

7655 West Mississippi Avenue, Suite 300, Lakewood, CO 80226-4332, P.O. Box 260263, Denver, CO 80226-0263 (303) 980-0070, Fax: (303) 980-0836

# EMC 2016 Demo

Jennifer Kitaygorsky, PhD Tim McDonald, PhD <u>tim@ema3d.com</u>

May 15, 2019

EMA has updated these simulation instructions. Contact us anytime with questions or concerns.

#### Background

This demonstration is the simulation portion of a DO-160 Cable Susceptibility Measurement. The actual test was done in July 2014, and the files "measCurrent.dat" and "measVoltage.dat" are the results from that test.

#### Import the Model

- 1. Open CADfix.
- 2. Navigate to the "File" drop down menu; click "Open".
- 3. Navigate to whatever folder you want to work out of.
- 4. Type "demo.fbm" in the "File name" section.
- 5. Click "Import".

## Create the Box Geometry

- 1. Type "SETO GPLANE" in the command window. Here you are opening and creating a set called GPLANE, to create the planar box part of our geometry.
- 2. Open the Tools  $\mathbf{z}$  tab on the left side of the screen.
- 3. Expand the "Build" Euild menu.
- 4. Select "Create/edit points".
- 5. In the "Position" box on the lower left, enter "0.0" in X, Y, and Z and click "Apply" and note that Q1 is created (Important: the name of your point may be different than Q1, this may happen because CADfix automatically names points, lines, etc. Generated names will appear in the lower left hand corner of the window after the feature is generated. Please keep careful track of names and do not solely rely on this document. For example if your point is named Z5 instead of Q1 replace all instances of Q1 in this document with Z5).
- 6. Select "Create/edit lines".
- 7. Find the "Swept line" tab \* 7.
- 8. Enter "Q1" as the Sweep Point.
- 9. In the "Vector/Axis" section click "Define".
- 10. In the "Define Vector/Axis" dialog, find the "Type" field and click the "Absolute" radio button.

Туре		
۲	Absolute	
$\bigcirc$	Relative	

11. In the "Translation" field, enter "20.0" in the "dY" field, as shown below.

Define Vector / Axis		ß
Type Absolute	Translation dX 0.0	
Relative	dY 20	
Method	dZ 0.0	
<ul> <li>Two points</li> <li>Point, vector</li> <li>*</li> </ul>		
ОК	Cancel	

- 12. Click "OK" in the Dialog and then click "Apply" in the "Create/edit lines" tool.
- 13. Line "U1" should be displayed.
- 14. Select "Create/edit surface".
- 15. In the "Create/edit surface" tool, select the "Swept surface" tool 🔗.
- 16. Type line "U1" in the "Sweep line" field.
- 17. In the "Vector/Axis" section click "Define".
- 18. In the "Define Vector/Axis" dialog, find the "Type" field and click the "Absolute" radio button.



19. In the "Translation" field, enter "72.0" in the "dX" field and "0.0" in the "dY" and "dZ" fields, as shown below.

Define Vector / Axis	ß
Туре	Translation
Absolute	dX 72.0
Relative	dY 0.0
Method	dZ 0.0
<ul> <li>Two points</li> <li>Point, vector</li> <li>*</li> </ul>	
·	
ОК	Cancel

- 19. Click "OK" in the Dialog and then click "Apply" in the "Create/edit surface" tool.
- 20. Surface "V1" should be displayed.



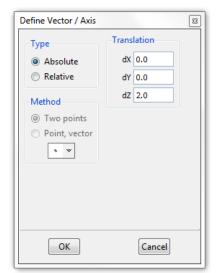
- 21. Type "SETC GPLANE" in the command window. This closes the GPLANE set, which contains the surface V1 and all of the lines and points it contains.
- 22. Type "CLIP BOTH OFF". This removes the clipping planes so that can see all of the geometry.
- 23. Type "PLUS SI GPLANE". This displays the surfaces of GPLANE.

## Create the Cable

- 1. Type "SETO ENDL" in the command line. This creates a new set, named "ENDL", which will be the left vertical piece that connects the cable to the GPLANE.
- 2. Open the Tools  $\Xi$  tab on the left side of the screen.
- 3. Expand the "Build" Euild menu.
- 4. Select "Create/edit points".
- 5. In the "Position" box on the lower left, enter "6.0" in X, "10.0" in Y, and "0.0" in Z and click "Apply" and note that point Q5 is created.
- 6. Select "Create/edit lines".
- 7. Find the "Swept line" tab \* 7.
- 8. Enter "Q5" as the Sweep Point.
- 9. In the "Vector/Axis" section click "Define".
- 10. In the "Define Vector/Axis" dialog, find the "Type" field and click the "Absolute" radio button.



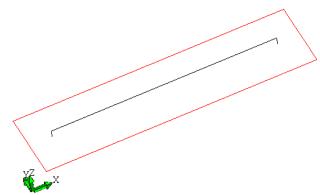
11. In the "Translation" field, enter "0.0" in the "dX" field, "0.0" in the "dY" field, and "2.0" in the "dZ" field, as shown below.



- 12. Click "OK" in the Dialog and then click "Apply" in the "Create/edit lines" tool.
- 13. Line "U5" should be displayed.
- 14. Type "SETC ENDL". This closes the set named "ENDL".
- 15. Type "SETO ENDR". This creates and opens the set named "ENDR", which will be the right vertical piece that connects the cable to the GPLANE.
- 16. Open the Tools  $\Xi$  tab on the left side of the screen.
- 17. Expand the "Build" EBuild menu.
- 18. Select "Create/edit points".
- 19. In the "Position" box on the lower left, enter "66.0" in X, "10.0" in Y, and "0.0" in Z and click "Apply" and note that point Q7 is created.
- 20. Select "Create/edit lines".
- 21. Find the "Swept line" tab \*\* .
- 22. Enter "Q7" as the Sweep Point.
- 23. In the "Vector/Axis" section click "Define".
- 24. In the "Define Vector/Axis" dialog that pops up, find the "Type" field and click the "Absolute" radio button.



- 25. In the "Translation" field, enter "0.0" in the "dX" field, "0.0" in the "dY" field, and "2.0" in the "dZ" field.
- 26. Click "OK" in the Dialog and then click "Apply" in the "Create/edit lines" tool.
- 27. Type "SETC ENDR" in the command window. This closes the "ENDR" set.
- 28. Type "SETO CABLE". This creates and opens the "CABLE" set, which will be the horizontal piece of the cable, that connects ENDL and ENDR.
- 29. Select the "Two Points" option 🖍 in the "Create/edit lines" tool.
- 30. Click on the endpoints of lines U5 and U6 (points Q7 and Q8) in the tool and click "Apply".
- 31. Note line U7 is created as shown below.



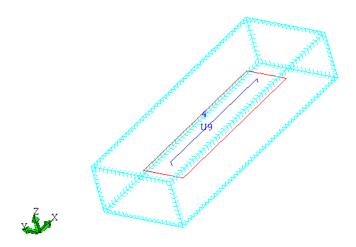
- 32. Type "SETC CABLE" in the command window.
- 33. Type "SETA ENDS ENDR ENDL" in the command window. This creates a new set, called "ENDS", and adds sets ENDR and ENDL to that set.

## Create the Define the Units and the Lattice

- 1. Navigate to Tools->Toolbox->EMA3D\_#2\_SpecifyUnits in the top menu.
- 2. Go to the button on the right labeled "Units"
- 3. Select the "Inches" radio button and click "OK".
- 4. Navigate to Tools->Toolbox->EMA3D\_#3\_DefineLattice in the top menu.
- 5. Go to the button on the right labeled "ConLatt"
- 6. Enter the values as shown below and click "OK". These are defining the size of the lattice. For other problems, you can click "Minimum Boundary" and EMA3D will automatically selected the minimum lattice boundaries for you.

Lattice Parameters			
	Start Coordinate:	-18	
X-Lattice	End Coordinate:	90	
	Cell Size:	2	
	Start Coordinate:	-10	
Y-Lattice	End Coordinate:	30	
	Cell Size:	2	
	Start Coordinate:	-10	
Z-Lattice	End Coordinate:	12	
	Cell Size:	2	
Mininum Boundary			
App	bly	Close	

7. Type "PLUS LATE" in the command window and inspect the computational lattice.

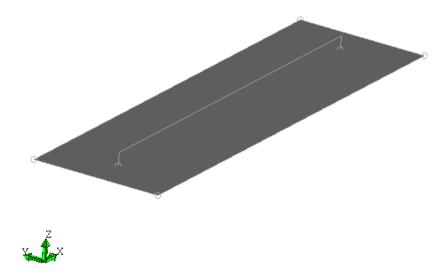


# Mesh the Geometry

- 1. Navigate to Tools->Toolbox->EMA3D\_#5\_MeshGeometry in the top menu.
- 2. Select "Mesh" \_\_\_\_\_\_ from the right menu.
- 3. Go to View -> Display Settings -> Mesh and set the edge color and shrink factor both to "Off".

Display settings	B
Basic Colours Selection Lighting Framerate Qua	lity Mesh
Element edge colour	OFF 💌
Mesh shrink factor	FF 💌
Repaint picture on OK or Apply	
OK Apply Ca	incel

- 4. Click OK.
- 5. Type "PLOT TWI ALL" in the command window and inspect the mesh to make sure everything is present, as shown:



## Define Time Step

- 1. Open Tools->Toolbox->EMA3D\_#6\_DefineTimeStep.
- 2. Select "TimeStep" \_\_\_\_\_ from the right menu.
- 3. Enter "8.5e-11" in the "Time Step (s)" field. (This is less than 90 % of the Courant criterion for a 2" uniform mesh).
- 4. Enter "5.015E-4" for the final simulation time, as shown below. EMA3D will automatically calculate the maximum time step based on the Courant condition. You can enter any time step that is less than or equal to the provided number.

Simulation Time Information					
In order to obey the Courant condition based on the cell size for x,y,z coordinates, the simulation timestep must be less than					
t <= 9.7	765e-011 s				
A good value to use for a	A good value to use for a simulation timestep might be				
t = 9.00e-011 s					
Time step	8.500E-011				
Final Simulation Time	5.015E-004				
Apply	Close				

- 5. Click "OK".
- 6. Select "MagTSteps" MagTSteps from the right menu.
- 7. Enter the following to gradually increase the permittivity value as the quasi-magnetostatic condition is met.

Magnetostatic Time Step	Tool	[	23
Magnetostatic	Increasing Time S	Step Parameters	
Number	Time	Factor	
1	1e-7	1.0	
2	5e-5	1000	
3			
BACK		NEXT	
APPLY		CLOSE	

8. Click "Apply".

## Define the Properties

- 1. Open Tools->Toolbox->EMA3D\_#8\_DefineProperties.
- 2. Select "PROPERTY" PROPERTY from the right menu.
- 3. Click "New".
- 4. In the "Name" field of the "New Property" window, type "ALUM".
- 5. In the "Type" spin box, change the definition to "Isotropic Surface", as shown below.

New Property		B
Name: ALUM		
Type:	Isotropic Surface	_
Close	Apply	Ok

- 6. Click "Apply".
- 7. Change the "Electric Conductivity (sigma)" field to "3.8e7".

Isotropic Surface	X
Name:	ALUM
Electric Conductivity (sigma):	3.8e7
Permittivity (epsilon):	8.854e-12
Permeability (mu):	1.257e-6
Magnetic Conductivity (sigmam):	0.0
Default Values	5
Close Apply	ОК

- 8. Click "OK".
- 9. Click "New".
- 10. In the "Name" field of the "New Property" window, type "PECL" as shown below.
- 11. In the "Type" spin box, change the definition to "PEC Line".

- 12. Click "OK".
- 13. Leave the other fields as the default, and click "OK".
- 14. Click "Close" on the EMA3D Property Editor.

#### Assign the Materials

- 1. Open Tools->Toolbox->EMA3D\_#8\_DefineProperties.
- 2. Select "PROPERTY" PROPERTY from the right menu.
- 3. Highlight "ALUM" in the "Property Editor" popup window.
- 4. Click "Assign" in the "Property Editor" popup window.
- 5. Type "GPLANE" in the "Assign Property" popup window.
- 6. Click "OK".
- 7. Highlight "PECL" in the "Property Editor" popup window.
- 8. Type "ENDR" in the "Assign Property" popup window.
- 9. Click "OK".
- 10. Click "Close" on the EMA3D Property Editor.

#### **MHARNESS** Cables

- 1. Select the "Harness Editor" tool in the MHARNESS toolbox.
- 2. Right click on "Routes" and select "New Shield with conductors"
- 3. Enter "TSP1" as the "Route Name"
- Type "CABLE" into the "Select Route Segments" Box as shown below. Then press "Enter."

below. Then press "E Route Editor	Enter."		Routes	Routes	New - Conductor New - Shield	
	Route Name TS	P1	Segments Sources		New - Shield with conductors	
Select Route Segments			Probes			
CABLE D 🗘	<u>^</u>		Tools			
Al /		Select Route Properties	Settings			
	~					

MHARNESS Editor

Harness set name:

HARNESS

- 5. Press "New" for the "Select Route Properties".
- 6. Enter "TSP" as the "Property Name". Select "Twisted Shielded Pair" as the "Cable Type", change the "AWG" to "22", and leave all other values as default as shown below.

MH Cable Definitions			
Pro	perty Name TSP		
Cable Type: Twisted Shie	lded Pair (TSP)	AWG:	22 🗸
Define Cable Properties			
Wire radius	3.800000e-004	m	
Wire jacket thickness	2.550000e-004	m	
Wire resistance	5.310000e-002	Ohms/m	
Wire jacket dilectric value	3.000000e+000	]	
Shield filler thickness	1.700000e-004	] m	
Shield filler dielectric value	1.000000e+000	]	
Shield thickness	2.300000e-004	] m	
Shield dilectric value	3.000000e+000	]	
Shield resistance	3.280000e-002	Ohms/m	Estimate as overbraid
Shield transfer inductance (begining)	6.000000e-009	H/m	Estimate as overbraid
Shield transfer inductance (middle)	6.000000e-009	H/m	Estimate as overbraid
Shield transfer inductance (end)	6.000000e-009	H/m	Estimate as overbraid
Apply	Reset		Cancel

- 7. Click "Apply"
- 8. Select both ends of the cable for the "Termination Points" as shown below.

Select `	Termination	Doints-
Jereet	remnation	FUILLS

□         ∴         Q6         ∧           All ◆         ⊠         ⊡	Define Boundary Conditions Resistive 1e-6 Capacitive 1.0e-7
Use Second Terminatio	Inductive 1.0e-10
Apply	Cancel

- 9. Click "Apply"
- 10. Currently the cables are shorted to the end lines along with the shields. While the shield should stay shorted, we will need to open one end of the cable.
- 11. Expand the "Routes" tree and the "ROUTE1" tree in the harness editor.
- 12. Right-click on the first cable "TSP1C1" and select "Edit."

MHARNESS	Editor		×
	Harness set name:	HARNESS	
Routes	Routes     G-TSP1		<b>A</b>
Segments	TSP1C1	Edit	
Sources	TSP1C2	Delete Copy	
Probes		Сору	1
Tools			
Settings			

13. Check the "Use Second Termination Boundary Condition" box and enter the values as shown below.

elect Route Segments	7 ^	Select Route Pro	
	~	TSPMAT	New
Select Termination Po			
	Q6	<ul> <li>Define Bounda</li> <li>Resistive</li> </ul>	1.000000E-06
All •		Capacitive	1.0e-7
		Inductive	1.0e-10
Use Select Termination Po		tion Boundary Conditio	n
<u></u> 口 0	and the second s	Define Bounda	ary Conditions
All • 🔀 🌁		Resistive	1e6
		Capacitive	1.0e-7
			01.0e-10
		~	

- 14. Click "Apply"
- 15. Repeat this process for "TSP1C2."
- 16. Open the "Sources" tab.
- 17. Right click on "Cable Current" and select "New".
- 18. Enter the values as shown below.

MHARNESS Current Probe Tool	
Define Current Probe Options	
Probe Name:	CURPROBE2
Probe Start Time:	0.0
Probe End Time:	0.0005015
Probe Step Time:	8.500e-9
Select which segment you want to apply	this probe to U7
Select the probe location(s) along the segment	Middle Cell 🗸
Select the conductor to probe	TSP1C1 V
Apply	Cancel

- 19. Click "Apply"
- 20. Right Click on "Cable Voltage" and select "New".
- 21. Enter the values as shown below.

MHARNESS Voltage Probe Tool	x
Define voltage Probe Options	
Probe Name:	VOLTPROBE1
Probe Start Time:	0.0
Probe End Time:	0.0005015
Probe Step Time:	8.500E-9
Select which segment you want to apply t	his probe to U7
Select the probe location(s) along the segment	End Cell 🗸
Select the conductor to probe	TSP1C1 ~
Apply	Cancel

- 22. Click "Apply".
- 23. Click "Write" at the bottom of the window.
- 24. A popup window will appear, press "Save"

**Define Sources** 

1. Type "PLUS L ENDL" in the command window. This displays the ENDL line name.

```
Fb> prnt l endl
* Message 564 *
Server terminated for coprocessor 1
LINE START END DEF3 DEF4 DEF5 DIV ELEM TYPE PHYS MAT CONS CSYS
US Q5 Q6 4 +BE2/1 STR
```

2. Observe the line name which in this case is "U5".

EleCurDen

- 3. Click "EleCurDen" from the right menu (Electric Current Density Source).
- 4. Enter an Identification Name in this case it is "currden."
- 5. Enter "injected\_tsp\_current.dat" in the "Source File Name" field, which is the experimentallygenerated cable current source.
- 6. Select "Geometry" as the "Specification Type."
- 7. Select "Sigle Line" in the "Assign To" field.
- 8. Click "Beg to End" as the "Assignment Type" field.
- 9. Enter "U5" in the "Line Name" field, where U5 is the name of the line above. Change the multiplier to "1.0."

Electric Current Density Source Creation Tool				
Identification Name (12 char max):		CURRDEN		
Source Origin:	Source File     O Analytic Waveform			
Source File Name:		injected_tsp_curr		
Specification Type:	Geometry		O Nodes	
Assign To:	C Point	Single Line	C Set of Lines	
Line Name:		U5		
Assignment Type:	C Define Components	Beg to End	C End to Beg	
Multiplier:		1		
OK	APPLY		CLOSE	

10. Click "OK".

# Select Boundary Condition

- 1. Open Tools->Toolbox->EMA3D\_#11\_SelectBdyConds.
- 2. Select "BOUND" from the right menu.
- Click the "Low Freq (EFLD)" radio button in "ALL" column of the "Boundary Condition Editor" popup window.
- 4. Click "APPLY".

#### Prepare the 3D Simulation Input File

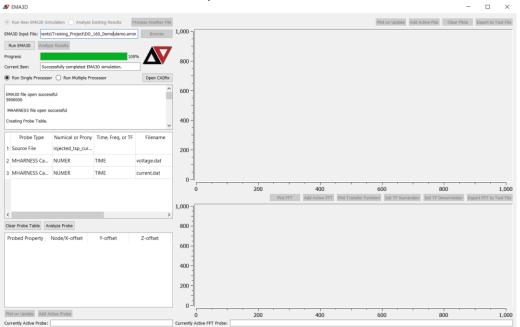
- 1. Click the Save button.
- 2. Open Tools->Toolbox->EMA3D\_ReviewTool.
- 3. Select "Review" from the right menu.
- 4. The Model Review Tool should contain the same elements as shown below:

CADfix Model Name: "DO160.fbm"			Approximate EMA3D Model Size (MegaBytes): 0				
GENERAL	UNITS	LATTICE	TIME STEPS	MATERIALS	SOURCES	BOND. COND.	PROBES
'Num of Prc: 1'	'Inches'	'55 x 21 x 12'	'Mag: 8.500e-011'	'Num of Prop: 4'	'Num of Srcs: 1'	'E Low Freq'	'Num of Prob: 0'
REVIEW	REVIEW	REVIEW	REVIEW	REVIEW	REVIEW	REVIEW	REVIEW
REFRESH	RESTORE	Create EMA	3D Input File				CLOSE

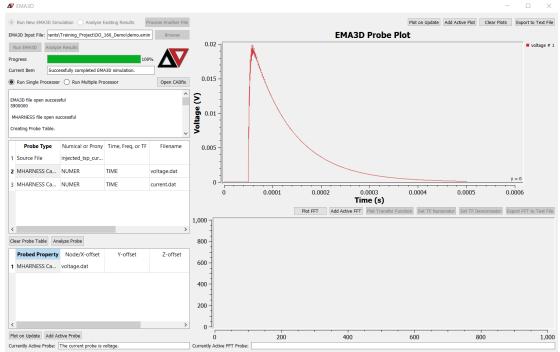
- 5. Review the simulation details, and click "Create EMA3D Input File" when ready to proceed.
- 6. Note the file name and click "OK" in the popup window.
- 7. Click Save in CADfix to preserve the model.

#### Run EMA3D Simulation

- 1. Select Tools -> Toolbox -> EMA3D\_Run from the top menu.
- 2. Select "Run\_EMA3D" From the right menu.
- 3. Select the "Run New EMA3D Simulation" radio button.
- 4. Click "Browse".
- 5. Select the .emin file by the same name created in the step above (demo.emin) and click "Open".
- 6. Click "Run EMA3D".
- 7. Wait for the simulation to complete.

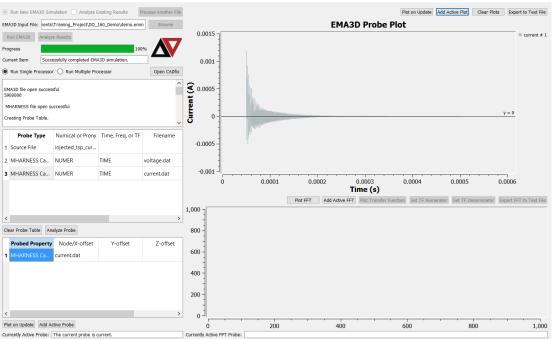


- 8. Double click the "Probe Type" number 2 entry.
- Double click the "Probed Property" number 1 entry. Observe the plot on the right hand side. The
  resulting curve is the voltage coupled to the inner pins. It should have a peak voltage of about
  20 mV as shown below.



#### 10. Repeat the previous two steps for the number 3 entry in "Probe Type".

**▲** EMA3D



Using a plotting program of your choice or the included 'compare.py' file you can plot the files "voltage.dat" and "measVoltage.dat" on the same plot. The result is a comparison of the simulated

voltage just calculated, and the voltage measured during the actual experiment. If you choose to use the included python file 'compare.py' make sure you have Python Version 2 as well as the NumPy, SciPy, and Matplotlib libraries. Also make sure the file is in the same folder as voltage.dat and measVoltage.dat when it is executed. The python script will automatically generate the image once it is run. The image below is what you should expect to see:

