



Shielding Effectiveness of a Box

**EMA3D Demo for Creating a Basic HIRF Problem in a
Box**

*Prepared by Casey Peirano, Kellan Kremer,
and Jason Stanko*

January 2020

CONTENTS


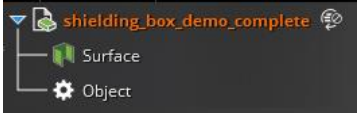


Contents	i
Problem Overview	1
Exploring the Discovery Interface	1
Constructing the model geometry	2
Defining the Simulation Domain.....	5
Defining the Plane Wave Source	6
Defining and Assigning Material Properties.....	7
Creating the Field Probe	8
Meshing the Model	9
Running the Integrated Simulation	10
Analyzing Results.....	11

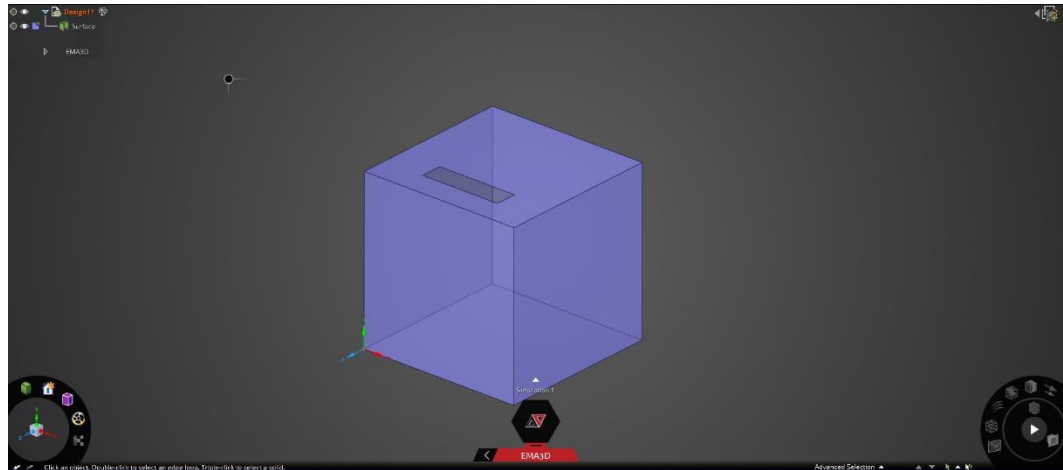
Problem Overview

This tutorial will demonstrate the EMA3D workflow for a high intensity radiated field (HIRF) simulation in Discovery. This workflow includes setting up the problem space, assigning properties, defining probes, and basic post-processing.



This guide begins with a brief exploration of the Discovery interface, followed by instructions for setting up the geometry of the problem.

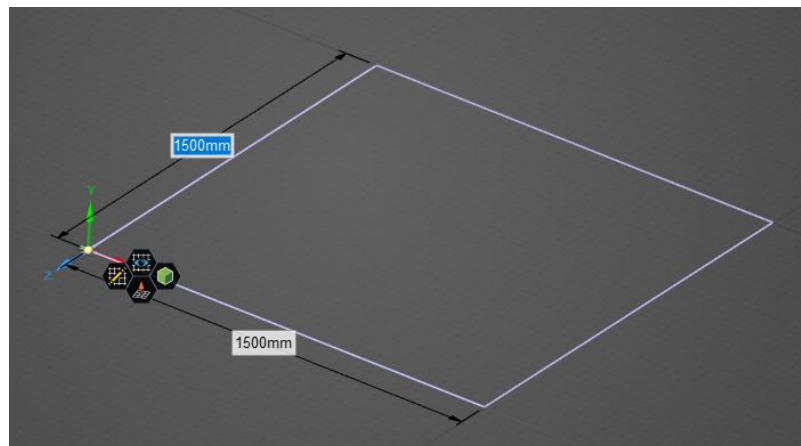
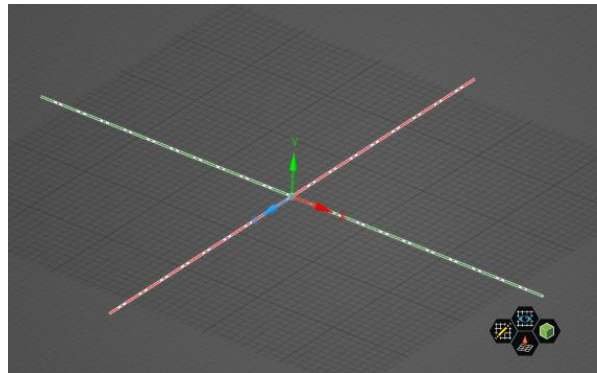
Exploring the Discovery Interface



1. To start, navigate to the **EMA3D**  stage tab at the bottom of the screen.
2. Individual components of the model can be viewed in the **Structure Tree**. To better visualize components of the model, components of the model may be hidden. Clicking the eye next to the component name hides it from view. 
3. 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.
4. The **Simulation tree** can be found after navigating to the **EMA3D**  stage tab at the bottom of the screen (users may need to collapse 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.

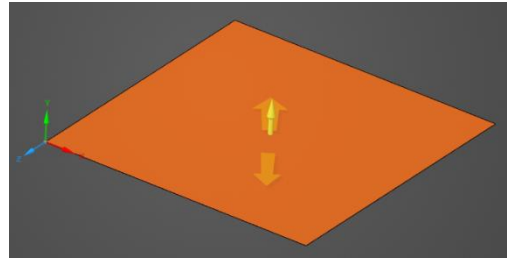



Constructing the model geometry

1. At the top of the Discovery window, select the design tab in the ribbon to open the design tools.
2. Click Sketch , then click New Plane in the model window. Finally select the positive Y axis at the origin to set the plane. Rotate the view so that it looks like the figure at right with Positive Y axis pointed up.
3. In the Sketch section of the design tools ribbon, select rectangle , then click the center of the grid, where all three directional arrows meet, in the model window (a green dot will appear when you hover over this point).
4. Drag the cursor diagonally toward the upper right of the window (in the XZ plane). One of the dimension boxes will highlight blue. Type 1500 then press Tab and enter 1500 in the other dimension box. Press Enter.

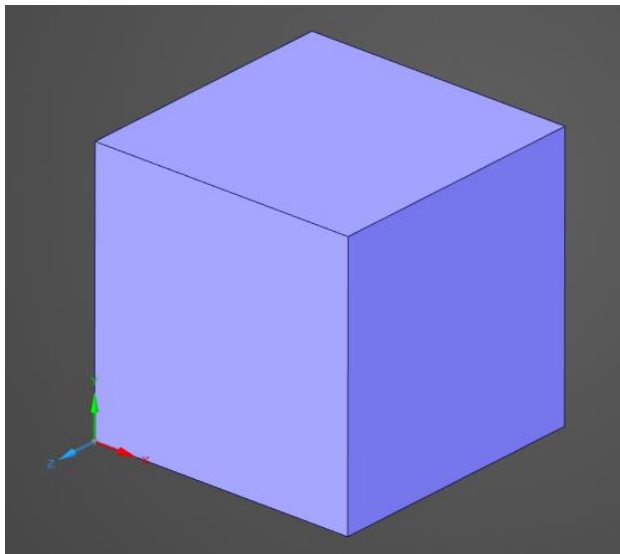





5. Zoom out to view the entire square and hit escape.
6. Select Pull  in the ribbon and select the square in the model window. It will turn orange and a menu will pop up next to the cursor. Select the Ruler  button in that menu, select the square again, type 1500, and press enter. This will form the square into a solid cube. Hit escape.




Alternatively, you can use the pull  tool to drag the square upward until the dimension box reads 1500.

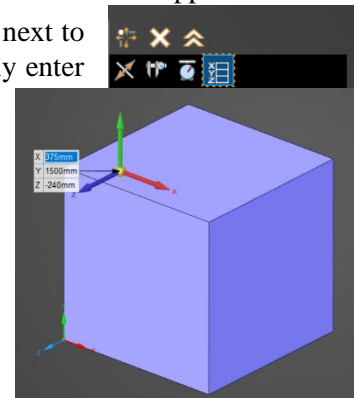
Now we are going to add an open slit on the top of the cube.





7. Select the point  tool from the Sketch area and click on the top of the cube. The drawing grid will move to the top surface. Click again anywhere on the top face of the cube to draw a point. Press D to return to 3D mode.
8. Select the move  tool and click on the point.
9. In the bottom left of the SpaceClaim window, an Options menu will appear. Click on the coordinates  button. A popup menu will appear next to the point in the model window where you can manually enter the coordinates to move the point. Enter:
 - a. **X:** 375 mm
 - b. **Y:** 1500 mm (unchanged)
 - c. **Z:** -240 mm

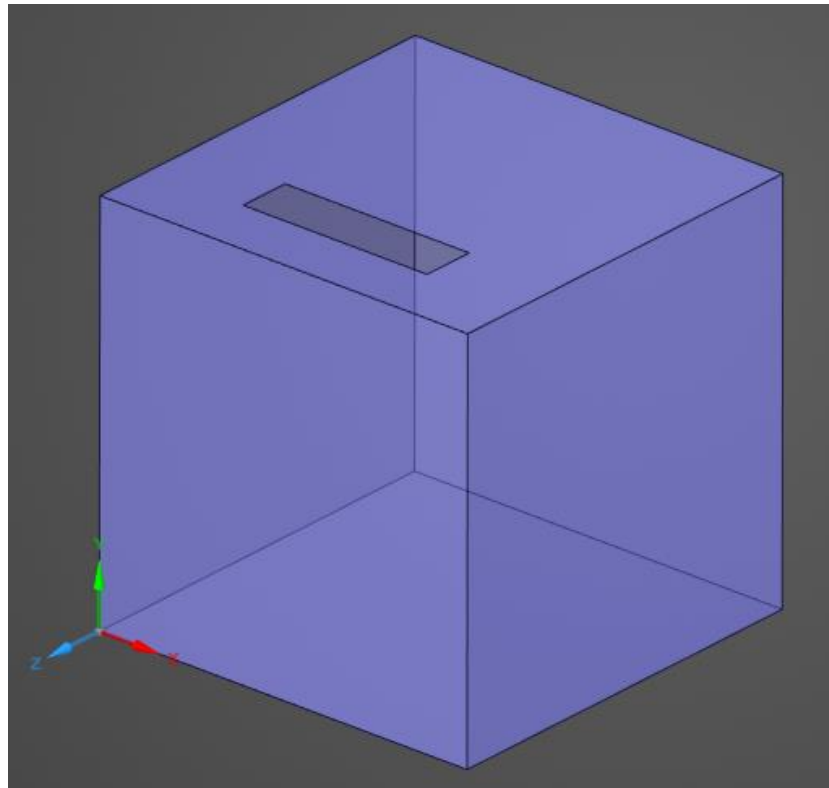
Press enter and then escape when finished.

10. Using the rectangle  tool, click on the point and drag the cursor toward the top right corner of the grid. Enter 750 in the highlighted dimension box and use Tab to



enter 200 in the other such that the long side is along the front face of the cube (see figure below).


11. Click the 3D mode  button at the bottom of the model window (or press the D key) then click select  in the ribbon.
12. Right click on the new, small rectangle and click delete in the pop-up menu. Notice the difference in transparency indicating that the solid cube has become a surface.
13. Right click on the point in the corner of the cutout and select delete from the popup menu to remove it. Hit escape.



Defining the Simulation Domain

To begin, load the file “shielding_box_demo_cad_blank.dsc0” into Discovery.

The first step in preparing for simulation will be to define the computational domain.


1. Click **Domain**  under **EMA3D** in the **Ribbon**.
2. Six directional handles will appear.
3. Under **Time** in the **Properties Panel** adjust the lowest and highest frequency values to be:
 - a. **Lowest Frequency:** 10 MHz (if no units are specified, Hz is default value. Units are case sensitive; if capitalization is incorrect the default unit will be used.)
 - b. **Highest Frequency:** 500 MHz
4. Drag the handle pointing in the -x direction to change the **Minimum X** value to **-420**. To see the value, click the arrow to the left of **Minimum** under **Lattice** in the **Properties Panel**.
5. The lattice parameters can also be manually adjusted in the **Properties Panel**.
6. Under **Lattice** in the **Properties Panel** adjust the **Minimum** values to be:
 - a. **X:** -420 mm
 - b. **Y:** -420 mm
 - c. **Z:** -1920 mm
7. Under **Lattice** in the **Properties Panel** adjust the **Maximum** values to be:
 - a. **X:** 1920 mm (if no units are specified, mm is default value)
 - b. **Y:** 1920 mm
 - c. **Z:** 420 mm



The screenshot shows the software's Properties Panel with the following sections and values:


- TIME**
 - Stability Margin: 15 %
 - Start: 0 s
 - Step: 1E-10 s
 - End: 1E-07 s
 - Lowest Frequency: 10000000 Hz
 - Highest Frequency: 500000000 Hz
 - Increment Method: Constant
 - Magnetostatic Steps: ...
- LATTICE**
 - MINIMUM**
 - X: -420 mm
 - Y: -420 mm
 - Z: -1920 mm
 - MAXIMUM**
 - X: 1920 mm
 - Y: 1920 mm
 - Z: 420 mm
 - STEP SIZE**
 - X: 60 mm
 - Y: 60 mm
 - Z: 60 mm
 - DIVISIONS**
 - X: 1
 - Y: 1
 - Z: 1
 - RENDERING**
 - BOUNDARY**
 - BACKGROUND**

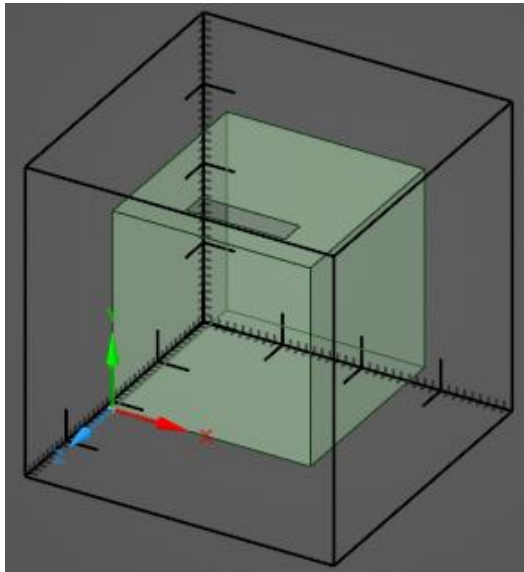
NOTE: The spatial step size, temporal step size, start time, and end time, will be automatically calculated based on the specified minimum and maximum frequencies. In the future, these items will be manually adjustable by toggling “advanced options.”

8. Click **Ok**  to complete the domain setup. Upon completion of the domain parameters the cube is fully encapsulated by the lattice. Press escape.

Defining the Plane Wave Source

Next, the HIRF plane wave source will need to be defined.

1. Click **Plane Wave**  under **EMA3D** in the **Ribbon**.

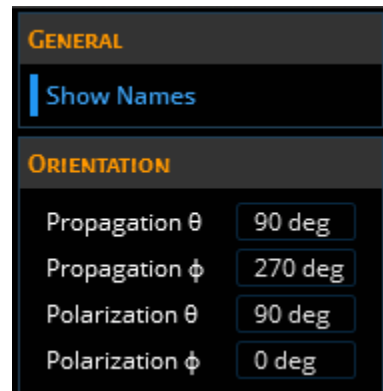


2. In the **Properties Panel** enter the following values (a definition of each variable is available at the bottom of the **Properties Panel** when the variable is selected):
 - i. **Propagation $\Theta = 90$**
 - ii. **Propagation $\Phi = 270$**
 - iii. **Polarization $\Theta = 90$**
 - iv. **Polarization $\Phi = 0$**

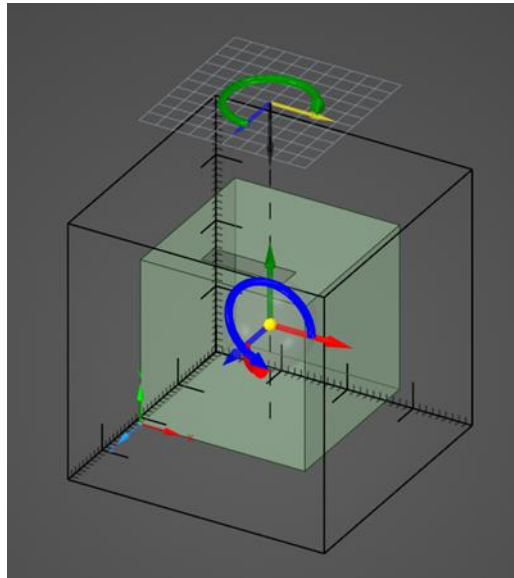
At this time, it is possible that some variables may change values after entering another variable to ensure orthogonality. If this occurs, simply reenter the values.

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 +**X** direction, **Propagation Vector** (K, straight black arrow) in the -**Y** direction and **Magnetic Field Vector** (H, straight blue arrow) in the +**Z** direction.






4. Click **Ok**  to complete the plane wave setup.

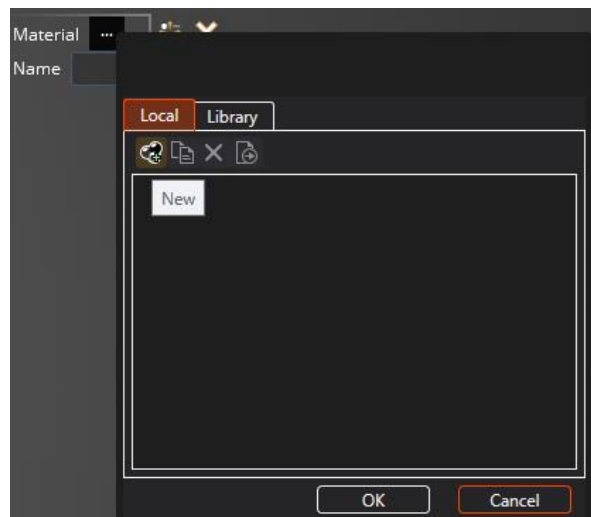



Defining and Assigning Material Properties

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

The geometry in the model will need properties assigned to it in order to simulate the problem. The entire model is composed of surfaces.

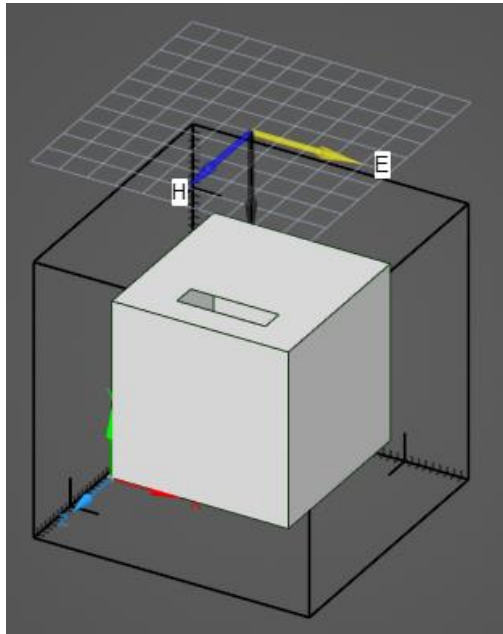
1. Click **Materials**  under **EMA3D** in the **Ribbon**.
2. Click **Isotropic**  in the dropdown list.
3. In the **Properties Panel** select the ellipsis next to **Material**. A new window will appear.
4. In the popup window, under the **Local** tab, select **New** .
5. Double click to select the first definition in the list “**Isotropic 1**.” To view its properties, click the arrow to the left of **Properties** in the **Properties Panel**.
6. To the right of the **Name** parameter, double click “**Isotropic 1**” to rename the material. Rename the material **Aluminum** and press the enter button to rename the definition.



7. Change the color to gray by selecting **Color** and then selecting the dropdown menu arrow.
8. Change the **Electric Conductivity** (σ_e) to **1.2e7**.
9. Click on the **Select surface tool**  in the model window to restrict our definition assignments to surfaces.
10. In the **Structure Tree** select **Surface** to assign the **Aluminum** material properties to all **Surface** structures.

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


11. The cube should now be recolored in the model window. Click the check box on the **Isotropic Node** under **Materials** in the **Simulation Tree** to hide the color and revert to the original, more transparent view.



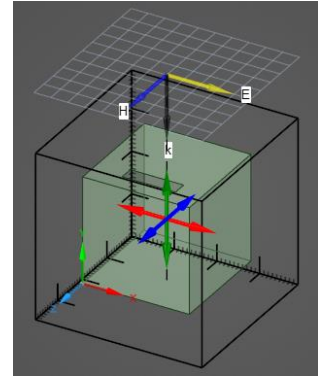
12. Click **Ok** when finished assigning materials.


Creating the Field Probe

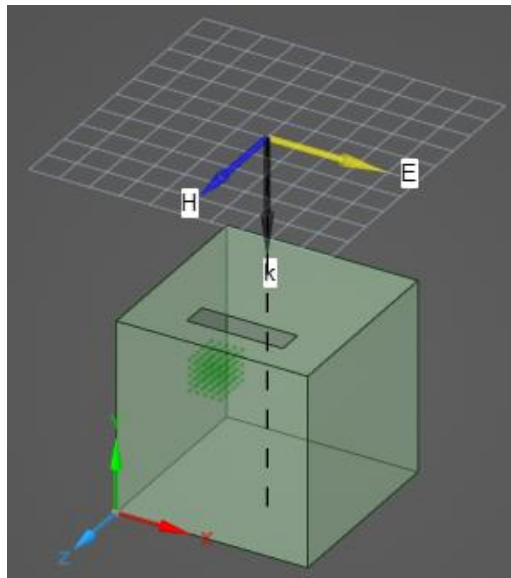
Next, we'll create the electric field probe that will capture the fields inside the cube.

1. Click **Field Probe**  under **EMA3D** in the **Ribbon**.

2. Click on **Boxed Region** in the dropdown list. The default probe type, which we are using in this demo, is electric. The probe can be changed to magnetic in the **Properties Panel**.
3. Several arrows and a small box (the probe region) will appear at the center of the box. The size and position of the boxed region for the probe can be adjusted by dragging the red, blue, and green directional arrows. For this demo, we are going to specify the coordinates of the probe by entering values in the **Properties Panel**.
4. Under **Minimum** in the **Properties Panel** adjust the values to be:
 - a. **X**: 540mm
 - b. **Y**: 1080 mm
 - c. **Z**: -360 mm
5. Under **Maximum** in the **Properties Panel** adjust the values to be:
 - a. **X**: 840 mm (if no units are specified, mm is default value)
 - b. **Y**: 1380 mm
 - c. **Z**: -60 mm



6. Click **Ok**  to create the probe. Then hit escape.
7. The probe should now be visible in the model window and should be added to the **Simulation Tree** under the **Probes Node**.





Meshing the Model

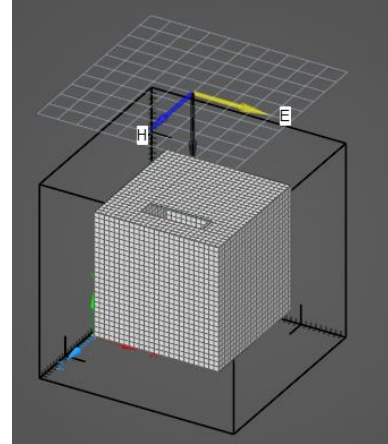
Next, we need to mesh the model geometry.

1. Click **Mesh**  under **EMA3D** in the **Ribbon**.

2. 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 Discovery window will track the meshing progress.

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).


3. 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. Users can also adjust the mesh transparency by changing the material color to have 4 numbers in the A-RGB code, where the first number is < 255 . Then make sure “Allow Transparent Mesh” is checked in the  Settings menu. To bring the mesh back into view after closing the mesh menu, select *Display Simulation Mesh* from the dropdown menu beneath **Mesh**  under **EMA3D** in the **Ribbon**.

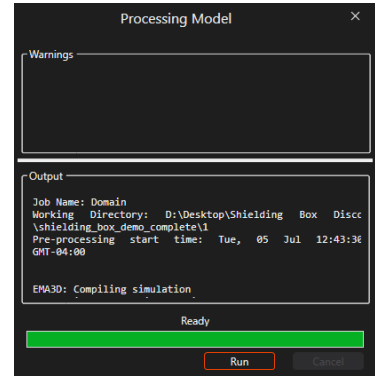


NOTE: If you would like to compare your model to a fully meshed, completed model, one is provided in `shielding_box_demo_complete.dsco`. Load the file into Discovery to see the model. You may also follow the next 2 sections to run the provided completed model, though the results are provided at the end of this document.

Running the Integrated Simulation

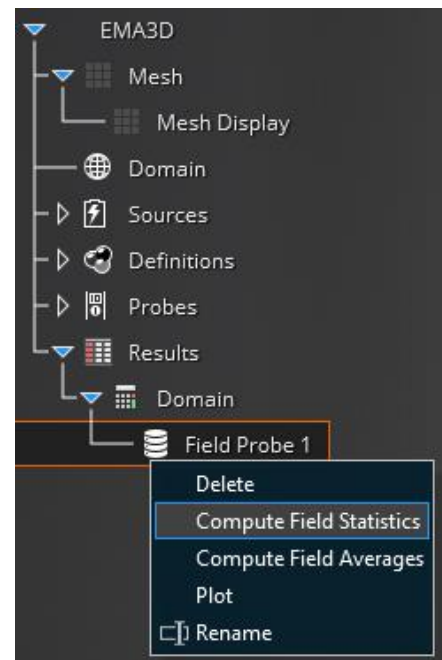
We are now prepared to run the simulation.

1. Click **Start**  in the **EMA3D** section of the **Ribbon**.
2. EMA3D will immediately begin pre-processing the simulation. Any errors and warnings will be logged in the pop-up window.
3. The EMA3d .emin file, and other relevant files, will be output in the current working directory at this time.
4. In the pop-up menu, click Run.
5. In 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.
6. Once completed, click the X in the upper right of the pop-up window to close the menu.

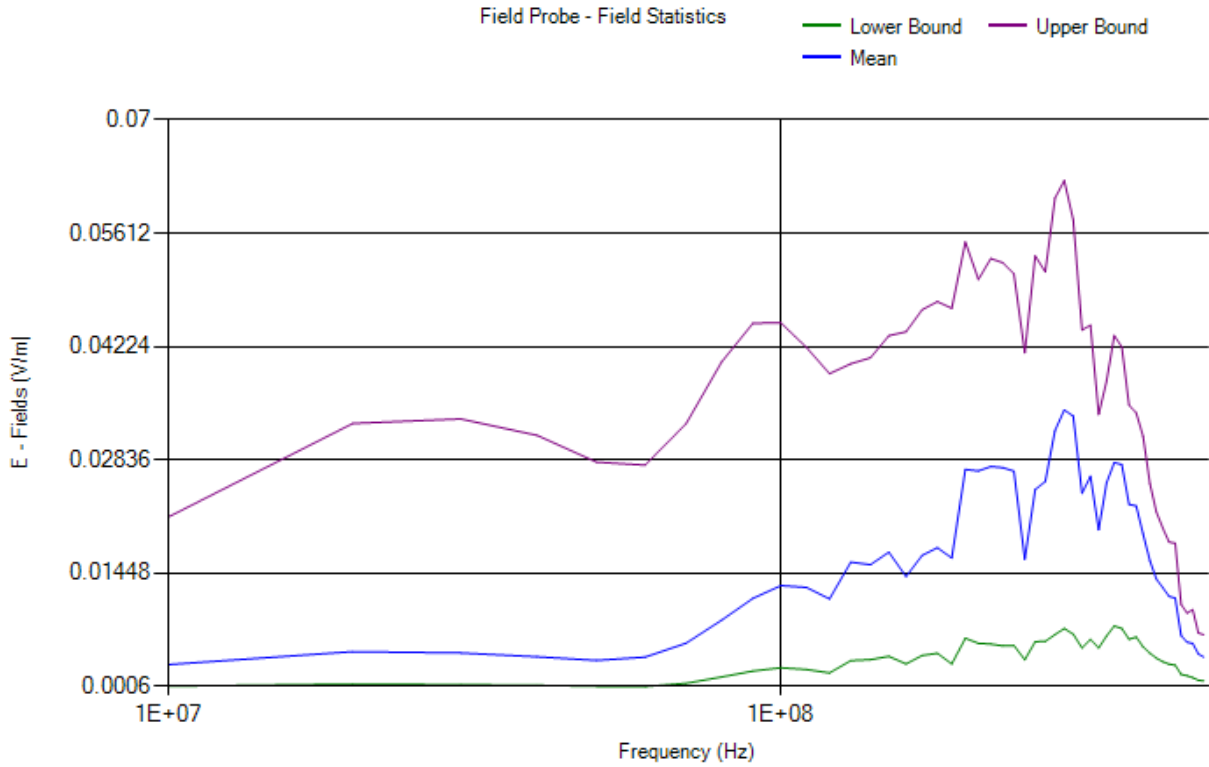


Analyzing Results

1. Expand the **Results Node** then the **EMA3D Simulation Node** in the **Simulation Tree**.
2. Right click on **Field Probe 1** and click **Compute Field Statistics** in the dropdown menu.
3. In the pop-up window, select `planewave.dat`, which will be used to compute the grid statistics.
4. Click the check boxes next to X, Y, and Z to compute the fields for all vectors.
5. Expand the **Visualization Node** then **2D Plots Node** in the **Simulation Tree**. There should be two new plots created called **Field Probe 1 - Field Statistics 1** and **Field Probe 1 - Shielding Effectiveness 1**.
6. Right click on **Field Probe - Field Statistics** and click **Show** in the dropdown menu.
7. A plot window will now appear with the E-Field Data.
8. Change plot options by clicking **configure** → **chart** in the dropdown menu.
9. In the pop-up window various chart elements can be modified.
10. Repeat steps 4-9 for **Field Probe - Shielding Effectiveness**.



NOTE: Right clicking on a series in the Legend or in the Plot Area Will allow you to configure individual series options.



NOTE: Frequencies outside the range of frequencies specified in the domain section should be interpreted with extreme caution, as the time and spatial steps were not calculated to properly capture frequencies outside of those specified.

