Explanation of S11 and S21 in Spice Simulation

KSKelvin Kelvin Leung
Created on 4-26-2023
Last Updated on 8-6-2025

Basic of S11 and S21

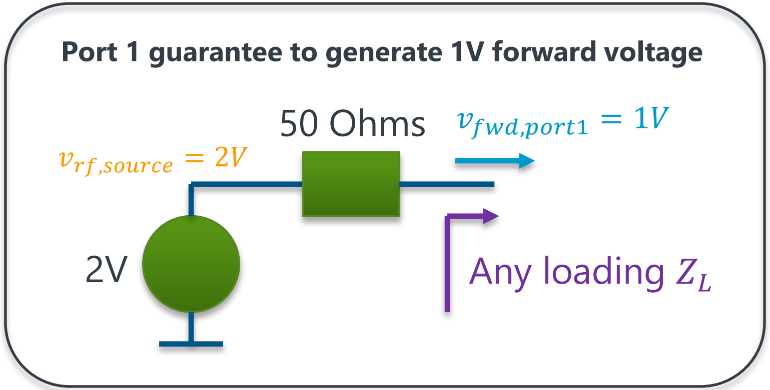
- Definition of S11 and S21
  - $S_{11} = \frac{v_{rev,port1}}{v_{fwd,port1}} = \sqrt{\frac{P_{rev,port1}}{P_{fwd,port1}}}$
    - Given $v_{rev,port2} = 0V$ ($p_{rev,port2} = 0W$)
  - $S_{21} = \frac{v_{fwd,port2}}{v_{fwd,port1}} = \sqrt{\frac{P_{fwd,port2}}{P_{fwd,port1}}}$
    - Given $v_{rev,port2}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

- 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

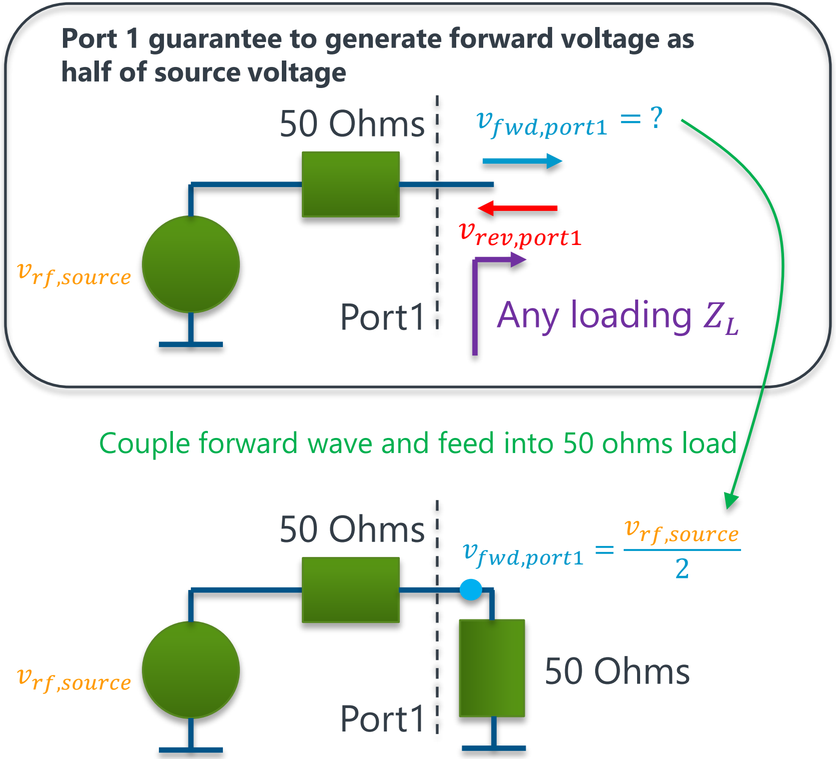
- Refer to S21 Definition
  - $S_{21} = \frac{v_{fwd,port2}}{v_{fwd,port1}} = \sqrt{\frac{P_{fwd,port2}}{P_{fwd,port1}}}$
    - Given $v_{rev,port2} = 0$ ($p_{rev,port2} = 0$)

- 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



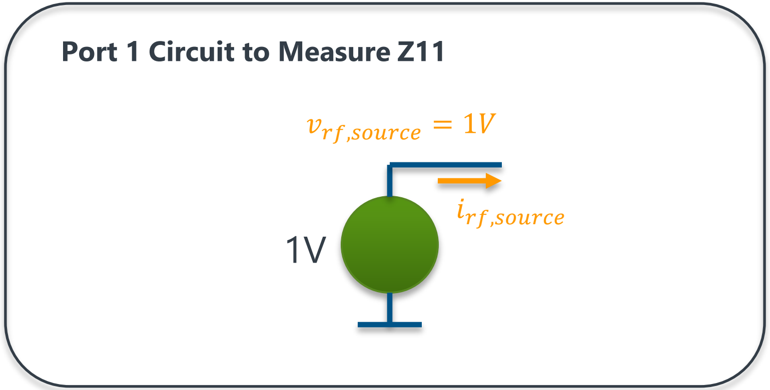
How to Simulate S21 in Spice with AC Sweep

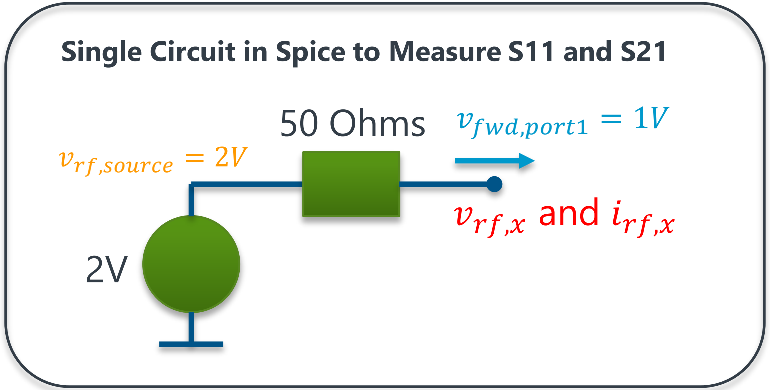
- Why this circuit must generate a forward voltage equal half of source voltage
  - In theory, forward voltage is the voltage wave that travels forward and terminates into the characteristic impedance Zo (50 ohms)
  - For a circuit with a voltage source in series with a 50-ohm resistor, regardless of the loading condition, if we only need to know the forward voltage, we can assume we couple the forward wave and feed it into the 50-ohm impedance
  - The forward-only circuit now becomes a simple voltage divider, and the forward voltage must equal half of the source voltage $v_{fwd,port1} = \frac{v_{rf,source}}{2}$
  - Therefore, the advantage of having a 50-ohm in series with the source is that it terminates the reverse wave and forced forward voltage to equal half of the source voltage
  - This is also why signal generators are designed to have a source impedance equal to the characteristic impedance Zo, as their voltage or power setpoint can always be related to the forward voltage or forward power with only a scaling factor



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 $Z_{11} = \frac{v_{rf,x}}{i_{rf,x}}$

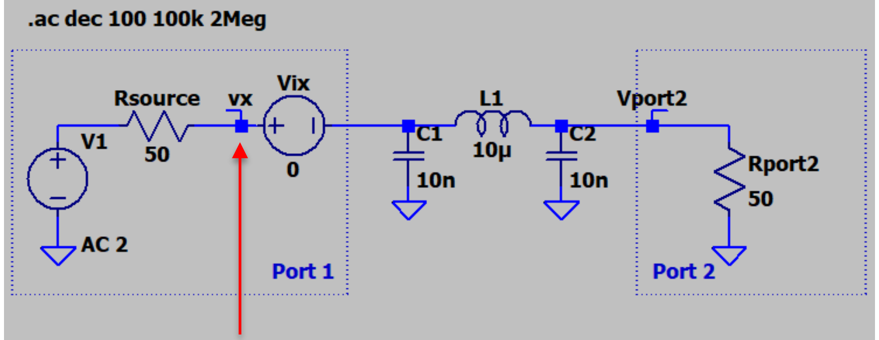




Spice Simulation for S11 and S21

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

Schematic



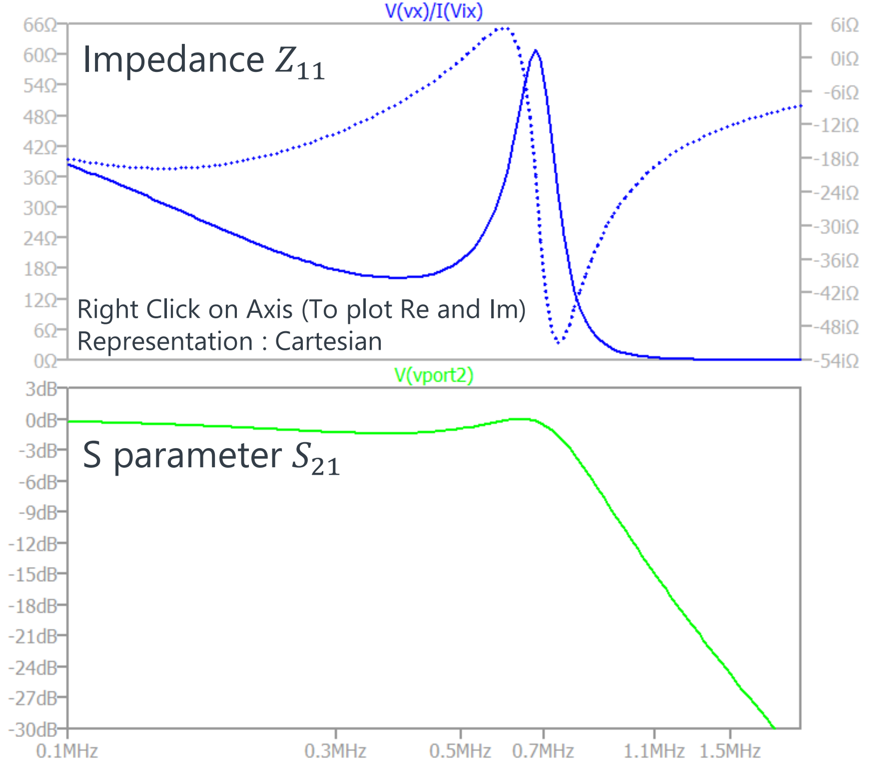
** important concept

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

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

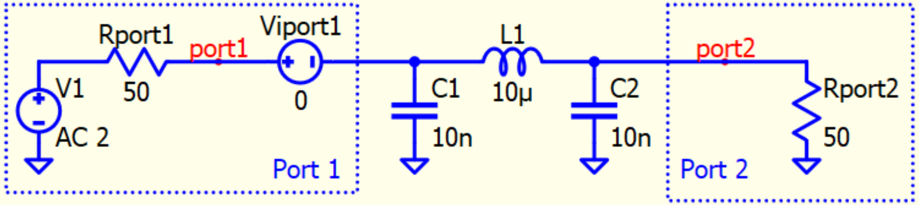
Direct measure voltage as Port 2 is $S_{21} = \frac{v_{fwd,port2}}{v_{fwd,port1}} = \frac{v_{rf,port2}}{1}$

Simulation Result



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

Schematic

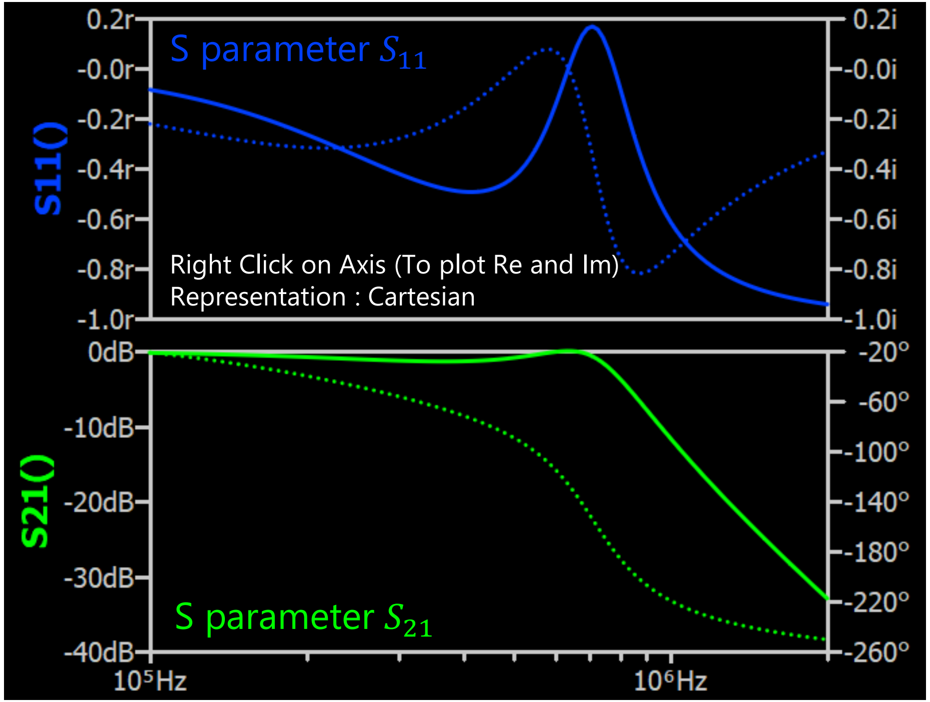


.param Zo=50
.func Z11() V(port1)/I(Viport1)
.func S11() (Z11()-Zo)/(Z11()+Zo)
.func S21() V(port2)

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

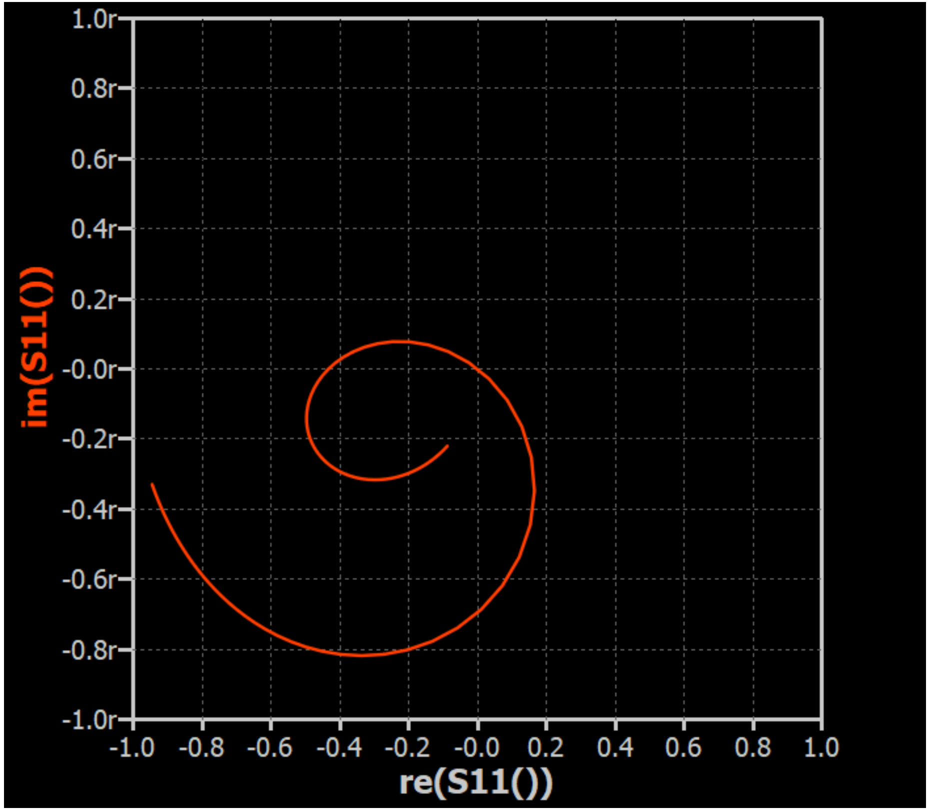
** important concept
V(port1) is direct voltage as a summation of $v_{fwd,port1} + v_{rev,port1}$
As AC Sweep assigned V1 as 2V source, $v_{fwd,port1} = 1V$
Direct measure voltage as Port 2 is $S_{21} = \frac{v_{fwd,port2}}{v_{fwd,port1}} = \frac{v_{rf,port2}}{1}$

Simulation Results



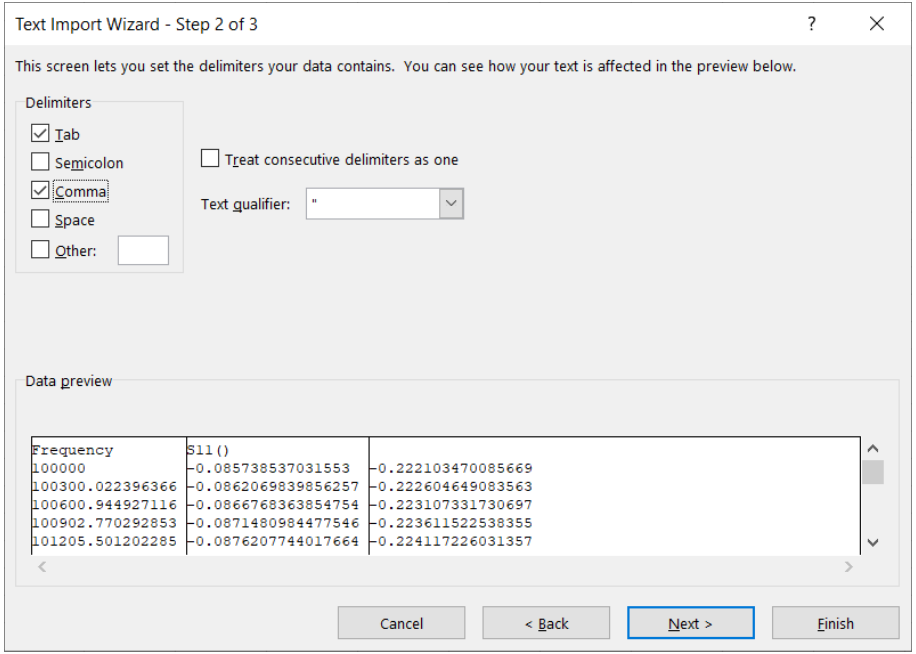
QSpice Simulation for S_{11} and S_{21} : QSpice_Sparam_Simulation.asc
How to Plot S11 in QSpice

- Plot S11
  - Qspice supports Catesian Representation, S11 can be plotted in a X-Y plot format
  - Add plot expression : im(S11())
  - Right Click left y-axis
    - change to Cartesian Representation
    - Change range to Top=1r, Tick=0.2r, Bottom=-1r
  - Right Click right y-axis
    - select No Imaginary
  - Right Click x-axis
    - Change Quantity Plotted to re(S11())
    - Dis-select Logarithmic
    - Change range to Left=-1, Tick=0.2, Right=1

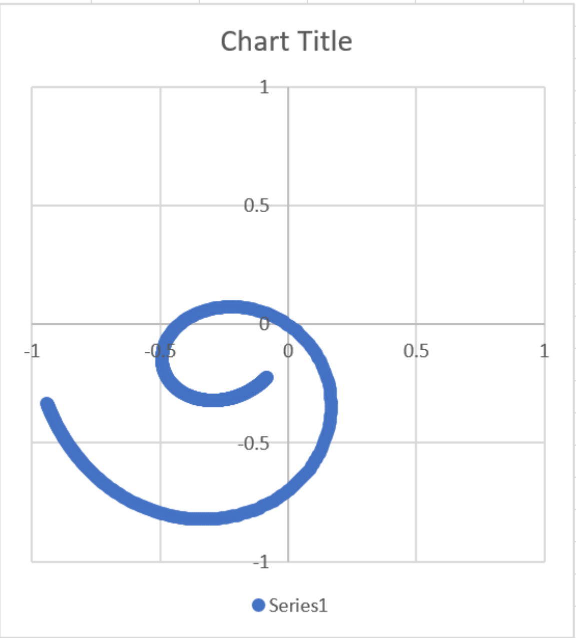


QSpice Simulation for S_{11} and S_{21} : QSpice_Sparam_Simulation.asc
S11() in SmithChart Representation

** User can also 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)

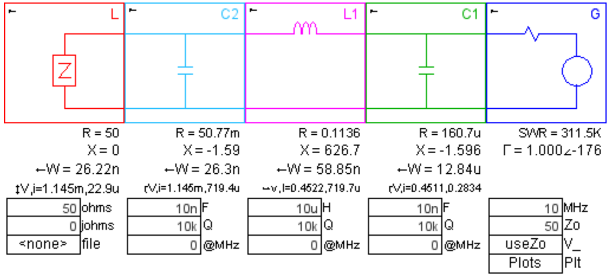


Frequency	S11() Real	Imag
100000	-0.08574	-0.2221
100300	-0.08621	-0.2226
100600.9	-0.08668	-0.22311
100902.8	-0.08715	-0.22361
101205.5	-0.08762	-0.22412
101509.1	-0.08809	-0.22462
101813.7	-0.08857	-0.22513
102119.2	-0.08905	-0.22564
102425.5	-0.08953	-0.22615
102732.8	-0.09001	-0.22665
103041.1	-0.0905	-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

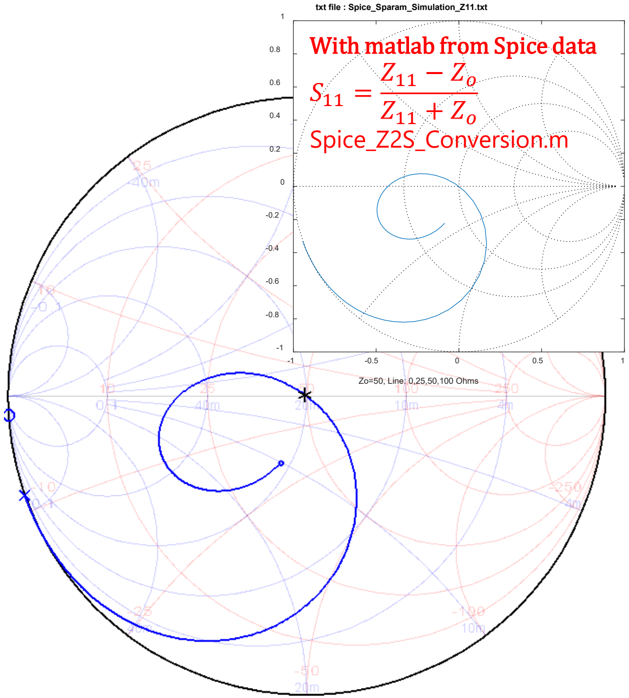


SimNEC Simulation for S_{11} and S_{21} : SimNEC_Sparam_Simulation.ssn

Schematic



SmithChart S11



Gain Chart S21