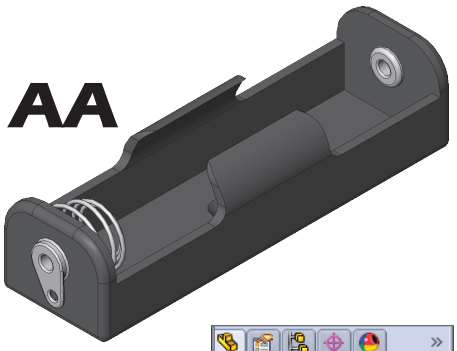
Chapter 9

Boat

Battery Holder AA



A. Front Extrude.

Step 1. Click File Menu > New, click **Part** and OK.

Step 2. Click **Front Plane** in the Feature Manager and click **Sketch** from the Content toolbar, **Fig. 1**.

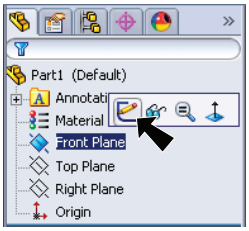


Fig. 1

Step 3. Click **Centerline** **(S)** in the **Line flyout**.

Step 4. Draw a centerline up from the Origin, **Fig. 2**.

Step 5. Click **Smart Dimension** **(S)** on the Sketch toolbar.

Step 6. Dimension **centerline .54** as shown in **Fig. 2**.

Step 7. Click **Zoom to Fit** **(F)** on the View toolbar.

Step 8. **Right click drawing and click Select** from menu to unselect Centerline Tool.

Step 9. Click the centerline to select it, **Fig. 3**.

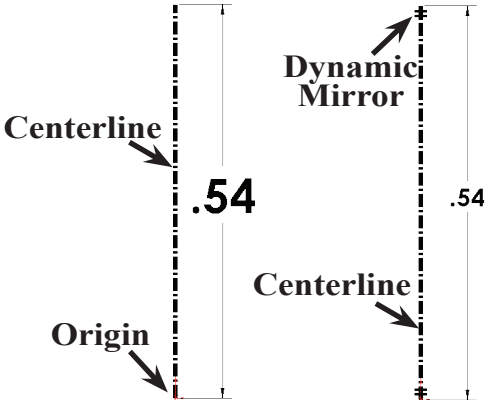


Fig. 2 Fig. 3

Step 10. Click **Dynamic Mirror** on the Sketch toolbar or Tools Menu > Sketch Tools > Dynamic Mirror, **Fig. 3**. Symmetry symbols appear at both ends of the centerline. Geometry drawn on one side of mirror line will mirror onto the other side.

Step 11. Click **Line** **(L)** on the Sketch toolbar.

Step 12. Draw **three lines** on one side of the dynamic mirror centerline, **Fig. 4**. Keep horizontal or vertical by using the inferencing line, the dotted line that appears when you draw the lines.

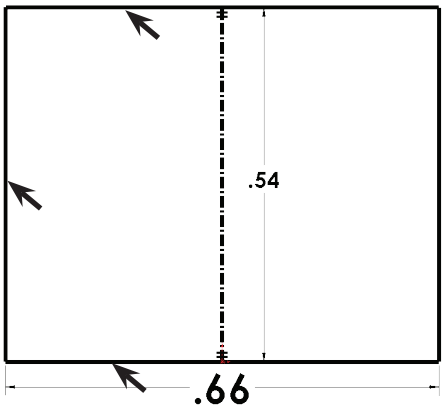


Fig. 4

Step 13. Click **Smart Dimension** **(S)** on the Sketch toolbar.

Step 14. Dimension **width .66**, **Fig. 4**.

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Step 15. Click **Dynamic Mirror**

on the Sketch toolbar to **turn off** Dynamic Mirror. Or Tools Menu > Sketch Tools > Dynamic Mirror.

Step 16. Click **Sketch Fillet** on the Sketch toolbar.

Step 17. In the Property Manager set:

Radius to **.2**, **Fig. 5**.

click **both top corners**, **Fig. 6**

click **Yes** to continue

click OK **twice** .

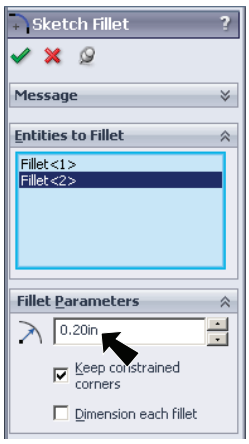


Fig. 5

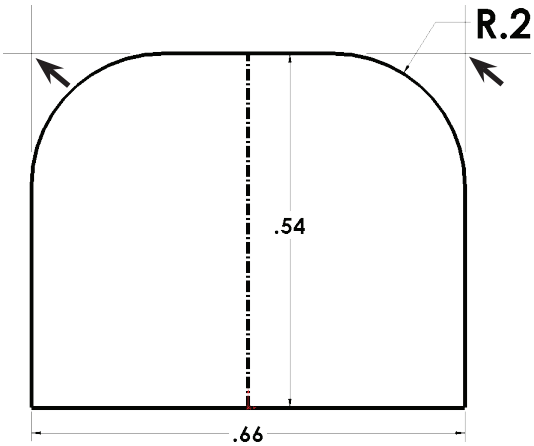


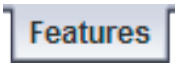
Fig. 6

B. Save as "BATTERY HOLDER".

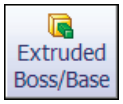
Step 1. Click File Menu > Save As.

Step 2. Key-in **BATTERY HOLDER** for the filename and press ENTER.

C. Extrude Front Cut.

Step 1. Click **Features** 

on the Command Manager toolbar.

Step 2. Click **Extruded Boss/Base** 

on the Features toolbar.

Step 3. In the Property Manager set:

under **Direction 1**

End Condition to **Mid Plane**

Depth to **2.22**

click OK , **Fig. 7** and **Fig. 8**.

Step 4. Click **Zoom to Fit** **(F)** on the View toolbar.

Step 5. Save. Use **Ctrl-S**.

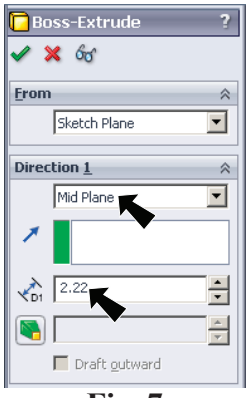


Fig. 7

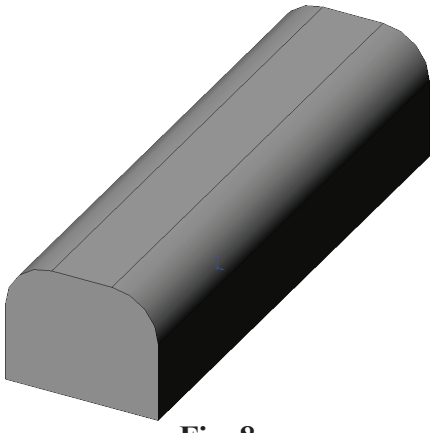


Fig. 8

D. Side Cut.

Step 1. Click **side face of holder** and click **Sketch** on the Content toolbar, **Fig. 9**.

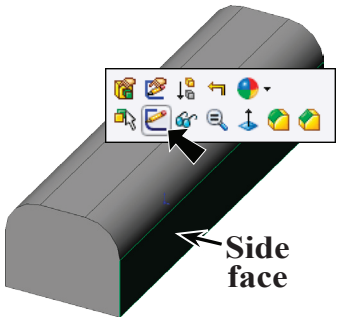


Fig. 9

Step 2. Click **Normal To** on the View toolbar. (**Ctrl-8**)

Step 3. Click **Line** (**L**) on the Sketch toolbar.

Step 4. Draw **four** lines as shown in, **Fig. 10**. Keep the top line horizontal by using the inferencing line, the dotted line that appears when you draw the lines.

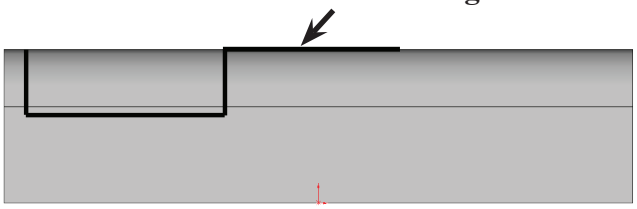


Fig. 10

Step 5. Click **Smart Dimension** (**S**) on the Sketch toolbar.

Step 6. Add the dimensions as shown in **Fig. 11**.

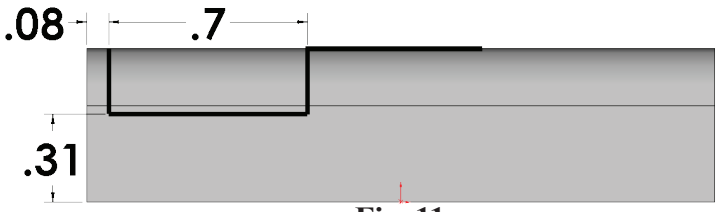


Fig. 11

Step 7. Click **Sketch Fillet** on the Sketch toolbar.

Step 8. In the Property Manager set:

Radius to **.07**, **Fig. 12**.

click the **two corners,** **Fig. 13**.

click OK **twice** .

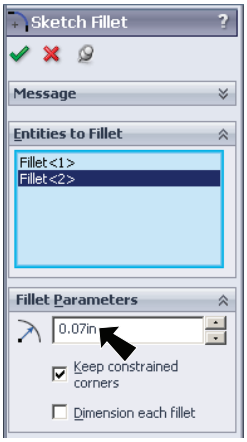


Fig. 12

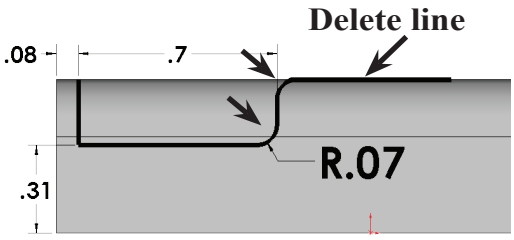


Fig. 13

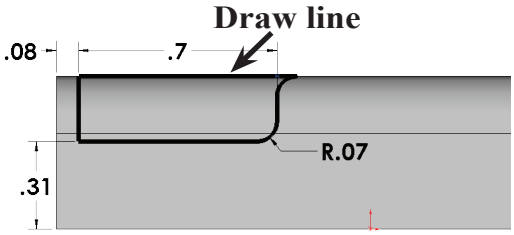


Fig. 14

Step 9. Delete the line to the right of top fillet, **Fig. 13**.

Step 10. Click **Line** (**L**) on the Sketch toolbar.

Step 11. Draw a line across top of sketch to close sketch, **Fig. 14**.

Step 12. Click **Centerline** in the **Line flyout** .

Step 13. Starting from the Origin , draw a centerline up through part, **Fig. 15**.

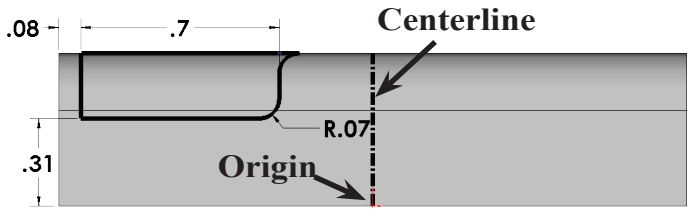


Fig. 15

Step 14. **Right click drawing and click Select** from menu to unselect Centerline Tool.

Step 15. **Drag selection around sketch** to select all entities (lines, centerline and dimensions), **Fig. 16**. To drag selection, click above and to left of sketch and drag down and to right to drag around all.

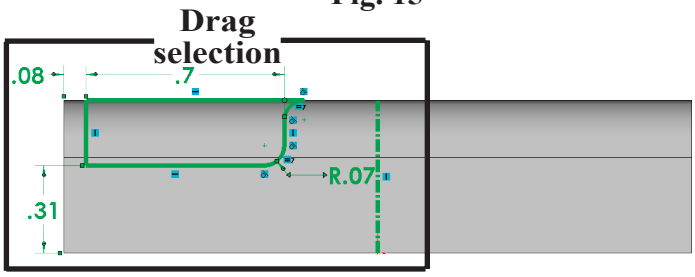


Fig. 16

Step 16. Click **Mirror Entities** on the Sketch toolbar, **Fig. 17**.

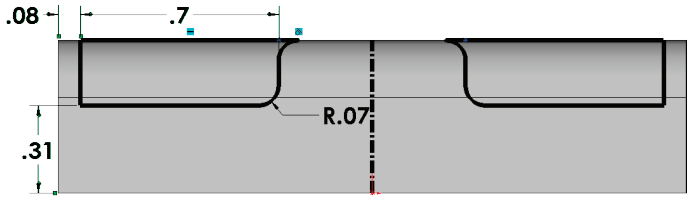
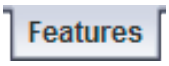
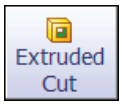


Fig. 17

Step 17. Click **Isometric** on the Standard Views toolbar. (**Ctrl-7**).

Step 18. Click **Features**  on the Command Manager toolbar.

Step 19. Click **Extruded Cut**  on the Features toolbar.

Step 20. In the Cut Extrude Property Manager set:

under **Direction 1**

End Condition to Through All

click OK .

Step 21. Save. Use **Ctrl-S**.

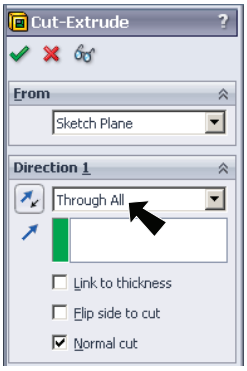


Fig. 18

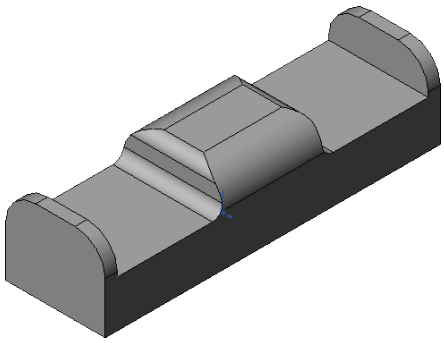


Fig. 19

E. Inside Cut.

Step 1. Click **inside back face of holder** and click **Sketch** on the Content toolbar, **Fig. 20**.

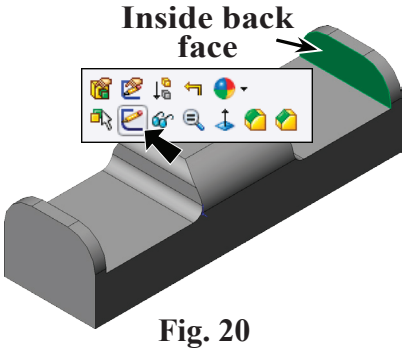


Fig. 20

Step 2. Click **Normal To** on the View toolbar. (**Ctrl-8**)

Step 3. Click **Centerline** in the **Line flyout**.

Step 4. Starting from the Origin, draw a centerline up through part, **Fig. 21**.

Step 5. **Right click drawing and click Select** from menu to unselect Centerline Tool.

Step 6. Click the centerline to select it, **Fig. 21**.

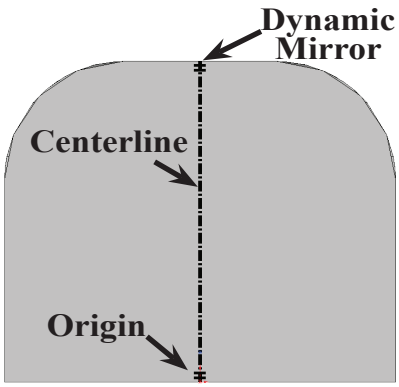


Fig. 21

Step 7. Click **Dynamic Mirror** on the Sketch toolbar or Tools Menu > Sketch Tools > Dynamic Mirror, **Fig. 21**. Symmetry symbols appear at both ends of the centerline. Geometry drawn on one side of mirror line will mirror onto the other side.

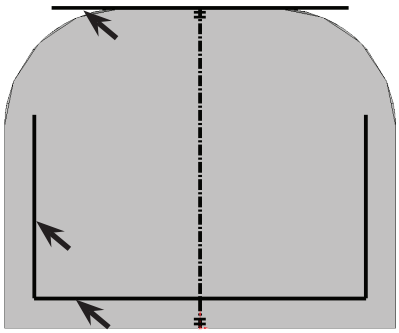


Fig. 22

Step 8. Click **Line** (**L**) on the Sketch toolbar.

Step 9. Draw **three lines** on one side of the dynamic mirror centerline, **Fig. 22**. Keep the top line horizontal by using the inferencing line, the dotted line that appears when you draw the lines.

Step 10. Click **Smart Dimension** (**S**) on the Sketch toolbar.

Step 11. Add the dimensions as shown in **Fig. 23**.

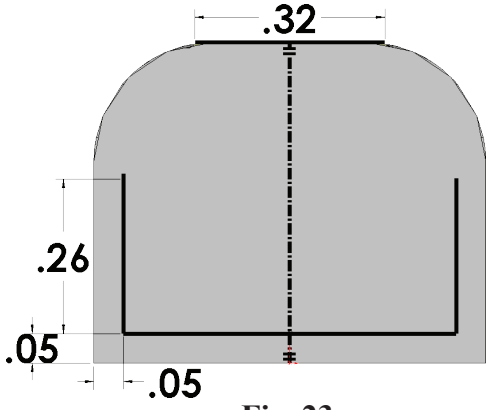


Fig. 23

Step 12. Click **Tangent Arc** in the **Arc flyout** on the Sketch toolbar.

Step 13. Draw an arc between Position 1 and Position 2 in **Fig. 24**. Start arc at Position 1 and swing up along path shown in **Fig. 24**.

Step 14. Click **Isometric** on the View toolbar. (**Ctrl-7**)

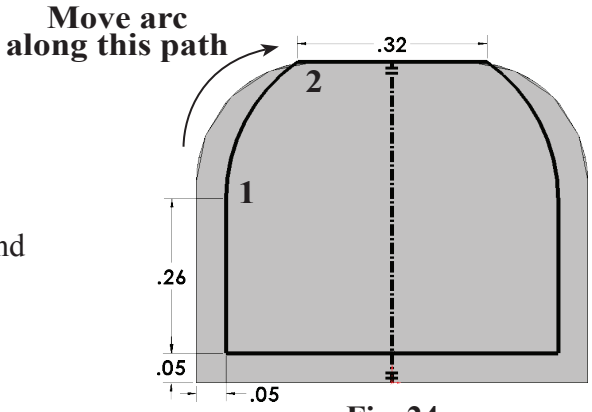
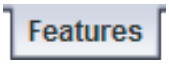
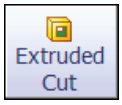


Fig. 24

Step 15. Rotate view to view **inside front face at the other end of holder**, hold down middle mouse button (wheel) and drag to rotate view, **Fig. 25**.

Step 16. Click **Features**  on the Command Manager toolbar.

Step 17. Click **Extruded Cut**  on the Features toolbar.

Step 18. In the Property Manager set:
under **Direction 1**
End Condition to
Up To Surface

to set **Face/Surface**, **click inside front face of holder**, **Fig. 25**

click OK , **Fig. 27**.

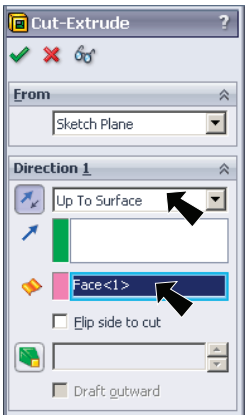


Fig. 26

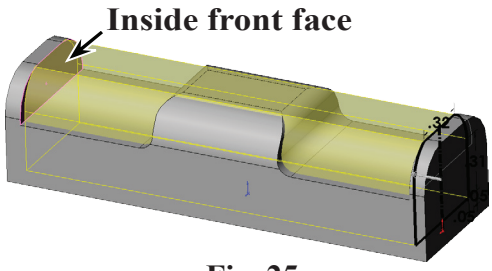


Fig. 25

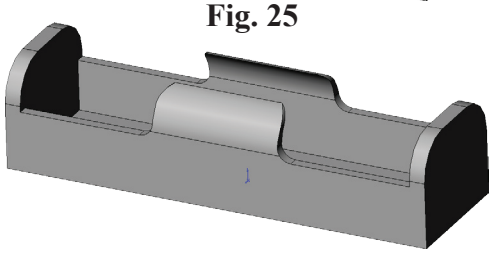


Fig. 27

Step 19. Save. Use **Ctrl-S**.

F. Screw Hole.

Step 1. Click **Top** on the Standard Views toolbar. (**Ctrl-5**)

Step 2. Click **Hole Wizard** Hole Wizard on the Features toolbar.

Step 3. In the Property Manager on the Type tab set:
under Hole Specification:
Click **Countersink**, **Fig. 28**
under Standard:
select **Ansi Inch**
under Hole Specification:
Size: #2

click **Positions tab** Positions at top of Property Manager.

click **bottom face of holder** to place Point, **Fig. 29**

click **Point** on the Sketch toolbar to **turn off** Point tool

grab the center of Point in the drawing and drag to the Origin, **Fig. 30**

click OK in the Point Property Manager

click OK in the Hole Wizard Property Manager, **Fig. 31**

Step 4. Save. Use **Ctrl-S**.

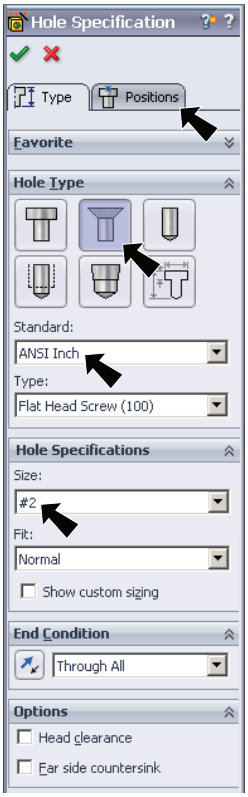


Fig. 28

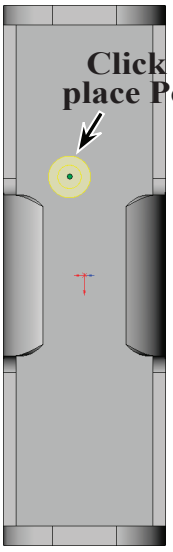


Fig. 29

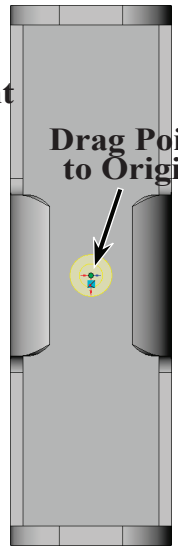


Fig. 30

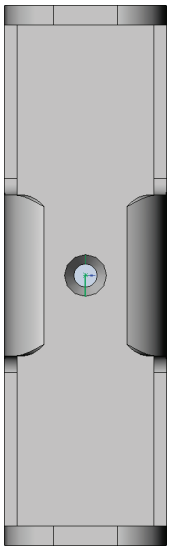


Fig. 31

G. Lug Holes.

Step 1. Click **Isometric** on the View toolbar. (**Ctrl-7**)

Step 2. Click **outside front face of holder** and click **Sketch** on the Content toolbar, **Fig. 32**.

Step 3. Click **Normal To** on the View toolbar. (**Ctrl-8**)

Step 4. Click **Centerline** in the **Line flyout**.

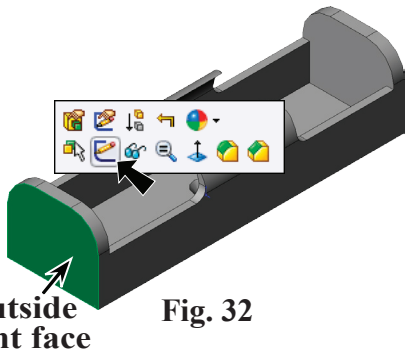


Fig. 32

Step 5. Starting from the Origin , draw a centerline up through part, **Fig. 33**.

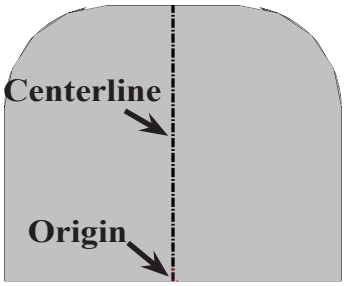


Fig. 33

Step 6. Click **Circle** (**S**) on the Sketch toolbar.

Step 7. Draw a circle on the centerline, **Fig. 34**.

Step 8. Click **Smart Dimension** (**S**) on the Sketch toolbar.

Step 9. Add dimension as shown in **Fig. 34**.

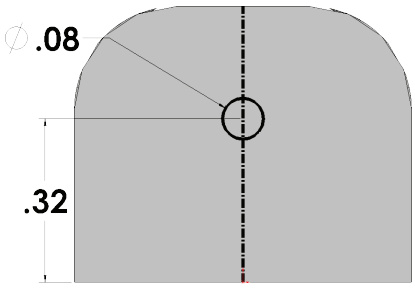


Fig. 34

Step 10. Click **Features** on the Command Manager toolbar.

Step 11. Click **Extruded Cut** on the Features toolbar.

Step 12. Click **Isometric** on the View toolbar. (**Ctrl-7**)

Step 13. In the Property Manager set:
under **Direction 1**
End Condition to **Through All**
click OK , **Fig. 35**
and **Fig. 36**.

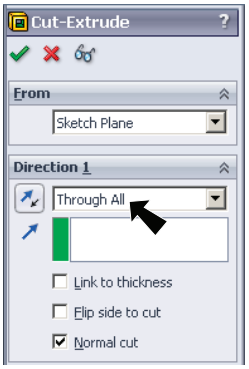


Fig. 35

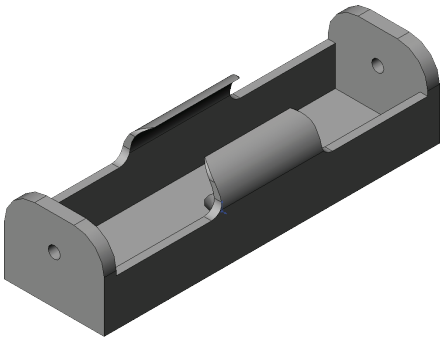


Fig. 36

Step 14. Save. Use **Ctrl-S**.

H. Sketch for Helix Spring.

Step 1. Rotate view to **view inside front face of holder (other end)**, hold down middle mouse button (wheel) and drag to **rotate view**, **Fig. 37**.

Step 2. Click the **inside face of front of holder** and click **Sketch** on the Content menu, **Fig. 37**.

Step 3. Click **Zoom to Selection** (**Q**) on the View toolbar to zoom to inside face.

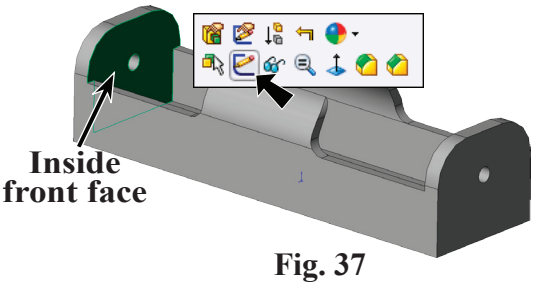


Fig. 37

Step 4. Press **Escape key** to unselect the face.

Step 5. Click **Offset Entities**

on the Sketch toolbar.

Step 6. In the Offset Entities Property Manager set:

Distance to .045

click edge of lug hole, **Fig. 39**

click OK .

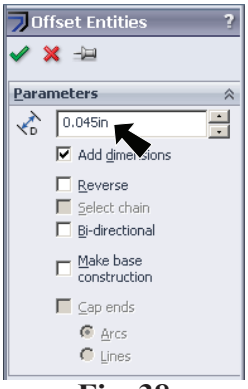


Fig. 38

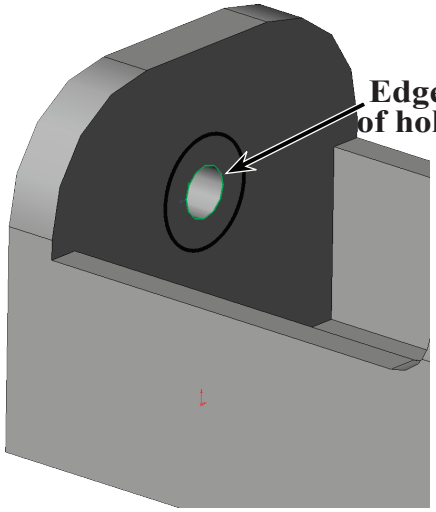


Fig. 39

I. Helix for Sweep Path.

Step 1. Click Insert Menu > Curve > Helix/Spiral.

Step 2. In the Helix/Spiral Property Manager set:

under Defined By: **Fig. 40**

select **Pitch and Revolution**

under Parameters

select **Variable Pitch**

under Region Parameters set:

use **Tab key** to move to next field

	P	Rev	Dia
1	.05	0	.17
2	.04	1	.42
3	.12	2	.38
4	.12	3	.35
5	.03	4	.3
6	.01	5	.25

use Tab key to move to next field

uncheck **Reverse direction**

Start angle 180

check **Clockwise**

click OK , **Fig. 41**.

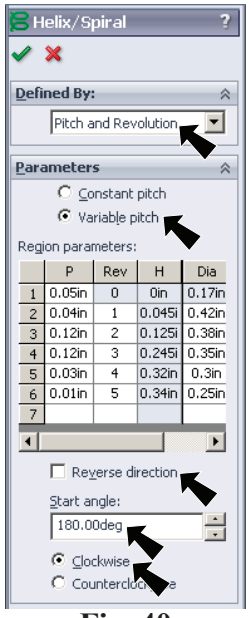


Fig. 40

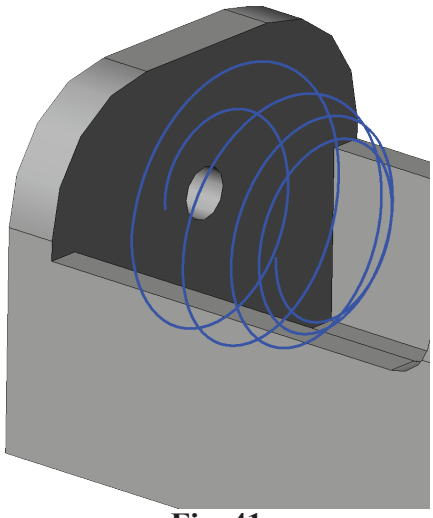


Fig. 41

J. 3D Sketch Helix Profile.

Step 1. Click **Sketch**

on the Command Manager toolbar.

Step 2. Click **3D Sketch**

in the **Sketch flyout**

on the Sketch toolbar. Be sure to click the **flyout arrow** to select 3D Sketch.

Step 3. Click **Circle** on the Sketch toolbar.

Step 4. Press **Tab key** on keyboard until cursor changes to **ZX** plane (**ZX** is Top Plane)

Step 5. Draw circle starting at the endpoint of helix, **Fig. 42**.

Step 6. **Right click drawing and click Select** from menu to unselect Circle Tool.

Step 7. Click circle to select it.

Step 8. In the Circle Property Manager set:

Radius .01 , **Fig. 43**

click OK , **Fig. 44**

Step 9. Exit the **3D Sketch.** To Exit, click

in the **Sketch flyout**

on the Sketch toolbar. Click the **flyout arrow** then 3D Sketch.

Step 10. Save. Use **Ctrl-S**.

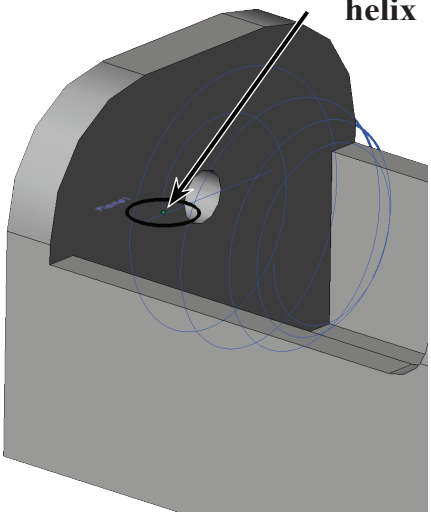


Fig. 42

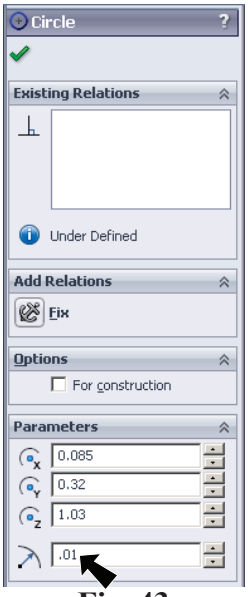


Fig. 43

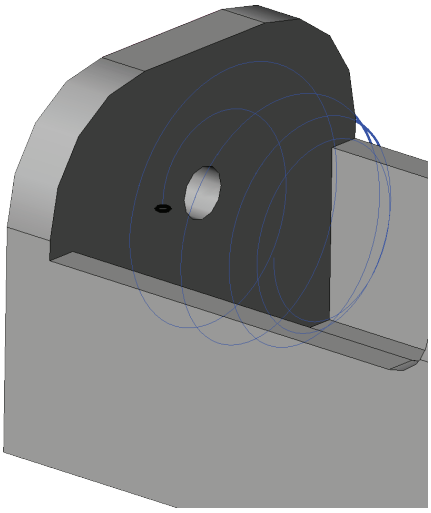
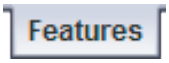
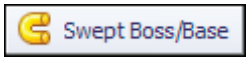


Fig. 44

K. Sweep Helix.

Step 1. Click **Features** 

on the Command Manager toolbar.

Step 2. Click **Swept Boss/Base** 

on the Features toolbar.

Step 3. In the Swept Boss/Base Property Manager set:

for **Profile** , **Fig. 45**
click **small circle** (3D sketch), **Fig. 46**

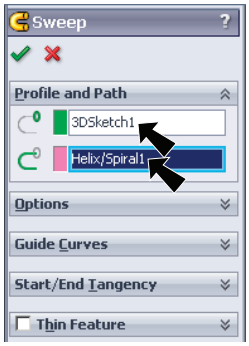


Fig. 45

for Path
click **helix**, **Fig. 46**

click OK .

Step 4. Save. Use **Ctrl-S**.

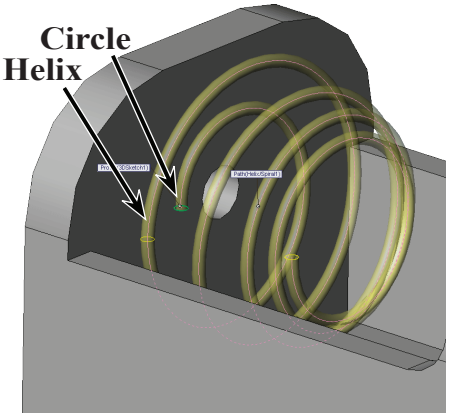


Fig. 46

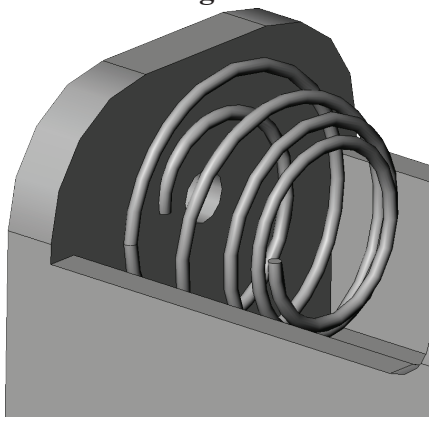


Fig. 47

L. Lug Rivet.

Step 1. Click **Isometric** on the Standard Views toolbar. (**Ctrl-7**)

Step 2. Click the **front face of holder** and click **Sketch** on the Content menu, **Fig. 48**.

Step 3. Click **Zoom to Selection** (**Q**) on the View toolbar to zoom to inside face.

Step 4. Press **Escape key** to unselect the face.

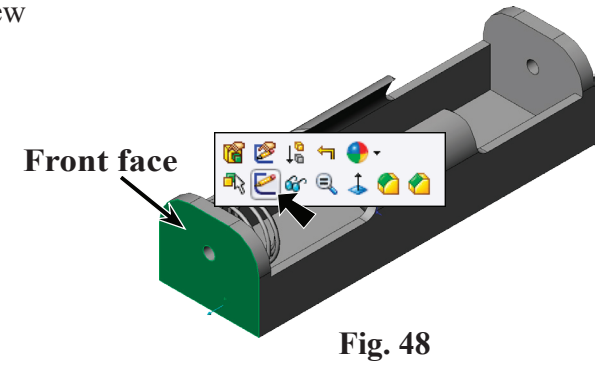


Fig. 48

Step 5. Click **Offset Entities**

on the Sketch toolbar.

Step 6. In the Offset Entities Property Manager set:

Distance to .06

click edge of lug hole, **Fig. 50**

click OK .

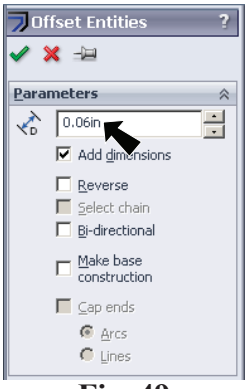


Fig. 49

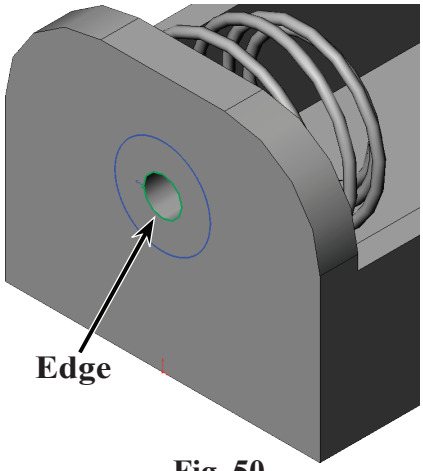


Fig. 50

Step 7. Click **Convert Entities**

on the Sketch toolbar.

Step 8. In the Convert Entities Property Manager:

click edge of lug hole, **Fig. 51**

click OK **twice** .

This converts the edge of hole to a circle.

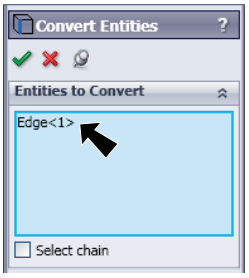


Fig. 52

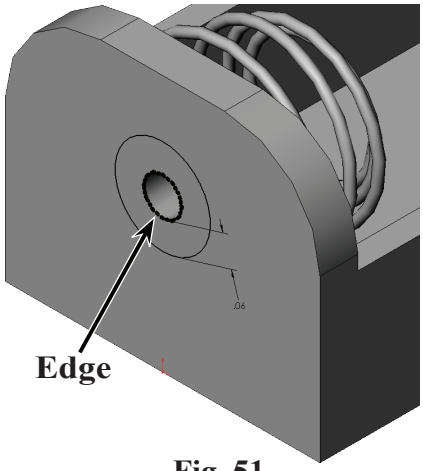
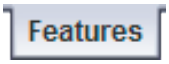
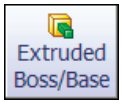


Fig. 51

Step 9. Click **Features** 

on the Command Manager toolbar.

Step 10. Click **Extruded Boss/Base** 

on the Features toolbar.

Step 11. In the Property Manager set:
under **Direction 1** **Fig. 53**

Depth to **.02**

check Direction 2 and set:

Depth to **.11**

and click OK , **Fig. 54** and **Fig. 55**.

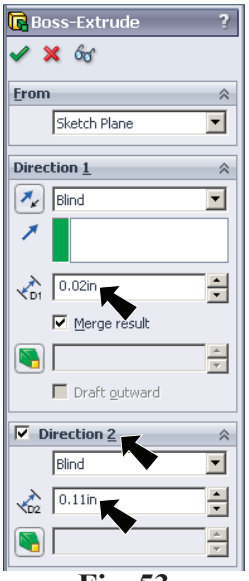


Fig. 53

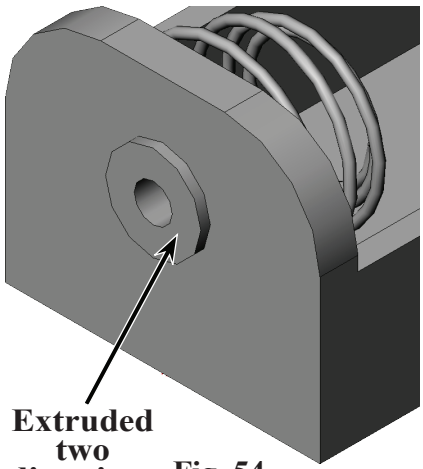


Fig. 54

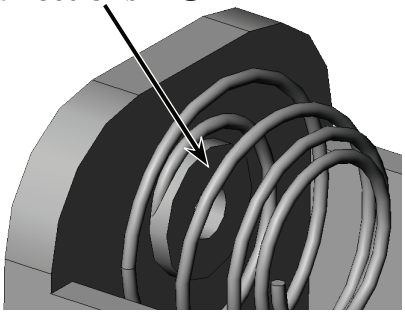


Fig. 55

Step 12. Save. Use **Ctrl-S**.

M. Soldering Lug.

Step 1. Click the **front face of lug rivet** and click **Sketch** on the Content menu, **Fig. 56**.

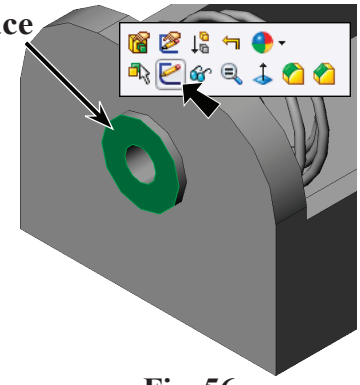


Fig. 56

Step 2. Click **Normal To** on the View toolbar. (**Ctrl-8**)

Step 3. With the face still selected, click **Convert Entities** on the Sketch toolbar.
This converts the outer edge of face to a circle, **Fig. 57**.

Step 4. In the Convert Entities Property Manager:
click edge of lug hole, **Fig. 59**
click OK **twice**
This converts the edge of hole to a circle.

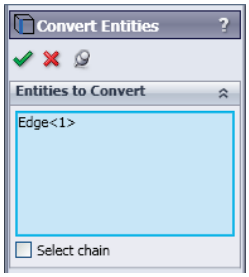


Fig. 58

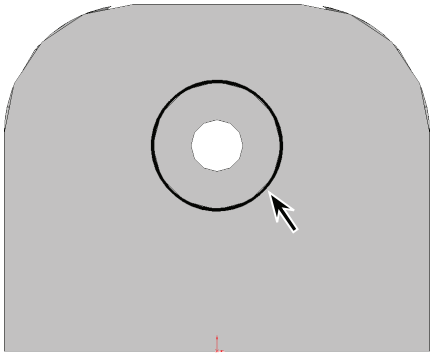


Fig. 57

Step 5. Click **Centerline** in the **Line flyout**.

Step 6. Draw **two centerlines**. Draw centerline up from the Origin and through the part. Draw a second centerline from the center hole to the bottom right of sketch, **Fig. 60**.

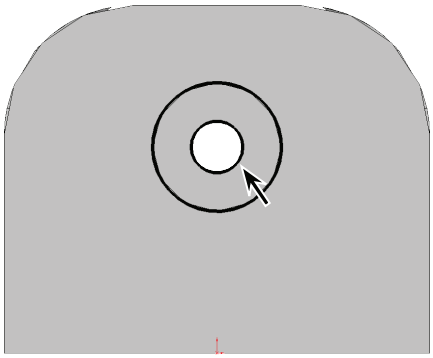


Fig. 59

Step 7. Click **Smart Dimension** (**S**) on the Sketch toolbar.

Step 8. Dimension the angle between centerlines **15 degrees** as shown in **Fig. 61**. **To Smart dimension angle**, click both centerlines then move the cursor between lines and click. Key-in **15** for the dimension and press ENTER.

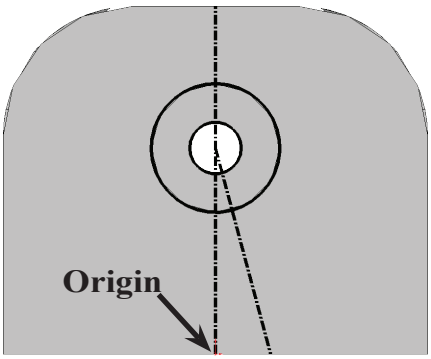


Fig. 60

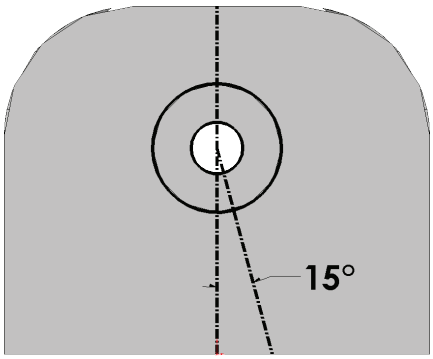


Fig. 61

Step 9. Click **Circle** **(S)** on the Sketch toolbar.

Step 10. Draw a circle on angled centerline, **Fig. 62**.

Step 11. Click **Smart Dimension**

(S) on the Sketch toolbar.

Step 12. Add the two dimensions as shown in **Fig. 62**.

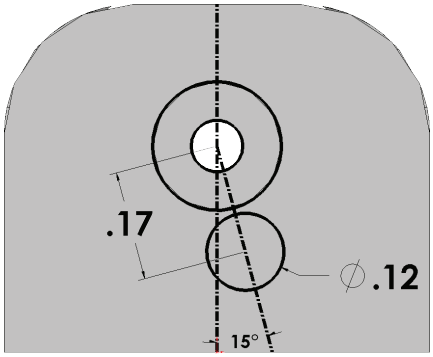


Fig. 62

Step 13. Click **Line** **(L)** on the Sketch toolbar.

Step 14. **Draw tangent line on left side** from big circle to bottom circle, **Fig. 63**. To draw tangent line, start line on big circle above the center at **Position 1**. Move cursor along lower circle at **Position 2** and when cursor change to **Tangent icon** , click circle.

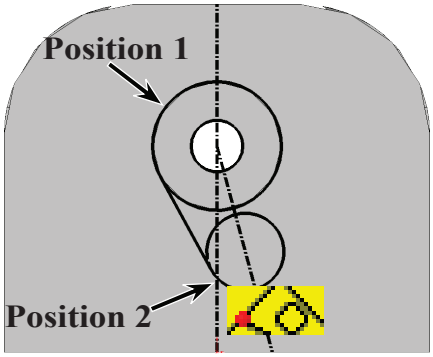


Fig. 63

Step 15. **Draw tangent line on other side** from big circle to bottom circle, **Fig. 64**. To draw tangent line, start line on big circle above the center at **Position 3**. Move cursor along lower circle at **Position 4** and when cursor change to **Tangent icon** , click circle. To create the line Tangent to circle the Tangent icon must appear before you click the small circle.

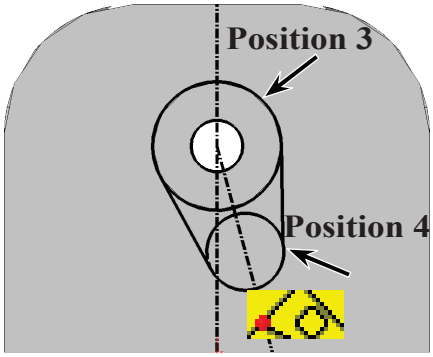


Fig. 64

Step 16. Click **Trim Entities**

on the Sketch toolbar.

Step 17. In the Property Manger select **Power Trim** .

Step 18. Trim away section of circles between tangent lines. To Trim, drag cursor across the **sections of the circles between the tangent lines, do not drag across centerlines**, **Fig. 65**. Results shown in **Fig. 66**. Click OK in the Property Manager when done.

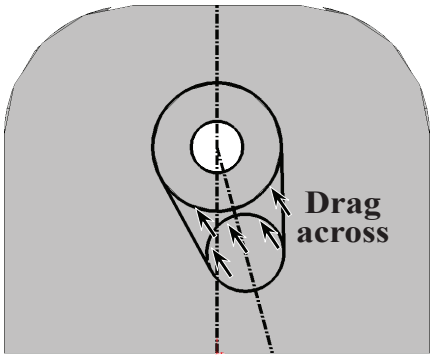


Fig. 65

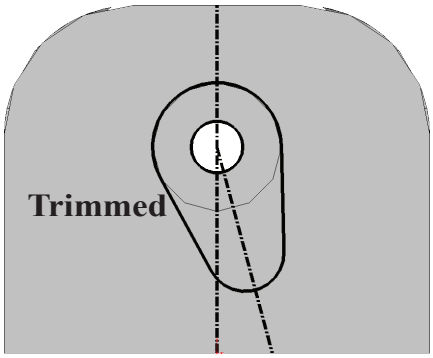


Fig. 66

Step 19. Click **Circle** (**S**) on the Sketch toolbar.

Step 20. Draw a circle at center of lower circle (now an arc), **Fig. 67**.

Step 21. Click **Smart Dimension** (**S**) on the Sketch toolbar.

Step 22. Dimension circle **diameter .05, Fig. 67**.

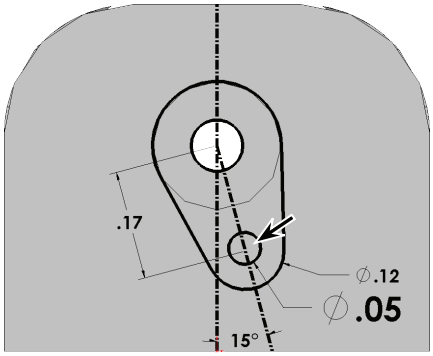


Fig. 67

N. Extrude Soldering Lug.

Step 1. Click **Isometric** on the Standard Views toolbar. (**Ctrl-7**).

Step 2. Click **Zoom to Selection** (**Q**) on the View toolbar to zoom to sketch.

Step 3. Click **Features** on the Command Manager toolbar.

Step 4. Click **Extruded Boss/Base** on the Features toolbar.

Step 5. In the Property Manager set:

Depth to **.02**

click OK , **Fig. 69**.

Step 6. Save. Use **Ctrl-S**.

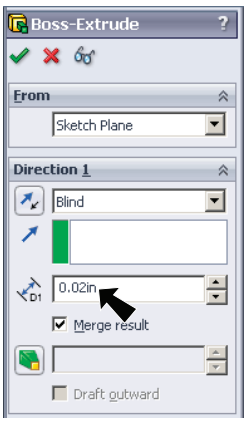


Fig. 68

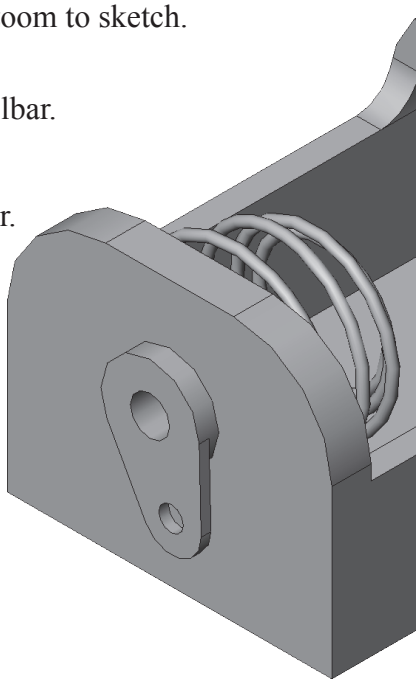


Fig. 69

O. Mirror Rivet.

Step 1. Click **Zoom to Fit** (**F**) on the View toolbar.

Step 2. **Ctrl click** the **Front Plane** and **Boss Extrude2** (**Rivet**) in the Feature Manager to select both, **Fig. 70**. To Ctrl click, hold down the Ctrl key, click **Front Plane** and **Boss Extrude2**.

Step 3. Click **Mirror** on the Features toolbar.

Step 4. In the Property Manager click OK, **Fig. 71** and **Fig. 72**.

Step 5. Save. Use **Ctrl-S**.

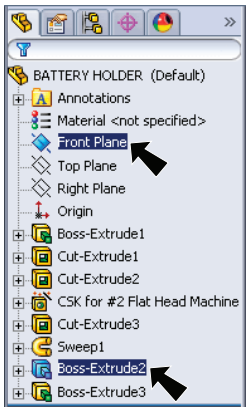


Fig. 70

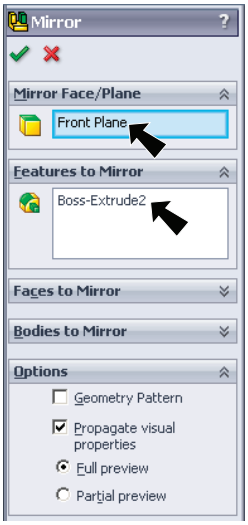


Fig. 71

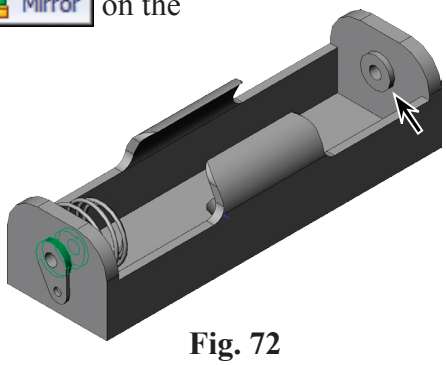


Fig. 72

P. Mirror Soldering Lug.

Step 1. Rotate view to view both **rivets**, hold down middle mouse button (wheel) and drag to **rotate view**, **Fig. 75**.

Step 2. **Ctrl click** the **Front Plane** and **Boss Extrude3** (**Soldering Lug**) in the Feature Manager to select both, **Fig. 73**. To Ctrl click, hold down the Ctrl key, click **Front Plane** and **Boss Extrude3**.

Step 3. Click **Mirror** on the Features toolbar.

Step 4. In the Property Manager click OK, **Fig. 74** and **Fig. 75**.

Step 5. Save. Use **Ctrl-S**.

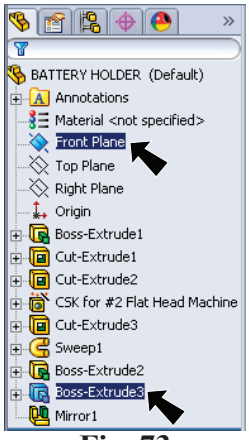


Fig. 73

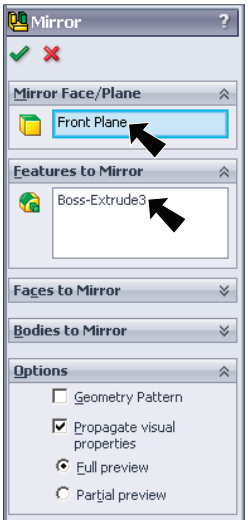


Fig. 74

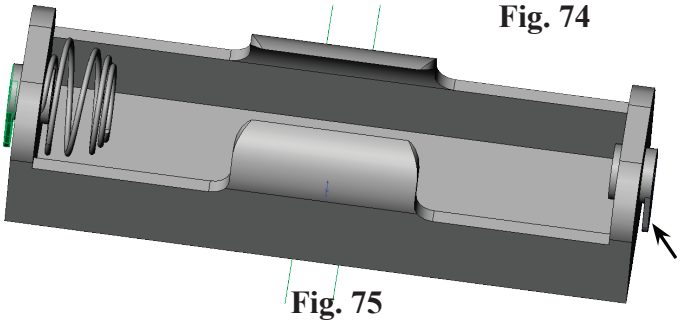


Fig. 75

Q. Fillet Soldering Lugs.

Step 1. **Ctrl click** both Soldering Lug features, **Boss Extrude3 and Mirror2** in the Feature Manager to select both, **Fig. 76**.

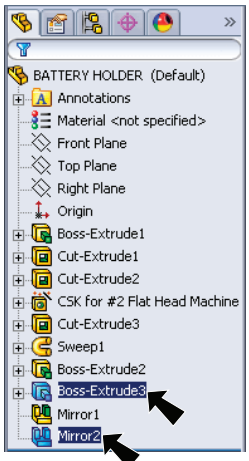


Fig. 76

Step 2. Click **Fillet** on the Features toolbar.

Step 3. In the Fillet Property Manager:

Select **FilletXpert**, **Fig. 77**

Set the **Radius** to **.009**

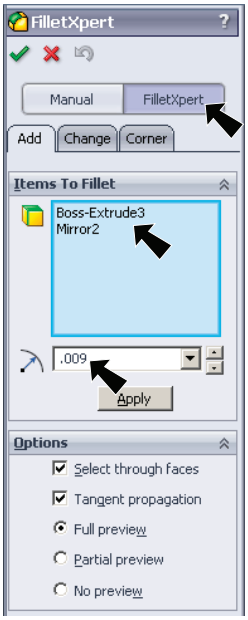


Fig. 77

Click Apply, **Fig. 78** and **Fig. 79**

Rotate view to view **rivet in inside of holder**, hold down middle mouse button (wheel) and drag to **rotate view**, **Fig. 81**.

Zoom in to **rivet in inside of holder**, **Fig. 81**. To **zoom**, hold down **Shift key** and drag with middle mouse button (wheel). To **pan**, hold down **Ctrl key** and drag with middle mouse button (wheel).

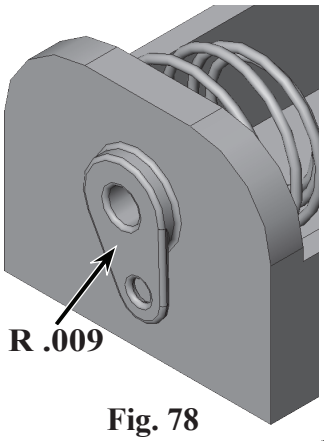


Fig. 78

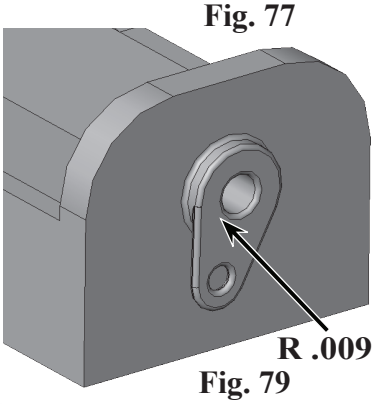


Fig. 79

Set **Radius** to **.02**
click **outer face of rivet on inside of holder**, **Fig. 81**.
Click Apply

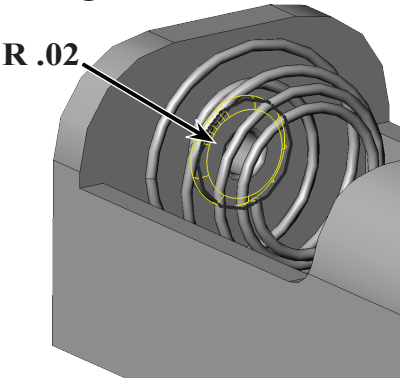


Fig. 81

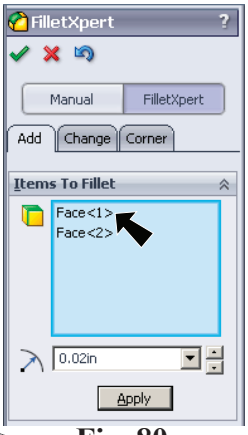


Fig. 80

Click **Isometric** on the Standard Views toolbar. (**Ctrl-7**).

Zoom in to **other rivet**, **Fig. 82**. To **zoom**, hold down **Shift key** and drag with middle mouse button (wheel).

click **outer face of rivet on inside of holder**, **Fig. 82**.

click OK .

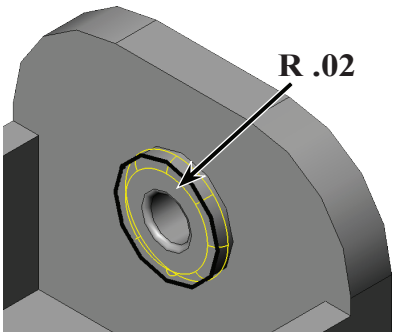


Fig. 82

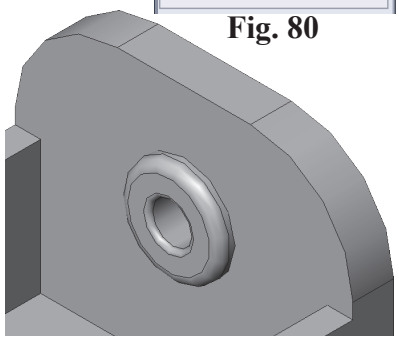


Fig. 83

R. Appearance.

Step 1. Click **Zoom to Fit** (**F**) on the View toolbar.

Step 2. Click holder to select the part, click **Appearance Callout** on the Content menu and click **BATTERY...**, **Fig. 84**.

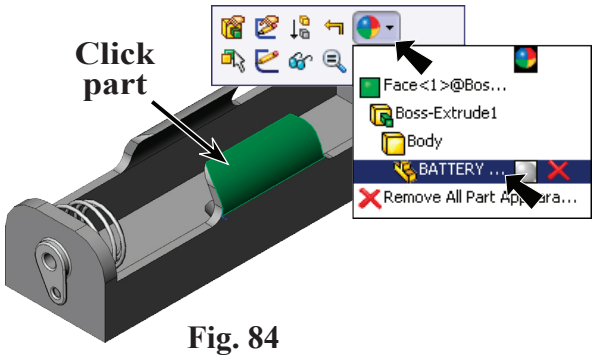


Fig. 84

Step 3. In the Appearances Task pane, expand **Plastic**, click **High Gloss** and in the lower pane select **dark gray high gloss plastic**, **Fig. 85**. Click OK in the Property Manager.

Step 4. **Shift click** Sweep and last Fillet in the Feature Manager to select list of features to apply steel appearance, **Fig. 86**. To Shift click, hold down the Shift key, click first feature, Sweep and click last feature, last Fillet.

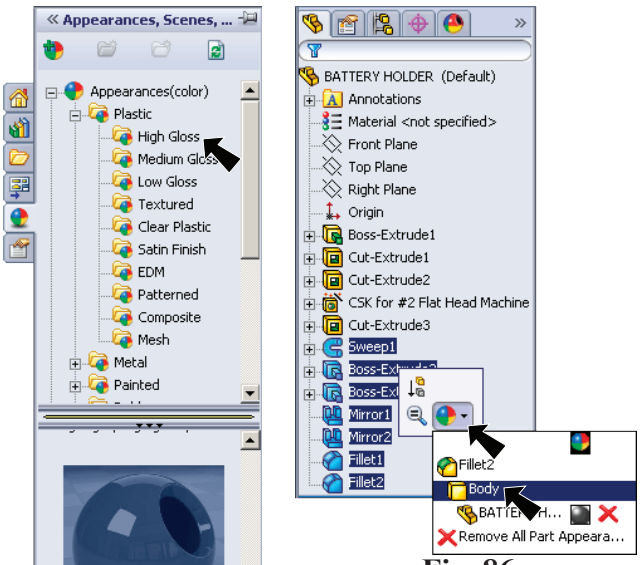


Fig. 85

Fig. 86

Step 5. Click **Appearance Callout** on the Content menu and click **Body**, **Fig. 86**.

Step 6. In the Appearances Task pane, expand **Metal**, click **Chrome** and in the lower pane select **chromium plate**, **Fig. 87**

click inside lug hole, Fig. 88

click OK in the Property Manager.

Step 7. Save. Use **Ctrl-S**.

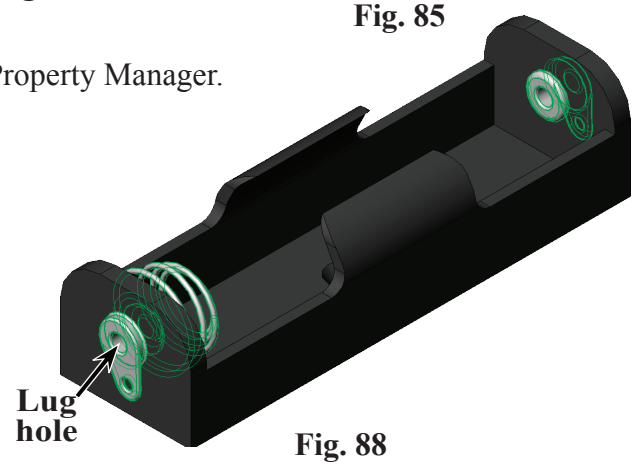


Fig. 88

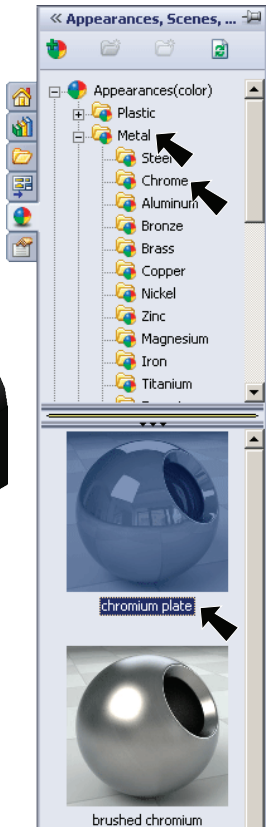


Fig. 87

S. Fillet Holder Edges.

Step 1. Click **Fillet**

on the Features toolbar.

Step 2. In the Fillet Property Manager:

Select **FilletXpert**, **Fig. 89**

Set the **Radius** to **.04**

Click outside edges of holder **Fig. 90**.

Rotate view to view **bottom**, hold down middle mouse button (wheel) and drag to **rotate view**, **Fig. 91**.

click **bottom edges**, **Fig. 91**.

click OK .

Step 3. Save. Use **Ctrl-S**.

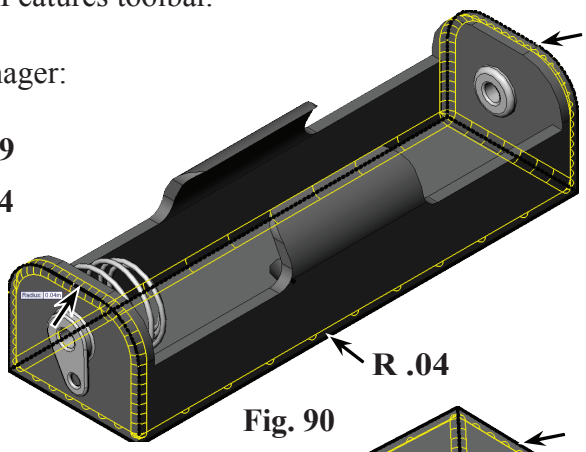


Fig. 90

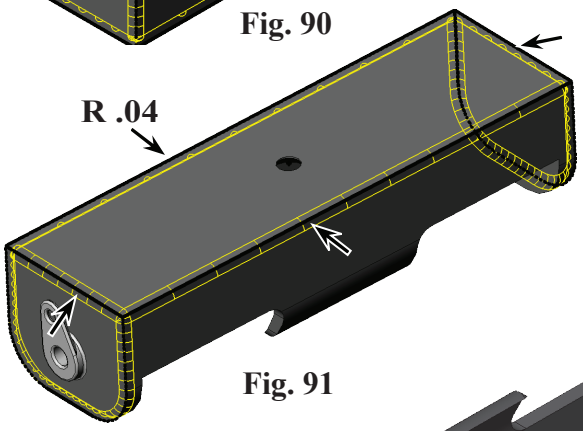


Fig. 91

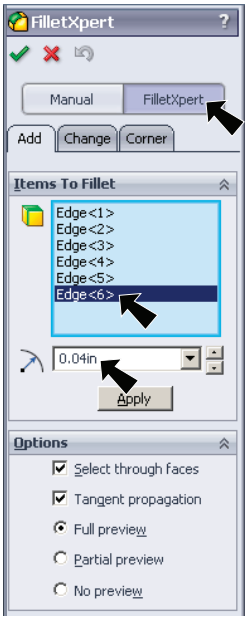
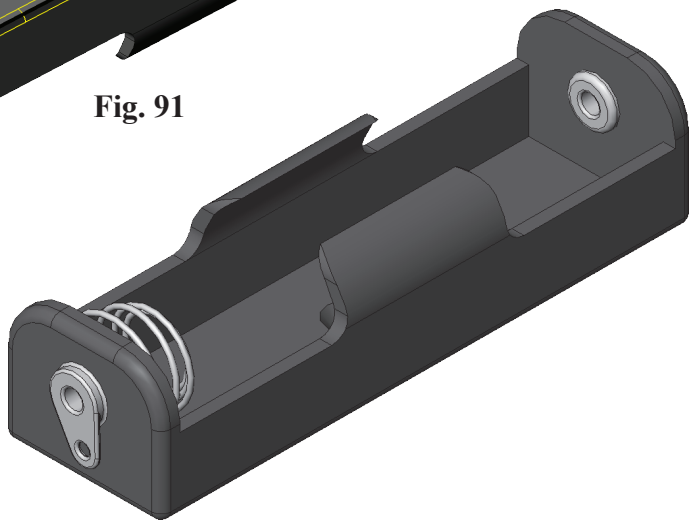



Fig. 89