



Basic CADfix and EMA3D Training

Prepared by Eric Miller

January 6, 2017

Contents

Introduction	4
Things to Keep in Mind	4
Getting Started.....	4
Constructing Geometry.....	7
Fuselage	7
Create a cockpit:	10
Wings	15
Vertical Stabilizer	20
Finishing Touches.....	23
Cockpit Window	23
Fuselage Windows	28
Creating a Cable	31
Geometry Notes.....	32
Steps to EMA3D Simulation	33
1) General Topics	33
2) Units.....	33
3) Define Lattice	33
4) Construct Source Geometry	36
5) Meshing	36
6) Time step	37
7) Background	37
8) Define Properties	38
9) MHarness Cables	41
10) Define Source.....	42
11) Boundary Conditions	44
12) Probes	44
Bulk Current Probe Geometry	44
Bulk Probe Properties	45
E-Field Probe Geometry.....	45
E-Field Probe Properties	45
EMA3D Review Tool.....	46

Running EMA3D	47
Post-Processing	48
EMA3D GUI Processing	48

Introduction

The purpose of this training module is to build familiarity with the basic functions of CADfix and EMA3D. By building a very simple plane model, you will learn basic geometry creation and modification techniques. Due to the limited nature of this training module, many CADfix techniques won't be shown. In later modules, more advanced techniques will be explored.

Almost everything in CADfix can be done using either a GUI (graphic user interface) or using a command line input. In this training, both methods will be shown for the applicable steps. Often, beginning users find the GUI easier than the command line. However, the command line can greatly increase efficiency and more naturally lends itself to the creation of user-written macros.

Once the plane geometry is constructed, you will follow the steps to turn the CADfix model into an EMA3D simulation. The process shown in this module is intentionally left basic to prevent tedium.

Note: This training was made using CADfix 10 SP2.0

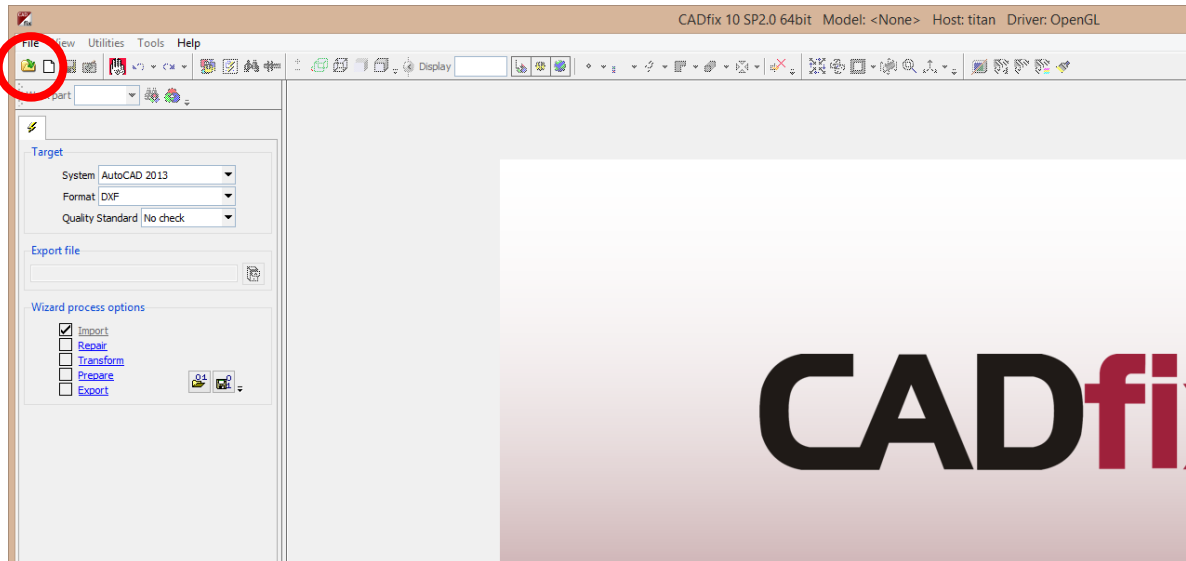
Things to Keep in Mind

There are some quirks to keep in mind when using CADfix/EMA3D:

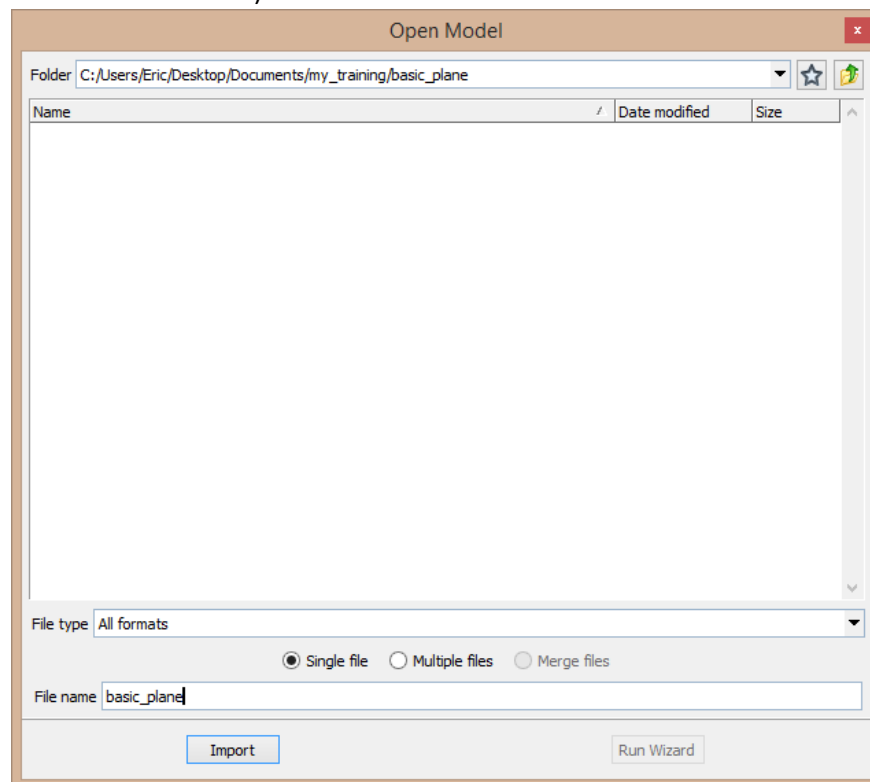
- The position of the mouse determines the focus of the program. In other words, keyboard entries will go into the window underneath the mouse cursor.
- Geometries are grouped into sets.
- There is a set manager that can be found in Utilities->Set Manager. This can be used as a GUI equivalent for the command line SET commands shown later in the document.
- While a set is open, it will collect all newly created geometry. More than one set can be open at a time.
- When using the CADfix command line, the "Delete" key acts as a single backspace.
- When using the CADfix command line, the "Backspace" key deletes the whole line.
- When using the CADfix command line, the back arrow deletes to the next whitespace.
- The escape button clears the current geometry selection/highlighting.

Getting Started

1. Open CADfix
2. Click the "Open Model" button at the top left of the screen

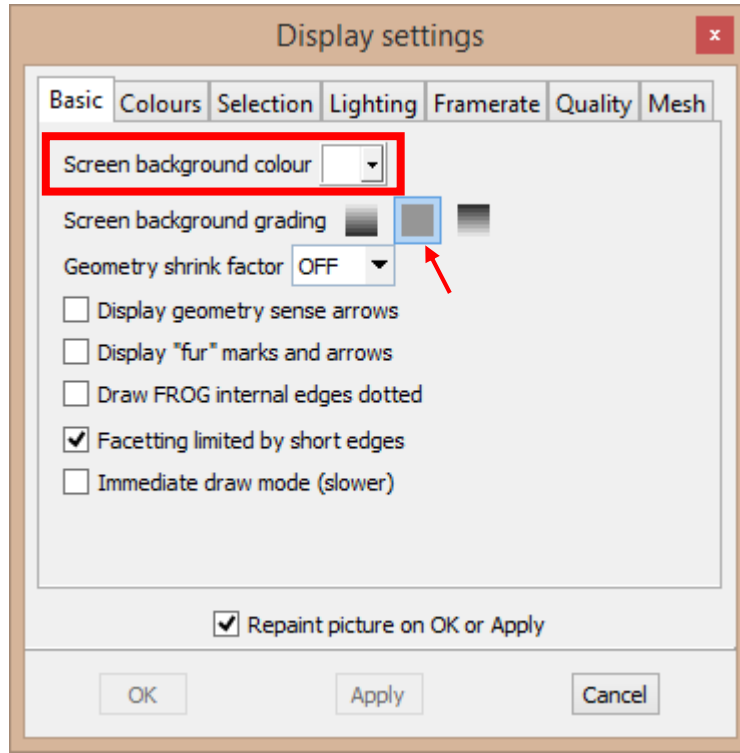


3. Navigate to a folder to hold the files created during this training
4. Type a file name into the “File name” text field and click “Import” (this will open the file entered or create it if it doesn’t exist)



5. An empty model should be opened. The background might be a dark gradient. To change it to white, do one of the following:

- a. Navigate to Display Settings (View->Display Settings). Under the “Basic” tab, change the screen background colour to white using the dropdown menu. Change the background shading to solid. Leave the other fields unchanged. Click “OK.”



- b. Alternatively, locate the command line at the bottom of the screen and type:

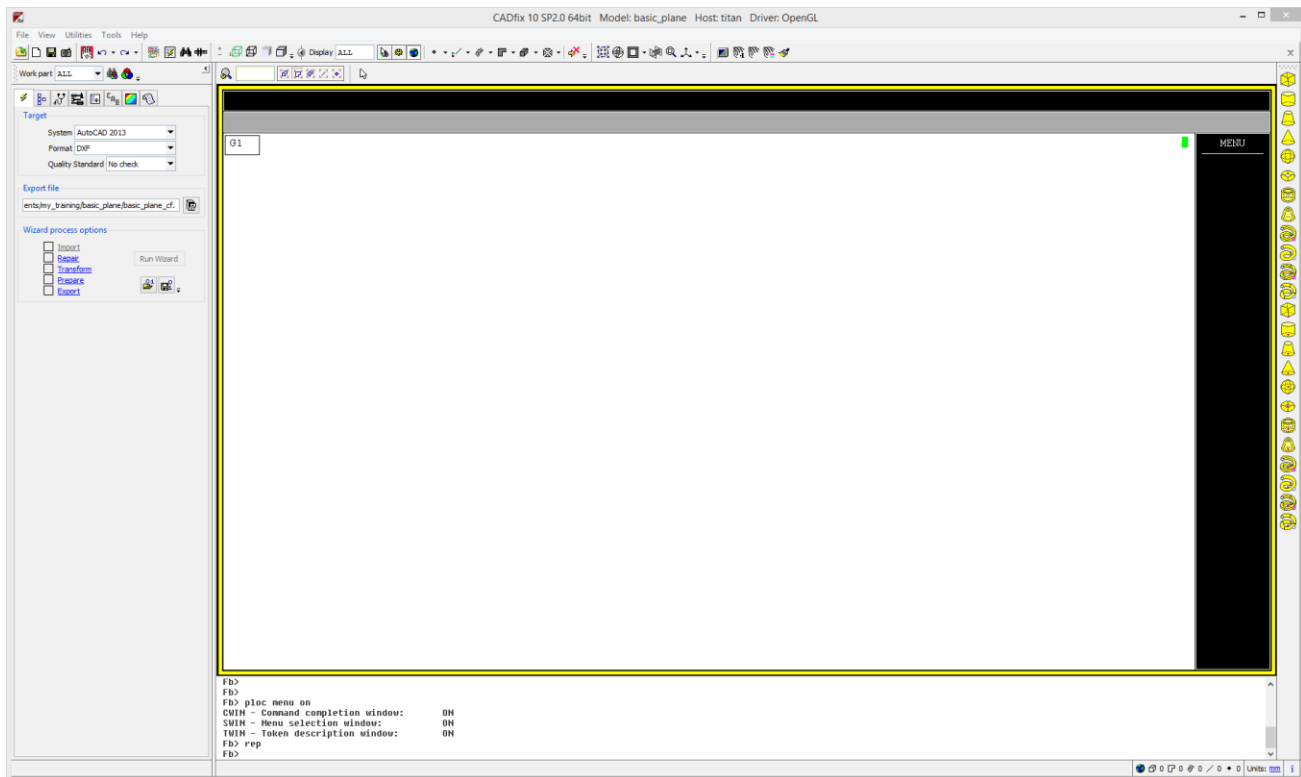
PLOC BGND W

Then hit enter. To see the change, type **REP** and hit enter. This will repaint the image.

“PLOC” accesses the display settings, “BGND” accesses the background settings, “W” is white



6. To get a helpful command menu on the screen (you'll want this), type **PLOC MENU ON** and hit enter. To force the menu to appear, you'll have to resize the screen. Now CADfix should look like:



Constructing Geometry

Now we're ready to start making a simple airplane. We'll make wings, fuselage, and empennage. The geometry used in CADfix is composed primarily of points, lines, shapes, surfaces, and bodies. Points are the lowest level entities that can be connected to build lines. Lines can be connected into surfaces (with an embedding shape). Surfaces can be connected to build bodies. However, bodies are computationally heavy for EMA3D simulations and will be avoided in most models.

Fuselage

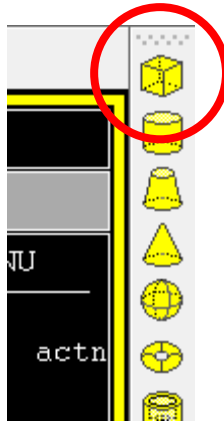
The fuselage will be represented by a rectangular box with rounded forward face, which will represent the cockpit. Our fuselage will be about 35 meters long. There are many possible methods to build a simple box:

1. Use "Cube" primitive in the "Primitives" toolbox to create a box body, then delete the body but keep the surfaces of the cube.
2. Create lower level geometries to make the box using the GUI.
3. Create lower level geometries to make the box using the command line.

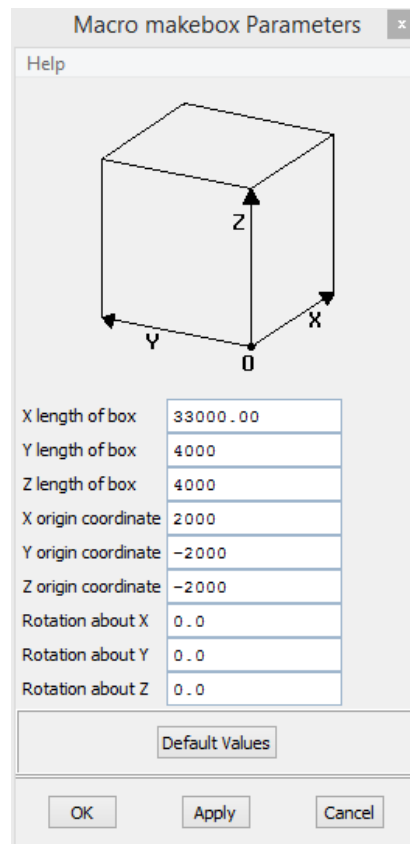
We'll start by using the primitive:

1. Make sure the primitives toolbox is open. It should be on the right edge of the window. If it isn't there, activate it by navigating: Tools->Toolbox->Primitives.

- Click the “Cube” tool. It should be the first tool in the toolbox

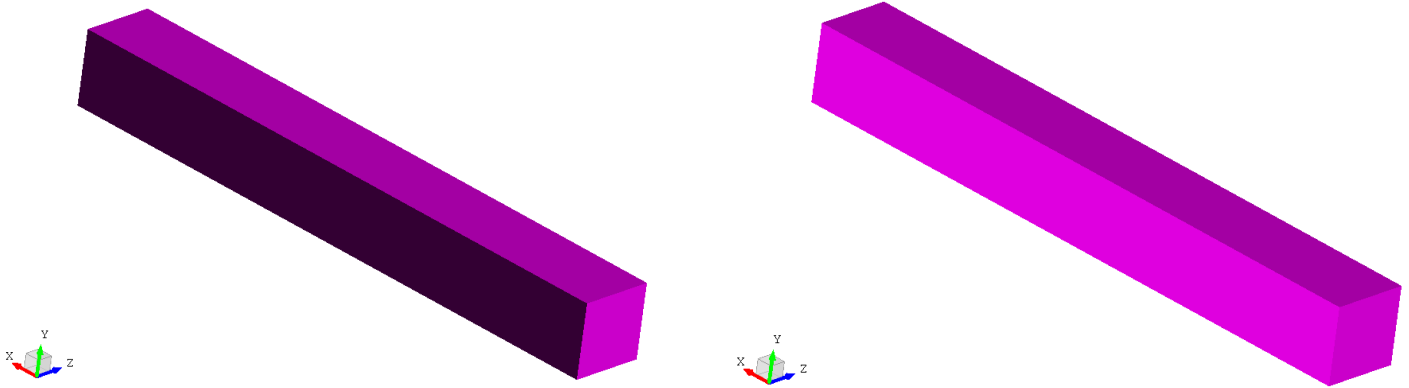


- The box will be 33 m long, anticipating 2 m of cockpit. Fill in the parameters as follows (units are in millimeters by default):



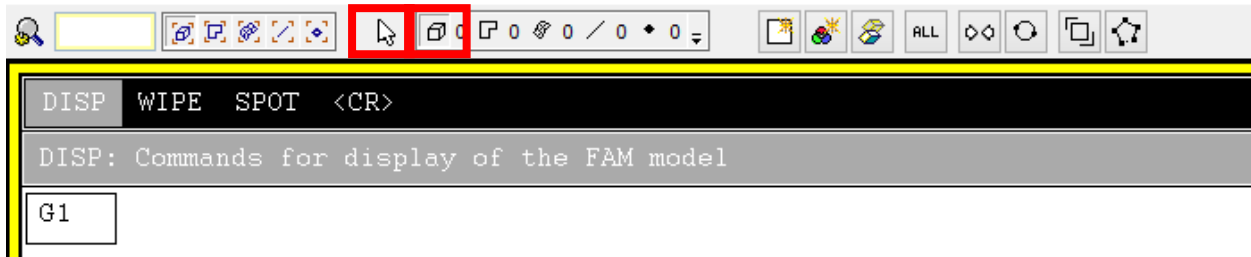
- Type **PLOT SI ALL** and hit enter.
 - “PLOT” tells CADfix to clear the image, then add something to the image, i.e. to plot it
 - “SI” is the type code for shaded surfaces
 - “ALL” tells CADfix what part of the model to plot, in our case “all” or everything
- You should now see a pink box. Using the left mouse button, you can rotate the model. As you rotate, you may notice one side is darker than the others. This is a result of CADfix’s default shading setting. We can change it to look better. Type **PLOC SHAD BOTH** and hit enter

- a. This tells CADfix to shade faces of surfaces the same regardless of normal direction. In other words, the inside of a box or surface will be lighted the same as the outside.
- b. You will need to repaint the picture by typing **REP** in the command line before you can see the changes.



Original image on left, image after changing shading setting on right

6. You'll want to keep track of the fuselage. Put it in a set named FUSELAGE.
 - a. Click on the "Pick bodies" tool. If it isn't available, click the "Basic selection" cursor.

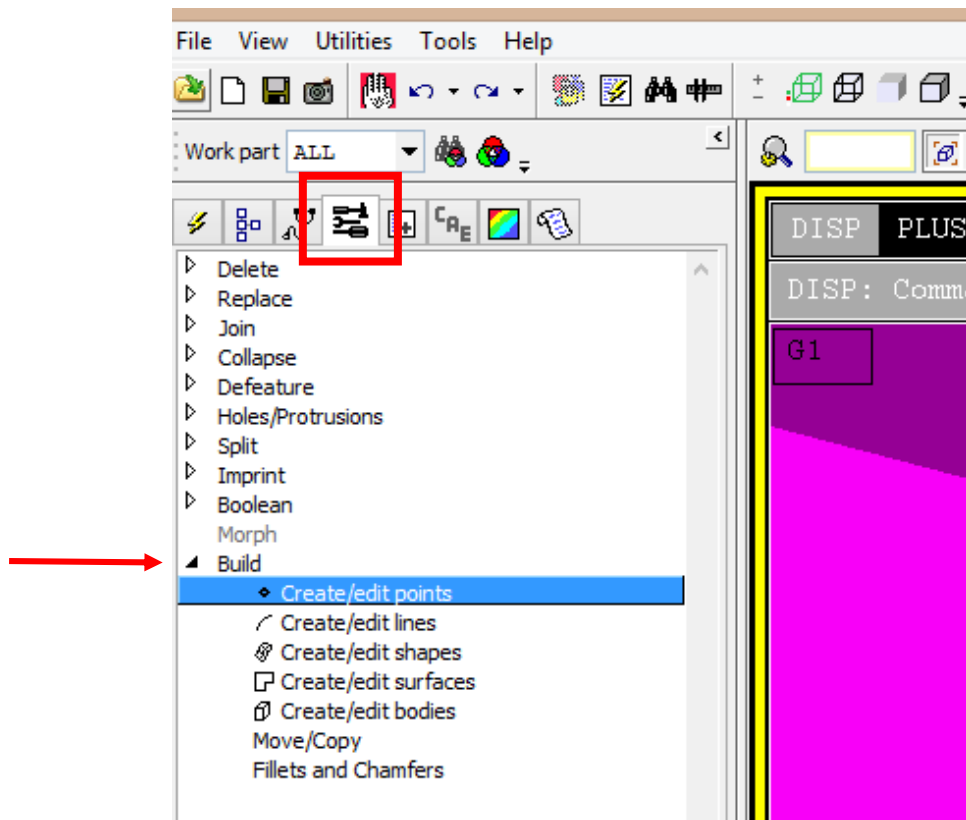


- b. Pick the box you just made.
 - c. With the box still selected, type **SETA FUSELAGE SELE**. This tells CADfix to add the selected items to a set named FUSELAGE.
 - d. Right now, FUSELAGE only contains the body. We want all of the points, lines, and surfaces, too. Type **COMP FUSELAGE**. This tells CADfix to complete the set, adding the lower level components.
(for more information on set management, see "CADfix Notes" in the appendix)
7. As mentioned earlier, bodies are computationally heavy and we don't want them.
 - a. Print the names of the bodies in FUSELAGE: **PRNT B FUSELAGE**
 - b. There should only be one body listed. Delete it: **DEL B name**, where "name" is the body name. For the default naming convention, it is **W1**.

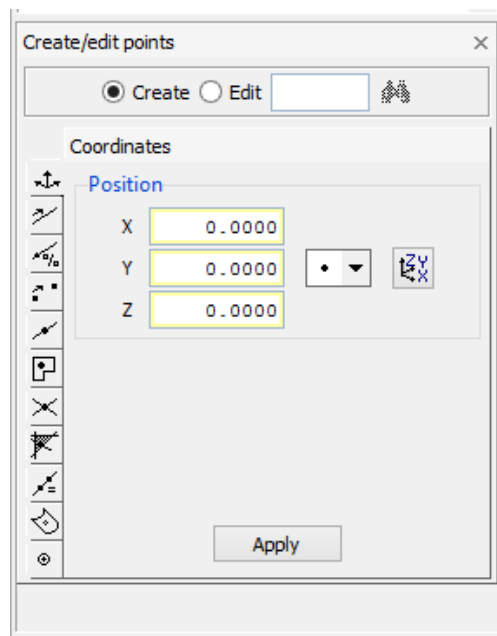
```
Fb>
Fb> prnt b fuselage
  BODY  SUR1  SUR2  SUR3  SUR4  SUR5  SUR6  ELEM      TYPE PHYS MAT  CONS CSYS
W1    +U1   -U2   -U3   -U4   -U5   -U6   +HE8 / 1  BOUN
Fb> del b w1
* Message *  BODY - W1 deleted
Fb>
```

Create a cockpit:

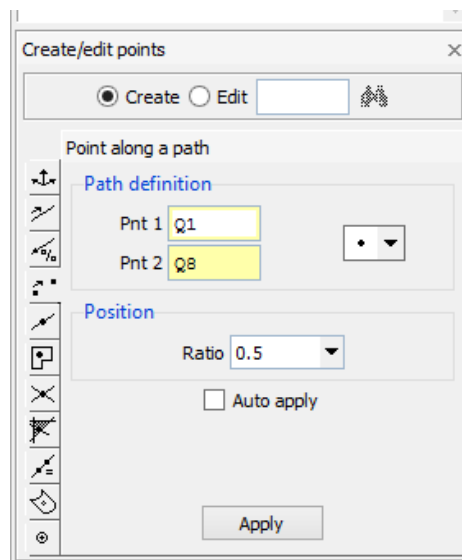
1. Open a set to hold the cockpit (an open set will put all new geometry inside it): **SETO COCKPIT**
2. Make a point at the origin. This can be done by either using the GUI or commands:
 - a. Command line: **PNT ! 0 0 0**. The “!” tells CADfix to automatically name the point.
 - b. GUI: Find the “Tools” tab, then enter the “Build” tree, then click on “Create/edit points.”



- c. Create a point at the origin using the tool by clicking “Apply.”

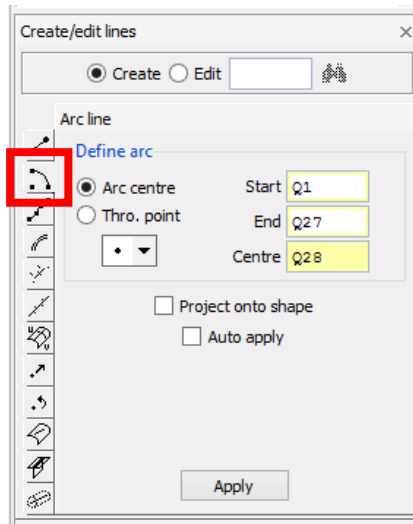


3. Add the cockpit points to the image: **PLUS PA COCKPIT**. “Plus” adds items to the image without removing other entities from the image. The “PA” means points annotated with names.
4. Create a midpoint between opposite corners of the square face by doing one of the following:
 - a. Command line: **PNT ! pnt1 pnt2 0.5**
 - i. Creates point halfway between points “pnt1” and “pnt2.”
 - ii. When selecting points, you can use the cursor to select them rather than typing their names if you use the arrow keys to highlight “-- pnt” in the menu on the right.
 - iii. “pnt1” and “pnt2” are opposite corners of the square face (Q1 and Q8 for example).
 - b. GUI: In the “Create/edit points” tools, select the “Point along a path” tool. Select the corners of the square face and leave the ratio at 0.5. Click “Apply.”

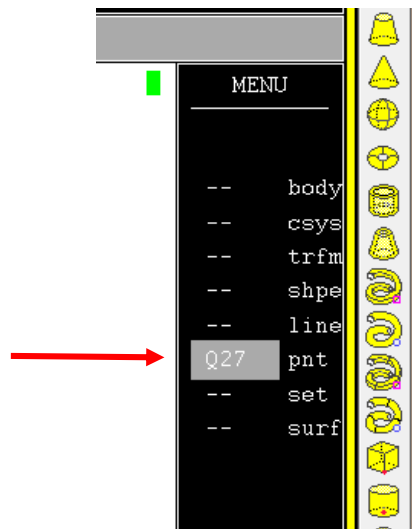


5. Create arcs connecting the corners of the square with the point at the origin. This can be done with command line or GUI:
 - a. Command line:
 - i. **LINE ! pnt1 pnt2 centerpnt**, where “pnt1” and “pnt2” are arc endpoints and “centerpnt” is the center of the arc. The center point is the midpoint of the square face.
 - ii. Points can be selected with the mouse without typing their names.
 - b. GUI:

- i. Navigate to the “Create\edit lines” tools under Tools->Build. Select the arc tool.

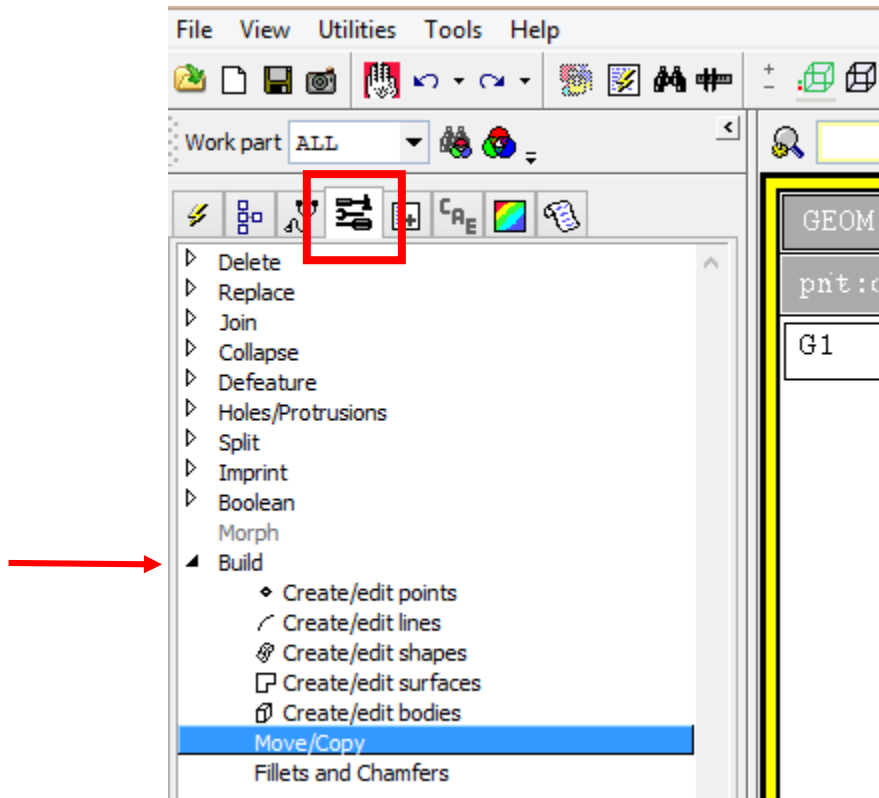


- ii. Select the start, end, and center points. Click “Apply.”
6. Add the new geometry to the image: **PLUS G COCKPIT**
 7. At this point, you can see that the cockpit isn’t very pointy. Let’s move the forward point to make the airplane sleeker.
 - a. Command line: **MOVE pnt TRA -1000 0 0**
 - i. To select the point, use the up/down arrow keys to highlight “pnt” in the side menu. Now you can use the mouse to pick the points instead of typing their names.

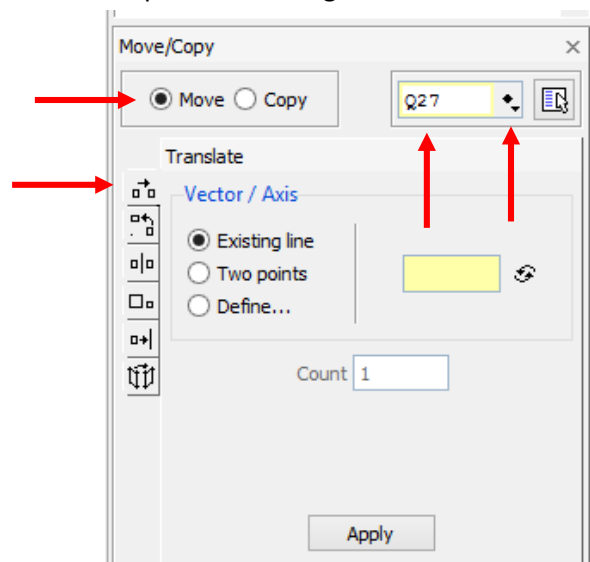


- ii. “TRA” indicates a TRAnslation is to be used (there are options for other kinds of transformations, too).
- b. GUI:

- i. In the “Tools” tab, find the “Move/Copy” tool under the “Build” tree.

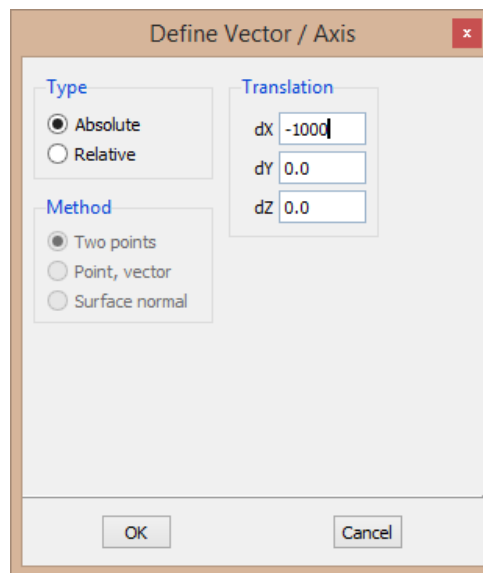


- ii. Select “Move.”
iii. Select the “Translate” tool (the default).
iv. From the dropdown menu, choose point entity types.
v. Select the point at the origin.



- vi. Click “Define...” A new window will pop up.
vii. Select “Absolute.”

viii. Enter -1000 in the “dX” field.

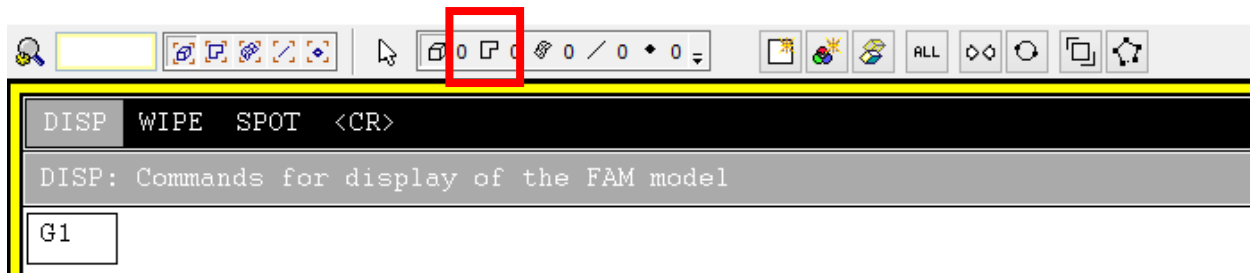


ix. Click “OK.”

x. Click “Apply.”

8. We’re about to create surfaces with the new lines. First, let’s delete the square face so the cockpit and fuselage share the same interior.

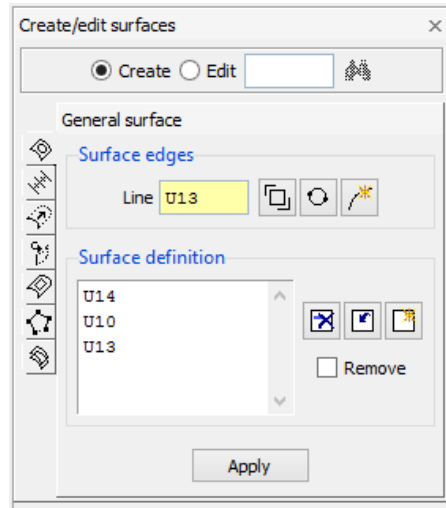
a. Select the “Pick surfaces” tool.



b. Pick the square face. Make sure that the surface is the only thing selected.

c. **ZAP SELE** (“zap” is similar to the **DEL** delete command except that it operates on sets)

9. Make cockpit surfaces:
 - a. Find the “Create/edit surfaces” tools under the “Tools” tab, “Build” tree.
 - b. In the “General Surface” tab (the default), select the lines that will be the edges of the new surface. These will be two adjacent arcs and the connecting straight line on the fuselage.



- c. Click “Apply.”
 - d. On the pop-up window, click “OK.”
 - e. Repeat steps a-d for the remaining three surfaces.
10. Repaint the picture to see all of the new geometry.
11. Close all open sets to keep it organized: **SETC ALL**

Wings

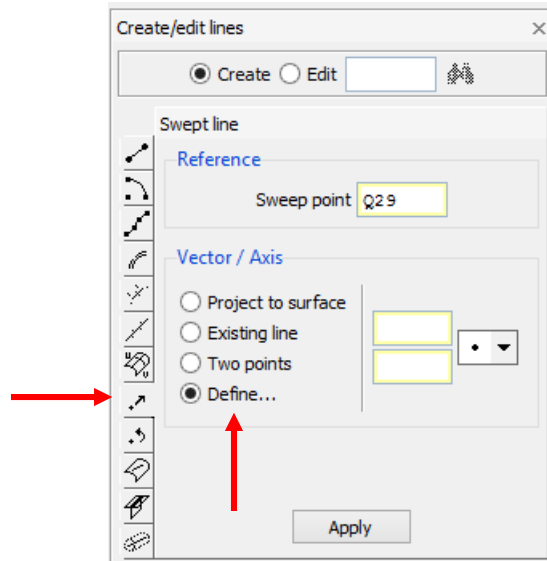
The wings will be 1 meter tall, 1/2 meter from the bottom of the fuselage, and have a total wingspan of 30 m, and start 18 m aft of the forward tip of the cockpit. One wing will be created, then it will be mirrored to the other side. As with the fuselage, there are several ways to start:

1. Use “Cube” primitive in the “Primitives” toolbox to create a box body, then delete the body but keep the surfaces of the cube.
2. Create lower level geometries to make the box using the GUI.
3. Create lower level geometries to make the box using the command line.

Because the “Cube” technique was already shown for the fuselage, the wings will be built by constructing the lower level geometries. The sweep technique will be extensively used, where a lower level entity is swept through space to create a higher level entity.

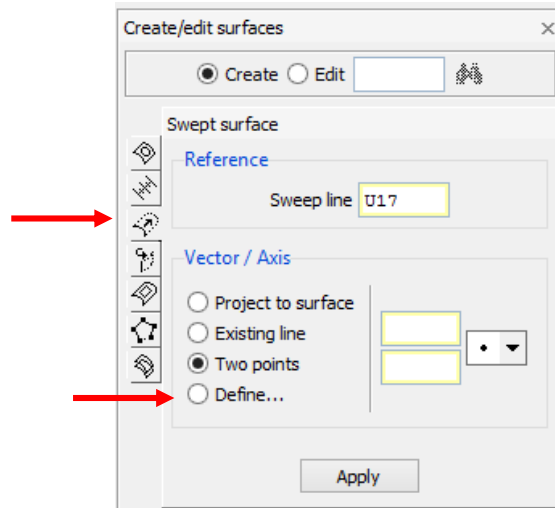
1. Open a set for the left wing: **SETO LEFTWING**
2. Clear or wipe the display: **WIPE**
3. Create a master set for the airplane and add the cockpit and fuselage. This will help organize things for visualization: **SETA PLANE COCKPIT FUSELAGE**
4. Plot the lines of the plane: **PLOT L PLANE**
5. Open the master set: **SETO PLANE** (this will automatically put the left wing in the master set).
6. Create a point at (17000, -500, 2000):

- a. **PNT ! 17000 -500 2000**
 - b. Use these coordinates in the GUI tool (shown in Fuselage Section).
- 7. Add the new point to the image: **PLUS PA LEFTWING**
- 8. Sweep the new point aft 5 meters:
 - a. **SWEP pnt ! TRA 5000 0 0**, where “pnt” is the point you just made.
 - b. Or, use the GUI tool to sweep:
 - i. Under “Create/edit lines,” use the “Swept line” tool.
 - ii. Pick the start point.
 - iii. Click “Define...”



- iv. In the pop-up, select “Absolute.”
 - v. Enter 5000 in the “dX” field.
 - vi. Click “OK.”
- 9. Repaint the picture: **REP**
- 10. Sweep the new line down 1 meter to make a surface:
 - a. Command line: **SWEP line ! TRA 0 -1000 0**, where “line” is the new line.
 - b. Or, use the GUI tool:
 - i. Under “Create/edit surfaces,” use the “Swept surface” tool.
 - ii. Pick the start line.

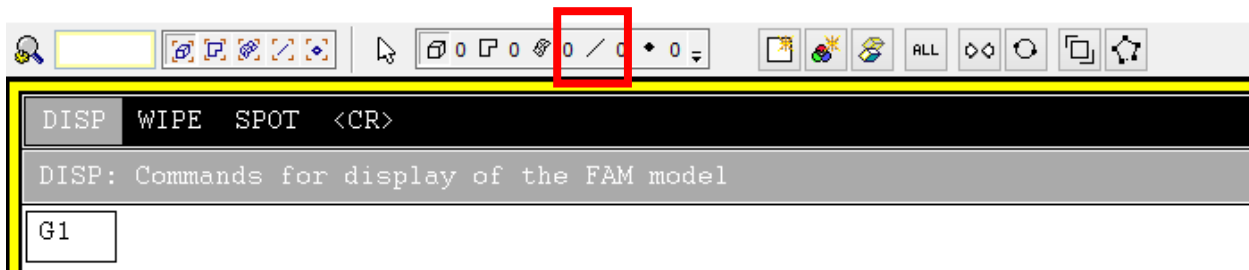
- iii. Click "Define..."



- iv. In the pop-up, select "Absolute."
- v. Enter -1000 in the "dY" field. The other fields should be zero.
- vi. Click "OK."

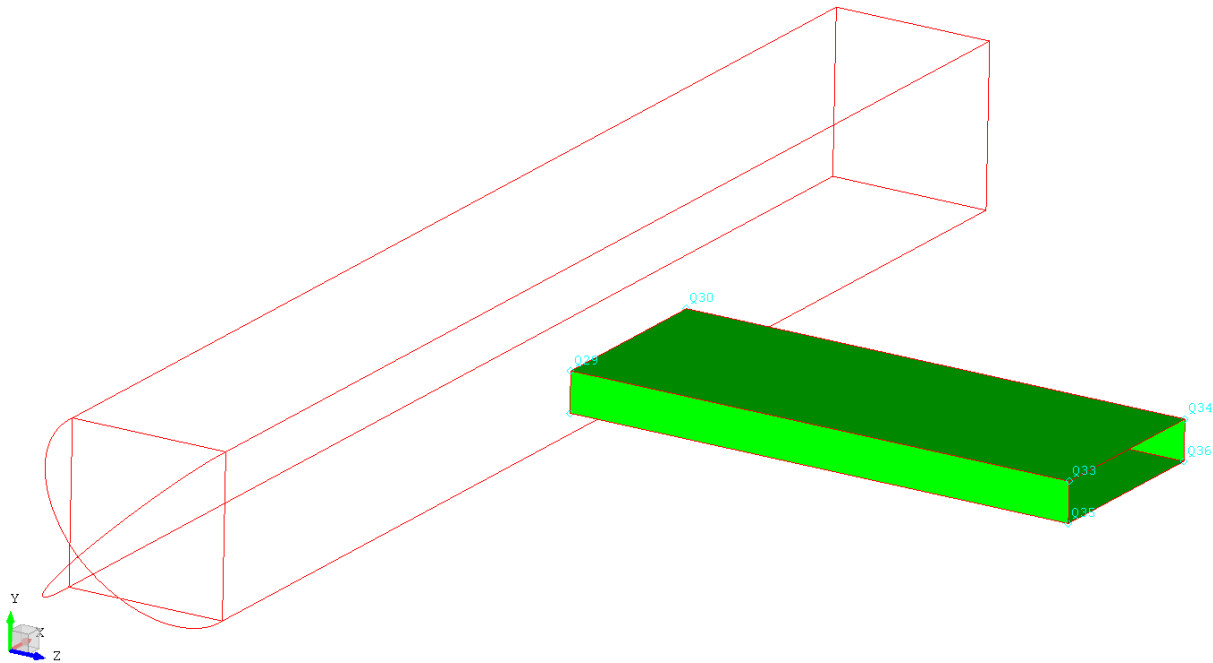
11. To construct the wing, can either sweep the surface you just made to create a body or sweep the lines to create surfaces. If you create a body, you'll have to delete the body. Because this process was already shown when making the fuselage, this document will now show how to sweep the lines to make surfaces. Another surface will be needed to close the wing.

- a. Sweep with command line:
- b. activate the "Pick lines:"



- i. Pick the lines that bound the surface created in step 10 (if they aren't visible, make sure to repaint the picture)
- ii. Enter: **SWEP SELE ! TRA 0 0 13000**
- iii. Repaint the picture: **REP**
- c. Sweep with GUI:
 - i. Follow steps in 10b, choosing the lines created in step 10 as starting points. Instead of entering -1000 in the dY field, enter 13000 in dZ (leaving the others zero).
 - ii. You will need to sweep each line individually with the GUI.
- d. Close the wing:

- i. Add wing surfaces to the image, colored green: **PLUS SI LEFTWING G**



- ii. Use the “General surface” tool to close the gap in the wing. See steps 9a-9d from the Cockpit section for more details.
- iii. Repaint the picture

12. Check what sets are open and close the ones that aren’t “PLANE”:

- a. Type **PRNT SE**

```

Fb> prnt se

```

	P	SH	L	C	S	B	N	E
LOOK	0	0	0	0	0	0	0	0
DISP	0	0	0	0	0	0	0	0
FAIL	0	0	0	0	0	0	0	0
FUSELAGE	26	6	12	0	5	0	0	0
COCKPIT	2	4	4	0	4	0	0	0
LEFTWING	11	6	12	0	6	0	0	0
PLANE	39	16	28	0	15	0	0	0
Y2	0	0	4	0	0	0	0	0

Open sets are: PLANE LEFTWING

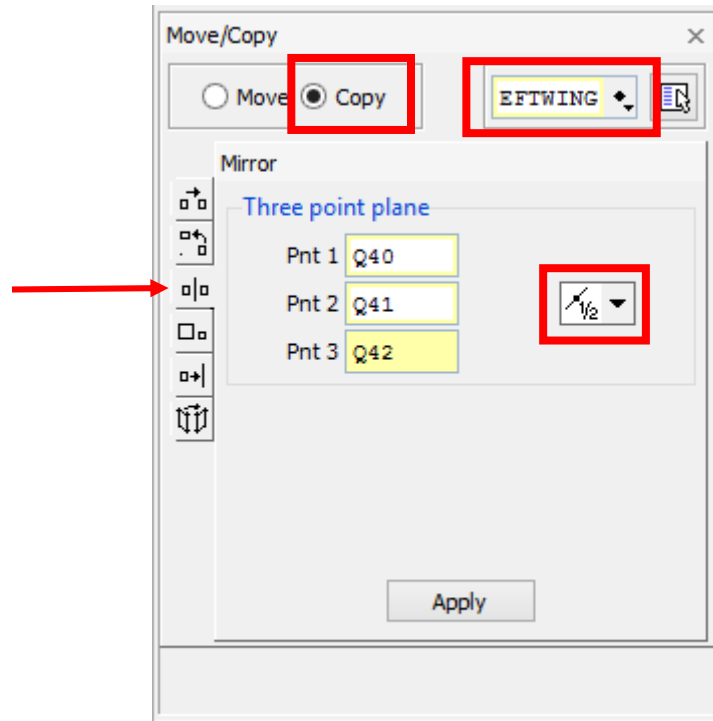
* Message * Some sets were hidden. To see all sets use PRNT SE ALL

Fb>

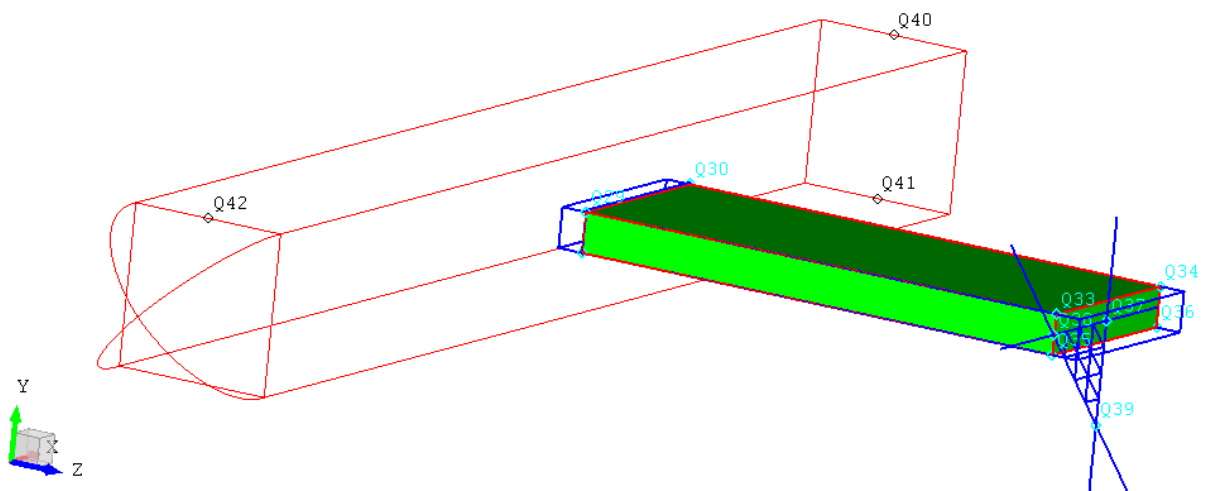
- b. Close any sets that aren’t “PLANE:” **SETC LEFTWING**

13. Mirror the leftwing to the right side (X-Y transformation):

- a. Using the GUI:
 - i. Navigate to the “Move/Copy” tool in the “Build” tree, under the “Tools” tab.
 - ii. Click “Copy.”
 - iii. Switch to the “Mirror” option.
 - iv. Type “LEFTWING” into the selection field and hit enter. This will select everything in the LEFTWING set for copying.
 - v. In the dropdown menu for point selection, choose “Pick mid-point of line.”

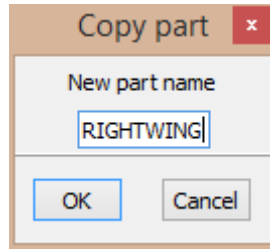


- vi. Select the upper and lower lines at the rear face of the fuselage, as well as the upper line where the cockpit joins the fuselage. Midpoints of these lines will make an X-Y plane.

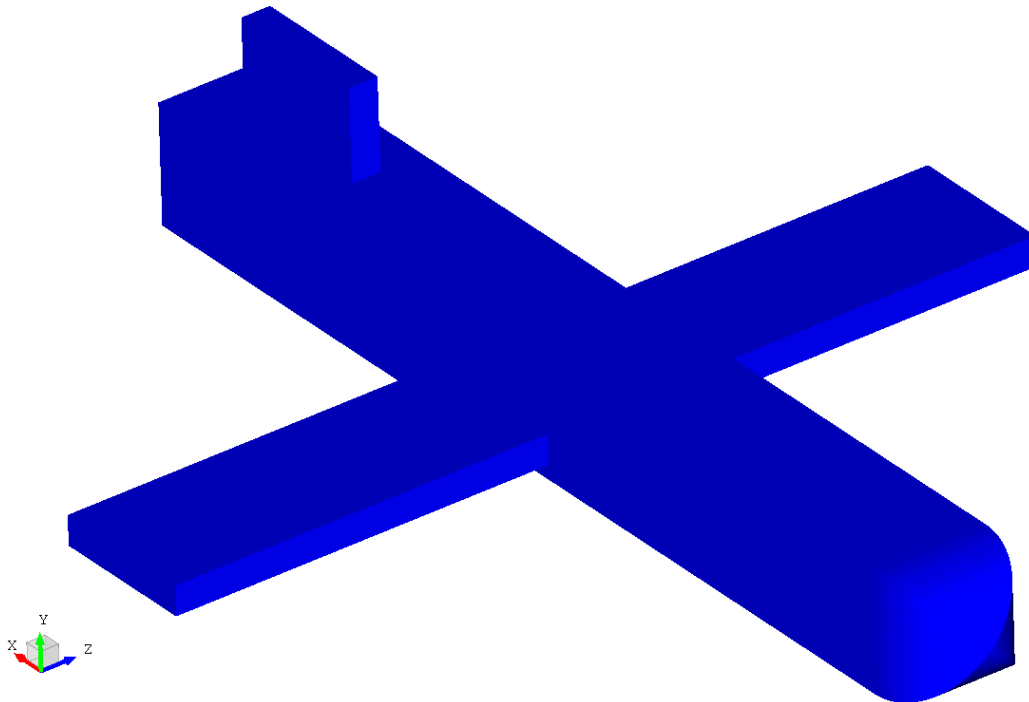


- vii. Click "Apply."

- viii. A popup will ask for a “New part name,” enter “RIGHTWING.” This will put the new wing in a set named RIGHTWING.



- ix. Click “OK.”
- b. Command line: First make a mirror transformation, then copy using that transformation.
- Create three points that form an x-y plane using your preferred method. Note the names.
 - Create a transformation named XYMIRROR:
TRFM XYMIRROR MIR pnt1 pnt2 pnt3
 - Copy the leftwing into RIGHTWING using the XYMIRROR transformation:
COPY LEFTWING RIGHTWING XYMIRROR
You can use this same transformation for all x-y mirrors (for future mirroring, skip straight to step iii).
14. Plot all of the shaded surfaces in PLANE in blue: **PLOT SI PLANE B**

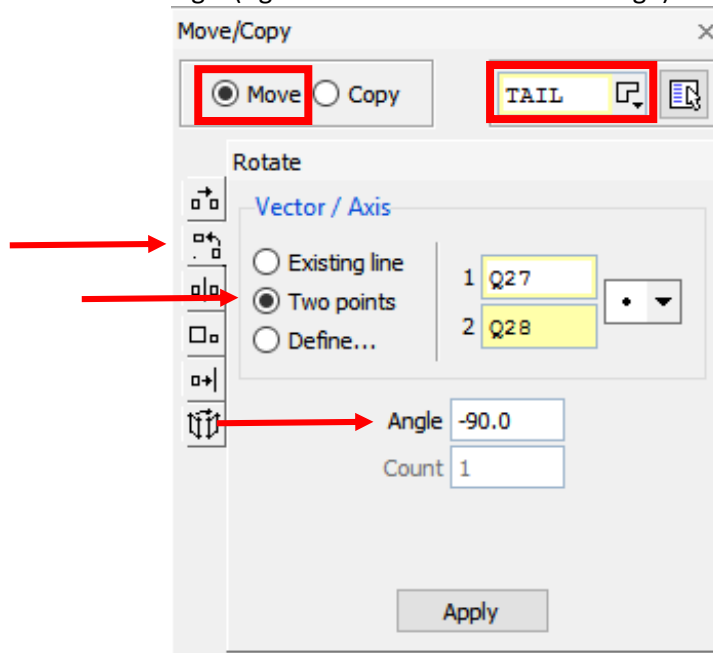


Vertical Stabilizer

To make the vertical stabilizer, we will copy, rotate, and move one of the wings. Because our wing is off-center with respect to the centerline of the fuselage, we can't conveniently rotate the existing wing

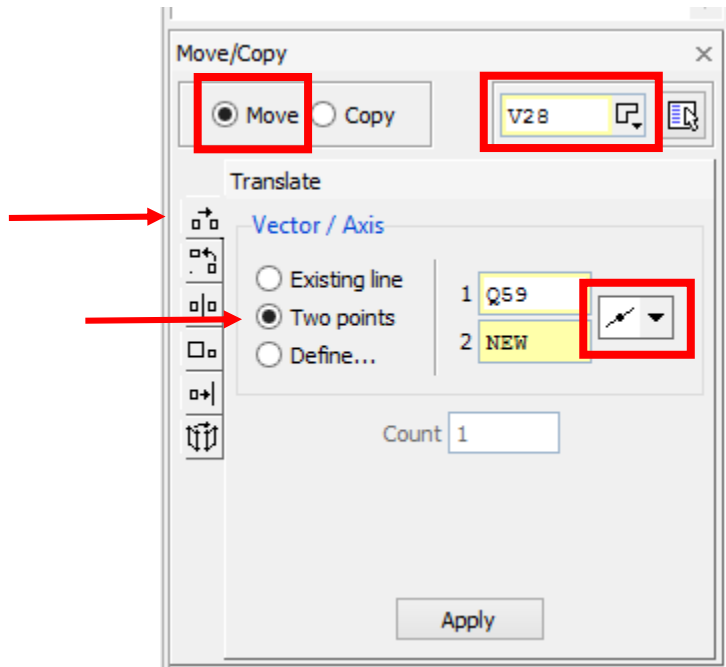
using the centerline to create a centered and flush stabilizer. To remedy this, we will create a copy of the wing so it is centered with the fuselage, then rotate it.

1. Plot the lines in PLANE: **PLOT L PLANE**
2. Add the points (with labels) to the image: **PLUS PA ALL**
3. Create a copy of LEFTWING and move it 1.5 meters up. Call the copy TAIL:
COPY LEFTWING TAIL TRA 0 1500 0 (for GUI instructions, see the Cockpit section step 7)
4. Navigate to the “Move/Copy” tool.
5. Select “Move.”
6. Type “TAIL” into the selection box.
7. Switch to the “Rotate” option.
8. Select “Two points” to define the axis of rotation.
9. Choose the point at the tip of the cockpit and a point directly aft of the tip.
10. Enter “-90.0” as the angle (right hand rule determines the sign).

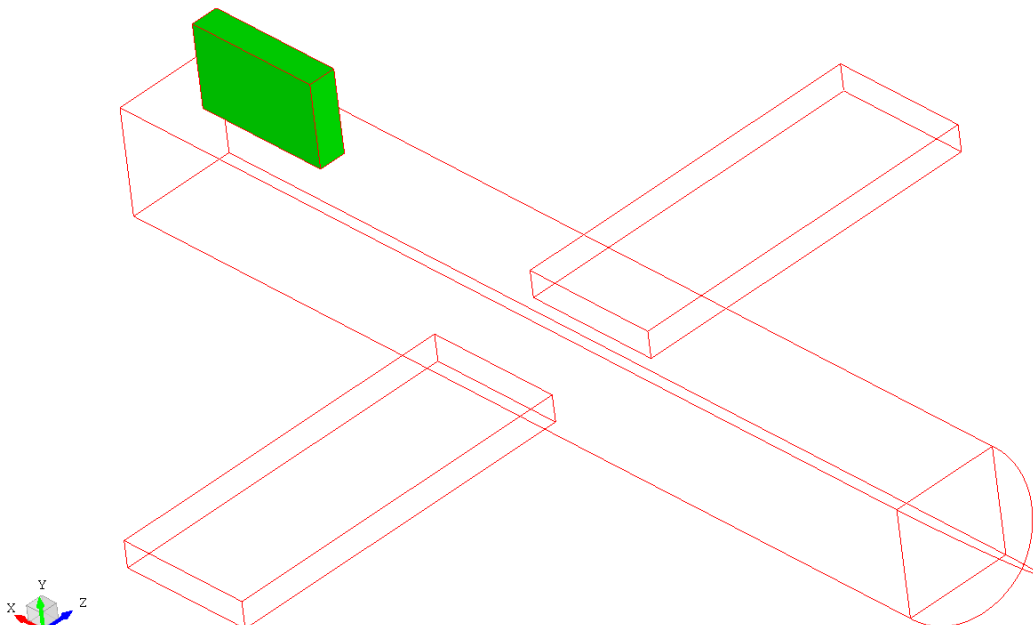


11. Click “Apply.”
12. Wipe all points from the image to help visibility: **WIPE PA**
13. Add the surfaces of the tail to the image, in green: **PLUS SI TAIL G**
14. Move the tail aft 11 meters:
 - a. **MOVE TAIL TRA 11000 0 0**
 - b. Repaint the image
15. Shorten the tail:
 - a. Navigate to the “Move/Copy” tool.
 - b. Select “Move.”
 - c. In the selection dropdown, pick surfaces.
 - d. Select the upper face of the tail.
 - e. Stay on the “Translate” option.

- f. Select “Two points” for the vector definition. (When using “Two points” for a vector definition, the relative difference between the two points creates the vector. The absolute position of the points don’t make any difference.)
- g. For the first point, select a corner of the upper face.
- h. For the second point, use the dropdown to change to “Position on line” and pick a position along the line underneath the first point. The position you pick will be the new height of the tail, so pick a height that looks reasonable (around 1/3 the total height works well).



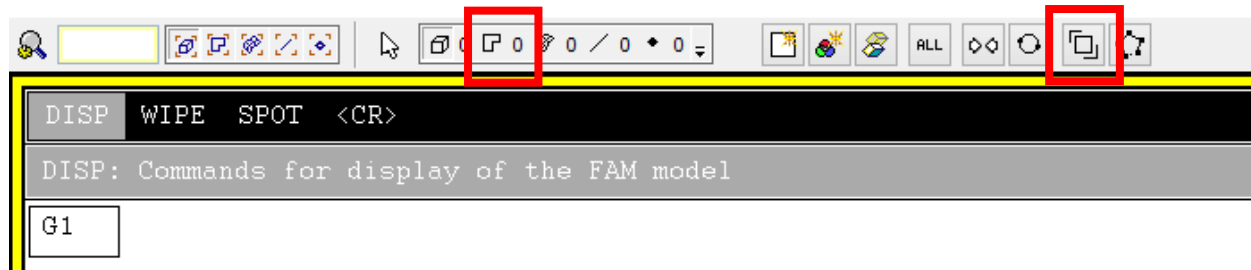
- i. Click “Apply.”
- j. Repaint the picture.



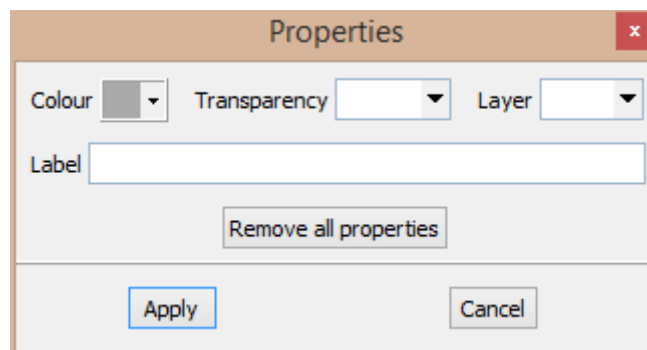
Finishing Touches

The airplane doesn't look much like an airplane. Let's assign a realistic color to it.

1. Plot the plane using assigned colors: **PLOT SI PLANE ASGD**
2. Notice the pink coloring. Pink is the default if no color is assigned.
3. Select the "Pick surfaces" tool.
4. Fit the entire plane in the image.
5. Then activate "Box select" by clicking on it. This will let you drag the mouse over an area to select it.



6. Select the entire plane.
7. Right click anywhere on the image and select "Properties."
8. In the color dropdown menu, pick a color. In the example, a dark gray was chosen.
9. Click "Apply."



10. Repaint the picture.

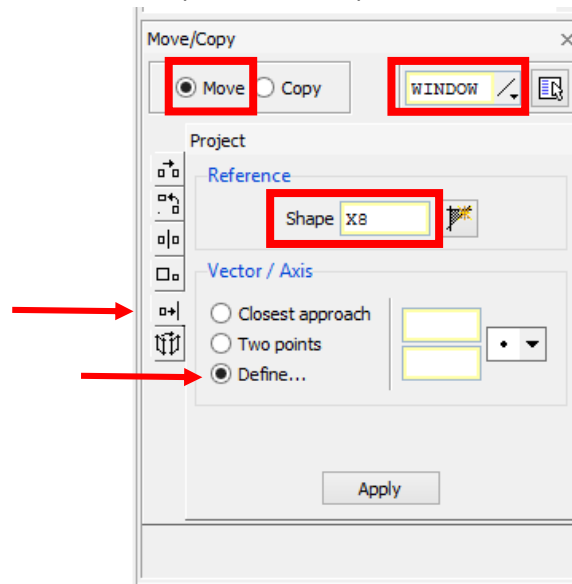
Cockpit Window

We have a basic plane, but no one could fly it. Let's add a window to the cockpit. This step will show how to use a very powerful variant of the "Move/Copy" tool and how to modify existing surfaces. Methods like these are extremely useful in constructing a CEM.

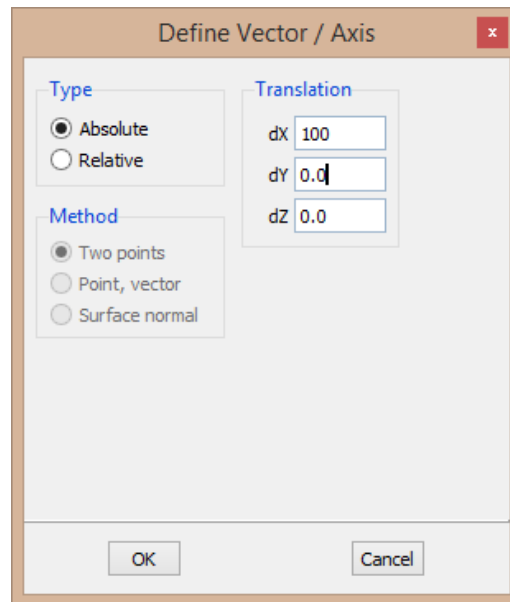
1. Plot the cockpit surfaces with assigned color: **PLOT SI COCKPIT ASGD**
2. Add the cockpit lines: **PLUS L COCKPIT**
3. Check which sets are open: **PRNT SE**
4. Close any open sets that aren't PLANE using **SETC setname**.
5. Open a new set, WINDOW: **SETO WINDOW**
6. Create the lines for the new window:
 - a. Navigate to "Create/edit points" in the "Build" tree.
 - b. In the default "Coordinates" option, change the dropdown to "Pick position on surface..."

- c. A window will pop up. Select the upper face of the cockpit, being mindful that the positive y-axis is up.
- d. The mouse should now look like a crosshair. Select a position on the surface to be the lower right corner of the window (from the pilot's perspective).
- e. Click "Apply."
- f. Add the point to the image: **PLUS PA WINDOW**
- g. Sweep the new point up 750 mm using either the command line or the GUI, whichever you prefer. For convenience, the command is: **SWEP pnt ! TRA 0 750 0**, where **pnt** is the point made in the previous step.
- h. Add the new line to the image, in blue: **PLUS L WINDOW B**
- i. Mirror the line using the "Move/Copy" tool in the "Build" tree:
 - i. Make sure "Copy" is selected
 - ii. Select the new line
 - iii. Use the mirror option
 - iv. The plane should be defined by the midpoints of the upper and lower edges of the cockpit, along with the tip of the cockpit.
 - v. No part name is necessary.
- j. Repaint the picture.
- k. Connect the end points of the two new lines to each other to make a rectangle:
 - i. Command line: **LINE ! pnt1 pnt2**, where **pnt1** and **pnt2** are the points to connect.
 - ii. GUI: Navigate to "Create/edit lines" in the "Build" tree. Select the two end points and click "Apply."
- l. Repaint the picture.
7. Move the window lines onto the cockpit surface:
 - a. Navigate to "Move/Copy"
 - b. Make sure "Move" is selected.
 - c. Use the "Project" option.
 - d. Type WINDOW into the selection field in the top right and hit enter.

- e. For the reference shape, click the top surface of the cockpit.



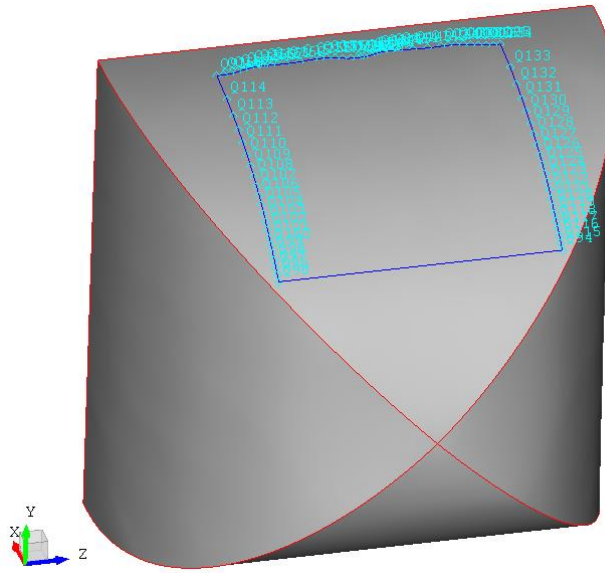
- f. Click “Define...” so we can pick the direction of projection.
- g. In the top left of the popup, click absolute.
- h. Type any positive number into the “dX” field. In the example, 100 is used. Leave the other fields as zeroes. This tells CADfix to only change the X coordinates of the things being moved and only in the positive direction.



If your lines extend past the top of the cockpit, you will have to project a little downwards (add a negative number in the dY, -20 for example).

- i. Click “OK.”
- j. Click “Apply.”

- k. Repaint the picture. It should look like this:



You'll notice that many more points have been created. This is a result of how CADfix projects lines onto embedding shapes. You might also notice that the upper line isn't perfectly straight. That is because the embedding shape isn't perfectly flat. This detail likely won't be noticeable when the model is meshed into a finite difference problem space. You might notice a line or two without many points (the lower line in the picture above). The lack of points is indicative of a failed projection. For any failed projections, use the move tool again, this time using "Closest approach" instead of "Define..." Once finished:

8. Close the WINDOW set: **SETC WINDOW**

9. Open the COCKPIT set: **SETO COCKPIT**

10. Split the upper cockpit surface using the window lines:

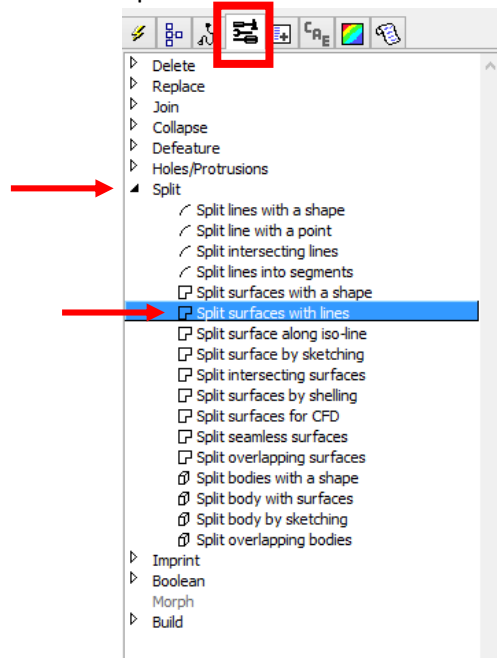
a. Command line: **SSPL surf WINDOW 5**

i. Recall that you can use the arrow keys to highlight "surf" in the menu on the right side of the screen, then you can use the mouse to select the surface.

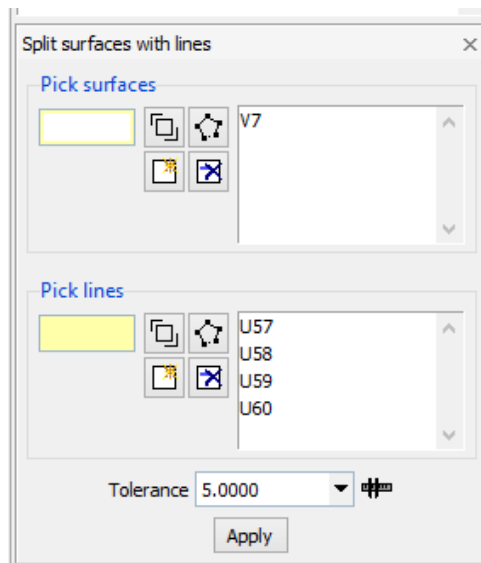
ii. The "5" at the end of the command is the tolerance. This option tells CADfix how accurately to do the split. Units are in model units (mm for this model).

b. GUI:

- i. Navigate to the “Split” tree of the “Tools” tab and select “Split surfaces with lines.”



- ii. Pick the top surface of the cockpit.
- iii. Type WINDOW into the “Pick lines” field and hit enter.
- iv. Change the tolerance to 5 mm. If the split fails, increase the tolerance.



- v. Click “Apply.”
 - vi. Repaint the picture.
11. The cockpit now has a window shaped division, but isn’t grouped properly in sets. The window needs to be added to the WINDOW set and removed from the COCKPIT set.
 - a. Press escape to ensure that nothing is selected
 - b. Activate the “Pick surfaces” by clicking it above the image.
 - c. Pick the window of the cockpit.
 - d. Type: **SETA WINDOW SELE**
 - e. Type: **SETR COCKPIT WINDOW**

- f. Repaint the picture. The window should disappear. That's because only the surfaces in COCKPIT are being displayed and we just removed the window from that set.
 - g. Add the window to the image using assigned colors: **PLUS SI WINDOW ASGD**
 - h. Change the assigned color by selecting the window, right clicking, and entering properties. Use the dropdown "colour" menu.
12. Close all sets: **SETC ALL**
 13. Plot the entire plane, using assigned colors: **PLOT SI PLANE ASGD**

Fuselage Windows

Let's add windows to the fuselage. The windows will act as apertures for energy to enter the aircraft for the HIRF simulation. The quickest way to create the apertures is:

- Make the outline for one window on one side of the fuselage
- Mirror (copy) the outline to the other side of the fuselage
- Then copy both several times down the length of the fuselage
- Remove the windows from the fuselage

Creating the outline

First, create an outline of a window. For this simple model, assume the windows are square. Use ½ meter side length. Make the outline the forward-most window on the left side of the fuselage.

1. Close all open sets: **SETC ALL**
2. Open a new set called APERTURES: **SETO APERTURES**
3. Plot the surfaces of the fuselage, then add the lines of the plane for reference.
4. Create a point at 6000, 1500, 2000: **PNT ! 6000 1000 2000**
5. Add the annotated points in APERTURES to the image: **PLUS PA APERTURES**
6. Sweep that point towards the tail 0.5 meters: **SWEP point ! TRA 500 0 0**
7. Add the lines in APERTURES to the image in a difference color: **PLUS L APERTURES C**
8. Sweep the new line down 0.5 meters: **SWEP line ! TRA 0 -500 0**
9. Add the surfaces in APERTURES to the image: **PLUS SI APERTURES ASGD**
 - a. You can now assign a color if you'd like to (the color assignment will be copied with the outline, so it's easiest to change it now rather than later).

You should now have a little square surface as a window.

Mirroring the outline

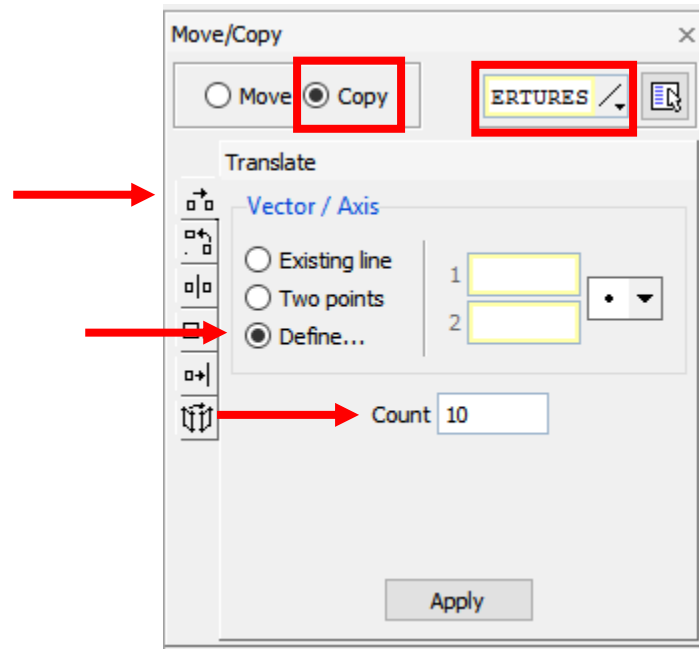
Leave APERTURES open. Using your preferred method, mirror a copy of the outline to the other side of the fuselage. Refer to the "Wings" section, part 13 for instructions for command line and GUI options. Regardless of the method, use the default name for the mirrored part (use ! for the command line **COPY** command – if using the GUI, leave the name popup prepopulated).

Copying the outlines

There are now outlines on each side of the fuselage in the APERTURES set. Leaving the APERTURES set open, copy the outlines through the fuselage. In both the GUI and command line options, there is an optional count option. If provided, CADfix will repeat the copy action that number of times, propagating along the translation vector.

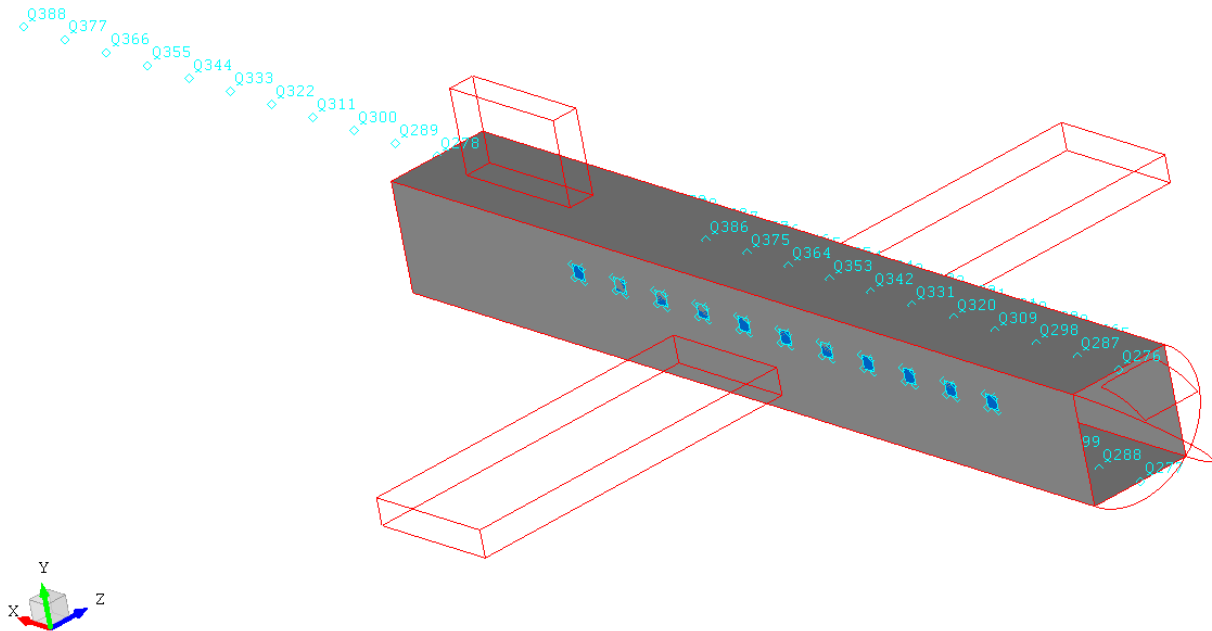
Copying use the GUI:

1. Open the “Move/Copy” tool.
2. Enter “APERTURES” as the item to copy.
3. Use the “Define...” option to define a 2 meter vector pointing in the positive x axis.
4. Change the “Count” entry from 1 to 10.



Copy using the command line: **COPY APERTURES ! TRA 2000 0 0 10**

Depending on how you mirrored the first outline, you might have many extraneous points in APERTURES.



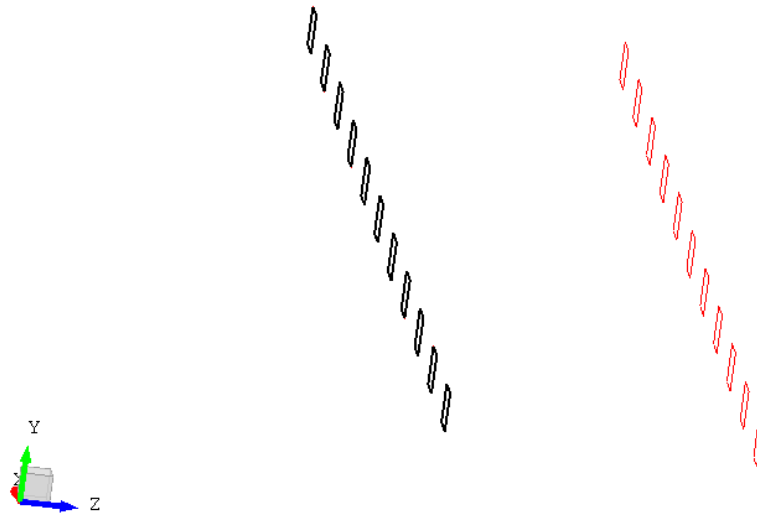
Remove those extra points using: **SETR APERTURES FREE P ALL**
The above command will remove all free (read “unused”) points in the set APERTURES.

Removing the windows from the fuselage

The “Create/edit surfaces” tool is powerful and provides a lot of usability improvements over the command line equivalent. This is one of the few times where the author suggests the GUI over the command line. As such, only the GUI option will be presented here.

First, we need to group the windows into left and right sections:

1. Plot just the lines of the APERTURES.
2. Use the box select tool to select the lines on the right side of the fuselage.

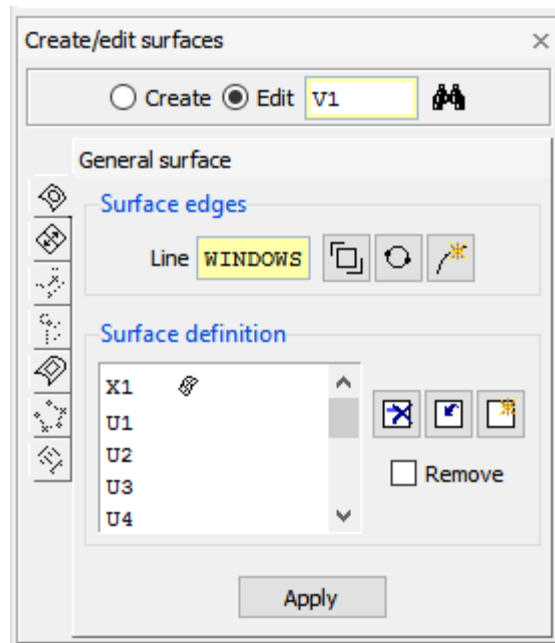


3. Add the selected lines to a set, RWINDOWS: **SETA RWINDOWS SELE**
4. Repeat steps 2 and 3 for the left side of the fuselage, this time using set name LWINDOWS.

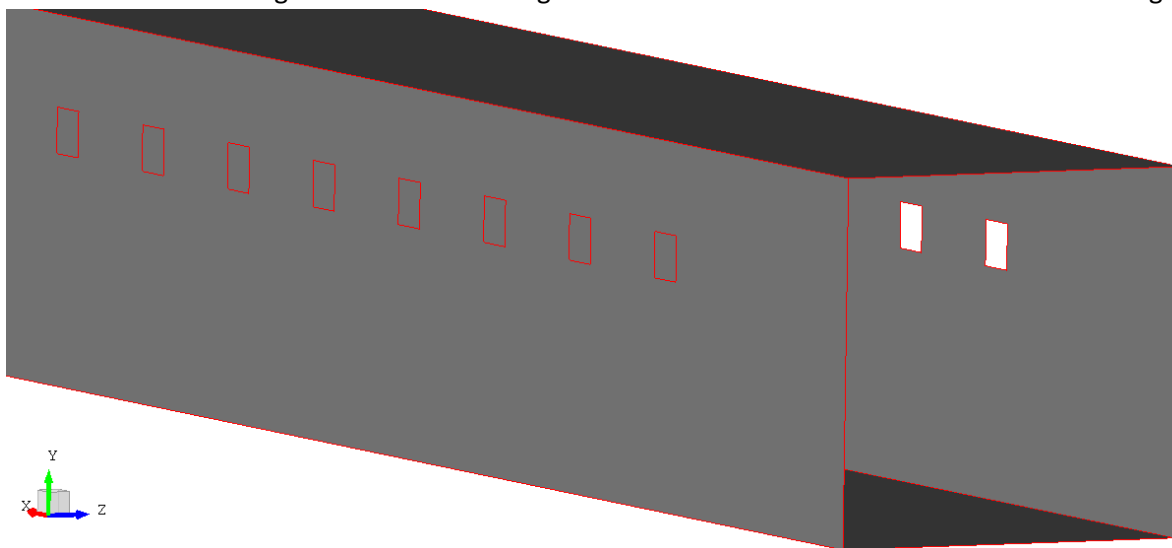
With the lines grouped, we can change the definitions of the fuselage surfaces to include gaps where the windows are.

5. Add the lines of the fuselage to the image.
6. Open the “Create/edit surfaces” tool.
7. Select the “Edit” bubble and pick the right side of the fuselage as the entity to be edited.
The surface definition should populate with four lines and a shape.

8. Add the lines of the RWINDOWS set to the definition by typing RWINDOWS in the “Surface Edges” entry box and hitting enter.



9. Hit “Apply.”
10. Click “OK” in the popup window, making now changes to the embedding geometry.
11. Repeat steps 7-10 for the left side of the fuselage, this time using LWINDOWS for the new edges.
12. Add the fuselage surfaces to the image. There should now be holes where the windows go.

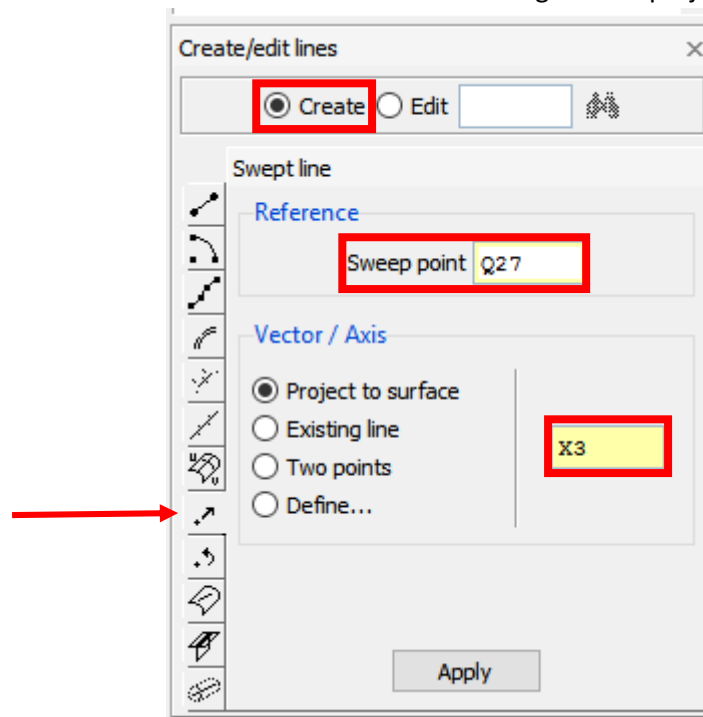


Creating a Cable

Induced currents on cable bundles are often of interest for HIRF problems. We will create a cable that goes nose to tail in the simple airplane. The cable will be represented with a line (a special EMA3D property will give it some 3D material properties).

The cable will be a straight cable that connects the nose to the tail:

1. Close all sets: **SETC ALL**
2. Open a new set, CABLE: **SETO CABLE**
3. Plot the lines of the cockpit and the fuselage.
4. Sweep the point of the nose onto the very back surface of the fuselage:
 - a. GUI:
 - i. Open “Create/edit lines”
 - ii. Select the “Swept lines” option on the left hand side.
 - iii. Leave the button on “Create.”
 - iv. Select the forward most point of the nose as the sweep point.
 - v. Select the “Project to surface” option.
 - vi. Select the back surface of the fuselage as the projection surface.



- vii. Click “Apply.”
 - b. Command line:
 - i. **SWEP point1 ! surf**, where point1 is the point to sweep and surf is the surface onto which point1 will be projected.
5. Close all sets.

Geometry Notes

The intent of this training module is to familiarize the reader with the CADfix and EMA3D interfaces. By this point, the reader should’ve seen most of the commonly used GUI aspects of CADfix and been introduced to the powerful CADfix command line. Manipulating the model (rotation, translation, and zoom) should be comfortable. The reader should understand the basics of controlling what entities are shown, or “plotted.” Set management should be beginning to make sense, too. Other training modules will teach techniques and methods required to simplify a complex model and make it suitable for a CEM.

Steps to EMA3D Simulation

The process of setting up an EMA3D simulation is outlined in the labelling of the toolboxes. This training module will follow the order laid out in the numbering of the toolboxes.

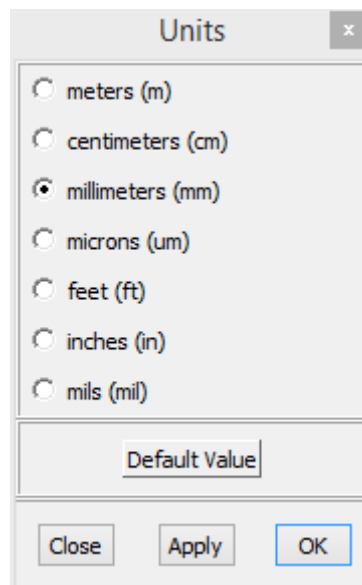
1) General Topics

In the top left of the screen, click “Tools->Toolbox->EMA3D_#1_GeneralTopics.” This will open the first toolbar for EMA3D, replacing the primitives opened at the beginning of this training. From this toolbar, you’ll find “MPIBlocks” and “EFldnit.” These are easy ways to change the blocking for parallel runs with MPI and define an initial E-field value, respectively.

This airplane model doesn’t need either of those features, so we’ll proceed to number two.

2) Units

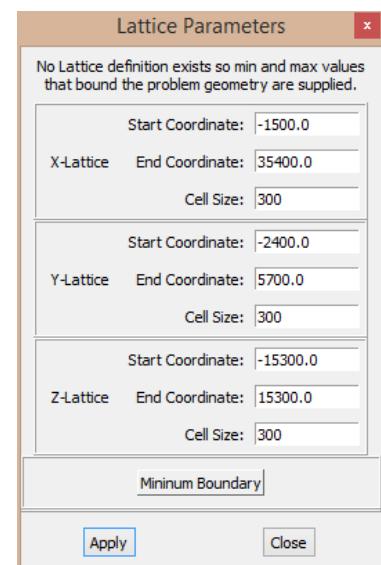
Click “Tools->Toolbox->EMA3D_#2_SpecifyUnits.” This opens the second toolbar. There’s only one tool, labelled “Units.” Click it and confirm millimeters as units. Click “OK.”



3) Define Lattice

Click “Tools->Toolbox->EMA3D_#3_DefineLattice.” There are two tools in this toolbox. The first, “ConLatt,” is used to construct 3D lattices. We’ll use this one because we have a 3D model. The second tool, “TwoDim,” is for 2D problems. Click “ConLatt.” The lattice will define the problem space to be simulated and the size of the finite difference elements.

The default values (based on all model geometry) are shown to the right. We will change the cell size to 50 mm for all directions. We will also change the start and end coordinates for all directions. We will add 4-5 cells of space (200-250 mm) between the airplane and the boundary of the problem. We can use small boundaries because we will use PML boundary conditions, which (when used correctly) don’t create many reflections.



To check the dimensions of the plane, type: **PRNT DIMS PLANE**

This command will print the dimensions of the set PLANE. Use the global minimums and maximums of the "Line ends." The section for "All cords" includes points and shapes that don't need to be meshed.

```
Fb> prnt dims plane
Coordinate system:   GLOB (CART)

656 coords (points and NURBS/FROGS control points)
All coords: Global: Min x   = -1441.342854
All coords: Global: Min y   = -16620.770139
All coords: Global: Min z   = -16620.770139

All coords: Global: Max x   = 35120.770139
All coords: Global: Max y   = 15650.000000
All coords: Global: Max z   = 15650.000000

All coords: Diameter       = 58477.288588

All coords: Centroid = 5243.338649  660.850890  -232.341927
All coords: Stan Dev = 8854.309549  2395.752770  3769.837313
All coords: Median  = 1648.156152  531.144190  -17.458902
All coords: 1st %   = -1167.126510  -2018.246651  -15668.782420
All coords: 99th %  = 34992.802745  15633.864615  15633.864615

Ff lines
Line ends : Global: Min x   = -1000.000000
Line ends : Global: Min y   = -2000.000000
Line ends : Global: Min z   = -15000.000000

Line ends : Global: Max x   = 35000.000000
Line ends : Global: Max y   = 5112.068000
Line ends : Global: Max z   = 15000.000000

Line ends : Diameter       = 47398.117170

Line ends : Centroid = 18937.304038  877.107285  4.756857E-07
Line ends : Stan Dev = 11817.459513  1939.562784  7086.527625

Shortest line           = U18
Minimum line length     = 1000.000000
Maximum line length     = 33000.000000
Average line length     = 6778.864691
Standard deviation      = 8220.640996

Largest dim. or coordinate = 63983.696861
Smallest useable tolerance = 7.627451E-02
Fb>
```

Be mindful that your maximum y coordinate will likely be different than the one shown above. We will want the start and end coordinates of the lattice to be even multiples of the step. If we increase the boundaries by about 200 mm and use the next greatest even multiple, the parameters are shown to the right. Click “Apply.”

Lattice Parameters

Start Coordinate: -1200

X-Lattice

End Coordinate: 35200

Cell Size: 50

Start Coordinate: -2200

Y-Lattice

End Coordinate: 5350

Cell Size: 50

Start Coordinate: -15200

Z-Lattice

End Coordinate: 15200

Cell Size: 50

Minimum Boundary

Apply

Close

4) Construct Source Geometry

Click “Tools->Toolbox->EMA3D_#4_ConstructSrcGeom.” An empty toolbox will replace the previous one. At this step, the source and detach geometry need to be made. This step is important for simulating lightning and ESD problems. However, this model will be used for HIRF. Sources for HIRF are typically plane waves, which don’t require constructing any new geometry for attachment or detachment.

There’s nothing to do in this toolbox for this model.

5) Meshing

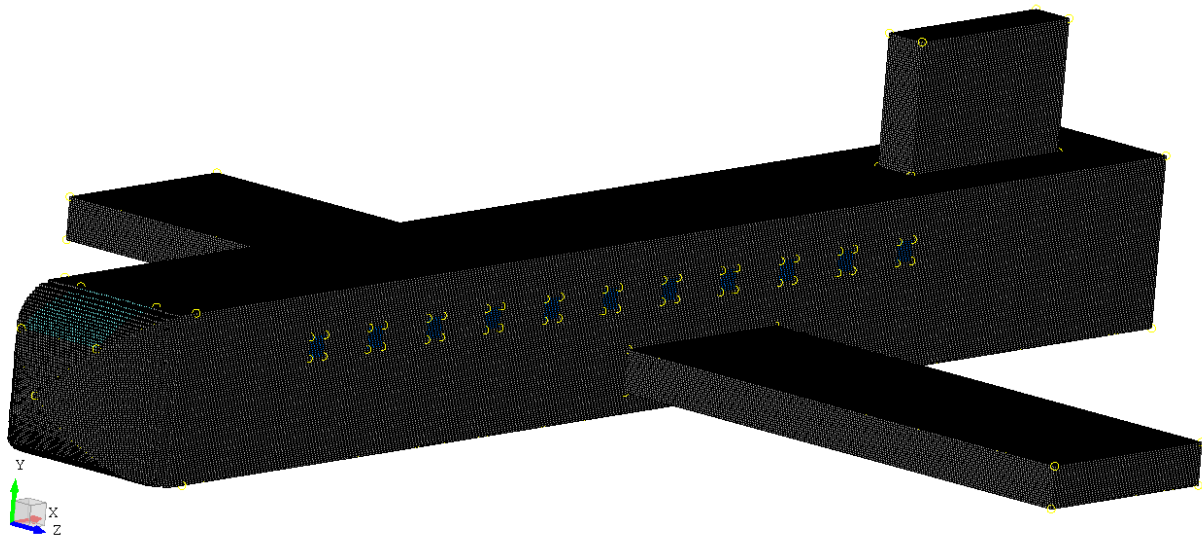
Navigate to “Tools->Toolbox->EMA3D_#5_MeshGeometry.” Three tools should now be visible in the toolbox on the right side of the screen. Click the first, “Setup.” This prepared the model for being meshed.

To mesh everything, click the second tool: “Mesh.” Alternatively, you can type **MESH** into the command line.

On larger and more complex models, it is often advantageous to mesh only a portion of the model to reduce computation time. To do this, type: **MESH entity** into the command line. The **entity** can be a set (DETACH, LEFTWING, etc.), or a piece of geometry (V11, U30, etc.).

Mesh the entire model: **MESH**

Plot the meshed twinkles with the assigned color: **PLOT TWI ALL ASGD**



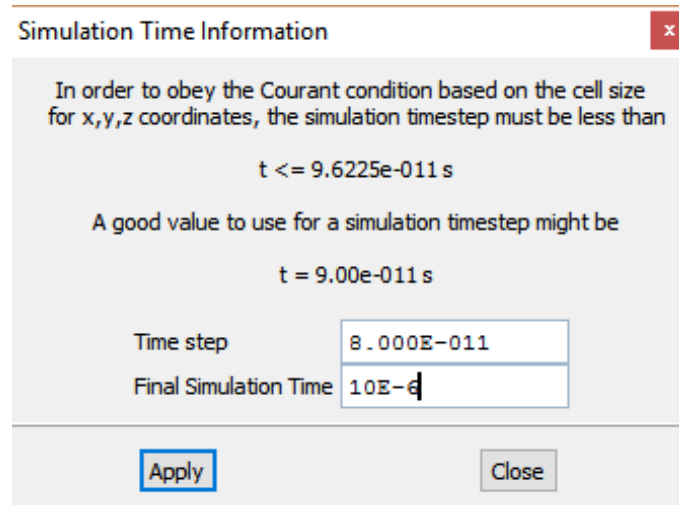
For practice, mesh just the left wing: **MESH LEFTWING**.

Repaint the picture. Just the left wing should be showing now.

We need the entire model, so re-mesh the entire model: **MESH**. Repaint the picture to confirm the entire plane is showing.

6) Time step

Click “Tools->Toolbox->EMA3D_#6_DefineTimeStep.” Two tools should be in the toolbox. Click the first button, “TimeStep,” in the toolbox. A window will pop up. Enter 8.0e-11 for the time step and 10e-6 for the final simulation time.



The dialog box titled "Simulation Time Information" contains the following text and input fields:

In order to obey the Courant condition based on the cell size for x,y,z coordinates, the simulation timestep must be less than

$$t \leq 9.6225e-011 \text{ s}$$

A good value to use for a simulation timestep might be

$$t = 9.00e-011 \text{ s}$$

Time step:

Final Simulation Time:

Buttons: Apply, Close

The time step was chosen to satisfy the courant condition without being within 10% of the courant value. The margin will provide a more numerically stable simulation.

Click “Apply.”

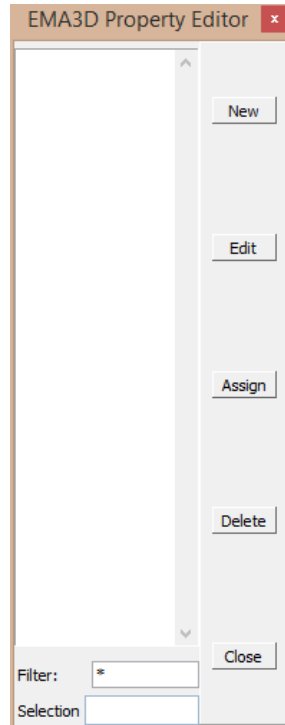
The “MagTSteps” button sets the permittivity scaling for the simulation. This model won’t use this feature. Permittivity scaling enforces a quasi-magnetostatic assumption to increase the permittivity of free space. Scaling the permittivity allows EMA3D to increase the time step without violating the courant condition. The nature of HIRF problems is not magnetostatic, so no permittivity scaling will be used.

7) Background

Using EMA3D, the background material can be changed. However, we will be using the default values. This tool isn’t needed for this model.

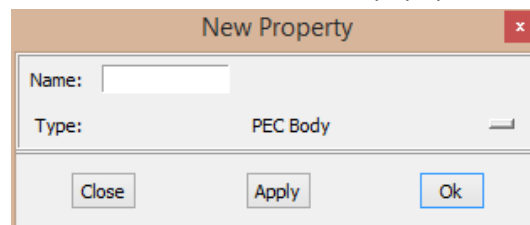
8) Define Properties

To run a simulation, EMA3D needs parameters for material properties. To define properties, click “Tools->Toolbox->EMA3D_#8_DefineProperties.” Only one tool will occupy the toolbox: “PROPERTY.” Click it. A window will pop up.



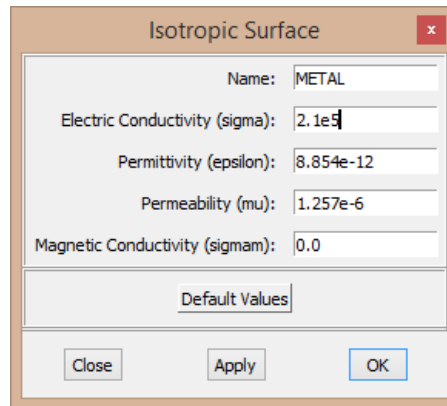
Three material properties will be needed for the simple airplane: conductive metallic surfaces, non-conductive glass surfaces, and perfectly electrically conductive lines.

1. To make a new material, click “New.” A window will popup:



2. Enter “METAL” as the name.
3. Select “Isotropic Surface” for the type.

- Click "Apply." Another window will popup. Fill in the values as follows:



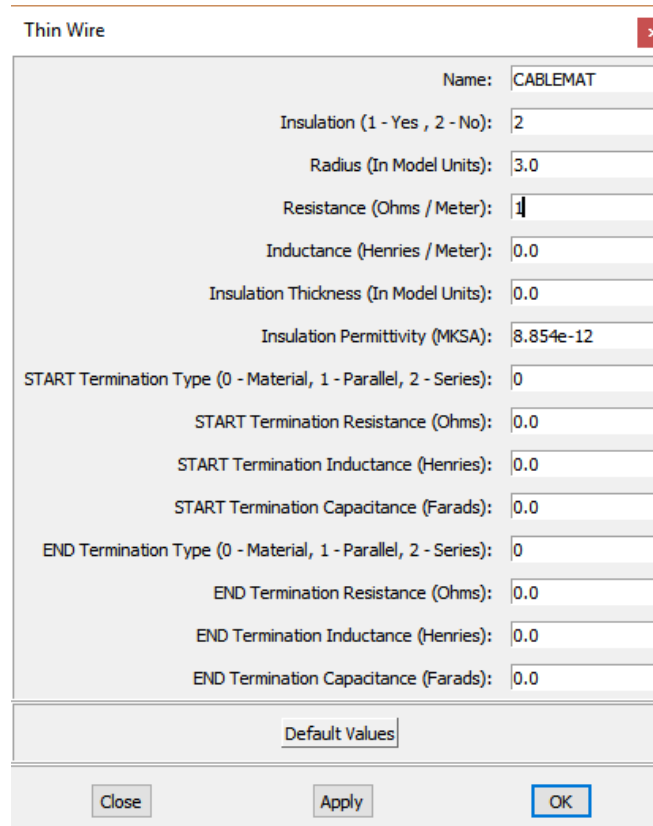
The "Isotropic Surface" dialog box is shown with the following values:

Property	Value
Name	METAL
Electric Conductivity (sigma)	2.1e5
Permittivity (epsilon)	8.854e-12
Permeability (mu)	1.257e-6
Magnetic Conductivity (sigmam)	0.0

Buttons: Close, Apply, OK

The conductivity is that of a typical airplane aluminum scaled for use with 500 mm cells.

- Click "OK." METAL has been defined as a material.
- In the "New Property" window, enter CABLEMAT as the name and choose "Thin Wire" as the type.
- Enter the values shown in the picture. Note that the cable is assumed to be 3 mm in radius (which is very big) and the resistance is assumed to be 1 ohm/meter (which is very big). The high resistance was chosen so that the induced currents would taper to zero more quickly.



The "Thin Wire" dialog box is shown with the following values:

Property	Value
Name	CABLEMAT
Insulation (1 - Yes, 2 - No)	2
Radius (In Model Units)	3.0
Resistance (Ohms / Meter)	1
Inductance (Henries / Meter)	0.0
Insulation Thickness (In Model Units)	0.0
Insulation Permittivity (MKSA)	8.854e-12
START Termination Type (0 - Material, 1 - Parallel, 2 - Series)	0
START Termination Resistance (Ohms)	0.0
START Termination Inductance (Henries)	0.0
START Termination Capacitance (Farads)	0.0
END Termination Type (0 - Material, 1 - Parallel, 2 - Series)	0
END Termination Resistance (Ohms)	0.0
END Termination Inductance (Henries)	0.0
END Termination Capacitance (Farads)	0.0

Buttons: Close, Apply, OK

- Click "OK."
- In the pop up, click "OK;" there are no parameters to change.

You may have noticed that no material was created for the windows. The windows will not get material assignments. Without a material, EMA3D will ignore those surfaces and they will act like apertures. This is a reasonable approximation for how a window acts in HIRF environments.

The materials we need have been defined. Now we need to assign them to the parts. The cable will get the CABLEMAT material. The rest of the plane will be assigned METAL properties.

Notice we don't have a good way of referring to the metallic portions of the plane. The metallic portions are the fuselage, cockpit, leftwing, rightwing, and tail. We could either assign the same property to each piece (which would be tedious), or we could make a better way to refer to just the metallic portions. In other words, make a set that holds just the metallic portions of the plane.

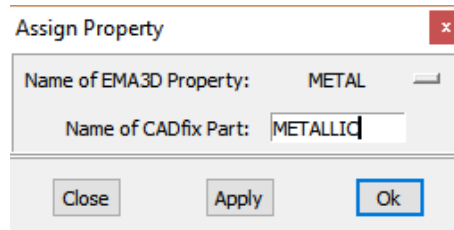
As mentioned earlier, the metallic portion can be made by adding up the fuselage, wings, and tail. The command to do this would be:

SETA METALLIC FUSELAGE COCKPIT LEFTWING RIGHTWING TAIL

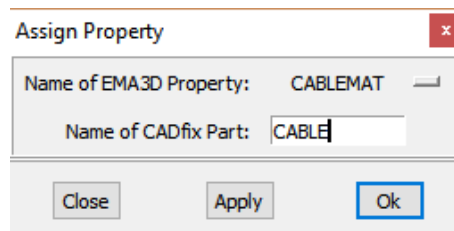
It's worth noting that there is a limit to how many additional arguments can be used with **SETA** and all other set commands, so with a larger model this option will require many lines. Alternatively, we know that the metallic portions of our model include anything that isn't a window. We can create a duplicate of the PLANE set named METALLIC and remove the windows from METALLIC:

**SETA METALLIC PLANE
SETR METALLIC WINDOW
SETR METALLIC APERTURES**

Return to the EMA3D Property Editor window. If you closed it, open it again using the "PROPERTY" tool in the toolbox. Click "Assign." The Assign Property window will pop up. Choose "METAL" from the dropdown of available properties. Enter "METALLIC" as the CADfix part. Click "Apply."



After clicking "Apply," change the property to "CABLEMAT." Enter CABLE as the CADfix part. Click "OK"



to close apply the change and close the window.

All material definitions are now applied. Close the property editor.

9) MHarness Cables

Some EMA3D purchases include MHarness. MHarness is used for complex pipe and cable routing and probing. The cable in this model is simple enough that MHarness isn't needed. The MHarness tool deserves its own tutorial and is beyond the scope of this training.

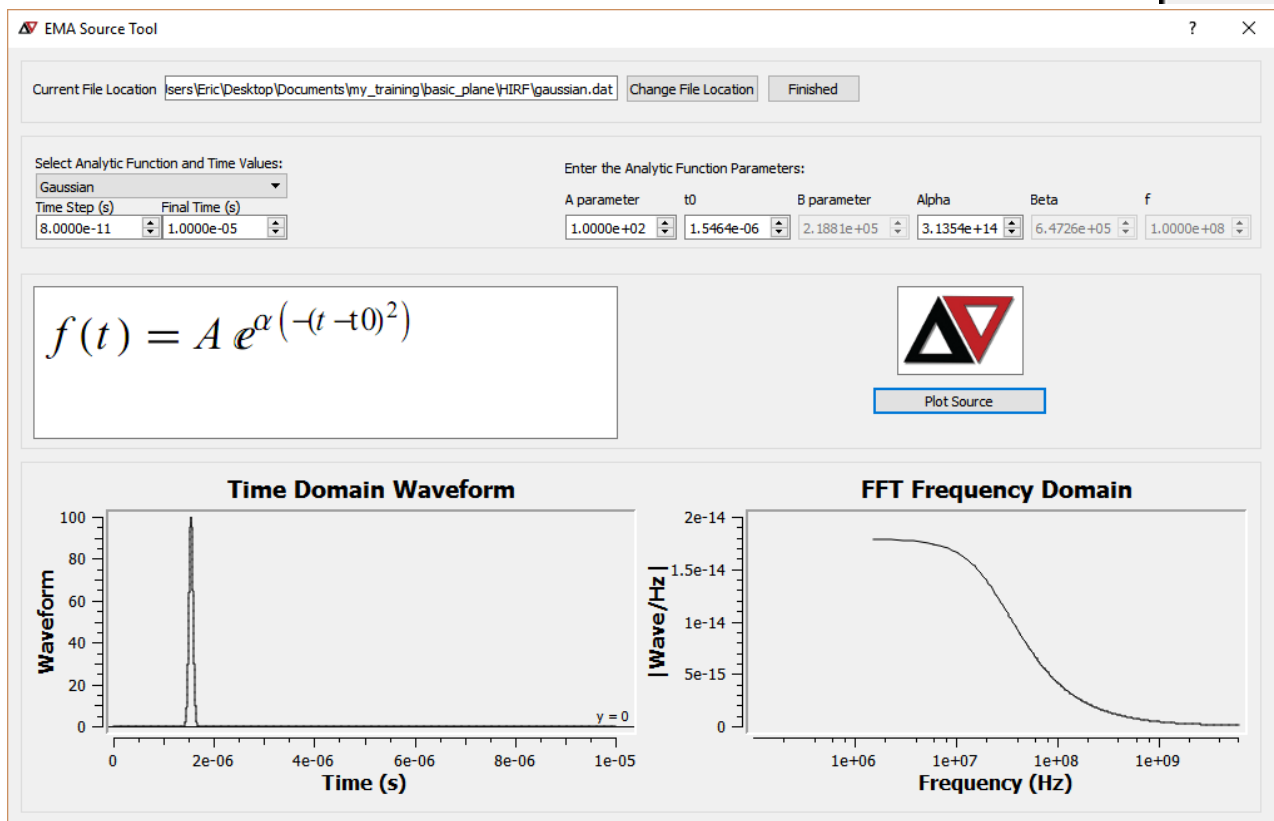
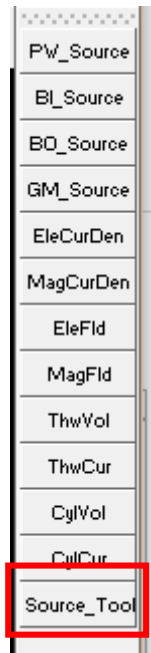
10) Define Source

Click “Tools->Toolbox->EMA3D_#10_DefineSources.” The toolbox will now hold many tools. Each tool is used for different kinds of sources. Sources can be plane waves, magnetic current densities, among many others. We will use a plane wave source. Before the source definition is created, we need to make a source waveform file.

Click on “Source_Tool.” It is the last tool in the current toolbox. A new window will open. It is the EMA3D source tool. Use the “Gaussian” option in the “Select Analytic Function and Time Values:” dropdown. Change the final time to 10e-6 seconds. Fill in the rest of the values based on the picture below.

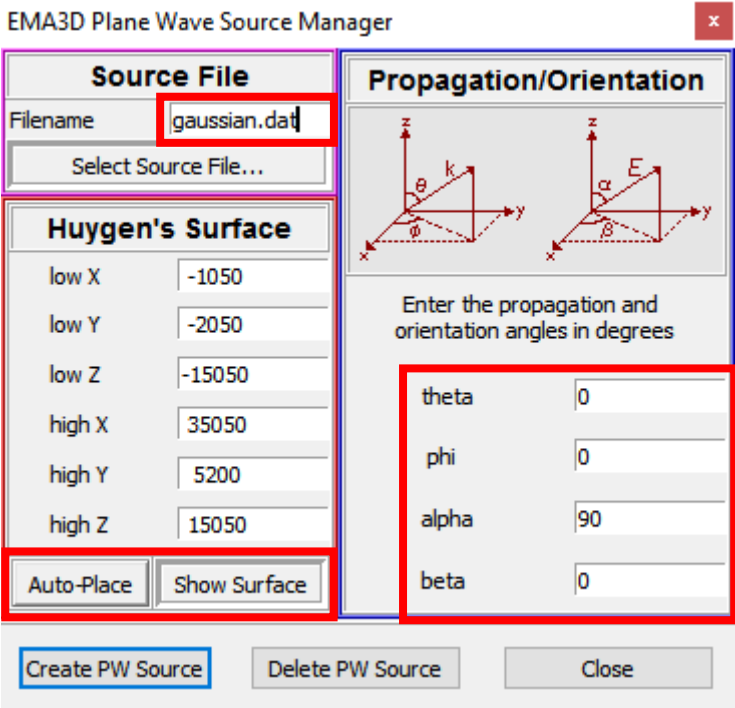
The peak time parameter was chosen so that the first value in the source file has a value of zero. EMA3D will throw an error if a source file doesn’t start with zero.

Note the name of the file created. In the example the name is “gaussian.dat.” Click “Finished.” In the popup, click “OK.”



To define a plane wave source:

1. Click on the “PW_Source” button
2. Enter “gaussian.dat” as the source filename
3. In the “Huygen’s Surface” section, click the “Auto-Place” button.
4. Click the adjacent “Show Surface” button and ensure that no parts of the plane extend past the Huygen’s surface.
5. For a broadside propagation vector, enter zero degrees for theta and phi.
6. To align the E-field with the long axis of the aircraft, enter 90 degrees for alpha, zero degrees for beta.



The image shows the "EMA3D Plane Wave Source Manager" dialog box. It is divided into several sections. The "Source File" section at the top left has a "Filename" field containing "gaussian.dat" and a "Select Source File..." button. Below this is the "Huygen's Surface" section, which contains input fields for "low X", "low Y", "low Z", "high X", "high Y", and "high Z" with values -1050, -2050, -15050, 35050, 5200, and 15050 respectively. At the bottom of this section are "Auto-Place" and "Show Surface" buttons. To the right is the "Propagation/Orientation" section, which features two 3D coordinate system diagrams. The first diagram shows a vector k with angles θ and ϕ . The second diagram shows a vector E with angles α and β . Below the diagrams is the text "Enter the propagation and orientation angles in degrees". A table of input fields follows: "theta" (0), "phi" (0), "alpha" (90), and "beta" (0). At the bottom of the dialog are three buttons: "Create PW Source", "Delete PW Source", and "Close".

Source File	
Filename	gaussian.dat
Select Source File...	

Huygen's Surface	
low X	-1050
low Y	-2050
low Z	-15050
high X	35050
high Y	5200
high Z	15050
Auto-Place Show Surface	

Propagation/Orientation	
Enter the propagation and orientation angles in degrees	
theta	0
phi	0
alpha	90
beta	0

Create PW Source	Delete PW Source	Close
------------------	------------------	-------

7. Click “Create PW Source.”

11) Boundary Conditions

Click “Tools->Toolbox->EMA3D_#11_SelectBdyConds.” Click the only tool in this toolbox. A window will appear. Check “ALL” on the “PML” row. The PML (Perfectly Matched Layer) option is an absorbing boundary condition has been used with HIRF simulation models with great results.



Leave the bottom three entry fields as the defaults. Click “Apply.”

12) Probes

For a simulation to be useful, it needs to write data. Probes determine what data is written as outputs from the simulation. For this simple airplane, we will make bulk current probes near the front and rear of the cable. We will also make e-field probes in space near the bulk current probes.

Creating any probe will require going into: “Tools->Toolbox->EMA3D_#12_DefineProbes.” The toolbox should now have many tools in it.

Bulk Current Probe Geometry

The first probe will be a bulk current probe on the cable, near the nose of the aircraft. Bulk current probes work by integrating the magnetic fields around rectangular region defined by two points. The two points are the opposite corners of the rectangle. To measure current on a cable, only one point is needed (the magnetic fields are integrated around that point). The should lie exactly on the mesh grid.

1. Plot the cockpit lines in one color, then the cable in another:
PLOT L COCKPIT
PLUS L CABLE G
2. Open a set to contain all probes: **SETO PROBES**
3. Open another set to contain just the forward current probe: **SETO CURPROBE1**
4. Using your preferred method, create a point at (-200, 0, 0): **PNT ! -200 0 0**
5. Close the forward current probe set: **SETC CURPROBE1**
6. Open a set for the rear current probe: **SETO CURPROBE2**

- Using your preferred method, create a point at (34000, 0, 0): **PNT ! 34000 0 0**
- Close the set for the rear current probe: **SETC CURPROBE2**

Bulk Probe Properties

To apply the bulk current probe definitions to the points, change the toolbox back to the Define Probes toolbox: Tools-Toolbox_EMA3D_#12_DefineProbes.

- Click “BulkCurr” in the toolbox.
- A window will pop up. Fill it out as shown:

Bulk Current Probe Set Name:	CURPROBE1
First Bounding Point Name:	Q406
Second Bounding Point Name:	
Current Type:	<input checked="" type="radio"/> Electric <input type="radio"/> Magnetic
Start Time (Seconds):	0.0
Stop Time (Seconds):	10e-6
Step Time (Seconds):	8e-11
Normal Direction:	<input checked="" type="radio"/> X <input type="radio"/> Y <input type="radio"/> Z
Skip Factor:	10000
<input type="button" value="OK"/> <input type="button" value="APPLY"/> <input type="button" value="CANCEL"/>	

- The bounding point is the point in the set CURPROBE1. To find its name, either:
 - Add the labelled point to the image: **PLUS PA CURPROBE1**
 - Print the point in RWPROBE: **PRNT P CURPROBE1**
- Enter the point name in both of the bounding point fields.
- Click “APPLY.”
- Change the set name field to CURPROBE2.
- Change the bounding point to the point in the CURPROBE2 set. Leave the time fields and normal direction unchanged.
- Click “OK.” The bulk current probes have been defined.

E-Field Probe Geometry

The e-field probes will also be located at a point. Create these points:

- With the PROBES set still open, copy the CURPROBE1 up 400 mm. Name the set EPROBE1:
COPY CURPROBE1 EPROBE1 TRA 0 400 0
- Repeat step one for CURPROBE2, creating EPROBE2. Use the same translation.

E-Field Probe Properties

- Click the “New2” button in the Probes toolbox.
- Enter EPROBE1 as the probe set name.
- Pick or enter the point name for EPROBE1.
- Click the checkbox for “Ex-Field Physical Property Name”

5. Leave the radial buttons unchanged.
6. Enter the start time as 0.0
7. Enter the stop time as 10e-6
8. Enter the step time as 8e-11

New Probe Creation Tool

Step (1) If new probe points or nodes are already in a set, enter set name.

New Probe Set Name:

If the new probe points are not in a set, enter the desired set name above and select the points:

Pick Points

Step (2) Select the Physical Property for this New Probe set

Ex-Field Physical Property Name:	EXCP	<input checked="" type="checkbox"/>
Ey-Field Physical Property Name:	EYCP	<input type="checkbox"/>
Ez-Field Physical Property Name:	EZCP	<input type="checkbox"/>
Hx-Field Physical Property Name:	HXCP	<input type="checkbox"/>
Hy-Field Physical Property Name:	HYCP	<input type="checkbox"/>
Hx-Field Physical Property Name:	HZCP	<input type="checkbox"/>

Thin Wire, Cylinder, or MHARNESS Segment Current:
Thin Gap or Seam Voltage

TW, CY, MH, TG, SM ☐

Step (3) Define the New Probe output Parameters.

Numerical Results:	<input checked="" type="radio"/> Yes	<input type="radio"/> No
Prony Results:	<input type="radio"/> Yes	<input checked="" type="radio"/> No
Time Results:	<input checked="" type="radio"/> Yes	<input type="radio"/> No
Frequency Results:	<input type="radio"/> Yes	<input checked="" type="radio"/> No
Transfer Function Results:	<input type="radio"/> Yes	<input checked="" type="radio"/> No
Start Time (Seconds):	<input type="text" value="0.0"/>	
Stop Time (Seconds):	<input type="text" value="10e-6"/>	
Step Time (Seconds):	<input type="text" value="8e-11"/>	
	<input type="text"/>	
	<input type="text"/>	
	<input type="text"/>	
	<input type="text"/>	

APPLY **CANCEL**

9. Click "Apply."
10. Repeat steps 2-9 for EPROBE2.

EMA3D Review Tool

Change the toolbox to the Review Tool: "Tools->Toolbox->EMA3D_ReviewTool." Click the "Review" tool. A window will pop up. It should look like:

Click “Create EMA3D Input File.” Another popup window will appear. Nothing needs to be changed; click “OK.”

EMA3D Review Tool - Version 4.0

CADfix Model Name: "basic_plane_HIRF.fbm"		Approximate EMA3D Model Size (MegaBytes): 2374	
GENERAL	UNITS	LATTICE	TIME STEPS
'Num of Proc: 1'	'mm'	'729 x 152 x 609'	'Con: 8.000e-011'
REVIEW	REVIEW	REVIEW	REVIEW
MATERIALS	SOURCES	BOND. COND.	PROBES
'Num of Prop: 3'	'Num of Srcs: 2'	'PML'	'Num of Prob: 4'
REVIEW	REVIEW	REVIEW	REVIEW
REFRESH		RESTORE	Create EMA3D Input File
			CLOSE

When the emin file is created, click “OK” again. Close the review tool window.

Running EMA3D

Change the toolbox to the Run EMA3D toolbox: “Tools->Toolbox->EMA3D_Run.” Click the only button, labelled “Run_EMA3D.” The EMA3D window will appear, although it might take a moment.

Click the “Run New EMA3D Simulation” radio button.

Click “Browse” to find the emin file created earlier.

Click “Run EMA3D.” This simulation will likely take a day or two.

Run New EMA3D Simulation ☒ Analyze Existing Results ☐ Process Another File

EMA3D Input File: desktop\Documents\my_training\basic_plane\basic_plane.emin **Browse**

Run EMA3D Analyze Results

Progress: 0%

Current Item: Press Run EMA3D to Begin Processing.

☒ Run Single Processor ☐ Run Multiple Processor **Open CADfix**

Probe Type	Numical or Prony	Time, Freq, or TF	Filename	# of Columns

Post-Processing

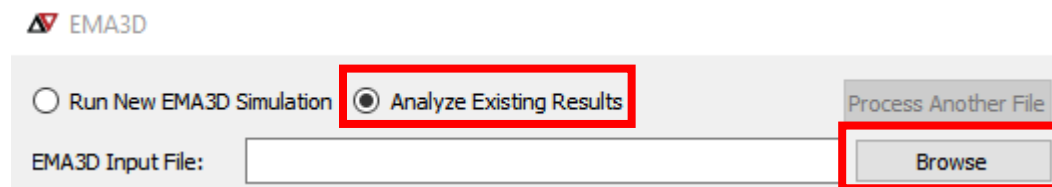
There are many ways to post process data output by an EMA3D simulation. The EMA3D GUI has built in options for plotting probe outputs. However, large models have hundreds of probes and it is not practical to rely on GUI's for such large projects. It is common to build application specific, high volume scripts and tools for large projects. Many EMA3D probes output data as .dat files. These files, if in time domain, have time values in the first column and data values in the subsequent columns. Columns are separated by whitespace. These files are easy to use with MATLAB, Mathematica, Python, etc.

Not all EMA3D probes output .dat files. Probes that have visual output will often use postscript or .emout file types. For example, animation probes output emout files and slice probes output postscript files. This training document didn't have any such files. Only .dat files were generated by the simple simulation.

EMA3D GUI Processing

Once the data files have been created, you can double click them in the Probe window to access the data. Because all of the data made by this simulation only have one column, only one possible plot in the data window. Double click a column to see its plot. Probes can be plotted one on top of another. Probes can be plotted while the simulation is still in progress.

If the simulation has finished, use the "Analyze Existing Results" button in the EMA3D GUI. Use the "Browse" to find the simulated emin file and click "Open."



49

[illegible]

Double clicking an entry in the probe list (upper of the two tables on the left side) will populate the lower table with details and column numbers of the selected probe. Instead of double clicking, you can click the “Analyze Probe” button.

☒ Run Single Processor
 ☐ Run Multiple Processor

Open CADfix

Number in Prony Tran vector is: 0

Number in Prony Comp vector is: 0

Number in Bulk Probe vector is: 2

Number in Voltage Difference Probe vector is: 0

Number in MH Probe vector is: 0

	Probe Type	Numerical or Prony	Time, Freq, or TF	Filename	# of Columns
1	Source File	gaussian.dat			
2	New Probe	NUMER	TIME	nprnumertime1...	1
3	New Probe	NUMER	TIME	nprnumertime2...	1
4	Bulk Current	NUMER	TIME	bpr1.dat	1
5	Bulk Current	NUMER	TIME	bpr2.dat	1

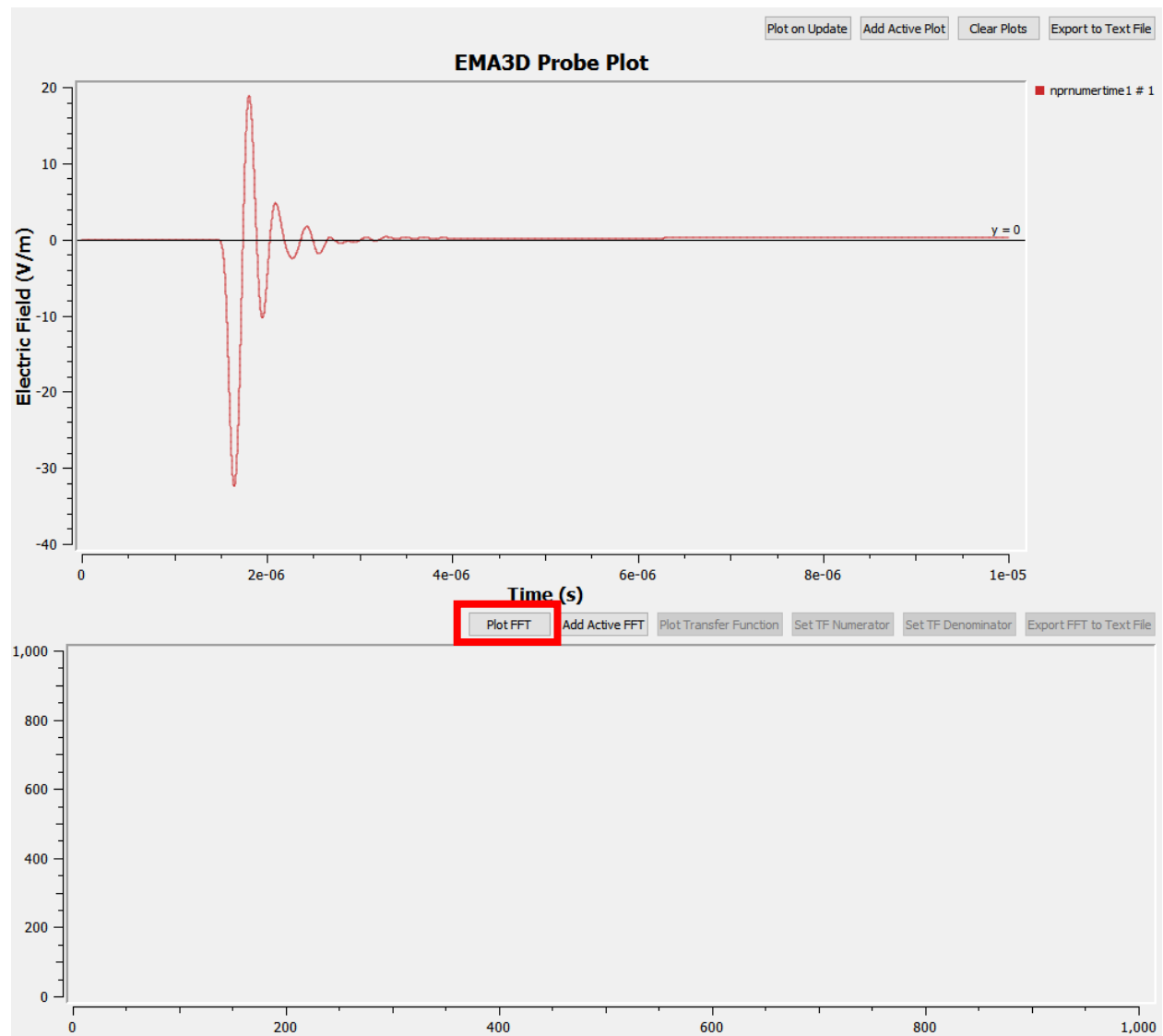
Clear Probe Table
Analyze Probe

	Probed Property	Node/X-offset	Y-offset	Z-offset
1	EX	21	53	305

Plot on Update
Add Active Probe

Currently Active Probe: The active row is # 2, which is probe nprnumertime1.

All of the probes used in this demo only have one column, so the lower table will only have one row. Double clicking an entry in the lower table will plot the entry on the right side of the window. Instead of double clicking, you can use the “Plot on Update” button.



With a probe plotted in the time domain, you can use the “Plot FFT” button to plot the frequency components of the probe. Once the FFT has been calculated, you can use the “Set TF Numerator”/”Set TF Denominator” buttons to assign the currently plotted FFT as the numerator/denominator of a transfer function. Typically, the source will be the denominator. With both the numerator and denominator assigned, you can plot a transfer function. You can also export the time domain, frequency domain, or transfer function data as text files using the other buttons in the GUI.