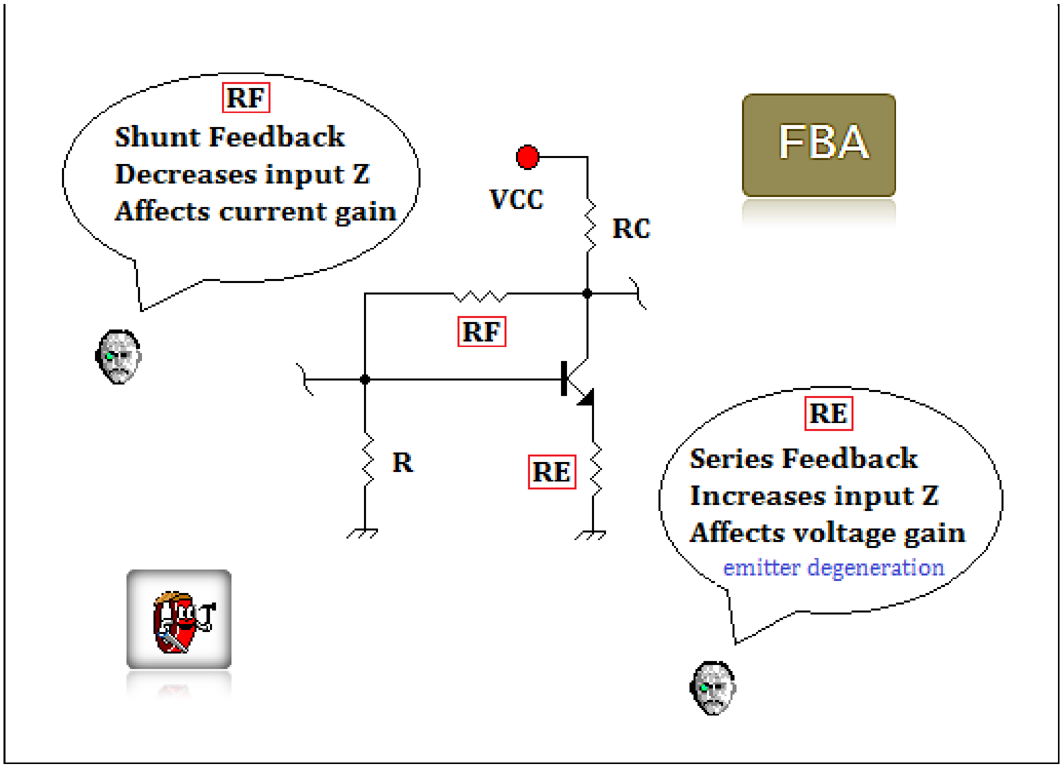


Above — Logarithimic base current plotted against VBE. If we want this current to increase, we need to put more DC voltage on the BE junction. On the bench, we may easily measure base voltage to confirm our calculations — measuring base current proves more difficult. Whether I'm setting up amplifer bias with voltage dividers, a current source, or even biasing it with a downstream AGC voltage, I prefer to think in terms of voltage control — although I get that V and I truly just coexist.

Current or voltage modelling — it's your choice and the math will guide you. Look for these equations on the Web, or in second hand bookstores. I've got 8 or 9 transistor theory books now and they're really timeless.

---

5. Miscellaneous Figures and Photographs



RF

Shunt Feedback
Decreases input Z
Affects current gain

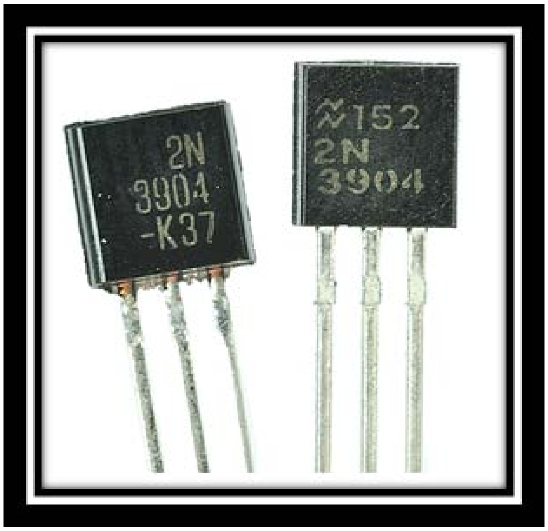
RE

Series Feedback
Increases input Z
Affects voltage gain
emitter degeneration

6. References

1. Communications Receivers - Principles & Design. Rohde, L.R and Bucher, T.T.N. 1988. McGraw-Hill.
2. Technical Correspondence QST Magazine (ARRL) Aug 1990. Hayward W.
3. Experimental Methods in RF Design (ARRL) 2003. Hayward W. Campbell R. Larkin B.
4. Microwave Handbook Vol 1. Components and Operating Techniques. 1989 RSGB.
5. Emails with Wes, W7ZOI and Professor Ken Kuhn, Winter 2010-2011 — Thanks! Никогда не забуд.

Hobby and Fun 2011

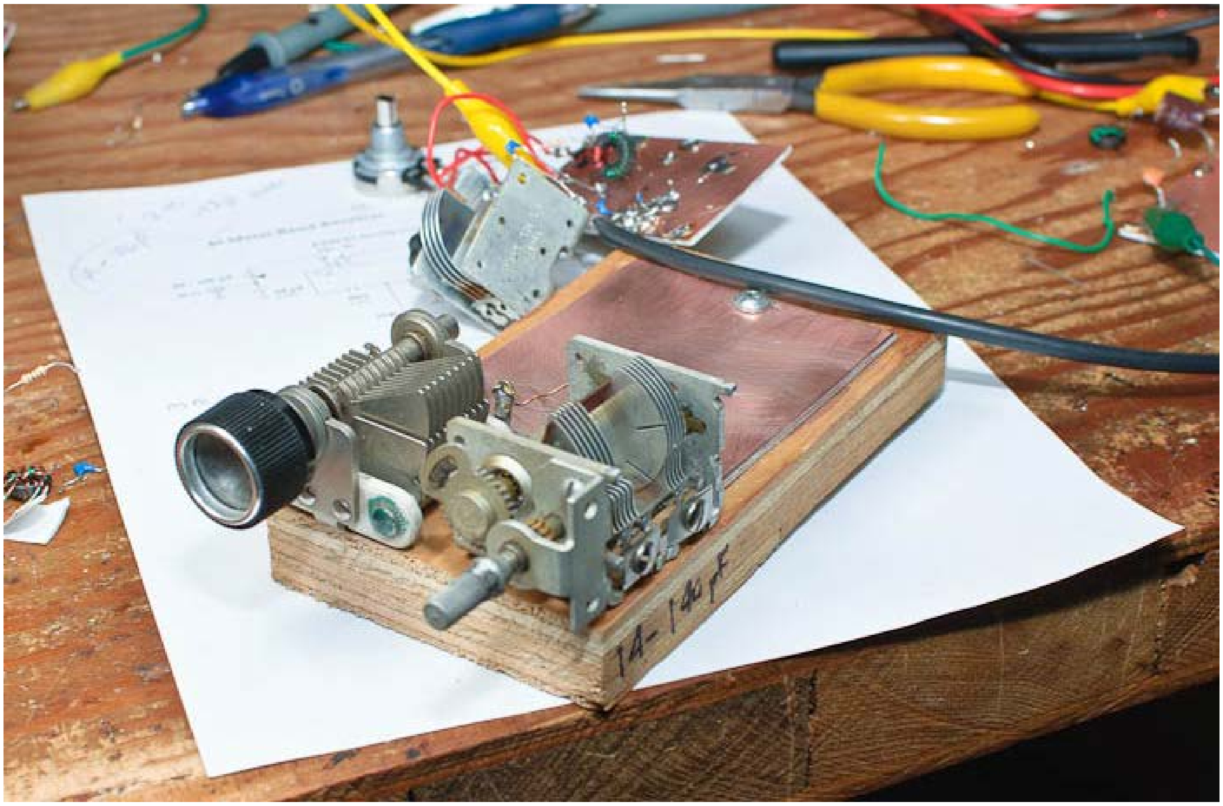


Introduction

Hobbies are supposed to be for recreation. Electronics should be fun, not stressful — heavy math, big parts counts and complexity are more likely to scare away experimenters than recruit them. This page avoids the measurement focus of my latest stuff and simply promotes fun and discovery.

You might be interested to know that my simple experiments/projects garner the most emails. Many wrote "I'm rediscovering electronics", or, "I want a simple and fun hobby". Hobby and fun are my goals too.

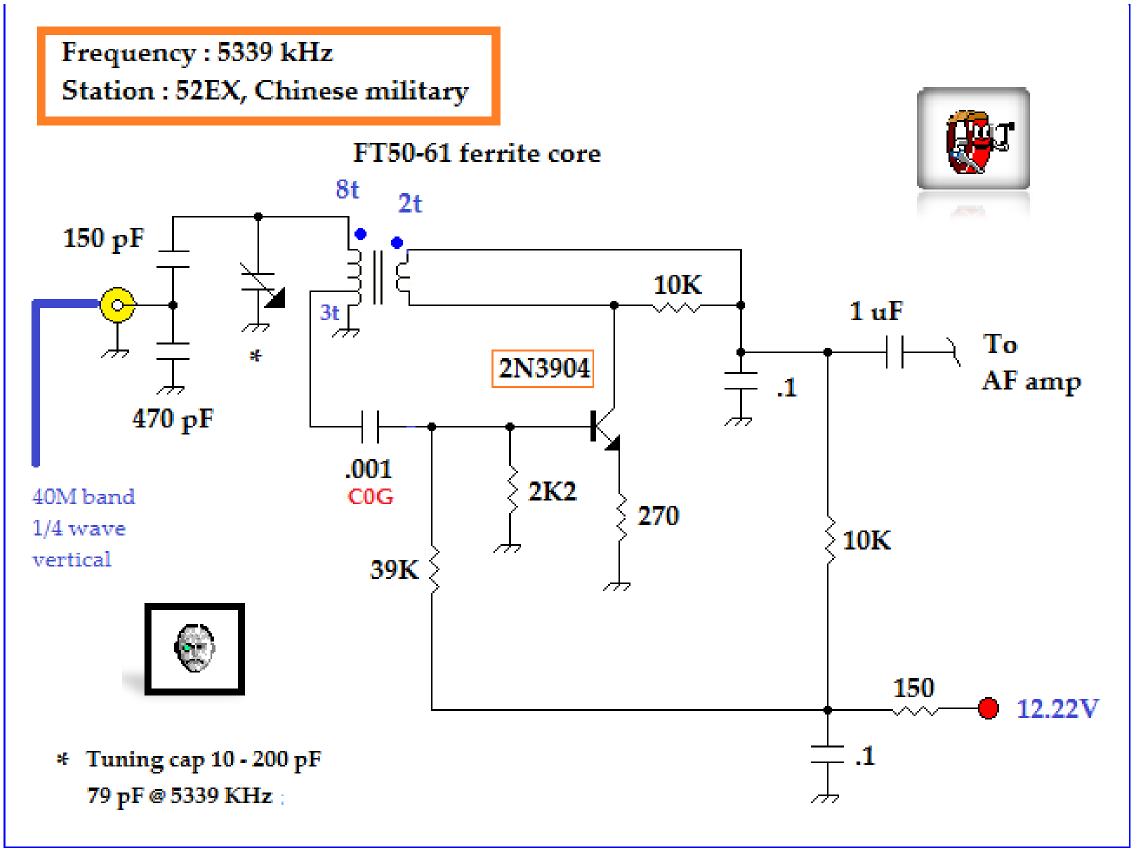
Simple Regenerative Receiver Experiments



Above — 2 air variable capacitors and a copper clad board screwed onto a piece of wood for my bench musing. Regenerative receivers delight and amaze — some builders take them very seriously. I respect this, but to me; they should be as simple as possible. I wanted a 2 stage "genny" receiver for this page and present 2 different receivers; 1 is my design, the others is a JFET variant of a favorite W7ZOI circuit. Quoting Wes, W7ZOI "feedback your imagination".

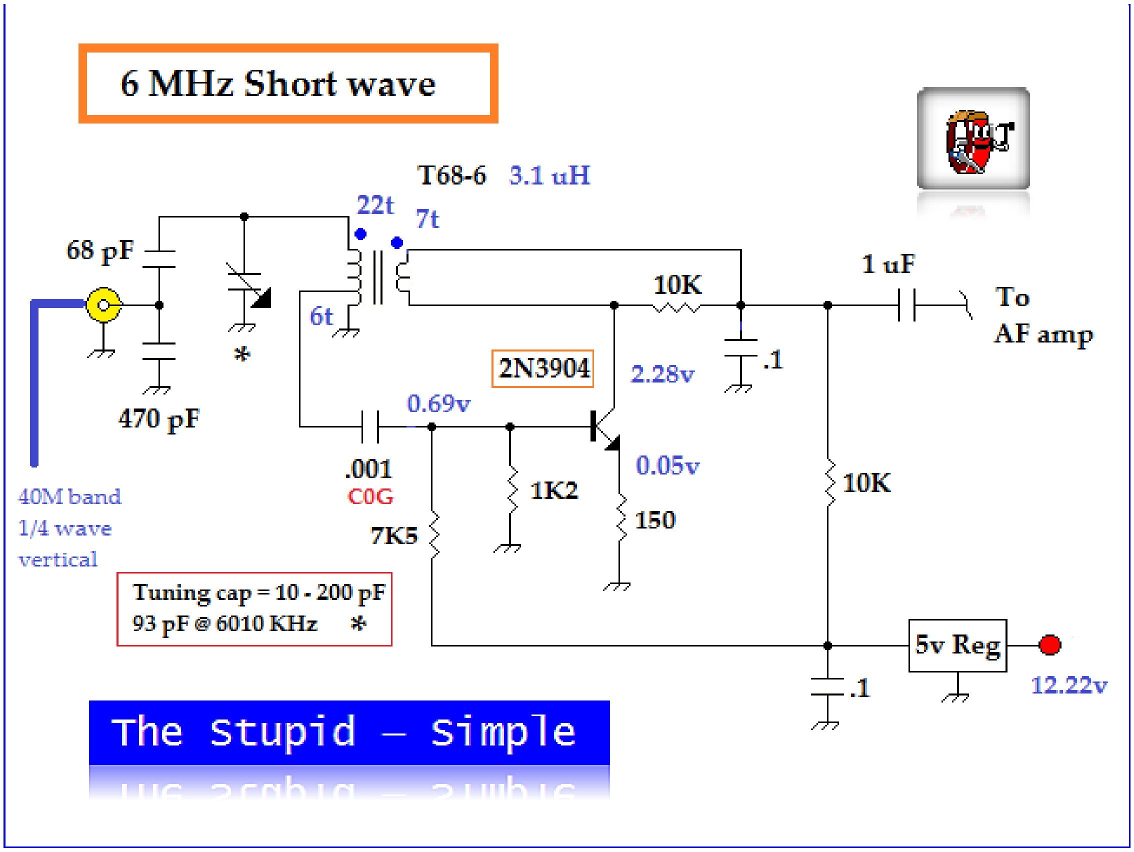
Some builders place a simple common gate or a common base RF preamp on the input to boost gain and reduce antenna radiation of the RF oscillator, while others place an RF gain control on the input — usually a potentiometer; to prevent overloading the RF stage. I won't prescribe what to do — that's up to you. There are countless example of regenerative receivers on the web and you can many spend hours viewing them. Some of the most intriguing are those built by Russian speaking experimenters. Example link.

My circuit ideas are meant as fodder for your own experiments.



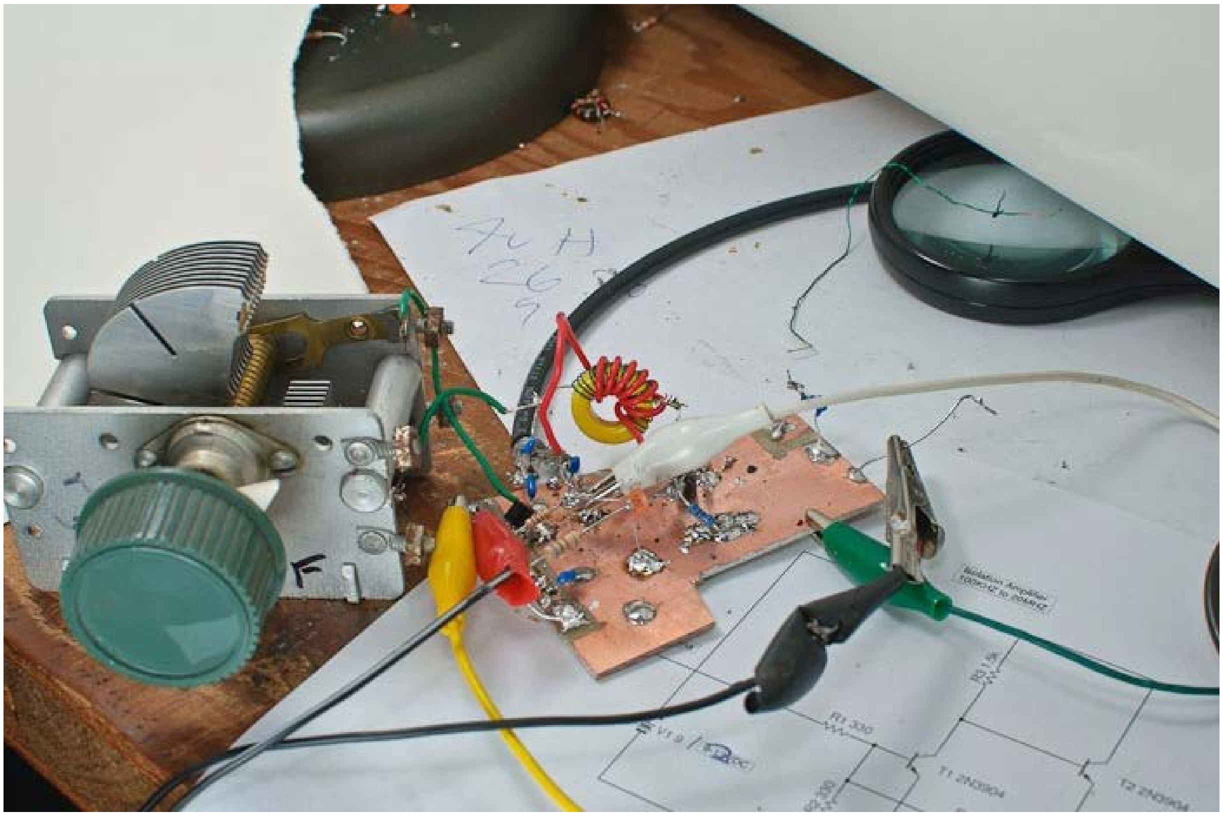
Above — An experimental, ultra-simple "CW" receiver. At 5339 KHz I hear strong Morse code each night. It's suggested to be from China, but I'm unsure. Connected to my 1/4 wave 40M band vertical - a simple matching network and trimmer tuning capacitor were fitted to the input. Here's some audio. I like the beat note of this receiver - it has no regeneration control and is fixed for CW. Minimalist circuits are fun — some hardcore regen builders might freak out; no voltage regulator (here's a version with that + a T68-6 inductor), no regeneration control (here's a version with that), a relatively low Q coil etc. I wanted to try my hand at design and not just copy someone else's receiver.

I call it the Stupid— Simple receiver and although it emits crisp, warm audio, some bench work is required to get the correct bias and appropriate amount of positive RF feedback — an experimenter's circuit that explores DC bias and AC feedback. The sort of thing a father can build with his son. We need more circuits fitting this profile.



Above — The Stupid—Simple experimental receiver set up for ~6 - 7 MHz. 2011 marks the 50th anniversary of Radio Habana Cuba. We tune RHC at 6010 KHz, and as long as I've been listening to SW radio, I've tuned this station. Here are 3 sound bytes from around 6 MHz recorded from 2:30-2:40 GMT on Feb 28, 2011, including an old repeated episode of DX'ers Unlimited by Arnie Coro which had faded out by the time I located and turned on my audio recorder. Audio1 Audio2 Audio 3. The audio stage in these recording was a discrete transistor AF amp I designed, however, an LM386 was chosen for the final amp to keep the parts count and difficulty down. I got a little too close to the receiver a couple of times during the recording and made the open circuit breadboard squeal.

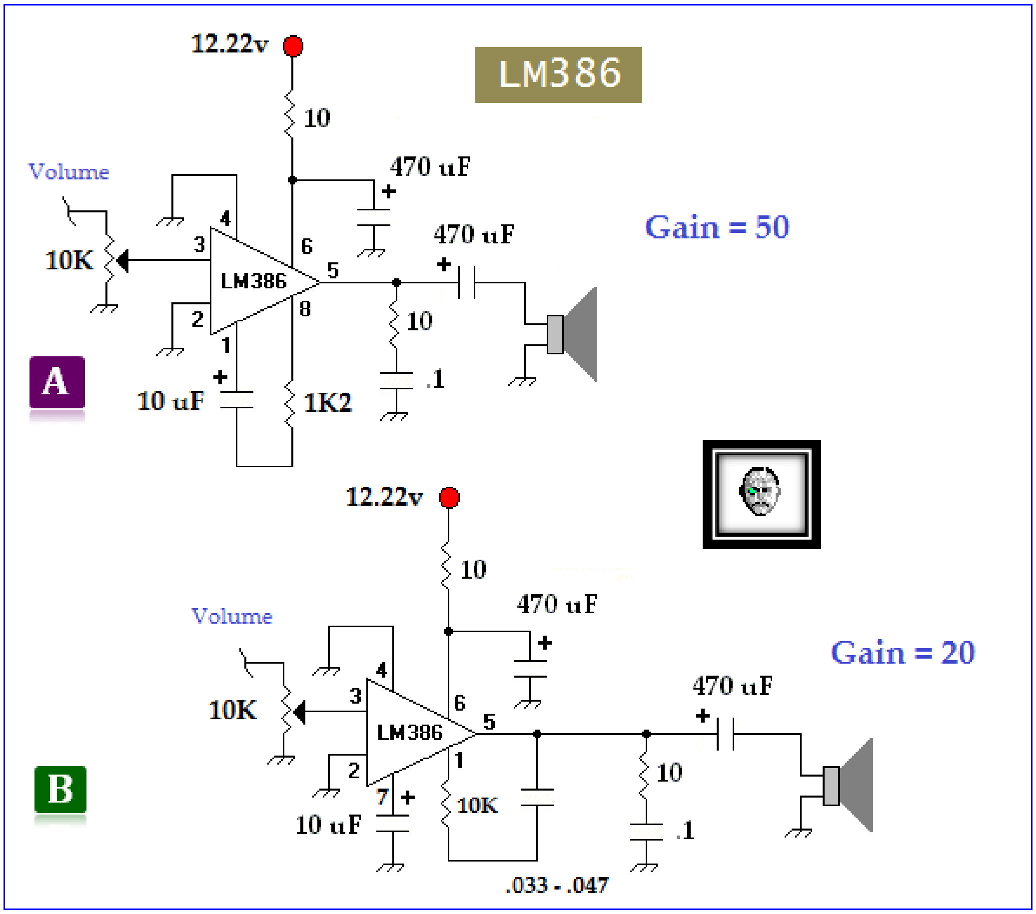
Arnie Coro talked about a "regenerodyne" receiver in Sound Byte *Audio3*. Very cool. Here's a link. Now this is radio!



Above — The Stupid-Simple regenerative receiver breadboard for 6 MHz. You can see the T68-6 — the red secondary windings are wound in the same direction as the 22 turns of primary. I started with 9 turns and unwound a link and tested sequentially until I had the right amount of feedback for AM. I just used normal hook up wire for the secondary winding. The white colored cable goes to the audio amp. This is a prototype experimental layout — a regenerative receiver should have short connections around the tank circuit and be in metal box for best results.

On some stations, my little 1 RF + 1 AF stage receiver sounded better than my superheterodyne receivers. The 2N3904 is just barely turned on — I determined that a base bias of 0.66-0.69v provided maximal sensitivity. The 150 ohm emitter resistor can be a 500 ohm pot and used to fine tune the regeneration. 150 ohms gave the best compromise gain and feedback + current for the 2N3904.

Audio Amplifier



Above — I chose an LM386-N for my AF power amplifier. The LM386 exhibits less peak signal distortion when run in the low-power (X20) gain mode and a higher VCC such as 12 volts. My receiver used the schematic denoted B — a 10K and 0.033 to 0.047 uF RC network is used to reduce the amplifier high frequency response. Click for a sound byte of me tuning around 6 MHz with a 10K + 0.047 uF RC network between pins 1 and 5. Figure B is my favorite way to use the LM386 and comes right off the National Semiconductor LM386 data sheet. Look for this data sheet with your favorite search engine.

Because I have a big antenna, Figure B provides adequate volume to a speaker. Connecting pins 1 and 8 via a 10 uF capacitor bypasses some emitter resistance and gives X200 gain. A resistor in series with the capacitor pin 1 and 8 will reduce the gain. Figure A shows a gain = 50 configuration. You'll have to choose the LM386 set gain to suit your particular regenerative receiver, however, the greater the gain setting, the greater the chance of distortion, unwanted noise and audio feedback.

I generally build my audio power amps around op amps or discrete transistors, but the LM386 exalts this web page's theme. Distortion in all these small power amplifiers is dependent on input signal amplitude as much as anything else.

LM386 Motorboaters

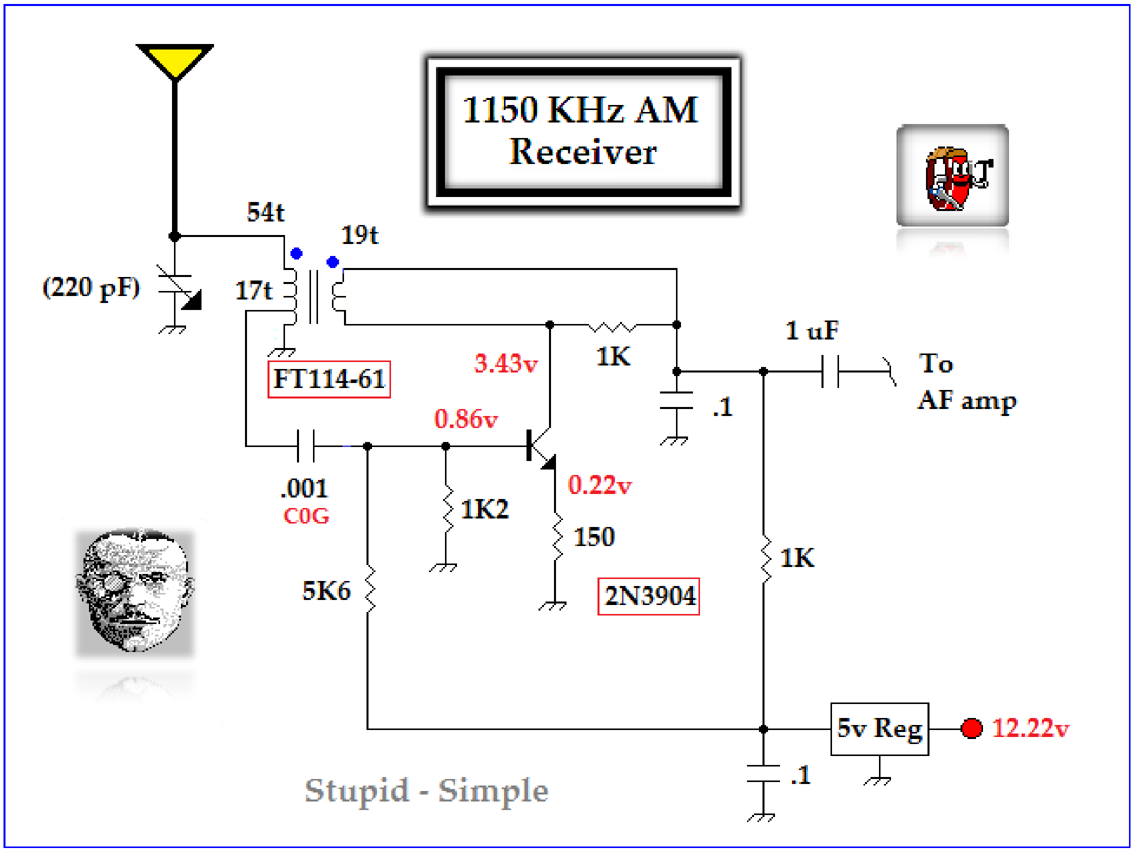
Expect AF feedback motorboating via your DC power lines as you increase the AF gain pot in *many* LM386 circuits. If this happens, try better bypass and decoupling on the LM386 power line: 100 to 470 μF shunt bypass on the DC line + a 10-22 series decoupling resistor, plus 100 - 470 μF shunt bypass on pin 6. Click for 1 example. ypically 470 μF bypass is required if you suffer 60 hertz hum. You may model your simple RC filter with application E on the javascript applet page to see the 3 dB cut off frequency of your particular DC line low-pass filter.

Stupid—Simple Notes

The Stupid—Simple circuit really needs the adjustable 10K regeneration control if you wish to tune both AM and CW. The number of turns on the feedback winding varies with factors including transistor beta, how you wind the primary and secondary windings (greatly affects the coupling between the primary and secondary windings) and whether you want AM, or CW reception — or both. Experiment with the number of turns to figure it out.

You can try "matching" the tank circuit to your antenna by decreasing the 470 pF cap to as low as 68 pF. This will affect the tuning capacitor range. For an air variable tuning capacitor, use anything you can find. Consider connecting fixed parallel and/or series capacitors to reach or limit the desired capacitance. Many good examples are published on the web.

Some builders float the tuning capacitor across the inductor so the cap is ungrounded. The stator (body) of the capacitor should be connected to the circuit ground to help minimize the effects of hand capacitance. A grounded metal case further helps.



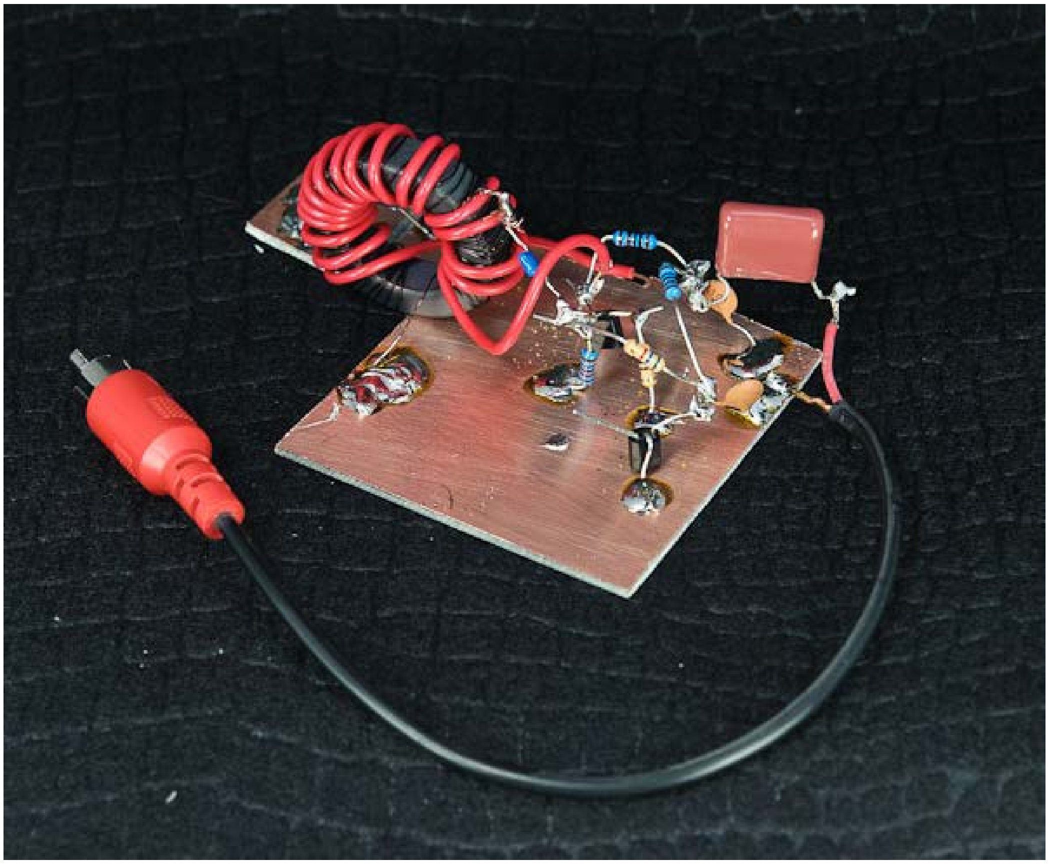
Above — My popcorn regenerative design "The Stupid-Simple" set up for broadcast band radio at 1150 KHz.

A reader from Brazil enquired about putting the Simple Stupid on MW. I had some time for a couple of experiments but only wanted 1 frequency — 1150 KHz, the local 10 KW sports radio station. Using 28 gauge wire, I wound 54 turns (about 230 uH) on a A FT-114-61. Most builders won't have this toroid, however ferrite rods from AM radios are plentiful and a great substitute.

This design relaxed the regeneration to improve audio quality (no whining or hissing). The bias and feedback loop were wound for the best sounding audio. For example, at the bias shown, if you increased the 19 turn link to 21 turns, the bass response increases; decreasing to 17 turns reduces the bass response.

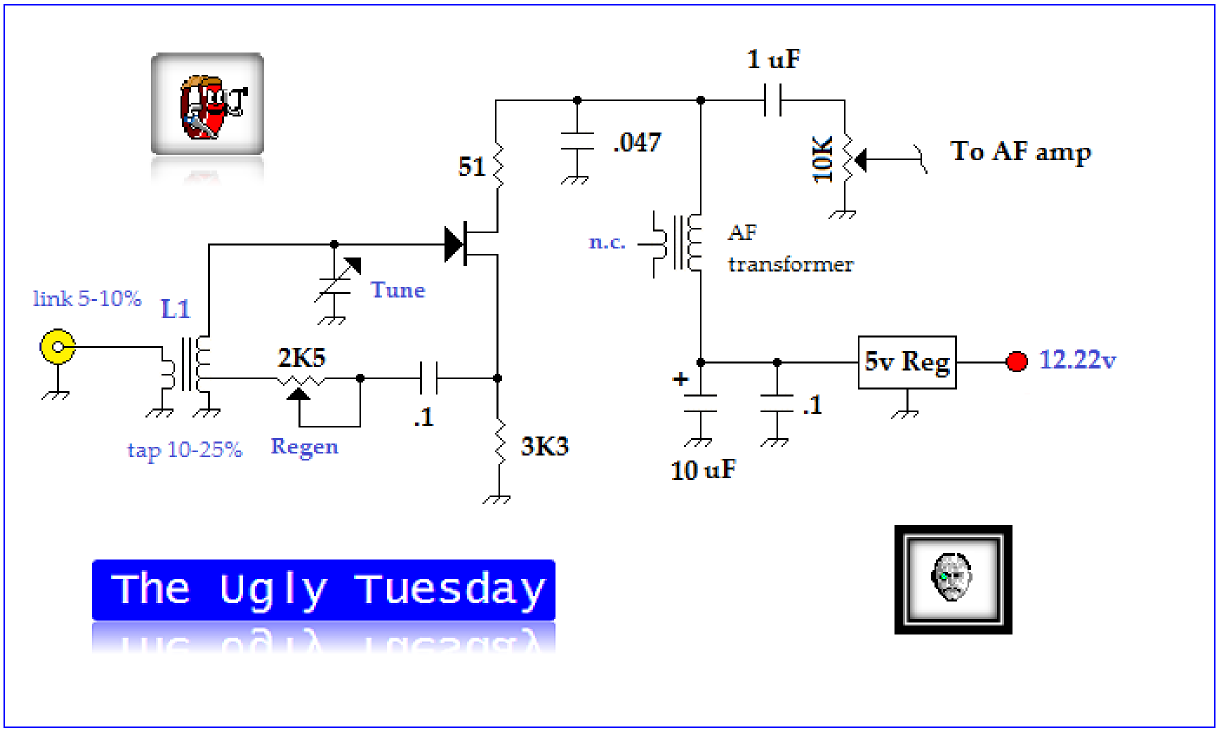
As a result of lowered positive feedback, the selectivity is down, however, a variable capacitor is needed to peak the station. After peak tuning, I removed and measured the air variable cap at ~ 200 pF and then substituted a 220 pF fixed capacitor to simplify things.

For audio, I used a bench AF power amp into a speaker. It sounds nice for 1 transistor. Audio sample



Above — The breadboard of the 1 channel receiver for my workshop - I'll use it to keep track of the Canadian Football League statistics. My test antenna was a long piece of outside wire. Red hook-up wire forms the 19 turn feedback winding.

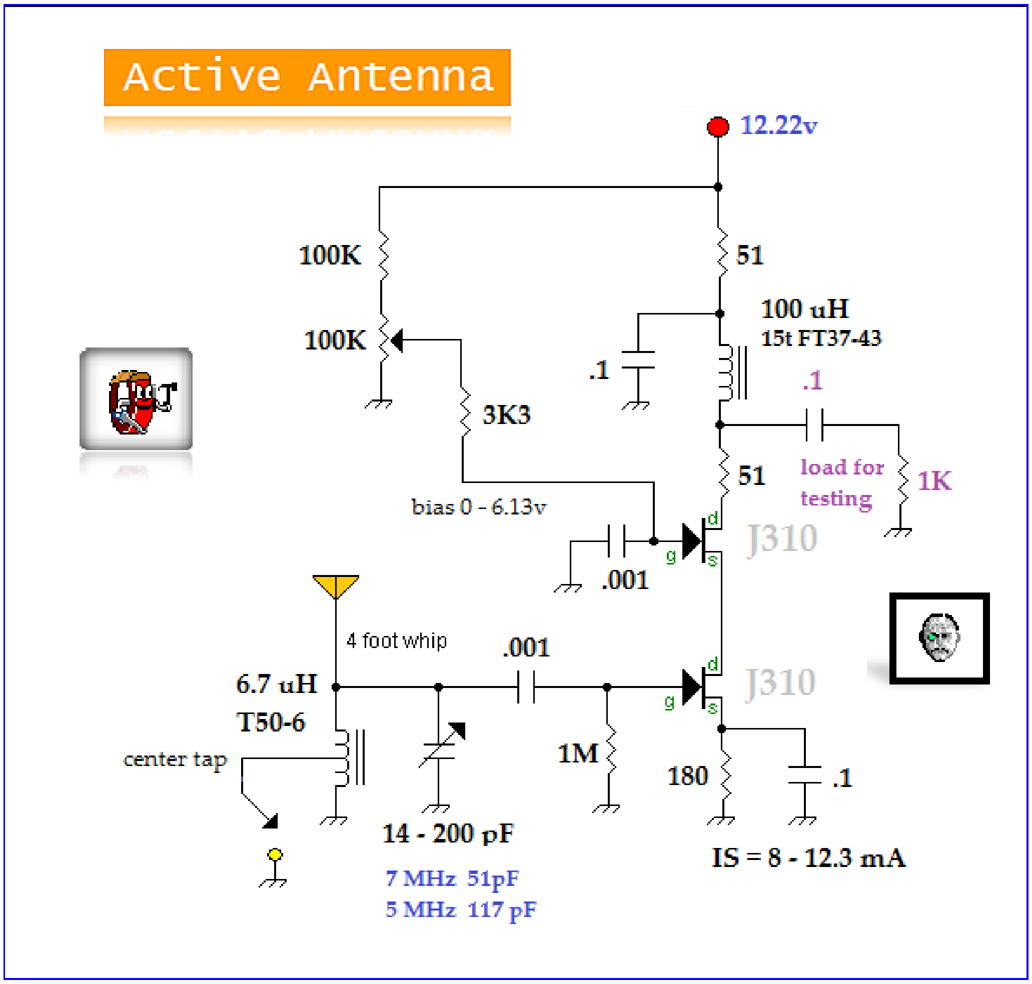
For cities with multiple AM stations (AM stations are dying out in Canada), you'll have to add more regeneration and probably move to a better design. This radio is simple, but not extraordinary.



Above — Another regenerative design that tuned AM, SSB and CW from 5-11 MHz with different toroid coils wound on a T50-6. It's based on a favorite design by Wes, W7ZOI. I suggest tapping L1 at 10 - 25% of the total number of turns. The secondary link for the antenna connection depends on the impedance of the antenna, but 5 - 10% of the total number of L1 primary turns worked well at my QTH. Please experiment with the secondary link to determine the optimal coupling to your antenna. My L1 inductance ranged from 1.5 to 5.6 uH. You may have to add a fixed capacitor in parallel with your air variable capacitor when using low inductance coils such as 1.5 uH.

The 51 ohm resistor suppresses UHF parasitic oscillations. The AF transformer is a transistor radio output (1000 : 8 ohm) junk box special and serves as an RF choke. I tried various AF transformers harvested from old transistor radios in this slot and they all worked fine. Nothing's really critical on this receiver — that's why I like it. Truly junk box radio.

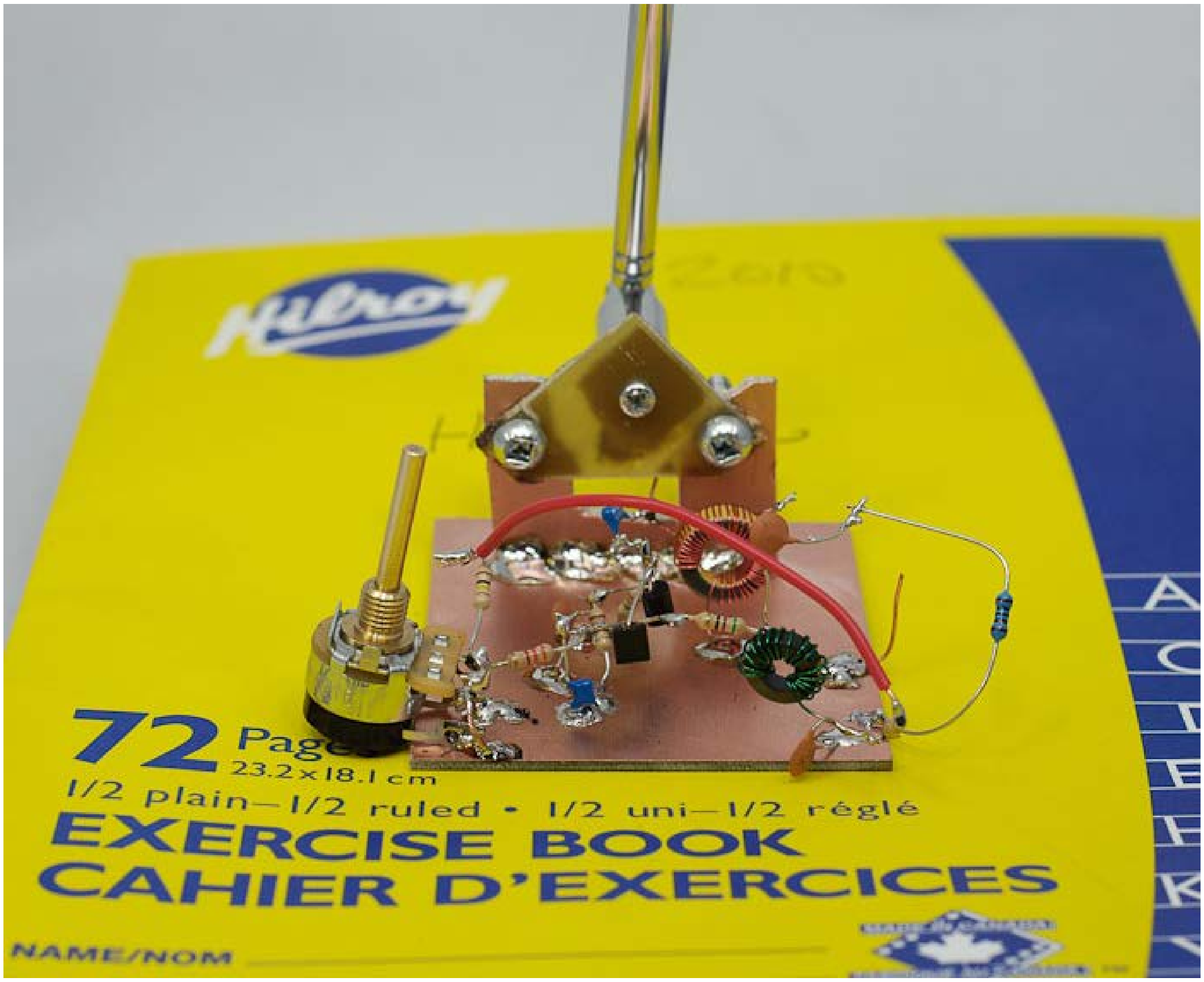
Simple Active Antenna Experiments



Above — A voltage probe or active antenna using a telescopic whip. It's been awhile since I built one in keeping with a minimalistic circuit theme. I tested this VPA from 5 to 14 MHz. The center tap on the coil allows the peaking at ~10 MHz and higher. The L value is non-critical; choose a value that will work with your tuning variable capacitor or varactor. The L - C values can be *roughly* determined from a chart like this, or just do the math (XL = XC at the desired frequency). Account for stray inductance. If you wish to perform return loss measurements on this circuit, you'll have to short the 6.7 uH inductor as the whip antenna can tune in RF from the RF signal generator used for the return loss bridge.

Without the 1K load, the circuit will oscillate. I thought about some ways to match the output transformer to a regenerative receiver tank circuit. The 100 uH drain choke could be replaced by a (bifilar) 2:1 transmission line transformer or two. Transformer experiments this Winter clearly illustrated the superior coupling of transmission line transformers and mandates using them over conventional transformers whenever possible.

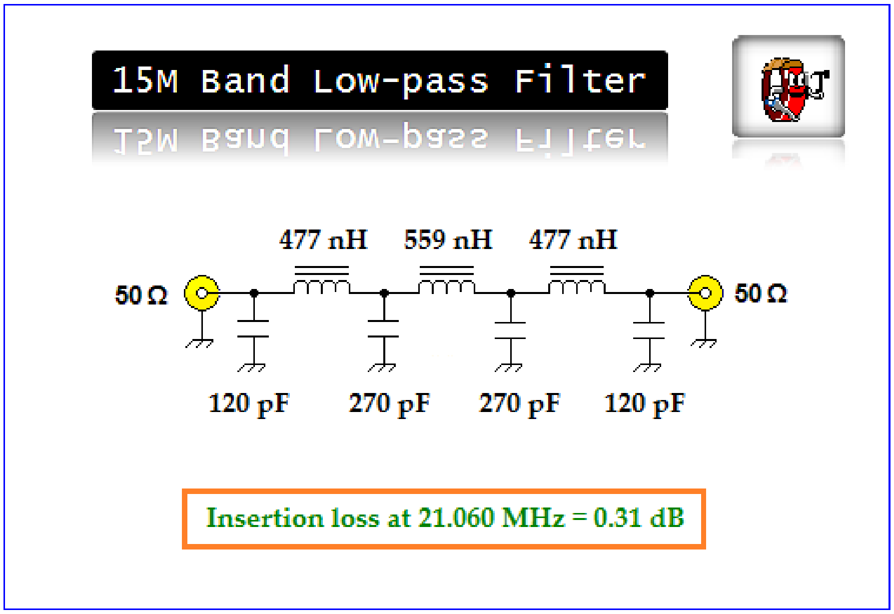
Using a conventional transformer with a shunt resistor across the transformer would also work, but the resistor reduces gain. I built and tested this output circuit with an 8:1 transformation using two 2:1 transmission line transformers. The output impedance at the JFET drain is somewhere around 4300-4500 ohms at 7-14 MHz. The transformed output impedance is somewhere around 250-330 ohms at 7- 14 MHz. Connecting the VPA output to a tap in the regenerative main tuning inductor might work — being careful not to load down the regenerative tank coil.



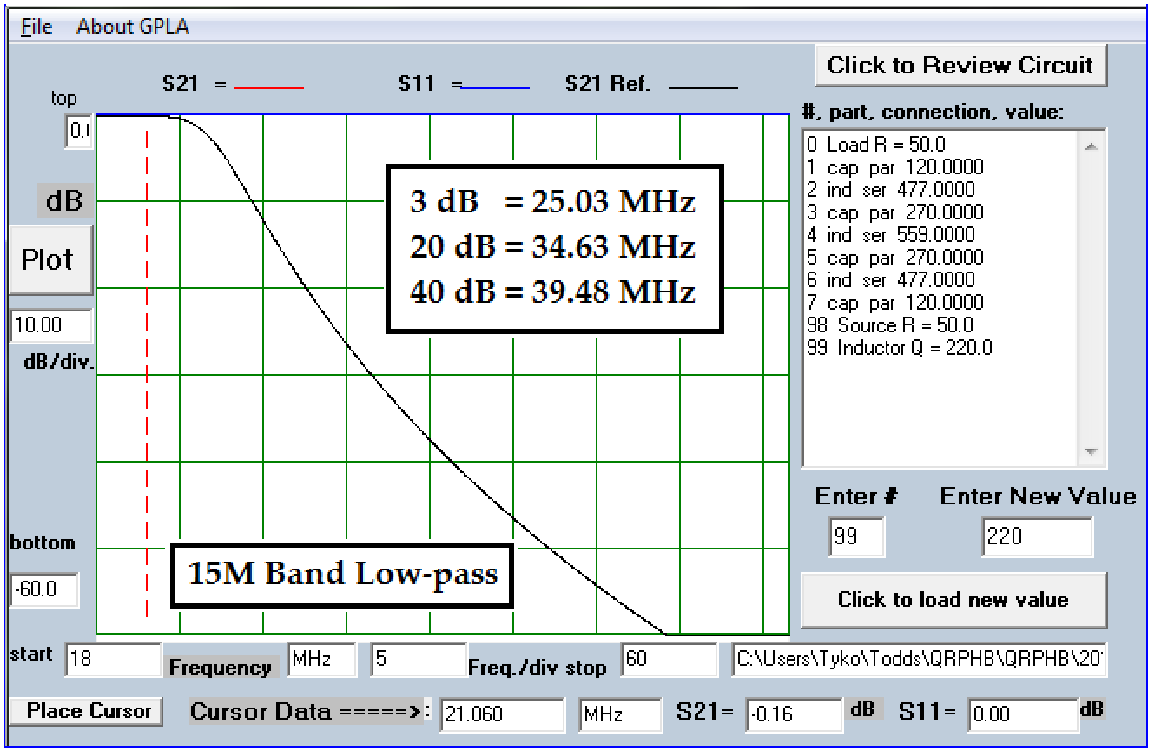
Above — The VPA breadboard on my latest notebook. The 100K pot sets the stage gain. While simple, it works okay.

Above — the VPA built March 12, 2011 (the 14-200 pF air variable cap is not shown). It took about 45 minutes to design, build and test it. The 1K load used for testing is the blue resistor to the extreme right.

Low-pass Filter for 21 MHz



Above — A 7 element Chebyshev low-pass filter for the 15 Meters Ham band (fCo = 25.03 MHz to allow the use of standard value capacitors). A builder requested a band-pass filter design for his 15M band receiver. In order to accurately test my design, I decided to make a permanent, low-pass filter module to follow my signal generator. 15 Meters is a favorite Ham band, so I'm certain to use it in the future.



3 dB = 25.03 MHz
20 dB = 34.63 MHz
40 dB = 39.48 MHz

15M Band Low-pass

#, part, connection, value:
0 Load R = 50.0
1 cap par 120.0000
2 ind ser 477.0000
3 cap par 270.0000
4 ind ser 559.0000
5 cap par 270.0000
6 ind ser 477.0000
7 cap par 120.0000
98 Source R = 50.0
99 Inductor Q = 220.0

Enter # 99
Enter New Value 220

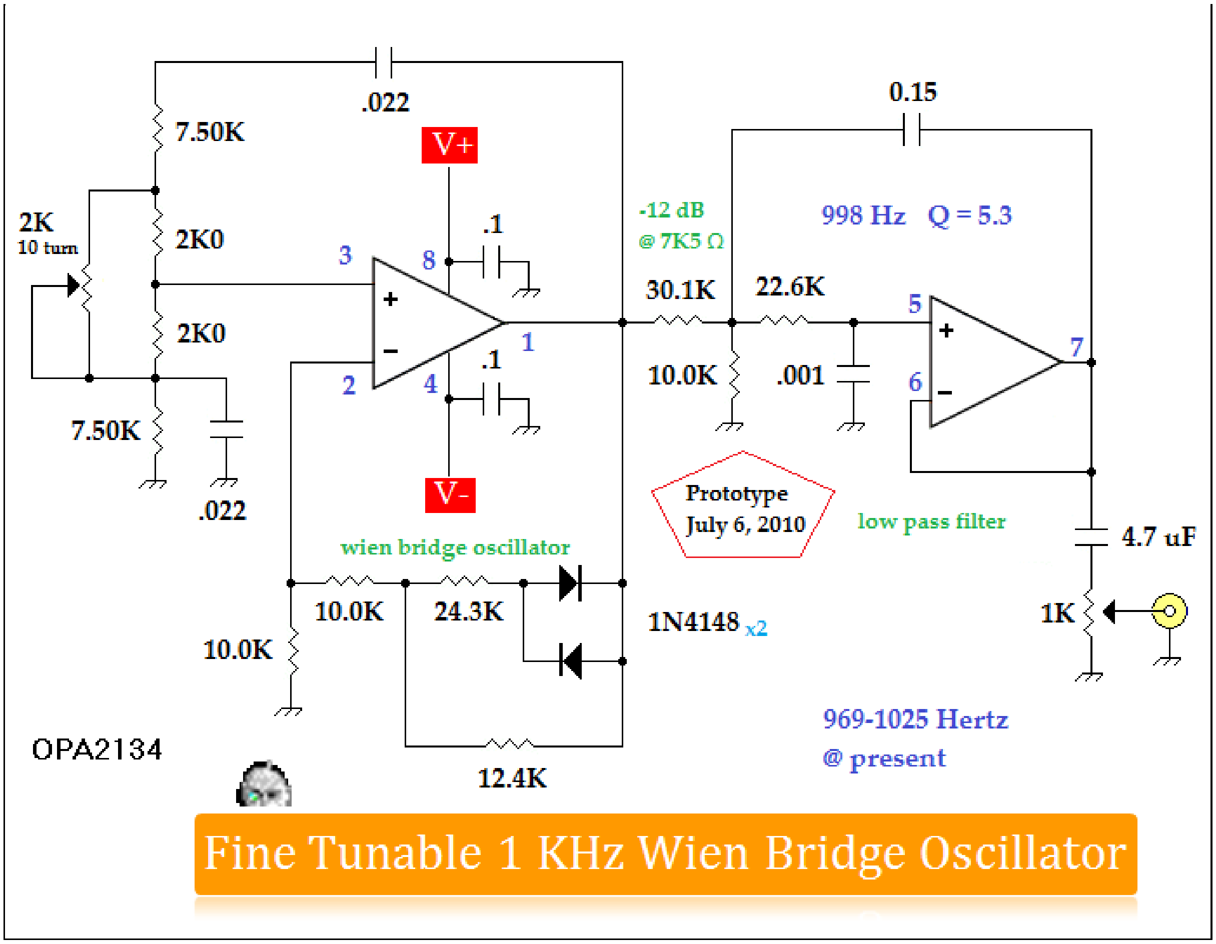
top 0.(
10.00 dB/div.
bottom -60.0
start 18 Frequency MHz 5 Freq./div stop 60
C:\Users\Tyko\Todds\QRPHB\QRPHB\20

Cursor Data =====>: 21.060 MHz S21= -0.16 dB S11= 0.00 dB

Above — A GPLA plot of the filter. The frequency cut-off at -3, -20 and -40 dB are shown. Perhaps this filter is overkill, but I had all the parts on hand and love a serious low-pass filter. Click for the bread board photo.

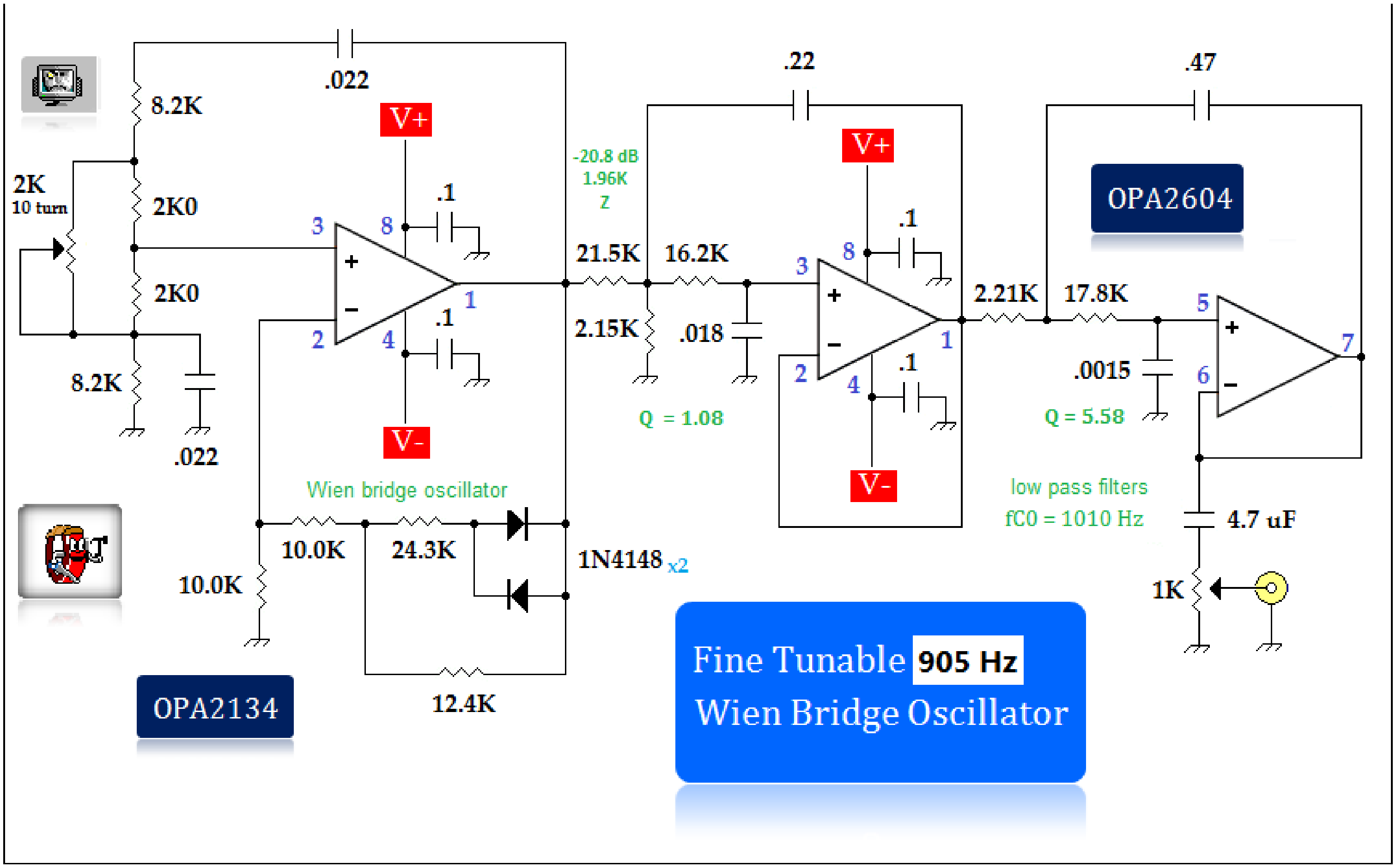
Fine-Tuneable 1 KHz Wein Bridge Oscillator

In 2010, I wanted a fine-tuneable Wien bridge oscillator to drive a notch filter in an AF distortion analyzer. Ken Kuhn drew me up a schematic on his coffee break and emailed it the same morning.

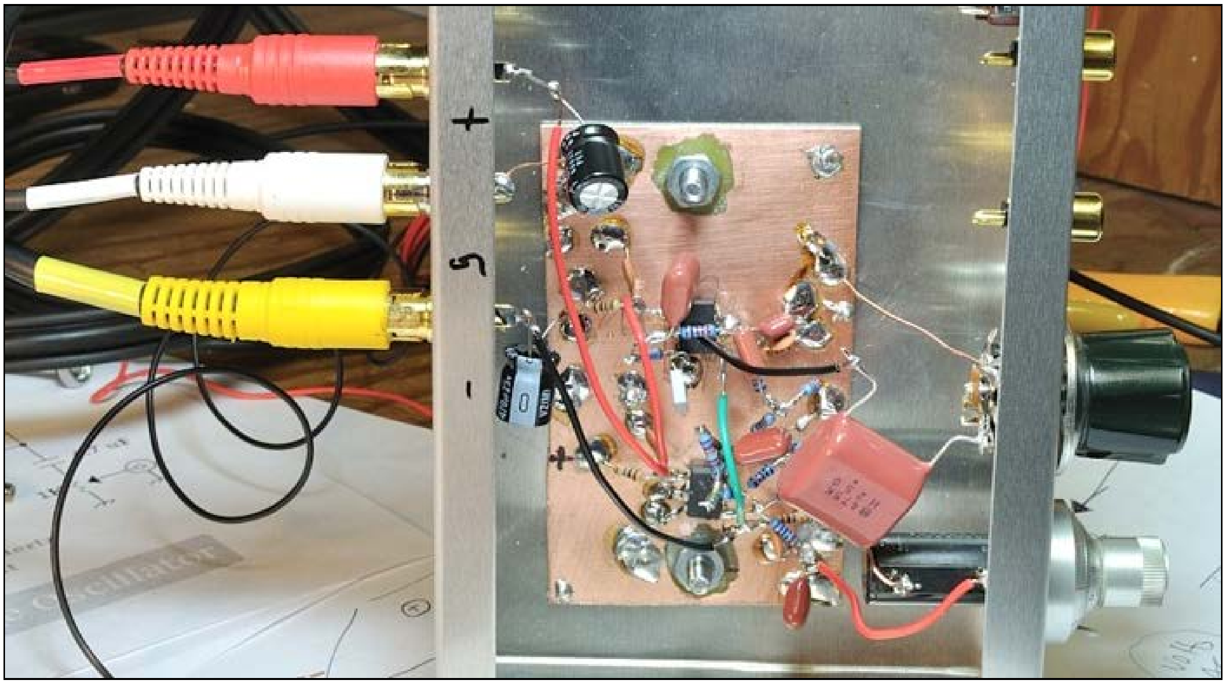


Fine Tunable 1 KHz Wien Bridge Oscillator

I applied anti-parallel diodes instead of the classic incandescent bulb for amplitude stabalization in the feedback loop; probably a mistake leading to higher distortion. Ken's fine-tuning circuit works perfectly. I matched the 7K5 + 2K0 + 0.22 uF components on each filter half. 1% parts go in this circuit.



A version for 905 Hertz built with crazy expensive op-amps. I chose 905 Hz to match my notch filter frequency.



My breadboard.

RF — Test and Measurement

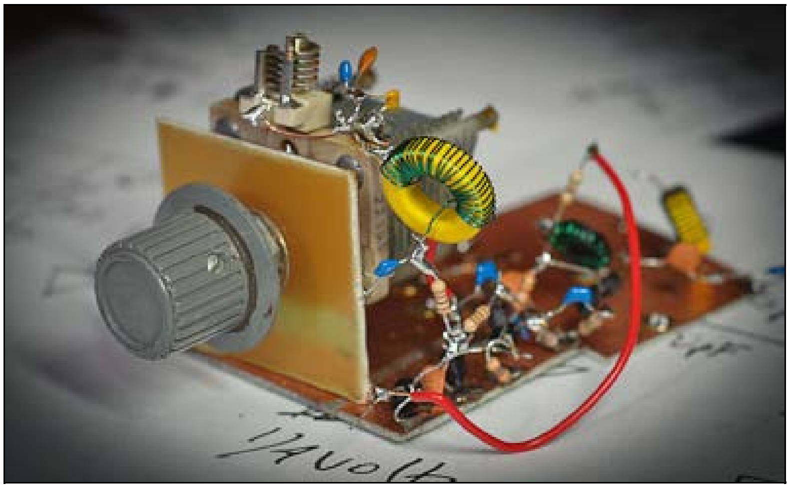
QRP Modules 2011

Introduction

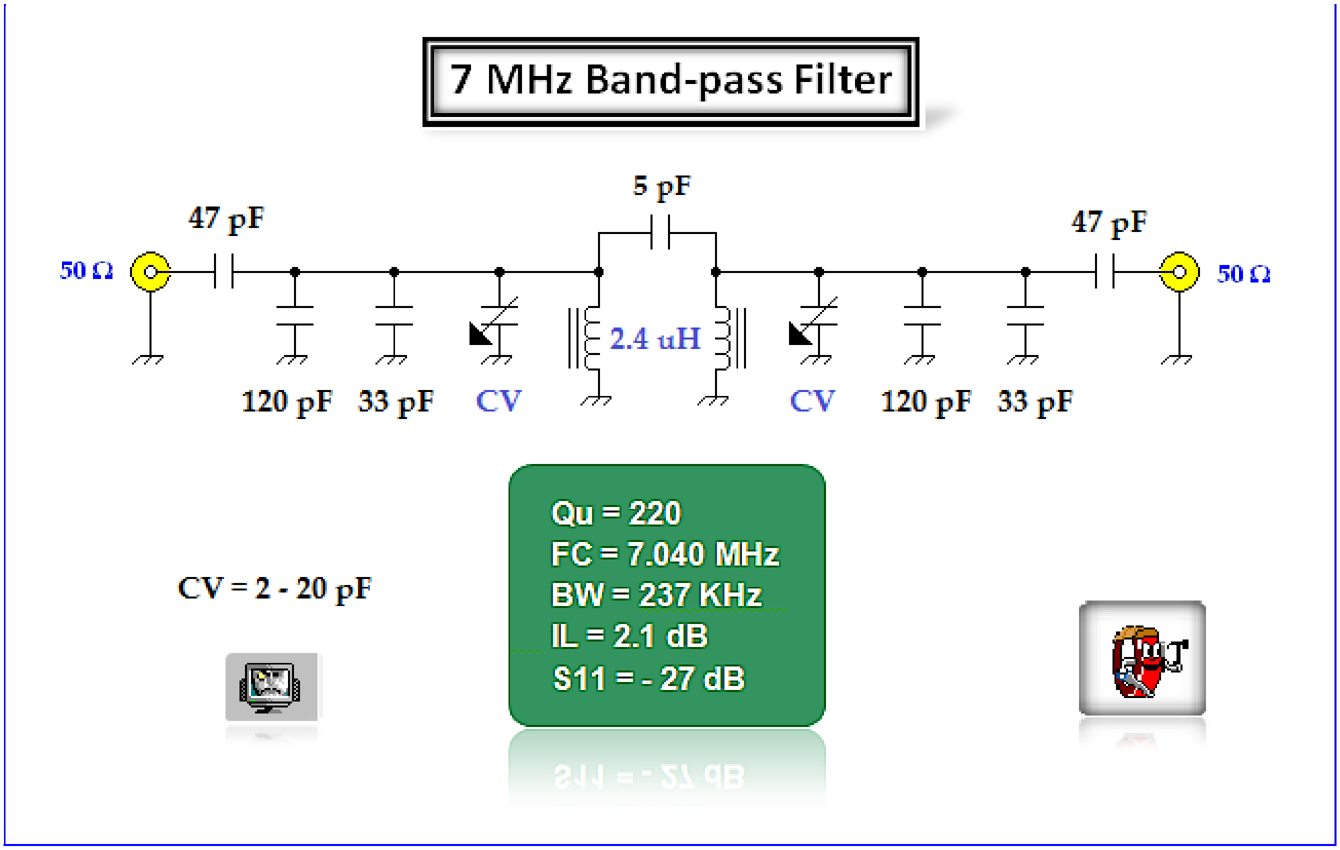
As experimenters, we rebuild core circuits over time. I decided to increase my collection of stock modular circuits to avoid re-inventing the wheel. This web page serves as a module repository for the website.

Since our needs differ, I've shared these circuits more for interest sake and really not as schematics to copy. All modules were carefully built and tested.

1 great virtue of the metal encased module is strong shielding. RF modules use a 50 ohm port impedance and BNC connectors. RCA jacks interface the AF modules.

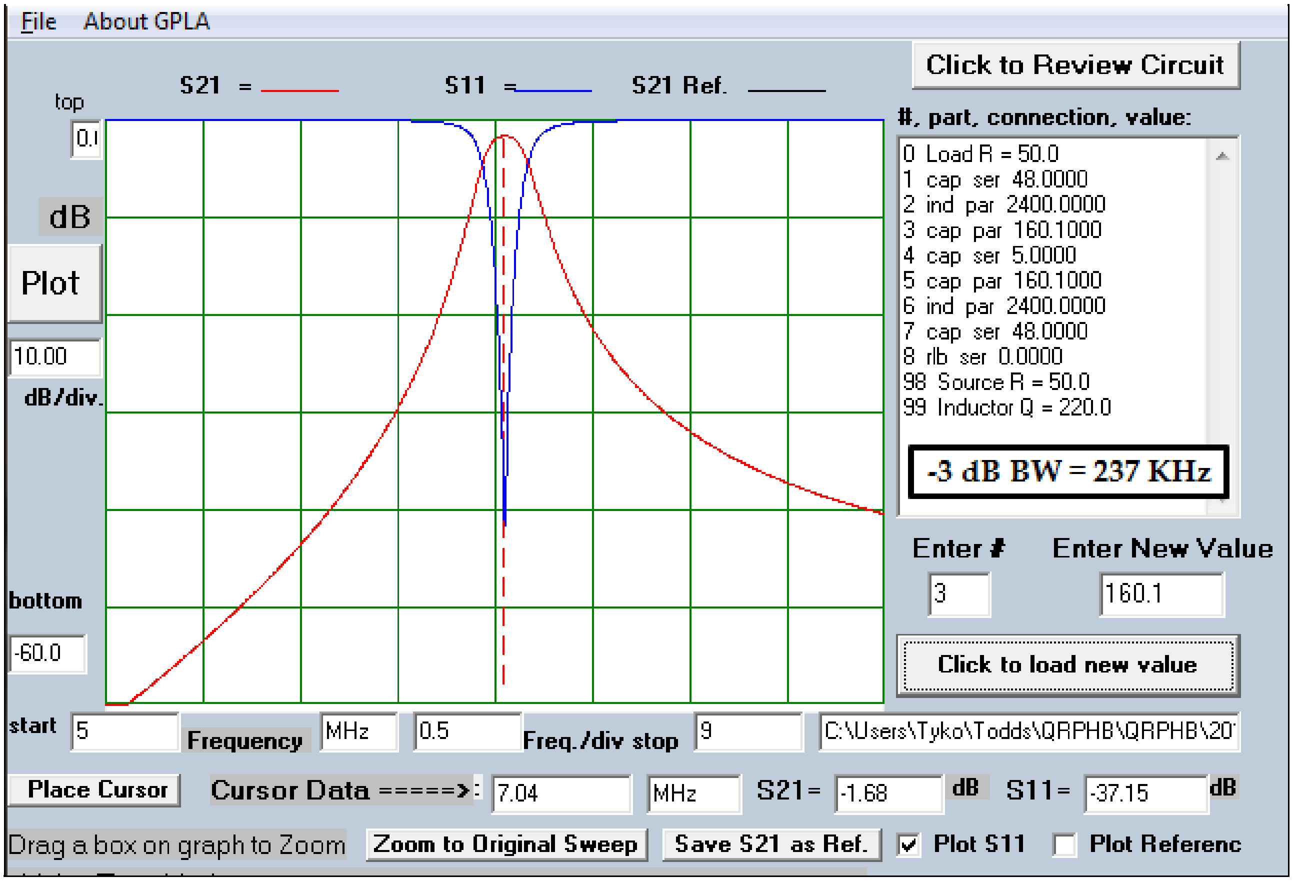


40 Meter Band-pass Filter



Above — A 40 Meter Ham band double-tuned band-pass filter. I designed this circuit using 2 programs that came with EMRFD and describe the process on this web page. The 2.4 uH measured coils were wound using #22 AWG wire on T68-6 powdered iron toroids and all fixed caps were ceramic C0G type. I centered my filter at 7.040 MHz. You should be able to peak it anywhere on the 40 Meter CW sub-band by tweaking the variable capacitors.

I peaked the trimmer capacitor while looking at the peak-to-peak voltage on a 50 ohm terminated oscilloscope. The filter input was connected to a 7.040 MHz signal generator with a 30 dB return loss, low harmonics (-55 dBc) and 50 ohm cables.

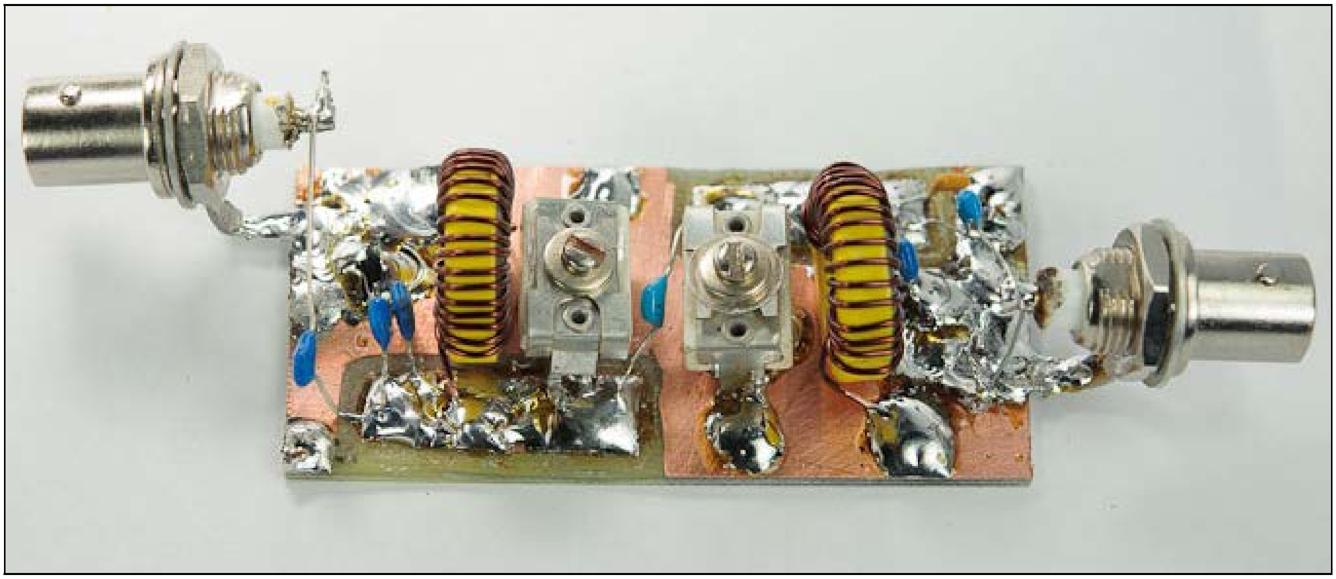


0 Load R = 50.0
1 cap ser 48.0000
2 ind par 2400.0000
3 cap par 160.1000
4 cap ser 5.0000
5 cap par 160.1000
6 ind par 2400.0000
7 cap ser 48.0000
8 rlb ser 0.0000
98 Source R = 50.0
99 Inductor Q = 220.0

-3 dB BW = 237 KHz

top: 0.(
dB/div.: 10.00
bottom: -60.0
Enter #: 3
Enter New Value: 160.1
start: 5 MHz
Freq./div: 0.5
stop: 9
C:\Users\Tyko\Todds\QRPHB\QRPHB\20
Cursor Data =====> 7.04 MHz
S21= -1.68 dB
S11= -37.15 dB
☑ Plot S11
☐ Plot Referenc

Above — A simulation of my filter design in in GPLA08. The calculated IL was 1.68 dB, I measured the IL at 2.1 dB The calculated return loss or S11 was 37.2 dB; I measured 27 dB. A good filter.

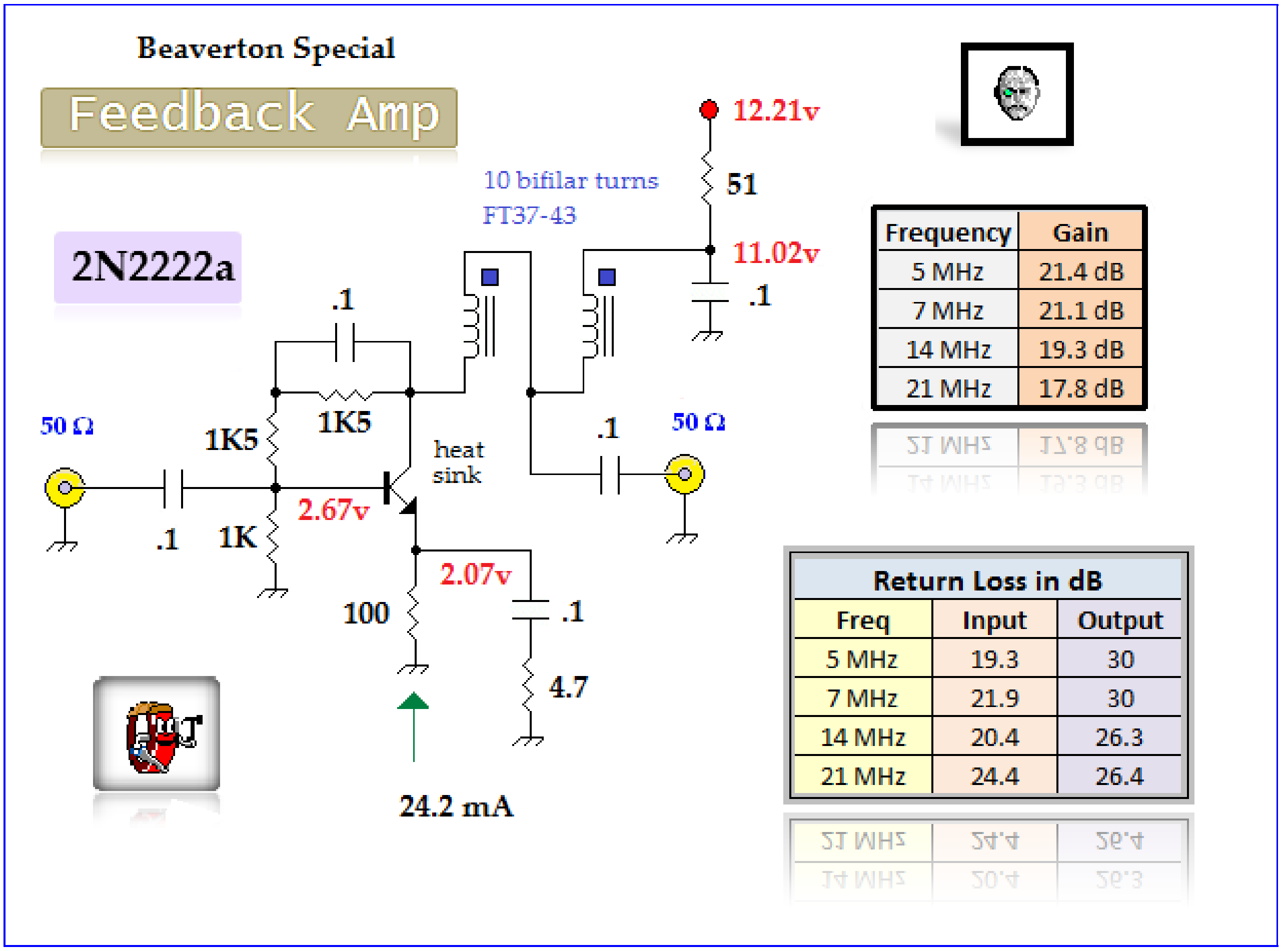


Above — the 40 meter double tuned band-pass filter breadboard with temporary BNC connectors and series caps. Since this filter will serve as my main front-end filter for all future 40M band receiver bench design, I blinged out and put in big toroids and high Q, air-variable trimmer capacitors. While I could have just use a single 150 pF tank capacitor and a wide range trimmer cap such as common, ceramic 10-70 pF, the small range, high Q trimmer capacitors offer better performance and fine tuning.

Click for a spectrum analyzer +tracking generator sweep where the center frequency = 7.040 MHz. Graticules: Horizontal = 1 MHz per division, Vertical = 10 dB per division. You can see why it tunes so sharply.

After testing the bread-board, I removed the temporary BNC connectors and series caps, I stuck it in a Hammond box and wired in permanent, short leaded 47 pF capacitors. Final testing in the sealed box varied minimally from the open bread board. This board looks especially ugly because it held a previous filter and contained lots of remnant solder.

Broadband Feedback Amp



Frequency	Gain
5 MHz	21.4 dB
7 MHz	21.1 dB
14 MHz	19.3 dB
21 MHz	17.8 dB

Return Loss in dB

Freq	Input	Output
5 MHz	19.3	30
7 MHz	21.9	30
14 MHz	20.4	26.3
21 MHz	24.4	26.4

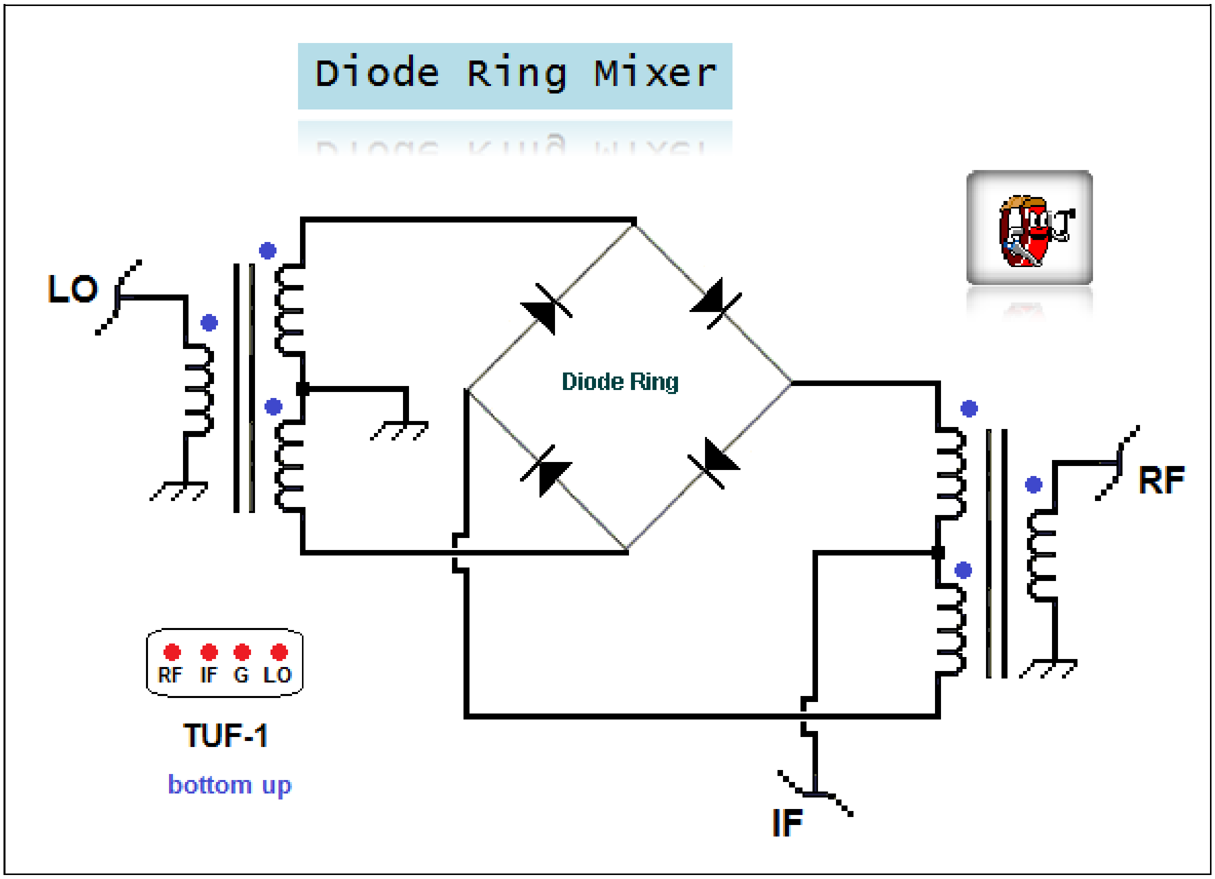
Above — A "Beaverton Special" feedback amp with analysis. As experimenters, we often need a go-to, broadband 50 ohms input and output RF amplifier. This is it!

Popularized by Wes, W7ZOI and Doug DeMaw, W1FB, this amp has stood the test of time and fits perfectly into the 50 ohm module concept. A bevy of transistors were tried — a 2N4401, 2N5179, 2N3904, 2N3866 or 2N5109 all worked fine. For the greatest return loss and signal handling possible, current over 21 mA is required and thus a 2N3904 isn't the best choice. Collector current = heat, so heat sink the BJT as appropriate.

I found that a 2N2222a biased with over 22 mA emitter current gave a stellar output return loss and low distortion. Within reason, for different transistors, keep the bias and feedback resistors constant and change the emitter resistor (100 ohms in my amp) to set the current you want or need.

Many builders follow this amp with an attenuator pad to preserve the input return loss.

Diode Ring Mixer

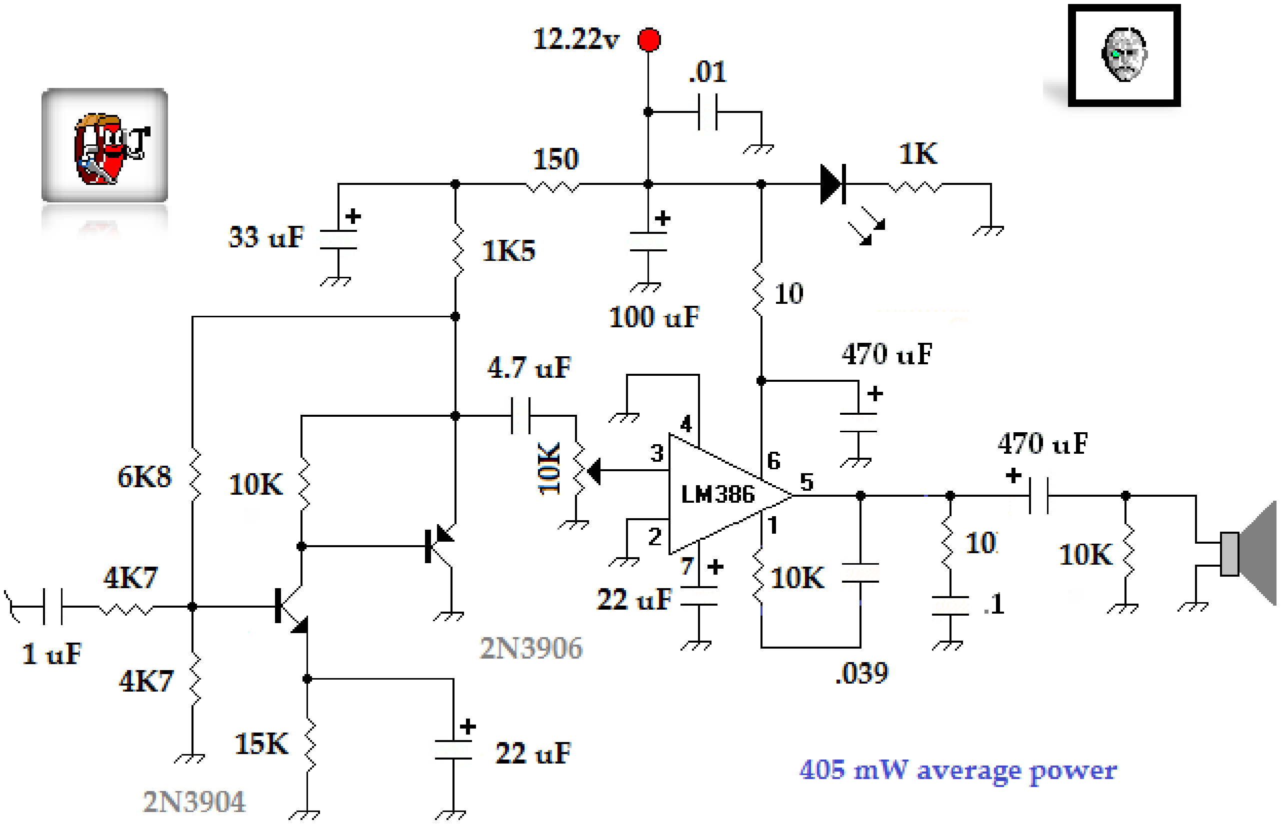


Above — A Minicircuits TUF-1 diode ring mixer was used in standard configuration. 7 dBm LO drive.

Above — The DRM module. It's hard to photograph inside a solder laden chassis. Connections are short.

Popcorn Audio Frequency Power Amplifiers

Popcorn Receiver Audio Amp



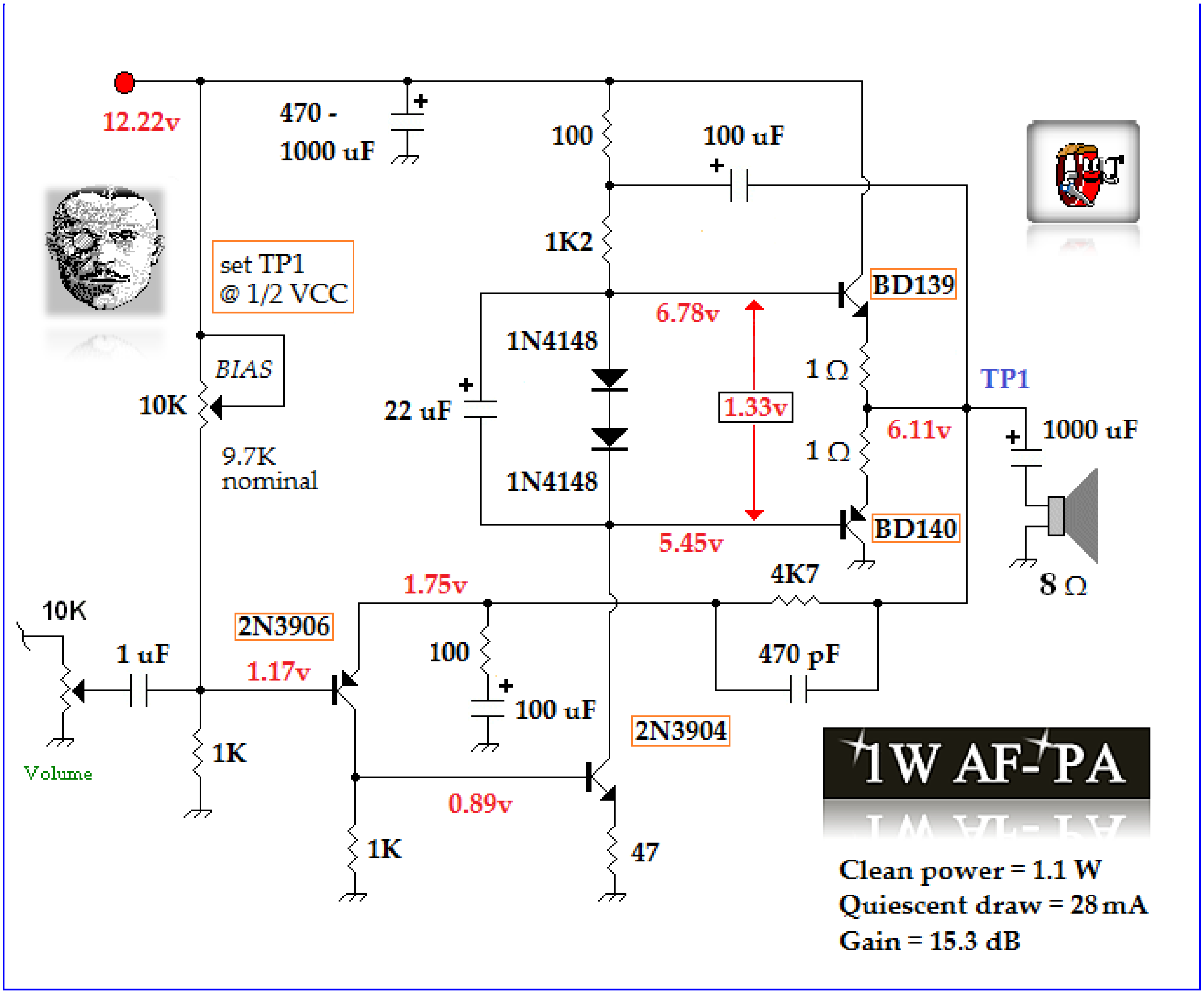
Above — Popcorn receiver audio power amp. I wanted a simple audio stage for testing popcorn receivers (to follow a high output impedance preamp device). Completing this module means never having to build such an amp again. The voltage gain is provided by BJTs to keep the noise down, but the popcorn factor up. The preamp impressed me with its strong signal handling capacity via feedback and careful biasing. The NPN is center biased so that when its intentionally distorted during testing, the positive and negative halves of the AC waveform distort equally — it provides a nice, big, AC voltage swing. An LM386 in X20 gain mode with some bass boosting comprises a reasonable power amp section. The 10K resistor on the output discharges the 470 uF cap when no speaker is connected to avoid a loud pop. The 4K7 series input R can be lowered, or omitted for more sensitivity.

Some builders might employ the LM386 in a higher gain mode at the expense of fidelity, or just wire up a TDA7052. I think in popcorn circuits, what really matters is that you understand what you're doing and try to *design* rather than just copy the "usual circuits".

Above — The popcorn AF amp in a clear blue chassis. Phono jacks provided a connection for the input and output — they 're inexpensive and readily available. The DC supply is connected to uninsulated banana jacks on the rear; it's well decoupled (resistor) + bypassed (capacitor) to help stop parasitic AF feedback. This amp is pretty quiet, considering its junk box legacy.

Above — The project with the top cover removed. The board is secured by the ground wires connecting it to the pot, jacks and DC voltage posts. The input is on the right.

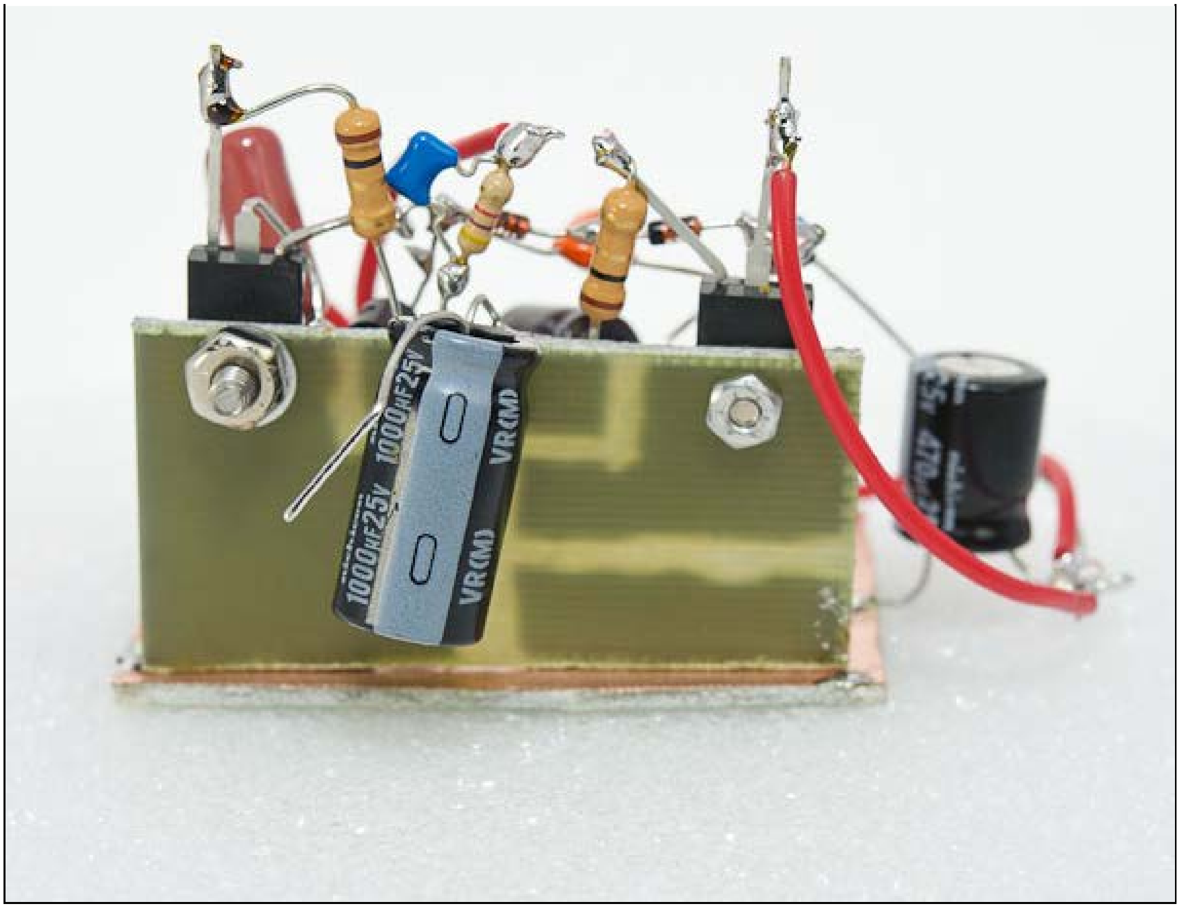
I like a relatively simple, lower gain AF amp on the bench for receiver development. You can use such an amp to decide on how much overall AF gain is needed, how you'll distribute it, and not have to deal with unwanted AF feedback.



Above — 1 watt popcorn audio power amplifier. Built around the BD139/140 complimentary pair - I achieved a clean I KHz sine wave at 1.1 Watts power after testing + tweaking my prototype design. I chose the familiar series diode pair to bias the power followers into Class A/B; an amplified diode (transistor level shifter) might be a better choice.

Bootstrapping the 2N3904 voltage amp pumps up the clean signal power capacity. The 2N3906 establishes the bias for the 2N3904 and the BD139-140 pair. Set the 10K bias pot so that the DC voltage at TP1 is 1/2 of the VCC. During testing, the AC voltage was centered perfectly between the DC rails and when pushed into clipping, the positive and negative AC waveform distorted nearly equally. Quiescent current = 28 mA ; not meant for a field-portable receiver.

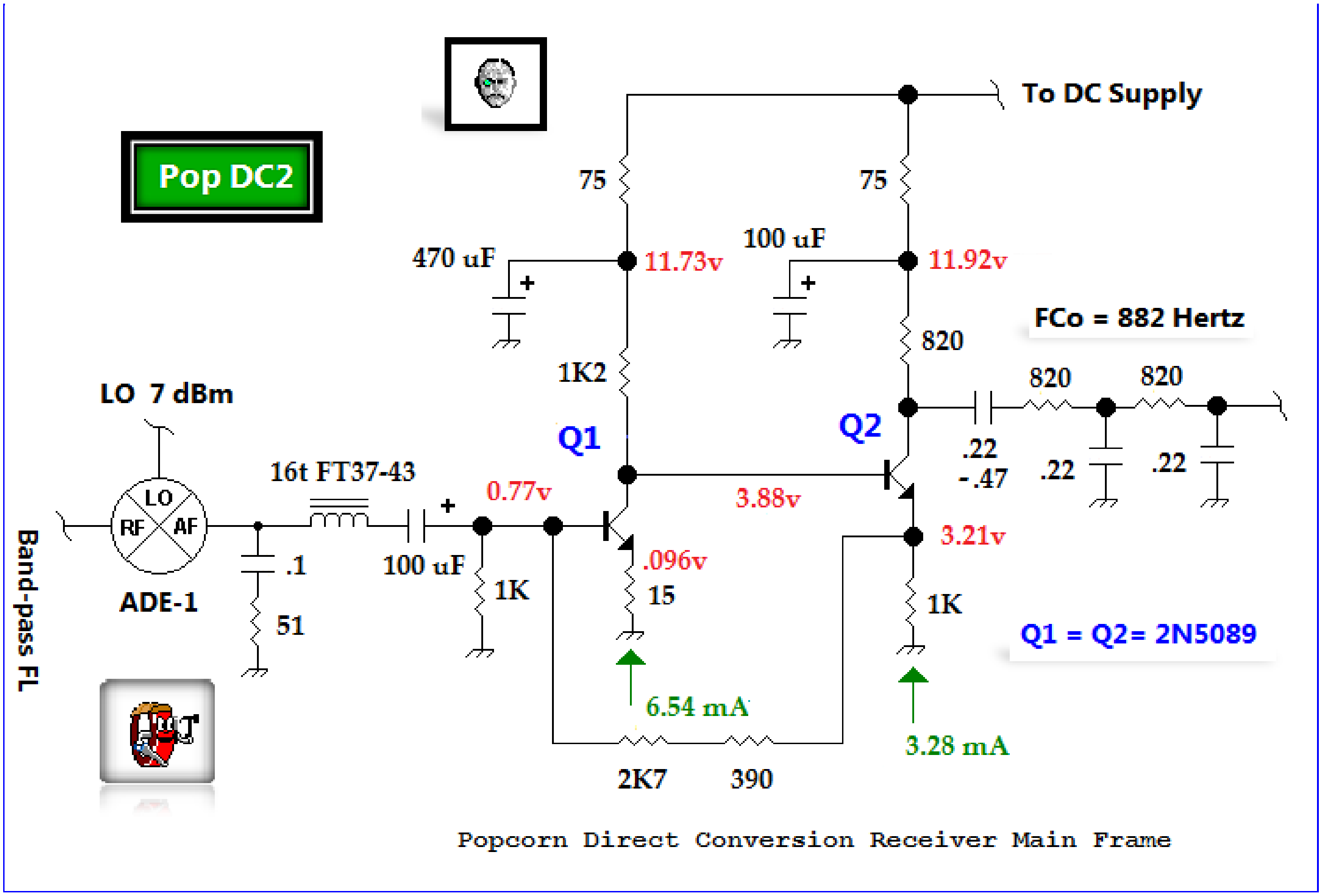
Click for a photo of my breadboard. Copper clad board serves as heat sinks for the power followers. The BD139 and BD140 make great complimentary transistors for audio frequency power amplifiers. With an Ft of 190 MHz, the BD139 can work okay as a driver or even the final in modest power QRP transmitters.



Above — 1 watt Audio PA (reverse view).

Pop DC2 — Popcorn Direct Conversion Receiver Main Frame

These circuits update the Popcorn DC receiver from 1998 and includes all components from the product detector through to the speaker, minus the VFO and band-pass filter.



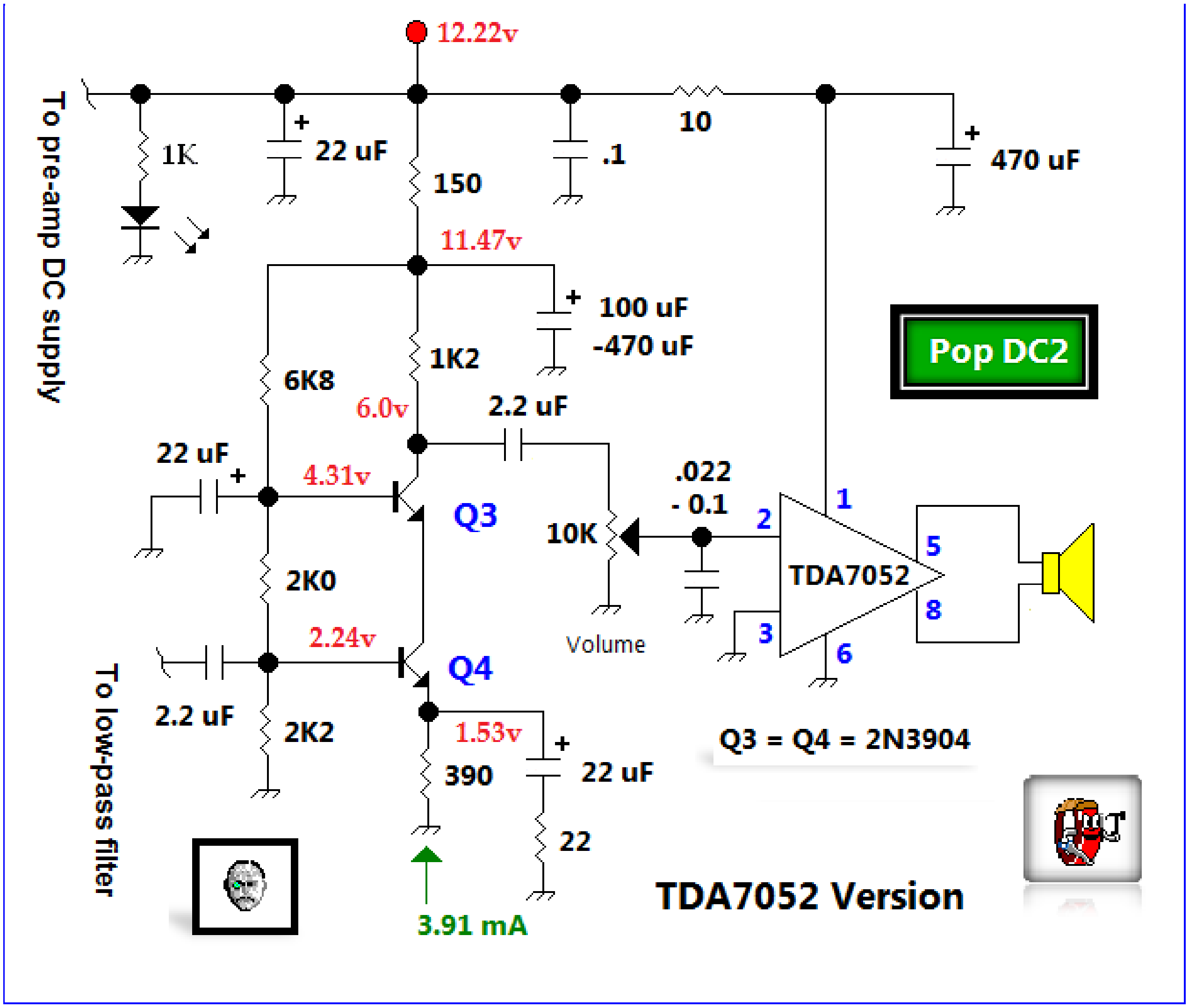
Above — The mixer and first audio preamplifier The 0.22 uF to 0.47uF cap connecting Q2's collector to the low-pass filter network exerts a high-pass response to remove low frequency noise and potentially any hum. I heard no hum, although a 470 uF filter capacitor on Q1 helps ensure that. Increase the 100 uF filter capacitor filtering Q2 if you hear hum or motor boating. The diode ring mixer exhibits AF that's hard to beat — very dynamic, vibrant and lively. I enjoyed the low microphonics with the double balance + a return loss of over 25 dB on all of 3 of its ports. Alternate photo.

I've read negative comments about my use of "those big filter capacitors" — 1 thing radiophiles can learn from audiophiles is that to adequately decouple and bypass means we need to stop fooling around with the usual 22 - 47 uF capacitors and *really* bypass. Viewing well designed AF amplifiers informs us so; these designers *really* filter their amplifiers from the DC supply. You can always increase the decoupling resistor value to allow use of a smaller capacitor value, however, we only have a single power supply at around 12 VDC, and I dislike giving up too much of it for DC filtering purposes. Do what ever amuses you.

Click for some analysis of the preamp. The MPSA18 went obsolete in 2011, so I chose the low-noise 2N5089 for Q1 and Q2.

The Popcorn DC2 receiver keeps the format of the earlier version; discrete transistors for all but the power amp and R-C low-pass filtering. The filter still allows you to listen to SSB, as there aren't many poles and the cutoff is nearly 900 Hertz —it just removes the ice-pick in the ear often heard in unfiltered DC receivers. You can change the capacitor values for a different cutoff frequency. Applet E performs this function.

Second Preamplifier Stage with TDA7052 final



Above — The 2nd pre-amp and AF power amp. Experimenting with a number of audio stages, I decided on this cascode common emitter / common base amp biased to provide temperature stability, high gain, low distortion + proper termination of the low-pass filter. (The input Return Loss = 19 dB in my 820 ohm bridge set up). Increase the 100 uF filter capacitor on Q3 up to as high as 470 uF if you hear motor boating (low frequency thumping). This stage is prone to feedback since it's directly connected to the power amp. Photograph.

The simple and effective TDA bridged amp has a fixed gain of 40, so this receiver isn't crazy loud, however, it sounds okay. The bypass capacitor on Pin 2 filters hash noise and *can* remove some of the high frequency din from off frequency stations. Experiment to find the best value for your ears; even 0.015 uF might be your preference. I chose a 0.047 uF for my final version.

The 2 uF coupling caps between the power amp and Q3 can be lowered or raised to suit your parts collection. All the audio path coupling or bypass capacitors were "polysomething" types in my bread boards.

This is a base station receiver since the quiescent current draw listening to noise = 37 mA.

For low parts count or beginner's receivers, IC audio power amps make sense; 1 chip and you're done. Consider, for example, the TDA7052 — 2 bridged amplifiers supply reasonable power and headroom in an 8 pin DIP package. A good, but imperfect part. Depending on your goals and abilities, the limitations of the 7052's fixed 40 dB gain and/or the inability to drive grounded loads or insert additional feedback networks may constrain your designs.

Sound Bytes on 40 Meters:

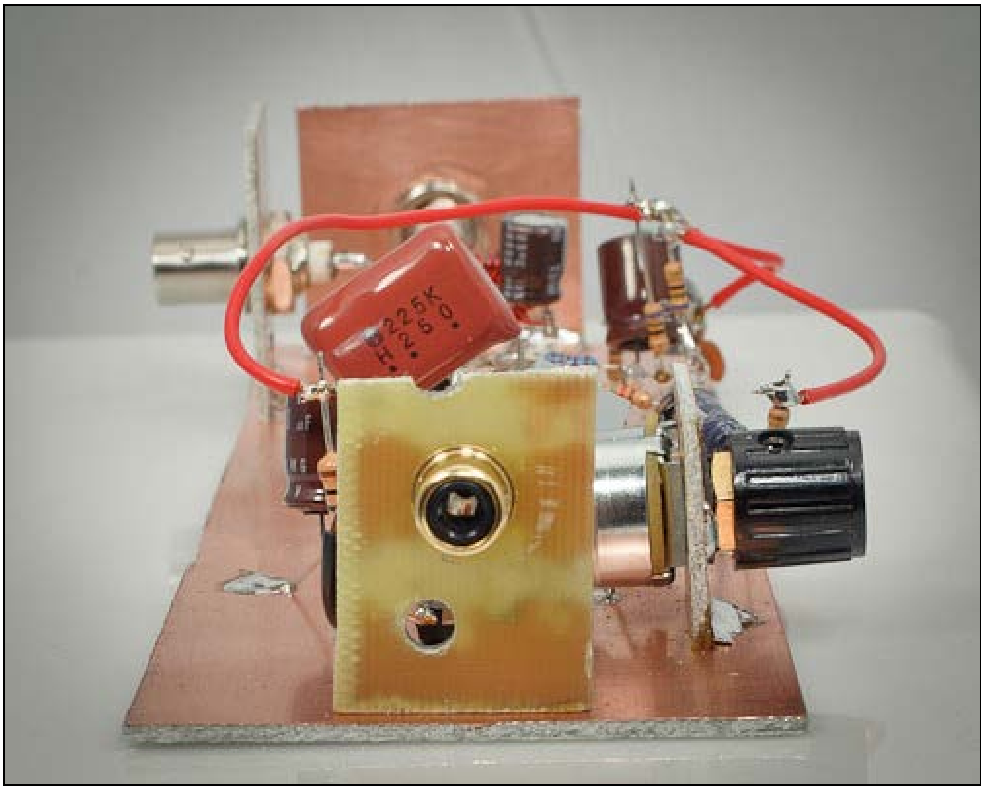
I recorded these sound bytes prior to adding a 0.047 uF bypass capacitor to pin 2 of the 7052 chip and increasing the coupling cap on Q2 from 0.22 to 0.47 uF.

For a control — An ICOM superheterodyne receiver with digital IF filtering set to wide (2.2 KHz) was recorded immediately after recording the Pop DC2 receiver (although I pressed the middle (900 Hz) and narrow (600 Hz) filter selection briefly, but they made the noise worse), The antenna is a 1/4 wave vertical in a city lot with noisy conditions. I don't believe in artificially making my stuff sound better than real, and present warts-and-all audio files. I compressed these files heavily so you'll hear the noise phase shifting a little. Normally with this antenna, a "quiet" QRN level is S9; it doesn't bother me. Icom

[Pop DC2](#) — I was tuning through a pile up to hear how the receiver copes with all the signals (twice as many with a DC receiver!)

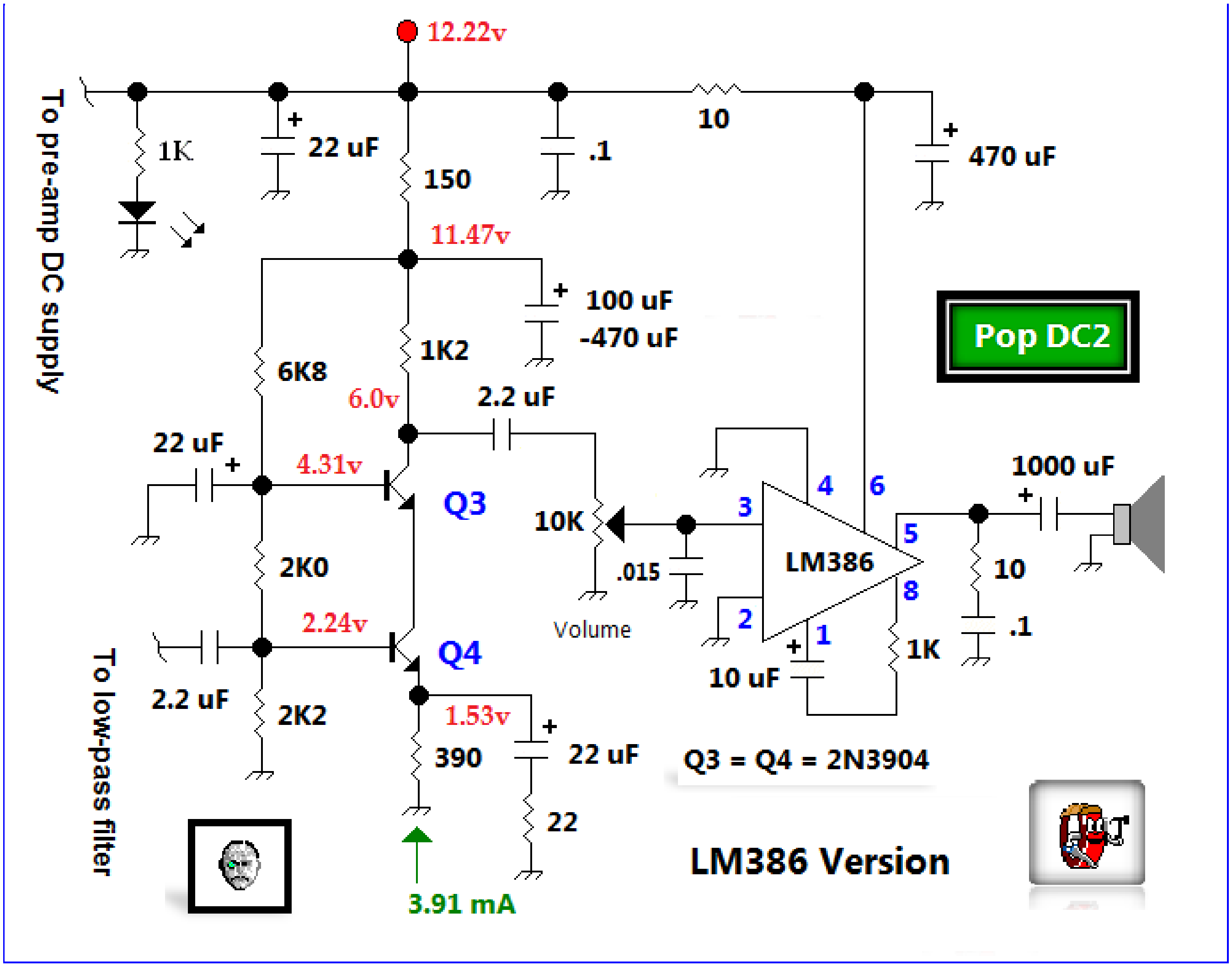
[Pop DC2](#) — More QRN, QSB and pile ups.

[SSB](#) - After this, I changed the .22 coupling cap between Q2 and the R-C filter .47 uF to add a little more bottom end.



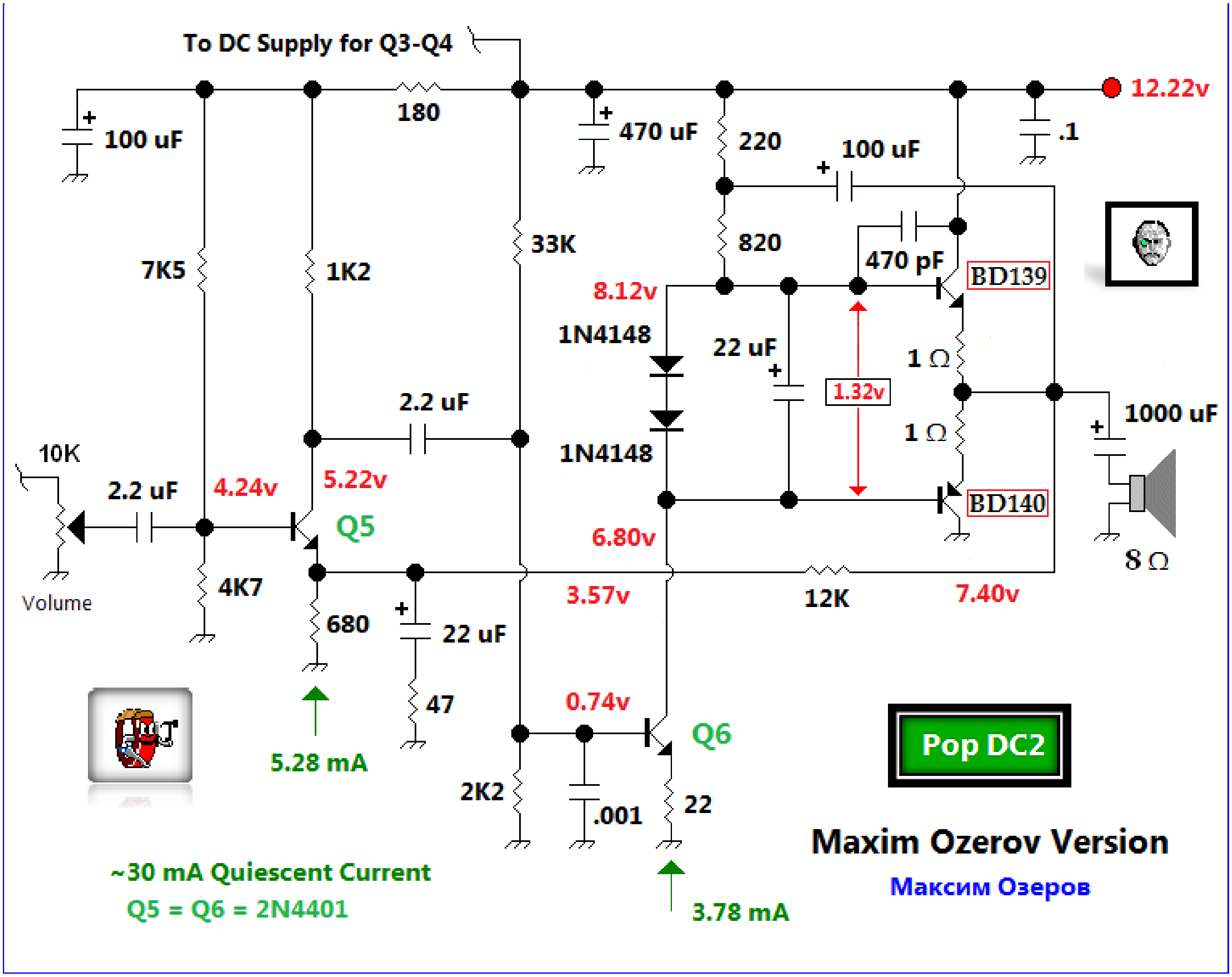
Above — Speaker terminal (an RCA jack isolated from ground on the TDA7052 version). Volume pot at right.

Second Preamplifier Stage with LM386 final



Above — The 2nd preamp and final with an LM386 set for just under a gain of 50. Click for a Canadian SSB sound byte.

Alternate Final Amp Stage that connects to the Q3 volume potentiometer



Above — A reader called Maxim Ozerov requested a discrete semiconductor version of the power amp after I posted the 7052 version on my blog. Placing 2 voltage amps inside the negative feedback loop proved challenging, since I'm no expert and learn on the bench. The gain = ~42 and the maximum pure sine wave power before clipping begins to occur = ~625 mW. This amp is louder and sounds warmer than the 7052 version.

2N3904s work fine for Q5 and Q6, but I found that the 2N4401 had a consistently higher DC beta and this helps ensure the bias and collector resistors shown will provide the widest possible, pure AC signal swing.

This amp replaces the earlier IC power amps (connects to 10K volume pot after Q2, however a 2.2 uF coupling cap is required after the volume potentiometer). If you need more **voltage** gain, increase the value of the 12K negative feedback resistor. Above 75K, the gain will approach 50 and *greatly* increase the possibility of distortion.

My dummy load for development and testing = three 1/2 watt resistors in parallel: 75, 82 and 10 ohms.

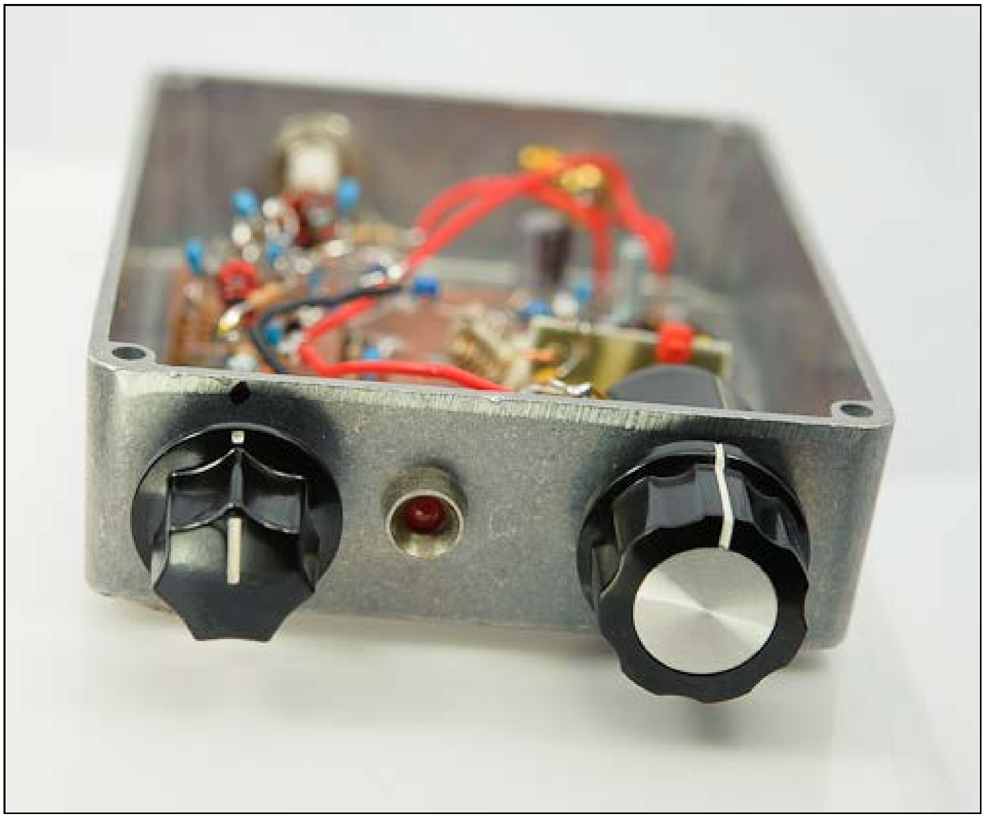
Certainly you can craft better - louder - quieter audio stages with low noise op-amps, however, my readers write that they enjoy building up discrete transistor designs, and for popcorn receivers; I do too.

Sound bytes from November 1, 2011

40 Meters - QRN is lower tonight. Some audio from a lineout tape deck (no tone controls nor equalization). I have only 2 cassette tapes, This one is Russian language from 1983 - the 1 strong accented syllable generates good peaks for AF listening tests. Audio recorded from an 8 ohm, 18 cm (7 inch) speaker mounted in a wooden frame with no back. Speaker choice and cabinets are critical and often overlooked; again we may look to audiophiles for guidance

Click for some 50 ohm AF preamplifier experiments cut from this page.

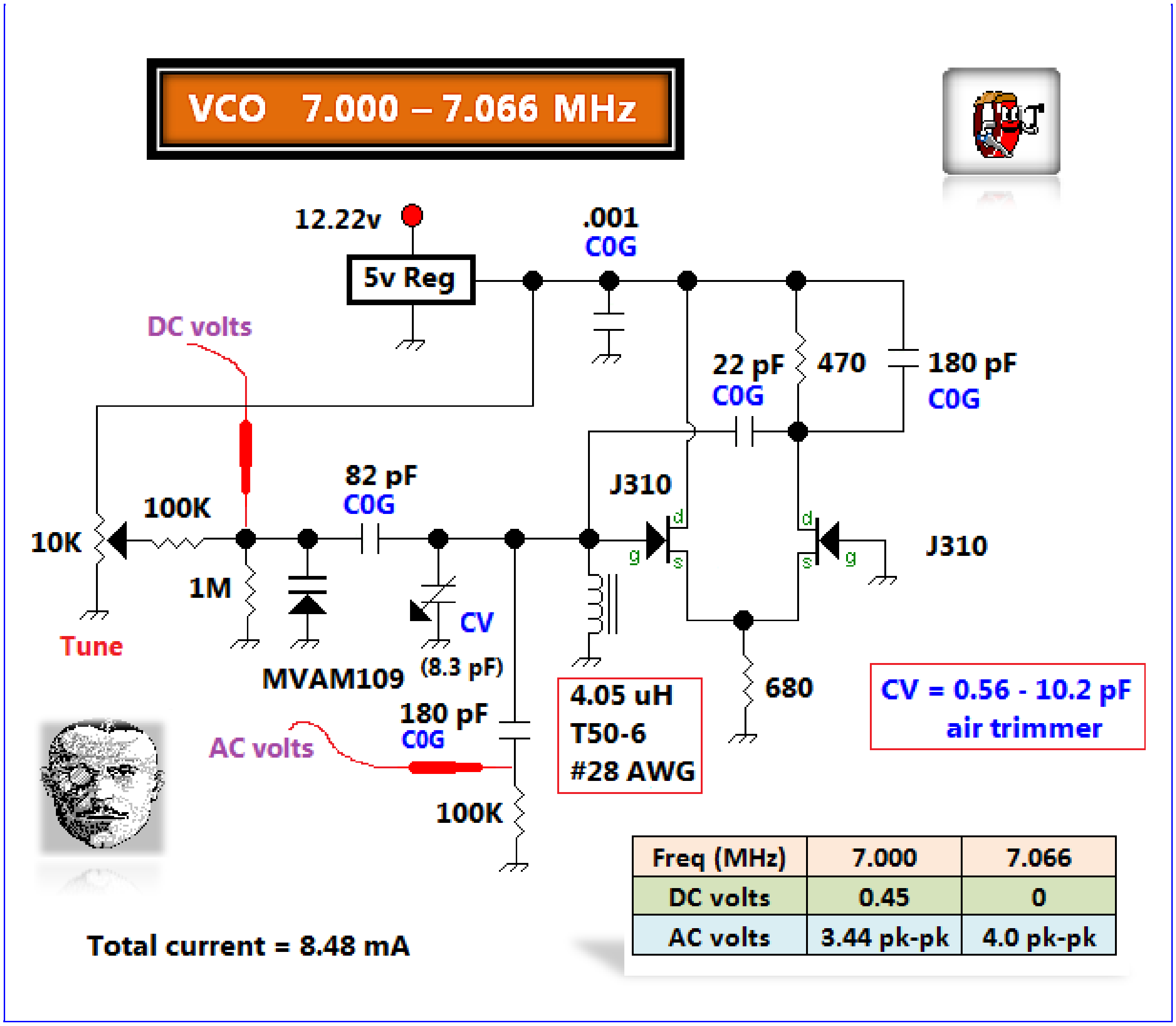
7 MHz VCO Experiments



As RF designers and builders, we rely on signal generators for nearly every experiment. I sought a reliable 7 MHz voltage controlled oscillator and built 1 after some effort. I'll describe and critique a VCO I rapidly designed for a reader and then present a better VCO with some design ideas.

7 MHz VCO Experiments: A rapidly developed Popcorn 7 MHz VCO

A reader needed a 7 MHz VCO in a hurry (3 hours); he only had 1 MVAM109 varactor and wanted to cover the bottom 60 KHz of the 40 Meter Ham band using a linear taper 10K potentiometer for tuning. He planned to use a dual-gate MOSFET cascode buffer (good choice), so I didn't have to bother with a buffer.



Freq (MHz)	7.000	7.066
DC volts	0.45	0
AC volts	3.44 pk-pk	4.0 pk-pk

Above — The VCO with a 100K resistor as the temporary buffer. He'll use a 100K resistor on G1 of the 2-gate MOSFET buffer. With a Q of 150 at 1 MHz; high noise level and a hyper-abrupt capacitance-versus-voltage curve designed for tuning AM radios, the MVAM109 varactor ranks poorly. The C of my MVAM109 with no reverse DC voltage was 725 pF.

Still, this VCO tuned in a linear fashion, showed a nice sinusoidal output and proved frequency stable. I wanted the AC voltage at the varactor anode at under 1 volt pk-pk (it was 752 mV) to help reduce forward conduction during the positive AC voltage swing. I was bad and ran the tuning DC voltage from 0 to 0.45 volts which greatly increases the potential for forward conduction in a varactor. To mitigate this somewhat, an 82 pF couples the varactor to the tank and drops the AC voltage and reactance seen by the varactor.

In VCOs on the web and print, you'll often see builders connect their varactor to a high Z, and high AC voltage point in the VFO tank; whoa!

At HF, if a varactor is forward biased by the positive half of the AC signal, varactor leakage current and voltage-source loading increases momentarily and lowers Q + broadens tuning. Further, serious harmonic energy and phase noise might be generated as the varactor is biased positive and negative alternately. You can sometimes see distortion in your scope during experiments with extreme AC voltage swings across the varactor. The varactor coupling capacitor should be as low as possible.

Balanced varactor tuning (anode to anode) provides another way to reduce AC signal effects at the cost of reduced maximum capacitance since the 2 varactors are in series. With back-to-back varactors, as the AC signal swings, the varactors are driven into high and low capacitance alternately, but the net capacitance remains constant. Thus applied reverse DC voltage sets the varactor capacitance rather than AC signal amplitude.

The reader for whom I made this impromptu circuit can lower the AC tank voltage by decreasing the VCC or increasing the 680 ohm source resistor after installing the buffer and tweaking things for a 7 dBm output voltage. This topology suffers from an amplitude versus frequency issue — at 7.0 MHz, the output = 3.44 volts pk-pk and at 7.066 MHz the output rises to 4.0 volts pk-pk.

Stuck with an MVAM109 constraint and 3 hours to design/build a VCO, I share this circuit as a raw experiment; not an example of good design because it is not. I took the signal off the gate to derive the best sine wave; this requires a lightly coupled, high impedance buffer with strong reverse isolation to prevent the pulling of the VCO frequency by downstream changes.

A lower L + higher C in the tank, and/or a higher Q varactor could turn this VCO into something reasonable. Popcorn versus high performance? You choose!

7 MHz VCO Experiments: A Suitable 7 MHz VCO

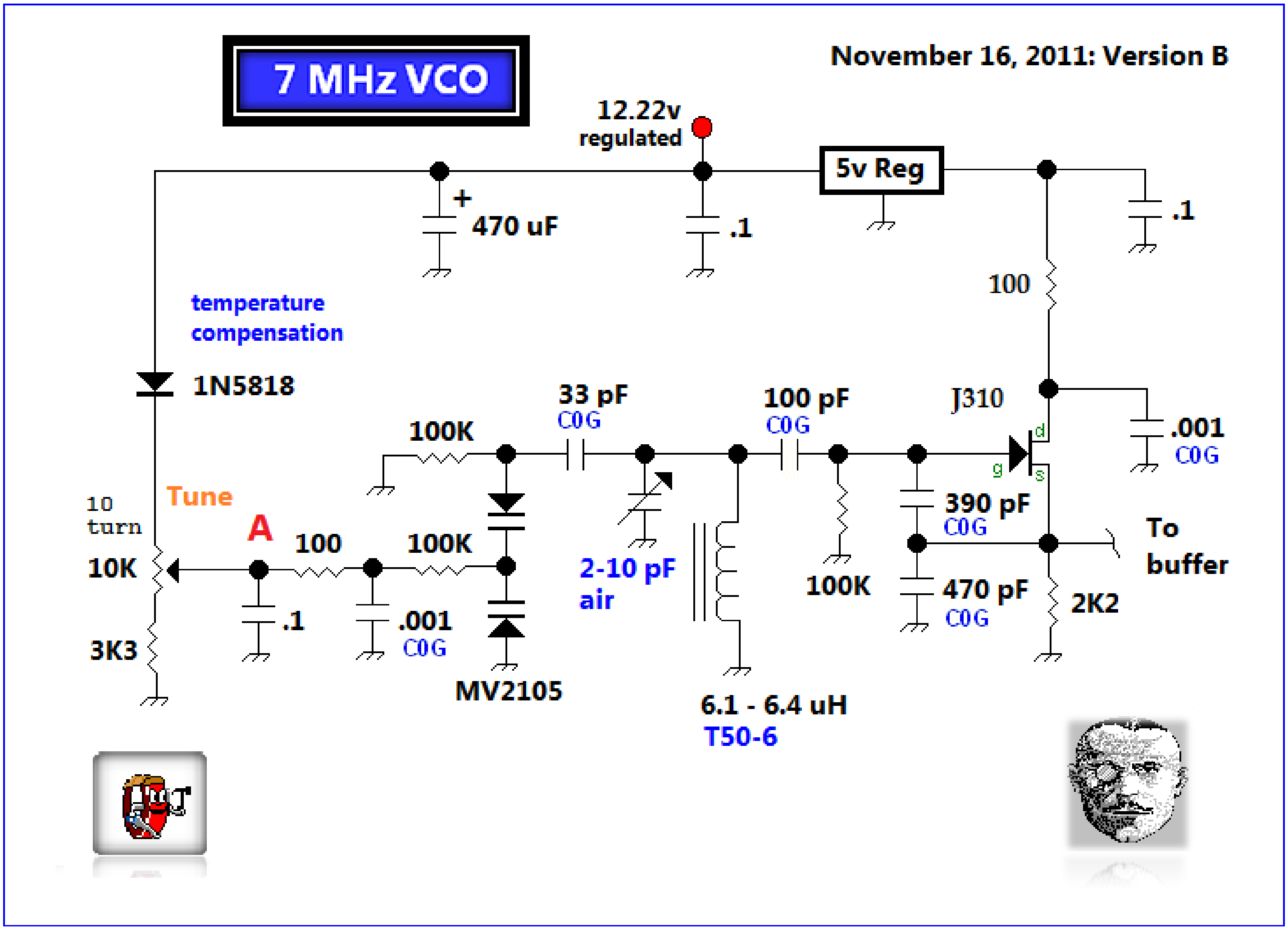
PART 1: Introduction

During my Fall 2011 VCO experiments I studied books including EMRFD and built versions of EMRFD Figures 4.33 and 4.34. Figure 4.33 is a common-base Colpitt's Oscillator using a hyperabrupt varactor. On Q1, the 33 ohm resistor in series with the 0.1 uF cap "de-Q" the 2N3904 to reduce UHF oscillations. Wes also employs current limiting with a 1K5 emitter resistor.

The temperature drift compensation circuit involving a temperature sensitive reference diode + op-amp fascinated me — astute temperature compensation design. I built and tested the whole circuit; the VCO has some amplitude versus frequency and phase noise issues, but it's okay for general use and *great* for varied environments. After tackling Figure 4.33, I built and tested the JFET Colpitts oscillator in Figure 4.34 and share my experiences developing this VCO with an alternate buffer.

These circuits are not cookie-cutter / carbon-copy: they show raw design ideas from the bench.

PART 2: The Voltage Controlled Oscillator



Above — A JFET Colpitts VCO picked after after trying 5 different topologies. This VCO is my version of EMRFD Figure 4.34; originally designed by Wes, W7ZOI.

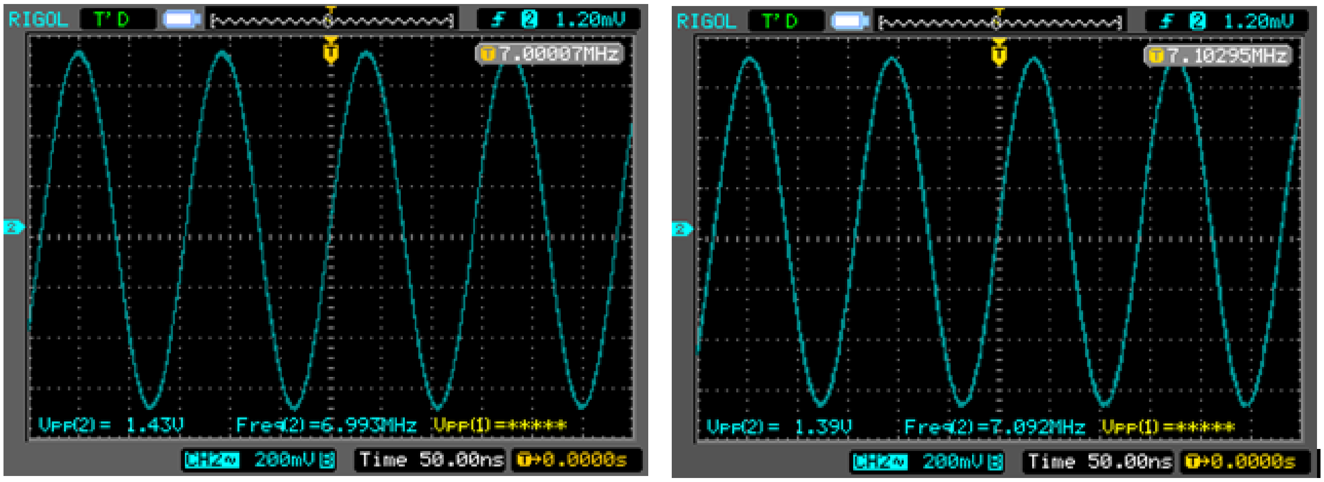
This JFET Colpitts oscillator exhibits a flat output versus frequency, low noise, scales easily to other frequencies and accomodates a wide variety of varactors. For example, you may scale it to other frequencies by changing the L and tweaking the "Colpitt's capacitors" up or down as needed.

I employed a small air variable trimmer capacitor to set the lower band edge and this meant experimenting with the inductor to find 1 that allowed me to set the band edge with such a small trimmer capacitor. I built 2 versions; in 1 the required L= 6.09 uH and in the other, L= 6.4 uH. It would be much easier to use a trimmer cap with a larger capacitance range as it makes chosing the inductor less exacting.

With the trimmer shown set to half its range, I started with a 6.6 uH coil and remove 1 turn at a time until the output in a counter was close to 7.00 MHz. After permanently fixing the inductor, I tweaked the trimmer cap so the lower band edge was 7.000 MHz with the chassis lid on.

To further drop phase noise, you could reduce the 33 pF coupling cap, add another pair of anti-parallel varactors, run a higher C to L ratio, or perhaps decrease the source resistor to increase the current limiting. Also low resistance, high Q, SMT varactors would help lower phase noise — SOD parts are tiny, but test your hand steadiness and vision.

Tuning Range



DC @ Point A = 3.0 volts

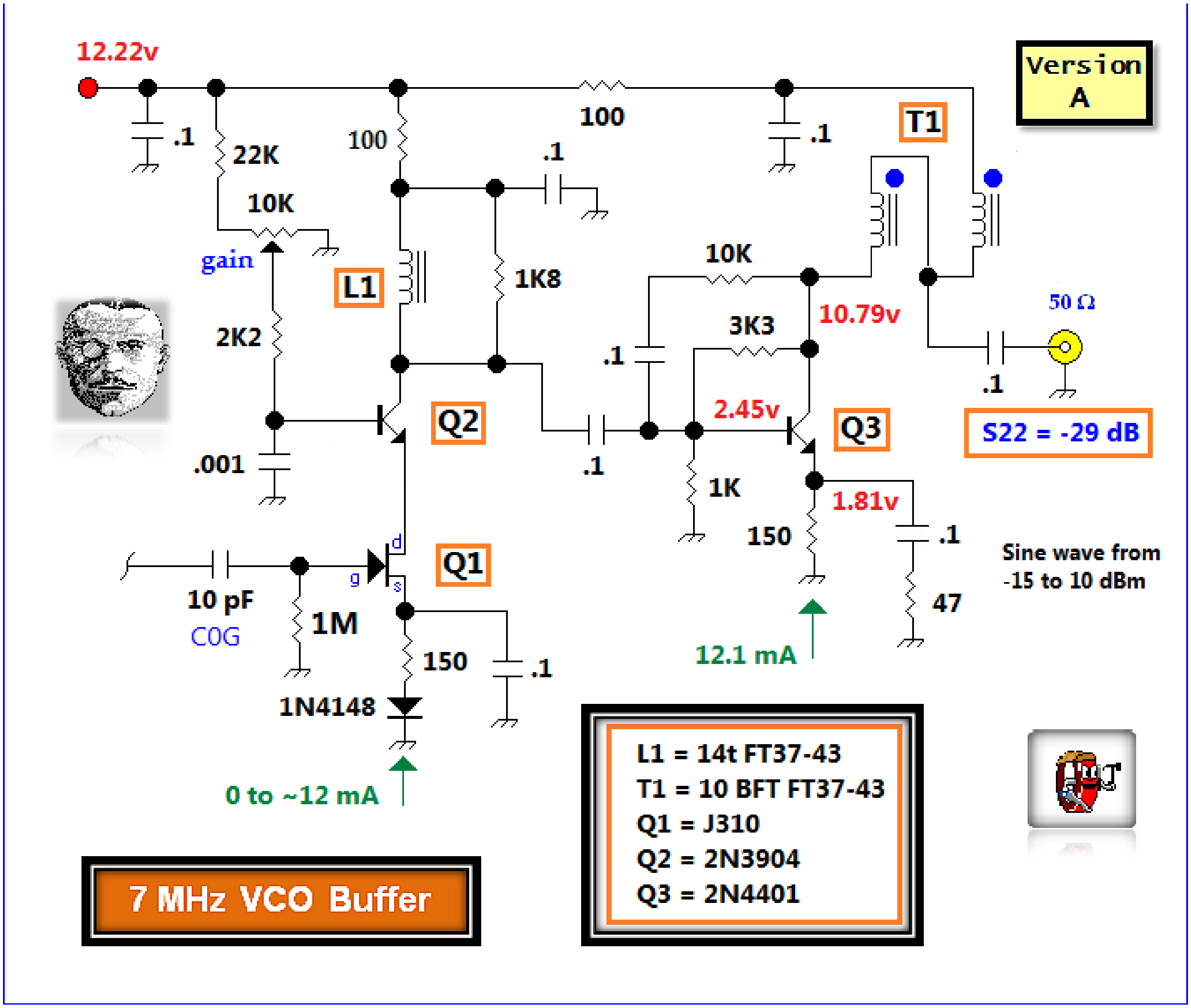
DC @ Point A = 12.21 volts

Above — When tuning from the minimum frequency and tuning voltage (7.0 MHz / 3.0 VDC) to the maximum tuning voltage and frequency (7.103 MHz / 12.21 VDC) the signal amplitude only changes 0.04 volts peak-peak.

I kept a minimum of 3.0 VDC on the varactors at the minumum frequency to provide reasonably linear tuning, keep the applied reverse voltage away from 0, and improve temperature stability. All were bench determined and are not factors you can generalize to all VCO circuits. Change the minimum DC voltage on your VCO control by adjusting the resistor on the grounded end of the pot; 3K3 in my case.

Click for a moderate resolution photograph of the VCO and buffer prior to adding the temperature compensation parts.

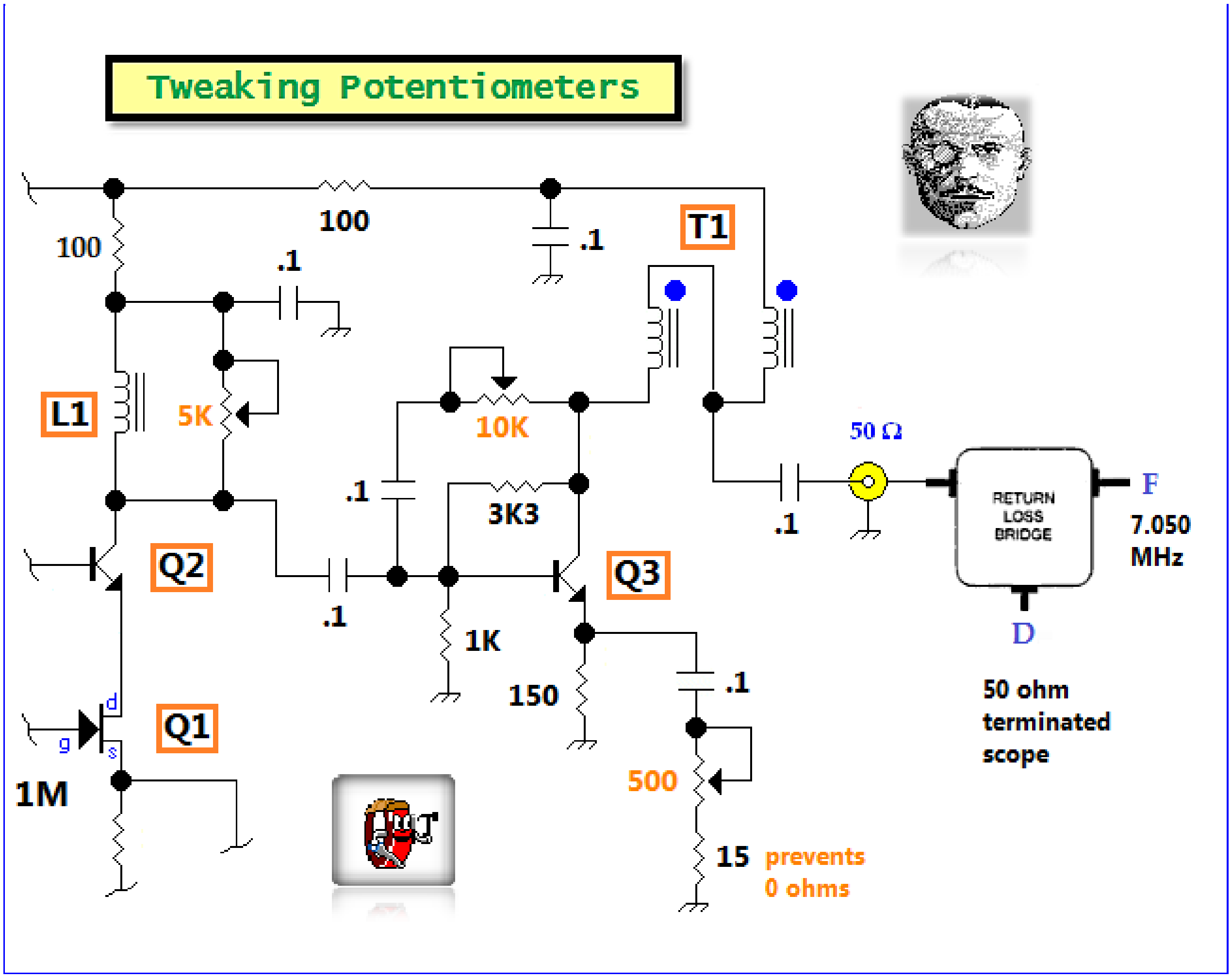
PART 3: The Buffer/Amplifer



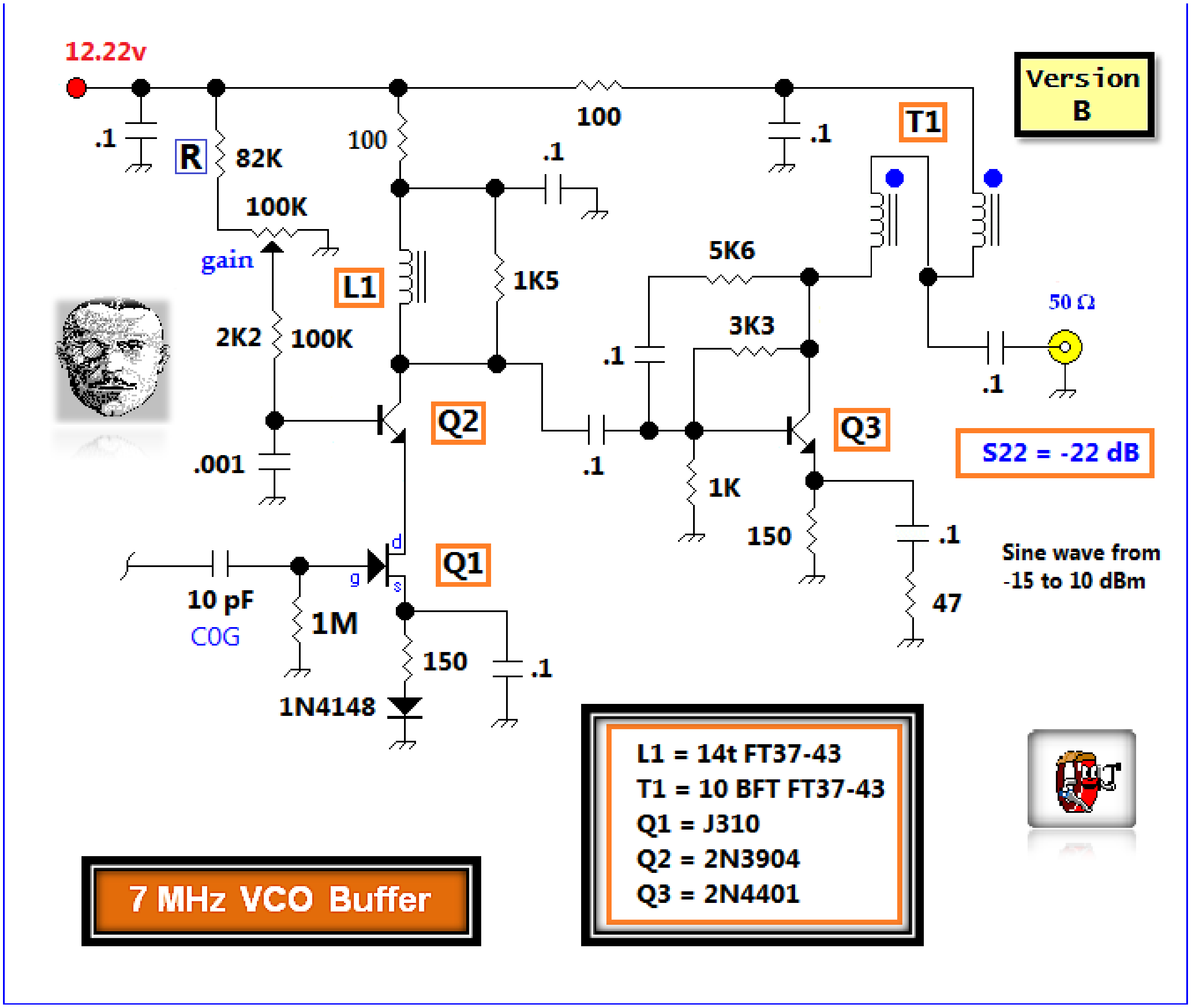
Above — The Q1-Q2 hybrid-cascode amp gives strong reverse isolation (nearly 70 dB) and front panel gain control. You could also employ a dual gate MOSFET or JFET cascode with either fixed bias, front panel control, or a trimmer resistor to adjust the bias on Q2

I enjoyed designing the Q3 final amp amp and matching its input impedance to the output Z of Q2. One way to establish a fixed + known output impedance in order to to get a strong return loss without tuned circuits/networks is to feedback some signal from the collector to the base. The difficulty lies in finding how much negative feedback to apply, while still DC biasing the amplifier for good temperature stability. I set up a crude experiment to determine the Scattering Parameter S22. The goal is to set up a good Q3 output return loss using feedback + matching the Q3 input impedance by tweaking the inductor resistor across L1 and adjusting Q3's emitter degeneration.

The return loss in my first prototype without any attenuator pad = 29 dB; some of this was pure luck.



Above — Q2 and Q3 with 3 variable **orange colored** resistors in-situ and a Return-Loss bridge connected to the output. The potentiometers are tweaked while watching the detected output in an oscillocope. Adjust all the pots for the lowest peak-peak voltage and then carefully remove each pot and measure its resistance with an ohm meter. Replace all 3 pots with the nearest equivalent standard value resistor. Then measure and calculate the return loss (negative of S22). Watch the Q3 emitter resistance — too little R might bring distortion.



Above — A seperate buffer built with 100% different parts that required different AC feedback plus shunt resistor across L1. The parts in this circuit weren't as hot as Version A, and the maximum output voltage was only 1.8 volts pk-pk. In order to get the AC output voltage to just above 2 volts, I had to tweak the resistor labelled **R**.

To keep the heat and current down in the final amp, I decided to keep the maximum clean output to 2 volts peak-peak (= 10 mW = 10 dBm) with an emitter current of ~ 12 mA. If you want higher clean output than 10 dBm, you'll have to run more Q3 emitter current and maybe choose a different BJT, plus apply a heat sink.

When cranked to maximum DC voltage, the Q2 gain pot allows a peak output AC voltage of ~2.2 volts pk-pk into 50 ohms and distortion is evident. At or below 2 volts pk-pk all is well — I'll use this VCO mostly from 0 to 7 dBm.

Since the circuits uses 2 BJTs and a JFET and many 5% tolerance resistors , the Q3 output will vary according to your parts. Tweak the resistor labelled **R** to provide a maximum AC signal just over 2 volts peak-peak into 50 ohms. This translates to around 3.8-5.5 volts DC bias for Q2 with your gain pot cranked fully clockwise.

Return loss variations. You probably noticed the return loss in Version B = 23 dB, while Version A = 29 dB.

Version B originally had the 1K8 shunt resistor across L1 and the 10K + 0.1 uF AC feedback arm just like version A and I measured a return loss of 22 dB. I stuck in 2 tweaking potentiometers (did not bother tweaking the the emitter series feedback element). After pot tweaking, the best return loss I could obtain with 5% tolerance resistors was 23 dB and this probably represents what the average builder will obtain. An S22 of - 22 to -23 dB works fine for the QRP work bench.

If you don't plan to do any potentiometer tweaking, I recommend building circuit A since it has a little more gain due to the slightly higher shunt resistor, and also I built 3 versions of Version A with an S22 of -22 dB or higher.

PART 4: Temperature Compensation

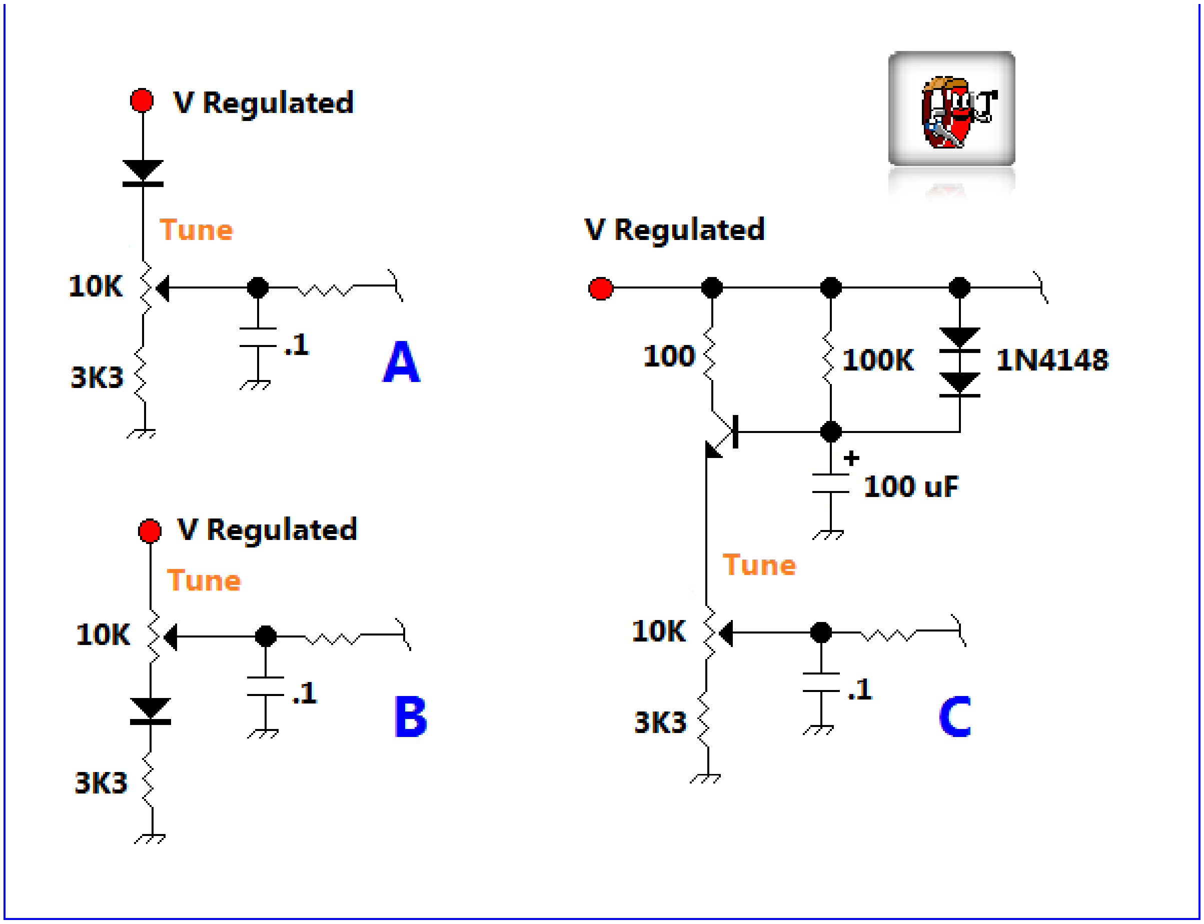
Before temperature compensation, my VCO slowly drifted down in frequency and was unusable.

If you look through the Ham Radio VFO/VCO literature, you will see that many builders use polystyrene caps as the Colpitt's capacitors, and/or in parallel with other NP0/C0G tuning capacitors. Negative temperature compensation caps like an N750, or the polystyrene types temperature compensated the oscillator. Negative temperature co-efficient caps are hard to obtain for many builders; especially in small quantities, however, they are worth their weight in gold.

Diode Compensation

Stabilize your VCO as much as possible with compensating capacitors and by following prudent temperature stability techniques before adding diode compensation. *See the VFO 2011 web page and EMRFD.* Temperature compensation is best performed in a homebrew oven (see EMRFD) and normally takes an incredible amount of time and patience.

Temperature compensating diodes are far from static — a diodes temperature co-efficient is dynamic and may vary with current and also unfortunately, with temperature and even while tuning your VCO !



Above — Simplistic diode temperature compensensation schemes.

The late, great, Doug DeMaw advocated sticking a 2N3904 or 2N2222a (wired as a diode) between the control potentiometer and the varactor decoupling network since the forward biased P-N junction exhibits a negative temperature co-efficient and should stop the decrease in frequency. It can help, however, as you tune and swing the control DC voltage from minimum to maximum the forward bias on the diode increases and the diode temperature coefficient decreases.

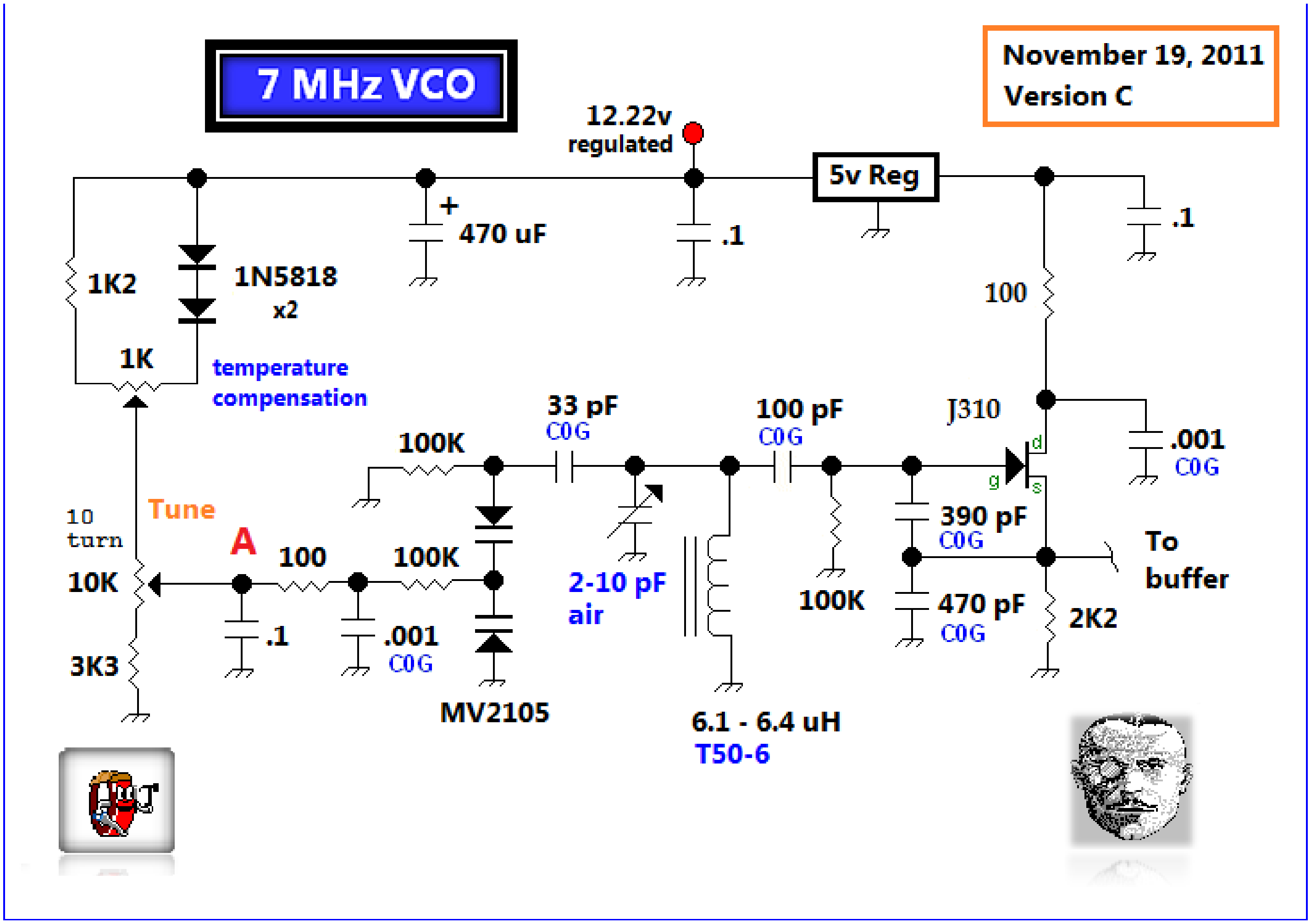
I've never had success using a transistor in this way; the BJT caused the VCO frequency to increase in an erratic manner that varied along with the DC control voltage. When watching drift in a frequency counter set to sample every second or so, a stable design will slowly change frequency in 1 or occasionallly 2 Hertz increments — some people call this "linear drift". if you see your VCO dropping down frequency in 10 - 20 Hertz jumps per second, you'll have a bad time temperature compensating.

I experimented with the above 3 designs that keep a constant current on the diode. Figures A and B work. I tried both and confirmed that a given diode compensated slightly differently when in circuit A or B. This gives you a bit of a tweaking room for your chosen compensation diode. I tried Figure C, but it had too much negative temperature coefficient and sent the VCO drifting upward about 1-2 Hertz each second.

I settled on circuit A and then tried some diodes: the initial best was a grubby old Germanium from my junk box. The best choice turned out to be a Schottky barrier rectifier (1N5818). I connected the VCO to a receiver and could listen to CW QSOs without groaning. My VCO now drifted up in linear, 1 Hertz hops at about 105 Hz per hour. It took a long time to tack solder in and wait 10 or 15 minutes for each diode to stablize before I finally settled on the 1N5818.

The better solution is to choose a suitable diode and vary its current to tweak the temperature compensation. Wes did this in EMRFD Figure 4.33.

Advanced designs may use a reference voltage + a temperature dependent voltage that is applied to op-amps in a proportional way to temperature compensate the DC control voltage. Then, too, some builders ovenize their VCO container to maintain a very stable environmental temperature.



Above — A simple and elegant diode compensation scheme proposed by Ken Kuhn. Basically, it lets you tweak the degree of compensation to what is really needed rather than accept what you get from a diode. Adding more diodes will increase the effect — but the 1.2 K resistor should be increased accordingly to roughly match the overall voltage drop of the diodes. Hopefully there is a point on the 1K potentiometer where temperature compensation can be very good at a tuning point of interest. The diodes should be located to thermally match the rest of the oscillator circuitry.

Set the band edge *after* finding the sweet spot on the 1K potentiometer since it will affect the tuning frequency. This experimental circuit cannot be casually copied and it took a while to converge to the desired operating point on the 1K potentiometer. Generally you start with the 1K pot towards the 1.2 K resistor and then adjust for the best stability after warm-up. Then repeat and adjust as necessary over time.

Temperature compensatiing an oscillator like this is a challenge as all parts have some temperature drift and it takes a lot of measurements (and often, some dumb luck) to determine the overall compensation curve that is needed.

My 1 hour drift up in frequency is now ~ 60 Hertz per hour at various tuning frequencies across the tuning range. I stuck with the 1N5818 diode, and probably should have tried other diodes and also changed the 1K2 resistor to observe any effects, however, I have spent an inordinate amount of time on this circuit and leave it to others, or future bench work to improve. *See QRP — Posdata below.*

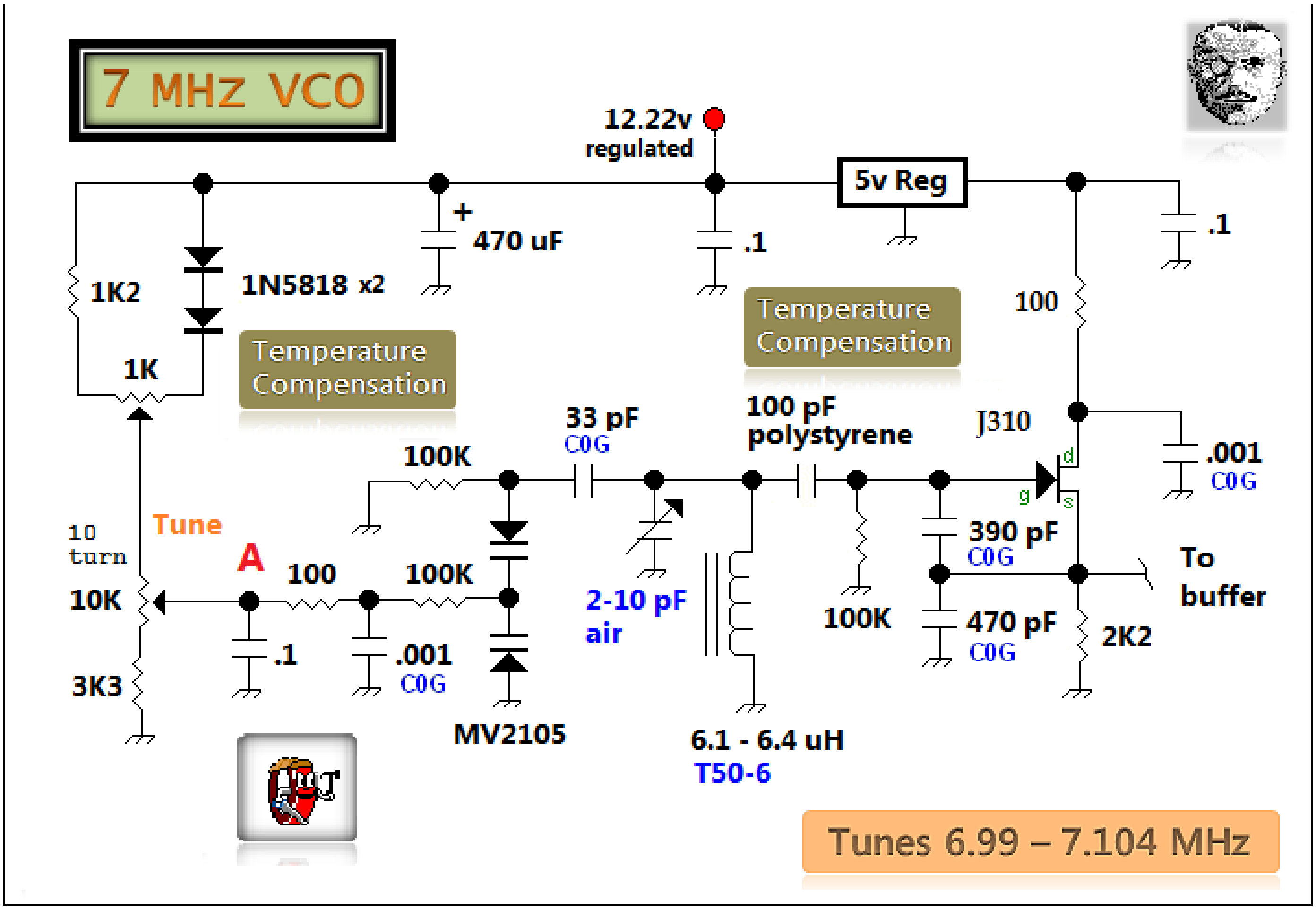
Above — Version C of the VCO. When I built the first versions, I drilled a hole to accomodate a third potentiometer, but filled it with a LED holder that was temperature sealed with epoxy glue. The 1K pot *jus*t fit into my chassis.

QRP— Posdata for January 2012

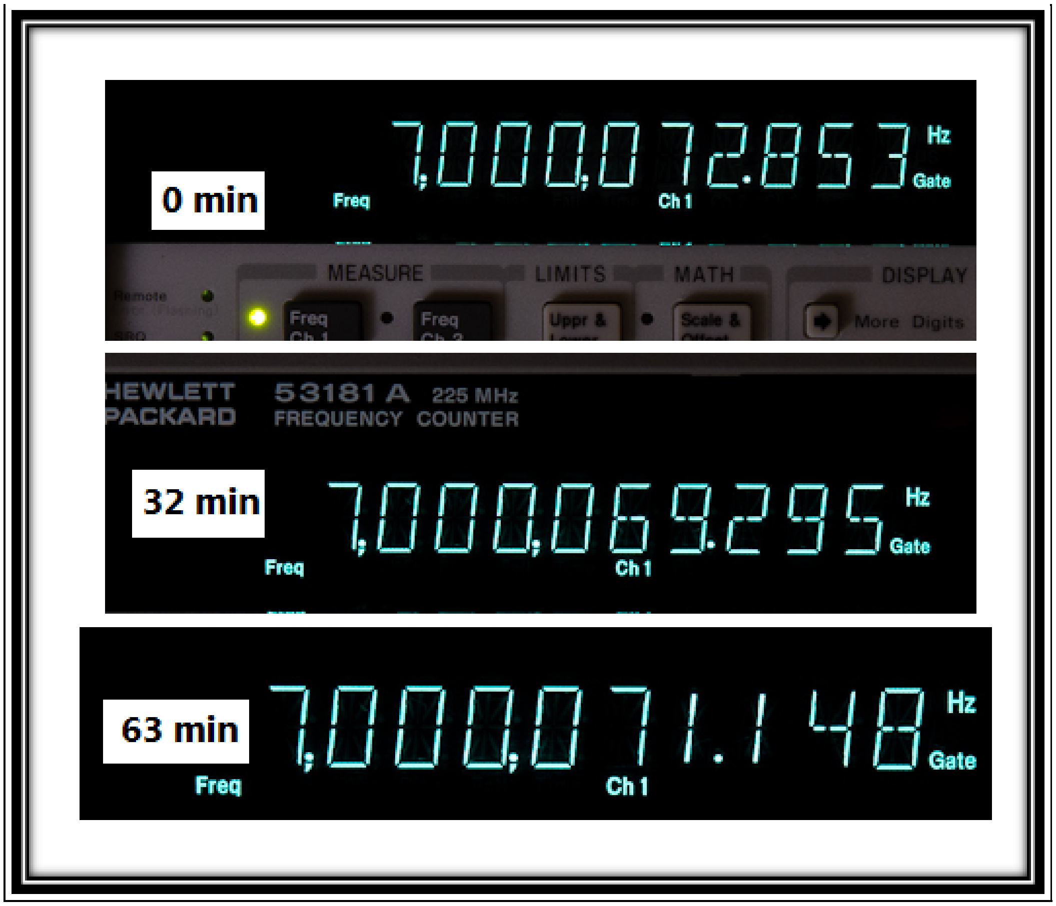
In late 2011, I shopped on eBay to build up a small quantity of 10 - 270 pF polystyrene, plus some 56 pF N750 ceramic temperature compensation capacitors. After 2 simple, but time-consuming experiments, I temperature stablized my 7 MHz VCO frequency drift to under 10 Hz per hour in the relatively constant temperature of our basement.

I didn't feel like re-doing the whole resonator circuit and thus focused on the tank to FET coupling capacitor. Placing a N750 capacitor in parallel with a fixed NP0/C0G cap to make 100 pF resulted in over-compensation and no amount of tweaking on the "adjustable diode" circuit worked. A few hours late, I swapped in a 100 pF polystyrene capacitor and after further hours of waiting and tweaking, I nailed the frequency stability sought.

An oven provides the best way to temperature compensate, however, whether you choose the oven method, or the bench method like I did, great patience is required to see if a change to your TC circuitry works or not.



Above — Final version of the 7 MHz VCO. I changed the 100 pF capacitor coupling the JFET to the resonator circuitry from C0G to polystyrene and slowly tweaked the 1K temperature compensation pot to find the point of convergence.



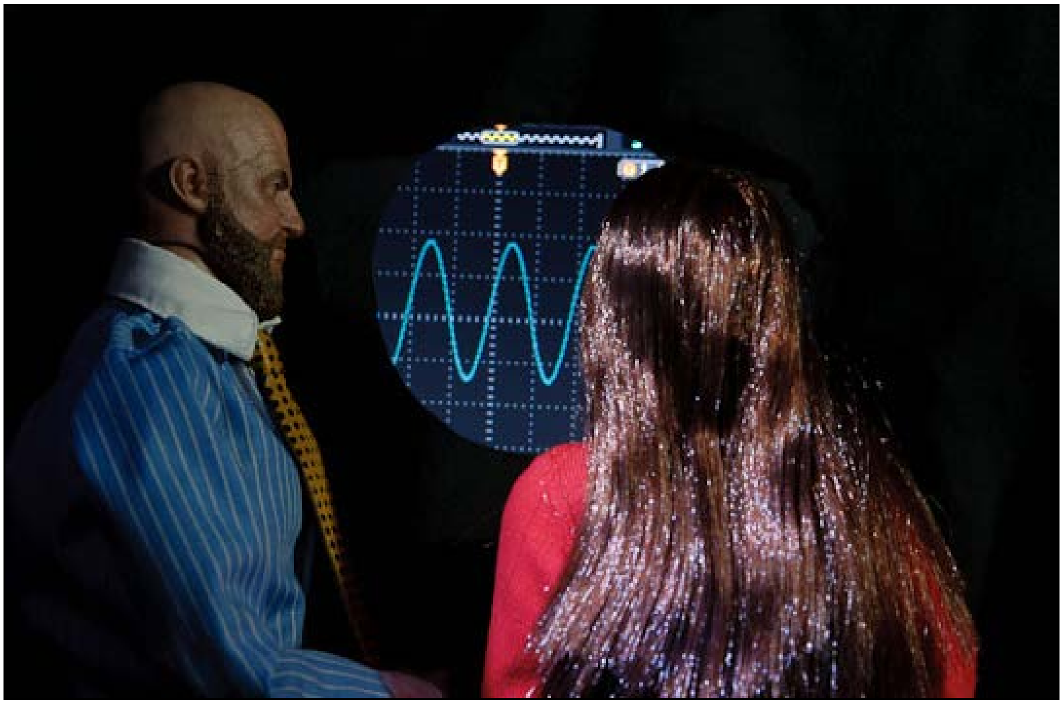
Above — The ~1 hour drift after a 30 minute warmup period for the 7 MHz VCO. Love this.

Above — My new temperature compensation capacitor parts drawer. I'll keep an eye out for further bargain temperature compensation parts on eBay and at Ham Radio festivals.

Fearless Leader and Hero   Храбрый вождь и герой

Above — Professor Vasily Ivanenko ( ), fearless leader (ТЫ МОЙ ГЕРОЙ)

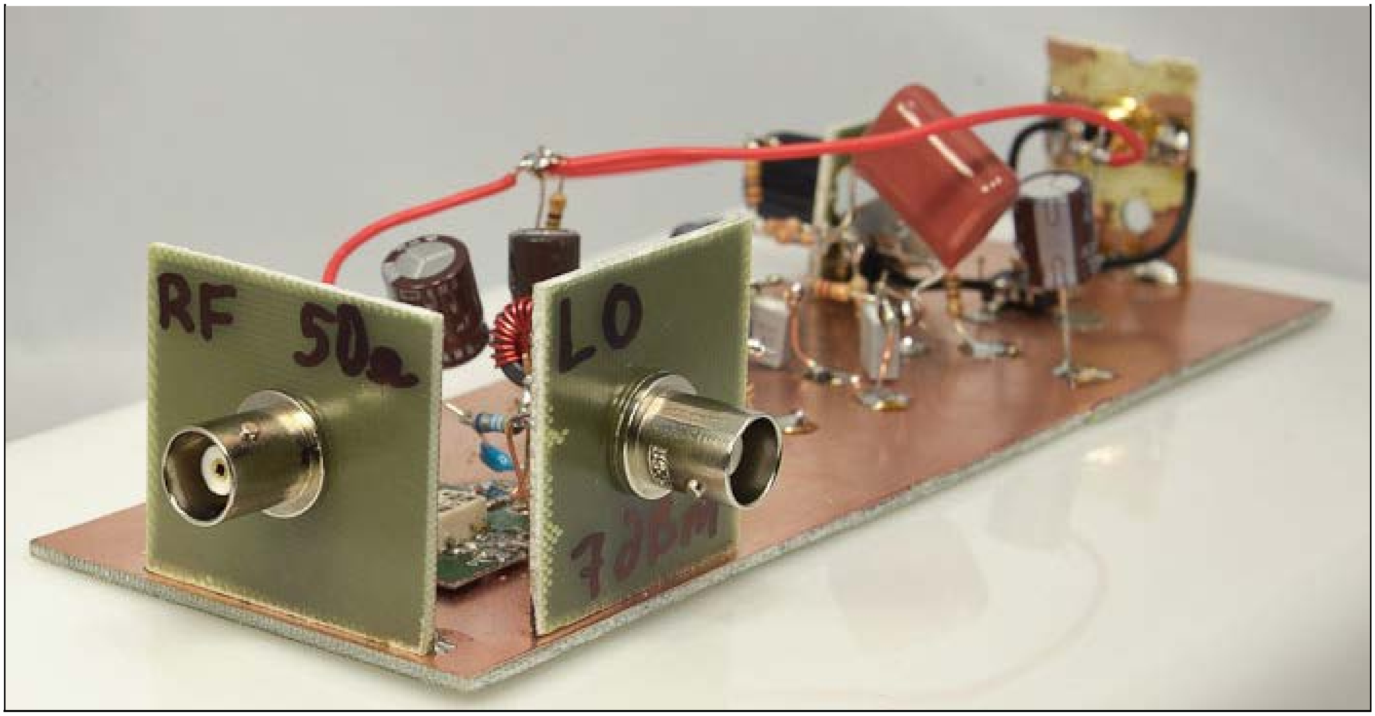
He's my hero because he's humble, fallible, well-intentioned and moral. Professor Ivanenko lives for learning — fame is filler — hollow and distracting. His current *ego* lags his voltage by 90 degrees. Is he part inductor / part human?



Miscellaneous Photos and circuits

Above — 3 types of adapters. A BNC male to SMA female, a BNC male to PL-259, and a BNC female to S0-239 allow the RF modules to be connected to a variety of equipment.

Click


RF 50Ω
LO
7dBm

Double Tuned Receiver Band-pass Filter Design Center

This web page is for builders who own EMRFD. Assisted by 4 of the Ladpac programs from the EMRFD compact disk, and the information presented in EMRFD Chapter 3, I share some experiments building popcorn receiver band-pass filters. Prior to diving into this material, please read the help file ***Ladpac2008 Manual.pdf*** and a file on the EMRFD compact disk called The Double Tuned Circuit: An Experimenter's tutorial by Wes, W7ZOI.

Preface

Derived from experiments, my web content reflects the efforts of a lay-person, hobby-level designer — I make mistakes. I say this not to make excuses or avoid accountability, but to share the truth. My hope is that my experiments inform yours and we all improve over time. I correct reported mistakes and rely on your eyes to see them.

Arduous and requiring good math skills, filter design is out of reach for many builders. Software changes this and learning to apply computer programs in real-world situations is part of our hobby. This web page shares some bench experiences, plus my thoughts about using some programs written by Wes, W7ZOI. I present suggestions and examples based more on empiricism and from reading about band-pass filter design than scientific methodology.

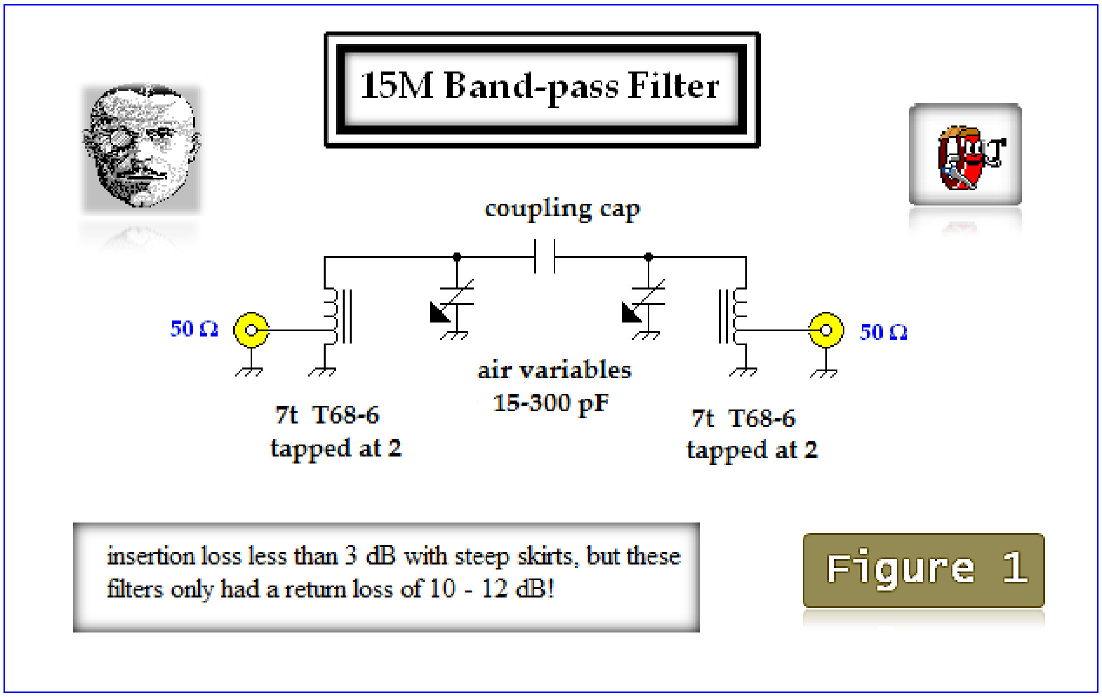
From email regarding my VFO and RF Workbench pages, I have become aware that I've lead many builders to think that a perfect sine wave and a high return loss are "must have" bench outcomes. This is false. A clean sine wave proves useful for accurate measurement, but is not a *de rigueur* bench outcome. A desire for high return loss reflects my own personal *obsession*; in simple QRP rigs, this may represent folly. Please don't overestimate the importance of return loss from my bias; decide for yourself.

Part 1: Experiments with 2 coupled L-C tanks.

Goal: A 15 Meter band band-pass filter with an insertion loss < 4 dB and a return loss of >= 20 dB.

Software: Ladbuild08 and GPLA08.

The simplest band-pass filter is an L-C tank. To get a decent stop band we generally couple 2 or 3 tanks together with series capacitor(s). Other filter topologies were ignored. In Part 1, I just connected up a couple of tanks on the bench without the use of software. Some attempts at impedance matching via transformer links were also trialed.


15M Band-pass Filter

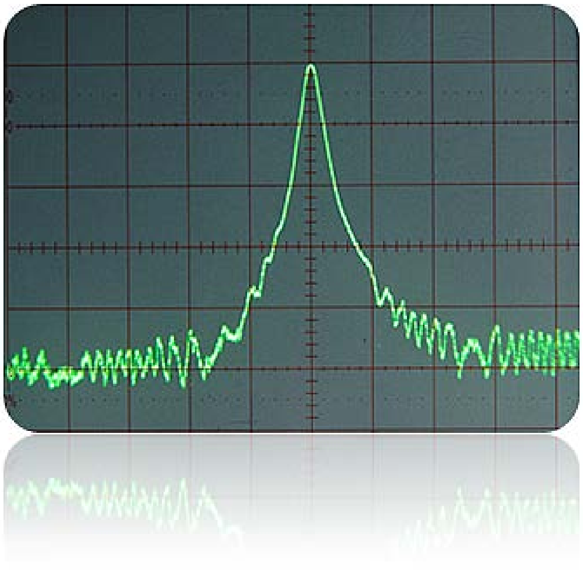
insertion loss less than 3 dB with steep skirts, but these filters only had a return loss of 10 - 12 dB!

Figure 1

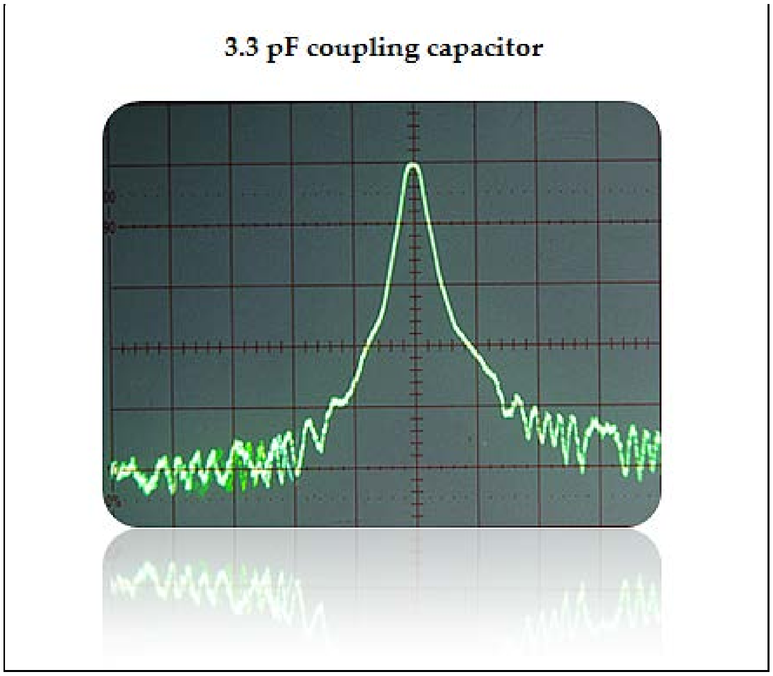
Above — I built a 5 component filter for the base experiment. Inductors = 7 turns of #22 AWG on a T68-6; tapped at 2 turns from ground. The inductors turns were expanded or compressed until L= 300 nH. Tuning capacitors = large 15 to 300 pF air variable capacitors. Coupling capacitors trialed = 2 pF, 3.3 pF, 5 pF, and 7.5 pF.

After soldering in a coupling capacitor, each tank (also called resonator) was tuned to resonance by looking at the peak-peak output voltage in a 50 ohm terminated oscilloscope. After tuning, I measured insertion and return loss and then swept each filter with a tracking generator + spectrum analyzer. On the bench I determined that the greatest return loss occurred with 2 transformer taps from ground; the result — a dismal 10-12 dB.

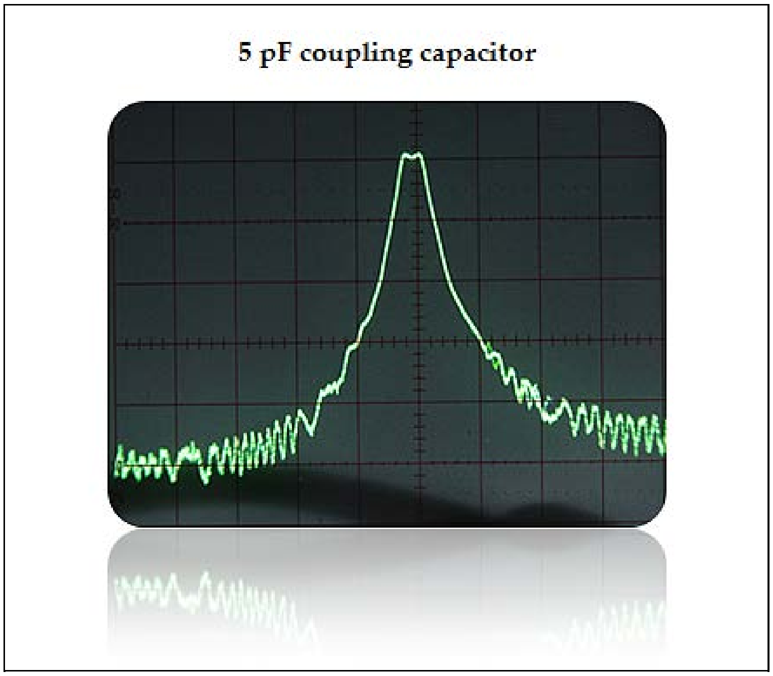
What effect does changing the coupling capacitor have?


2 pF coupling capacitor

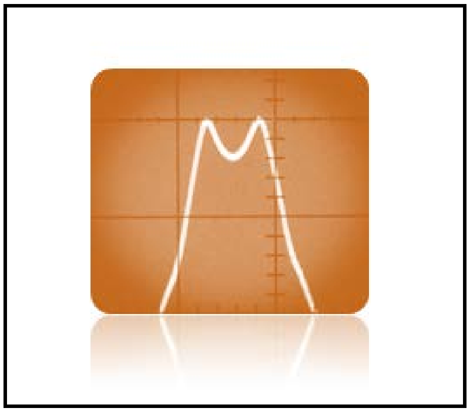
Above — A spectrum analyzer + tracking generator sweep of the filter response with a 2 pF coupling capacitor between the inductors. Graticules = 2 MHz per horizontal division and 10 dB per vertical division. Click on this zoom to better see the - 3 dB bandwidth. The sweep revealed a sharp peak response with steep skirts and a 3 dB down BW of ~220 KHz or so. Some of the noise arose from the big air variable caps connected to each tank with short hook up wires, plus no shielding.



Above — The SA + TG sweep with a 3.3 pF coupling capacitor. The peak isn't as sharp, but still looks good. As shown, increasing the coupling capacitor value increases the 3 dB filter bandwidth with all other components equal.



Above — With a 5 pF coupling capacitor, a double humped response appeared. The bandwidth further increases.

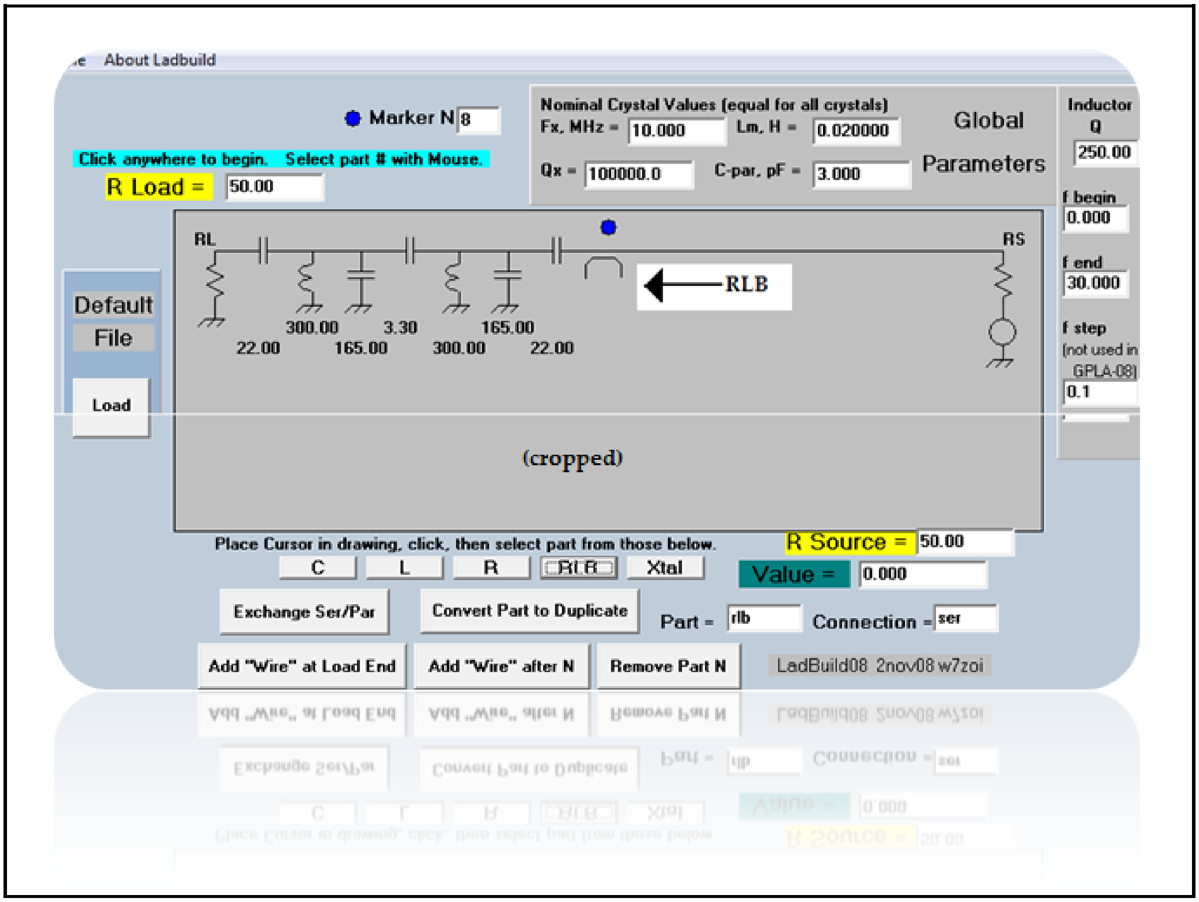


Above — A zoom of the double humped filter response employing a 7.5 pF coupling capacitor. Imagine the difficulty tuning this band-pass filter in a receiver by listening to band noise. Tuning in either peak skews the filter bandwidth. Additionally, the 3dB bandwidth now = ~ 1.6 MHz — Not a good filter!

Optimizing Return Loss

Despite trying, I could not obtain a better return loss than 10 -12 dB by changing the tap point on the 7 turn inductors. In part this was due to limited potential autotransformer ratios on a 7 turn coil. I emailed Wes, W7ZOI and he sent this file. I learned that adding a series capacitor to each end will tune the filter to 50 ohms impedance. What capacitor value should we use?

The answer can be found purely experimentally, or with Ladbuild08 to make a digital file of your filter and GPLA08 to analyze it.



Click anywhere to begin. Select part # with Mouse.

Marker N 8

R Load = 50.00

Nominal Crystal Values (equal for all crystals)

Fx, MHz = 10.000

Lm, H = 0.020000

Qx = 100000.0

C-par, pF = 3.000

Inductor Q 250.00

f begin 0.000

f end 30.000

f step (not used in GPLA-08) 0.1

Place Cursor in drawing, click, then select part from those below.

R Source = 50.00

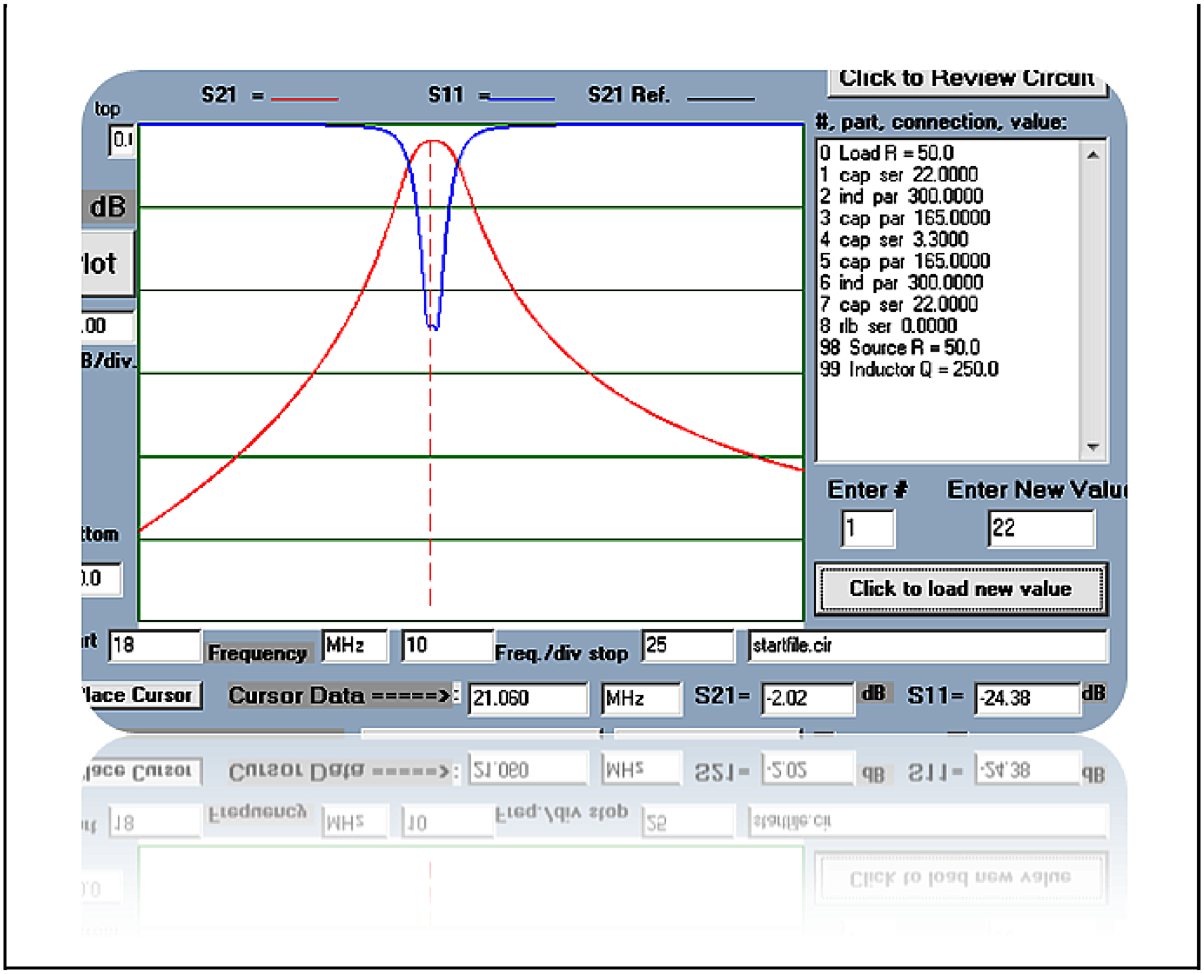
Value = 0.000

Part = rlb

Connection = ser

Above — I "built up" my Figure 1 filter in Ladbuild08 with a 3.3 pF coupling capacitor. Initially I guessed at the values for the series end capacitors and knew my tuning capacitor were ~ 165 pF because I removed and measured them from the peaked filter from Figure 1 and added a few pF for stray capacitance. Any of these values can be changed in GPLA, so educated guessing is okay.

For size 50 to 68 toroidal inductors, many builders choose a Qu value from 200 - 250 with # 6 material. Qu affects insertion loss and to some extent, return loss. Click for a tutorial from Wes', W7ZOI site and consult EMRFD for more information.
In order for GPLA08 to display an S11 plot (return loss), a return loss bridge (RLB) must be added as shown. Also check the **Plot S11** check box in GPLA08.



#, part, connection, value:

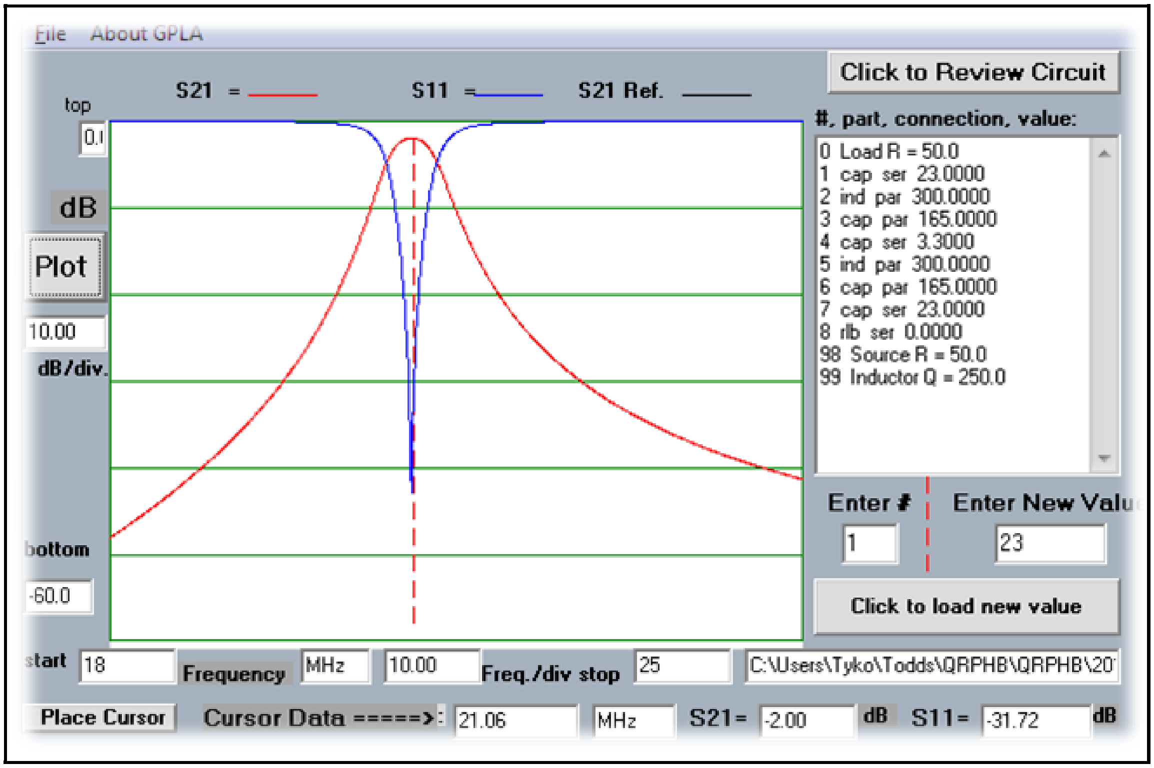
0 Load R = 50.0
1 cap ser 22.0000
2 ind par 300.0000
3 cap par 165.0000
4 cap ser 3.3000
5 cap par 165.0000
6 ind par 300.0000
7 cap ser 22.0000
8 rlb ser 0.0000
98 Source R = 50.0
99 Inductor Q = 250.0

Enter # 1 Enter New Value 22

start 18 Frequency MHz 10 Freq./div stop 25 startfile.cir

Cursor Data =====> 21.060 MHz S21= -2.02 dB S11= -24.38 dB

Above — The GPLA 08 filter simulation of the filter "built" with Ladbuild08 above.



#, part, connection, value:

0 Load R = 50.0
1 cap ser 23.0000
2 ind par 300.0000
3 cap par 165.0000
4 cap ser 3.3000
5 ind par 300.0000
6 cap par 165.0000
7 cap ser 23.0000
8 rlb ser 0.0000
98 Source R = 50.0
99 Inductor Q = 250.0

top 0.0 10.00 dB/div. bottom -60.0

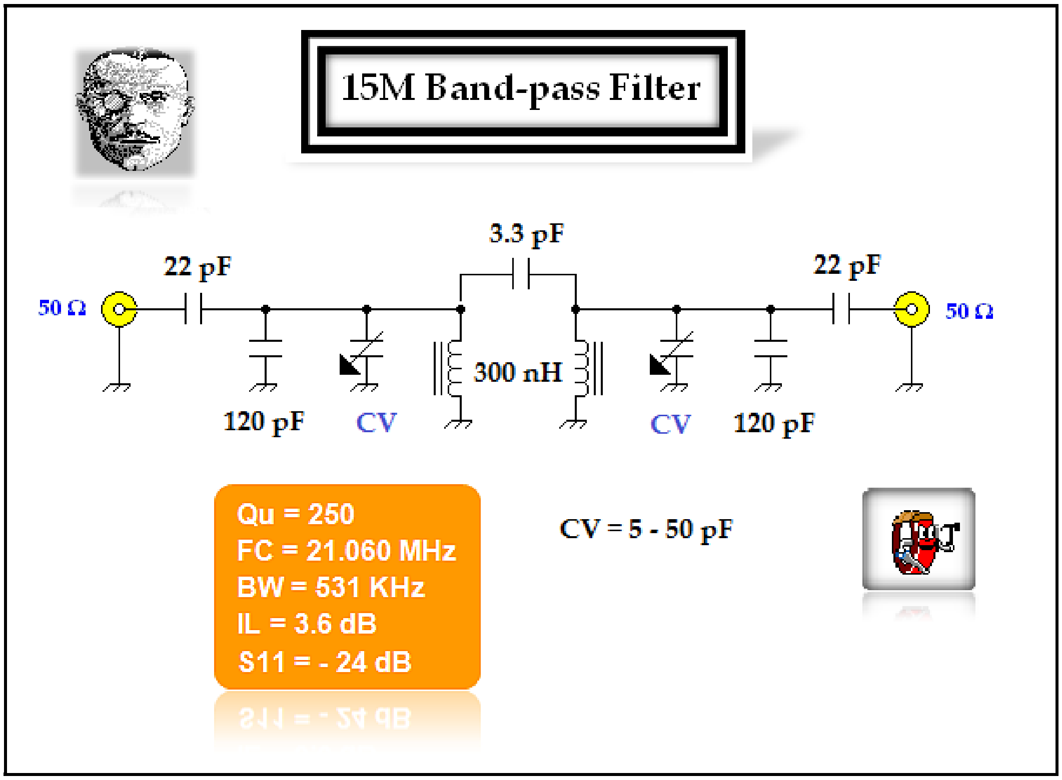
Enter # 1 Enter New Value 23

start 18 Frequency MHz 10.00 Freq./div stop 25 C:\Users\Tyko\Todds\QRPHB\QRPHB\20

Cursor Data =====> 21.06 MHz S21= -2.00 dB S11= -31.72 dB

Above — In this filter simulation, I tweaked the end capacitors (parts #1 and #7) from 22 to 23 pF and watched the return loss (S11) improve by 7.34 dB — if wanted, you can optimize the end capacitor values to improve the match into 50 ohms. To re-establish the center frequency, slight retuning of parallel capacitors #3 and 6 is required when changing the series end capacitors; although I specifically didn't change them for this example.

Increasing the 2 end capacitors to increase S11 renders an option only; you don't have to go for the best S11 in your filters. Increasing the series end capacitors to bump up return loss tends to increase the 3 dB bandwidth and reduce insertion loss.



Above — I built and measured the filter with 22 pF end capacitors since these are common, standard values. In another experiment, a 1 pF cap was soldered in parallel with each series capacitor and the return loss increased by about 4 dB. Click for a bench photo of an alternate version of the above filter. Clearly, GPLA08 simulation furnishes us popcorn builders with a starting point to make top-notch band-pass filters.

Click for another simulation of a filter employing a 2 pF coupling capacitor, with the end capacitors tweaked for the best S11. *S-11 is just the negative of the return loss.* I would certainly use this filter in the front end of a popcorn direct conversion receiver.

An easier way to design your band-pass filters involves using DTC08 to design a raw filter and GPLA08 to substitute in standard value capacitors and tweak your filter. That's part 2. The material presented in this section supports the discussion in Part 2 and 3.

Part 2: Band-pass Filter Design using DTC08

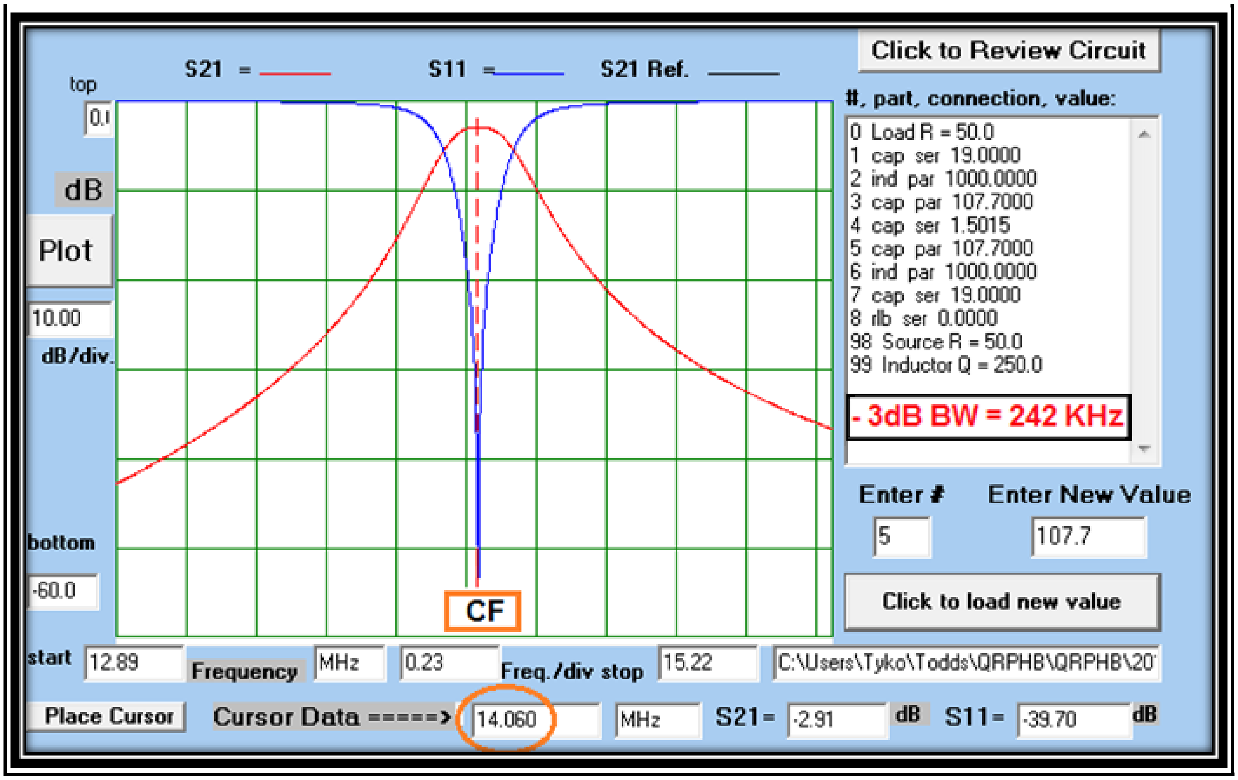
Prior to using these Ladpac programs, some numbered design points and a preamble follow.

More than anything else, our parts collection dictates what filter parameters we choose and end up with. For example, if you want filters with a low bandwidth such as 150 KHz and under, you'll require inductors and capacitors that provide really high Q, or you might suffer from punishing insertion loss.

The following are general starting points only — *your* needs, parts and abilities drive *your* filter design. Example variances include: if a low noise amplifier follows a filter, a higher insertion loss might be okay; a high return loss is not always required for a low noise figure; especially in popcorn receivers. Also, it's a viable choice to trade off insertion loss for steep skirts in some filters.

1. A reasonable 3 dB bandwidth = 100 to 500 KHz, but this depends on the purpose of the filter.
Numerous considerations challenge us. Will this be a whole band (CW + SSB) filter, or a CW only filter? As a CW op who uses simple equipment, I tend to design moderate bandwidth (200-300 KHz) CW-only filters. If you need CW + SSB, then a bandwidth of 350 KHz or greater might suit you. It's really up to you.

Other factors affecting bandwidth choice include whether the filter drives a superheterodyne or a direct conversion receiver. In superheterodyne receivers, your intermediate frequency informs your filter bandwidth choice. Consider the following 2 diagrams:



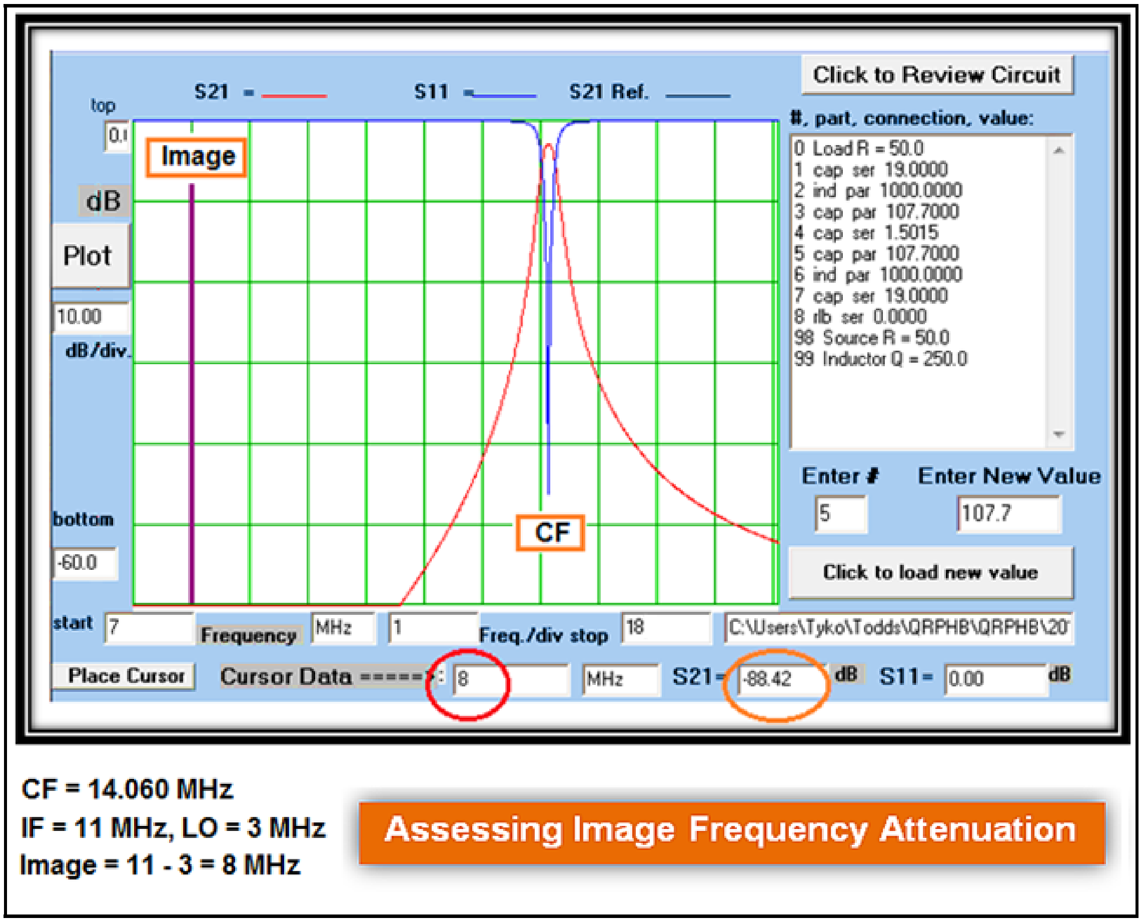
#, part, connection, value:

0 Load R = 50.0
1 cap ser 19.0000
2 ind par 1000.0000
3 cap par 107.7000
4 cap ser 1.5015
5 cap par 107.7000
6 ind par 1000.0000
7 cap ser 19.0000
8 rlb ser 0.0000
98 Source R = 50.0
99 Inductor Q = 250.0

- 3dB BW = 242 KHz

top 0.I
10.00 dB/div.
bottom -60.0
Enter # 5
Enter New Value 107.7
start 12.89 Frequency MHz 0.23 Freq./div stop 15.22 C:\Users\Tyko\Todds\QRPHB\QRPHB\20
Cursor Data =====> 14.060 MHz S21= -2.91 dB S11= -39.70 dB

Above — Using DTC08 and GPLA08, I designed an example filter for the front end of a 14 MHz superheterodyne receiver with an 11 MHz IF. BW = 242 KHz. CF = 14.060 MHz; a frequency some QRP operators favor.



#, part, connection, value:

0 Load R = 50.0
1 cap ser 19.0000
2 ind par 1000.0000
3 cap par 107.7000
4 cap ser 1.5015
5 cap par 107.7000
6 ind par 1000.0000
7 cap ser 19.0000
8 rlb ser 0.0000
98 Source R = 50.0
99 Inductor Q = 250.0

top 0.I
10.00 dB/div.
bottom -60.0
Enter # 5
Enter New Value 107.7
start 7 Frequency MHz 1 Freq./div stop 18 C:\Users\Tyko\Todds\QRPHB\QRPHB\20
Cursor Data =====> 8 MHz S21= -88.42 dB S11= 0.00 dB

CF = 14.060 MHz
IF = 11 MHz, LO = 3 MHz
Image = 11 - 3 = 8 MHz

Assessing Image Frequency Attenuation

Above — Assessing filter attenuation at the image frequency using GPLA08. *To keep the arithmetic simple, I employed a frequency of 14 MHz for the image frequency calculation.* As shown, the simulated attenuation of my 8 MHz image frequency is 88.42 dB. Since I personally target an image frequency suppression of 60-70 dB; at 88 dB, if I wanted, I could increase the bandwidth of this filter for broader coverage and reduced insertion loss.

How much image frequency rejection is needed for superheterodyne receivers? I'm uncertain, for I have seen competent authors choose between 50 and 100 dB. I feel a good target = 60 - 70 dB, and 50 dB is the bare minimum. To realize image attenuation above 50 dB, shielding is usually required.

Three or more L-C tank band-pass filters may be required when your image frequency is close to the IF frequency. Choose both your intermediate frequency and your bandwidth wisely.

2. After selecting your bandwidth, *tweak the inductance* and only if necessary, make minor adjustments to your set 3 dB bandwidth to give standard, or near-standard value coupling capacitors that you *own.* Obviously, you can place fixed capacitors in series or parallel, or even couple your resonators with a variable capacitor.

3. I favor size 50 to 80 powdered iron toroids with number 2, 6, or 10 material for a reasonably high Qu.

4. I aim for an insertion loss of of 3-4 dB; especially above the 40 Meter band; *consider the variances discussed earlier*

5. I aim for a return loss of at least 20 dB; *consider the variances discussed earlier*

6. If you can, measure your bread boarded filter bandwidth to confirm or improve the GLPA simulation. Insertion and return loss are easily measured — see EMRFD and the RF Workbench web pages on this site for methods.

I provide no graphic tutorial of DTC08; however, some work flow suggestions follow:

Open up DTC08, choose your center frequency, Qu, inductance and bandwidth and then press the **Calculate** button. Adjust the L until you get close to a standard value coupling capacitor from your parts bin. If required, you may also tweak the bandwidth value to get the needed coupling capacitor. It's wise to change the L before BW since changes in inductance don't cause too many complications within limits.

Name and save your filter to a specific file system directory or folder; or simply save it as the default file.

Open GPLA08 and load your recently saved filter file. Press the **Plot** button and then the **Click to Review Circuit** button. In some cases, you will have to type the CF in the *Cursor Data* text box and press **Plot** to set the cursor at your center frequency.

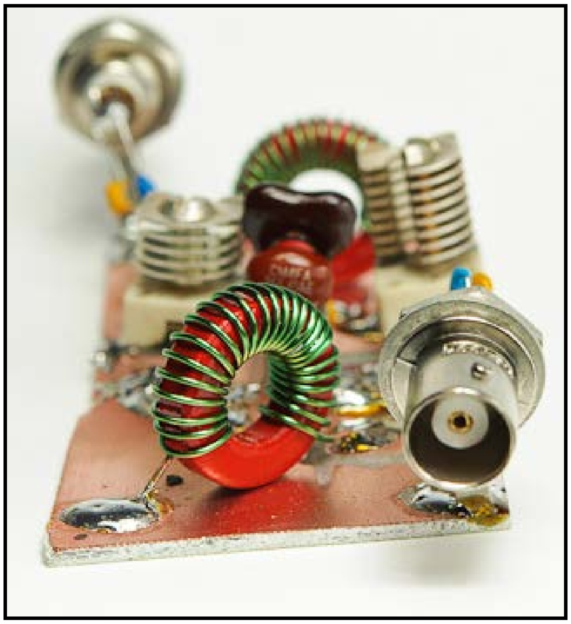
Change the coupling capacitor(s) to a standard value using the **Enter New Value** data entry controls.

Adjust the series end coupling capacitors to standard values. and if S11 is an issue for you, tweak them up and down while observing S11. Re-establish your center frequency by tweaking the parallel tank tuning capacitors and then re-plot to ensure the CF is lined up with the center of the plot.

In Part 3, I provide 3 filter design examples. Your own filter designs will be the most important examples to study.

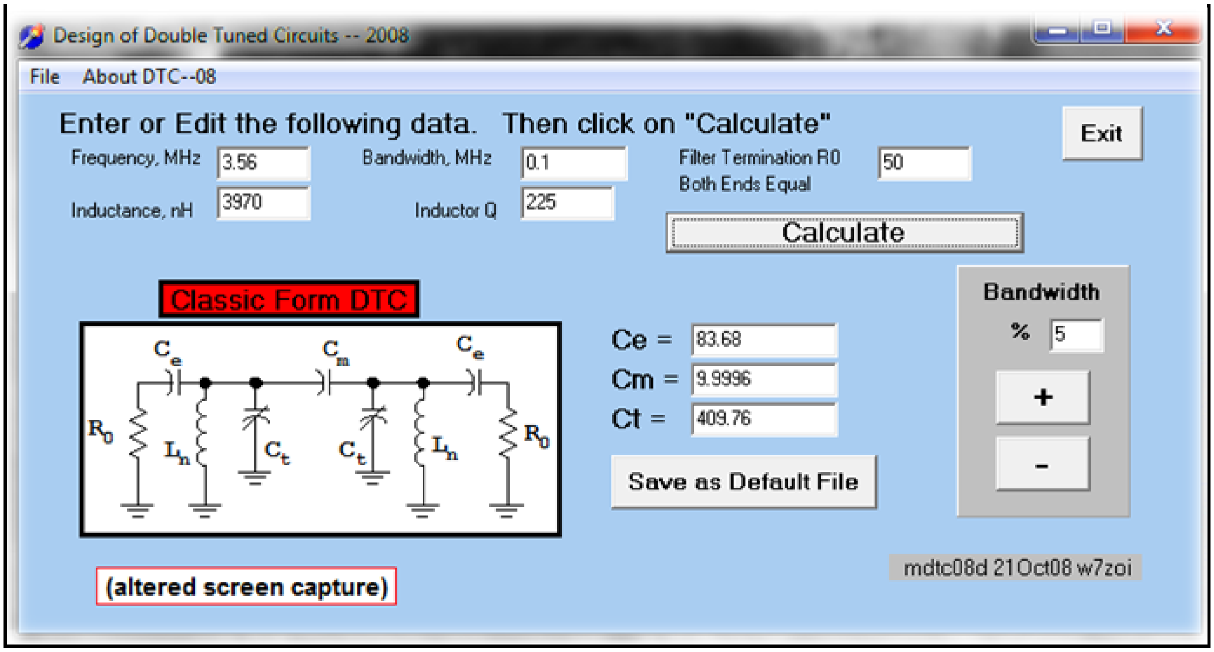
Part 3: Band-pass Filter Examples

Example 1: An 80 Meter Band Filter



Above — Breadboard photograph of the 80M filter. This example filter may hit home for you — I like listening to CW at and below the 3560 KHz QRP calling frequency, however, another local Ham likes to talk on 75 meters SSB at or above 3790 KHz. This situation calls for a narrow band-pass filter. With my filter, the attenuation at 3790 KHz = ~ 24 dB; had I built a wide bandwidth filter, for example, 350 KHz BW; the attenuation at 3790 KHz, would only be ~4 dB. Perhaps a 3 resonator filter with even steeper skirts would be better?

I'll show the design process from start to test.



Frequency, MHz 3.56

Bandwidth, MHz 0.1

Filter Termination R0 Both Ends Equal 50

Inductance, nH 3970

Inductor Q 225

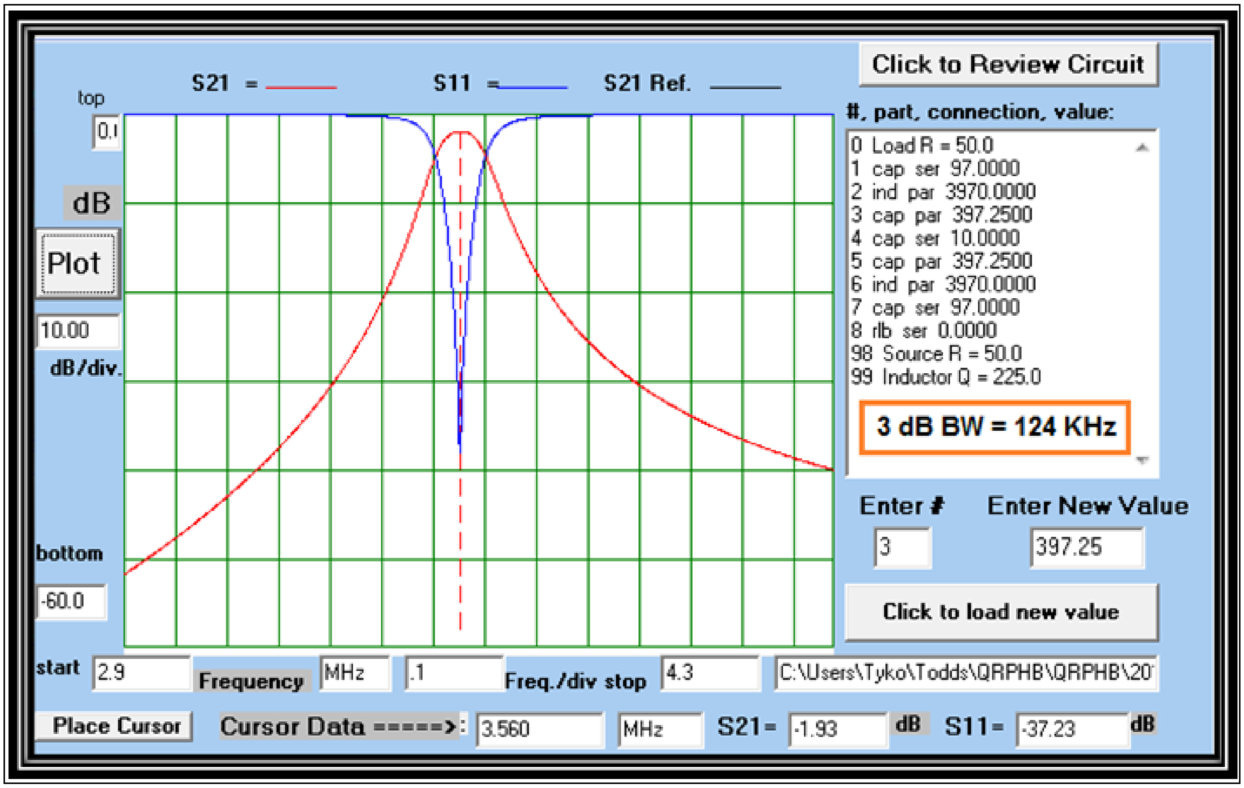
Ce = 83.68

Cm = 9.9996

Ct = 409.76

Bandwidth % 5

Above — The basic DTC08 data entry fields were populated. I chose a 100 KHz bandwidth and tried different L values until Cm = 10 pF, since I have a whole drawer of 10 pF capacitors. I believed my Inductor Q would be at least 225 and wound 25 turns of #22 AWG on a T68-2 toroid and expanded or contracted the windings until I measured 3970 nanohenries. In reality, we should measure the inductor Q and in future I will, however, my sense is that few builders do.



top 0.0

10.00 dB/div.

bottom -60.0

#, part, connection, value:

```
0  Load R = 50.0
1  cap  ser  97.0000
2  ind  par  3970.0000
3  cap  par  397.2500
4  cap  ser  10.0000
5  cap  par  397.2500
6  ind  par  3970.0000
7  cap  ser  97.0000
8  rlb  ser  0.0000
98  Source R = 50.0
99  Inductor Q = 225.0
```

3 dB BW = 124 KHz

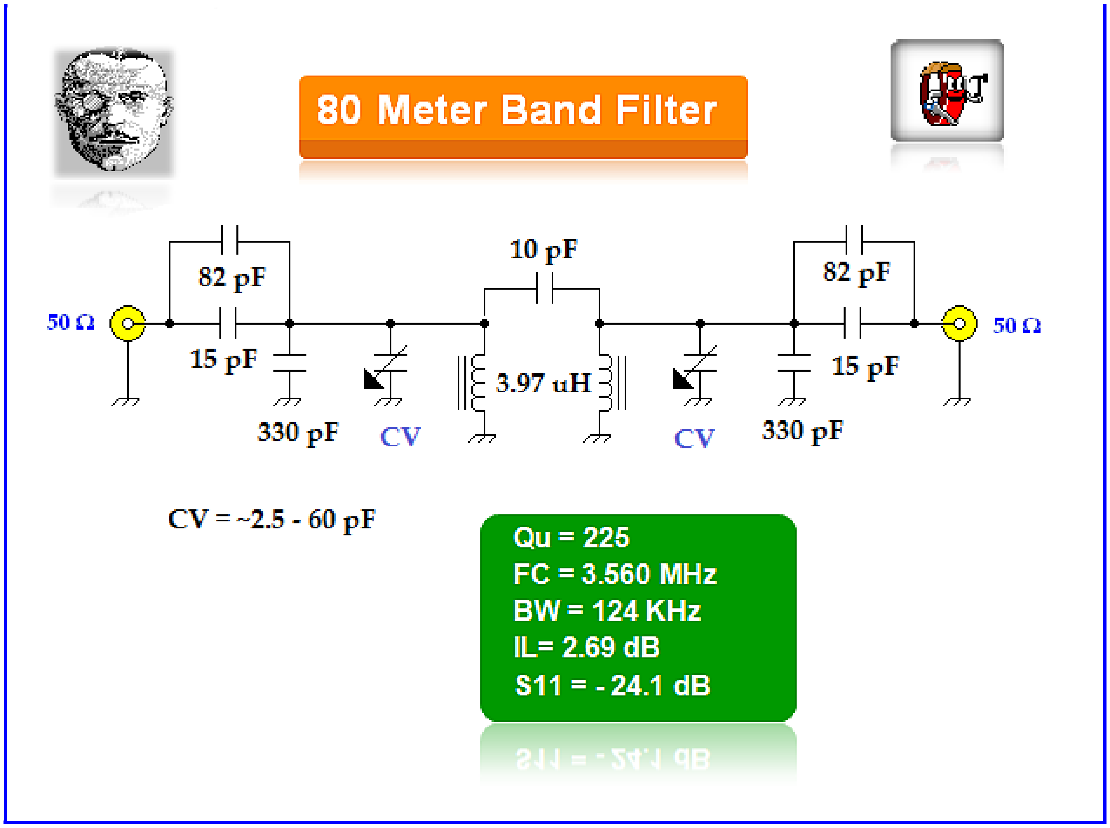
Enter # 3

Enter New Value 397.25

start 2.9 Frequency MHz .1 Freq./div stop 4.3 C:\Users\Tyko\Todds\QRPHB\QRPHB\20

Cursor Data =====>: 3.560 MHz S21= -1.93 dB S11= -37.23 dB

Above — After saving my filter, I opened it up in GPLA08 as above. I replaced #4 with a standard 10.0 pF value, and started tweaking the end caps; parts #1 and #7 to gain a better S11 per my obsession with return loss. Retuning #3 and #7 re-establishes the center frequency and allows the S21 and S11 values to be interpreted. I settled on this filter and headed for the bench.



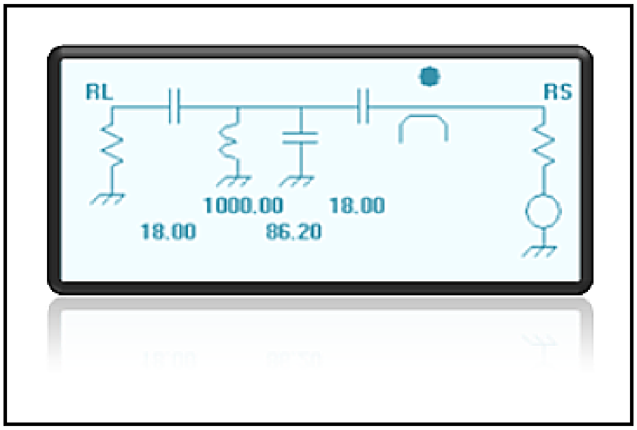
Above — Schematic and analysis of the breadboard. Click for another photograph. In reality, I bench determined the exact capacitance needed to tune each tank at 3.56 MHz with 2 large air variable capacitors that I removed and measured after peaking the filter. For each tank, I try to get just below this value with fixed value capacitors and add a small (2.5 - 22 pF, or so) air variable trimmer capacitor for peaking.

You need to test with capacitance under and over that required to ensure you properly tuned each L-C tank to resonance. Your parts collection, stray capacitance, mistakes or inductance variations in the toroids necessitate custom tuning of your tanks on the bench. I give capacitance values that should work, but it's up to you to ensure resonance of each tank. I find narrow BW filters require a steady hand to tune.

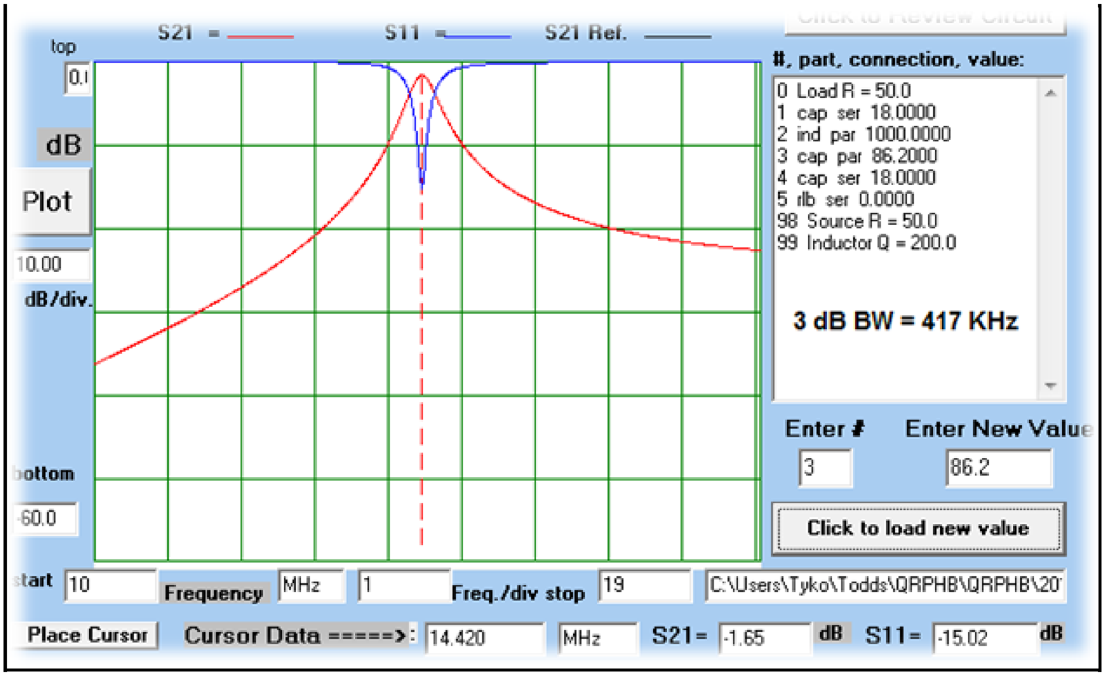
After peaking the tank in your oscilloscope, record the peak-to-peak voltage. Remove the filter and connect your signal generator to your scope with an RF barrel connector and again record the peak to peak voltage. The difference between the 2 is your insertion loss. You can calculate IL with Applet H on this page . Next, perform return loss measurements. If you can, determine the true 3 dB bandwidth of your filter by sweeping it with SA plus a generator. My filter 3 dB BW = 124 KHz.

Example 2: 20 Meter Band Superheterodyne Receiver Filter

A fictitious builder wants a superheterodyne receiver that covers 14.0 - 14.350. His IF = 2 MHz. The local oscillator = 12 MHz. The image frequency = (12 - 2) = 10 MHz. He centers his filter at 14.020 MHz. In this simulation-only example, we'll go from 1 resonator to 3.



Above — A single resonator with series matching capacitors "built up" in Ladbuild08.

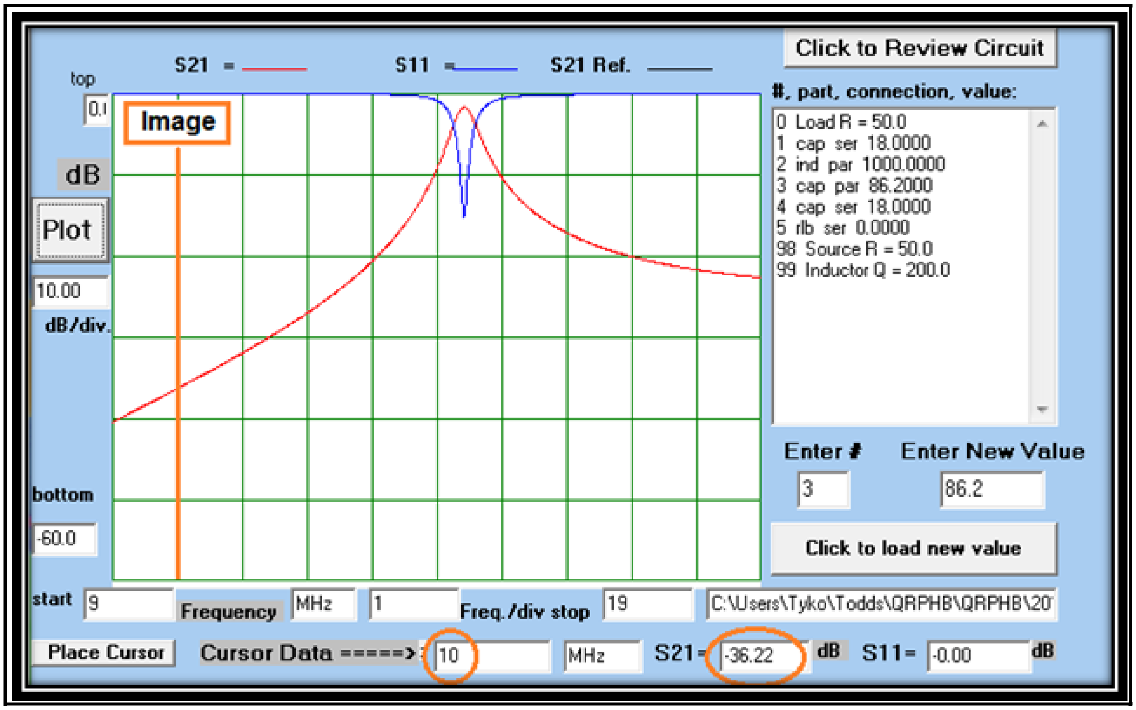


#, part, connection, value:
0 Load R = 50.0
1 cap ser 18.0000
2 ind par 1000.0000
3 cap par 86.2000
4 cap ser 18.0000
5 rlb ser 0.0000
98 Source R = 50.0
99 Inductor Q = 200.0

3 dB BW = 417 KHz

top 0.0
10.00 dB/div.
bottom -60.0
start 10 MHz 1 Freq./div stop 19
C:\Users\Tyko\Todds\QRPHB\QRPHB\20
Cursor Data 14.420 MHz S21= -1.65 dB S11= -15.02 dB
Enter # 3 Enter New Value 86.2

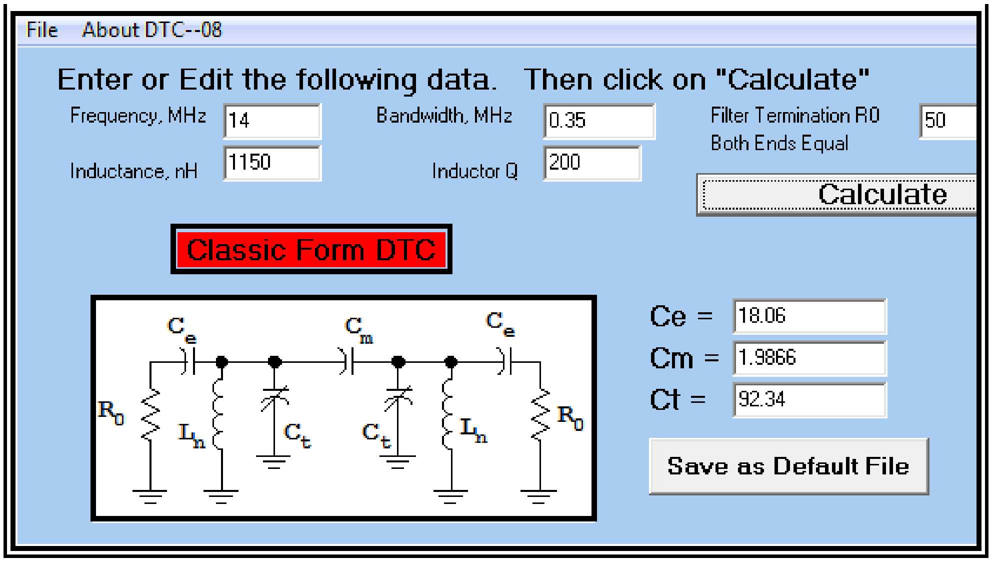
Above — The GPLA08 plot of the single tank filter. The bandwidth = 417 KHz. Increasing the end capacitors to 22 pF to try to increase return loss increases the 3 dB bandwidth as shown here, so we better stick with the original design.
In GPLA simulations with a perfectly centered filter, S21 = the insertion loss and S11 is negative of the return loss.



#, part, connection, value:
0 Load R = 50.0
1 cap ser 18.0000
2 ind par 1000.0000
3 cap par 86.2000
4 cap ser 18.0000
5 rlb ser 0.0000
98 Source R = 50.0
99 Inductor Q = 200.0

top 0.0
10.00 dB/div.
bottom -60.0
start 9 MHz 1 Freq./div stop 19
C:\Users\Tyko\Todds\QRPHB\QRPHB\20
Cursor Data 10 MHz S21= -36.22 dB S11= -0.00 dB
Enter # 3 Enter New Value 86.2

Above — Assessing image frequency attenuation in GPLA08; this sucks — only 36.2 dB down. We need to add a tank.

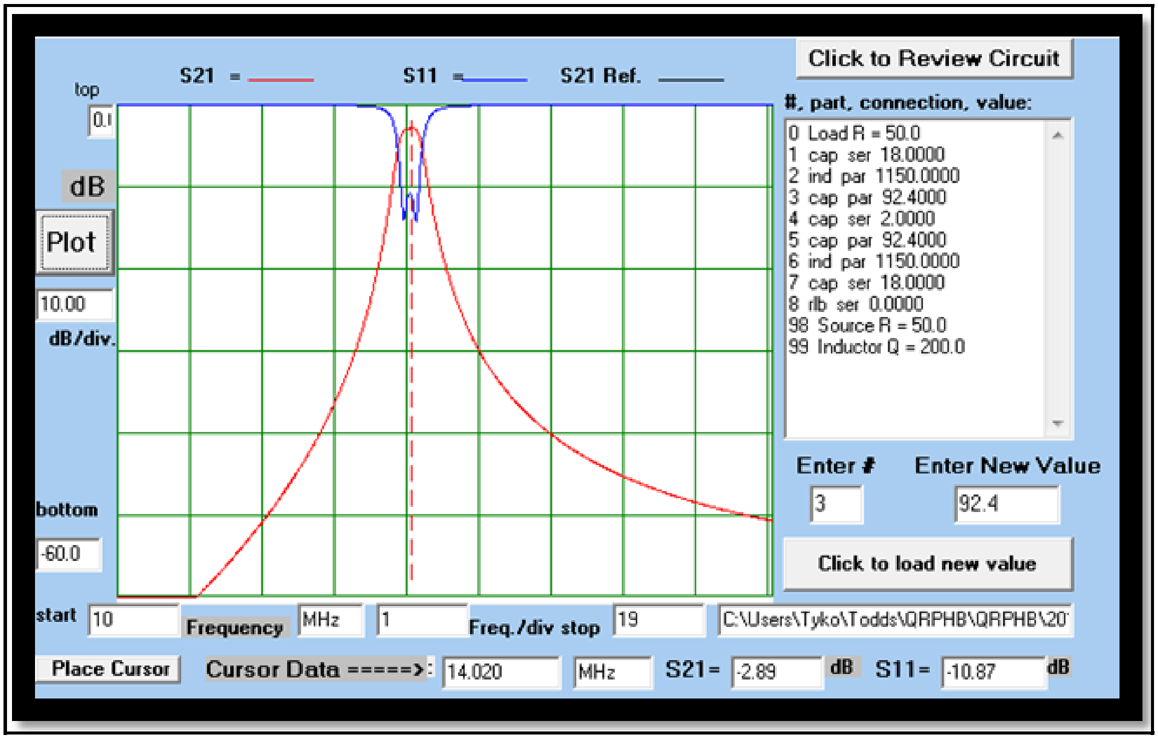


Enter or Edit the following data. Then click on "Calculate"

Frequency, MHz: 14
Bandwidth, MHz: 0.35
Filter Termination R0 Both Ends Equal: 50
Inductance, nH: 1150
Inductor Q: 200

Ce = 18.06
Cm = 1.9866
Ct = 92.34

Above — Building up a filter in DTC08, I increased the L from 1000 to 1150 nH to give a Cm near to a standard value.



#, part, connection, value:

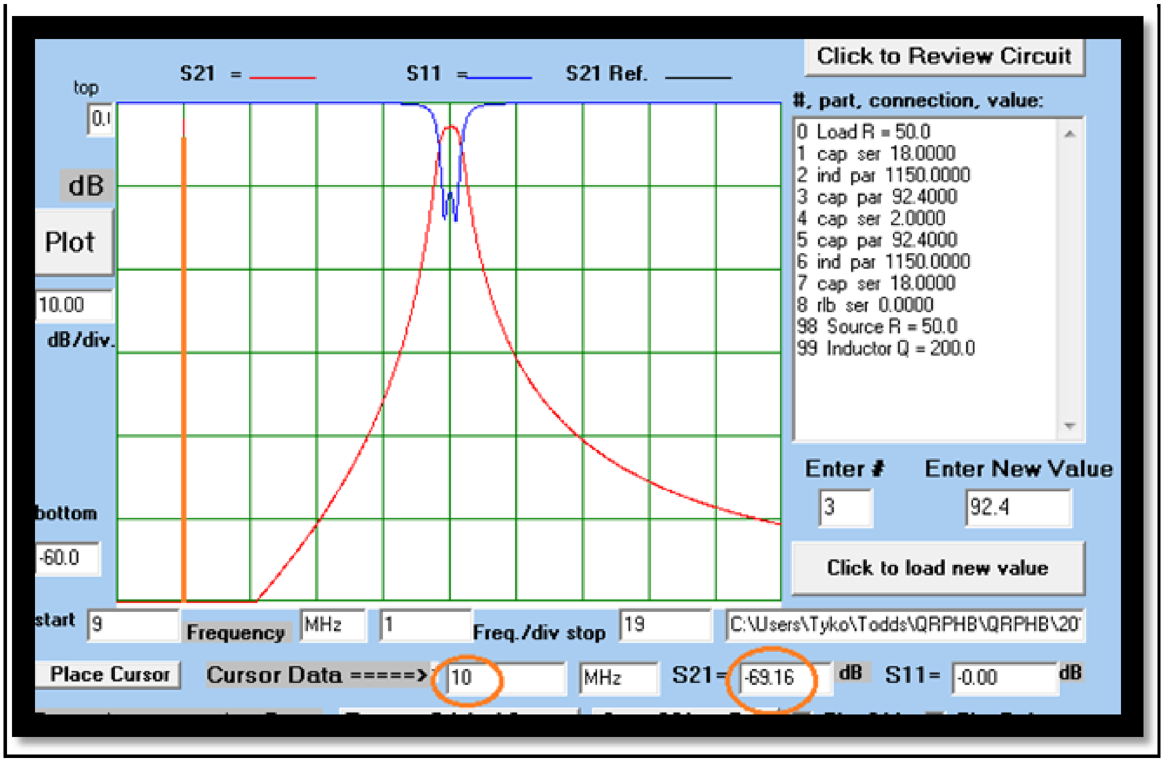
```
0  Load R = 50.0
1  cap  ser  18.0000
2  ind  par  1150.0000
3  cap  par  92.4000
4  cap  ser  2.0000
5  cap  par  92.4000
6  ind  par  1150.0000
7  cap  ser  18.0000
8  rlb  ser  0.0000
98  Source R = 50.0
99  Inductor Q = 200.0
```

top: 0.
dB/div.: 10.00
bottom: -60.0
start: 10 MHz
Freq./div: 1
stop: 19
C:\Users\Tyko\Todds\QRPHB\QRPHB\20

Enter #: 3
Enter New Value: 92.4

Cursor Data =====>: 14.020 MHz S21= -2.89 dB S11= -10.87 dB

Above — The GPLA08 plot of the double tuned filter. I set #4 to 2 pF and #1 and #7 to 18 pF (nearest standard values). #3 and #5 were slightly tweaked to center the filter. The simulated IL is only up 1.24 dB from the single resonator version. You are probably wondering why I didn't design the filter for a CF = 14.020 MHz in DTC08 above to keep consistency. I probably should have, but wanted to illustrate the versatility of GPLA08 to center filters "on the fly".



#, part, connection, value:

0 Load R = 50.0
1 cap ser 18.0000
2 ind par 1150.0000
3 cap par 92.4000
4 cap ser 2.0000
5 cap par 92.4000
6 ind par 1150.0000
7 cap ser 18.0000
8 rlb ser 0.0000
98 Source R = 50.0
99 Inductor Q = 200.0

Enter # 3

Enter New Value 92.4

top 0.0

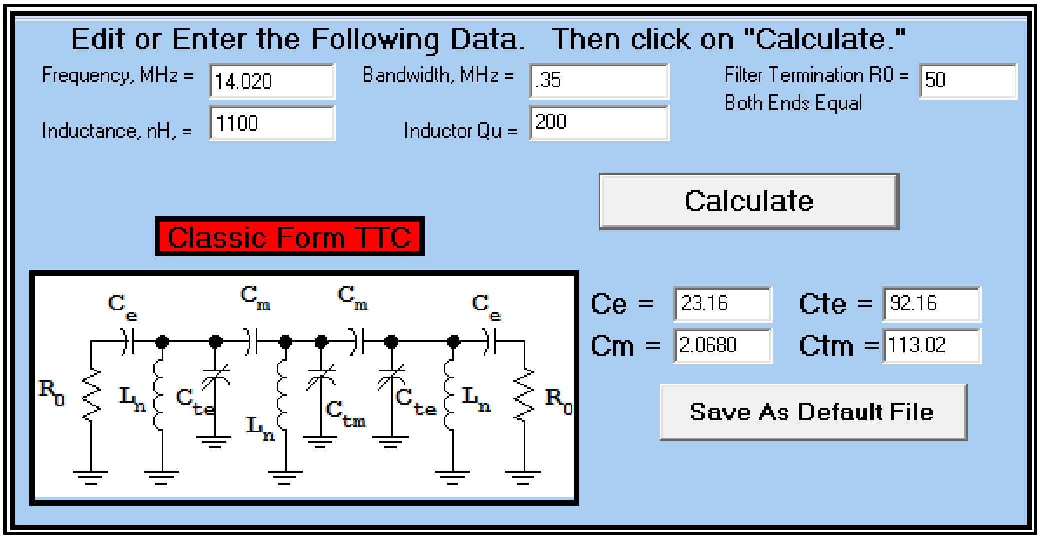
10.00 dB/div.

bottom -60.0

start 9 Frequency MHz 1 Freq./div stop 19 C:\Users\Tyko\Todds\QRPHB\QRPHB\20

Cursor Data =====> 10 MHz S21= -69.16 dB S11= -0.00 dB

Above — Assessment of the 10 MHz image — now it's 69.2 dB down. Although this filter will work well for his particular receiver specifications, this fastidious builder wants even greater image attenuation and decides to add a third resonator!



Edit or Enter the Following Data. Then click on "Calculate."

Frequency, MHz = 14.020

Bandwidth, MHz = .35

Filter Termination R0 = 50

Both Ends Equal

Inductance, nH, = 1100

Inductor Qu = 200

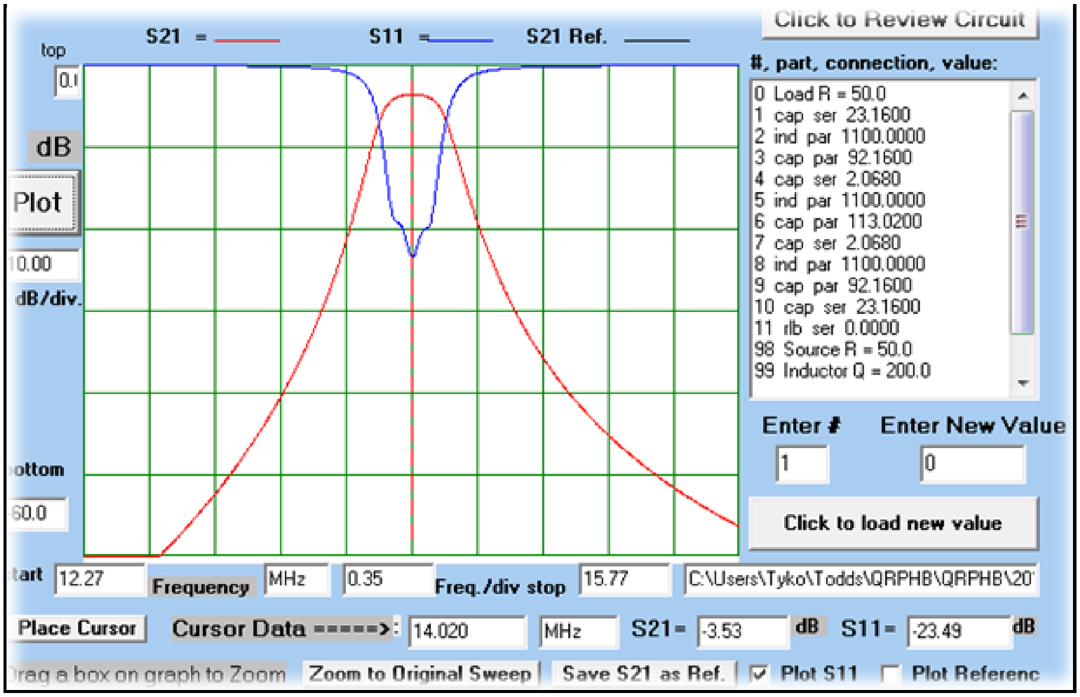
Ce = 23.16

Cte = 92.16

Cm = 2.0680

Ctm = 113.02

Above — Building up a filter in TTC08, I chose an L of 1100 nH to give a Cm close to a standard value.

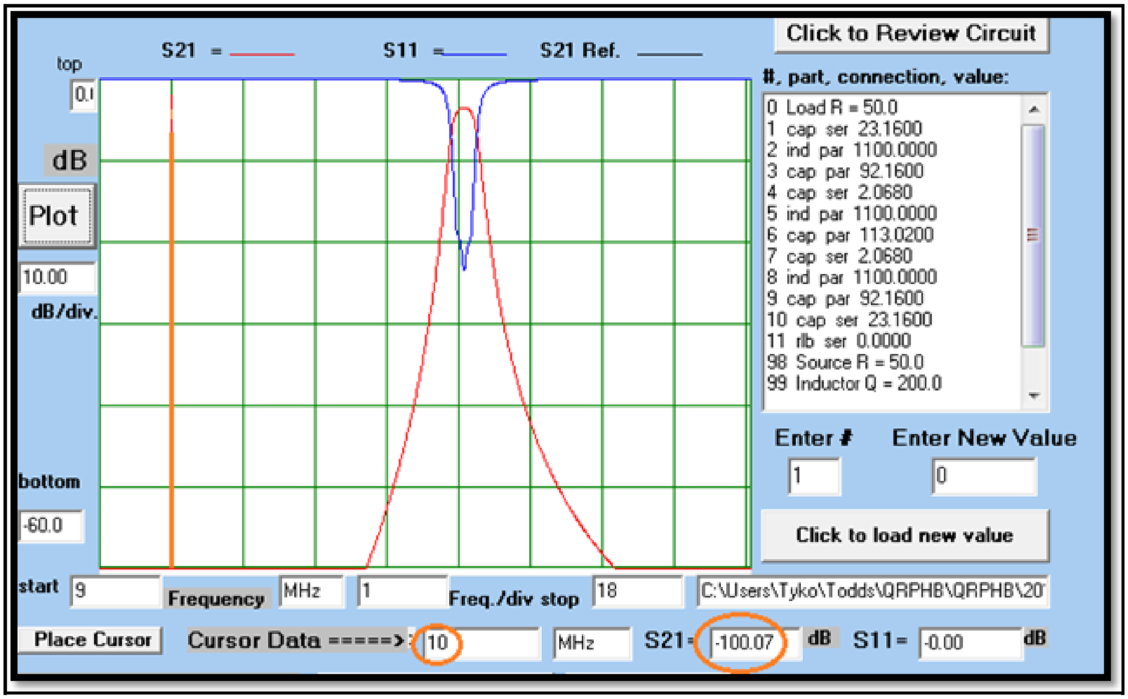


#, part, connection, value:

0 Load R = 50.0
1 cap ser 23.1600
2 ind par 1100.0000
3 cap par 92.1600
4 cap ser 2.0680
5 ind par 1100.0000
6 cap par 113.0200
7 cap ser 2.0680
8 ind par 1100.0000
9 cap par 92.1600
10 cap ser 23.1600
11 rlb ser 0.0000
98 Source R = 50.0
99 Inductor Q = 200.0

Enter #: 1
Enter New Value: 0
top: 0.(
0.00 dB/div.
ottom: 60.0
tart: 12.27 MHz
Freq./div: 0.35
stop: 15.77
C:\Users\Tyko\Todds\QRPHB\QRPHB\20
Cursor Data: 14.020 MHz
S21= -3.53 dB
S11= -23.49 dB

Above — The GPLA plot of the 3 tank filter. I performed no parts tweaking — it's up to you from here on in. The simulated IL remains quite reasonable.

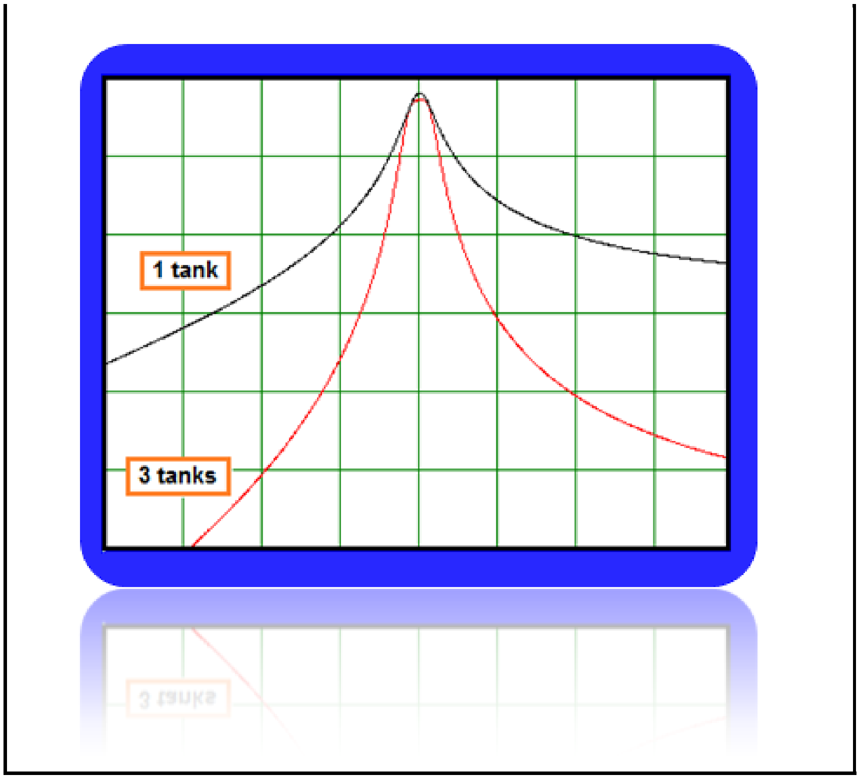


#, part, connection, value:

0 Load R = 50.0
1 cap ser 23.1600
2 ind par 1100.0000
3 cap par 92.1600
4 cap ser 2.0680
5 ind par 1100.0000
6 cap par 113.0200
7 cap ser 2.0680
8 ind par 1100.0000
9 cap par 92.1600
10 cap ser 23.1600
11 rlb ser 0.0000
98 Source R = 50.0
99 Inductor Q = 200.0

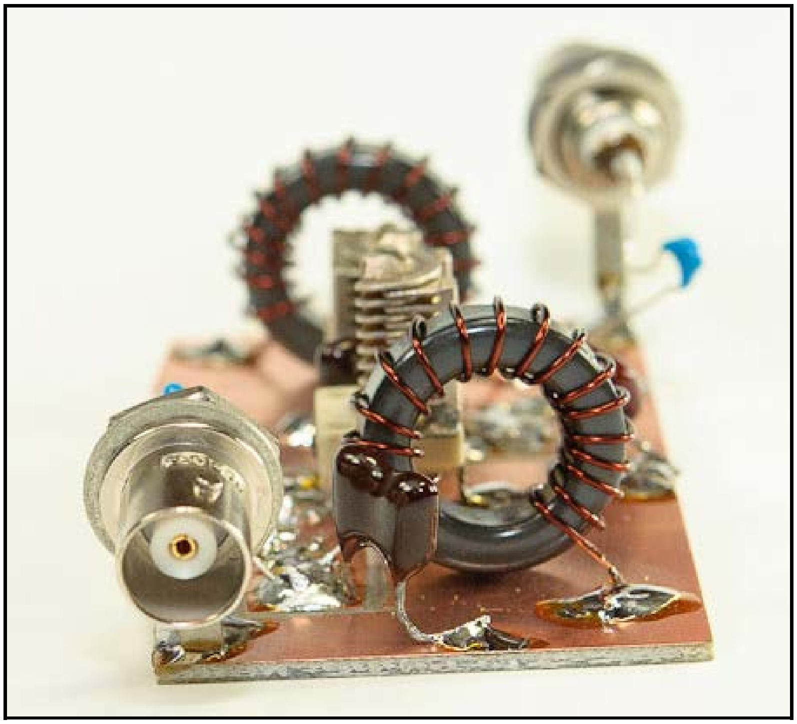
Enter #: 1
Enter New Value: 0
top: 0.(
10.00 dB/div.
bottom: -60.0
start: 9 MHz
Freq./div: 1
stop: 18
C:\Users\Tyko\Todds\QRPHB\QRPHB\20
Cursor Data: 10 MHz
S21= -100.07 dB
S11= -0.00 dB

Above — The GPLA08 assessment of the 10 MHz image frequency. Now 100 dB down!



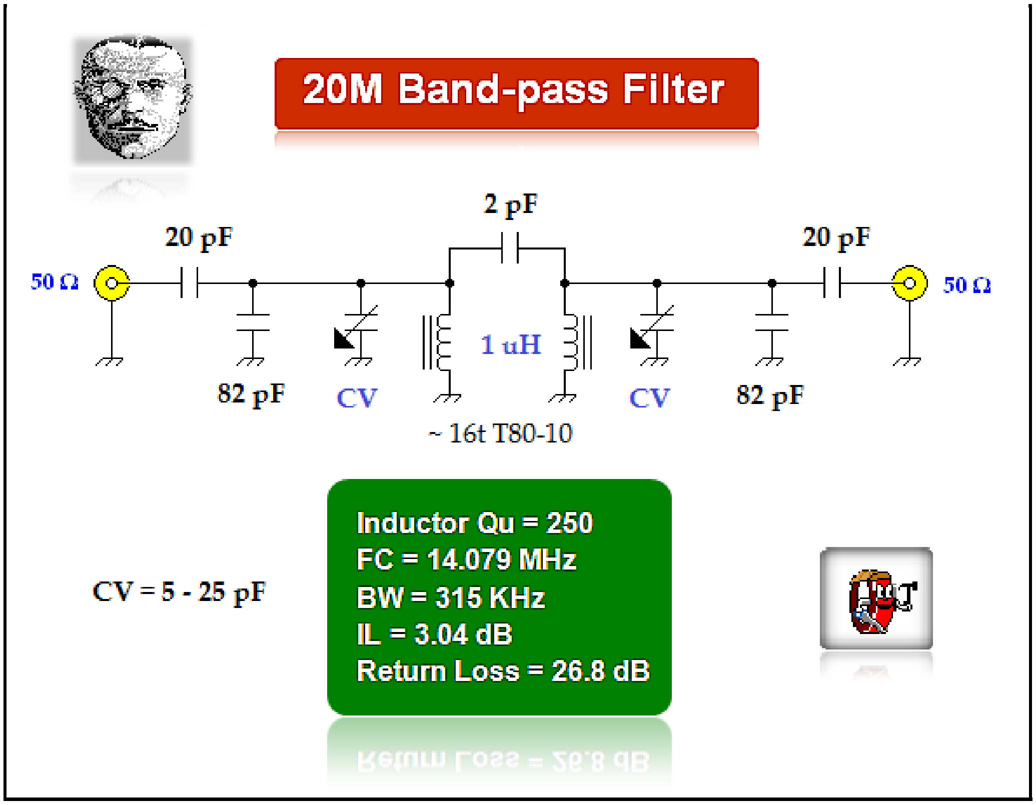
Above — The 1 tank and 3 tank filters superimposed to show the skirt action. The 3 dB bandwidth is the same!

Example 3: A 20 Meter Band-pass Filter for a Builder from Argentina



Above — An Argentinean builder emailed that he wanted a band-pass filter optimized for 14.070-14.095 MHz RTTY but also usable for the CW sub-band and lower SSB frequencies. He wanted a center frequency in the RRTY sub-band and I chose 14.079 MHz. Tuning this filter to a center frequency as low as 7.030 MHz for CW should be possible with the variable capacitor value shown, but as mentioned, you really need to do this carefully on your bench. I employed T80-10 toroids and scrunched or expanded the 16 turns of # 22 AWG wire until they measured exactly 1000 nH.

The best return loss will only occur when your filter is perfectly tuned to the test frequency, so tune carefully.



Above — The schematic + bench analysis for the 20 Meter band double tuned band-pass filter. My original design called for 22 pF series end capacitors to get a decent return loss. After building and measuring the circuit, the results were disappointing: insertion loss = 3.7 dB and a return loss = 17 dB. I wanted a better S11 and IL, so I decreased the end capacitors to 20 pF and savored the measured data shown in the schematic.

Simulating this tweaked design in GPLA08 unveiled a lower return loss than the original design simulation with 22 pF end capacitors; exactly opposite to my bench observations.

Bench work reveals the truth — The filter *you* get is dependent on factors such as parts types + tolerances, stray reactance, layout, test gear and any bench errors. For example, I don't know the Qu of my 1 uH inductors, but suspect that the Qu is greater than the 250 specified. Also my intended - 3 dB bandwidth was 350 KHz, yet my filter = 315 KHz; in part, because I lowered the series end capacitors, but also due to other fore mentioned factors.

Many popcorn builders can't easily measure their filter bandwidth. Does it really matter? Probably not, however, the big realization for me is that unless you measure, you won't actually know your data like insertion loss, return loss, or bandwidth — simulations are great, but don't obliterate the need for bench testing as possible.

Consider this; with the SPICE program you can design a circuit with a 2N3904 and run 400 mA of current through it — the transistor won't smoke 1 bit ! Project outcomes depend on understanding and employing best practices, experience and measurement on the bench. Finding best practices proves difficult in a day and time when general scientific literacy, the number of expert mentors and interest in analog electronics are all waning.

Click for another photo of the filter. On my actual filter, I used high Q, air variable trimmer caps that only had a capacitance variation of 15 pF or so. I soldered in fixed capacitors to get close to the capacitance needed to tune each tank. If possible, I think its better use smaller value trimmer caps because they permit finer tuning. The air variable trimmer offers high Q plus you can see when the capacitor is fully meshed (maximum C). This signals that you need to add more fixed capacitance to that tank for peaking.

Conclusion

To repeat; our parts collection dictates our band-pass filter outcomes. Size 50 to 68 #6 material toroids will work fine for most HF frequencies above 3 MHz. Don't stress out too much if your insertion or return loss is a little higher than you wanted; in all likelihood your filter will work fine and you'll be glad you didn't just copy some else's design and rob yourself of the design experience.

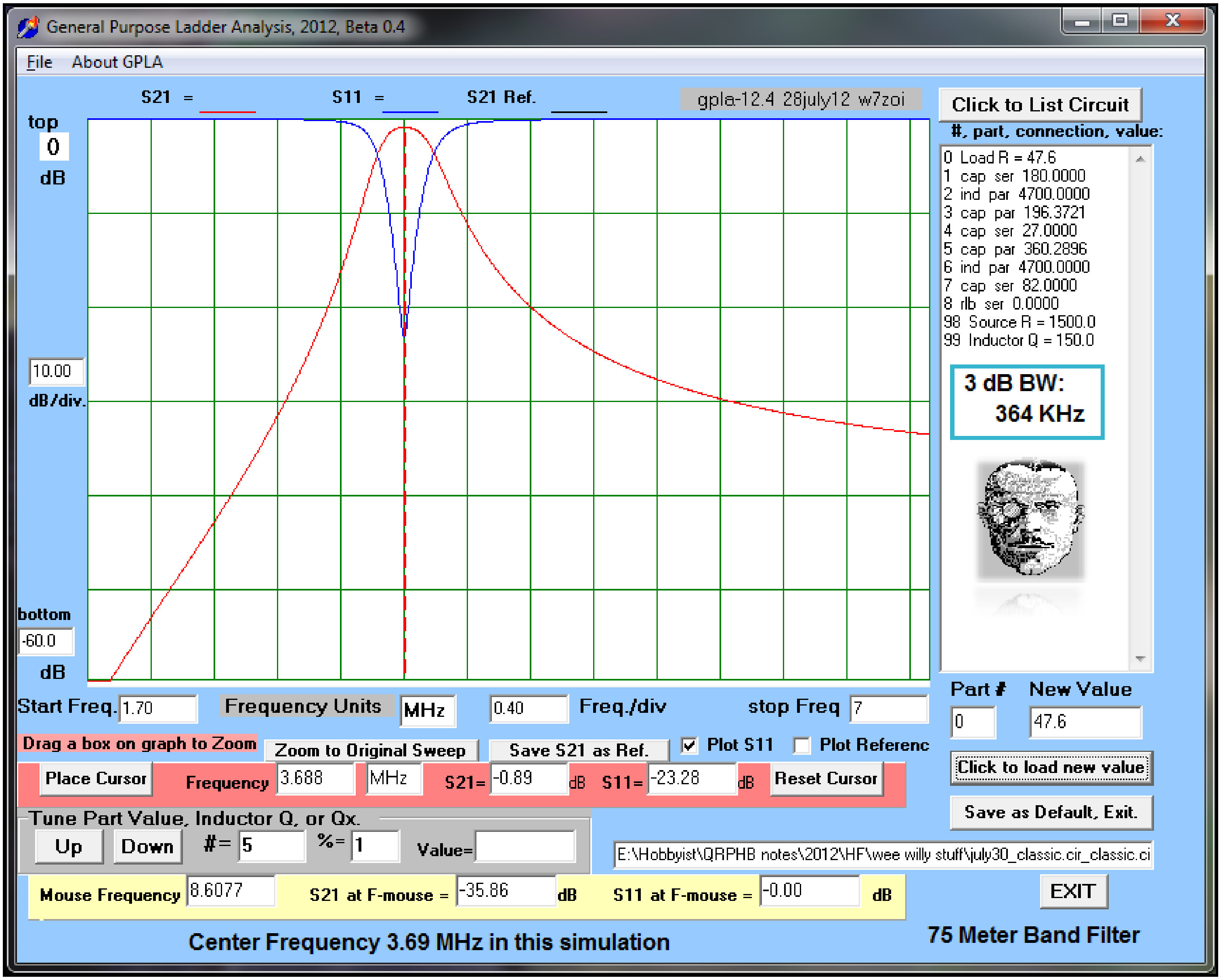
I am hopeful, this web page will inspire a few builders to experiment with band-pass filters for their receivers and other applications. My sincere thanks to Wes, W7ZOI for his guidance with filter design.

QRP — Posdata for August 2012 — NE612 Mixer Band-pass Filters

I designed some band-pass filters for NE612 based front-ends with LadBuild and GPLA and show my 75 Meter band filter design figures below. I'm not a fan of employing an NE612 as a receiver mixer since it easily overloads and spews harmonics when mixing strong input signals. The 1K

'RF gain' pot found in many receivers, or the more conventional switchable attenuator pad prove essential when receiving 'booming' signals with a NE602/NE612 mixer in your front-end.

Still, for field-portable tranceivers/receivers, the NE612 mixer keeps the current and radio size down nicely.



gpla-12.4 28july12 w7zoi

top 0 dB

10.00 dB/div.

bottom -60.0 dB

#, part, connection, value:

0 Load R = 47.6
1 cap ser 180.0000
2 ind par 4700.0000
3 cap par 196.3721
4 cap ser 27.0000
5 cap par 360.2896
6 ind par 4700.0000
7 cap ser 82.0000
8 rlb ser 0.0000
98 Source R = 1500.0
99 Inductor Q = 150.0

3 dB BW: 364 KHz

Start Freq. 1.70

Frequency Units MHz

0.40 Freq./div

stop Freq 7

Part # 0

New Value 47.6

☑ Plot S11

☐ Plot Referenc

Frequency 3.688 MHz S21= -0.89 dB S11= -23.28 dB

Tune Part Value, Inductor Q, or Qx.

#= 5 %= 1 Value=

E:\Hobbyist\QRPHB notes\2012\HF\wee willy stuff\july30_classic.cir_classic.ci

Mouse Frequency 8.6077 S21 at F-mouse = -35.86 dB S11 at F-mouse = -0.00 dB

Center Frequency 3.69 MHz in this simulation

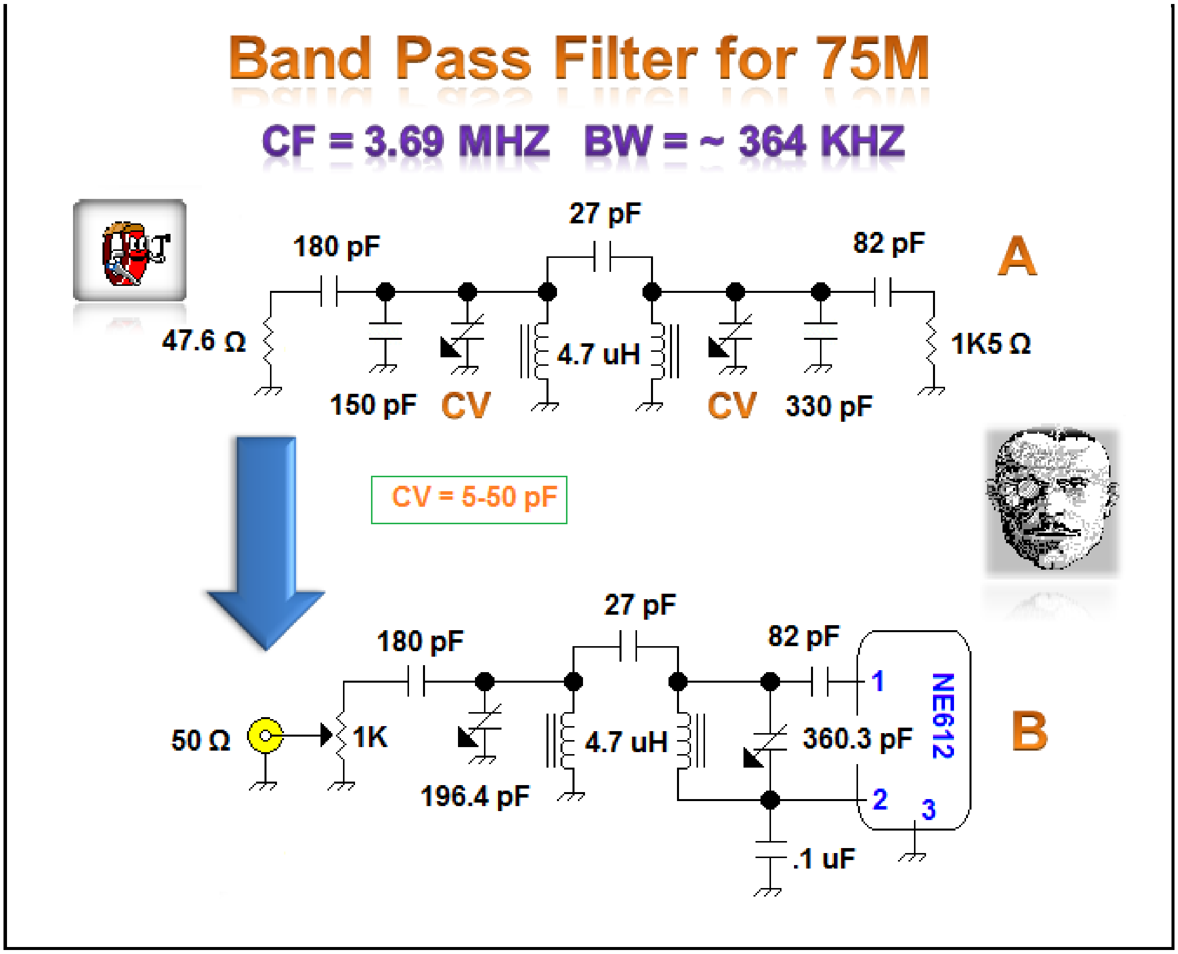
75 Meter Band Filter

Above — My filter centered at 3.69 MHz. I set the 3 dB bandwidth higher than my usual 200-300 KHz so I could tune a good chunk of the 80-75M band without losing too much signal.

To establish some starting L and C values, I built a classic form filter in DTC08 with a 50 Ω input and output impedance centered at 3.69 MHz. After some tweaking, I settled on L = 4700 nH, Ce = 148 pF, coupling capacitor Cm = 27 pF and about 220 pF (Ct) to resonate each tank.

Next I changed the termination R's to 1500 Ω in DTC08 to simulate the right half of the filter matched into the 1K5 input impedance of an NE612. This gave me some Ct and Ce values to start with. I started Ladbuild 08 and built up a schematic.

In a seperate experiment, I determined that the resonator Qu of a 3.7 MHz L-C tank with an L of 4700 nH wound on a T50-2 core, was ~150.



Above — The completed schematic. My filter exhibits an attenuation of ~70 dB at the top of the AM radio broadcast band (1500 KHz) providing I shield it in an RF-tight box. I took the 47.6 Ω input Z from the 1K pot and my 50 Ω antenna in parallel. An input Z of 50 Ω would work just as well in simulation and on-bench.

In Part A, I show a possible way to resonate each tank with 1 fixed C and a trimmer capacitor. In Part B, I omitted this detail and just show the calculated C needed for resonance as a variable capacitance.

To make this filter with GPLA, I tweaked the capacitor values to nearest standard value parts and tuned the filter with the GPLA Tune Part Value controls while looking at the waveform and my 3 dB bandwidth. I love tweaking values in GPLA and over the years have designed several hundred RF filters for readers.
The rubber hits the road on the bench however! You can get an E.E. degree without melting solder in this day and time — but only bench measurements tell the truth.

Please tune each tank carefully like I mentioned earlier... For example, say a tank needs 180 pF for resonance, *but you don't know this*. You solder in a 100 pF cap and a 5-50 pF *enclosed* ceramic trimmer capacitor into the L-C tank. While watching the 'scope this tank will "peak" since the tank will exhibit its highest peak-peak voltage when the trimmer cap is set to 50 pF and fall off as you decrease the C of the trimmer. You might think you peaked the tank, however you' re actually under by 30 pF!

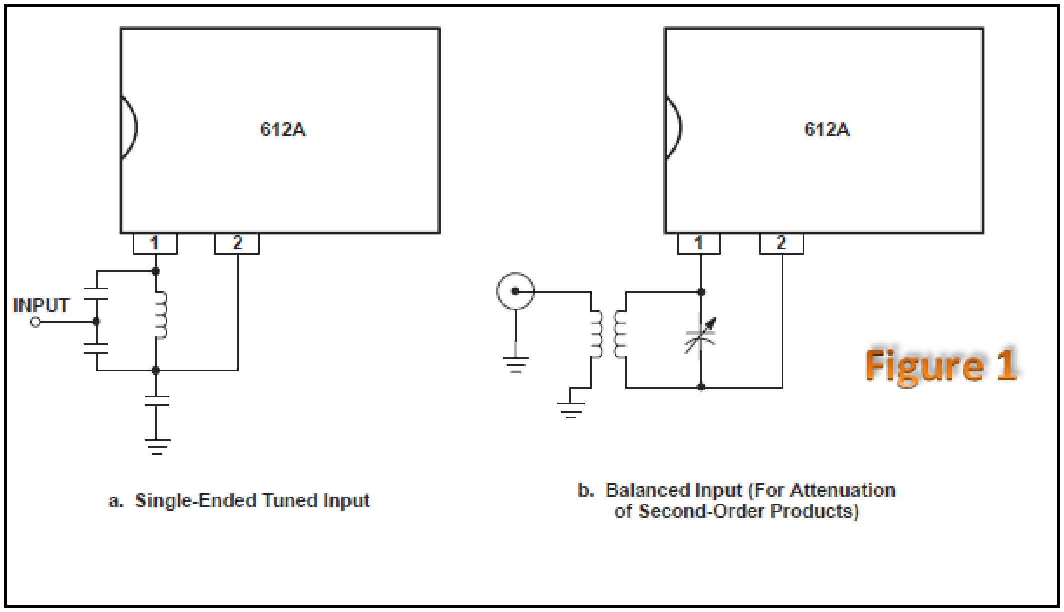
While leaving the trimmer set to maximum C (peak-peak voltage) in this theoretical example, if you tack solder in another 10 pF cap your 'scope will show an even greater pk-pk voltage. , If you remove this 10 pF cap and then place in a 27 pF cap, the pk-pk voltage will go even higher since you're almost at the target 180 pF. If you removed the 27 pF cap and tack soldered in a 47 pF cap, the pk-pk voltage in the 'scope will go down since your now at 197 pF. Thus you know that resonance is somewhere between 177 and 197 pF.

Of course you could decrease the trimmer cap C and stil use the 47 pF cap, however, my description isn't a prescription to follow, just some things to think about. Sometimes I remove a trimmer cap and measure it to ensure the cap is not set to maximum C; that would tell me I need to add more fixed capacitor(s) to the tank. Air variable trimmer caps give visual indication since maximum C occurs with maximum mesh. Unfortunately they are rare and expensive.

On my bench I keep a pair of small 12 to 400 pF air variable caps and temporarily solder them into my tanks. After peaking, I remove and measure them — then I have a good idea of what capacitance is needed to resonate the tank at my test frequency.

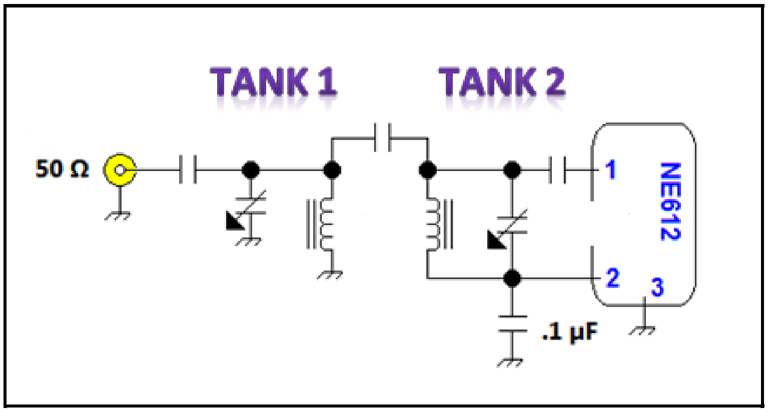
It's all an experiment.

Click for another low loss, well matched example: CF = 5.17 MHz, 3 dB BW = 196 KHz. 50 ohm version. NE612 final version.



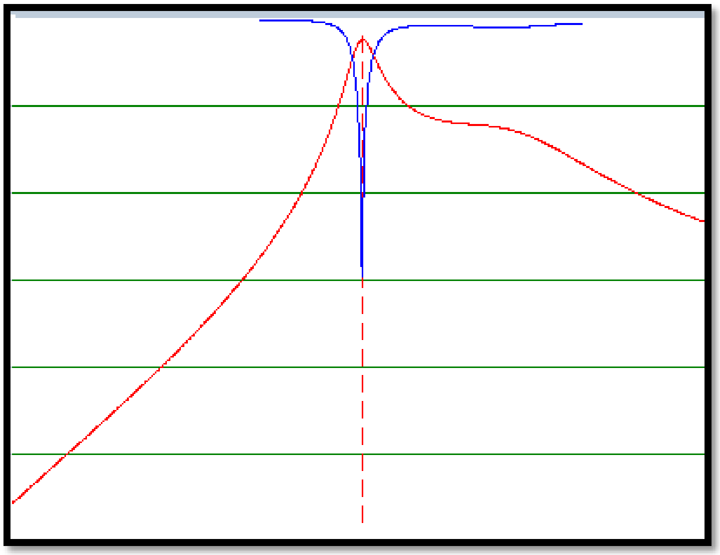
Above — NE612 input circuits. The NE612 datasheet specifies a 3 pF input capacitance + a 1K5 input resistance.

If you look around the Web, many builders just run a single tank for band-pass filtering. While okay for novelty-grade rigs, the poor filter stopband may unleash some ugly problems in the mixer and on down the receiver chain.

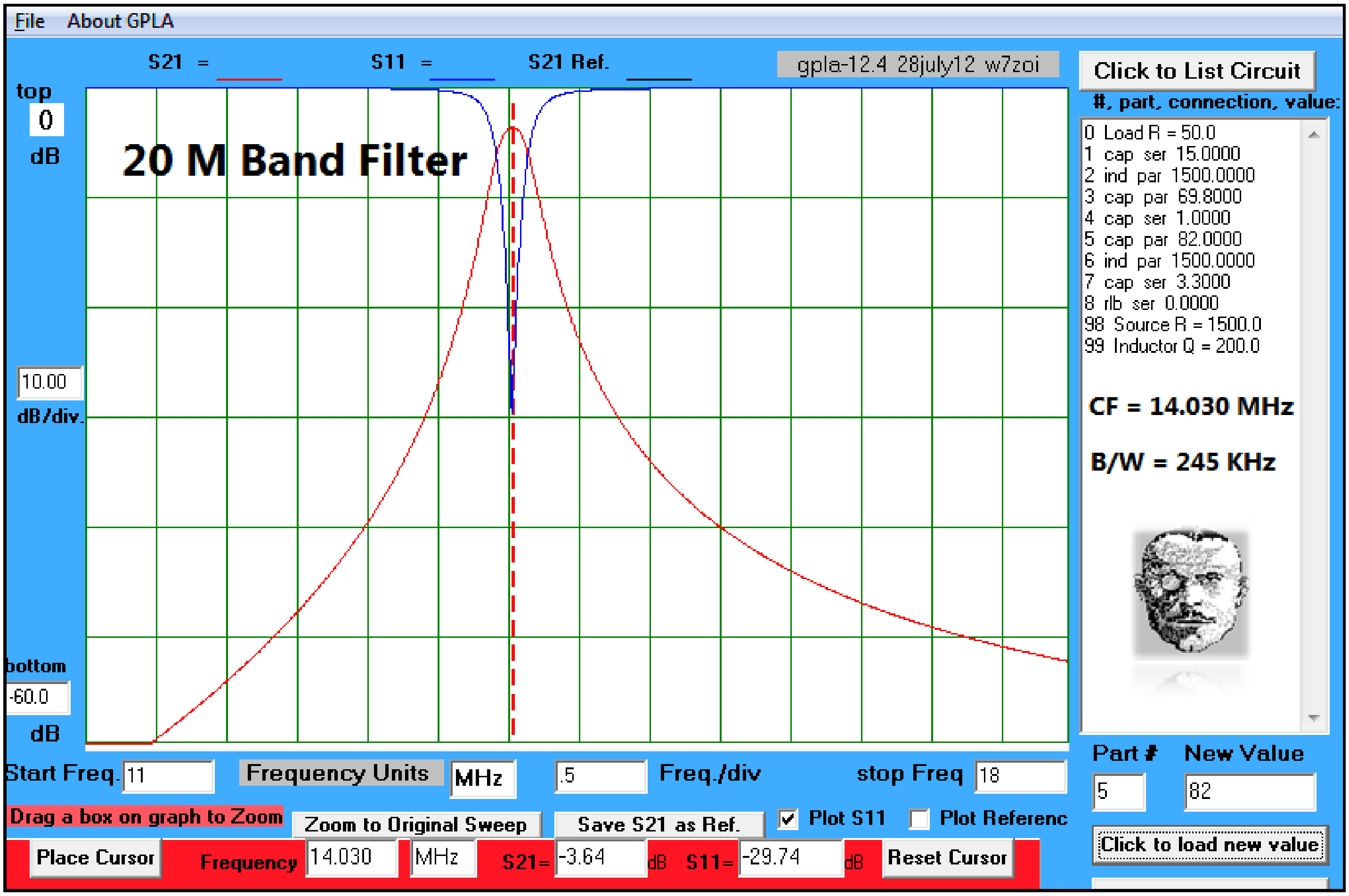


Above — A double tuned circuit with the L-C tanks named 1 and 2 and a series capacitor to match Tank 2 to the NE612.

Most builders match Tank 1 into its 50 Ω source with a capacitor divider, or a matching transformer. For Tank 2, some enthusiasts just connect the Tank 2 coil directly to pin 1 as shown in Figure 1a. Without the matching series capacitor, unfortunate side effects may arise...



Above — The low-pass skirt of a double-tuned filter may attenuate higher frequencies poorly when no series capacitor (or other network) matches Tank 2 to the NE612 input. I perfectly matched Tank 1 to its 50 Ω source just connected Tank 2 to a 1500 Ω resistive load in this simulation. I wonder how bad things get in the real world when a complex impedance is involved?



#, part, connection, value:
0 Load R = 50.0
1 cap ser 15.0000
2 ind par 1500.0000
3 cap par 69.8000
4 cap ser 1.0000
5 cap par 82.0000
6 ind par 1500.0000
7 cap ser 3.3000
8 rlb ser 0.0000
98 Source R = 1500.0
99 Inductor Q = 200.0

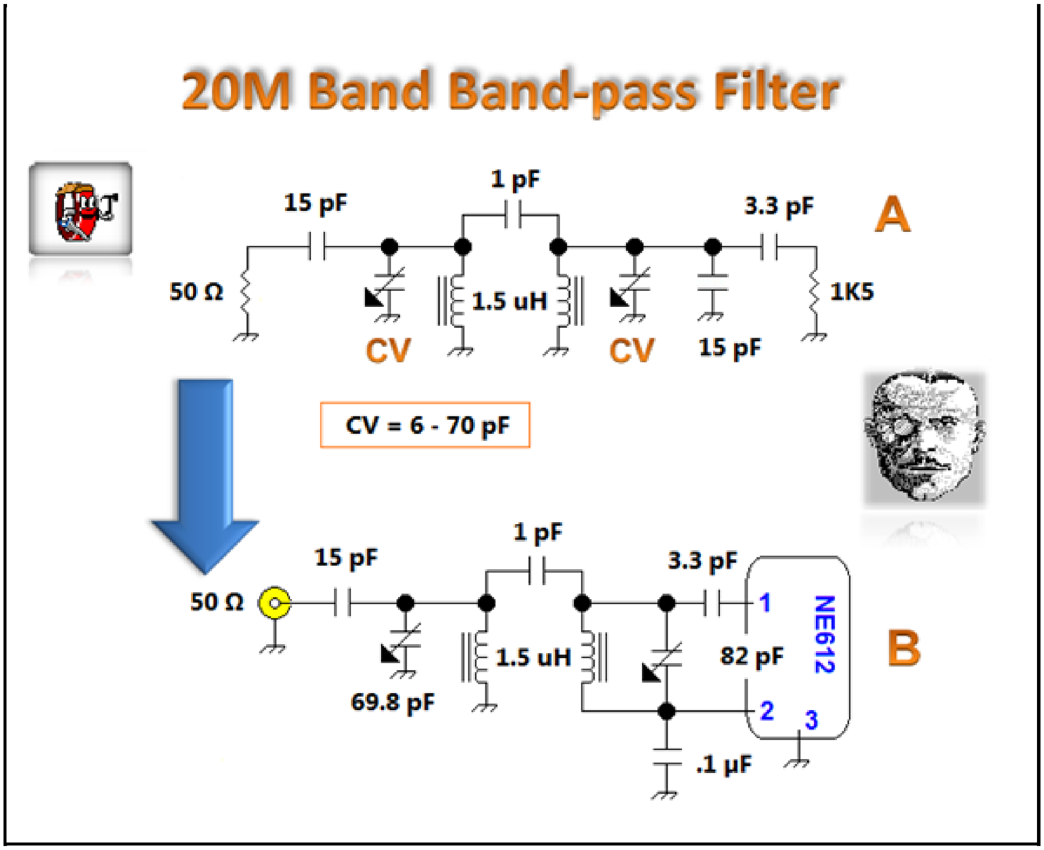
CF = 14.030 MHz

B/W = 245 KHz

top 0 dB
10.00 dB/div.
bottom -60.0 dB
Start Freq. 11
Frequency Units MHz
.5 Freq./div
stop Freq 18
☑ Plot S11
☐ Plot Referenc
Frequency 14.030 MHz
S21= -3.64 dB
S11= -29.74 dB
Part # 5
New Value 82

Above — I designed a filter for a 20 Meter band CW receiver centered at 14.030 MHz with DTC and GPLA. The design 3 dB bandwidth = 245 KHz. NE612 filter design was discussed in QRP — Posdata 1.

I then breadboarded the filter with T68-6 inductors, but common lower Qu ceramic trimmer capacitors.



Above — The DTC/GPLA filter design with Tank 2 evolved to provide single-ended input for the NE612.

I wanted to test 2 questions:

1. Does the 0.1 µF coupling cap connected to the cold end of Tank 2 and Pin 2 change the bandwidth or filter skirt shape?
2. Will the 3.3 pF cap really match the NE612?

I expect that worldwide, the NE612 input impedance may vary slightly from part to part; different breadboards will exhibit different reactances and that although the datasheet specifies 1500 Ω , we may be dealing with a complex impedance that varies with the aforementioned factors plus perhaps, input frequency. I simply want a good filter with clean skirts.

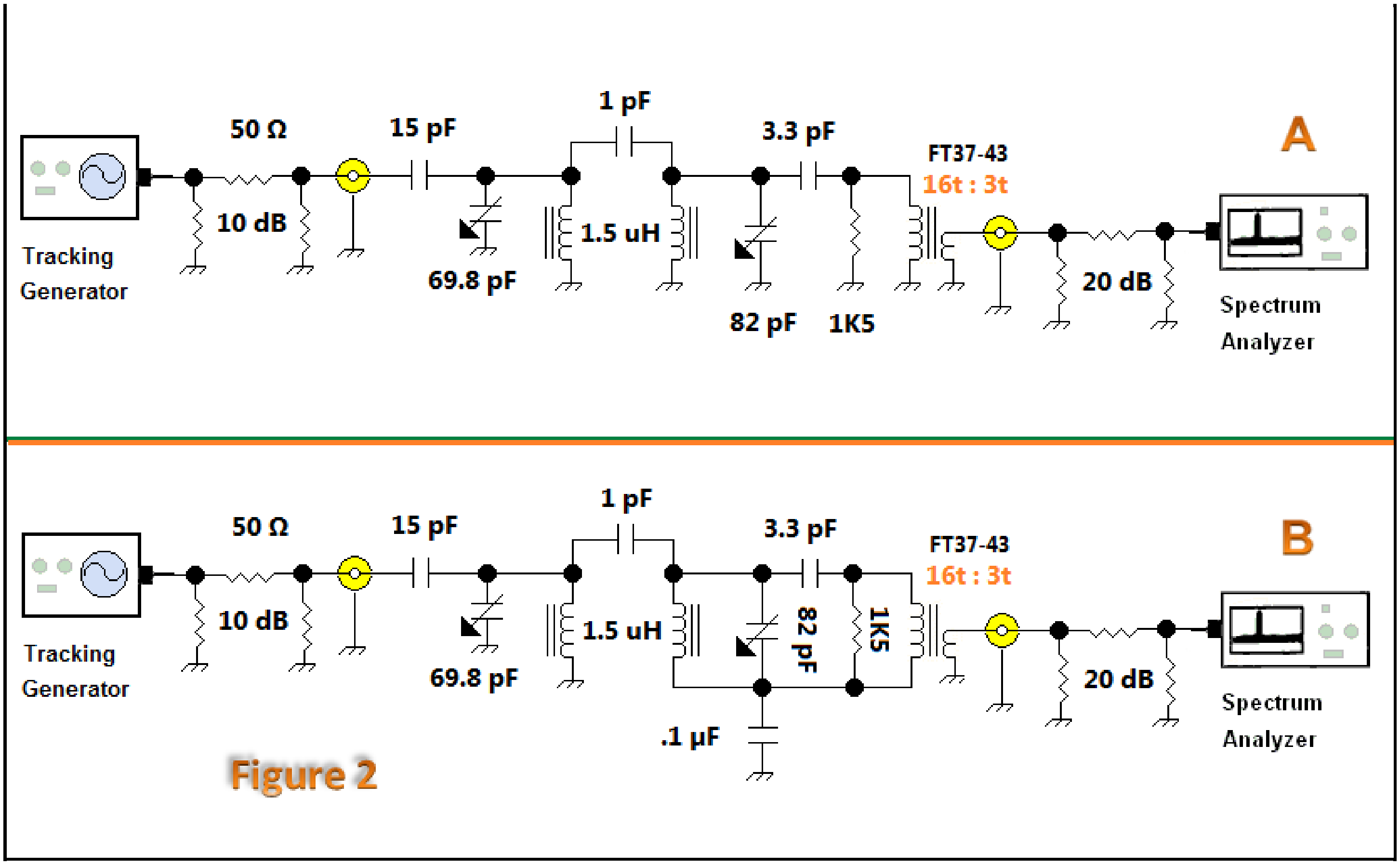
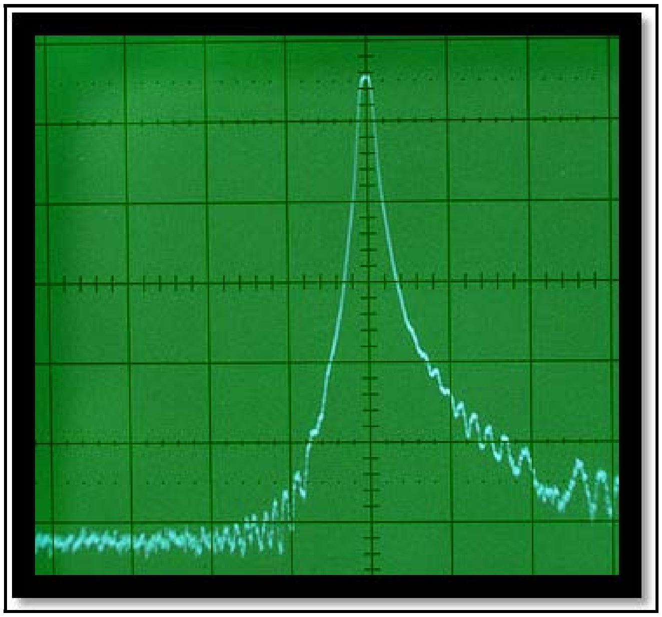


Figure 2

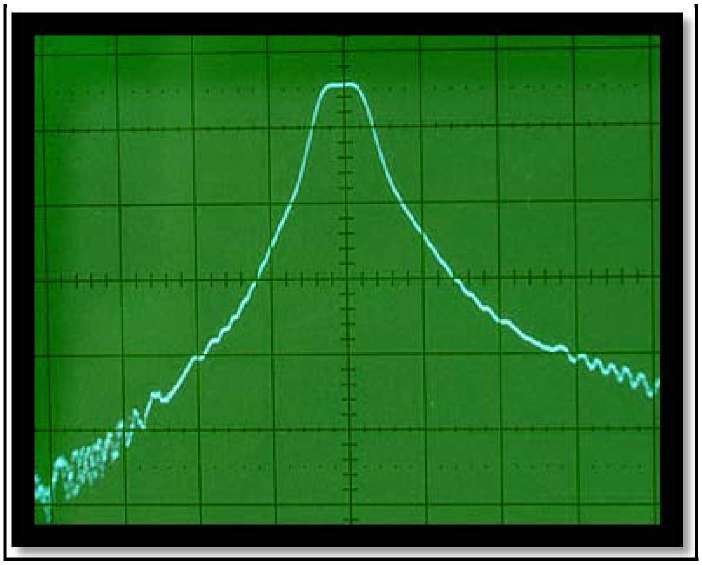
Above — Experiment #1. I compared spectrum analysis of circuit A with B. I built and measured A and then cut away some copper to isolate the copper board grounding the Tank B parts. This "island" was AC coupled to the rest of the ground plane with a short leaded 0.1 µF ceramic capacitor.

I saw no significant difference between Circuit A and B — it appears the 0.1 µF capacitor does not affect the filter parameters to any extent.

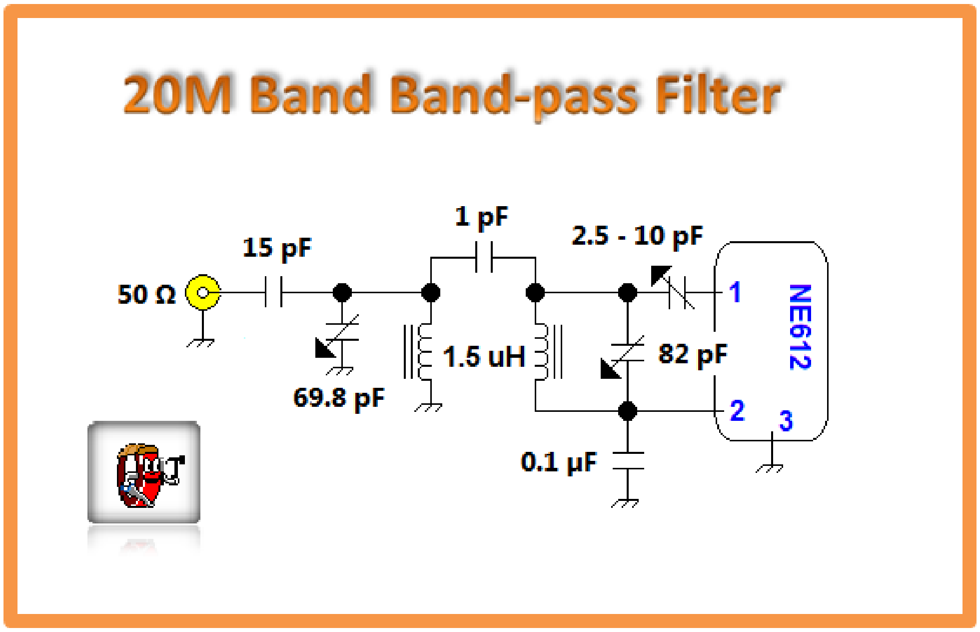
To simulate a NE612, I soldered a 1K5 Ω 5% resistor across Tank 2. Tank 2 was transformer coupled to the 50 Ω Z required for spectrum analysis. A 20 dB pad ensured a strong return loss and a safe input amplitude for the SA.



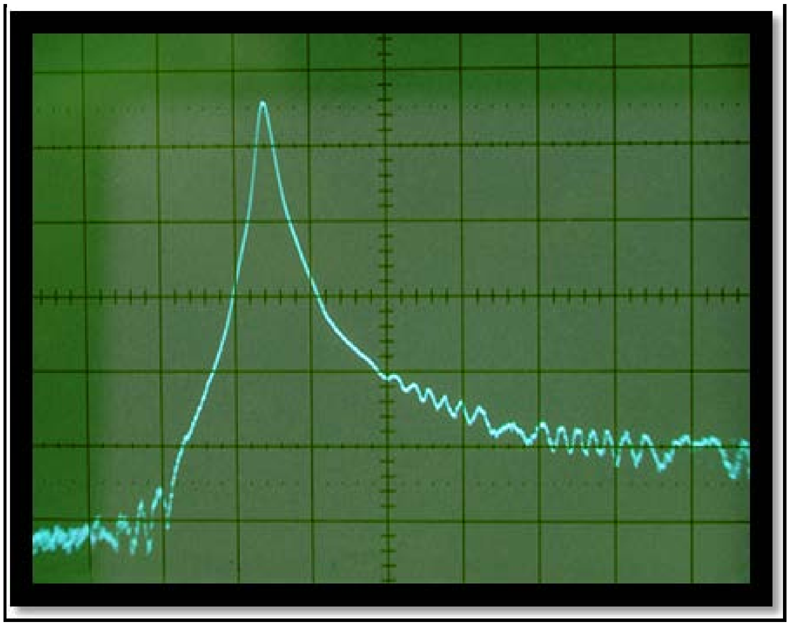
Above — Spectrum analysis of Figure 2A or B. I saw flat topping of the waveform – almost double humping with a higher than wanted 3 dB bandwidth; this bothered me. Fixing this problem was Experiment #2.



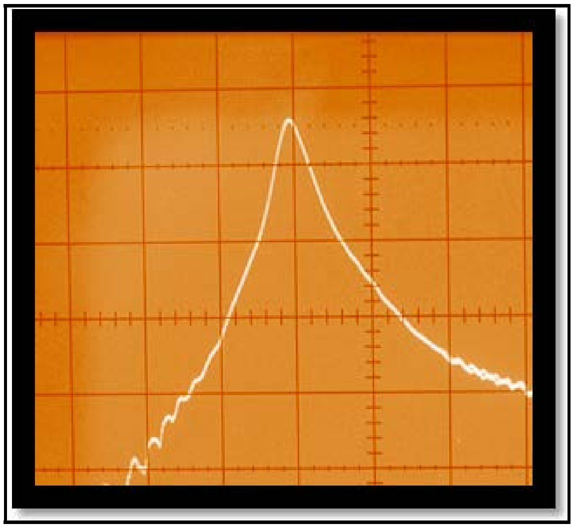
Above — A zoom of the poor coupling of Figure 2A or B. From my experience building filters, I suspected a termination resistance mismatch in Tank 2 ; exactly what I was trying to avoid!



Above — I swapped a trimmer capacitor for the series 3.3 pF fixed cap in Tank 2. Then I reconnected the circuit into my test set up.



Above — A photo of my TG + SA measurement after tweaking the newly added trimmer capacitor and peaking each tank. I shifted the SA screen center over so the tracing could be seen without all the hash marks in the center. Wonderful.



Above — A zoom of the now matched Tank 2. Due to the bench-altered Tank 2 match, my low-Q variable capacitors and other factors, the 3 dB bandwidth now is just over 300 KHz.

I'm not sure if these experiments reflect what actually happens with a NE612 input band-pass filter, however, I plan to match my second tank with a series trimmer capacitor in future NE602/NE612 work. I also want to explore balanced input.

RF Workbench Page 4

The 4th installment of a QRP/SWL HomeBuilder series exploring *basic* RF measurement

Part 4 describes a method to calculate reverse isolation in the **50 Ω environment** after converting measured peak-to-peak AC voltages to dBm. I tested 2 common amps at ~7 MHz to show the concepts and calculations.

In this series, I gratefully borrow from the work of Wes, W7ZOI per correspondence, direct contributions and from EMRFD.

Tools Needed

1. 50 Ω terminated scope (or a spectrum analyzer) and a 50 Ω signal generator
2. 50 Ω RF cables with RF connectors (such as short cables with female BNC connectors)
3. 6 dB 50 Ω attenuator pad; plus an adjustable attenuator if you use a fixed output signal generator.
4. BNC through-response connector(s)

Procedure

1. Measure the amplifier forward power gain. This is S21.
2. Measure amplifier reverse power gain. This is S12. Like S11, express S12 as a negative value.
3. Reverse isolation using dB values = S21 - S12.

Step 1 : Measure the forward power gain

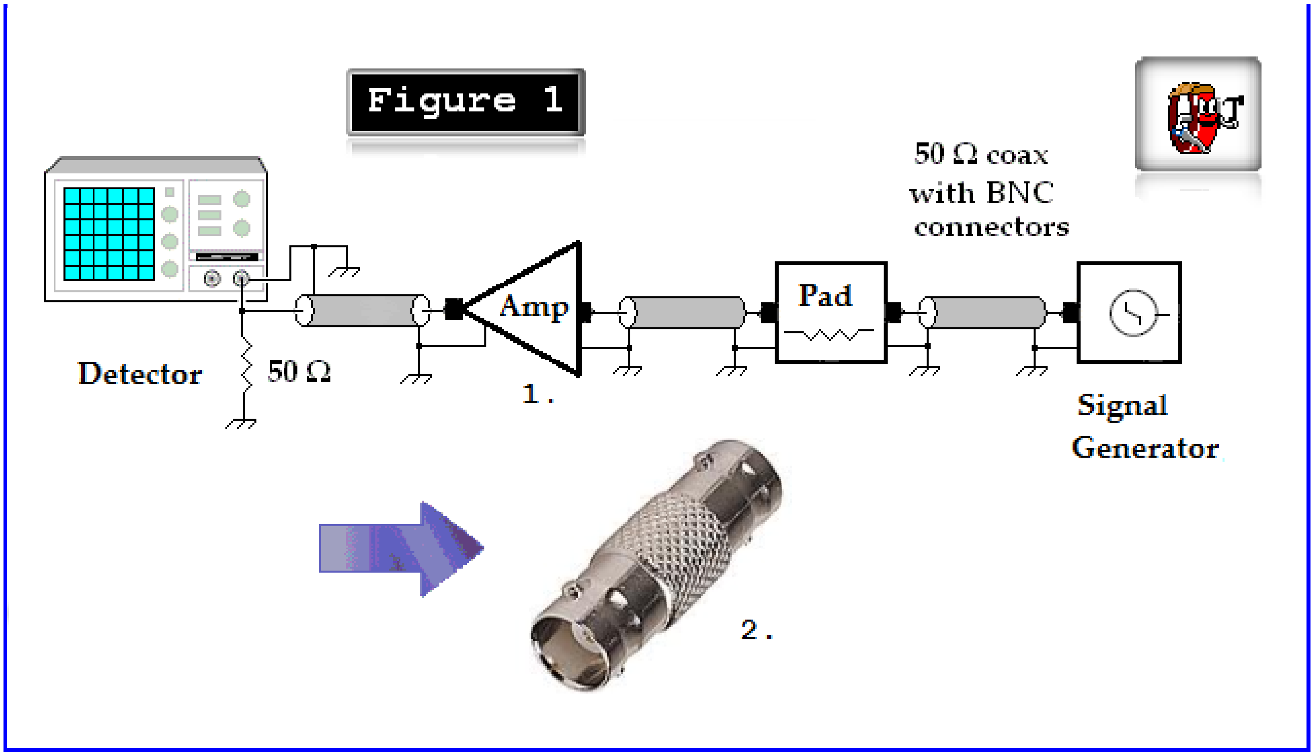


Figure 1

Figure 1 shows how to measure forward gain in a 50 Ω environment — **1**. Convert the measured AC voltage to dBm, **2**. Disconnect the amplifier, insert a through-response connector and convert this measured AC voltage to dBm. The difference between the 2 values = S21. Applet H will do these calculations from the peak-peak voltages. *The attenuation pad following the signal generator in Figures 1 and 2 signify that the signal generators have a 50 ohm output impedance and is optional.*

Choose a signal generator level that ensures the output of your amplifier is linear while providing a good signal to noise ratio for measurement. With an oscilloscope, I generally test amplifiers with an input power of between 0 and -11 dBm; although choose whatever level that works for you consistent with linear amplification.

After measuring the forward gain of your amp, a good way to test for linearity is to add a fixed 6 dB pad between your signal generator and your amplifier to drop the applied signal by half. *[A 6dB pad drops the peak-peak voltage by 1/2 . A 3 dB pad drops the power in dBm by 1/2]*. The power gain should be equal or nearly equal to the measured dB value obtained before you added the 6 dB pad. If they vary significantly, you are likely driving the amp too hard and causing some non-linear output products. S21 = complex *linear* gain.

Step 2 : Measure the reverse power gain

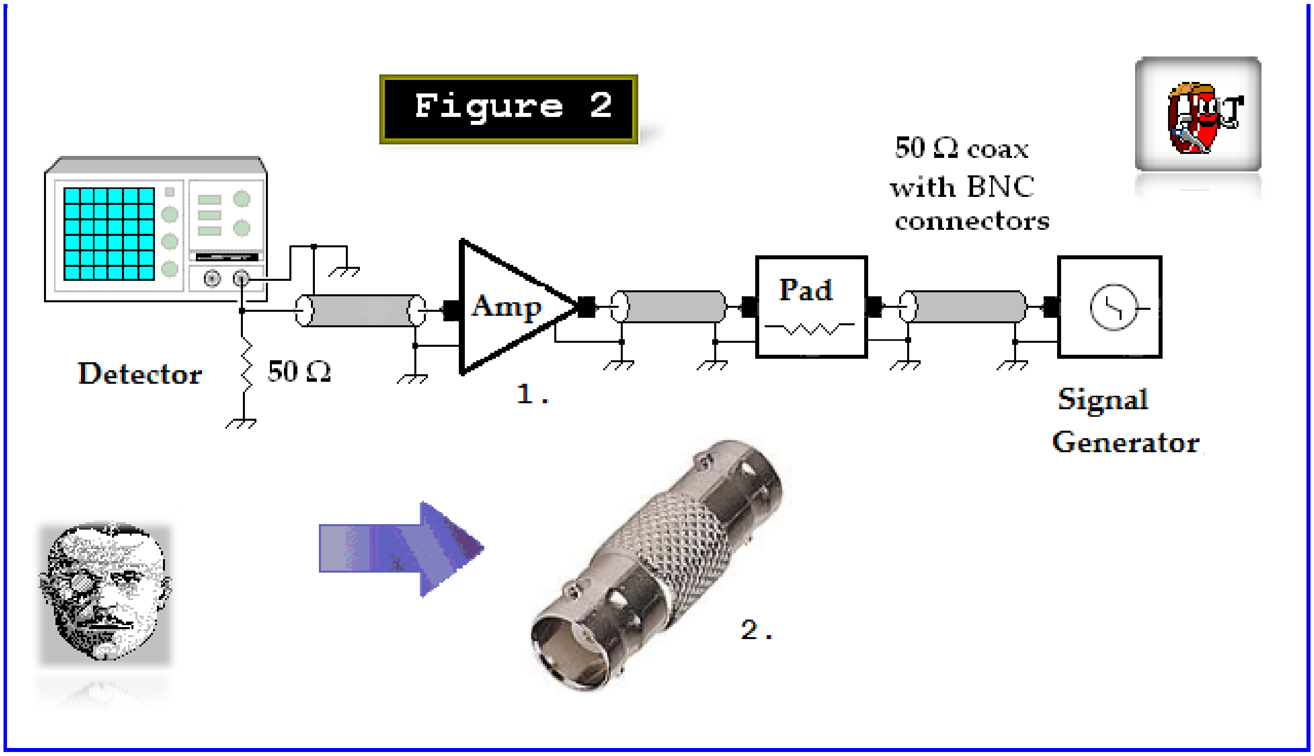


Figure 2

Figure 2 shows how to measure reverse gain in a 50 Ω environment — **1**. Convert the measured AC voltage to dBm, **2**. Disconnect the amplifier, insert a through-response connector and convert this measured AC voltage to dBm. The difference = S12. Ensure that you employ the same drive level used to measure the forward gain.

Measuring reverse gain may be tough. When the amplifier under test requires a low drive level and/or has strong reverse isolation, you may not have enough signal to accurately measure with your oscilloscope. The tool of choice for low reverse voltage measurment is a spectrum analyzer (SA) — a narrow band SA may be required to distinguish the weak signal from random noise with low signal voltages.

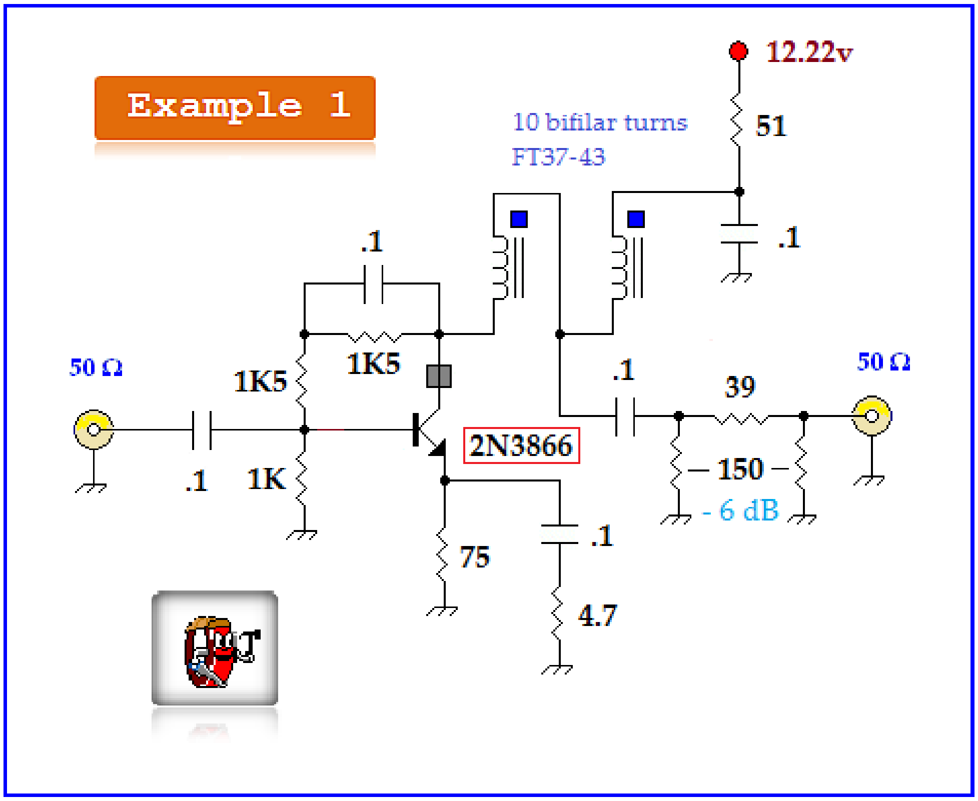
Summary

A **practical** bench work flow goes something like:

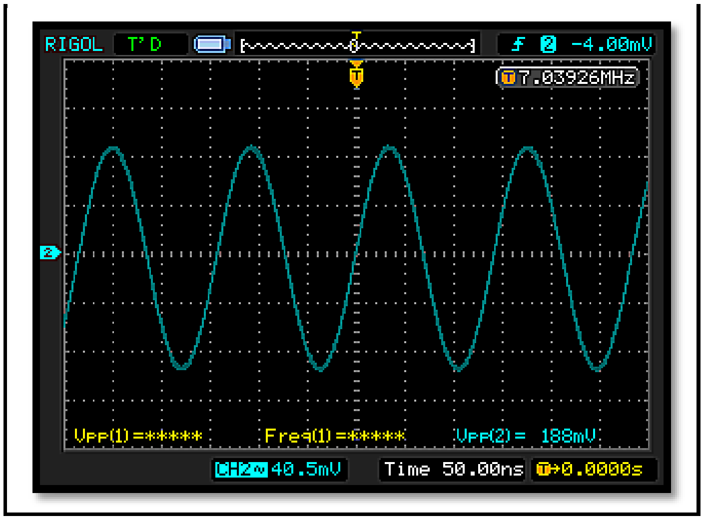
1. Measure the through-response peak-peak voltage.
2. Measure and record the peak-peak AC voltage while driving the amplifier input port (forward gain set-up).
3. Reverse the amp so you're driving the output port and then measure the peak-peak AC voltage (reverse gain set-up).
4. Calculate S21 and S12.
5. Calculate reverse isolation.

Example 1: Feedback amplifier

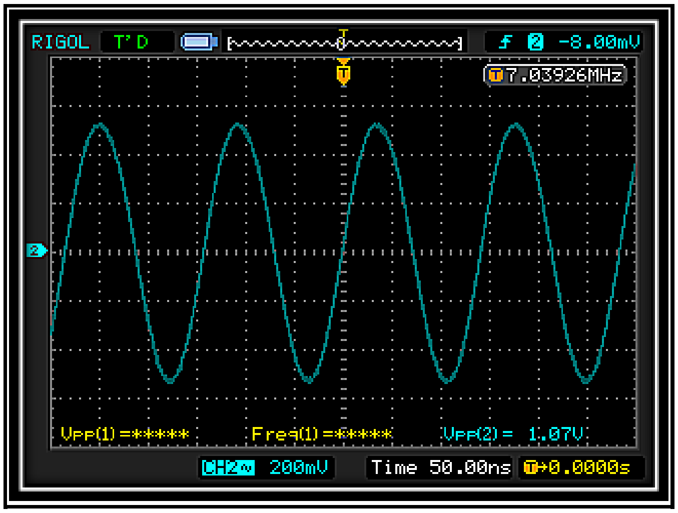
In the first example, I measure the reverse isolation of a Beaverton Special feedback amp. The schematic is shown below. Tested with a 7.039 MHz signal generator possessing a return loss of 30 dB.



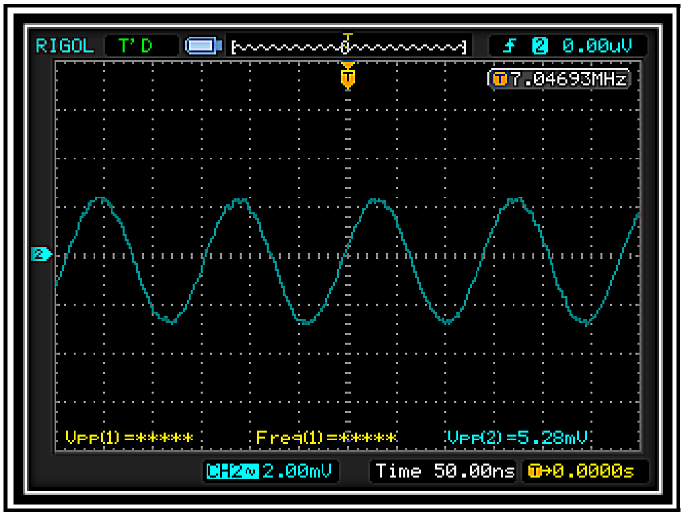
50 ohm Voltage Measurements:



Above — AC voltage with a through-response connector in-situ: 188 mV pk-pk

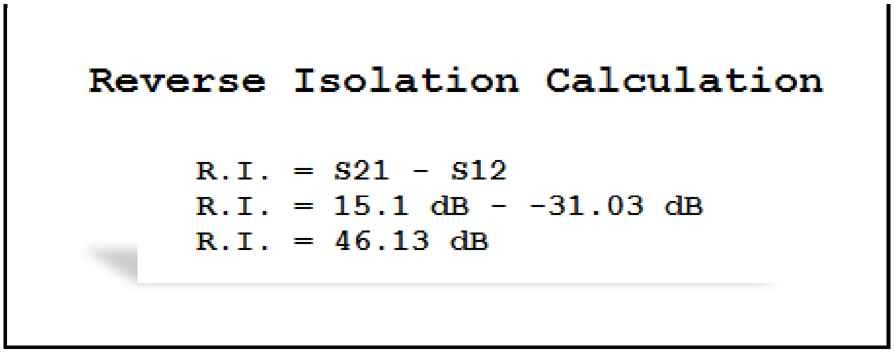


Above — AC voltage with forward gain set-up: 1.07 V pk-pk



Above — AC voltage with reverse gain set-up: 5.28 mV pk-pk.

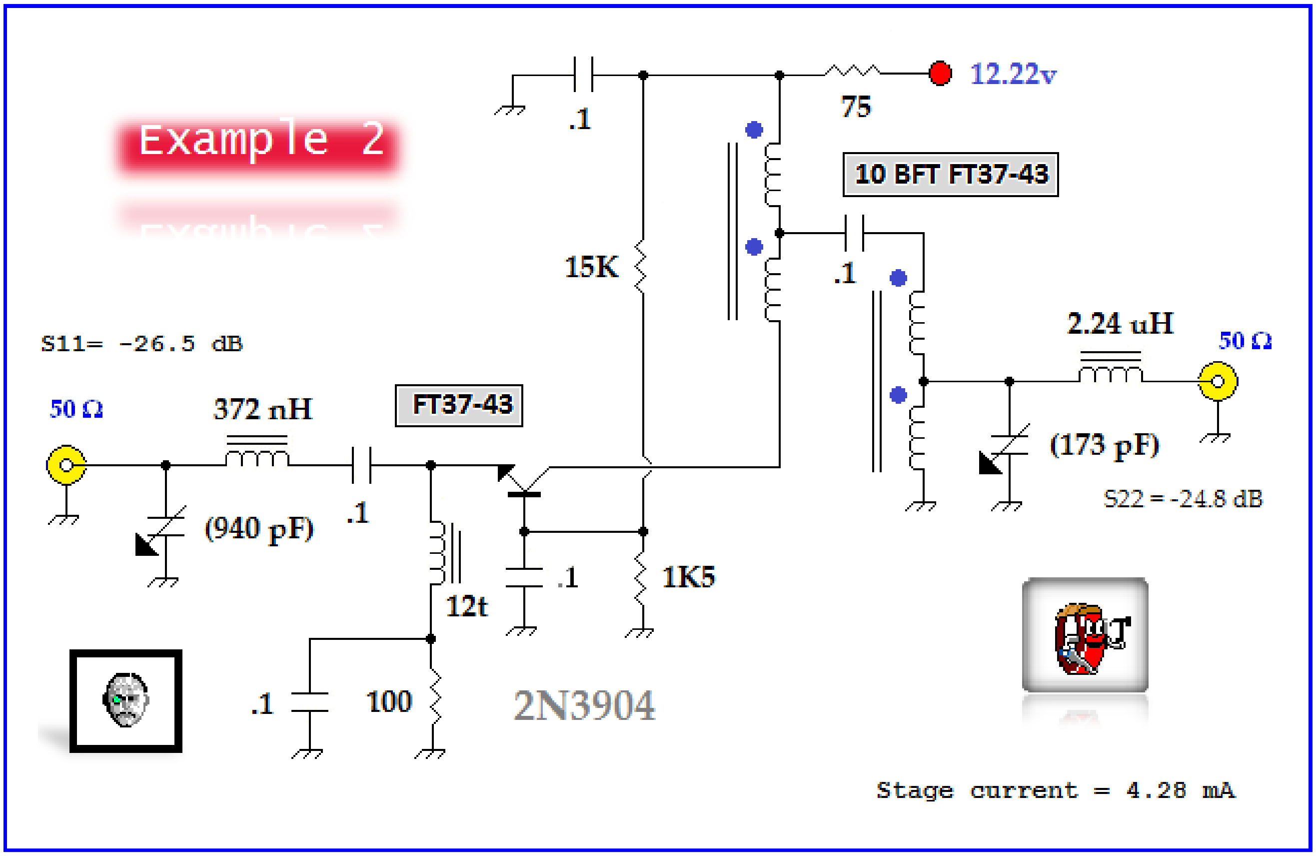
S21 = 15.1 dB. S12 = -31 dB. I tested my driven amp's linearity by adding a scrap, standard value 6 dB pad in between the signal generator and the amplifier — the S21 was 15.2 dB with the pad and 15.1 dB without the added 6 dB pad. It's linear.



Reverse Isolation Calculation

R.I. = S21 - S12
R.I. = 15.1 dB - -31.03 dB
R.I. = 46.13 dB

Example 2: Common Base Amplifier



Above — A common base amp employing L- networks for a strong return loss in and out at 7.039 MHz. S11 and S22 = the negative of return loss.

Through connector voltage = 192 mV peak-peak.

S21 = (1.88 volts peak-peak AC voltage) = 19.82 dB.

S12 = (1.68 mV peak-peak AC voltage) = -41.16 dB.

Reverse isolation at 7.039 MHz = (S21 - S12) = 61.42 dB

I confirmed the linearity of my S21 using the aforementioned 6 dB attenuator pad — I couldn't increase my signal generator output level above the indicated -10.35 dBm, since gain compression emerged in the common base amplifier (reduced AC voltage was measured).

My reverse AC voltage was under 2 mV; about the threshold where my oscilloscope waveform becomes rather ugly and sits in the noise. Clearly, the limitations of measuring reverse isolation with an oscilloscope must be factored. Still, you got to love 60 dB + of reverse isolation in a popcorn circuit.

1 local professional EE told me if you can measure it with a 'scope, you don't have spectacular reverse isolation - no doubt, a spectrum analyzer

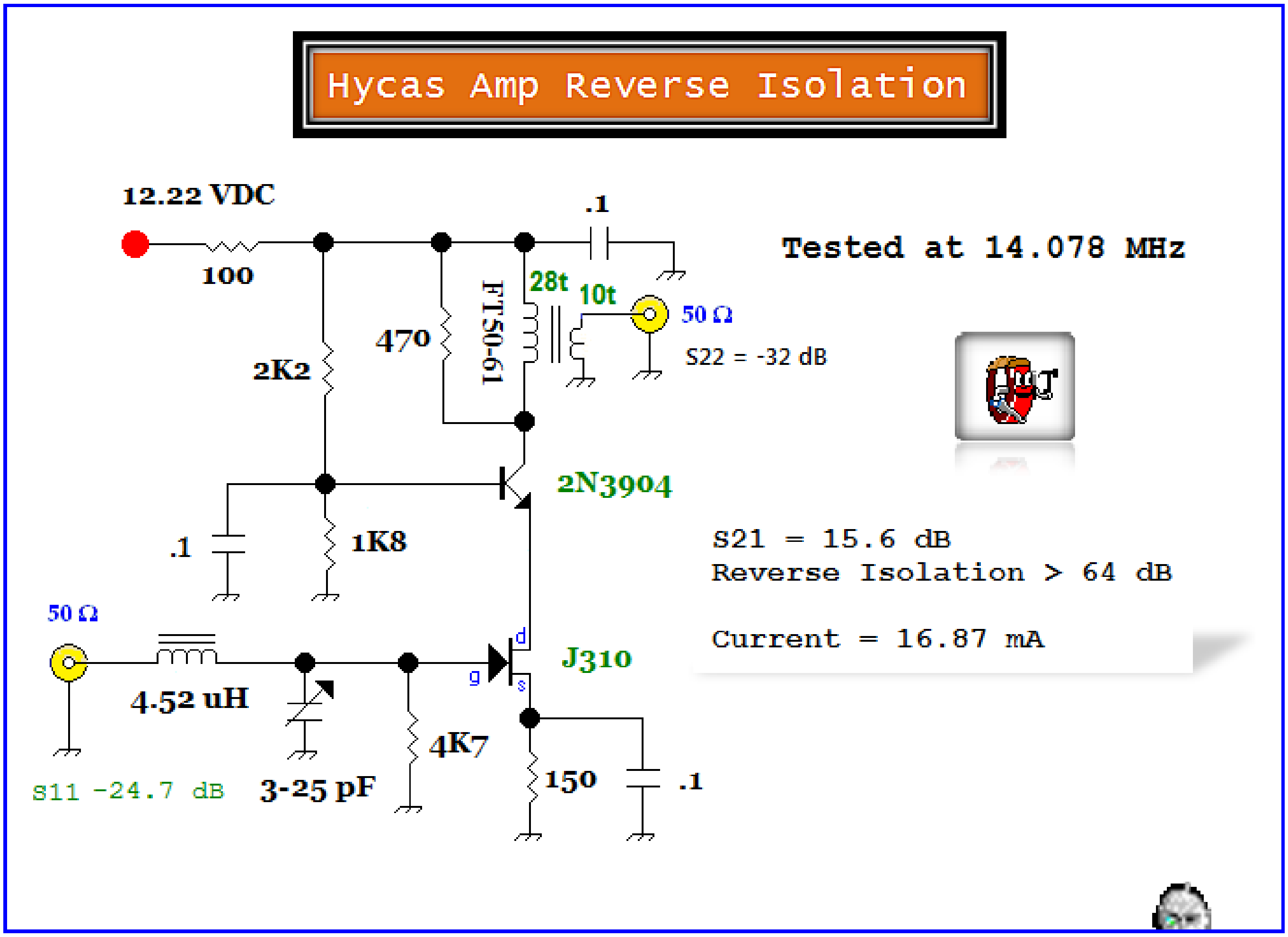
pumps up the measurement quality in circuits with high reverse isolation, but more amateur designers have 'scopes than spectrum analyzers, so just do your best.

Involving Scattering Parameters as possible on your workbench can only lead to circuit improvement. You can't better your outcomes if you can't or don't measure them — applying and more importantly, understanding test equipment is just 1 component of our hobby. A hobby unto itself; test equipment activity complements amateur radio design. I have met test equipment focused builders who make radio gear just as an excuse to apply their test gear!

Improved bench practices are the corollary of striving to learn more about measurement techniques and increasing our collection of measurement devices.

QRP — Posdata 1: Hycas Amplifier

I love the hybrid cascode (hycas) as a general purpose RF amplifier. What's not to love about a using a common source FET followed by a common base bipolar amp? I attempted to measure the reverse isolation of a version just using an oscilloscope.



Above — A hycas amp set up for high return loss on both ends at 14.078 MHz. Too much current may cause gain compression and harmonic distortion, so please test your hycas amps for both. I tested using a signal generator with a 30 dB return loss driving a 50 Ω terminated oscilloscope. Since the hycas amp contain a high impedance input JFET and a common base amp, the reverse isolation should be reasonably high, or at least as good as a common base amplifier.

My testing failed — The reverse isolation was too high to measure with an oscilloscope. Using proper bench techniques (linear amplification + honest scope reporting), I determined the highest reverse isolation I could measure = 64 dB. In fact, injecting a whopping signal of 1.08 volts peak-peak into the hycas output port only gave an S12 of 1.84 mV — whoa!

The problem is such a strong signal (1.08 volts peak-peak) at the input port results in severe limiting and distortion; so valid reverse isolation measurement isn't possible. Even a 350 mV peak-peak signal may give some gain compression during S21 measurement depending on your matching. Thus, I can only accurately say that the reverse isolation of my hycas amp is greater than 64 dB.

Strong reverse isolation is 1 reason I favor hycas amps as VFO buffers. They make pretty good I.F. amps also.

The hycas IF amp system by Wes, W7ZOI and Jeff, WA7MLH offers amazing performance and features an excellent JFET bias scheme. I built 1 in 2008 — amazing design.

QRP — Posdata 2: Doesn't S12 = Reverse Isolation?

Many web sites, books and people report that reverse isolation = S12, yet above, I depart from this argument. In truth, I think reverse isolation equals S12, but reverse isolation may also equal S21 - S12.

I'll let you decide what to do, but explain why I enjoy the latter.

S12 is a negative value.

I prefer to turn that negative value into a positive 1 — RF Workbench 4 concerns measuring and applying amplifiers with the goal of high reverse isolation and not just measuring S12.

The main purpose to quantify amplifer reverse isolation is to strive to *improve reverse isolation* and an amplifier is but 1 component in our 50 Ω block. I believe in a creative, systems approach; open minded and positive (pun intended).

The whole RF Workbench series attempts to present 50 Ω bench measurements in a vibrant way devoid of excessive + boring engineer-style content that could blank the eyes of the budding Hams/SWLs designers that visit my site. I imagine this web site bores more advanced RF designers *to tears*.

Our goal is to obtain high reverse isolation while applying a 50 Ω systems approach.

Break away from strict + "stodgy" math-driven methods to fuel creative thought and experimentation.

I posit that an appropriate figure of merit in a well designed isolation amplifier is the difference between S21 and S12; and therefore, the term reverse isolation can be more than just S12.

Ham/SWL component-level experimentation by commoners like me is ~~slowly~~ dying and being replaced by a new generation of skilled, code writing experimenters. Although, some builders just copy other people's code and then apply it to kitted hardware. Reverse isolation impacts both our analog and digital designs.

Please consider this example:

I'm switching a level 7 diode ring mixer with 12 MHz and per normal, create lots of internal harmonic energy. With my spectrum analyzer connected to the mixer RF port I measure my 12 MHz LO signal at 50 dB below a +7 dBm signal, or -43 dBm. That's a 50 over S9 signal — very high amplitude in context!

I require strong LO isolation in my circuit and thus stick in a 50 Ω input/output amplifier. I measure this amp: S21 = 15 dB and S12 = -31 dB.

So, the signal at the amplifier input is -43 dBm plus -31 dB = -74 dBm. But, alas, -74 dBm isn't good enough me — I want to use that amplifier to elicit greater isolation. However, I don't want any gain in my system, so I insert a 15 dB attenuator pad after the amplifier. For this pad, both S12 and S21 = -15 dB. For my amp, the net cascade is S21 = 0 and S12 = -46 dB.
Since S21 = 0, the block has 0 impact on the signal amplitude applied to the mixer, but the signal at the input of my isolation amplifier is -43 dBm plus - 46 dB, or -89 dBm. This isolation I like — it also illustrates a systems approach that gets you thinking about measurement in your own 50 Ω blocks.

The figure of merit for making a good isolation amplifier is now the difference between S21 and S12. If you want, go ahead and just use S12 for reverse isolation, but you'll probably measure S21 plus S12 anyway and that's what this web page is about! Onward.

QRP — Posdata 3

Comments From the Workbench

I'm no amateur electonics expert — I'd like to be one, but this is a tough field; RF and AF *design* is quite scientific, under-resourced and a bit overwhelming. How do we experimenters advance and stay motivated? Reading works by professionals like Chris Trask, N7ZWY, Bob Larkin, W7PUA, Doug Self, Rod Elliott and others may highlight our lack of knowledge and scientific methodology — a realization which can distress and demotivate us lay-designers. To a degree, this is irrational thinking; personal growth is always about hard work, problem solving and overcoming barriers.

Unlike the white belted Karate student, who studies and practices under the guidance of a master to attain black belt skill level, most amateur designers, excluding electrical engineering students, can't access good teachers. As a lay-person, with few face-to-face mentors (nobody in

Canada), I try to learn by experimenting and incorporating whatever knowledge, advice or schematics I can find. Fortunately, some Electrical Engineers give me advice by email and in turn I'm able to share this information via experiments on QRP / SWL HomeBuilder.

Our dusty, analog hobby fades palpably — the number of analog electronics gurus dwindles each decade and modern electronics embraces miniature circuitry often involving digital ICs controlled by lines of code.

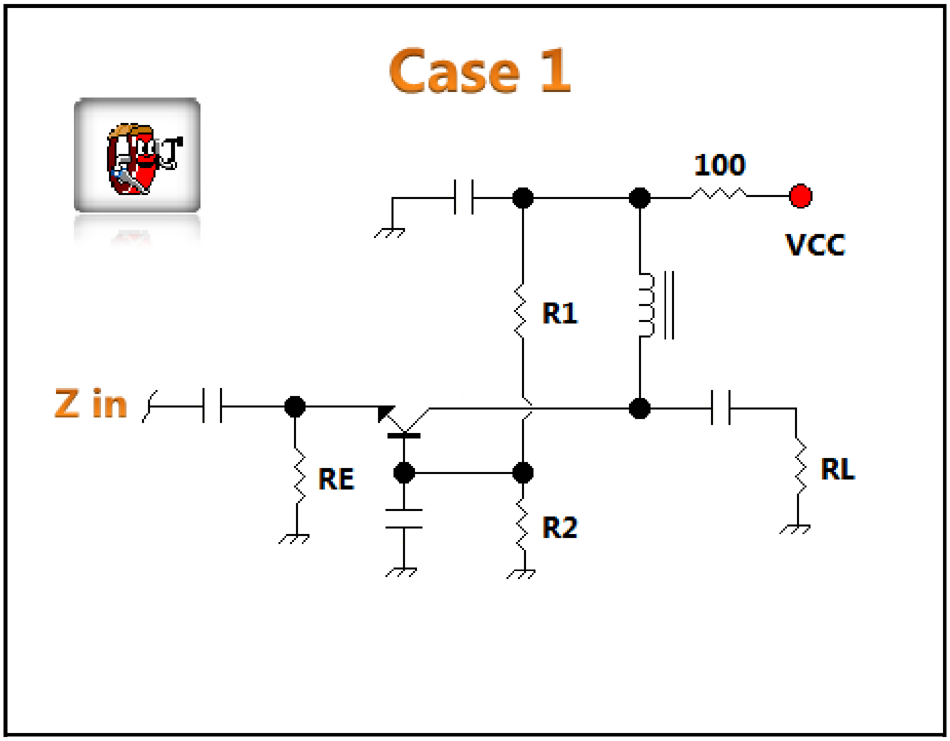
Current electronics hobbyist magazines rightfully focus on topics that are contemporary or important to their advertisers; for example, promoting mixed-signal ICs, DSP, microcontrollers and the kits they describe and then sell for income. Nuts and Volts is 1 example. Both analog RF and AF design increasingly lies in the hands of a small group of specialists, enthusiasts and students.

Yet, we *persevere*. Sharing our knowledge, circuits, experiences and references on the Internet helps sustain our small global community. That's the site purpose— sharing the (warts and all) experiments + basic information of a lay-person.

The Emitter Choke in Common Base RF Amps

This web page covers reverse isolation — a really important topic. 2 principle amplifiers we employ for strong reverse isolation are the common base BJT and common gate JFET alone or in cascode with other amplifier topologies.

Some comments regarding using a radio frequency choke in the common base amplifier follow.

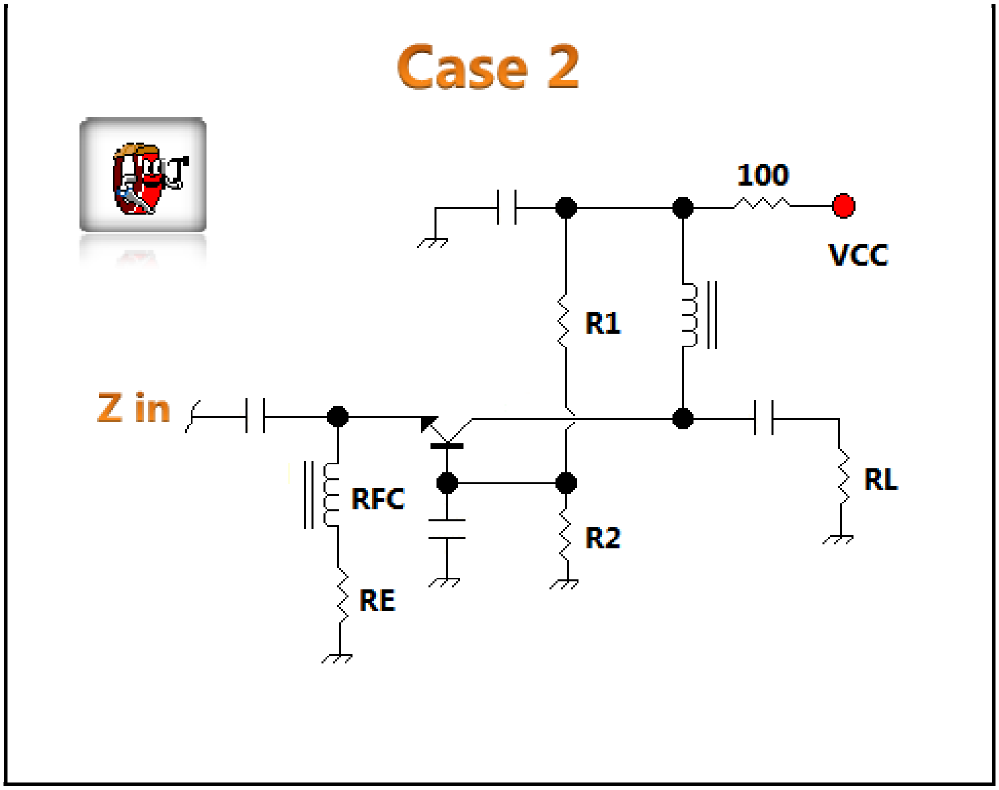


Above — Case 1: Emitter resistor only.

Apart from providing DC bias along with R1 and R2, emitter resistor RE plays another important role. Despending on its value, a portion of the input AC signal may pass through RE to ground instead of going through the transistor — degrading signal amplitude and noise figure. To minimize this, the resistor value should be many times (~10X or more) than the input impedance of the amplifier.

Although we might bias a common base amp to give an input Z of 50 Ω, often we'll choose a much lower input Z to get higher voltage gain. Input Z = 26 / Ie where Ie = mA; so if you bias for 5 mA, you are looking at an input Z of ~5 Ohms. In that case, a low value bias resistor such as 100 Ω won't shunt much of the input signal to ground, nor will it likely contribute much noise.

For most common base RF amps, a correctly chosen emitter resistor is all that's needed to decouple the AC signal and using an emitter choke proves hard to justify. However, it's important to understand how to apply an emitter choke since the basic principle also extends to the common gate JFET amplifer and other circuits.



Above — Case 2: Emitter resistor plus a choke.

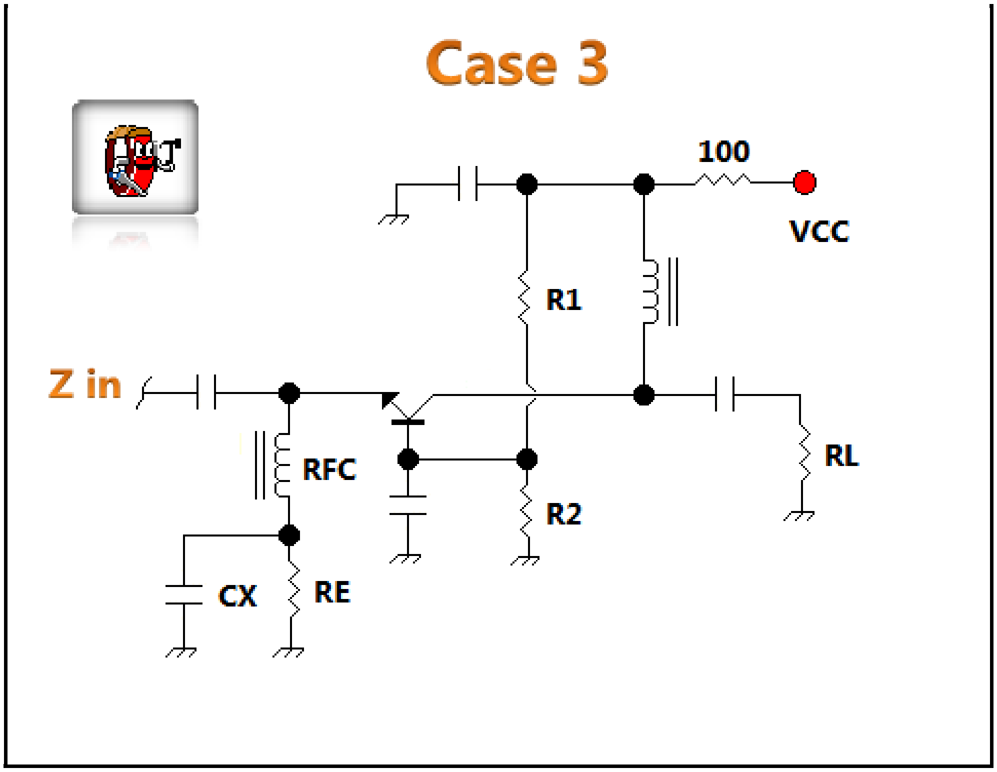
The choke's main purpose is to block or choke RF from passing to ground. The ideal choke would present infinite impedance to AC signals, plus 0 resistance to DC voltage. In reality, "ideal" = fantasy electronics and you can simply estimate a choke's inductive reactance using the classic formula (XL =2*PI * Freq * L).

Using a coil (and not just a resistor) is generally better for decoupling — although how much better might be debatable. If the inductive reactance (XL) of the coil is significantly higher than the input impedance of the transistor, then all of the input signal power goes to the transistor.

By convention, a minimum choke XL should be at least 3 times the input resistance, however, the self-resonant frequency of the coil must be significantly higher than the applied frequency. Thus, an ideal range of inductive reactance exists, and too little or too much can degrade performance. Many builders target an XL around 10 times higher than the transistor input impedance at the lowest operating frequency.

Example: For a common base amp biased for an input Z of 50 ohms, the minimum inductive reactance (XL) for the choke = 500 ohms. To calculate the inductance of an emitter choke for this amp at 50 MHz, we re-arrange the formula to solve for L.

L = XL / 2* Pi * F
L minimum = 1.59 uH.
Winding the choke on a ferrite core, or possibly a bead for VHF often means less turns, less winding capacitance and a higher self-resonant frequency.



Above — Case 3: Bypassed emitter resistor plus choke.

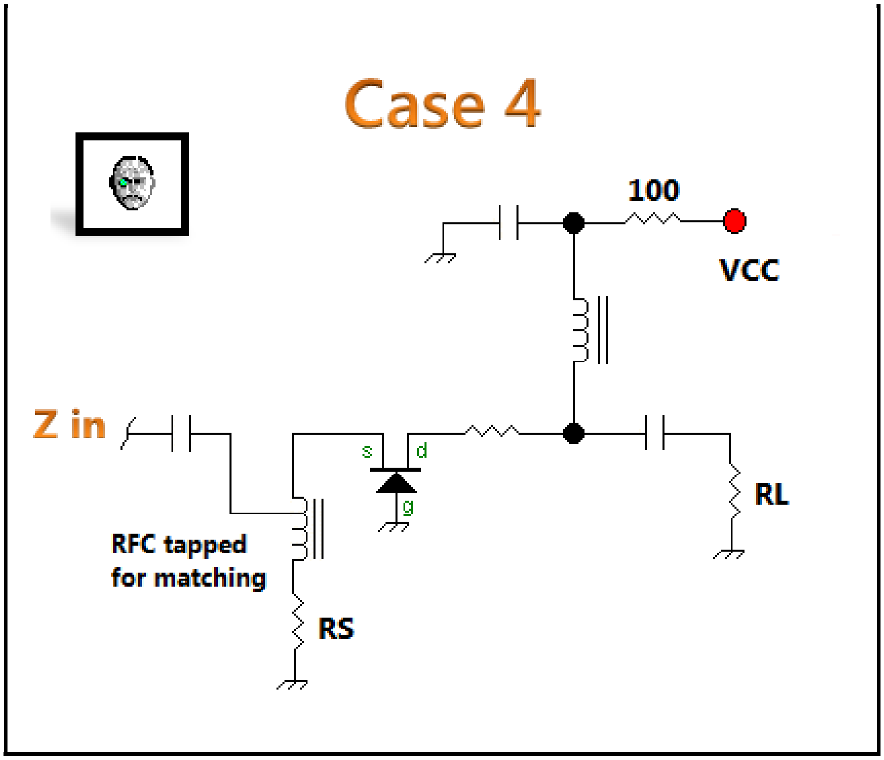
The primary purpose of the capacitor across RE is to filter resistor noise — but that is only an issue well below the frequency of interest and it should not be relevant at high frequencies where the choke reactance is significant. There may be some useful effect if the self-resonant frequency of the capacitor Cx is above the frequency of interest.

You can only use bypass capacitor **Cx** when a choke is implemented.

A 0.1 uF may be useless at high frequency. In error, I've used this value previously on the site; after 14 years of experimenting, I've learned a lot from my design mistakes.

The case of the common gate JFET amplifier

This discussion also informs common gate JFET amplifer design. The JFET source requires signal decoupling similar to the emitter of the bipolar transistor discussed above.



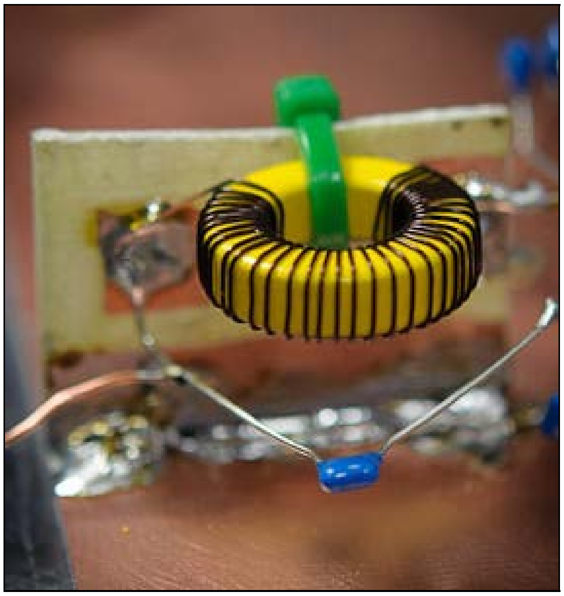
Above — Case 4: A choke plus source resistor will commonly be the "go to" design. Things get a bit more complicated with some, but not all JFET circuits — engineers often match the JFET input for a low noise figure rather than just the "correct" input impedance.

A good example follows: We might place a common gate JFET amp after a diode ring mixer because of the wideband load it presents to the mixer's RF port. The best noise match may occur with a hypothetical input Z of ~70 Ω (this argument represents an advanced topic).

After measuring the JFET pinchoff voltage and Idss, you would likely find that a source bias resistor of ~100 Ohms would be needed. This R value is so close to the JFET input Z that signal losses to ground would occur — demanding a choke for signal amplitude preservation, plus impedance and noise figure control.

RF — Test and Measurement

VFO - 2011



Building VFOs in 2011 might seem an irrelevant exercise given the move to and evolution of digital signal generators laden with bells and whistles like memories and audio or video frequency displays.

A successful L-C VFO requires skill, patience and some good parts to pull off — else, a "drift monster" may result. Despite their limitations, it's possible to build L-C VFOs with low frequency drift, distortion and phase noise; our typical VFO performance markers. L-C VFOs don't require programming skills or equipment to encode a microprocessor — making them a good choice for people who don't build or can't afford kit oscillators. Most of all, they kindle creativity, problem solving and pride when your oscillator actually works as planned. Junk box radio; my passion.

This material reflects lots of empiricism; not pure science. It's really *your* VFO design odyssey; a chance to think creatively and critically to sort out what works and what's folly.
Countless web pages discuss VFO design and I encourage you to search for and read them. Wes' EMRFD oscillator and temperature compensation notes = essential reading. Only your first 25 VFOs will prove difficult — it gets easier after that.

VFO 2011 Topics:

1. Frequency Stability Notes
2. Vackar VFO Experiments
3. HF Signal Generator
4. Miscellaneous Bits

---

1. Frequency Stability Notes

Building an oscillator that stays on frequency purports our greatest challenge and goal in L-C VFO design. Since drifting VFOs pose a source of frustration, I cover some topics that may help your VFO stay on frequency — do they help?

What is good drift parameter?

I'm uncertain, for after warm-up, I've measured kits that drifted 50-150 Hertz per hour, built L-C VFOs that drifted under 20 Hz per hour and every once and a while, build a drift monster VFO that sweeps upward at 2 - 8 hertz per minute! Likely under 20 Hertz per hour after warm up = a gold standard to compare against. You should be able to listen to a 10-20 minute QSO with no re-tuning, however, this assumes the transmitting stations are locked on frequency.

1. Unloaded Q and Frequency Stability

The number 1 reason to employ high resonator Q in oscillators is to obtain low phase noise. Secondly, the very steep phase slope through high Q resonance minimizes the effect of amplifier phase shifts caused by temperature changes and this in turn, minimizes any amplifier-induced frequency instability.

Long term frequency stability is chiefly dependent on the temperature, environmental and age stability of the resonator components regardless of Q.

I often see designs featuring high Q inductors wound on powdered iron toroids complimented with trashy, low Q variable and/or fixed capacitors, If you design for a high Q tank to minimize phase noise, consider using a high Q coil plus appropriately temperature stable, high Q capacitors.

2. Temperature Stable Inductors

Knowing that I'm venturing into a topic of great debate and lore, the inductor is 1/2 of the VFO resonator and thus a major source of temperature drift in L-C VFOs. Since MF and some HF VFO designs *may* preclude using the *inherently more temperature stable air wound inductor*, powdered iron toroids dominate our evermore compact designs. Many builders choose #6 material, although the lower temperature coefficient of #7 material theoretically should be better — however, my experiments have failed to measure a significant difference between these 2.

Some builders prefer size 68 inductors, for the bigger core is less affected by heating than smaller size toroids. My experience suggests that providing the VFO amplifier current is kept low, both size 50 and 68 are both suitable and the inductance needed should inform the core size.

I used to think that heavier gauge wire created greater frequency stability than smaller gauge wire until Wes, W7ZOI, woke me up. As it turns out, smaller gauge wire is often better for thermal stability because smaller gauge wire lies closer against the toroid core. Winding stiffer, heavier gauge wire creates more air gaps than smaller gauge wire and air gaps expand and contract during temperature changes. Smaller gauge wire will have a reduced Q, but it won't be as significantly lower as you might guess. As possible, I prefer tightly wound number 28 wire. 26 gauge wire tends to be my maximum size wire for VFO coils, however I suggest you make your own conclusions.

Wash your hands before winding and use both hands to actively move both the toroid and wire for tight turns. Take your time, ensure steady wire pressure and avoid kinking your wire. Taps increase the likelihood for air gaps — mitigate this by stripping the 2 tap forming wires as close to the toroid as possible and twist them into 1 wire right down tightly to the toroid edge to reduce any air gap.

The thermal stability characteristics of wire can be mitigated somewhat by annealing the wire with temperature cycling or by dunking it in boiling water. Roy, W7EL first reported annealing coils in 1980 and this has been confirmed during experiments by builders using temperature controlled ovens. I don't boil my coils any more.

3. Double Stacked Toroids

I noticed a new trend in VFO design is to stack 2 powdered iron toroid inductors. This allows the builder to double the inductance per number of windings over a single toroidal inductor. In an L-C VFO, the goal of these builders possibly is to reduce heating effects, increase unloaded Q, or perhaps to reduce core magnet flux density. For me the goal is far simpler, I just want to make compact, large L value inductors for 3 MHz and less.

Above — A T68-6 hamburger. The two T68-6 cores were epoxy glued together and compressed lightly in a vice for several hours. One of the initial tests I performed was to see if boiling the stacked coil affected the epoxy glue. The glue was not effected by annealing wire on a stacked coil with 5 or even 10 minutes of boiling in water. As mentioned, I stopped boiling my VFO inductors as tightly winding them with 26 gauge wire seems to work well.

I hold concern that stacked toroids may create more wire-air gaps when compared to a single toroid and stay with 1 toroid as possible. In compact antenna tuners and other non VFO projects, this isn't an issue.

4. VFO Tank Capacitors

We choose VFO tank capacitors to avoid temperature change caused frequency drift, or to counter drift during our temperature compensation process.

Many authors have published guidelines for long term temperature stability. It's important to consider these guidelines, but also try whatever works. I believe the following arguments are accurate based upon my experiments:

1. Multiple NP0 or C0G (0 temperature-compensation) tank caps: Most builders minimally use 4 or more C0G or NPO capacitors to reduce heating effects and to average out temperature coefficient variations.

2. No VFO tank capacitors from online surplus parts stores; buy new stock from known and reputable manufacturers. Grab bags and musty, old, surplus parts can obviate good design.

3. Trimmer and tuning caps need to be temperature stable. Air variable capacitors = my favorite, as possible.

4. Varactor, or diode tuning generally = more drift and a greater need for temperature compensation.

5. Employ short, stiff capacitor leads. I use 100 volt or higher voltage C0G tank caps as they tend to have thicker leads that stay put — perfect for Ugly, Manhattan, or Chuck Adam's MUPPET construction.

5. Temperature Compensation

The goal of temperature compensation is to cancel the tendency of the VFO to drift in 1 direction — easier said than done + very time consuming. A web search for VFO temperature compensation will yield many good write-ups. I feel it's partly art, partly luck and partly science. Your net VFO temperature coefficient can be affected by so many variables, so no 1 recipe will ensure a low drift VFO. Experiment, allow a lot of time to assess your changes and be patient — you'll figure it out.

The simplest way to test for drift involves watching a frequency counter, but if you don't have one, you might use a commercial, frequency stable (synthesized) receiver set in the SSB/CW mode. I use both. Experienced builders often employ an oven to test their temperature compensation at different, controlled temperatures. Wes, W7ZOI employs a styrofoam cooler housing a light bulb heat source controlled by a Variac. See EMRFD for more details and a photograph.

In 2011, I decided to build up a supply of temperature compensation capacitors and keep them in their own parts bin.

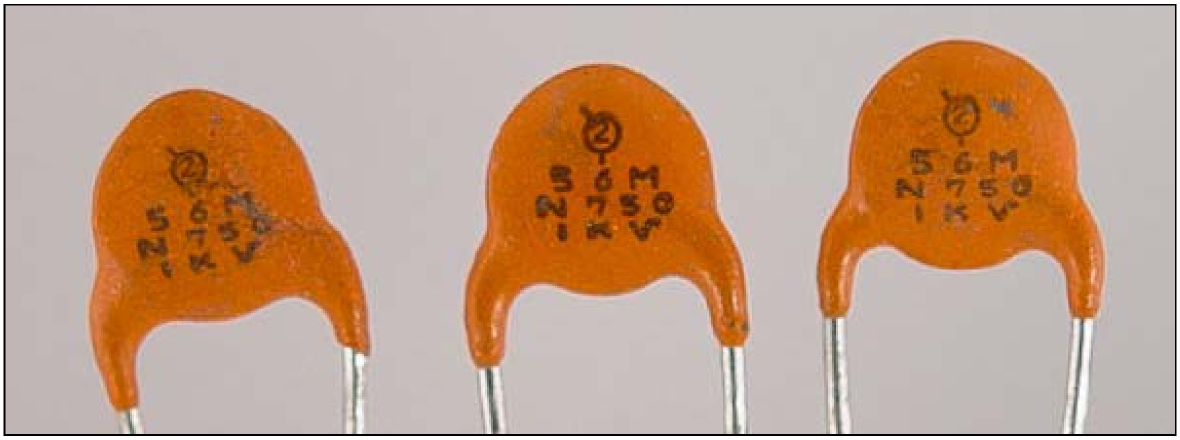
Above — "Tempco caps". A parts drawer containing polystyrene capacitors from 10 to 270 pF plus some 56 pF ceramic N750 capacitors for negative temperature compensation. I purchased these capacitors on eBay.

For capacitors other than NP0 (which use 0 instead of a ppm value), the temperature coefficient = **P** for positive and **N** for negative, followed by a 3-digit value specifying ppm/°C. For example, N220 is - 200 ppm/°C. and P100 is +100 ppm/°C.

I use NP0 and C0G ceramic capacitors interchangeably for both tuning and RF bypassing the VFO tank resonator. For C0G/NP0 *temperature compensation bypass*, I normally apply 0.01 or 0.001 μF caps, however, the more expensive 0.1 μF COG ceramic capacitors are still sold if you need C0G/NP0 bypass <= 7 MHz.

If your VFO is drifting upward you might insert 1 or more positive coefficient capacitor(s). If your VFO drifts downward, then try using negative coefficient value(s). Sometimes just 1 capacitor will do the job.

Since I don't stock any positive coefficient capacitors for positive coefficient compensation, I might try a adding a silver mica capacitor. ***Caution*** silver mica capacitors are extremely non-predictable and can't be universally recommended in temperature compensation schemes. You might also try swapping out 1 or more of your main tank NP0 or C0G capacitors in case they are bad; sometimes it gets frustrating. I provide some temperature compensation examples on the QRP Modules 2011 web page in the 7 MHz VCO section.



Above — 56 pF N750 ceramic capacitors rated at 1KV

6. Mechanical Rigidity

Movement of your VFO tank parts may lead to frequency instability. For example,

1. Well secure your single-sided only copper board. I use at least 4 number 8 bolts — 4-40 hardware is too light. Boards can warp over time if not lashed down properly. Aggressively bolt down any variable capacitors. No tank parts should move.

2. Anchor your inductor so it cannot budge: nylon bolts, zap-straps, glue - whatever.

3. Consider placing the VFO in a strong chassis with rubber feet.

4. Buss wires should be made from thicker gauge, well anchored wire.

7. Miscellaneous Points

1. Regulate the VFO amplifier DC voltage and wideband filter it. Voltage regulators require RF and often AF bypass to attenuate any noise or ripple riding on the DC.

> A decoupling resistor with a bypass cap on either side will widen your DC supply filtering bandwidth and deserves strong consideration. A poorly filtered DC supply can easily transmit the VFO tank energy to other stages along your DC lines and also may allow noise on the DC supply to modulate your VFO and increase phase noise.

2. You should have the buffer + a load resistor connected to your VFO when testing. **Do your temperature stability work after the buffer is built and the VFO is in its case.**

3.Stick your VFOs in an air tight, RF tight case to minimize air temperature changes and RF leakage respectively. Sometimes a VFO will drift once in a case because any radiated buffer amplifier heat will warm up the inside of the chassis. This usually levels off after warm up.

4. Modern voltage regulators may significantly reduce noise compared to a zener diode regulator. Specific low noise and low temperature coefficient voltage regulators are available, but maybe overkill for you. Whatever you use — filter it well —
The Micrel MIC5209-5.0BS in SOT223 sits in a couple of my reference oscillators.

5. JFET Gate clamping diodes may increase phase noise, but not prohibitively so in most popcorn designs.

6. When winding toroid inductors, wind 2 extra turns. When finished, unwind the first 2 turns since they are usually loosely wound and prime culprits for air gaps.

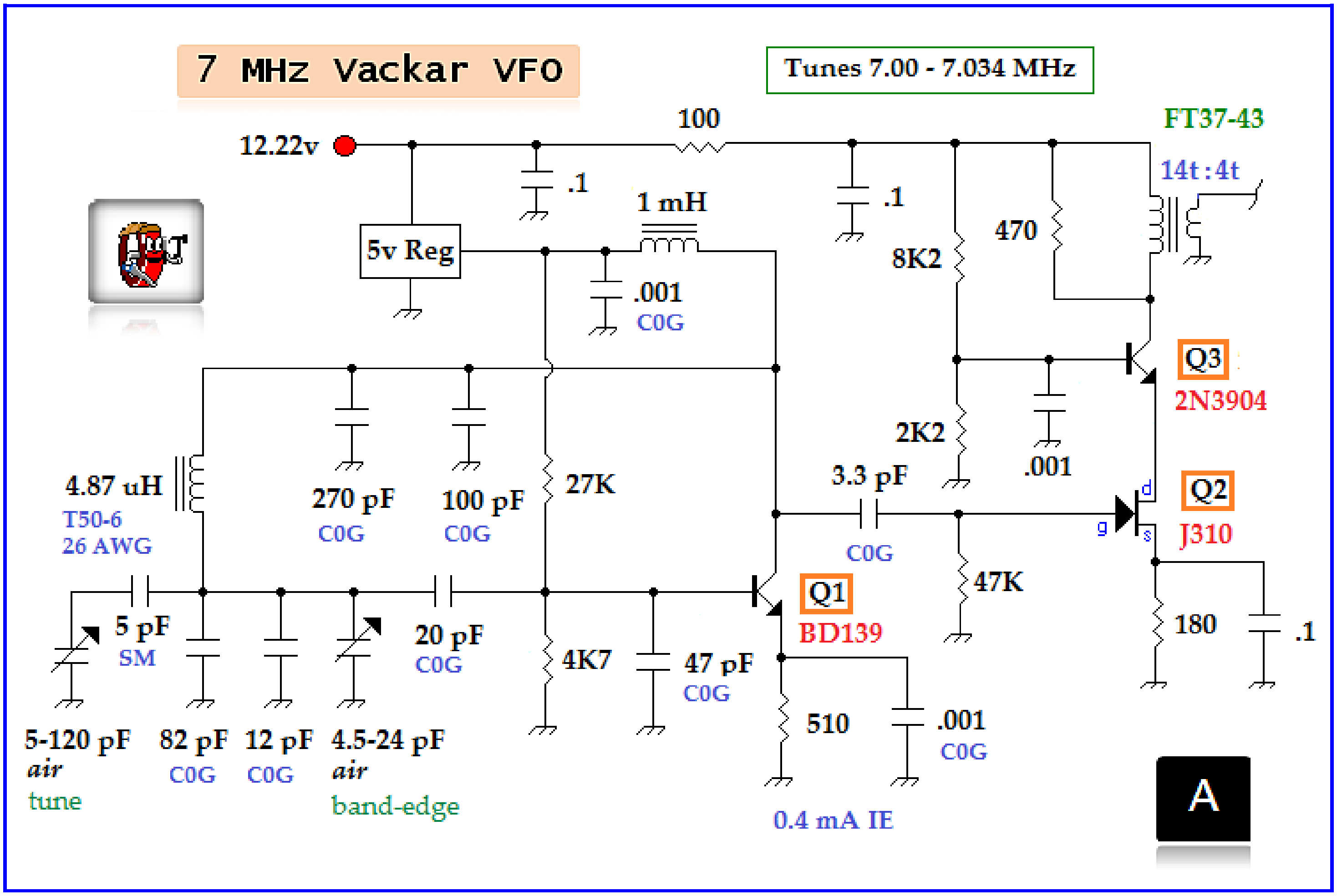
7. Since magnet wire comes off small spools, wire has a natural curve or radius — ensure you wind your coils according to the natural curve of the wire.

8. The need to secure powdered iron windings with dope, wax, goop, etc. is over-emphasized and usually unnecessary.

---

2. Vackar VFO Experiments

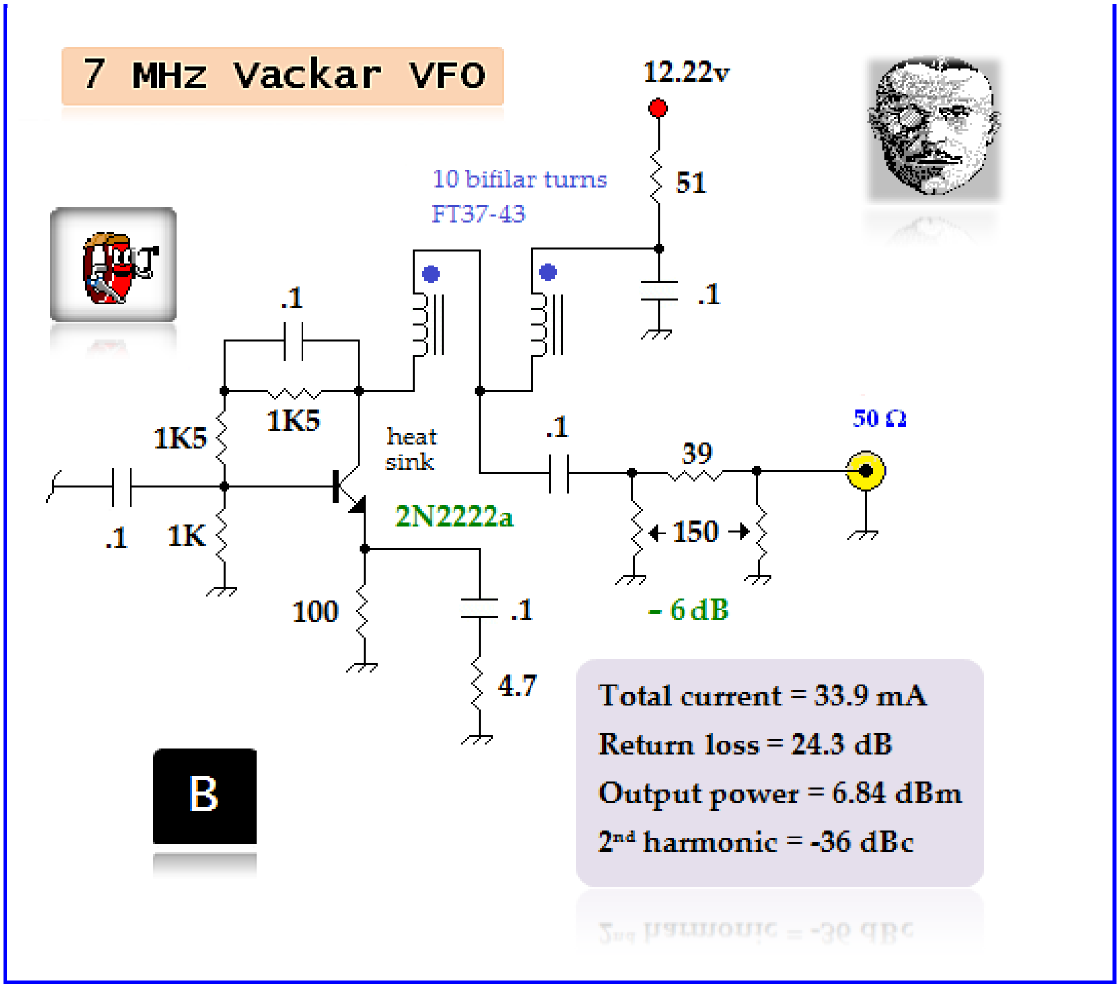
Some builders proclaim the Vackar as the "King of VFOs". I built a couple and became impressed by the low distortion and less than 5 Hertz per hour long-term drift achieved in my 2 designs. Inspired by work from Iulian, YO3DAC entitled Very Low Phase Noise Vackar VFO for HF Transceivers *(link and reference used by permission of Iulian)*, I crafted my version from his notes and schematic.



Above — Schematic of the Vackar VFO employing a BD139 — a large area transistor, to reduce 1/f noise. Iulian shared many design pearls in his paper and I won't repeat them. I ran Q1 with 0.4 mA emitter current to reduce heating and flicker noise. It's difficult to measure flicker noise, so no objective comments can be made.

I limited the tuning range to 34 KHz since the tuning capacitor lacked reduction gear and I was born with fumble fingers. As a CW operator - you'll find me down at the bottom of the band away from the RRTY anyhow. Increasing the 5 pF cap coupling the tuning capacitor to the tank increases the tuning range as expected.

Temperature compensating my VFO with the 5 pF silver mica capacitor proved a gamble since SM caps are unpredictable and often best avoided. In *my* VFO, it worked perfectly, however. This circuit is difficult to replicate and *not recommended* for new builders. Temperature compensation provided the sublime frequency stability.



Above — The final amp and measured output data. Measuring the 2nd harmonic 36 dB down without any tuned circuit or low-pass filter rocked. I ran nearly 22 mA of emitter current to bump up the return loss and spectral purity. A 2N5109 or 2N3866 would likely do a better job with less current. Total current = the entire VFO current. I glued a drilled copper penny on the 2N2222 to dissipate heat.

A photo of my version of a Vackar VFO. My design goals included low phase noise, low distortion, a return loss over 20 dB, good reverse isolation and ~7 dBm output power. I believe all VFOs are experimental; you build to suit whatever tuning capacitor or varactors you have, plus design around constraints such as total current, tuning range and other personal criteria.

Unlike harmonic distortion, oscillator phase noise, being close to the oscillation frequency, cannot be removed by filtering nor limiting — you must design for low phase noise. Modern digital VFOs are well harmonically filtered, and any phase noise depends on the DDS clock employed, so check the DDS specifications carefully if you go the DDS VFO route.

I'll be the first to state I'm no expert with VFOs, however, likely the only way to become expert is to build many and learn from your mistakes.

Above — A 7 MHz Vackar VFO with the lid off

Sound Test?

Although this technique raises the ire of some builders, I test my VFOs in a nearby receiver. The VFO output was terminated with a 51 ohm resistor that was also attached to my frequency counter via alligator clips and wire. I tuned a nearby CW superheterodyne receiver to 7.00 MHz with the audio beat note centered in its 600 hertz wide I.F. filter and watched the counter plus listened to the receiver.

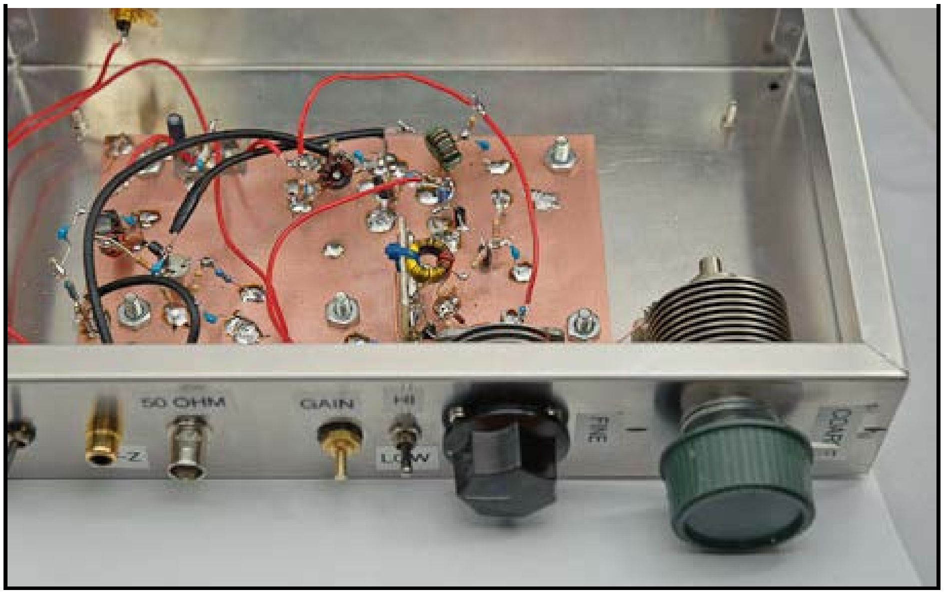
Click for a 1 minute 32 second audio file of the result (it stayed perfectly on frequency for ~5 hours of testing before I got a bad headache from listening to it and shut it off). *You can initially hear a station in the back ground despite only having a 45 cm piece of wire as the receiver antenna.* The VFO slowly drifts down to 6999996 Hz and then slowly back up to 7000000 Hz. You can hear the signal amplitude decrease as the VFO drifts down. So it doesn't stay perfectly on frequency, but slowly cycles up and down a few hertz. This VFO is my lab temperature stability benchmark for an L-C VFO.

A badly drifting VFO will move out of the test receiver I.F. pass band and sound like a Theremin as it does. Testing in a receiver; places the VFO in the exact circumstance it will be used — beating RF to mix to another frequency; in this case, base band audio.

The temperature stability and compensation of any VFO schematic are rarely reproducible since there are just too many variables. Try your best to get the drift out of your VFO using low temperature coefficient capacitors (NP0/C0G) and then after that, temperature compensate. Even today, I occasionally build a drift monster VFO and become frustrated. VFO design is not for the faint of heart and it's no wonder that many builders make a VXO, or cave in and build or buy a DDS signal generator.

---

3. HF Signal Generator



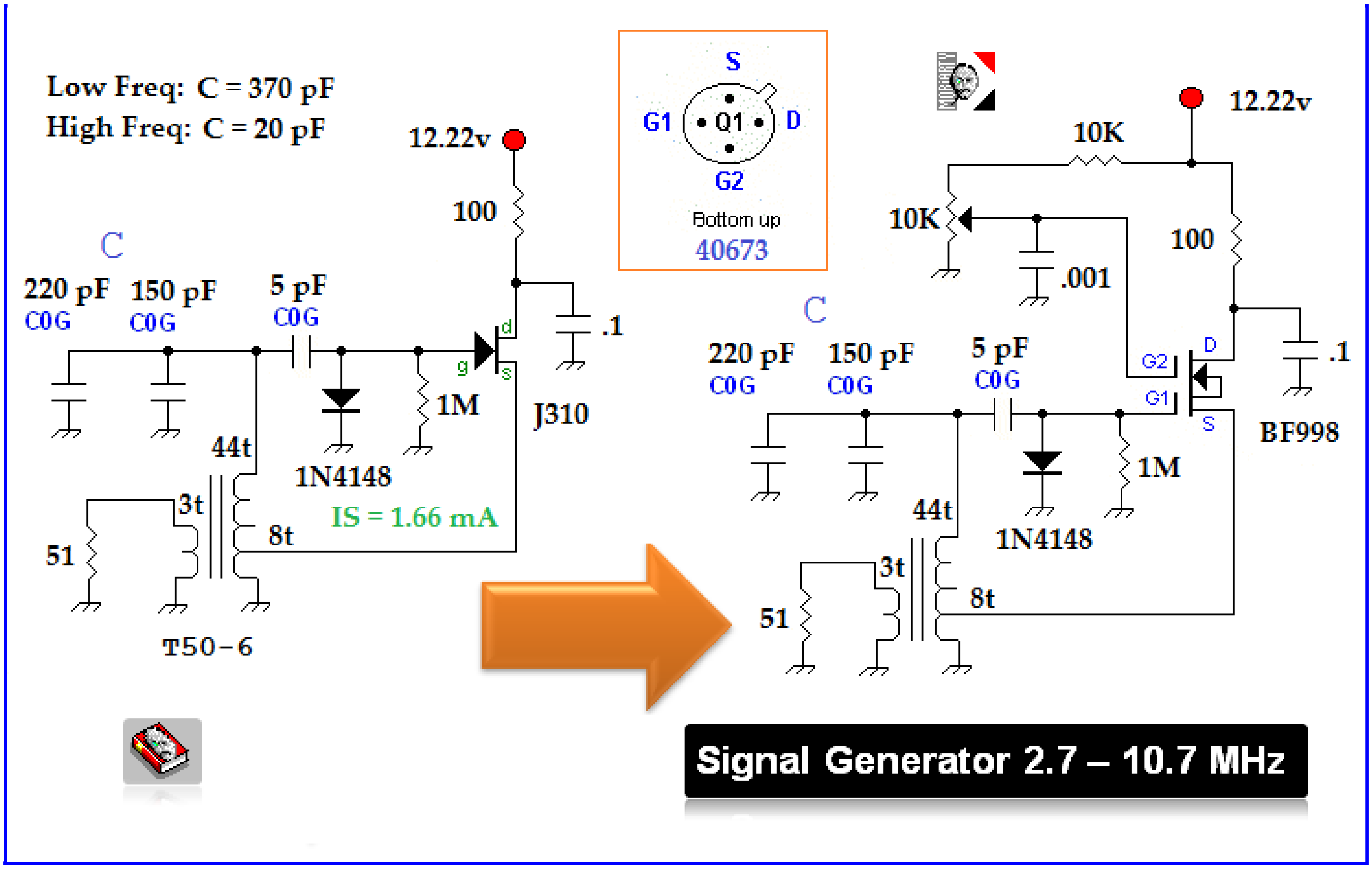
Above — I built a general purpose ~2.8 to 10.8 MHz signal generator (SG) for my lab in 2011. The first VFO topology tested was the Vackar. In my version, while employing a 100K ohm resistor as the buffer, the VFO only tuned from about 4 to 8 MHz and suffered from *extreme* amplitude variation as I changed the frequency across its range. For sweeping filters or measuring Q, a signal leveling circuit would be needed as normally we like our SG output to be flat across its frequency range. I later changed to a Hartley VFO because of its flatter output and the wider available frequency range with any given resonator.

This initial Vackar VFO experiment wasn't a total waste as I learned a way to accurately sweep a **D**evice **U**nder **T**est with an unlevel amplitude SG. Measure the peak-peak voltages of the DUT with a signal generator and an oscilloscope in the same manner we measure insertion loss or gain in a 50 ohm system: Measure the peak-to-peak voltage with the DUT in line; disconnect the DUT, insert a barrel connector and then re-measure.

The dBm difference between the 2 becomes the dB value to plot for *that* frequency. To a sweep a filter, say for example, a band-pass filter, find the center frequency and then sweep below and above that CF while plotting the dB versus frequency.
This "in and thorough" measurement described takes time, but resolves any frequency versus amplitude issues and can be used to test signal generators. We tend to ignore things like cable loss versus frequency and scope or spectrum analyzer ripple.

Still, it will be easier to just use a Hartley VFO where our sweeps are assumed to be level due to the flatness of the amplitude versus frequency for **small excursions** such as 3 dB band-pass filter sweeps. Do not expect amplitude flatness over wide excursions however — this requires additional circuitry.

To reduce noise and boost fidelity, this SG runs modest current and was not designed for battery use.



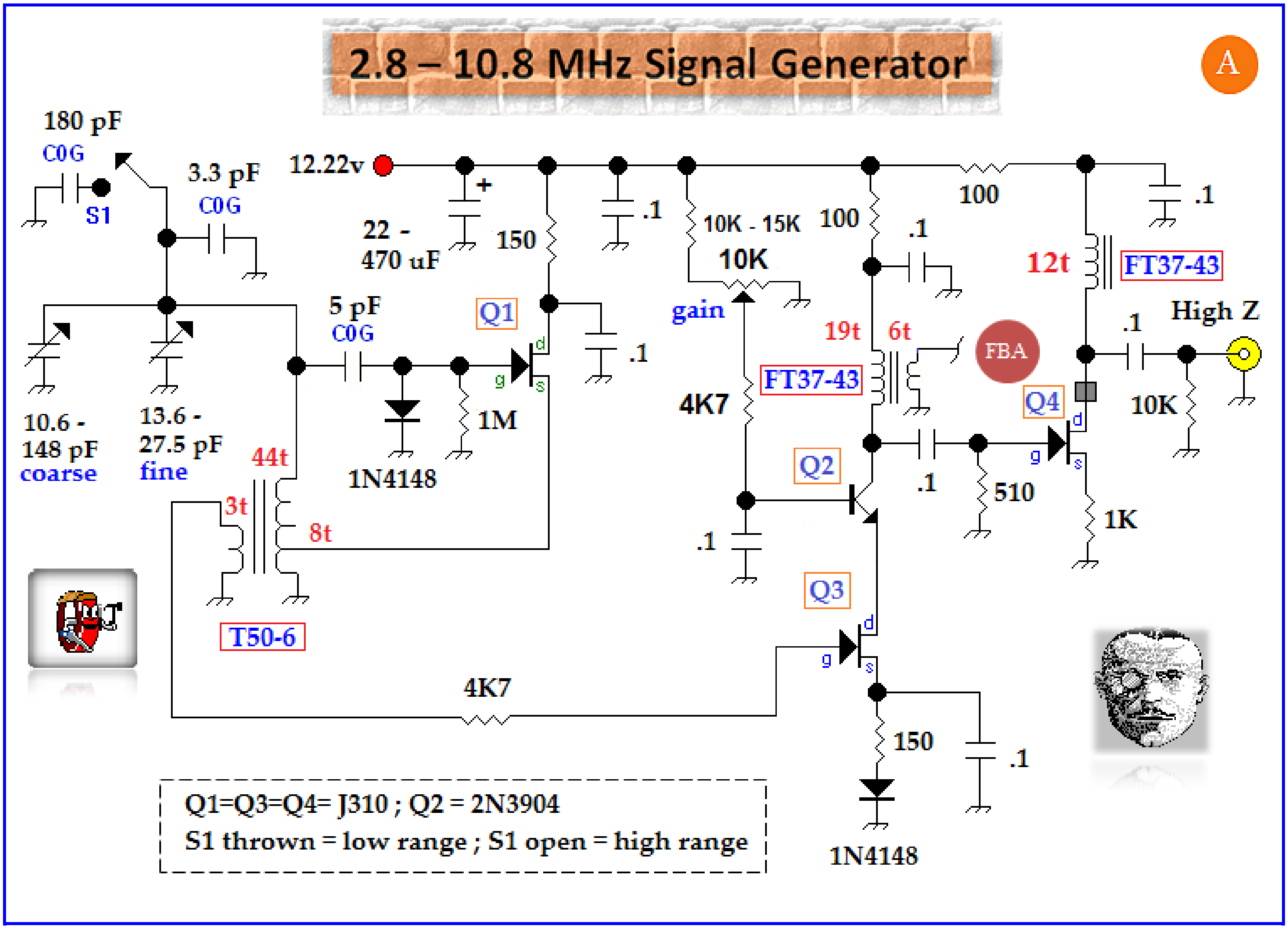
Above — My Hartley VFO is morphed into a double-gate MOSFET VFO; this was a mistake and I make lots of them.

When venturing out, it's often best to confirm a proven design is working before morphing it to something untried. Shown above left is the Hartley oscillator from EMRFD Chapter 7 sans buffer. Fixed "tuning" capacitors; either 20 pF (not shown) or 370 pF (150 pF + 220 pF) represent the intended high and low frequency swing of my air variable tuning capacitors.

I wanted a variable amplitude VFO and thus replaced the JFET with a double-gate MOSFET using a simple variable voltage divider to control gate 2. I showed this to Wes, W7ZOI and he informed me that the flicker noise of MOSFETs precludes their use in oscillators. I have always wondered why I've never seen MOSFET VFOs in any radio literature.

Above — My project chassis fitted with hardware. I employed 2 air variable tuning capacitors — the fine tuning capacitor ranges 13.6 to 27.5 pF, features built in 6:1 reduction gear and was purchased from Doug DeMaw many years ago. I secured the main copper board with 6 number 8 bolts. Rubber feet provide a stable, shock resistant base for the sheet metal box.

Final Build

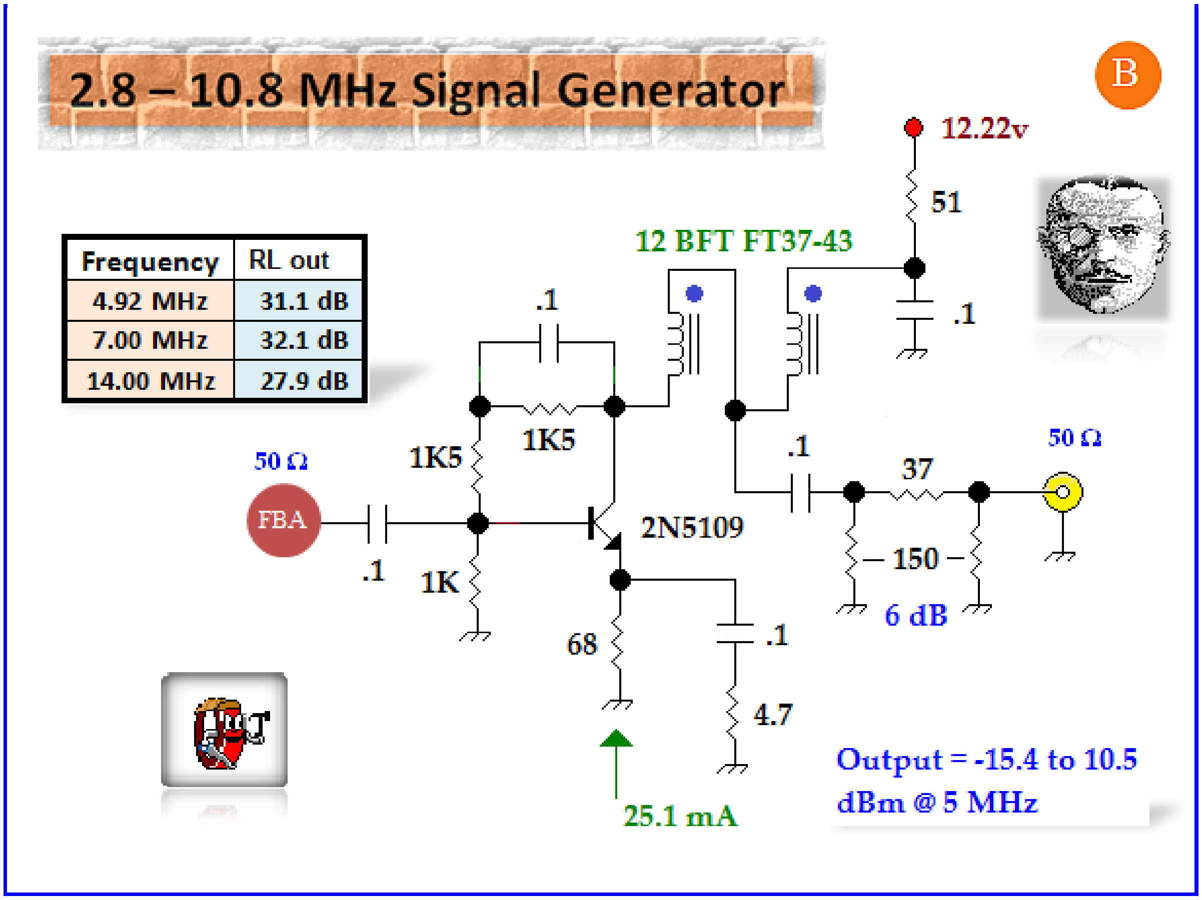


Above — Oscillator + buffer schematics of the latest version of my signal generator. I spent 1 evening playing with VFO designs and settled on the simple Hartley from EMRFD, Figure 7 .27. The 3 turn link provided lower distortion than coupling the oscillator to its buffer by the JFET source or gate.

Regulated 12.2 VDC powers the oscillator; avoiding the typical 5-9 VDC voltage regulator we normally use. A 22 to 470 μF cap should be employed to filter any voltage regulator noise from our DC supply. Mine has a 470 μF capacitor.

1 hour drift lies under 40 Hertz when averaged from 15 different frequency points between minimum and maximum. The Q2/Q3, Q4 and Q5 transformer inductances were optimized to allow good signal and/or matching performance in the ~ 2.5 to 10.8 MHz frequency range.

A hybrid cascode (hycas) buffer with variable base bias on Q2 forms the amplitude control for both the high impedance and low impedance outputs. The 510 ohm gate resistance on Q4 terminates the hycas amplifier and sets up a known output impedance to drive the 50 ohm feedback amp. I measured a greater than 22 dB return loss on the output of the 19:6 turn transformer from 4 to 14 MHz — indicating it drives the feedback amp reasonably well.



Frequency	RL out
4.92 MHz	31.1 dB
7.00 MHz	32.1 dB
14.00 MHz	27.9 dB

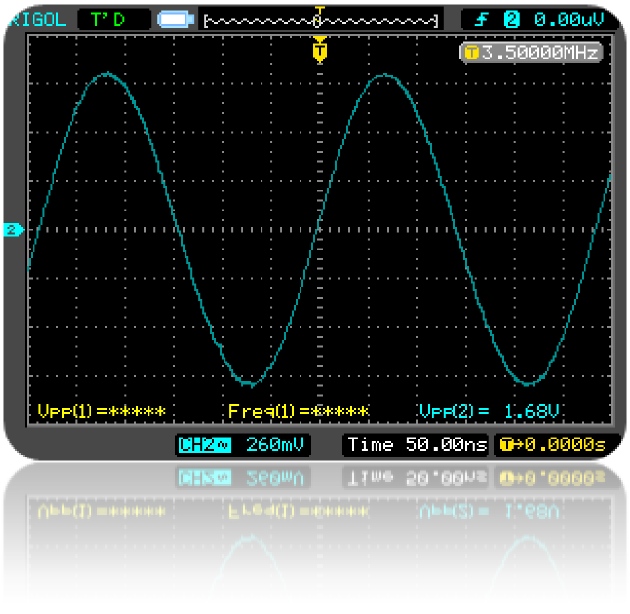
Above — The 50 ohm impedance feedback amp. Running 25.1 mA current allowed a clean sine wave output up to 2.12 volts-peak to peak into a 50 Ω terminated oscilloscope, plus an output return loss of over 30 dB across the SG tuning range. 3 tabled output return loss measurements are shown; including an out-of-range 14 MHz measurement.

Two series resistors made up the 37 ohm "resistor" depicted in the 6 dB pad, although a 39 Ω resistor would work fine.

At low output amplitudes, I typically stick an external 6, 10 or 20 dB attenuator on the output since the hycas amp can distort the signal a little when the gain control is set to a really low bias voltage on Q2. Then I fine tune the output power with the gain control.

Above — A "lid off" front panel photograph. Click or click for other photos. I'm now using miniature pots with a shaft diameter of 3.18mm. The potentiometer shaft lacks a knob and I'll purchase some on my next parts order.

Above — The completed signal generator.



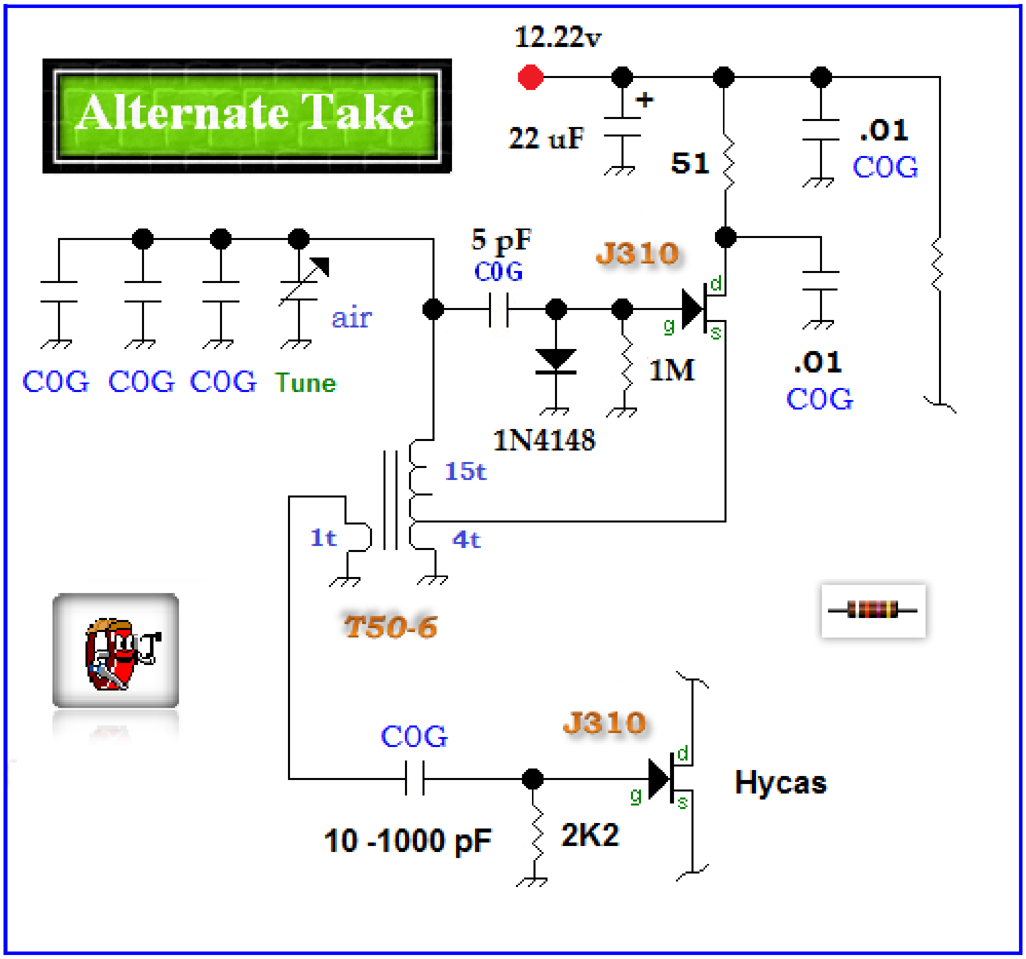
Above — Signal generator output at 3.5 MHz.

I appreciate that the VFO tank would be difficult to replicate since the 2 air variable capacitors are unique, however this is true of most VFOs. Wes wrote some great notes in EMRFD Chapter 7 regarding copying signal generators and the versatility of the Hartley VFO. I hope this project furnishes some ideas that spawn you to build something better than I did. *(I've received over 350 related emails since posting this page in 2011* *and many readers have built really great VFOs — Way To Go !)*

QRP — PosData for December 17, 2013

I slightly boosted the tuning range from 2.8 to 10.8 MHz by dropping the 5 pF resonator capacitor to 3.3 pF in October 2013. The schematics now reflect this changes plus clarify 1-2 stumbling blocks readers had. For example: my use of a 100K gain pot plus a 150K maximum voltage limiting for the Q2 bias. My build still has these, however, I altered the schematic to show a common 10K gain pot, plus a fixed 10 to 15K resistor used to limit the Q2 bias to between 5 and 6 VDC maximum.

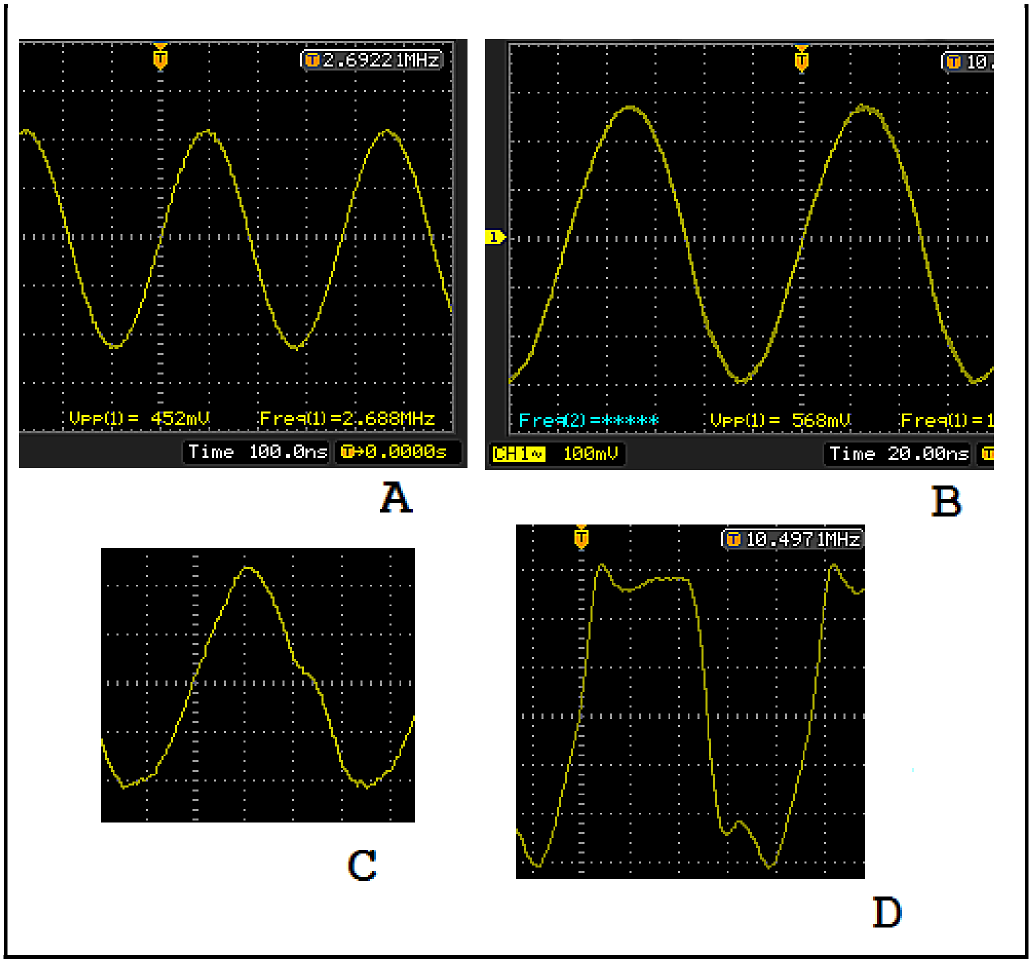
In reality, any reasonable pot and resistor will do since they function as simple voltage dividers. With a 12 volt supply, we don't want to drive the Q2 bias with more than ~ 5 to 6 VDC since this will just distort the AC signal as the hycas stage saturates. Measuring with a voltmeter, solder either a 10K, 12K, or 15K resistor to limit the maximum Q2 bias with the 10K potentiometer turned fully clockwise.



Above — Alternate way to couple the Hartley oscillator to the hycas buffer. Ground the JFET gate with a shunt resistor and lightly AC couple the JFET gate to the Hartley secondary coil with a series capacitor. You choose the capacitor value to limit the signal amplitude as needed. Our main goals are to lightly couple the Hartley tank to its buffer and avoid overdriving the hycas stage. Numerous examples of this "Alternate Take" circuit may be found on the QRP / SWL HomeBuilder web site.

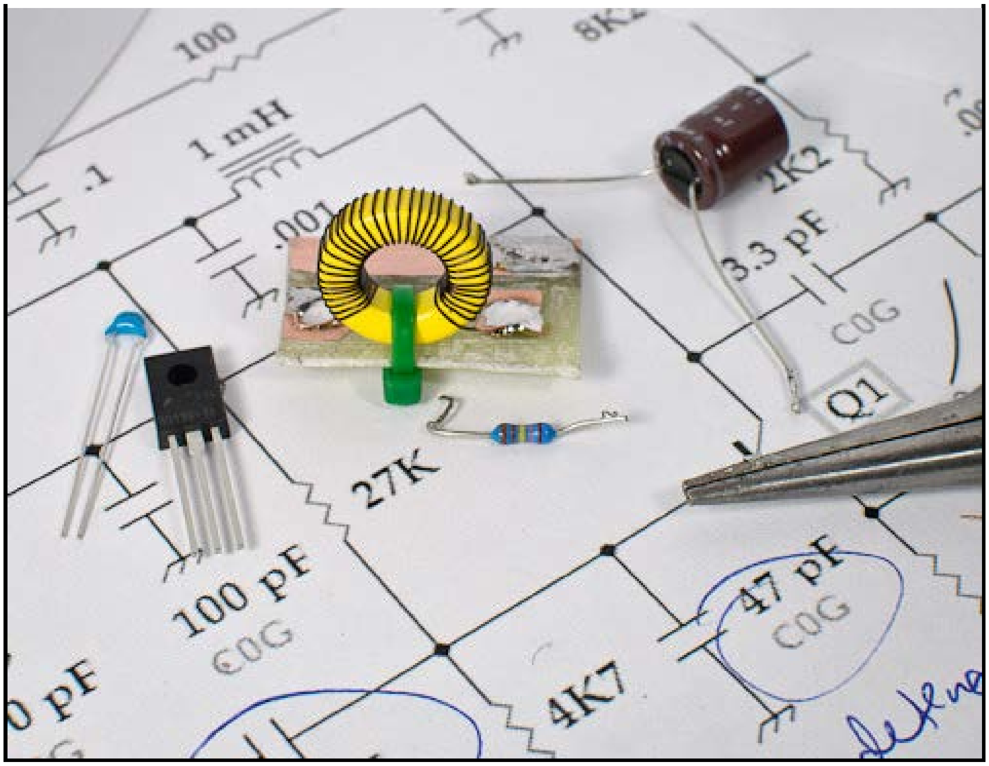
---

4. Miscellaneous Bits

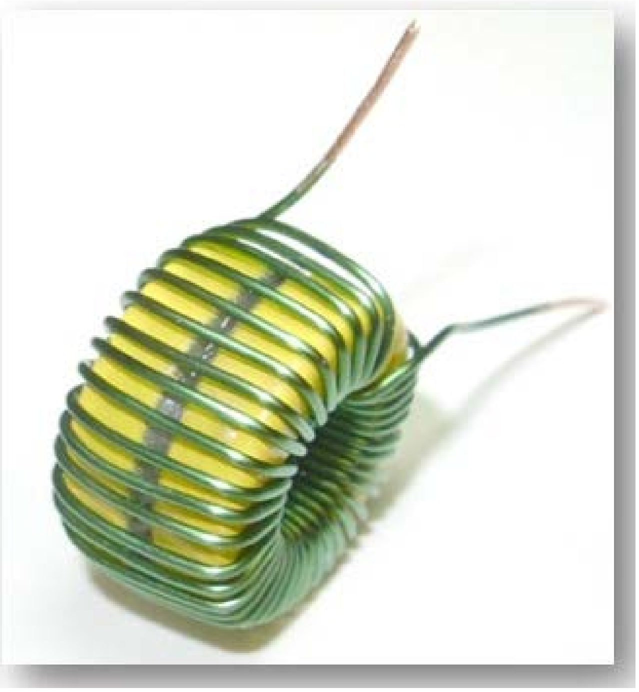


Above — Scope traces of an unbuffered Hartley oscillator with a X10 scope probe across a 51 ohm resistor across the 3 turn link. The unbuffered Hartley sine wave isn't harmonic free, but cleans up when properly buffered with a higher impedance amplifier. Figure A = the lowest frequency (2.7 MHz) — the distortion increased with frequency (Figure B was measured at 10.5 MHz). Figure C illustrates how slightly stronger output coupling with a 6 turn link trashes the output waveform — the strategy of using 2-3 links over the center of the main inductor works well.

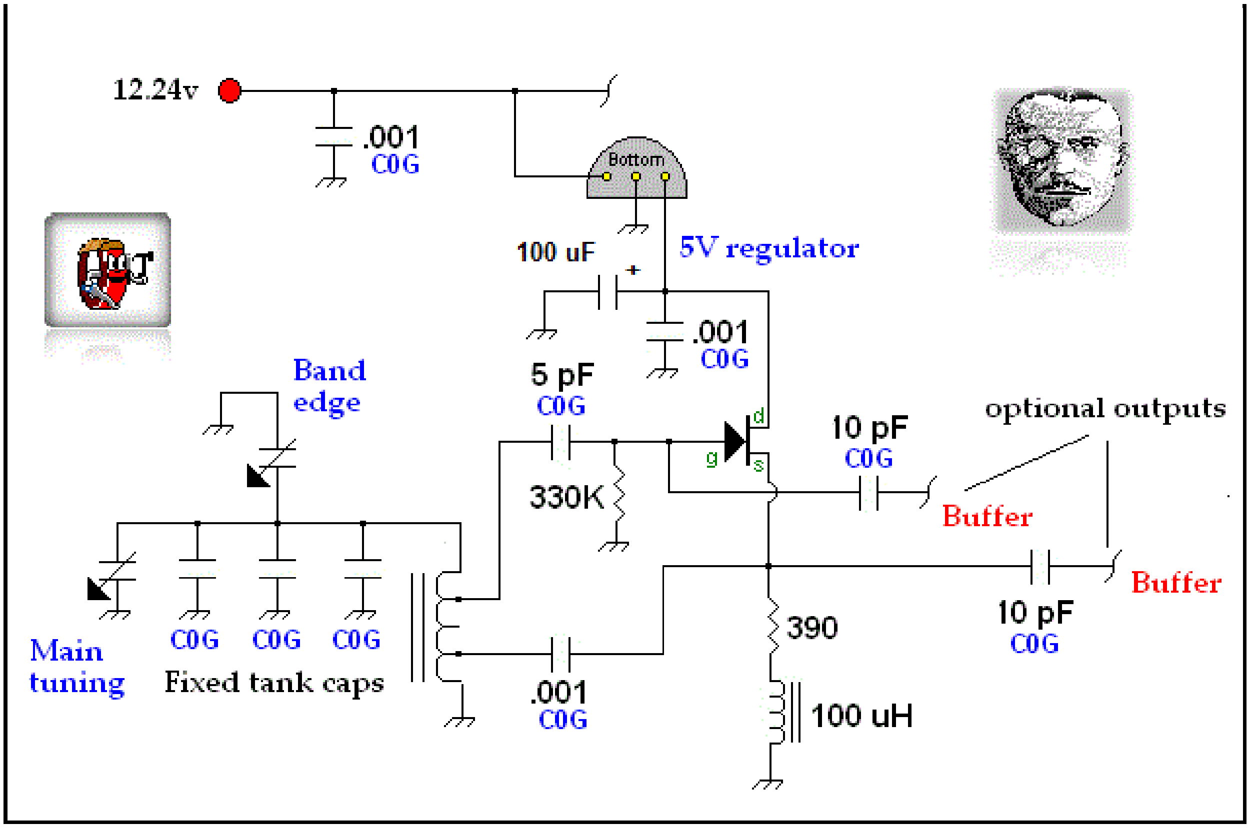
Figure D is the 6 link coupled Figure C oscillator with the gate clamping diode removed; yikes! I spent 4 hours studying what different current, voltages, coupling and so forth do to the Harley oscillator. I recommend the Hartley topology because it's simple, always starts and versatile.



Above — In my various signal generator experiments, I zap strapped the toroid to a small piece of thick copper board that was later soldered to the main board. The L seems robustly secured.

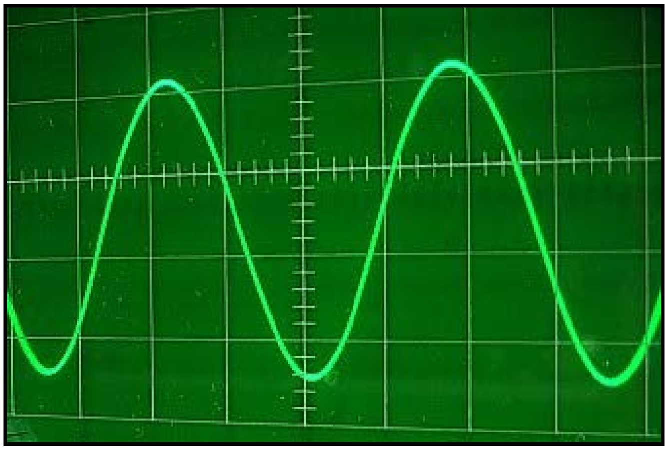


Above — A stacked toroid from the deleted VFO-2008 web page. I incorporated some of the information from the VFO-2008 page into this web page

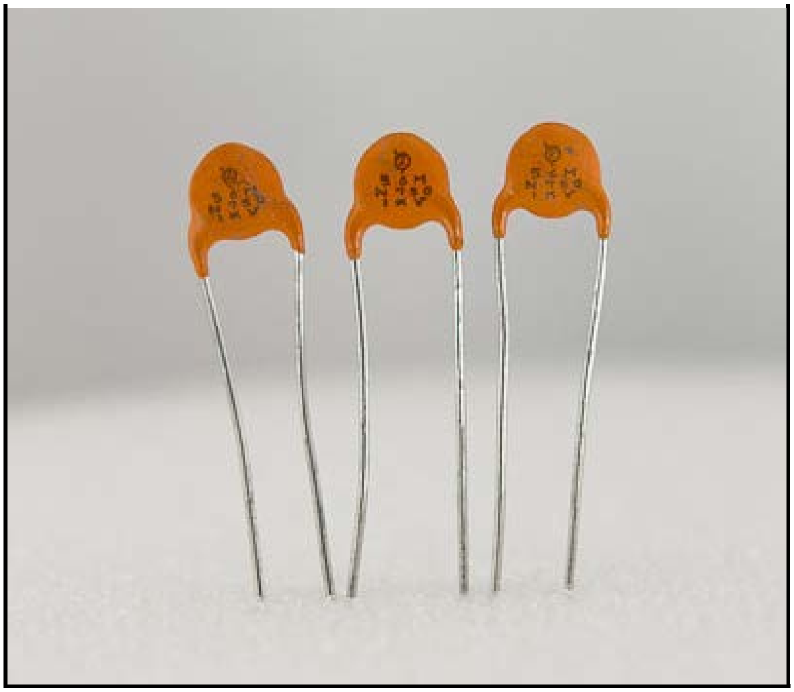


Above — In order of preference, 3 ways to couple a Hartley oscillator to its buffer. From now on, I'll couple with a 1-3 turn link since it gave a lower distortion signal than with source or gate coupling. This figure omits the gate clamping diode seen earlier— tapping the inductor as shown keepings the FET gate AC voltage at a reasonable level when not using a gate clamping diode. Some builders leave off the gate clamp diode that clips positive signal peaks for lower phase noise. The diode acts as an AGC and offers benefit. Reverse biasing this diode was suggested by Dr. Ulrich Rohde: see — Key Components of Modern Receiver Design - Part 2: Dr. Ulrich Rohde, KA2WEU , QST for June 1994.

A formula to use for the inductor taps: Divide the total turns by 1.45 to get the first tap and by 7.25 to get the second tap (near ground link).

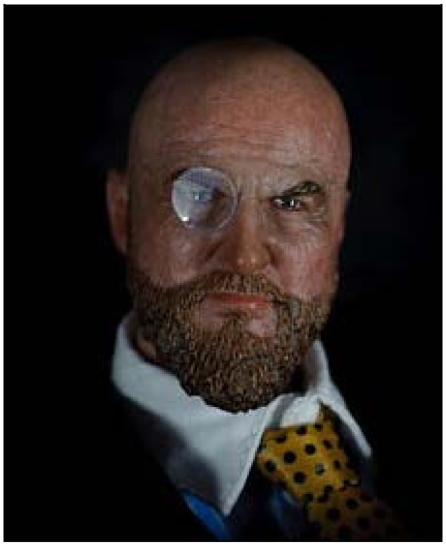


Enjoy your VFO experiments.




ON
HI-Z
50 OHM
GAIN
HI

The Butler Did It ! - First VHF Experiments 2011



Venturing into VHF, I felt like a beginner with no experience or confidence — however, the excitement and allure of new parts and circuits kept me going.

After performing a literature review, talking to some colleagues and renovating my QRP work bench, my first tasks involved buying some VHF parts + exploring the Butler crystal oscillator. Specifically, I'll cover my experiences with the common base version of the Butler oscillator.

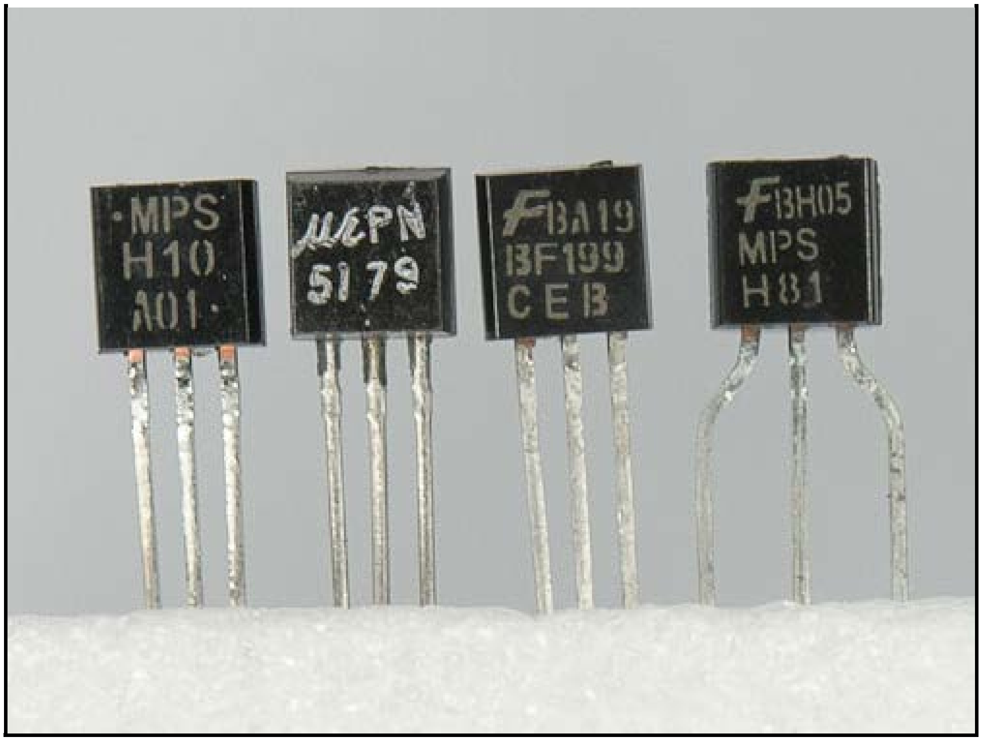
Like HF, the VHF knowledge base contains ever-present lore. Consider the Butler oscillator — I have read arguments stating that the emitter follower version of the Butler oscillattor is vastly superior to the common base version because the latter is prone to UHF and other spurs. These comments seem to have originated from a good book entitled Crystal Oscillator Circuits. *Revised Edition* by Robert J. Matthys published in 1992 by the Krieger Publishing Company.

While examining the schematics of professional/world class gear using a Butler, the common base version clearly dominates. Spectrum analysis and other measurements indicate that when common UHF oscillation management techniques are applied, common base Butler crystal oscillators work well.

Suppressing UHF oscillations with ferrite beads (and small value resistors), feedback, neutralization, limiting gain, etcetera are routine practices for us experimenters applying active devices that have strong gain into UHF on up. This is vanilla, or matter-of-fact construction for us; no worries. While fun and often convenient, lore ultimately stifles our progress.

Increasingly, I'm adopting the philosophy of Bob, K3NHI; "TMITK" — to measure is to know. Consider, too, you have to know what to measure and possess the required gear.

For BJTS, I purchased PN5179, MPSH10, 2SC3355, BF199, 2SC3357-RE, 2N5109/2N3866 and NE46134. The PNP part = MPSH81.



For JFETs, the J310 in TO-92 and SMT will remain my workhorse FET part along with a couple of other *low noise* JFETs and 2-gate MOSFETs. At VHF, the noise figure in a receiver chain is established by the first amplifier so a low noise preamp ranks important.

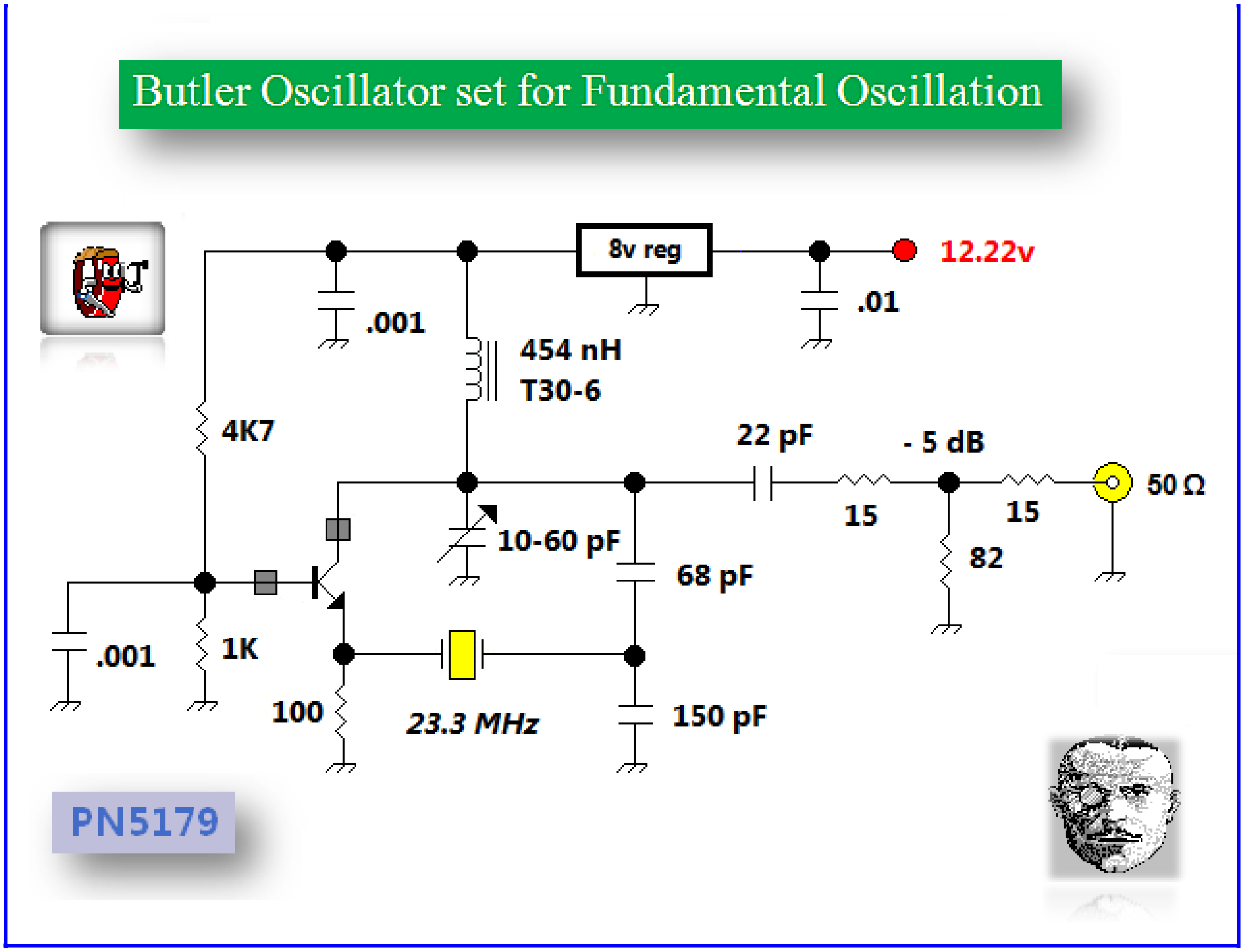
A collection of 100 volt NP0 capacitors ranging from 1 pF to 22 pF were added along with some chip and SMT caps as low as 0.5 pF. You might need a few air trimmer capacitors with a minimum capacitance ~ 2 pf — I applied 2 - 20 pF trimmer capacitors in most of the circuits that follow.

---

1. Experiments with a Butler Oscillator with a 23.3 MHz Fundamental Crystal

Fundamental Frequency

Long ago, I pulled a crystal marked 70.00000 MHz from a Drake Transmitter. I keep a fundamental oscillator based upon EMRFD Figure 4.23 on hand and verifed the fundamental frrequency at 23.3 MHz.

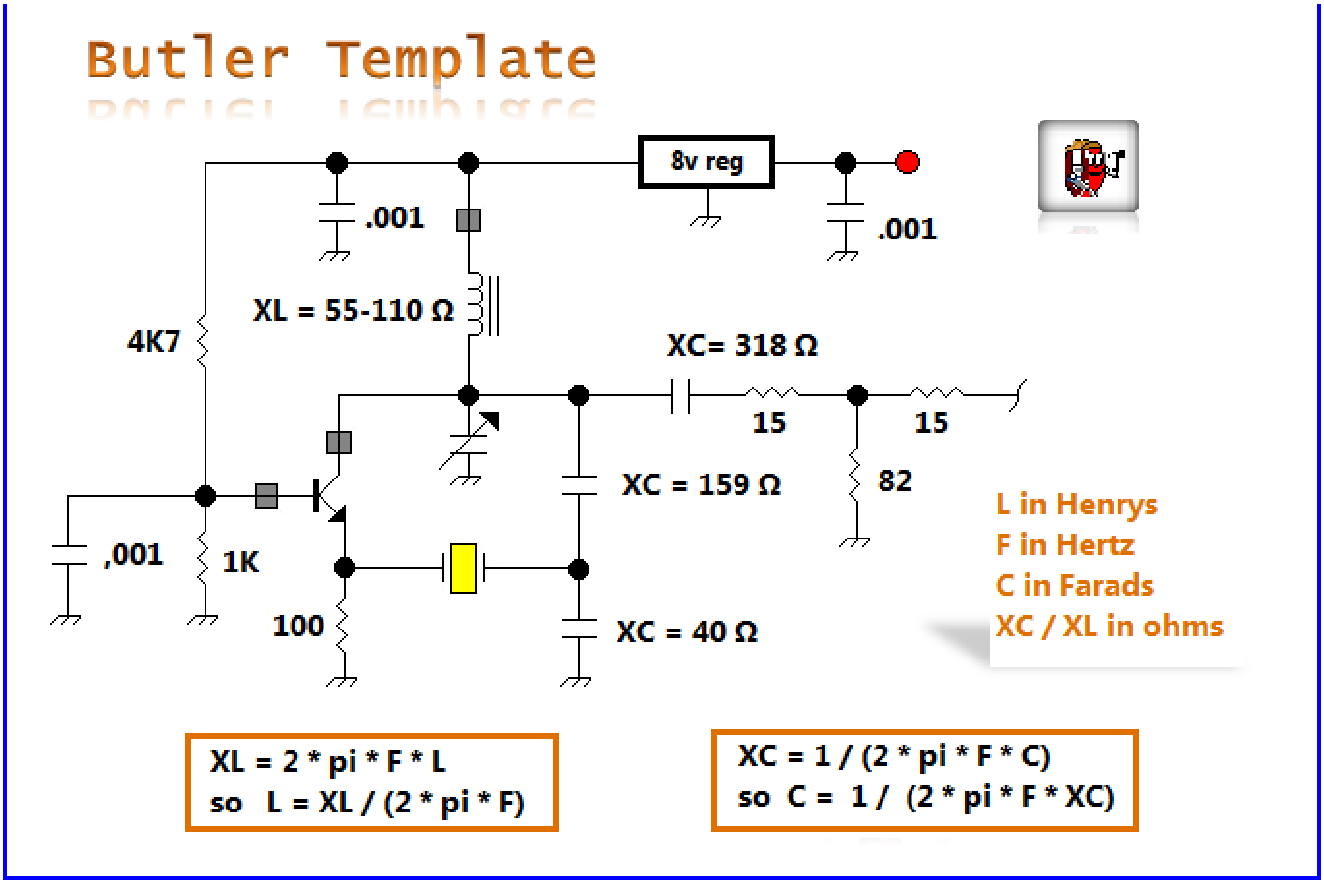


Above — A Butler oscillator arranged for output at the fundamental crystal frequency. While commonly arranged as an overtone oscillator, the Butler is a good oscillator for any application. Consider, for example, EMRFD Figure 7.32. Wes applied the Butler at a 14 MHz fundamental because he wanted the lowest phase noise and IMD prone signal source possible. Tellingly, his buffer circuitry also conforms to this high standard.

My initial waveform looked distorted and prompted a solution. My experience yields that the L value needs to be adjusted for the best looking waveform in the Butler circuit. The inductor wire, wrapped around a T30-6 toroid was either scrunched to increase the inductance, or expanded to decrease the L while re-peaking the trimmer cap. Eventually, with patience, a beautiful sine wave emerged on my 'scope. I removed and measured the L with an ADE inductance meter. Consider all of my reported inductance values as nominal — gentle expansion or contraction of the inductor coils might be required to get an agreeable sine wave.

Click for the oscillocope tracing at 23.3 MHz.

The Butler at Overtone Frequencies



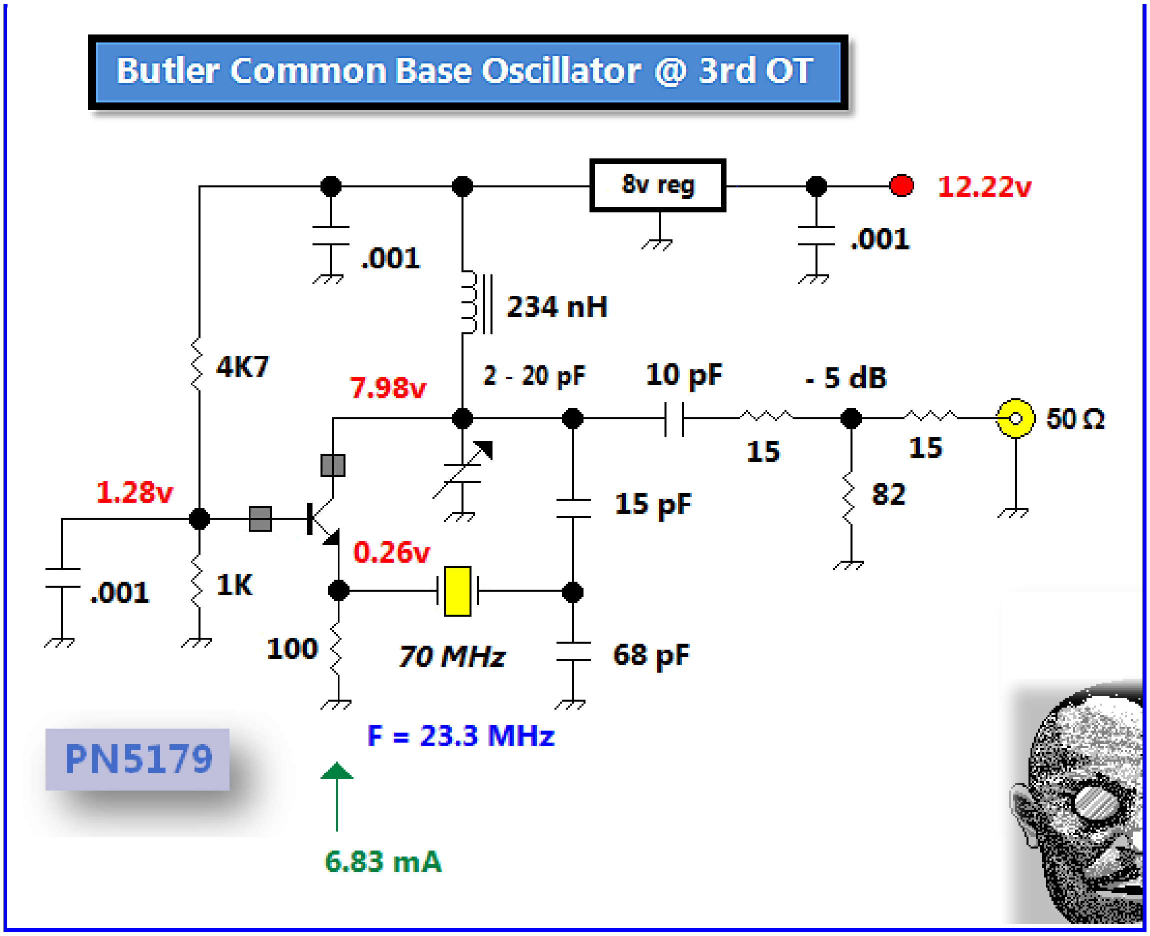
Above — A template (of sorts) for calculating Butler capacitor and coil values. The concept, rather than the absolute value matters most. I examined some well-designed Butler oscillators from professional equipment and determined their average XL and XC values. From the reactances shown, calculate the L and C values for the overtone frequency of interest with the 2 formulas in orange boxes. *Remember these XL or XC values just serve as starting values for experiments.*

For example : At 50 MHz with an XL of 108 Ω : L = 108 Ω / (6.28 * 50000000 Hz) = 0.000000344 H or 344 nH.

Fine tuning of the capacitor and inductor values might be required since factors including buffer input resistance +/- reactances, the overtone frequency and/or your breadboard layout may affect your Butler oscilator function.

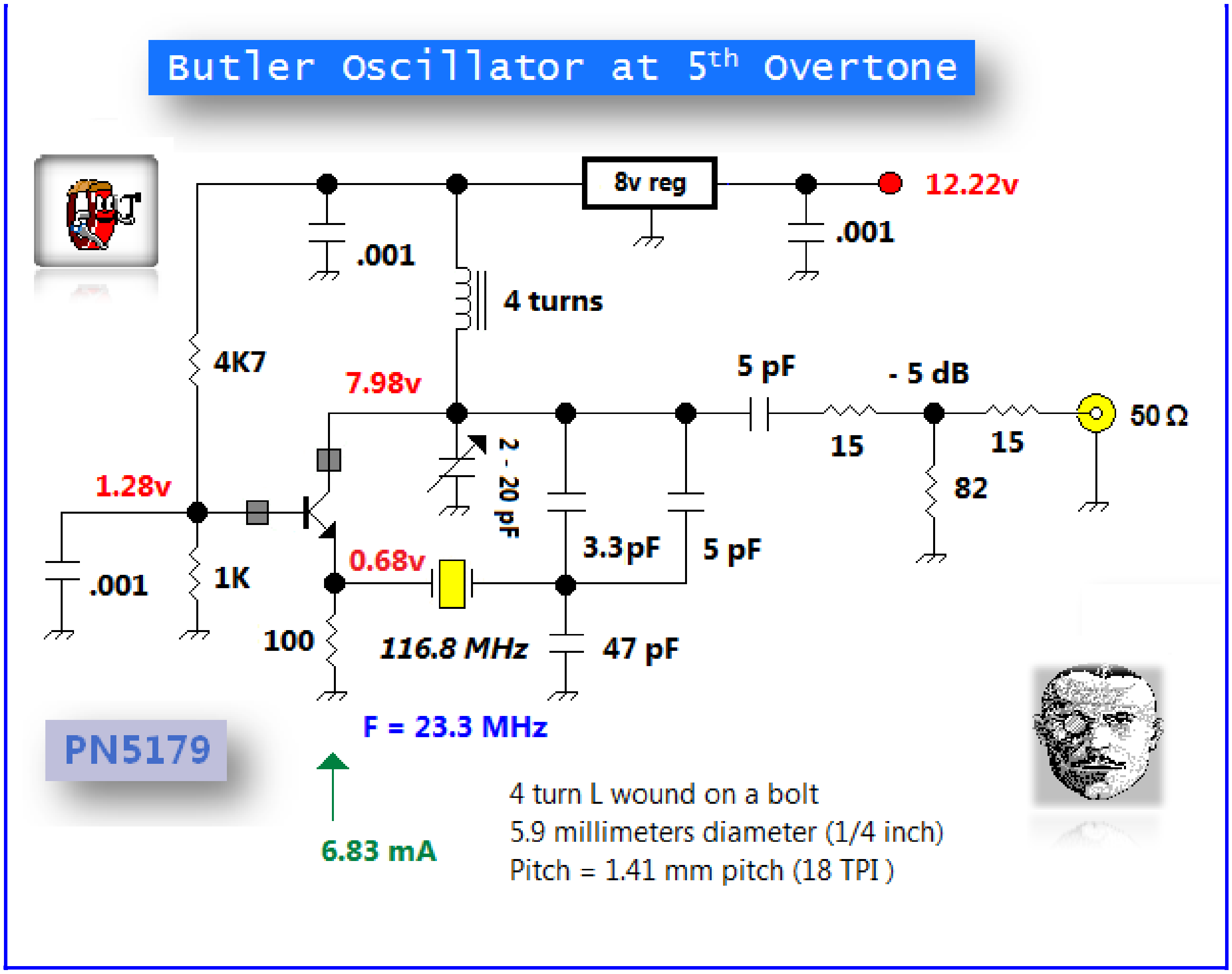
In the experiments from the 3rd to 9th overtone frequency, the L = an air inductor wound with 21 to 22 AWG wire on a bolt.

Third Overtone Frequency



Above — The Butler oscillator now arranged for output at the third overtone. Click and click for the 'scope outputs at the third crystal overtone. The first scope tracing was slightly mistuned. Once again, the inductor had to be gently squished or contracted to obtain a pristine sine wave.

Fifth Overtone Frequency



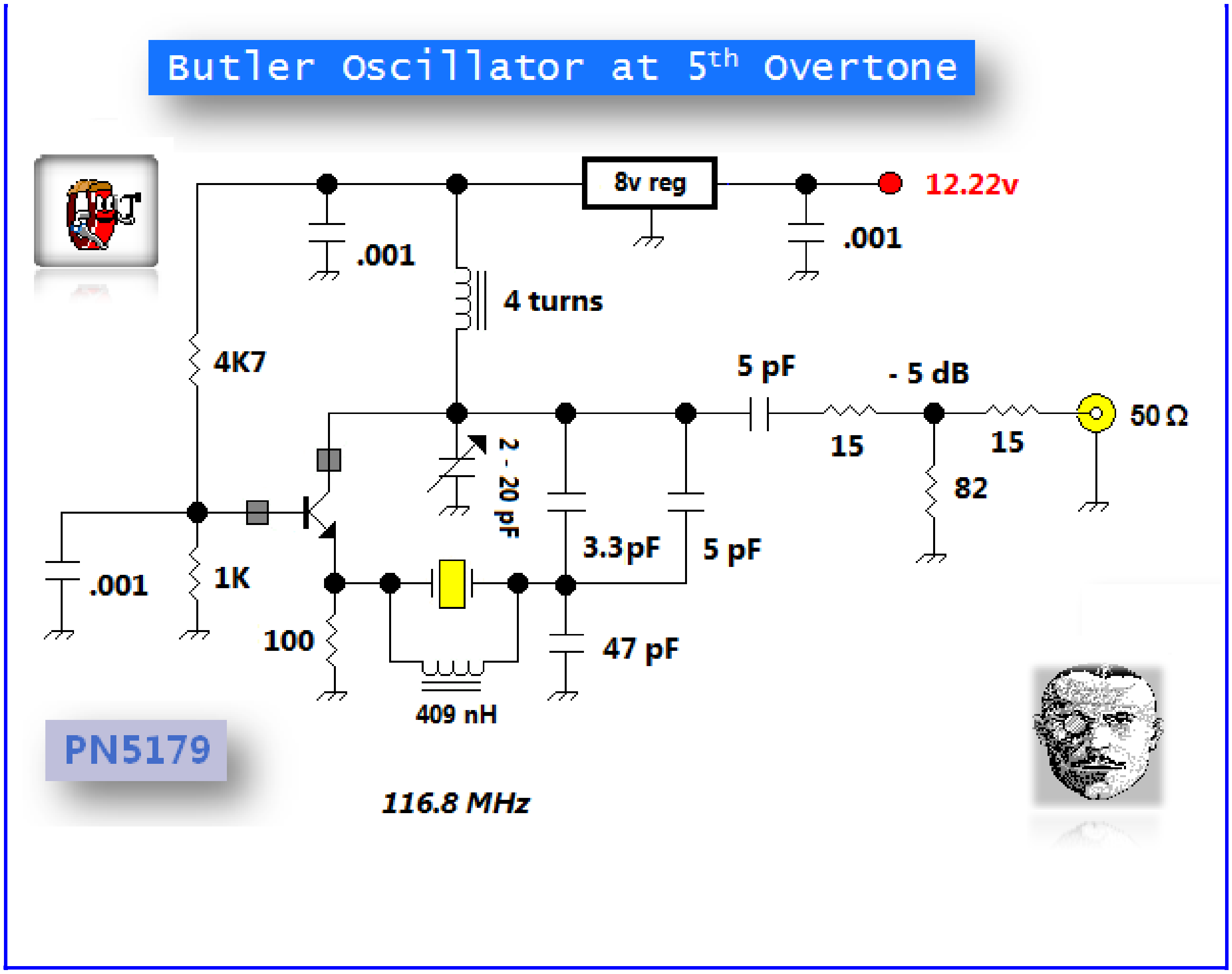
Above — The Butler oscillator at the 23.3 Mhz crystal's 5th overtone. Here's the 'scope tracing and an Ugly Construction photograph. The output = 7 dBm — very cool.

Some authors directly connect the attenuated output to the Local Oscillator port of a diode ring mixer. The signal is adjusted to the desired 7 dBm power by tweaking the 50 Ω pad attenuation, and/or the NPN's current. A good example = Single-Conversion Microwave SSB/CW Transceivers by Rick Campbell in QST for May, 1993.

The circuit above was measured with a 50 Ω terminated oscilloscope however, I also tested it with a 10X probe attached to termination resistors from 51 to 1 Meg ohms. When changing the 51 ohm termination resistor to a higher value such as as 47K, a previously working Butler may stop oscillating. The buffer input impedance and capacitance greatly affected the oscillator in my experiments.

When I wired up a Butler, 1 of 3 things happened: it did not oscillate, it gave a distorted output waveform, or it wowed me with a nice sine wave. Tuning the L-C tank is critical + finicky and may test your patience.

Adding an Inductor Across the Crystal



Above — The 5th overtone Butler with an inductor in parallel with the crystal

Above — My 116.8 MHz oscillator breadboard with an inductor wound on a T50-6 with wide spacing to allow scrunching and expanding of the windings across the crystal. I roughly determined my L should be ~ 400 nH and wound this on a # 6 toroid. While observing the output in my scope, I scrunched and then expanded its windings and adjusted the trimmer capacitor. The goal was to find a clean signal that snuffed out immediately when the trimmer cap was tuned off resonance. After finding the optimal L, I later removed and measured the coil. There is very little "wiggle room" — the oscillator tunes up and then dies very sharply as you tweak the trimmer cap. No sidebands were observed.

The inductor across the crystal is optional — some suggest it might only be needed above ~ 70 MHz.

At frequencies above ~70 MHz, the parallel capacitance of the xtal (C par) approaches the internal series resistance of the xtal and this provides an alternate path around the crystal for the signal and may short-circuit the crystal. The parallel inductor resonates with the crystal's parallel capacitance and tunes it out, so the crystal remains unbypassed.

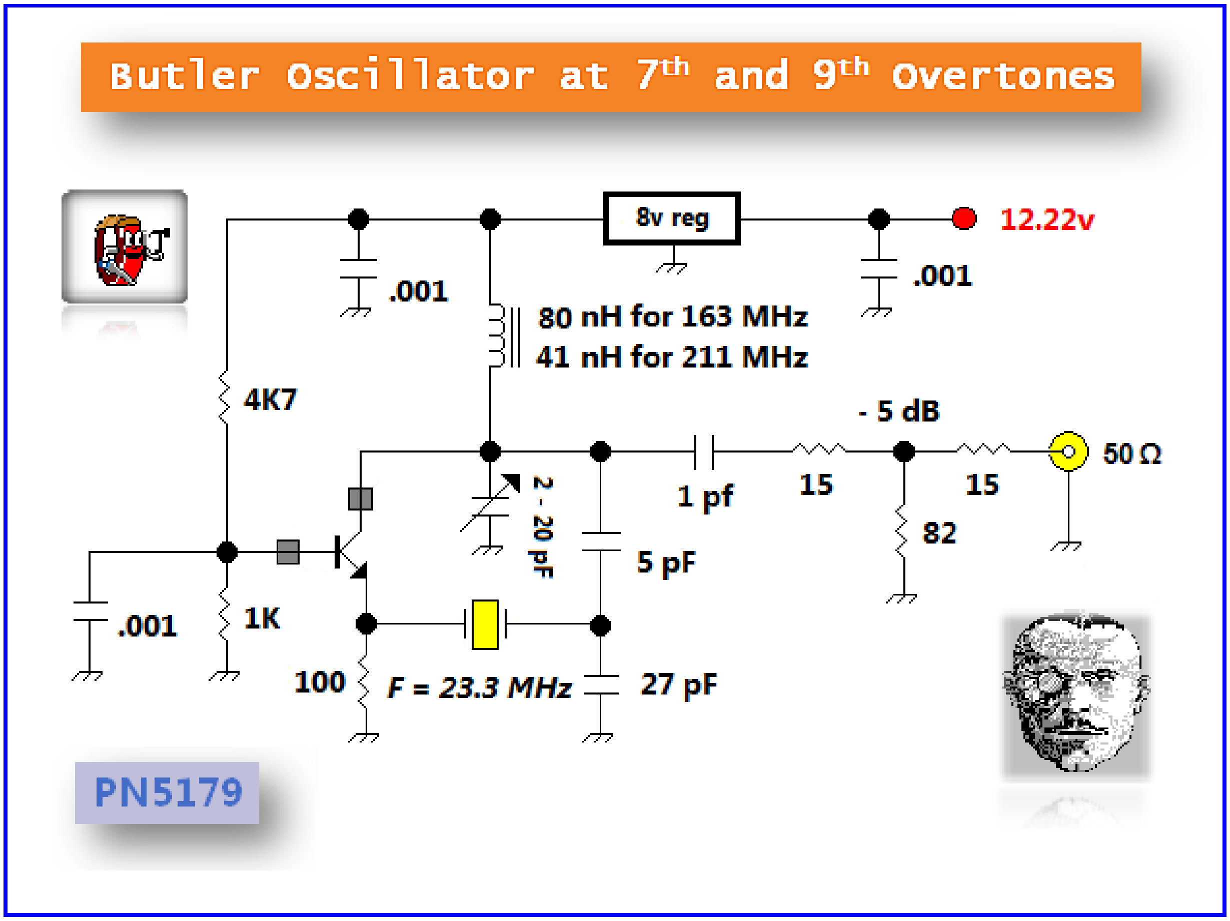
Our teacher, Wes, W7ZOI published a document covering the Butler parallel crystal inductor here.

Simply put — the inductor allows clean tuning and output. That is, when you tune the trimmer to one side or another, the oscillator just dies and doesn't produce the sidebands that are shown in Wes' web article.

Since many of us choose computer, or other surplus crystals, a high C par + low Q crystal might give you tuning woes depending on your overtone frequency. In this case, adding the L to your circuit may improve tuning and ward off any unwanted sidebands.

With my particular crystal the parallel inductor is not needed, however, I can report that even slight mistuning just snuffed out my oscillator with the added inductor.

Seventh and Ninth Overtone Frequency



Above — I decided to take the Butler up to the 7th and 9th overtones. The 1 pF coupling capacitor proved the most critical part; for example, if I raised it to 5 pF, the oscillator would not tune above the 5th overtone. I wound the inductors on bolts and then compressed or stretched the links to get the perfect inductance. At these frequencies, stray inductance becomes quite significant and my coils were 1 to 1.5 turns less than indicated on a spreadsheet coil inductance calculator. Click for the seventh overtone 'scope shot. Click for the ninth OT 'scope tracing.

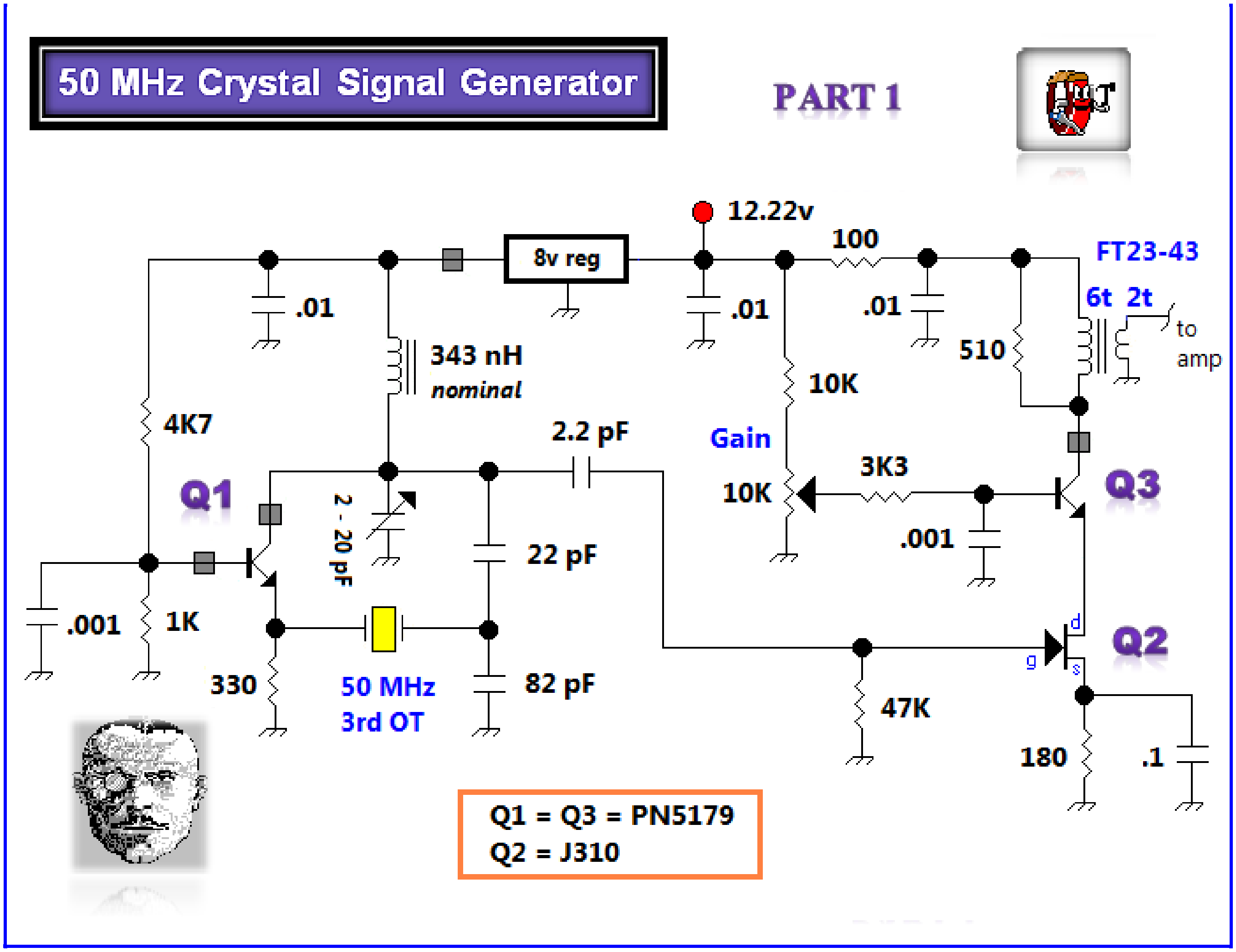
I'm confident that if my 'scope bandwidth was higher, I could have resonated the 11th overtone.

---

2. Butler Oscillator-based 50 MHz Signal Generator

Lacking a 6 Meter band signal generator, I decided to build a 1 frequency device employing a Butler oscillator. Some may laugh at a 1 frequency signal generator — I won't since I'll use it to design and align amplifers, filters, a new 6 Meter band VCO and measure scattering parameters. Besides, you can double, triple, VXO or mix single frequency generators with another variable oscillator — this web site has roots in humble, simple test equipment.

In my bag, I found a crystal labelled 50.0000 MHz and measured its fundamental at 16.67 MHz in a simple Colpitt's oscillator — perfect . Design goals included variable amplitude, strong reverse isolation + reasonable shielding and return loss. Since, I lack another 6 Meter band signal generator I could not measure return loss, however, choosing proven circuits + a terminal attenuator pad will help.

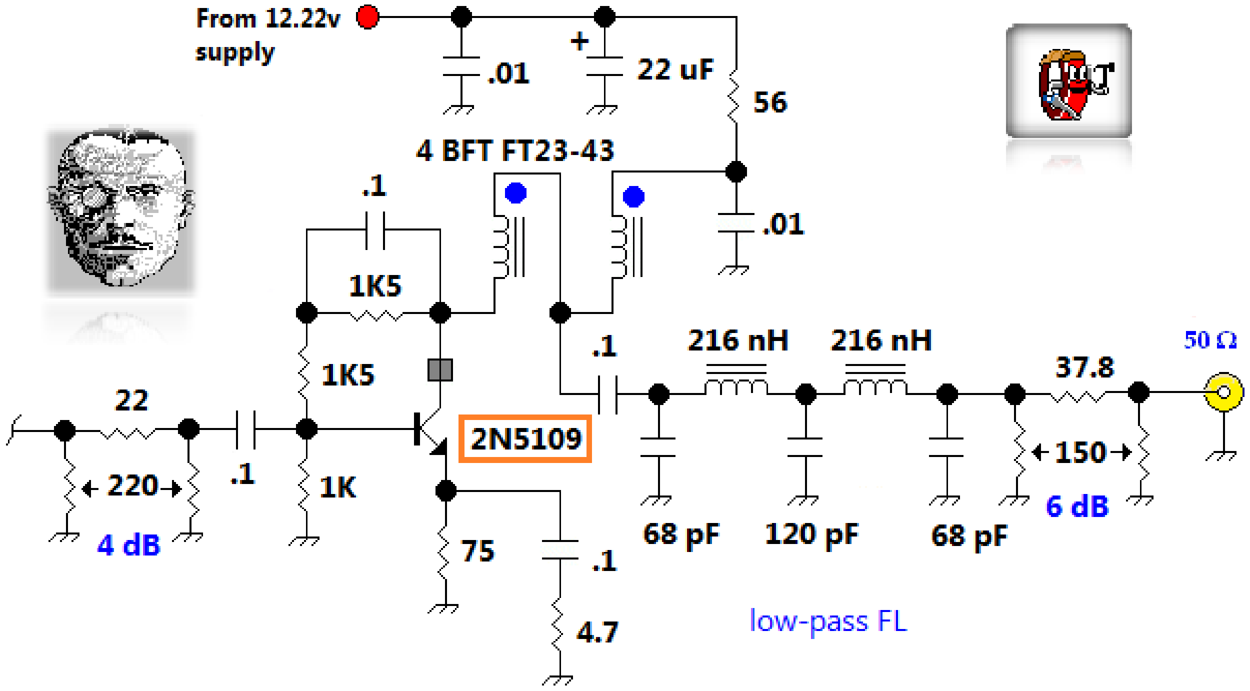


Above — Butler oscillator + hycas buffer. The 22 pF cap in the Butler was originally 15 pF, but when tested with various coils, oscillations proved a litle sluggish, so the 22 pF was substituted. I experimented with the coil, but ran out of time, so I wound a few turns of wire on a T50-10 toroid and soldered it in. After some careful manipulation of the windings, a glorious sine wave arose. I removed the L, measured it at 343 nH and then re-soldered it in place. An air coil or other size 6 or 10 powdered iron inductor should work fine.

Click for a moderate resolution photograph of the entire project.

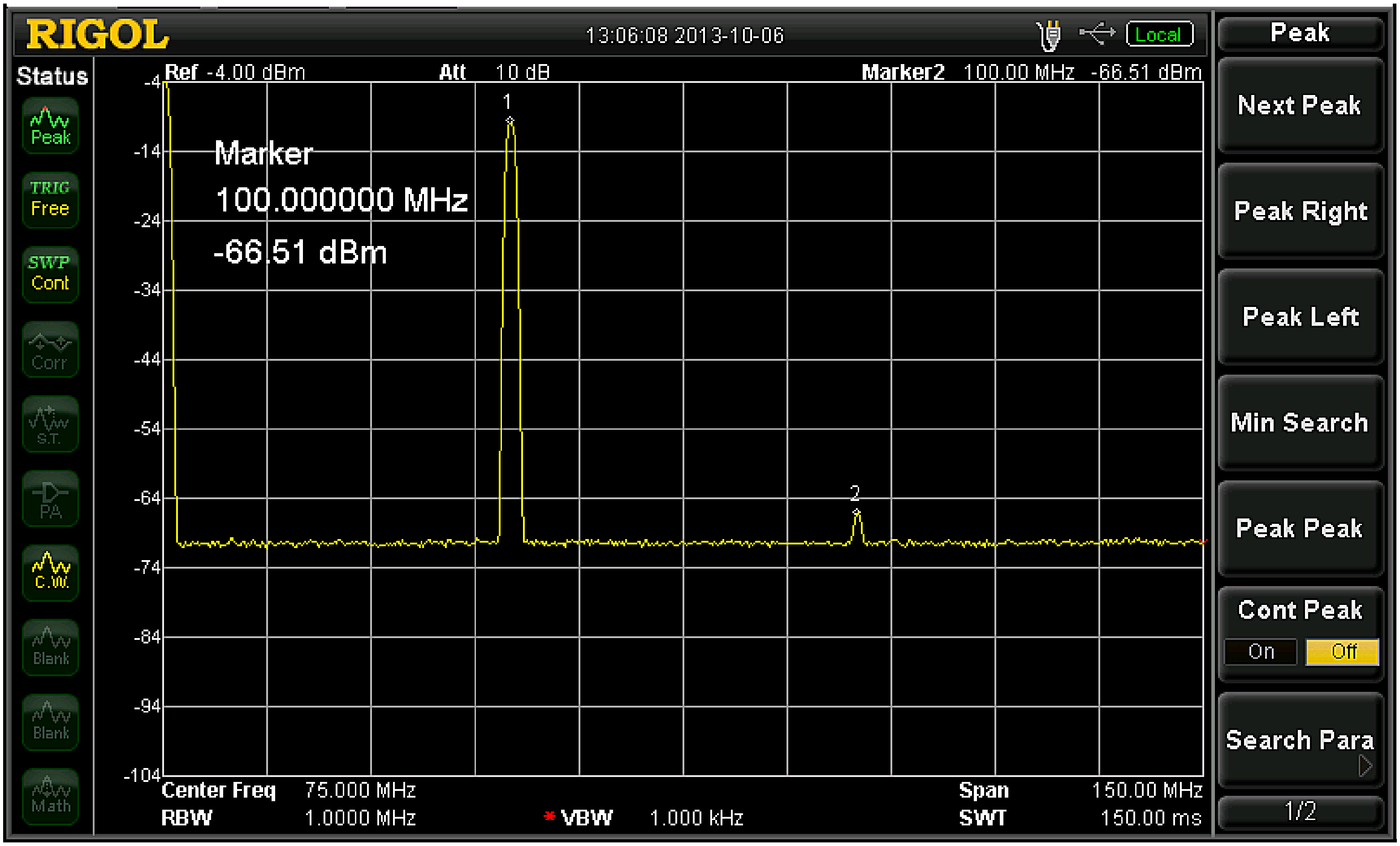
50 MHz Crystal Signal Generator

PART 2



Above — The final amplifier and low-pass filter. The maximum output of the Q2/Q3 amp is hot and can overdrive the 2N5109, therefore a 4 dB pad was added. This pad also improves the input return loss of the 2N5109. An alternate technique might be to further reduce the maximal DC bias voltage into the Q3 base and remove the 4 dB pad.

Click for the minimum and maximum output voltage waveforms from my 'scope



13:06:08 2013-10-06

Ref -4.00 dBm
Att 10 dB
Marker2 100.00 MHz -66.51 dBm

Marker
100.000000 MHz
-66.51 dBm

Center Freq 75.000 MHz
Span 150.00 MHz
RBW 1.0000 MHz
* VBW 1.000 kHz
SWT 150.00 ms

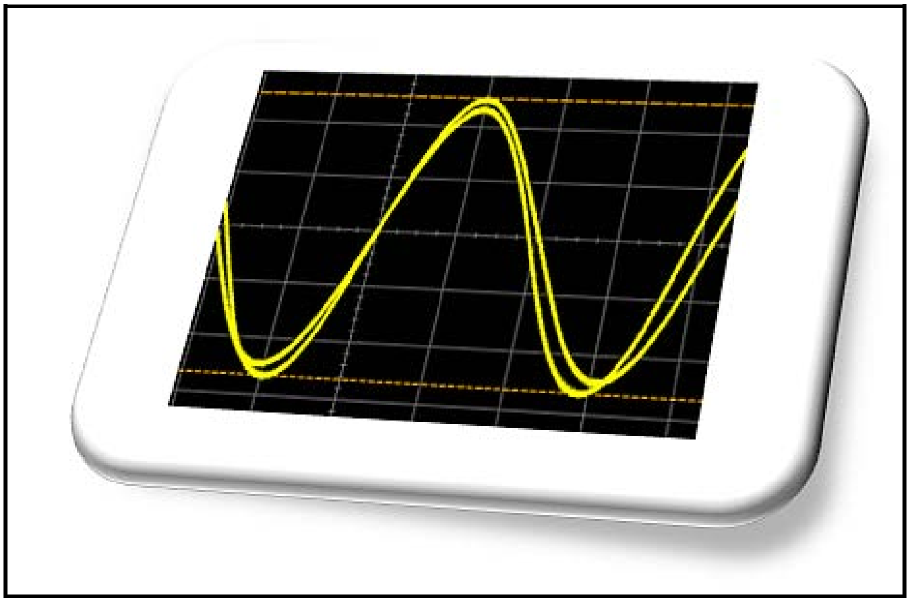
Above — Spectrum analysis with the generator output at -10 dBm. I placed Marker # 2 on the 2nd harmonic peak (100 MHz) and measured the power. The low-pass filter only attenuates the 2nd harmonic by ~ 25 dB, however, with the nice sine wave from the Butler, the 2nd harmonic measured -66.51 dBm = -56.51 dBc.

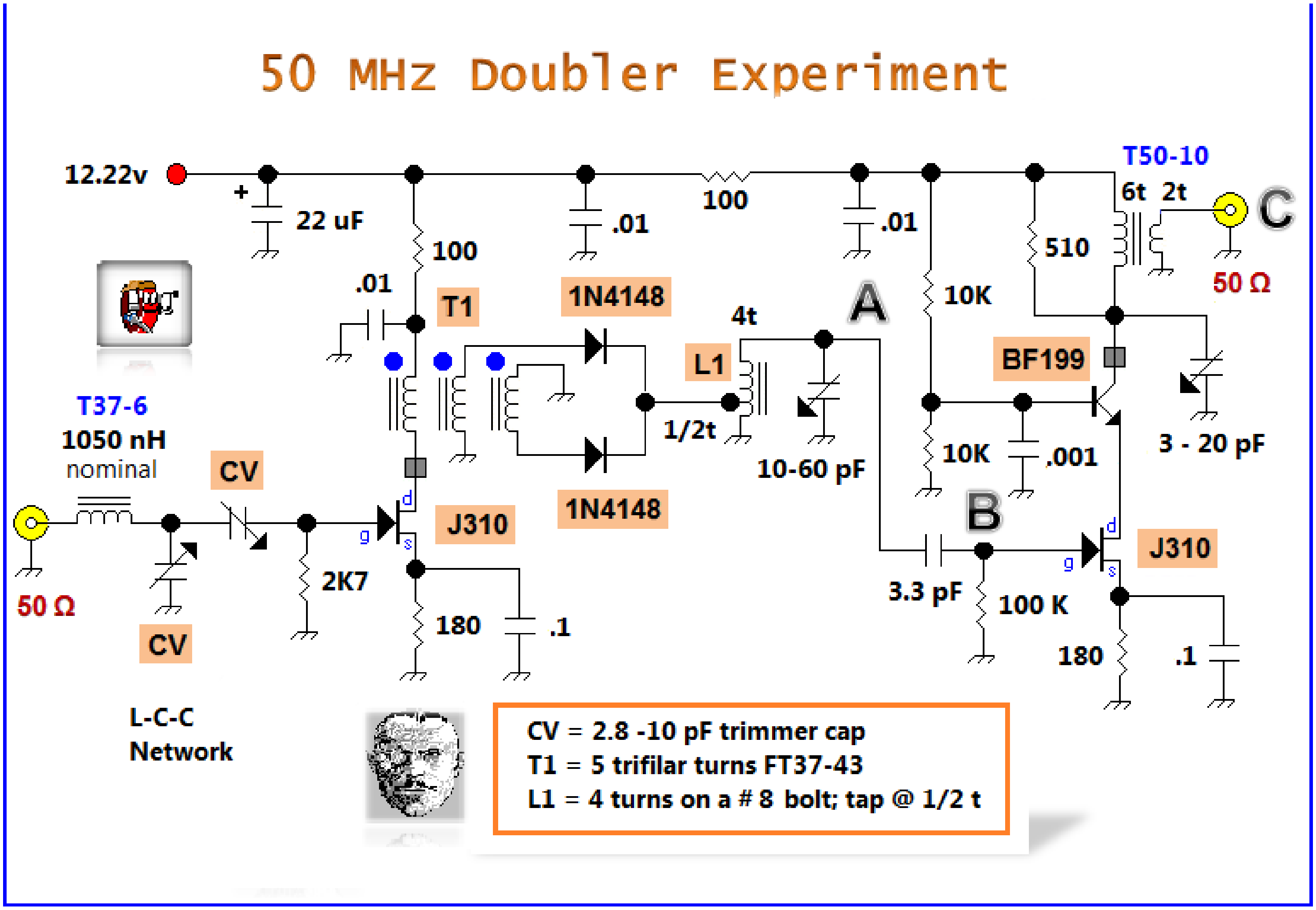
The 37.8 resistance indicated in the pad = 2 resistors in series. Many just use a 37 or 39 Ω R.

I look forward to advancing my VHF skills with this little signal generator.

---

3. Doubler for the 50 MHz Signal Generator





Above — An experimental frequency doubler for the Butler Oscillator-based 50 MHz Signal Generator shown above.

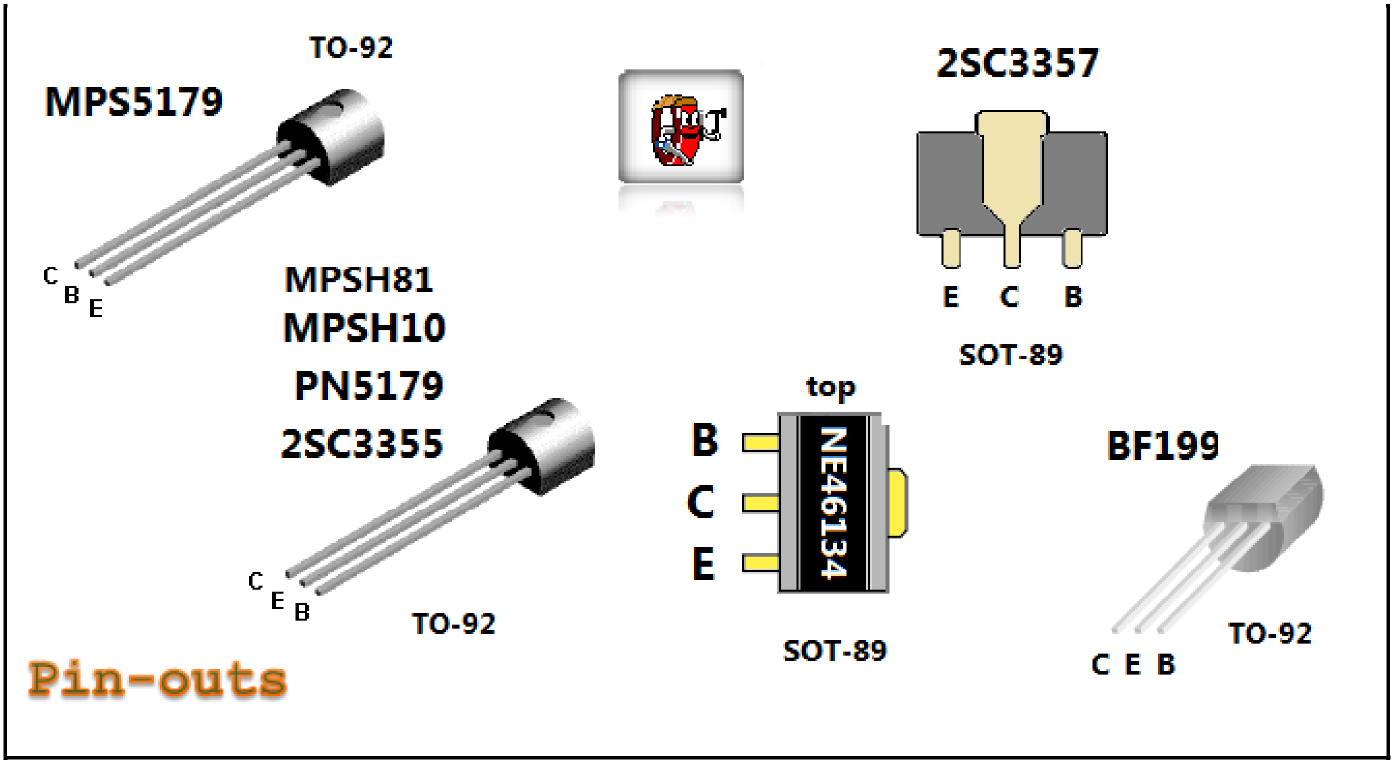
After trying a few circuits, I settled on a simple full-wave doubler featuring matched 1N4148 diodes driven by a JFET amplifier. Chapter 3 of Solid State Design for the Radio Amateur by Hayward and DeMaw for the ARRL = my key reference. Matched diodes (and a little luck), may suppress the 50 MHz signal up to 60 dB so only a single-tuned circuit follows the diodes.

The first JFET amp drives the diodes to improve harmonics + output voltage — an L-C-C Tee network matches the 2K7 Ω input to the 50 MHz signal generator output. I designed this matching network on the bench with the 2 diodes disconnected to avoid distortion during signal measurements. Using a 10X probe, I peaked the capactors for the greatest signal amplitude after finding the optimal L by educated trial and error. The inductor wound on a T37-6 was scrunched a little to further peak the L-C-C network.

I wound L1 with bare copper 26 AWG wire on a #8 bolt with coarse threads. To make enough space to solder on the 1/2 turn tap, I stretched the last turn with the other 3 turns still on the bolt to prevent stretching these coils. Click for a 'scope tracing at Point A. My 10X scope probe has ~ 15 pF capacitance and this affected the tuning — you can see some harmonics in the signal. With the 10X probe at Point B, I was nearly able to turn the circuit to 100.0 mHz, but still the 10:1 probe upsets the circuit somewhat.

This L - C tank tunes sharply and best with a non-conductive screw driver. I final tuned the L1 tank when the hycas amp was completed and connected to a 50 Ω terminated 'scope — eliminating the earlier tuning problems caused by the 10X probe. It seems that VHF requires more thought and care than HF when tuning resonators (tanks).

Point C is the maximum output (10.34 dBm) into a 50 ohm terminated 'scope. The 100.0 MHz signal is okay, but some builders might want to add low-pass filtering +/- an attenuator pad; or perhaps drive another feedback amp?


TO-92
MPS5179
C
B
E
MPSH81
MPSH10
PN5179
2SC3355
C
E
B
TO-92
2SC3357
E
C
B
SOT-89
top
B
C
E
NE46134
SOT-89
BF199
TO-92
C
E
B
Pin-outs

RF — Test and Measurement

VHF to the Max — Miscellaneous 2012 Experiments

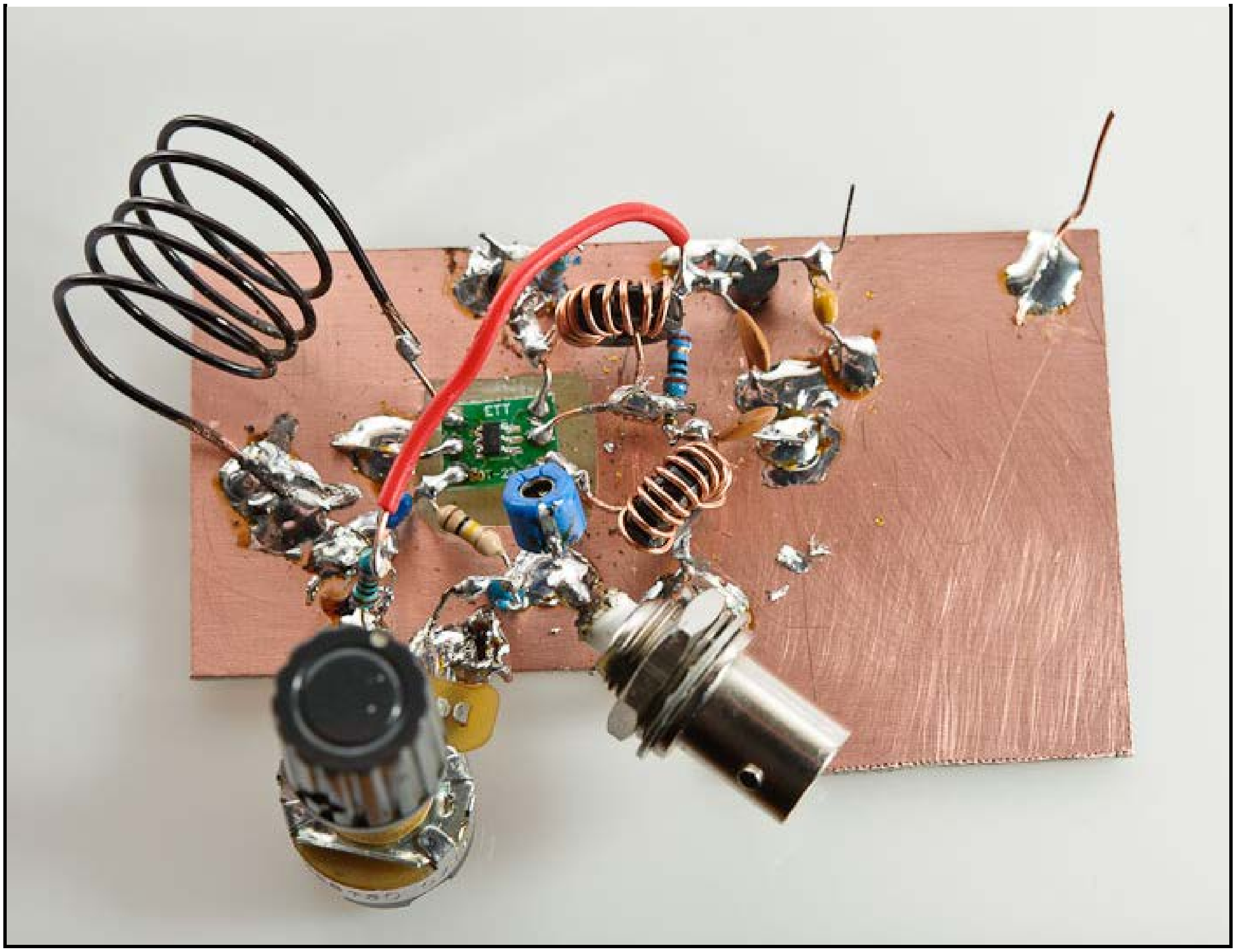
As a VHF newcomer, I need to make lots of circuits, measurements, mistakes and maybe — I might advance. My literature review revealed a big gap between popular, "for-fun", novelty-grade projects and the blinged-out circuits such as ultra low-noise LNA's featuring GaAs, MESFET, and pHEMT devices. Where do we find the middle ground projects? Likely on our own RF work benches.

Numerous questions arose — Will Ugly Constuction work? Do I have the right test gear? Am I measuring the proper things? What about noise figure? All a bit overwhelming — but even improbable discoveries began as simple questions and observations. We solve far more difficult problems each day.

I'm learning that whether your a beginner, or a master, bench fundamentals rank supreme. Want to avoid oscillations in your high fT BJT VHF amplifier? Work towards high reverse isolation, strong S11 and S22, careful layout and employ techniques that suppress instability — nothing earth shattering huh? I'm told that acquiring the needed skills takes time and practice.

In early Winter 2011-2012, I built some VHF signal generators, amplifiers and a lots of junk. This web page documents a few of these experiments.

Section 1. Early MAX2606 Experiments

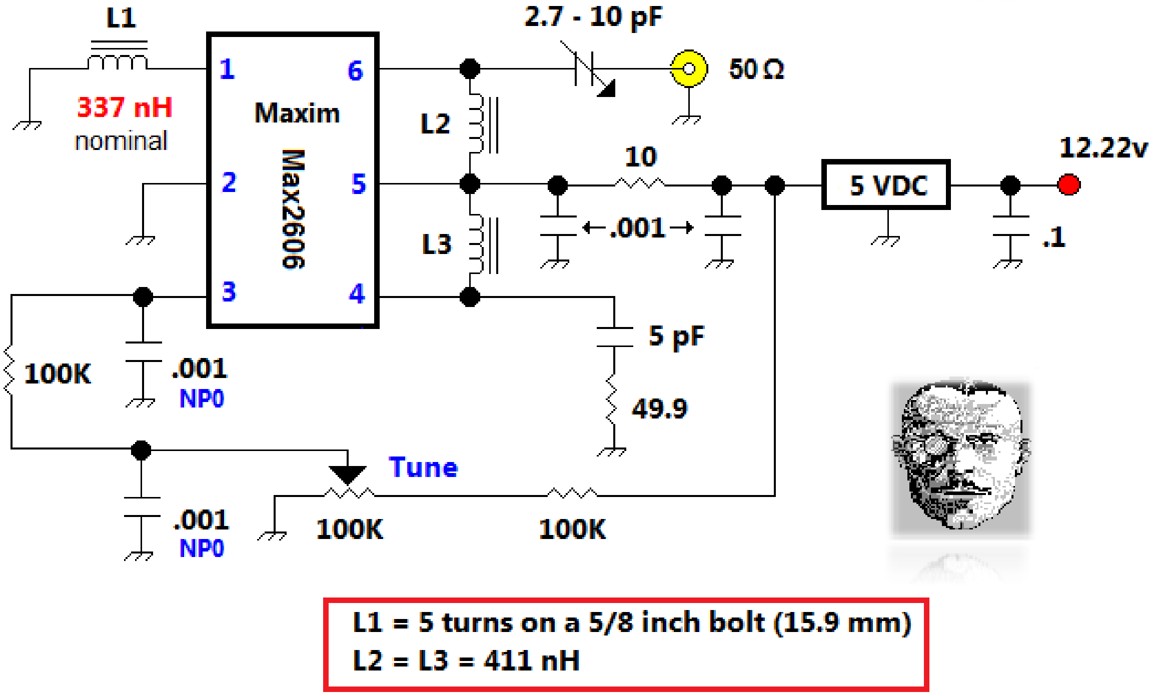


Above — My first of many MAX2606 VCO Experimental breadboards.

Maxim makes a series of cool SOT23-6 VCOs ranging from 45 to 650 MHz. Wanting a VCO to cover from ~100 to 106 MHz, I just had to try this chip. The datasheet provided all the online information I could find; save for a few homebrew, flea-powered FM transmitter projects that proved unhelpful.

After soldering the IC on a breakout (prototype) board using about 20X magnification, the proto-board was dropped onto a copper clad board with some of the copper ground away.

Max2606 VCO Experiments



Above — My first MAX2606 VCO experiment. A coil lying on the workbench was soldered in as L1. I stretched it a little to set the lower band edge. Going from CCW to CW on the 100K tuning pot, I measured from 86.3 — 106.3 MHz. I'll write about L1 a little later, for making a good output network consumed my initial experiments.

The Maxim datasheet shows a simple output network consisting of 2 pull-up resistors with a maximum R of 1K. All the online FM transmitter designs I saw used 1K pull-up resistors and made no attempt at matching or even employing inductors in place of the resistors. I applied two 560 ohm pull-up resistors in place of L2 and L3 and the output looked distorted and low in amplitude. Click for a 50 Ω terminated 'scope tracing and click for a tracing with a 10X probe connected to a 10K load resistor. Unacceptable for even us scratch-builder RF experimenters.

Wanting to match the output into a 50 Ω load with high-pass L-network instead of just resistors, I calculated that an L of 250 to 800 nH, plus a capacitor from 2 to 10 pF might work, however, as a VHF newbie, my hopes weren't high. I went for a single-ended output and not knowing what to do, terminated the unused port with an AC-coupled 49.9 Ω resistor.

Choosing a 100 MHz test frequency (where I own a doubled 50 MHz xtal oscillator), I removed L1 to snuff the oscillator and measured an output return loss of ~ 17 dB with L2 = L3 = 298 nH and the variable cap set to 7.3 pF (I removed it and measured it). It took about 1 hour of trying different L values to get this return loss. I also learned that the L value and L to C ratio of L2 and its series capacitor greatly affects the purity of the output waveform.

Like all L-networks, the L and C *must* be correct to obtain a strong, clean output. Since C is variable, I tried various inductor values and further fine-tuned them by scrunching or expanding the L2 windings. *Compress or expand the L2 windings + trim the variable capacitor to peak the L-network at your desired frequency.*

From my experiments, the L3 inductance should be close to L2 for the best output voltage and symmetry, however, don't bother scrunching or expanding L3 because this won't greatly affect the output signal as long as the L3 inductance is close to L2.

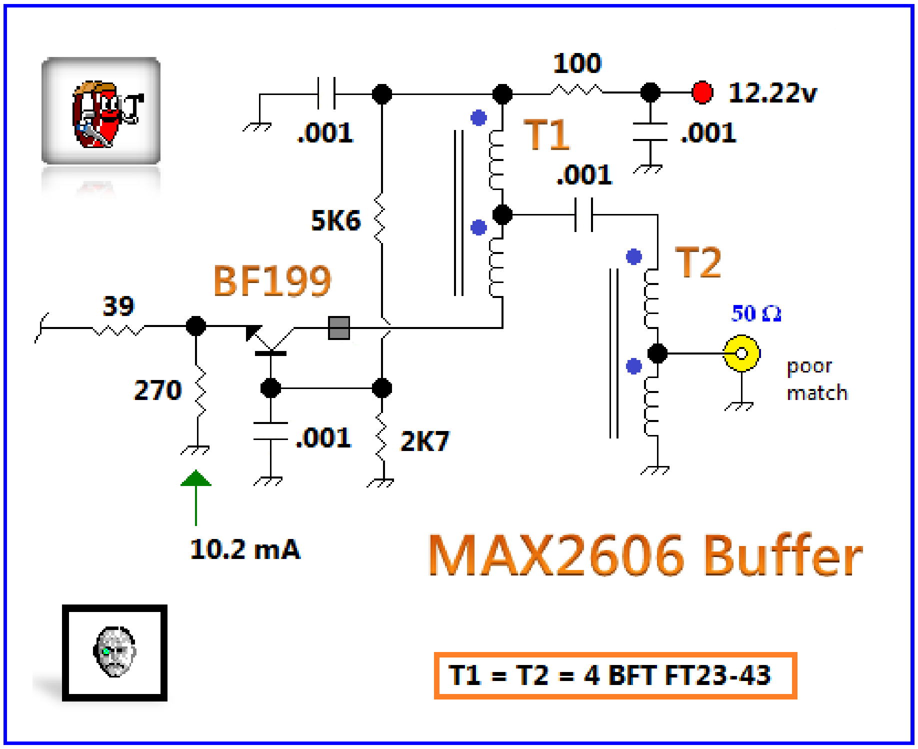
Click or click for some 'scope captures of poorly matched output L-networks — low gain and/or distortion appeared with mistuning.

Click for the output where L2 = L3 = 298 nH. I had the tuning set to 100.6 MHz for this screen capture, however, you can see the nice waveform and strong peak-peak voltage when compared to the pull-up resistor only versions.

The problem with a high Q L- network = low bandwidth. I wound L2 and L3 on T37-10 powdered iron toroids and later tried T37-6 toroids. The higher Q of the number 10 material seemed to translate into higher output voltage, but narrower bandwidth over the number 6 material toroidal inductors. Alternate breadboard. When tuning the MAX2606, signal amplitude changes with tuning frequency and a single L-network peaked somewhere near the middle of the tuning range further worsens this amplitude issue.

The *unbuffered* VCO waveform distorts somewhat as you move farther away from the peak L-network frequency. After trial and error, I settled in a compromise of L2 = L3 = 411 nH. This gave the best overall signal purity + amplitude from about 98 to 106 MHz with a set peak at ~ 102 MHz. Tuning below 98 MHz tended to really distort the waveform. Click for the *unbuffered* minimum and maximum frequency 'scope tracings.

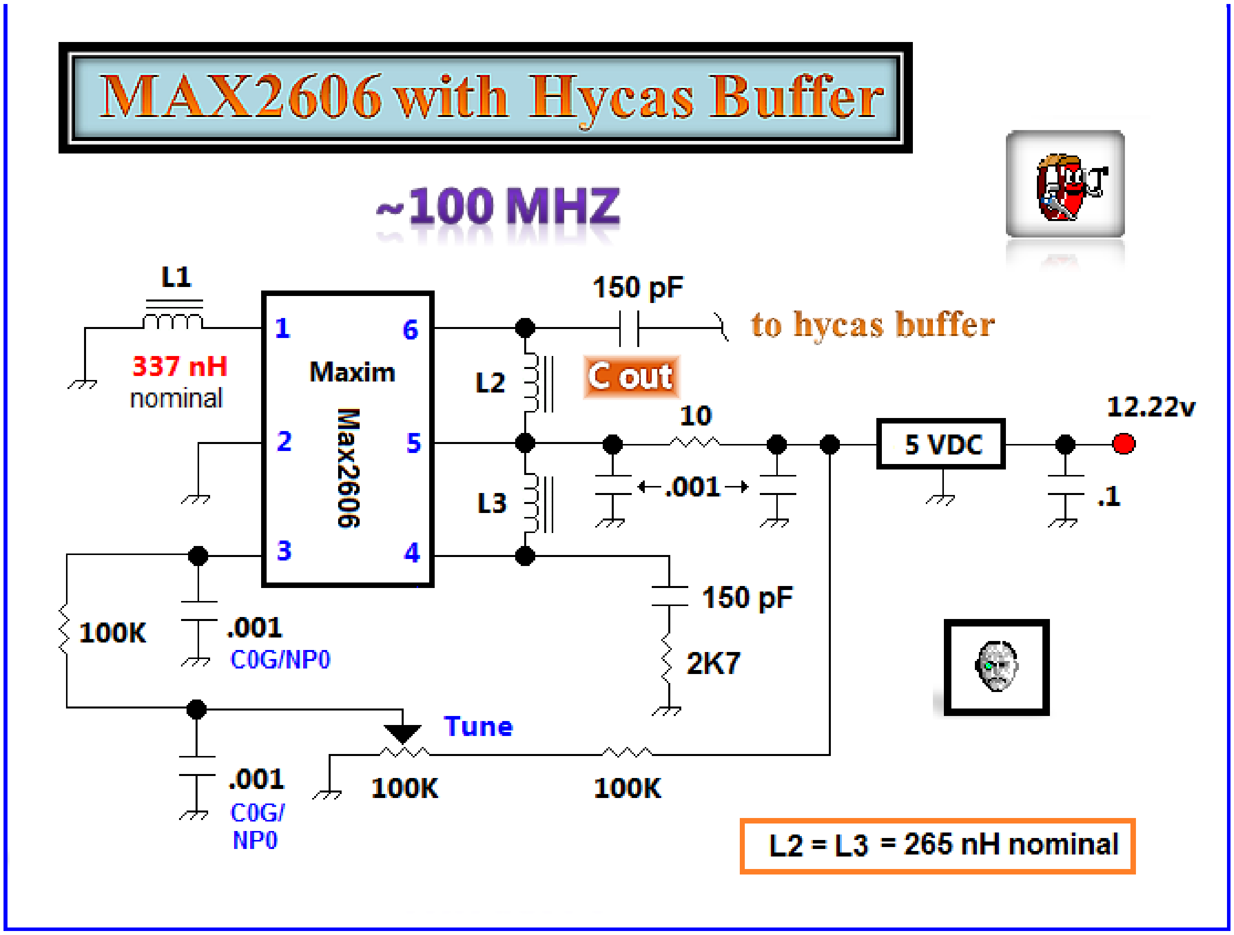
Placing an attenuator pad plus a buffer amp after the VCO dramatically reduced this distortion in my experiments, assuming the L-network components are correct and tuned.

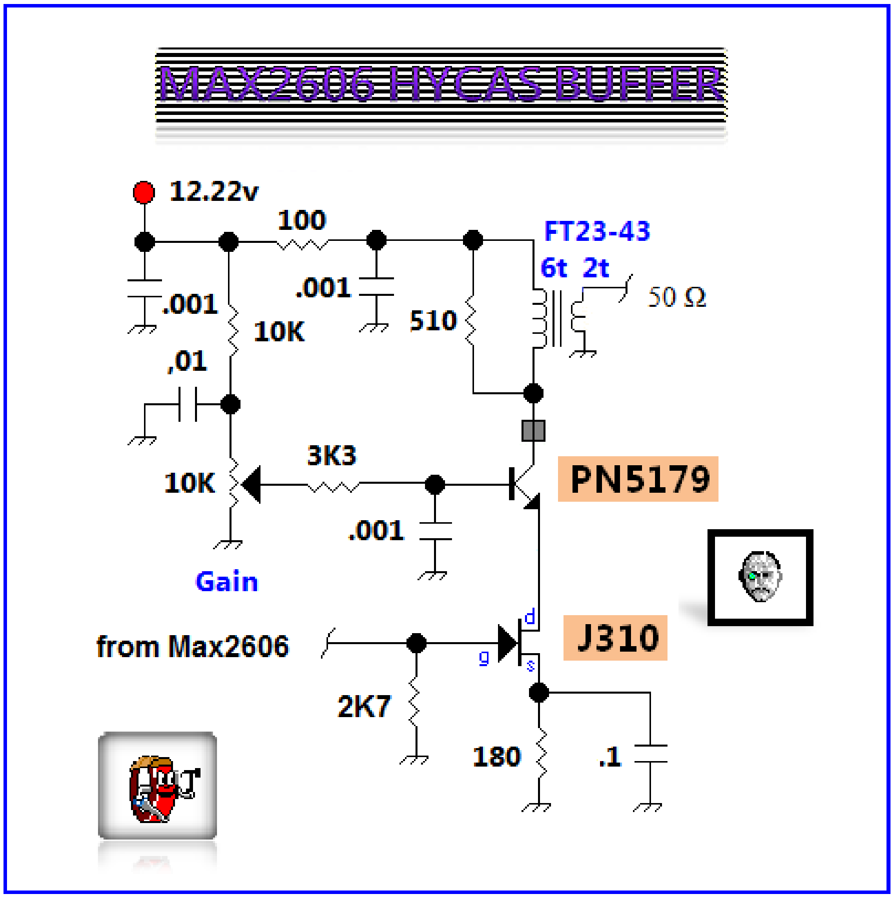


Above — A trial buffer first connected to output of the VCO schematic above. Two series-connected 2 transmission line transformers dropped the output impedance down for examination in my 50 Ω terminated scope — unfortunately the output return loss was poor (less than 10 Ω). Unlike the common base stage in a hybrid cascode amp, varying the bias to adjust gain worked poorly and lower applied bias caused distortion. I found it preferable to just fix the bias to get the greatest voltage gain + lowest harmonic distortion.

This buffer was discarded since the return loss was too low for my needs.

Testing a Hybrid-Cascode Buffer





Above — MAX2606 VCO into a hybrid cascode buffer at ~ 100 MHz. I also tried a buffer with a cascode of PN5179 transistors, however, the input Z of the buffer changed when adjusting the gain potentiometer and wrecked the input matching, so I stuck with the hycas buffer shown above.

L1 and L2 = ~ 265 nH wound on T37-10 toroids. To peak the L-match at ~102 MHz, I originally placed a 100 pF in parallel with a 10 - 70 pF trimmer cap for *C out*. After tuning, I removed, then measured these 2 caps to discover that *C out* = ~150 pF; so I just substituted a 150 pF capacitor. The L1 windings were expanded a little to re-peak the output network — expanding the coils raises fCo, while compressing the windings lowers the cut-off frequency. I don't recommend omitting a trimmer capacitor unless your confident with your measurements.

Click for the maximum power 'scope tracing of the hycas amp with the L-match peaked at 102.2 MHz (3.44 dbM)

Once again, I terminated the secondary output of the MAX2606 with an load resistance equal to the main output with an approximately equivalent fixed-value capactor — I'm not sure if it's needed, but it works okay and I stuck with it.

Up at ~ 144 MHz

Click for another experiment with the fore mentioned MAX2606 into the hycas buffer shown above at ~ 144 MHz. I peaked the L-match for 142.2 MHz and the output power = a surprising 7 dBm. The output is a clean sine wave from 135 to 169 MHz, although the signal amplitude varies widely. I also peaked it at ~144 MHz. I wound L1 - L3 with #21 AWG on a #10 bolt. Photo 1 Photo 2.

Let's build something useful with the MAX2606 based upon the experiments thus far...

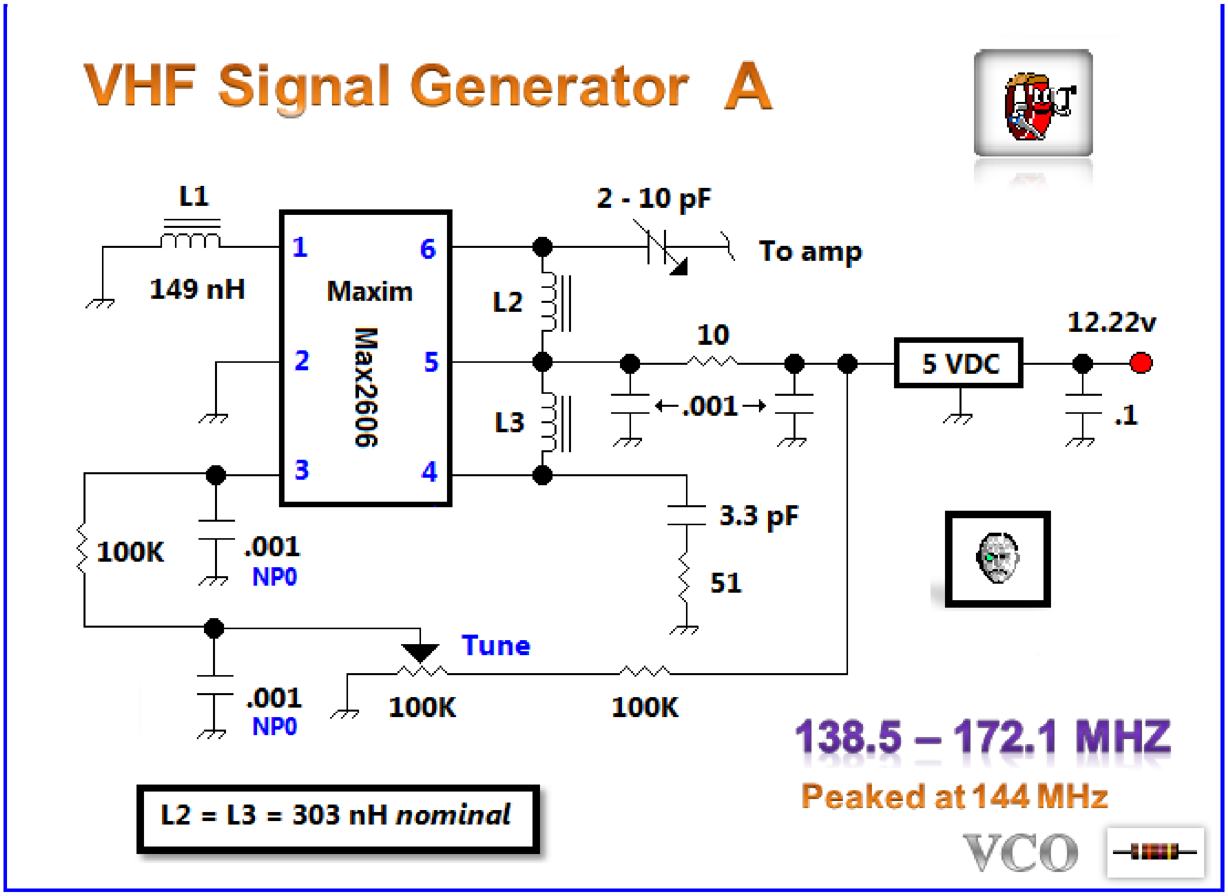
Section 2.   A 2-band Signal Generator Based Upon the MAX2606 VCO

Above — My dual VCO based on the MAX2606. This general-purpose signal generator will start my VHF circuit development in these 2 frequency bands. The importance of owning good signal generators can't be overstated — while not engineer grade, this box features a clean sine wave, strong output return loss and 1 KHz or better tuning resolution.

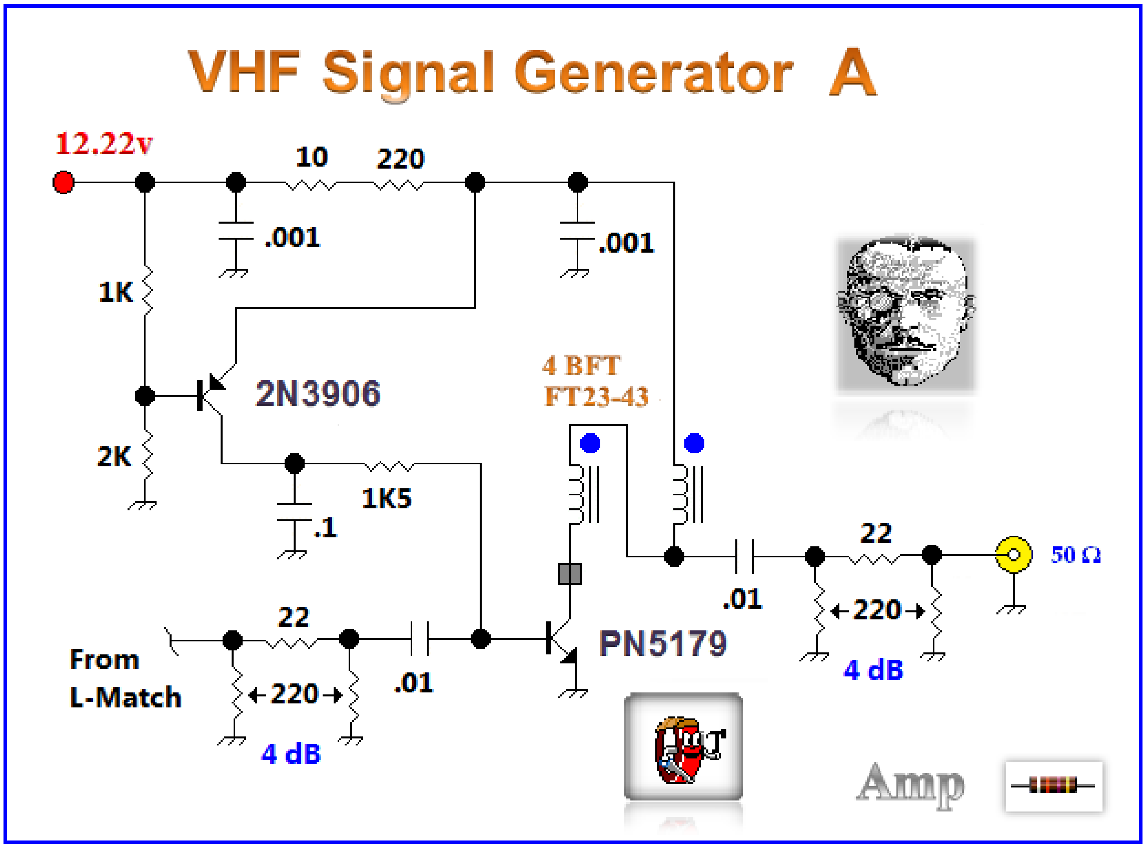
The center ON-OFF-ON toggle switch only turns on 1 oscillator at a time. The top 2 (ten-turn) tuning potentiometers lack knobs *(I'll get some later)* and the black knobs below them are attached to *unused* potentiometers. Initially, I planned to employ front panel gain control and drilled holes and fitted 10K pots in the chassis — I later decided to control the output power with outboard variable attenuators and skipped front panel power control for simplicity sake. Click for a side view of the unfinished project. Click for an early photo of the VCO **A** breadboard — I attached a shield to the copper clad board to help isolate the 2 VCOs.

I chose the frequency band **A** (138.5 - 172 MHz) to include the Ham 2 meter band + local commercial/service VHF segment and band **B** (98.919 - 109.06 MHz) to capture the FM broadcast band above 98 MHz.

Band A: 138.5 - 172 MHz



Above — The VCO schematic **A**. An L-match peaks the output at 144 MHz into 50 Ω. The beauty of this circuit = simplicity; just 2 active devices give low distortion and a strong output return loss (S22) on the 2 Meter Ham band.



Above — The output amplifier for VCO "A". Click for a schematic with some analysis at 144 MHz (the frequency I'll use the most). I spent a lot of time trying to develop a 50 Ω output Z voltage amp up at ~ 144 MHz. My attempts to employ shunt and series feedback gave generally poor results — stray reactances plague the standard FBA designs that work great under ~70 MHz.

Since the MAX2606 has an output L-Match, getting a high output return was my only goal — grounding the PN5179 emitter, employing a 4:1 transmission line transformer and biasing for ~15 mA emitter current did the trick. A 4 dB pad on the NPN input further establishes a strong S22 (output return loss).

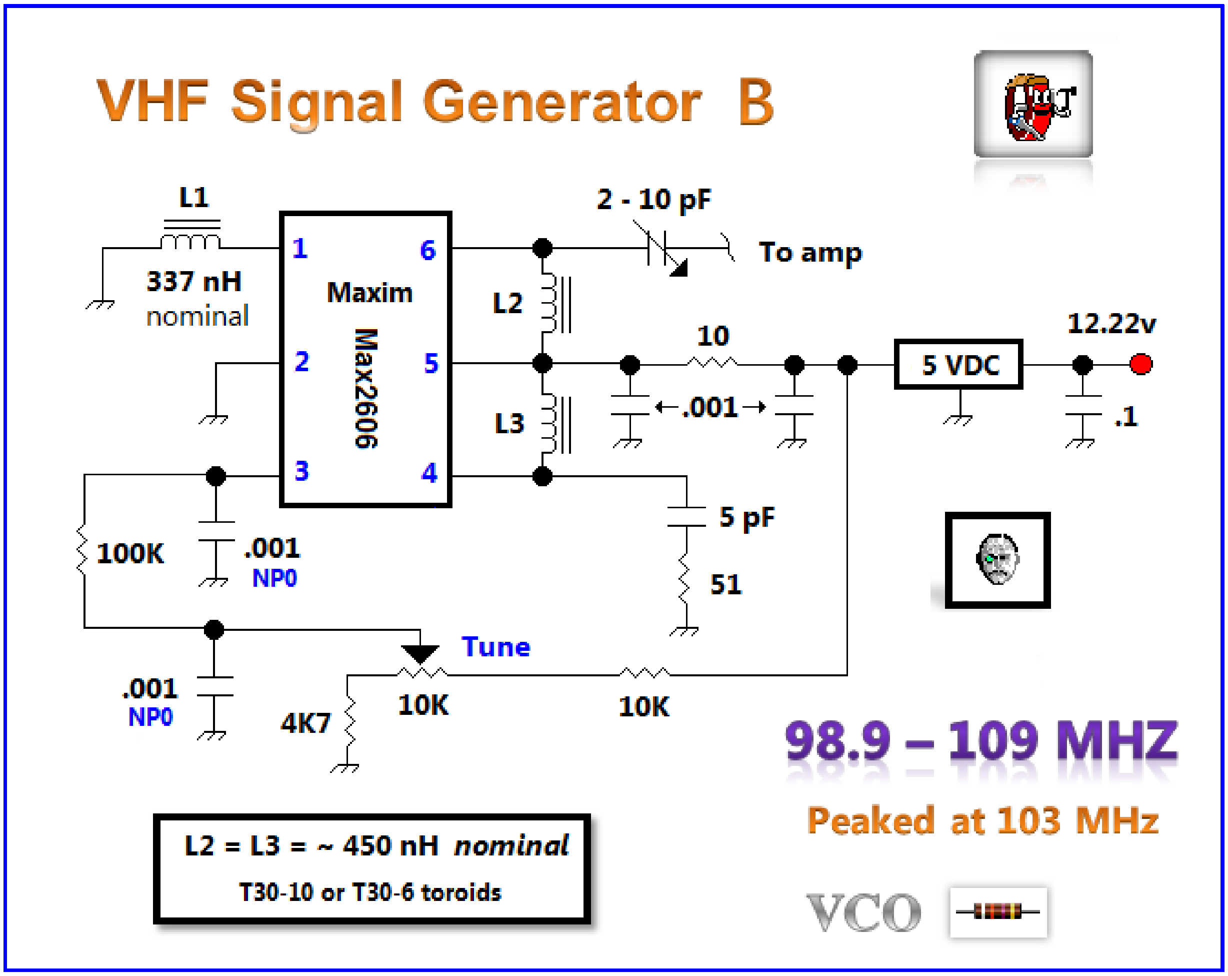
As possible, I attach additional 50 Ω outboard attenuators on my signal generators, however, the S22 on the stock generator should be okay across the VCO range due to the two 4 dB pads.

I measured the S11 on a prototype NPN amplifier by using the MAX2606 VCO shown to drive a MMIC with a 16 dB 50 Ω attenuator pad on the output and connecting it to the RF port of my return loss bridge. Thus, the actual VCO helped me design the final amplifier which buffers it in my final build. I kept the PN5179 and all other leads short as possible.

The output power looks like a sine wave when tuned from CCW to CW: With the 10-turn tuning potentiometer set to CCW [138.5 MHz], the output power ranges from -1.9 dBm; hits a peak of 0 dBm at 144 MHz and then gradually drops to -11.2 dBm at CW [172.1 MHz].

I moved away from the hycas buffer amp to simply my design and increase reproducibility. A MMIC might also work well, however the high current drain, potential for instability and biasing considerations introduce new problems — a simple wrap around PNP-biased NPN amplifier works okay. Sometimes the best solution = the simplest.

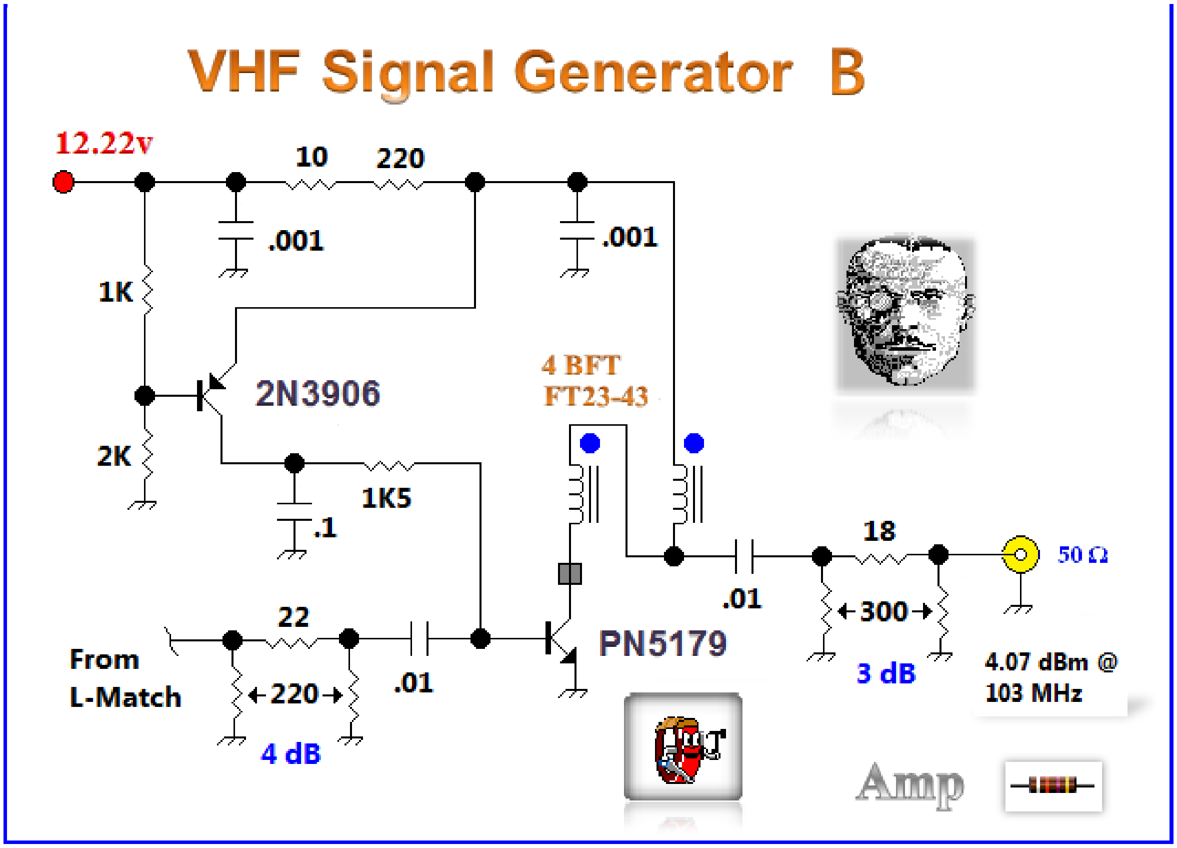
Band B: 98.919 - 109.06 MHz



Above — The VCO schematic **B**. An L-match peaks the output at 103 Mhz, although compressing or expanding L2 and tweaking the 2 -10 pF capacitor can peak the L-network anywhere in the tuning range. I limited the tuning range by adding a 4K7 R to the tuning pot to enable better matching and fine tuning.

In all cases L1 = an air inductor wound on a coarsely threaded bolt using 20 or 21 guage enamel-coated magnet wire for mechanical stability. I set the lower band edge of the VCO by setting the tuning pot to CCW and compressing or expanding the L1 inductor to get the frequency shown.

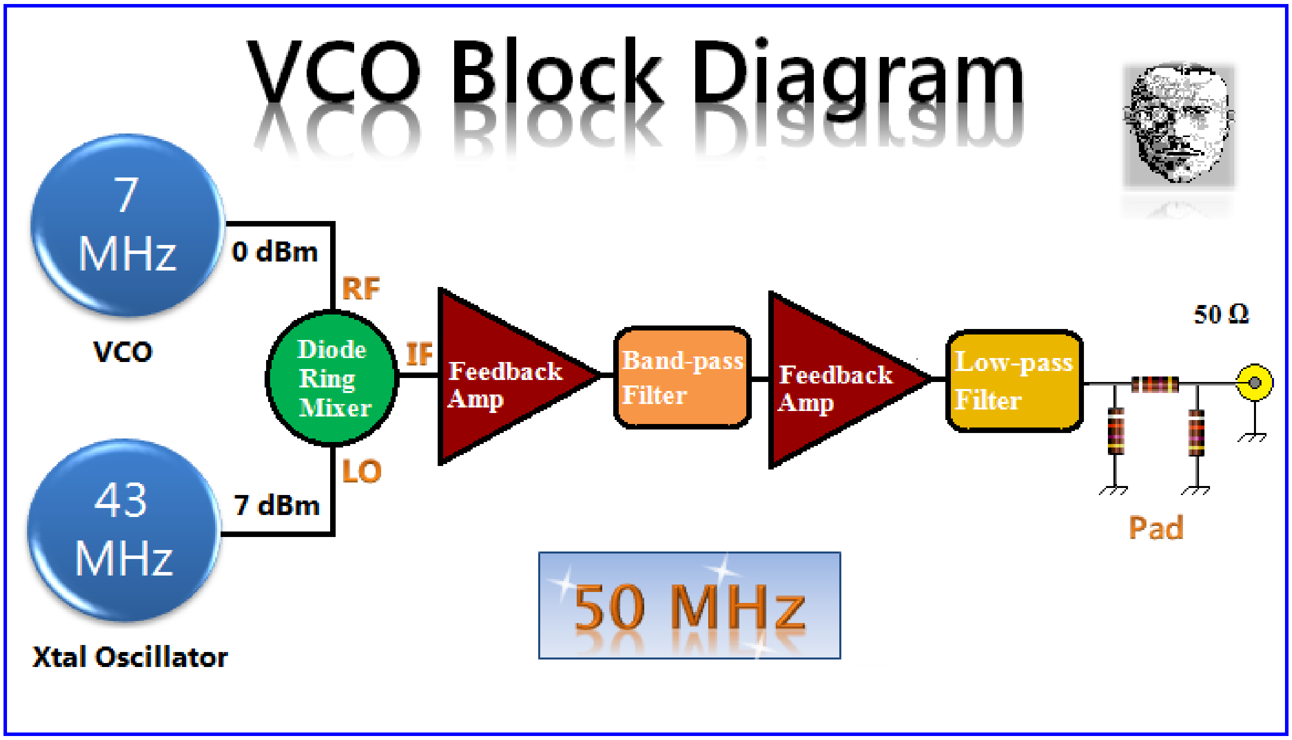
For L1, I found that excessively long leads can create unwanted oscillations and my coils are just a few mm above the copper board. Some builders properly mount their coils in a upright "smokestack" fashion and/or well away from the nearby metallic chassis or copper PCB to minimize Q losses + inductance changes. My coils lie well away from the metal chassis walls.



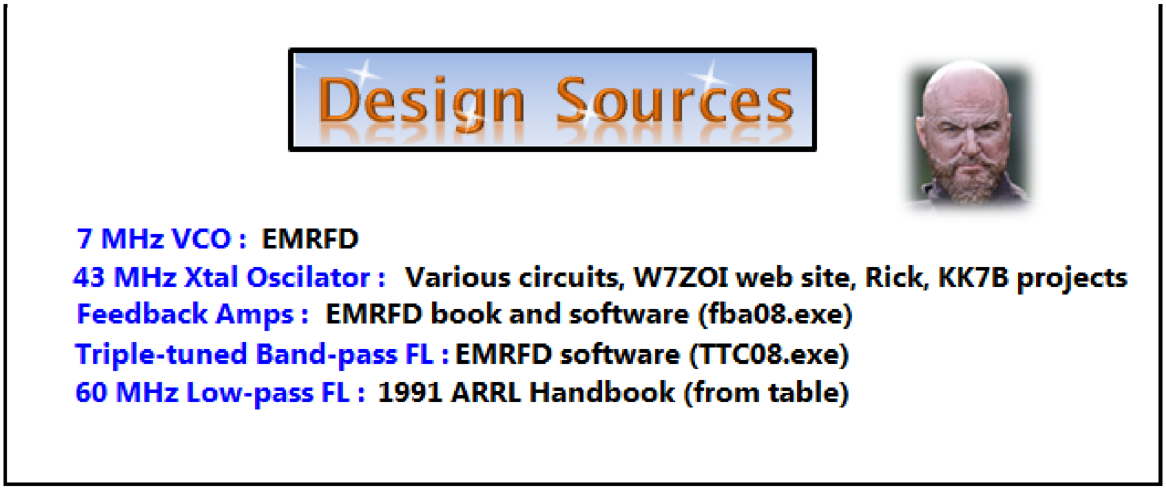
Above — The VCO **B** buffer/amplifier schematic. A clean sine wave appears across the entire tuning range — output return loss tuning from minimum to maximum was > 21 dB before I added the 3 dB pad.

Click for a photo of the completed, partially labelled project during the final tune up with all the boards bolted in.

Section 3. 50 MHz VCO



Above — Block diagram of the 50 MHz VCO I designed and built in February-March 2012. Click for a photo.



Design Sources

7 MHz VCO : EMRFD

43 MHz Xtal Oscilator : Various circuits, W7ZOI web site, Rick, KK7B projects

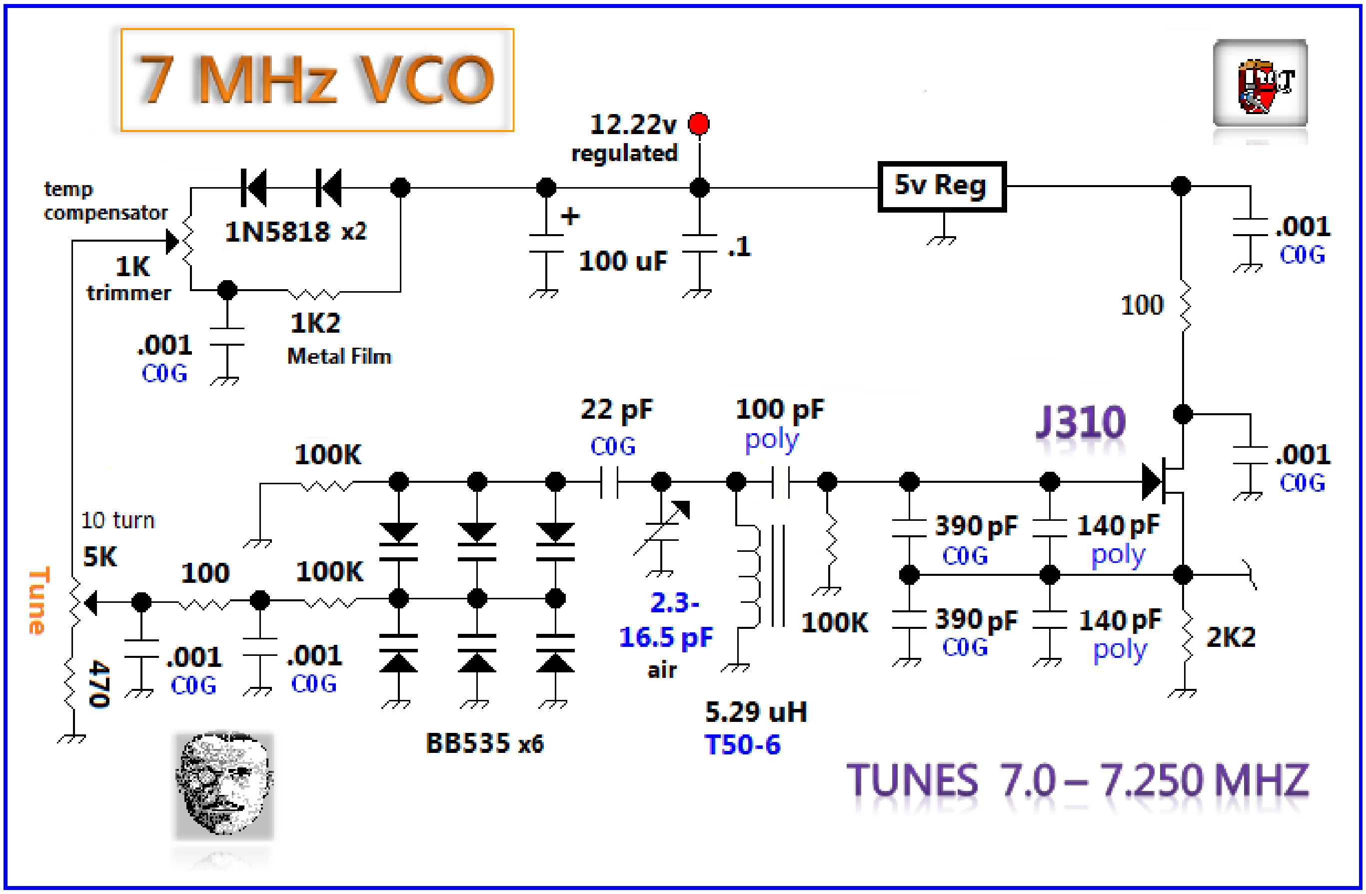
Feedback Amps : EMRFD book and software (fba08.exe)

Triple-tuned Band-pass FL : EMRFD software (TTC08.exe)

60 MHz Low-pass FL : 1991 ARRL Handbook (from table)

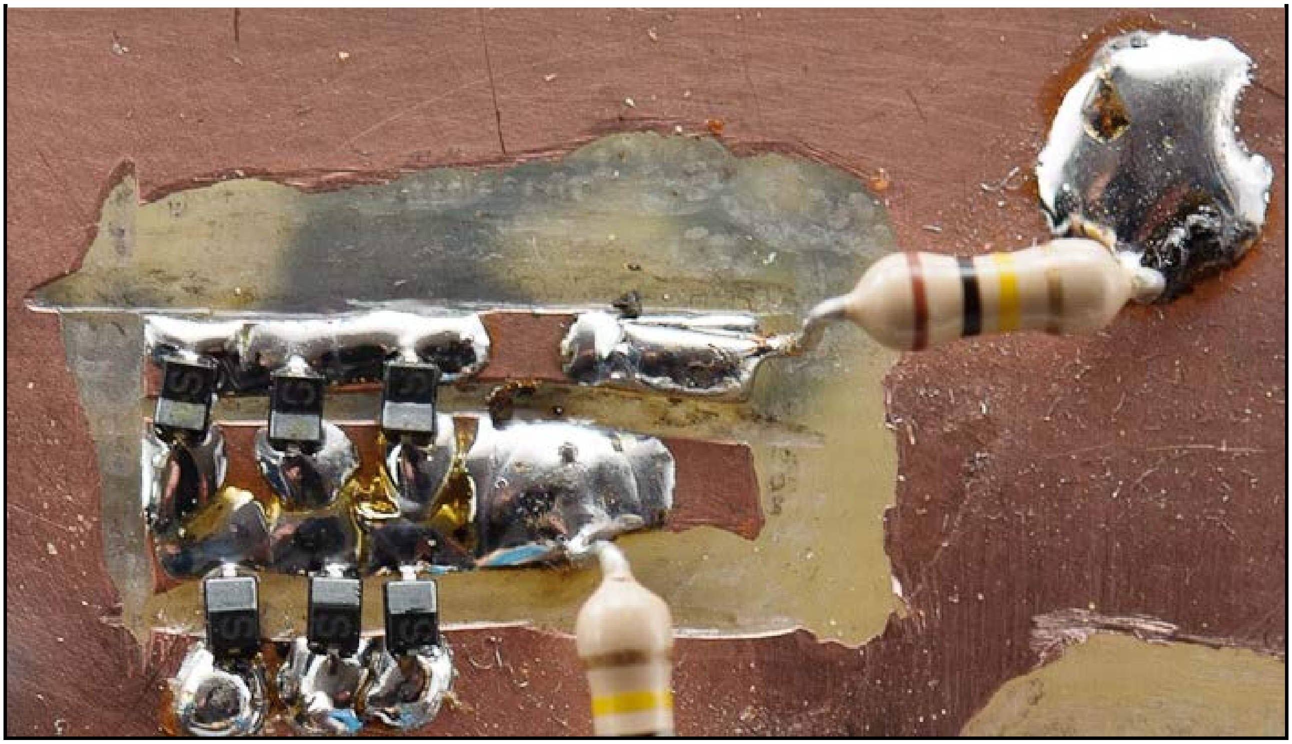
Above — Like most of you, I'm just an amateur designer who relies on others for example circuits, design procedures and inspiration. These cited references plus hard work drove my experiments. This project succeeds the Miscellaneous RF Experiments web page from 2011 — QRP SWL HomeBuilder evolves as I do.

7 MHz VCO + Buffer Amplifier [0 dBM output power]



Above — 7 MHz Colpitts VCO schematic.

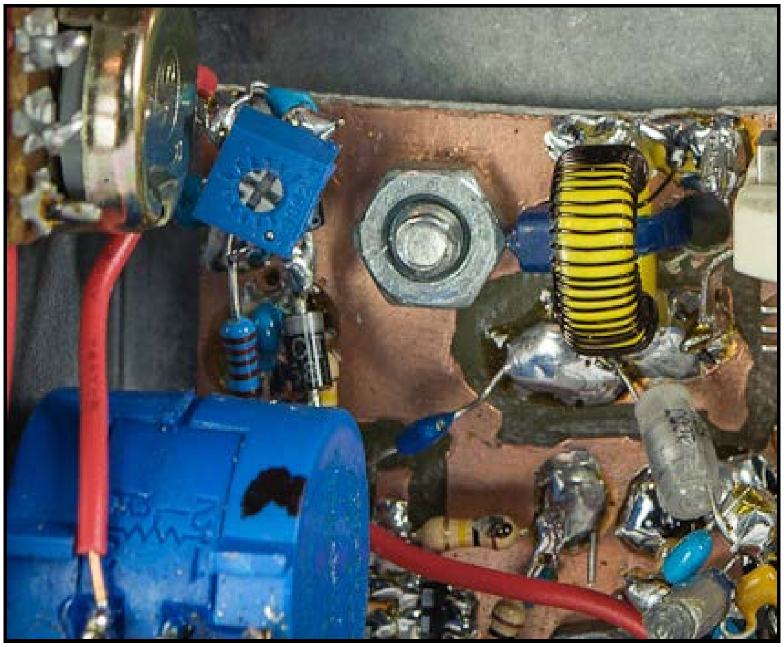
This VCO tunes ~7.00 to 7.250 MHz, although a wider tuning range occurs if you allow the tuning diodes to drop to 0 VDC with the 5K tuning pot cranked CCW. A 470 ohm resistor keeps about 1 VDC on the varactors at the lowest tuning frequency/applied reverse DC voltage.



Above — A macro photograph of the six BB535 varactors soldered on the VCO breadboard. With 0 applied reverse DC voltage, their total C = 43.5 pF. I left room for up to 4 more diodes, but didn't need them.

Macro photography provides an excellent way to inspect SMT parts — apart from all the fiberglass dust on the board, no shorts or other problems arose when soldering. Next to pF-value chip capacitors, these SMD varactors proved the most difficult surface-mount parts I've breadboarded to-date. Using clear tape, I tape my SMT parts to the PC board when soldering. With tape, you can still make tiny device placement adjustments with a pick or tweezers and yet the device holds steady enough to solder. I recently obtained a microscope for SMT work, although didn't need it for these diodes.

Striving for lower phase noise meant properly applying high Q tank parts — I soldered in 3 pairs of high-grade BB535 varactors and arranged them anti-parallel to avoid forward conduction + even harmonics. I also limited the AC voltage swing they "see" by connecting them to the L with a 22 pF capacitor. Tight windings of #28 gauge magnet wire on a T50-6 toroid formed the inductor. 4 number 8 bolts anchor the 7 MHz VCO board to the chassis and prevent board warp + movement.

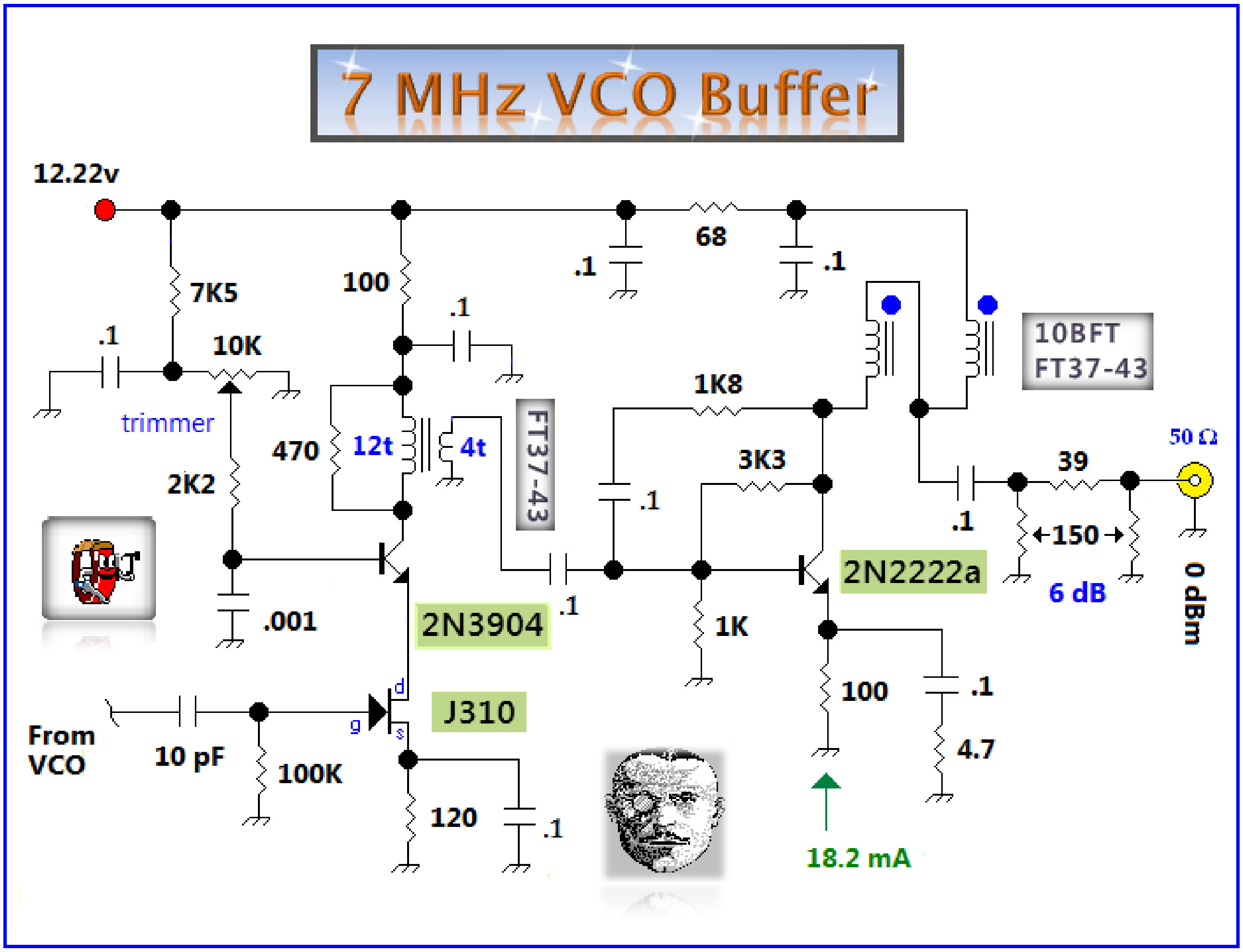


Above — a view of the square blue 1K temperature compensation (tempco) trimmer potentiometer.

To aid temperature compensation, I included 3 polystyrene capacitors in the base VCO — the tempco circuitry represents about 16 hours of work from December 2011. Click or click for photos of the bread board before the tempco parts were soldered on — the temporatry BNC connector lies in the background was removed after testing.

With care and patience my lid-on 1 hour temperature drift = ~10 Hz. My temperature compensation strategy worked because I took the time to measure and then determine how to cancel temperature drift in this 1 circuit — your results will vary and *experimentation* remains the key to temperature compensating VCOs and VFOs.

See EMRFD, and the VFO-2011 + QRP Modules 2011 web pages for more tempco information.

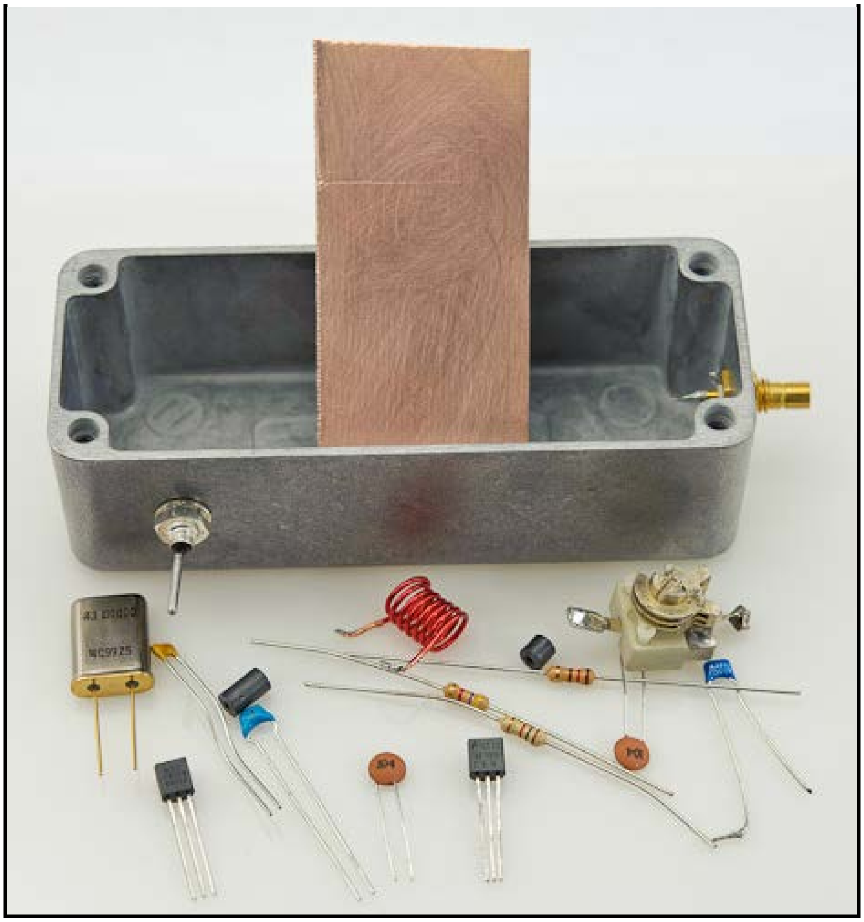


Above — 7 MHz VCO buffer/amplifier. I adjusted the 10K trimmer pot on the hycas buffer for exactly 0 dBm drive. Even before adding the 6 dB attenuator pad, the output return loss = 23.8 dB @ 7.0 MHz.

Originally, I wanted a VCO output of 7 dBm and applied ~18 mA emitter current in the final amp to preserve signal fidelity and eliminate the need for a low-pass filter. This buffer works great up to an output power of ~10 dBm: above 10 dBm or so, distortion occurs and you'll need to add a low-pass filter.

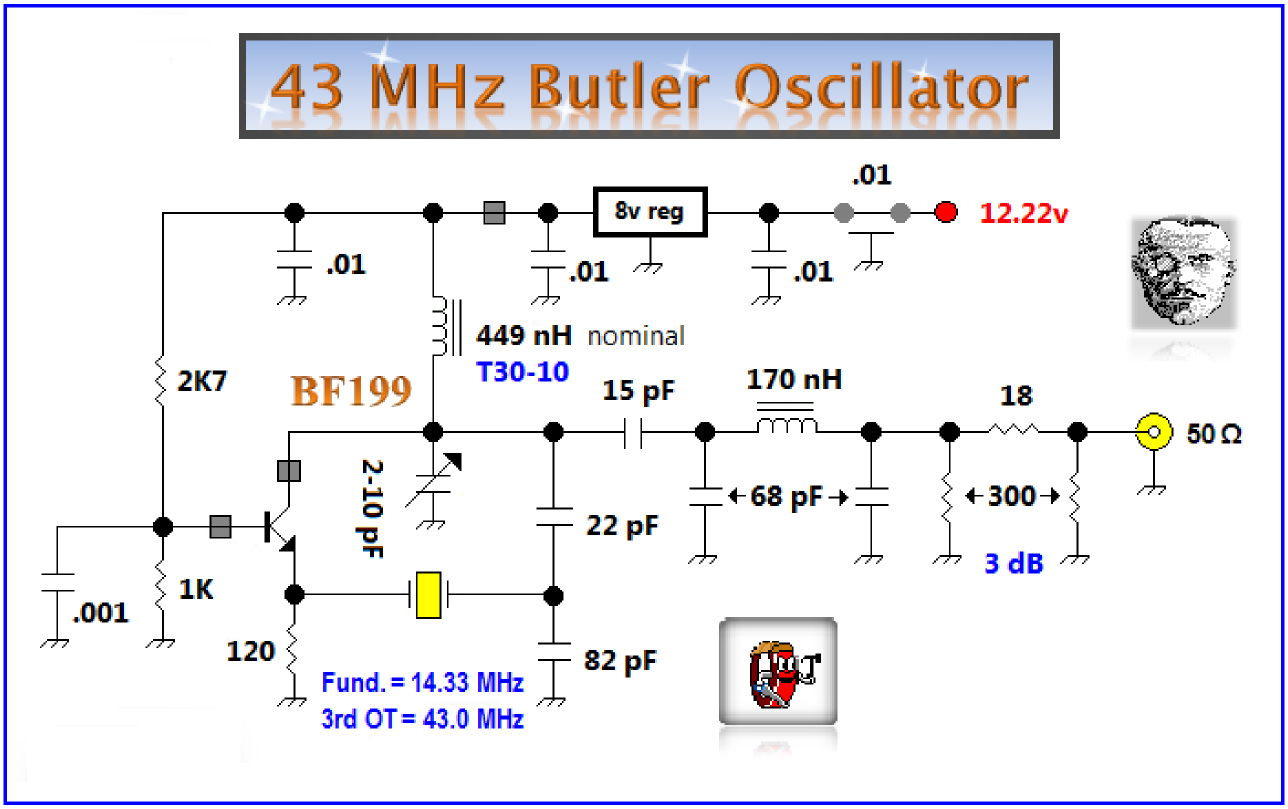
Adjust the hycas trimmer pot for whatever output power you seek, but If you're ever using this buffer for 7 -10 dBm output power, drop the 6 dB attenuator pad to 3 dB. This drops the drive level to maintain low harmonic distortion (2nd harmonic down > 35 dBc).

43 MHz Butler Xtal Oscillator [6.4 dBm output power]



Above — Some Butler crystal oscillator parts prior to the build. Since this Butler will go inside a box containing a VCO and some high gain amplifiers, it would be foolish to not stick it in an RF-tight box. On the Hammond chassis above, you'll notice a feedthrough capacitor for the 12 VDC and an gold colored SMA connector for the output.

Since the required L = over 400 nH, I opted for a toroidal inductor wound on a T30-10 instead of the air coil shown in the photo.



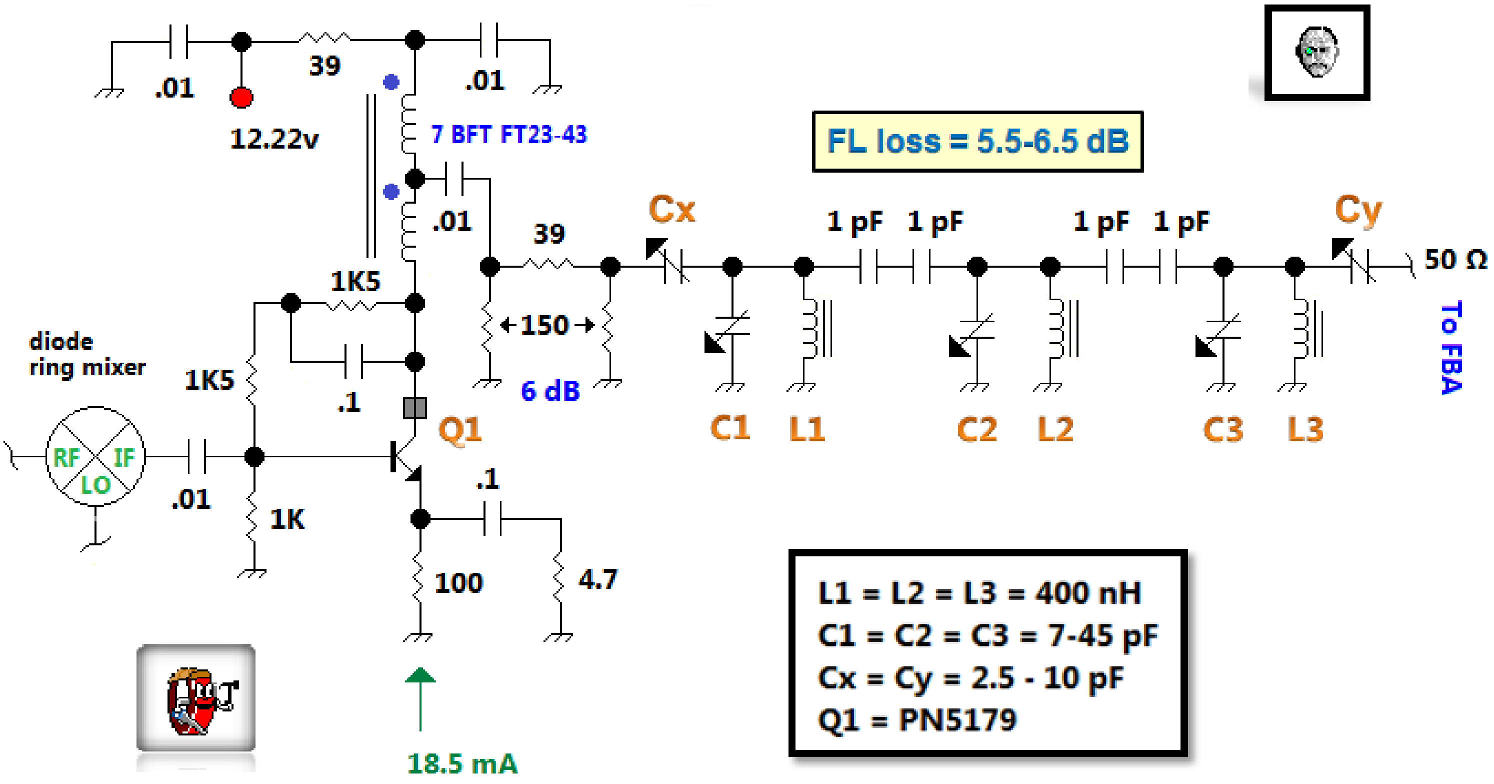
Above — 43 MHz Butler overtone oscillator schematic. The highest power I could muster = 6.4 dBm (close enough to 7 dBM).

This Butler looks good on FFT. Click, Click or click for 3 'scope captures. Despite trying to milk maximal power, the 2nd harmonic is over 40 dBc down. Click for a snap shot of the completed oscillator.

Above — The original Butler oscillator before adding the pi low-pass filter. The bolt (seen at top right) will also pass through the outer VCO chassis to hold this sub-chassis in place. Click for a bigger photograph.

Post-Mixer Amplifer and Triple-Tuned Filter

Post–mixer Amp and Band-pass Filter



Above — Schematic of the diode ring mixer, Q1 feedback amp and the triple-tuned filter. *I used a MCL SBL-1 mixer*. L1 - L3 were wound on T30-10 toroids. I bought my #10 and some #12 toroids from the great folks at Debco Electronics.

The post-mixer feedback amp data at 50.0 MHz (isolated from the mixer and pad + filter) : Emitter current = 18.5 mA, S21 =18 dB, S11 = - 24.4 dB, S22 = - 21.5 dB. *(S21 = power gain; S11 = negative of the input return loss; S22 = negative of the output return loss).*

The 6 dB pad helps absorb signal reflections from the filter caused from stray reactance plus capacitance variations caused by coupling the 2 tanks with only 0.5 pF (2 series 1 pF capacitors with a +/- 0.25 pF tolerance!)

Preliminary filter alignment: Peak your filter however you want — but here's how I peaked my filter with a crystal controlled 50.0 MHz signal generator connected via a temporary BNC connector tack soldered onto the copper board and wired to the Q1 input. Terminate the filter with a ~50 Ω resistor, or a temporary BNC connector plugged with a 50 Ω resistive terminator.

Connect the signal generator to Q1 and peak C1, C2 and C3 (in that order) using a 10X 'scope probe. It's better measure with your probe at point C2 when tuning C1 since this reduces mistuning caused by probe capacitance — measure at point C3 when tuning C2 etc.

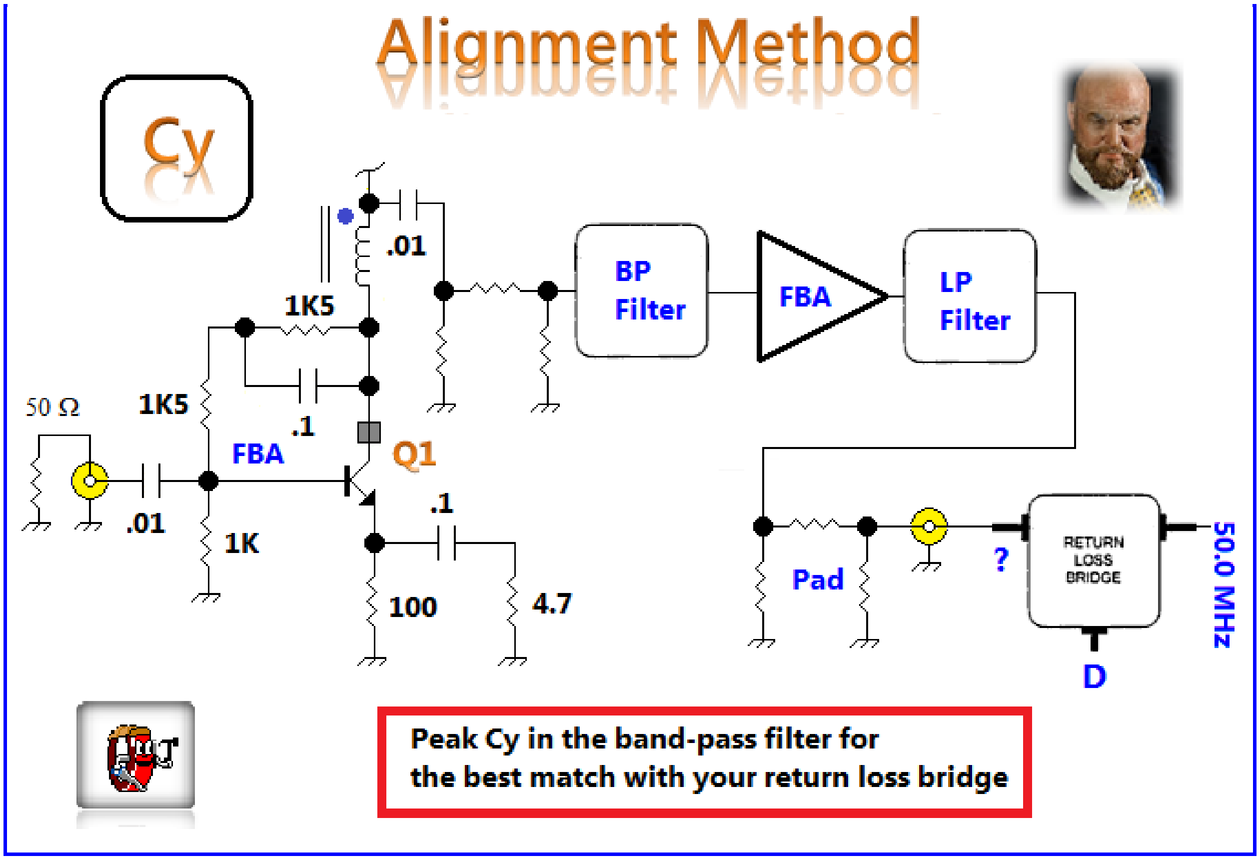
Then peak C3, Cx and Cy with the probe touching the terminating 50 Ω resistor. *It easier to perform the first tune-up with a 10X probe going sequentially from C1 to C3 since these peaking capacitors tune pin sharp.*

After the preliminary tune-up, if possible, connect a temporary BNC connector to the output and re-peak all the caps with a 50 Ω terminated scope; this boosts sensitivity and eliminates 'scope probe capacitance.

Perform pentultimate 50.0 Mhz alignment after you add the post-filter amp, low-pass filter and the 3 dB pad. Capacitor Cy critically sets the output return loss of Q2 and when properly matched, establishes a 50 Ω termination for the triple-tuned band-pass filter.

You can also match Cx by connecting a return loss bridge to the input of Q1 and terminating the RF chain with a 50 Ω resistor, although tuning Cx only changes the input S11 a little. In 1 experiment, I replaced Cx with 6 pF and it worked okay.

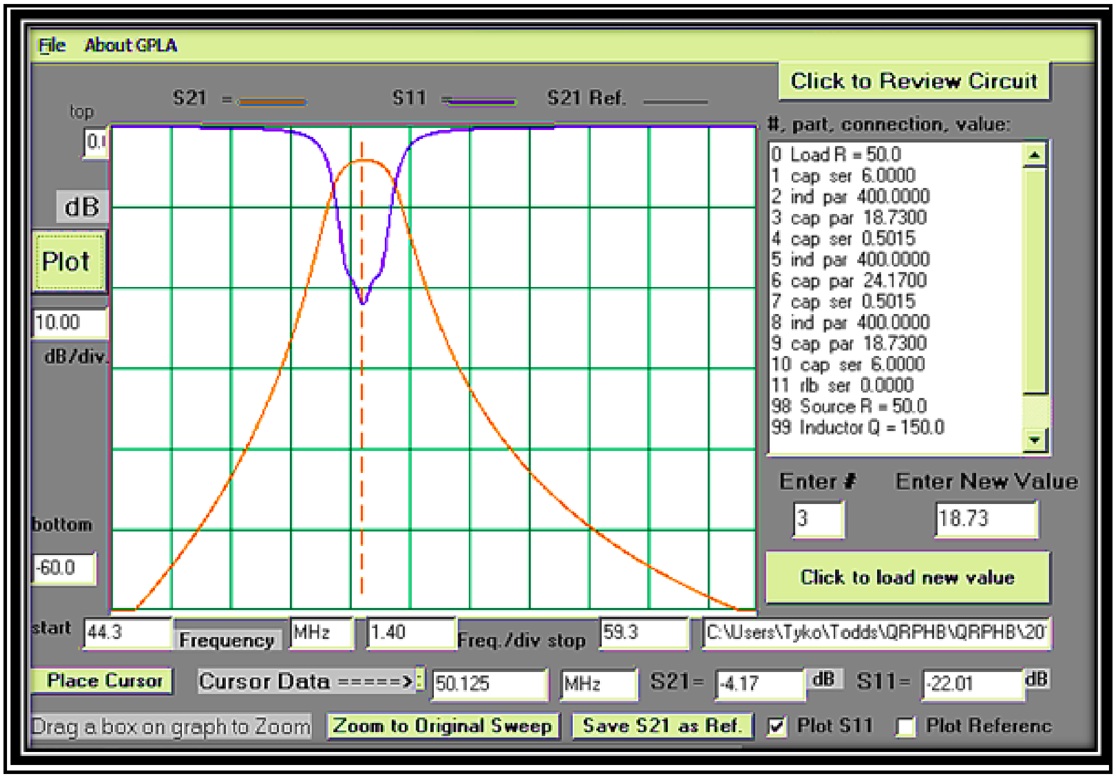
I wonder how I ever managed before making a return loss bridge: the workhorse of the QRP workbench.



Peak Cy in the band-pass filter for the best match with your return loss bridge

Above — Schematic depicting how to tune up Cy.

Tuning Cy matches the band-pass filter output to the Q2 input impedance — it's fascinating to examine the interdependence of these 50 Ω stages. After setting Cy, I connected the 50 MHz signal generator to the Q1 input and a 50 Ω terminated 'scope to the output and re-peaked Cx, C1, C2 and C3 — finally I tweaked Cy 1 last time with the whole stage in a return loss measurement set-up.



#, part, connection, value:

```
0  Load R = 50.0
1  cap  ser  6.0000
2  ind  par  400.0000
3  cap  par  18.7300
4  cap  ser  0.5015
5  ind  par  400.0000
6  cap  par  24.1700
7  cap  ser  0.5015
8  ind  par  400.0000
9  cap  par  18.7300
10 cap  ser  6.0000
11 rlb  ser  0.0000
98 Source R = 50.0
99 Inductor Q = 150.0
```

top 0.0

10.00 dB/div.

bottom -60.0

start 44.3 Frequency MHz 1.40 Freq./div stop 59.3 C:\Users\Tyko\Todds\QRPHB\QRPHB\20

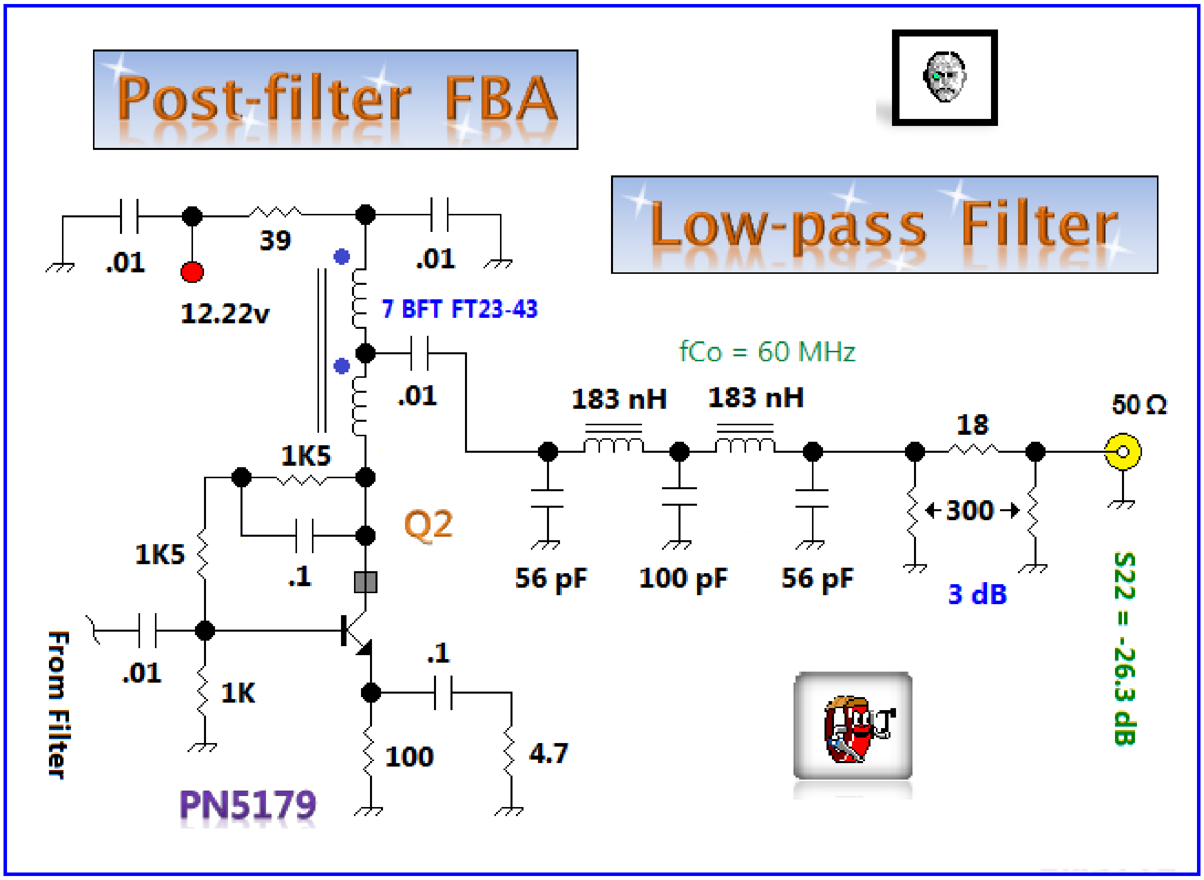
Enter # 3 Enter New Value 18.73

Cursor Data =====> 50.125 MHz S21= -4.17 dB S11= -22.01 dB

☑ Plot S11 ☐ Plot Referenc

Above — A GPLA simulation of the triple-tuned band-pass filter. CF = 50.125 MHz. I substituted 6 pF (the nominal value) for the 2 series end capacitors in my simulation. отлично!

Post-filter Feedback Amplifer, Low-pass Filter and Pad



Above — Q2, the post filter feedback amp (FBA), an N=5 Chebyshev low-pass filter plus a 3 dB pad.

For Q2, I copied Q1 to deliver a strong input and output return loss. In many circuits employing cascaded FBAs, you increase emitter current in each successive FBA to reduce distortion, however, increasing emitter current affects both the input and output impedance and may trash your amplifier's S11 and S22.

I spent days studying, simulating + bench testing different amplifier designs in the Q2 slot — I generated enough material for another web page and plan to show this work in an update to my Popcorn superhet receiver some day.

It's possible to overdrive Q2 depending on your amplifer power and stage matching. If so, you might consider placing a 3-4 dB pad after the band-pass filter. Some might opt for a 7 element low-pass filter; experiment — as always.

Low-pass filter inductors = turns on T30-10 toroids, although #6 material toroids, or air coils will work fine.

Outputs

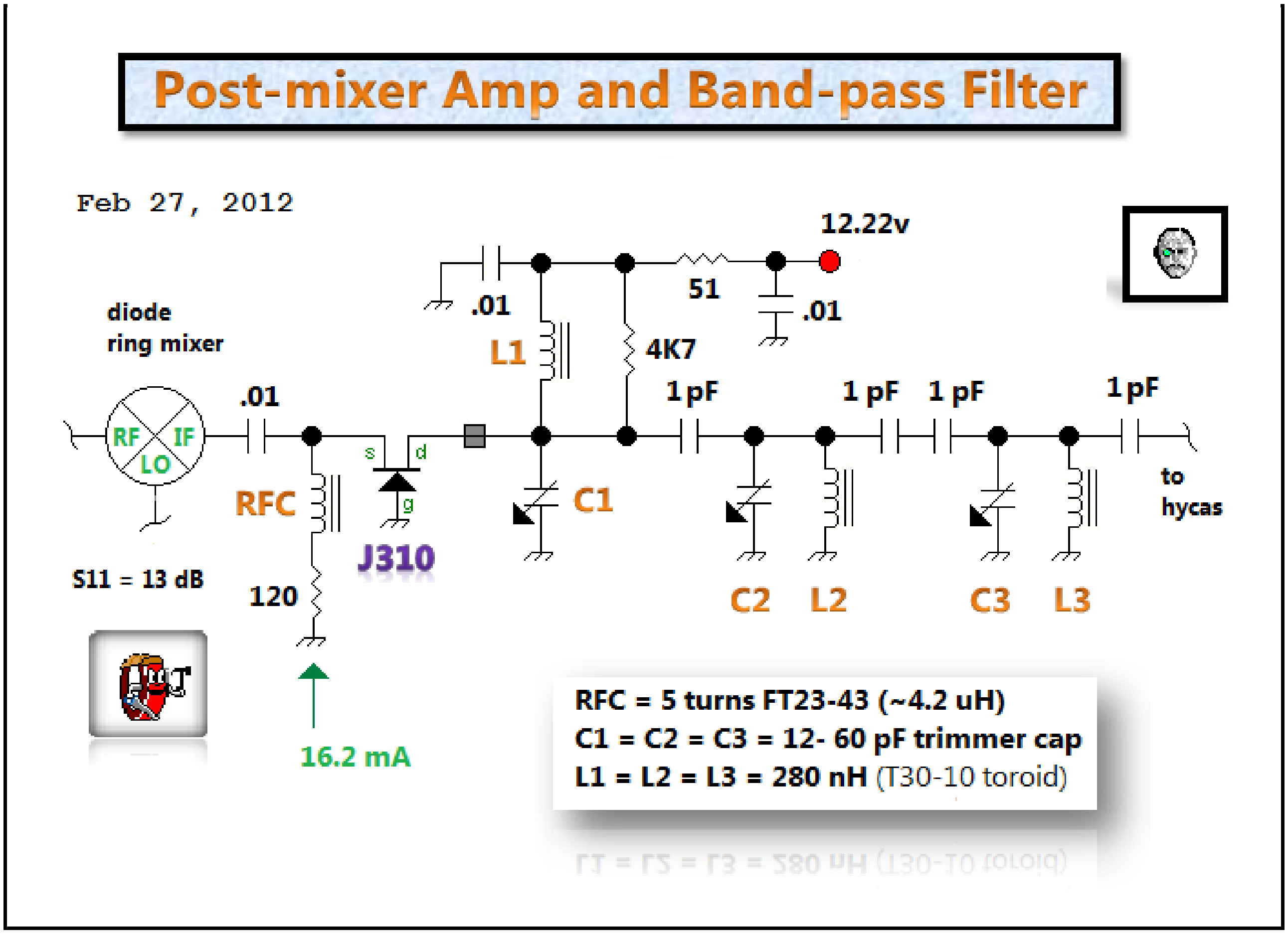
After bolting down the boards, wiring the DC and RF and confirming it worked, I finalized alignment. Using a frequency counter, I tuned the VCO to 50.125 MHz (the half-way point) and peaked C1, C2, C3 for the maximum peak-peak voltage into my 50 Ω terminated 'scope.

Click for the output at 50.125 MHz — 10.09 dBm. I normally hang an outboard 50 Ω attenuator on the output of my signal generators and keep 3 dB, 6 dB, 10 dB and 20 dB BNC-connected pads handy. With a 3 dB pad, the output power = 6.84 dBm — perfect for switching Level 7 diode ring mixers. Click for the 'scope shot with a 3 dB pad applied.

Click or click for an FFT of the output signal. The second harmonic is > 50 dB down. What fun!

The "vestigial" RF gain control shown on the chassis remains unused; wastage.

Miscellaneous Photos and Figures

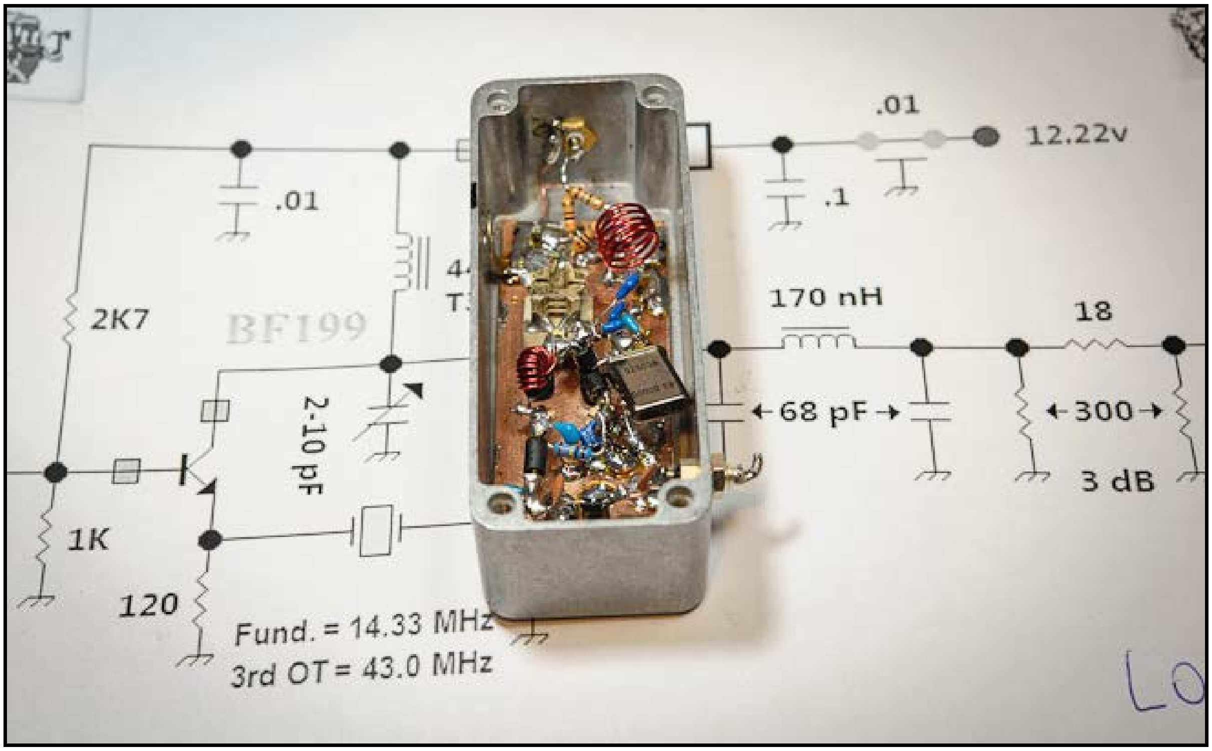


Above — A **failed** experimental JFET post-mixer amp with tuned output driving a double-tuned filter.

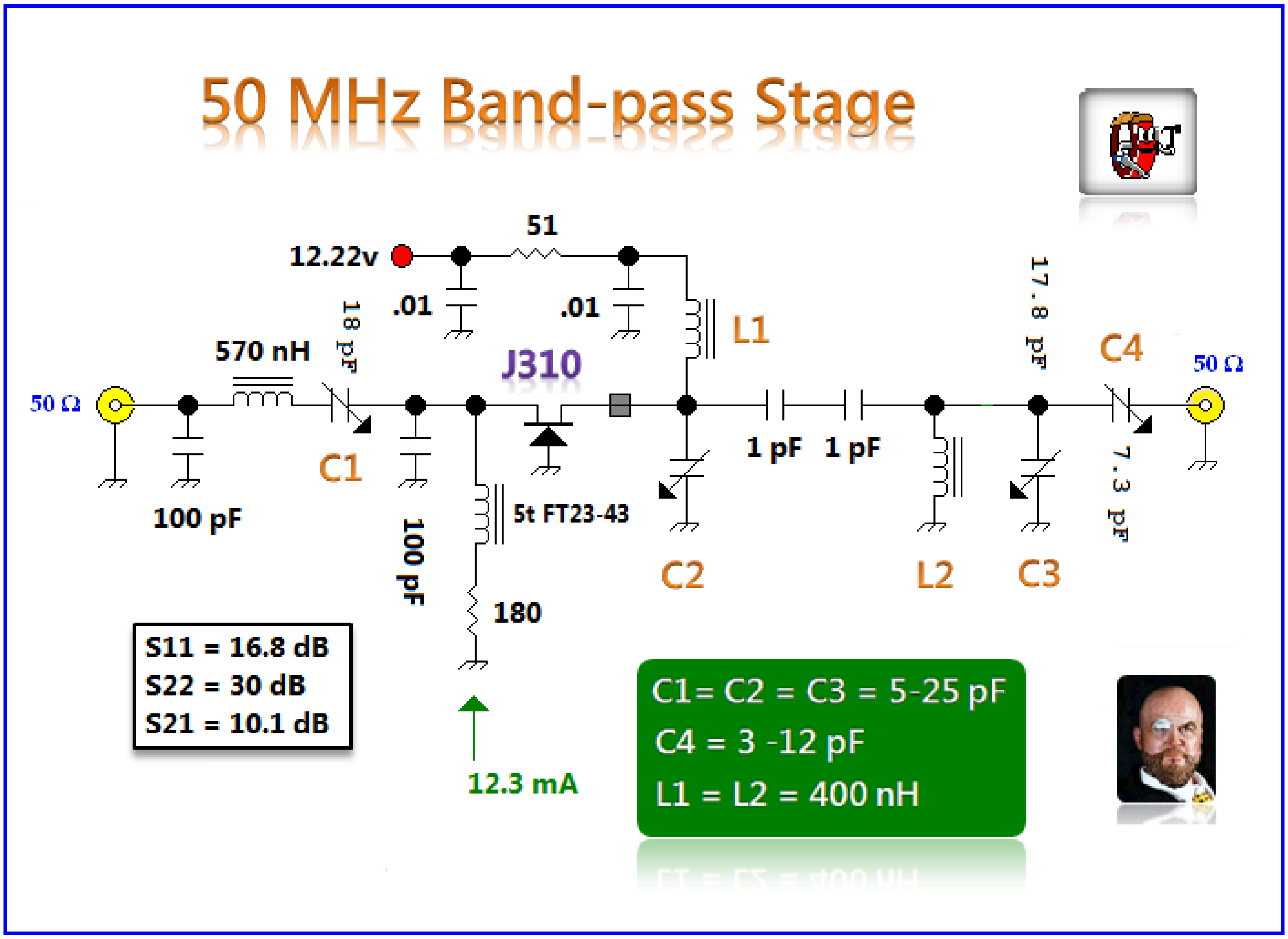
Click for a GPLA simulation of the double-tuned filter. The common gate JFET amp provides a great way to terminate a diode ring and obviates the need for a diplexer network. Click for a breadboard photo of the above stage.

The amplifier input match @ 50 MHz is only ~ 13 dB, however, we're not interested in a narrow band match — the tuned output network makes strong input matching at 50 MHz impossible *(for me at least)* without additional L and C (narrow band components that we don't want!). I tried a few tapped inductor schemes, however, at VHF, adding turns added significant capacitance and things got ugly fast.

The common gate JFET amp/filter goof-up shattered my expectations. The 4K7 input/output impedance drove instability through unwanted coupling between the inductors. I learned my lesson: at or above 50 MHz, *stick to 50 Ω stages* for stability.



Section 4. 50 MHz Receiver Pre-amp and Filter


50 MHz Band-pass Stage

S11 = 16.8 dB
S22 = 30 dB
S21 = 10.1 dB

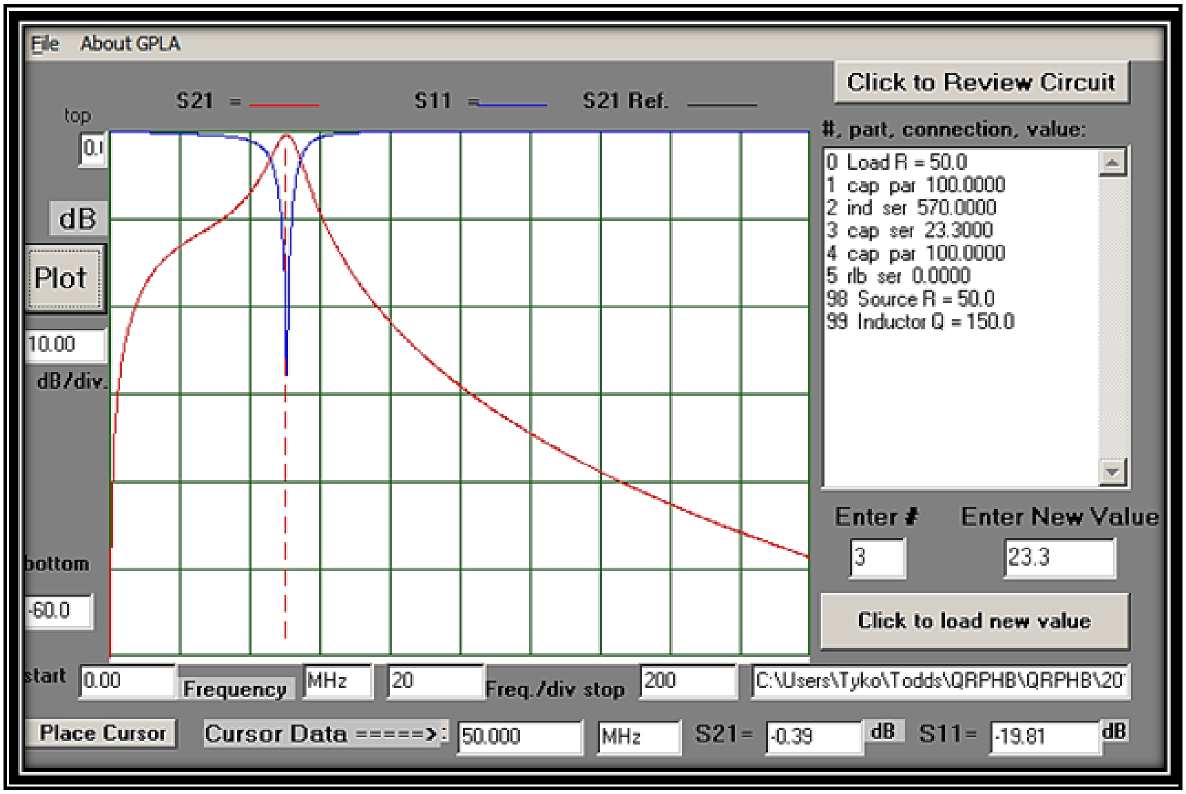
C1= C2 = C3 = 5-25 pF
C4 = 3 -12 pF
L1 = L2 = 400 nH

Above — A 50 MHz receiver front end filter with embedded common gate amplifer.

Inspired by the General Purpose Monoband Receiver Front End from Figure 6.69 in EMRFD, I applied inductive and capacitive reactance modeling, DTC08, Ladbuild08 and GPLA08 from the EMRFD ladpac series and built a 50 MHz equivalent.

Connect an antenna to the input and a 50 Ω impedance mixer to the output.

I tested the stage at 50.0 MHz and wound my inductors on T30-10 toroids, although #6 material cores would work okay. You'll find all the measurement techniques in my RF Workbench series 1-4 available though the top-level menu.



#, part, connection, value:

0 Load R = 50.0
1 cap par 100.0000
2 ind ser 570.0000
3 cap ser 23.3000
4 cap par 100.0000
5 rlb ser 0.0000
98 Source R = 50.0
99 Inductor Q = 150.0

top: 0.0
dB/div.: 10.00
bottom: -60.0
Enter #: 3
Enter New Value: 23.3
start: 0.00 MHz
Freq./div: 20
stop: 200
C:\Users\Tyko\Todds\QRPHB\QRPHB\20
Cursor Data =====> 50.000 MHz
S21= -0.39 dB
S11= -19.81 dB

Above — GPLA simulation of the peaked low-pass filter "built" in Ladbuild08.

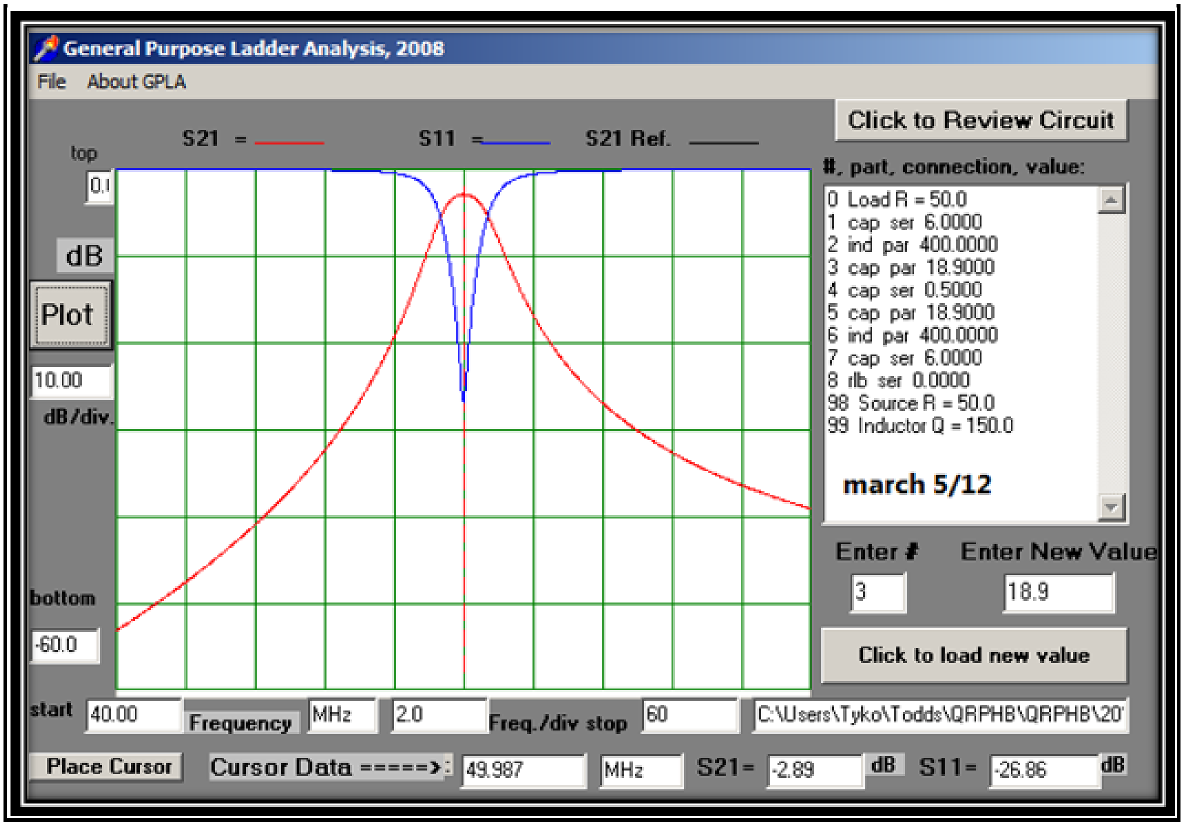
Wes often employs a peaked low-pass filter and after studying his work, I can see why — way better attenuation than a simple 3 element low-pass filter. The FM broadcast band runs from about 87.5 to 108 MHz and in Russia, they call it "YKB" (Ультракороткие волны) or ultra-shortwave. At 87.5 MHz, attenuation = 25 dB; pretty good for such a simple filter. At 144 MHz, filter attenuation rises to ~ 40 dB.

This peaked low-pass filter acts as a preselector for the JFET amp that follows it. *Please read the text describing Figure 6.69 in EMRFD for some great notes by Wes.*

In the simulation above, a 50.0 MHz peak response occurred with C1 at 23.3 pF, while in my real circuit, the capacitor was set to ~ 18 pF. Stray L and C + the input Z of the JFET amp caused this variance, but assuredly; GPLA gets you close.

To peak the low-pass filter, I connected a return loss bridge to the input port and tweaked C1 for the lowest possible peak-peak voltage (tuned for the the best return loss which = 16.8 dB in my circuit). *You may also compress or expand the 540 nH inductor to aid tweaking.*

Since common gate amplifiers often exhibit a lower noise figure with a slight mismatch, an S11 of -16.8 dB works fine .I wish I had the gear to set the input match for the lowest possible noise figure — perhaps 1 day I will.



#, part, connection, value:

0 Load R = 50.0
1 cap ser 6.0000
2 ind par 400.0000
3 cap par 18.9000
4 cap ser 0.5000
5 cap par 18.9000
6 ind par 400.0000
7 cap ser 6.0000
8 rlb ser 0.0000
98 Source R = 50.0
99 Inductor Q = 150.0

march 5/12

top 0.0

10.00 dB/div.

bottom -60.0

start 40.00 MHz

Freq./div 2.0

stop 60

C:\Users\Tyko\Todds\QRPHB\QRPHB\20

Enter # 3

Enter New Value 18.9

Cursor Data =====> 49.987 MHz S21= -2.89 dB S11= -26.86 dB

Above — A GPLA simulation of the 50 MHz double tuned band-pass filter "built" in DTC08. The bandwidth = ~1.8 MHz and varies slightly with the tuning of C4.

I peaked both C2 and C3 with a 50 Ω signal generator and a 50 Ω terminated scope connected to the input and output ports respectively.

Next, I connected my return loss bridge to the output and tweaked C3 and C4 for the lowest peak-peak voltage — the best return loss — and since you tweak 2 capacitors, a strong output return loss delights you.

Finally, I measured the peak-peak voltage with the amp in-line, and after removed the amp and reconnected the 50 Ω cables with a through-connector. Inputting the 2 pk-pk voltages into Applet H on the Design Center web page gave a gain *or S21* of 10.1 dB. I repeated all of the steps above a couple more times to ensure I had set C2, C3 and C4 perfectly.

I found tuning the resonators difficult due to the sharp tuning and wide capacitance range of C1-C3. Assuming your tanks are peaked, the best amplifier gain correlated to the highest input and output port return loss. Have I stressed the importance of a return loss bridge enough?

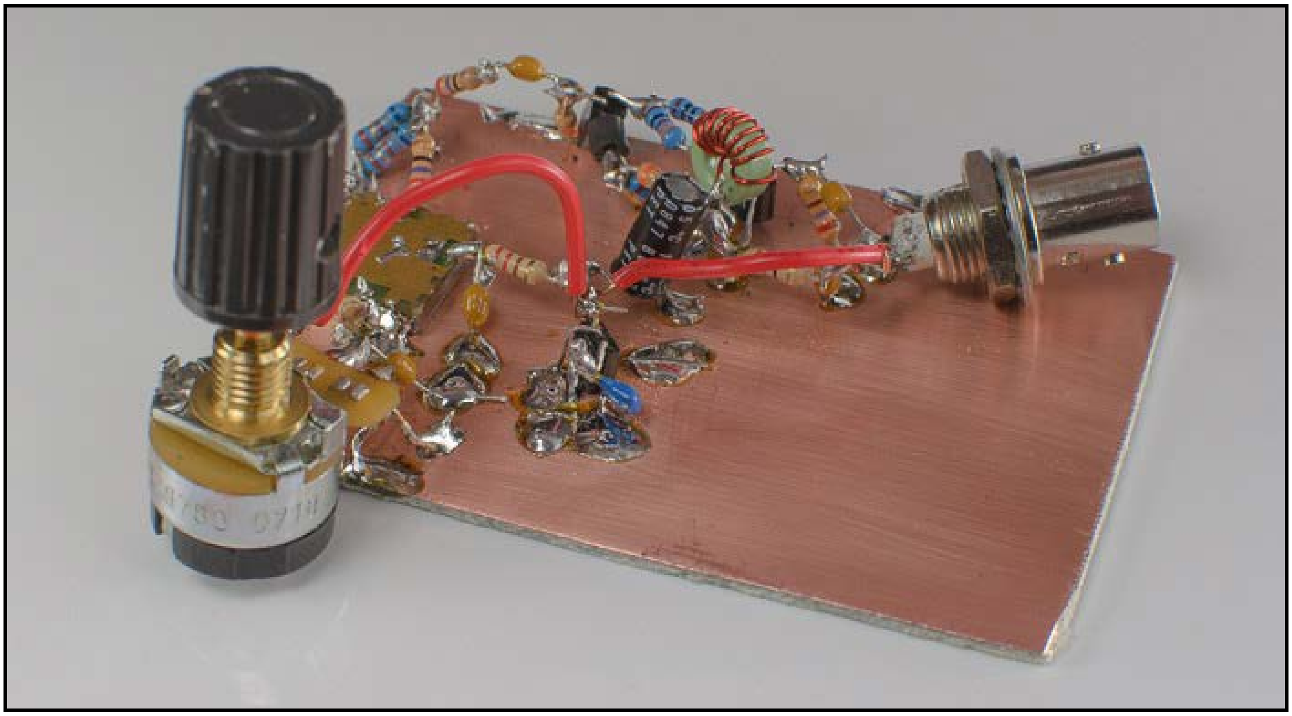
10.1 dB gain should be enough gain for listening to terrestrial 6 Meter band signals with my 5 element Yagi antenna.

Above — A photo of my protoype 50 MHz pre-amp breadboard. In my "keeper" version, I'll swap in a U310 JFET and bias it for ~15 mA.

Section 5. QRP — POSDATA: Z-Communications VCO Experiment

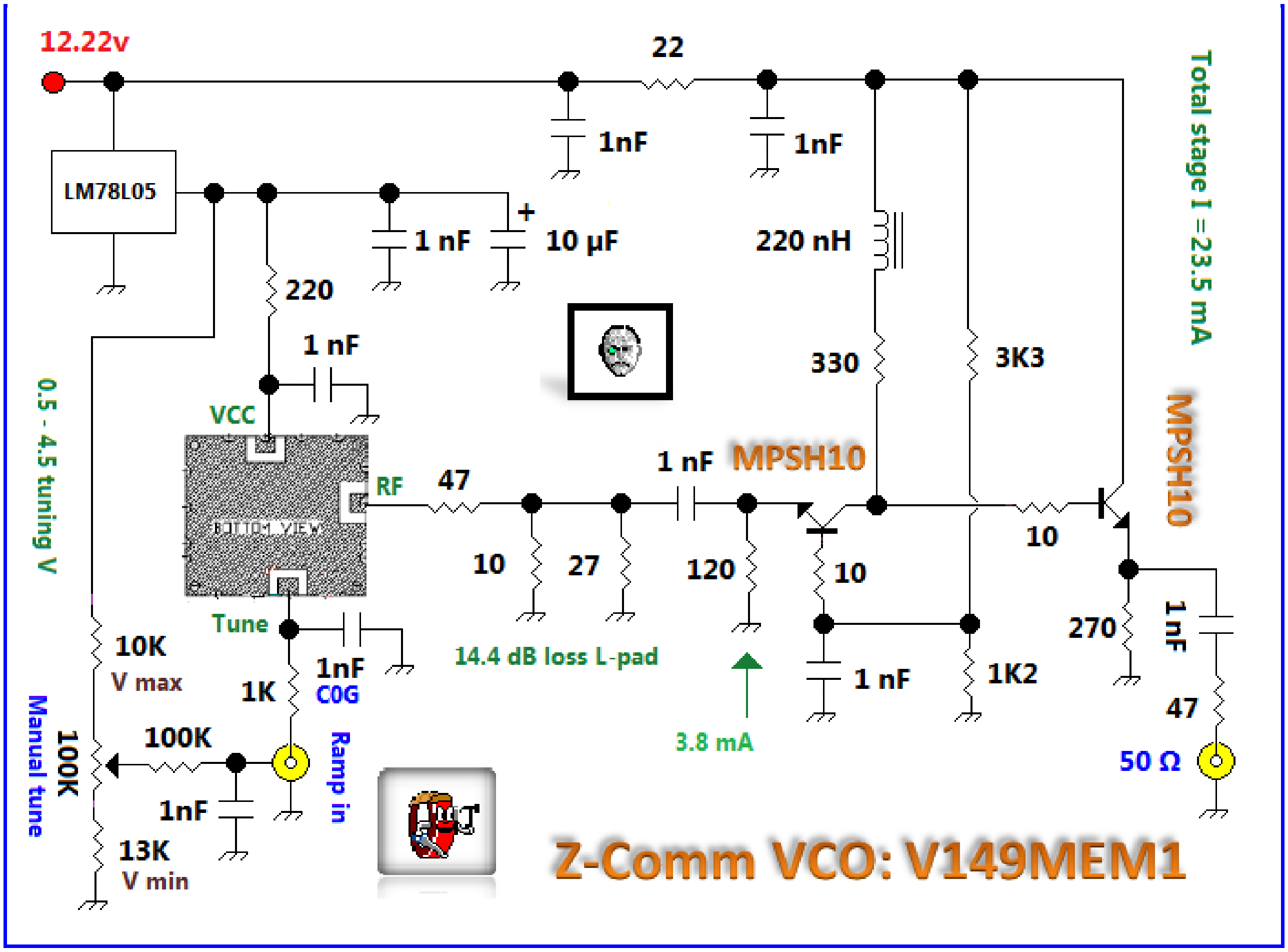
Looking on eBay, sellers list numerous VCOs, although most are surface mount and go well above VHF. My favorite VCO comes from Mini-Circuits Labs: the POS series. Click for an example: the POS-75. These "plug- in" VCOs come in same package as the SBL-1 mixer and are likely obsolete, but still for sale. If you're building a frequency synthesizer with low phase noise requirements, MCL VCOs seem hard to beat. You can still order them from MCL, but the high product and shipping costs might alarm you.

I've looked for cheaper alternatives and the Z-Comm VCO raises 1 possibility. Last year, I purchased a V149MEM1 device for 5 dollars including shipping. Some experiments follow:



Above — My first breadboard. Lacking the MINI-16 receptable, like with MCL POS VCOs: I turned it upside down and soldered the metal case to my ground plane. If I were to keep this circuit. I wound solder all 4 sides to the copper clad board, plus run some copper de-soldering brade from the bottom to the ground plane, or even cut a square hole and flush mounted the VCO on is back.

While mounting it upside down deviates from the recommendations found on the Z-Comm mounting datasheet, I fiigured that for VHF at least, it might work okay. We desire low inductance grounding, but creativity might allow dead bug construction techniques to work.



Above — My complete VCO. The Z-comm VCOs require at least a 10 dB pad on the output to keep port return loss high. Without a pad, you might see something like this plus boost the phase noise. In my circuit, I applied a resistor L-network with ~ 14.3 dB loss to pad the output and provide a match into a common base amp with an input impedance of ~ 6.8 Ω.

The 2-stage buffer is the brainchild of Bob, K3NHI and I love it. This buffer features a common base stage driving a emitter follower yielding high bandwidth and great reverse isolation. Normally, at VHF, the buffer is followed by more such stage(s), or a MMIC. The 220 nH inductor wound on a T30-12 toroid improves the high frequency response of the common base amplifer — experiment with this L to suit whatever VCO you wish to buffer. The gain of the 2 amp buffer is typically around 9 dB and the return loss at the input and output ports lies under 11 dB, so apply attenuator pads to boost S11/S22 as required.

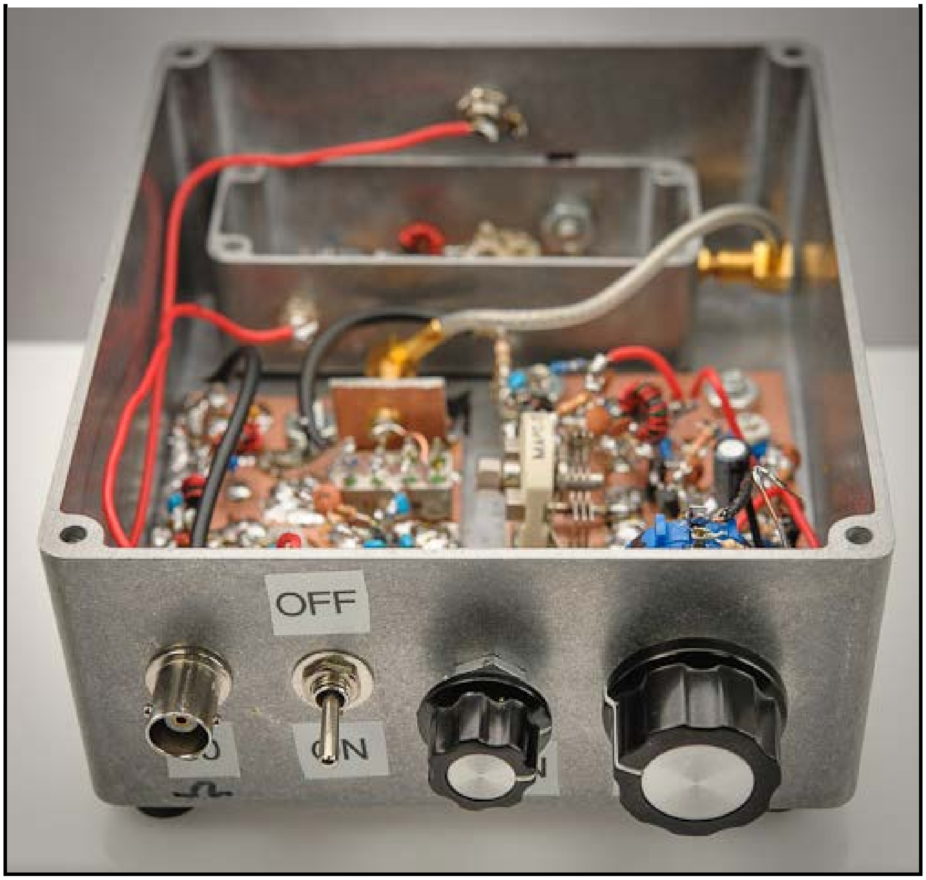
Click for the scope tracing at 0.5 VDC tuning voltage. Click for the 4.5 VDC tuning voltage 'scope tracing. The harmonic distortion at the lowest tuning voltage = ~ -19 dBc and decreases to -28 dBc at the highest tuning voltage; better than specified. Notice that power decreases as frequency increases. All the commercial VCOs I tested do this. A higher fT amp like the PN5179 or other BJT might be a better choice to offset the power change versus frequency *contribution* of the buffer/amp.

For a sweep circuit, I would mix this VCO with another low level, single frequency VCO with its current controlled by a downstream leveling circut to derive a flat amplitude over the range of the VCO. I plan to try the Z- Comm V150S015 in such an arrangement to make a 70 - 150 MHz VCO for sweeping.

Please refer to the datasheet for the pin out on the Z-Comm VCO: I chose the pinout shown in the schematic to make an efficient drawing. The two 10 Ω resistors in the buffer/amp snub UHF oscillations first measured by Bob and confirmed by me. Ferrite beads might work as alternates.

Section 6. Miscellaneous Photos

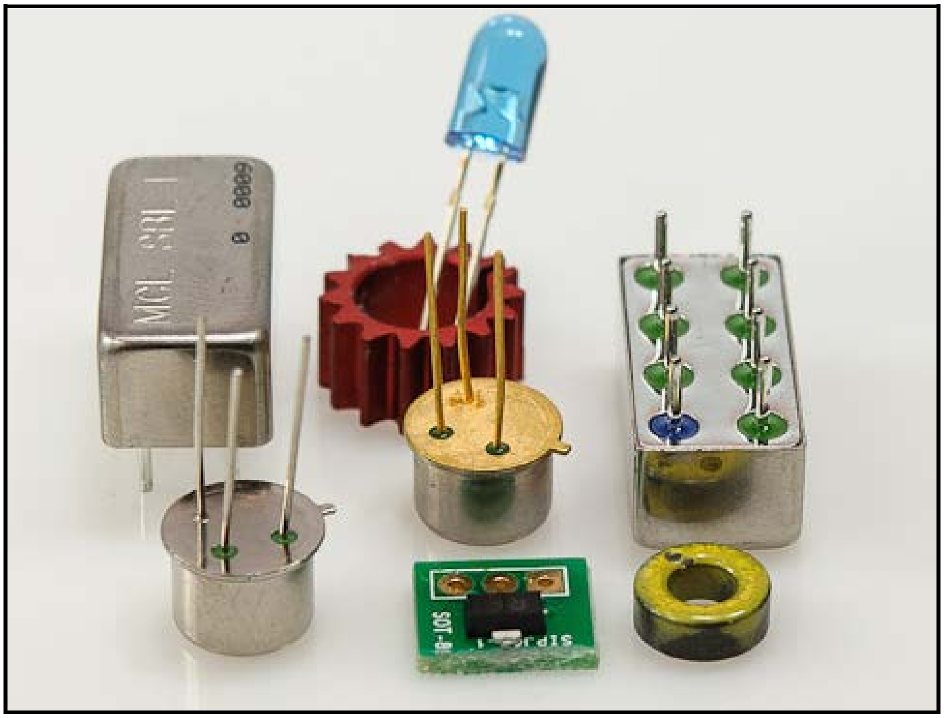

VHF SIGNAL GENERATOR
138-172 MHz
T
T
98-109 MHZ
ON | ON
OFF
50Ω
50Ω


OFF
ON


MALE TERMINATOR 50Ω

RF — Test and Measurement

HF Ragbag



It's easier to present short topics on catch-all web pages — HF Ragbag shows some 2012 non-VHF experiments in no particular order. I also share thoughts on circuit building and writing: we can think and work better.

1. Comments from the Workbench - The Need for Clarity

On Building

In 2012, I boosted my circuit and writing quality to improve your experience: a genuine, return-to-basics approach in amateur, component-level electronic design. As possible, RF circuits will feature 50 Ω input and output ports — totally adopting a 50 Ω environment — for I'm convinced this is the best way to go. The 50 Ω building and measuring standard offers much: an easy-to-interface modular approach; 10 dB improved sensitivity over a 10X 'scope probe and if wanted, measurement with commercial or homebrew test equipment such as a spectrum analyzer, network analyzer or RF power meter.

Like many, I started out by collecting and copying circuits with little emphasis on true understanding. I wanted a completed circuit — quickly as possible — failing to develop my design skills. Without design skills honed by studying and properly measuring our circuits, we bide in hit-and-miss electronics — a frustrating repetition of trial and error, over and over again. We ought to adopt the attitude and thinking of engineers while keeping our design work —including the math — fun. However, embracing scratch-homebrew electronics with the overall goal of trying to understand each stage takes effort. "There is no substitute for hard work" wrote Thomas A. Edison. Scratch homebrew involves reading, simulating, collecting parts, mastering new techniques and building or buying test equipment.

This is more than knack, an abused noun that often means "hack". Our key tasks: to measure, analyze and understand the circuits we copy or create takes patience and practice. Dissecting circuits to understand their function means to hypothesize and reflect — to apply science on paper, with software, and finally, through careful bench experiments. Often we lack the math skills or test equipment to fully investigate some aspects of our circuits, but try our best: measure what we can measure, seek help and grow. I hope this site shows our hobby can be less about making stuff and more about the rewards of actual design: an authentic, personal journey to get better at something you love.

I've never been much of a kit builder; it's too much like Max Klein's Paint by Number for my tastes. But kits dominate HF QRP homebrew and may offer a cost effective way to make gear; especially test gear. Stuffing parts in a printed circuit board won't teach you much about design, but might get your feet wet. Some people remain perfectly happy building kits or madly copying circuits — all the power to you! Do whatever you want. One day you might awaken, but don't worry; I won't try to goad, or convince you.

My favorite builders include people over 50 who suffer the often crippling symptoms of 'appliance apathy' — an epiphany reminds them why they first got into radio: homebrew experiences. Maybe a crystal radio set, or a simple superhet receiver they breadboarded long ago. *Then they come back full circle*; like a loop antenna. Oh-boy — "Bob" rediscovered his radio roots and needs to unleash his creativity and passion to learn and improve. I write for people like Bob. Heck; I am Bob.

On Writing

You'll notice improved narrative writing too: I prefer to read and write crisp statements in short sentences and paragraphs. Brief, yet descriptive text accompanied by ample white space, clear headings and bulleted lists invites you to read on. Plain language writing — simple, clear, writing that is easy to read and understand — signals a refreshing move away from the turgid, word-filled claptrap I learned in grade school. Making your prose easier to read requires greater effort writing and re-writing. My first..to...fourth drafts always suck.

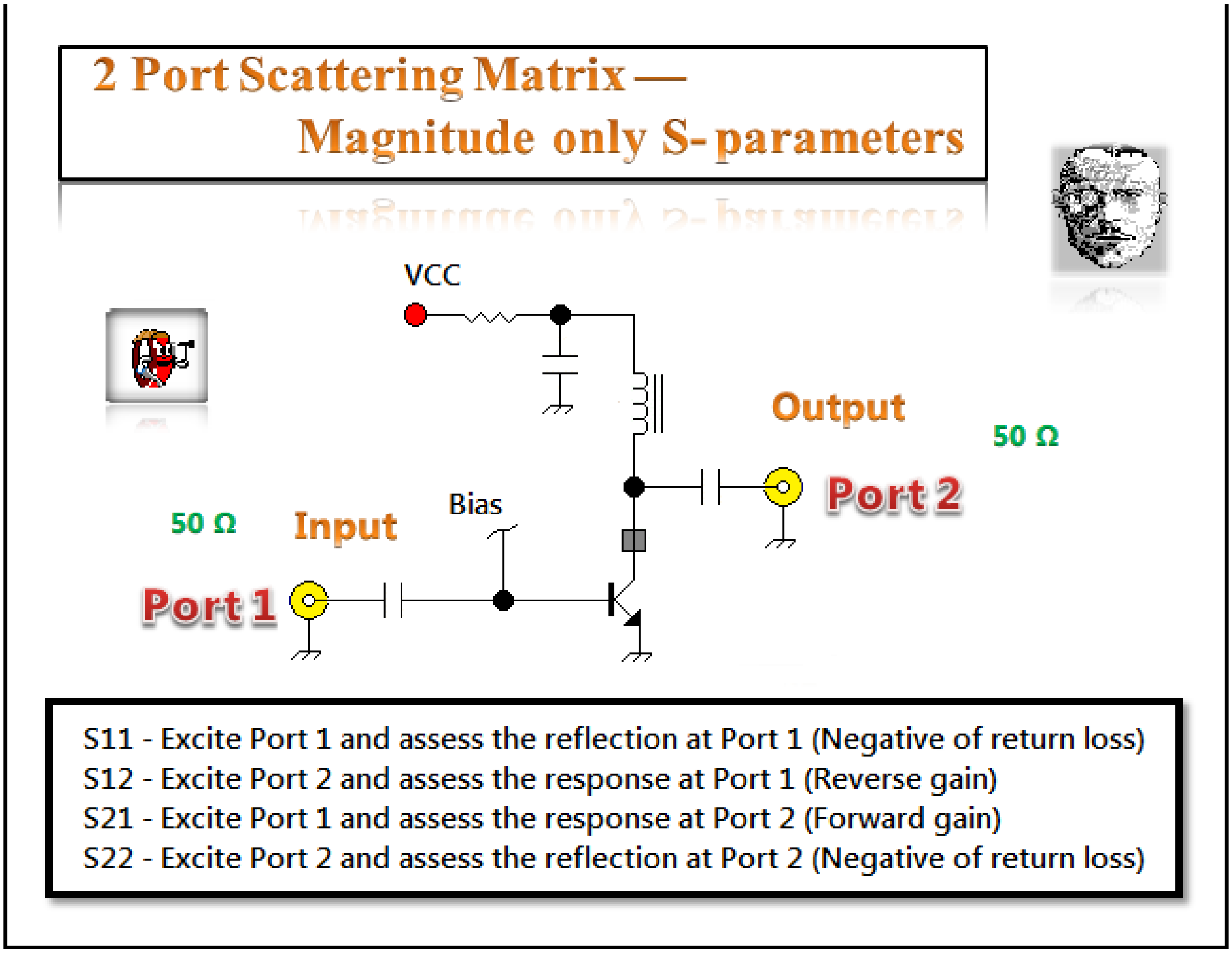
Passive verbs, or nouns and adjectives that function as verbs with no clear subject confuses readers and boosts wordiness: I employ active verbs to invigorate my writing — active verbs connote me or some else performing an activity you can visualize or feel. Actions that may inspire, persuade, or even vex you! Ours' is an emotional hobby.

RF electronics contains rich amounts of jargon. Of course, we must learn some jargon to communicate our ideas as hobbyists, but writing jargon to impress, or to place yourself above others lacks humility and alienates people. Do you know anyone who likes being talked down to? The first step towards becoming humble is to admit you're not humble and then work on it — and I'm working on it.

Although I enjoy writing about electronic experiments, I'm not sure it's worthwhile — Does anyone actually design circuits anymore? Well, back to my 1970's-style analog experiments...

---

2. Magnitude Only Scattering-Parameters


2 Port Scattering Matrix — Magnitude only S-parameters

S11 - Excite Port 1 and assess the reflection at Port 1 (Negative of return loss)
S12 - Excite Port 2 and assess the response at Port 1 (Reverse gain)
S21 - Excite Port 1 and assess the response at Port 2 (Forward gain)
S22 - Excite Port 2 and assess the reflection at Port 2 (Negative of return loss)

Above — A simple model describing the S-parameters displayed on QRP / SWL HomeBuilder in a Class A amplifer with 50 Ω ports.

S-Parameters

Any device with 2 connectors may be modelled at AC for a specified frequency with just 4 scattering parameters: forward gain, reverse gain + input and output impedance (match or VSWR).

S-parameters address voltage ratios: comparing the amplitude of different signals at the 2 ports. For example, S21 is the magnitude of forward gain and equals the ratio of output voltage to input voltage.

S-parameters are vectors; a mathematical quantity that may be visualized as an arrow anchored at 1 end that pivots around its base. The length of the arrow represents magnitude while the angle it makes with another vector or its base line decribes its phase in degrees. In addition to phase and magnitude, S-parameters allow analysis of gain, stability, complex impedance (resistance + reactance), admittance and other vector quantities.

Measure S-parameters with all ports terminated in a 50 Ω impedance.

Some of us worry only about the gain, losses or "match" in our 50 Ω circuits and could care less how the signal phase changes as it passes through our amplifiers or attenuators. I express only S-parameter magnitude in logarithmic form (dB) and take this *Über simplified* approach because builders can easily measure S11, S12, S21 and S22 on a 50 Ω test bench with a small staple of bench accessories + a 50 Ω 'scope or detector.

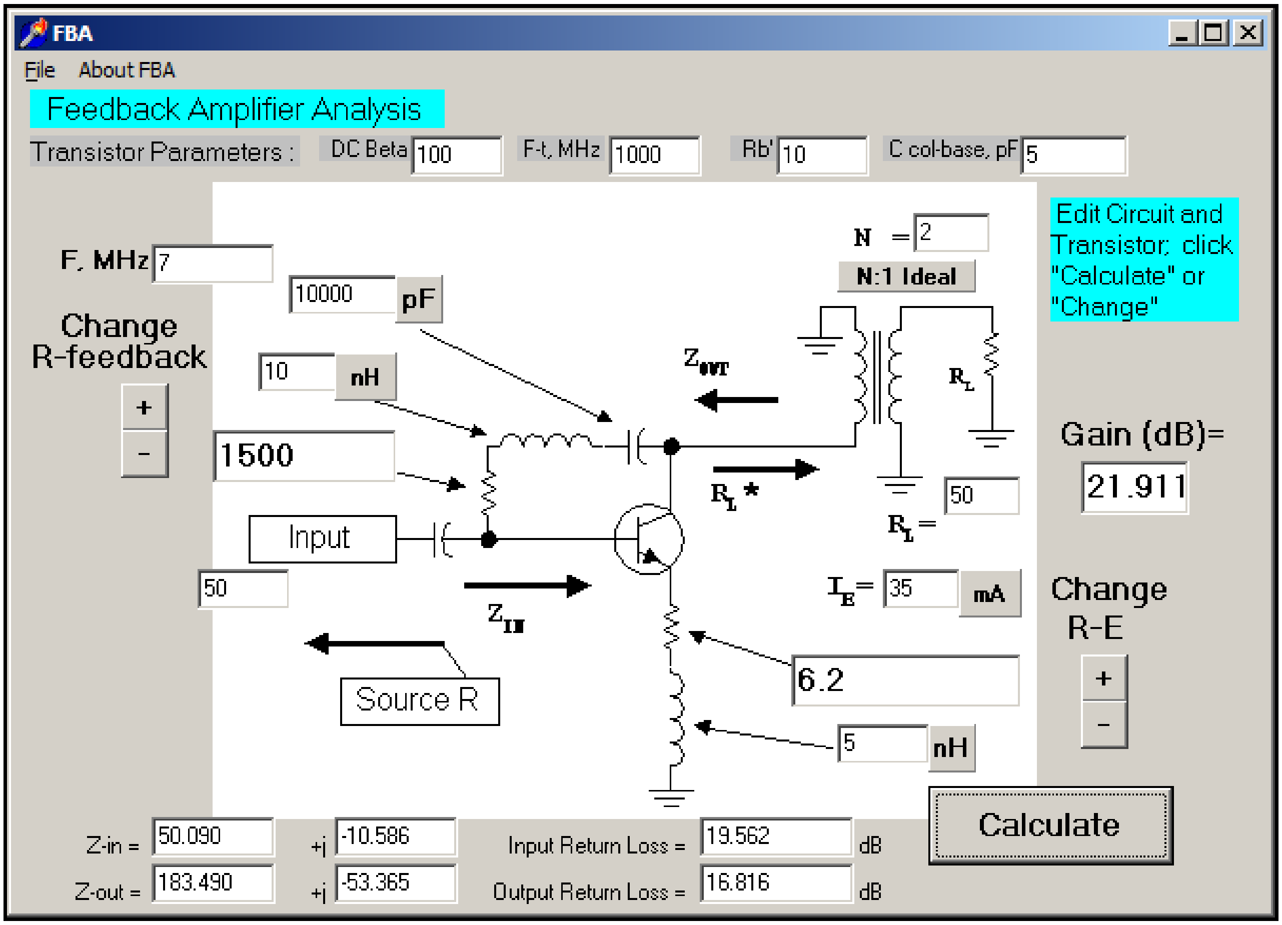
Topics like matrix theory, vector math, the "jay" operator, converting S-parameters into other matrices, Smith charts etc. may turn *off* the average amateur designer. You advanced readers, may raise your 2 port network skills by visiting better web sites + reading books, simulating with SPICE, or better yet, measuring your port parameters with a vector network analyzer.

---

3. More on Feedback Amplifiers (FBA)

Many builders (myself included) copy feedback amps rather than design their own. By tweaking the emitter current, shunt and series feedback while measuring S11 and S22, plus simulating with a program called FBA08.exe, I've learned it's possible to design good feedback amps FBA08 is 1 of the Ladpac programs that ships with EMRFD.

I wanted a FBA with ~35 mA emitter current for improved IMD and low distortion on strong signals. Such an amp might follow a diode ring mixer in a receiver I.F. chain.



DC Beta 100

F-t, MHz 1000

Rb' 10

C col-base, pF 5

F, MHz 7

10000 pF

10 nH

1500

50

N = 2

R L = 50

I E = 35 mA

6.2

5 nH

Gain (dB)= 21.911

Z-in = 50.090 +j -10.586

Input Return Loss = 19.562 dB

Z-out = 183.490 +j -53.365

Output Return Loss = 16.816 dB

Above — My 7 MHz FBA set up. Wes, W7ZOI suggested using 5 nH as the default emitter inductance and 10 nH for the default collector to base inductance in FBA08. These represent stray inductances in your circuit breadboard. Emitter inductance affects the input impedance more.

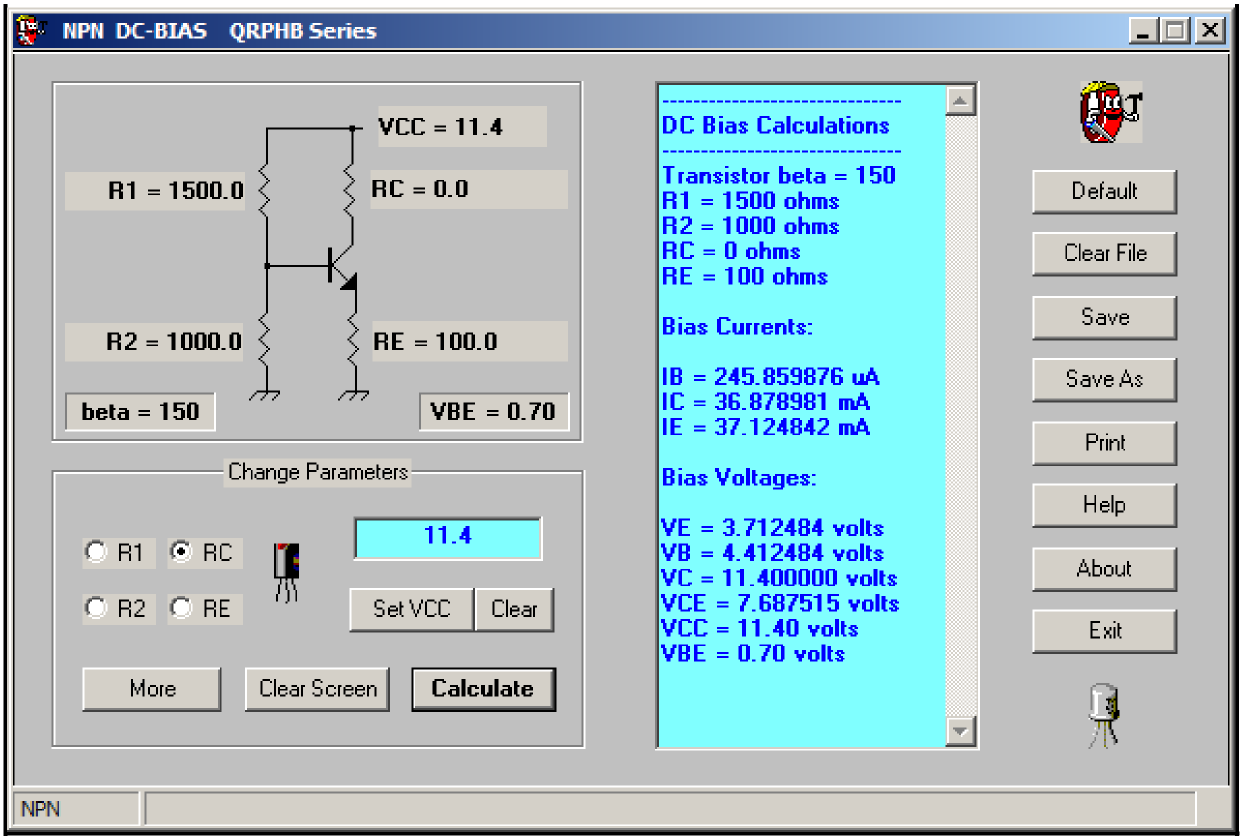
Zin = input impedance. Zout = output impedance.

Explore this program to learn how changing the emitter resistor, feedback resistor and emitter current affect the input and output return loss.

Adjusting the transformer N and load values only affect the calculations for Zin because this app wasn't really designed to crunch output transformer Z ratios for Zout manipulation. The default output Z = 200 Ω and thus for the N parameter with a 50 Ω RL, RL is multiplied by N^2 to set the amplifer load impedance.

From FBA08 simulations: with an emitter current of 35 mA, my series feedback = 6.2 Ω and shunt feedback = 1500 Ω.

I chose a simple voltage divider bias network to set up the ~35 ma and ensure reasonable temperature stability.



VCC = 11.4
R1 = 1500.0
RC = 0.0
R2 = 1000.0
RE = 100.0
beta = 150
VBE = 0.70
11.4

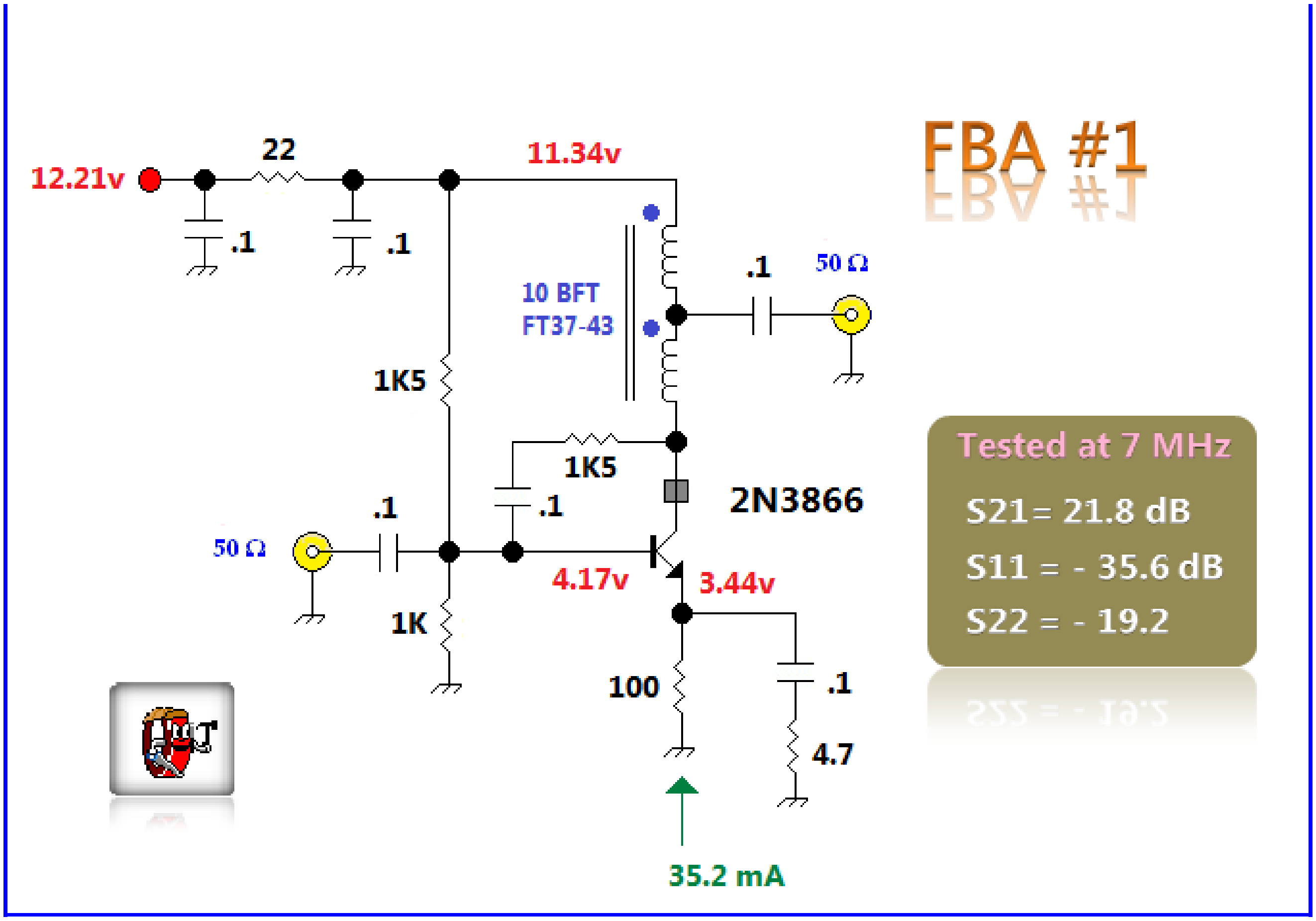
```
-------------------------------
DC Bias Calculations
-------------------------------
Transistor beta = 150
R1 = 1500 ohms
R2 = 1000 ohms
RC = 0 ohms
RE = 100 ohms

Bias Currents:

IB = 245.859876 uA
IC = 36.878981 mA
IE = 37.124842 mA

Bias Voltages:

VE = 3.712484 volts
VB = 4.412484 volts
VC = 11.400000 volts
VCE = 7.687515 volts
VCC = 11.40 volts
VBE = 0.70 volts
```

Above — Choosing the emitter and nearest standard value bias network resistors to set up ~ 35 mA emitter current with a program. Actual biasing requires you to set up the correct emitter current + establish reasonable temperature stability.

Click and scroll to #5 for some basic transistor biasing notes. While this supplement shows a simple method for stable bias networks, it probably understates that **Beta** bias stabiility is a function of the ratio of RB to RE, where RB = the 2 base resistors in parallel. The lower the ratio the better, but then more input power is lost in those resistors. A higher ratio reduces stability but wastes less input power — another trade off we must negotiate! See Ken Kuhn's web site for thorough, expert-level information on voltage divider biasing your BJT amplifiers.

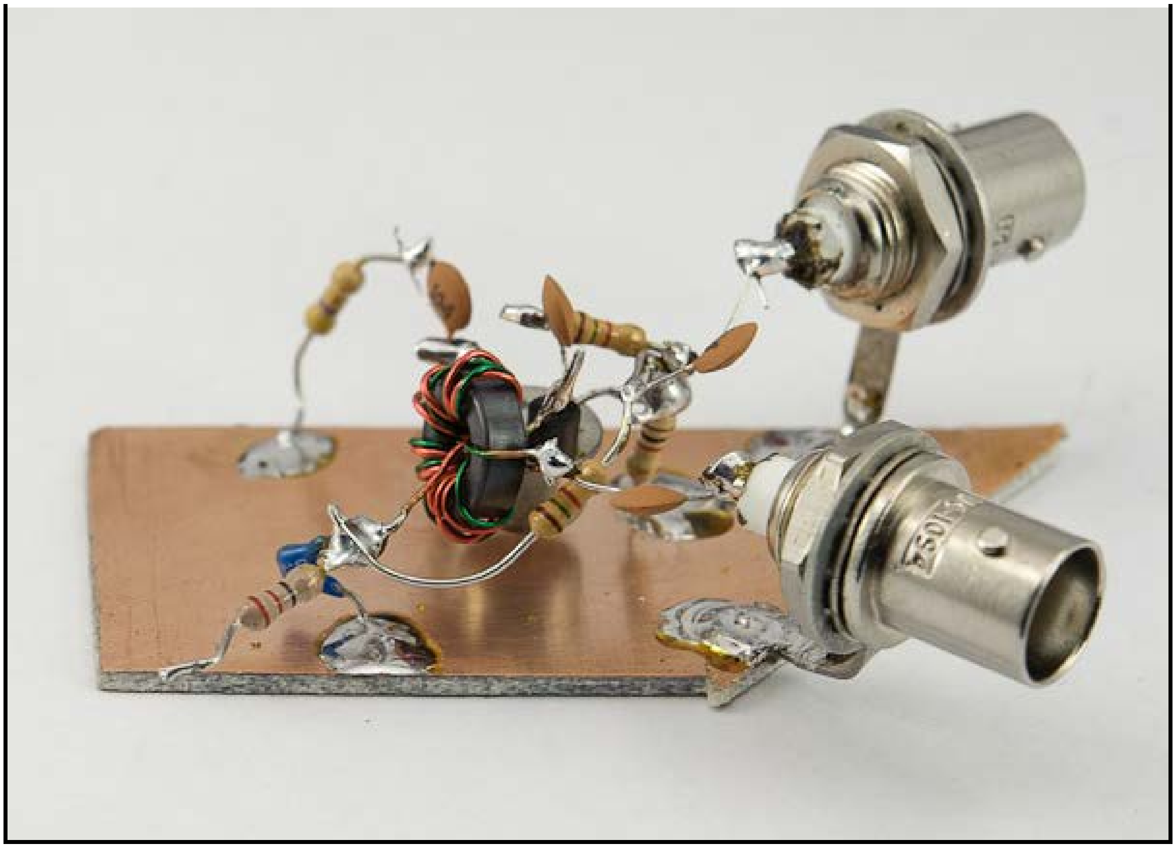
I use NPN DC BIAS, a program I wrote, however, Wes included 1 in the Ladpac software called Biasnpn08.exe that's also good. Determine the VC for the program by first multiplying the value of your decoupling resistor by the emitter current in Amperes to learn the voltage drop across the R. Then, subtract that voltage drop from your power supply voltage: 12.22V - (.0371 A X 22 ohms) = 11.4 VDC.

Our software allows you to pick approximate base and emitter resistor values to set up a desired current in your amplifier breadboard, but you must still choose reasonable values for temperature stability. Tweak them as needed, or choose some other bias method such as a current source. Let's move to the bench...



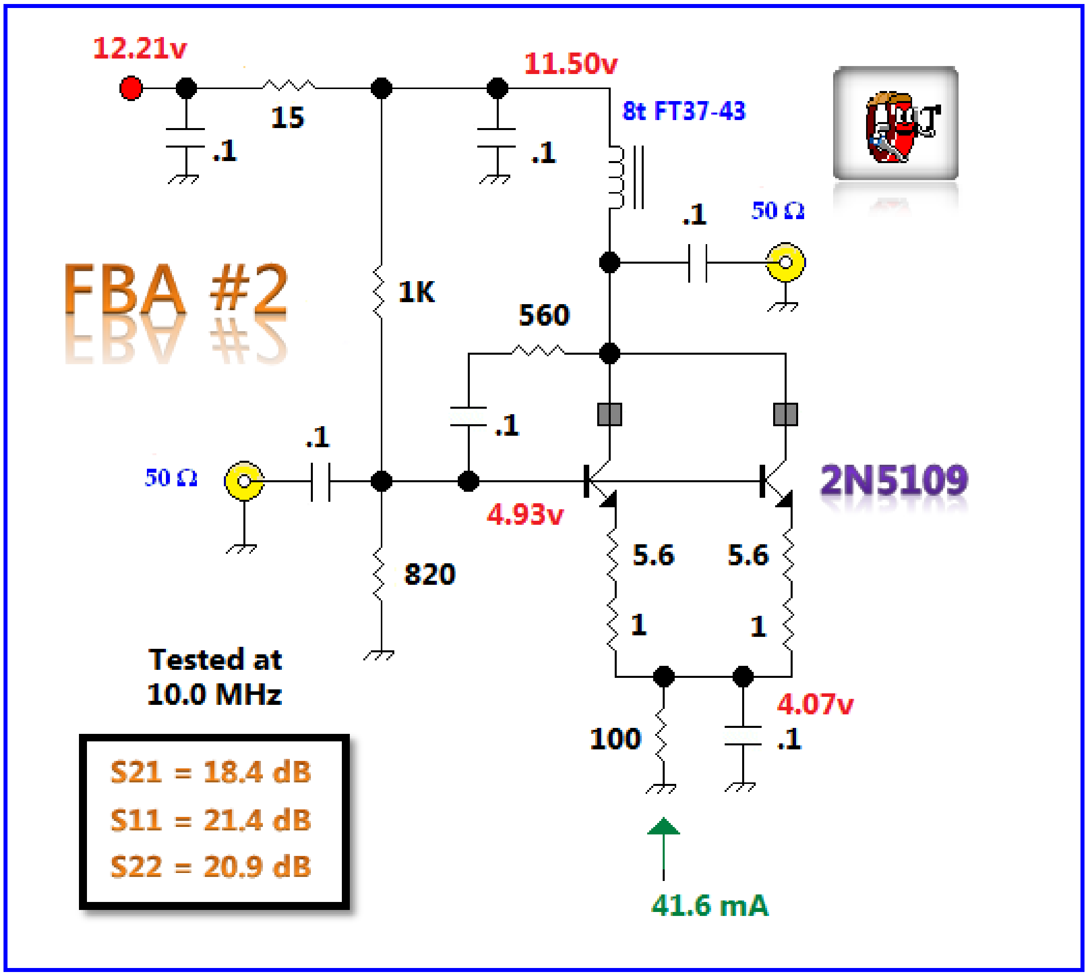
Above — My 7 MHz FBA with some measured S-parameters. On the bench, I lowered the 6.2 Ω series resistor to 4.7 Ω because the voltage divider bias network also affected Z in. I tried 3.3, 4.7 and 5.7 Ω resistors for series feedback and settled on 4.7 Ω since an S11 of -35.6 dB wins the prize!

The S22 of -19.2 dB bettered the value predicted by FBA and seems quite acceptable considering we normally follow a FBA with a 6 dB pad that raises the output return loss another 12 dB. FBA08 gets you close, however, only bench experiments will realize the amplifer you want, and sometimes, a decent S11 and/or S22 may elude you.



Above — A photo of the 35 mA feedback amp built on scrap of copper clad board.

Parallel Transistor Feedback Amp



Above — A feedback amp with two 2N5109 transistors wired in parallel. Click for a photograph of this prototype. I lacked 6.8 Ω resistors and placed 1 Ω + 5.6 Ω to make the needed R for a strong S11.

Originally, I built FBA #2 with a 4:1 Z transmission line transformer, but measurements of S22 disppointed me. Later, a L wound with 8 turns around an FT37-43 ferrite toroid drove an S22 of 24 dB, but S11 was only 18.5 dB.

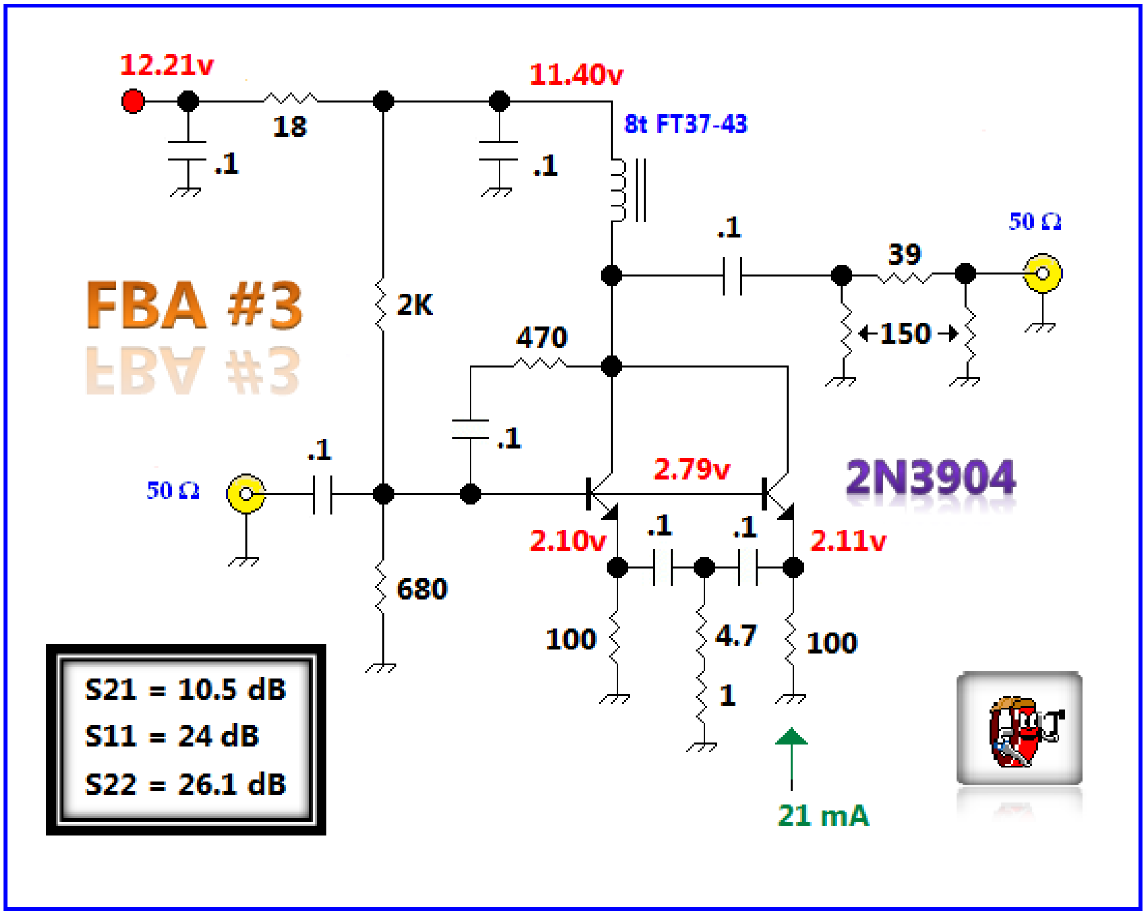
With the amp set up to measure return loss on the input port, I placed a 500 Ω potentiometer in series with a 100 Ω resistor between the collector and base terminals and tweaked the pot to obtain the lowest peak-peak voltage in my 'scope (lowest return loss). After, I removed the pot and measured its resistance at 572 Ω. Finally, I soldered in a 560 Ω resistor and re-checked S11. Perfect. With my goal of at least 20 dB for S11 and S22 obtained, I powered down my bench and took some photos.

Wes, W7ZOI displayed parallel transistor FBAs in EMRFD and other works and recently I noticed Lyle, KK7P employed a parallel NE46134 FBA as a post-mixer amplifer in the Elecraft K3.

Wes wired 2 parallel 2N3904s to avoid using an expensive medium power BJT like the 2N5109. Doing so splits the heat between 2 devices, but does not deliver better IMD performance beyond what is offered by increasing the emitter current. In a typical FBA bias setup, you may measure as much as 10 volts between the collector and emitter terminals and with a supply of 12 VDC + a standing current of 20 mA, the collector dissipation = ~200 mW. This is about maximum for a TO-92 device likes a 2N3904, but only half of maximal dissipation for 2 in parallel.

Then, too, the K3 applies 2 parallel medium power BJTs get power dissipation with an SMT transistor. For strong IMD performance, Lyle and crew are throwing 80 mA or so into the pair — hard to do with SMT parts, so they overcome heat and power dissipation issues with 2 devices. Cool (literally).

Heat sink BJTs when you crank up the emitter current.



Above — An attempted 2N3904-based parallel feedback amp. Each BJT draws ~21 mA emitter current. Without the 6 dB output pad, the output return loss = 14 dB — I *failed* to realize both a strong (raw) S11 and S22.

The power gain including the 6 dB pad = 10.5 dB. I'll discard this design since it's substandard — without failures, victory may taste bland.

For bench designers, making a parallel FBA where both the raw S11 and S21 are > 20 dB is difficult and bench failures may either frusturate you, or enhance your resolve to succeed. With success, great satisfaction arises and I'm addicted to that feeling.

An FBA bench triumph means you managed to establish the perfect combination of series + shunt feedback, emitter current and the correct output transformer ratio for *that* transistor plus its biasing circuitry — no small task.

A well matched amplifier = a thing of beauty! The fetching trio of high S11, S22 and S21 rewards your efforts and boosts your confidence to experiment further. And so it goes...

Sadly, only a fraction of hobbyists create and evaluate their own circuits.

---

4. Microphonics in Direct Conversion Receivers

LO = local oscillator or VFO. DC Receiver = direct conversion receiver.

Microphonics are induced electrical responses that arise from a mechanical vibration on the DC receiver chassis or circuitry. The audio amp, acting like a transducer, makes a clicking, or popping noise when you do things like tap the chassis, or unplug components — the disturbance throws out a burst of DC voltage that's amplifed by the AF chain and pops the speaker.

We may read or hear inexperienced builders tell us to expect microphonics in our DC receivers — *de trop* folklore strikes again! As a student of EMRFD and those wise designers who live in and around Beaverton, Oregon, I share some of their best tips to decrease microphonics in your DC receiver projects. " Keep Your LO From Radiating to the Outside World and Keep Unwanted RF from the Outside World Getting Into Your Receiver" seems the appropos title for the bulleted notes that follow:

Read EMRFD pages 8.7 to 8.11 and then build or apply the presented examples. Wisdom is experiential; it comes by doing, not just reading. It's no accident that Chapter 8 author Rick, KK7B mentions microphonics and hum in the same section. I've never read more thorough notes regarding DC receiver nuances anywhere; for example, did you consider that an ungrounded air variable capacitor shaft poking outside the LO box will radiate LO signal per Figure 8.18 ? I didn't in my early days.

Stick your LO in a RF-tight enclosure with RF-grade connectors and coax to patch the AC signal to the product detector. Bypass RF with feedthrough capacitors on any DC voltage lines that pass through the LO chassis wall. Many enthusiasts have only operated kitted or homebrew DC receivers where the LO and receiver guts lie on the same circuit board — this *ensures* microphonics. Wes and Roger built the historic Ugly Weekender VFO, transmitter and receiver in seperate boxes — resulting in low microphonics and no pulling of the VFO when keying the transmitter. Nothing in that 2 part QST series was done by accident. Read these articles to "go to school".

Reciprocally important; keep unwanted outside world RF from getting inside your DC receiver! Apply resistors plus capacitors, or inductors plus capacitors to decouple and bypass RF from moving along on your DC voltage lines, key line, microphone cables etc.

Keep product detector port-to-port isolation high. Typically, we employ double balanced mixers to obtain high port-to-port isolation. I cover mixer balance on this page . For diode ring mixers, measure the return loss of the circuits that you connect to the product detector LO, RF and AF ports — I aim for 20 dB or greater return loss on my LO output, RF output and AF amp input circuits to help preserve the product detector balance and keep port isolation as high as possible. Along with 50 Ω amplifers, attach attenuator pads, AF diplexers, or whatever to help increase port return loss as required.

LO-RF port isolation: Consider a common gate amp with an output matching network to get a high output return loss (S22). The common gate amp provides strong reverse isolation without adding much noise.

Avoid end-fed wire antennas where there is a strong antenna field right next to your radio.

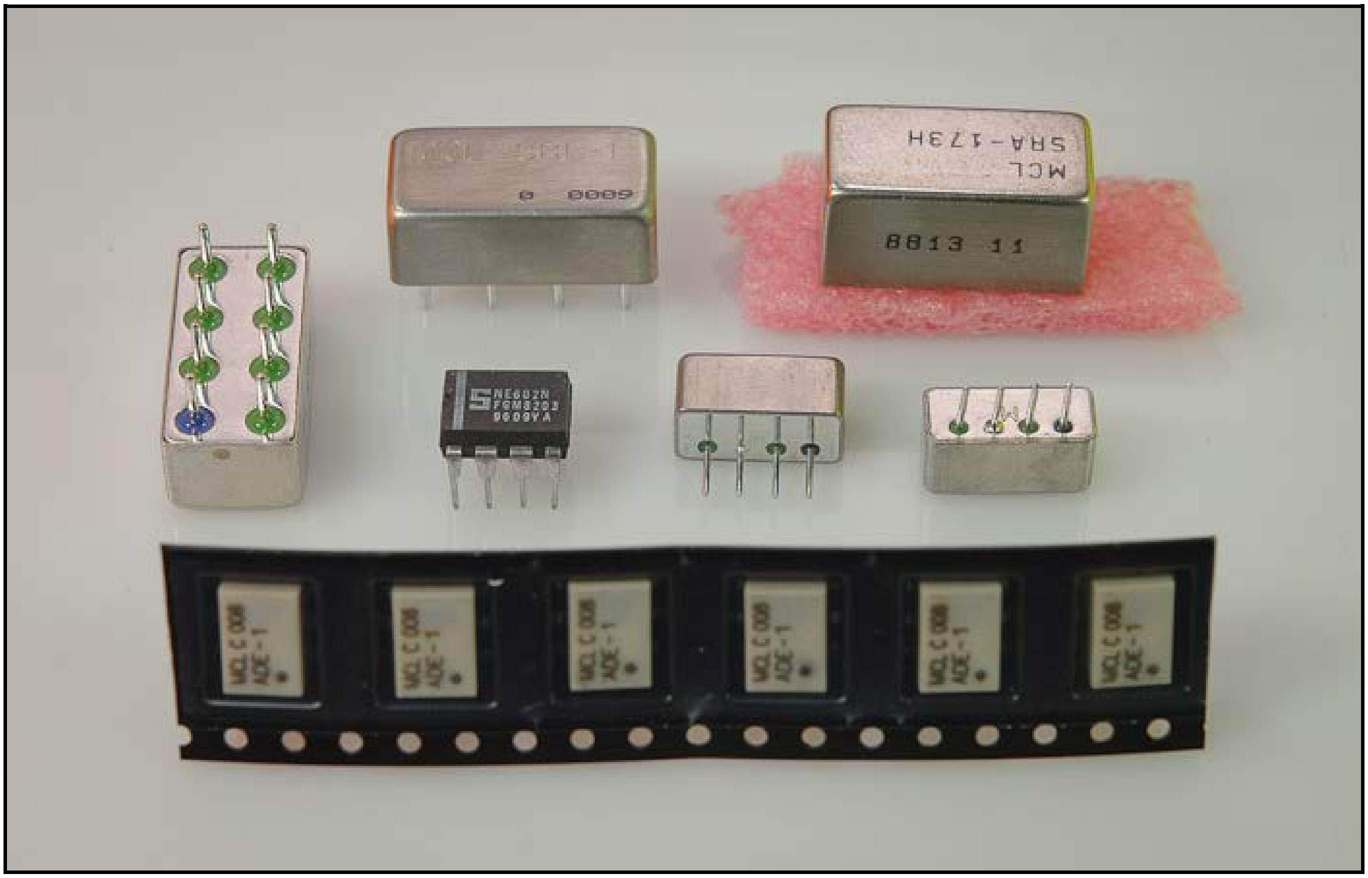
I favor sturdy chassis/cabinets with rubber feet. Homebrew copper clad board or die-cast aluminum cabinets may work best as joints and screwed connections won't corrode. *This is a weak recommendation.*

Double the LO frequency or apply a heterodyne VFO. Often microphonics arise in the VFO tank. EMRFD cover this well. If the VFO operates at a significantly different frequency than any of the signals reaching the balanced mixer, leaked LO won't cause as much havoc as when a LO tank is tuned to the mixer RF port frequency.

Despite proper techniques, RF can exit via the antenna port and make its back into our rig through power supply cables (often modulated by our house AC electricity). In some cases, we require special power supply decoupling to decrease hum and microphonics. We might need to add a common mode choke (+/- capacitors) for common mode noise suppression in addition to the usual differential mode choke(s) and capacitors. In my main shack power supply, I run a common-mode choke plus I soldered a 0.01 uF capacitor across each bridge rectifier diode to bypass RF.

Some radio operators just run battery power supplies.

Above — Feedthrough capacitors. I prefer hole mount over solder mount parts, however, quality feedthrough capacitors of any kind tend to be expensive. As a hobbyist, I'm constantly searching for bargains and when I find 1, I'll purchase a bunch to meet my current and future needs.



Above — Some double balanced mixers from my collection: ADE-1, NE602, TUF-1, TUF-2, SBL-1 and a SRA-173H; a MiniCircuit Lab's Level 17 diode ring mixer.

You owe it to yourself to listen to a DC receiver designed and built to reduce microphonics — music to our ears.

---

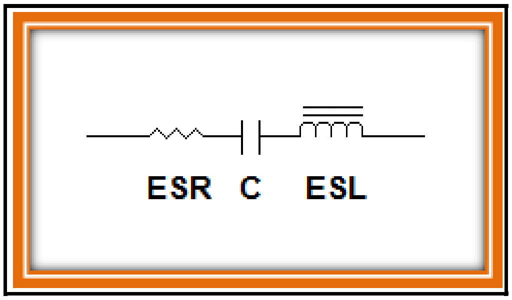
5. Some Experiments with RF Bypass Capacitors

Introduction

Bypass implies a low impedance path to ground for RF at 1 or more frequencies. After reading EMRFD pages 2.28 - 2.31, I decided to explore this subject for the first time. My bench measurements from Spring 2012 punctuated how little I knew about RF bypass and I share these notes as something for me and others to build on.

In these experiments, I

1. observed the self resonant frequency of MuRata RPE Series, 50v, 5% capacitors with X7R temp compensation at 0.1, 0.01 and 0.001 µF.
2. examined a wire short, plus 1 and then 2 Johanson Dielectric 0.01µF, 50v, X7R, size 1206 chip capacitors.
3. tested a 0.1 µF RF cap plus a parallel 2.2 µF electrolytic capacitor to look at parallel resonance side effects.
4. attempted to reduce the Q of some parallel capacitors to reduce unwanted high impedance peaks.



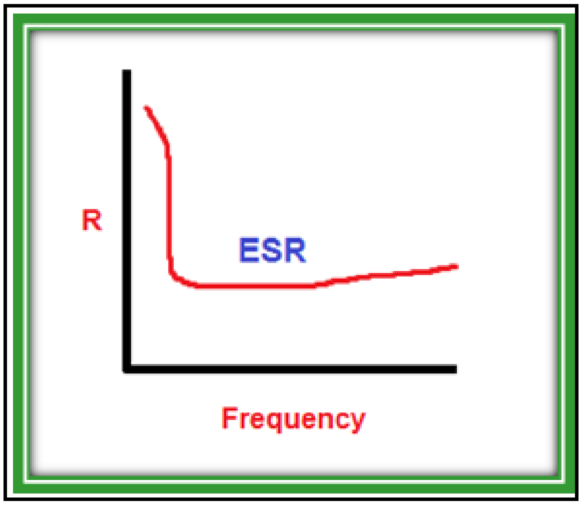
Above — The frequency dependent components of a capacitor are shown in this capacitor equivalent circuit schematic; essentially an RLC network. Engineers use mathematical formulae to describe the components of a capacitor along with reactance and with this math, you might derive an unknown variable from available data so it's worth diving into on your own.

ESR or equivalent series resistance = the sum of all of a capacitors' resistive components. Expressed in ohms, ESR acts like a resistor in series with the capacitor. Normally we desire capacitors with an ESR as low as possible. Consider reading the capacitor datasheets for those your stock and/or searching for information regarding low ESR capacitors on the Internet.

ESL refers to the equivalent series inductance; the sum of all the capacitor's inductive components. This includes lead length in hole-through parts.

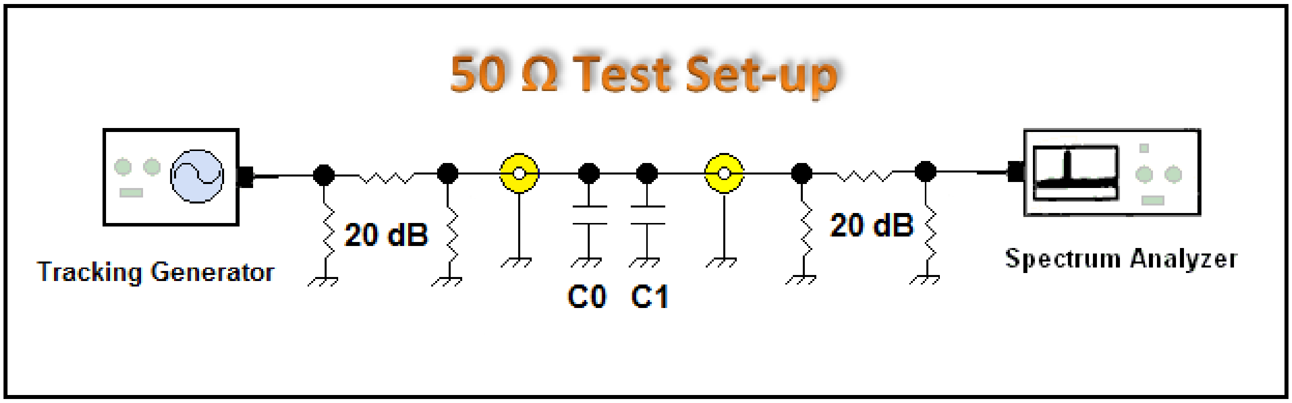
In a given capacitor, the series resonant frequency is the frequency where the inductive reactance from the ESL = the capacitive reactance, but since the 2 reactances are 180 degrees opposite in phase, they cancel to drop the impedance to 0 and the capacitor acts like a resistor at its ESR.

The series inductance of a capacitor may be determined using a network analyzer and unfortunately this in unattainable by most average builders. When designing RF bypass with network analysis, we strive for a low impedance over a wide frequency range, although small ripples typically occur.



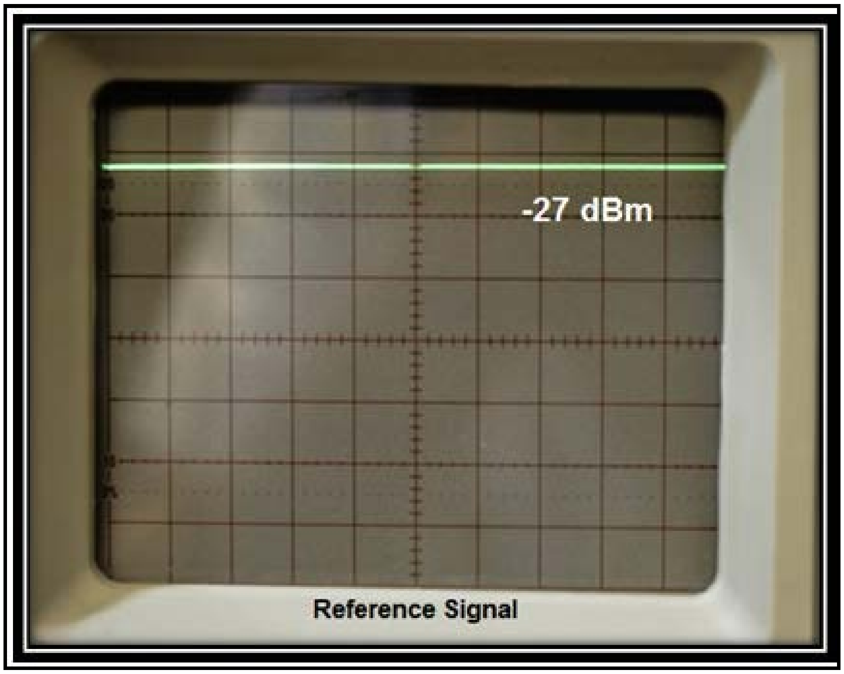
Above — A plot of equivalent series inductance. ESR tends to increase with frequency.

Methods



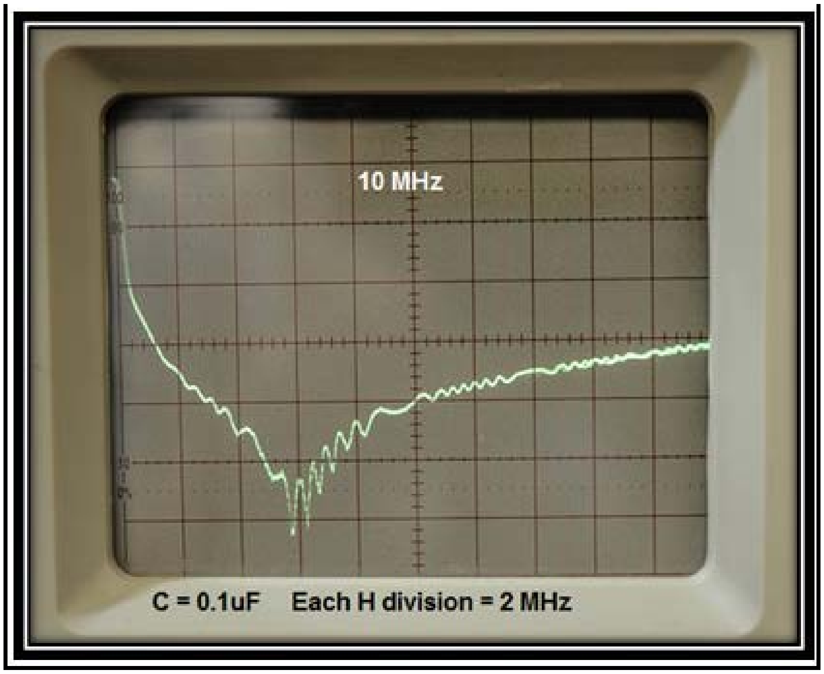
Above — My test set up. I performed all analysis with a tracking generator plus spectrum analyzer. The 50 Ω system used short coax patch cables fitted with BNC connectors with 20 dB attenuator pads before and after the capacitors under test. The capacitors shown as C0 and C1 were soldered on a copper board with short leads and BNC connectors. C1 is omitted when evaluating only 1 capacitor.

You may also perform capacitor self resonant frequency testing with a vector network analyzer, a signal generator plus a 50 Ω terminated scope, or with a sweep generator ramp-driving the oscilloscope X input while simultaneously driving a VCO with logarithmic output to the Y oscilloscope input. SPICE simulations may also yield insight.

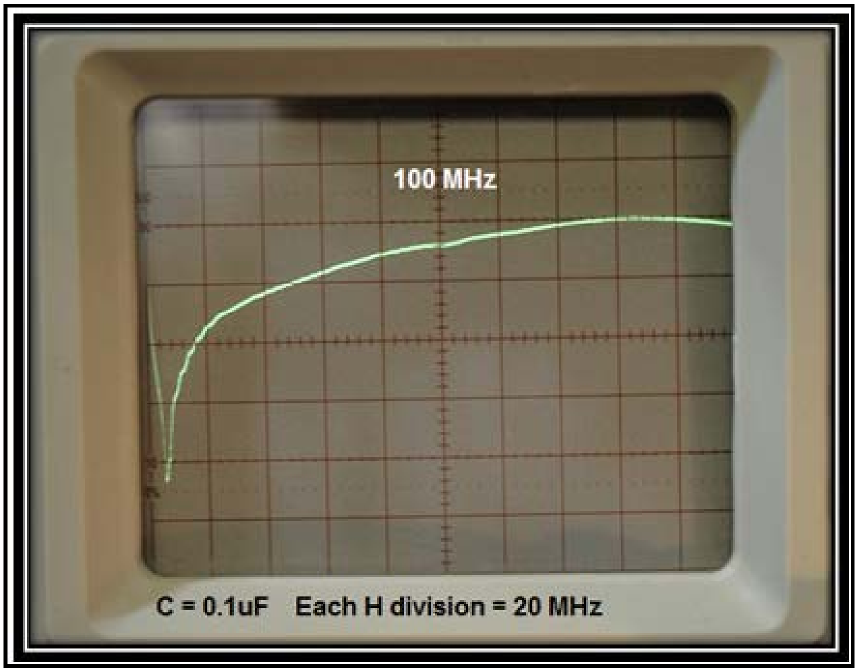


Above — The -27 dBm reference with a through-connector between my 2 coax patch cables (C0 + C1 board removed).
To save time I shot these SA photos handheld and prefer a slower shutter to capture a nice CRT tracing, so some of the photos show a little hand jitter.

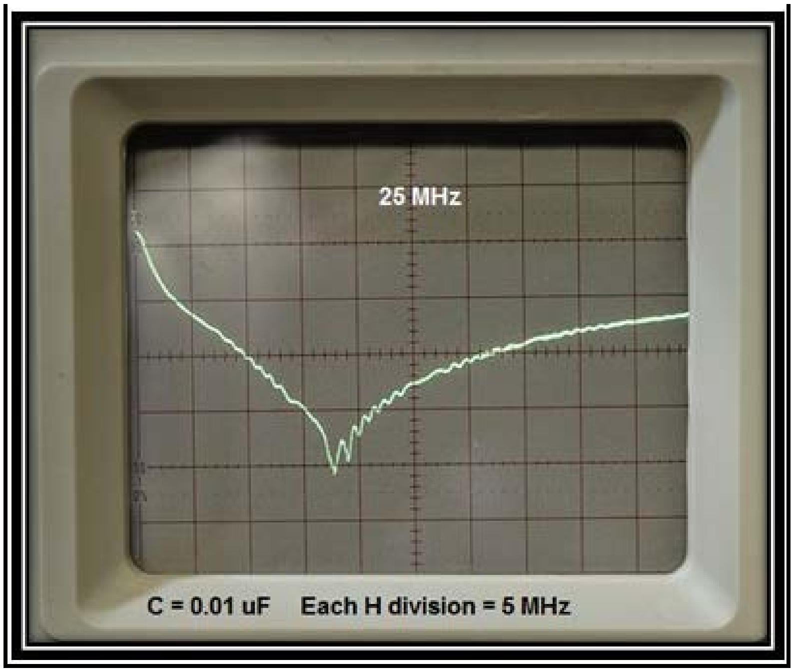
Single Shunt Capacitors



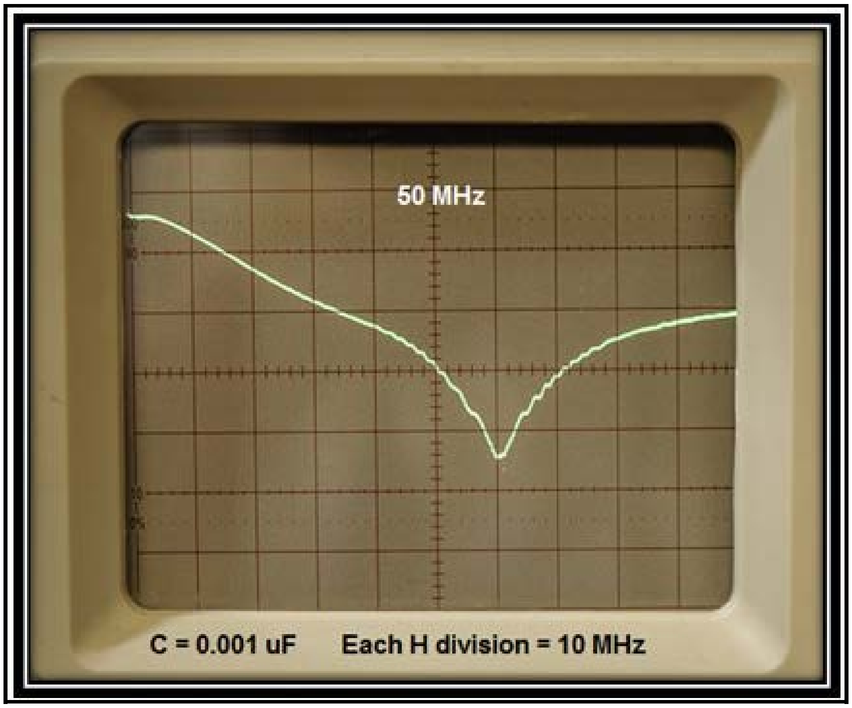
Above — C0 = 0.1 µF. I view the capacitor like a trap. At almost 5.8 MHz lies the peak attenuation, or lowest impedance — this is C0's self resonant frequency. The peak bypass frequency lies ~ 60 dB down. At 20 MHz, the attenuation is only ~ 30 dB.



Above — Another shot of the 0.1 µF bypass cap with a 200 MHz span. At 50 MHz, the reference signal lies only ~ 17 dB down. At 100 MHz, the attenuation is only ~ 11 dB — this hardly qualifies as “bypass” much above the self resonant frequency. Above the self resonant frequency, a capacitor's XL affects impedance more than the ESR and XC of the capacitor.



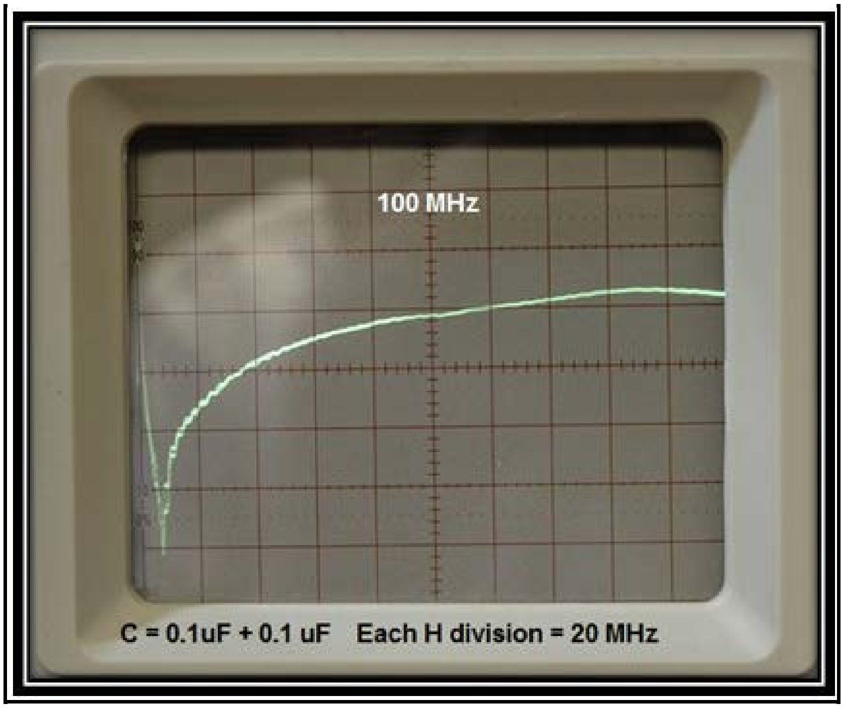
Above — C0 = 0.01 μF. The peak bypass frequency (capacitor self resonant frequency) is centered at 17.5 MHz and is ~ 50 dB down; not as deep as with the 0.1 μf cap. At 50 MHz, the attenuation is ~ 21 dB.



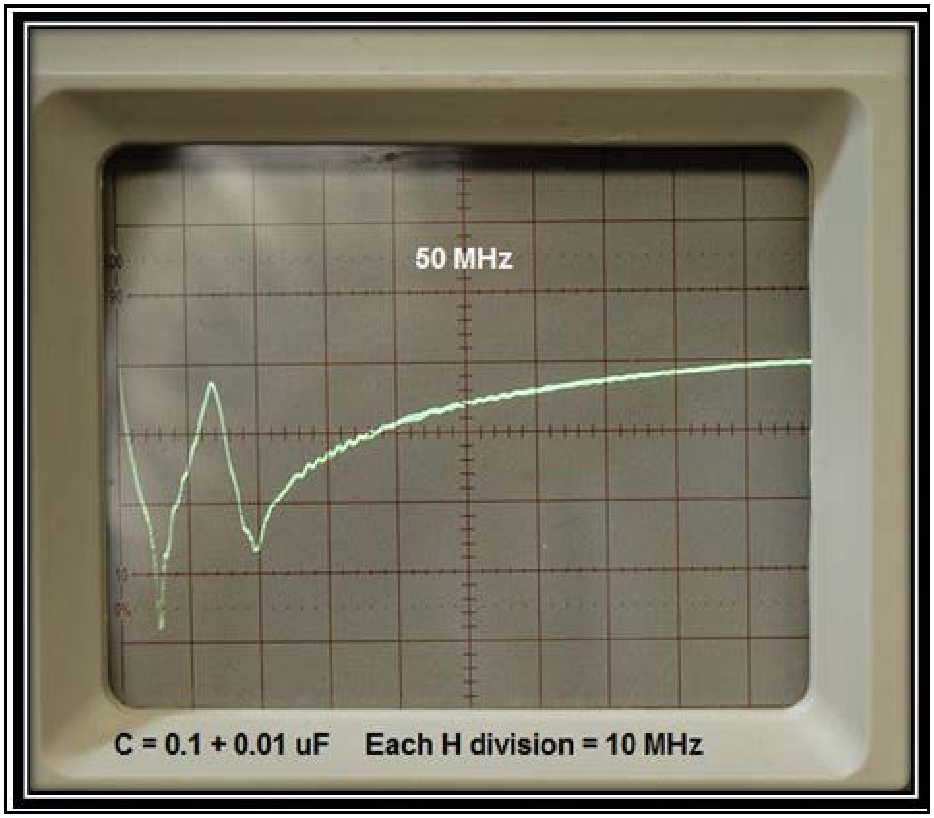
Above — C0 = 0.001. The response is peaked at 62 MHz with an attenuation of ~42 dB. At 100 MHz, the signal is 17 dB down. Again, the peak attenuation looks diminished compared to that of the 0.1 μF and the 0.01μF caps.

Capacitors in Parallel

Now I placed 2 caps in parallel (C0 + C1) as some builders do to try and garner a wider attenuation bandwidth.

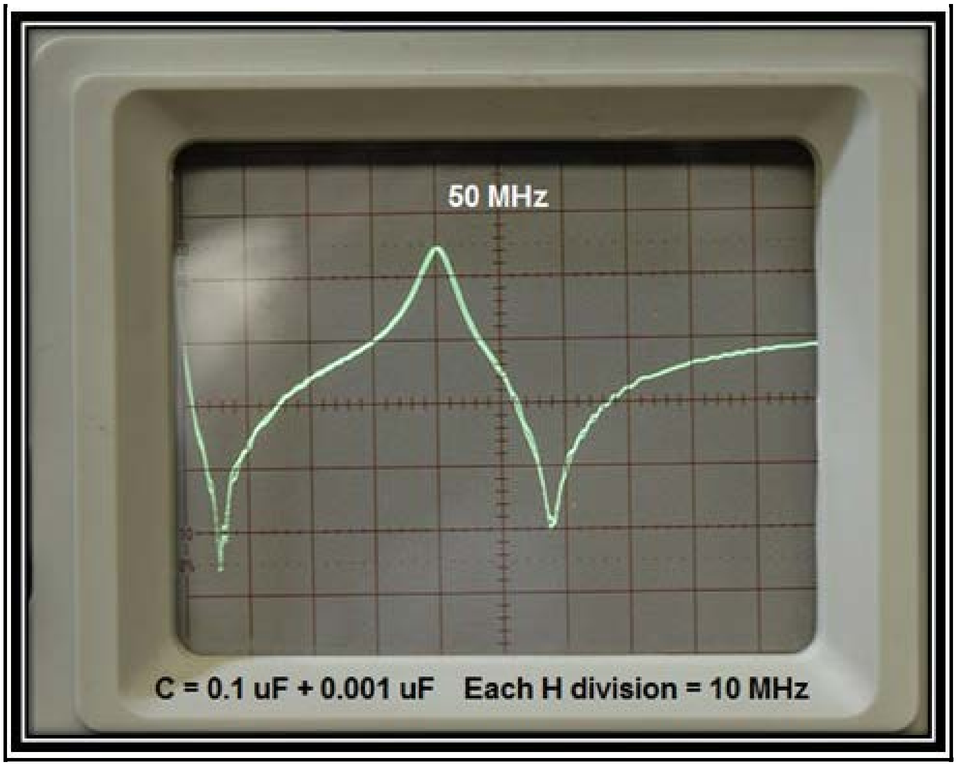


Above — C0 = 0.1 µF + C1 = 0.1 µF. The peak attenuation = 60 dB at 8 MHz; up 2 MHz from that of the single 0.1 µf capacitor. At 100 MHz, attenuation = ~ 19 dB — better than a single 0.1 µF but still low.



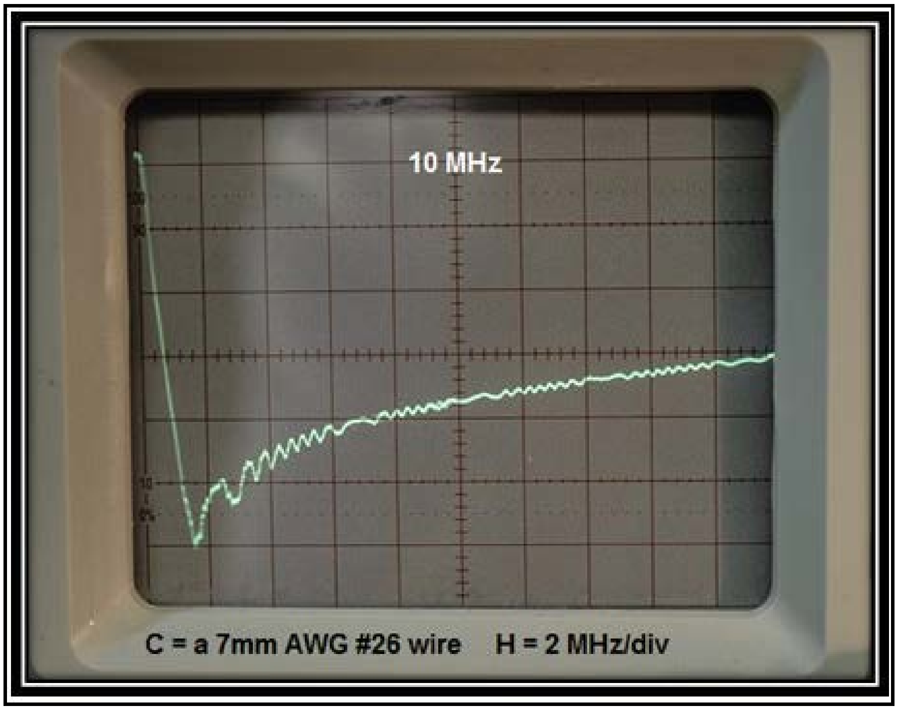
Above — C0 = 0.1 µF + C1 = 0.01 µF. Yikes! With the 2 different cap values, we get an unfortunate high impedance blip peaking at 13 MHz. Each capacitor exerts its self resonant frequency, but in between these self resonant frequencies, lies a disaster.
When placed in parallel, the inductance of 1 capacitor resonates with the capacitance of the other to form a parallel resonance — leading to a high impedance — that blocks RF bypass and peaks at a specific frequency.

But wait. Things can get worse:

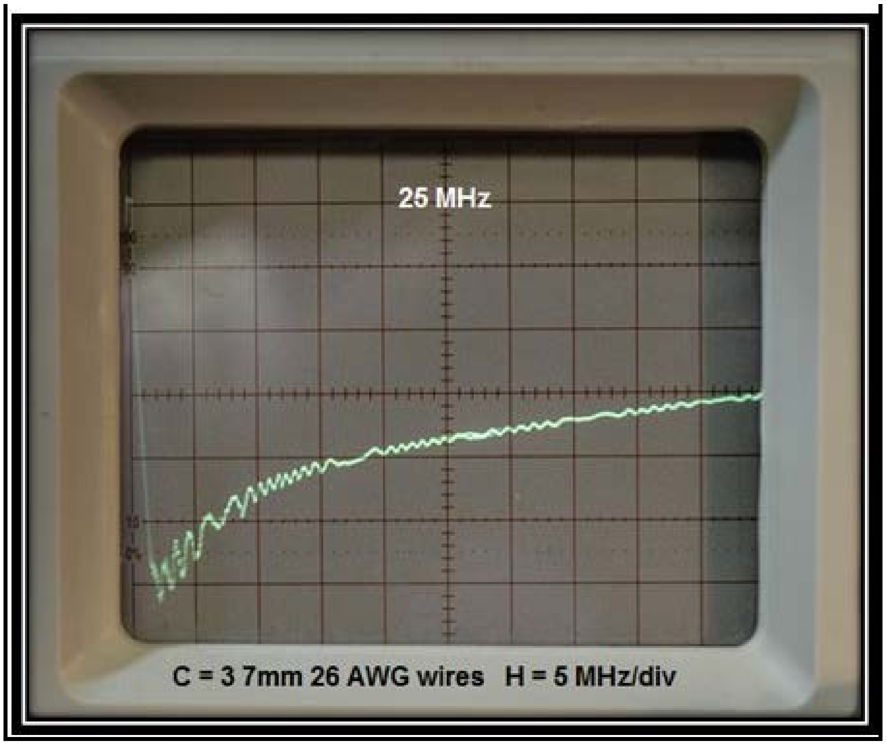


Above — C0 = 0.1 µF + C1 = 0.001 µF. The wide value variance between these 2 capacitors creates a huge, high impedance spike where the attenuation is only about 6 dB at 40 MHz. Catastrophic bypass indeed. катастрофа.

A 7 mm Length of Copper Wire



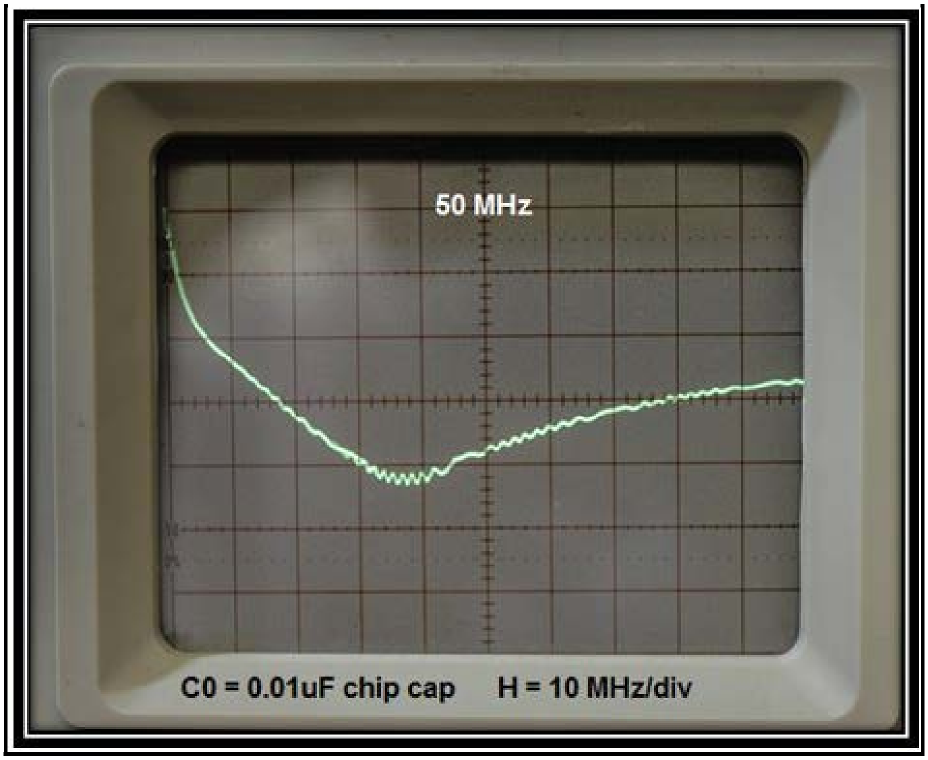
Above — A 7 mm piece of 26 gauge copper wire was shorted to ground instead of C0. This wire measured at ~7 nH of inductance and I saw that attenuation decreases with frequency from 62 dB at 8 MHz to ~33 dB at 20 MHz. Even a short piece of wire doesn't exhibit a flat, wideband bypass.



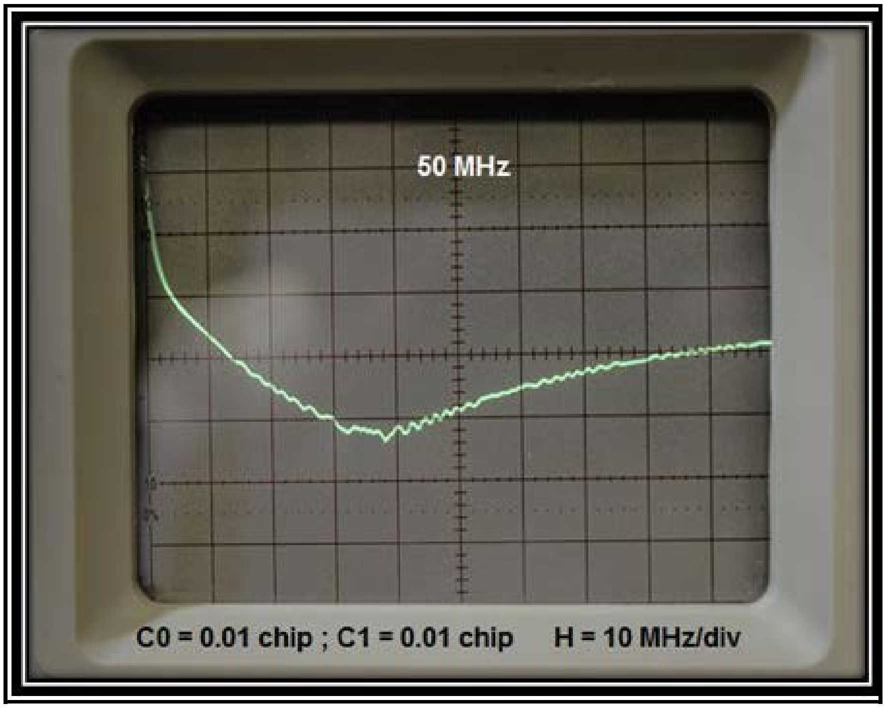
Above — This spectrum analysis shows three 7 mm wires shunted to ground – not much different than 1 wire.

0.01 µF Chip Capacitor(s)

Above — The magnified copper board that I tested one or two 0.01 µF chip capacitors. You can see 1 capacitor soldered in.

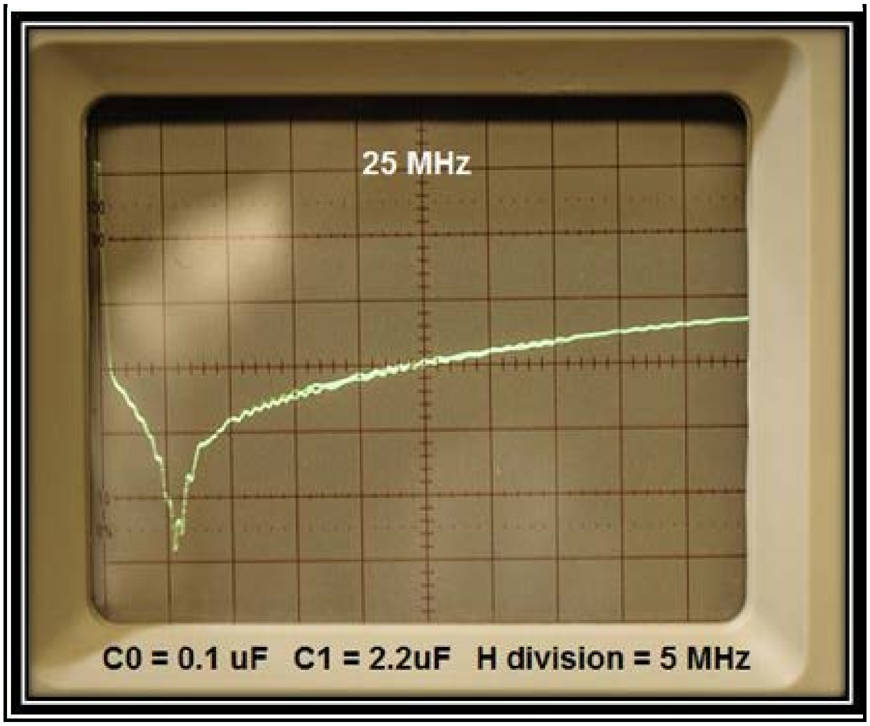


Above — C0 = 0.01 µF SMT cap. The SMT parts exhibited a peak attenuation of 45 dB at ~37 MHz. The attenuation dip lacks the sharp peak of the hole-through 0.01 µF cap shown eariler and exhibits a somewhat wider bandwidth. The self resonant frequency of the chip capacitor is 5 MHz higher than the particular hole-through capacitor I measured. Click for a side by side photo.



Above — C0 = 0.01 µF + C1 = 0.01 µF. The SMT parts exhibited a peak attenuation of 45 dB at ~37 MHz; similar to the single 0.01 µ F chip cap, but with a few more dB attenuation between 10 and 20 MHz.

0.1 µF Ceramic + a 2.2 µF Electrolytic Capacitor



Above — C0 = 0.1 µF and C1 = 2.2 µF. The low Q 2.2 µF C1 electrolytic cap did not create the a parallel resonance with C0. Shaky photo — sorry. I also tested a 10 and 22 µF cap in parallel with C0 and saw no disturbance caused by a parallel resonance between a big AF capacitor and C0 (an RF value cap) with my RF spectrum analyzer.

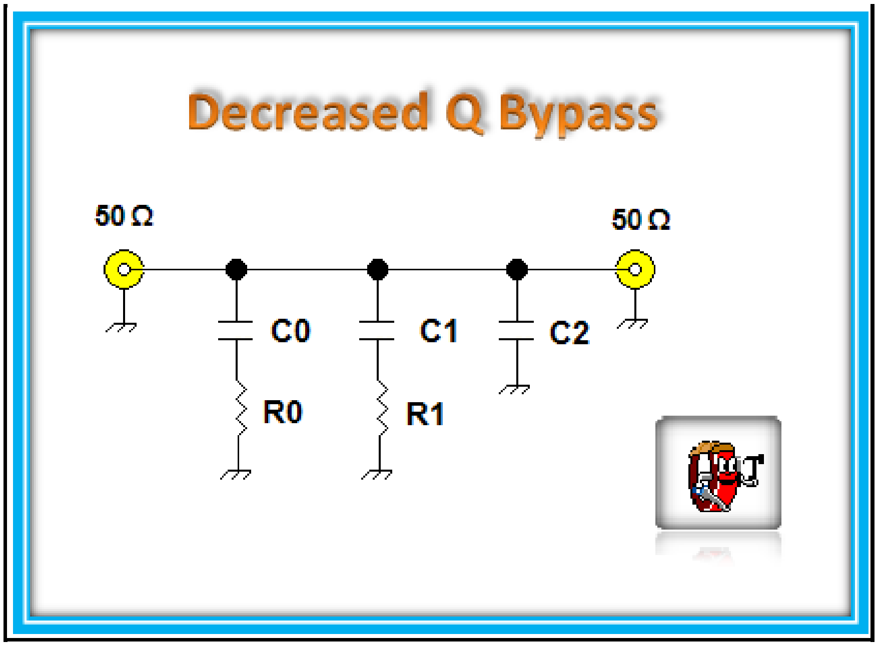
Some additional experiments applying a low Q *A*F capacitor plus a ceramic *RF* capacitor for wideband bypass yielded some interesting results and I'll present these in a future project.

Capacitors in Parallel with a Series Resistor to Lower Q

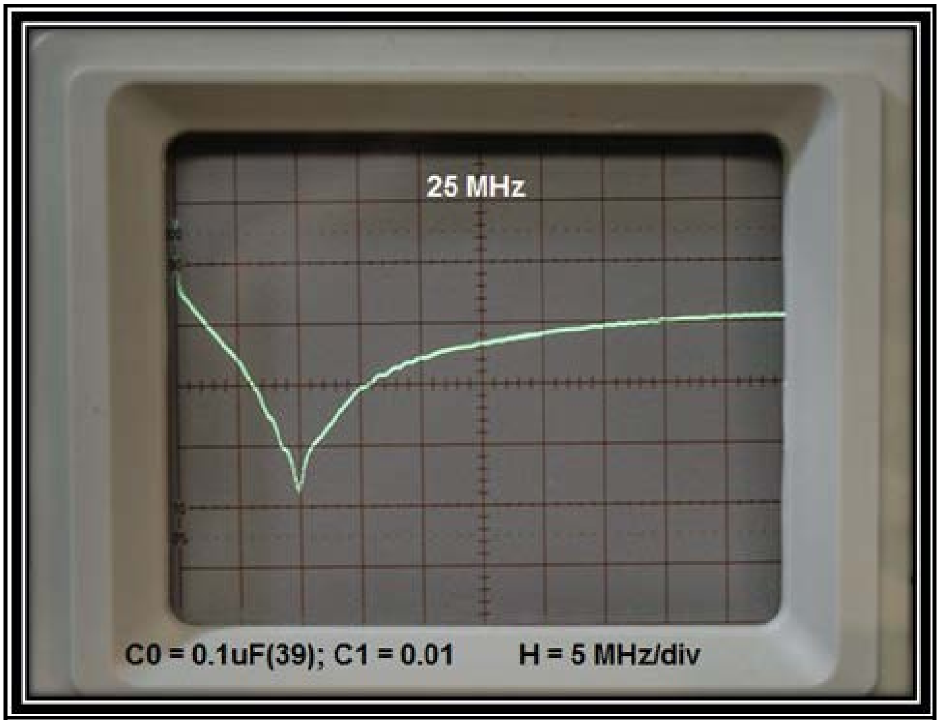
In previous experiments, placing 2 RF capacitors in parallel led to the formation of a peaked high impedance blip between the low impedance peaks set by the self-resonant frequency of the 2 capacitors. If multiple capacitors are soldered in parallel, the series inductance of each capacitor will resonate with the capacitance of the next smaller C value.

One solution is to put a resistance in series with all but 1 of the parallel capacitors so that the Q of resonance formed by this capacitor's series inductance and the capacitance of the next smaller capacitor is low. If capacitors exhibited 0 inductance then putting capacitors in parallel would be fine, however, since capacitors exhibit inductance, a parallel resonant frequency may occur with capacitors in parallel.

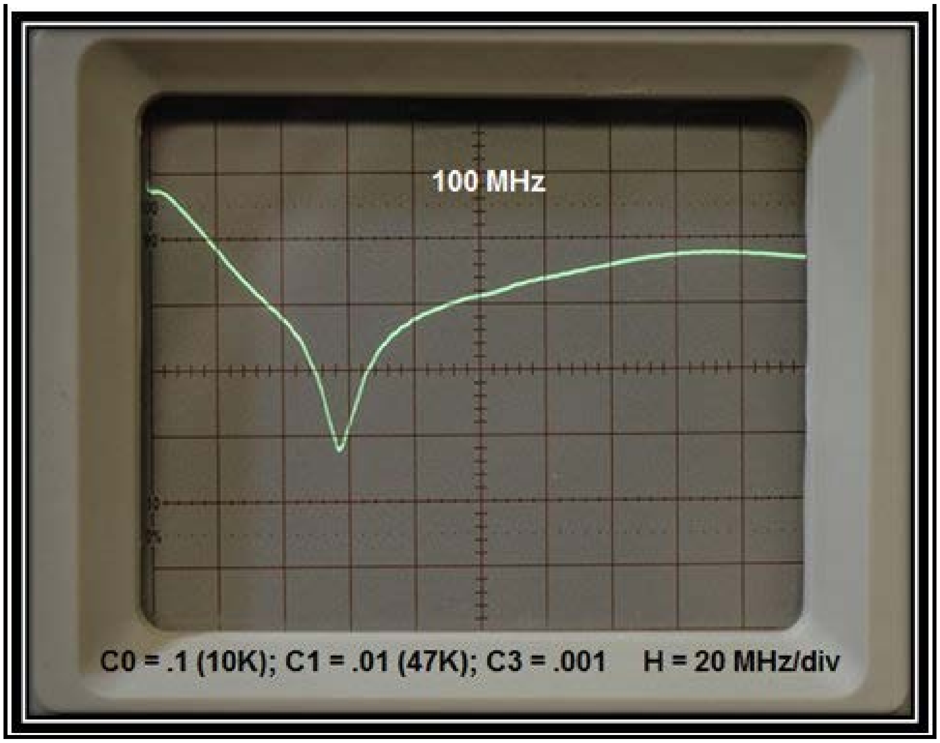
I found applying a series resistance to lower Q may flatten the impedance versus frequency response of the bypass network, but didn't decrease the impedance at any 1 frequency. Optimal bypassing or achieving the lowest impedance over a wide frequency range presents a complex topic that might even challenge some engineers.



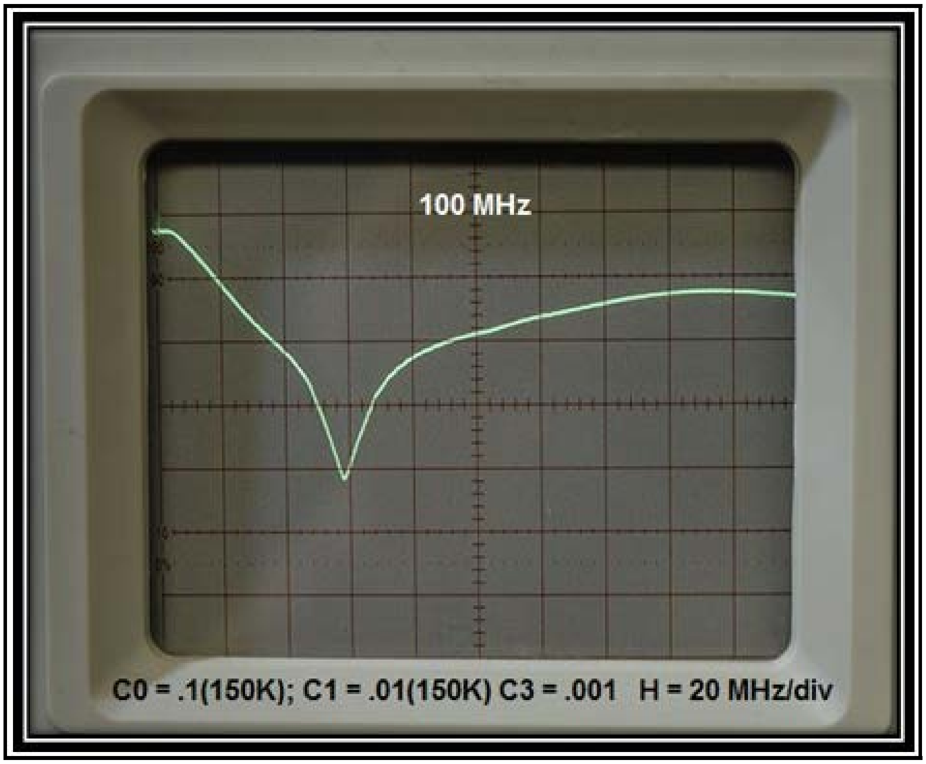
Above — A method to exact wideband bypass.



Above — My first try with C0 = 0.1 µF, R0 = 39 Ω and C1 = 0.01 µF. I arbitrarily placed the 39 Ω resistor in the R0 slot and saw that the high impedance peak seen earlier disappeared. This gave me the confidence to try 3 capacitors. I had no idea what R value to use and really just wanted to see what happens.

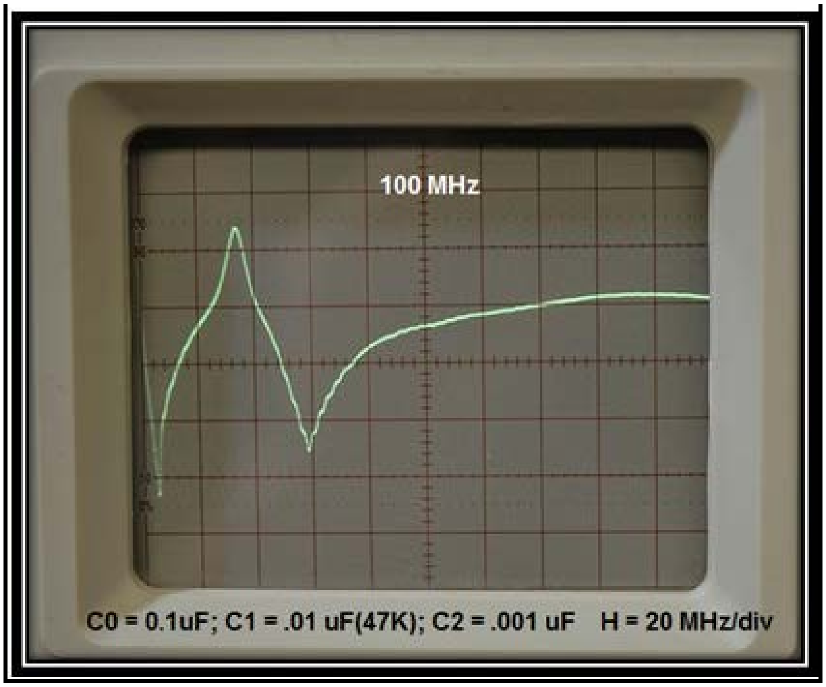


Above — The spectrograph with C0 = 0.1 µF, R0 = 10K, C1 = 0.01 µF, R1 = 47K and C3 = 0.001 µF.



Above — C0 = 0.1 µF, R0 = 10K, C1 = 0.01 µF, R1 = 47K and C3 = 0.001 µF. Again, no high impedance peak response; the self resonant frequency is close to that measured with a single 0.001 uF earlier, however, the peak bypass frequency moved from to 57 MHz from 62 MHz. Changing the resistor values moved the self-resonant frequency and the peak attenuation value a little, but I fell kilometers short of setting a wide band bypass. My approach lacks any real science and I need to step it up.

I hope to learn what capacitor values and types, plus R values to apply. This sounds like a job for simulation as well as further on-bench experiments? After writing this material, I learned that Ken Kuhn wrote an Excel spreadsheet to examine the net impedance of up to 3 capacitors in parallel Click to download.



Above — Just as a gag, I removed R0 from capacitor C0 in 1 circuit and then hooked up the board. The high impedance peak re-emerged.

A Commerical Example

I found a wideband MMIC employing (R1 + C1 and C2) as part of a bypass strategy. Cick for the datasheet excerpt. Note the size of the SMD capacitors; 0603 — tiny caps! My experiments showed some high gain MMICs require careful low inductance grounding and correct part choices or crippling oscillations and other bypass issues might arise.

My Learnings

When we think bypass, we really should think frequency dependent attenuation. The bypass cap is actually a network where impedance versus frequency varies significantly. At its self resonant frequency, a capacitor will exhibit the lowest possible impedance making a single capacitor a relatively narrow-band bypass device. Intuitively, we might want to choose a capacitor with a series resonant frequency at the frequency we wish to bypass, however, if we require a wideband bypass, the need to evaluate our bypass capacitor(s) increases.

In short, above the series resonant frequency of a capacitor, its bypass is basically useless and we should likely ensure that the self-resonant frequency of the particular capacitor we're using is above the highest frequency to be bypassed.

Bypassing with 2 or more unmatched RF caps will lead to an attenuation gap with peak(s) determined by the parallel resonance of these capacitors. Going above a 10:1 capacitor ratio, for example, greater than a 0.1 and a 0.01 µf, may cause a **severe** gap in attenuation at the parallel resonant frequency generated by the 2 capacitors.

Mine and work from more reputable authors clearly shows we should avoid applying parallel **RF** bypass capacitors of *different* values unless we apply a Q-reducing resistor to the capacitor(s) in parallel with a given RF bypass capacitor. Please read EMRFD page 2.3 for more information and watch out for abundant folklore concerning RF bypass.

The need for measurement and analysis challenges us; in some cases, you may realize good attenuation in the radio band of interest, while poorly bypassing the frequencies above it and compromise an otherwise good design.

Capacitor lead length may affect self-resonance at RF.

Future Work

It would be awesome to learn more about getting a wide-band bypass. I want to order some low or ultra-low ESR caps and measure them. My MuRata RPE Series caps specify low inductance; low is relative — how low is low? Should we apply chip capacitors for bypass in our critical circuits such as low noise VHF amps or MMICs?. Am I fussing about nothing? Lots of questions that folklore just won't answer.

Per EMRFD page 2.3, bypass is only half the equation — we need to *decouple* + bypass to filter RF from moving along our DC lines and so forth.

---

6. Some Experiments with Chokes plus Decouple and Bypass Filters

Introduction

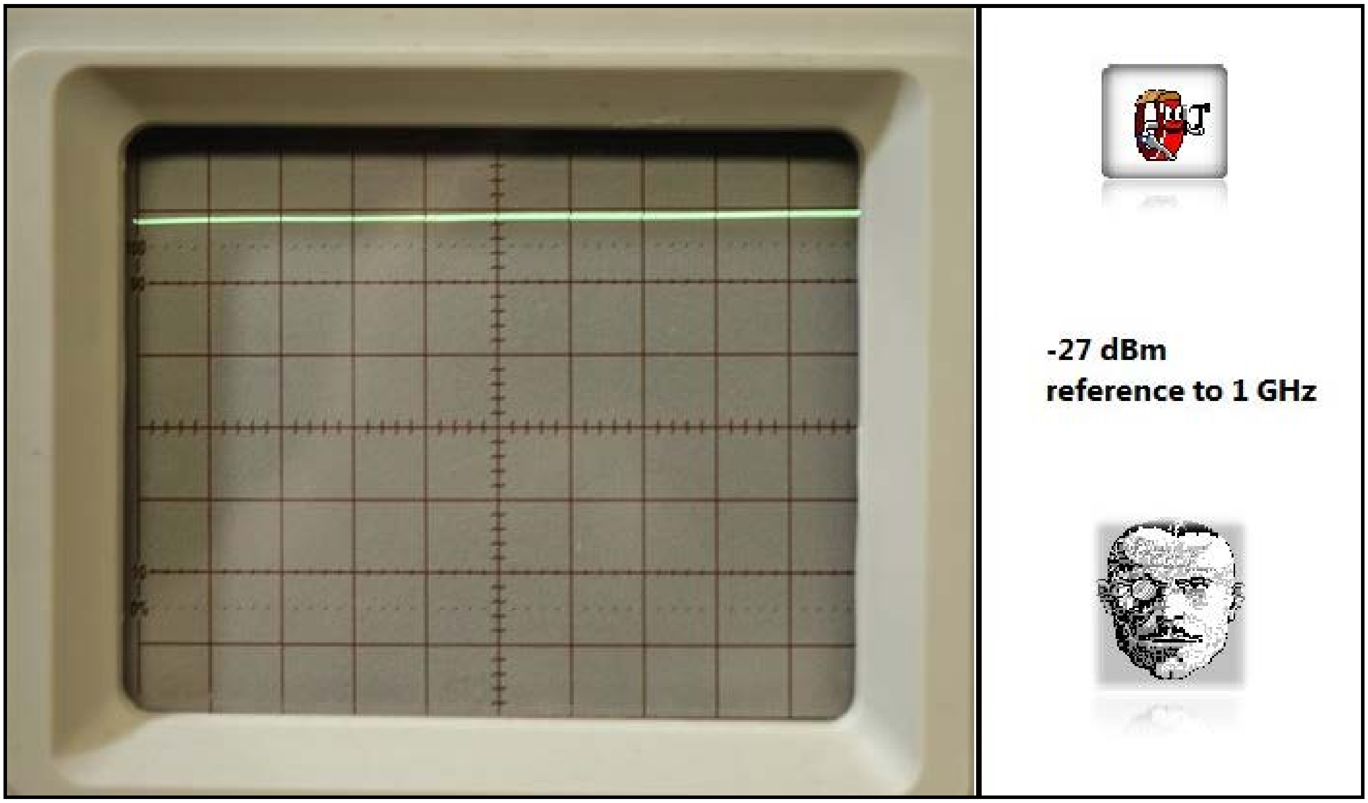
SRF = self resonant frequency; XL = inductive reactance; XC = capacitive reactance. L = inductor; C = capacitor; R = resistor.

Like the capacitor, inductors are networks with R, L and C and possess a SRF. R, L and C may vary with factors including the number of windings, frequency, or whether the L is wound on a ferromagnetic material, or air wound.

Considering R, L and C:

at frequencies below the SRF, XL dominates;
at frequencies above the SRF XC dominates;
at the SRF, the magnitude of XL and XC are equal but 180 degrees out of phase leaving resistance to dominate.

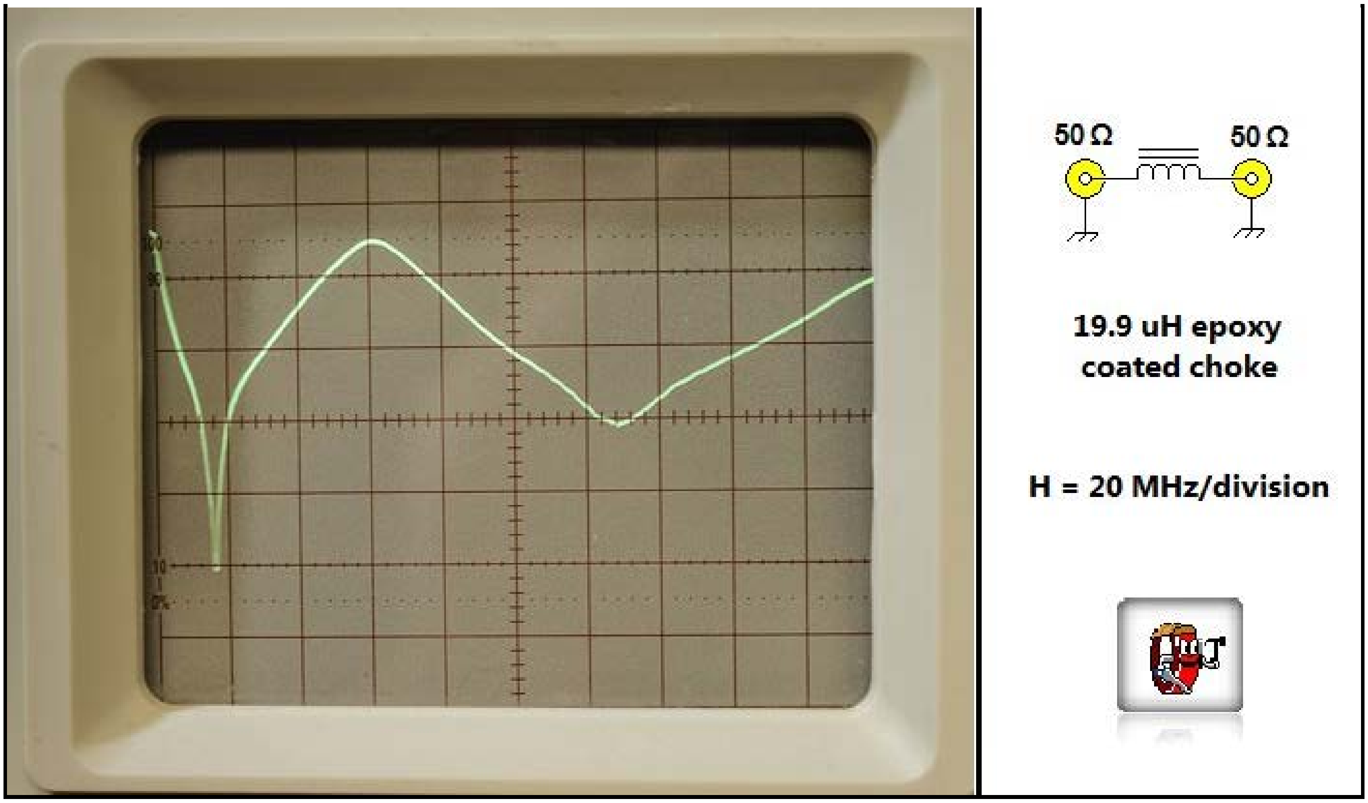
I encourage you to learn more by visiting the fabulous web site of David, G3YNH.



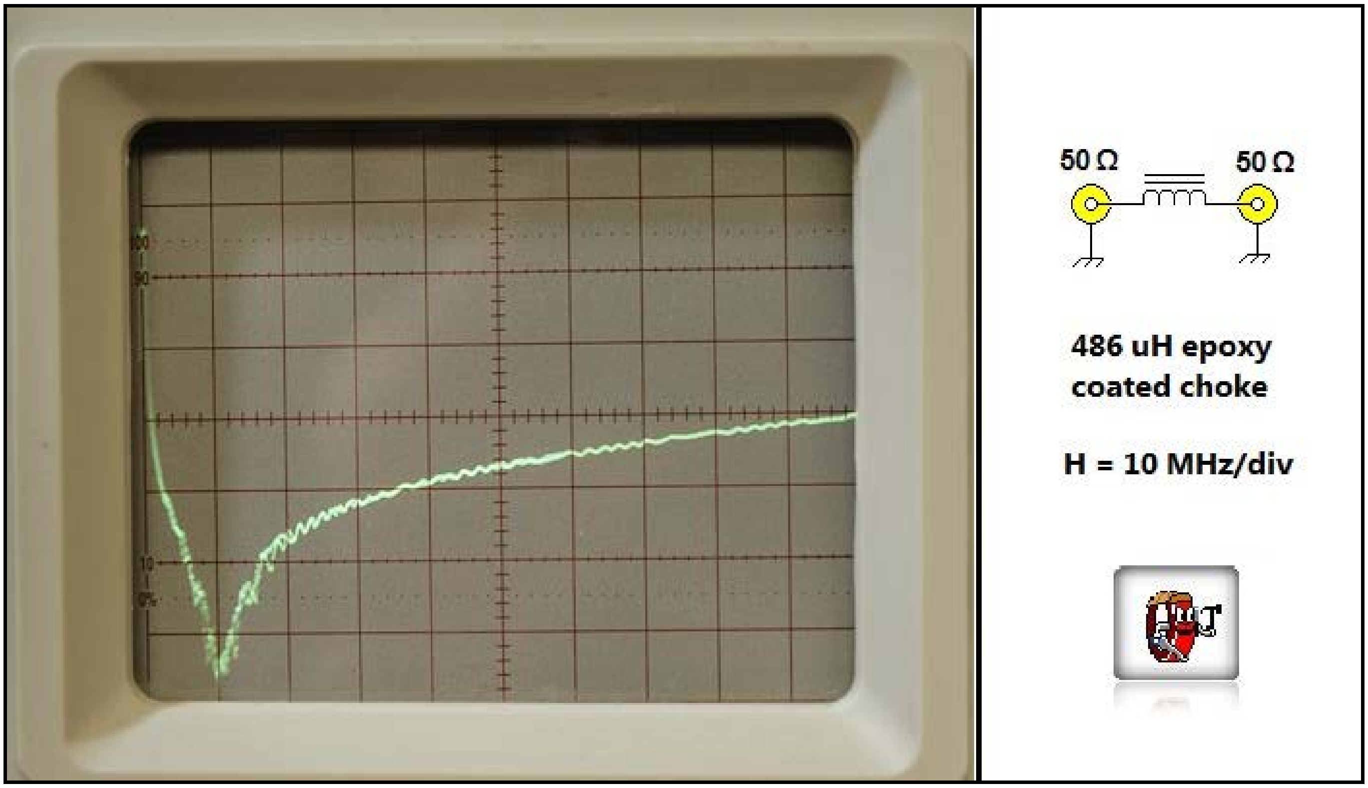
Above — Reference signal at -27 dBm. I used the exact test method shown in Section 5.

For those unaware, the spectrum analyzer screen is divided into 10 by 10 graticules. Each vertical division represent a 10 dB change; read down from the reference -27 dBm to measure the attenuation of the reference signal in dB. Horizontal divisions represent frequency; start at 0 on the left hand side and increment as specified on each figure.

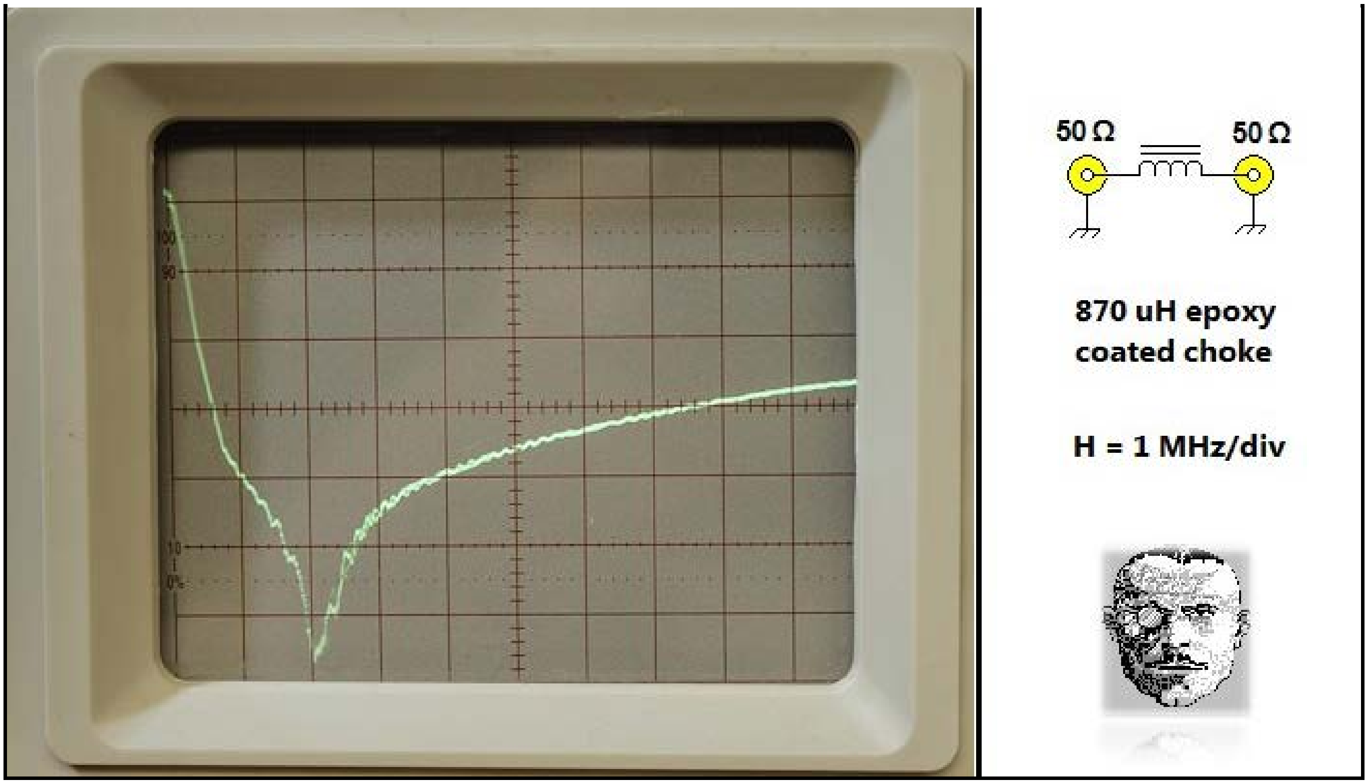
A Few Inductors



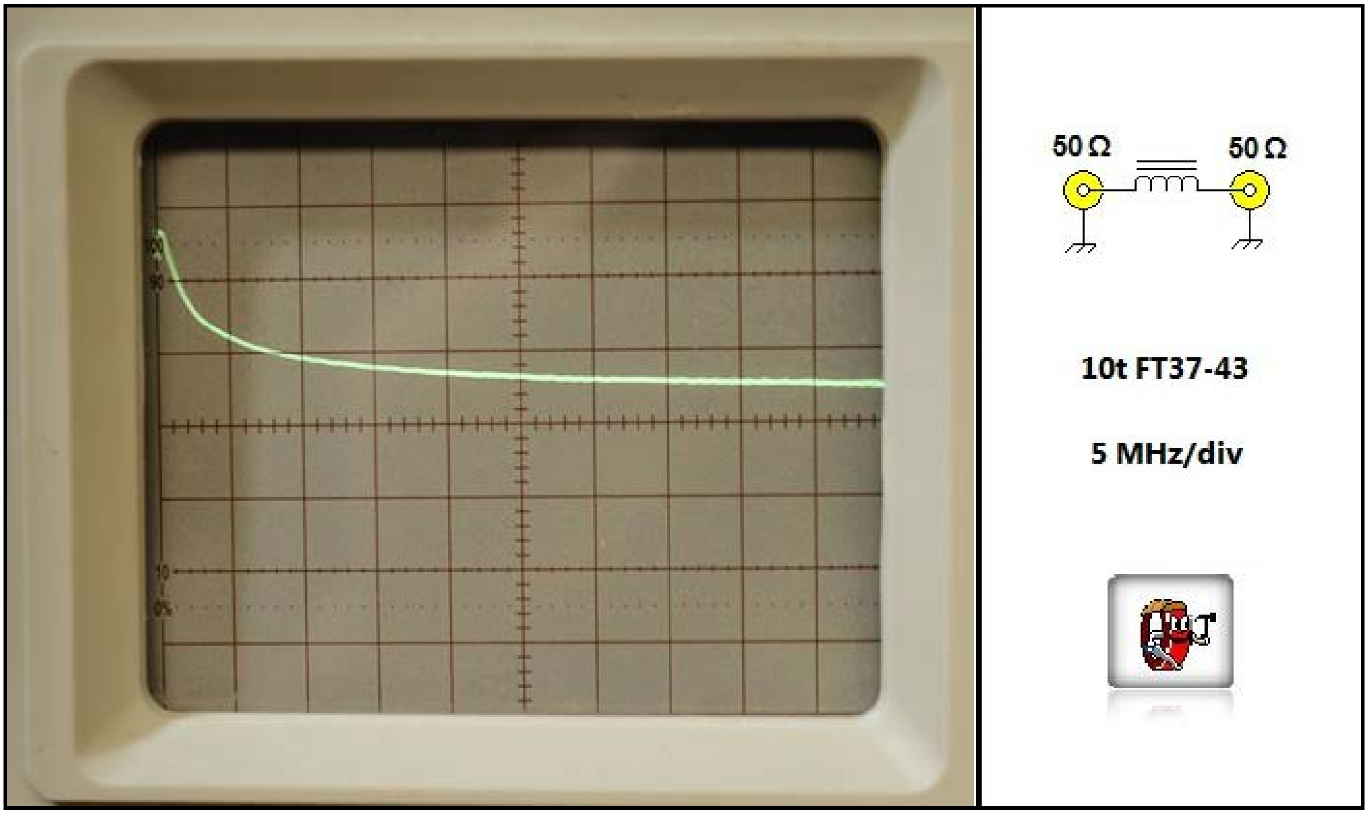
Above — A 19.9 uH epoxy coated choke that exhibits a primary SRF at 18 MHz and a second, smaller SRF at ~128 MHz. This wretched L gave me grief at 63 MHz. After measurement, I tossed it in the garbage can.



Above — A large, junk box choke with an SRF at about 10 MHz.



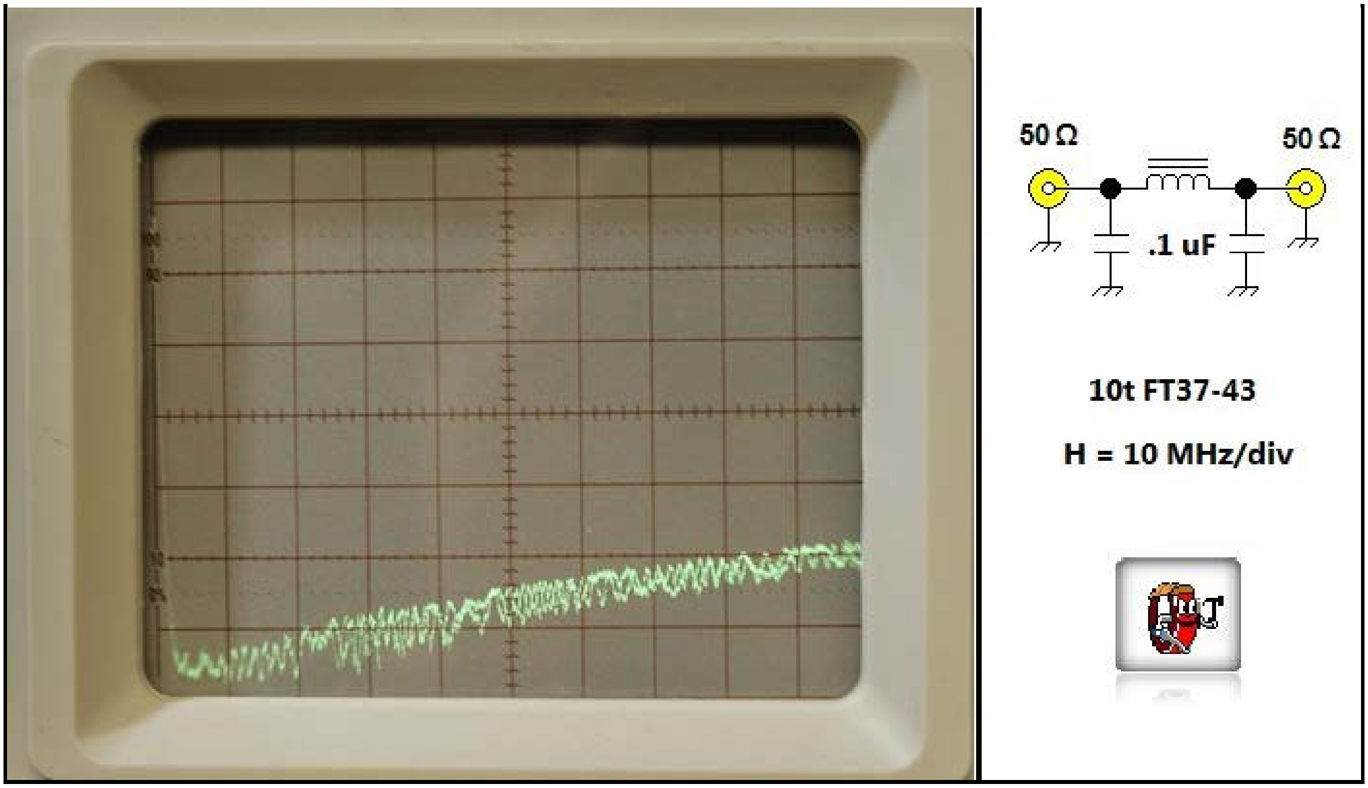
Above — I rarely use these big chokes: 870 μH with a SRF at about 2 MHz.



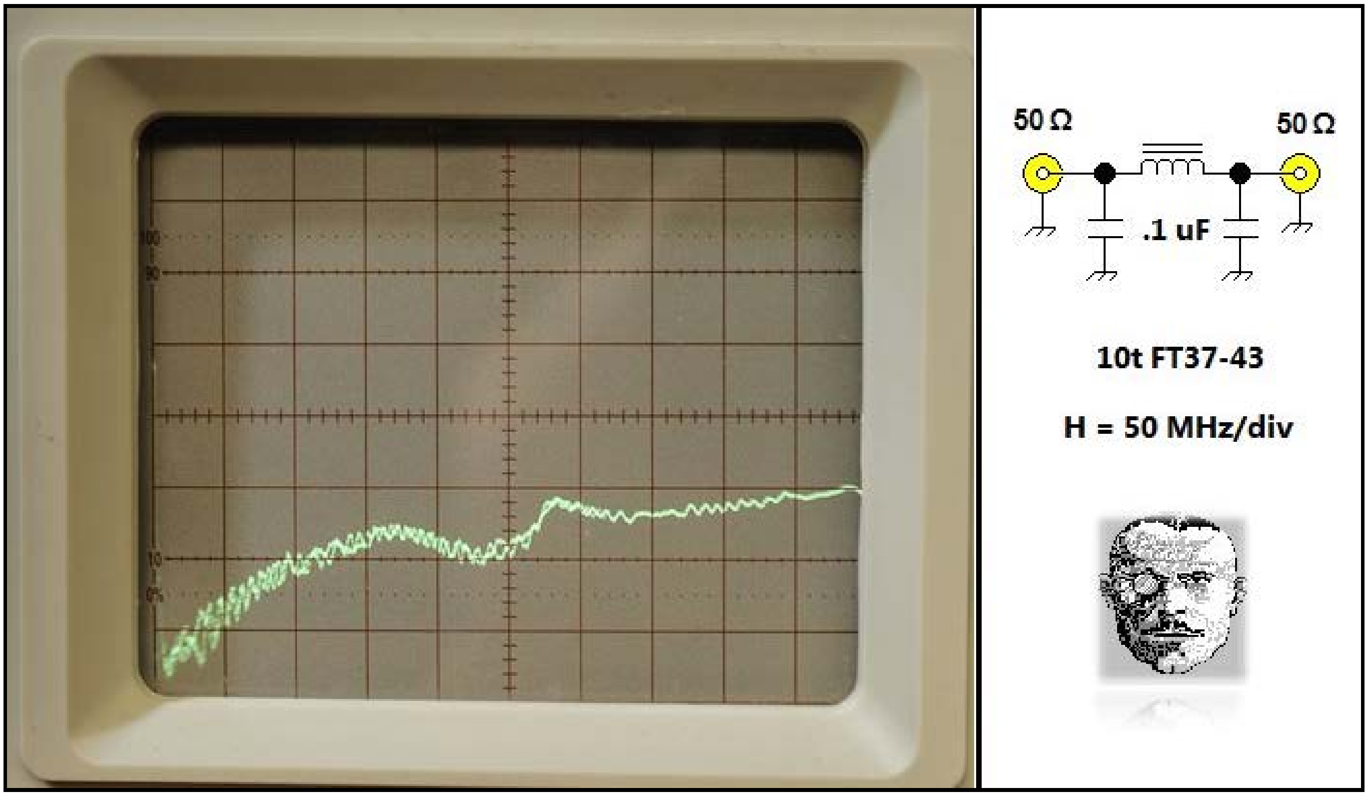
Above — A common L on our benches — 10 turns of #26 AWG on a FT37-43 ferrite toroid.

I couldn't measure the SRF with any span on my spectrum analyzer. I expect that a parasitic capacitance lies in parallel with the inductance, but the #43 material, with its low Q and high losses blankets the usual deep notch we see when the L exhibits a higher Qu.

10 Turns on a FT37- 43 with a Bypass Capacitor Shunting Each End

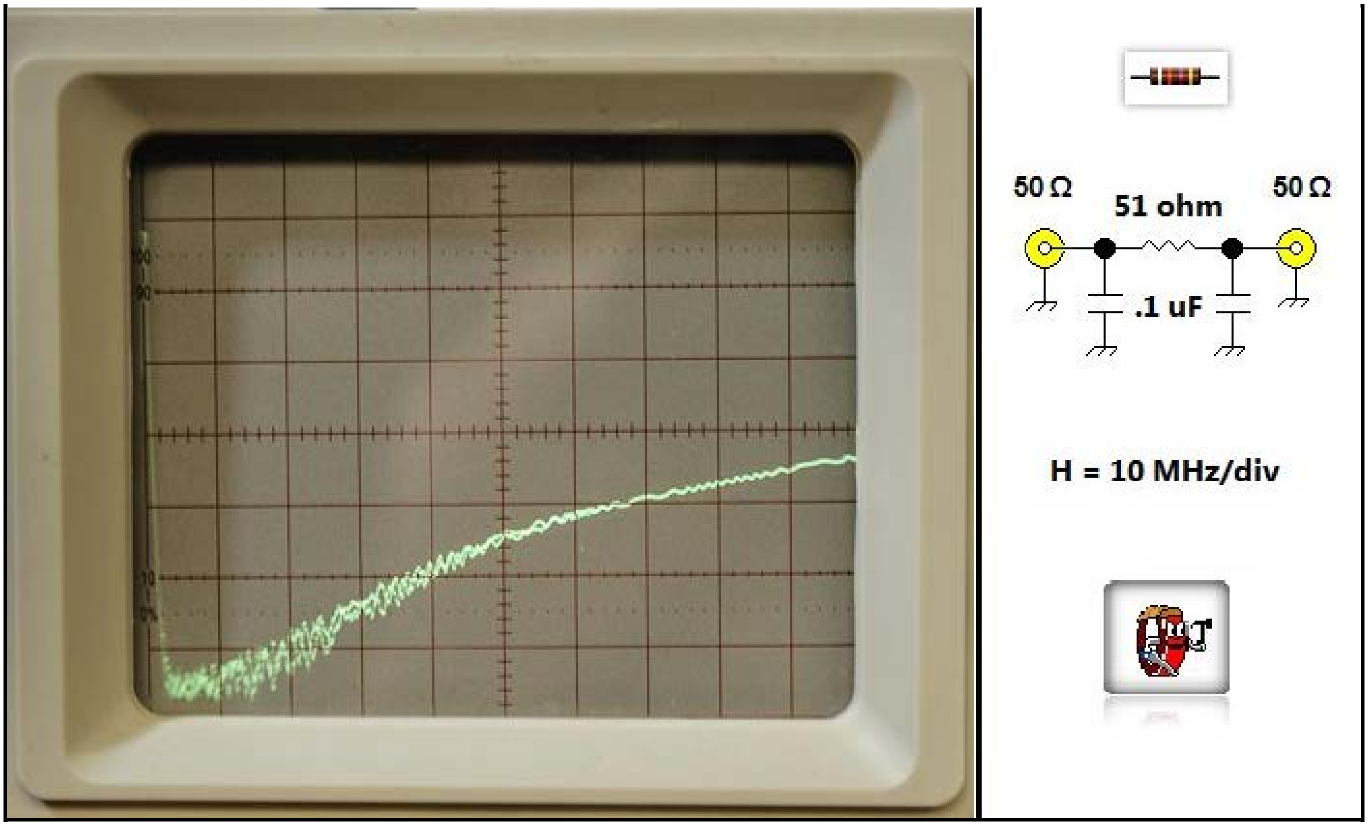


Above — Look at the big difference after adding shunt capacitors to a 10 turn FT37-43! Even at 100 MHz the attenuation lies nearly 50 dB down. Now I understand why Wes says **decouple plus bypass** when filtering our DC lines and so forth.

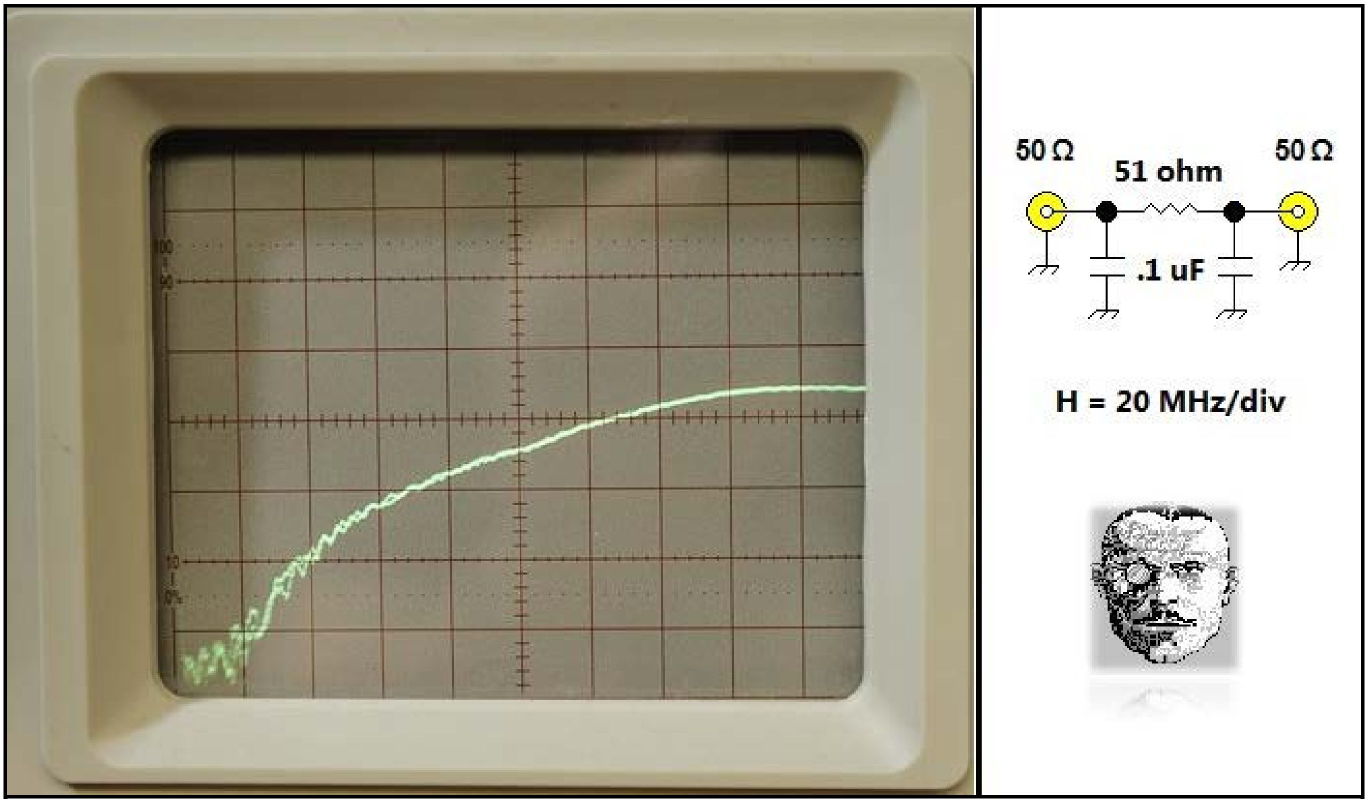


Above — The 10 turn FT37-43 coil with 0.1 μF shunt caps measured out to 500 MHz. Pardon the camera shake; I took all the photos hand held to save time.

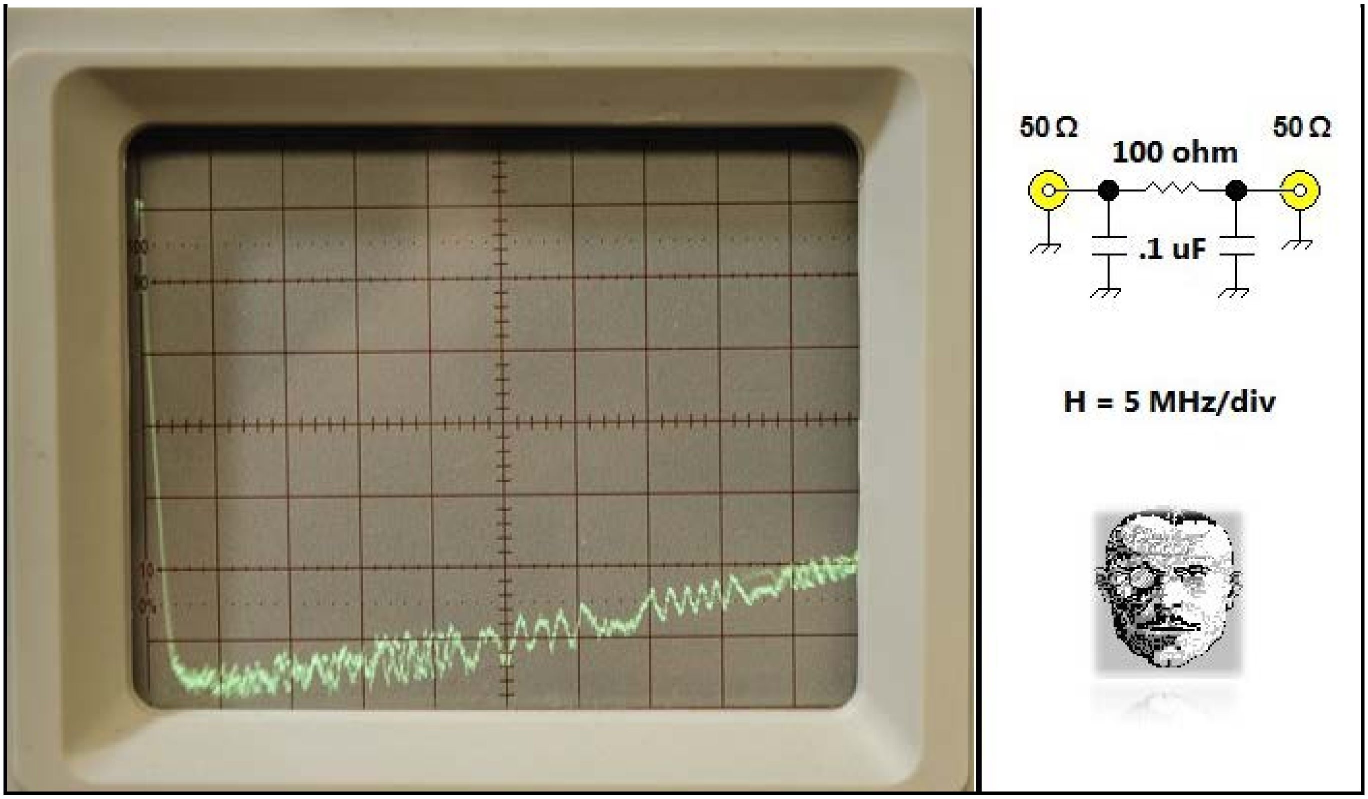
A Resistor with a Bypass Capacitor Shunting Each End



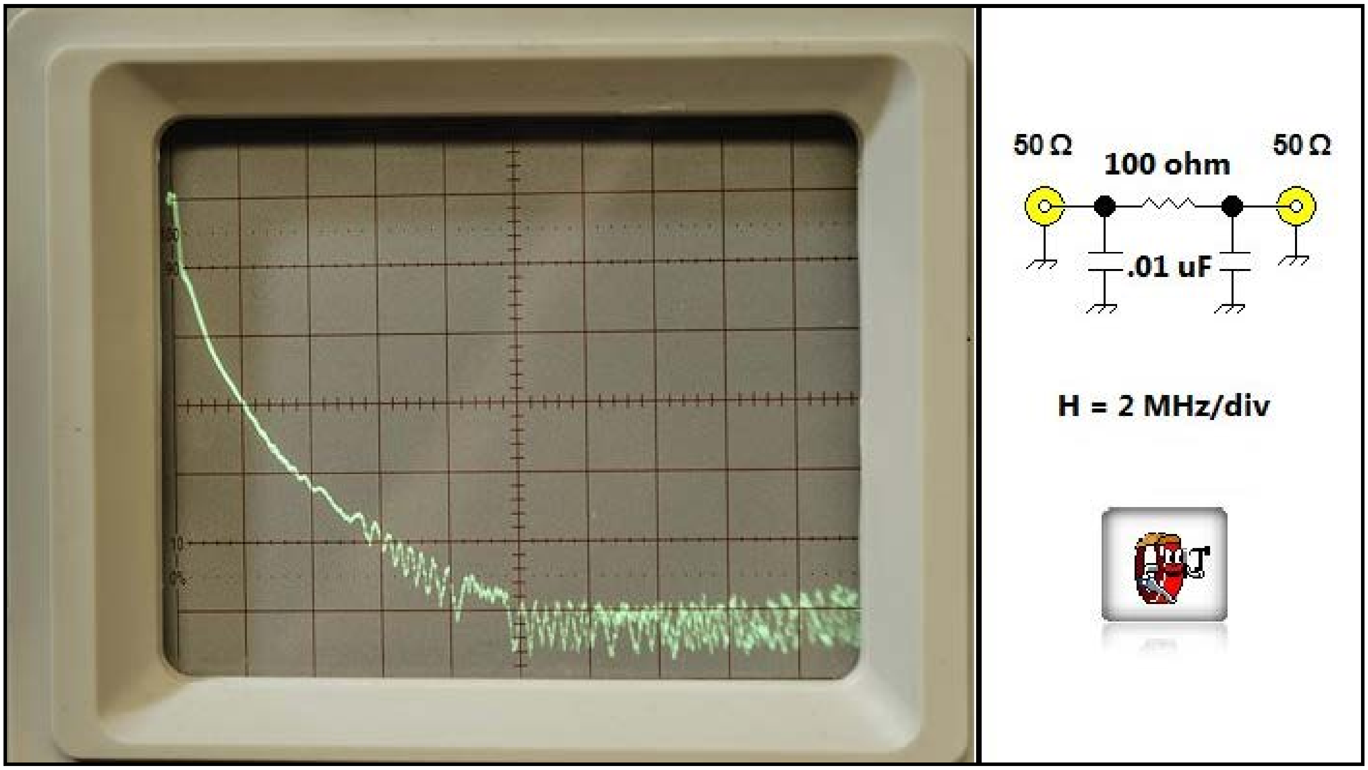
Above — A 51 Ω resistor bypassed with 0.1 µF capacitors at each end spanned out to 100 MHz. Even at 30 MHz, the attenuation looks stellar.



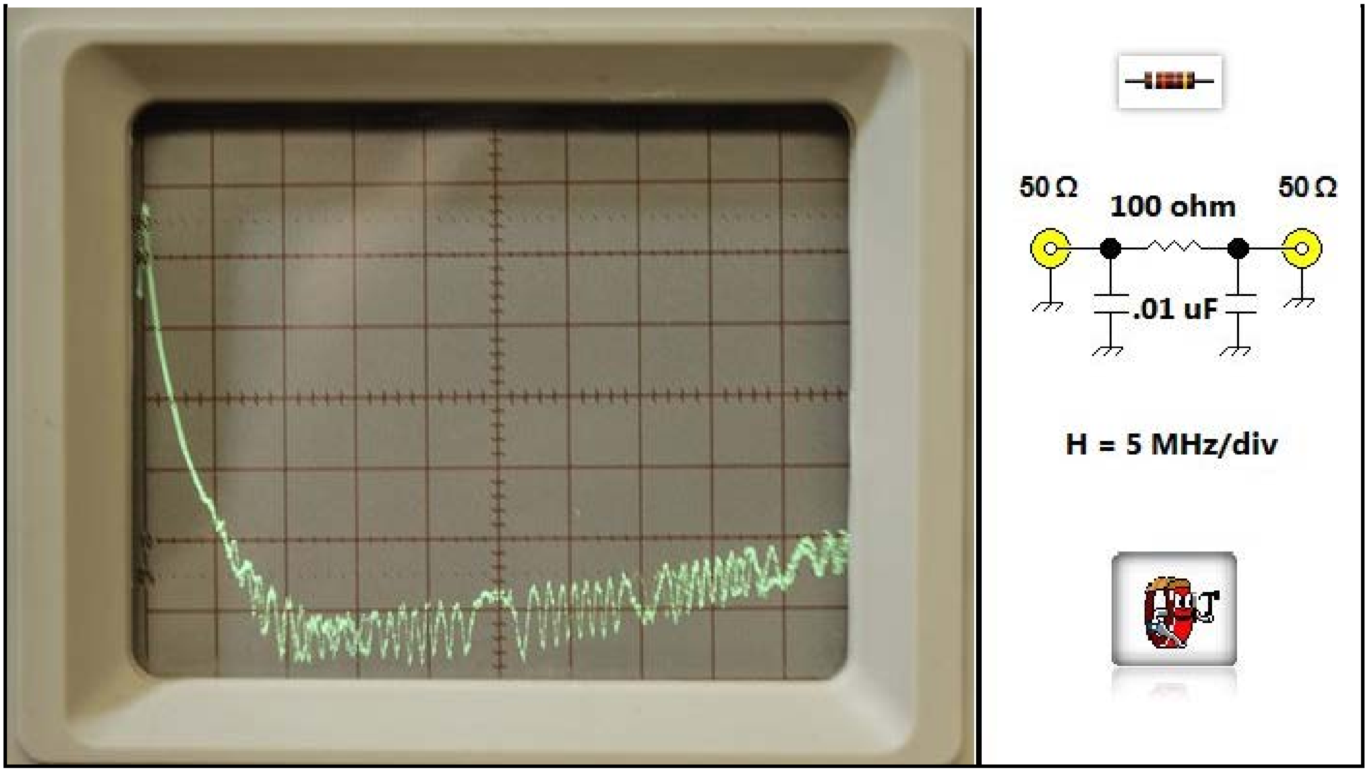
Above — The 0.1 µF bypassed 51 Ω resistor out to 200 MHz. I often use a 51 Ω decoupling resistor with appropriate capacitor values in active circuits that draw from 10 - 18 mA.



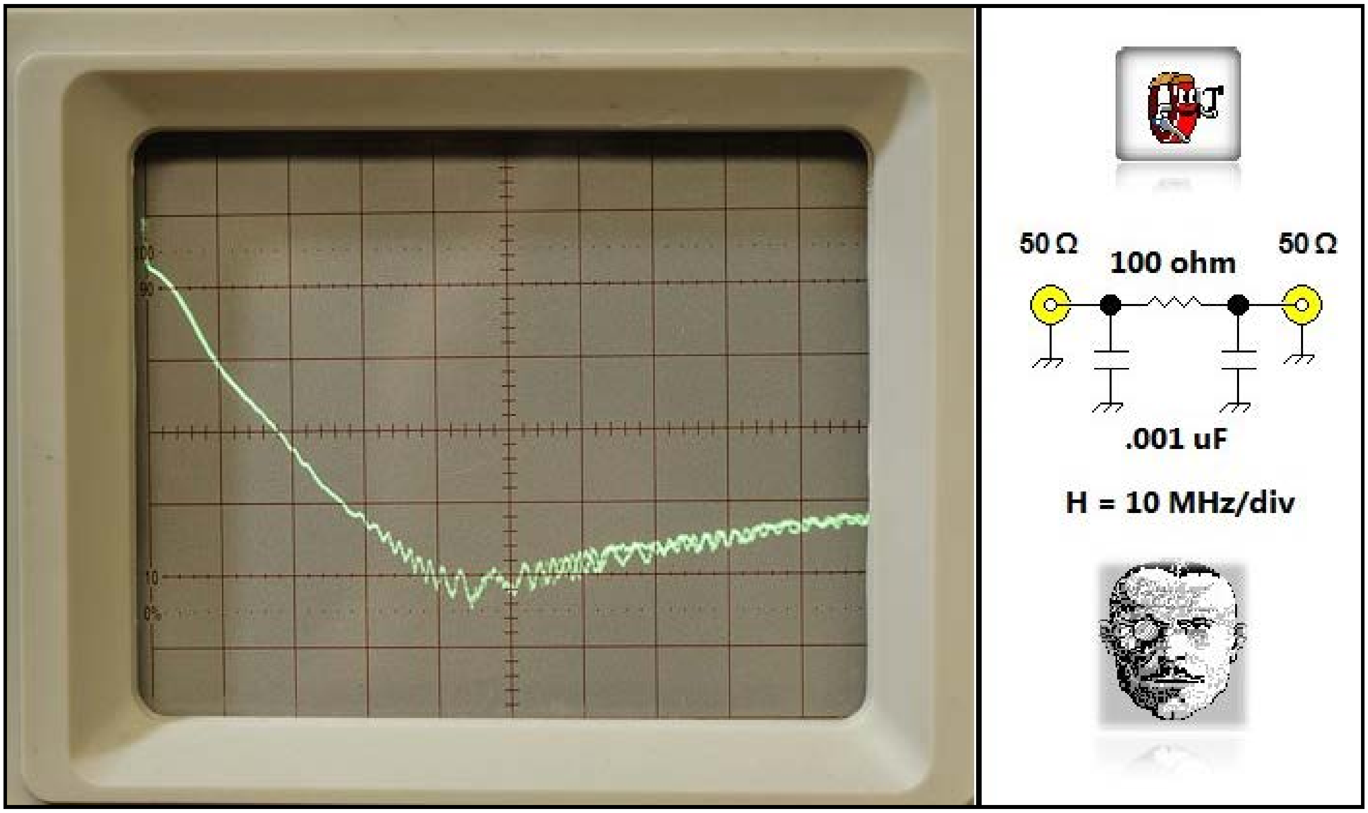
Above — The "bench standard"; a 100 Ω R with a shunt 0.1 µF at each end. We use this all the time. Even at 25 MHz, the attenuation looks around 55 dB down.



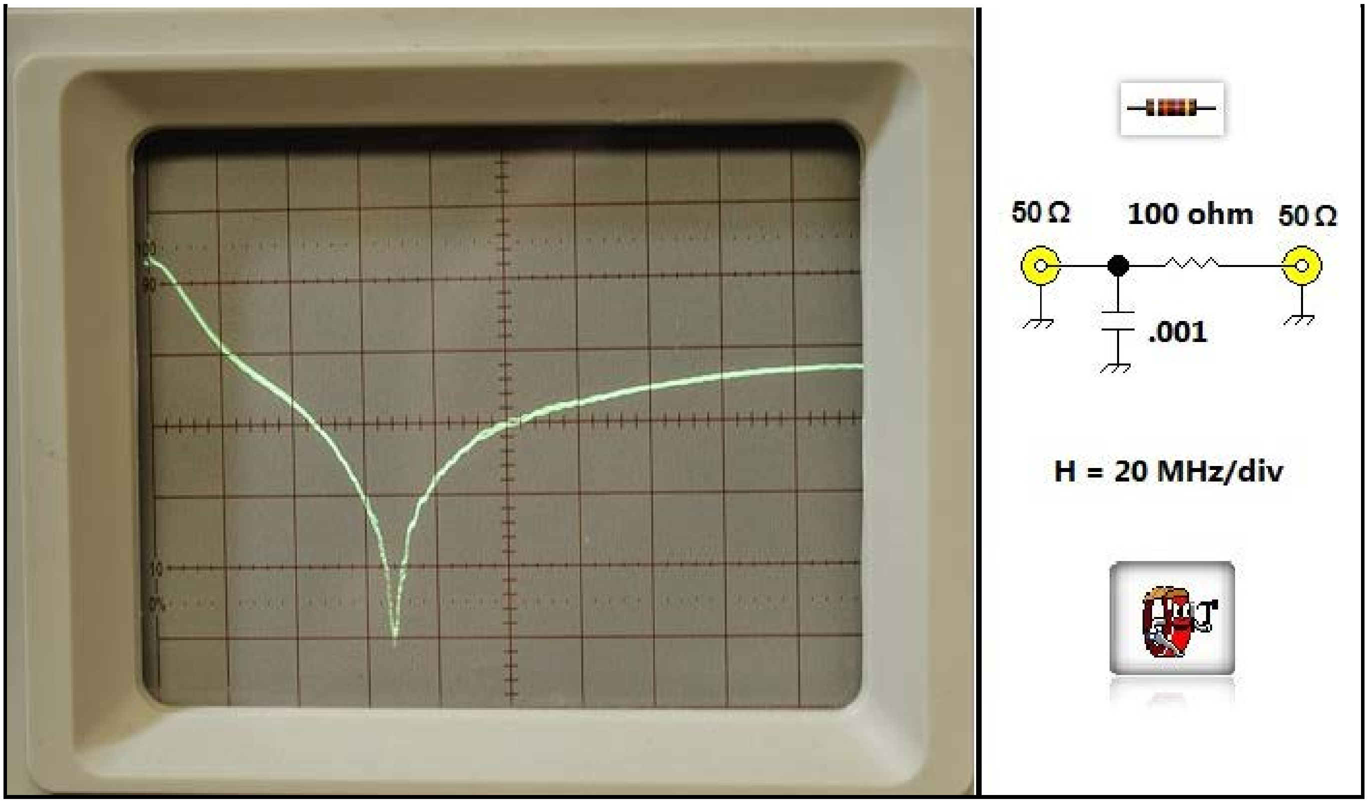
Above — The 100 Ω R with the shunt capacitors decreased to 0.01µF. At 6 MHz, we're about 50 dB down. From 10 to 20 MHz, the attenuation is about as high as I can measure.



Above — 100 Ω R plus 0.01 μF caps out to 50 MHz. I've used this combination of R and C for filtering at 50 MHz a lot.



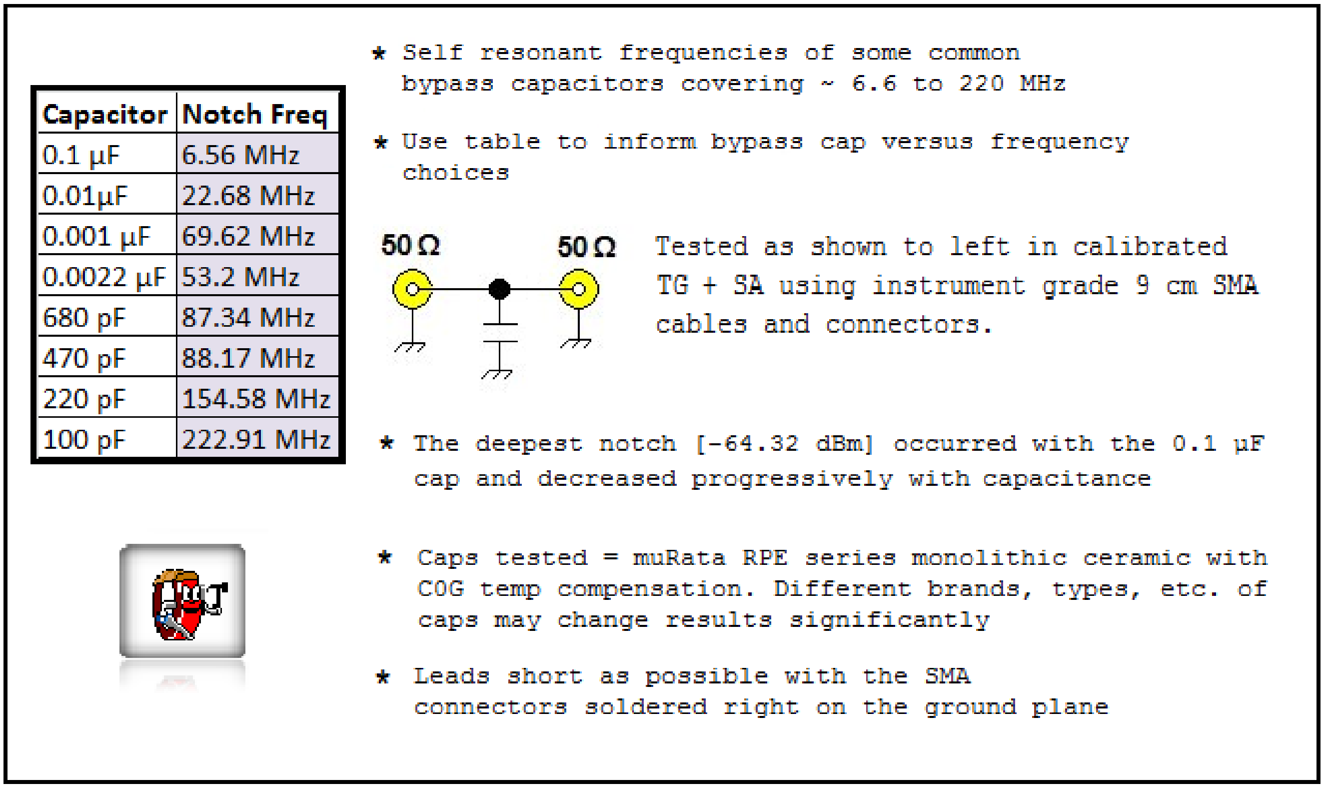
Above — 100 Ω R plus 0.001 μF capacitors out to 100 MHz. In my particular circuit, the attenuation at 50 MHz equals that of the 100 R + 0.01 μF C low-pass filter shown directly above.



Above — A 100 Ω R plus a single 0.001 μF capacitor. If you leave off 1 capacitor, a serious notch appears at ~ 68 MHz. If you flip the filter around so the bypass cap is on the right hand side, the tracing appears the same. This problem occurred with all the filters tested in all experiments. As possible, solder a suitable bypass capacitor on *both* sides of the R or L.

I encourage you to experiment with the SRF of coils and wideband decouple + bypass filters on your own.

QRP Posdata for Oct 2013 — SRF of some common bypass capacitors



★ Self resonant frequencies of some common bypass capacitors covering ~ 6.6 to 220 MHz

★ Use table to inform bypass cap versus frequency choices

Capacitor	Notch Freq
0.1 μF	6.56 MHz
0.01μF	22.68 MHz
0.001 μF	69.62 MHz
0.0022 μF	53.2 MHz
680 pF	87.34 MHz
470 pF	88.17 MHz
220 pF	154.58 MHz
100 pF	222.91 MHz

Tested as shown to left in calibrated TG + SA using instrument grade 9 cm SMA cables and connectors.

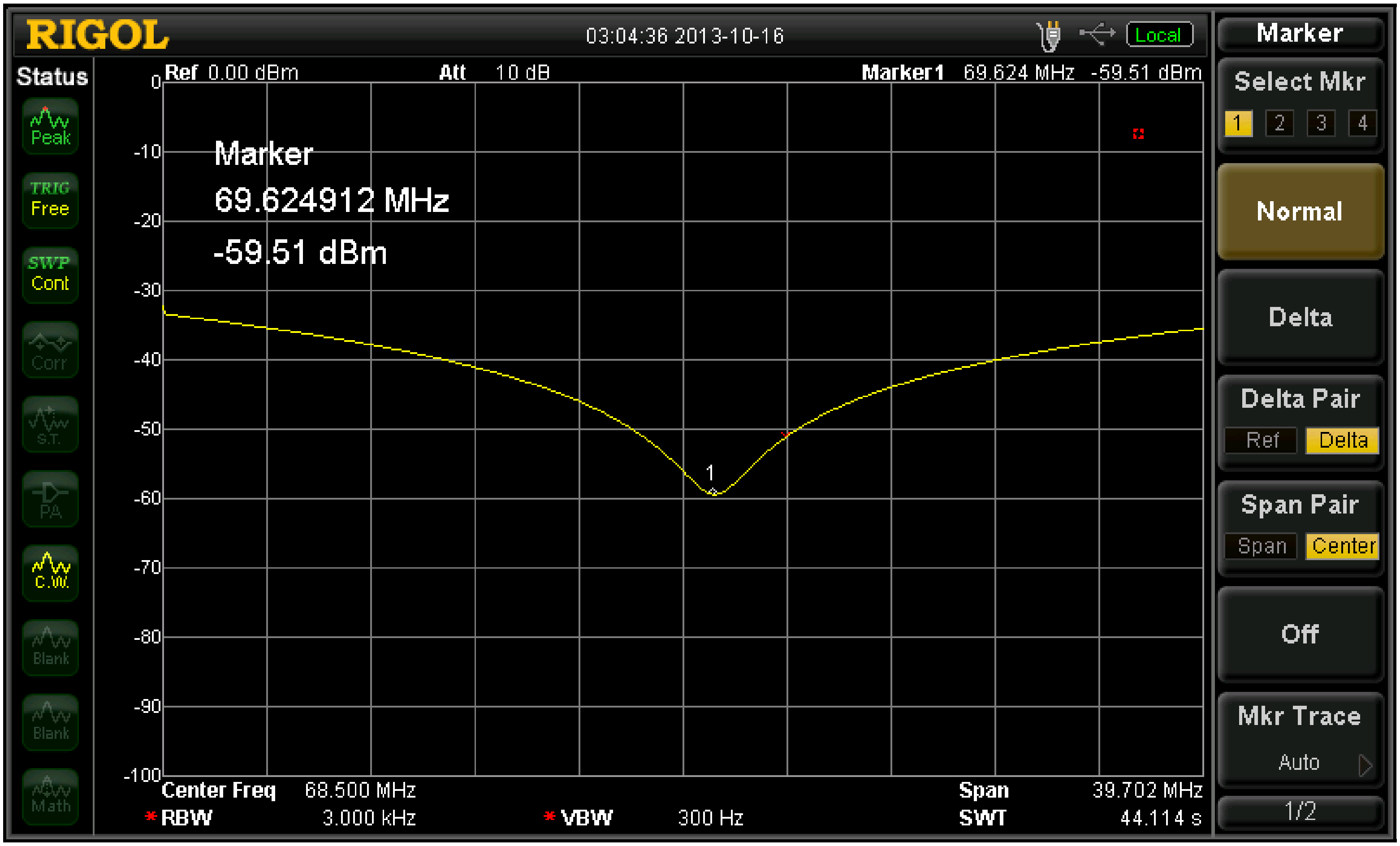
★ The deepest notch [-64.32 dBm] occurred with the 0.1 μF cap and decreased progressively with capacitance

★ Caps tested = muRata RPE series monolithic ceramic with C0G temp compensation. Different brands, types, etc. of caps may change results significantly

★ Leads short as possible with the SMA connectors soldered right on the ground plane

Above — A reference table showing the self resonant frequency of several comon value bypass capacitors in my parts collection. For example, if

I'm making a 21 MHz circuit, the best bypass capacitor choice from the table above = 0.01 µF. If possible, sweep the capacitors in your own collection to determine their SRF; or whether they're even suitable.

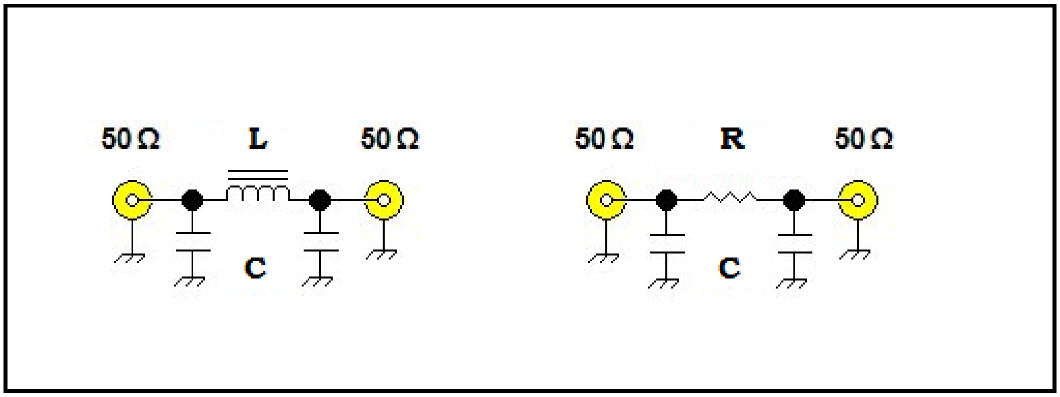


Ref 0.00 dBm
Att 10 dB
Marker1 69.624 MHz -59.51 dBm

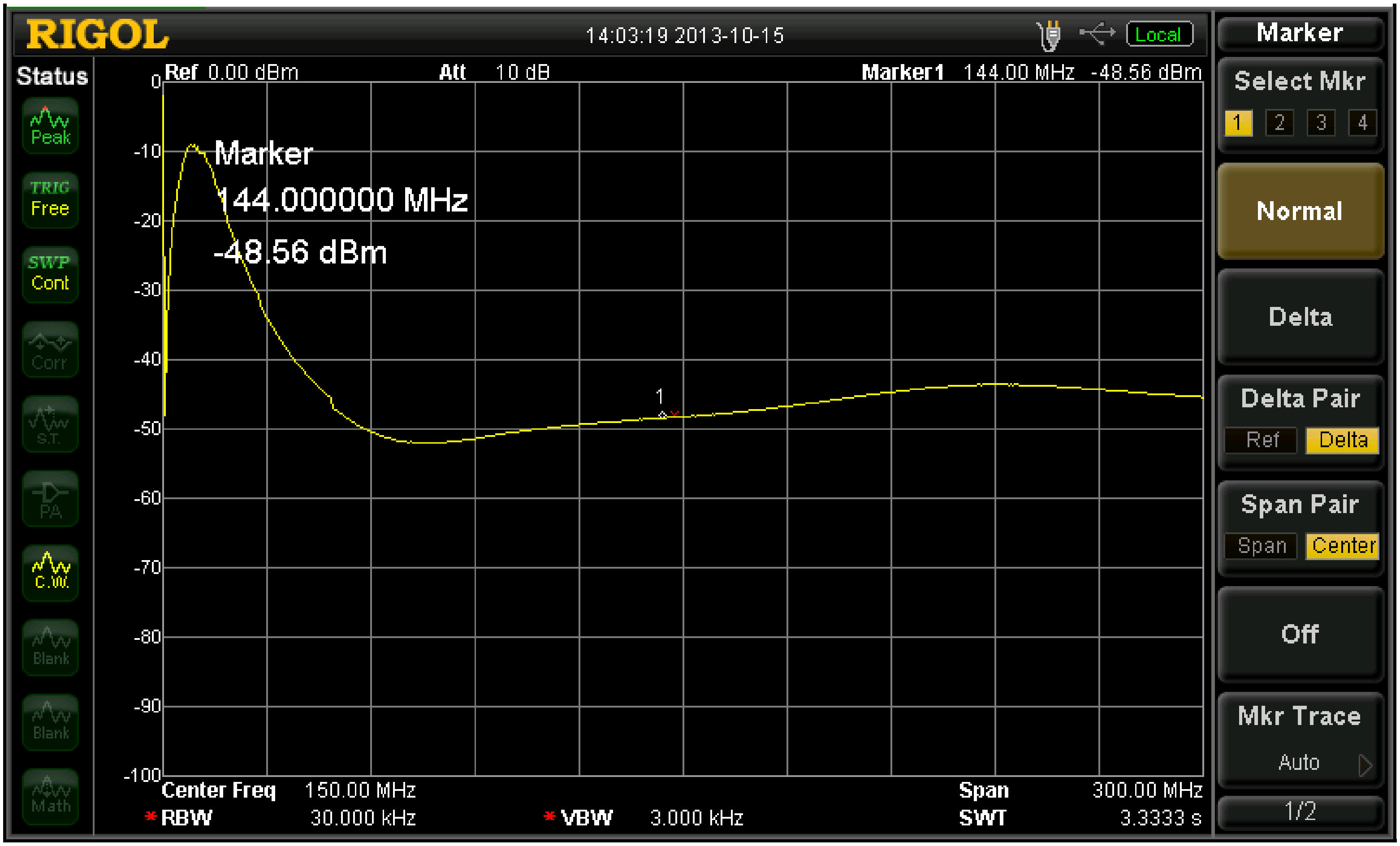
Marker
69.624912 MHz
-59.51 dBm

Center Freq 68.500 MHz
Span 39.702 MHz
*RBW 3.000 kHz
*VBW 300 Hz
SWT 44.114 s

Above — The close-in sweep of the 0.001 µF capacitor tabled above.



Above — As possible, stick 2 of your bypass cap values in a pi filter network with a series decoupling L or R to derive wideband filtration. For example, to filter your DC power lines.



14:03:19 2013-10-15

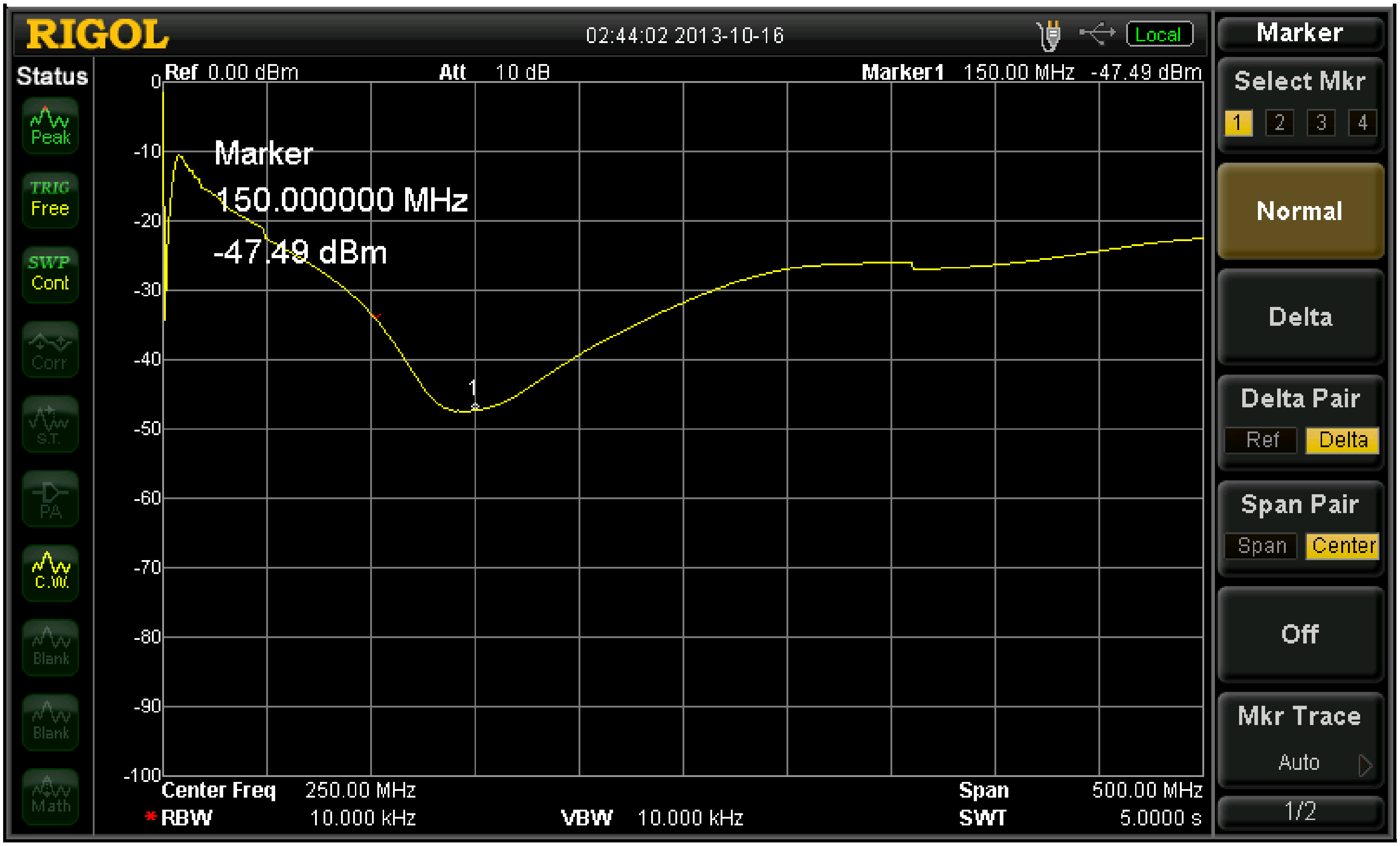
Ref 0.00 dBm Att 10 dB Marker1 144.00 MHz -48.56 dBm

Marker
144.000000 MHz
-48.56 dBm

Center Freq 150.00 MHz Span 300.00 MHz

*RBW 30.000 kHz *VBW 3.000 kHz SWT 3.3333 s

Above — A 300 MHz sweep of a pi filter [220 pF + 1.2 μH + 220 pF] for the DC supply line of a 150 MHz oscillator. The SRF peak lies at 76.69 MHz, but this filter works okay out to about 200 MHz. I placed a marker at 144 MHz and could use this filter for the 2M band as well.



Ref 0.00 dBm

Att 10 dB

Marker1 150.00 MHz -47.49 dBm

Center Freq 250.00 MHz

Span 500.00 MHz

RBW 10.000 kHz

VBW 10.000 kHz

SWT 5.0000 s

Above — The network described above, except I replaced the 1.2 µH L with a size 0805 10 Ω resistor [220 pF + 10R + 220 pF] and swept to 500 MHz. I set Marker 1 on 150 MHz; the frequency of the oscillator I wanted to DC powerline filter. The resistor gives a bit more filter bandwidth around 150 MHz. A 51 or 100 Ω resistor will further increase the bandwidth while decreasing the attenuation depth somewhat. Although resistors incur a DC voltage drop, they avoid the potential of an unwanted SRF in your filter arising from a renegade inductor— and so, a resistor may pose a better choice for pi filtering DC lines and so forth. It's your call.

At HF and lower VHF, I've found a hole-through capacitor may sometimes filter better than a "garden variety" SMT counterpart. Click for a graphic that shows this. Presumably, the SMT cap exhibited a lower Q than the equivalent hole-though part. At some frequency above 200 MHz, the lead inductance of the hole-through capacitor may cause the opposite effect.

Further, on the VHF — Véronique web page, Section 6: I swept 3 capacitors including an ultra-high Q SMT part.

RF Workbench Page 5

Welcome to part 5 of a web series exploring basic RF measurement and bench practices. This installment builds on the information from RF Workbench Parts 1 - 4.

In RFWB #5 I share a hodge-podge of thoughts and circuits concerning power measurement on the beginner-level RF workbench. Consult EMRFD for more support. Big thanks to my mentors: Wes, W7ZOI. Bob, K3NHI and John, K5IRK for their support as I advance to the basics.

Power Measurement Empowers You

Before embracing the 50 Ω RF environment, I misjudged the need to quantify small signal power — now I get that we measure lots of low-level signals on the 50 Ω RF workbench. Whether you're driving a mixer RF port with -30 dBm to reduce spurs, or tweaking an amplifer-under-test to exact the best S21, low-level RF power measurement is fundamental to fruitful RF design.

1 way to measure low-level RF power includes building a log linear RF power meter (PM) based on the Analog Devices AD8307. The basic circuit I show posits that most of you measure from MF to HF and don't need a PM that reads flat into UHF and further; a simple, 2 chip circuit might even prompt you to actually build a barebones PM for your QRP workbench.

Search for and download the Analog Devices AD8307 Revision D datasheet — it's definitely worth a read. Kudos to the design team that brought us a truly milestone device for low-cost power measurement.

Analog Devices offers a whole family of log-amps at different frequency ranges — for example, the AD8311 Log Amp/Detector covers from ~100 MHz to 2500 MHz. A sister product, the AD8302 2.7 GHz RF / IF Gain Phase Detector looks amazing.

1. A Barebones RF Power Meter

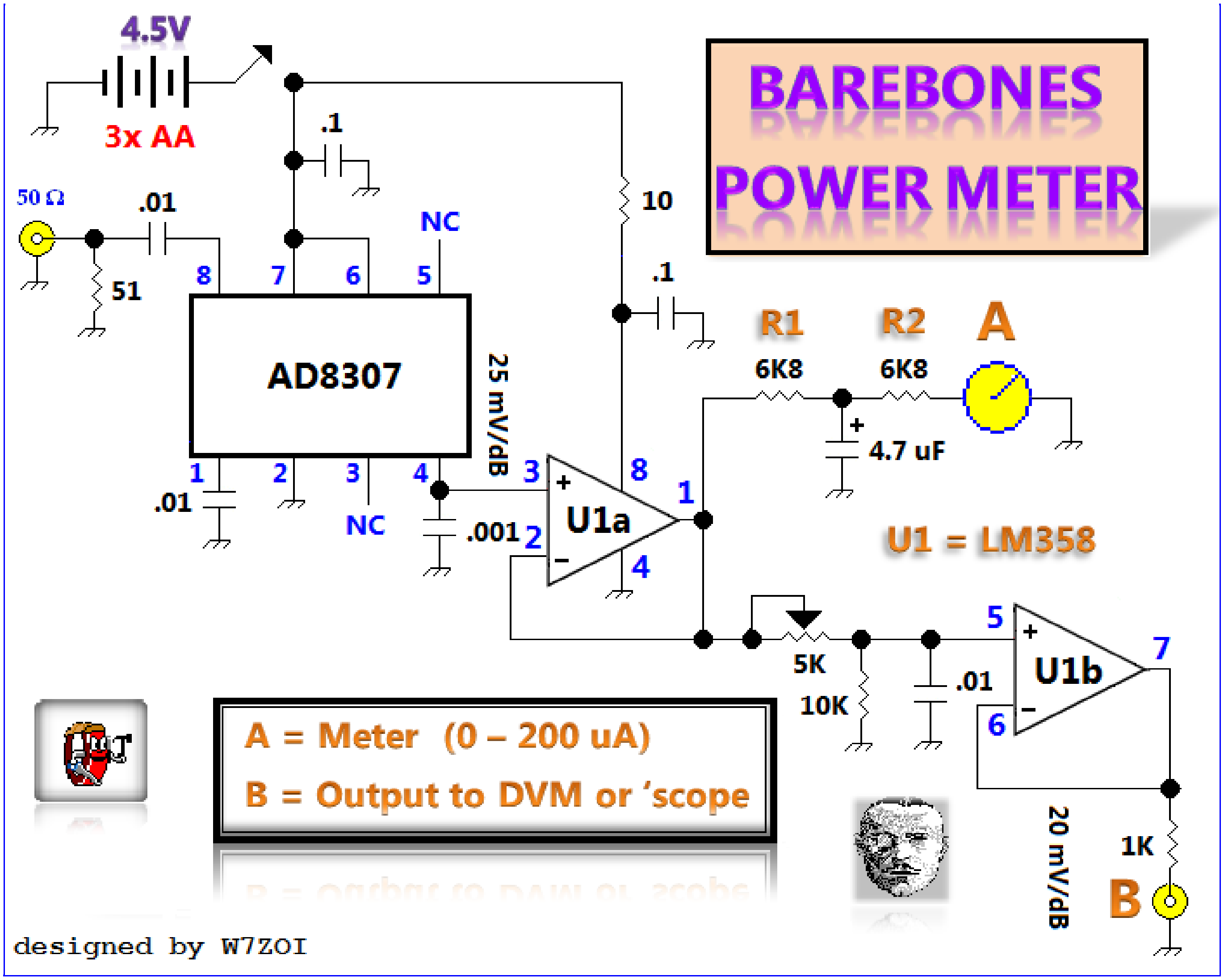
Lacks the usual input frequency compensation network needed to keep power measurement flat over several hundred MHz — since most input compensate by attenuating HF, this simple version offers stronger sensitivity at HF than usual.

Good from MF up to about 100 MHz.

4.5v B+ to eliminate power supply decoupling issues and to give greater battery life than a 9v battery.

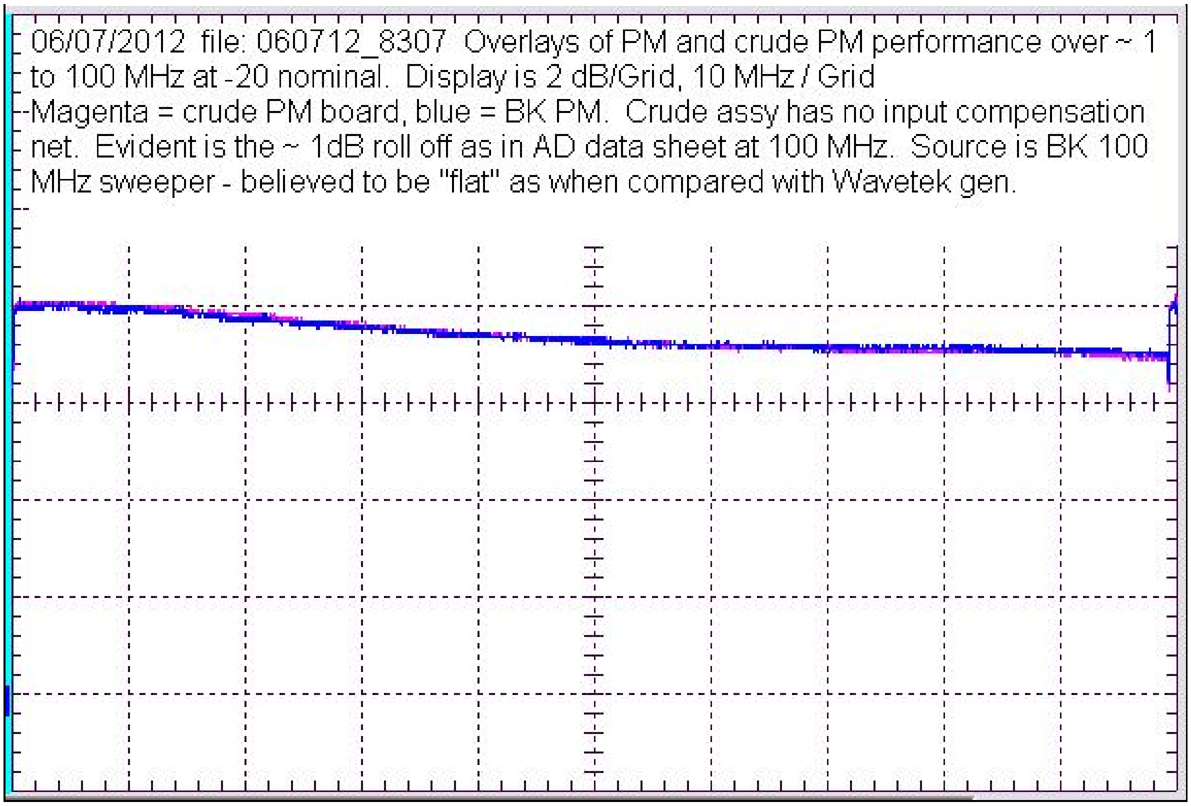
Minimum input power about -70 dBm. Maximum input power +15 dBm. Requires a metal box.

Big thanks to Wes, W7ZOI for letting me present his simple RF power meter. I found *numerous* AD8307-based RF power meter designs in periodicals and on the web and the writers devised simple to elaborate input compensation networks to establish a flat response out to 500 MHz — involving parts such as nH-level inductors, chip resistors and capacitors. This PM goes the opposite direction; a plain circuit for you builders who measure under 100 MHz; particularly at HF.



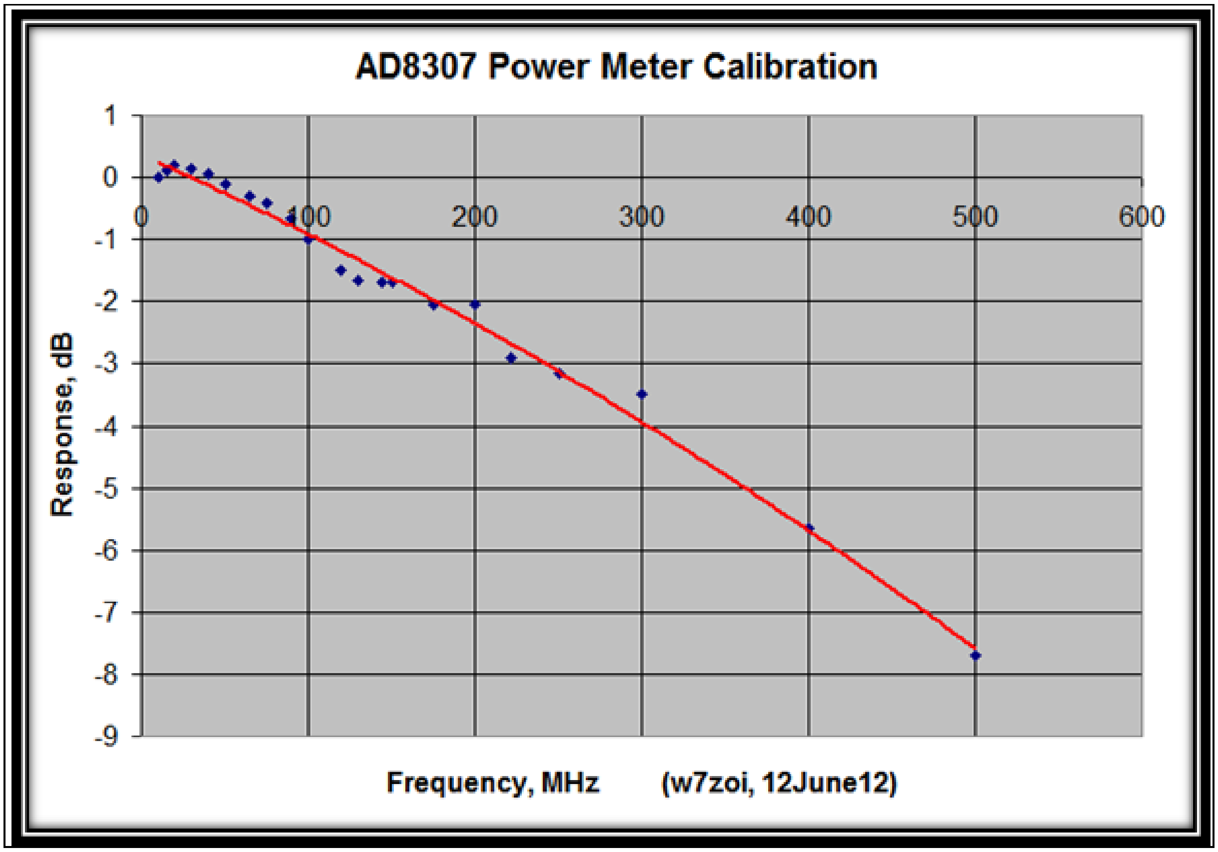
Above — the complete Barebones PM schematic. Presented with the permission of Wes, W7ZOI.

AD8307 Input Pin 8: Some Graphs and Notes:



06/07/2012 file: 060712_8307 Overlays of PM and crude PM performance over ~ 1 to 100 MHz at -20 nominal. Display is 2 dB/Grid, 10 MHz / Grid
Magenta = crude PM board, blue = BK PM. Crude assy has no input compensation net. Evident is the ~ 1dB roll off as in AD data sheet at 100 MHz. Source is BK 100 MHz sweeper - believed to be "flat" as when compared with Wavetek gen.

Above — The lack of input compensation might raise a few eyebrows! A sweep of the Barebones PM by Bob, K3NHI shows a reasonably flat response out to 100 MHz that's comparible to some of the 0-100 MHz range of wider sweeps in the published compensated circuits I've read.



Above — A different Barebones PM plot from Wes, W7ZOI that goes out 500 MHz. For HF work, this power meter proves adequate for QRP HomeBuilders. By all means, add some input compensation if you want to — good circuit examples abound. For example, EMRFD Figure 7.13, or Bob Kopski's — An Advanced VHF Wattmeter referenced in Section 5.

AD8307 specifications allow a typical +/- 0.3 dBm "ripple" in the input to output transfer characteristic. Bob, K3NHI verified this ripple in his lab.

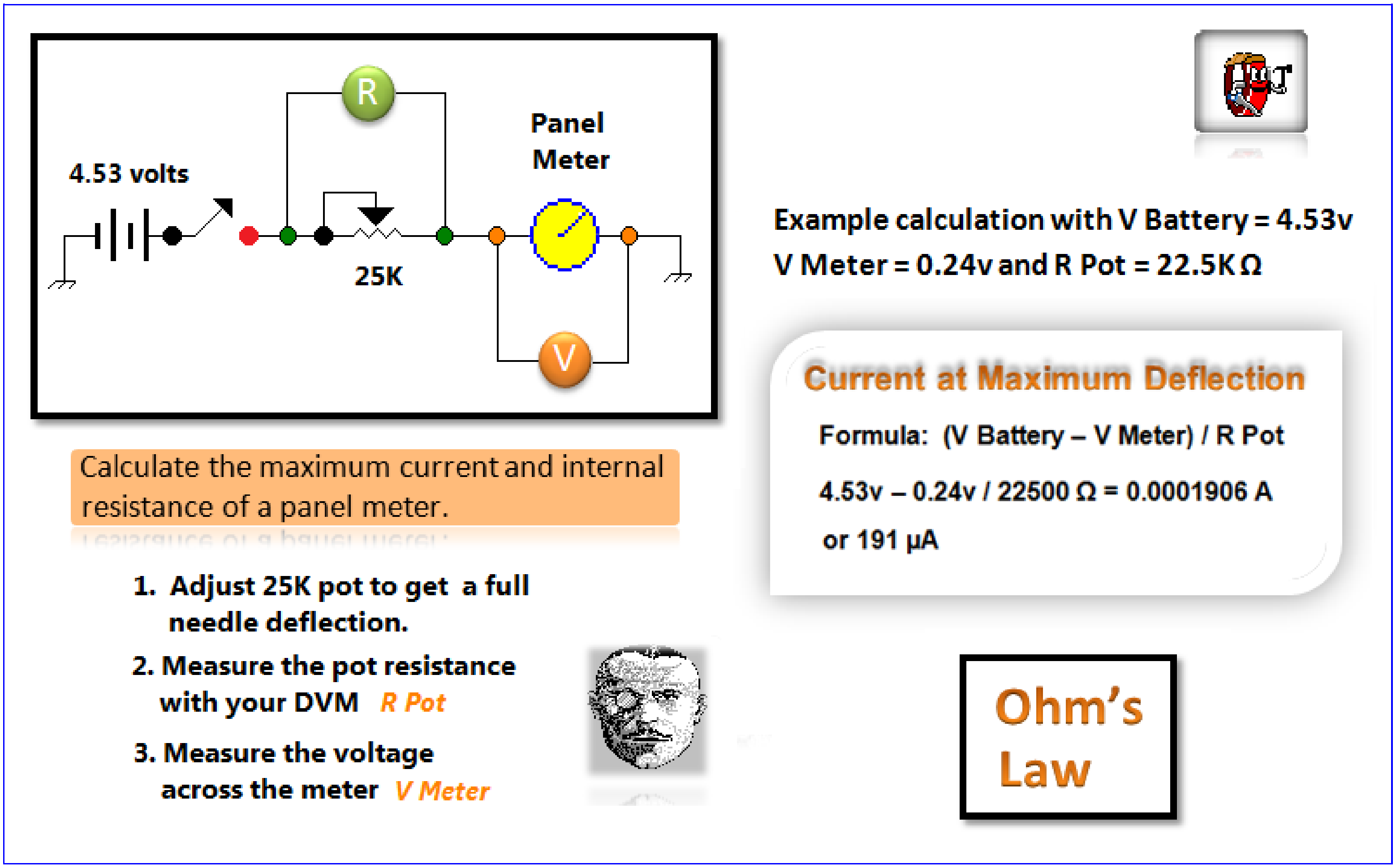
Therefore; depending on the position of a signal in the transfer curve, a move between 2 different power levels may yield as much as 0.6 db peak difference error. Figure 8 of the Rev D datasheet shows this. Figure 24 plus associated text tells why this occurs: the transfer characteristic is a chunk or segmented realization of a log transfer characteristic. It's really good — just not perfect.

If you consider the cost of thermal sensor power meters from Agilent, Techtronix, or even the Mini-Circuits PWR-6GHS, the AD8307 seems a bargain.

Measuring U1a Pin 1 With a Panel Meter

The first output at Pin 1 drives nearly any junkbox DC panel meter you might own The schematic specifies a 0-200 uA movement, however a 1 mA meter movement also works by tweaking R1 and R2 to establish the correct current. In the circuit shown the R = 13.6K. This R establishes the current required to drive the 200 uA meter Wes used in his design — nothing more. A wide variety of panel meters work because the drive current comes from an op-amp. Don't order an expensive panel meter just because you want to employ a 100 or 200 uA model — choose your meter because it affords good needle movement and resolution to allow accurate power measurement.

Although elementary for some, the following diagram shows you how to measure the maximum current for your meter: 100 uA, 200 uA, etc.



Example calculation with V Battery = 4.53v
V Meter = 0.24v and R Pot = 22.5KΩ

Current at Maximum Deflection

Formula: (V Battery – V Meter) / R Pot

4.53v – 0.24v / 22500 Ω = 0.0001906 A

or 191 µA

Calculate the maximum current and internal resistance of a panel meter.

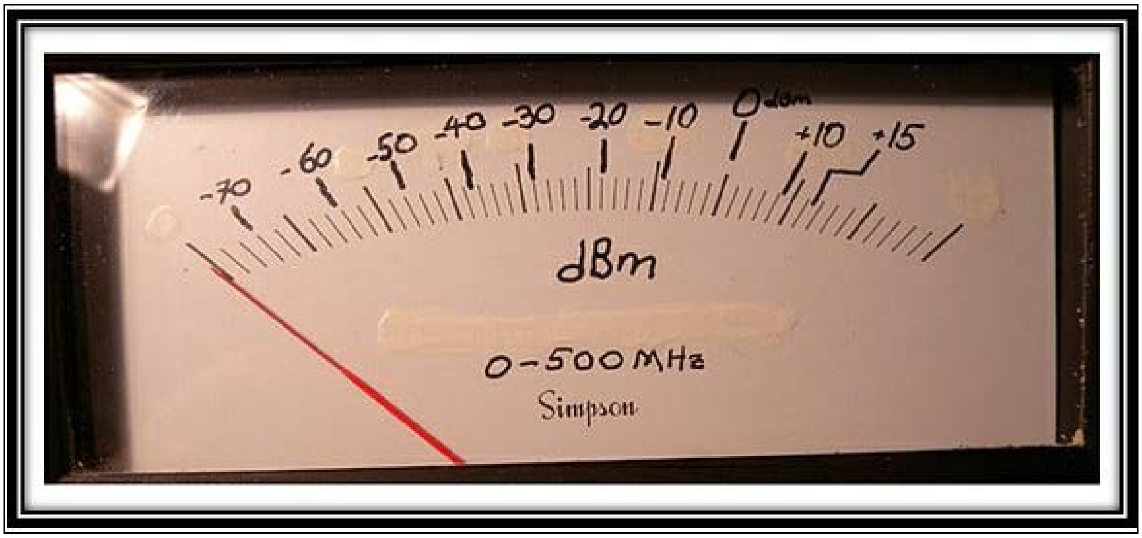
1. Adjust 25K pot to get a full needle deflection.
2. Measure the pot resistance with your DVM *R Pot*
3. Measure the voltage across the meter *V Meter*

Ohm's Law

Above — In my shoebox sat 4 different panel meters. I measured 1 meter and show the math above — a 191 µA meter.

Above — An antique 1 mA panel meter driven with 1 mA to achieve full deflection.

Meter resolution presents the biggest problem for junkbox panel meters and if you look around the web, you'll see some great examples how builders calibrated and/or marked the scale on their AD8307 circuit panel meters.



Above — Panel meter markings on my friend Peter's power meter.

I personally like my panel meters to read about 75% of full deflection at 7 dBm, but it's really your choice. Adjust R1 and/or R2 up or down to give the desired amount of meter deflection for whatever panel meter you own.

You may apply Ohm's law to figure out the maximum in-situ current for any panel meter. In the schematic the total R is 13.6K, so when you apply +10 dBm applied to the power meter input, the current in the panel meter will be 2.0v /13.6K = 147 uA. The maximum 200 uA will occur with a Pin 1 voltage of 2.72v. The op-amp won't go all the way to the positive rail, but If it did go all the way to 4.5v, you would hurl 331 uA into the panel meter. This exceeds the meter's uppermost scale, but likely won't destroy it.

The 4.7 uF cap can be any small uF capacitor value and low-pass filters the DC to smooth out the meter movement. I used a 10 uF in my breadboard.

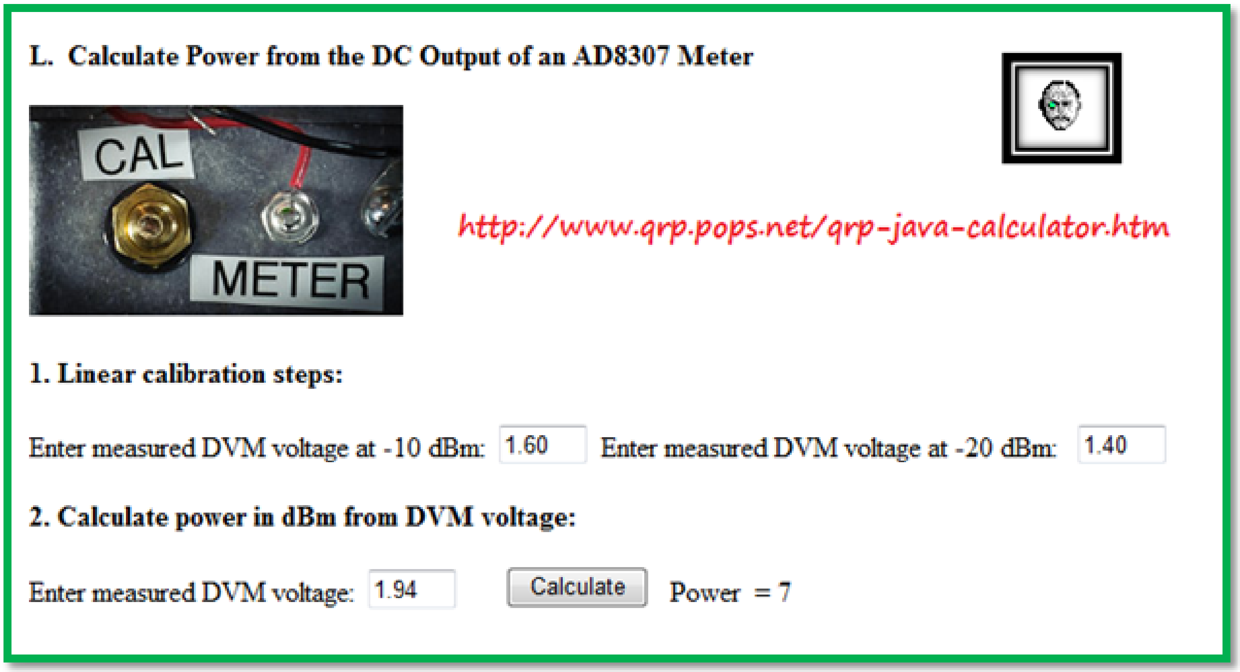
I normally view the panel meter to tune resonant circuits and observe trends; but not to precisely measure power — it's often more accurate to quantify power readings at Port B since this eliminates panel meter resolution issues.

How to Measure Power at Port B

We measure the DC voltage at port B and use equations or graphs to translate this voltage into an actual power reading after reference calibration. U1a and U1b are unity-gain voltage followers to buffer the AD8307 output. U1B features a 5K potentiometer in series with a 10K shunt resistor to form a voltage divider that changes the 25 mV/dB AD8307 output to 20 dB/mV. Tweak the 5K pot to calibrate to get as close to 20 dB/mV as possible, although since the LM358 buffers just pass on the DC voltage changes of the AD8307, the pot does not technically alter log linearity.

Wes published some essential notes on his web site. On page 8, you find his formula to convert a measured DC voltage into dBm. I wrote a program that incorporates his formula: Applet L on my Design Center web page; except that it takes calibration power at -10 dBm and -20 dBm.

Let's run through 1 example:



L. Calculate Power from the DC Output of an AD8307 Meter

http://www.qrp.pops.net/qrp-java-calculator.htm

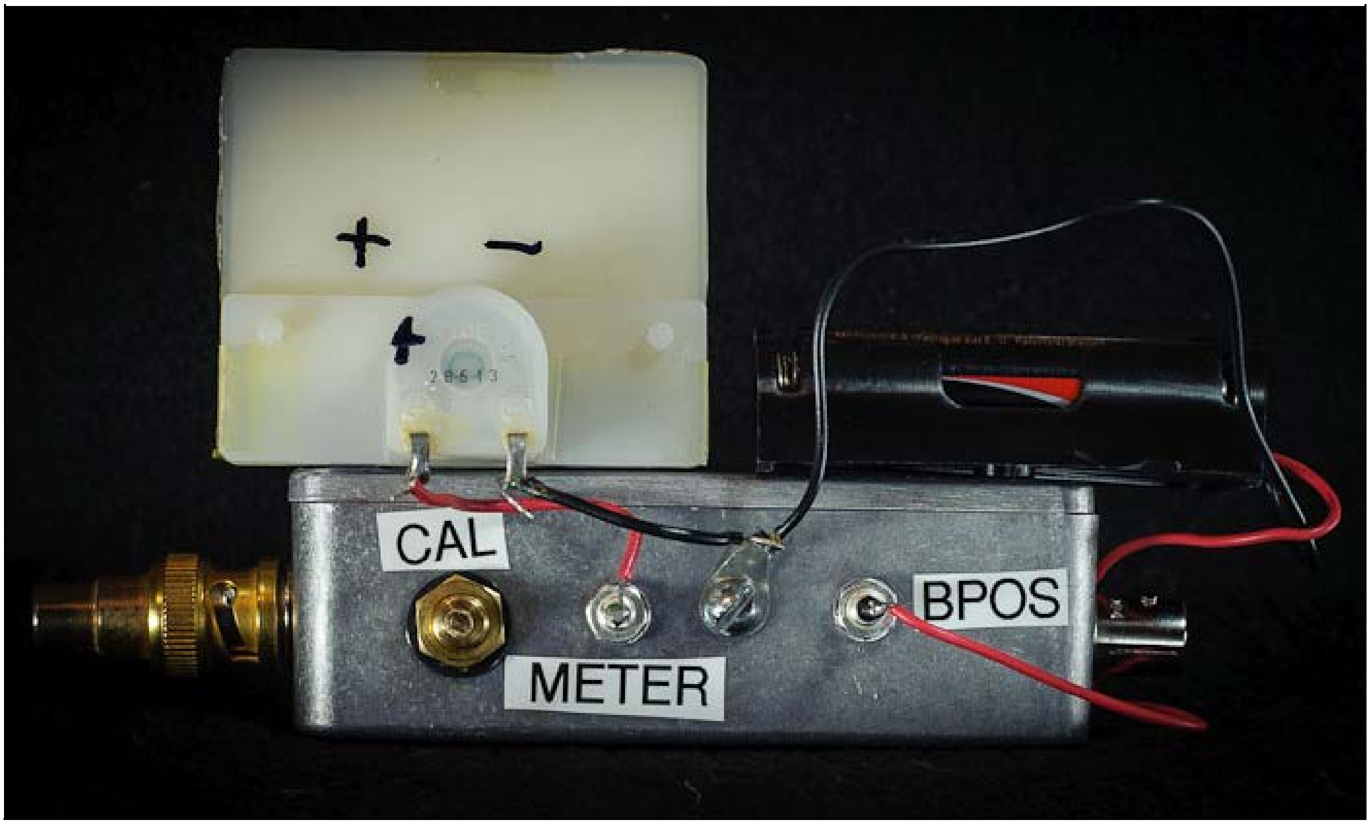
1. Linear calibration steps:

Enter measured DVM voltage at -10 dBm: 1.60 Enter measured DVM voltage at -20 dBm: 1.40

2. Calculate power in dBm from DVM voltage:

Enter measured DVM voltage: 1.94 Calculate Power = 7

Above — A screen capture of JavaScript Applet L. I calibrated my instrument by connecting a 10.0 MHz sine wave calibrator to the 50 Ω input port of my PM. At -10 dBm input power, my Port B reading was 1.6 VDC; at -20 dBm, I measured 1.4v at Port B. These 2 points establish the log linearity per Wes' notes. After this calibration, I measured the output power of a 4 MHz sine wave VFO that I designed to run at 7 dBm output power to drive the LO port of a diode ring mixer. Port B read 1.94 volts and when I entered this into Applet L and pushed <<Calculate>>, the VFO output power was indeed 7 dBm. More on this later...



Above — My prototype RF-tight breadboard of the Barebones PM. I ran my 'trademark' die case box, BNC RF input and Port B connectors, a common ground lug and feedthrough capacitors for the B+ and panel meter connections. I placed a BNC to RCA adaptor on Port B to allow the insertion of a standard-type positive DVM probe. A black alligator clip terminates the distal end of the negative probe on all my DVMs. I just clip it onto the ground lug.

Above — A view of my breadboard showing my 100 uA panel meter and the 4.5v battery pack. I raised R2 in the original schematic up to 8K2 to set my preferred needle movement in this 100 uA panel meter. A meter with linear markings might be better?

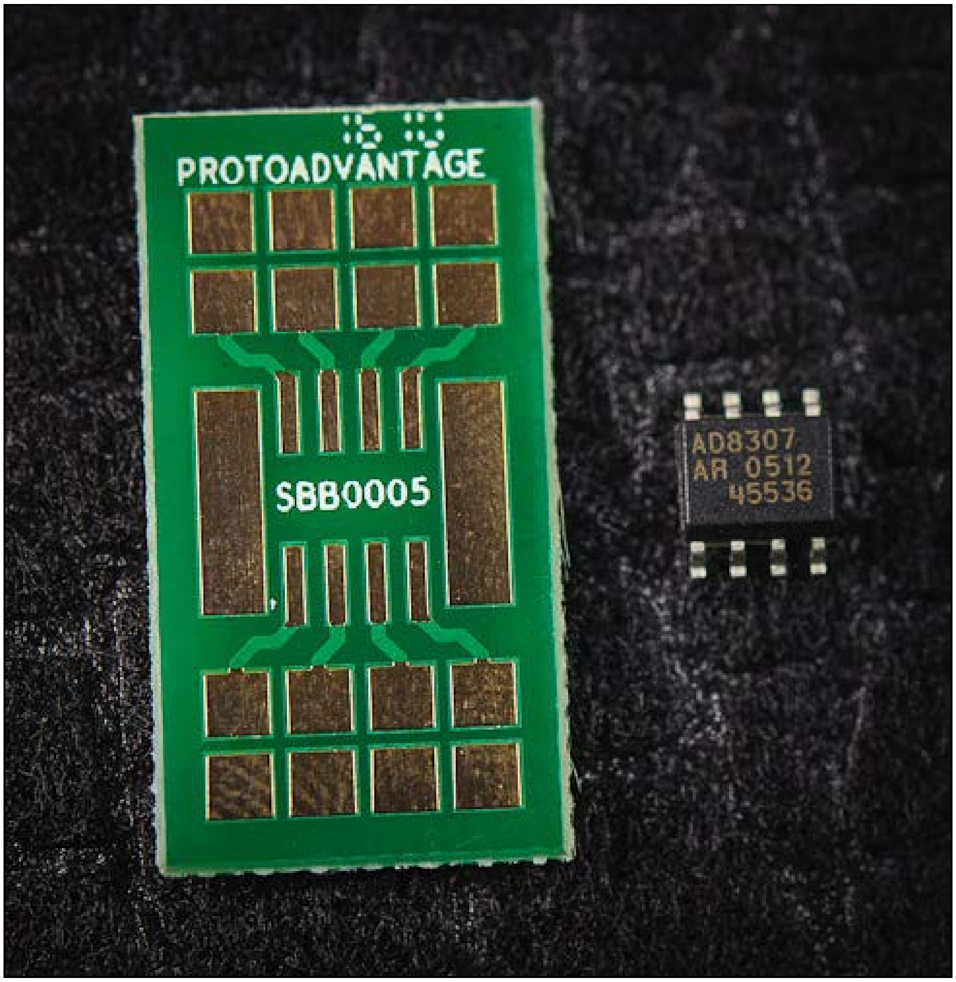
Some Further AD8307 Notes

After power up, you'll notice a DC output voltage around 0.22v or so with no input signal; this arises from wideband noise caused by resistors and amplifiers in the AD8307— all normal.

If possible, verify a 25 mV per dB power change with a manual sweep using a sine wave signal generator on the input and a DVM to AD8307 Pin 4. If you don't have the test gear to perform this function check, no problem.

The AD8307 input resistance = 1100 Ω , so we must place a resistor in parallel with the input to establish 50 Ω. Many builders just shunt a short-leaded 51 Ω resistor from input to ground like Wes did, however, you might also see builders place a 1% tolerance 53.2 Ω R in that slot to derive an input Z of 49.9 Ω. I did this in another "blinged-out", frequency compensated AD8307 meter I built for future UHF circuit experiments.

With 20 and/or 40 dB taps, attenuator pads and some 50 Ω cables, a Barebones AD8307 PM can measure everything HF you might build or buy for your *QRP* workbench. I posit that a simple AD8307 power meter may form the heart of a basic, first QRP workbench. Lacking a oscilloscope when I started in radio electronics, I measured RF with a germanium diode RF probe and a DVM — I would have enjoyed a simple log power meter plus a basic calibrator, however, the AD8307 did not exist back then.



Above — Some builders lament because an AD8307 (in DIP) costs around $10.00. I bid for and bought the AR version chip shown above for $4.42 USD. The above green Proto Advantage breakout board cost $1.00. To compare; some people spend $5.00 for a boutique coffee in Canada — it's all good.

The SMD packaged AD8307AR may offer better performance above 100 MHz with its lower lead inductance.

QRP — PosData for Dec 3, 2013

Realize your bench acumen — adapt and build some test equipment to suit *your* needs. Steve, VK2SJA crafted a version of the AD8307 PM. Click for his beautiful box featuring a 1mA meter movement. Aided by the Tonne software "Meter" program and the AD8307 datasheet, his ranks among the best I've seen. Kudos Steve.

---

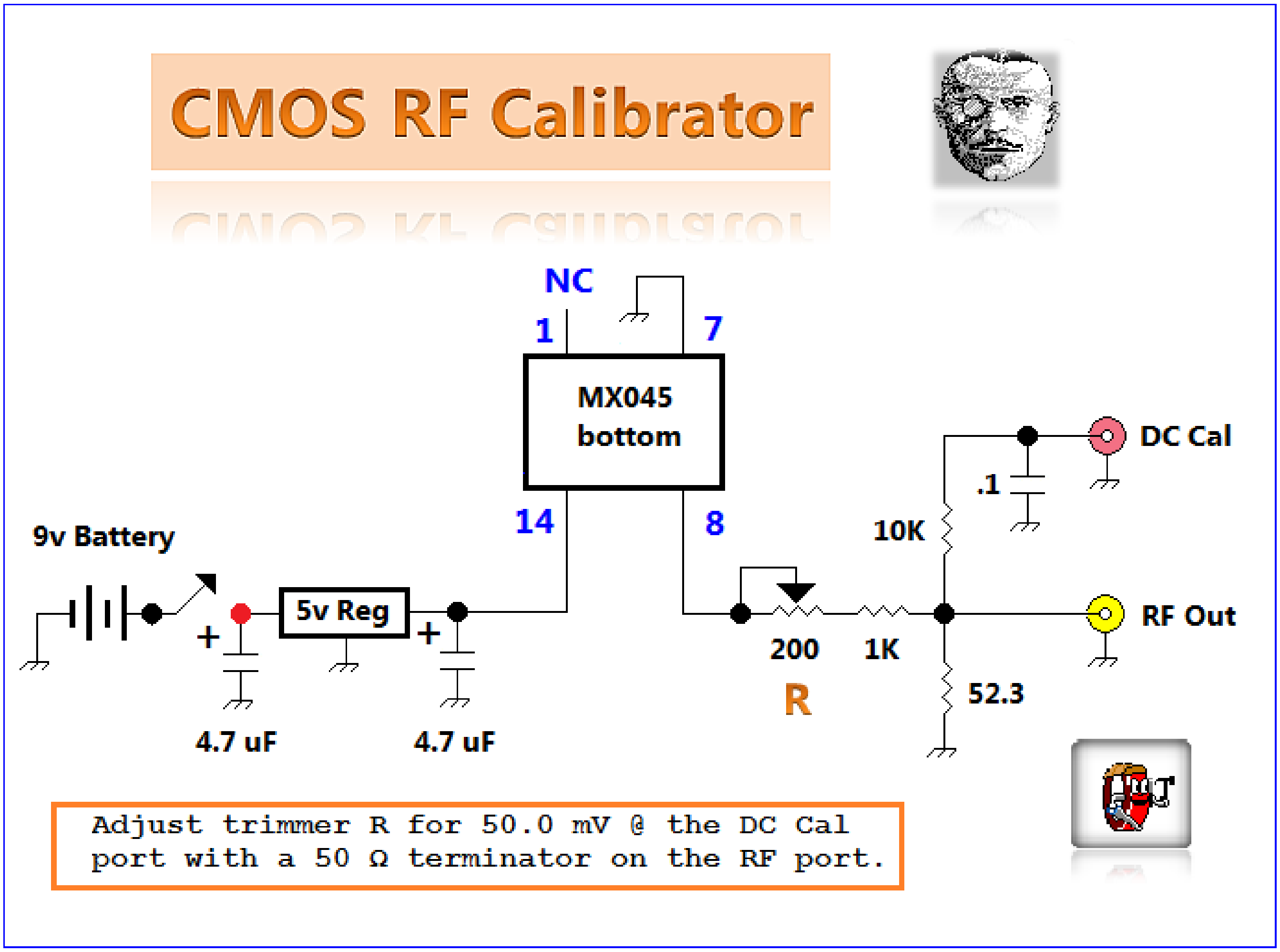
2. Power Meter Calibrators

1. CMOS Clock Oscillator RF Calibrator @ -10 dBm

Bob, K3NHI designed a 10.0 MHz reference oscillator for -10 dBm that might help equip the beginner bench. This CMOS signal source does not need an AC power reference to calibrate it — just a DVM to make a DC measurement and your good to go!

In addition to calibrating an AD8307-based PM, you might use this -10 dBm reference to calibrate other gear including homebrew sine wave signal generators; that's what I'll do later.

Above — The recommended CMOS clock oscillator for the CMOS RF calibrator. Digi-Key part # CTX772-ND. Data sheet.
If you substitute another CMOS clock, it must swing nearly rail-to-rail for accuracy. Bob intended this calibrator for those who lack the bench instruments needed to precisely calibrate RF devices since only a DVM is needed for calibration. This generator also offers a range of calibrated harmonics as Bob described in his January-February QEX article A Simple RF Power Calibrator — great when you want to examine a spectrum analyzer over a limited span.



Adjust trimmer R for 50.0 mV @ the DC Cal port with a 50 Ω terminator on the RF port.

Above — My version of the K3NHI CMOS signal source (presented with the permission of Bob, K3NHI). The 52.3 Ω resistor is a standard 1% part and I bought 5 for all my AD8307 projects and this little signal generator. In my first version, the trimmer resistor was 500 Ω and worked okay, but the 200 Ω trimmer improved calibration. I ordered 10 Bourns 200 Ω and 10 Bourns 500 Ω trimmers on eBay for a few dollars and after studying Bob's published work and applying his influences to my own, I now love to precisely calibrate or bias circuits with a 200 or 500 Ω trimmer R as appropriate.

QRP — PosData for November 22, 2012

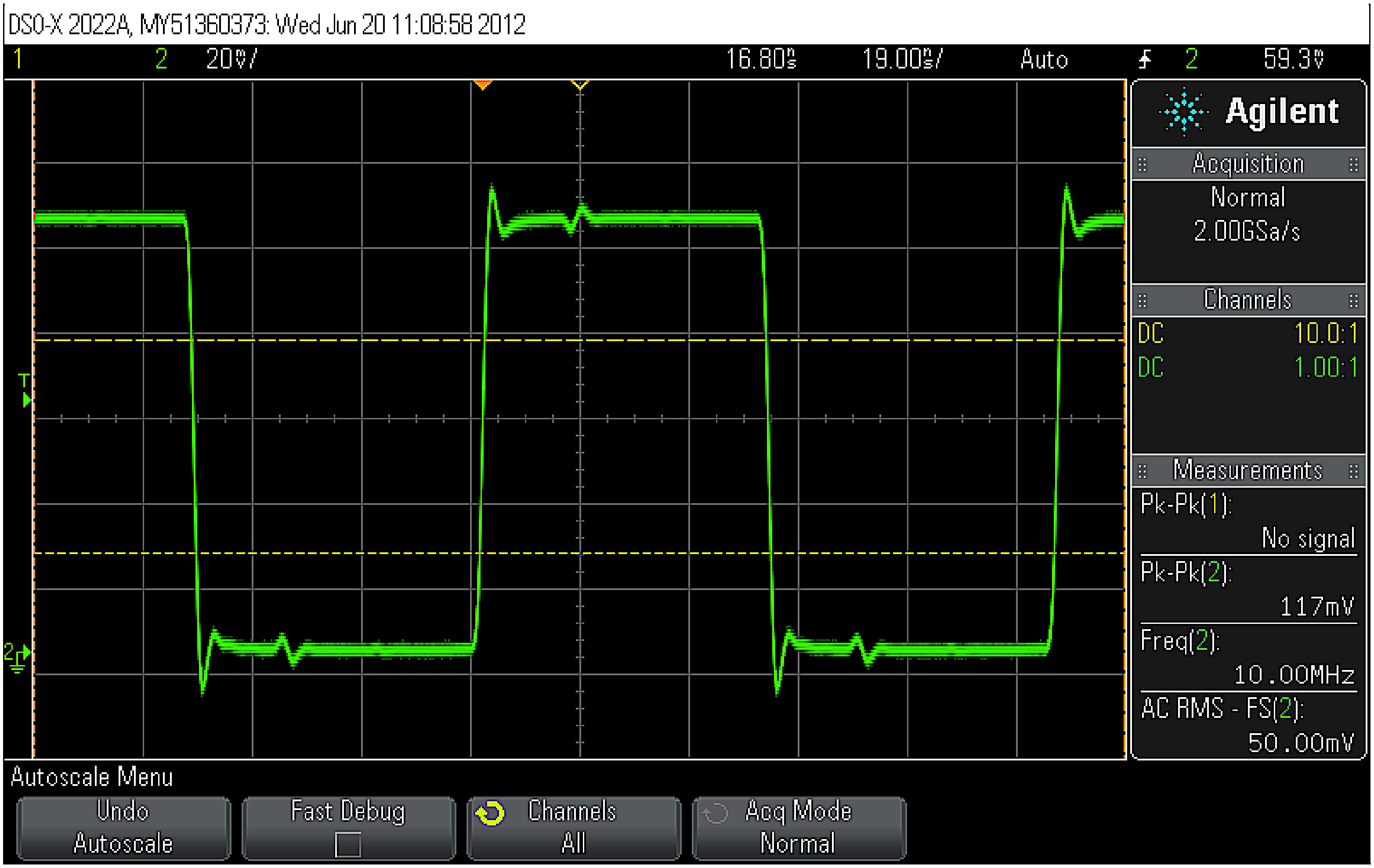
If required, you may substitute a 49.9 to 51 Ω resistor for the 52.3 Ω specified.

Per Bob's QEX 2010 Tech Notes and his emails specifically about calibrating the AD8307 with a CMOS *square wave***:** Modern AD8307 chips are better calibrated with a -10 dBM square wave. To test log linearity after CMOS signal generator calibration, apply a sine wave signal generator to your power meter and adjust its output to get the same power meter DC output voltage as with the CMOS generator. Then insert attenuator pads on the now calibrated sine wave generator to assess the mV/dB change with different power levels.

Bob's original CMOS calibrator outputted - 20 dBm, however, he updated it to output -10 dBm in 2010 as reported in his QEX 2010 Tech Notes.

Some readers have asked why calibrate the AD8307 at 10 MHz? Calibrate at whatever frequency you want, or more than one. However, at 10 MHz, the AD8307 exhibits its best log performance compared to other frequencies. Click or click for datasheet graphs.

Above — The DVM calibration port reading (2 volt scale) from my CMOS clock calibrator after calibration. That was easy!



DSO-X 2022A, MY51360373: Wed Jun 20 11:08:58 2012

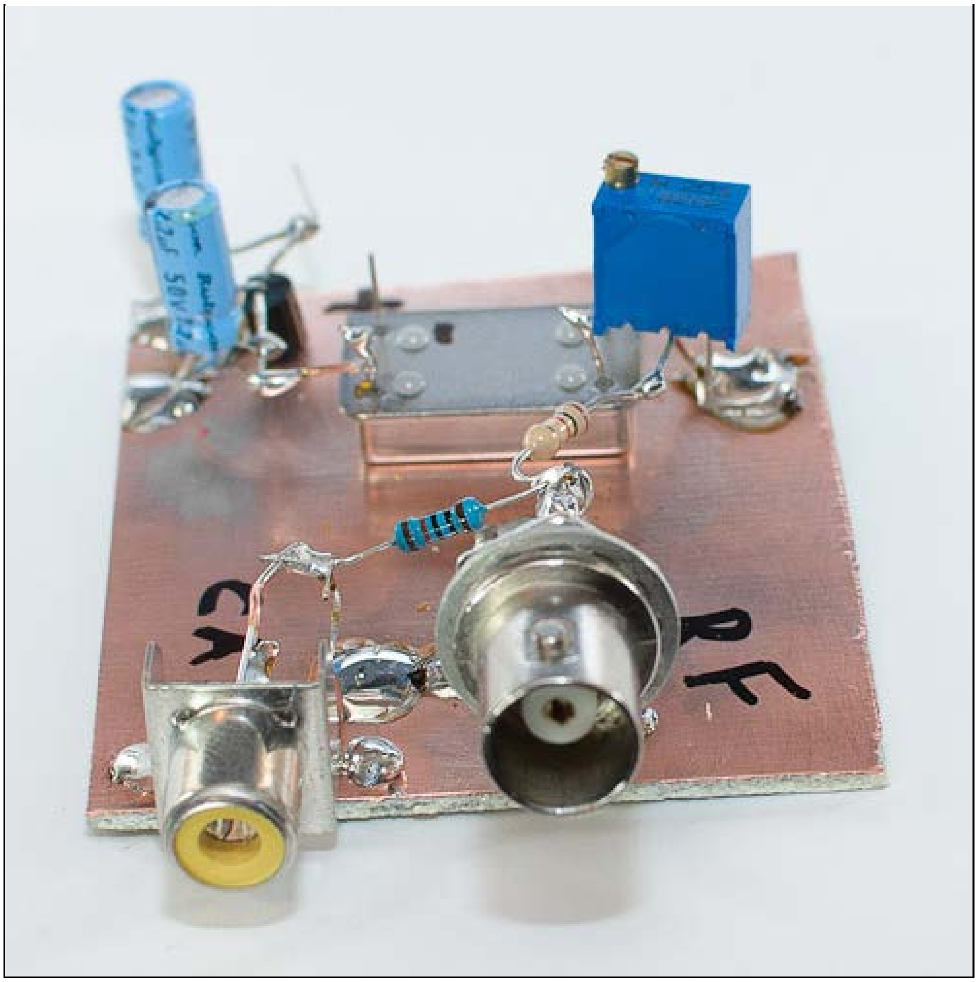
1 2 20mV/ 16.80ns 19.00ns/ Auto 2 59.3mV

Acquisition
Normal
2.00GSa/s

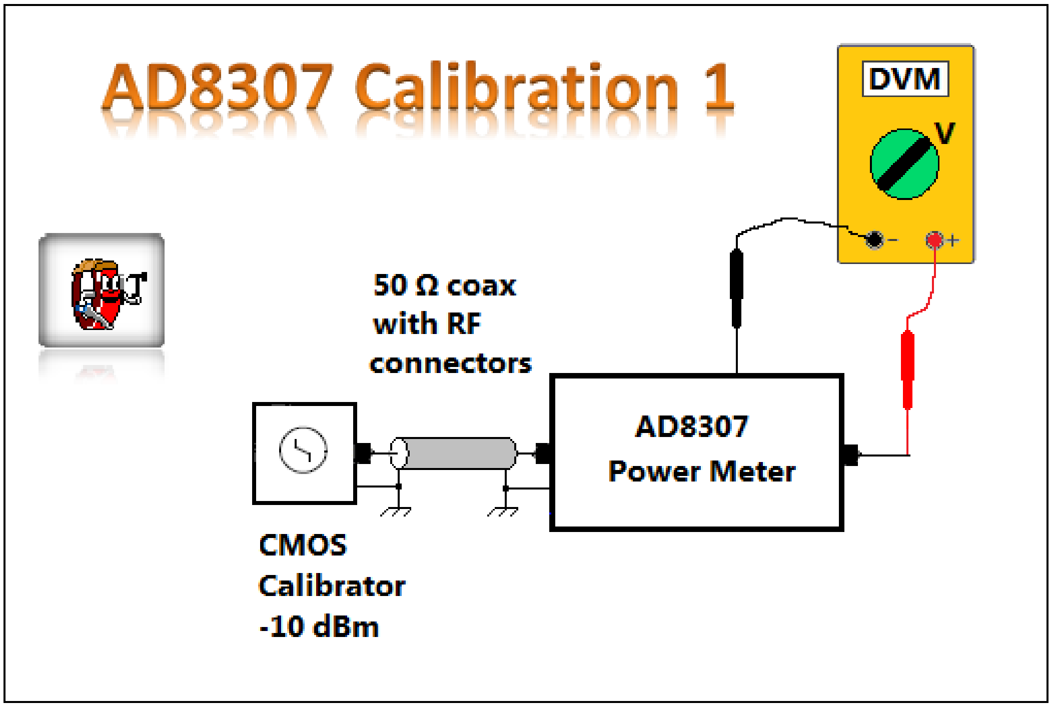
Channels
DC 10.0:1
DC 1.00:1

Measurements
Pk-Pk(1): No signal
Pk-Pk(2): 117mV
Freq(2): 10.00MHz
AC RMS - FS(2): 50.00mV

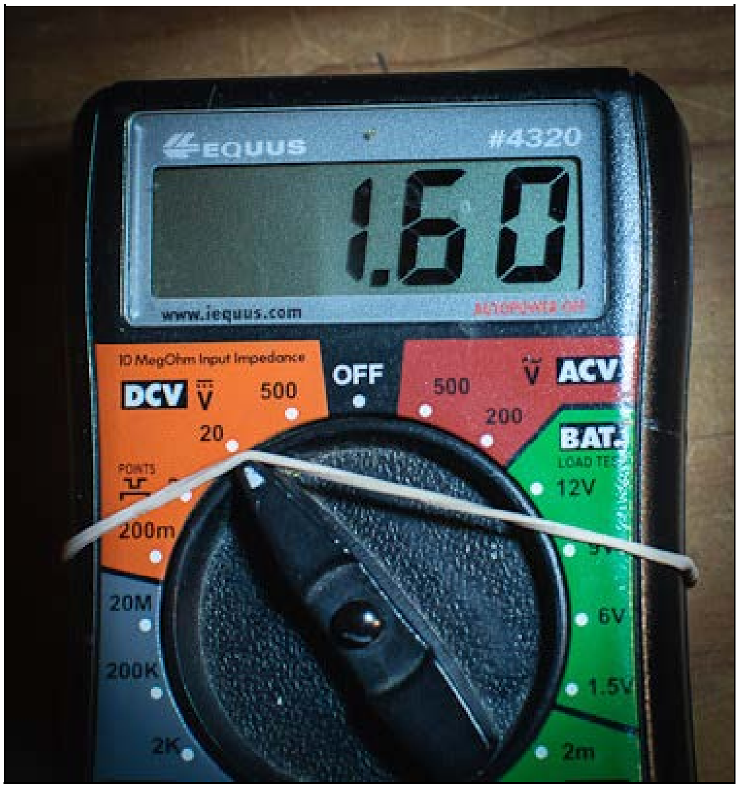
Above — The calibrated CMOS clock RF calibrator in my 50 Ω terminated scope.



Above — The breadboard of my version of the K3HNI CMOS -10 dBm RF calibrator. For best results, stick it in a shielded box. When correctly calibrated, it outputs -10 dBm on an AD8307 PM, -14 dBm on a spectrum analyzer and -13 dBm on a conventional, thermal-sensor power meter.



Above — The procedure for calibrating an AD8307 PM with the -10 dBm square wave CMOS signal source.



Above — When I connected the calibrated CMOS clock RF calibrator to my build of the Barebones PM, I measured 1.60v with my DVM; ***the -10 dBm reference voltage.*** Since the log-linear power changes 20 mV/dB; for my calibration reference voltage: 1.40v = -20 dBm, 1.20v = -30 dBm, 2.0v = 10 dBm etc.

Remember that the Barebones PM runs on a battery pack and over time the B+ will change. *Each tim*e I measure power with the meter I first calibrate it to establish the -10 dBm reference voltage.

Here's a simple formula that only works for 20 mV/dB @ my particular 1.60v calibration voltage, but gives you the general idea:

Power in dBm = 50 x (V - 1.8)

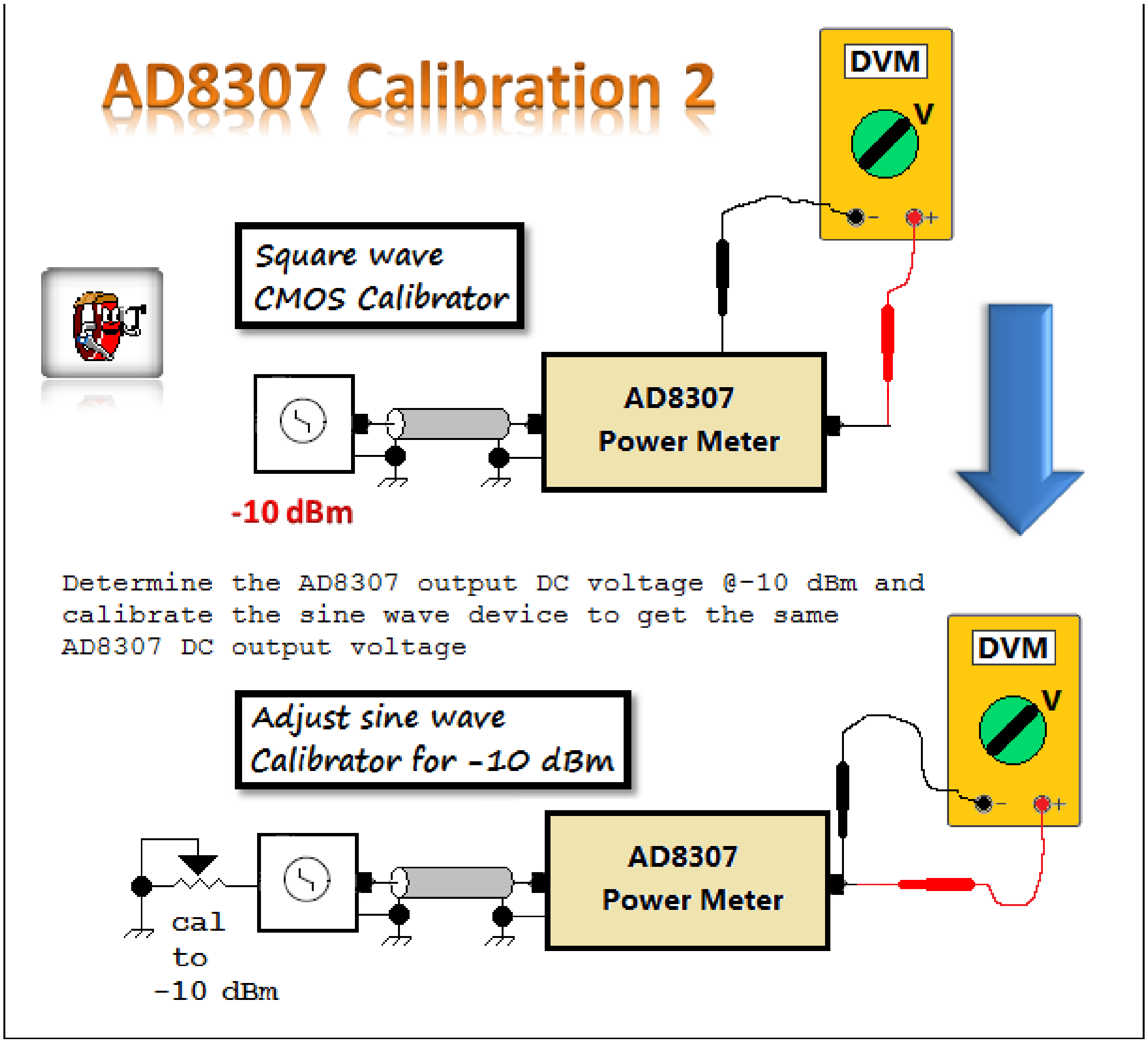
So if I measure 1.94v: 50 x (1.94 - 1.8) = 7 dBm.

2. Sine Wave Oscillator for Calibration @ -10 dBm and -20 dBm.

You may also calibrate your AD8307 PM with a calibrated sine wave signal generator. Advanced builders who own the gear needed to measure RF power tend to use a sine wave for calibration.

It's easy to calibrate the AD8307 with a sine wave signal source. Normally we calibrate our sine wave signal generators with instruments such as a 50 Ω terminated 'scope, a spectrum analyzer, a calibrated power meter, or a 49.9 to 51 Ω terminating resistor plus a 10X probe etc., but if you lack these instruments, your stuck.

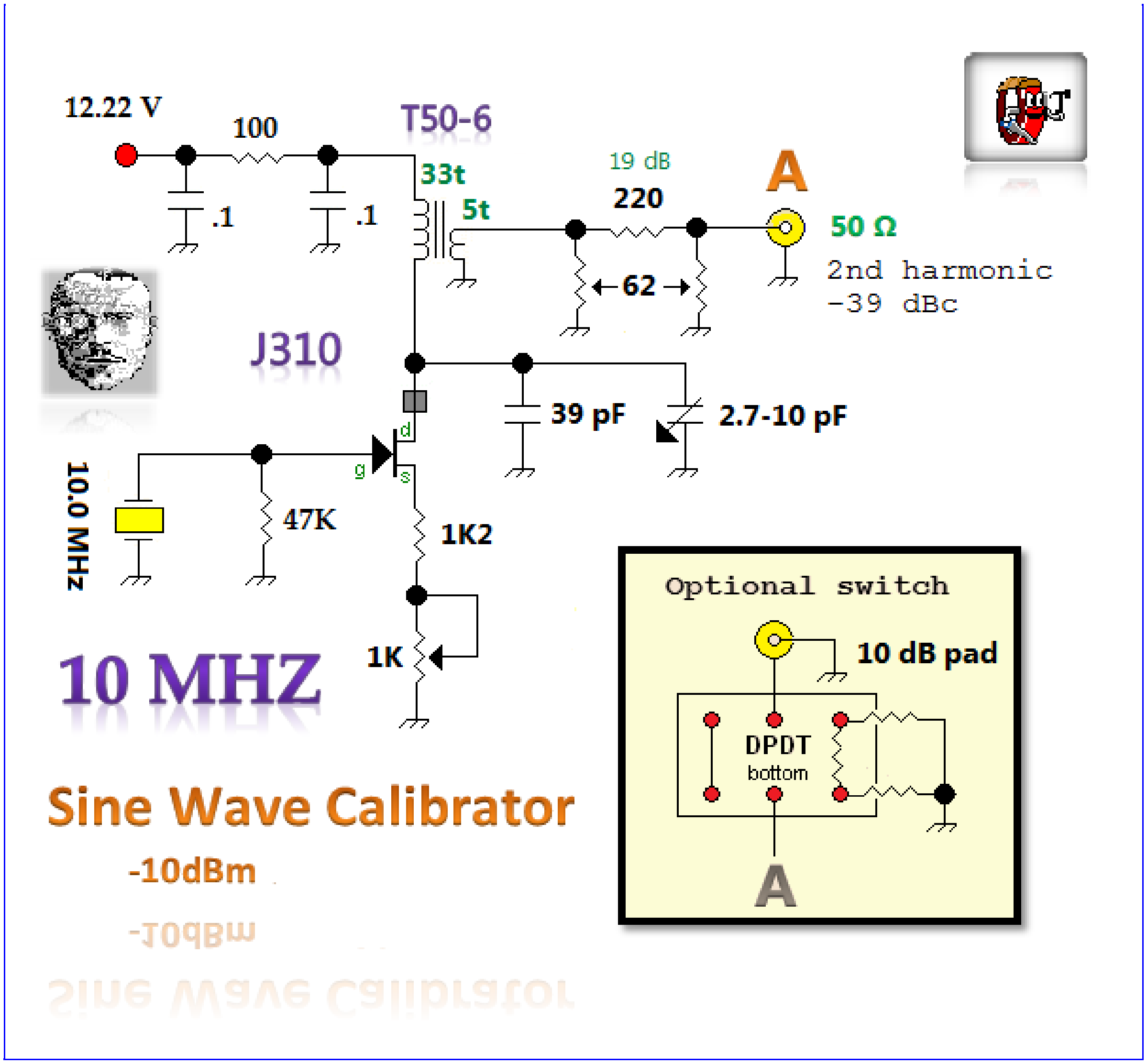
No problem. You may calibrate any appropriate sine wave oscillator at -10 dBm with your Barebones PM and the CMOS RF Calibrator shown earlier. Let's examine the procedure:



AD8307 Calibration 2

Determine the AD8307 output DC voltage @-10 dBm and calibrate the sine wave device to get the same AD8307 DC output voltage

Calibrate the AD8307 PM with the CMOS square wave reference and record the DC voltage at the output of Port B. Connect up your sine wave generator and adjust its output until you get the same Port B DC voltage as the reference CMOS RF signal generator — your sine wave oscillator should now be calibrated to -10 dBm.

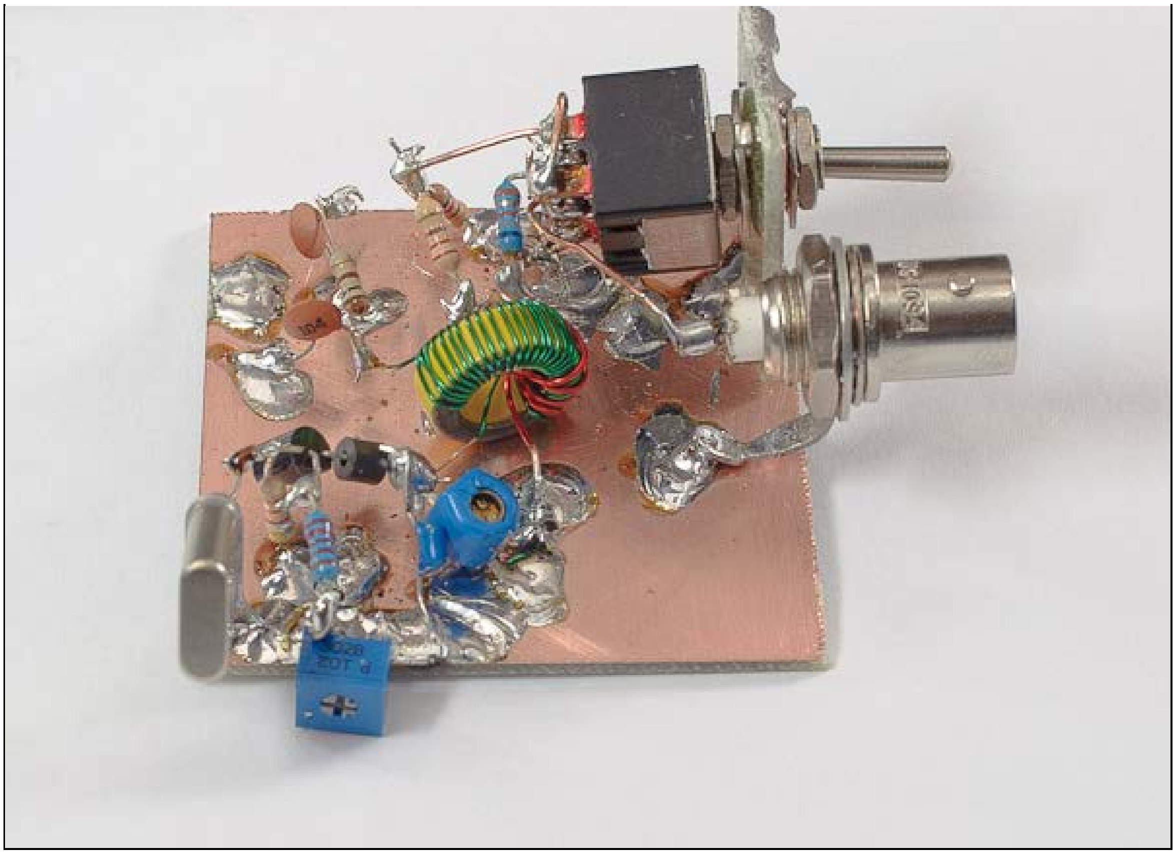


Above — I designed this simple 10.0 MHz sine wave calibrator to evaluate Bob's CMOS square wave RF calibrator and serve as an example sine wave reference oscillator. When running a regulated VCC of at least 12v and the *tank is perfectly* tuned, low distortion arises. I measured the second harmonic @ 39 dBc down. The L with 33 turns = 4.43 uH.

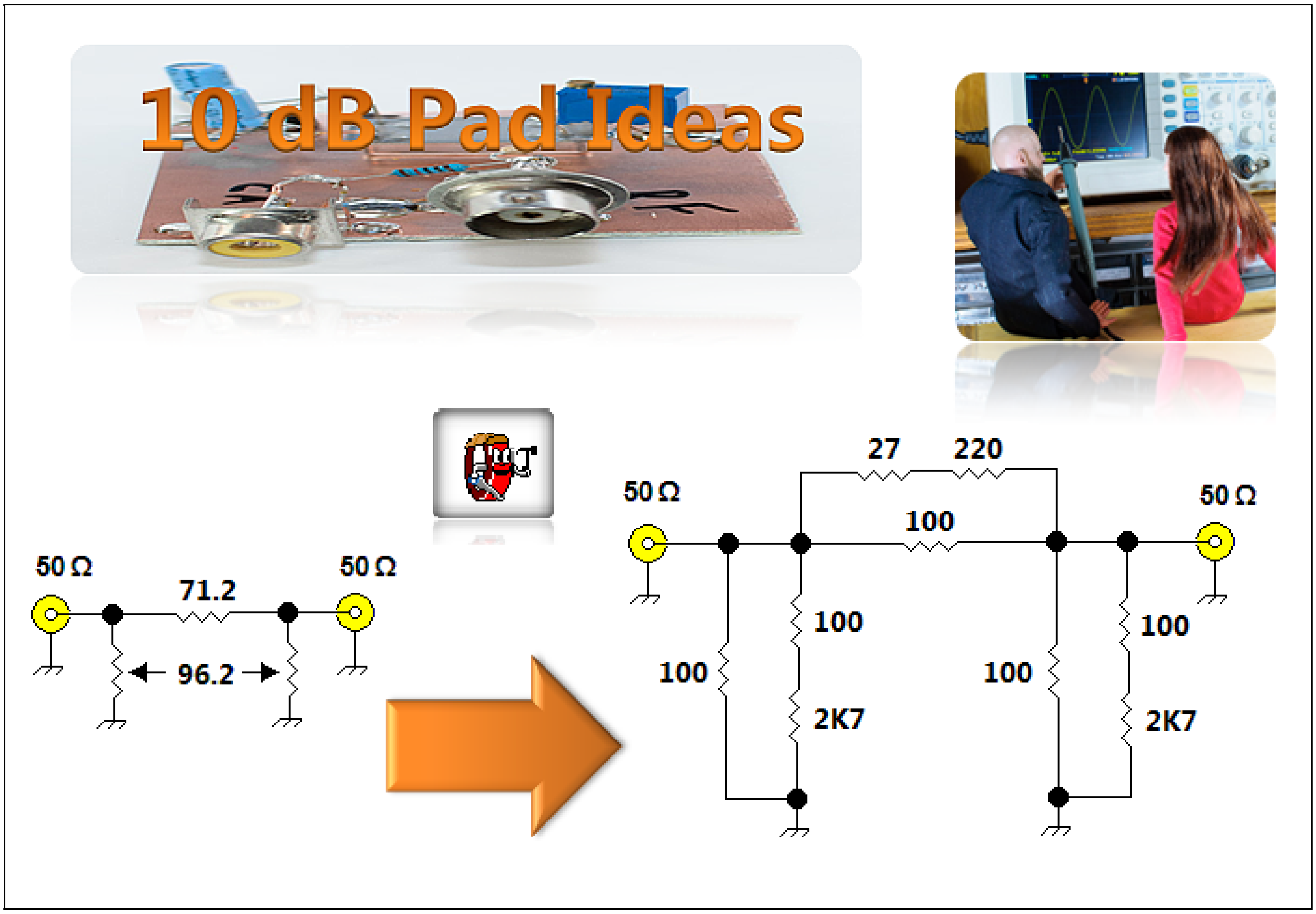
The initial tuning procedure goes like this:

1. Terminate the output with a 50 Ω resistor terminator, or a 50 Ω terminated 'scope, or your AD8307 PM.
2. Connect an ammeter between the VCC node and the regulated power supply.
3. Adjust the emitter trimmer R so that the circuit draws around 2.7 mA — then disconnect the ammeter leads.
4. Adjust the trimmer cap for the highest pk-pk voltage (and/or or best looking waveform) in the 'scope, or highest power in the AD8307 PM. *Nominal total C to resonate my particular circuit was ~ 45 pF.*

If not already done, connect the tuned-up sine wave signal source to a AD8307 PM. Adjust the trimmer potentiometer so the Port B DC voltage = the reference voltage measured during your CMOS RF generator power meter calibration. Your sine wave signal source is now calibrated to -10 dBm.

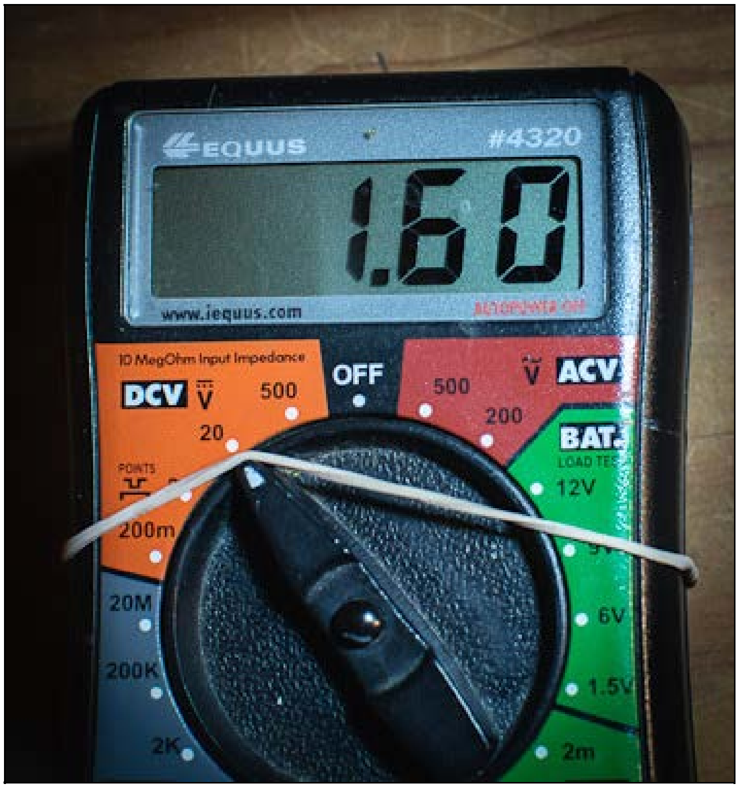


Above — The breadboard of my 10 MHz sine wave signal source. I included the optional switched 10 dB attenuator shown on the schematic inset. In the end I decided to just stick a removable **10 dB** pad like this 6 dB pad in-line for my -20 dBm measurement.

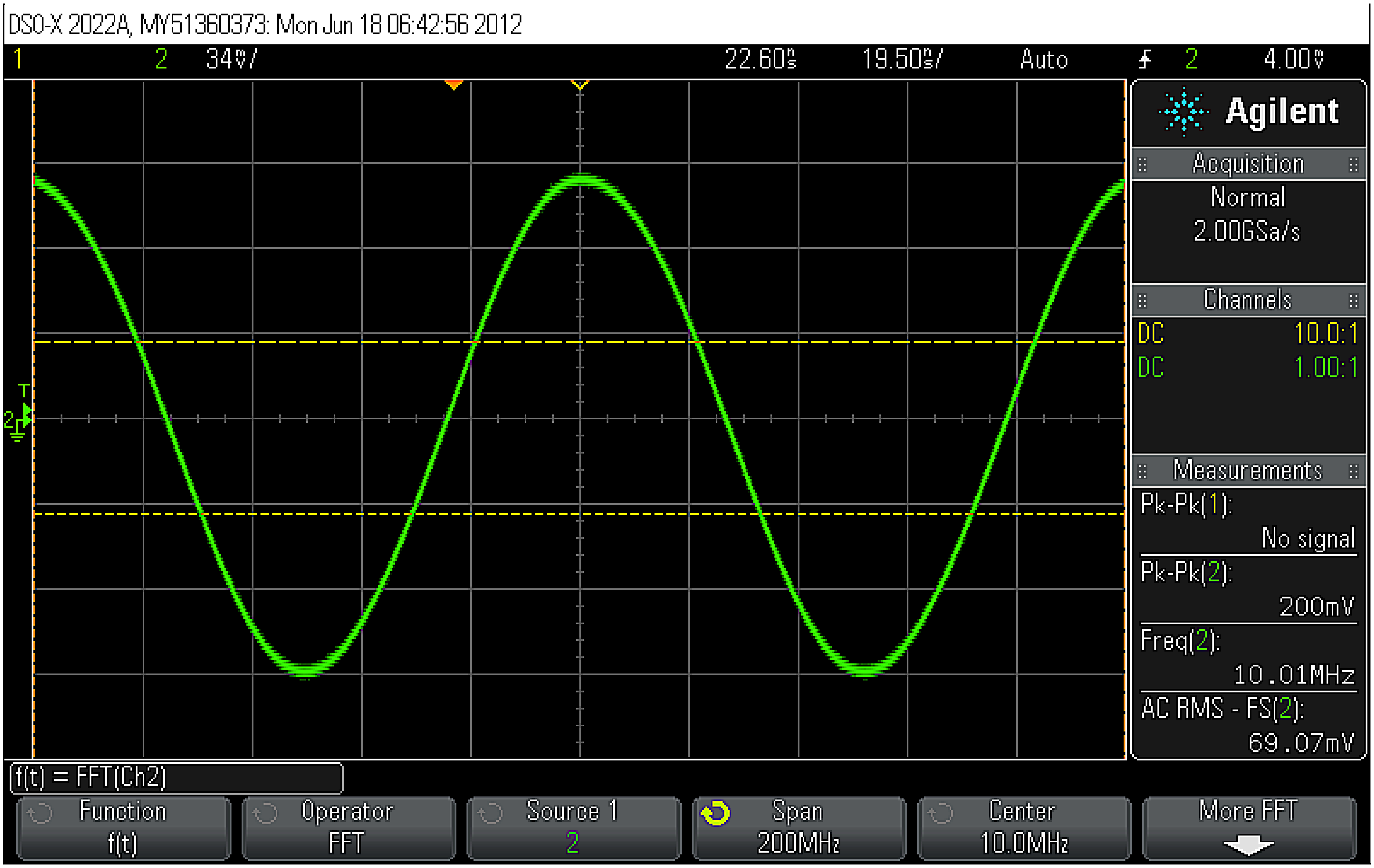


Above — Craft accurate attenuator pads with parallel + series resistors and some creative energy. Lacking the proper 1% parts, I hand selected some 5% resistors among the values shown to the right and built a pad that *precisely* gave a 10 dB power drop at 50 Ω.

Now let's check the calibration of this little sinusoidal RF generator...



Above — When connected to the Barebones PM, I calibrated the output power of the sine wave generator by tweaking the trimmer pot to give an output of 1.60v; the same Port B voltage yielded by the CMOS RF calibrator.



DSO-X 2022A, MY51360373: Mon Jun 18 06:42:56 2012

1 2 34mV/ 22.60ns 19.50ns/ Auto 2 4.00mV

Acquisition
Normal
2.00GSa/s

Channels
DC 10.0:1
DC 1.00:1

Measurements
Pk-Pk(1): No signal
Pk-Pk(2): 200mV
Freq(2): 10.01MHz
AC RMS - FS(2): 69.07mV

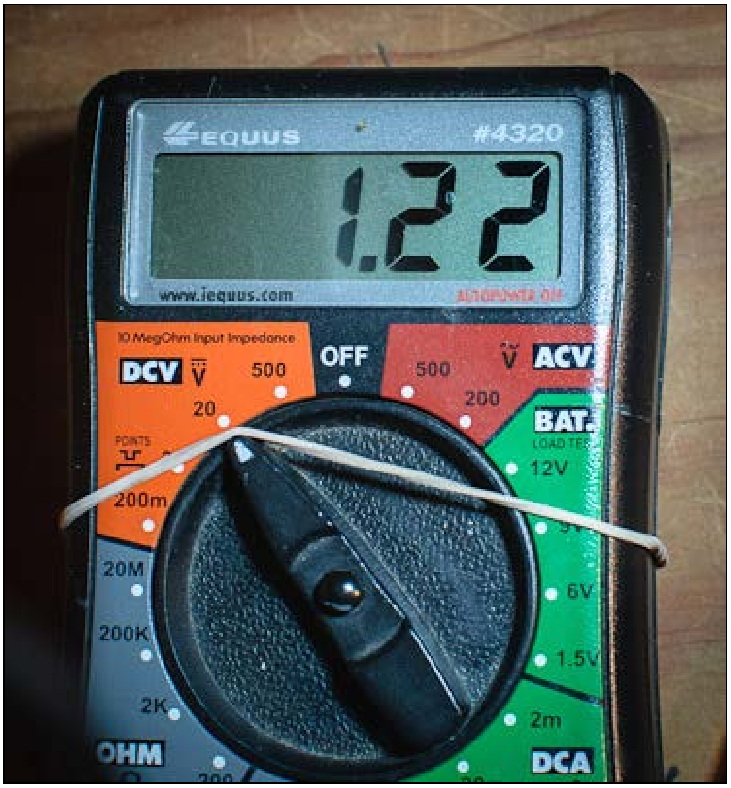
f(t) = FFT(Ch2)

Above — The 50 Ω terminated 'scope verification of my calibrated 10 MHz sine wave signal source: 200 mV pk-pk = -10 dBm. Wow, thanks for this Bob!

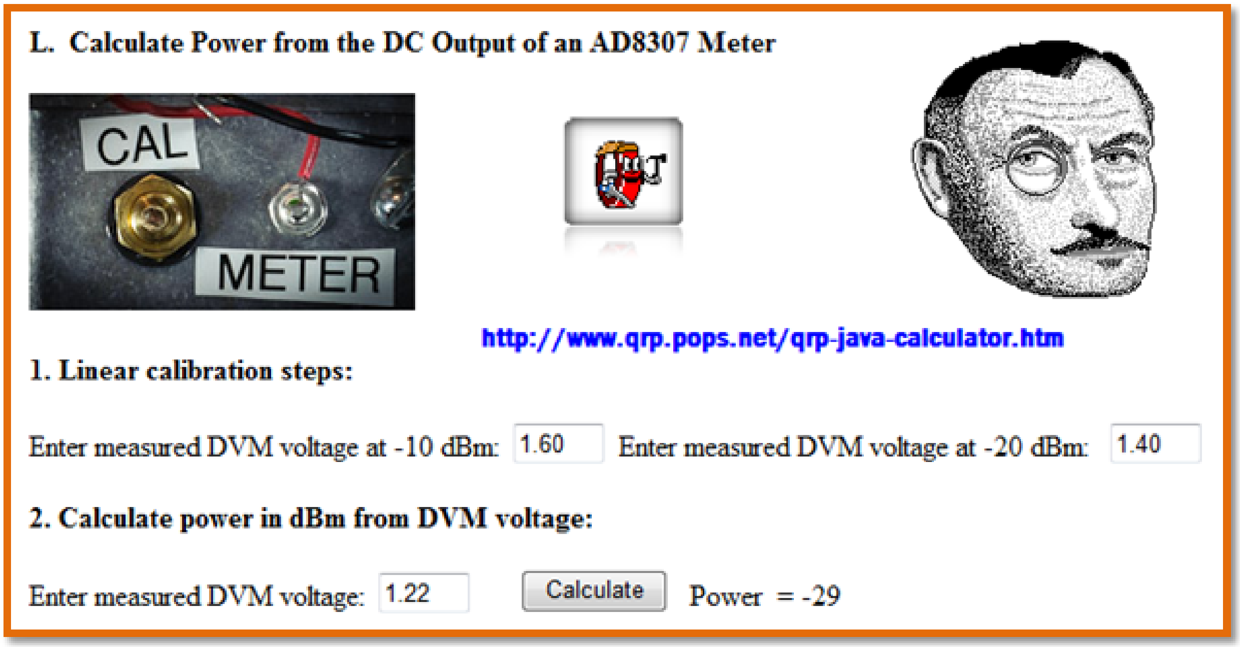
Above — The Port B voltage when a 10 dB attenuator pad was connected to the calibrated -10 dBm sine wave signal source: Measured power = -20 dBm. Notice the 20 mV/dB power drop — right on specification.

Refer to the section titled **<How to Measure Power at Port B>** Recall Wes' calculation needed 2 points to set the log linearity. I chose -10 dBm and -20 dBm instead of 0 dBm and -10 dBm so the CMOS RF calibrator could be used to calibrate any sine wave signal generators on hand.

The -10 dBm calibration reference power serves as 1 of the calibration points in Applet L while we derive the other by adding a 10 dB attenuator pad to a *sine wave* signal source output port. ***Don't connect an attenuator pad to the CMOS square wave calibrator —*** error in the AD8307 arises. Why? Analog Devices mysteriously changed the AD8307 at some point after 2004 and altered its crest factor — this disallows us performing log linearity calibration with a square wave.



Above — A 20 dB pad connected to the -10 dBm sine wave signal source yielded 1.22v at Port B in the Barebones PM. See the power calculation with Applet L below:



L. Calculate Power from the DC Output of an AD8307 Meter

http://www.qrp.pops.net/qrp-java-calculator.htm

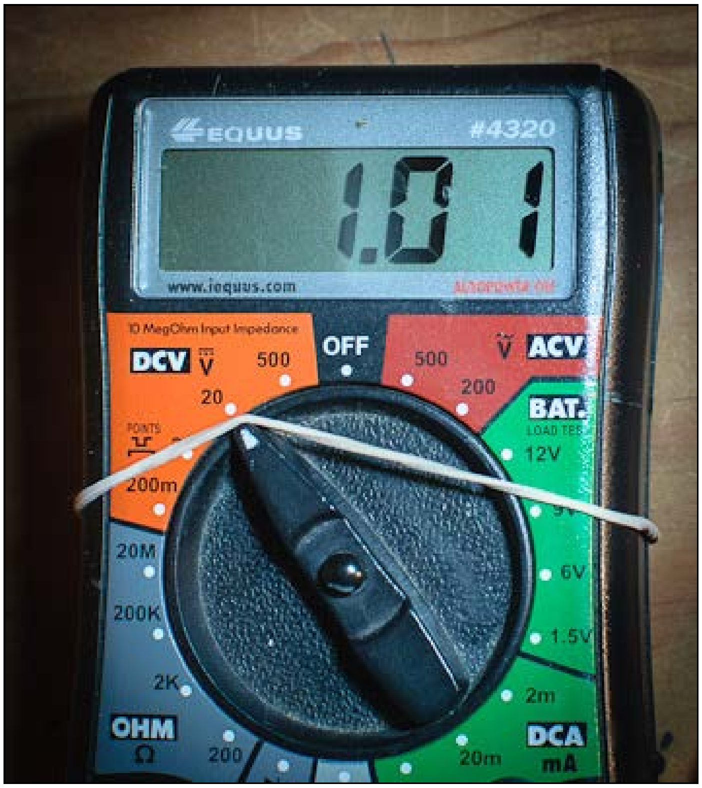
1. Linear calibration steps:

Enter measured DVM voltage at -10 dBm: 1.60 Enter measured DVM voltage at -20 dBm: 1.40

2. Calculate power in dBm from DVM voltage:

Enter measured DVM voltage: 1.22 Calculate Power = -29

Above — The calculation of the output power with a 20 dB attenuator pad on the sine wave signal source output. We're off by 1 dB since it should have calculated a power of -30 dBm. Could this be error caused by my attenuator circuit, or non-linearity by the AD8307 PM, or a bit of both? We have to live with such problems. Still, the Barebones power meter seems quite impressive for a simple circuit that I scratch built and calibrated in about 25 minutes.



Above — The Barebones PM DC output voltage with a 30 dB attenuator pad on the output of my -10 dBm, 10 MHz sine wave signal generator: Measured power = -39.5 dBm. Again; a very good — but not perfect power meter.

A Barebones power meter allows builders with modest equipment to measure power, gain/loss, and with a return loss bridge, return loss at HF. I wish I owned this little gem back when I started out.

Richard, "Dick", N4HAY, posted some great notes on his blog. I recommend following his blog since he really digs deep and likes math — thanks Dick!

Best!

---

3. A Basic RF Workbench

Since January 2012, a handful of readers asked what I consider a good basic *RF* work bench. Again, I'm just an amateur hobbyist, so my opinion might show my ignorance.

A stand-alone 50 MHz oscilloscope with at least one 10X probe. *More bandwidth if you plan to work above 50 MHz*

If no oscillocope, an AD8307-based power meter *The modern version of the diode RF probe*.

3-4 50 Ω coaxial cables with BNC connectors; a 50 Ω scope feed-through terminator, *1-2* 50 Ω BNC port terminators, a though-connector and some BNC connectors to solder onto temporary circuit boards or mount in a chassis.

A homebrew return loss bridge.

3, 6, 10 and 20 dB BNC connector equipped attenuator pads, or a step attenuator.

Signal generator(s) that cover most of HF; +/- VHF signal generators described in the next section.

AADE L/C Meter IIB. Click

12 volt regulated power supply good for at least 1 amp.

Digital multimeter. *I use 2 and keep 1 set up for current measurement only.*

Frequency counter: homebrew or commercial. *I ran a 40 year old, ovenized, accurate HP counter until 2012.*

With these devices, as possible, you can work in a modular, 50 Ω environment and measure gain or loss in dB, return loss in dB and absolute power in dBm. Starting small and expanding your bench around 50 Ω input and output impedance devices will provide a lifetime of challenge and excitement in RF design.

Later, the big toys can follow: spectrum analyzers, VNAs, commercial signal generators and other lab quality stuff.

Equipping an RF bench presents quite a financial burden. I started small and slowly added pieces over time. Many pieces such as my L/C Meter IIB were gifts for holidays or my birthday. Other pieces were old, inexpensive equipment that I restored and calibrated.

Above — RF tools of the trade. We're RF experimenters! As scratch homebrew builders, gear like BNC, SMA and through-connectors, 50 Ω terminators and inline attenuators lie scattered on our benches; our fodder. Alternate photo.

4. VHF Signal Generators

Having only started at VHF in November 2011, my knowledge suffers, however, a search for accessible, affordable, good quality VHF signal generators disheartened me. Ten year old or newer signal generators covering the VHF band work up to several GHz and cost a small fortune.

Lamenting old timers often recommend the vaccum tube HP-608 series that covered ~10-480 MHz. These heavy, glowing beasts sometimes come up in estate sales or on eBay for $200-400.

Then, too, the HP8640 series seems attractive, however, they are full of decaying parts. Ken Kuhn and others restore old HP gear as a hobby and this direction certainly gives us a valid option.

I've investigated 1 or 2 new, low-cost, commercial signal generators that work into VHF, but they failed to excite me; especially after I downloaded the schematics and sat in disbelief over their poor design.

Some minimum commercial signal requirements might include stable, linear tuning, a metal chassis, 50 Ω output with a return loss greater than 20 dB and low harmonic distortion at all frequencies. Like the rest of our lives, our budget usually determines what we buy.

I decided to build my own VHF signal generators and document them on this web site. I've learned that home building signal generators between 50 and 200 MHz requires skill and care, but can be done.

What about digital clocks?

At VHF, DDS spurs get extreme as you get closer to the maximum clock frequency .

The Si570 looks intriguing, however, still requires an MCU + components, I haven't read any lab quality evaluations of the Si570 as part of a engineer-grade VHF-UHF signal generator and if you know better, please email me.

5. L - C Meters

If you search for opinions about which L/C meter to get, you'll find an abundance of super write-ups including those that cover measuring with Kelvin probes, SMD tweezers; or statements suggesting that if you really need maximum accuracy, purchase a VNA. I encourage you to research this yourself and find the best L/C meter for *your* bench. Here are my 2 cents worth of opinion and please remember — I'm often wrong.

I use an AADE L/C IIB meter to test inductors and capacitors for HF and even some VHF work. Yes, the AADE L/C IIB doesn't measure large-value electrolytic caps and so forth, however, considering the cost versus performance — it's accurate enough for the popcorn RF workbench.

The AADE L/C meter uses the method described by William Carver, W7AAZ in an article called The LC Tester published in Communications Quarterly, Winter 2003 . EMRFD page 7.12 briefly examines Bill's circuit and shows his original oscillator along with an extended range Colpitts oscillator designed by Wes. For brevity sake, I'll just discuss inductance measurement with Carver-style meters. With care, an inductance resolution of 5 - 20 nH might be realized with such a device.

We normally don't consider that our inductor is actually a network with L, a parallel C and losses that might be modelled as R in series with the L, or a R in parallel with the C depending on our model. The inductor also exhibits a self-resonant frequency and for our design purposes, we usually *ignore* all these details and just consider it a "pure L". My L/C meter's oscillator runs from a few tens to a few hundred KHz and generally lies below the self-resonant frequency of the inductors I measure with it. I've learned by sweeping/analyzing my completed filters, that as long as you avoid the coil's SRF, *the low frequency* Carver-style meter proves a stalwart inductance meter for most HF and some VHF applications.

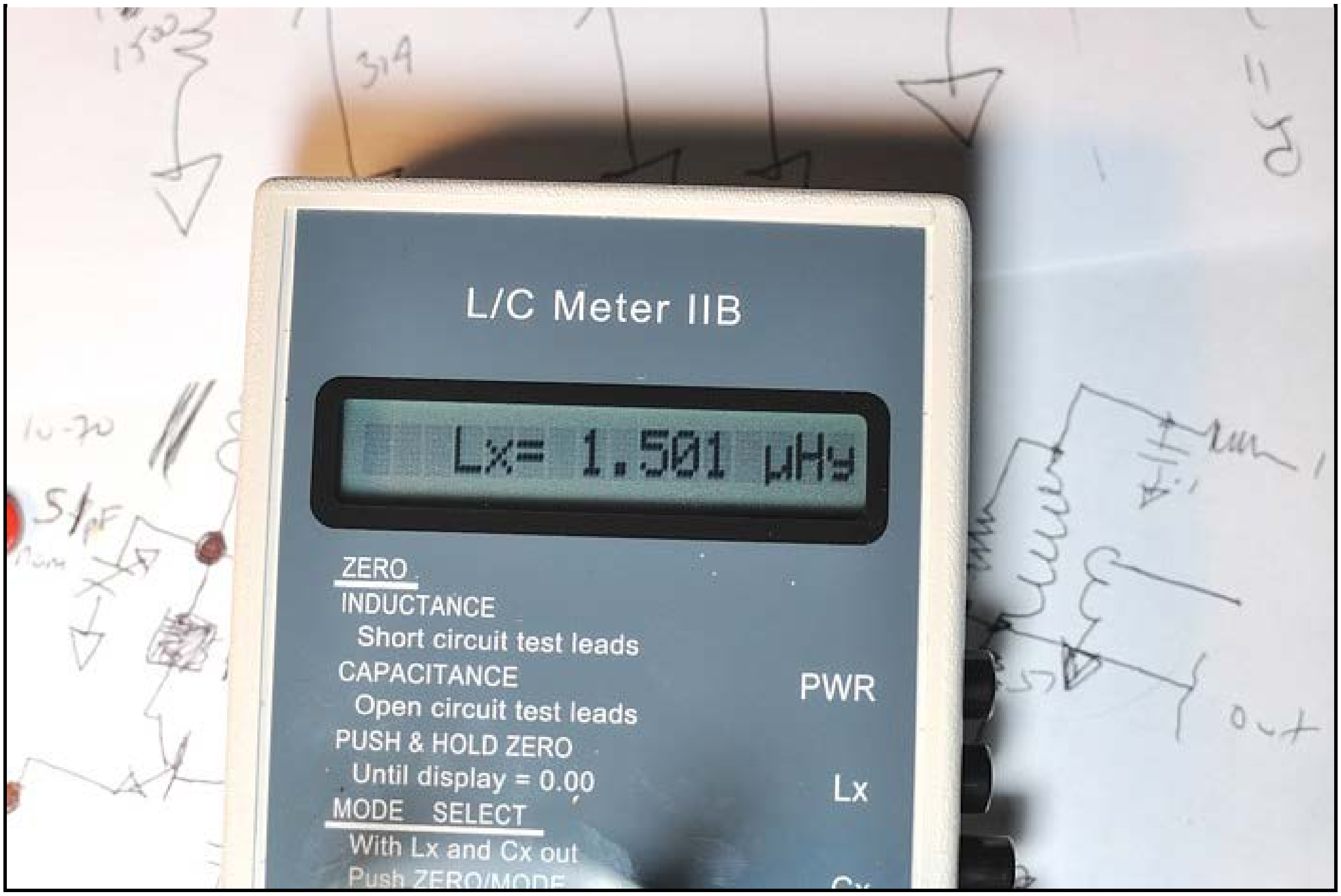
Often, we popcorn builders want to make a filter, an oscillator, or a pi, or L match and we apply software or tables to calculate the L and C values needed to resonate our filter tank(s) or matching networks, or to synthesize a low-pass filter. These math-driven programs/tables assume the *pure* "pixie dust" L described eariler — disregarding the stray C and R. So getting all worked up about whether our coil is 4.50 or 4.59 microHenries seems moot.

Further, we man-handle our inductors [changing the inductance somewhat] into a breadboard laden with 5% (or greater tolerance) capacitors, copper board pads/paths that exhibit C, active devices and so forth. Then, too, we connect these resonators, or filters to other blocks with sometimes reactive ports, plus or minus shielding. Despite all these variables, miraculously, we make the filter with our "measured" L work!

As we move up in frequency, the effects of stray L effect magnify and at some point our filter networks may behave poorly.

For band-pass filters with Carver-style device measured inductors, we need only adjust each trimmer capacitor to get the highest possible peak-peak voltage, or RF power with our filter between a signal generator and a 50 Ω terminated scope, power meter *respectively*, or whatever. After sweeping these peaked filters, rarely do I need to compress/expand, or add or remove windings to tweak the L get the desired filter response when the filter input and output ports are well matched.

In the case of single frequency matching networks like the L-match, we might need to tweak up or down the L to derive strong port matching. In all cases, our software and the L/C meter can get us into the ballpark, but in-situ bench measurement with other instruments will garner the home run.



---

6. Bob, K3NHI — RF Power Meter Follow-on

Bob built a follow-on detector and power meter to his 2002 QEX power meter *[Reference 3 in Section 7].* Bob gave me the green light to share his creation on my site. Thanks awfully Bob!

Click for the schematic in pdf format. Per typical K3NHI fashion, it's laden with trimmer pots allowing precise calibration — Bob's stuff contains loads of tweaks and wiggles! The new PM offers more flexibility + features including bigger battery supplies that won't quickly die when you fail to turn it off. He also included a means to measure battery voltage on the analog meter. Click for a photo showing the batteries. Low noise, high-speed, rail to rail, CMOS op-amps for IC1 and IC3, allow you to capture PEP during sideband transmitter measurement with appropriate attenuation.

The TS922 op-amp might be hard to find in DIP since that package went obsolete, however, it's available in SMD. If you can't get any TS922, likely other modern, high speed, rail to rail, CMOS op-amps will work fine — consult datasheets to ensure you meet or beat the TS922's performance for IC1 and IC3. Some of the latest design op-amps offer truly sublime specifications and evoke joy in our breadboards.

Bob critically *isolated* the entire RF sub assembly **outlined in green on the schematic** with metal shielding and feed through capacitors for the B+ and DC output. This helps ensure stable and accurate AC voltage measurement. Click or click or click for more of Bob's photographs.

7. References

[1] Roger Hayward, KA7EXM — A PIC-based HF/VHF Power Meter, QEX May/June 2005

[2] Wes Hayward, W7ZOI, Bob Larkin, W7PUA, Simple RF-Power Measurement, QST, June 2001

[3] Bob Kopski, K3NHI — An Advanced VHF Wattmeter, QEX, May/June 2002.

[4] Bob Kopski, K3NHI — A Simple Enhancement for the Advanced VHF Wattmeter, QEX, Sept/Oct 2003.

[5] Bob Kopski, K3NHI — A Simple RF Power Calibrator, QEX, Jan/Feb 2004 **+** Tech Notes article, QEX, May/June 2010.

RF — Test and Measurement

VHF FM

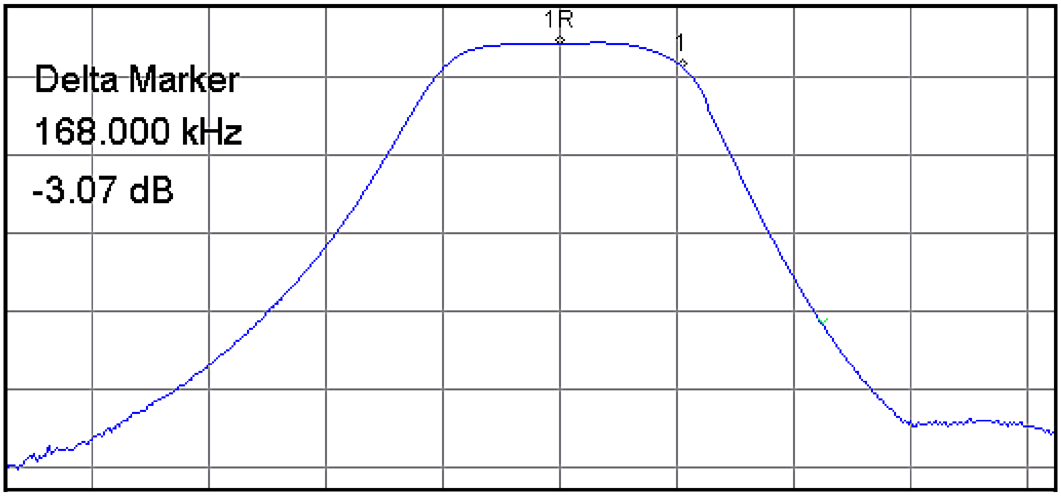
Yes - I could purchase a FM radio-in-an IC for $2.00 and be done in 35 minutes, but what would I learn?

Repository for FM superhet receiver experiments conducted from 2012 to 2014.

1. 10.7 MHz IF Filter Experiments
2. A Basic Colpitts VCO
3. DC-DC Converter for VCOs
4. Supplemental Page #1 - it's time to make some receivers
5. Miscellaneous Photos or Figures

---

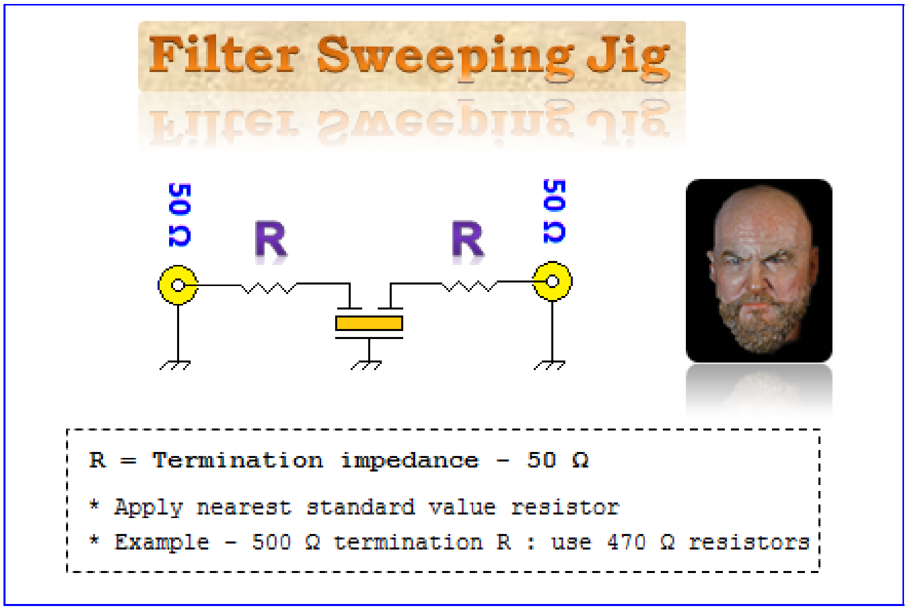
1. 10.7 MHz IF Filter Experiments



As a FM receiver design newbie, I read about and experimented with some 10.7 MHz IF filters to learn common practices, what's available and which measurements might help me to reach my goals. Over time, I've collected a variety of crystal and ceramic filters for hopeful future work. Click for 2 exotic examples. IF filters might be purchased at Ham festivals, surplus electronic parts stored and/or online. Prior to paying for a filter, I've found it useful to politely request a sweep of the filter, or, better yet, perform this task myself. To sweep a filter in your lab, you'll need a tool such as a spectrum analyzer with tracking generator, a VNA, or some other analog/digital sweep system.

I'll homebrew some crystal filters for narrow band FM in future installments — I ordered some 20 MHz xtals.

Sometimes a filter in your junkbox will state the IF and perhaps the 3 or 6 dB bandwidth, but not the input/output port termination impedance. How do we determine this impedance? I've learned we can figure this out by testing differerent termination resistors with this simple test jig:



Filter Sweeping Jig

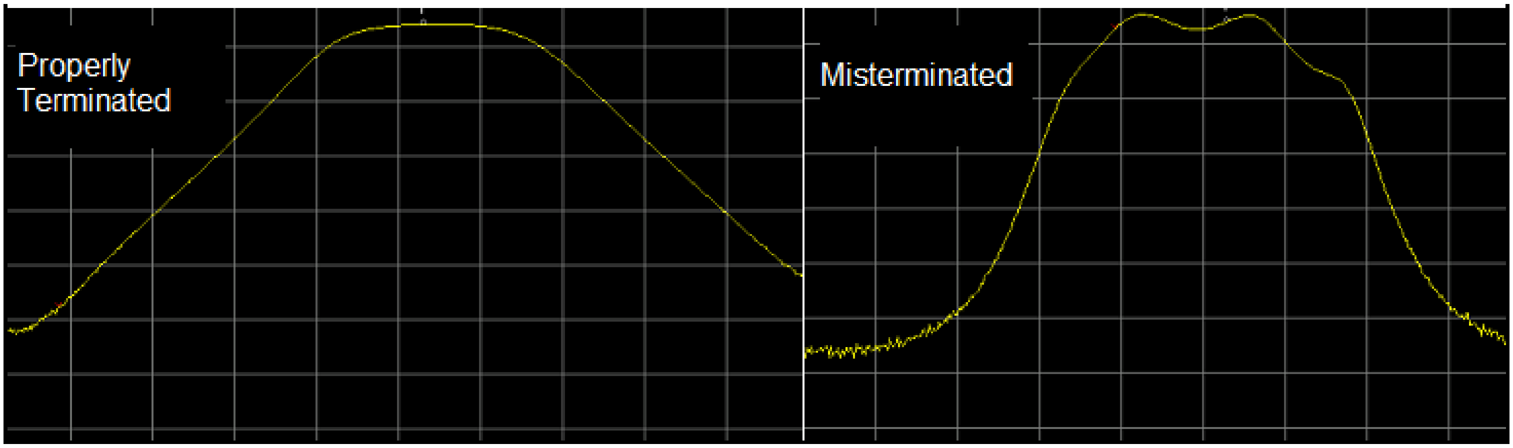
R = Termination impedance - 50 Ω

* Apply nearest standard value resistor

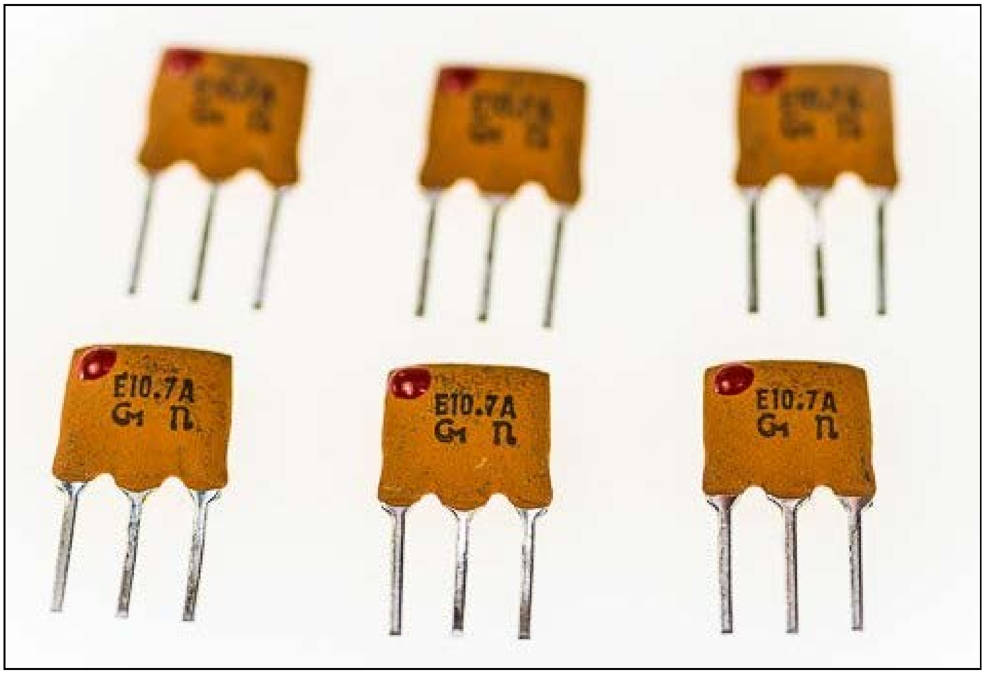
* Example - 500 Ω termination R : use 470 Ω resistors

Above — A simple crystal or ceramic IF filter sweeping jig. Since the series resistors attenuate the signal, losses occur; but the shape should look clean with minimal ripple. Normally, we builders will also place (or switch in) 50 Ω attenuator pads on both the signal source and detector within our sweep system to buffer impedance mismatch. Comparisons of this simple jig with more precise and complicated matching methods suggest that for many filter sweeps, it might work fine.

Above — My test jig with a Murata ceramic filter soldered in-situ. Keep the resistors *close* to the board. I've pretty much moved to SMA connectors in my lab: they're cheaper than BNC, plus we can buy a wide variety of quality 50 Ω patch cables donning various connectors for low cost. For example, a 30 cm cable with a male BNC and SMA connector on either end.



Above — A poor termination may result in improper bandwidth and ripple — easy to spot in this trace. Click for a trace from a 'gone bad' instrumentation crystal filter: 10.7 MHz @ 30 KHz with 2200 Ω Z in/out. Not really usable with ~ 10 dB ripple.

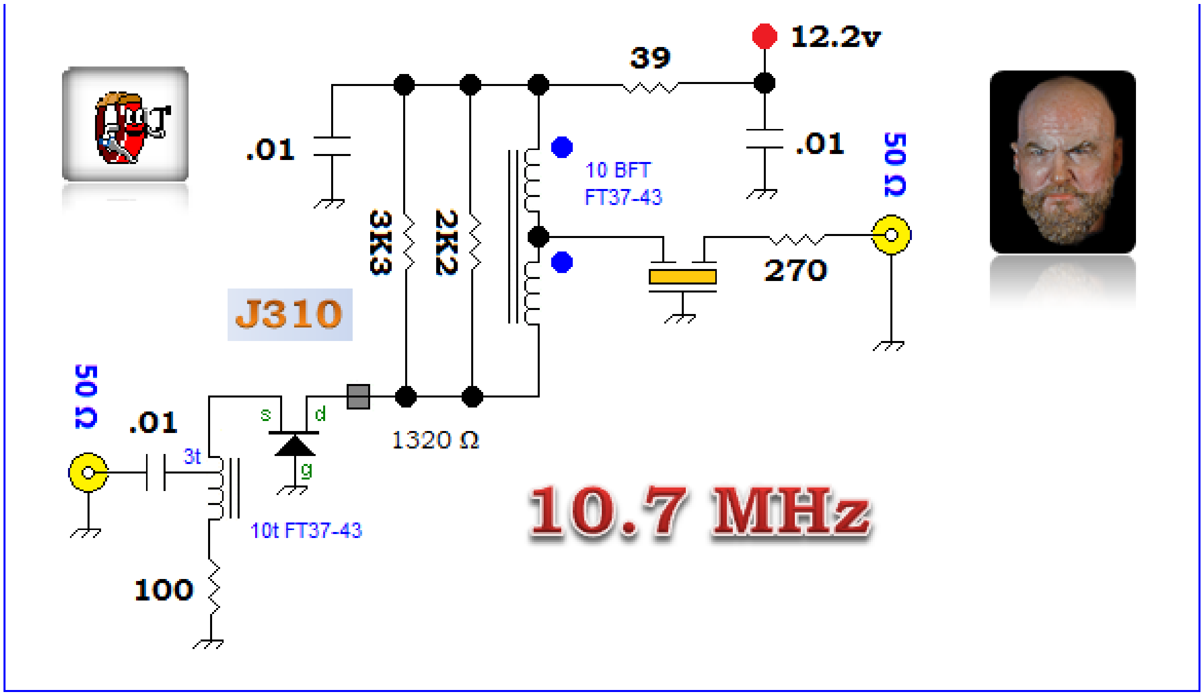


Above — Older 280 KHz Murata 10.7 MHz IF filters purchased long ago. Low cost = their main attraction, although they too will suffer total obsolescence and a price increase.

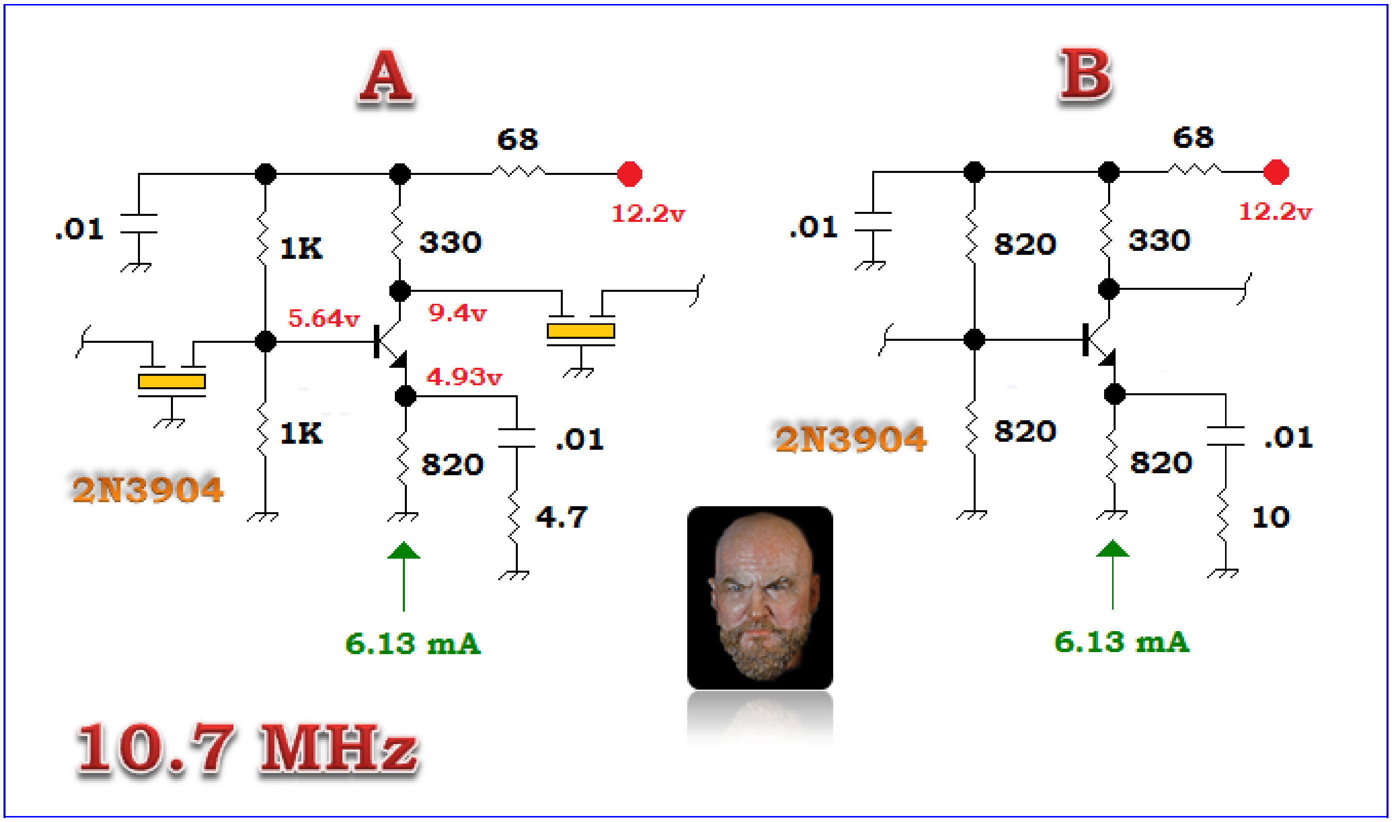
I bought some newer, lower insertion loss ceramic filters in the following bandwidths: 280 KHz, 230 KHz, 150 KHz, 25 KHz and 20 KHz. Check their datasheet — most Murata ceramic filters require a 330 Ω termination (preferably resistive) and I keep a filter sweeper jig with 270 Ω resistors as a regular bench tool. 280 KHz was a popular WBFM filter bandwidth in many older high-end FM receivers including my 1980's T-85 Yamaha receiver; my benchmark FM receiver.

Many of us hopeful FM builders, smitten by modern digital gear, fail to recognize the fantastic design achievements made by FM receiver engineers back in the day. All those air-variable, ganged band-pass preamp stages, low noise amplifers and often incredibly complicated and great sounding FM multiplex circuits just blow me away. Perhaps I'm a hopeless analog nostalgic? My T-85 sports 5 ceramic filters [280 KHz and 230 KHz B/W Muratas] and the narrow filters are listener switchable for narrow band Dx.

Ways to Match These Filters with Amplifers

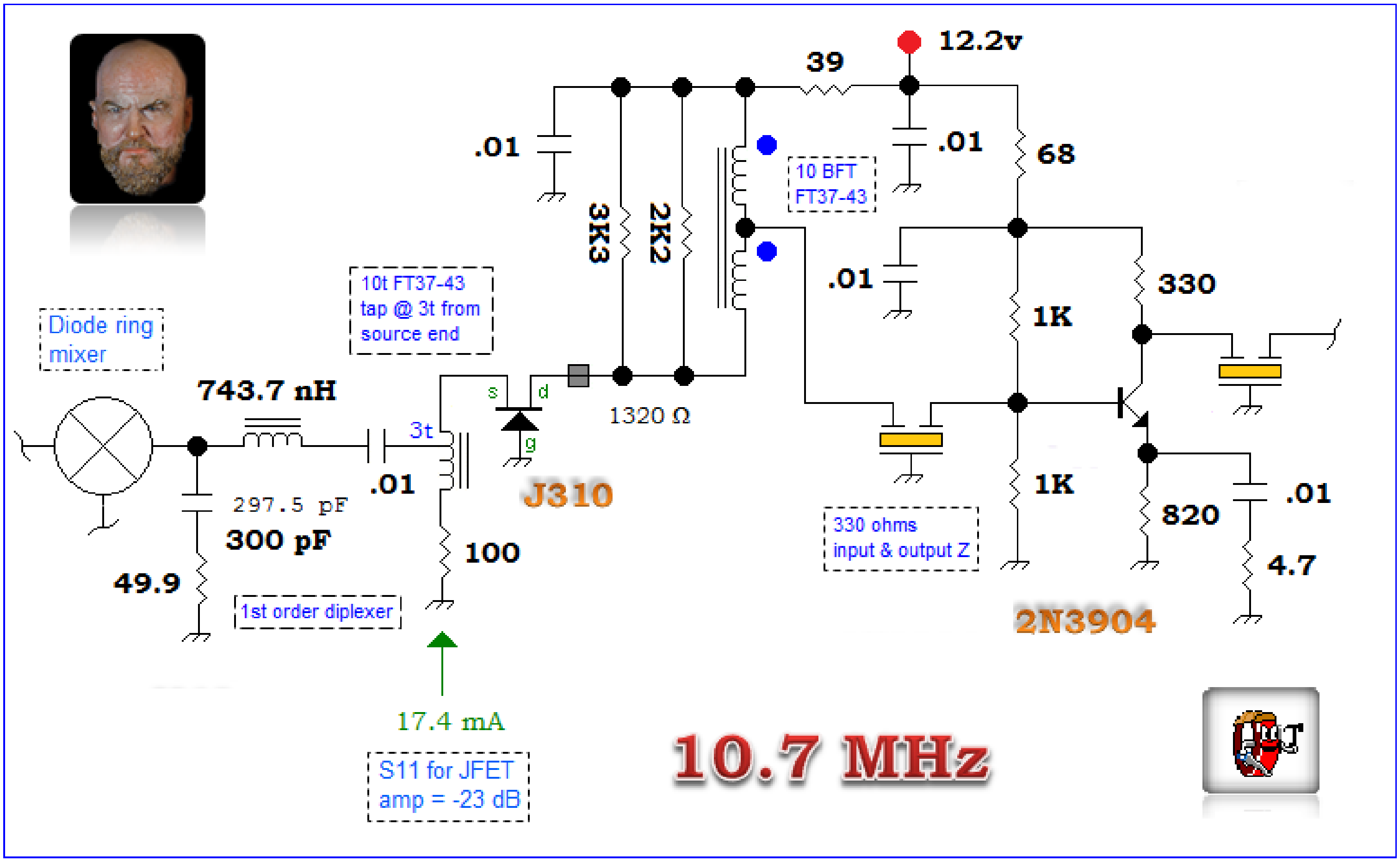


Above — A common gate JFET amplifer drives a 330 Ω ceramic filter. I placed 2 resistors in parallel to get the needed shunt R of 1320; my 2 resistors measured 1316 Ω. The bifilar transmission line transformer provides the 330 Ω Z to drive the filter. Details of the JFET amp come in a later schematic, but the input return loss at 10.7 MHz = 23 dB. I swept this circuit and it looked similar to the tracing with the same ceramic filter in my 270 Ω filter sweeping jig. This particular filter exhibited 6.4 dB of insertion loss.

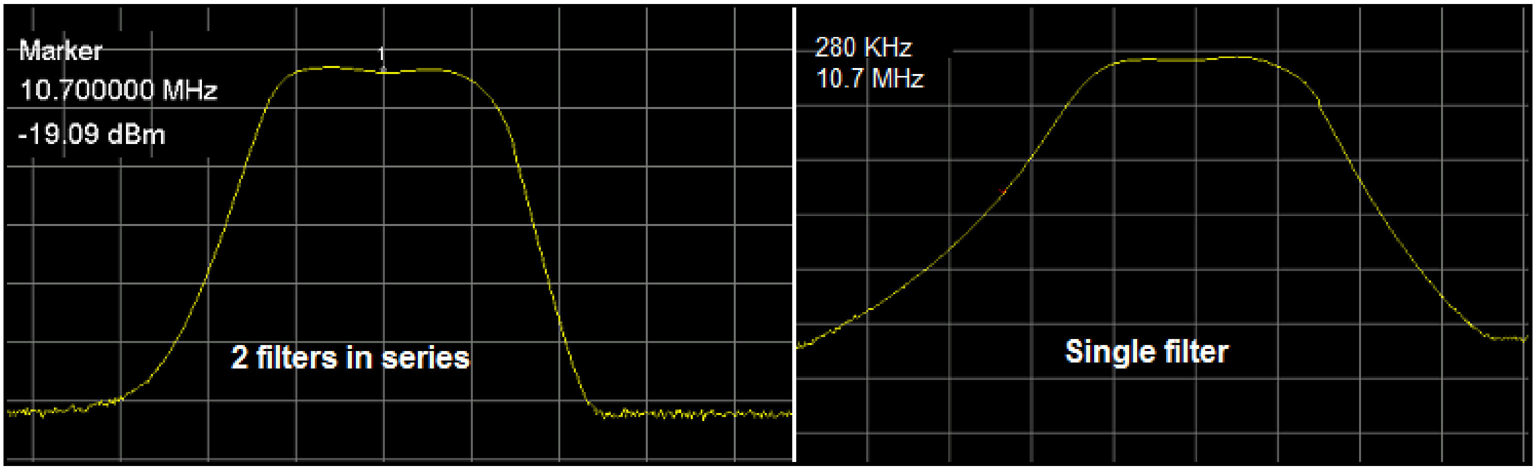


Above — 2 versions of a BJT amp with a 330 Ω input and output impedance. If you read schematics of good FM receivers, often the designers drive the filter with a 330 Ω collector resistor. Click for an example. By keeping the bias and degeneration resistors low and the current moderate, an amplifer with 330 Ω input Z is easy to design [although the input Z will vary with Beta].

I felt surprised that version **A** exhibited a voltage gain of 11.7 despite those low bias and collector resistors. You can stick a filter on either side as shown. Murata recommends a buffer amp between cascaded ceramic filters and you'll see this often in FM receiver schematics from the 1980s or so. Resistors provide wideband termination. Version B is the same amp with a little more degeneration to lower the gain and serves as a design example. I've got the procedure documented here under '*Calculating the input resistance of a common emitter stage'.*

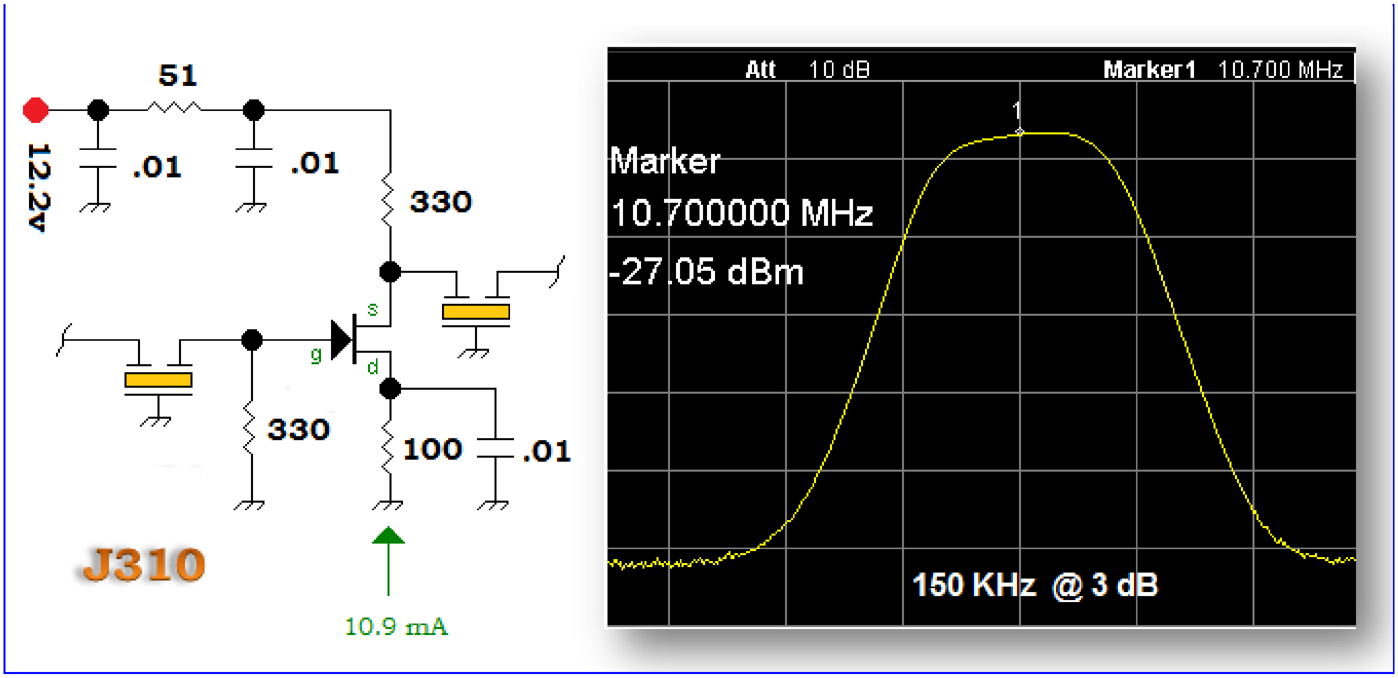


Above — An IF block using the designs shown earlier. I terminated this stage with a 270 Ω resistor and of course removed the mixer and diplexer. 2 sweeps lie below. I'm tempted to tune the JFET drain and couple the transformer with a few links as needed to get a 4:1 impedance ratio. Anyhow — food for thought.



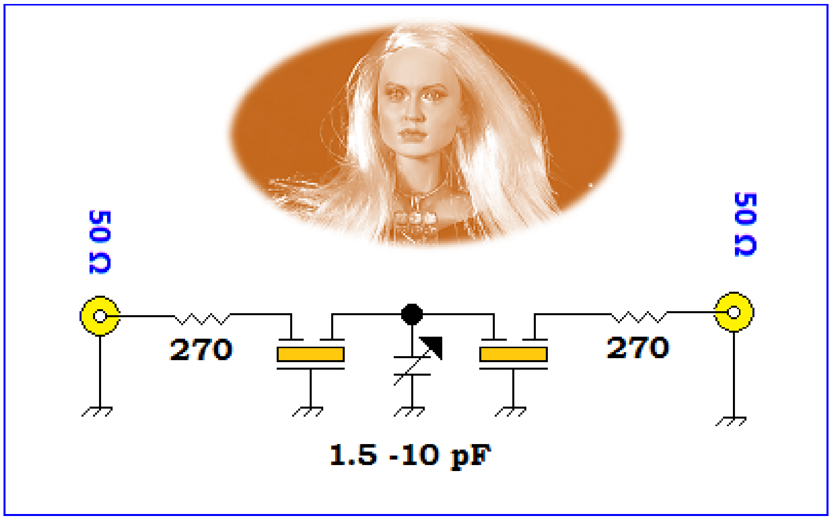
Above — A sweep of the IF block shown above left sans the mixer + diplexer (*two* 280 KHz ceramic filters). On the right lies the trace of the common gate amp driving a *single* 280 KHz filter with its output terminated with a 270 resistor. The advantages of 2 cascaded filters seems apparent, although the slight downward dimple at the center frequency might represent some capacitive loading at the output of the common gate amplifier.

I built a number of other amplifers and swept all of them Click or click for 2 early examples that use active devices instead of series matching resistors on the output. In these circuits, R Term was changed and then the circuit was swept. The tracings looked good.



Above — I built Brian, K6STI's nJFET IF amp. He used it to offset the losses associated with 2 ceramic filters. Click for Brian's fabulous website. I placed 150 KHz 3dB BW filters before and after a J310 and swept — my circuit exhibited a 2 dB net loss which seems quite reasonable.

The 330 input resistor is a load/termination on the input filter and will dissipate some energy and lower the AC input voltage to the gate compared to the usual high Z input resistors we apply in our JFET common source amps — from open circuit to full termination would incur a 6 dB voltage drop. Still, for simplicity versus performance, Brian's circuit looks hard to beat.



Above — 2 ceramic filters in series. I added a small trimmer between the pair in hopes to mitigate any filter skirt distress or ripple. Click for a tracing with and without the trimmer capacitor. You might experiment with the filter coupling and the filter block termination impedances to better their skirts and passband The losses of the above filter block may reach 12-14 dB.

If you don't have a sweep system, I was able to crudely test the amplifers + filters with my 10.7 MHz signal generator and a DSO.

Resistance Bridge

If you go with a BJT IF amplifer, it's possible to measure the input impedance with a bridge and tweak the emitter current and/or degeneration resistor to get very close to a 330 Ω Zin. I keep a drawer with through-hole resistors rated between 1 and 10 Ω for tweaking my emitter resistor values to change series feedback in my common emitter amps.

I first designed a simple 330 Ω bridge for measurement with my DVM. It worked, but the null lacked the depth and resolution we need. Later I improved the sensitivity by adding another coil and changing to a 'scope or SA detector, but after building EMRFD Figure 7.36, I abandoned my bridge. Figure 7.36 just blew me away. The null of a 330 resistor was only a few 10s of microvolts during calibration.

I placed a small 500 Ω pot in parallel with a 120 Ω resistor for the variable resistance. After some basic testing, I calibrated it with a 330 Ω resistor; adjusting the pot for the deepest null and just left it there for testing my 330 Ω IF amps @ 10.7 MHz.

I plan to make Figure 7.36 for VHF and maybe UHF with chip caps plus a small screwdriver adjustable trimmer pot [to get the lowest possible L] calibrate it and make it a part of my test bench arsenal. After getting a null, we measure the pot's resistance with an ohm meter to learn the impedance at the ? port

Considering that our predecessors measured just about everything RF with a bridge, this little circuit suddenly become relevant. A series L and C "add-on" circuit shown as Figure 7.39 may be placed in series with the ? port and device under test to deepen the null in the face of reactance. Bridge circuits form the very essence of RF measurement. Yes Bobby, we can measure impedance without a VNA.

---

2. A Basic Colpitts VCO

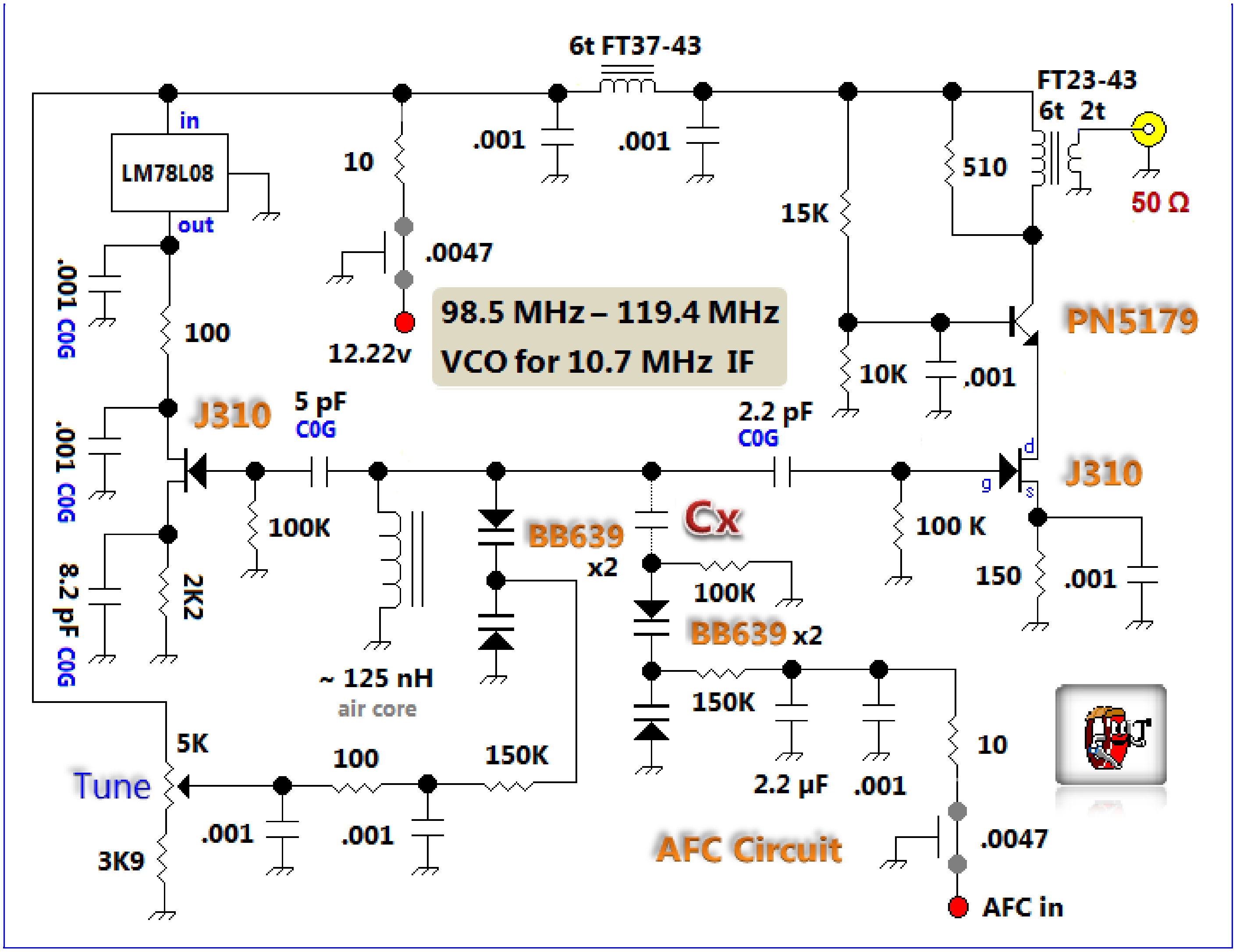
Above — My completed Colpitts VCO. *I installed the unlabelled, left-sided pot in case a potentiometer is required for future AFC circuitry changes. It's not hooked up.*

I reviewed some 1970's FM receiver schematics to learn that before PLL-locked VCOs dominated, often Colpitts VCOs were locked onto a strong frequency with Automatic Frequency Control (AFC). Local oscillators tanks often employed a inductor plus an air variable capacitor that tuned from ~77 to 119 MHz with a varactor for AFC. All the tuning and front-end filter air variable capacitors were ganged together and I'm sure alignment took some skill.

Some VCOs tuned with varactor(s) instead of an air variable cap — this is what I wish to do. Varactor tuned VCOs usually suffer more thermal drift than air variable capacitor versions.

AFC compensates for VCO thermal drift by a seperate varactor with its control voltage line DC coupled to the FM detector through an R-C low-pass filter. Any difference between the VCO frequency and the desired FM frequency produces a proportional DC voltage. The DC control voltage changes the oscillator to the desired frequency by re-tuning the AFC varactor within this feedback loop, albeit over a limited range. AFC is unsuitable for weak signal DXing, since it may pull the receiver onto a strong adjacent signal. Many 1970's FM receivers supplied an AFC defeat switch.

I remember 1 old FM receiver in my parent's home that stayed locked on 1 frequency for years thanks to AFC.



Above — The schematic of my version of a JFET Colpitts VCO (with AFC) that lacks the standard gate to source feedback capacitor; the intrinsic capacitance from the J310 gate to source provides the feedback needed for oscillation.The 8.2 pF bypass cap was determined on the bench — too little, or too much C decreases output voltage, or snuffs out the oscillator.

I just couldn't bring myself to make a VCO with a BJT, since on my bench at least, they suffer more thermal drift than JFET-based oscillators. I built with a mixture of SMT and hole-through capacitors and resistors. The anti-parallel arranged hyperabrupt varactors were found on eBay. Click for a rear photo of the project chassis. The gold colored jack is an SMA connector.

I bench designed this VCO and it took many hours to find the correct amount of L and C for the resonator to give a low distortion, sine wave output across the ~21 MHz tuning range. This meant soldering in and removing these tank components frequently. Click for the lowest frequency output. Click for the highest.

In the example local oscillators I reviewed, the engineers made no attempt to level off the signal that normally increases in AC voltage as you increase frequency. I also ignored levelling. Presumably the designers didn't worry with leveling the oscillator output in their superhet receiver as long as the output voltage sufficiently drove the mixer into complete switching. Levelling would add cost and complexity. This isn't a lab grade RF signal generator — that's for sure.

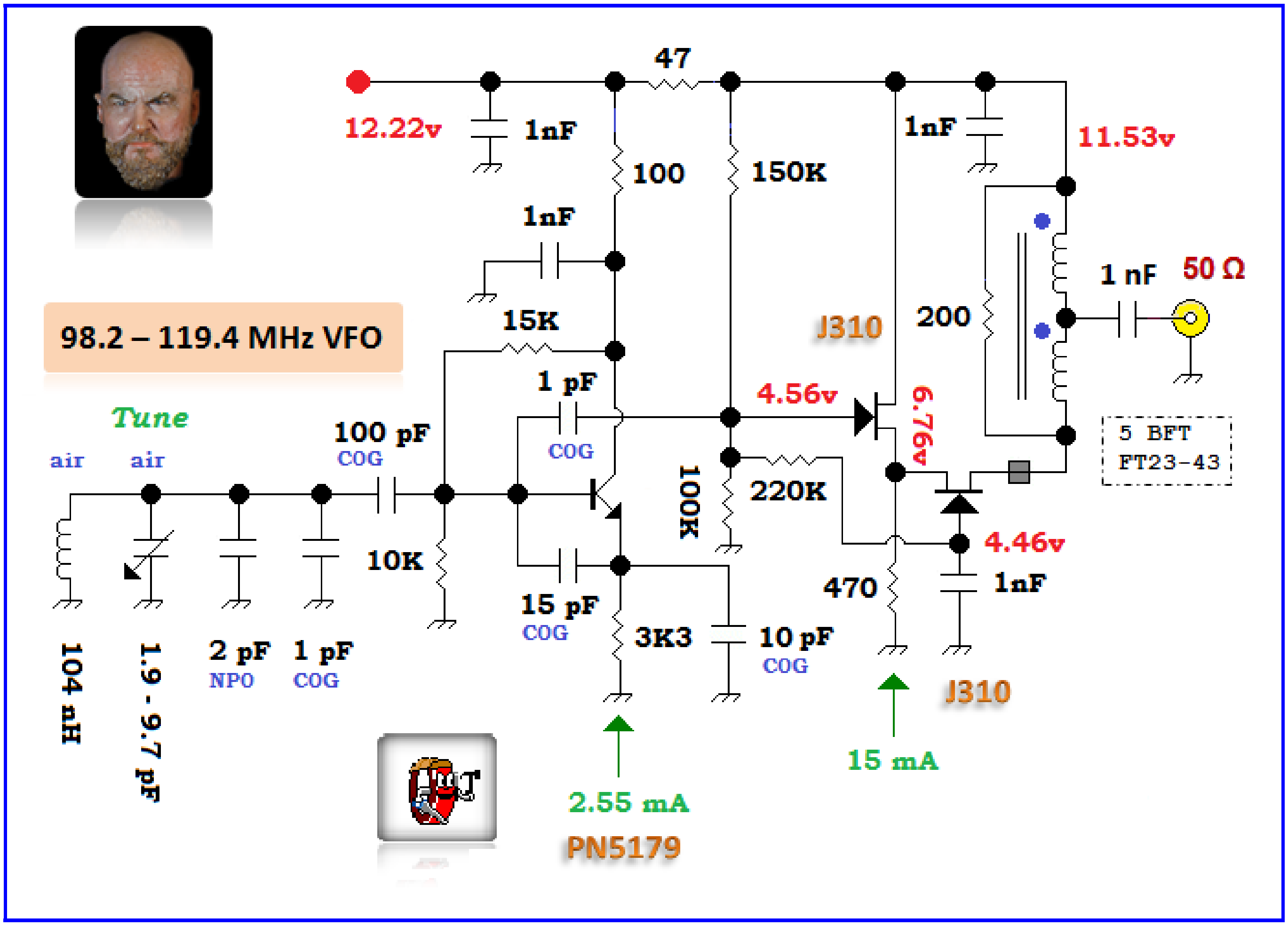
At present, the AFC varactor pair is disconnected since I won't know how strongly to couple it with Cx until I have a working detector. Also I will need to experiment to determine the best R-C time constant for the low-pass filter; likely the 2.2 uF capacitor will need an increase in value.

With the 3K9 Ω resistor under the 5K tuning pot, I keep at least 5 VDC on the tuning varactors or the VCO would stop running as I tuned the pot towards CCW. The coil = about 3 turns of 16 gauge wire on a 5/8 inch bolt. (Despite Canada going metric in ~1975, they still sell nuts and bolts in inches at our hardware stores). The stiff wire prevents the inductor from turning into a "microphonic" spring when the VCO is bumped. Click for a photo. The nominal L = ~ 125 nH, although I bent and manipulated the coil so it sat attached to the copper clad board with no tension and then squished or expanded the turns to establish my lower band edge.

In many FM receivers, either a single or balanced dual-gate MOSFET mixer was driven by a high impedance buffer/amplifer. If I mix with a 2-gate MOSFET, I'll insert a common gate JFET amplifier on the IF strip to boost the LO output impedance and AC voltage.

The feedthrough capacitors are 0.0047 µF - they were on sale so I bought them. To prevent a parasitic high impedance when placed in parallel with my standard 0.001 µF bypass caps, I placed a series 10 Ω resistor.

I enjoyed this crazy design; trying to replicate a relic, but popular local oscillator idea from decades ago. Let's hope I did it justice. Perhaps future VHF stuff on the FM and even 2 meter band will involve an Si570 and PIC, Arduino or other microcontroller? This simple VCO will do for now. My greatest passion lies in designing and building the front end.



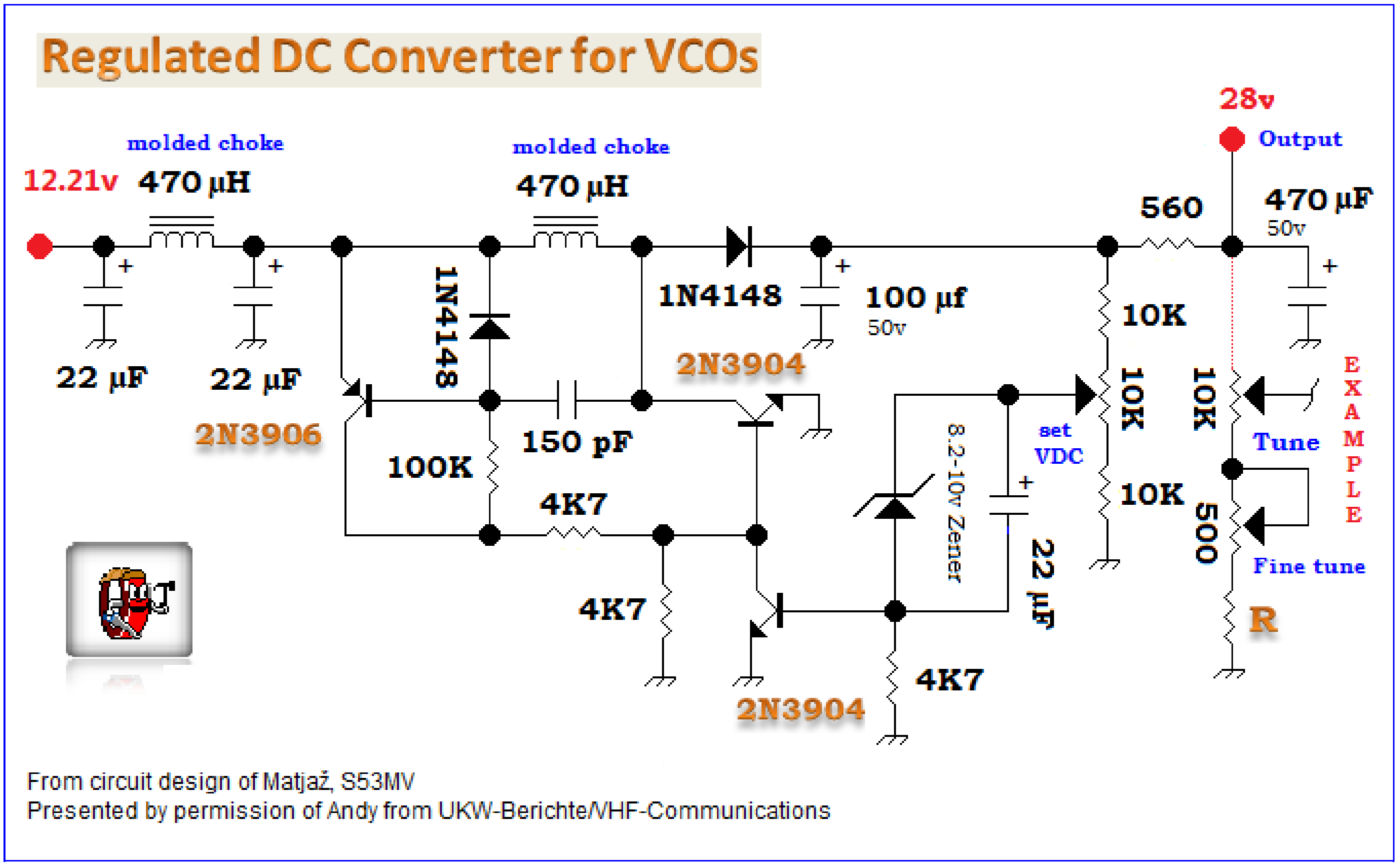
Above — A well-buffered, bench-module, high-side VFO I sometimes use for broadcast FM band mixing into a 10.7 MHz IF. The output at 98.5 MHz = -5.35 dBm, perfect for switching Gilbert cell mixers with a little padding or amplitude tweaking. Click for the output of the Colpitts only with a 10X probe @ 120 MHz. With care, you can see the second harmonic in the 'scope tracing — click for the SA tracing that shows the 2nd harmonic 27.5 dB down from the carrier. Click for a 'scope tracing with my MMIC bench module amplifier from VHF Veronica connected; the amp exerts some low-pass filtering that cleans up the signal somewhat.

---

3. DC-DC Converter for VCOs

Until now, I ran a maximum reverse DC voltage of ~12 volts in my varactors. For wider VCO or L-C filter tuning, builders may chose 28 volt varactors such as the BB535 or BB149A and boost the 12v supply up to 28v with a DC-DC converter. Some build inductorless converters pulsed from 555 timers, or use CMOS voltage converters like the CL7662, or Si7661 to make a doubler. As an RF constructor, I like working with coils and built the following circuit:

Above — Bench module: 28v DC to DC converter. While containing no tuning control pot, my build places the zener diode regulator control potentiometer on the front panel to allow fast-tweaking of the output voltage from ~21-30 VDC depending on the load. Click for the breadboard photo.


Regulated DC Converter for VCOs

From circuit design of Matjaž, S53MV
Presented by permission of Andy from UKW-Berichte/VHF-Communications

Above — My regulated 28v converter for varactor tuning adapted from a design by Matjaž, S53MV. I pulled this circuit from his *amazing* 2-part article with circuits that span from 11 GHz RF to DC. *See the reference articles below*. I filtered heavily and at switch-on, my circuit draws ~ 50 mA, but then drops to ~ 11 mA after the capacitors charge. The 10K *[set VDC]* trimmer pot allows you to dial in your desired output voltage and thus this converter may work over a wide range of DC power supply voltages.

The tuning control(s) might be a single potentiometer, or even seperate pots for tuning 2 different VCOs. In the above schematic, I show 1 possible tuning scheme: a 10K coarse tuning in series with a 500 Ω fine tuning potentiometer. R keeps some minimal reverse DC on the varactor(s) and is optional. Again, my bench module DC converter omits any tuning controls — these are built into the circuit containing the varactor diode(s).

The oscillator frequency varies slightly with the set output voltage. Click for a screen capture at ~32 VAC with a 10X probe placed on the PNP emitter. In another test, with no load, I watched the coil's magnetic field collapse and ring in this cool 'scope capture. This is why I love deep memory DSOs so much.

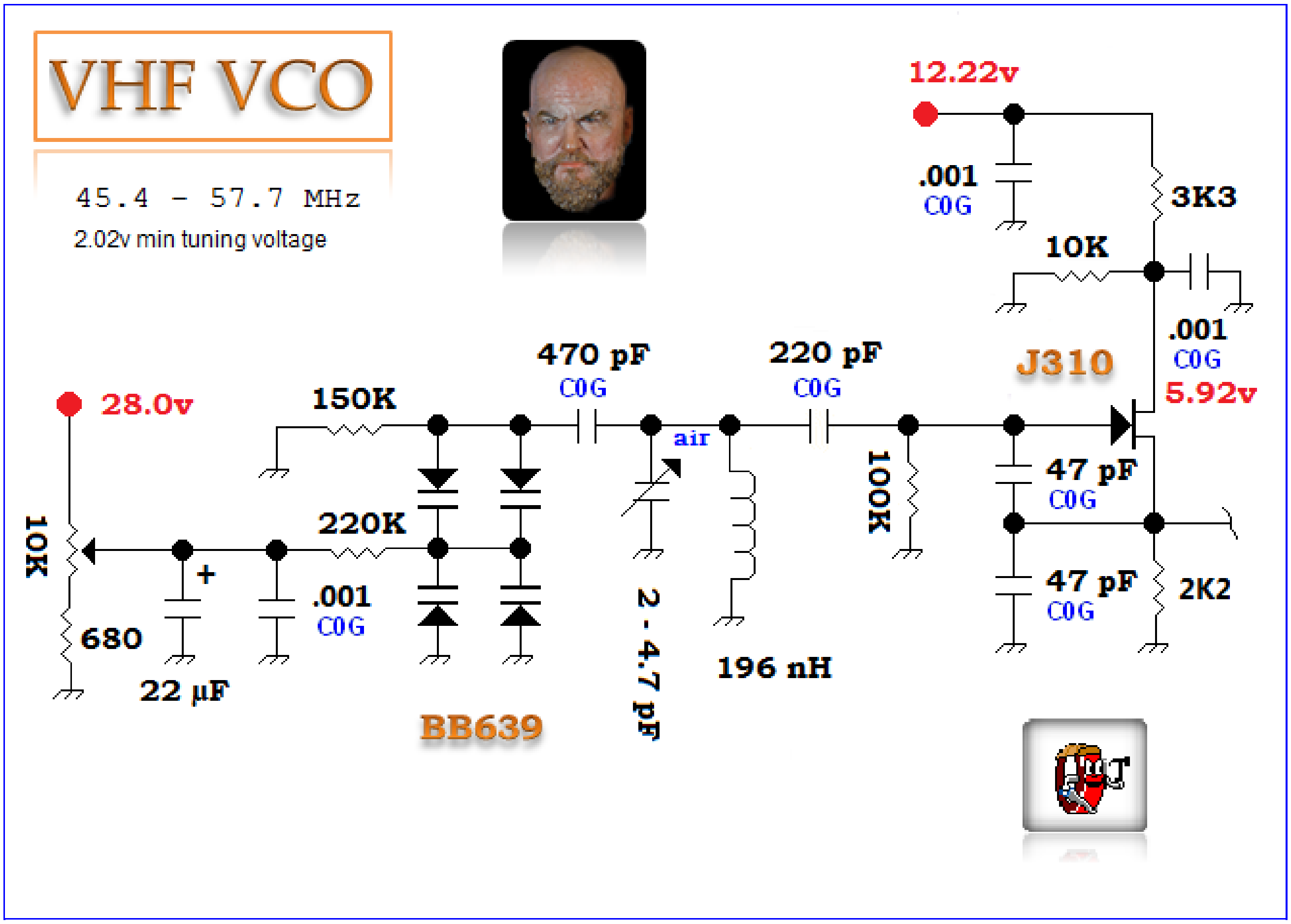
VHF COMMUNICATIONS

TV Satellite Receive System Part 1 : Low Noise 11 Ghz Down Converter ---- issue 4/86

TV Satellite Receive System, Part 2 : Indoor Unit ------- issue 1/87

http://www.vhfcomm.co.uk/

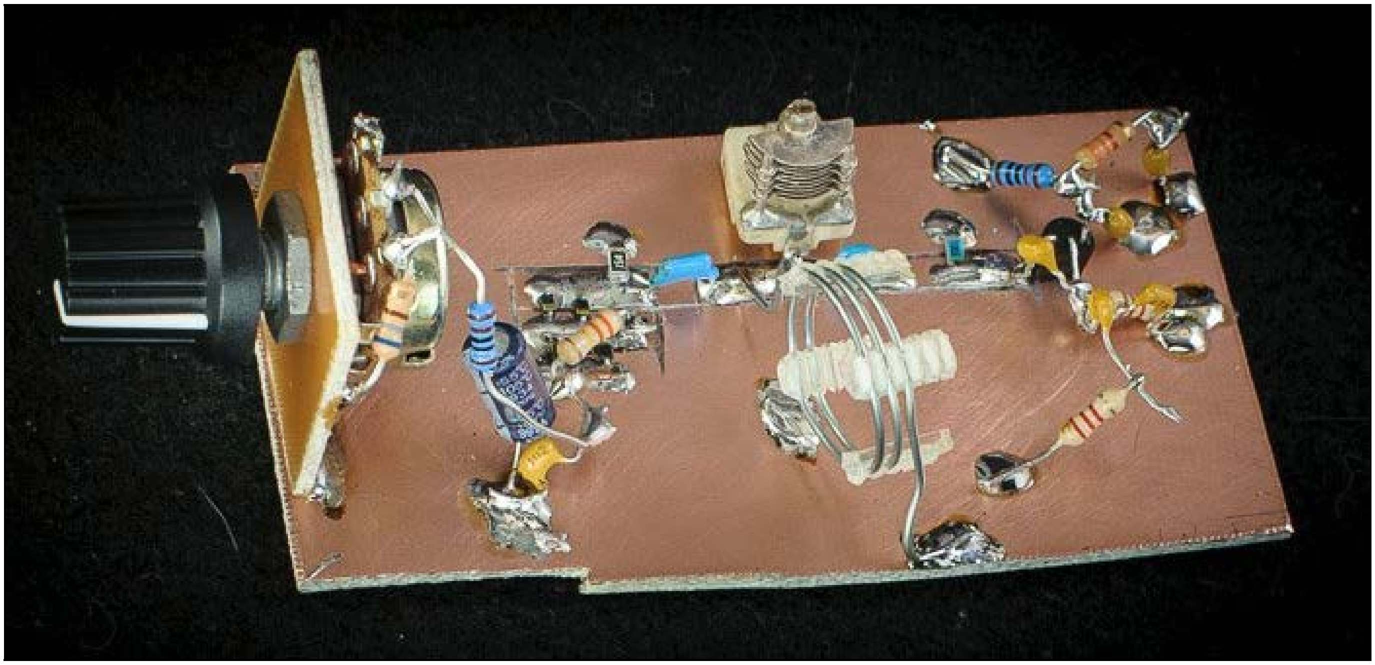
Above — Andy, G8ATD who owns VHF Communications magazine granted me permission to show the DC converter circuit. His magazine archives provide a treasure trove of useful circuits from VHF to Microwave and it's clear Andy passionately spent lots of time publishing the magazine until 2013, plus scanning and organizing the archived material.
Although VHF, UHF and microwave focused, much of the concepts and learning can also enrich your HF exploits.



Above — A "quicky" VCO thrown together to test the DC-converter. Click and click for the output with the tuning pot set to fully CCW and then

CW. The 680 R keeps about 2 volts on the varactors with the tuning pot set to CCW.

In the reference articles cited above, you'll find 2 HF-VHF Hartley VFO designs that tune over a 20 MHz span thanks to 28 volt varactors and careful design. In yet another UHF circuit, a 1 octave tuning span is realized with the author's specially designed VCO. Truly hardcore design from a great teacher — I crave exposure to the work of such authors.



Above — A photo of my "quicky" VHF VCO. 73!

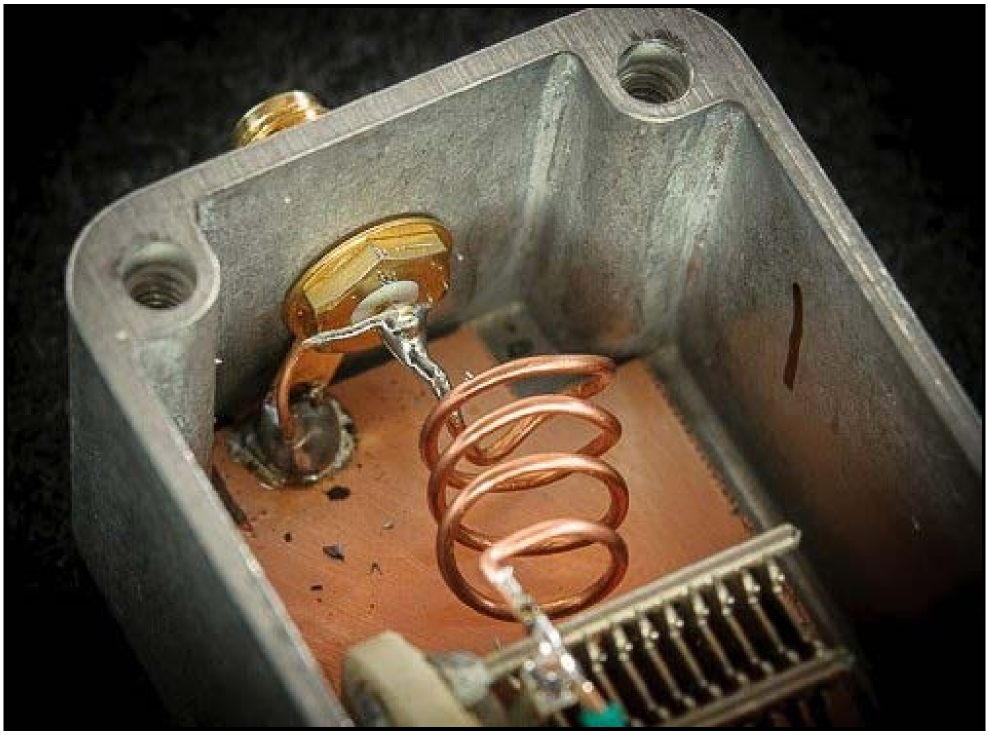
---

4. Supplemental Web Page #1

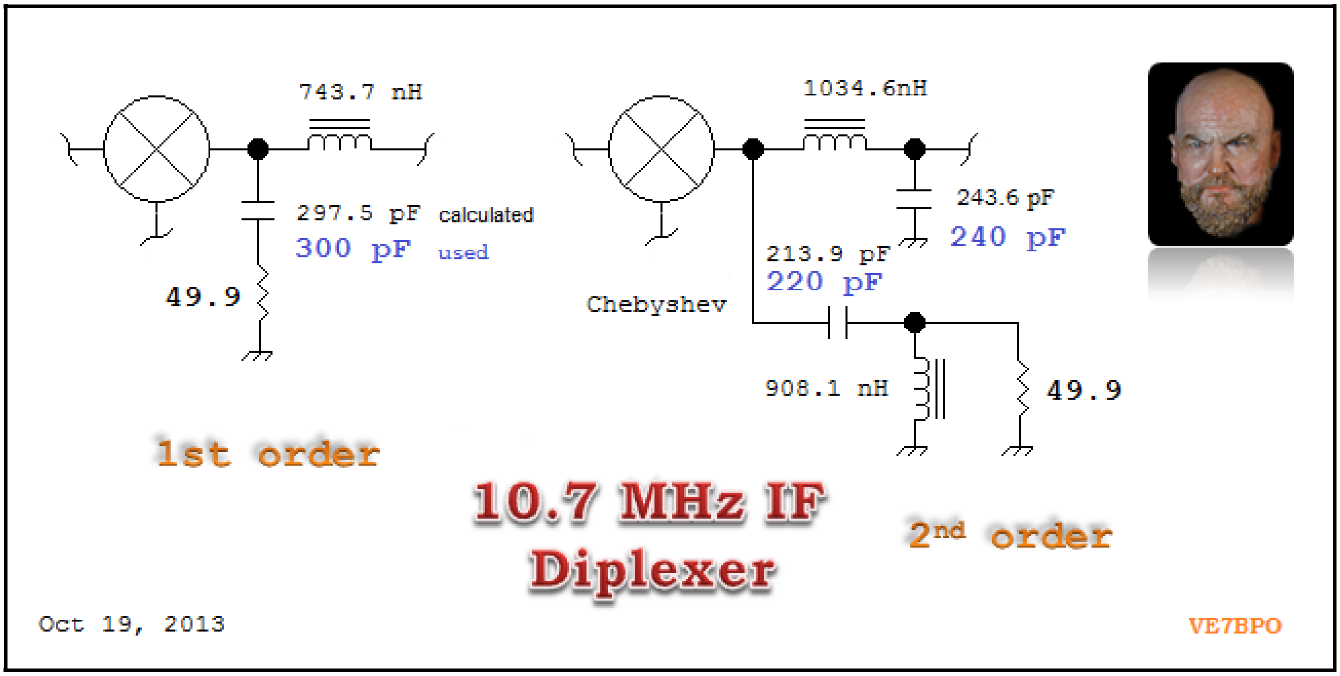
Click for the first supplemental web page.

---

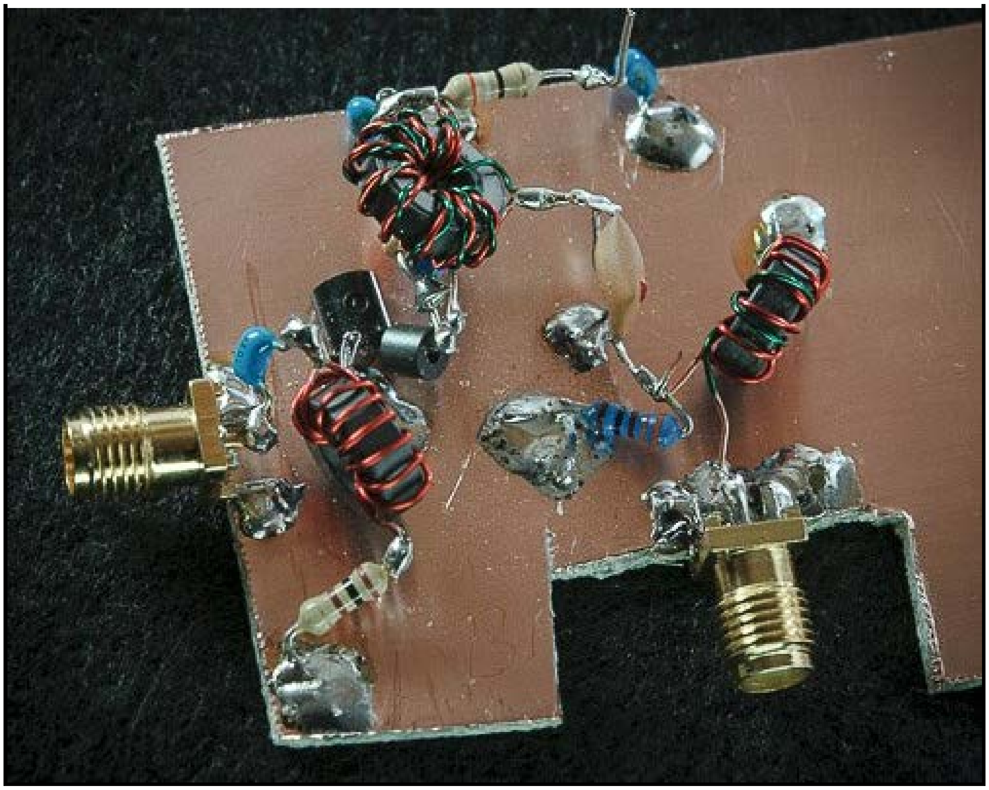
5. Miscellaneous Photos or Figures



Regency
TRANS
OFF | VOL
SQUELCH
MIC
F1
F2
F3
F4
BTL-304
VHF-FM
IVER



743.7 nH
297.5 pF calculated
300 pF used
49.9
1st order
1034.6nH
243.6 pF
240 pF
213.9 pF
220 pF
Chebyshev
908.1 nH
49.9
2nd order
10.7 MHz IF
Diplexer
Oct 19, 2013
VE7BPO



RF — Test and Measurement

Sundry Experiments 2012 - 2013

This page shows some of my better non-VHF experiments for 2012-2013.

Although VHF and UHF excite me greatly, It's always fun to build at HF, or even AF.

Section 1: I explain why you might see sweeps that look like oscilloscope tracings on QRPHB: they're devices swept with equipment designed by Bob, K3NHI. Bob's work simply amazes me — full-on, creative precocity.

Section 2: An LM1875 AF power amplifer test.

Section 3: Three Questions with Jason, NT7S.

Section 4: EMRFD Experiments — A 1-on-1 Tracking or Offset Phase-locked Loop.

Section 5: Boot-strapped Popcorn AF Feedback Pair.

Section 6: Non-Mechanical Iambic Paddle.

Section 7: A Journey Above HF.

Section 8: Popcorn AF Amplifier — Reprise

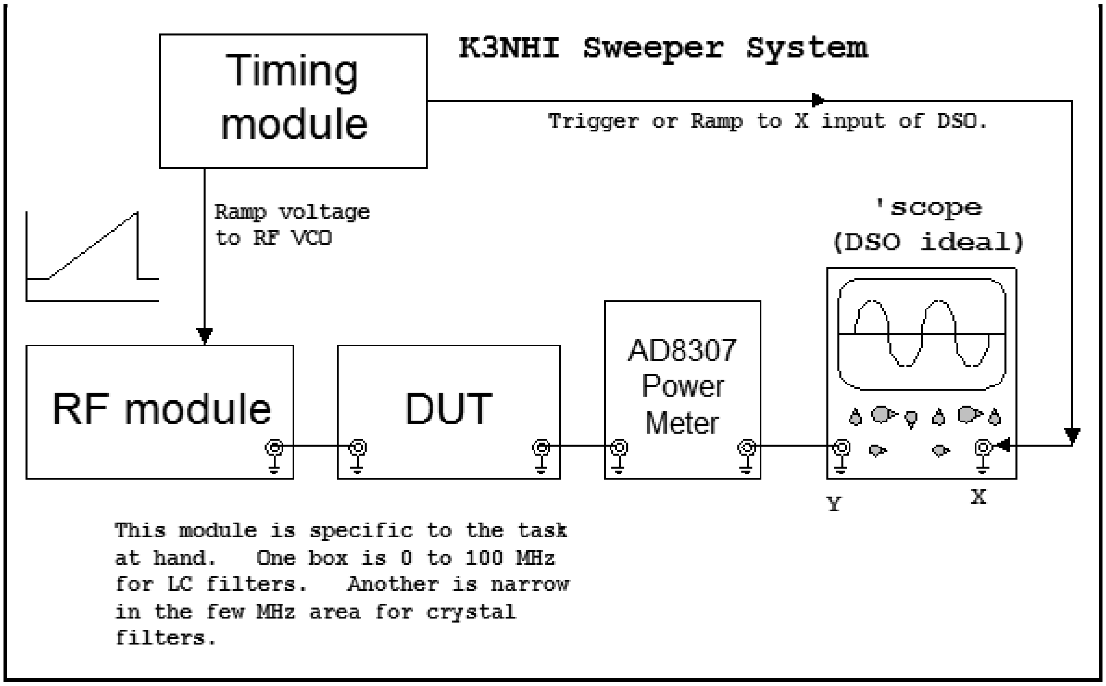
Section 9: The Progressive Receiver by John, K5IRK and Wes, W7ZOI

Section 10: Miscellaneous Pictures and Figures

---

1. Analog Sweep System

Today, advanced experimenters might build a network analyzer/sweeper incorporating a microcontroller, a DDS, or Si570 based frequency synthesizer, plus the needed analog RF circuitry. I went another direction: the Bob Kopski, K3NHI sweep system — all analog, no lines of code and probably 4X the bench work. I show some photos, traces and text in hope it might inspire you to pursue your own sweep system — digital-based or otherwise.



Above — The K3NHI Sweep System macro diagram drawn by Wes. The resultant trace looks like the output of a tracking generator plus spectrum analyzer. What I like most is that I'm measuring with my "tough" 'scope and need not worry about input power and so forth like we do with expensive RF test gear. I simply love measuring signals with my oscilloscope. Testing circuits with Bob's sweep system compels me to treasure component-level analog design and renews my passion afresh.

Above — 3 components of my K3NHI sweep system. Click for a higher resolution photo. To date, I've made the Utility Sweep Generator (time base) *on the bottom*, a 1-118 MHz VCO with clean and level output *top left*, an AD8307 power meter optimized for sweeps *top right* and a crystal filter VCO *shown later*.

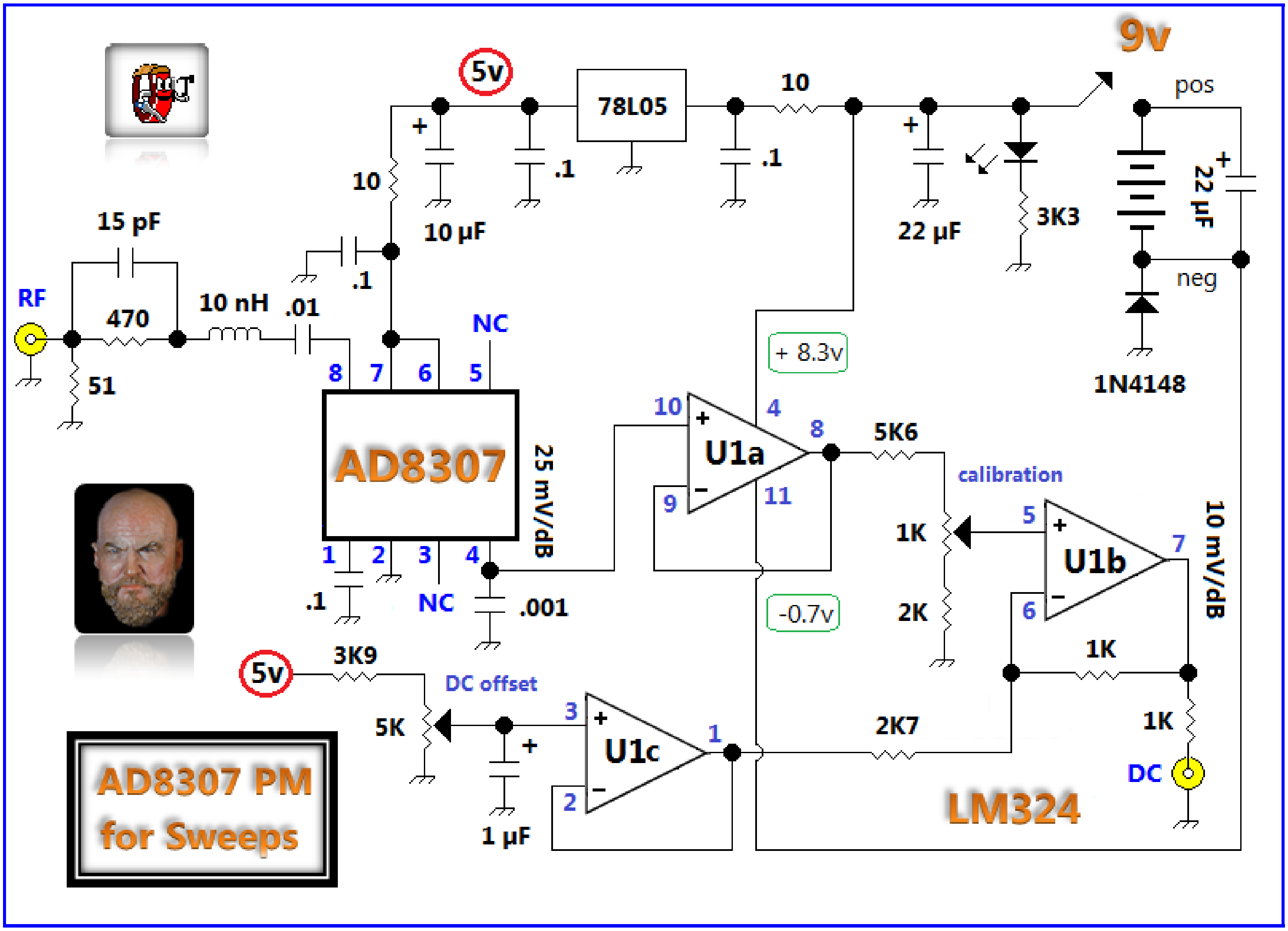
Building Bob's Utility Sweep Generator proved difficult. Although technically just a ramp generator, this 1 is calibrated, provides high isolation between the X and Y channels and will sweep *anything*. The power supply has 8 different regulated DC voltages including ~ - 3V. I may use it as the time base for a spectrum analyzer project 1 day.
Bob gave me permission to post the schematics: One Two Three IC1-5 is an LM324 op-amp. Doc 1 Doc 2

Email me for some build notes.

Click, click, or click for photos of the 1- 118 MHz VCO. Click for the schematics. One Two

I show a bare-bones AD8307-based Power Meter (PM) on the **RF Workbench 5** web page, however, to augment the PM for sweeping, I added input compensation, plus some tweaks from Bob's QEX articles: Bob Kopski, K3NHI — An Advanced VHF Wattmeter, QEX, May/June 2002 *and* Bob Kopski, K3NHI — A Simple Enhancement for the Advanced VHF Wattmeter, QEX, Sept/Oct 2003.

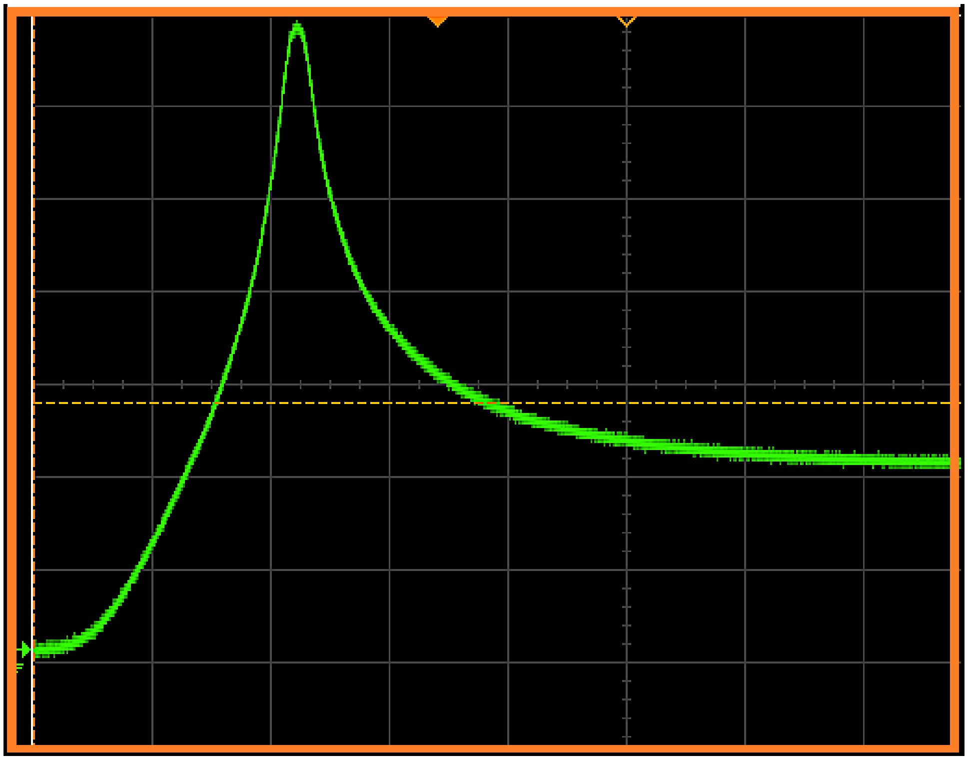
I strongly recommend you build Bob's power meter (referenced above) if you're contemplating a power meter build.



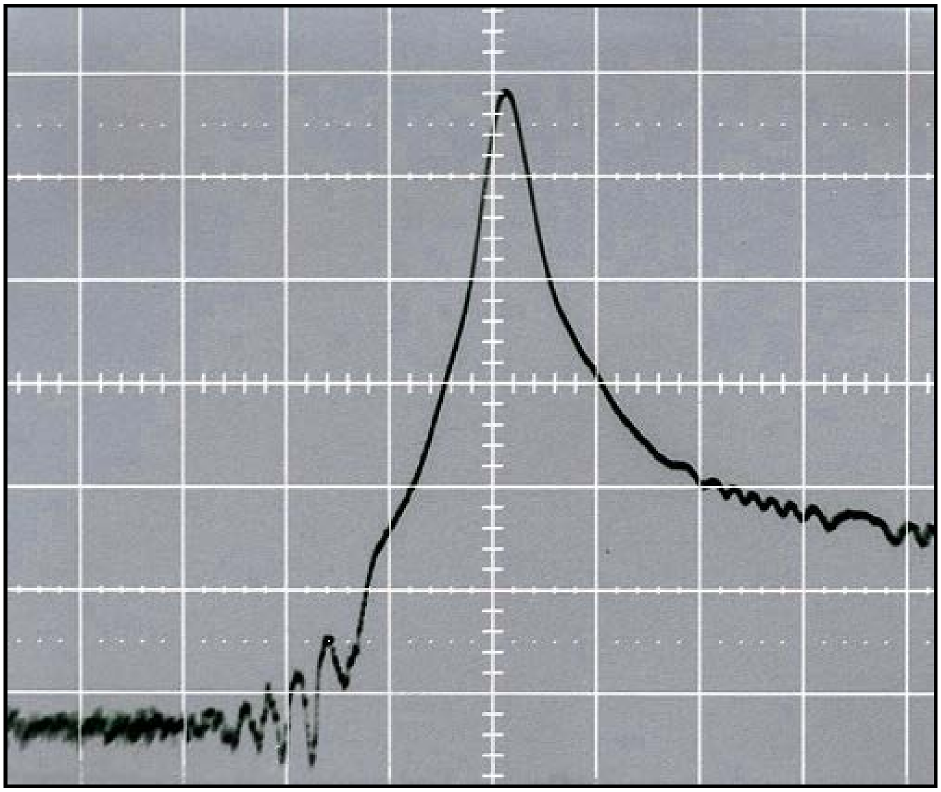
Above — My AD8307-based power meter. Click for a photo of an early version lacking the level shifter. The level shifter, or DC offset control allows precise Y axis control to enable a resolution up to 1mV/dB when set up properly in the DSO. With this resolution, it's possible to see filter ripple.

Above — The K3NHI sweep system in action. This photo shows the power meter with the (later) added DC offset control. Using the offset control potentiometer, I'm able to examine the top of a signal peak at 1 dB per division with various spans.

Above — I swept a 7 MHz band-pass filter bench module.



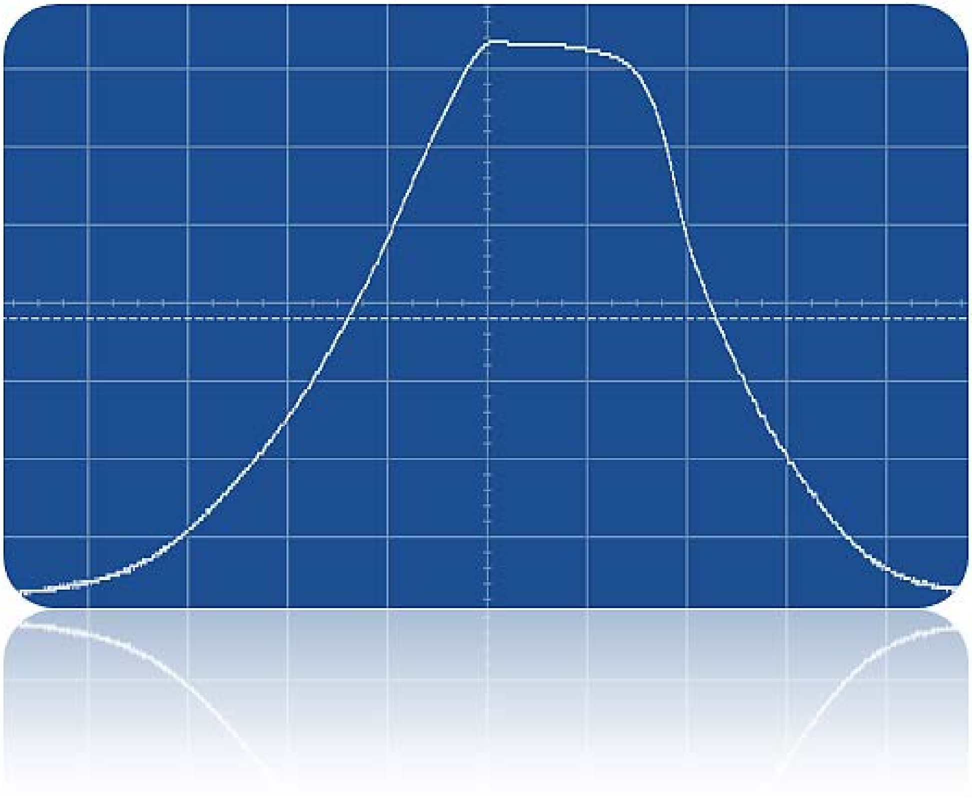
Above — A sweep of the above 7 MHz band-pass filter. At the time I was still learning system calibration and remember feeling blissful that I made such a cool sweep system.



Above — A sweep of the 7 MHz band-pass filter with a tracking generator plus spectrum analyzer for comparison. The span differed from that used with the K3NHI sweep system, however, I'm sure I'll get better with the K3NHI system over time.

Above — My build of the K3NHI *Hartley* VCO for sweeping crystal filters. I ordered a 4 mm tuning knob for the 10-turn "Manual Tune" potentiometer used to help center the sweep in my 'scope. This 10-turn pot, a DIP switch with C's and a secondary L, plus two MV209 varactors allow narrow resolution sweeps (< 5 Hz nominal) within a ~2.5 - 18 MHz range.

I monitor the sweep frequency on a counter via the VCO monitor port and all the tweaks on this VCO and the Utility Sweep Generator allow easy filter centering. Click for a breadboard of the VCO with the first buffer and a temporary BNC connector for testing. Click for the whole project. The secondary wideband buffer provides strong signal fidelity, reverse isolation and output return loss (22.3 dB) — it draws 61 mA. Click for the VCO schematic courtesy of Bob, K3NHI. Click for a side-by-side of a xtal filter as measured with Bob's sweep system plus an N2PK VNA.



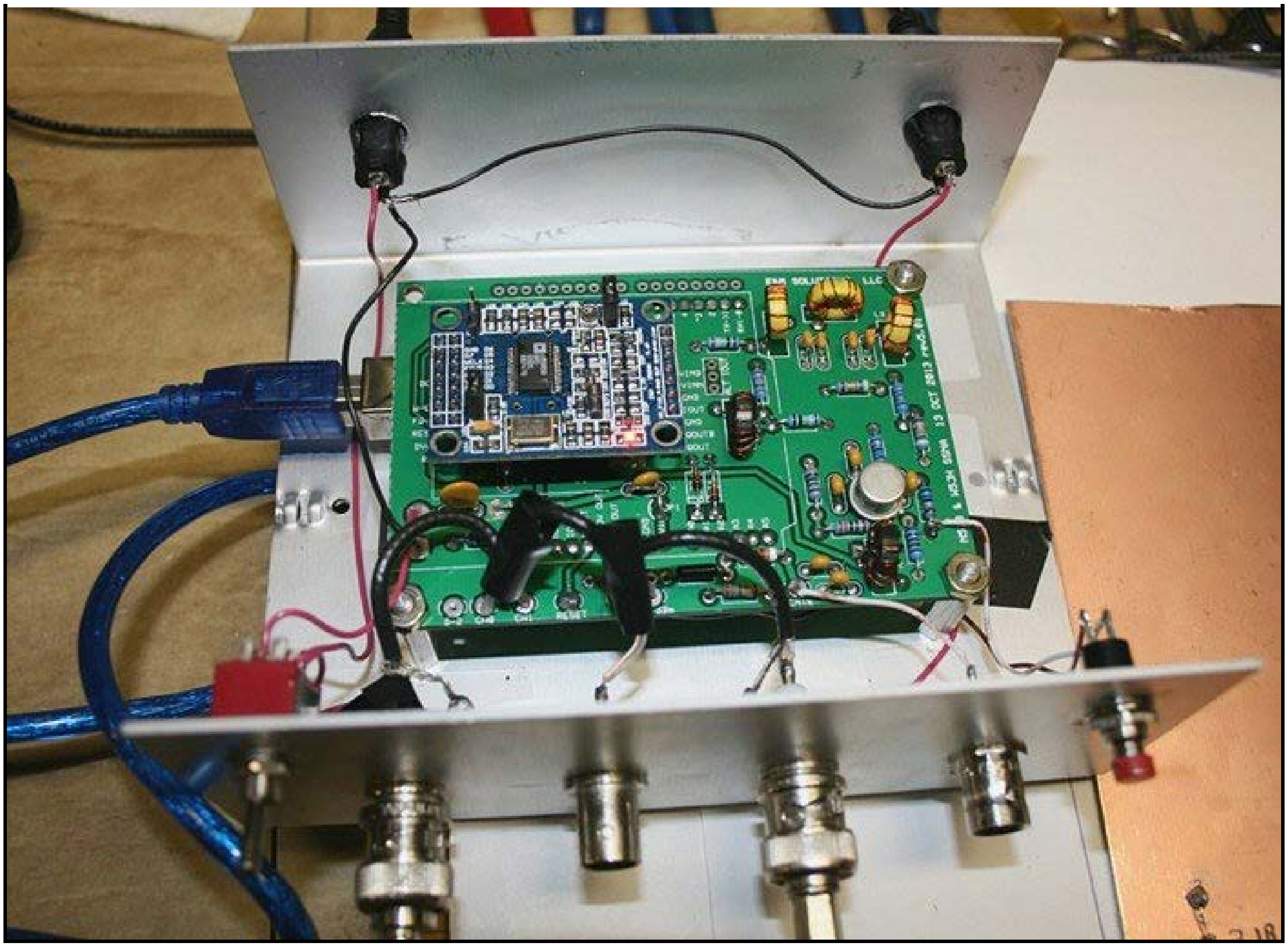
Above — I made and then swept a simple ~500 Hertz wide 4.9152 MHz Cohn or Min-Loss filter using 4 crystals.

So now, if you see sweeps on the site that look like oscilloscope tracings, you'll understand how they were created. I've learned so much from Bob's work and his mentorship last Spring. John, K5IRK coached me also.

QRP-POSDATA for October 2013

3 builders incorporated Bob's sweep system circuitry into projects including a receiver, a spectrum analyzer and the following build of the Utility Sweep Generator (USG) by Jay: Jay built some PC boards to simplify the wiring challenge this USG presents. Click 1 Click 2 Click 3. Great stuff — thanks for sharing.

QRP-POSDATA for March 2014 — Poor Hams Scalar Network Analyzer (PHSNA) —



Above — The PHSNA built by Mikey, WB8ICN.

Jerry W5JH, along with Jim, N5IB and Nick, WA5BDU developed this low-cost sweeper/ lab toolkit.

The Poor Hams Scalar Network Analyzer consists of an Arduino UNO R3, plus an AD9850, or AD9851 DDS, a W7ZOI/W7PUA Power Meter and a MS Windows OS based computer. Builders can scan and plot L-C filters, crystal filters, RF amplifiers and such — much like Bob's sweep system — or an HF tracking generator + spectrum analyzer. The PHSNA also measures crystal parameters with little fuss. Connect a return loss bridge to easily sweep return loss measures of the input or output ports of filters, amplifiers, antennas and more. The total cost to build the PHSNA is approximately $50-60 USD.

Mikey graciously sent me some photographs of his PHSNA build. Complete system in his lab with the chassis lids removed for these photos. Monitor photo showing menu choices. Power meter. Mikey's jig to examine crystals with a 12.5 Ω termination. Once you own a calibrated sweep system, you'll wonder how you ever managed without 1.

Thanks again to Mikey for the photos. I built and tested the return loss bridge using the PCB from the PHSNA Yahoo group. Click for a 613 KB pdf file of my build. Nick, WA5BDU web site.

Hats off to Jerry and crew for this open-system project! A Yahoo group called PHSNA serves as the communications hub and houses superb, detailed documentation. You need to join Yahoo to access this group. Then search for PHNSA and while your at it, also sign onto the EMRFD group.

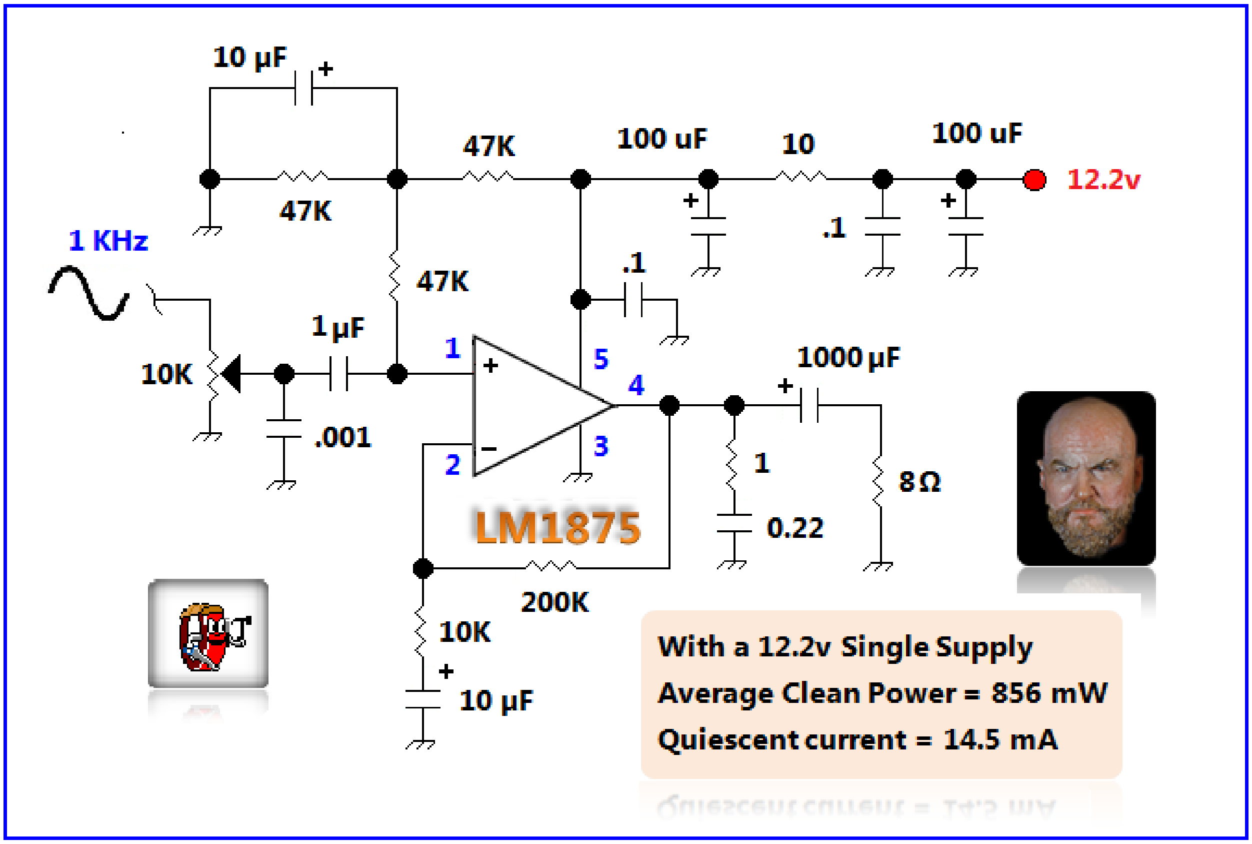
---

2. LM1875 Audio Power Amplifier

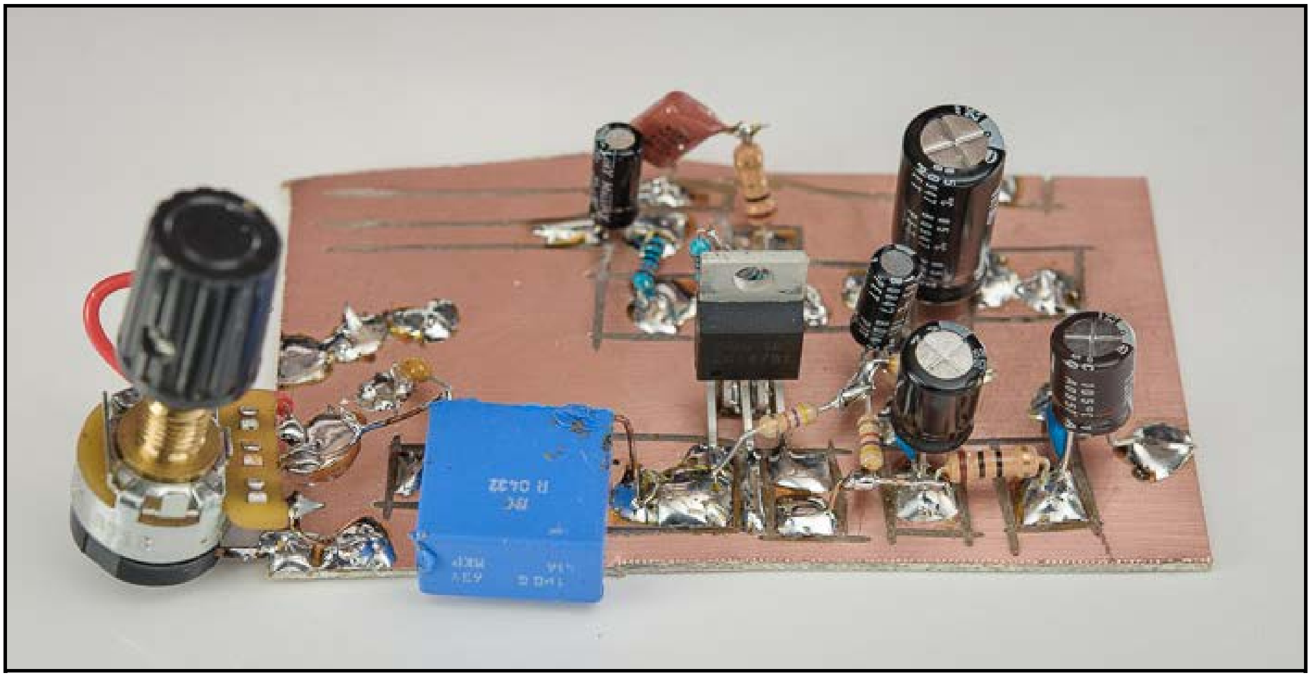
I tested the LM1875 AF power amplifer because its specifications look great: 20 watts into a 4Ω or 8Ω load on ±25V supplies and a TO-220 package for easy heat sinking. Of course, for this web site, I tested it with a typical radio experimenter bench power supply; a single-supply at ~ 12 VDC.

This is probably not a great part for Ugly Construction and I *attempted* to return the load ground, the output Zobel RC filter network, feedback loop and input grounds to a central grounding point through separate paths cut paths into my copper board. A better breadboard method might include the so called "star grounding".

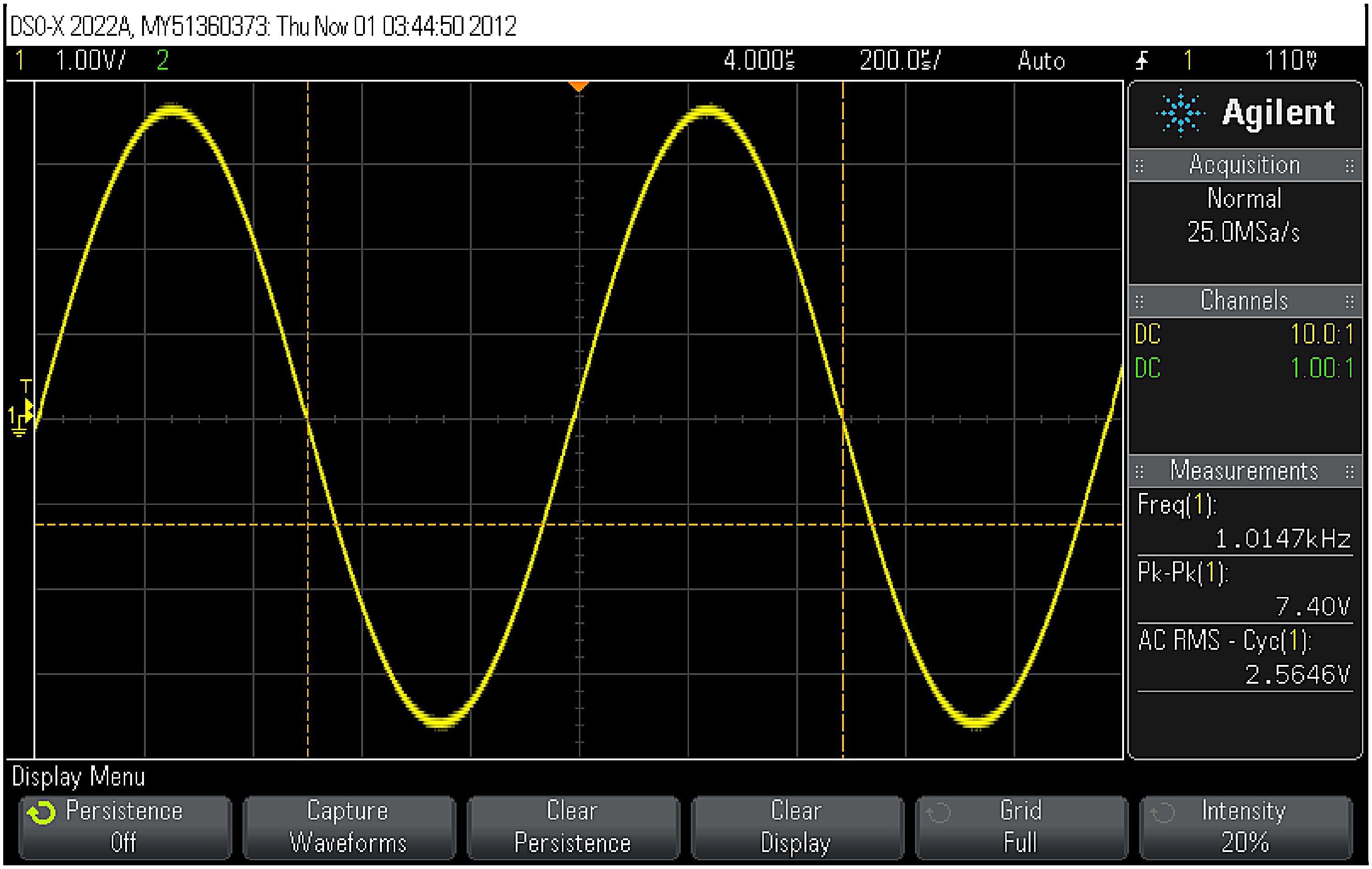
I saw RF oscillations on the 'scope and removed them by soldering a 0.001μF bypass capacitor across the input. A 470 pF bypass capacitor did not work well enough. The datasheet describes specific causes and cures for RF oscillations and I've learned they must be heeded. I once found similar problems with an LM380.



Above — The LM1875 in the ~suggested datasheet, single-supply set up. This amplifer reminds me of setting up an op-amp. Unlike the LM380, within limits, you may choose the gain to suit your needs. As shown the gain = 25.6 dB. Dropping the feedback R to 100K dropped the gain down to 20.5 dB. For clean output power capacity; it blows away the LM380.



Above — My first LM1875 test breadboard. After the photo, I moved the 0.1 µF RF bypass cap right onto lead 5 — we should carefully RF bypass device power supply leads, but I got sloppy.



DSO-X 2022A, MY51360373: Thu Nov 01 03:44:50 2012

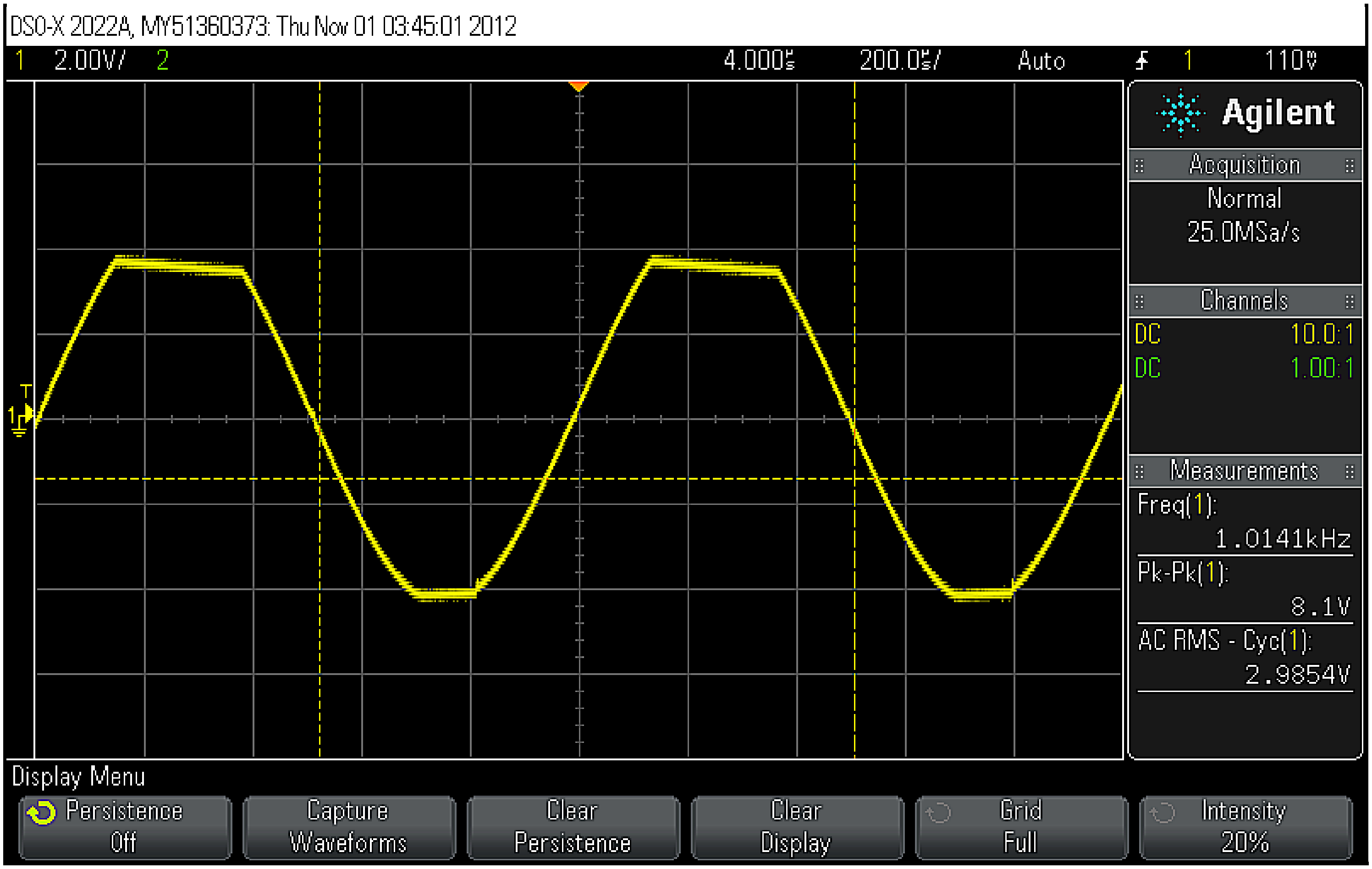
1 1.00V/ 2 4.000µs 200.0µs/ Auto 1 110mV

Acquisition
Normal
25.0MSa/s

Channels
DC 10.0:1
DC 1.00:1

Measurements
Freq(1): 1.0147kHz
Pk-Pk(1): 7.40V
AC RMS - Cyc(1): 2.5646V

Above — At the maximum power before visible sine wave distortion appears; 856 mW. I listened to this amp while connected to a line-level tape player and an 8 Ω speaker load: very nice. I want to try it with a split +/-15 VDC supply and a star grounded breadboard since a 12v single-supply limits the output power so much. Still, at 12 volts single-supply, this IC yielded the highest clean, average output power of any AF Power chip I've tested: a worthy consideration for a high-grade receiver.



DSO-X 2022A, MY51360373: Thu Nov 01 03:45:01 2012

Acquisition
Normal
25.0MSa/s

Channels
DC 10.0:1
DC 1.00:1

Measurements
Freq(1): 1.0141kHz
Pk-Pk(1): 8.1V
AC RMS - Cyc(1): 2.9854V

Above — 'Scope tracing; I advanced the 10K volume pot to drive the amp into clipping.

---

3. Interview — Three Questions with Jason, NT7S from Etherkit

I follow 2 English language amateur radio blogs — 1 is Ripples in the Ether by Jason, NT7S

Possessing a modern flare, Jason, the blogger, gently but resolutely challenges some of the cliquish, dogmatic thought and behavior that tarnishes amateur radio, or just blogs about fun stuff. He writes well — creating an emotional dialog that stimulates thought and reflection. We get a sense that he cares about our radio hobby and wants it to grow and improve.

Jason, the man behind Etherkit, champions a modern, open-source vision that I find both positive and refreshing.

1. Tell me about your decision to embrace the open source software philosophy for your hardware in a time where proprietary code, copyrights and patents still hold strong. How do Ham Radio equipment sellers benefit from code sharing?

I believe that the open sharing of knowledge has always been one of the cornerstones of our amateur radio community, going back to its earliest days. So the open source/open hardware ethos has always resonated with me in regard to our hobby. I started Etherkit with the intention of providing a small bit of income to my family and as a way to promote the idea of open hardware within the ham radio community at large. I have no illusions of becoming the next Elecraft, but I hope that I can build up a stable of affordable and fully-open ham radio kits that will be "hackable" and extensible for the motivated experimenter. I do this by providing the full source code for my microcontroller firmware, all of the PCB design files, Creative Commons licensed documentation, and programming ports for my products. I've already seen some neat examples of customers extending my first product (the OpenBeacon MEPT kit) by doing things such as adding in WWVB time discipline and pairing it up with a Raspberry Pi for cheap automation. I hope that others will take my code or my circuits and re-purpose them in their own work, even if they don't buy my products.

I am not an open source zealot and do not begrudge the large majority of vendors who choose to keep their intellectual property closed. However, most of what us smaller companies do is not on the cutting edge of radio. We leverage the knowledge and works of those who came before us. Perhaps if I created something wholly-new that would be patent worthy, I would consider keeping it closed, but that's not the kind of products that I'm able to develop as a one-man operation. We do not copy the designs of others, but we do take concepts that are for the most part well-tested and come with new ways to implement them. Because of that, it's my personal opinion that I have a duty to keep my designs open.

In the open hardware world at-large, there is a discussion about whether open source hurts your own business prospects. There are still some debatable points in that discussion, but I think it has been shown that if you look at the entire balance, open hardware is a good thing for smaller companies. One of the largest concerns is that under most open source licenses, a competitor can just clone your hardware and undercut your sales. That is a genuine concern, but I think that products such as Arduino have shown that if you make a quality product, most folks will recognize that quality and stick with the original.

Even if others buy a clone of your hardware, in all likelihood, that may be strengthening your brand identity (as long as that vendor isn't stealing your name). Another concern is that a customer can just copy your product for themselves. To that, I say good!. Because of the work and costs involved (economy of scale), it's going to be time and/or money consuming to make that copy. It's probably cheaper and faster to just buy the kit. The reason you copy it for personal use is because you love working with the technology. Which is exactly what I want to encourage. You may lose a small bit of sales, but I think it gives you more name recognition in the end.

2. What’s it like being a vendor at Dayton?

To be clear, I wasn't a vendor at Hamvention in 2012 (hopefully I will be there by 2014), but I was a vendor at Four Days In May at the Fairborn Holiday Inn. It was a wonderful experience to get to sit with the big names in the QRP world, selling my wares. I got the opportunity to meet tons of QRPers and build up some good relationships. Online sales are wonderful for the ultra-small operations such as myself, but nothing beats actually meeting your customers face-to-face, especially when you are at the world's most well-known QRP convention.

3. In industry, SMT parts are normal and hole-thru might better be called “hold-over”; what’s your view on kitting products with SMT parts?

We've seen some SMT kits within the QRP world, but they still are more of an oddity than anything else. I understand the concerns that people have with SMT assembly, but I think that there is still a lot of trepidation that builders needn't have. It's my opinion that SMT construction with "larger" components such 1206 or 0805 is well within the capabilities of the average kit builder. I also believe that once you are comfortable with SMT construction, it is probably faster and more efficient than through-hole construction.

OpenBeacon is a through-hole product, but I have had a QRP CW rig in development for the last two years that is a SMT design. In beta testing, I've found that one of the biggest challenges in kitting is that I have to clearly identify each and every component. With a through-hole kit, you can just throw all of the resistors or all of the capacitors together because they are clearly marked. Not so with SMT. You have to have a system to keep each value separated from the others and marked with a value. SMT resistors and semiconductors have a laser-etched value, but it's nearly impossible to see by naked-eye, and SMT capacitors generally have no markings at all.

So I have had to compartmentalize each strip of components of the same value cut from a reel, and mark them with a sticker. That is pretty costly and time-consuming. I'm hoping to find ways to streamline this process so that I can release SMT kits without the large time investment that it currently takes.

---

4. EMRFD Experiments — A 1-on-1 Tracking Phase-locked Loop

I built the 1-on-1, or offset phase-locked loop circuitry described on EMRFD page 4.22 and share these schematics in faith you'll create your own. Rich with wisdom and reason, this section lies among the best topics from EMRFD. Please read Wes' notes since I won't repeat his narrative — only supply a few ideas and measurements. In the article closer, Wes suggests some modern parts to raise performance and I applied all of them with the exception of the 14 MHz VCO.

Rather than building the main VCO with divide by N circuitry to allow multiband use, I copied the original 14 MHz oscillator verbatim. Why? Well, I wanted to test this VCO: a design that wisely doesn't expose the varactor to high impedance or signal amplitude and thus avoids forward-biasing the single tuning diode. I've discussed this before on the QRP Modules 2011 web page under **7 MHz VCO Experiments**. Also, I really just wanted to learn about PLL circuitry. The Figure 1 macro schematic below illustrates this project.

In my circuit, a frequency stable 14 MHz VCO = **the goal**; the rest of the circuitry supports this.

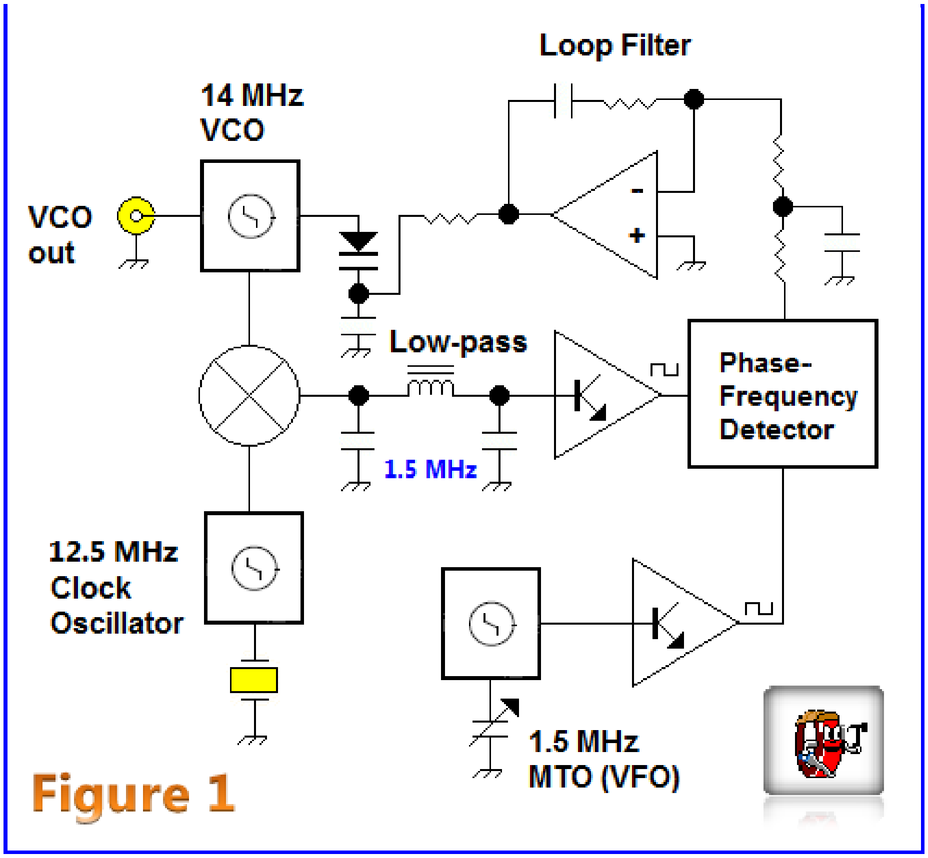
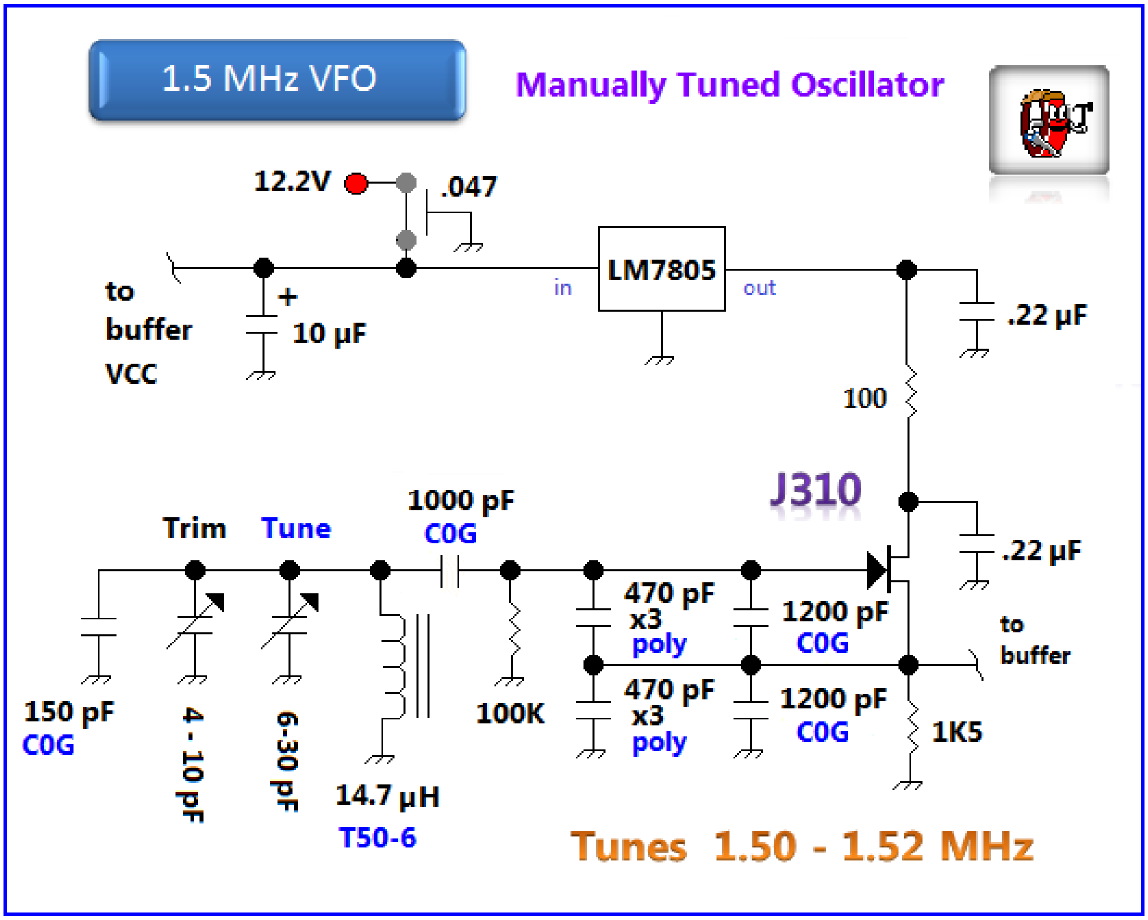


Figure 1

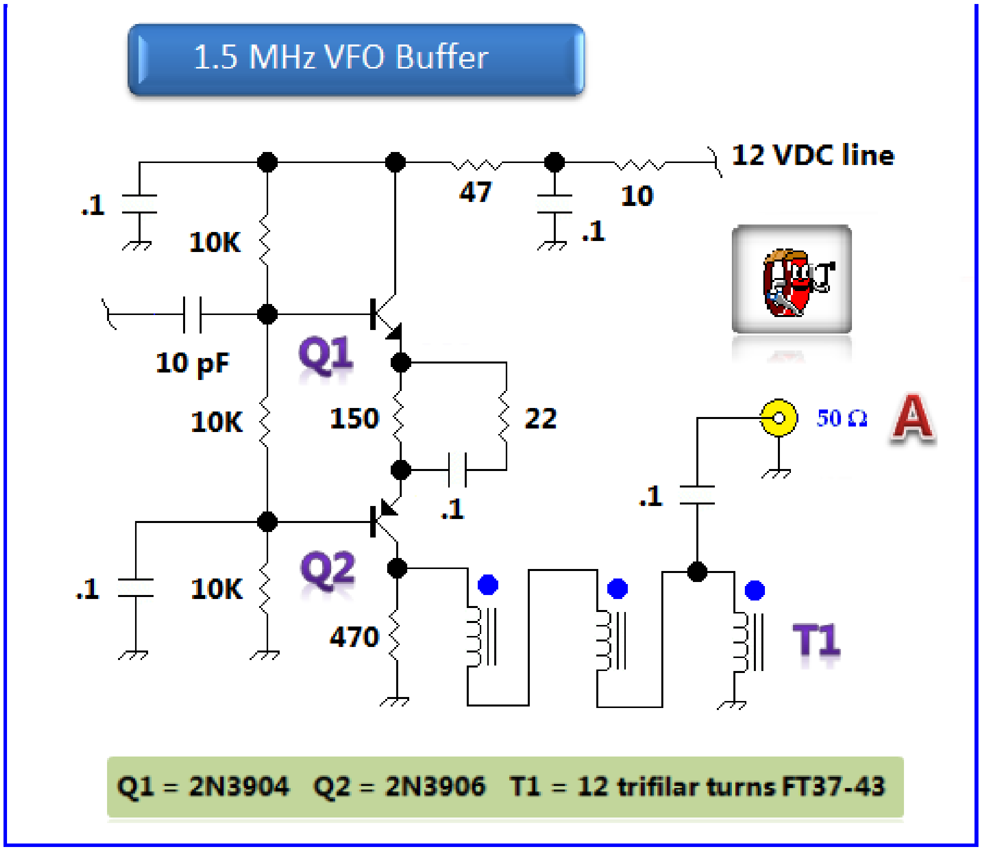


Above — The 1.5 MHz VFO. In his modern writing, Wes calls this the MTO, or Manually Tuned Oscillator *in the context of a tracking PLL.* I wound the L with # 30 AWG wire on a T50-6 toroid.

My MTO exhibits a low tuning range (only 1.50 to 1.52 MHz) since I built in a box with a small air-variable capacitor that swung only 24 pF and I ran the "Colpitts capacitors" at 2610 pF to keep phase noise low. This box normally holds a VHF oscillator and I just removed the main board and swapped in a 1.5 MHz equipped copper board. I won't keep this PLL and thus sticking the 1.5 MHz MTO in an existing oscillator chassis with a grounded tuning shaft and feedthrough capacitor helped save money and time.

Above — The built 1.5 MHz MTO. With temperature compensation from 6 stiff-leaded, 600 VDC, 470 pF polystyrene capacitors, my frequency drift measured between 3 and 4 hertz per hour upward at room temperature. Properly designed + built + temperature compensated L-C oscillators at 1.5 to 3.5 MHz may exhibit stellar temperature stability. See the VFO - 2011 web page for some tips.

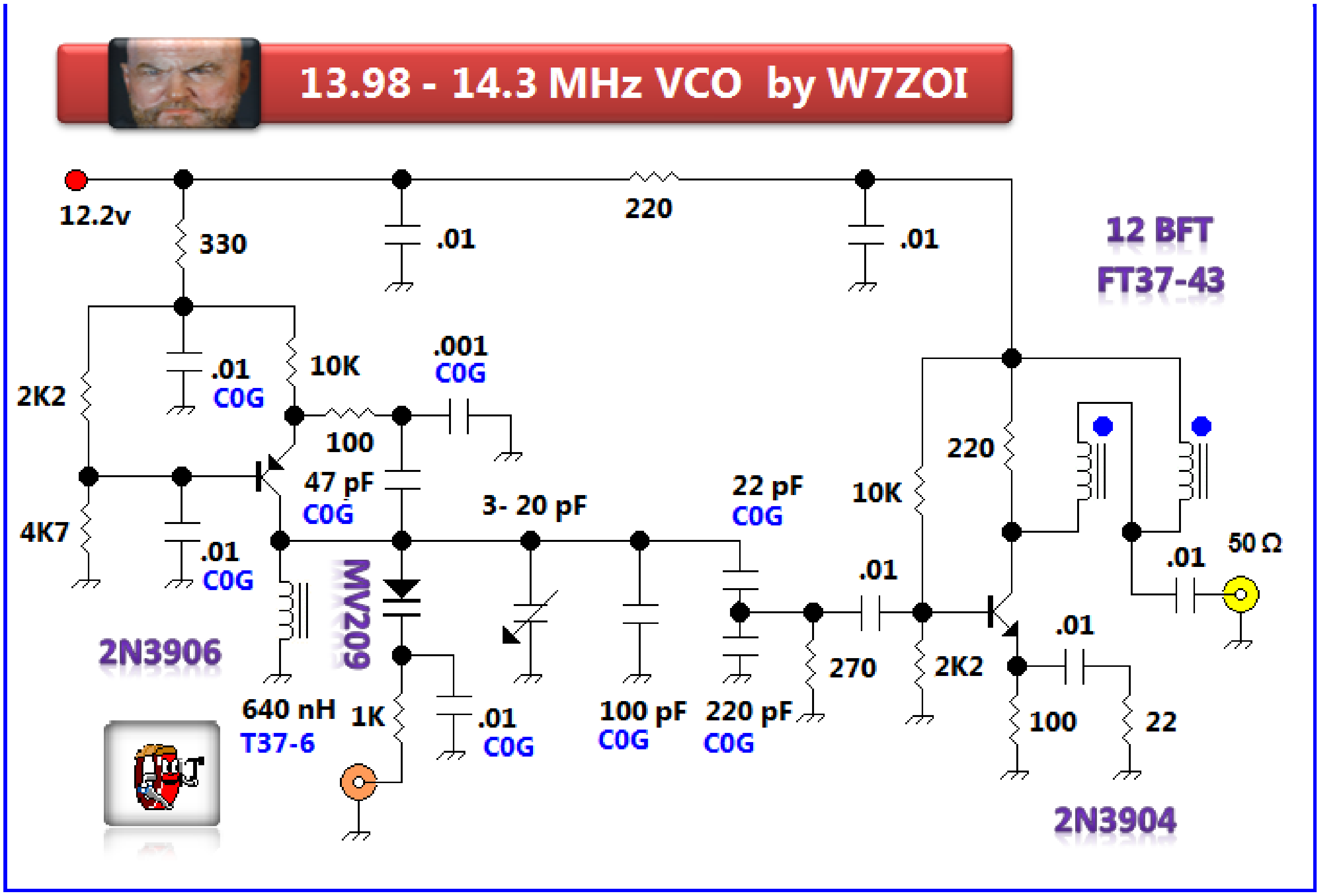
Since this VFO was sublimely frequency stable, I didn't possess the guts to change up the L-C ratio to garner a wider tuning range from the small air-variable tuning capacitor. A 100 pF, or greater delta-F air-variable tuning capacitor would stretch the VFO (MTO) tuning range nicely.



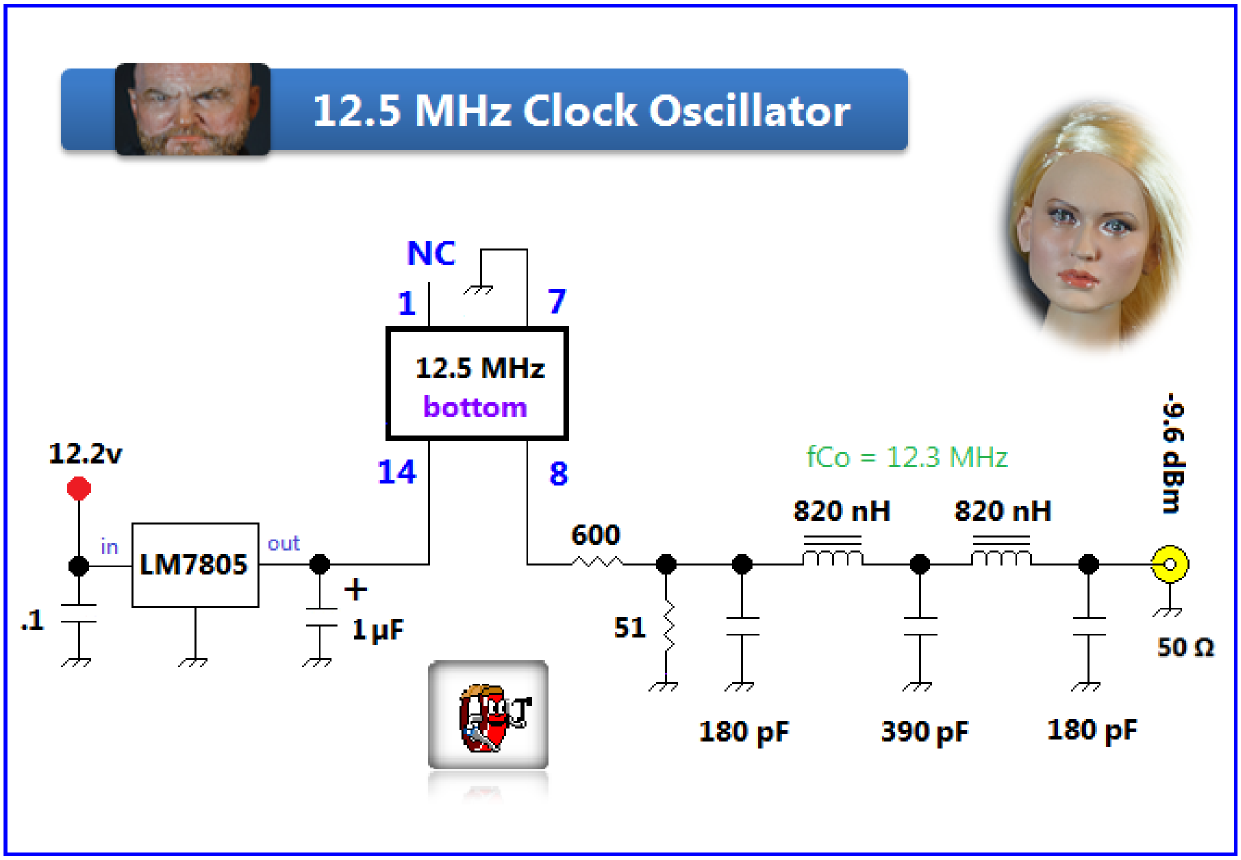
Above — I designed this buffer last year and it's my new favorite. Click for the original. A 10 pF C0G/NP0 capacitor lightly couples the MTO output to the high impedance of Q1, an emitter follower. Further, a common base amp provides gain and *essential* reverse isolation. You may adjust Q2 gain by changing the degenerative feedback offered by the 22 Ω resistor and 0.1 μF capacitor. For example, decreasing the R to 18 Ω may provide 7 dBm output for a diode ring mixer.

MTO output power = 6.71 dBm.

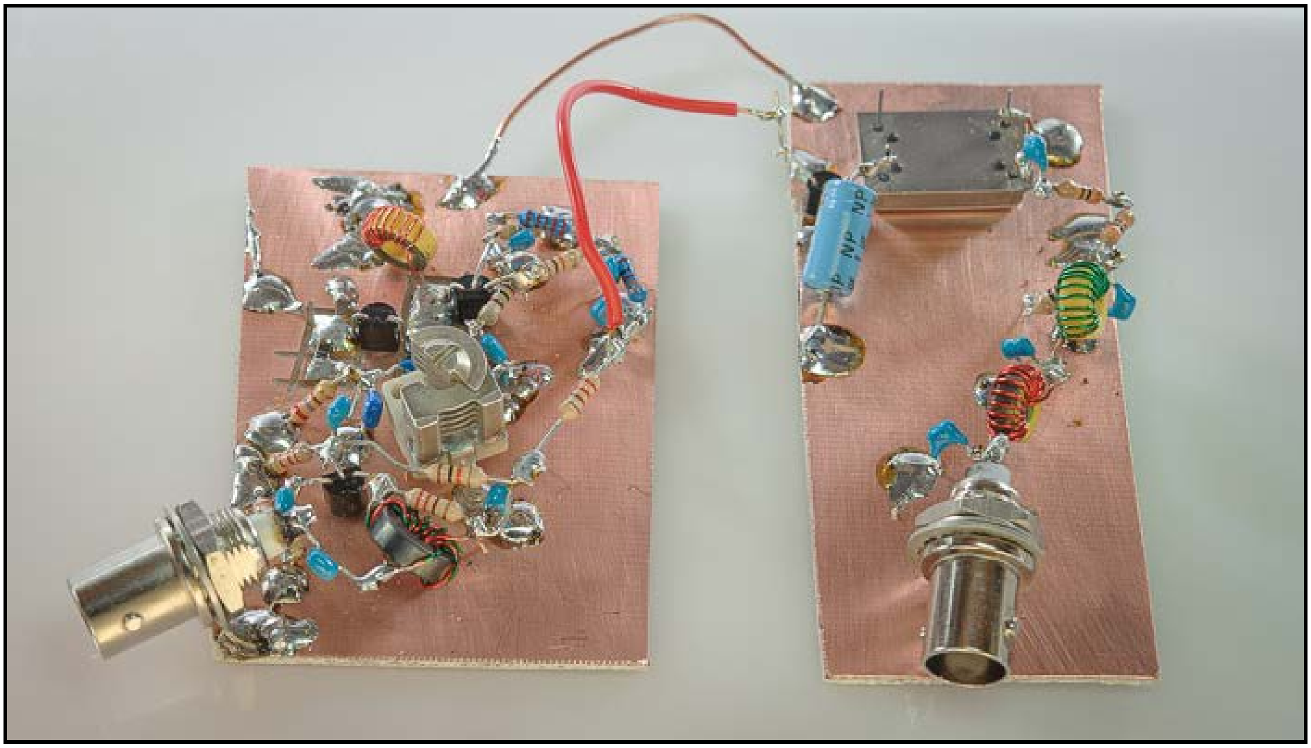
I transformed the 470 Ω collector resistor impedance to 50 Ω with a transmission line transformer. Even though part of the PLL circuitry involves logic gates, or is at DC; as possible, my circuits employ a 50 Ω input or output impedance to allow measurement with my 50 Ω modules and/or instruments, plus transmission via 50 Ω cables.



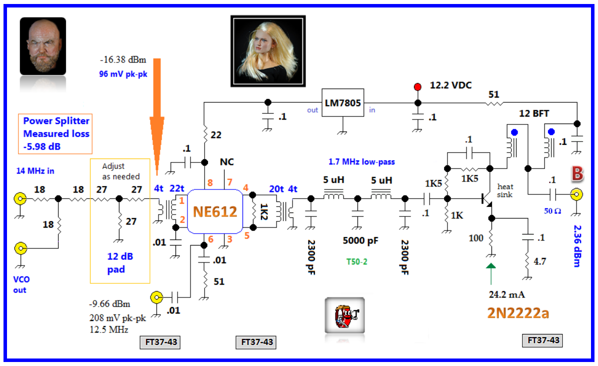
Above — The 13.98 to 14.3 MHz VCO by Wes, W7ZOI (Figure 4.43 in EMRFD). The connector in series with the 1K varactor resistor was an RCA type. Output power = 1.62 dBm. I employed a 3 - 20 pF air variable for the trimmer.



Above — In the original circuit, Wes built his 12.5 MHz crystal oscillator with a single 2N3904. Lacking a 12.5 MHz crystal, I built my xtal oscillator from an old, junkbox 12.5 MHz clock oscillator. A resistor L- matching network drove a low-pass filter to scrub off harmonics, Click for the clean output 'scope tracing at 211 mV pk-pk in my first version. Later, some tweaks gave a final power of -9.6 dBm (208 mV pk-pk). Many authors switch their NE612 mixers with a peak-peak voltage of ~200-300 mV. An AC-coupled 51 Ω resistor on the NE612 pin 6 properly terminates the oscillator to establish the desired drive power and filtering.



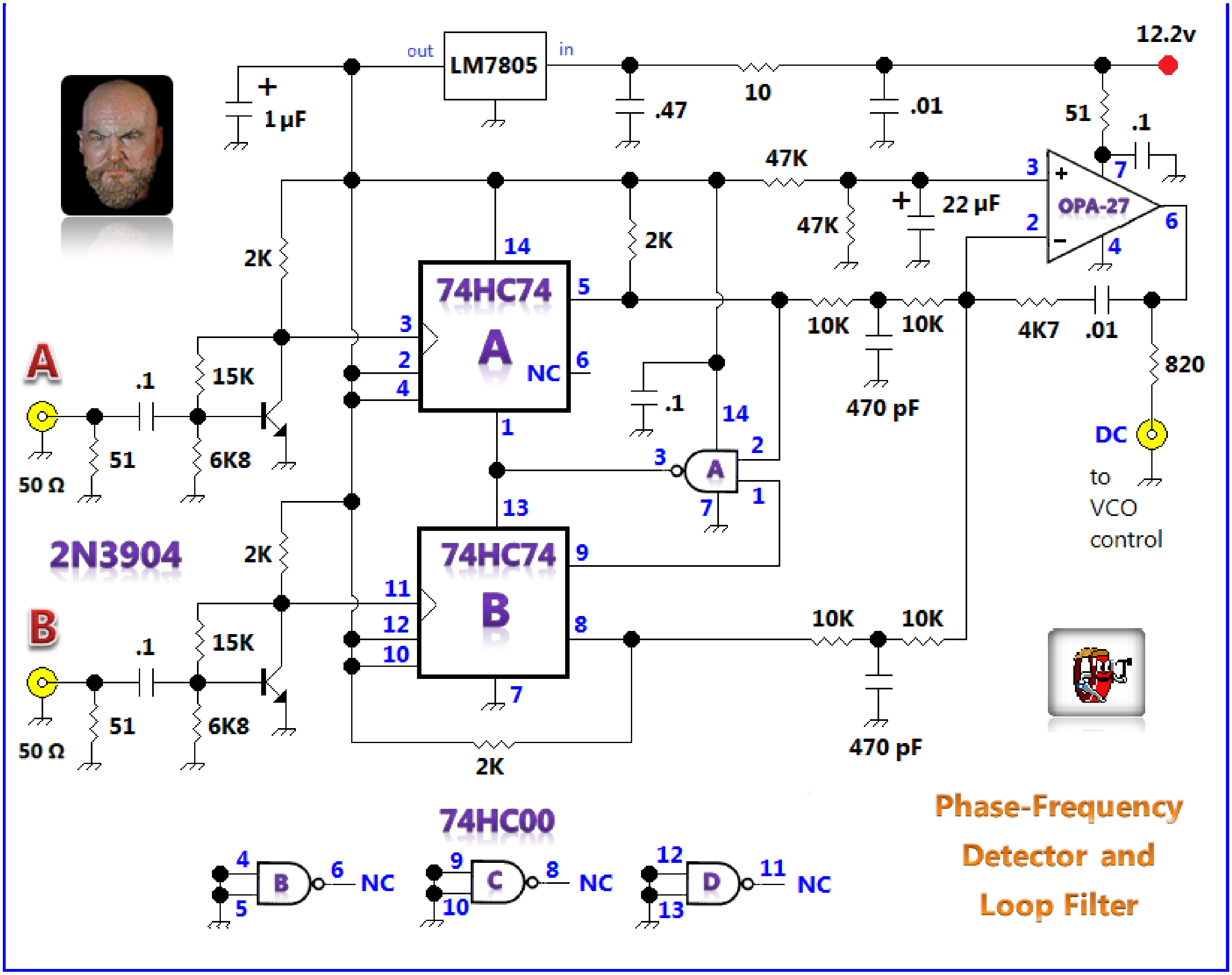
Above — Oscillator breadboards: 14 MHz VCO (left) and the 12.5 MHz clock oscillator (right).



Above — Click to view the power splitter, mixer, low-pass filter and amplifier schematic. Click for an FFT of the clean 1.5 MHz output sine wave. At this point, the 14 MHz VCO has no DC voltage connected to its frequency compensation varactor. As shown, the mixer products are seriously attenuated by the simple, low-pass filter + keeping the mixer RF port signal amplitude low. The power splitter provides the input for the mixer and also the main output for the 14 MHz VCO. The main VCO output requires 50 Ω buffer/amplifer(s) to drive a receiver mixer, transmitter chain, or whatever.

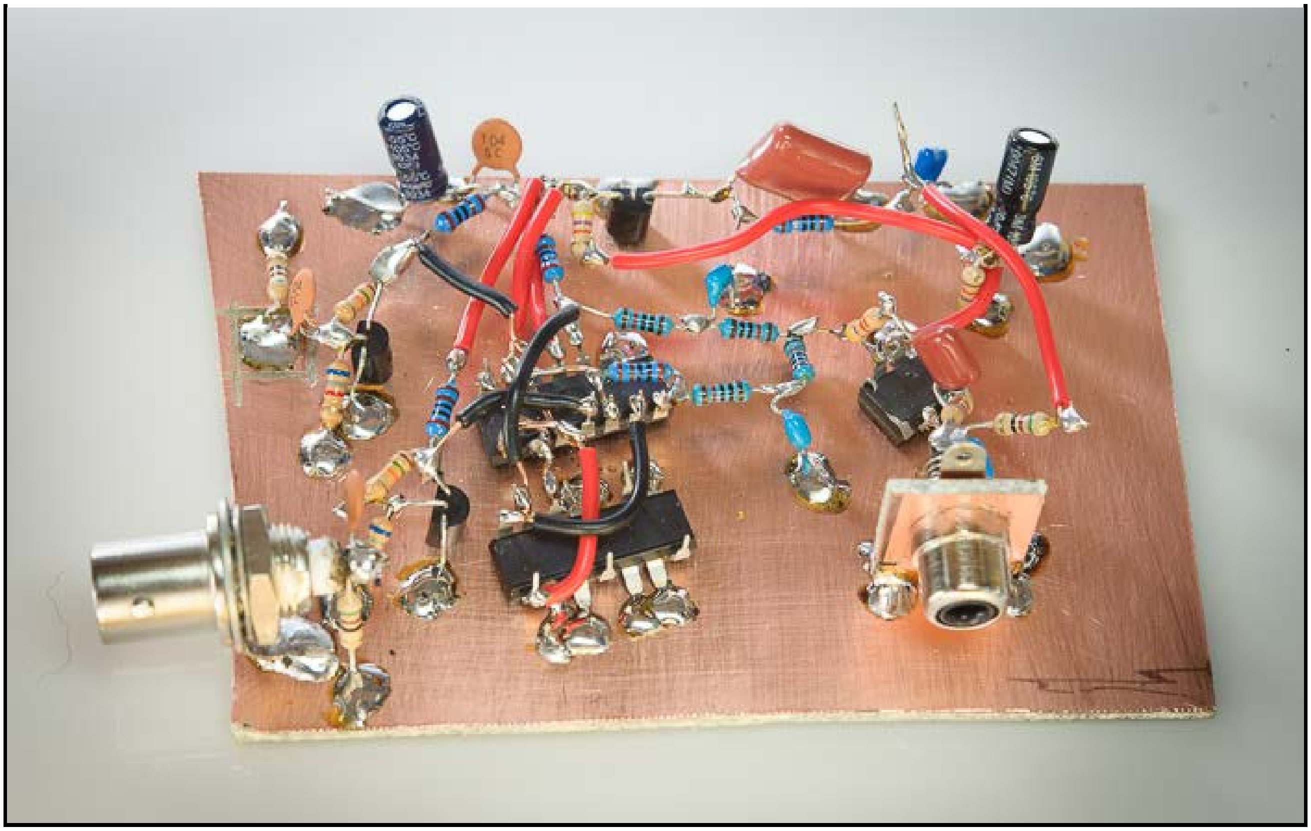
I inserted the 12 dB attenuator pad to keep my mixer RF port signal low to drop the mixer products amplitude down; further losses occur in the transformer. You can change this pad to whatever is required. I belong to the camp of builders who drive their transmit mixers with low-level RF signals to avoid messy outputs at the IF port.

Click for a breadboard photo of my initial bench tests with the mixer board. A 50 Ω resistive terminator shunts the main VCO output port during this testing. I temporarily insert BNC connectors along my development breadboards to measure output signals with my 50 Ω terminated 'scope, spectrum analyzer, or power meters and *rarely* measure RF circuits with a 10X probe.



Above — Phase-frequency detector and loop filter schematic. 51 Ω resistors terminate each 1.5 MHz input and drive two 2N3904 switches per EMRFD Figure 4.41. The loop filter design from EMRFD works as described, however, if you make a loop filter for a different circuit, casual copying goes out the window. Engineers design their loop filters according to factors including the crossover frequency, VCO gain, the N-division for the loop, etcetera *with software.* Some people and companies offer such software on the Web.

My loop filter 0.01 µF cap was a 1% polyester capacitor, although Wes specs a 10% tolerance in EMRFD. No cheapo ceramic bypass caps here please.



Above — Phase-frequency detector and loop filter breadboard. Click for a photo of the scattered, ugly, working boards on my workbench. Many prototypes look like this on our benches, however, sometimes, they work perfectly until we stick them in a box! Do you relate? Each oscillator belongs in its own metal box with strong bypass and decoupling networks (feedthrough caps reign supreme here) since the 3 oscillators might decide to party together and create havoc.

Above — My VCO frequency with the 1.5 MHz MTO set at full mesh. Since my MTO only tuned from 1.50 to 1.52 MHz, my VCO only tuned from

14.00 to 14.020 MHz, but that's easily fixed as I've stated. I'm very happy — it locked perfectly and my 14 MHz VCO stayed on frequency at the exact previously measured frequency drift of the 1.5 MHz MTO. When I connected the 14 MHz VCO to my counter without the PLL circuitry, it drifted willy-nilly.

Conclusion

The sense of awe and joy arising from locking a VCO on frequency won't be understood by many. The concepts and circuitry offers many possibilities. If the MTO and VCO exhibit low phase noise, short-term oscillator stability may be fantastic.

The 14 MHz VCO could be a 56 MHz VCO with sequential division by flip-flops to provide output at 28, 14 and 7 MHz with the 14 MHz portion going to the offset mixer. In EMRFD, Figure 4.44, Wes offers 14 and 7 MHz output by dividing the 14 MHz signal from the main power divider output port. The 7 MHz band is low-pass filtered to remove harmonic energy.

Wes extended this circuit by dividing the MTO by a hardware programmable frequency divider so that the difference from the mixer and low-pass filter is 170 kHz nominal. He uses this 'Almost Synthesizer' on the air for his QRP adventures.

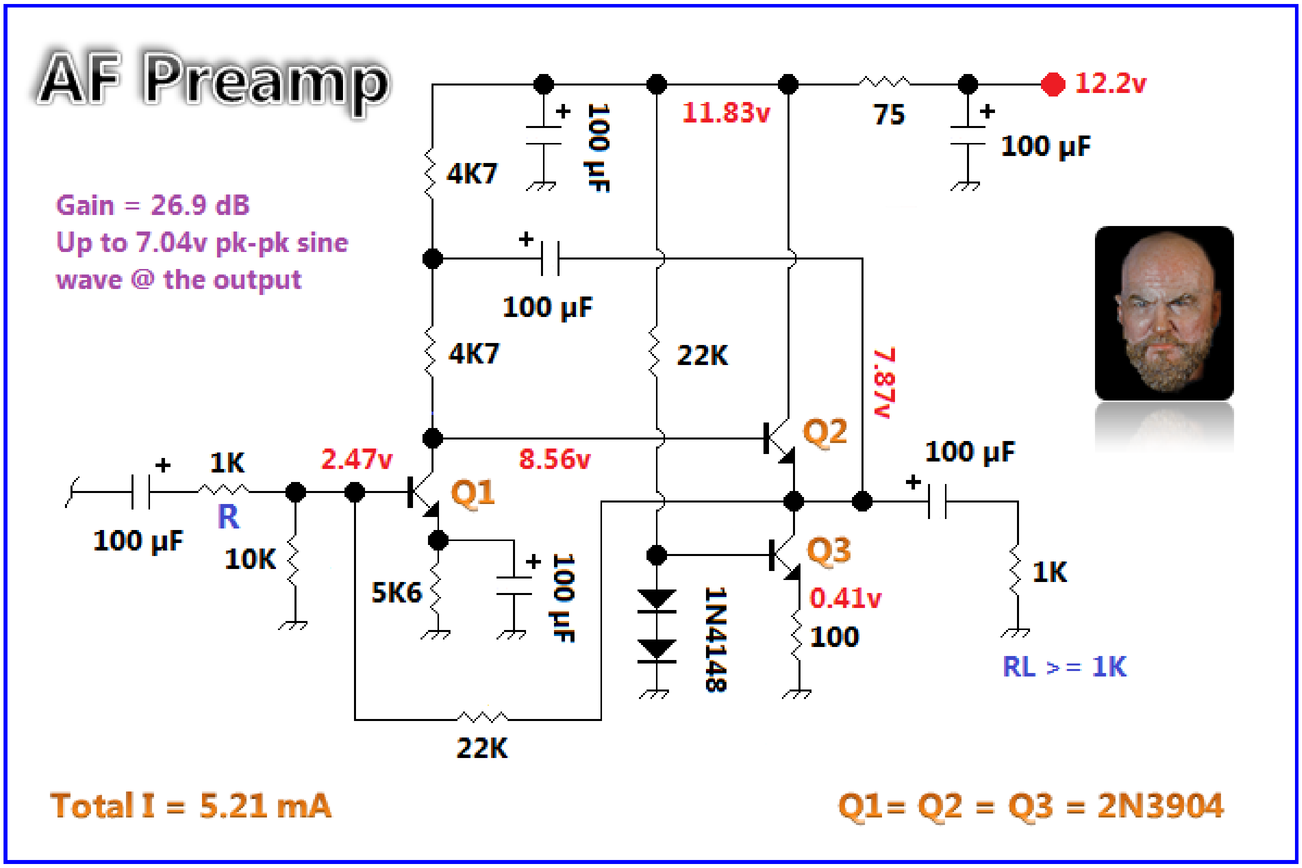
While most builders will sensibly jump from an L-C VFO to a kit containing a programmed microcontroller plus a DDS or Si579, it's also fun to play with hardware to learn and ingrain synthesizer concepts + gain bench wisdom.

QRP — PosData for April 17, 2013

For a good read on the offset PLL, consider studying Wes' book **Introduction to Radio Frequency Design**, ARRL, 1994, page 320 and on. *This book is now out of print*. Wes ported the PLL active loop filter design program he wrote for IRFD from DOS to Windows in April 2013. Click and scroll for it. Thanks for this Wes!

---

5. Boot-strapped Popcorn AF Feedback Pair



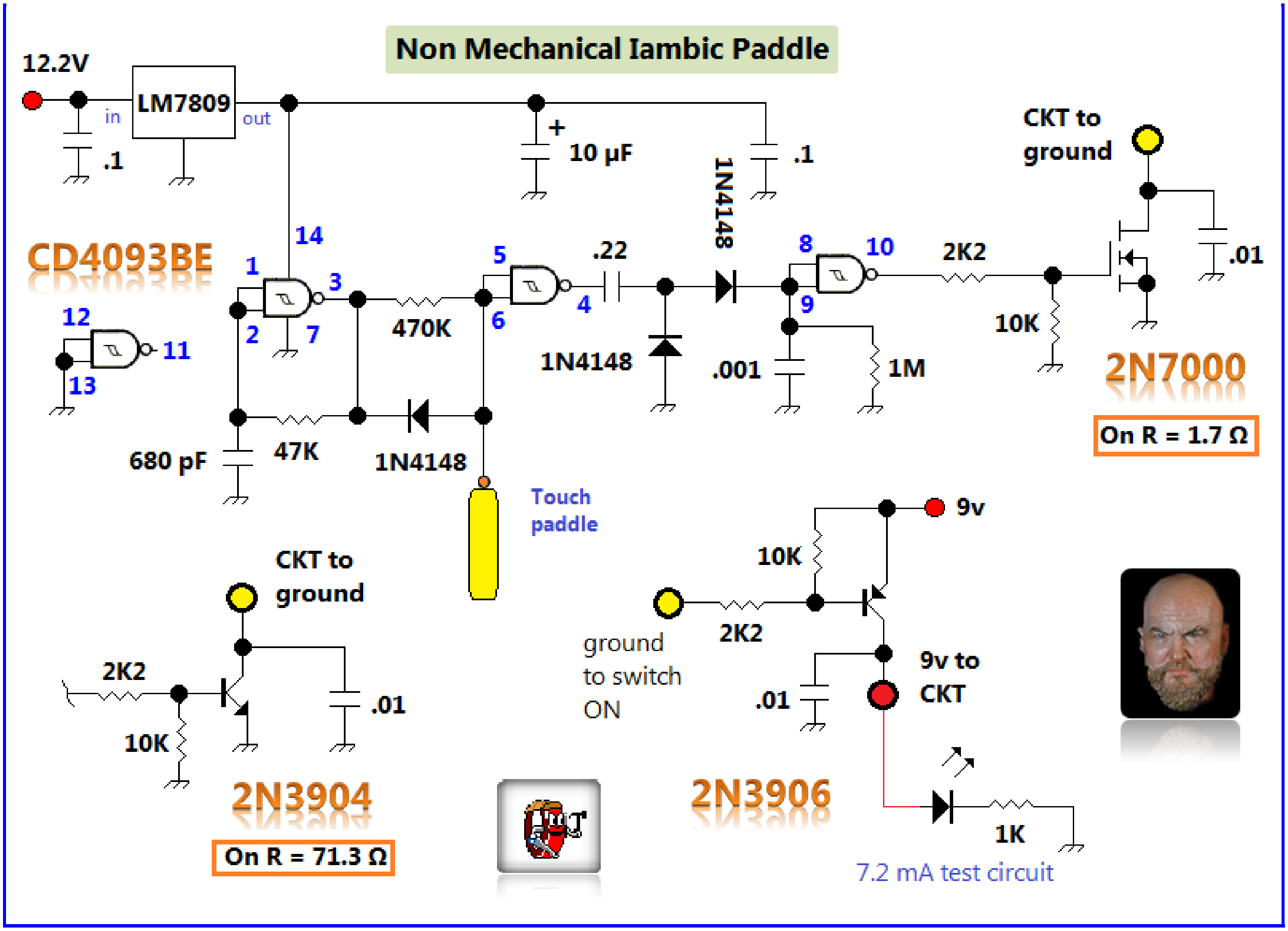
Above — I designed this AF stage for a builder from Indonesia; a popcorn AF shunt feedback amp based on the work of Douglass Self. Despite only drawing ~ 5 mA, this amp stayed clean until the output voltage exceeded 7.04 volts peak-peak on my test bench. Boot-strapping increases gain and lowers distortion in Q1.

Q2 buffers the Q1 voltage amp from external loading and increases gain.The Q3 current source boosts the load-handling capacity of the Q2 emitter follower. The input R can be raised to reduce sensitivity. The 1K output R could be a 5-10 K volume pot.

---

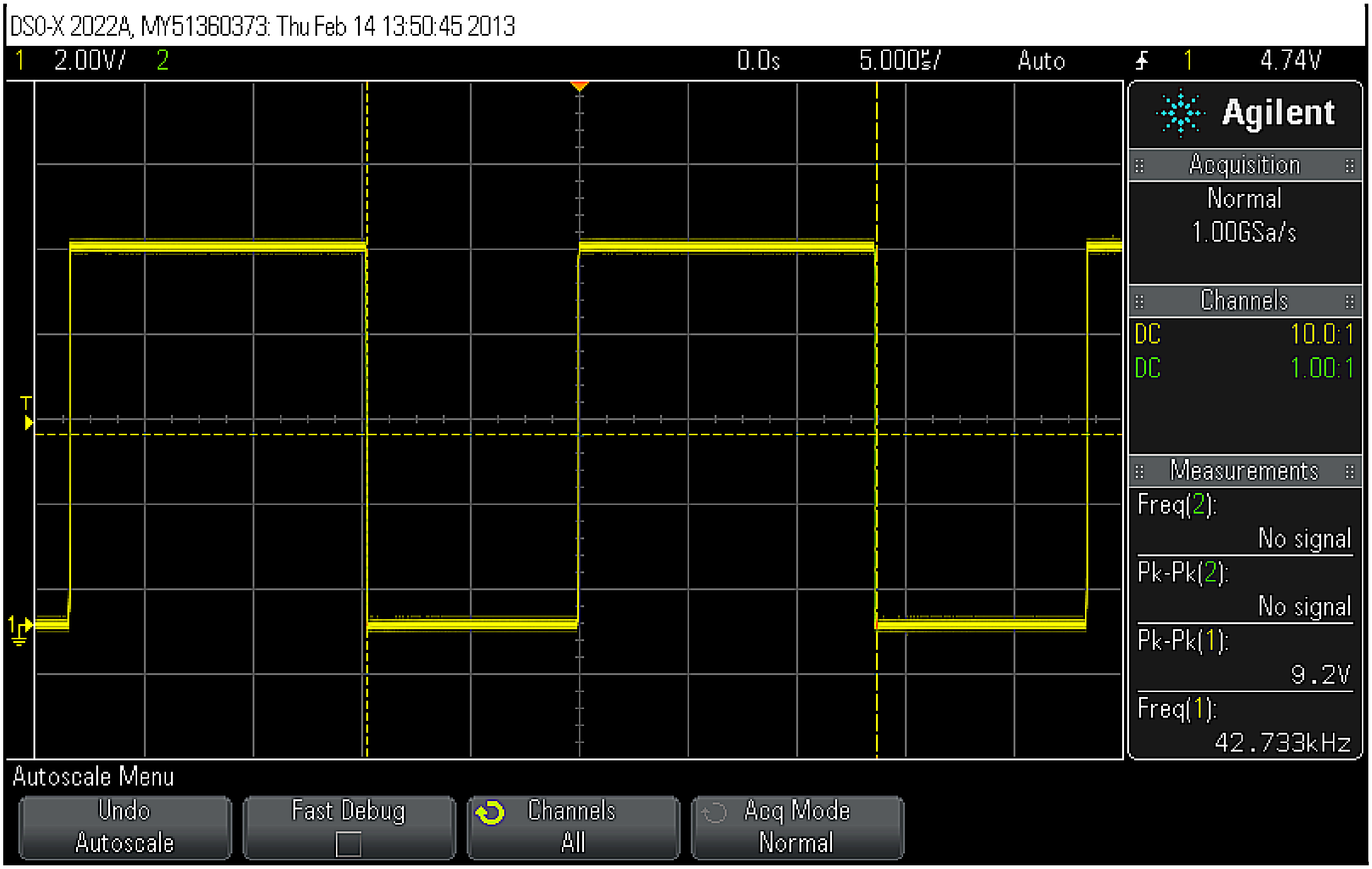
6. Non-Mechanical Iambic Paddle

Above — The very ugly development proto-board of my half-done non-mechanical Iambic paddle. At some point I'll build the other half (the dah paddle switch) and press it into service. You might also use this circuit as a non-mechanical straight key.



Above — The schematic for 1 of the paddle circuits. I compared the ON resistance of the BJT switch with the enhancement mode FET and the 2N7000 won: only the FET could key the continuity tester on my DVM.

You may extend this circuit with a 2N3906 switch for paddle-switched 9 volts (or whatever VCC you want). In the bottom right, I connected the PNP collector to an LED and flashed it for fun. The 0.01 µF capacitor on the switch drain or collector bypasses any RF to ground. With higher power RF, you may have to place a similar bypass cap in parallel with the shunt 10K resistor on the 2N3904/2N7000 base and gate respectively.



DSO-X 2022A, MY51360373: Thu Feb 14 13:50:45 2013

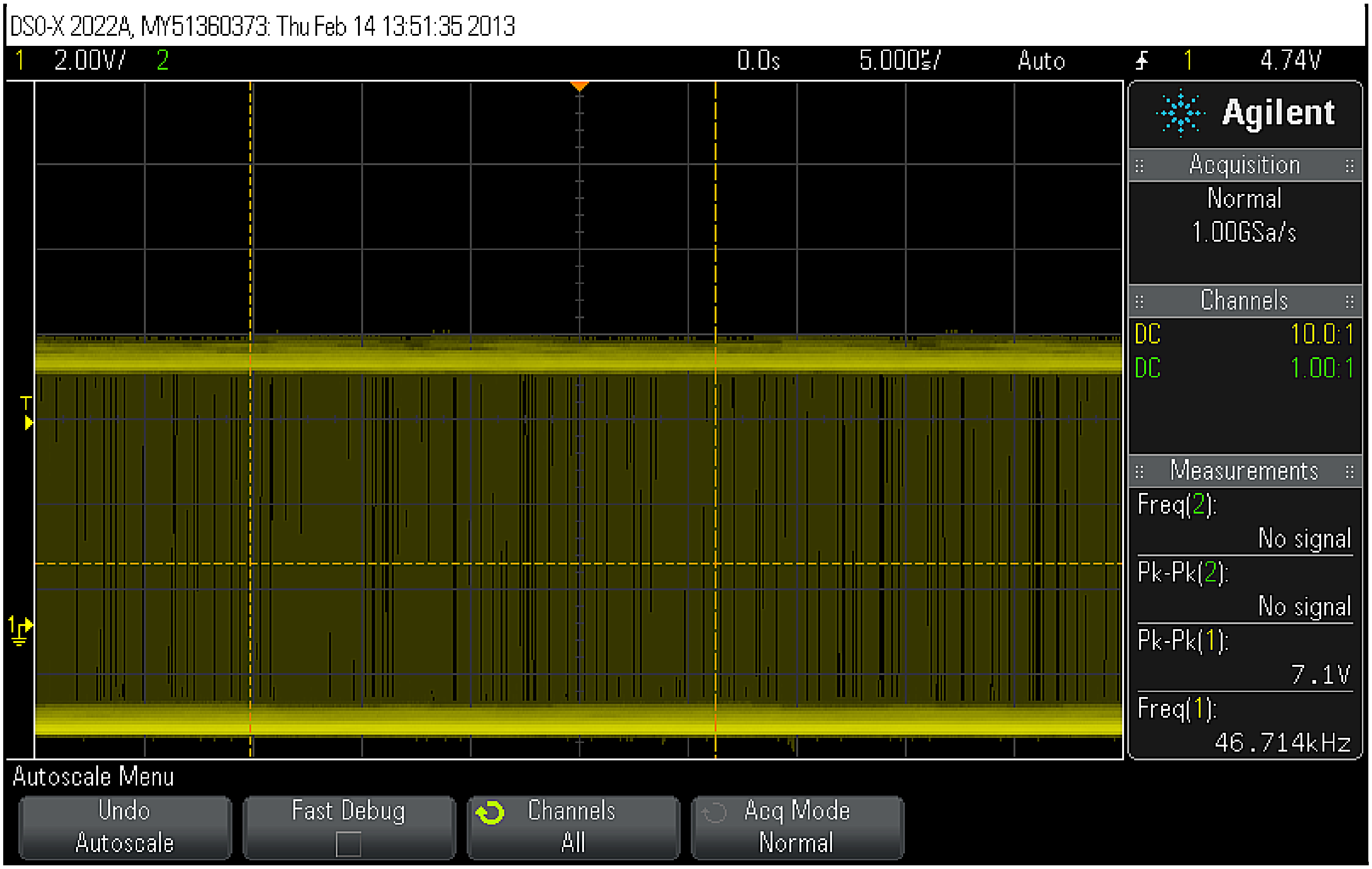
1 2.00V/ 2 0.0s 5.000µs/ Auto 1 4.74V

Acquisition
Normal
1.00GSa/s

Channels
DC 10.0:1
DC 1.00:1

Measurements
Freq(2): No signal
Pk-Pk(2): No signal
Pk-Pk(1): 9.2V
Freq(1): 42.733kHz

Above — A 'scope shot of the ~ 43 KHz oscillator generated in the first 4093 Schmitt trigger NAND gate.



Normal
1.00GSa/s

DC 10.0:1
DC 1.00:1

Freq(2): No signal
Pk-Pk(2): No signal
Pk-Pk(1): 7.1V
Freq(1): 46.714kHz

Above — Here's the disturbed oscillator output *just* after the paddle is touched: this stops the signal at pin 5 and 6 of gate 2 and kills the AC output at pin 4. The DC voltage across the 0.001 μF cap discharges through the 1 megohm resistor pushing pin 10 HIGH to turn on the 2N7000 (or the 2N3904). Normally, pin 10 is LOW since the rectified output of the undisturbed ~43 KHz oscillator goes to both pin 8 and 9.

A fun circuit for a Saturday afternoon...

---

7. A Journey Above HF

This project began as a 14 MHz low-noise amplifier build, but ended up with me learning more about SMT breadboard techniques and suppressing spurs. A short exploratory/descriptive account of my bench journey plus some photos follow.

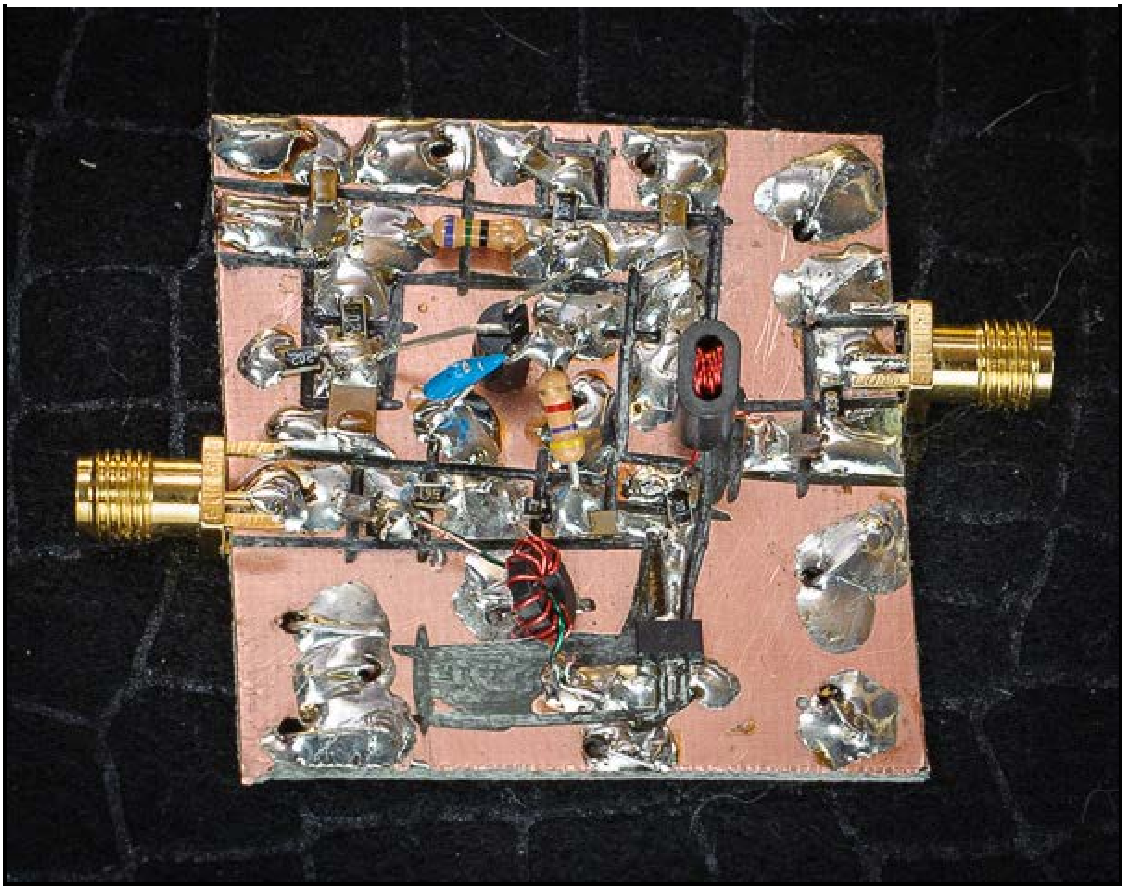
Above — I'm slowly adding SMA connectors and pieces. Since modern consumer digital network engineers use them, they're abundant and often rated from DC to 18 GHz; more bandwidth than I'll ever need.

I'm also building with evermore SMT components and just love it. Through-hole (I prefer to say hole-through) stuff continues to disappear like lemonade on a hot August afternoon.

Above — The schematic of my version of Victor 4Z4ME's feedback amp (FBA) as tested at 14 MHz: he emailed me a paper and provided some online support. Click for another version from December 2012. Typical noiseless FBAs suffer from poor reverse isolation, however, Victor runs the collector to base feedback through an asymmetrical 3 dB power combiner/splitter that boosts port isolation, defines the gain, plus sets the input and output impedance.

Strong virtues of the asymmetrical power splitter — fully utilized in this design include a very low loss on the input side (the 1 turn side) and a much higher loss ont the feedback side that allows the feedback to defines the input impedance on one side while exerting a negligible impact on noise figure and dynamic range on the other side. Victor measured a noise figure of 1.5 dB using a MRF586 BJT. Thanks to Victor for the information and design.

For strong IMD properties, I ran 50.6 mA total stage current into a gorgeous, low-noise, NE46134 NPN transistor with a fT of 5.5 GHz. Using VHF-UHF techniques, I built with mostly SMT parts on 2–sided board using copper wire vias to connect the 2 copper surfaces. I discussed the wrap-around bias technique in 2011 as number 1.



Above — My prototype breadboard with dremel cut islands for soldering the size 1206 or 0805 SMT parts, plus a few hole-through items. Carving an island for the SOT89 transistor package proved difficult, but even I (a challenged dremelist) did it.

Woe to Oscillations: Like misplaced car keys, oscillations may remain hidden unless you search for them. Often, the only difference between a proper oscillator and a regular amplifier is we *want* the former to oscillate. To check for instability, we might use our high bandwidth scope, or a spectrum analyzer, but many will have to find spurious RF with basic, DC - HF bandwidth test equipment. In any case, just do your best. To some extent, unwanted oscillations are the *elephant in the room* that few talk about. Well, it's okay to think, talk and feel some emotions about them.

Sure enough, when I connected a 14 MHz signal to the amp's input and a 50 Ω terminated 'scope to the output to measure gain, my sine wave had 2 or 3 others on top of it. In the 4Z4ME amp, the PNP bias transistor can be a source of AF to HF oscillations.

Victor wrote: "The circuit has a low frequency amplifying loop that goes through both transistors. The PNP transistor does not invert the signal (it is a common base amplifier) and the RF transistor inverts so it is a loop with 180 degrees phase shift (negative feedback). The various decoupling and RF coupling capacitors in this loop add phase shift on this low frequency loop. If the accumulated phase shift adds to an additional 180 degrees and gain is larger than 1 you have oscillations. The simplest way to solve it is to make one of the capacitors very large so it will add only 90 degrees phase shift but it will drop the gain at the higher frequencies where the other capacitors start to add phase shift to be less than 1 so there are no conditions for low frequency oscillations. This technique is called "Dominant Pole". That's the reason that I suggested to connect a very large capacitor to the PNP transistor".

I found my oscillations disappeared with a 0.1 μF collector bypass cap on the PNP (Cx on my schematic). The 0.1 μF cap on the PNP collector was critical – a 0.22 μF failed to work, as did a .001 μF --- but a 0.1 μF held it stable. In another 2N2222a-based 4Z4ME amp with 0.01 μF input and output caps, it took a 10 μF capacitor on the PNP collector to snuff out some ~766 Hz oscillations.

We don't use a wrap-around PNP bias with our RF oscillators — that's asking for trouble.

I aso measured oscillations at ~ 372 MHz with my spectrum analyzer. A collector 10 Ω R killed these UHF oscillations and after that I saw no spurs from .001 to 1 GHz. (I should have made the dremel cut right close to the NPN collector for the 10 Ω resistor. I hoped there were no oscillations above 1 GHz because I can't measure them.

Finding oscillations:

Many builders lack a spectrum analyzer, let alone 1 that goes up into UHF bandwidth. I'll share a few tips I've learned on the bench that don't require expensive test gear:

A 10X scope probe on the drain or collector of an amp may sometimes reveal oscillations up to the maximum 'scope bandwidth — set your 'scope vertical scale for high sensitivity. This provides direct measurment of oscillations.

Indirect methods to infer unwanted oscillations also lie in our armatorium. I learned this trick from Wes: Place the circuit under test in your normal gain measurement set up with an oscilloscope. Then vary the DC power supply voltage slowly and smoothly — your measured 'scope voltage changes should also track slowly and smoothly. You may see an AC voltage jump as the amplifier goes into and out of oscillation with the DC power supply tweaking. After finding this oscillation caused AC voltage spike, you work to remedy it with a variety of means such as better bypassing, changing bias voltages, shielding and locating breadboard errors.

Sometimes if you put your finger near the active part while watching the bias voltage or current you may see the bias jump around *if* oscillations are present.

My final indirect oscillation busting technique: If you measure the specified/expected gain and return loss on the input/output port, this *may* signal your device is stable — I've noticed this with MMICs where I saw oscillations on my SA, stabilized them and *only then*, measured the expected S21, S11 and S22.

Sometimes, eliminating a hot part proves the best fix! In 2012, a new builder wrote to say that he soldered in a Mini-Circuits DC - 6GHz MMIC; the ERA-2SM in SOT-86 as a buffer for his 3.5 MHz VFO. Anyhow, in the photo were long leads plus no decoupling resistors etc. It sounds like the circuit behaved hyperreactively and *vibrat*ed in spasm. The cure was to eliminate the microwave part and put in a hycas amp built with a J310 + a 2N3904 — we encounter risk when plying the latest, hottest, super-high fT amplifiers sold on eBay with casual abandon.

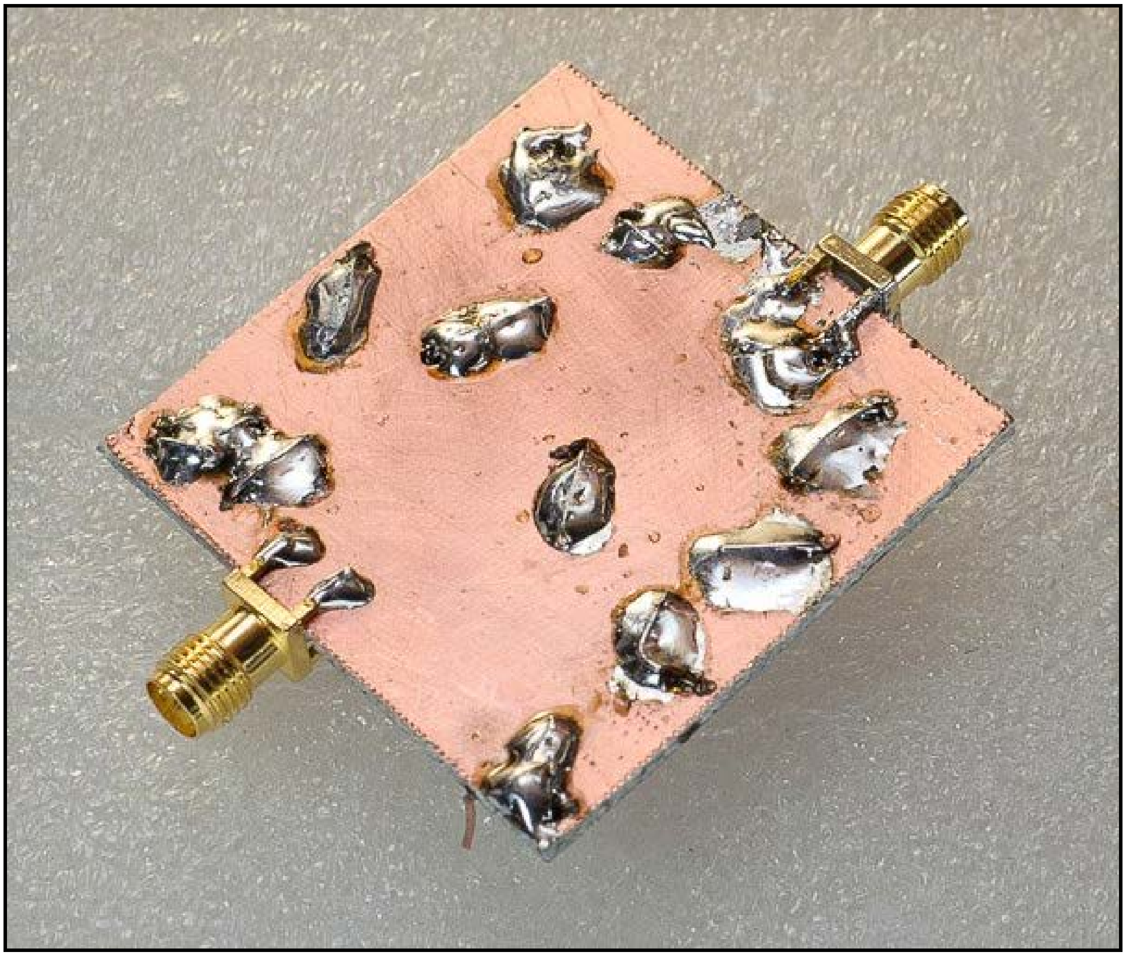
Practice makes perfect. if you believe learning is experiential and build to learn, you'll learn to build.

Finally, as an amateur, I struggled to choose a SMT ferrite bead and after reviewing many datasheets and application notes I ordered a size 0805, 800 mA part with 120 ohms Z at 100 MHz and its peak impedance at 340 MHz. I'll let you know how that works out.

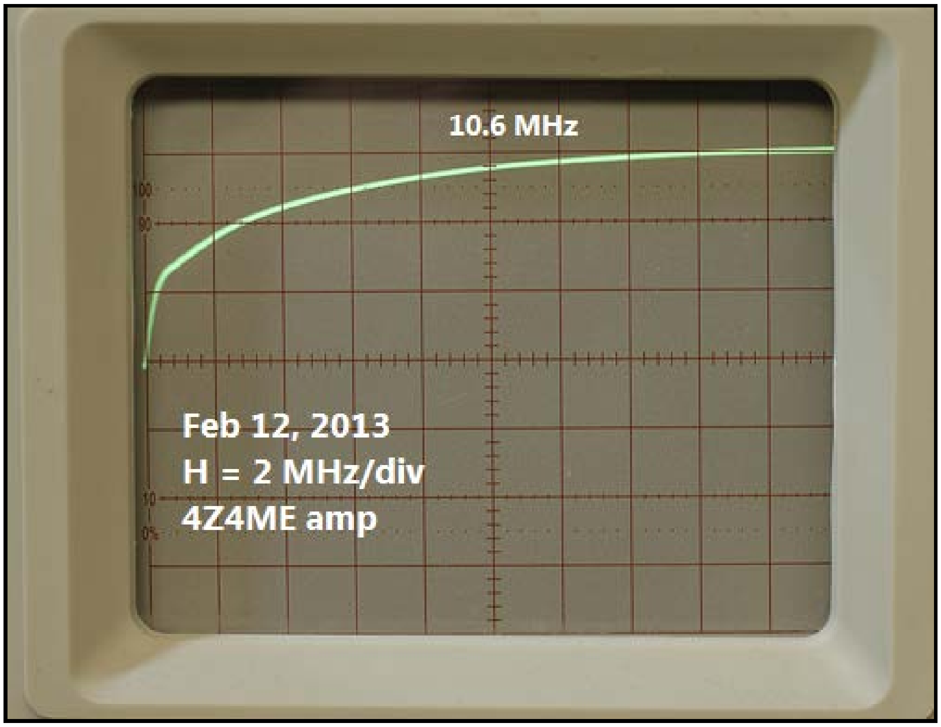
RF Bypass on our DC lines: As possible, we ought to provide a broadband RF bypass to provide a low impedance to RF from low frequency up to the maximum frequency wherever our FET, BJT, MMIC, etc. operates. For example, you can't just swap a higher gain BF998 (1 GHz) for a 40673 (VHF) dual-gate MOSFET and expect the same stability and bypass requirements can you? At the very least, I bypass G2 of the BF998 with a size 0805 0.01 or .001 μF SMT capacitor and the drain with RF bypass good for 1 GHz.

Wideband RF bypass may solve oscillation issues too.

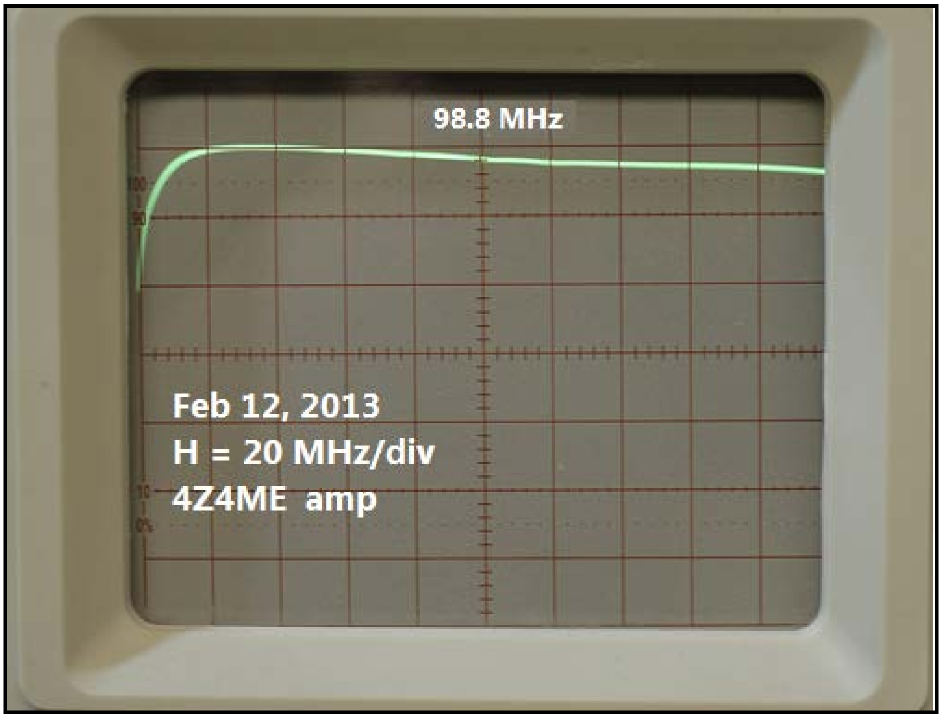
I tried to apply a broadband bypass in my breadboard, although it gets extremely difficult to think about bypassing RF at > 1 GHz for the QRP homebuilder. Our hobby should include reflection and proper intention at the very least.



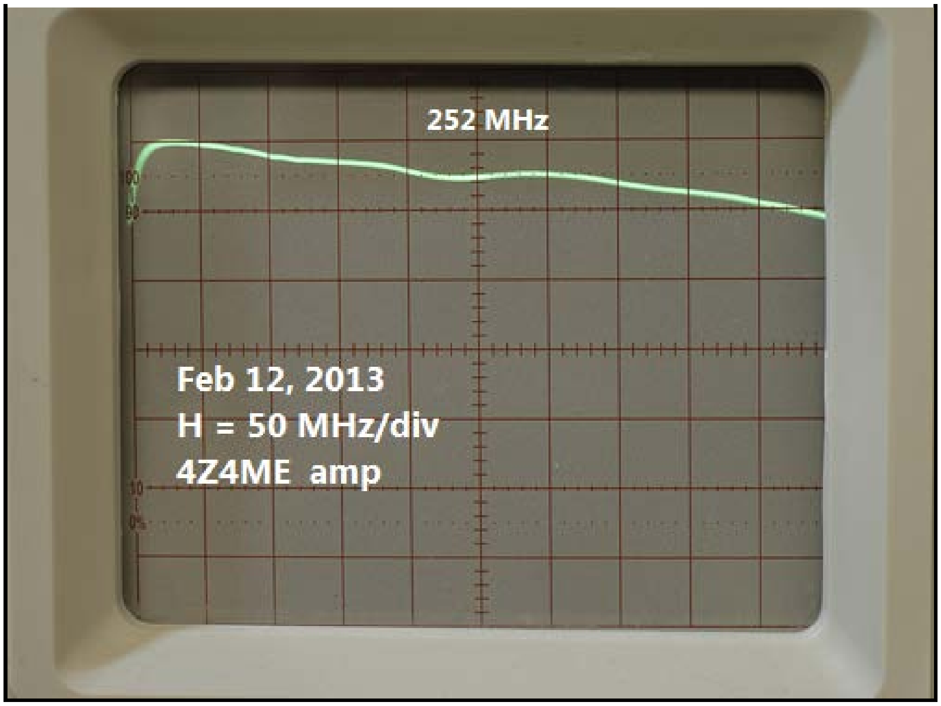
Above — A photo of the bottom of my breadboard showing the vias. I made mistakes: we should try to keep the via holes as close as possible to all bypass caps, my 10 Ω collector snubber resistor, collector port, or whatever we need to put at RF ground. The vias connect circuit areas to the large area, low impedance ground plane to minimize inductance. We should also try to place bypass capacitors as physically close to the pins of whatever we're bypassing.



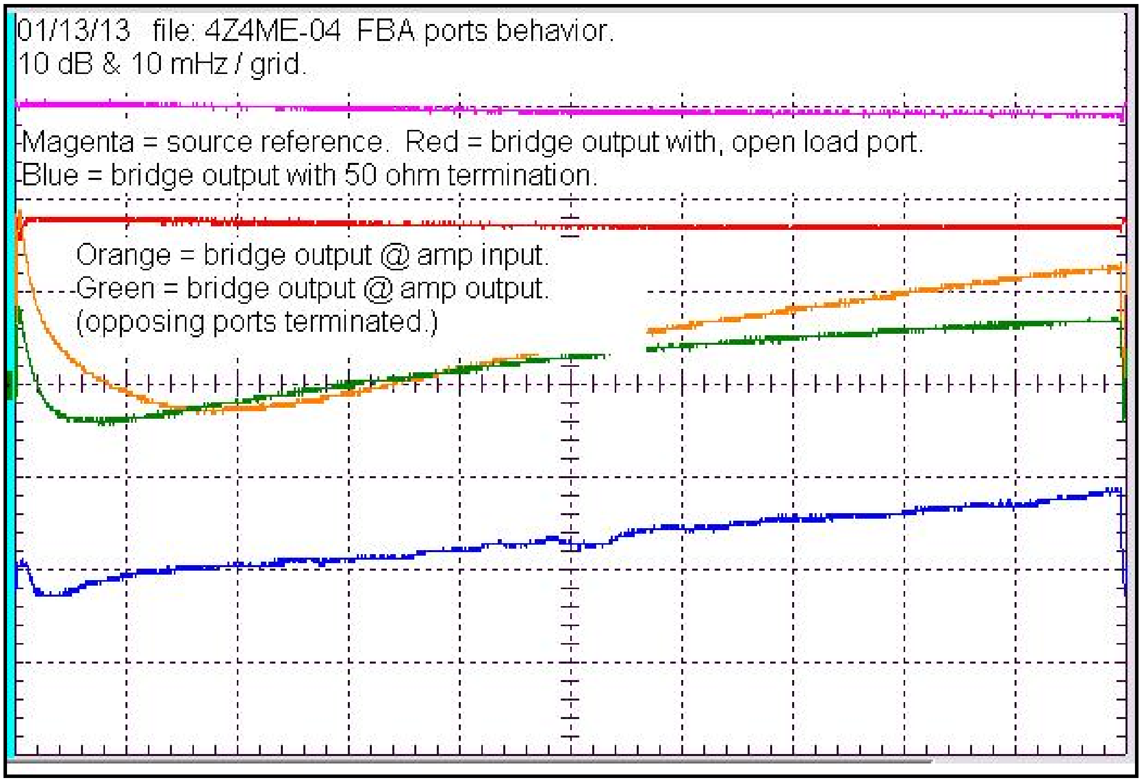
Above — A TG + SA sweep of the gain of my 4Z4ME NE46134 FBA from ~1 to 20.6 MHz. Each vertical quare denotes 10 dB. Each horizontal square = the value specified in the photo. In this case; 2 MHz per division. Maximum gain was ~16 dB.



Above — A sweep from ~ 1 to 200 MHz. This would also make a good 6 meter band amplifer or ????.



Above — A sweep from ~ 1 to 500 MHz.



Above — Bob K3NHI made and swept a 2N5109 version of the 4Z4ME amp biased for ~47 mA emitter current. Here he swept return loss at the amp's input and output from about 1 to 100 MHz. The output return loss of my build was down, however, I didn't own many SMT resistors between 33 and 68 Ω. For example, if the output impedance at the collector is 10 Ω, then the series resistor should be 40 (39) Ω.

I've found that in my FBAs, changing the current and also the transistor type (2N5109 , 2N2222a etc.) also affects the input and output return loss. At HF, it's possible to measure RL with a simple bridge, so optimization is possible.

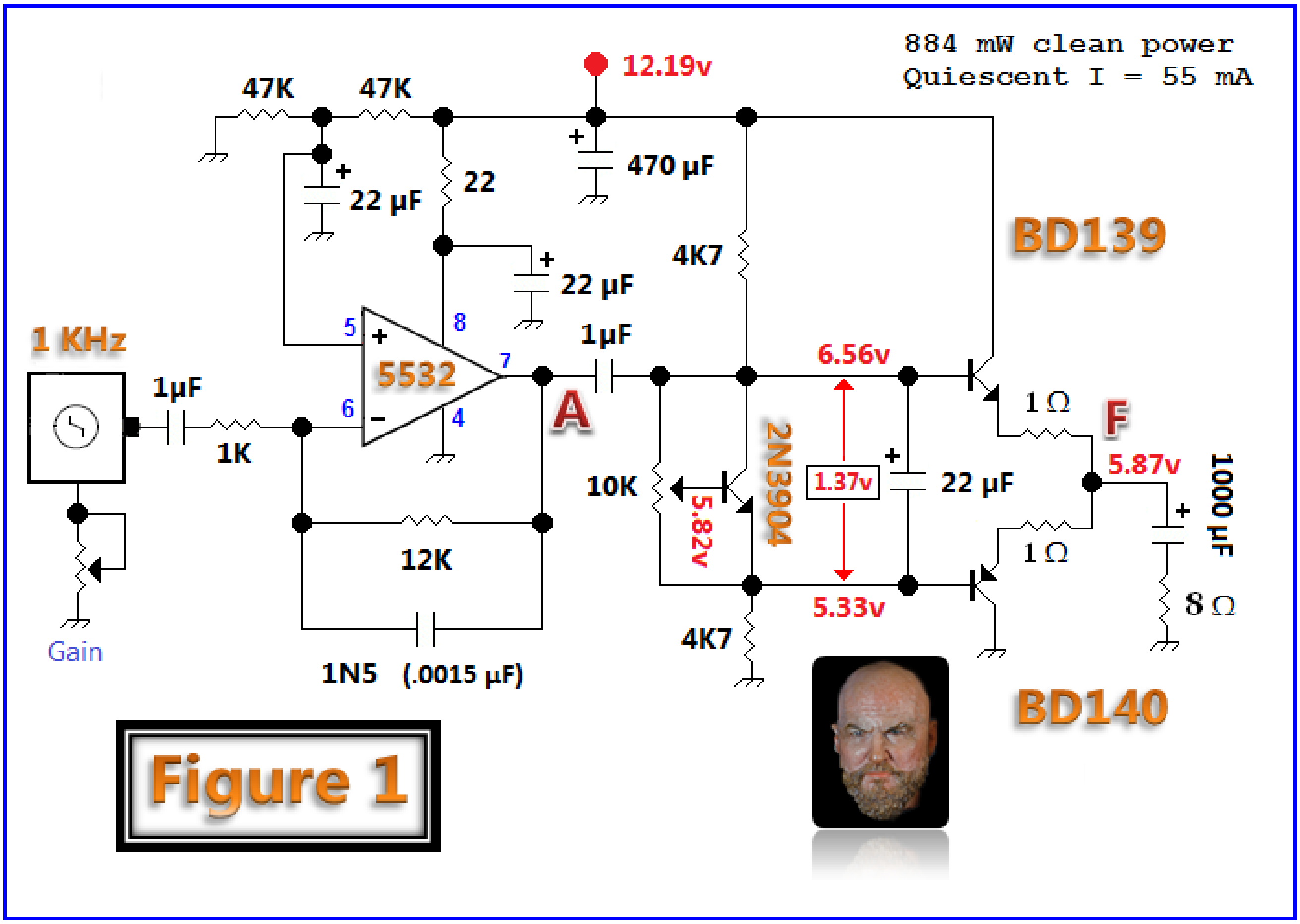
I learned a lot by building just 1 amplifer and discussing my findings by email with friends. Hopefully the next version I make will show improved understanding and skill.

---

8. Popcorn AF Amplifier for Receivers — Reprise

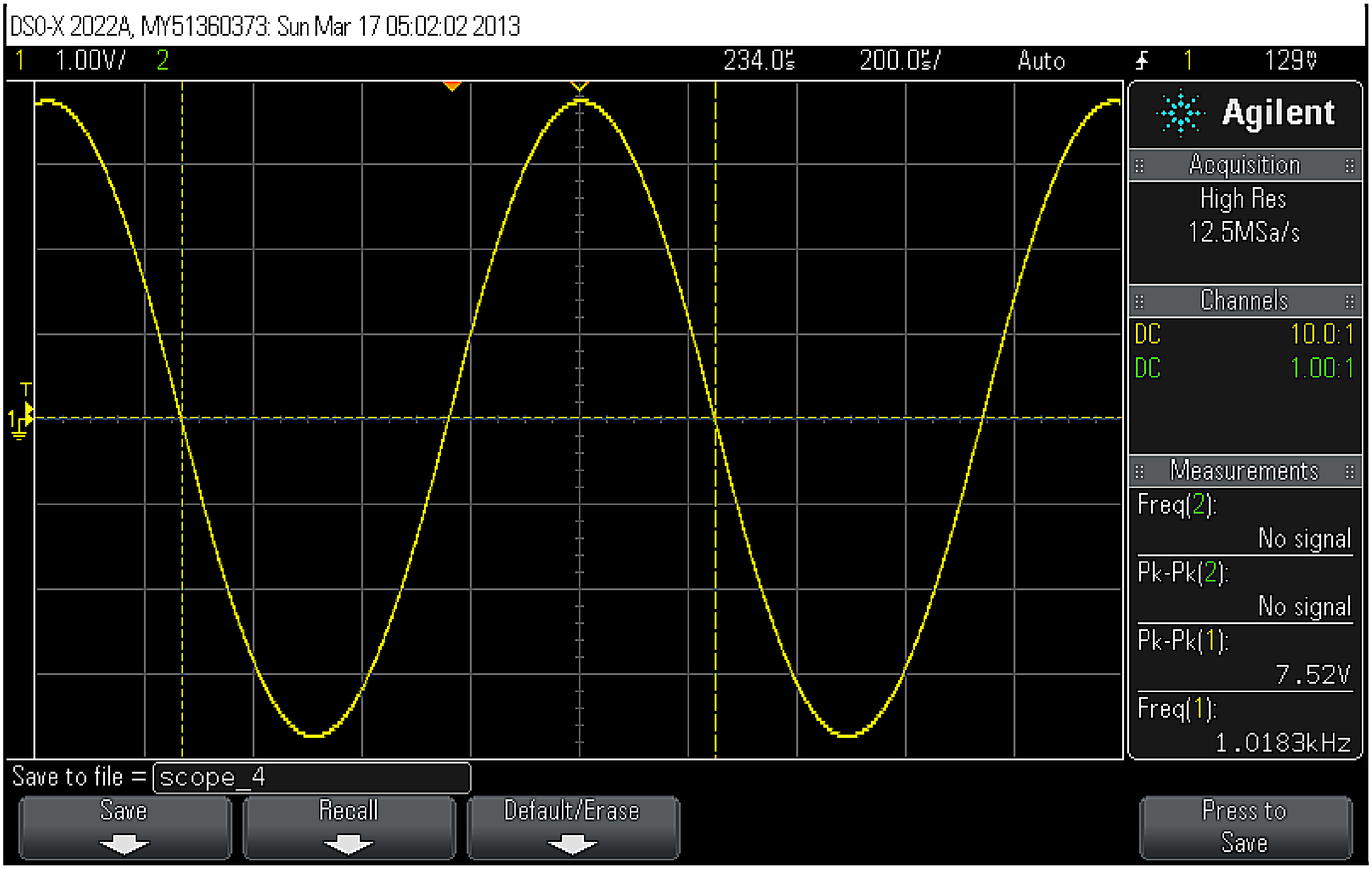
I've worked on a popcorn audio power amplifier (PA) since 2008 and offer my latest experiments. There's only so much you can do with a single-supply 12 volt AF power amp, but I enjoy improving my circuit.

My power measurement technique is shown as Figure 4 here. To enhance versatility, the following PA's may be coupled to whatever preamplifer you choose. In all cases, I drove the power amp stage with a 5532 op-amp voltage amplifer. The power followers were biased with a 2N3904 amplified diode (also called NPN shifter bias amplifier, or DC level shifter) rather than just a pair of series diodes, since this allows you to dial in just the right amount of bias as you watch the AC signal in your 'scope. I wrote a tutorial that explains how to bias complimentary-symmetry power followers in 2008: Click for the link.



Above — Figure 1: A popcorn AF power amplifer in full bench *test* mode. Measure the AC with a 10X 'scope probe across the 8 Ω resistor and the DC voltage and current with a multimeter. A distortion analyzer proves useful, but not essential for popcorn circuitry. I also listened to each amplifer connected to a line-level cassette player and an 8 Ω, 15 cm speaker. A 4 Ω speaker doubles the maximal clean power, but I don't own any and stuck to 8 Ω.

Containing no negative feedback, the power amplifer stage runs from the red-colored designator points **A** to **F**. You can AC or DC couple point F to your preamplifer stage as required to apply negative feedback.
As mentioned, you can use the 5532 preamp shown with any reasonable gain (i.e. change the 12K resistor), or opt to replace it with your own design. A low output impedance amplifer best drives the power stage.



DSO-X 2022A, MY51360373: Sun Mar 17 05:02:02 2013

1 1.00V/ 2 234.0µs 200.0µs/ Auto 1 129mV

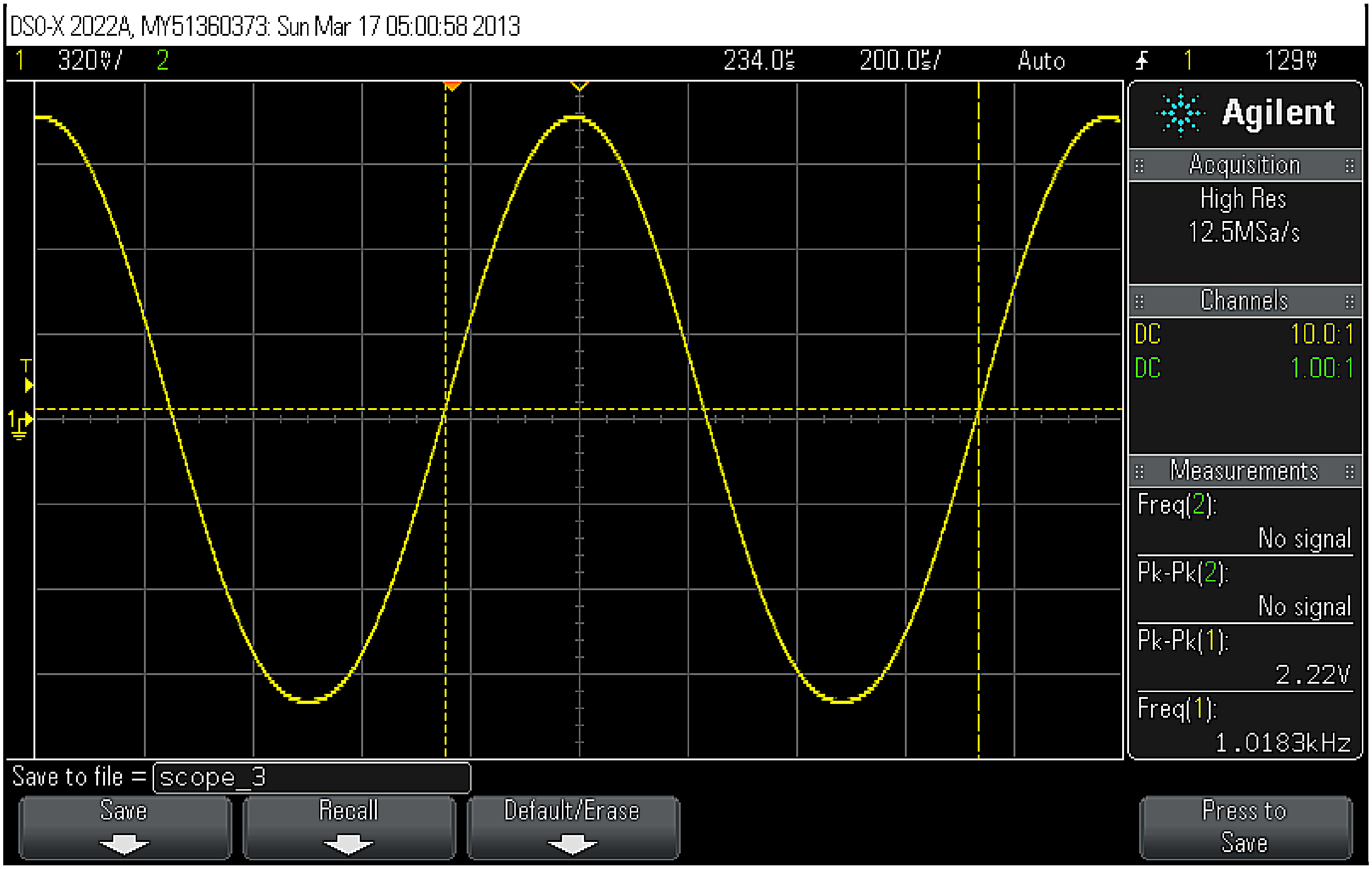
Acquisition
High Res
12.5MSa/s

Channels
DC 10.0:1
DC 1.00:1

Measurements
Freq(2): No signal
Pk-Pk(2): No signal
Pk-Pk(1): 7.52V
Freq(1): 1.0183kHz

Save to file = scope_4

Above — The output of Figure 1 in my 'scope driven to the maximal pk-pk voltage just before distortion begins to appear. Obviously this task is somewhat subjective, however, allows comparison of the amps you build on your bench.



DSO-X 2022A, MY51360373: Sun Mar 17 05:00:58 2013

1 320mV/ 2 234.0µs 200.0µs/ Auto 1 129mV

Acquisition
High Res
12.5MSa/s

Channels
DC 10.0:1
DC 1.00:1

Measurements
Freq(2): No signal
Pk-Pk(2): No signal
Pk-Pk(1): 2.22V
Freq(1): 1.0183kHz

Save to file = scope_3

Above — A 'scope screen capture with the 22 µF level-shifter filter capacitor from Figure 1 removed. Look what happened; the maximum clean signal fell from 7.52v pk-pk to 2.22v pk-pk. That capacitor is essential to get the maximal possible headroom.

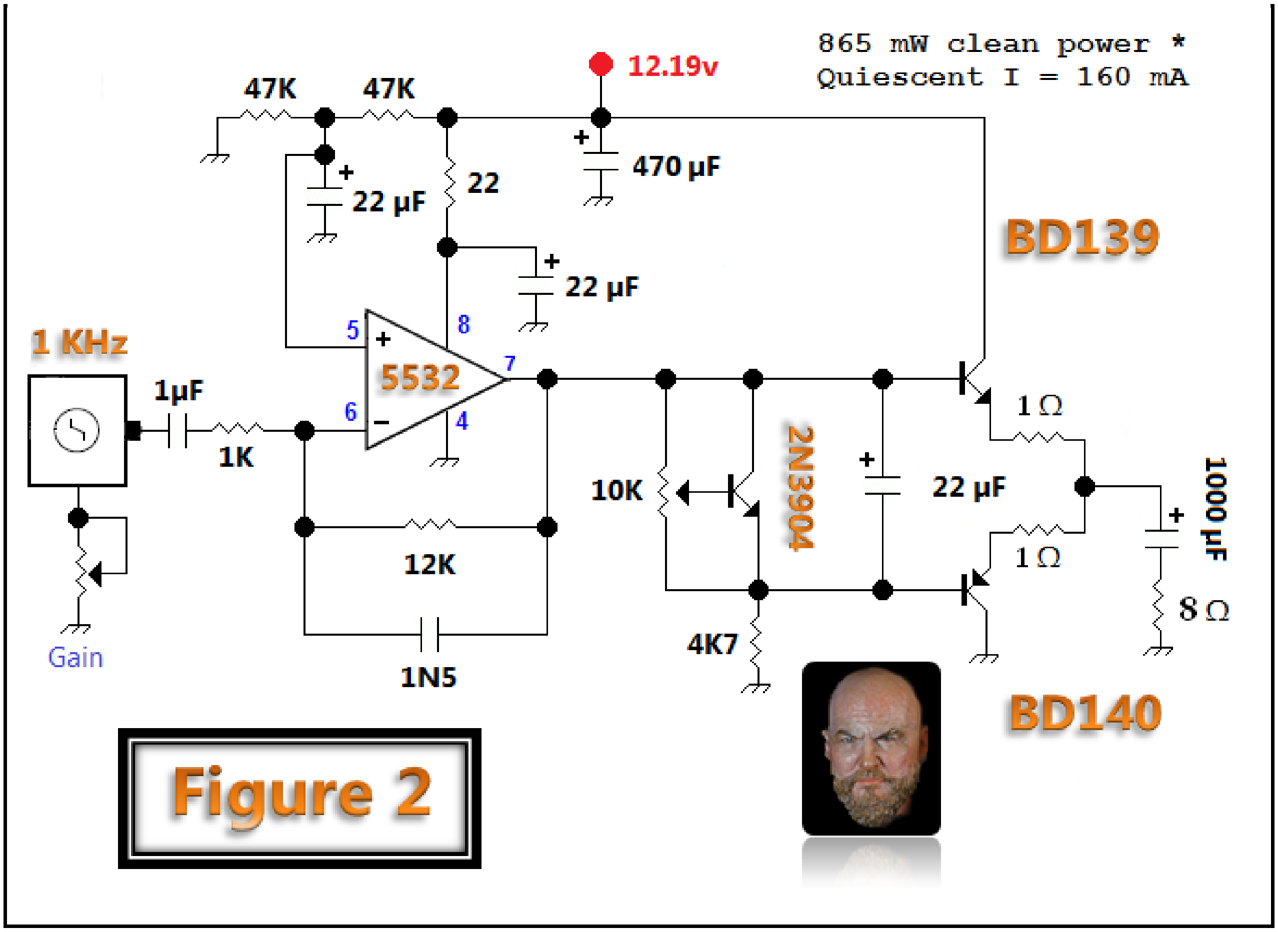
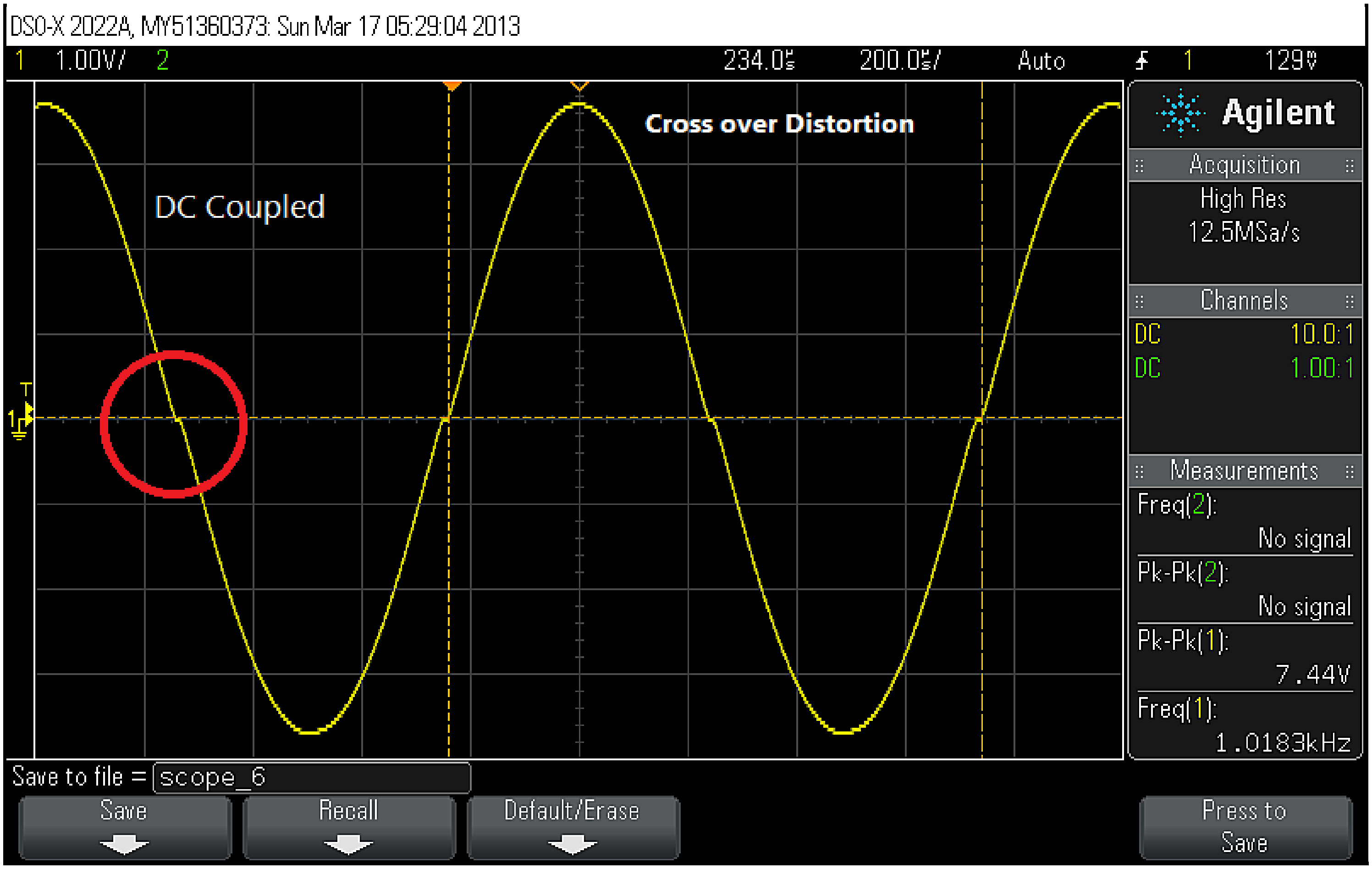


Figure 2

Above — Figure 2 is Figure 1 with the op-amp DC coupled to the level-shifter. I tested the circuit with and without the 4K7 resistor connecting the base of the 2N3904 to the DC supply: it didn't boost the amplifer headroom, nor reduced crossover distortion, so I removed that R.



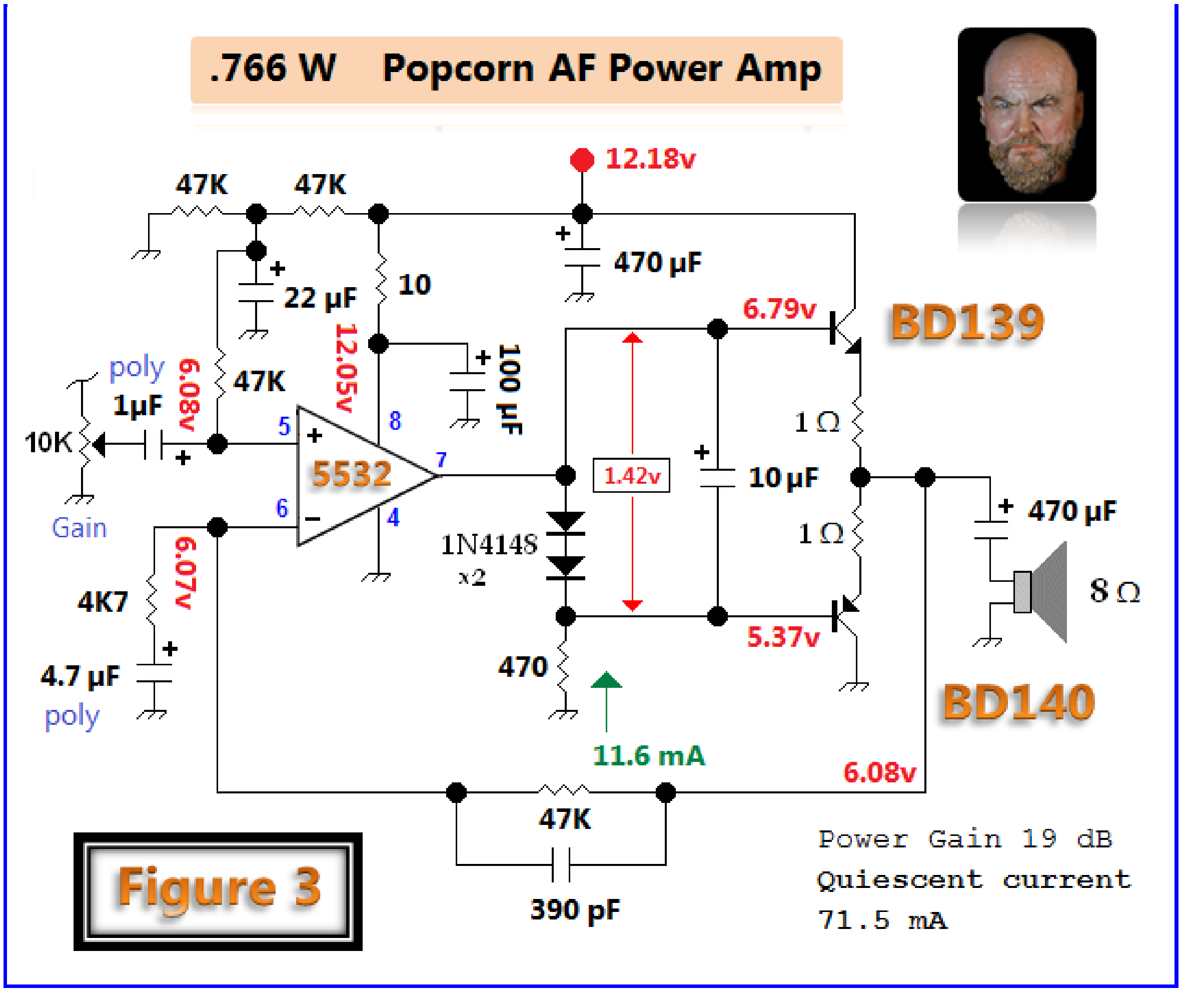
Acquisition
High Res
12.5MSa/s

Channels
DC 10.0:1
DC 1.00:1

Measurements
Freq(2): No signal
Pk-Pk(2): No signal
Pk-Pk(1): 7.44V
Freq(1): 1.0183kHz

Save to file = scope_6

Above — The Figure 2 amplifer 'scope tracing. At *maximum power*, crossover distortion appeared and I've seen this before. Likely, there is not enough base drive to keep the power followers forward biased. By adjusting the level shifter, I almost removed the crossover distortion, but never eliminated it. This drove the quiescent current up to 160 mA. Yikes!

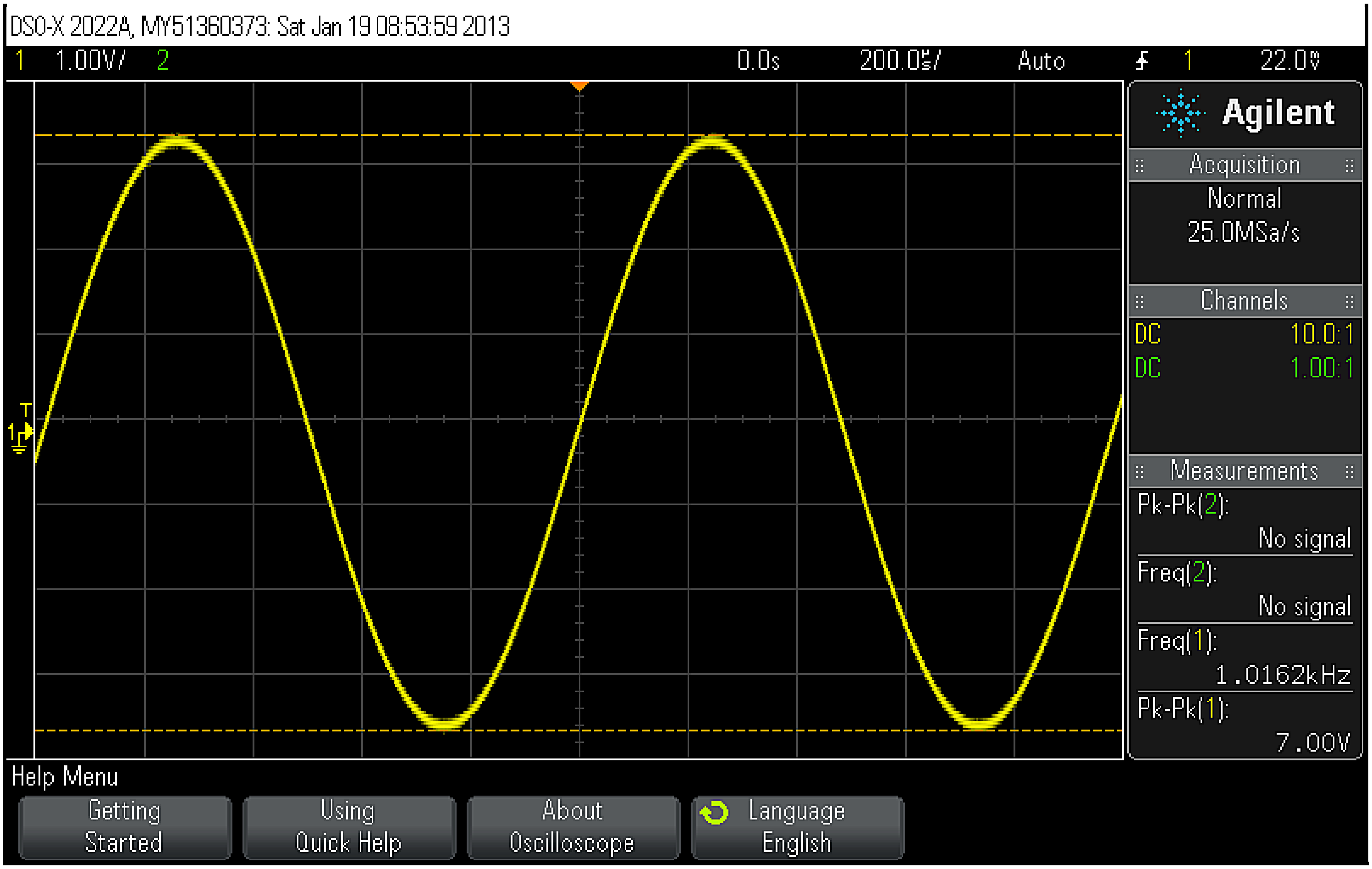


.766 W Popcorn AF Power Amp

Figure 3

Power Gain 19 dB
Quiescent current
71.5 mA

Above — A variation of Figure 2 employing diodes instead of an NPN level-shifter. To kill the cross-over distortion, I lowered the 4K7 resistor by a magnitude of 10. This gave a maximum clean power of 766 mW with a quiescent current of nearly 72 mA. Head room and quiescent current are inferior to the Figure 1 circuit.



DSO-X 2022A, MY51360373: Sat Jan 19 08:53:59 2013

1 1.00V/ 2 0.0s 200.0µs/ Auto 1 22.0mV

Acquisition
Normal
25.0MSa/s

Channels
DC 10.0:1
DC 1.00:1

Measurements
Pk-Pk(2): No signal
Freq(2): No signal
Freq(1): 1.0162kHz
Pk-Pk(1): 7.00V

Above — The Figure 3 'scope tracing. Click for a 'scope tracing with the signal generator amplitude increased *slightly* to push this amp into clipping.

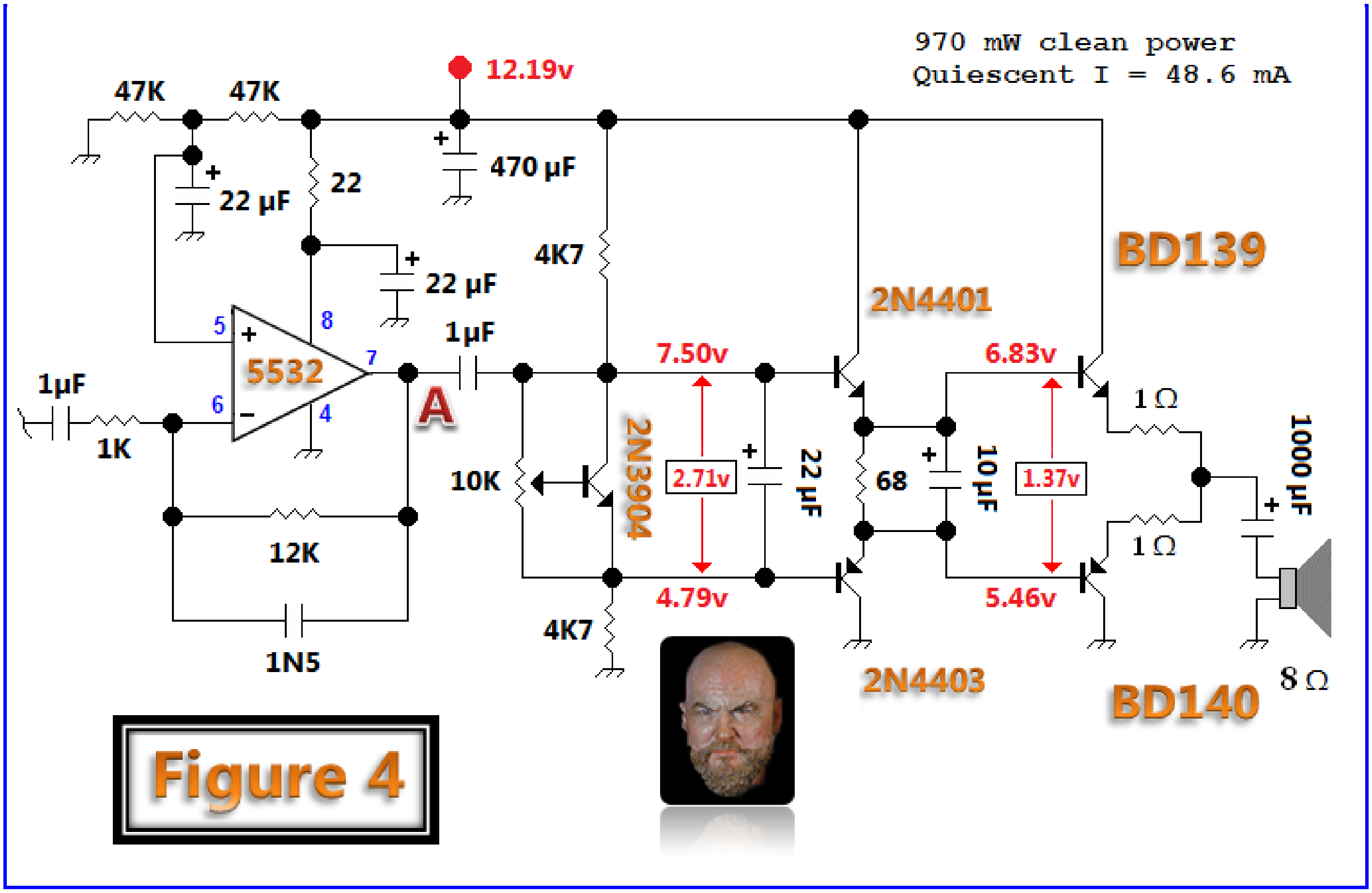
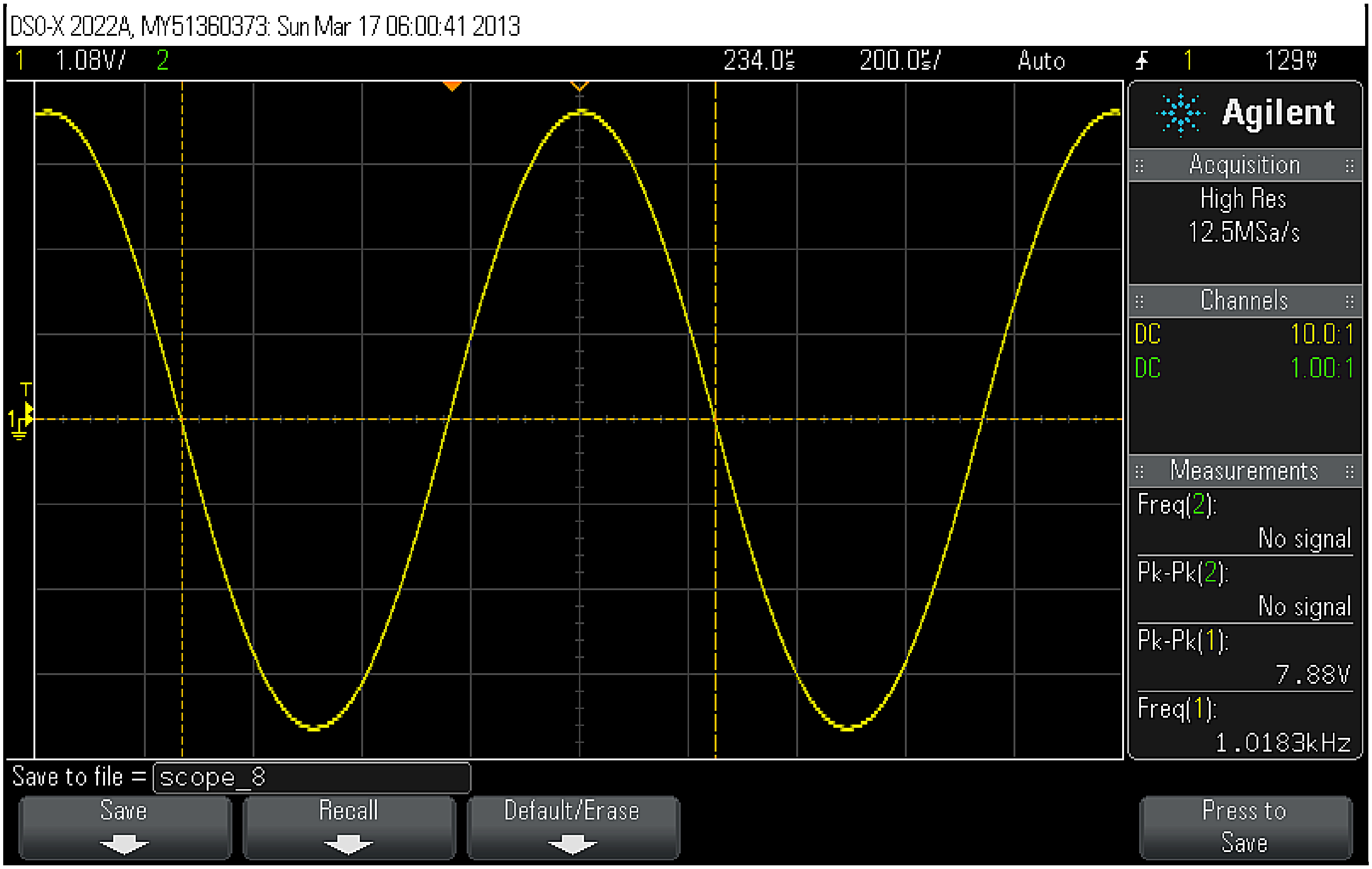


Figure 4

Above — Back to an AC coupled power amplifer like Figure 1. I added a set of intermediate followers built with a 2N4401/2N4403 pair. The clean output power now lies at 970 mW with a quiescent current under 50 mA. Adjusting the trimmer potentiometer on the level-shifter even a tiny amount may change the quiescent current dramatically.

I found a bias of 1.37v across the BD139/140 pair removed all trace of cross-over distortion at maximum clean signal power. Just tweak the 10K trimmer potentiometer while looking at your 'scope and decide what bias you prefer. I lower the bias until crossover distortion appears and slowly tweak it to find the sweet spot. Then measure the DC voltage across the power follower base terminals, plus the total stage quiescent current with the signal generator switched off. You might have to repeat this procedure a few times, since trimmer pot adjustment is quite sensitive.

Jerry, W5JH made a PC board. Click for front photo. Click for rear photo.



DSO-X 2022A, MY51360373: Sun Mar 17 06:00:41 2013

1 1.08V/ 2 234.0µs 200.0µs/ Auto 1 129mV

Acquisition
High Res
12.5MSa/s

Channels
DC 10.0:1
DC 1.00:1

Measurements
Freq(2): No signal
Pk-Pk(2): No signal
Pk-Pk(1): 7.88V
Freq(1): 1.0183kHz

Save to file = scope_8

Above — The Figure 4 'scope tracing.

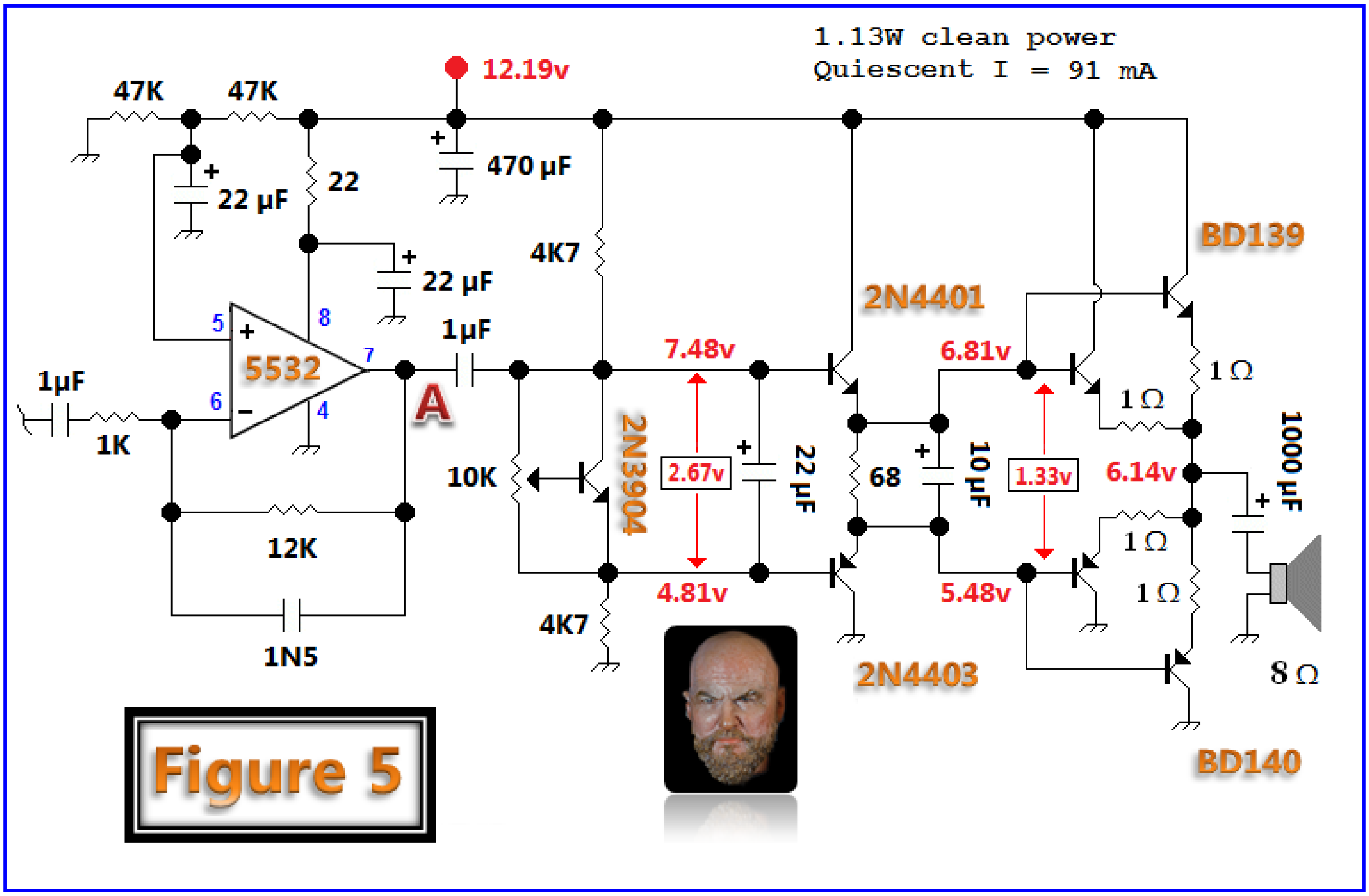
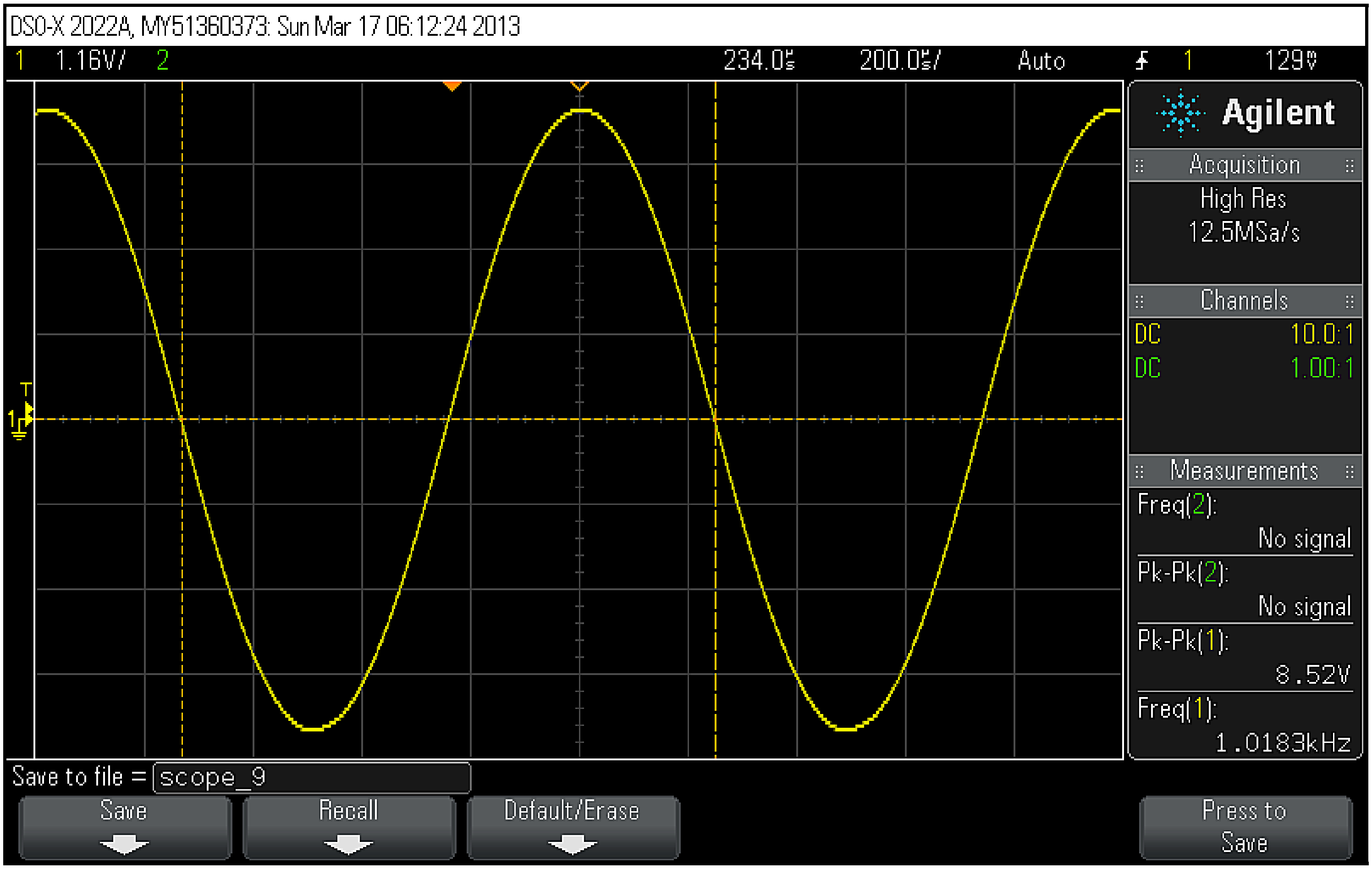


Figure 5

Above — I added another BD139/140 power follower pair in parallel. The boost in headroom over Figure 4 was small, but it was nice to break the 1 Watt barrier. This amp sounded great and blew away an LM386 set up for a gain of 20 — more headroom, less noise and boosted warmth.



DSO-X 2022A, MY51360373: Sun Mar 17 06:12:24 2013

1 1.16V/ 2 234.0µs 200.0µs/ Auto 1 129mV

Acquisition: High Res 12.5MSa/s

Channels: DC 10.0:1; DC 1.00:1

Measurements:

Freq(2): No signal

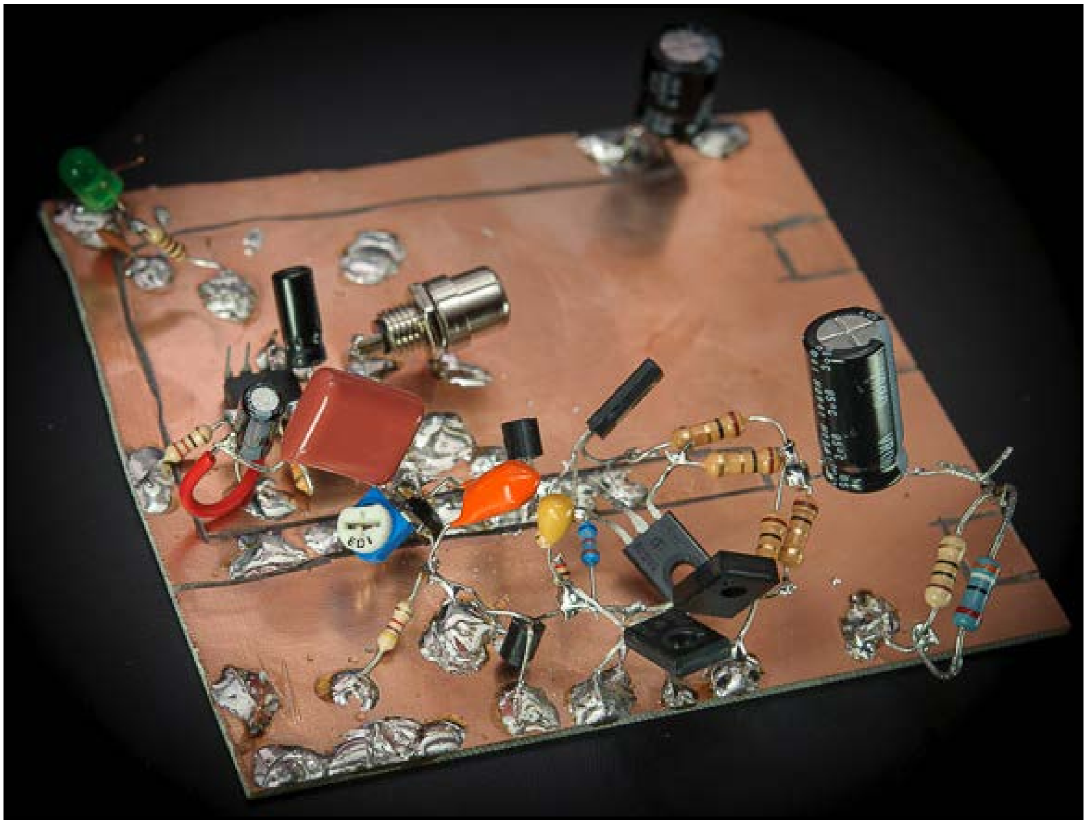
Pk-Pk(2): No signal

Pk-Pk(1): 8.52V

Freq(1): 1.0183kHz

Save to file = scope_9

Above — The Figure 5 'scope tracing.

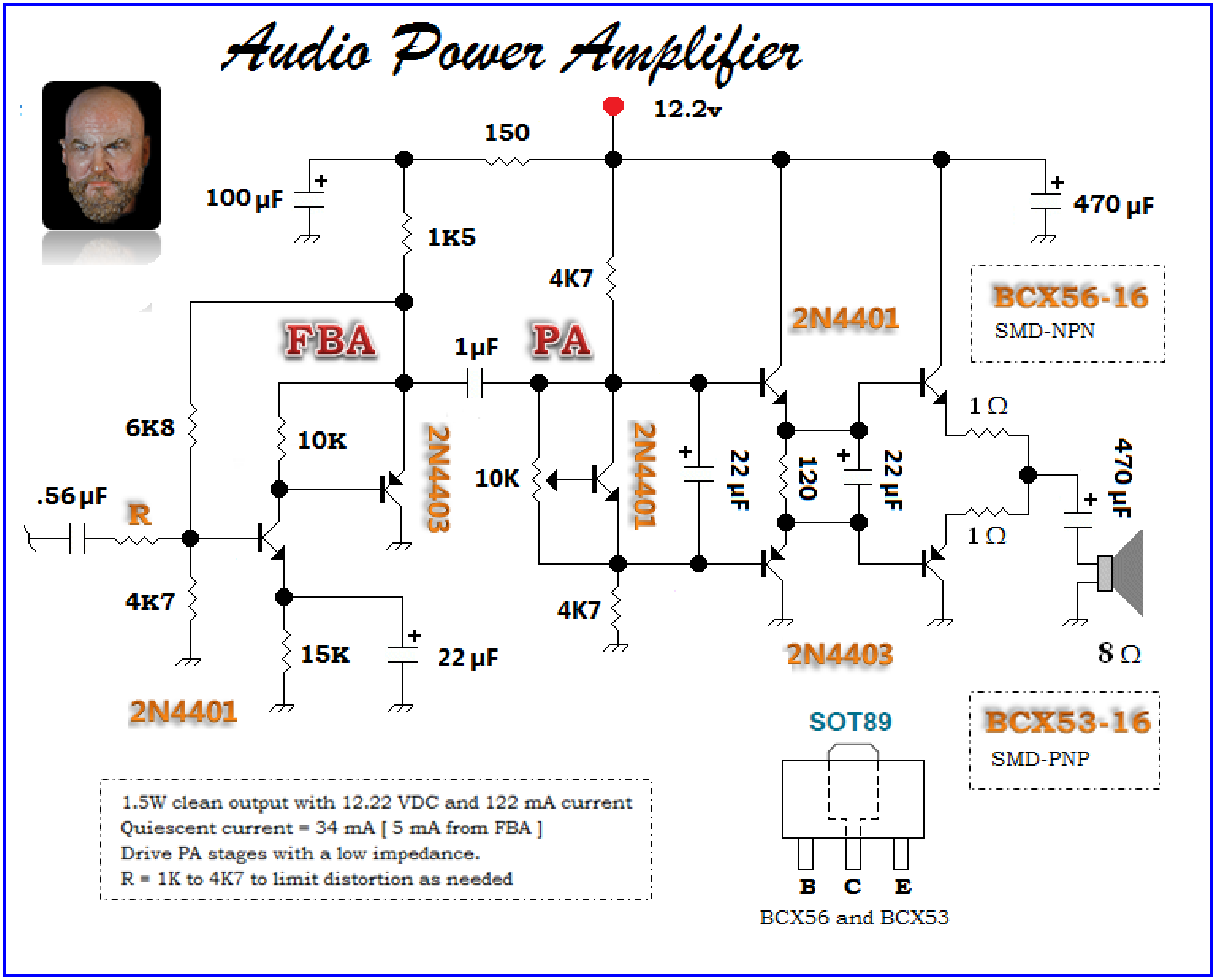


Above — The Figure 5 breadboard. I built all the AF power amps on this board. Signal caps <= to 1 μF were polyester film, while I employed 10 or 22 μF tantalum caps for the level shifters. Electrolytic caps work fine; especially for the level shifter capacitors. The green power indicator LED drew 10 mA and I subtracted this from the quiescent current measurements.

Depending on the AF gain of your receiver, you might wish to add the familiar Zobel filter; a 10 Ω R in series with 0.1 μF C from the positive end of the output capacitor to ground, or more AF bypassing/ decoupling to the circuitry.

No component values were critical — imbuing the spirit of homebrew radio, substitute parts and measure outcomes.

QRP — PosData for August 18, 2014



Above — I built a 100% discrete version of Figure 4 power amplifer with a 2N4401/2N4403 feedback voltage amp that drives the PA stage with a low impedance. Without a low impedance drive, the PA [all the components right of the 1 μF capacitor where you see **PA**] the headroom of the PA section falls down. Thus an op-amp or emitter follower drive works great.

I moved to SMD power transistors: the NXP BCX56 + BCX53. These give a little more output than the BD139/140, and take much less board room. I ran no heat sink other than 8 mm by 4 mm PC board traces for the collectors. A better choice might be the related BCP56 + BCP53 pair in SOT223 since the bigger package of this version better sinks heat.

I determined the optimum emitter resistor for the 2N4401/2N4403 followers = 120 Ω [not 68 Ω as shown in Figure 4]. This drops the the PA quiescent current in half and yields the same performance. If you build Figure 4, change to 120 Ω.

I meant Figure 4 and the QRP POS-Data amp as easy-to-make power amplifiers with a *little* gain. You will have to add voltage gain in some AF projects. For Figure 4, it's easy to change up the 12K resistor, or apply the other half of the 5532. Decouple and bypass better if you get motor boating or hum. 73 de VE7BPO

---

9. The Progressive Receiver by Wes, W7ZOI and John, K5IRK

When introduced in QST for November 1981, the Progressive Communications Receiver (PR) by Wes, W7ZOI and John, K5IRK set a dynamic range benchmark for dual-conversion homebrew receivers: 94 dB in CW mode. 31 years later, few other home-built radios have ever reached this benchmark.

The PR is a single conversion superheterodyne 80m receiver; or perhaps it's a direct conversion receiver with an extra mixer ahead of the product detector? It's both and that's the point. Further, Wes and John added other bands with another mixer, plus a crystal oscillator, RF filter +/- an RF amp for each band.

You'll see the PR listed as A High-Performance Communication Receiver in ARRL Handbooks from the early 90s or so. ARRL staffers built and enjoyed the PR in their test lab for many years.

After purchasing the 1991 ARRL Handbook and reading about this project, I slowly adopted and entrenched the PR's progressive (modular) approach. When I built and web published my 2 TRF WWV receivers with crystal IF filters, a couple of people wrote that I "grossly over-designed" them, however, other, more astute builders placed a mixer on the front end and turned their TRF into a multi-band superhet like I later did. These builders understood how progressive circuit building works — all PR inspired.

I hope my introduction renews your interest in the PR. Wes and John's article is thorough and complete — I can't add to it, however, I'll share my thoughts along with those from Wes and John all these decades later.

The IF Stage gain comes from 2 dual gate MOSFET amplifiers. The final MOSFET amp drives a BJT differential pair providing 9 MHz RF to the product detector at 50 Ω and signal for the AGC circuit. Wes recalls using a 3N211 in the original MOSFET slots since Doug DeMaw owned a pile of them and contributed some. In some areas, the 40673 was the dominant 2-gate MOSFET soldered by radio home builders back in the day.

An evolved version of the PR Intermediate Frequency stage appeared in EMRFD as Figure 6.50. Wes replaced the now hard-to-find and *expensive* leaded dual-gate dual gate MOSFET with a cascode of J310s. I built this stage and it performed well with my 12.2 VDC power supply.

Jeff, WA7MLH built the general purpose IF system and noticed if the power supply dropped below 12 VDC, the gain control and maximum gain fell off and he wrote to Wes about his findings. Wes later confirmed dysfunction with a lower DC supply. I asked him to recollect this time:

"…. Frankly I don't remember if I went immediately to computer simulations or if I built a single stage. I think I built. Anyway, Jeff was correct, depending upon the FETs that were used. Then I got to plowing into the details of the cascode, this time with SPICE. I was using PSPICE for some simulations, but was in the process of switching over to LT-SPICE. That makes no difference, for I used the same models in both.

Anyway, playing with a single stage showed that the mechanism for gain reduction was that when the voltage on the upper gate was reduced, it did nothing to the upper stage, but it compressed the supply on the lower FET. So far as signal goes, the upper part was nothing more than a common gate stage that passed whatever signal current was there in the drain of the lower part on into the source of the upper part.

But these were depletion mode FETs, the normal thing for most of the JFETs we use. As such, you have to get the gate down pretty low to get the source voltage low enough to be effective in reducing the gain of that lower FET. That's when it became clear that one could use other parts in the upper slot. An enhancement mode FET such as a MOSFET would work well. And just a common bipolar would do the job nicely too..." *Wes, W7ZOI per an email - January 2013.*

In QST for 2007, a refined version of the EMRFD Figure 6.50 IF stage appeared with a hybrid cascode amplifier instead of the cascode JFETs (and the 2 original dual gate MOSFETs of the PR). Further, Wes added a third amplifier to widen gain control. The hybrid cascade stage has since gone in 100s of receivers across the globe; ensuring the PR legacy lives on as the modernized hycas version.

Roger, KA7EXM sold kits of the hycas amp for years, however, these kits sold out. PC boards may still be available via his web site. Roger received permission to publish the QST hycas article online:

Above — The now *sold out* hycas IF System kit once offered by Roger, KA7EXM.

To underscore my love of the original PR IF amp, I built a version using SMT dual gate MOSFETs in 2013 and feel the SMT version might be perfect for builders who prefer to manually control IF gain in most situations, but want AGC control now and then. Versions built with the BF998 MOSFET may suffer parasitic ocillations at UHF and so meticulous attention to decoupling and bypassing out to UHF and in some cases double-sided copper board may be required to prevent unwanted ringing at UHF.

If you build the entire PR with the BF998, the VFO output runs about 10 dBm and requires attenuation. Again, please consider UHF oscillation precautions.

Wes and others have built both leaded and all-SMT versions of the hycas amplifer.

John, K5IRK Recalls

I wanted John's recollection of the PR design and build and received the following narrative in late March 2013:

"…One night in late 1979 I had telephoned Wes to chat about QRP and building rigs. During the conversation he told me he had a project and related QST article in mind and asked me if I wanted to participate. He was in the process of writing a book (IRFD) and didn't have much free time, so he needed some help. I told him yes. A few weeks later I received the first schematics and began to gather parts.

Recall that the project started as direct conversion receiver. But it then progressed (hence the title of the article) to a superhet. I choose to build my superhet with a SSB filter as I had already built Wes' CW Competition Grade RX from SSD. During the next year he sent me schematics and a few critical parts thru snail mail. (The Internet and e-mail would come later.) I would build the circuits and report the results back to him. As I recall, we only had about three phone conversations throughout the whole project.

The design was an iterative process. For example, the IF circuitry began with a single 40673, but grew to include two stages. Wes bread boarded this in a mono band version of the receiver at his end. An early audio derived AGC system was replaced by the IF derived circuit that appears in the article. The VFO was also designed twice, as were the Front end Filters and the BFO. More is said below about the design procedure.

I designed the circuit board layouts. The traces on my boards were drawn by hand and etched at home. I sent hand drawn sketches of each of the boards to Doug DeMaw; Circuit Boards Specialists (CBS) then created the commercial boards for those folks who wanted boards. I recall that we sent out layouts to any readers for a SASE. All of my boards were designed and built on double sided PC material with the exception of the VFO board and the Audio Filter boards. This can be seen in the photos. The boards eventually sold by CBS were single sided. All layouts and functionality were confirmed prior to publication.

The traditional design process for a homebrew ham receiver in 1980 started with schematic sketches based upon the intuition and experience of the designer. The total circuit would then be built. Measurements were merely things that were done afterward, something to characterize the result. The PR was different. Individual stages were designed, built, and measured. Negative feedback was used in the critical amplifiers to guarantee that the gains were high enough for reasonable noise figure, but low enough to preserve input intercept.

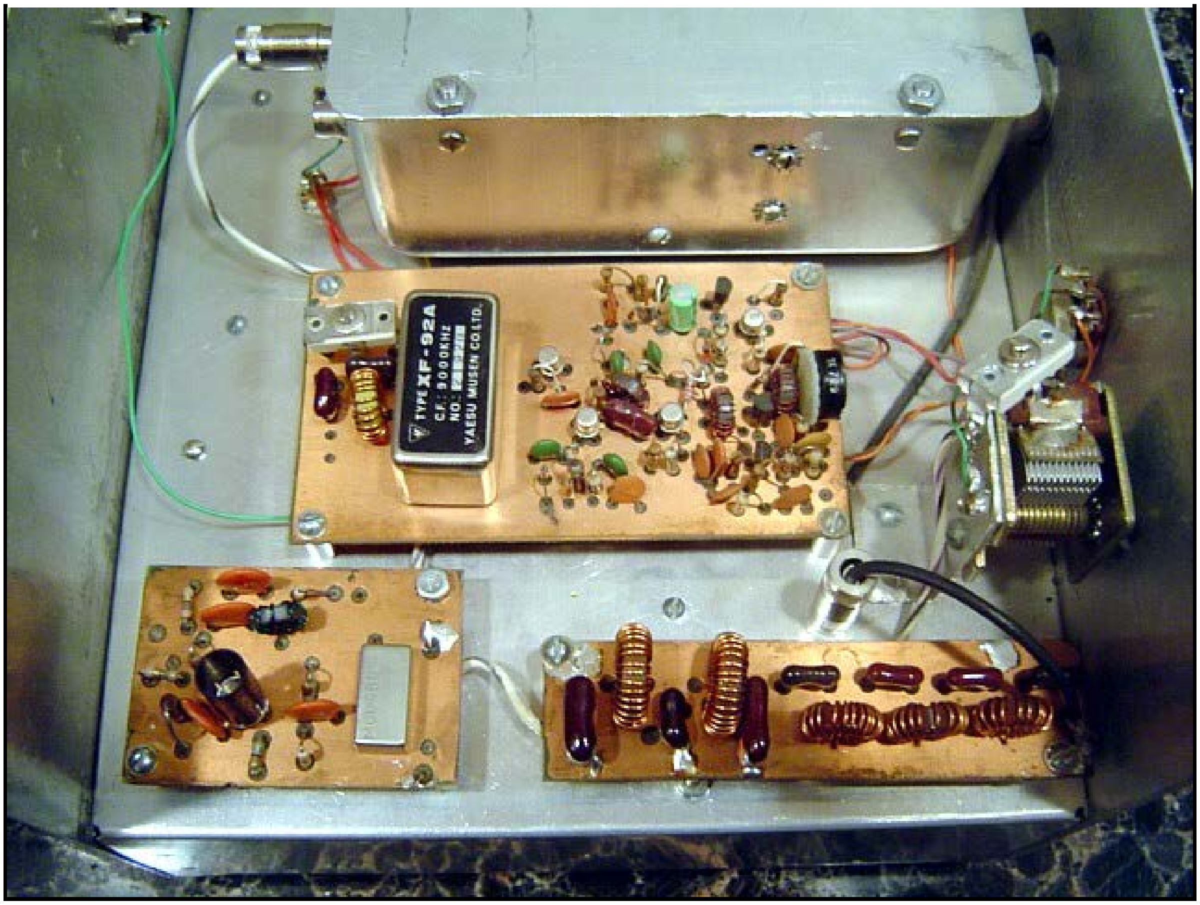
Only after the individual stages were operational, were they assembled to form a working receiver. Some stages were further modified during system assembly and evaluation. We had no computers available for circuit simulation, although the gain distribution was optimized with a hand calculator. The goal was not just a receiver that sounded good, but a box with good two tone dynamic range.

Once my receiver was completed, I sent it to Wes for MDS and DR measurements. A goal was to compare these measurements with those already done in Oregon. Roger (KA7EXM) took many (if not all) of the photos. Wes then forwarded the receiver to ARRL. They returned it to me when they were finished. Wes refined the article during the final months of the project before it was finally published. I received copies of his drafts, and then offered my feedback.

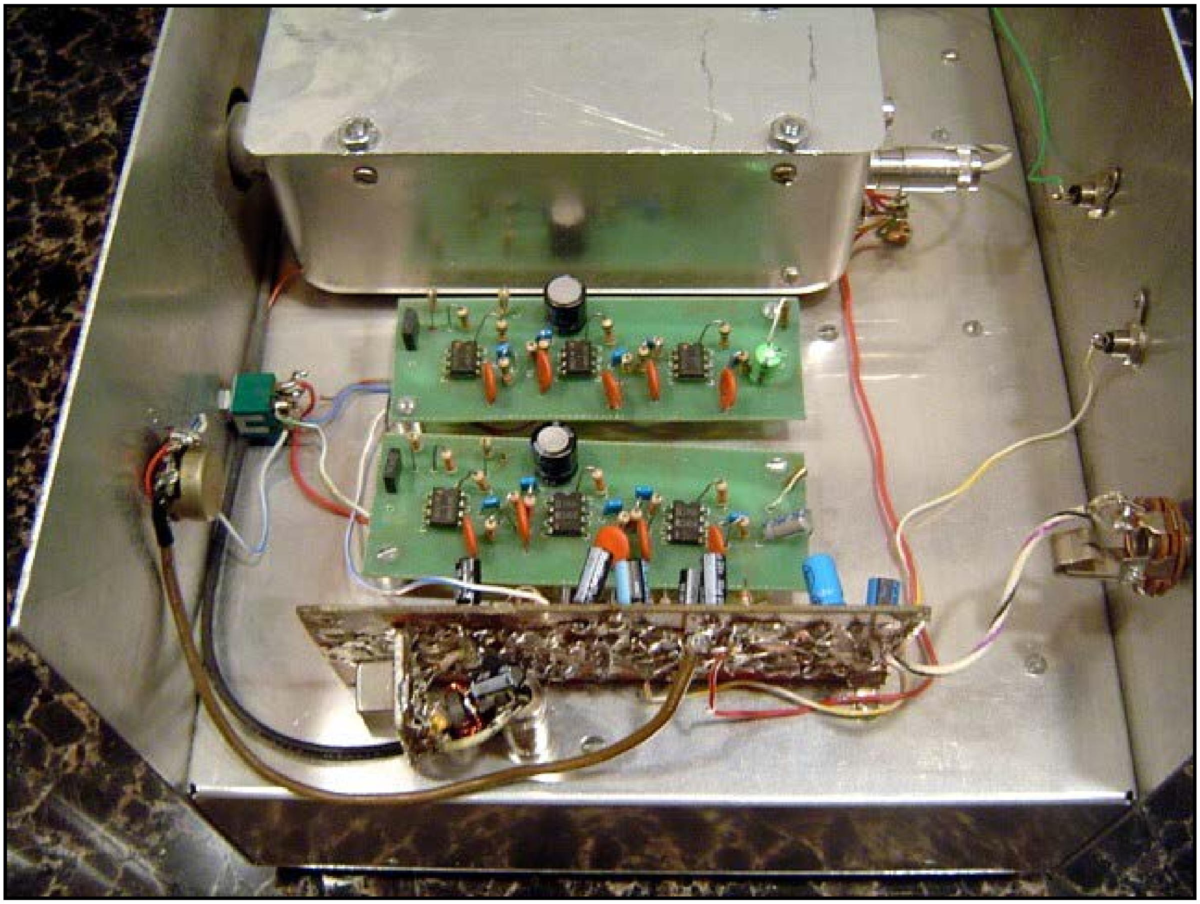
We had hoped that a "few" experimenters would enjoy building the receiver, but had no idea that it would be as successful as it was. We eventually learned why this occurred: First, the receiver held up well when it was measured at ARRL Headquarters, exceeding the performance of most of the appliances being evaluated at the time. This prompted some League staff members to build the receiver for their own use. This, in turn, prompted them to include it in the Handbook for several years.

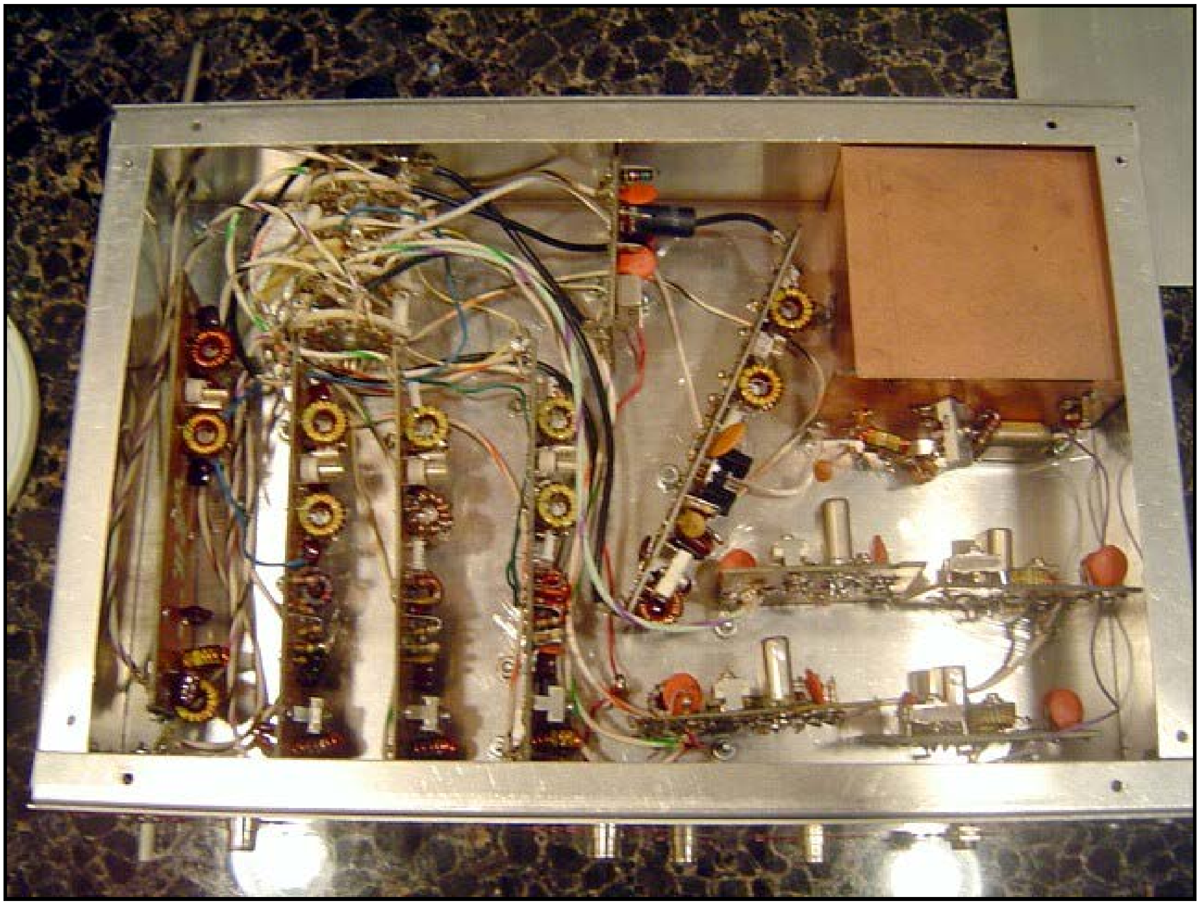
My original receiver plays as well today as it did in the beginning. The electrolytic caps on the audio board have been replaced, for they were beginning to go south on me, but that's the only change. I do have a second version that is used for experiments..."

Some PR photos taken by John in March 2013



"...You will notice a couple of things different in these photos than in the photos in the article. First, tacked on to the Front Panel 365 pF variable cap is a mica trimmer cap.....this is of no significance as I was just messing with lowering the BW upper and lower frequencies to see what difference it would make....None is the answer..." *John, K5IRK.*



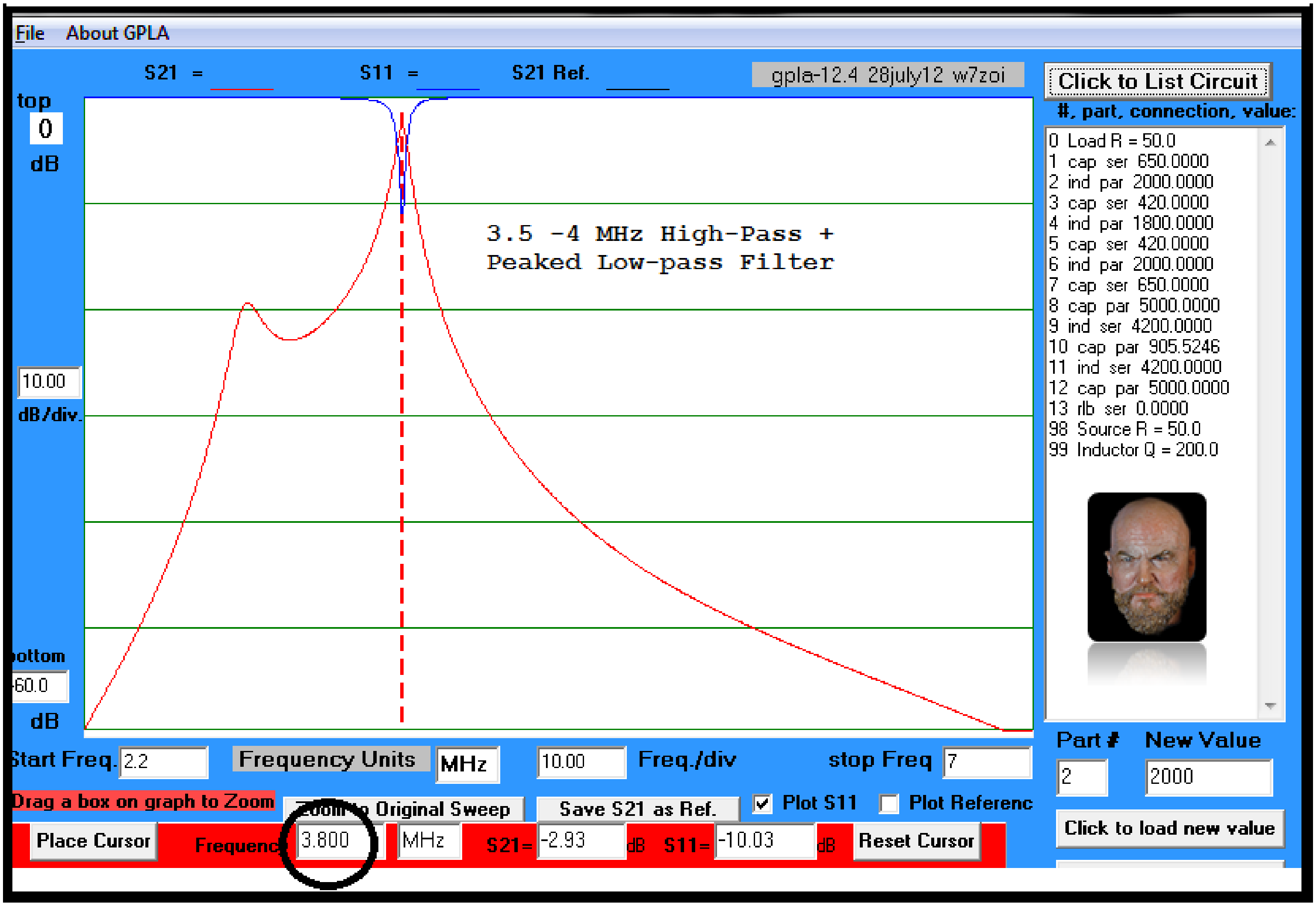


"...In the above photo you will see the addition of the 30 meter band to the RX with the diagonal front end filter board and the Xtal Oscillator attached to the BFO Box's wall that was added some years after the article was published…." *John, K5IRK.*

High-pass plus Peaked Low-pass Filter

If you know me, you know I love the peaked low-pass filter both at RF and AF. This adoration came from studying the PR and other work published and shared privately to me by Wes. *A good reference = The Peaked Lowpass: A Look at the ultraspherical filter by Wes for Ham Radio, June 1984.*

While the high-pass + peaked low-pass filter placed ahead of the second mixer in the PR wouldn't likely go in a modern W7ZOI design, its narrow-band LC filtration generates a roofing action when the receiver uses converters for bands other than 80 meters. The result is that much of the DR obtained in the single conversion version is retained in the dual conversion receivers.



top 0 dB

10.00 dB/div.

ottom -60.0 dB

tart Freq. 2.2

Frequency Units MHz

10.00 Freq./div

stop Freq 7

☑ Plot S11

☐ Plot Referenc

Frequenc 3.800 MHz

S21= -2.93 dB

S11= -10.03 dB

#, part, connection, value:

```
0  Load R = 50.0
1  cap  ser  650.0000
2  ind  par  2000.0000
3  cap  ser  420.0000
4  ind  par  1800.0000
5  cap  ser  420.0000
6  ind  par  2000.0000
7  cap  ser  650.0000
8  cap  par  5000.0000
9  ind  ser  4200.0000
10  cap  par  905.5246
11  ind  ser  4200.0000
12  cap  par  5000.0000
13  rlb  ser  0.0000
98  Source R = 50.0
99  Inductor Q = 200.0
```

Part # 2

New Value 2000

Above — A GPLA simulation of the front end high-pass/peaked low-pass filter centered for 3.8 MHz by tweaking capacitor #10 in the software. Click for another simulated filter that covers 3-4 MHz. Look at the sublime low and high-pass skirt action with a 3 dB bandwidth of 73 KHz. Love this!

Click for a screen shot of an entire 40 Meter band version shown with the low-pass filter section centered at 7.003 MHz. I also made 1 for WWV 10 MHz — by scaling the original version's XL and XC and tweaking with GPLA, you may build 1 for any HF band.

AF Amplifier

The post product detector AF chain sounds great. Many of us later replaced the Q5 mute switch with something quieter, however, even today — this AF block holds its own against most discrete component headphone-level circuits. I particularly love the crisp fidelity of the Class A feedback pair Q3 and Q4. Today, I would substitute a BD139 for Q4.

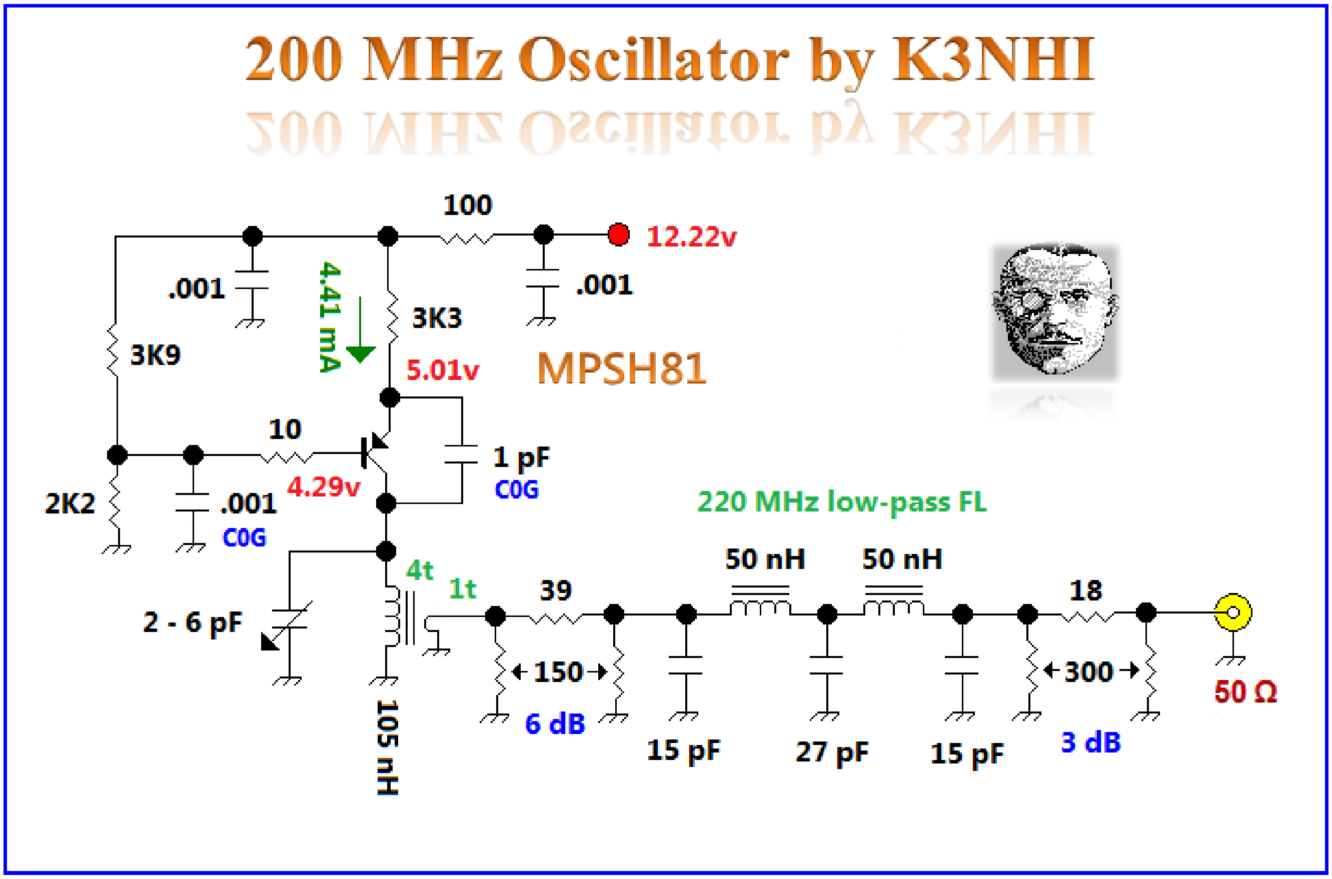
Wrap up

I could go on, but this web page is already too long. Studying + building stages from the PR; a receiver designed more than 3 decades ago, might raise your game today. Go team!

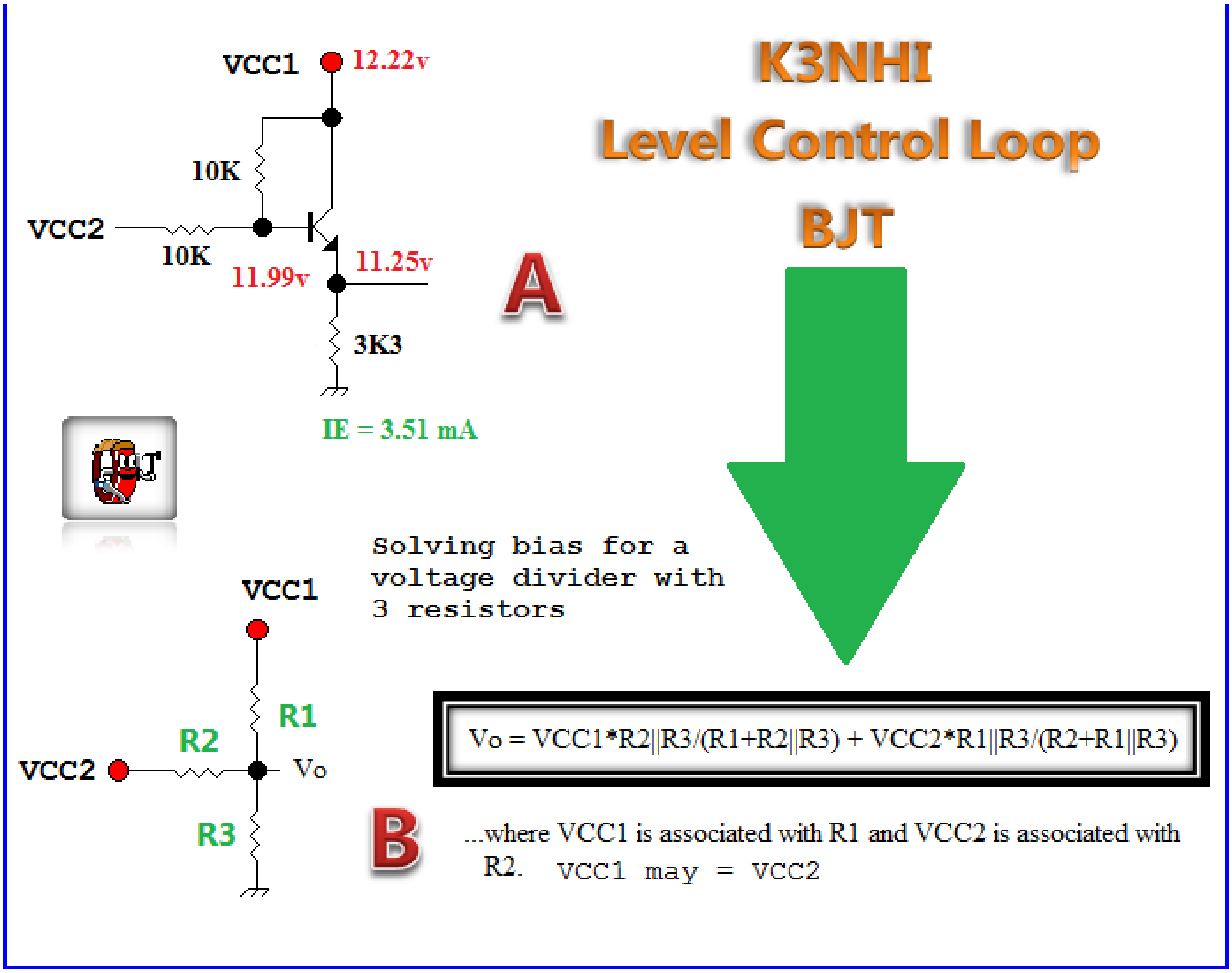
---

10. Miscellaneous Pictures and Figures

Above — I built a prototype 200 MHz oscillator with trailing low-pass filter before making the 210 MHz version placed in the 1-118 MHz VFO for the K3NHI sweep system. (Section 1). The secondary coil floats and may be positioned between any of the 4 primary links to change coupling and thus output power.



Above — The schematic of my 200 MHz local oscillator. Click for another version I designed that tuned from ~135-208 MHz.

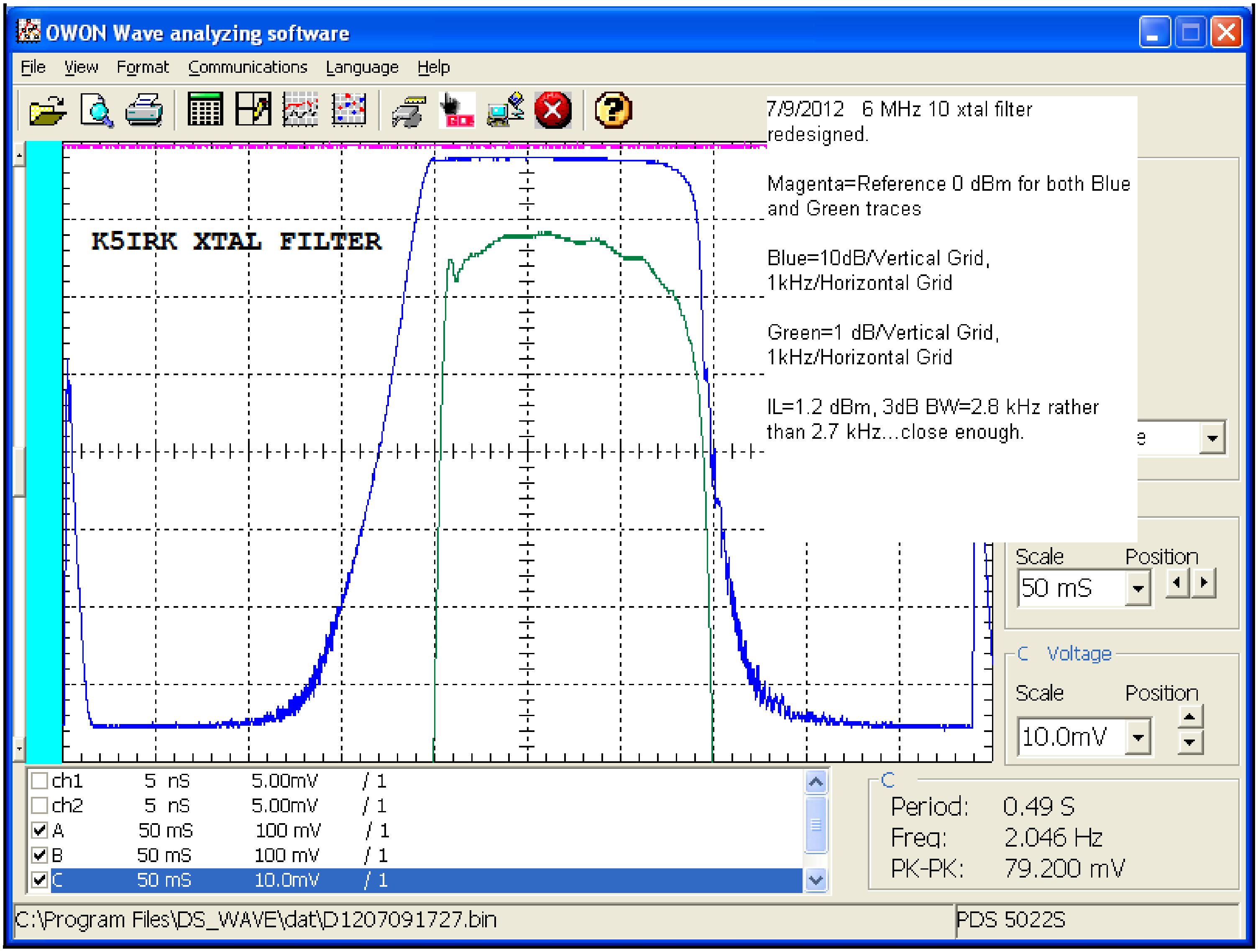


Solving bias for a voltage divider with 3 resistors

$$Vo = VCC1*R2||R3/(R1+R2||R3) + VCC2*R1||R3/(R2+R1||R3)$$

...where VCC1 is associated with R1 and VCC2 is associated with R2. VCC1 may = VCC2

Above — In the K3NHI 1-118 MHZ VCO lies a fabulous leveling circuit that involves a CMOS rail-to-rail op-amp controlling a BJT level shifter. To understand the bias of this BJT, I made a breadboard (A) and then a simple model (B) and developed the equation shown that involves 3 bias resistors. Bias or Vo is determined by superposition. VCC1 and VCC2 may be equal or not. VCC2 comes from the op-amp. I also learned it's a good thing to surround myself with smart people.



7/9/2012 6 MHz 10 xtal filter redesigned.

Magenta=Reference 0 dBm for both Blue and Green traces

Blue=10dB/Vertical Grid, 1kHz/Horizontal Grid

Green=1 dB/Vertical Grid, 1kHz/Horizontal Grid

IL=1.2 dBm, 3dB BW=2.8 kHz rather than 2.7 kHz...close enough.

☐ ch1	5 nS	5.00mV	/ 1
☐ ch2	5 nS	5.00mV	/ 1
☑ A	50 mS	100 mV	/ 1
☑ B	50 mS	100 mV	/ 1
☑ C	50 mS	10.0mV	/ 1

Scale 50 mS

Voltage Scale 10.0mV

C

Period: 0.49 S
Freq: 2.046 Hz
PK-PK: 79.200 mV

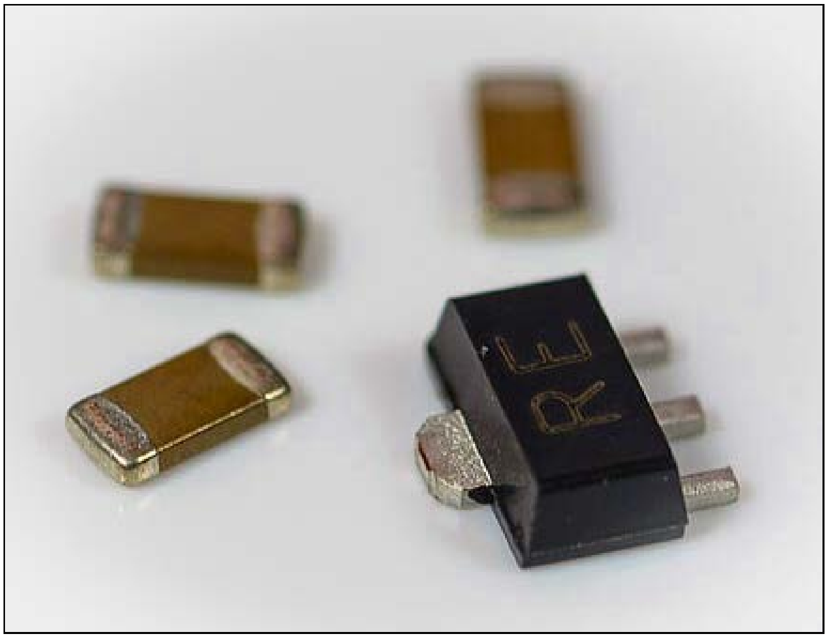
C:\Program Files\DS_WAVE\dat\D1207091727.bin

PDS 5022S

Above — John, K5IRQ designed and swept a 10 crystal SSB filter with an insertion loss of 1.2 dB in the K3NHI sweeper that I just had to show you. Click for his build up in Ladbuild08 and then GPLA08 from the EMRFD CD. Bob's xtal sweeper from Section 1 and Wes' software arm us experimenters with solid tools. Nothing can replace measurement and reason.

RF — Test and Measurement

VHF — Véronique



Repository for VHF experiments conducted in 2013 and 2014.

1. Ferrite beads
2. BFS17A transistor
3. 50 Ω MMIC Bench Amplifer
4. Breadboarding Double-Sided Copper Boards
5. Simple Preamp Filter
6. VHF Band-pass Filter Experiments
7. NE612 Mixer Diddy
8. Miscellaneous Photographs

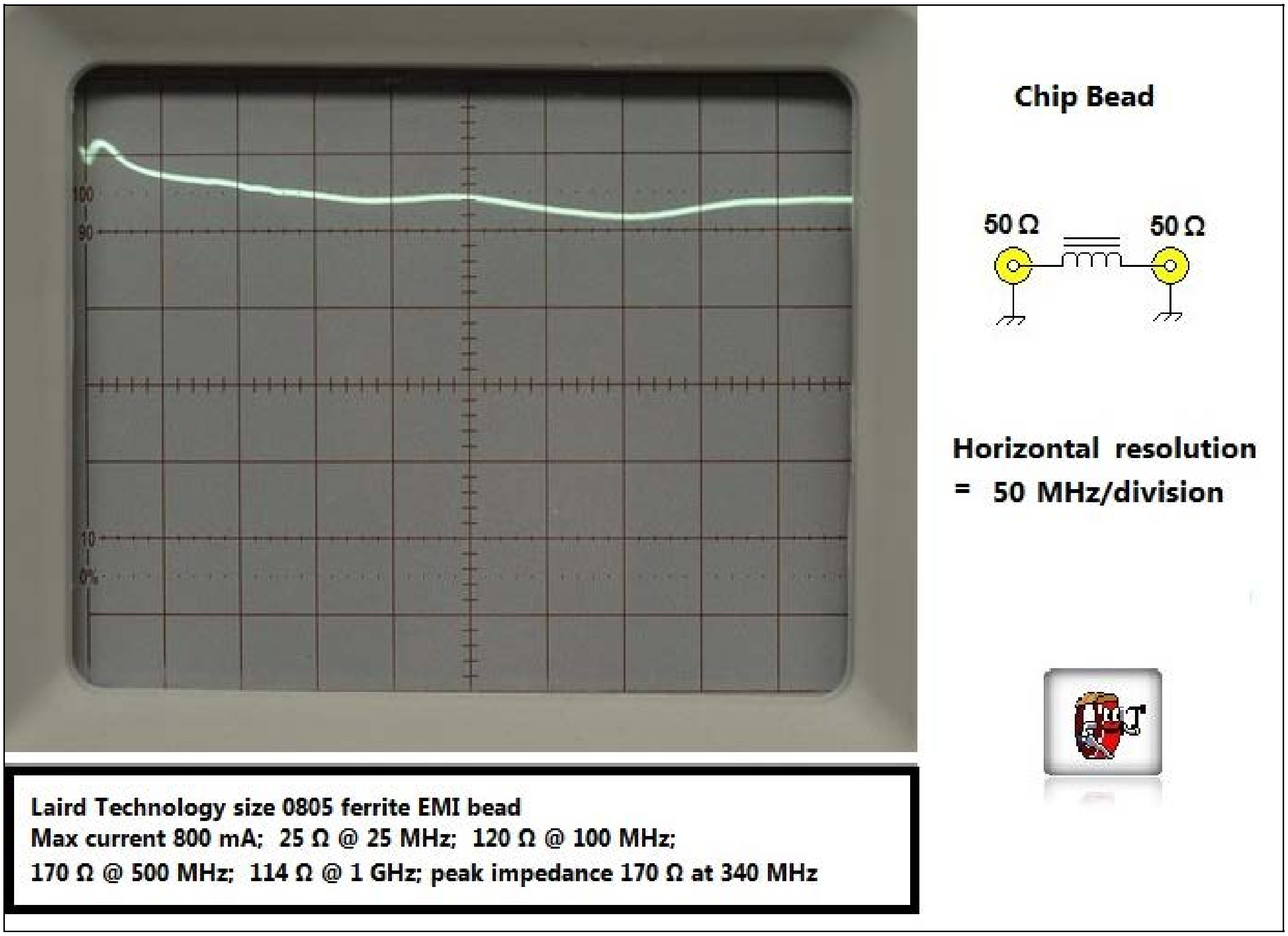
---

1. Ferrite Beads

I toiled to choose an SMT ferrite bead for my junk box. Selecting suitable chip beads not only troubles us, but even some engineers I've read — which to choose? Chip beads act as high frequency resistors that present a low impedance to DC plus our desired RF signal while impeding and dissipating UHF through resistive losses caused by minute heating of the ferrite material from eddy currents. SMT beads are typically rated by their maximum current, resistance at DC, and the impedance they present at 100 MHz.

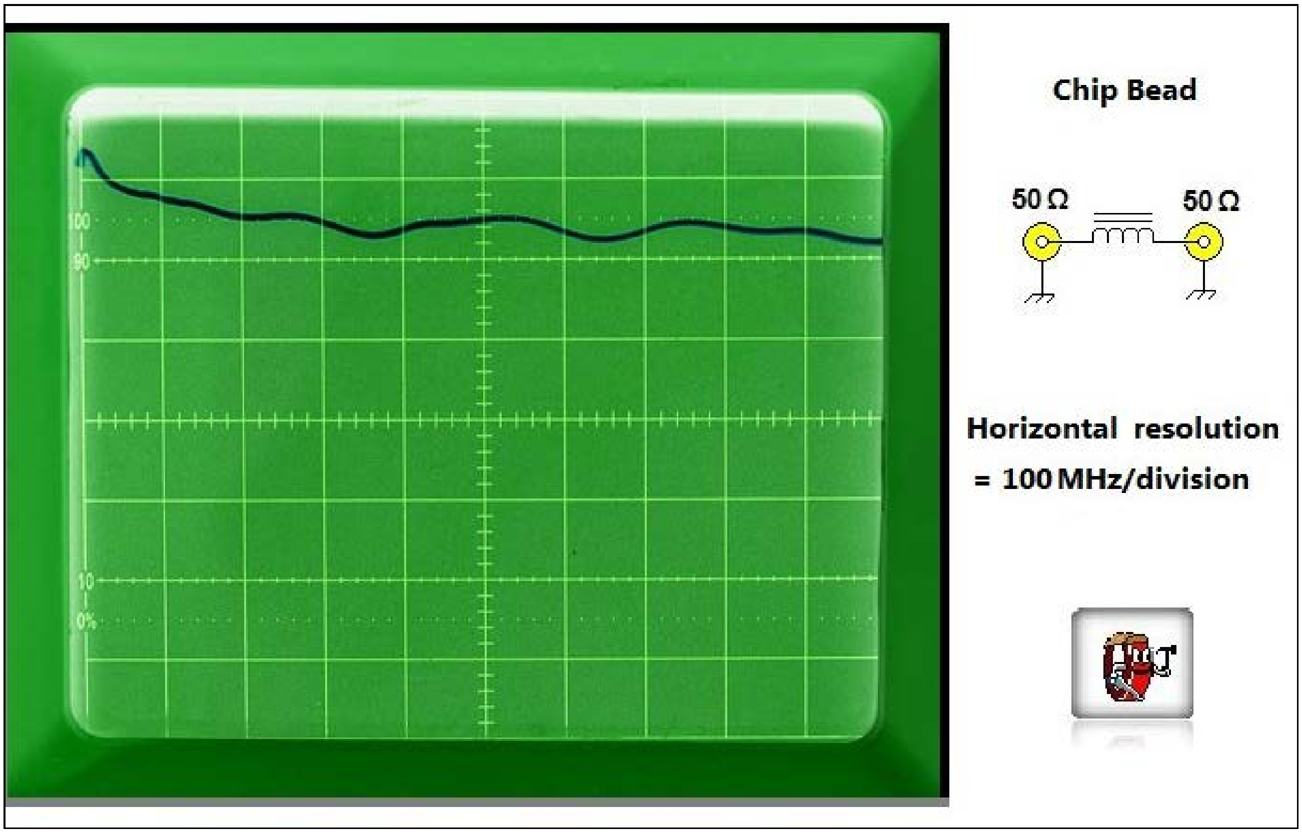
Since a ferrite bead's impedance is essentially resistive to parasitic UHF oscillations, I remembered that the model for a bead is actually an inductor paralleled by a resistor. It's the "resistor" we're after, for that's the extra load that tends to stabilize the amplifier. From past success in UHF supression with low-value (10-51 Ω) resistors, I chose my SMT bead to have a relatively low Z at 100 MHz. Consider, too, that many chip beads sold today serve as RFI suppressors for high speed digital lines where the Z at 100 MHz and maximum current lie well above my requirements.

I found a chip bead that appealed to me on eBay and bought 50: size 0805; 800 mA; with a Z at 100 MHz of 120 Ω.



Laird Technology size 0805 ferrite EMI bead
Max current 800 mA; 25 Ω @ 25 MHz; 120 Ω @ 100 MHz;
170 Ω @ 500 MHz; 114 Ω @ 1 GHz; peak impedance 170 Ω at 340 MHz

Above — A sweep of my bead from 1- 500 MHz. Sadly, I did not perfectly center the sweep at 250 MHz, but the peak impedance occurred close to the specified 340 MHz.

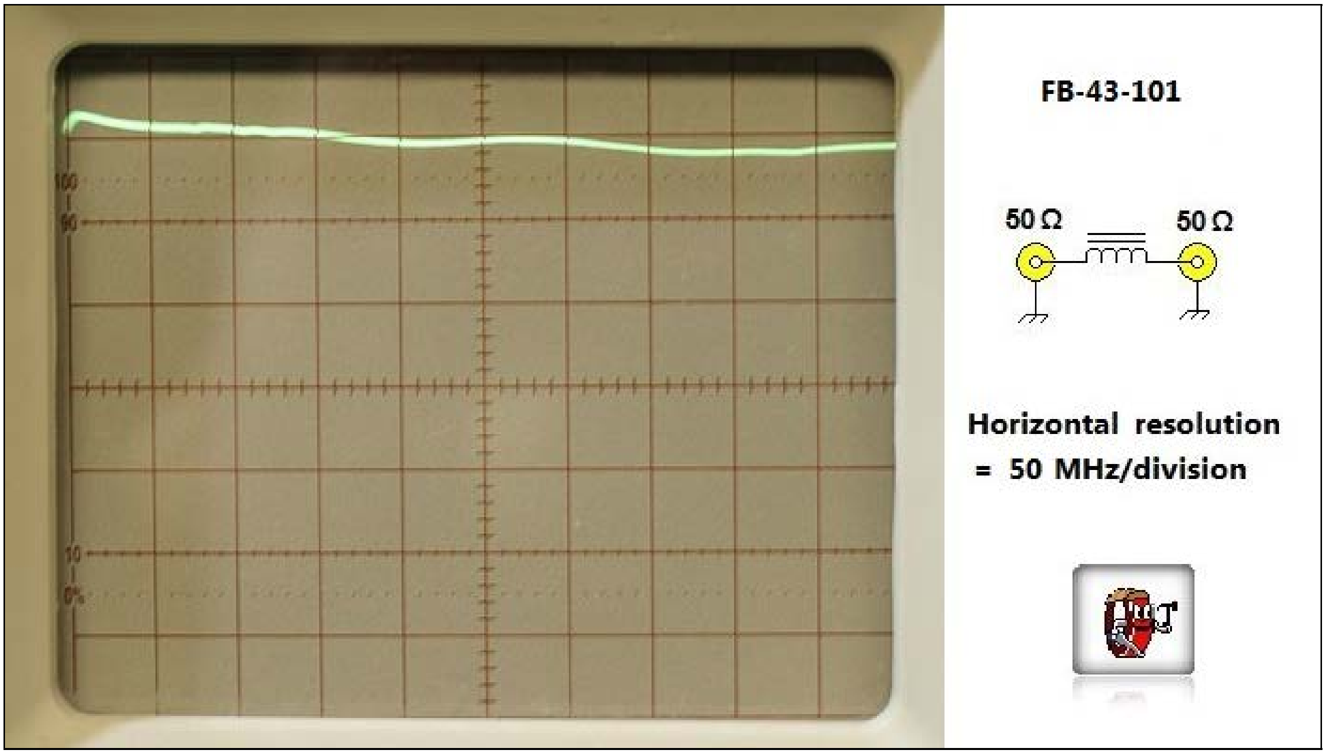


Above — I swept my Laird techlogies chip bead out to 1 GHz. Ultimately, I'll have to verify its function with in-situ experiments. For example, before and after chip bead analysis of a low noise amplifier that's oscillating at UHF.

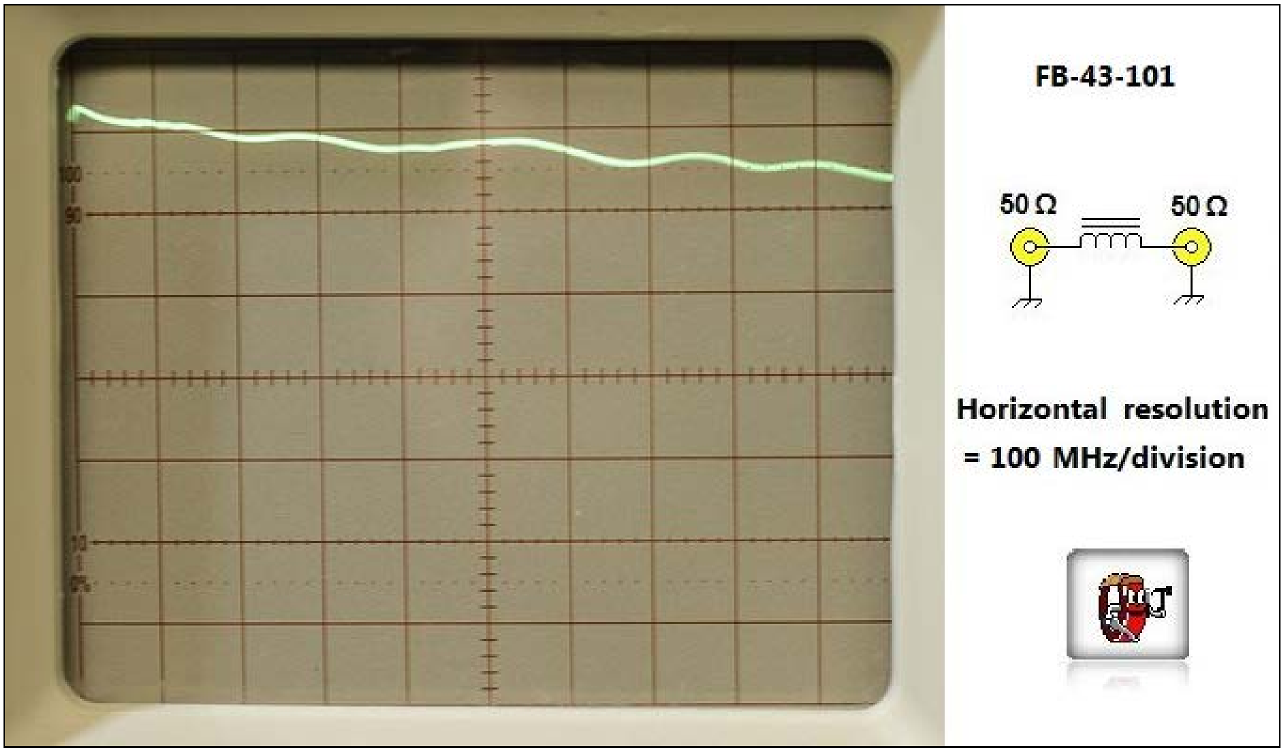
FB43-101

If you sweep the floor under my QRP work bench, you'll find 2N3904s, J310s and a few FB-43-101s in the dust pan. In our hobby, FB-43-101s are common as fleas on a dog; many builders like to stick them on the drain, collector, or base/gate lead(s) of an active part to squash UHF oscillations. I've never seen a datasheet for this part and decided to sweep 1 for reference purposes.

2 sweeps of the FB-43-101 follow:



Above — A 1 - 500 MHz sweep of the FB-43-101 slide-on bead. This tracing shows very subtle attenuation that peaks between 400 and 450 MHz. Probably this minimal attenuation at UHF provides the reason we often need to put 2 on the drain or collector of our amplifiers?



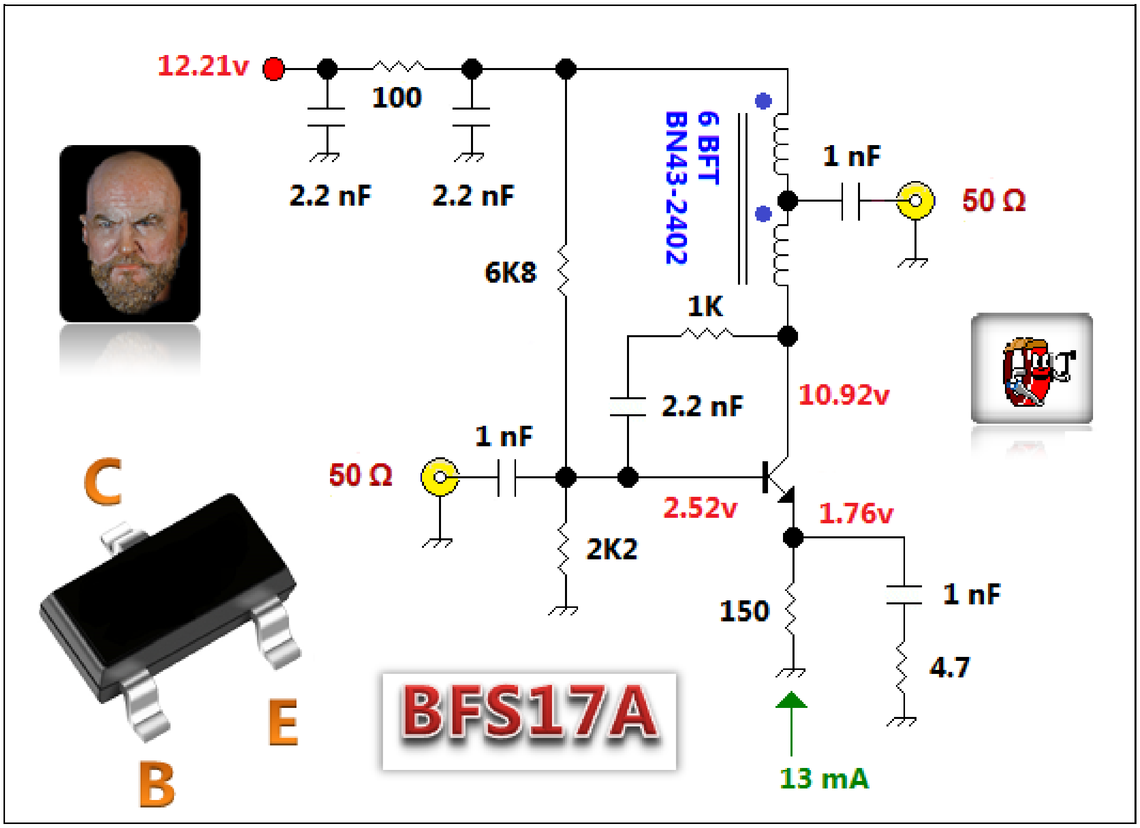
Above — A 1 - 1000 MHz sweep of the FB-43-101.

---

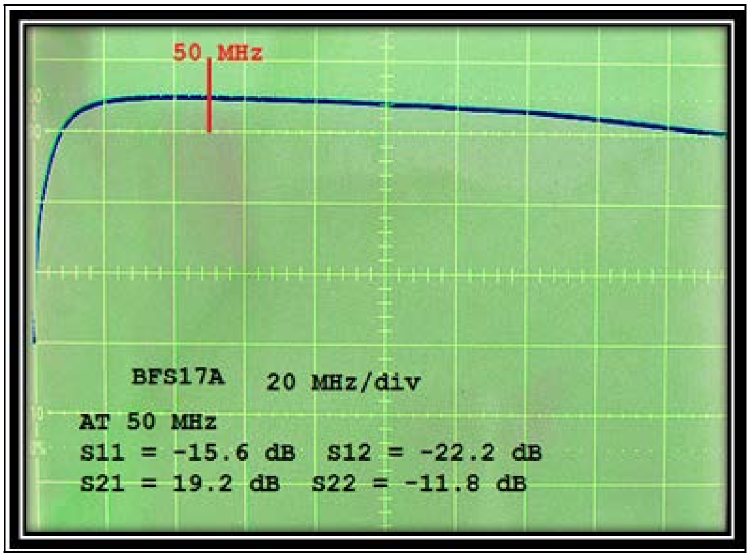
2. BFS17A Transistor

I sought a surface mount NPN transistor to supplant my leaded transistors such as the PN5179, BF199 or MPSH10. Serving as a general purpose transistors biased for ~5 to 20 mA collector current — they'll work as buffers + oscillators + as the BJT for hybrid-cascode amplifiers built with a SMT J310 on the bottom.

My requirements were SOT-23 (3 well *seperated* leads allow easily carved islands in a copper board with a dremel tool); an fT that's not too high to help reduce higher UHF oscillations; and a noise figure < 3. Other factors included price and availability. We enjoy many choices with such criteria, but I settled on the BFS17A. Click for the data sheet . In summary, it's a 2.8 GHz part with a NF of 2.5 dB at 800 MHz.

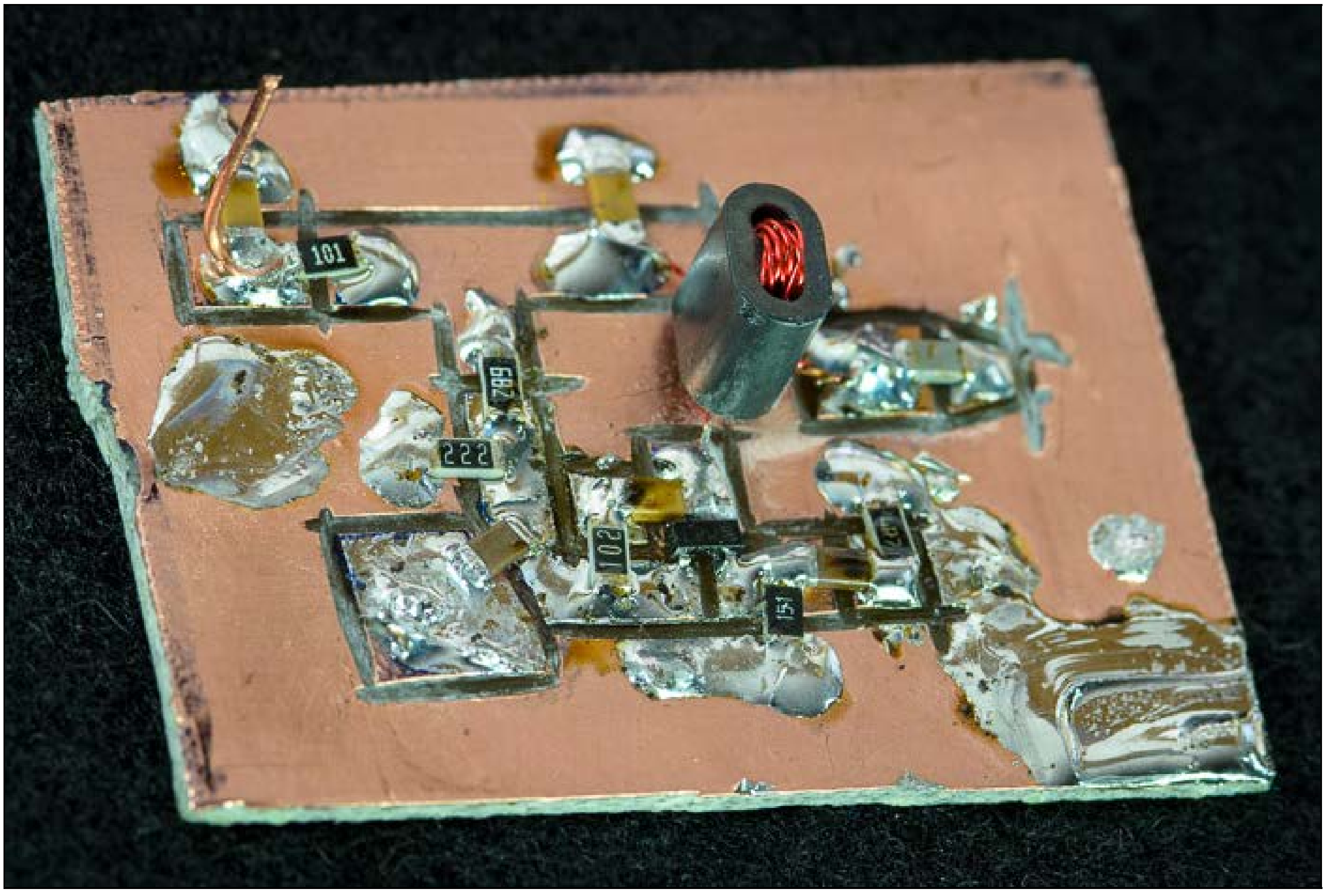


AAbove — My test schematic; a simple feedback amp with capacitors oriented to 50 MHz and above. A limited selection of SMT resistors constrained my experients, but the results seem okay. I strove for the best possible input and output return loss at 50 MHz. I wanted a emitter current of ~ 12 mA and got 13 mA with the bias resistors on-hand.



Above — A 200 MHz wide sweep of my breadboard with tracking generator + spectrum analyzer. Please view the various 2-port measurements at 50 MHz in the lower half. Despite some input and output port mismatching, a gain of 19.2 dB was measured at 50 MHz.

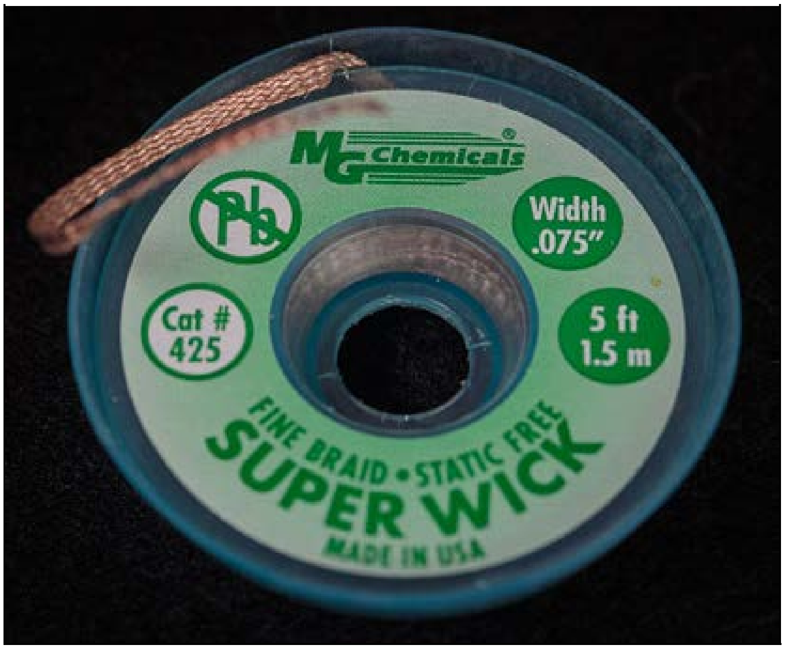
I wound my output transformer within a BN43-2402 binocular ferrite, but ordered some #61 material equivalents for assessment at VHF.



Above — My magnified breadboard looking messy after many parts substitutions from experiments to find the best possible S11 and S22. I removed the shiny RF connectors prior to photography since they often create blown highlights. Alternate image. Despite a limited RF bypass before and after the 100 Ω decoupling resistor + crude bread boarding, I measured no parasitic oscillations from 0.001 to 1 GHz — likely due to the heavy shunt feedback. I liked this transistor, found it easy to breadboard and ordered 100. You'll see the BFS17A in my experiments hereafter.

By all means, order whatever transistor *you* want. Past emailers wrote to say they enjoy reading how other experimenters think about and assess their parts such as RF beads or transistors.

Surface Mount Part Soldering



Above — Solder wick. On my SMT workbench, wick proves the most important tool period. A piece of desoldering wick can transform a monster mistake into a perfect circuit in under 2 seconds.

I've read many accounts of how builders anchor the parts they're soldering. We need to keep the SMT part flat on the board to avoid mechanical tension and poor solder joints — a system is required. For size 0805 and 1206 parts, I often just use my index finger fingernail to hold the part in place while I solder 1 lead with my other hand. After a few seconds of cooling, I solder the other lead and then usually touch up the first joint. For tricky formats like SOT-23, or SOT-143, I usually tape down the part while ensuring even pressure on all leads and just solder 1 lead. After cooling, I remove the tape and easily solder the other leads. Again, the intial lead may require some retouching.

SMT Size

Above — A 5X magnified roll of 50 BFS17A in SOT-23. Compact or what?

Above — BFS17A in SOT-23

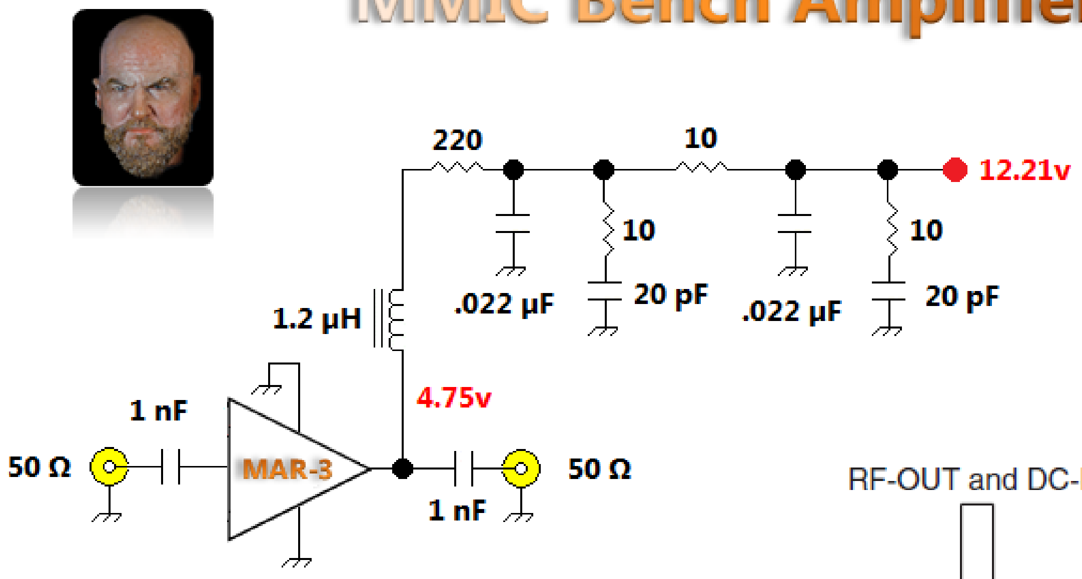
---

3. 50 Ω MMIC Bench Amplifer

To boost low-level signals during experiments, every 50 Ω workbench needs a utility RF amp module. With strong wideband gain + RF port return loss (S11 and S22), MMICs make a good choice. Since I own 15 pieces, I built an amp around the MCL MAR-3, a modest gain + noise figure, 2 GHz plastic part in the Micro-X package. We enjoy a bevy of MMICS to choose from and as they increase in fT and gain, so does the need for solid, low inductance grounding techniques.

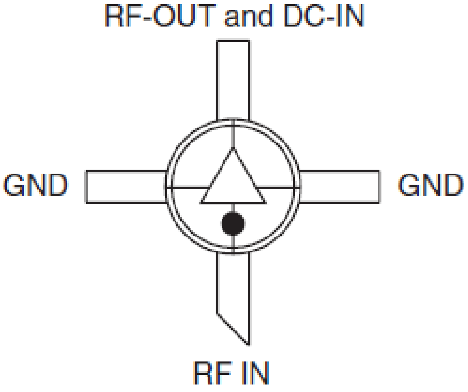
Above — The completed amplifer module in a Hammond box with an RCA jack for the DC voltage.

MMIC Bench Amplifier



Frequency	S21
7 MHz	11 dB
14 MHz	11.2 dB
50 MHz	11.7 dB
100 MHz	12.1 dB
144 MHz	11.4 dB

I = 32.5 mA



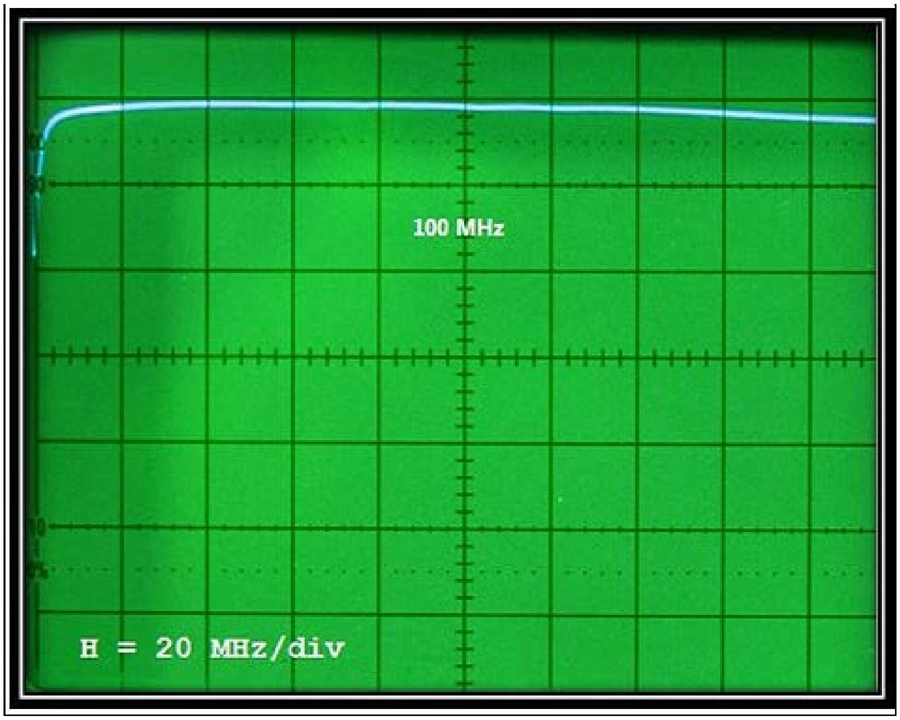
Above — Schematic with a table showing gain versus frequency. I built all SMT (size 0805 mostly) except for *short* leaded 1 nF input and output coupling capacitors. I applied sturdy wideband DC filtering since I can't risk RF flowing on my DC lines during bench experiments. I measured no oscillations from 100 KHz to 1.5 GHz.

I chose a 1.2 µH SMT choke in an attempt to boost gain from ~66 to 150 MHz. Many builders leave off the choke, but some RF will flow through the 220 Ω bias resistor to ground and thus, to get maximum gain, a high impedance RFC might help. Recall this RFC should exert an inductive reactance of at least 500 Ω (>= 10 X the load impedance) at the lowest amplifer operating frequency; in my case this occurs at ~66 MHz. A better choice for the RFC might be 1.6 µH which exerts ~ 500 Ω at 50 MHz, but I just own size 0805 1.0 and 1.2 µH chip inductors.

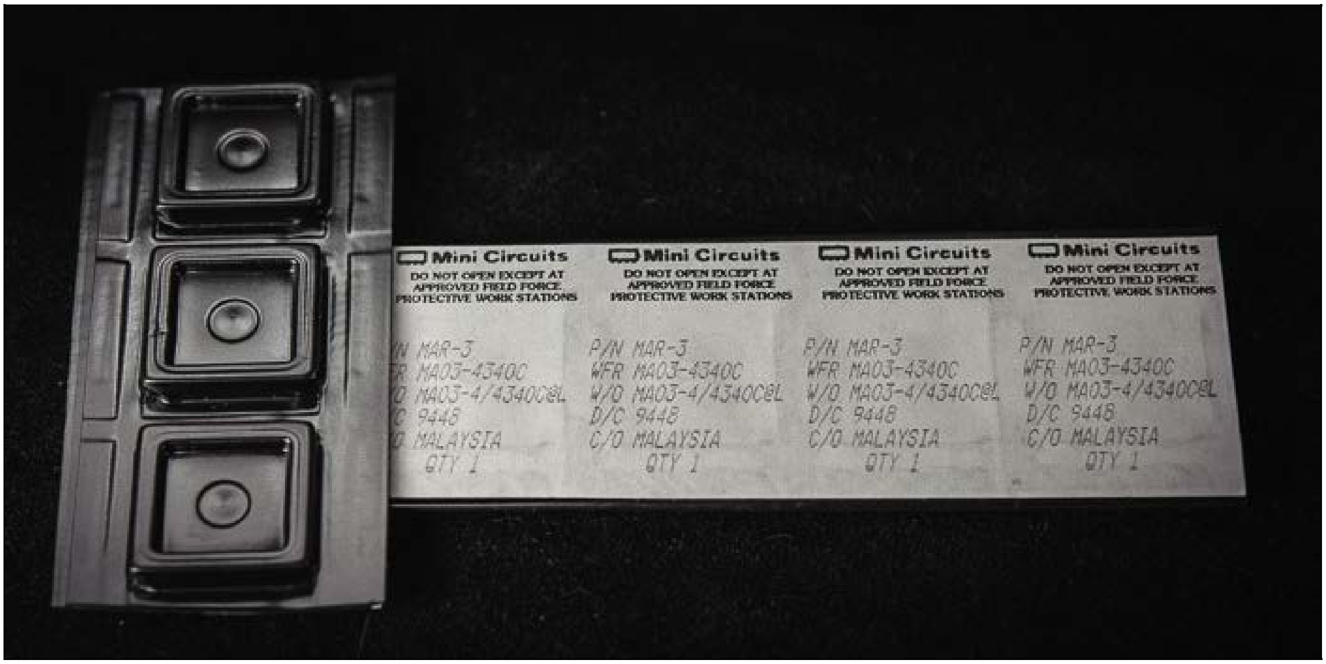
My particular amp delivers ~10 to 12 dB gain from about 5 to 180 MHz. Since the coupling capacitor value determines the low frequency response, if you want a MMIC amp for mostly HF, try reducing your input coupling caps from 0.001 to 0.01 µF or so. You mighty increase the RFC to say 10 µH as well — that's why we don't just build kits; design to suit *your* needs.

The 220 Ω bias resistor should really be 200 Ω for a VCC of 12.2v, however, I had to substitute the 220 Ω R due to a low selection of SMT parts. This dropped the current and also my gain at 100 MHz = 0.4 dB less than that specified in the MAR-3 datasheet — no big deal.

I built the bread board on 2-sided FR4 copper clad board. I soldered thin copper foil around all 4 edges to adjoin the top and bottom copper surfaces. Around each MMIC ground lead, I drilled 4 via holes and soldered copper wire from top to bottom. 2 vias were positioned near every bypass capacitor and all along the input and output paths. 23 total via wires.



Above — Tracking generator plus spectrum analyzer sweep out to 200 MHz. Click for a sweep from 1 to 10 MHz showing a rapid fall off below 5 MHz. This little amp will serve me well for most HF and especially my VHF experiments. A UHF amplifier module is planned and all knowledge gained from VHF circuit building will flow forward.

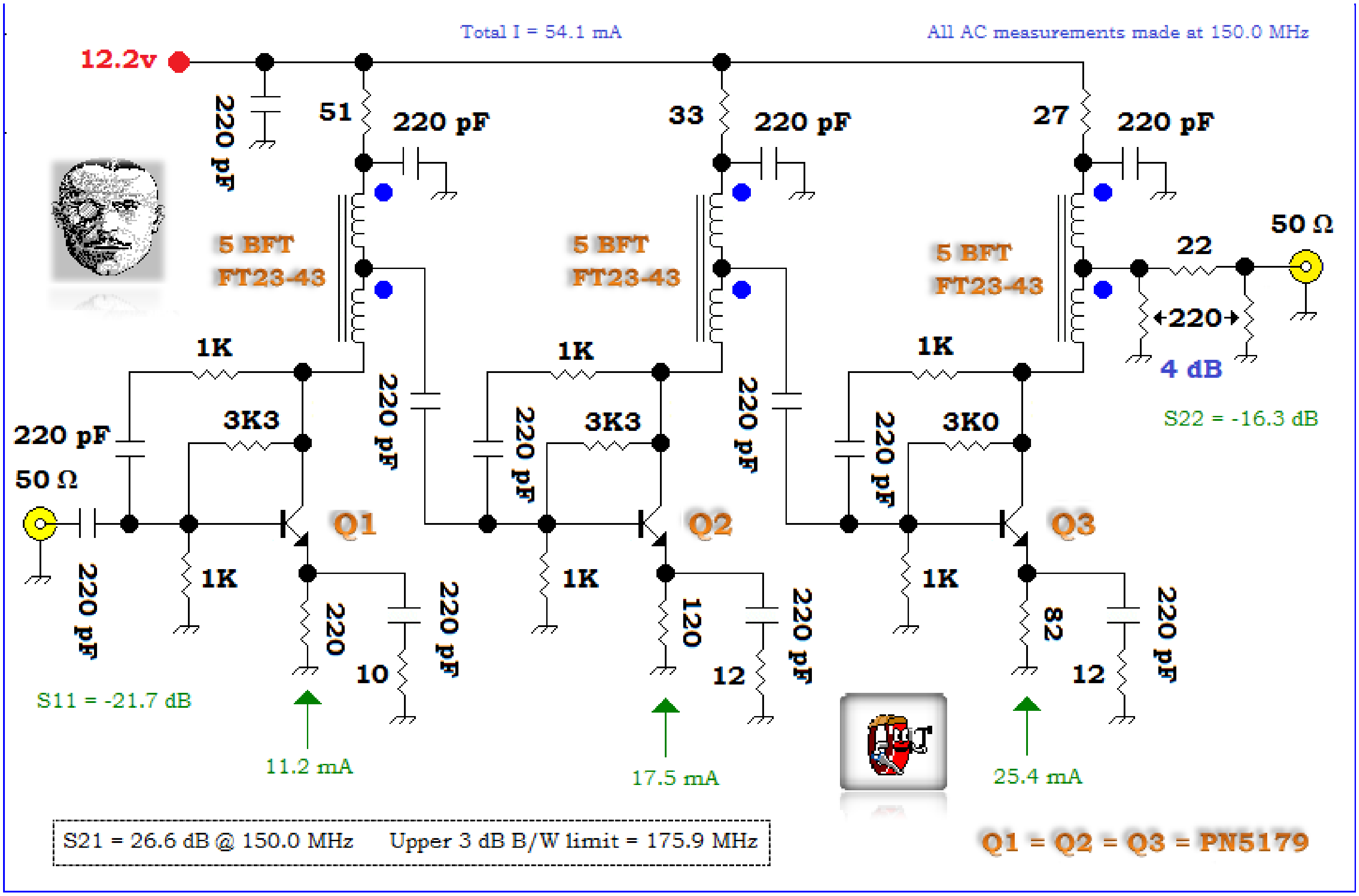


Above — Part of a strip of MCL MAR-3 amplifiers.

Feedback Amps (FBAs)

MMICs like the MAR-3 use a Darlington feedback pair. Still, too, we shouldn't write-off discrete component FBAs wielding transformers + shunt and series feedback at VHF. These amps; staples of countless W7ZOI and W1FB projects since the 1970s; evolved commercially into GaAs FBAs built on a tiny wafer by companies like TriQuint Semiconductor.

In broadband amplifiers, negative feedback permits a wideband (flat) gain response, reduces input and output VSWR (S21 and S22) and controls performance changes from S-parameter and other variations from transistor to transistor. In 2013, I needed an amp chain with >=25 dB gain at 144-165 MHz [~150 MHz mostly] for low level stuff : -25 to -10 dBm input power. Inspired by Wes' 144 MHz CW/DSB transmitter FBA chain *(**SSD** -- Chapter 8 -- Figure 30 in Solid Stage Design for the Radio Amateur; ARRL published in 1986 and out of print)*; I cascaded 3 PN5179 BJTs as FBAs:



Above — My 150 MHz (design frequency) FBA module. Since the fT of the PN5179 = 900 to 2000 MHz @ a 100 MHz test frequency, gain drops steadily as we move up above 100 MHz. Centered at 150 MHz, my upper 3 dB drop off point was 175.9 MHz — thus this amp works okay from 144-165 MHz where I need it.

I built and tweaked each stage to derive the best S11 and S22 . *Click for a snapshot taken of my test of Q1 with a BNC connector attached to both ports — I did this for each stage.* It's a trade off since if you boost the emitter degeneration R, the S11 will improve but the gain decreases. Typical stage S11 = -18.5 dB and S22 = -17 dB. I progressively bumped up the current in each stage to reduce gain compression. I get ~12-13 dBm power out of the whole amplifer before compression/distortion occurs.

Securing the best S11 and S22 for each stages might seem silly, but invoked learning. The S11 and S22 of the *total amplifer* counts the most. An interesting thing happened to S22 – with a RF connector soldered right on the board without the pad, S22 = -23.8 dB. When I added the 4 dB pad and stuck it in the box and then ran a ~ 5mm copper wire to connect an SMA output jack, the S22 decreased to -16.3 dB! At 150 MHz — life differs from 7 MHz.

Placing the 4 dB pad between stage 2 and 3 will stabilize + enhance S22, gives a 4 dB higher 1 dB compression point and output intercept, which may make a FBA chain more useful for experiments like transmit driver applications. I can report I measured no oscillations out to 1.5 GHz and this amp works okay for my intended purpose with gain of 26.6 dB at 150 MHz.

I employed 220 pF caps because the plus the self resonant frequency of my 220 pF Murata caps = 154.6 MHz. Therefore, gain at lower HF is low.

Above — FBA breadboard photograph.

QRP — Posdata for October 27, 2013

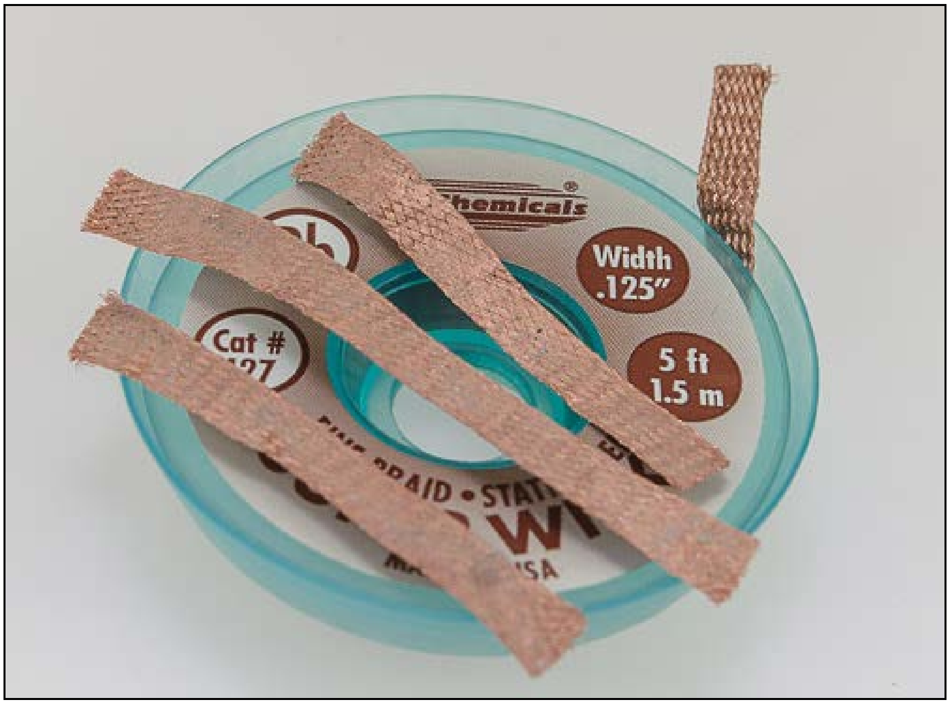
I first learned radio electronics from SSD, EMRFD and the ARRL Handbook and still remember the joy when my copy of SSD arrived in 1990. Feedback amplifiers litter this book — and for good reason — they offer stable 50 Ω blocks for building up our RF signal. Wes' 144 MHz CW/DSB transmitter still intrigues me and inspired the circuit shown above.

I asked Wes if he kept that 2M transmitter, and if so, could he snap a couple of photos for me? Wes opened it up to see that the VXO and the frequency multiplier chain were missing. Click Click Click . Thanks to Wes, W7ZOI for the photos.

---

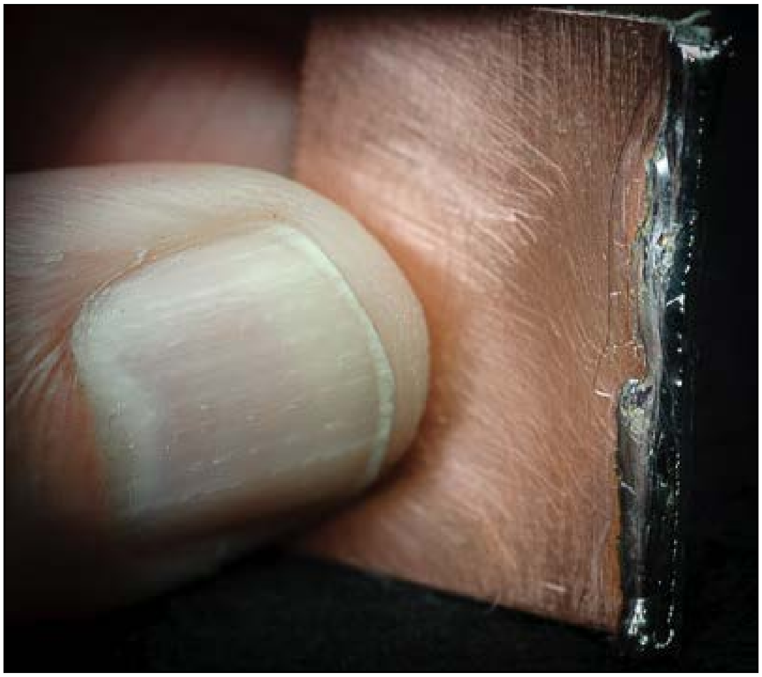
4. Breadboarding Double-Sided Copper Boards

Seeking a low inductance AC ground for some of my VHF and UHF FR-4 prototype breadboards, I join the top and bottom copper surfaces with solder wick along its 4 edges and copper wire vias in the main board area.



Above — A bench staple, solder wick, joins the 4 edges in true Ugy fashion. Many prefer copper foil for this task, but it's not cheap, nor readily available prompting a pragmatic approach. I take a hammer and pound the solder wick [0.125 inch minimum] so it becomes wider and thinner.

Above — Take a piece of wick and tack it in 2-4 places along the top surface of the copper board. The center solder shown above looks perfect — too much solder will flow into the wick and make a difficult bend job around the board edge. Tack just enough wick so you have enough left to go over the edge and to solder on other side of the copper board.

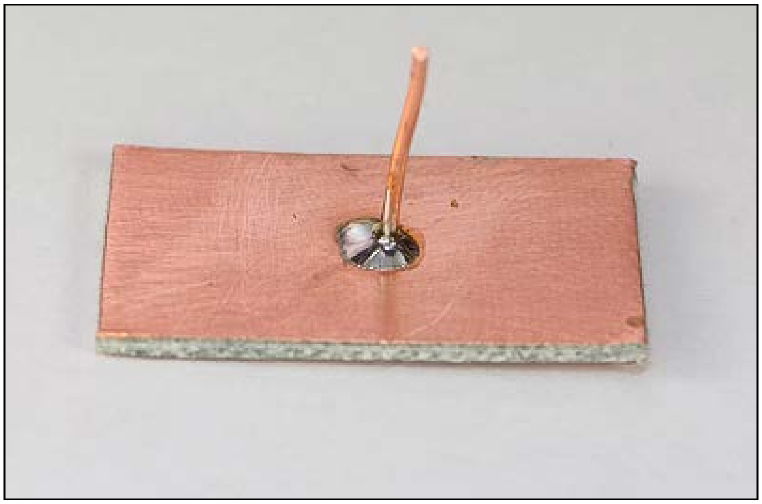


Above — With hand and/or pliers manipulate the wick around the other side to completely bridge the top to bottom. Finally, liberally solder the wick on both surfaces, move to the next board edge and repeat. Click for a rather messy UHF board with copper wick on 2 edges and numerous vias around a prescaler chip, some input + DC lines and a switch.

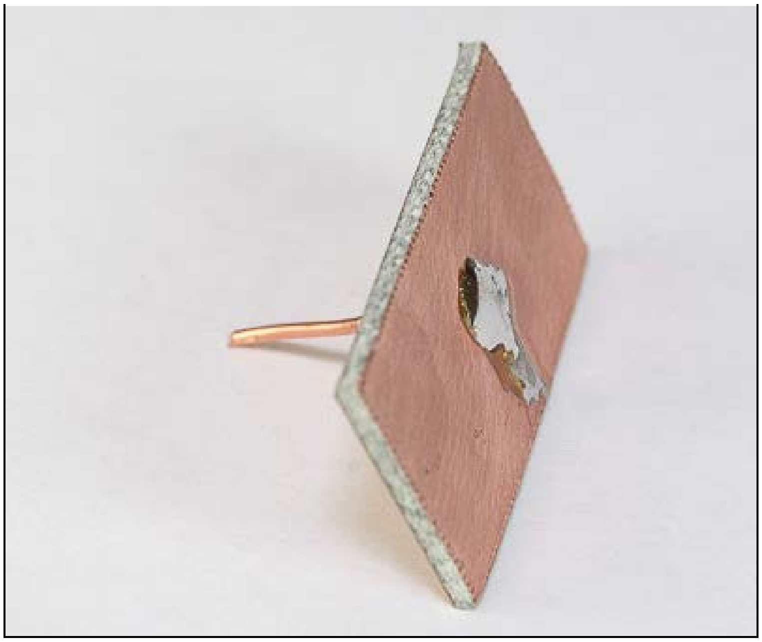
Above HF, most don't make pretty prototype boards to admire — rather, builders strive for good AC grounding and generally stick their board in a metal box for shielding and this hides the breadboard. No one's ever told me I make pretty circuits for any frequency and I'm okay with it.

Onto wire vias:

Above — Your drill chuck will need to accommodate small bits such as those shown above (3/64 inches = 1.2 mm) . I keep these and others that vary from 0.5 mm to 1.2 mm for making via holes.



Above — Magnified via wire soldered at the top side. I found making a hole close to the same size of the via wire will hold the wire in place for rapid soldering. This wire is normally clipped flush at this point. Click for a photo of my flush cutters.



Above — The board shown above was flipped over to solder the bottom copper surface. I try to drill and solder in all vias prior to soldering active devices, since the board heat may damage some parts.

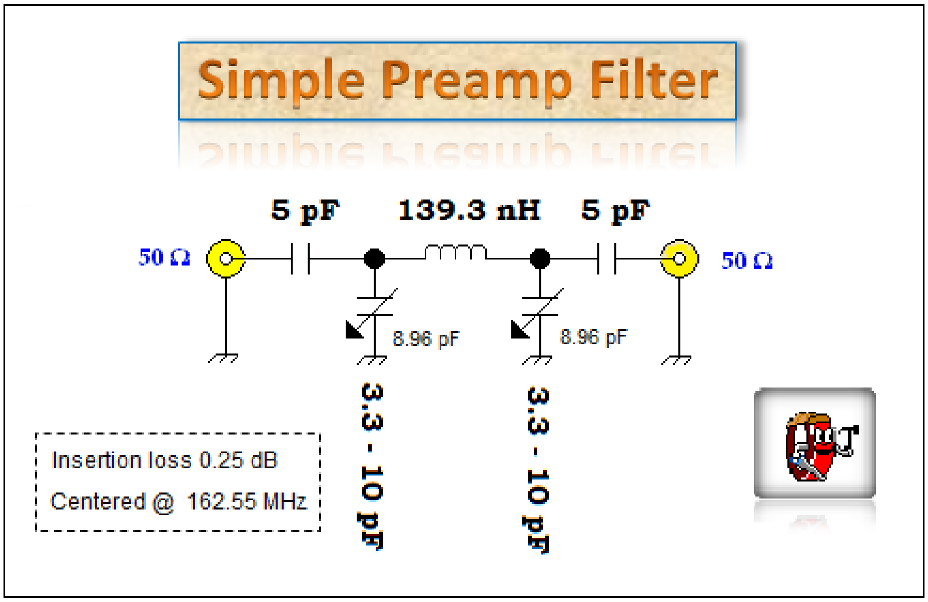
---

5. Simple Preamp Filter

Some RF filtering should proceed our receiver low noise amplifer (LNA) input — but what? Low-pass, high-pass, band-pass, 2 poles, many poles? Factors informing this decision might include our receive frequencies, QRM, selection of IF and thus our image stripping requirements, plus maximum tolerable signal loss before the LNA. We might not want image rejection from this filter since the post LNA filter can tackle this function.

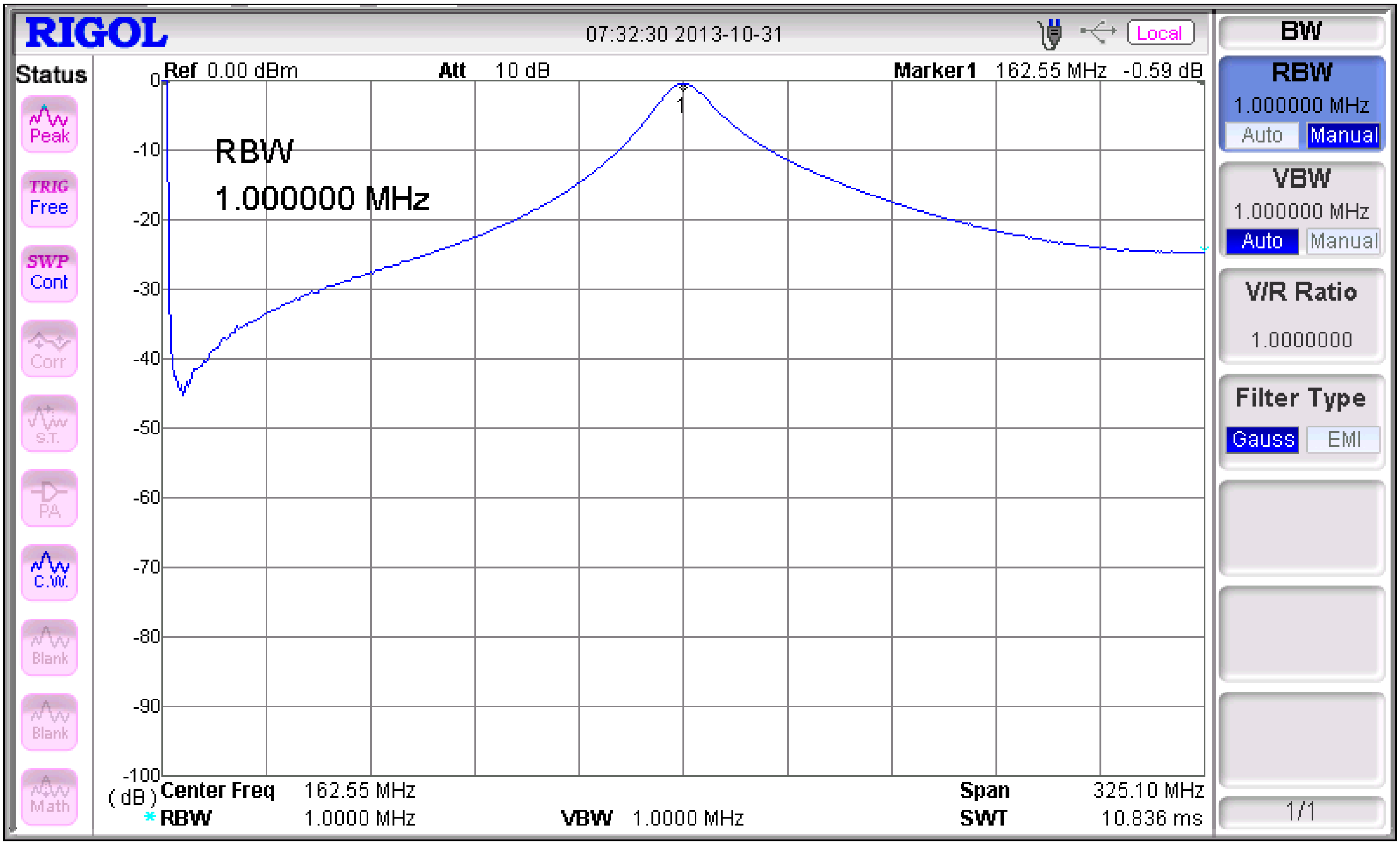
I examined an input filter presented by Joe Reisert, W1JR in the November 1986 Ham Radio. Joe wrote a fantastic column called VHF/UHF World and his context was a high dynamic range 2M receive converter to a 28 MHz IF.

I'm uncertain how to classify this filter: it looks like a standard pi low-pass filter except the input and output series capacitors exert a high-pass skirt. Joe built his filter for 144 MHz. After "building" Joe's filter in Ladbuild08 I tweaked the filter in GPLA *[software from EMRFD]* to center it at our local NOAA weather channel: 162.55 MHz. Click for the GPLA plot.



Above — Input filter schematic with values shown to center this rather symmetrical filter at 162.55 MHz. The half power (3 dB down) bandwidth = 18 MHz. You can easily scale it to other other frequencies — by tweaking the trimmer caps and perhaps squishing or expanding the inductor links some, this filter will tune widely including the 2 Meter Ham band. The IL blew me away (better than calculated) — the resonator Q was high due to the air wound L and Q >= 700 trimmer caps along with good port matching.

Click to view a larger photo of the breadboard. I wound the coil on a 3/8 inch diameter bolt but then spread the turns to get the L and length I wanted. Built on double-sided copper clad board, a few copper wire vias connect top to bottom. I joined the LC circuit to its ports with leaded 5 pF ceramic caps.



07:32:30 2013-10-31

Ref 0.00 dBm   Att 10 dB   Marker1 162.55 MHz -0.59 dB

RBW 1.000000 MHz

Center Freq 162.55 MHz   Span 325.10 MHz

RBW 1.0000 MHz   VBW 1.0000 MHz   SWT 10.836 ms

RBW 1.000000 MHz (Manual)

VBW 1.000000 MHz (Auto)

V/R Ratio 1.0000000

Above — Tracking generator plus spectrum analyzer sweep. Under sweep set up, I minutely squished or expanded the L and tweaked the caps to peak the filter at 162.55 MHz while obtaining the lowest insertion loss. Further, I had to tweak the trimmer capacitors to center the filter with the Hammond lid on since putting the lid on changed the center frequency slightly. A *TG equipped* spectrum analyzer proves the ultimate bench tool for VHF and UHF in my humble opinion.

Above — Zero calibration of my AADE L/C IIB meter. By sweeping numerous LC circuits after measuring the L with this tool, I've learned it's reasonably accurate with low inductance coils. I zero the meter prior to *each* measurement and zero calibrate it with the alligator clips butted end-to-end while gripping a small piece of copper wire. Try to keep the same relative alligator clip position during coil measurement. I'm uncertain if this is protocol, but it seems to work for me.

Thanks to Joe, W1JR for this circuit and the opportunity to learn more about component-level VHF design and construction.

---

6. VHF Band-pass Filter Experiments

Build Notes

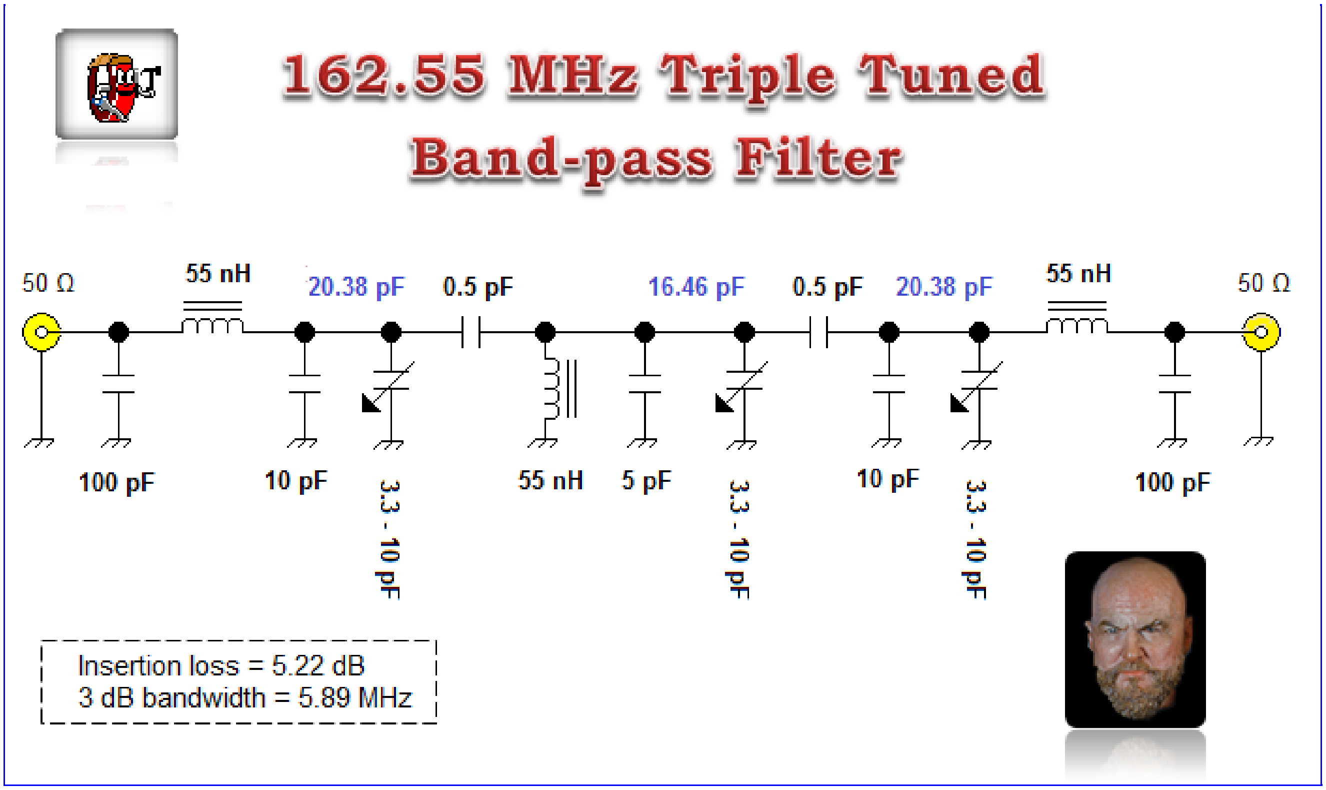
1. All filters designed with the EMRFD ladpac programs DTC08, or TTC08 and then tweaked in GPLA.

2. Seeking a band-pass filter for my experiments with the local NOAA weather channel at 162.55 MHz, I designed, built and tested 3 filters. With a 10.7 MHz IF superhet receiver, the image frequency = 141.15 MHz.

3. My VHF sweep system = a tracking generator plus a spectrum analyzer.

Filter 1

To avoid the need for proper electromagnetic/electrostatic shielding, I kept my HF experimenter hat on and built my first filter with powdered iron toroids (T30-12) knowing I would pay a Qu penalty that may wreck filter insertion loss, bandwidth and port matching. I hoped that the passive electromagnetic shielding offered by toroidal inductors would reduce resonator coupling.

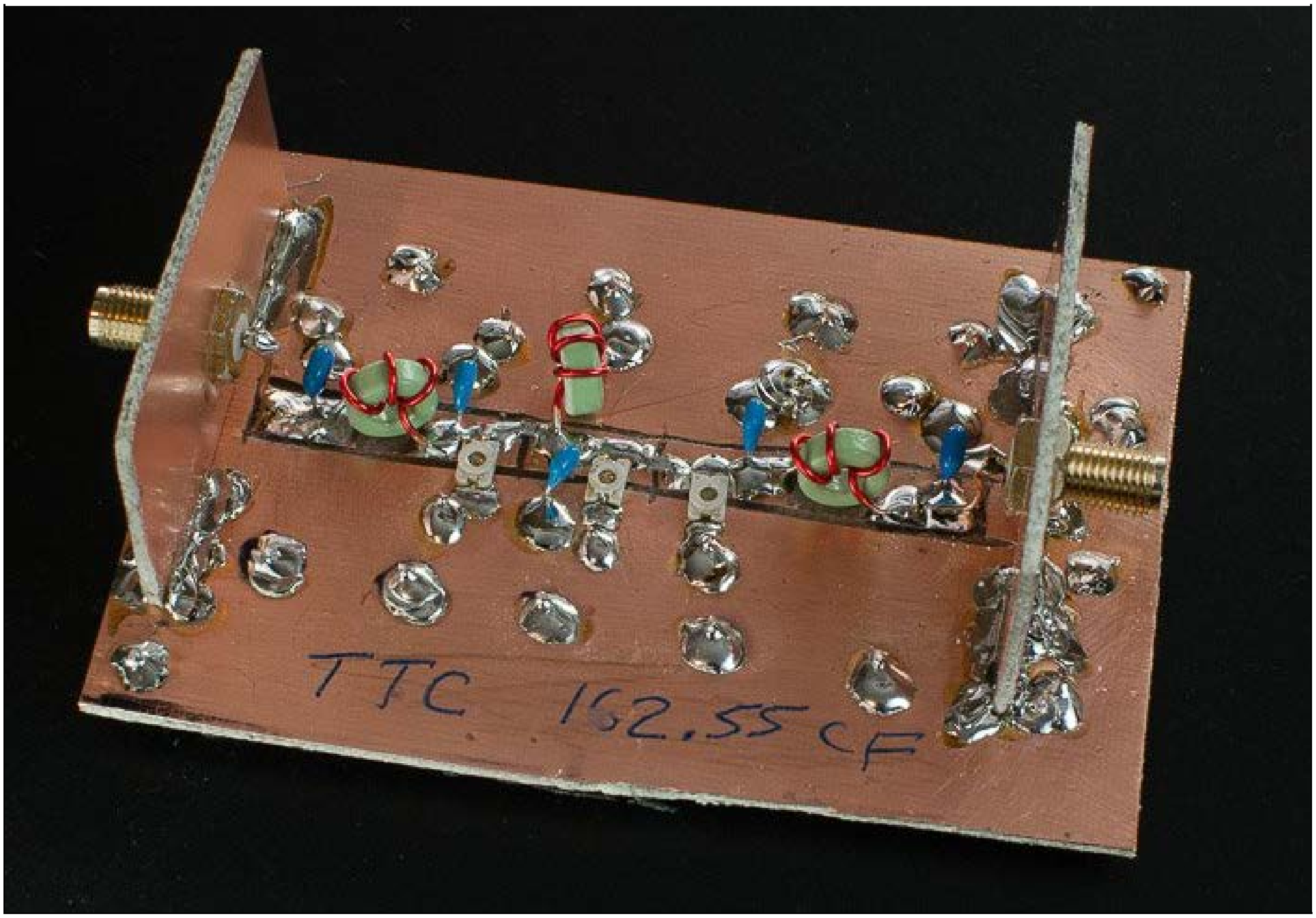


Above — Mixed-mode triple-tuned filter schematic. Click for the GPLA simulation. When we want to go up in frequency plus desire a narrow bandwidth, we face 2 options: reduce our inductance and/or coupling capacitor(s). This taxes our parts collection and breadboard skills since capacitors less than 1 pF are relatively uncommon and require SMD breadboard techniques. I adjusted my design bandwidth and inductance in TTC08.exe to allow the 0.5 pF capacitors plopped in my parts bin to work. For a change of pace, I chose a mixed-mode filter topology.

I strive to place a single high Q variable capacitor in my band-pass filter tanks, however, my Q = 700 SMD trimmers only ranged 3.3 to 10 pF so I stuck a leaded ceramic capactor in parallel with each trimmer.

Common SMD capacitors exhibit low to medium Q and I try to avoid them in my at HF band-pass filters: often I'll just solder a short leaded though-hole caps instead. I also applied this logic to my VHF filter, however, this might prove foolish at VHF since the self-resonant frequency of a given leaded cap is lower than that of an equivalent SMD cap and capacitor XL becomes more significant as we move up in frequency.

Click for a pdf file showing a simple experiment with 3 capacitors . I'll occasionally use ultra-high Q SMD caps in my future experiments based upon these results.

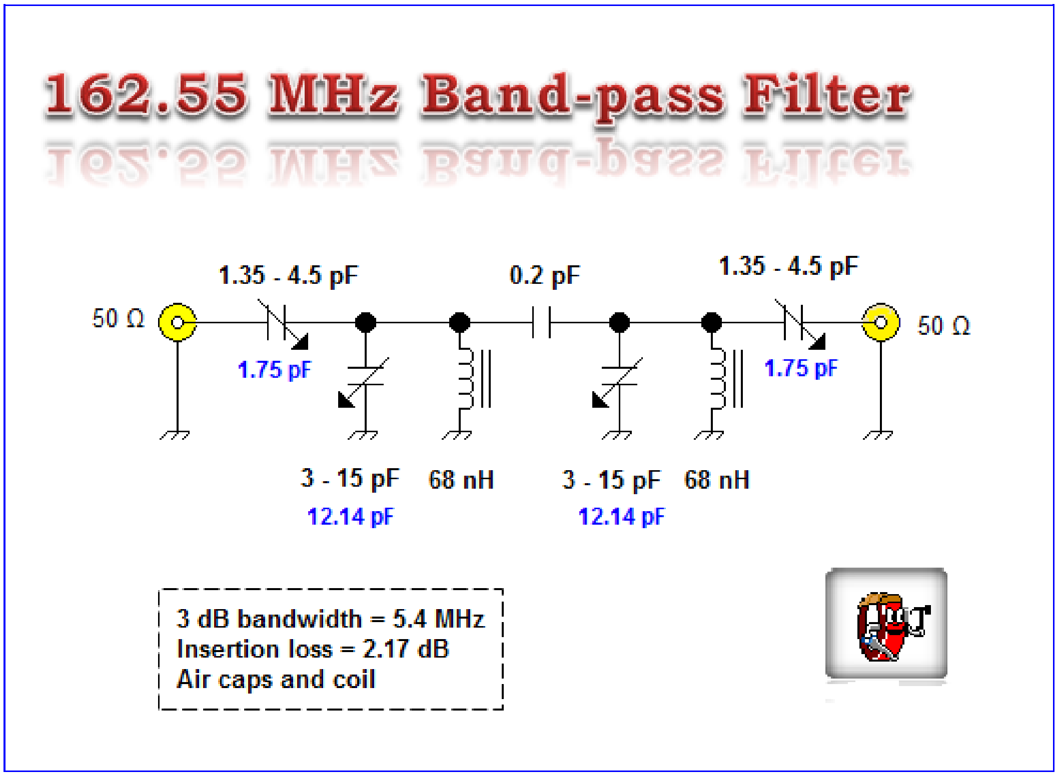


Above — My TTC breadboard. Click for a larger photo. Click for a bench sweep to view a feculent skirt peak in the filter spectogram. Some resonator coupling occurred despite spacing and placing the inductors at right angles. This circuit possessed 6 tweaks: the 3 trimmers and the 3 Ls — I just kept tweaking them until I got the best shape and lowest IL. That's 21 gauge wire on the toroids. My image frequency = 47.28 dB down. Since the sweep yielded such a wide discrepancy from my GPLA simulation, I discarded this filter and moved on to design #2.

Filter 2

Attempting to obtain a resonator Qu of at least 300-400, I moved to air core inductors wound with bare 22 gauge copper wire *plus* air-variable capacitors. Double-tuned classic topology. Without stout shielding, air wound inductors will couple and trash the stopband.

Above — Copper and brass sheets I bought for making filter boxes after Wes recommended using metal sheets or strips for shielding filters. In North America, builders can purchase a sheet metal hand brake plus a rotary cutter/shear for under $100 and make all sorts of boxes for radio gear. I'll equip my QRP workshop with such equipment over the next year. This was my first experience man-handling sheet metal or strips.

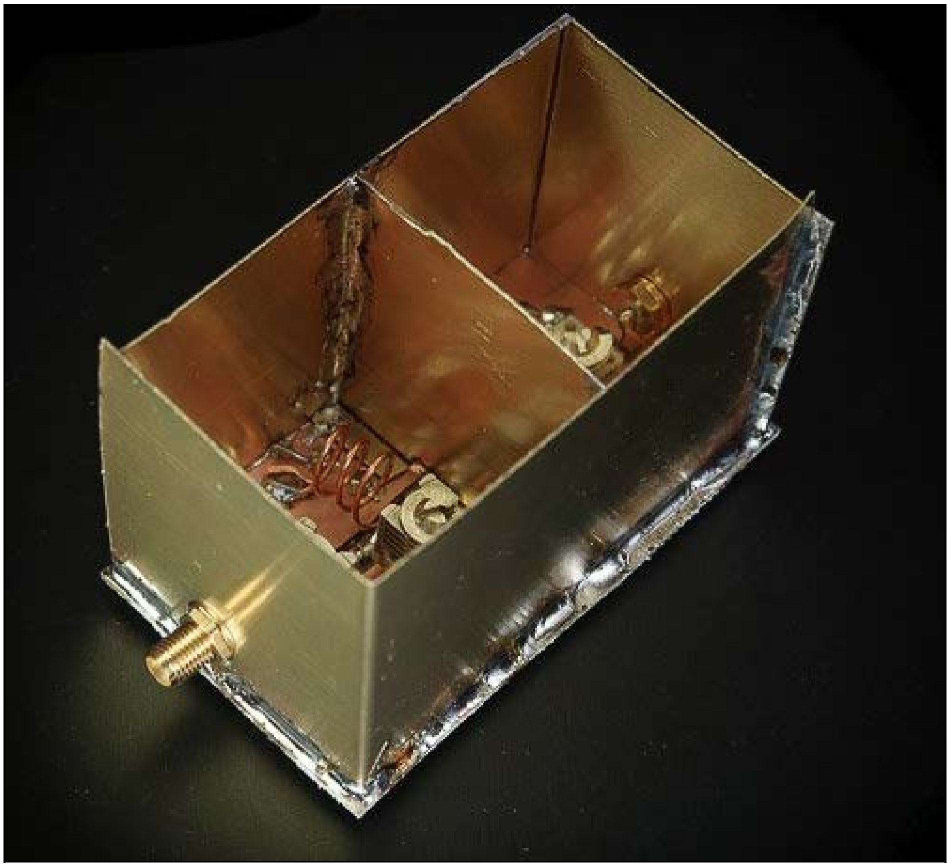


Above — Schematic diagram of Filter #2 with tabled IL and half-power bandwidth. Click to view a macro photograph of the filter. I soldered the brass box onto a double sided copper-clad FR4 bread board laden with some copper wire vias to enhance the ground plane. The 0.2 µF coupling cap = an ATC microwave cap.

Click and click for the sweeps. Notice the splendid skirt symmetry at the filter top and also the fantastic peak attenuation in the low-pass skirt

(~81 dB from the filter peak to the lowest point in the low-pass filter skirt). I'd never seen deeper attenuation on my bench before: quite gratifying. My double-tuned filter had essentially matched the 3 dB BW of triple-tuned filter #1 with less insertion loss.

Not all appeared perfect however; attenuation of frequencies above the passband looked so-so and I was unable to get a lower 3 dB bandwidth by manipulating L and C in each tank.



Above — My first ever brass box soldered on a 2-sided copper clad board: Filter #2. Lacking proper tools and also the knack for making beautiful chassis like Dave, AA7EE does, I just did my best. My only concern was getting tank isolation. I chiseled out a small cut in the center brass divider just high enough to clear the SMD 0.2 pF chip capacitor and center copper strip. Overlaps allowed soldering of brass walls After tweaking the L and C parts, I placed a brass lid on top during sweeps. Okay, onto filter # 3:

Filter 3

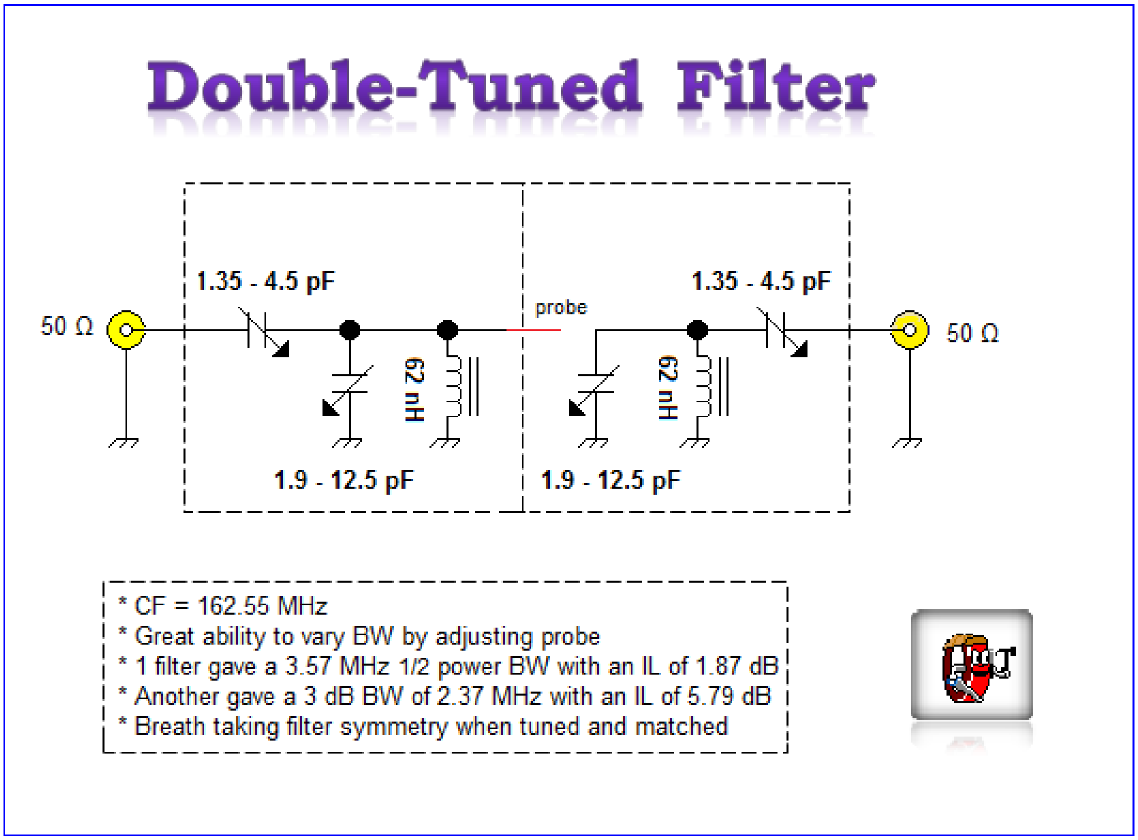
When coupling caps fall below 1 pF, some builders place their resonators in close proximity to couple the tanks, however, these usually involves slug-tuned powdered iron or ferrite inductors with a generally low Q. The coil consists of cup core and a threaded center core which combine to give a magnetic shield. A tin plated copper can surrounds the core for electrostatic shielding. I think after these experiments — placing each resonator in its own sealed box seems a better option.

I noticed another approach in The Double Tuned Circuit: An Experimenter’s Tutorial in QST for December 1991 by Wes, W7ZOI. Wes drilled a hole in the tank compartment divider and passed a wire soldered to 1 tank through the hole and positioned it near the variable capacitor of the other resonator. We change coupling by adjusting this wire.

The wire may be thought of a "tweakable" capacitor, however, the key point is we need to couple energy from 1 resonator to the next; whether by wire, capacitor or inductor, proper resonator coupling will give the required skirt shape within the the limits imposed by Q and the matching to the 50 Ω ports. When establishing a filter passband, the end Q and resonator coupling pretty much dictate the outcome.

Filter #3 involved more mechanics than electronics. I made a U shaped box from a small sheet of copper.

Above — My copper box drilled for the 2 air-variable tuning capacitors. I built the walls, top and divider from thinner gauge brass strips. Rosin flux aids soldering your metal sheets together — I soldered mine with a Weller 80 Watt iron and began with the center divider. Click for a side shot after completion.



Double-Tuned Filter

- * CF = 162.55 MHz
- * Great ability to vary BW by adjusting probe
- * 1 filter gave a 3.57 MHz 1/2 power BW with an IL of 1.87 dB
- * Another gave a 3 dB BW of 2.37 MHz with an IL of 5.79 dB
- * Breath taking filter symmetry when tuned and matched

Above — My filter schematic with some tabled data. Click for a top off photograph where you can easily see the black wire probe. To change coupling, you change the wire length and/or its distance from the second tank capacitor. I really enjoyed moving the wire and watching the outcome on the screen. Per usual, I tweaked the coupling, Ls and end caps to derive many 3 dB bandwidths. In 1 arrangement with a 3 dB bandwidth of 3.57 MHz, the insertion loss was only 1.87 dB — I had evidently found a perfect ratio + combination of L and C yielding a low IL.

I'll increase the coil wire gauge in future experiments to lower solenoid resistance and try to garner even more Qu.

This is the *best* filter I've ever built. After peaking the tanks to 162.55 MHz, resonator adjustments weren't really needed. By moving the wire probe I saw that BW and IL change inversely. While tweaking the series caps and moving the probe, it felt like déjà vu from my experiences simulating these exact changes in GPLA over many years — except now they were alive and kicking. With some inductor manipulation (or as needed, changing the inductors) this filter can be centered from ~140 to well over 200 MHz. A high-octane, dramatic and versatile filter indeed.

Click for a sweep where I moved the wire very close to the neighboring tank capactor and over coupled the filter.

Click for a screen shot of another sweep.

Conclusion

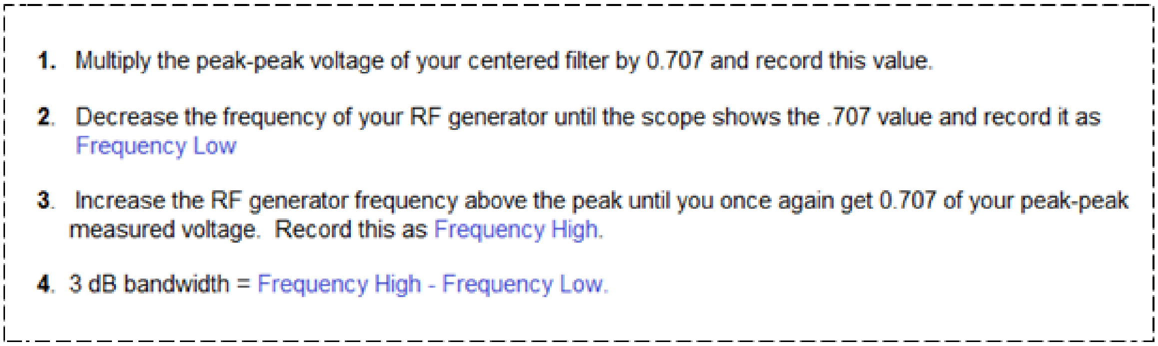
With new awareness that resonator coupling and other issues may *also* occur in HF band-pass filters, making these 3 filters surely boosted my future band-pass filter construction standards at any frequency. Over the years, builders have emailed me sweeps of HF band-pass filters (usually VNA sweeps) with also sorts of extra resonances +/- poor insertion loss. Since I'm just an amateur, often nimrod RF enthusiast, I won't pretend I possess all the knowledge to critique their filter woes.

However, photos of their HF filters usually look messy and rushed: opposite to our needs — eschewing important factors like symmetrical layouts, inductor spacing, a first-rate ground plane, larger gauge + well spaced wire wound on big as possible toroid cores Yikes, I better stop now lest I venture into filter folklore.

Today, designing a filter with software is merely half the battle — construction poses the harder task. Unsurprisingly, measurement and experimental methods will elevate your outcomes.

Filter analysis with a sweep system is the ultimate, however I built and then tested many band-pass filters with a frequency counter, a signal generator, a 'scope and a return loss bridge during the first 12 years of my RF life.

Assuming you have a filter that peaks and isn't overcoupled: to measure the 3 dB bandwidth, first peak the filter to resonance for the highest peak-peak voltage in your 'scope. Here's the rest of the procedure:



1. Multiply the peak-peak voltage of your centered filter by 0.707 and record this value.
2. Decrease the frequency of your RF generator until the scope shows the .707 value and record it as Frequency Low
3. Increase the RF generator frequency above the peak until you once again get 0.707 of your peak-peak measured voltage. Record this as Frequency High.
4. 3 dB bandwidth = Frequency High - Frequency Low.

You may also wish to measure the insertion and return loss as described in EMRFD, Radio Society Handbooks, or the RF Workbench series on this web site.

Whether you run superhet, low IF, SDR, or Zero IF (DC) receivers, or need to filter a transmit chain, well made band-pass filters just might boost your project performance. VHF poses additional challenges but offers great learning opportunities.

QRP — Posdata for Nov 12, 2013

A. Notes for Filter #2

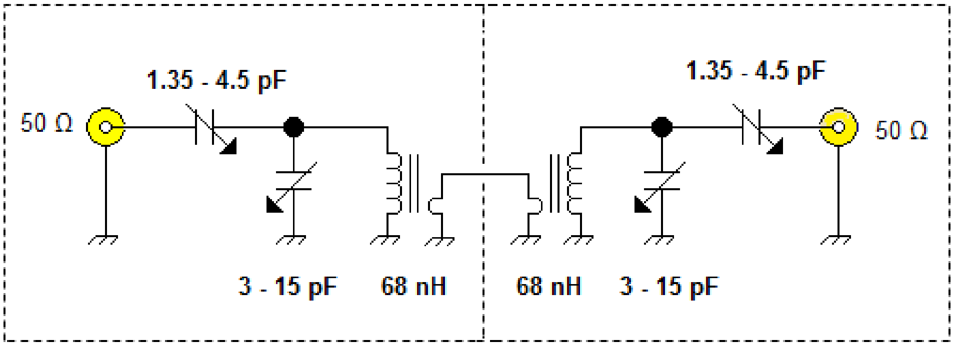
Victor, 4Z4ME wrote to explain why the attenuation above the passband in Filter #2 looked mediocre. Filter topology and component values yielded the poor high-pass attenuation and not unwanted coupling as I originally shared.

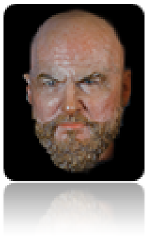
Victor wrote: "The problem is that all the couplings are of one nature, in this case capacitive.
At very low frequencies the inductors short the signal to ground while the capacitors impedance gets higher so it is clear that the output signal goes to zero.

At high frequencies the inductors may be regarded as disconnected because their impedance get very high, however although the capacitors impedance get lower , you have them both at the series arms and at the parallel arms, so the signal gets to be attenuated by their impedance ratio and not get shorted to ground.

This can be shown in analytic calculation, or easier by simulation. See the attached file (LTSpice simulation). You can see that the filter (similar to your filter) gets closer to a finite attenuation at high frequencies, getting closer to the attenuation of a similar circuit with the inductors *removed*.

You can get better high frequency results by trading the center 0.2pF capacitor coupling to an inductive coupling. Remove the coupling capacitor and insert a wire through the center shield, ground its ends close to the main inductor. Shape the wire to a small loop near the grounded end of each of the resonator inductors to get small inductive coupling, and you get again a good BPF but with better high frequency attenuation. Instead of loops you can connect the coupling wire at a tap at the bottom of the main resonator inductors, close to the ground".





Filter 2 Pos-data Experiment

Above — Following Victor's suggestion, I inductively coupled the Filter #2 resonators with a wire looped near the grounded end of each tank inductor. Please view the schematic above. High-pass attenuation above the passband improved significantly. Click for a sweep out to ~ 511 MHz that shows the peak high-pass attenuation much closer to the low-pass skirt peak reponse.

At 2X center frequency (325.1 MHz) the high-pass skirt peak attenuation closed within 14 dB of the low-pass peak attenuation compared to the original Filter #2 response where the delta was around 30 dB. Further improvement occurs as we move above ~400 MHz. Big thanks to Victor for his support.

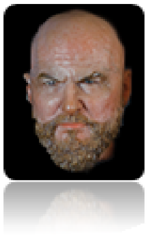
B. Sheet Metal Safety

A cut finger while working sheet metal prompted the following remarks: Jagged edges from cut sheet metal (especially brass stock it seems) may lead to deep cuts. Filing a burred edge with the metal clamped in a vice between some thicker metal stock with its raspy edge barely showing will reduce the "knife edge effect". When drilling, clamp the metal piece to your bench on a wooden backing board . If you jam your hand drill in an unclamped piece of metal it could spin and really gore you. Safety first.

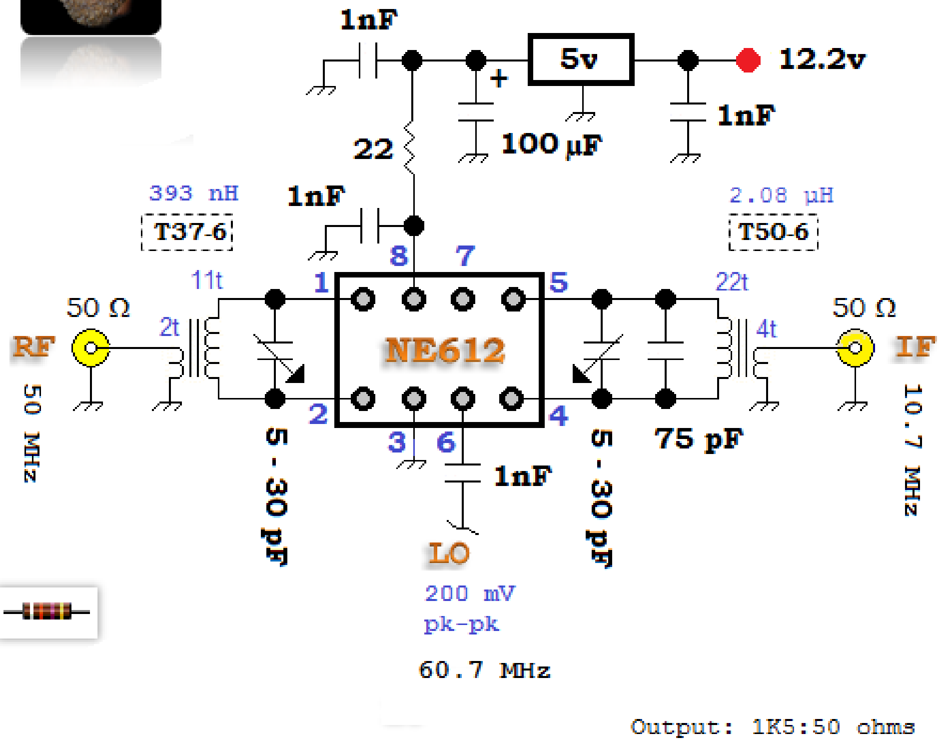
---

7. NE612 Mixer Diddy

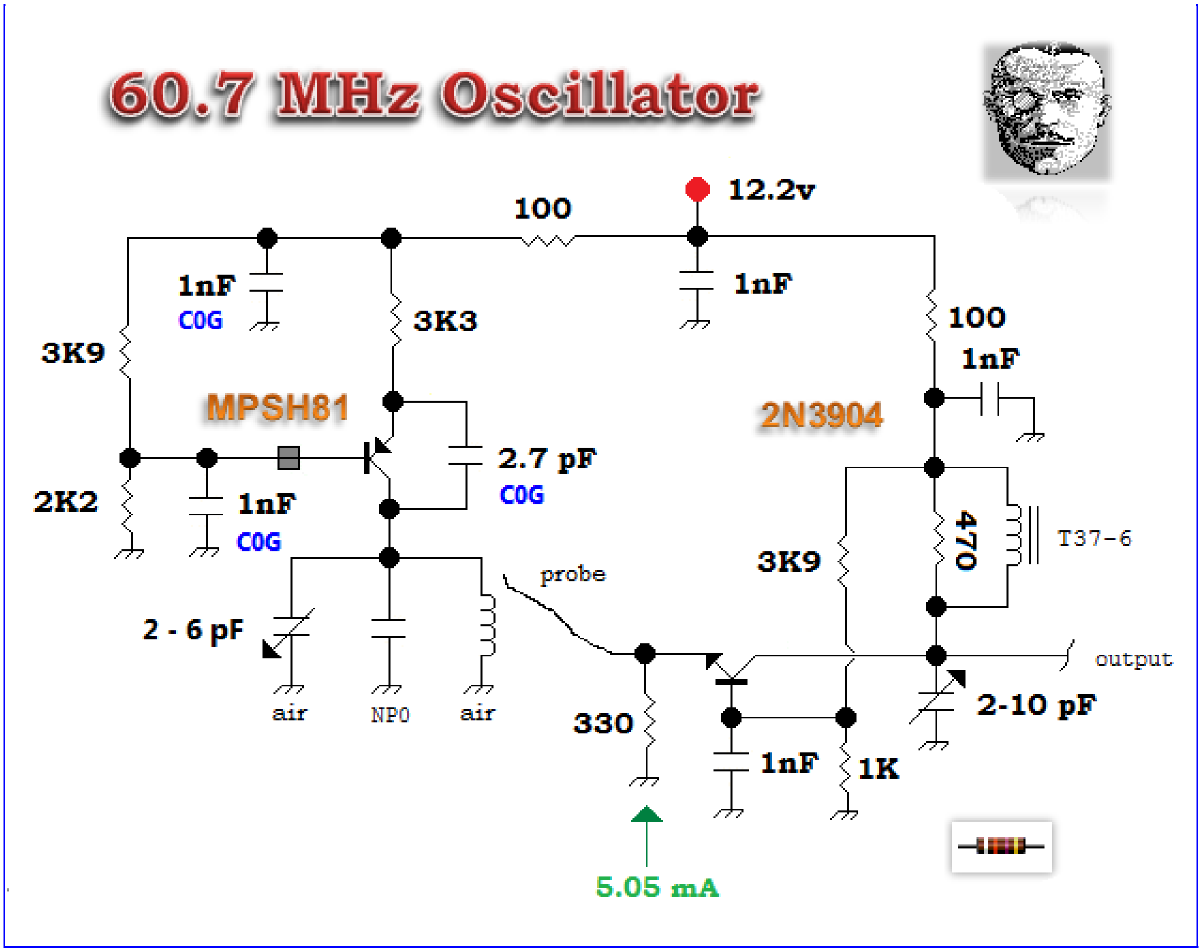
The Signetics NE612 and ilk enjoys wide use by the amateur radio community in novelty-grade Ham band receivers, or in somewhat better receivers laden with abounding front end filters + *stout,* switchable RF attenuators. Do you think of the NE612 as an ersatz receiver mixer? Some only consider them for the transmit mixer slot. Read this link why.



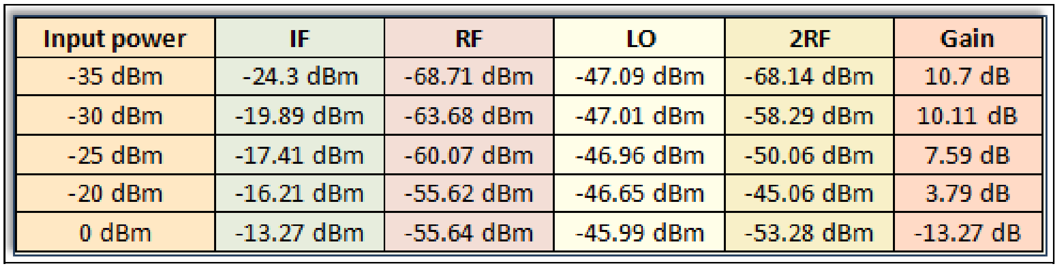
NE612 Evaluation



Above — My base schematic. I applied differential input to cancel 2nd order mixer products and balanced output to obtain maximum conversion gain. Click for the breadboard. Powdered iron toroids, Q = 1500 trimmers and a fixed C on the output formed the resonators. I chose a 10.7 MHz IF to drive low-cost ceramic filters in further (unshown) experiments. Normally, we don't bother tuning the input since our bandpass filter should launder the input signal sufficiently.



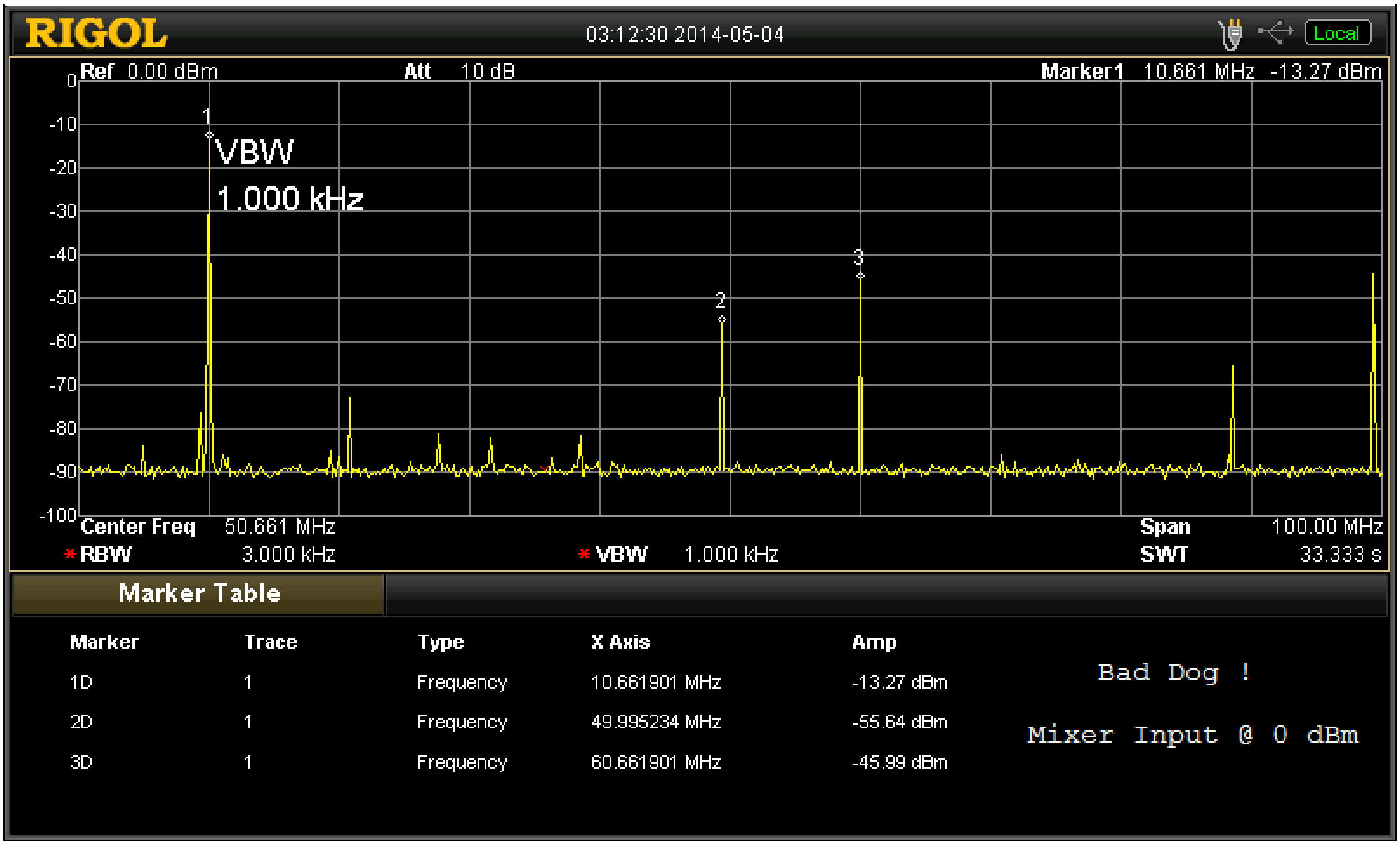
Above — My 'thrown together' 60.7 MHz local oscillator. A wire on the common base buffer amp input avoids the series R and C usually required and I moved the wire probe to set the output power between 180 and 250 mV peak-peak into a 1K resistor load.



Input power	IF	RF	LO	2RF	Gain
-35 dBm	-24.3 dBm	-68.71 dBm	-47.09 dBm	-68.14 dBm	10.7 dB
-30 dBm	-19.89 dBm	-63.68 dBm	-47.01 dBm	-58.29 dBm	10.11 dB
-25 dBm	-17.41 dBm	-60.07 dBm	-46.96 dBm	-50.06 dBm	7.59 dB
-20 dBm	-16.21 dBm	-55.62 dBm	-46.65 dBm	-45.06 dBm	3.79 dB
0 dBm	-13.27 dBm	-55.64 dBm	-45.99 dBm	-53.28 dBm	-13.27 dB

Above — A table showing the mixer input, IF, RF, LO, 2xRF [the worst spur] power readings, plus calculated conversion gain. This table echos the datasheet + work by others — input power >= -25 dBm drives this mixer into compression. The NE612 runs low current plus low voltage across the output transistors — thus the maximum output power is limited to somewhere between +2 dBm to +5 dBm from my experiments.

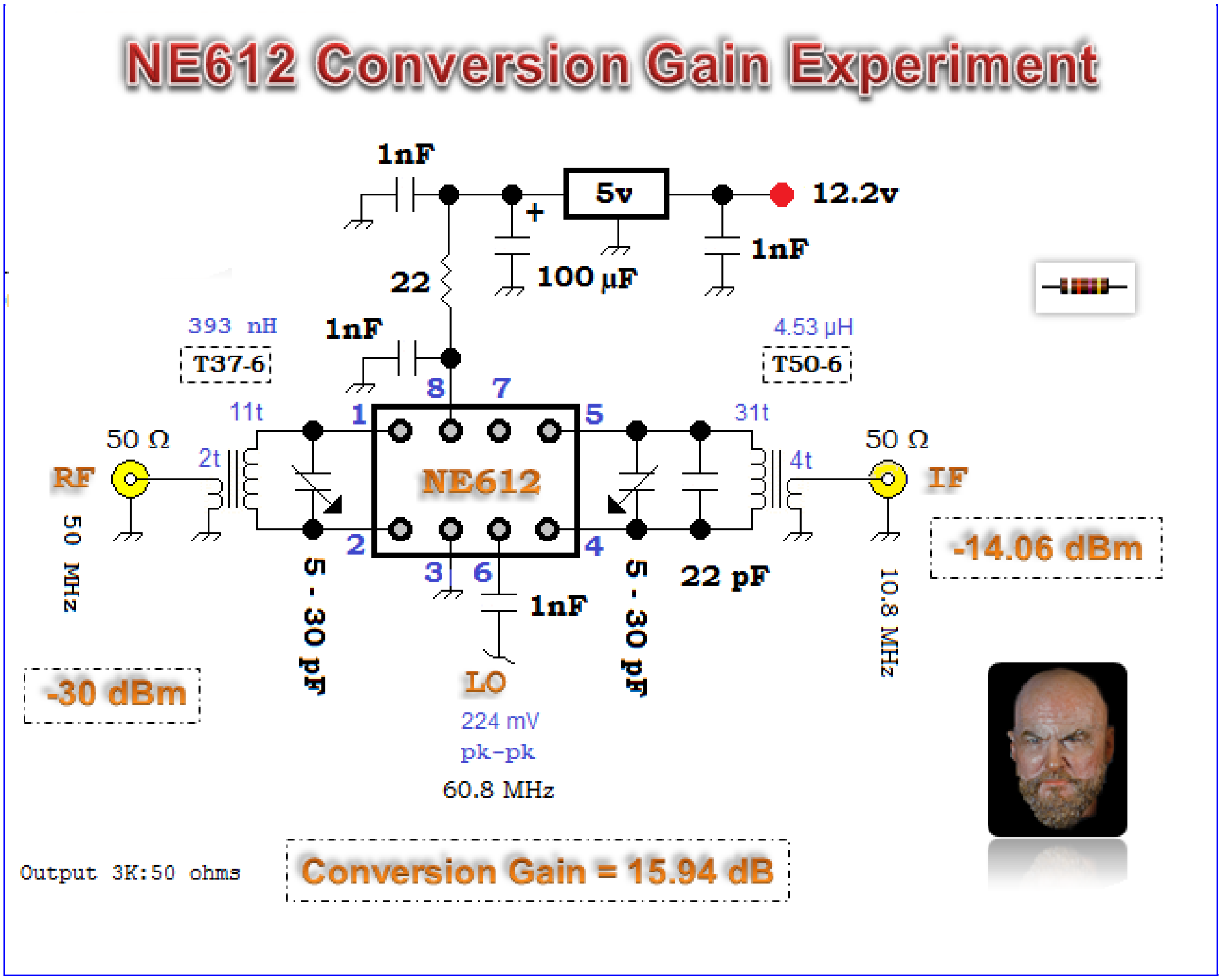
Even with -35 dBm input power, I measured a conversion gain much lower than the typical 14-18 dB enjoyed by others who applied differential output at ~ 50 MHz. Turns out, I goofed — each output transistor contains a 1K5 Ω resistor, so when wired differentially, the output Z = 3K. Back to the bench!



Ref 0.00 dBm
Att 10 dB
Marker1 10.661 MHz -13.27 dBm
Center Freq 50.661 MHz
Span 100.00 MHz
*RBW 3.000 kHz
*VBW 1.000 kHz
SWT 33.333 s

Marker	Trace	Type	X Axis	Amp
1D	1	Frequency	10.661901 MHz	-13.27 dBm
2D	1	Frequency	49.995234 MHz	-55.64 dBm
3D	1	Frequency	60.661901 MHz	-45.99 dBm

Above — SA of the mixer output with 0 dBm RF input. Something we would never do in real-life, but I wanted to see the spurs with such a drive: the spurious output actually looks great! The NE602, NE612, MC1496 and SO42P [all Gilbert cell mixers] output cleanly; especially when compared to diode-based mixers.

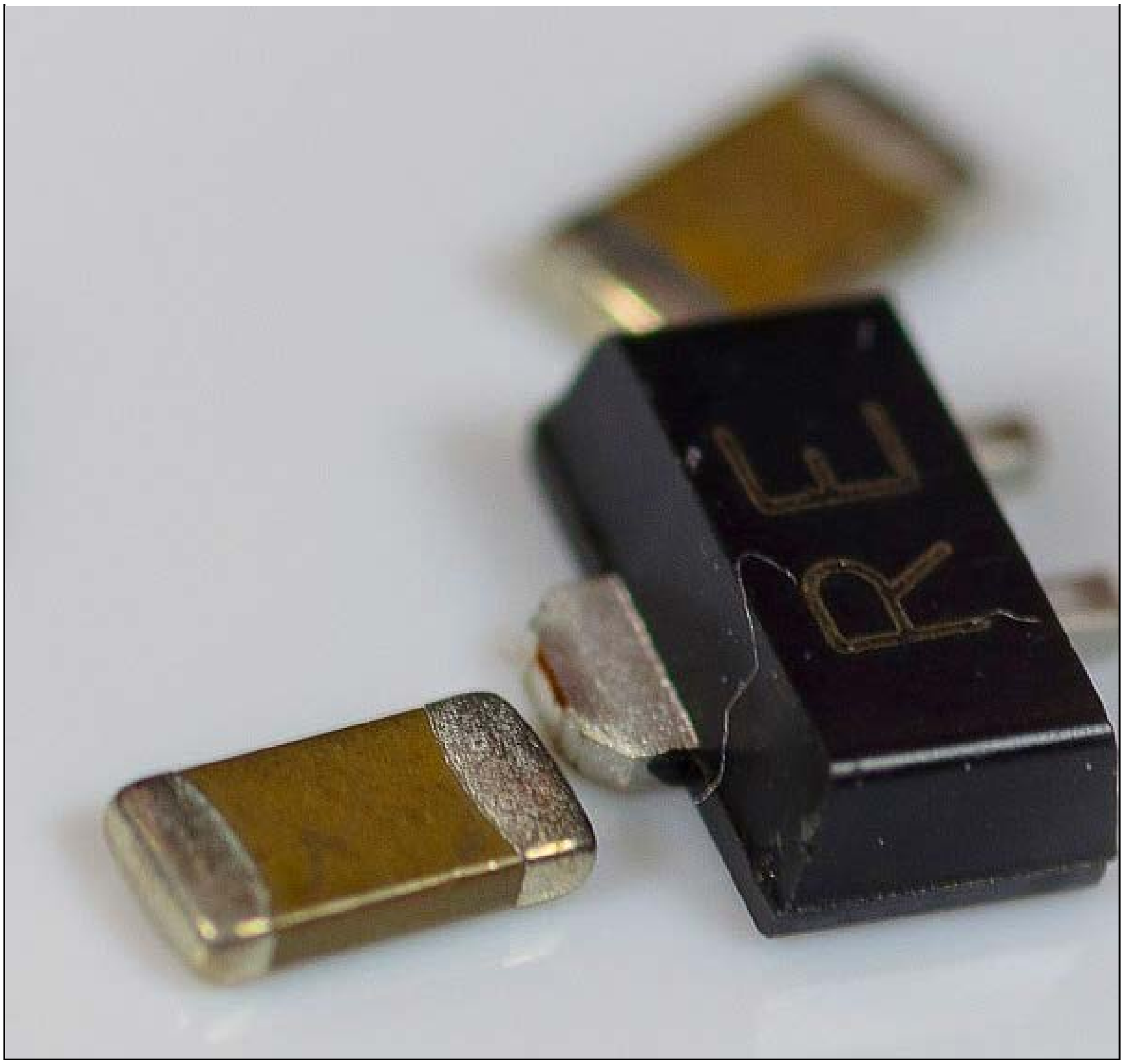


Above — My mixer with output Z corrected at 3K Ω. Wow, a conversion gain of 15.94 dB with -30 dBm drive. All good now. Click for the IF measurement on my SA. The LO had drifted up to 60.8 in this experiment, however, I left it and peaked the output resonator for maximum smoke. I also extracted another 0.5 to 0.8 dB gain by tweaking the LO drive. In my breadboard, a drive of 224 mV pk-pk gave the highest conversion gain.

Hereafter, I'll drive my NE612 transmit mixers with between -26 to -30 dBm on the RF port with differential input and output.

---

8. Miscellaneous Photographs



Above — Hair or lint in my SMT macro photo

Above — Instrument grade N-male to SMA female

Above — Instrument grade N-male to SMA female; gorgeous machine work.

RF — Test and Measurement

RF Workbench Page 6 — QRP Transmitters

RF Workbench 6 explores some questions regarding basic transmitter power amplifer (PA) design. Since I write for both Ham and non-Ham readers, I feel badly that RF Workbench 6 excludes pure radio listeners; please consider getting your amateur radio licence to join the fun.

For Class A, AB and C transmitters in current source operation, radio builders knew most everything presented on this web page back in the 1980's — humbled and guileless, I've but a skosh of new ideas to further elevate this topic. Sometimes I feel bound in a creative straightjacket because puzzling design questions paralyze my understanding and progress. This web page explores a few transmitter questions born on my QRP work bench and I hope you'll relate and go design your own high-octane transmitters. As always, I hope to share insightful + actionable info for the DIY scratch builder; albeit from a lay-person perspective.

I borrow heavily from the work of Wes, W7ZOI per correspondence, direct contributions, from EMRFD and the references listed in Section 5. My sincere thanks to Wes for his support and inspiration.

Preface

You require an amateur radio operator licence to operate a radio transmitter and it's up to you to know and ensure your transmitter spurious emissions meet or beat the regulations in your country.

Although I measure my circuits with calibrated equipment, expect your measured voltage and current to differ from mine. **Why?** Variables may include your DC power supply voltage and current rating, dummy load resistance, stray L and C, cables and wire effects, bench errors, parasitic oscillations, temperature effects and parts tolerances.

I unabashedly declare my love for experiential learning: to build and measure, discover and develop — and best of all, to tune, tweak, and transform my RF power amps to unleash the "smoke" and power we all crave.

Contents

Section 1: 21.06 MHz Bipolar Transistor Transmitter in Class C: Dig out your old transistors and explore some Class C topics.

Section 2: PA Measurement with Examples: Measure collector V and I

Sections 3, 4: Incomplete and coming in 2013-14. I'll post any additions on the QRP Log -- Thanks

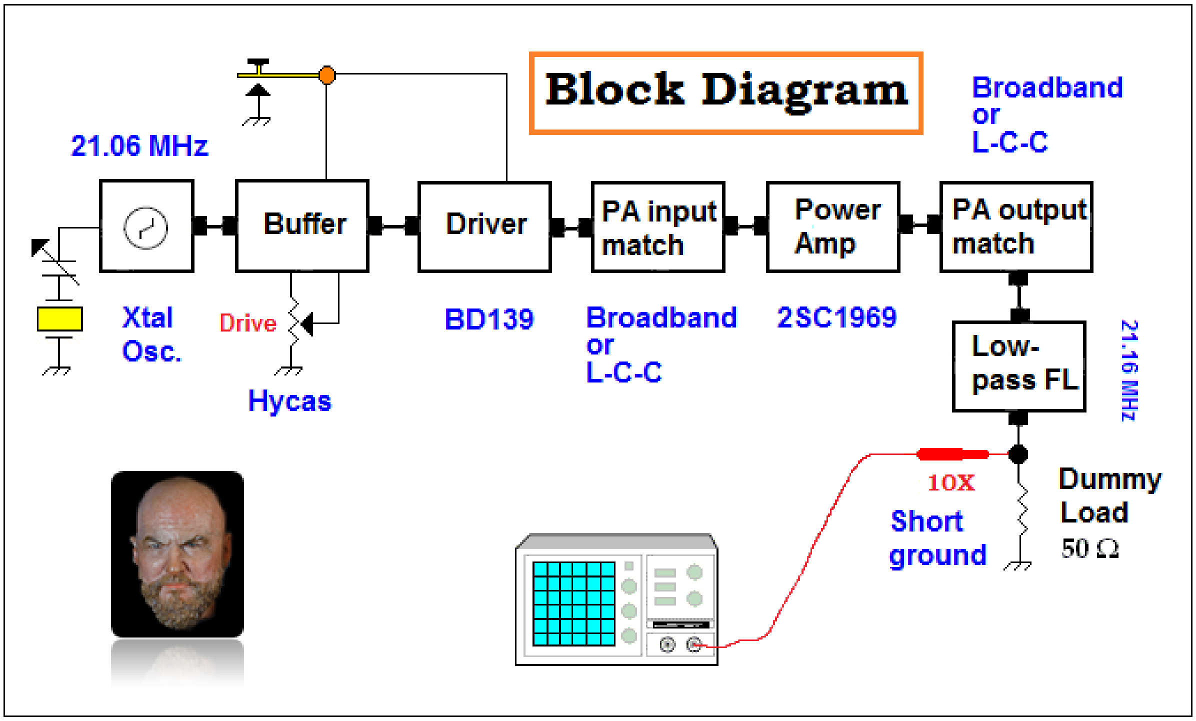
Section 5: I'll list all the references for this web page

Section 6: Photographs and Miscellany. I'm told at least ~~3~~ (now 4) people enjoy my photos.

Section 1: 21.06 MHz Bipolar Transistor Transmitter in Class C

Despite all the buzz about PA efficiency and the trend towards more efficient switched-mode power amplifiers, my email inbox fills with content regarding those Class A to C power amps. Consider too, most Hams, including QRP buffs, power their rigs with an AC connected DC supply that makes battery life moot. Perhaps our love of PAs in current source operation harkens from our historic tube days: do we simply enjoy making heat and wasting power? Like me, you might just love making and measuring bipolar transistor circuits.

A Class C power amplifer excited by a crystal oscillator offers both new and reawakening homebrewers the chance to make a simple but relatively powerful, 1 frequency transmitter with only 2 - 4 RF stages. Keying a Class-C PA embodies the Ham glory days: put some RF into the sky with basic, analog circuitry.



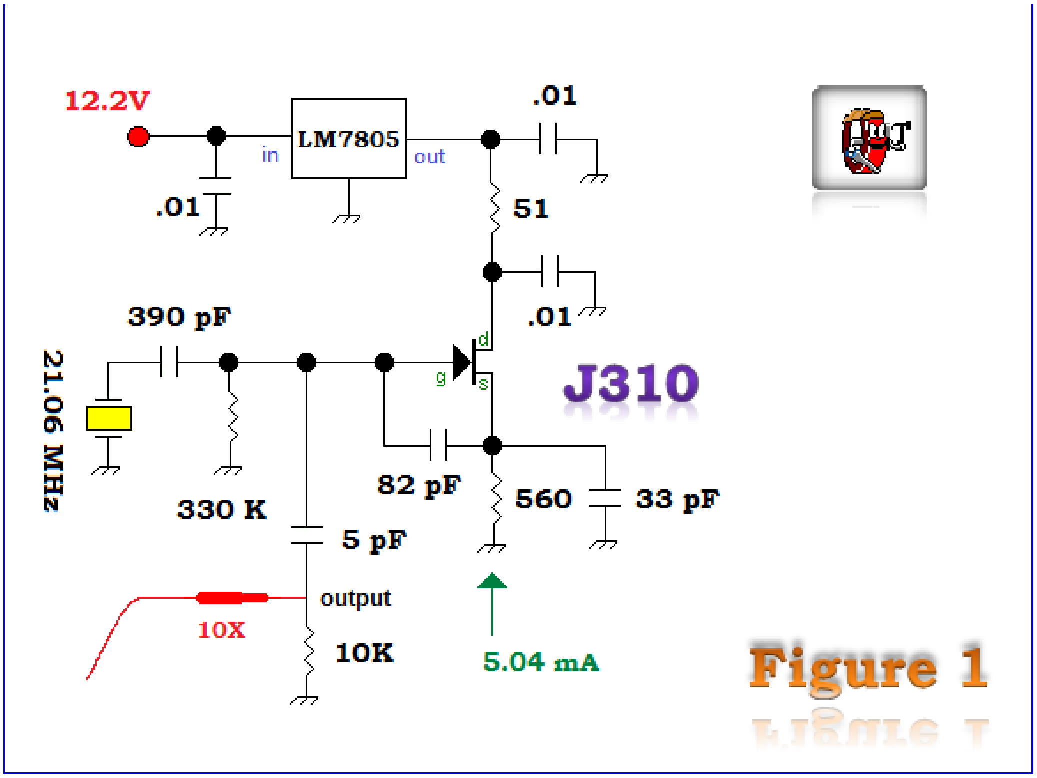
Above — The block diagram for all the experiments in Section 1. Each stage involving transistors was built and tested from left to right before making the next stage.

Crystal Oscillator, Keying Switch and Hycas Buffer Amplifer

As more a listener than a talker, I wanted a low-level crystal oscillator to keep my *key up* current draw low. In the class C power amplifers based on the above block diagram, I measured only 7.5 mA. *key up* and several hundred mA *key down*. I also wanted my back wave to be > 40 dB when I eventually converted the PA to Class AB during another set of experiments.

Design Question 1: What backwave should I strive for?

Backwave = the dB difference in output power between key up and down. Opinions differ and context may partly explain why. If you're working DX, backwave is less important than when your working nearby ops on 80 meters. From asking respected authors, experiments and operating, I feel 40 dB should be the minimal backwave to strive for. Operating Class C, your final(s) are turned off during key up and a >= 40 dB backwave proves an easy goal.

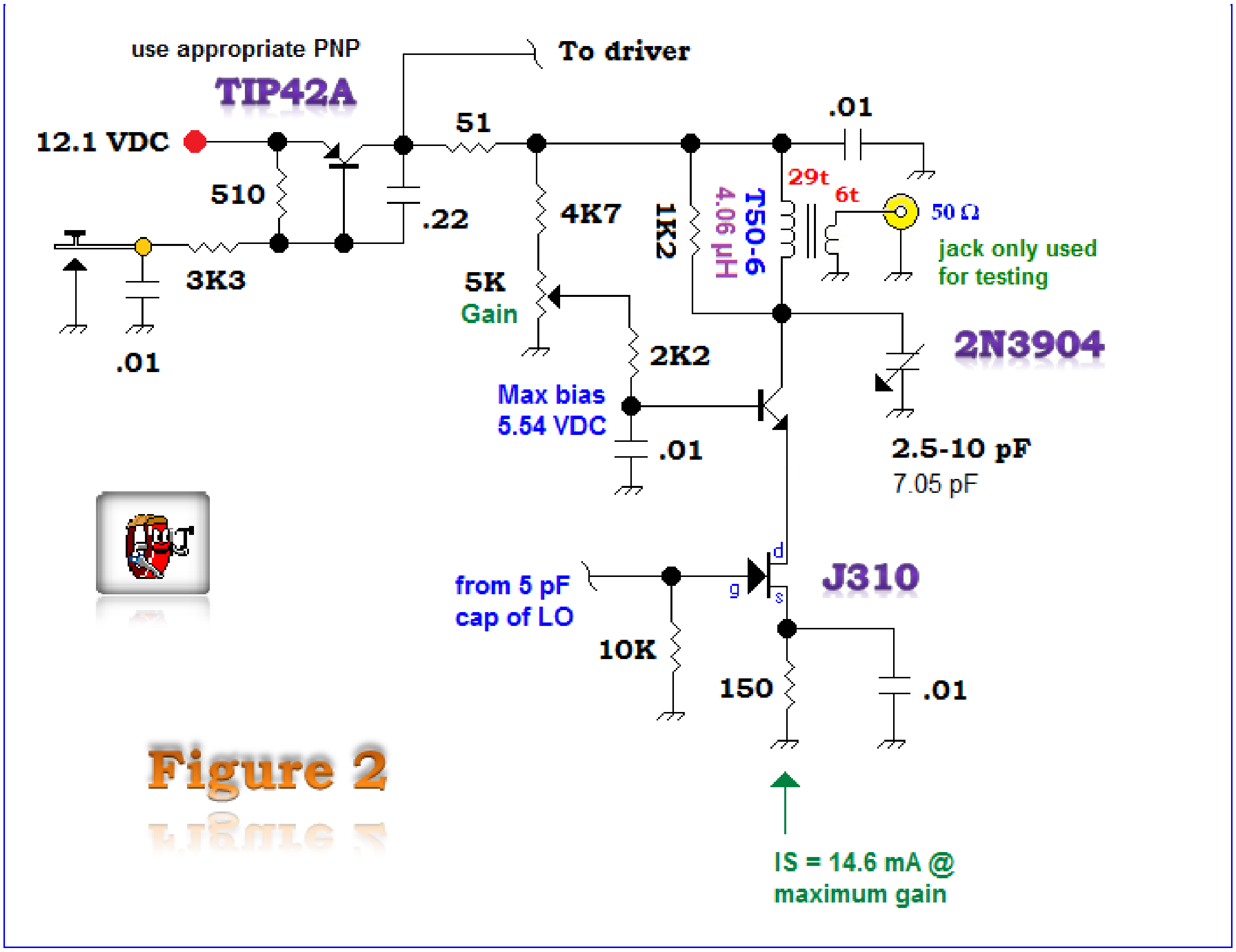


Above left — The 21.06 MHz crystal oscillator. Click for the output voltage into 10K Ω. Click for the FFT. Seeking low distortion and output power, I placed variable capacitors in the feedback and source slots and also played with the 560 Ω source resistor. While watching in my 'scope and tweaking , I chose the the lowest distortion signal under 1 Vpp. The variable capacitors were removed, measured and replaced with nearest standard 5% parts. The 10K load of Figure 1 corresponds to the gate resistor of the hycas amplifer input.

Above — Without the 390 pF capacitor, the crystal oscillator vibrated at 21.058 MHz.

Above — The crystal oscillator measured frequency after experimentally adding a 390 pF cap. Should I have tried a 470 pF capacitor and attemped to get 21.0600 MHz?

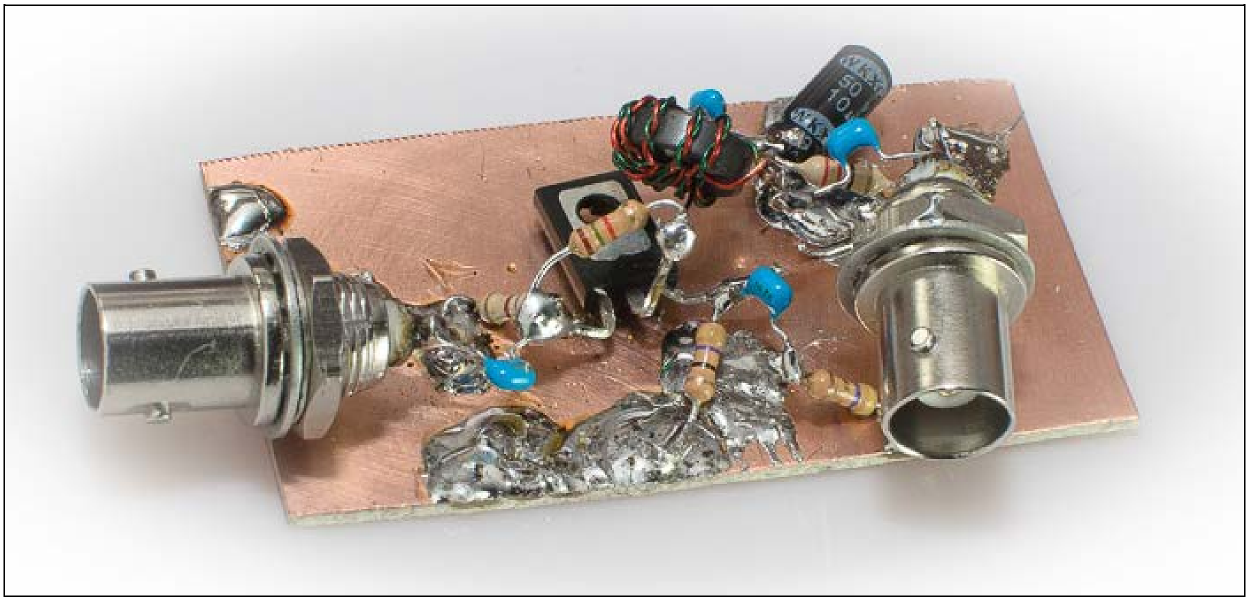
My oscillator looks unconventional and in some cases you might want 1 with higher output voltage. Check EMRFD and the other W7ZOI references in Section 5 for xtal oscillator design ideas.



Above — Figure 2, the hybrid cascode buffer. I've never tuned, nor keyed a hybrid cascode amp before and it works well. Click for the oscilloscope output. Click for the FFT measured with a long wire grounded 10X probe. I applied a junkbox TIP42A switch lying on my bench since I keyed a number of *unshown* high current stages during my experiments. Only about 10% of my experiments ever make it onto the QRP/SWL HB web site. For the PNP switch, just use whatever can handle the *measured* current with key down: 2N3904, 2N4403, 2N2907, BD140, etc.,etc..

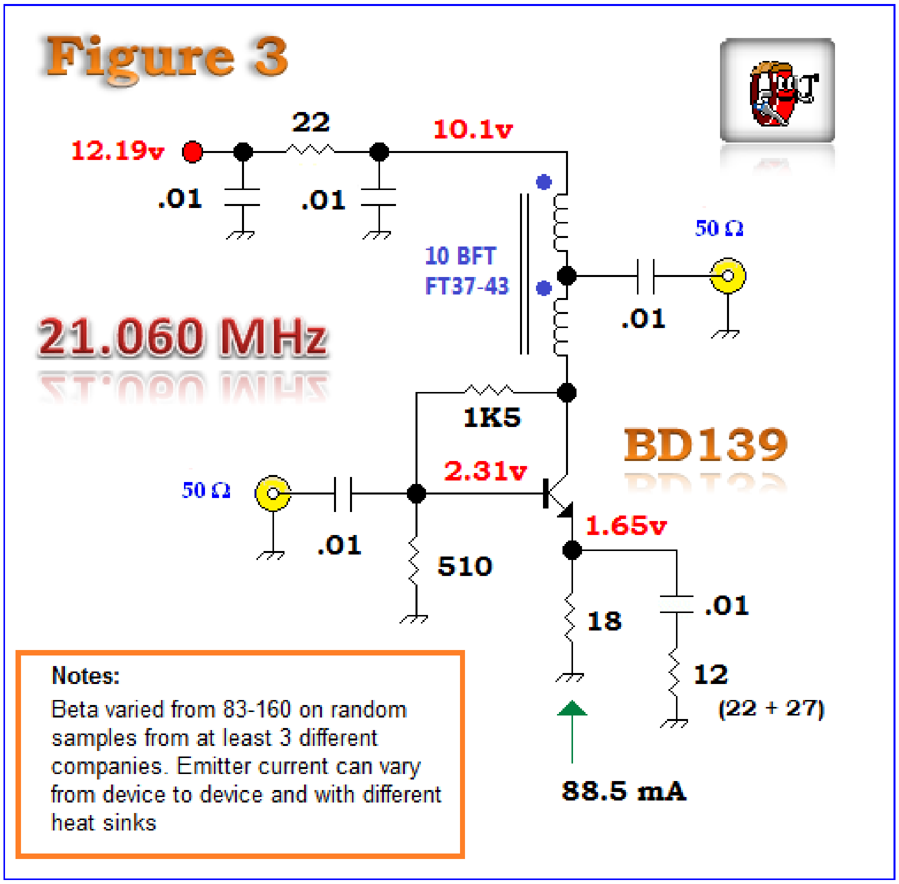
Yes dear reader, measure everything.

The Driver Amplifer



Above — A BD139 feedback amp breadboard. Click for the schematic. Wanting an output power of at least 22 dBm to drive my final, I built 3 varieties with both the BD139 and a 2N3866. Normally, for receiver amplifers, I focus on a strong input and output return loss and work hard to maintain linearity — but mostly keep the emitter current under 50 mA. At 21.06 MHz, the biggest *clean* signal I could muster with such a broadband feedback amp was +16 to +18 dBm — well short of my goal. Click for a DSO screen shot of the 41.3 mA amp at 6.42 Vpp or ~20.13 dBm power; quite distorted.

I needed to try something else and saw the Figure 3 amplifier in some of Wes' work; wielding bold emitter current to boost signal handling and power.



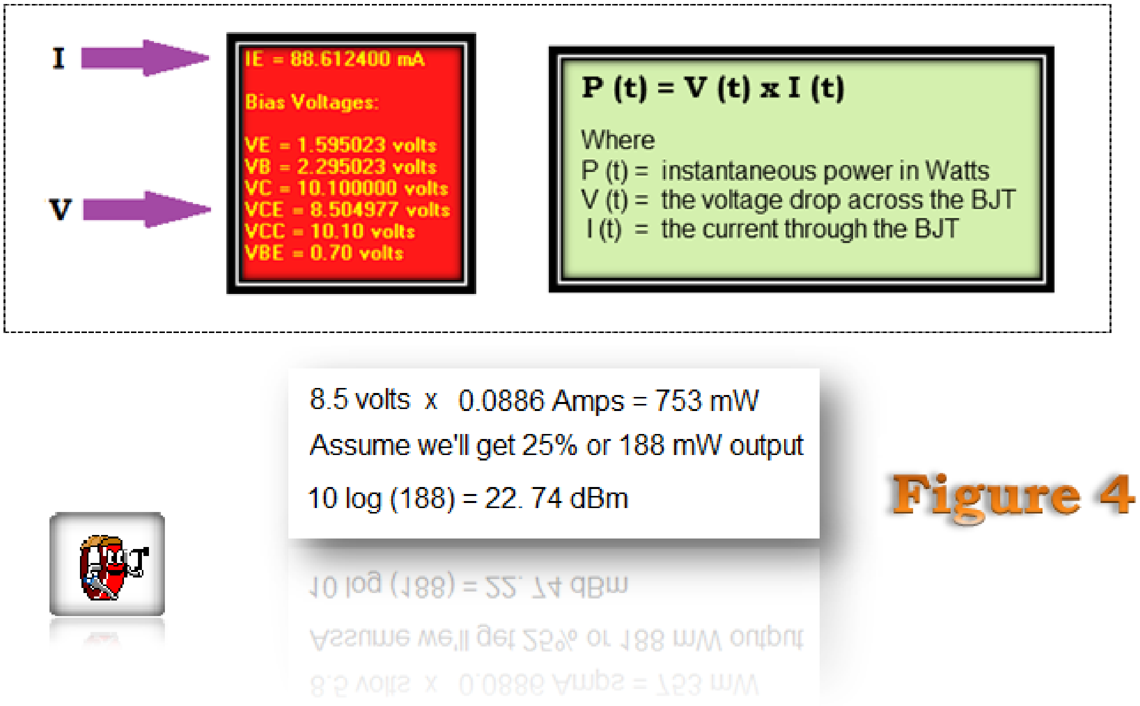
Notes:
Beta varied from 83-160 on random samples from at least 3 different companies. Emitter current can vary from device to device and with different heat sinks

Above — The W7ZOI designed feedback amp used for a driver in all Section 1 power amp experiments. Love it! Many builders feel tempted to abandon series and/or shunt feedback and just "go for gain". My experiments suggest this approach often leads to instability within the driver circuit and possibly on down the RF chain. We should strive for linearity and stability; 2 big reasons we apply feedback. Flirt with a driver *lacking* feedback if you dare.

Click for the maximum output power attained by turning my hycas bias pot fully clockwise: 7.77 Vpp, or 21.79 dBm. You can see distortion, but since this stage will drive a Class-C transistor PA (i.e. it will switch "a diode" on and off and generate clipping), I decided it will work fine. A VHF-UHF transistor such as the 2N3866 or 2N5109 may yield as much as 3-4 dBm more power, however, you'll see more clipping at higher driver levels. I stuck with the BD139 and tried several pieces from at least 3 different manufacturers (The BD139 was made by at least 10 companies at 1 point). A Philips version purchased in 2008 yielded the 21.8 dBm, so I bolted it to a home built copper clad heat sink and soldered it in place.

Calculating Transistor Output Power Off the Bench

Design Question 2: How do I mathematically estimate my Class A amplifier output power?



IE = 88.612400 mA

Bias Voltages:

VE = 1.595023 volts
VB = 2.295023 volts
VC = 10.100000 volts
VCE = 8.504977 volts
VCC = 10.10 volts
VBE = 0.70 volts

P (t) = V (t) x I (t)

Where
P (t) = instantaneous power in Watts
V (t) = the voltage drop across the BJT
I (t) = the current through the BJT

8.5 volts x 0.0886 Amps = 753 mW

Assume we'll get 25% or 188 mW output

10 log (188) = 22. 74 dBm

Figure 4

Calculate Output Power from VCE and IE

Above — Estimating output power from my Figure 3 amplifier. I measured a static DC Beta of 125 as shown in Section 5 on this page. Then I calculated VCE (the voltage across the transistor) and emitter current with my program NPN-DC Bias. Wes wrote a similar and likely better program called Biasnpn08.exe that's included on the EMRFD compact disk.
You may also measure your VCE and IE directly and improve the above calculation with measurements. I do this frequently.

Class A amps exhibit about 25% efficiency and also imagine that Figure 3 will bear some gain compression.

My measured power = 21.8 dBm — it amazes me how close a non-bench calculation using DC analysis can get to a AC bench measurement. The Figure 4 procedure is apparently a bread and butter RF design tool.

The Power Amplifier Input

Design Question 3: How do I match my driver output to the Class-C power amp input to maximize power?

This question provided a few nights of reading and experiments; time well spent. We have 2 main ways to match our driver output to the PA input: broadband or tuned. First, lets explore the problem theoretically:

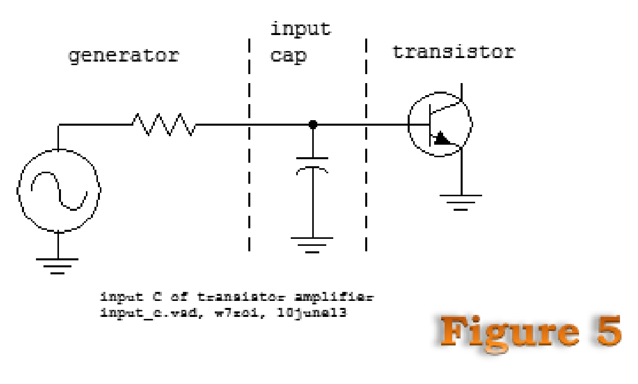


Figure 5

Above — A simple model sent by Wes: driving an amplifier with a power source. As amateur designers, simple models help us understand complex concepts.

A pure voltage source doesn't really exist. We don't apply a voltage to the base of the PA; rather, we apply a power source (our signal generator) — a signal voltage followed by a series impedance that's usually just a source resistance for a load. The power source + source impedance together *source* the load that includes the load capacitance represented as an input cap in Figure 5. You may wish to read about Thévenin's theorem and its dual, Norton's theorem to learn more about linear network analysis.

The load presents an impedance where the power is delivered and either dissipated in its resistance, or reflected back to the source when the load contains a reactance . At our PA input, this capacitive load reflects some of the signal back and induces a pole that corresponds to a "capacitor" time constant.

If you consider our power source, its source resistance, the PA input capacitance and just omit the effects of Q you'll see that the the voltage across the capacitor decreases as frequency increases. If you insert a 2:1 turns ratio transformer after the driver, you'll drop the AC voltage by 2 and its resistance by 4. For the same capacitance, the 3 dB cutoff frequency of this single pole filter increases. Also, the lower output impedance drops the driver amplifier voltage gain and may boost stability.

In short, we need a low driver source impedance since the Class C bipolar amplifier base-emitter junction has lots of capacitance To get current flowing into the PA base, we drop our driver output source impedance down *low*.

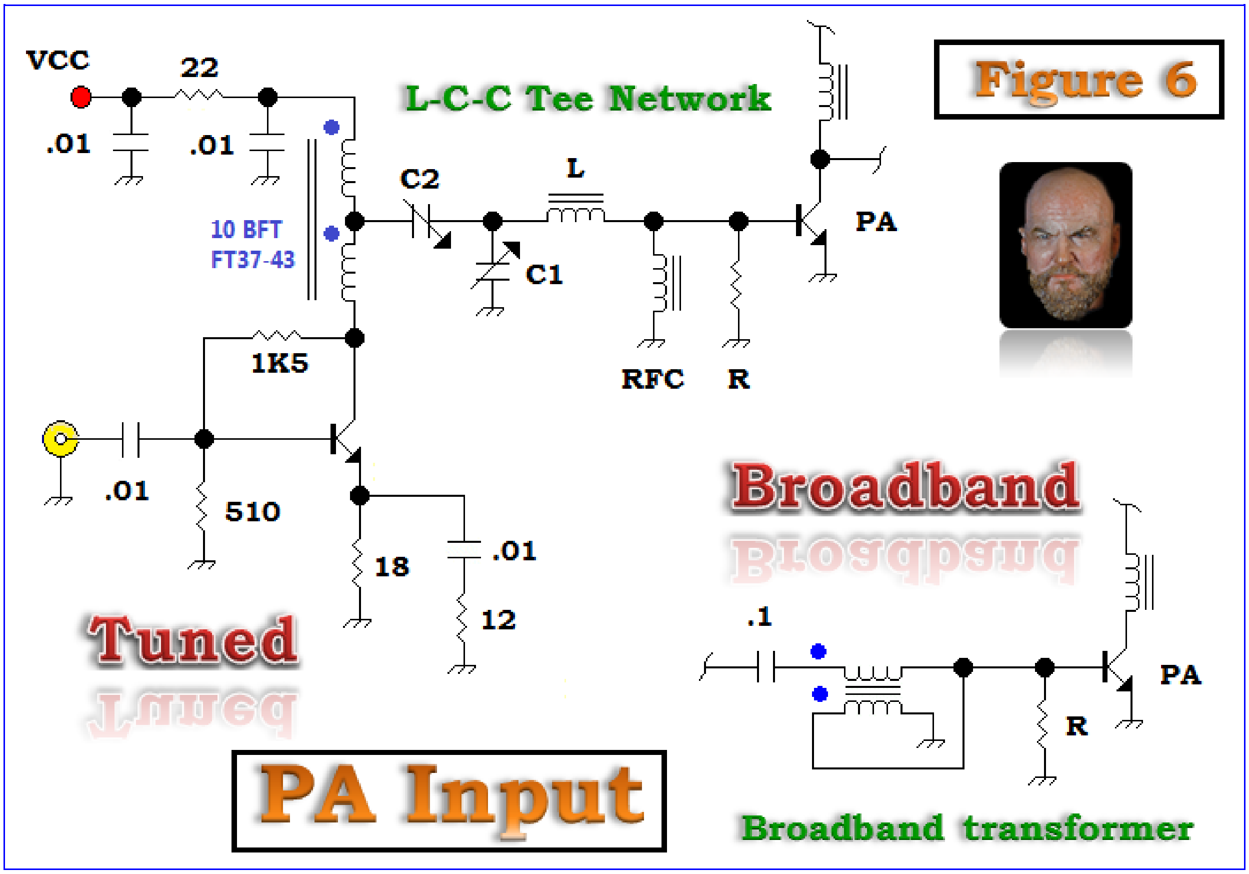
I thought — couldn't we just neutralize this capactance with an inductive output impedance? Thus our PA driver stage would drive a resistive load and everything would be "ducky". I consulted Wes, plus viewed some math and circuitry.

You can't effectively neutralize a bipolar transistor amplifier, since the capacitance isn't constant and behaves more like like a varactor diode rather than the C shown in Figure 5.. Even when forward biased, diode capacitance depends on the instantaneous voltage that appears across the diode. In contrast, 'FETs prove less troublesome and thus it's possible to achieve some neutralization with a 'FET amplifier. Please view the power stage from EMRFD Figure 11.14. Check out the push-pull pair of IRF511s. C5 and C6, at 22 pF each, are neutralizing capacitors.

In some cases, the capacitive reactance may be absorbed as part of a resonant circuit on the PA input. For example, a low-pass filter might soak up some of the C.

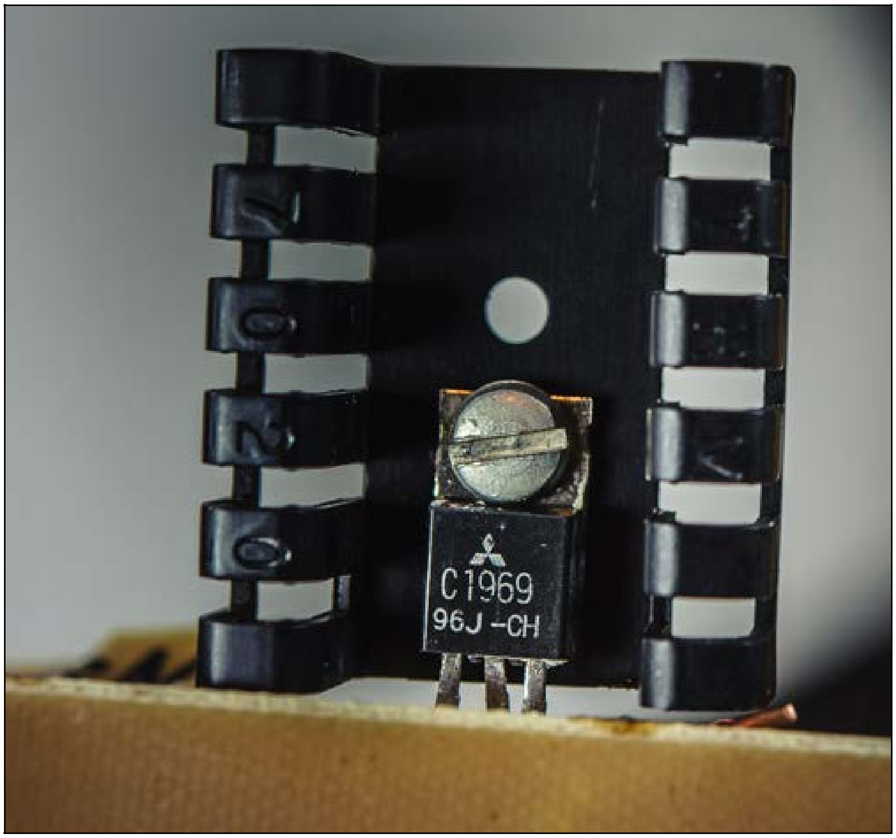
In all cases, our goal is drive in some power and switch the PA transistor solidly on and off (i.e. technically approaching class B but definitely class C because conduction will be less than 180 degrees) — a low output impedance can drive that input capacitance.

Driving final(s) with a very low output Z rings true even if your driving the latest (super-costly) ultra-low loss, high breakdown voltage, silicon GaN FETs to output >100 Watts at 1296 MHz in Class D, E, F, or Inverse Class F. Just get it done!



Above — Figure 6 shows 2 basic methods to drive the PA with a low impedance: single frequency network or broadband step-down transformer. I'll discuss both methods in detail later. Let's move away from theory and lighten things up for a couple of minutes.

Above — A short 10X probe ground lead boosts Tx measurement accuracy; in particular, for FFT analysis with your DSO. You can read or see **why** by searching the web for text and/or videos concerning oscilloscope probes.



Above — The long gone 2SC1969 works great. Beware of counterfeits however.

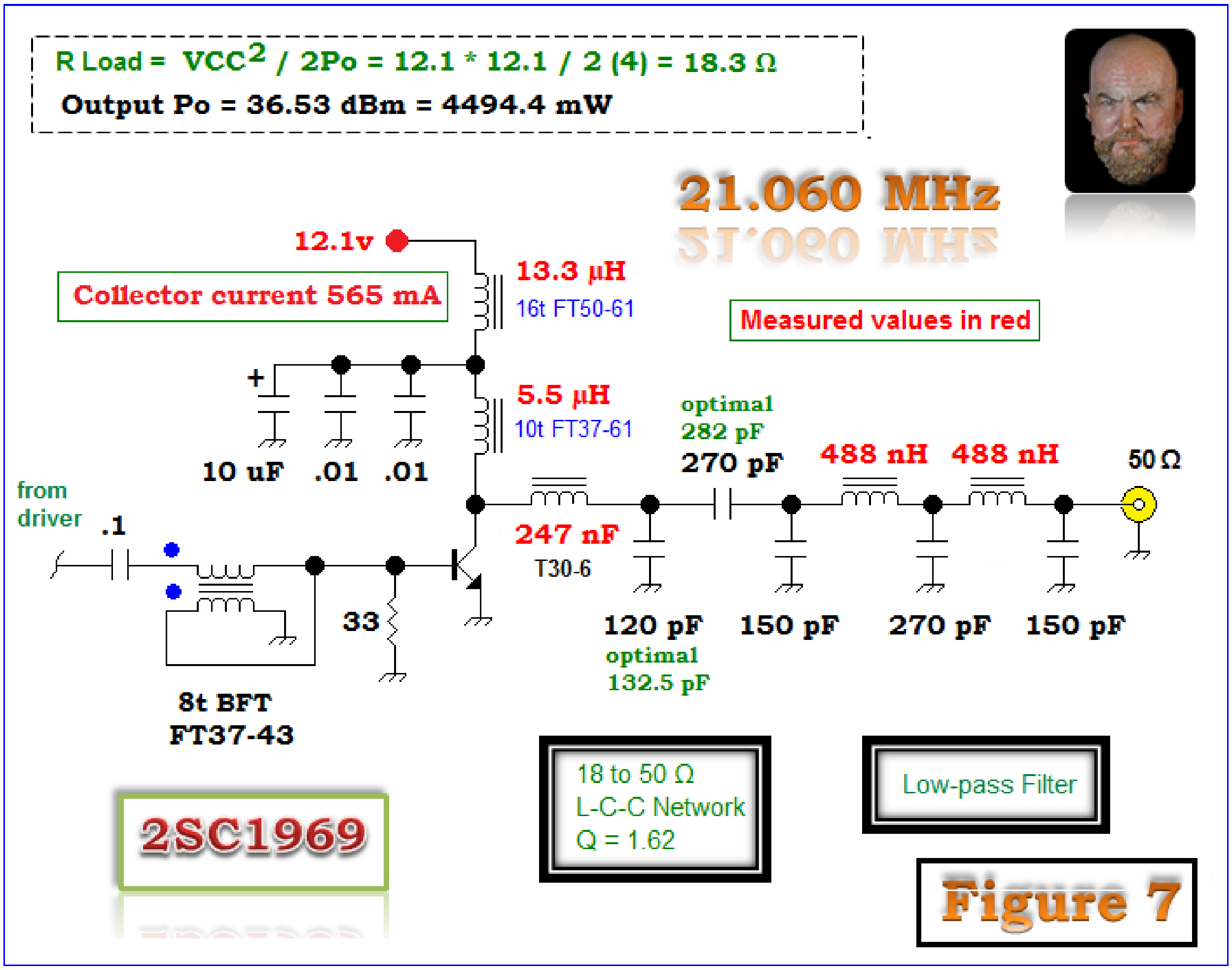
Bipolar transistor selection troubles us since most of the 'old standbys' went obsolete way back when. I've been collecting dusty, old *QRP* driver and PA transistors for decades and my bag includes BD139, MJE181, 2SC2509, 2SC1969, 2SC2166, NTE123, 2N3053, 2N4037 and 2N2219A. For VHF, apart from the 2N5109/2N3866 standards, I've kept a few MRF607, 2SC1971 and 2N3553.

VHF/UHF transistors exhibit tremendous gain at lower HF, but they might oscillate without due care.

Some rules I've gleaned: The fT of a BJT should be at least 3-5 times the operating frequency; the VCEO should be at least double the VCC (more is better) ; don't exceed the maximum heat dissipation, nor maximum current. In short, read the datasheet. I won't cover heat sinking, nor gain compensation for multi-band amplifers on this web page; for it's been done by far better authors.

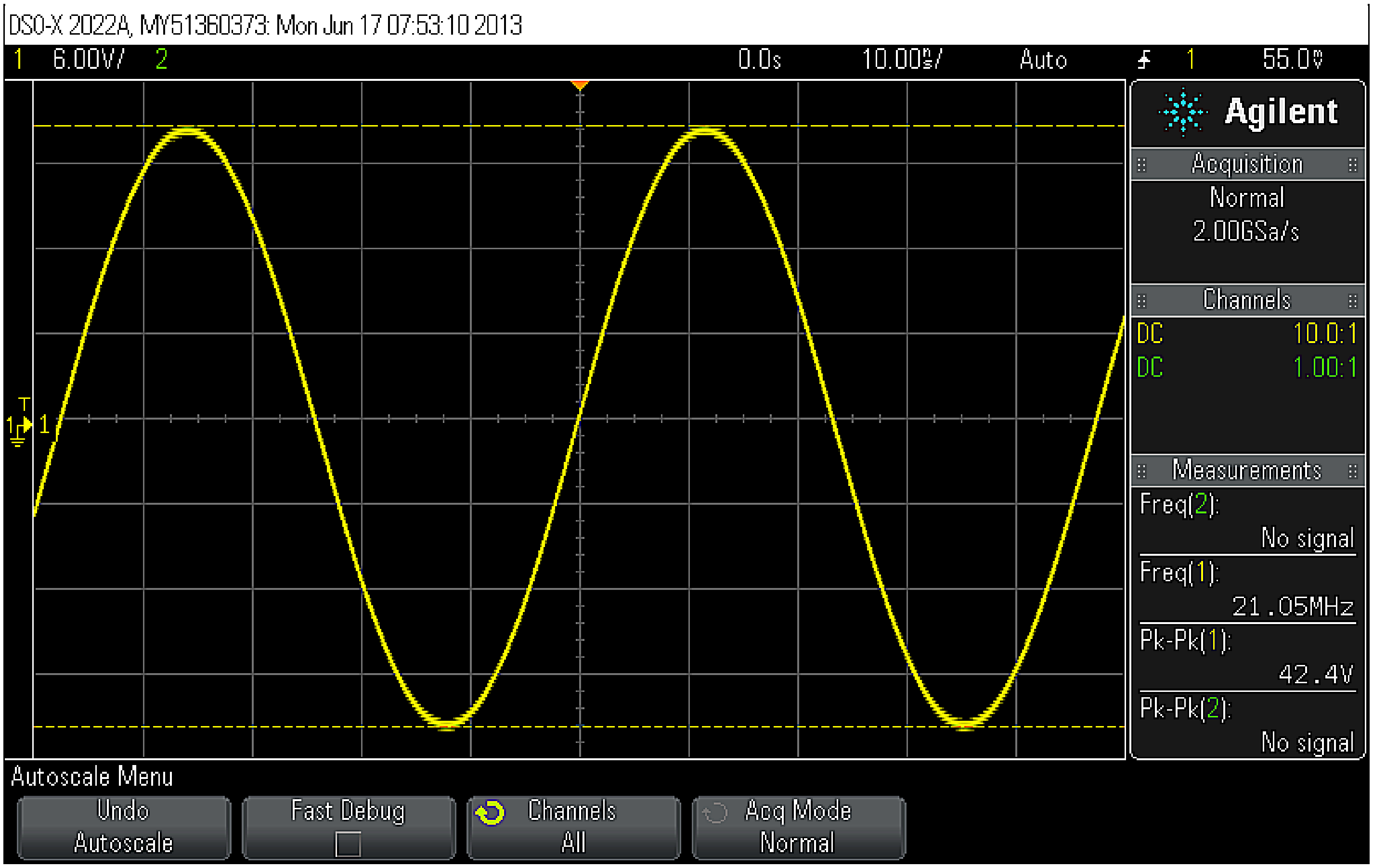
Lets make a basic Class C power amplifier based upon the RF blocks presented thus far.

Class C PA with Broadband Input and L-C-C Tuned Output



Above — Input matched PA via a broadband transformer (a bifilar wound transmission line transformer with a 4:1 Z ratio). A great example of this design includes the Figure 2 of Ugly Weekender transmitter by Wes. The 50 Ω output Z of the driver is transformed to 12.5 Ω. to drive the PA input. While a conventional link-coupled transformer works okay, the closer coupling of a twisted wire transmission line transformer often affords a better impedance transform and lower losses. You might experiment with a 9:1 Z transformer instead of the 4:1 shown. Click for a highly compressed photo of 1 of the full breadboards.

The 33 Ω resistor helps lower gain and stabilize the amp. I ran 33 to 100 Ω applying experimental methods. Read more about the PA base resistor and coil XL requirements in the text following Design Question 6.



DSO-X 2022A, MY51360373: Mon Jun 17 07:53:10 2013

1 6.00V/ 2 0.0s 10.00ns/ Auto 1 55.0V

Acquisition: Normal 2.00GSa/s

Channels: DC 10.0:1, DC 1.00:1

Measurements:
Freq(2): No signal
Freq(1): 21.05MHz
Pk-Pk(1): 42.4V
Pk-Pk(2): No signal

Above — The 'scope output from Figure 7. 42.4 Vpp = 36.53 dBm = 4.49 Watts.

L-C-C Output Matching Network

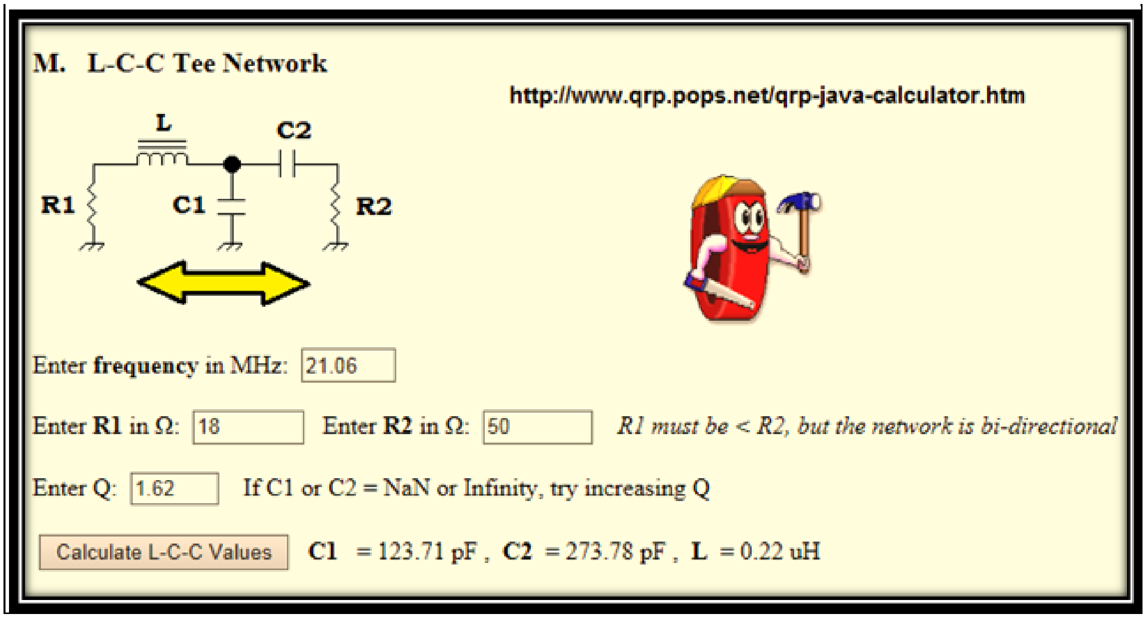
3 discussion points arise from Figure 7: [1] the output matching network. [2] the collector choke + decoupling circuitry, and [3] the low-pass filter. I'll discuss choosing a collector choke and decoupling in Section 2. We design our low-pass filters with tables or software; I used an old ARRL Radio Amateurs Handbook and then tweaked it in GPLA to get standard value capacitors. Click for a GPLA simulation of my filter. The L-C-C network also provides some low-pass filtering.

To calculate the collector output resistance, we use the formula R(load) = Vcc^2/ (2*Pout) as shown in Figure 7 for 4 Watts. For 1.5 Watts at 12 VDC --- R(load) = 12 * 12 / 2(1.5) = 48 Ω --- no output matching network is required; making 1.5 watts a popular transmitter power for new builders eager to learn about Class C.

Design Question 4: How do I make an L-C-C matching network?

Let's stick with 4W power and make an L-C-C Tee network for a 18 to 50 Ω transform. I prefer the L-C-C network over the L-network because you get 2 capacitors to tweak and I find it much easier to match a wide range of impedances compared to the L-network. Since we may pinch, or expand the inductor windings, in reality, the L-C-C gives you 2.5 possible tweaks.

Normally I wind the L on a number 6 powdered iron toroid with 26-28 gauge magnet wire in size T25, T30, T37, or T50. 26 gauge wire allows for good coil squishing and expansion with thumb and forefinger. For design cycle capacitors, I mostly solder in those horrible, low Q, orange, green, or red muRata trimmer caps such as the TZ03Z500F169B00. Although rare, ceramic-mica trimmers work well.



M. L-C-C Tee Network

http://www.qrp.pops.net/qrp-java-calculator.htm

Enter **frequency** in MHz: 21.06

Enter **R1** in Ω: 18 Enter **R2** in Ω: 50 *R1 must be < R2, but the network is bi-directional*

Enter Q: 1.62 If C1 or C2 = NaN or Infinity, try increasing Q

C1 = 123.71 pF , **C2** = 273.78 pF , **L** = 0.22 uH

Above — To calculate L, C1 and C2 we may employ software to obtain starting L and C values. I recommend Wes' ZMAT08.exe, a Ladpac program that ships on the EMRFD CD. I wrote my own java-script applet with less prompting and error checking and put in on the Design Center web page as **M**. Note I used 18 Ω as the load resistance in my calculations.

During actual bench assessment of your network, output your PA to a 50 Ω dummy load rated for that power and preferably measure the peak to peak voltage with a short-grounded 10X probe.

For Figure 7, I wound and tack soldered in a measured L of about 230 nH and placed a 100 pF cap plus 6-50 pF trimmer for C1 and a 220 pF + 22 pF cap + 6-50 pF trimmer cap for C2. The tuning procedure goes like this:

Tune the capacitors for maxmum power. Try squishing the L, then re-tweak C1 and C2 and see what happens. Did the Vpp go up or down? We want it to go up, so if the Vpp went up you needed more L, and perhaps still need more. If you squish the coil again (or remove it, add 1-2 turns and then replace it) and the Vpp goes down, you increased the L too much and need to go back down. If the original Vpp goes up when you expand the coils, you had too much L and need to do the opposite procedure just described. Re-tweak C1 and C2 after every inductance change and calculate the power in dBm and/or watts on your bench.

Through trial and error you'll find the optimal L. If nothing works, suspect a bench error, or device failure. Occasionally, I add too much, or too little fixed C in parallel with the trimmer cap. Another time, my transistor was fried.

Once you've extracted the maximum power, remove the cap(s) that make up C1 and measure them. Do the same for C2. I replace C1 and C2 with the nearest equivalent standard value capacitor or (parallel capacitors) and then repeat the tuning procedure to find the optimal L for these 2 new capacitors. Normally, after optimizing the Vpp, I'll remove and measure L to confirm my L-C-C network design. You can use the L just removed or make a tidier version to permanently solder in.

Once you solder in the "keeper" L, you'll need to expand or compress the links a tad to peak your Vpp once again. Although it sounds like a chore, I love measuring a higher Vpp with each successful manipulation of the L-C-C network; it's quite a rush actually.

In the end, my optimal measured values were: L = 247 nH, C1 = 132.5 pF and C2 = 282 pF. In my final, I substituted C1 = 120 pF and C2 = 270 pF as shown in Figure 7. Although I designed for 4W, I squeezed out 4.49W with my L-C-C network.

Design Question 4: How do I choose my L-C-C matching network Q?

The easy answer is the degree of impedance transformation sets the minimum Q and thus choosing the best Q becomes the central question.

The theory regarding loaded versus unloaded Q + network losses and such may bog down many, but in context, a lower Q leads to a more efficient network with less loss. Low Q values also means we tweak larger value capacitors that minimize the effects of stray capacitance. In the collector output network of Figure 7, the Q of 1.62 gave more power than an earlier Q = 5 network. Further, during design, I sometimes slightly adjust the Q in software so I get closer to standard value caps for C1 and C2.

Don't be afraid to remove C2 and just make an L-network. Again, just experiment on your bench to gain confidence.

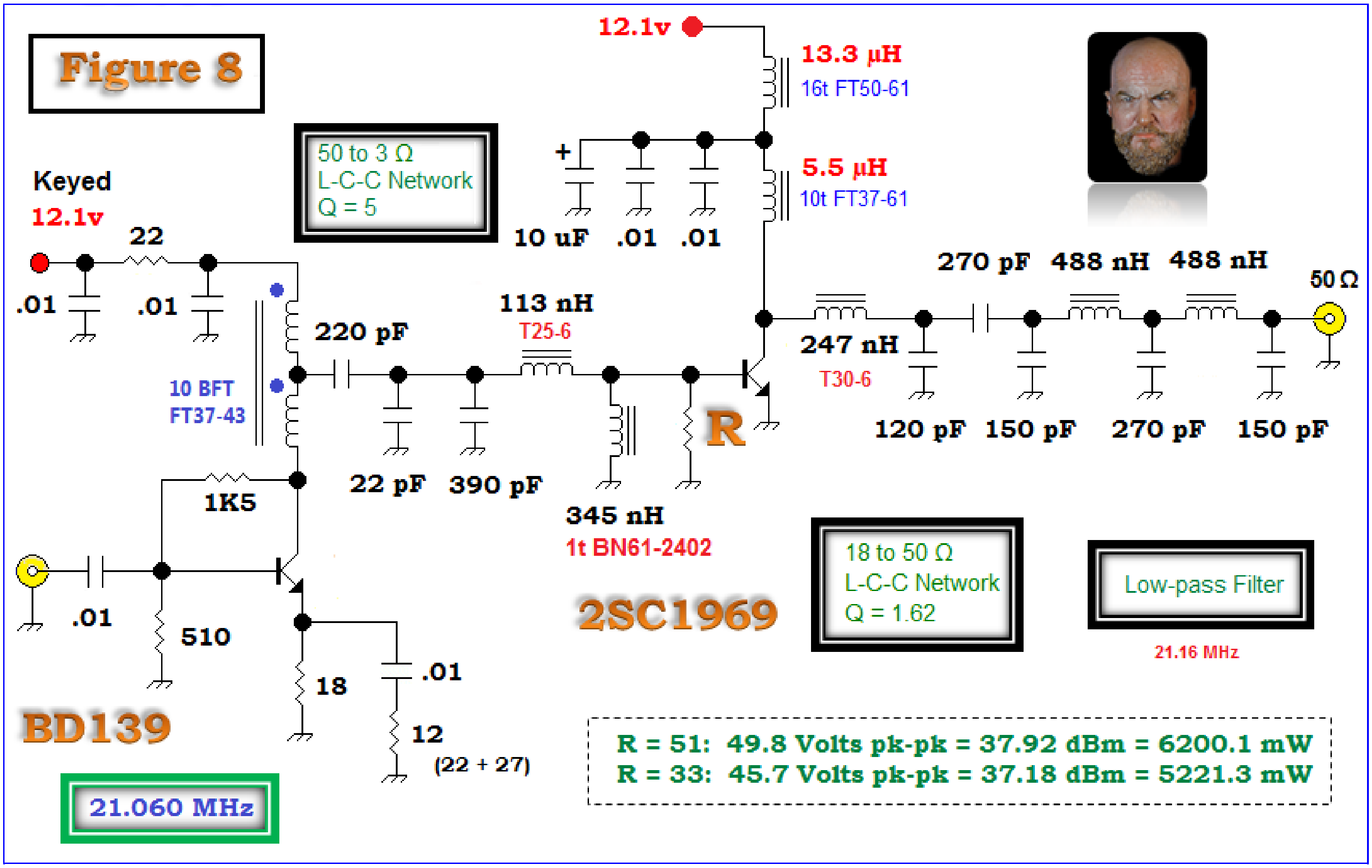
I tried making this circuit with a BD139 in the PA slot and managed to squeeze a mere 3.1 Watts power at 12.1 VDC. Others, such as Wes and Jason, NT7S delivered more power into 50 Ω at and even above 21 MHz with a BD139 PA.

Factors such as transistor specifications, frequency, VCC, and driver power may affect the output power. In the end, the 2SC1969, a final I harvested from an old Cobra CB more than 20 years ago delighted me. I constantly scrounge for old driver and PA transistors in my travels. Sadly, many old CB radios given to me over time suffered from fried finals.
You might try checking eBay, but watch out for fakes.

The BD139 grew to notoriety because it's widely available, cheap and generally does the job at or below ~28 MHz, depending on what output power your seek. I found wide variation in Beta (or hFE) and fT in the random batch I tested. Also remember we're hitting them with lots of current and they heat up and specs change. At 7 MHz, the BD139 works great. The BD139 reminds me of the MPF102 JFET; you never know what your going to get. The best I've tested were Fairchild BD139s and woefully, they're all gone.

Class C Power Amp with Broadband Input and L-C-C Tuned Output

Let's make a PA with an L-C-C Tee network tuned input: Figure 8.



Above — Figure 8. The Figure 7 PA with a tuned input network smashed my world with 6.2W power. Click for the FFT showing the second harmonic down at least 55 dB. I later confirmed this with spectrum analysis.

Design Question 5: How do I choose my PA input impedance?

PA Input Impedance

Calculating the input Z of a transistor with 100's of mA emitter challenges us.The familiar calculations like Zin = (Beta +1) x 26 /I e (mA) don't work out perfectly since our transistor is hot and Beta changes, and in my case I'm way up in fT. I asked Wes and he suggested I assume an input Z of 3 Ω (with about 200 pF of parallel C). Thus , I chose a 50:3 Ω transform to design my L-C-C network.

From my experiments, bench tweaking an L-network, or L-C-C network will accommodate anything even close to 3 Ω input impedance — 3 Ω seems quite actionable. Tune your L-C-C network in the manner described for the PA output match. Since your transforming 50 Ω to 3 Ω, the minimum Q will rise.

Design Question 6: How do I choose my PA input choke and resistor when fed with an L-C-C network?

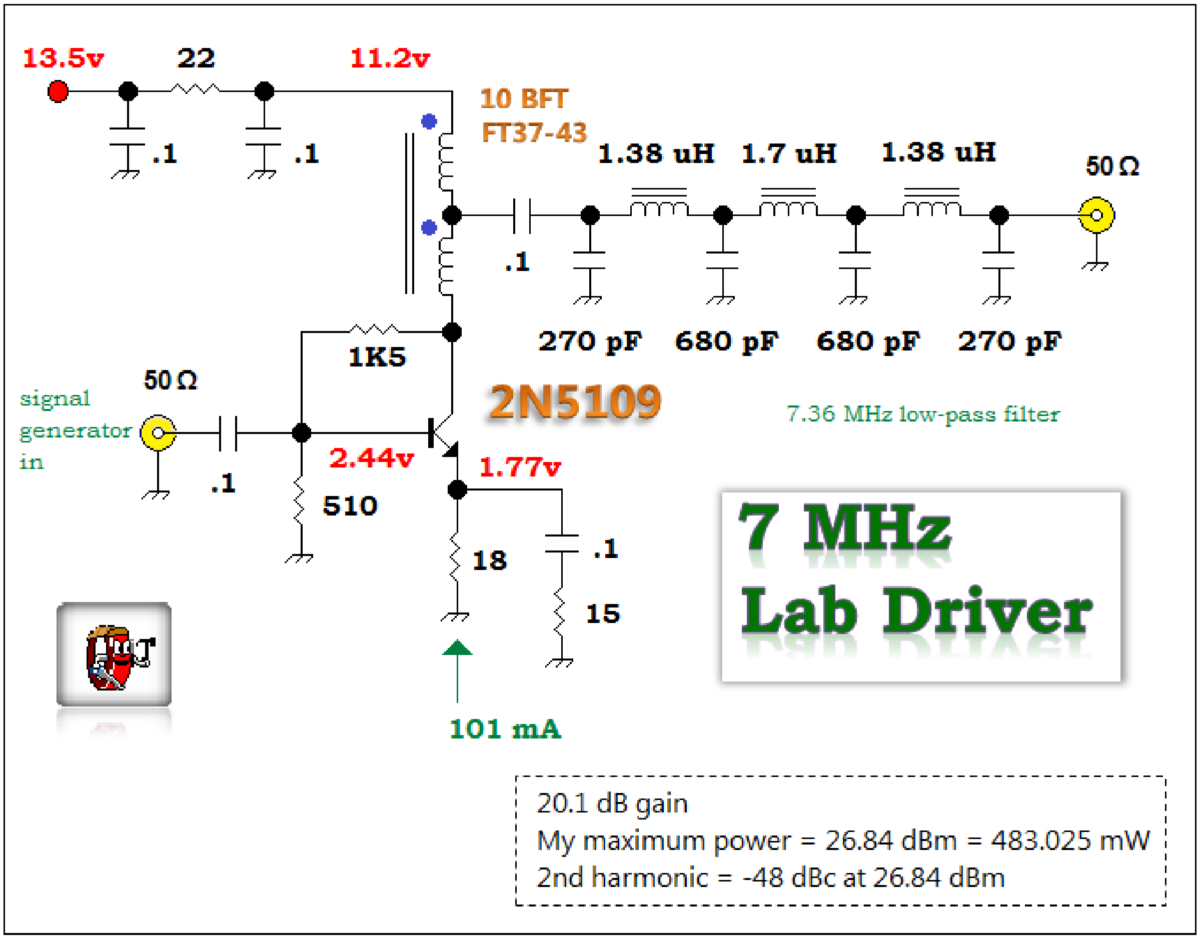
The RFC serves as a DC return path for the PA base just like in the transformer in the wideband version. We normally apply a rule that the minimum inductive reactance of a choke should be 10x the impedance it's connected to, so that's 30 Ω XL.
Since F = 21.06 MHz, the minimum choke value will be 30 Ω / 21.06 MHz x 6.28) = 227 nH. In order to keep the SRF up and for ease, I ran a single turn of 26 gauge wire through a BN61-2402 binocular core and measured 345 nH. Good enough. The # 61 mix will exhibit some Q. Likely # 43 material will work fine too.

The resistor labelled R serves to reduce the Q of *my* particular RFC, lowers gain and boosts stability. With R = 33 - 51 Ω, the amp held stable, but if I tried to increase R above 51 Ω spasm erupted. The resistor affected output power as shown in Figure 8. Again the resistor should minimally be about 10X the input Z or so. I tried an R of 10 Ω and the output power dropped down to a little under 3W as more of my driver output just flowed through the resistor to ground.

To conclude, I executed the time honored tradition of choosing parts and circuits to extract power from a crystal oscillator-driven analog RF amplifer chain. Much of this information extends to linear operation (Class A, AB) and general RF design. Have fun designing your own transmitters and as always, apply experimental methods!

Section 2: PA Measurement with Examples

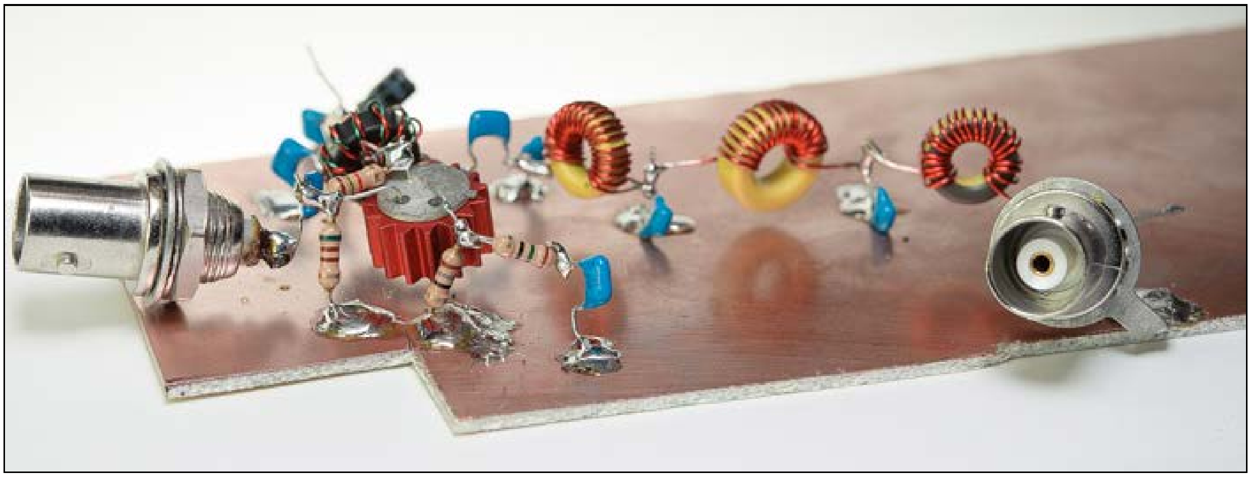
In this section, I posted some PA measurement ideas + experiments and comments about PA collector chokes.



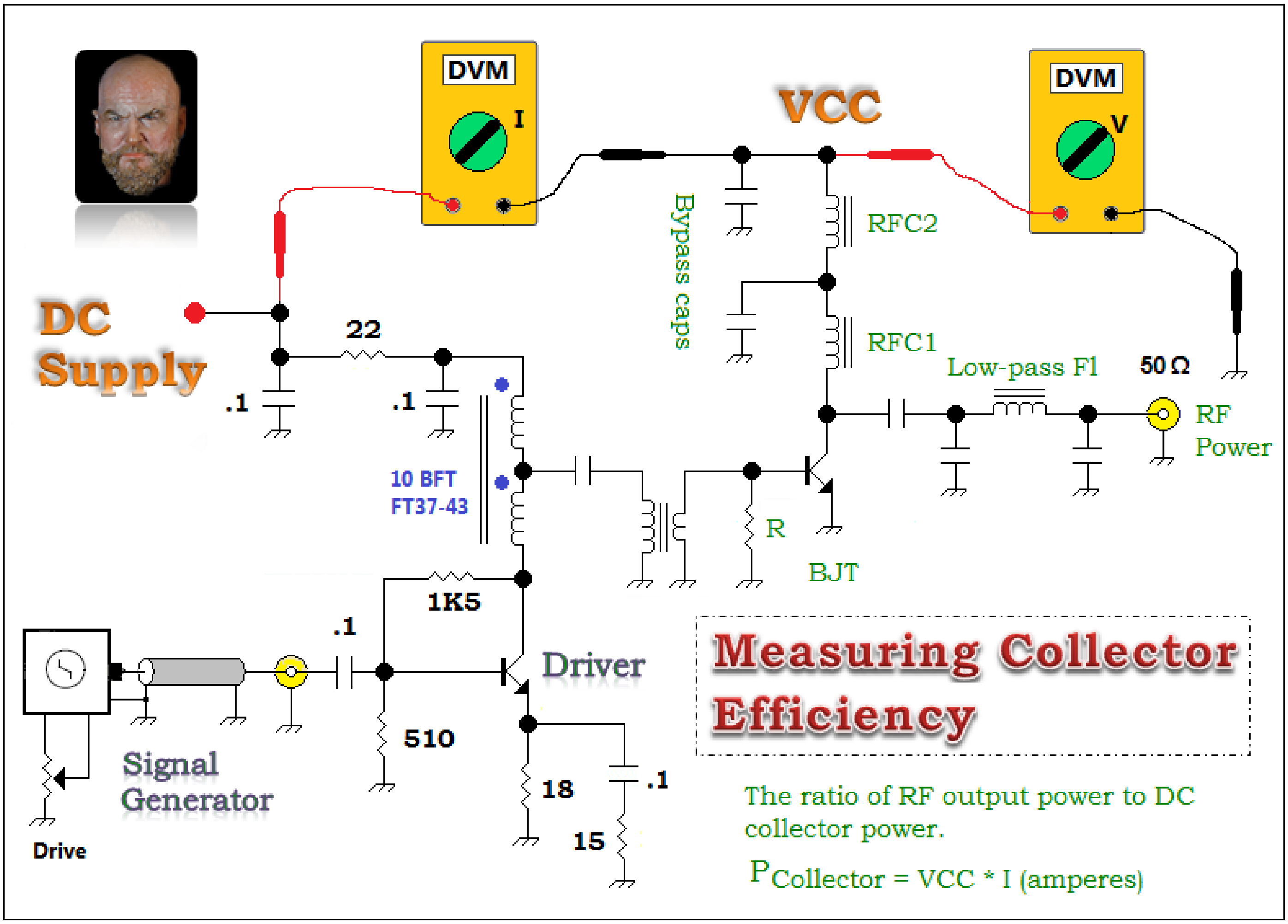
20.1 dB gain
My maximum power = 26.84 dBm = 483.025 mW
2nd harmonic = -48 dBc at 26.84 dBm

Above — To boost my signal generator output for driving *any* experimental PA up to about ~20W output, I built a high-current, sine wave driver. I excite this amplifer with my variable gain homebrew HF signal generator — in the version shown above, a 7 MHz low-pass filter is soldered on the amplifier breadboard. I'll stick this amp in a metal box sans filter and then apply the correct bench module low-pass filter via a BNC connector. Click for a photo of 3 of my HF low-pass filter bench modules.

Normally the low-pass filter output is pad attenuated to buffer and reduce the output voltage since it's rare I need up to 483 mW drive on the QRP workbench.



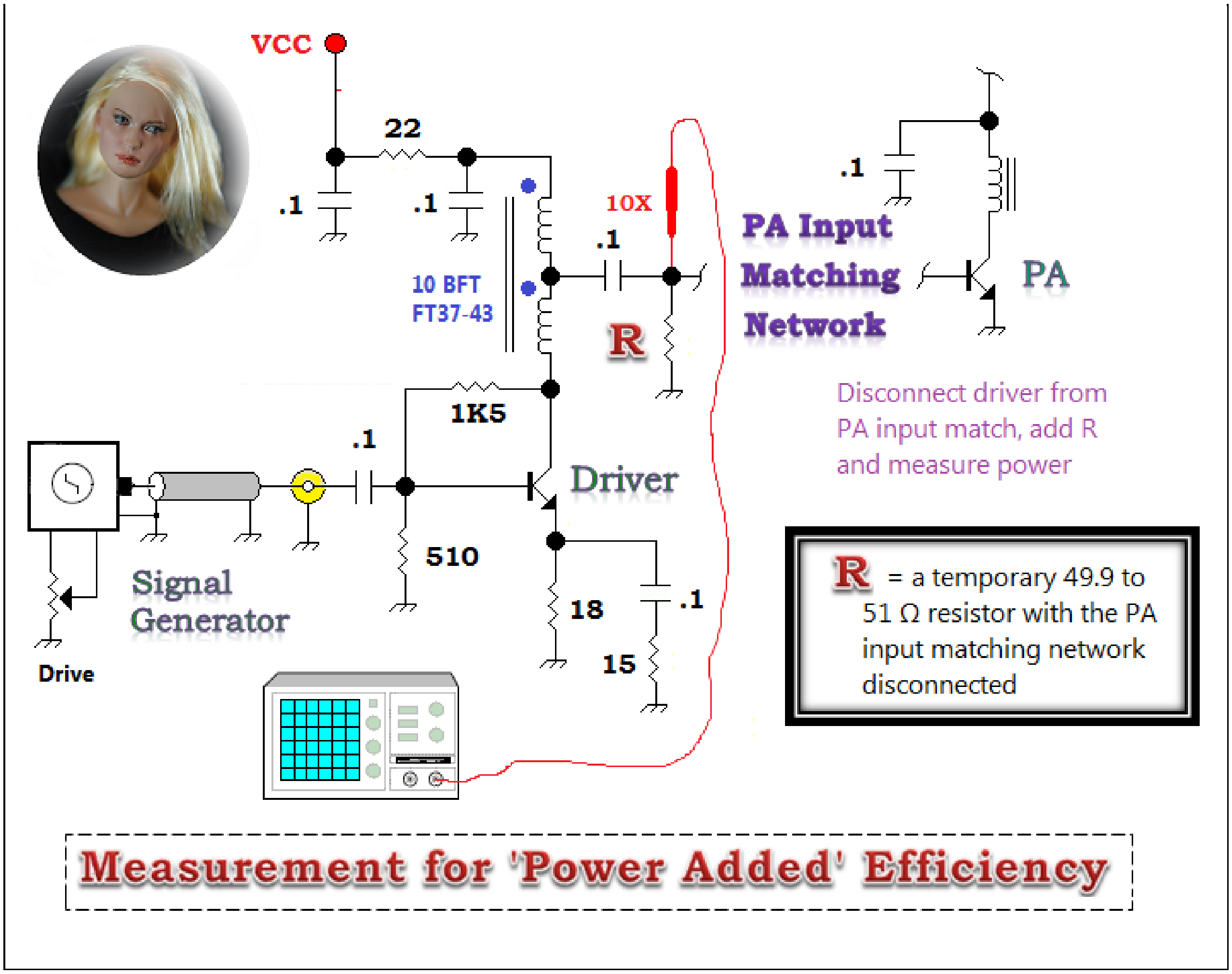
Above — A photo of the PA driver amplifer breadboard.



Above — A method to simultaneously measure collector **V** and **I** to calculate DC collector power. Two DVMs boost measurement fidelity since the current measuring ammeter's internal resistance will drop the DC supply ~ 0.2 to 0.5 volts depending on the device and current. Collector efficiency = RF output power / (V * I) .

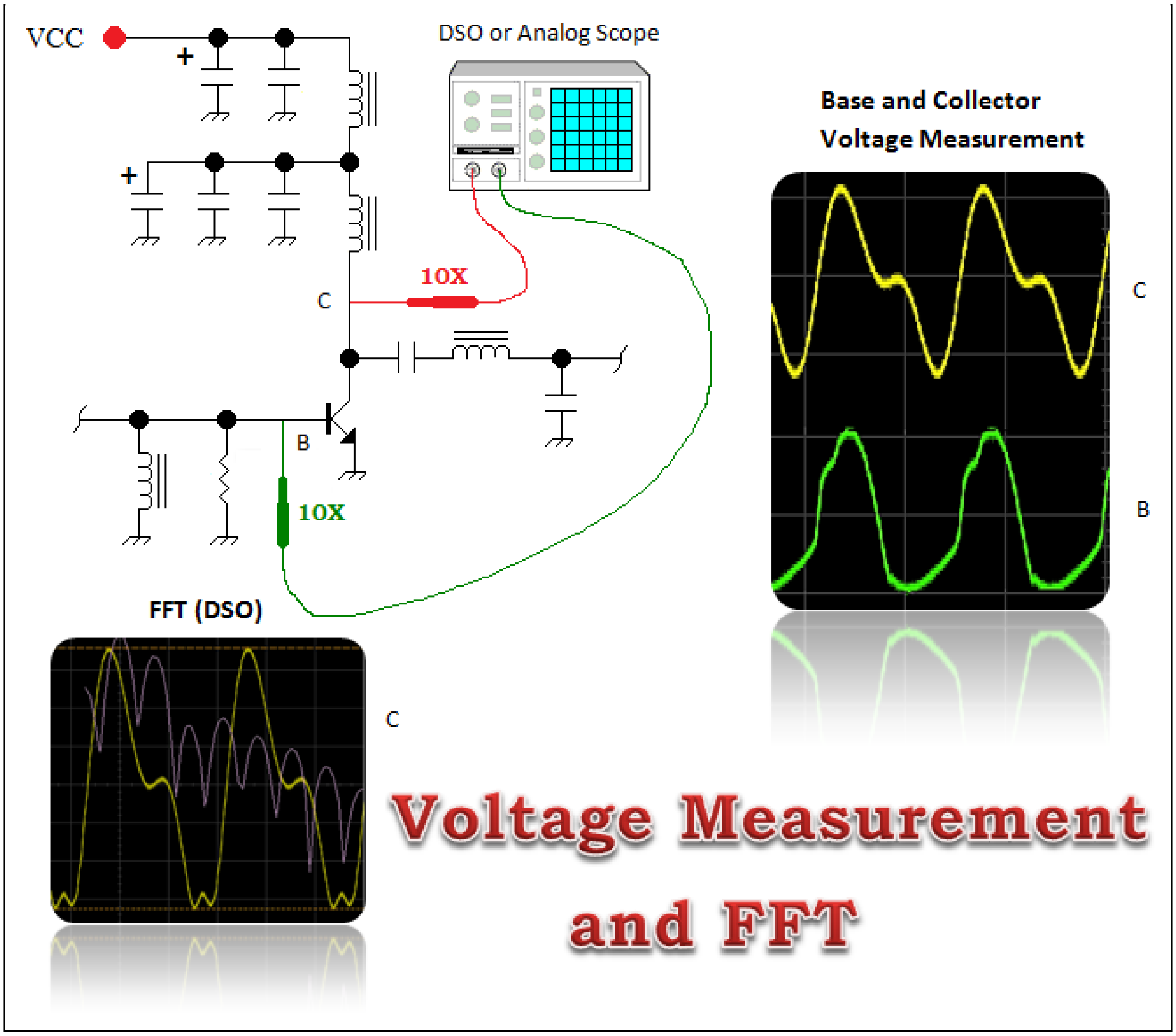
This method provides a great excuse to purchase another multimeter.

Measure VCC where shown since the decoupling and bypass filters help prevent the AC signal from affecting the DC measurement. We only touch our 10X 'scope probe on the collector. Exert due care and wear eye protection in case you blow up the transistor and create shrapnel.



Above — Power Added Efficiency considers the base drive power in addition to the DC collector power. The formula = RF output power / (DC Power + RF drive power). Although uncommon, measuring drive power helps you to assess and compare both input matching and PA efficiency metrics.

At the very least — I like knowing my PA drive power and so employ driver amplifiers with 50 Ω output impedance to allow this power measurement. I often permanently solder a 49.9 or 51 Ω resistor to the breadboard ground plane and temporarily connect this R ; plus disconnect the input matching circuit to measure driver power when I've finished tweaking my collector to antenna port circuitry.

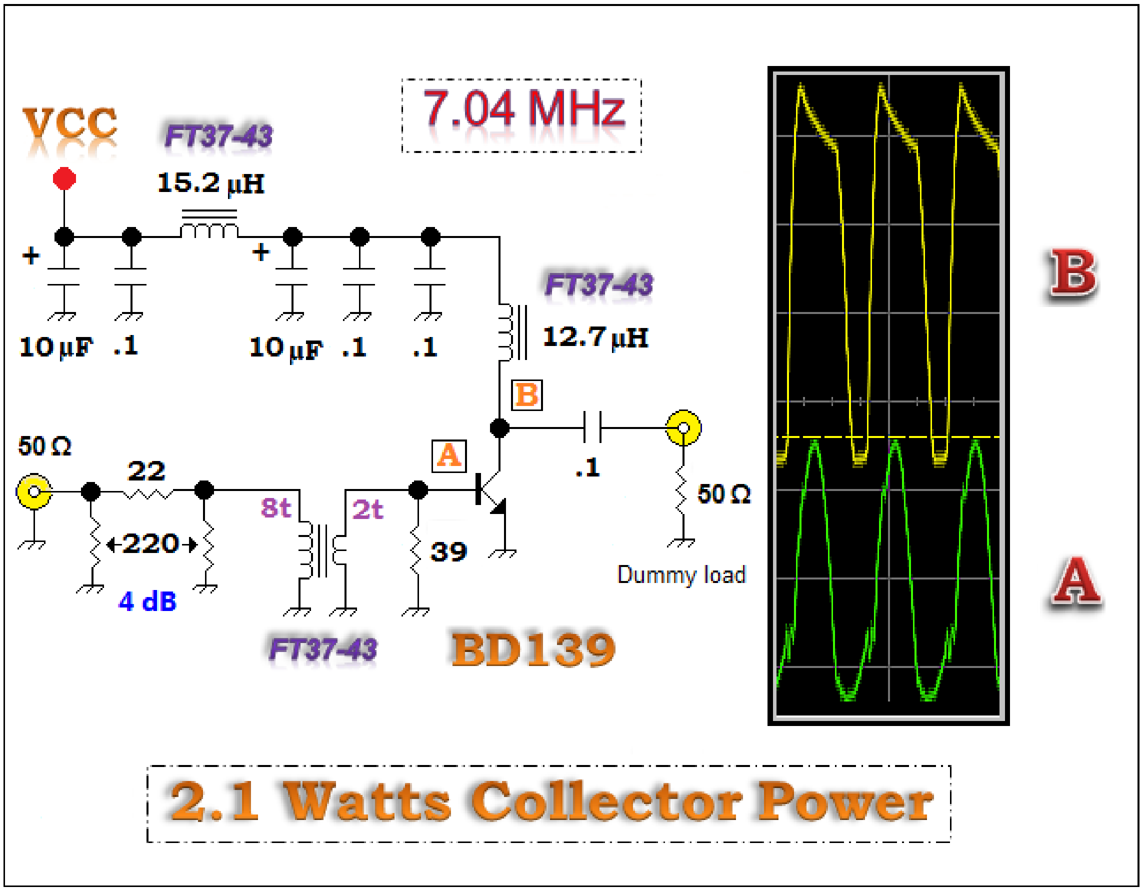


Above — Oscillocope measurements of the base and collector voltages. Most of us enjoy simultaneously running 2 channels on our 'scopes. With a good quality DSO, FFT analysis of the collector waveform may yield insight, however, this knowledge might also confound us since interpreting our measures is fully half of the task.

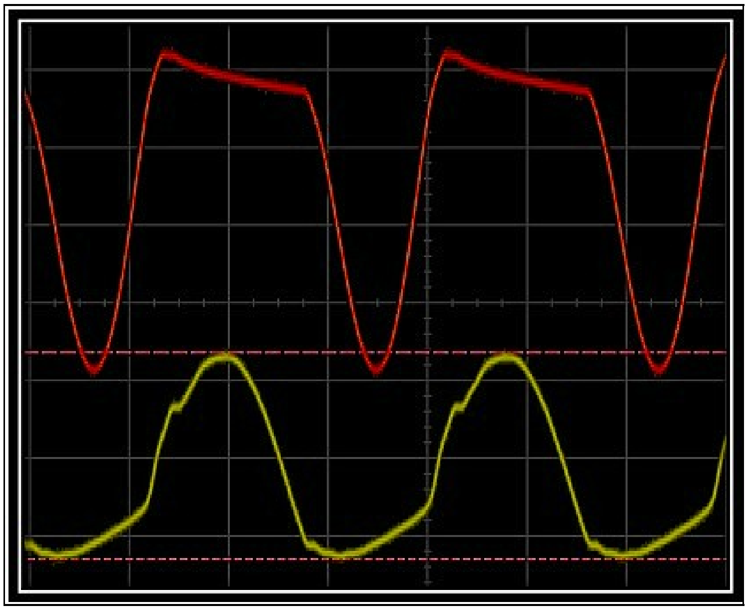
I often struggle to properly interpret measures. Some builders blog about or email me that they enjoy an innate gift at understanding electronic circuitry — I don't. I guess I lack the knack? For some of us, problems and complications block progress, but more often than not, we eventually cross the finish line.

Above — 1.5 Watt, Class C breadboard used for the experiments that follow. I changed the collector choke (RFC1) and applied various filters to examine some of the measurement techniques aforementioned. Click for a bigger photo.

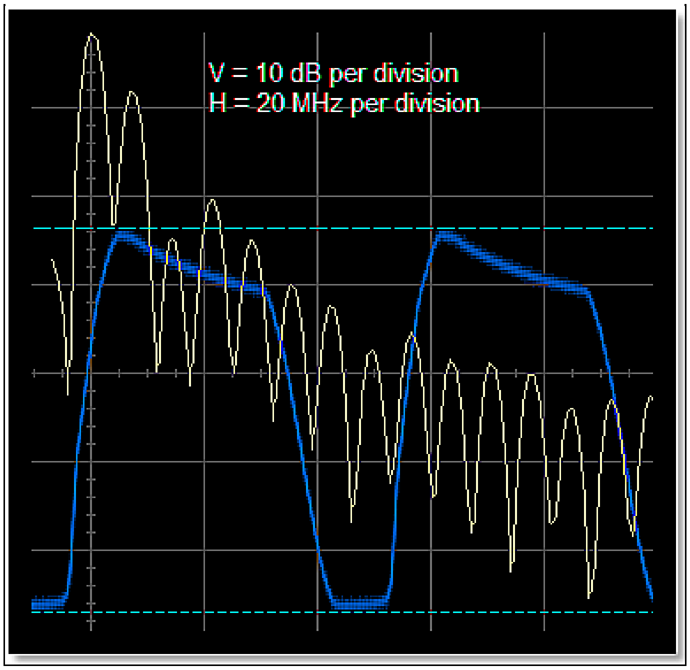
The Collector Choke



Above — A Class C amplifer with it's collector AC coupled to a 50 Ω resistor load. *No filter.*



Above — Base and collector waveforms. The unfiltered collector waveform (top) rich in even and odd harmonics ~resembles a square wave. The maximum collector voltage occurs when the current is 0 and in concert, the maximum collector current flows where collector voltage falls to near 0. As the BJT shuts off, the collector circuit kicks and rings so that the DC collector voltage may rise as high as twice VCC.



Above — An FFT of the unfiltered collector showing F1, F2, F3, F4, F5, etc. Distortion city!

The Collector Choke

I've read web pages suggesting we might boost PA efficiency by decreasing the collector choke inductance. While that's possible, in reality, it's not that simple. Experiments involving factors like collector load resistance, angle of conduction, RFC inductance/Q, matching networks, base drive configuration and output filtration are ripe with complicated math and theory that I struggle to understand — entire books and research papers explore these topics as author(s) configure their PA into Class D, E, etc. to boost efficiency. The published design work and PA devices out today are truly breath taking.

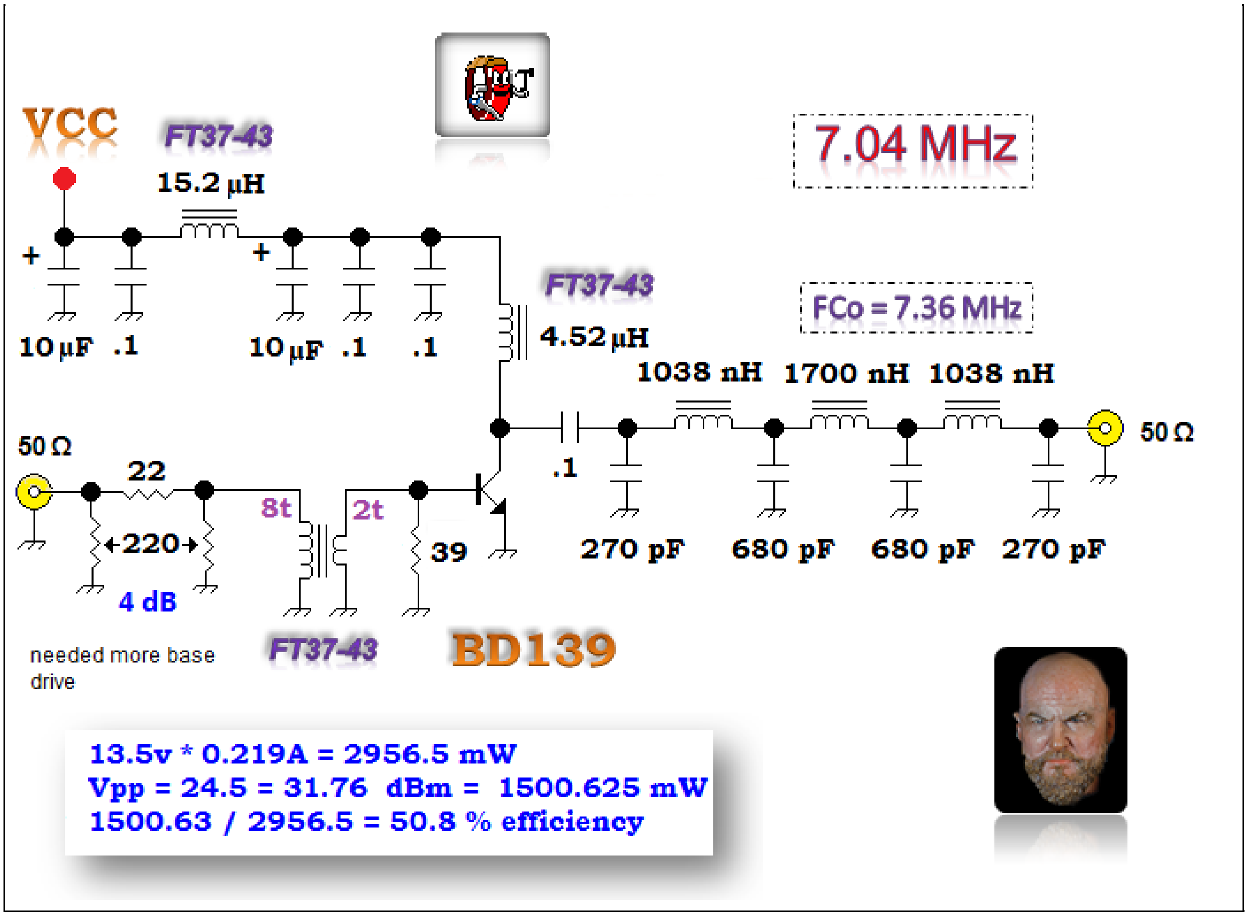
We should likely think of the BJT, collector current + voltage, the RFC, base drive and output networks as a system of parts that interact — changing 1 part might change collector current, impedance, efficiency, harmonics, etcetera. As such, the experiments that follow do not construe science; only some experiments by a neophyte to ponder and build on.

The ideal RFC passes only DC — i.e. exhibits infinite impedance to AC, but a short circuit to DC. The primary purpose of the choke is to block our high-power, distorted AC signal from flowing on our DC power lines. Some authors suggest that the minimum inductance for the inductor should be 5-10X the collector resistance and I'll leave that up to you. At HF, I wind my RFCs on #43 or #61 material ferrites while keeping the turns number low as I remember that the coil will exhibit a self resonant frequency that might cause problems if it lies near a frequency I need to filter.

1.5 Watt Amp with Three Different Collector Chokes

The 1.5 Watt obviates the need for a matching network since the collector load resistance = 48 Ω with a VCC of 12v. I rebelled and ran 13.5 V supply for a calculated 60.8 Ω collector load R. I probably should have this matched this collector load to 50 Ω at 13.5v, but wanted a "real world" amp without a matching network. At the very least, these amplifiers illustrate the PA measurement techniques shown above. I wound the RFC on a common FT37-43.

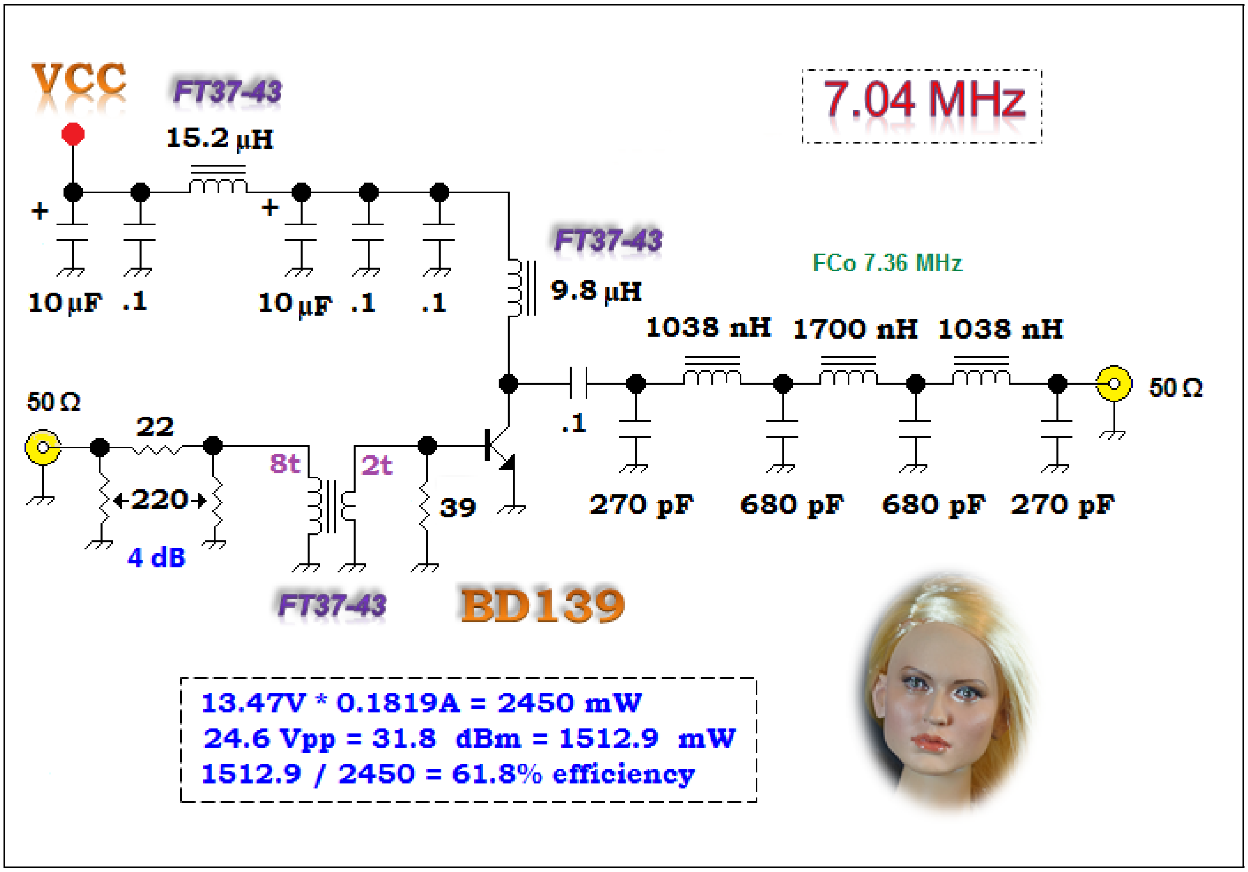
In all cases, I adjusted the base drive so that the sine wave PA output read ~24.5 Vpp while measuring collector V and I. I made DSO measures of the the base and collector voltage and collector FFTs, although forgot to save some of them as screen captures.



needed more base drive

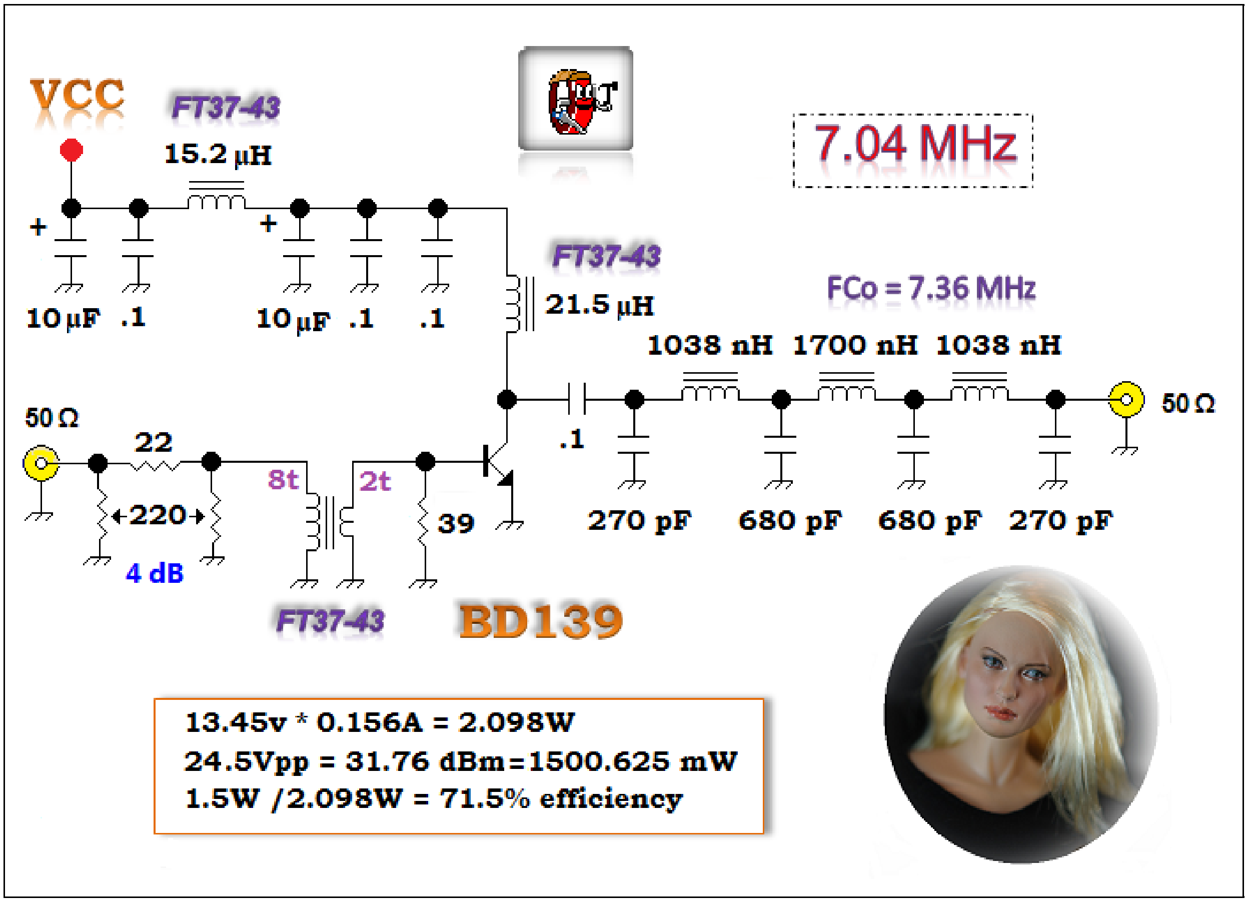
13.5v * 0.219A = 2956.5 mW
Vpp = 24.5 = 31.76 dBm = 1500.625 mW
1500.63 / 2956.5 = 50.8 % efficiency

Above — The collector choke = 4.52 µH or an XL of ~ 200 Ω @ 7.040 MHz. My measurements indicate an efficiency of 50.8%. With a choke XL of only ~ 200 Ω, some of my collector AC might be flowing to ground through the bypass caps and that'll kill efficiency. Click for an FFT of the collector. This circuit required more base drive than the other 2.

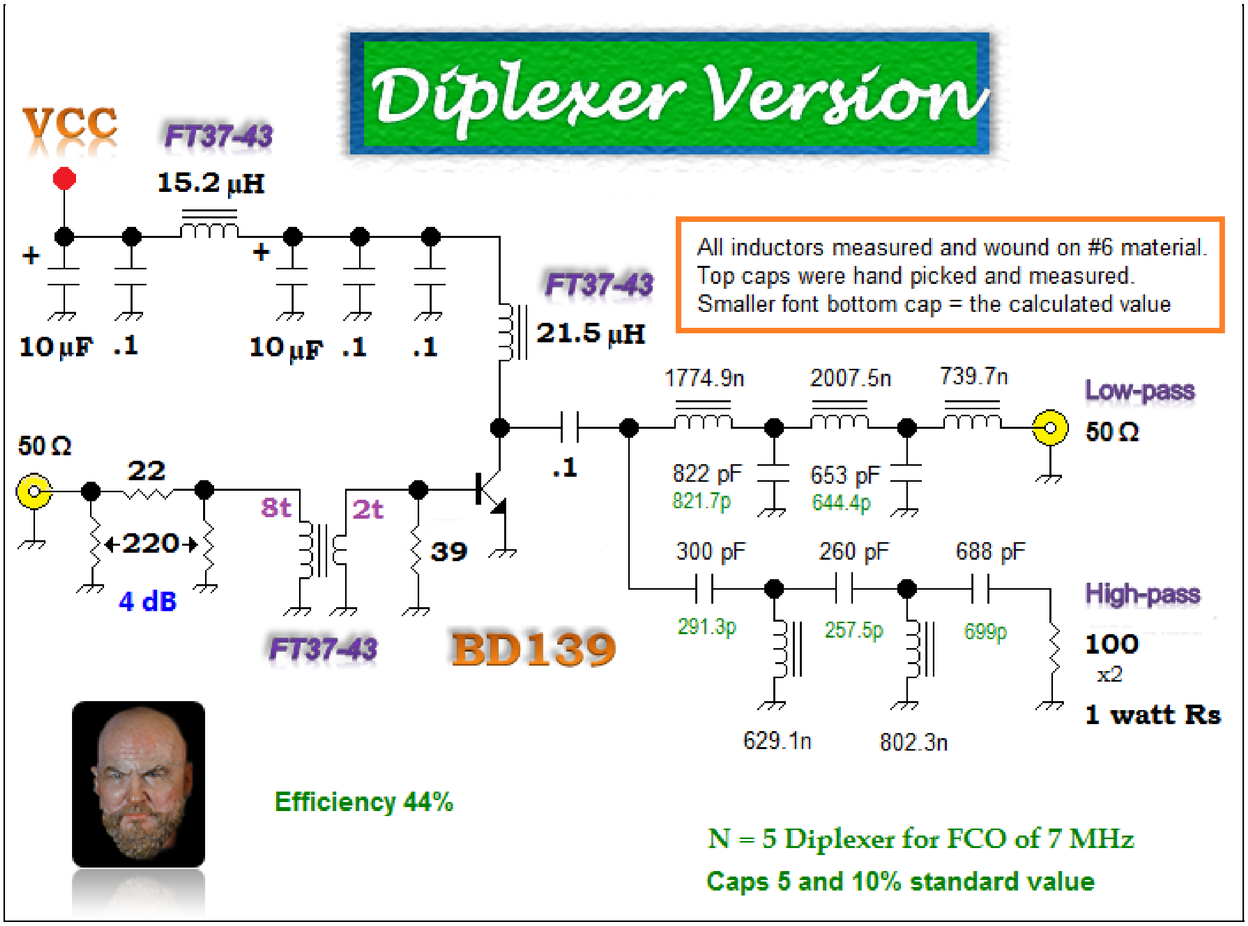


13.47V * 0.1819A = 2450 mW
24.6 Vpp = 31.8 dBm = 1512.9 mW
1512.9 / 2450 = 61.8% efficiency

Above — The collector choke = 9.8 µH = an XL of ~ 434 Ω. Efficiency now 61.8%. Base and collector tracing. Collector FFT.



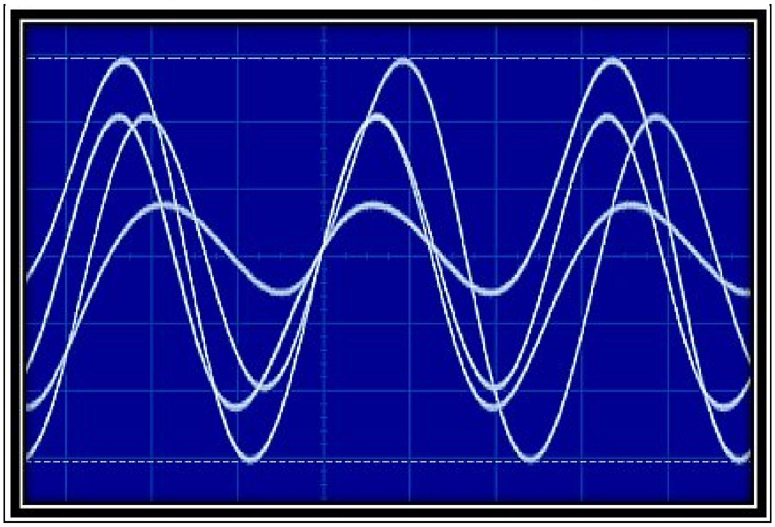
Above — The collector choke = 20.5 µH = an XL of ~ 951 Ω. 71.5% efficiency. Base + collector. Collector FFT. Sine wave FFT. To date, 71.5% is the best Class C efficiency I've measured.



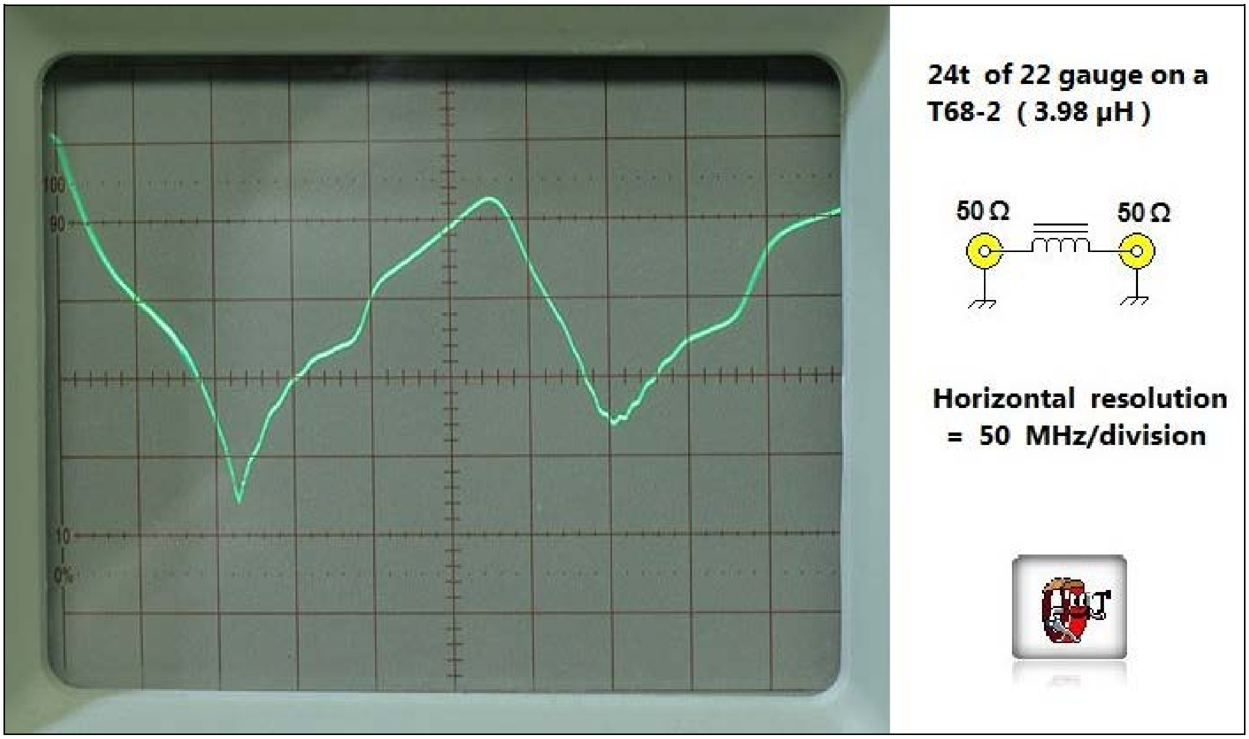
Above — A PA terminated in a N= 5 diplexer for 7 MHz. Click for the FFT of the collector. The collector waveform looks telling — resembling the 50 Ω resistor terminated version shown earlier. Unshown; I lowered the signal generator to 6.99 MHz and drove it to a sinusoidal 1825 mW to obtain a true 50 Ω collector load resistance plus get the greatest low-pass output signal and efficiency (44%) . Although diplexers will 50 Ω terminate the collector at all frequencies, they don't afford high collector efficiency.

Rogues Gallery

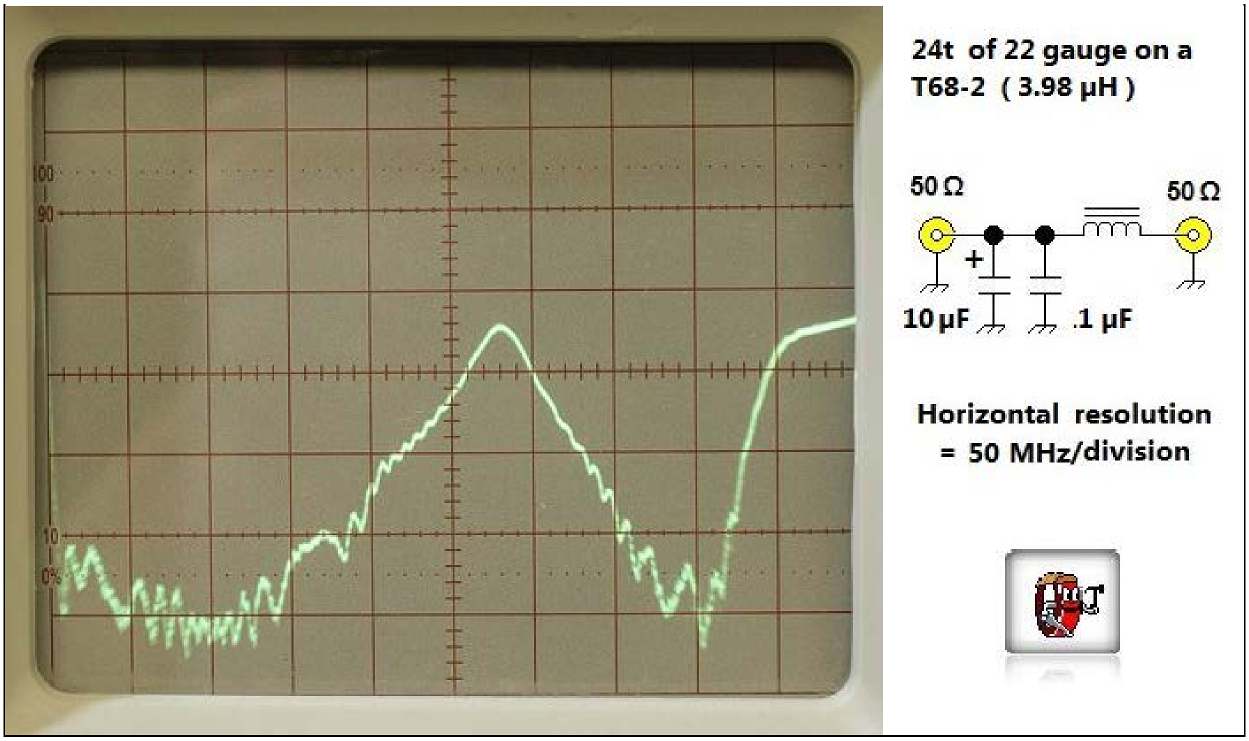
Above — Some 5W CW amplifier collector waveforms as I tried to boost efficiency by manipulating a matching network to drive up the ringing; all failures. These seemingly simple Class C PA's aren't so simple after all. Although failures — I felt thrilled by these experiments and vow to get better at it.



Above — A sine wave output of a very unstable PA.



Above — To observe a renegade series resonant frequency, I wound a choke for a 21 MHz amplifer on a T68-2 powdered iron toroid: a casual 24 turns of wire. Sweeping it shows 2 SRFs at roughly 115 and 350 MHz. 2 SRFs jarred me.



Above — I placed a couple of caps on the cold side like we normally do for our PA chokes. Likely I should have used a 0.01 µF cap instead of the 0.1 µF shown for 21 MHz, but the result would be the same: 2 SRFs remain and poor wideband bypass arises. We clearly need to consider SRF in our chokes and how it might affect our PA function and DC line filtering.

Conclusion to PA Measurement

With an oscilloscope and 2 DVMs you can easily explore boosting your Class C PA efficiency. I've seen precious little info on this topic in the amateur RF experimenter literature. 1 reputable reference = a section in EMRFD called **A Look at Some High Efficiency Amplifiers**. For now, I'll temporize. Likely you'll advance faster than me, since I'm more a receiver guy and don't devote enough time to study transmitter design.

The math and concepts of efficient PA design within Class C - E flummox me. Most have just switched to driving FETs with a square wave built up from parallel inverters (high-speed logic gates in the now obsolete 74AC series) — but hopefully our experiments tweaking BJTs in Class C will carry us forward into other Tx modes.

Measuriing collector power with 2 DVMS while watching the sine wave output in your 'scope gives you incredible feedback: you can even observe thermal runaway. After purposely advancing the base drive to "heat up" a BJT, I saw the Vpp progressively drop in the 'scope, the DC voltmeter drop somewhat and then the DC current meter 'take off' : 400, 500, 890, 1200 mA and suddenly, poof, the BJT was on fire, exploded and flung 2 leads into the air. Normally, they just smoke and stink up your lab. I put a picture of a couple of blown BD139s in the photo section below. Expect to burn up some BJTs during experiments and please work safely.

Section 4: Class E Amplifier

This section unposted.

Section 5: References

1. Solid State Design for the Radio Amateur by Wes Hayward, W7ZOI and Doug DeMaw W1FB. American Radio Relay League 1986.

2. An Updated Universal QRP Transmitter by Wes Hayward, W7ZOI. American Radio Relay League: QST for April 2006.

3. Class E Amplifiers Part 1: Class E Basics. Dan Tayloe, N7VE / Ozarkcon Class E Presentation, 1994. *Search for it online*

4. The Handiman's Guide to MOSFET "Switched Mode" Amplifiers by Paul Harden, NA5N Part 1 Introduction to Class C,D,E and F. First Published in the journal "QRPp". Paul's Site

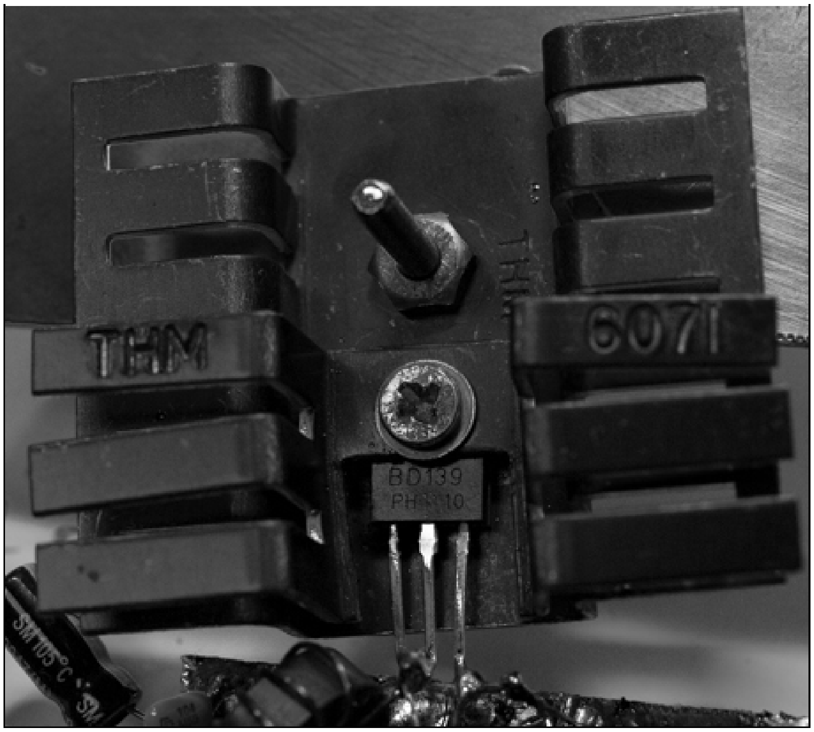
---

Section 6: Miscellaneous Figures and Photos


8047
C3039
M0H4


2N3553
K0350
2N3553
K0350


2I 06000
WC9838


THM
6071
BD139
SM 105°C

HF - Embarcadero

Respository web page for HF experiments performed in Fall 2013 and Winter 2014.

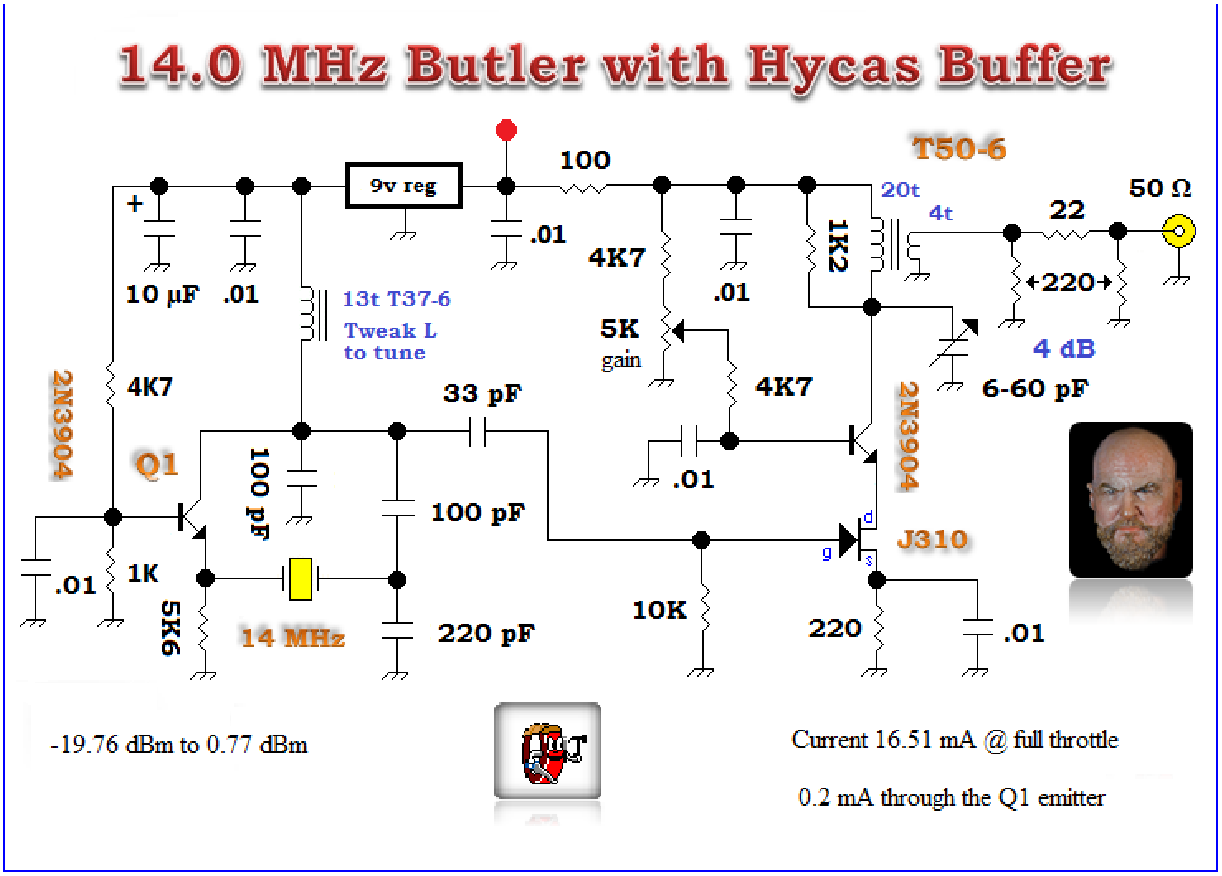
---

1. IMD Oscillator Quest

To measure IMD and calculate IP3 in receivers and components, we need 2 oscillators that typically lie 20 KHz apart, although some apply other frequency spacings. We join these oscillators with a 6 dB (or 3 dB) hybrid combiner and often follow the combiner with an RF attenuator and/or low-pass filter before connecting to a DUT and detector. I sought an oscillator possessing low phase noise, low distortion, strong return loss + high reverse isolation and documented my experiments to fuel your own. Nestled half-way within the HF band, 14 MHz is a popular IMD oscillator frequency choice. I chose 14.0 MHz and 14.020 MHz for classic 20 KHz spacing.

Wes, W7ZOI well covers intercept measurements in EMRFD and in part, popularized receiver DR measurements with his QST article entitled Defining and Measuring Receiver Dynamic Range in QST for July 1975. Click for the article header only. After publishing his article, receiver DR testing by the major Ham transceiver manufacturers became standard.

If you own 2 crystals spaced 20 KHz apart, you're set. A Butler oscillator set up for vibration at the xtal fundamental frequency works well. See EMRFD Figure 7.32 for a great example — A Butler carefully biased to stand reasonable current for low phase noise with a low impedance in the Colpitt's tank. Wes placed a 100 Ω resistor on the link between base terminals of the differential pair. The low impedance and the bias levelling provides current limiting and prevents the oscillator BJT from going into saturation.



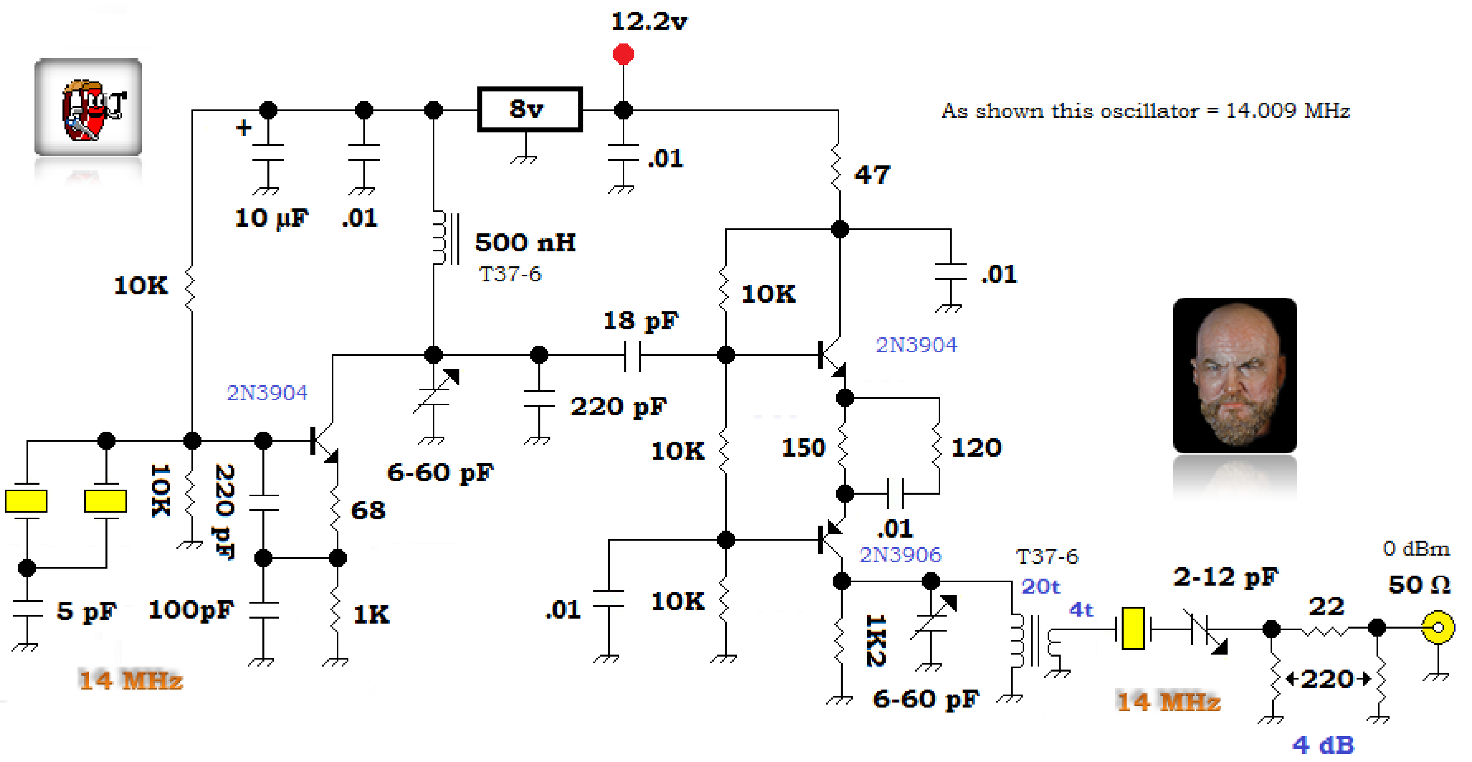
Above — A fundamental frequency Butler with hybrid-cascode (hycas) buffer I built for 14.0 MHz some years ago. Rather than tuning the oscillator tank with the usual low-Q trimmer capacitor, I designed for a fixed C of ~100 pF and thumb and finger tuned the tank to preserve the Q.

Alternately, a standard Colpitt's crystal oscillator with the crystal connected to ground via a shunt capacitor of >= 200 pF provides a great way to get a low phase noise, low distortion oscillator. Take the oscillator output from the junction of the xtal and shunt capacitor and ensure you lightly load this node. See EMRFD Figure 4.24 or my Low Noise Crystal Oscillator web page for examples.

A problem arose. I couldn't find 2 crystals 20 KHz apart, so running 2 fundamental Butler or Colpitts oscillators for IMD measurement seemed out of reach. A web search yielded 1 solution; pull the xtal oscillators to a desired spacing around 14 MHz and crystal filter the output to drop phase noise.

My next experiments mirrored the work of Christian, DL9NL on this web site. It appears he based his design on that of Stuart, KI6QP who authored A Precision Two-Tone RF Generator for IMD Measurements for QEX for April 1995. Stuart used crystals cut 20 KHz apart to make 2 Colpitt's oscillators running moderate current with output filtering by 1 xtal filter tuned to the oscillator frequency to reduce phase noise.

In his project, Christian, DL9NL wired up two 14.318 MHz microprocessor filters in a super VXO topology to cleverly move the xtals 5 or 20 KHz apart. He also output filtered his 2 oscillators with a xtal filter.



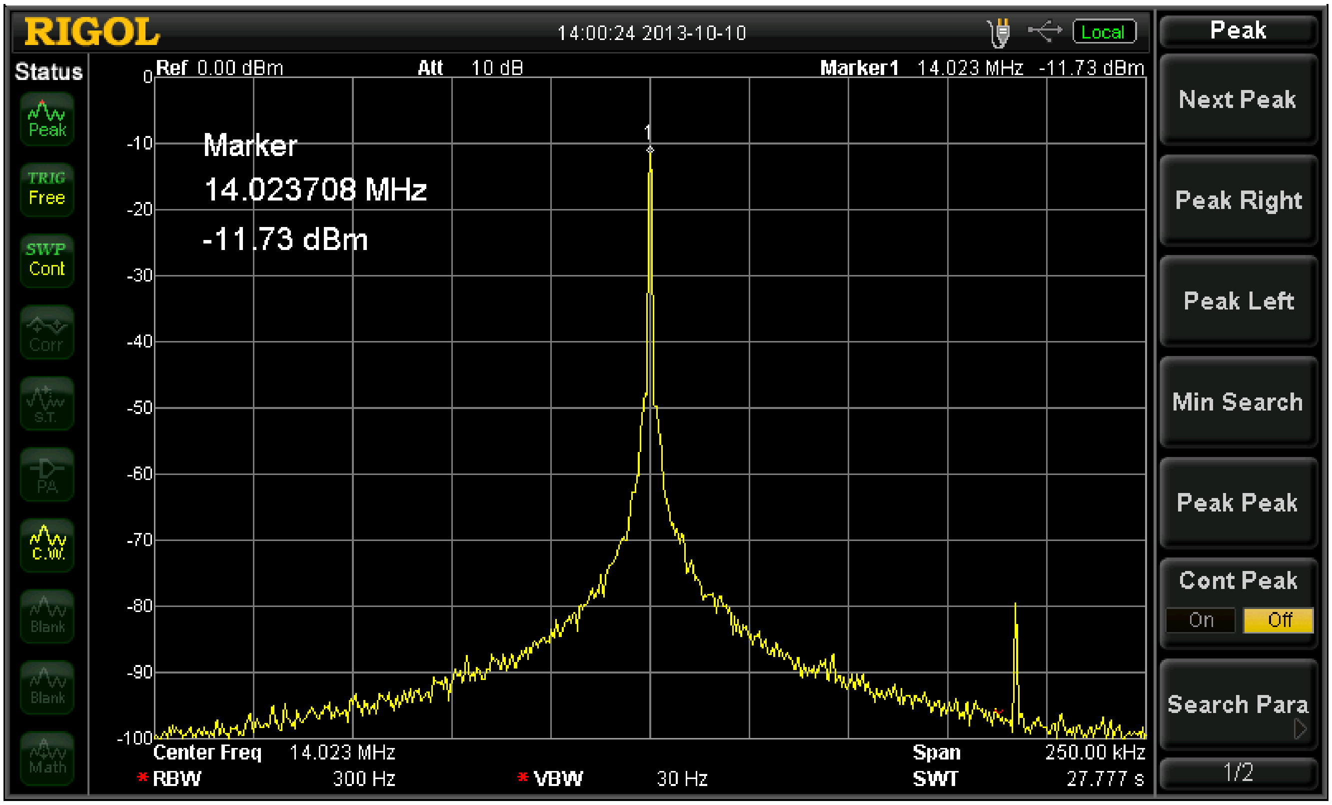
General Idea:

1. Use a super VXO with added L and/or C on 2 different oscillators to pull them 20 KHz apart
2. Apply an external xtal filter to lower phase noise

Above — 1 of my rudimentary experiments involving a super VXO plus output xtal filter. Not a great design. In this experiment, the 2 crystals were connected to ground via a 5 pF capacitor to yield 14.009 MHz. I learned that xtal frequency and xtal parameters determined the available frequency spread and output filter function.

After thought and the above bench work, and also because I need to make oscillators for IMD measurements at 50-100 MHz, I decided to go with an LC oscillator. They work fine for IMD measures with proper design and construction. For example, the signal generator shown in EMRFD Fig 7.27 sports a phase noise approaching -140 dBc/Hz @ a 10 kHz spacing; getting awfully close to the cherished vintage HP-8640B signal generator.

With LC oscillators, I learned that a metal enclosure with a feedthrough capacitor for the DC line proved important.Strong reverse isolation also ranks 'very important'. A case in point follows:



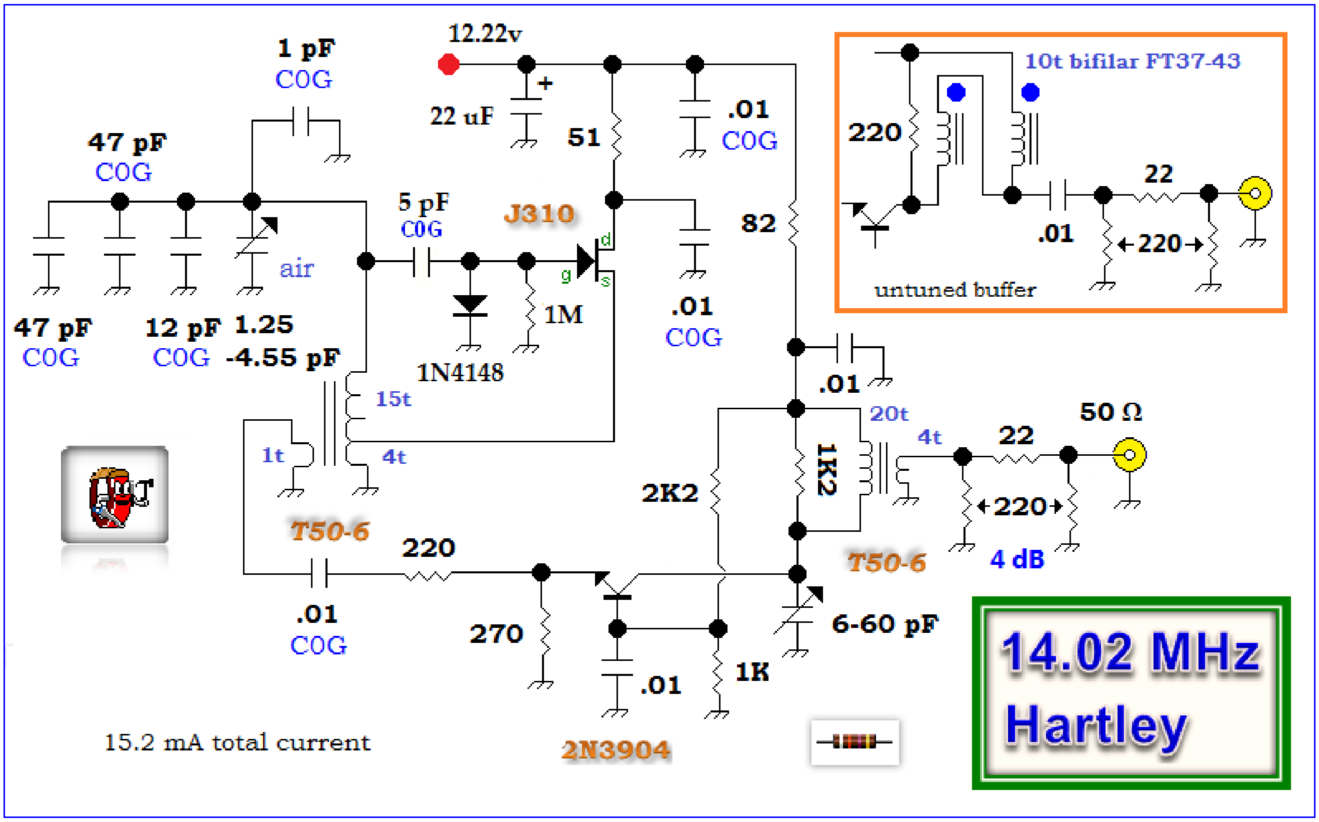
Ref 0.00 dBm
Att 10 dB
Marker1 14.023 MHz -11.73 dBm

Marker
14.023708 MHz
-11.73 dBm

Center Freq 14.023 MHz
Span 250.00 kHz
RBW 300 Hz
VBW 30 Hz
SWT 27.777 s

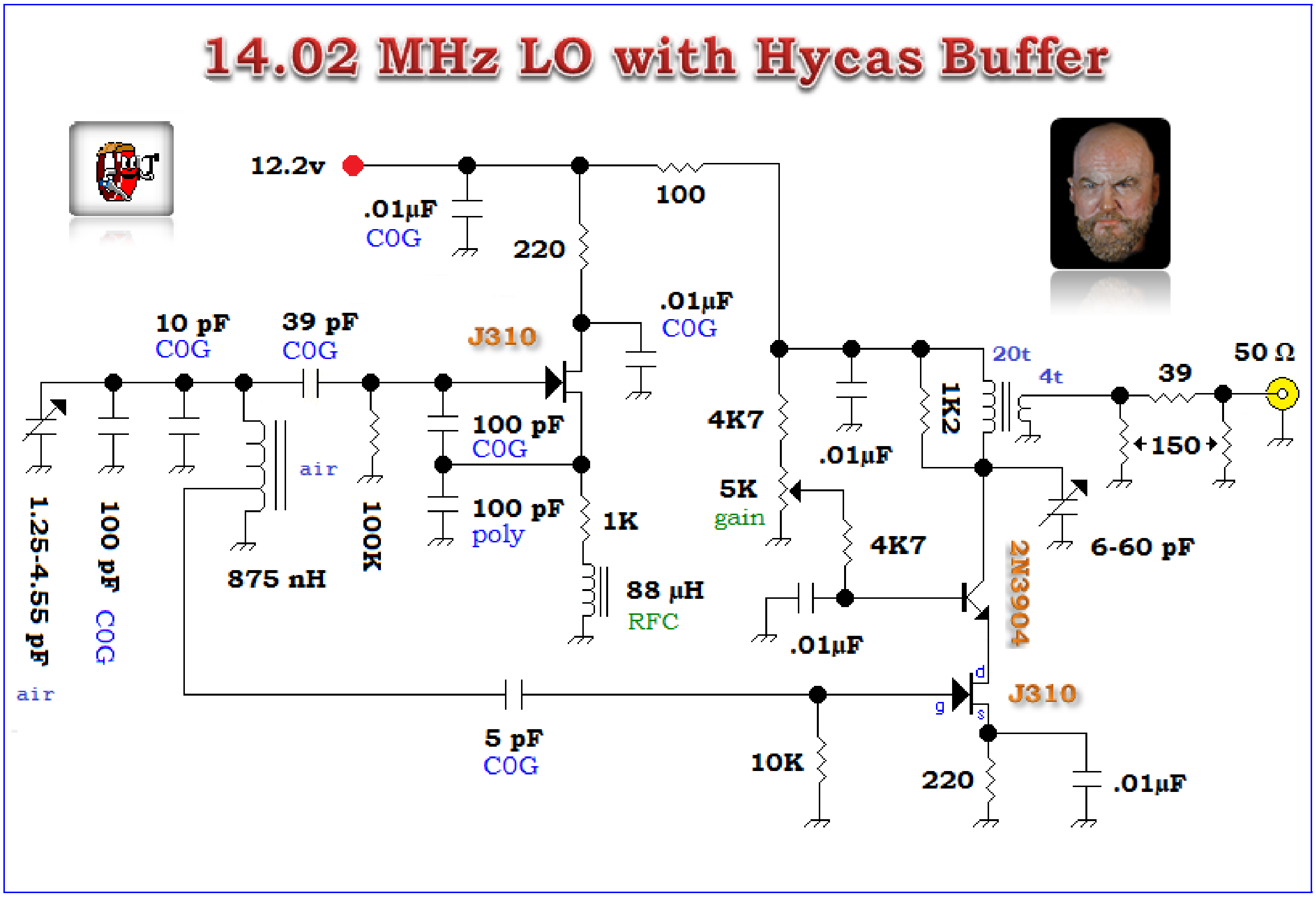
Above — A close-in spectrum analysis of 1 of my 14.02 MHz LC oscillators. Span = 250 KHz; RBW = 300 Hz and my video bandwidth = 30 Hz to "cut the grass". You can see a spur about 92 KHz right of center; a signal from my SA or environment is getting back into my VCO and low-level modulating it. Yikes! Another RF-tight LC oscillator with a common-base isolation amp showed no such spur.

Close in spectrum analysis may allow you to measure phase noise with your spectrum analyzer, however the wide-range, first oscillator of a Rigol spectrum analyzer operates in the range of 2- 4 GHz and is varactor tuned; as such, the best phase noise we can measure with it = -88 dBc/Hz @ 10 KHz spacing.



Above — 1 of my experiments: a Hartley with common-base buffer amp right out of EMRFD Figure 7.27 sans the final amp. I made 2 versions: 1 with a tuned buffer amp and another with a wideband output (see orange inset). The output power of the tuned amp and wideband amp = -6.7 dBm and -11.7 dBm respectively.

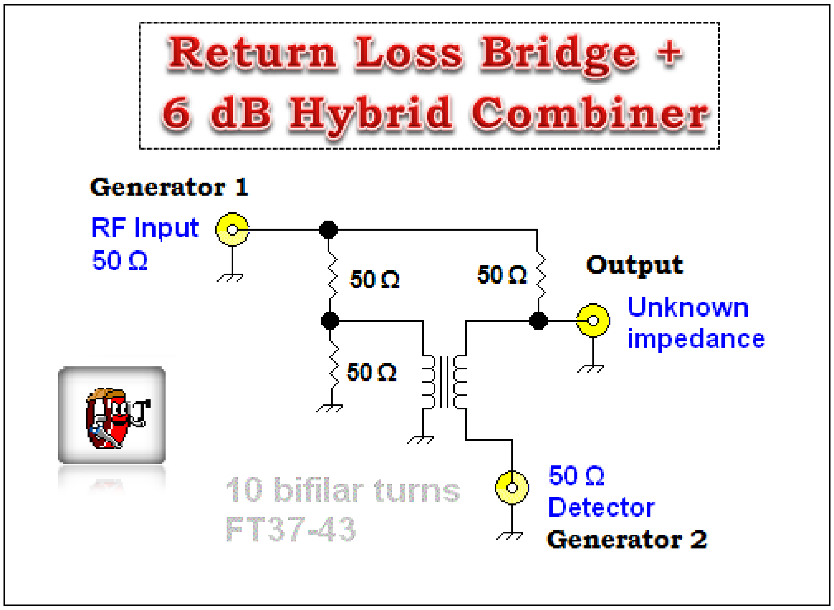
You may want to add a final amplifer — I've got a couple of amplifer modules if I need to get to 0 dBm or above. Prior to tweaking the lid-on frequency to 14.02 MHz, I measured the 2nd harmonic at 55 -dBC in the tuned output version— yay! Building with solid temperature drift techniques, the frequency drift in my counter went just a few hertz *up* over time. The Figure 7.27 oscillator remains a go-to circuit for us builders.



Above — A Colpitts with hycas buffer allowing variable power output. During measurement of IMD, the 2 oscillators need to exert the same amplitude, so only 1 needs to have variable output power. A wideband bench generator plus a 1 frequency oscillator like this might be all you need.

Above — A photo of the completed Colpitts LC oscillator. Running a phono jack for the DC input was a mistake. A feedthrough capacitor = the best choice. The epoxy glued air wound coil and air trimmer cap plus the regular temperature stability techniques and a 100 pF polystyrene capacitor yielded a temperature drift under 40 Hertz per hour. 5 bolts anchor the copper clad board.

I connected the 5 pF coupling cap to the lowest turn on the air coil. Mounting an air coil (solenoid) in smokestack fashion over a copper board probably upsets its Qu, but it worked okay.



Above — Our old friend the Return Loss Bridge is also known as a 6 dB hybrid combiner and 1 serves double-duty on your QRP workbench as an isolator and a return loss bridge.

Phase Noise and Commercial Signal Generators

The worst of today's ham transceivers transmit a CW carrier with noise that is down by 117 dBc/Hz @ 10 kHz spacing. This is bad enough that if you had a Ham operating this rig 0.8 km away you would hear this noise, even with a perfect receiver. In a typical oscillator, the phase noise @ 100 kHz spacing lies ~20 dB lower than @ 10 kHz out to a noise floor that is around -140 dBm/Hz or better. The Elecraft K3 impresses us with phase noise at -138 dBc/Hz @ 10 KHz [Sherwood].

We either build, borrow or buy our signal generators. Second hand commercial products range from good old HP/Agilent gear or LC boxes once used by TV/FM receiver repairmen. The TV service equipment covers ~300 kHz to 150 MHz and tends to be inaccurate with a limited output amplitude range, while the vintage HP stuff often is heavy, decaying and crowds your bench.

Most of hobbyists can't afford nor justify spending $8-18k on boxes that go down to -127 dBm amplitude and -146 dBc phase noise at 1 GHz @ 20 kHz offset [Agilent N5181B MXG X-Series RF Analog Signal Generator]. These generators feature components like a YIG oscillator phase locked to an ovenized crystal reference. Professional, high-end gear.

Another choice = the Arbitrary/Function Generator or ARB that sell for a fraction of the cost of the aforementioned boxes. A few friends own them, in particular, products made by Rigol and GW Instek. These DDS generators range in frequency and price: higher price generally = better phase noise.
For example, the popular Rigol DG1022A phase noise = -108 dBc/Hz @ 10 KHz offset and the DG4162 = ≤-115 dBc/Hz @ 10 KHz offset; not good. Often ARBs will feature a host of modulation functions, sweeps, frequency counters and the 2 output channel models may offer I/Q outputs and the ability to perform 2-tone IMD tests. Search around the T Equipment website to read some equipment datasheets

My question to 2 informed EEs — Is the phase noise of a -115 dBc/Hz ARB *good enough* to reliably measure IMD? Their answer: yes, if you have a high end, modern spectrum analyzer with a minimum RBW of 100 Hz, deft analyzer skill and understand the measurement. Often us hobbyists have to make due with whatever test equipment we can build or buy.

You might just build a low noise oscillator with a crystal at 14.0 MHz and provide the other tone with your bench signal generator or some other combination of 2 devices. This is test equipment junkie fodder — band-of-brother's stuff. Thanks for reading.

---

2. Almost Popcorn Superhet

I published this rogue web page with hope you might find it interesting. It houses notes and photos for an incomplete update to my 1998 Popcorn 7 MHz, CW, superhet receiver.

Click for the Almost Superhet web page.

Click for the Almost Superhet Supplement page

---

3. VXO — VCXO Notes

VXOs and VCXOs provide a useful main or reference oscillator for HF and when multiplied, in VHF-UHF designs. Ignorant of such circuits, I executed some crude experiments to discover how they work and find ways to improve my designs. Increasingly clinical on my bench, I'm losing patience with the radio mystique — although the limits of my lab equipment, brain power and bench skills humble and tumble me into deeper understanding why educated people might submit to electronics folklore and go in blind faith.

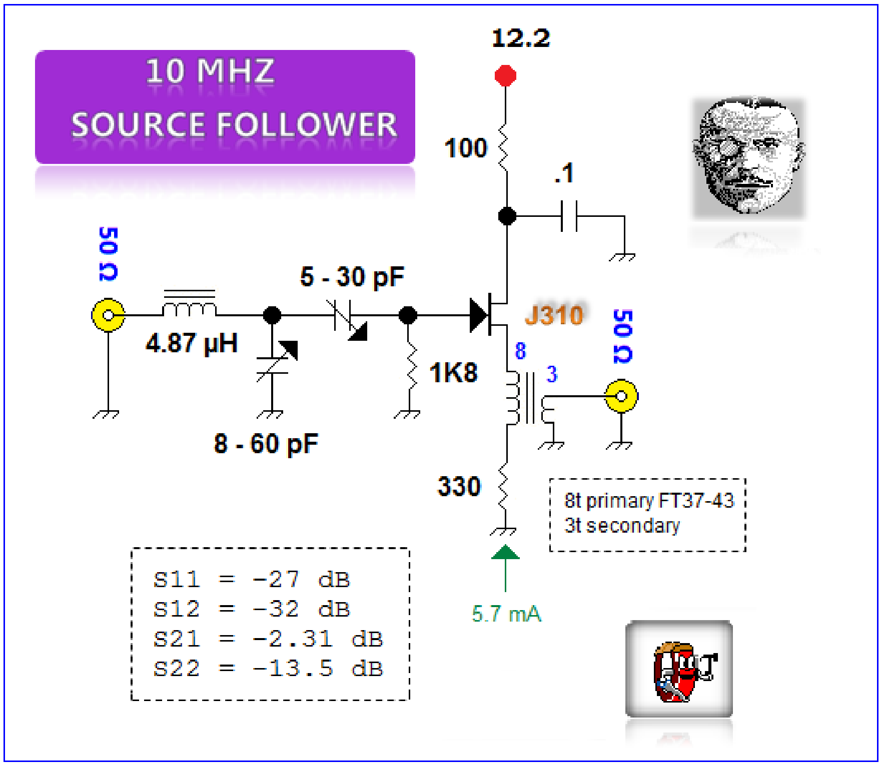
I'm not sure my bench work accomplished much, but if I can add 1 factoid to the RF enthusiast VXO knowledge base, then, I'll feel happy. I'm officially adopting **System D**; the slang term that originated in the French-speaking African continent and Caribbean Plate. In Down and Out in Paris and London, George Orwell called these System D practitioners "débrouillards" — low-level kitchen help, who achieve culinary success in the worst of circumstances. Television hero MacGyver practiced System D.

System D in the context of the QRP Homebuilder describes an effective, practical and resourceful builder who skillfully adapts to any situation and *gets it done*. System D builders don't just kludge together crap — these science-informed débrouillards innovate and shine. Sounds hopeful doesn't it?

Nearly all my bench adventures begin with a web search and the key word *VCXO* yielded many espléndido files + presentations by people and companies who passionately share their knowlege and experiences. Thanks to all!

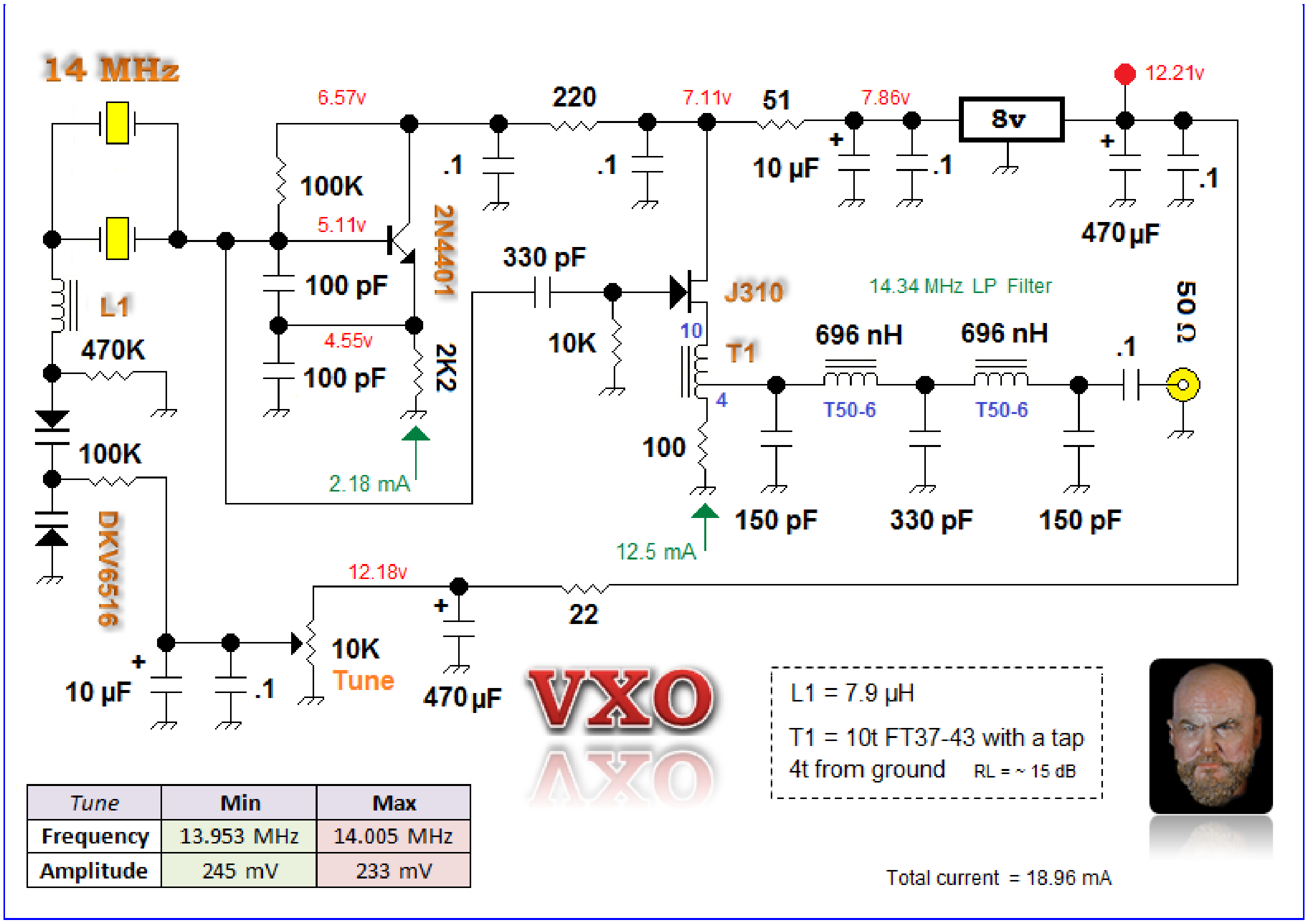
Experiments

It started with an email from Paol — what's the S12 of the source follower in his 14 MHz VXO buffer amplifier? I built 1 for 10 MHz and tested it since I didn't know.



Above — Device Under Test. Paol copied a source follower he found on a web site (complete with an über kitsch 1 mH choke on the source). I tweaked the current and replaced that big, old MF choke with a tapped FT37-43 giving an output return loss of ~ 13.5 dB. Although mediocre, it's probably better than the match provided by the series resistor we typically stick between the JFET source and its 50 Ω load. Further, for my S12 measurement, I matched the input with a L-C-C network to get a strong return loss @ 27 dB. Measured with a spectrum analyzer, S12 = -32 dB; paltry reverse isolation. The matching networks incurred some losses but raised measurement fidelity.

-32 dB — not great, but okay if followed by another amp. For a diode ring mixer, I might couple it to a common base amp, or eliminate the source follower altogether and replace it with a hycas buffer amp. Paol drove a NE612 transmit mixer with his VXO + a source follower and it worked okay in his bare-bones TX. He AC-coupled a 51 Ω resistor to ground on pin 6 of the NE612 that well terminated the LP filter. Intrigued, I then helped him with his VXO.

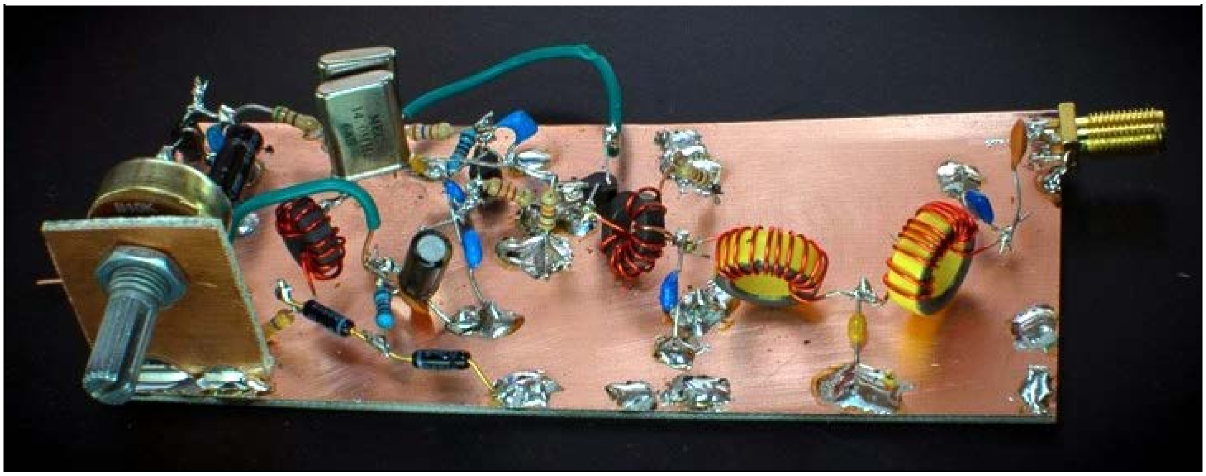


Tune	Min	Max
Frequency	13.953 MHz	14.005 MHz
Amplitude	245 mV	233 mV

Above — My first VXO experiment with varactors instead of an air variable cap that technically creates a voltage-controlled crystal oscillator (VCXO). The tabled values were measured with L1 wound on a FT50-61 ferrite toroid.

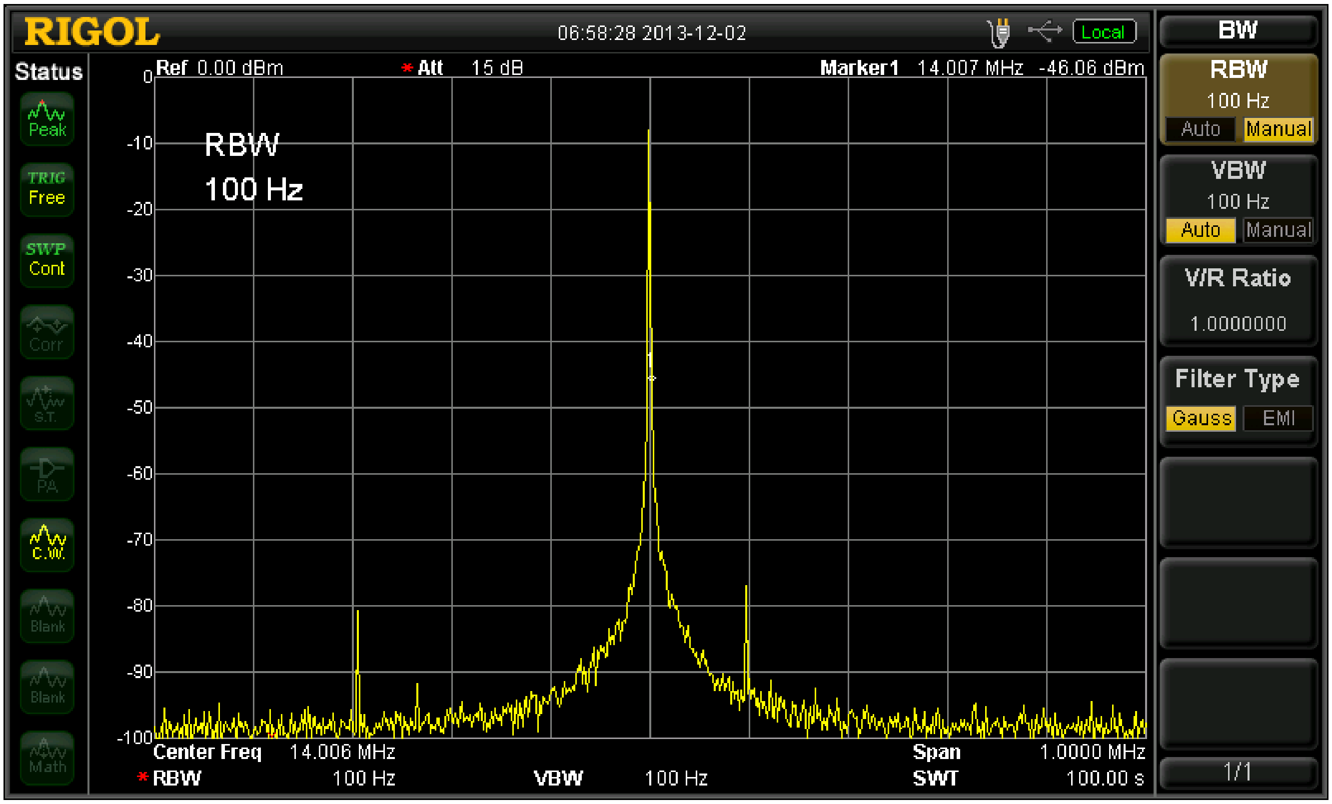
Paol had varactors but no air-variable capacitors. Both of us owned some Q = 200, DKV6516 silicon varactor diodes that with a reverse DC voltage of 2 to 10 volts give a nice tuning range of 18 to 140 pF for a single diode. We cheated and ran the reverse DC voltage from 0 to 12.2 volts, although later increased the 22 Ω R to limit the peak tuning voltage to 10v. Paol also put a fixed resistor on the cold end of his 10K tuning pot to keep a minimum 2 VDC on the varactors: I didn't bother, since my "this is only a test" breadboard will be dismantled and junked.

Above — DKV6516



Above — VCXO breadboard. Paol and I each made a VCXO and he with 14.060 MHz crystals. Lacking these, I just used 2 junk box 14 MHz crystals with a measured Qu of ~80K. The tapped FT37-43 gave an output return loss of ~ 15 dB thus providing a reasonable match into a simple 50 Ω low-pass filter. Click for a sweep showing the 2nd harmonic suppression.

My initial build didn't contain the two noise filtering 470 µF caps on the 12 volt supply: I originally soldered a solitary 22 µF cap cap close to the 8 volt voltage regulator input. My 12.2 volts comes from a regulated DC supply. More on filtering noise later.



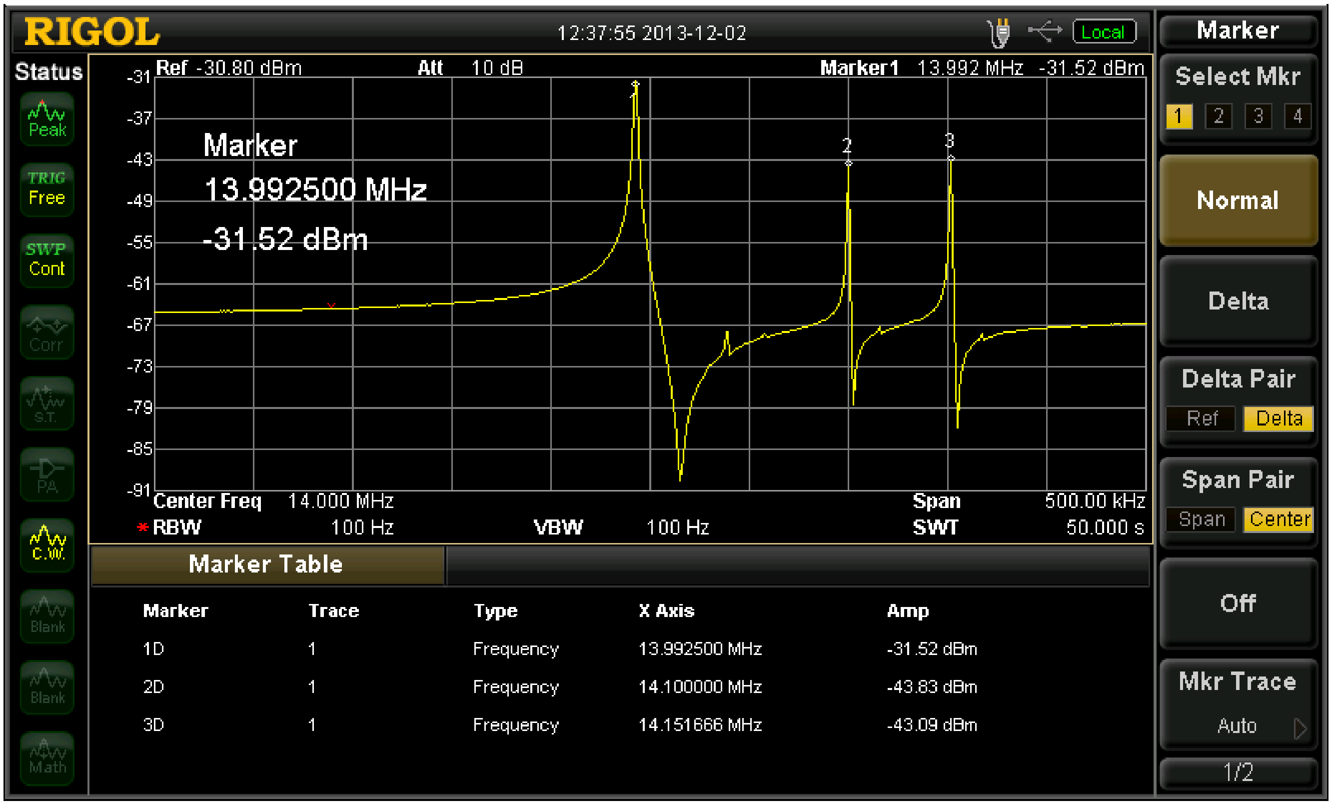
06:58:28 2013-12-02

Ref 0.00 dBm
*Att 15 dB
Marker1 14.007 MHz -46.06 dBm

Center Freq 14.006 MHz
Span 1.0000 MHz
*RBW 100 Hz
VBW 100 Hz
SWT 100.00 s

RBW 100 Hz Auto [✓ Manual]
VBW 100 Hz [✓ Auto] Manual
V/R Ratio 1.0000000
Filter Type [✓ Gauss] EMI

Above — My VCXO spectrogram showed 2 big spurs: 300 KHz down and 100 KHz up from center frequency (CF). I exchanged the varactors with an elegant, Q >= 1000 air variable, plus changed to a higher Q inductor, but could not get rid of these spikes. I assumed these spurs came from the crystals, or the BJT. Then, too, might they arise from my spectrum analyzer?



12:37:55 2013-12-02

Ref -30.80 dBm Att 10 dB Marker1 13.992 MHz -31.52 dBm

Marker
13.992500 MHz
-31.52 dBm

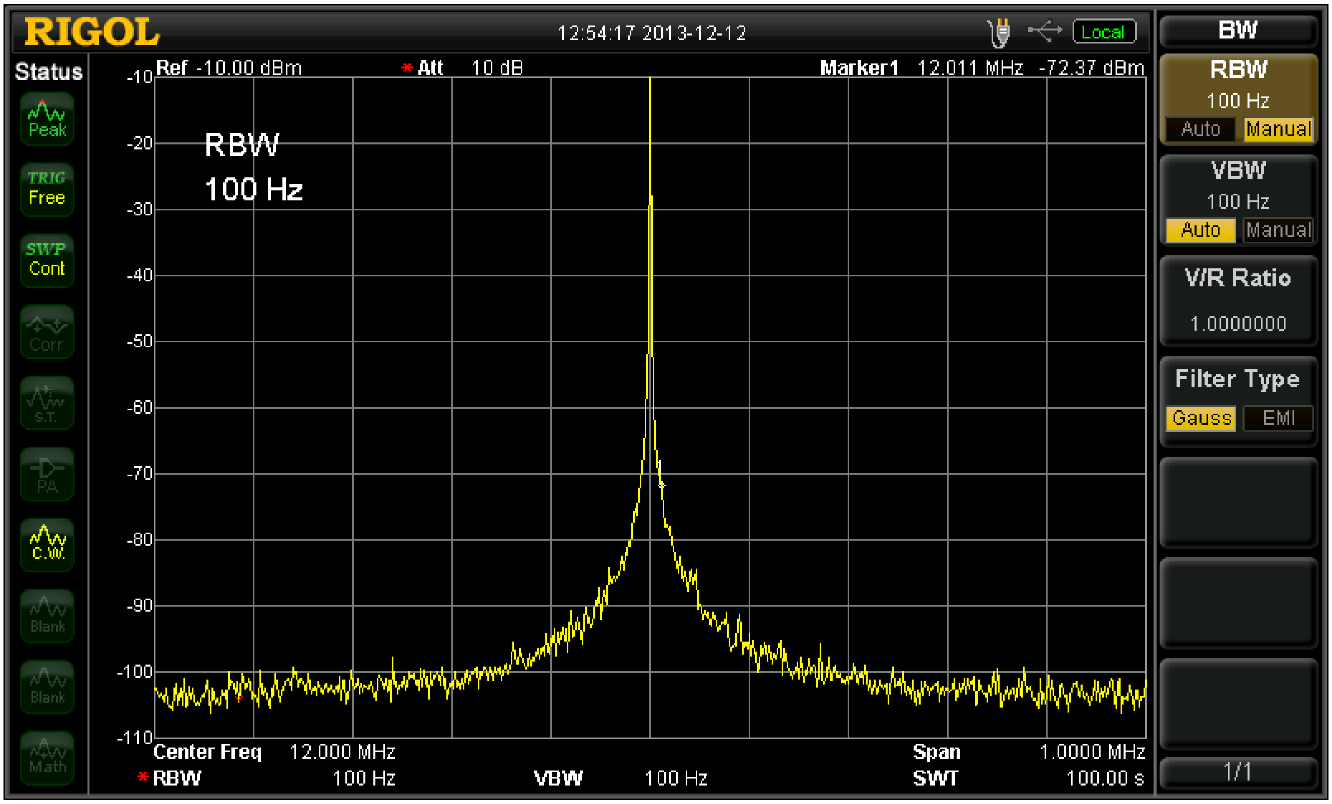
Center Freq 14.000 MHz Span 500.00 kHz
*RBW 100 Hz VBW 100 Hz SWT 50.000 s

Marker Table

Marker	Trace	Type	X Axis	Amp
1D	1	Frequency	13.992500 MHz	-31.52 dBm
2D	1	Frequency	14.100000 MHz	-43.83 dBm
3D	1	Frequency	14.151666 MHz	-43.09 dBm

Above — 14 MHz crystal sweep spectogram. To find out why these spurs occurred, I swept 1 crystal at a time from my batch and saw spurs on all of them. However, after some reading, I discovered that the mere presence of a crystal spurious response doesn't signal causality since this is almost a normal happening. Wes, W7ZOI explains this on this web page. Crystals usually have a spurious response and from Wes, the spur is more often than not about 80 or 100 kHz above the main signal. This happens with fairly good crystals, less so with crappy, cheap crystals and even less often with good quality crystals. Since many of us buy computer-grade crystals from Mouser-Key, or online auction sites, who really knows what we're getting? 1 day I hope to test a Q = 1 million crystal — that will be fun.

However, renograde crystals could still be the cause. Sometimes when pulling a crystal(s), we're living on the edge — the main resonance is pulled so far that the circuit balks at the resonant frequency, or lies near the edge of such behavior and spurs start to show up. *Back to System D*. Simple substitution with 2 other crystals from another batch could tell me if these particular crystals were behaving poorly. It turns out, they were.



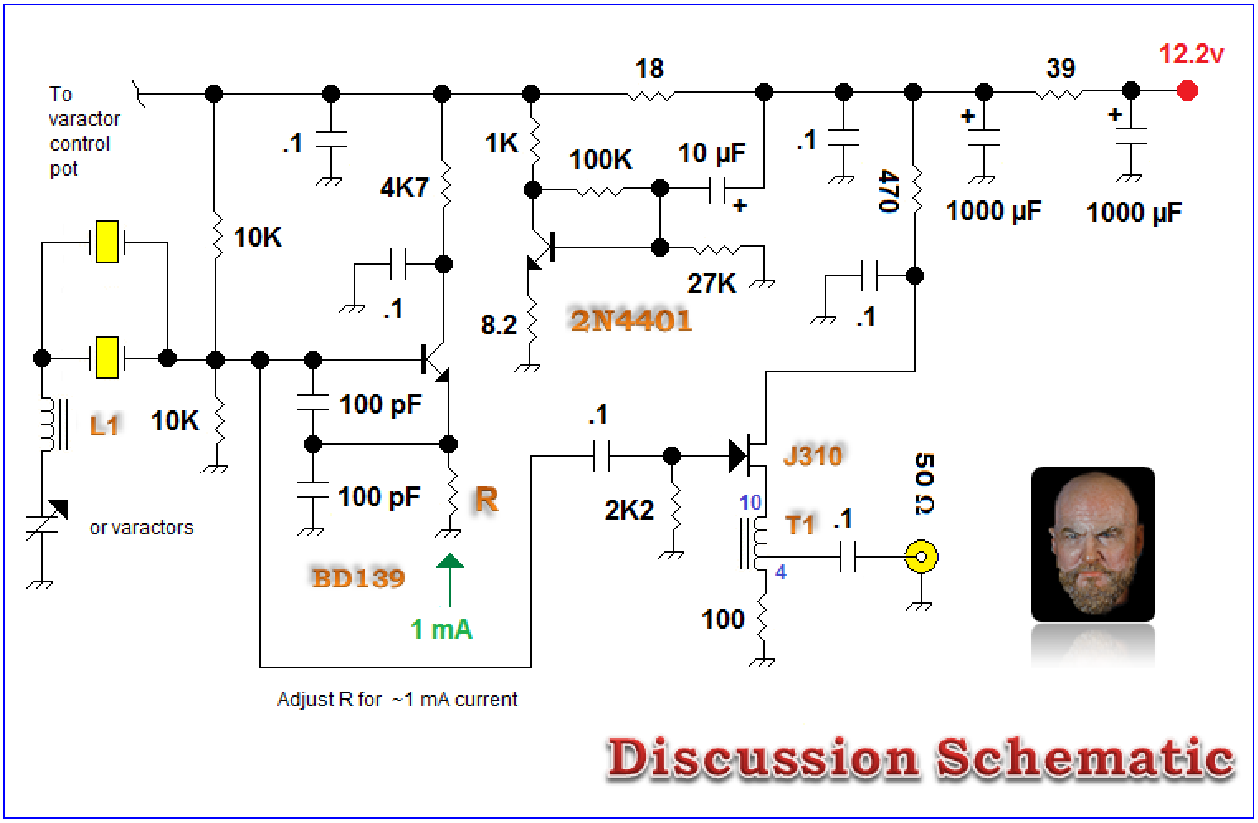
12:54:17 2013-12-12

Ref -10.00 dBm *Att 10 dB Marker1 12.011 MHz -72.37 dBm

RBW
100 Hz

Center Freq 12.000 MHz Span 1.0000 MHz
*RBW 100 Hz VBW 100 Hz SWT 100.00 s

Above — Spectogram of the VCXO with 12 MHz crystals. I wanted to try other 14 MHz crystals, but didn't own any. My nearest value crystals were 12 MHz jobs with a Qu of 120K. Eh voila! No spurs — yay! Let's talk about noise:



Above — Schematic for sideband noise discussion

System D design poses a real challenge with repect to noise. Measuring noise proves an advanced technique beyond the scope of this popcorn RF web site. We can't measure noise with our spectrum analyzer since the oscillator in the analyzer usually exhibits more noise than the VXO we want to test.

On page 4.12 of EMRFD version 1 is a small photo of 2 spectograms that show a clean crystal trace contrasted with a really bad oscillator trace. Baffled, because I could not replicate this experiment, I emailed Wes and asked him how he did it. Wes wrote we're able to see the noise in those traces because his SA featured a multiple pole, wideband filter with steep skirts that allows us to see the baseline noise close-in to the carrier. In contrast, most modern SAs feature synchronously tuned resonator type, or digital filters that produce a narrow peak with skirts that spread way out. The noise gets swept under the skirts.

By all means, advanced builders may apply notch filter techniques to measure oscillator sideband noise down low, however, a consideration for others might be to apply techniques proven to reduce noise and spurs — and just build the best oscillator possible plus measure what you can.

Some basic techniques to make a low-noise VCXO are well described elsewhere and may include: a high Q resonator, high Q varactors [with low series resistance], low power supply noise [including varactor tuning voltage], a low flicker noise amplifer, avoid overdriving the crystal and try to swing the highest AC voltage in the tank without reaching saturation and/or causing reverse breakdown in the varactor(s). Even the ratio of the Colpitts capacitors can affect phase noise, but without a method to measure phase noise, this gets esoteric. Some of these techniques also apply to VXOs.

Power Supply Noise

Bob, K3NHI and others have measured significant noise from the LM317 or related voltage regulators that we amateurs typically use. Power supply noise can FM our resonator and boost oscillator sideband noise. We may filter power supply noise with *large* value capacitors plus/minus series decoupling. See the work of Leif, SM5BSZ on this web page under the heading "The +12V supply is carefully decoupled like this". In addition, Charles Wenzel, published this web page.

I employed both techniques in my discussion schematic above. I went with the pi filter shown above 1000 µF - 39R - 1000 µF at the voltage input. A inductor might be used instead of the R if the voltage drop is too high for you. The Wenzel circuit fascinates me. The series resistor (18 Ω in my version) samples the current that flows into the oscillator. A voltage is then developed that is applied to the amplifier to generate a voltage that is out of phase with the noise current.

With good DC noise filtering I could measure small improvements in sideband noise in my SA despite only able to measure down to -88 dBc/Hertz at 10 KHz offset. In contrast, you rarely ever see even 1 electrolytic capacitor in the VXO schematics featured on many Ham radio builder sites.

Another trick is to avoid a regulator altogether. Since my DC supply already has a voltage regulator set to 12.2v, placing a second 5 to 8 volt regulator after it might just add another noise source.

Overdriving the Crystal

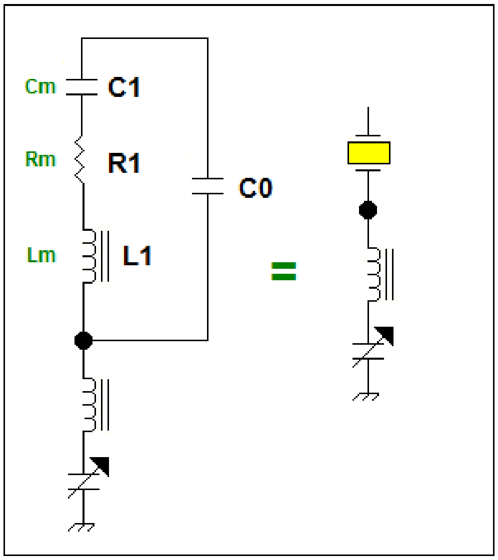
I heed the advice of Konstantinos, SV3ORA. He found that gently driving his crystal generated less noise and spurs compared to overdriving the crystal. Thus I adjusted my emitter resistor to about 1 mA current. For the schematic above, a typical emitter resistor starting value = 1K8. Just measure your emitter current.

I employed a large area, slower BJT, the BD139 to hopefully drop flicker noise. Expensive, super high Q crystals (Q = 500K to 1 million) may be driven with up to several mA of current, however, it's likely not so for the computer-grade xtals we typically use. I can't measure the outcome of these interventions, but System D thinking might just prevail.

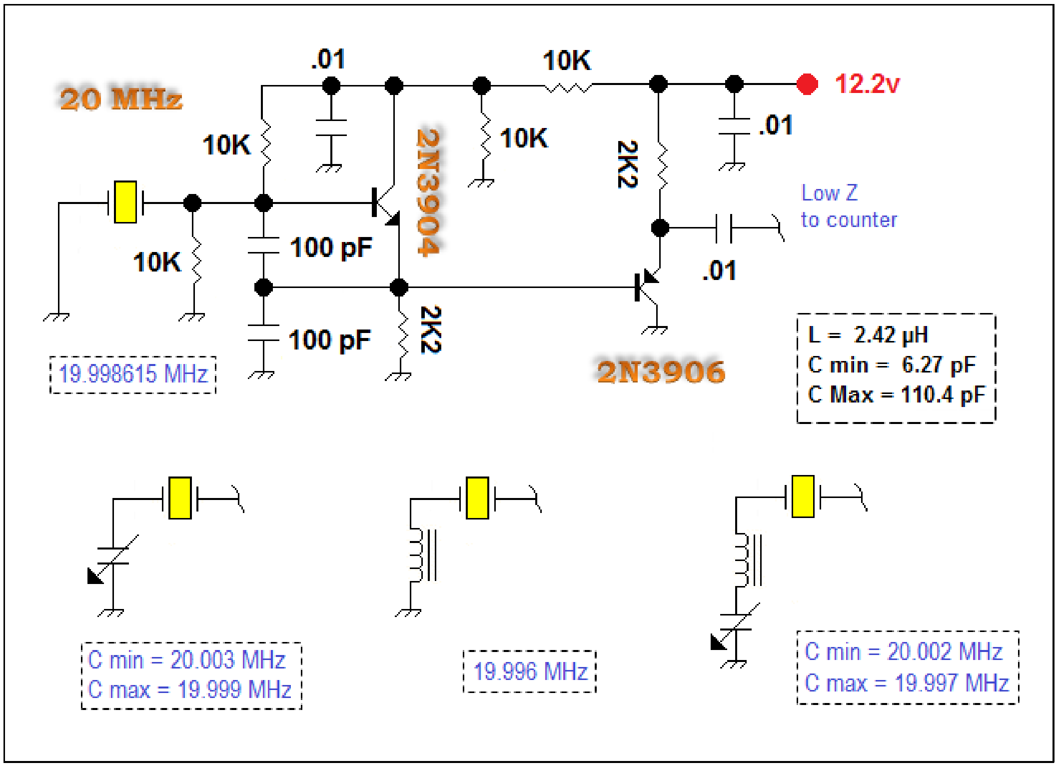
Varactors

Varactors tend to be low Q and their Q deteriorates as we go up in frequency. I've seen very low phase noise VCXOs employing varactors, however, these were engineering marvels. Still, as a VHFer and hopeful UHF enthusiast, this is the direction I wish to go. We ideally need to keep some reverse DC voltage on the varactors at all times and not go to 0 — this also limits tuning range in our simple 12v supply radios. Read your datasheet. Air variable capacitors (a VXO) might be a better solution for your application.

How Does a VXO work?



Above — The crystal equivalent electrical circuit has 2 arms: The impedance of the motional arm Zm lies in parallel with Zo, the impedance resulting from the parallel capacitance C0 (formed by the 2 crystal leads.) We measure and/or calculate these parameters when designing our crystal filters. In the above schematic I added the external L and C used to move the crystal’s series resonant frequency around in our VXO, VCXO, or Super VXO circuits.



Above — I made some simple experiments to explain why we place both a L and C in series with the crystal and how it changes the crystal frequency in our oscillator. From these experiments, some theory from EMRFD and wisdom from Ken Kuhn, I made the following simple conclusions:

Electrically a crystal looks like a large inductance in series with a small capacitance and in parallel with C0. An external inductor adds to the crystal’s series inductance and *lowers* the resonant frequency.

The external variable capacitor adds in series with the crystal internal capacitance to lower the net capacitance and *raise* the resonant frequency.

The purpose of adding an external inductor is so that we can tune both lower and higher than the natural crystal frequency. In contrast, if we only apply an inductor, the oscillator frequency would always be lower and if we just apply a capacitor, the oscillator frequency would always be higher.

Since the external inductance has much lower Q than the effective internal inductance, it adds to noise and signal losses. The same goes for the external C.

As we raise the inductance ever higher to boost the delta F, at some point, the parallel C of the crystal resonates with this "too high" L and the series resonant frequency has nothing to do with the series resonance of the crystal. This kills the normally sublime frequency stability of a crystal-based oscillator.

Q of the VXO/VCXO Inductor

I experimented to learn that the Q of that L doesn't need to be that high to allow the frequency to be moved up or down by an air variable cap or varactor.

For example, if I wound a coil on a T68-2 and shunted it with a resistor no less than 1K8 — it worked fine (sometimes the frequency shift would even improve 2-3 KHz for a given L). A #61 ferrite toroid wound inductor also worked well and requires much less turns allowing thicker wire for mechanical stability. Also, the lower number of turns helps avoid significant SRFs. Temperature stability remains a question with the this and all ferrites.

An FT37-43 or FT50-43 ferrite inductor worked poorly – I measured high signal losses plus the maximum frequency swing dropped when compared to the same inductance value wound on a T-68-2, or FT-50-61. Avoid #43 ferrites in any resonator circuit unless you're purposely building a noisy oscillator for learning purposes.

Some guys use those little molded chokes with a Q of ~ 50 for the L and they also work well. Predictably, as I lowered the Q of the external L, or went from an air variable C to a varactor, signal losses increased resulting in lower signal output amplitude.

Decreasing resonator Q increases phase noise. For a popcorn radio, I’m not sure the change in phase noise would be a deal breaker in some homebrew projects, but as possible, I think it's wise to employ high Q resonator components: crystal, inductor and the variable capacitance. Thus I'll wind my VXO/VCXO inductors on powdered iron toroids hereafter.

Since a quartz crystal resonator is driven by the piezoelectric effect, fundamental plus overtone frequency modes (3/5/ 7/11) may be derived. I unbypassed the collector and placed a collector choke on it to examine the output with my 'scope. The waveform rung with many harmonics. Lots to learn.

Go débrouillards!

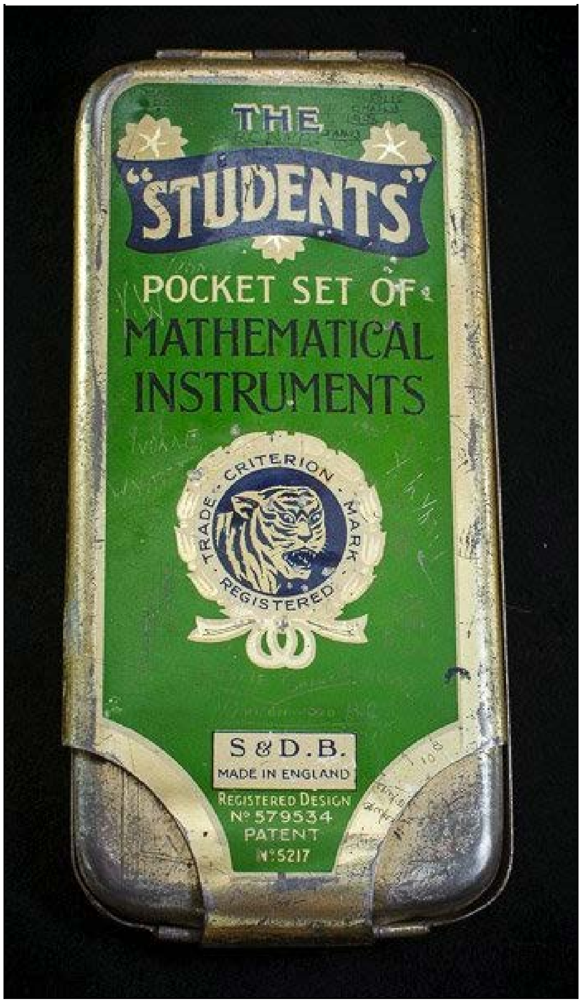
---

4. Funster Line — A Simple 40M Band QRP Trans-receiver

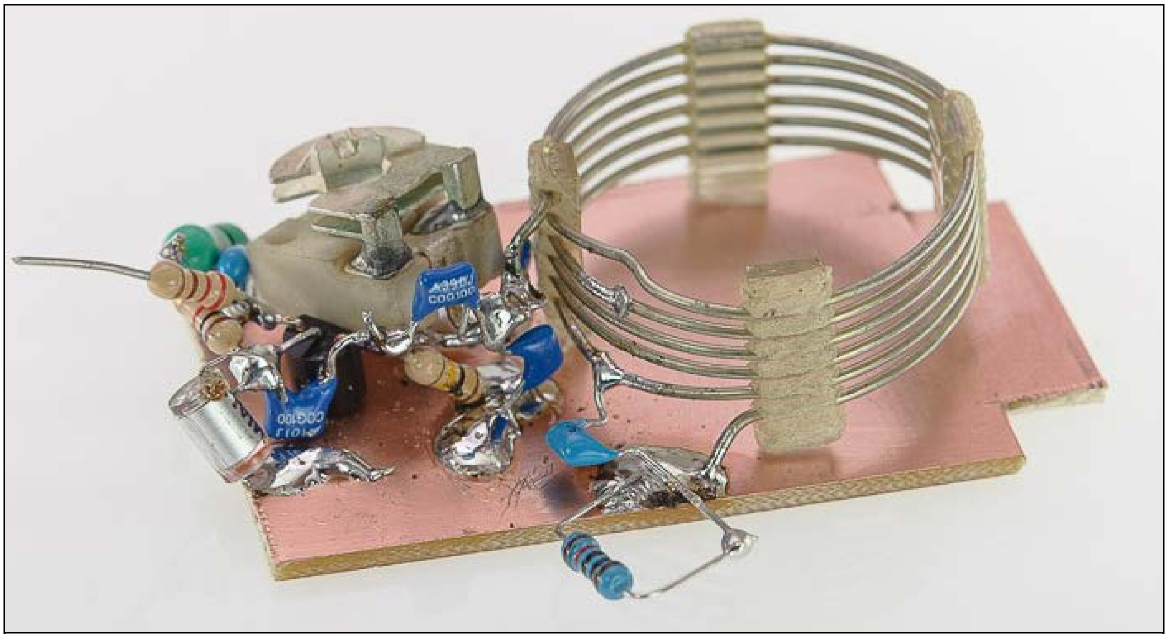
Click for the Funster transmittter notes.

---

5. Miscellaneous Photographs and Figures

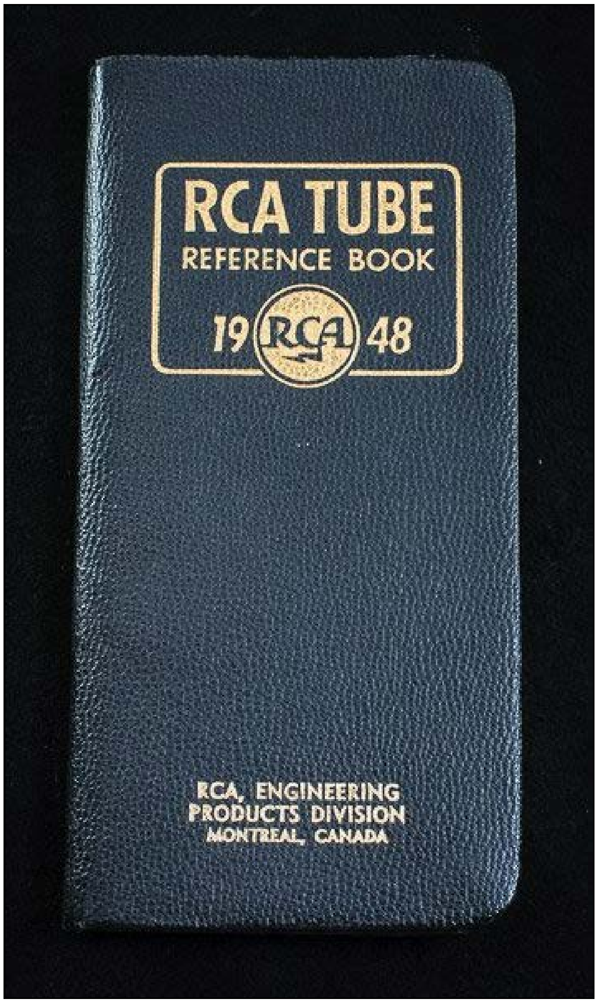


Above — My handed-down geometry set — purchased in 1938. Click for larger picture.



Above — My prototype Colpitts LC oscillator with only a 10K resistor as the load.

Above — Hi Q glass encased crystals. Click for another photo.



April 2014: New player on the scene — Bar Bar Kitty — chief science officer and QC consultant.

April 2014: Other new pet: Munchkin. Clear-spoken and direct — pet me, or feed me... and oh yah, Q2 is oscillating.

RF — Test and Measurement

Caitlyn 310 — UHF Beginnings

Respository web page for beginner UHF experiments in 2014-2015.

1. References
2. Transistors, RFCs, Capacitors and a 50 Ω Detector
3. Return Loss Bridge Experiments
4. VCO Experiments
5. Frequency Counter Kit

---

1. References

After a modest quest, I found an excellent microwave primer called Microwaves Made Simple. Principles and Applications by W. Stephen Cheung and Fred H. Levien. Published in 1985, the authors brilliantly teach the basics in plain language (no small task). My copy was printed in December 2013 by Artech House who print single copies of many classic texts on demand. Click for a photo of my book.

The ARRL published 2 UHF compendium books entitled: The ARRL UHF / Microwave Experimenters Manual [1990] and The ARRL UHF / Microwave Projects Manual [1994]. While out of print, used copies seem prevalent and might serve as good resources for builders who hunger for circuits and Ham-related UHF circuit building information. Click and click for photos.

Instrument companies provide continuing education and materials to educate & support their customers. With luck you'll find helpful free application notes and other tools. Two recent RF library additions include an Agilent CD containing all their current application notes and webcasts [my image is truncated and the CD includes applying SA's, 'scopes and signal generators], and this old Spectrum Analyzer Fundamentals booklet by Tek.

---

2. Transistors, RFCs, Capacitors and a 50 Ω Detector

I want to grasp circuits from 300 to 1296 MHz. This meant acquiring some UHF parts, making some bench tools and buying a 50 Ω detector to sweep and/or measure power in my UHF stages. Click for a snapshot of some of my parts cabinets.

BJTs

I'll employ the BFR93A, 2SC3356, and 2SC3583 as my mainstay NPN bipolars; at least to start with. Numerous transistors will work — price, availability, NF and shipping costs informed my choice. I'd also feel happy with MRF901, BFR92 and BFR96 NPNs and it's always fun reading datasheets. My PNP choice = BFT93 by NXP. Later on, I'll add in discrete GaASFET and other non-BJT species. Only 2 MMICs lie in the parts cabinet: MAR-3 and MGA-61563.

Radio Frequency Chokes (RFC)

Although, I'll also wind, or perhaps ply carefully dimensioned copper board traces to make RF chokes, I built up a stock of lower Q chip inductors in size 0805 and 0603. These include 8.2 nH, 33 nH, 68 nH, 220 nH, 330 nH, 470 nH and a few other values. For RFCs we need to consider L, tolerance, SRF, Q , maximum current and temperature stability — it's probably better to only buy clearly specified parts so you can study their datasheet to glean this information. I've swept nearly all my RFCs and recorded their SRF.

Choosing RFCs — SRF (series resonant frequency)

An RF choke exerts its greatest impedance at its SRF and in the case of a first-order RFC such as a single inductor, we should aim the SRF at or near the frequency we wish to attenuate the most. Consider a choke in the emitter lead of a BJT Colpitts oscillator with a frequency range from 375 to 400 MHz: The choke's ideal SRF should lie somewhere above 400 MHz. As possible, the SRF of a first-order RFC should exceed the operating frequency of the device it's choking.

In higher-order filter applications such as a C-L or C-L-C (pi filter) applications, engineers tend to target the choke's SRF to >= 10 times the operating frequency. Challenges such as limited inductor choices, needing to match stages, or coping with stray reactances will arise, however, as amateur UHFers, we just try our best to fit an RFC.

UHF Capacitors

I swept numerous size 0805 + some 0603 and 1206 capacitors in a microstrip line jig to determine their SRF. Math formulas enable us to calculate and plot attentuation, Q and ESR at self-resonance if we wish. Click for 1 example. Although we rarely apply capacitors at their SRF in practical circuits, considering Q, ESR and attenuation gets us thinking that capacitor SRF is a dynamic value that may vary widely with part size, capacitance and datasheet specified Q.

For instance, if you take a size 1206 capacitor, measure its SRF and then replace it with a same-value size 0805 capacitor, the SRF of these two caps might vary by as much as 200 MHz or more. Size does matter! Even the same value capactor by a different manufacturer may be a SRF game changer. Bob, K3NHI tells it correctly: to measure *is* to know.

Datasheet specified capacitor Q is usually measured at a low frequency like 1 MHz. Since Q decreases as frequency increases, a capacitor that specs a Q of 2500 at 1 MHz may only sport a Q of 87 at its SRF way up at 512 MHz. During sweeps, I measured a deeper attenuation notch and often a higher SRF for "very high Q" caps when compared to the common, low-medium Q, popcorn caps sold as mystery parts by online stores or auctions. I'm now hypervigilent about finding very high Q SMD capacitors for sale at a low price.

I wrote the SRF of all my SMD caps in my notebook and use this data to inform my capacitor choice for RF bypass and tuning. Click for a photo of 3 UHF parts drawers. I've pretty much gone to size 0805 caps to try and garner a higher SRF at VHF and UHF. Still, too, we might have to solder down 0603 parts up above 1 GHz. Yikes!

Did you notice that when you drop a 0805 part on the floor, you never ever find it again? Even a little cough can expunge a size 0603 part from your QRP workbench.

50 Ω Detector

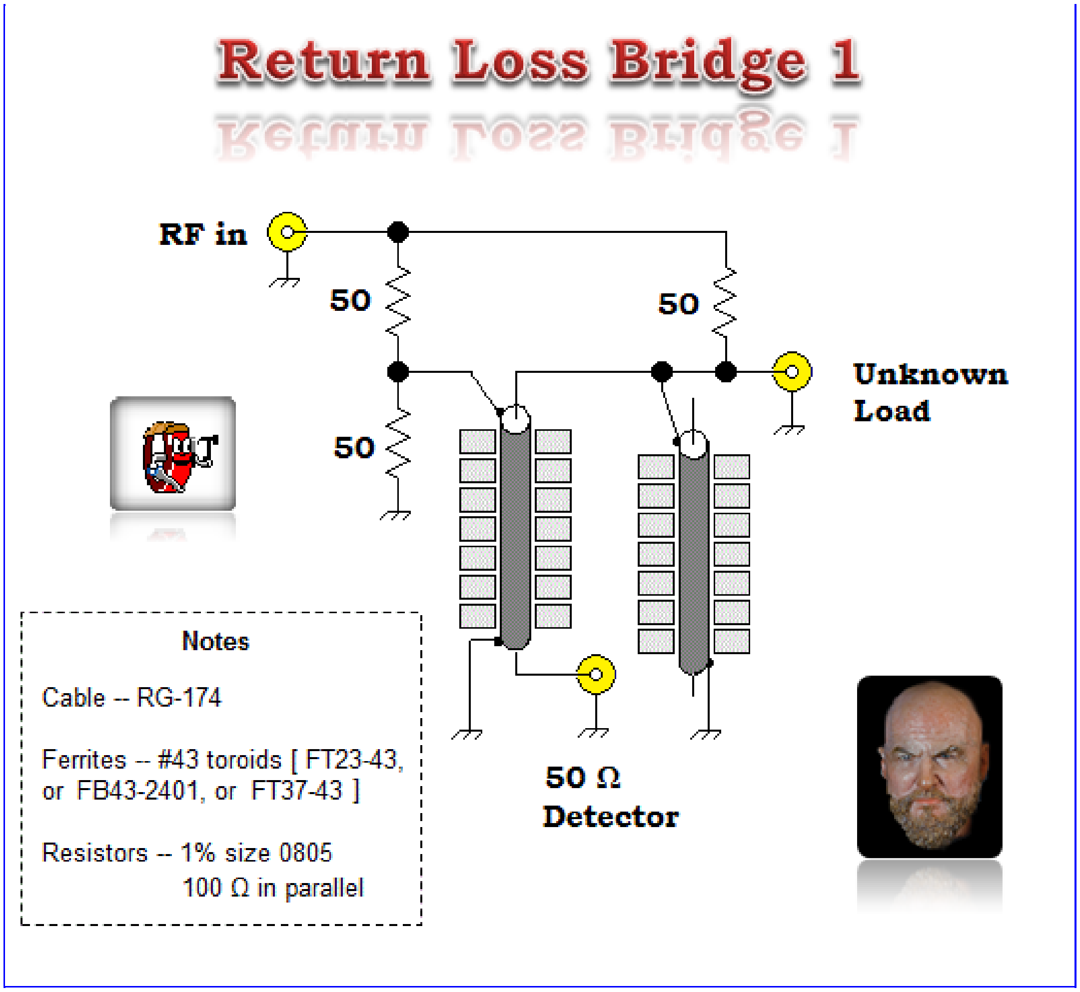
From 0.5 - 200 MHz, my 50 Ω terminated input DSO forms the heart of my test bench. An analog, or digital storage oscillocope with a bandwidth >= 1 GHz, plus a suitable wideband probe costs dearly and for many, won't be an option. UHF bandwidth, thermistor-based 50 Ω power meters are also expensive and lack signal viewing.

Thus, a spectrum analyzer seems the best choice for the amateur UHF enthusiast. A spectrum analyzer with a built-in tracking generator adds serious value — giving us a MF-UHF sweep system with profound utility. One popular, modern TG + SA choice is the Rigol DSA-815. I'm fortunate to have access to gear that measures up to 3 GHz.

3. Return Loss Bridge Experiments (RLB)

I'd be lost without a return loss bridge at UHF. Directivity marks the touchstone of a return loss bridge — and up at UHF grabbing a minimum 20 dB directivity may pose quite a challenge. Experiments to find a bridge with good directivity from 250 to 1750 MHz follow:

Above — A macro photo of RLB#1: 1 of the 3 "string of pearls" type return loss bridge built this Winter. I tried different numbers, sizes and mixes of ferrites. This particular bridge had 3 and then 2 FT37-61 toroids as shown, although I did not shorten the RG-174 coax when reducing to 2 toroids. The 61 mix gave poor directivity and was abandoned.



Notes

Cable -- RG-174

Ferrites -- #43 toroids [FT23-43, or FB43-2401, or FT37-43]

Resistors -- 1% size 0805
100 Ω in parallel

Above — Return loss bridge #1: the "string of pearls" variant. Several web sites feature great "string of pearls" RLB experiments for you to review and many place enough ferrite to extend from HF to UHF. I swept each of my builds in a Rigol tracking generator + spectrum analyzer with 4 frequency markers set. I made an open-circuit sweep [nothing on the unknown impedance port] followed by a sweep with a Mini-Circuits Labs DC - 18 GHz, 50 Ω load threaded on the unknown Z port. The difference between the open-circuit and 50 Ω measurement = the directivity at that frequency.

Ref 0.00 dBm Att 10 dB Marker4 1.7500 GHz -35.10 dB

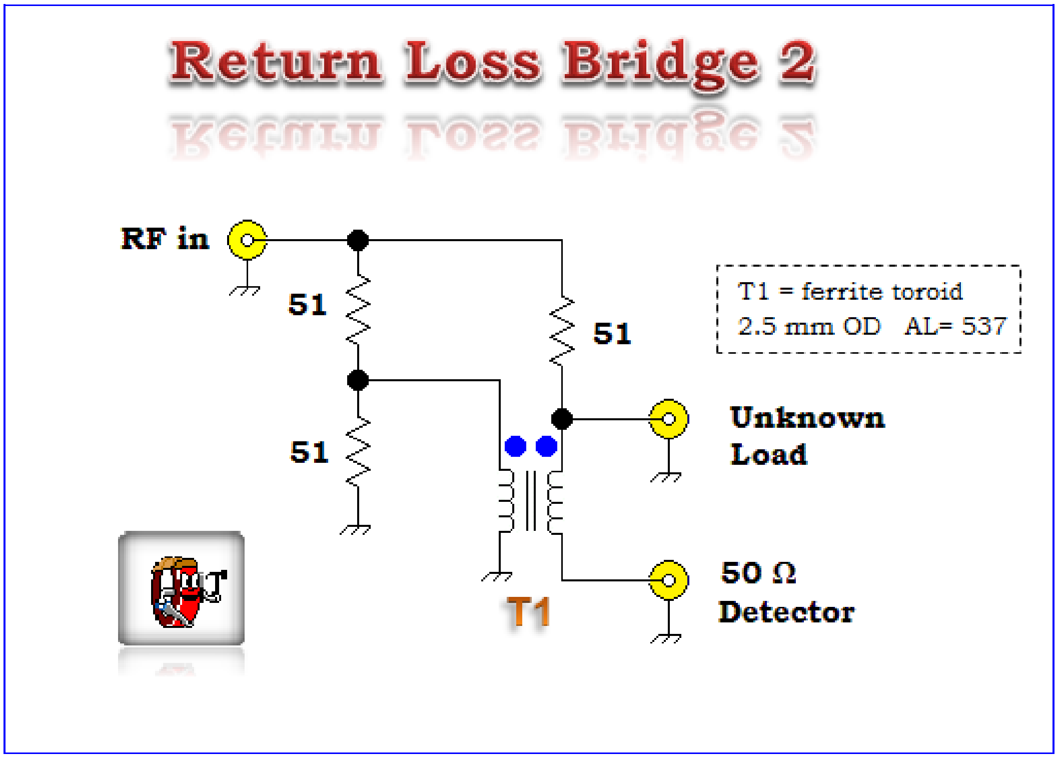
Marker
1.750000000 GHz
-35.10 dB

Center Freq 1.0000 GHz Span 1.5000 GHz
RBW 100.00 kHz VBW 100.00 kHz SWT 150.00 ms

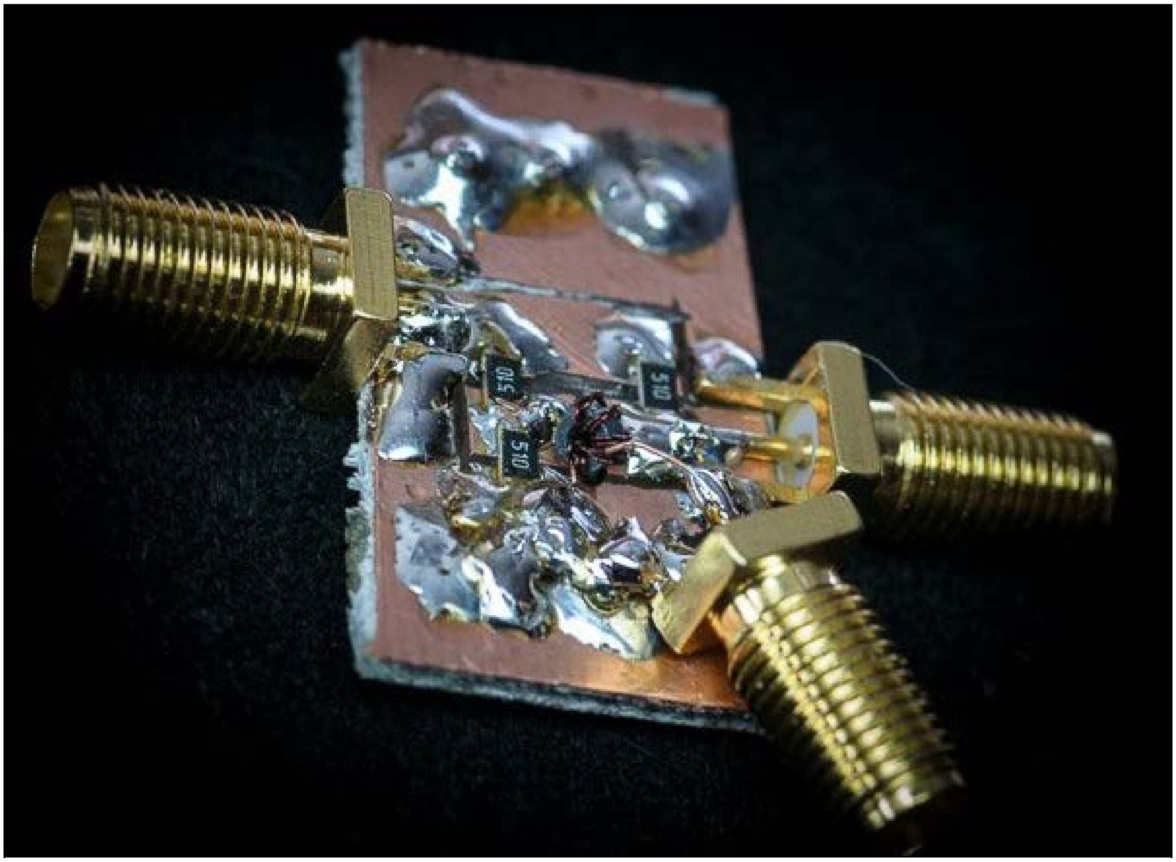
Marker Table

Marker	Trace	Type	X Axis	Amp	Directivity
1D	1	Frequency	250.000000 MHz	-42.26 dB	25.1 dB
2D	1	Frequency	752.500000 MHz	-38.28 dB	22.55 dB
3D	1	Frequency	1.207500000 GHz	-39.27 dB	23.68 dB
4D	1	Frequency	1.750000000 GHz	-35.10 dB	18.64 dB

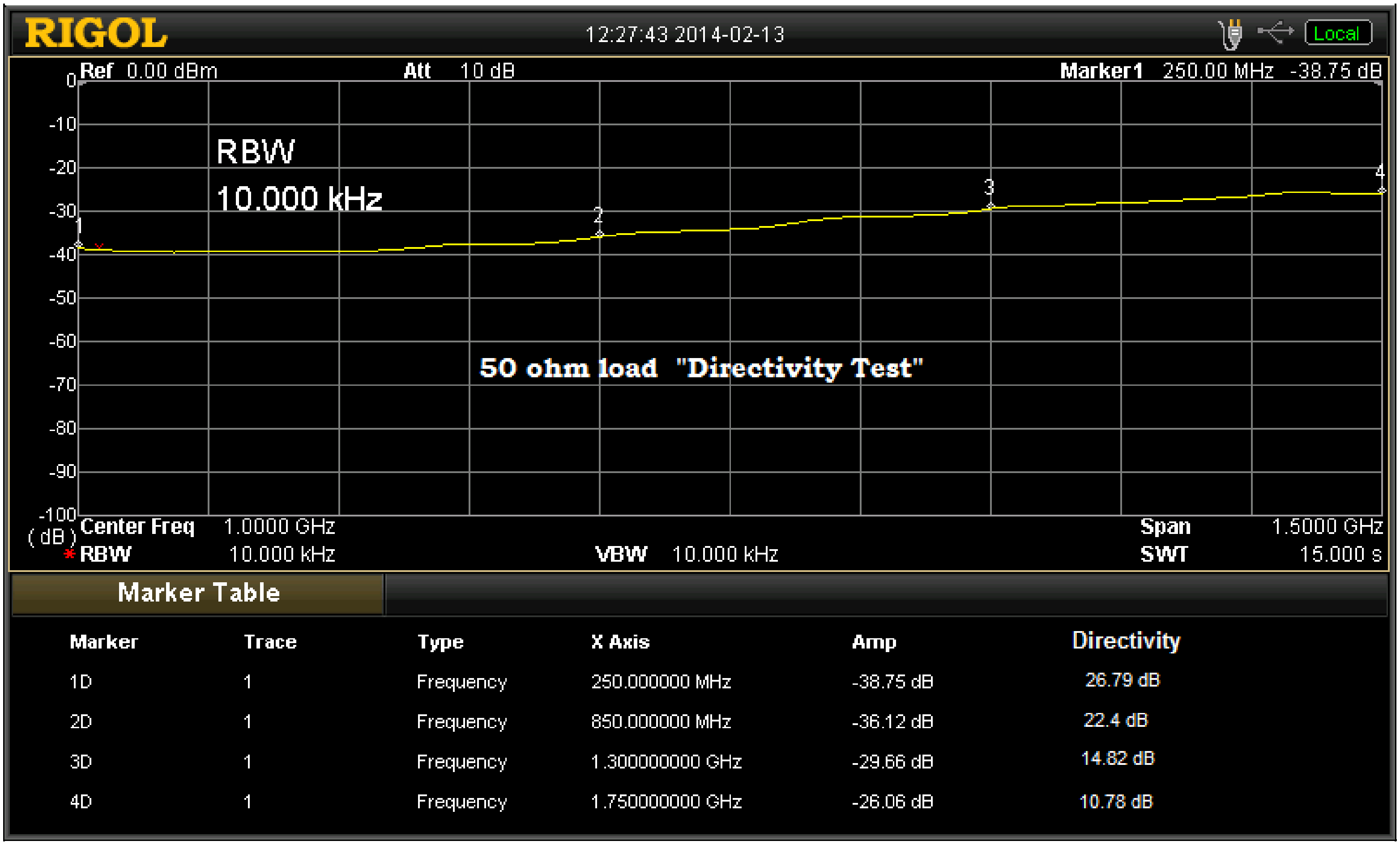
Above — The sweep of my best "string of pearls" bridge from 250 MHz to 1.75 GHz showing the directivity at 4 frequency points. Click for the open circuit sweep. I felt encouraged by this sweep with five FB43-2401 beads over each piece of coax cut just long enough to fit the beads. I built on 2-sided FR4 copper clad board with at least 6 via wires connecting the 2 copper surfaces.



Above — Schematic of RLB #2: the bridge topology featured in EMRFD. T1 = 5 bifilar windings of 32 gauge wire wound on a *tiny* ferrite toroid boasting a strong AL. I pulled this ferrite from a blown up mixer removed from an old 1.5 GHz spectrum analyzer. I started with 5 toroids, but during winding, got spastic and dropped 2 of them on the floor and concluded that the SMD part gnome who lurks beneath my bench swallowed them up, since I couldn't find them after an intensive search.



Above — Magnified return loss bridge #2 breadboard. Serious UHF enthusiast Paul Wade, W1GHz found that the smaller your breadboard, the better the UHF directivity in his bridge experiments: My findings support his. In RLB #2, I went with plain, size 1206, 5% resistors. I just ordered some size 0603 49.9 Ω 1% resistors for future experiments in my quest to make my ultimate UHF RLB.



Marker	Trace	Type	X Axis	Amp	Directivity
1D	1	Frequency	250.000000 MHz	-38.75 dB	26.79 dB
2D	1	Frequency	850.000000 MHz	-36.12 dB	22.4 dB
3D	1	Frequency	1.300000000 GHz	-29.66 dB	14.82 dB
4D	1	Frequency	1.750000000 GHz	-26.06 dB	10.78 dB

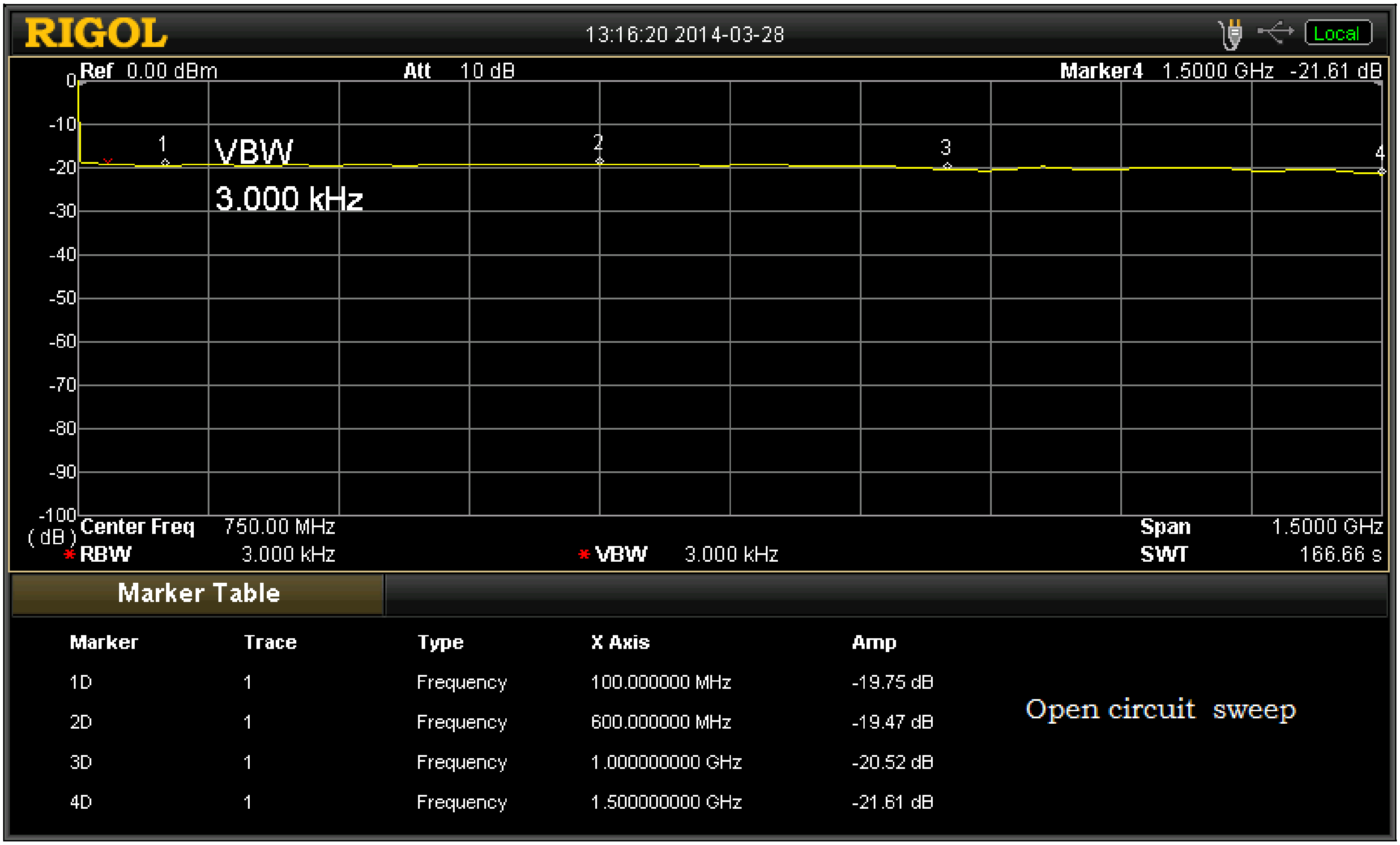
Above — The sweep of RLB #2 from 250 MHz to 1.75 GHz showing the directivity at 4 frequency points. Above ~900 MHz, directivity falls right off, so this bridge won't cut it for a serious UHF bench. I tried reducing T1 to 3 bifilar windings but this worsened the directivity. Click for a different sweep of RLB # 2 from 50 MHz to 1.5 GHz. Directivity = 27.5 dB at 50 MHz, so this bridge would work great for VHF. I imagine that SMD 49.9 Ω 1% resistors might boost directivity a little.

Of course, we're always impressed by the work of Kostas , SV3ORA.

Return Loss Bridge 3

Some experiments with the 3-bead RLB from Sam Wetterlin follow:

Above — My build of the Wetterlin 3-bead return loss bridge. This PC board was the only 1 left over from a 2006 joint project between the 4 Hams whose calls are etched on the PCB. Wes sent me this board and some of the parts and I'm grateful. My 3 dB pads = 294 Ω 1% resistors plus an 18 Ω 5% in size 0805. My 49.9 Ω 1% resistors = size 0603. The transformer = a Minicircuits Labs TC1-1-1-13M+. Click for the ferrite description. I used 24 gauge copper via wires to connect grounded nodes to the ground plane on the back of the PC board. I chose to permanently solder a 49.9 Ω resistor on the reference port.



13:16:20 2014-03-28

Ref 0.00 dBm
Att 10 dB
Marker4 1.5000 GHz -21.61 dB

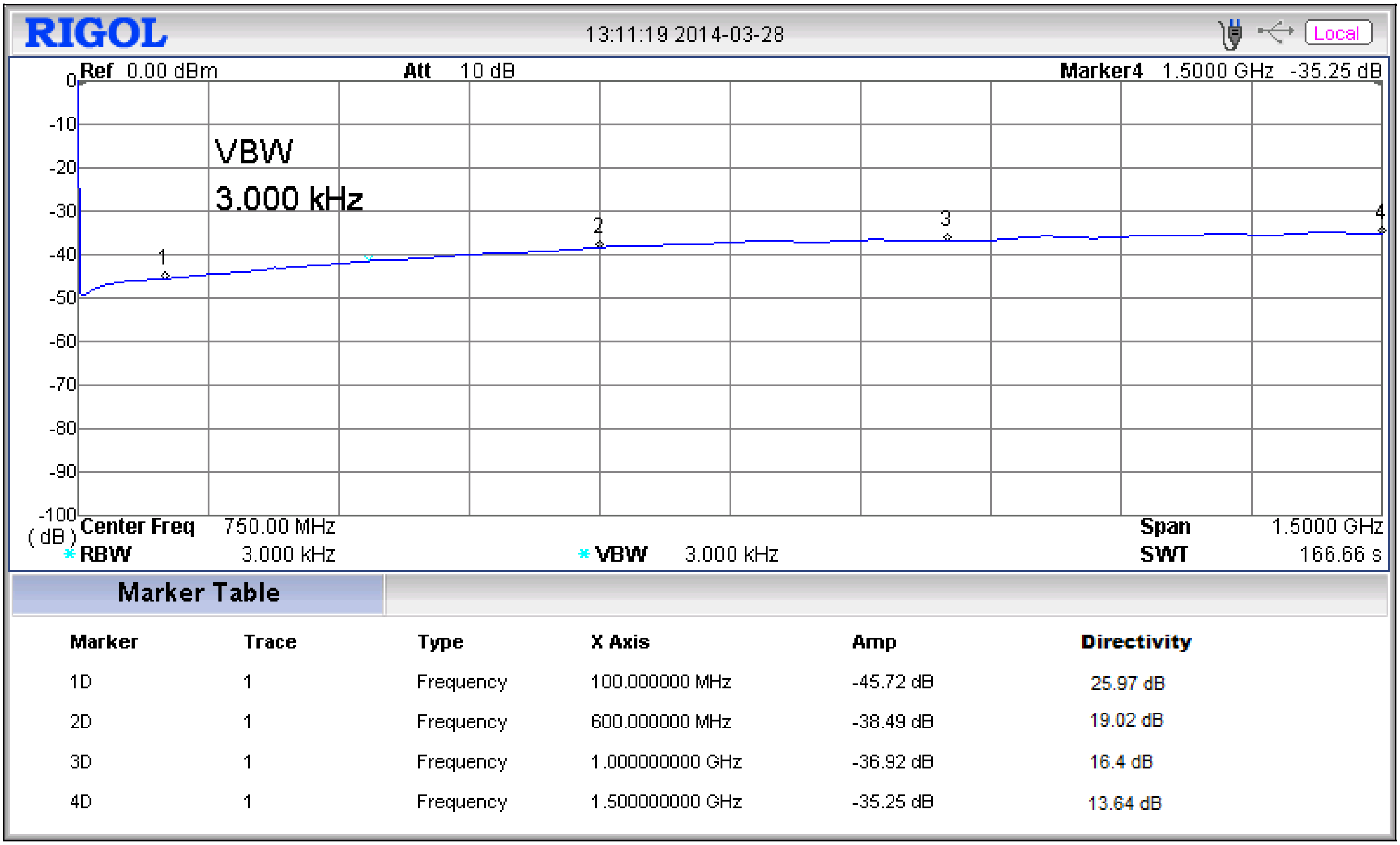
Center Freq 750.00 MHz
Span 1.5000 GHz
RBW 3.000 kHz
VBW 3.000 kHz
SWT 166.66 s

Marker Table

Marker	Trace	Type	X Axis	Amp
1D	1	Frequency	100.000000 MHz	-19.75 dB
2D	1	Frequency	600.000000 MHz	-19.47 dB
3D	1	Frequency	1.000000000 GHz	-20.52 dB
4D	1	Frequency	1.500000000 GHz	-21.61 dB

Open circuit sweep

Above — Open circuit sweep with 4 markers set. Span = 1.5 GHz.



13:11:19 2014-03-28

Ref 0.00 dBm Att 10 dB Marker4 1.5000 GHz -35.25 dB

VBW
3.000 kHz

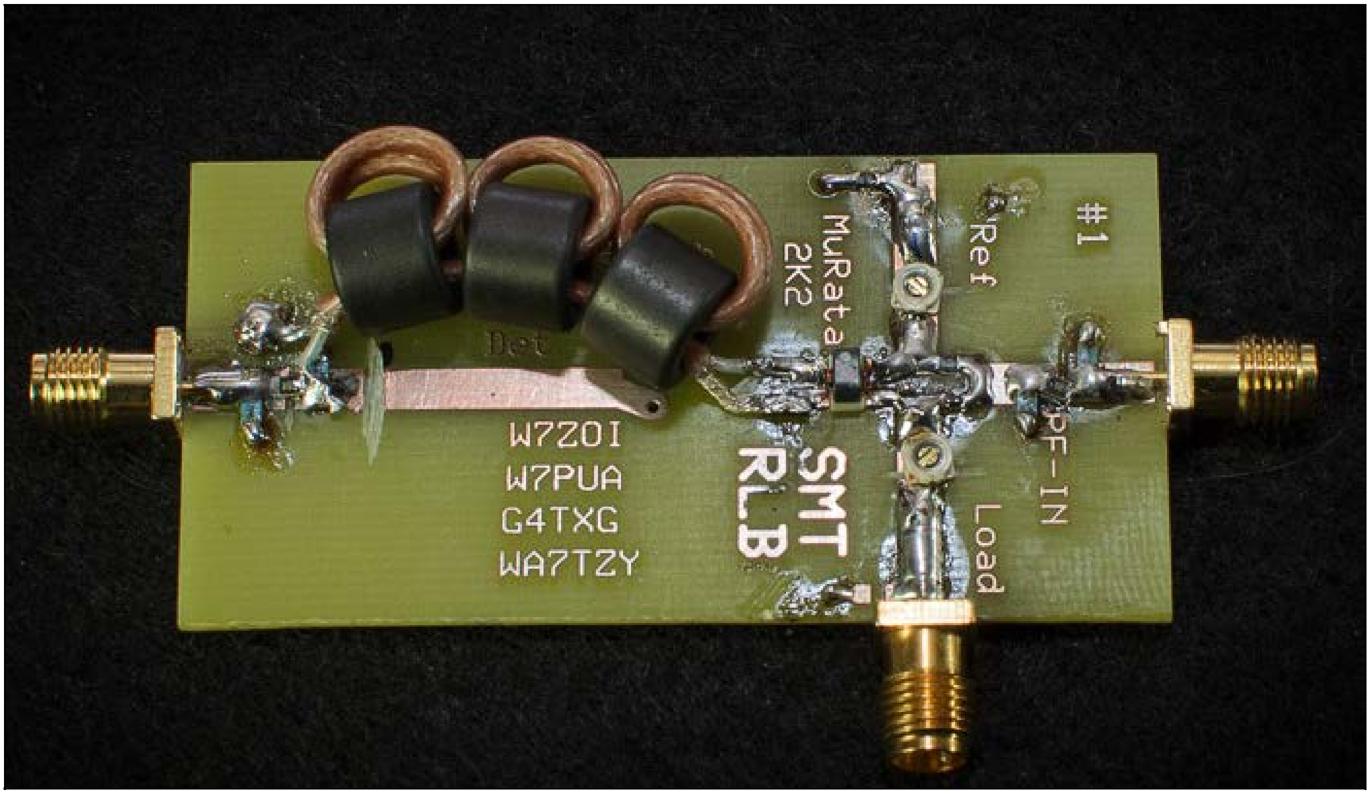
Center Freq 750.00 MHz Span 1.5000 GHz

RBW 3.000 kHz VBW 3.000 kHz SWT 166.66 s

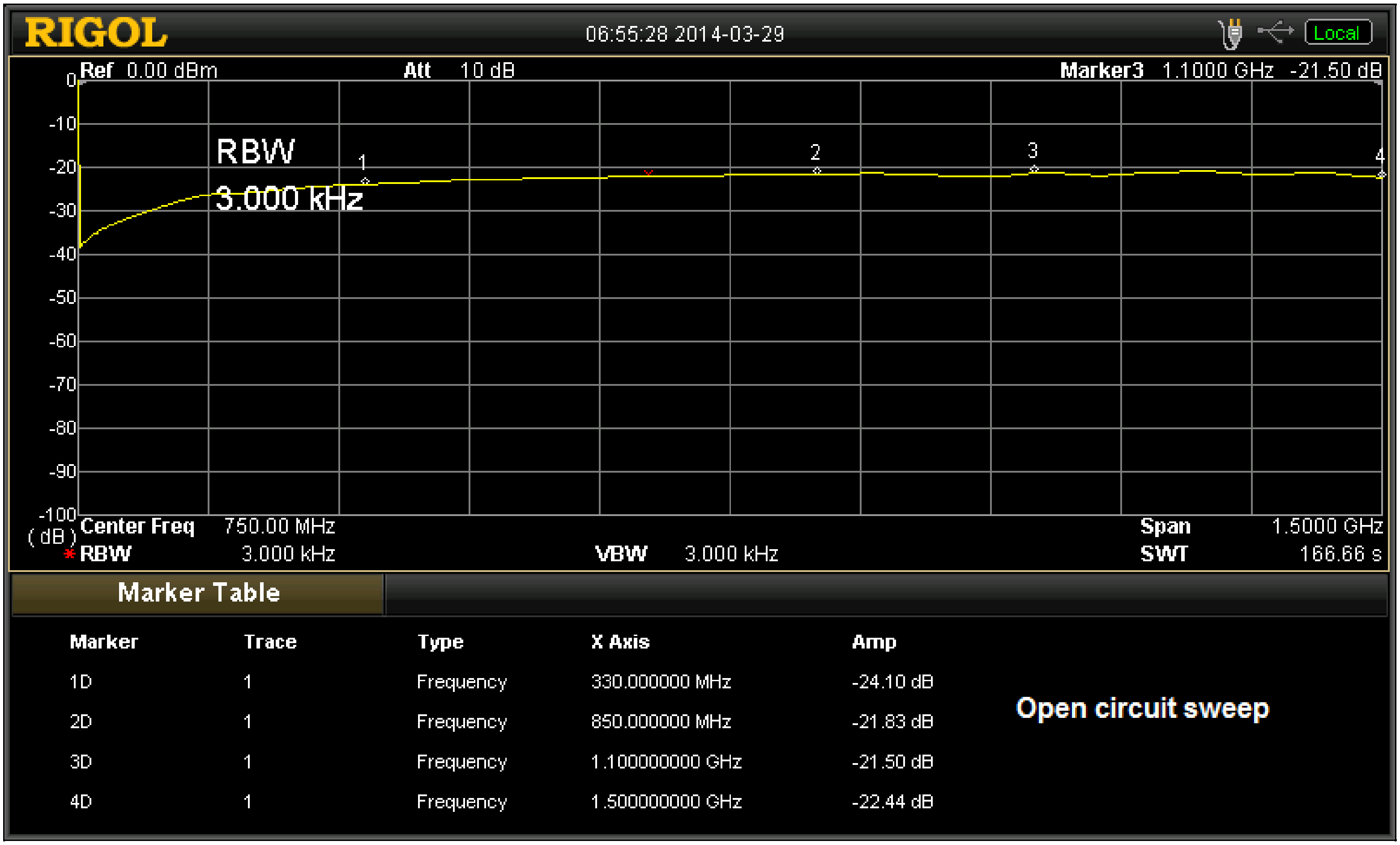
Marker Table

Marker	Trace	Type	X Axis	Amp	Directivity
1D	1	Frequency	100.000000 MHz	-45.72 dB	25.97 dB
2D	1	Frequency	600.000000 MHz	-38.49 dB	19.02 dB
3D	1	Frequency	1.000000000 GHz	-36.92 dB	16.4 dB
4D	1	Frequency	1.500000000 GHz	-35.25 dB	13.64 dB

Above — 50 ohm load termination sweep with calculated directivity. Click for the 50 Ω sweep where the span = 200 MHz. I was disappointed with the UHF directivity in *my* build.



Above — The 3-bead RLB with added series trimmer capacitors on the load and reference ports. SMT trimmer caps = Voltronics JR300 5.5-30 pF with a Q of 1500. I tried both shunt and series caps to try and boost UHF directivity.



06:55:28 2014-03-29

Ref 0.00 dBm **Att** 10 dB **Marker3** 1.1000 GHz -21.50 dB

Center Freq 750.00 MHz **Span** 1.5000 GHz

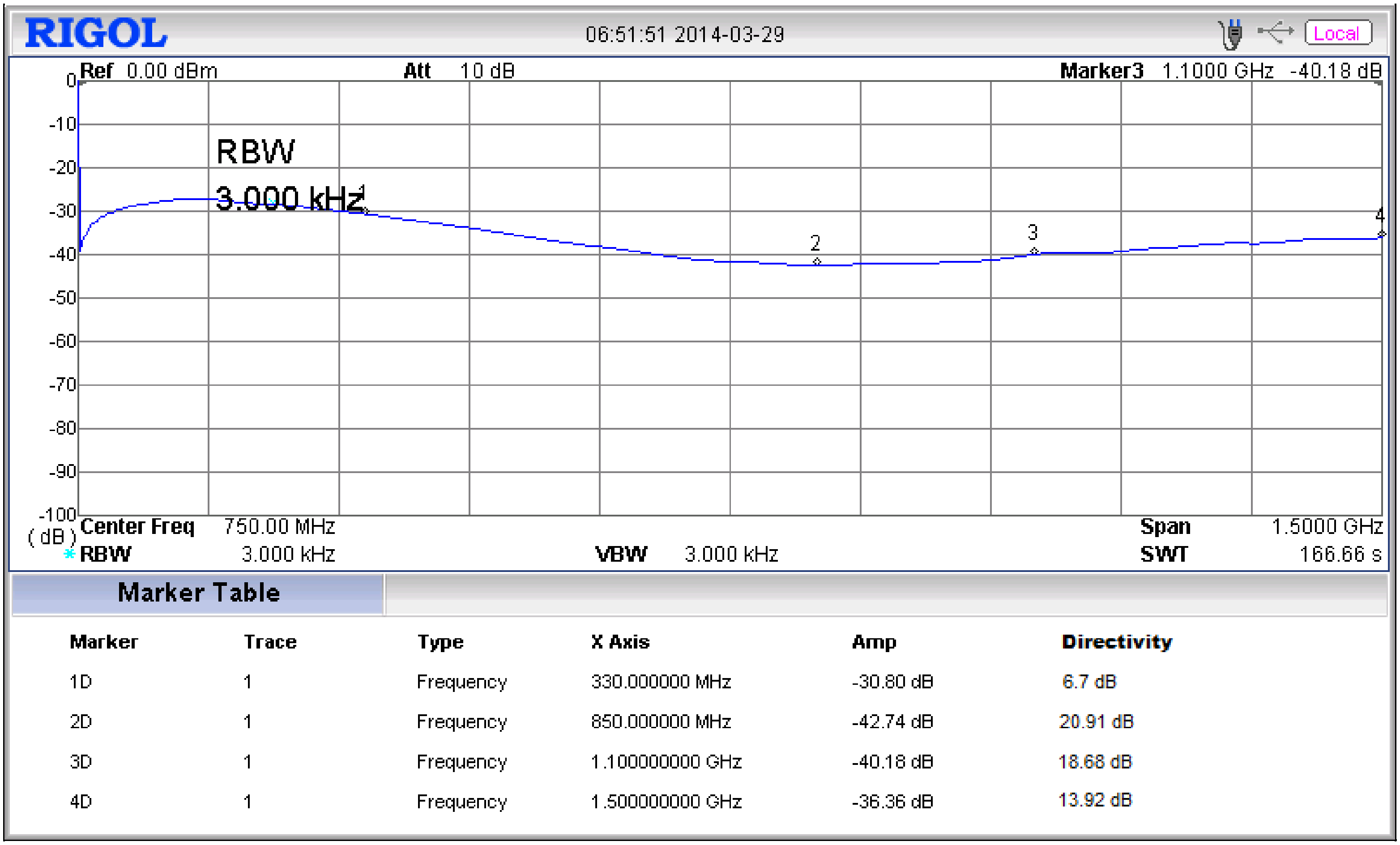
RBW 3.000 kHz **VBW** 3.000 kHz **SWT** 166.66 s

Marker Table

Marker	Trace	Type	X Axis	Amp
1D	1	Frequency	330.000000 MHz	-24.10 dB
2D	1	Frequency	850.000000 MHz	-21.83 dB
3D	1	Frequency	1.100000000 GHz	-21.50 dB
4D	1	Frequency	1.500000000 GHz	-22.44 dB

Open circuit sweep

Above — Open circuit sweep of the 3-bead bridge with series capacitors on the load and reference ports.

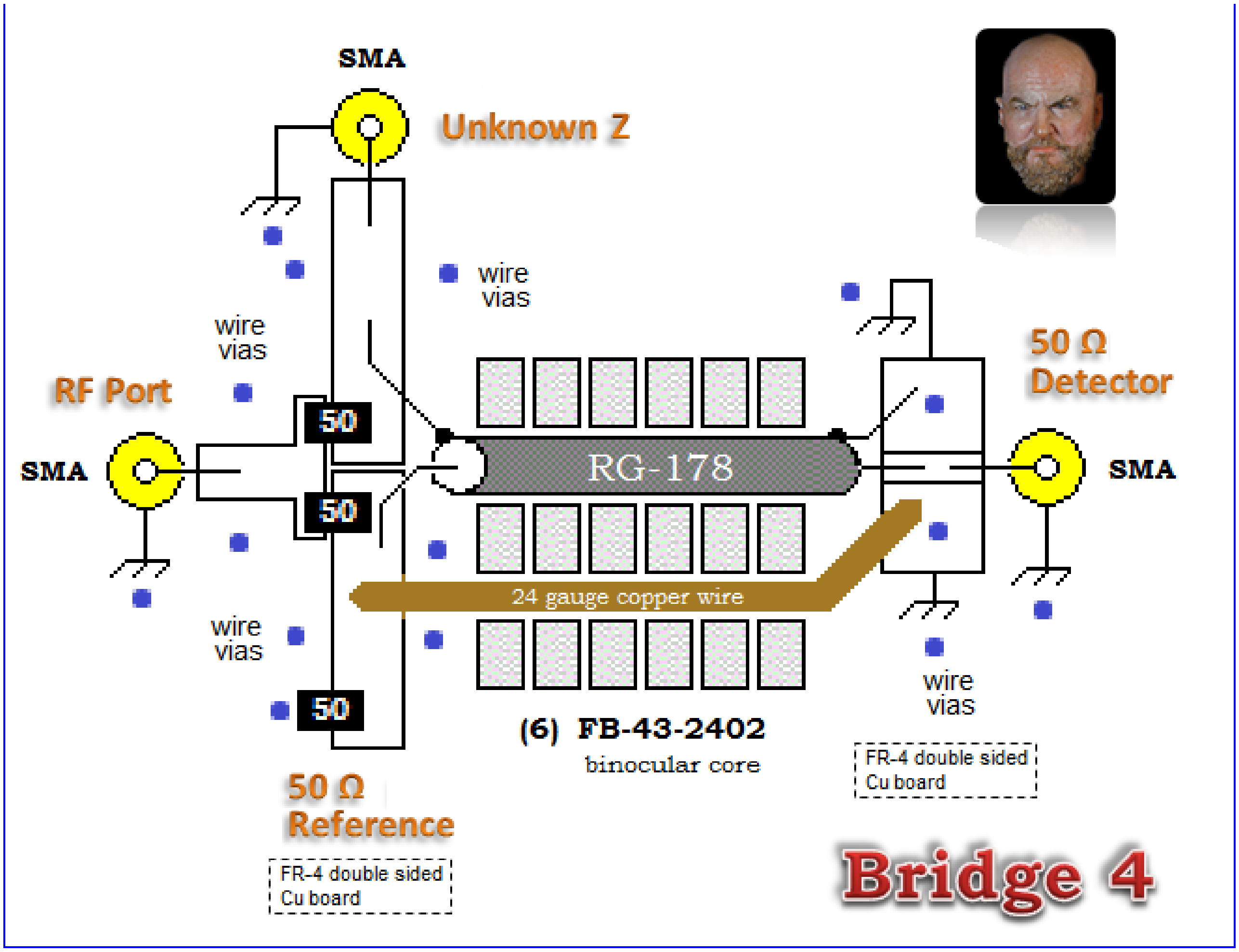


Marker	Trace	Type	X Axis	Amp	Directivity
1D	1	Frequency	330.000000 MHz	-30.80 dB	6.7 dB
2D	1	Frequency	850.000000 MHz	-42.74 dB	20.91 dB
3D	1	Frequency	1.100000000 GHz	-40.18 dB	18.68 dB
4D	1	Frequency	1.500000000 GHz	-36.36 dB	13.92 dB

Above — 50 Ω load sweep. By carefully tweaking the 2 trimmer caps, I set the deepest NULL at ~ 850 MHz; although it's not as deep as I want. Low end directivity was compromised.

Return Loss Bridge 4

Since my bench experiments with the "string of pearls" (SOP) return loss bridge showed promise, I kept going and eventually built a version closer to my goal: A bridge for 300 MHz to 2 GHz with decent directivity.

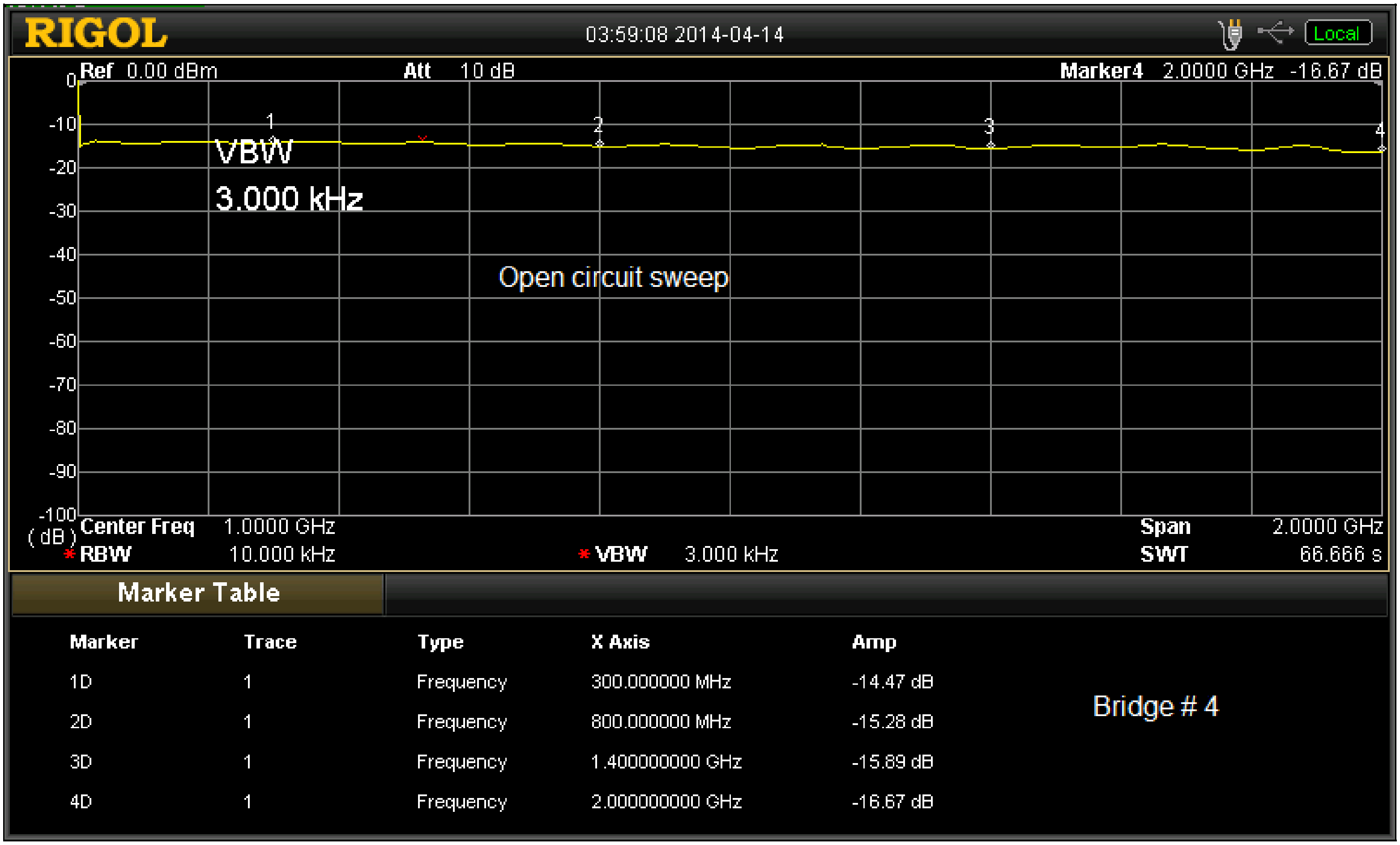


Above — My SOP build diagram of Bridge #4. I cut traces in 2-sided FR-4 Cu board to make the needed microstripline paths.The two 49.9 Ω 1% bridge resistors = size 0603, however, my reference port 49.9 Ω, 1% resistor was size 1206. The braid of the coax goes to the unknown impedance port and the center core to the reference port. Six FB-43-2402 binocular cores just passed the diameter of the RG-178 coax and binocular ferrites boosted balance compared to seperate beads in my experiments.

Rather than run coax through the remaining hole in the ferrite string, I just used a 24 gauge piece of Cu wire for balun symmetry. I left the green insulation on the wire to help keep it in place during soldering — I used the same wire spool to make the via wires.

Above — Breadboard photograph. Click for a larger photo. I soldered some thin brass sheet metal to electrically join the ground planes of the 2 carved boards [represents the normal RF-tight metal chassis we apply].The brass sheet boosted directivity 8 dB by improving balun function.

Comments by a prominent figure in our hobby ....."and the thing is 'ugly to the core'. Way to go."



03:59:08 2014-04-14

Ref 0.00 dBm **Att** 10 dB **Marker4** 2.0000 GHz -16.67 dB

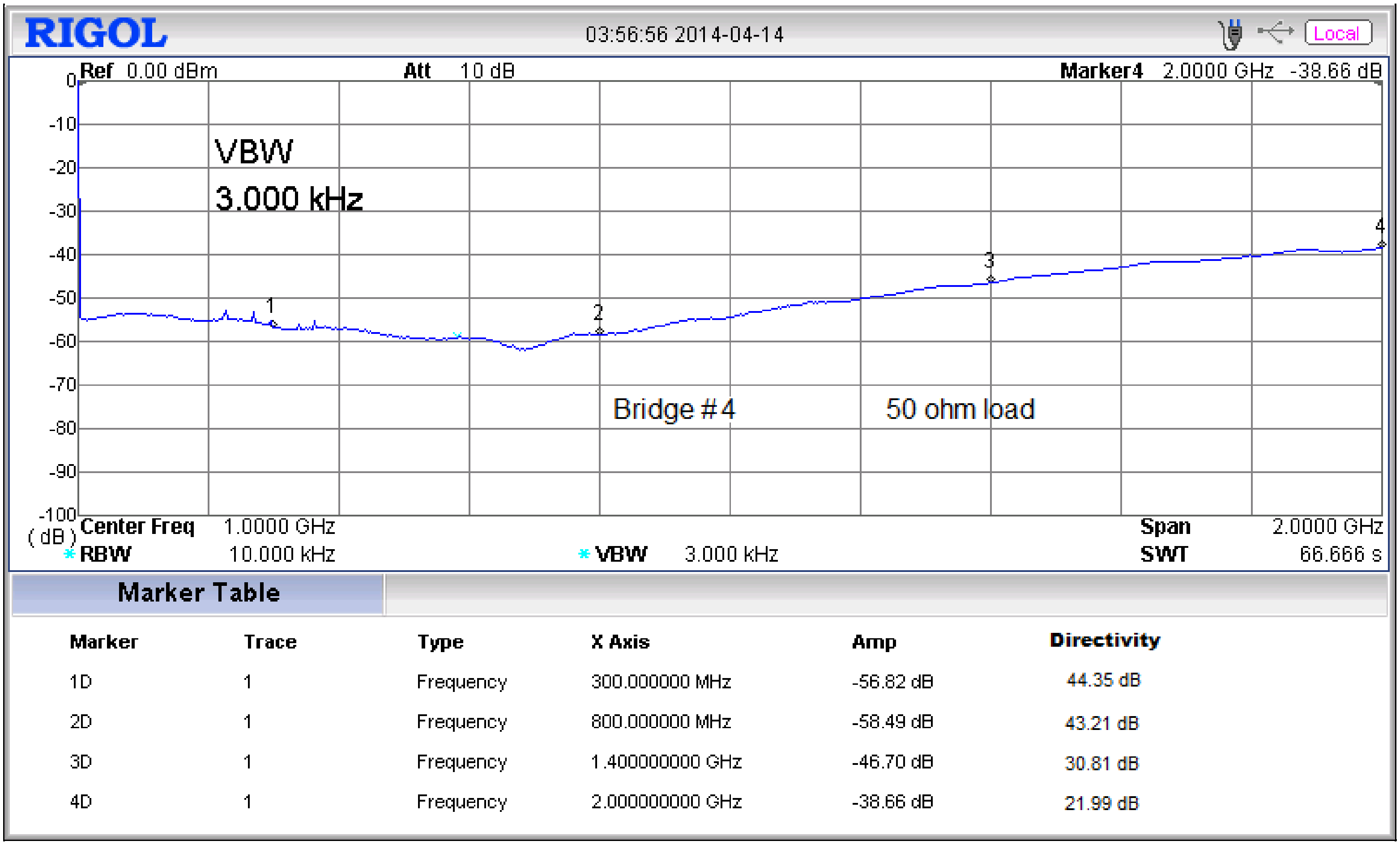
Center Freq 1.0000 GHz **Span** 2.0000 GHz

RBW 10.000 kHz **VBW** 3.000 kHz **SWT** 66.666 s

Marker	Trace	Type	X Axis	Amp
1D	1	Frequency	300.000000 MHz	-14.47 dB
2D	1	Frequency	800.000000 MHz	-15.28 dB
3D	1	Frequency	1.400000000 GHz	-15.89 dB
4D	1	Frequency	2.000000000 GHz	-16.67 dB

Bridge # 4

Above — Open circuit sweep with a 2 GHz span and center frequency of 1 GHz.



03:56:56 2014-04-14

Ref 0.00 dBm **Att** 10 dB **Marker4** 2.0000 GHz -38.66 dB

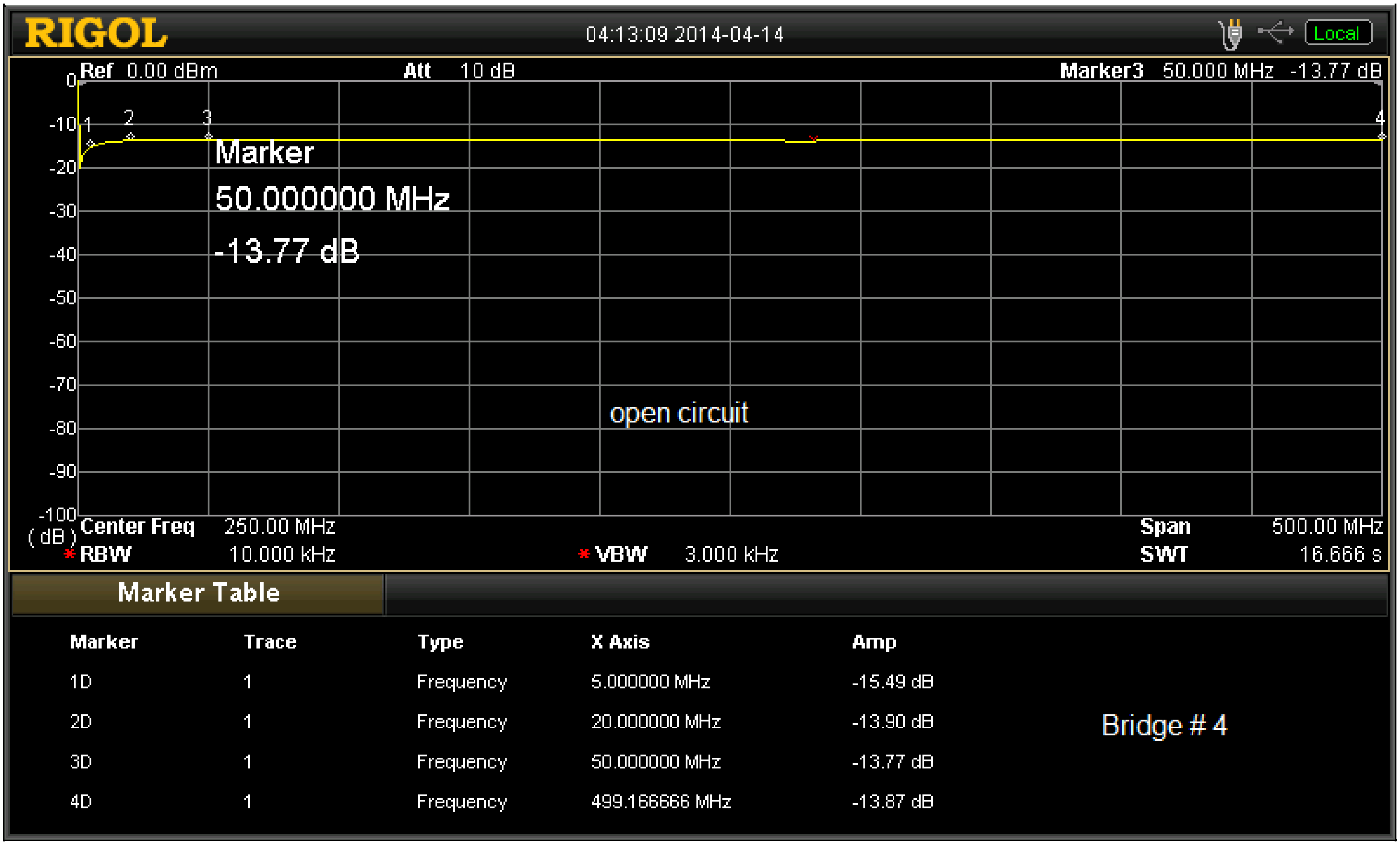
Center Freq 1.0000 GHz **Span** 2.0000 GHz

RBW 10.000 kHz **VBW** 3.000 kHz **SWT** 66.666 s

Marker Table

Marker	Trace	Type	X Axis	Amp	Directivity
1D	1	Frequency	300.000000 MHz	-56.82 dB	44.35 dB
2D	1	Frequency	800.000000 MHz	-58.49 dB	43.21 dB
3D	1	Frequency	1.400000000 GHz	-46.70 dB	30.81 dB
4D	1	Frequency	2.000000000 GHz	-38.66 dB	21.99 dB

Above — Sweep with a 50 Ω load threaded on the unknown impedance port and 4 markers showing directivity at those 4 frequencies. Yay — this is much better! I wanted at least 30 dB directivity at 1296 MHz and got it.



Ref 0.00 dBm **Att** 10 dB **Marker3** 50.000 MHz -13.77 dB

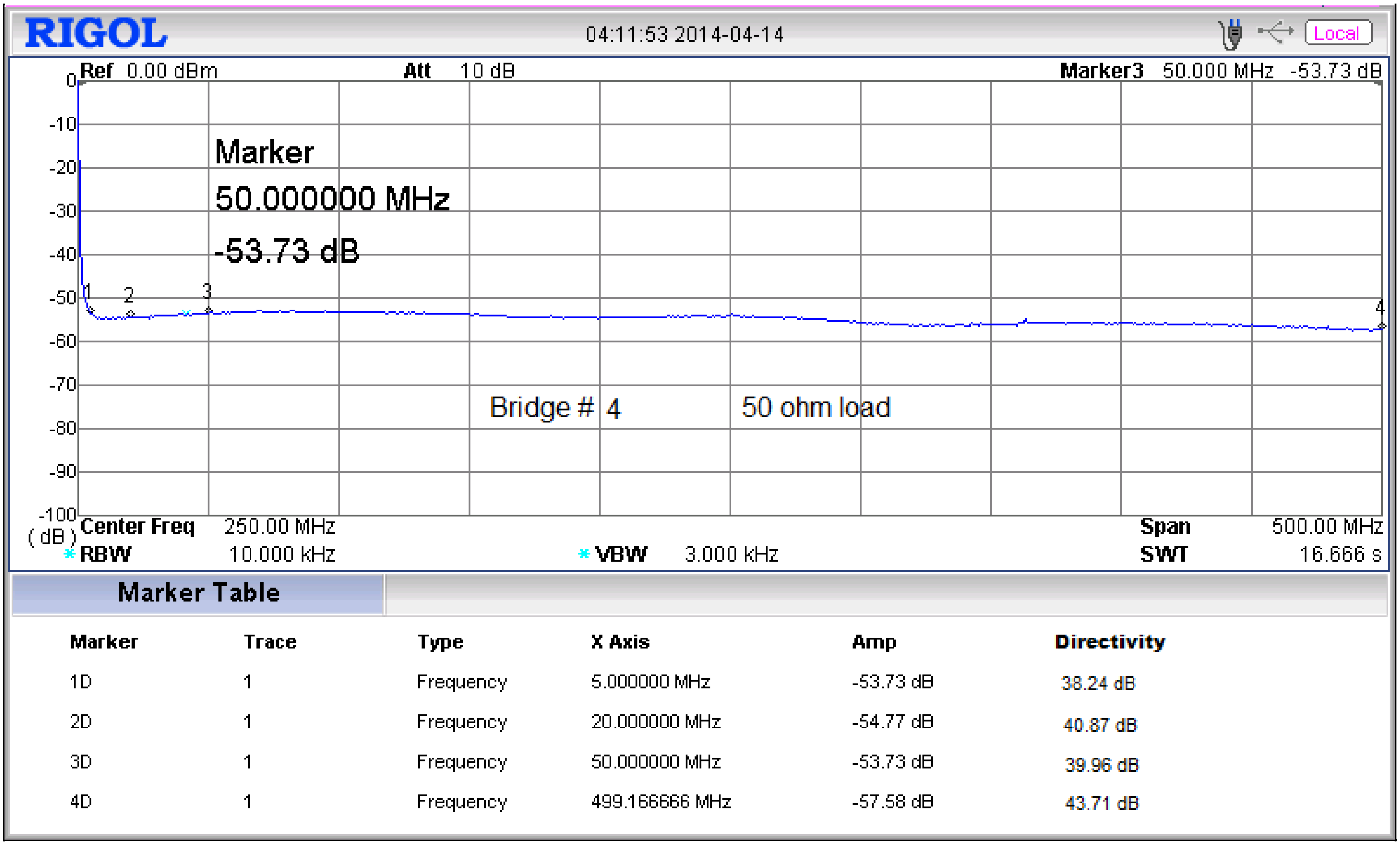
Center Freq 250.00 MHz **Span** 500.00 MHz

*RBW 10.000 kHz *VBW 3.000 kHz **SWT** 16.666 s

Marker Table

Marker	Trace	Type	X Axis	Amp
1D	1	Frequency	5.000000 MHz	-15.49 dB
2D	1	Frequency	20.000000 MHz	-13.90 dB
3D	1	Frequency	50.000000 MHz	-13.77 dB
4D	1	Frequency	499.166666 MHz	-13.87 dB

Above — Open circuit sweep with a 500 MHz span and center frequency of 250 MHz. I set 2 of the markers to evaluate the directivity at HF.



Ref 0.00 dBm
Att 10 dB
Marker3 50.000 MHz -53.73 dB

Center Freq 250.00 MHz
Span 500.00 MHz
RBW 10.000 kHz
VBW 3.000 kHz
SWT 16.666 s

Marker Table

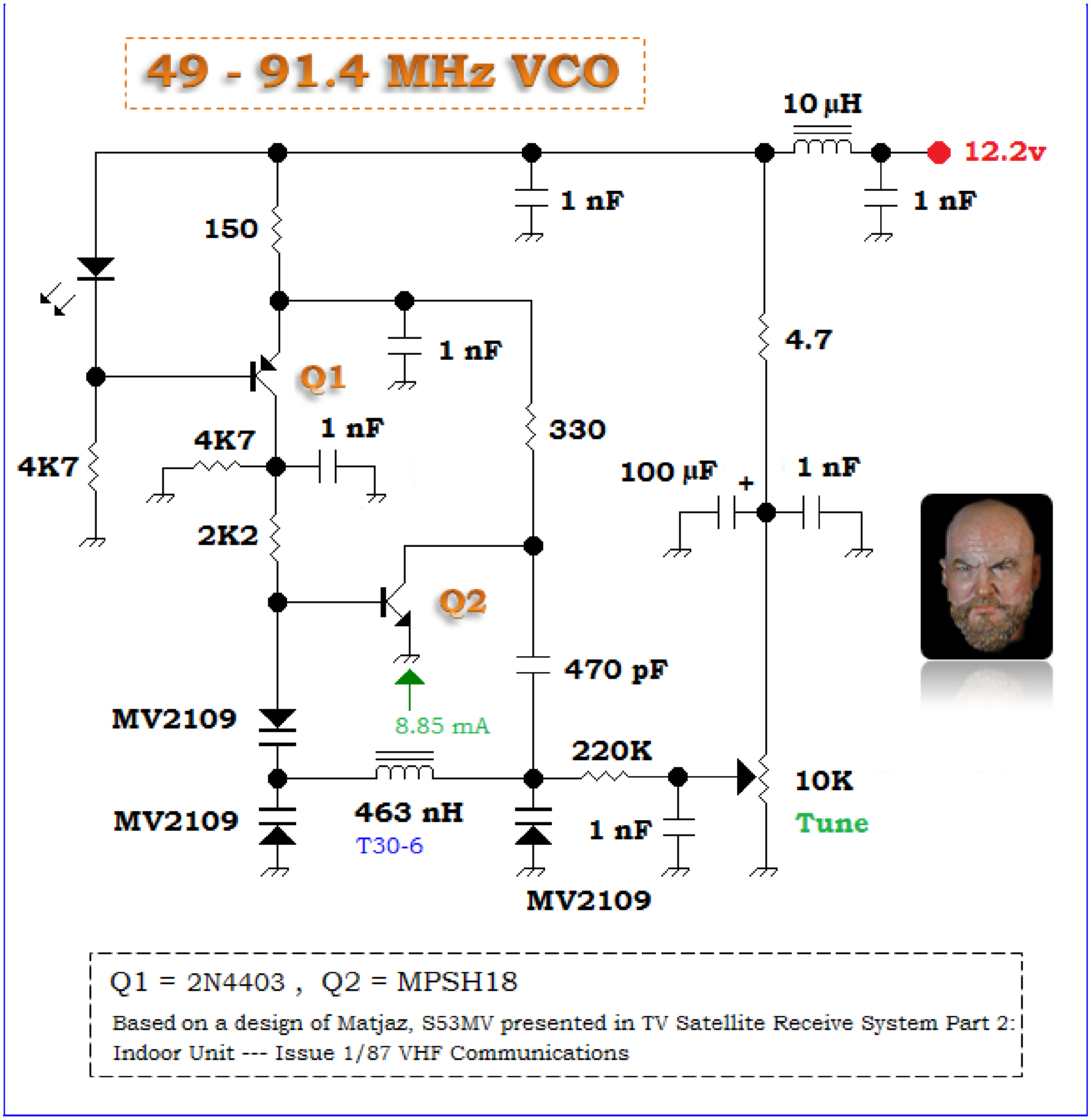
Marker	Trace	Type	X Axis	Amp	Directivity
1D	1	Frequency	5.000000 MHz	-53.73 dB	38.24 dB
2D	1	Frequency	20.000000 MHz	-54.77 dB	40.87 dB
3D	1	Frequency	50.000000 MHz	-53.73 dB	39.96 dB
4D	1	Frequency	499.166666 MHz	-57.58 dB	43.71 dB

Above — 500 MHz sweep of Bridge #4 with a 50 load on the unknown impedance port. The directivity at HF surprised me. Although better suited for VHF-UHF, this bridge will work okay from 5 MHz to ~1.5 GHz.

---

4. VCO Experiments

Ask experimenters about homebrew RF oscillators and you might hear about a pleasing paradox — vexing and satisfying in the same breath! At UHF we usually make a VCO and keep it on frequency by phase/frequency locking to a xtal oscillator reference. I sought some wideband, free running VCOs in hopes I'd inch up the learning curve.



Q1 = 2N4403 , Q2 = MPSH18

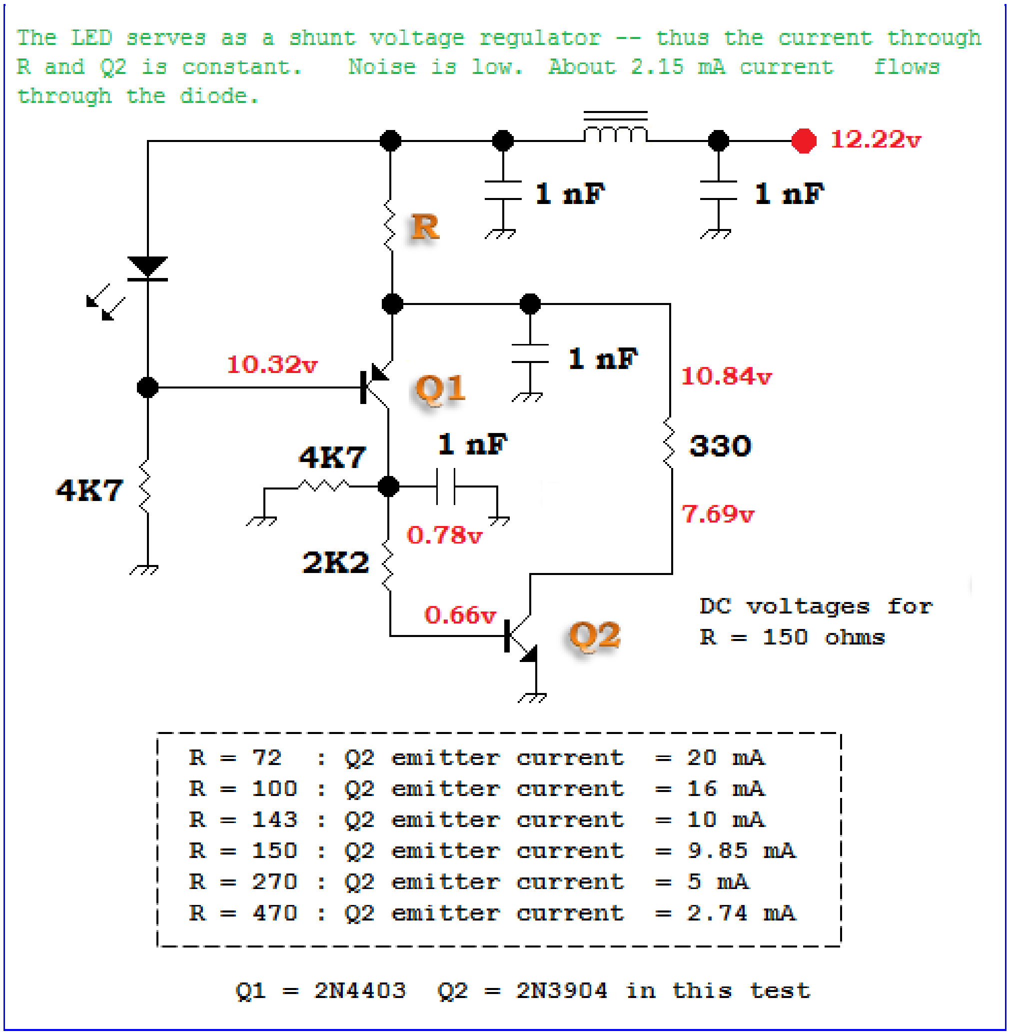
Based on a design of Matjaz, S53MV presented in TV Satellite Receive System Part 2: Indoor Unit --- Issue 1/87 VHF Communications

Above — My first attempt at a VCO circuit described by Matjaž, S53MV in a 2-part article published in VHF Communications magazine. The original circuit contained a tunable RF filter, mixer, buffer amplifer + more and I recommend you purchase the 2 articles for your own analysis. Shown with permission from Andy Barter of VHF Communications magazine, I ordered all issues from the 1980s which came on this DVD.

With 28v varactors, this VCO will tune 1 octave minimum.

To bench investigate this stunning common emitter Colpitts variant, I first built it at VHF using through-hole parts with a standard 12 volt tuning voltage that tuned from 49 to 91.4 MHz. The output came from 1 link around the 463 nH inductor terminated in a 1K resistor at the hot end + ground at the cold end to allow 10X probe measurement with my DSO.

Matjaž, S53MV describes his circuit function here. A variable C on the base in combination with the input capacitance of Q2 operates as a tuned impedance matching network and preserves a steep phase slope (which means you have higher Q) — a requirement for low phase noise. In essence, the inductor with a varactor at each end forms an adjustable pi-network to provide correct phase shift tracking.



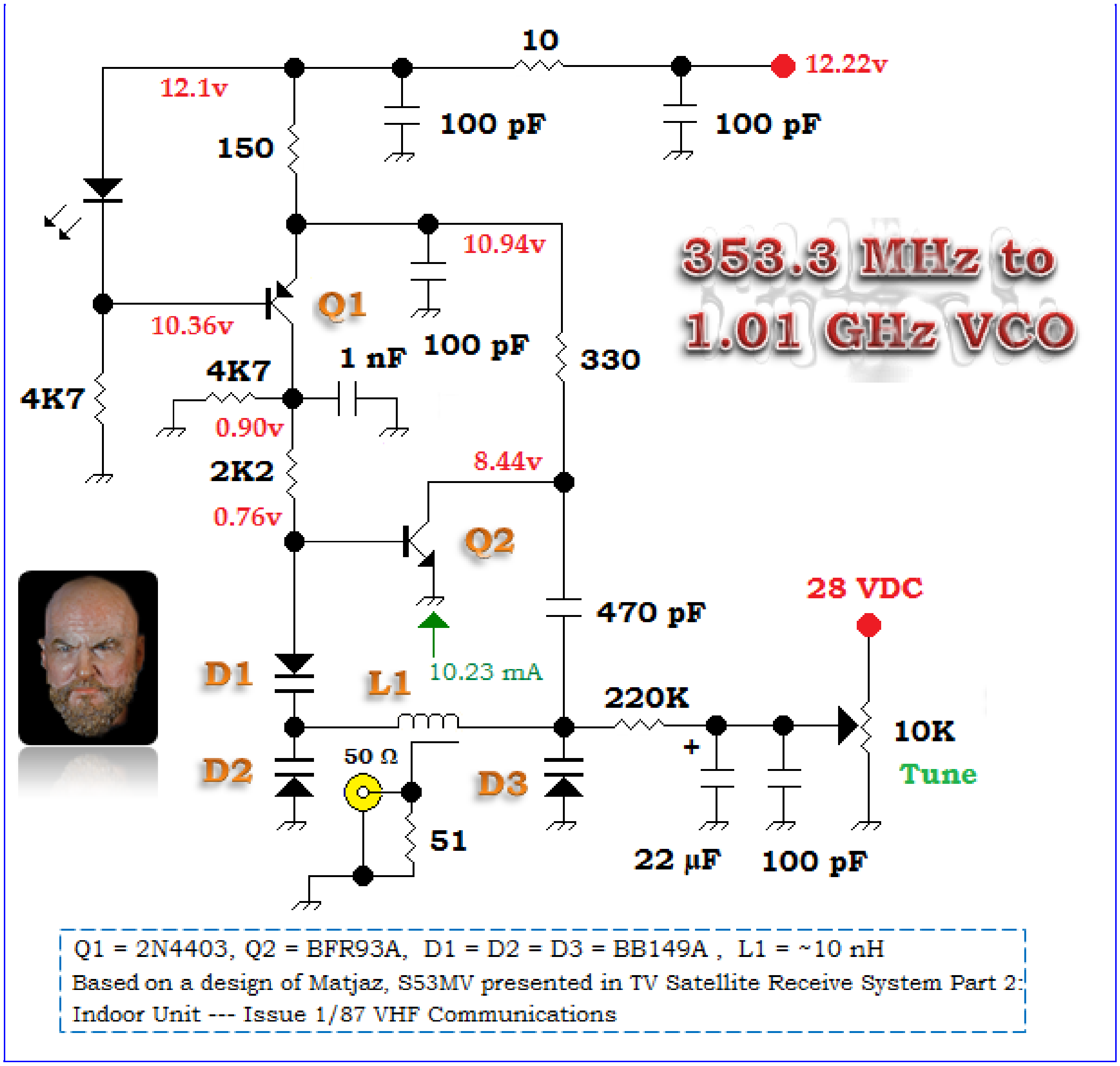
The LED serves as a shunt voltage regulator -- thus the current through R and Q2 is constant. Noise is low. About 2.15 mA current flows through the diode.

DC voltages for
R = 150 ohms

R = 72 : Q2 emitter current = 20 mA
R = 100 : Q2 emitter current = 16 mA
R = 143 : Q2 emitter current = 10 mA
R = 150 : Q2 emitter current = 9.85 mA
R = 270 : Q2 emitter current = 5 mA
R = 470 : Q2 emitter current = 2.74 mA

Q1 = 2N4403 Q2 = 2N3904 in this test

Above — I built the low-noise, temperature-stable, PNP current source using though-hole parts. R provides a convenient way to vary collector current, although changing the PNP base bias also works.



Q1 = 2N4403, Q2 = BFR93A, D1 = D2 = D3 = BB149A , L1 = ~10 nH
Based on a design of Matjaz, S53MV presented in TV Satellite Receive System Part 2:
Indoor Unit --- Issue 1/87 VHF Communications

Above — My UHF VCO schematic. Click for the breadboard photo. A wire positioned near L1 provided the output into my spectrum analyzer. Click for a sweep with the reverse DC voltage set at 0v. Click for the a sweep with the tuning at 28v to establish the lowest possible varactor capacitance. Markers indicate the carrier frequency. As shown, a common emitter VCO exhibits rich harmonic energy and viewing the second harmonic at 2 GHz felt really cool in the 28v sweep. Not to worry — the tracking generator in the Rigol TG + SA units also exhibit strong harmonics and work fine for sweeping.

Click for a pdf file showing how I laid out my board. I cut traces on the top of 2-sided FR4 board and connected any grounded points on top to the bottom ground plane with a 24 gauge copper via wire. A 28v DC-DC Converter to supply the varactors lies on VHF-FM.

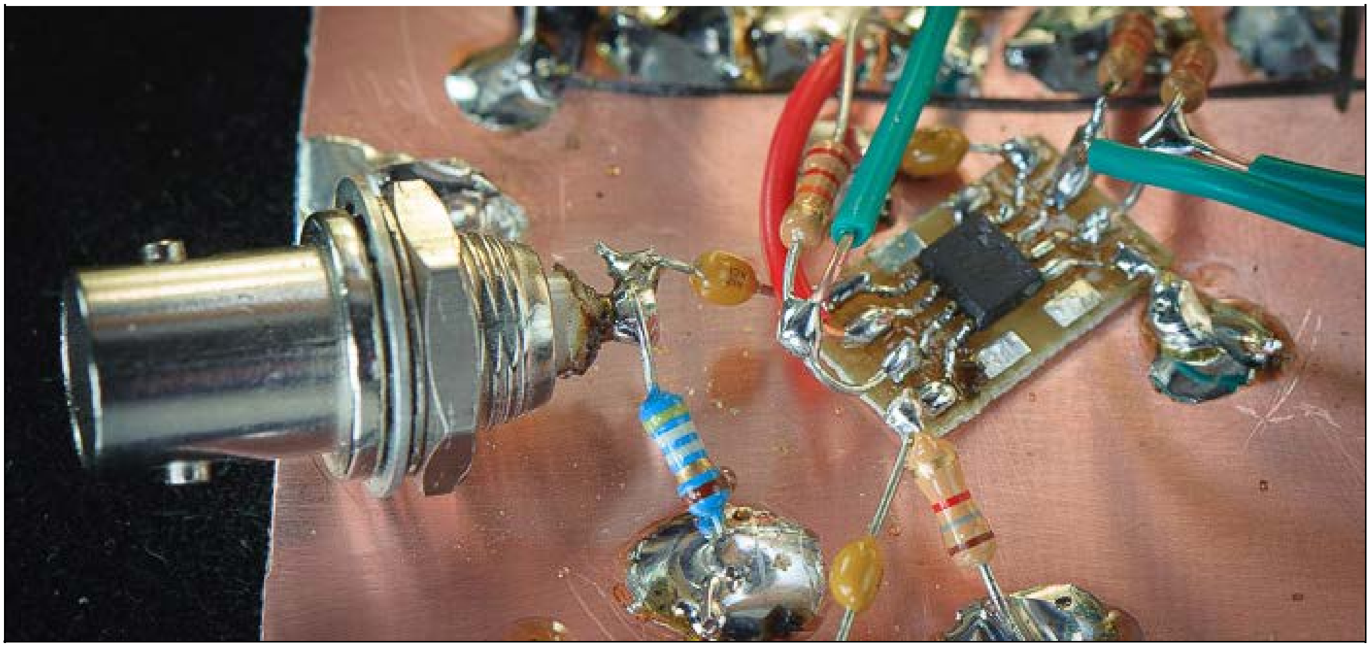
I varied L and even tried a 3 mm by 8 mm copper sheet metal inductor but this failed to oscillate. In the end, a single wire with ~10 nH inductance worked to give me the tuning span shown. L1's distance above the ground plane was determined by tuning the VCO for the smoothest transistion across its range. This circuit begs further experiments including Q2 current, L1 and the 470 pF fixed feedback capacitor.

---

5. Frequency Counter Kit

A sensitive UHF bandwidth frequency counter makes an important tool for the UHF homebrew experimenter. With a search engine, you'll find commercial and kit counters for sale along with DIY projects for builders handy with microcontrollers.

If you already own a decent counter, then a crude prescaler tool employing an IC like the MC12079 might work okay if you can tolerate or manage modulus division by 64, 128, or 256. The MC12079 requires an RF signal between -17 dBm to +3 dBm to count properly, so many builders place amplifier(s) along with anti-parallel clamping diodes before the digital IC. The Motorola datasheet shows a basic circuit to build off.



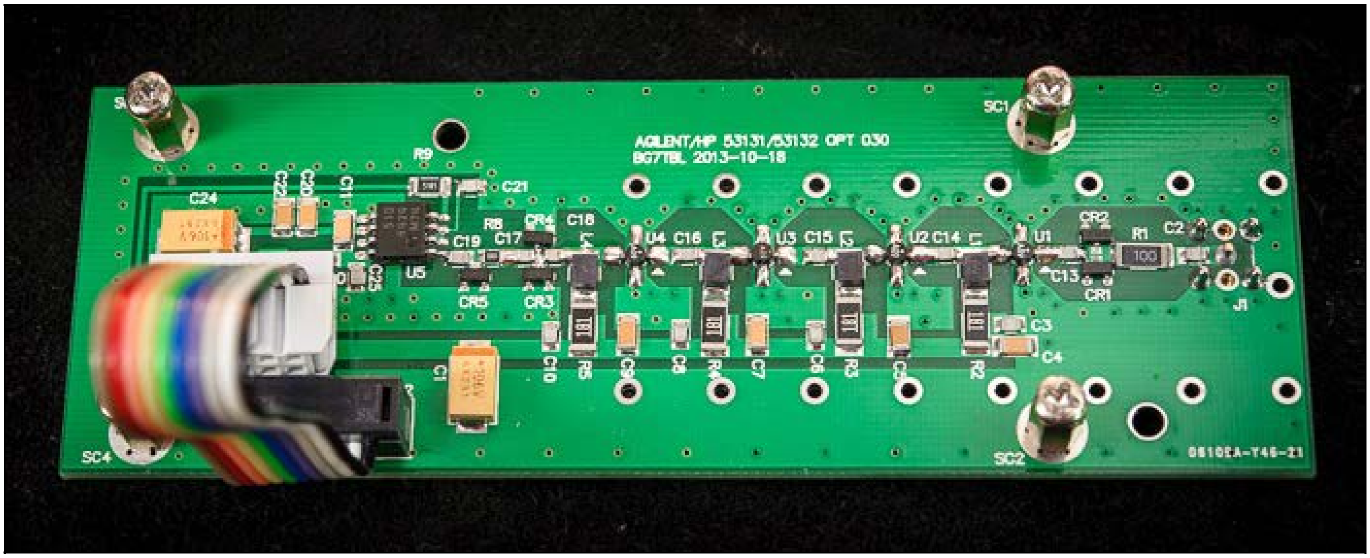
Above — My first UHF counter experiment involved the Motorola MC12079 and 12080 prescalers. Eventually I made a proper double-sided copper board device with anti-parallel diodes, a MMIC, the MC12079 and some 74AC series logic — it worked okay for casual use. The 2 big drawbacks were MC12079 self-oscillation with 0 applied input signal and having to remember to multiply measures by my set prescaler division integer. I sought a long-term solution.

A Solution Appears

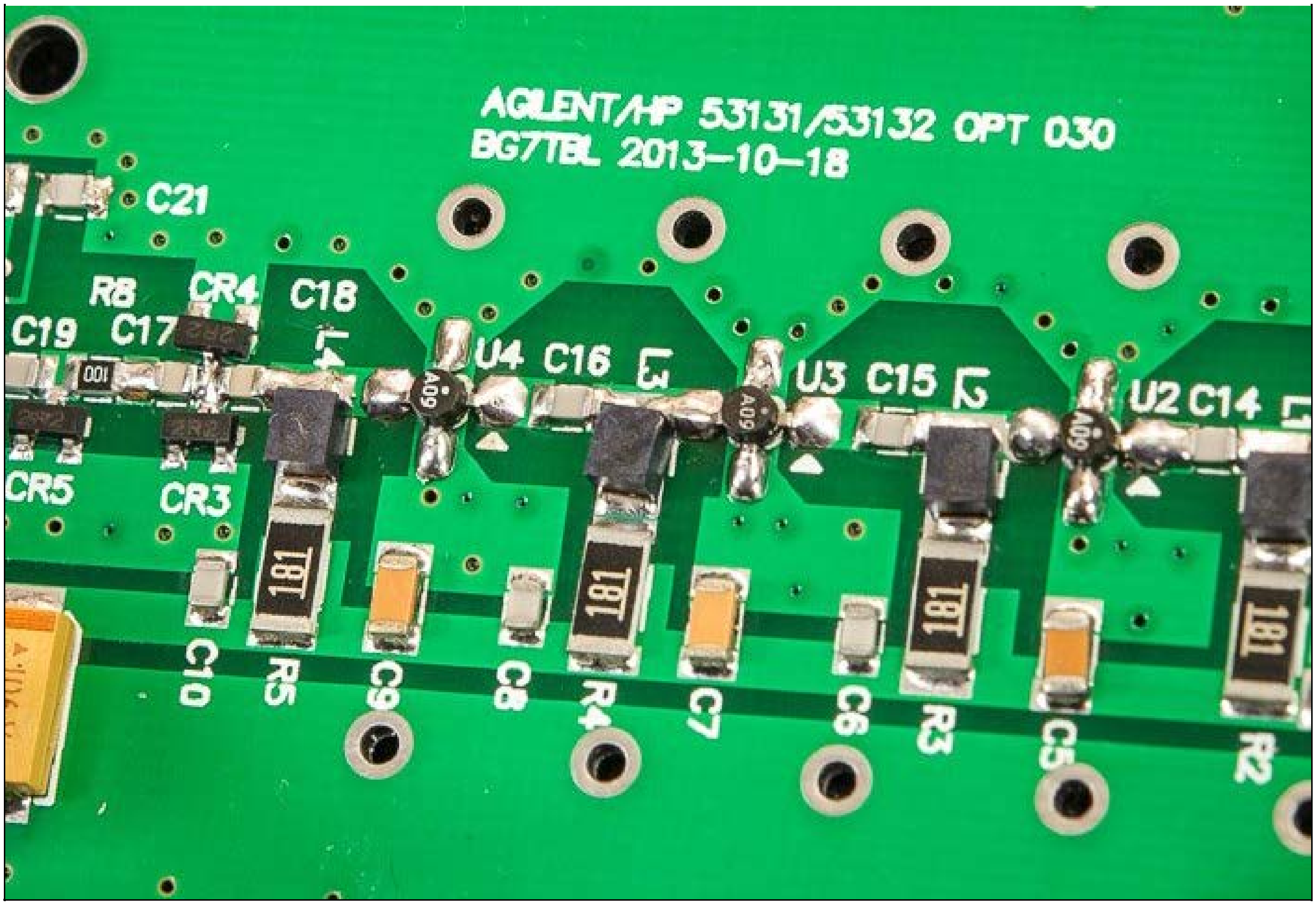
In 2012 I bought a second-hand, calibrated HP 53131A with the optional A6 High Stability Timebase from an EE for ½ the going price at the time. 225 MHz bandwidth poses its only limitation; it even came with the original spiral bound manual.

This summer, Shahriar from TheSignalPath.com reviewed a Chinese kit to take the HP 53131A to 3 GHz with a clone of the original HP OPT30 Channel 2 kit. Based on his quantitative review and warm thoughts of using my sturdy HP counter at UHF, I got 1. Mine sold on eBay for $87 Canadian including shipping. Click for TheSignalPath.com video. Click for a larger photo of the kit I bought.

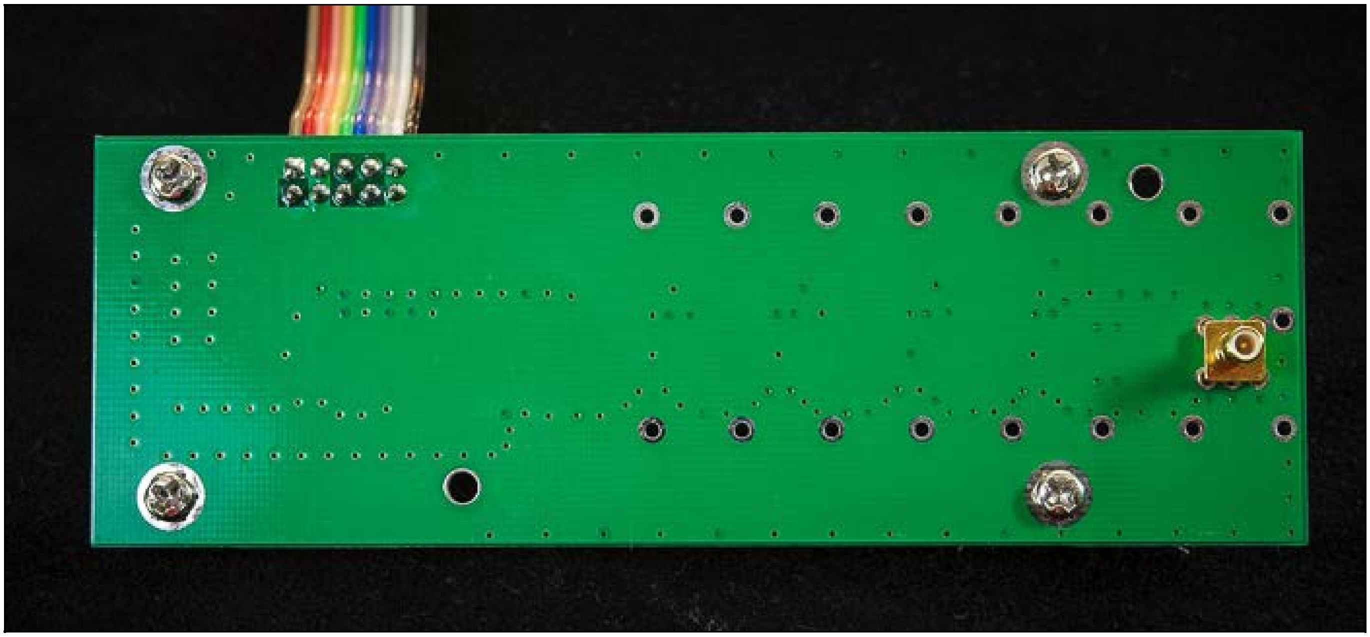
Above — My HP counter with the kit installed as Channel 2.



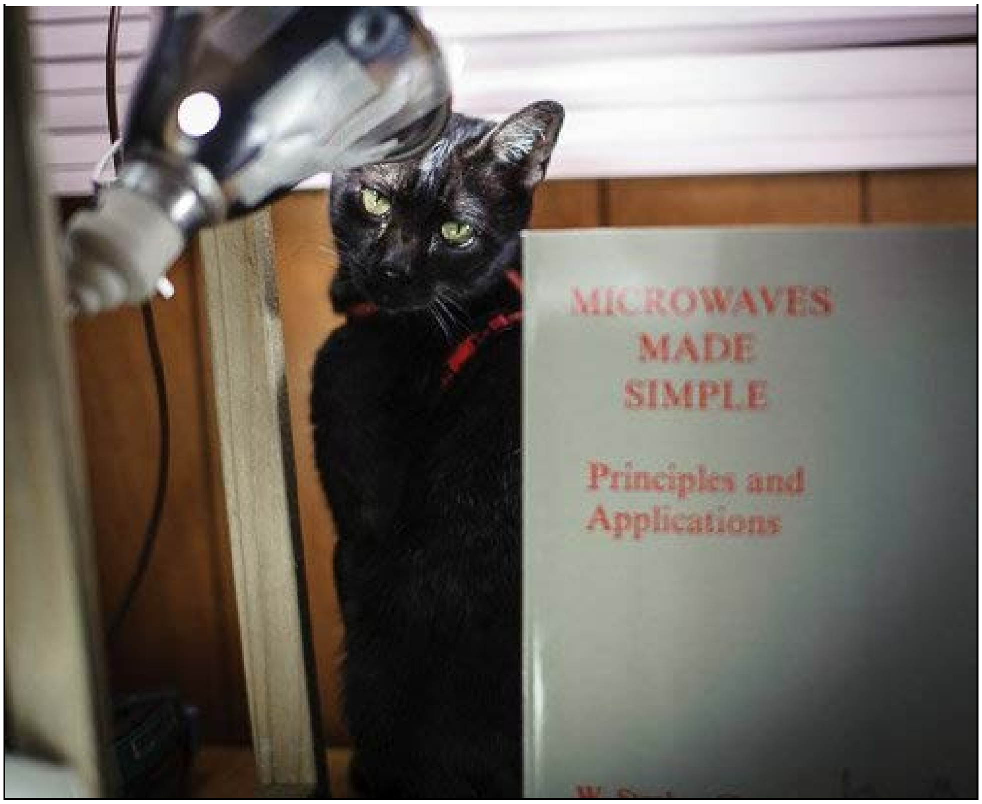
Above — Top of the circuit board. Anti-parallel diodes, 4 MMICS, passives and then even more anti-parallel diodes go before the prescaler chip.



Above — Close up of SMD circuitry.



Above — Rear view of the circuit board showing stiching vias that join the top ground plane to the bottom ground plane along the signal path. My tests showed that it works okay to 3.1 GHz. Click for a picture of the installed board. I'm very happy with my HP counter's boosted bandwidth.


MICROWAVES
MADE
SIMPLE
Principles and
Applications

RF — Test and Measurement

Funster Line — Transmitter

Menu Item 4 on the HF Embarcadero web page. Click here to go back.

The receiver will go on its own page and hopefully I'll complete and write it up this Fall.

1. Transmitter
2. References

Introduction

I began the Funster transceiver line in 1995 and show the 2014 version [now back to a CW trans-receiver on the 40M band]. Judging by reader feedback, plus analog scratch-homebrew threads in social networks, email lists and print, many of us feel a nostalgic pull towards old-school radio. Me too.

Now at version 8 — Funster shares my joy in making a simple, old-style radio with all discrete parts. In some ways, this radio set is a twisted version of Wes and Roger's Ugly Weekender from QST for August 1981.

1. Transmitter

Summary

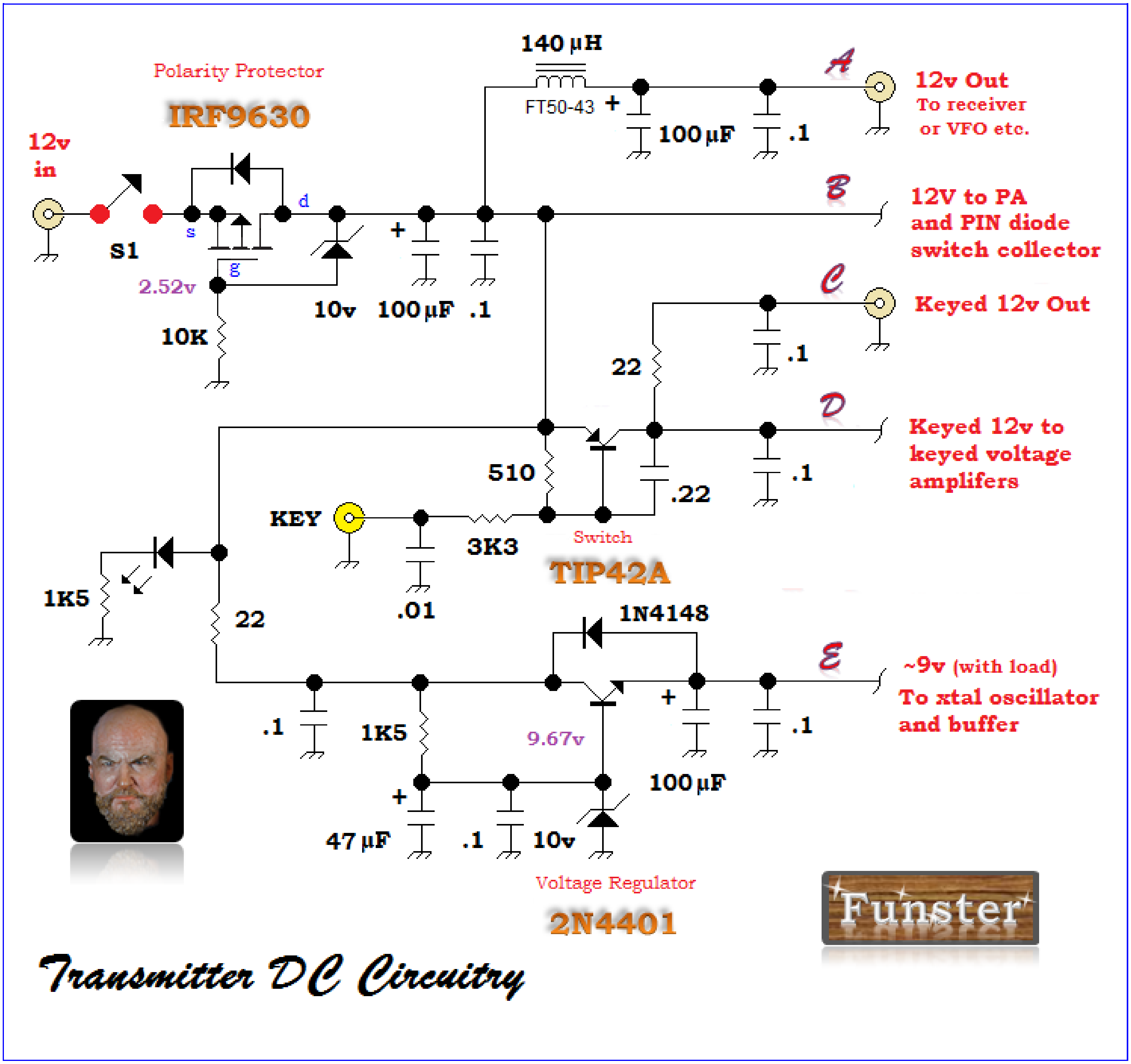
Variable power: 128 to 3312 mW @ 13.6 VDC. Switchable 7.04 MHz xtal oscillator or external VFO input port. Solid state transmit/receive [T/R] circuit with PIN diode. External continuous and switched 12 VDC power jacks. Single-ended BJT Class C power amplifier with 67% efficiency.

Note: I state 12 VDC in my DC circuits, however, this nominal value might better be termed 12 -13.6 VDC.

Key up power current draw @ 13.6 VDC = 17 mA. Key down current @ 3W RF power with 13.6 VDC = 420 mA.

DC Circuitry

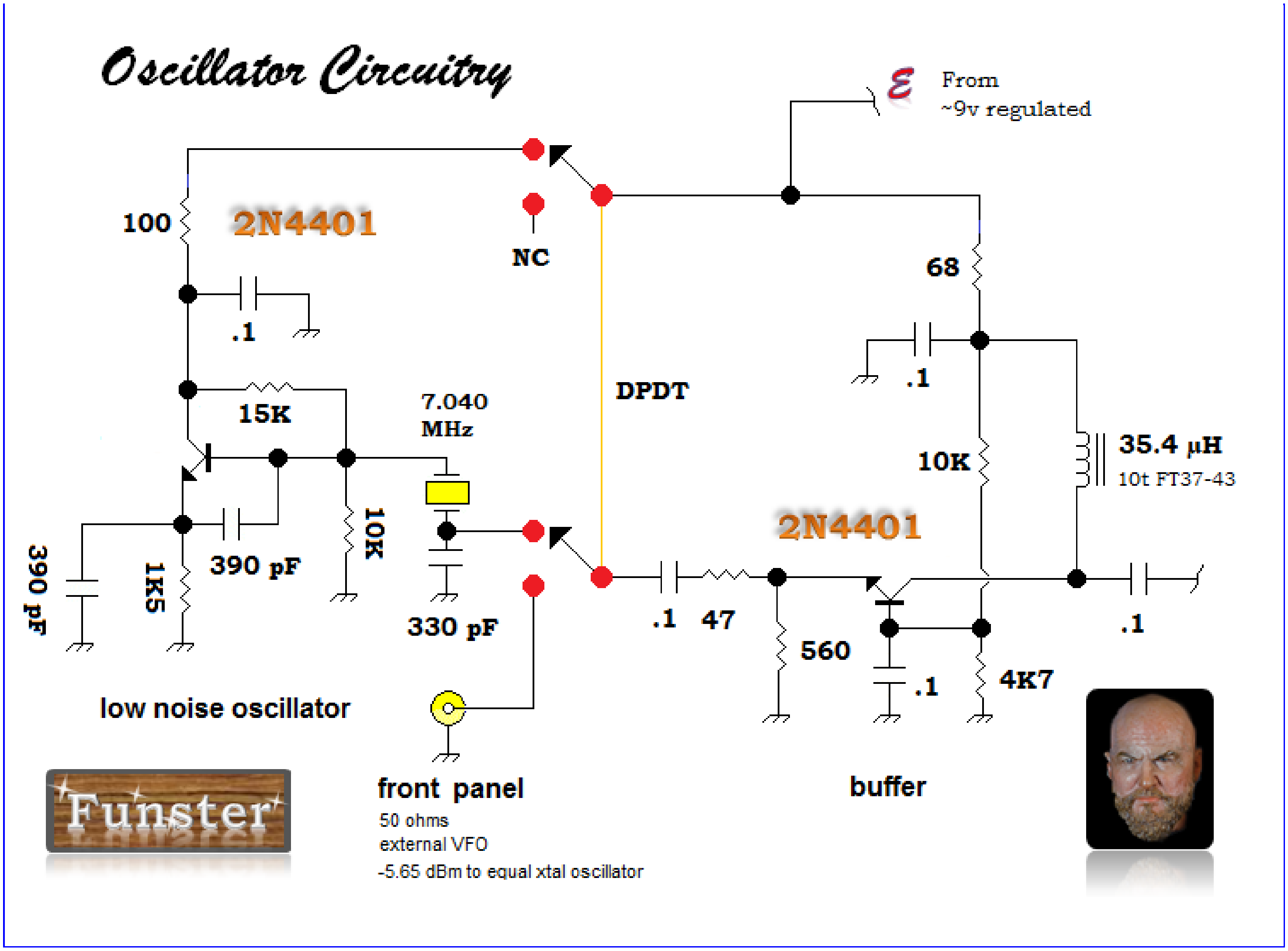
It's fun to march off the signal path and focus on DC circuitry — switching, regulating and filtering the DC voltage feeding our RF or AF stages with finesse. Trans-receivers contain DC, RF, AF and external patch cords that benefit from mindful DC line filtering + selective high-isolation amps in the signal path to prevent RF flowing where it's unwanted. You'll see heaps of decoupling with RF + AF bypass and a few strong reverse-isolation amplifers in this radio set.



Above — The DC circuit board. Click for the breadboard photo. A reader sent me 2 IRF9630 power MOSFETS: avoiding the voltage drop of the usual single rectifier diode, I employed 1 as a reverse polarity protector. Another reader sent me a long strip of 2N4401 BJTs, so I ran them instead of the equally good 2N3904. Thanks for the parts guys!

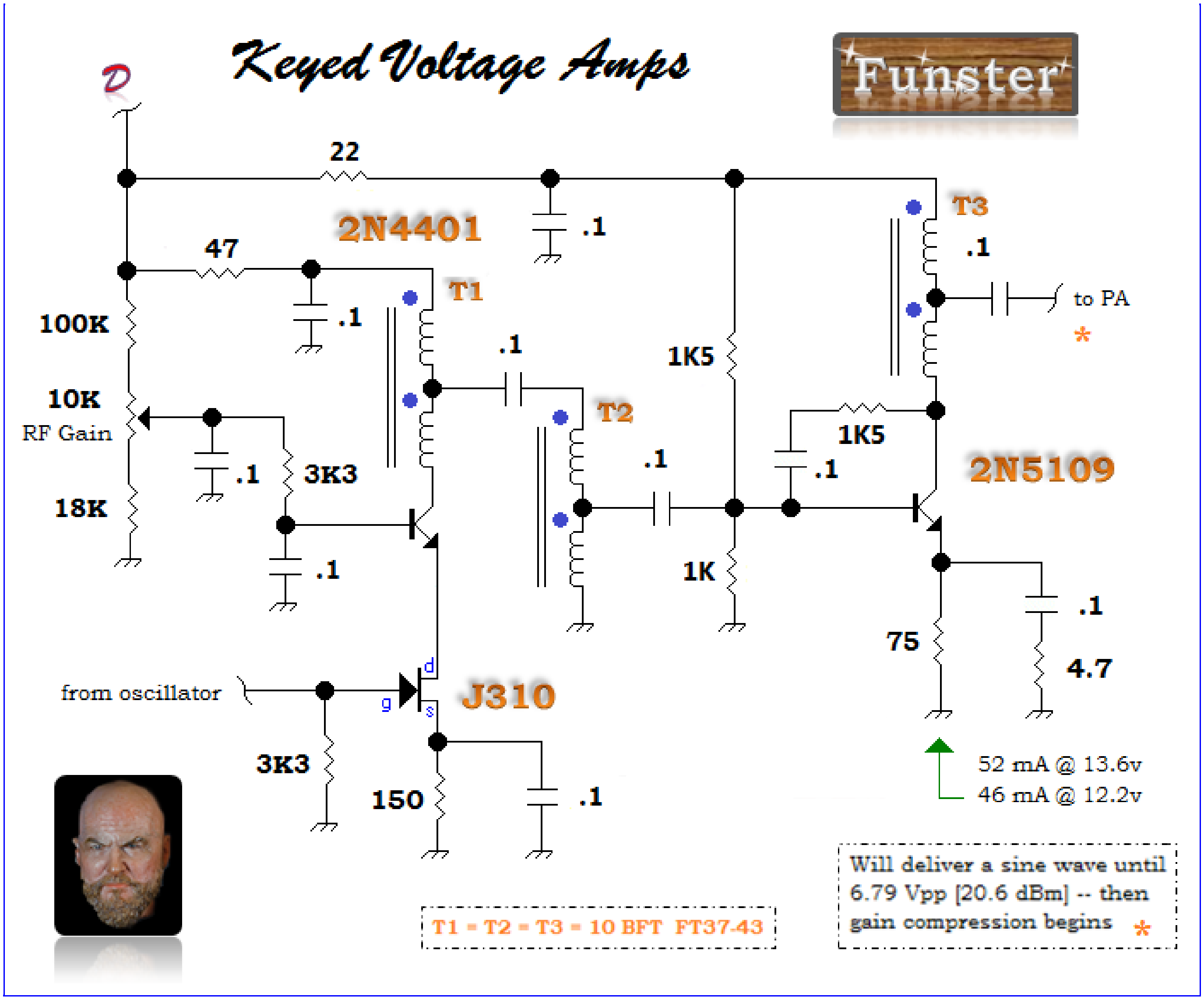
A simple series voltage regulator with a zener diode reference supplies DC to the crystal oscillator and its buffer. The base-emitter voltage drop of the 2N4401 emitter follower lays in series with the load, so load current changes alter the regulator output voltage. Vout = Vzener - VBE : so the regulated voltage is roughly 9 VDC.

Oscillator Circuitry



Above — Oscillator circuitry. I soldered my oscillator circuitry on the DC control board. Click for the breadboard. Click for a close-up of the old NorCal crystal supplied by NT7S — thanks Jason! To equilibrate to the xtal oscillator AC output voltage, the VFO input port should run about -5.65 dBm. I might stick a 6 dB attenuator pad on the front panel switch and use a 7 MHz VFO with 0 dBm output power.

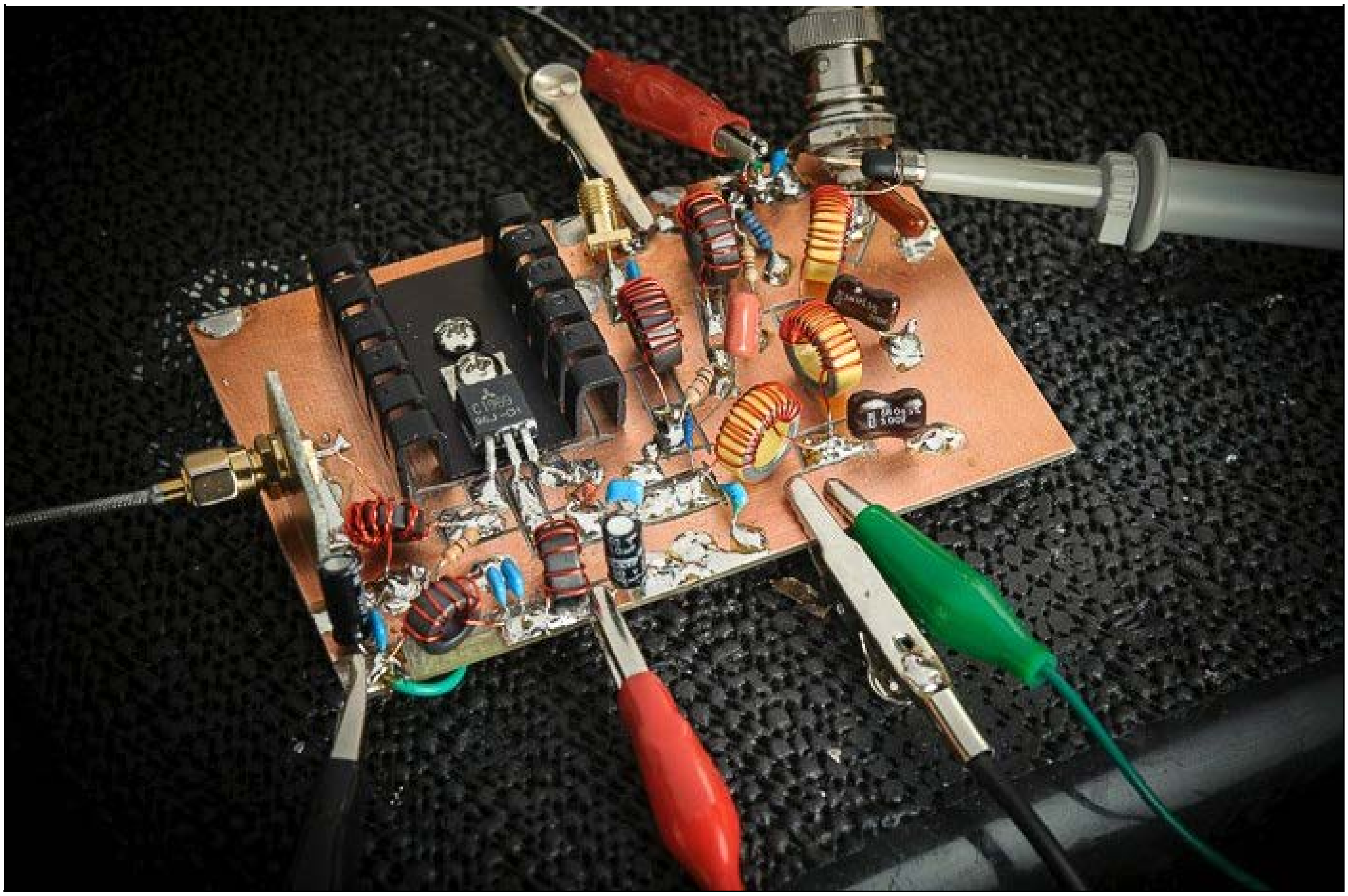
Keyed Voltage Amps



Above — The 2 keyed voltage amplifers. Click for a breadboard photograph. I set the value of the RF gain pot series resistors by placing a 100K pot before and after the 10K gain pot and tweaked them back and forth to find close to 3W maximum and 125 mW minimum RF power with the gain pot set @ CW and CCW respectively. The 100K pots were removed, measured and nearest standard value resistors substituted: 100K and 18K.

By carefully choosing the 75 Ω emitter resistor, the 2N5109 feedback amp output remains a sine wave at the drive levels needed for a PA output at 3 Watts. At 3 Watts PA output, the feedback amp output voltage = 5.91 volts pk-pk [19.4 dBm]. The emitter current shown will cleanly drive a PA for more power if desired. Boost it as needed.

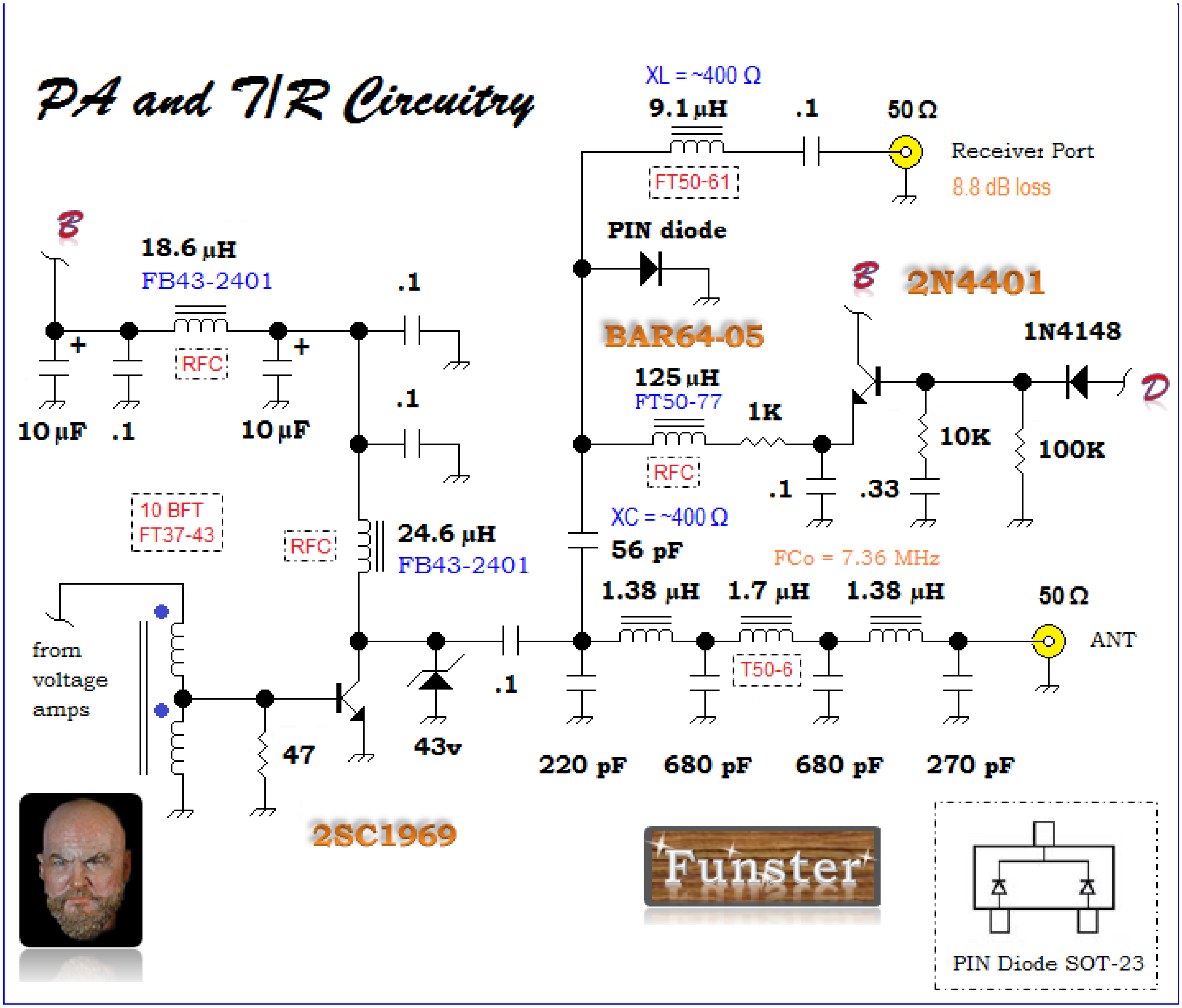
Power Transistor and T/R Switching



Above — Final testing the PA and T/R circuitry before installing it into the Hammond chassis. Click for a bigger photo of my test setup.

Three temporary RF connectors were soldered on this board for bench measurement. I connected the (SMA) receiver port to Channel 1 of my 50 Ω terminated 'scope to measure the RF not getting shunted to ground by the PIN diode. The SMA cable/connector to the far left is a temporary cable connected to the keyed voltage amp board output in the Tx chassis — thus I tested the PA stage with its own xtal oscillator and RF amplifers. The right-sided BNC RF port connects the PA low-pass filter to a 10W, 50 Ω dummy load via a 22 cm 50 Ω patch cable.

Further, I measured collector voltage + collector current with 2 DVMs and also PA output power in my oscillocope (DSO) Channel 2 with a 10X probe employing a tiny ground lead.



Above — PA and T/R circuitry schematic. Click for another breadboard photograph taken before the 2N4401 switch was added. Although we might run a BD139 or other cheap BJT for the final, I chose my favorite HF classic, the 2SC1969. This transistor will survive a high VSWR, however a 43 volt - 500 mW zener diode further protects it from such.

In 1 version with a 1:4 Z collector matching network and no zener diode, I measured 6.25W RF power after tweaking the 100K and 18K RF gain limiting resistors in the preceeding hycas voltage amp to drive the PA harder. At ~3W maximum power in my final experiment, the 2SC1969 barely warms up.

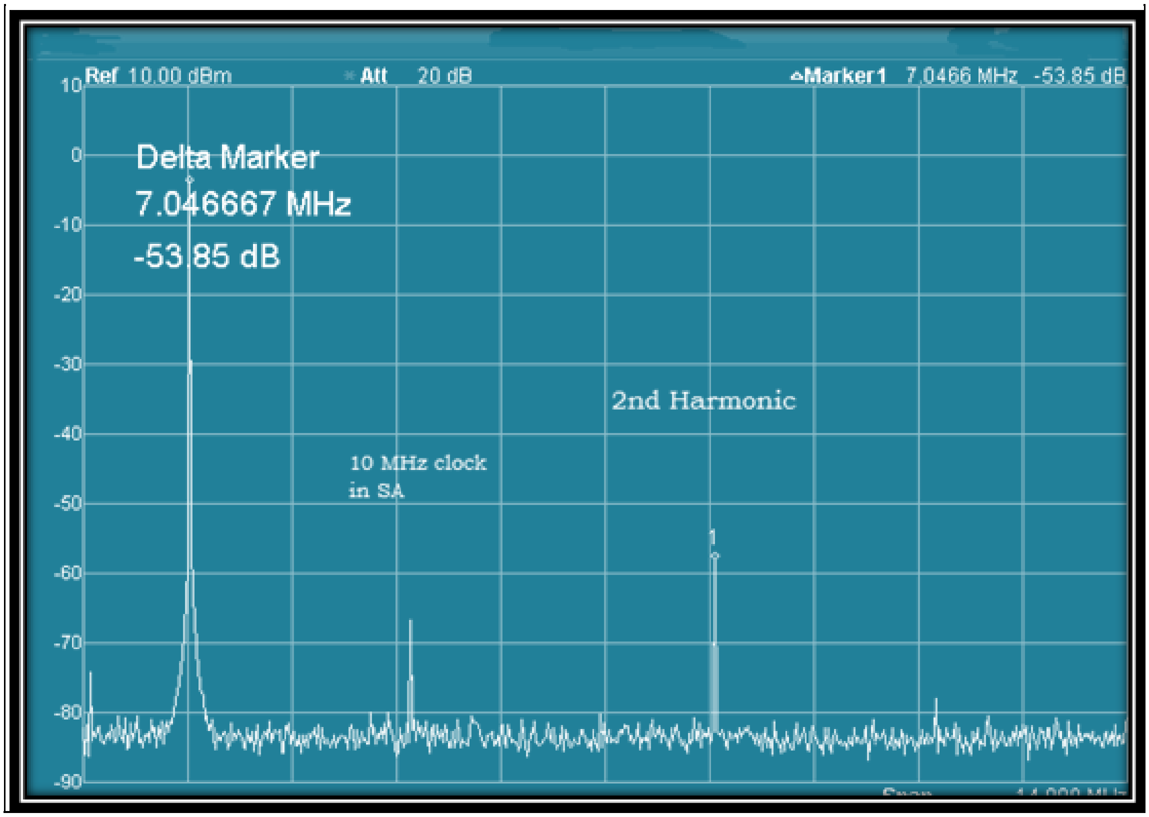
Collector Matching

After a number of experiments, I learned that from tens of mW up to ~ 3.3 Watts RF power, we may omit collector matching and still get reasonable efficiency if you run a collector choke >= 15 µH. Thus, I made the maximum power ~ close to 3W — easy peasy.

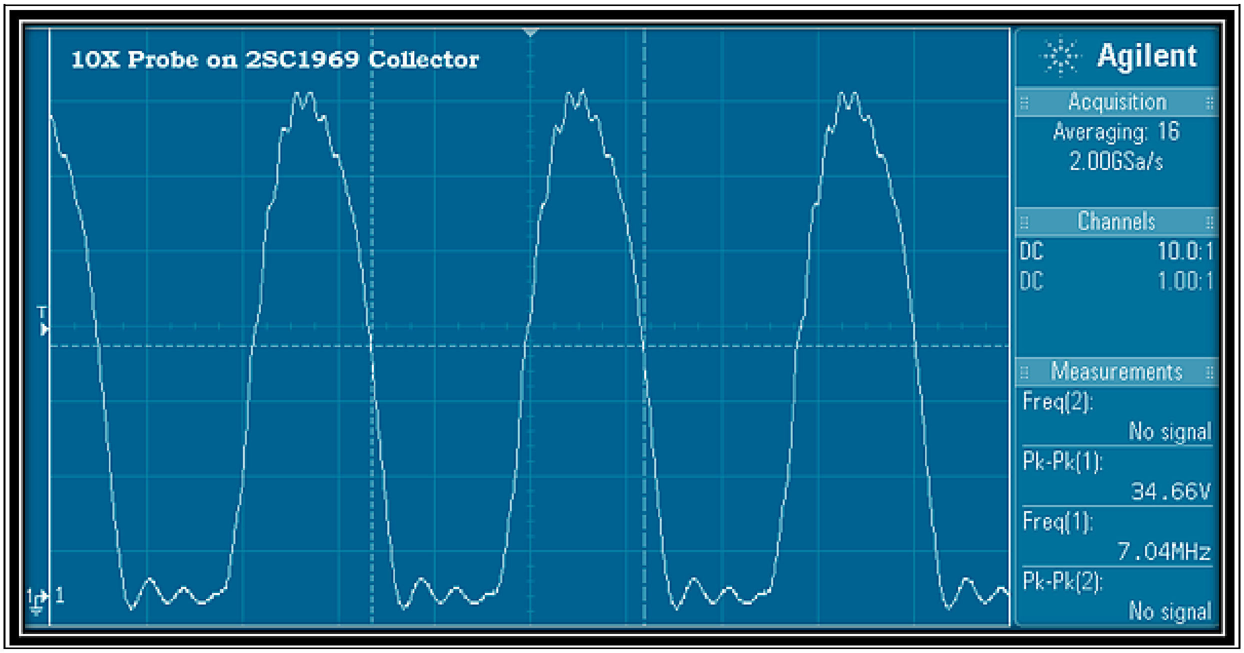
Ask QRP enthusiasts what QRP is all about and they might extol the thrill of exacting the most kilometers per Watt — in keeping with that spirit, I love turning down my Tx power and the Funster front panel RF gain control makes it a breeze.

The PA circuit is standard fodder. I wound the 24.6 µH and 18.6 µH RFCs on FB43-2401 ferrites [a single hole bead that looks like a fat FT37-43]. An FT37-43 or FT50-43 makes a great substitute.

PA Measures



Above — Spectrum analysis of the Tx output. The 10 MHz clock signal from my spectrum analyzer lays between the fundamental and 2nd harmonic. The 2nd harmonic = -53.85 dBc @ 3W power output.



Above — PA collector waveform at 3W output measured with my DSO's 10X probe.

Above — Set your meters on stun! DC measures to calculate PA efficiency [See RF Workbench 6]: DC power = 13.58V x 0.331A = 4.49W. RF power = 3W. Efficiency = RF power / DC power = 3W / 4.49W = 66.8%.

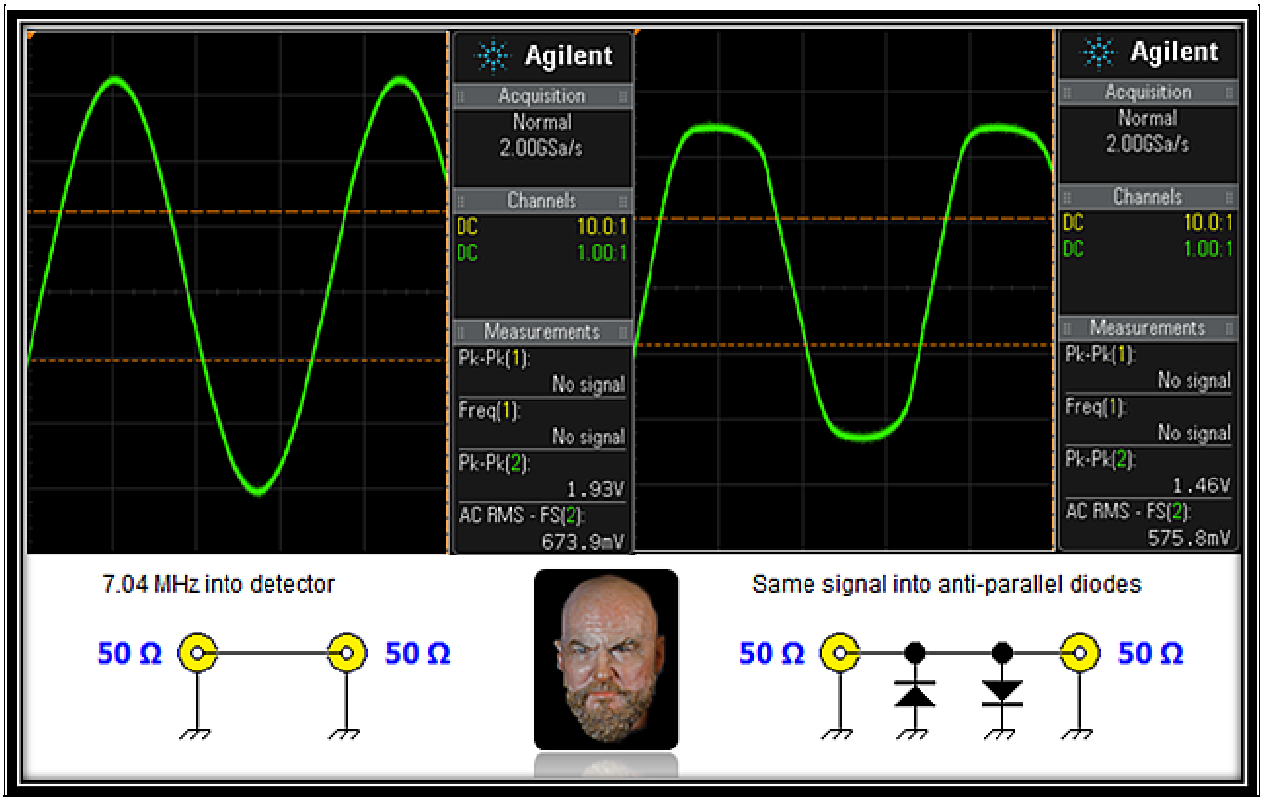
T/R Discussion for Single Band Transceivers

Previous Funsters ran a simple QSK T/R circuit developed by Wes, W7ZOI in the early 1980s: a series X = 500 Ω resonator with anti-parallel diodes between the L and C. The series capacitor presents 500 Ω reactance to the 50 Ω transmit line so that only a small portion of the transmitter power flows to ground via the diodes and even less reaches the receiver input. In short, the 500 Ω capacitive reactance + clamping action of the anti-parallel diodes protect the receiver front end.

For 1 band, this simple circuit sizzles. Two potential concerns:

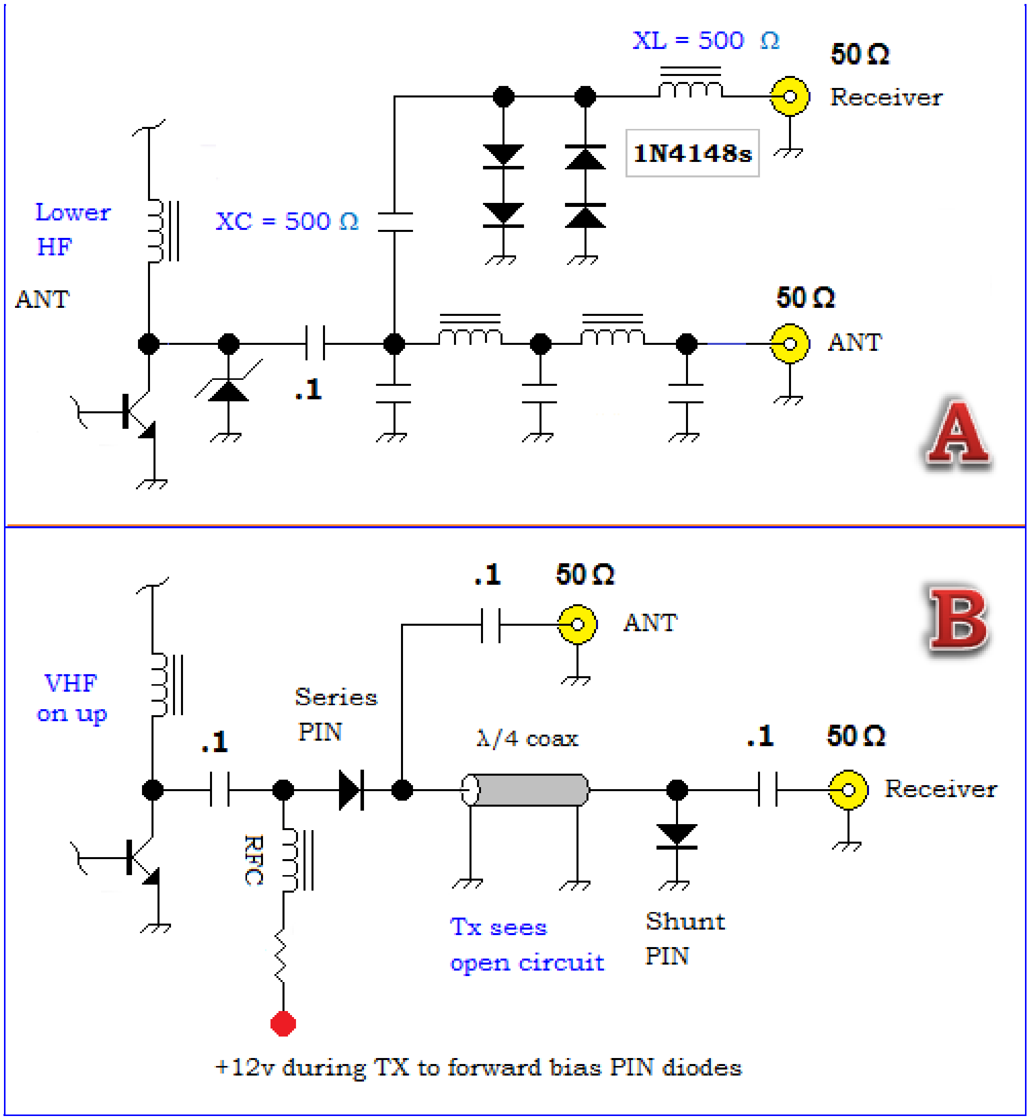
~6-9 dB loss of the receive signal making it better for lower HF, although I've seen builders add a receive preamp and use this T/R system on 12 and 10 meters. Since both band noise and signals get attenuated simultaneously, I've rarely had readability issues on 160-30 meters in my popcorn transceivers.

The anti-parallel diodes may cause IMD. The intercept for the 500 Ohm series reactances with shunt diodes is around +6 or +7 dBm. This might exert problems in a high performance receiver front end, but poses a non-issue with typical NE612 front ends. Many builders, including Wes, reduce IMD potential with 2 or 3 series diodes on each leg. Another solution includes running a PIN diode as the shunt element. That's what I did.



Above — The DSO tracing of an experiment showing the before and after effect of 2 anti-parallel diodes. The voltage across a silicon diode varies with the current going through it. Silicon diodes don't conduct much current until the voltage across them reaches a threshold [typically ~0.6 to 0.7 volts].

In situ, the forward threshold voltage of each diode limits the circuit voltage by clipping off AC voltage above its threshold. Clamping refers to the limiting of voltage in *2* directions using anti-parallel pair(s) of clipping diodes. For example, if a negative-going AC dips more than 1 diode drop below ground, the left hand diode will begin to conduct and clip off any voltage above its threshold. This, of course, describes the "ideal diode" — in real life, factors like switching speed, current-handling and temperature are all in play.



Above — 2 possible T/R schemes. Figure A shows the classic W7ZOI T/R system with 2 diodes per leg. See EMRFD Fig 2.112, for an example with 3 diodes per leg. For 1 band, it's often our go-to circuit.

Figure B shows a scheme well suited for VHF on up: Both PIN diodes get forward biased during transmit mode, however, a 1/4 wave transmission line lays between the series and shunt PIN diode. 1/4 wave transmission lines exert magic [actually it's pure physics that may be verified with a Smith chart]. When the shunt PIN diode gets biased *on* and grounds the receiver port end of the transmission line, the other end looks like an open circuit and thus power flows to the antenna port and not into the forward biased shunt PIN diode to ground. I hope to apply this system in future VHF and UHF projects and sadly, it's impracticable at HF.

Funster Version 8 T/R

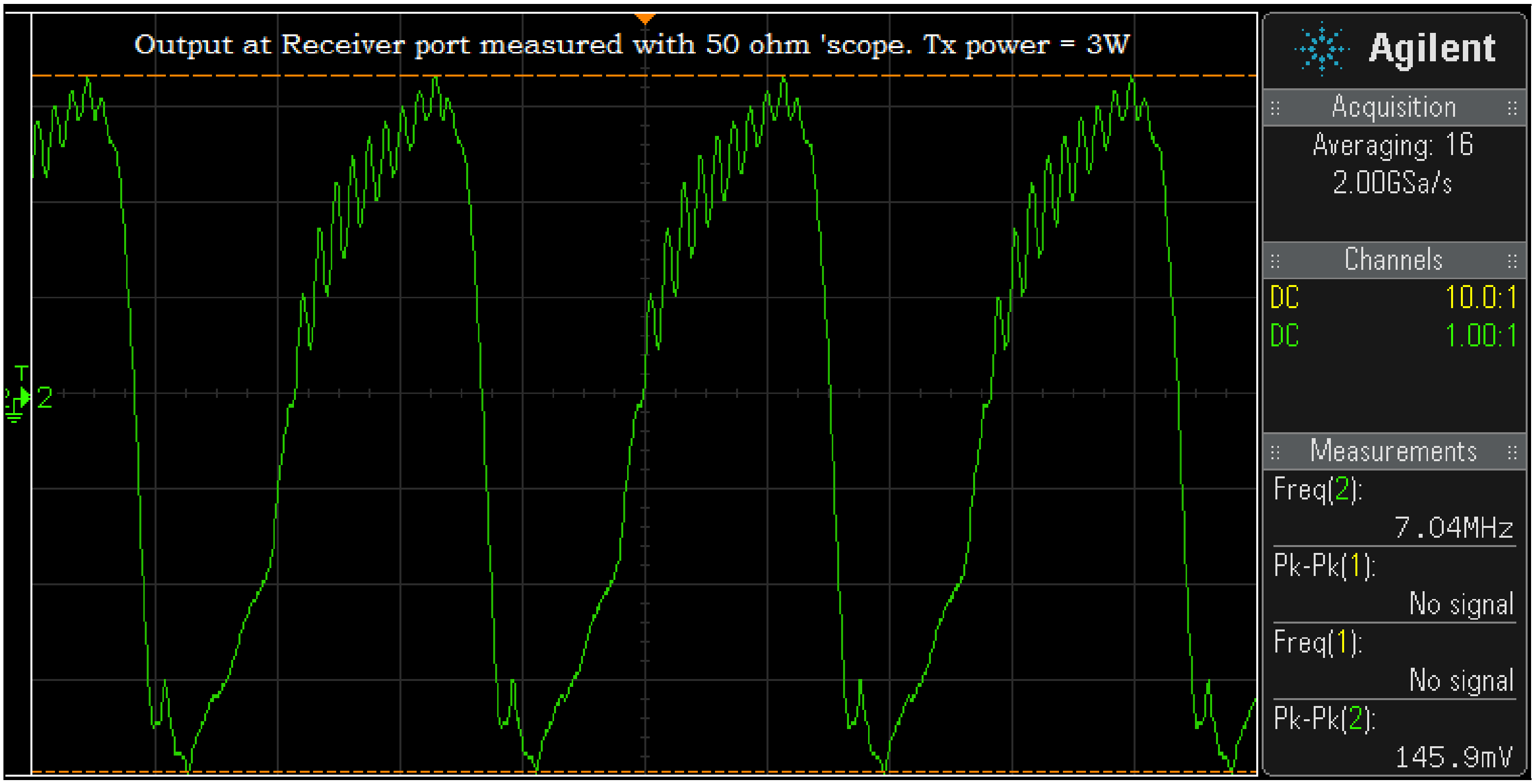
As it happens, I seek experience with PIN diodes and therefore shunted a PIN diode instead of running anti-parallel diodes in the classic W7ZOI T/R scheme. This meant adding a transistor switch to quickly bias on the PIN diode @ *key down* and run a little hang time @ *key up*.

Please refer to the PA-T/R schematic. During key down, keyed 12 volts saturates the 2N4401 switch and 11 ma current biases *on* the PIN diode. At key up, the BJT switch's base voltage is held on by the 10K plus series 0.33 μF capacitor just long enough to allow the stored energy in the large PA collector choke to collapse — protecting the receiver.

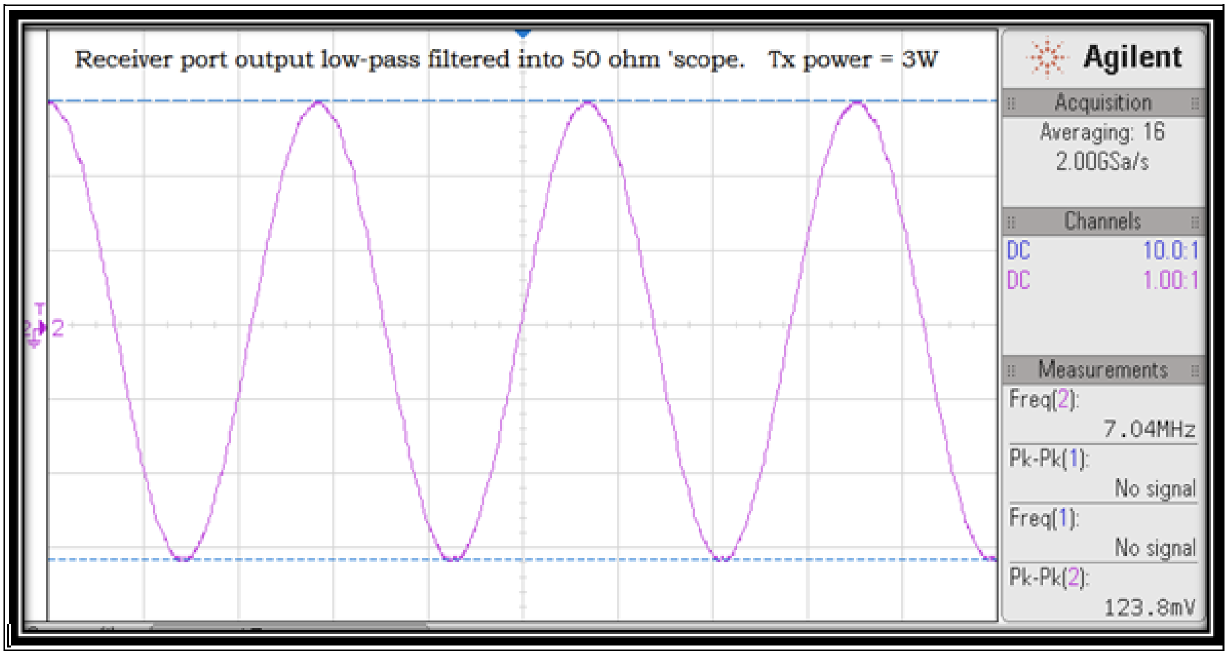
Feel free to experiment with the timing capacitor value. Previously, Wes, W7ZOI suggested connecting the T/R receiver port to a 50 Ω teminated 'scope for general measurement and also for clocking up RC time constants in DC switching circuits. With a 2 channel 'scope you can monitor the receiver port switch function in 1 channel while watching the PA collector AC voltage in the other. Now that's experimenting! Click for a sample from my QRP workbench.

I chose an XC / XL of 400 Ω for the series L-C network. The 9.1 μH coil shouldn't be wound on a 43 material ferrite [the Qu is just too low]: a bigger powdered iron toroid like size 68, or even a 61 ferrite material toroid works okay from my experiments [Q's as high as 162 were measured on a FT50-61 @ 1 MHz in 2013]. Although this coil tunes coarsely, try tweaking the 9.1 μH inductor with thumb and forefinger during actual signal testing in a receiver.

I wound the 125 μH choke connected to the pin diode cathode on a 77 material ferrite toroid to get it done with just a few turns and avoid a self resonant frequency close to 7 MHz.



Above — 50 Ω terminated DSO measurement of the receiver port @ 3W transmitter power. Theoretically all power should shunt to ground through the PIN diode, however, the diode exhibits a little resistance and the series L-C circuit contains reactance, so isolation is never perfect.



Above — A low-pass filtered measure of the receiver port while in transmit mode. The beauty of a 50 Ω measurement environment shines: I placed a 7.36 MHz-bench module low-pass filter between the receive port and my DSO to quantify the leaked power to the receiver port as -14.17 dBm when the transmitter delivers 3W. Safe for my receiver.

My sincere thanks to those who offered advice and feedback during the build of this transmitter.

Above — Funster in the field.

Above — Rear view of the Funster Tx.

---

2. References

T/R References

Perfecting a QSK System. QEX for Mar/Apr 2006. Markus, VE7CA. Click for pdf.

Electronic Switching and How it Works, QST for September 1984. Doug, W1FB

7 MHz receiver -- discussion begins on page 215 -- On page 214 in Fig 1 see the 33 pF input cap plus receiver protecting anti-parallel diodes. Also view the 555 timer and other T/R control circuitry in Figure 2 on page 215. *This receiver realizes the first-ever printed version of the W7ZOI T/R system described earlier.* Solid State Design for the Radio Amateur. ARRL,1986 [out of print] by Wes, W7ZOI and Doug W1FB.

Transmit - Receive Antenna Switching. EMRFD. First published by the ARRL in 2003. Page 6.68.

Schematic for the 30-W, 7 MHz power amplifier. Figure 2.112 [page 2-42] EMRFD. Design by Wes, W7ZOI

RF — Test and Measurement

VHF FM — Supplement 1

This page supplements VHF FM

1. 10.7 MHz IF Strip - with example broadcast band FM receiver
2. NOAA Receiver Experiments @ 162.55 MHz

---

1. 10.7 MHz IF Strip

I'm inexperienced with making FM receivers and lack confidence. I began with a utility module to allow basic assessment of the circuit gain required for different detectors, ceramic filter bandwidths, and more so — to get something working to boost my confidence.

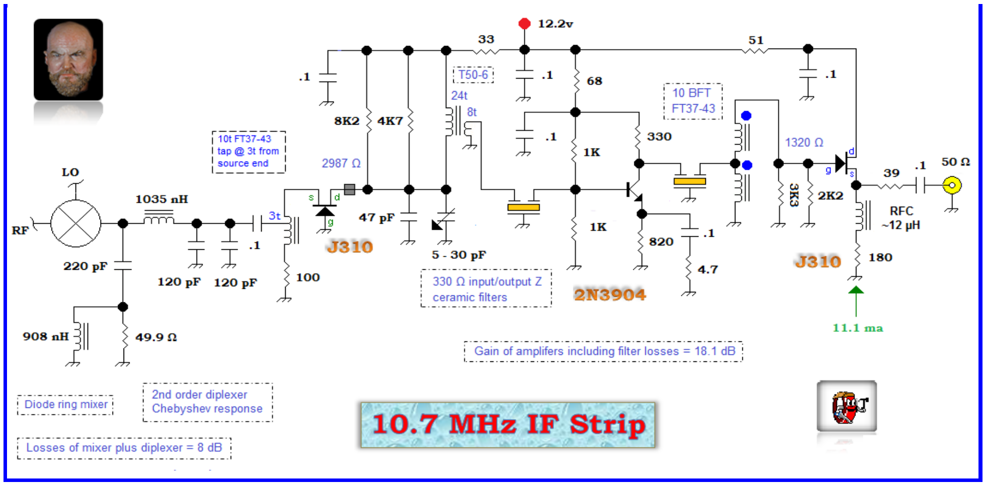
The FM receiver 10.7 MHz IF standard began soon after World War 2 and 10.7 MHz still makes sense for the primitive radio builder — surplus IF filters still seem available in a variety of bandwidths. Although vexxing, if we suffer from receiver images at 10.7 MHz, we can always filter the input better, or jump up the IF and make a crystal filter for say 20 MHz. Above 20 MHz good, inexpensive, fundamental-mode crystals get hard to find.

Mixer

I chose a diode ring mixer; again. After trying other mixers such as balanced FETs, or an NE612 , I kept coming back to the diode ring. Partly I'm biased and partly it's from from measures like this one. Although the oscillators weren't perfectly set for a 10.7 MHz IF, the LO and RF signals are both > 40 dB down from the desired IF — love this.

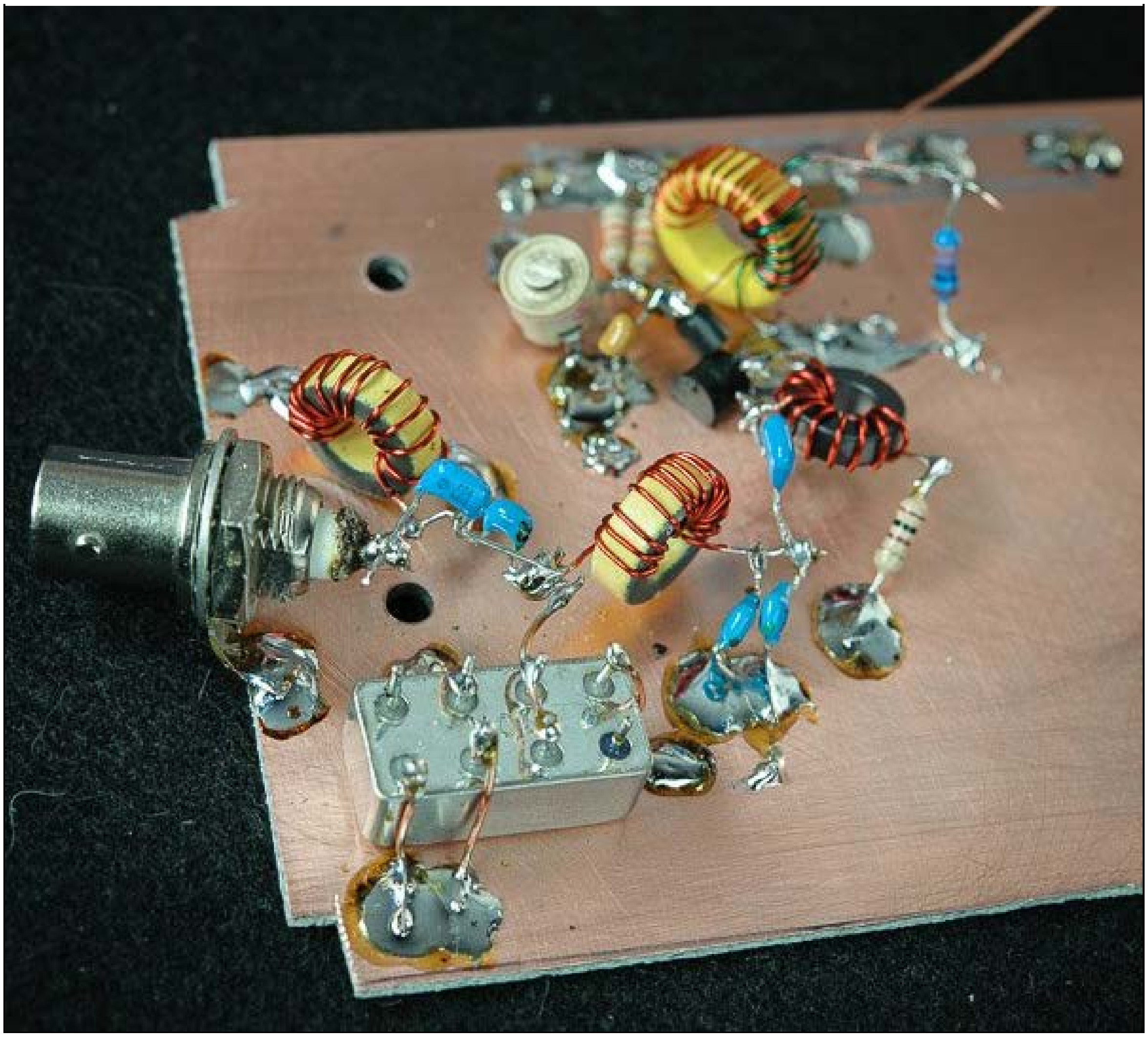
Since I designed a 50 Ω bench module, the IF output contains a source follower to drive the IF port. In an actual "keeper" receiver, you probably wouldn't bother since you might just directly drive the detector, or perhaps another 330 Ω input/output IF amp before the detector.

Although I humbly build simple/primitive circuits, my design goal is never for a *low parts count* — each part serves its purpose. A low parts count ideology plus minimalism exploits the frugal and primitive homebrew nature of our brother hobbyists, however, often gets out of hand. Some builders just cite a low parts count *design goal* to rationalize a poor design. Worse yet, inexperienced minimalists often exclude vital parts such as DC supply decoupling and bypassing, temperature stablizing bias networks, or adequate band-pass filtering. To each his own. Please discern all you see or read in homebrew radio; we're builders, not lemmings.



Above — Click on the IF strip schematic to view it. Most all the stages were discussed on VHF-FM. My net module gain = 10.1 dB. The 2 ceramic IF filters = 230 KHz Muratas with a measured average loss of 6.15 dB each.

Above — The boxed up circuit contains 3 SMA jacks. Click for a photo of the completed breadboard.



Above — A close up of the mixer, diplexer and JFET amp, although the SBL-1 mixer is disconnected. This board contains a temporary BNC jack to allow testing of the non-mixer parts such as JFET gain, diplexer loss, return loss, to confirm the tuning cap choice and also to look for any UHF oscillations eminating from the FET, a UHF part. Click to zoom in on the common gate amplifer with its gate lead soldered as close to the plastic case as possible.

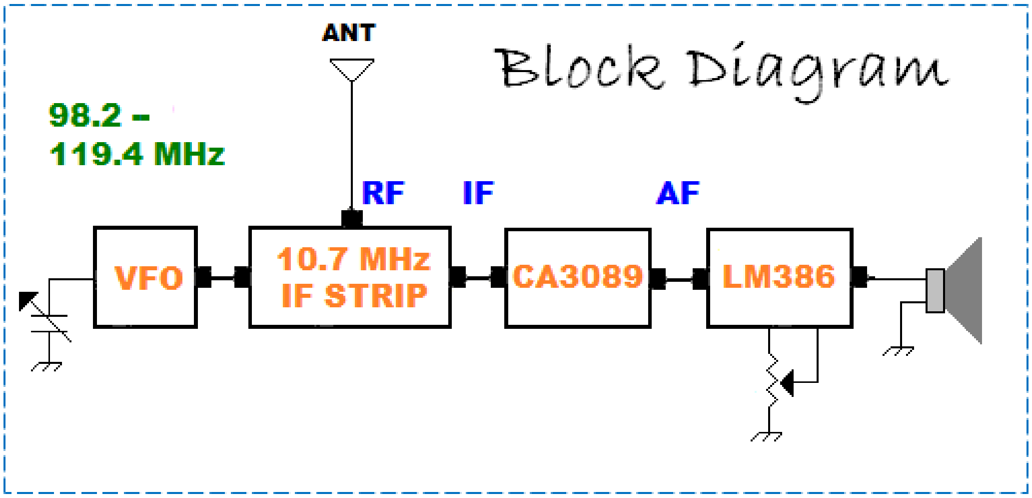
Detector

I tried to homebrew my first IF amp +/- limiter and detectors , but felt unhappy with their performance, failures, circuit complexity and/or noise issues. Click click or click for 3 raw, miserable design examples from Winter 2014. Discouraged; after 2 months of lamenting, I opted for an IC detector subsystem. 3 relic part choices might include the CA3089, SA604A or LA1150.

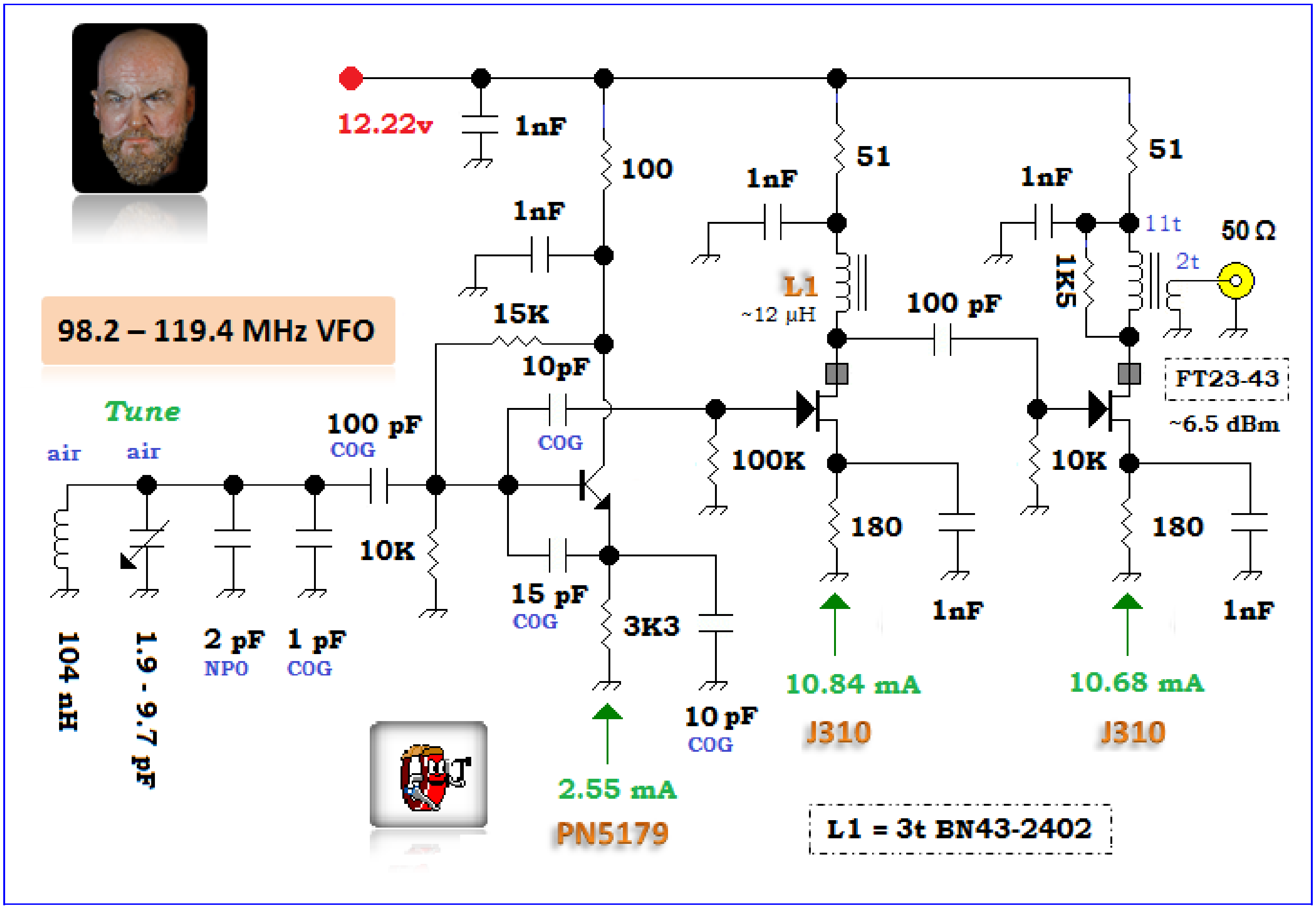
In my case, the CA3089 suited me best and offers a lot: a double-balanced quadrature FM detector with limiter + IF ampifiers; squelch, AFC, AGC, AF preamp and the potential for high-fidelity FM reception. All that's required is an outboard detector resonator — easy to build with a powered iron toroid plus a fixed + trimmer cap.
Price + shipping varies widely, but last year, I got 5 shipped to me from Europe for $7.25 USD and these will make 5 receivers — not bad at all. What hobby isn't expensive?

I've even read that some builders used the CA3089 for an S-meter amplifer, although its logarithmic compliance is far from stellar. You'll need to download the CA3089 datasheet.

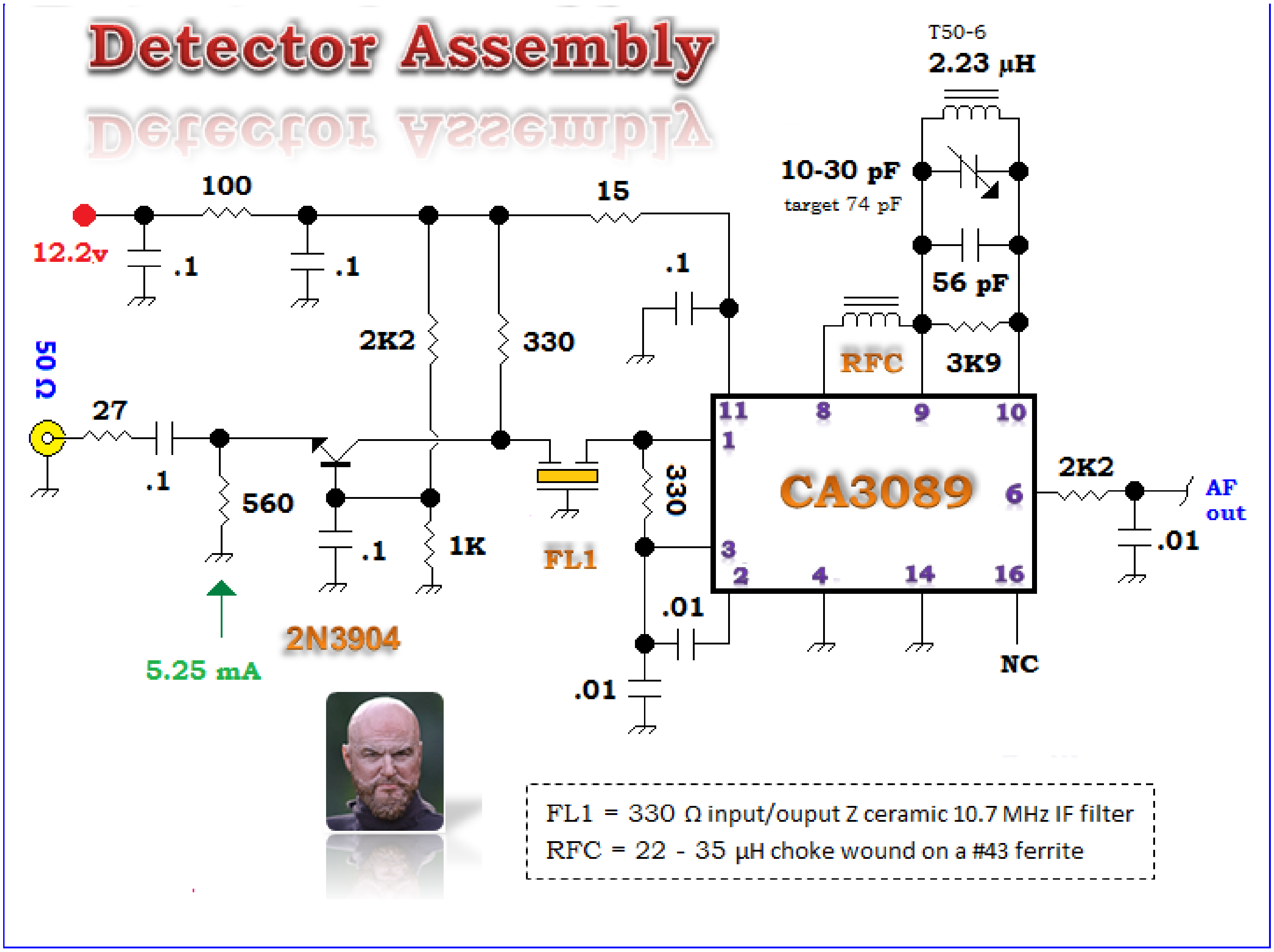
Let's build a receiver:



Above — The block diagram. The antenna = 1 meter piece of wire soldered to a SMA jack that was threaded into a brass sheet to fashion a crude ground plane. No LNA or input band-pass filtering was employed.



Above — The temperature compensated, free running VFO. Normally we should run a VCO-PLL, digital contraption, or a VFO/VCO with ALC feedback, but this thing worked fine to learn on.



FL1 = 330 Ω input/ouput Z ceramic 10.7 MHz IF filter
RFC = 22 - 35 µH choke wound on a #43 ferrite

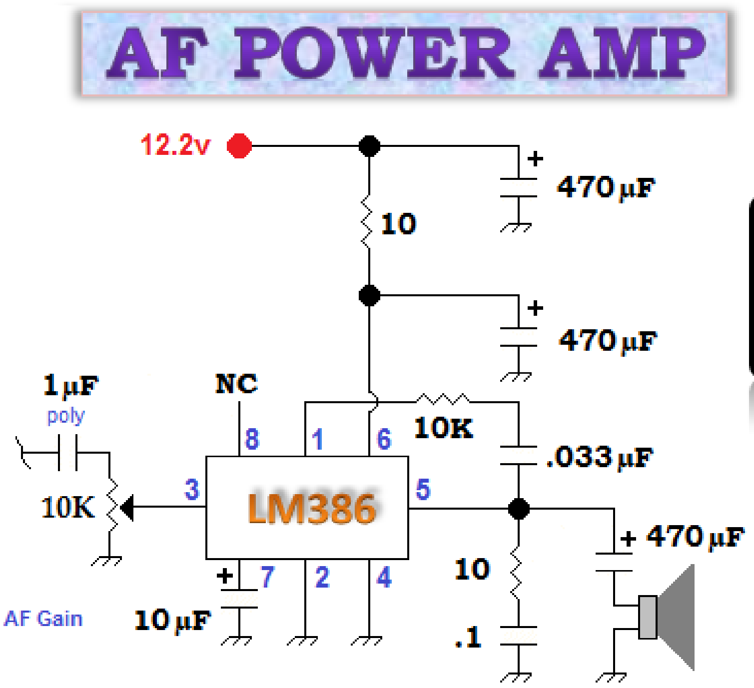
Above — The detector assembly. To match to the 50 Ω IF strip port, I employed a common base amp with a 330 Ω collector resistor driving a final 230 KHz wide "tail end or mop up" filter. In many designs you could simply drop this amp as the gain isn't really needed. On the other hand, the CA3089 contains a limiter and it won't hurt at all.

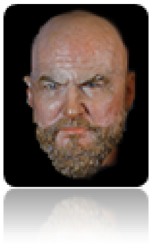
To make the detector coil, a small square island was carved on the copper clad board. The island was bisected so that pin 9 soldered to 1/2 the island and pin 10 to the other half. The 3K9 R, 56 pF C and a small, high Q SMT trimmer capacitor bridged the 2 islands. Finally, a 2.23 µH, T50-6 toroidal inductor was soldered across the 2 islands.

Tune a station and peak the trimmer cap for the most crisp sounding audio — it's set and forget. When driven with enough signal for full quieting, the CA3089 delivers very low distortion — the FM receiver in my truck sounds worse.

An RFC provides a DC short across pin 8 to 9 and keep ~5.5 VDC bias on the pin 9 transistor base. The minimum inductance should be 10X the detector coil. I wound my RFC with 6 turns on a BN43-2402 [35 µH] , but any old #43 ferrite wound choke >= 22 µH should work. For my chokes, I prefer a low # of turns on a binocular core to avoid any SRF issues.

Since the CA3089 contains a powerful audo preamplifer, only an AF power amplifer with low gain is required. I opted with the old LM386, but in future, better, "keeper rigs" I'll add an op-amp tone control stack driving a pair of power followers for good measure.





Above — Final audio power amp: an LM386 set for a voltage gain of 20 with some low-pass filtering and stout DC line filtering.

Click for a photo of the detector and LM386 assembly. Click for a better view of the outboard resonator and nearby RFC.

Outcomes

After connecting the VFO, antenna, IF strip, detector assemby, speaker and powering them up, loud FM radio blared through the loudspeaker. I tuned every local FM station with full quieting and felt surprised that despite no band-pass filtering, plus no RF preamp, it worked and sounded so well.

Further experiments showed that inserting as much as 15 dB attenuation between the IF stip and the detector assembly still kept full quieting on all the stronger stations. In some cities with multiple, strong FM stations, input band-pass filters will be needed. I'll cover that in a later installment.

I connected this VFO to the IF strip and tuned in my local NOAA weather station at 162.55 MHz. Click for a photo of the VFO built on a piece of bent copper sheet metal.

Future experiments may include changing ceramic filter bandwidth, trying different mixers, applying varactor-tuned input band-pass filters for strong FM environments and for FM Dxing. Finally, I'll design a low-noise preamp with a common gate JFET and/or a dual-gate MOSFET. Utimately, I hope to make a dedicated NOAA weather receiver, a broadcast FM Dx receiver and then some narrow band FM jobs for 2M or 6M. The future looks good.

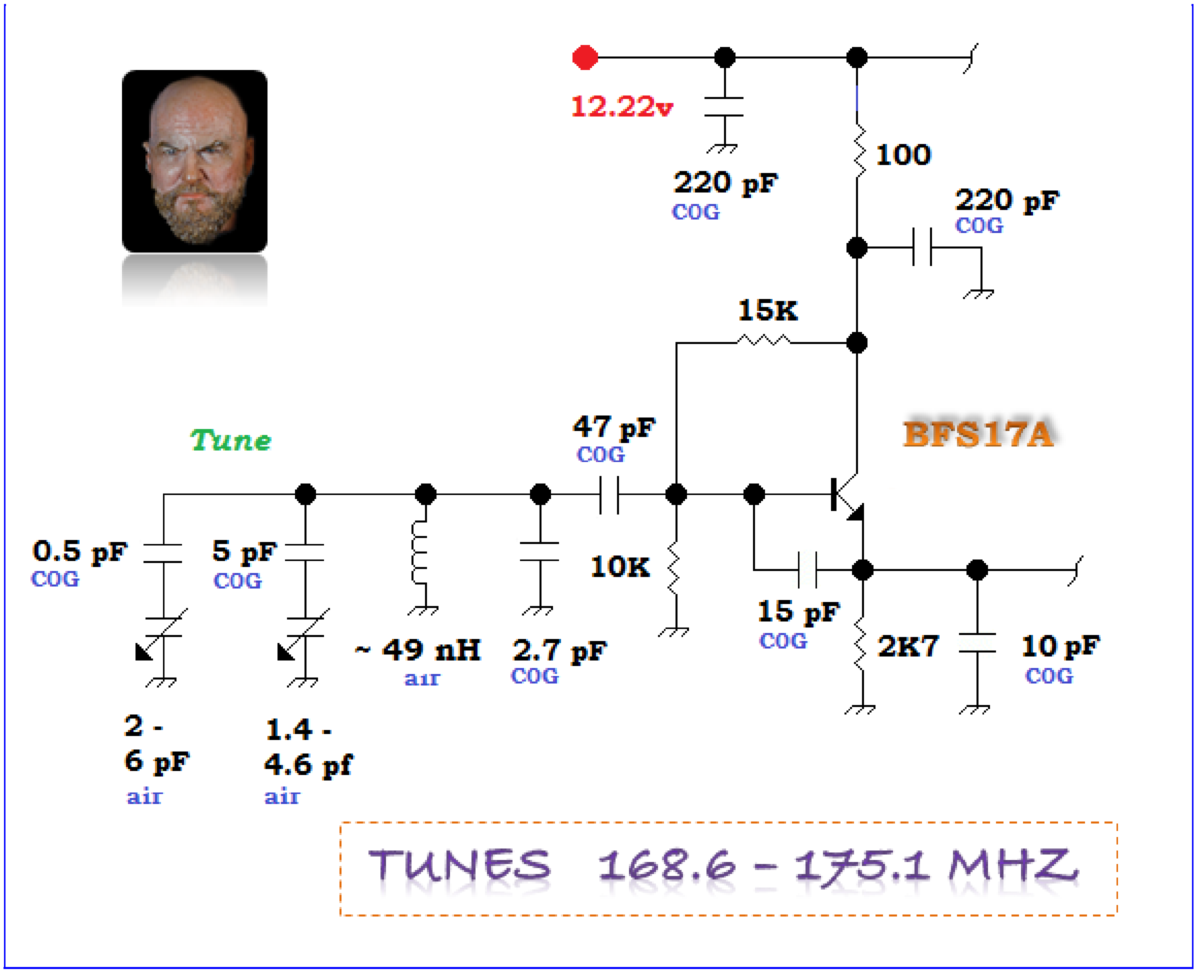
Audio sample one : a few local stations [mostly recorded before I added the negative feedback on the LM386]
Audio sample two: a weak signal [our provincial Highway Report] that doesn't reach full quieting.

Thanks.

---

2. NOAA Receiver Experiments @ 162.55 MHz

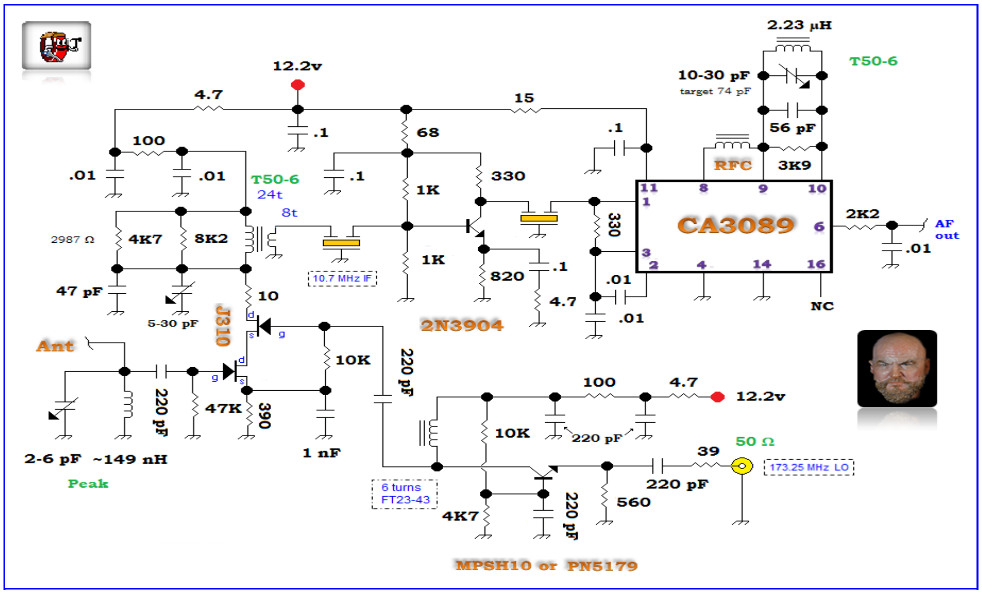
I document some *raw* experiments and hopefully 1 day will make a good receiver for my local NOAA FM broadcast.



Above — I built a VFO for 173.25 MHz with fine tuning for my local, NOAA weather report receiver @ 162.55 MHz. The fine tuning 2- 6 pF variable capacitor in series with an ATX 0.5 pF chip cap resolves 10's of kilohertz to permit tuning narrow band signals. You'll especially need fine resolution tuning when applying ceramic IF filters <= 150 KHz.

Above — The 2 VFO air variable tuning capacitors. Click for the circuit breadboard with a tuned, common emitter buffer, that I'd rather not show [because it sucks]. I ran the VFO into a bench module to boost the power up between 5 and 10 dBm in my experiments.

Since @ VHF, my VFO went on 2-sided FR4 board with several wire vias joining the 2 surfaces. We normally avoid double sided boards for VFOs since doing so makes a low Q, potentially moisture absorbant, undefined temperature coefficient "capacitor" that may worsen temperature drift. For bench learning and in particular, FM reception, temperature stability tested adequate and I can listen for hours without retuning.



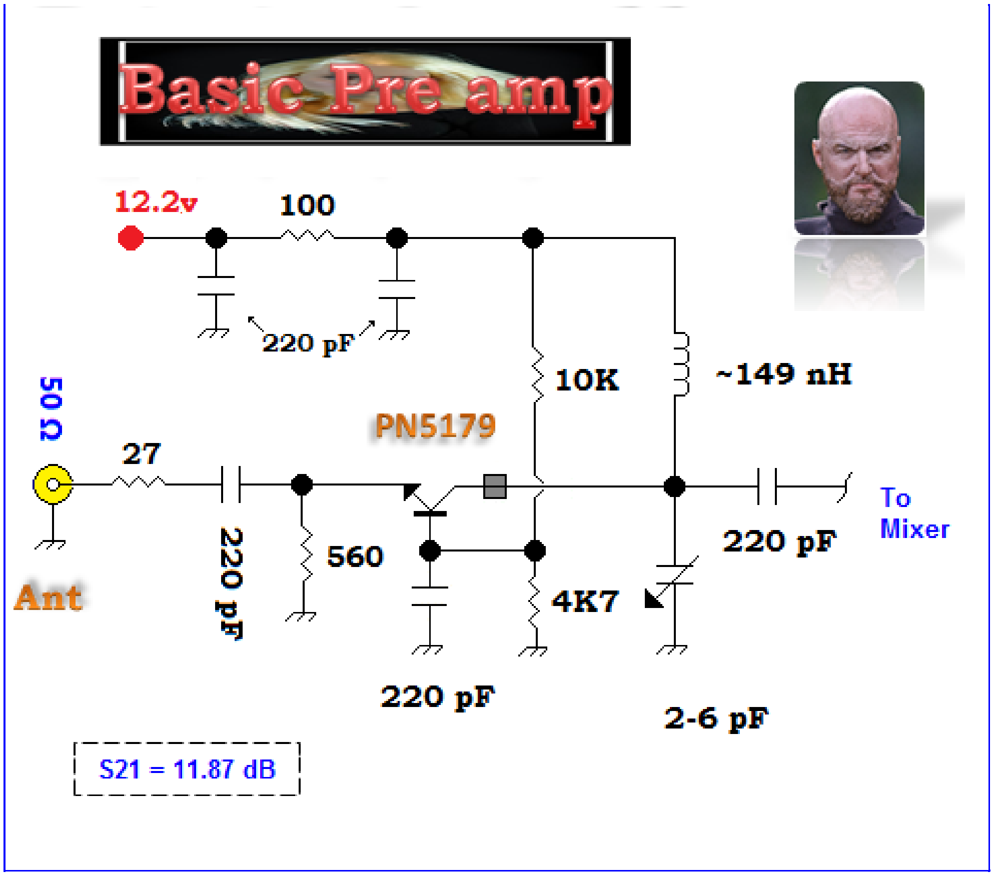
Above — The entire receiver with a cascode mixer from EMRFD. In experimenter's mode; I tried out many ideas. For example: different bandwidth IF filters [settled on 230 KHz] , 1 versus 2 ceramic filters, or eliminating the 2N3904 amp [2 better drove the CA3089]. One salient learning — IF filters may possess a wide variation in center frequency and thus if you have a sweeper, matching them might be useful. In this receiver topology, you don't need to match 230 KHz or 280 KHz ceramic filters; the mismatch became noticeable with a <= 150 KHz 3 dB filter bandwidth.

I boosted the VFO signal with a common base amp — I varied the power into this LO amplifier by applying different strength attenuator pads. Between 1.5 to 2 Vpp on the mixer gate gave a reasonable compromise of conversion gain, noise and easy-to-get VFO power. From EMRFD, maximum conversion gain occurs when driven with 5 Vpp.

This unbalanced mixer worked fine, however, without a metal case, I heard a little hum from my DC supply and noises from household appliances such as the clothes washer and fridge. This receiver suffered from microphonics as well. The AF amp was a LM386 set for gain = 50.

A simple LC resonator tuned the mixer RF input and proved essential. I tried different ways to couple a 25 cm long wire antenna such as tapping the coil and an L- network, however, my main antenna peeve was severe hand capacitance effects. Although this receiver does not require a preamp for RF gain, the isolation seems needed. If you put your hand near the telescopic whip antenna of my Realistic Pro 2020 scanner, no noise or RF disturbance arises.

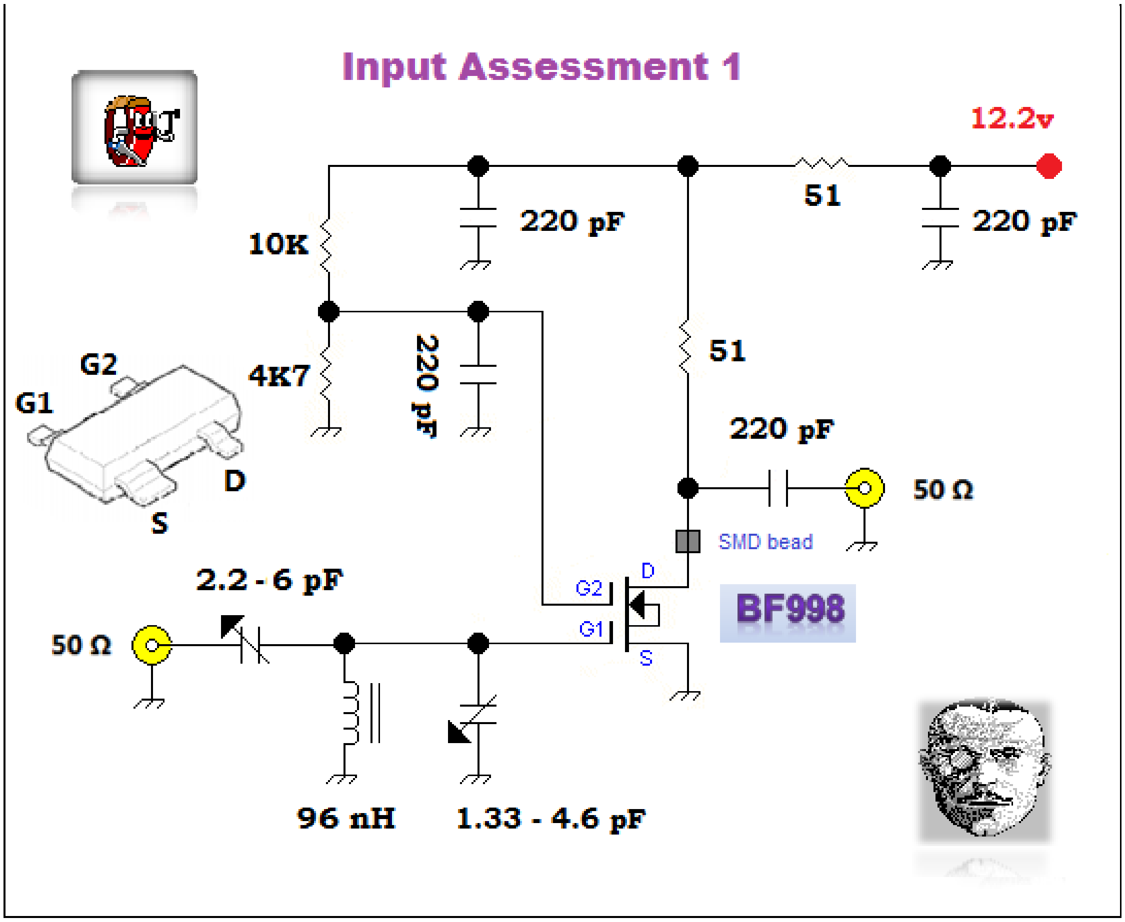
From studying scanner and FM receiver schematics, most rigs contain a dual-gate MOSFET preamp for a low noise figure and high input isolation. These MOSFET amplifiers feature input and output tuning, although with slug-tuned inductor cans and the oh-so-typical varactors, the resonator Q runs low.



Above — I quickly built a tuned-output common base amp to study front end isolation. I removed the existing resonator and connected this to the JFET mixer gate. It eliminated antenna hand capacitance effects. Click for a sound byte of the receiver with this preamplifier and a 20 cm wire antenna attached to its input. [206 KB download]. I reckon some of my receiver problems arose from the mess of unshielded boards, modules and wire on my bench.

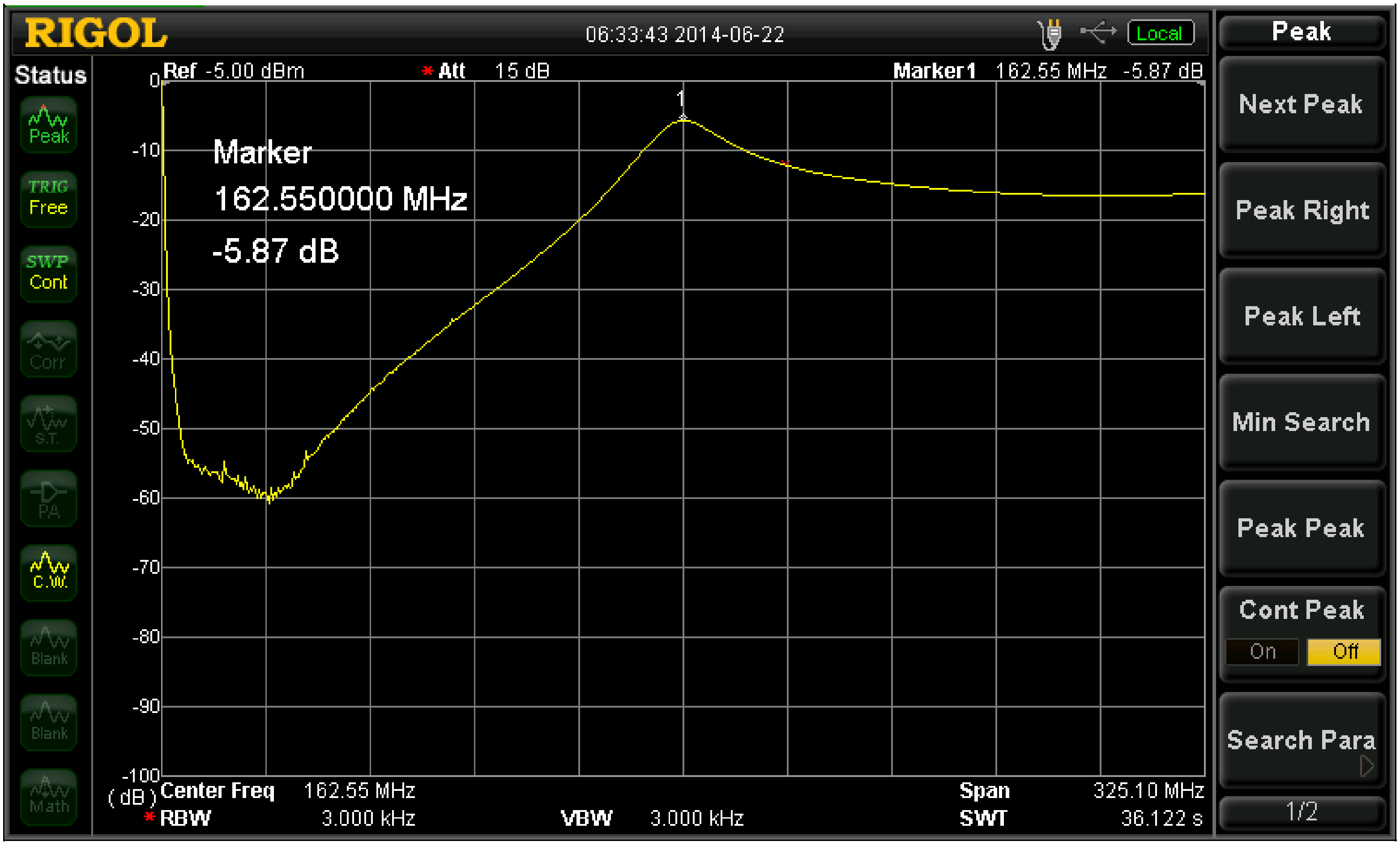
Tuned Input with a 2-Gate MOSFET [162.55 MHz]

A dual gate MOSFET preamp concluded this day's experiments. Tests of a simple input network follow:



Above — Input Assessment 1. To examine the input without any output network effects, I loaded the BF998 drain with a 51 Ω resistor and matched the input parallel resonator for the greatest input return loss [-S21] by tweaking the series SMD trimmer capacitor. After setting the best input return loss [21 dB], I swept this amp.

The XL of the 96 nH air wound coil = 98 Ω and the XL and XC of this tank may be scaled to other frequencies by keeping the reactance values shown or choosing your own. For example; a coil for 50 MHz where XL = 98 Ω = ~312 nH. I discussed and swept the ferrite bead soldered on the FET drain in Section 1 of the VHF Veronica web page.

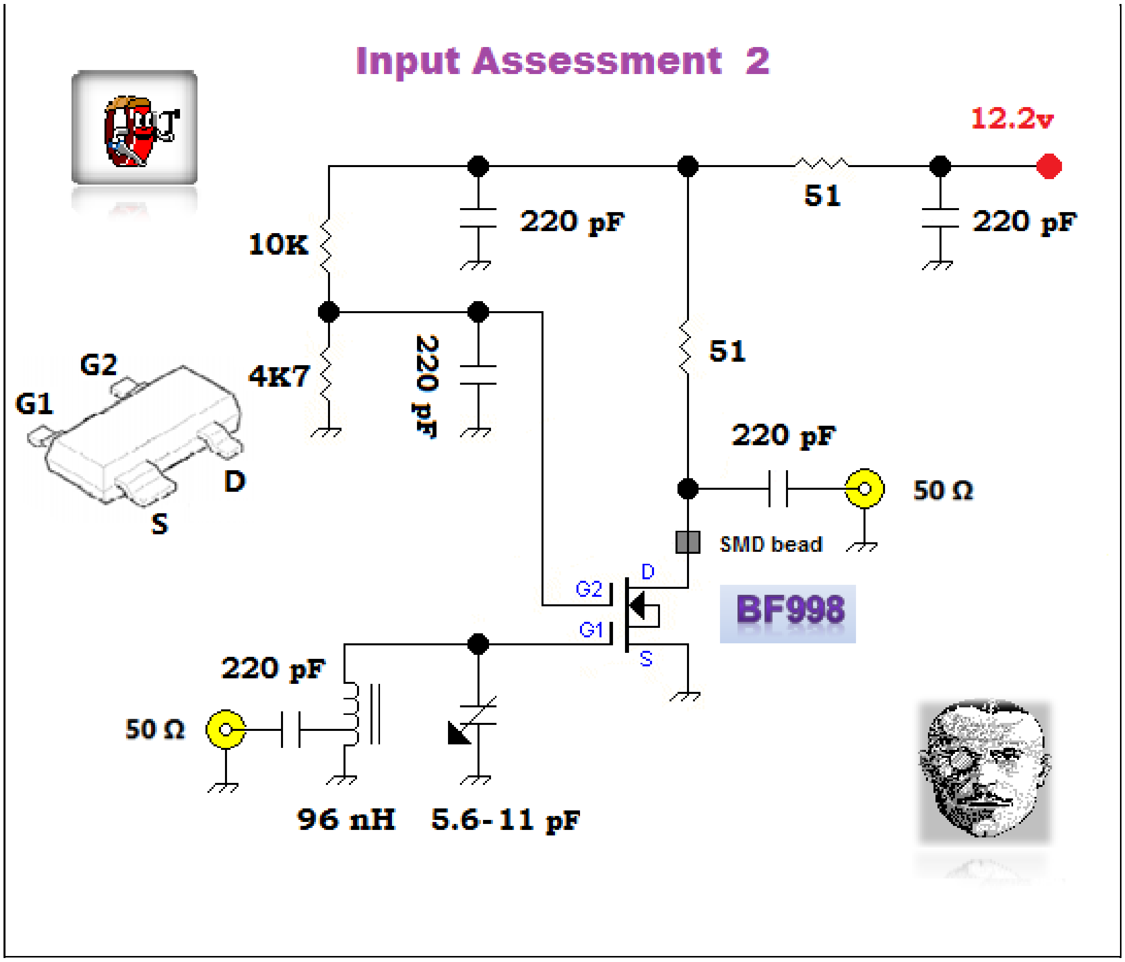

06:33:43 2014-06-22

Ref -5.00 dBm
Att 15 dB
Marker1 162.55 MHz -5.87 dB

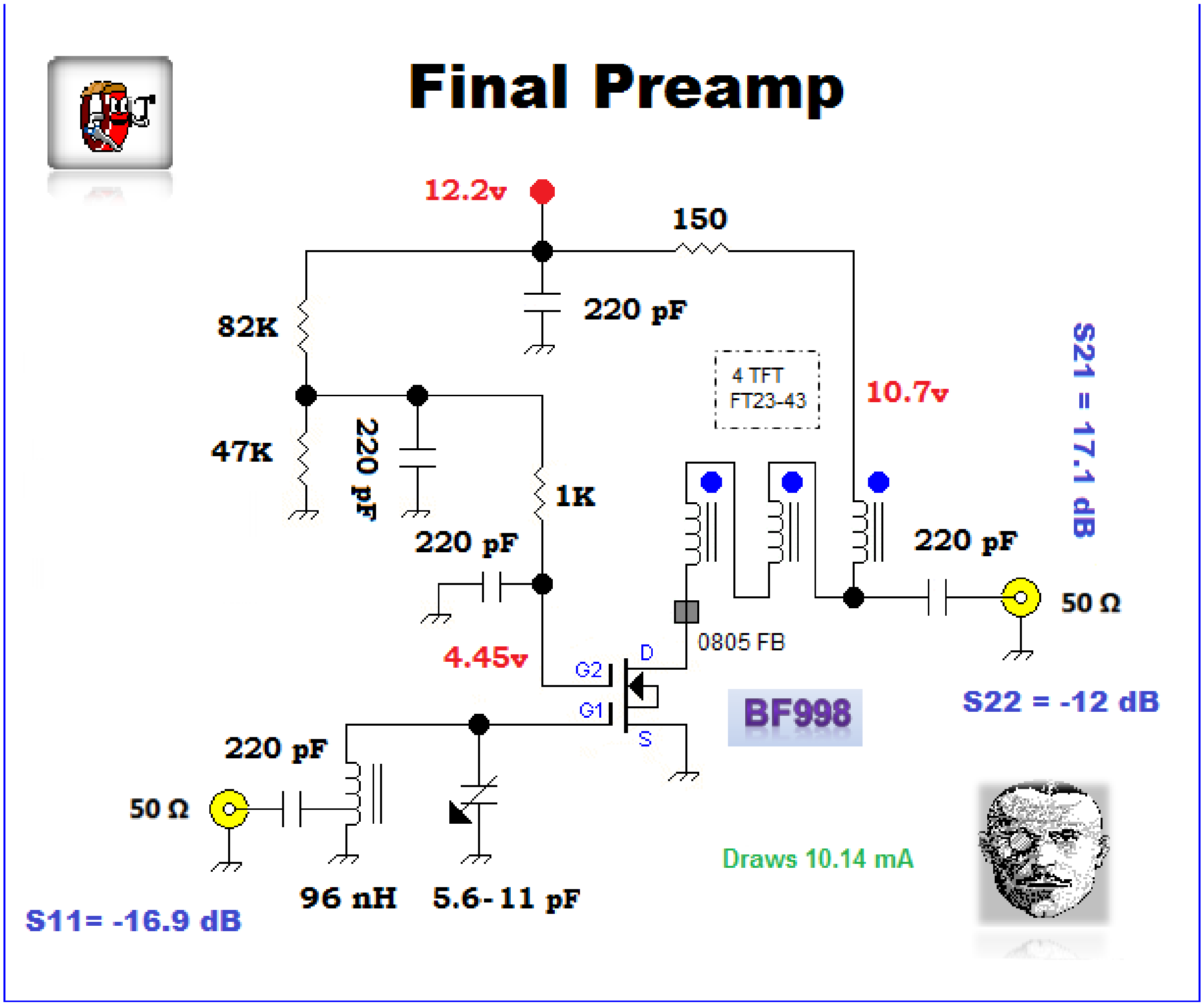
Marker
162.550000 MHz
-5.87 dB

Center Freq 162.55 MHz
Span 325.10 MHz
RBW 3.000 kHz
VBW 3.000 kHz
SWT 36.122 s

Above — A sweep of the Input Assessment 1 amplifer. Although easy peasy to adjust, series matching the resonator to its input load results in a slipshod right-hand [low-pass] skirt. Click for a screen shot showing the amp's 3 dB bandwidth.



Above — Input Assessment 2. After measuring a poor low-pass skirt in Input Assessment # 1, I opted for the conventional tapped transformer to steepen the low-pass skirt response versus frequency. My sweeps *did* show a better low-pass skirt shape . Finally, I built a "real" amp and soldered in a broadband output network, tweaked the bias scheme + limited the VDD with a series 150 Ω resistor. A higher value input resonator trimmer cap is required for this amplifier versus that in Input Assessment # 1.



Above — The final 162.55 MHz preamp schematic + some measures. Click for a breadboard photo. Click for the gain measurement with a TG & SA. I measured the same S21 [within 0.3 dB] using my VHF signal generator plus my 50 Ω input DSO. I didn't push too hard with the gate bias or VDD, since neither maximal gain nor parasitic oscillations were sought. Click for a zoom of the filter peak response. Click for a wider sweep to see the right-hand skirt response.

Built on 2-sided FR4 with short cut paths to fit mostly size 0805 SMD parts, I placed via wires on every ground point and detected 0 unwanted oscillations from HF to 3 GHz.

A bare copper wire coil allows us to set the input impedance tap point anywhere along the wire — unlike the *integer only* tap points provided by magnetic toroid-based transformers. In LNA design, many builders set the input tap point for the lowest NF with the amp in noise figure test mode. Another vanilla flavored option is to set the tap with the amplifier input connected to a return loss bridge while adjusting the tap point for the best input return loss — that's what I did. My S11 = - 16.9 dB with my final tap around 1.2 turns of the total 7 links on my 96 nH coil.

The output transformer presents a poor match to 50 Ω with a S22 of just -12 dB. I plan to put a 4 dB pad after it for my NOAA receiver, so it's okay. Some might chase a better output return loss and also perhaps tune the output for better band-pass filtration.

Your FM receiver preamp might need better input filtration, however, if a simple design works, then simple rules! Have fun with *your own* experiments.


473
270
472

RF — Test and Measurement

Pin Outs

Mini-Circuits Labs SBL-1 and TUF-1 diode ring mixers

...more later

Above — Normal set-up: **Ground** -- Pins 2, 5, 6 and 7. **LO** -- Pin 8, **IF** -- Pins 3 and 4, **RF** -- Pin 1

Above — **RF** -- Pin 1, I**F** -- Pin 2, **Ground** -- Pin 3, **LO** -- Pin 4.

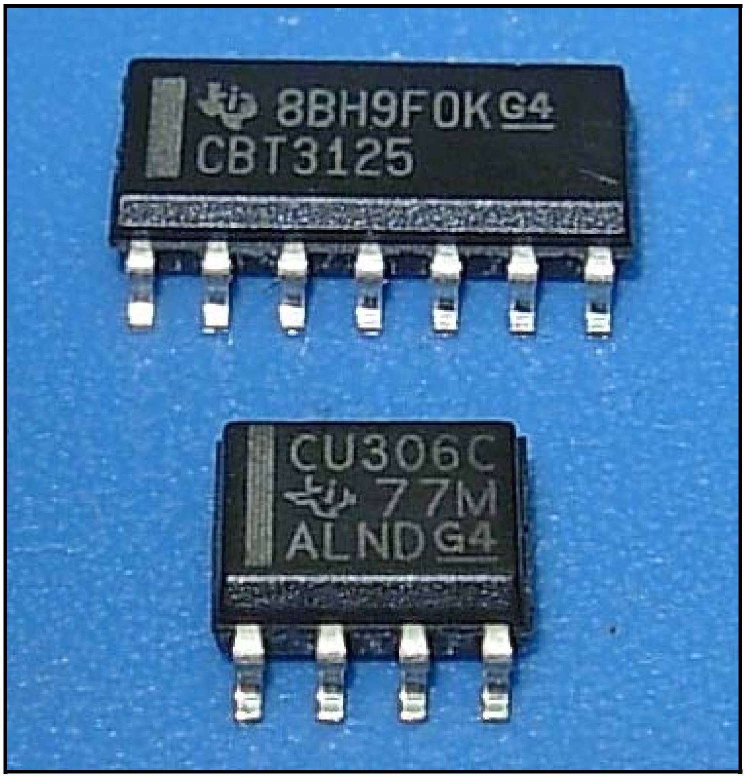
VHF-FM Supplement — Amplifiers

Deleted October 26, 2013

Amateur Radio Electronic Design

KL7R Memorial Receiver Experiments

Introduction



This web page is a memorial to Mike Caughran, KL7R, who died suddenly in January 2007. Mike was a passionate experimenter who was embraced by the homebrew radio electronics community. He was best known as the co-creator of and sidekick to Bill Meara, M0HBR on the podcast Solder Smoke. Mike was a hardcore science and technology buff. His knowledge of general science and curiosity about minimalist RF designs was amazing. I worked Mike on 40 and 80 meter CW and later by voice on eQSO. Mike was the first person to refer to this web site as the "popcorn site". He held an interest in digital circuits. In tribute to Mike, a series of receiver experiments which includes some digital circuits are presented. Mike Caughran will be remembered as a remarkable, kind and passionate homebuilder.

Navigation

There are 3 linked web pages associated with this KL7R tribute web page.

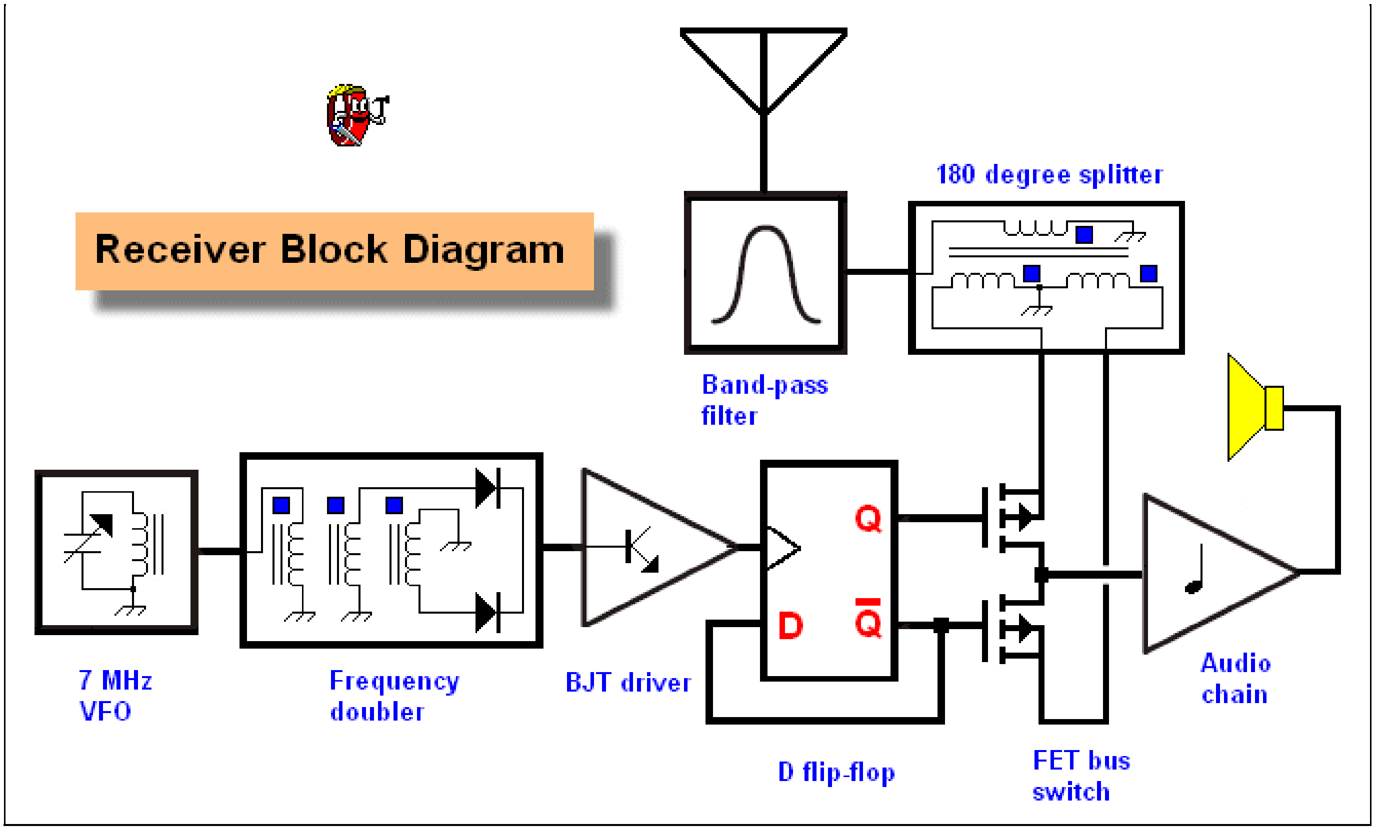
1. Mike's personal web page His own web page memorializes him best

2. Supplemental web page Additional schematics and photos which supplement this main web page

3. VFO web page Describes the VFOs used for these experiments

My special thanks to Wes, W7ZOI for his coaching and suggestions to improve many of the circuits on this web page. This web page borrows heavily from his designs as presented in EMRFD.

Receiver Description



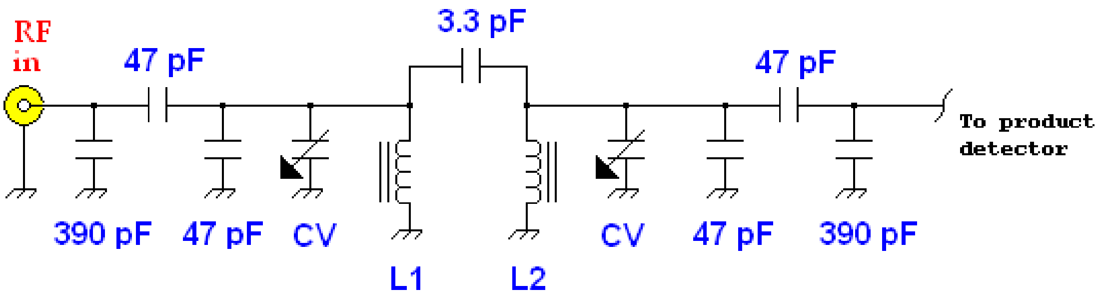
Shown above is the receiver block diagram. The KL7R memorial receiver depicted on this web page is the final output of many hours of experimentation. Most of the circuits or circuit ideas originated in EMRFD.

From our conversations, Mike was always challenging himself; experimenting, testing and pushing his knowledge threshold. The joy of discovery motivated him. Fired by his spirit of inquiry, I explored methods to build a receiver containing at least 1 digital circuit. On many days, I accomplished nothing. The circuits did not work and little to no progress was made. These were the difficult dry spells all experimenters must endure. Design and circuit failures can be very disheartening. I also wasted a lot of parts. However, I kept going and slowly successes occurred and my confidence rose. The end result was a little more knowledge and a cool, popcorn, direct conversion receiver which I hope will provide ideas and inspiration for your own experiments.

Double Tuned Band-pass Filter

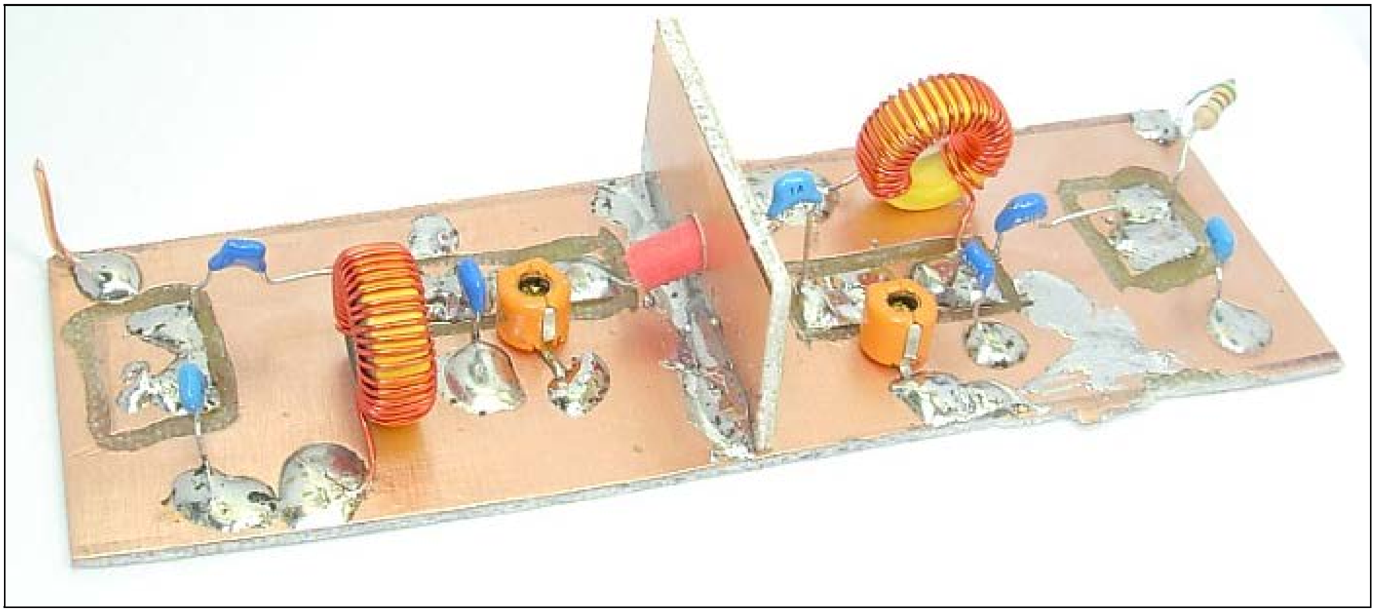
Figure 1

40 Meter Band-pass Filter

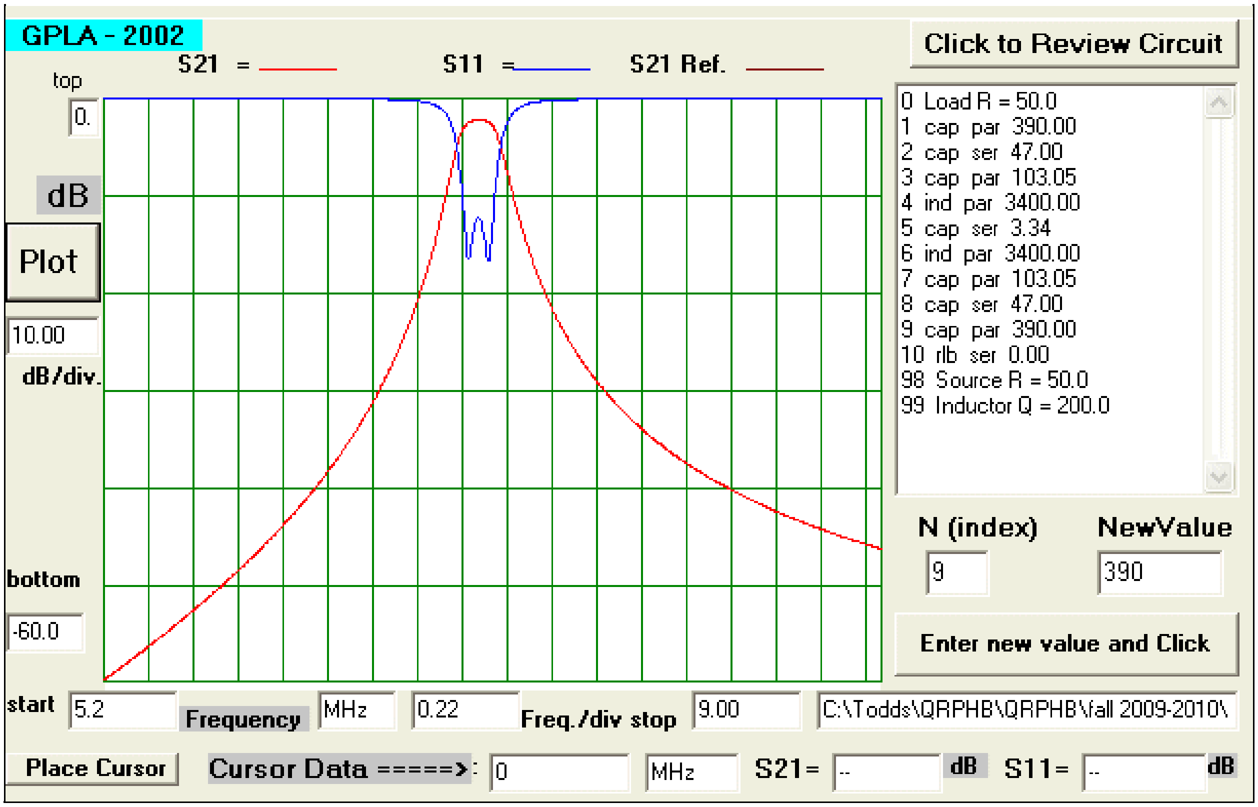


CV = 6.0 - 50 pF trimmer cap
L1 = L2 = 3.4 uH = 29 t #24 AWG on a T50-6 toroid
Input and output impedance = 50 ohms
-3 dB bandwidth = .220 MHz
Insertion loss = 2.3 dB

Figure 1 shows the front end band-pass filter. If you can't obtain a 3.3 pF coupling capacitor, try this other 7 MHz band-pass filter circuit or perhaps just design your own. See the Webmaster's page for information concerning many of the parts used on this web page.



Shown above is a photograph of the Figure 1 breadboard. The inductors were spaced apart at right angles to reduce unwanted coupling. The copper clad board L-C tank divider is not necessary. The 51 ohm load resistor seen to the right was removed after testing.



top 0.

10.00 dB/div.

bottom -60.0

start 5.2 MHz

Freq./div 0.22

stop 9.00

C:\Todds\QRPHB\QRPHB\fall 2009-2010\

0 Load R = 50.0
1 cap par 390.00
2 cap ser 47.00
3 cap par 103.05
4 ind par 3400.00
5 cap ser 3.34
6 ind par 3400.00
7 cap par 103.05
8 cap ser 47.00
9 cap par 390.00
10 rlb ser 0.00
98 Source R = 50.0
99 Inductor Q = 200.0

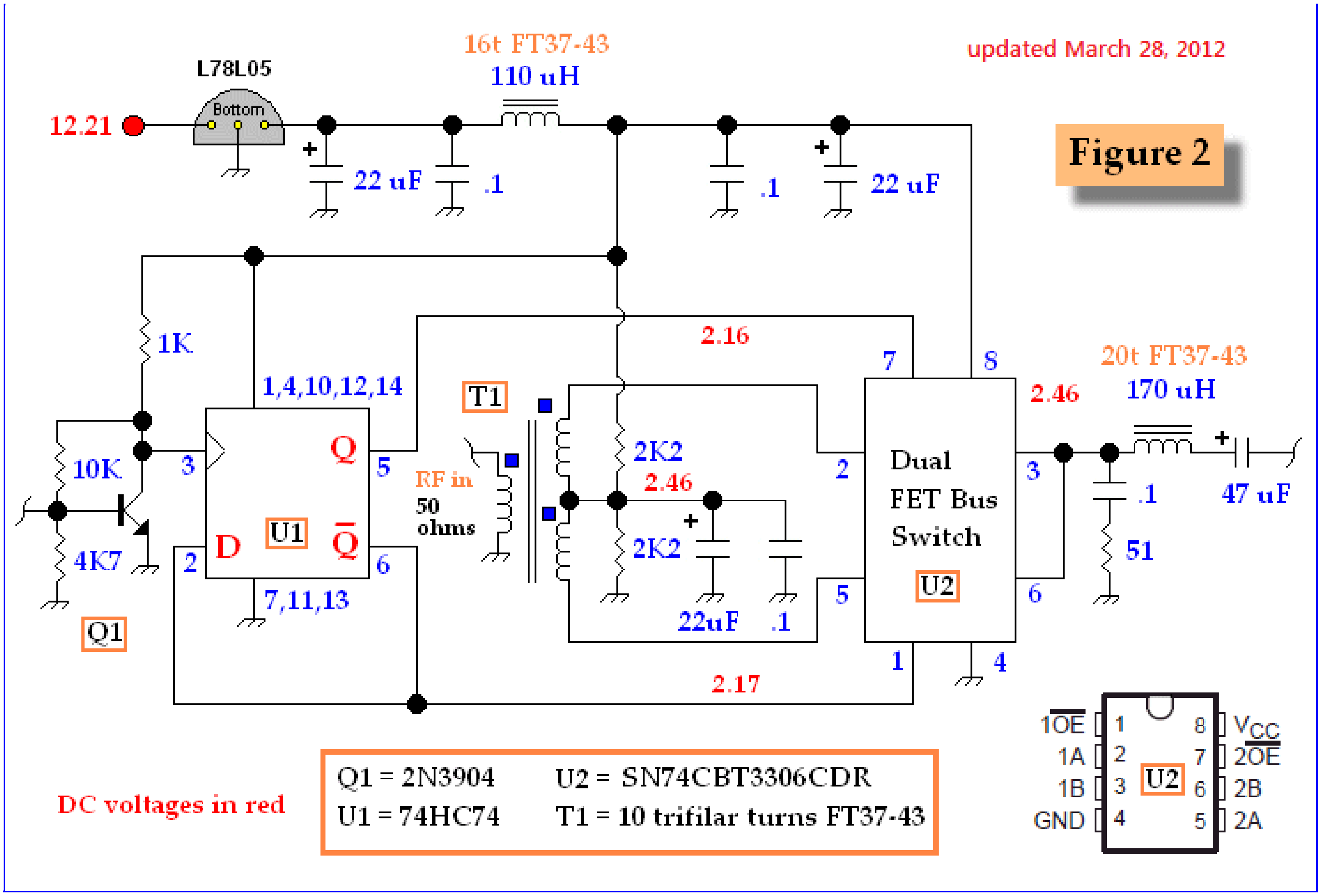
N (index) 9

NewValue 390

Cursor Data =====>: 0 MHz S21= -- dB S11= -- dB

Shown above is a GLPA simulation of the Figure 1 band-pass filter. Хорошо.

Product Detector Experiments

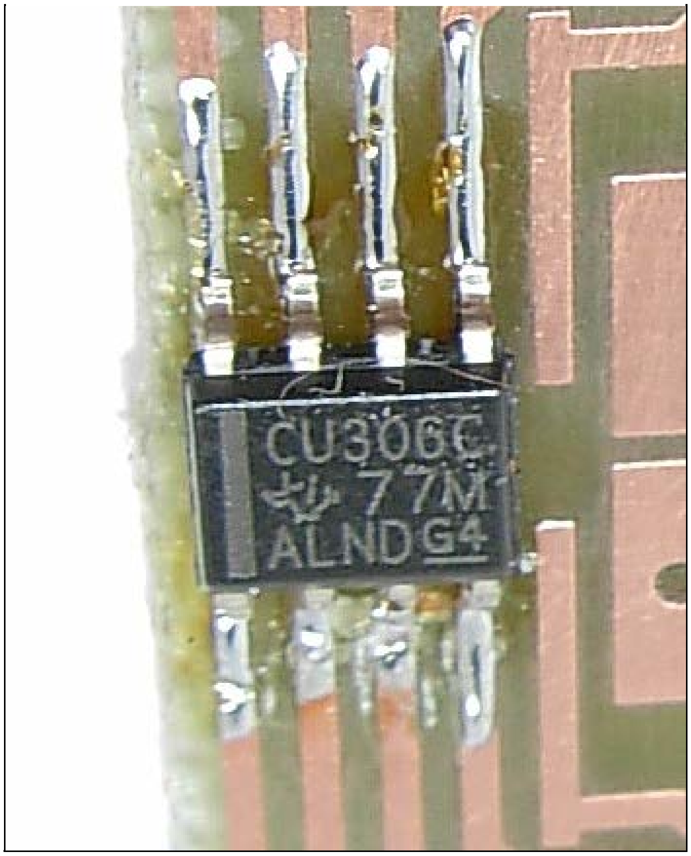


Shown above in Figure 2 is the BJT driver, D flip-flop and the CMOS switch product detector. The 14 MHz VFO connects to Q1 via a 0.1 uF coupling capacitor that is shown on the VFO schematic. A dual FET bus switch (CBT3306) serves as the product detector. The on-resistance of this switch is only around 3 ohms! If you are ambitious, you might try using a 14 pin SOIC switch such as the QUAD FET SN74CBT3125DR with 2 pairs of the 4 switches wired in parallel. I tried 3 different CMOS switches in the U2 slot. The other switches were the MAX4066CPD and a 74HC4053 (wired up appropriately using their datasheets). The insertion loss and performance of these 2 switches was disappointing. My bench standard for comparison was a 7 dBm diode ring mixer. Numerous experiments were performed. For example, I tried running the 4066 at 12 volts VCC to minimize its on-resistance and had to modify most of Figure 2 as well. Being new to digital, blending 5 volt and 12 volt logic IC required great effort to get it working properly. These experiments consumed the better part of 2 days. My conclusion was that if you are going to go to the trouble of make a CMOS switch work, you might as well use a part that has a low on-resistance. Hence, I have since abandoned using DIP IC CMOS switches (4053, 4052, 4066 etc.) as mixers and product detectors. They may still be a good choice in your own context.

The CBT3306 is outstanding and very similar to the diode ring mixer with respect to insertion loss and audio quality as a product detector. There are some other good CMOS switches you might try. I chose the CBT3306 because it was SOIC (the largest of the common SMT topologies), costs only 77 cents (Canadian dollars) and only has 8 pins to deal with. Pragmatism on the workbench is always good!

SMT versus VE7BPO

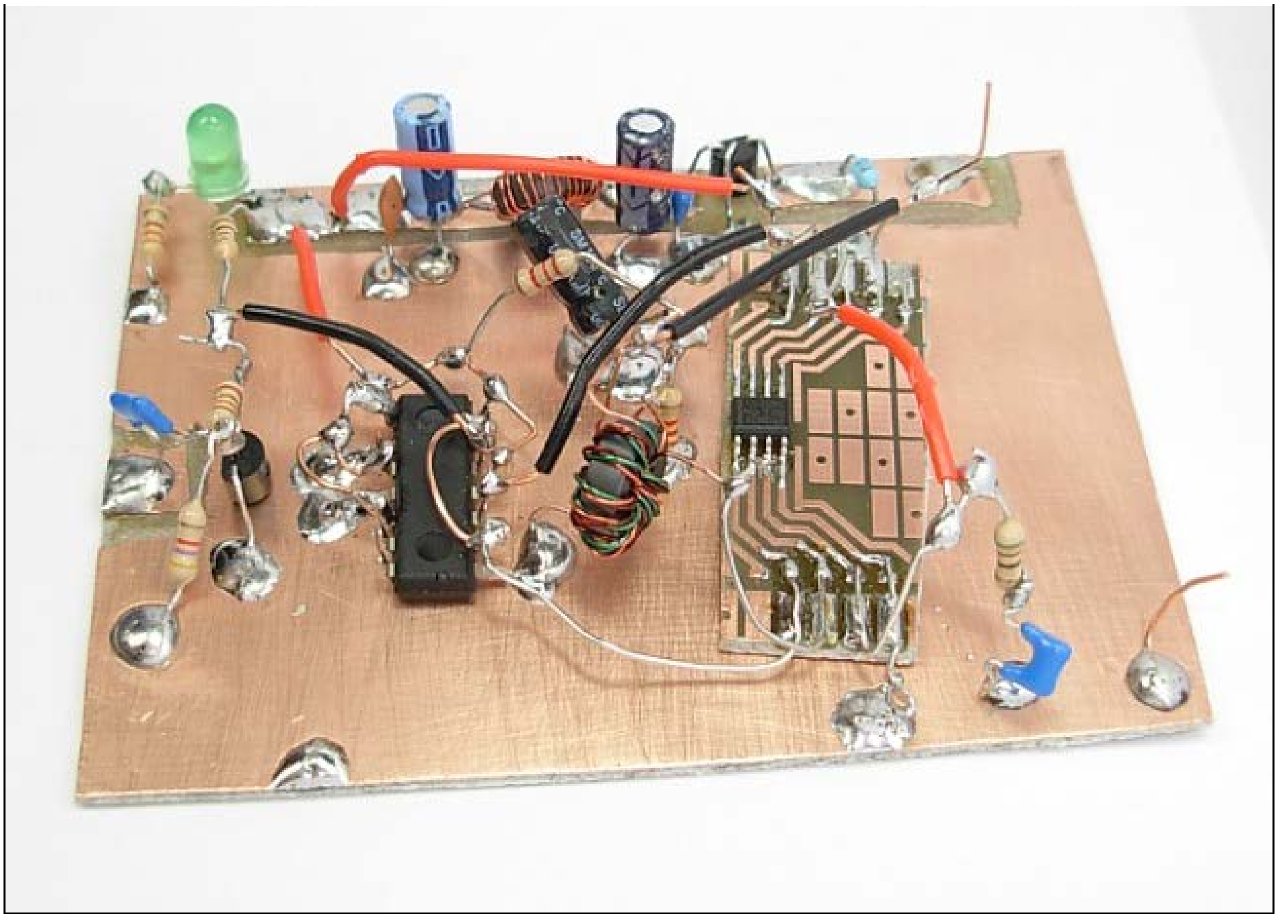
The difficulty with using the CBT3306 was that I had to learn about and equip for SMT. I ordered the switches and after their arrival, hesitated to do anything with them for 5 days. I managed to borrow a magnifier and bought a SOIC prototype board and some flux. Still, I seemed to be paralyzed with fear about soldering U2. I was stuck. For inspiration, I went back to Mike's web site and found this web page. Here Mike was working with a 18 pin SOIC chip and I was worried about a mere 8 pin IC! Part of the problem was my relatively poor eyesight. I learned this can be managed with a magnifying visor. I bought mine here , but only after U2 was soldered in. The borrowed table magnifier was okay, but constrained arm movement and reduced lighting. The visor seems to be a better choice.



Shown above is the CBT3306 soldered on my prototype board. I put flux on the board traces and then tinned them. Following that, I lined up U2s leads on the traces and began soldering. The bottom pins (pins 1-4) did not go well, but they were soldered all the same. Following that, the soldering of pins 5-8 went very well. I was very happy; the SMT monster had been tamed! It was learned from Wes, W7ZOI that many builders use Surfboards for SOIC applications. I will get some for future SMT IC work. Using SMT parts in your Ugly Construction allows you to use parts which are unavailable otherwise. Increasingly, good old DIP ICs are disappearing from catalogs and some new parts are appearing as SMT only. It makes sense to jump in and use SMT parts when it is advantageous for spec reasons, or if miniaturization is required.

With some effort, the Figure 2 circuit could be morphed into Colin, G3SBI's H-mode mixer format. Not on this web site though! If you Google "H-mode mixer" , many good websites will be returned.

The product detector's baseband audio output is at the 47 uF capacitor which connects to Q1 of Figure 3. A simple 51 ohm, low pass network is used to terminate U2. This is from W7EL's An Optimized QRP Transceiver from QST for Aug 1980. I have some more information regarding AF termination circuits on this web page. You may wish to increase the 170 uH inductor value somewhat to get more low pass filtering. The product detector circuit is the number one potential source of hum and noise in this receiver. Take the time to plan your layout to minimize wire length and crossing and provide some physical symmetry. Decouple well.



Shown above is the product detector. At this point, I had not decide how to terminate U2 and had soldered a shunt 0.1 uF cap and 51 ohm resistor to the switch output. U2 is dwarfed by the FT37-43 toroid; a part we normally think of as small in size.

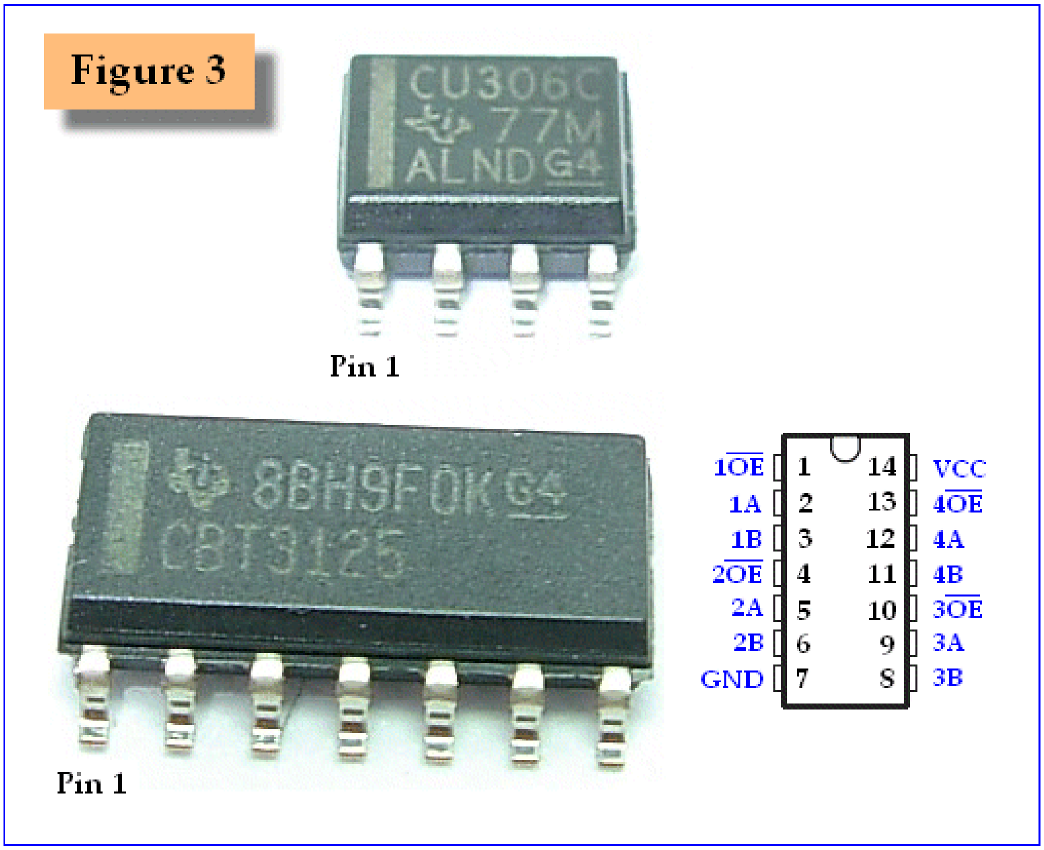
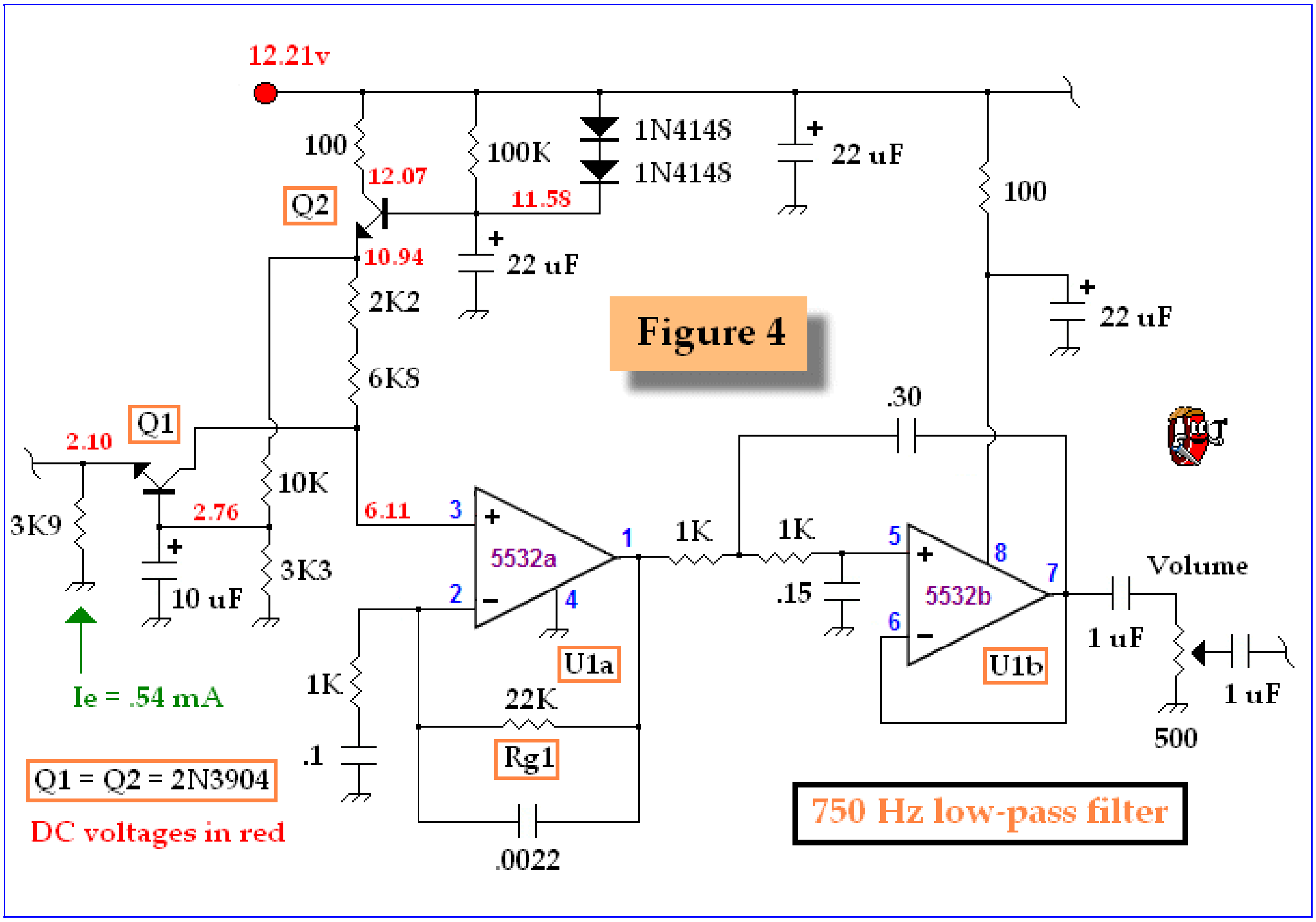


Figure 3. Photographs of the FET bus switches I have in my collection and the pin-out for the 14 pin CBT3125

A diode ring mixer was used as a reference mixer in the product detector experiments.

Audio Amplifier Chain



Shown in Figure 4 is the audio preamp and first low pass filter. Direct conversion receivers are all about audio. High performance DC receivers have become more common since Rick, KK7B unleashed his R1 on the world in 1992. High performance receivers are out of scope for this popcorn website, however, this design is welcome. The entire audio chain uses low-cost 5532 op amps, cheap BJTs, plus fairly common resistor and capacitor values. Poly"something" capacitors were used for all audio AC coupling and shunting capacitors of 4.7 uF or less.

For hum immunity, the familiar BJT capacitive multiplier popularized by Roy, W7EL was used at Q2. Using a high-end spectrum analyzer, Wes, W7ZOI demonstrated that this circuit can oscillate. He detected oscillations at UHF. Thus, Wes recommended using a 100 ohm snubber resistor on the Q2 collector to alleviate these parasitic oscillations. Additionally, 2 diodes were added in parallel to the 100K base resistor. This provides an instant-on feature for the audio preamp, as normally there is a time-constant delay when you switch on the receiver. I like this feature, but it is purely optional.

Q1 is a common base amp biased for 0.54 mA. Therefore, the input impedance is 26/0.54 or about 48 ohms. The 6K8 + 2K2 collector resistors were paralleled so that the quiescent collector voltage was close to 6.1 volts. You could also substitute a single 9K1 resistor, although this is an uncommon value. This voltage provides the approximate VCC/2 bias needed for U1a and U1b. Connecting the Q1 collector directly to the pin 3 op-amp input allows the exclusion of a coupling capacitor and the usual VCC/2 resistor network used to bias the 5532 op-amp from a single power supply. I borrowed this from EMRFD.

The gain of U1 is set by the resistor labeled Rg1. If after testing, the AF gain is too high (for example if this AF stage was used in a superheterodyne receiver), simply lower this resistor value. I chose a 22K ohm resistor to allow enough gain for weak signal listening. On louder stations, you will need to lower the volume control to prevent distortion in the stages that follow as overall, there is a lot of gain in this AF chain. Adjust the Rg1 value to suit your needs. If you use a switch such as a 4052 for U2, or your receive antenna is small, you may want to increase this resistor a little for more gain. This is an experimenter's receiver after all. The 0.0022 uF capacitor in the op-amp feedback loop provides a single pole of low pass filtering. I ran this capacitor as high as 0.0082 uF. This gave a theoretical 3 dB cutoff frequency of ~ 880 Hz, but the receiver lost its sizzle. Experimentally, I learned that using a 0.0022 uF feedback capacitor on both U1A and U2A dropped some of the high frequency noise while preserving some sparkle in the received audio.

The audio preamp chain is concluded with a 750 Hz low-pass filter. Resistor values were kept low to minimize noise. All the active low pass filters are low Q, Sallen-Key designs with a Butterworth response. They are stable, easily scalable and brain-dead simple. These filters are fatigueless due to their gentle cut off slope. I actually had a 0.30 uF capacitor in my parts collection, although it was a 600 volt rated part from my tube amp parts bag. You could use two 0.15 uF capacitor in parallel or just substitute a 0.33 uF capacitor. If you can't find a 500 ohm volume control potentiometer, swap in whatever you have, but try to keep the resistance low.

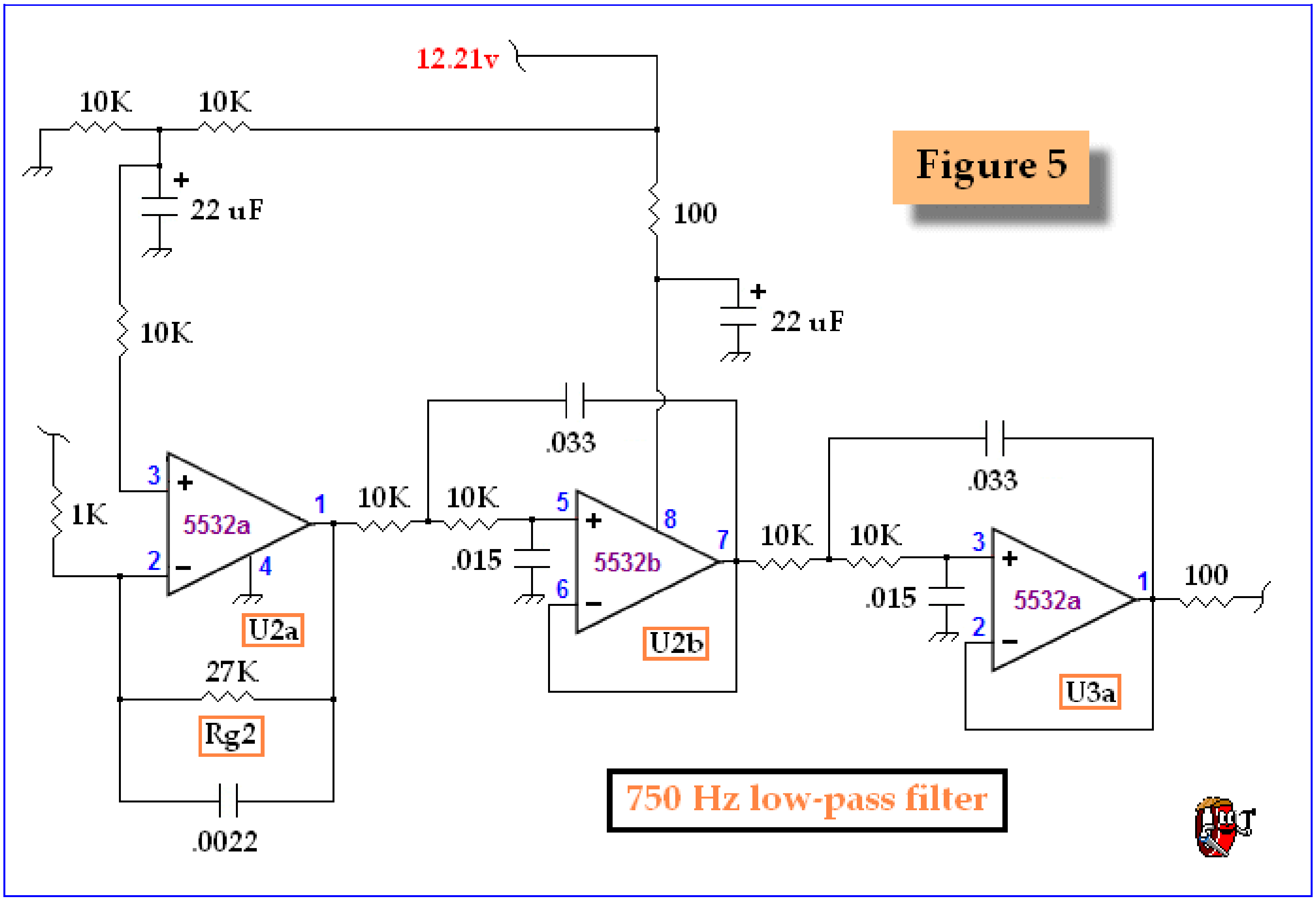


Figure 5

750 Hz low-pass filter

In Figure 5, the remaining preamplifier stage and low-pass filters are shown. Rg2 was chosen for the same reason as Rg1 in Figure 4. Please experiment with these values. The low pass filter stages are scaled up by a factor of 10 as resistance values are less critical at this point in the audio chain. You could use the same resistor and capacitor values used in the first low-pass filter or use the Figure 5 values for all of the low-pass filters. Very often, our parts collections dictate how we experimenters design and build circuits.

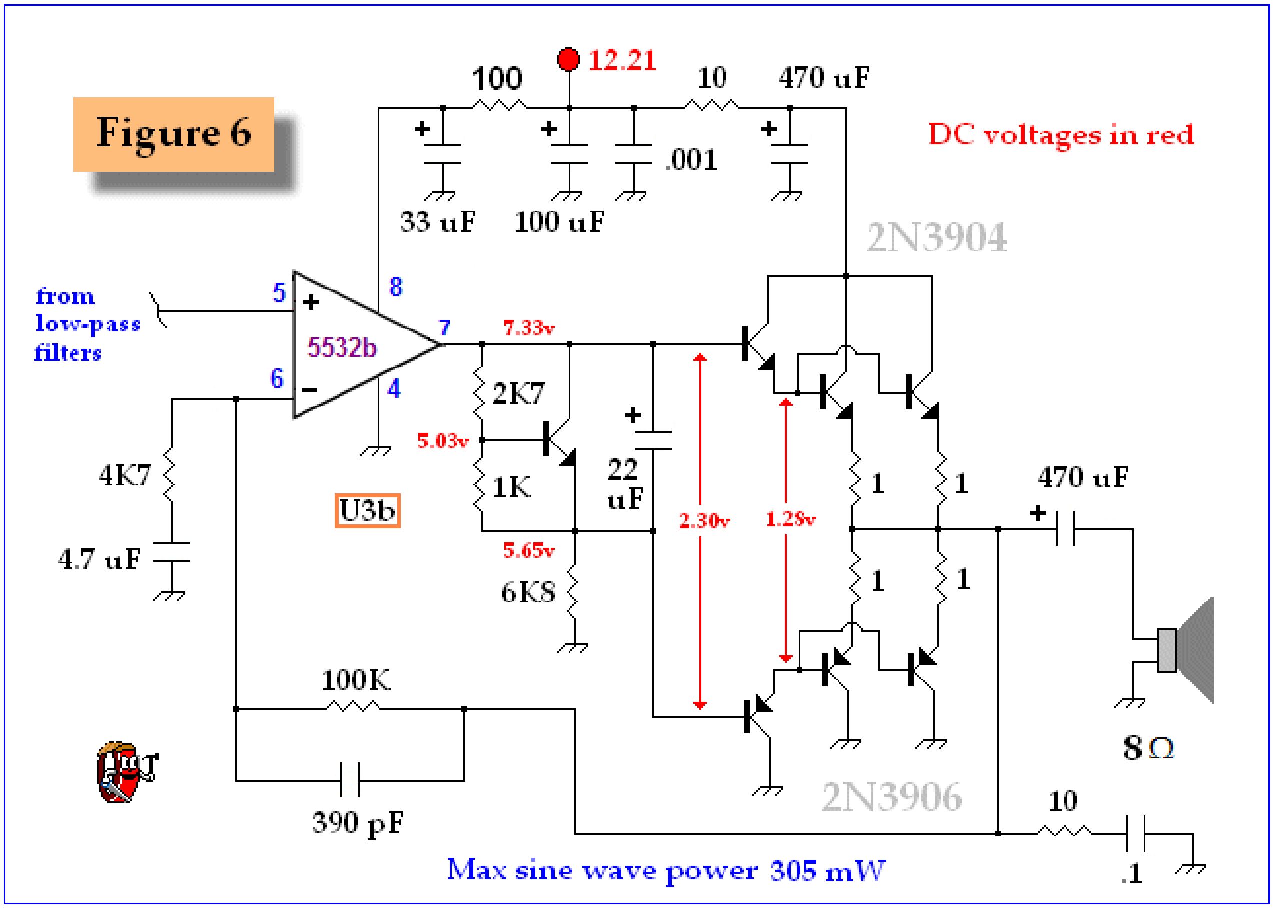
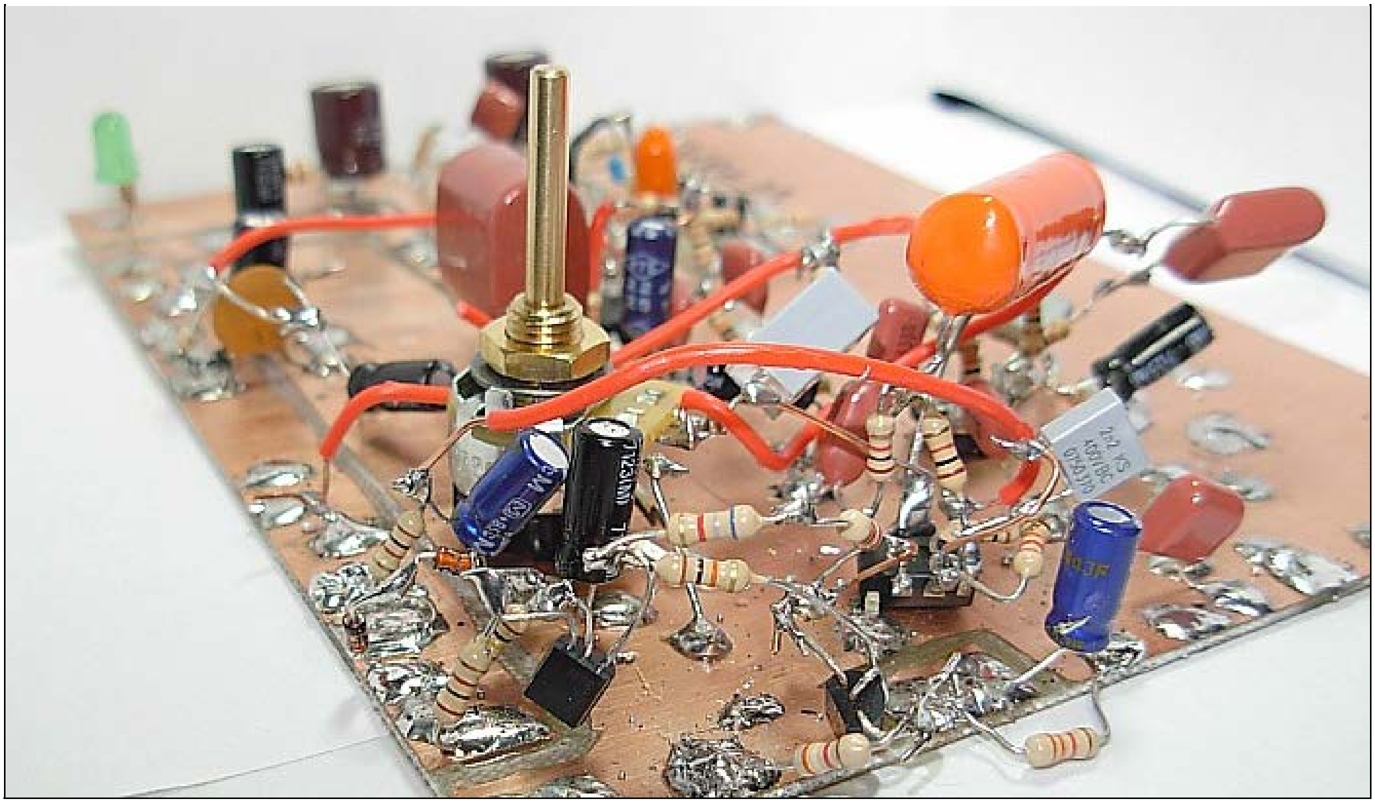


Figure 6 is the power amplifier schematic. Apart from field portable transceivers where headphones are used to save battery power, I exclusively listen to my receivers via a loudspeaker. This safely allows the exclusion of AGC circuits. You may have noticed that AGC circuits have not been presented on this web site. I rarely use them. The extensive VCC decoupling in Figure 6 is needed in this high gain AF chain. The 1 ohm emitter resistors in the finals invite instability, but were tamed with the 10 ohm / 0.1 uF low pass filter. My original design called for 2.2 ohm emitter resistors, but none were available. This final amplifier is moderately loud and very quiet. When it was first built (the AF chain was built backwards), you could not tell it was on when no signal was applied. With no antenna connected to this receiver and the volume on full, there is only a little noise. This is a good way to test a receiver AF chain for noise. Speaker choice is also important. Speakers of a greater power rating and size sound better; especially when mounted in a wooden cabinet.

Biasing power amps has been discussed extensively on this web site. Ensure you measure and record your quiescent DC voltages as shown in Figure 6. If you hear cross over distortion or the quiescent voltage between the bases of the paralleled final transistors is less than 1.25 volts, try decreasing the 6K8 resistor to 5K6 ohms.



The final preamp breadboard is shown above. You can tell by all the grunge and the solder marks, that many different configurations were trialed. Despite looking haggard, the AF chain is quiet and does not hum nor detect broadcast radio.

Shown above is an early breadboard of Figure 6. In this version, the volume control was at the input and a voltage divider network was wired to pin 5 to provide VCC/2 bias. Testing with an audio signal generator, a tape player and other sources were performed. Later, the biasing resistors and the potentiometer were removed and the Figure 5 stage was added to the copper clad board and tested.

Final Thoughts

We are remembered best by those whom we affect. Certainly Mike's unique perspective and enthusiasm inspired me to dig deeper into this hobby. I asked Bill, M0HBR, for a quote to conclude this web page. Bill wrote this: "Just last week somebody was asking me for background info on Mike's DSB modification of the Heath HW-8... I think it is a real tribute to Mike that years after his passing, hams around the world are still talking about him fondly, still visiting his web site, still following his lead on homebrew radio projects. Certainly among the 3,000 + listeners of the SolderSmoke podcast, Mike is present in spirit every time a soldering iron is heated up." Thank you Bill.

My heartfelt regards to Mike's family.

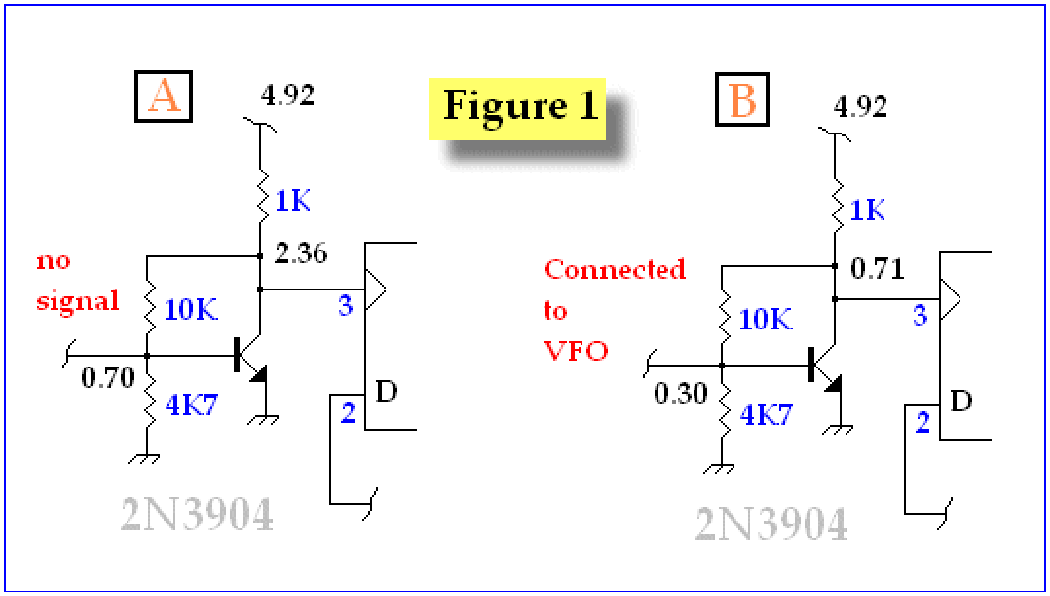
Amateur and Short Wave Radio Electronics Experimenter's Web Site

VFO Experiments 2009

Supplement to the VFO 2009 Web Page

This web page is a supplement to the VFO 2009 Web Page

Figure 10 Bipolar Transistor Notes

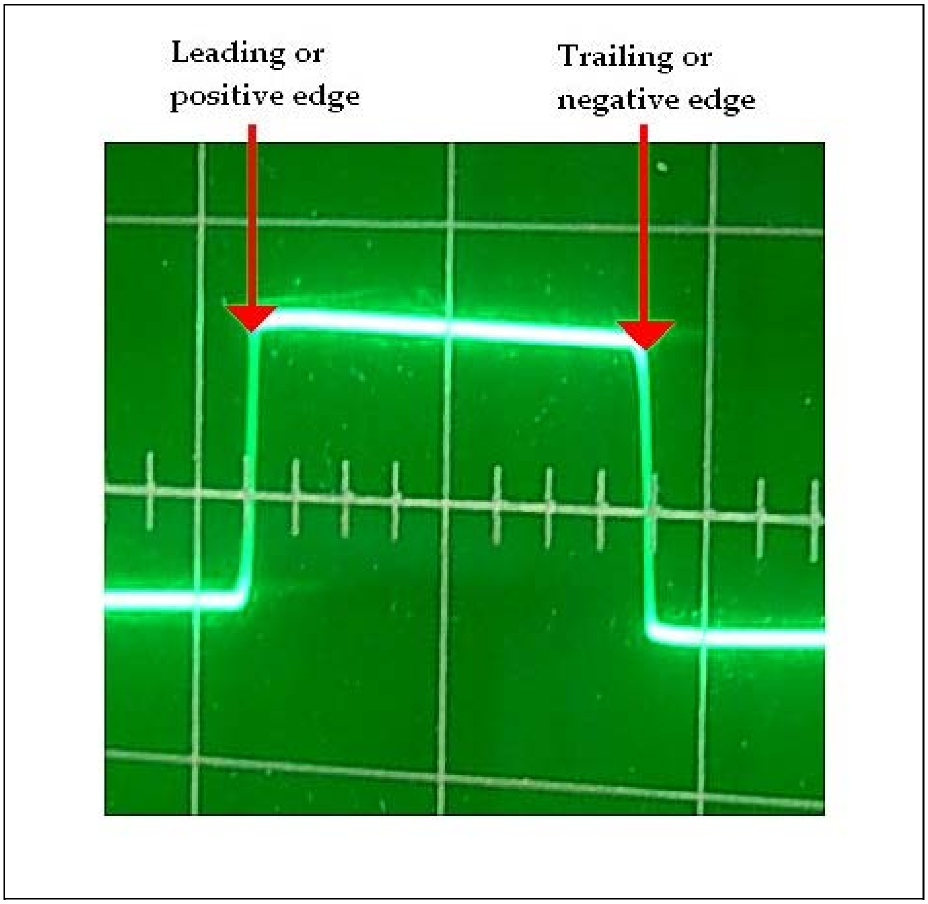


This circuit is from EMRFD and is an analog to digital interface for the VFO. I did not understand the circuit, so I asked Wes, W7ZOI to explain his design criteria and the basic circuit function. This transistor circuit is a driver. The purpose of this transistor driver is to positive edge trigger a D flip-flop. The paramount criterion was that the transistor collector has a quiescent bias voltage that fell between the minimal acceptable logic high and logic low DC voltages on the 74HC74 clock input. See this chart for details about this topic. The correct DC collector voltage is set via feedback from a voltage divider (in this case, a 10K and 4K7 resistor). The collector DC voltage (shown above in Figure 1A) was 2.36 and lies perfectly between the logic low and high state of the 74HC device that it drives.

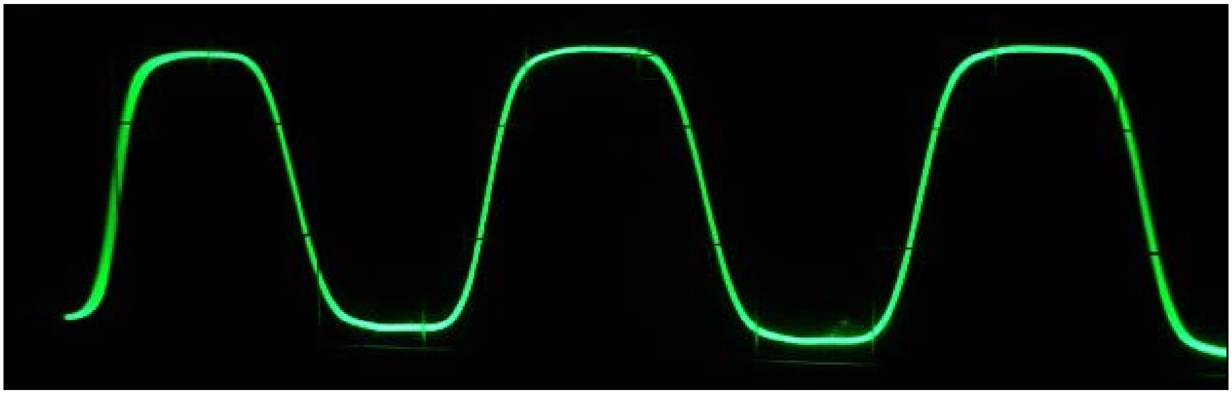
This transistor circuit is also a high gain amplifier, however, its primary function is a driver. Refer to figure 1B. When connected to the VFO output, the base voltage drops to 0.30 volts. This is an average DC voltage because it fluctuates as the AC waveform swings up and down. More or less AC drive on the input will change this DC voltage.
When connected to the VFO output, the collector voltage dropped to 0.71 VDC. Again, this is just an average, as when the AC swings positive, the the collector voltage will drop down to saturation (where collector voltage is less than base voltage) and then increase towards the positive supply. It does not have to go into cutoff; just to 3.2 volts or so. It also does not have to go into saturation; just to below the 1.3 volts or so. Thus, the transistor remains between cutoff and saturation when appropriately driven with a sine wave.

Our VFO serves as the clock and data on the D flip-flop inputs are only transferred to the flip-flop's outputs on the positive edge of a clock pulse. Positive edge triggering means that the output only responds to the input changes when the clock signal transitions from logic low to logic high. What a fantastic little circuit! You can find additional information and some practical examples in EMRFD. My special thanks to Wes, W7ZOI for helping me to better understand his design.

Positive Edge Triggering



The triggering of a positive edge triggered flip flop is better understood viewing a graphic like above. The instant the waveform goes positive, the flip flop is triggered. It is easy to understand why a square wave is desirable for clean triggering.



Radio Amateurs of Canada Support Page

Commentary for Canadian Radio Amateurs

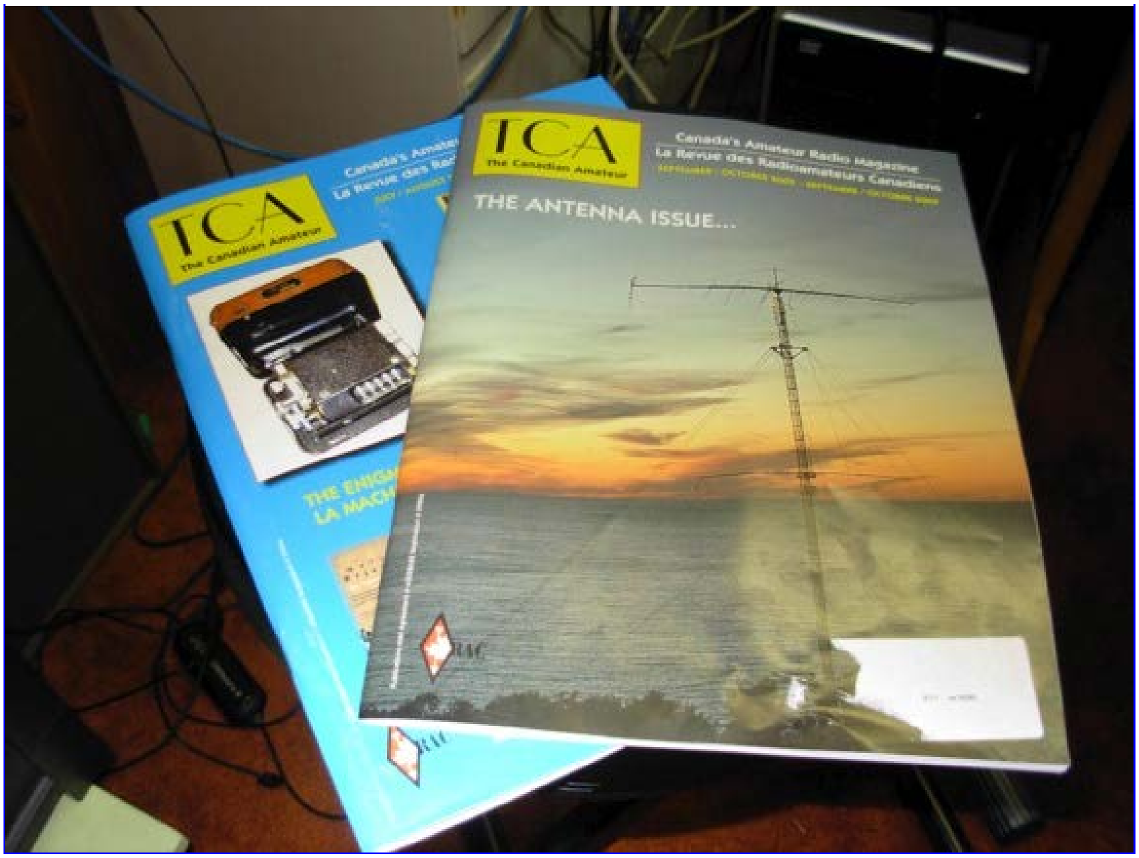
A significant number of licensed Canadian amateur radio operators do not support their national amateur radio society: Radio Amateurs of Canada Typically support is by membership, but may also include volunteering time to perform the various functions required to run a large, non-profit organization. It has been demonstrated repeatedly, that a small percentage of people are doing most of the work in modern volunteer organizations.

Some thoughts concerning volunteerism

- Volunteerism is down in many sectors including service clubs, meal delivery programs, church groups, volunteer firefighters radio clubs, etc. etc.
- The population is aging and our current volunteers are retiring; Younger folks have so many distractions and things they can do today. We now live in a world with 200 plus TV channels! The spirit of volunteerism seems to be decreasing in the western world.
- There is some evidence that people who donate their cash to charities also tend to participate in groups, associations and organizations. Is this a characteristic or trait?
- People volunteer for causes they believe in. What do people believe in now? Is our culture too self-absorbed, individualistic and focused on our own personal agendas and pleasure?
- Economics. Are young to middle aged people less able to volunteer and contribute because they are spending more of their time working?

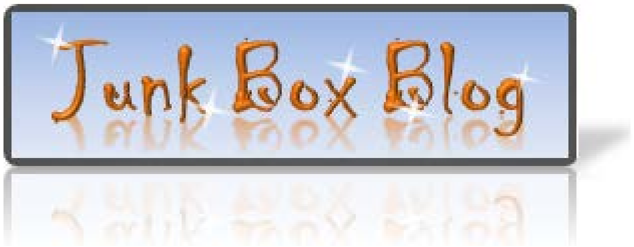
RAC Membership

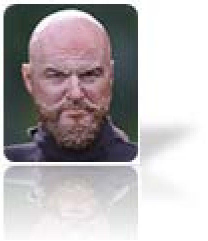
If you haven't, please consider joining the RAC. I joined out of respect to the many RAC volunteers who have devoted their personal time and continue to toil to keep amateur radio viable in Canada. We cannot take our currently allocated radio frequency bands for granted. There are groups who for profit, want to apply technologies which will consume or interfere with the radio frequency bands we enjoy. Canadian amateur radio needs a strong, united voice to survive into the future.
Apathy and lack of awareness may significantly decrease the rights and privileges we enjoy as radio amateurs today. The least we can do is support our national amateur radio society. It would be even better if more of us contributed by volunteering our personal time towards our wonderful radio hobby and serving the RAC.

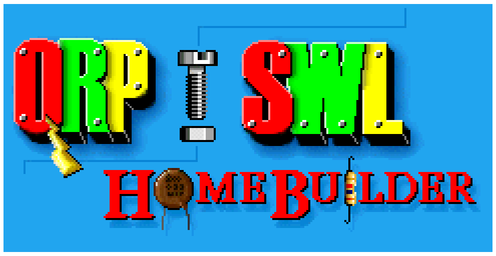


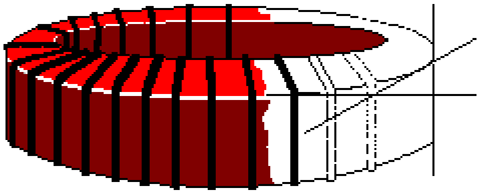
The RAC Journal The Canadian Amateur is an excellent resource. Sept-Oct 2009 marked the first ever antenna issue.

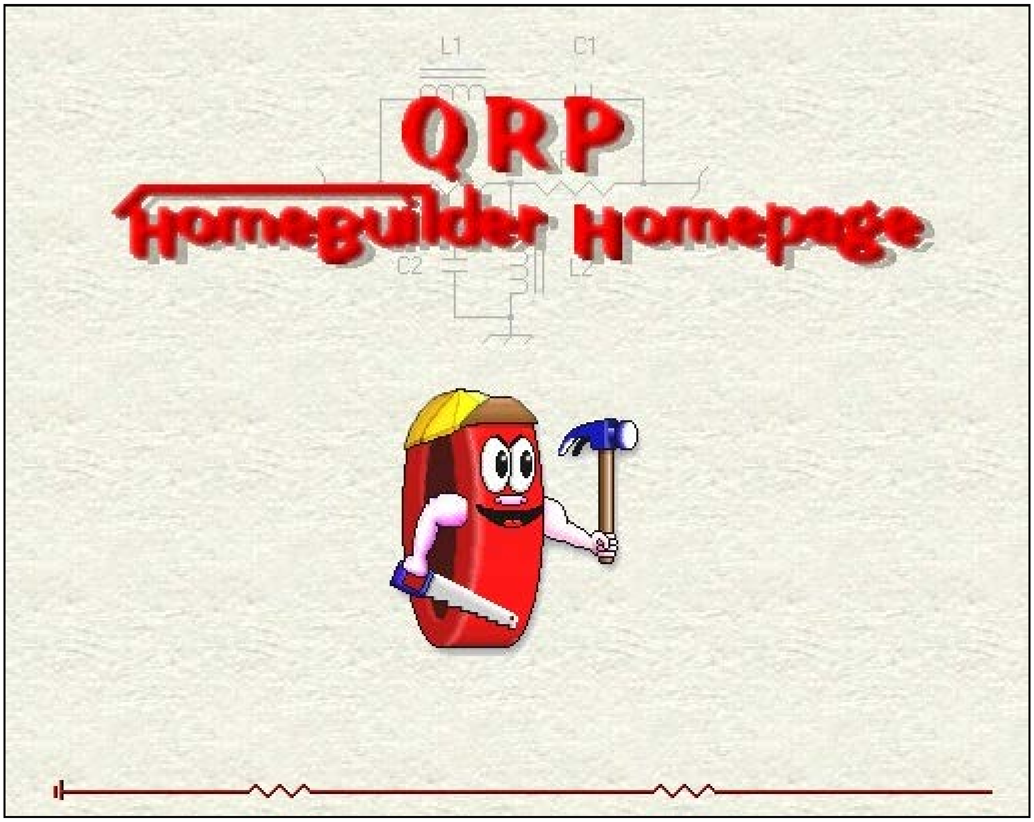
Old QRPHB Web site graphics...

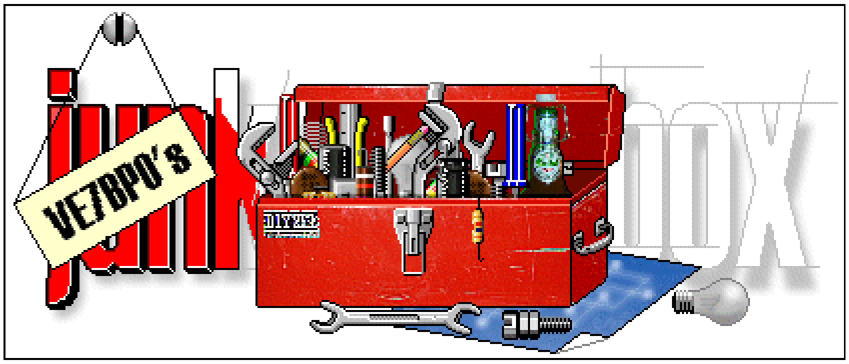


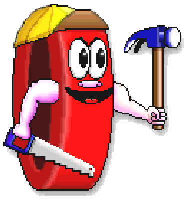







L1
C1
QRP
Homebuilder Homepage
C2
L2

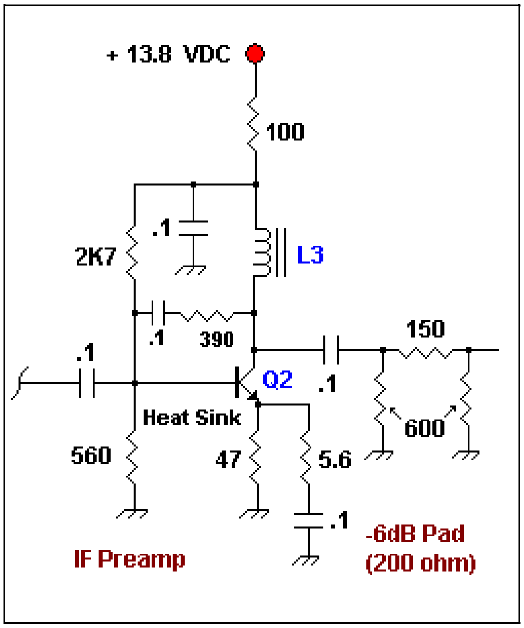

VE7BPO's
junk box



This page last updated: September 21, 2013

Info on the Popcorn Superhet RF Amp

Discussion



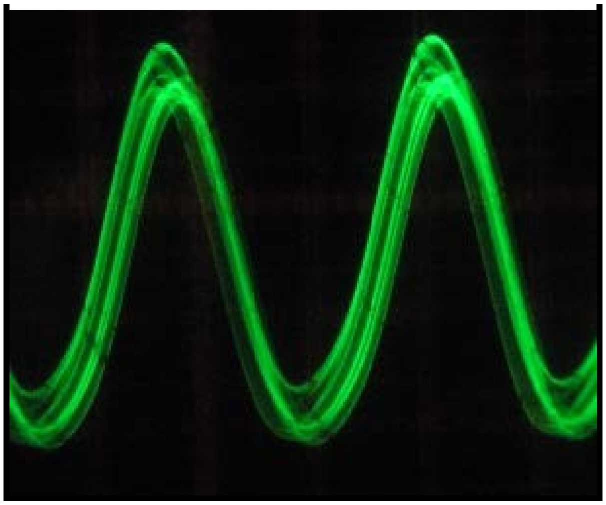
I have received a lot of emails regarding this RF amp from the popcorn superhet receiver. This amp was designed using a calculator and as it turns out, my input and output impedance was off by several ohms from the target 50 ohms input Z and 200 ohms output Z. Today we have computer programs to calculate the resistor values, so crunching the math is not a factor any more. This RF amp is preceded by a 50 ohm diplexer and followed by a 200 ohm pad, so it actually matches the preceding and following stages very well considering that it is clearly depicted to be a popcorn receiver. If I had to redesign this amp in 2008, I would use software to design a CE amp using "noiseless" Norton feedback, Still, this was a good sounding receiver at the time.
Probably the weakest section of this receiver is the audio preamp from Q5 on through to the LM386. I would likely never use these noisy stages in 2008-9. Instead I would use low noise op amps. This web site like my own abilities has grown over time. In the following section, I have paraphrased some writing from Wes, W7ZOI concerning the Q2 RF amp.

An Analysis by W7ZOI

The input impedance of a common emitter bipolar transistor amplifier is related to the standing current, the emitter degeneration resistance, the transistor beta, the F-t, and the operating frequency. Most of those details can be well modeled with the hybrid pi model. If you put quite a bit of emitter degeneration in such an amplifier, the gain drops while the input impedance increases. However, it is much less dependent upon beta and frequency. The emitter degeneration is one form of negative feedback. But now we introduce a second form of negative feedback by applying some of the collector voltage to a resistor that generates a current that is fed back to the base. This feedback tends to reduce gain just as does emitter degeneration. But it also tends to reduce input impedance. By playing a careful balancing act, you can design an amplifier that has a 50 Ohm input and output impedance with a reasonable amount of gain. This particular circuit has a gain of 21.7 dB at 10 MHz and an input return loss of 14 dB. The output return loss is even better at 18.6 dB. Z-in is 39-j14 while Z-out is 172-j34.

These are calculated values using that hybrid pi model. I have devoted a few pages to a discussion of this feedback amplifier topology in the ARRL 2003 book, Experimental Methods in RF Design. This was co-authored by KK7B and W7PUA. You can get the book on line from the ARRL. The book includes computer programs that will handle the design chores. One deals with bias issues while another does the RF design and analysis.

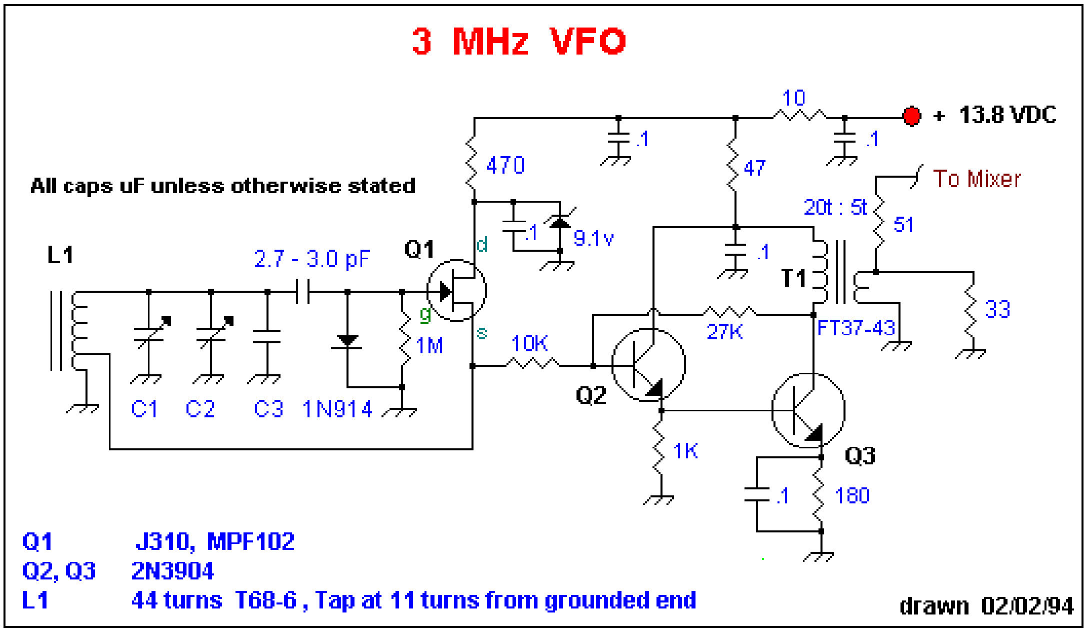
The RF amplifier under discussion may be found on this web page



---

3 MHz VFO for 40M Popcorn Superhet Receivers

Discussion



This VFO was first popularized by Roy Lewallen, W7EL and has been used in countless homebrew and commercially sold QRP rigs. For the 40 meter ugly super hets, you have a choice of a VFO frequency of 3 MHz or 11 MHz and I chose the former for this website. For L1, you can use the T68-6 as shown or redesign your own inductor using a T50-6 core. The T68-6 core allows the builder to wind the coil using # 24 AWG wire for enhanced Q and stability. The main tuning capacitor, C1 that was used had a built in reduction drive and went from 4 - 19 pF. If you use a tuning capacitor with a greater capacitance swing, you may have to connect it to the top of L1 via a small-value NP0 capacitor to reduce the tuning frequency range. This can be done using math or by just plain experimentation. C2 is a small ceramic air-variable trimmer from my junk box used to set the lower band edge of the VFO. It can be omitted if the user wants to go frugal and experimentally set the lowest frequency of the VFO using small-value ceramic NP0 caps. Air variable caps for C1 and C2 are mandatory for minimal drift VFO operation. C3 actually refers to 4 NP0 ceramic caps which were used to place the VFO on the correct frequency. Four caps were used to minimize heating and thus drift in the VFO. For the prototype VFO which tunes something ~ 3.00 to 3.67 MHz , I used the mentioned air-variable, four NP0 ceramic caps (20pF,100pF,5pF,100pF) and C2 the trimmer cap that went from 2 - 50 pF. These values should be used only as a guide for prospective builders as there are many variables at play. Following a 10 minute warm up period, the VF0 frequency stability is excellent. Again, keeps all component leads as short as possible. T1 is a broadband transformer wound by using 20 turns of # 26 AWG over a FT37-43 toroid core and then distributing 5 secondary turns spaced evenly over the primary windings. Do not omit the 33 ohm load resistor. The 2.7 to 3.0 pF coupling cap should also be of the NP0 ceramic type. Q1 can be the J310 (my favorite) , 2N4416 , MPF102 or other correct substitutes.

This VFO should be in a separate sealed, shielded box from the rest of the receiver

.

VFO Stability Hints

Numerous tips for enhancing VFO stability have been generated from Wes Hayward, W7ZOI, Roy Lewallen, W7EL, Doug DeMaw, W1FB and others. Here are a few summarized:

1. Use air wound or powdered-iron toroidal inductors made from number 6 material.
2. Use the heaviest gauge of wire possible to wind your inductor.
3. Anneal the inductor by boiling it in water for around 5 minutes after winding.
4. Use ugly construction above a SINGLE-SIDED copper side up ground plane.

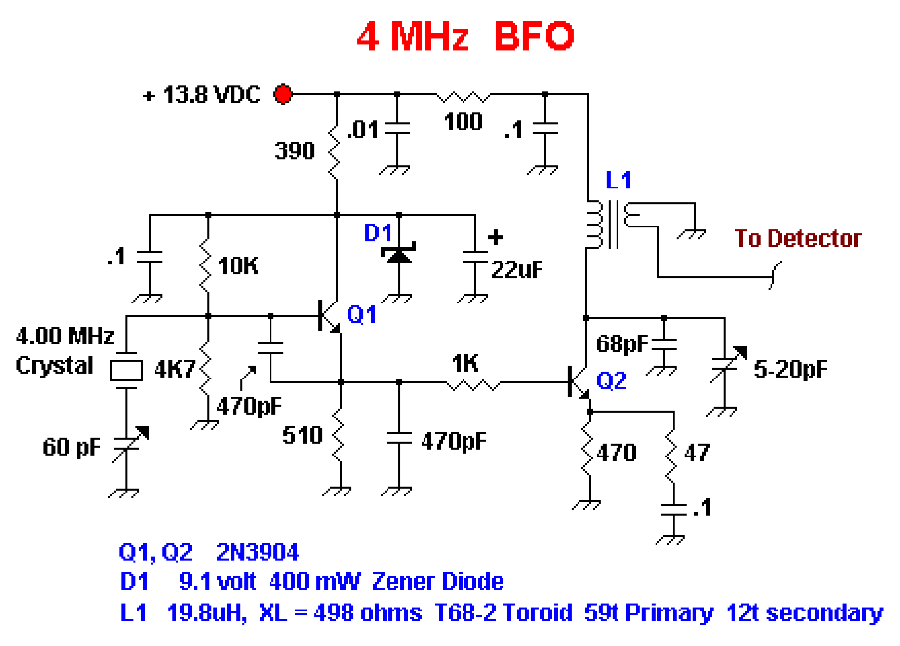
5. Capacitors in the L-C circuit should be NP0 ceramic type.
6. Use air variable capacitors for tuning that have double bearings and no backlash.
7. The VFO should be operated at a lower regulated voltage.
8. Do not use cheap low-Q trimmer caps.
9. Completely encase the VFO to prevent RF leakage and to minimize environmental temperature changes.

Amateur and Short Wave Radio Electronics Experimenter's Web Site

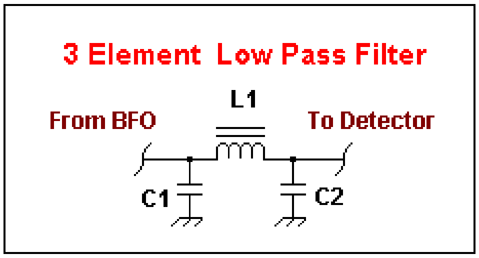
4 MHz BFO for 40M Popcorn Superhet Receivers

Discussion

Shown is a 2 stage BFO for use with the Ugly 40 meter superhets with a 4.00 MHz IF. This is a design by W7ZOI which I have used from 3.5 to 9 MHz by only changing the crystal and the Q2 output coil and capacitor values to suit the frequency of choice. The output coil is 59 primary and 12 secondary windings on a T68-2 toroid core. You will need in the order of 79 pF to resonate this coil, thus a 68 pF plus a 5-20 pF variable works well. If you do not have a T68-2 core, a T50-2 could be used with 30 AWG wire which is a bit difficult so you could opt for an XL of 200 ohms which would be 40 primary and 8 secondary turns on a T50-2 core. You would need ~200 pF to resonate such a coil and the BFO would have reduced output power, but would still work okay. If you design your own tank, you probably should not use an XL/XC less than 190 ohms for the output stage at 4 MHz. Use a 5:1 turns ratio for primary to secondary windings.



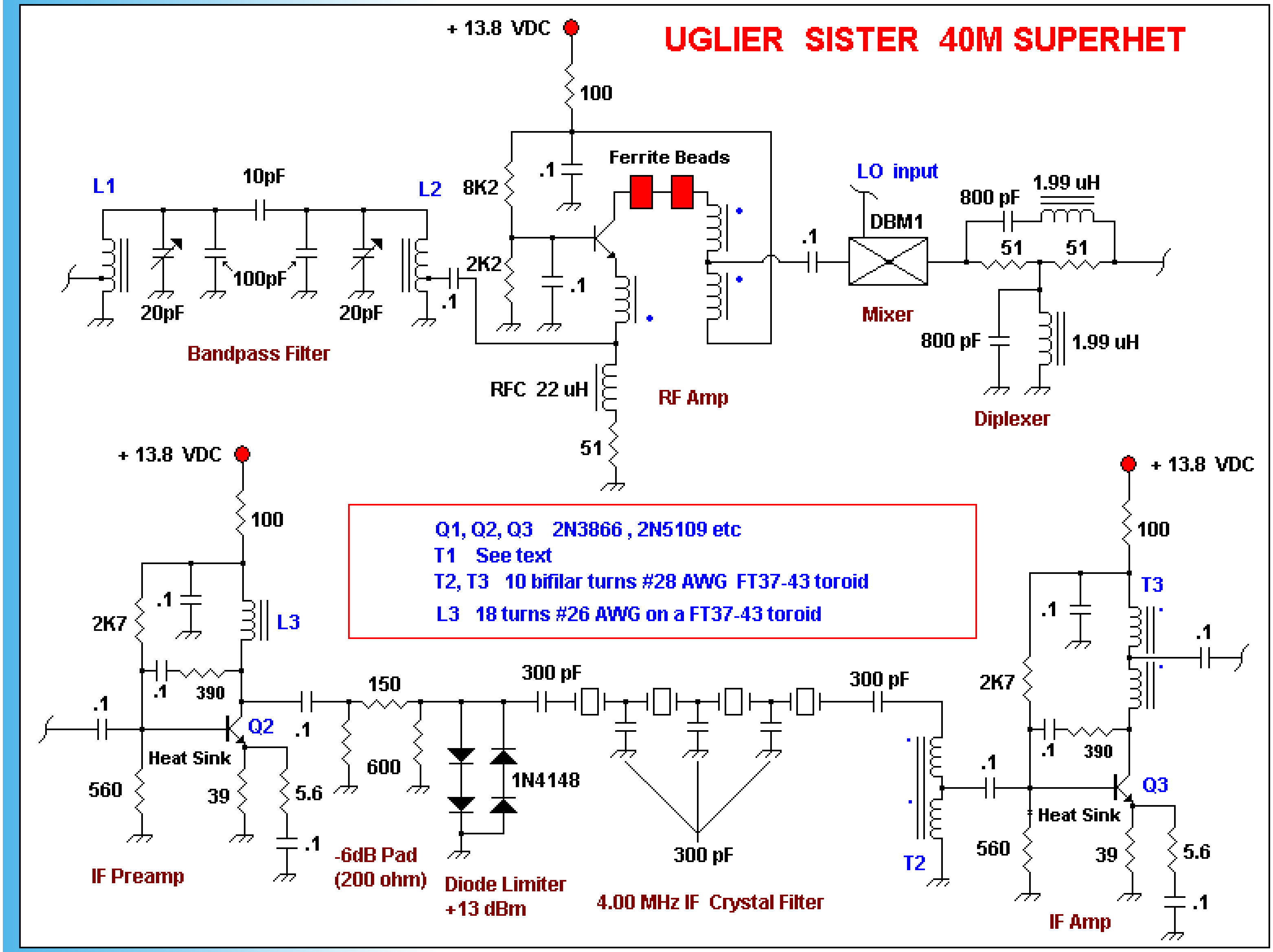
It wouldn't hurt to follow the Popcorn Superhet BFO with a low pass filter. A simple pi-type 3 element filter is suitable. Since I had a couple of 470 pF caps left over, I made a low pass filter using a 6.8 MHz cutoff frequency with a 1.17 uH inductor. A 1.17 uH inductor using a T37-2 core requires 17 turns while on a T50-2 core requires 15 turns. Caps were inexpensive monolithic ceramics from Digi-Key. The filter is shown below, L1 = 1.17uH , C1 and C2 = 470 pF. If you want, you may use other cutoff frequencies to suit any junk box caps you have on hand and use an XL/XC of 50 ohms.



Construction

Keep component leads as short as possible to promote stability. Do not omit the zener diode D1. I thought of using a small ferrite bead on the base of Q2 and will try it on the next version of this BFO that I make. The BFO, if unstable can break into all sorts of weird AF sound effects that are very annoying. The BFO should ideally be in a shielded box, but many amateurs have good results by just placing the BFO strategically in the receiver chassis. When winding L1, make sure you wind the secondary coil so that the secondary's grounded end is on the cold side of the primary coil that connects to the 100 ohm resistor and positive voltage. I just pick one end of the primary coil and close wind the secondary coil from this point in the same direction for the correct number of windings. Then the starting point of the secondary can be grounded and the other end connected to the detector or a low pass filter. After building the coil, temporarily place a 51 ohm resistor from the secondary output lead to ground. Then you can test and peak the tuning of the output tank into a load. Leave the resistor in place for all your testing work such as crystal matching or RF stage testing until you are ready to connect it to the product detector. The 60 pF variable cap connected between the crystal and ground is used to set the beat frequency of the BFO. Adjust it to get a good sounding beat note when listening to a station. You will notice that the beat frequency can be tuned roughly by just listening to 40 meter band noise in the headphones and then tweaked on an active QSO.

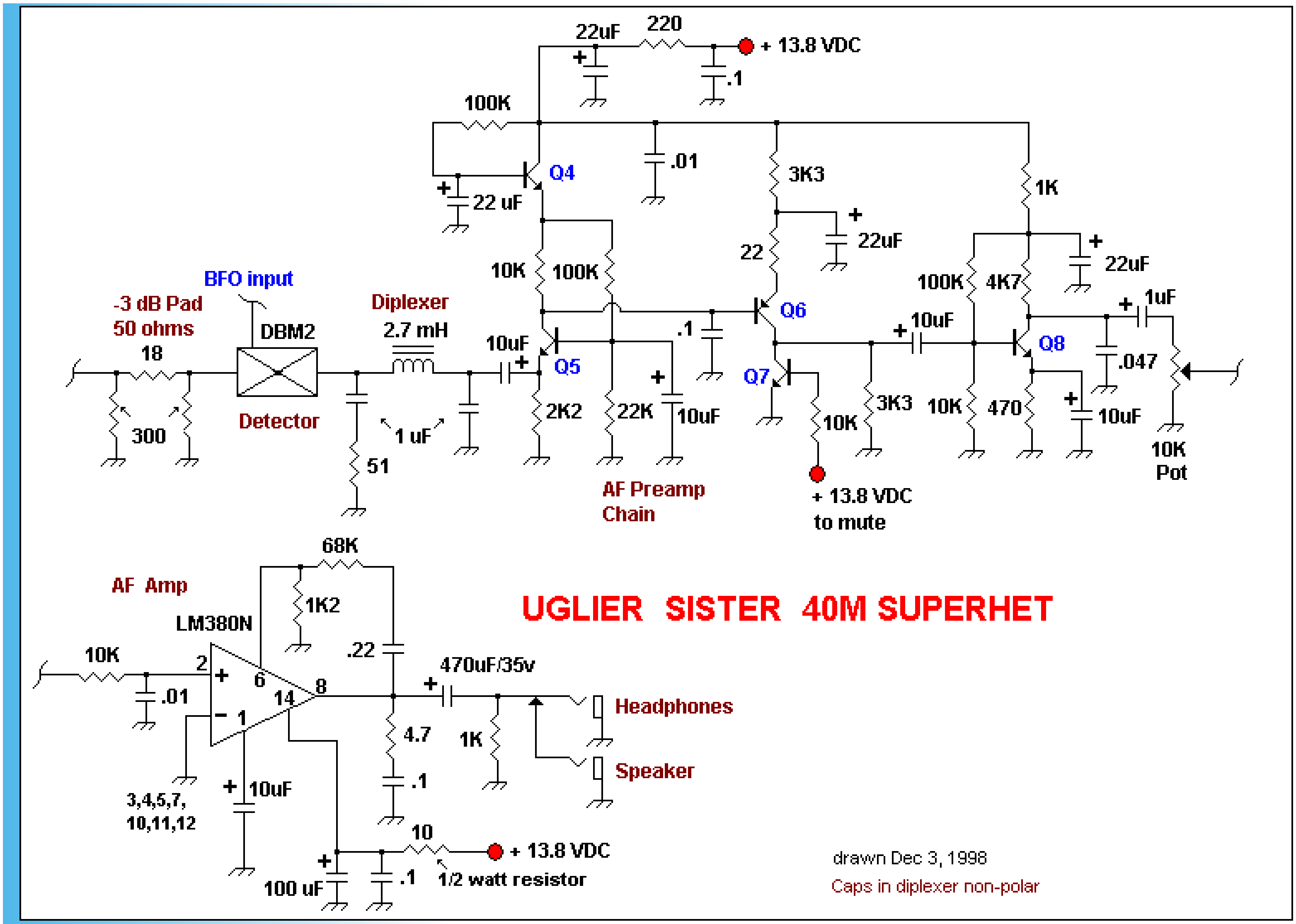
QRPHB Archive Schematics



UGLIER SISTER 40M SUPERHET

Q1, Q2, Q3 2N3866 , 2N5109 etc
T1 See text
T2, T3 10 bifilar turns #28 AWG FT37-43 toroid
L3 18 turns #26 AWG on a FT37-43 toroid

<\p>



<\p>

Uglier Sister Superhet Receiver

This is a 40 M superhet that I built a few years ago. While I was happy with this receiver, it has been discontinued from the main listings on the QRPHB website. I have had a couple requests for this schematic, so I have created this archival web page.
I stood a lot of current in the IF preamp and main IF amplifiers, which is not really necessary, particularly in the case of the post-filter amp. Of course raising the emitter resistor values is easy to do in these amps if you do wish to lower the current. There also is no audio filtering in the AF preamp stages, which is something that I often use in better quality receivers that I build now.

VFO and BFO

[VFO schematic]
[BFO schematic]

Construction Information

[Refer to this web page for details on the xtal filter, diplexer and construction information.]

The Norton RF preamp is discussed on the RF Preamp web page.

[Return to the QRP HomeBuilder Homepage]

Amateur and Short Wave Radio Electronics Experimenter's Web Site

Diplexer Supplemental Page

More Information on the Bridge-Tee Diplexer

Introduction

No web page on the QRPHB web site has furnished more email and controversy than the Diplexer Web page. This page has received much work to improve its content and accuracy and the email received has been greatly appreciated. Presented is the detailed math and some information regarding the Bridge-Tee Diplexer by Helmut Strickner, VK4STR and Wes Hayward, W7ZOI.

VK4STR

The T-bridge diplexer math calculation (also known as the W1JR diplexer)

The formulae presented and used in the program for the Bridge-T Diplexer uses a Q of 1. Using a Q = 1 resonator is not optimal because we don't want a "broad peaking response" of the series LC resonator in the T-bridge diplexer. Our goal should be to only let the desired IF frequency pass and attenuate all other frequencies as much as possible by properly terminating them into 50 Ohms in the diplexer.

The optimal Q for HF frequencies is around 10, less for higher frequencies. The higher Q of the series resonator effectively increases the attenuation of undesired harmonic frequency components from the mixer IF-port, and the reflected products from the crystal back to the IF-port. The IM performance (IP3) of the receiver is improved.

The possible improvement in IP3 depends mainly on the following stages. The post-mixer amplifier following the diplexer has to handle huge impedance variations at the input of the crystal filter. Crystals do not absorb but reflect any products they cannot handle. These products are then reflected back to the IF-port of the mixer deteriorating the IM performance.

A low Q diplexer lets these reflections from the crystal pass unhindered and most mixers will react with degraded intermodulation performance. It is also important to note that a good termination insensitive post-mixer amplifier should follow the diplexer.

The best results for a high intercept (IP3) receiver can be obtained by choosing:

1. High level mixer
2. T-bridge diplexer with optimised Q (Q=10 for HF)
3. Termination insensitive post-mixer amplifier (see N6NWP - QEX July 1995)

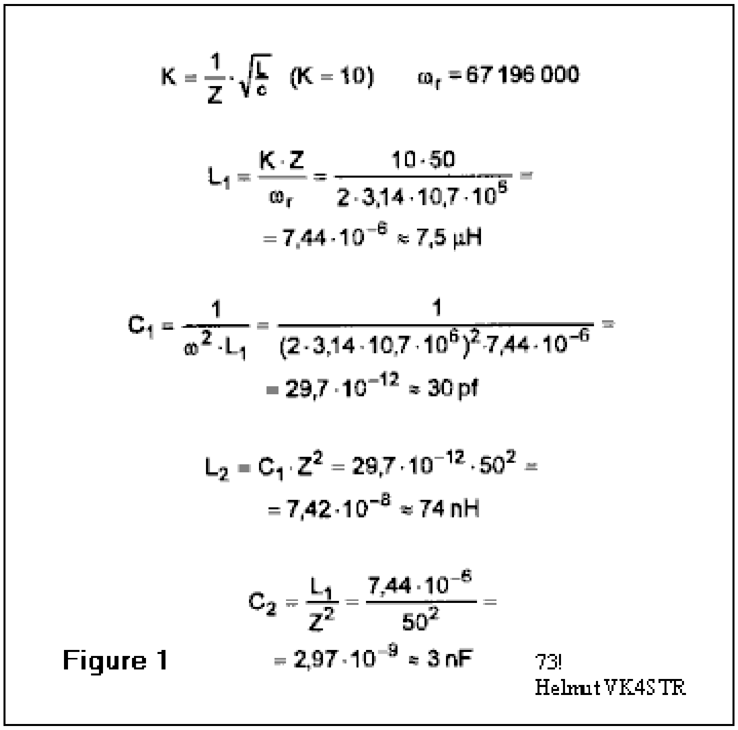
I believe that the intermodulation performance of many receivers can be improved by some simple measures like a properly designed diplexer. The T-bridge diplexer from W1JR is the best one around, go for it.

A brief example of the calculation for a 10.7MHz diplexer follows in Figure 1. The value of K should be 10 for short wave and 2 to 4 for VHF (2 meter band). K is the ratio of L to C.

I hope that you can use this example to calculate the values for other diplexer frequencies.

To fully understand the basis of this calculation please refer to Meinke/Grundlach: "Taschenbuch der Hochfrequenztechnik", Springer-

Verlag Berlin/Heidelberg. Page 127.



$$K = \frac{1}{Z}\cdot\sqrt{\frac{L}{c}} \quad (K = 10) \qquad \omega_r = 67\,196\,000$$

$$L_1 = \frac{K\cdot Z}{\omega_r} = \frac{10\cdot 50}{2\cdot 3{,}14\cdot 10{,}7\cdot 10^6} = $$
$$= 7{,}44\cdot 10^{-6} \approx 7{,}5\ \mu H$$

$$C_1 = \frac{1}{\omega^2\cdot L_1} = \frac{1}{(2\cdot 3{,}14\cdot 10{,}7\cdot 10^6)^2\cdot 7{,}44\cdot 10^{-6}} = $$
$$= 29{,}7\cdot 10^{-12} \approx 30\ pf$$

$$L_2 = C_1\cdot Z^2 = 29{,}7\cdot 10^{-12}\cdot 50^2 = $$
$$= 7{,}42\cdot 10^{-8} \approx 74\ nH$$

$$C_2 = \frac{L_1}{Z^2} = \frac{7{,}44\cdot 10^{-6}}{50^2} = $$
$$= 2{,}97\cdot 10^{-9} \approx 3\ nF$$

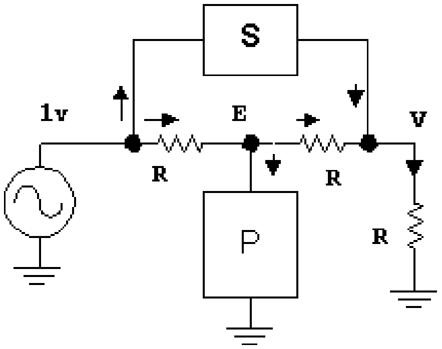
Figure 1

73!
Helmut VK4STR

W7ZOI

Attached are some outputs from Mathcad, Version 7.0. The "proper" equations are derived, along with some other results. Feel free to put these on the web, although this is mathematics rather than the usual "formulas" and there might not be any interest in such stuff.

As it turns out, the formula that Todd presented in completely accurate for the case of Q=1. That is, both the series and parallel tuned circuits have inductors and capacitors with reactance of 50 Ohms. Evaluation then shows that S11=0 for all frequency, while S21 peaks broadly at the resonant frequency. If you pick a higher Q, the inductor in the series tuned circuit increases by Q and the capacitor in the parallel one increases by Q. Resonance is maintained in both. Again, you then get a perfect match at all frequencies and a S21 peak at resonance. But I don't know if this is what Joe had in his original work.



Shown is a general bridge T type network with series impedance S and parallel impedance P. We wish to find the input impedance seen by the generator. But to do this, we need to know the input current. The input voltage is set at 1, so Zin=1/Iin. To calculate this, we must know E and V. These are obtained with a nodal analysis.

We write the nodal equation at E :

$$\frac{(1-E)}{R}=\frac{E}{P}+\frac{(E-V)}{R}$$

We solve this for E:

$$\frac{(1-E)}{R}=\frac{E}{P}+\frac{(E-V)}{R}\ \text{solve},E\ \rightarrow \frac{-\left(\frac{1}{R}+\frac{1}{R}\cdot V\right)}{\left(\frac{-2}{R}-\frac{1}{P}\right)}$$

We now write the equation at V and solve it for E:

$$\frac{(1-V)}{S}+\frac{(E-V)}{R}=\frac{V}{R}\ \text{solve},E\ \rightarrow -\left(\frac{1}{S}-\frac{1}{S}\cdot V-\frac{2}{R}\cdot V\right)\cdot R$$

We now equate the two expressions for E to each other and solve that for V.

$$-\left(\frac{1}{S}-\frac{1}{S}\cdot V-\frac{2}{R}\cdot V\right)\cdot R=\frac{-\left(\frac{1}{R}+\frac{1}{R}\cdot V\right)}{\left(\frac{-2}{R}-\frac{1}{P}\right)}\ \text{solve},V\ \rightarrow \frac{-\left[\frac{-R}{S}+\frac{1}{\left[\left(\frac{-2}{R}-\frac{1}{P}\right)\cdot R\right]}\right]}{\left[\frac{R}{S}+2+\frac{1}{\left[\left(\frac{-2}{R}-\frac{1}{P}\right)\cdot R\right]}\right]}$$

which simplifies to $V=\dfrac{\left(2\cdot R\cdot P+R^2+P\cdot S\right)}{\left(2\cdot R\cdot P+R^2+3\cdot P\cdot S+2\cdot S\cdot R\right)}$

We now take one of the expressions for E and put V in it to obtain a complete solution.

$$E=\ 2\cdot P\cdot\frac{(R+S)}{\left(2\cdot R\cdot P+R^2+3\cdot P\cdot S+2\cdot S\cdot R\right)}$$

The next chore is to evaluate the input impedance, Zin. We get this by calculating the input current. The input voltage is set at 1, so Zin=1/ i-in.

$$i\ in=\ \frac{(1-V)}{S}+\frac{(1-E)}{R}\qquad \text{So } Z=\ \frac{1}{\left[\frac{(1-V)}{S}+\frac{(1-E)}{R}\right]}$$

which simplifies to $Z\ in=\ \dfrac{-1}{(-R+R\cdot V-S+S\cdot E)}\cdot S\cdot R$

But to get a meaningful expression, we must sub V and E into this. When we do so, we get

Figure 2

$$Z\,\mathrm{in} = \frac{\left(2\cdot R\cdot P + R^2 + 3\cdot P\cdot S + 2\cdot S\cdot R\right)}{\left(2\cdot R\cdot P + 3\cdot R^2 + P\cdot S + 2\cdot S\cdot R\right)}\cdot R$$

This is now a useful result. We can, for example, let S, P, and R be resistor values. We can then design bridge T attenuators. But let's continue with the analysis. Let's ask the question "What are the conditions that will provide an input impedance of R?" We merely set the Zin expression equal to R, and then solve that equation for S or P. We will solve for P.

$$\frac{\left(2\cdot R\cdot P + R^2 + 3\cdot P\cdot S + 2\cdot S\cdot R\right)}{\left(2\cdot R\cdot P + 3\cdot R^2 + P\cdot S + 2\cdot S\cdot R\right)}\cdot R = R \text{ solve}, P \rightarrow \frac{R^2}{S}$$

Hey, that's a cool result, and it's even rather simple. This is an expression for P in terms of S. If we use it for attenuators, it says that our pad is matched with these conditions. We can combine these results with the equation for V to get the attenuation. A couple of interesting matched pads are shown:

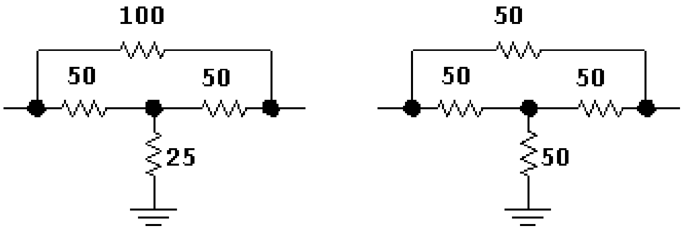


Figure 3

The expression for V was $\frac{\left(2\cdot R\cdot P + R^2 + P\cdot S\right)}{\left(2\cdot R\cdot P + R^2 + 3\cdot P\cdot S + 2\cdot S\cdot R\right)}$ But the conditions for a match were

$P = \frac{R^2}{S}$ So, we put this in for P.

$$\frac{\left(2\cdot R\cdot P + R^2 + P\cdot S\right)}{\left(2\cdot R\cdot P + R^2 + 3\cdot P\cdot S + 2\cdot S\cdot R\right)} \text{ substitute}, P = \frac{R^2}{S} \rightarrow \frac{\left(2\cdot\frac{R^3}{S} + 2\cdot R^2\right)}{\left(2\cdot\frac{R^3}{S} + 4\cdot R^2 + 2\cdot S\cdot R\right)}$$

which simplifies to $V = \frac{R}{(R+S)}$ So the two pads have voltage outputs of 0.5 and 0.333 and the resulting attenuations are 6 and 9.54 dB.

But enough about the attenuators. Let's get back to the original question, that of the diplexer. The circuit of interest is one where the series network is a series tuned circuit while the parallel one is a parallel tuned circuit. But we want to know the conditions that will generate a perfect match. With such a circuit, it is clear that a match will exist at resonance so long as both circuits are resonant at the same frequency. The impedance of the series tuned circuit goes to zero while that of the parallel TC becomes infinite (neglecting loss.)

We write the susceptance (imaginary part of admittance) directly for the parallel tuned circuit as

$B_p = \omega \cdot C_p - \dfrac{1}{\omega \cdot L_p}$ But the reactance is the recipricol of this, $\dfrac{1}{\left(\omega \cdot C_p - \dfrac{1}{\omega \cdot L_p}\right)}$

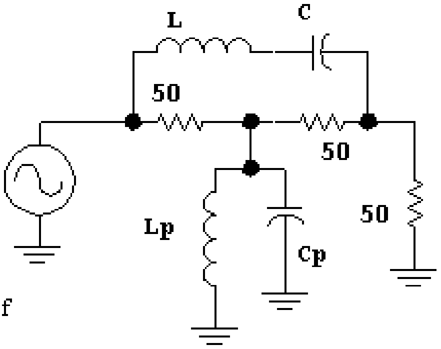
which simplifies to become $\dfrac{1}{\left(\omega^2 \cdot L_p \cdot C_p - 1\right)} \cdot \omega \cdot L_p$

We know that this is resonant at the same frequency as the series tuned circuit with L and C. So, $L \cdot C = L_p \cdot C_p$ and $L_p = \dfrac{L \cdot C}{C_p}$ We substitute these in the above expression:

$X_p = \dfrac{1}{\left[\omega^2 \cdot (L \cdot C) - 1\right]} \cdot \omega \cdot \dfrac{L \cdot C}{C_p}$ But earlier analysis showed us that this reactance was related to the series reactance:

$P = \dfrac{R^2}{S}$ with $S = \omega \cdot L - \dfrac{1}{\omega \cdot C}$ This lets us set up an equation that is then solved for Cp.

$$\frac{1}{\left[\omega^2 \cdot (L \cdot C) - 1\right]} \cdot \omega \cdot \frac{L \cdot C}{C_p} = \frac{R^2}{\left(\omega \cdot L - \dfrac{1}{\omega \cdot C}\right)} \text{ solve}, C_p \rightarrow \frac{L}{R^2}$$



This is a particularly simple, yet useful result

1) Pick a crossover frequency f, Hz. Then, $\omega = 2 \cdot \pi \cdot f$

2) Pick a series inductor L. Use one with a reactance of around 50 Ohms if you want a very broad response. Make it larger for narrower bandwidth.

3) Resonate the series L with a cap, $C = \dfrac{1}{\omega^2 \cdot L}$

4) Pick a parallel cap $C_p = \dfrac{L}{R^2}$

5) Resonate that cap with a parallel inductor $L_p = \dfrac{1}{\omega^2 \cdot C_p}$

Figure 4

Amateur and Short Wave Radio Electronics Experimenter's Web Site

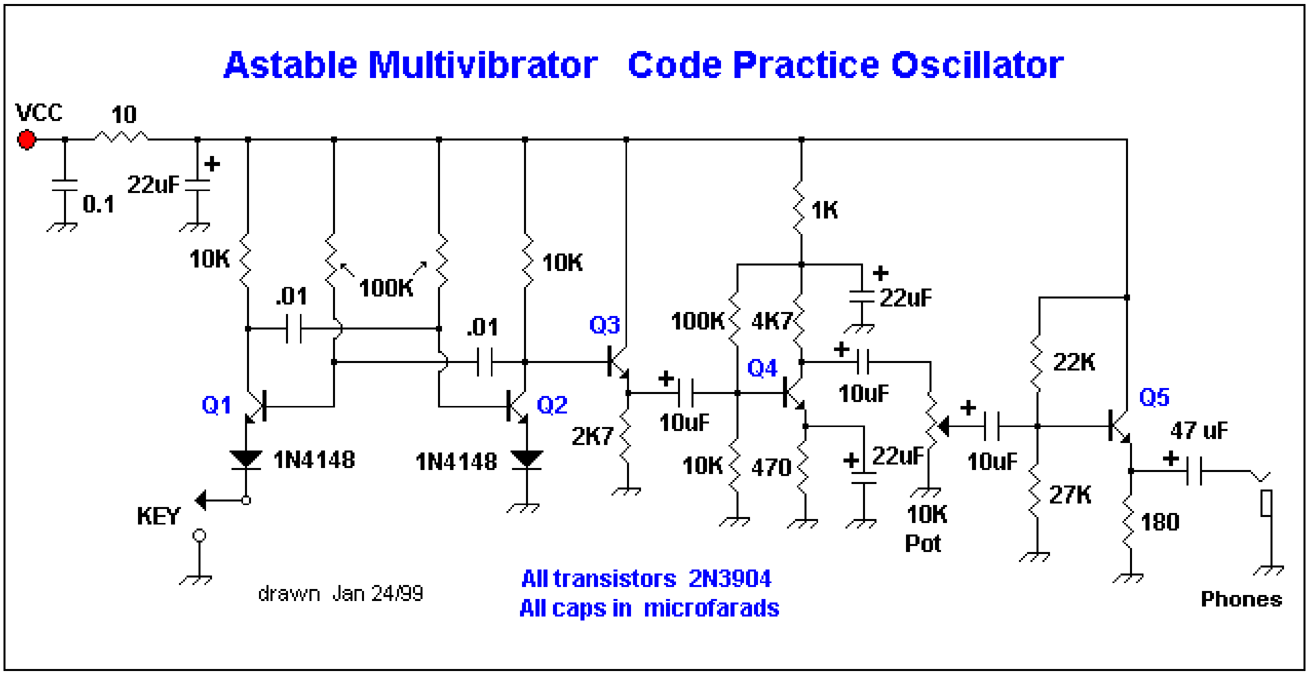
Astable Multivibrator Projects

Discussion:

Astable or free-running multivibrators have been used in home-built amateur radio equipment for many years. The basic circuit is a two stage amplifier with AC-coupled feedback from output to input. One transistor stage is on (conducting current) while the other is off (not conducting current) until the stages switch conducting states repeatedly at a specific frequency. The oscillation frequency is set by the resistor and capacitor values connected to the base terminal of each stage. This RC network determines how long the transistor stays in the off position.

Presented are two projects which utilize astable multivibrators built using the ubiquitous 2N3904 BJT. The first project is a code practice audio-frequency oscillator while the second is a simple , no-frills electronic keyer for keying a transmitter. Either circuit would be a great first project to learn how to build circuits using Ugly Construction.

Code Practice Oscillator



Above is the schematic for a simple Morse code practice oscillator. This circuit was originally built with 2N3904 transistors, however many different NPN transistors could be substituted as required.

Tracing the circuit from left to right first brings us to the multivibrator circuit which is composed of Q1 and Q2. The oscillation frequency of the multivibrator is ~ 700 hertz and is set by the RC network formed by the 100K resistor and the 0.01 uF capacitor connected to each transistor base terminal. The approximate time off for each transistor is given by the following formula:

[Time 0ff = 0.7 * R * C] with R in ohms and C in farads.

It maybe more practical to leave the resistance value fixed and vary the capacitor value to obtain a desired oscillation frequency. Rearranging the above formula allows this :

[Total Time Off = 1 / Frequency] with Total Time Off being the total number of seconds that both transistors are off and Frequency is in hertz.

Once the total time off is known, you must divide that answer by 2 as each transistor is off half of the total time off in this symmetrical circuit. Then you simply solve for the capacitor value:

[Time Half = Total Time Off / 2]
[Capacitor = Time Half / (0.7 * R)] with Capacitor answer in farads.

Lets run the numbers to solve for the capacitor values in the schematic;
R = 100K, desired frequency = 700 hertz.
Total Time Off = 1 / F ----> 1 / 700 = 0.00143 seconds.
Time Half = Total Time Off / 2 ----> 0.00143 / 2 = 0.00071 seconds.
Capacitor = Time Half / (0.7 * R) ----> 0.00071 / (0.7 * 100000) = 0.0000001 farads = 0.01 microfarads.

For 600 hertz, the capacitors would be 0.012 uF and for 400 hertz, 0.018 uF.

As you can see it is maybe necessary to adjust the base resistor value to achieve a specific oscillation frequency. The rule of thumb is that the base resistor should be ~ten times the value of the collector resistor assuming the base-bias resistor is connected to VCC as shown.

The output of the multivibrator is buffered by the high input impedance of an emitter follower Q3. This serves to prevent the oscillation frequency from changing when the output load is changed. The AF stage connected to the emitter-follower is a standard high gain common-emitter amp that has been used in many of the projects on this web site. As the multivibrator is buffered by 2 amplifier stages, good frequency stability is realized and frequency changes are negligible when turning the volume control pot. The final stage is a common-collector amplifier which can drive low impedance headphones with reasonable volume.

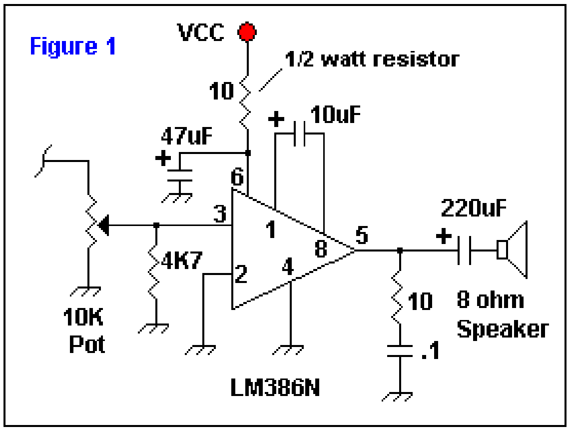
Code Practice Oscillator Project Notes

The voltage / time output waveform of the astable multivibrator is largely a square wave which some people find harsh. Many users prefer listening to a sine wave although that is beyond the scope of this web page. The multivibrator shown has real advantages in that, it is both dependable and tolerant with respect to parts substitutions. Keying the oscillator as shown practically guarantees that the multivibrator will start running each time you hit the key.

VCC can be 9 to 13.8 volts DC and the larger the voltage the greater the volume in the headphones. The B+ decoupling circuit at the top left can be omitted for battery operation or if you prefer not to bother with it. The basic multivibrator and emitter-follower circuit can be used in a transceiver as a sidetone for monitoring keying. A series resistor from the emitter-follower maybe necessary to attenuate/match the sidetone to the transceiver's AF amp.

If you desire speaker level output, the Q5 common-collector final can be omitted and the circuit shown below used. This circuit uses the LM386N and provides up to ~ 0.5 watts into an 8 ohm speaker. Connect the Figure 1 circuit as shown to the 10 uF coupling capacitor connected to the collector of Q4. Do not connect the 10 ohm half-watt resistor to the decoupled VCC shown in the schematic. The power supply to the LM386N AF amp should be directly connected to the VCC, not like the common-collector AF stage shown in the schematic to the right.

Figure 1

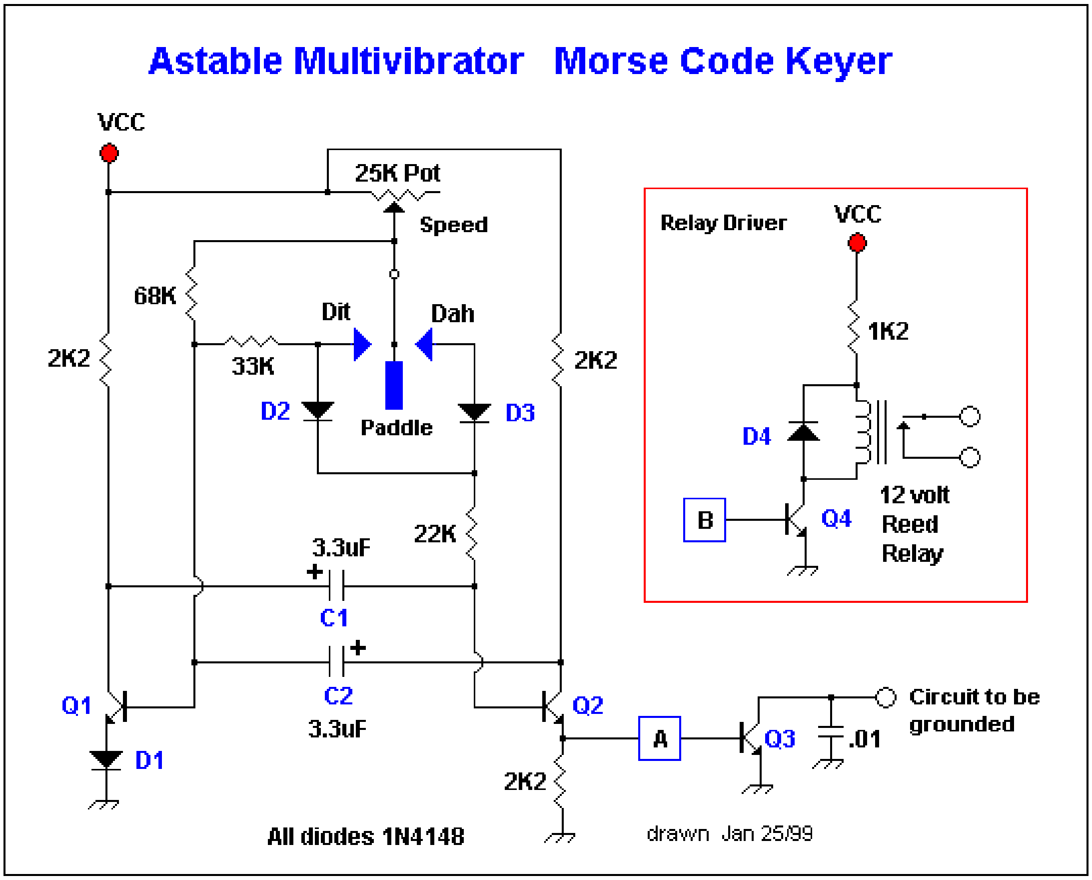


Simple Electronic Keyer

A simple keyer suitable for a popcorn QRP transmitter can be built around an astable multivibrator and an example of such is shown above. The basic design of this keyer is from notes, however the original author of the circuit is unknown. The notes were written in 1973 . I modernized the circuit, added a variable speed control and designed an additional output driver stage.

Keyer Operation

Keying this circuit generates either dits and spaces or dahs and spaces. When the keyer is idle, Q1 is on and Q2 and Q3 are switched off. When the user sends code, Q1 turns off and Q2 and Q3 switch on and in



turn key any device appropriately connected to the collector of Q3. The off-time of Q1 sets the on-time of Q2 and Q3. The off-time of Q2 and Q3 is set by the 22K Q2 base resistor. This off-time is the set time of the spaces and is constant.

The 68K resistor on the base of Q1 is about three times the resistance of the 22K base resistor on Q2 and consequently dahs are ~ three times the length in duration than dits. Spaces and dits are of the same length of time because when sending dits, the 68K base resistor is paralleled with the 33K resistor and effectively the resistance is ~22K ohms. If the optional relay driver transistor Q4 is used instead of Q3, the theory is the same, just substitute Q4 wherever you see Q3.

Keyer Speed

The keyer speed is very sensitive to the power supply voltage and any keyer speeds mentioned are ball-park values. Your results may vary depending on your VCC and component tolerances. If the 25K speed control pot is turned to minimum resistance, the actual power supply voltage will be present on both the paddle common and the top end of the 68K base resistor. This will be the maximum speed for the keyer. In fact, the speed control pot could be omitted if you want to economize and the keyer will run at the maximum speed as determined by C1 and C2. If you do not want the speed control feature, connect the paddle common and the 68K resistor to the VCC supply. Another alternative is to build a two speed keyer by using a switch to switch in or out a fixed resistor to vary the voltage instead of using a potentiometer. A trimmer resistor may also be used for "lid-off" speed adjustments.

Varying the base-bias voltage with a pot changes the charging rate on capacitors C1 and C2. Although, I experienced no problems be careful with some resistances/VCCs as the circuit may be unable to provide enough current to saturate the transistors when the base-bias voltage is at its minimum setting (pot set to maximum resistance). Smaller pots such as 10K can also be used with a more limited ability to reduce the keyer speed below the maximum rate.

To set the maximum rate for the keyer, it is necessary to vary the value of C1 and C2. For this circuit to function correctly, C1 must equal C2. An experiment was conducted with the 25K pot removed and the paddle common and the supply end of the 68K resistor connected to the main B+ terminal. VCC was measured at 13.8 volts. C1 = C2. The words per minute were counted for four different standard capacitance values and the results were as follows:

2.2 uF = 27 WPM
3.3 uF = 23 WPM
4.7 uF = 17 WPM
10 uF = 9 WPM

For this project, I settled with the 3.3 uF value, although personally, I use a 2.2 uF capacitor for C1 and C2. Sending speed can be reduced with the speed control pot or by increasing the time interval between characters and words. With 3.3 uf caps for C1 and C2, turning the 25K speed control pot to maximum resistance dropped the sending rate down to 12 words per minute. If you needed to slow down below 12 WPM, an amateur could send code using the Farnsworth method as mentioned above. At any rate this circuit allows you to determine the maximum speed rate by choosing the C1 and C2 capacitance value to suit your needs.

Q3, Q4 Output Stages

Two different output stages maybe used with this keyer and they will be referred to as the Q3 or Q4 stage.The Q3 stage is a simple transistor switch which will ground any component(s) connected to its collector when turned on during code sending. A variety of transistors maybe used here and care must be taken to ensure that you do not exceed the maximum dissipation of a given BJT.

The Q4 stage is a relay driver. The 12 volt relay used during bench testing was a Radio Shack reed relay that I had in my parts collection. Specifications were SPST 1A @ 125 VAC , 1050 ohms DC coil resistance, part number 275-233. My VCC was 13.8 volts so a 1K2 current limiting resistor was placed between the relay and supply voltage. The relay has suffered no harm despite significant torture but be careful when you are using a VCC greater than 12 volts. This resistance may be dropped down to 470 ohms or omitted if you are using lower voltages. Any relay with a DC coil resistance of 500 to 3000 ohms should work in this circuit.

I have never used this keyer with the Q4 relay driver for QRP operation as I prefer solid-state switching. The Q3 stage in turn can be connected to trigger a PNP transistor switch to supply DC voltage to a keyed transmitter driver and/or to pull a VFO to its offset frequency. A great example of PNP transistor switches can be found in The Ugly Weekender article by KA7EXM and W7ZOI. This article is referenced in the recommended reading list on this web site. Another bonus of the Q3 stage is that it draws less current than the Q4 stage.

Sending Code

If you are used to a deluxe iambic keyer, this keyer will take some getting used to. The dahs in particular can be problematic as they do not self-complete like they do on my station homebrew TTL iambic keyer. After some practice, however most people should be able to send some good morse code with it despite the lack of iambic luxury.

Miscellaneous

When you power up this keyer, it sends a dit. In one version of this keyer, used in a 40M transceiver, I had a 10K pot for the speed control with a built-in switch on the 10K pot to turn the keyer on or off during station setup.

The diodes D1 - D4 can be any common switching diode such as the 1N914. Q1 to Q4 can be any NPN transistor with a Beta greater than 50 such as the 2N3904 or 2N2222. In the test keyer, all BJT's were 2N3904.

For fun, I connected the multivibrator code practice oscillator to the keyer and gave the keyer a workout. They both work great together and are very complimentary. Have fun with astable multivibrators !

Update Oct 2007:

Giovanni, IW7EHC has posted a great web page on this schematic including PCB layout files in FidoCAD format

Check out his excellent Italian and English language web site

---

Amateur and Short Wave Radio Electronics Experimenter's Web Site

Supplemental Page for the 10 MHz WWV TRF receiver

I have received a tremendous amount of feedback regarding the 10 MHz WWV TRF receiver. Many builders have constructed their own versions and have improved it. This captures the spirit of the QRP/SWL Homebuilder web site; take an idea and evolve it into something better on the bench.

I will post some of the feedback as well as provide additional comments and images regarding the 10 MHz WWV project on this supplemental web page. Please continue to provide project feedback and help me to improve the QRP/SWL HomeBuilder web site.

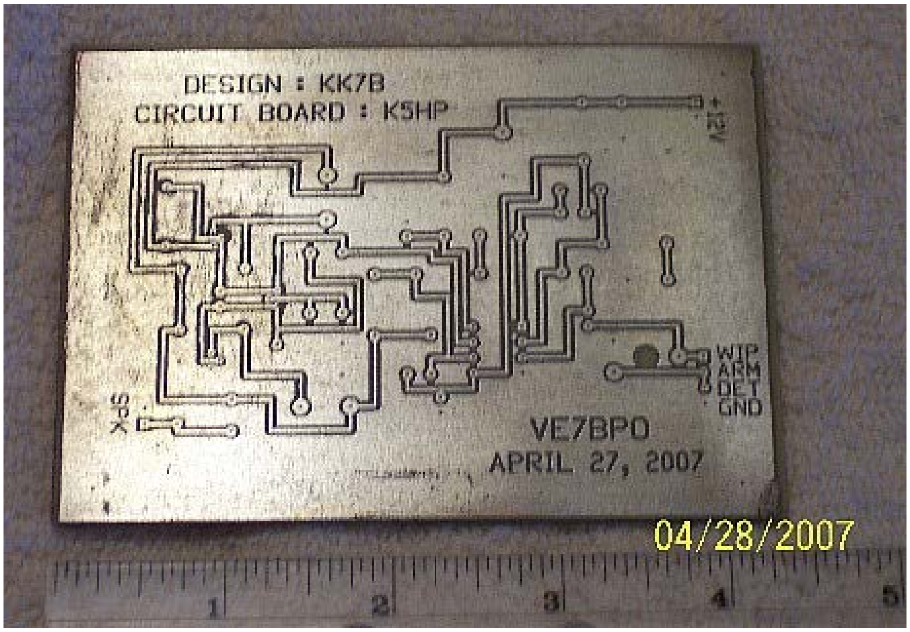
Terry, K5HP
Hi Todd,

I have just finished (few days ago) your design for the 10 MHz WWV Receiver. I reduced you schematic diagrams to CirCad schematics, drug out the rat-net, positioned and connected everything, and etched the boards. I'll include a photo of the 4 finished boards. They are very roomy for extra tinkering. Except for two solder bridges on two very big pads on the tweeker capacitors in the front end, it all worked perfectly first try. Hmmm, I cannot say that for all my projects in the past. After repairing the two shorts to ground of the RF path, the front end then tuned and worked perfectly also.

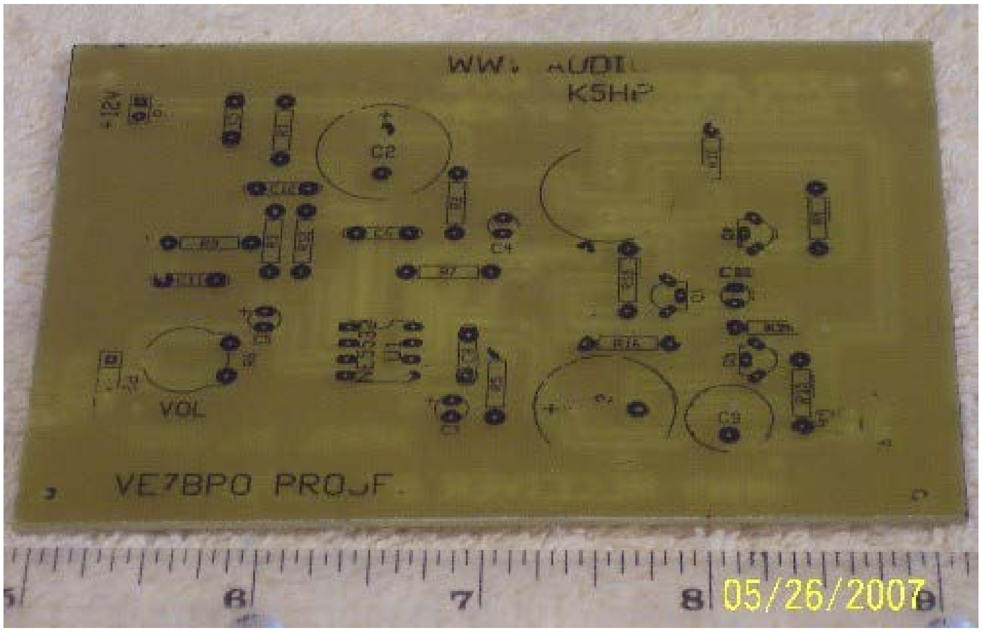
I have not taken the project out to the Lab (barn) and put it on my Motorola Service monitor, but the input to the filter board is roughly 500 uV. It can easily hear WWV at this level with a decent antenna connected to the filter input when WWV is fairly strong at my location. Not bad, considering the mismatch at that location.

Have you or someone else tinkered with an AGC/AVC circuit? At one time I had a nice circuit using an National chip (don't remember the number) which is no longer available that was easy to build and easy to retro-fit into something. It was an AVC which they called an audio AGC.

When I built out the boards I placed proper credit to you and the other developers mentioned on the boards. Should you want my work product it's free for the asking. I'll be glad to send it to you. CirCad in ver 4 is a fairly



Above. Etched Audio amplifier PC board from K5HP.



Above. Audio amplifier PC board overlay for parts placement from K5HP.

complete PCB cad program and it's free too. There is a version 5, but I have not tried it out yet.

My sincere congratulation for producing and error free web article. That is a heck of a more than you can say about many other projects in the past. Most of those in the past had errors a plenty, and some just did not flat work as described. Someone just thought they would as it turned out. Thanks for the fine article and I'll check your site over for more interesting thing to build or learn from. Best wishes and good luck.

"I use the CirCAD to produce the artwork, then take the laser printer output and bond the plastic toner onto the PCB. Works good most of the time. Sure easier than trying to do photo etch, and silk screening is pretty expensive. The bottom copper is viewed as you would look down onto the top of a circuit board, through to the copper side. The side you are seeing is bonded to the copper surface. Print one out, hold it up to a light source looking through non printed side and it becomes apparent."

Above. A layout of the 4 completed boards by K5HP.

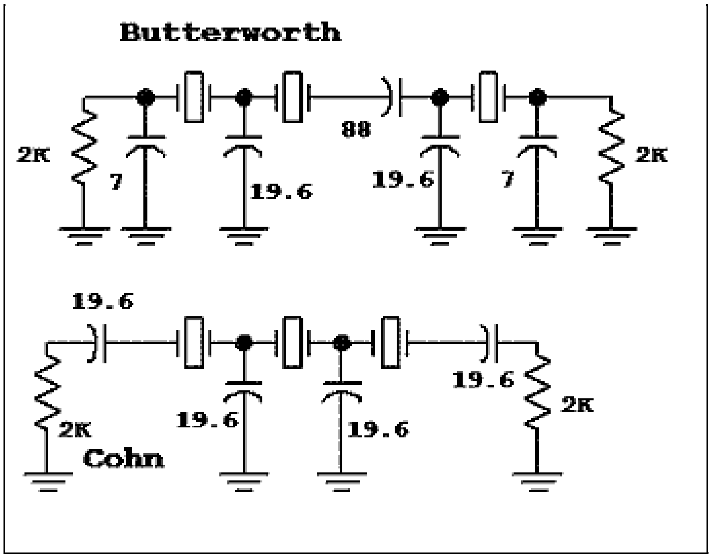
Terry designed and built printed circuit boards. His photographs, etching and overlay files are all bundled together in convenient zipped files which can be downloaded below.

Download the front end zip file

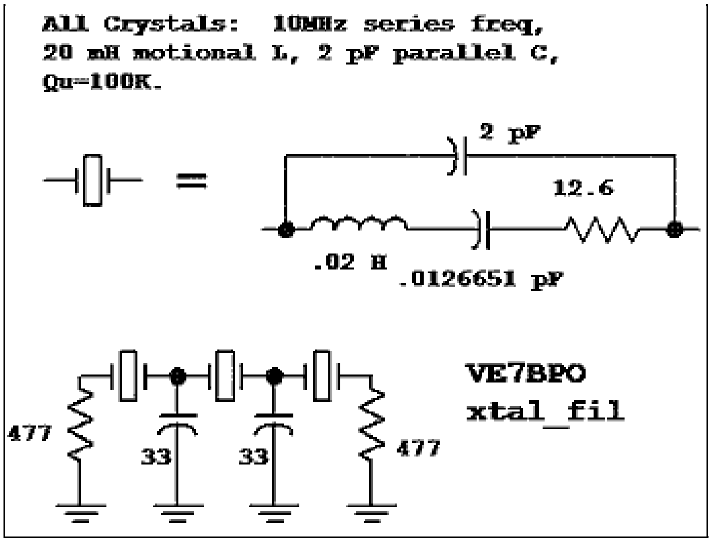
Download the Crystal filter zip file

Download the Detector zip file

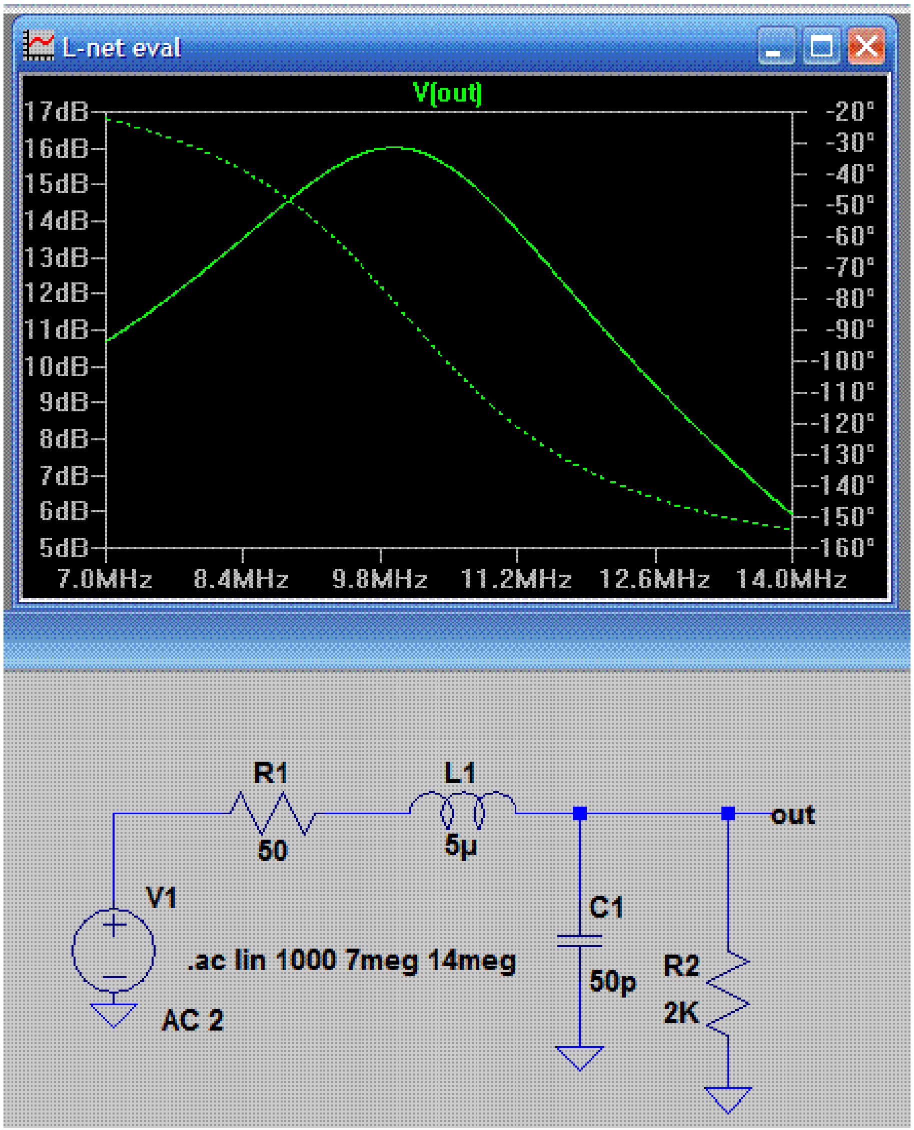
Download the AF amp zip file



Above. Schematics diagrams of 10 MHz crystal filters by Wes, W7ZOI. I did not experiment with the Butterworth type filter.



Above. Schematics diagrams by Wes, W7ZOI.
A schematic representation of the 10 MHz xtal is shown along with the "popcorn" 10 MHz xtal filter I designed.



Above. A simulation of an L network depicted in the L-Match and AM Detector schematic for those interested in driving this detector with a 50 ohm output impedance.

SWL HOMEBUILDER

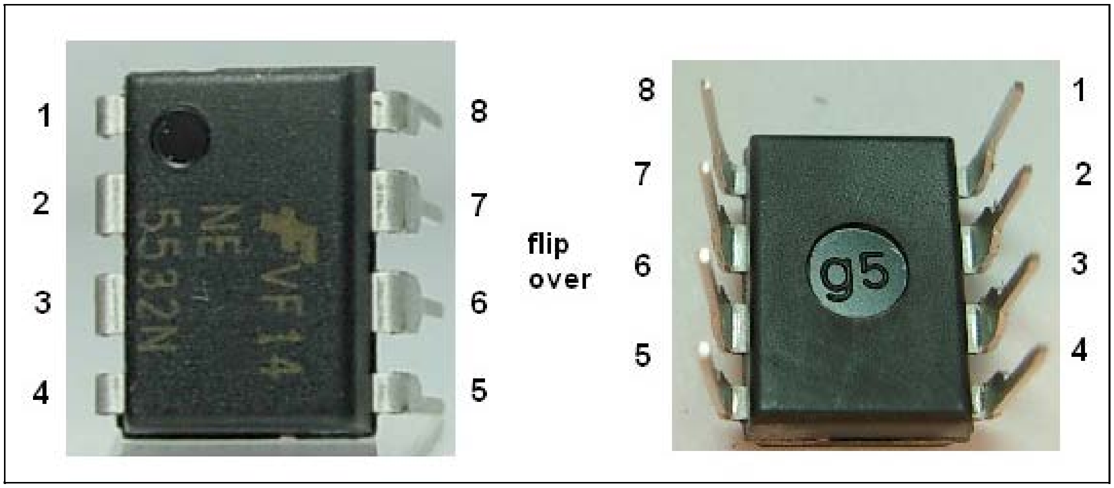
Amateur and Short Wave Radio Electronics Experimenter's Web Site

MF TRF Receiver Supplemental Page

Click here to go to the main project web page This supplemental page is under construction

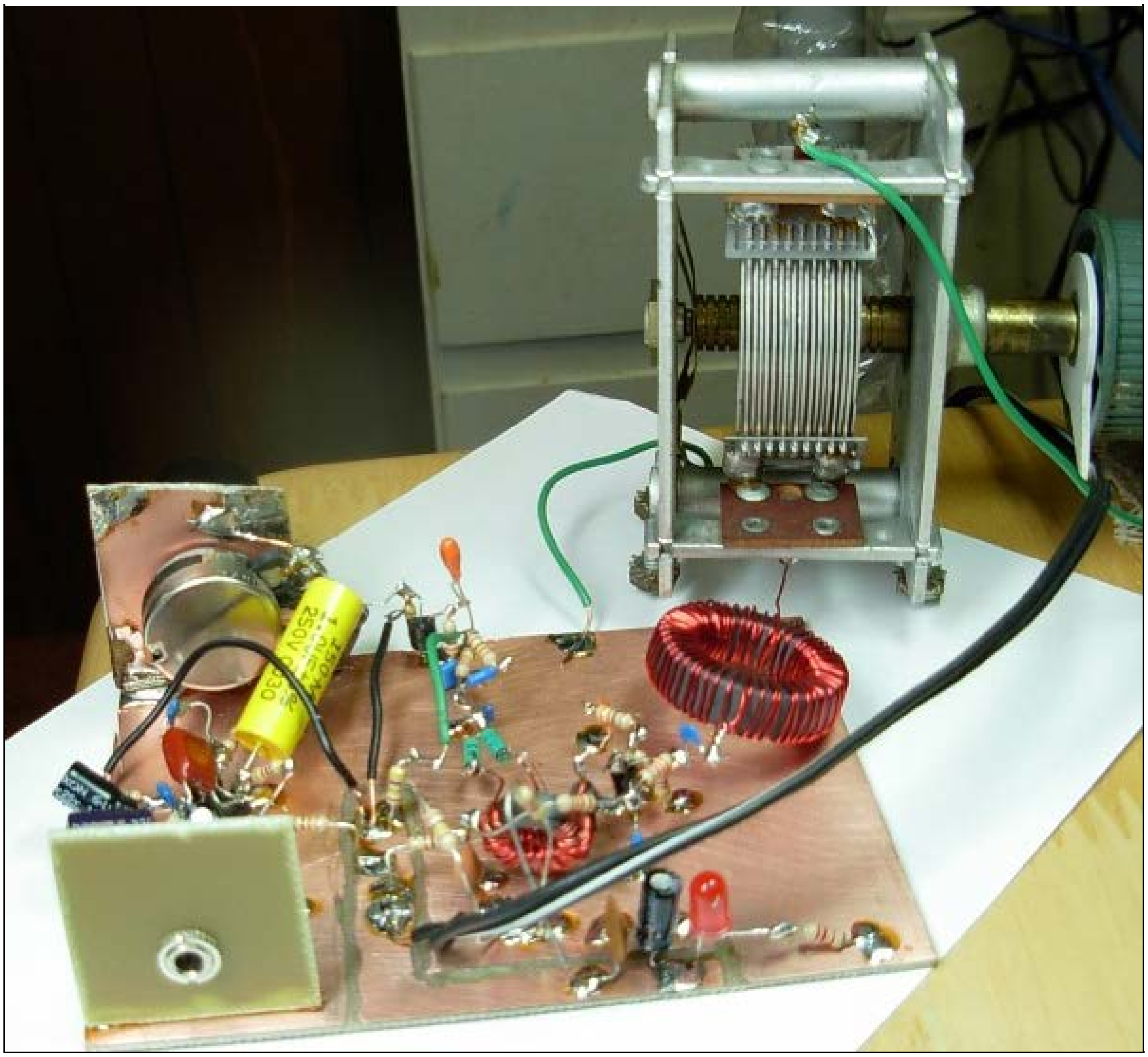
---

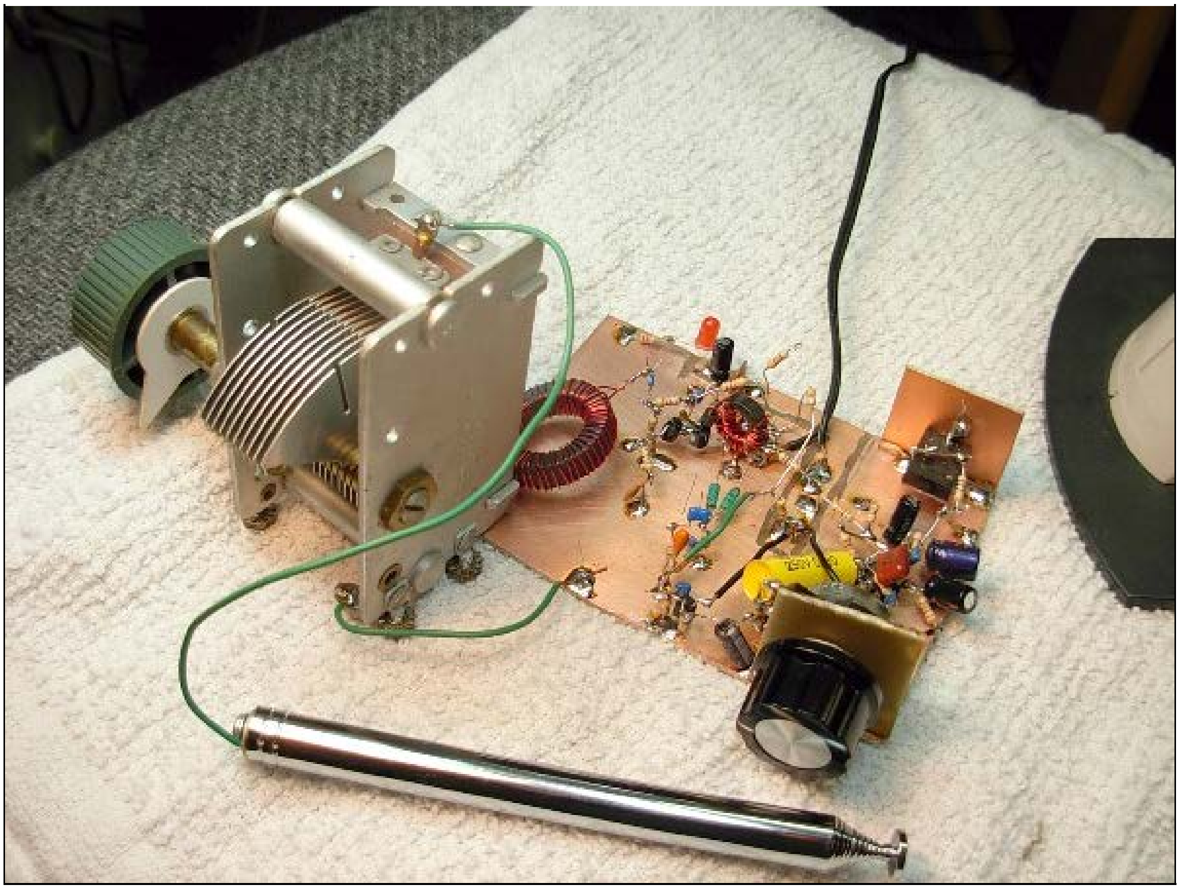
5532 Op amp Pin Out



A Version of the Receiver Using Just L1 and C1 as the band pass filter

You can see a red LED and current limiting resistor in the photos. This is a power on indicator. The telescopic whip antenna can be seen in the lower photo.



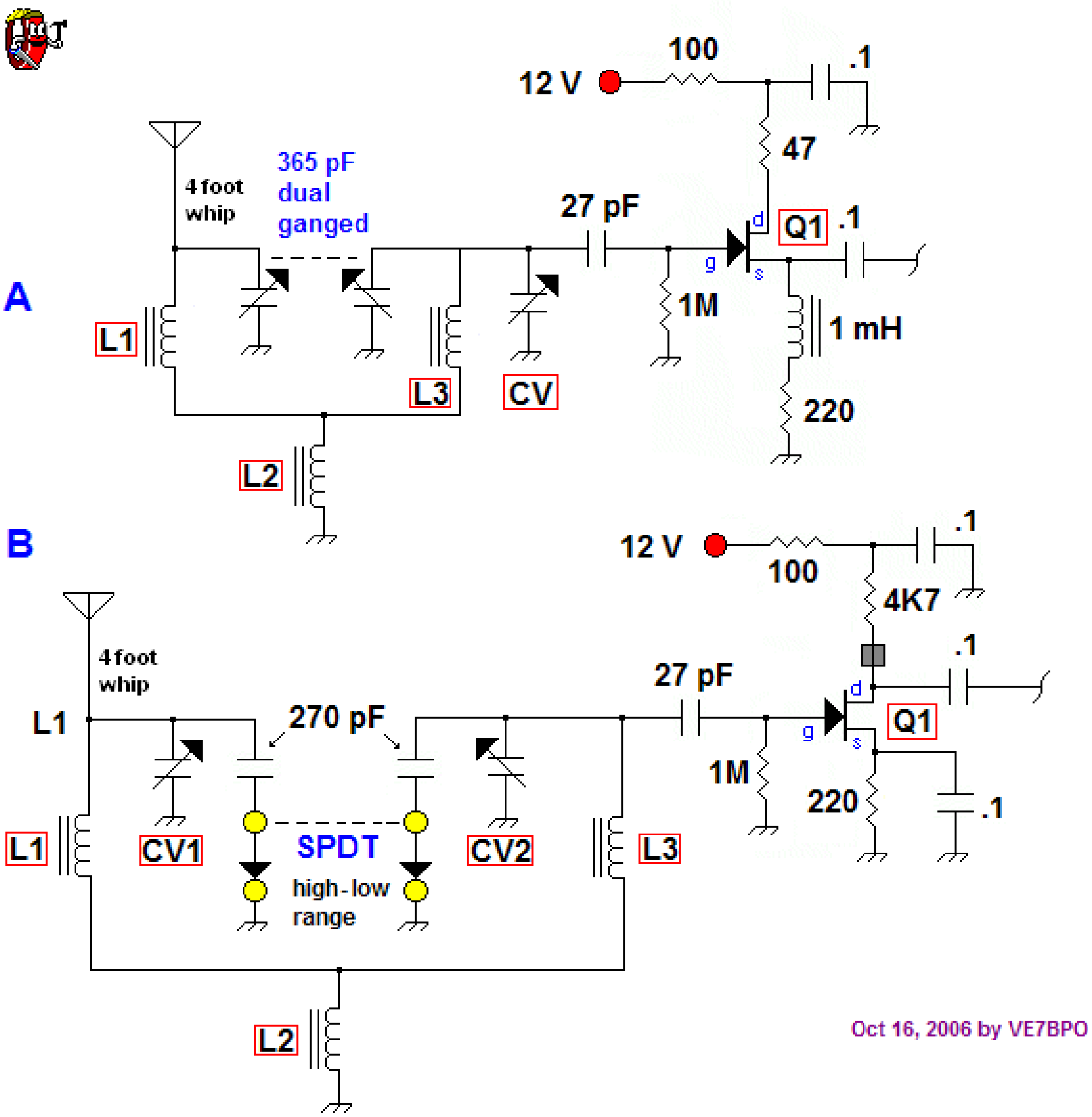


Early Versions of the Project Front End

In section A of the schematic below, a dual-ganged 365 pF air variable capacitor was used for tuning. CV, a small compensating capacitor may be required to adjust for the capacitive differences of the 2 L-C tanks in the front end filter. It may have to be placed on the L1 side of the filter (opposite to what is shown). A JFET source follower was used for Q1 but discarded because stage gain was insufficient with a short whip antenna.

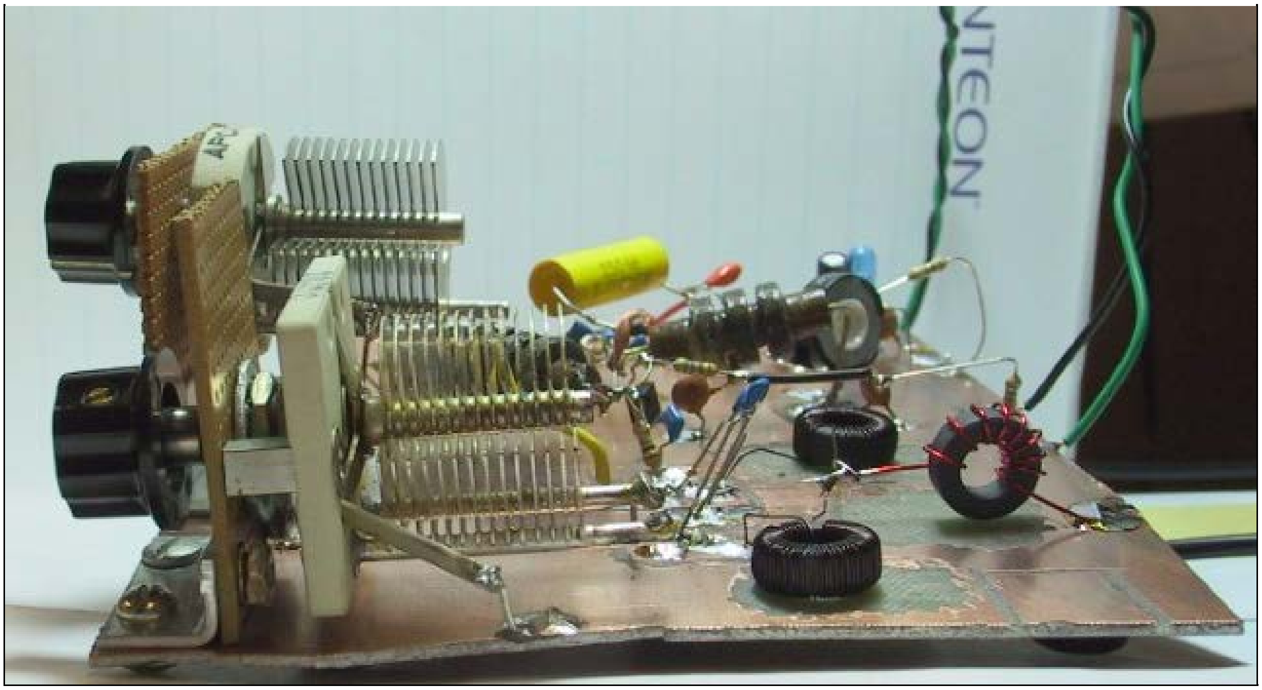
In the version depicted as section B, two 12-110 pF air variable capacitors were used along with switchable 270 pF capacitors to provide 2 ranges of tuning. Although this worked, the switching mechanism was found to degrade selectivity somewhat. The JFET was reconfigured as a common source RF amp. This amp caused some distortion of the RF signal on stronger stations unless the gate resistor was lowered to 47K or less. Ultimately a cascode BJT RF amp was chosen for the project. A photograph of the section A breadboard is shown below the schematic.

Broadcast Radio Band Front End

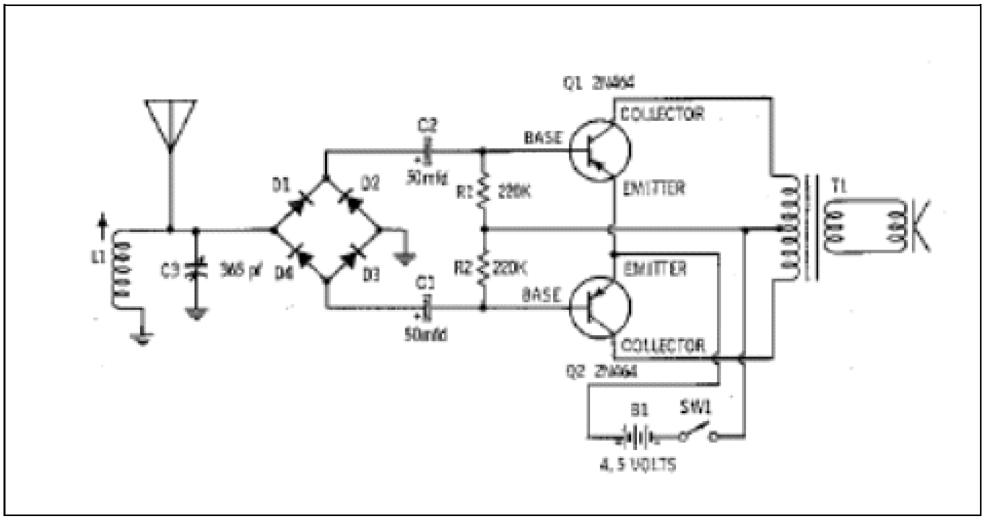


Oct 16, 2006 by VE7BPO

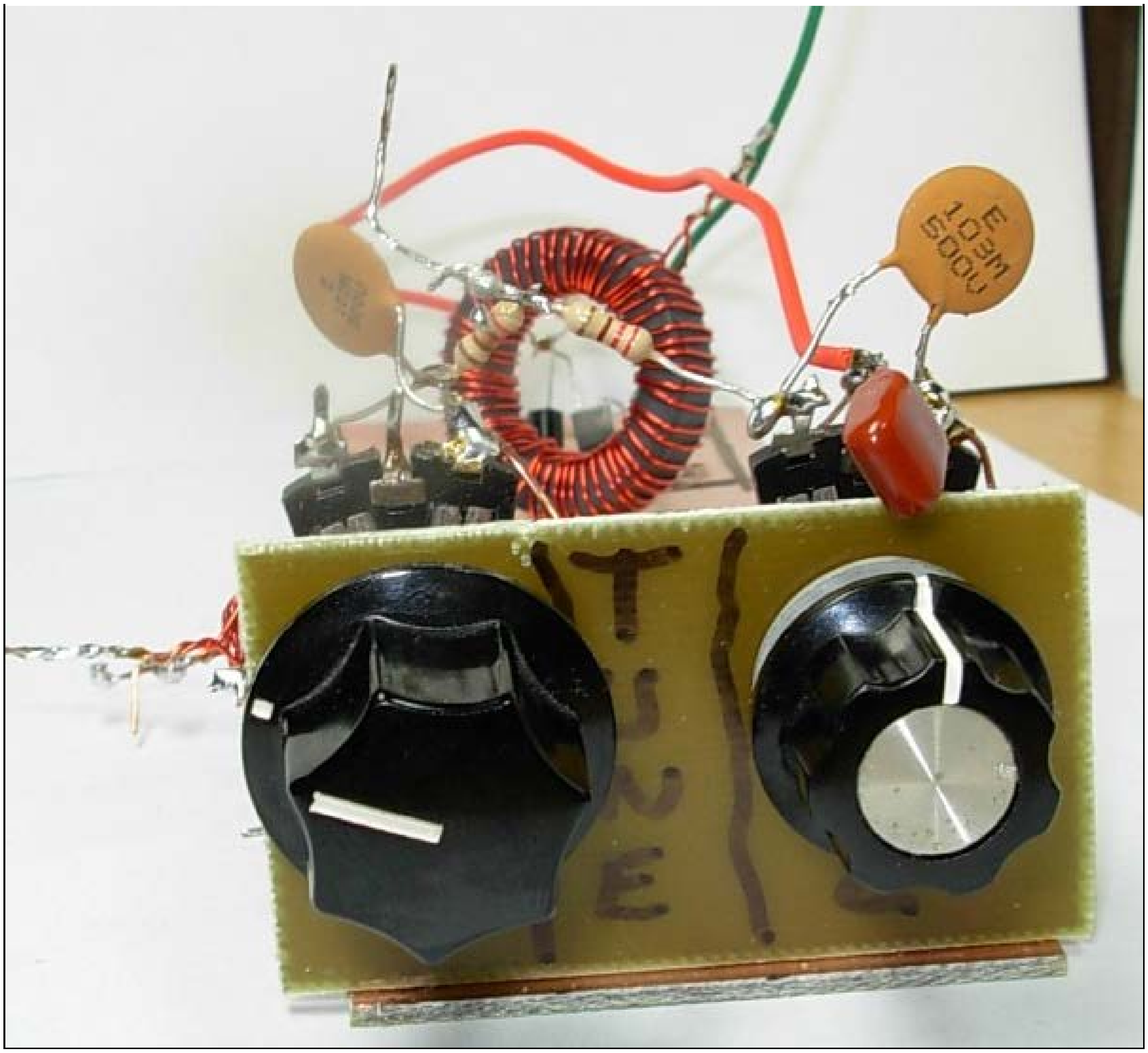
L1 = L3 = 200 uH or 55 turns #28 AWG on a FT50-61 ferrite core
L2 = ~ 5 uH = 9 turns #22 AWG on a FT50-61 ferrite core
CV = small value trimmer capacitor for tracking - 4.5 - 20 pF
CV1 = CV2 = 12 - 110 pF air variable capacitor
Q1 = nJFET; MPF102 or J310



The inspirational detector schematic used for motivation to develop the full wave detector



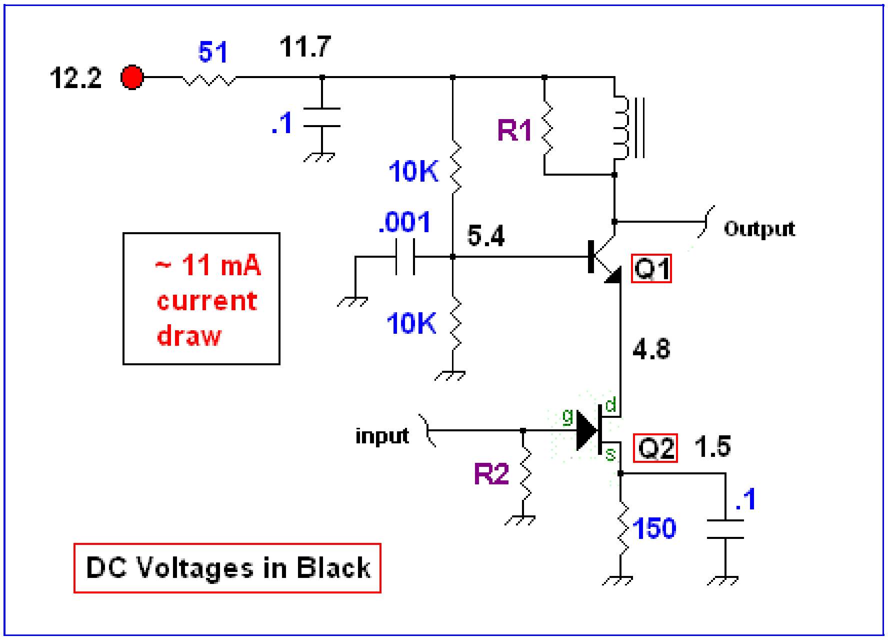
Varactor Tuned Front-end Band pass Filter



Above photo. The 2 potentiometer varactor controlled band pass filter.

Supplemental to the Cascode Hybrid 5 MHz WWV Receiver

DC Voltages of the Cascode Hybrid Amp



On the right is the basic hybrid cascode amplifier schematic. Note that the Q1 voltage divider bias has two 10K resistors as compared to the 12K and 10K combination used in the final project. A slightly lower Q1 bias voltage was ultimately chosen along with a 150 ohm Q2 source resistor to enhance stability. Early bread boards of this amplifier used a 47 - 100 ohms Q2 source resistor along with the slightly higher Q1 bias, but suffered from parasitic oscillations.

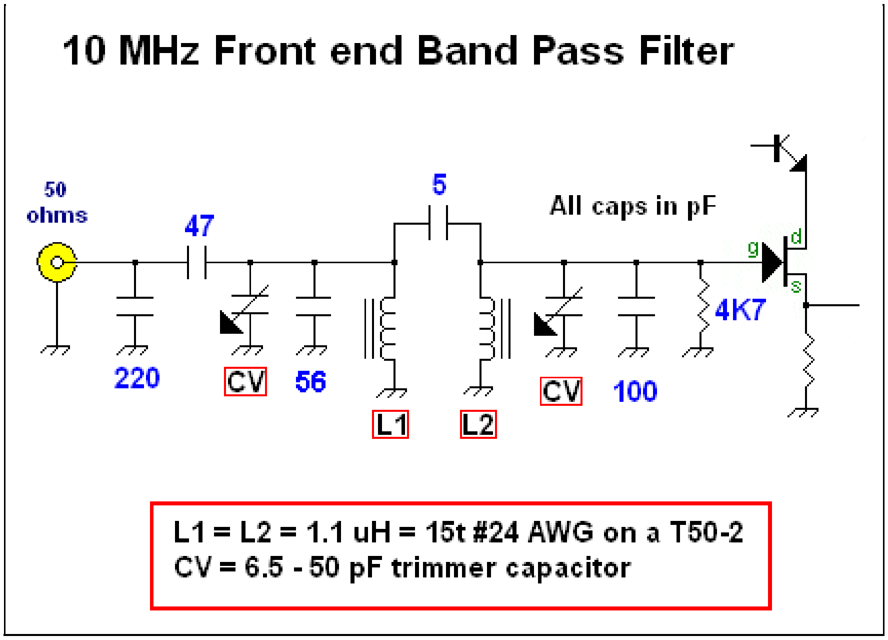
The DC voltages are rounded to 1 decimal point. These are ball park values for reference purposes. The typical stage current draw was around 11 mA. R1 sets the output impedance and is optional. R2 sets the input impedance. This amp is very versatile in that it can be used to match a wide variety of input and output impedances.

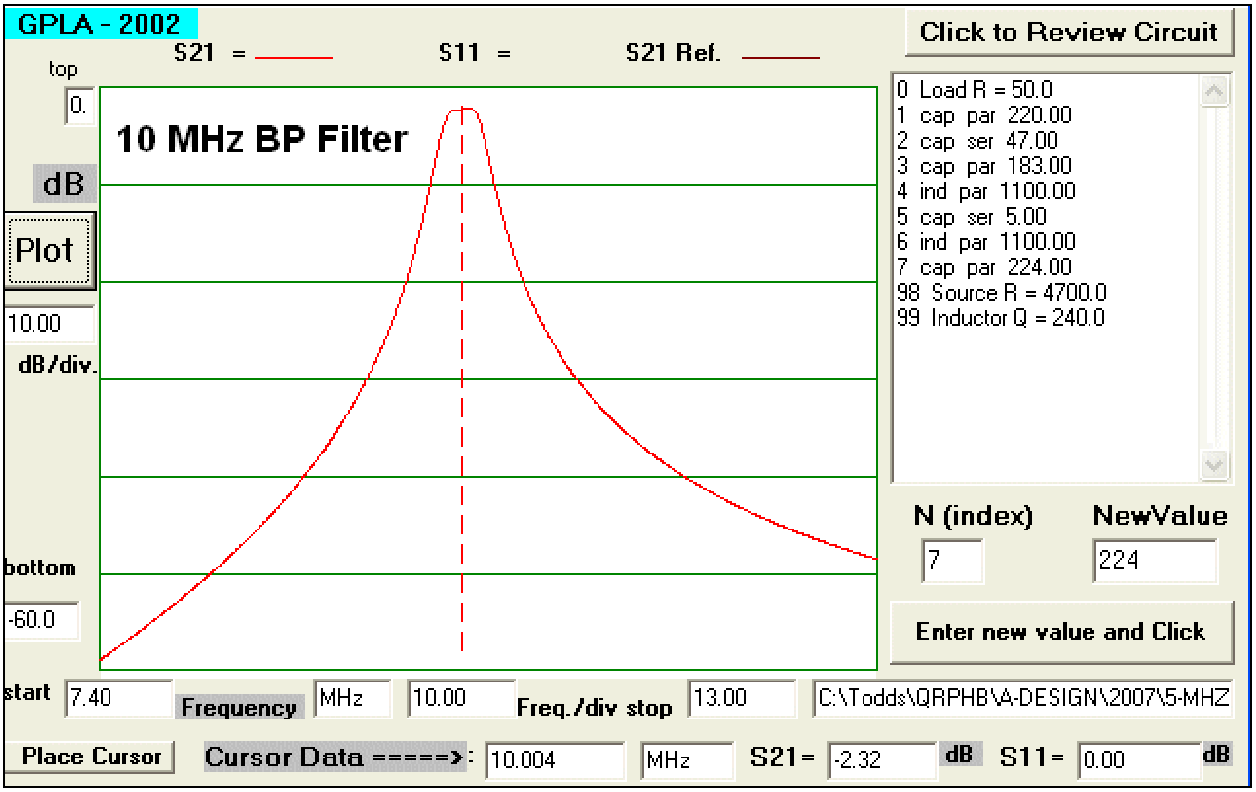
The **main 5 MHz WWV receiver web page** which this web page supplements is here.
Click here for a recording of 5 MHz WWV made with the Icom R-75 receiver using the 6 KHz AM filter and the stock speaker. You may wish to compare it to the recorded 5 MHz WWV receiver project audio files linked here , here and here.

---

10 MHz Components

For those who wish to experimentally develop a 10 MHz version, I have included some potential band pass and crystal filter schematics you may wish to consider. In addition, my preliminary experiments indicated that another RF amplifier stage might be necessary for 10 MHz WWV. During development of these band pass and crystal filters, 10 MHz WWV reception was very poor and due to lack of time and the poor conditions, further experimentation was abandoned.
To the left is a 10 MHz band pass filter schematic with a -3dB bandwidth of around 100 KHz.



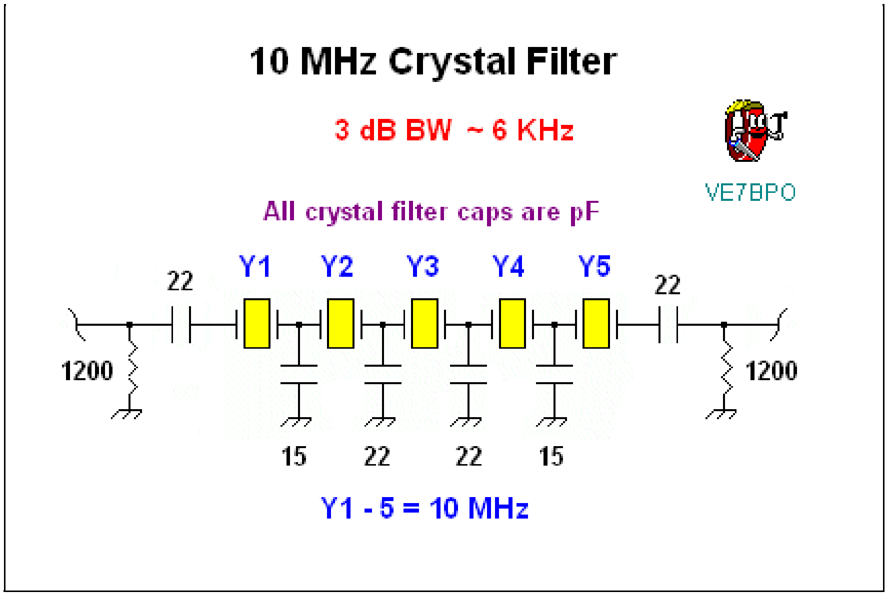


```
0  Load R = 50.0
1  cap  par  220.00
2  cap  ser  47.00
3  cap  par  183.00
4  ind  par  1100.00
5  cap  ser  5.00
6  ind  par  1100.00
7  cap  par  224.00
98  Source R = 4700.0
99  Inductor Q = 240.0
```

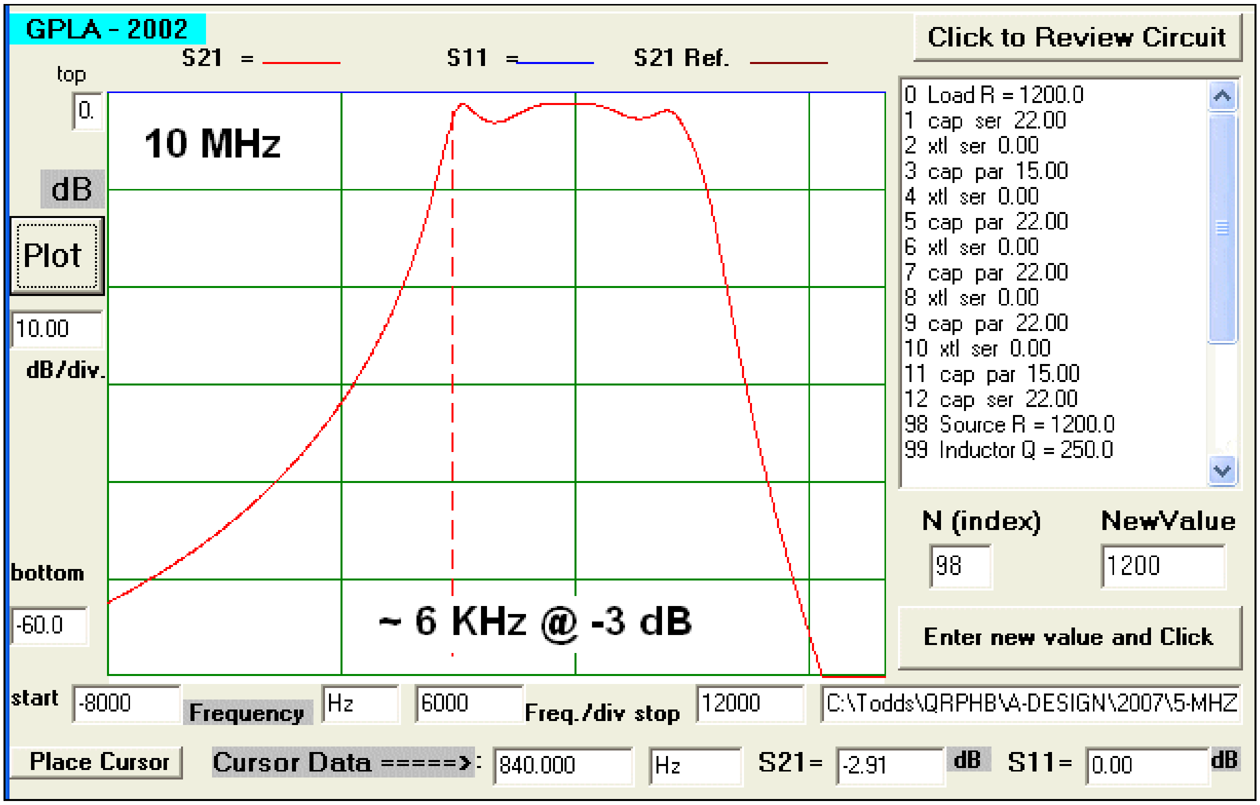
top 0.
10.00 dB/div.
bottom -60.0
N (index) 7
NewValue 224
start 7.40 MHz
Freq./div 10.00
stop 13.00
C:\Todds\QRPHB\A-DESIGN\2007\5-MHZ
Cursor Data =====>: 10.004 MHz S21= -2.32 dB S11= 0.00 dB

The GPLA simulation of the 10 MHz front end band pass filter. Tuning is very sharp with this filter.

To the left is a crystal filter design for 10 MHz. The realized bandwidth of these simple min-loss crystal filters is dependent on your crystal parameters.



A photograph of the experimental 10 MHz crystal filter is shown to the right. Leads were kept long so that parts could be reused in future experiments.



```
0  Load R = 1200.0
1  cap  ser  22.00
2  xtl  ser  0.00
3  cap  par  15.00
4  xtl  ser  0.00
5  cap  par  22.00
6  xtl  ser  0.00
7  cap  par  22.00
8  xtl  ser  0.00
9  cap  par  22.00
10  xtl  ser  0.00
11  cap  par  15.00
12  cap  ser  22.00
98  Source R = 1200.0
99  Inductor Q = 250.0
```

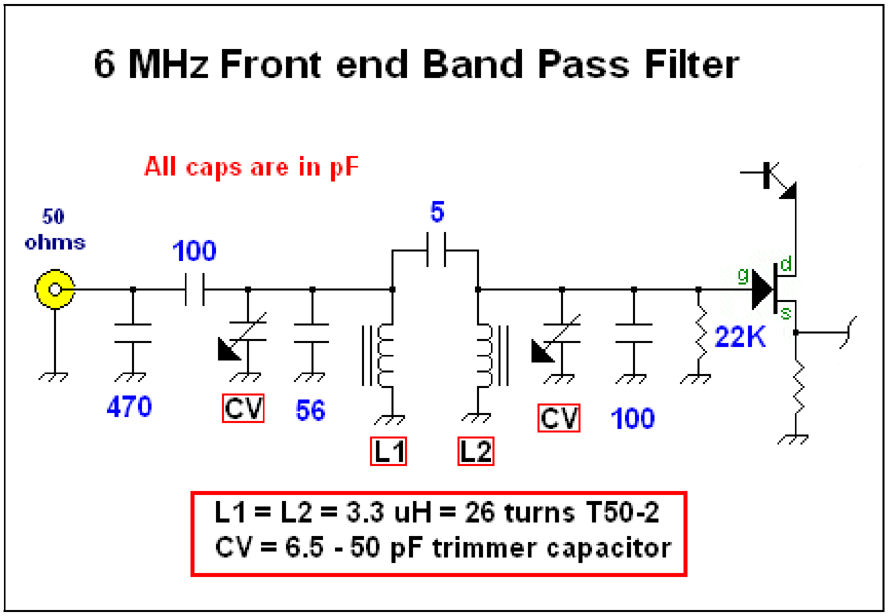
top: 0.
dB/div.: 10.00
bottom: -60.0
N (index): 98
NewValue: 1200
start: -8000 Hz
Freq./div: 6000
stop: 12000
C:\Todds\QRPHB\A-DESIGN\2007\5-MHZ
Cursor Data =====>: 840.000 Hz
S21= -2.91 dB
S11= 0.00 dB

Above. The GPLA simulation of the 10 MHz crystal filter.

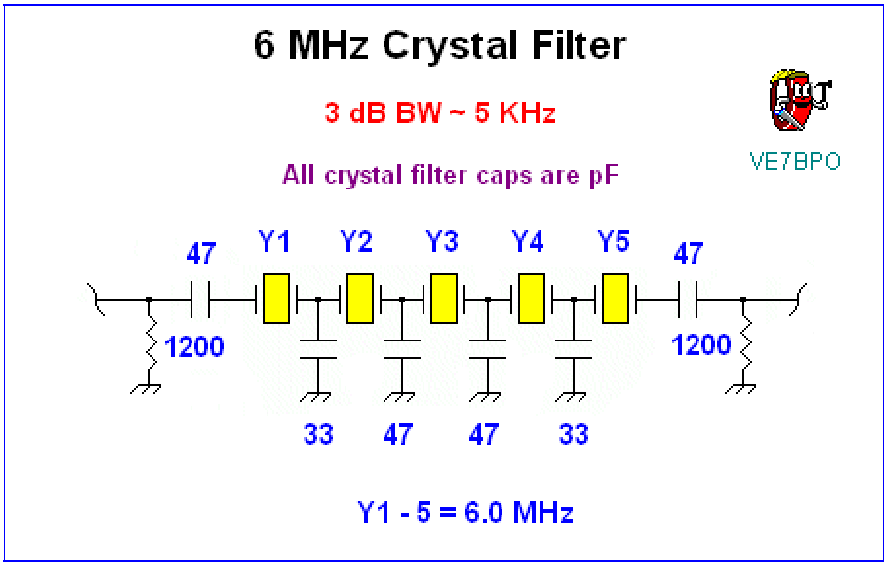
---

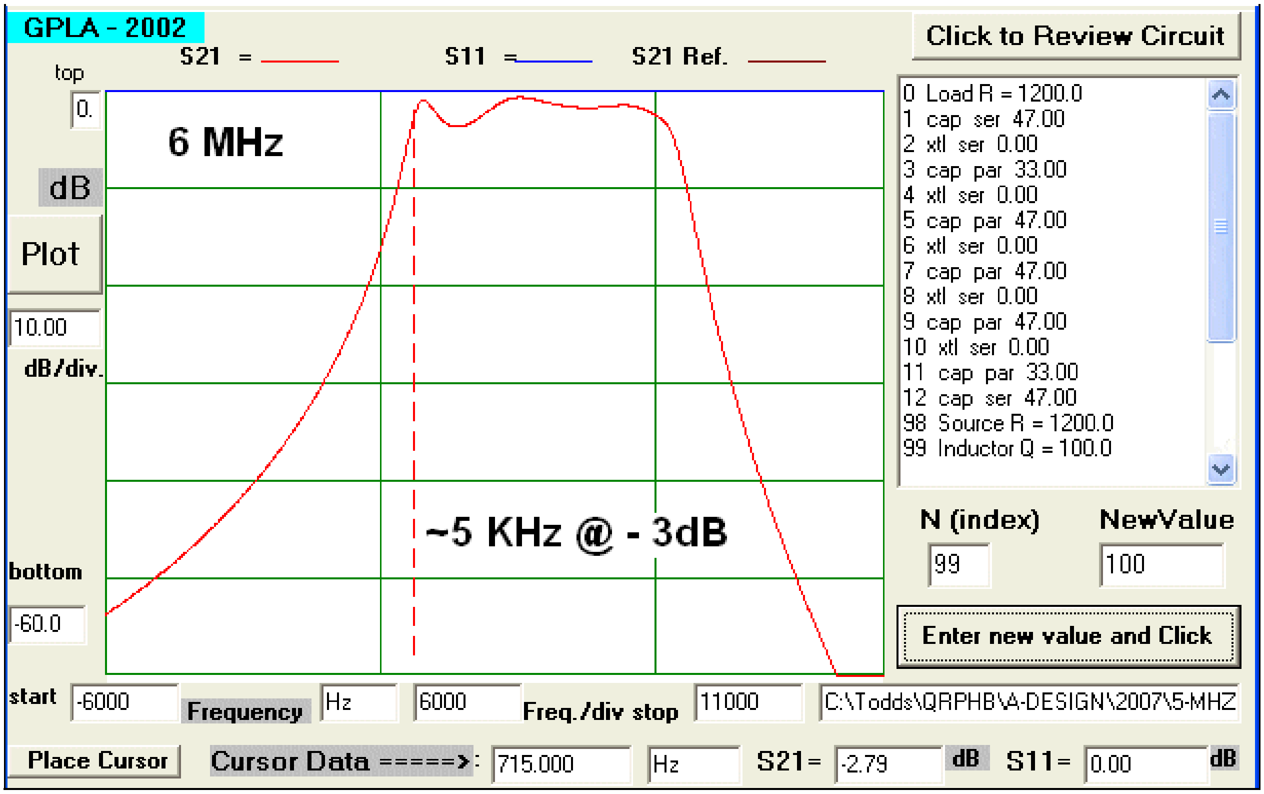
6 MHz Components

Since it is easy to find 6 MHz crystals and Radio Habana Cuba is on 6.0 MHz, band pass and crystal filters were designed for reference purposes. To the right is the front end, band pass filter schematic.



To the left is a potential crystal filter for 6 MHz AM reception. The bandwidth is ~ 5 KHz which is starting to get too narrow for some listeners, however, your crystal parameters will determine your filter's actual bandwidth.



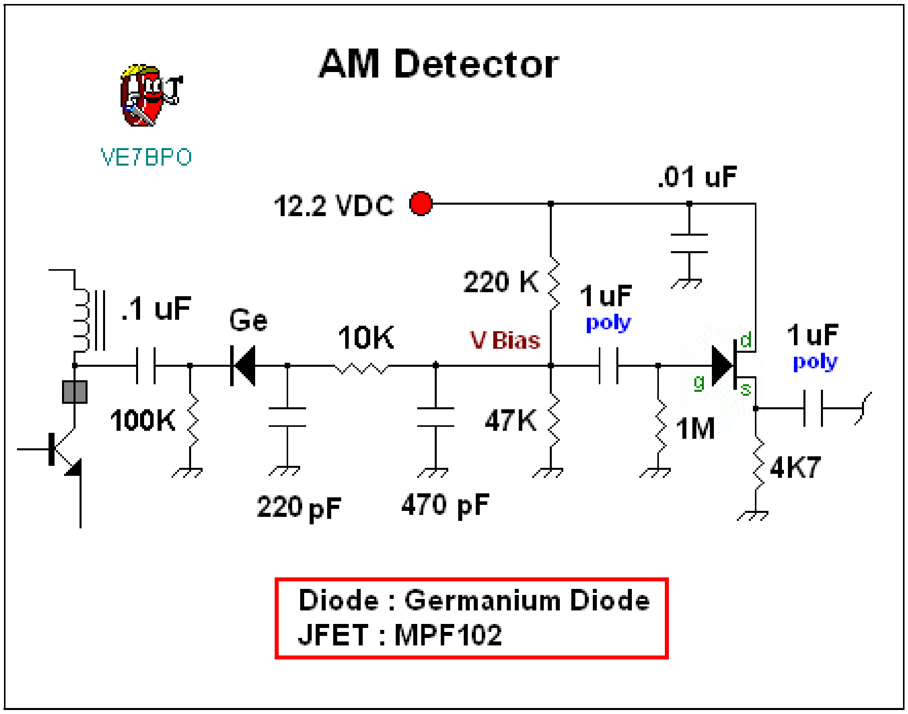


0 Load R = 1200.0
1 cap ser 47.00
2 xtl ser 0.00
3 cap par 33.00
4 xtl ser 0.00
5 cap par 47.00
6 xtl ser 0.00
7 cap par 47.00
8 xtl ser 0.00
9 cap par 47.00
10 xtl ser 0.00
11 cap par 33.00
12 cap ser 47.00
98 Source R = 1200.0
99 Inductor Q = 100.0

N (index): 99
NewValue: 100
top: 0.
dB/div.: 10.00
bottom: -60.0
start: -6000 Hz
Freq./div: 6000
stop: 11000
C:\Todds\QRPHB\A-DESIGN\2007\5-MHZ
Cursor Data: 715.000 Hz
S21= -2.79 dB
S11= 0.00 dB

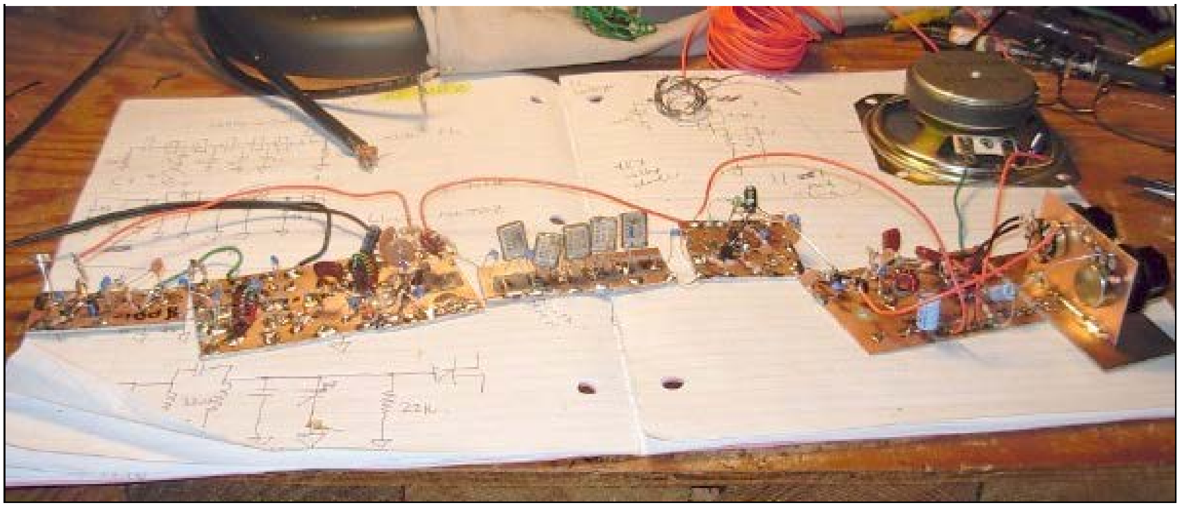
The GPLA simulation of the crystal filter using the parameters from the 6.0 MHz crystals I had in my parts collection.

VE7BPO Envelope Detector



To the left is the schematic of another detector experimentally developed for the 5 MHz WWV receiver. The germanium diode has incredible sensitivity when biased, although noise tends to increase with applied DC voltage. This basic design also worked well with a hot carrier diodes, although had less sensitivity. V bias can be changed by adjusting the 220K, 47K and 100 K resistor values. Note diode polarity.

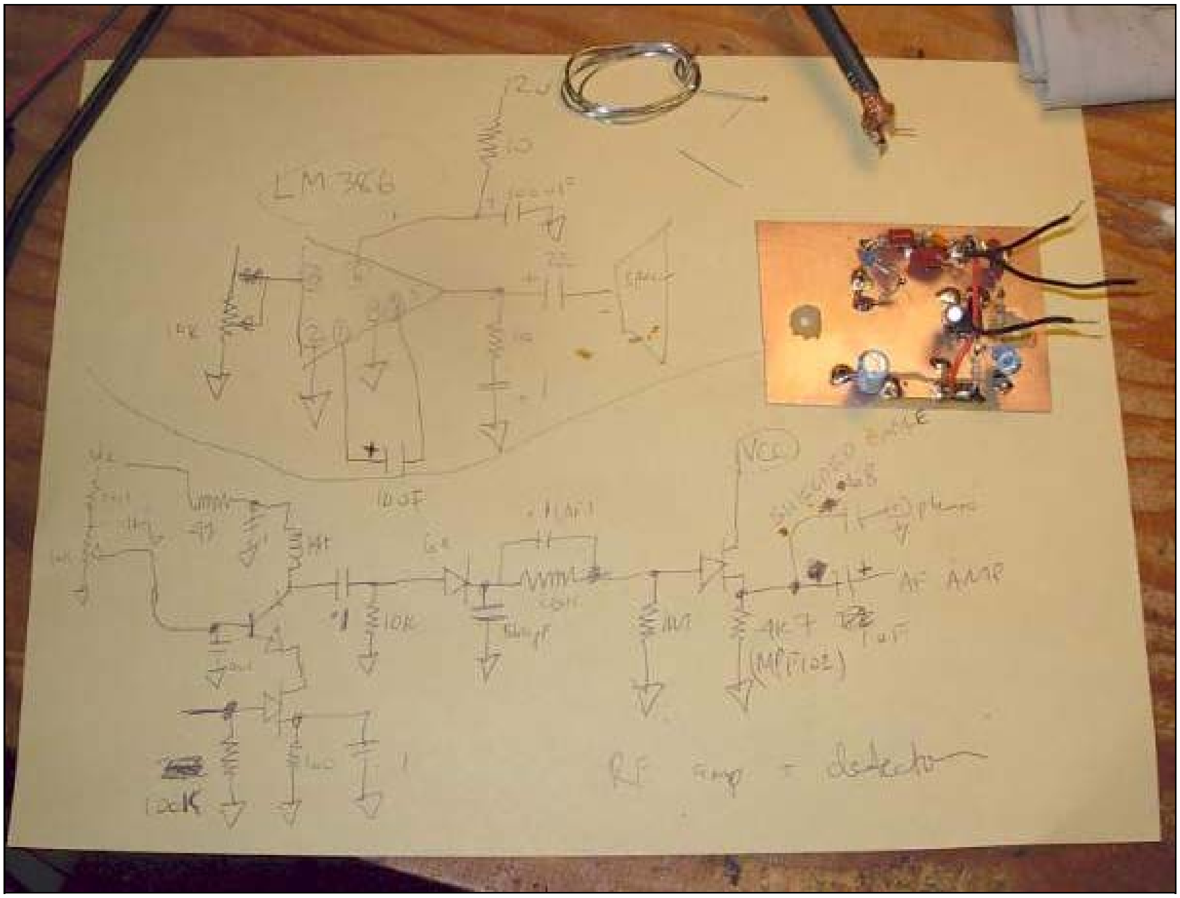
Miscellaneous Photos and Circuits



Above. A work bench version of the basic receiver bread board built in distinct stages from the antenna input to the speaker.

Above, A breadboard of the KK7B audio stage which is preferred to the LM386 stage used in the 5 MHz WWV receiver. The LM386 does take up less room and certainly increases the popcorn factor.

Above. The AF stage shown above in a chassis containing a speaker. This is my lab AF amp for testing receivers on my work bench.



A work bench photo of the first version of the W7ZOI detector built in October 2007 is shown above.

RF — Test and Measurement

My Links

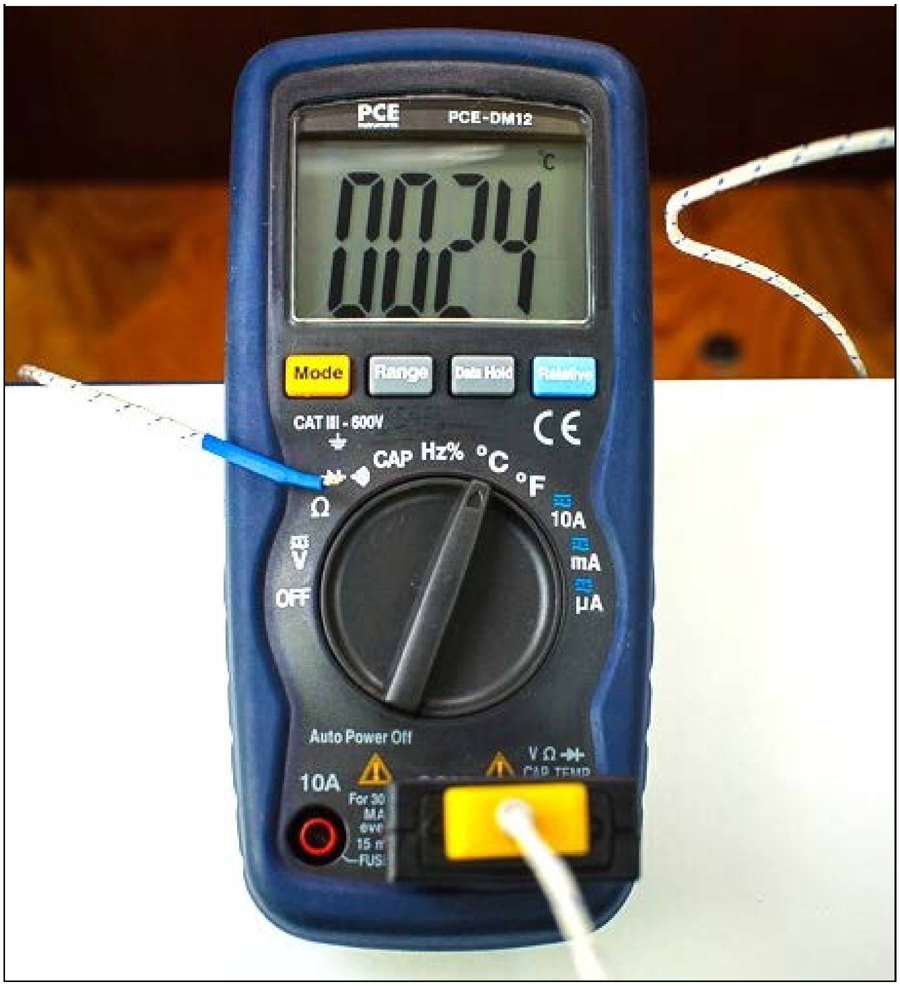
Greetings! I enter the QRPHB / SWL HomeBuilder site here and linked a few of my favorite tools, part suppliers and URLs I like and wish to remember. This page might feel intransigent.

In the following section called **Graphics**, I show the odd newer schematic or photo unpublished as permanent site content. A schematic might just represent my mistaken notion or whimsy: a sense of what's caught my fancy — not that my opinion is better than anyone elses.

Quick Navigator

Graphics

Russian giant sunflower — grew to over 3 meters tall



My gorgeous new PCE multimeter just in time for Season 16 experiments starting October 3. Click for another photo. Click for the PCE website — test equipment galore.

RF Transistor Data Spreadsheets

FET

BJT

Test Equipment

Test equipment

USA Rigol Dealer T Equipment

Online Tools and Publications

Standard resistor and cap values

Smith Chart information

Good inductive/capacitive reactance calculator other tools available

Resistive pad attenuator calculator

Ohm's Law 1 pager

VHF Communications website UKW-Berichte/VHF-Communications

The Signal Path EE video blog They review and show the DSA1030A SA + TG

DUBUS Technical magazine for 50 MHz and above

Favorite Experimenters

Esteemed QRP homebrew mentor - Wes Hayward's, W7ZOI web site

🍁 Iulian, YOUDAC-VA3IUL Comprehensive design information and amazing circuits web site

🍁 Markus Hansen, VE7CA A venerable Room 19 Radio School graduate. He has authored for QST. See his receiver featured in QST for March 2006

🍁 Blog of fellow VE7-land experimenter Steve, VE7SL. Look for a link to Steve's web site on his blog

Professor Ken Kuhn's web site. Perhaps start on his site map. A treasure trove of useful information

Ripples in the Ether An always interesting blog from our friend Jason who runs Etherkit

SV3ORA's Site Friend Konstantinos from Greece. Design info galore. Check out his microwave links.

FM Tuner Site Vintage FM Tuner Information

Paul, KE7HR Caver website

Matjaž, S53MV Microwave and RF. My favorite microwave designer. See this page for some inspiration

Paul Wade, W1GHZ authors my favorite QST column *Microwavelengths* His site

Jacques, VE2AZX web site Retired EE with great ideas

Michigan QRP Club Publish *The 5 Watter* Quarterly

Website of Michel, **F6FEO** from France. F6FEO project pages: click and click

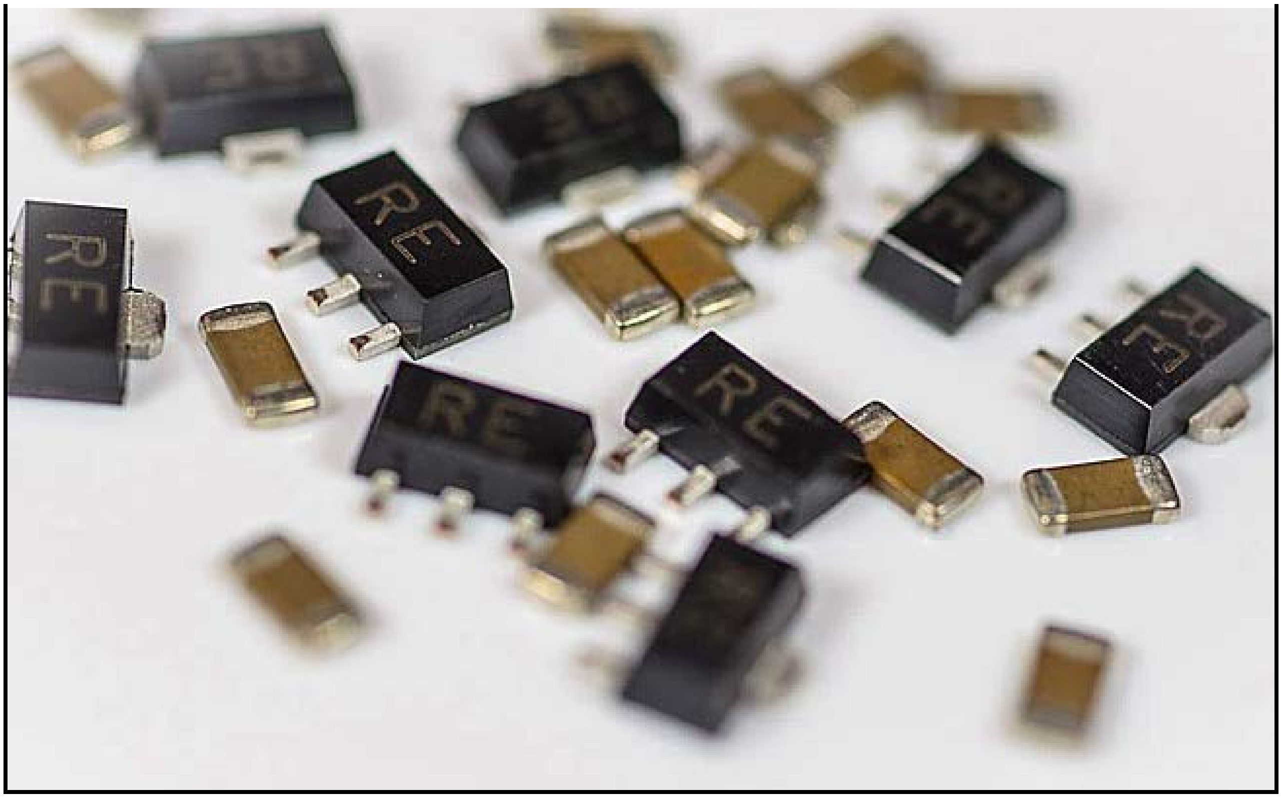
Roland, a retired EE who lives in France keeps this wonderful test equipment page

JF1OZL Mega site that first inspired me years ago. He once wrote me a letter - great guy

Brian, K6STI 88-108 MHz web site.

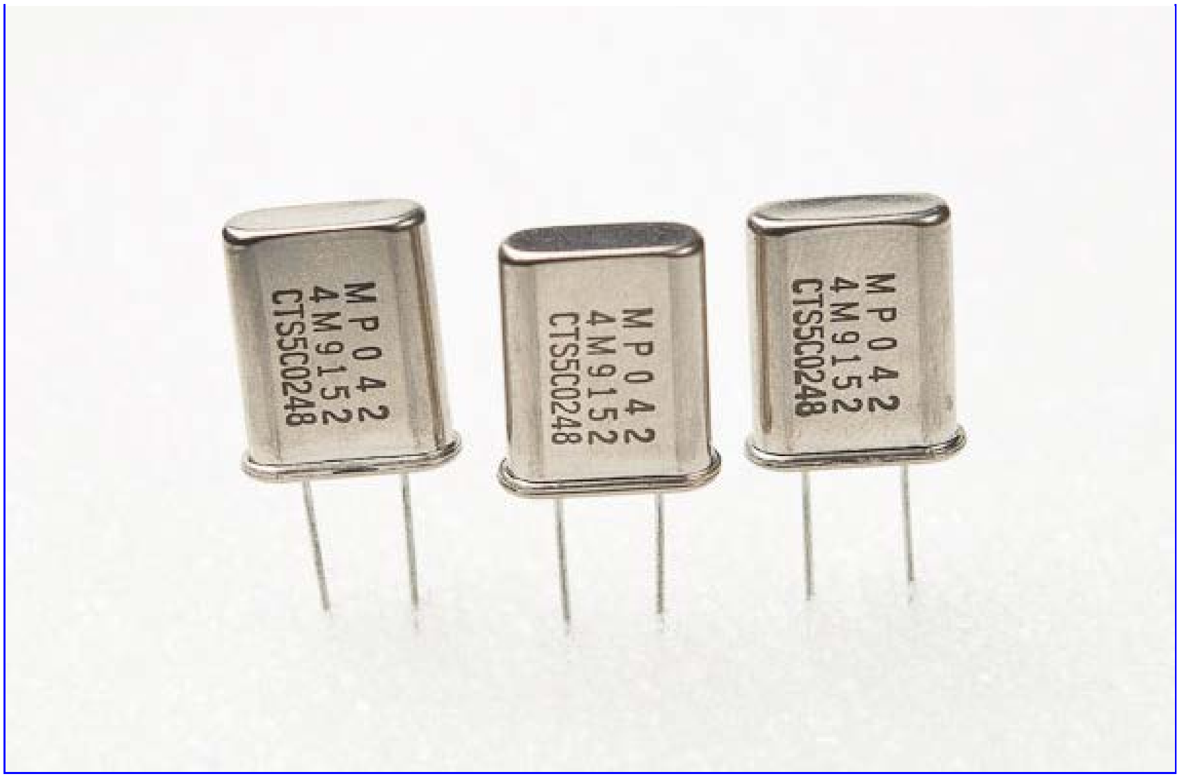
Leif, SM5BSZ Amazing work

KA7OEI Microwave Page web site


RE

Memorial Museum of the Leningrad Siege, St. Petersburg, Russia VE7BP0 photos 2006

Государственный музей обороны и блокады Ленинграда

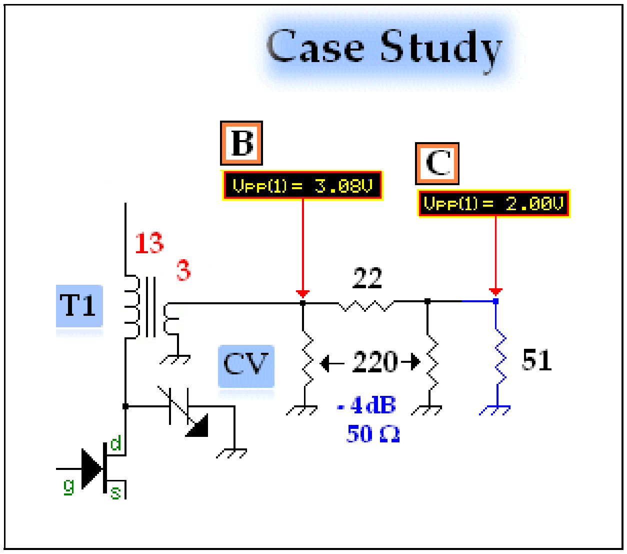

MP042
4M9152
CTS5C0248

RF Workbench Page 1 - Case Study Answers

This web page supplements RF Workbench Page 1

Answers:

1. Calculate the power in dBm at point B **= 13.75 dBm**
2. Calculate the power in dBm at point C **= 10 dBm**
3. What is the attenuation in dB of the 50 ohms attenuator pad? **= -3.75 dB**
4. What is the output power in mW of this stage? **= 10 mW**

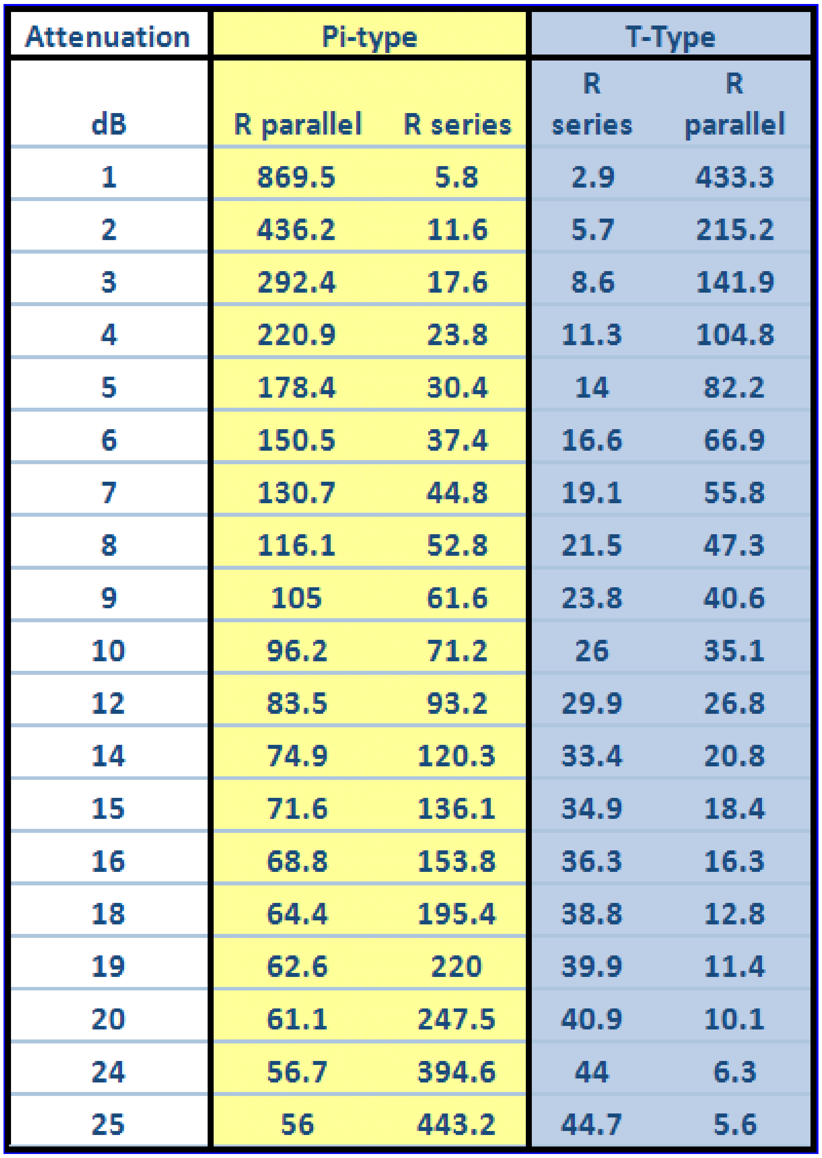


The design contained a "theoretical" - 4 dB, 50 Ω attenuator pad using standard value resistor substitutions

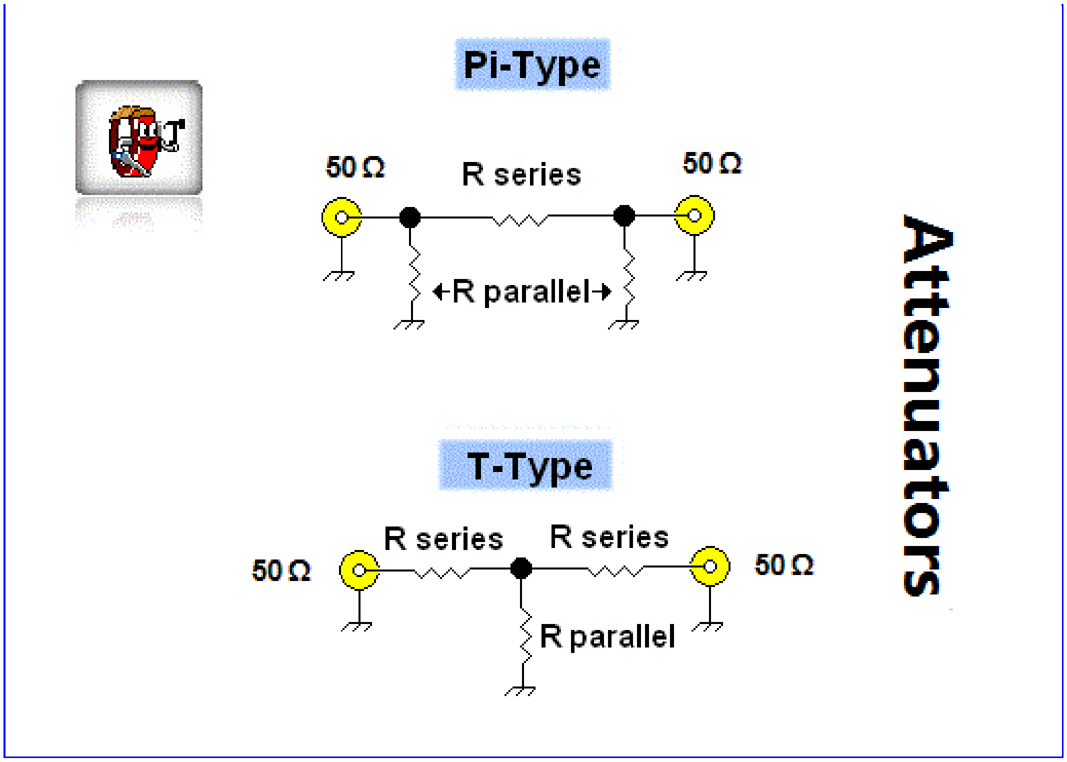
50 Ohms Attenuator Pads

This web page supplements RF Workbench Page 2

Table For Pi and T- type pads



Attenuation	Pi-type		T-Type	
dB	R parallel	R series	R series	R parallel
1	869.5	5.8	2.9	433.3
2	436.2	11.6	5.7	215.2
3	292.4	17.6	8.6	141.9
4	220.9	23.8	11.3	104.8
5	178.4	30.4	14	82.2
6	150.5	37.4	16.6	66.9
7	130.7	44.8	19.1	55.8
8	116.1	52.8	21.5	47.3
9	105	61.6	23.8	40.6
10	96.2	71.2	26	35.1
12	83.5	93.2	29.9	26.8
14	74.9	120.3	33.4	20.8
15	71.6	136.1	34.9	18.4
16	68.8	153.8	36.3	16.3
18	64.4	195.4	38.8	12.8
19	62.6	220	39.9	11.4
20	61.1	247.5	40.9	10.1
24	56.7	394.6	44	6.3
25	56	443.2	44.7	5.6



To build attenuator networks, we normally choose use nearest standard resistor values, or parallel/series 2 or more resistors from our resistor collection to approximate the tabled R. If you're building with 5% parts, consider measuring a few resistors with an ohmmeter to find the best match to the tabeled R.

Resistors in the 8 dB T-type or the 4, 14 and 19 dB Pi-type networks are close to 5% standard resistor values.

Expected Return Loss Table (50 ohm bridge)

This web page is a supplement to Transistor Input Impedance Experiments

Expected Return Loss in a 50 ohm Bridge

Resistor (Ω)	Return Loss (dB)
10.00	3.52
15.00	5.38
22.00	8.20
27.00	10.50
33.00	13.77
39.00	18.16
47.00	30.19
62.00	19.40
68.00	16.33
75.00	13.98
82.00	12.31
100.00	9.54
120.00	7.71
150.00	6.02
170.00	5.26
200.00	4.43
220.00	4.02
270.00	3.25
330.00	2.65
390.00	2.20
470.00	1.86
510.00	1.71

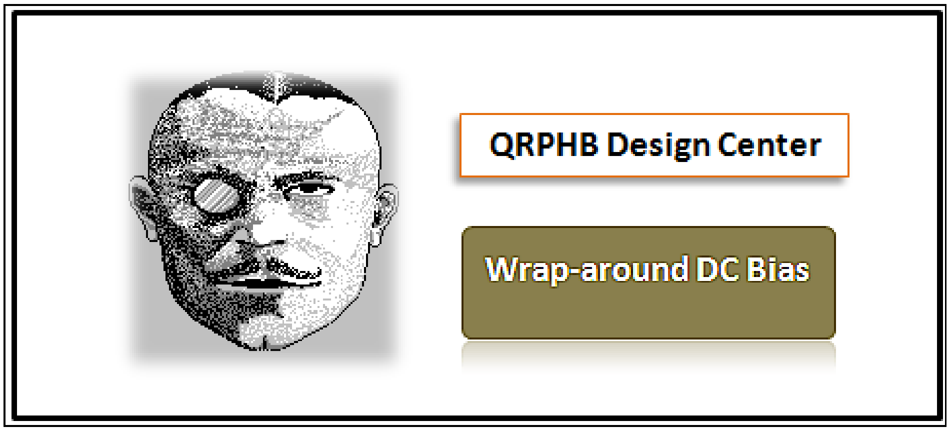
RF — Test and Measurement

Miscellaneous RF Topics 2011 Supplement

Introduction

This web page supplements the Miscellaneous RF Topics 2011 web page.

1. Design Center - Wrap Around Bias



PNP Wrap-around Bias for an NPN RF Amp

There are entire books written on RF power amps. I don’t pretend to be an expert and offer a simple approach to help move novice builders up the learning curve. Big thanks to Wes, W7ZOI for the theory and explanation for EMRFD Figure 2.16; the wrap-around PNP bias of an NPN transistor with grounded emitter. This bias circuit is fantastic and helps prevent you from burning up too many expensive medium power transistors such as the 2N3866 or 2N5109 during experiments and operation. From our emails, the following experiments were performed and information gleaned.

Since the NPN emitter terminal is grounded or near to ground on my web page examples, the current can really run high — Don't forget to use heat sinks and appropriately rated resistors on the NPN circuit when you really go for power.

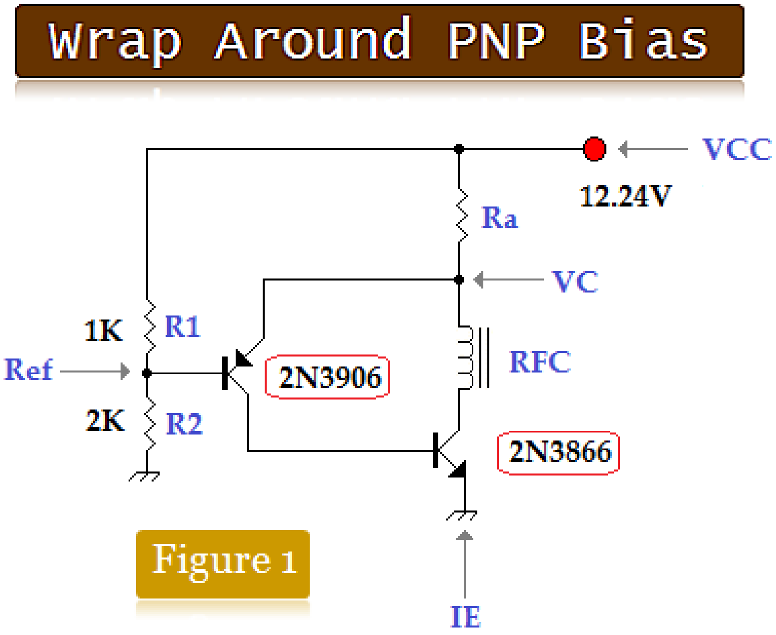
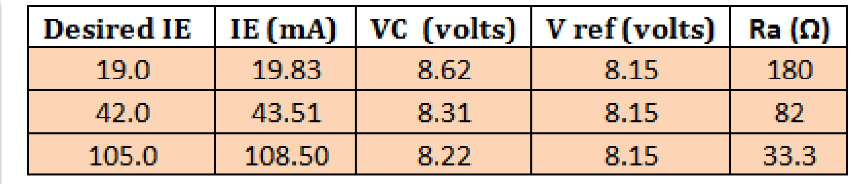


Figure 1



Desired IE	IE (mA)	VC (volts)	V ref (volts)	Ra (Ω)
19.0	19.83	8.62	8.15	180
42.0	43.51	8.31	8.15	82
105.0	108.50	8.22	8.15	33.3

Above — The schematic and data table from some biasing experiments. The normal decoupling parts were removed to isolate the biasing circuitry. All voltage and current values were measured. Another photo. A folded piece of paper is under the wire connecting the NPN emitter to the copper clad board. This is how I measure emitter current. The paper acts as an insulator and allows you to press hard with the ammeter probe to ensure good contact. After current measurement, the paper was discarded and the emitter soldered to the copper ground plane.

Biasing Procedure

Step 1: Choose a Target VC

Using collector resistor Ra, we want to drop about 2/3 of the power supply voltage across the NPN transistor. For a 12 volt supply or VCC, this is 8 volts VC. The formula is Target voltage = VCC x 0.66. Why choose 2/3 VCC instead of ½ VCC like we do for Class A audio amplifiers? In audio amps we just use a resistor as the load and the collector voltage can't go above the power supply unless we insert some reactances in the circuit. In simple AF voltage amps, the half VCC supply allows the greatest undistorted, peak-to-peak AC signal swing.

In RF power amps, a transformer or RFC lies on the collector terminal, so the instantaneous collector voltage (sum of the supply voltage plus the peak AC voltage) can go well above the supply voltage. If you have 8 volts on the collector, VC can easily rise to 16 volts or so on signal peaks. It's often prudent to choose a VC closer to VCC than the classic ½ VCC of audio amplifiers.

Hereafter, Target VC is referred to as **VRef**

Step 2: Choose R1 and R2 to bias the PNP @ VRef

Choose the PNP biasing transistors R1 and R2 so that the PNP bias = VRef. The formula follows:

$$\mathrm{VRef} = \frac{\mathrm{VCC} \times \mathrm{R2}}{\mathrm{R1} + \mathrm{R2}}$$

I wrote a Java Applet that does the math Applet A ; just plug in a VCC and play with the resistor values until you get your target VRef with the resistors from your parts collection. Try to get a bias as close as possible to the targeted VRef, but don’t stress out over it.

Step 3: Choose Ra to give (VRef + 0.6)

Ra = VCC – (VRef + 0.6) / desired current in amperes. Use or make up Ra with nearest standard resistor value(s) with an appropriate power rating. In a PNP, the emitter terminal is 0.6 VDC higher than the base, so need to we add that to VRef. The desired current is any reasonable current that you want to set up; for example 25 mA or 75 ma.

Example Calculation

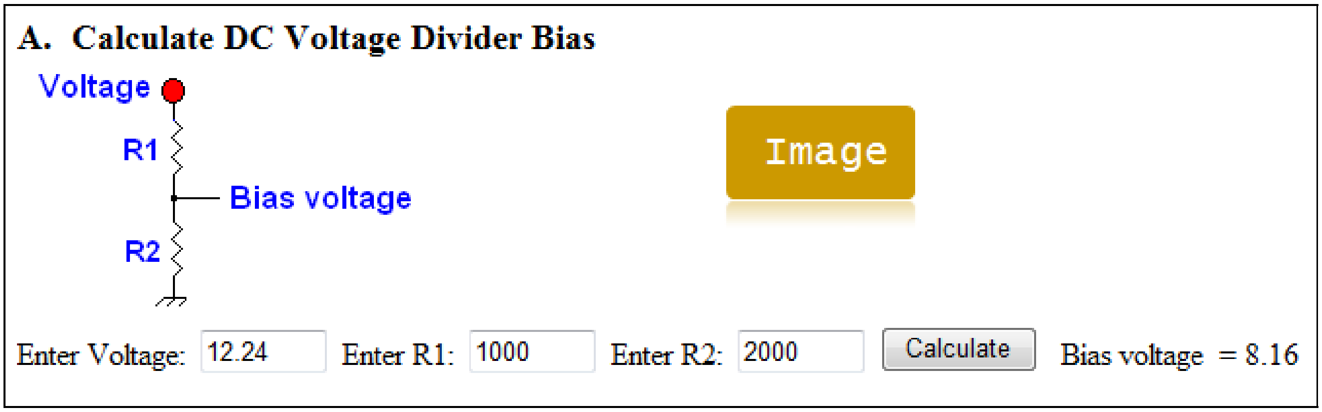
I wanted a emitter current of 105 mA from the Figure 1 circuit. VCC = 12.24. Let's run through the steps.

Step 1: Choose a Target VC

Target voltage = 12.24v x 0.66 = 8.1 VDC -> Now called VRef

Step 2: Choose R1 and R2 to bias the PNP at VRef

I just plugged some resistor values into [Java Applet A](). R1 is normally lower than R2.



A. Calculate DC Voltage Divider Bias

Enter Voltage: 12.24 Enter R1: 1000 Enter R2: 2000 Bias voltage = 8.16

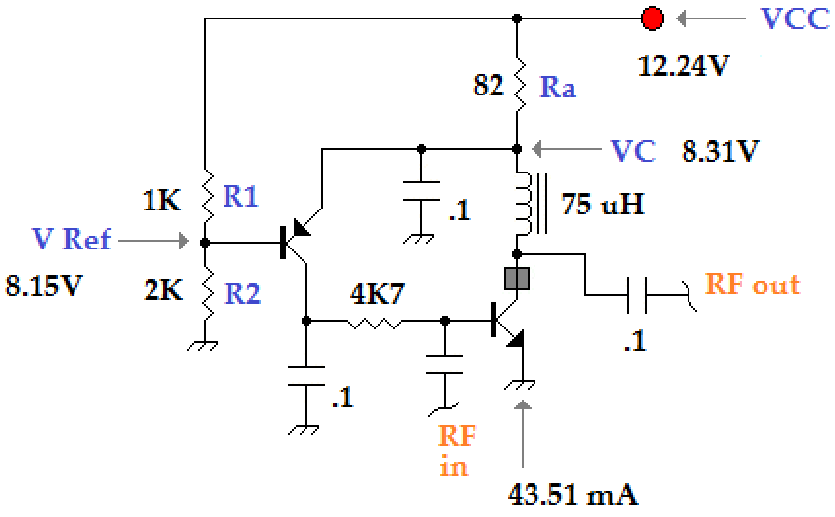
Above — A practical bias network. I have lots of 1 and 2K resistors. 8.16 v is close to my target VRef.

Step 3: Choose Ra to give (VRef + 0.6)

12.24v – (8.16v + 0.6v) / 0.105 A desired = 33.1 ohms. I put three 100 ohm ½ watt resistors in parallel for Ra. [Click]() for lateral view photo showing the 3 resistors.

The exact measurements of this example and 2 other (19 and 42 mA) are tabled in Figure 1. I used an 82 ohm 1/2 watt resistor for Ra in the 42 mA version. The bench values closely approximate the calculated values. This circuit is quite instructive. In a real circuit (Figure 2) we add in the decoupling parts. Ra performs double duty as the decoupling resistor as well as the current regulator resistor.

Figure 2



Above — A working version of the desired 42 mA medium power amp from Figure 1. Refer to EMRFD Chapter 2 for further information.

How does the PNP wrap-around bias loop work?

Using the sample conditions from Figure 2: VC = 8.16v calculated, Desired IE = 42 mA and VCC = 12.24v
The main decoupling/current limiting resistor Ra is established at 12.24v – (8.16v + 0.6v) / 0.042 mA = 82.9 Ohms.

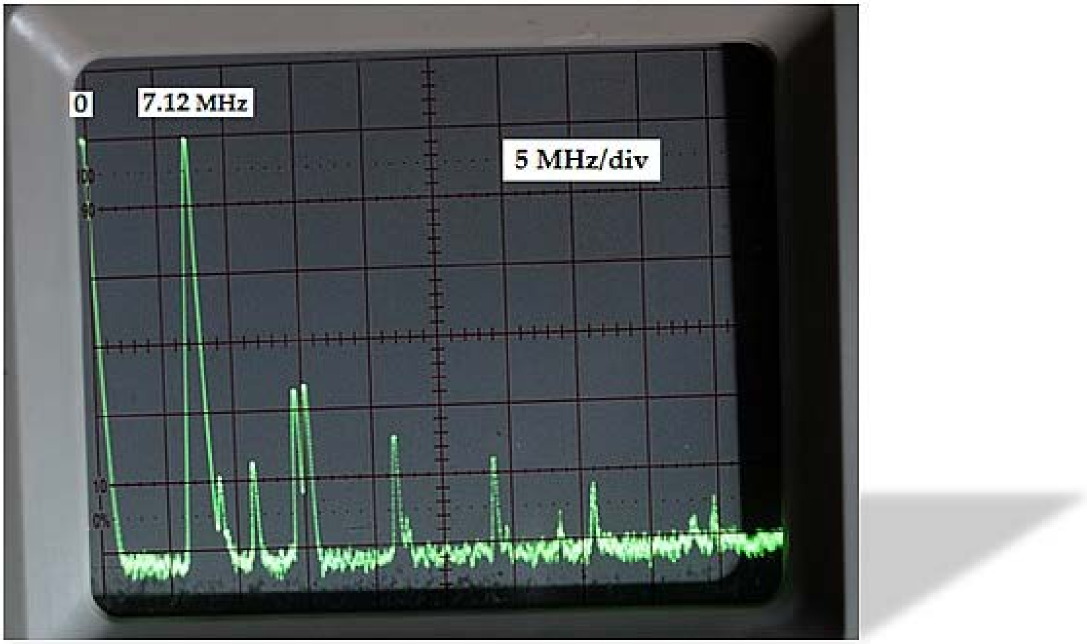
No matter what, the PNP VRef = 8.16 volts. All of the desired conditions are established and the current is flowing in the NPN. Let's change these conditions and see what happens.

First, instead of 42 mA, assume that the NPN current drops to something less. Hence, there will not be as much of a drop across Ra. This will cause VC to increase. This is the voltage on the PNP emitter. When this voltage increases, there is more voltage between the base and the emitter of the PNP, which will make it draw **more** current. That current comes out of the PNP collector runs to the base of the NPN. This causes the NPN to draw more current (beta times) which increases its collector current and causes the collector voltage to drop because of the I x R drop in Ra. It drops until it reaches the 8.16 volt level.

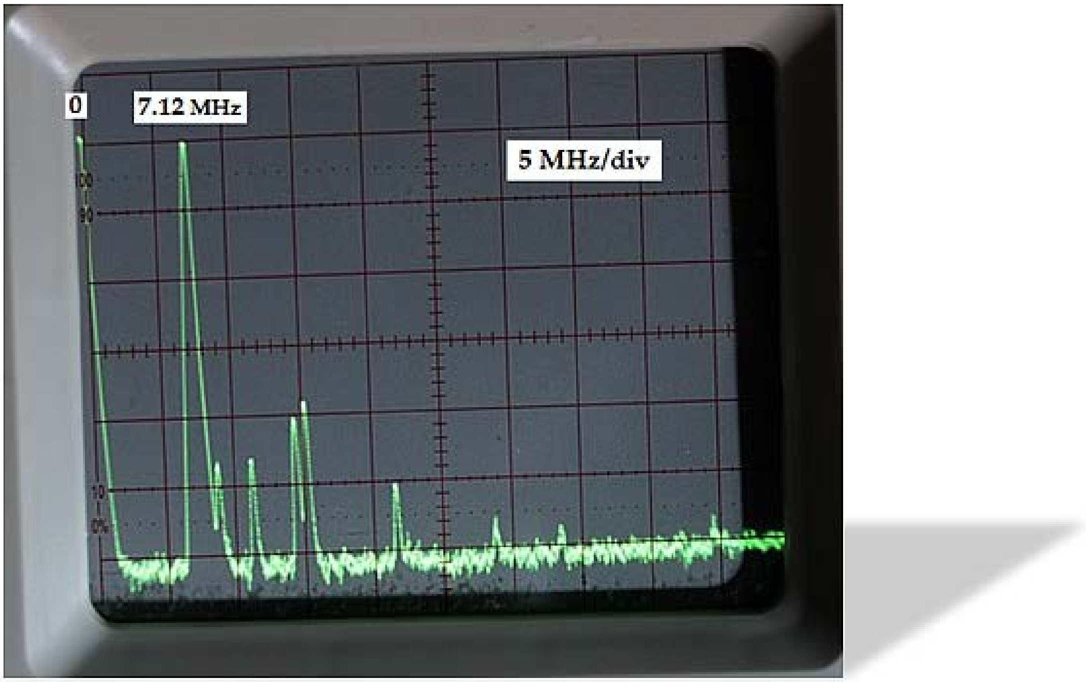
Let's now assume the opposite. The NPN draw too much current, more than the desired 42 mA. This means that there is too much I x R drop in Ra and VC goes **below** 8.16 volts. This tends to turn the PNP off. The PNP collector current drops, so the NPN base current also drops, causing the NPN collector current to drop. This causes VC to again increase to reach 8.16 volts. Is this cool or what?

---

2. RC Low-pass Network

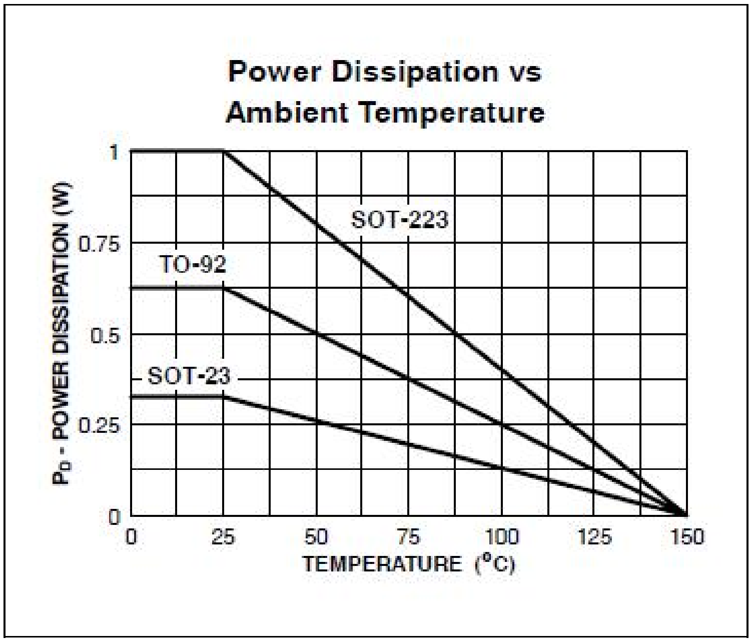


Above — Spectrum analysis without the RC low-pass filter.



Above — Spectrum analysis with the RC filter in-situ. Above 20 MHz, the spurs are ~5-6 dB down.

3. Transistor Power Dissipation in the 2N3904



Above — Power Dissipation versus Temperature for the 2N3904 in 3 packages.

Power dissipation in the collector resistor

For example, VC (the voltage across the transistor) = 10 volts and the emitter current = 20 mA

P = I x E = .020 amps x 10 volts = 200 mW

Power dissipation in the emitter resistor

P = I x E = .020 amps x the measured emitter voltage

or P = I x I x R

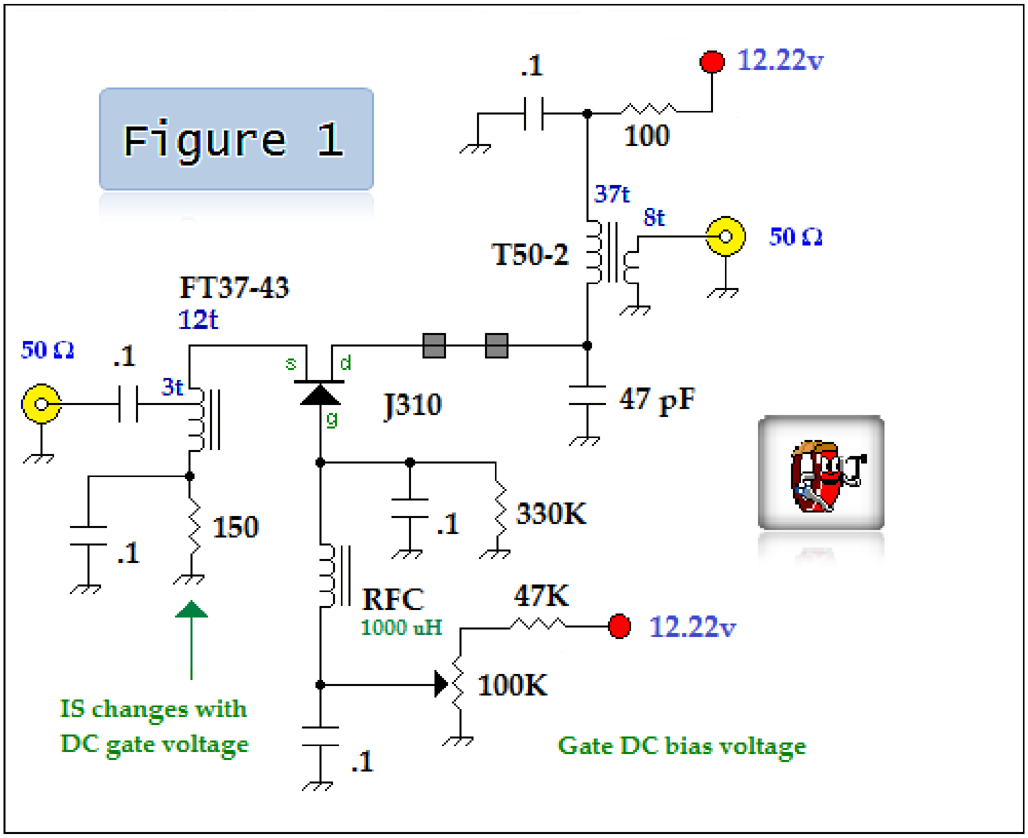
For example, emitter current = 20 mA with a 5 ohm emitter resistor; P = .020 amps x .020 amps x 5 ohms = 2 mW

We assume IE = IC in the above examples. At 200 mW collector dissipation, the 2N3904 temperature temperature climbs to about 65 C. A metal can transistor such as 2N2222 has more surface area and better heat dissipation. A heat sink also improves heat dissipation. Temperature dissipation charts are referenced to ambient temperature which may be dependent on the air circulation around the transistor. If you plan on running 200 mW or so collector dissipation with a 2N3904, glue on a heat sink. A pinch of the finger and thumb, or a infrared thermometer are the most common popcorn ways to test for the need for more heat sinking, or a part with better power dissipation.

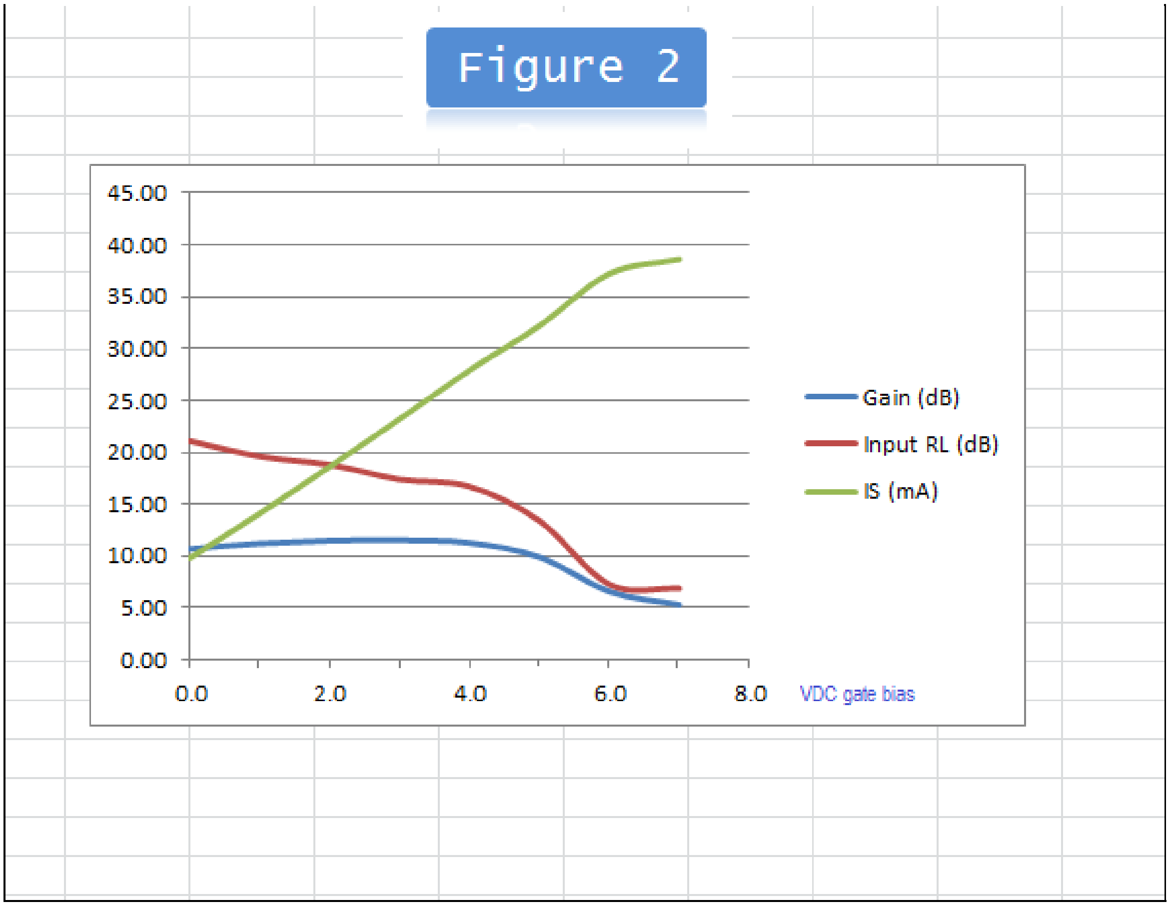
Above — I learned this from a Russian builder. 100 heat sinks for a dollar. Typically, a small blob of epoxy with the transistor positioned to the lower edge of the penny works best. I used too much epoxy and didn't wait for it to completely dry before taking this photograph — the penny slid down.

4. Some Factors Affecting Common-Gate Amplifier Input Impedance

A question arises: with all the devices and topologies available, why build a common-gate JFET RF amp in 2011 and on? For the J310 JFET at least: they're cheap, the input impedance is close to 50 ohms and wideband, the amp demonstrates good stability, signal handling capability and linear performance — all without running mega-current. Мне нравится. Many authors state the common-gate noise figure is inferior to that obtainable with the common-source topology, but for the popcorn builder, the common-gate noise figure is probably reasonable.



Above — A JFET gate biasing experiment. Here I applied 0-7 volts DC to the gate of a common-gate FET amp. The results proved interesting; see the graph below. The drain was tuned with a fixed 47 pF cap and manually "scrunch tuned" by pinching the T50-2 toroid with fingers and thumb. The input RFC serves to decouple the DC from the AC signal and the tap is used to improve the input impedance match.

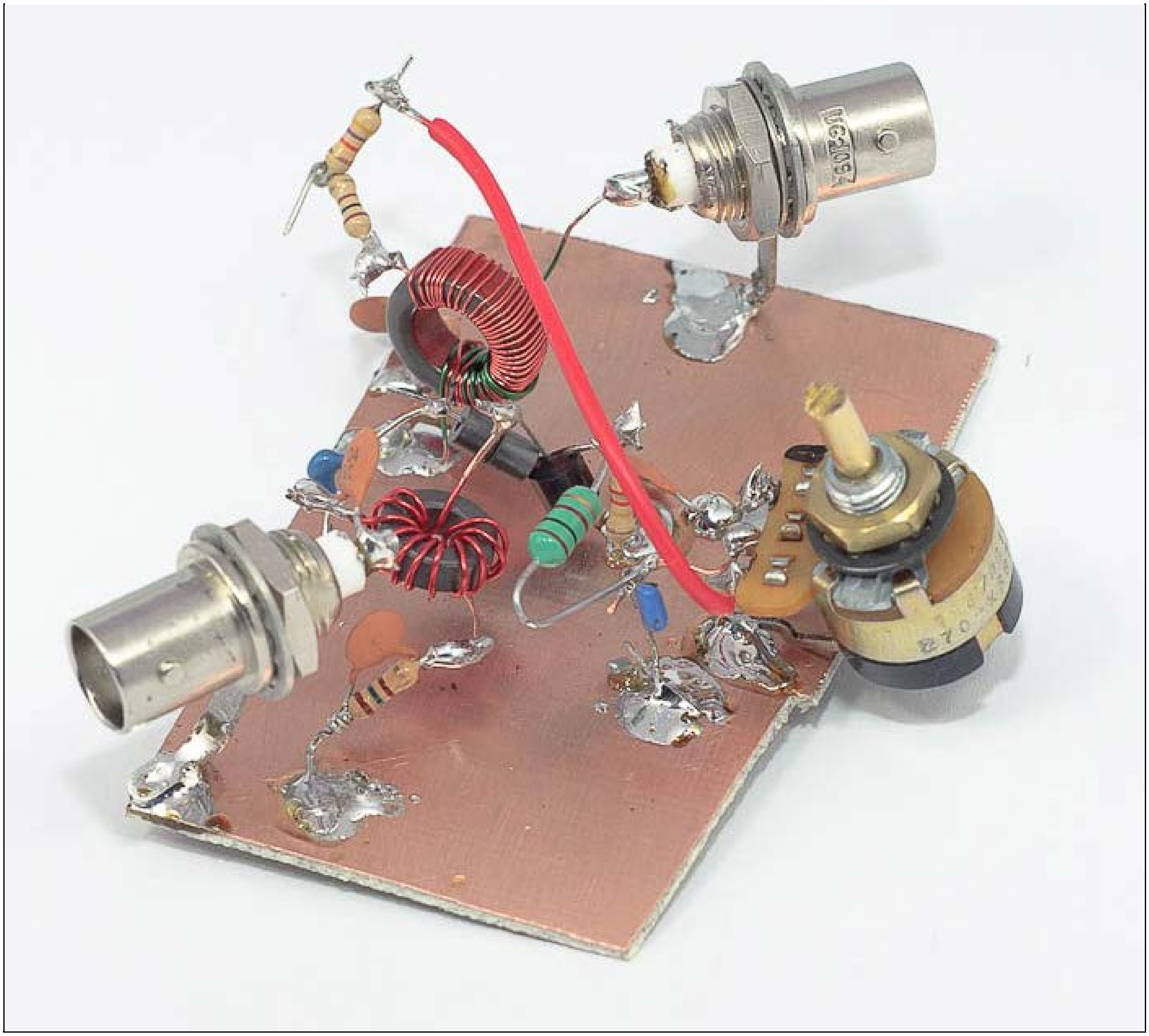


Above in Figure 2 — A graph of the DC gate voltage versus gain, input return loss and source current from Figure 1.

Increasing the DC gate voltage increases the source current as shown in the graph — ranging from 9.8 to 38.6 mA. From a DC gate voltage of 0 to about 4 volts, amplifier gain changes very little. Technically, the gain of this amplifier should increase along with drain current, however, the input resistance simultaneously goes down, creating greater voltage division loss at the input plus reduced return loss from impedance mismatch. This tends to cancel the increased gain as the plot shows.

The AC signal current injected into the FET comes out of the drain with power gain because that same current is now flowing into a higher impedance. The gain is dependent on the ratio of the impedance at the drain to that at the source. Click for a SPICE plot of DC gate voltage versus source current for 0 - 5 volts from Wes, W7ZOI.

From experiments, I learned that at least 6 things effect input return loss: the source resistor value, stacking or paralleling JFETS, the gate DC voltage(all which change current and/or transconductance), whether or not you bypass the source resistor, negative feedback and also the tap on the input RFC. Like other amps, the impedance of the device before and after the JFET amp can also change the input match, but those are external and normal considerations.



Above — Breadboard of the DC gate bias experiments of Figure1. Properly biased JFETs act like a current source where drain current is controlled by the gate-to-source DC voltage. The input impedance of the common gate amp is the reciprocal of the transconductance. Ken Kuhn posted some great JFET design notes in the yellow JFET section on his web site. EMRFD provides another essential reference for RF JFET design.

Noise Figure and the Common Gate Amplifier

Noise figure is important in low-level RF amplifiers such as a preamp or post-mixer amp because noise generated by these amplifiers contributes to the overall noise figure of a receiver. The input impedance an amp like the common-gate is considered an *active* impedance. Often, with active impedances, obtaining the maximum power transfer and lowest noise figure don't occur simultaneously. That is, the optimum input impedance match for gain is not the optimum input match for low noise performance. With pure resistances, the best noise figure possible is 3 dB; however, when using an active impedance, it's possible to achieve a noise figure less than 3 dB by applying negative feedback or other techniques to manipulate the JFET active input impedance and lower its voltage noise.

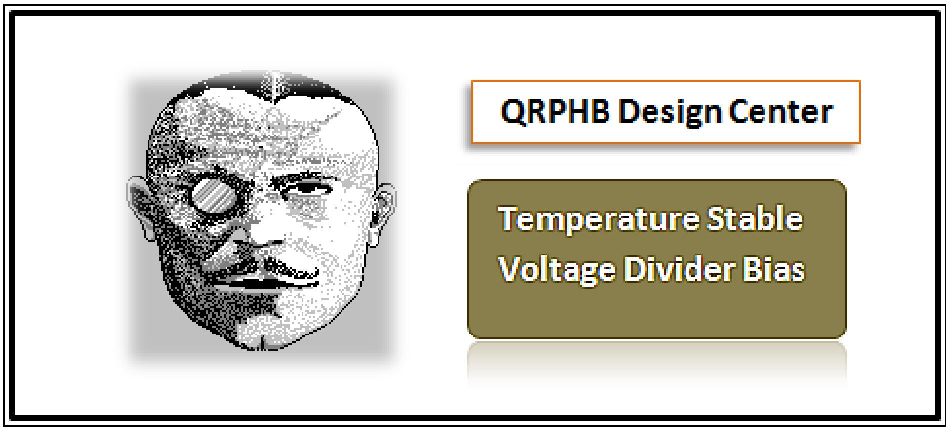
A stellar example by Bill, W7AAZI employs DC gate bias, noiseless negative feedback and paralleled JFETs to achieve a noise figure in the 1.2-1.5 dB range. This mind-boggling circuit is described in EMRFD as Figure 6.94. I wanted to explore this topic: can amateur builders design *common-gate* amplifiers for a lower noise figure? The answer is perhaps, however, unfortunately, there is no cookie-cutter approach.

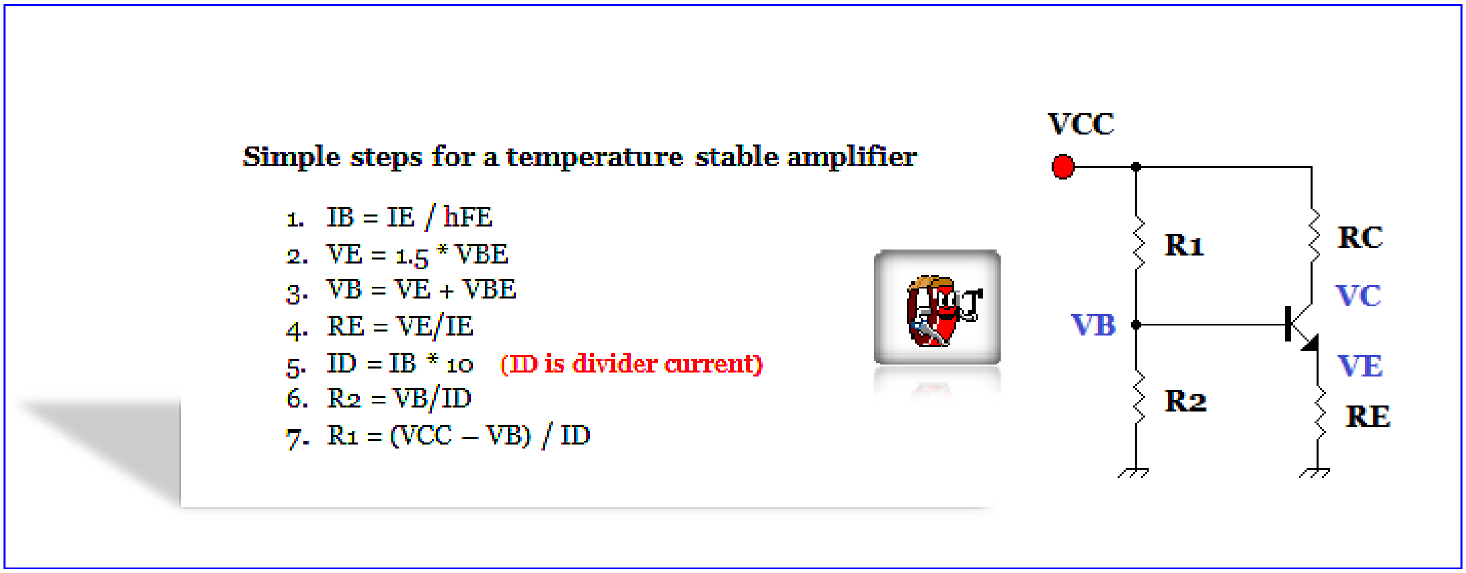
Low noise amplifier design imposes an advanced topic and many builders lack the math skills and/or test equipment to design and generate results. Each design must be approached on a case-by-case basis, although similar fundamentals apply to all cases — using active impedance manipulation techniques and paralleling up JFETs while paying attention to all the resistances in a circuit — even loss resistance in the inductors. I learned from Professor Ken Kuhn that when amplifiers are paralleled the output signal power adds linearly, whereas the internal noise adds statistically (square root of the sum of the squares) and thus noise figure improves. In short, paralleling JFETs reduces voltage noise, but only to a point, as input capacitance also increases which can degrade high frequency performance and noise figure. Ideally, the JFETs paralleled should be matched for IDSS and VP.

Further, all devices have a voltage noise specifications (in volts per root Hz) and a current noise specifications (in amperes per root Hz). The source impedance that produces the lowest noise is the ratio of the two — derived from Ohm's law. The input impedance of the device may be higher or lower; thus the optimum source impedance for low noise is not necessarily the optimum source impedance for maximum power transfer. It's unlikely that these two impedances differ widely, however, only careful bench measurements or closely copying proven designs will ensure the desired results.

---

5. Design Center - Temperature Stable Voltage Divider Bias





Simple steps for a temperature stable amplifier

1. IB = IE / hFE
2. VE = 1.5 * VBE
3. VB = VE + VBE
4. RE = VE/IE
5. ID = IB * 10 (ID is divider current)
6. R2 = VB/ID
7. R1 = (VCC – VB) / ID

Above — A simple way to design a temperature stable BJT voltage divider bias. Assumes that IC = IE. You specify IE.

I sought to develop a simple way for builders to design a temperature stable BJT amp using voltage divider bias. It wasn't so easy. Here is the full math from Ken Kuhn. As shown, it gets complicated and over-the-head of the target readers of this web site.

My simple approach makes assumptions, and certainly an astute builder could improve my algorithm - I developed other formulae, however settled on this one because it well fits the scope of the QRP/SWL HB web site. Most engineers design their bias networks in SPICE and I recommend this.

To choose a Beta or hFE, you can measure the Beta or use the transistor datasheet Beta near to the emiiter current you intend to apply. Many builders debate this and even go as far to state that spec sheet Beta values are useless. This may be true, but you *have* to decide on some approach. Ultimately, you will build the amplifier and can tweak 1 or more of the bias resistor values to set your bias voltage (emitter current). Click for a snippet from a 2N3904 datasheet showing hFE at a range of currents.

Biasing BJTs is a great example of building to suit your design requirements. The average reader likely operates their radios at room temperature where a minimum temperature stability factor is needed. On the other hand, you may operate field portable, or use a crammed chassis where transistors heat threatens stability. As always, your design choices must fit your needs, parts and abilities.
The above 7-step method forms a simple **QRPHB Design Center** method to achieve a reasonable temperature stability factor at normal

operating temperatures. I considered 2 factors: VBE and Beta (hFE) changes.

VBE

We normally attribute the voltage difference from base to emitter (VBE) as 0.7 volts for silicon transistors. VBE changes inversely with temperature and at AF to HF, we mitigate VBE creep by applying an emitter resistor to increase the emitter voltage VE. The emitter resistor provides a feedback loop so as collector current starts to increase due to a rise in temperature, the voltage drop across RE also starts to increase. This voltage tends to reverse bias the base-emitter junction. The net effect is decreased collector current.

How do I choose an emitter resistor value for VBE bias stability asked the little grasshopper (novice)?

The VE value should typically be about 10 to 25% of VCC. VE recommendations really vary from author to author — some say VE should be at least 5X VBE while other authors suggest 2-4 volts as the minimum acceptable VE value for bias stability in typical AF-HF amplifiers. I use 1.5 * VBE (0.7) = a VE of 1.05 volts, but it's up to you. Since you know IE for your transistor, calculate the resistor RE to provide the target VE using ohm's law: RE = VE/IE.

At VHF on up, emitters are typically directly connected to ground since that path must have as near zero inductance as possible. In these amps, the bias is derived by a feedback circuit that controls the base current to stabilize the collector current at a specific level. Inductance in the emitter lead causes significant loss of gain at higher frequencies and also upsets the input impedance to the base. Similar effects occur in JFETs and even vacuum tubes. 10s of nH of inductance at 144 MHz represents a reactance of 10s of ohms. At UHF on up this becomes very serious — that's why construction methods are so critical for these builders.

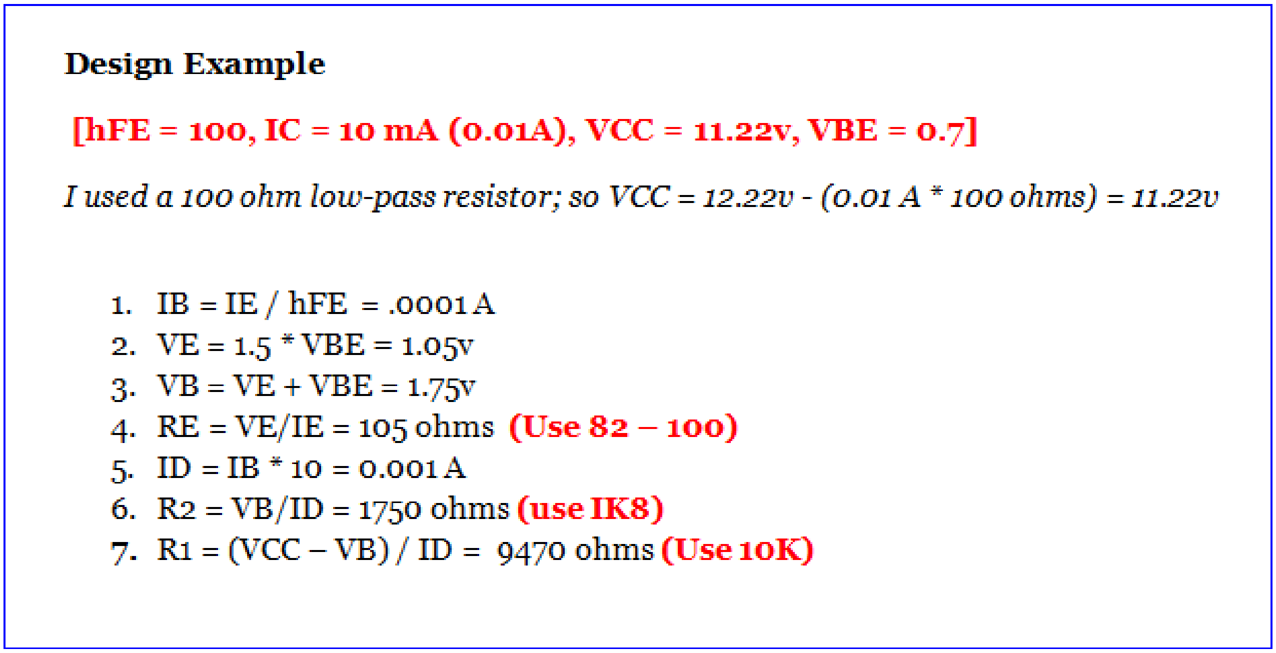
In conclusion; to influence VBE stability against temperature changes and to some extent leakage current, at AF- HF, apply series feedback to a raise emitter voltage (VE). Accounting for variations arising from 5% tolerance parts, I chose a minimal VE of 1.05 volts (equation 2 of 7 in my "Simple Steps..." algorithm).

A tradeoff challenges us — increasing the emitter resistance, increases AC degeneration, lowers AC signal gain and raises the transistor input impedance. Bypassing all or part of the emitter resistance offsets these problems.

Beta

Beta variation arising from temperature changes must be mitigated. I briefly discuss the factors that change Beta in the bullets under the #5 Bipolar Junction Transistor Beta Tester on this page.

To design "beta independent amplifiers", the builder alters the resistance ratio RB/RE. Decreasing this ratio improves the Beta stability while decreasing the amplifier current gain. The ratio can be confusing, so the Design Center equations 5, 6 and 7 just make the current in the divider 10X the base current. It's simple — you're done. Feel free to choose whatever ratio you think is best. Good values for the RB/RE ratio lie in the 12 - 20 range.



Design Example

[hFE = 100, IC = 10 mA (0.01A), VCC = 11.22v, VBE = 0.7]

*I used a 100 ohm low-pass resistor; so VCC = 12.22v - (0.01 A * 100 ohms) = 11.22v*

1. IB = IE / hFE = .0001 A
2. VE = 1.5 * VBE = 1.05v
3. VB = VE + VBE = 1.75v
4. RE = VE/IE = 105 ohms (Use 82 – 100)
5. ID = IB * 10 = 0.001 A
6. R2 = VB/ID = 1750 ohms (use IK8)
7. R1 = (VCC – VB) / ID = 9470 ohms (Use 10K)

Above — A design example employing my "Simple steps for a temp stable amp" bias algorithm. My VCC = 12.22 volts, but, as always I decoupled and bypassed the VCC — in this case, I applied a 100 ohm /0.1 uF network. At 10 mA emitter current, the voltage drop across the 100 Ω resistor results in a VCC of 11.22 volts.

Big thanks to Wes, W7ZOI and Professor Ken Kuhn for helping me with my math problems and to better understand biasing concepts.

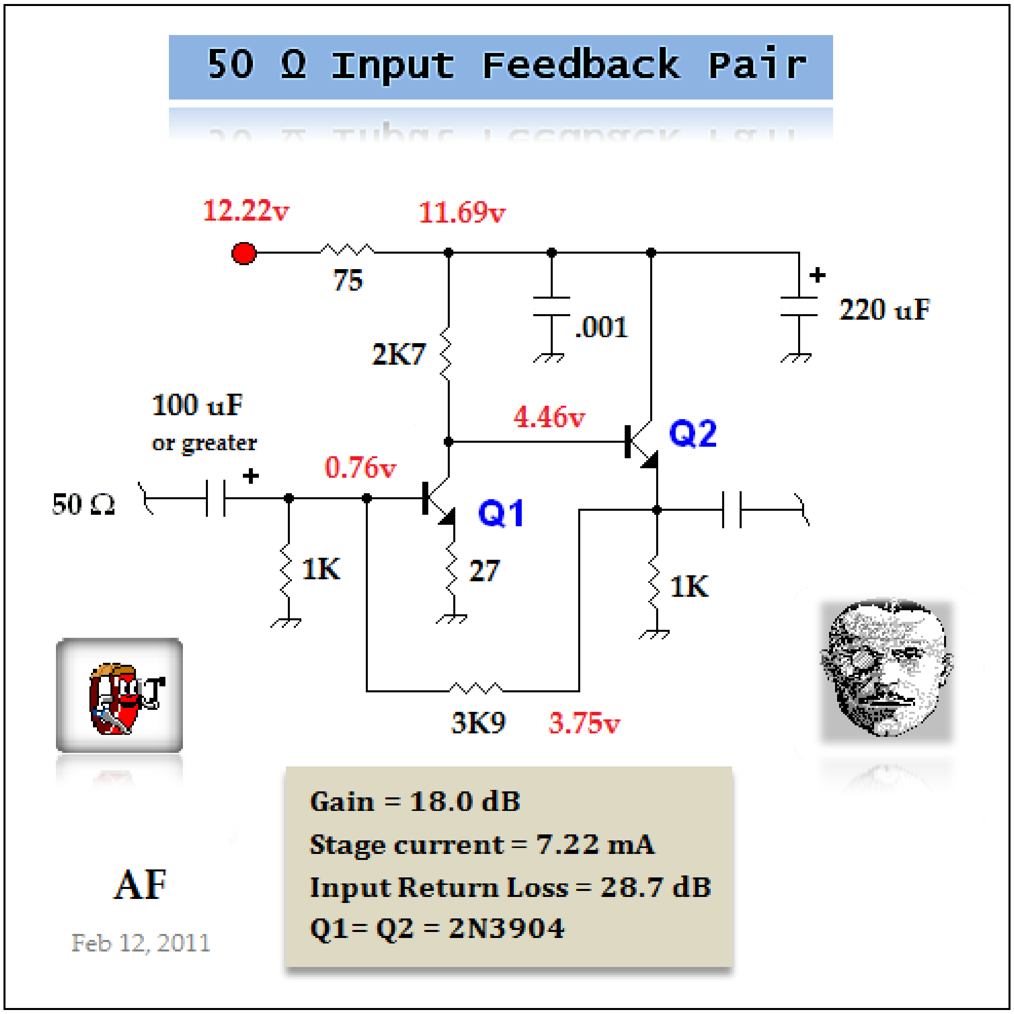
A friend made a Design Center spread sheet for MS Excel. In this application you enter the desired IC rather than IE.

Amateur Radio Electronic Design

50 Ohm Common Emitter Audio Preamp Experiments

Introduction

This page supplements the Pop DC2 Receiver Experiments from this web page.



Above — A 50 ohm audio preamp to follow a diode ring product detector.

Many seek an alternative to the common base AF preamp that typically follows a diode ring product detector. Winter experiments yielded some possible ideas. The simplest is shown above.

Using all different parts, I built and measured another version to ensure reproducibility of the input return loss while just using standard 5% resistor values. The second amp proved similar: 18.4 dB gain, 7.27 mA stage current (3 .8 mA is consumed by the emitter follower), a return loss of 27.6 dB and a nice sine wave output. A 2K2 resistor AC coupled to the emitter follower provided a load. Note in the photo, I used 1% resistors in some slots, but the values were standard 5% and not unusual values like 15.4 K etc.. The 3K9 feedback R is the most critical value — it's a 5% part.

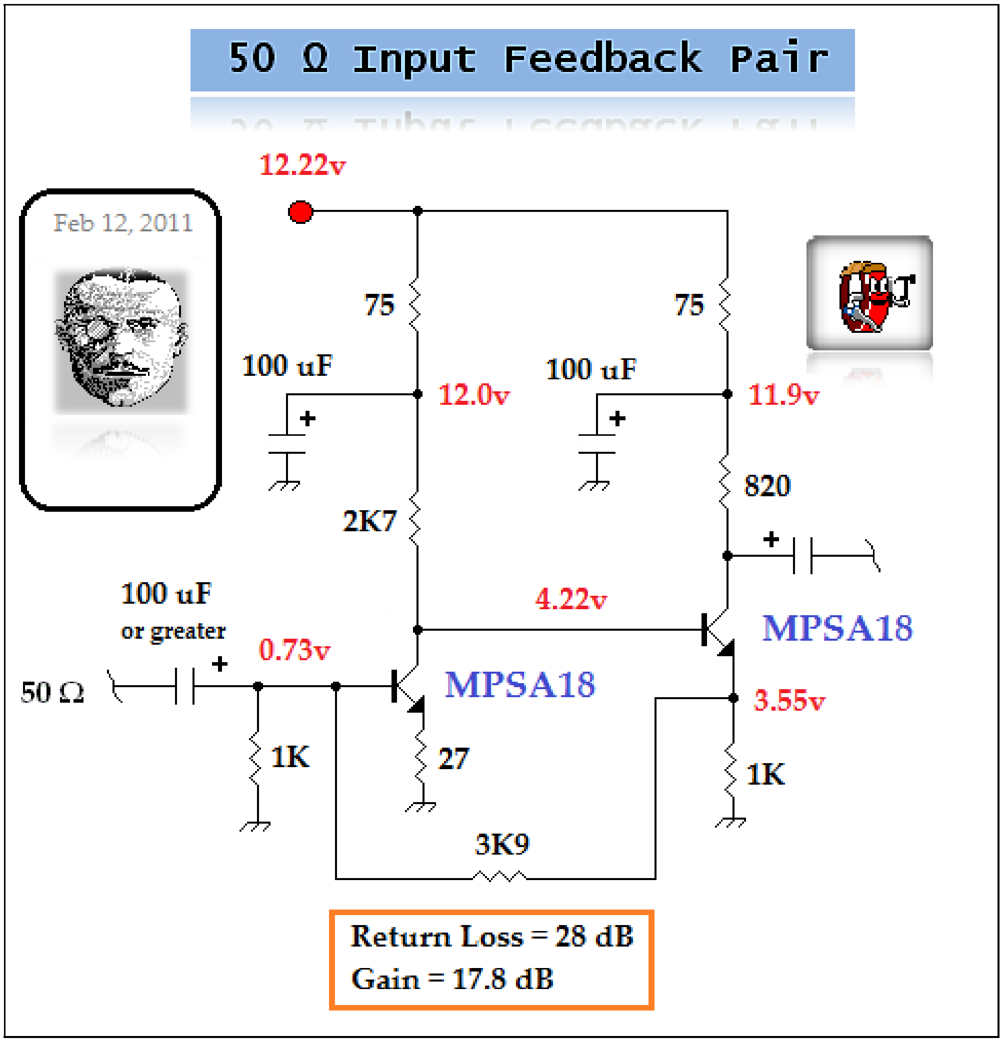
About 1/4 of my resistor collection are 1% metal film, as I don't like stocking both 5% and 1% tolerances of the same R value. I envision this amp driving a 5532 amplifier/low-pass filter built with 5% resistors with the op-amp output connected to a 1-10K volume control potentiometer.

The amp above is pure popcorn fluff, yet represents over 8 hours of reading, calculations and bench discovery. In another version, I soldered in low noise transistors (MPSA18 for example) and 1% metal film resistors to try and drop the noise figure. Yet another stage had Q1 collector bootstrapping (3 layers of feedback). I'm not sure what I'll do with these experiments, however, at least 1 of these will end up on the module page as the update to the popcorn DC receiver mainframe.

Glenn, VE7DNL built a landmark common emitter preamp with parallel 2N4401s, shunt feedback and some 1% bias resistors. I built a version of his pre-amp without the diplexer. The input return loss in my version was only 5 dB — likely the diplexer network helps establish the input match. It appears Glenn's goal was low NF and the best match and noise figure don't always correlate. I matched Q2 and Q3 as well as I could since only 6 were available. An Hfe of 220- 240 is not uncommon for the 2N4401 — a good part.

Here's a SPICE plot of the amp performed by Wes, W7ZOI. The two plots are S21 and S11. S21 is the forward gain in dB. S11 is the input reflection coefficient, which is just the negative of the return loss. At 1 kHz, the calculated gain = just under 42 dB and the input match = ~17 dB return loss.

To improve return loss in my breadboard, I need to tweek Q2 and Q3 bias (lower the 6.04K resistor), plus add some emitter degeneration. Emitter degeneration (series feedback) would worsen the noise figure. The 100 uF emitter caps offer some series feedback and capacitor values like 1000 uF are needed if "true" AF bypass is wanted.

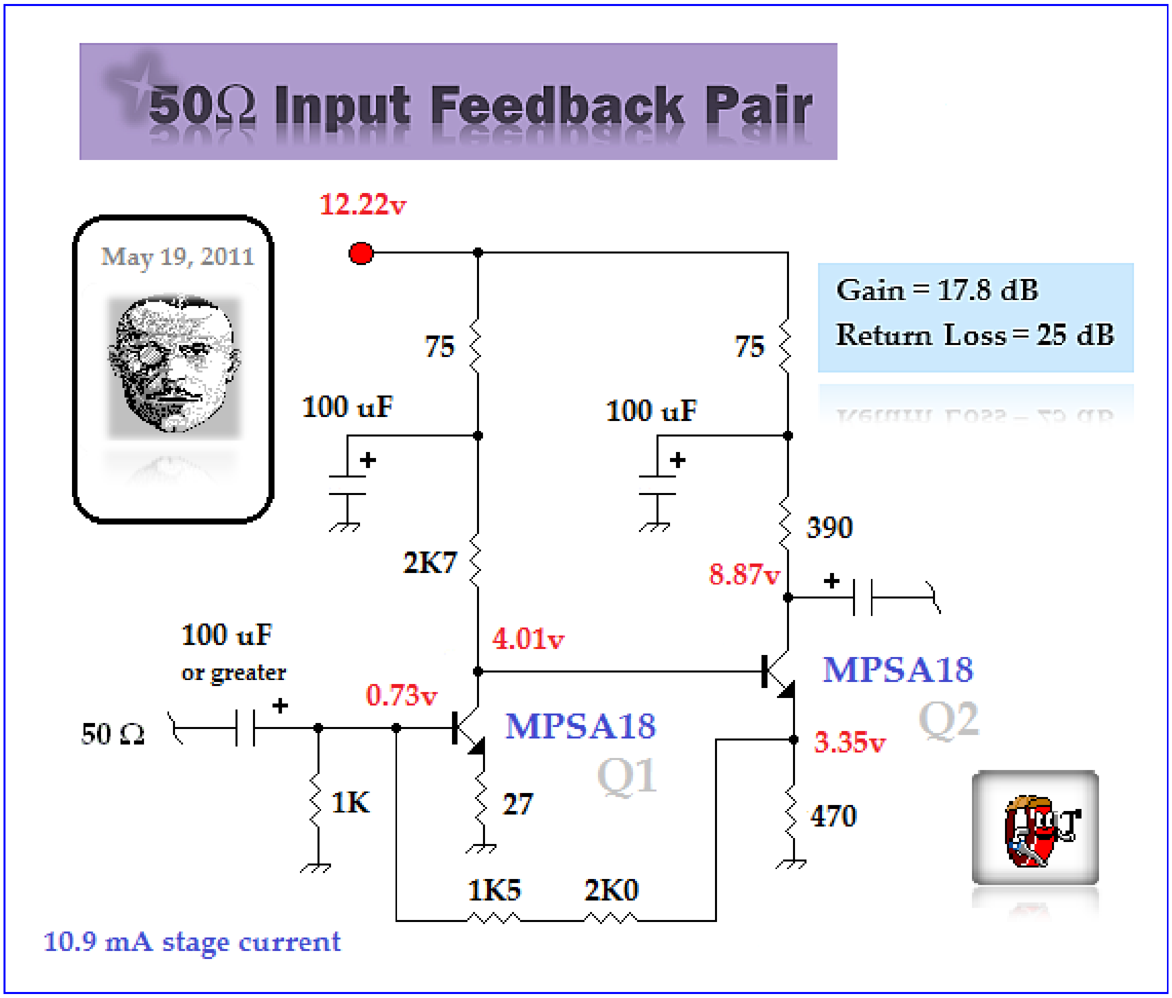


Above — Alternate experimental 50 ohm audio preamp to follow a diode ring product detector. The high impedance output enables small value output coupling capacitors for pseudo high-pass filtering. I simply added a collector resistor plus a decoupling network to Q2. The low noise MPSA18 BJT is a favorite and predictable part. The 2N3904, 2N4401 and others should work fine.

Don't even think about using an input capacitor less than 100 uF at 50 ohms — you'll trash the input Return Loss. I measured the highest Return

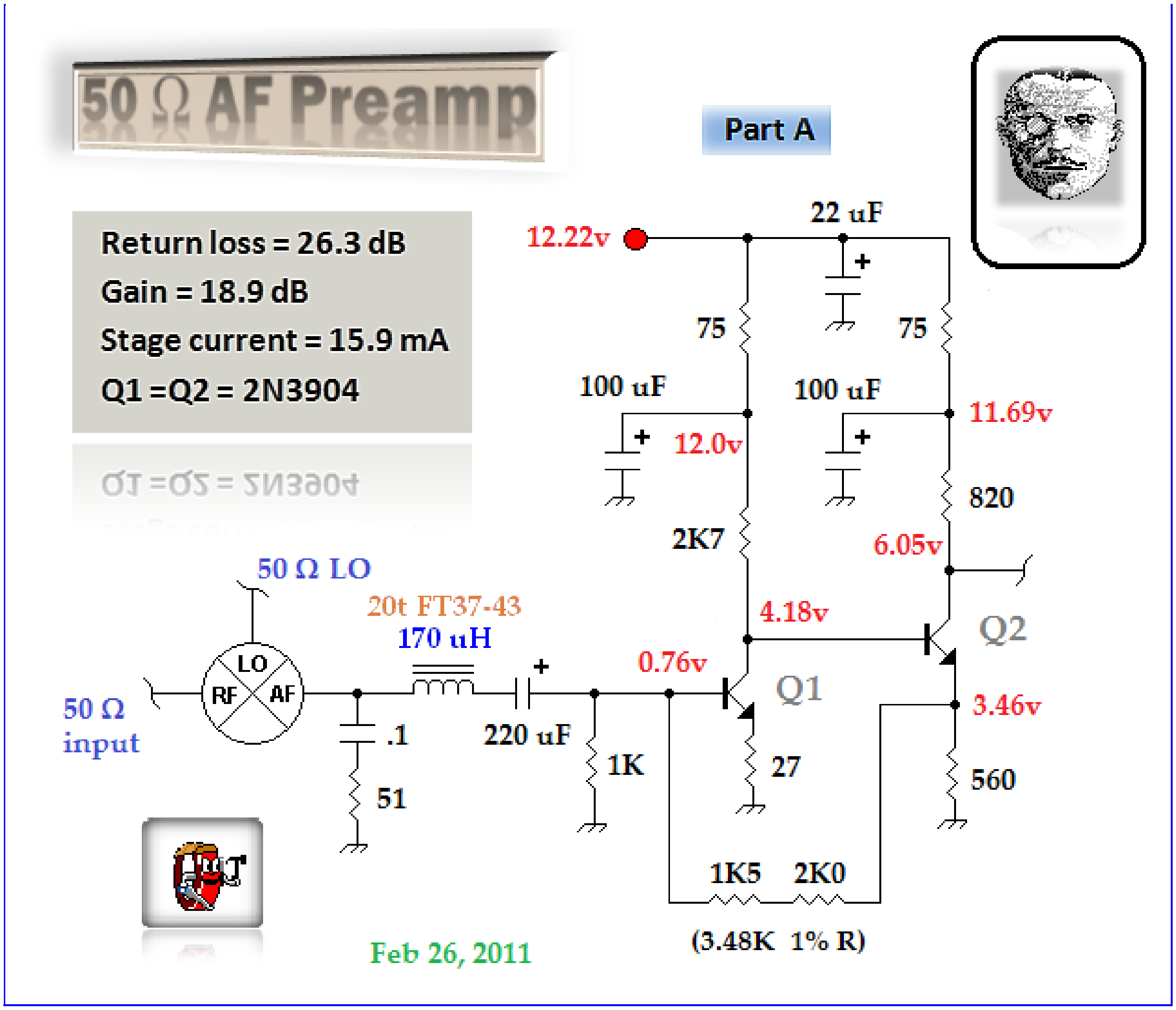
Loss with a 470 uF coupling capacitor, but the 100 uF cap shown worked almost as well.

I'm currently decreasing the (Q2) 1K emitter resistor with hopes of squeezing out a little more gain while trying to preserve strong signal handling capacity. I've found that heavy feedback is your friend.



Above — Another version; trying to increase stage gain. The Q2 collector resistor dramatically affects clean signal handling — lowest distortion occurred with a 367 ohm collector resistance. I substituted a 390 ohm collector R. This dropped the gain a little, but increasing distortion over gain is never my goal.

During experiments, dropping the the Q2 emitter resistor below 470 ohms made the other biasing resistors more critical, decreased return loss and invited distortion. The best return loss occurred when critically biasing with 1% resistors — I'm avoiding that!

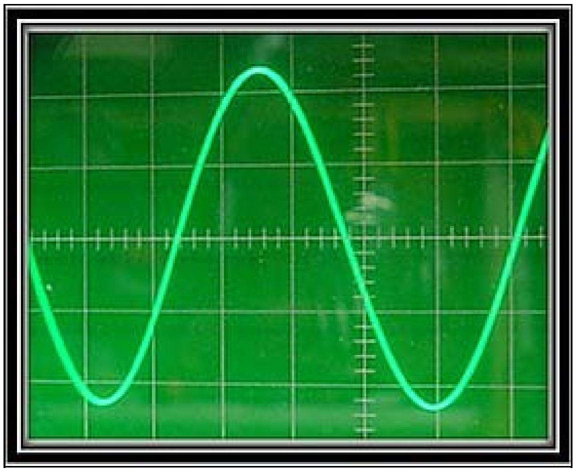


Above — Another popcorn part experiment from February. A low-pass network terminates a diode ring mixer and drives a 50 ohm input feedback pair arranged so the Q2 collector DC voltage is approximately 1/2 VCC — this provides the DC bias for a 5532 op-amp low-pass filter arranged in the Sallen-Key topology.

The Q2 collector voltage removes the need for a separate single supply op-amp bias network and an AC coupling cap. I didn't try this circuit in a receiver; arranging the feedback network and testing the possibility of such a design drove this experiment. A perfect 1/2 VCC Q2 collector voltage might be obtained with a standard 1% value 806 ohm R. I lack this part.

RF — Test and Measurement

Almost Popcorn Superhet



1. Receiver Concept and Macro Diagram

Not done.

Maybe a flight-of-fancy that never crosses the finish line? This page seems to repeatedly find the back burner; a good intention that never sees completion. Sadly this is normal for some of us builders — how many incomplete projects adorn your home and workshop? Still, this web page might fuel your *own* experiments.

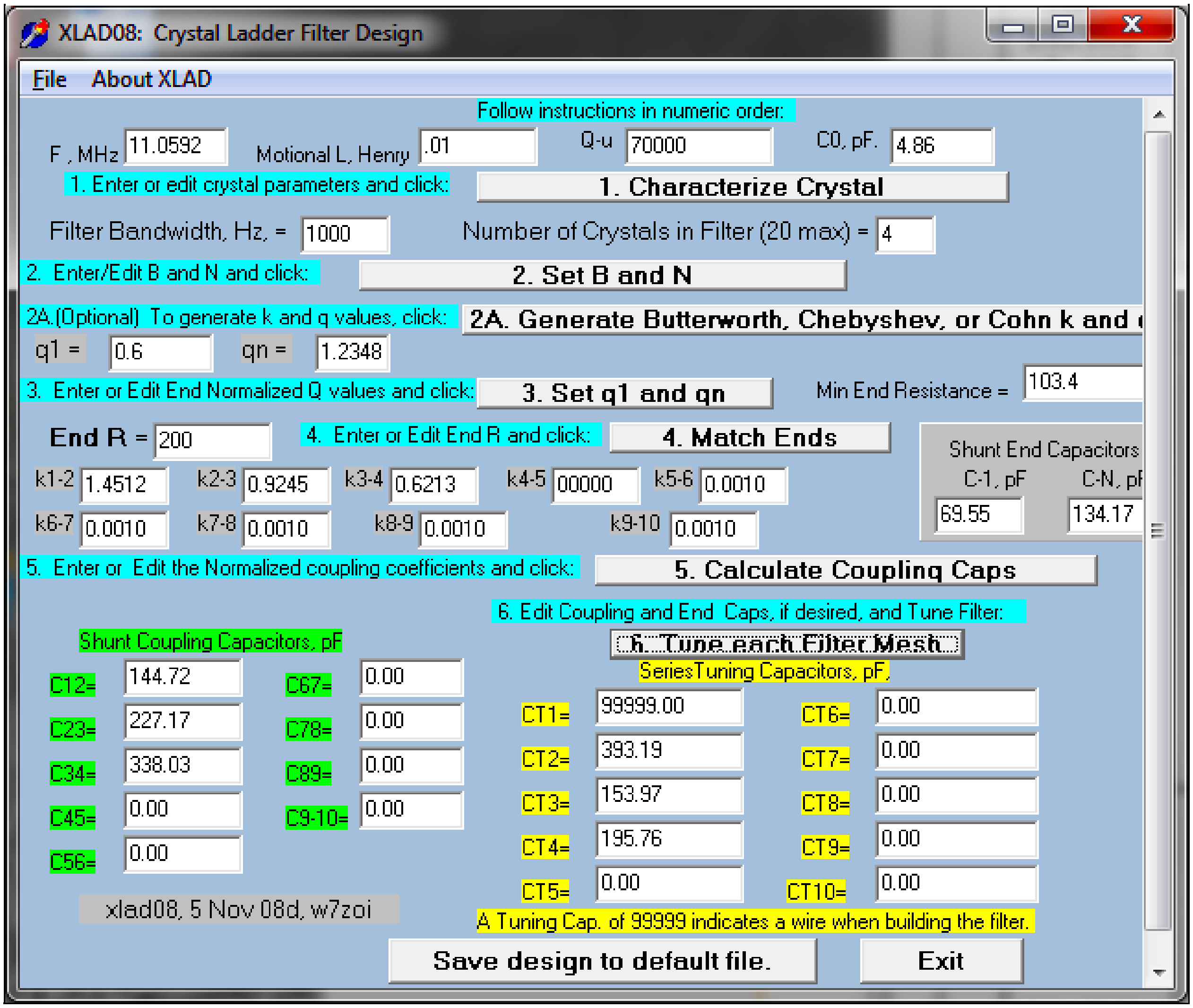
For years, I've attempted to update my Popcorn Superhet from 1998 but always seem to lose interest from numerous distractions such as VHF and UHF experiments and other non-radio stuff. So here it is so far — raw and unrefined: more experiments from my analog-centric workbench. To clarify — I love the new digital radio stuff, but that's well covered on the net by numerous code writers and those bleeding edge, frontier-pioneer types. I prefer a 50 Ω lab with all the visceral enjoyment and learning it has to offer. To each his own.

This web page remains unlisted on the main top level menu — and for good reason.

Click for some supplemental lab notes.

2. N = 4, Gaussian-to-6 dB Crystal Ladder Filter

I frequency matched four 11.0592 crystals, characterized them and took an average of those crystal parameters plus some data from Zverev and inputted this into xlad08.exe, a program that ships with EMRFD.



Follow instructions in numeric order:

F , MHz 11.0592
Motional L, Henry .01
Q-u 70000
C0, pF. 4.86

1. Enter or edit crystal parameters and click:

Filter Bandwidth, Hz, = 1000
Number of Crystals in Filter (20 max) = 4

2. Enter/Edit B and N and click:

2A.(Optional) To generate k and q values, click:

q1 = 0.6
qn = 1.2348

3. Enter or Edit End Normalized Q values and click:

Min End Resistance = 103.4

End R = 200

4. Enter or Edit End R and click:

Shunt End Capacitors

C-1, pF	C-N, pF
69.55	134.17

k1-2 1.4512
k2-3 0.9245
k3-4 0.6213
k4-5 00000
k5-6 0.0010
k6-7 0.0010
k7-8 0.0010
k8-9 0.0010
k9-10 0.0010

5. Enter or Edit the Normalized coupling coefficients and click:

6. Edit Coupling and End Caps, if desired, and Tune Filter:

Shunt Coupling Capacitors, pF

C12= 144.72
C23= 227.17
C34= 338.03
C45= 0.00
C56= 0.00
C67= 0.00
C78= 0.00
C89= 0.00
C9-10= 0.00

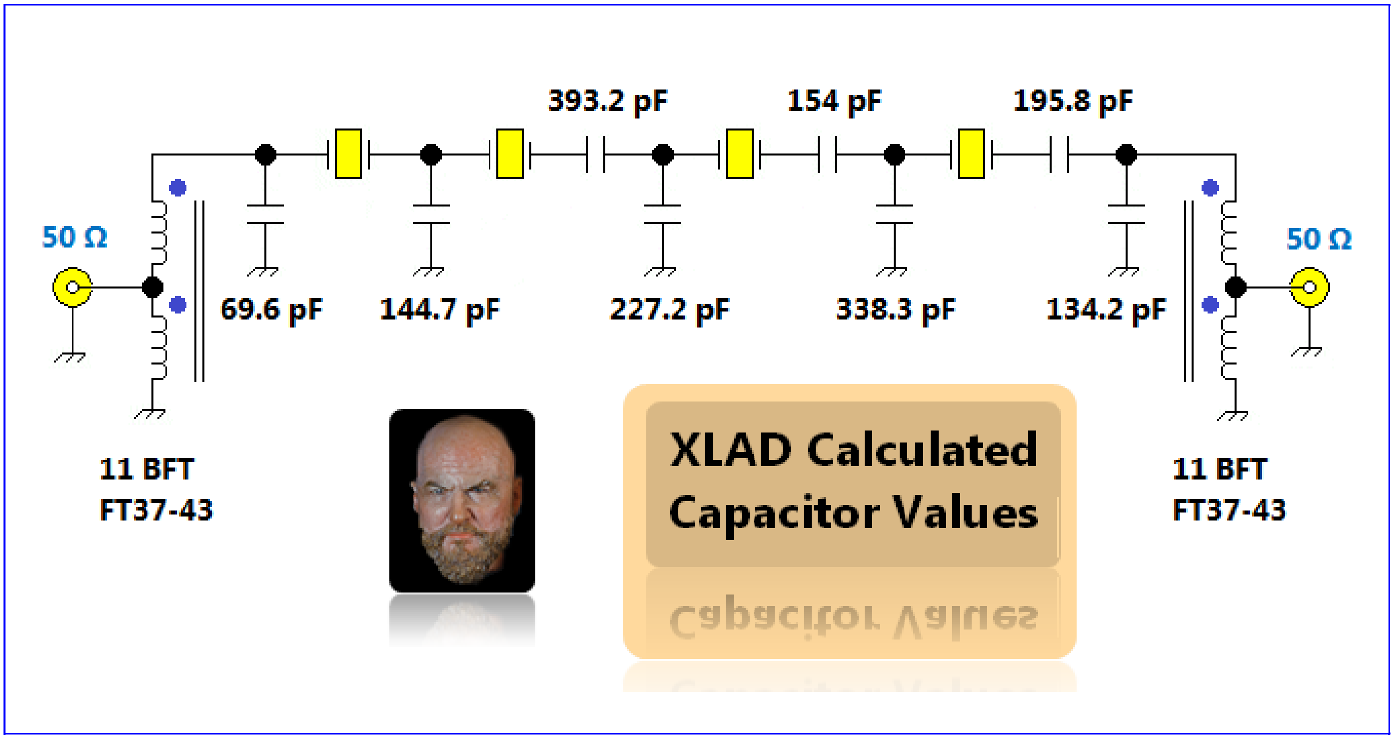
SeriesTuning Capacitors, pF

CT1= 99999.00
CT2= 393.19
CT3= 153.97
CT4= 195.76
CT5= 0.00
CT6= 0.00
CT7= 0.00
CT8= 0.00
CT9= 0.00
CT10= 0.00

A Tuning Cap. of 99999 indicates a wire when building the filter.

xlad08, 5 Nov 08d, w7zoi

Above — Xlad08.exe screen shot.

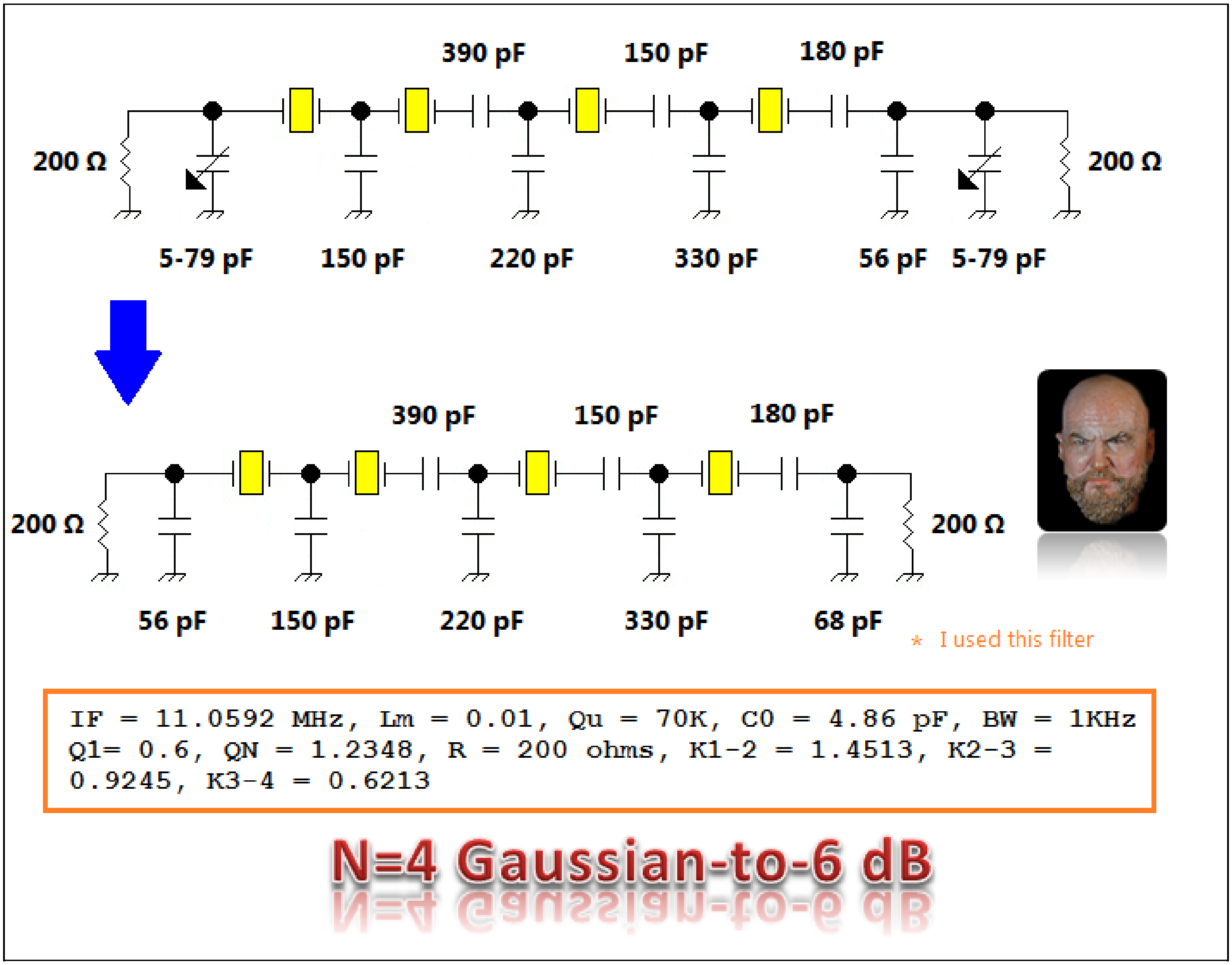


Above — The xlad filter expressed as a schematic. Click for a GPLA plot of the filter.

I built and hand tuned this crystal filter with my analog sweep system. Click for a photo of the optimized filter breadboard — I used 1 or 2 parallel capacitors to tune each capacitance to derive the best possible skirt shape. Click for a sweep of this optimized filter in my 'scope. 2 bifilar transmission line transformers provided the required 200 Ω port termination in my 50 Ω sweep system.

I sought a crystal filter based on the xlad08 calculated values using only 1 nearest standard value in each capacitor slot.

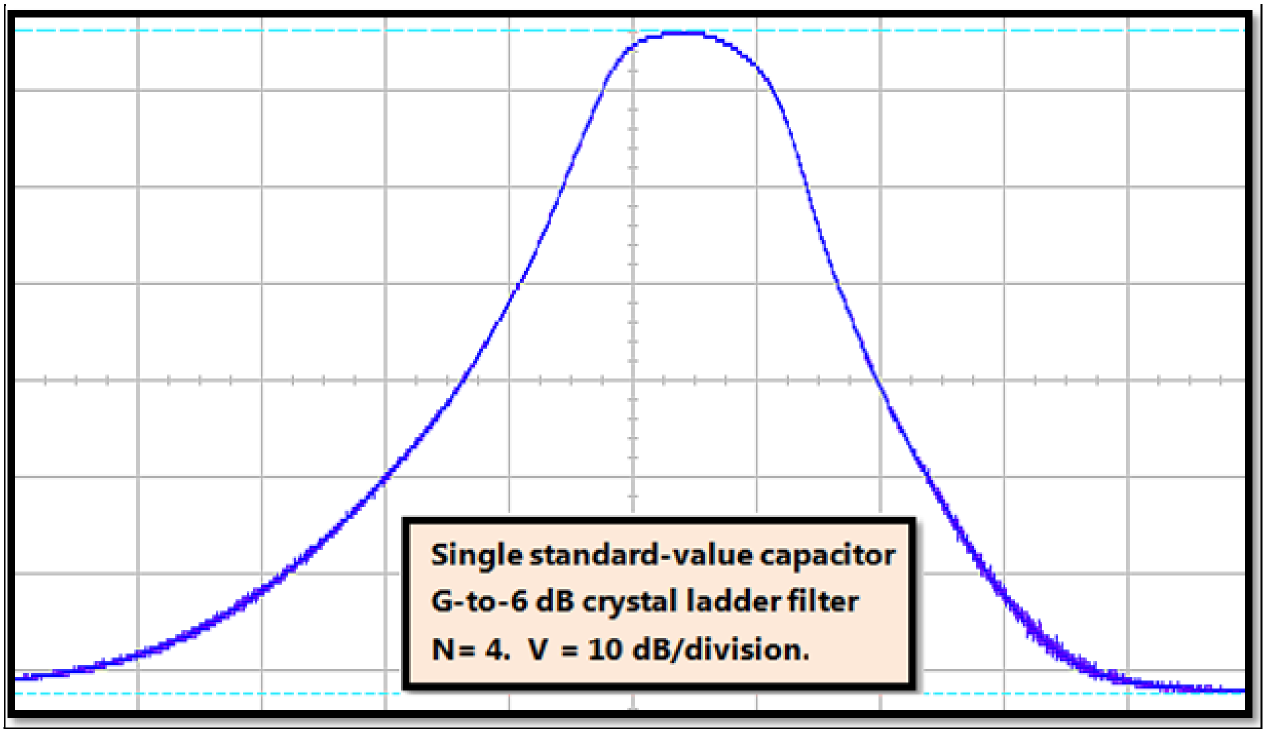
Transitional filters like the G-to-6 dB, are generally tolerant of part substitutions and thus I sought a filter with only 1 nearest 5% standard-value capacitor in *every* slot instead of 2 sticking 2 close in value parallel caps to make up the needed capacitance as needed.



IF = 11.0592 MHz, Lm = 0.01, Qu = 70K, C0 = 4.86 pF, BW = 1KHz
Q1= 0.6, QN = 1.2348, R = 200 ohms, K1-2 = 1.4513, K2-3 = 0.9245, K3-4 = 0.6213

Above —The 2-step transition to a 1 capacitor, nearest standard value in each capacitor slot. In the top diagram, I placed a trimmer cap on each end to allow filter tweaking while looking at the swept filter in my 'scope. I've learned that adjusting the parallel cap nearest to each port a simple way to tweak a G-to-6 dB filter. After setting the best looking skirt, I removed and measured the end parallel capacitance and substituted the nearest standard fixed capacitor. Manipulating the capacitors values will change filter bandwidth and return loss.

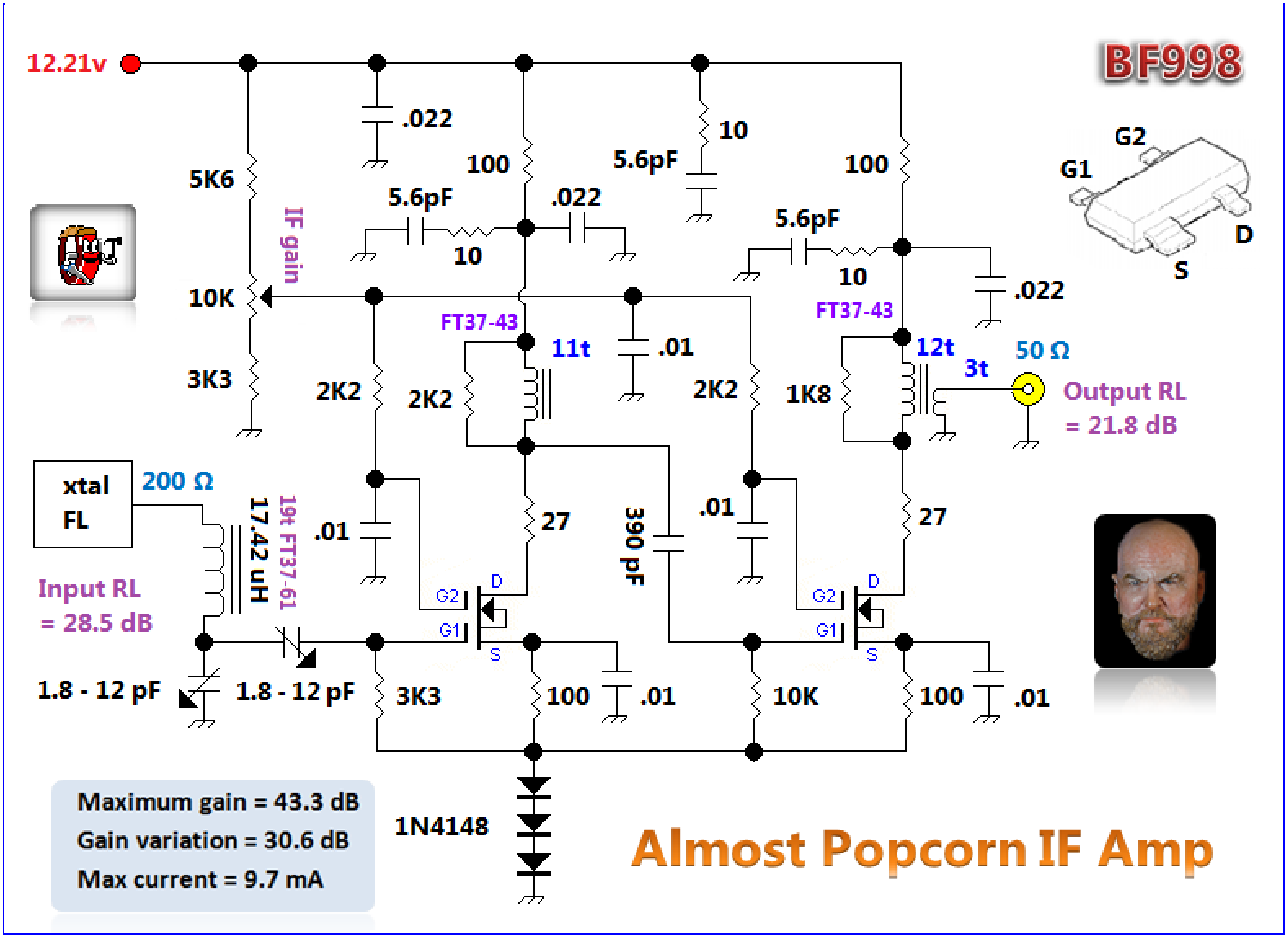
You might compare the lower filter to the xlad calculated values to see how tolerant the circuit is.



Above — A sweep of the single value capacitor version that's used in the Almost Popcorn receiver. While the 1 KHz filter skirt isn't perfect, it's reasonable. Click for the test bench breadboard. This particular filter measured 1/2 power bandwidth = 1227 KHz and the insertion loss was ~3.6 dB.

---

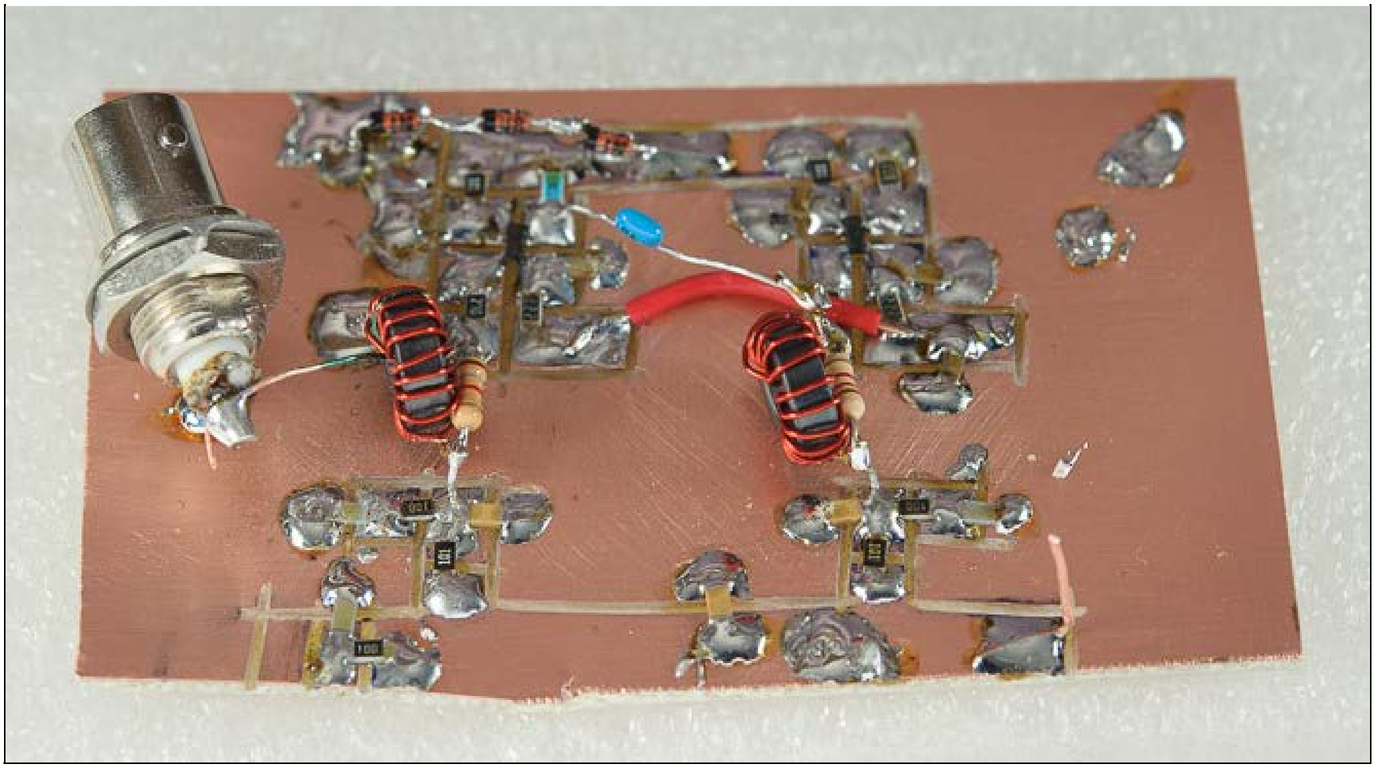
3. IF Amplifier



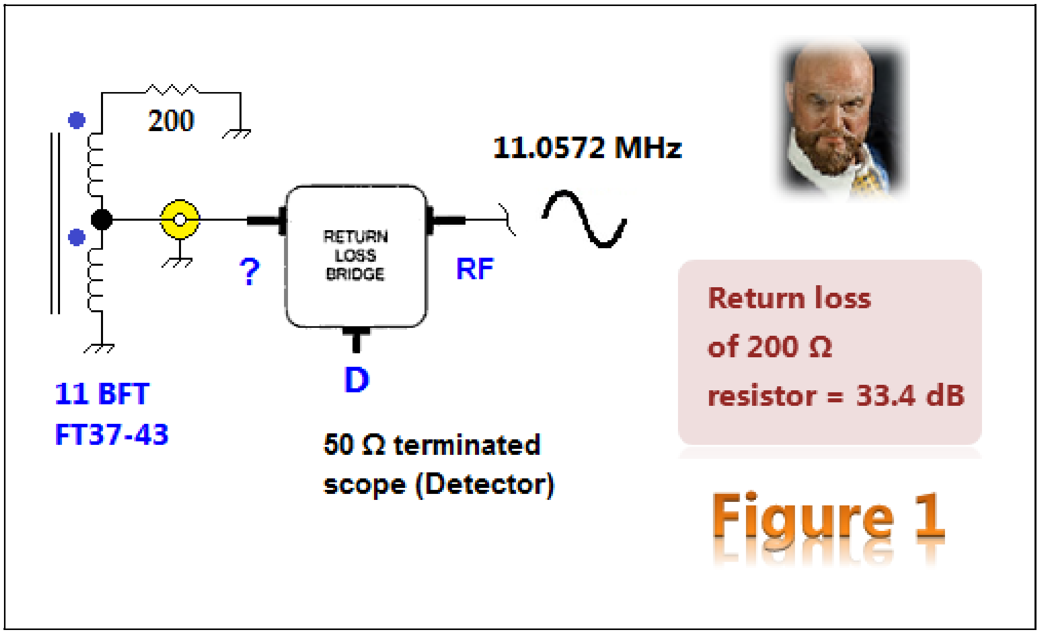
Above — Version 2 of a popcorn IF amplifer idea with BF998s. IF = 11.0572 MHz.

I have no use for AGC in my personal receivers since I listen through a speaker and *always* ride the RF gain control with my finger. Since the BF998 has gain into UHF, wideband decoupling + bypass SMT R-C networks filter each MOSFET in case of oscillations. Click and click for photos. This circuit will operate fine with just 0.022 μF bypass caps.

A network matches the 200 Ω xtal filter output impedance to the 3K3 input Z of the IF amp cascade. The L-C-C network has become a favorite matching weapon — 2 tweaks is often better than the 1 offered by a garden variety L-network. The product detector that follows this stage is a Level 7 diode ring mixer soldered onto the audio preamp board. This receiver will feature some modish circuits aimed at low noise + high fidelity but still stay "almost popcorn". Anyhow, that's my hope.



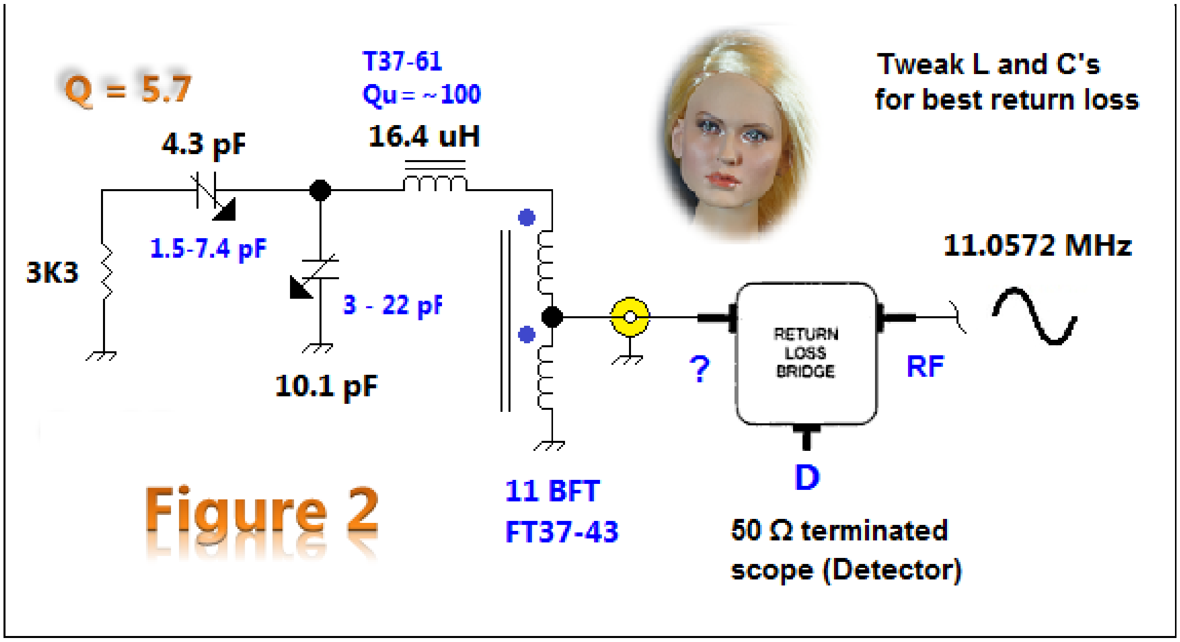
Above — A breadboard of the Popcorn IF amp sans the input network. I tested this stage using an oscillator with a 3K3 output impedance and liked what I saw. Click for a reverse angle view. After this build, I ordered some SMT versions of the 1N4148 diode. The BNC connector was only soldered on the board for testing purposes.



Above — Setting up to measure the return loss of the 200 Ω input Z of my popcorn IF amp. 1 difference between scratch homebrew and kit building is all the measurements you need to make to optimize your scratch receiver. A joie de vivre stems from our descriptive/exploratory analysis before and after making our breadboard. We just learn from mistakes and move forward to improve both our knowledge and gear.

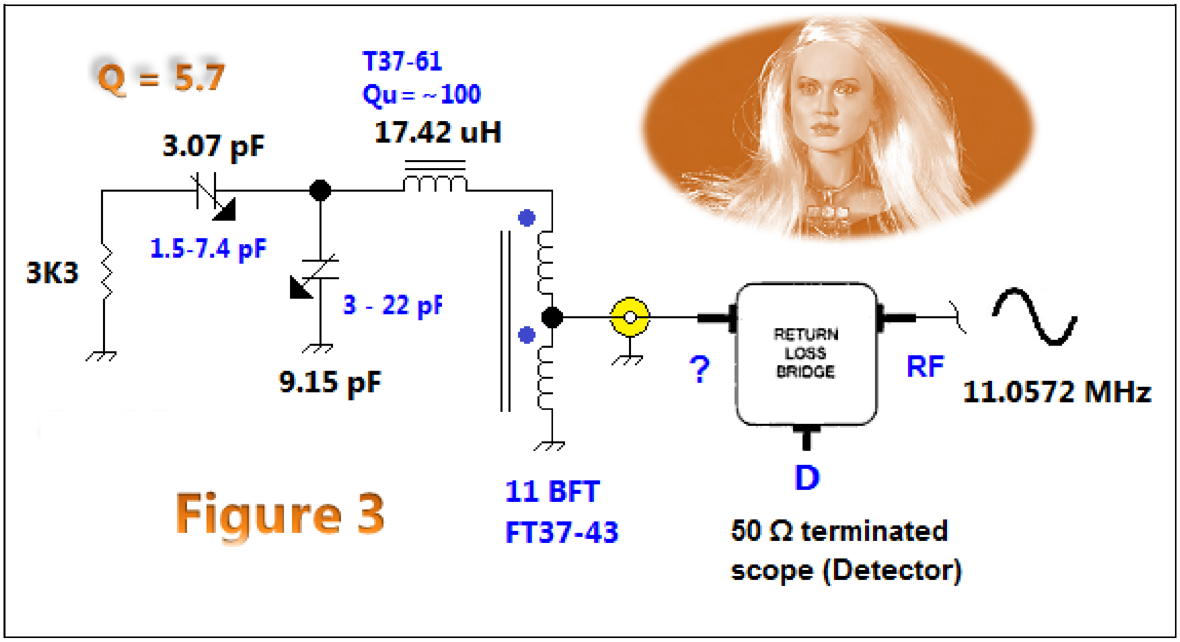
I chose a 200 Ω input impedance for a reason — it's easy to apply a standard 50 Ω return loss bridge via a 4:1 transmission line transformer to establish the correct input impedance for the single frequency matching network.

From the set-up shown above, I found that with a 200 Ω resistor, my return loss at 11.057 MHz = 33.4 dB. This signals the very best we can achieve with our network: directivity. Click for a photo of the above experiment breadboarded on a scap of copper clad board.

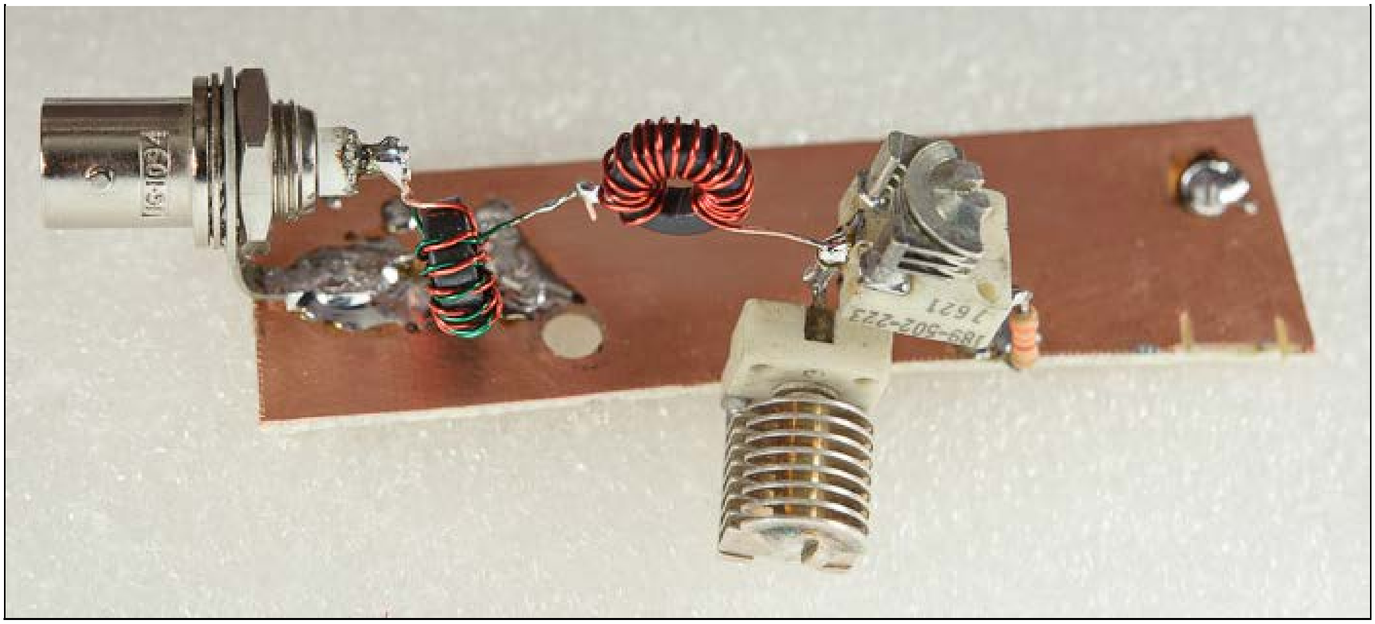


Above — The schematic of my 200 to 3K3 Ω matching network. I designed it with zmat08.exe, a program by Wes that I first used as a console app since the days of DOS. *Zmat08 program comes with EMRFD.* Click for a screen shot of from Zmat08.

The calculated values from Zmat08 only consider a pure resistance and not a complex impedance, so on-bench adjustments of the design L and C values are normally required and fun to do. Variable trimmer caps make tweaking the C easy.



Above — The final— *measured* schematic — While watching the peak to peak voltage on my 50 Ω terminated scope, L and both capacitors were tweaked until I found the highest return loss. L is tweaked by scrunching or expanding turns, or adding/removing turns as needed: in this case, I added a turn and expanded the windings slightly. With the L-C-C match, a return loss greater than 25 dB may arise plying good bench practices — mine measured a lovely 29.2 dB. I removed and measured the inductor plus 2 capacitors to get the values shown **black** in Figure 3.



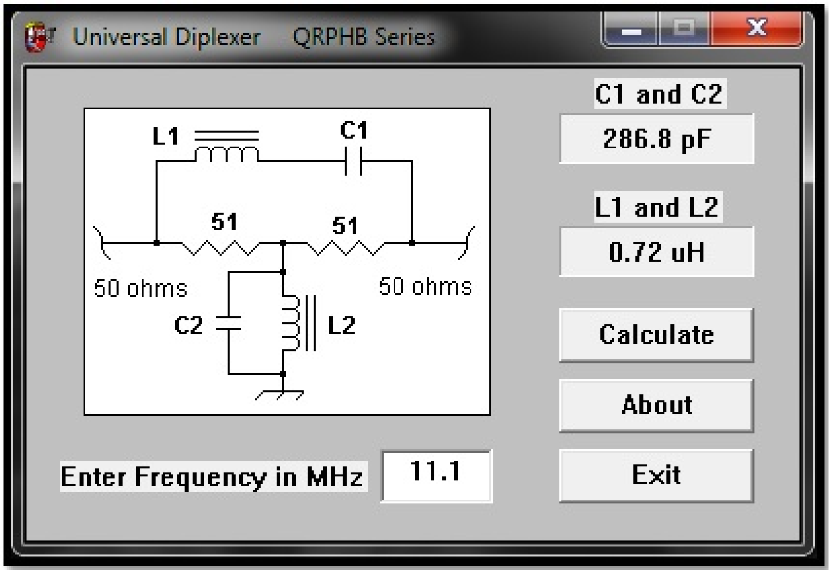
Above — A breadboard of the Figure 3 network before I experimented to find the best L (and shortened its leads).

After I making and tweaking the Figure 3 network, I took the coil along with 2 cheaper trimmer caps and added them to my IF amplifer breadboard. While measuring the return loss of the IF strip input, I had to expand a couple of links on the inductor, but after tweaking the L and the 2 trimmer caps, I measured a 28.5 dB return loss. This ensures my crystal filter will see close to a 200 Ω termination at the IF input. I love well matched stages!

Further, I made a L network and then a L-C-C network to match the output port, however, UHF oscillations erupted. I then went back to a simple broadband transformer and measured a RL of 21.8 dB with a 12:3 turns ratio. Done.

---

4. Mixer and Post-Mixer Amplifer

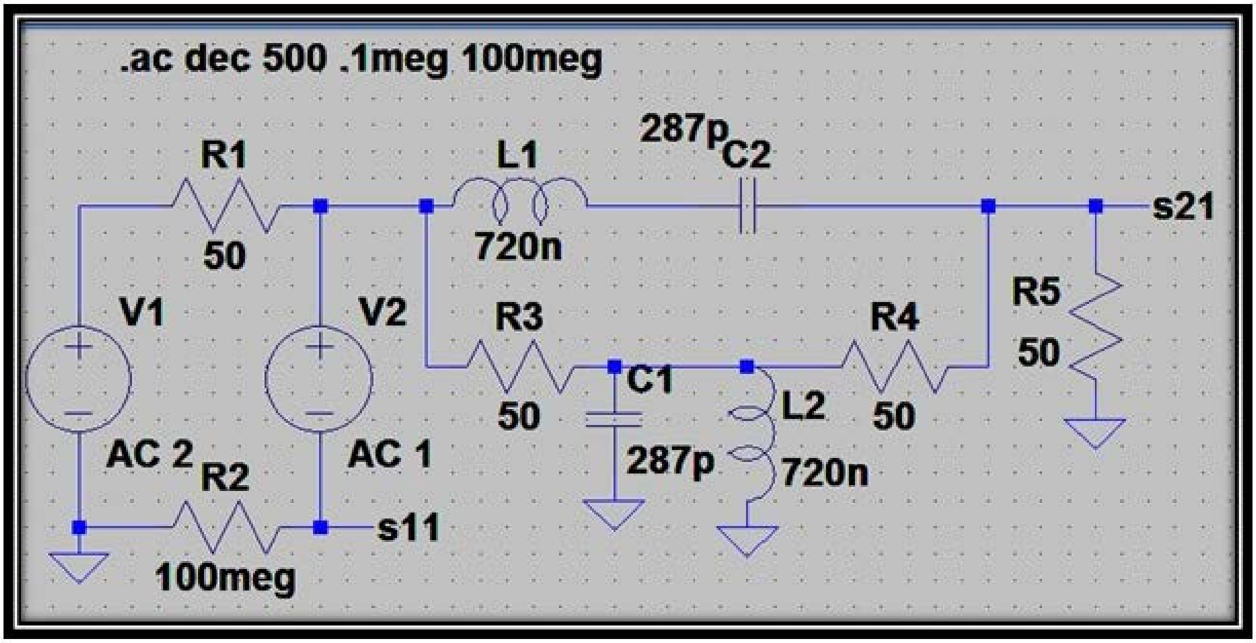


C1 and C2: 286.8 pF

L1 and L2: 0.72 uH

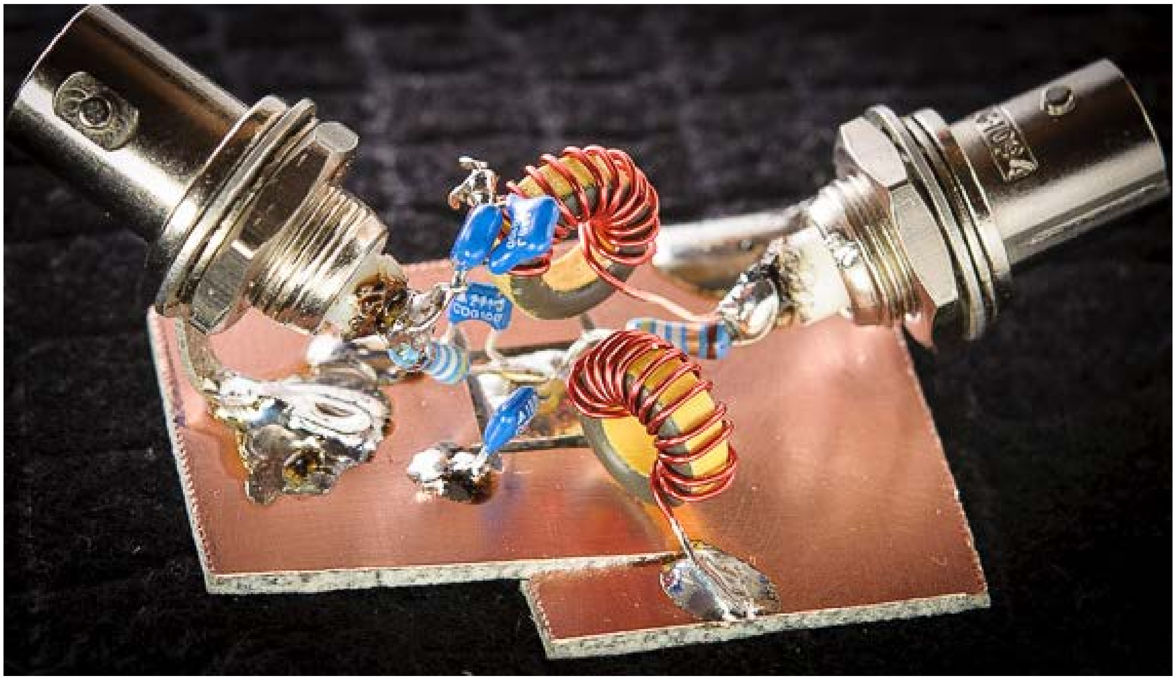
Enter Frequency in MHz: 11.1

Above — I tested the W1JR (Joe Reisert) bridged-tee diplexer against another design used by Ten-Tek. I'll show the outcomes on the Almost Popcorn receiver page some day, but the W1JR better fullfills its purpose; termination from DC to daylight at the mixer IF port. I calculated the L and C values for a 11.0592 MHz IF with my Universal Diplexer application from many moons ago.

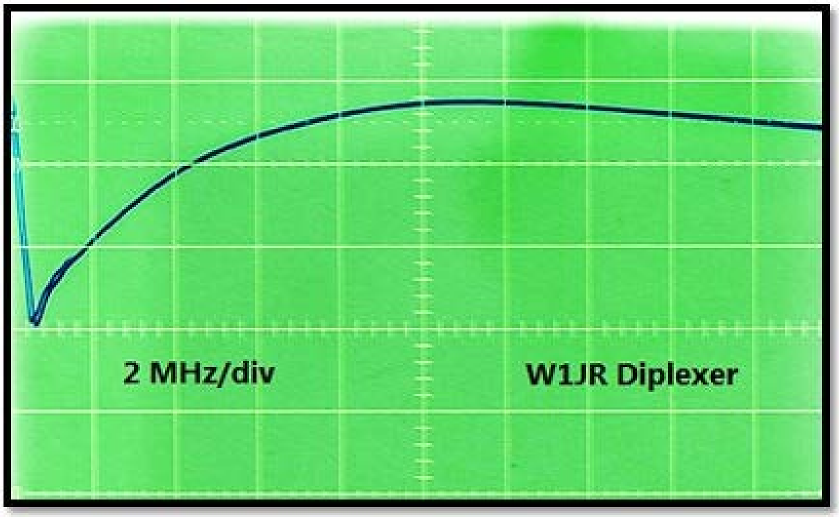


Above — SPICE provides a great way to assess a diplexer since you can plot both S21 and S11 against frequency.

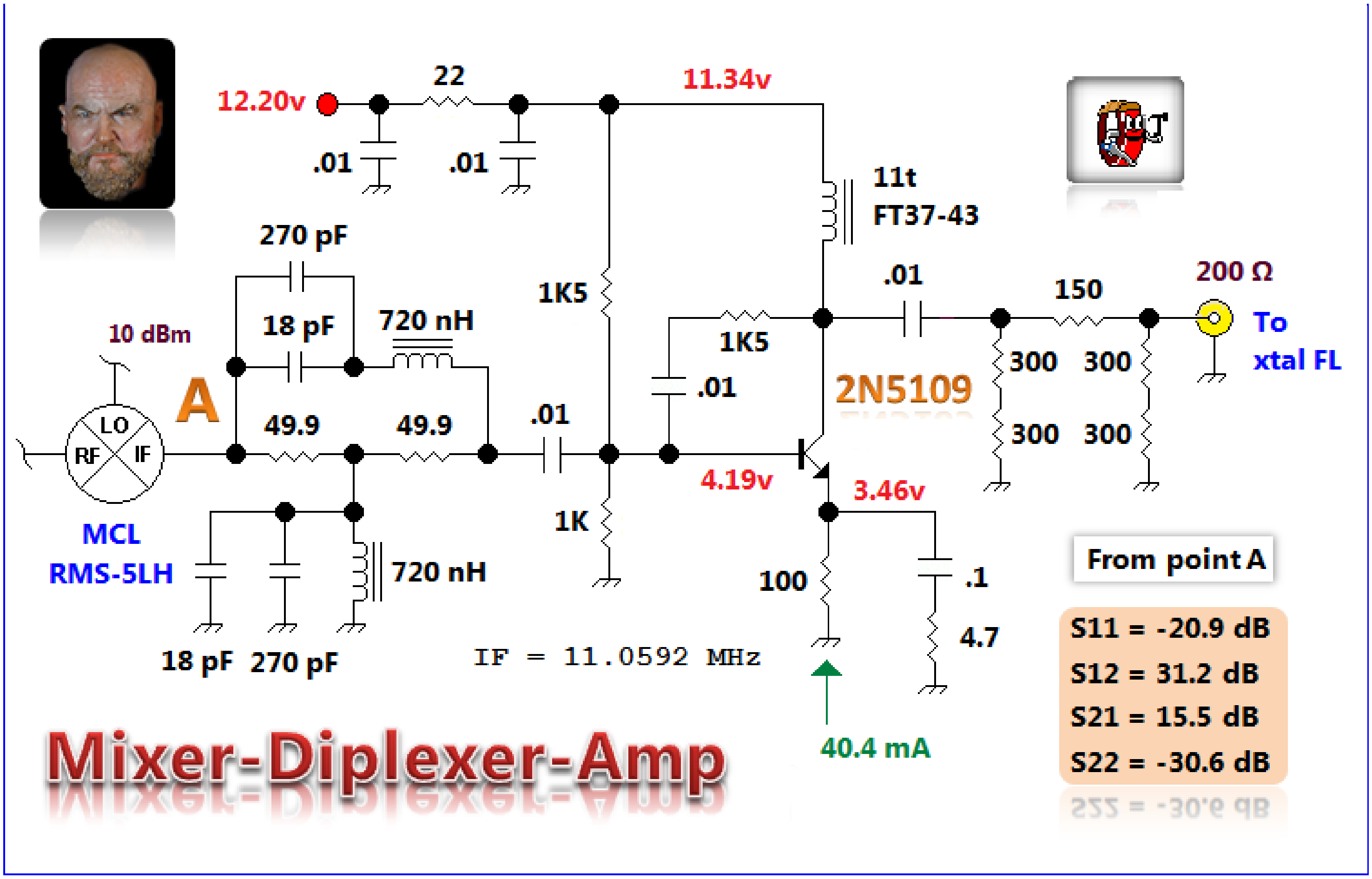
Above — A fantastic match! Thanks Joe for the design and to **Wes, W7ZOI** for his help with diplexers and the SPICE plots through many years of related experiments. Since web publishing the diplexer page, at least 500 diplexer-related emails have come in. Human translated versions of the web page exist in at least 3 languages.



Above — The evaluation breadboard of my W1JR diplexer. Those are T30-6 toroids but they look huge in this photo.



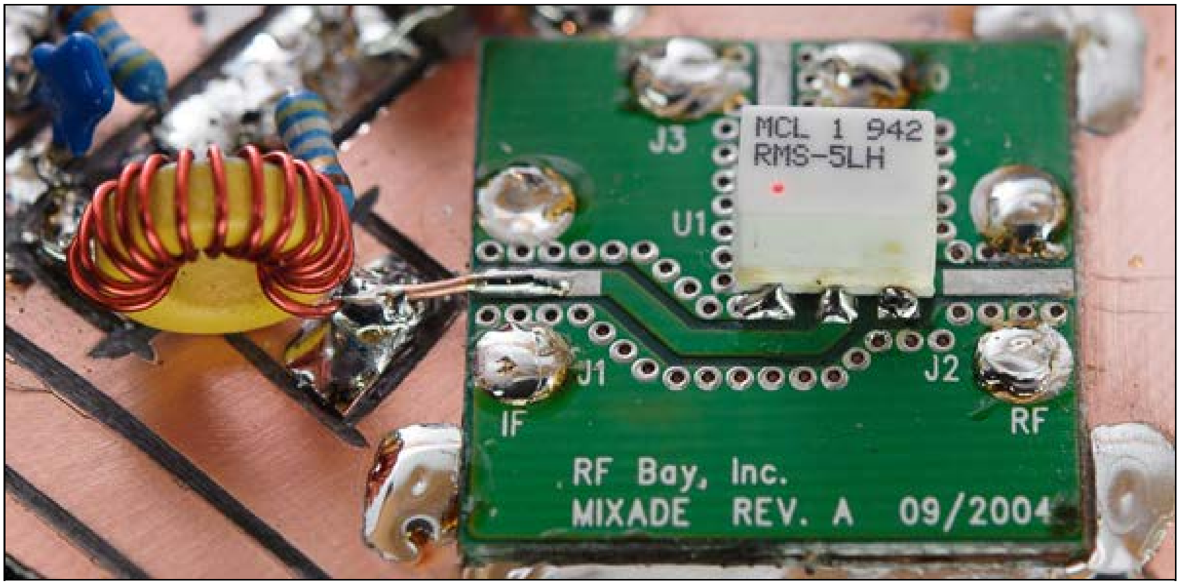
Above — Tracking generator plus spectrum analysis sweep. SPICE does a better of examining the high-pass side.



Above — A look at the diode ring mixer, diplexer and post-mixer amp from the latest version of my Almost Popcorn Receiver. I tested the diplexer and RF amp with pad as one unit (see the S-values).

It did not make sense to use a transmission line transformer to get 50 Ω on the amp collector and then use another transformer to build it back up to the needed 200 Ω impedance for the crystal filter, so I just employed a single choke driving a 200 Ω Z attenuator pad.
S parameters are 50 Ω values, thus S22 = the 200 Ω transformed into 50 Ω. After terminating the stages 200 Ω output with a 4:1 Z transmission line transformer, I tested it like any other 50 Ω circuit andmeasured S22 as -30.6 dB — my xtal filter input Z is well established.

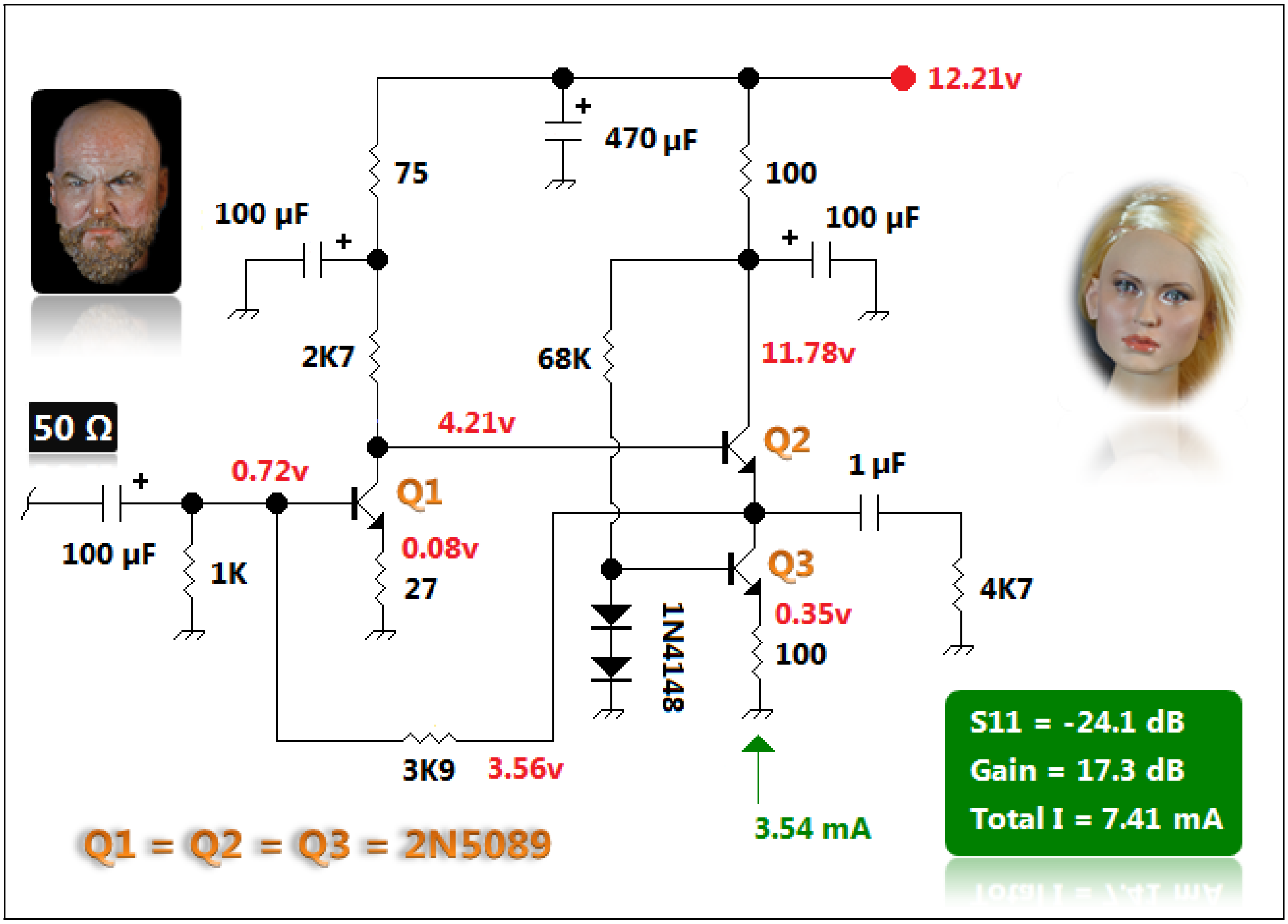
Even with the 6 dB *(200 Ω)* attenuator pad and losses from the diplexer, the stage gain from point A = 15.5 dB. Click for to view a bigger picture of the breadboard. Losses from a diode ring mixer are typically 5 - 7 dB.



Above — Mixer and diplexer assembly. The RMS-5LH takes 10 dBm LO drive.

5. Product Detector and Post Product Detector AF Amplifer

I kept the diode ring for a product detector: an SBL-1 driven with a xtal oscillator (BFO).



Above — My post-product detector amplifier: a common emitter/ emitter follower pair arranged for a 50 Ω input impedance with shunt and series feedback. Getting the biggest, clean sine wave in and out posed my only goal — the 3.54 mA current source on the follower helps that cause. The input Z tested quite insensitive to output load variations and the (S11) return loss = 24.1 dB.

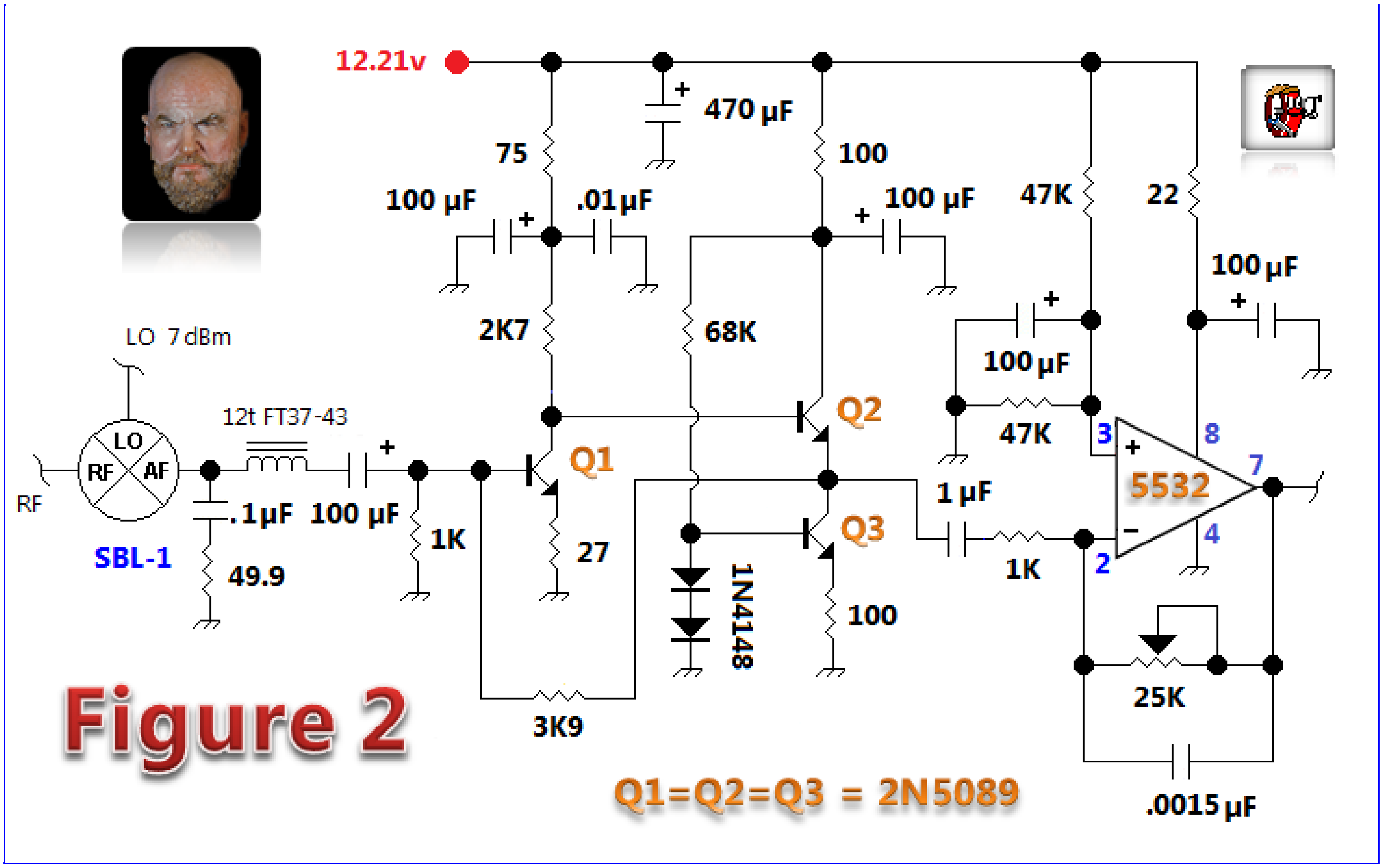
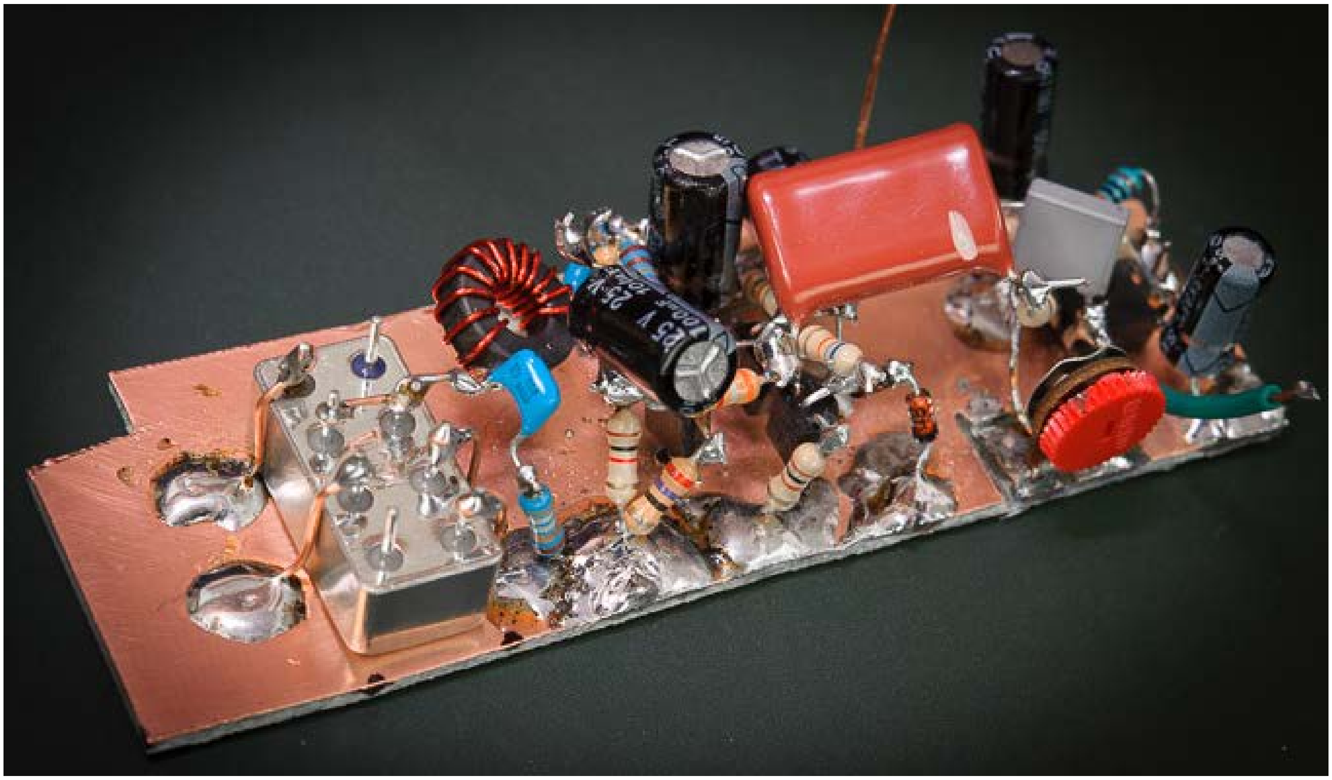


Figure 2

Above — The "Full Monty product detector to AF preamp". A MCL SBL-1 diode ring mixer drives the 50 Ω input Z feedback pair and finally a 5532 op-amp with adjustable gain. The 25K gain trimmer pot allows the builder to accommodate factors such as whether the stage is a DC receiver, or an RF-AF block for a superheterodyne receiver like in this radio. I'll set the gain as low as possible and hope to build up my AC voltage with active, gain producing low-pass filter op-amp stages.

The output t pin 7 is directly coupled to an op-amp AF filter. With software we can easily design a myriad of audio filters: band-pass, low-pass, high-pass, all-pass etc. I only run low-pass filters in this slot and even here we must decide whether to make our reponse Gaussian to 6 dB (or 12 dB), Bessel, Butterworth, or a Chebyshev with some degree of ripple. I'm testing out a few designs with an eye towards simplicity and 10% parts.

Spurred on by curiosity, with my ears as the main evaluation tool; I currently have a 5532 paper design to deliver 20 dB gain with a 500 Hz pass-band, Gaussian to 12 dB low-pass response. Click for schematic, I chose a 500 Hz cut off frequency since the skirt shape by definition is subtle and the Q low. This trick should better roll off the AF above 1 KHz a little faster without the ugly group delay associated with steeper filters employing more poles. We shall see!

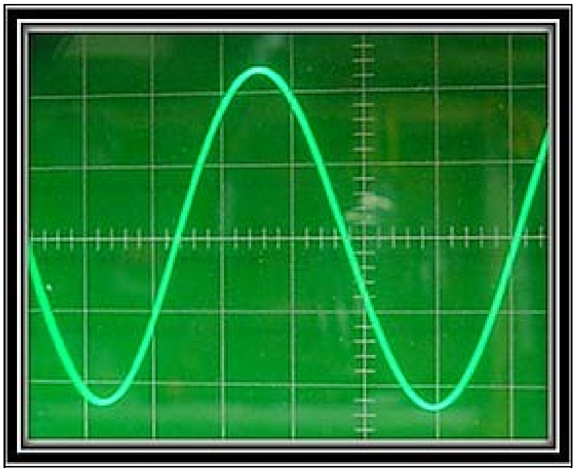


Above — The compact breadboard of Figure 2.

Audio filters and AF power amp are incomplete.....

RF — Test and Measurement

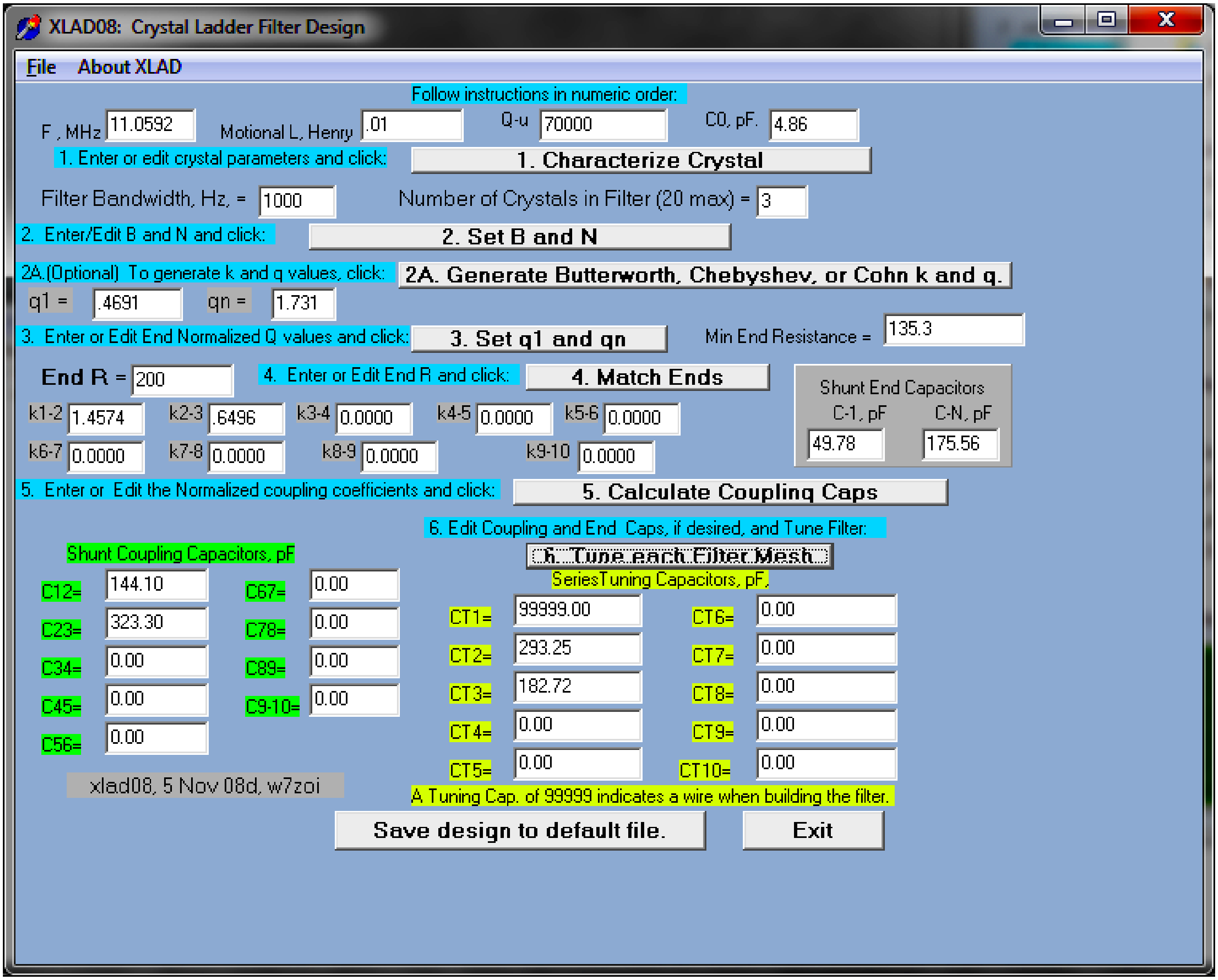
Almost Popcorn Superhet — Supplemental Lab Notes



These lab notes supplement the Almost Popcorn Superhet Receiver Web page which has remained incomplete for greater than 10 years. I'm sorry for the rough writing and procrastination.

N = 3 Gaussian-to-6 dB Crystal Ladder Filter

I spent a pleasant Saturday afternoon designing, building and testing Gaussian-to-6 dB crystal filters. I inputted some xtal parameters + constants from Zverev into Xlad08.exe from EMRFD. Here's the N = 3 version



Follow instructions in numeric order:

F , MHz 11.0592
Motional L, Henry .01
Q-u 70000
C0, pF. 4.86

1. Enter or edit crystal parameters and click:

Filter Bandwidth, Hz, = 1000
Number of Crystals in Filter (20 max) = 3

2. Enter/Edit B and N and click:

2A.(Optional) To generate k and q values, click:

q1 = .4691
qn = 1.731

3. Enter or Edit End Normalized Q values and click:

Min End Resistance = 135.3

End R = 200

4. Enter or Edit End R and click:

Shunt End Capacitors
C-1, pF 49.78
C-N, pF 175.56

k1-2 1.4574
k2-3 .6496
k3-4 0.0000
k4-5 0.0000
k5-6 0.0000
k6-7 0.0000
k7-8 0.0000
k8-9 0.0000
k9-10 0.0000

5. Enter or Edit the Normalized coupling coefficients and click:

6. Edit Coupling and End Caps, if desired, and Tune Filter:

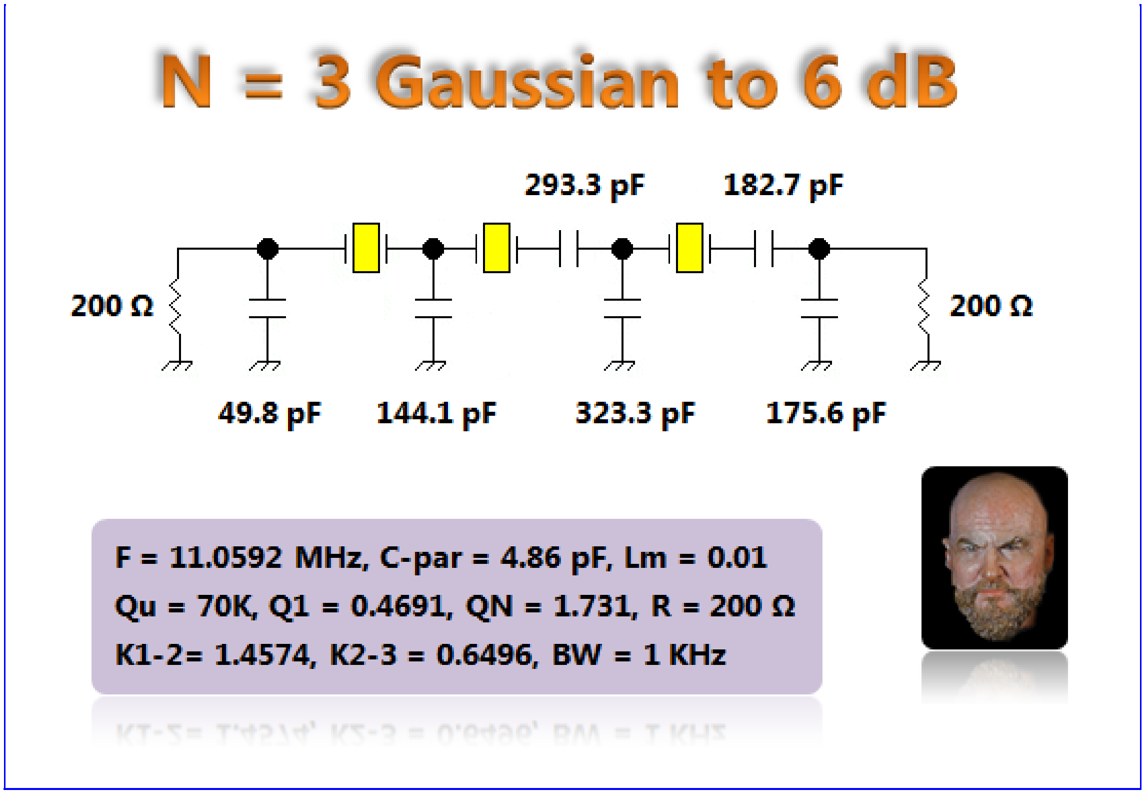
Shunt Coupling Capacitors, pF
C12= 144.10
C23= 323.30
C34= 0.00
C45= 0.00
C56= 0.00
C67= 0.00
C78= 0.00
C89= 0.00
C9-10= 0.00

SeriesTuning Capacitors, pF,
CT1= 99999.00
CT2= 293.25
CT3= 182.72
CT4= 0.00
CT5= 0.00
CT6= 0.00
CT7= 0.00
CT8= 0.00
CT9= 0.00
CT10= 0.00

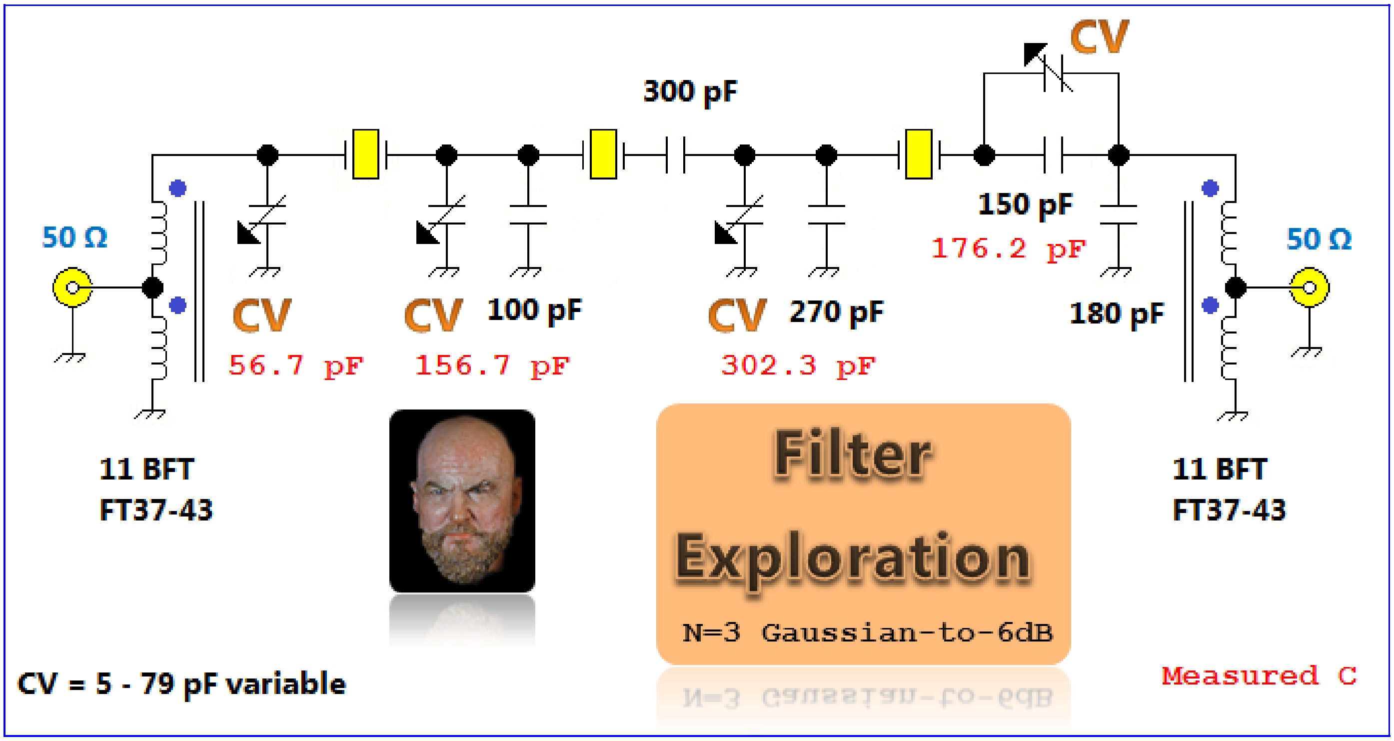
A Tuning Cap. of 99999 indicates a wire when building the filter.

xlad08, 5 Nov 08d, w7zoi

Above — A screen capture from Xlad. This app allows skilled builders to depart from the typical Cohn (Min-Loss) filter and tune filter poles to derive beautiful skirt shapes and low insertion loss if wanted. I'm not a great crystal filter designer, but I've seen work from friends who have a knack for it: imagine making a N= 8 to 10 crystal CW filter with just ~1 dB insertion loss like these builders! Click for the GPLA simulation.



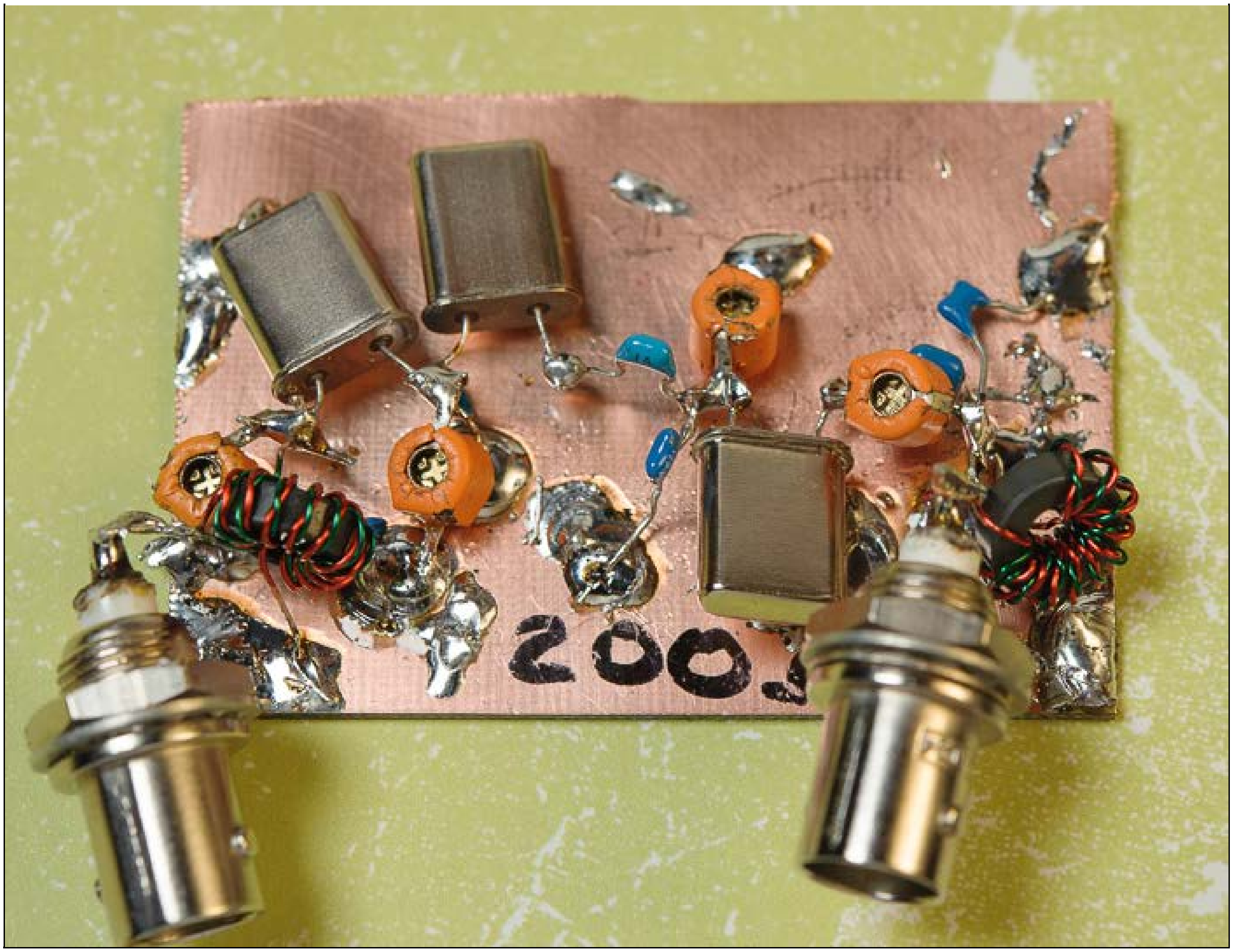
Above — Gaussian-to-6 dB xtal filer schematic where N = 3.



Above — Circuit to allow evaluation of the filter in my 50 Ω sweep system. Transmission transformers provide the needed 200 Ω filter termination. I inserted 4 trimmer caps to permit tweaking of the filter under test to establish the very best filter shape. With trimmer capacitor adjustment, I was able to trash the filter or peak it well. Click for 3 different swept bandwidths: Sweep-1 Sweep-2 Sweep-3.

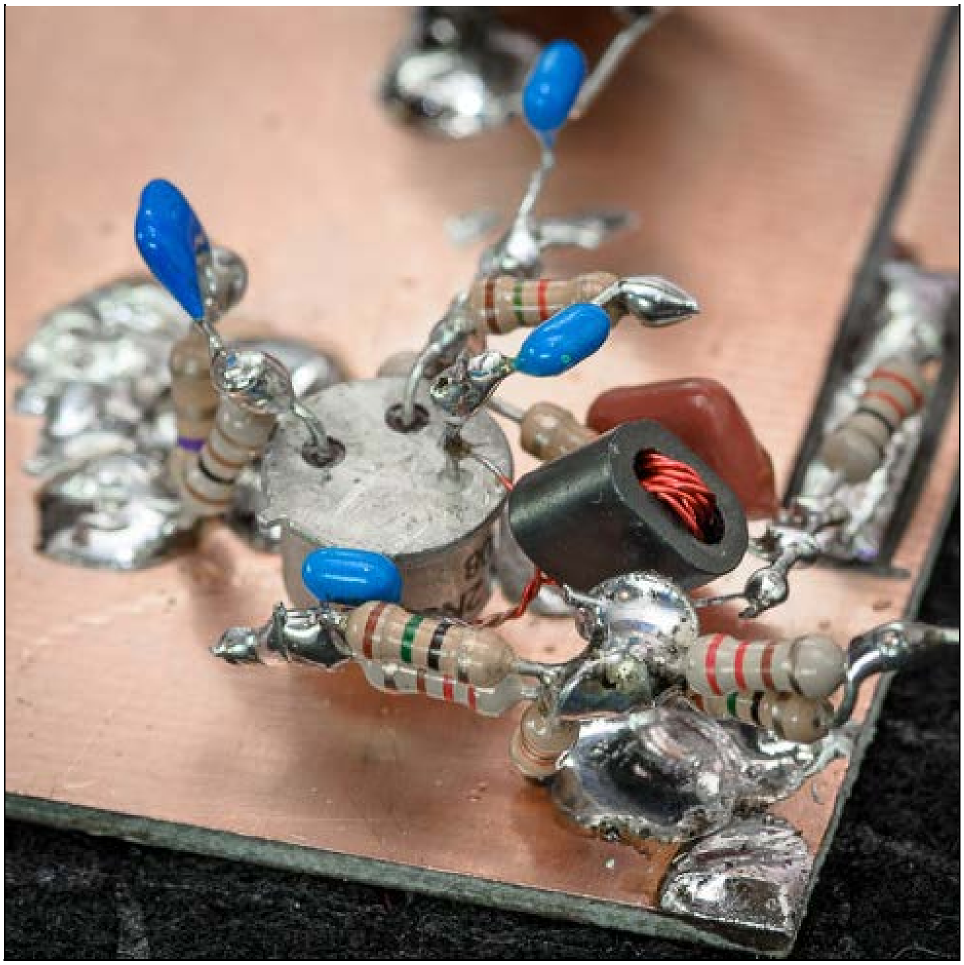
After peaking, I removed *all* the caps and measured them as shown in **red font**. I'll replicate these values with nearest standard-value fixed capacitors. The trimmer caps nearest to the ports proved more sensistive compared to the 2 middle trimmers capacitors during tuning.

This might be the coolest thing I've ever done on my test bench.

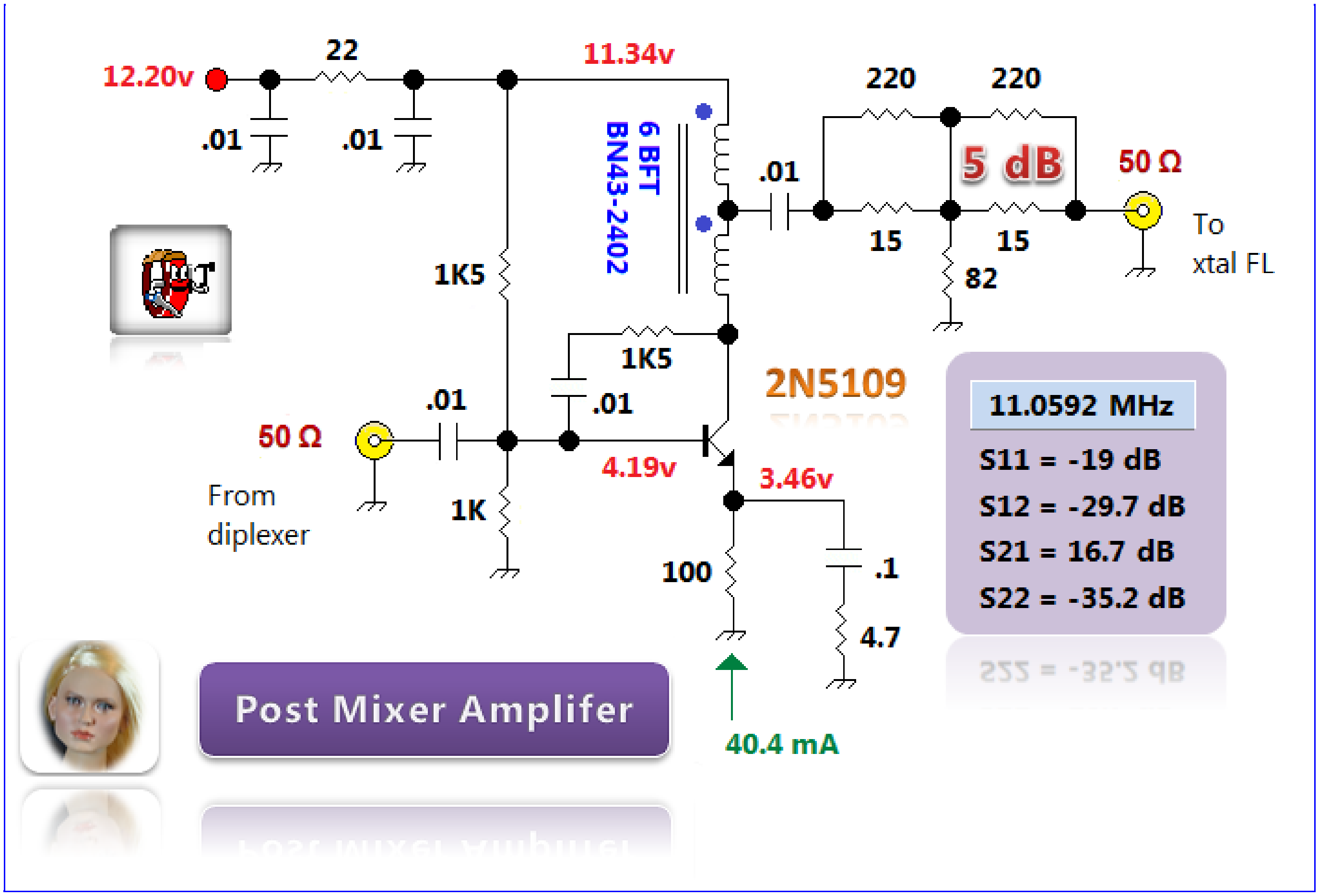


Above — My N = 3 filter breadboard. I left the capacitor leads long for re-use. I consume huge numbers of parts and recycle as possible to keep costs down.

Mixer and Post Mixer Amplifer Development



Above — Ugly breadboard shot with my ancient 105 mm lens for a change. It's a Beaverton Special feedback amp (FBA) biased for ~40 mA. For my post-mixer amp. If you need a stout RF amp, this one will do in a storm.

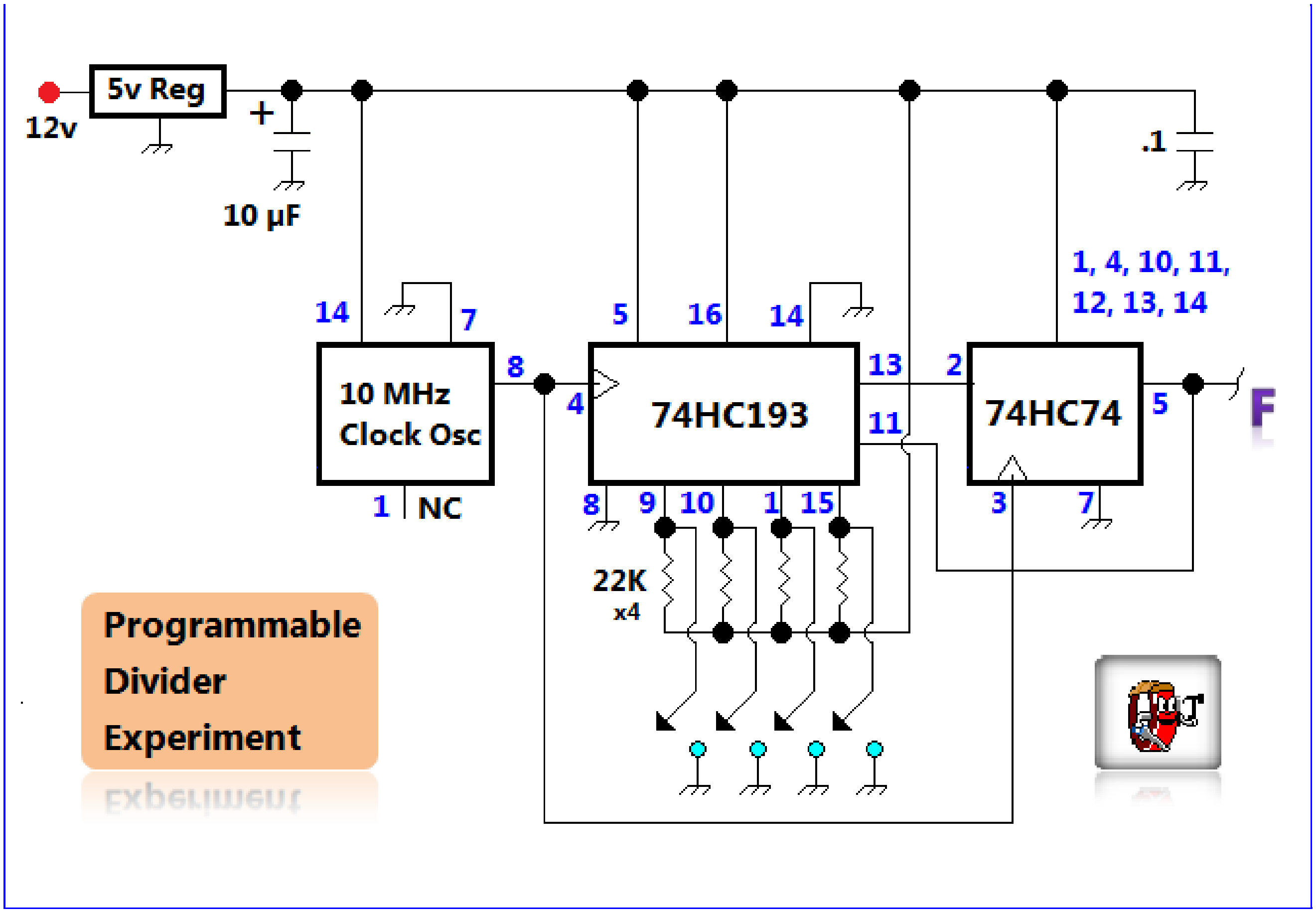


Above — The Beaverton Special feedback amp biased for ~40 mA with a 50 Ω output Z. The FBA with no 5 dB pad: S11 -18.5 dB; S12 -34.6 dB; S21 22.1 dB and S22 -31.8 dB. I love this FBA for its simplicity and well defined port impedance. My IF = 11.0592 MHz. Click for a discarded (but cool) photo.

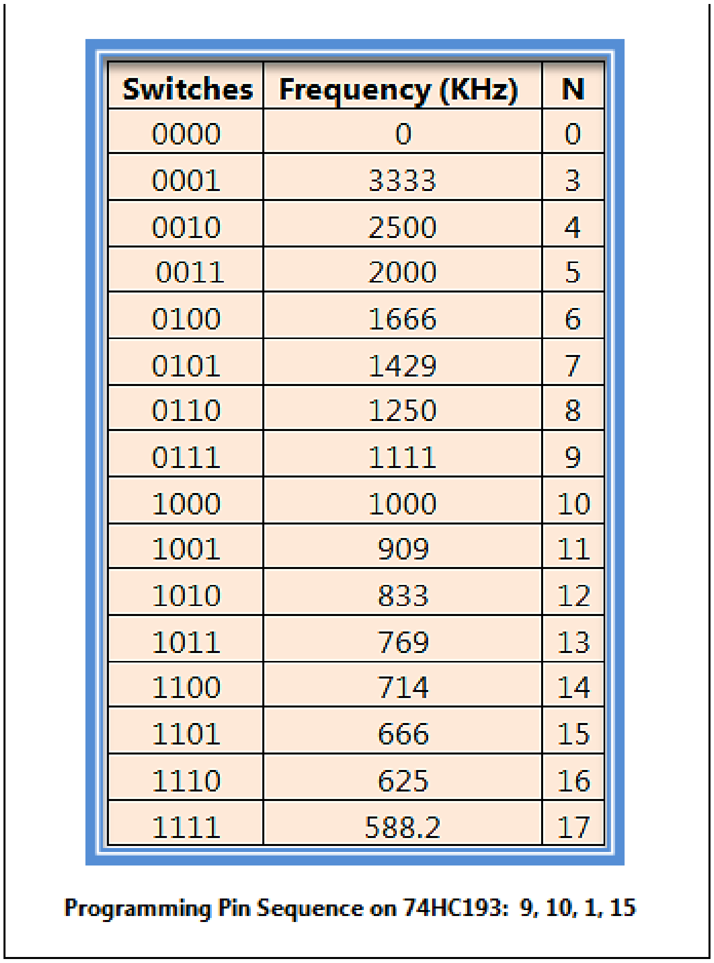
To drive the 200 Ω input Z of my xtal filter I later converted the output to 200 Ω by changing the collector transformer and pad.

Frequency Synthesizers and Logic

I've spent a great deal of time reading about & playing with some HC series logic I bought in 2011-2012. I even purchased a few 74AC74, a D flip-flop you can clock up to 160 MHz or so. Most of my reading concerned frequency synthesizers: PLLs, dividers, prescalers, etc. Since most modern devices run a clock at GHz frequencies, the evolution of frequency synthesis technology from around the mid 1970's to now makes for a fascinating read.



Above — Using a CMOS 10 MHz clock oscillator, I tested out a positive edge-triggered programmable frequency divider designed by Wes, W7ZOI in 2011. It works perfectly.



Switches	Frequency (KHz)	N
0000	0	0
0001	3333	3
0010	2500	4
0011	2000	5
0100	1666	6
0101	1429	7
0110	1250	8
0111	1111	9
1000	1000	10
1001	909	11
1010	833	12
1011	769	13
1100	714	14
1101	666	15
1110	625	16
1111	588.2	17

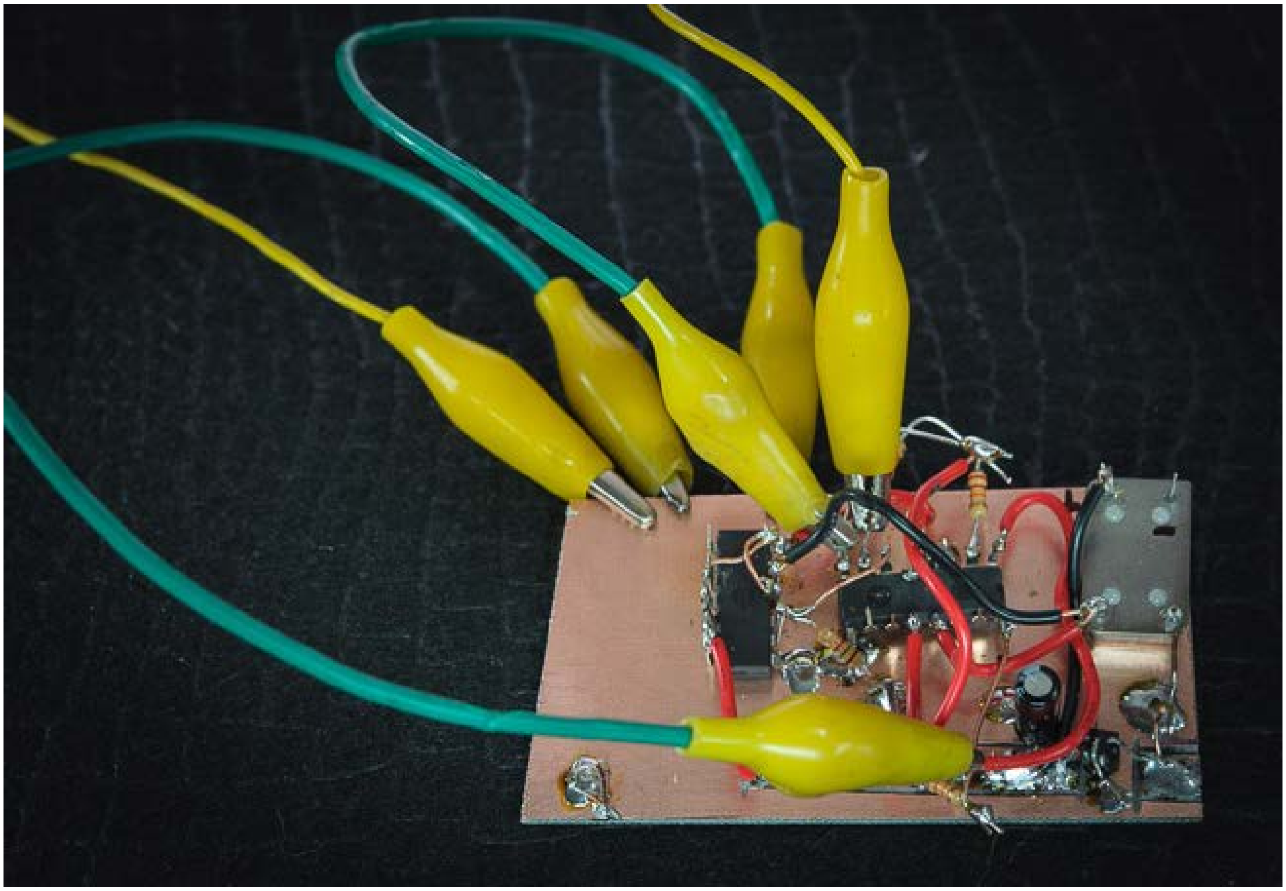
Programming Pin Sequence on 74HC193: 9, 10, 1, 15

Above — A test of all 16 combinations of the 4 programming switches with the 10,000 KHz input. I show the measured output frequency in KHz and the integer generated by dividing the output F into 10,000 KHz. You may leave pin 9 HIGH and just run it as a divide by 10 to 17 counter with 3 panel mounted programming switches — that's what I hope to do.

While working at Tektronixs, Wes designed the frequency synthesizer for the 492 Spectrum Analyzer; a portable SA that went to 21 GHz or so. His experiences informed the design of this simple, low-cost, edge-triggered divider.

Here, Wes applies the 74HC193 as a down counter where a number is loaded and then it count downs until zero is reached. The number gets boosted by 2 — 1 comes from the 74HC74 D flip-flop, while the other arises from the phasing of the signals. The 74HC74 makes the overall output coherent with its clock and avoids the flicker noise that otherwise might be generated in the long divider chain of the ripple counter.

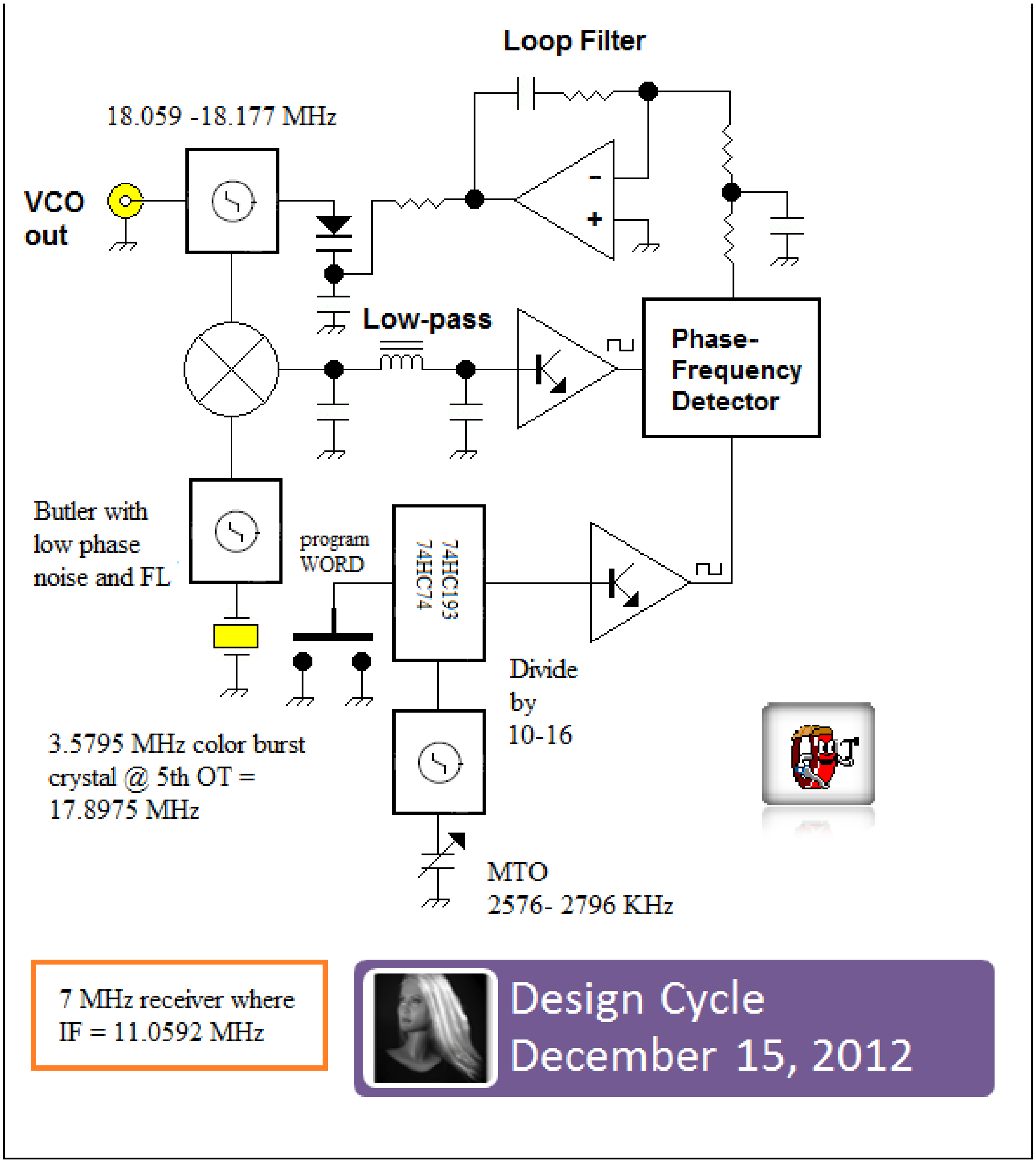
The standard binary to decimal conversion is altered in this circuit.



Above — My ugly breadboard. Instead of switches, I relied on bench jumpers to set a pin LOW.

Check out the standard binary switch combinations on this cool website: Math Is Fun. I also enjoyed their hexidecimal drum machine tool - Click. Actually, the whole web site looks great and I will recommend it to our friends with children and maybe even a few radio enthusiasts.

The Local Oscillator



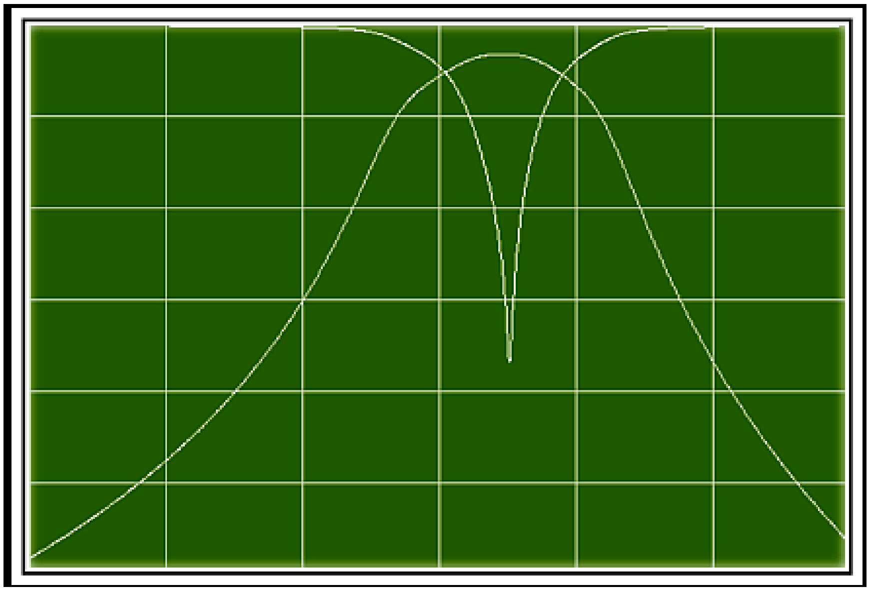
Above — An Almost Synthesizer based upon the design currently in use at W7ZOI.

I picked the MTO, xtal oscillator and VCO frequencies and will design the loop filter once I get the parts and a few small die-cast boxes on hand. The MTO division integer is low and quiet — plus this scheme enhances MTO frequency stability and spectral purity. A 4-bit ripple counter + flip flop perform the division where N = 10 - 16. I might go higher with N , but don't want to get too greedy. For CW, I'm generally in the bottom 20 KHz of the band anyway.

The MTO tunes about a 13-14 KHz range and the N gives 6 different tuning ranges (with some overlap) to span a total range of ~118 KHz. The 4-bit WORD is changed by 3 chassis-mount toggle switches and give a PLL reference of ~~175 to 280 KHz.

This almost synthesis seems a little crude in a time when people use microcontrollers to input a WORD into a chip with with the simple turn of a rotary controller, however this set-up fosters learning, huge fun, and when the oscillators plus loop filter are well designed; low phase noise and reciprocal mixing.

The Crystal Filter



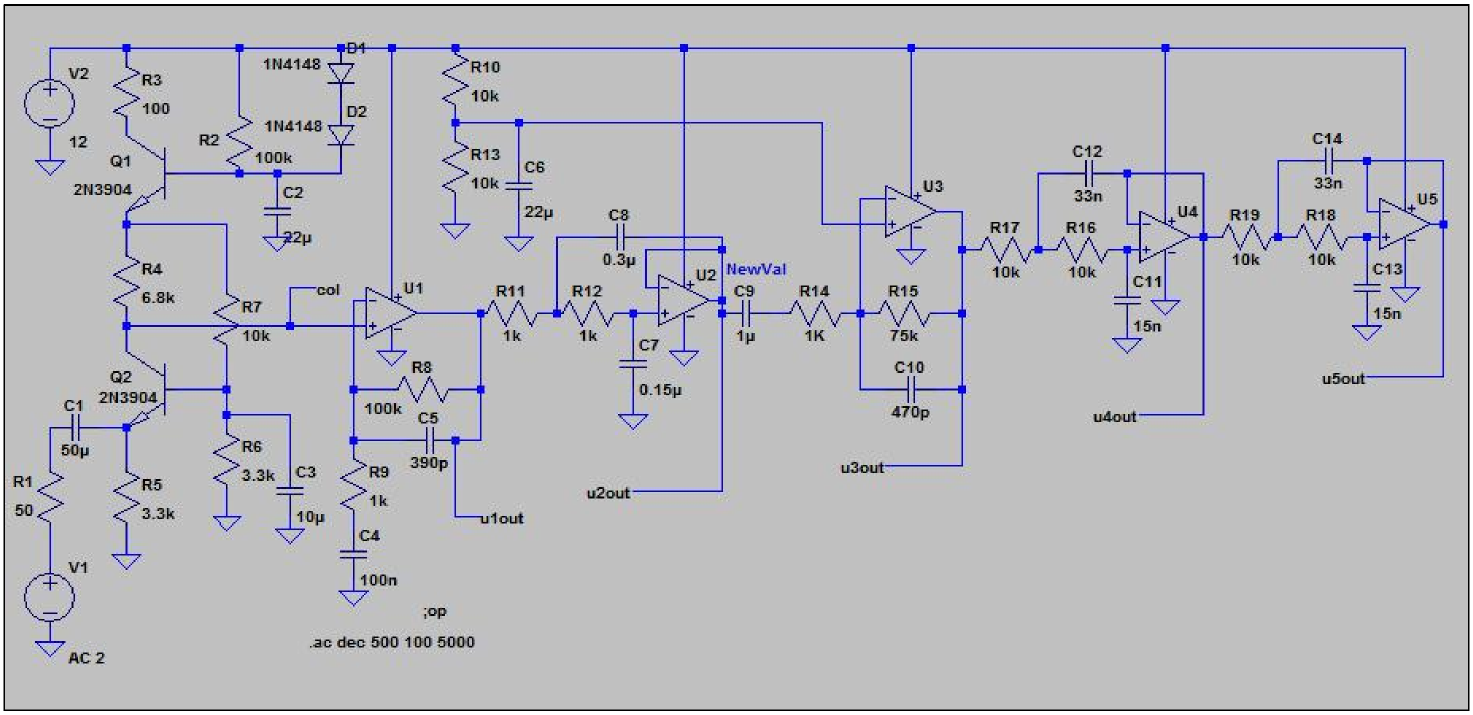
Above — A GPLA sweep of my 4 crystal filter where C parallel = 4.86 pF, Lm = .01 H, IF = 11.0572 MHz, BW = 1 KHz and R term = 200 ohms. This Gaussian to 6 dB filter design was chosen for speaker audio that sounds full + dynamic without the hollow ringing so often heard in homebrew receivers. I filter in my brain and first learned to copy CW across a room while listening to a speaker.

My code mentor Doug, could make coffee, walk around the room, talk to us and still 100% copy CW at 40 WPM. He taught us the ear to brain bandwidth is ~ 100 Hz when copying weak signal CW through a sideband crystal filter. To this day, I prefer listening through wider CW filters with brain band-pass filtering.

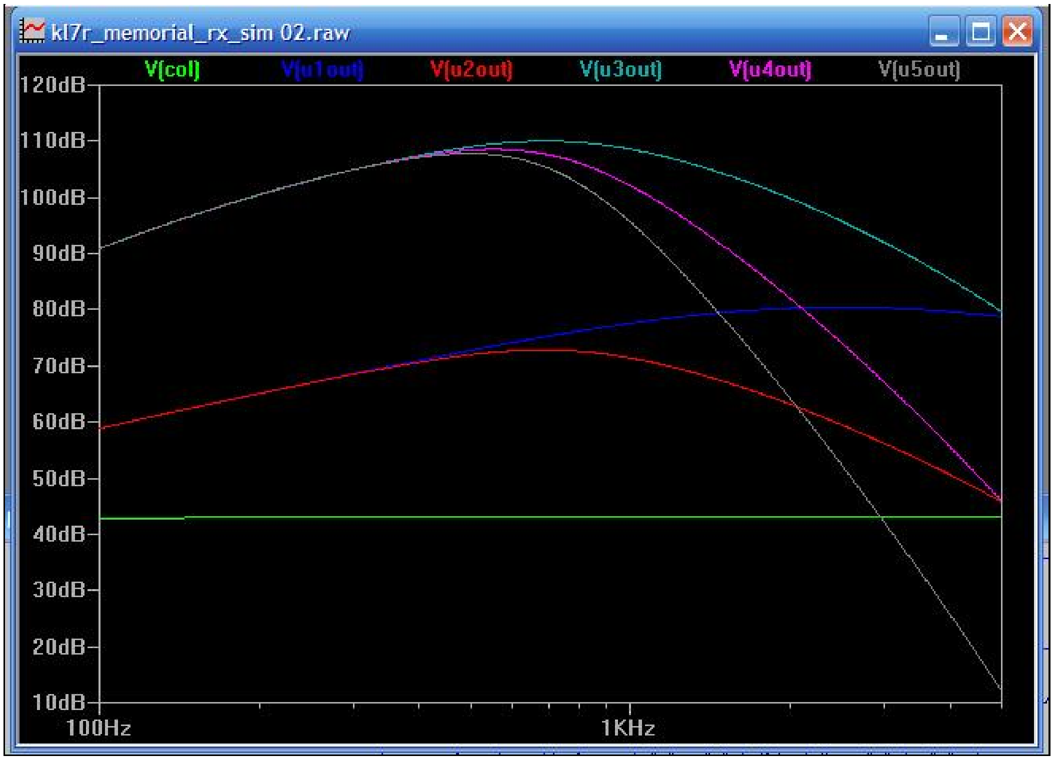
KL7R Memorial Supplemental Page

This web page is a supplement to the K7LR Memorial Receiver Experiments Web Page

Audio Preamplifier Focus

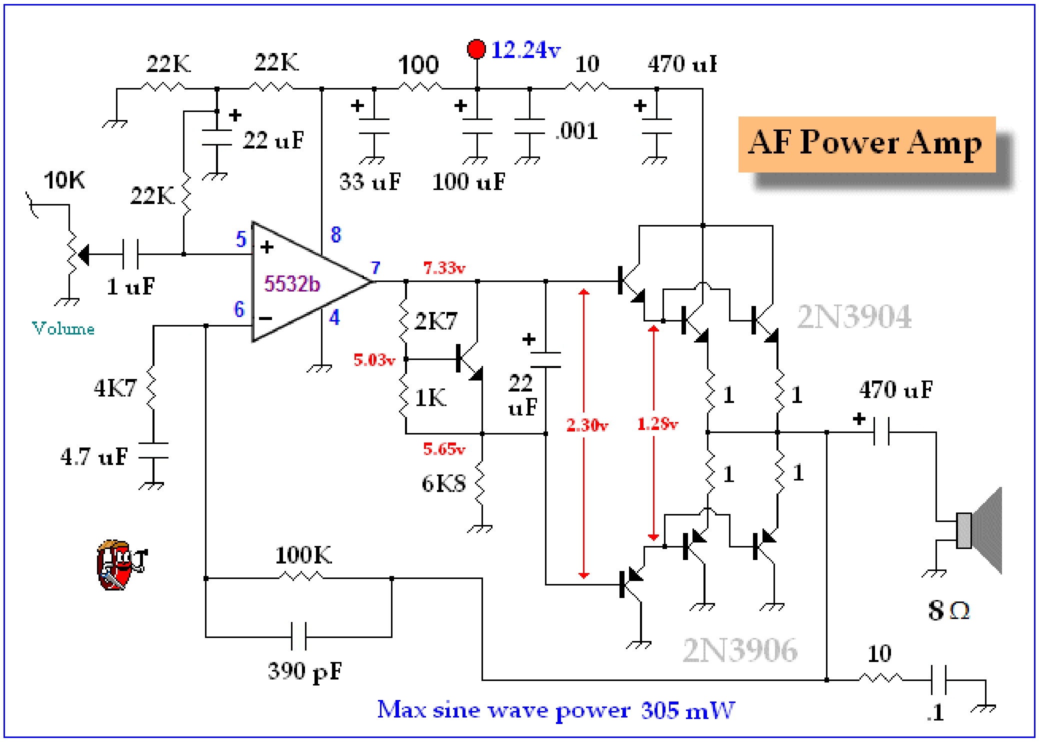


Shown above is a SPICE build of an earlier prototype of the audio preamp chain which was ultimately used in this receiver. This SPICE work was performed by Wes, W7ZOI. Note that at this point, I had 100K plus 390 pF feedback in U1 and a 75K plus 470 pF feedback resistors in the op-amp amplifiers (really massive gain). I spent 4-5 days trying different audio stages before settling on the circuits shown.



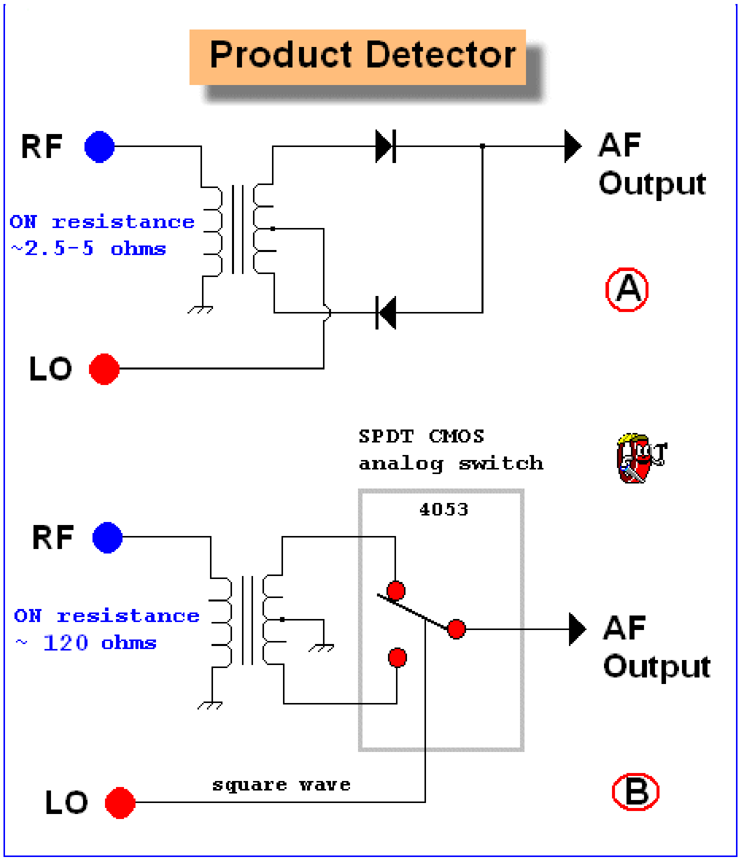
Shown above is the voltage versus frequency plot from a SPICE run of the schematic shown above. The gentle slope (decibels per octave) of the low pass filters can be seen. Certainly Chebychev filters with 1-2 dB of ripple would have provided steeper filtering, but the filters would have odd resistor values and be less easily modified or reproduced by popcorn builders. The V(u5out) shows a reasonable low pass response for the AF preamp.

Audio Power Amp

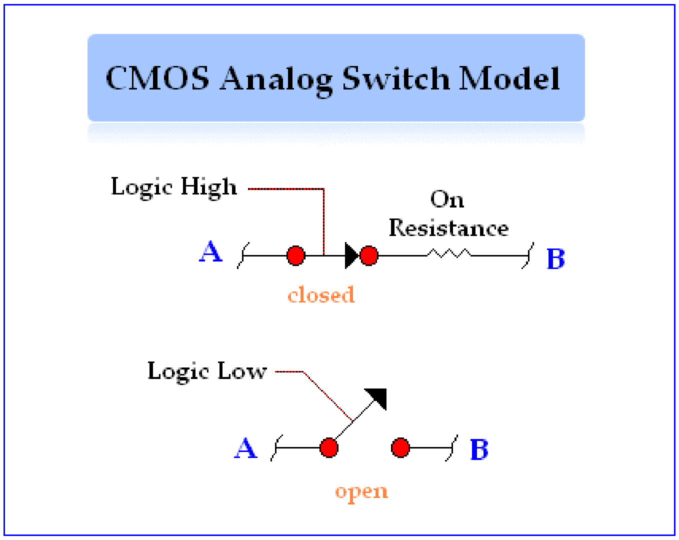


This schematic shows a stand-alone version of the audio power amp. If only 1/2 of the 5532 is used, connect pins 3 and 2 of the unused op-amp to the point where all the 22K resistors connect (VCC/2 bias point).

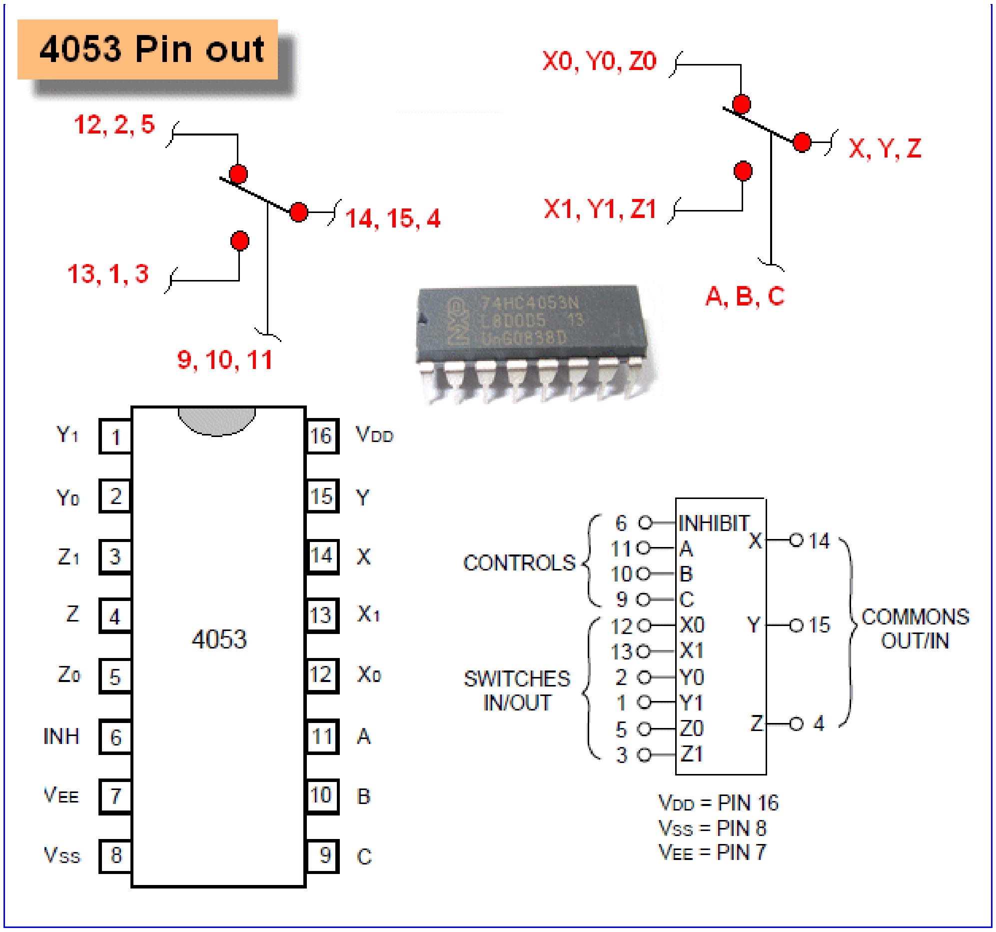
Product Detector Notes



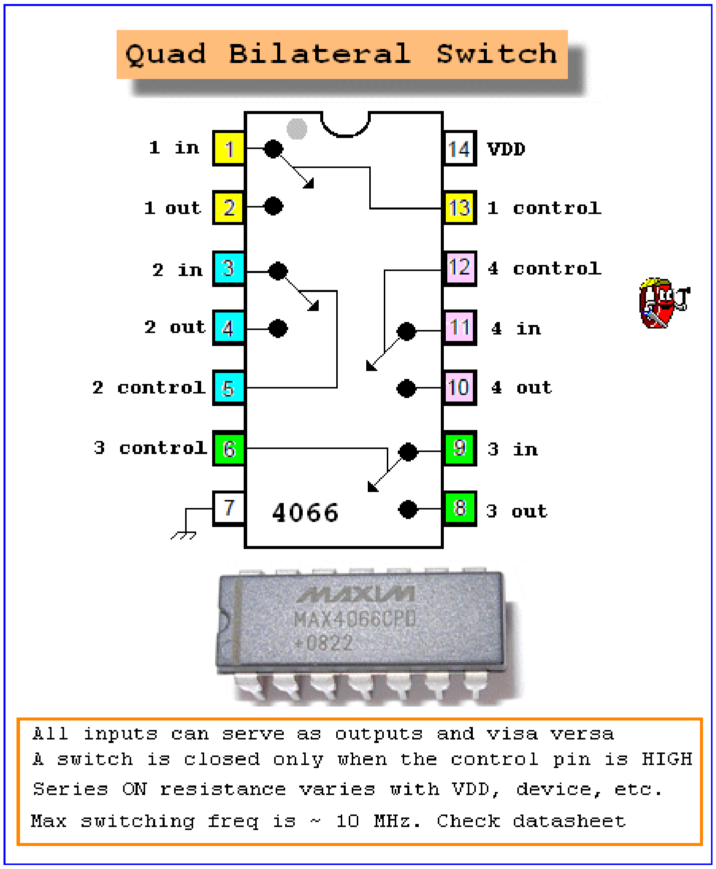
Shown above is the analogy between an SPDT CMOS switch and a single-ended, diode product detector. In both cases (A and B), the LO causes 1 diode (switch) to conduct, while the other is OFF. Then, on the other half cycle, the other diode (switch) conducts. If you wish to actually using the product detector/mixer depicted as "A" go with Figure 5.19B in EMRFD which has the LO applied to the transformer primary. As shown, "A" has poor LO to RF isolation, but it helps model the CMOS SPDT analog switch function. Function and performance are completely different topics... The on-resistance of the 4053 is much higher and thus performance is not identical.



CMOS analog switch model. A or B are bidirectional and can be used interchangeably for the input or output. Ideally the on-resistance should be low to reduce propagation delay of digital signals plus resistance and perhaps distortion of analog signals. They can be used to switch digital bus data or analog signals. CMOS switches can replace mechanical reed switches in some low power circuits.

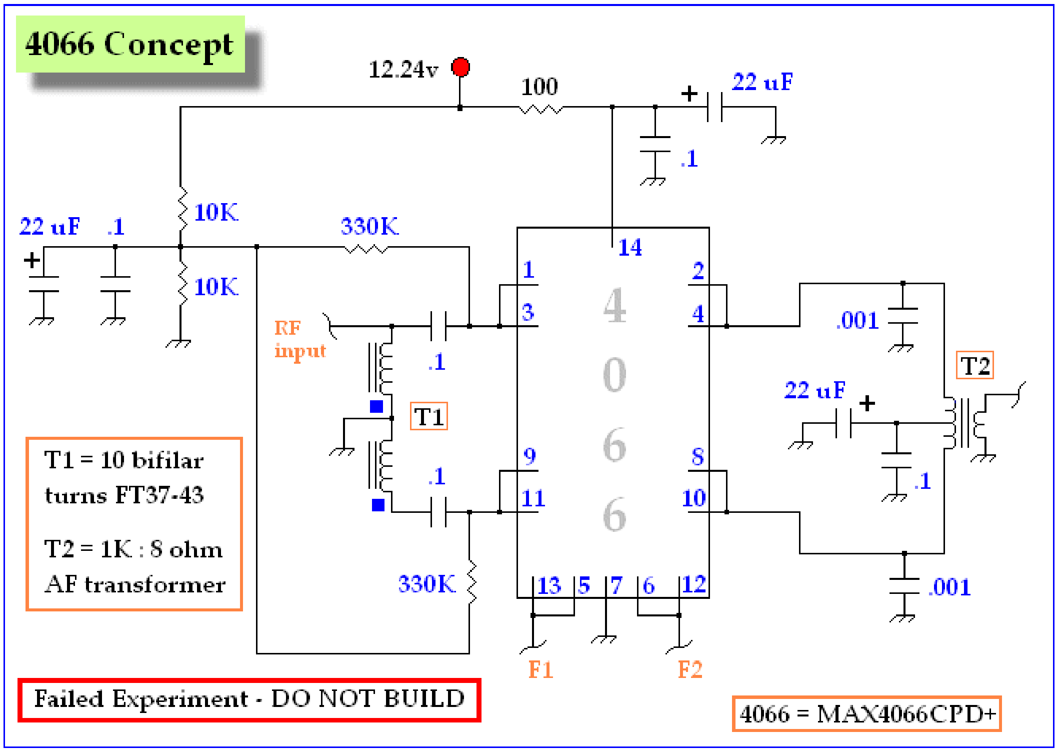


The CD4053BC is a triple 2-channel multiplexer having three separate digital control inputs; A, B, and C, and an inhibit input. Each control input selects one of a pair of channels which are connected in a single-pole double-throw configuration. A high on resistance makes the 4053 a compromise part for RF mixing. It might be perfectly okay for your design.

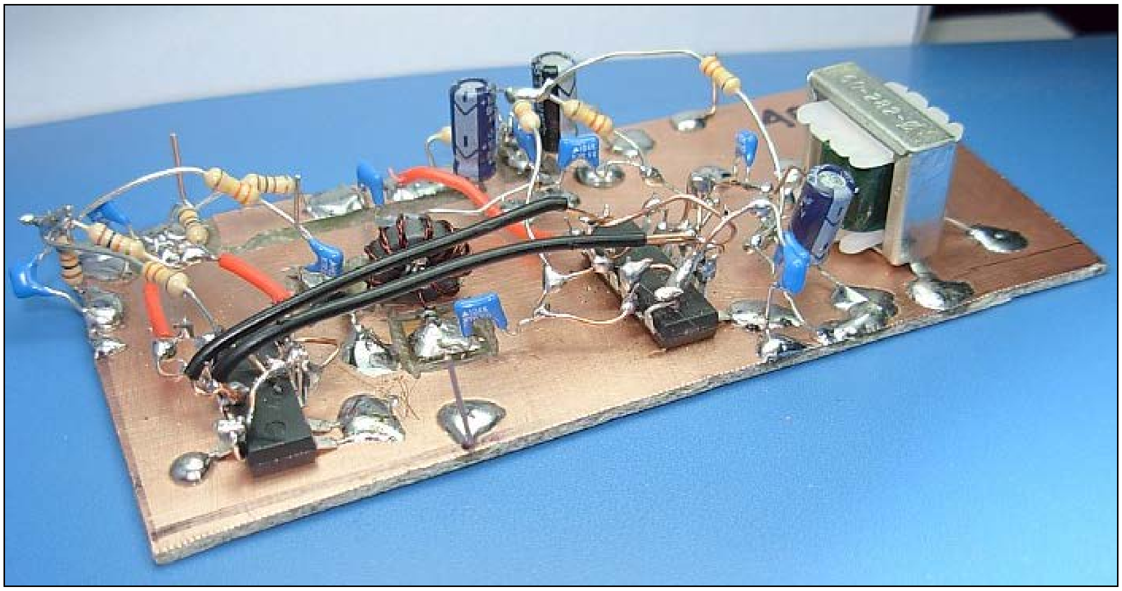


All inputs can serve as outputs and visa versa
A switch is closed only when the control pin is HIGH
Series ON resistance varies with VDD, device, etc.
Max switching freq is ~ 10 MHz. Check datasheet

The 4066 is more easily understood when drawn in a semi-schematic. The on-resistance of this MAXIM version is better than the usual HC4066. About 45 ohms with a VCC of 12 or so.



A 4066 experiment that was a total failure. It is both humbling and useful to show your failures and not just the good fruit. This circuit had hum, noise and low output. The input transformer was a bifilar type, which was really cool.



The breadboard of the 4066 Concept schematic shown above.

The DIP IC switches used as product detectors in the K7LR memorial receiver experiments.

My work bench during some final experiments for this project.

Amateur and Short Wave Radio Electronics Experimenter's Web Site

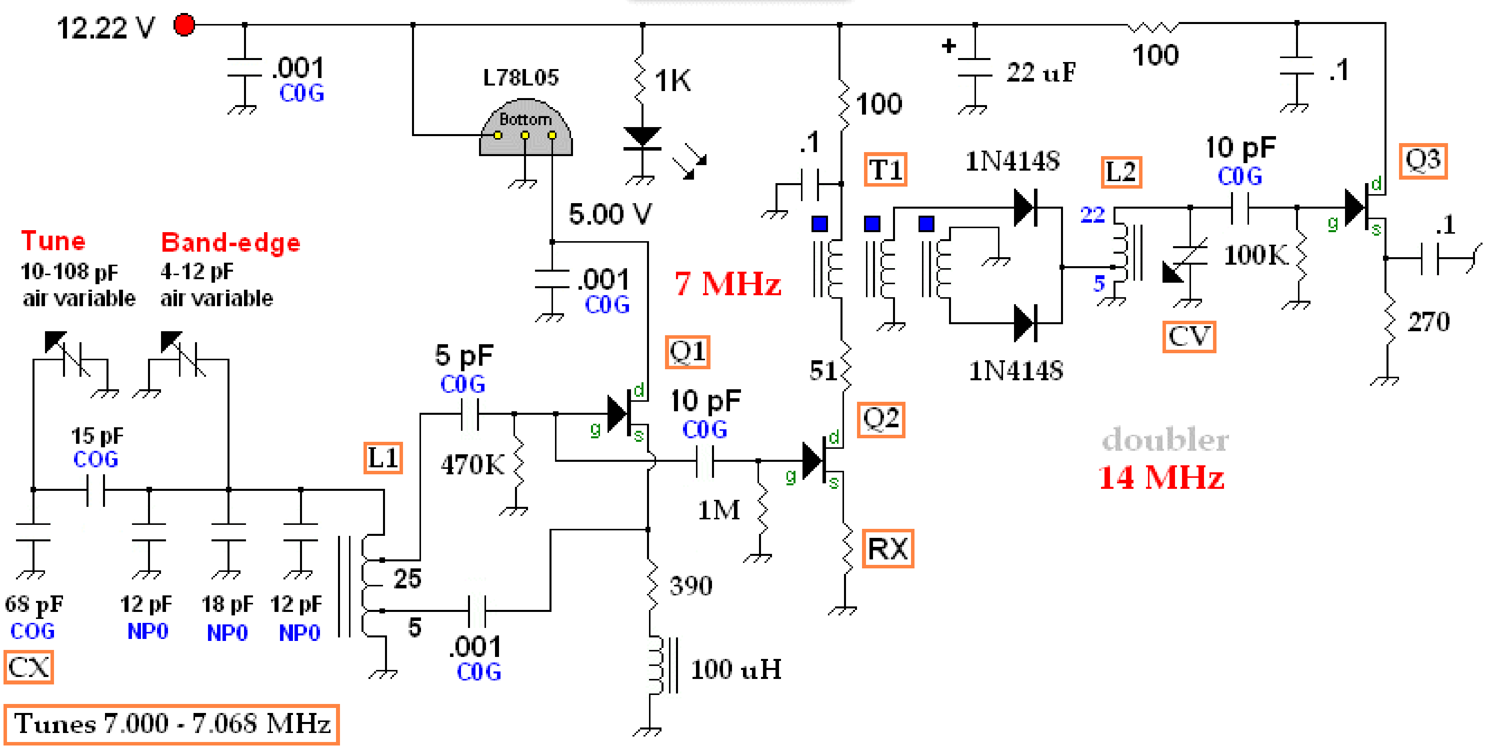
---

K7LR MEMORIAL RECEIVER VFOs

This web page is a supplement to the K7LR Memorial Receiver Experiments Web Page

14 MHz VFO Circuit

Figure 1



L1 = 6.1 uH = 36t on a T68-6 toroid. Tap at 25 and 5 turns. (used 26 AWG)
Q1 = Q2 = Q3 = J310
100 uH RFC that I used was wound as follows: 38t on a FT50-61 ferrite toroid
T1 = 10 trifilar turns on a FT37-43   L2 = 2 uH = 22t on a T50-6; tap @5t
CV = 6.0-50 pF trimmer capacitor
RX = 470 ohms

Figure 1 is the 14 MHz VFO used to switch (clock) the CMOS switch product detector. The tuning range can be easily changed by adjusting capacitor CX. As shown, the the tuning capacitor has a frequency range of about 68 Hz. If you decrease CX to 47 pF, the tuning range increases to around 99 Hz. Increasing CX to 100 pF gave about a 40 Hz tuning range. When you use a larger tuning range, fine tuning can become difficult. A geared reduction dial is one possible solution. For simplicity, I prefer to just keep the tuning range low as reasonably possible and use a big knob on the main tuning variable capacitor. Even the 68 Hertz tuning range shown can prove difficult for fine tuning. As experimenters, we are continually problem solving. Choose the tuning range and/or methods which suit your personal needs.

The base 7 MHz oscillator is doubled since a D flip flop is used to clock the CMOS switch product detector. This oscillator has low harmonic content and this is important for suppressing the 7 MHz fundamental frequency and its harmonics in the frequency doubler. Some astute builders may even match the two diodes used in the doubler. Diode matching is discussed on this page. CV tunes very sharply and the output at Q3 is a clean sine wave. This in turn provides a well balanced square wave at each output of the D flip-flop. The Q3 output voltage was measured at -0.46 dBm (0.6 volts peak to peak into 50 ohms).

Choosing Capacitor Values

There are a number of ways to determine the capacitor values required to tune your VFO L-C tank circuit. It is pretty much essential to have a frequency counter and nice perhaps, to own a capacitance meter. Main tuning capacitors are typically harvested from an old radio or from the junk box. Its tuning range could be anywhere from 30 to 300 pF, or more. To limit the tuning range of this variable capacitor, normally you parallel a fixed-value capacitor and then series connect it to the top of L1 with a small, fixed-value capacitor. I performed this task entirely by trial an error using a frequency counter connected to the collector of Q2. Some fixed value NP0 or C0G capacitors plus a small trimmer capacitor are also required for tuning and to set the lower band edge respectively. Finding the right combination of capacitor values is painstaking, but with practice, gets easier. Once you have the basic capacitor values sorted out and your tuning range set, frequency stability experiments are then performed. This is known as VFO temperature compensation. Some times temperature compensation can be achieved by finding the right combination of NP0 and C0G capacitors. Additionally some negative or positive temperature coefficient capacitors may have to be soldered in and tested. Your final capacitor leads should be short as practically possible to reduce stray lead capacitance and for mechanical rigidity. Temperature compensation is discussed on this web page.

A 7 MHz VFO Circuit for Diode Ring Product Detectors

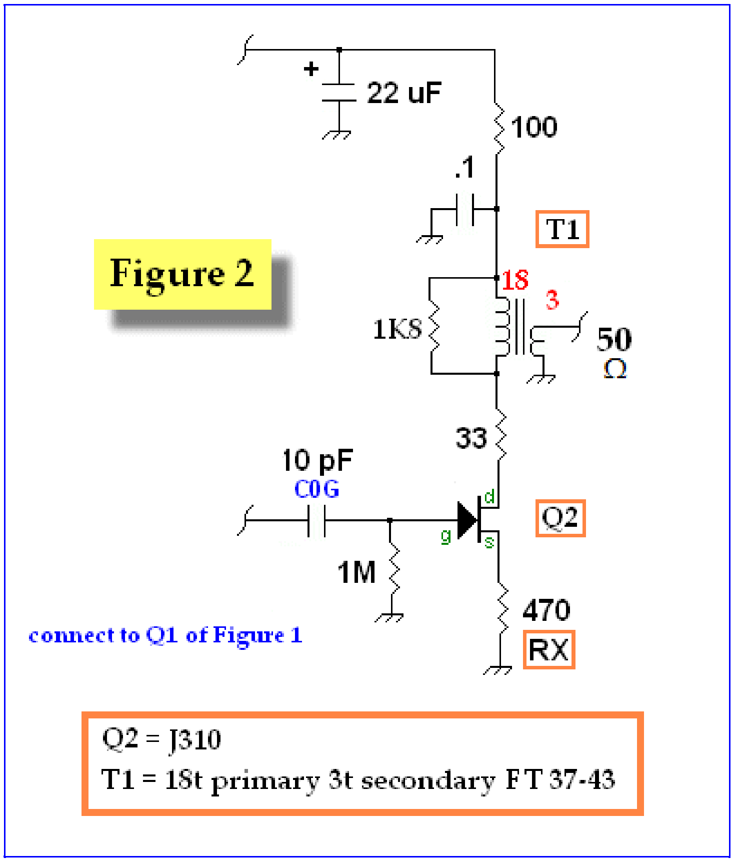
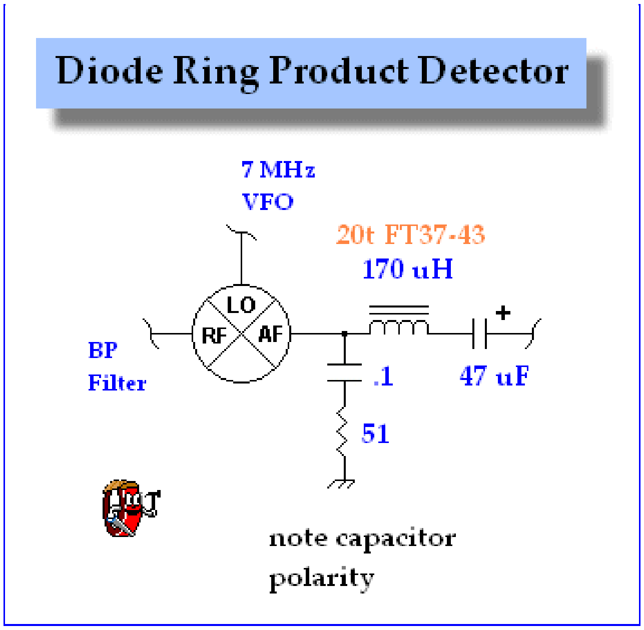


Figure 2

Q2 = J310
T1 = 18t primary 3t secondary FT 37-43

If you want to build a simpler version of the K7LR memorial receiver, a diode ring mixer may be substituted for the digital switch. The VFO requires modification as it will be run at 7 MHz. A different buffer is used and the Q2 to Q3 frequency doubler circuit is excluded. The diode ring product detector version is very nice. You could use a Mini-circuits TUF-1 , SBL-1 or alternate, or perhaps homebrew your own.

In Figure 2 is a VFO buffer configured to drive a 50 ohms input impedance, 7 dBm level, diode ring mixer. Change Rx to change the output voltage. With RX at 470 ohms, the output was somewhere around 5 dBm. This is useable for many situations. If you want exactly 7 dBm, the AC peak to peak voltage with a **51** ohm load resistor connected to the 3 turn link should be 1.43 volts. Adjust RX to achieve this voltage in your oscilloscope.



Note that if you build a diode ring product detector receiver version using the simple W7EL low pass termination network, the polarity of the 47 uF electrolytic capacitor will need to be changed as shown.

Key goals of this website include providing ideas, basic support and encouragement. I am delighted when builders make their own stuff and not copy my circuits. This is why we homebrew; to create, explore and share. Mike, K7LR did all of these very well.

QRP SWL HOMEBUILDER

Amateur and Short Wave Radio Electronics Experimenter's Web Site

---

Photography Links

Nikkor 50mm lenses

HENRI CARTIER-BRESSON - shot a lot with 50mm - video to watch over and over

The genius of photography -- BBC4 series about the history of photography

http://www.freestylephoto.biz
