

F-16 18 GHz HIRF Training

EMA3D Primer for Creating a Basic HIRF Problem on an F-16 Cockpit

Prepared by Sean Rozowski

**December 2018**

# Contents

[Contents i](#_Toc531608169)

[Problem Overview 1](#_Toc531608170)

[Opening the File 1](#_Toc531608171)

[Viewing the Model 1](#_Toc531608172)

[Defining the Model Units 2](#_Toc531608173)

[Defining the Simulation Domain 2](#_Toc531608174)

[Meshing the Geometry 3](#_Toc531608175)

[Defining the Simulation Time Steps 3](#_Toc531608176)

[Defining Background Properties 3](#_Toc531608177)

[Defining Material Properties 3](#_Toc531608178)

[Assigning Material Properties 4](#_Toc531608179)

[Defining the Plane Wave Source 5](#_Toc531608180)

[Defining Boundary Conditions 5](#_Toc531608181)

[Creating Probes in EMA3D 5](#_Toc531608182)

[Creating the EMA3D Input File 6](#_Toc531608183)

[Running the Integrated Simulation 6](#_Toc531608184)

[Run EMA3D Simulation in Parallel 6](#_Toc531608185)

[Analyzing Results 8](#_Toc531608186)

[Python Analysis 8](#_Toc531608187)

[MATLAB Analysis 8](#_Toc531608188)

## Problem Overview

This tutorial will introduce the user to the EMA3D workflow for a HIRF simulation. This workflow includes setting up the problem space, assigning properties and defining probes. This training module assumes that the user has a basic understanding of the CADfix and EMA3D GUI. It is recommended to see other trainings as well to help develop stronger understanding of CADfix and EMA3D.

A pre-simplified F-16 cockpit model will be used to simulate this problem. The F-16 cockpit will be exposed to a plane wave of a gaussian pulse to simulate multiple frequencies at once. This will allow the canopy shielding effectiveness to be measured.

## Opening the File

1. Under the Windows start menu launch **CADfix** **for** **EMA3D**
2. Click on the **Open Model** button.
3. Select **CADfix GDX** as the **File type**
4. Import the file titled **“F-16\_18GHz\_Demo.gdx.”**

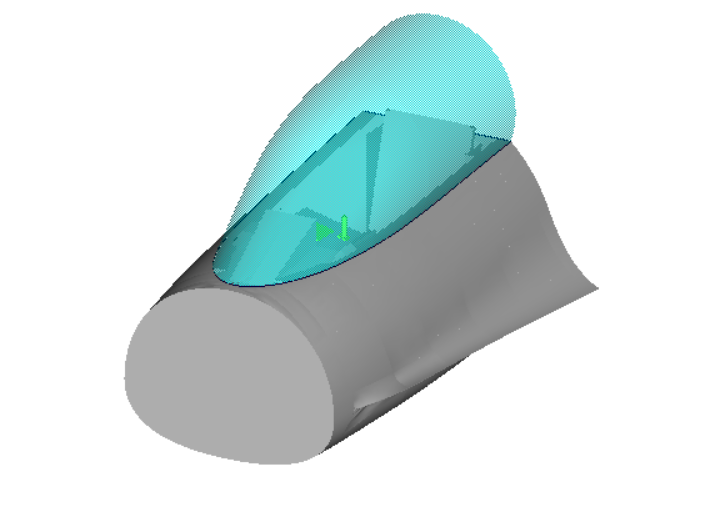
## Viewing the Model

The imported model will automatically plot the line connectivity. To see the entire model the surfaces will need to be plotted.

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

**PLOT SI ALL ASGD**

**PLOC SHADE BOTH**

**REP**

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

The model is currently in an assembly and needs to be broken up into CADfix sets.

1. Enter the following into the command line to create the needed sets:

**SETA SKIN Z2**

**SETA CANOPY Z4**

**SETA BOXES Z6**

**SETA SEAM Z8**

**SETA PINS Z10**

**SETA GRID1 Z12**

**SETA GRID2 Z14**

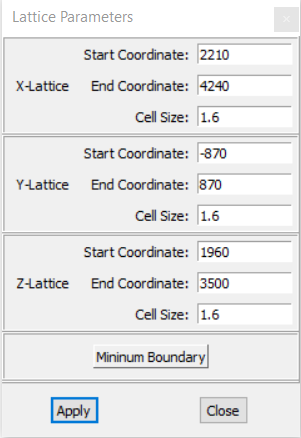
**SETA GRID3 Z16**

## Defining the Model Units

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

1. Open the **General** toolslist in the **EMA3D** tab and click **Specify Units**
2. Select **Millimeters** in the pop-up window
3. Click **OK**

## Defining the Simulation Domain

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

1. Click **Define** **Lattice** in the **General** tools list of the **EMA3D** tab
2. Click on the **Lattice** **Parameters** tool in the expanded section
   1. Define the X-Lattice to be 2210 to 4240
   2. Define the Y-Lattice to be -870 to 870
   3. Define the Z-Lattice to be 1960 to 3500
3. Change the cell size to **1.6** for the **x, y**, and **z** directions.

Ensure the parameters are entered correctly as seen below.

1. Click **Apply**
2. The boundary of the problem space can be seen by copying the following into the command line:

**PLUS LATE**

## Meshing the Geometry

Next, we need to mesh the geometry.

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

## Defining the Simulation Time Steps

Next the simulation time steps must be defined. Our simulation will run to 0.02 μs with a time increment of 1.6e-12 s.

1. Click **Define** **Time** **Step** in the **EMA** tab
2. Click **Time** **Step**
3. Enter **1.6e-12** into the box labeled **Time step**
4. Type **2.0e-8** in the box labeled **Final Simulation Time**
5. Click **Apply**

## Defining Background Properties

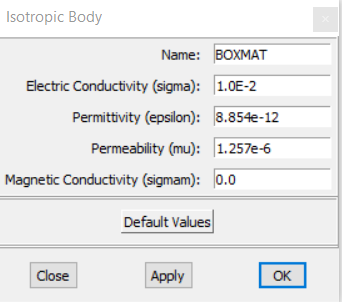
Next, we need to define the property of empty space in the problem.

1. Click **Define Background**

The default values are for air which will be used for this problem.

1. Click **Ok**

## Defining Material Properties

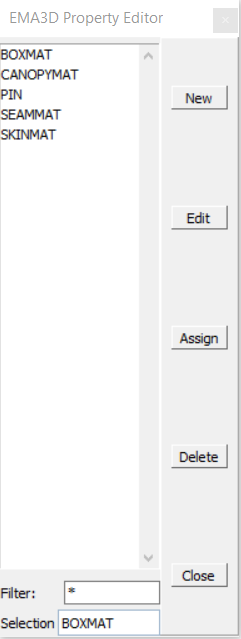
The geometry in the model will need properties assigned to it in order to simulate the problem. The entire model is composed of surfaces, so isotropic surface materials will be defined along with a PEC line material.

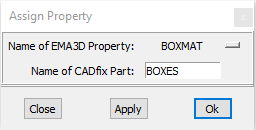
1. Click **Define Properties** in the **EMA** tab
2. Click **New** in the “EMA3D Property Editor” window
3. Enter **BOXMAT** into the **Name** field
4. In the **Type** spin box, change the definition to **Isotropic Body**
5. Click **Ok**
6. In the popup window, enter **1.0e-2** in the **Electric Conductivity** field
7. Leave the other fields as the default and click **OK**

Repeat Steps 2-7 with the names, material types and electric conductivities from the following table:

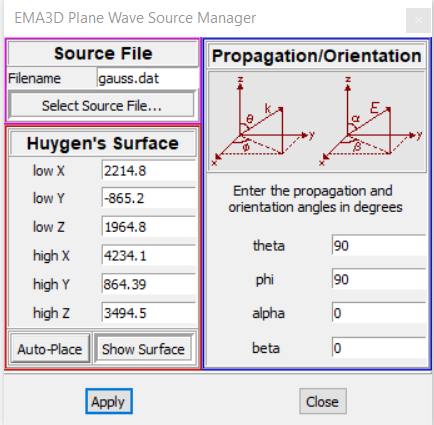
|  |  |  |
| --- | --- | --- |
| EMA3D Material Name | EMA3D Material Type | Electric Conductivity |
| SKINMAT | **Isotropic Surface** | 6.85E+06 |
| SEAMMAT | **Isotropic Surface** | 0 |
| CANOPYMAT | **Isotropic Surface** | 1.85E+01 |
| PIN | **PEC Line** | N/A |

## Assigning Material Properties

Now that we have our properties defined, we can assign the properties to the geometry itself.

1. Select **BOXMAT** in the “Property Editor” popup window
2. Click **Assign**
3. Type **BOXES** into the field labeled **CADfix Part**
4. Click **Apply**
5. Select **PIN** as the **EMA3D Property**
6. Type **PINS** as the **CADfix Part**
7. Click **Apply**
8. Select **CANOPYMAT** as the **EMA3D Property**
9. Type **CANOPY** as the **CADfix Part**
10. Click **Apply**
11. Select **SEAMMAT** as the **EMA3D Property**
12. Type **SEAM** as the **CADfix Part**
13. Click **Apply**
14. Select **SKINMAT** as the **EMA3D Property**
15. Type **SKIN** as the **CADfix Part**
16. Click **Ok**

## Defining the Plane Wave Source

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

1. Click **Plane Wave** in the **Define Sources** section of the **EMA** tab
2. Enter **gauss.dat** for the **Source File**
3. Click **Auto-Place** for the **Huygen’s Surface**
4. For the **Propagation** values enter the following:
   * 1. **Θ = 90**
     2. **Φ = 90**
     3. **α = 0**
     4. **β = 0**
5. Click **Apply**

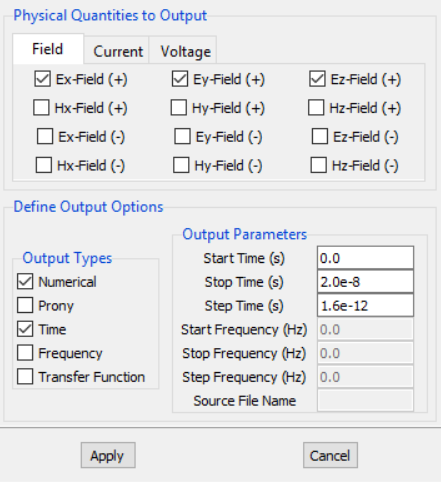
## Defining Boundary Conditions

Now we can define the boundary conditions for the problem. For HIRF, the perfectly matched layer boundary condition will be used for all boundaries.

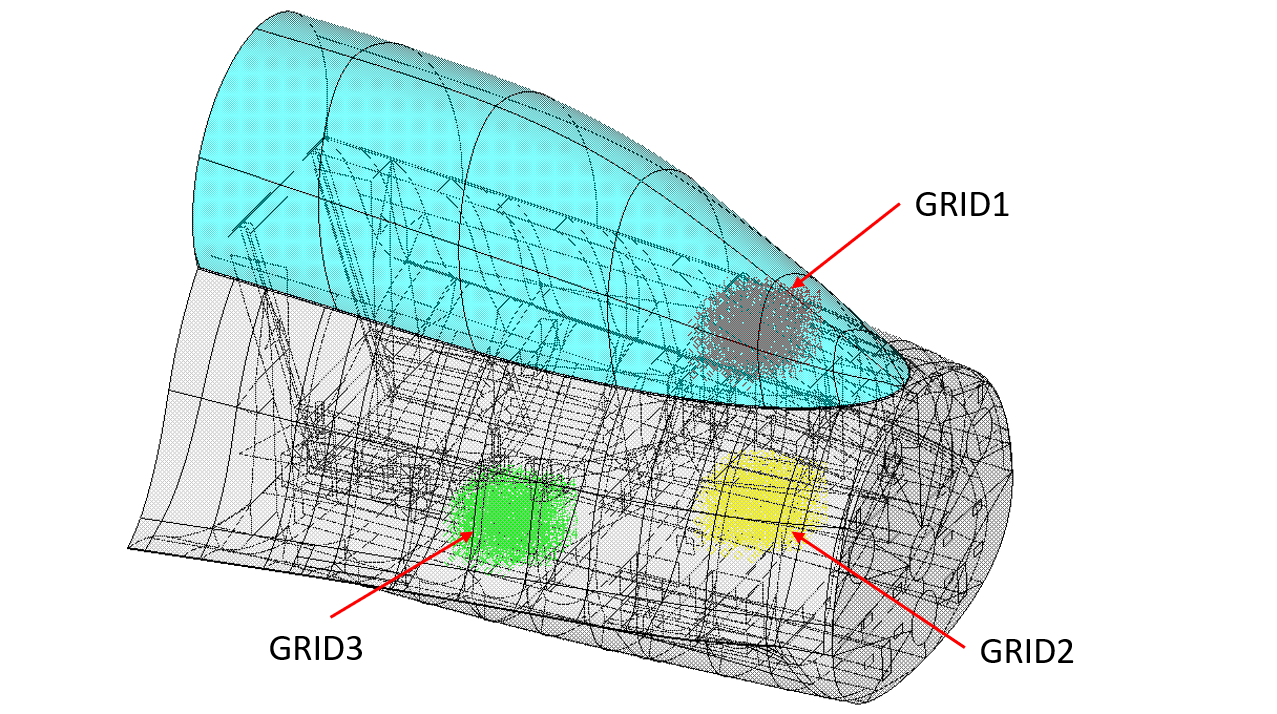
1. Click **Define Boundary Conditions** in the **EMA** tab
2. Select **PML** for all boundaries
3. Enter **32** for the **# Layers** and press **Apply**

## Creating Probes in EMA3D

Next, we’ll create the electric field probes that will capture the fields inside the canopy and be used to compute the shielding effectiveness.

1. Click **Define Probes** in the **EMA** tab
2. Right Click on the **New Probe** and select **New**
3. Enter **GRID1** as the **Set containing points**
4. For the **Probe Points** enter the set **GRID1** and press **Enter**
5. Select positive **Ex, Ey** and **Ez** fields as the outputs
6. The **Output Types** needed are **Numerical** and **Time**
7. Enter **1.6e-12** into the box labeled **Step Time**
8. Type **2.0e-8** in the box labeled **Stop Time**
9. Click **Apply**

Repeat steps 2-9 two more times, entering **GRID2** and **GRID3** as the **Set containing points.**



## Creating the EMA3D Input File

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

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

**NOTE**: Due to the relatively small mesh size in the model it may take a while to create the input file.

## Running the Integrated Simulation

We are finally prepared to run the EMA3D simulation.

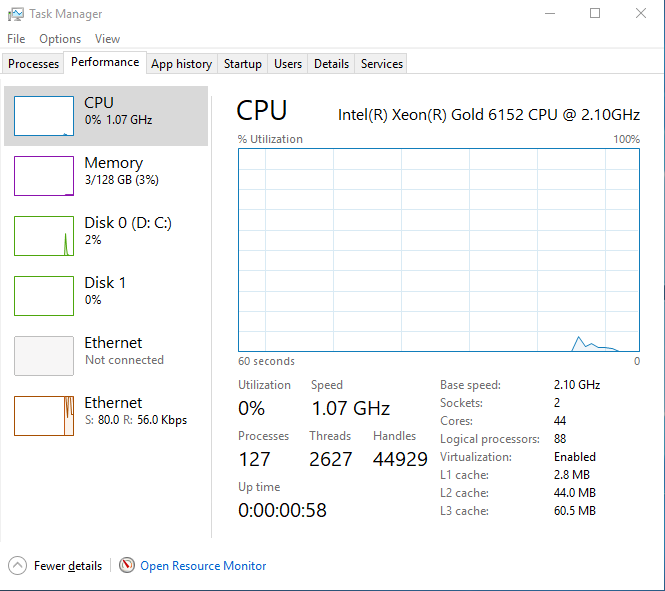
1. Navigate to the **General** toolslist under the EMA3D tab
2. Click **Run EMA3D** to launch the **EMA3D** **Run** **Tool**. This may take several seconds to launch
3. Select the **Run** **New** **EMA3D** **Simulation** radio button
4. In the Entry titled **EMA3D** **Input** **File** click the **Browse** button
5. Select the **.emin** file we created earlier
6. Click **Run** **EMA3D** to start the simulation

**NOTE**: The simulation will work in the EMA3D run tool, but it will take multiple days to finish the simulation. It is suggested to run this simulation in parallel if at all possible.

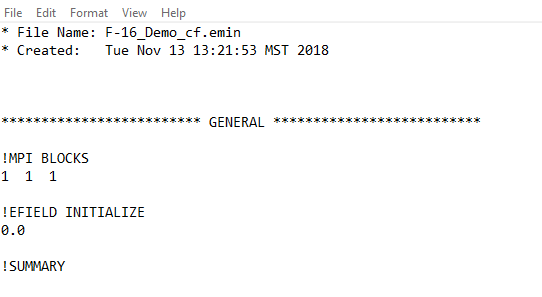
## Run EMA3D Simulation in Parallel

EMA3D allows for the simulation to be parallelized across multiple computing processors. This can dramatically speed up simulation time and is suggested should multiple computing cores be available for use.

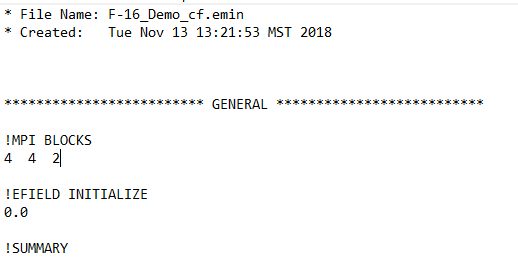
1. To check how many cores are available on any compute, open the task manager and click on the performance tab and select the tab labeled **CPU**. The total amount of **logical processors** can be found in the bottom right corner.



1. The computer this demo was run on has 88 processors available, but for this example 32 processors will be used.
2. Open the file named **F-16\_18GHz\_Demo.emin** in any text editor.



1. Edit the line under **!MPI BLOCKS** from **1 1 1** to an even distribution of the number of processors you are going to run this on. For example, since this computer will run the problem on 32 cores the line will be changed to **4 4 2.** It is important to note that this distribution will most likely not be the fastest way to solve this problem. The fastest way to solve the problem can be found by running the first 15 – 20 time steps of the problem using different distributions in the **!MPI BLOCKS** section.



1. Open a command prompt window and navigate to the folder you were working out of.
2. Type **mpiexec.exe -n # ema3d F-16\_18GHz\_Demo.emin** where **#** is the number of cores you are running the problem on, in the case of the computer the example was run on this would be 32.
3. Hit **Enter** and the program should begin.

**NOTE:** If this is your first time running MPI the program will ask for an account user name and password, this is just the user name and password for the computer you are using. Once that information is entered the program should begin.

1. EMA3D will check for issues in the input file and then prompt you if you would like to start the sim. Type **Y** and hit **Enter** and the program will begin to run the simulation.

## Analyzing Results

The data that is captured in the simulation can be looked at in using the **EMA3D Run** tool. However, the statistical analysis for the shielding effectiveness of the canopy must be done in an outside program. Two scripts have been prepared to do this analysis, one in python and one in MATLAB. Please select whichever script you prefer.

### Python Analysis

1. Open the file named **F-16\_18GHz\_demo.py.**
2. Scroll down to line **110** and change **di** to the directory containing the data.
3. Press the **Run**  at the top of the screen and allow a couple of minutes for the simulation to run.

### MATLAB Analysis

1. Open the file named **F\_16\_18GHz\_Demo.m.**
2. Ensure the data is in the same directory as the script and press **Run.**

****

****

****