

LAB GUIDE

TEM TRANSMISSION LINE – PART #1 (INCIDENT WAVES)

- Complete the pre-test in your TL Lab Workbook #1 before you start working on the lab exercises.
- The TL Lab Workbook #1 must be submitted to the TA for marking at the end of the 3-hour session. Late submissions are unacceptable.
- This Guide will help you work with the software (MEFiSTo-2D Classic).

Experiment #1: Propagation of a Gaussian Pulse in a Matched Lossless Line

NOTE: You will be required to make a number of changes to the projects you will be working with. You do not have to save the changed projects. You can modify a project and run it without saving it to the hard disk.

Step 1: Project Geometry

In this step, you will get acquainted with the structure to be simulated. If MEFiSTo-2D is not started for you already, you can do so by choosing

Start/Program Files/MEFiSTo-2D/MEFiSTo-2D

Go to **File/Open** and open the file `tline.tlm`. If you have difficulties locating the file, ask the TA for help.




What you see is a section of a parallel-plate line, which stretches along the z -axis. It supports a uniform plane wave, which has only two non-zero field components: E_y and H_x . The y -direction is orthogonal to the screen plot (going into it). In the parallel-plate line, E_y points from one strip-plate to the other, which is exactly the direction perpendicular to the screen plot. The x -direction is the one along the short side of the plate, i.e., the vertical axis of your screen (from top to bottom). The z -axis is from left to right on your screen. The computational region is defined as vacuum, i.e., $\mu = \mu_0$ and $\varepsilon = \varepsilon_0$.

Move the cursor over the 2-D structure and observe the digital counter in the **(Z-X)** Coordinate Window in the toolbar. Using this feature, you can measure the width and the length of the line in terms of number of cells. The cells are clearly visible. They form a 2D grid which the simulator uses to represent the otherwise continuous space as a discrete grid of very short transmission lines. The length of these very short transmission lines is exactly the size of the cell ΔL . Here, $\Delta L = 1$ mm.





You will also notice three probes (the green dots). The probes indicate points at which you can observe the desired field component as a function of time as well as its spectrum.

Measure the width and the length of the transmission line as well as the distances between Probe 1 and Probe 2 as well as between Probe 2 and Probe 3. Record your measurements in TABLE 1 of your Workbook.

Step 2: Field Animation Observations


You can start the simulation by clicking on the **Forward**  button in the toolbar. If you do not see the Gaussian pulse propagating, the simulation may not have been reset. See whether the **Reset**  button in the toolbar is active. If it is, click on it to erase all simulation data. Then click  again.

You should clearly see a Gaussian pulse propagating down the line. It represents the electric field E_y between the two parallel strips of the line. You are in fact observing the E_y field component in the x - z plane as time runs forward. The elapsed time can be observed in the lower left corner of the application window. *Answer Questions 1 and 2 in the Workbook.*

NOTE: To stop the simulation at any time, click the **Stop** button . To start the simulation again from the beginning, click on , and then click on  again. You can also run the simulator forward only one step at a time using **Forward One Delta_t** . Delta_t is a very small period of time indicating how often the field is being sampled in time by the simulator.

You can change the **Field Display Attributes** in the **Field** menu at any time. To do so, right click anywhere in the animation window and select **Field Display Attributes**. An additional list of options appears, which will allow you to modify the plot appearance.

Step 3: The Excitation Function

Reset the simulation by clicking . To inspect the characteristics of the excitation function, select **Source Waveform** from the main menu. You should be seeing a tick next to **Gaussian (f)**. Select it. Its characteristics will be displayed as follows: Magnitude: 0.707, Bandwidth [GHz]: 40, Gaussian-Modulated Carrier: Constant. Press **OK** and inspect the function plot. *Answer Questions 3 to 6 in your Workbook.*

Select again **Source Waveform** → **Gaussian (f)**. Change the bandwidth of the pulse to 20 GHz and click **OK** to inspect the function plot. *Answer Questions 7 and 8 in your Workbook.*


Select again **Source Waveform** → **Gaussian (f)**. Restore the bandwidth of the pulse to 40 GHz.


You can return to the **Field/3D** view at any time by a right click on your mouse, and by selecting **Field** from the list of options. Once in **Field**, again with a right-click, you can select a **2D** or a **3D** view option. Alternatively, select **View** → **Field**, then **Field** → **3D** from the main menu.

NOTE: There are three main menus in MEFiSTo-2D: the **Draw** menu (layout and settings), the **Field** menu (3D and 2D animation), and the **Graph** menu (probe and source waveforms). When one is active, the other two are deactivated on your main menu bar. However, you can conveniently toggle between them by right clicking anywhere in the current window, and by choosing the desired menu from the appearing list of options.

Step 4: Measuring the Velocity of Pulse Propagation

The original project in `tline.tlm` represents loss-free TEM line ($\epsilon_r = \mu_r = 1$, $\sigma = 0$). The phase velocity and the group velocity are identical in this case (assuming that numerical dispersion is not a factor). To measure the velocity of propagation, you will have to measure the time delay between **Probe 1** and **Probe 2**. You will also need the distance between the probes but you already have this information in TABLE 1.

Reset  the solver. You will re-run the simulation but this time you will observe different plots. Select **View** → **Graph**. Choose **Graph** → **Probe Responses**. You now see nine plots on your screen. The first column of three plots shows the measurements from Probe 1, the second column from Probe 2, and the third column from Probe 3. The top plots show the time-responses measured by the probes. The second row of plots show the amplitude spectra of the signals recorded by the three probes, and the third row of plots show their phase spectra.

Run the simulation by clicking . You will see three signals appearing one after the other (with some delay) in the top graphs.


It is time for measurements. Right click in the screen. This will display a list of options: click on **Probe 1** → **V(t)**. This will display the waveform of the Gaussian pulse, recorded by Probe 1. For better view, zoom in the peak of the pulse. To do so, left click and hold while pulling out a rectangular region around the pulse peak. Release the click to obtain a zoomed-in view. Click on the tip of the highest bar (the maximum of the pulse). Look at the **(Z-X)** Coordinate Window. The first number, when rounded off, will tell you the exact time step (sample number in time), at which the maximum occurred at Probe 1. **Record it in TABLE 2 in your Workbook**. Right click anywhere in the screen and select Probe 2. To see the time waveform recorded at Probe 2 the option **V(t)** must be selected. If it is not, you can select it again by a right-click from the available list of option. Read out the time step at which Probe 2 signal is maximum in the same manner as you did for Probe 1. **Complete TABLE 2 and answer Question 9**.

Step 5: Computing Distance from Phase Delay

By a right-click select for viewing Probe 1, **V(f)** phase. Read out the phase at 10 GHz by clicking on the respective point in the plot and observing the values in the **(Z-X)** Coordinate Window. Repeat for Probe 2, **V(f)** phase. **Fill in TABLE 3 in your Workbook and answer Question 10**.

Experiment #2: Propagation of a Sinusoidal Wave in a Lossless Line

Step 1: Setting Sinusoidal Excitation

Initialize the simulator with the **Reset** button . From the **Source Waveform** menu select **Sin(f)**. Set its characteristics as follows: Magnitude = 0.707, Frequency [GHz] = 6. Click **OK** and inspect the excitation waveform. **Answer Questions 11 to 13 in the Workbook**.

In the **Simulation Control Data** window (choose **Simulation Control** → **Control Data**) set the number of time steps to **2000** and the time steps between updates to **1**. Click **OK**.

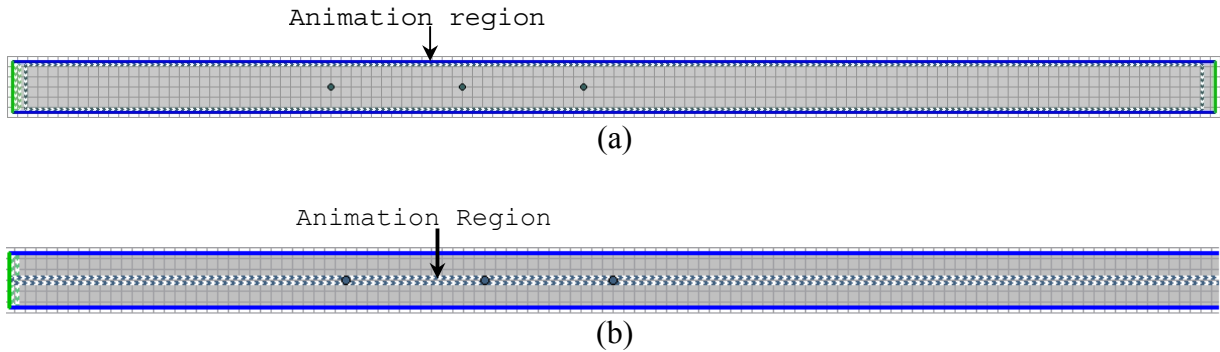






Fig. 1. The animation region must be only one cell wide in order to access **Field/2D** and the envelope plot. (a) The animation region before modification is 5 cells wide. (b) The animation region after the modification must be 1 cell wide.


Step 2: Field Animation Observations

To start the simulation, click **Forward** . The progress of the iterations is shown in the bottom left corner of the window, where the counter **Time Step Number** runs. You can stop the simulation when desired by pressing . *Answer Questions 14 and 15 in the Workbook.*

Step 3: Observing the Envelope of the Traveling Wave



To observe the envelope of the traveling wave, you will need to limit the animation area to one-cell width. Only then, the **Field/2D** animation feature of the simulator is accessible. First, you have to delete the current **Animation Region**, which is the rectangle with a dark-green/white contour (see Fig. 1). This is done by re-drawing it. Go back to **Draw** mode by selecting **View/Draw** or by right clicking and selecting **Draw**. Right click anywhere in the window and select **Animation Region** from the appearing menu. You have to draw a rectangle exactly on top of the existing rectangle with a dark-green/white contour. Drawing is done by a left click, which you have to hold while pulling out the rectangular region. If you are successful, the previous **Animation Region** will disappear. Now, you have to draw a new **Animation Region**. It should start right at the source region on the left, extend throughout the line along its axis, and finish where the transmission line ends. Also, it has to be only one cell wide as in Fig. 1.

You are now ready to observe the traveling wave and its envelope. Go to **View** → **Field**. Once **Field** is activated, choose **Field** → **2D**. You will now see a 2D plot: the line voltage vs. position along the line (in spatial steps ΔL). Click on the **Show Signal Envelope Display** button . Start the solver by a click on . Observe that the envelope of the traveling wave is constant along the line. This means that the maximum of the voltage along the line V_{\max} and its minimum V_{\min} are the same. This is expected because there is no reflected wave, only an incident one.

Note: You can reset the envelope whenever you wish during the simulation using the **Reset Signal Envelope** button .

Answer Questions 16 to 19 in the Workbook.

Step 4: Measuring the Frequency and the Wavelength of a Traveling Wave

You will measure the frequency of the propagating voltage using the amplitude spectrum of one of the probe's responses. Remember that each probe acts not only as an oscilloscope but also as a spectrum analyzer since it can display the magnitude and phase spectra of the recorded signal. When a sinusoidal wave passes through the probe, the recorded signal is clearly sinusoidal and we can expect a sharp peak in the magnitude spectrum at the frequency of the signal. To read out the frequency correctly, the simulation has to be re-run with a different setting for the Discrete Fourier Transform (DFT), which calculates the complex spectrum of the signal from its time waveform. Reset the simulator with . In the main menu, click on **Simulation Control** → **Control Data**. Under Frequency Domain Data, choose Number of frequency points as 100, Lower frequency as 5 GHz, and Upper frequency as 7 GHz. Select **Graph** → **Probe 1** → **V(f) Magnitude**. Re-run the simulation by clicking on . The amplitude spectrum gradually takes shape as more and more time samples are being added to the recorded waveform. The frequency of the sinusoidal signal is the one where the amplitude spectrum is maximum. Read this frequency by a click and a readout from the (Z-X) Coordinate Window. **Record the value in TABLE 4. Compare the value with the one which you set up previously in Step 1.**



The wavelength will be measured directly from the **Field/2D** plot of the voltage distribution along the line. To measure the wavelength, stop the simulation and select **View** → **Field**, then **Field** → **2D**. Now measure the distance from maximum to maximum using the readouts from the (Z-X) Coordinate Window. **Record your measurements in TABLE 4.** Compare the measured wavelength with the wavelength calculated using the frequency f , which you measured above, and the speed of the EM wave in the line (see Step 4, Experiment 1). **Answer Question 20.**

Experiment #3: Propagation of a Sine Wave in a Lossy Line

Replace the loss-free **Computation Region** by a new one of equal size, which has a specific conductivity $\sigma = 0.1$ S/m. To replace the **Computation Region**, proceed as follows.

- 1) Select **View** → **Draw**. Right click and select **Computation Region**. Draw a rectangle right on top of the existing computation region (the existing computation region is the grey-shaded rectangle, which covers the whole line and the source region on the left). This deletes the existing computation region. Draw the rectangle of the new computation region exactly where the previous one was.
- 2) In the appearing dialog box, enter the **Relative Dielectric Constant** as **1**, and the **Specific Conductivity** as **0.1**. Press **OK**.


Step 1: Observation and Measurement of the Attenuation of the Sinusoidal Wave

Start the simulation with . To observe the wave better, you may want to amplify the magnitude: right click and select **Field Display Attributes**. In the attributes bar click two or three times on **Magnify Y**. Reset  and run again. Now, switch your view to see the output of the three probes: **View** → **Graph**, then **Graph** → **Probe responses**. Reset and run simulation again. **Answer Question 21 in the Workbook.**

Fill in TABLE 5 in your Workbook. There, you will have to measure the magnitudes of the

voltage wave at Probe 1 and Probe 3, V_{m1} and V_{m3} . Also, you will need the distance between Probe 1 and Probe 3, $D_{1,3}$, which you can calculate using data from TABLE 1. This is sufficient to calculate the attenuation constant α in Np/m. In order to read out more accurately the wave magnitude at Probe 1 and Probe 3, you may want to change the view to Probe 1 (select **Graph** → **Probe 1** → **V(t)**) and then to Probe 3 (select **Graph** → **Probe 3** → **V(t)**). *Answer Questions 22 to 24 in the Workbook.*

Step 2: Observation of the Envelope of Attenuated Sinusoidal Wave

Reset simulator and switch to 2-D field view: **View** → **Field**, then **Field** → **2D**. Switch on the **Envelope Display Option** . Run the simulation. Once it is over, use the **(Z-X) Coordinate Window** to read out the envelope value at two positions as well as the positions themselves. It is up to you which positions you choose. Fill in *TABLE 6 in your Workbook and answer Question 25.*


Step 3: Measurement of the Wavelength of the Attenuated Sinusoidal Wave

In the **Field/2D** view, measure the wavelength as the distance between two neighbouring maxima. Remember that the horizontal axis of the plot is the distance along the line in terms of ΔL where $\Delta L = 1$ mm. *Fill in Table 7 in your Workbook and answer Question 26.*


Experiment #4: Propagation of a Gaussian Pulse in a Lossy Line

Reset simulator. Define a Gaussian pulse excitation by choosing **Source Waveform/Gaussian (f)**. Set **Magnitude** to 0.707, **Bandwidth in GHz** to 40, and the **Gaussian-Modulated Carrier** to Constant. Click **OK**. In **Simulation Control** → **Control Data**, set the **Total number of time steps** to 300.

Step 1: Observation of the Attenuated Traveling Pulse

You will observe the field (voltage) propagation and its decreasing envelope in the 2-D field view (**View** → **Field**, **Field** → **2D**). Start the simulation and display the envelope with . Now switch the view to see the three probe outputs: **View** → **Graph** → **Probe Responses**. *Answer Questions 27 and 28 in the Workbook.*

Step 2: Measuring the Attenuation of the Pulse

Go back to the 2-D field view (**View** → **Field**, **Field** → **2D**) where you will measure two points on the envelope (use ) with the **(Z-X) Coordinate Window**. *Fill in TABLE 8 in your Workbook and answer Question 29.*

Submit your Workbook to the TA now!