10

This chapter shows how to use LineCalc and the Transient simulator to make basic delay and TDR measurements.

Lab 10: TDR and LineCalc with the Transient Simulator

OBJECTIVES

- Simulate the delay through a line
- Simulate a TDR (time domain reflectometry) measurement
- Use the Data Display as a calculator (equations)
- Use LineCalc to analyze impedance and synthesize a matched line

PROCEDURE

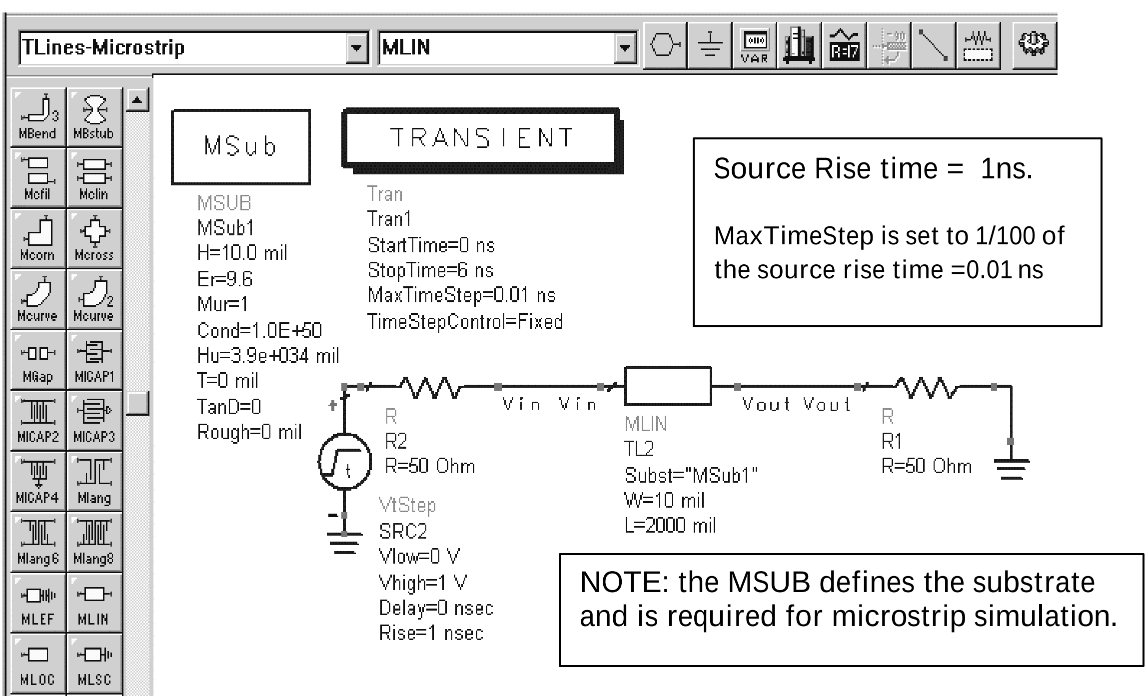
1. Open a new schematic window and setup the circuit

a. Insert a **VtStep** source (Time Domain) and set Vhigh = 1V and Rise = 1 nsec with no delay as shown.

b. Insert a Transient controller set as shown: **StartTime=0ns, StopTime = 6 ns** and **MaxTime Step = 0.01 ns** (this is the sampling time step). Also, be sure the TimeStepControl is Fixed. Use the Display tab to make settings visible.

c. From Tlines-Microstrip palette, insert an **MSUB** (substrate definition) using the default values. Then insert an MLIN (scroll down to find it) set to **W=10** and **L=2000** mils. This is a narrow and long line (2 inches) used to route a signal.

d. Add node names **Vin** and **Vout**, two 50 ohm resistors, and wires.



Source Rise time = 1ns.

MaxTimeStep is set to 1/100 of the source rise time =0.01 ns

NOTE: the MSUB defines the substrate and is required for microstrip simulation.

e. Save the schematic as: **transient_simple**.

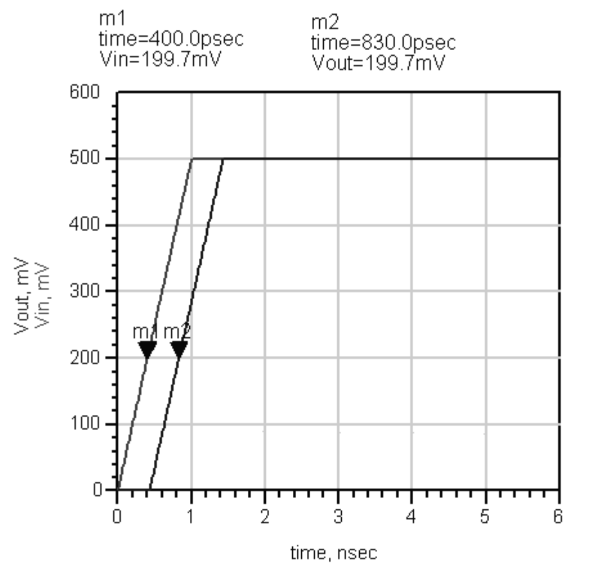
2. Simulate and plot the response

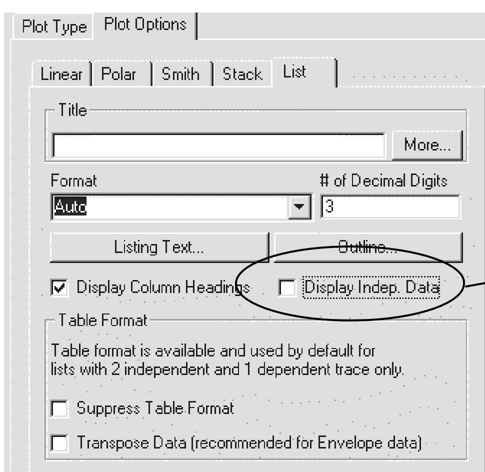
a. Simulate the schematic and open a new data display, saved as: **transient_simple**

b. Insert a plot of: Vin and Vout

c. Put markers on the plot (as shown) where the voltages are equal for the two markers, near the lower portion of the rising edges.

d. Write an equation to compute the difference between the marker x-axis values. Use the **indep** function and list the marker delay as shown. To remove the invalid variable from the list, click the Display Indep Data box check as shown.



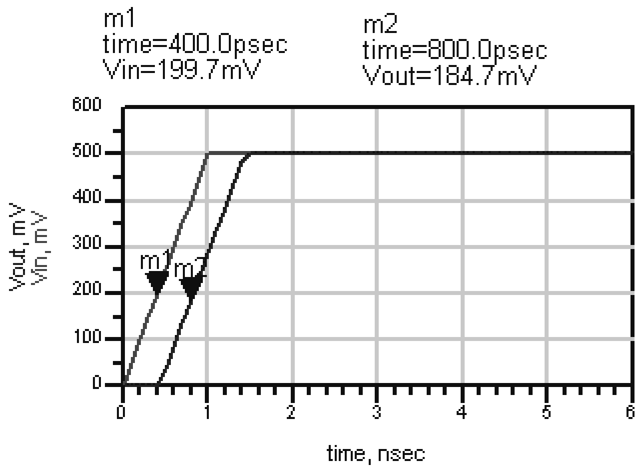


Eqn marker_delay=indep(m2) - indep (m1)

marker_delay
4.300E-10

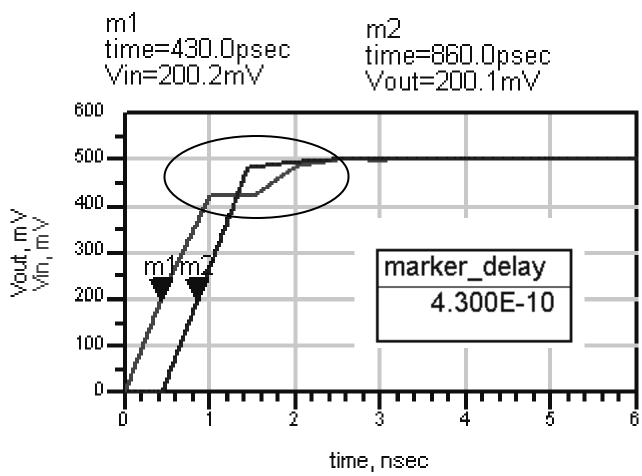
e. Try selecting both markers (use SHIFT key) and move them with an arrow key. As the time increases, the marker delay value will update but you should see almost no change because the delay is constant at about 0.43 nanoseconds shown here.

3. Change the simulation time step to less resolution



Change MaxTimeStep to 0.1 nsec (this is less resolution) and simulate again. Now try placing the markers at the same voltages. Because the sample time was not small enough, you will not be able to set the markers to the same voltage. This is why the sample time is critical.

4. Change the time step to 0.01 ns and change the MLIN width to 20 mils and simulate



With the increased width, the response appears (circled area) to act like a capacitor charging: the input Vin trace takes more time to reach 500 mV. Of course, increasing the width of the line will increase capacitance, creating a mismatch to the 50 ohm load. Also, the delay is not constant as you can see by moving the markers along the rising values of Vin and Vout and noting the change in marker delay.

5. Save the schematic and data display (trans_simple)

Delay Note: The last few steps demonstrated how microstrip transmission line delay can be measured using a fast rising pulse in the time-domain. Of course, this is a simple circuit but the concept can be applied to more complex lines or even filters. In general, for delay, the sampling time step must be set with enough resolution to capture the rise of the voltage at the input and output so that the levels can be tracked as in the last few steps above.

Additionally, it is also better to characterize the circuit prior to measuring delay so that any mismatches can be resolved. For this purpose, LineCalc can be used as you will see later on. However, before using LineCalc, you will use the same MLIN and set up a TDR measurement to determine the character of the mismatch.

6. Create a new schematic

a. Save the last schematic (trans_simple) as: **trans_tdr**.

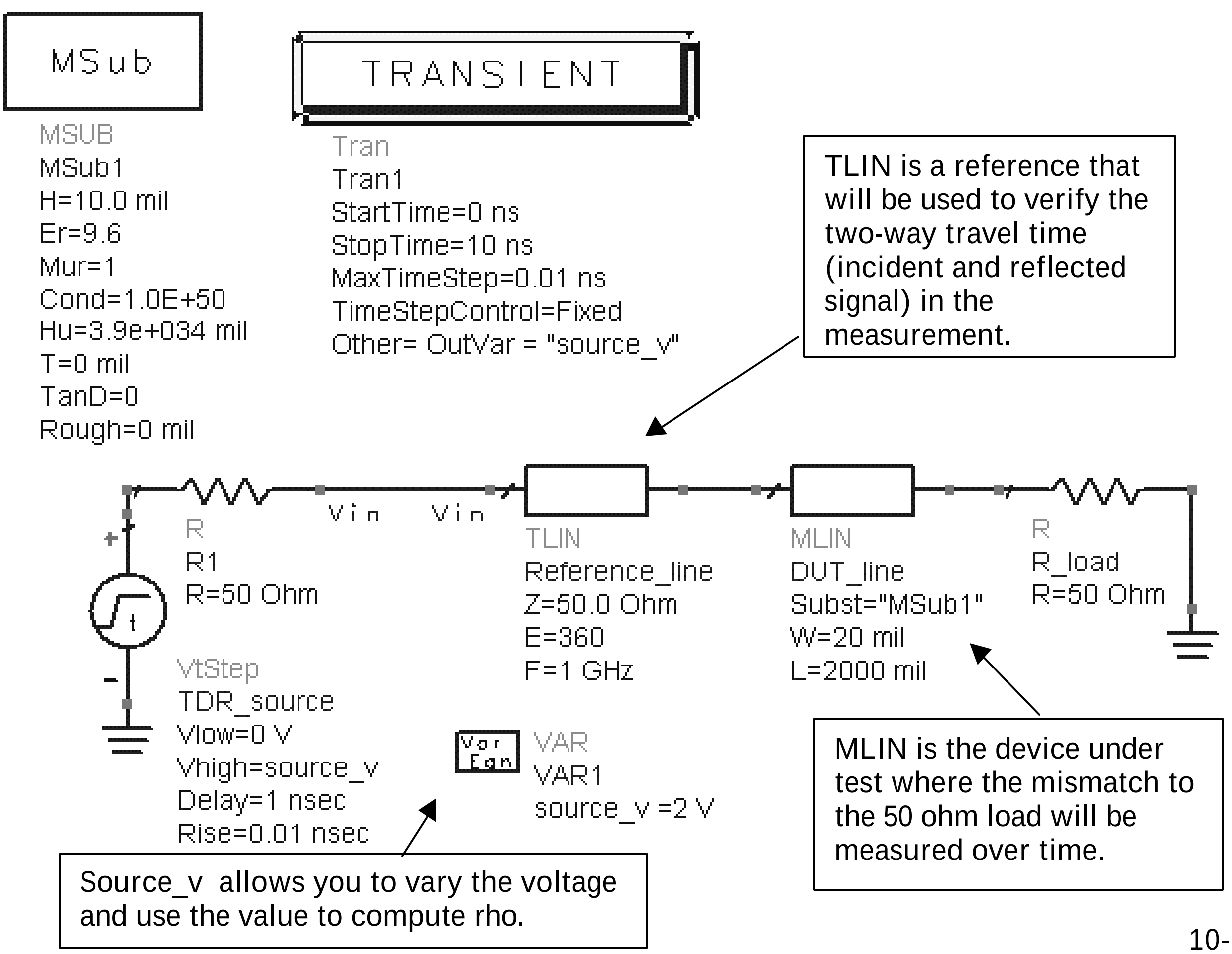
a. Use the same MSUB, simulation controller, and the MLIN which you just measured for delay or transit time.

b. Add a VarEqn for the source voltage rise: source_v = 2 V

c. Set up the Transient controller as shown: Stop to 10 nsec with 0.01 time step. Also, set up an OutVar for the source_v as shown. To do this, go to the display tab and check the box marked Other. Then type in the term OutVar = “source_v”.

d. Setup the source (similar to last schematic) but set the voltage rise as Vlow = 0 to Vhigh = source_v, which is 2 volts set in the VAR. Also, set 1 nsec delay and a rise time of 0.01 nsec.

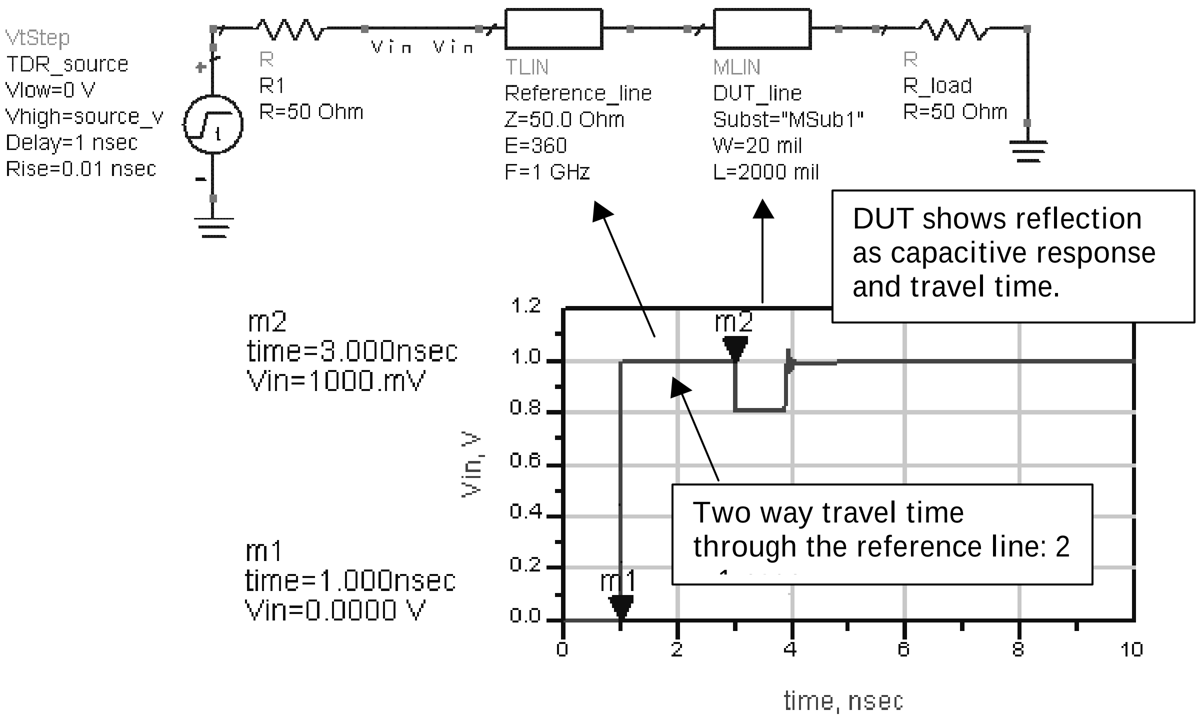
e. Complete the circuit: go to the Tlines_ideal palette, insert a TLIN, and set it as shown. This ideal line will be used as a reference for measuring the MLIN. Remove the Vout node, check the circuit, and then save it.



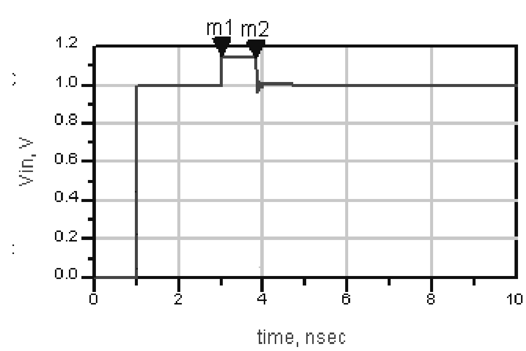
7. Simulate and plot the response at Vin

a. Open a new data display and plot Vin. You should see the response of the reference line and the mismatched MLIN.

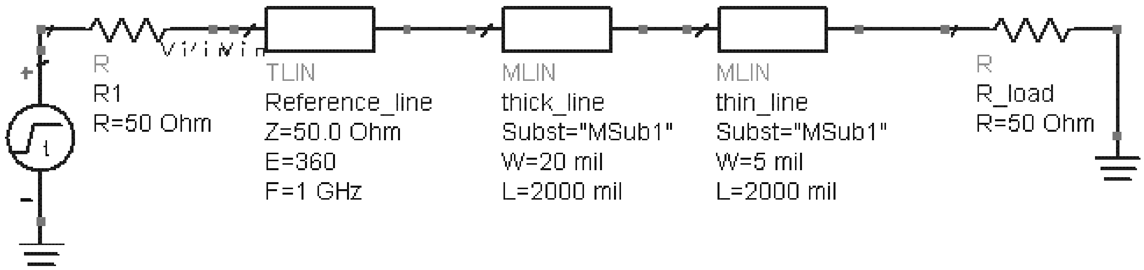
TDR measurement note: As you should see, the trace is delayed for 1 ns. Then the pulse rises and stays at one-half the voltage for 2 nsec as it travels though the reference line (1 nsec), reflects off the mismatched DUT (marker 2), and then travels back through the reference line for a total of 2 nsec. The DUT response occurs over a little less than 1 nsec which means that its travel time is less than ½ nsec. Also, the DUT shows a capacitive response, followed by a small ringing effect as the pulse encounters the mismatch between the DUT and the 50 ohm load.



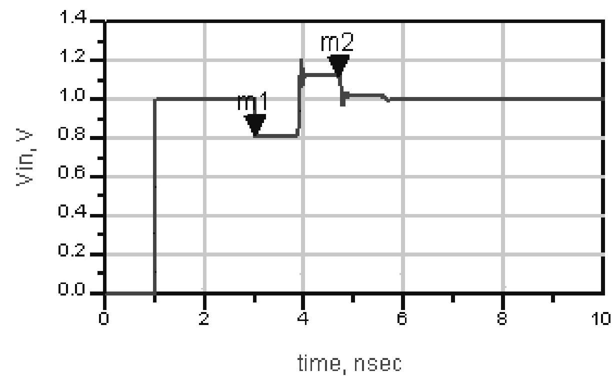
b. Change the MLIN to 5 mils wide, simulate, and plot the response. You should see the inductive effects of the narrow line as its impedance increases.



c. Add another MLIN so that you have both a thick line (20 mils) and a thin line (5mils).



d. Simulate and view the response. As you will see, the mismatches appear both capacitive and inductive for the 2 MLINS. After about 6 nsec, the voltage at Vin settles back to half of the source voltage.



Design Note on BW: For the last steps or measurements, the frequency bandwidth where the circuit operation is valid can be determined by the pulse rise time. Using the standard rule-of-thumb, 0.35 / rise time, you can calculate the bandwidth which will be valid in the frequency domain. For a rise time of 1 nanosecond, this is about 350 MHz. For 0.01 nsec, this would include tones up to 35 GHz.

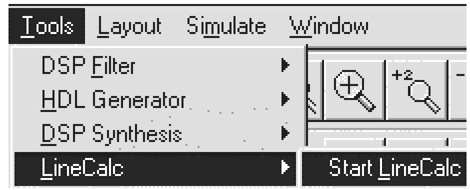
Eqn bw=0.35/1e-9

bw
3.500E8

e. Calculate the BW in the data display as shown here but do it for a rise time of 0.01 nsec.

Next Steps – At this point you can use LineCalc to 1) analyze the impedance of the MLIN just measured and 2) synthesize a line that is a match to the load based on your substrate and the frequency of interest.

8. Use LineCalc to determine the MLIN impedance and to create a matched line



a. From the schematic window start LineCalc: **Tools > LineCalc > Start LineCalc**. The main window will appear as shown here.

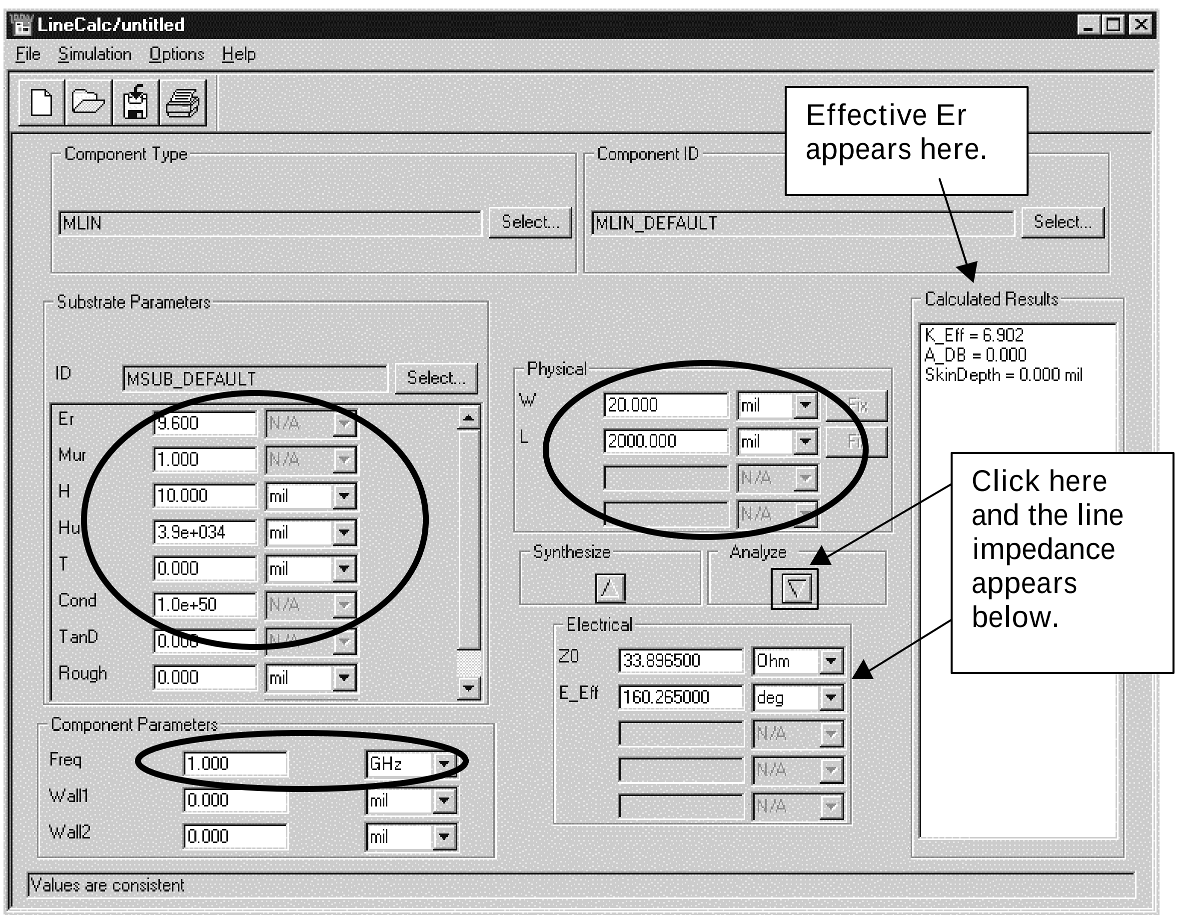
NOTE on LineCalc: Notice that the default component type is MLIN and that the default substrate is an ADS MSUB but it may <u>not</u> be set to the values on your schematic.

b. **Substrate Parameters**: Set the MSUB values to your schematic MSUB values, such as H=10, etc.

c. **Component Parameters**: Set Freq to 1 GHz (this will be easier to use with the reference line and the numbers will be easier to use).

d. **Physical**: Set the width and length of the MLIN to be analyzed. Here, set W = 20 mils and L = 2000 mils.

e. Click the **Analyze** button and the Electrical characteristics of the line will be calculated as characteristic impedance Z0 and E_Eff which is the effective electrical length in degrees.

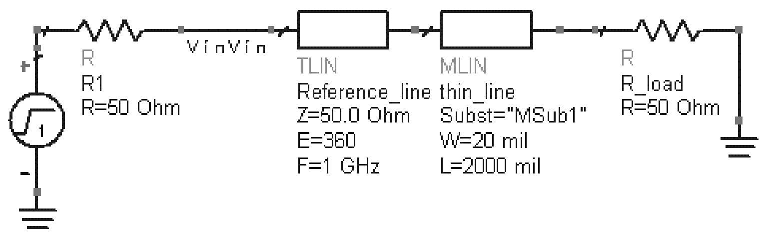


Note on MLIN Z0 and E_Eff – As you can see, the analyzed value of the line is not 50 ohms but about 34 ohms with an effective electrical length of about 160 degrees.

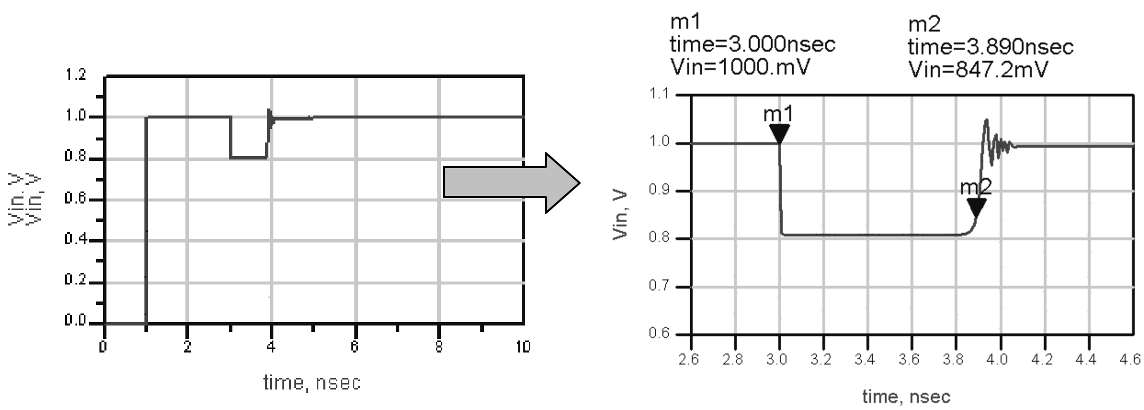
9. Minimize LineCalc (you will use it later)

10. Reset the circuit and set up equations to calculate distance

a. Reset the circuit to contain only the 20x2000 mil MLIN.



b. Simulate again and zoom into the plot where the MLIN response is shown. Insert two markers on either side of the response – marker m2 should be placed before the ringing (due to dispersion) begins.



c. In the data display, insert 4 equations as shown here. Er and velocity are required to calculate the speed of the signal through the line, based on the dielectric constant and the speed of light (3x10e8 meters/sec). Afterward, list the two distance equations.

Eqn Er = 6.902

Eqn velocity = 3e8 / sqrt(Er)

Eqn distance_mtrs = (velocity * (indep(m2) - indep(m1))) / 2

Eqn distance_mils= distance_mtrs * (1e5/2.54)

distance_mtrs	distance_mils
0.051	2000.599

d. You can move the marker notice the variation in the results.

11. Add more equations to calculate rho and line impedance

a. Add the following equations to the data display. Z_line will be the calculated impedance at any point where you move a maker (m3). Zo is the specified system impedance and rho (reflection coefficient) is based on the reference voltage which is ½ the source voltage indexed to the first value in the dataset [1].

Eqn z_line = Zo * ((1 + rho) / (1 - rho))

Eqn Zo = 50

Eqn rho = (m3 - ref_v) / (ref_v)

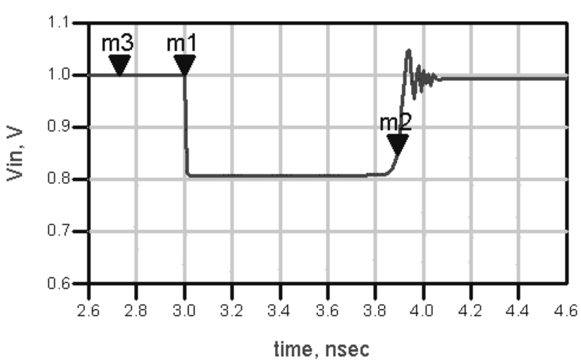
Eqn ref_v = source_v [1]/ 2

b. Add another marker (M3 shown here) to the plot of the DUT MLIN (20x2000 mils) and move the other marker text (m1 and m2) aside.

c. Insert a list with rho, z_line, and ref_v. Move the marker (m3) across the trace and you will see how the values change. This means that you can measure rho or Z_line at any point in the circuit.

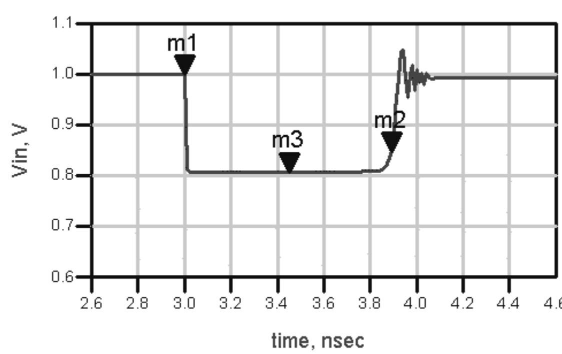
m3
time=2.730nsec
Vin=1.000 V

rho	z_line	ref_v
0.000	50.000	1.000



m3
time=3.450nsec
Vin=808.3mV

rho	z_line	ref_v
-0.192	33.916	1.000



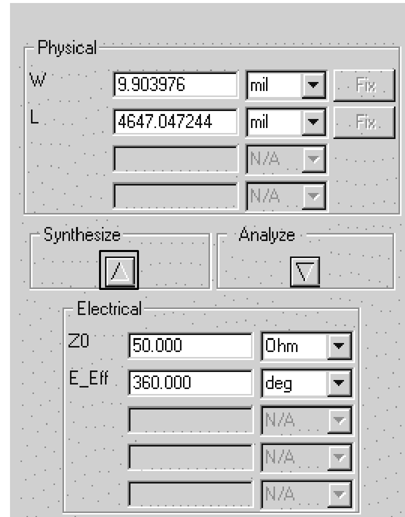
d. Change the DUT MLIN width to 5 mils or add two DUTS in series (as before) and use this technique to measure the response.

12. Use LineCalc to synthesize a matched line

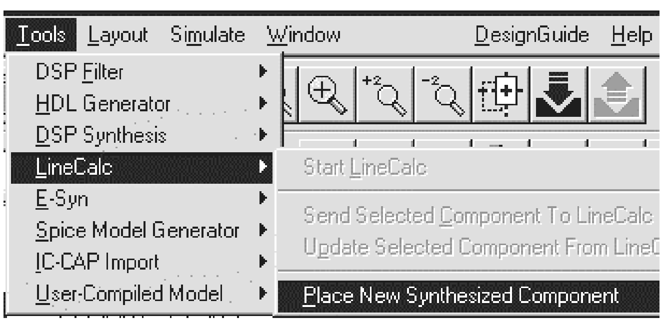
a. Go back to the LineCalc program and reverse the process by entering the value of impedance and electrical length in degrees for a 1 GHz signal, using the same substrate definition:

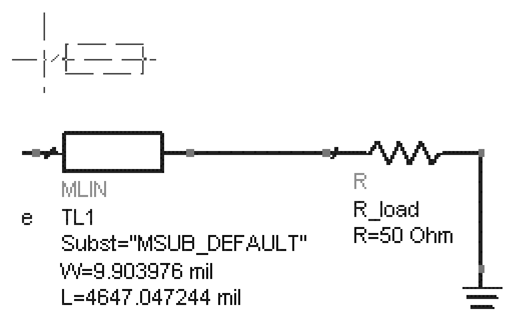
b. Click the Synthesize button and the program will generate the length and width as shown here.

c. Return to the Schematic and remove the existing DUT MLIN.



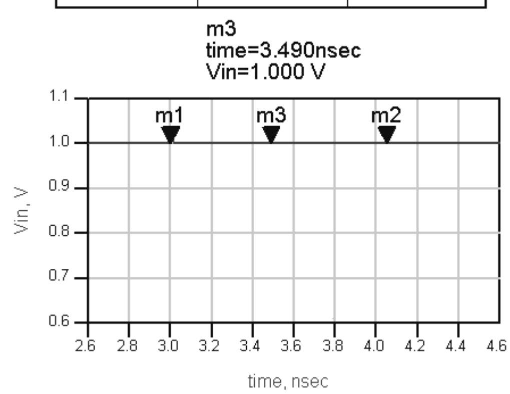
d. In the schematic menu bar, click the command**: Tools > LineCalc > Place New Synthesized Component**. The component will be attached to you cursor. Go ahead and insert it as shown.





e. Simulate again and you should see that the new MLIN is a perfect match as shown in the plot here: no discontinuity, rho is very close to zero and z_line = 50 ohms.

rho	z_line	ref_v
1.682E-5	50.002	1.000



13. At this point, close LineCalc and and save the schemtatic and data display.

EXTRA EXERCISES:

1. Insert several lengths of line into the existing schematic and measure the response as in the lab steps: delay and TDR.

2. Create a filter using the microstrip components library and then measure the delay as in the first part of the lab.

3. Try writing an expression and listing VSWR: 1 + mag (rho) / 1 – mag (rho) for the MLIN of 20x2000 – it should be about 1.5.

4. Example file for TDR: Create a new project directory copy the example project named RF_Board/TDRmeas_vs_model. Run the simulation and then change or tune the transmission line values (L and W) to see the response. Also, if you make all W and L and H (lines and substrate) 1/10 of their values, you should get the same results.