Explanation of S11 and S21 in Spice Simulation

KSKelvin Kelvin Leung
Created on 4-26-2023
Last Updated on 12-9-2023

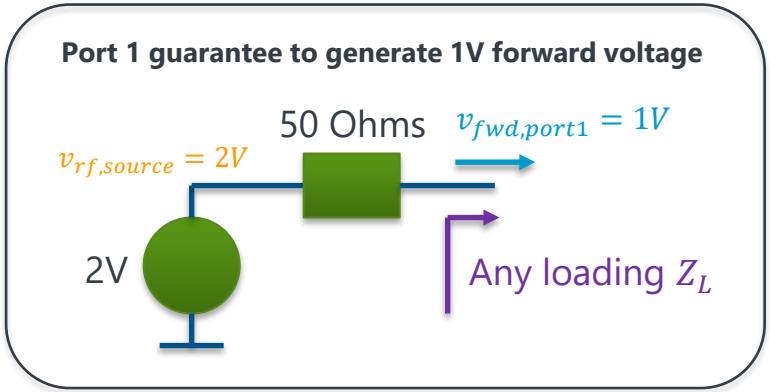
Basic of S11 and S21

- Definition of S11 and S21
  - $S_{11} = \frac{v_{rev,port1}}{v_{fwd,port1}} = \sqrt{\frac{P_{rev,port1}}{P_{fwd,port1}}}$
    - Given $v_{rev,port2} = 0V$ ($p_{rev,port2} = 0W$)
  - $S_{21} = \frac{v_{fwd,port2}}{v_{fwd,port1}} = \sqrt{\frac{P_{fwd,port2}}{P_{fwd,port1}}}$
    - Given $v_{rev,port2} = 0V$ ($p_{rev,port2} = 0W$)

- Remark
  - Measurable voltage (v_{rf}) is summation of forward and reverse voltage
    - $v_{rf} = v_{fwd} + v_{rev}$

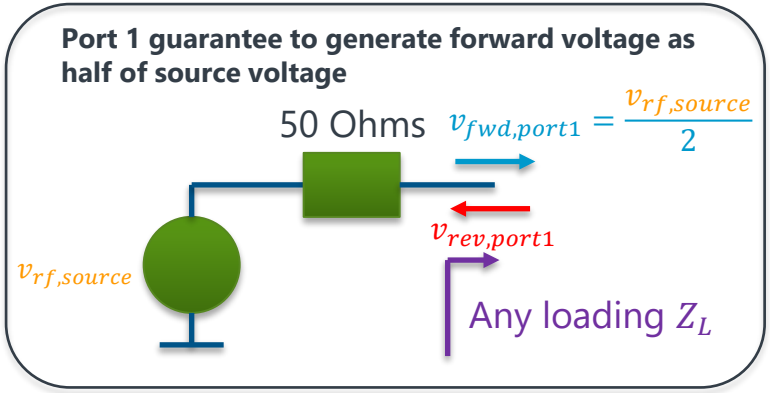
How to Simulate S21 in Spice with AC Sweep

- Refer to S21 Definition
  - $S_{21} = \frac{v_{fwd,port2}}{v_{fwd,port1}} = \sqrt{\frac{P_{fwd,port2}}{P_{fwd,port1}}}$
    - Given $v_{rev,port2} = 0$ ($p_{rev,port2} = 0$)
- Simulation Idea
  - As no reflection at port 2, system must be terminated by characteristic impedance (50 Ohms)
  - As no reflection, direct voltage ($v_{rf,port2}$) measurement at system output equal $v_{fwd,port2}$
    - $v_{rf} = v_{fwd} + v_{rev}$, if $v_{rev} = 0$, $v_{rf} = v_{fwd}$
  - If Port 1 is setup to give a forward 1V voltage (i.e. $v_{fwd,port1} = 1V$), AC Sweep voltage at port 2 is S21 as
    - $S_{21} = \frac{v_{fwd,port2}}{v_{fwd,port1}} = \frac{v_{rf,port2}}{1}$
  - By Port 1 as a 2V voltage source with 50 ohms source impedance, it can guarantee $v_{fwd,port1} = 1V$ in regardless of port 1 loading condition

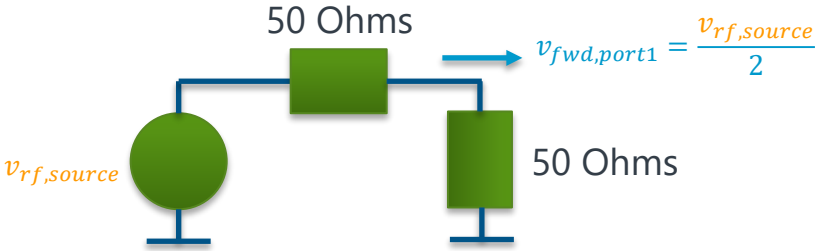


How to Simulate S21 in Spice with AC Sweep

- Why this circuit must generate a forward voltage equal half of source voltage
  - In theory, forward voltage is voltage wave travel forward which terminated into characteristic impedance Zo (50 ohms)
  - In this circuit, no matter what loading condition is, if we couple forward wave and terminate into 50 ohms, it always $v_{fwd,port1} = \frac{v_{rf,source}}{2}$
  - This is also the reason why signal generator designed to have source impedance equal to characteristic impedance Zo, as its voltage or power setpoint can always be equal to forward voltage or forward power



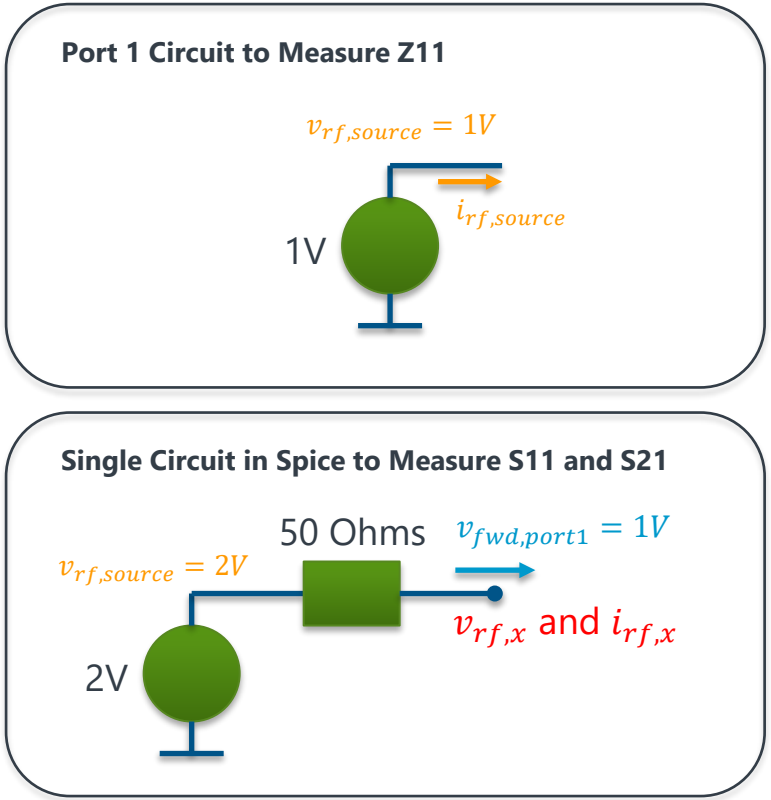
If forward wave is coupled into 50 ohms



How to Simulate S11 in Spice with AC Sweep

- S_{11} is rely on conversion from Z_{11}
  - Z_{11} is impedance measured from port 1 with port 2 terminated at 50ohms
    - If no output from circuit, port 2 not exist and both forward and reverse at port 2 must be 0
  - Z11 is a direct measurable parameter in AC Sweep by $Z_{11} = \frac{v_{rf,source}}{i_{rf,source}}$
  - Conversion is $S_{11} = \frac{Z_{11}-Z_o}{Z_{11}+Z_o}$

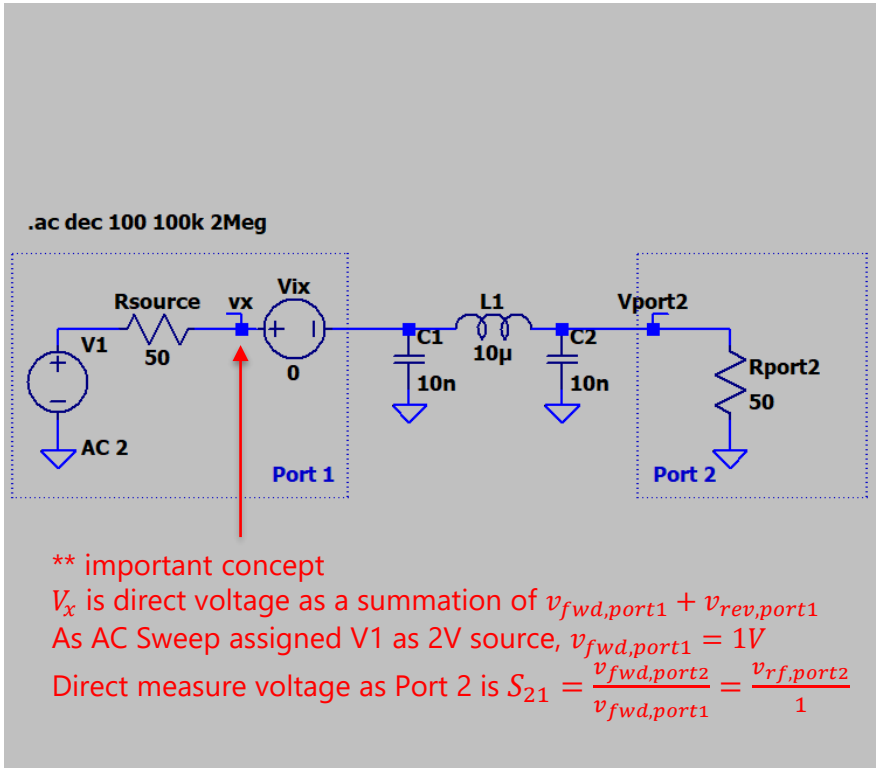
- A single circuit in Spice to measure both S11 and S21
  - It is possible to use S21 port 1 circuit but measure v_{rf} and i_{rf} after source impedance and calculate $Z_{11} = \frac{v_{rf,x}}{i_{rf,x}}$



Spice Simulation for S11 and S21

LTSpice Simulation for Z_{11} and S_{21} : LTSpice_Sparam_Simulation.asc

Schematic



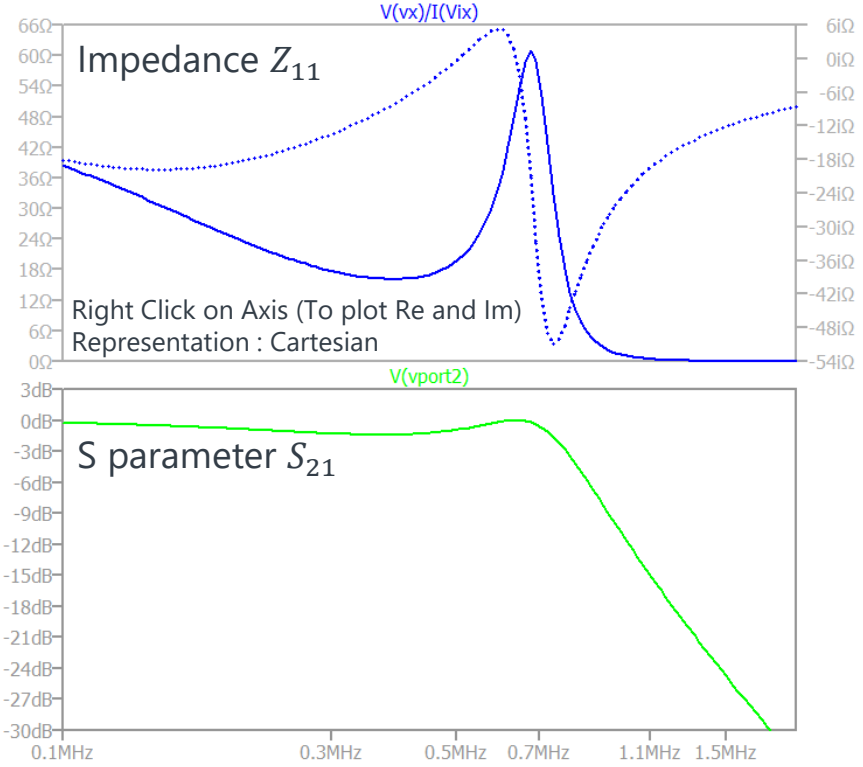
** important concept

V_x is direct voltage as a summation of $v_{fwd,port1} + v_{rev,port1}$

As AC Sweep assigned V1 as 2V source, $v_{fwd,port1} = 1V$

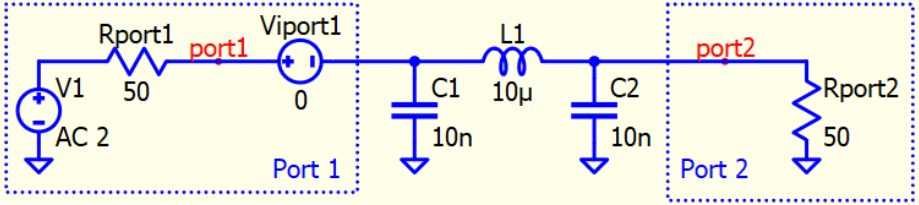
Direct measure voltage as Port 2 is $S_{21} = \frac{v_{fwd,port2}}{v_{fwd,port1}} = \frac{v_{rf,port2}}{1}$

Simulation Result



QSpice Simulation for S_{11} and S_{21} : QSpice_Sparam_Simulation.asc

Schematic




```
.param Zo=50
.func Z11() V(port1)/I(Viport1)
.func S11() (Z11()-Zo)/(Z11()+Zo)
.func S21() V(port2)
```

```
.ac dec 100 100K 2Meg
.plot S21()
.plot S11()
```

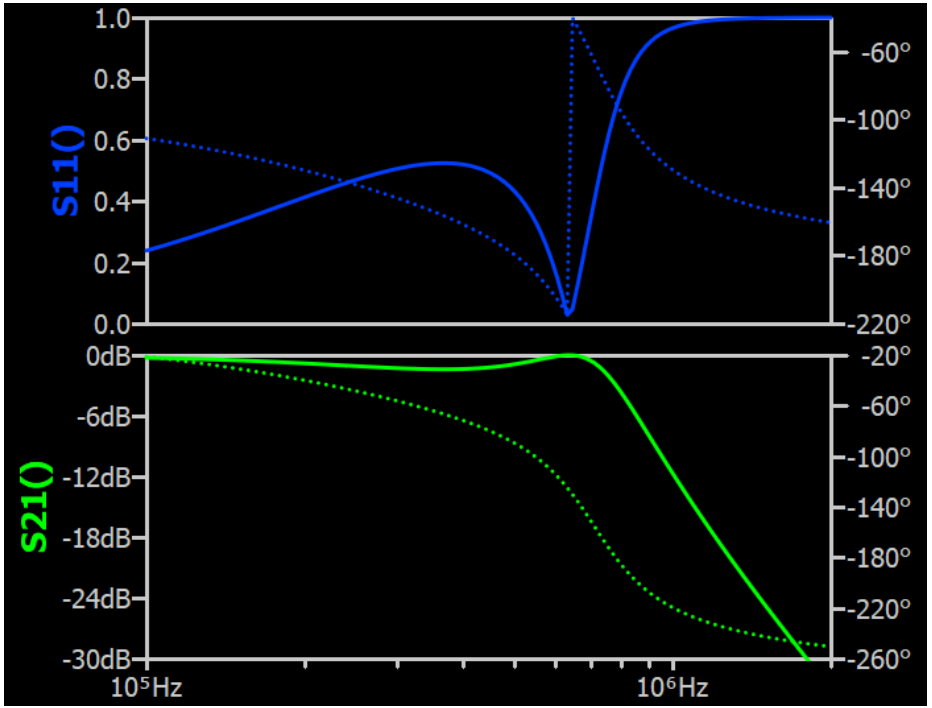
** important concept

V(port1) is direct voltage as a summation of $v_{fwd,port1} + v_{rev,port1}$

As AC Sweep assigned V1 as 2V source, $v_{fwd,port1} = 1V$

Direct measure voltage as Port 2 is $S_{21} = \frac{v_{fwd,port2}}{v_{fwd,port1}} = \frac{v_{rf,port2}}{1}$

Simulation Results



QSpice Simulation for S_{11} and S_{21} : QSpice_Sparam_Simulation.asc S11() in SmithChart Representation

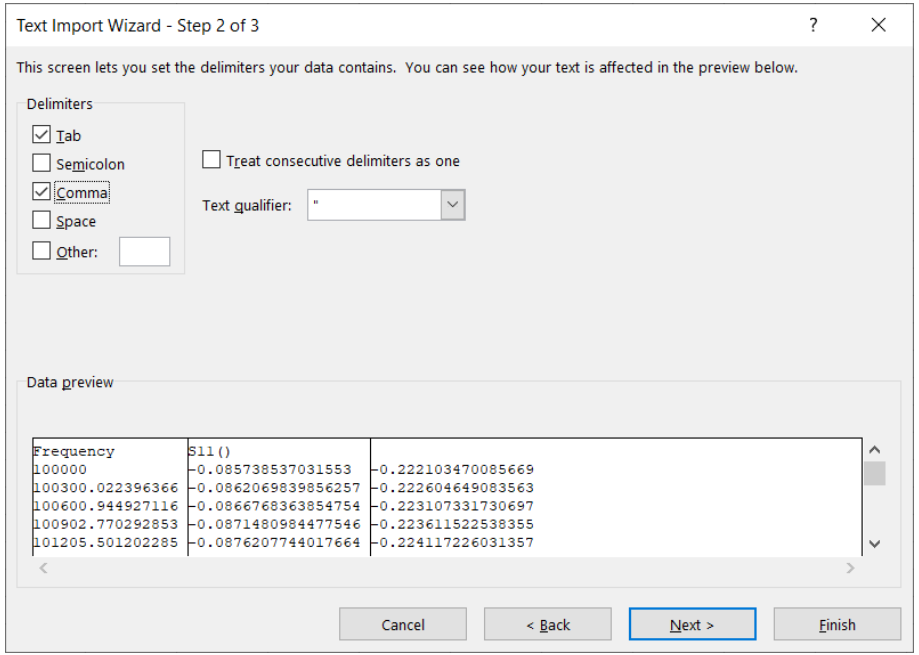
** Qspice not supports to plot in Cartesian representation yet, it requires post-processing data to plot in SmithChart

Method to plot S11 in Microsoft Excel

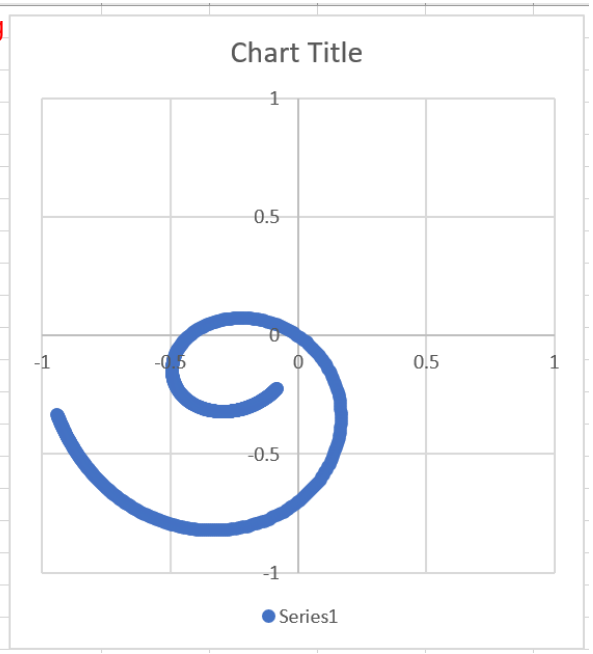
[1] In Qspice Waveform Viewer, File > Export, Select to export S11() in csv format. Exported S11 is in format of [re(S11), im(S11)]

[2] Rename .csv to .txt, use Excel to import this .txt. In Import Wizard, it will ask for delimiters, select both “Tab” and “Comma”

[3] Use X-Y Scatter plot to plot with x-axis Re(S11) and y-axis Im(S11)

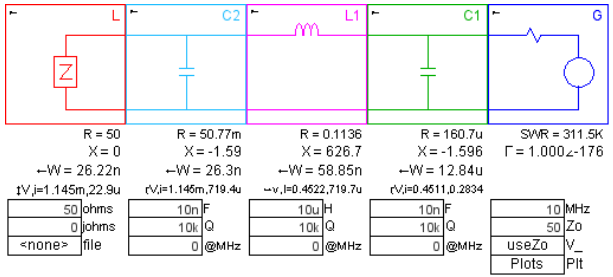


Frequency	S11()	Real	Imag
100000	-0.08574		-0.2221
100300	-0.08621		-0.2226
100600.9	-0.08668		-0.22311
100902.8	-0.08715		-0.22361
101205.5	-0.08762		-0.22412
101509.1	-0.08809		-0.22462
101813.7	-0.08857		-0.22513
102119.2	-0.08905		-0.22564
102425.5	-0.08953		-0.22615
102732.8	-0.09001		-0.22665
103041.1	-0.0905		-0.22716
103350.2	-0.09099		-0.22767
103660.3	-0.09148		-0.22817
103971.3	-0.09197		-0.22868
104283.2	-0.09247		-0.22919
104596.1	-0.09296		-0.2297
104909.9	-0.09346		-0.23021
105224.7	-0.09397		-0.23072
105540.4	-0.09448		-0.23122

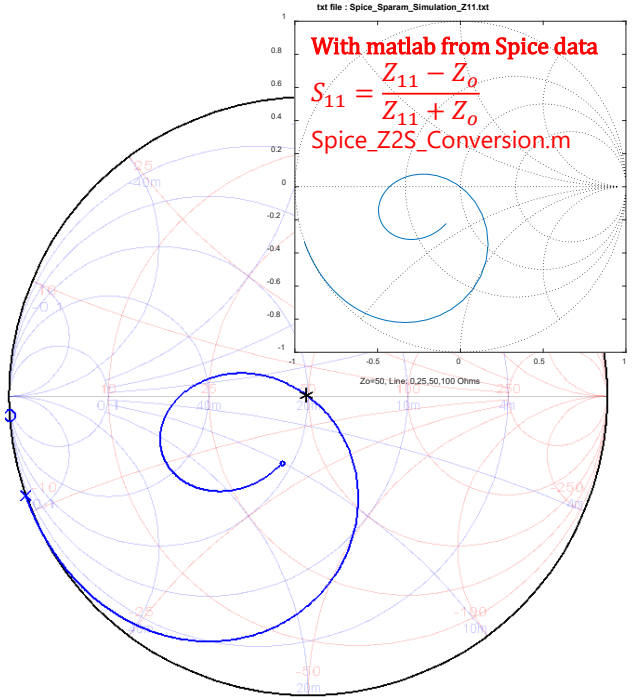


SimNEC Simulation for S_{11} and S_{21} : SimNEC_Sparam_Simulation.ssn

Schematic



SmithChart S11



Gain Chart S21