Norton Amplifier (Common Base Transformer Feedback) Simulations in Keysight ADS

James Fletcher, *G8YYH*

Introduction

Common Base Transformer Feedback Amplifiers (CBTFs), also known as Norton Amplifiers, are a popular amplifier topology for use as high linearity, low noise amplifiers, particularly for the HF bands. There has been some success implementing this topology for higher frequencies, such as Chris Haji-Michael's use of transformer feedback in a MMIC LNA (http://www.sunshadow.co.uk/LFA.pdf)

Through the use of an amplifier with negative feedback, we can improve the noise figure and linearity of an amplifier.

The seminal patent by Norton on transformer feedback amplifiers describes a method of adding non-resistive feedback to produce an amplifier that has lossless feedback and low noise figure.

If we add resistive feedback, we will increase the noise figure of the amplifier due to noise in the resistor being added.

The primary goal of this report is to introduce why Noise Figure, IP2 and IP3 are important parameters for any RF amplifier design, and why we care about these figures of merit in the context of receiver design. Then, we investigate the basic iteration of the Common Base Amplifier, how we might be able to improve it through impedance matching and other techniques. We then investigate the performance of the lossless transformer feedback amplifier in a single ended and push-pull configuration. Further discussions on optimising the source impedance of the amplifier for Noise Figure, and stability are included.

RF Figures of Merit: Noise Figure, IP2, IP3

To quantify how much extra noise is generated by a device in our RF chain, we can do this with a figure of merit known as Noise Figure. In any device we add to our hypothetical receiver, we want to have as low as possible additional noise contribution. This is because any additional noise generated in addition to thermal noise (-174dBm/Hz), has the potential to swamp any low level signal we might want to receive that enters through our antenna.

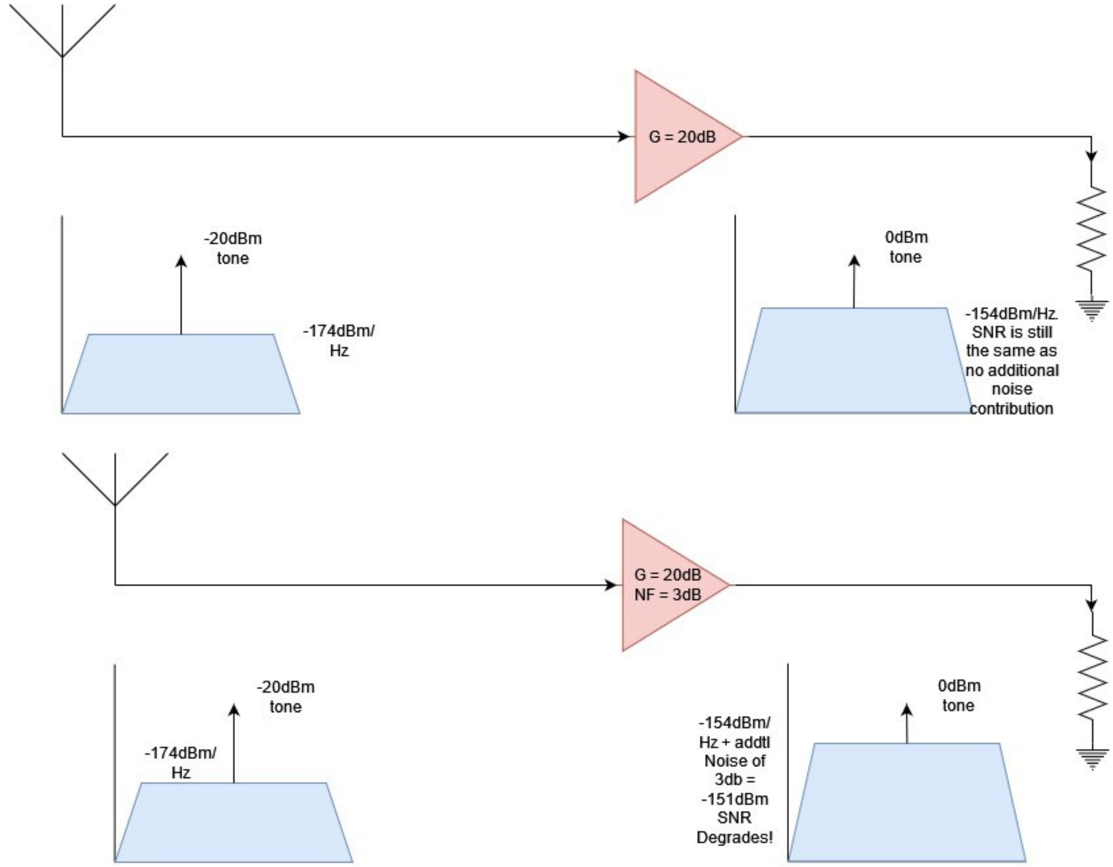


Figure 1: A Visual Representation of a Perfect Vs. Non-Ideal Amplifier in the Context of Noise Figure

Let us imagine a "perfect" amplifier with an input signal of -20dBm. We will assume the noise floor is at thermal noise, -174dBm/Hz. If the amplifier has a gain of 20dB, our input signal will rise to 0dBm at the output of the amplifier, but so will our noise floor, to -154dBm/Hz. The noise must get amplified too. No amplifier is perfect though, and we will have some additional noise generated internally by the amplifier due to the mechanism of flow of electrons through the semiconductor junctions (There's various types, such as shot noise, flicker noise etc.). Our noise floor will be therefore raised by the gain plus the additional noise generation of our real world amplifier.

We can therefore define noise figure as:

$$NF = Output\ Noise\ Floor\,(dBm/Hz) - Gain\,(dB) + (-174\,dBm)$$

This equation effectively measures the additional noise generated by the device by subtracting away the gain of the device itself.

Noise figure can also be thought as a degradation of SNR, or the ratio of output to input SNR. Which in any real case will always degrade unless the NF of a component is 0dB.

Due to the Friis formula for cascaded amplifiers and their total noise figure, the first component in a receive chain mostly determines the noise figure for the overall receiver. Ideally we want as low noise figure as possible, followed by high gain stages to reduce the system NF.

$$Cascaded\ NF\ via\ Friis\ Equation = F1 + F2 - 1/(G1) + F3 - 1/(G1G2) + \ldots$$

Intermodulation Distortion (IP2 and IP3)

IP2 and IP3 characterise the non-linearities in any RF device. Because we assume non-linear devices follow a power series of an nth order polynomial of form $f(x)=a+bx+cx^2+dx^3+...$, any two tones at the input to a non-linear device will interact non-linearly. If we input $x=\sin(wf_1t)+\sin(wf_2t)$ into this equation,the expansion of the two sinusoidal terms will give rise to terms that have the frequencies of 2f1-f2, 2f2-f1 (expansion of x^3), and f1+f2, f1-f2 (expansion of x^2). These terms appear as spurious tones at the output of the amplifier.

More needed here on IP2 and IP3.

What parameters define a good low noise BJT?

We need to decide what BJT we want to use for our common base application, seeing as Norton Amplifiers are used for low Noise Figure applications, we want a BJT that is going to contribute as little noise as possible to the circuit if we want to use it as an amplifier in an RF frontend. From my asking on the Loop Atennas Groups.io forum, it seems there's a number of parameters that determine the noise performance of a BJT:
https://groups.io/g/loopantennas/topic/2sc5551_and_bfu590g/104249915?p=,,,20,0,0,0::recentpostdate/sticky,,,20,2,0,104249915,previd%3D9223372036854775807,nextid%3D1705663079786417902&previd=9223372036854775807&nextid=1705663079786417902

In the two presentations linked, there's a lot of maths to unpack. I'll write about this another time probably once I have some idea what on earth that's all about…

"For example, when degenerative feedback is applied to an amplifier, its
noise figure can be reduced to as close to unity as desired (for example,
bypassing the entire amplifier with short circuits yields unit noise figure).
But its gain is also reduced in the process." from Circuit theory of linear noisy networks

Ic is a major factor in noise of CB amps. Shown later.

It seems most good low noise devices have been found through experimentation and measurement, some transistor manufacturers do give noise parameters in their datasheets though.

More to elaborate on here really.

Resources on Common Base Amplifiers

This investigation will be based on the circuits provided in the paper on Norton Amplifiers by Dallas Lankford, found here: http://www.thegleam.com/ke5fx/norton/lankford.pdf

Simulations

Keysight ADS is a circuit simulation tool similar to LTSpice and others on the market but intended primarily for the research and commercial engineering user base. With this comes advanced simulation tools that are not available with tools like LTSpice. This includes the ability to simulate what is known as “Harmonic Balance” simulations. These are simulations that are able to simulate the performance of a circuit in the time domain, while analysing the performance of harmonics generated in non-linear parts of the circuit. This enables us to simulate the IMD performance of an amplifier by using a two-tone source. Keysight ADS also has the ability to simulate S-Parameters and Noise Figure.

For the investigations that follow, a BFU590G was chosen as the transistor of choice to investigate the varieties of common base amplifier. The 2SC5551 is a well known BJT that provides low noise figure, but has since gone obsolete. The BFU590G was recommended in a Groups.io post in the loop amplifier group. https://groups.io/g/loopantennas/topic/use_of_2sc5551_in_other/84252803?p=

Luckily NXP (Suppliers of the BFU590G) offer a collection of models of their whole BFU series for use in Keysight ADS.

Initially, a common base amplifier using a BFU590G was simulated in ADS to approximate the gain and IMD performance to set a baseline to compare against the different flavours of common base feedback amplifiers. For now, we will only simulate at a single tone power of 10MHz. This is done by using a 1 Tone source, with a source impedance of 50 Ohms, and an input power of -50dBm. The biasing network for the BFU590G is based off the circuit diagram in the Lankford paper (here). For now, we will use ideal “DC Block” and “DC feed” components provided by Keysight ADS for our biasing network, these are effectively perfect inductors and capcitors with no frequency dependence. Current probes are set up so we can plot Ie,Ib and Ic once the simulation is running. Nets Vin and Vout are used to plot the input and output voltages.

Common Base Amplifier Simulations

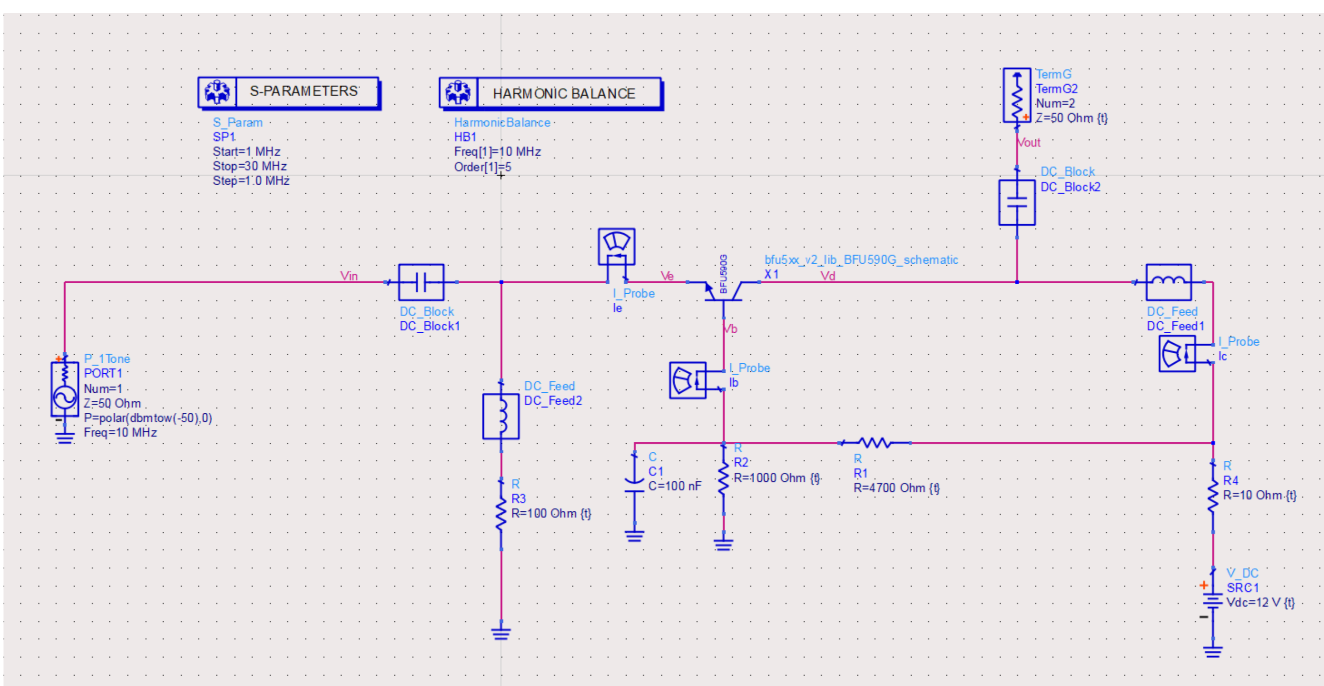


Figure 2 ADS Schematic for Common Base Amplifier

We can plot the gain (S_{21}), input match (S_{11}) and time domain plots of the input and output voltages of this amplifier using ADS. We will only look at a single tone of 10MHz in the time domain, and will perform an S-Parameter sweep from 1 to 30MHz.

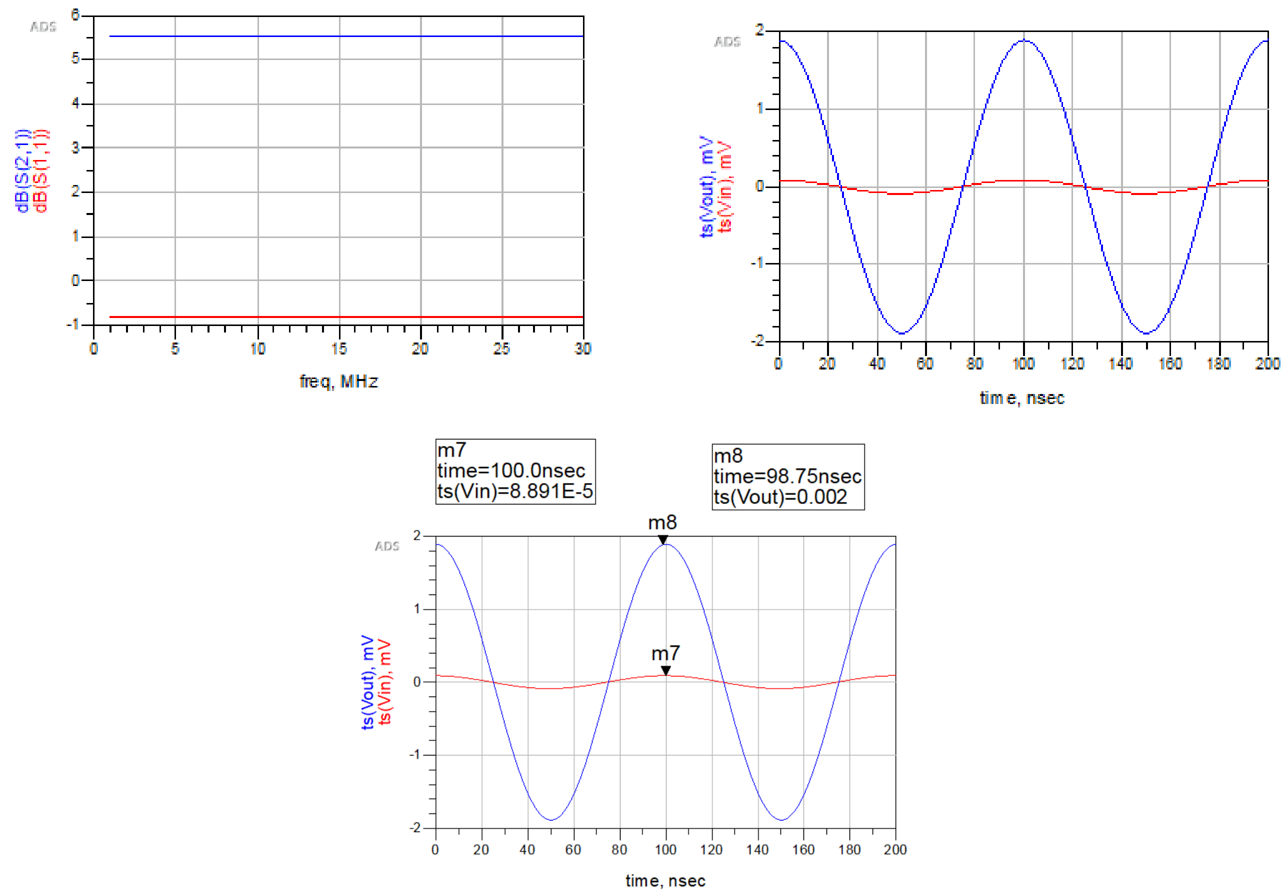


Figure 3 S21, S11 and time domain voltages for the initial common base amplifier

As can be seen above, while the gain is acceptable, the S_{11} is really poor. Less power therefore is being delivered into the emitter of the transistor, as nearly 80% of the power is reflected back to the source. The gain is still being measured relative to the source power though. I.e the incident power coming out of port 1.

So even though our net input power is lower than the actual -50dBm we want, our gain is being measured relative to this initial -50dBm input power from port 1. What we are actually measuring with the S_{21} is the *transducer* power gain, i.e the output power delivered to the load referenced to how much power is *available* from the source, rather than the gain referenced to the power actually incident at the input of the amplifier (By input I mean at the emitter of the BJT, so after the AC coupling capacitor and DC bias network of the DC feed and R3). This means that our power gain relative to the input of the emitter is actually *higher* than we see from the S-parameters, if you measure the power at the emitter, it's about -71dBm, the output power is -47dBm. So if we can match our input power to 50 Ohm, therefore delivering more of our available input power into the emitter we might be able to obtain better power transfer with regards to the amount of power actually delivered from the source to the emitter. Note that *transducer power gain* is equal to the S_{21} of an amplifier only if ZL and ZS are terminated in the reference impedance Z0.

I had some confusion with regards to power vs voltage gain that stumped me, and it's probably worth me writing here how I understand it. There is a term that crops up in RF design textbooks known as *transducer power gain* that we've just discussed. S-Parameters are defined in terms of forward and reflected voltage waves at an n-port network. What confused me was that textbooks define the *transducer power gain* as $(|S_{21}|)^2$, and I had never calculated gain this way, and always took whatever S_{21} I measured to be the gain of the amplifier, so I was very confused that all of a sudden I was reading that the power gain of an amplifier was defined differently. The reason behind the confusion is as follows:

The S_{21} is defined as the ratio of the incident voltage to the terminated load at the output of port 2 to the incident voltage wave at port 1. If we want to find the power ratio of these two voltages in linear form, we need to find the absolute value of these complex waves, square them, and calculate the ratio. In RF textbooks, S-Parameters are defined as linear voltage ratios (I.e without taking the logarithm). Due to the definition of voltage gains in dB being $20\log_{10}(\frac{V_1}{V_2})$ and power gains being $10\log_{10}(\frac{P_1}{P_2})$, you can either say the gain of a device in dB in terms of S-Parameters is $20\log_{10}(\frac{V_1}{V_2})$ or $10\log_{10}(\frac{V_1^2}{V_2^2})$. This is because we decibel is strictly a *power ratio*. Therefore, the 20Log term for voltages comes from the fact you are actually taking the 10log of the voltage ratio squared. For correctness, we should always talk about S-Parameters in terms of voltage waves, so when we convert S-Parameters from linear to log, we are taking $10\log_{10}(\frac{V_x^2}{V_y^2})$. Where $V_x$and V_y are the relevant forward/reflected voltages waves for a defined S-Parameter. It's best to be explicit about this otherwise people like me get confused!

https://blog.teledynelecroy.com/2014/12/what-s-parameters-reveal-about_9.html
The bottom of the above linked article explains this better than I do. 1st chapter of Microwave Component Measurements by Dusnmore also explains this well.

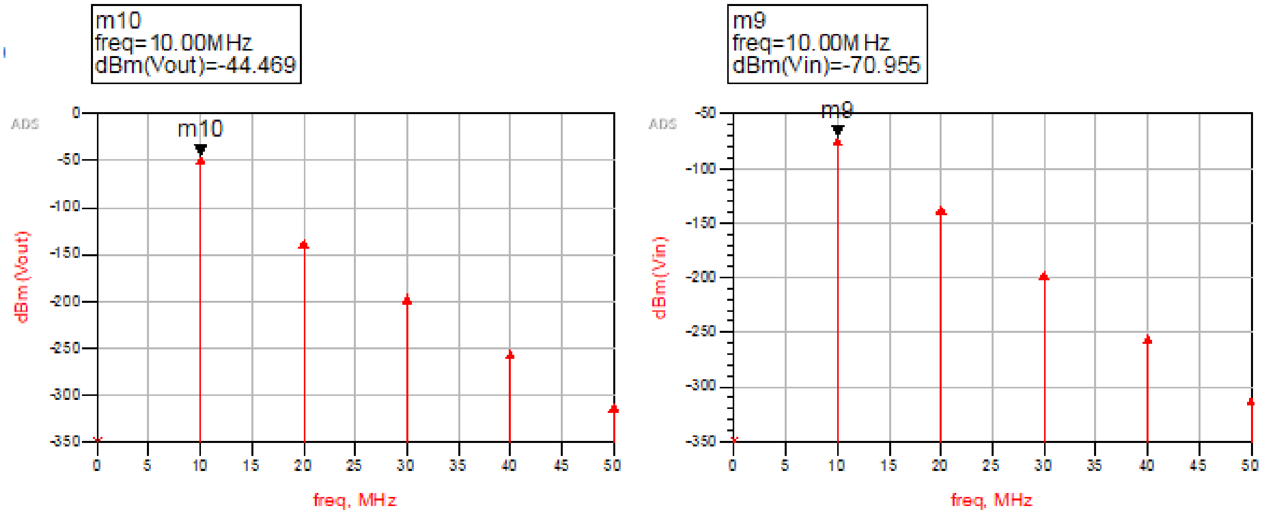


Figure 4: Input Vs Output Power. R3 = 100Ohm, Port 2 Impedance = 50 Ohms.

Looking at the Smith Chart plot of the input reflection coefficient, we are looking into nearly a short circuit. This has been calculated to be an input impedance of 3 Ohms using the Zin function.

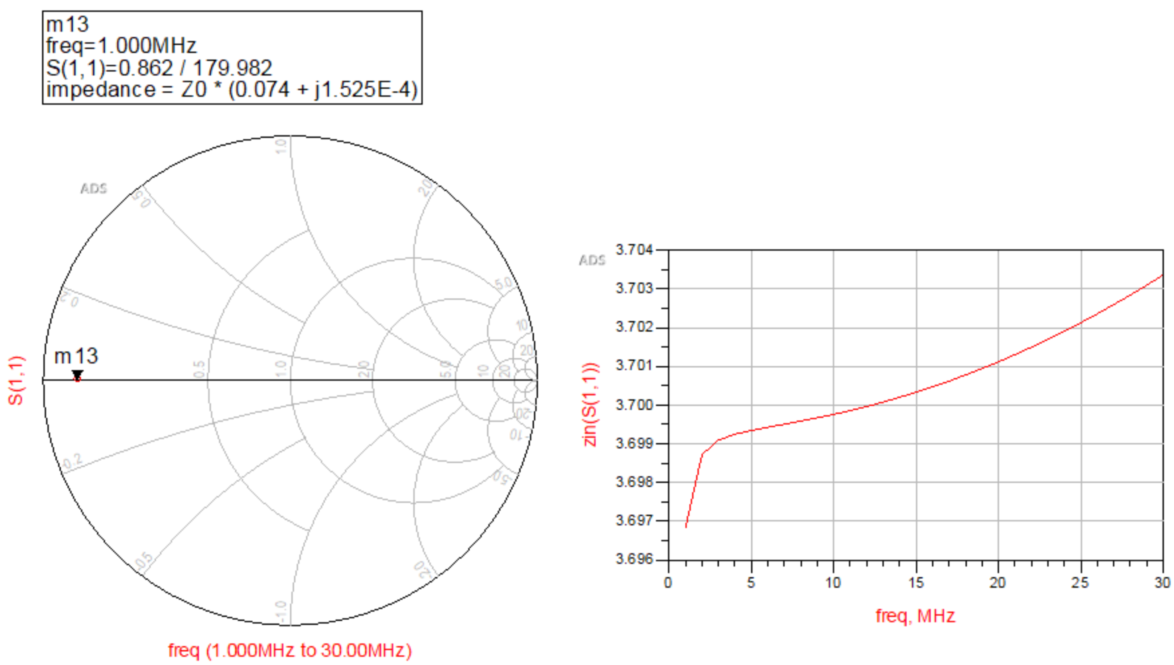


Figure 5 S11 of the Common Base Amplifier with R3 = 100 Ohm

ADS offers a tuning feature. This allows us to tune each selected component value while the simulation changes in real time. This is a handy way to either fine tune and optimise a design, or to change one parameter and see it's effect on the whole circuit.

There is a trap that you should be careful not to fall into with using the tuning interface, which I've fallen into myself. Don't mindlessly tune components hoping to at some point to find an optimum for whatever parameter you want to perform best. Take a step back, think about the effect of tuning that resistor, capacitor, whatever is going to have on your circuit. Then tune the component. There is nothing to be learned from mindlessly tuning components. This is part of the reason behind writing this report, it has allowed me to write down what's going on and think about what I'm changing, and the consequences of changing certain parameters.

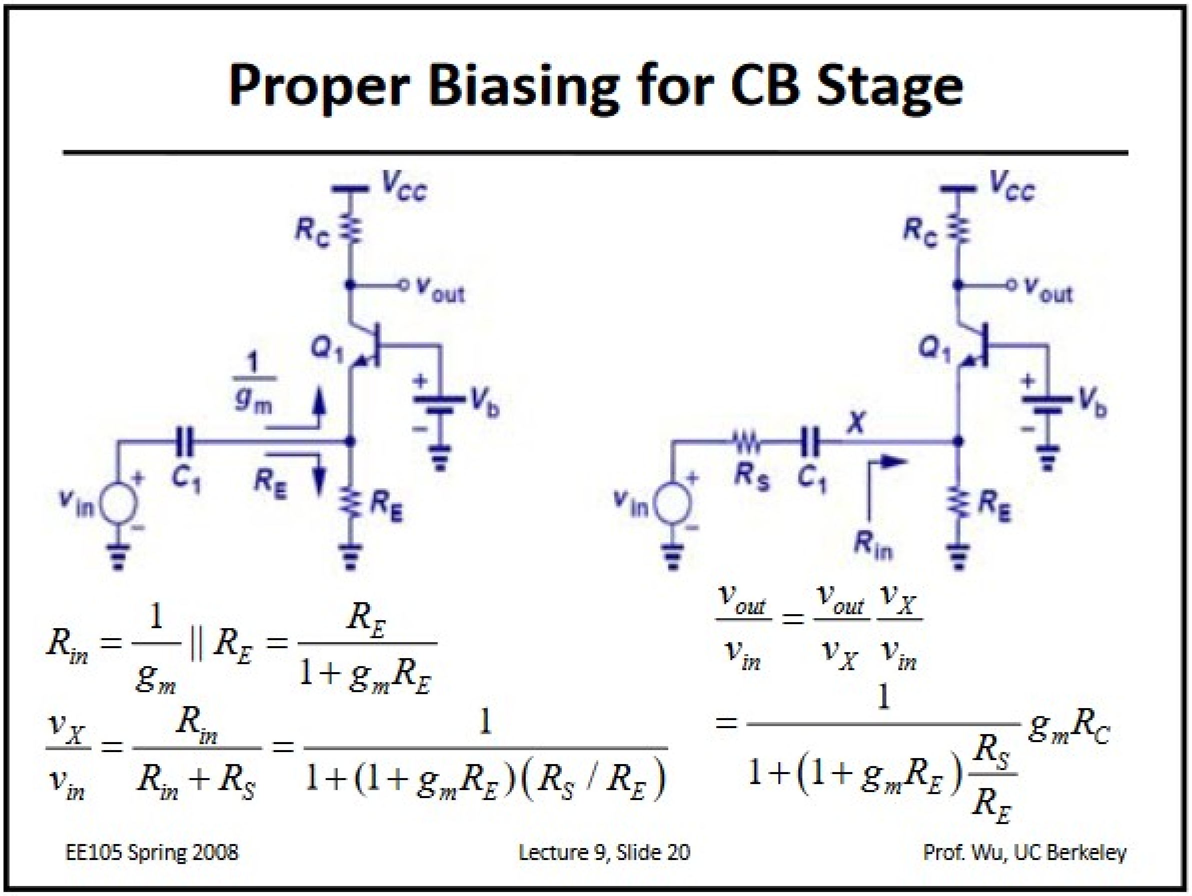


Figure 6 Biasing for a CB amplifier and how the Input Impedance is determined (gm is usually in the order of mS so gm dominates)
https://inst.eecs.berkeley.edu/~ee105/sp08/lectures/lecture9_6.pdf

The above picture shows how we have biased our common base amplifier. Of note here is the equation for our input impedance, *Rin*. Because we need the external emitter resistor to ground for biasing the transistor properly, this resistor Re is in parallel with the emitter resistance of the BJT, which is *re (or* $1/g_m$*)*. g_m is determined by the equation:

$$g_m = \frac{I_c}{V_t}$$

V_t is the thermal voltage of the BJT and taken to be ~ 26mV. I_c is biased around the 24mA range when R3 = 100 Ohms, giving a g_m of 1 Siemens, or a $\frac{1}{g_m}$ of 1 Ohm. When talking about Re in this report, I will use Re and R3 interchangably, as I have set this to be R3 in ADS.

g_m therefore dominates this parallel combination, thus lowering the overall input impedance. We will show how the input impedance changes by using the tuning interface to see how the S11 changes as we change R3.

Below in Fig.6 is the best compromise between gain and input impedance I can produce. This was done by increasing Re, because our original input impedance was very low. We needed to increase Re to bring us closer to a 50 Ohm input impedance. This was found at an Re of 1400Ohm. By doing this, we are decreasing Ic (so increasing $\frac{1}{g_m}$). If I_c decreases, V_b must decrease (because $I_b = \beta I_c$), therefore if V_{be} is a fixed 0.7V (with some variation over temperature) then V_e must drop by the same amount. A better way to think about this is with the following equation : $I_c \approx \frac{Vb-0.7}{R_e}$ therefore, if R_e decreases, so must I_c.

While increasing R_e, we are effectively increasing the total parallel impedance seen by the source, because $\frac{1}{g_m}$ is typically very small, R_e needs to be increased quite a bit to increase the parallel impedance. By improving the match, more of the RF power available to us is now being delivered to the emitter of the BJT.

But by increasing R3, we are decreasing the total R_e. So while our input match has improved, the gain has dropped, because the $\frac{R_e}{R_c}$ ratio has decreased. If we are calculating gain relative to our -50dBm, we don't care so much about S_{11} in this specific case, because the gain is primarily determined by the ratio $\frac{R_c}{R_e}$. So there's not much we can do to improve the input match and the gain in this specific topology as both of these parameters are linked to each other, and it's in our best interest to have as low input match here as possible, to maximise the ratio. The S_{11} does have an effect on stability and optimum Noise Figure, this is investigated later.

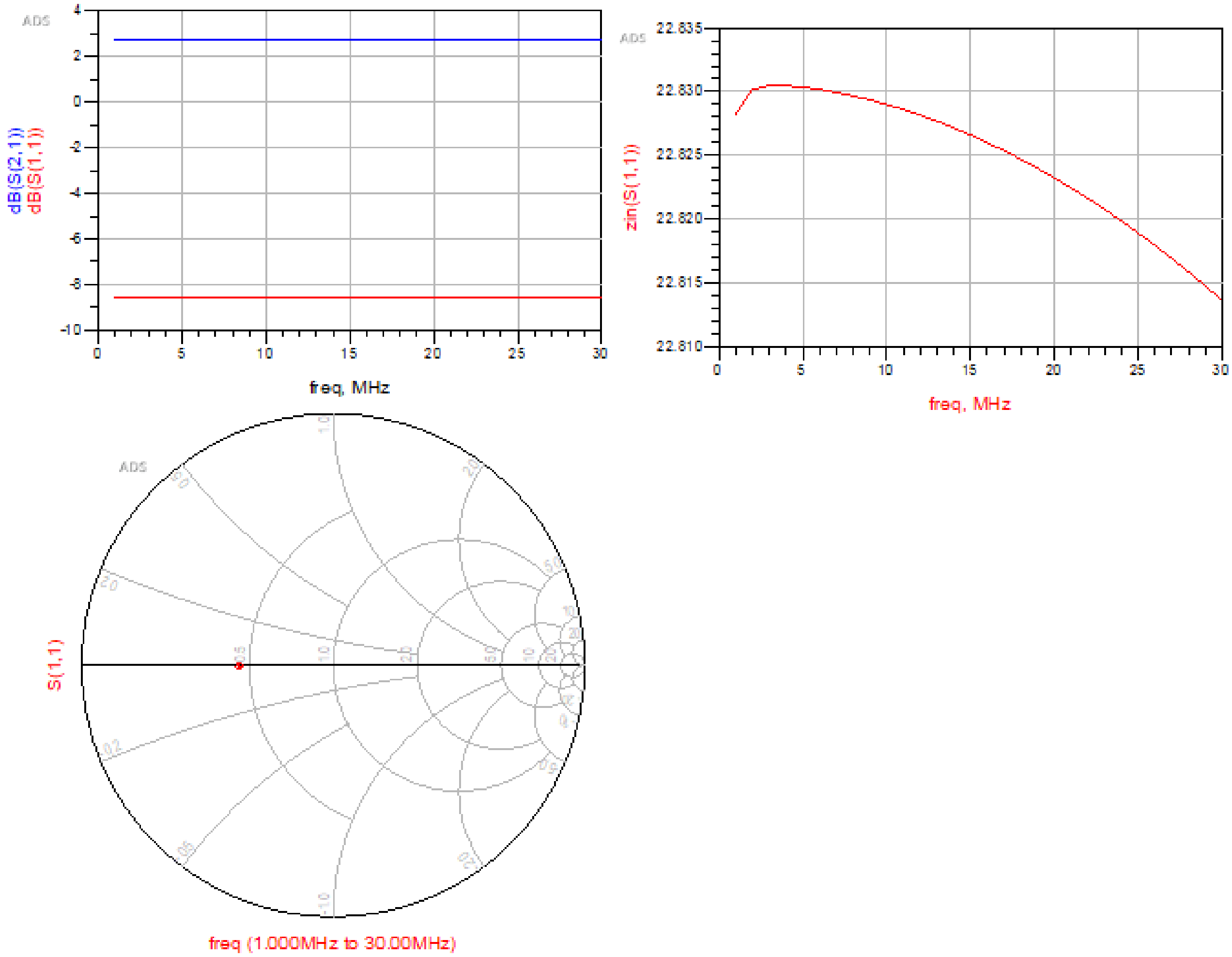


Figure 7 Best compromise between input match and S21 I could get. Note, we've improved the S11 but the gain has dropped.

We can also plot the Noise Figure, and IP2 and IP3 of this amplifier. To do this, we change our Harmonic Balance simulation to now simulate a two tone 50 Ohm impedance source, and add in simulation commands to find IP2, IP3 and noise figure of this amplifier. Note that all IP2 and IP3 measurements in this report are *Output Referred,* i.e. these are the IP2 and IP3 measurement points at the output of the amplifier. If we know the gain, we can simply subtract the gain to find IIP2 and IIP3. I have arbitrarily chosen 10 and 12MHz as my two input tone frequencies. A better choice for future testing may be two tones that are more representative of what may be seen for a receiver being used in the HF band. For example, two tones spaced where you may expect a weak signal you want to receive, on 40 meters for example (10MHz), and another strong in band tone. You are unlikely to see a strong 12MHz signal, due to the preselection in the RF front end rejecting anything out of band. With the nominal biasing values from the Lankford paper, we get the following results, shown in Fig.9.

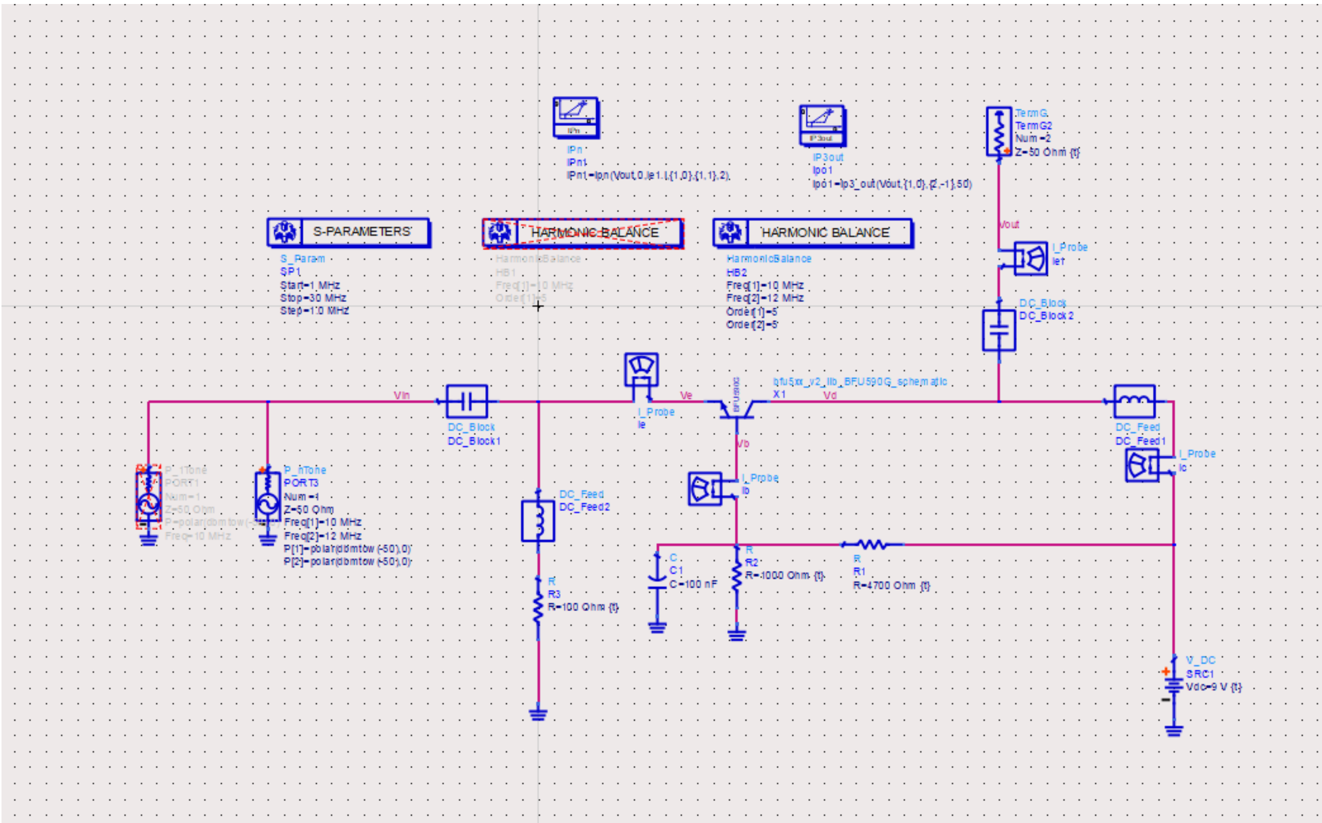
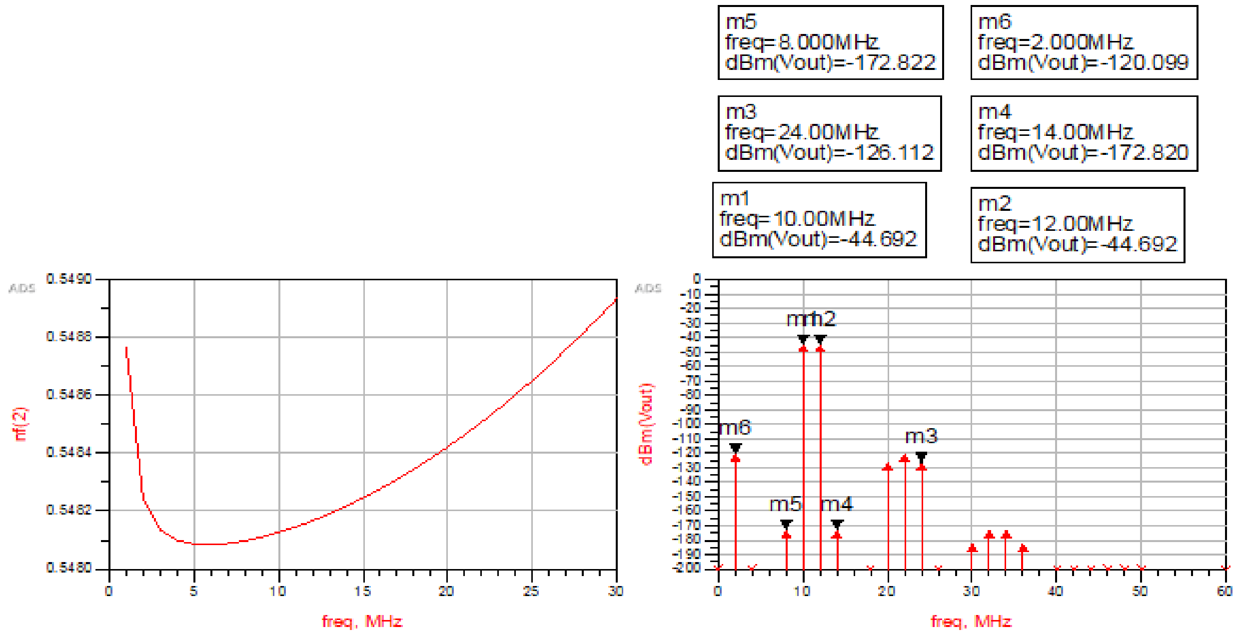


Figure 8 ADS Schematic for CB amp with 2-Tone harmonic balance added. Tones at 10 and 12MHz



freq	IPn1	ipo1
<invalid>Hz	30.708	19.372

Figure 9 IP2, IP3, Noise Figure and output spectrum in dBm for CB amp, R3 = 100 Ohms.

IP2 and IP3 are respectable, and the Noise Figure is very impressive (A task for another day, but it would be worth me comparing some different transistors and the Noise Figure obtained with the same bias). I found by decreasing R3, I was able to improve linearity in terms of the IMD products. I think this is because we are operating further away from the saturation region maybe, as Ic increases, Ve must be higher. I was also able to confirm Dallas Lankford's findings that linearity worsens as you decrease Ic (By increasing R3), while Noise Figure improved. So that means there is a compromise you have to make between input match and linearity. Parameter "Ipo" is the IP3 measurement, "IPn1" is the IP2 measurement parameter. Our baseline to compare all the other topologies against is now the following (with R3 set to 100 Ohms):

Figure of Merit	**Value**
IP2	31dBm
IP3	19dBm
Noise Figure	0.55dB
Gain	5.5dB

Is there a way to optimise the gain further? Well one idea is to make the collector “see” a higher impedance. As we saw earlier, if the load impedance increases relative to the input impedance, Re, so does the gain of the common base amplifier. We can make the load impedance at the collector “look” higher by adding in an impedance transformer. This is shown below. I’ve set this up with 2 mutually coupled inductors, with the useful ADS tuning interface I can quickly change this impedance ratio and see the effect on gain and other parameters. The red boxes around component values show which values I’ve selected to be able to tune.

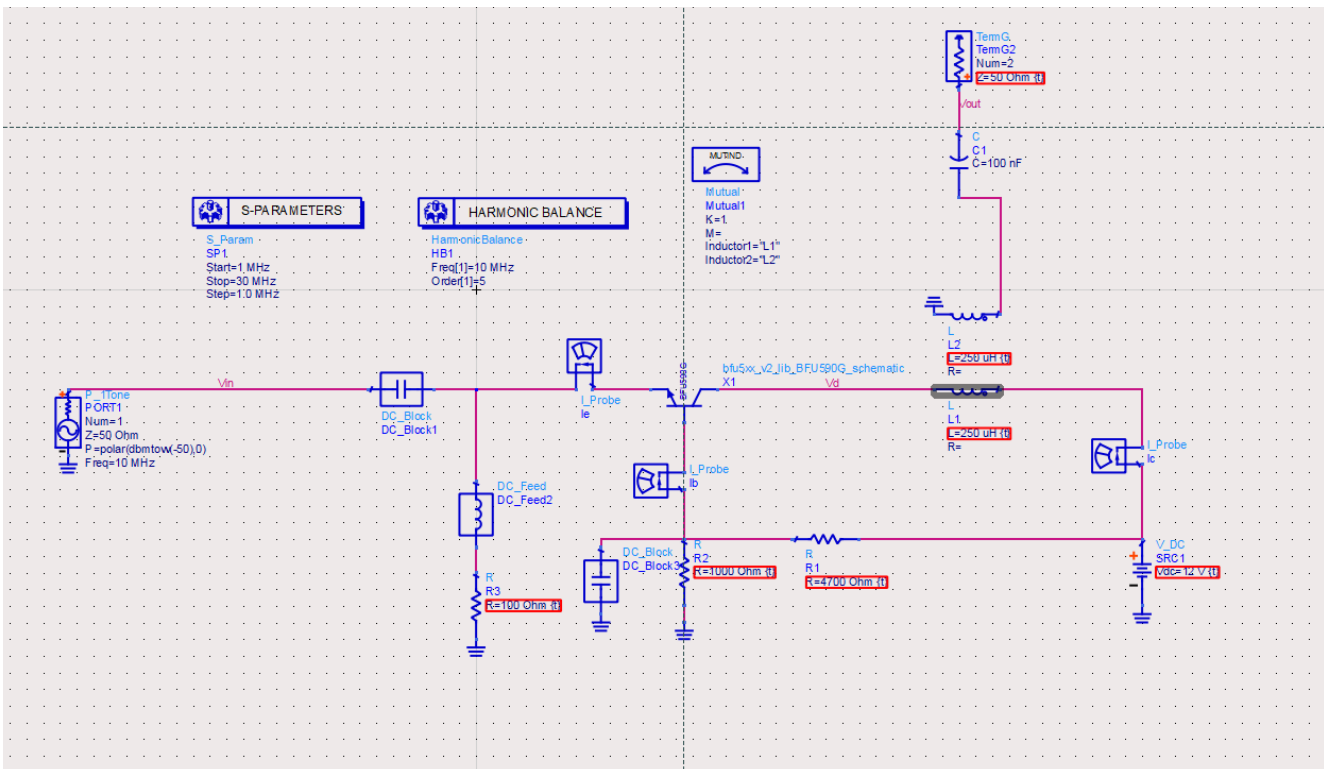


Figure 10: ADS Schematic for CB amp with impedance transformer through mutually coupled transformer windings

With an Equal impedance transformation ratio of 1:1, we get the following results:

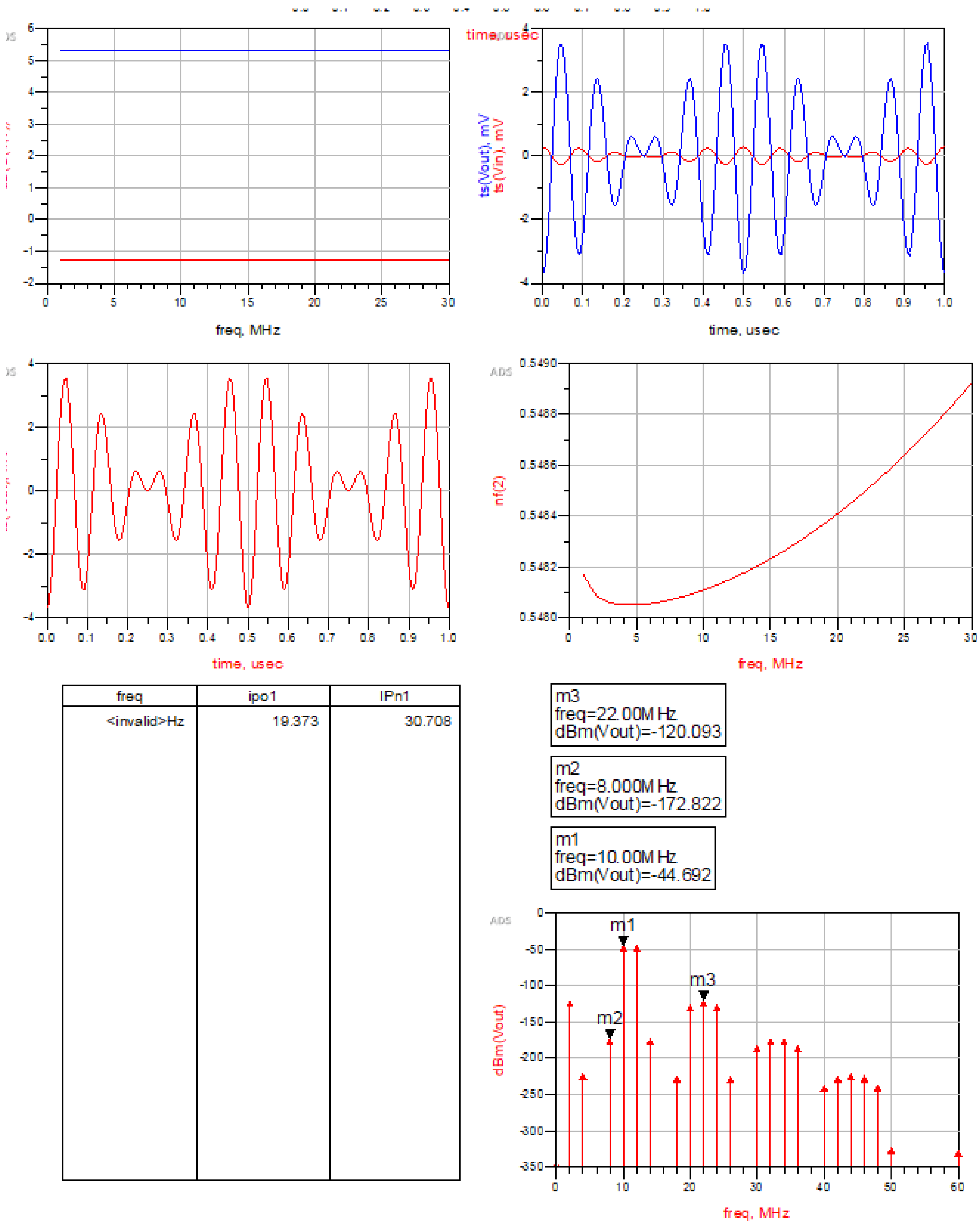


Figure 11: RF Figures of merit for a 1:1 impedance transformation (plus some time domain plots of input vs output voltage and output voltage)

With the equal impedance ratio, all the figures of merit are the very close to the same as before.

With a really high impedance ratio, we get the following results:

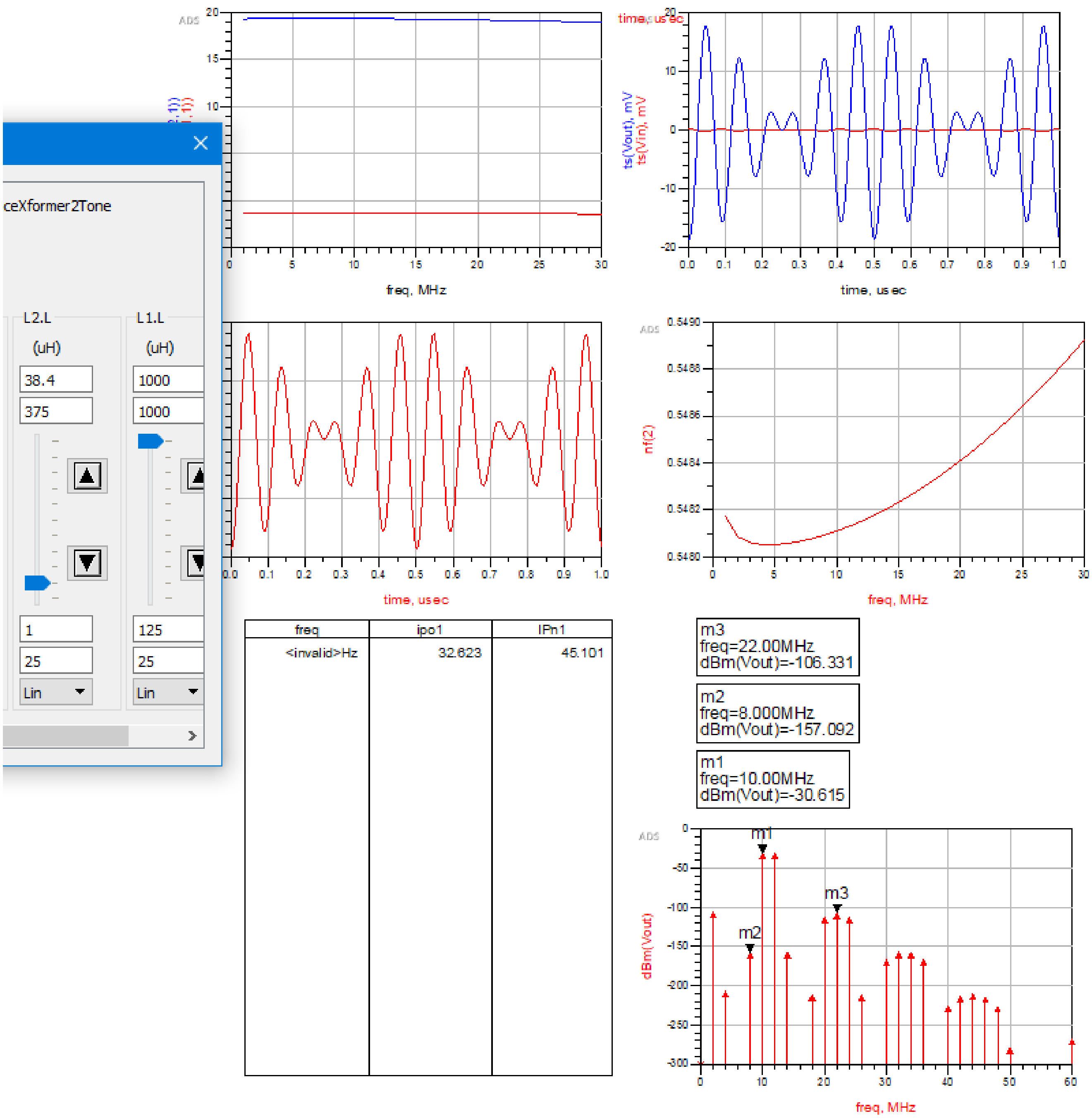


Figure 12 RF Figures of Merit for a collector to load impedance ratio of 1000/38.4 = 26.

Tuning parameters for L2 and L1 are attached in Fig 12. IP2 and IP3 actually improves quite a bit. Noise figure stays about the same and gain is huge at nearly 20dB. If we assume, like Lankford's paper, that we use Amidon FT-50-75 to wind this impedance transformer, this gives an Al of 3100. This equates to winding about 6 turns for an inductance of 100uH. The impedance transformation ratio is equal to the ratio of turns on the secondary to primary squared, same with the inductance ratio. So a 1:3 inductance ratio would give us 3x the output impedance (looking into 50Ohm). We've transformed the output impedance from 50 to 150 Ohms at the collector. I went back to the basic common base amp and tuned the 50 Ohm output impedance to 150Ohm, and we get similar results, but about a 1dB increase in gain in comparison, probably due to the fact that no voltage is being dropped at the collector due to the inductive reactance. One thing to note is that increasing this impedance ratio causes IP2 and IP3 to improve. A ratio of 1000uH to 36uH is unlikely to be able to be wound on an Amidon FT-50-75 core, as 1000uH needs 60 turns.

With a more realistic impedance ratio of 5:1 at the collector, we get the results shown in Fig.13. Our gain has dramatically improved, IP2 and IP3 also have improved relative to the common base amplifier, our Noise Figure has degraded by about 0.05dB.

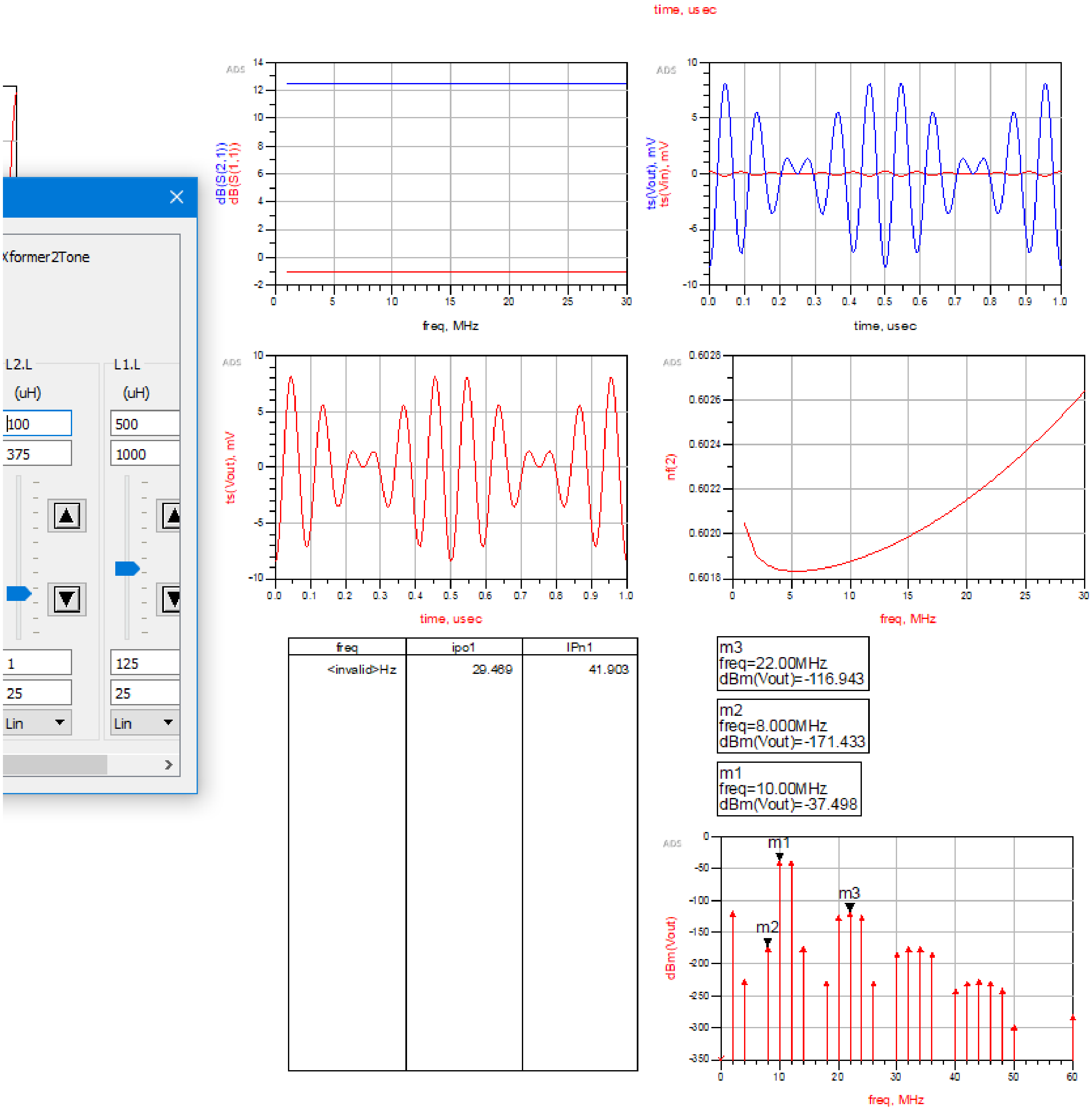


Figure 13: RF Figures of Merit for a More Realistic and Realisable Impedance Ratio of 5

Another way to improve the input match, rather than altering Re, is to use an impedance transformer at the input to improve the S11. One way we can do this is through the use of a 1:4 Ruthroff impedance transformer. This will transform the current very low input impedance we are seeing of about 3 Ohms up to 12 Ohms, which will help improve the S11 of the amplifier circuit. Because we were originally changing R3 to improve the output match, and thus reducing the gain by increasing this, maybe using an impedance matching method will help to increase our gain by preventing a large amount of reflecting away from the emitter of the BJT.

A Ruthroff impedance transformer makes use of a two windings around a ferrite that approximates a transmission line. There's lots of resources online to understand this, including the paper by Ruthroff himself : https://www.qsl.net/kp4md/ruthroff.pdf .

Jerry Sevick's book *Transmission Line Trasnformers* is also a good resource. https://www.okdxf.eu/files/Noble%20Publishing%20-%202001%20-%20Transmission%20Line%20Transformers,%204ed.pdf .

V1 exists across the input to the transmission line transformer. V2 is "bootstrapped" by V1, causing the output voltage to double(https://www.n5dux.com/ham/files/pdf/Transmission%20Line%20Transformer%20Basics.pdf). For conservation of power to occur, Rl must increase by a factor of 4. (P = V^2/R). There's a more rigorous mathematical derivation of this here…/explain this better. Michael steer vol 4. explains this well.

We want to transform our 50 Ohm source impedance down to the lower impedance of Re, so will be using this in a 4:1 configuration. This is shown in the schematic in Fig.15.

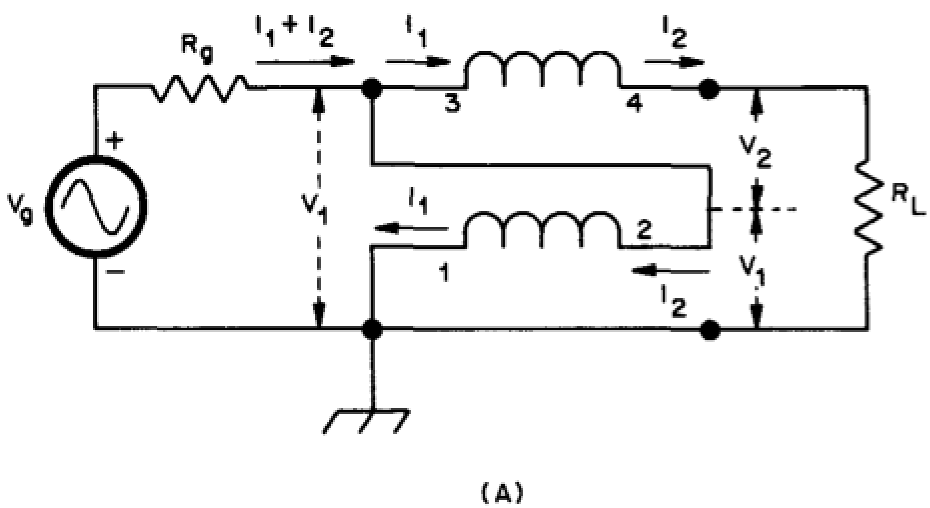


Figure 14 The 1:4 Ruthroff Transformer.

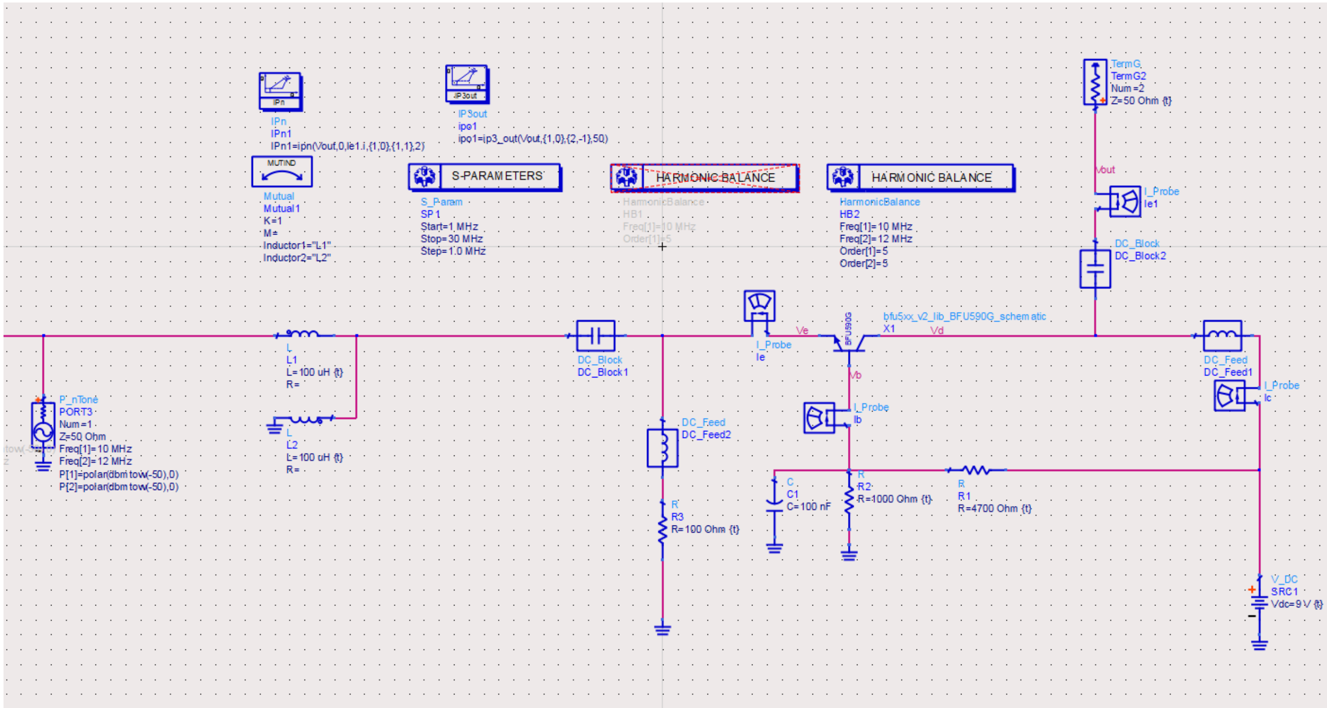
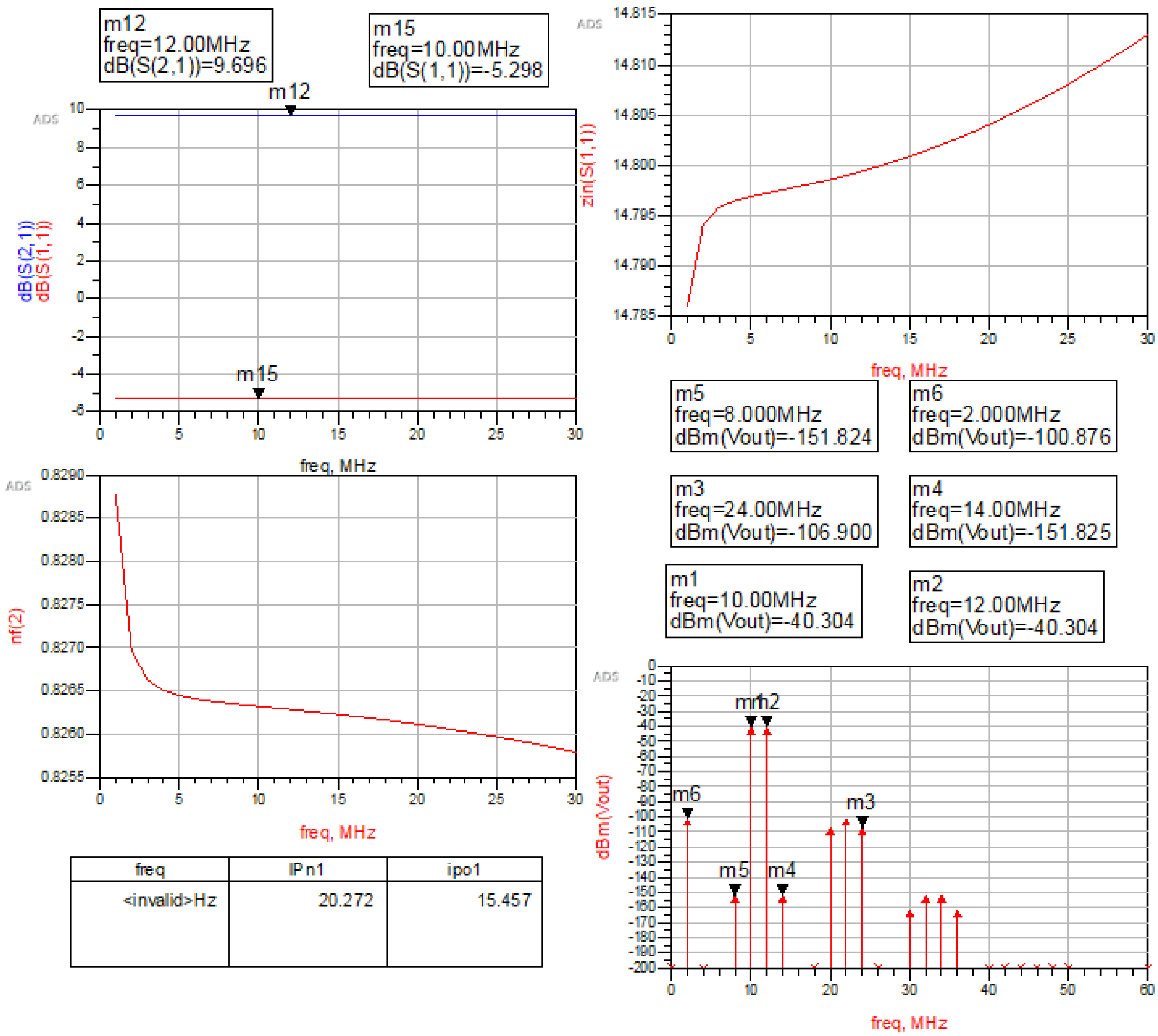


Figure 15: Schematic of the Common Base Amplifier with 4:1 Ruthroff Impedance Transformer

As can be seen in Fig.16, Zin(S(1,1)) has increased to about 14 Ohms, bringing us further away from the short circuit end of the smith chart. As a result, the input match has improved, and less power is being reflected, so the gain has improved by about 3dB in comparison to the initial amplifier setup with R3 = 100 Ohms. Note that we haven't altered the emitter resistance through this method, and therefore this impedance matching network has no effect on the biasing of the circuit and has no impact on the gain. As a result of this gain increase, the IMD performance has decreased. For the first common base amplifier that was simulated, IP2 was ~30dBm and IP3 ~ 20dBm.



freq	IPn1	ipo1
<invalid>Hz	20.272	15.457

Figure 16: Results of using the 4:1 impedance transformer, R3 = 100 Ohm.

You could also use a traditional impedance transformer with two mutually coupled transformer windings to obtain the input impedance match. Ideally, we want to bring up the input impedance even higher compared to the Ruthroff transformer, as we still only had an input impedance of 14 Ohms, so we want a higher impedance ratio than 4:1. Increasing the impedance transformer's ratio further with this topology, I achieved about 2dB more gain out of the Common Base amplifier, shown in Fig. 18, at the expense of Noise Figure and IMD performance. You could now enhance this topology further by using an impedance transformer at the collector like before, to increase the gain. I've done a simulation of this too. Schematic shown in Fig. 19, results in Fig. 20.

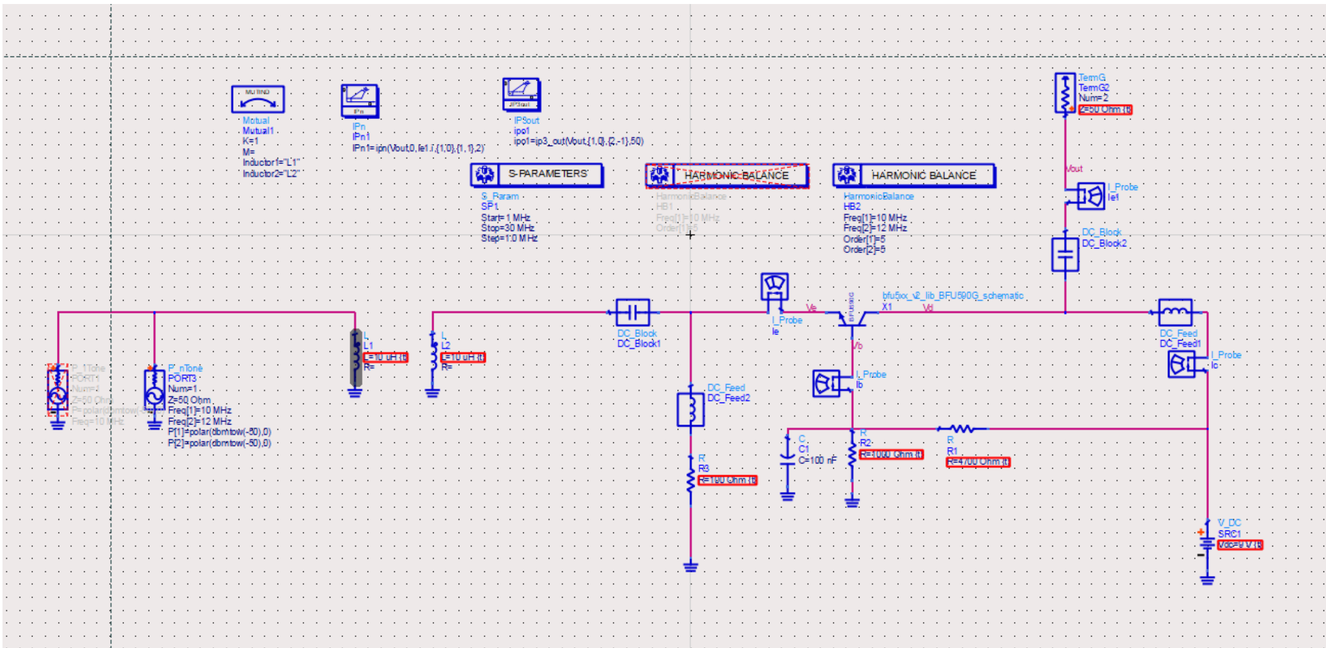
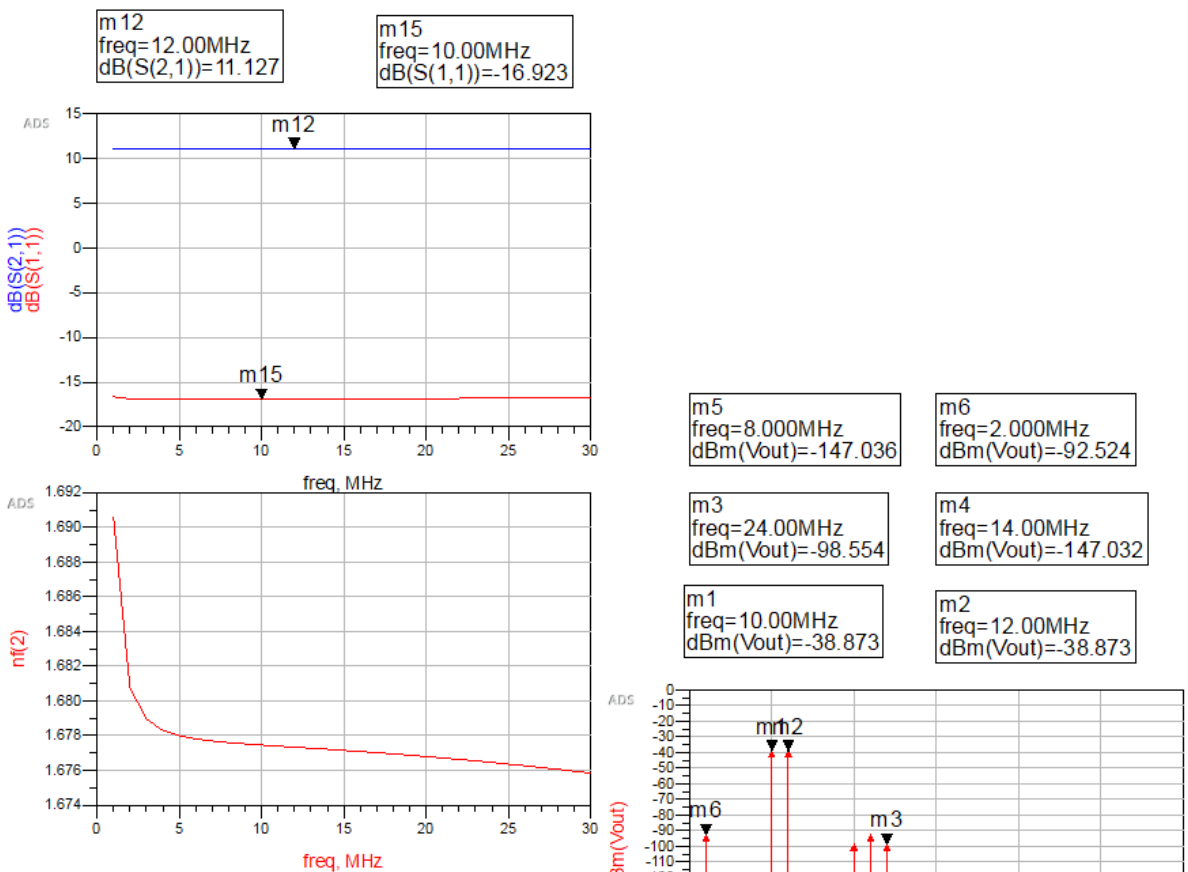


Figure 17: Common Base Amp with Input Impedance Transformer



freq	IPn1	ipo1
<invalid>Hz	14.787	15.208

Figure 18: Results for 10:1 Input Impedance Transformation

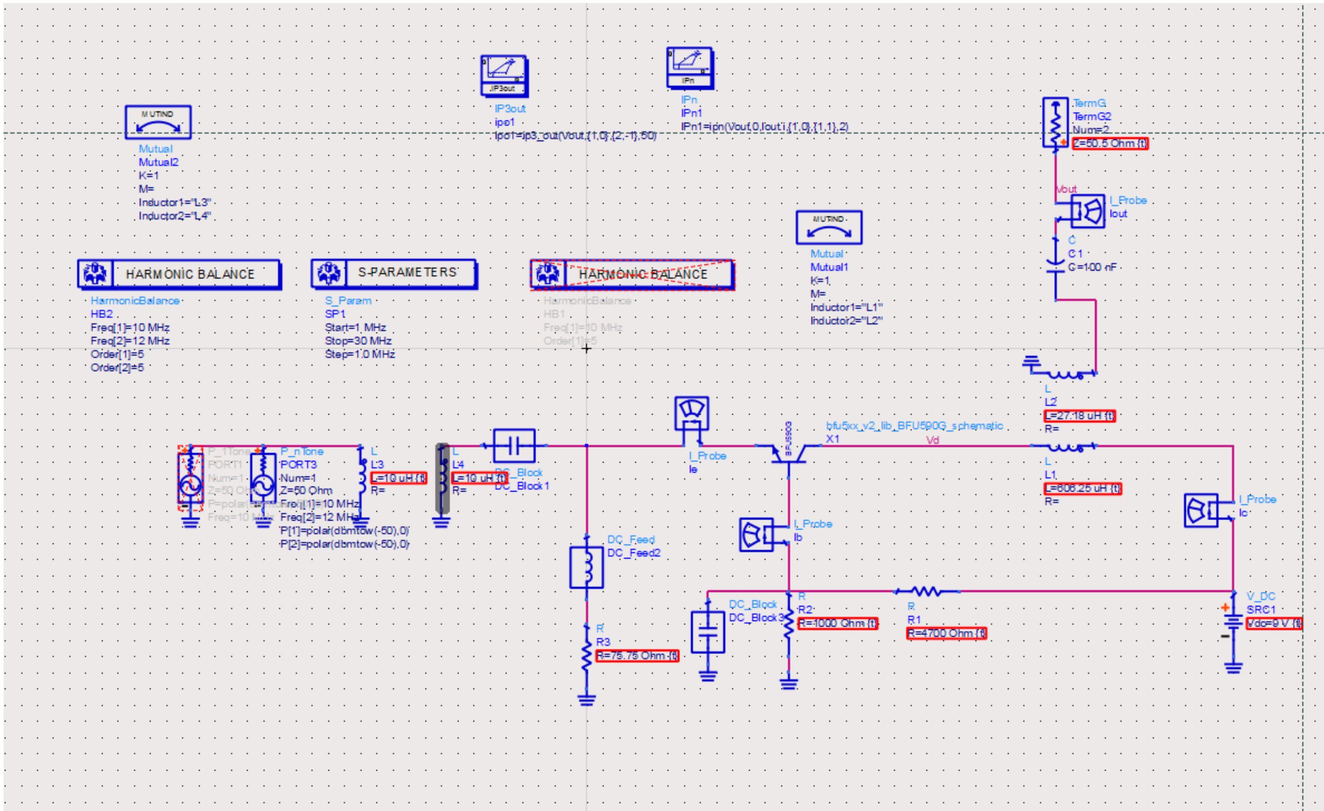


Figure 19: Common Base Amplifier Schematic with Impedance Transformation at the Input and Output

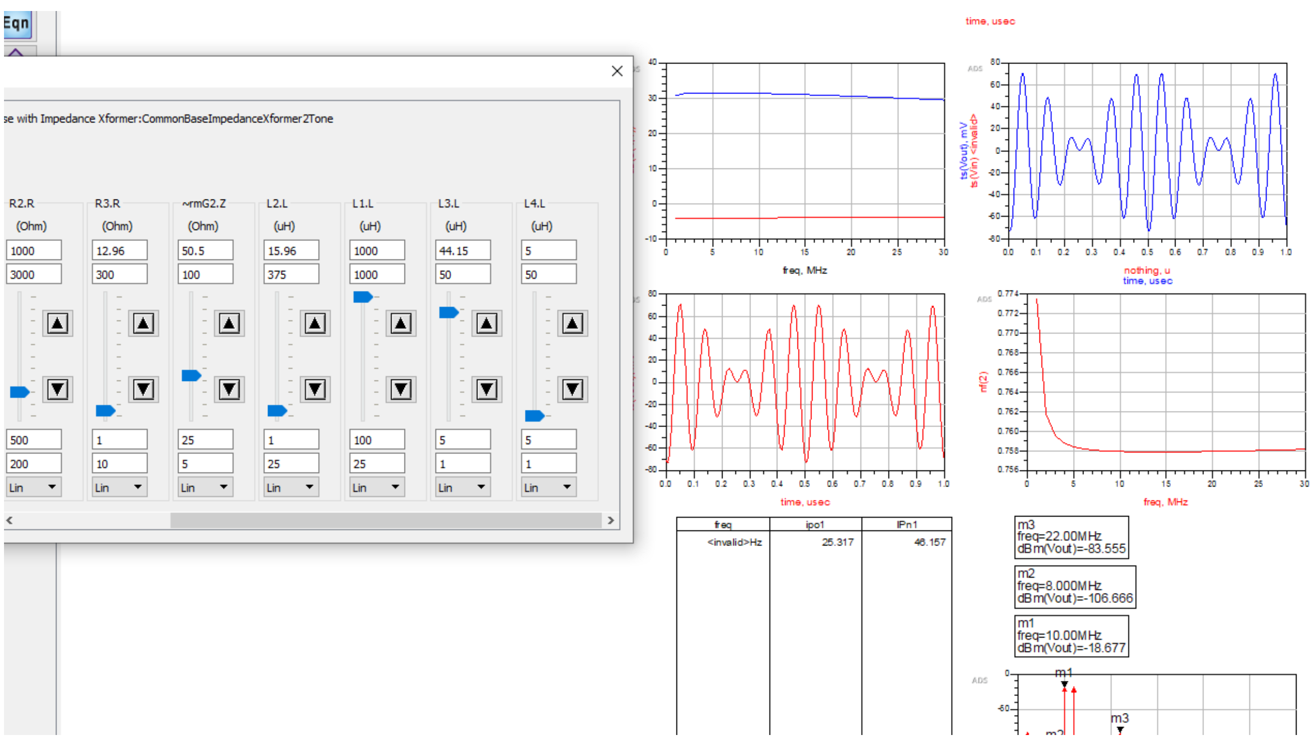


Figure 20: Common Base Amplifier with Input and Output Impedance Transformation Results. Input transformation of 9:1, output transformation of 66. (Extreme values).

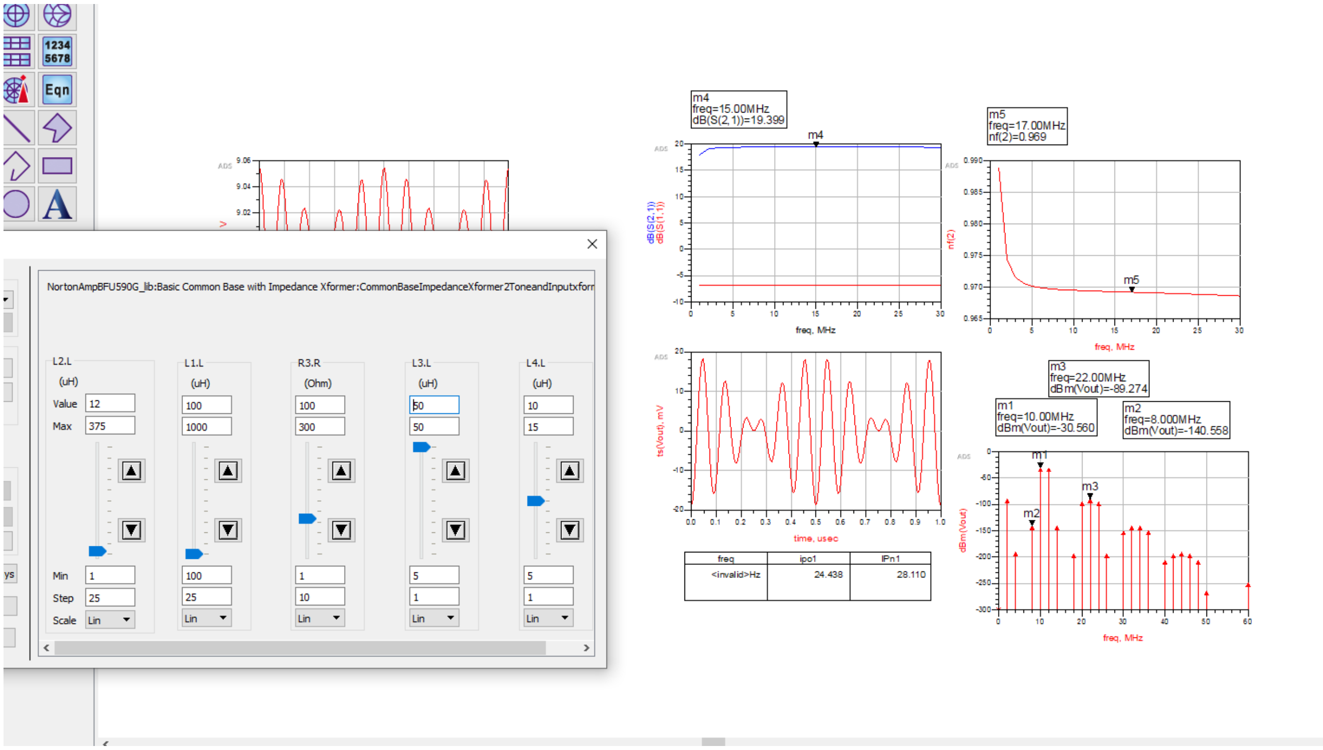


Figure 21: Results for a more realistic output impedance transformation ratio of 10:1 and input transformation of 5:1

Conclusion to the Common Base Amplifier Simulations

We have seen that the most basic form of common base amplifier has a gain that's determined by the ratio Rc/Re. We saw a gain of about 5.5dB for an Re set to 100 Ohms. Noise figure was ~0.55dB, IP2 31dBm, IP3 19dBm for this bias point. The input at the emitter has a very low impedance, we can alter Re to improve the input match, but at the expense of gain.

With an impedance transformer at the collector, we were able to improve the gain of the amplifier by having a high impedance transformation at the collector, to effectively increase the ratio of Rc/Re. The best gain seen was 20 dB but this was with an inductance that is unlikely to be possible to be wound on an Amidon core.

We also investigated using impedance matching to see if we can improve the gain of the amplifier by improving the input match to the amplifier (without increasing R3 and therefore decreasing the gain). With a 4:1 impedance transformation at the input, using a Ruthroff impedance transformer. A traditional 2-winding transformer at the input to vary the impedance match was also investigated.

Then, these two impedance transformations were combined to try and see if we could extract even more gain. With an impedance transformation ratio of 9:1 at the input, and a 20:1 impedance transformation at the output, a gain of >30dB was seen, with an IP2 of 46dBm and IP3 of 25dBm.

The gains seen by some of the optimised amplifier topologies are quite high at their extremes. There is a trade-off that needs to be investigated further, which is the final critical figure of merit for any amplifier design, and that is stability, which will be mentioned toward the end of this report.

Table 1: Comparison of all topologies investigated (all values rounded to nearest integer apart from NF which is rounded to nearest tenth)

Amplifier Topology	**Gain (dB)**	**IP2 (dBm)**	**IP3 (dBm)**	**NF (dB)**
Common Base	6	31	19	0.5
Common Base with output impedance transformer (realistic 5:1 transformation)	12	29	42	0.6
Common Base with Ruthroff Input impedance transformer (4:1 input impedance transformation)	10	20	15	0.9
Common Base with input and output transformer (10:1 O/P and 5:1 I/P)	19	28	24	1

In conclusion, the input and output transformer combination seems to be the best compromise in terms of gain, IMD and noise figure performance.

Common Base Amplifier with Transformer Feedback

We will now move on to the main purpose of this investigation after finding a baseline against which to compare. We will now start simulating the “Basic Amp” and Push-Pull configurations of the Common Base Transformer Feedback Amplifier. The same simulation parameters as before apply. -50dBm input tone, single tone harmonic balance with S-Parameters and Noise Figure, along with two tone harmonic balance for IMD. To start with, I have simulated a 1:2.5:2.5 inductance ratio. This is because the Coilcraft WB series of transformers offers a surface mount part with a secondary that you can centre tap. These transformers offer a quick and easy part to put down on a PCB. The part itself is quite expensive at about £6 for 1-off. For a low volume board design, it’s probably best to go with hand wound transformers on a ferrite core, as otherwise the transformers will cost nearly as much as it would to have the PCBs made!

Wideband Transformers

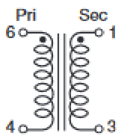
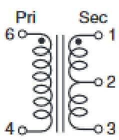
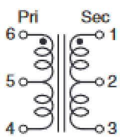
Schematic	Part number SM[1]	Part number TH	Impedance ratio[2] pri:sec	Bandwidth (MHz)	Insertion loss max (dB)	Pri (pins 4-6) L min[3] (µH)	Pri (pins 4-6) DCR max[4] (mOhm)	Sec (pins 1-3) L min[3] (µH)	Sec (pins 1-3) DCR max[4] (mOhm)	DC Imbalance[5] max (mA)
	WB1-1SL_	WB1-1L	1:1	0.150–500	0.70	27	75	27	75	—
	WB1-6SL_	WB1-6L	1:1	0.100–350	0.50	25	100	25	100	—
	WB1.18-3SL_	WB1.18-3L	1:1.18	0.040–300	0.50	90	300	108	330	—
	WB1.5-6SL_	WB1.5-6L	1:1.5	0.050–325	0.26	56	120	84	150	—
	WB2-1-2WSL_	WB2-1-2WL	1:2	0.080–700	1.00	38	100	75	150	—
	WB2.5-6SL_	WB2.5-6L	1:2.5	0.080–225	0.26	30	100	75	130	—
	WB4-6SL_	WB4-6L	1:4	0.100–125	0.50	25	100	100	200	—
	WB9-1SL_	WB9-1L	1:9	0.125–125	0.57	25	100	225	250	—
	WB16-1SL_	WB16-1L	1:16	0.050–100	0.60	56	75	896	330	—
	WB36-1SL_	WB36-1L	1:36	0.100–45	0.50	25	50	900	180	—
	WB1-1TSL_	WB1-1TL	1:1	0.100–375	0.51	25	100	25	100	30
	WB1-6TSL_	WB1-6TL	1:1	0.050–200	0.20	70	150	70	150	18
	WB2-1TSL_	WB2-1TL	1:2	0.070–400	1.00	38	100	75	150	29
	WB2.5-6TSL_	WB2.5-6TL	1:2.5	0.050–125	0.28	56	120	140	200	13
	WB3-1TSL_	WB3-1TL	1:3	0.040–500	0.40	96	110	270	200	4.0
	WB4-1HSL_	WB4-1HL	1:4	0.100–500	0.50	25	120	100	160	15
	WB4-6TSL_	WB4-6TL	1:4	0.050–200	0.50	43	120	172	160	5.0
	WB5-1TSL_	WB5-1TL	1:5	0.050–400	0.30	48	220	240	500	13
	WB8-1TSL_	WB8-1TL	1:8	0.150–400	0.76	18	100	144	270	17
	WB13-1TSL_	WB13-1TL	1:13	0.150–125	0.72	17	90	221	200	10
	WB16-6TSL_	WB16-6TL	1:16	0.050–100	0.60	56	75	896	330	25
	WBT1-6SL_	WBT1-6L	1:1	0.040–200	0.25	70	150	70	150	19
	WBT1.5-1SL_	WBT1.5-1L	1:1.5	0.040–350	0.30	48	150	70	180	19
	WBT2.5-6SL_	WBT2.5-6L	1:2.5	0.050–100	0.26	70	150	175	200	11
	WBT4-1SL_	WBT4-1L	1:3	0.040–150	0.26	45	120	135	160	13
	WBT4-1ASL_	WBT4-1AL	1:4	0.040–350	0.40	96	110	384	220	3.5
	WBT16-1SL_	WBT16-1L	1:16	0.100–100	0.50	25	100	400	300	7.5
	WBT25-1SL_	WBT25-1L	1:25	0.100–65	0.50	25	100	625	350	6.0

Figure 22: Coilcraft transformers for HF. We want a transformer in an SMT package that has a centre tap in the secondary for our feedback transformer. The downside of this is that you can only have a equal turns ratio, so the feedback transformer ratio will always be of the form 1:m:n

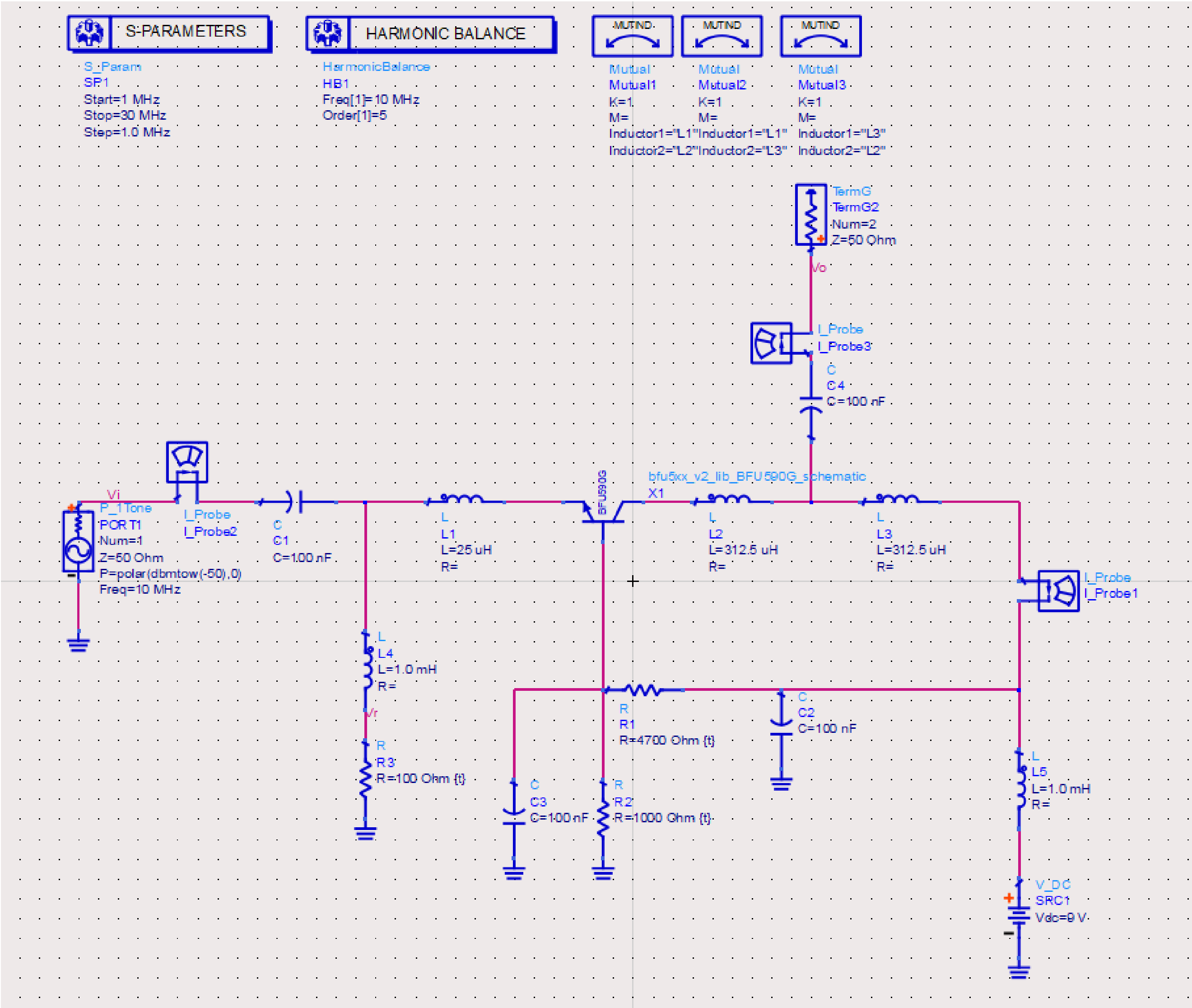


Figure 23 Common Base Amplifier with Transformer Feedback, from Dallas Lankford Paper

The input match has now vastly improved compared to the original Common Base configuration with no feedback, as can be seen by the S11 plot on the Smith Chart. Noise figure has also improved by about 0.1dB. Gain has improved too. One thing of note is that R3 has little to no effect now on the input match. The output load impedance has an effect on the input impedance now. R3 now has a huge effect on IMD and NF though. High R3 → (150Ohm), lowest NF of 0.4dB. Low R3 → good IMD performance, but poor NF.

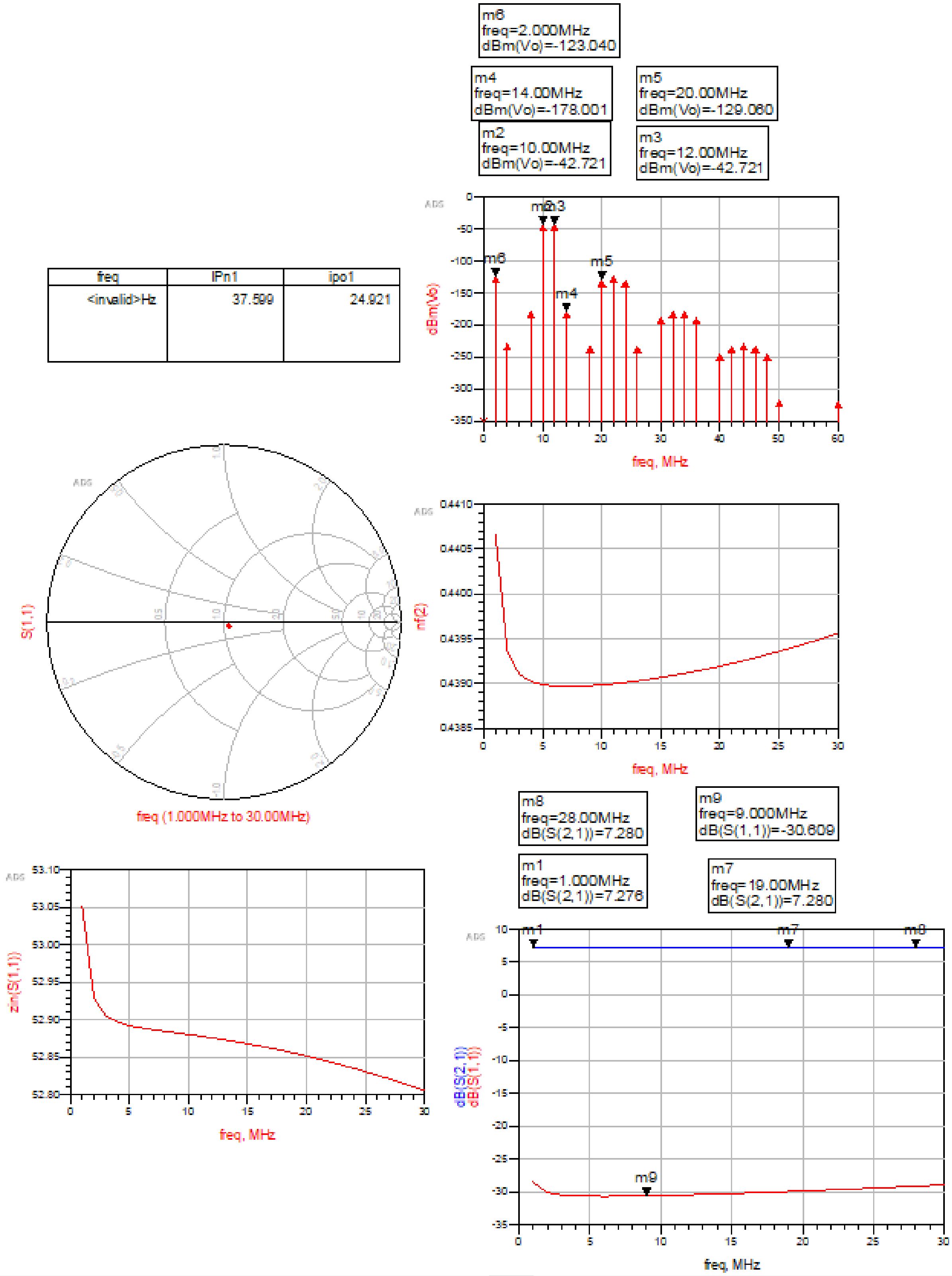


Figure 24: Results for 1:2.5:2.5 Turns Feedback Transformer

1:5:3 on Amidon FT-50-75 inductance values calculated and simulated below, the gain is spot on compared to Lankford's calulated values for this same turns ratio.

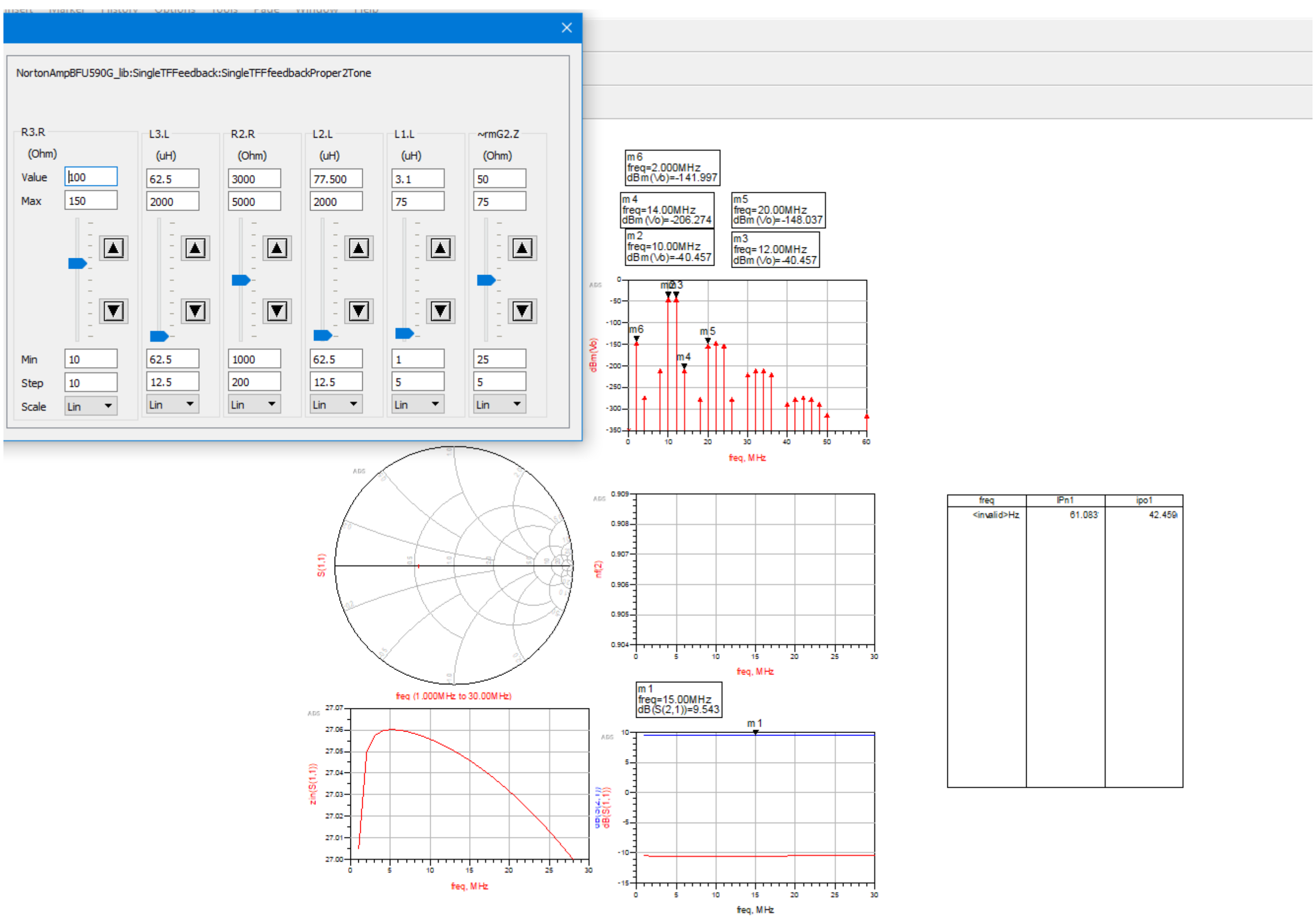


Figure 25: Common Base Transformer Feedback Amplifier with Turns 1:5:3, R3 = 100 Ohms

To make it easier to change the turns ratio of the feedback transformer, I set up some variables to automatically calculate the inductance of each inductor. Turns_m and Turns_n control the turns ratio to L1, and we can then vary L1's inductance (and therefore L2 and L3's).

The inductance of L1 (and therefore the inductance of L2 and L3) seems to have minimal effect on the operation of the circuit as a whole.

1:11:4: Interesting to note that increasing the gain/the turns ratio reduces IMD performance.

By varying this ratio, it was seen that a slight deviation away from the condition n = M^2-m-1, which is the condition for a “two way impedance match” (stated in the Lankford paper and Norton’s patent), the match worsens, but not by much.

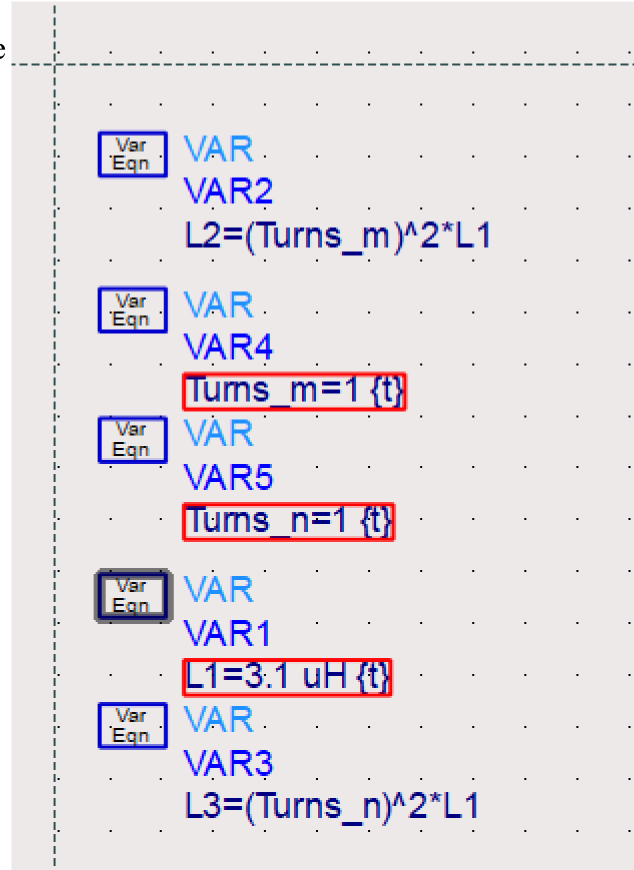


Figure 26: Variables to automatically calculate the inductance of the feedback windings when you set a desired turns ratio, making it easier to tune components. You can now just set M and N in the tuning interface rather than having to manually calculate the inductance

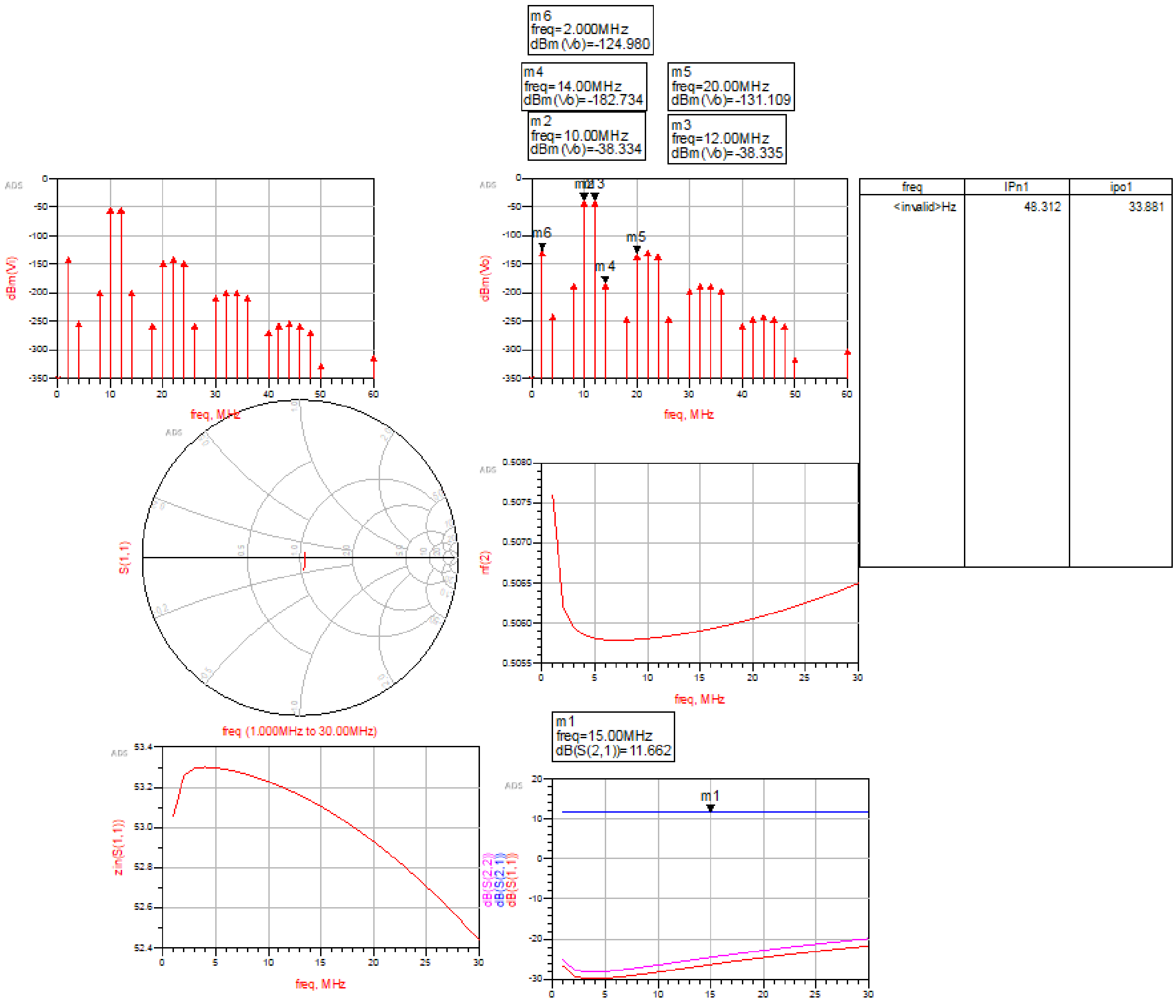


Figure 27: Common Base Transformer Feedback Amplifier with Turns 1:11:4, R3 = 100 Ohms

Another interesting parameter to tune is K, which is the mutual coupling between each trasnformer winding. From some quick research on what the expected coupling coefficient might be, I'm not still 100% sure what an expected coupling coefficient might be, but I'll stick with 0.9 as the lowest for now.

Some resources on coupling coefficients :

https://www.w0qe.com/Technical_Topics/coupling_between_coils.html
https://www.kn5l.net/Measure-Transformer-k/
https://maker.pro/forums/threads/transformer-coupling-coefficients.66258/

A 1:5:3 transformer feedback ratio with k=0.9 is shown in Fig. 28. Previously, I saw that the inductance values of the individual windings had no effect on any part of the circuit, when k = 1. Now, when k=0.9, inductance has a large effect. My only idea here is that when k = 1, the transformer is perfectly coupled, and so the impedance of the winding itself has no effect. To understand what's really going on, I need to understand how mutually coupled inductors are simulated.

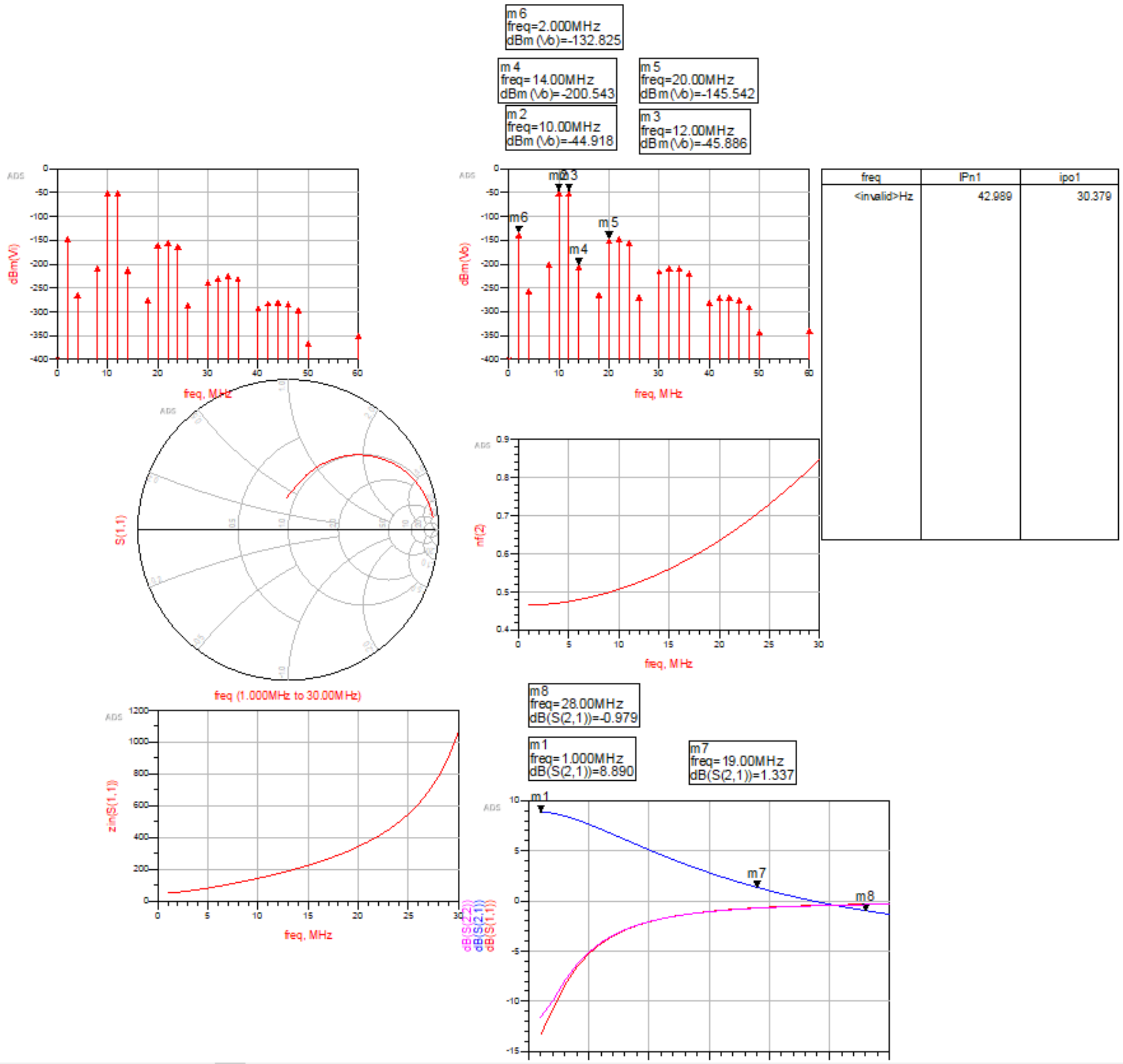


Figure 28: Transformer Feedback Amplifier with Mutual Coupling of 0.9 Between Feedback Transformer Windings

We can now optimise our single ended transformer feedback amplifier for NF and also for IMD and see what our optimum for both is. We will do this with a 1:5:3 turns ratio for the feedback transformer.

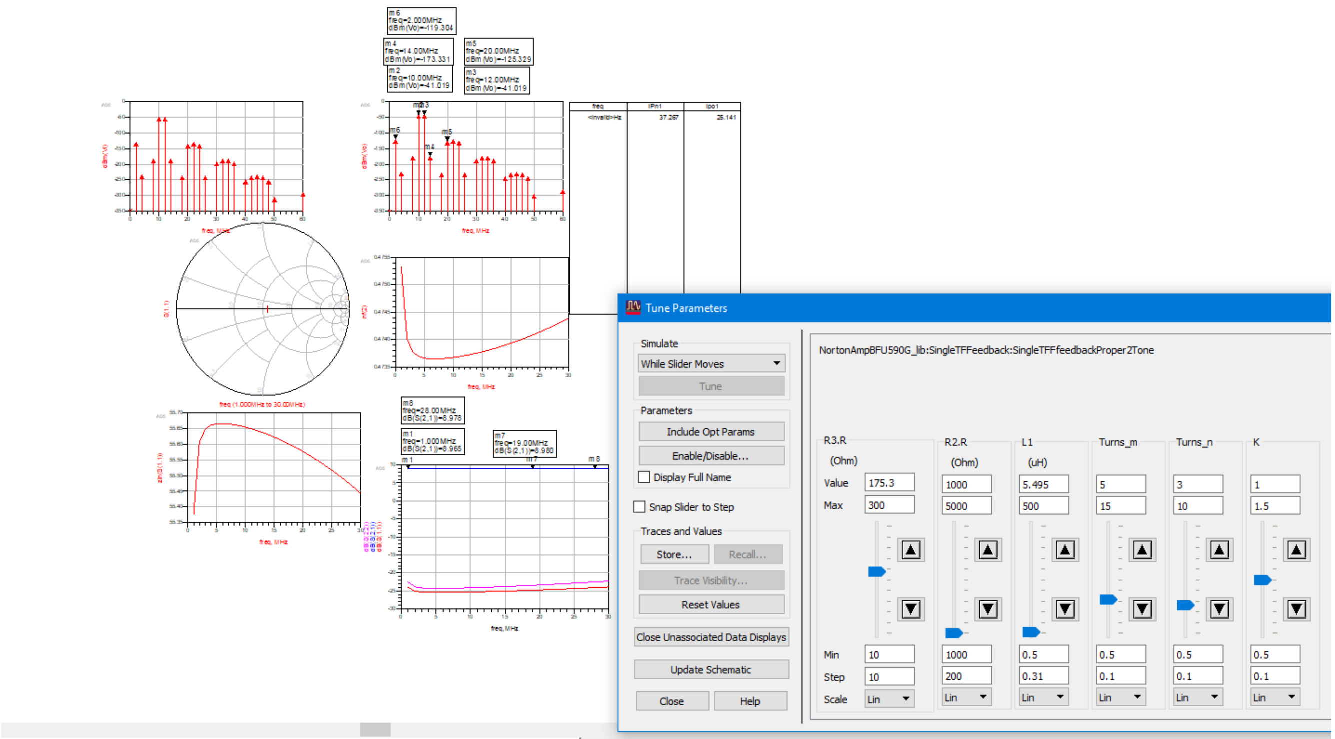


Figure 29 Optimum NF for single ended CB Transformer Feedback 1:5:3 ratio with R3 = 175 Ohms.

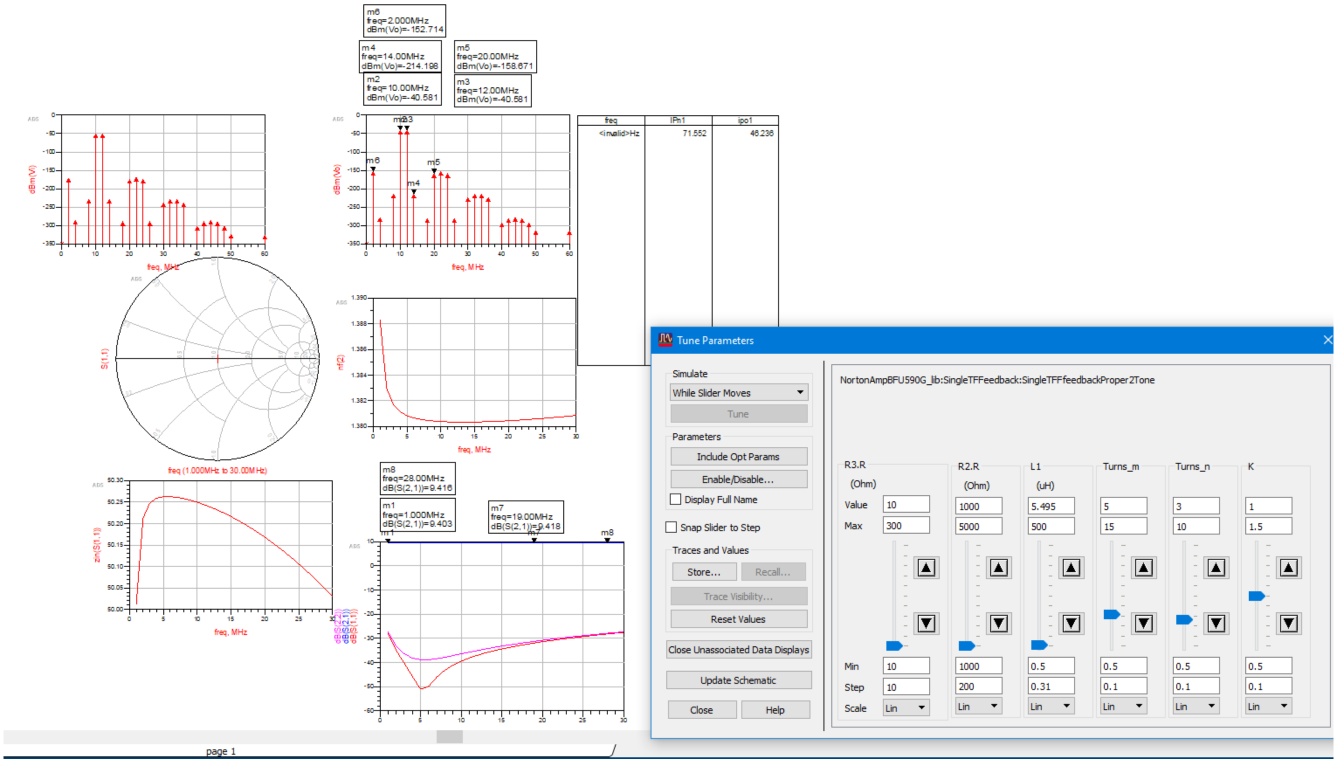


Figure 30: Optimum IP2 IP3 for 1:5:3 transformer feedback ratio, R3 = 10R.

Conclusion for Single Ended Transformer Feedback Amplifiers

The single ended transformer feedback amplifier has shown an improvement over the traditional common base amplifier. We've seen that as R3 decreases, Ic increases, and with this the IMD performance improves at the expense of a higher noise figure.

Table 2: Table of Results for Single Ended Transformer Feedback Amplifier Topologies (all with K = 1)

Amplifier Topology	Gain (dB)	IP2 (dBm)	IP3 (dBm)	NF (dB)
1:2.5:2.5 feedback	7	38	25	.4
1:11:4	12	48	33	.5
1:5:3 (R3 = 175 Ohms)	9	38	25	.5
1:5:3 (R3 = 10 Ohms)	9	71	46	1.3

Push-Pull Transformer Feedback Amplifiers

We now investigate the push pull configuration. This configuration should vastly improve the IP2 performance, because any even order distortion should cancel out. As shown below (R3 is at 5 Ohms so max IMD performance) The IP2 and IP3 are incredibly good. The reason that IP2 is particularly impressive at 238dBm goes back to the power series mentioned at the start of the report. The IP2 intermodulation distortion products are generated from the x^2 term of the power series. Both amplifiers are fed with an input signal 180 degrees out of phase. If we assume one input has a sinusoid of $\sin(wf_1 t)$, the other amplifier's input will have $\cos(wf_1 t)$.This is equivalent to $\sin(x) = -\cos(x)$. This means one input is the negative of the other due to their phase relationship. When we expand the power series with $f(x) = -x$, then squared term will always be positive. When both output add, they add out of phase, so are subtracted, causing the squared term to cancel out. A better explanation can be found here : https://www.highfrequencyelectronics.com/Jan14/1401_HFE_HarmonicSuppression.pdf . The reason our IP2 is incredibly good compared to Lankford's results is that everything in the simulation is perfectly matched. In reality, the transistors and components won't have the exact same characteristics between amplifier paths, so the IP2 terms won't perfectly cancel.

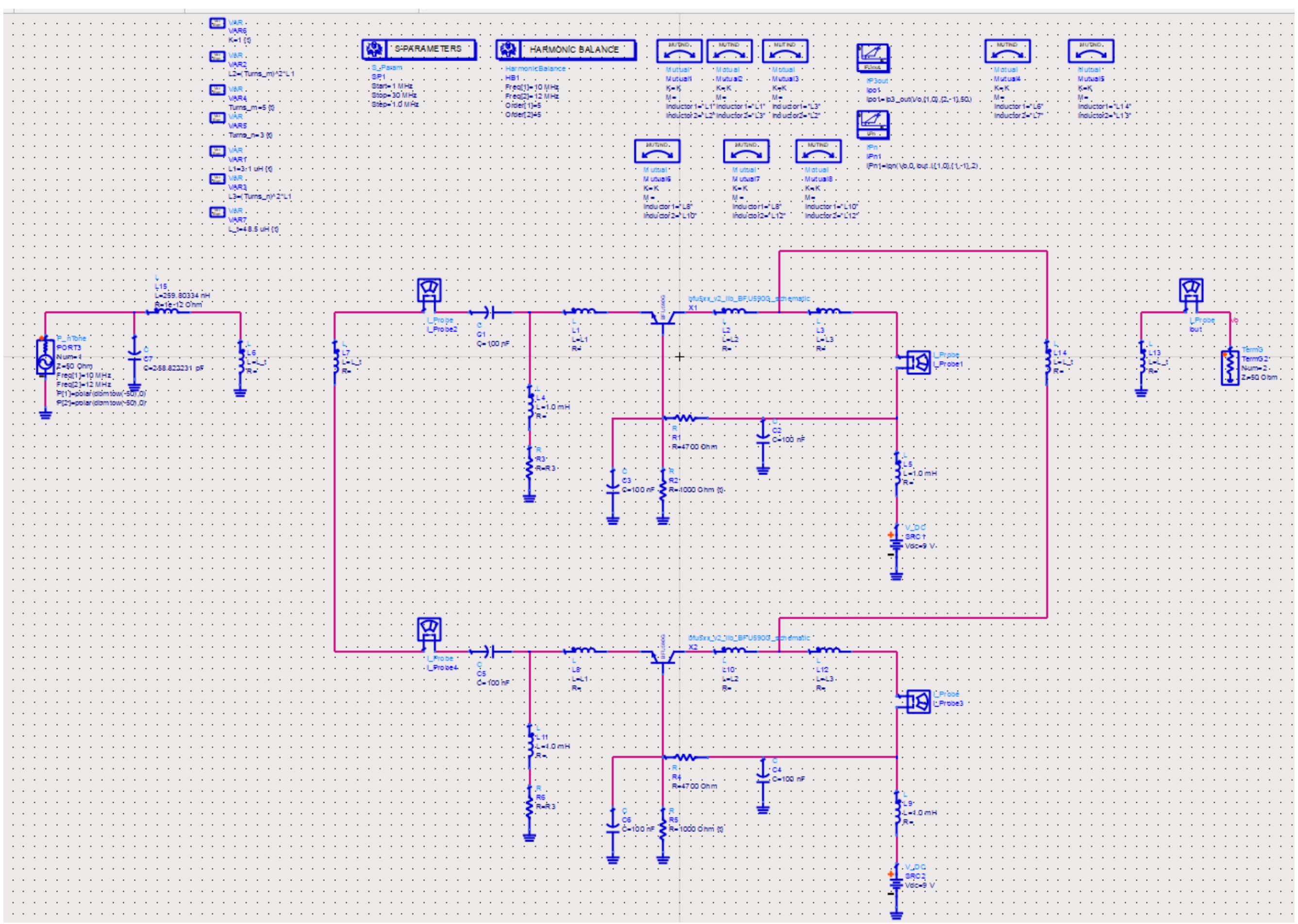


Figure 31: Schematic for the Push Pull Transformer Feedback Amplifier

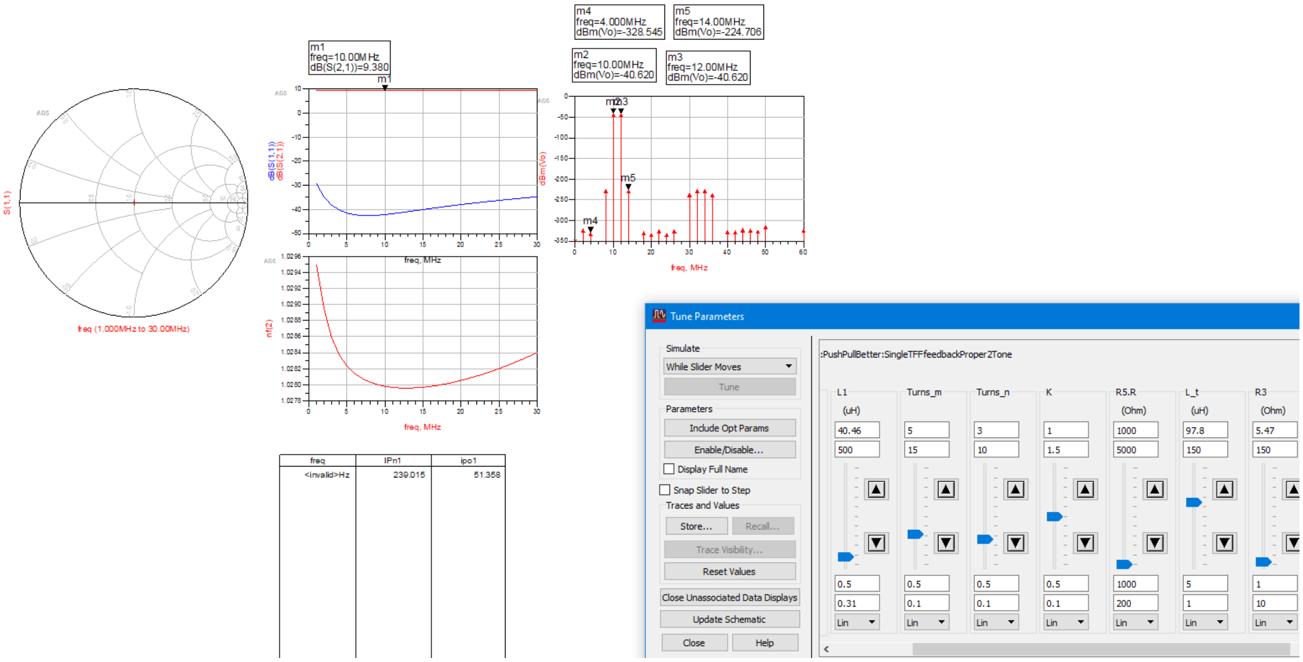


Figure 32: Results for the 1:5:3 Push-Pull configuration. R3 = 5 Ohms, biased for optimum IMD Performance

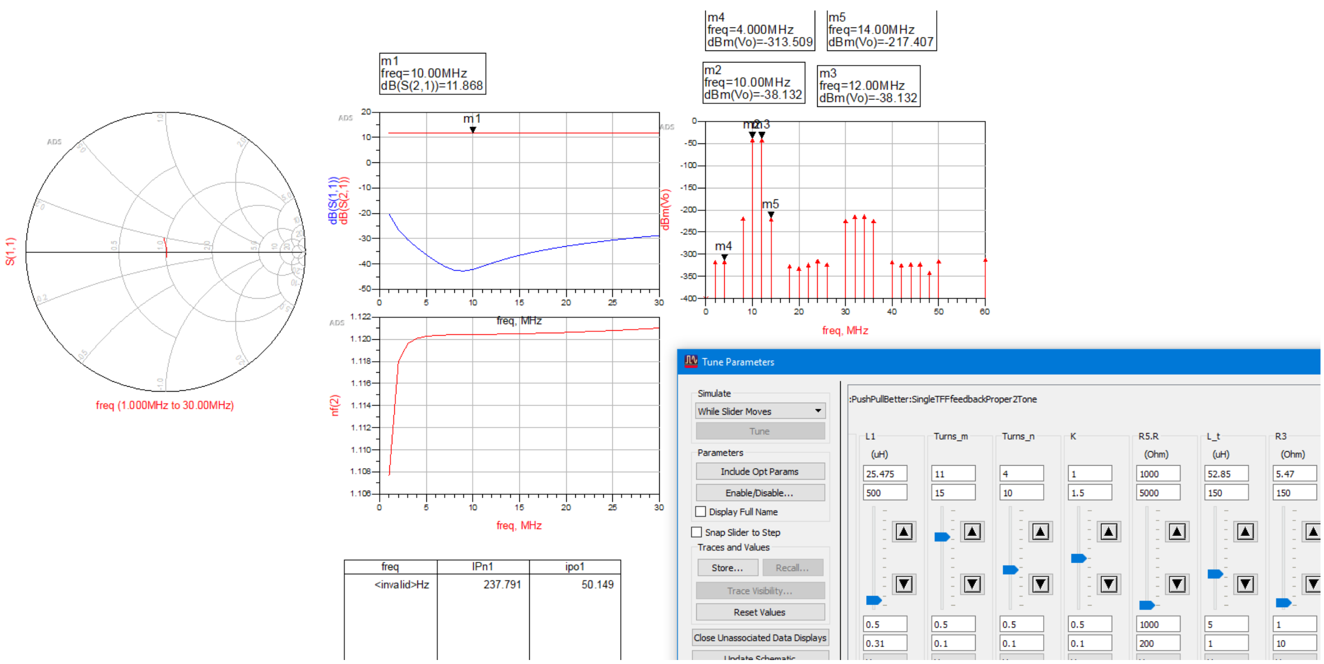


Figure 33: Results for the 1:11:4 Push Pull Amplifier. R3 is biased for optimum IMD performance at 5 Ohms

Comparing the two results for the different turns ratio feedback amplifiers, we can see that there is minimal impact on IP2 and IP3, gain has improved by about 2dB to the value predicted by the equations in the Lankford paper. There is also minimal change in Noise Figure.

Conclusion for Optimising NF and IMD with R3

We have seen throughout this report that increasing R3 has improved our measured Noise Figure and decreased the IMD performance of the amplifier. This has been seen across all topologies of common base amplifier. By changing R3, we are changing the collector current, Ic. We have seen increasing R3 (decreasing Ic) has improved the Noise Figure, and increasing Ic improves the IMD performance.

IMD products are generated at the output of a common base amplifier due to the non-linear relationship between the base-emitter voltage and the collector current. The two articles linked below provides a good treatment on why IMD performances changes with Ic. Our input for the CB amp can be considered to be the base-emitter voltage. Vbe consists of the quiescent Vbe, along with our input voltage excursion away from this bias point.
which is directly propotional to the IMD performance via the non linear equations stated in the expansion of the Ebers Moll equation as shown in the link.

http://g4kno.com/hardware/theory/bjt_distortion2.html

https://web.archive.org/web/20210216123151/http://rfic.eecs.berkeley.edu/142/pdf/module13.pdf (pg 41.)

Through decreasing R3, Ic must increase. As a result, Ib must increase, increasing the voltage drop across the base biasing resistors, and thus increasing Vbe. OIP2 and OIP3 are shown in the first article to be directly propotional to the quiescent current Iq. Thus if we increase Iq through decreasing R3, we improve the IMD performance of the amplifier.

Noise figure is found to vary with Ic also. The lower Ic is, the better the Noise Figure performance, as seen in prior results.

Shot noise from the random motion of charge carriers in the P-N junctions contributes to excess noise in BJTs (we care most about this for noise figure, as the power spectral density of shot noise is white, I.e frequency independent and is wide band), along with “real” resistances in the hybrid pi model of the BJT (such as the base spreading resistance). In finding the equivalent noise at the input in the slides linked : https://www.uio.no/studier/emner/matnat/ifi/INF5460/h16/undervisningsmateriale/f8-1p.pdf
The equivalent noise is found to be directly proportional to Ic and Ib, so by increasing Re, we are decreasing Ic, and therefore the lowering the equivalent input noise generated by the BJT and therefore the noise figure.

Shot noise was derived by Schottky when investigating noise in vacuum tubes. The Schottky equation states that shot noise is directly propotional to the current flowing

Lecture on shot noise:
https://www.youtube.com/watch?v=hDjvOZHcQOI

He explains shot noise in a concise way within the first 3 minutes.

Shot noise from the perspective of photography :

https://www.strollswithmydog.com/photons-poisson-shot-noise/

University of Oslo

Minimizing the noise factor

We found previously the following expressions for optimum noise factor:

$$F_{opt} = 1 + \frac{E_n I_n}{2kT\Delta f}$$

(This can only be achieved when *Rs*=*Ro*=*En/In*.)

We insert the frequency independent terms we found for *En* and *In* and get:

$$F_{opt} = 1 + \sqrt{\frac{2r_x}{\beta_0 r_e} + \frac{1}{\beta_0}}$$

To achieve low noise we must:

- Reduce r_x
- Increase β_0
- Reduce I_C ($r_e \sim 1/I_C$)

Normally you will achieve the lowest noise when the collector current is less than 100µA. If the collector current is very small, we are left with:

$$F_{opt} = 1 + \frac{1}{\sqrt{\beta_0}}$$

INF5460 Noise in bipolar transistors 17

Figure 34: Conclusion from Linked Article on Lowering NF for BJT

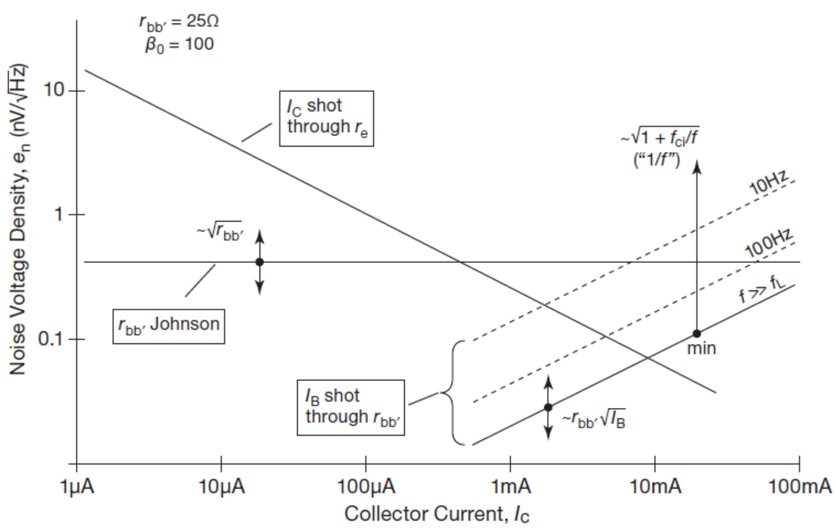


Figure 8.11. Input noise voltage e_n in a BJT. At low currents the shot noise in I_C through r_e dominates; otherwise the Johnson noise in $r_{bb'}$ is the dominant term. However, at low frequencies and high currents, the base-current shot noise through $r_{bb'}$ causes e_n to rise again. These curves assume typical values of flicker-noise breakpoint (f_{ci} ~1 kHz) and base resistance ($r_{bb'}$~25 Ω), with double-sided arrows indicating variation with $r_{bb'}$, and the arrow upward from "min" indicating $1/f$ rise with decreasing frequency.

Figure 35: Diagram showing noise voltage density vs values of IC

Optimising for Noise Figure with Nfopt

Now the question is, can we “have our cake and eat it too” with regards to NF and IMD? Well, with ADS, we can ask it to find the optimum Noise Figure we might be able to achieve at a certain bias point for an amplifier (this is finding our optimum match for Gamma Opt, I.e the source impedance that gives us the best NF). So, we can bias R3 for optimum IMD, and see what ADS says our optimum noise figure can be. The compromise we have to make here is that we’ll have to be make an impedance matching network, to obtain our optimum S11, and this will be, by design, frequency dependent.

Of note is that when R3 is set to about the optimum noise figure, NF opt is reported to be about 0.02dB lower. So there’s not much point trying to optimise NF at this bias point, as we already are near to our optimum anyway (with IMD being sacrificed at this bias point, specifically IP3 worsens by 10dB, R3~73 Ohm). What’s more of interest is seeing if we can have a situation where both IMD performance and NF is really good.

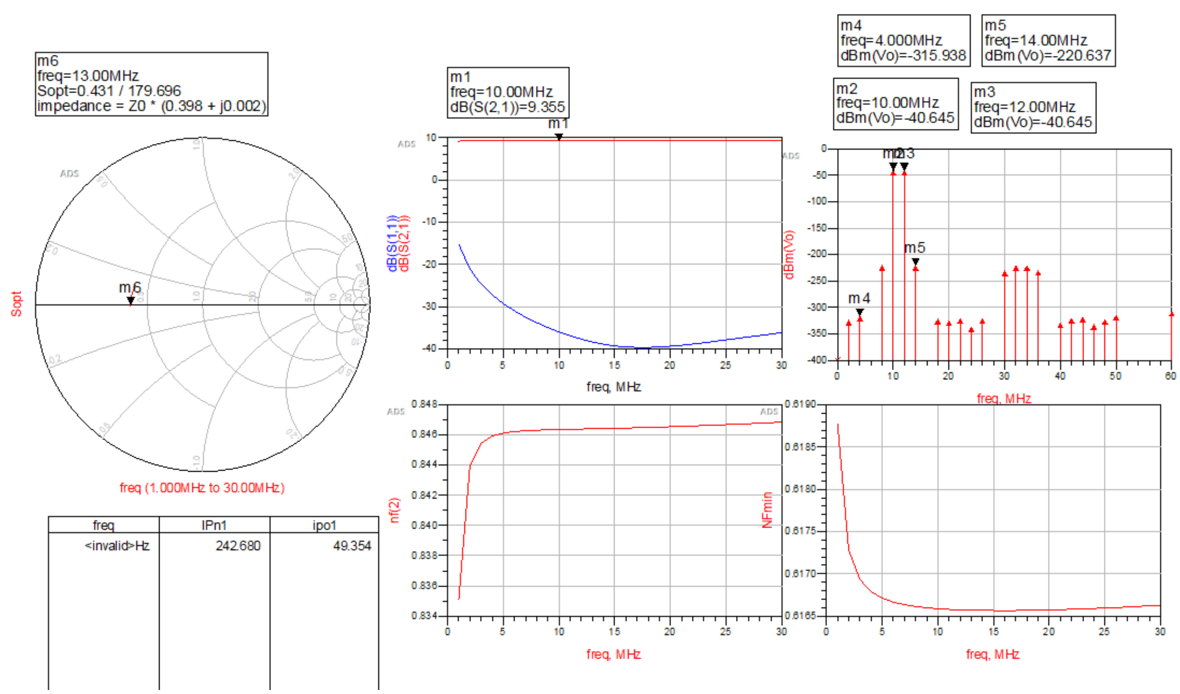


Figure 36: RF Figures of Merit for push-pull variation with additional plot of Sopt and NF Min. Plotted at low R3 of 10 Ohm, where IMD performance is near optimum

We have plotted the minimum possible Noise Figure, along with the optimum impedance match that this noise figure occurs at (Sopt and Nfmin). We can cheat a bit and change our port 1 source impedance from 50 Ohm to the optimum source impedance that ADS has calculated at say halfway through the band of measurements, so 15MHz (Marker 6), and see if this produces the optimum NF. Z0 = 50 Ohms.

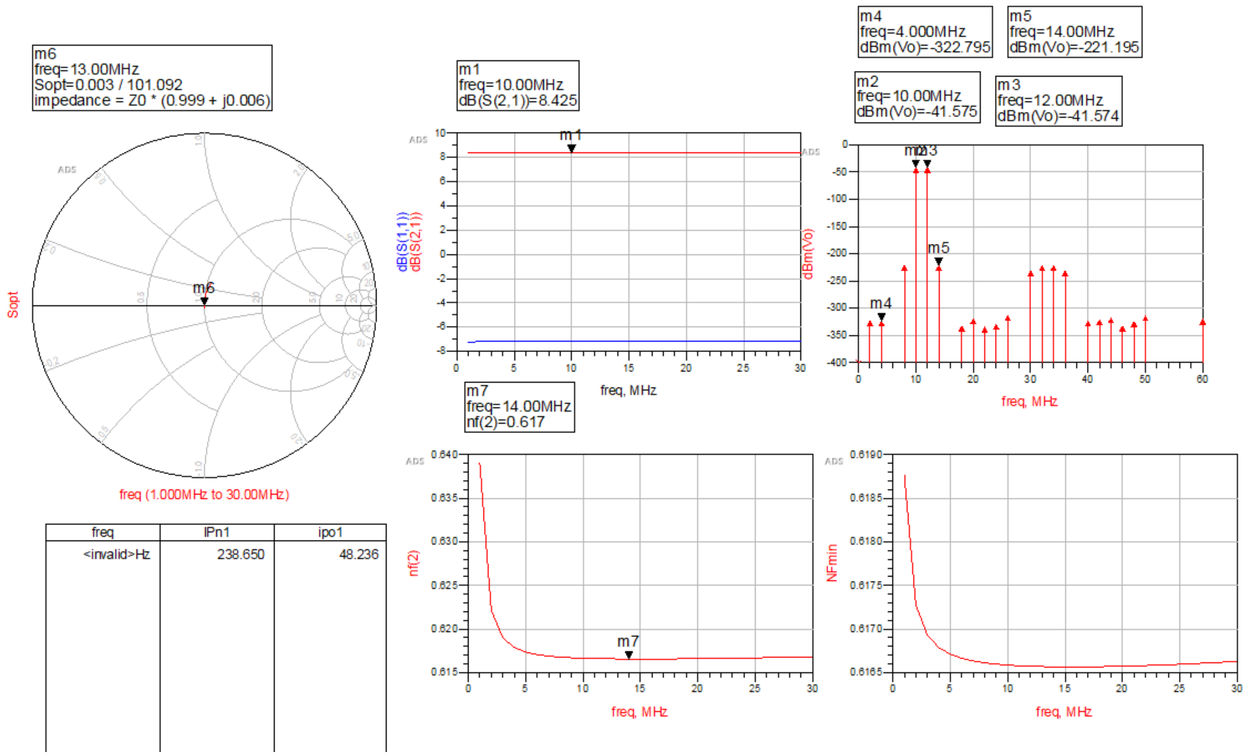


Figure 37: RF Figures of merit for push pull amp with source impedance set to 19.9+j*0.002 Ohm. R3 = 10 Ohms.

This has indeed dropped our Noise Figure to near the optimum that was predicted by Nfmin. IP2 has dropped slightly, and so has the gain. But apart from that, we've managed to improve noise figure by 0.2dB.

We can use the Smith Chart tool in ADS to determine the topology and LC values for the matching network. The idea is to travel along the constant impedance and admittance circles on the Smith Chart, eventually reaching 50 Ohms (centre point of smith chart). We start at the load impedance, and transform this to the desired 50 Ohm point, which means this network will look like 50 Ohms from the source. To start with, I used a series inductor to travel into the inductive upper half of the Smith Chart, until I hit a constant admittance circle which crossed through the 50 Ohm centre point. From there, I used a shunt capacitor (As a shunt capacitor is effectively an admittance) to travel down to the 50 Ohm point. Therefore producing an impedance transformation from our port 1 output impedance of 50 Ohm to the optimum source impedance for Noise Figure This is shown in Figure 37.

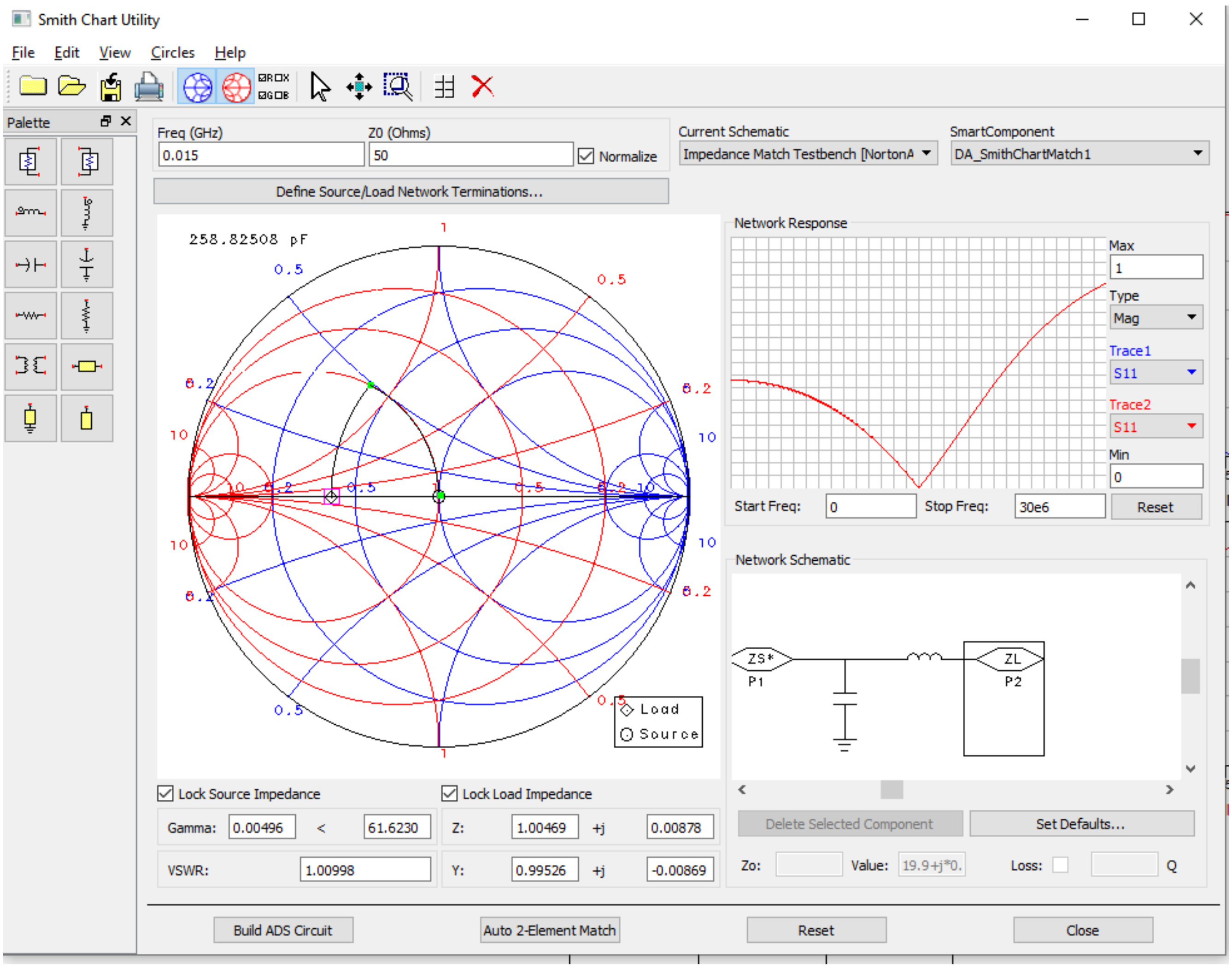


Figure 38: Matching a 50 Ohm source impedance to an Sopt of 19+j0.1 at 15MHz

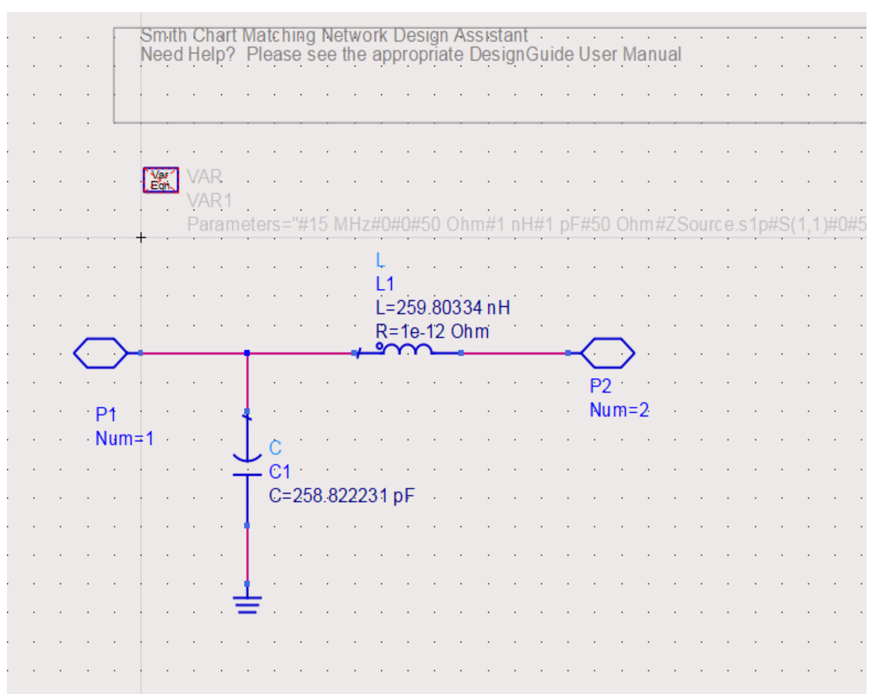


Figure 39: Resultant Impedance Matching Circuit for NFopt at 15MHz

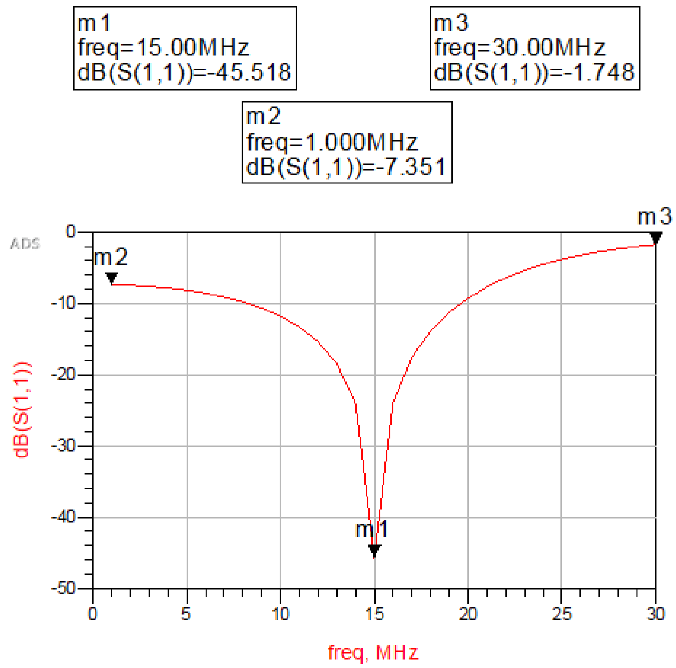


Figure 40: S21 of Impedance Matching Network

The results of this new impedance match on the RF figures of merit are shown below. We see that we have improved the Noise Figure at 15MHz through this impedance match, but because of the inherent frequency dependent nature of the matching network (which is very narrowband. We have simulated this matching network with perfect components so the Q of the matching network will be very high). We have only improved NF near to 15Mhz. Whereas with setting the Port 1 impedance previously to the optimum source impedance, this was setting this impedance at all measurement frequencies, whereas with the matching network, we are using a frequency dependent filter to make our 50 Ohm source impedance "look like" 19.9+j0.1 Ohms. So this won't be functional at all frequencies, which is seen by the S11 plot of the matching network. Only at 15MHz were we matched to 19.9+j0.1 Ohms. At the extremes, I.e 1 and 30MHz our input match was -7 and -1dB respectively. Noise Figure really shoots up as we deviate our frequency away from 15MHz. **It's probably worth investigating wideband matching networks, such that a match to Nfopt can be found over a large range of frequencies.**

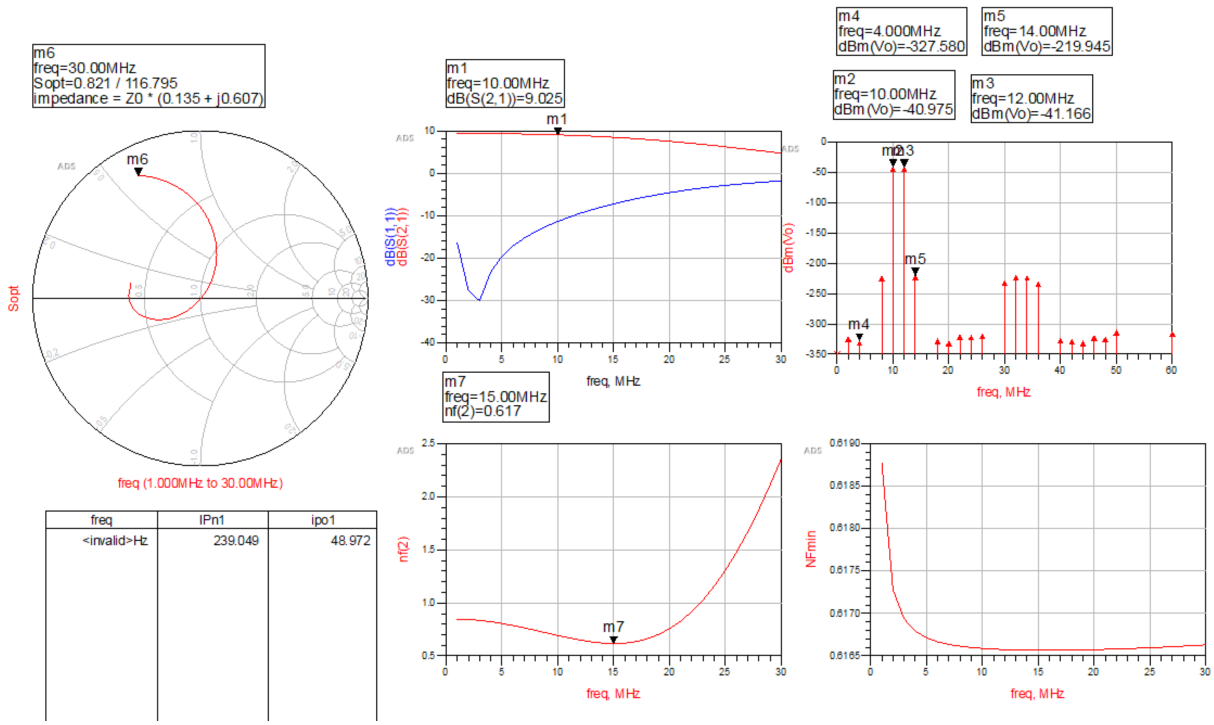


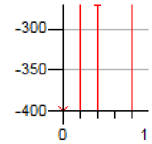
Figure 41: RF Figures of Merit for NFopt/Sopt matching at 15MHz

Stability Factor Analysis

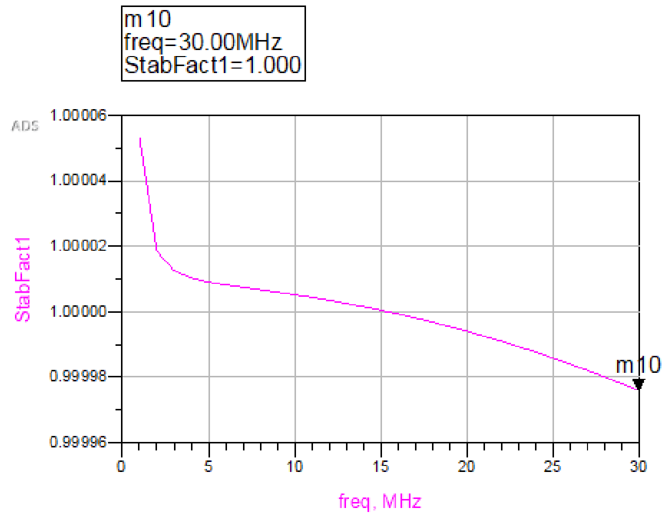
One thing that hasn't been plotted and is worthwhile mentioning is the *stability factor* of these amplifiers. With the high gains I was getting out of some of the topologies (20-30dB), I was a little bit suspicious that there was no trade-off we were making. Well there is, and due to my naivety in the earlier simulations I missed out on analysis this. It's to do with the stability factor. I will start by going back and analysing one of the topologies that yielded very high gain. I've selected the input transformer and output transformer Common Base amplifier that yielded a gain of 30dB.

Stability of an amplifier quantifies if the device is likely to oscillate. If the stability factor is greater than 1, the amplifier is unconditionally stable and will not oscillate. I found that the transformer feedback amplifiers all had a stability factor of 1, but the original common base amplifiers had a stability slightly less than one from approx 20MHz up.

Needs to be investigated further and more theory to read.



freq	StabFact1
1.000 MHz	1.000
2.000 MHz	1.000
3.000 MHz	1.000
4.000 MHz	1.000
5.000 MHz	1.000
6.000 MHz	1.000
7.000 MHz	1.000
8.000 MHz	1.000
9.000 MHz	1.000
10.00 MHz	1.000
11.00 MHz	1.000
12.00 MHz	1.000
13.00 MHz	1.000
14.00 MHz	1.000
15.00 MHz	1.000
16.00 MHz	1.000
17.00 MHz	1.000
18.00 MHz	1.000
19.00 MHz	1.000
20.00 MHz	1.000
21.00 MHz	1.000
22.00 MHz	1.000
23.00 MHz	1.000
24.00 MHz	1.000
25.00 MHz	1.000
26.00 MHz	1.000
27.00 MHz	1.000
28.00 MHz	1.000
29.00 MHz	1.000
30.00 MHz	1.000



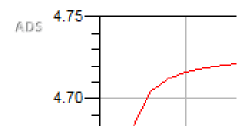


Figure 42: Stability factor of 1:5:3 transformer feedback ratio amplifier.

Limitations of my Simulations

All the simulations I've done have been very much in the ideal situation. We haven't considered the Q factor of any of the resonant components, the only non-ideal factor really considered in this report was the K factor of coupling between the transformer feedback windings.

Really, I should investigate non-idealities in the matching network for finding Nfopt, etc. I think once I've settled on a single design to go for, then it's worth fully investigating any non-ideal situations, otherwise there would have been too many deviations at each discussion of an individual amplifier topology, and this report would have been a lot longer. This report has served as an initial investigation into each potential amplifier topology.

PCB Design

To test each topology, I propose to make a test board with all topologies of Common Base and Norton Amplifier discussed here.

Each topology should have it's bias to be able to be disconnected/connected to the supply via a jumper or similar method so we don't draw a load of current. Ideally R3 for each amplifier should be a potentiometer in the range of 0-150 Ohm for each, but I want to put a fit/no fit bias parallel to this potentiometer on the board so that we can put a precise resistor value down. Ideally the potentiometer should be linear, not logarithmic.

Some designs have a ferrite bead at the base and collector of the BJT, these should be put in parallel or on the same footprint as a 0 Ohm resistor, to see the effect the ferrite bead has. There's also the question of what our impedance peak for the ferrite bead should be. Further Investigation needed.

Further Work to Do

-More extensive stability analysis
-Further investigation into noise

Further Reading

Michael Steer : RF Design Series. This is a series of free RF design books that are really good.
https://repository.lib.ncsu.edu/items/c920fb47-8458-433e-9cd8-b5cacf70e859

Pozar, Microwave Engineering.

Boylestad Nashelsky

Niknejad's lecture notes https://web.archive.org/web/20210216114442/http://rfic.eecs.berkeley.edu/142/lectures.html

Lathi, Modern Analog and Digital Comms systems. Good for the statistics section.