

# Operations Manual for Creating Latitudes for Plug Fabrication from Longitudes of a Solar Car

Course: EML 4451/2, Senior Design

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# Abstract

## ***Definitions***

- Design plug: (n.) The framework or skeleton of a solar car.
- Orthogonal: (adj.) Perpendicular, normal
- Profile / Longitude / Centerline / Edge: (n.) This is what would appear if the solar car was cut vertically in lengthwise direction and viewed from a perspective orthogonal to the cutting plane.
- Projection / Latitude / Cross-Section: (n.) This is what would appear if the solar car was cut vertically in the cross-wise direction and viewed from a perspective orthogonal to the cutting plane.
- Object: (n.) Any shape drawn in AutoCAD
- Polyline: (n.) Joined objects
- Loft: (n.) A complex shape that you create through a series of planar closed loops or sections using Mechanical Desktop.

## ***Theory of the System***

- System Purpose: This system is a series of steps outlining a method of creating a 3D computer model of a solar car using AutoCAD and Mechanical Desktop (MDT).
- Fundamental Assumptions (regarding the educational foundation of the user):
  - The user should understand how a scale model or full-size solar car is constructed using the method employed by the Florida Agricultural and Mechanical University-Florida State University College of Engineering (FAMU-FSU CoE).
  - The user should understand basic concepts of two-dimensional sketching and three-dimensional solid modeling.
- Instruction Overview: The user begins with two longitudinal profiles (drawn in AutoCAD) representing the shape of the edge and centerline profiles. The user must decide upon locations for each of the latitudes. Generally, there will be a

higher concentration of latitudes where there curvature of the centerline is more extreme. Next the user will draw lines marking the position of the latitudes and various functions in AutoCAD to manipulate these markers such that they can be used to create a latitudes. The user will then use MDT to position the latitudes and finally to loft them into a solid object.

## **Warnings**

- Please observe the following
  - Only operate a computer when fully alert
  - When using the a computer, sit using an ergonomically correct posture (i.e. back is straight and upright, only fingers touch the keyboard)
  - Take frequent breaks
  - Do not wait until the last minute to attempt this process
  - Save your files frequently
  - When in doubt, ask for help
  - Do not use this manual as a doorstop, hand fan, doodling pad, etc. (somebody worked very diligently to create it)

## Frequent Operations

### ***To Open a AutoCAD or Mechanical Desktop***

1. Click the Start Menu, highlight Programs, then highlight and click on AutoCAD or Mechanical Desktop.
2. Click the Continue button when prompted.
3. Click the Open button at the top left portion of the screen.
4. The Open dialogue box will appear.
5. Navigate to the location of the AutoCAD file containing the centerline and edge longitudes, highlight the file name by clicking on it once and then click the Open button.

### ***To create a NEW file***

1. Click File → New (or press **Control + n** simultaneously).
2. Press **Enter**.

### ***To OPEN a file***

1. Click File → Open (or press **Control + o** simultaneously).
2. The Open dialogue box will appear.
3. Navigate to the directory/folder containing the file that is to be opened.
4. Click **OK**.

### ***To SAVE a file***

1. Click File → Save (or press **Control + s** simultaneously).
2. The Save dialogue box will appear.
3. Navigate to the directory/folder where the file will be saved (if necessary).
4. Enter a name for the file.
5. MDT and AutoCAD will add the proper file extension (.dwg) to the file automatically.
6. Click **OK**.

### **To SAVE a copy of a file**

1. Click File → Save As.
2. The Save As dialogue box will appear.
3. Navigate to the directory/folder where the file will be saved (if necessary).
4. Enter a new name for the file
5. MDT and AutoCAD will add the proper file extension (.dwg) to the file automatically.
6. Click *OK*.

### **To COPY objects within a file**

1. Type 'copy' or 'co' in the Command Line at the bottom of the window.
2. Select the objects to be copied with the mouse and press **Enter**.
3. When prompted to specify a base point or displacement, pick a point in the workspace to place the object by clicking once.
4. Move the cursor (and hence, the objects) to their desired location.
5. Click once more to define the second point of displacement.

### **To COPY objects to another file**

1. Select the objects to be copied with the mouse and press **Control + c**.
2. Open the second file if it is not already open and press **Control + v**.
3. When prompted to specify a base point or displacement, pick a point in the workspace to place the object by clicking once or enter Cartesian coordinates.

### **To MOVE objects within a file**

1. Type 'move' or 'm' in the Command Line at the bottom of the window.
2. Select the objects to be moved with the mouse and press **Enter**.
3. When prompted to specify a base point or displacement, pick a point near the objects to be moved in the workspace by clicking once.
4. Move the cursor (and hence, the objects) to their desired location.
5. Click once more to define the second point of displacement or enter a distance.

6. It is often helpful to toggle on the ORTHO Command (F8), which permits vertical and horizontal straight-line motion only.

### ***To TRIM objects***

1. Type 'trim' or 'tr' in the Command Line and press ***Enter***.
2. Click an object to select it as the as the cutting edge and press ***Enter***.
3. Click the portion of each of the latitude position markers that extends above and below (if applicable) the edge longitude and press ***Enter***.

## Creating Lines to Construct Latitudes

1. Make sure that the ORTHO button at the bottom of the screen is activated (press F8 to toggle ORTHO on and off).
  - a. Copy the edge longitude and latitude position markers.
2. Trim the latitude position markers such that they are contained within the copied edge longitude.
3. Arrange the drawing by moving, copying and pasting objects as necessary.
4. Arrange the drawing as shown by moving, copying and pasting objects as necessary.
5. Copy box #3,
  - a. Place it below box '3'.
  - b. Label it '4'.
  - c. Draw three short lines.
  - d. Select the three reference lines and delete them.
6. Copy box #4,
  - a. Place it below box '4'.
  - b. Label it '5'.
  - c. Copy the set of three short lines.
  - d. At this juncture, each centerline height and edge height is known for all latitudes.

## Constructing Latitudes

1. Move the objects as shown at [http://www.eng.fsu.edu/ME\\_senior\\_design/2002/folder11/stage\\_1.htm](http://www.eng.fsu.edu/ME_senior_design/2002/folder11/stage_1.htm)
  - a. Repeat step 1 in this section for each remaining latitude.
2. When these latitudes are copied into MDT it will be easier to move them if they are joined as one object instead of five objects (as they currently are). Joined objects are called polylines.
  - a. To make the latitudes polylines, type 'pe' or 'pedit' in the Command Line and press **Enter**.

- b. Pick the upper left white line by clicking on it.
  - c. When prompted to change this line to a polyline, press **Enter** to accept the default response (yes).
  - d. Type 'j' in the Command Line and press **Enter** when prompted to enter an option to join the line that was picked in step 2b with the rest of the lines comprising the latitude.
  - e. Select the lines and press **Enter twice**.
  - f. The two red lines at the left and right will change to white lines.
  - g. Delete the vertical red line by selecting it and pressing **Delete**.
  - h. To change the entire polyline to red, select it and move the cursor to the color box and select 'red'.
3. If it is necessary to add radii to the latitudes type 'fillet' in the Command Line and press **Enter**.
  - a. Type 'r' in the Command Line and press **Enter**.
  - b. Enter a value for the radius and press **Enter**.
  - c. Pick the two lines that form the upper left "corner" of the latitude.
  - d. Repeat this fillet process for each of the "corners".
  - e. Repeat this fillet process with a different radius for the vertex.

## Creating a Loft from Latitudes

1. Open Mechanical Desktop
  - a. Click File → New
2. Press **Enter** when prompted for a template file name.
  - a. This step selects the default template file, acad.dwt
3. Click the Continue button when prompted.
4. Click Insert → Block to insert the centerline longitude, which is used to properly align the latitudes
5. The Insert dialogue box will appear.
  - a. Click the Browse button located at the top of the Insert dialogue box.
6. Navigate to the file containing the centerline longitude that you want to use and double-click on the file name.
7. Click the OK button located at the bottom of the Insert dialogue box.
8. When prompted to specify an insertion point, enter 0,0,0.
9. Click View → Zoom → All
  - a. The centerline longitude will probably not be displayed on the screen, so it is necessary to zoom out.
10. Click the Toggle Shading / Wireframe button located at the upper right portion of the screen.
  - a. The x- and y-axes will change from white to red and green and the z-axis will appear as blue.
11. Click the 3D Orbit button located three buttons to the left of the Toggle Shading / Wireframe button at the upper right portion of the screen.
12. A green circle with rotation handles will appear, rotate as necessary.
13. Before the latitudes can be inserted, work planes must be created at each of their locations. To allow these work planes and latitudes to be inserted orthogonal to the centerline longitude, click the New Sketch Plane button located at the left side of the workspace.

- a. When prompted to select a work plan or planar surface, press **y** to select the YZ-plane, which is orthogonal to the World XY-plane, in which the centerline longitude was drawn.
  - b. When prompted to select an edge with which to align the x-axis, left-click three times to rotate the x-axis.
  - c. Right-click once to accept this orientation.
  - d. Notice that the red, green, and blue coordinate axes have changed orientation.
14. To insert work planes, click the Work Features – Work Plane button located at the left side of the workspace.
- a. The Work Plane dialogue box will appear.
  - b. Select the radio button corresponding to Planar Parallel in the 1<sup>st</sup> Modifier area.
  - c. Select the radio button corresponding to Offset in the 2<sup>nd</sup> Modifier area.
  - d. Enter an Offset of corresponding to the distance between the nose of the solar car and the first latitude, which in this case is 300 and press OK.
  - e. When prompted to select a work plane or planar surface to offset the new work plane from, press the **y** to select the World YZ-plane.
  - f. Right-click once to accept this orientation.
  - g. Left-click three times to change the orientation of the blue XY-axes.
  - h. Right-click once to accept this orientation and the work plane will appear.
  - i. Repeat this process until all necessary work planes have been created.
15. For the loft to work properly, each latitude must be inserted/sketched in its corresponding work plane.
- a. To make the first latitude's work plane the active sketch plane, click the New Sketch Plane button and when prompted click the first latitude's work plane.
  - b. Left-click three times to change the orientation of the blue XY-axes
  - c. Right-click once to accept this orientation.
16. Open the AutoCAD file containing the latitudes.
- a. Select the first latitude.

- b. Press ***Control + c*** to copy the first latitude from the AutoCAD file.
- c. Press ***Control + v*** to paste the first latitude into Mechanical Desktop.  
When prompted to specify an insertion point, simply move the latitude near the first work plane and click once to perform the insertion
- d. Click View → Zoom → Window and draw a box around the latitude and nose of the centerline longitude.
- e. Type 'move' in the Command Line at the bottom of the window and press ***Enter***.
- f. When prompted, select the first latitude and press ***Enter***.
- g. Make sure that the Midpoint Object Snap option is selected by right-clicking on the *OSNAP* button at the bottom of the window and clicking *Settings* from the pop-up menu that appears upon right-clicking.
- h. The Drafting Settings dialogue box will appear. Click the box next to Midpoint to activate the Midpoint Object Snap and then click *OK*.
- i. When prompted to specify a base point or displacement, select the midpoint of the vertex of the latitude.
- j. When prompted to specify a second point of displacement, select the end point of the latitude position marker on the centerline longitude.
- k. Click View → Zoom → All to see the entire drawing and switch the sketch plane to the second work plane created.
- l. Repeat this process until all latitudes have been inserted to their desired location.

17. Before the loft can be performed, the sketches must be profiled within their respective work planes.

- a. Click the *New Sketch Plane* button at the left of the workspace and click on the first work plane.
- b. Click the *Profile a Sketch* button at the left of the workspace.
- c. Click the first latitude.
- d. Press ***Enter***. A profile will be created in the model tree at the left of the workspace.
- e. Repeat this process until all sketches have been profiled.

18. Once the profiles have been completed, it is useful to create a separate file that contains the loft because the loft can be undone in some cases and not in others.
- When starting from scratch (i.e. inserting the centerline longitude, creating work planes, inserting latitude sketches, profiling the latitude sketches, and finally lofting the profiled sketches), the loft can be undone, yielding the set of profiled latitude sketches. Mechanical Desktop keeps track of each command performed by the user starting from when the program was opened.
  - However, when opening a saved loft, there are no saved commands to undo. Thus the loft cannot be undone.
  - Click File → Save As and give the file a name such as model\_x\_profiled\_latitudes.
  - Click File → Save As and give the file a name such as model\_x\_lofted\_latitudes.
19. In all likelihood, the profiles will not be able to be lofted in one single operation because of the complex geometry of the shape. The user will receive an error message, thus the loft must be performed in two or three steps. During the 2001-2002 academic year, this exercise's emphasis was placed mostly on creating the geometry from the nose of the car to the canopy area to make sure that the curvature of the body was gentle enough to permit fastening of photovoltaics. Consequently, the resulting lofts are done in sections from the first latitude to the eighth latitude and then from the eighth latitude to the eleventh latitude.
- To loft the profiled latitude sketches *click and hold* the Sketched Features button and click on the Loft button from the pop-out menu that appears.
  - When prompted, click each of the profiles in the groups stated above starting at the front of the drawing and moving backwards and press **Enter**.
  - The Loft dialogue box will appear.
  - Click OK.
  - The lofts should appear as shown at [http://www.eng.fsu.edu/ME\\_senior\\_design/2002/folder11/3d\\_body.htm](http://www.eng.fsu.edu/ME_senior_design/2002/folder11/3d_body.htm)

## **Closing Remarks**

After using this comprehensive operations manual for the process of creating latitudes from longitudes and lofting in MDT, the user should be able to complete the following tasks:

- Open AutoCAD and MDT
- Open, save, and create new AutoCAD and MDT files
- Sketch, copy, paste, move, and trim objects in AutoCAD or MDT
- Zoom in and out, and toggle on and off OSNAPS and Shading in MDT
- Create work planes, change the orientation of the User Coordinate System, make profiles, and create lofts in MDT

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