Summary of Program on Circuit Analyzer Software

Presented by Jeff Harman 7-14-2020

After I gave this program I was asked to put it on the RST website.

Opening Blather

Most of you know me as the old pack rat that crawls out from under the prickly pear to pound brass on field day. However I am a retired analog circuits and control systems engineer. I started my career designing hf receiver circuits at Collins Radio and finished as an adjunct teaching sophomore EE at Pima Community College. I was a practicing engineer for 56 years. As such I was exposed to the growth of circuit analysis programs. After a brief history, as I remember it, TINA (Texas Instruments Network Analysis) and PSpice will be demonstrated.

History, as I remember it; Skip this if you aren't interested!

The first circuit analysis program I was aware of was ECAP, Electronic Circuit Analysis Program, which was an IBM program in the mid- 1960s. I didn't get to play with it, but my understanding is that it did a good job on transient analysis of linear systems; and not much else.

In 1968 I joined Scientific Data Systems. They had their own program, CIRC (I don't know source of the acronym.) and it had the same capability and limitations as ECAP. The output device was a model 28 TTY, and data was presented either as a series of dots or crosses, users choice. Typical simulations were 10 feet long!

In 1980 I was a consultant at Cambrian Consultants; I used a Hewlett Packard program that was a great improvement, it would do transients, and amplitude, phase, and group delay frequency sweeps; a tremendous aid when designing analog signal conditioning. Graphics were printable on 8 ½ X 11 sheets. However only linear analysis was performed, and you had to model transistors yourself using small signal models.

At about this time a graduate student at UC Berkeley developed a program called SPICE; I don't know what this stands for either. Spice did linear and non-linear analysis of analog circuits. Accurate non-linear models were used. With the possible exception of Cadence and Sabre the programs that followed were at least partially Spice based.

Cadence developed an excellent, but highly expensive, program. The initial Cadence program required 2 mainframe IBM computers and carried a multi-million dollar price tag. I believe that Intel used, and is using, Cadence for their microprocessor design. Sabre is known as the "poor man's Cadence" having similar capability and a slightly lower price tag. NCR uses Sabre. I have spent a lot of time reviewing Sabre schematics and simulations, but it's too pricey for mere consultants.

PSpice managed to fit a Spice based program into an IBM PC, hence the name. I believe they were the first. Others soon followed.

OrCAD was a circuit layout program, and PCad, I think; was a schematic capture program. PSpice bought PCad, then OrCAD bought PSpice, and then Cadence bought OrCAD. They have merged the program so that the designer can us the OrCAD software to do analog and digital mil-spec schematics, analog and digital analysis, and layout the circuit board.

Other programs are Microcap, my favorite, ICap, TINA, and many more.

Demo 1, TINA

Circuit analysis programs ANALYZE circuits; they do not synthesize them. To get started you must have a circuit to analyze. I chose to compare a 9 pole Butterworth filter (maximally flat amplitude response) with a 9 pole Tchebycheff filter flat with a 0.1 dB amplitude ripple. (I have seen Tchebycheff spelled 3 ways in "authoritative "texts.) I used tables in "Handbook of Filter Synthesis" by Anatol Zverev to obtain normalized values for both. For this talk a pole is the number of equivalent inductors and capacitors in a circuit. When capacitors or inductors are merely in parallel they count as 1 pole.

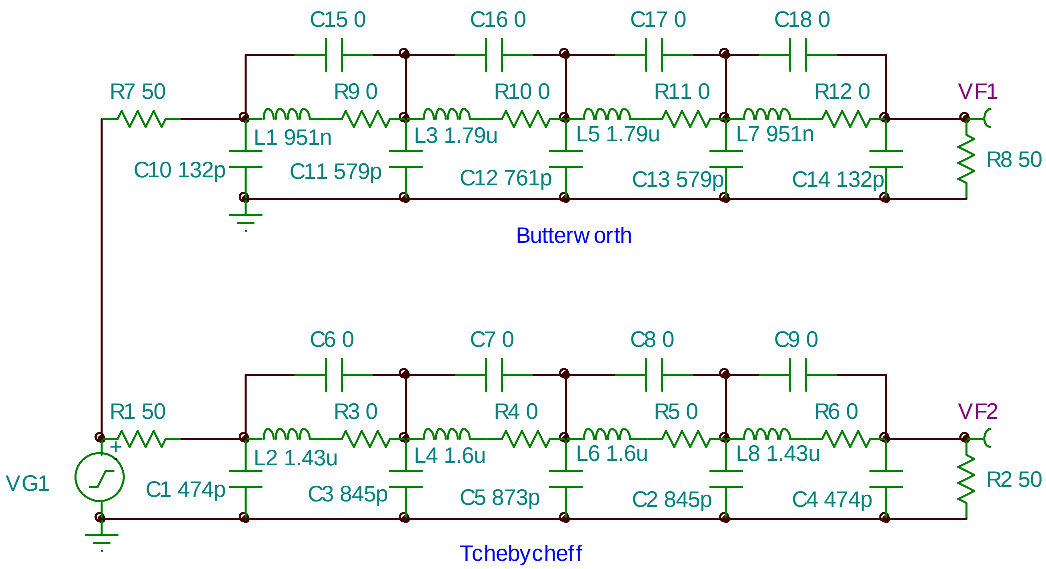
Zverev died in 1983; he wrote his handbook in 1967. It goes in and out of print. I obtained my latest copy in 2008, it was in print then. A Wiley publication. Most other filter texts and handbooks that I've seen reference Zverev.

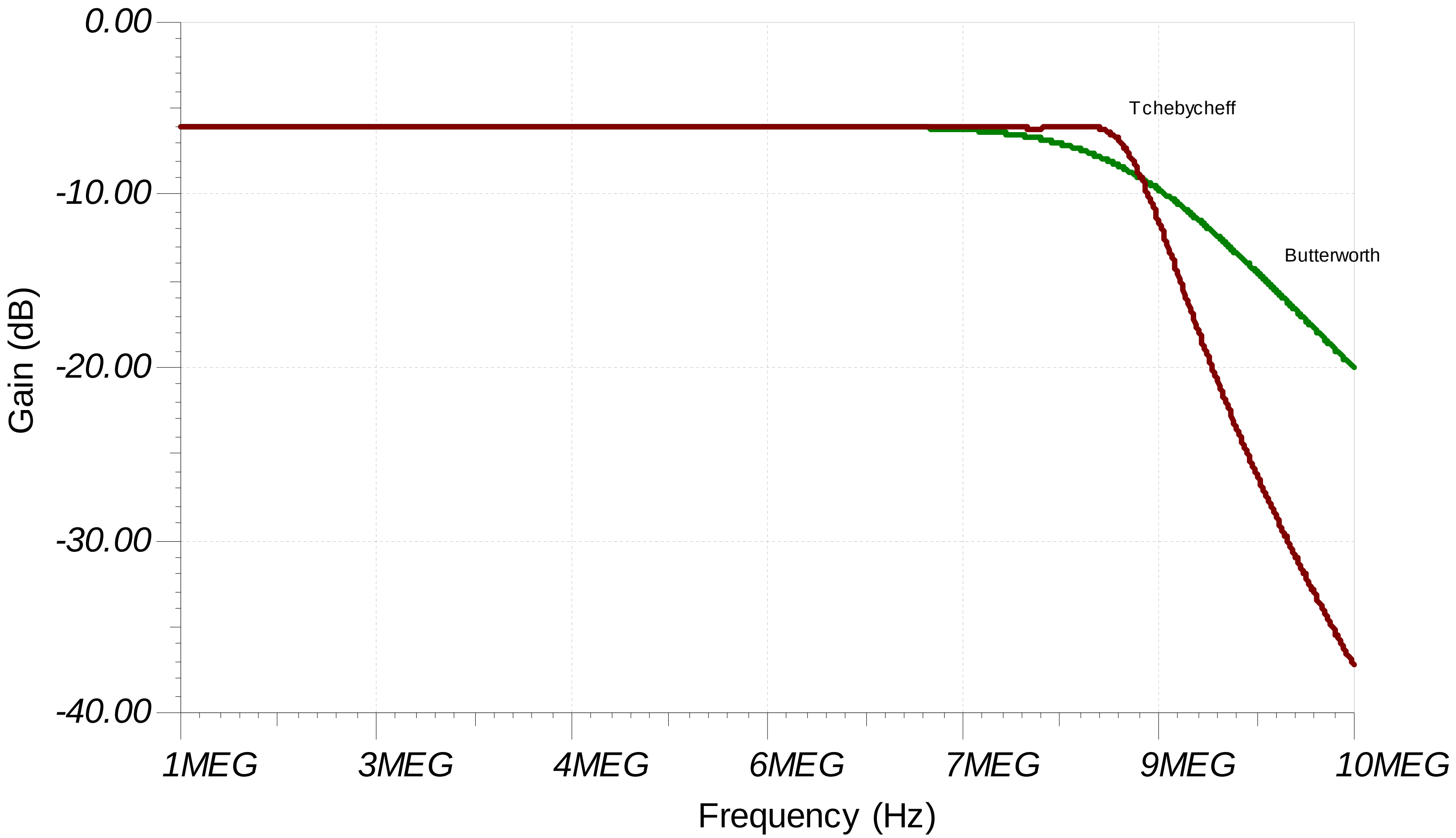
The tables in Zverev are normalized to 1 ohm and 1 radian per second; to de-normalize the values you must impedance scale by multiplying inductor values by the desired impedance and dividing capacitor values by the desired impedance. Then you must frequency by dividing the impedance normalized values by the desired radian frequency. Since this must be done 9 times, I set up spread sheets to do this.

Butterworth scaling spreadsheet

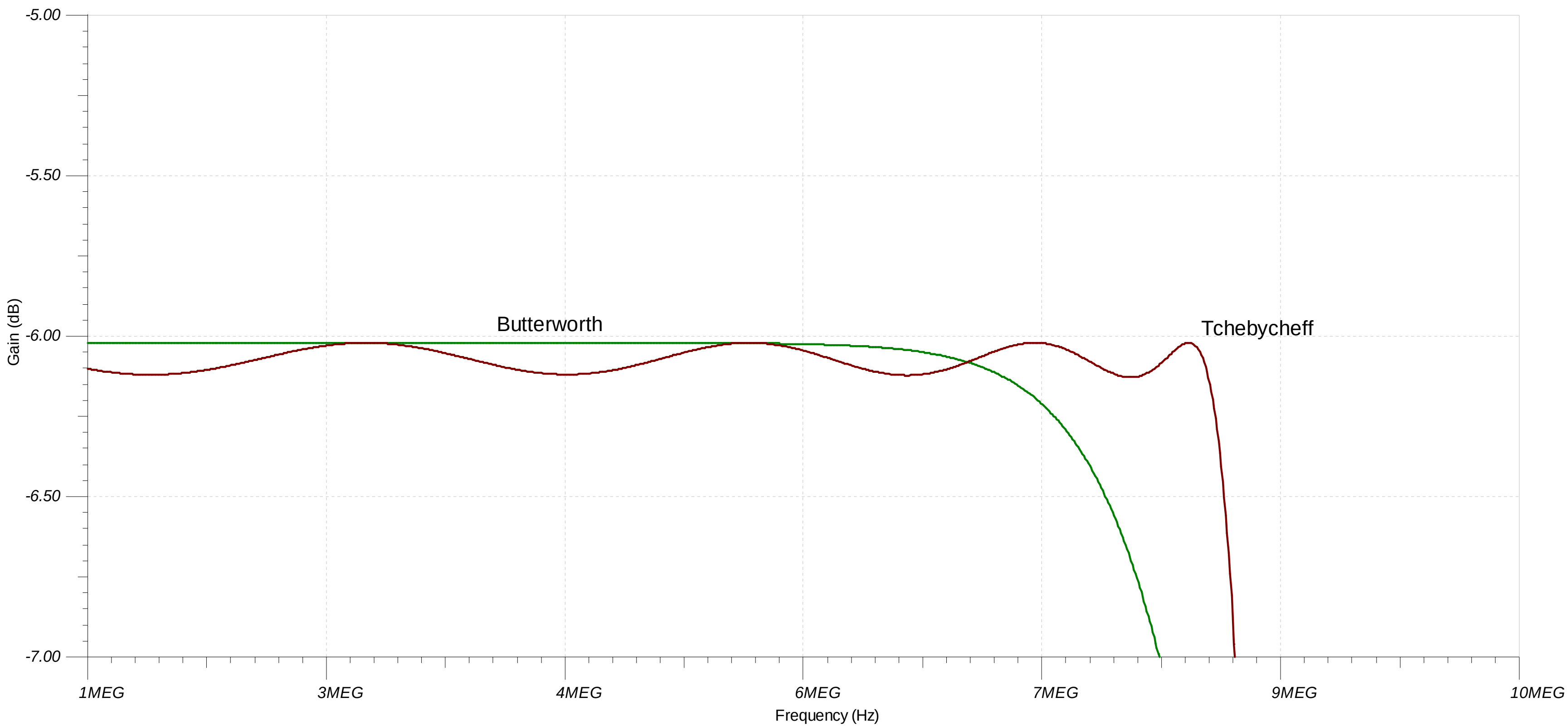
F Scale	R	C1	L2	C3	L4	C5	L6	C7	L8	C9
Zverev	1	0.3473	1	1.5231	1.8794	2	1	1.5231	1	0.3473
Impedance	50	0.006946	50	0.030462	93.97	0.04	50	0.030462	50	0.006946
frequency	8366600	1.3213E-10	9.5113E-07	5.79468E-10	1.78756E-06	7.60906E-10	9.51133E-07	5.79468E-10	9.51133E-07	1.3213E-10
frequency scale factor	1.90227E-08									

Finally a circuit

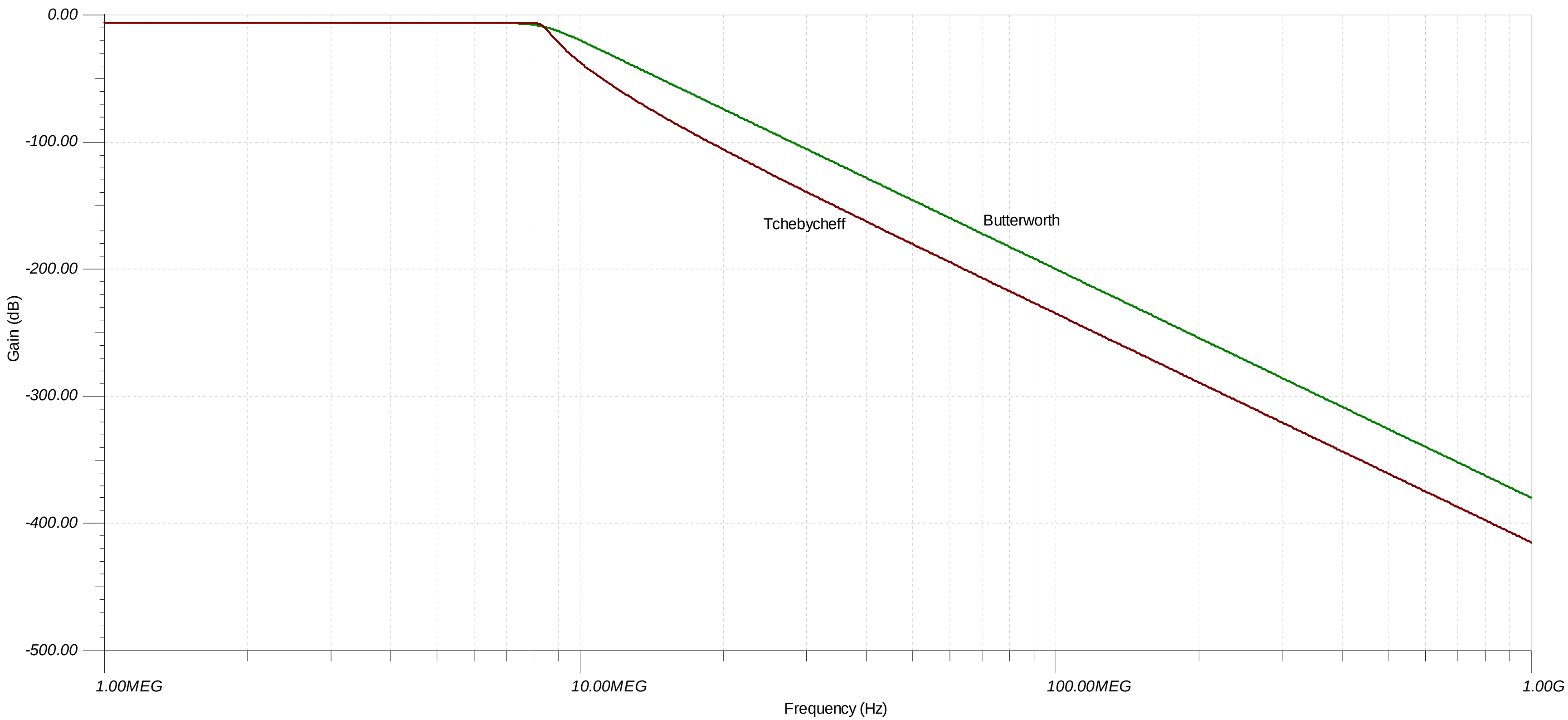

C15 0
C16 0
C17 0
C18 0
R7 50
R9 0
R10 0
R11 0
R12 0
VF1
L1 951n
L3 1.79u
L5 1.79u
L7 951n
R8 50
C10 132p
C11 579p
C12 761p
C13 579p
C14 132p
Butterworth
C6 0
C7 0
C8 0
C9 0
R1 50
R3 0
R4 0
R5 0
R6 0
VF2
L2 1.43u
L4 1.6u
L6 1.6u
L8 1.43u
VG1
+
R2 50
C1 474p
C3 845p
C5 873p
C2 845p
C4 474p
Tchebycheff



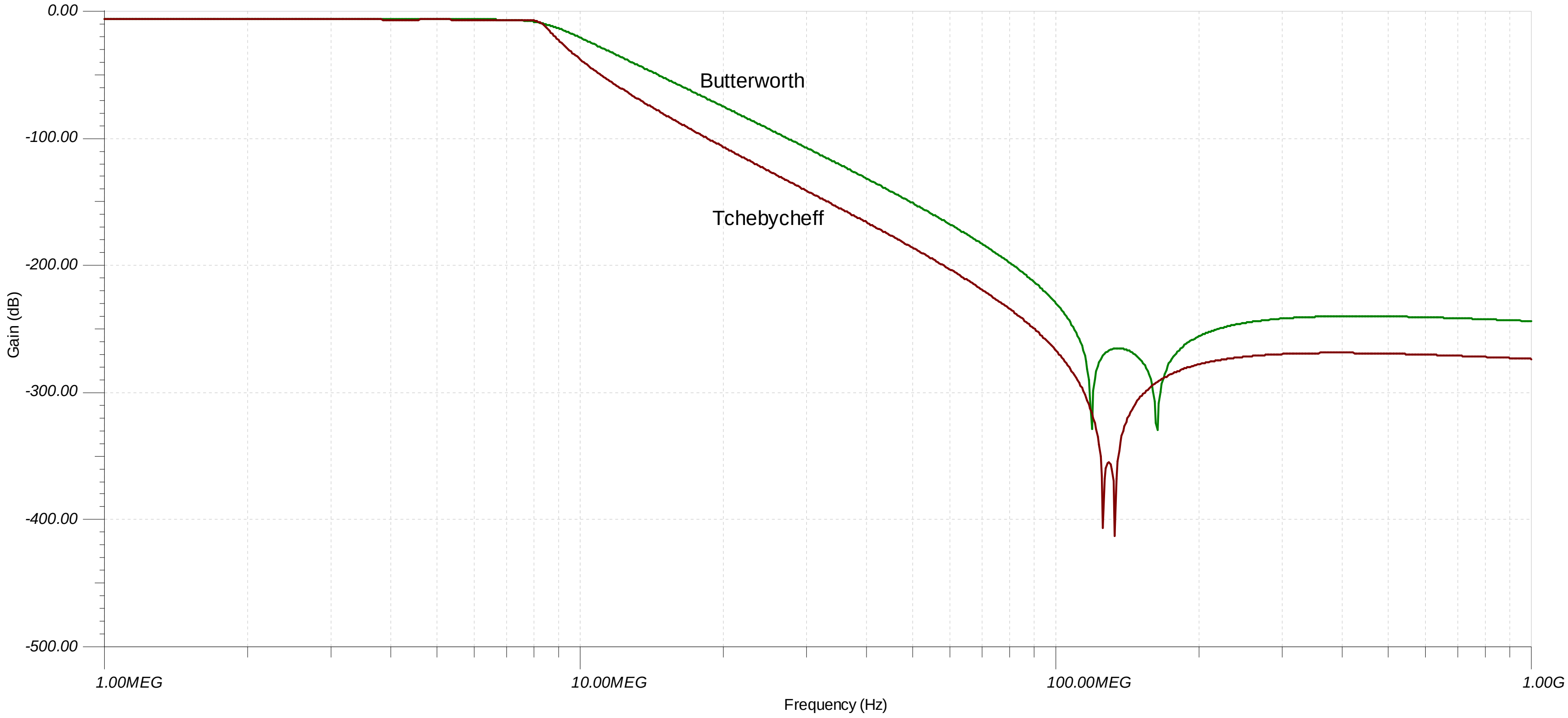
The Tchebycheff wins for sharpness of cutoff and flatness close to the 3 dB frequency. Note that since both filters were designed to have the same cutoff, the responses cross at the 3 dB point. The 6dB insertion loss is simply the voltage divider action of a 50 ohm source and a 50 ohm load. Now let’s look at the ripple. The input and output terminals may be interchanged without affecting performance.


-5.00
-5.50
-6.00
-6.50
-7.00
Gain (dB)
Butterworth
Tchebycheff
1MEG
3MEG
4MEG
6MEG
7MEG
9MEG
10MEG
Frequency (Hz)

Let’s look at a wider bandwidth, and with a logarithmic time base.

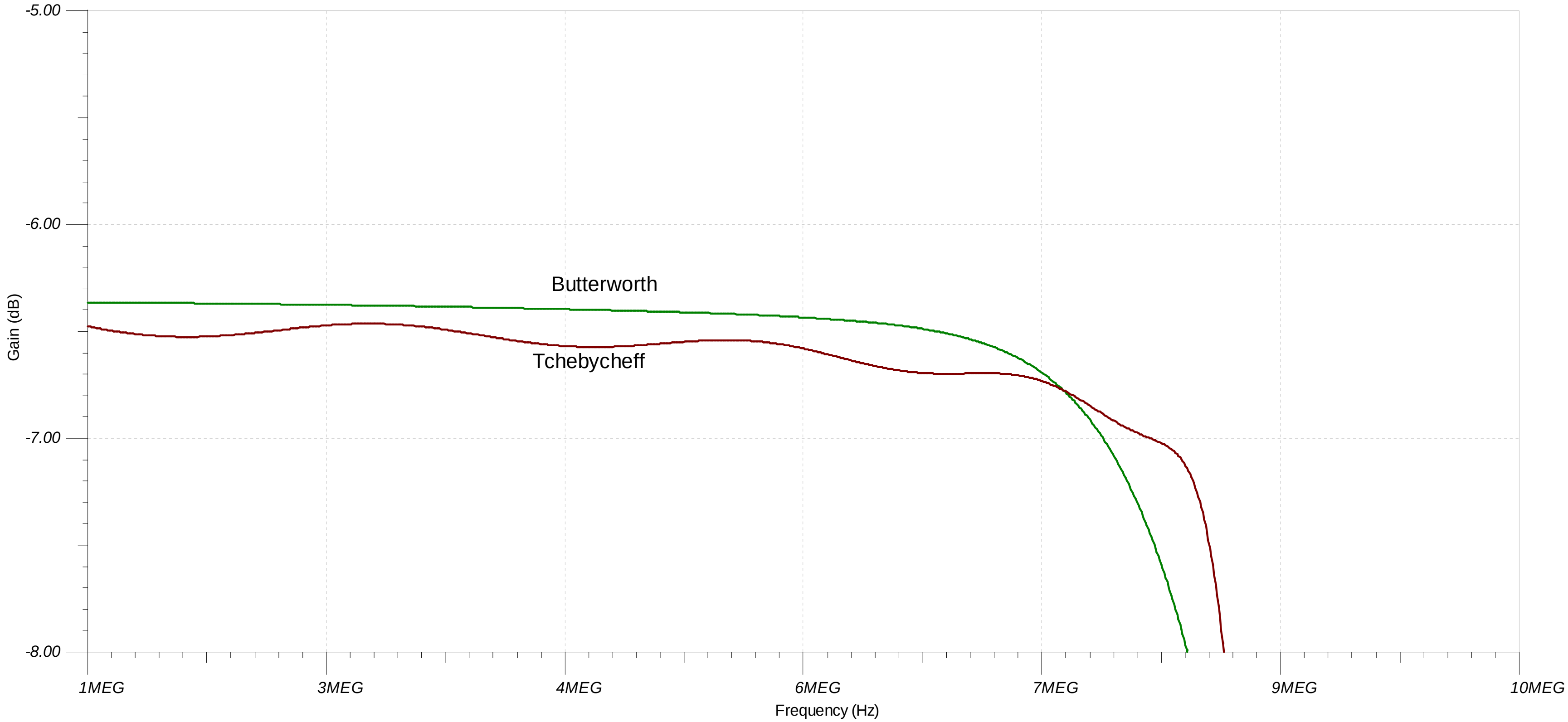


We are dreaming; you can’t buy these components. Inductors have a series resistance and a self-resonant frequency. The self-resonant frequency should be specified, or measured if you wind your own, and Q at some test frequency is specified. The resistance at the test frequency is simply the reactance divided by Q. Yes, it is frequency dependent due to skin effect. I calculated values of 1.12 ohms and 1.4 pf for the 1.78 uH inductor assuming a Q of 100 and a self-resonant frequency of 100 MHz, what the heck, I’m lazy I’ll use 1 ohm and 1 pf for all inductors; this is close enough to reality. Capacitors for filters have equivalent series resistance and self-resonant frequency also; usually much better than inductors; I’m lazy, they won’t be modeled. The next 3 demos were not presented at the meeting because of time constraint.

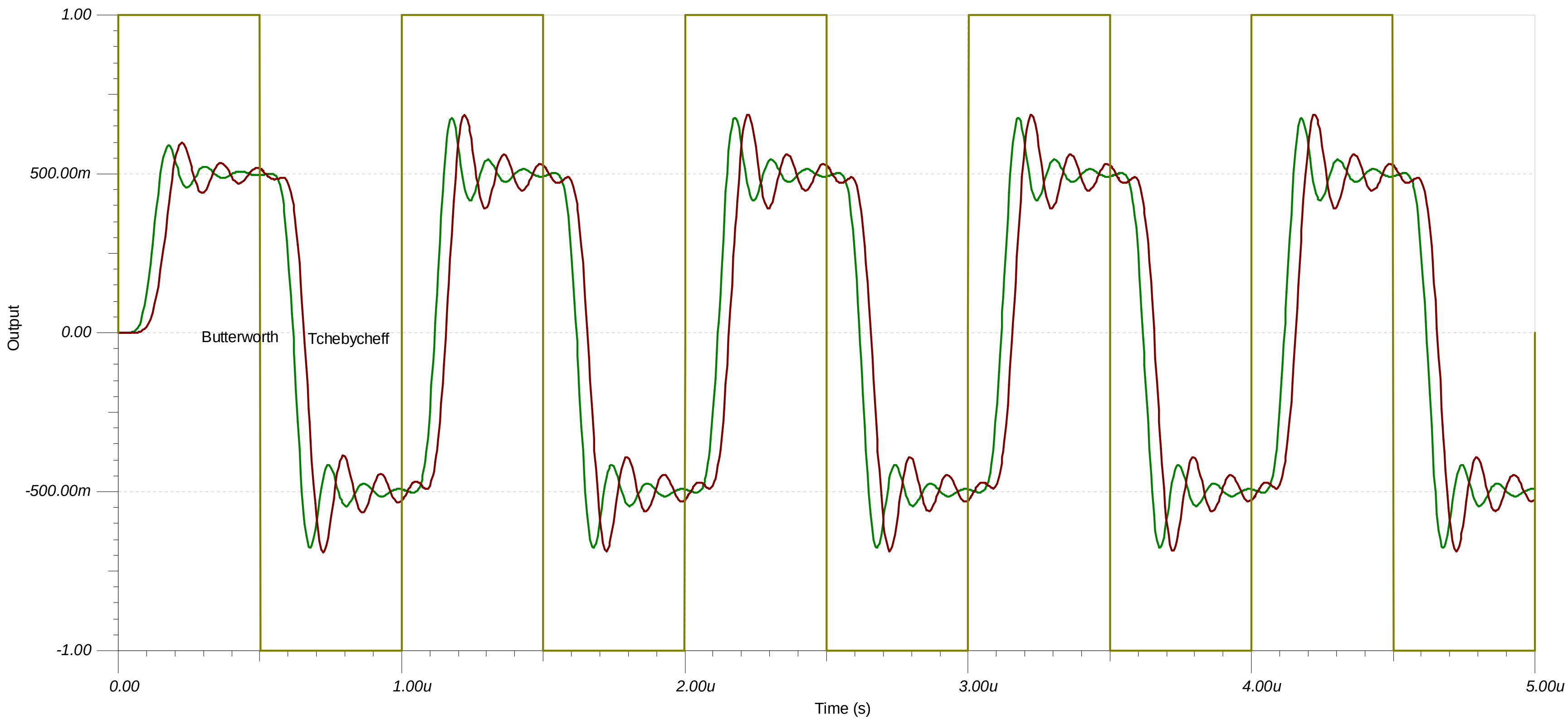


0.00
-100.00
-200.00
-300.00
-400.00
-500.00
Gain (dB)
Butterworth
Tchebycheff
1.00MEG
10.00MEG
100.00MEG
1.00G
Frequency (Hz)

We should look at ripple with the more realistic values.



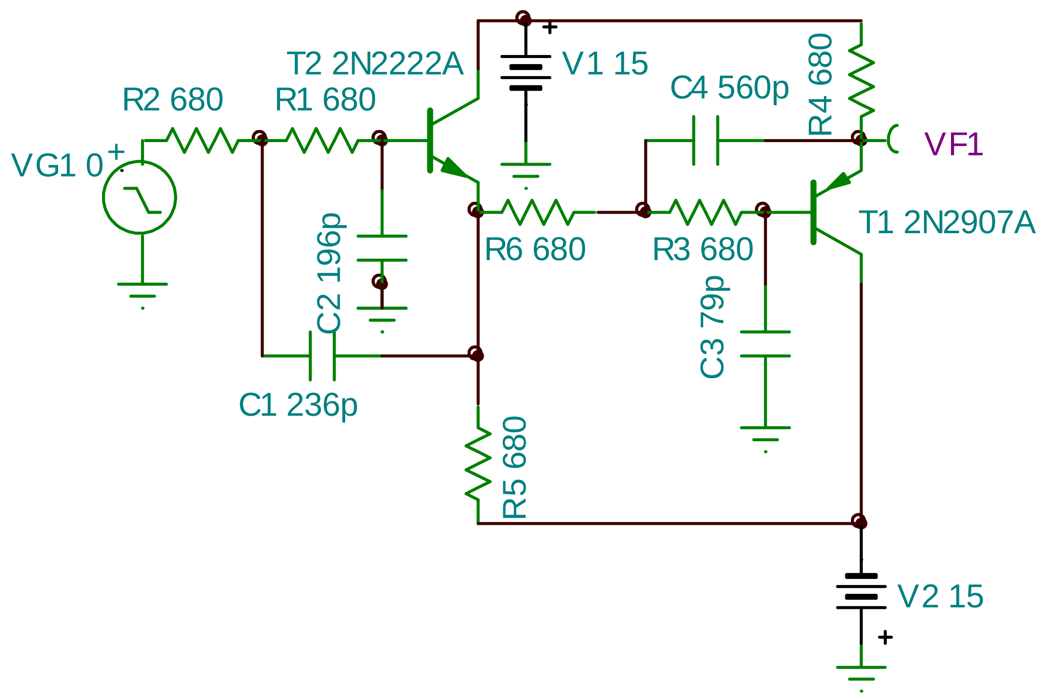
Just for completeness, let’s do a transient analysis.



These filters don’t look good for pulse work.

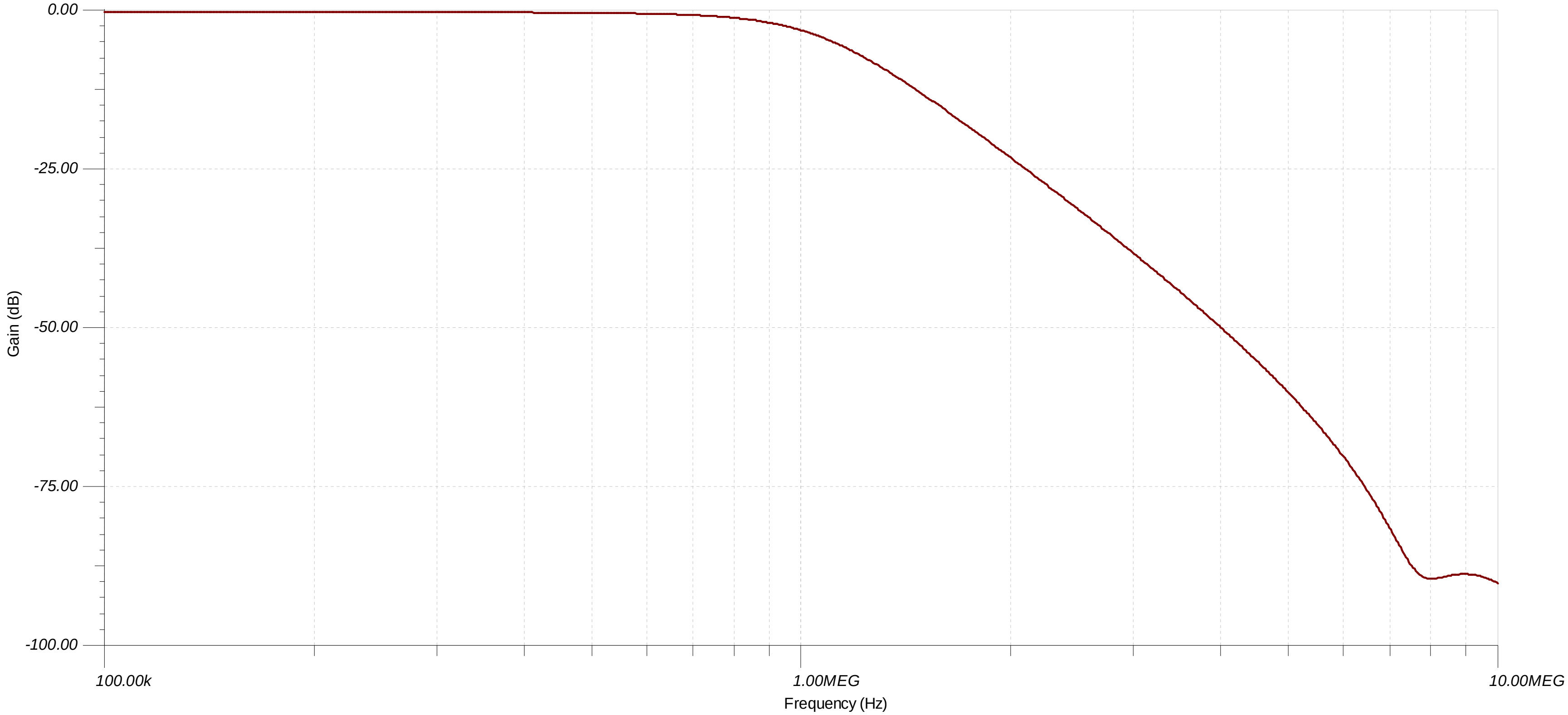
All of this analysis could be done with the early HP program I used. The next circuit could not, it did not have good non-linear models of transistors, FETs, diodes, SCRs, and Opamps. Tina does!

Since I have gotten us into filters, let’s analyze an active filter. I chose to analyze a 4 pole Butterworth filter formed by two Sallen and Keyes active filters in cascade. I used emitter followers instead of opamps because at 1 MHz they have much higher gain than opamps. Note that the Inductor-capacitor can be implemented with components that will stand the voltage and current levels associated with a transmitter final. Active filters are used at signal levels small enough not to blow the active components!



The transistors are operated at the same currents, thus the voltage drops due to base emitter voltage and base current should offset. Since it’s a 4 pole filter the cutoff is not as sharp as the 9 pole inductor capacitor filter.

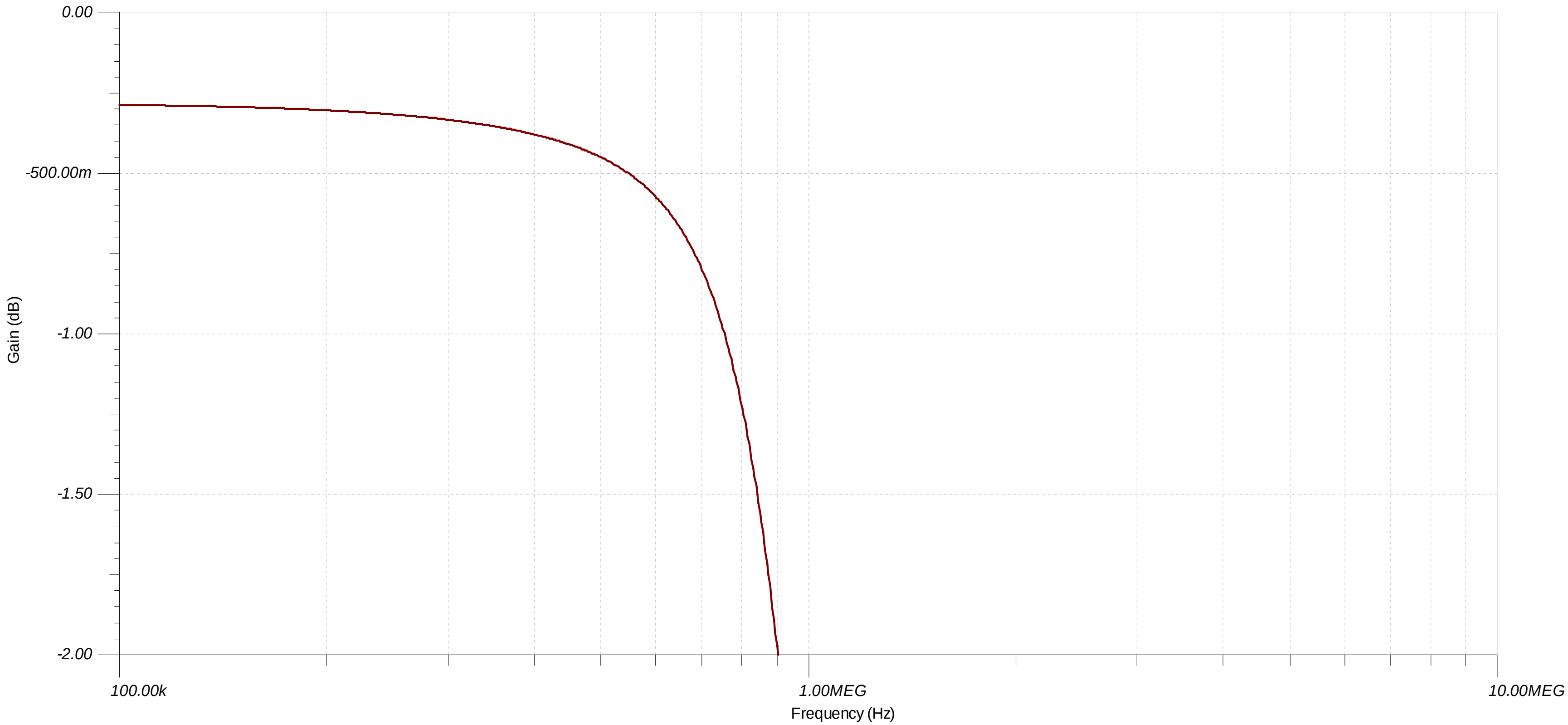
Frequency response for active filter



The floor for this filter is -80 dB, this is more than adequate for most applications.

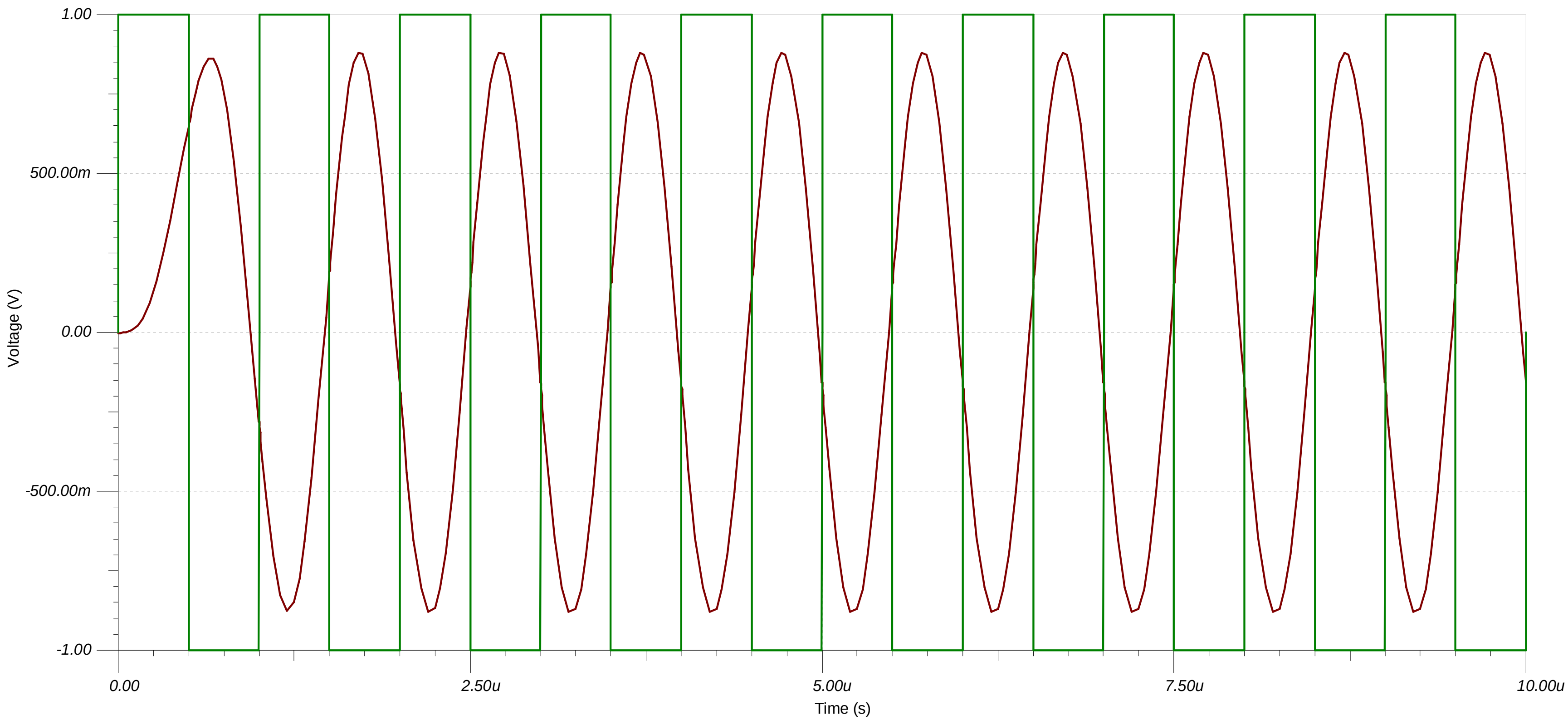
Expanded to show smoothness of response, Butterworth characteristic.

,

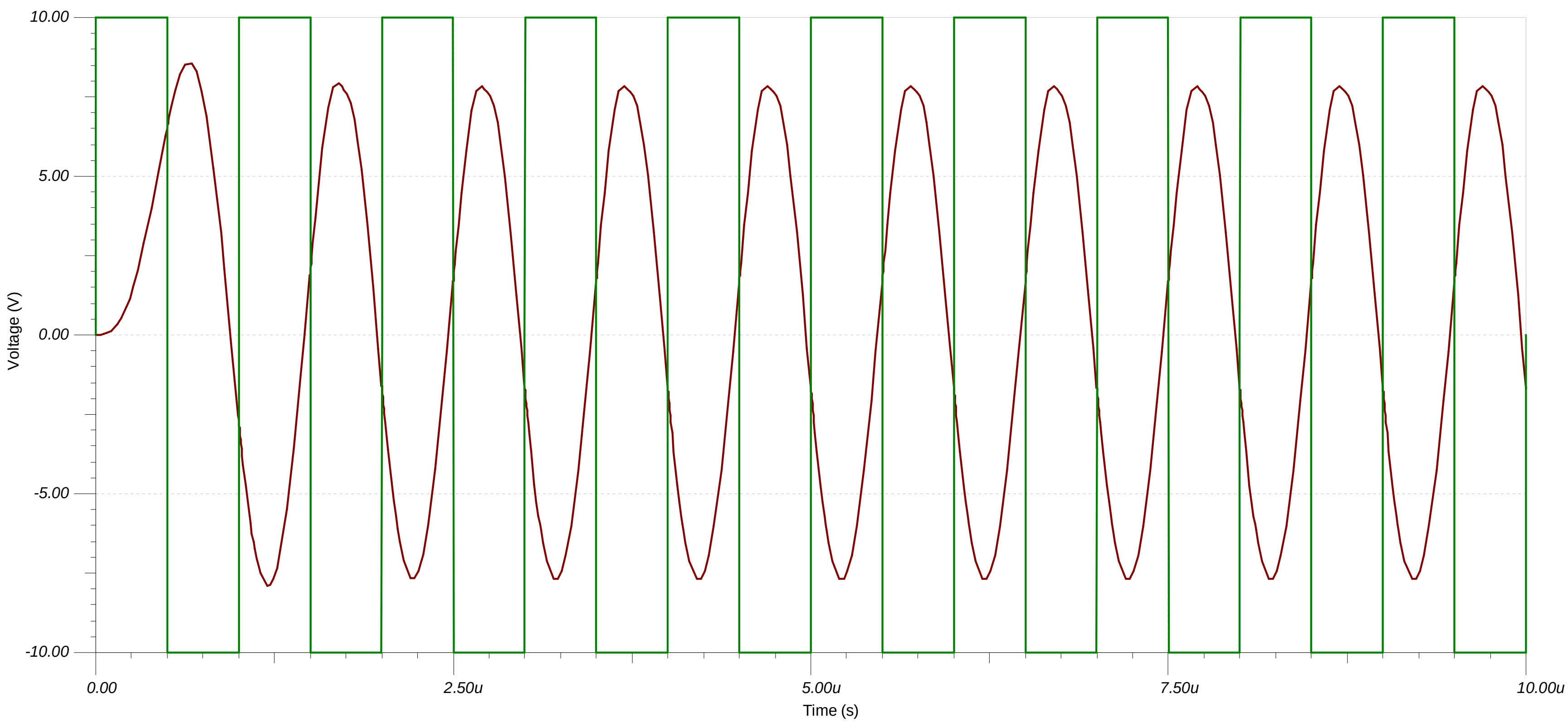


The overload power for the inductor capacitor filter is the failure point of the components, for the active filter overload is the dynamic range of the active device. This is somewhat less than the power supply voltage.

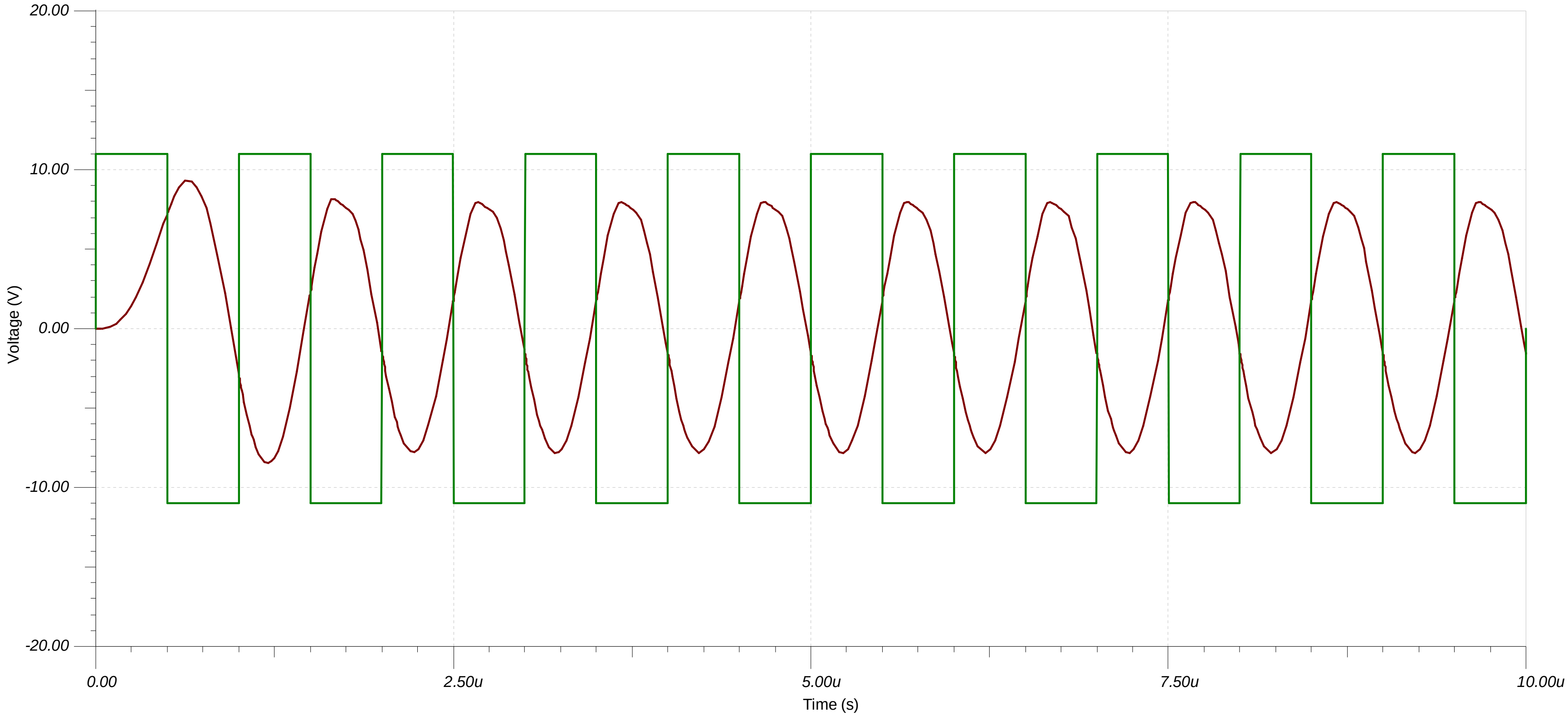
Removing the harmonics from a square wave. The square wave frequency is at the cut off treqeency, therefore the attenuation of the fundamental.



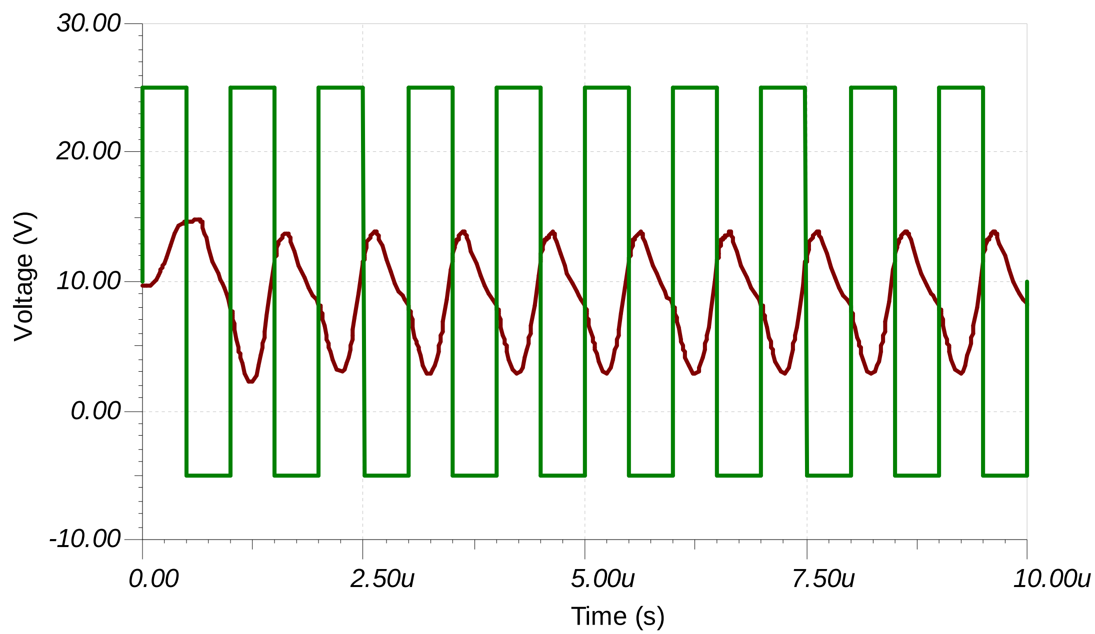
10V peak input



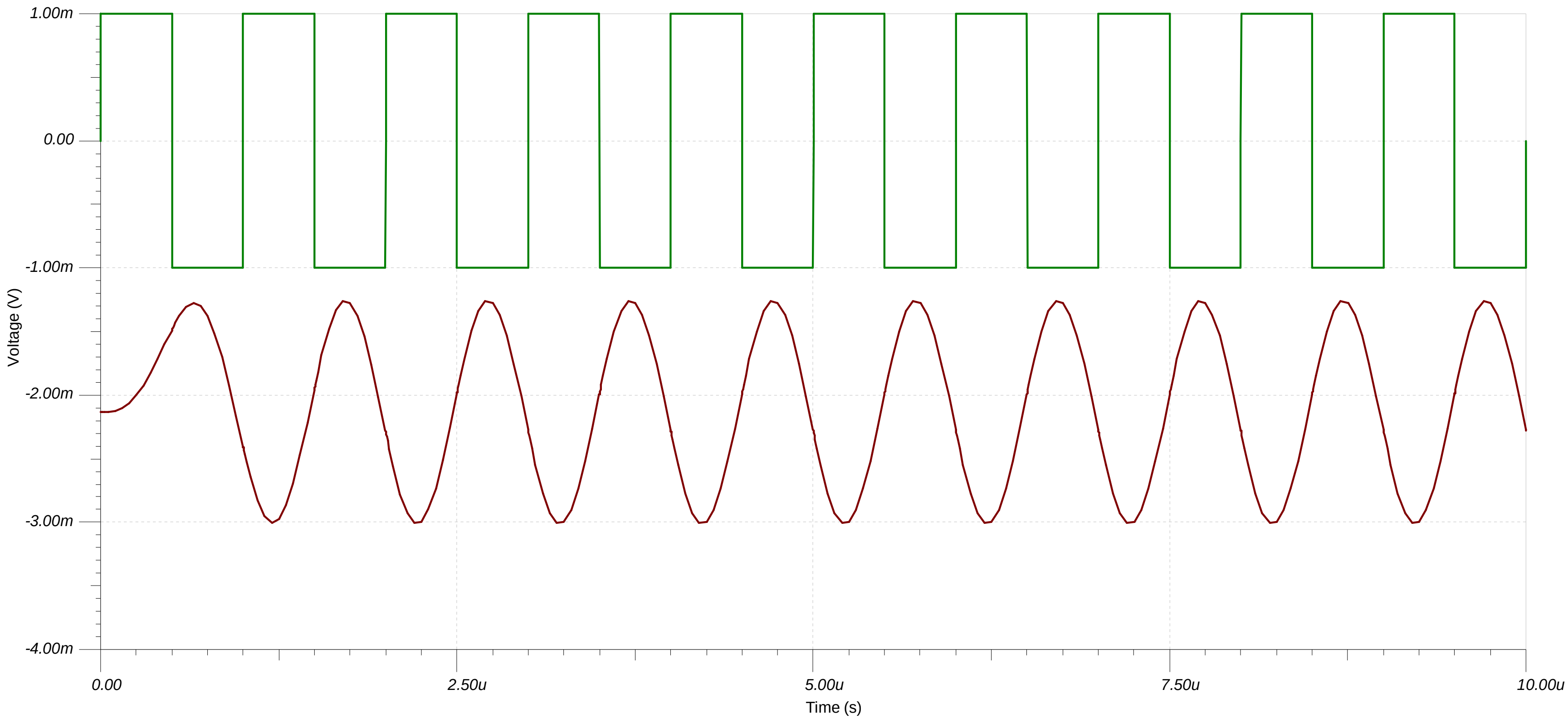
11V peak input



Note clipping begins, we are adding Harmonics to the output.



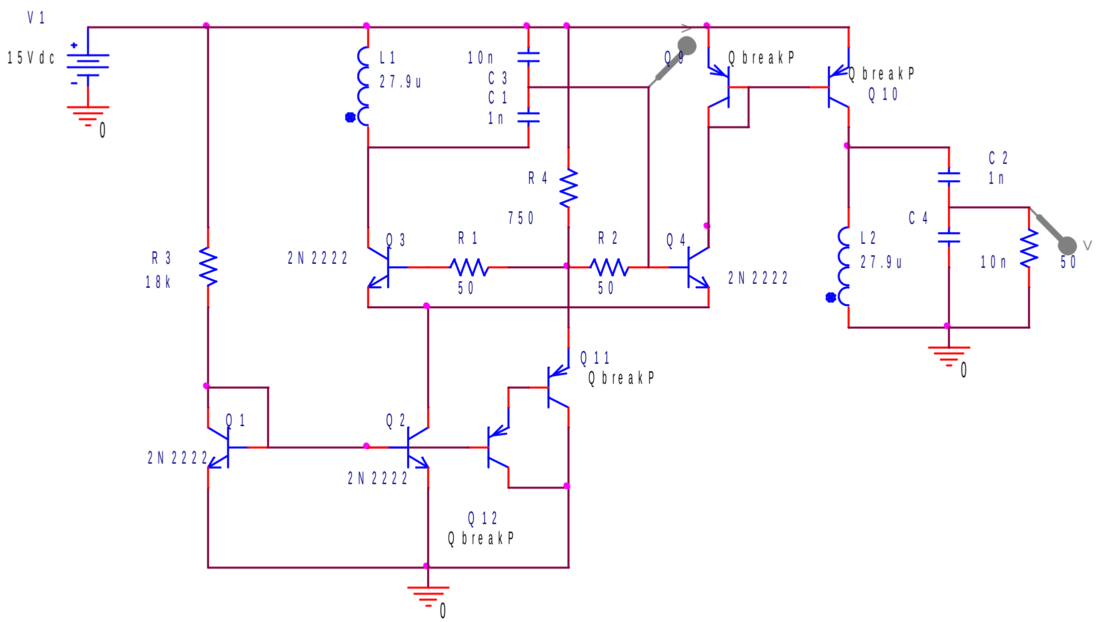
Now we’ve really got distortion

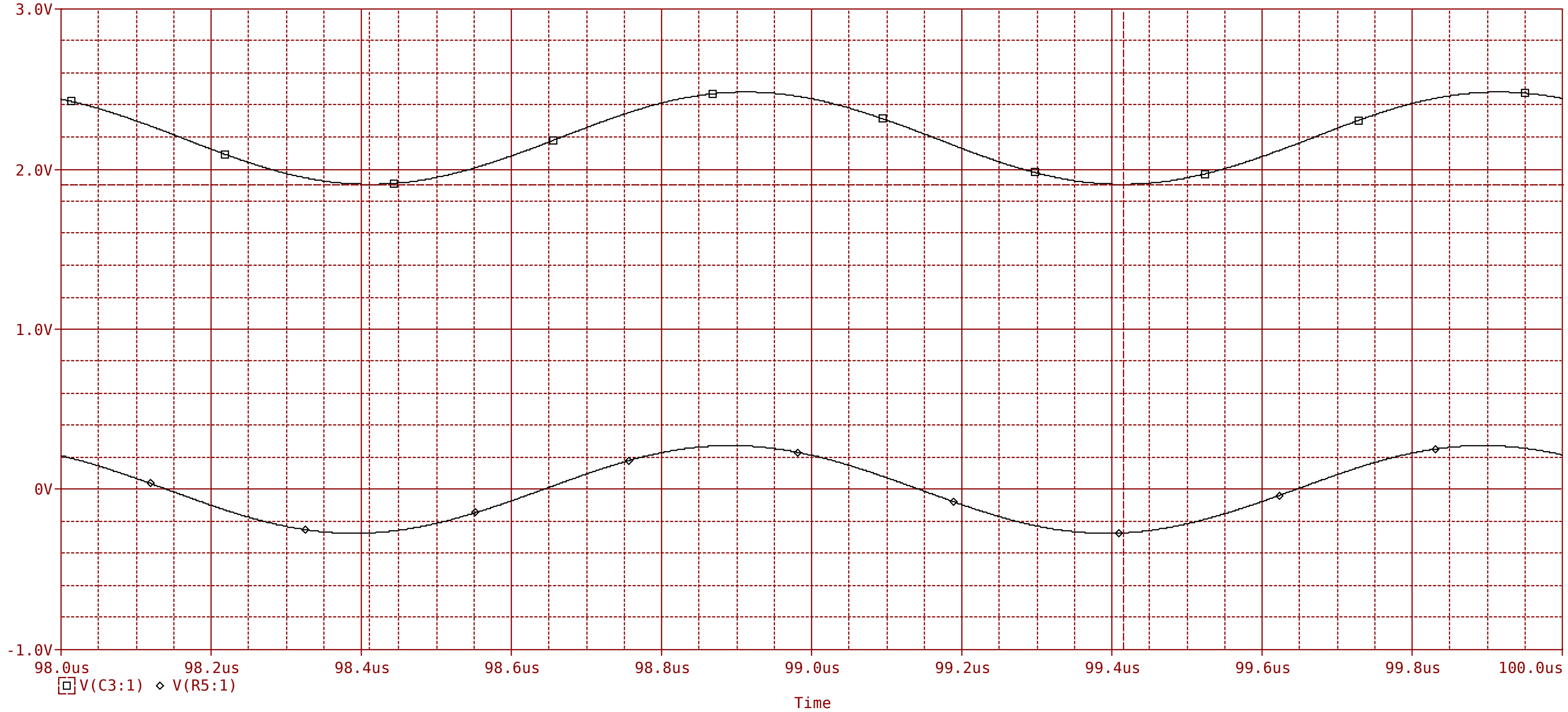


DC offset only 2 mV; comparable to opamps.

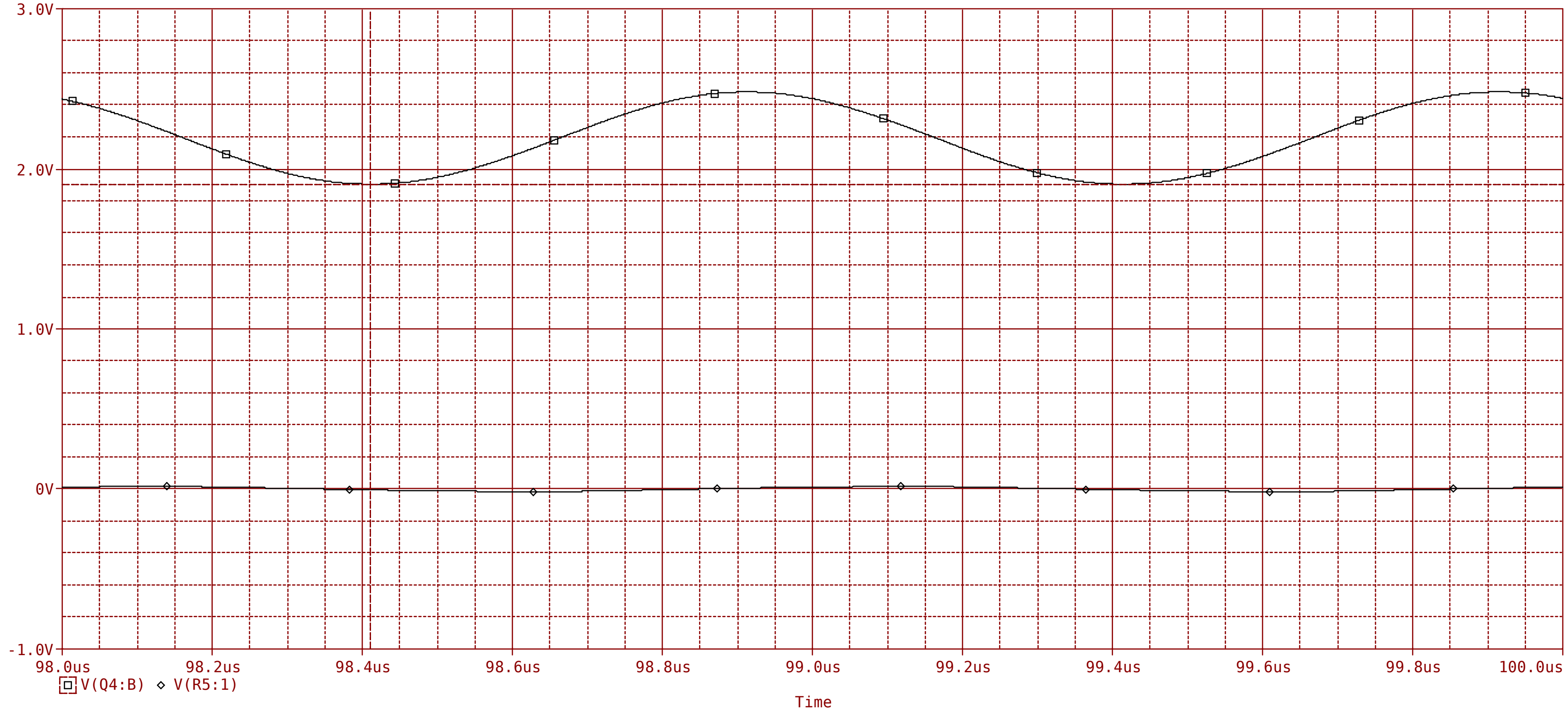
That was a pretty good workout of TINA, The following circuits are analyzed with PSpice.

The emitter coupled oscillator shown has excellent waveform purity and the current mirror buffer provides outstanding isolation.



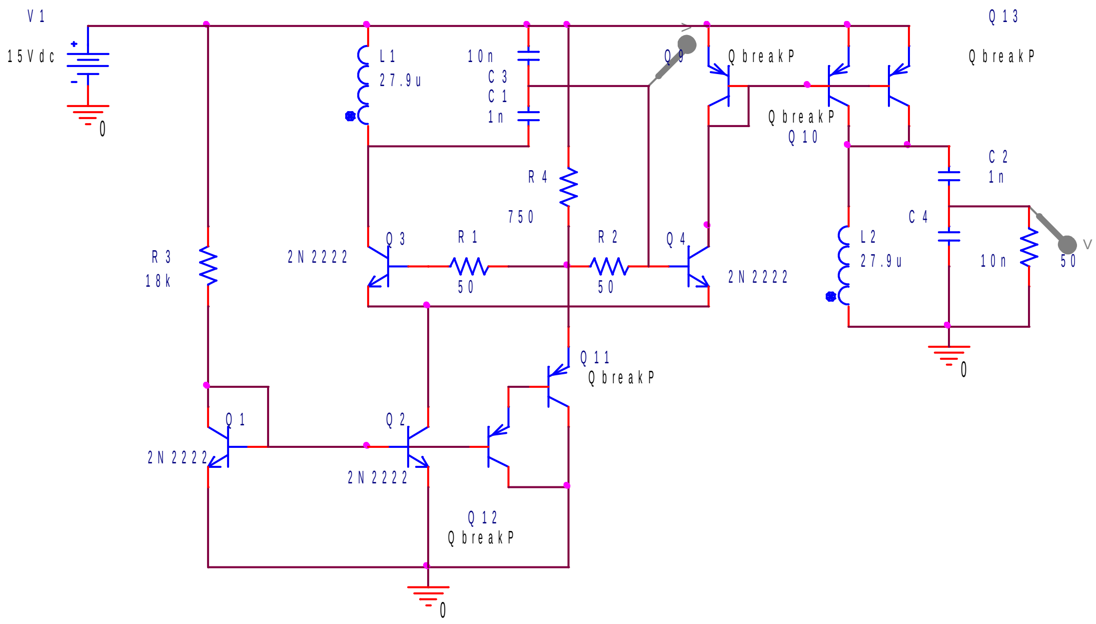


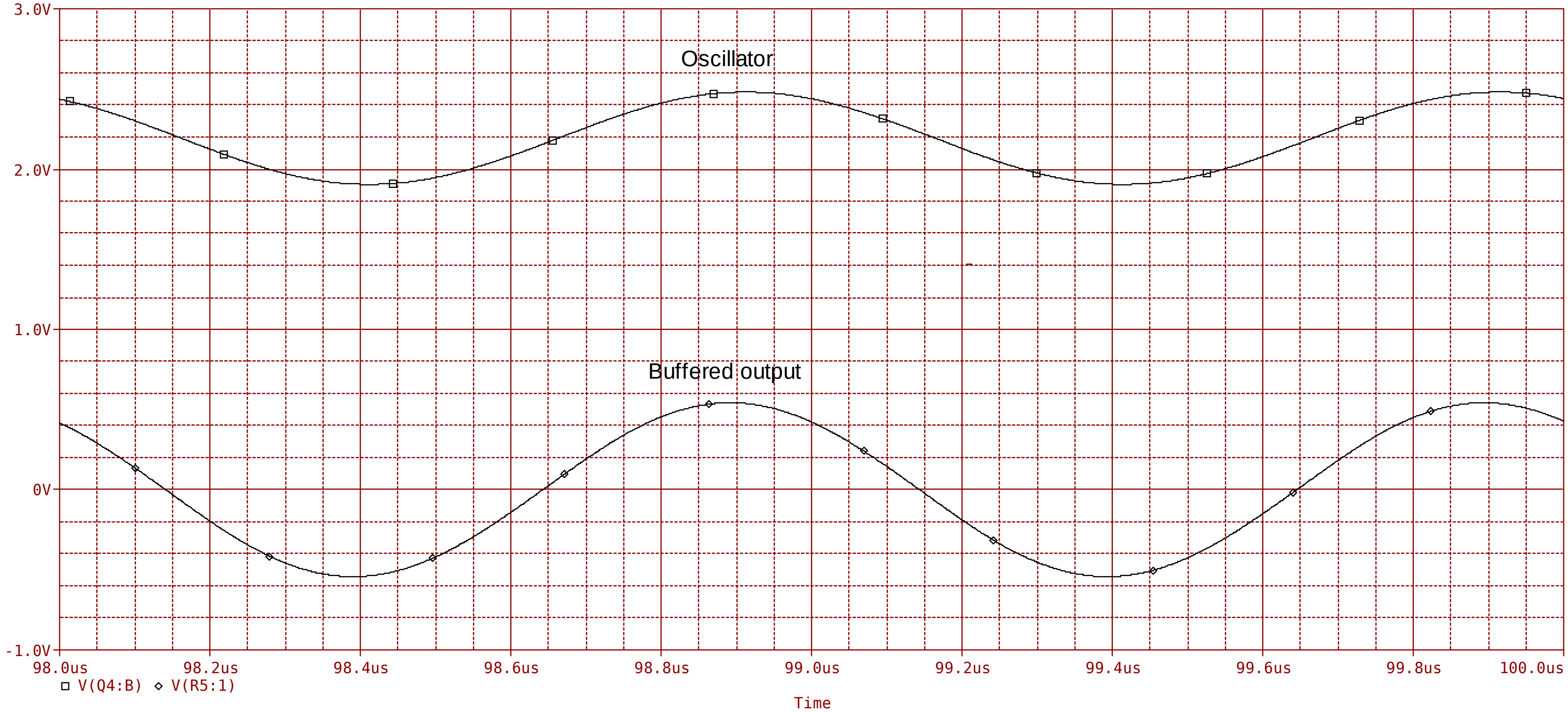
The period measured with cursors is 1.0021 us



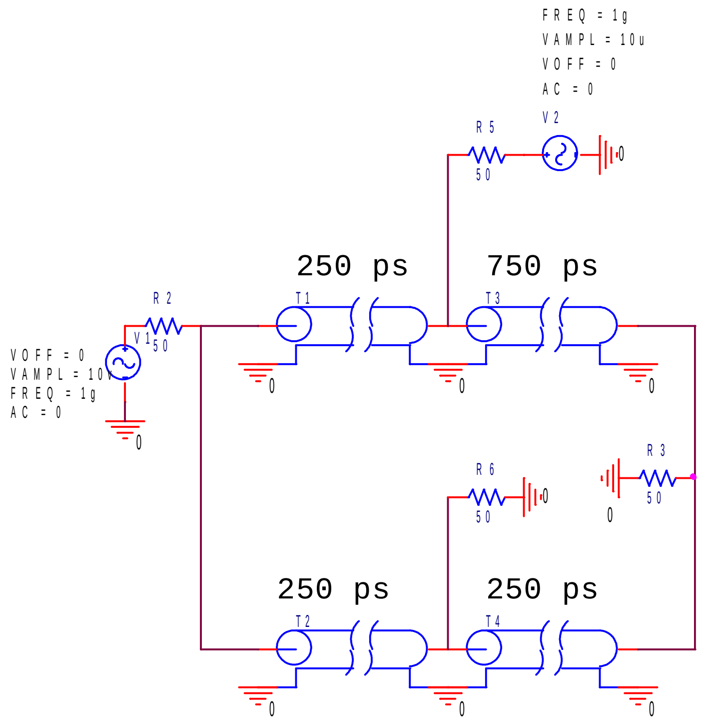
Crushing the amplitude and phase of the buffered output by loading it with a capacitor, the output of the oscillator itself has a period of 1.0021us, the same as when the buffer was tuned to the oscillator frequency. Neither the amplitude or frequency was changed at the oscillator; the buffer did its job!

Let’s add a transistor to the current mirror, or in a monolithic IC double the area of the buffer transistor.



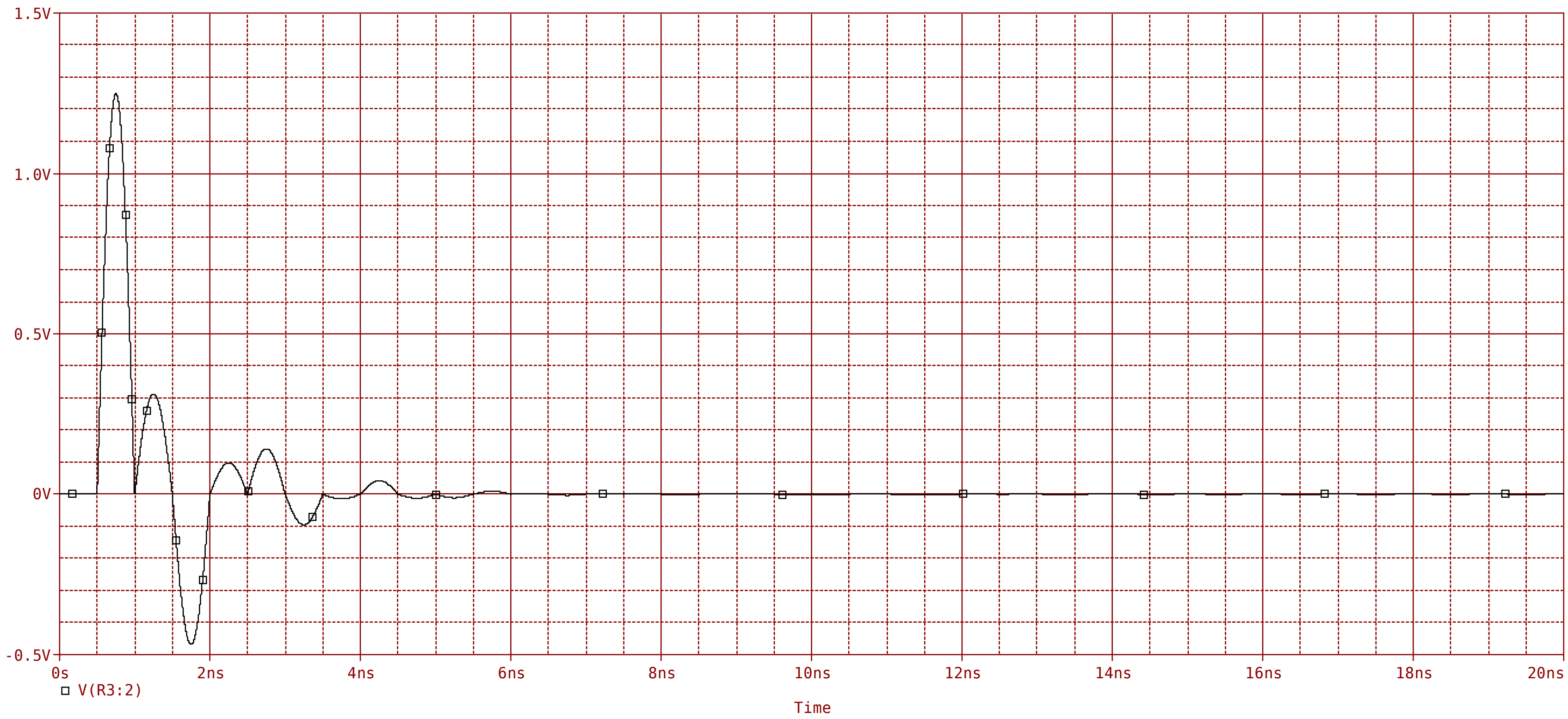

Oscillator
Buffered output
3.0V
2.0V
1.0V
0V
-1.0V
98.0us
98.2us
98.4us
98.6us
98.8us
99.0us
99.2us
99.4us
99.6us
99.8us
100.0us
V(Q4:B)
V(R5:1)
Time

PSpice has good models of transmission lines, let's model a 1 GHz transmit/receive isolator.



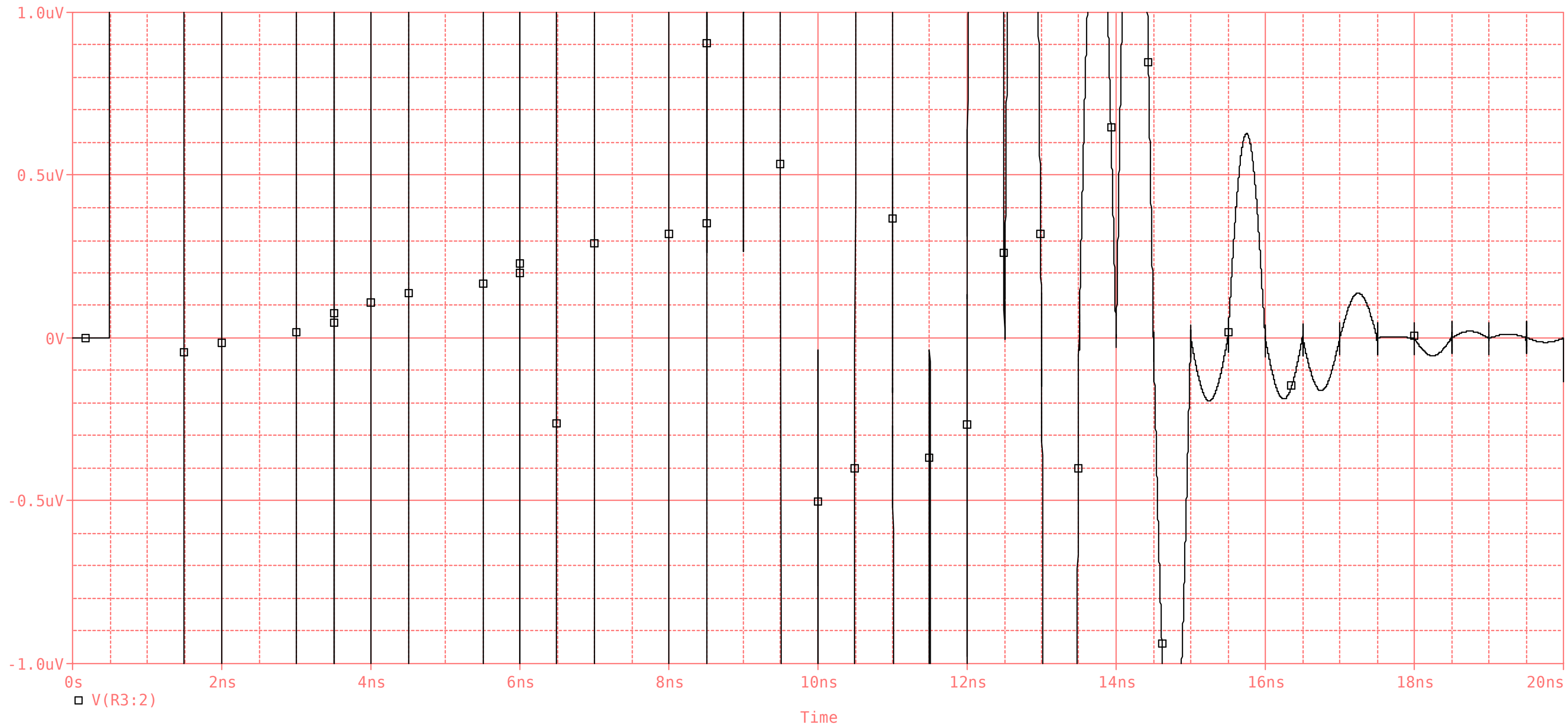
The large waveform is the voltage at the antenna, the small is the voltage at the receiver during transmit. Let's look at that more closely.

an

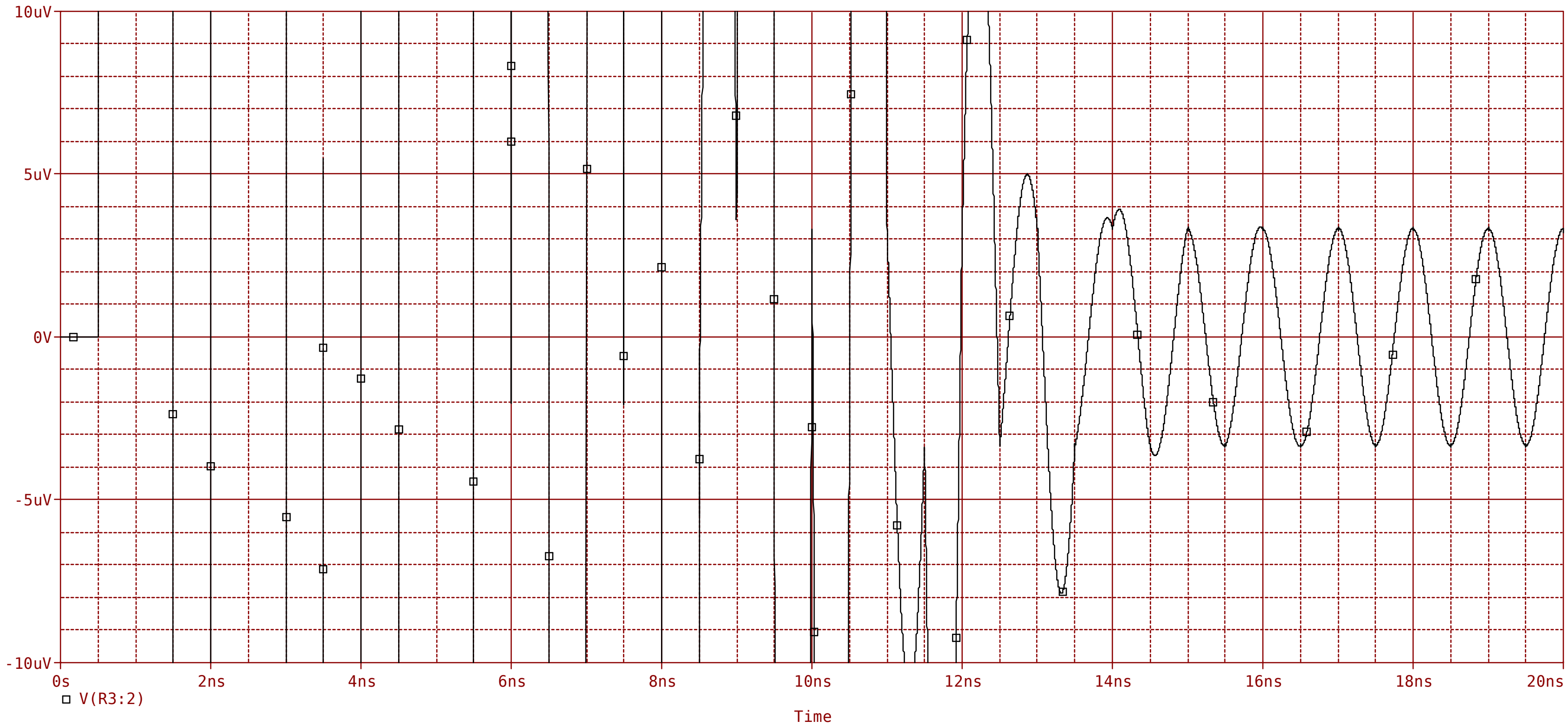


There's a wicked transient at the start which would surely overload the receiver, after a few cycles isolation occurs. However, if the rise-time on the pulse is long compared to the frequency it shouldn't be a problem. 30 ns, for instance, is a pretty long rise-time. The transient has died to less than a microvolt in 20 ns.

The plot below is the same simulation expanded to 2uv full scale. Transmit signal transient has dropped to less than a u-v in 20 ns

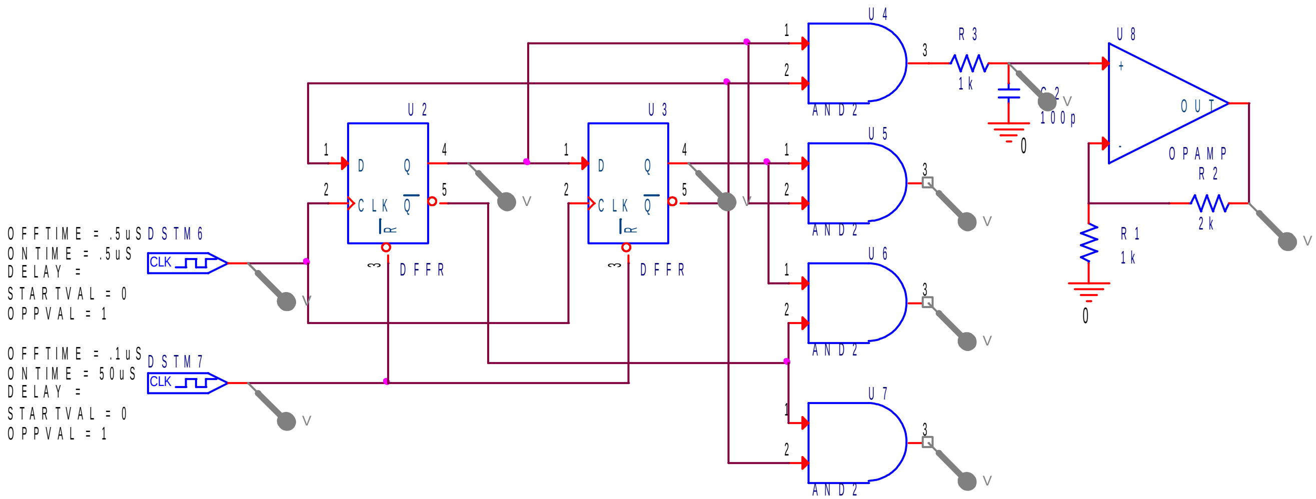


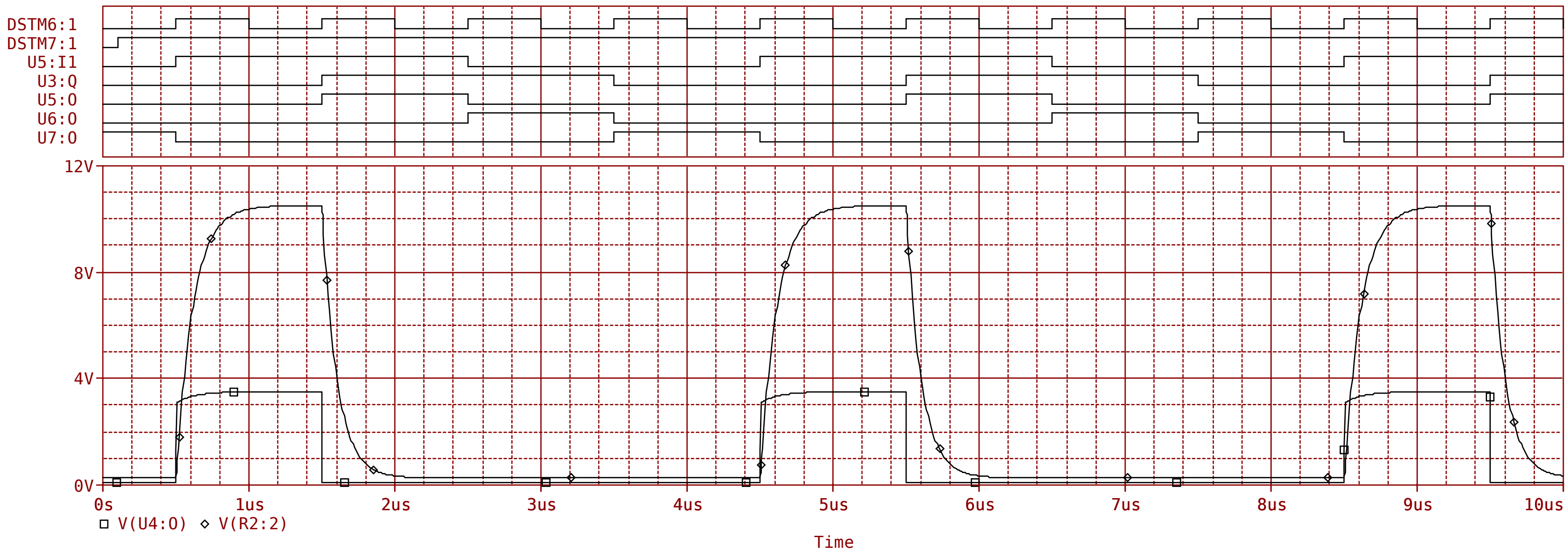
The simulation below shows receiving and transmitting on the same antenna at the same time. (Doppler radar?)



That shows the ability of PSpice to analyze transmission line circuits. It also has a logic analyzer built in.

The circuit below is a 2 stage twisted ring (Johnson) counter with decode logic to produce a pulse corresponding to the selected counter state at each gate output. This is buffered with a gain of 3 opamp buffer just to show mixed signal capability.





Top line is the clock being counted.

Second line is the clear pulse to initialize the counter. A two stage Johnson counter is self-clearing, but the simulator isn't.

Third line is the Q output of ff1.

Fourth line is the Q output of ff2.

Fifth line is state 2 pulse

Sixth line is state 3 pulse

Seventh line is the state 4 pulse.

Small analog pulse is the state 1 output, since it's an analog interface, the simulator shows the ttl voltage.

Large analog pulse is the filtered and amplified output.

Wrap up

The student versions of TIN.A and OrCAD (PSpice) were demonstrated. I got them free because I was teaching at PCC. The students also got OrCAD free when we used it, and TINA free when we switched to it. I think the student versions are sold to the general public for around $500.00, but I'm not a rep for either company. I think the price for full OrCAD is around $20,000.00. I don't know about TINA but I'm sure it's in the same ballpark. TINA is easier to use.

PSpice is a little bit geeky to use. (7 steps to place a part) and I often had to refer to the manual to do simple tasks. However it comes with printed circuit layout software. The accuracy is excellent, I obtained 6 figure match with differential equation solutions using Excel for arithmetic. TINA is more intuitive and easier to use. The student versions don't have all the functions of the industrial versions and won't analyze nearly as complex circuits; however for ham radio projects a few thousand nodes is more than adequate.

I preferred Microcap when I was consulting. I purchased Microcap 4 for my consulting business at $3500.00. (PSpice, without the layout capability was $5000.00.) I upgraded Microcap 5 which had a $5000.00 price tag; but Spectrum Software, who develops and sells Microcap, gives a credit of the purchase price of the version you have when upgrading. 10 years ago Microcap had a $15000.00 price tag. It was easy to use.