Study Notes for AWR Tutorial

Your Name

August 1, 2024

Abstract

These are study notes for the Advanced Writing and Research (AWR) tutorial.

Contents

1 Introduction

This is an introductory tutorial to use AWR from the Roland device.

2 Videos

2.1 Design Troubleshooting for Stability Part 1

Instability arises from the use of feedback and gain together. This can be controlled by adding loss or adjusting bypassing in the lab to stabilize the circuit.

From AWR DE Tutorial 5

- After selecting the new schematic, name it and then press `Ctrl + L` to add elements.
- To see the details of a component, right-click and select `Help`. [?]
- You can add equations as in QUCS and use the factors in the component details.
- To tune an item or a factor on the graph, click the `Tune` tool above and then click on the factor; it will change to a blue color.
- Set the limit of the chosen factor and then click the tuner. It is behind the identifier tool that chose the factor to tune. You will see the variable tuner has limits and then you can change it above and below.

- To plot the output voltage (V_{out}), you need a voltmeter. Connect it in shunt with the load resistor R chosen from the element menu.
- To measure the DC characteristics of the amplifier (V_{out} vs V_{in}), you need a linear graph. Add a new measurement, select `Nonlinear` and then `Voltage`. Choose V_{DC} on the measurement component and place the voltmeter on the vertical axis. For the x-axis, use the voltage from the source.
- Click `Apply` and then choose the simulation tool on the schematic.

From AWR DE Tutorial 6

- From the elements menu, you can also find the voltage source (V source). If it is AC, to measure it you will need a voltmeter similar to the one used on the load resistor.
- The working frequency is a global value defined for the overall project from the project options menu. Define the frequency range and then from the global units icon above, choose the frequency and change its unit to kHz in this case.
- The netlist file that can be imported into AWR has the extension `.cir`. From the netlist menu, select `Netlists` and then choose `Import Netlist`.
- With `Ctrl + K`, you can add the important netlist.
- After adding the symbols, you can change them from square and normal device shapes from `Properties > Symbols`, then choose the needed shape.
- To add the voltage source, add `DCVS` and `ACVS` from the elements.

From AWR DE Tutorial 7

- The models that can be used in AWR are the PSpice models with `.cir` extension.
- These are called AWR netlist files.
- From `Steer 03.Wideband Amplifier Design Using the Negative Image Model, Part F`, we can see that the model used with the transistor is added in the data files above the graphs.

From Impedance Matching AWR

- From project options, you can change many of the item units.
- You can use the cookbook for ADS and check the LNA simulation part and try to replicate it in AWR.
- To measure the input impedance, it is a linear measure, and you can find it on the graph after adding a new measure. Choose Z_{in}.
- Define Z_{in} unit, the port number, and the source name. If there is a transmission line after the input port, it will appear.
- It is important to sweep the frequency and define the number of points for measurement to determine at which frequencies the device should read a measurement. This can be done from the project options window.
- To add another plot on the graph, click `Duplicate Measurement` and then from the window called `Modify Measurement`, you can add the changes between the two lines.
- For the 50-ohm resistor, if you change the Z_{in} measure from 75 ohms with the transmission line to 50 ohms, you will see matching at 50 ohms for Z_{in} real and the imaginary value is zero, indicating matching.
- In the case of 75 ohms with a 50-ohm transmission line, you will see the imaginary part increase with frequency from -20 to +20. If the imaginary part is greater than 0, it means positive reactance, indicating inductance, and negative reactance means it is capacitive.

- If you remove the resistor and use just the transmission line and simulate, you will see the real and imaginary parts are 0 ohms at 2.4 GHz if the transmission line is open, not grounded. If grounded, there will be a very large Z_{in} at 2.4 GHz.
- In the matching case, measure the impedance on the Smith chart and check where the point is for your frequency. Connect a series capacitor and a parallel inductor if the point on the Smith chart is on the capacitive part under the middle line. Then tune the values of C and L and check which one has a larger effect on the point. Tune it to get the point above on the inductive part, then with the capacitor value change it to reach the middle line indicating the point is totally matched.
- After adding a microstrip line with T-line used as a load, it should be matched when the length and the width give a 50-ohm for the material used on the substrate. In this case, the microstrip line and the T-line are 50-ohm, meaning they should be matched around the frequency of the T-line, which is 2.4 GHz. Matching can be seen from the impedance linear graph or from the Smith chart.
- If you have just one port, you will measure on Smith just S_{11}. When measuring, it is a must to use the MSUB on the schematic to define the substrate for the transmission line. The values of the variables on the MSUB are taken from the window where we define the width and the length on the MSline, meaning ϵ_r and tan δ and so on.
- When matching with MSline, define the length and then you can add a parallel one in front of the MSline and define the length and tune it to get the matched point. Adding the admittance might help on Smith to get a line to move with it. On the new line, you can play with the width and the length.
- The microstrip TEE or MTEE is used to connect the MS lines instead of connecting them together, which might lead to parasitic capacitance. So the idea is to make the design and see the Smith chart and match with the MS lines, and then control the length and width of the MS lines to get the matched point. Then connect them with the MTEE with a dollar sign which will connect the three sides with the equivalent width for the line.
- If an MS line is not grounded, you need to connect the Mopen line, which is an MS line with open termination.
- To see the length of the MS lines, you can see the layout graph and then import the other component models that define the layout. AWR requires some experience because it does not always have the best documentation.

Microstrip Antenna AWR Lab Video

- This video is important for working with T-lines.

3 Technical Note from the CEL Blog

The CE3512K2 is an air cavity package structure which is effective in reducing RF power loss consumed by the packaging material. For HEMT, the gate has high impedance at DC, so there is only a need for bias at the drain and connecting V_g directly. The bias at the drain is sufficient to use just the inductor, and it should be high enough to provide good RF isolation at the operating frequency. The inductor can also be part of the matching network. A shunt capacitor provides a low impedance point, creating a low pass filter for noise and unwanted signals from the DC source. As frequency increases, parasitic effects become dominant, necessitating transmission line-based components in both bias and matching circuits.

Transmission Line Matching The quarter wavelength ($\lambda/4$) transmission line is a crucial element in RF design. the ($\lambda/4$) can replace he inductor on rf isolation. and for this technique there is no frequency limit. Radial and rectangle open stub are the most common Tlines on rf. thr coupling capacitors has to undesirable effects it has high insertion loss as a fraction of dB directly degrades the noise figure by the same amount and the matching network is become complicated due to the parasitic effect of the high frequency capacitors.

output series capacitor has negatable impact on the circuit performance

you will need bias tee if you you wanna exclude using the input capacitors to prevent dc coupling with the instrument during the measurement. this only on noise figure measurement.

Tlines component on the matching network usually used on the cas for the freq higher than 10Ghz

Ampleon is a good company for rf transistor with application note and references. as stated on the website edaboard.com after you have the transistor model you can chose a bias point then stability analysis with Rollet factor then design the input and output matching. also the power transistor has optimum load impedance that maximize the delivered power or the gain.

one simple measurement is to make representation for the S param and then compare between QUCS and AWR

for Simulation

add S2p on the data files by import then add it on the schematic as rectangle then click and from Properties change the symbol. then follow the circuit stubs as on the Matching with the AWR video. the bias circuit is to design the bias point and the matching to work on 50ohm on HEMT transistor has highrt breakdown voltage and high operational temp. $\Gamma(S) = \Gamma(opt)$ must be first for LNA connect the bias circuit and define the bias voltaged and calculate Rg and Rd and the capacitor and the inductors

4 From github (https://github.com/aliruveycan/Maximum-Earnings-using-AWR-Microwave-Office-Amplifier-Design-and-Simulation.git)

to simulate the transistor on the data files upload the s parameter file and on the graphs plot all S parameters graphs.

then you can match with the Tlines length as on the AWR 1 hour tutorial for Matching.

5 Diffrent pdf notes from google Image

From Ultra wid band amp by chengcheng Xu Rf amplifier is used to amplify small signal amplitude to a large amplitude in transmission and receive signal from antennas.

if the BW from 1.5 to 3.5 Ghz it means that the amplifier can be called 2 GHz amp so our amplifer can be called 1GHz or 800 MHz or 600MHz amplifer for the SNSPDs

Not understood phrase is the final gain is 20 dB with the power gain slightly below 30 dBm.

if the transistor IV curve is mostly linear then the output signal gain will be linear otherwise gain will be non linear.

Class A amplifier has poor efficiency when converting the dc power to high freq output power the lost power will br converted to heat on the device this power that is not converted to rf power but it provide great output to input linearity

Class B more efficient than A but it scarifies the output linearity to input signal ratio. lead to distortion on the output signal half of the signal only amplified

push pull configuration with two transistor may solve the class B amplifier issue.

if the spice model is not provided so the setup a bias network is not important you will use on the s2p file form the provider then you need to match the output of the transistor to the output matching circuit also consider the stability of the amplifier if it is conditional on unconditional stable.

the S22 matching with the output matching network is created first than the inpupt matching network

if S11 is not matched all the input power will not be deliver to the amplifier

the biasing circuit or bias point use one that is recommended from the the provider bias points means the points for the s2p files that provided Vds and Vgs and Id

chose the bias point that has gain near to your interested freq on the thesis they use s2p at 3v 60 mA bias this can be show from the graph for MSG/MAG and S21 dB you see that S21 has gain at the needed frequency.

this bias point also avilable on the AWR software so no need to import the file on the Microwave office software

P1dB compression point is also important as the transistor it self is not linear so the output / input ratio can't be linear

it is defined as the point where the device output power and the linear model output different by 1dB.

this point where the device also start to provide Nonlinear behavior so you look at the p1dB point on the data sheet

if you wanna use another transistor you should follow the same process.

the FPD2250 has 1dB point 31dBm and if the gain is 10dB so the max power input to the transistor is 21dBm so you need to see the conversion of the power values to dBm and check the P1dB effect. the 21dBm will be above the output of the 1 st stage.

the stability is not related to which gain value of BW is the device it self is unstable this will not has an effect on what gain or other parameter so we need to use stable network to prevent oscillating output

the stability on Microwave office is tested by K() and B1 parameter and also by SCIR1() and SCIR2() parameters of the device

K>1 and B not < 0

SCRR 1 will create solid line and dashed lines circles the inside or out side the l´solid line circle

the solid circle will be the stable one. the goal is to be inside the stable region

Design steps

- add the transistor model and change it to symbol or leave it as it is.
- check also if it is avilable on AWR or not at the beaning
- design a bias point circuit should be stable or should be called stability circuit. ther is no voltage sources as on fig on the tutorial just Rd and Rout and Rg and the two ports also you might need to use MSline and control it's length tune the component then see the stability for K and B values.
- the above step is reoeated for the two the stages on thier own.
- this stability test can be done on smith plot with the circles need te studied alone.

Wideband Matching

- the goal is to has a Wideband at the first gain is not important at this point as the BW.
- so Gmax is not needed now check table one on the tutorial pdf
- matching method are Chebyshev matching and lumped elements matching
- Chebyshev matching is based on Chebyshev polynomial and the matching network is mostly consist of quarter wave transformer and from the BW and quarter wave transformer section number the impedance of each quarter wave transistor is generated Chebyshev Matching is on figure 12
- then you need to see the matching on smith see S11 and S22 it should be nearly perfect match means it is circulated around the center point smith plot.
- as shown on figure 12 Chebyshev has 6 Tline for each S11 and S22 and the and the length of the quarter wave transistor need to be quarter of the signal wavelength. so the reault now for Rf application is not realestic and the and the design will be very large and amplifier detentions will be large
- so the Chebyshev matching is not implemented here.

lumped element match

-

6 matching

on smith for any point to match it move with circle until you get a point on a circle that pass by the smith center which is 50 ohm matched.

on impedance smith series R go go right toward the center line series C goes down series L goes up

on the admittance plane the shunt R,L,C goes the same L up C down but R left.

on Lec 7 impedance match with smith video is very nice

if

$$z_o = 50ohmandZ_s = 50ohmsoZ_ithenormalizedimpedanceis1meansfullmatched$$

Z_i which is represented on smith chart

adding resistor will add loss to your design so using L and C is is enough and better. impedance match with the Tlines is very diffcult compared to Lumped element but the idea is the same there is numerating on smith for the matching lines. bit the middle line for the real is the same but the imagnary numbers are written now interms of Λ.

Txline feature from AER Videos

from tools chose txline it is used to know how much width should the line be to be 50ohm or how length should be to 90 deg at 80 Ghz for example on the txline window change to mm on length and width using the arrow on the middle to change between the calcuation then use the length and width on thes chematic then simulate to see the smith chart you will see the point is on the middle if smitt

Sweep feature on the parameter you want sweep click right then chose sweep define a start and end and steps. and define also by equation a value to initialize the variable. **Reflection coefficient measurement** from the graph and S11 measure but at the beginning you have to add measure port at the input all of that is on the polar plot not smith then tune the Tline length. and see the effect on S11

if you change the port to harmonic balance port you will be able to see the I V values on this device. time waveforms of I V. on elements it is called port1 you will see the wave source.after it you need a V meter or I meter from element chose V$_meterandalsoImeterImeteronseriesafterportandvmeteronshunt.fromthegraphrecta$

on plot chose properties then measurement then chose which graph on left and which on right. side.

on the Tline is length 0 the I V time relation are on phase when change the length the VSWR is the same and the real part of the reflection coefficient is the same but hte phase deg changes always this means the phase difference between IVtime relation will change.

due to the phase changes the Tline is considered as network from inductors series and capacitor parallel with the the voltage source.

VSWr is the ration between

$$V_max$$

/

$$V_min$$

there are some simulation AWR videos on the Microwave labcast channel
Simulation

- import the sparmeter and make graph for the device at the beganing before anything.
- then you can make your graph from the graph tool
- first stabilize your circuit connet in out series R and shunt R at the out this will leads to uncoditional stability.
- then you can check that by the by K measure or SCIR1 ans SCIR2 for stability circle.
- now need to match over the wide band this is most important than getting gain. so BW is the important now
- the Match step is the most difficult this can be by the TLines and the lumped elements
- most popular on AWR the Tline method from the Txline tool you can characterize your Tline properties.
- Match output before the input it is easy and important for the gain means match s22 of the transistor with the output load or matching.
- then Match the input S11 for the transistor with the input circuit to be sure that the power will be deceived to the transistor otherwise it will not.
- now Design input match and output match sepertly and check them on smith the trick here is to make the whore freq range around the match point middle of smith
- and define only the operating or project frequencies on fron Dc to 1 ghz and define for th txline frequencies the middle point 500 MHz or less it means to make the Wideband match you need to match around the middle frequency.
- after you make the matching circuits make blocks and connect them to the in and out of the transistor
- then check the gain now.
- then change the port to source nd simulate the power on the device to see the total power output on this case you will see only the part where the S parametr measured mean the linear region not the whole characterstcis this due to no Nonlinear model avilable for the transistor

- and due to no Nonlinear model the Nonlinearity mean P1dB and OPI3 points of the device will be read only from the datasheet as mentioned on the pdf tutorial by chengcheng zu. on the Tlines from the txline which is considered to be real Tline insted of the ideal one. you will use the stability T junctions when connect 3 Diffrent Txlines.

Matching Circuit Notes

- **From AWR Matching Tutorial**
- An ideal transmission line (Tline) is lossless and provides a perfect phase delay at a specific frequency, similar to what was discussed in class.
- Under the transmission lines menu, you will find the phase menu, where you can choose 'Tlin' and 'Tloc' options. These are used for the lossless case.
- For the Tline, Z_0 is the characteristic impedance. Keep it at $50\,\Omega$. EL is the electrical length, which is not the physical length, but the phase delay in degrees that will apply to the signal. F_0 is the center frequency, which is crucial for determining EL.
- To simulate the input impedance in ohms from the linear curves, choose 'Zin' and select only the magnitude. Then, generate your graph and measure from Port 1.
- The real part of Zin represents the resistance in ohms, while the imaginary part of Zin shows the reactance, which defines whether the circuit behaves capacitively or inductively (negative or positive reactance). This is where the matching process begins.
- If you measure the Zin for the Tline with only a ground connection, it will show a large value at the center frequency, indicating an open circuit due to the high impedance.
- If you want to control a frequency on the Smith chart, set it as the central frequency on the Tlines, observe its point on the Smith chart, and then move it up and down based on the admittance and impedance plots. Note that using resistors is not preferred, as they introduce losses.
- When starting to match with lumped elements, always begin with the L-network of an inductor (L) and a capacitor (C) together, then adjust it. Notice that increasing the capacitance lowers the impedance since $Z = \frac{j}{\omega C}$.
- After matching on the Zin Smith charts with a complex magnitude, you can create an S11 graph and observe the value at the matched frequency in dB.
- The problem with ideal Tlines is that they are not practical; they don't account for losses, junction connections, or various parasitics.
- To work with real Tlines, you need a new schematic.
- Real Tlines rotate around the center point by a specific amount, unlike LC components.
- After defining the microstrip line (MSline) based on the Txline tool, prepare the substrate for your measurements using the same tool.
- To achieve matching, add series and parallel Tlines and adjust their length and width.
- Simulate a T-shaped Tline on the layout to check the design before fabrication. When connecting more than one Tline with a T-junction, ensure that the widths of the junction sides and the Tlines are consistent. Only change the length. The length should not exceed 20 mm to maintain the dimensions of the PCB after fabrication.
- The junction is called MTEE, and there is another variant called $MTEE$\$, which automatically adjusts the widths if you have different width Tlines.
- For the open-ended Tline, use 'Mopen' to terminate the Tline.
- There is a tool called 'Snap Together' on the toolbar. Just select all Tlines on the layout and press it, and everything will be connected automatically. The tool is located next to the 'Tuner' tool.

- If you want to solve everything in a 3D simulation, you will need an EM simulation, which is quite difficult.

from edaboard

- power transistor have optimum load impedance that maximize either the delivered power or the power gian
- if you connect the the pspice model for thr transistor you can use the IVVURVE tool to mesure the graph for the transistor.just connect the drain and the gate.
- also you could add the models for lumped elements like C or L and R you need to import the model on the net files. **tips**
- you can add series R at the gate to get stable device the output match is enough to use the output capacitor and the bias Ld , Rd with capacitor or without.
- also the gate the bias with Lg and series and parrellel resistor.
- also the matching of the inpupt is the most important mostly done with LC circuits
- as on the figure 2 sreies L and 2 parrellel C with the coupling capacitor. note the capacitors are infront the inductors.
- also the output match is done on the same way but it arranged this way series L parrellel C and then repeated one time.
- on smith you measure S11 and S11*
- tuners helps alot to find the points on smith.
- search about LTuner tool that show smith chart as like boxing the the matching circuit. it is connect after the ports directly means the capacitor are inside it.
- also omn this circuit the inductor and the reistor and are in series for the biasing.
- the port impedance musst be conjugate of the optimum load ot sourse impedence
- there are power matching and conjugate matching

optimzation

- to begin optimization from view menu chose variable browser then mark on optmize foor the varible you want.
- also there constrain option to define min and max values no need for it now
- after you mark to optimize chose the optimzer goal from the project list
- then chose the parametr you want control just S11 or S22 or any other measuremet
- define the value at goal start option
- then define goal type $>$ or $<$ or $=$
- then define the bandwifth you want get this value on it on min and max options
- then you will find a mark on the graph you can move it on the graph and change the optimization goal by mouse
- repeat the process with other curves then see the other marker on the graph for the other otimizer goal .
- then in simulate menu click optimize
- on the menu chose Random local for the optimization method
- Mac iteration means how many times the circuit simulated greater number better result but takes longer

Dc biaing at the end

- when you chose a s parameter file it has a Id and Vds values the bias Vdd is not Vds
- do use Rd which will give you the needed Id and use the Vdd to get the nedded Vds
- using High Rg is nice to don't didturb the measurments for matching.
- use I meter and V meter to meaure these values.
-

From Microwave Labcast Lecture 9
From Microwave Labcast Lecture 9

- Oscillating occurs due to Amplifier + Feedback. Some slides from the video are shown in figs. 1 and 2.

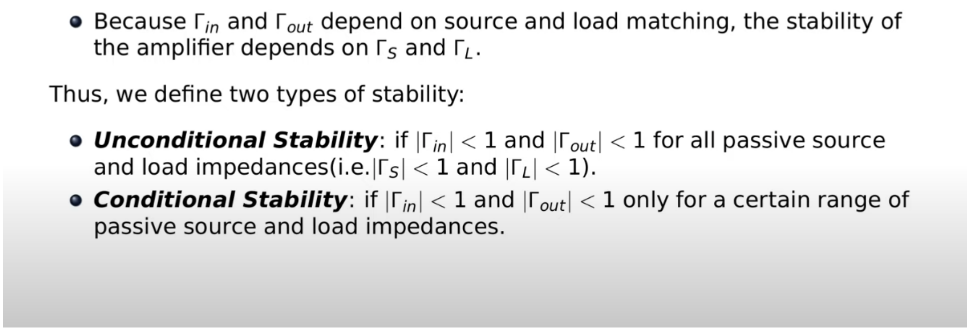

Figure 1: Stability Figure 1

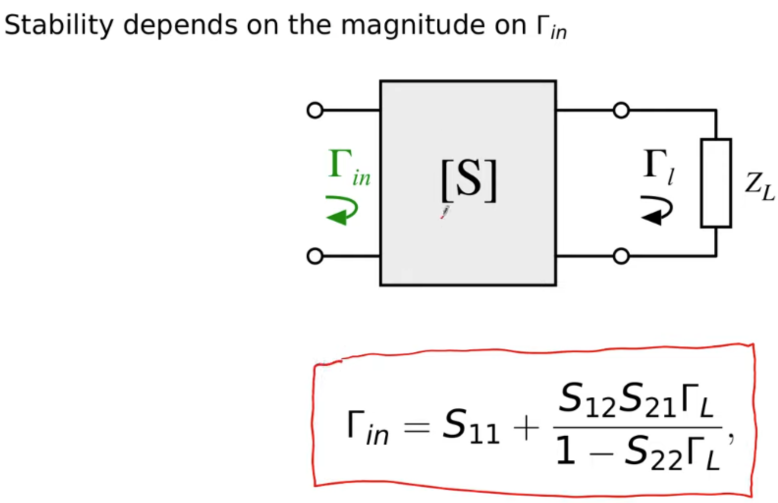

Figure 2: Stability Figure 2

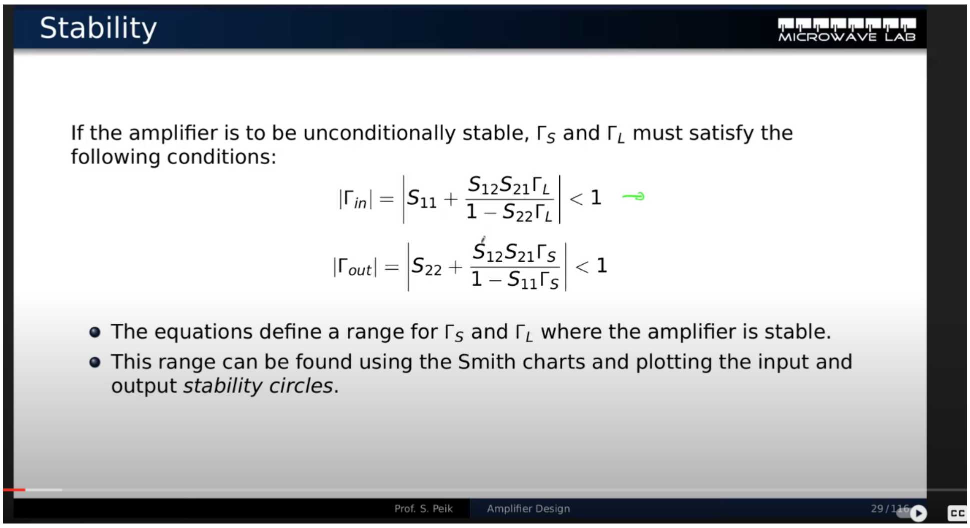

Figure 3: Stability Figure 3

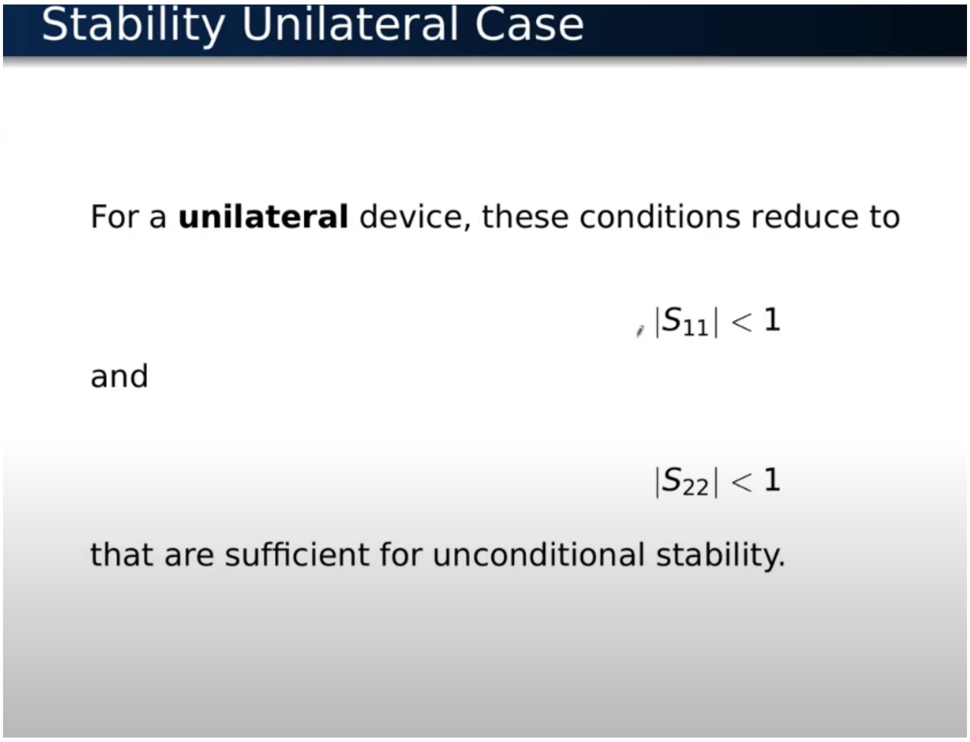

Figure 4: Stabiliy for transistor and amplifier

- after the derivation for the Gamma in aout you get this for the C and R C is the center of complex numbers and R is the radius for the real numbers

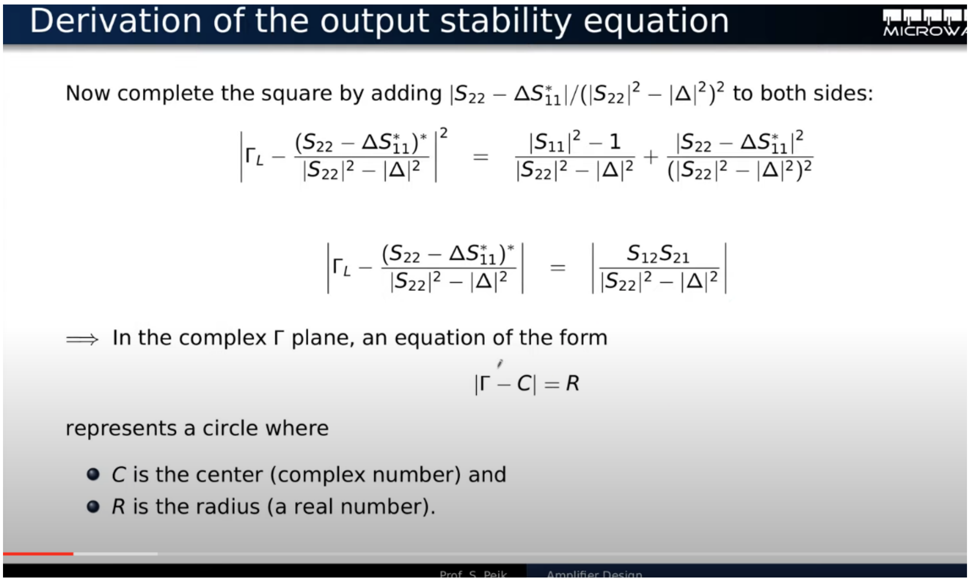

Figure 5: Stabiliy for transistor and amplifier

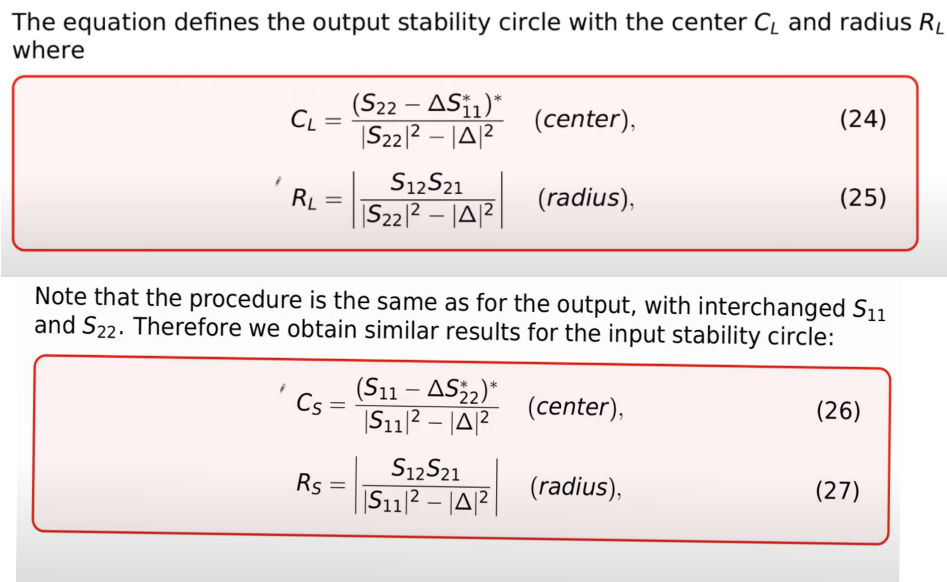

Figure 6: Stabiliy for transistor and amplifier

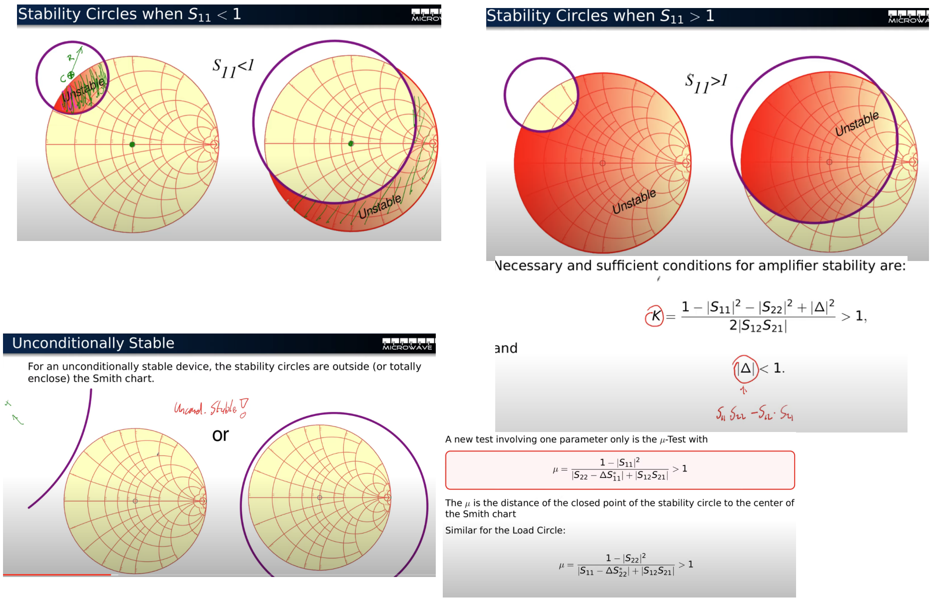

Figure 7: Stabiliy for transistor and amplifier

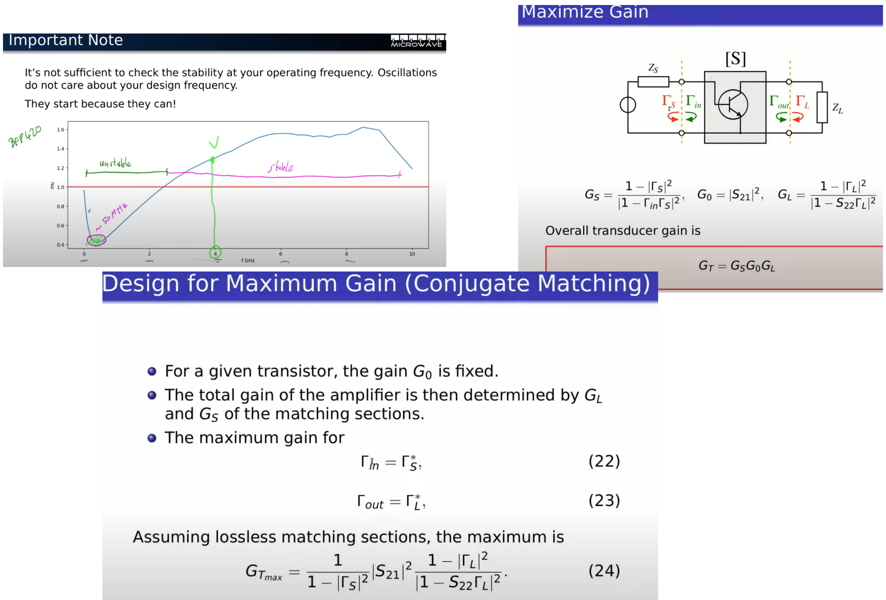

Figure 8: Stabiliy for transistor and amplifier

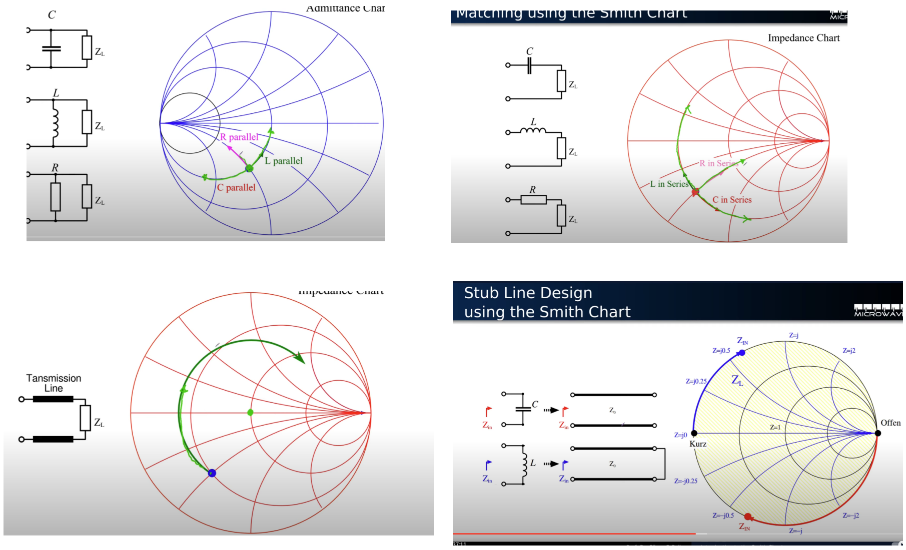

Figure 9: Stabiliy for transistor and amplifier

figures/stability 10.pdf

Figure 10: Stabiliy for transistor and amplifier

What to do to design

- stability ba K and delta or SCIR$_IJ(1,2)andSCIR_IJ(2,1)onsmithtobesurethatallcirclesoutsidethesmithmeansunc$

stability wit the SCIR1 and SCIR2 for 22 and 11 should the spesified freq circle should be outside the the smith plot. all stability circles should be outside. the shunt resistor takes the circles outside to the open side right side of smith the series resistor takes the circles outside of smith from the left side or the short circuit side.

stability with MU1 ans MU2 and chose port 22 for the two values for the output stability A also this repeated with thr inpupt match. don't chose a unit leave it normalised . MU $>$ 1 for the stability with MU test. MU is the distance of the closed point of the stability circle to the center of the smith chart if MU $<$ 1 the design will be potentialy Instability.

stability with K>1 and delta <1 I think also the it works for 11 and 22 stabilize the design could reduce the gain also.

for the stability test add to the transistor inpur and output capacitors and the L shocks inductor on the line for the biasing circuit. then amke the stability test define the s parameter simulatioon from the points for the freq range for the transistor

like this circuit on fig. 11

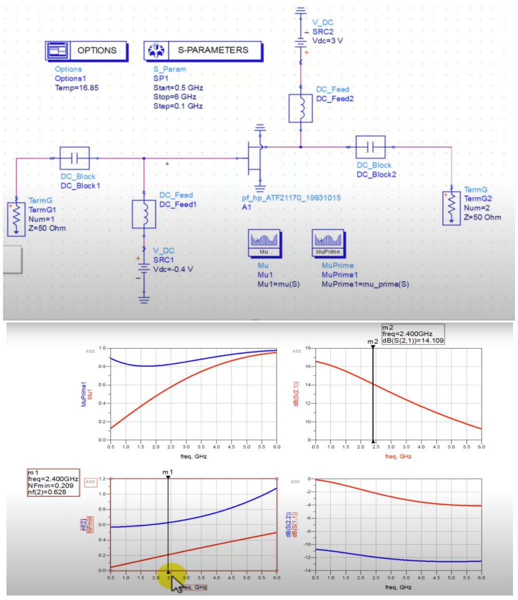

Figure 11: Stabiliy for transistor and amplifier

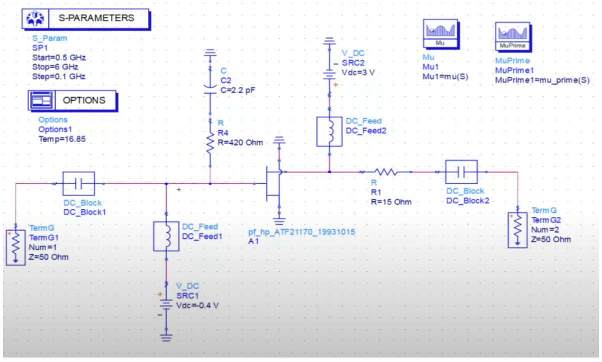

Figure 12: Stabiliy for transistor and amplifier

so get the stability points vs the the noise value. add series R at the in and out of the transistor dierctly like the R0 but use higher values.

avoid any noise components at the lna input which could lead to higher NF so the R0 for the stability could be at the output side only to avoid noise but the resistor the lower the better

adding the Vds = Vdd for the bias point fot s2p file and also with Vgs

if the design is slightly stable or there value is lower than 1 by small value it could be solved during matching this is called potential stable but unconditional stability is preferred this for the unspecfied freq range but for the BW it must be unconditional stable als using the feedback is food for the stability and the shunt RC line which is on input side this also can improve the stability but R shunt should be as high as possible for provide a load for the gate of the device the capcitor shuct is blocks the DC from going to the ground throug the resistor with the stability

Anurag RF design 9

- some notes about the noise
- LNA Theory Fundamentals - Quick Review

  What is a Low Noise Amplifier?

  A low-noise amplifier (LNA) is an electronic amplifier that amplifies a very low-power signal without significantly degrading its signal-to-noise ratio. An amplifier will increase the ower of both the signal and the noise present at its input, but the amplifier will also introduce some additional noise. LNAs are designed to minimize that additional noise. V Designers can minimize additional noise by choosing low-noise components, operating points, and circuit topologies. v Minimizing additional noise must balance with other design goals such as power gain impedance matching. An LNA is a key component at the front-end of a radio receiver circuit to help reduce unwanted noise in particular. In most receivers, the overall NF is dominated by the first few stages of the RF front end.

  By using an LNA close to the signal source, the effect of noise from subsequent stages of the receive chain in the circuit is reduced by the signal gain created by the LNA, while the noise created by the LNA itself is injected directly into the received signal.

  The LNA boosts the desired signals' power while adding as little noise and distortion as possible. The work done by the LNA enables optimum retrieval of the desired signal in the later stages of the system.

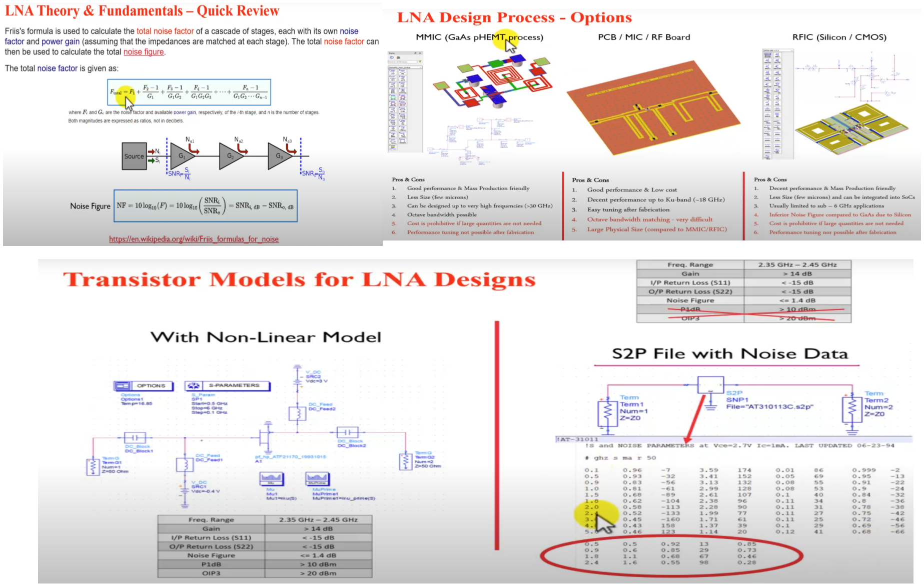

Figure 13: Stabiliy for transistor and amplifier

-
- if you are luky you will find the Nonlinear model of your transistor and if not you will have the S2p file that has noise parameter at the end of the file and mention the bias point and you will not be able to simulate the Nonlinear points p1dB and oip3
- with thr nonlinear model first thinf add biasing for the transistor and get IV curves and check the point you look for it which should give as low as possible power consumption which is defined by Vds * Id if with s parametr you can't so that as you have s parameter which is extracted at a particular bias point . with s parameter start from the stability point step 2

With the dielectric function ϵ, which describes the response of the material to external electromagnetic fields. $\epsilon = \epsilon_1 + i\epsilon_2$ is a complex function, which will in general depend strongly on the frequency of $\vec{E}$ and also on the direction, if the material is anisotropic. The sometimes more intuitive complex refractive index $\tilde{n}$ can be gained from ϵ throug [?]:

$$\sqrt{\epsilon} = \tilde{n} = n + i\kappa \tag{1}$$

6.1 Key Paper 1

Discuss the first key paper. [1]

6.2 Key Paper 2

Discuss the second key paper. [2]

7 Conclusion

Summarize the key points discussed in the tutorial and their implications.

References

[1] A. Author. Title of the key paper 1. *Journal Name*, 10(2):123–456, 2021.

[2] B. Author. Title of the key paper 2. *Journal Name*, 11(3):789–1011, 2022.