



MHARNESS

Crosstalk

**EMA3D-MHARNESS Primer for Creating a Basic
Crosstalk Simulation**

Prepared by Jason Stanko

August 2018

CONTENTS

Problem Overview	1
Preparation	1
Defining the Model Units.....	1
Creating the Harness Geometry	1
Defining the Simulation Domain.....	2
Meshing the Harness	3
Defining the Simulation Time Steps.....	3
Creating the Harness Structure.....	3
Route 1 – Shield.....	3
Route 2 – TSP1.....	4
Routes 3,4,5 – Conductors	5
Verifying Harness Cross-Sections	6
Creating the Excitation Sources.....	9
Constructing the Input Signals	9
Applying the Source	11
Creating the Probes	11
Voltage Probe.....	12
Finalizing the Harness Settings.....	12
Creating the MHARNESS Input File.....	13
Creating the EMA3D Input File	13
Running the Integrated Simulation	13



Problem Overview

This tutorial will introduce the user to the basics of setting up a MHARNES simulation. The user will learn the how to design a simple cable harness, create MHARNES sources, and create MHARNES probes. This training module assumes that the user has a basic understanding of the CADfix and EMA3D GUI. It is recommended to see other trainings as well to help develop stronger understanding of CADfix and EMA3D.

A basic cable harness composed of six MHARNES segments will be designed in this tutorial. The harness will contain one shield with a nested twisted shielded pair (TSP) and three adjacent conductors. A differential voltage source will be applied to the ends of the TSP inner conductors. The crosstalk between the TSP inner-conductor and one of three conductors adjacent to the TSP will be measured using a Voltage Probe. The harness hierarchy will look as followed:


- **SHIELD**
 - **TSP**
 - **TSP inner-conductor 1**
 - **TSP inner-conductor 2**
 - **CONDUCTOR 1**
 - **CONDUCTOR 2**
 - **CONDUCTOR 3**

Preparation

1. Under the Windows start menu launch **CADfix for EMA3D**
2. Click on the **New Model**  button
3. Click **Save**  and save the model to an appropriate place on your computer.

Defining the Model Units

We first need to define the model units for our problem.

1. Open the **MHARNES** tools list in the **EMA3D**  tab and click **Specify Units**
2. Select **Meters** in the pop-up window
3. Click **OK**

Creating the Harness Geometry

Our next task is to create the lines that will define the harness routes. To do this, we first need to define the harness points. We will do this through the CADfix command line.

1. Copy and paste the following into the CADfix command line:

```
PNT P1 0 0 0
```

```
PNT P2 4 0 0
PNT P3 -1 0 1
PNT P4 -2 -1 1
PNT P5 -4 -1 1
PNT P6 -2 1 1
PNT P7 -4 1 1
```

NOTE: More detailed information about using the CADfix command line usage and manipulating CADfix geometry can be found in *Introduction to EMA3D and CADfix* Tutorial.

Now we must connect the points. We will again do this through the CADfix command line.

2. Copy and paste the following into the CADfix command line:

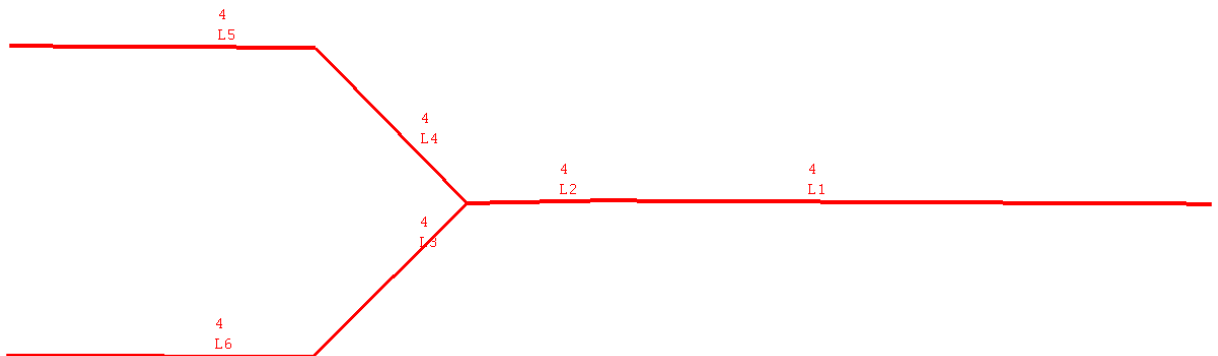
```
LINE L1 P1 P2
LINE L2 P1 P3
LINE L3 P3 P4
LINE L4 P3 P6
LINE L5 P6 P7
LINE L6 P4 P5
```

It would be beneficial to plot the lines and their names

3. Write the following into the CADfix command line:


```
PLOT LA ALL
```

Our Harness geometry should like the image below




Defining the Simulation Domain

Now that our geometry has been defined. It is time to define the computational domain or Lattice.

1. Click **Define Lattice** in the **MHARNES**s tools list of the **EMA3D**  tab
2. Click on the **Lattice Parameters** tool in the expanded section.
3. Change the cell size to **0.05** for the **x**, **y**, and **z** directions.
4. Click **Apply**.


Meshing the Harness

Next, we need to mesh the harness geometry.

1. Click **Mesh Geometry** in the **MHARNES** tools list of the **EMA3D**  tab to expand the section.
2. Click **Setup**.
3. Click **Mesh**.


Defining the Simulation Time Steps

Next the simulation time steps must be defined. Our simulation will run to $1\mu\text{s}$ with a time increment of $3.0\text{E-}11$.

1. Click **Define Time Step** in the **EMA**  tab .
2. Click **Time Step**.
3. Enter $3.0\text{e-}11$ into the box labeled **Time step**.
4. Type $1.0\text{e-}6$ in the box labeled **Final Simulation Time**.

Creating the Harness Structure

Now that the top-level harness geometry has been created, we can construct the harness cross sections by defining the routes that will inhabit each segment.

- Click **Harness Editor** in the **MHARNES** tools list of the **EMA3D**  tab to launch the MHARNES Harness Editor.

Route 1 – Shield

The first route we will create is a top-level shield.

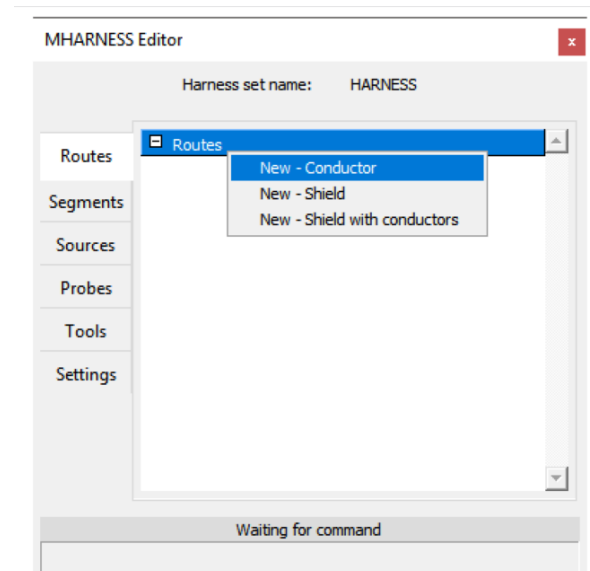
1. Right click on **Routes** in the **Route** tab of the **Harness Editor** and click **New - Shield** in the pop-up menu.
2. Type **SHIELD** in the box labeled **Route Name**.

We need to select the segments which will define the shield's path. The shield will occupy all segments in the harness.

3. Select Lines **L1 – L6**.

Next, we need to define at which points this route terminates. Boundary conditions can be any combination of resistive, capacitive, and/or inductive. At a minimum, boundary conditions must be defined at all “open” ends of a route. We will use the default value of a purely resistive boundary condition for this route.

4. Do not change the **Boundary Condition** definition.

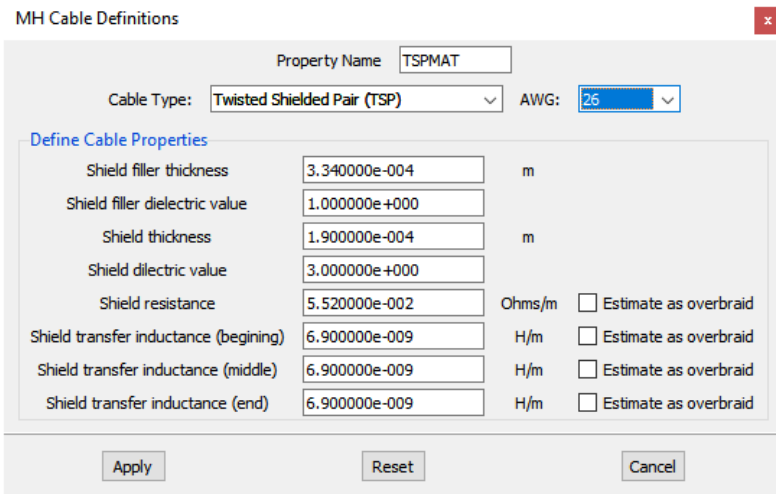
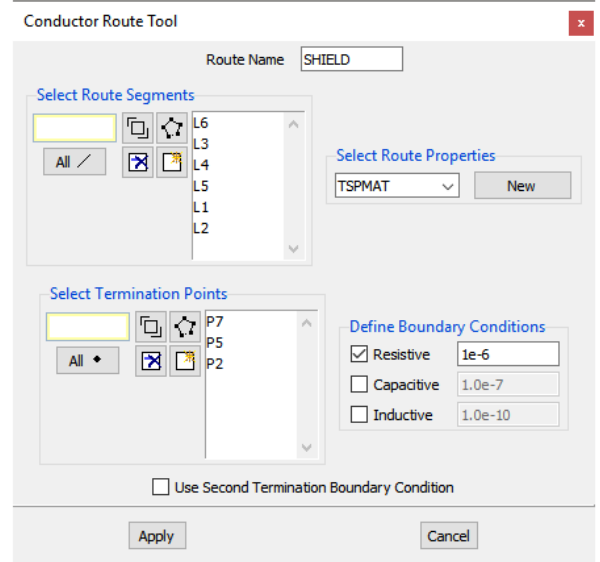


5. Select **Termination Points P5, P7** and **P2** for our route **SHIELD**.

NOTE: Although uncommon, junctions between segments can also be specified as termination points provided that all adjacent routes also have a boundary condition specified at that point.

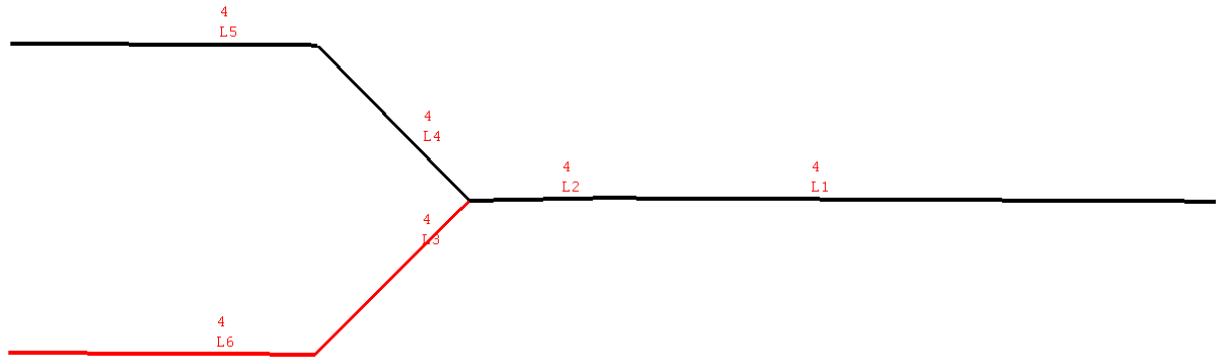
Finally, we need to specify the route's physical properties. We will select our shield's parameters to match that of a twisted shielded pair with 26-gauge inner conductors.

6. Click the **New** button under the **Select Route Property** section to launch the **Route Property Tool**.
7. Select **Twisted Shielded Pair (TSP)** from the **Cable Type** drop down menu.
8. Leave the **AWG** at **26**.
9. Type **TSPMAT** in the property name and click **Apply** to create the property.
10. Click **Apply** to create the route.
11. The route will now be added to the **Harness Editor**.



Route 2 – TSP1

We will now nest a twisted shielded pair inside of our shield. We will re-use the route property **TSPMAT** that we created earlier. However, we will modify the inner conductor resistances. The TSP will occupy the upper path of our harness shown in black below.



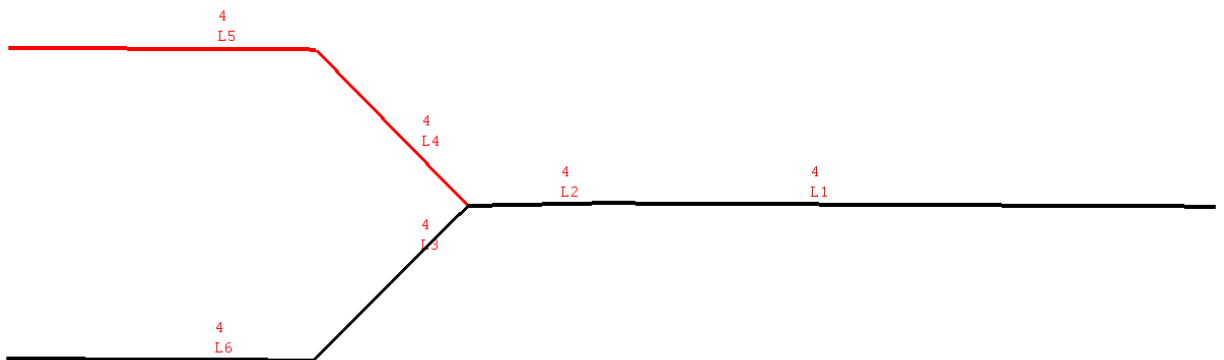
1. Right click on the **SHIELD** in the Harness editor and click **New Child – Shield With Conductors**.
2. Type **TSP** in the box labeled **Route Name**.
3. Select segments **L1**, **L2**, **L4**, and **L5**.
4. Select **Termination Points P7** and **P2**.
5. Right-click on **TSPMAT** in the Route Property list and click **Edit**. Several new parameters related to the inner conductors are now available for modification.
6. Change the box labeled **Wire resistance** to **1.25e-1**.

NOTE: Changing any of the Shield properties for this material will also change those of our previously created route **SHIELD**.

7. Click **Apply** to save the modified property.
8. Click **Apply** again to finish creating the route. A route **TSP** will be created and added to the tree along with two inner-conductors **TSPC1** and **TSPC2**.

Routes 3,4,5 – Conductors

We will now nest three more conductors inside of **SHIELD**. These conductors will travel the lower path outlined below. These three conductors will be adjacent to the route **TSP** in Lines **L2** and **L1** and will diverge at point **P3** into lines **L3** and **L6**.



1. Right-click on the **SHIELD** in the Harness editor and left-click **Add Child – Conductor**.
2. Type **CONDUCTOR1** in the box labeled **Route Name**.
3. Select segments **L1**, **L2**, **L3**, and **L6**.
4. Select **Termination Points P5** and **P2** for this route.
5. Click the **New** button in the **Select Route Property** section.
6. Change the **AWG** to **22**.
7. Type **CONDMAT** in the **Property Name field**
8. Click **Apply** to create the new property.
9. Click **Apply** again to finish creating the route.
10. Right-click on the newly created route **CONDUCTOR1** and left-click **Copy** to duplicate the route.
11. Copy **CONDUCTOR1** again to create the third and final conductor.

NOTE: MHARENES requires a common dielectric constant for adjacent shields and/or jackets. This means that the shield and/or jacket dielectric constants for all adjacent routes in a segment will be averaged together into a common value. If a bare conductor (0 jacket thickness) is adjacent to the shield/jacket of another route, a shield/jacket dielectric constant of 1.0 will be used for that group of adjacent routes.

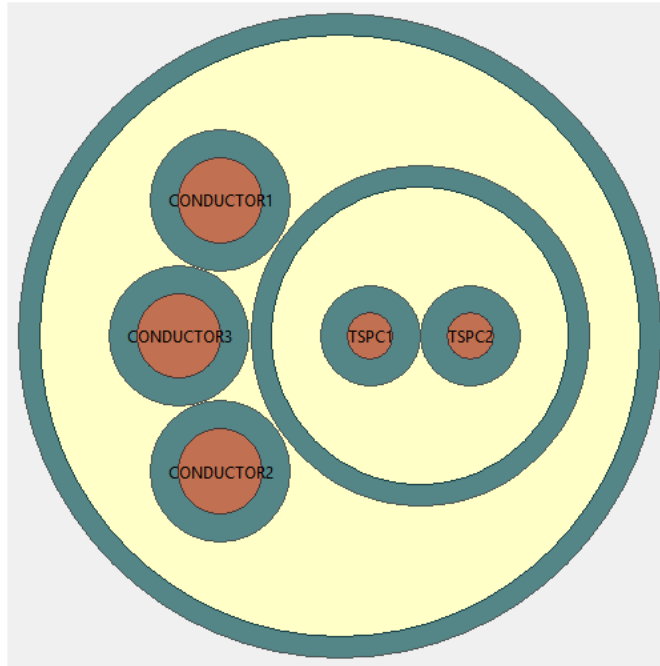
Verifying Harness Cross-Sections

Now that the harness routes have been defined, we can verify that the cross-sections of individual segments are as expected.

1. Open the **Segments** tab in the **Harness Editor**.

Each MHARNES segment in the tab corresponds to a CADfix line in our harness. Left clicking on any individual segment should highlight the corresponding line. We will first verify the segment corresponding to line **L6**. This line should contain the route **SHIELD**, with **CONDUCTOR1**, **CONDUCTOR2**, and **CONDUCTOR3** nested inside of it.

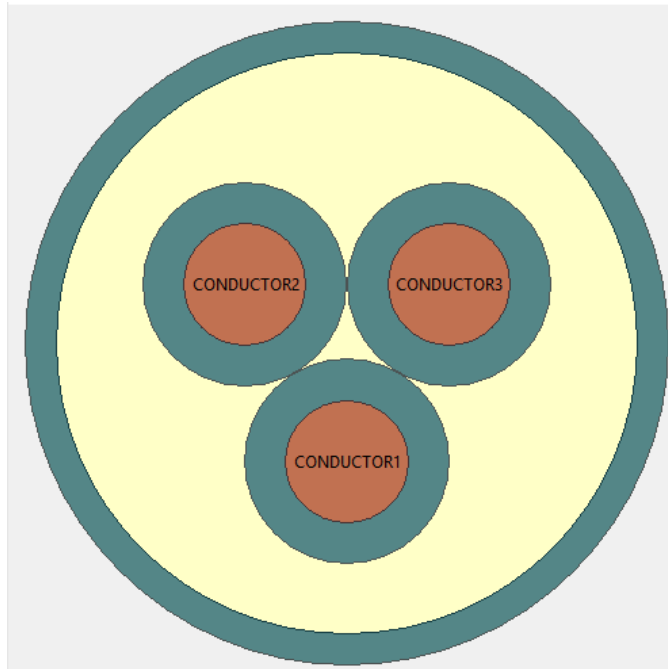
2. Find the segment corresponding to line **L6**.
3. Right click on the segment and click **Inspect Topology**.
4. When the pop-up window is fully loaded, it should look like the image below.



NOTE: There are several references to “adjacent routes” in this tutorial. The term adjacent routes refers to a group of routes whom can directly touch each other within a harness and are not separated by a level of nesting. **TSP CONDUCTOR1**, **CONDUCTOR2**, and **CONDUCTOR3** are considered adjacent in segment **L6**. **TSPC1** and **TSPC2** are also considered adjacent. Routes may branch into different paths causing them to be adjacent in some segments and not in others.

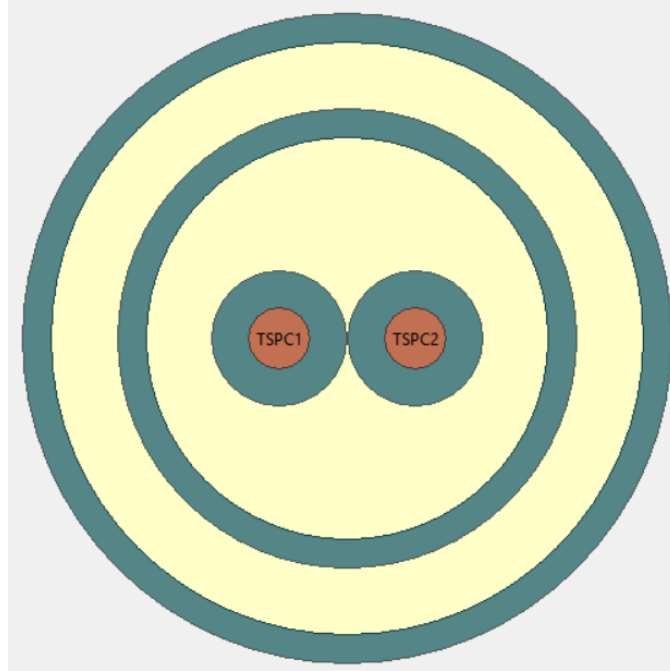
We will next verify the segment corresponding to line **L5**. This line should contain the route **SHIELD** with only the nested route **TSP** inside of it.

5. Close the **Segment Visualizer** window that popped up if you have not done so already.
6. Find the segment corresponding to line **L5**.
7. Right click on the segment and click **Inspect Topology**
8. When the pop-up window is fully loaded, it should look like the image below.



Finally, we will verify the segment corresponding to **L1**. This segment should contain all the routes we have created.

9. Close the **Segment Visualizer** window that popped up if you have not done so already.
10. Find the segment corresponding to line **L1**.
11. Right click on the segment and click **Inspect Topology**.
12. When the pop-up window is fully loaded, it should look like the image below.



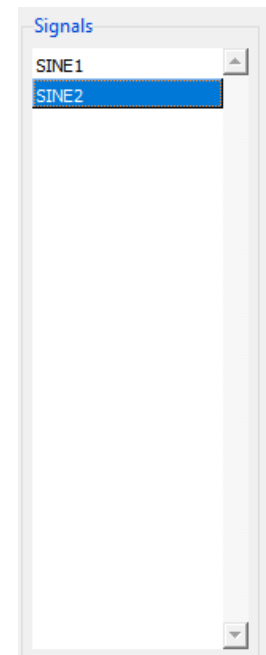
Creating the Excitation Sources

MHARNNESS, by default, supports several different types of excitation types and source waveforms. More complicated signals can be specified from a data file. We will be constructing a send and receive signal composed of two superimposed Sine waves using the **Signal Generator tool**. The sources available in the **Harness Editor** differ from other EMA3D probes in that they can be applied directly to individual conductors/shields within a cable harness.

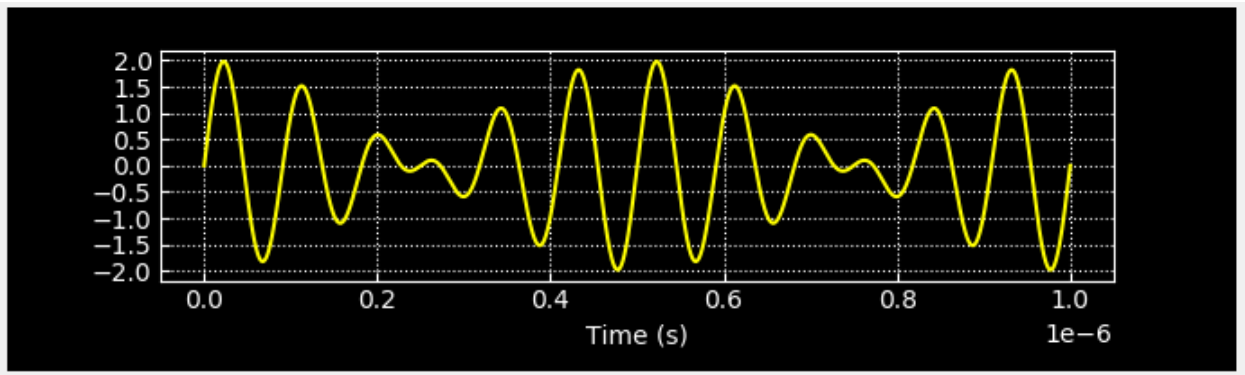
Constructing the Input Signals

Send Signal (V+)

1. Open the **Tools** tab in the **Harness Editor**.
2. Under **Miscellaneous**, click on **Signal Generator**, a window will pop-up.
3. Click on the **Sine** tab.
4. Type **SINE1** in the box labeled **Signal Name**.
5. Change the **Step Time** to $3.0e-11$.
6. Change the **Stop Time** to $1.0e-6$.
7. Change the **Frequency** to $1.0e7$.
8. Click on **Plot** under the **Create Signal** section to visualize the signal before adding it.
9. Click **Add** under the **Create Signal** section. The signal **SINE1** should show up in the **Signals** list on the left-hand side.
10. Type **SINE2** in the box labeled Signal Name.
11. Change the **Stop Time** to $1.0e-6$.

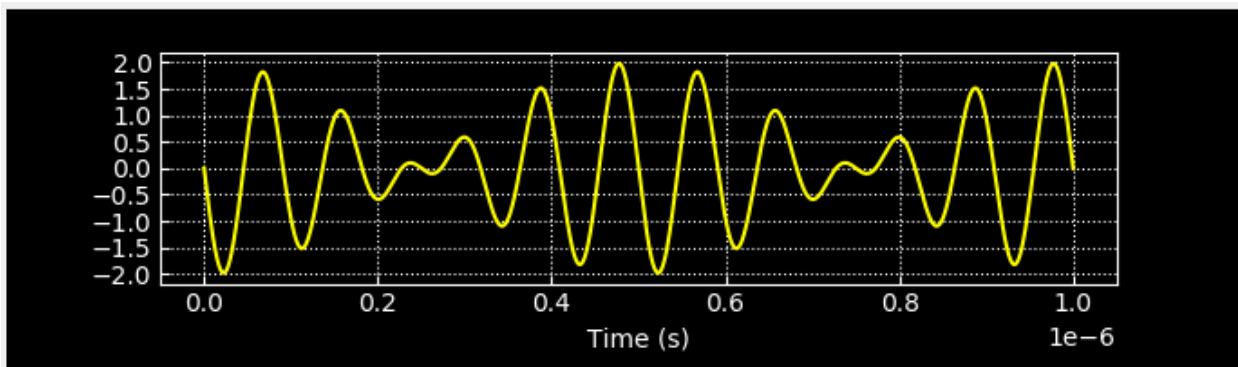


12. Change the **Frequency** to 1.2×10^7 .
13. Click **Add** under the **Create Signal** section. The signal **SINE2** should be added to the Signals list on the left-hand side.
14. Click **Plot Resultant Signal** at the bottom of the window. The two waves should be superimposed and look like the image below.
15. Click **Write To File**.
16. Leave the file name as **send**.
17. Click **Save** to write the signal to the file **send.dat**.
18. Do not close the **Signal Generator Tool** yet, we will re-use our signal in order to create a negative version of our signal to act as the receive side of our differential signal.



Receive Signal (V-)

19. Right Click on **SINE1** in the **Signals** list and click **Edit**.
20. Change the **Amplitude** to -1.00 .
21. Click **Add**.
22. Right Click on **SINE2** in the Signals list and click Edit.
23. Change the **Amplitude** to -1.00 .
24. Click **Add**.
25. Click **Plot Resultant Signal** at the bottom of the window. The two waves should be a negative version of the previous signal and look like the image below.
26. Click **Write To File**.
27. Change the **file name** to **receive**.
28. Click **Save** to write the signal to the file **receive.dat**

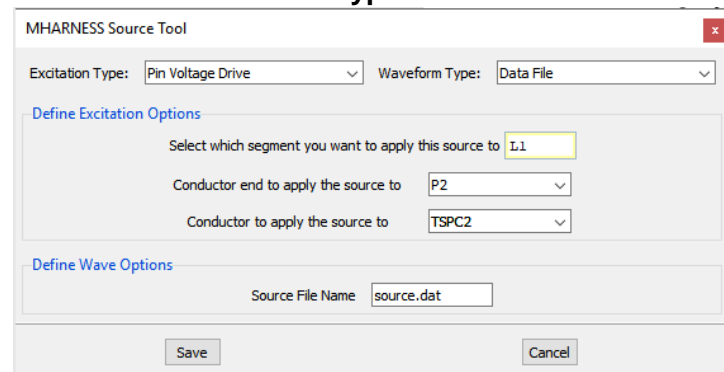


Applying the Source

We will apply the signal we constructed as a **Pin Voltage** source. Pin Voltages are exactly as they sound, a voltage source applied to a shield or conductor at a termination point. We will create two sources, a send and receive signal, which will be applied to the two inner conductors of **TSP**.

NOTE: Pin Voltages can only be applied at route ends who also end in a termination.

1. Open the **Sources** tab in the **Harness Editor**.
2. Right-click **Pin-Voltage**.
3. Left-click **New** in the pop-up menu.
4. Leave **Excitation Type** as **Pin Voltage Drive** and the **Waveform Type** as **Data File**.
5. Select **L1** as the segment to apply the source to.
6. Select **P2** as the conductor end to apply the source to.
7. Select **TSPC1** as the conductor to apply the source to.
8. Change the **Source File Name** to **send.dat**.
9. Click **Save**.
10. Repeat the following steps for conductor **TSPC2** at point **P2** on segment **L1** using the **Source File Name** **receive.dat** to create the receive signals.
11. Expanding the **Pin Voltage** section in the **Excitation** list should show two sources **MHSRC1** and **MHSRC2**



NOTE: MHARNES is imbedded with several different types of common waveforms by default such as **Gaussian Impulses** and **Damped Sine Waves** which do not require a source file to be generated first. These waveforms are available under the **Waveform Type** dropdown menu.

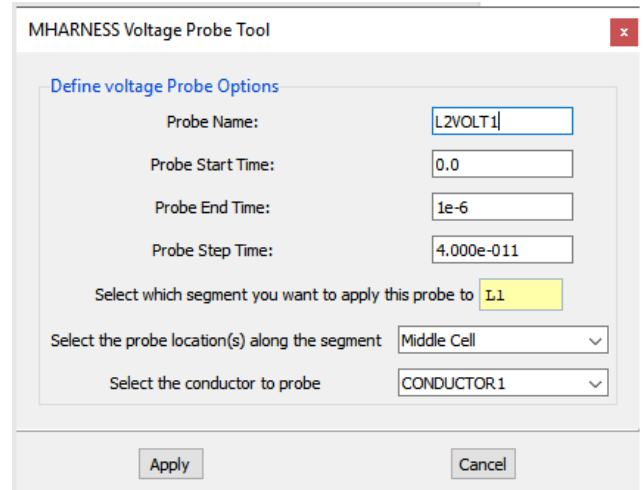
Creating the Probes

We will next define our probes. We will create a single voltage probe in this tutorial. The probes available in the **Harness Editor** differ from other EMA3D probes in that they can output the current or voltage on any individual conductor/shield in a segment. Structure probes are also available; structure probes allow users to verify segment's topology by creating a POSTSCRIPT file of a segment's cross section during simulation.

Voltage Probe

We are mainly interested in examining the crosstalk between the conductors in **TSP** and **CONDUCTOR1**. For the sake of simplicity, we will only create a single voltage probe on **CONDUCTOR1** in line **L2**; feel free, however, to create additional probes in any other area of interest using the below procedure.

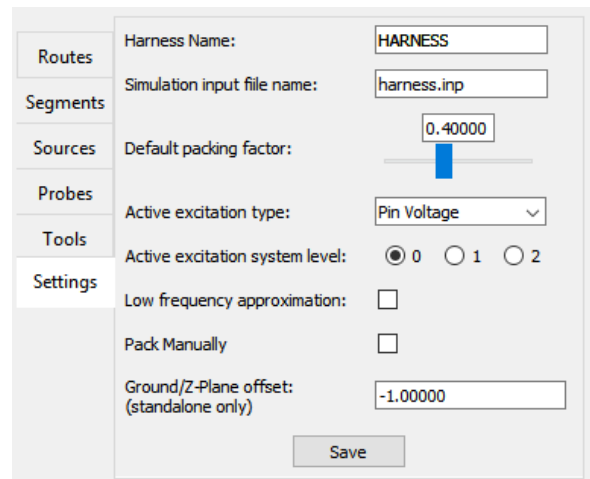
1. Open the **Probes** tab in the **Harness Editor**.
2. Right click **Cable Voltage** in the **Probes List**.
3. Left-click **New**.
4. Change the **Probe Name** to **L2VOLT1**.
5. Leave the **Probe End Time** as **1.0e-6**.
6. Change the **Probe Step Time** to **3.0e-11**.
7. Select line **L2** as the segment to probe.
8. Change the probe location to **Middle Cell**.
9. Select **CONDUCTOR1** as the conductor to probe.



Finalizing the Harness Settings

Before generating the MHARNES input file we need to first define what type of excitation source we wish to use and what system level will be driven. MHARNES, currently, only allows one type of excitation to be used in driving one level of route nesting to be driven at any given time.

1. Open the **Settings** tab in the **Harness Editor**.
2. Leave the **Active excitation type** as **Pin Voltage**.
3. Change **Active excitation system level** to **0**.
4. Click **Save**.



A NOTE ON SYSTEM LEVELS:

System levels are defined with respect to the most deeply nested route. The most deeply nested shield/conductor is always level 0, followed by level 1 and level 2. Segment topologies are currently limited to a maximum system level of 2. The system levels for our topology tree are as followed.

- SHIELD
 - TSP
 - TSP inner-conductor 1
 - TSP inner-conductor 2
 - CONDUCTOR 1
 - CONDUCTOR 2
 - CONDUCTOR 3

- Level 2
- Level 1
- Level 0
- Level 0
- Level 1
- Level 1
- Level 1

Creating the MHARNESS Input File

We now need to generate the MHARNESS input file. This input file is required for both the MHARNESS standalone solver as well as EMA3D-MHARNESS integrated simulations.


NOTE: The MHARNESS input file must be written prior to EMA3D input file creation for EMA3D-MHARNESS integrated simulations.

1. Click the **Write** button at the bottom of the **Harness Editor** to generate the MHARNESS input file. This may take some time for more complex harnesses.

* **TIP:** This is the final step required for MHARNESS standalone simulations. Some users may find it convenient to use the **Run Standalone** button to help verify simulations are properly set up before moving to EMA3D-MHARNESS integrated simulations as standalone simulations often run in a fraction of the time.

Creating the EMA3D Input File

We now need to generate the EMA3D input file which is required for EMA3D-MHARNESS integrated simulations.

1. In the **EMA3D**  tab navigate to the **General** tools list.
2. Click **Review Tool** to expand the toolbox.
3. Click **Review**.
4. Click **Create EMA3D Input File**.
5. Click **OK** to generate the **.emin** file.

Running the Integrated Simulation

We are finally prepared to run the EMA3D-MHARNESS integrated simulation.

1. Navigate to the **General** tools list under the EMA3D  tab.

2. Click **Run EMA3D** to launch the **EMA3D Run Tool**. This may take several seconds to launch.
3. Select the **Run New EMA3D Simulation** radio button.
4. In the Entry titled **EMA3D Input File** click the **Browse** button.
5. Select the **.emin** file we created earlier.
6. Click **Run EMA3D** to start the simulation. The simulation should be able to complete in a few minutes on a modern laptop.

Tabulated results will be written to the file **L2VOLT1.dat**, where the first column of each line is the time step and the second column is the voltage information at our probed point. More information is available in the MHARNES User Manual which can be accessed through the Windows start menu. The results from the simulation have been plotted below using Microsoft Excel

