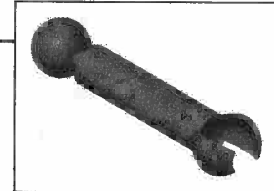


CHAPTER 5

Revolved Parts

Revolved Parts

Ball Joint Arm



- The Revolve command rotates one or more sketch profiles around a centerline, up to 360° to create a thin or a solid feature.

Open profile
= Thin Feature



Closed profile
= Solid Feature



- The revolved sketch should have a continuous closed contour and it can either be a polygon, a circle, an ellipse, or a closed spline.

Valid profile
1 continuous,
closed contour.



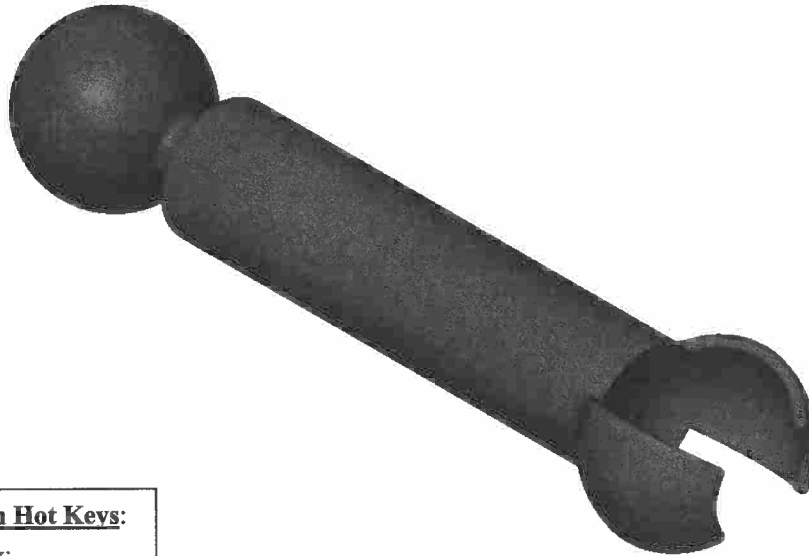
Invalid profile
Multiple closed
contours.



- If there is more than one centerline in the same sketch, the center of rotation must be specified when creating the revolve.
- In the newer releases of SolidWorks, the center of the revolve (centerline) can be replaced with a line, an axis, or a linear model edge.
- The revolve feature can be a cut feature  (which removes material) or a revolve boss feature  (which adds material).
- This chapter will guide you through the basics of creating the revolved parts.

Ball Joint Arm

Revolved Parts



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

Dimensioning Standards: **ANSI**

Units: **INCHES** – 3 Decimals

Tools Needed:



Insert Sketch



Line



Circle



Rectangle



Sketch Fillet



Trim



Add Geometric Relations



Dimension



Centerline



Base/Boss Revolve



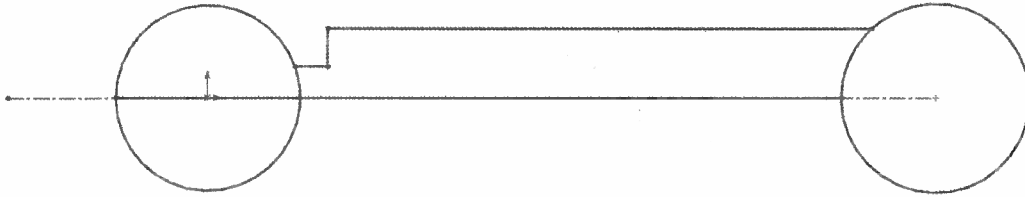
Fillet/Round



Mirror Features

1. Creating the Base Profile:

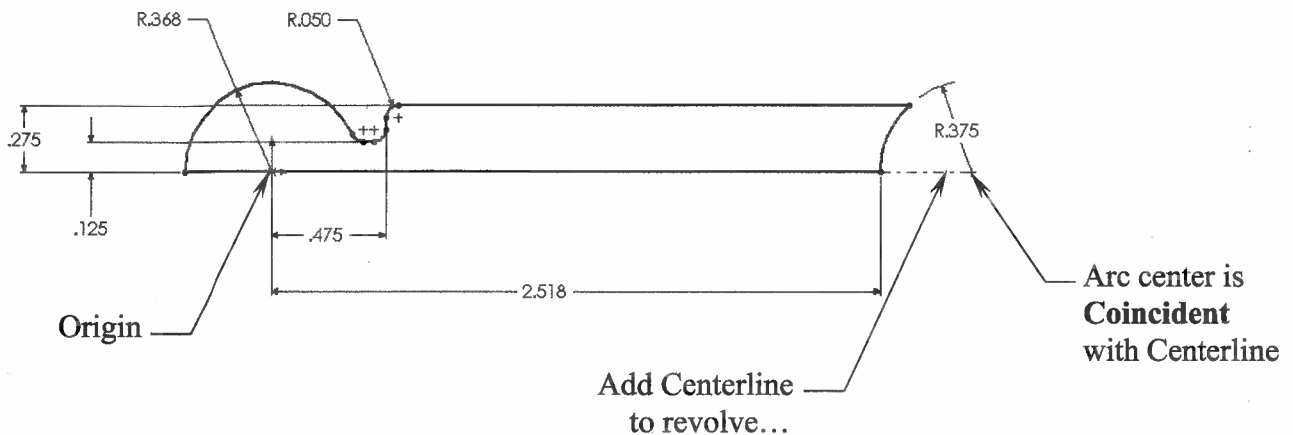
- Select the Front plane from the FeatureManager tree.
- Click  or select **Insert / Sketch**.
- Sketch the profile using the Line  and Circle  commands.



- Trim  the circles as shown.

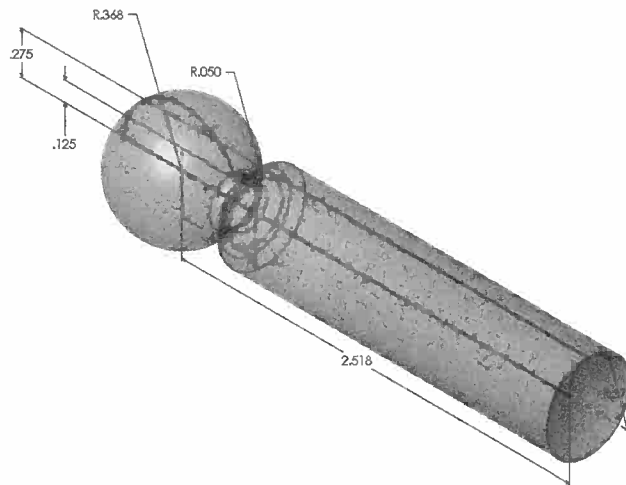


- Add Dimensions  and Relations  needed to fully define the sketch.
(It is easier to add the R.050" fillets after the sketch is fully defined).



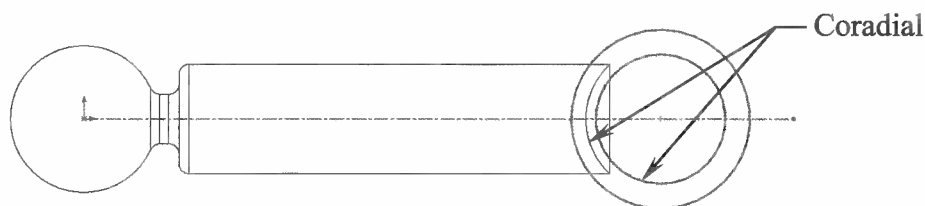
2. Revolving the Base Feature:

- Click **Revolve**  or select **Insert / Boss-Base / Revolve**.
- Revolve Type: **Blind** .
- Revolve Angle: **360 deg.** .
- Click **OK** .

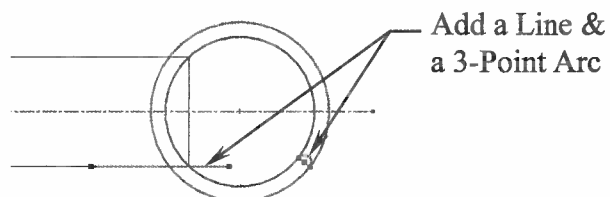


3. Sketching the Opened-End Profile:

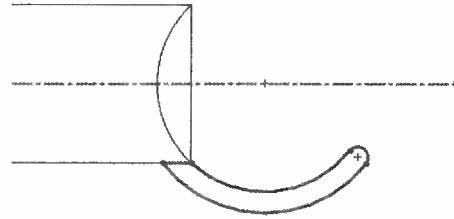
- Select the Top plane from the FeatureManager tree.
- Click  or select **Insert / Sketch**.
- Switch to Hidden Lines Visible mode: click  on the VIEW toolbar.
- Sketch 2 circles  as shown.
- Add a **Coradial** relation  between the small circle and the hidden edge.



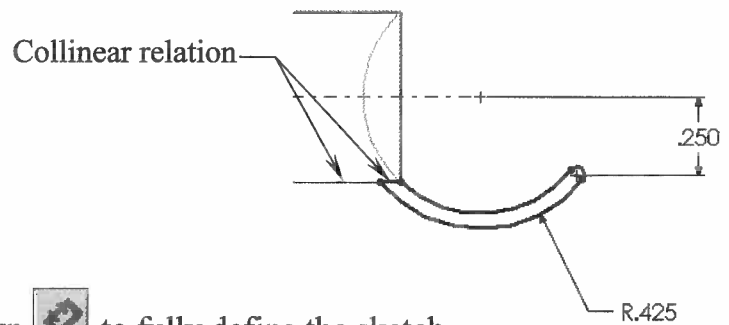
- Add a line along the bottom edge and a 3-Point Arc. The Arc is tangent with both circles.



- Trim  the line and the 2 circles.



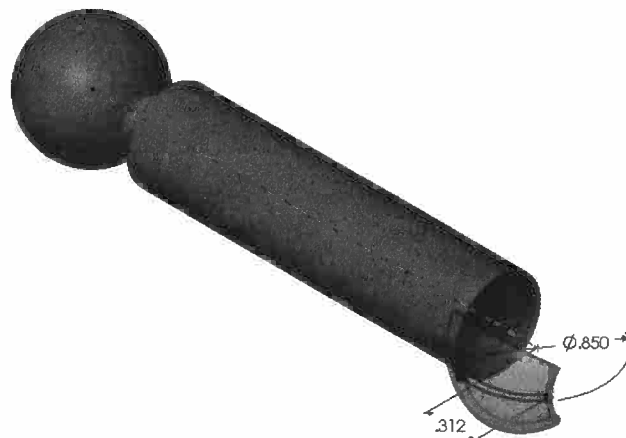
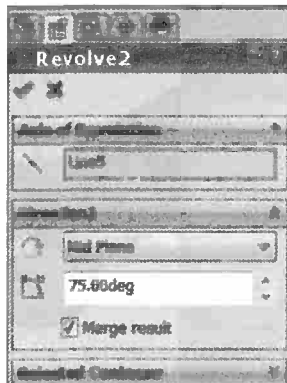
- Add a Collinear relation  between the line and the bottom edge of the part.



- Add the dimensions shown  to fully define the sketch.

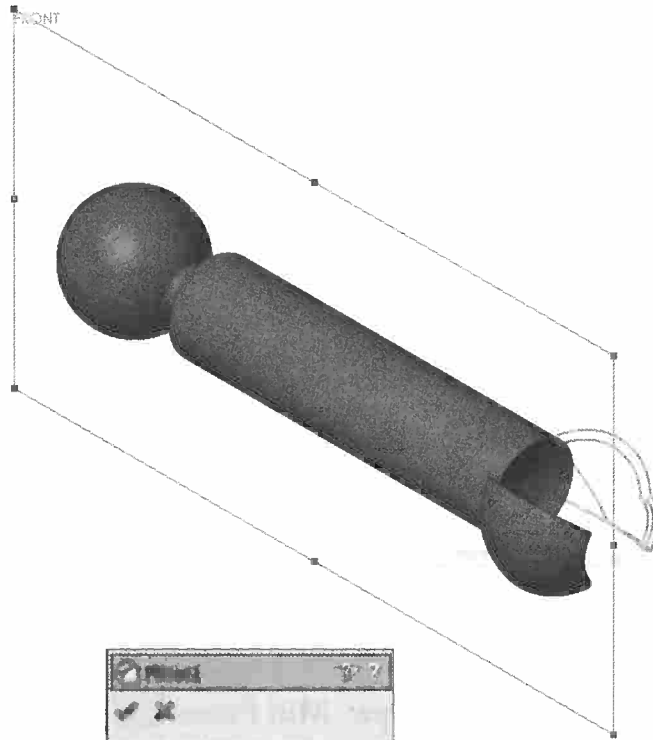
4. Revolving the Opened-End Feature:

- Click **Revolve**  or select **Insert / Boss-Base / Revolve**.
- Revolve Type: **Mid Plane** .
- Revolve Angle: **75 deg.** .
- Click **OK** .



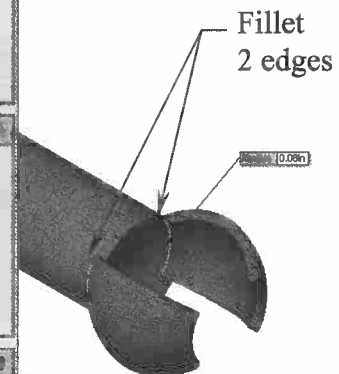
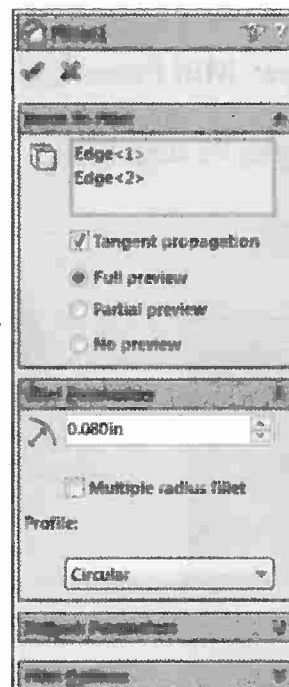
5. Mirroring the Revolved feature:

- Click **Mirror**  or select **Insert / Pattern Mirror / Mirror**.
- Select the Front plane from the FeatureManager tree as mirror plane .
- Select the Revolve2 feature either from the graphics area or from the FeatureManager tree, as Features to Mirror .
- Click **OK** .



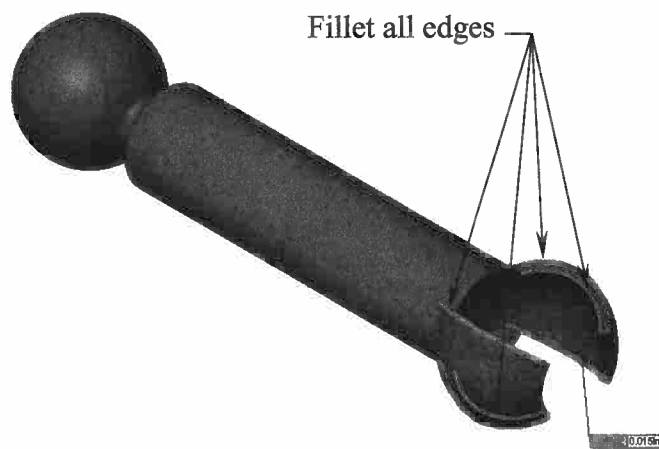
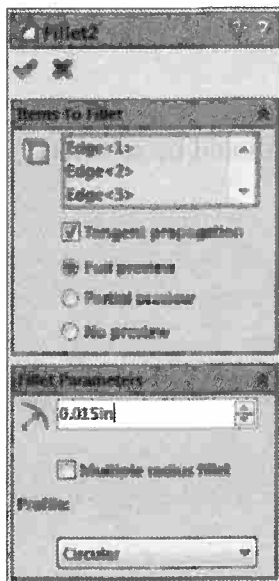
6. Adding the .080" Fillets:

- Click **Fillet**  or select **Insert / Features / Fillet / Round**.
- Enter **.080 in.** as the Radius .
- Select the two edges as shown for Items to Fillet .
- Click **OK** .



7. Adding the .015" Fillets:

- Click **Fillet**  or select **Insert / Features / Fillet / Round**.
- Enter **.015 in.** as the Radius .
- Select the edges of the 2 revolved features as shown.
- Click **OK** .



8. Saving Your Work:

- Select **File / Save As / Ball-Joint-Arm / Save**.

