Creo Revolve Tutorial

Setup

1. Open Creo Parametric
   
   Note: Refer back to the Creo Extrude Tutorial for references and screen shots of the Creo layout

2. Set Working Directory
   
   a. From the Model Tree navigate to your folder > right mouse button > Set Working Directory

3. Click on the icon New > Part > name file Revolvetutorial (remember is must be on string no spaces) > Ok

4. Basic setup features of mouse and hotkeys
   
   a. Solid Model Navigation Tools
      
      i. Mouse
         
         1. **Left Mouse Button**: Selects objects
         2. **Roll Bar**: Zooms in and out on the model screen.
         3. **Hold Down Roll Bar** (Middle Mouse Button): Rotates Model
         4. **Hold Right Mouse Button Down**: Gives pop out menu based on the design tab.

      ii. Hotkeys
         
         1. **a**= Autoscale
         2. **i**= Isometric
         3. **d**= Turns Datums on and off

Revolv

Revolving is the method of creating a solid by drawing a closed profile and rotating around an axis line. The profile may only be drawn on one side of the axis line, this will avoid intersecting material as the closed profile rotates around the axis

1. Under the Model Tab > click on the Revolve Icon

2. Hold Right Mouse Button in the work area (Black Background) > select Define Internal Sketch > select Front Datum > Click Sketch on the Pop-up Window

3. Place Axis Line: This is the line you want your profile to spin around
   
   a. Under the Sketch Tab select Centerline Icon to the right of the Arrow tool. Note there is another Centerline Tool that says “Centerline” next to the icon, this will not work for an axis
   
   b. Left Click on the Origin (Center of the workspace, where the two Datums cross each other) > Move the cursor so the centerline is horizontal > Left Click again to place the centerline. This will place the centerline on top of the Top Datum. You may not be able to see it. If the user does not place this line then the software will not be able rotate Profile
4. Draw the following profile. Avoid creating Equal Length Lines (Software will show an L next to each line if equal)
   a. Note Profile
      i. Solid: Placing the Profile on top of the axis line will create solid material
      ii. Hole: Placing the Profile offset from the axis line will create a hole in the center of the object
   b. Profile Edges
      i. Edges Parallel with the axis line will become curved surfaces on the solid model
      ii. Edges Perpendicular to the axis line are linear edges

If Diameters Do NOT show: Placing Diameter Dimensions
   c. Four Clicks of the mouse will be needed to place a dimension
   d. Under Sketch Tab click on Normal Icon
   e. Mouse Clicks
      i. Left Click on Axis Line
      ii. Left Click on Parallel Edge
      iii. Left Click on Axis Line (second time)
      iv. Middle Mouse Button anywhere to place the dimension

   Note Dimensions will appear to measure into blank space. The software is showing where the other side of the revolve edge will appear

   Sketch Tab > Click the Green Check to lock in the sketch profile > This will take you to the Revolve Tab. Buttons are similar to the Extrude (See Extrude Tutorial)
A Solid Object should show up rotate the part around to examine the part. If no solid shows up it means you did not set your axis line. To fix Click on Placement (on the Revolve Info Box) > Click Edit next to sketch > this will take you back into the sketch > Place Axis line> Green Check

f. Note Blind Distance represents number of degrees of rotation (We will always rotate 360 degrees)
g. Note this tutorial is creating a solid, but one can create revolve as a cut. See Challenges at the end of the Tutorial

5. Revolve Tab Click Green Check to lock in the object

Offset Datum

Offset Datums allow users to create new Datums or work surfaces inside or outside of the part. This allows the user to place a profile in space other than on the default Datums or predefined solid.

6. If Datums are off turn them on Hotkey D

7. Go to Model Tab > Click on the icon Plane (Pop-up Window will appear, this shows what Datums have been selected, Allows user to create offset or angled Datums) > Select Top Datum > In the Datum Pop-up menu the Datum will appear; In the Translation Box enter 0.75 (If Datum does not go above the TOP Datum, then type in -.75) If you are having a hard time seeing which direction it the object is going click Hotkey I (isometric) or Go to View Tab > Named Views > Default Orientation (Isometric) or any of the others
8. Extrusion Cut
   a. Model > Extrude > Select DTM1 > View screen will rotate 90 degrees to the viewer
   b. Now we will define some edges to measure from
      i. Sketch Tab > select References Icon
      ii. Click on the following edges seen below
      iii. Close References window when done

c. Sketch Tab > draw a rectangle as seen below > Adjust Dimensions > Green Check for Sketch > Extrude Info Tab Select the following
   i. Select the Cut Icon
   ii. Cut Through All
   iii. Make the cut go up (If wrong direction select Flip Direction)
d. Select Green Check (If Final Part does not look correct > go to model tree Left Click on the last extruded > Right Click on that extrude > Select Edit Definition; this will place you back into the extrude info box.)

9. Adding Material
   a. Extrude Icon > Hold Right Mouse Button > select Define Internal Sketch > Select the back surface of the part > Draw the following Rectangle with given dimensions

   
   Ignore the hole. Step comes later

   Extrude 1.00 into the part
10. Extrude circle cut in the following two location (Note you will do TWO separate extrudes)

**Location 1**
Depth: 0.25

**Location 2**

Place Reference Circle: Use Circle Tool to draw > Set Dimension diameter 3.5 > Select circle > Right Mouse Button > Select Construction

Depth of Hole = Blind 1.00
Part Should Look as Follows

Round Edges

11. Model Tab > select Round > Change size to 0.125 > Hold Control select the following edges. (Holding Control Key will make all of the edges the same size, not Holding Control Key will make the edges different sizes)

12. Green Check
13. Creating a Full Round from an existing edge
   a. Model Tab > select Round > select the following edges. Hold Control Key while your selecting

   b. Go to Sets Tab in the Round Info bar > select Full Round. This will automatically create a full round edge without having to type in a radius size > Green Check

**Grouping**

Grouping allows the user to put multiple features (extrude, round, chamfers, etc.) together to clean-up the Model Tree or modify the object (Mirror or Pattern)

14. First we need to reorder the model tree
15. Hold Left Mouse Key down on Round 2 and Drag it below Extrude 2. This will reorder the Model Tree allowing the grouping of the correct two operations. NOTE: If you have more than 2 Rounds listed you have missed a step. Go back and reread placement of the rounds.
16. Hold Control Key down > select Extrude 2 and Round 2 > Right Mouse Button > select Group > this will create a group of these two operations which will allows us to mirror both operations together

**Mirror**

Mirroring allows the user to select one or multiple existing Solid (Extrude, Revolve, etc.) and reflect the shape over a giving datum. In this tutorial we will be using a pre-existing datum

17. In the Model Tree Left Click on Group Local_GROUP resulting in selecting it.
18. Make sure the Datums are on (Hotkey D or select Datum Icons from View Tab)
19. Go to Model Tab > Select Mirror Icon (make sure Local_Group is still selected) > Click on Front Datum either in the Model Tree or in the work space > Click Green Check to complete the mirror.
Pattern

Patterning allows the user to equally space a given extrude. There are multiple ways to pattern.

1. Direction- Allows user to equally space a solid in a specified linear direction (Length, Height, or Depth). User can specify one or two directions creating a grid pattern
2. Axis- Allows user to rotate a profile based on an axis
3. Table- Allows User to create a table and equally or unequally space a solid

Direction Pattern

20. In the Model Tree select Extrude 3 > Go to Model Tab select Pattern
21. Rotate the view to Top View
c. Go to View Tab > select Named Views > Top View > this will rotate your image to a Top View Orientation
22. Go to Pattern Tab > Click on Dimension Drop Down Menu
23. Select the Linear .500 Dimension that measures from the back edge of the part to the center of the hole. (when selecting the dimension you are not selecting the value for the software but the direction) (we will only be patterning in one direction)
24. In the Dimension Drop Down Menu for Direction 1 place a value of 0.75 and change the number of iterations to 3 (See screen shot below) Black dots will show where the holes will appear

25. Green Check the Pattern
**Axis Pattern**

26. Click on Extrude 4 on the Model Tree > Model Tab > select Pattern > change the pattern type from Direction to Axis
27. Turn on Datums
28. Select the Center Axis running down the middle of the part
29. Change the iterations to 8 and the degrees to 45 degrees

View changed to Right Side from View > Named Views > Right Side

30. Green Check Pattern

**Final Part**

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**Challenges**
1. Optional Try to create a revolved cut.
   Hint:  1. Create a solid to cut away from first.
          2. Use an offset datum to place cut in the middle of the part