Steps to draw 3D Worm Gear using SolidWorks 2013
EGN 1110C: Engineering Drawing

1.) Draw two circles on the “Front Plane” centered at the origin, give the correct dimension using “Smart Dimension” and Click “Exit Sketch”

2.) Select the sketch you have drawn, and go to Features Toolbar, Select “Extrude” using “Mid Plane” mode to make it symmetry with the front plane. Define the distance = 1.00in
3.) Turn the view orientation to the “Right Plane”

4.) Create new sketch by selecting “Circle” and select right plane to draw (To select the right plane, expand the design tree manager appeared on the workspace)
5.) Draw the circle centered on the upper edge of the extruded feature and using smart dimension to modify the diameter to 0.625 in. Then, click exit sketch.

6.) Go to Feature Toolbar, select “Reference Geometry” then select “Axis”
7.) On the property manager side bar, select “Cylindrical/Conical face” option and then select the surface of the cylinder as the reference to create the axis.

8.) After clicking “OK”, The reference axis is created as shown, we will use this axis to be the axis of revolution in revolved cut feature.
9.) On the design tree manager, select circle sketch the you have already created, then on the Feature Toolbar, Click on the “Revolved Cut”. Choose the “Axis1” that you created as the “Axis of Revolution”.

10.) After clicking “OK” (Check mark), the revolved cut is created as shown.
11.) Create the new sketch on the right plane by clicking on “Line” and select the “Right plane” by expanding the design tree appeared on the workspace.

12.) Draw the shape as shown on the edge of the object.
13.) Use “Smart Dimension” to modify the dimensions of the shape as shown. After that, click OK and exit sketch mode.

14.) Select the shape you’ve just drawn, go to Feature Toolbar, select “Revolved Cut”. Choose the “Axis1” as the axis of revolution. Then click “OK”.

15.) The Revolved Cut is finished at the back of the object as shown.

16.) Select the Revolved Cut that you've just created from the design tree manager, go to Features Toolbar, select “Mirror”. Choose the “Front Plane” as the mirror-referencing plane (Expand the Design Tree Manager). Then click “OK”
17.) The Revolved Cut is created at the front and the back of the object symmetrically as shown below:

18.) We will modify the outside perimeter to have the bevel edge using “Chamfer”. On the Features Toolbar, click the drop-down arrow below the Fillet, select the tool “Chamfer”.

19.) Select outer edges both the front and the back edge to chamfer, giving distance = 0.11in and the angle = 45 degree. Then click “OK”

20.) Once you finished the Chamfer, click on “Fillet”, we will modify four inner sharp edges of the worm gear.
21.) Select four inner edges of the worm gear as shown and define the fillet radius = 0.0625in. Then Click “OK.”

22.) The four round inner edges (Fillet) are created as shown.
23.) We can hide the “Axis1” since we no longer use it. On the Design Tree Manager, “Right click” on the “Axis1” and click on the “glasses icon” to hide. (You can unhide it later by also clicking on the glasses icon.)

24.) The “Axis1” is disappeared as shown below.
25.) Turn the design to the “Front View”, we will create the Keyway Cut.

26.) Draw the Rectangular sketch (0.150in x 0.400in) on the front-most surface of the object which its base are centered at the origin as shown using the “Smart Dimension” to modify the width and the height. Then click on the “Exit Sketch” to finish sketching.
27.) Select the rectangular you sketched from the design tree manager, On the Features Toolbar, select “Extruded Cut” and choose the “Through All” option. Then click “OK”!

The Worm Gear is completed!!