ΔVEMA Electro Magnetic Applications, Inc.
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

EMA3D Simulation Example Avionics Analysis Training

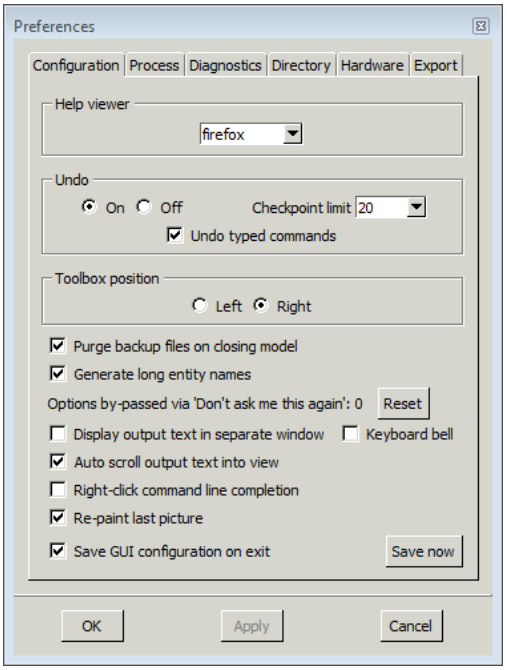
Cody Weber, cody@ema3d.com
Tim McDonald, PhD, tim@ema3d.com

June 30 - July 1 2014

Model 1 - Imported box with display, vent holes, connectors and seams

Import the Model

1. Open CADfix
2. Open File -> Preferences and make sure the “Generate long entity names” option is checked as shown below:



3. Navigate to the appropriate folder
4. Change File type to CADfix GDX
5. Highlight avionics_training.gdx

6. Check Import Only button
7. Click Go
8. Rotate and Inspect the imported Model

Specify Units

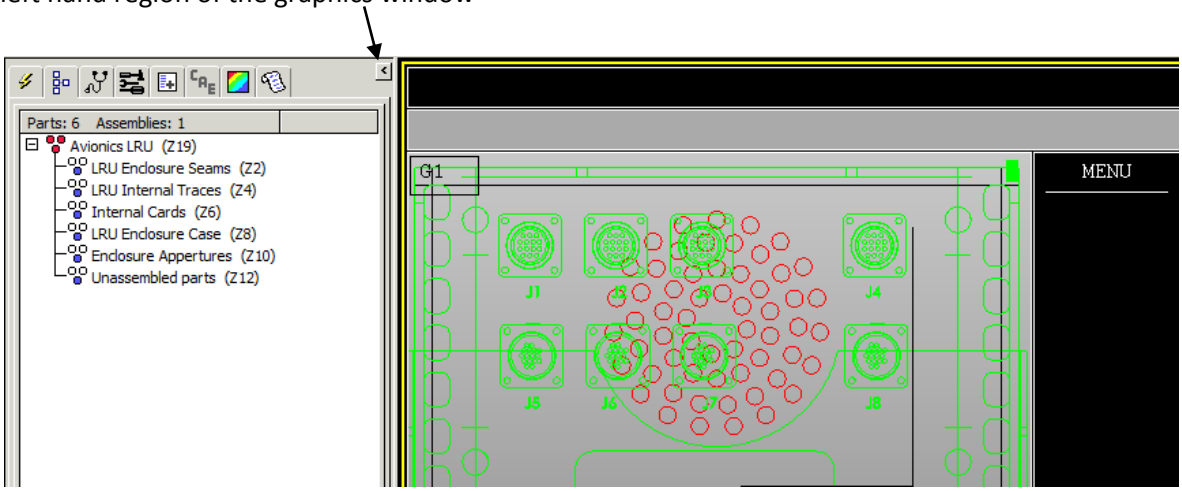
1. Open Tools -> Toolbox -> EMA3D_#2_SpecifyUnits
2. Click Units from the right side of the screen.
3. Select the millimeters (mm) radio button.
4. Click OK

Create the Major Sets from the Hierarchy

1. Click the hierarchy icon  on the left, then the assembly button at the bottom 
2. Type SETA SEAMS Z2 in the command window.
3. Type SETA TRACES Z4 in the command window.
4. Type SETA CARDS Z6 in the command window.
5. Type SETA CASE Z8 in the command window.
6. Type SETA HOLES Z10 in the command window.

Geometric Tolerance Check and Menu Command Guide Viewing

1. Examine the model -> **PLOT G ALL**
2. Plot surfaces with assigned colors -> **PLUS SI ALL ASGD**
3. Check the geometric tolerance -> **PROC GTOL**
4. Set the geometric tolerance to be 0.01 mm -> **PROC GTOL 0.01**
5. Make sure the CADfix menu is on to help guide CADfix command structure -> **PLOC MENU ON**
6. The menu may not appear, to show the menu, click the hide/show arrow tab  , at the upper left hand region of the graphics window



Initial Model Cleanup and Revision Control

1. Eliminate duplicate Points, Lines, Surfaces and Shapes by using the MERG command **MERG P ALL GTOL NOLOCK**

MERG L ALL GTOL SIMPLE MINSLOP 0.0
MERG S ALL GTOL EQSH
MERG SH ALL GTOL (takes a minute or so)

2. Save the model as avionics_training_mod1.fbm

Original Set Cleanup and Inspection Using Clipping Planes

1. Observe that the CARDS are in both the CARDS set and the CASE set
2. **PLOT G CASE**
3. **PLUS SI CASE ASGD**
4. Use clipping planes to inspect CASE interior -> **CLIP BOTH AUTO**
5. Press F9 on the keyboard to activate/deactivate clipping planes
6. Press and hold the left mouse button and slide the mouse left or right to control the position of the near clipping plane
7. Press and hold the center (scroll button) mouse button and slide the mouse left or right to control the position of the far clipping plane
8. Press and hold the right mouse button and slide the mouse left or right to control the position of the both clipping planes at the same time
9. Position the near clipping plane so that the cards inside of the box can be seen
10. Notice that the cards inside of the box appear dark in color on one side of the card, shade both orientations of surfaces to better see the CARDS surfaces ->**PLOC SHAD BOTH**
11. Repaint the image -> **REP**
12. Plot the surfaces of the CARDS set to realize that the CARDS surfaces are in both the CASE and CARDS set -> **PLUS SI CARDS C**
13. Remove the CARDS from the CASE set -> **SETR CASE CARDS**
14. Wipe the CARDS surfaces from the plotted image -> **WIPE -1**
15. Observe that the CARDS are no longer in the CASE set
16. Turn clipping planes off -> **CLIP BOTH OFF**

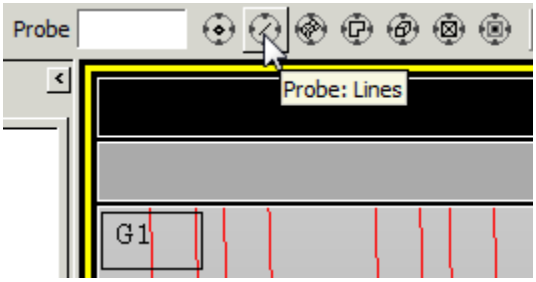
Create Connectors Set

1. Create a set for connectors using QADD command -> **QADD CONNECTORS**
2. Type **A** to adjust geometry
3. Type **N** to create multiple points around the connectors and generate a new geometry
4. When the surrounding geometry is nearly completed, type W to include only whole entities
5. Type **S** to grab all surfaces inside of the newly defined geometry
6. Investigate the newly created CONN set -> **PLOT G CONNECTORS** , **PLUS SI CONNECTORS ASGD**
7. Complete the connector set down to add all points, lines and surfaces that are used to create the connector surfaces -> **COMP CONNECTORS**
8. Remove the original connectors set from the CASE set -> **SETR CASE CONNECTORS**

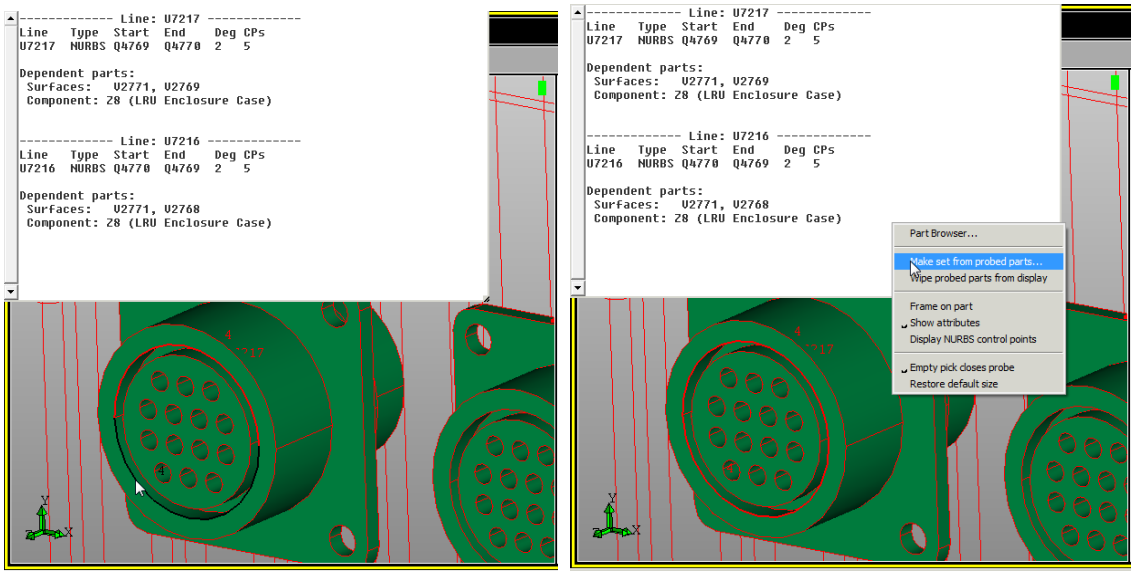
Add modified Connectors to the Case

1. Connectors J1 and J2 will be represented for simulation
2. Include the geometry of the case for reference -> **PLUS G CASE**

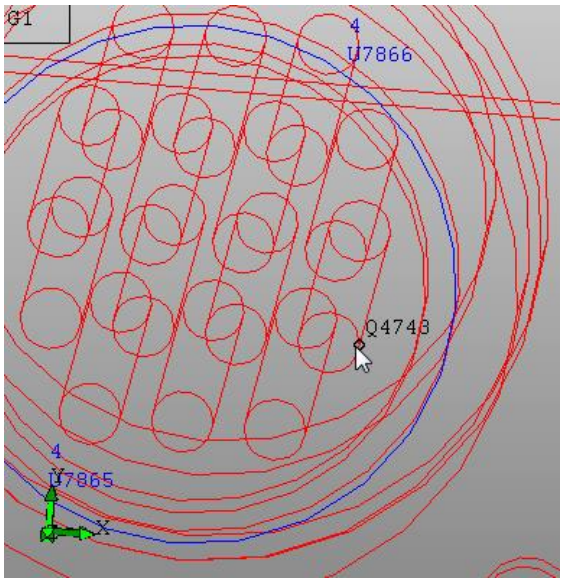
3. Make sure that the GMER option is off in CADfix -> **PROC GMER OFF**
4. Select the 2 lines U7216 and U7217 that form the outline for the interior channel of J1 using the Probe: Lines select tool as shown below:

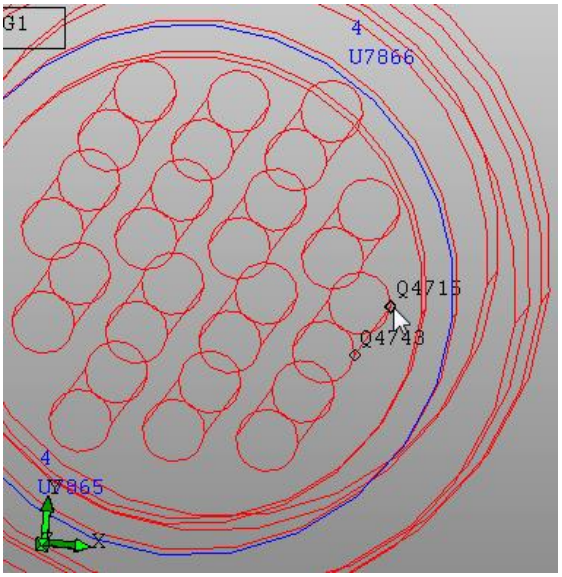


5. Right click in the probe entity window and chose make a new temporary set called TEMP1

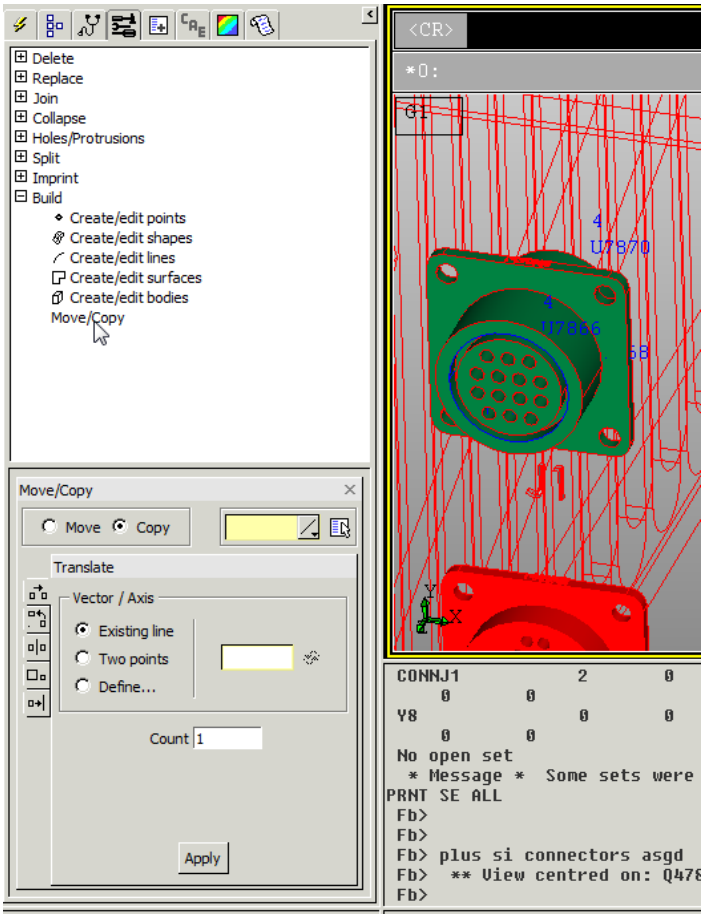


6. Copy the TEMP1 set of lines into the set CONNJ1 -> **COPY TEMP1 CONN1**
7. Plot the lines of the CONNJ1 set -> **PLUS LA CONNJ1 B**
8. Open the CONNJ1 set -> **SETO CONNJ1**
9. Now sweep the 2 lines in CONNJ1 so that they meet the end of the metal connector inside of the box using the menu point to point SWEP commands to select the points with the mouse cursor -> **SWEP CONNJ1 ! TRA Q4743 Q4715**

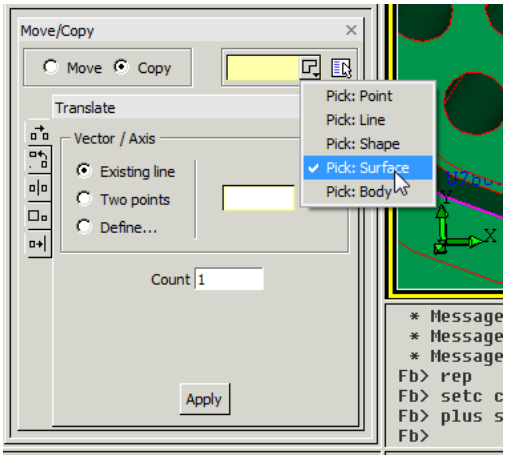




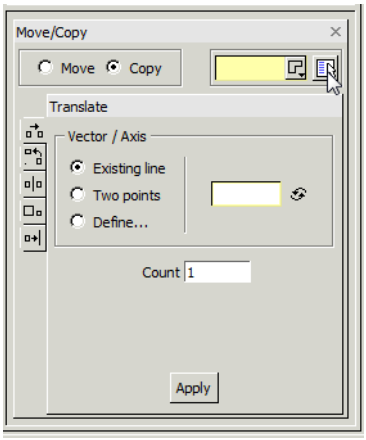
10. Close the CONNJ1 set -> **SETC CONNJ1**
11. Copy the CONNJ1 Set to the J2 (CONNJ2) position using the Move/Copy, set manager technique in CADfix as shown below.
12. Click the Move/Copy tool in the CADfix Build tools on the left side of the screen.



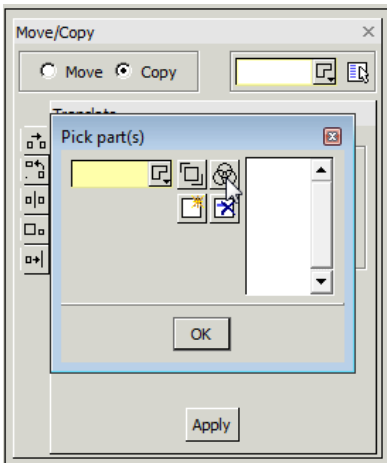
13. Click the drop down menu to select the Pick: Surface option:



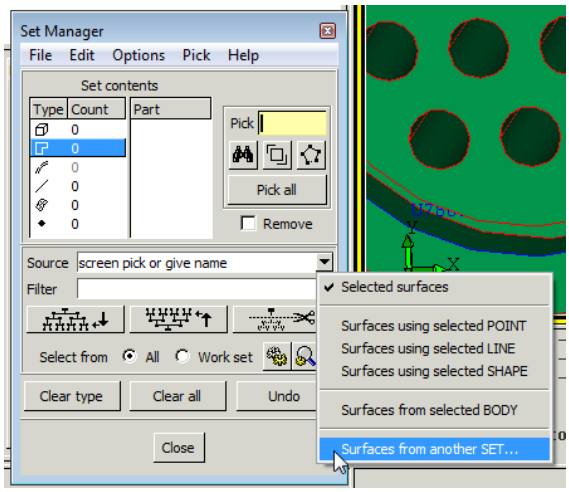
14. Click the list manager button in the upper right hand corner



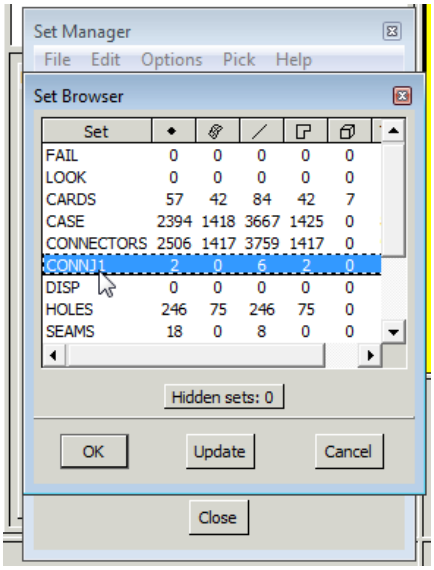
15. Click the Set Manager button in the Pick Part(s) window:



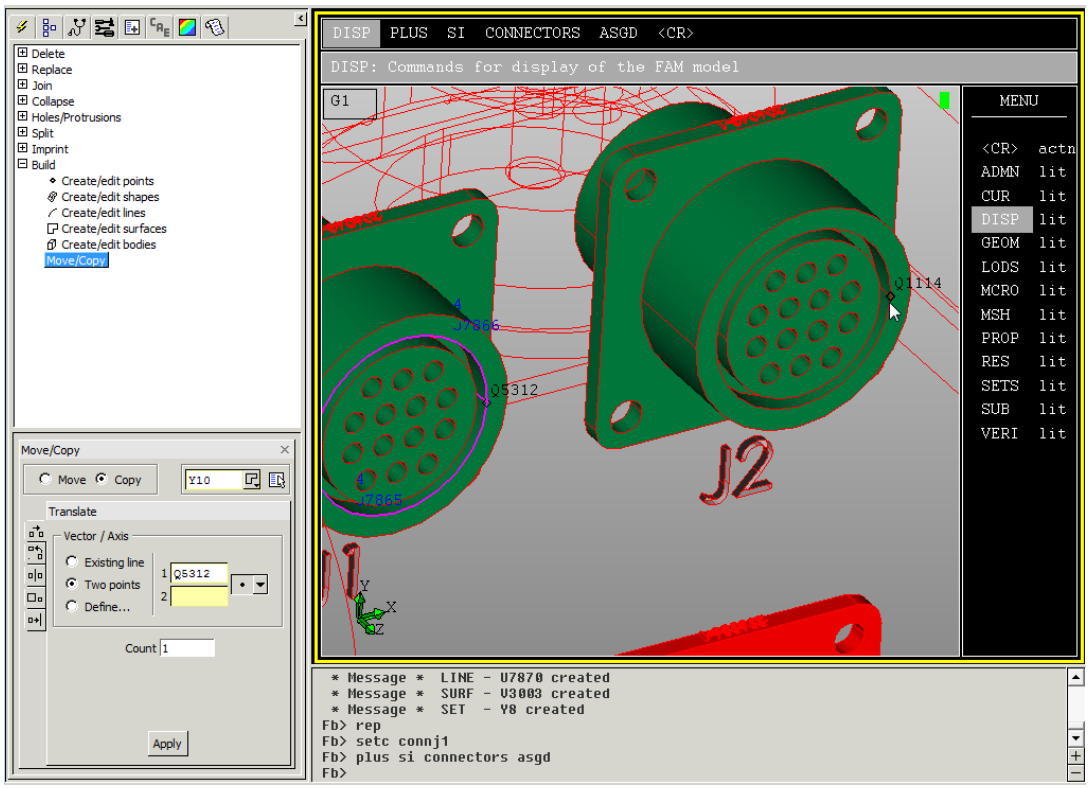
16. In the Source drop-down menu, select the Surfaces from another SET… option:



17. Select the CONNJ1 set and click OK



18. Click Close in the Set Manager Window
19. Notice the highlighted surfaces are V3002 and V3003 and click OK
20. Click the two points option the Vector / Axis section of the window and select the two inside wall points of connectors J1 and J2, Q5312 and Q1111



21. Click Apply to complete the copy sequence
22. Click Apply -> Type CONNJ2 for the name of the new part and click OK
23. Plot the case and newly created connectors ->

    PLOT G CASE

    PLUS SI CASE ASGD

    PLUS G CONNJ1 B

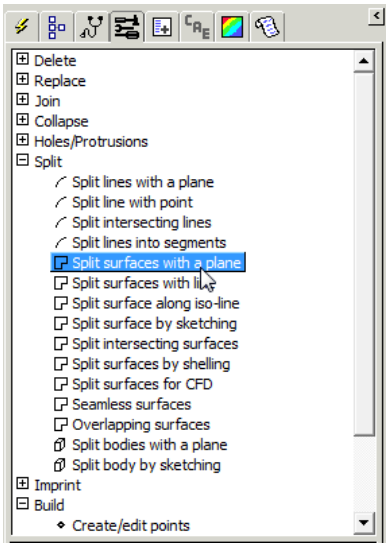
    PLUS G CONNJ2 B

    PLUS SI CONNJ1 C

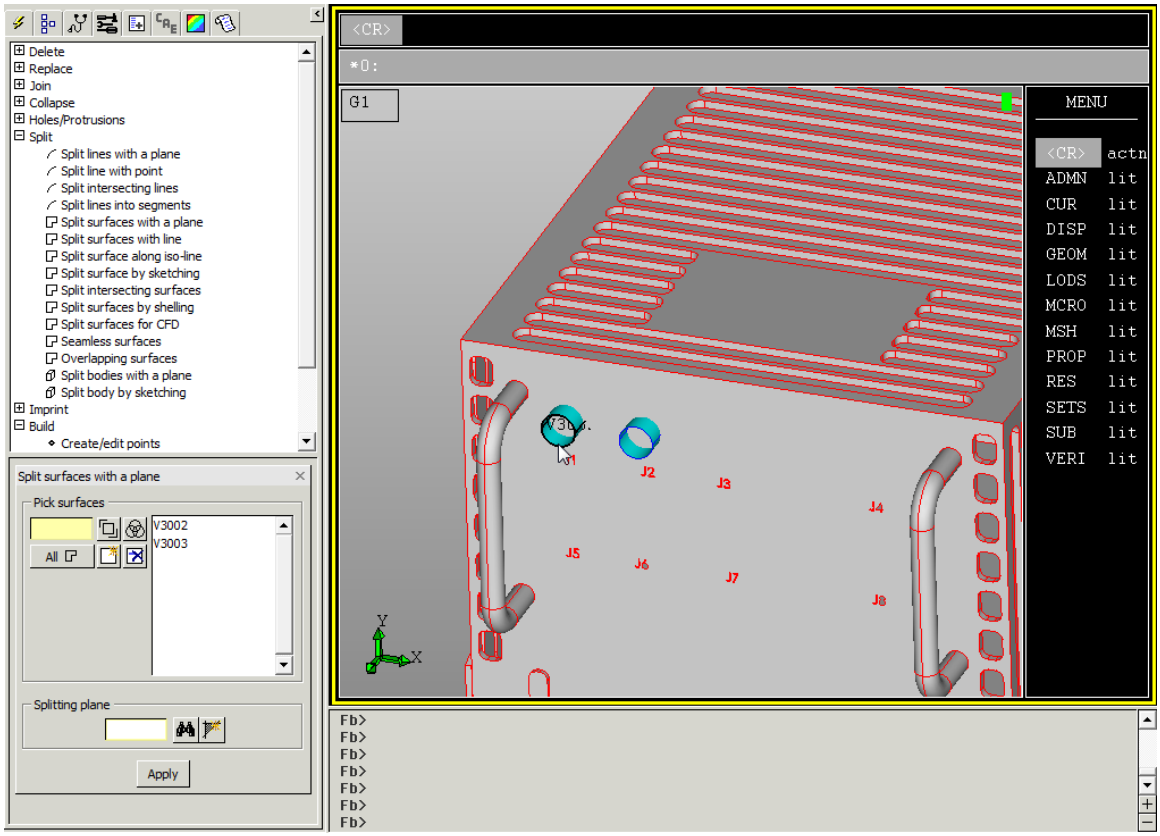
    PLUS SI CONNJ2 C

Split Connector Surfaces with Front Case Surface

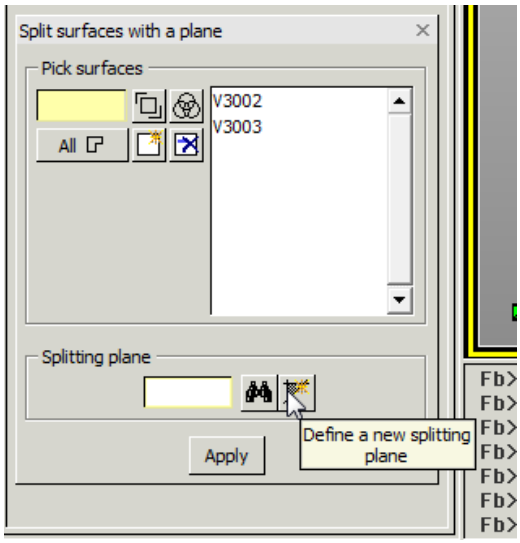
1. Open the CONNJ1 set -> **SETO CONNJ1**
2. Click the Split surfaces with a plane tool as shown below:plus



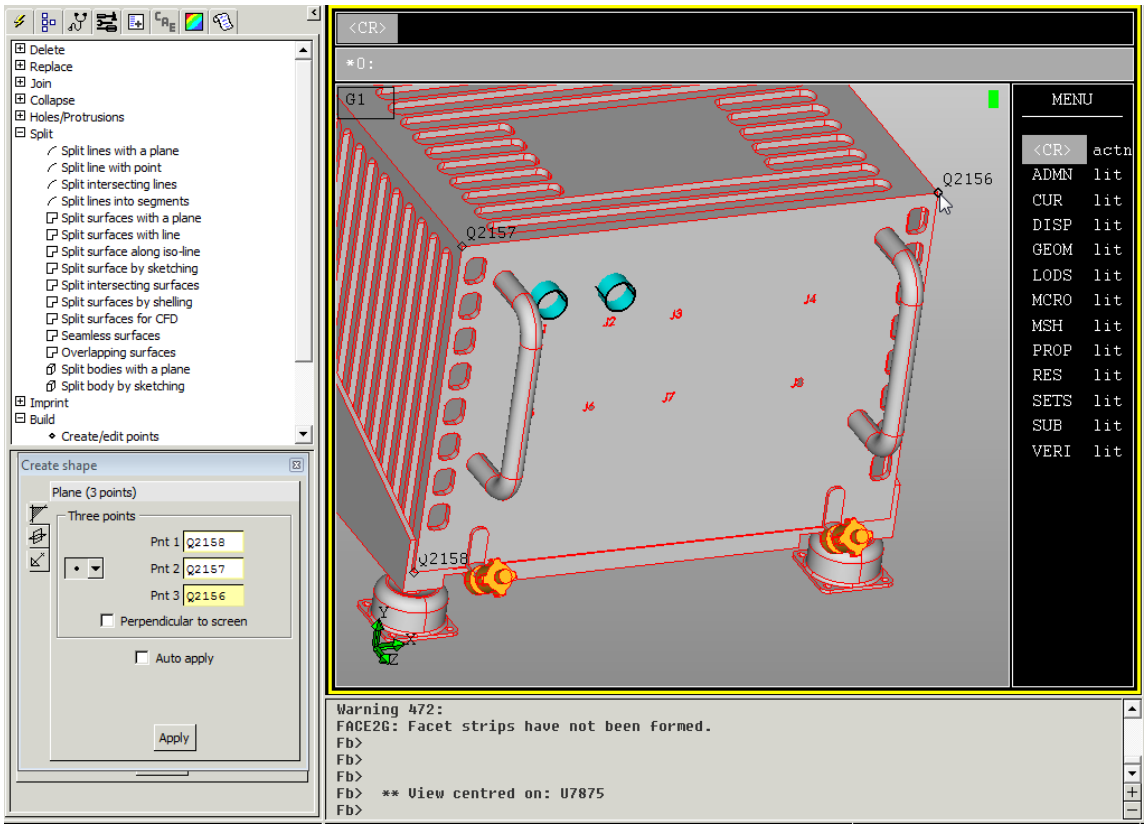
3. Select the 2 surfaces of the newly created connector CONNJ1 in the Pick surfaces section:



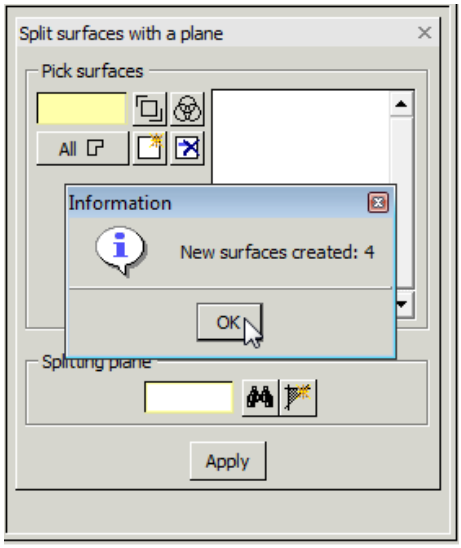
4. Click the Define a new splitting plane button for the Splitting plane section



5. Select three corner points on the face to define the plane as shown below:



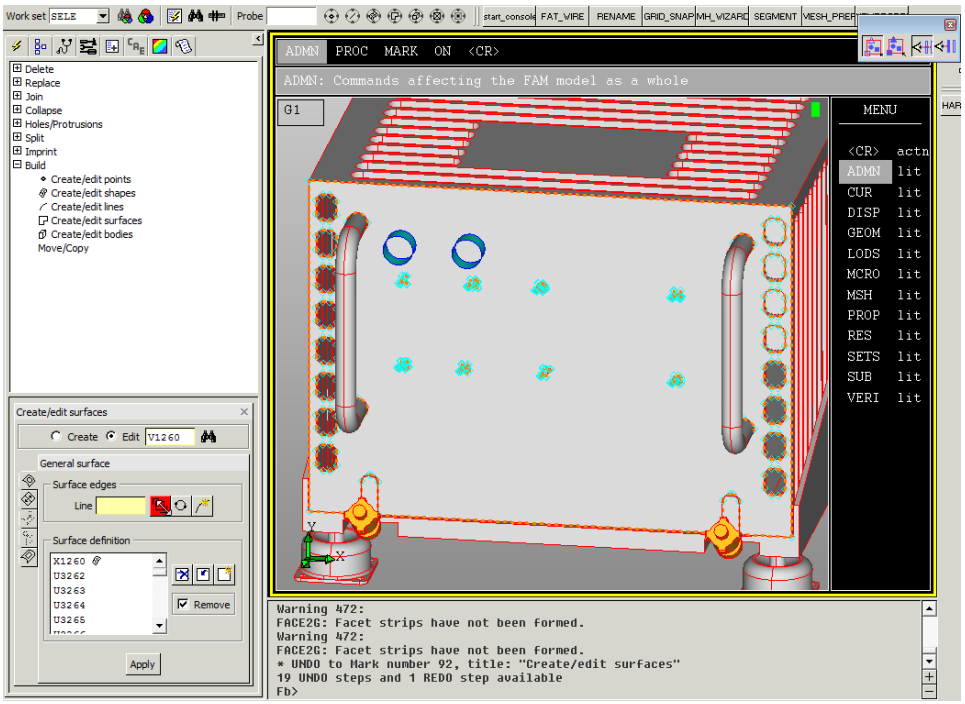
6. Click Apply in the Create Shape window.
7. Click Apply in the Split surfaces with a plane window.
8. A window should pop up indicating that 4 new surfaces have been created. Click OK.



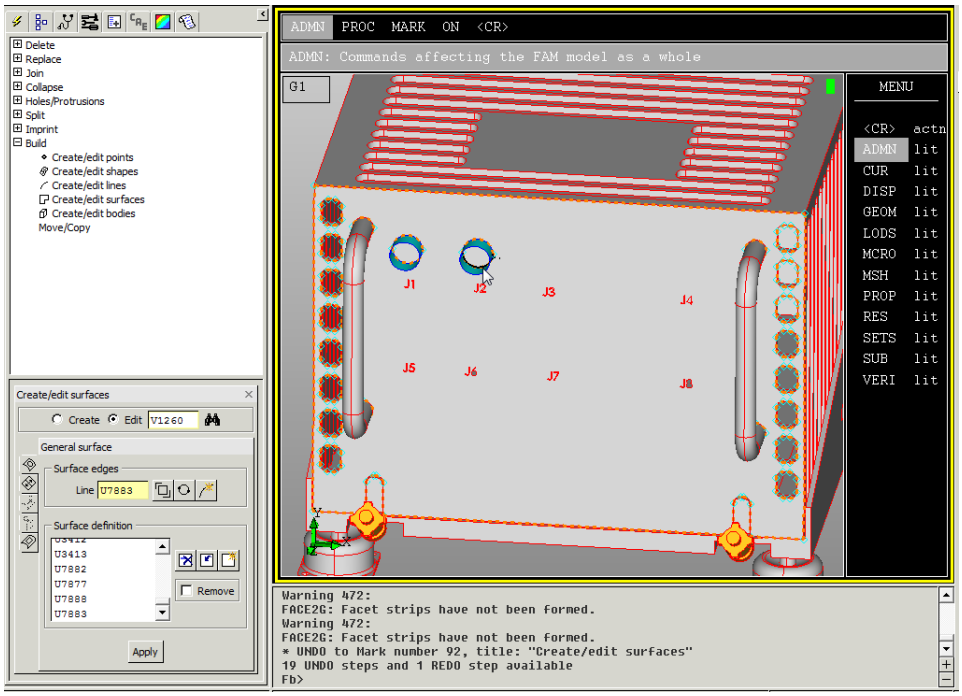
9. Close the CONNJ1 set -> **SETC CONNJ1**
10. Open the CONNJ2 set -> **SETO CONNJ2**
11. Repeat the above process to split the CONNJ2 surfaces
12. Close the set CONNJ2 after the surfaces have been split -> **SETC CONNJ2**

Modify the Case Face with Connectors

1. Open the CASE set -> **SETO CASE**
2. Remove the J1-J8 labels from the case surface using the surface edit tool as shown below.
3. Click the Edit button and select surface V1260.
4. Click the Remove box.
5. Click the box select tool to envelope all of the connector labels at once and remove them from the surface definition.



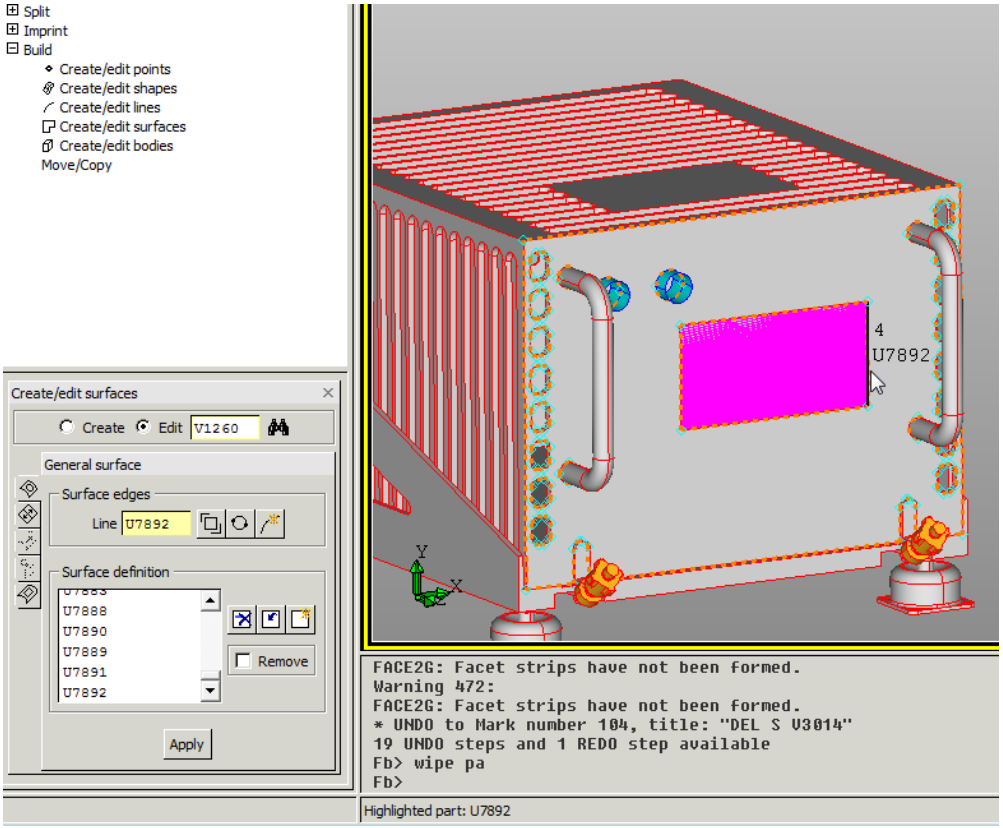
6. Uncheck the Remove box in the lower right corner
7. Click the newly created connector lines that lay on surface V1260.



8. Click Apply
9. In the Define embedding shape window, make sure the Existing button is selected and click OK
10. Notice the holes that have been created on the box surface in the connector channels
11. Save the model as avionics_training_mod2.fbm

Add Display Panel

1. Create a new point -> **PNT ! -50 100 0**
2. Sweep the new point to make a line -> **SWEP Q5697 ! TRA 0 75 0**
3. Sweep the new line to make a surface -> **SWEP U7889 ! TRA 150 0 0**
4. Type -> **REP**
5. Edit the front face surface of the CASE to exclude the new Panel surface area
6. Do this by adding the new lines to the surface definition in the Create/edit Surfaces tool:



7. Click Apply
8. Click OK when the Define embedding shape window pops up.
9. Add the display surface to a set called PANEL -> **SETA PANEL V3014**
10. Remove the display panel from the CASE set -> **SETR CASE PANEL**
11. Close all sets -> **SETC ALL**
12. Repaint the image and notice that the display panel is gone from the image -> **REP**

Add Vent Holes to Top Panel

1. Create lines that will be used as thin gaps to represent a grid of vent holes with the following commands:
   SETO VENTS
   SETO TEMP2
   PNT ! -65 250 -135
   SWEP Q5701 ! TRA 10 0 0
   COPY U7893 ! TRA 15 0 0 8

COPY TEMP2 ! TRA 0 0 10 7
SETC TEMP2
SETC VENTS
PLUS L VENTS B

2. Observe the lines on top of the box that will be a grid of vents

Create the Dipole Source

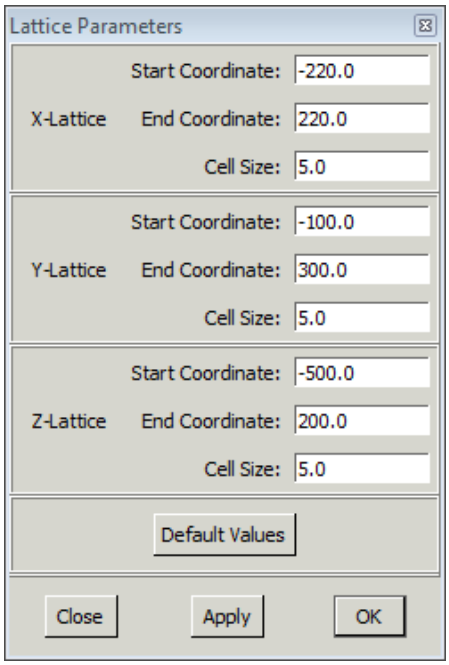
1. Create the lines for the dipole source using the following commands:
   SETO SOURCE
   PNT ! 0 100 150
   SWEP Q5845 ! TRA 0 5 0
   SETC SOURCE
   SETO DIPOLE
   SWEP Q5846 ! TRA 0 15 0
   SWEP Q5845 ! TRA 0 -15 0
   SETC DIPOLE

Make Master Mesh Set

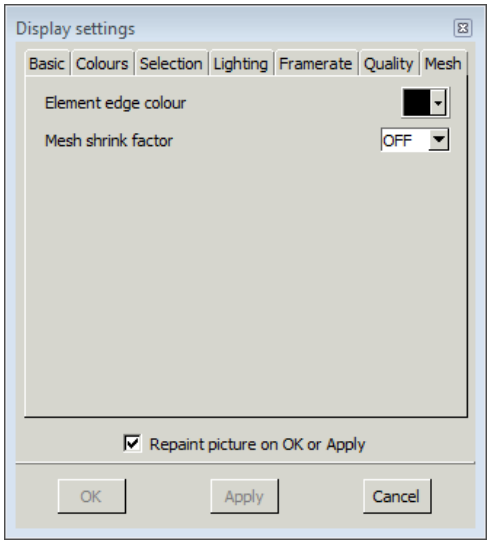
1. Create a master set that contains all of the components necessary for meshing and EMA3D analysis by adding previously defined sets to BOX:
   SETA BOX CARDS CASE CONNJ1 CONNJ2
   SETA BOX DIPOLE HOLES PANEL SEAMS
   SETA BOX SOURCE TRACES VENTS
   PLOT G BOX
   PLUS SI BOX ASGD

Define Lattice and Mesh Geometry

1. Open Tools->Toolbox->EMA3D_#3_DefineLattice
2. Select ConLatt from the right side of the screen.
3. Enter the following lattice parameters:

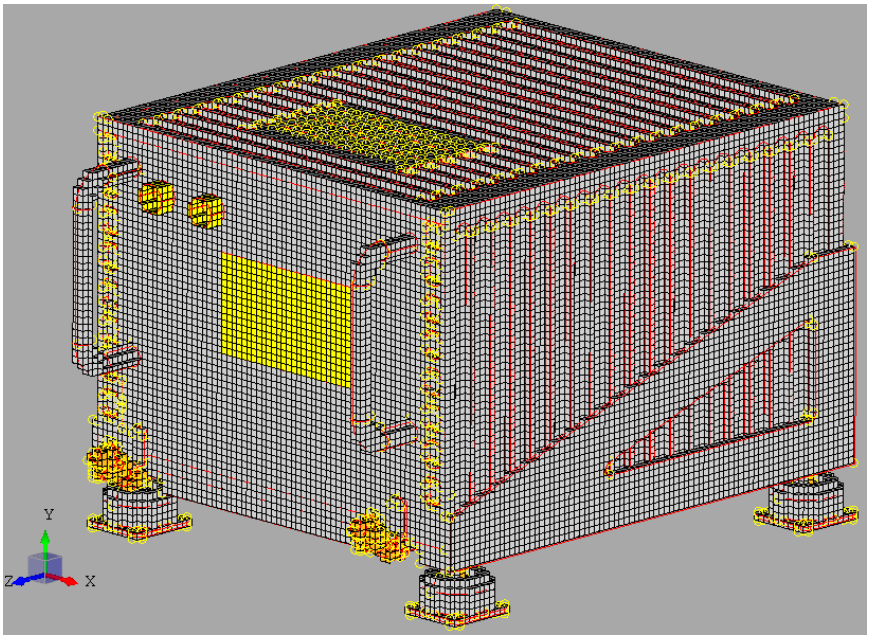


4. Click OK
5. Plot the lattice of the problem space -> **PLUS LATE**
6. Make sure the lattice surrounds the entire box with a few millimeters of padding on each side.
7. Make sure the geometric tolerance is still set to 0.01 -> **PROC GTOL 0.01**
8. Open Tools->Toolbox->EMA3D_#5_MeshGeometry.
9. Click the Mesh button
10. If geometry is changed or only the master mesh set is to be meshed, click the Setup button to prepare the mesh and type -> **MESH BOX**
11. Go to View -> Display Settings -> Mesh and set the edge color to Black and shrink factor both to Off



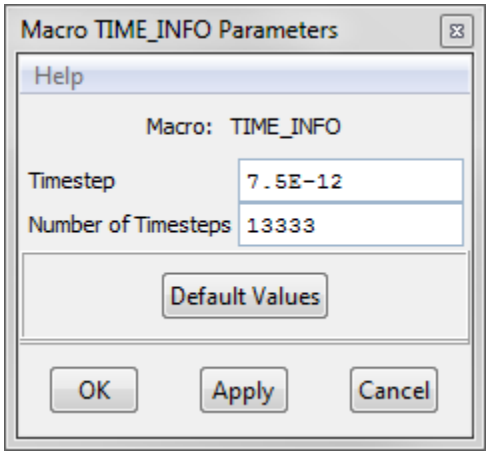
12. Click OK
13. Inspect the mesh to make sure everything is present -> **PLOT TWI BOX**

14. Save the model as avionics_training_mod3.fbm



Define Time Step

1. Open Tools->Toolbox->EMA3D_#6_DefineTimeStep.
2. Select TimeStep from the right menu.
3. Enter 7.5e-12 in the Time Step (s) field.
4. Enter 13,333 in the Number of Time Steps field, as shown below.

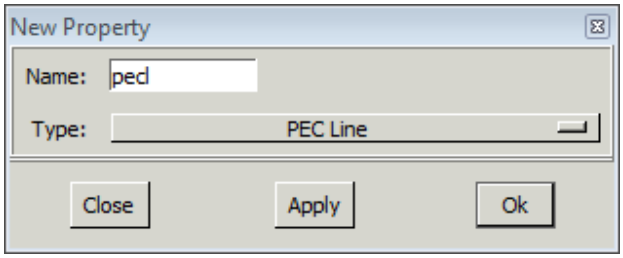


5. Click OK

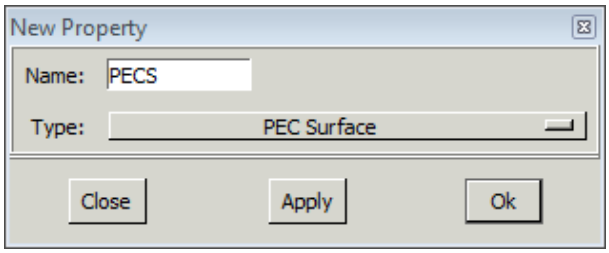
Define the Properties

1. Open Tools->Toolbox->EMA3D_#8_DefineProperties.
2. Select PROPERTY from the right menu.
3. Click New
4. In the Name field of the New Property window, type PECL

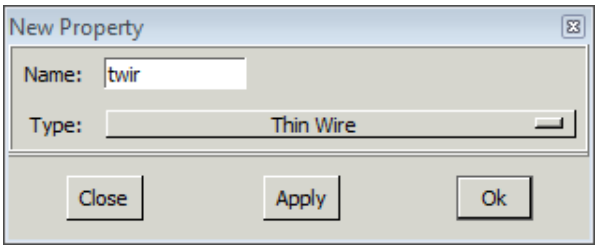
5. In the Type spin box, change the definition to PEC Line, as shown below



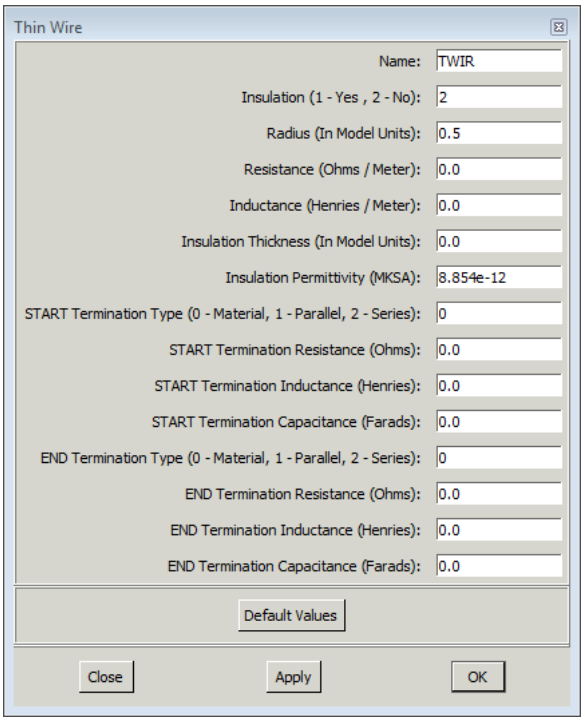
6. Click Ok in the New Property window.
7. Click Ok in the PEC Line window.
8. Click New
9. In the Name field of the New Property window, type PECS
10. In the Type spin box, change the definition to PEC Surface, as shown below



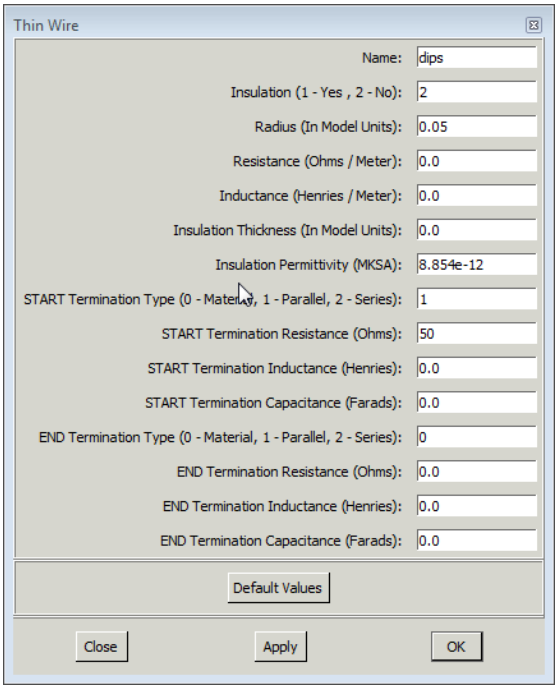
11. Click Ok in the window that pops up.
12. Click Ok in the PEC Surface window.
13. Click New
14. In the Name field of the New Property window, type TWIR as shown below.



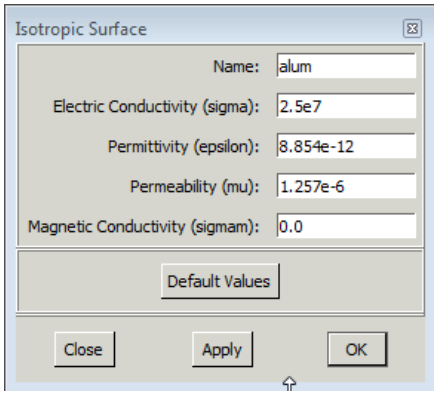
15. In the Type spin box, change the definition to Thin Wire.
16. Click OK
17. In the Thin Wire popup window, enter 0.5 in the Radius field.
18. Leave the other fields as the default, as shown below:



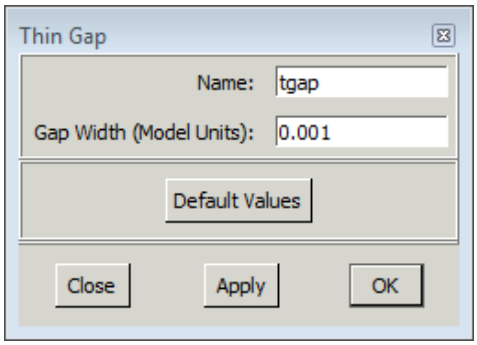
19. Click OK
20. Click New
21. In the Name field of the New Property window, type DIPS.
22. In the Type spin box, change the definition to Thin Wire.
23. Click OK
24. Change the START Termination Type to 1 - Parallel
25. Change the START Termination Resistance to 50 (Ohms)
26. Leave the other fields as the default, as shown below:



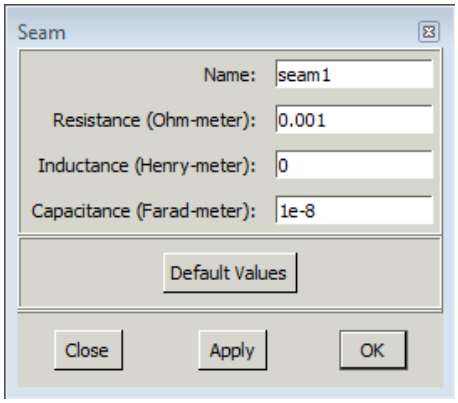
27. Click OK
28. Click New in the EMA3D Property Editor window.
29. In the Name field of the New Property window, type ALUM.
30. In the Type spin box, change the definition to Isotropic Surface.
31. Click OK
32. In the Isotropic Surface popup, enter 2.5e7 in the Electric Conductivity field, as shown below:



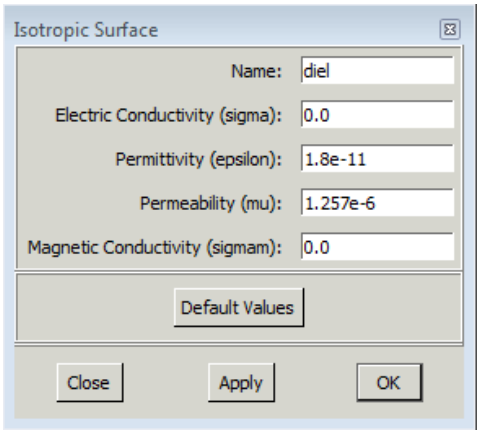
33. Click OK
34. Click New in the EMA3D Property Editor window.
35. In the Name field of the New Property window, type TGAP.
36. In the Type spin box, change the definition to Thin Gap.
37. Click OK
38. In the Thin Gap popup window, enter 0.001 in the Gap Width field, as shown below:



39. Click OK
40. Click New in the EMA3D Property Editor window.
41. In the Name field of the New Property window, type SEAM1.
42. In the Type spin box, change the definition to Seam.
43. Click OK
44. In the Seam popup window, enter the Resistance, Inductance and Capacitance values as shown below:



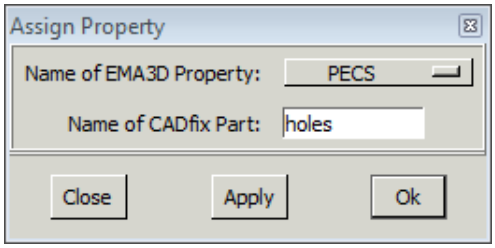
45. Click OK
46. Click New in the EMA3D Property Editor window.
47. In the Name field of the New Property window, type DIEL.
48. In the Type spin box, change the definition to Isotropic Surface.
49. Click OK
50. In the Isotropic Surface popup, enter 1.8e-11 in the Permittivity field, as shown below:



51. Click OK

Assign the Materials

1. Use the Property Editor to assign the materials to the appropriate sets.
2. Highlight PECS in the Property Editor window.
3. Click Assign in the Property Editor window.
4. Type HOLES in the Assign Property window.

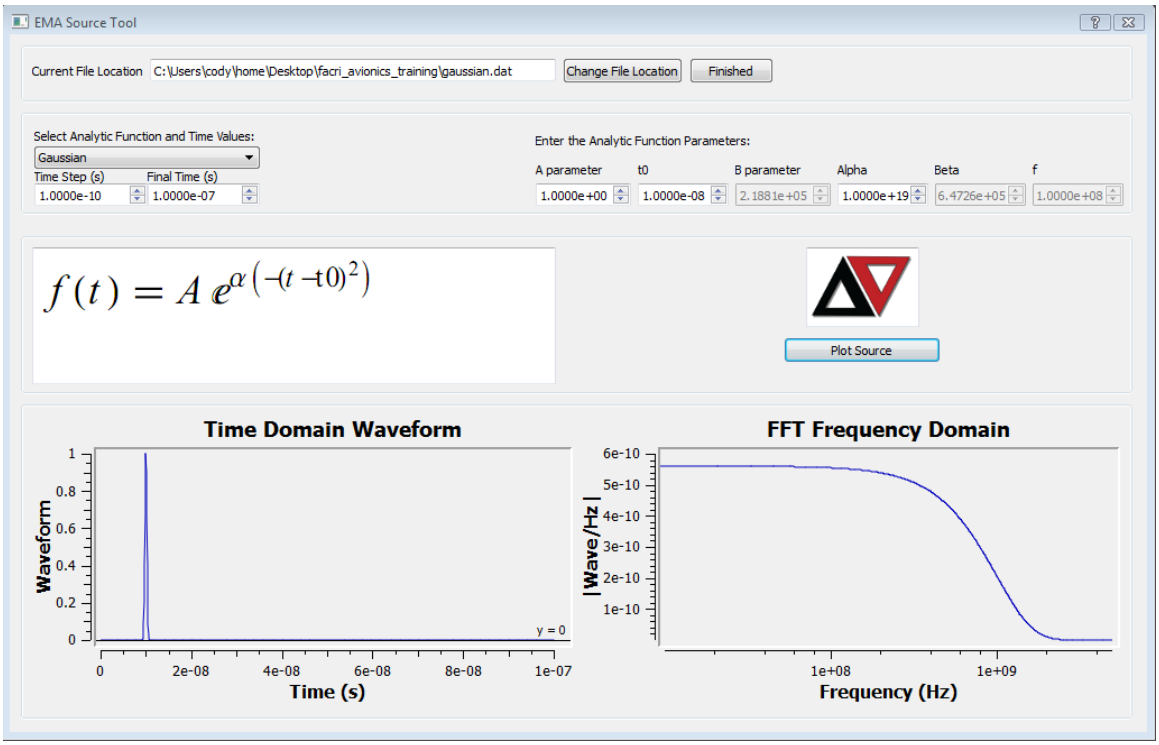


5. Click Apply
6. Use PECS again, type CASE in the Assign Property popup window.
7. Click Apply

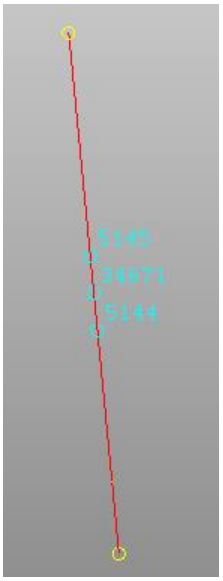
8. Select TWIR in the Name of Property spin box of the Assign Property popup window.
9. Type TRACES in the Assign Property popup window.
10. Click Apply
11. Select ALUM in the Name of Property spin box of the Assign Property popup window.
12. Type CARDS in the Assign Property popup window.
13. Click Apply
14. Type CONNJ1 in the Assign Property popup window.
15. Click Apply
16. Type CONNJ2 in the Assign Property popup window.
17. Click Apply
18. Select DIEL in the Name of Property spin box of the Assign Property popup window.
19. Type PANEL in the Assign Property popup window.
20. Click Apply
21. Select PECL in the Name of Property spin box of the Assign Property popup window.
22. Type DIPOLE in the Assign Property popup window.
23. Click Apply
24. Select TGAP in the Name of Property spin box of the Assign Property popup window.
25. Type VENTS in the Assign Property popup window.
26. Click Apply
27. Select SEAM1 in the Name of Property spin box of the Assign Property popup window.
28. Type SEAMS in the Assign Property popup window.
29. Click Apply
30. Select DIPS in the Name of Property spin box of the Assign Property popup window.
31. Type SOURCE in the Assign Property popup window.
32. Click Apply
33. Click Close in the Assign Property popup window.
34. Click Close in the Property Editor window.

Define Sources

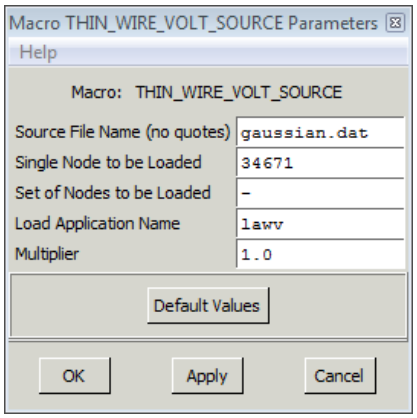
1. Open Tools->Toolbox->EMA3D_#10_DefineSources
2. Set the Current File Location to the current working project directory on your machine.
3. In the Select Analytic Function and Time Values spin box, change the value to Gaussian.
4. In the EMA Source Tool field, change the Time Step to 1.000e-10.
5. In the Final Time field, change the value to 1.000e-07
6. In the A parameter field, change the value to 1.0
7. In the t0 field, change the value to 1.000e-8
8. In the Alpha field, change the value to 1.000e+19
9. Click Plot Source. Note that the frequency domain content goes to about 1 GHz.
10. The results should be the same as below:



11. Click Finished
12. Click Ok in the EMA_Source_Tool popup window.
13. Open Tools->Toolbox->EMA3D_#10_DefineSources
14. Type **PLUS NA SOURCE C**
15. Observe the node number that will be the source at the center of the dipole



16. In this case, the node is 34671
17. Click ThwVol from the right menu (Thin Wire Voltage Source)
18. Enter gaussian.dat in the Source File Name field
19. Enter 34671 in the Single Node to be Loaded field as shown below:



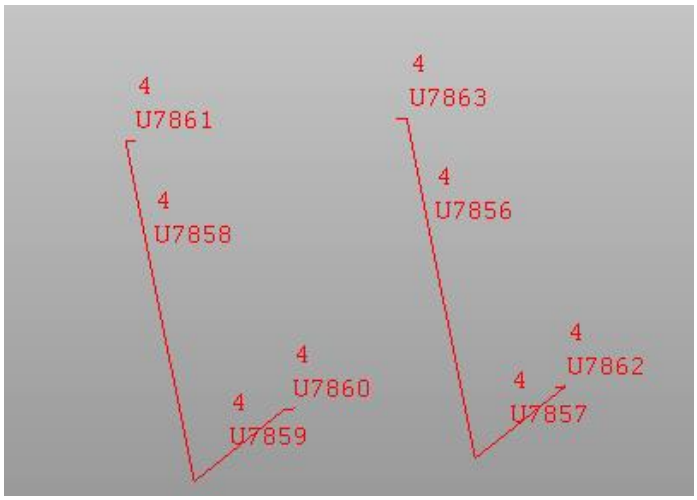
20. Click OK

Select Boundary Condition

1. Open Tools->Toolbox->EMA3D_#11_SelectBdyConds.
2. Select BOUND from the right menu.
3. Click the PML radio button in the Boundary Conditions popup window
4. Review the default values as shown below and click APPLY

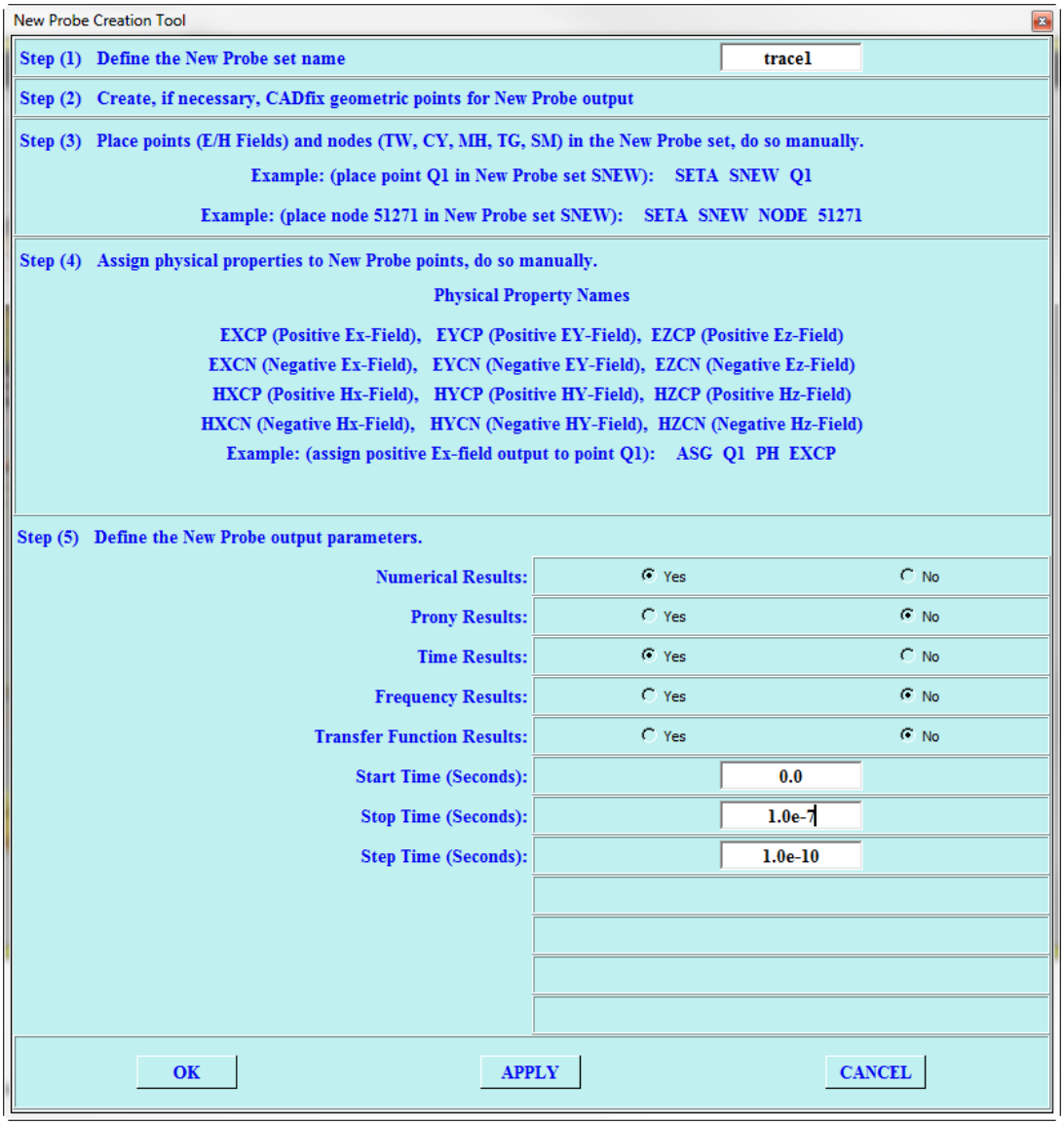
Define Trace Current Probe

1. Type **PLOT LA TRACES**
2. Observe the names of the lines plotted, such as U78567 and U7859.



3. Type **SETA TRACE1 N U7857** in the command window, where U7857 is the name of one of the lines echoed above.
4. Type **SETA TRACE2 N U7859** in the command window, where U7859 is the name of one of the other lines echoed above.
5. Open Tools->Toolbox->EMA3D_#12_DefineProbes.
6. Select New from the right menu.
7. Input TRACE1 in the New Probe Set Name field under Step (1)
8. Input 1e-6 in the Stop Time field under Step (5)
9. Input 1e-10 in the Step Time field under Step (5), as shown below

10. Click APPLY
11. Input TRACE2 in the New Probe Set Name field under Step (1)
12. Input 1e-6 in the Stop Time field under Step (5)
13. Input 1e-10 in the Step Time field under Step (5)
14. Click APPLY
15. Now, create two points to associate with an electric field probe inside and outside of the box. In the command window, type
    PNT FIPR1 0 100 50
    PNT FIPR2 0 100 -100
    SETA FIELDPR1 FIPR1
    SETA FIELDPR2 FIPR2
    ASG FIPR1 PH EXCP
    ASG FIPR2 PH EXCP
16. Return to the New Probe #2 dialog box.
17. Type FIELDPR1 in the New Probe Set Name field (1)



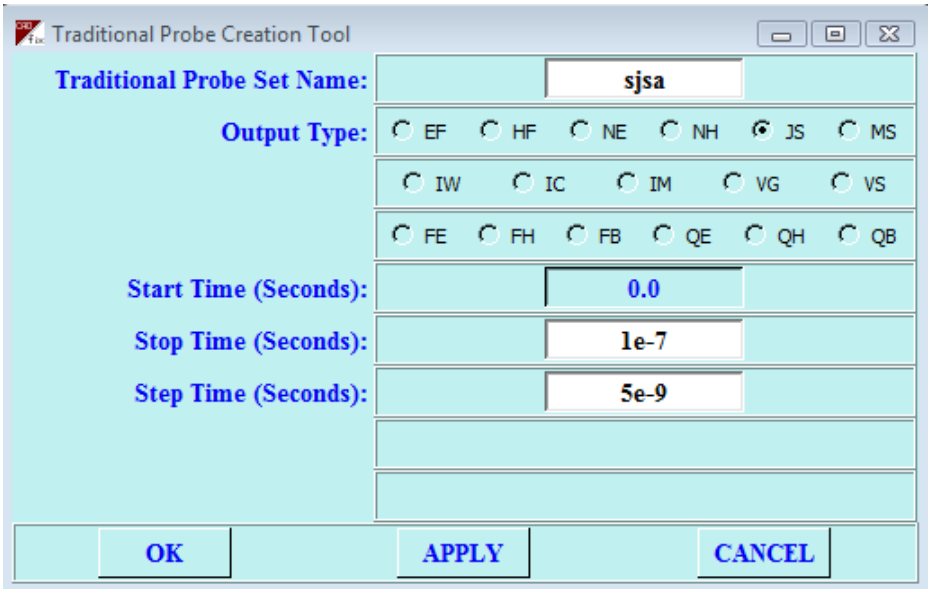
18. Retain a Start Time of 0.0, a Stop Time of 1e-6 and a Step Time of 1e-10
19. Click APPLY
20. Change the New Probe set name to FIELDPR2
21. Click OK

Define Surface Current Probes for Animation

1. Type the following into the command line:

   RNAM SJSA
   RNAM STP1
   SETA SJSA S BOX
   COMP SJSA GN
   SETR SJSA S BOX
   SETA STP1 L BOX
   COMP STP1 GN
   SETR SJSA STP1

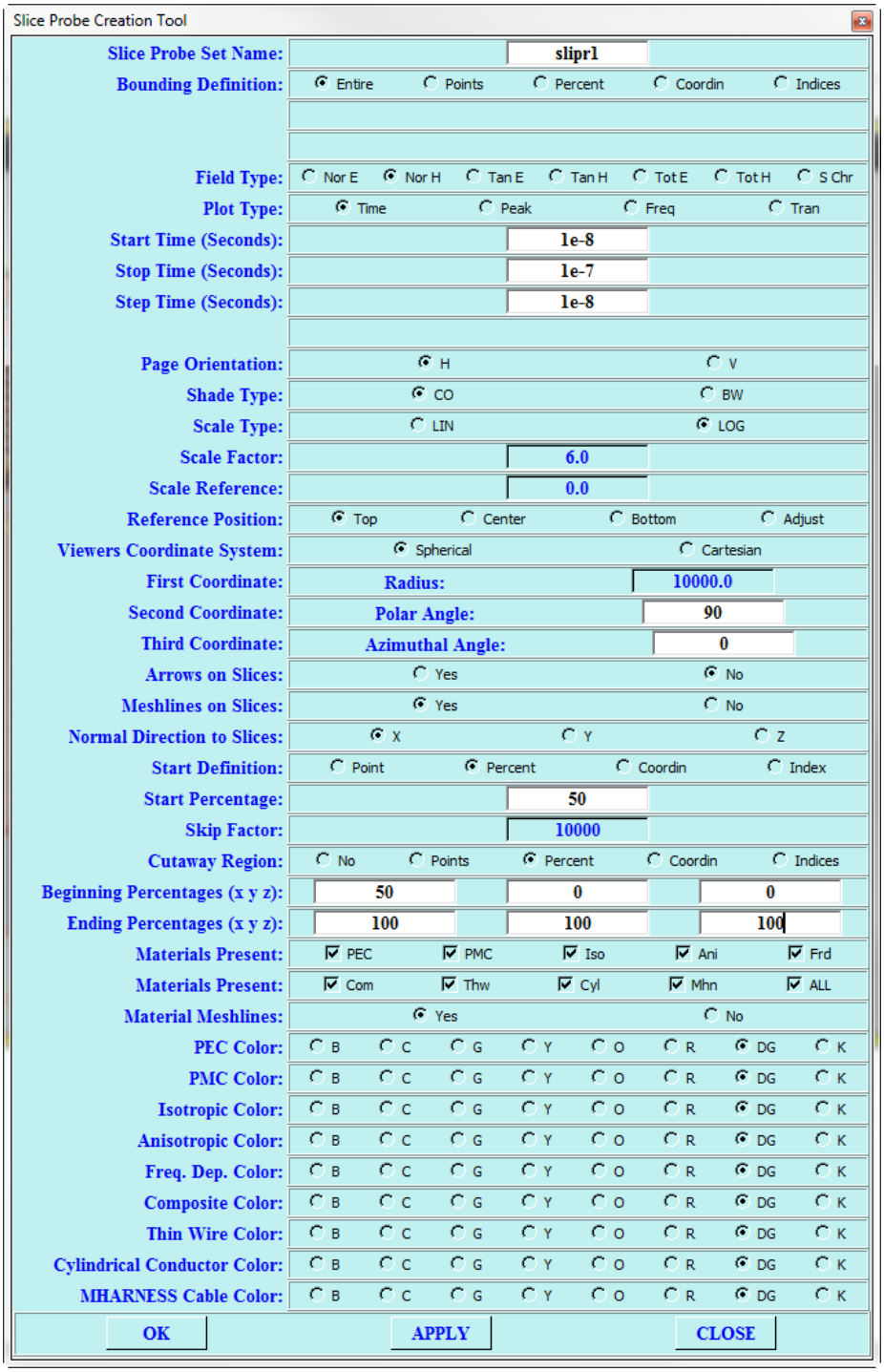
2. Type **PLOT TWI SJSA** into the command line and observe the box twinkles that will be used for surface current density animation.
3. Select Traditional from the right menu.
4. Type SJSA in the Set Name field.
5. Select JS in the Output type field.
6. Type 1E-7 in the Stop time field.
7. Type 5E-9 in the Step time field, as shown in the figure below:



8. Click OK

Define Slice Probe

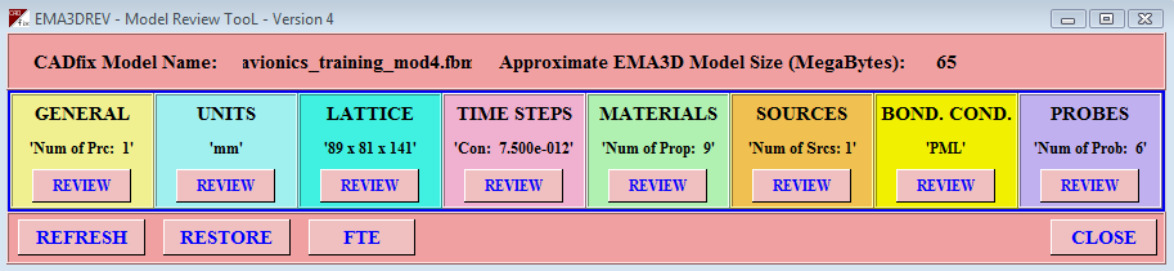
1. Click Slice in the right hand menu
2. Type SLIPR1 in the Set Name field.
3. Select Tan E in the Field Type field.
4. Select Time in the Plot Type field.
5. Type 1e-8 in the Start time field.
6. Type 1e-7 in the Stop time field
7. Type 1e-8 in the Step time field.
8. Type 90 in the Polar Angle field.
9. Type 0 in the Azimuthal Angle field.
10. Select X for the Normal Direction to Slices field.
11. Select Percent in the Start Definition field.
12. Type 50 for the Start Percentage field.
13. Select Percent in the Cutaway Region field.
14. Type 50 0 0 for the Starting Percentages (x y z) field.
15. Type 100 100 100 for the Ending Percentages (x y z) field.
16. The information in the Slice Probe window should look as follows:



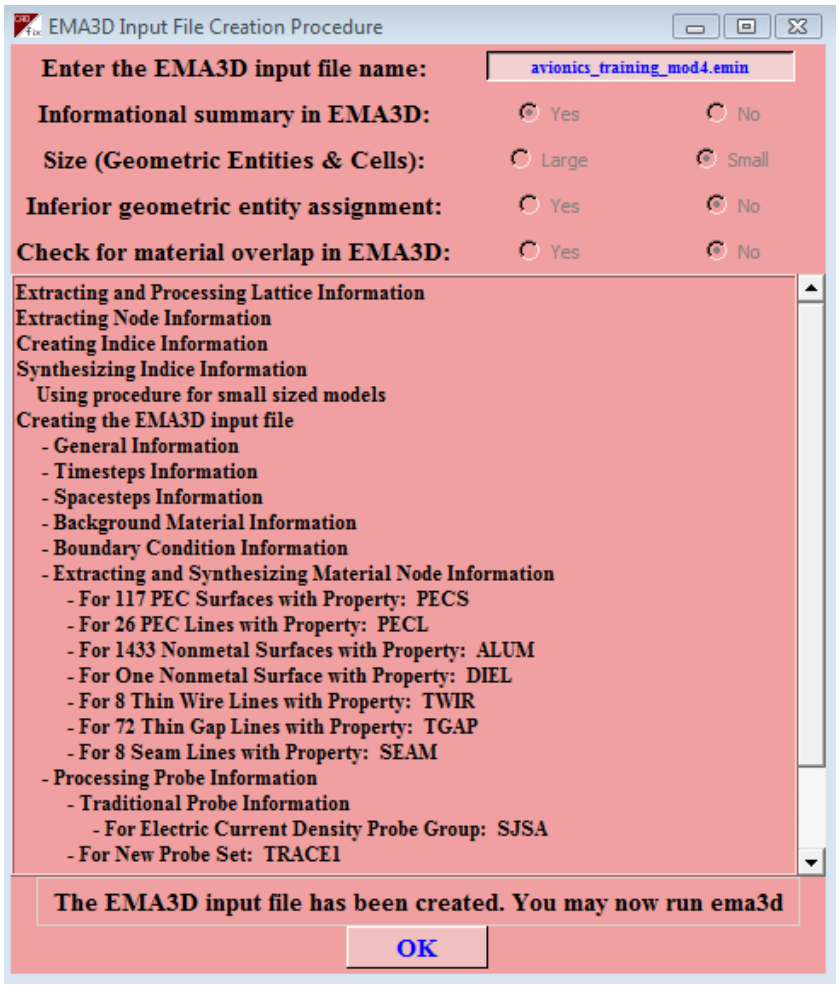
17. Click OK
18. Save the model as avionics_training_mod4.fbm

Prepare the 3D Simulation Input File

1. Open Tools->Toolbox->EMA3D_ReviewTool.
2. Select Review from the right menu.
3. The Model Review Tool should contain the same elements as shown below:



4. Review the simulation details, and click FTE when ready to proceed.
5. Note the file name and click OK in the popup window.

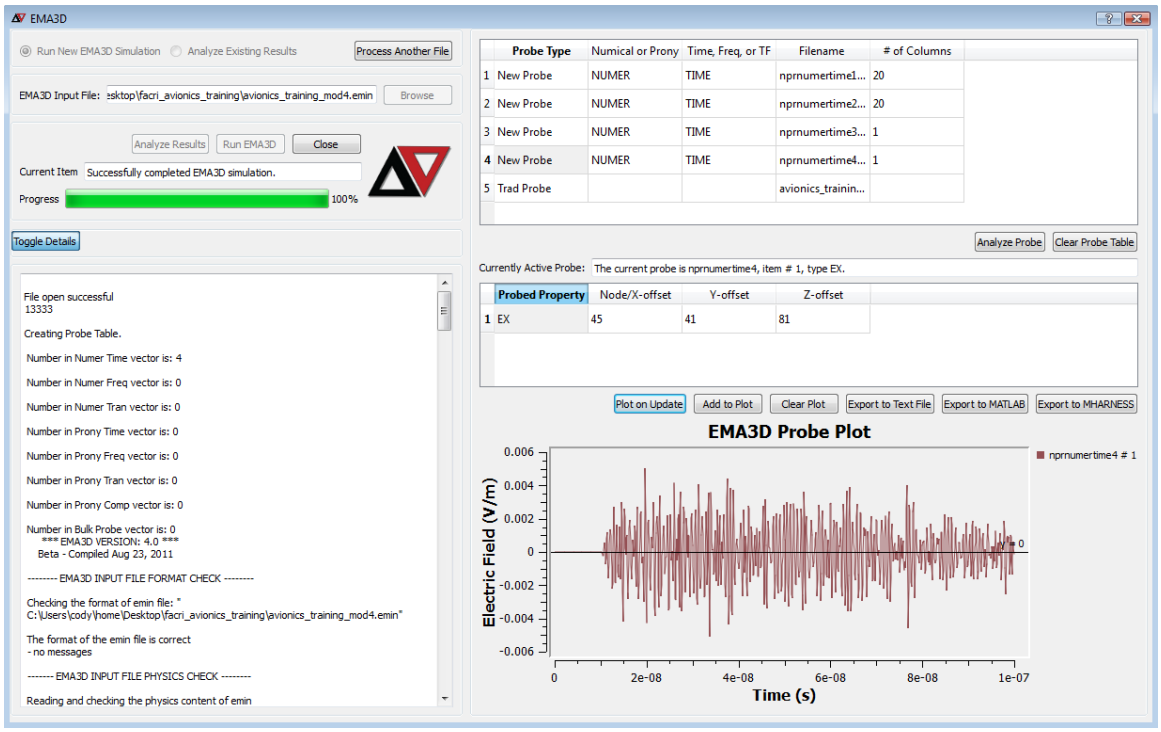


6. Click OK in the EMA3D Input File Creation Procedure window.
7. Click Close in the EMA3DREV window

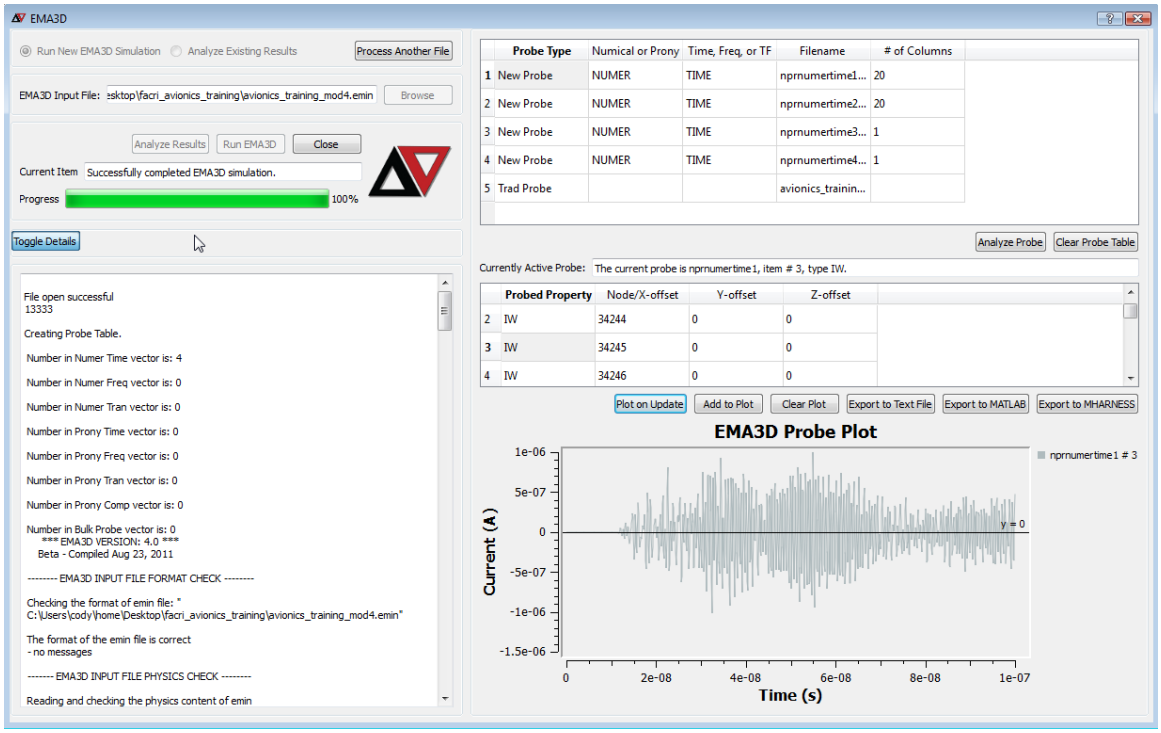
Run EMA3D Simulation

1. Open Tools->Toolbox->EMA3D_Execute
2. Select EMA3D_exe 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 FTE step above.

6. Click Open
7. Click Run EMA3D
8. Click the Toggle Details button to observe the command prompt sequence produced by EMA3D.
9. Wait for the simulation to complete. Observe the progress of the plot by clicking on the probe table items. The window is shown below:



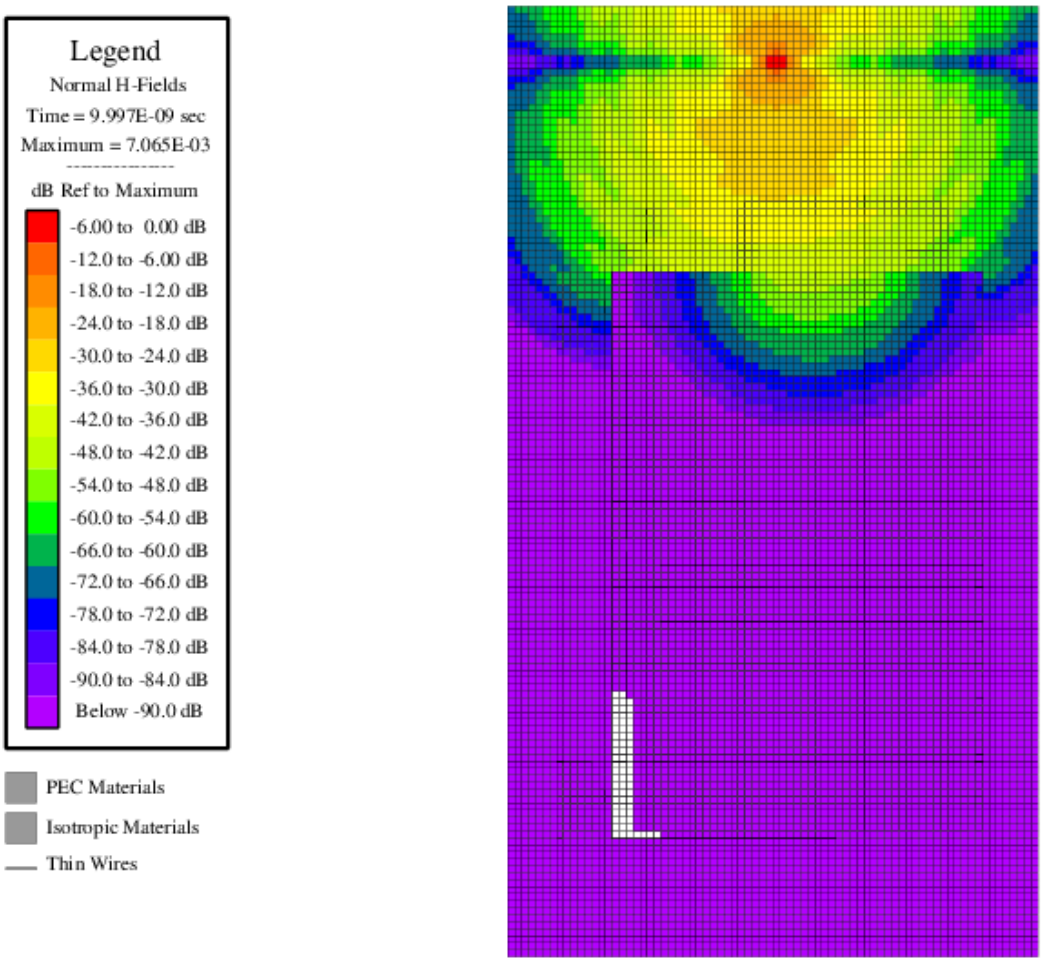
10. Highlight 4 New Probe from the list at the top right
11. Click Analyze Probe.
12. Click Plot on Update.
13. Observe the noisy electric field inside of the box.
14. Double click 1 New Probe
15. Highlight 3 IW from the Active Probe Table.
16. Click Plot on Update
17. The cable current as a function of time is displayed in the EMA3D Probe Plot, as shown below.



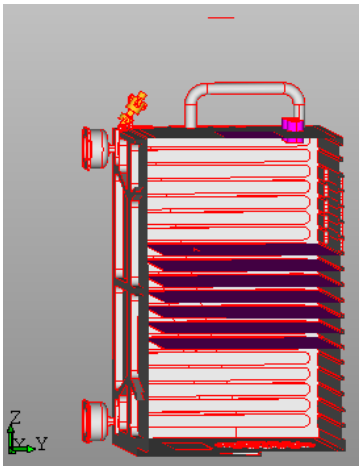
18. Investigate the other probes using the same procedure.

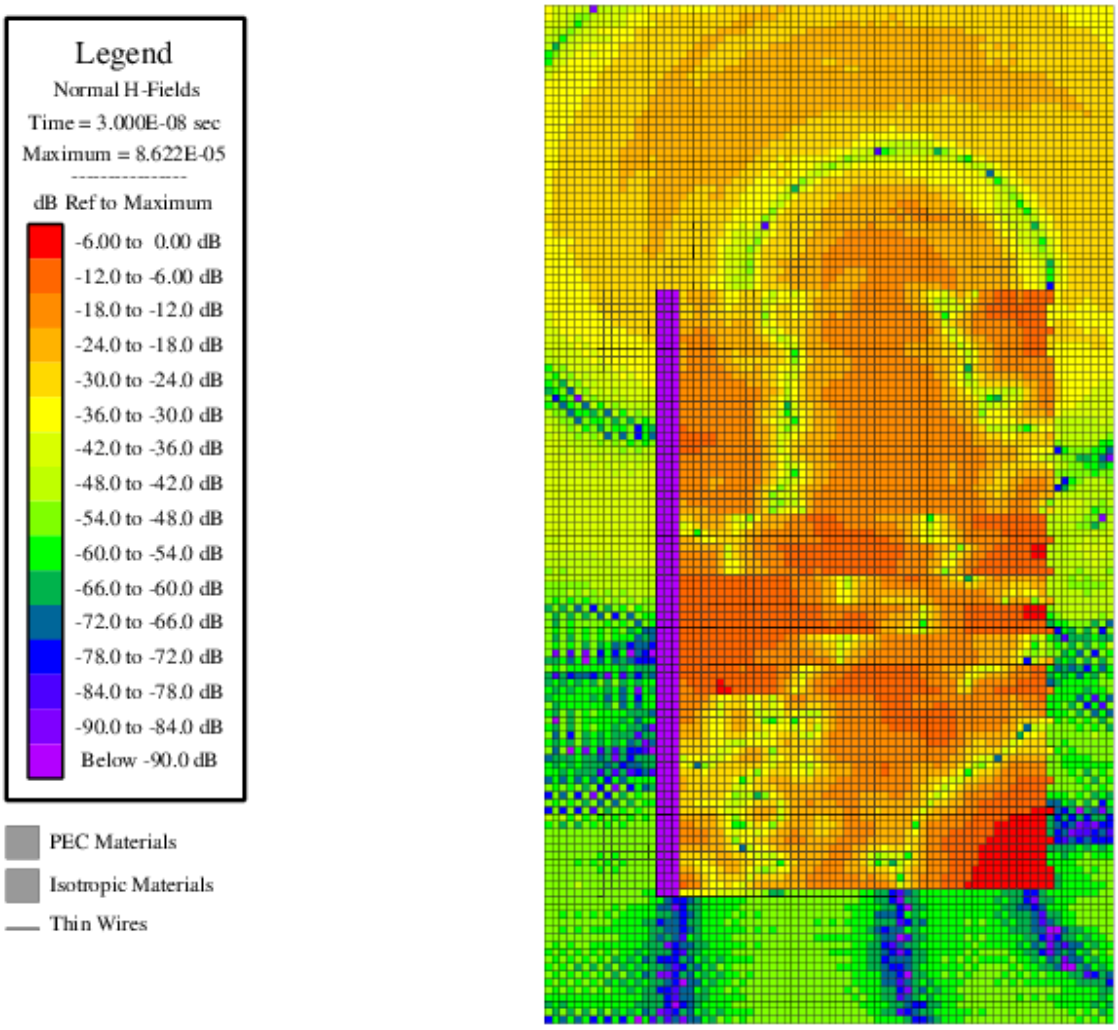
Open a Slice Probe to View Fields Inside of the Box

1. Use a file browser to navigate to the folder where the simulation was run
2. Open the post script file “avionics_training_mod4spr1-timepict000001.ps” with a post script viewer like GSiew, which is the graphical interface for the postscript interpreter Ghostscript, or RoPS Postscript Viewer
3. Observe that at 1e-8 s the source waveform has not yet fully spread throughout the problem space.



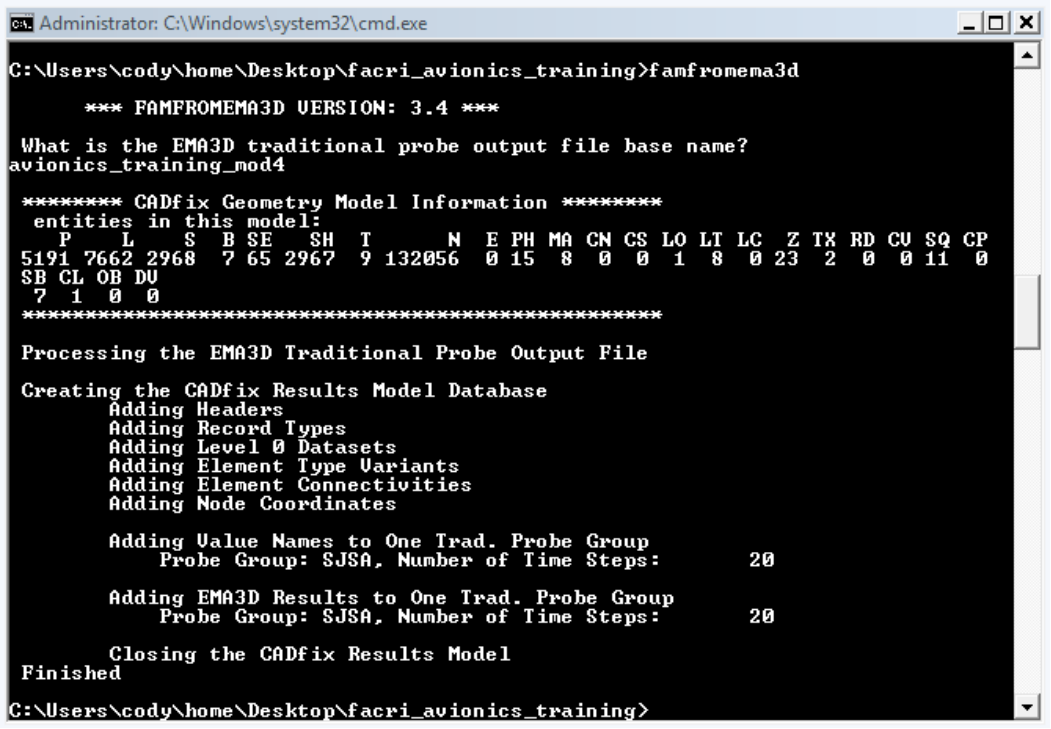
4. Open the post script file “avionics_training_mod4spr1-timepict000003.ps”
5. Notice that by 20 ns, the electric fields inside of the metal box are larger than the ones on the outside
6. The slice probe is approximately displaying the angle and orientation of the box as shown below. The image is using clipping planes to visualize the cards in the box interior.



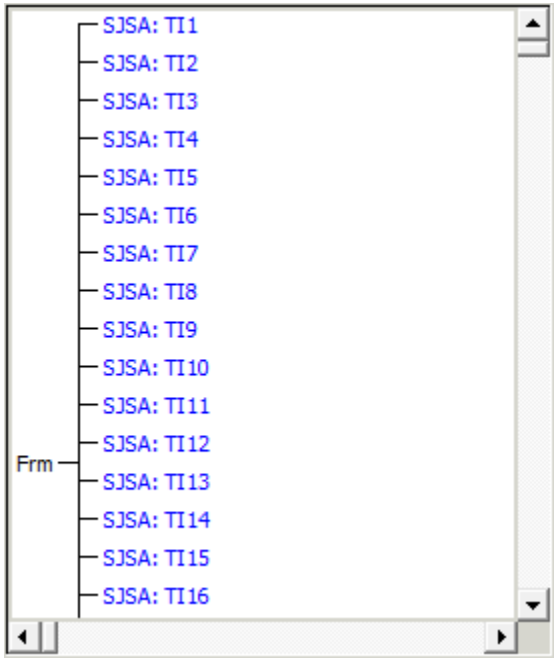


Create Surface Current Density Animation

1. Save the CADfix database.
2. Close CADfix
3. Open Windows Start menu and click run. On Windows 7, type directly in the Start Menu search box.
4. Type cmd.com to open a command window.
5. Type cd /d <folder location>, where <folder location> refers to the working directory from above.
6. Type famfromema3d into the command window.
7. Type the name of the .emin file, such as avionics_training_mod4 and press enter.
8. An example of the command line execution is below:



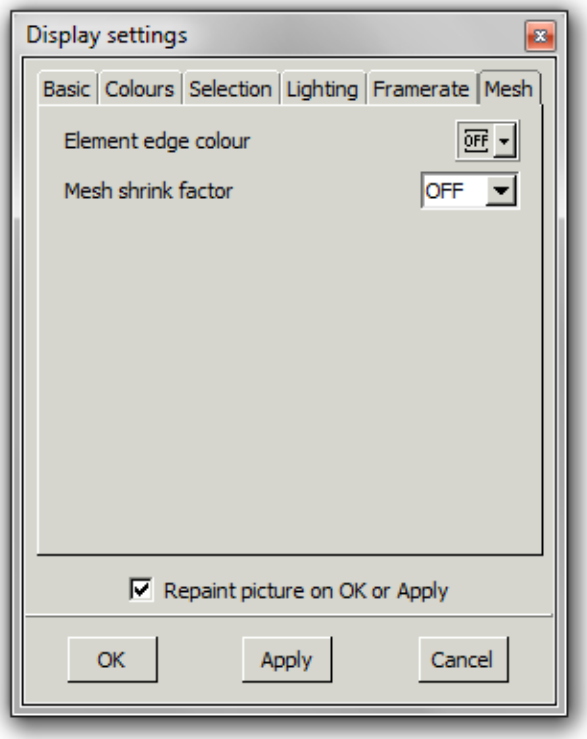
9. Open the CADfix database file saved above, such as avionics_training_mod4.fbm
10. Click the post-processing tab on the left side of the CADfix window 
11. Wait on the database to load.
12. Check to ensure the traditional probe of set SJSA is visible for each time point, as shown below:



13. Type CHEC to make sure that the results database matches the model.
14. Observe that the Result and model databases are consistent as shown below:

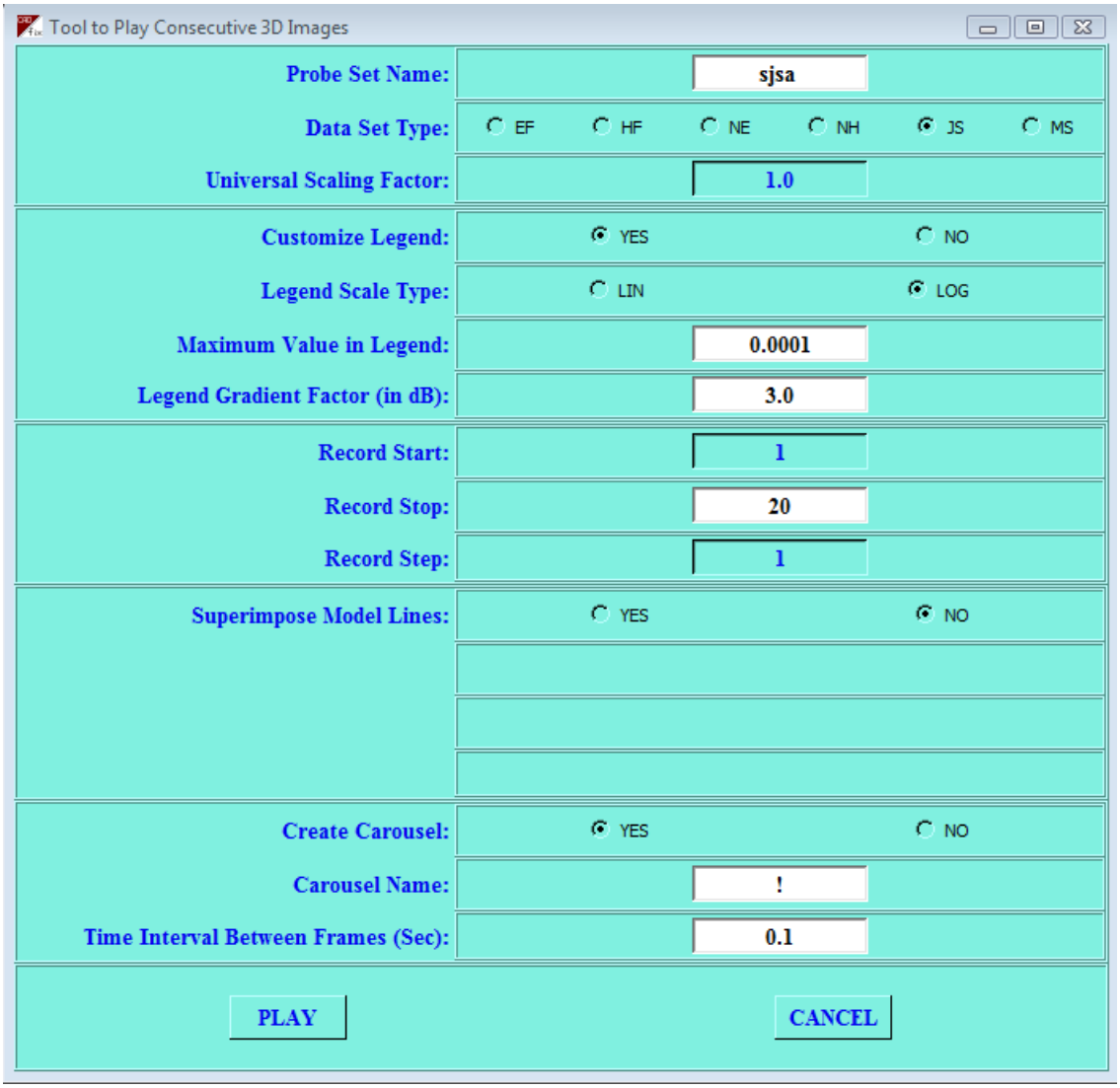
```
 50000 nodes compared; processing continues
 60000 nodes compared; processing continues
 70000 nodes compared; processing continues
 80000 nodes compared; processing continues
 90000 nodes compared; processing continues
 100000 nodes compared; processing continues
 132056 nodes compared.
 Result and model databases are consistent
 132056 nodes have coordinates coincident to within
1.0000E-02
  Maximum node coordinate difference:   0.000000
 Fr>
```

15. Type RNAM RD01 into the command window.
16. Type RNAM RD02 into the command window.
17. Open View->Display Settings and open the Mesh tab
18. Change the Element edge colour to off
19. Change the Mesh shrink factor to off



20. Click OK
21. Open Tools->Toolbox->EMA3D_PostProcessing.
22. Select “Animation” from the right menu.
23. In the “Probe Set Name” field, enter “SJSA”
24. In the “Data Set Type” field, select the “JS” radio button.
25. Click the “Yes” radio button in the “Customize Legend” field.
26. Set the “Maximum Value in Legend” to “0.0001”
27. Set the “Legend Gradient Factor (in dB)” to “3.0”
28. Set the “Record Stop” to “20”

29. Click the “Yes” radio button in the “Create Carousel” field.
30. In the “Carousel Name” field, type “!”
31. In the “Time Interval Between Frames (Sec):”, type “0.1”
32. Leave the other values at the default, as shown below:



33. Click Play
34. Type **PLAY C1** to run the animation

Plot the Simulation Data using Spyder

1. If the WinPython program is not already on the computer you are using, download it from the following website for free: http://sourceforge.net/projects/winpython/files/
2. The 2.7 version of WinPython seems to be more stable at this time and that is the version recommended for installation.
3. Navigate to the WinPython folder installed on your machine: C:\Program Files (x86)\WinPython-64bit-2.7.6.4

4. Open the Spyder.exe program
5. Open the supplied file: plot_avionics_training_data.py
6. Click the run file button at the top of the window

7. Observe the results data plots