Study Notes for AWR Tutorial

Your Name

July 30, 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

$$z_o = 50 ohm and Z_s = 50 ohm so Z_i the normalized impedance is 1 means full matched$$

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$_meterandalsoImeterImeteronseriesafterportandvmeteronshunt.fromthegraphrect$

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
Final Design

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.