

# Experiment 10 - Power Amplifier Measurements Using Vector Network Analyzer

## 1 Introduction

This experiment contains two portions: measurement and simulation of nonlinear characteristics of a power amplifier. You will learn how to measure the key figure-of-merit (FOM) of a power amplifier using the vector network analyzer (VNA) and prove the limitation of S-parameters in characterizing nonlinear device. We will be using an evaluation board (TB-409-1+) for a monolithic power amplifier (Gali-1+) in this experiment. Data sheets provide information about the PA and the evaluation board are both available on the web course.

In the first part, you will measure the S-parameters of the amplifier module to verify the small-signal parameters match with the data-sheet. Additionally, you will use ADS simulation to prove the limitation of S-parameters in nonlinear device measurements. In the second part, you will perform a power-sweep on the VNA to characterize the 1-dB compression point for the amplifier and simulate the nonlinear behavior using the Harmonic Balance simulator in ADS. Lastly, you will compare the simulated 1-dB compression point with the measured data and check if there is an agreement between those two.

## 2 Background

Over the years, the wireless communication market has experienced remarkable development since the first handheld phone introduced by Motorola. Nowadays, the smartphone is one of the most indispensable personal items, with a wide variety of applications that can benefit our daily life. Consequently, the thirst to achieving better wireless transceiving system design, with a low market cost, has gradually become the primary goal of modern RFIC manufacturers.

In all RF transceivers, the power amplifier (PA) plays a key role in driving the antenna on the transmitting end, while low-noise amplifier boosts the receiving end signal. Together these components account for the basic operation of a duplexed system. Among the various requirements in designing a PA, efficiency and linearity are the two most important characteristics. In the modern RF industry, engineers are sparing no effort to increase the PAE in order to increase the battery life; however, linearity requirements, such as ACLR for W-CDMA, E-UTRA for LTE, and ACPR for CDMA2K, must be obtained in order to achieve the basic PA functionality [1].

Typically, we design PAs to operate only within their linear region of operation but that results in an inefficient use of the available power. PA design is different than traditional microwave amplifier design as the goal is not to simply use simultaneous conjugate for max power, instead the designer has to satisfy figures-of-merit like power added efficiency (PAE) and TOI points that dictate the level of non-linearity in the amplifier. Since, PAs operate under large signal they are often driven into the non-linear regime and thus characterization of the nonlinear behavior is of utmost importance to the RF engineer.

## 3 Pre-lab

1. Refer to the data sheet of the PA used in this experiment, determine the maximum operating current that the PA could sustain?
2. Refer to the data sheet of the evaluation board, determine the power supply voltage required for the PA?
3. What is the physical significance of 1 dB compression point? Why is it a useful metric to characterize non-linear behavior of RF circuits?
4. What is the physical significance of PAE? Write down the equations to compute it.

5. What is Harmonic Balance? How is it different than traditional SPICE simulation? List some advantages for using Harmonic Balance for studying RF circuits/systems and mention some common potential problems that could arise when using this simulation engine.
6. Briefly talk about PA nonlinear properties.

## 4 Equipment

- Agilent E8358A VNA.
- Agilent 85052D 3.5mm SOLT calibration kit.
- 3.5mm cables.
- N-type to 3.5mm adaptors.
- Mini Circuit Gali+1 PA.

## 5 Procedure

### Part I - Small-Signal S-Parameter Measurement Procedure

1. Set up equipments as shown in Figure 1. Make sure the DC power source shows “OUTPUT OFF” during your set up. **Important:** Set the *current limit* according to the value you found in question 1, *pre-lab*. This is to make sure the DC supplier will never send out a current that is high enough to destroy the PA. If you couldn’t figure it out, ask your TA. Turn the DC source on by pressing “Output On/Off” before measuring the amplifier so that we measure an operating amplifier, not an unbiased one.

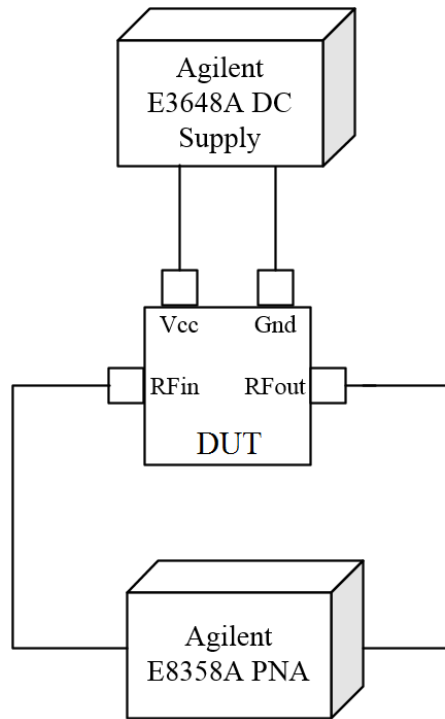


Figure 1: Vector Network Analyzer experiment setup diagram

2. As always, preset the VNA to the default state by going to System > Preset.

3. Set the frequency range from 600 MHz to 8 GHz. Choose 1601 frequency points. Remember to calibrate the VNA.
4. Connect port 1 of the PNA to the  $RF_{in}$  of the amplifier and port 2 of the PNA to  $RF_{out}$  of the amplifier.
5. To save data select File > Save As. Find a directory on your personal drive, give the file the appropriate name and select “Trace (\*.s2p)” as your Save as Type. Sample name can be “PA\_SmallSig.s2p”.
6. Open ADS, create a new workspace and import the measured data in, show all 4 parameters (both on Smith Chart and in Magitude scale).
7. Put markers on  $S_{21}$  as well as  $S_{11}$  for each of the points listed in the data-sheet to compare measured data with manufacturer’s data.

## Part II - Power sweep simulation using Harmonic Balance

In this section, we will implement a simple ADS environment for running power sweep simulations.

1. Create new workspace in ADS and call it ECE451\_lab10.
2. Create new schematic and call it Power\_sweep.
3. We will use the above .s2p file to represent the PA in this simulation. In ADS type S2P in the component search window above the upper left palette.
4. Now in the upper left pallet select Simulation-HB  $\Rightarrow$  HB.
5. Then select Simulation-HB  $\Rightarrow$  PrmSwp
6. In the upper left pallet select Sources-Freq Domain  $\Rightarrow$  P\_nTone.
7. Your schematic should look like the following as shown in Figure 2

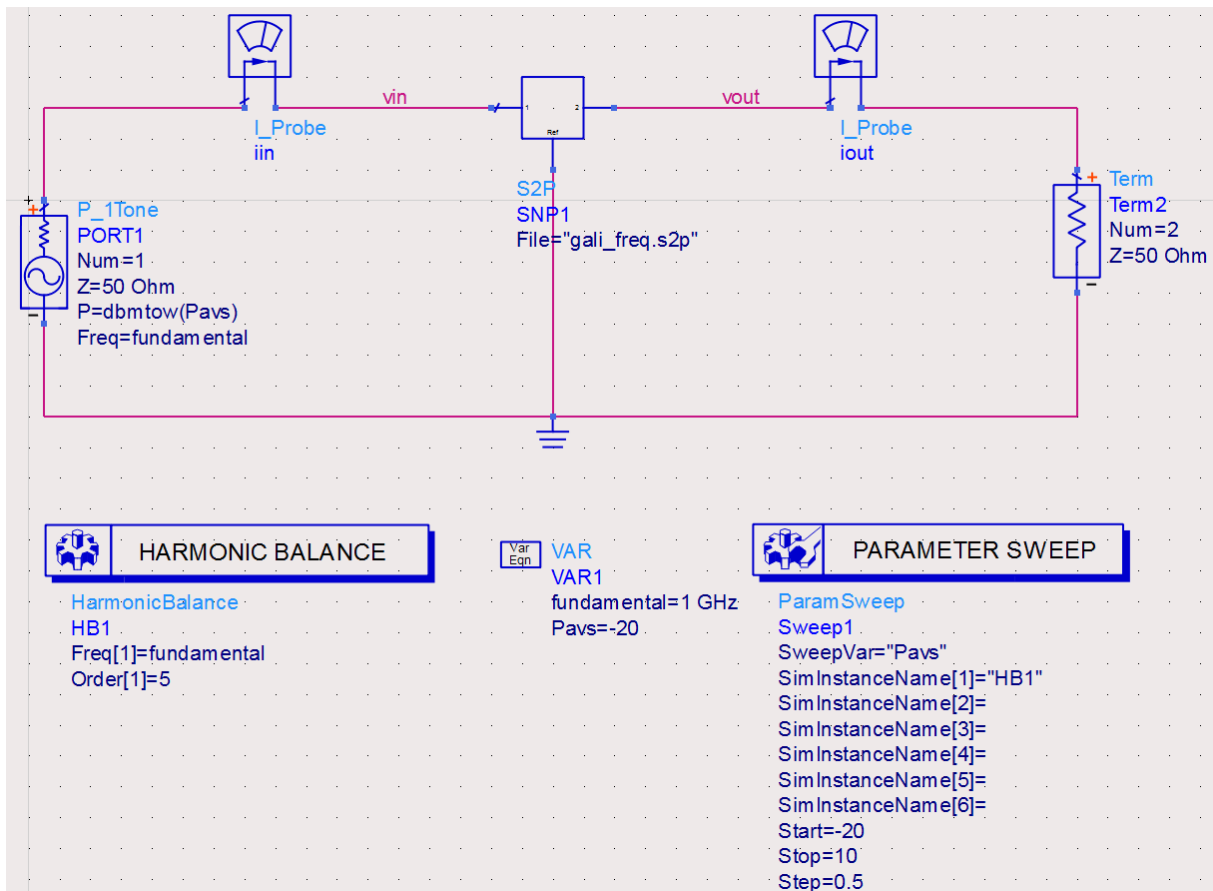


Figure 2: Harmonic Balance Simulation Schematic

8. Add equations as shown in Figure 3.

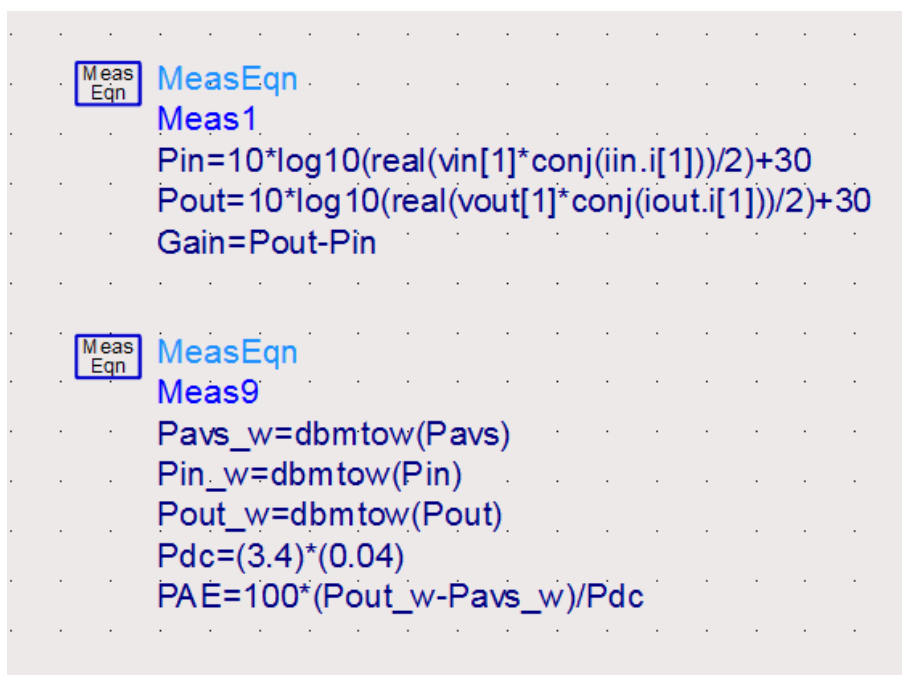


Figure 3: Necessary Equations for HB Simulation

- Before Simulation, go to Simulate ⇒ Simulate Settings from top menu and change the Dataset name to Power\_sweep\_Spara as shown in Figure 4

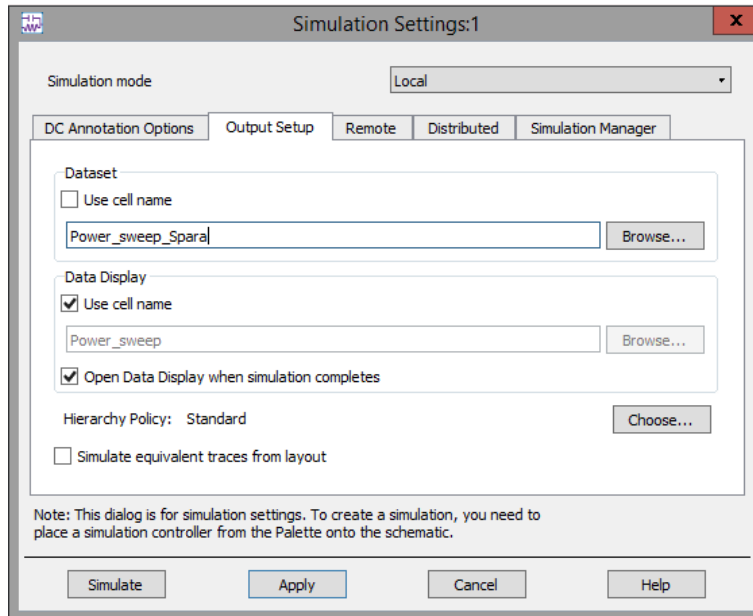


Figure 4: Simulation Settings

- Plot the Gain in ADS data display (DDS) window and change the Y-axis scale from auto to 10–13 with a 0.5 interval.
- Your Gain plot should be a straight line if everything was performed correctly. In the next part of this experiment, we will verify this result by direct measurement the VNA.

### Part III - 1-dB Gain Compression measurement

- Prepare the VNA for a Power Sweep:
  - Set the network analyzer to perform a power sweep by selecting “Sweep > Sweep Type” and choosing “Power Sweep” as the Sweep Type.
  - Set the frequency and power sweep range as in the simulation in the previous part.
  - As always, perform a 2-port SOLT calibration before your measurement.
- Set up the Amplifier:
  - Set up the PA as in Figure 1.
  - Now turn on the output of the power supply by pushing the “Output on/off” button and look at the VNA output. Right click using the mouse on the VNA screen and select AUTO SCALE. Plot the  $S_{21}$  and your PNA image should look similar to Figure 5



Figure 5:  $P_{out}$  vs.  $P_{in}$

3. Measure your 1 dB gain compression point:

- (a) View the log magnitude of S21 on the VNA. Set to markers to measure the 1 dB gain compression input power. You can add multiple markers by selecting “Marker-Select Marker” choosing the marker you want to select. Use one marker to indicate the max of Pout and the other marker to find the input power that gives a -1 dB output power. Record this 1dB gain compression Pout and Pin.
- (b) Save the data file as “PA\_Power\_Sweep.s2p”

## Part IV - Comparison Between S-parameters and Power Sweep Data Using ADS

1. Before importing “PA\_Power\_Sweep.s2p” data to the ADS, we need to make a small change to the .snp file. Change the dBm unit to Hz (because ADS can only read this unit), and delete Power Sweep after the semicolon as shown in Figure 6.

```
!Agilent Technologies,E8358A,US42080660,A.06.04.32
!Agilent E8358A: A.06.04.32
!Date: Wednesday, April 05, 2017 14:08:36
!Correction: S11(Full 2 Port(1,2)) S21(Full 2 Port(1,2)) S12(Full 2 Port(1,2)) S22(Full 2 Port(1,2))
!S2P File: Measurements: S11, S21, S12, S22: Power Sweep
# dBm S dB R 50
-20 -2.300681e+001 -1.400515e+002 1.196243e+001 7.547932e+001 -1.720868e+001 -8.886793e+001 -2.428563e+001 9.479536e+001
  Hz
```

Figure 6: A simple modification to import power sweep .snp file to ADS

2. Now you can import your snp file to the data display window from last section (in my case: Power\_sweep.dds)
3. Plot the new  $S_{21}$  to Gain plot
4. Your plot should look similar to Figure 7. Now, clearly, amplifier gain of small signal S-parameters measurement is linear throughout the sweeping power range; however, amplifier gain starts to compress at -5 dBm in the large signal power sweep measurement. This phenomenon is due to the nonlinearity nature of power amplifier. Therefore, to characterize the behavior of power amplifier, especially at relatively high input/output power, a S-parameters model is not enough. In next lab, we will introduce another parameter called *X-parameters*, which is a large signal parameter meant to characterize nonlinearity.

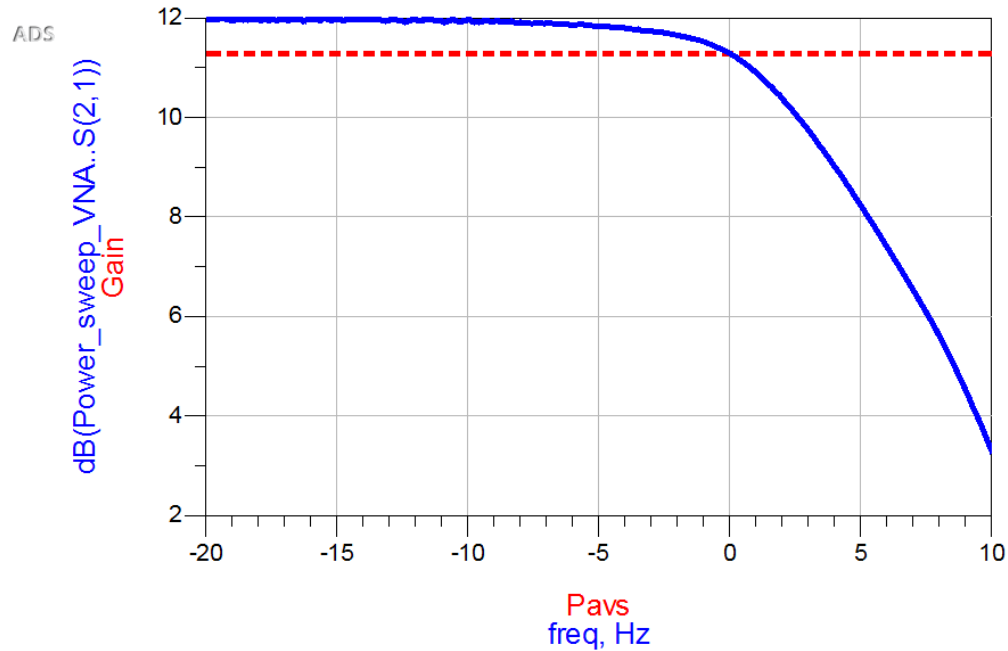


Figure 7: A Comparison Between Small and Large Signal Measurement

5. In the same data display window, plot the PAE and the output power of the VNA measurement by using the equations in Figure 8. Make sure to use the correct dataset file name in equations (in here powersweep.. should be replaced).

$$\text{Eqn } P_{\text{out}} = \text{powersweep}..freq + \text{dB}(\text{powersweep}..S(2,1))$$

$$\text{Eqn } \text{PAE} = 100 * (\text{dbmtow}(P_{\text{out}}) - \text{dbmtow}(\text{powersweep}..freq)) / (3.4 * 0.04)$$

Figure 8: Equations to Write in Data Display Window

## Part V - Harmonic Balance Simulation Using Circuit Model in ADS

In this section, you will use the circuit model of the PA to perform a HB simulation and compare it with your measured power sweep data from VNA.

1. Re-open your S-parameters simulation schematic and change the S2P component to the circuit symbol (with a 12 V dc source) provided on the website as shown in Figure 9.

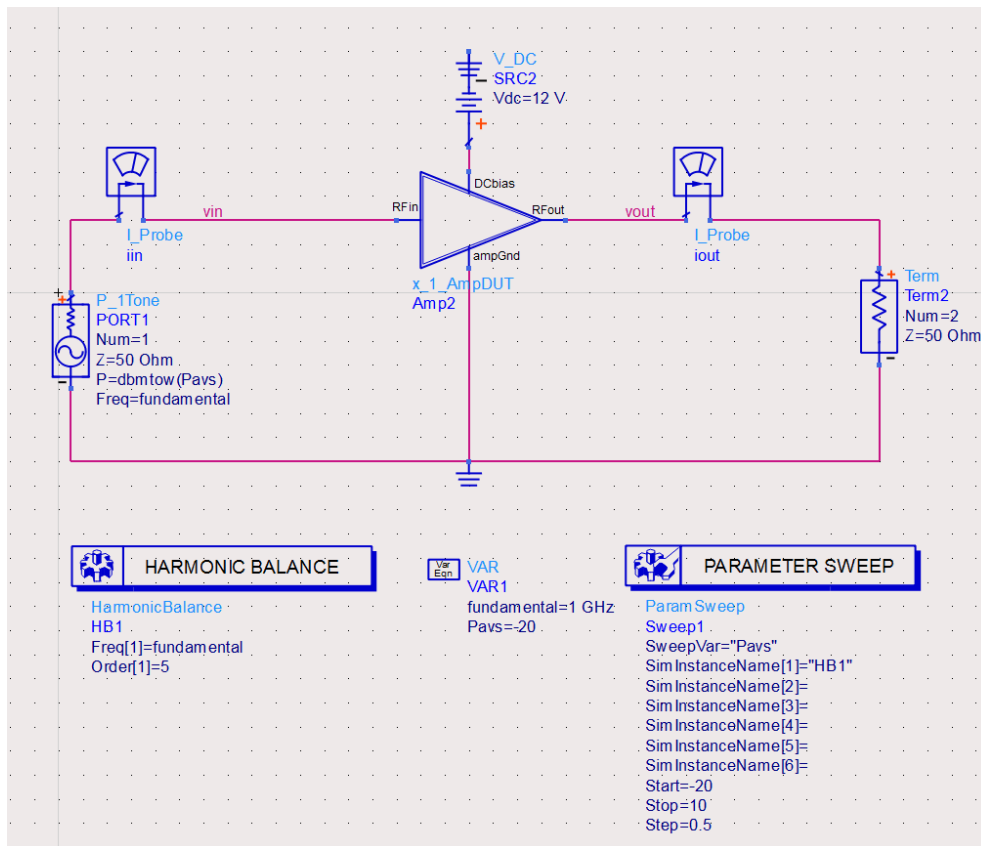


Figure 9: Harmonic Balance Simulation Schematic with Circuit Model

- Before running the simulation, go to Simulate ⇒ Simulate Settings from top menu and change the Dataset name to Power\_sweep\_Ckt as shown in Figure 10

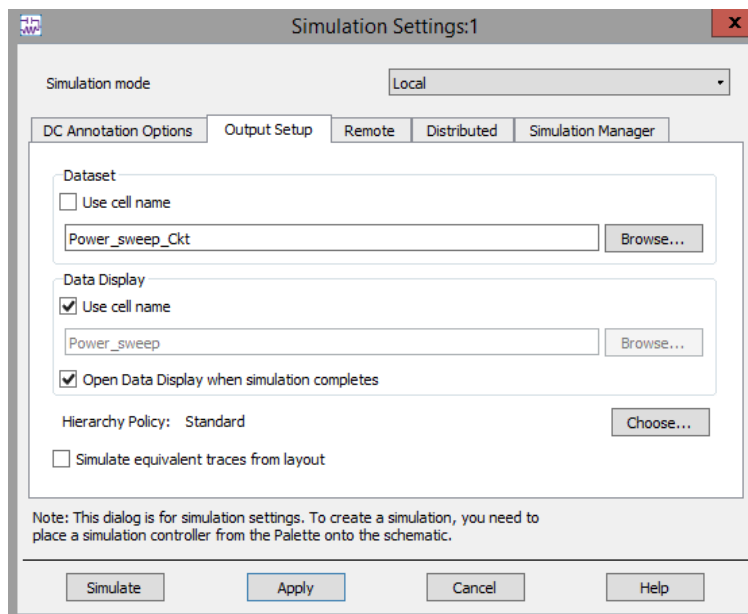


Figure 10: Simulation Settings

- After the simulation, plot Gain from VNA power sweep measurement and circuit model HB simulation on



the same graph and compare. Note that you need to choose the correct dataset from the drop down menu. Your plot should be similar to Figure 11.

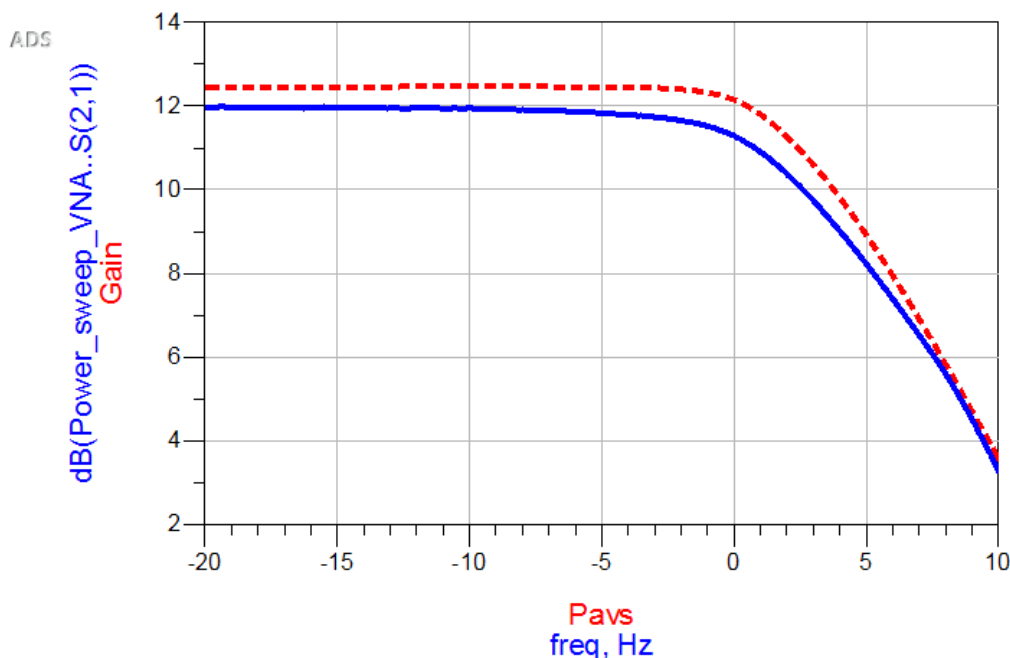


Figure 11: Gain Comparison Between Circuit Model Simulation (Red Dot) and Power Sweep Measurement (Blue line)

4. Do the same for plotting Pout and PAE.

## 6 Conclusion

1. Compare your S-parameters ( $S_{11}$ ,  $S_{21}$  and  $S_{22}$ ) small signal measurements with manufacturer's data-sheet. What may cause discrepancies? Compare all frequencies listed on datasheet.
2. Why are we not able to simulate the P1dB point from the S-parameter data measured on the VNA using Harmonic Balance simulation?
3. Can S-parameters help characterize non-linear behavior in amplifiers? If not, what potential shortcomings do you see in the S-parameter formalism?

## References

- [1] Z. Li, "Characterization of various types of power amplifiers," Master's thesis, University of Illinois at Urbana-Champaign, Champaign, IL, May 2017. 2