SPICE Simulation of Transmission Lines by the Telegrapher's Method

Part 3: Putting the Telegrapher's Equations into a Usable Sub-Circuit

Roy McCammon, Senior Engineer, 3M Communication Markets Division

Transforming the Circuit

While the circuit shown in Part 1, Figure 2 could be simulated directly, it is not optimal for SPICE.
SPICE has some idiosyncrasies:

- Current sources are better than voltage sources.
- Laplace transform components work better if they converge to zero at high frequencies.
- An inductor with an internal series resistance works better than a separate inductor and resistor.

Currents are proportional to voltages. Voltage sources can be replaced with current sources. Voltage dependent sources can be replaced with current dependent sources. By applying some circuit theory transformations, the circuit in Figure 2 can be converted to the circuit in **Figure 3** by the successive application of equivalence transformations. This circuit looks greatly different from the circuit on Figure 2, but the terminal voltages and currents are the same.

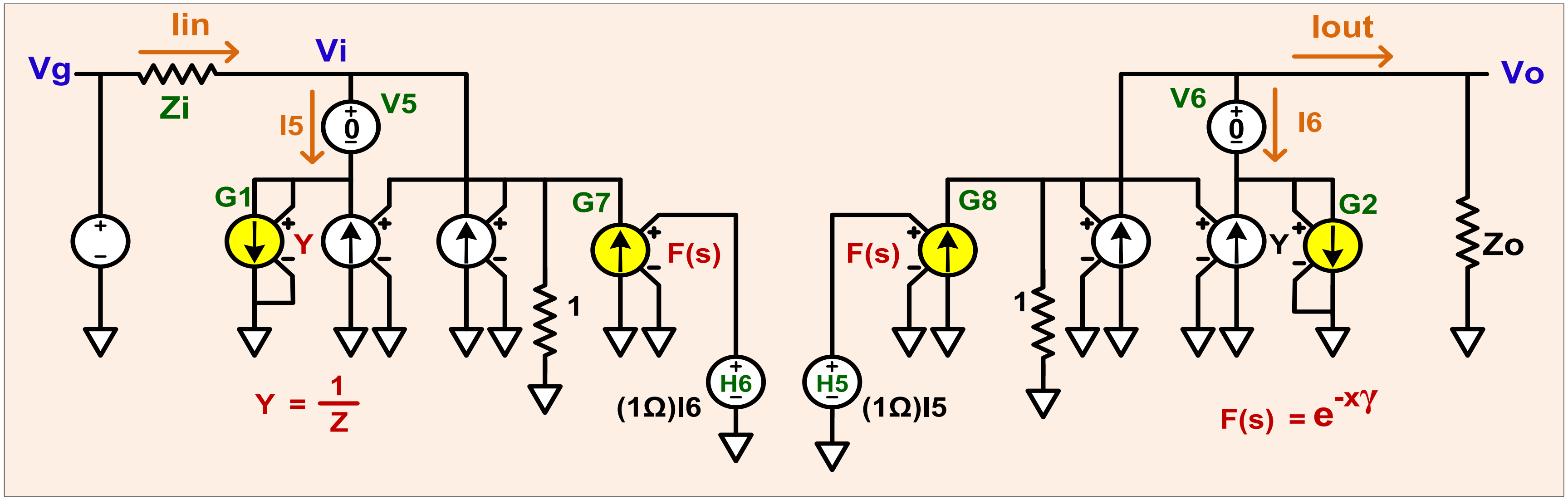


Figure 3. Two-port implemented with G components. The Laplace components are shown in yellow.

There are 4 Laplace components: G1, G2, G7 and G8. G1 and G2 approach Z_∞ instead of zero at high frequency. G7 and G8 do go to zero, but not quickly. Some improvement is needed.

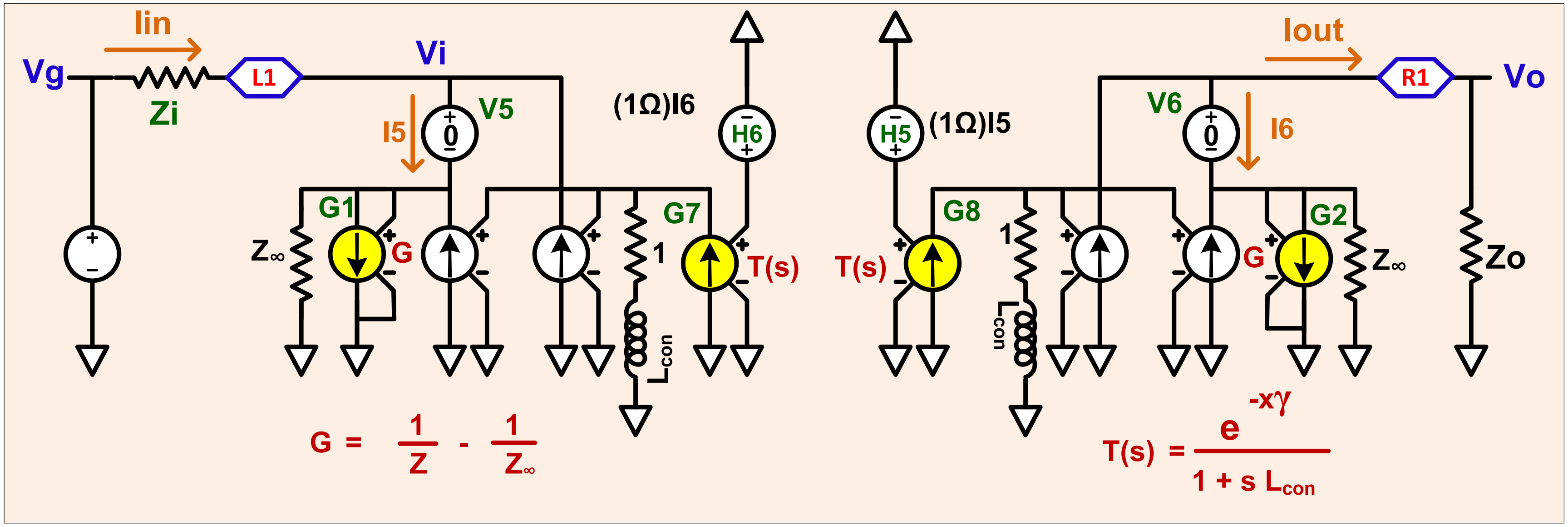


Figure 4. Unbalanced single mode transmission line in SPICE friendly form. The Laplace components are shown in yellow.

Examination of the Laplace parameter of G1 and G2 reveals that it converges to a real constant $1/Z_\infty$ which is a conductance. This constant will be subtracted out of the Laplace expression and added back as an ordinary resistor with impedance of Z_∞. G1 and G2 will now converge to zero. To make G7 and G8 roll off more rapidly, the Laplace expression will be divided by $(1 + s\ L_{con})$ which will cause the Laplace functions of G7 and G8 to converge quickly to zero. Then, to get the gain back, the output voltage of G7 and G8 is developed across an impedance equal to $(1 + s\ L_{con})$.

The result is shown in **Figure 4**. The 1-ohm resistor in series with L_{con} will be absorbed into L_{con} as a series resistance. The Tables' L1 and R1 will become the terminals of the sub-circuit. What value should be used for L_{con}? That will be discussed a little further down.

The DC case

Does the model work at DC? It is not obvious from the unfamiliar forms that the solution takes. The characteristic impedance is very high. How, one might ask, can one get any signal into the transmission line at all? The answer is that the line is mismatched at both ends and the reflections from the far end diminish the impedance at the near end and the reflections at the near end diminish the impedance at the far end. The two ends swap geometrically decreasing reflections that, with a little smoothing, look just like an RC-charging curve. The derivation and solution of the Telegrapher's Equations makes no assumption about the frequency of the signals and does not exclude DC. It ought to work. And in fact it does.

How about SPICE? Does it work with this model at DC? The answer is yes for AC analysis, a qualified yes for transient analysis and a conditional yes for DC operating-point analysis.

In the AC analysis, you cannot specify 0 Hz, but you can make the low frequency as low as you want (even one cycle per year ~ 31 nHz). The model works in the AC analysis with the understanding that the frequency must be positive.

In the transient analysis, you put a step function (from zero to one, for example) and you see the waveform settles down to a sustained DC level. The model appears works in the transient analysis.

What about DC operating point? If the equations work and the model works and the transient and AC analysis work then the operating point analysis ought to work. Right? Yes, it ought to work, but in some cases it does not. But it can be made to work, by increasing the inductance of $L_{\mathbf{con}}$.

But, how do we feel about this? We feel disquieted. We are of the opinion that this guiding principal should hold: if the model is a sufficiently accurate model of the physical system, then the simulator ought to produce accurate results. Thus, we should expend our effort on accurate modeling and not on appeasing the simulator. Once we get accurate simulation, we would be willing to tweak the model to run faster.

We are not bothered by documented, universal principals such as current sources are preferred to voltage sources or that Laplace functions should go to zero for infinite frequency. We are not bothered by undocumented, universal principals that we have discovered, such as balanced circuits should be modeled by balanced models (which also is in accordance with our guiding principal of attempting to accurately model the physical system). We are a little bothered if we have to have a tweak, such as $L_{\mathbf{con}}$, that may need to be adjusted depending on rise times and transmission line length. But we are greatly bothered by having to change the value of the $L_{\mathbf{con}}$ for different types of analysis.

Choosing L_{con}

How can L_{con} be determined? We offer the following suggestions:

Set up a simple operating-point circuit. Use a long (100 kft) transmission line. Drive it with a 1 Vdc source. Terminate it with Z_{dc}. Run the operating point analysis. The output should be:

8a. $$Vo = e^{-x\gamma_{dc}} = 0.9839$$

where

8b. $$Z_{dc} = \sqrt{\frac{R_{dc}}{G_{dc}}} \approx 324\ k\Omega$$

8c. $$\gamma_{dc} = \sqrt{(R_{dc})\,(G_{dc})} \approx 162\ u\ /Kft$$

8d. $$X = 100\ Kft$$

Reduce L_{con} as small as possible and still get acceptably close.

Set up a simple transient-analysis circuit with the length less than half of the shortest line you want to simulate. Match it at the source and leave it unterminated at the load. Drive it with a 1-volt step with a rise time about 1/10 of the propagation delay of the transmission line. Increase L_{con} if needed to suppress anything weird. Weird things that we have seen include:

1. Bursts of large amplitude chaotic oscillation, usually after the simulation has converged to its final value.
2. Simulations that go on for a while outputting reasonable results, and which suddenly hang.
3. Cases where the delay was not proportional to the length of the transmission line.
4. Cases where the results have frequencies much higher than the input.

You now have the minimum L_{con}. Now increase L_{con} until something changes significantly. If you can find a range of values for L_{con} of, say, 10-to-one, where the outcome of a particular analysis does not change significantly, then set L_{con} for the geometric mean of the range and use that value for that analysis. Certainly, when you do run any analysis you should try both twice and half the value for L_{con}.

Verifying the Simulation

First, an AC simulation is run to see if the two-port accurately reproduces the data from Table 2.

AC Simulation

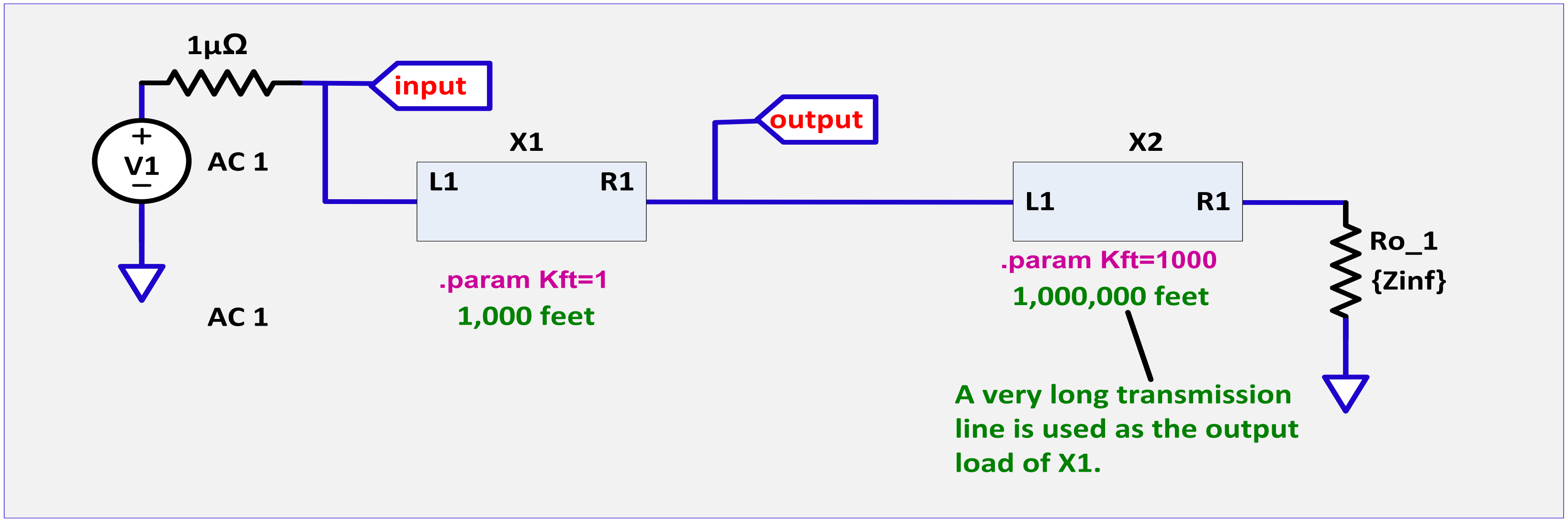


Figure 5. Frequency Response Schematic

Figure 5 depicts the frequency response circuit. The SPICE program we use supports hierarchical schematics. Our brilliantly designed two-port has been reduced to a rectangle with two terminals. But this allows us to concentrate on the application.

Table 2 gives loss in dB/Kft, so we will simulate a 1000-foot section. The propagation function is defined as the input-to-output ratio when the transmission line is terminated by its characteristic impedance. We will use a longer transmission line of the same type for the termination.
In the circuit of Figure 5, X1 is the 1000 foot section. X2 (1 million feet) serves as the load for x1.The results of the AC simulation are shown in **Figure 6**. The actual loss conformed exactly to the data in the Secondary Parameter Approximations columns of Table 2. For comparison, we run the same circuit using the SPICE O component with R, L and C from the highest frequency in Table 1. The O component is accurate for only one frequency.

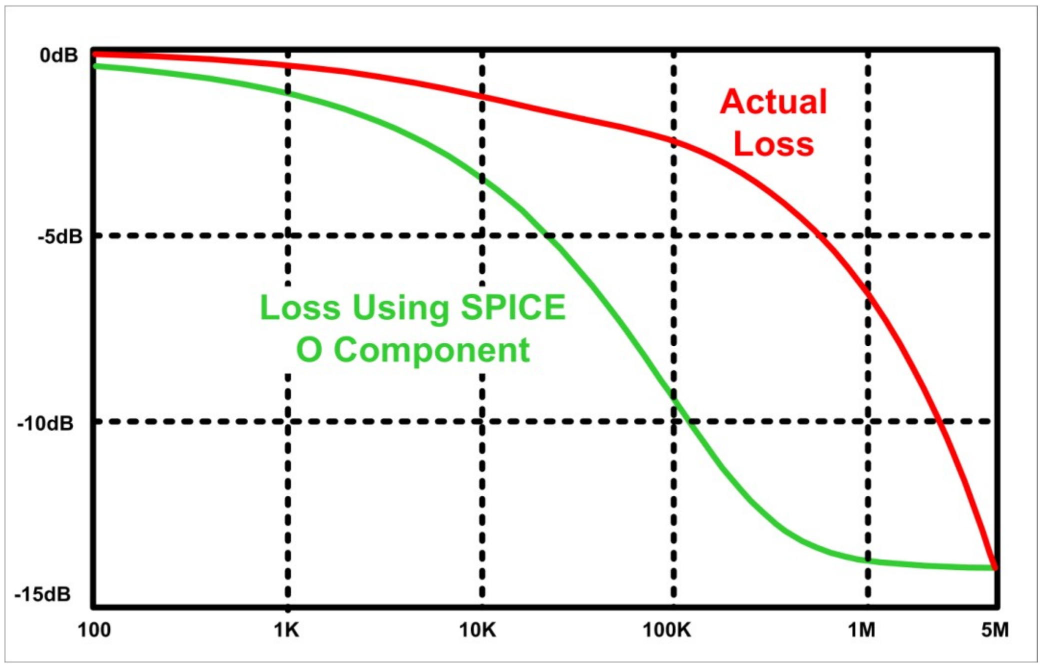


Figure 6. Matched Load Frequency Response Simulation: Telegrapher's Model in red and SPICE O component in green.

Transient Simulation

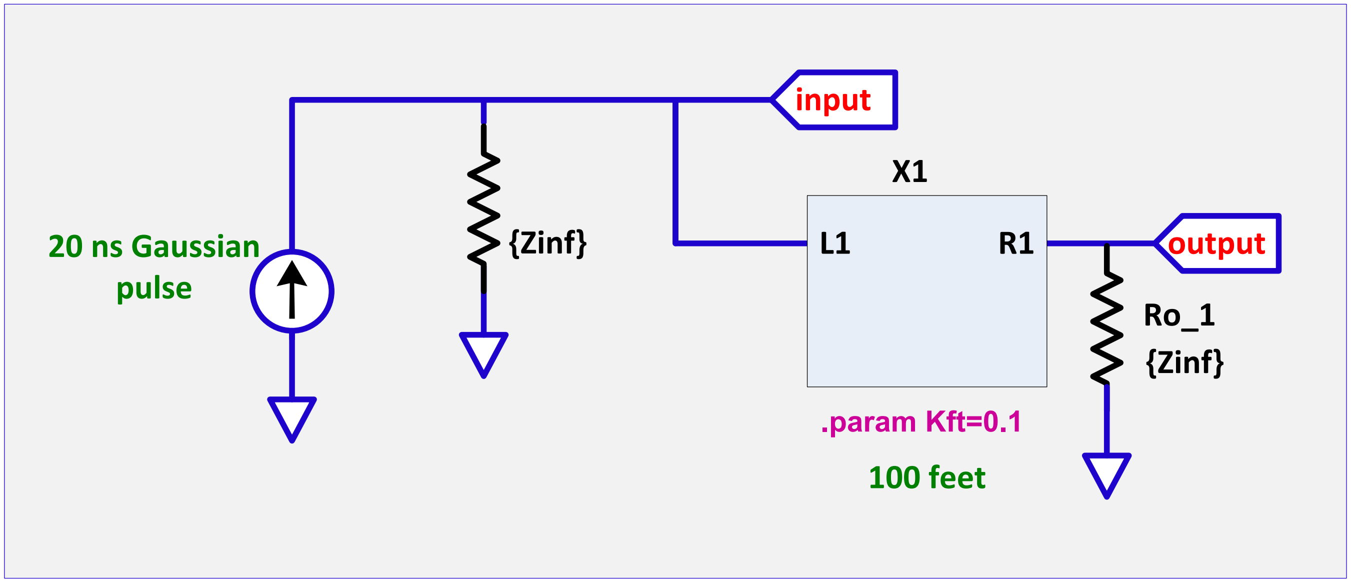


Figure 7. Transient Response Schematic.

We will use the circuit of **Figure 7** to try the Telegrapher's Model with transient simulation. The source, Ig, generates a Gaussian pulse of 20 ns width. We will try three cases: matched load, shorted load and open load. The circuit in Figure 7 has a matched load. Actually Z_{inf} is not a perfect match, except at infinite frequency, but it is close enough.

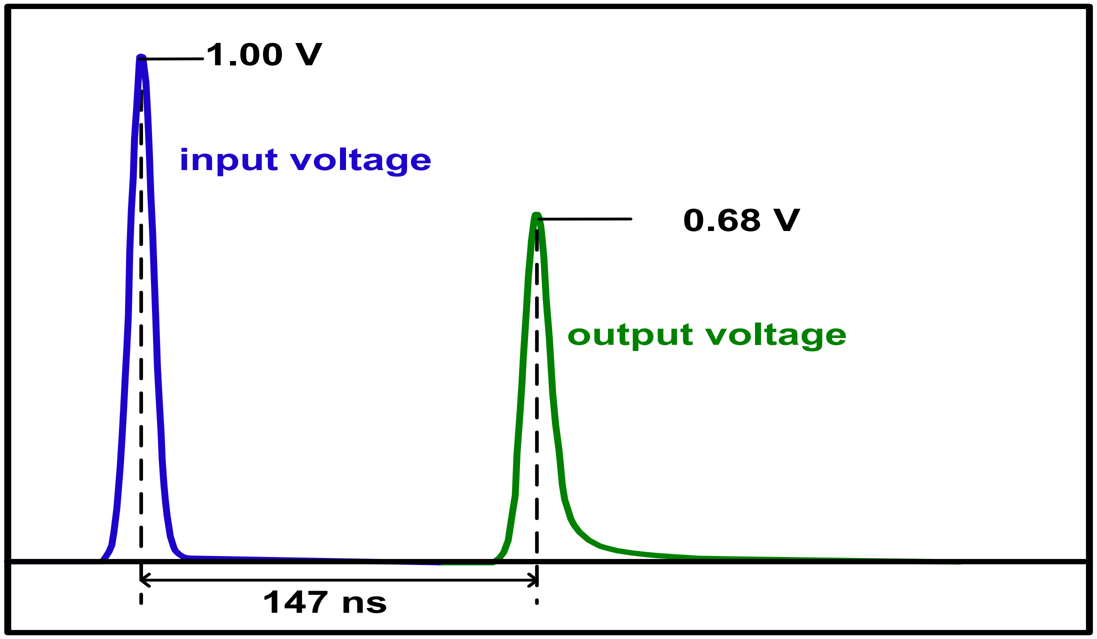


Figure 8. Matched Load Transient Simulation. Input and output voltages are in blue and green respectively.

Figure 8 shows the results for the matched load. The output pulse shows up about 147 ns after the input and there is no significant reflection. The output pulse is delayed, attenuated and slightly dispersed.

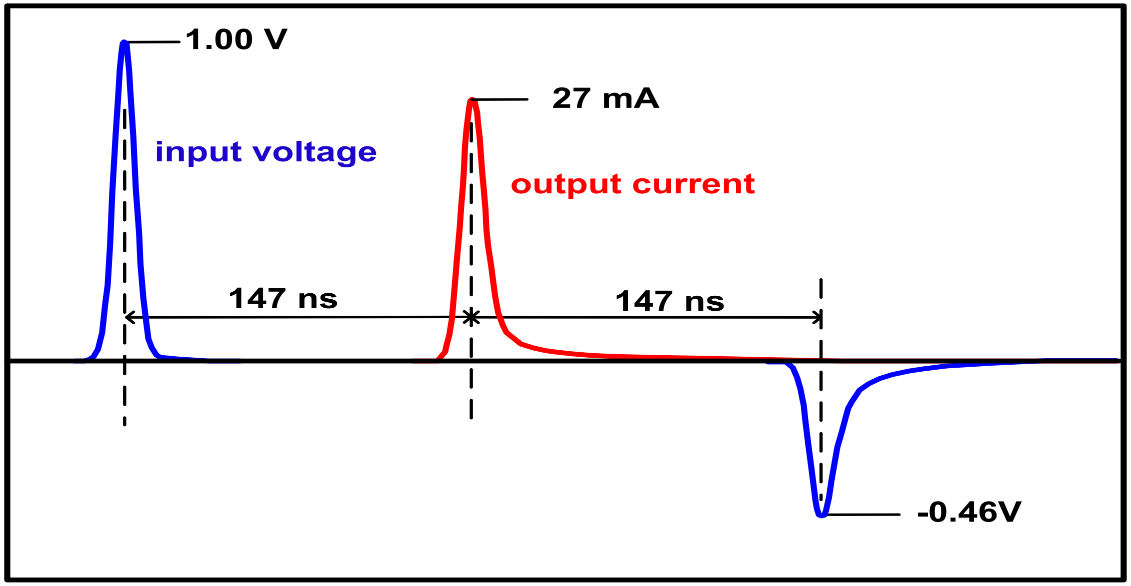


Figure 9. Shorted Load Transient Simulation. Ro_1 is changed to 1μΩ. Ro_1 current shown in red.

Figure 9 shows the results of a short-circuited load. We use 1 μΩ for a short circuit because it provides a component for measuring current. The current pulse shows up at about 147 ns after the input pulse and a reflected pulse of opposite polarity is seen at the input at the 294 ns later. The pulses show increasing dispersion as the travel time gets longer.

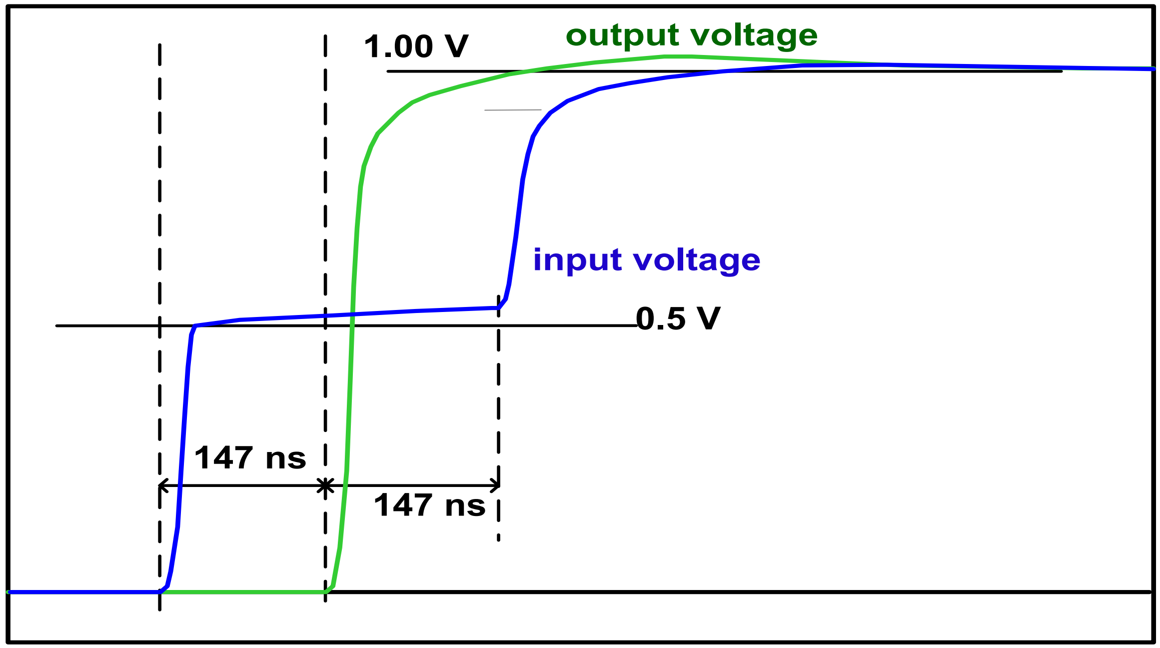
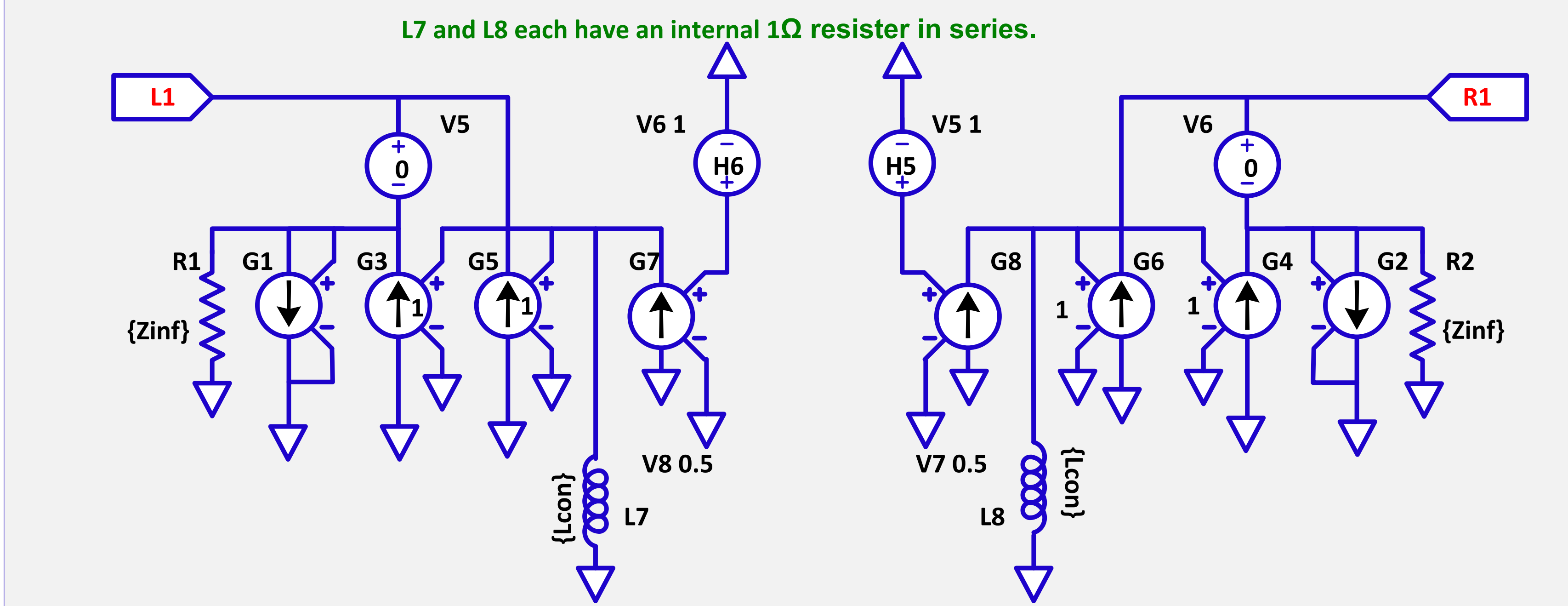


Figure 10. Open Load Transient Simulation with step input. Ro_1 changed to 10 kΩ.

Figure 10 shows the response to a one-volt step input on a line that is matched at the source and unterminated at the load. This is commonly called a "series termination" and is sometimes used for logic signals where there is a single source and a single load. Even though the input takes two steps and 294 ns to reach 1 V, the output cleanly swings to one volt only 147 ns after the input.

The sub-circuit

The last thing we will do (and the first thing we promised to do) is to put all this circuitry into a sub-circuit that you can drop onto your SPICE schematic and get accurate time and/or frequency domain simulations. **Figure 11** shows all of and nothing but the final sub-circuit.



Laplace=Exp(-Kft*((((Rdc*(1-(s/wR)^2)^.25)+s*(Linf+Ldel/(1+A*((-(s/wL)^2)^.5)-(s/wL)^2)^.25))*(Gdc+G2*(-(s/w2)^2)^k+s*C))^.5))/(s*Lcon+1) ; /* LT for G7 */

Laplace=Exp(-Kft*((((Rdc*(1-(s/wR)^2)^.25)+s*(Linf+Ldel/(1+A*((-(s/wL)^2)^.5)-(s/wL)^2)^.25))*(Gdc+G2*(-(s/w2)^2)^k+s*C))^.5))/(s*Lcon+1) ; /* LT for G8 */

Laplace=(((((Gdc+G2*(-(s/w2)^2)^k)+s*C)/((Rdc*(1-(s/wR)^2)^0.25)+s*(Linf+Ldel/(1+A*((-(s/wL)^2)^0.5)-(s/wL)^2)^0.25)))^0.5)) -Yinf ; /* LT for G1 */

Laplace=(((((Gdc+G2*(-(s/w2)^2)^k)+s*C)/((Rdc*(1-(s/wR)^2)^0.25)+s*(Linf+Ldel/(1+A*((-(s/wL)^2)^0.5)-(s/wL)^2)^0.25)))^0.5)) -Yinf ; /* LT for G2 */

Figure 11. Unbalanced Transmission Line Sub-Circuit Schematic.

The one-ohm resistors in series with L7 and L8 have been absorbed into those inductors as an internal series resistance. The Laplace expressions for G1, G2, g7 and G8 are shown at the bottom of the schematic. The chances of typing those expressions without error is low, so we are going to help you out with that.

When SPICE was created, there were no graphical user interfaces (GUI's). Everything was entered as a text file using a plain old word processor or even punched cards. The syntax was standard and it was easy to swap files. Various vendors have added GUI's with non-standardized file structures. Each flavor of SPICE has its own unique method for creating sub-circuits, creating symbol that represent the sub-circuit and passing parameters to those sub circuits. So, we cannot tell you how to create a sub-circuit on your version of SPICE. But we will help you.

The old SPICE with its text files still exists underneath the GUI. Sub-circuit text files are still the most common way to provide SPICE models. Every SPICE GUI implementation has a way of accepting sub-circuit files as text and allowing its users to create symbols that link to those sub-circuit text files. If you request a SPICE model of an op amp or diode, you will get a plain old text file that you have to somehow wrap up into a graphical symbol. The odds are that if you have read all three parts of this article then you know how to do this already. So, open your word processor, go to the WWW version of this article, copy and paste the .param statement and the sub-circuit definition into your word processor and make the necessary manipulations.

Or, if you want to create your sub-circuit graphically, we've done the hard part for you. Look at Figure 11 and draw it graphically. The hard part is the .param statement and the Laplace expressions. Once again the way to do that is to copy and paste it from the web version of this article. If you don't know how to do this, the Yahoo group listed as a resource is a good place to ask for help.

Here is the sub-circuit net list. Note, in a .param statement, ** means exponentiation. In a Laplace expression ^ means exponentiation. The following text (shown in green courier font) is what you can copy and paste into your simulation. Once you get your sub-circuit entered, set the Kft parameter to the length of your transmission line and you are ready to simulate.

```
.param Kft=1                ; 1 Kft = 1000 feet
.param Lcon=10n             ; convergence inductance
+ C=15.72e-9                ; the value of capacitance at dc
+ Gdc=0.5n                  ; the value of conductance at dc
+ Rdc=52.50                 ; the value of resistance at dc
+ Ldc=0.1868e-3             ; the value of inductance at dc
+ Linf=0.133e-3             ; inductance at infinite frequency
+ Ldel=(Ldc-Linf)           ; inductance parameter
+ Zinf=(Linf/C)**0.5        ; characteristic impedance at infinite  frequency
+ Yinf=1/Zinf               ; characteristic conductance at infinite frequency
+ F2=5e6                    ; the highest frequency in Hz
+ W2=6.28318*F2             ; the highest frequency in rad/sec
+ G1=23u                    ; the value of conductance at F1
+ G2=36u                    ; the value of conductance at F2
+ Rac=304.62                ; the value of resistance at F2
+ F1=3e6                    ; the second highest frequency in Hz
+ A=1.6                     ; inductance parameter
+ k=Log(G2/G1)/Log(F2/F1)/2 ; conductance parameter
+ WL=6.28318*161000         ; inductance parameter
+ WR=W2*(Rdc**2)/(((Rac**4)-(Rdc**4))**0.5) ; resistance parameter

.subckt single_mode_xline L1 R1
G1 N1 0 N1 0 Laplace=(((((Gdc+G2*(-(s/w2)^2)^k)+s*C)/((Rdc*(1-(s/wR)^2)^0.25)+s*(Linf+Ldel/(1+A*((-(s/wL)^2)^0.5)-(s/wL)^2)^0.25)))^0.5))-Yinf
G2 0 N1 N2 0 1
G3 0 L1 N2 0 1
V1 L1 N1 0 Rser=0
H1 N4 0 V1 1
G4 N6 0 N6 0 Laplace=(((((Gdc+G2*(-(s/w2)^2)^k)+s*C)/((Rdc*(1-(s/wR)^2)^0.25)+s*(Linf+Ldel/(1+A*((-(s/wL)^2)^0.5)-(s/wL)^2)^0.25)))^0.5))-Yinf
G5 0 N6 N5 0 1
G6 0 R1 N5 0 1
V2 R1 N6 0 Rser=0
H2 N3 0 V2 1
R1 N6 0 {Zinf}
R2 N1 0 {Zinf}
G7 0 N2 N3 0 Laplace= Exp(-Kft*( (((Rdc*(1-(s/wR)^2)^.25)+s*(Linf+Ldel/(1+A*((-(s/wL)^2)^.5)-(s/wL)^2)^.25))*(Gdc+G2*(-(s/w2)^2)^k+s*C))^.5))/(s*Lcon+1)
L1 N5 0 {Lcon} Rser=1
G8 0 N5 N4 0 Laplace= Exp(-Kft*( (((Rdc*(1-(s/wR)^2)^.25)+s*(Linf+Ldel/(1+A*((-(s/wL)^2)^.5)-(s/wL)^2)^.25))*(Gdc+G2*(-(s/w2)^2)^k+s*C))^.5))/(s*Lcon+1)
L2 N2 0 {Lcon} Rser=1
.ends single_mode_xline
```

References

1. *Engineering Electromagnetics*, 4th edition by William Hayt, 1981. Chapter 12, section 1, "The Transmission Line Equations".
2. Wikipedia.org, "Primary line constants", "telegrapher's equations" and "propagation constant".
3. *Subscriber Loop Signaling and Transmission Handbook*, by Whitman D. Reeve, 1995, IEEE Press.
4. *DSL Simulation Techniques and Standards Development for Digital Subscriber Line Systems*, by Walter Y. Chen, 1998, Macmillan Technical Publishing.
5. *Engineering Electromagnetics*, 4th edition by William Hayt, 1981. Chapter 12, section 3, "Transmission-Line Parameters".
6. *Electric Transmission Lines*, by Skilling, 1951. Chapter 7.
7. *Fields and Waves in Communication Electronics*, by Ramo, Whinnery and van Duzer, 1965. Chapter 5.
8. *Home Networking Basis: Transmission Environments and Wired/Wireless Protocols*, Chapter 2, section 1.2, by Walter Y. Chen, 2003, Prentice Hall.

Resources

1. Linear Technology Corporation LT Spice IV download available with a very generous license.
   http://www.linear.com/designtools/software/ltspice.jsp
2. Yahoo user's group
   http://tech.groups.yahoo.com/group/LTspice/

About the author

Roy McCammon is a senior engineer with 3M's Communication Markets Division, and a graduate of the University of Texas Department of Electrical Engineering. He has operated a satellite tracking station in Antarctica, designed astronomical instruments and telescope servos, and has spent the last 28 years designing test equipment used by telecommunications providers and thinking about transmission lines. His interests include dancing the two-step, pushing miniature battleships around a table top, and simulations.