



# **Car Radiated Immunity Demo**

**EMA3D Demo for Creating a Basic Radiated Immunity  
Model of a Car**

***Prepared by Grant Riley and Tim McDonald***

***January 2021***

# CONTENTS

Contents .....	i
Problem Overview .....	1
Downloading and Opening the File .....	1
Exploring the SpaceClaim Interface.....	1
Defining the Simulation Domain.....	2
Defining the Plane Wave Source .....	3
Defining and Assigning Material Properties.....	4
Importing the cable harness .....	6
Creating the Current and Voltage Probes.....	8
Creating the Bulk Current Probe (Harness Bundle Current) .....	10
Creating the Animation Probe .....	11
Meshing the Model .....	12
Running the Integrated Simulation .....	13
Analyzing Results.....	13

## Problem Overview

---

This tutorial will demonstrate the EMA3D workflow for a radiated immunity (RI) simulation in SpaceClaim. This workflow includes setting up the problem space, assigning properties, defining probes, and basic post-processing.

A pre-simplified car model will be used to simulate this problem. The car will be exposed to a plane wave of a gaussian pulse to simulate multiple frequencies at once. This will allow the car body shielding effectiveness and the coupling onto cables in the wiring harness to be measured.


## Downloading and Opening the File

---

1. In a web browser, navigate to [www.ema3d.com/training/CarRI\\_Demo/](http://www.ema3d.com/training/CarRI_Demo/) and click on the red hyperlink that reads **training document here** to download a zip file of the Car demo.
2. Using the File Explorer navigate to and open the downloaded zip file called CarRI\_Demo.zip. Unzip the file and open CarRI\_Demo\_Initial.scdoc to launch SpaceClaim.


## Exploring the SpaceClaim Interface

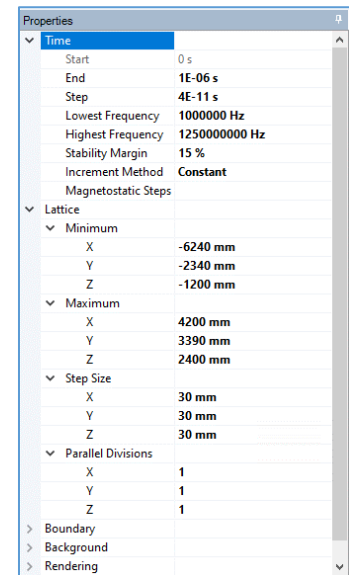
---

1. The Car model we have just imported is a full CAD assembly. To better visualize components of the model, individual components of the model may be hidden. Navigate to the structure tree by clicking on the **Structure** tab on the left-hand side of the screen, midway down the screen. Unchecking the box next to the component name hides it from view. For example, uncheck Carshell to see inside the car.
2. Controls for navigating the model can be found in the **Orient** section under **Design** in the **Ribbon**. These include:
  - a. **Spin**: rotate model about the cursor or center of grid (Shortcut: center mouse button, equivalent to clicking and holding scroll wheel on a standard mouse)
  - b. **Pan**: gives the ability to position the model within the model window (Shortcut: ctrl + center mouse button)
  - c. **Zoom**: zoom in or out on the model (Shortcut: scroll wheel or ctrl + +/-)
  - d. **Align coordinate system**: at the bottom left of the model window, click a specific axis on the axis widget  to align the view along an axis. Clicking and dragging an axis rotates the model.
3. The **Simulation tree** can be found along the same ribbon as the **Structure tree**. The Simulation tree can be used for a multitude of purposes that will be outlined throughout this guide. This tree is useful in tracking your progress as you prepare your model for simulation. Not all branches in the simulation tree are required to run a simulation, but their uses should be considered.


## Defining the Simulation Domain

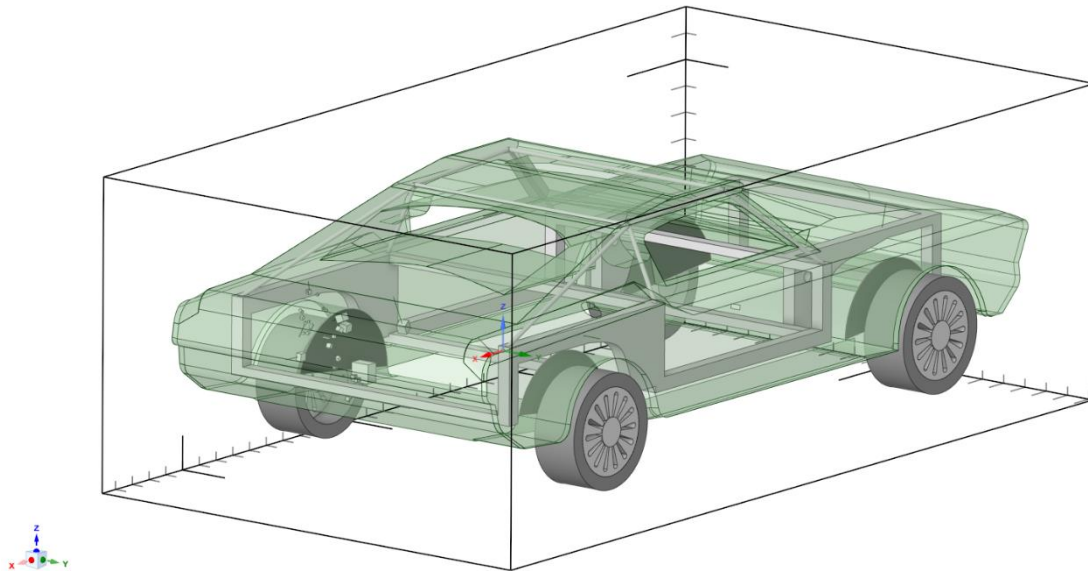
The geometry in this model has already been cleaned. So, the first step will be to define the computational domain.

1. Click **Domain**  under **EMA3D** in the **Ribbon**.
2. Check the box next to **Domain** in the **Simulation tree** to make the current domain visible.
3. Six directional handles and a small empty box will appear within the body of the car model.
4. Drag the handle pointing in the +x direction to change the **Maximum X** value to **4200 mm**. To see the value, click the arrow to the left of **Maximum** under **Lattice** in the **Properties Panel**.
5. The lattice parameters can also be manually adjusted in the **Properties Panel**.
6. Under **Time** in the **Properties Panel** adjust:
  - a. **End:** 1E-6s
  - b. **Step:** 4E-11s
  - c. **Lowest Frequency:** 1 MHz (if no units are specified, Hz is default value. Units are case sensitive; if capitalization is incorrect the default unit will be used.)
  - d. **Highest Frequency:** 1.25 GHz
  - e. **Stability Margin:** 15%
7. Under **Step Size** set each dimension to 30 mm.
8. Under **Lattice** in the **Properties Panel** adjust the **Minimum** values to be:
  - a. **X:** -6240 mm (if no units are specified, mm is default value)
  - b. **Y:** -2340 mm
  - c. **Z:** -1200 mm
9. Under **Lattice** in the **Properties Panel** adjust the **Maximum** values to be:
  - a. **X:** 4200 mm
  - b. **Y:** 3420 mm
  - c. **Z:** 2400 mm




NOTE: The spatial step size, temporal step size, start time, and end time, will be automatically calculated based on the specified minimum and maximum frequencies. In the future, these items will be manually adjustable by toggling “advanced options.”

10. Click **Ok**  to complete the domain setup.
11. Upon completion of the domain parameters, the car is fully encapsulated by the lattice.

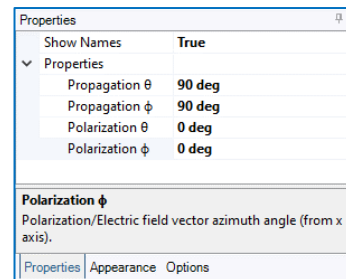



## Defining the Plane Wave Source

Next, a plane wave source will need to be defined.

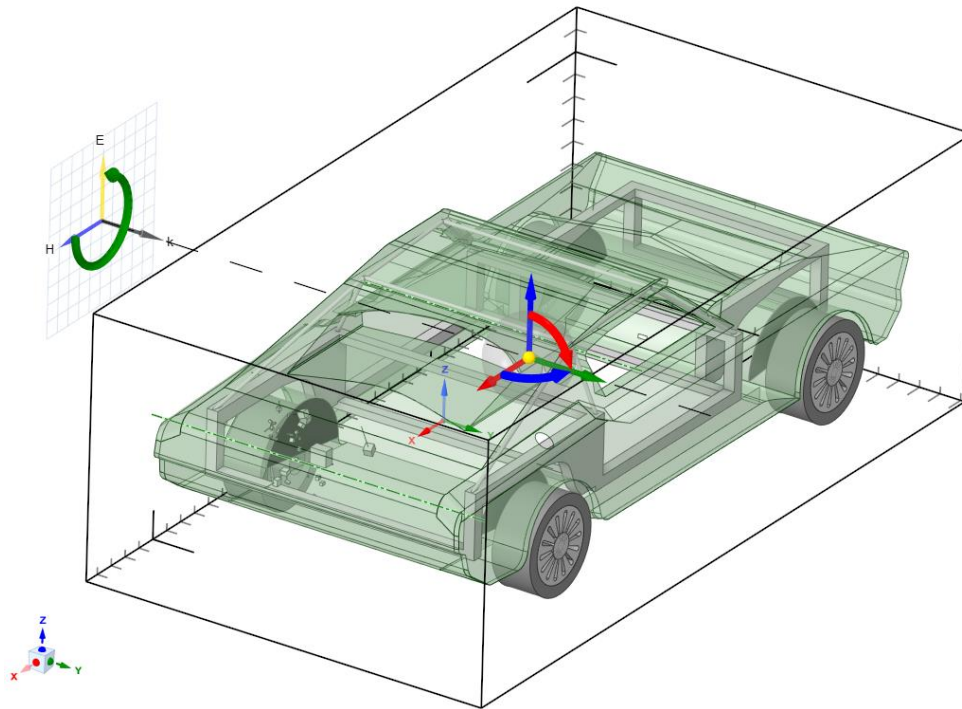
1. Click **Plane Wave**  under **EMA3D** in the **Ribbon**.
2. In the **Properties Panel** enter the following values (a definition of each variable is available at the bottom of the **Properties Panel** when the variable is selected):
  - i. **Propagation  $\Theta = 90$**
  - ii. **Propagation  $\Phi = 90$**
  - iii. **Polarization  $\Theta = 0$**
  - iv. **Polarization  $\Phi = 0$**

At this time, it is possible that some variables may change values after entering another variable. If this occurs, simply reenter the values.






3. In the **Properties Panel** change **Show Names** to **True** to see the vector labels. The plane wave should be oriented with **Electric Field Polarization Vector** (E, straight yellow arrow) in the **+Z** direction, **Propagation Vector** (k, straight black arrow) in the **+Y** direction and **Magnetic Field Vector** (H, straight blue arrow) in the **+X** direction.
4. Click **Ok**  to complete the plane wave setup.
5. A **Plane Wave** object should now be added to **Simulation Tree** under the **Sources Node**.

NOTE: You can also manually click and drag the angle handles to adjust the planewave's orientation.



## Defining and Assigning Material Properties

The geometry in the model will need physical properties assigned in order to simulate the problem. The entire model contains curves, surfaces, and solids.

1. First we need to make the material assignments visible. Click the check mark next to **Definitions** in the **Simulation tree**.
2. Click **Materials**  under **EMA3D** in the **Ribbon**.
3. Click **Isotropic**  in the dropdown list.
4. In the **Properties Panel** select **Material**. Select the arrow that appears directly to the right of materials for a dropdown menu. A new window will appear.
5. In the popup window, under the **Local** tab, select **New** .
6. Double click to select the first definition in the list “**Isotropic**.” To view its properties, click the arrow to the left of **Properties** in the **Properties Panel**.
7. To the right of the **Name** parameter, double click “**Isotropic**” to rename the material. Rename the material **Steel** and press the enter button to rename the definition.

8. Change the color to light blue, by selecting **Color** and then selecting the dropdown menu arrow. The color selection is arbitrary, but the colors used in this demo are provided for ease of following along.
9. Change the **Electric Conductivity** ( $\sigma_e$ ) to **1.45e6**.




10. Click on the **Select surface tool** in the model window to restrict our definition assignments to surfaces.


11. In the **Structure Tree** select **carshell**, **frame**, and **wheelshields** from within the car assembly then select **enclosures** from within the electronics assembly to assign the **Steel** material properties to all **Steel** structures on the vehicle.

Properties	
Material	Steel
▼ Properties	
Name	Steel
Color	120, 0, 114, 178
$\epsilon$	8.854E-12
$\mu$	1.257E-06
$\sigma_e$	1450000
$\sigma_m$	0
> Metadata	
<b><math>\sigma_e</math></b>	
Material electric conductivity	




NOTE: You can also manually click on surfaces of the model directly to assign material properties to specific surfaces.

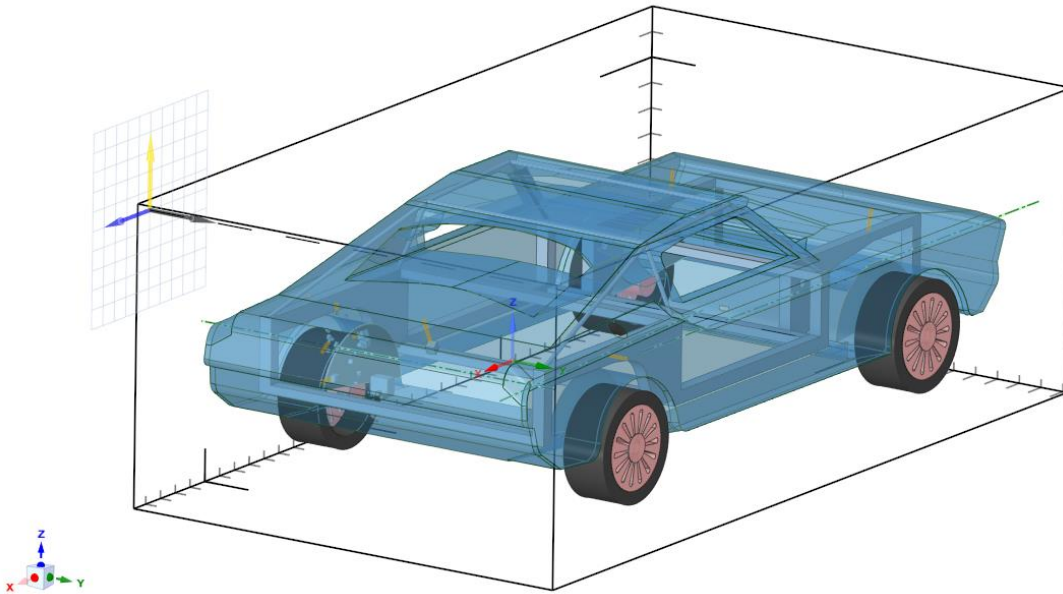
12. Several items should now be recolored in the model window. Click the checkmark on the **Isotropic Node** in the **Simulation Tree** to hide the colors and revert to their original.
13. Repeat steps 2-10 to create the following material definitions. Click **Ok**  when finished assigning all materials to exit the materials definition tool.

Material Name (Components)	Electric Conductivity $\sigma_e$ (S/m)	Permittivity $\epsilon$ (F/m)	Color
Rubber (tires)	0.2	2.66E-11	Gray
Aluminum (rims)	3.5E+07	8.854E-12	Light Red

NOTE: It is also possible to add many materials without assigning them to surfaces immediately. To assign materials later, follow steps 1-3 above. Then click **New**  multiple times. Now you can double click each material and adjust the properties for for each and assign to surfaces later following steps 10 and 11 above.

Next we will assign the properties of a PEC (Perfect Electric Conductor) to ground the electronic components to the body and frame of the vehicle.

1. Click **Materials**  under **EMA3D** in the **Ribbon**.
2. Click **PEC**  in the dropdown list.
3. Click on the **Select line tool**  in the model window to restrict our definition assignments to curves.
4. In the **Structure Tree** expand **electronics** and click on **groundwires** to assign the **PEC** properties to all ground wire segments on this portion of the vehicle harness.




## Importing the cable harness

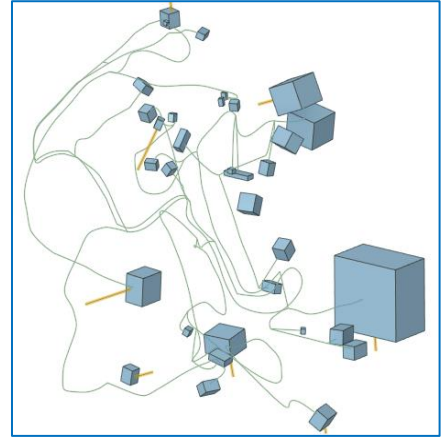
RI can produce undesired currents and voltages on the wiring inside of a vehicle. Using **MHARNES**, it is possible to import a complete pre-defined wiring harness model and measure induced voltages and currents on cables of interest.



1. Under the **MHARNES** tab in the **Ribbon** click the **Cabling** group and select **Import** from the drop-down menu. The .Kbl Import Wizard will open.
2. Click the ellipses button to the right of the File field.



3. Select the file “/CarRI\_Demo/Harness\_RearLeft.kbl” and click **Open**.
4. Select the **NURBS** (Non-Uniform Rational Basis Spline) option for the Line Type. Leave Parse Wire Types and Share Properties checked, Units in mm, and Unit Scale at .001. Then click **Next**.
5. Here the user has the option to select which wires to import. In this case we will leave all the wires selected. Click **Next**.
6. Now the user has the option to select which segments to import. Note that the wires routed through each segment can be viewed here. The geometry for each segment can be selected from the cable file or assigned to a curve in the model. In this case, we will right click on any segment and select **Set All From File**. Click **Finish**.
7. After the Harness is imported close the Import Wizard using the X.
8. In the **Structure tree** uncheck the box next to the **car** assembly to hide the car and better view the complete wiring harness and electronic assembly. The automatically generated segment labels can be suppressed by unchecking the box left of **Harness** in the **Simulation tree**. The complete harness should now look as shown in the figure.
9. Right click on any segment and click **Visualize** to explore the cables within that segment.
10. After exploring the harness, re-check the box next to **Harness** in the **Simulation tree** to make the labels visible again.
11. Next we will set the transfer inductance of two cable shields within the harness.
12. Click the drop-down arrow to the left of **Harness** in the **Simulation tree**. A full list of conductors and shields are shown. Search for the shield called “CL\_014” and right click on it and select Edit.
13. In the properties dialog, click the arrow next to Properties and then click inside the box to the right of Transfer Inductance. Click the down arrow that appears. This popup window describes the various shield properties that affect the transfer impedance.
14. Change the Specification Type from “**Calculated**” to “**Fixed Value**”. Leave the default values for each section. This will hold the transfer inductance at these values throughout the simulatoin.
15. Close the popup window then click the green check mark  within the design window to save the changes.
16. Repeat steps 12-15 for the shield called “CL\_016\_2”.

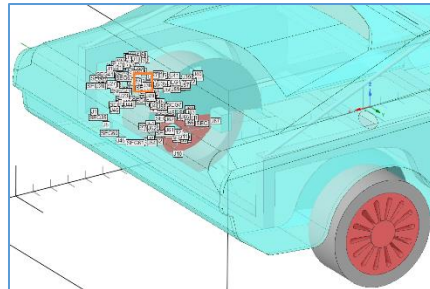


Properties	
Type	CL_014
> Metadata	
> Properties	
Radius	0.001 m
> Resistance	
Twisted	False
Pitch	0 twists/m
> Filler	
> Jacket	
> Transfer Inductance	
Specification Type	Fixed Value
> Connector (Beginnin	
Shield	
> Connector (End)	


## Creating the Current and Voltage Probes

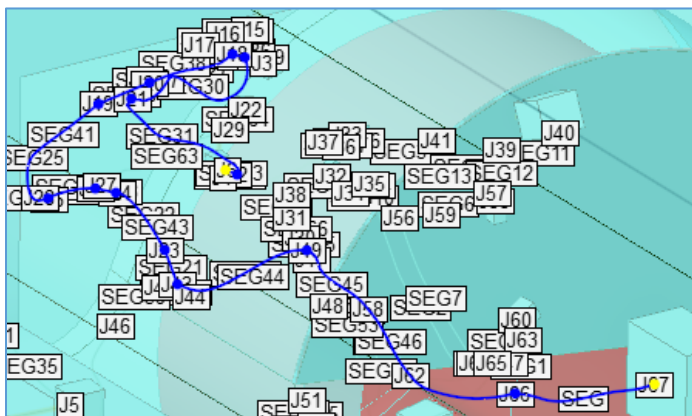
We will next define a current and voltage probe on each cable. We are interested in probing the current and voltage on specific conductors.

1. Zoom in to the portion of the harness near the rear of the car and find SEG33 (located within the orange box in picture at right). Right click on SEG33 and click **Visualize** to view the cables inside the segment. We want to measure the voltage on the wire with the label W\_200\_1. Press the **escape key** twice to exit the visualization.
2. To make a meaningful measurement, we need to add some realistic terminations to the conductor.




Click the arrow next to **Harness** in the **Simulation tree** to show all the components of the harness. Right click on conductor W\_200\_1 and click edit. In the design window, click the

Select Point button . Now click the red boxes at both ends of the highlighted segment – they will turn to yellow dots once selected.

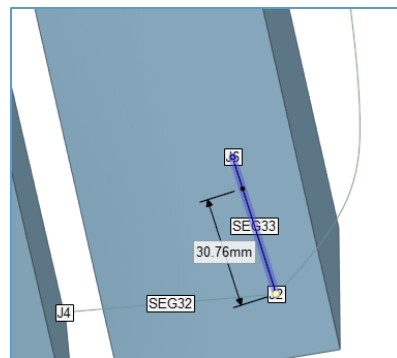


In the Properties dialogue, click the arrow to the left of Terminations and expand the subsections. In the Resistance property, enter 50 Ohms for each termination. Update the Method property to **Inline** for both terminations. This will connect the boundary conditions

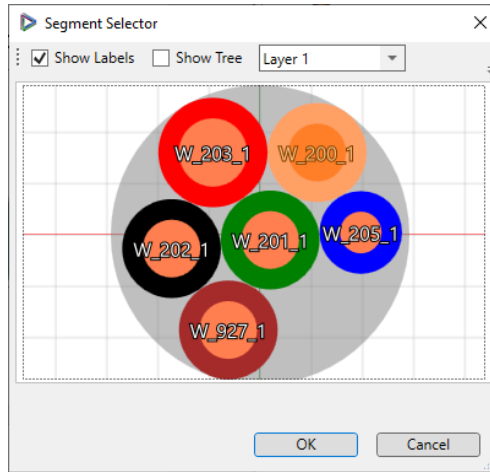
of MHARNESS and EMA3D. Click Ok  to save.

3. Now we can add the voltage probe. Under the **MHARNESS** tab in the **Ribbon** click **Cable**  and select **Voltage**.

4. Select segment SEG33. An arrowhead will follow your cursor along the cable. Click along the length of the cable close to the box to set the probe's location. (The voltage probe can be placed anywhere along the length of the selected cable.)





5. Once the probe has been placed, an image of the cable's cross section will appear in a new window. Select **Layer 1** from the drop-down menu above the image of the cross section.
6. Next, click **Show Labels** then click on the cross section of wire **W\_200\_1** to add the probe. It will be illuminated. Click **Ok**.




We will now add a Current probe to measure the current passing through one of the shields.

7. First, we need to add terminations to the conductors. Under **Harness** in the **Simulation tree**, expand the shield **CL\_016\_2** and right click on one of the conductors and click edit.

In the design window, click the Select Point button . Now click the red boxes at the junction at both ends of the highlighted segment. In the Properties dialogue, click the arrow to the left of Terminations and expand the sub-sections. In the Resistance property, enter 50 Ohms. In this case, we will leave the Termination Method set to Boundary because this conductor is inside a shield and we want the boundary condition to remain only in MHARNES.

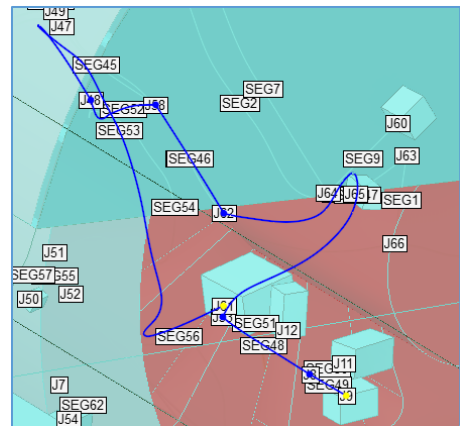
8. Click the green check mark  to save the settings and repeat this step for the other conductor.

9. Next, right click on the shield **CL\_016\_2** under **Harness** in the **Simulation tree** and click edit. Add terminations as above, by clicking the Select


Point button  and clicking on each end of the segment, denoted by a red-boxed U. We want the shield to interface with EMA3D so we will change the Termination Method to **Inline** for both terminations. We will leave the default resistance values unchanged. Click the green

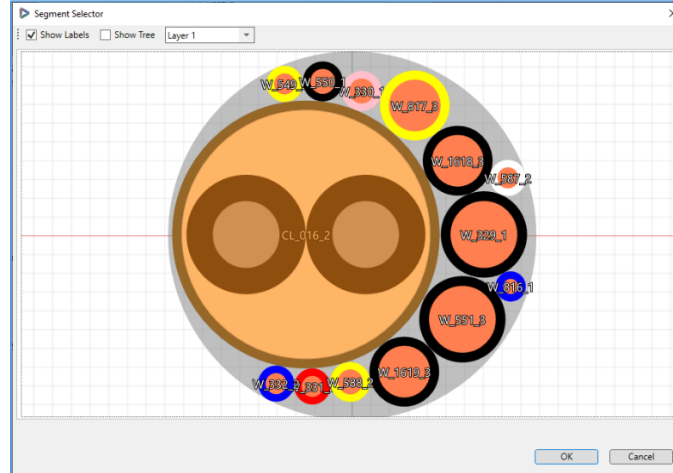
check mark  to save the settings.

10. Now we will add a current probe. Zoom in near the bottom of the harness and find **SEG49** attached to the box at junction **J9**. Right click on **SEG49** and click **Visualize** to view the




cables inside the segment. We want to measure the current through the shield around the two conductors. Press **Escape** twice to exit the visualization.

11. Under the **MHARNESS** tab in the **Ribbon** click **Cable**  above Probes and select **Current**.
12. Click the SEG49 cable and place the probe near the middle of the cable.
13. The SEG49 cross section appears in a new window. In the **Layer** drop-down menu select **Layer 1**, check **Show Labels** and then select the shield **CL\_016\_2**, and click **Ok**. A blue dot will appear on the Spline cable to indicate the probe location.





## Creating the Bulk Current Probe (Harness Bundle Current)

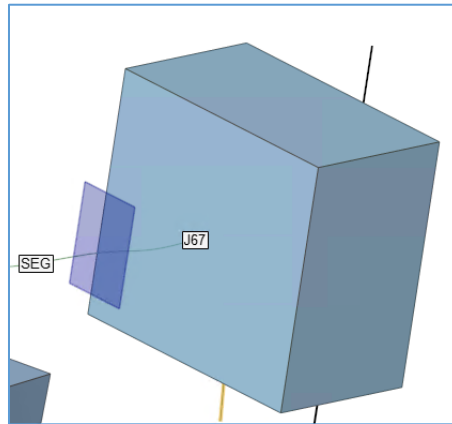
We will now creating a bulk current probe around the spline SEG which is located within the harness toward the front end of the car.

1. Click **Current**  within the **Probes Panel** under **EMA3D** in the **Ribbon**. A small box with three sets of arrows will appear.
2. In the **Properties Panel** set the following:
  - a. **Normal direction:** X
  - b. **Minimum:**
    - X: 1770 mm
    - Y: -750 mm
    - Z: -90 mm
  - c. **Maximum:**
    - X: 1770 mm
    - Y: -690 mm
    - Z: 0 mm

Properties	
Normal Direction	X
Skip Factor	1
Minimum	
X	1770 mm
Y	-750 mm
Z	-90 mm
Maximum	
X	1770 mm
Y	-690 mm
Z	0 mm
Time	
Start	
Value	0 s
Same as sim	True
Step	
Value	4E-11 s
Same as sim	True
End	
Value	1E-06 s
Same as sim	True
Skip	1
<b>Maximum</b> Maximum values for the probe bounds.	


3. The probe can also be created by dragging the red, blue, and green directional handles that appear after clicking the Current Probe button .

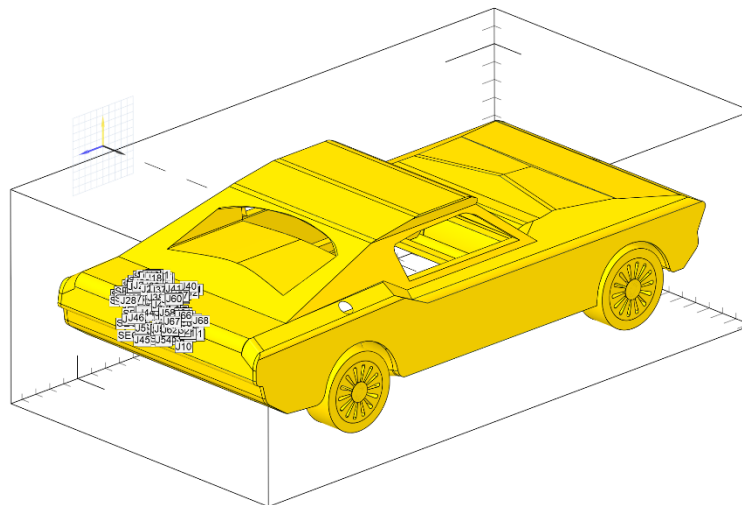
4. Click OK  to set the probe. The probe will appear as a blue plane in the model window encompassing SEG.



## Creating the Animation Probe

Next, we will create an animation probe that will capture the electric current on the surface of the car.

1. In the **Structure Tree**, click to make the car visible.
2. Click **Animation**  within the **Probes Panel** under **EMA3D** in the **Ribbon**.
3. In the **Structure Tree** click on the entire **Car** and **Electronics** assemblies. The selected animation surfaces will be highlighted.



4. To capture the main part of the EM wave passing over the vehicle, we will set the duration of the video to 1E-7s. In the properties dialog, under 'Time' and 'End', change the 'Same as simulation' to **False** and set the Value to 1E-7s.

5. To prevent the animation file from growing too large, we can reduce the number of steps to 50. In the properties dialog, under 'Time' and 'Step', change 'Same as simulation' to **False** and set the Value to 2e-9s.




6. Click **Ok** to create the probe.
7. The probe should now be visible in the **Simulation Tree** under the **Probes Node**.

Properties	
Probe Type	Electric Current
Time	
Start	
Value	0 s
Same as simulation	True
Step	
Value	2E-09 s
Same as simulation	False
End	
Value	1E-07 s
Same as simulation	False
Skip	1



## Meshing the Model

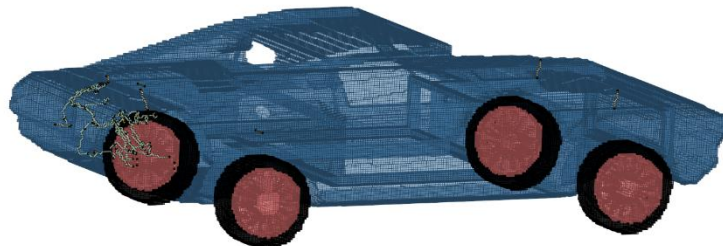
Next, we need to mesh the model geometry.

1. In the **Structure Tree**, make sure the checkboxes next to the components are all checked to ensure they will be meshed.
2. Click **Mesh**  under **EMA3D** in the **Ribbon**.

**NOTE:** The geometry will only be meshed if it lies within the lattice, has material properties assigned to it, and is visible (checked in the structure tree).

Some portions of this model may be too small to mesh and will throw a warning. In this case, the unmeshed segment do not significantly affect the results.

3. The amount of time to mesh the geometry will vary depending on the computer being used and the model geometry. A progress bar at the bottom of the SpaceClaim window will track the meshing progress.
4. To more easily view the mesh, right click anywhere in the model window and select **Inverse Visibility**. Alternatively, in the **Structure Tree**, deselect model components as desired to improve mesh visibility. To view the underlying structures without mesh (but without unmeshing them), make sure they are visible then hit Escape when in the **Mesh**  menu. To bring the mesh back into view, select **Inspect** from the dropdown menu beneath **Mesh**  under **EMA3D** in the **Ribbon**.

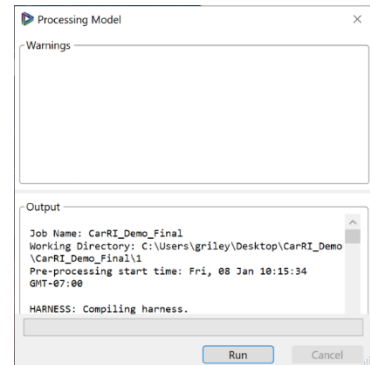


## Running the Integrated Simulation

We are now prepared to run the simulation.



Click **Start**  in the **EMA3D** section of the **Ribbon**.

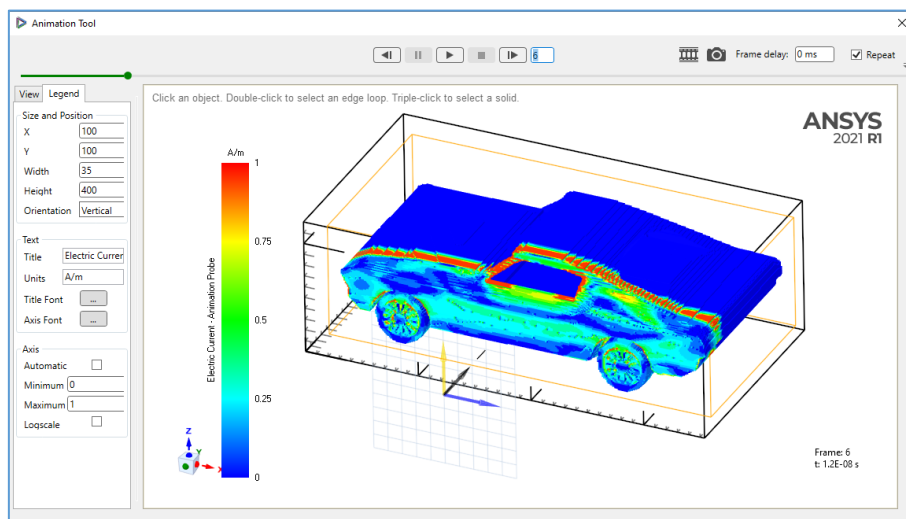
1. EMA3D will immediately begin pre-processing the simulation. Any errors and warnings will be logged in the pop-up window.
2. The EMA3d .emin file, and other relevant files, will be output in the current working directory at this time.
3. In the pop-up menu, click Run.
4. The **Analysis Panel** (located next to the **Structure** and **Simulation Panels**), the progress and status of the simulation are shown. Additionally, any runtime errors will appear under Output.



## Analyzing Results

As this is a large simulation and could take many hours to run, the results of an already completed simulation have been provided.

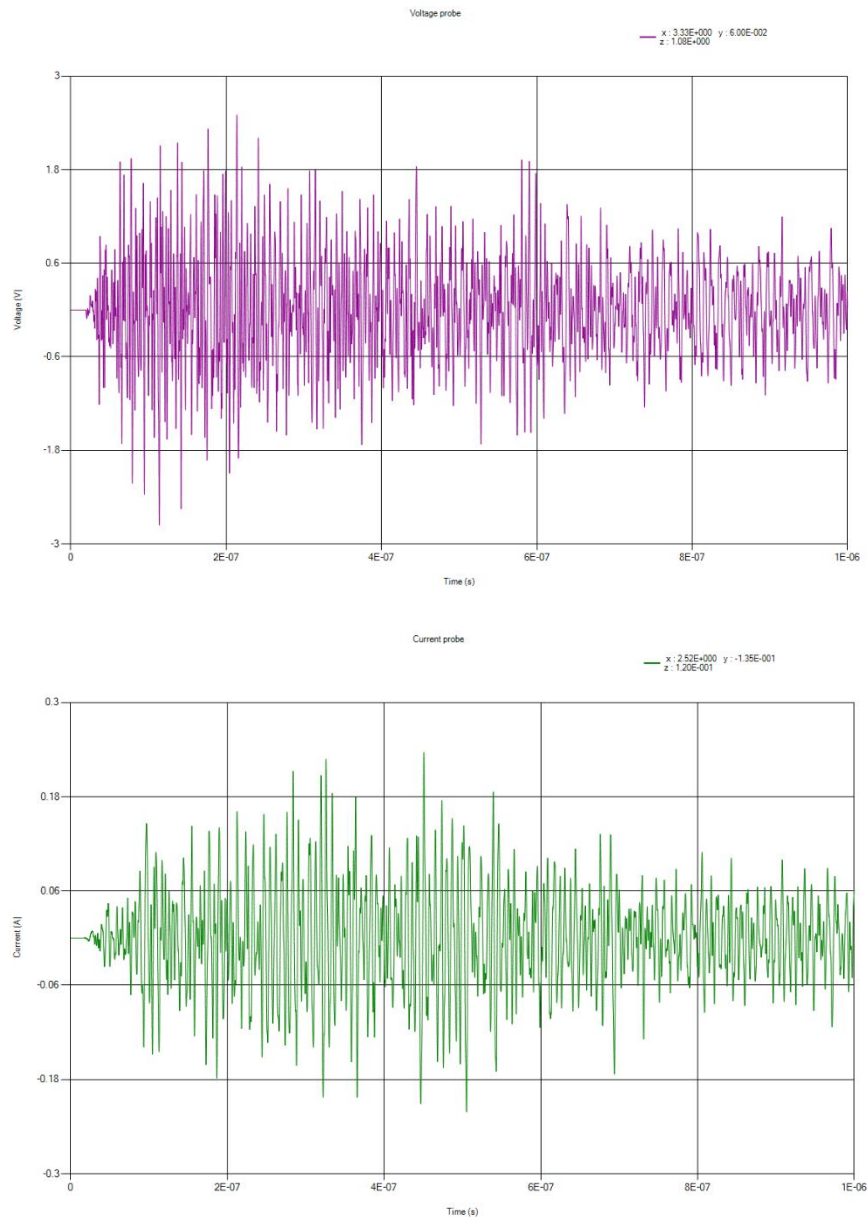
1. To load the results, open the file called CarRI\_Demo\_Final.scdoc in SpaceClaim.
2. Expand the **Results – CarRI\_Demo\_Final** tab in the **Simulation Tree**.
3. Right click on Animation Probe and click **Generate Animation** in the dropdown menu.
4. In the pop-up window, press the play button at the top. This will load the animation of the currents on the surface of the car. You may need to spin the model to find the best angle. The scale of the legend can be changed by clicking the Legend tab.
5. Geometry visibility can be toggled on/off in the usual way, by checking/unchecking the box to the left of the desired geometry leaving just the mesh behind.
6. The animation can be saved as a GIF file by clicking on  or a still shot with .





Now we will plot the results of the voltage and current probes.

1. Right click **Voltage probe** under **Results** and **CarRI\_Demo\_Final** in the **Simulation tree** and select Create plot from the drop-down menu.
2. Right click **Voltage probe** under **Visualization** and **2D Plots** in the **Simulation tree** and select Show from the drop-down menu.
3. The induced voltage as a function of time in wire W\_200\_1 is shown in the plot. The plot settings can be adjusted by right clicking in the plot area and clicking Configure and Chart. The data can be extracted by right clicking in the plot area.
4. The FFT of any signal can be generated by right clicking the Voltage probe under the results in the simulation tree then following the same process as above. This is a useful feature to explore the spectral content of a given signal.



5. Repeat steps 1 and 2 for the Current probe results on shield CL\_016\_2.



Now we will plot the result of the bulk current probe measurement.

1. Right click **Bulk Current** under **Results** and **CarRI\_Demo\_Final** in the **Simulation tree** and select Create plot from the drop-down menu.
2. Right click **Bulk Current** under **Visualization** and **2D Plots** in the **Simulation tree** and select Show from the drop-down menu. The net current passing through the bulk harness segment SEG is given as a function of time. Again, the plot settings can be adjusted by right clicking in the plot area and clicking Configure and Chart. For example, you may want to change the scale of the horizontal axis to be linear. The data can be extracted by right clicking in the plot area.

