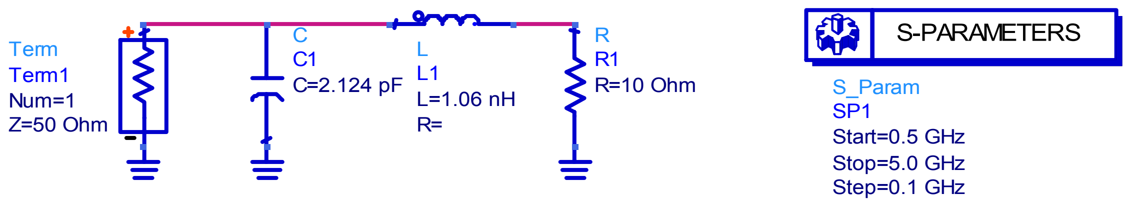
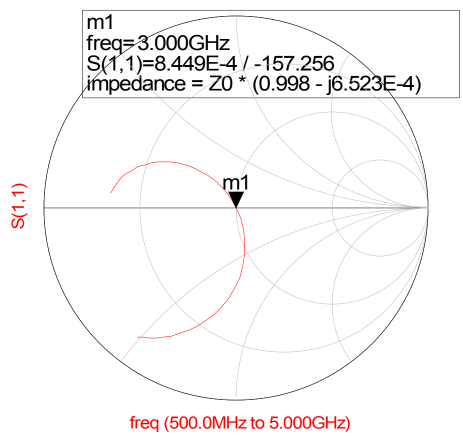
ADS Tutorial #3 Tuning and T-Line

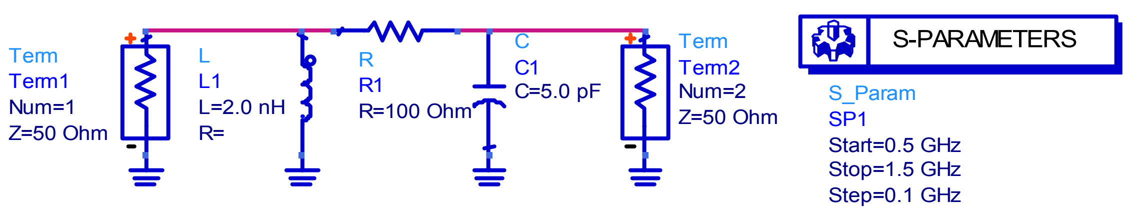
Oklahoma State University

1. Tuning Matching Network





2. Two-Port Network



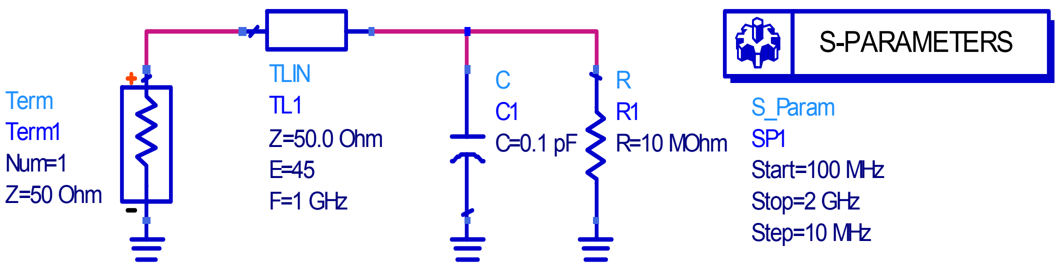
Matrix conversion functions:
a) Z-matrix: stoz(S)
b) Y-matrix: stoy(S)
c) H-matrix: stoh(S)
d) ABCD matrix: stoabcd(S)

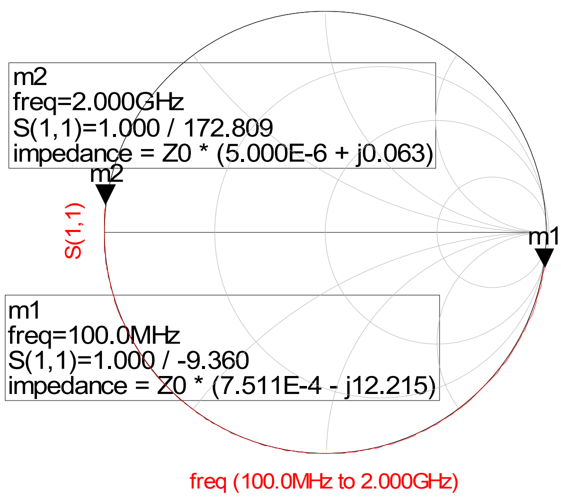
e) Z-to-Y: ztoy(Z)
g) Z-to-ABCD:ztoabcd(Z)
……

3. Transmission Line Circuit

1) Ideal T_Line

The components of the ideal transmission line can be found in the component pallet "Tlines-Ideal". The length of the T-line is specified by the "electrical length" (the number is the angle in degree) and the frequency. It is a one port network, so one can only get S_{11} (or Γ) in the S-parameter simulation. As the load resistance is very high, the termination is equivalent to a pure capacitor, which is reflected in the simulation result. It can also be found that the input impedance looks like a capacitor at low frequency range, but it becomes inductor-like in the high frequency range. A T-line with E=45 deg will become a quarter-wavelength (E=90 deg) T-Line at 2 GHz.

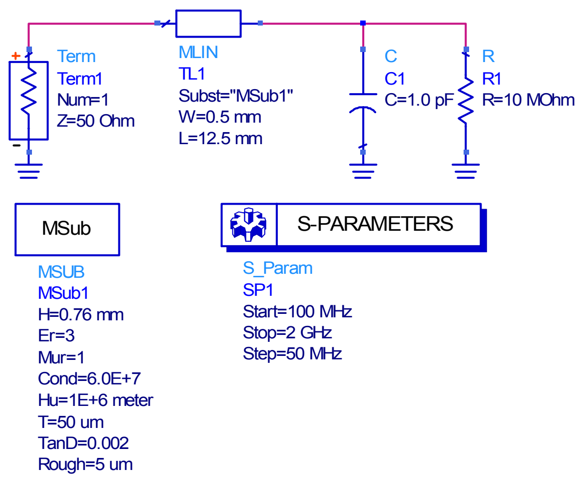


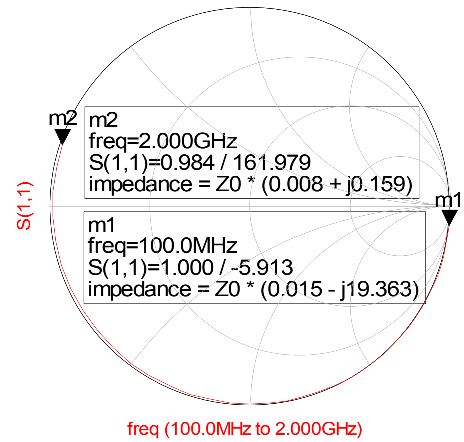


2) Microstrip

A) Schematic Design

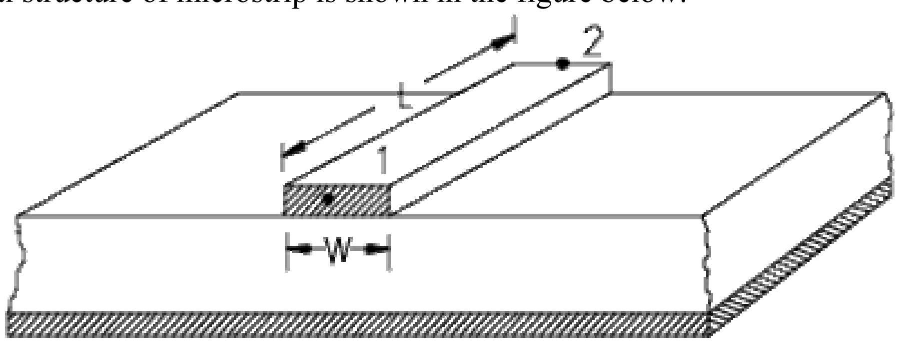
In a realistic microstrip T-line we have a number of parameter to set up. First we can construct the following simplest circuit. The microstrip T-line can be found in the component pallet "TLines-Microstrip". We can try to match the parameter of this microstrip T-line to the ideal T-line (by design L = $\lambda/8$, i.e. 45 degree; at the same time assume v_p = c/Er), the simulation result is shown below. They match pretty well, but we can see the effect of the loss, as $|S_{11}|$ becomes less than unity at high frequency.





B) Physical Design

The physical structure of microstrip is shown in the figure below:



Just like the VLSI design, there are two families of parameters:

1> The "design parameters" are listed under the symbol of "MLin", and one can select the width and the length. There are various design rules for the width, as too narrow or too wide stripes are not practical from the point of view in fabrication. We can use the following rule: $0.05H < W < 16H$, where H is defined in the following item.

2> The "technology parameters" are listed under the "MSub" icon, which describe the properties of the PCB from which the microstrip is made. This group of parameters should be provided by PCB vendors, and the designers are not allowed to change them. The meaning of these parameters are the following:

- H: substrate thickness.
- Er: relative dielectric constant of the insulating substrate.
- Mur: relative permeability of the conductor (it is unity unless ferromagnetic material).
- Cond: conductivity of the conductor layer (for copper $\sigma=6.33\times10^{7}$ S/m).
- Hu: height of the upper layer material (usually assume infinity for microstrip).
- T: thickness of the conductor layer.
- TanD: loss tangent of the dielectric substrate material.
- Rough: ripple amplitude of the surface (zero is usually assumed).

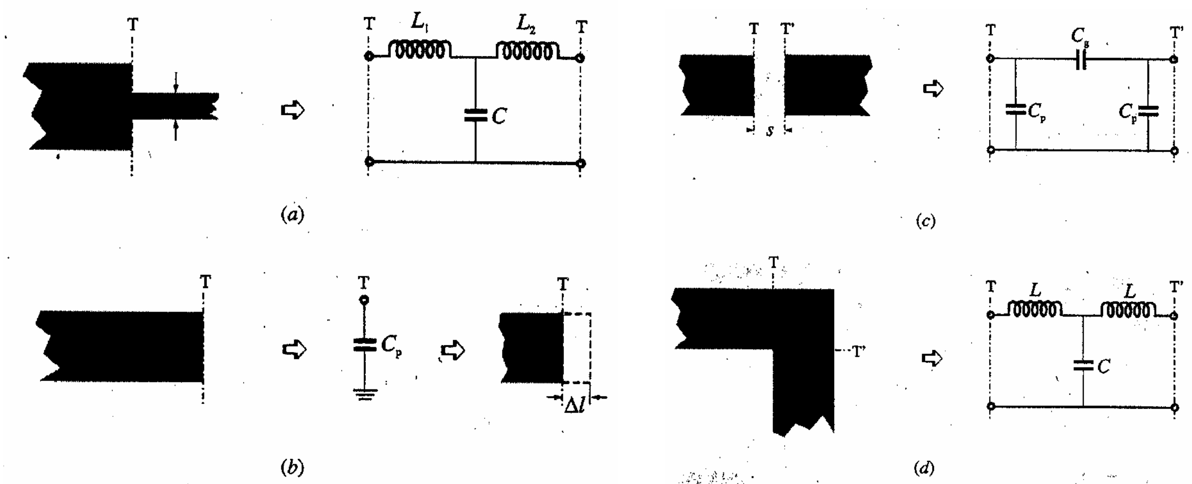
3) LineCalc

It is a built-in tool in ADS that can be used to find the characteristic impedance and the electrical length from the structural and material parameters of the transmission line. On the other hand, if the material parameter of the substrate is determined, it can also be used to find the design parameters of the T-Line from the desired characteristic impedance and electrical length. Besides microstrip, LineCalc can deal with many kinds of T-Lines, including coaxial cable, coplanar T-Line, etc.

LineCalc is located under "tools" on the menu bar. In the LineCalc window click "options" on the menu bar and choose preference, where one can select the unit. The unit "mil" is equal to 0.001 inch, and it is about 25.4 μm, which is widely used in the specification of microstrip. However, the IS units will be adopted gradually in the future. We can set the substrate parameter following the example on the previous stage.

After the width and the length of the microstrip is determined, one can place this component into the design schematic. In the schematic window click the "LineCalc" from the "Tools" on the menu on top, there is only one option "Place New Synthesized Component". Then this component can be move to a position in a circuit.

Usually one needs to resimulate the circuit, there are parasitic components in the microstrips, which are not taken into account in the schematic simulation. In the figure below some equivalent parasitic elements are shown.



Momentum method can be used to simulate the layout, which will be introduced in the next tutorial.