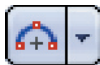


CO2 Shell Car Eye Screw

A. Sketch Centerpoint Arc.

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

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

Step 3. Click **Centerpoint Arc**  (S) in the **Arc flyout**  on the Sketch toolbar.

Step 4. Draw a slightly open arc starting from the Origin , **Fig. 2**.

To draw the arc, click the Origin to place the center of the arc. Start the first end point directly above the Origin, then swing the arc to the right around counterclockwise. Click to place the second end point leaving a small gap in the arc, **Fig. 2**.

Use the inferencing line, the dotted line that appears when you draw the arc.

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

Step 6. Dimension radius of the arc **2.5** as shown in **Fig. 2**. To Smart dimension click the arc then move the cursor out away from the arc and click. Key-in the dimension and press ENTER. Arrange the dimension as **Fig. 2**.

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

B. Save as "EYE SCREW".

Step 1. Click File Menu > Save As.

Step 2. Key-in **EYE SCREW** for the filename and press ENTER.

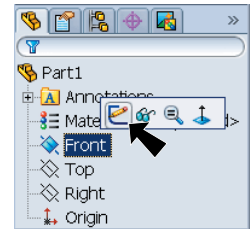
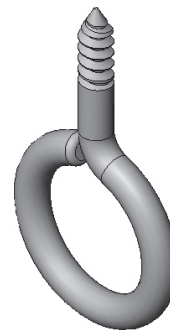


Fig. 1

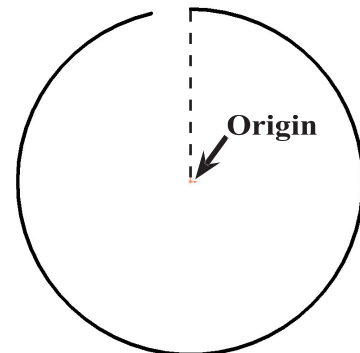


Fig. 3

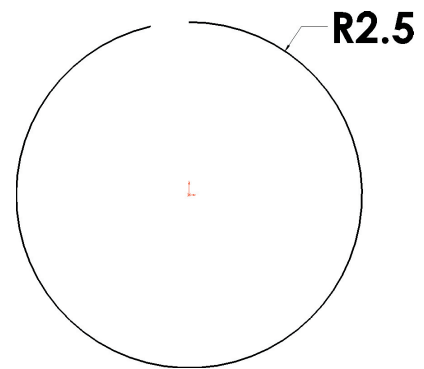


Fig. 3

C. Line.

Step 1. Click **Line**  (L) on the Sketch toolbar.

Step 2. Draw line as shown in **Fig. 4**. Draw the line up from the arc endpoint that is directly above Origin.

Step 3. Click **Smart Dimension** (S) on the Sketch toolbar.

Step 4. Dimension arc gap **.5** and the line **5** as shown in **Fig. 5**. To dimension the gap in arc, click both endpoints of arc, then move cursor out way and click. To Smart dimension the line, click the line, then move the cursor out away from the line and click.

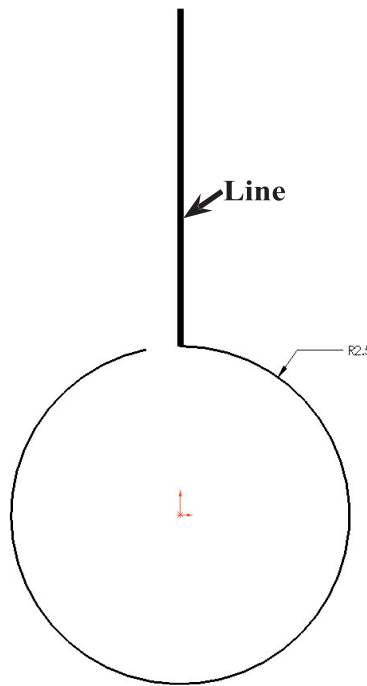


Fig. 4

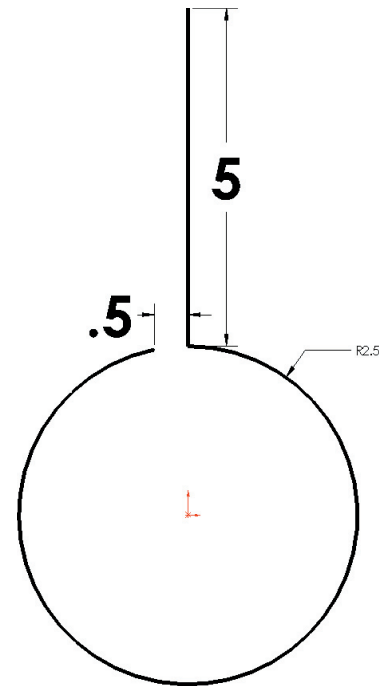


Fig. 5

Step 5. Click Zoom to Fit  (F) on the View toolbar.

D. Sketch Fillet.

Step 1. Click **Sketch Fillet**  on the Sketch toolbar.

Step 2. Set the **Radius**  to **1** in the Sketch Fillet Property Manager, **Fig. 6**.

Step 3. Click the corner shown in **Fig. 7** where the arc and line intersect.

Step 4. Click **Exit Sketch**  on the Sketch toolbar.

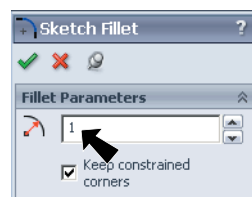


Fig. 6

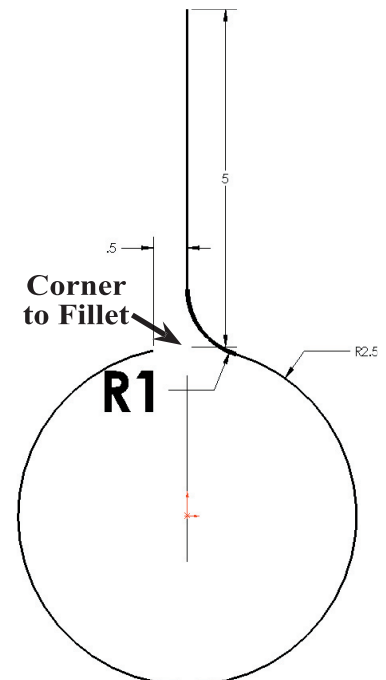


Fig. 7

E. 3D Sketch Profile.

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

Step 2. Click **Top** (plane) in the Feature Manager (left panel), **Fig. 8**.

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

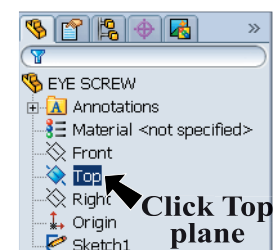


Fig. 8

Step 4. Click **3D Sketch**  **3D Sketch** in the **Sketch flyout** on the Sketch toolbar. Be sure to click the **flyout arrow**  to select 3D Sketch.

Step 5. Click **Circle**  on the Sketch toolbar.

Step 6. Draw a circle starting at the top end of the line in sketch, **Fig. 9**.

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

Step 8. Dimension the circle **1** diameter as shown in **Fig. 9**. To Smart dimension click the circle then move the cursor out away from the circle and click. Key-in the dimension and press ENTER.

Step 9. Exit the **3D Sketch**. To Exit, click  in the **Sketch flyout**  on the Sketch toolbar. Click the **flyout arrow** then 3D Sketch.

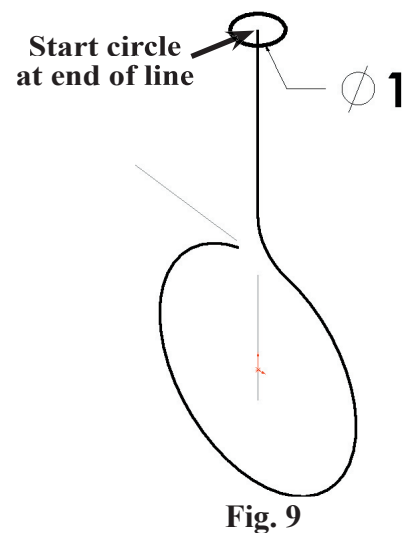


Fig. 9



Fig. 10

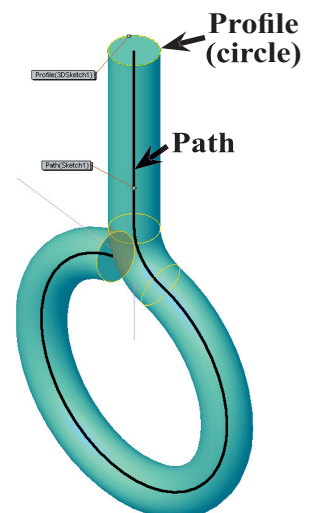


Fig. 11

F. Sweep.

Step 1. Click Insert Menu > Boss/Base > Sweep.

Step 2. Click **Profile**  field in the Sweep Property Manager, **Fig. 10** and click the **circle** in the 3D Sketch, **Fig. 11**.

Step 3. Click **Path**  field in the Sweep Property Manager, **Fig. 10** and click any geometry of **Sketch1** (lines and arcs), **Fig. 11**.

Step 4. Click OK .

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

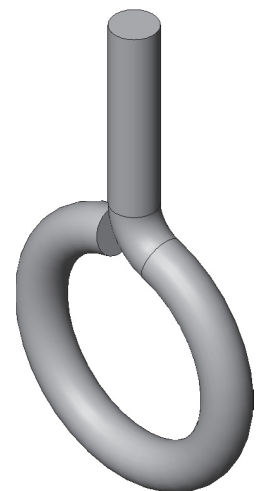


Fig. 12

G. Fillet Edge.

- Step 1. Click **Fillet**  on the Features toolbar.
- Step 2. Set the **Radius**  to .2 in the Fillet Property Manager, **Fig. 13**.
- Step 3. Click the edges at both ends of the axle and click OK  in the Property Manager, **Fig. 14**.
- Step 4. Save. Use **Ctrl-S**.

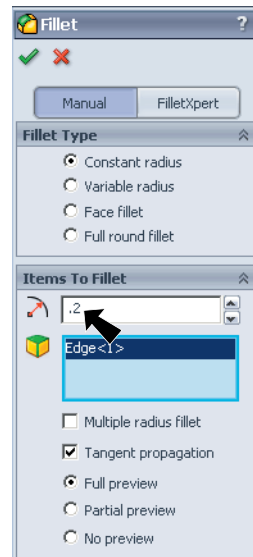


Fig. 13

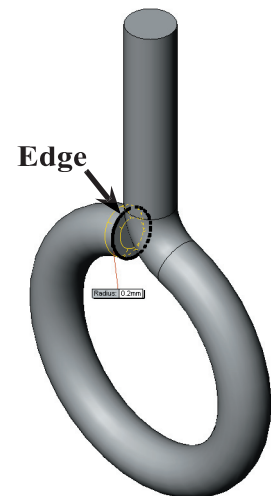


Fig. 14

H. Threads Sketch for Sweep Cut.

- Step 1. Click **Isometric**  on the Standard Views toolbar. (**Ctrl-7**)
- Step 2. Click the **top face** and click **Sketch**  on the Content menu, **Fig. 15**.
- Step 3. Click **Offset Entities**  on the Sketch toolbar.
- Step 4. In the Offset Entities Property Manager set:
Distance to .23

The yellow offset circle should be on the inside of the face, **Fig. 17**. If it is not, click Reverse. Click OK .

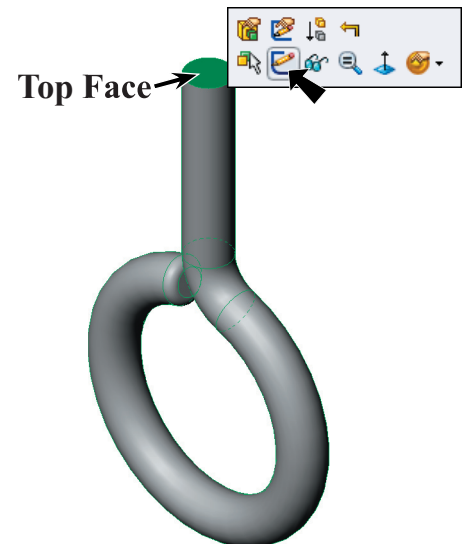


Fig. 15

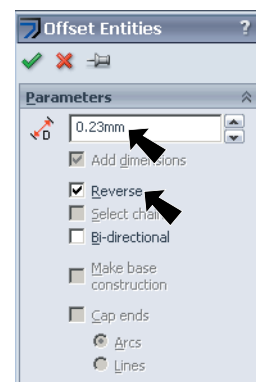


Fig. 16

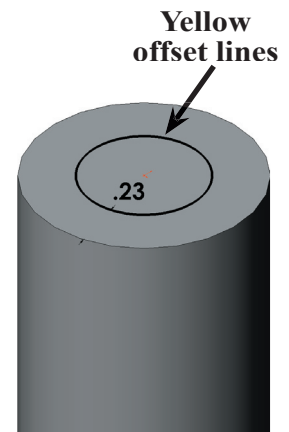


Fig. 17

I. Helix for Sweep Cut.

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

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

select **Pitch and Revolution**

under Parameters

select **Variable Pitch**

under Region Parameters

set **Pitch** to .37 all rows

click in Row 2 of Rev column and
set to 6

click in Row 3 of Rev column and
set to 7

set Row 3 Dia to 1 and P to .37

check **Reverse direction**

Start angle to 270

check **Counterclockwise**

click OK , Fig. 18.

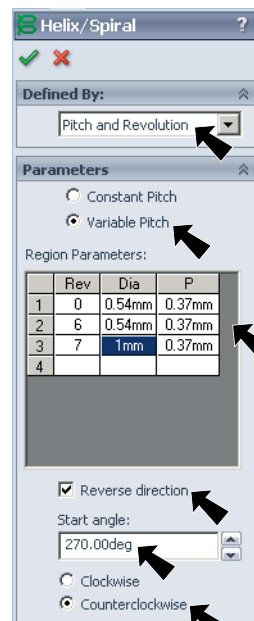


Fig. 18

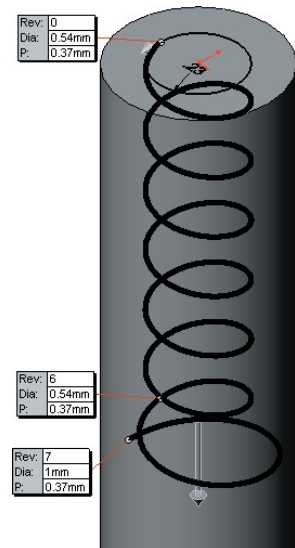


Fig. 19

J. Sketch Thread Profile.

Step 1. Click **Front** (plane) in the Feature Manager and click **Sketch**  from the Content toolbar, Fig. 20.

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

Step 3. Use the **Zoom to Area**  in the View toolbar to drag a zoom window around the **top end of the eye screw**, Fig. 21.

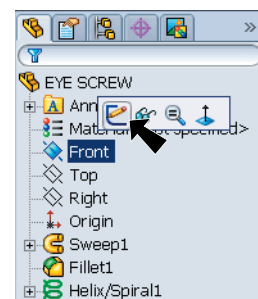


Fig. 20

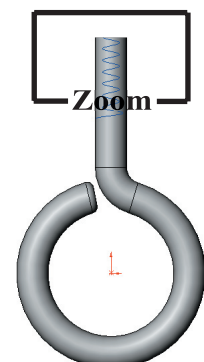


Fig. 21

Step 4. Click **Line**  (L) on the Sketch toolbar.

Step 5. Draw four lines as shown in **Fig. 22**. Keep the sketch out in space and not attached to the eye screw.

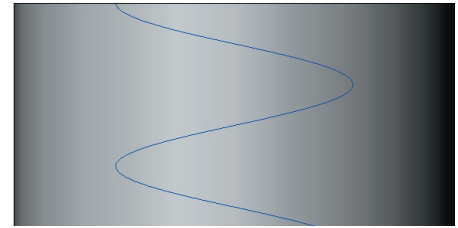
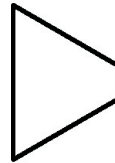


Fig. 22

K. Add Equal Relations.

Step 1. Press the Escape key to unselect **Line** tool.

Step 2. Select the **top and bottom lines**. Use **Ctrl click**, to Ctrl click, hold down the Shift key and click both lines, **Fig. 23**.

Step 3. In the Property Manager under Add Relations, click **Equal** , **Fig. 24**. Click OK .

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

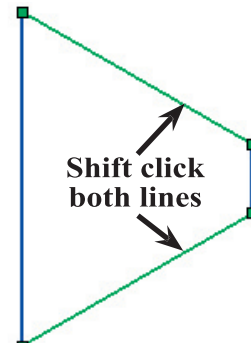


Fig. 23

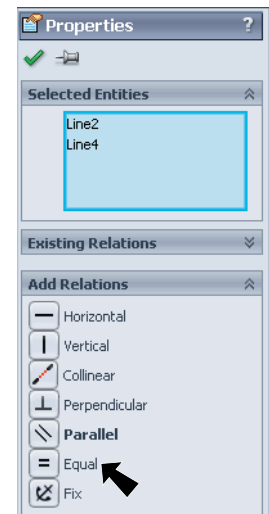


Fig. 24

L. Smart Dimension.

Step 1. Click **Smart Dimension**  (S) on the Sketch toolbar.

Step 2. Dimension the left line **.35** and the top **.24** as shown in **Fig. 25**.

Step 3. Dimension the angle **60° degrees** between the left line and bottom line as shown in **Fig. 26**. To dimension the angle, click both lines then move the cursor inside and click. Key-in **60** for the dimension and press ENTER.

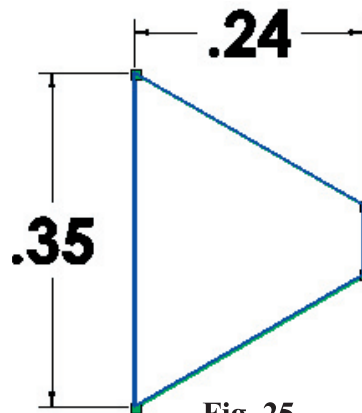


Fig. 25

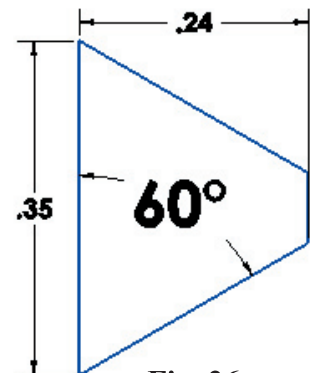


Fig. 26

M. Add Pierce Relation.

- Step 1. Press the Escape key to **unselect Smart Dimension**.
- Step 2. Select the **bottom right endpoint of the sketch and helix**. Use **Ctrl click**. To Ctrl click, click the **bottom right endpoint of the sketch**, then hold down the Ctrl key and click helix (not the endpoint of the helix), **Fig. 27**.
- Step 3. In the Property Manager under Add Relations, click **Pierce** , **Fig. 28**. Click OK .
- Step 4. Click **Exit Sketch**  on the Features toolbar.
- Step 5. Save. Use **Ctrl-S**.

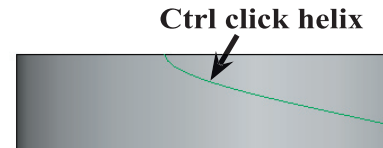
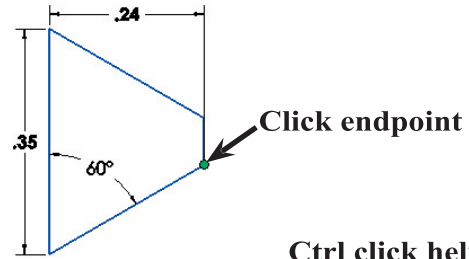


Fig. 27

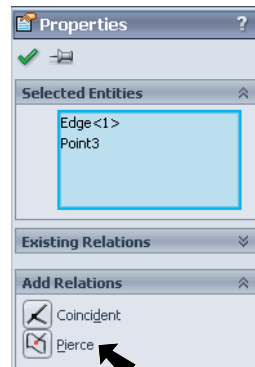


Fig. 28

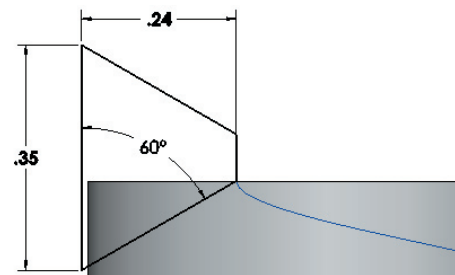


Fig. 29

N. Sweep Helix.

- Step 1. Click Insert Menu > Cut > Sweep.
- Step 2. Click **Profile**  field in the Sweep Property Manager, **Fig. 30** and click the **sketch** in the **Fig. 31**.
- Step 3. Click **Path**  field in the Sweep Property Manager, **Fig. 30** and click helix, **Fig. 31**.
- Step 4. Click OK .
- Step 5. Save. Use **Ctrl-S**.

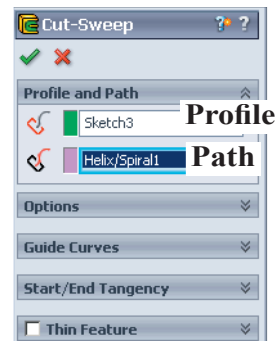


Fig. 30

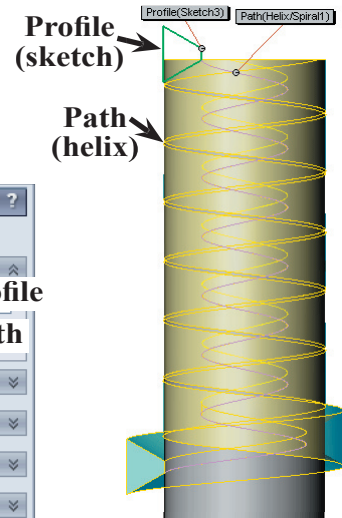


Fig. 31

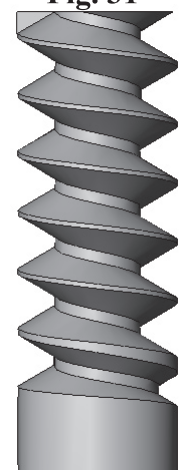


Fig. 32

O. Chamfer.

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

Step 2. Roll the rollback bar to above the Cut-Sweep. To rollback Cut-Sweep, click Cut-Sweep in the Feature Manager and click **Rollback**  from the Content toolbar, **Fig. 33**.

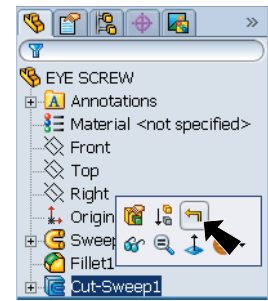


Fig. 33

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

Step 4. In the Chamfer Property Manager set:

Distance distance

Depth 1  **D1** to **.8**

Depth 2  **D2** to **.45**, **Fig. 34**.

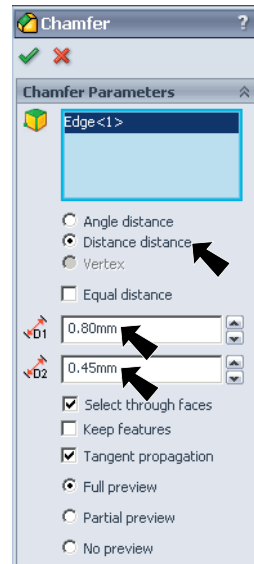


Fig. 34

Step 5. Click the **top corner edge** of the eye screw, **Fig. 35** and click OK  in the Property Manager, **Fig. 36**.

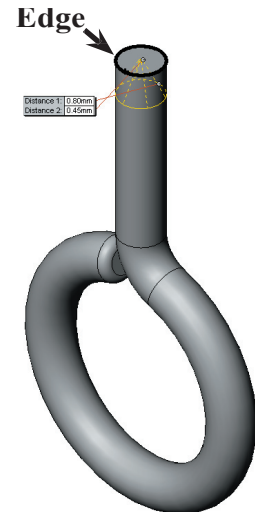


Fig. 35

Step 6. Roll forward to display the Cut-Sweep. To roll forward, **right click** Cut-Sweep in the Feature Manager and click **Roll Forward** in the menu, **Fig. 37** and **Fig. 38**.

P. Appearance.

Step 1. Click Edit Menu > Appearance > Material.

Step 2. Expand Steel, select **Steel AISI 304** and click OK , **Fig. 39**.

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

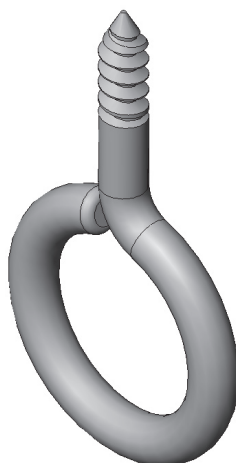


Fig. 38

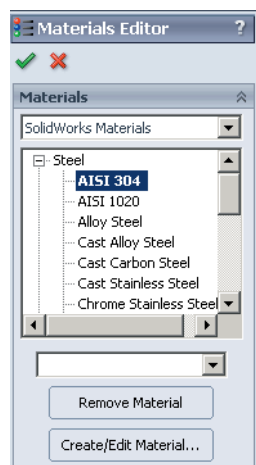


Fig. 39

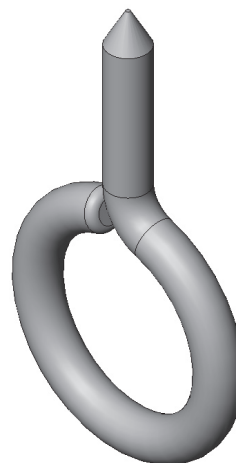


Fig. 36

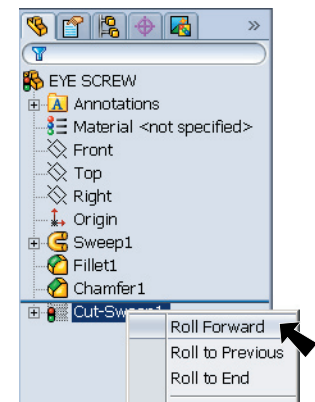


Fig. 37

Q. Open Assembly File.

Step 1. Click File Menu > Open. Select your ASSEMBLY file and click OK.

Step 2. Click **BODY** in the Feature Manager and click Open Part from the Content toolbar, **Fig. 40**.

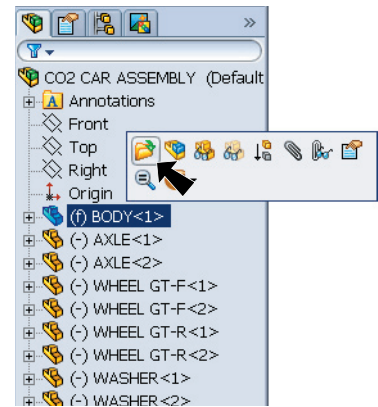


Fig. 40

R. Holes.

Step 1. Click **Bottom**  on the Standard Views toolbar. (Ctrl-6)

Step 2. Use the **Zoom to Area**  in the View toolbar to drag a zoom window around the **front wheel cutout**, **Fig. 41**.

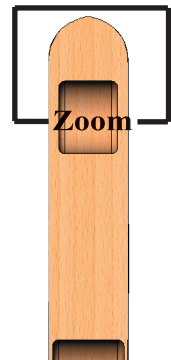


Fig. 41

Step 3. Click **Hole Wizard**  on the Features toolbar.

Step 4. In the Property Manager, on the Type tab and set:
under Hole Specification:

Click **Hole**  **Fig. 42**

under Standard:

select **Anis Metric**

under Size:

select **.9**

under End Condition:

set **Blind Depth Hole**  to 1.5

under Custom Sizing:

click the Position tab at the top of the Property Manager.

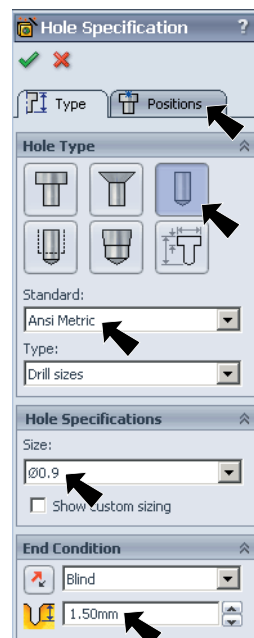


Fig. 42

Step 5. Click to place the hole approximately in the center of the body just above the wheel cutout, **Fig. 43**.

Step 6. Click **Point**  on the Sketch toolbar to **turn off** Point tool.

Step 7. Grab the point in the drawing and drag to activate the inferencing line, the dotted line that appears. Move the point to the inferencing line to align point with the center of the body, **Fig. 43**. Click OK

 in the Point Property Manager.



Fig. 43

Step 8. Scroll down to the rear wheel cutout, **Fig. 44**. To scroll hold down **Ctrl** and press **up arrow key**.

Step 9. Click **Point**  on the Sketch toolbar to **turn on** Point tool.

Step 10. Add a point for the rear eye screw. Click to place the hole approximately in the center of the body just below the wheel cutout, **Fig. 44**.

Step 11. Click **Point**  on the Sketch toolbar to **turn off** Point tool.

Step 12. Grab the point in the drawing and drag to activate the inferencing line, the dotted line that appears. Move the point to the inferencing line to align point with the center of the body, **Fig. 44**. Click OK  in the Point Property Manager.

Step 13. Click OK  in the Hole Wizard Property Manager, **Fig. 45**.

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

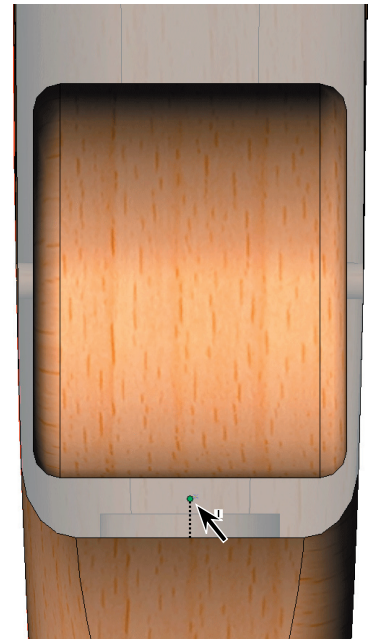


Fig. 44

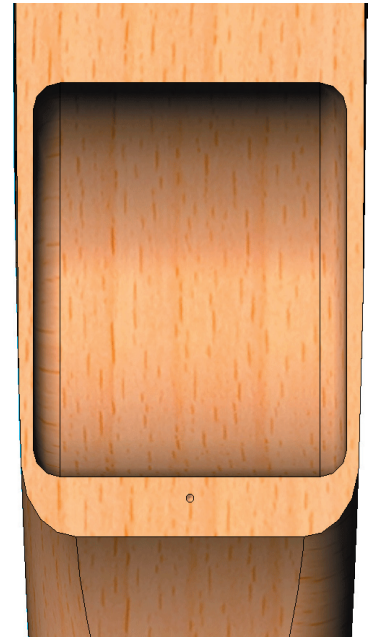
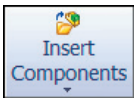


Fig. 45

S. Insert Eye Screws into Assembly.

Step 1. Switch back to the ASSEMBLY file. Use Window Menu > CO2 CAR ASSEMBLY.SLDASM. Click Yes to rebuild.

Step 2. Use the arrows on the keyboard to rotate assembly to view the bottom eye screw holes as shown in **Fig. 46**.

Step 3. Click **Insert Components**  on the Assembly toolbar.

Step 4. Click **Keep Visible**  in the Property Manager.

Step 5. Click **Browse** in the Property Manager.

Step 6. Select your **EYE SCREW** file and click Open.

Step 7. Insert two eye screws. Click approximately where the eye screws are positioned in **Fig. 46**. Click OK  in the Property Manager when done.

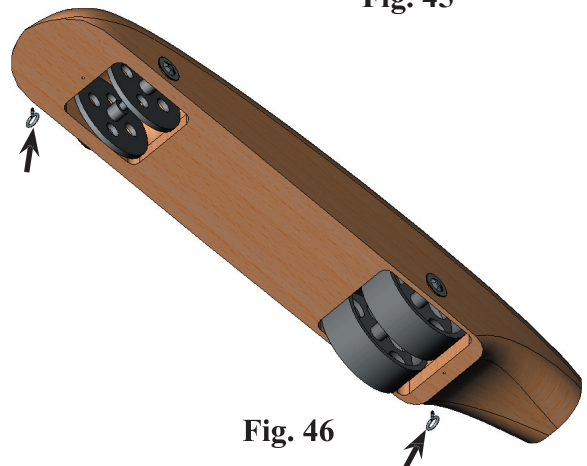
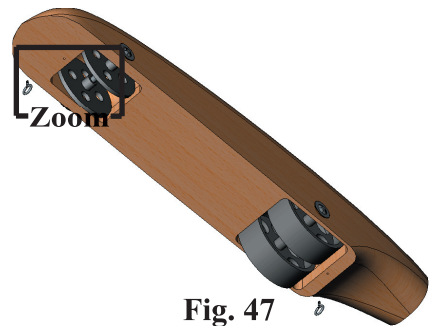


Fig. 46

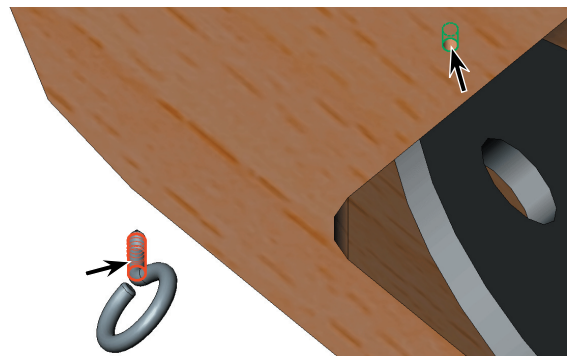
T. Mate: Eye Screw and Body.

Step 1. Use the **Zoom to Area**  in the View toolbar to drag a zoom window around the **front eye screw and hole**, **Fig. 47**.



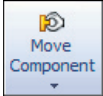
Step 2. Click **Mate**  on the Assembly toolbar.

Step 3. Click **cylindrical inside face of the hole in body** and **cylindrical face of eye screw** just above the threads, **Fig. 48**.



Step 4. Click Add/Finish Mate  in Mate pop-up toolbar to add a **Concentric** mate, **Fig. 49**.

If the becomes hidden inside the body, first close Mate Property Manager. Click the part in Feature Manager (first eye screw). Then,

click Move Component  on the Assembly toolbar and **drag down in drawing area away from any part**. Do not drag on any part. Click OK  in the Property Manager when done with move.



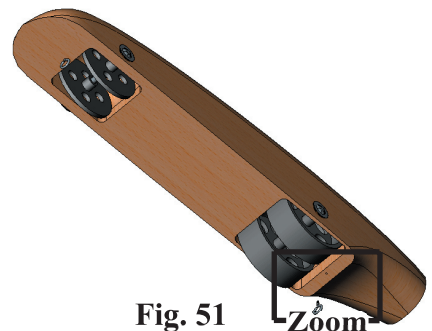
Step 5. If necessary, click **Mate**  on the Assembly toolbar. Click **bottom face of the body** and **round edge of the eye screw**, **Fig. 49**.



Step 6. Click Add/Finish Mate  in Mate pop-up toolbar to add a **Concentric** mate, **Fig. 50**.

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

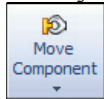
Step 8. Use the **Zoom to Area**  in the View toolbar to drag a zoom window around the **rear eye screw and hole**, **Fig. 51**. Click to Area  again or Escape to unselect Zoom to Area.



Step 9. Click **cylindrical inside face of the hole in body** and **cylindrical face of eye screw**, Fig. 52.

Step 10. Click Add/Finish Mate  in Mate pop-up toolbar to add a **Concentric** mate, Fig. 53.

If the eye screw becomes hidden inside the body, first click OK  to close Mate Property Manager. Click the part in Feature Manager (second eye screw). Then, click Move Component



on the Assembly toolbar and drag down in drawing area. Do not drag on any part. If eye screw does not drag out of the body, check that the eye screw is selected in the Feature Manager before you click Move Component.



Step 11. If necessary, click **Mate**  on the Assembly toolbar. Click **bottom face of the body** and **round edge of the eye screw**, Fig. 53.

Step 12. Click Add/Finish Mate  in Mate pop-up toolbar to add a **Concentric** mate, Fig. 54.

Step 13. Click OK  in the Property Manager when done.

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

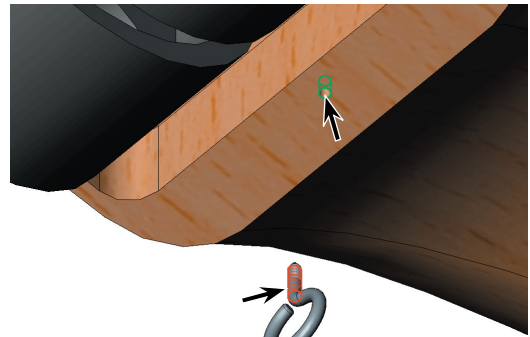


Fig. 52

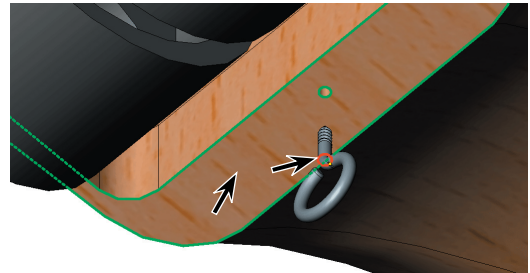


Fig. 53

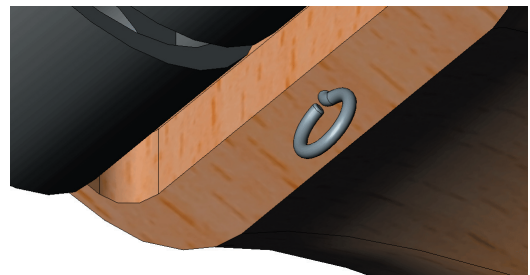


Fig. 54