CADfix and EMA3D Training

Part One

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.

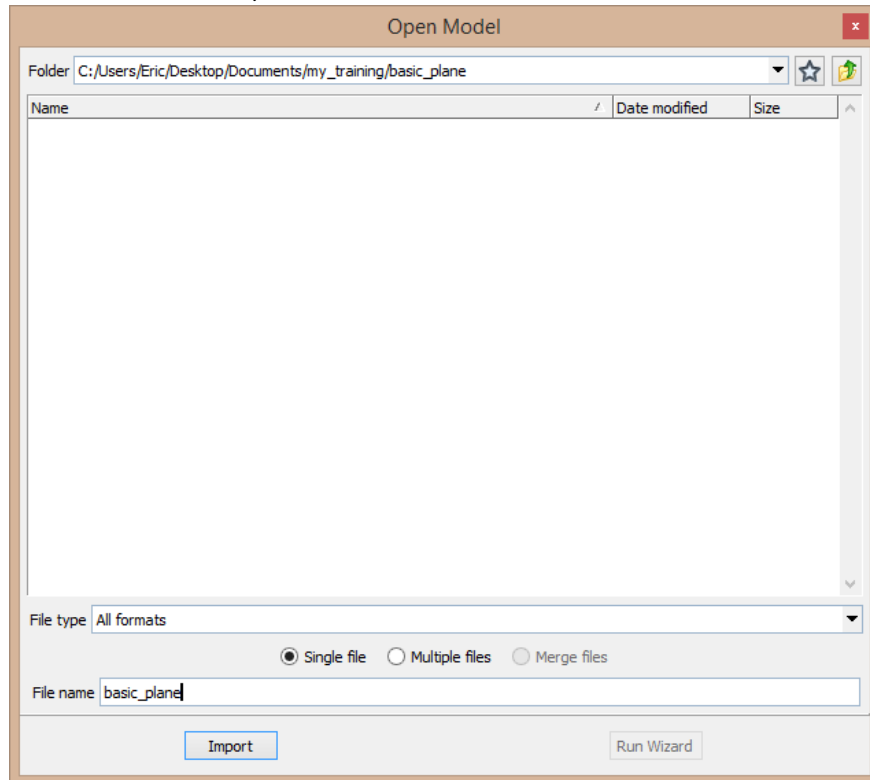Note: This training was made using CADfix 10 SP2.0

Getting Started

1. Open CADfix
2. Click the "Open Model" button at the top left of the screen
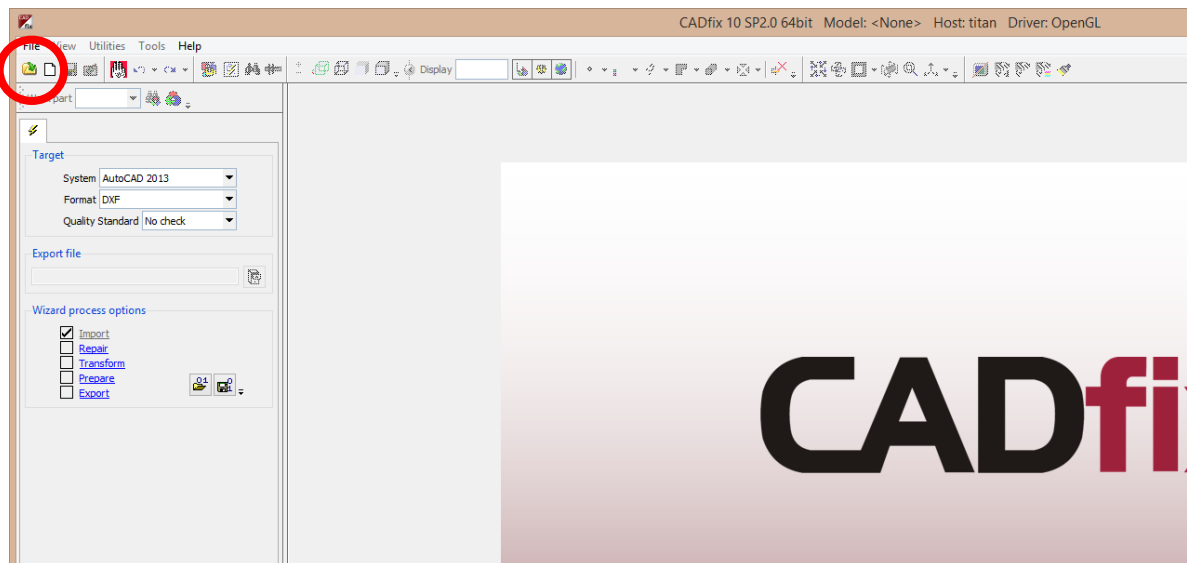
3. Navigate to a folder to hold the files created in this training

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."
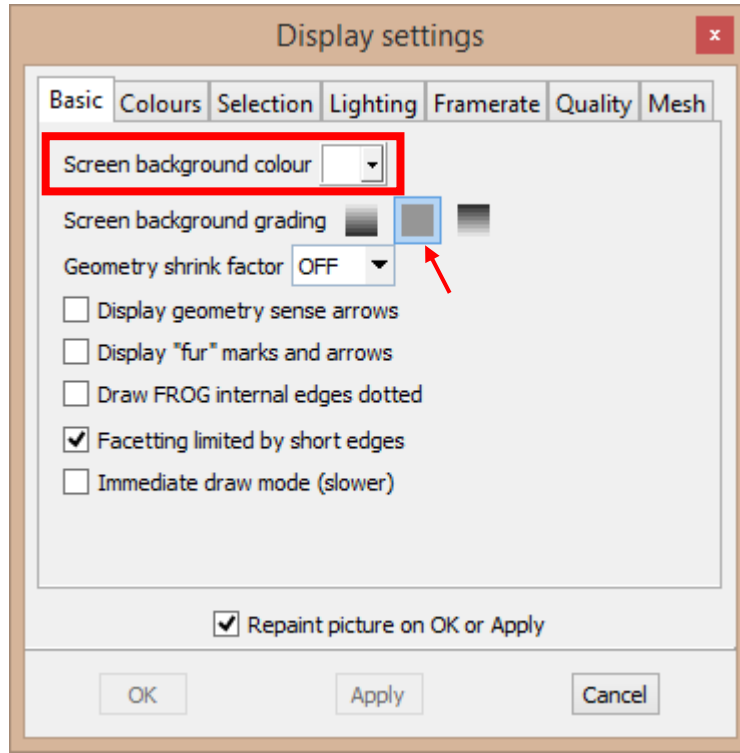
b. Alternatively, locate the command line at the bottom of the screen and type:
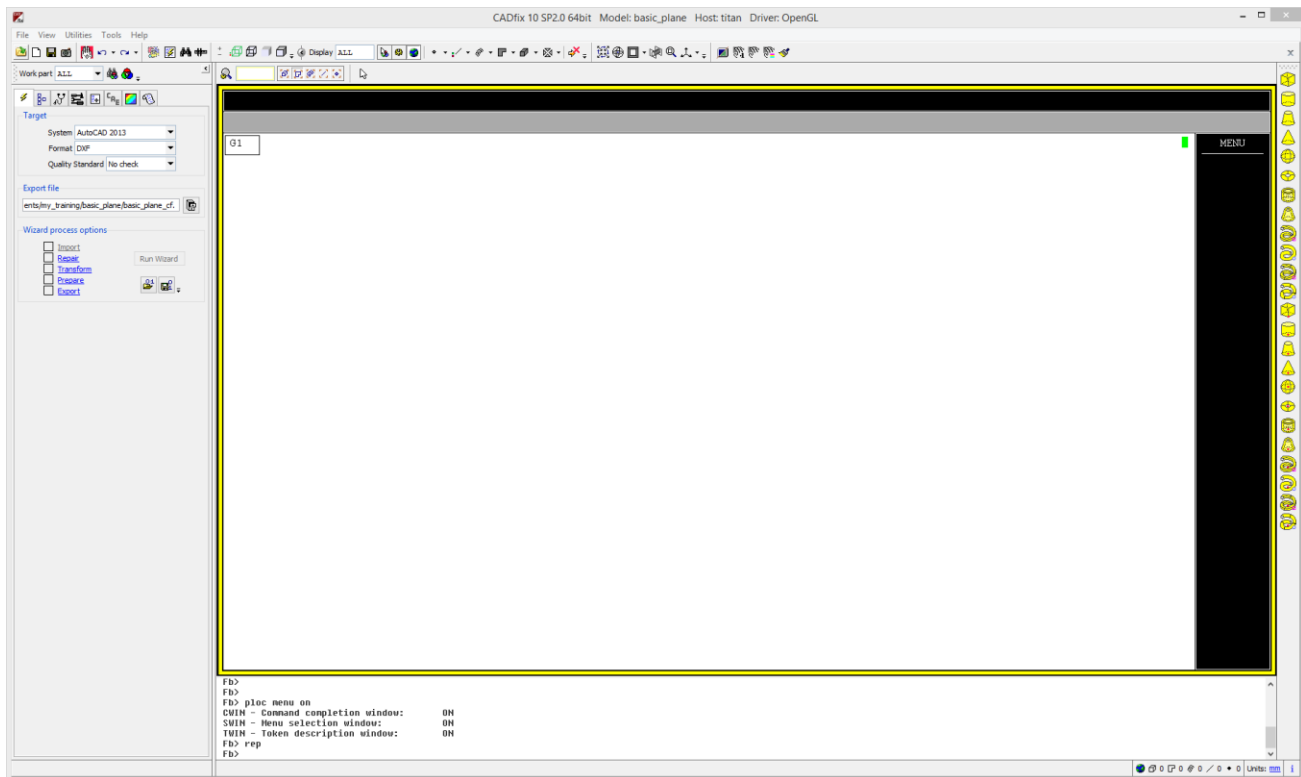   **PLOC BGND W**
    Then hit enter. To see the change, type **REP** and hit enter. This will repaint the image. "PLOC" accesses the display settings, "BGND" accesses the background settings, "W" is white

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 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.
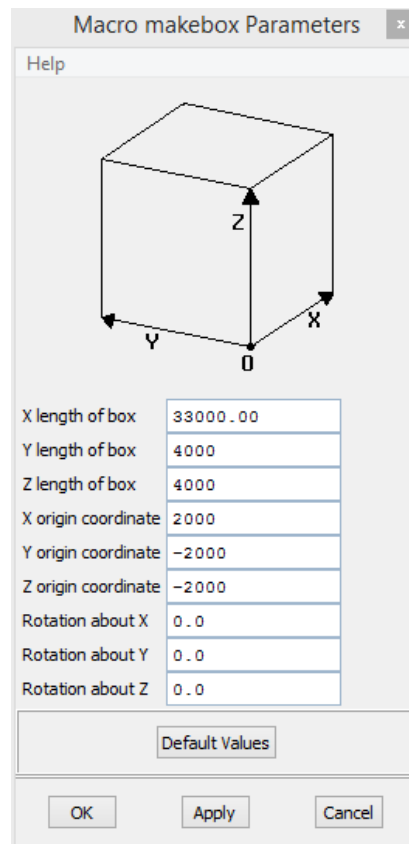3. Create lower level geometries to make the box using the command line.

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.

2. Click the "Cube" tool. It should be the first tool in the toolbox
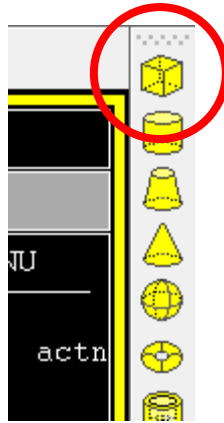
3. The box will be 33 m long, anticipating 2 m of cockpit. Fill in the parameters as follows (units are in millimeters by default):


Macro makebox Parameters

Parameter	Value
X length of box	33000.00
Y length of box	4000
Z length of box	4000
X origin coordinate	2000
Y origin coordinate	-2000
Z origin coordinate	-2000
Rotation about X	0.0
Rotation about Y	0.0
Rotation about Z	0.0

4. Type **PLOT SI ALL** and hit enter.
   a. "PLOT" tells CADfix to clear the image, then add something to the image, i.e. to plot it
   b. "SI" is the type code for shaded surfaces
   c. "ALL" tells CADfix what part of the model to plot, in our case "all" or everything
5. 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.
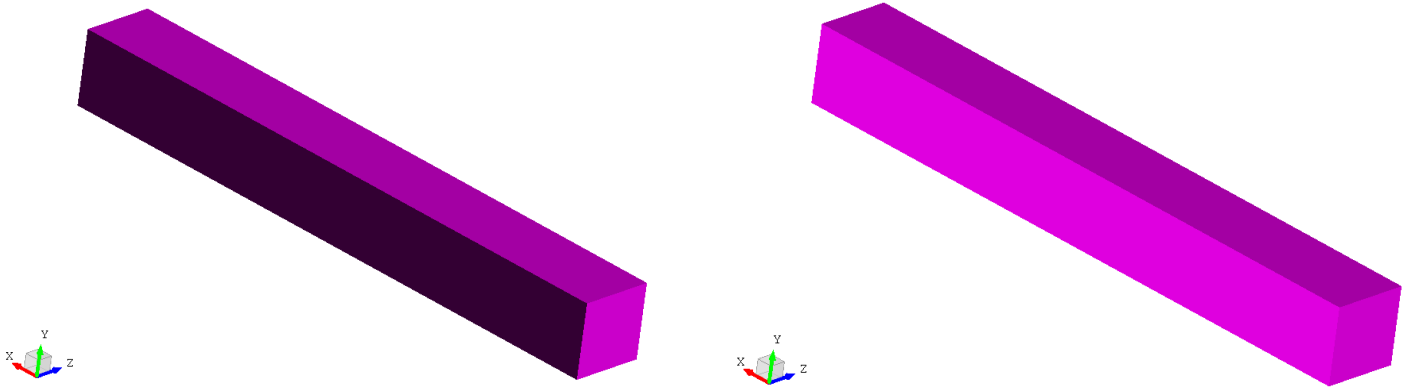
Original image on left, image after changing shading setting on right

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.
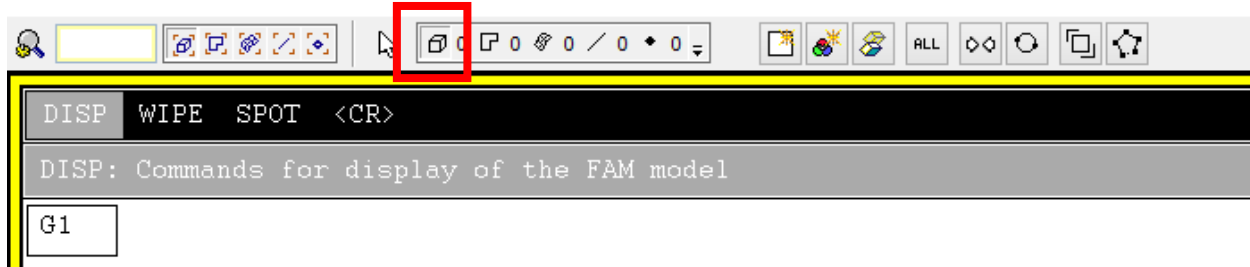
b. Pick the box you just made.
c. With the box still selected, type **SETA FUSELAGE SELE**. This tells CADfix to add the selected items to a set named FUSELAGE.
d. Right now, FUSELAGE only contains the body. We want all of the points, lines, and surfaces, too. Type **COMP FUSELAGE**. This tells CADfix to complete the set, adding the lower level components.
   (for more information on set management, see "CADfix Notes")
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."
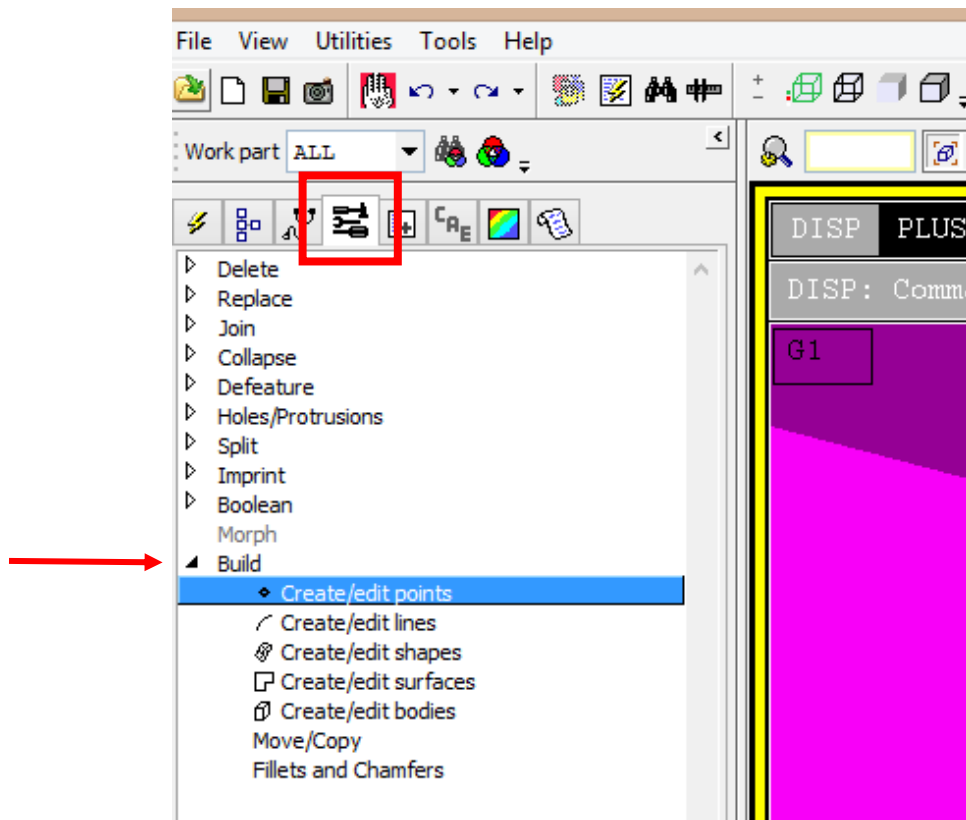
   c. Create a point at the origin using the tool by clicking "Apply."
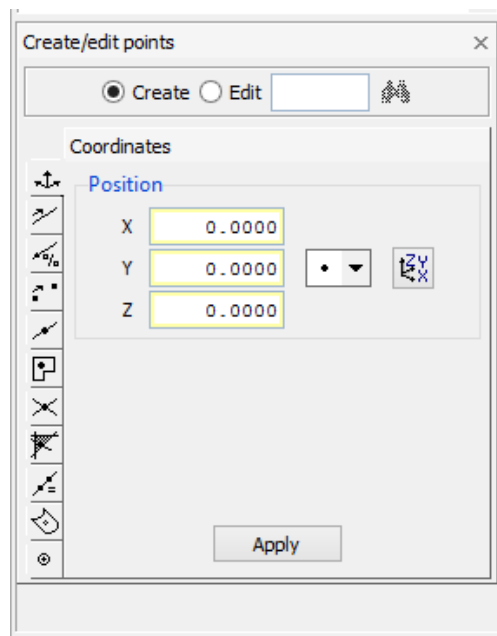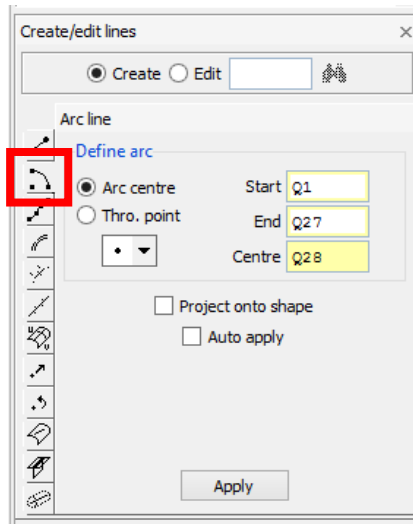
3. Add the cockpit points to the image: **PLUS PA COCKPIT**. "Plus" adds items to the image without removing other entities from the image. The "PA" means points annotated with names.
4. Create a midpoint between corners of the square face by doing one of the following:
   a. Command line: **PNT ! pnt1 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."
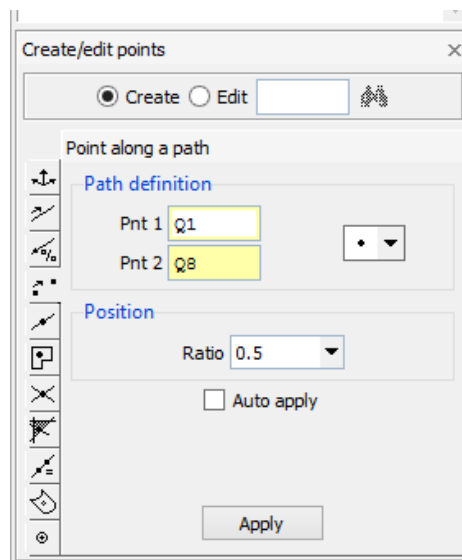

5. Create arcs connecting the corners of the square with the point at the origin. This can be done with command line or GUI:
   a. Command line:
      i. **LINE ! pnt1 pnt2 centerpnt**, where "pnt1" and "pnt2" are arc endpoints and "centerpnt" is the center of the arc. The center point is the midpoint 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.
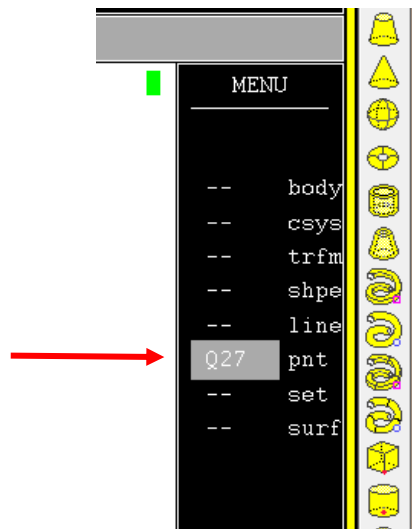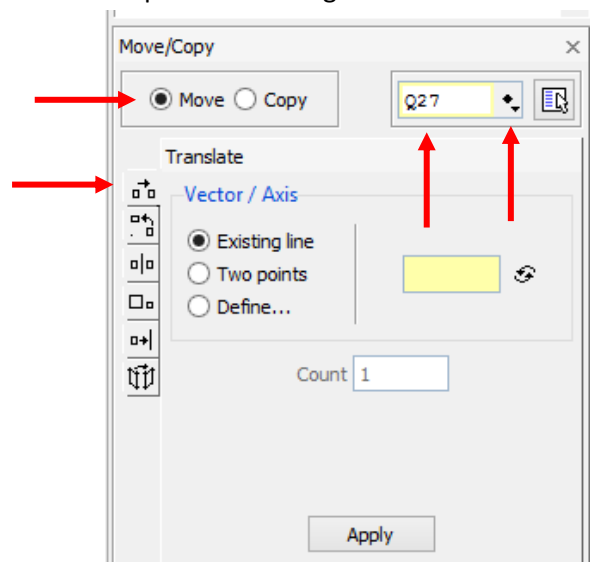

    b. GUI:

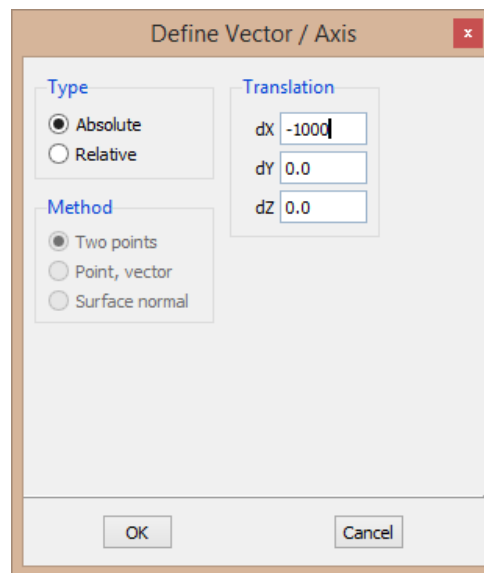i.   In the "Tools" tab, find the "Move/Copy" tool under the "Build" tree.
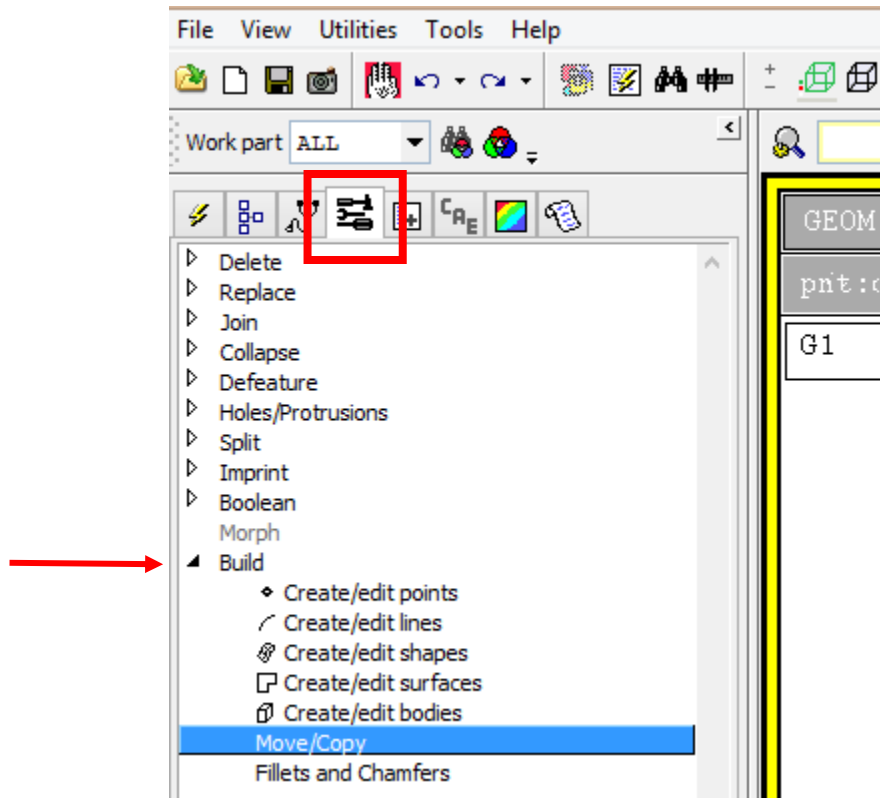
ii.   Select "Move."
iii.   Select the "Translate" tool (the default).
iv.   From the dropdown menu, choose point entity types.
v.   Select the point at the origin.

vi.   Click "Define…" A new window will pop up.
vii.   Select "Absolute."

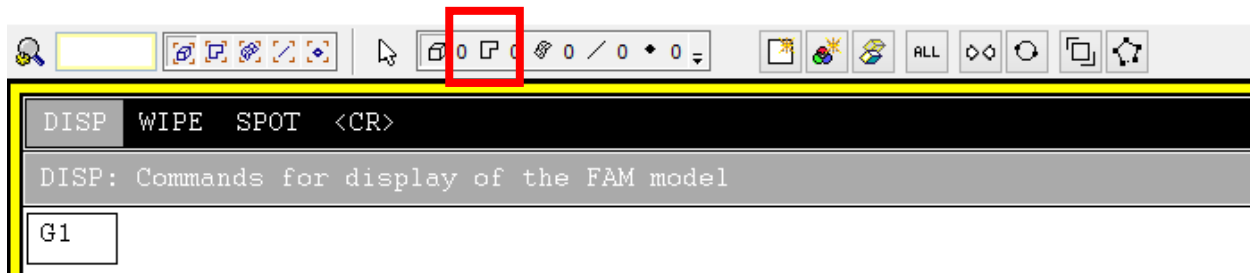viii.   Enter -1000 in the "dX" field.

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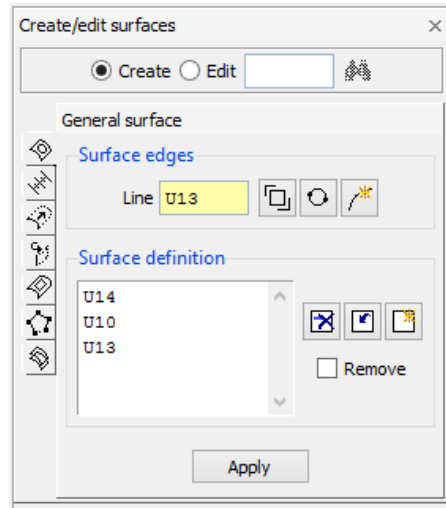
    c. Click "Apply."
    d. On the pop-up window, click "OK."
    e. Repeat steps a-d for the remaining three surfaces.
10. Repaint the picture to see all of the new geometry.
11. Close all open sets to keep it organized: **SETC ALL**

Wings

The wings will be 1 meter tall, centered along the height 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…"
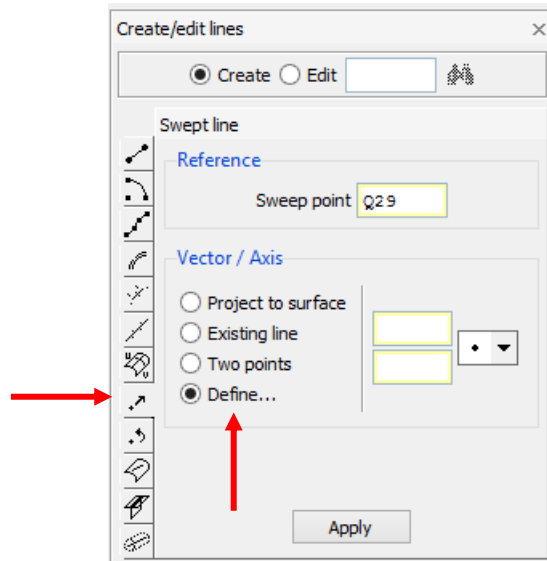
              iv.   In the pop-up, select "Absolute."

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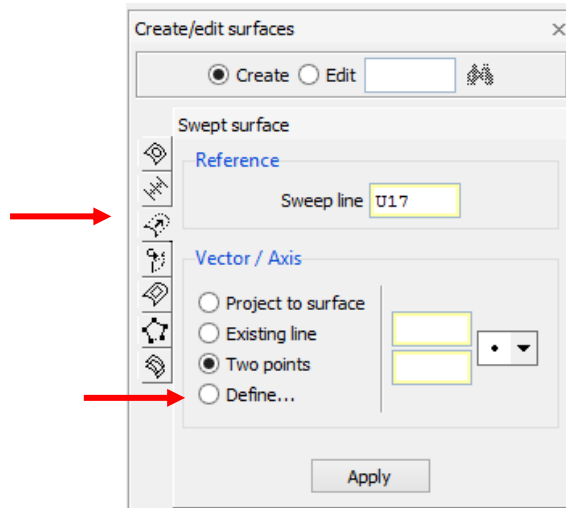
iv. In the pop-up, select "Absolute."
v. Enter -1000 in the "dY" field. The other fields should be zero.
vi. Click "OK."

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:"
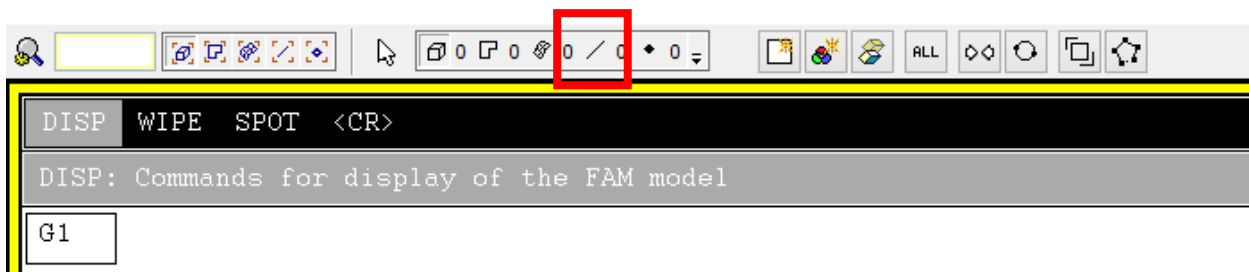
    i. Pick the lines that bound the surface created in step 10 (if they aren't visible, make sure to repaint the picture)
    ii. Enter: **SWEP SELE ! TRA 0 0 13000**
    iii. Repaint the picture: **REP**
    c. Sweep with GUI:
    i. Follow steps in 10b, choosing the lines created in step 10 as starting 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**
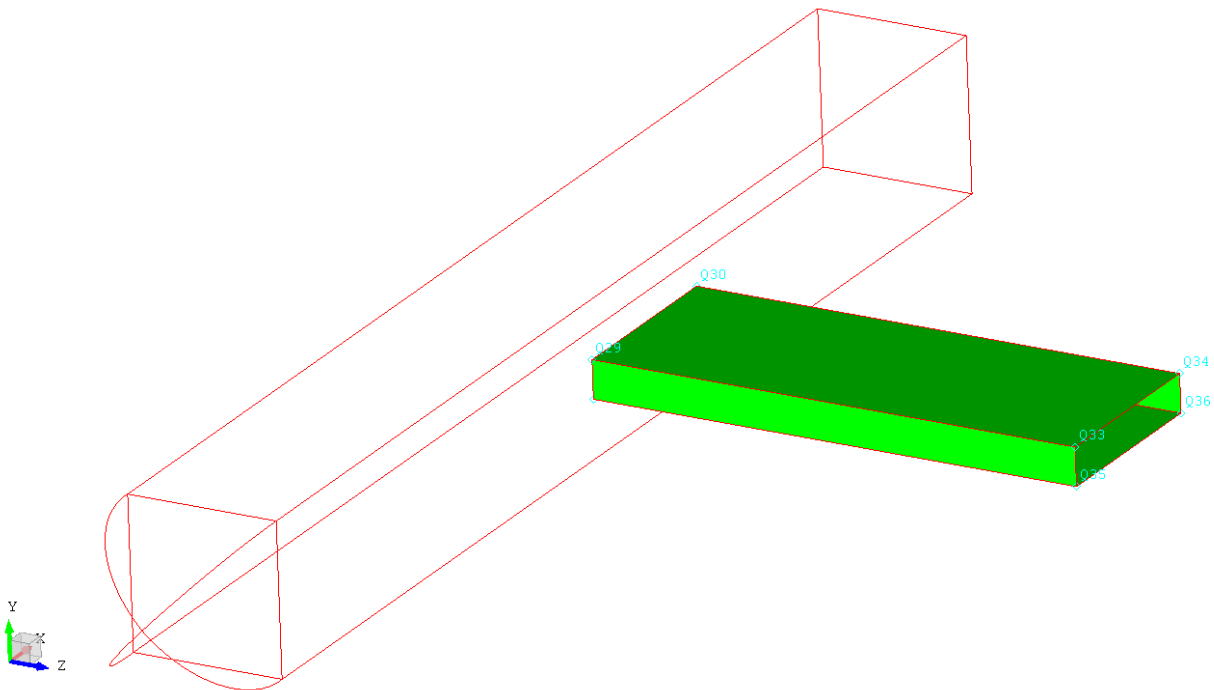
      ii.   Use the "General surface" tool to close the gap in the wing. See steps 9a-9d from the Cockpit section for more details.

      iii.   Repaint the picture

12. Check what sets are open and close the ones that aren't "PLANE":

    a.   Type **PRNT SE**

```
 Fb>
 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.   Navigate to the "Move/Copy" tool in the "Build" tree, under the "Tools" tab.

    b.   Click "Copy."

    c.   Switch to the "Mirror" option.

    d.   Type "LEFTWING" into the selection field and hit enter. This will select everything in the LEFTWING set for copying.

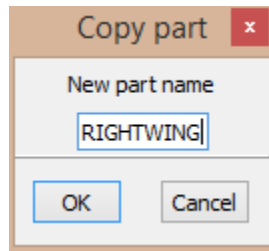   e.   In the dropdown menu for point selection, choose "Pick mid-point of line."
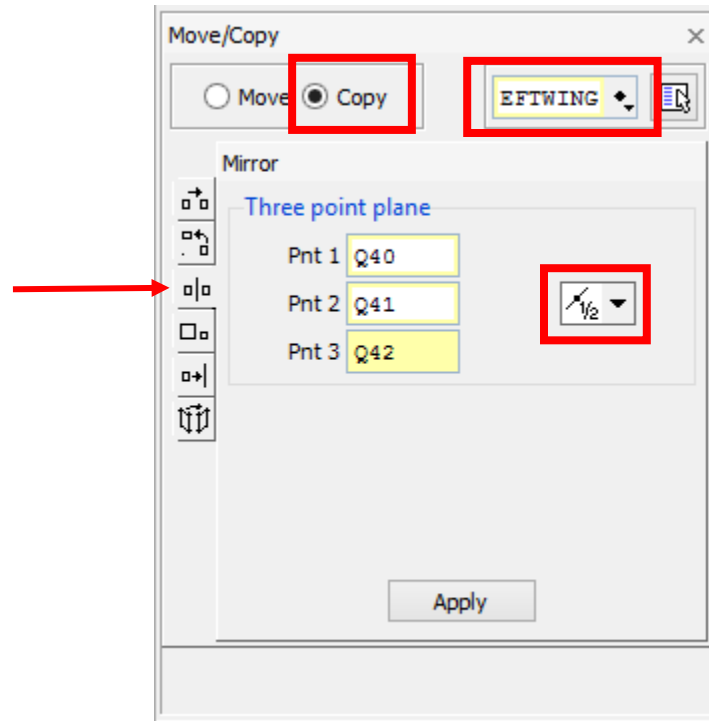
f. Select the upper and lower lines at the rear face of the fuselage, as well as the upper line where the cockpit joins the fuselage. Midpoints of these lines will make an X-Y plane.
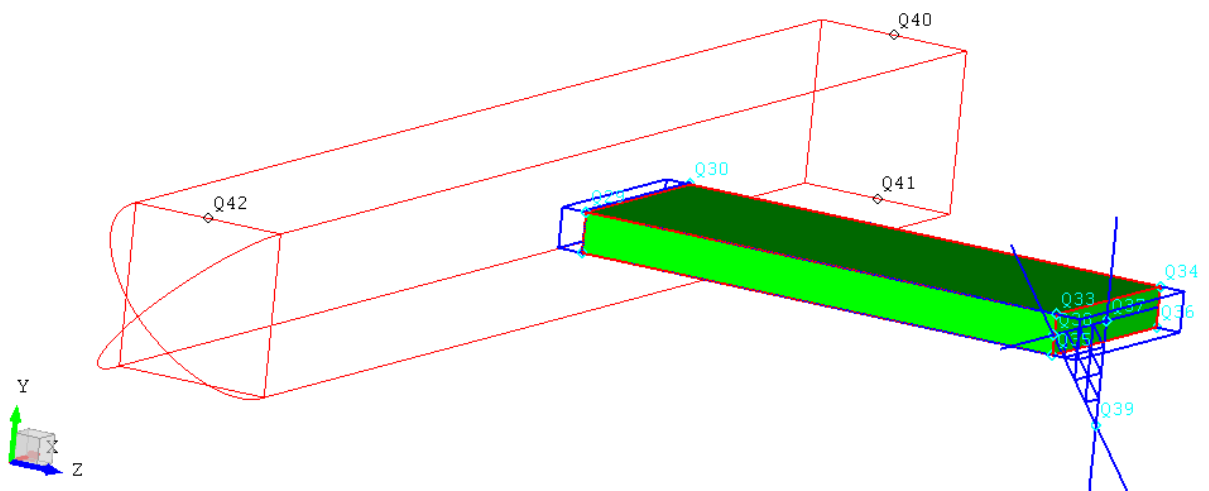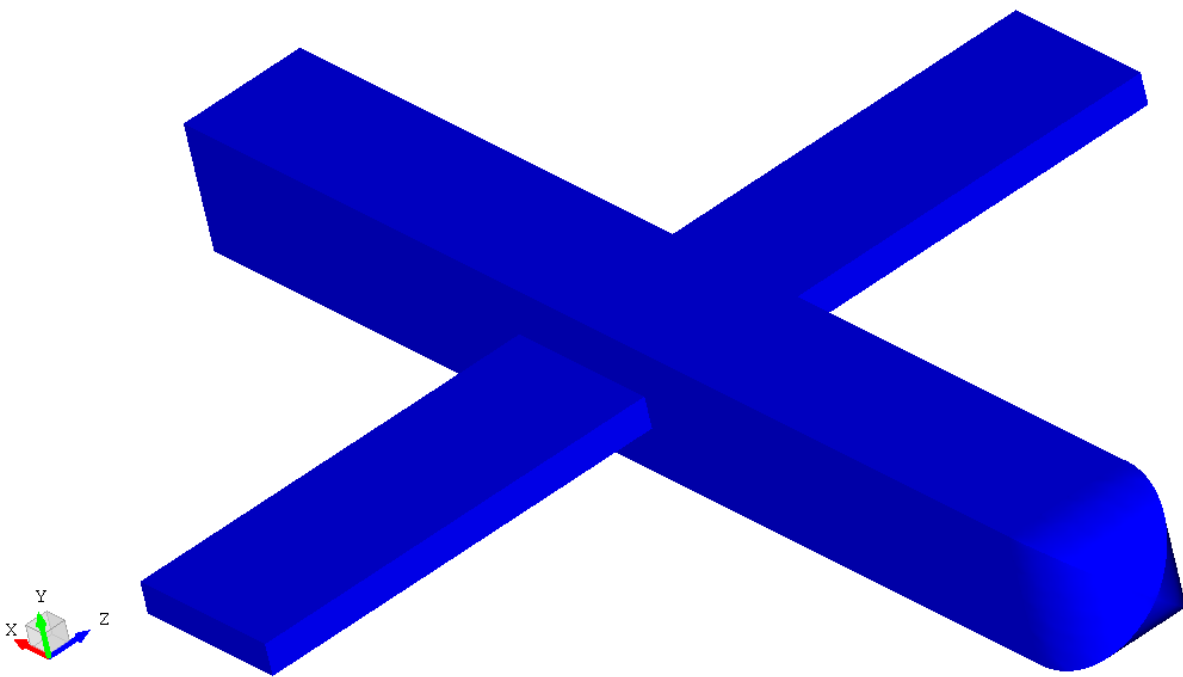

g. Click "Apply."

h. A popup will ask for a "New part name," enter "RIGHTWING." This will put the new wing in a set named RIGHTWING.

i. Click "OK."

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.

1. Plot the lines in PLANE: **PLOT L PLANE**
2. Add the points (with labels) to the image: **PLUS PA ALL**
3. Navigate to the "Move/Copy" tool.
4. Select "Copy."
5. Type "LEFTWING" into the selection box.
6. Switch to the "Rotate" option.
7. Select "Two points" to define the axis of rotation.
8. Choose the point at the tip of the cockpit and a point directly aft of the tip.

9. Enter "-90.0" as the angle (right hand rule determines the sign).
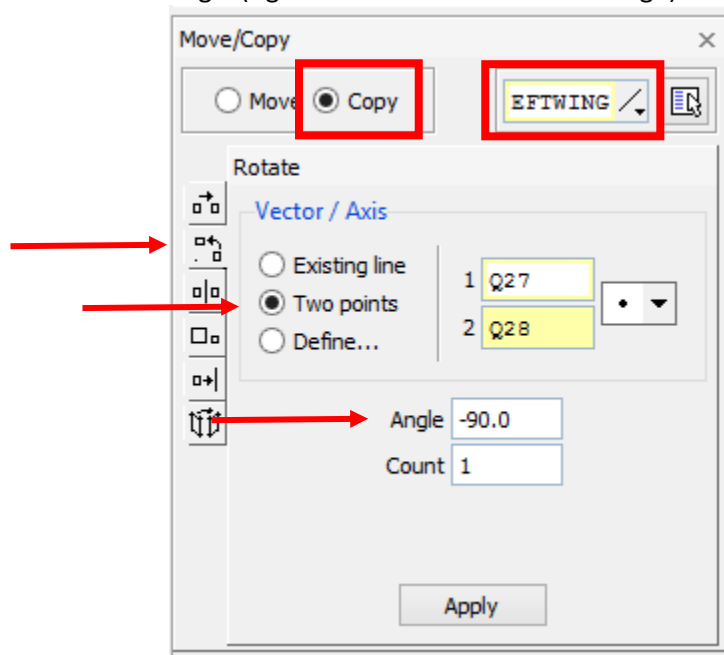
10. Click "Apply."
11. Enter "TAIL" as the part name in the popup.
12. Wipe all points from the image to help visibility: **WIPE PA**
13. Add the surfaces of the tail to the image, in green: **PLUS SI TAIL G**
14. Move the tail aft 11 meters:
     a. **MOVE TAIL TRA 11000 0 0**
     b. Repaint the image
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.
     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).
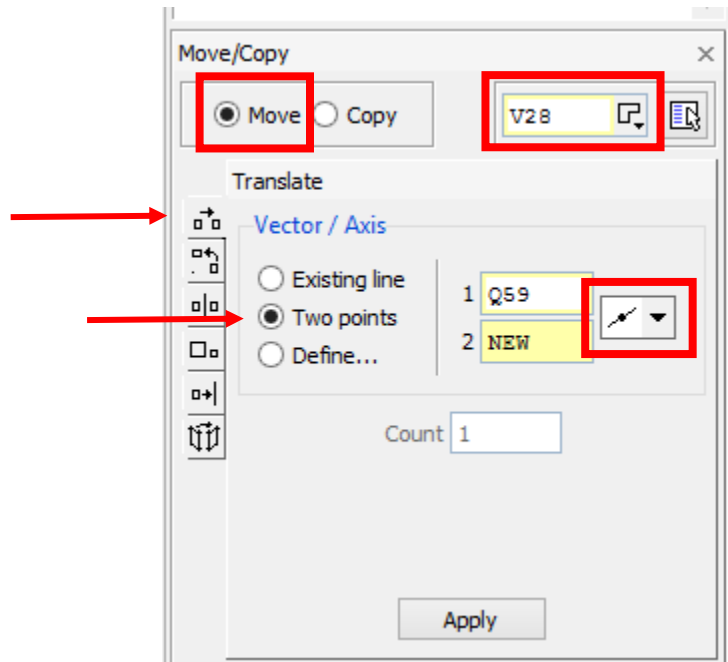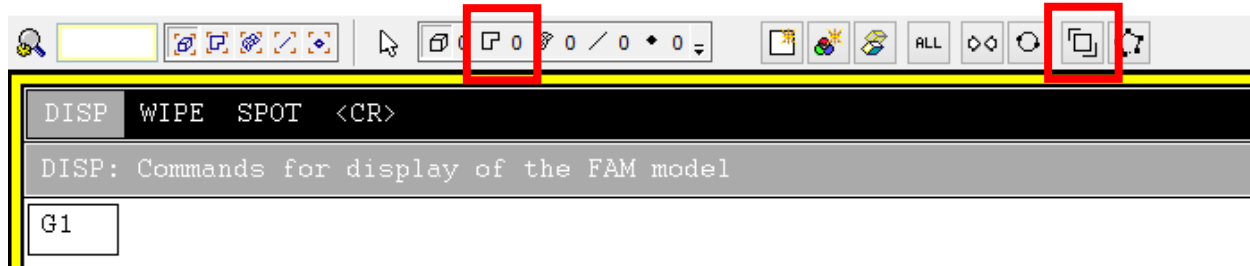

i. Click "Apply."
j. Repaint the picture.
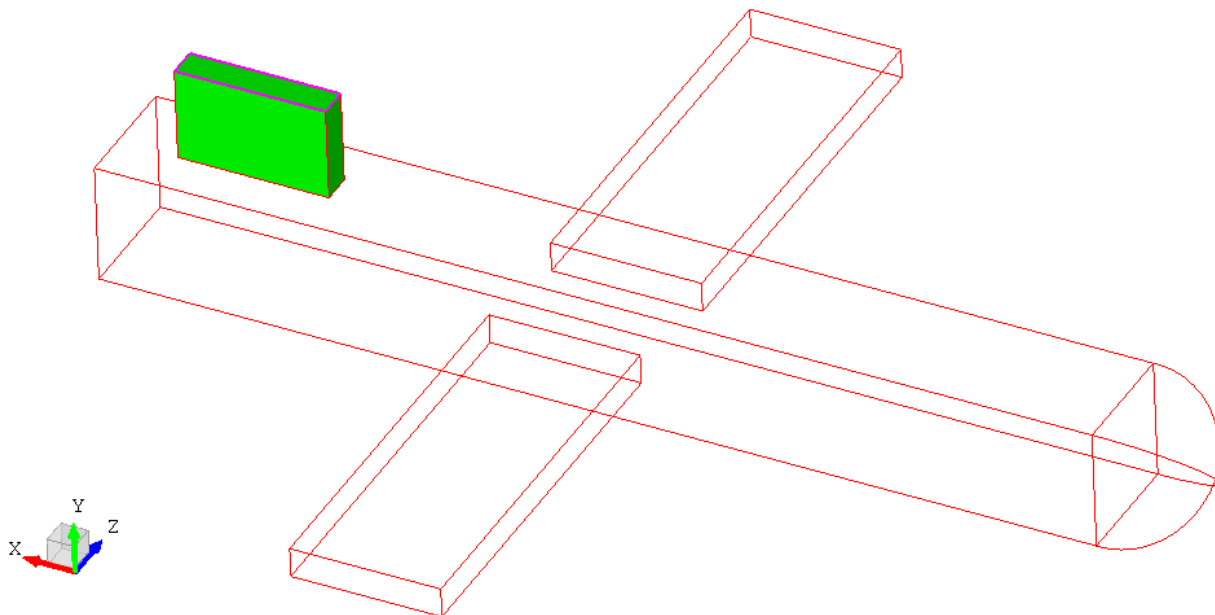
Finishing Touches

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."
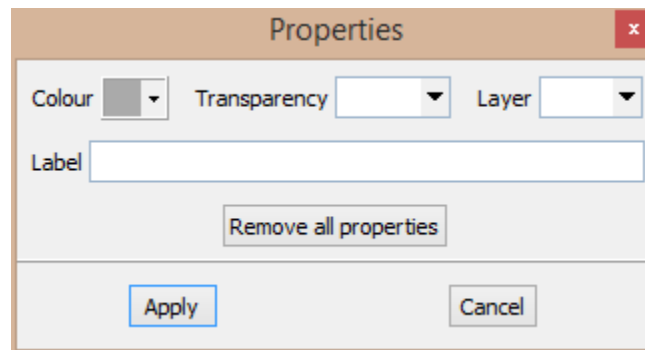
10. Repaint the picture.

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.
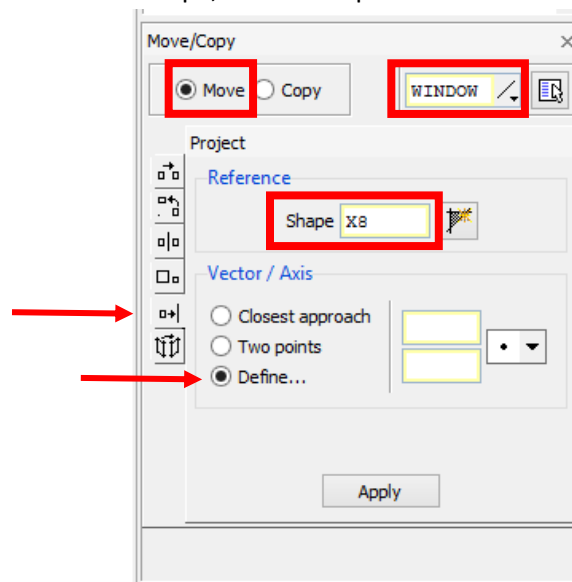
f.   Click "Define…" so we can pick the direction of projection.
g.   In the top left of the popup, click absolute.

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.
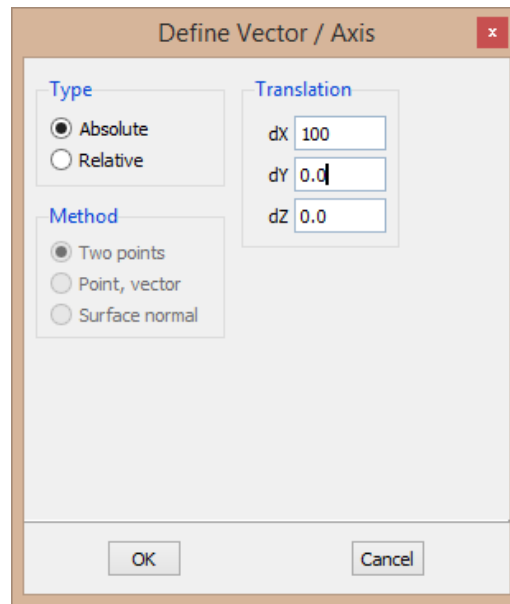
i. Click "OK."
j. Click "Apply."
k. Repaint the picture. It should look like this:
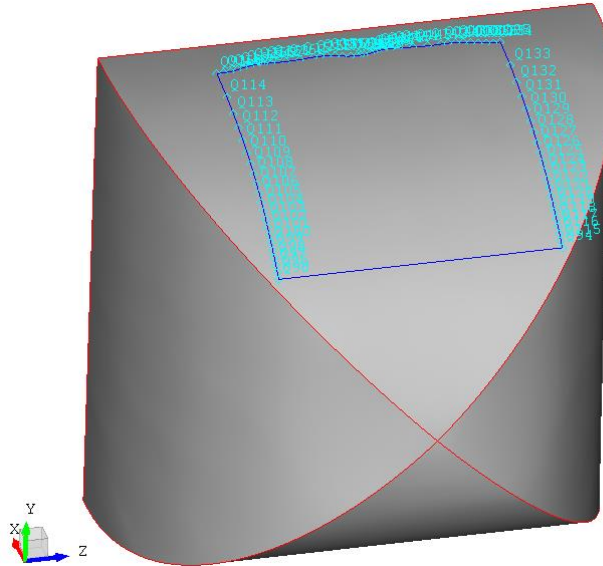
You'll notice that many more points have been created. This is a result of how CADfix projects lines onto embedding shapes. You might also notice that the upper line isn't perfectly straight. That is because the embedding shape isn't perfectly flat. This detail likely won't be noticeable when the model is meshed into a finite difference problem space.

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."
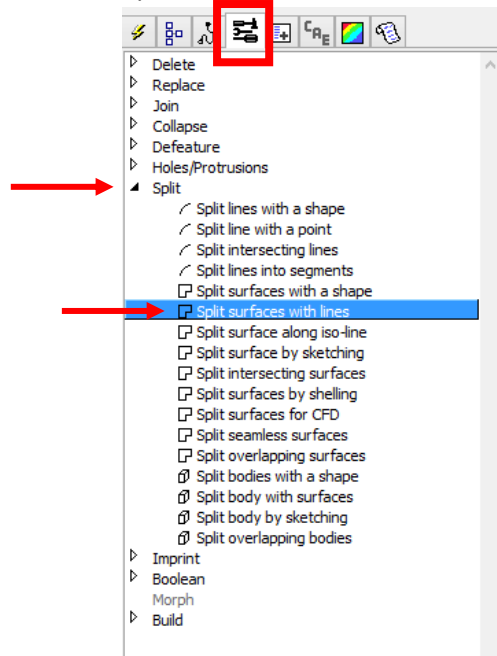
      ii. Pick the top surface of the cockpit.
      iii. Type WINDOW into the "Pick lines" field and hit enter.
      iv. Change the tolerance to 5 mm. This ensures that the split will succeed.
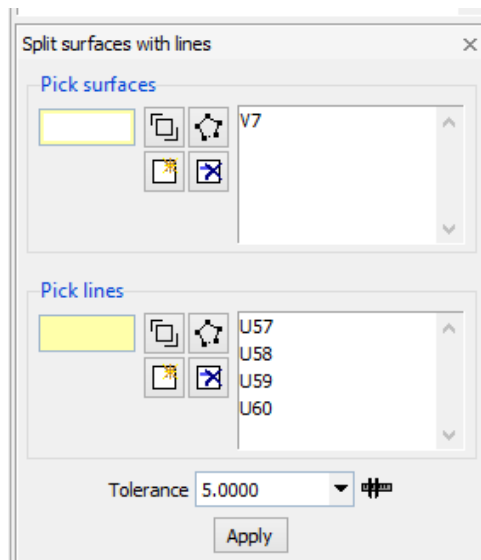
      v. Click "Apply."
      vi. Repaint the picture.

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**

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, as well as 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.

More detail can be added to this simple model (horizontal stabilizers, window apertures, etc.), but the scope of this model limits the techniques that can be exemplified. For practice, extra exercises are included at the end of this document. Instructions aren't included for those. However, all of the techniques necessary for the exercises have been presented in the "Constructing Geometry" section. It is encouraged to try the exercises to help enforce what has been learned and to bring attention to what wasn't understood.

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 "EFIdnit." 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."
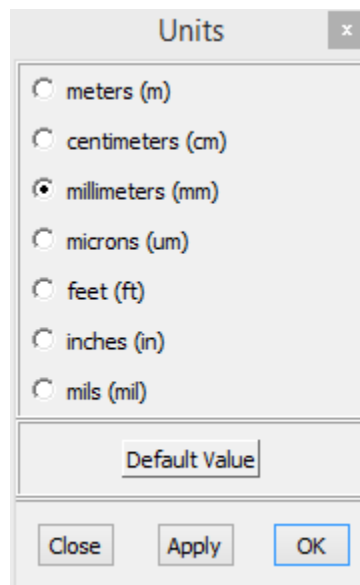
3) Define Lattice

Click "Tools->Toolbox->EMA3D_#3_DefiineLattice." 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 default values (based on all model geometry) are shown to the right. We will change the cell size to 500 mm for all directions. We will also change the start and end coordinates for all directions. We will add 2 meters of space between the plane and the boundary of the problem.

To check the dimensions of the plane, type: **PRNT DIMS PLANE**
This will print the dimensions of the set PLANE. Use the global
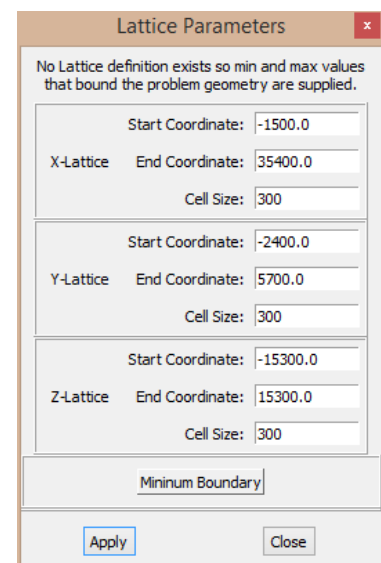
minimums and maximums of the "Line ends." The section for "All cords" includes points and shapes that don't need to be meshed.

```
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

54 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 add increase the boundaries by 2000 mm and use the next greatest even multiple, the parameters are shown to the right. Click "Apply."

Lattice Parameters

No Lattice definition exists so min and max values that bound the problem geometry are supplied.

X-Lattice
Start Coordinate: -3000
End Coordinate: 37000
Cell Size: 500

Y-Lattice
Start Coordinate: -4000
End Coordinate: 7500
Cell Size: 500

Z-Lattice
Start Coordinate: -17000
End Coordinate: 17000
Cell Size: 500

Mininum 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. We will make a wingtip-wingtip strike situation. A source line will carry the lightning current to the plane and a detach line will carry it away. In the process, we will make four new sets: one to hold just the source line, one to hold the detach line, one to hold both, and the last to hold the plane, source, and detach line.

Source Line:

First, open the new sets for the source line and the new master set:

1. **SETO SOURCE**
2. **SETO MASTER**

Next, make a point at the center of the left wingtip, either with GUI or command line:

3. Command line: **PNT ! corner1 corner2 0.5**
   a. Where **corner1** and **corner2** are points at opposite corners of the left wingtip, **0.5** is the ratio of distances between the two points.
4. GUI: navigate to the "Create/edit points" in the "Build" tree. Use the "Point along a path" option. Select the two corner points of the wingtip. Leave the ratio at 0.5. Click "Apply."
5. Add the new point to the image: **PLUS PA SOURCE**

Sweep the new point 2000 mm in the positive Z direction using your preferred method. For convenience, the command is provided below:

6. **SWEP pnt ! TRA 0 0 2000**
7. Add the source line to the image: **PLUS L SOURCE**
8. Repaint the picture: **REP**
9. Close the SOURCE set: **SETC SOURCE**

Detach Line:

We will mirror the source line to create the detach line.

1. Open the detach set: **SETO DETACH**
2. Navigate to "Move/Copy" in the "Build" tree.
3. Select the "Mirror" option.
4. Click the "Copy" radio button.
5. Type SOURCE into the selection box to select the source line for copying. Hit enter.
6. From the dropdown menu, select "Pick midpoint of line."
7. Pick lines whose midpoints make an X-Y plane (horizontal lines of the front and rear fuselage faces work well).
8. Because we've already opened a DETACH set, the part name doesn't matter. Leave it as the default when the dialogue pops up. Click "OK."
9. Add the detach line to the image: **PLUS L DETACH**
10. Repaint the picture.
11. Close DETACH: **SETC DETACH**

Wrap-up:

We will add the DETACH and SOURCE sets to a STRIKE set so we can access both quickly. We will also add the PLANE set to the MASTER set. Having separate sets for the model geometry and the master set is often useful. The master set will become the meshed set later. We will also close all sets.

1. **SETA STRIKE DETACH SOURCE**
2. **SETA MASTER PLANE**
3. **SETC ALL**

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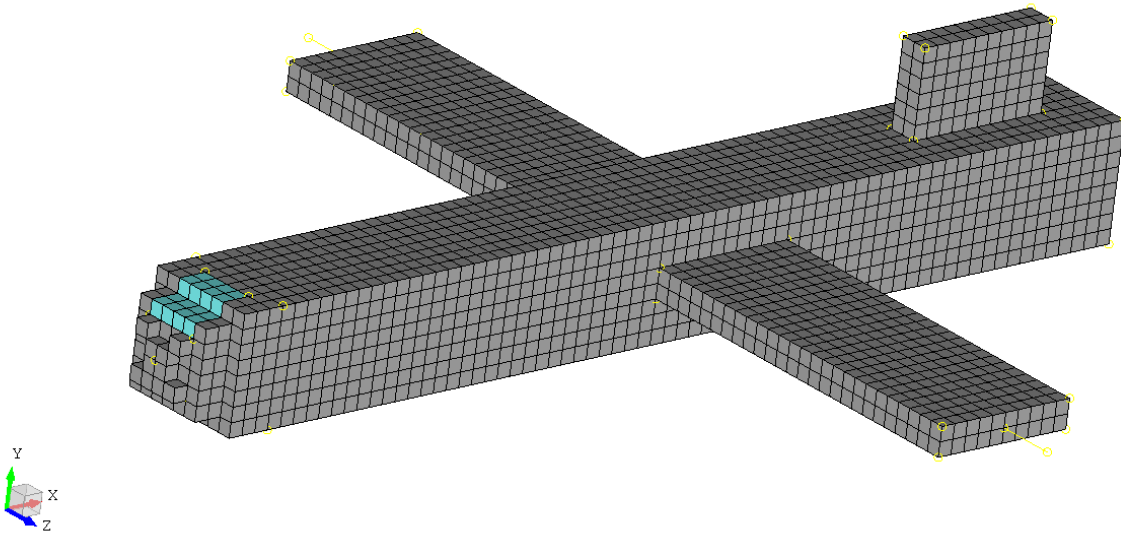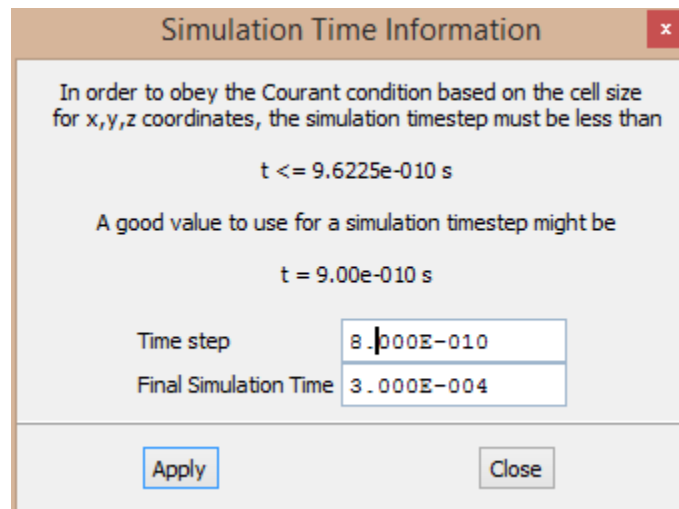
For practice, mesh just the left wing: **MESH LEFTWING**.

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-10 for the time step and 3e-4 for the final simulation time.

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."

Next, click "MagTSteps." This window controls the parameters for gradual permittivity scaling. Enter the values as shown below and click "Apply:"

Permittivity scaling allows EMA3D to increase the time step without violating the courant condition. Scaling reduces the computation time significantly and is a very important EMA3D tool.

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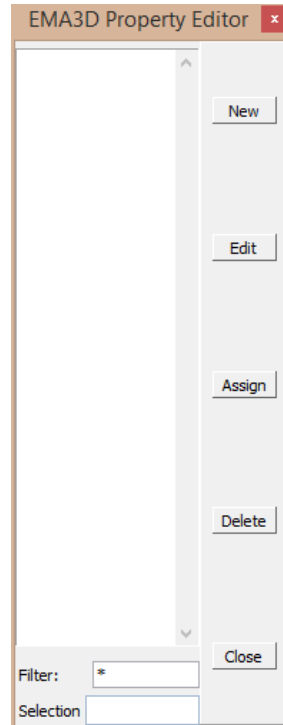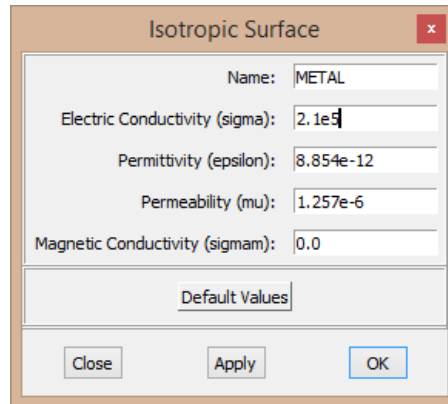
Three material properties will be needed for the simple airplane: conductive metallic surfaces, non-conductive glass surfaces, and perfectly electrically conductive lines.

1. To make a new material, click "New." A window will popup:
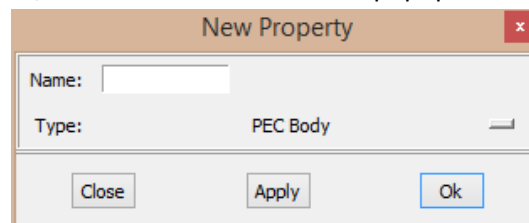
2. Enter "METAL" as the name.
3. Select "Isotropic Surface" for the type.

4. Click "Apply." Another window will popup. Fill in the values as follows:
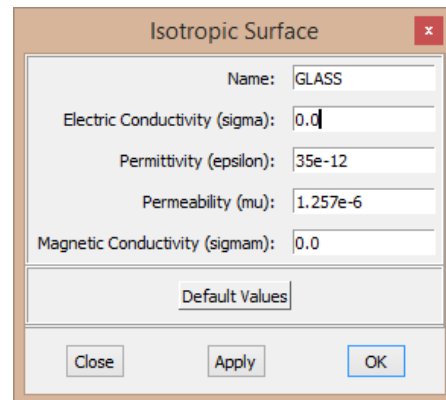
The conductivity is that of a typical airplane aluminum scaled for use with 500 mm cells.

5. Click "OK." METAL has been defined as a material.
6. In the "New Property" window, enter "GLASS" as a name and choose isotropic surface as type.
7. Click "Apply."
8. In the Isotropic Surface window, leave the conductivity at zero because glass is a very poor conductor. Change the permittivity to 35e-12, or roughly 4 times the permittivity of free space. Permeability can stay the same.

9. Click "OK."
10. In the "New Property" window, enter PECL as the name and choose "PEC Line" as the type.
11. Click "OK."
12. In the pop up, click "OK;" there are no parameters to change.

The materials we need have been defined. Now we need to assign them to the parts. The WINDOW set will clearly receive the GLASS material. The rest of the plane will be assigned METAL properties. The detach line will be assigned the PECL property (the SOURCE line will get a special current density property later).

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 the window. We can create a duplicate of the PLANE set named METALLIC and remove the window from METALLIC:

SETA METALLIC PLANE
SETR METALLIC WINDOW

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."
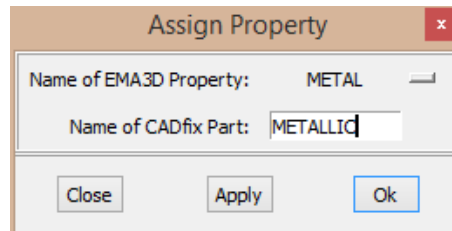
After clicking "Apply," change the property to "GLASS." Enter WINDOW as the CADfix part. Click "Apply."
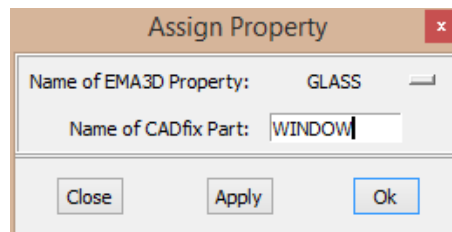
After clicking "Apply," change the property to "PECL." Enter DETACH as the CADfix part. Click "OK."
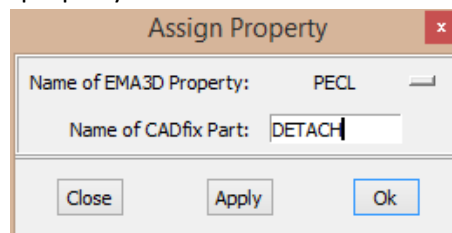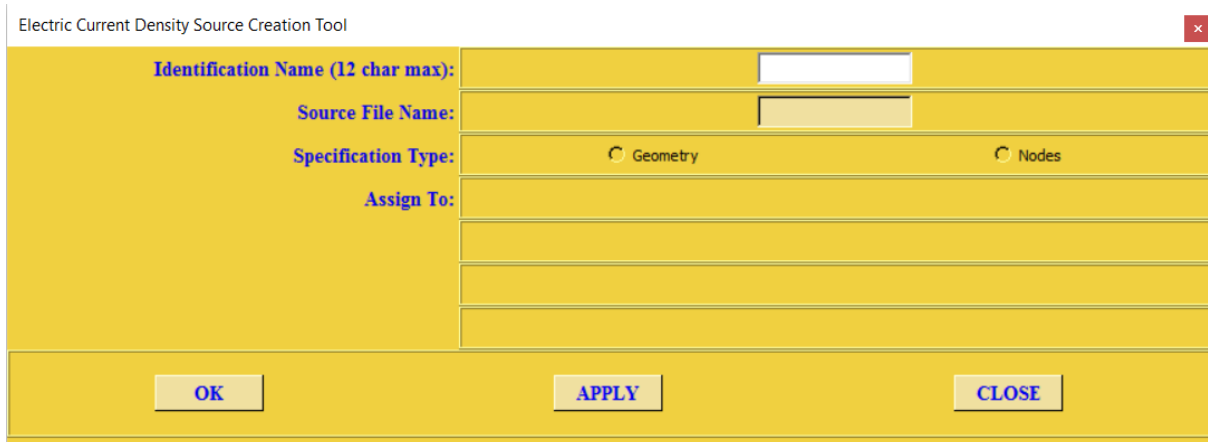
All material definitions are now applied. Close the property editor.

9) MHarness Cables

Some EMA3D purchases include MHarness. MHarness is used for complex pipe and cable routing and probing. The simple airplane in this training module has no pipes or cables, so this tool isn't needed.

10) Define Source

Click "Tools->Toolbox->EMA3D_#11_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. Electric current densities are used for lightning simulations. To define an electric current density source, find and click the tool labelled "EleCurDen." A window will pop up.

The source file is the file that has the source waveform. A source has been provided and is titled comp_a.dat. Enter "comp_a.dat" in the source file name field.

The identification name is a unique name used to reference the source. We will enter "SSOURCE" in the "Identification Name" field.

We wish to define the source using a line, so we will select "Geometry" as the "Specification Type".

Our source line is contained in the SOURCE set, so we can select "Set of Lines" in the "Assign To" field and enter "SOURCE" in the "Set Name" field. Alternatively, we can also select the "Single Line" option and reference the line directly in the "Line Name" field. Following the default naming convention, the line's name should begin with a "U" and can be found using the command: **PRNT L SOURCE**

The "Assignment Type" defines the source direction, our line was created starting at the wingtip and traveling outboard, so choosing the "End to Beg" option will define our lightning to travel inboard towards the aircraft. To verify this, the command **PLOC L SOURCE** followed by **PLOC SENSE ON** can be used to show the line direction. Our source line should point outwards like this:
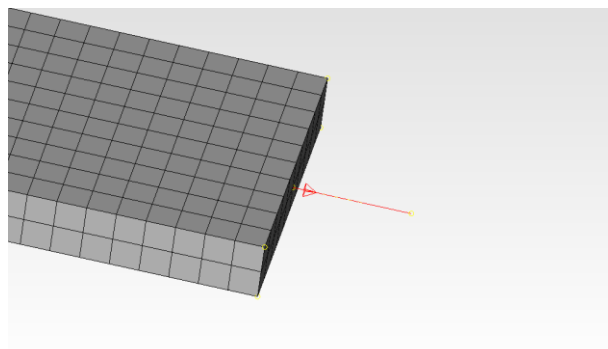
If the arrow points inboard, the "Beg to End" option should be used. Note: the source direction will not change the physics for this simulation, it will, however, change the sign of the final answer.

Finally, the "Multiplier" can be set to 1, which causes the filled-out window to look similar to:

Click "OK." The source has been defined.

11) Boundary Conditions

Click "Tools->Toolbox->EMA3D_#12_SelectBdyConds." Click the only tool in this toolbox. A window will appear. Check "ALL" on the "Mur1 (HFLD)" row. The Mur 1 (HFLD) option as a boundary condition that approximates an infinite extension of background problem space by absorbing magnetic fields at the boundary. Absorbing the magnetic fields reduces the reflections at the boundary. This boundary

condition has been used with lightning simulation models with great results.

Leave the bottom three entry fields empty; they are for a different boundary condition. 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 a bulk current probe on the left wing to measure attachment current, a bulk current probe on the right wing for detachment current, and a surface current animation probe.

Creating any probe will require going into: "Tools->Toolbox->EMA3D_#13_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 left wing. This probe will need to completely cover a cross section of the wing. The first step is to create two points that will be opposite corners of a rectangle of the probe.

1. Plot the left wing with the assigned color: **PLOT SI LEFTWING ASGD**
2. Open a set to contain all probes: **SETO PROBES**
3. Open another set to contain just the left wing probe: **SETO LWPROBE**

The points that define this probe should lie exactly on the mesh grid. They should also share the same Z coordinate so that the points make a predictable cross section. There are many ways to do this. One way is to:

4. Add the mesh twinkles of the left wing to the image: **PLUS TWI LEFTWING**
5. Navigate to "Create/edit points" in the "Build" tree.
6. Use the "Point on line" option.
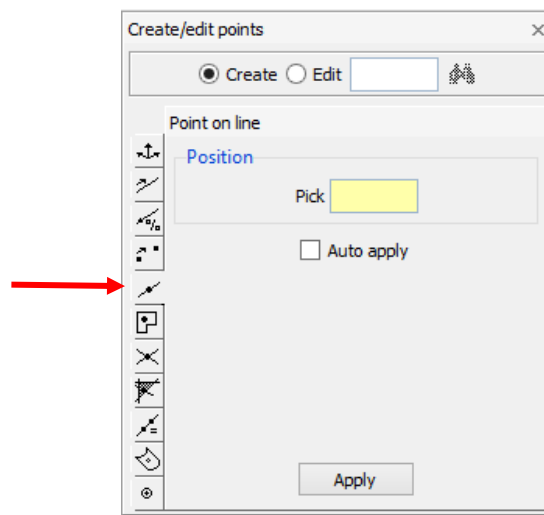
7. Now the cursor should look like a crosshair. Select a location on the lower forward line of the wing, about one cell in from the wingtip. It doesn't need to be exact.
8. Click "Apply" in the "Create/edit points" window.
9. Repeat steps 7 and 8 for the upper back line of the wing.
10. Close the left wing probe set: **SETC LWPROBE**

We will snap these points onto the grid later. Before that, we will make the right wing probe. You could mirror the left wing probe, but we'll use a different strategy for the right wing.

The special source property that attaches to the left wing means that it can't be probed directly. That's why the probe was placed on the wing one cell away from the tip. The detach line that connects to the right wing doesn't have any restrictions on it. Instead of probing the wing, the detach line can be probed. This means we only need one point in the probe to completely bound the line.

1. Plot the detach line: **PLOT L DETACH**
2. Open a set for the right wing probe: **SETO RWPROBE**
3. Using the same "Point on line" point creation tool used earlier, create a point anywhere in the
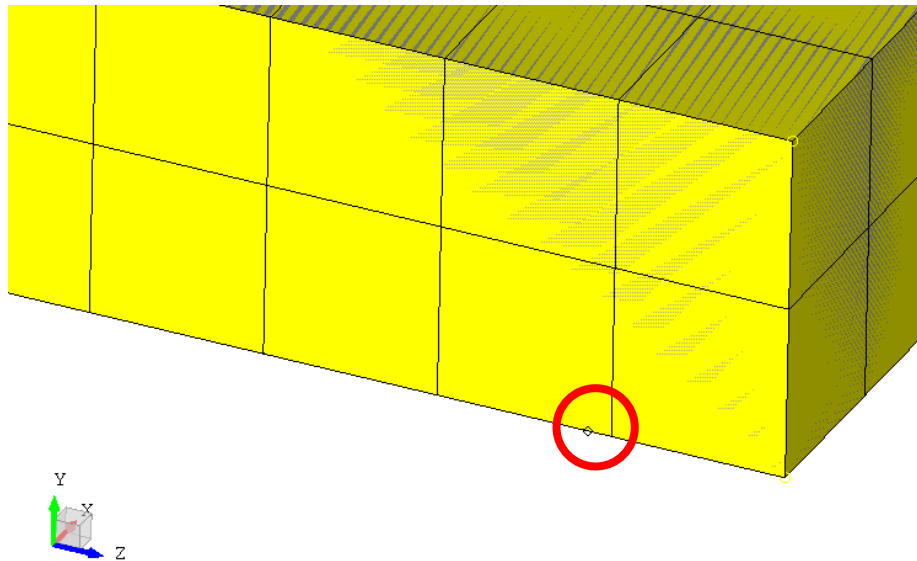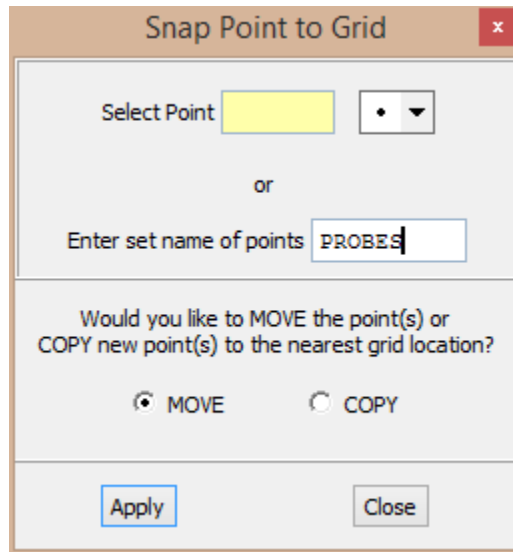

   interior of the detach line (avoid the 500 mm at either end).
4. Close the right wing probe set: **SETC RWPROBE**
5. Close the PROBES set: **SETC PROBES**

All of the points needed for the probes are made. To snap them to the mesh grid:

1. Change the toolbox: "Tools->Toolbox->Miscellaneous."
2. Click the "SnapToGrid" tool.

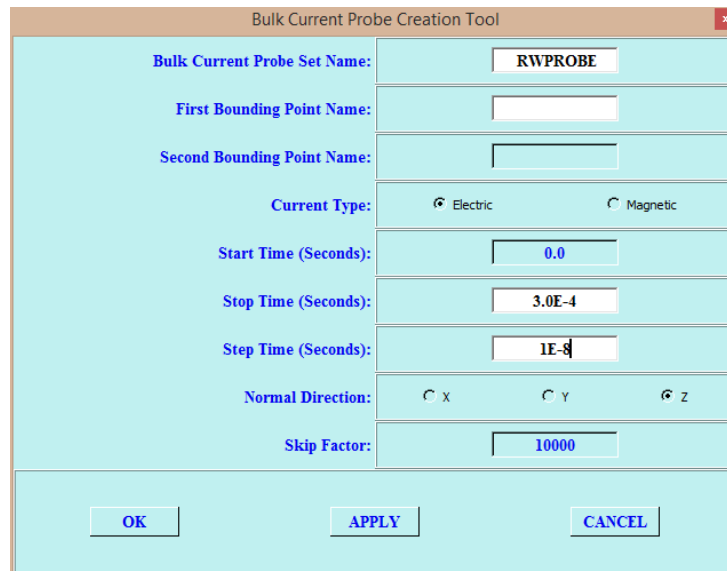3. Enter PROBES as the set of points to move.

4. Click "Apply."

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_#13_DefineProbes.

1. Click "BulkCurr" in the toolbox.
2. A window will pop up. Fill it out as shown:

3. The bounding point is the point in the set RWPROBE. To find its name, either:
   a. Add the labelled point to the image: **PLUS PA RWPROBE**
   b. Print the point in RWPROBE: **PRNT P RWPROBE**
4. Enter the point name in both of the bounding point fields.
5. Click "APPLY."
6. Change the set name field to LWPROBE.

7. Change the bounding points to the points in the LWPROBE set. Leave the time fields and normal direction unchanged.
8. Click "OK." The bulk current probes have been defined.

Animation Probe Set Creation

The animation probe needs a set of nodes from meshed surfaces. To make a set of just the surface nodes:

1. Create a set, SJSA, with all surfaces
2. Complete the set down to the nodes.
3. Remove all surfaces from the set.
4. Create a second set, REMTEMP, with all lines.
5. Complete the second set down to the nodes.
6. Remove the second set from the first (removes undesired line nodes).

The commands for the process are:

1. **SETA SJSA S ALL**
2. **COMP SJSA GN**
3. **SETR SJSA S ALL**
4. **SETA REMTEMP L ALL**
5. **COMP REMTEMP GN**
6. **SETR SJSA REMTEMP**

Use **PRNT SE SJSA** to print a summary of the contents of the set SJSA. There should only be nodes, labelled as "N."

```
Fb> prnt se sjsa
                  P       SH      L       C       S       B       N       E
SJSA              0       0       0       0       0       0       3918    0
No open set
Fb>
```

Animation Probe Definition

In the Define Probes toolbox, click "Traditional." A window will pop up. Fill it in as follows, changing the output type to "JS" for surface currents. The stop time is the end of the simulation. The step time is chosen to have 50 frames in the animation.

Click "OK." The animation probe has been defined.

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."

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 should only take a minute.
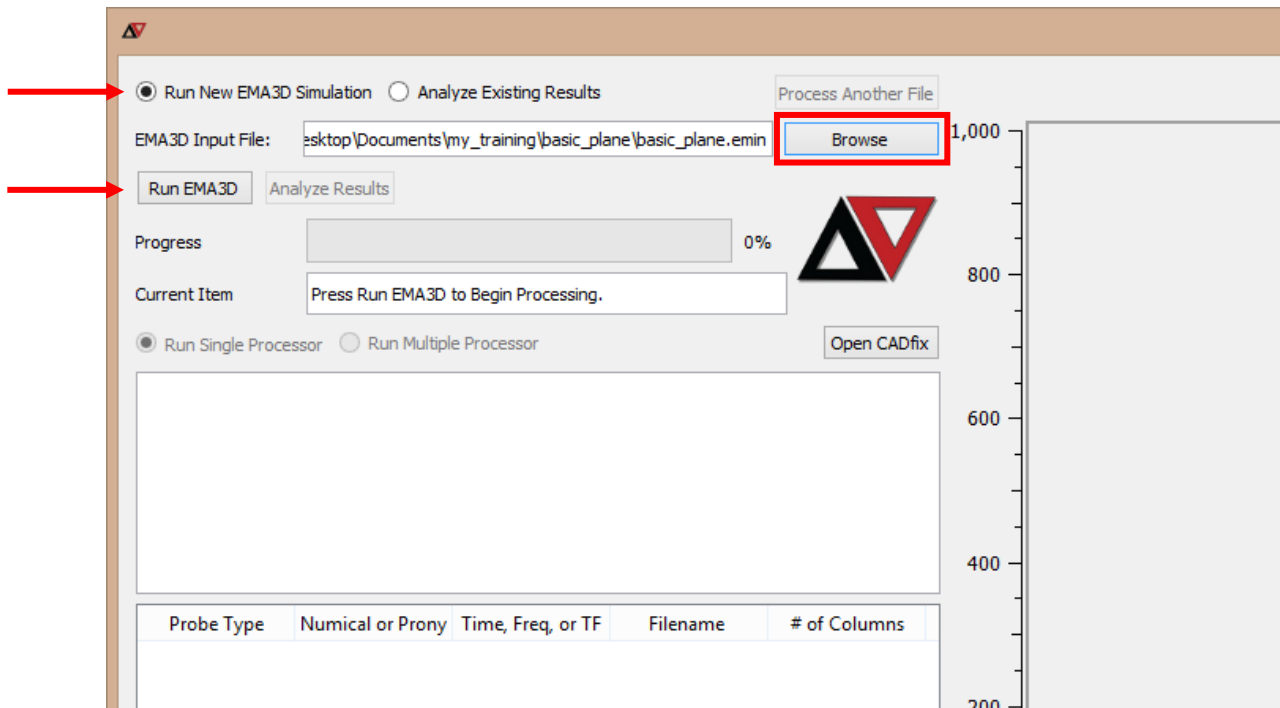
Post-Processing

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. For this reason, simple Python scripts have been included as examples of how process data outside of any GUI's. Also included is a more capable Python script that can process hundreds of current and voltage data files in succession.

Some EMA3D probes output files that aren't stored as .dat files. An example of such a probe is the animation probe. The simple airplane simulation used in this exercise has an animation probe, so the process will be shown.

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. Plots can be plotted one on top of another.
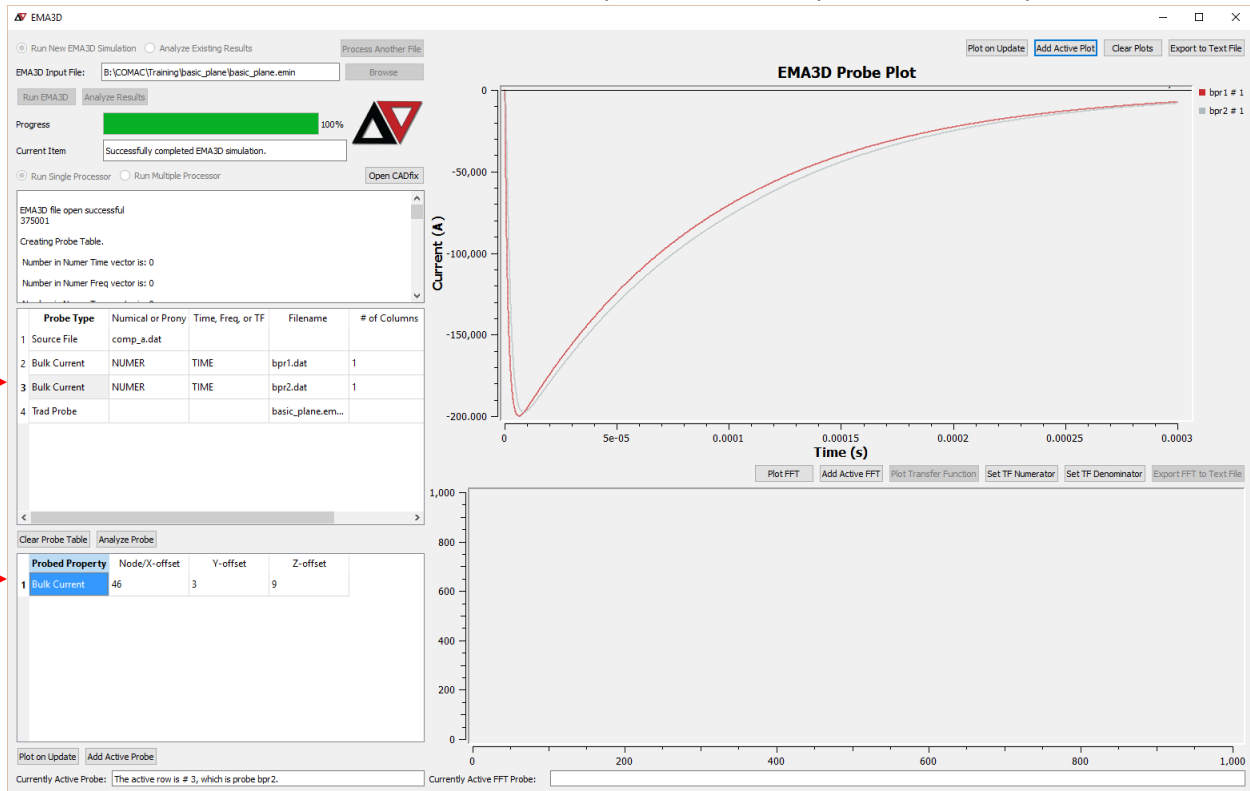
Simple Python Scripts

Our simulation only contained current probes and one animation probe. The Python scripts explored in this section are only applicable to the current probe outputs. Included with the documentation for this training module are three python scripts:

1. post_process.py
2. single_j_plot.py
3. single_j_plot_cmd.py

single_j_plot.py

The first, post_process.py, is the strongest of the included scripts. It will be discussed in more detail later. The other two are duplicates of each other except the inputs are entered differently. To use "single_j_plot.py," edit the script in your preferred Python IDE. Because the Spyder IDE is included in the EMA3D installation, this training will assume you will be using Spyder.

To open the script in Spyder, navigate to the simulation directory in a file explorer. Right click single_j_plot.py and select "Edit with Spyder." This will open the script in the Spyder environment.

If the "Edit with Spyder" option isn't available, try typing "Spyder" in a Windows search bar. If the program appears in the display, select it. If the search didn't find Spyder, try manually looking for it. By default, the EMA3D installation will put the installation in the user's Documents folder. Look in: ~\Documents\WinPython-64bit-3.4.3.6" where the version number (3.4.3.6) might vary. Also, on

systems that don't support 64bit look for a 32bit installation. Inside this folder there should be "Spyder.exe." Double click it. Using the Spyder GUI, open the Python script.

Once single_j_plot.py is open in Spyder, change the values for the filename and title in lines 16 and 19, respectively. The naming convention for current probes is to use the base name of the .emin file and append "bpr#" where # is the instance number of the probe. For our simulation, the base name is "basic_plane" which makes the current probe names "basic_planebpr1.dat" and "basic_planebpr2.dat." Current probe 1 is for the left wing. Current probe 2 is for the right wing.

Inside the Spyder window, make sure a console window is open. Click anywhere in the console to activate it. Now click the green triangle to run the python script in the console. The script will create png images of plots in the scripts active directory with the title you gave it.

single_j_plot_cmd.exe

Instead of editing single_j_plot.py, you can run single_j_plot_cmd.py. The "_cmd" version of the script will prompt the user for the file name and title from the command window, without any editing of the script.

An easy way to run the script is to find it using a file explorer, right click the script, then select "Run in interactive mode." A window will pop up and will ask for the file and title to use for the plot. Hit enter after each entry.

Just like the earlier script, a png will be made for the data file in the active directory of the script.

Stronger Python Script

The "post_process.py" script is a much faster method to process EMA3D probe outputs. It recursively looks through a directory for .dat files and processes each one according a set of labels provided as a csv file. This script can be run from within a GUI like Spyder or from a command prompt; use whatever you prefer.

When you run the script, it'll prompt you for the file name and path of the label file. A file named "labels.csv" is provided for the simple plane simulation. Only two .dat files were created by the simulation, so only two labels are contained in the csv. The csv holds data of the form: **_probe suffix, plot title._**

After you enter the path to the label document, the script will prompt you for the directory holding the .dat files. Enter the directory that holds the output of the simulation. As mentioned earlier, the search is recursive so make sure that there aren't any other .dat files anywhere in the directory. For this reason, I moved everything for this simulation into its own directory with no subdirectories. I named this directory "original."

```
C:\Users\Eric\Desktop\Documents\my_training\basic_plane>python.exe post_process.py
Enter the file name (with full path) to the csv holding the labels.
labels.csv
Enter the directory to search for .dat files.
C:\Users\Eric\Desktop\Documents\my_training\basic_plane\original
```

The script will automatically make a plot for each of the data files. It will also create txt files with peaks, peak times, and half peak times. When it's done, it'll display the number of seconds used for processing.

This simulation didn't make much data, but large models often have hundreds of probes. A quick tool like this is crucial when analyzing large models. Another important note is that the simple airplane simulation only created bulk current probes. The provided script will process current and voltage probes without any extra instruction.

If this script is used for other models, it may be necessary to change the cell size in the script. The cell size is needed to calculate voltage differences from electric field probes. The script is provided with a 25 mm cell size because that is a typical cell size for detailed aircraft models. The cell size definition can be found on line 90.

This script will be explored in more detail in later training modules.

Results

Any of the above methods should result in plots that look something like the below plots, except your titles may be different.
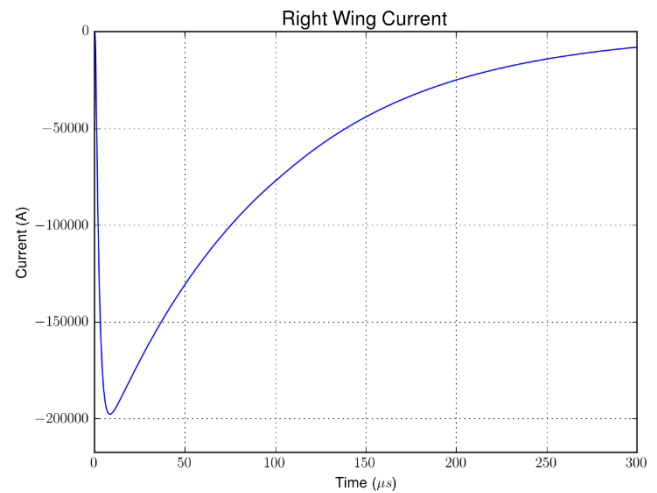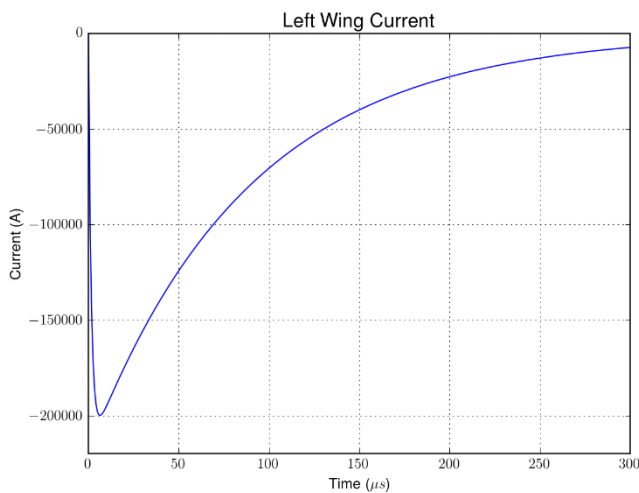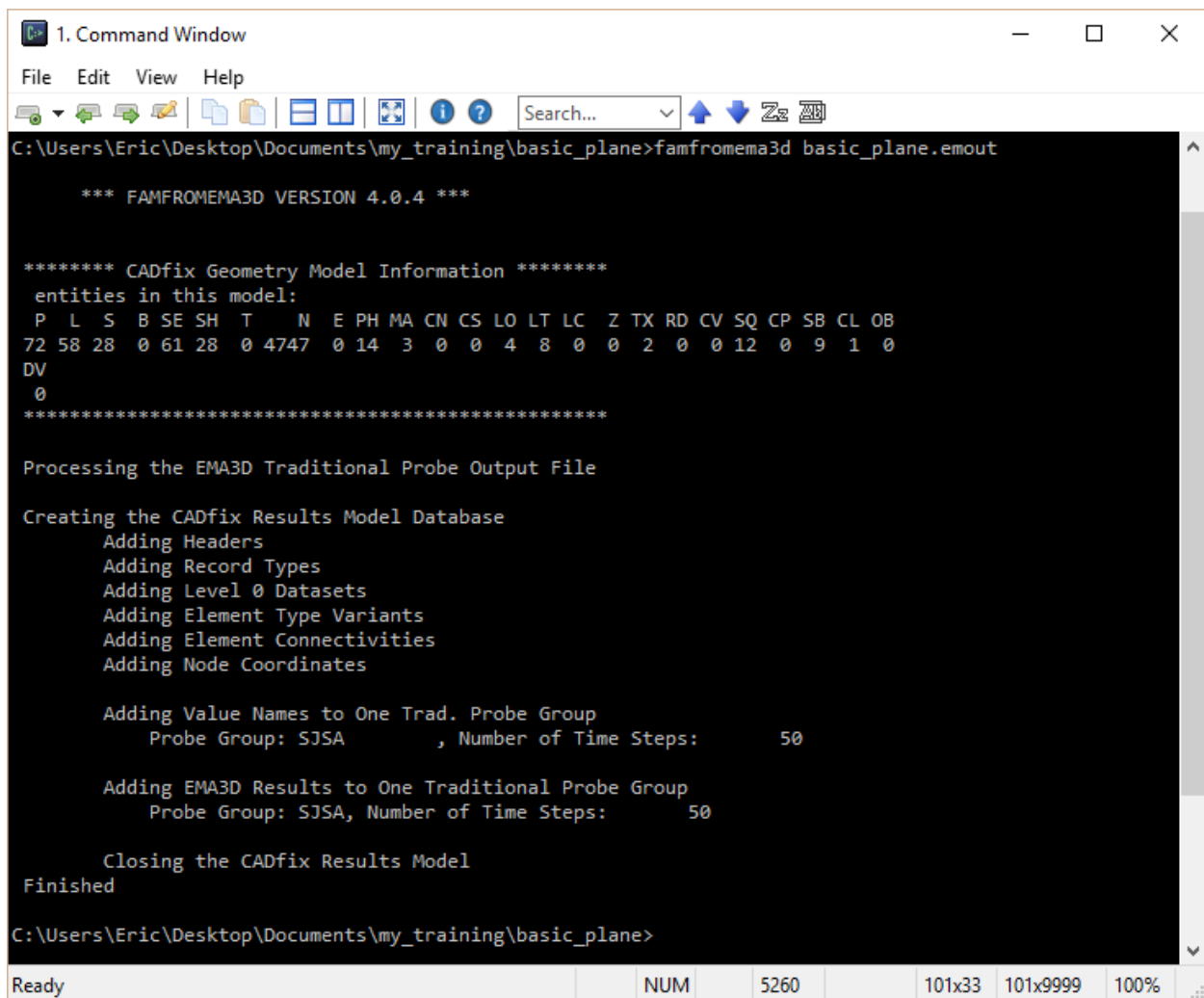
Creating an Animation

The simulation created an .emout file. The .emout has the surface currents recorded for every part in model for 50 time steps. When simulating large and complex models, the .emout can be many gigabytes large. This simple model will have a much more manageable size.

To create an animation, the data in the .emout file needs to be processed into a CADfix results database. To do this, open a Windows command window. An easy way to open a command window in the proper directory is to use a file explorer to navigate to the folder that holds both the CADfix .fbm and the .emout. Right click and select "Open an enhanced console here." However you open a command window, make sure to make the active directory the same directory that holds the emout, emin, and fbm files.

Once the command window is opened, type: **famfromema3d basic_plane.emout**. This command will turn the EMA3D results into a CADfix results database. If you named your .emin file something other than "basic_plane.emin," type that name instead (replace .emout for .emin). For large models, making the results database can take many minutes. The simple airplane just simulated is so simple, it should only take a few seconds on most modern machines.

Now that the EMA3D results are in a format usable by CADfix, open the CADfix model again. The following steps will outline how to make an animation

1. On the left side of the screen, navigate to the "Results" tab.
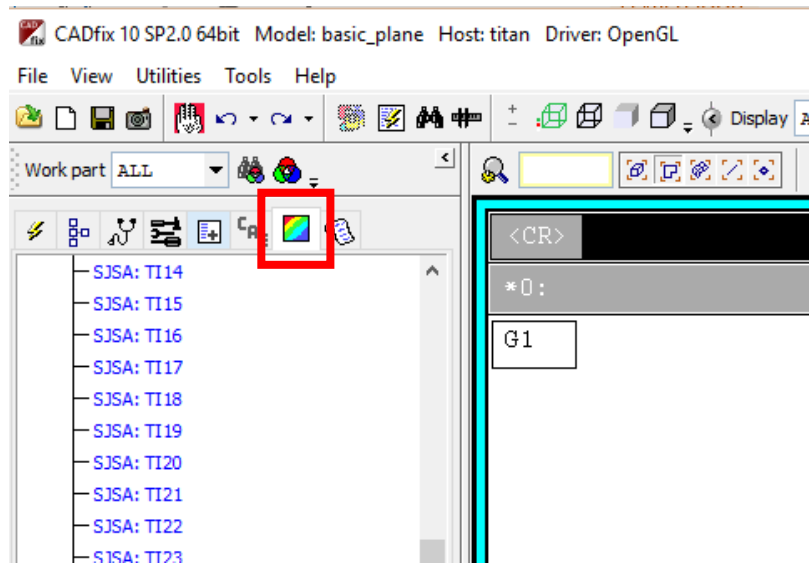
2. In the lower left corner of the CADfix screen, find the "Open results database" button and click it.
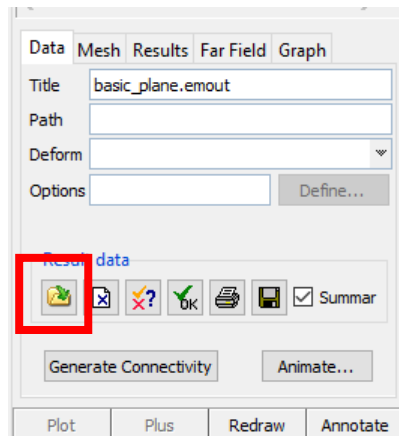
3. In the dialogue that opens, select the .frm file that was made by **famfromema3d** and click "OK."

4. In the lower left corner of the CADfix screen, find the "Check Results Database" button and click it. CADfix will check that the results database matches the geometry in the model. If the model has been modified and meshed since the simulation, this results might not match.
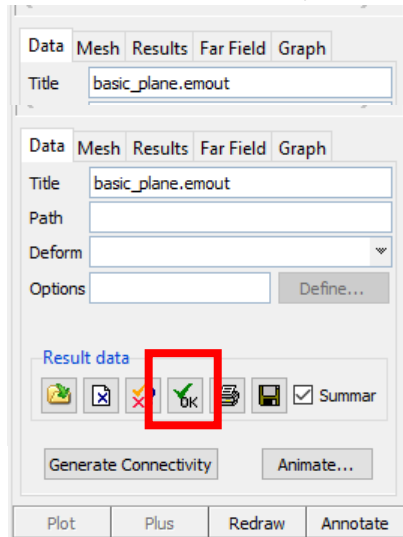
5. If the "Maximum node movement" displayed in the command line is 0.000, click the "Accept Results Database" button located right next to the "Check Results Database" button. Accepting a database will lock the mesh (it can't be changed) to keep the geometry and results paired.
6. In the left hand side of the CADfix screen, there should be entries for each time that the animation probe captured. Click one select the JCURRENT branch.
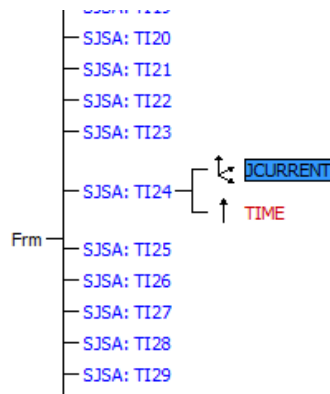
7. In the lower left corner of CADfix, navigate to the "Results" tab. Leave the sub-tab on "Contours." Select the "Tartan Contours" radio button.
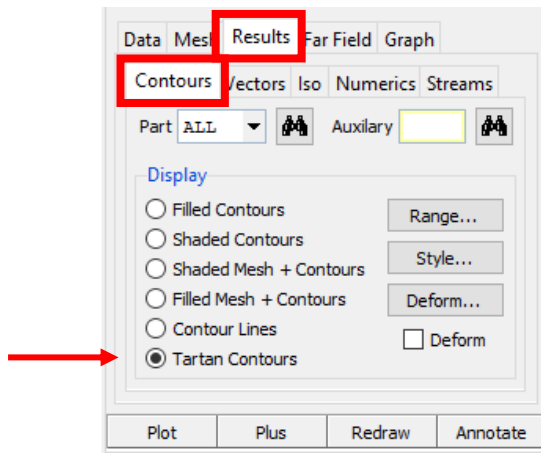
8. Click the "Plot" button to plot the surface current densities for the selected time step in the image.
9. CADfix automatically sets the color scale used. By default, it will change for each time step. To change it, click "Range…"
10. A warning will pop up. Click "OK." Nothing wrong has happened.
11. Change the radio buttons to "User defined" for both fields. Enter "0" in the "From" category and "4e4" in the "To" category. Click "Apply," then click "Close."
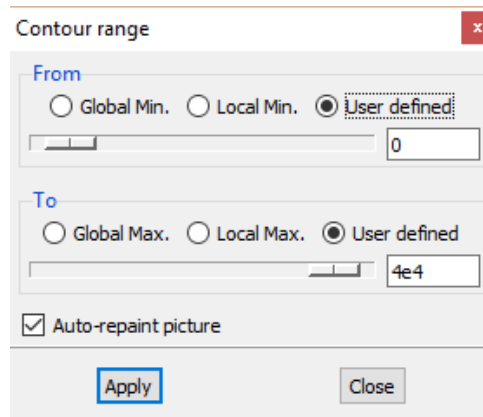
12. Rotate the image until the airplane is in a nice view. This will be the view used in the animation.

13. In the lower left corner of CADfix, navigate back to the "Data" tab. Click "Animate…"
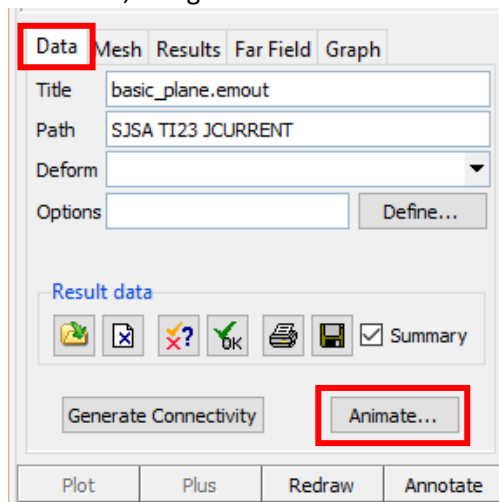
14. In the next window, stay in the "Multi-step" tab and select all of the animation steps. The contour range is saved from our previous edit (step 11). With all of the time steps selected, click "Capture."
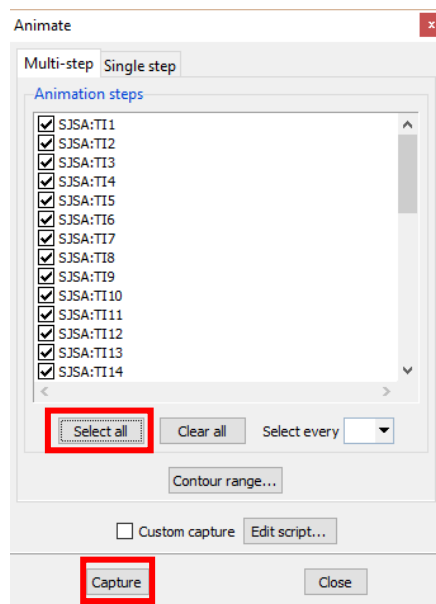
15. Watch as CADfix captures each frame of the animation.
16. Once all frames are captured, an "Animation Manager" window will pop up. To use it, close the "Animate" window.

17. The "Animation Manager" window has controls that allow you to step through the animation, play the animation, change the frame rate, and edit the animation.

Watching the animation and looking at the current probes, it's clear that some resonance is present in the simple plane (see current going back and forth between wings at later times).

18. When ready, click "File->Save As…" to save a .gif of the animation.

This marks the end of the required work for this training module. By now, you should be comfortable navigating the CADfix toolboxes and build tools. Basic command line controls (**PLOT, PLUS, SETA, SETO, SETC,** and **SWEP**) should also fee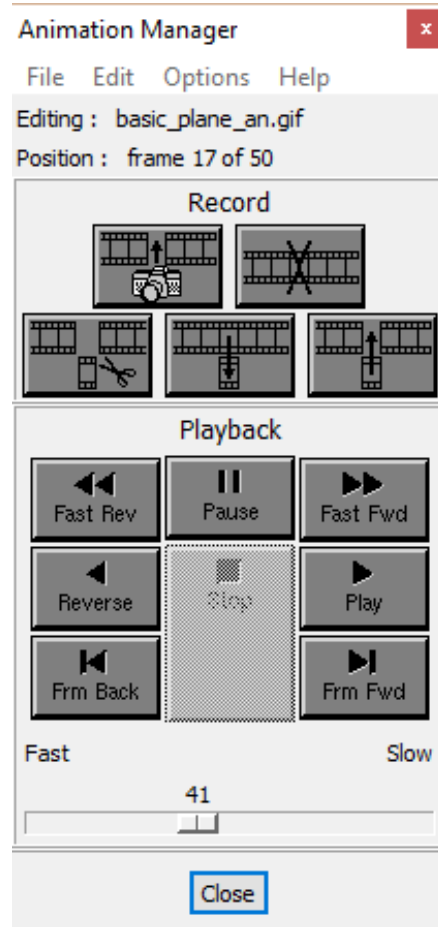l familiar. You should be able to perform simple geometry creation and modification using primitives, build tools, and/or typed commands (whichever you prefer).

The steps required to perform and analyze simple, one-off simulations have also been exemplified. It is useful to remember that the order of the steps are outlined in the labelling of the toolboxes. The EMA3D GUI should make sense and you should be able to run another .emin file without prompts.

As users become more comfortable with both CADfix and EMA3D, some of the processes shown in this module can be replaced with quicker, more purpose-built methods. Using GUI's to input thousands of material properties or to quickly generate hundreds of lines isn't reasonable, so quicker methods will be required. As these needs develop, CADfix and EMA3D come into their own with their customizability and versatility. User-created macros are a must-have in CADfix and batch control of EMA3D greatly increases

efficiency. As more complex needs arise, contact EMA for help and advice from their experienced scientists.

Practice Problems

Extra tasks are included in this module with minimal instructions. Hints will be provided with each task. In particular, general procedures and methods will be suggested. The methods needed to perform each task have already been exemplified earlier in the module.

Each task will be motivated to be useful beyond just getting more CADfix and EMA3D practice. Tasks will include geometry modifications and new simulation parameters. The motivation for each task will be presented.

Before making any modifications, save the model with a new file name to archive each step.

If a result database has been loaded and accepted to a model, to be able to make any changes you will have to type:

PROC LOCK OFF

Without the above command, CADfix won't allow any changes (it's trying to prevent mismatches between model and results).

Boundary Conditions

As an easy first problem, try changing the boundary condition to the electric field mur 1. After that, all you need to do is create a new emin file. No other changes are necessary. Compare the old and new results. The data using the electric field mur 1 is dominated by some sort of resonance. We will explore whether this is a result of numerical error (difference of similar numbers, operations of disparate numbers, etc.) Use these boundary conditions for the following exercises. We will explore the resonance and try to mitigate it.

The model and data can be found in the folder named "emur."

Swept Wing

The first simulation had wings perpendicular to the fuselage. Sweeping the wings might make the wings a less effective resonating chamber (which could contribute to the resonance we saw earlier). Add a 5 meter sweep to the wings.
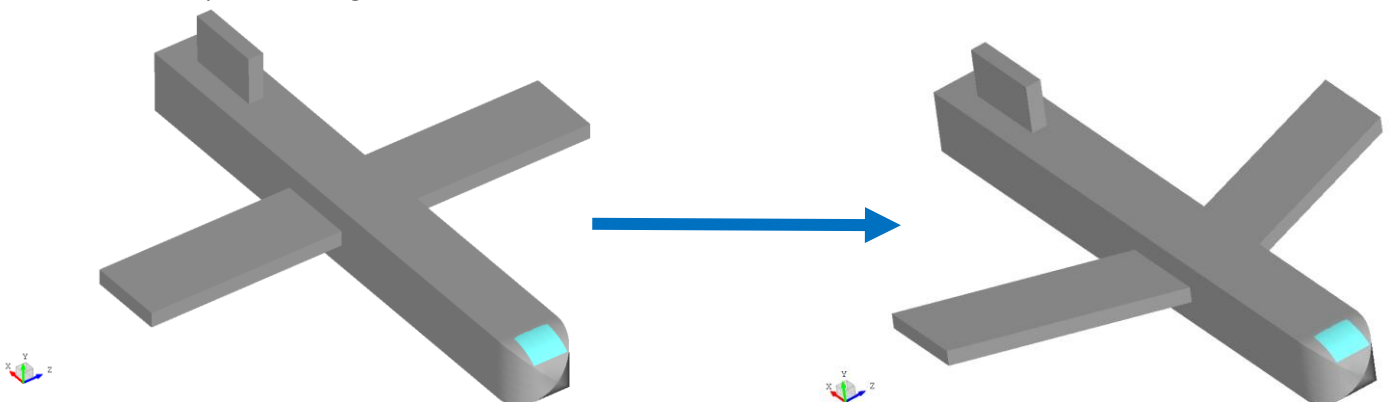
Hints: Move the wing root forward 5 meters using "Move/Copy" or **MOVE**. If after moving the wing root the shaded surfaces don't display well, type **SSIM S PLANE ANA** to fix the surfaces. You will need to mesh the model after modifying it. After meshing, check the probe locations. Redefine the source (this needs to be done because the node numbers changed since the last time it was defined).

After meshing, make a new emin file and run it. Check if the resonance is still present. If you want to have an animation, you will need to redefine the SJSA set (not its properties). Repeat the steps outlined in the EMA3D step 12 section of this module to put surface nodes in SJSA.

The wild oscillations are gone. This most likely means that the oscillations occurred due to numerical error. This can happen when very similar fields appear on adjacent nodes and are calculated. Fundamental imperfections in the way numbers are represented in computers limit the available precision. The result is that numbers that are similar to within a certain precision can lead to errors. This can be exemplified very simply:

```
>>> a=2+1e-10
>>> a
2.0000000001
>>> a-2
1.000000082740371e-10
>>>
```

It is likely that sweeping the wings created more strongly varying fields inside the wings, eliminating the numerical oscillations.

Changing the boundary condition reduced the boundaries' ability to absorb the currents generated in the problem space. Any instabilities using the magnetic Mur boundary condition flowed out of the problem space through the absorbing boundaries.

The data and model can be found in the folder "swept."

Cylindrical Fuselage and Hemispherical Nose

Maybe changing the fuselage shape will change the resonating characteristics of the plane, too. Change the fuselage to a cylinder instead of a box. After that, make the cockpit a hemisphere.

Fuselage hints: Be mindful of maintaining proper set organization. Start by making a cylinder using a primitive. Use the same length and total width (diameter). Remember to delete the solid body (leave the surfaces). After the cylinder is made, delete the old box. Use "Move/Copy" to project the wing root and tail onto the cylindrical shape of the fuselage. Project the surfaces using two points that will keep the projection purely in the X-Z plane.

Cockpit hints: To make a hemisphere, first make a sphere at a point (tool in the primitive toolbox). Delete the body, then delete the aft half of the sphere. Don't worry about making a window. In fact, just **ZAP** the window set. Don't forget to delete the old rectangular cockpit, too.

You will have to assign properties to any new geometry you created (fuselage and cockpit). You will also need to mesh the whole model again. After meshing, redefine the source. If you want an animation, redefine the SJSA set (not its properties).

Process the data in any way you prefer.

The data and model can be found in the set "cylinder."