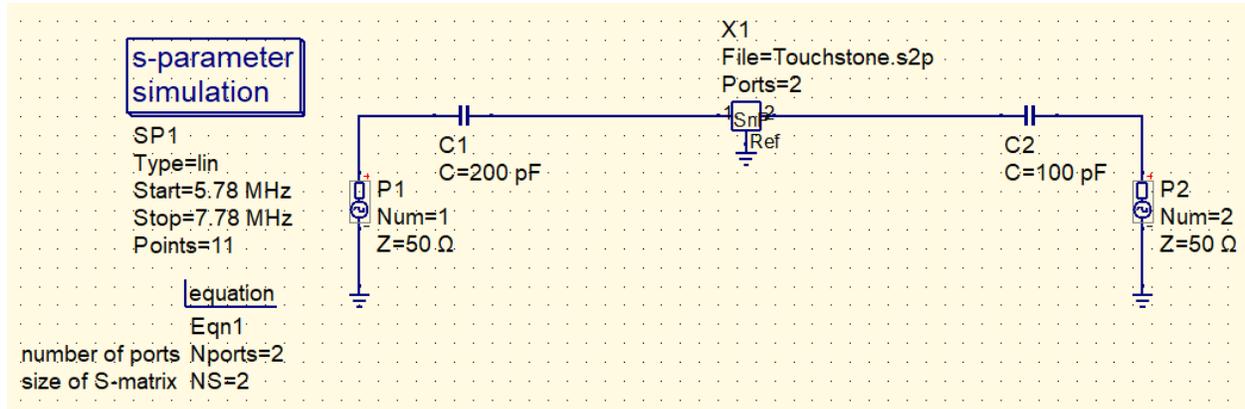
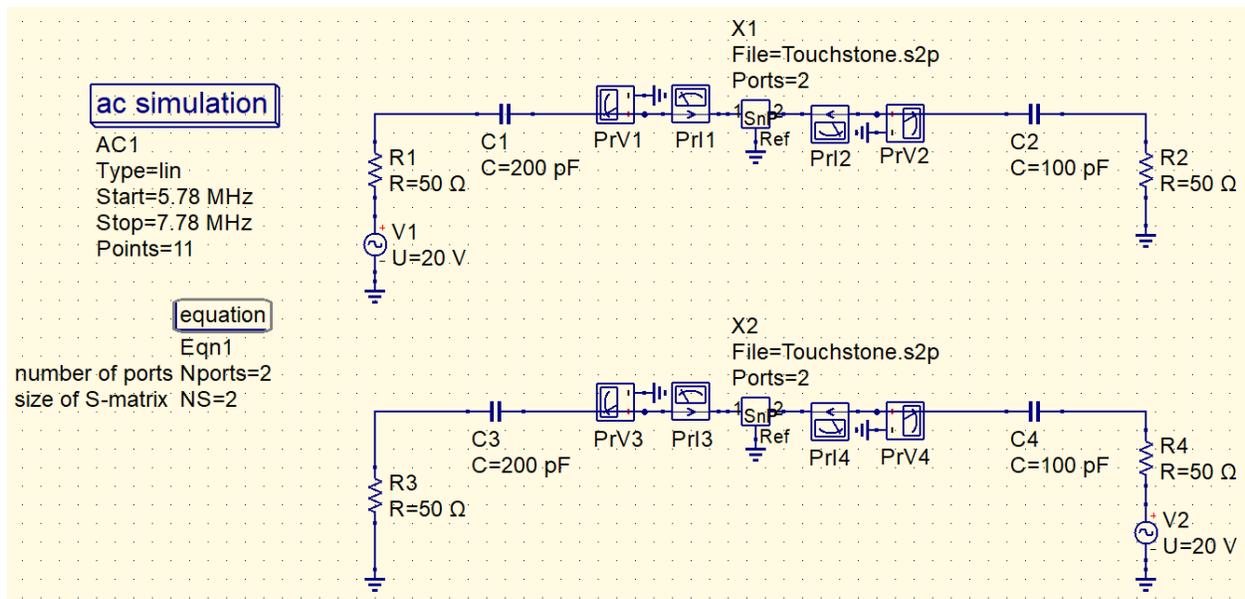


We have a two-port antenna phased array. We simulate it in full-wave software such as HFSS or in CST and save results as 2x2 S-matrix, as Touchstone S2P file. We now want to figure out the matching parameters for each of the two ports of the antenna array. We run the simulation in QUCS and find the capacitor values (C1 and C2).



The question that we face is as follows. If we input 1 W of power (which is  $\sqrt{2}$  input “a” wave) into P1 port, what would be the input power into both ports of 2-port X1 antenna array? Same question is when we input 1W into P2 port. We need to know these values, because we later use the input waves flowing into antenna array to calculate with ohmic power dissipated in antenna array, radiated electric and magnetic field. Input power waves serve as input provided as excitation in HFSS or CST.

So, at the moment, there is no direct way in QUCS to measure power waves “a” and “b” flowing into the X1 antenna array. I came up with this workaround. I created this AC simulation, where the circuit is copied two times:



Driving a circuit with 20V and 50 Ohm series resistor is equivalent to driving with 1W input power and equivalent to input power wave  $a = \sqrt{2}$ . On each of the 2 ports of the X1 antenna array I put one

Voltage Probe and one Current probe. I then load the data into Octave, find and save the power waves “a” and “b”.

The following code saves the output of the suggested Power Wave Probes.

```
clear all;

Z0=50;
FileName="Page_Nports2_NS2.dat";
Nports = loadQucsVariable (FileName, 'Nports');
NS = loadQucsVariable (FileName, 'NS');
for m=1:NS % NS is a number of Power Wave Probes
    Power_Probe_a{NS} = [];
    Power_Probe_b{NS} = [];
    for n=1:Nports
        V = loadQucsVariable (FileName, strcat('PrV', num2str(m+(n-1)*NS), '.dv'));
        I = loadQucsVariable (FileName, strcat('PrI', num2str(m+(n-1)*NS), '.i'));
        a = (V+Z0*I)/(2*sqrt(Z0));
        b = (V-Z0*I)/(2*sqrt(Z0));
        Power_Probe_a{NS} = [Power_Probe_a{NS} a];
        Power_Probe_b{NS} = [Power_Probe_b{NS} b];
    end
    Power_Probe{NS}=[Power_Probe_a{NS} Power_Probe_b{NS}];
    dlmwrite(strcat('Power_Probe', num2str(m), '.dat'), Power_Probe{NS})
end
```

So, technically, this workaround works. However, you see that I have to draw the circuit three times: once for S-parameter simulation and two times for AC simulation. In general case, if the circuit has N ports, then I would need to have N+1 copies of the same circuit. Lots of copy-paste operations. For each of the ports of the X1 antenna array, I would need to plug one Voltage Probe and one Current Probe. This means I need 2xN probes.

This process would be much simpler if we had a Power Wave Probe that we would plug into each X1 port. Then we would need to draw only one schematic as I showed to you previously:

## s-parameter simulation

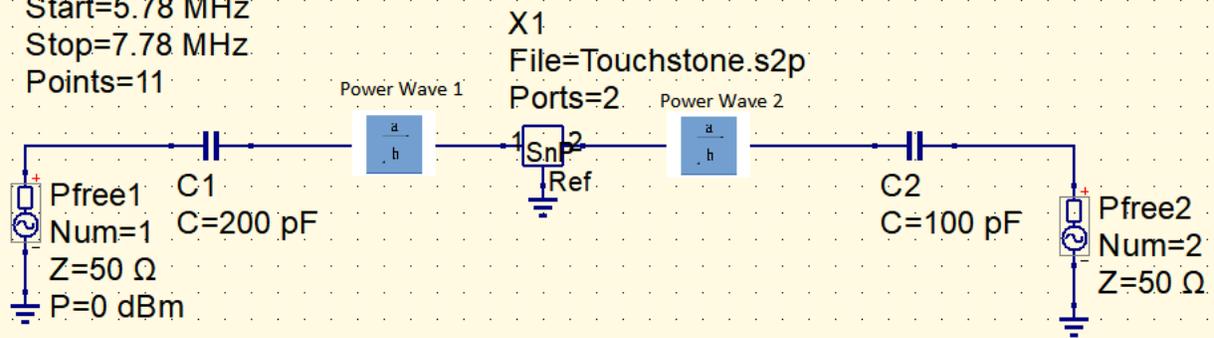
SP2

Type=lin

Start=5.78 MHz

Stop=7.78 MHz

Points=11



From this simulation, we would extract S-parameter and all power waves that we need.

I am sending to you ZIP file with this project. Feel free to open it and see how it works. The Octave code generates outputs "Power\_Probe1.dat" and "Power\_Probe2.dat". Each of the Power Wave Probes returns two "a" and two "b" power waves, because the circuit has two ports. This is for each frequency point. Also, Power Wave Probes could be inserted not necessarily on the input of X1 antenna array, but anywhere in the circuit (S-parameter simulation). Just as Current and Voltage Probes can be used anywhere in the AC Simulation.

Additional comments.

Power Wave Probe measures both (currents and voltages) and then converts them to power waves "a" and "b". We could also program it to also provide currents and voltages. Then there would be no need to extract them using Octave. The suggested Power Wave Probe looks very much like Current Probe, because it is also used in series. Modifying Current Probe to output power waves so that it would serve as a Power Wave Probe is possible way to go. I would still prefer to introduce a separate Power Wave Probe that is specifically designed for S-parameter Simulation. Using Current Probe in the S-parameter simulation may be confusing for other people, because in fact such Current Probe would need to measure not only current, but also voltage. May I suggest that Current Probe stays as is (only for AC Simulation) and we introduce a separate Power Wave Probe?

Power waves were introduced because from a certain perspective they are more convenient to work with than voltages and currents. In particular, if we work with 50 Ohm devices at RF frequencies. It is a different way to look at the circuit. In place of voltages and current, we now have power waves. This means, there should be a way to measure power waves in the circuit. Power Wave Probe would be a

natural way to do so. To use Current Probe to output power waves would be strange and confusing way of doing that.

I am not sure what you mean by saying "simulation parameter "output"". Could you explain?