Getting Started with AWR Design Environment (part 3)

We can create and simulate an antenna. Ho hum. (Just kidding, it's totally awesome that we can do that!) Now, we're going to also simulate other circuits and try to get the antenna and circuit to interact. Muahaha.

Away we go!

1) Launch AWR again and either open your Folded Rickroll project, or re-create the tutorial from the last couple of weeks. If you have your antenna ready, just use that.

2) We need some files, first. The circuit we will be putting on our boards is made up of some diodes. Specifically, Schottky diodes from Avago designed to work out higher frequencies. Download the spice file from their website. We're using the Avago HSMS285x series.

   http://www.avagotech.com/products/wireless/diodes/schottky/hsms-2852#documentation

   (it should be near the bottom. You can get the S parameters file if you want to try that, but we'll stick with the spice netlist for now).

3) Save the netlist somewhere convenient.

4) Add the netlist to AWR.

   Project > Add Netlist > Import Netlist

   You will need to look for "all files" since this is .psp. Just choose PSPICE as the format, and it should go through fine.

```
! NOTE: This file was translated from a SPICE file.
!
! SPICE orders the nodes of active devices as Drain Gate Source.
! The AWR convention is Gate Drain Source. The translator has reordered
! the nodes for active device elements (B, J, M and Q). The user should
! verify the node order for subcircuits representing active devices.
!
! Polynomial controlled sources have been decomposed to linear devices
! when possible.  The user should verify the use of these devices.
!
! SPICE model for HSMS-285x
! The parameters are for a single diode (HSMS-2850). Parameters also apply
! to the individual diodes within multiple diode configurations.
!
DIM
    TEMP   C
    FREQ   HZ
    RES    OH
    COND   /OH
    IND    H
    CAP    F
    LNG    M
    TIME   SEC
    ANG    DEG
    VOL    V
    CUR    A
    PWR    DBM

CKT
SDIODE 1 2 ID="DCD1" IS=3e-006 CJ0=1.8e-013 VJ=0.35 BV=3.8 IBV=0.0003 &
    EG=0.69 N=1.06 RS=25 XTI=2 M=0.5
DEF2P 1 2 "hsms"
```

5) This is the netlist for a schottky diode. When we use it in a circuit, it will look like a box. Port 1 will be the anode and port 2 will be the cathode.

   Make a new schematic. Right click on "Circuit Schematics" and choose New Schematic. Name it PowerHarvest. This will be the power harvesting circuit that will convert RF to DC.

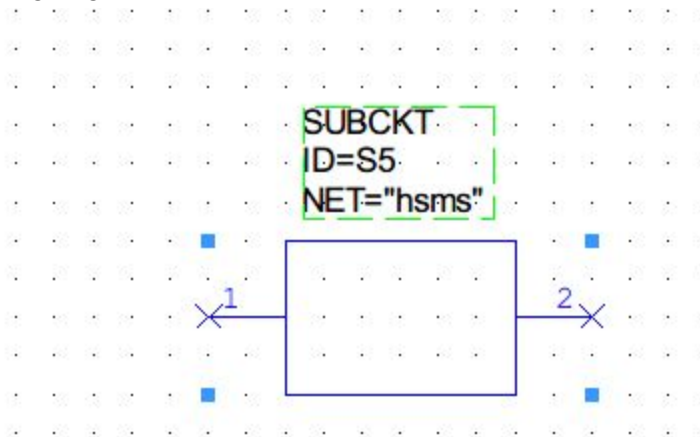6) Add in the HSMS285x series diode. To do this, from your new schematic, click on "Elements"
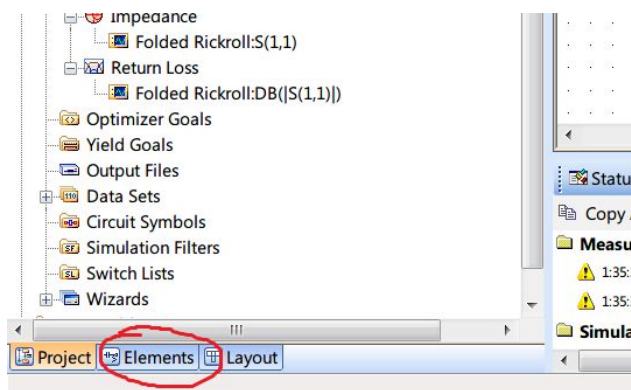
And select "Subcircuits" from the top window and "hsms" from the bottom window.
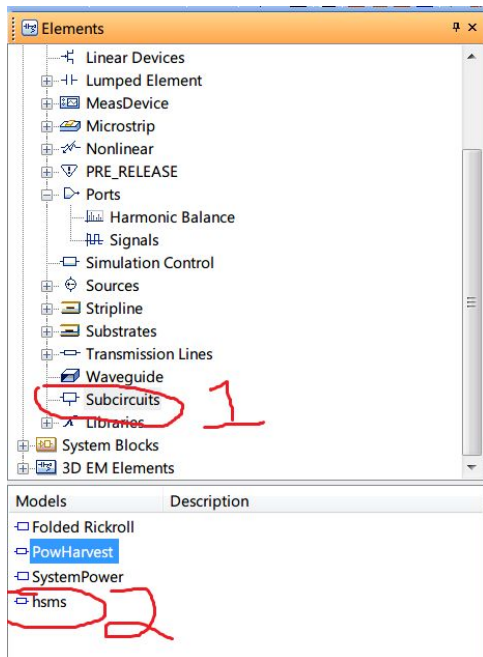
You can then click and drag the hsms element to your schematic.

7) It's going to look like a box.

8) Again, port 1 is the anode and port 2 is the cathode.

9) Draw the following circuit. Use the hsms subcircuit, the lumped element capacitor (just the basic one is fine), the port symbol, and the ground symbol. The last two come from the toolbar. When setting the capacitor value, you may need to go to the Project Options and change the global capacitor units to be pF instead of microFarads.

10) My final result looks like this:
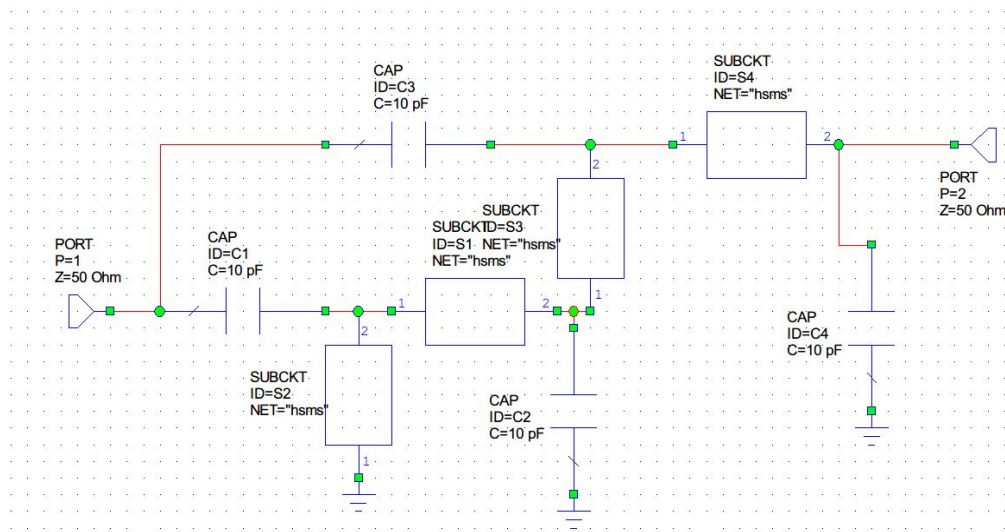
11) Now we will add this to yet another circuit. Create a new schematic called "SystemPower." Add in the "PowerHarvest" subcircuit.
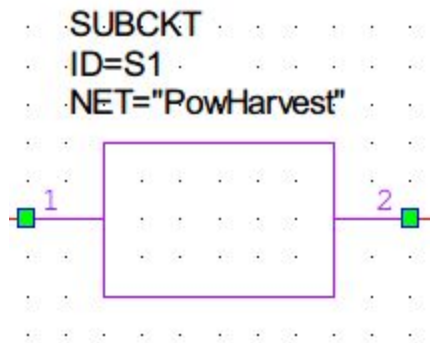

Note that Ports 1 and 2 correspond to the ports you added in the previous schematic.

12) Add the following elements: A "Harmonic Balance" from the "Ports" category -- The Port1 "Port with 1-tone HB Source". Harmonic balance is a name for a simulation that can handle nonlinear devices like diodes. Most Spice simulations are crippled by diodes. Even computers hate quantum mechanics. Connect this to port 1. Note the impedance is 50 ohms and the power is currently 0 dBm.

Add in a "Lumped Element" > Resistor > Load connected to port 2.

Add in a probe and have it connect to the **green probe node** next to the load. This probe CANNOT read voltage from the middle of a wire. The Measurement Probe is next to the Port icon.
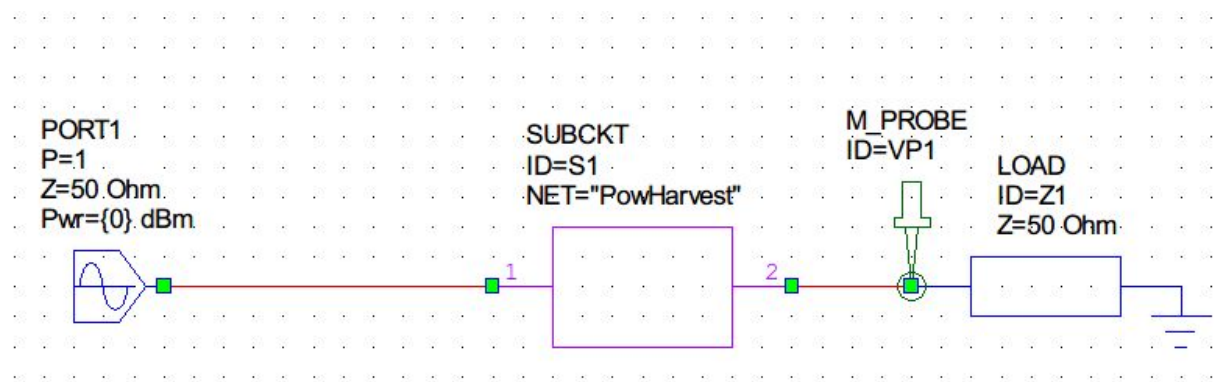
13) Now we can simulate this and get some data.

14) Create a new graph and call it Output Voltage. It's going to be a Rectangular plot.

15) Add the following measurement:
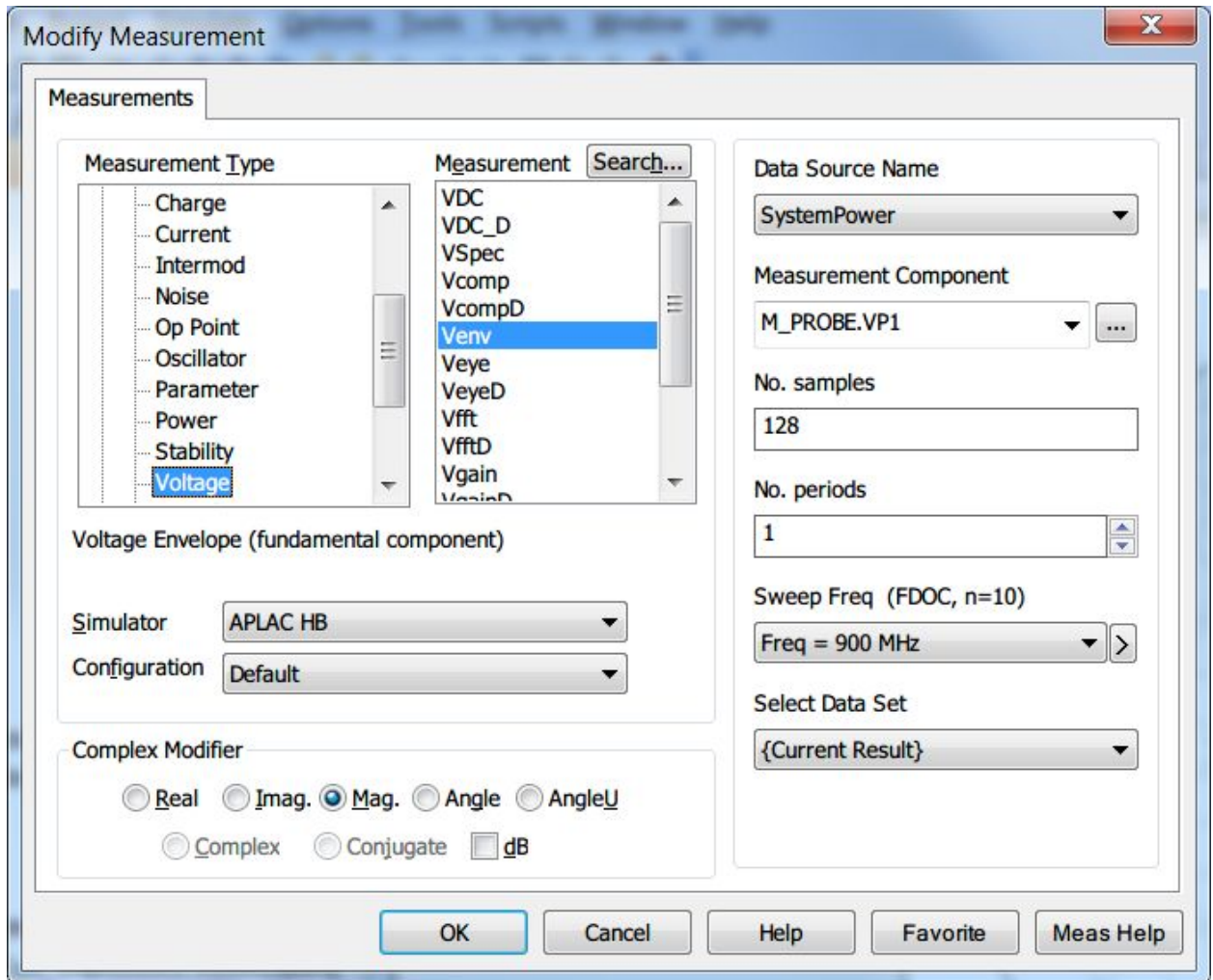
From Measurement Type, select Nonlinear > Voltage. For Measurement, select "Venv" The voltage envelope. Select the appropriate Data Source name (your top-most schematic) and the measurement component to be the probe you added. I would also suggest selecting a frequency close to your center frequency. For complex modifier, plot this as the Magnitude.

16) When you simulate, you should see voltage versus time. Mine hovers around 0.02 Volts. Which is basically abysmal. Wretched even. Or is it ratchet? I have no idea what's on fleec with y'all these days. That's right, I speak middle school. Boom.

17) So, let's investigate.

18) Add a new graph. Smith chart style. Texas Tea. (Ignore the last sentence, that's just your professor slowly losing it). Call it "Circuit S11"

19) For the measurement you add, create a "Linear", "Port Parameters" measurement type, set it to S, referenced from port 1 to port 1 of the SystemPower schematic. We can sweep in frequency, and its gonna be complex.

20) Run your simualation. What's your impedance point? You can add a marker annd have it display the value.
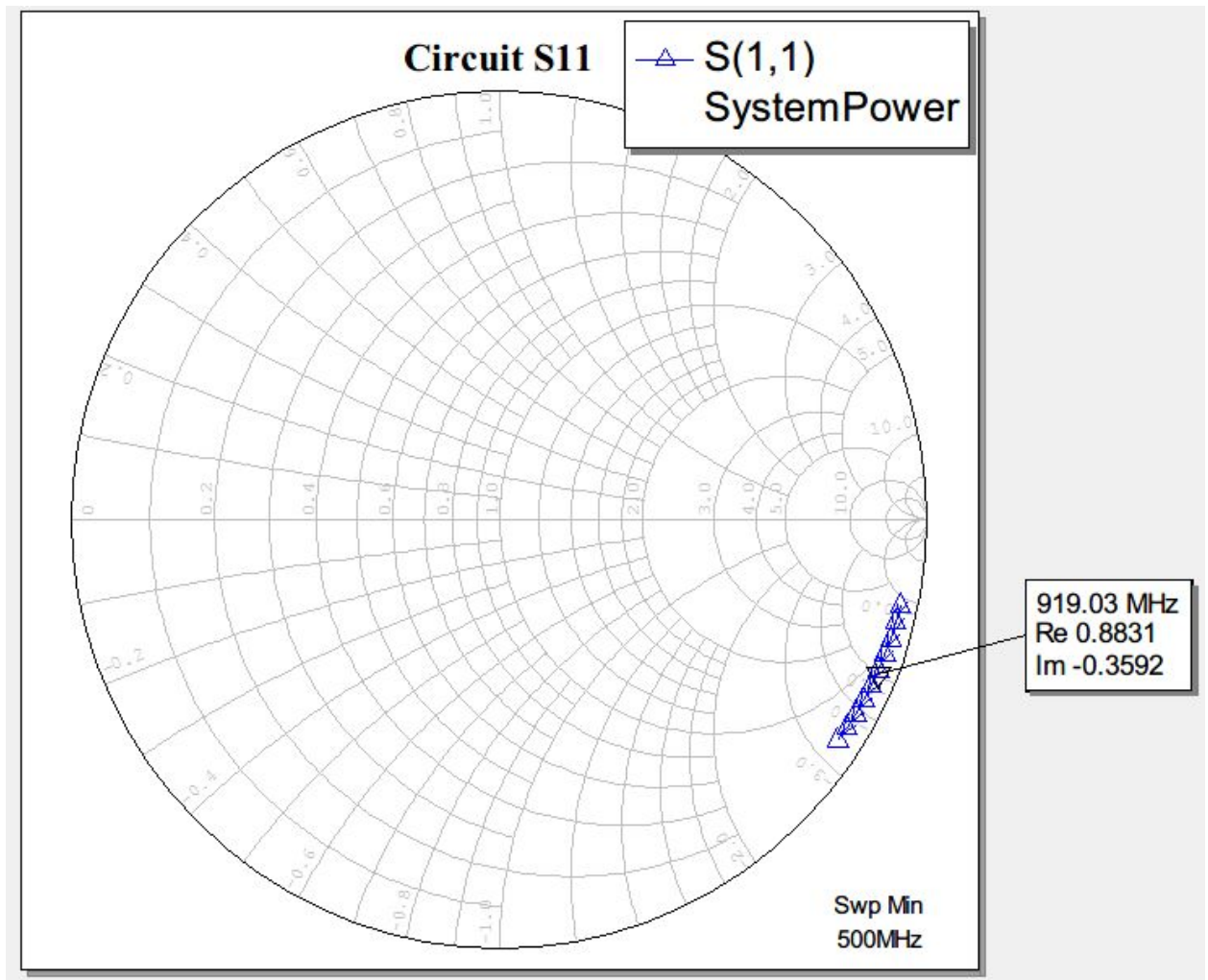
I set a marker close to 915 MHz, edited the marker to display the data point as a reflection coefficient in Real-Imaginary. However, you can also set it to display as an impedance "Denormalized to" 50 Ohms to see an actual Z value.

919.03 MHz
r 31.8992 Ohm
x -251.756 Ohm
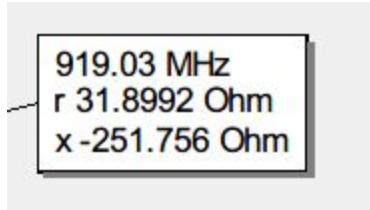
21) So that seems to be a problem. Match that circuit to 50 ohms by adding in lumped element components to your top level schematic, PowHarvest.

Then (You'll have to do this on paper unless you can figure out how to get AWR to plot Voltage versus Power) get some voltage points from between -30 to 10 dBm. And create voltage versus power plot.

What happens if you add a big (10 uF) capacitor in parallel with your load resitstance?

Your goal should be to have a plot of voltage versus power.