The following set of instructions are an optional replacement for the “Feature Based Modeling”. This demo should help prepare the students for the Out of Class HW (Note offset for 12mm hole(s) is 15mm).

Feature Based Modeling

Discuss the various construction options available and indicate that sketching on the FRONT PLANE greatly reduces the work involved. Show the full size picture of the object located near the end of this document.

1. Edit the Front Plane and draw infinite length Horizontal and Vertical construction lines through the origin and set the Unit System to IPS.

2. Draw 2 concentric circles on the origin, sketch the outline and dimension as shown and make the slot SYMMETRICAL about the Horizontal construction line. Set the paired horizontal lines equal.

3. Trim the larger circle and dimension the slot height and depth as shown. Note that the sketch is Fully Defined!

4. Use the Fillet tool with a radius of 0.25 to create the 1st internal round and click Yes on the dialog box to temporarily over-ride the equal constraints. Repeat for the 2nd round.
5. To create the shown **Fully Defined** sketch.

6. Exit the sketch, select ISO button (Ctrl+7) and select any line on the sketch and then Extrude Boss/Base 1” in both directions.

7. Select the top surface and Edit Sketch, orient using Ctrl 8 or Normal To in View Orientation and select Perimeter Circle and sequentially select the top, right and bottom edges to produce the shown sketch.

8. Using the Line tool, create the shown sketch which provides a "closed" surface for the next Extruded Cut and trim the left half of the circle.
9. Exit Sketch and under Feature select **Extruded Cut**, select the two surfaces as shown, select the "Through All" option and accept the cut.

10. To produce the object shown.

11. Select the **top surface** and open for editing. Click on "Normal To" in View Orientation or Ctrl 8 and draw a **Center Point Circle** at an **arbitrary location**. Note that this hole could have been drawn in the previous sketch but is separated here for ease of executing the Extruded Cut and for demonstrating the Concentric Constraint.

12. Add **Concentric** constraints to the 2 circles and **Dimension** as shown. Note that the sketch is **Fully Defined!**
13. Accept the sketch, select ISO button (or Ctrl+7) and click on Feature and the **Extrude Cut** selecting the "Through all" option and accept.

14. To prepare for cutting the slots, change the Display Style to **Hidden Lines Visible**.

15. Select the **Top Plane** and open for Editing. Draw 2 Corner Rectangles on the left hidden line as shown.

16. To prepare for the creation of the slots draw both **Horizontal** and **Vertical** construction lines through the origin.

17. Make all sides of the rectangles **EQUAL** and the 2 rectangles **SYMMETRIC** about the Horizontal construction line using either paired elements (see example in **RED**).
18. **Dimension** the rectangle side and offset as shown. Note that the sketch is **Fully Defined!**

19. Exit the sketch and select Isometric under View Orientation or Ctrl+7. Before executing **CRITICAL** that your previous Sketch remains **HIGHLIGHTED!**

20. With the **Sketch4 HIGHLIGHTED(BLUE)** select Feature and the Revolve Cut and click on the "Z" axis construction and accept the Revolved Cut.

21. Change Display Style back to Shaded with Edges to produce the object as shown.
22. Under Fillet, select **Chamfer** with a length of **0.05** and click of all 4 circle edges as shown. Accept the Chamfer.

23. To produce the final object as shown.
24. With all dimensions as shown in the Extracted Drawing below and **Fully Defined**. Do not produce Extraction (Extraction is addressed in SolidWorks 6).

Optional Feature illustrated on next page.
How would you add these features? [This additional feature is not required if this object is submitted as the in class assignment]

Positive Hemisphere

Negative Hemisphere

Draw this pattern and revolve around about axis and MIRROR to produce feature on both sides of object:

1. Revolved Boss/Base
2. Revolved Cut

To MIRROR, the FRONT PLANE must be made visible by selecting it and clicking on the eye glasses icon. Reverse process to hide the plane after completing mirroring.
25. (Use either steps 25 & 26 or discuss the in class assignment on the EEIC website) Select the Front surface and using Edit Sketch open a text box and complete label using point size of 22. Position the text by grabbing and moving the DOT. With the DOT still highlighted “BLUE”, under Display/Delete Relations select Add a Relation and click on FIX icon and accept with green arrow.

26. Select Zoom to Fit, print and submit object as optional in class assignment.