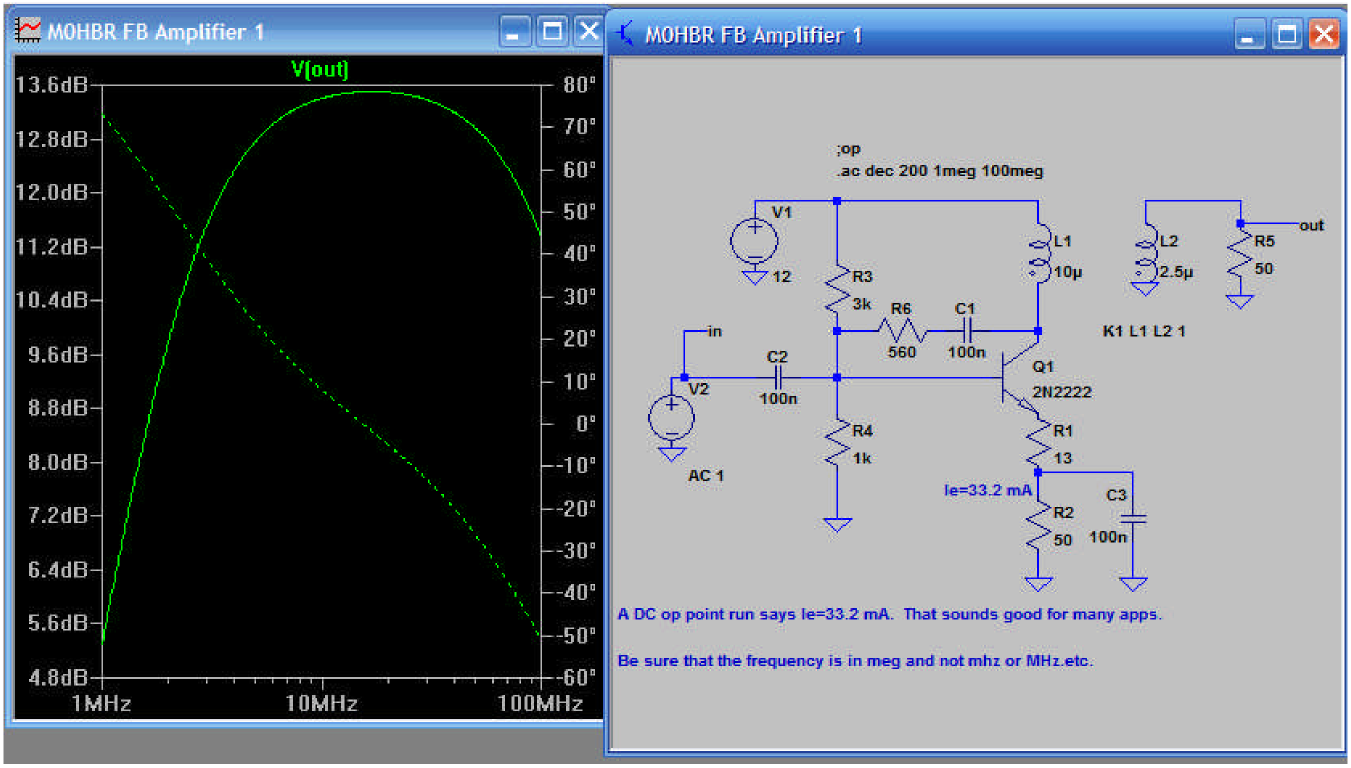
Basic Amplifier Analysis in SPICE

Wes Hayward, w7zoi, 12Jan07

This is a collection of simulations done to investigate a very basic feedback amplifier. The circuit includes both emitter degeneration and collector-to-base resistance. The development is presented in an evolution aimed at illustrating the methods. Generally, we will just insert the figures. Comments within them will then present most material.

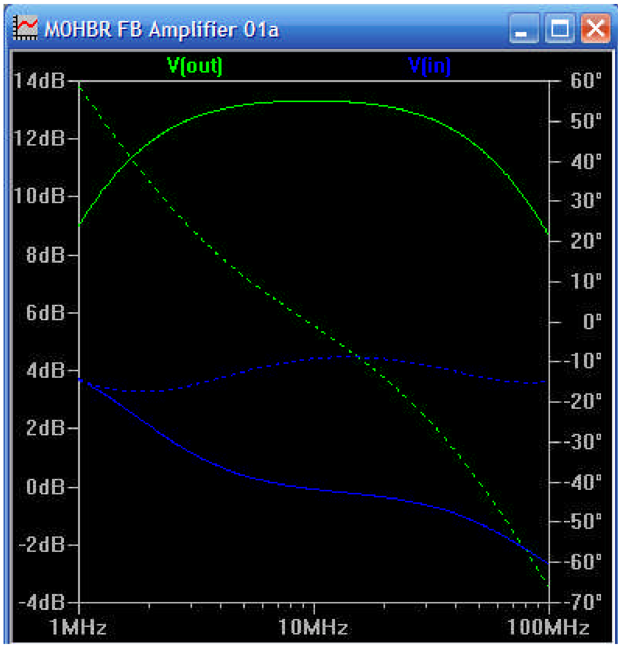
The simulations were all done with SWCad, a wonderful SPICE program offered by Linear Technology, Inc. I highly recommend the program to those wanting to do SPICE simulations. However, I urge the user to consider a personal observation that developed over 13 years of integrated circuit design with SPICE and other programs: **The Simulation is the Greater Experiment**. That is, the results are only as good as the models and the completeness of the simulation files. The *real truth* lies with experimental results.

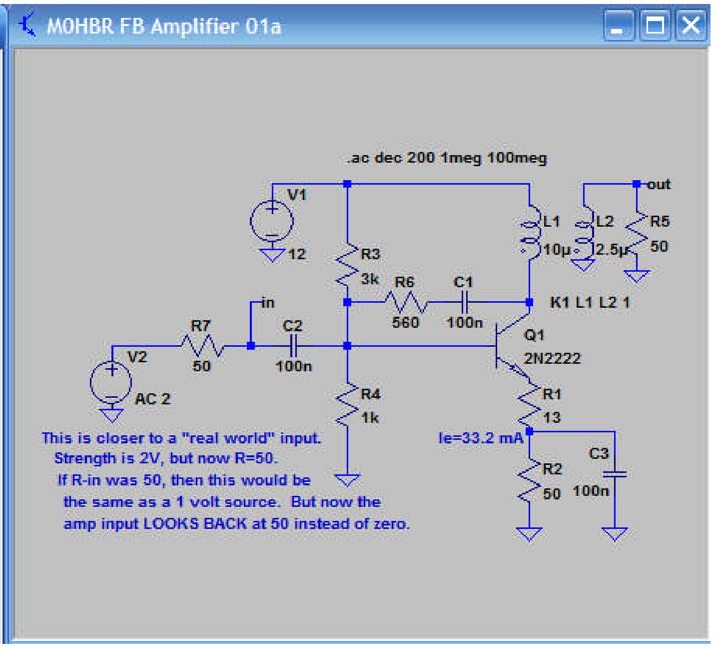


```
;op
.ac dec 200 1meg 100meg
```

A DC op point run says Ie=33.2 mA. That sounds good for many apps.

Be sure that the frequency is in meg and not mhz or MHz.etc.



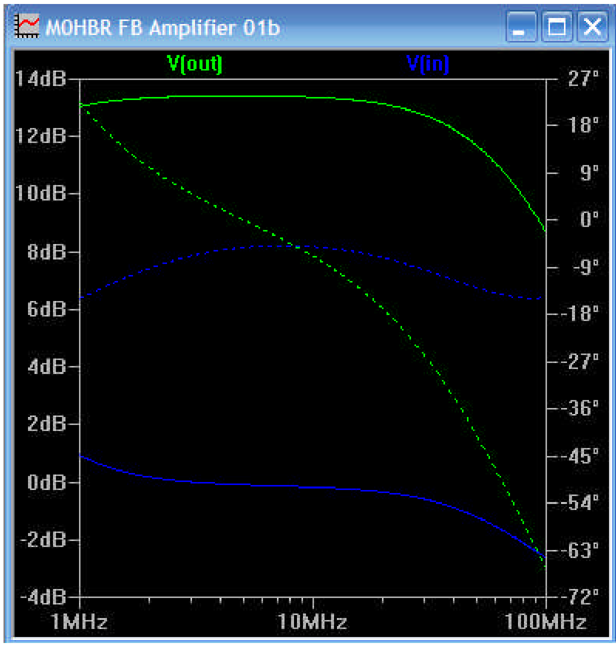


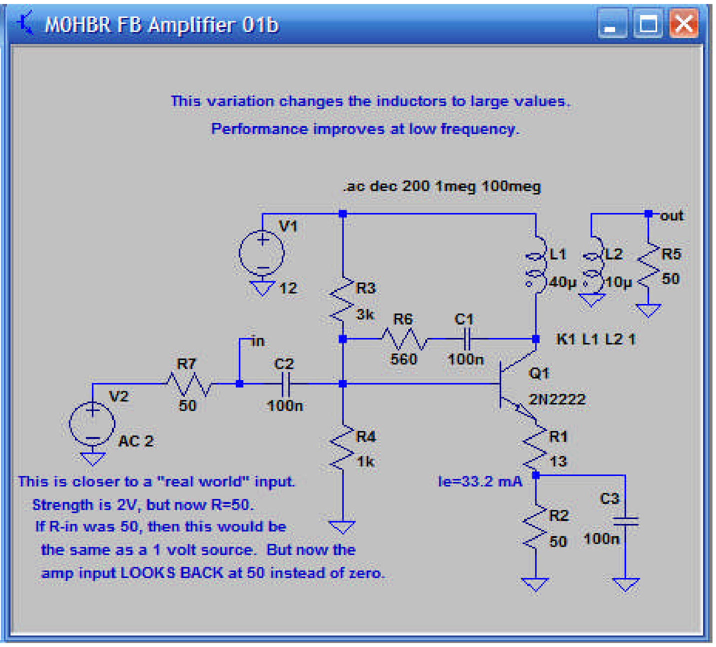
.ac dec 200 1meg 100meg

K1 L1 L2 1

Ie=33.2 mA

This is closer to a "real world" input.
Strength is 2V, but now R=50.
If R-in was 50, then this would be
the same as a 1 volt source. But now the
amp input LOOKS BACK at 50 instead of zero.





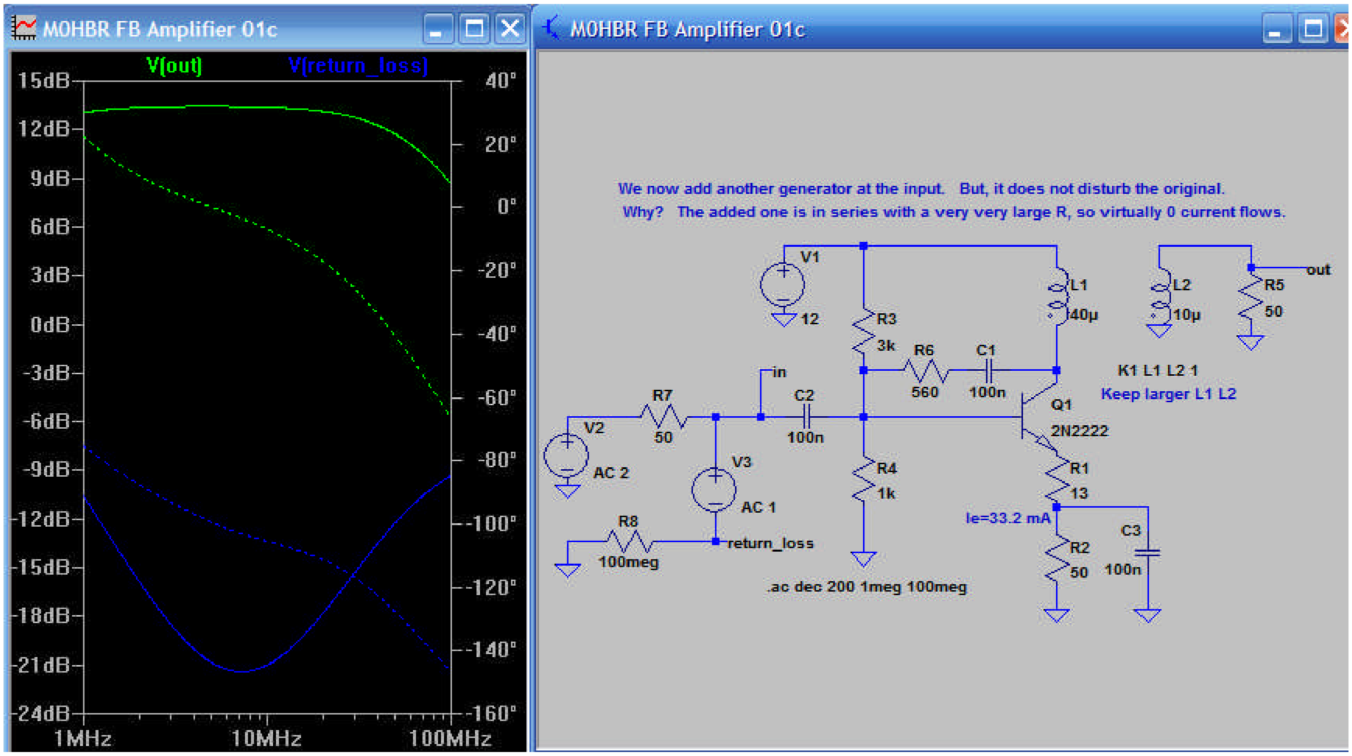
This variation changes the inductors to large values.
Performance improves at low frequency.

.ac dec 200 1meg 100meg

K1 L1 L2 1

Ie=33.2 mA

This is closer to a "real world" input.
Strength is 2V, but now R=50.
If R-in was 50, then this would be
the same as a 1 volt source. But now the
amp input LOOKS BACK at 50 instead of zero.



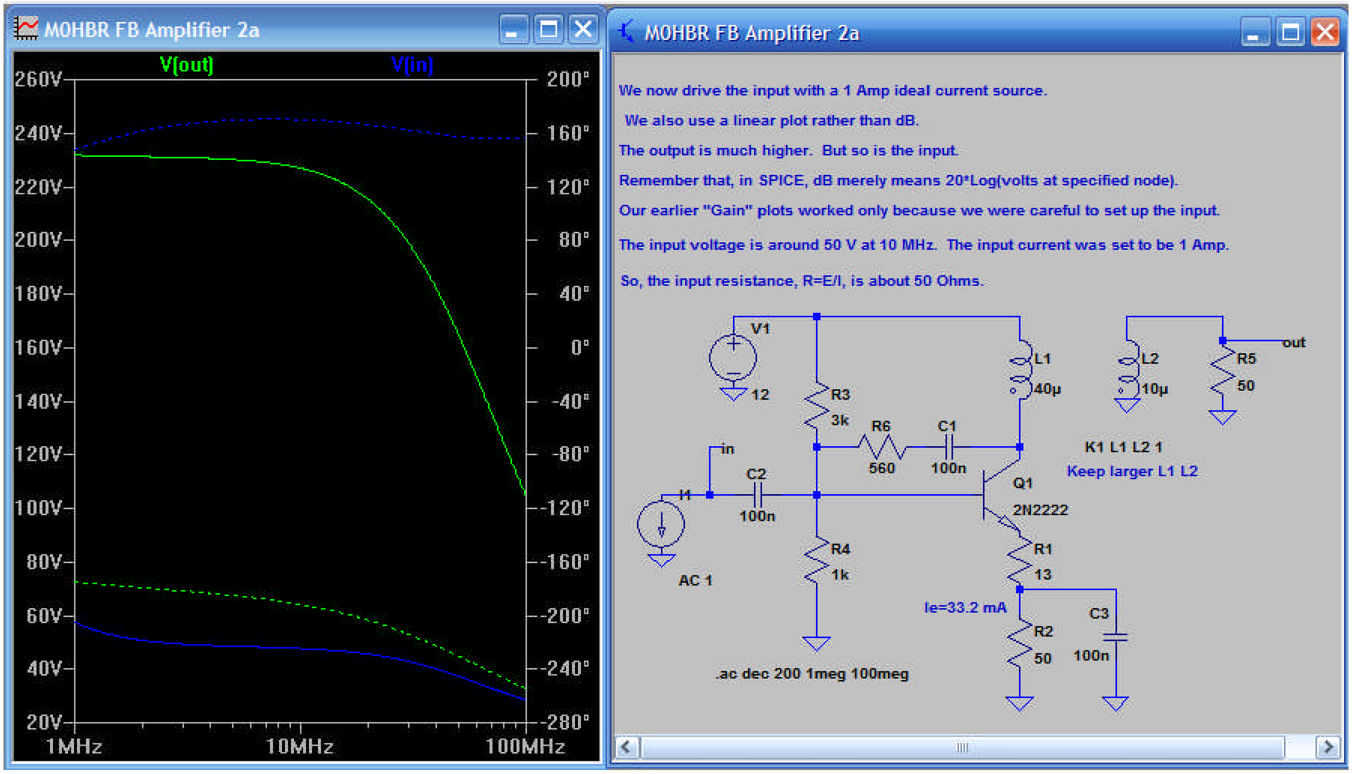
We now add another generator at the input. But, it does not disturb the original.
Why? The added one is in series with a very very large R, so virtually 0 current flows.

```
K1 L1 L2 1
```

Keep larger L1 L2

```
.ac dec 200 1meg 100meg
```

The methods used here are described in greater detail in “Reflections on the Reflection Coefficient: An Intuitive Examination,” QEX, January, 1993. The point labeled “return loss” should formally be named “S11” or “gamma”. In dB form, it is the negative of return loss.



We now drive the input with a 1 Amp ideal current source.

We also use a linear plot rather than dB.

The output is much higher. But so is the input.

Remember that, in SPICE, dB merely means 20*Log(volts at specified node).

Our earlier "Gain" plots worked only because we were careful to set up the input.

The input voltage is around 50 V at 10 MHz. The input current was set to be 1 Amp.

So, the input resistance, R=E/I, is about 50 Ohms.

The input impedance is easily calculated from Ohm's Law:

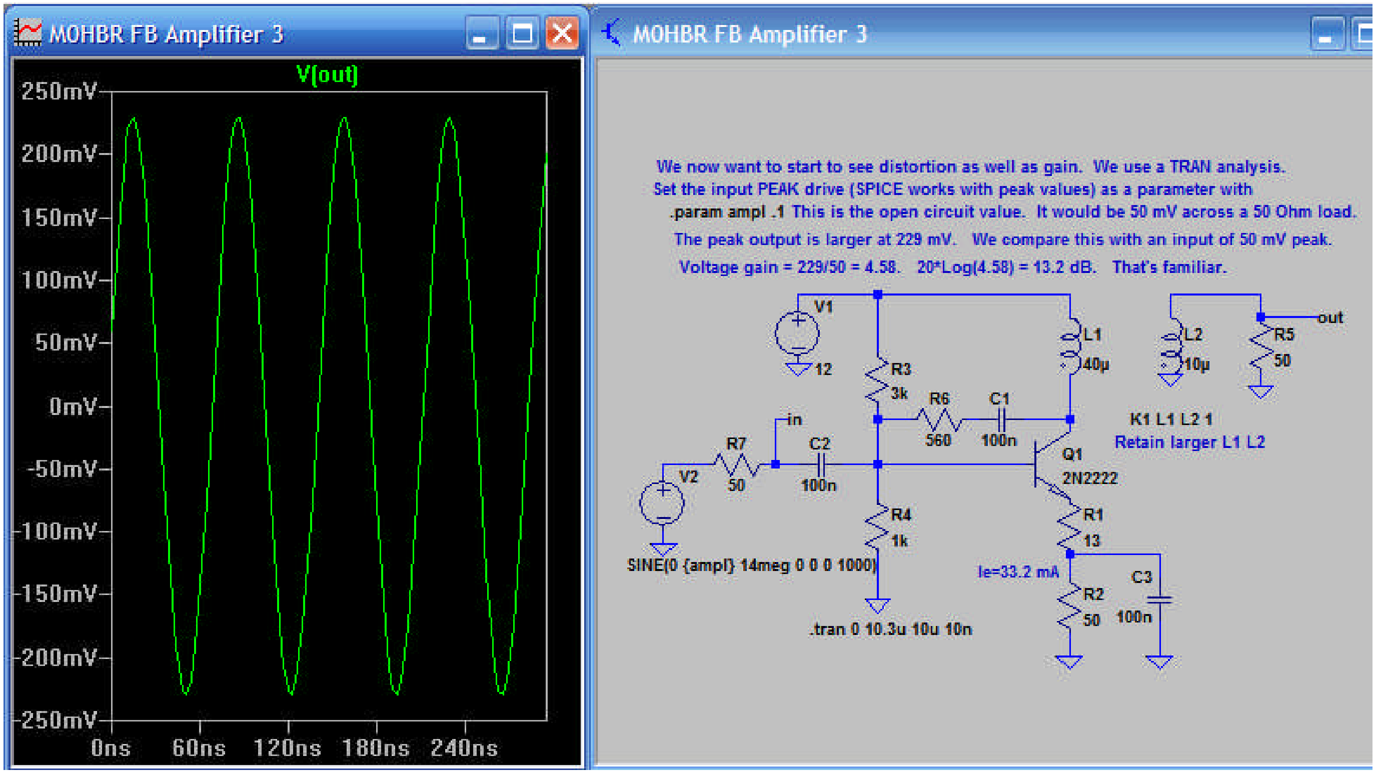
Impedance calculations

The voltage is 47.69 at 170.2 degrees. The current is 1 Amp, but it is ***out*** of the amplifier, so has phase of 180 degrees. The impedance is then the ratio, 47.69 with a phase of -9.8 degrees.

$$Z = A\cdot\left(\cos(\theta) + j\cdot\sin(\theta)\right)$$

$A := 47.69 \quad \theta := \frac{-9.8\cdot\pi}{180} \qquad \theta = -0.171$ radians

So, $Z = 46.99 - j\cdot 8.12$



We now want to start to see distortion as well as gain. We use a TRAN analysis.

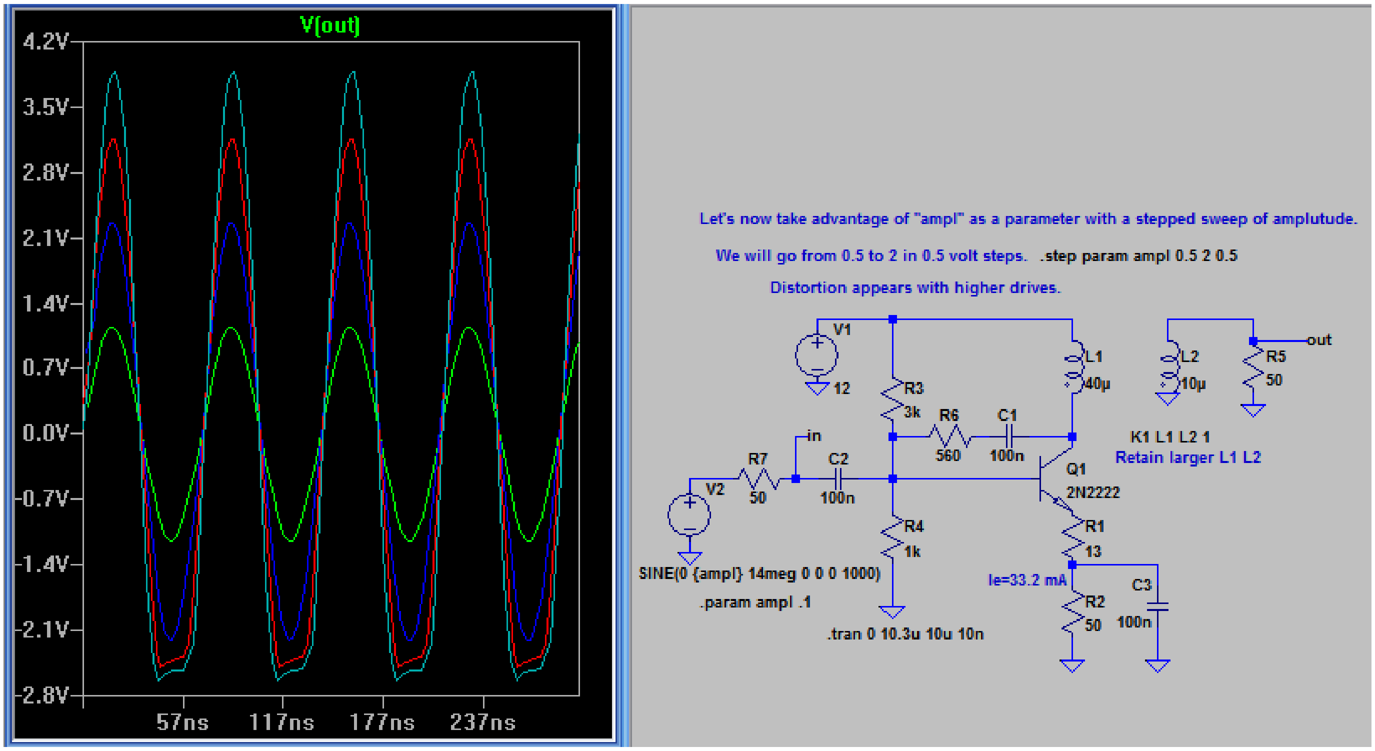
Set the input PEAK drive (SPICE works with peak values) as a parameter with

.param ampl .1 This is the open circuit value. It would be 50 mV across a 50 Ohm load.

The peak output is larger at 229 mV. We compare this with an input of 50 mV peak.

Voltage gain = 229/50 = 4.58. 20*Log(4.58) = 13.2 dB. That's familiar.

```
SINE(0 {ampl} 14meg 0 0 0 1000)
.tran 0 10.3u 10u 10n
```

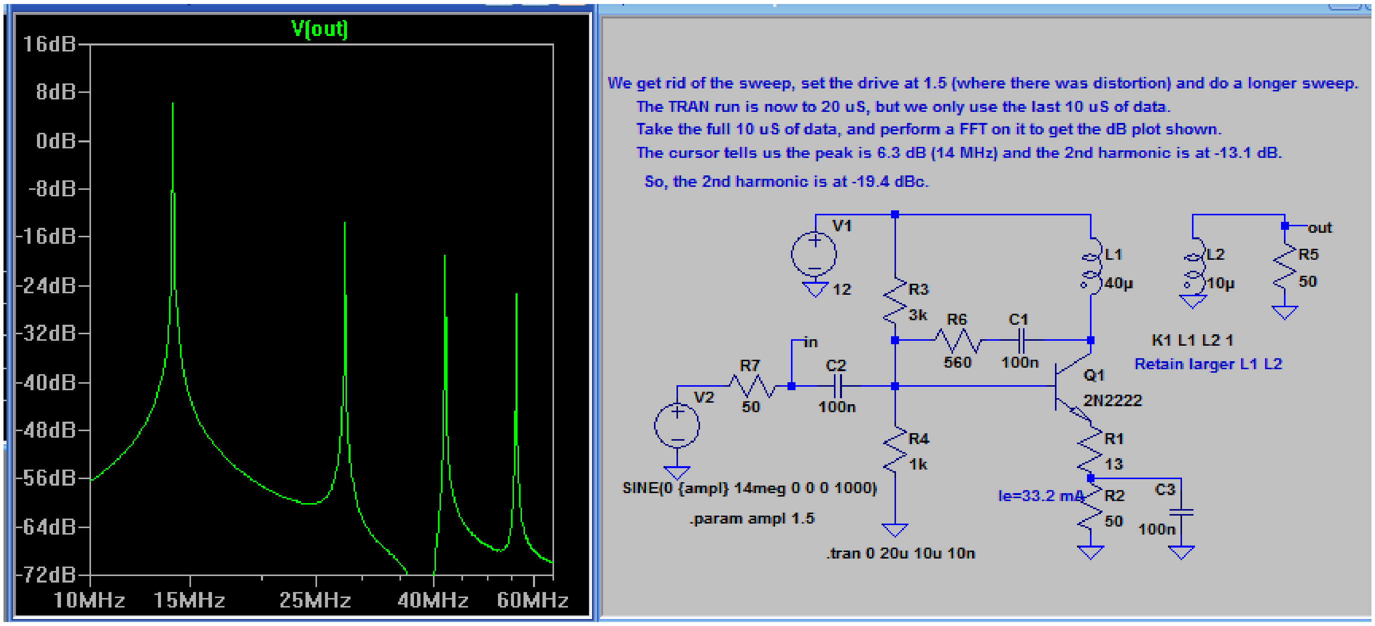


Let's now take advantage of "ampl" as a parameter with a stepped sweep of amplutude.

We will go from 0.5 to 2 in 0.5 volt steps. .step param ampl 0.5 2 0.5

Distortion appears with higher drives.

```
SINE(0 {ampl} 14meg 0 0 0 1000)
.param ampl .1
.tran 0 10.3u 10u 10n
```



We get rid of the sweep, set the drive at 1.5 (where there was distortion) and do a longer sweep.

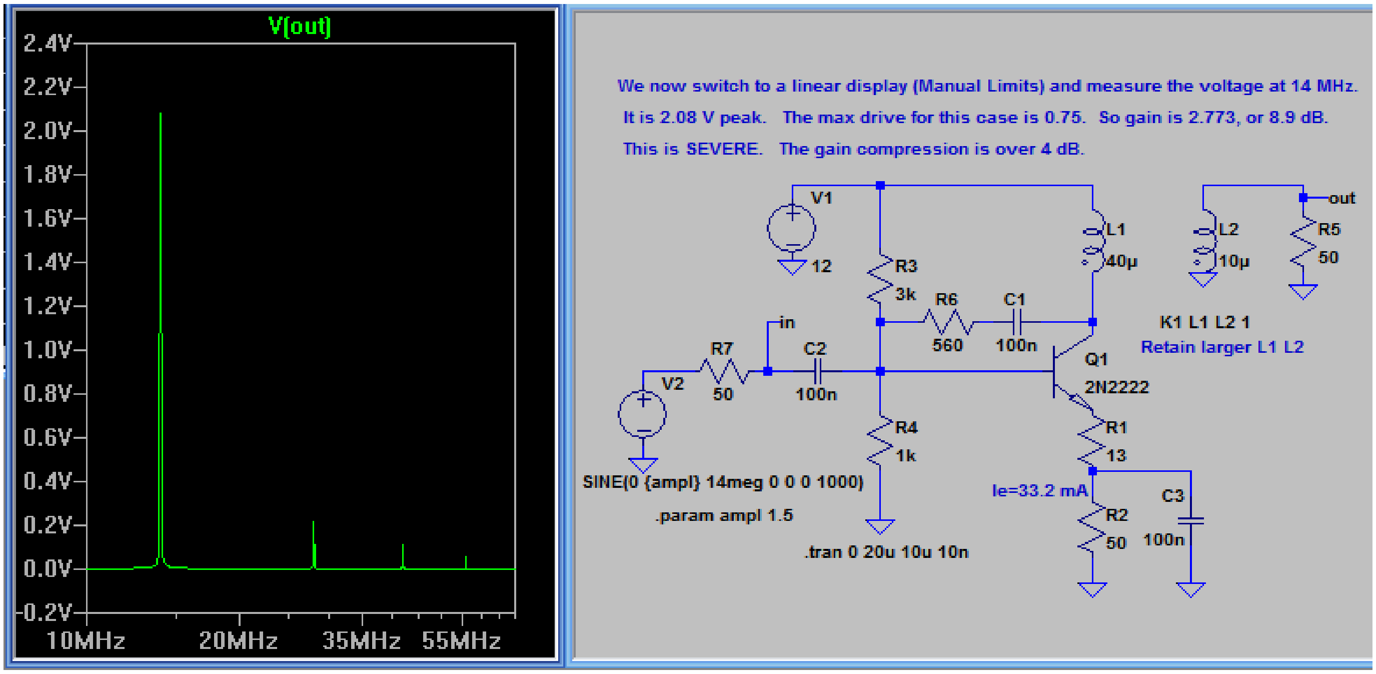
The TRAN run is now to 20 uS, but we only use the last 10 uS of data.

Take the full 10 uS of data, and perform a FFT on it to get the dB plot shown.

The cursor tells us the peak is 6.3 dB (14 MHz) and the 2nd harmonic is at -13.1 dB.

So, the 2nd harmonic is at -19.4 dBc.

```
K1 L1 L2 1
Retain larger L1 L2
SINE(0 {ampl} 14meg 0 0 0 1000)
.param ampl 1.5
Ie=33.2 mA
.tran 0 20u 10u 10n
```

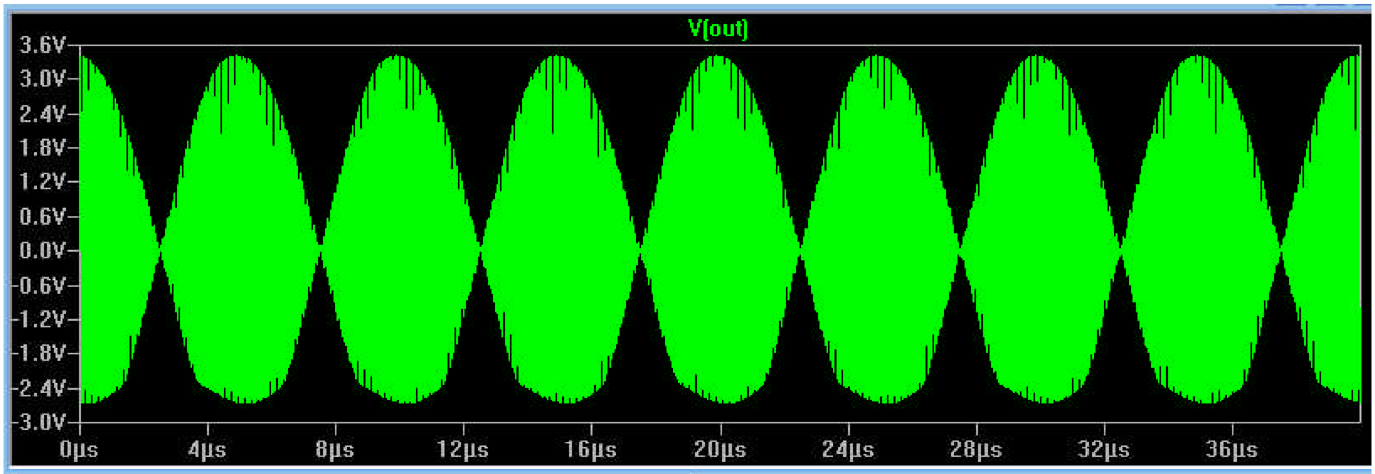


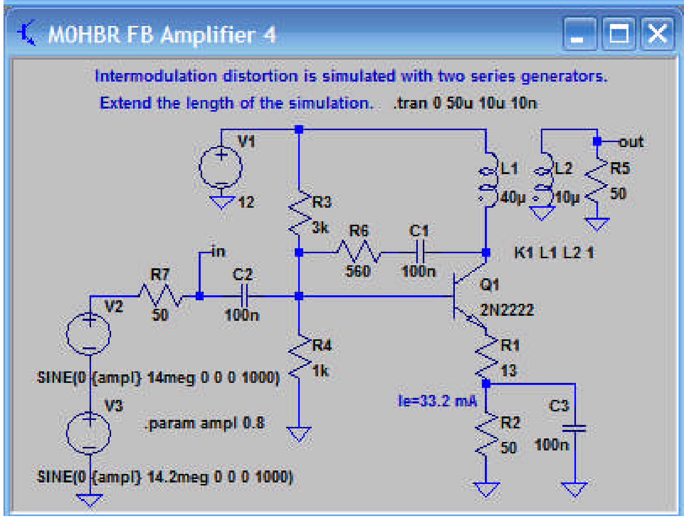
We now switch to a linear display (Manual Limits) and measure the voltage at 14 MHz.

It is 2.08 V peak. The max drive for this case is 0.75. So gain is 2.773, or 8.9 dB.

This is SEVERE. The gain compression is over 4 dB.

```
K1 L1 L2 1
Retain larger L1 L2
SINE(0 {ampl} 14meg 0 0 0 1000)
.param ampl 1.5
Ie=33.2 mA
.tran 0 20u 10u 10n
```





Intermodulation distortion is simulated with two series generators.

Extend the length of the simulation. .tran 0 50u 10u 10n

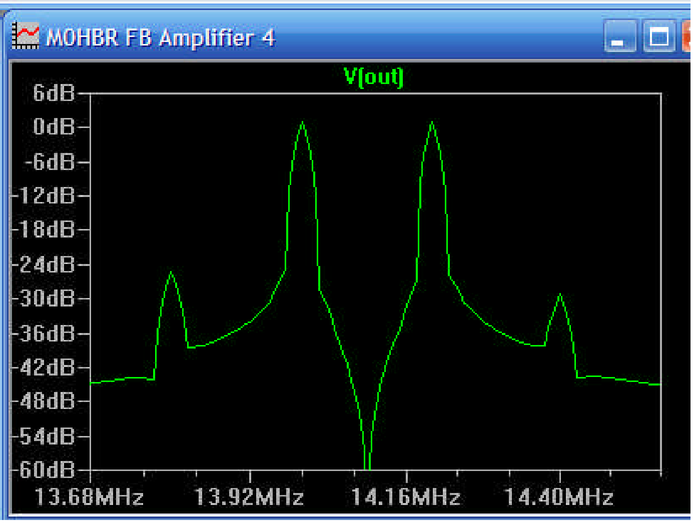
SINE(0 {ampl} 14meg 0 0 0 1000)

.param ampl 0.8

SINE(0 {ampl} 14.2meg 0 0 0 1000)

K1 L1 L2 1

Ie=33.2 mA



The spectrum plot shown above, lower right, was converted from a dB form to a linear presentation. We then put the cursor on the left major peak to read the voltage as 1.13 Volts. This is a peak value. Power is easily calculated, allowing the output intercept, OIP3, to be evaluated.

$$P = \frac{V_p^{\,2}}{2 \cdot R_{Load}} = \frac{1.13^2}{100} = 0.0128 \quad \text{Watts, or } +11.1\text{ dBm}$$

$$OIP_3 = P_{out} + \frac{IMDR}{2}$$

$$OIP_3 = 11.1 + \frac{26.5}{2} = 24.3 \quad \text{dBm}$$