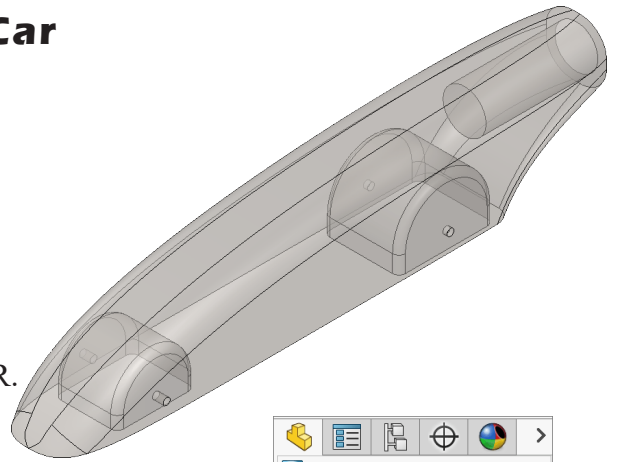




Chapter 2

CO2 Shell Car Body



A. Save as "BODY".

Step 1. If necessary, open your **BLANK** file.

Step 2. Click File Menu > Save As.

Step 3. Key-in **BODY** for the filename and press ENTER.

B. FRONT Wheel Shell.

Step 1. Click **Right Plane**  in the Feature Manager and click **Sketch**



from the context toolbar, **Fig. 1**.

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

Step 3. Zoom in around **front of the body at the axle hole**, **Fig. 2**. To zoom, place the cursor over the front axle hole and spin the wheel on mouse back. While spinning the wheel keep cursor on front axle hole.

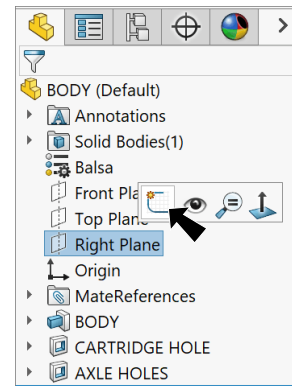


Fig. 1

Step 4. Click **Centerpoint Arc**  (S) in the **Arc flyout**  on the Sketch toolbar.

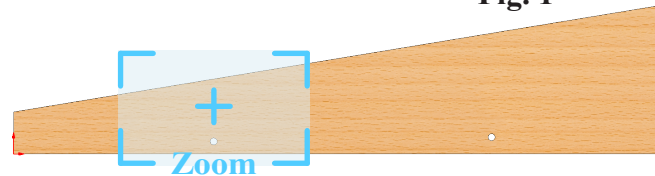


Fig. 2

Step 5. Click in the center of the axle hole to start the arc and move the cursor to the right. Click to place the first endpoint, then move cursor counterclockwise 180 degrees. Click to place the second endpoint, **Fig. 3**.

Tip: Confirm **Angle Snap** is turned on. To turn on, click **Options**  on the Standard toolbar. Click **System** tab and under **Sketch Relation/Snaps**, check **Angle** at the bottom of dialog box.

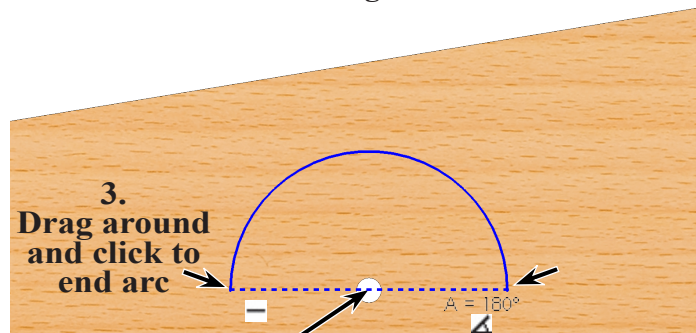


Fig. 3

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

Step 7. **Dimension arc 18**, **Fig. 4**. The front wheel diameter is 34.7 mm. We're setting the shell radius to slightly larger than 1/2 the wheel diameter.

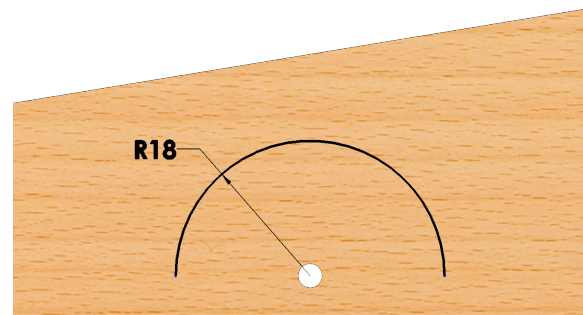
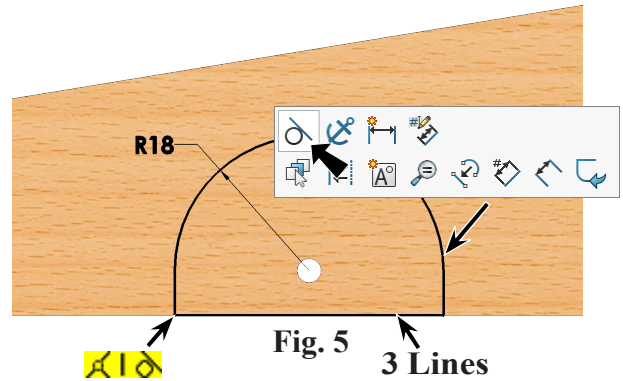


Fig. 4

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

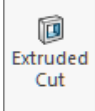
Step 9. Sketch the **3 lines**, **Fig. 5**. Start sketch at left endpoint of arc and sketch to bottom edge. Use the vertical tangent inferencing relations  to create tangent relation to arc. Continue line across edge until directly under right endpoint of arc, then click. Sketch vertical line up to arc endpoint and click **Make**

Tangent  on the Constraint pop-up toolbar, **Fig. 5**. The sketch entities should be all black indicating the sketch is fully defined.

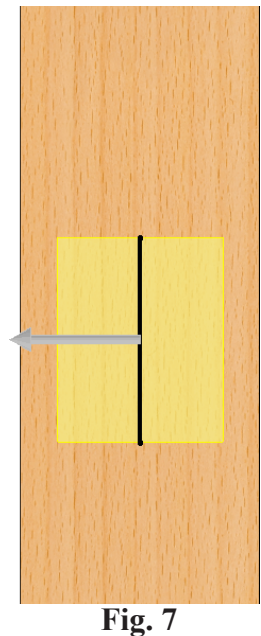
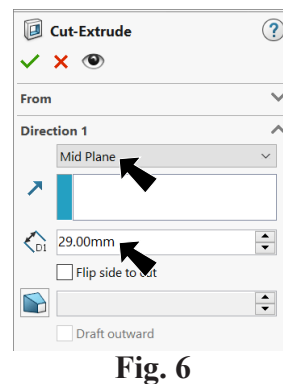


Step 10. Click **Bottom**  on the Standard Views toolbar. (Ctrl-6)

Step 11. Click **Features**  on the Command Manager toolbar.

Step 12. Click **Extruded Cut**  on the Features toolbar.

Step 13. In the Cut-Extrude Property Manager set:
under Direction 1, **Fig. 6**
End Condition **Mid Plane**
Depth  **29**
click OK .

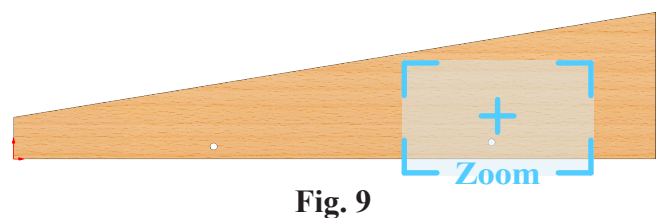
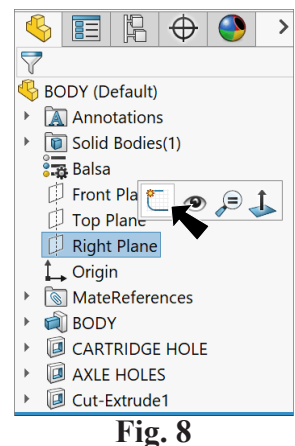


C. REAR Wheel Shell.

Step 1. Click **Right Plane**  in the Feature Manager and click **Sketch**  from the context toolbar, **Fig. 8**.

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

Step 3. Zoom in around **rear of the body at the axle hole**, **Fig. 9**. To zoom, place the cursor over the rear axle hole and spin the wheel on mouse back. While spinning the wheel keep cursor on rear axle hole.



Step 4. Click **Centerpoint Arc**  (S) in the **Arc flyout**  on the Sketch toolbar.

Step 5. Click in the center of the axle hole to start the arc and move the cursor to the right. Click to place the first endpoint, then move cursor counterclockwise 180 degrees. Click to place the second endpoint, **Fig. 10**.

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

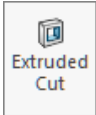
Step 7. **Dimension arc 22, Fig. 11**. The front wheel diameter is 34.7 mm. We're setting the rear shell radius to slightly larger than 1/2 the wheel diameter.

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

Step 9. Sketch **3 lines, Fig. 12**. Use the vertical tangent inferring relations  to create tangent relation to arc. Use **Make Tangent**  on the Constraint pop-up toolbar to add tangent relation to the right vertical line, **Fig. 12**.

Step 10. Click **Bottom**  on the Standard Views toolbar. (**Ctrl-6**)

Step 11. Click **Features**  on the Command Manager toolbar.

Step 12. Click **Extruded Cut**  on the Features toolbar.

Step 13. In the Cut-Extrude Property Manager set:
under Direction 1, **Fig. 13**
End Condition **Mid Plane**

Depth  **D1** 35
click OK .

Step 14. Save  (**Ctrl-S**).

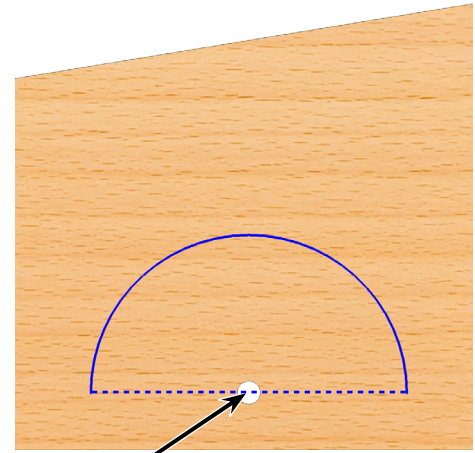


Fig. 10
Start at axle center

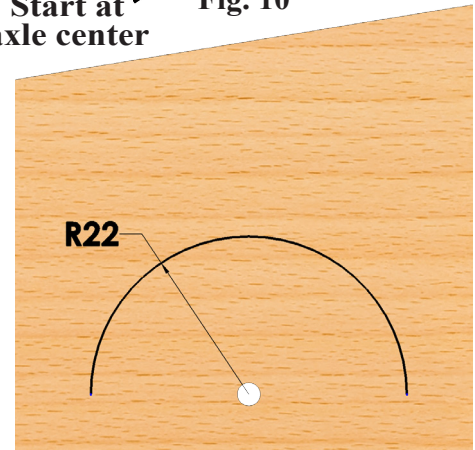


Fig. 11

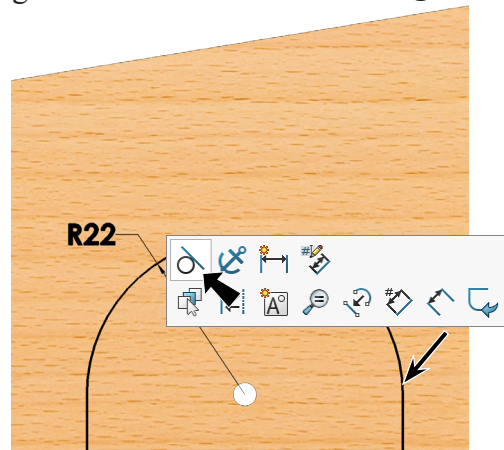


Fig. 12

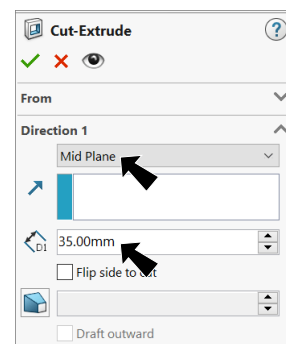


Fig. 13



Fig. 14

D. Rename Shell Features.

Step 1. Rename Cut-Extrude3 and Cut-Extrude4 to **FRONT SHELL** and **REAR SHELL** in the Feature Manager, **Fig. 15**. To rename, click feature and press **F2** on keyboard. Key-in new name.

E. Fillet Inside Edges Wheel Shells.

Step 1. Use **Left Arrow** key  on keyboard twice to rotate body to view **inside cylindrical edge** of shells, **Fig. 17**.

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

Step 3. In the Fillet Property Manager set:
select **FilletXpert**, **Fig. 16**

Radius  **3**

click an **inside cylindrical edge** of a shell, **Fig. 17**

and click **All concave, 12 Edges**  on the

Fillet pop-up toolbar

click to **unselect bottom edge** of Cartridge hole,

Fig. 18 and **Fig. 19**

click **OK** .

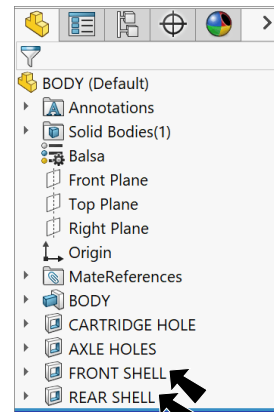


Fig. 15

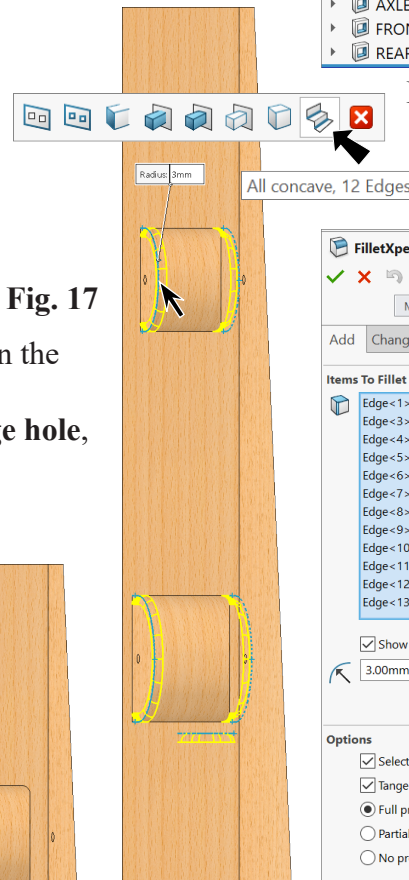


Fig. 17

Fig. 16

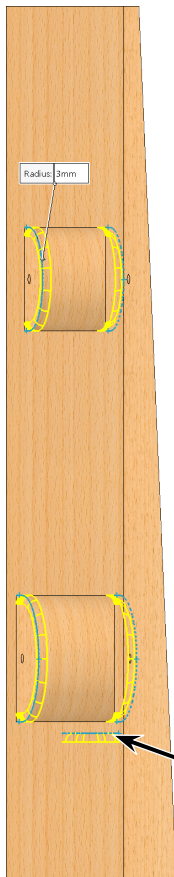


Fig. 18

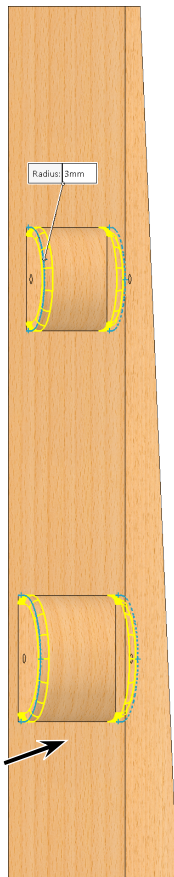


Fig. 19

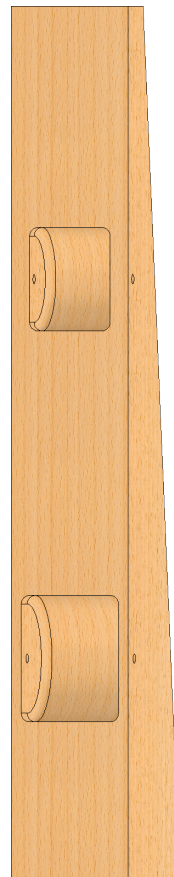


Fig. 20

F. Side Cut.

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

Step 2. Click the **side face** and click **Sketch**  on the context toolbar, **Fig. 21**.

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

Step 4. Click **Wireframe**  on the View toolbar.

Step 5. Click **Style Spline**  in the **Spline flyout**  on the Sketch toolbar.

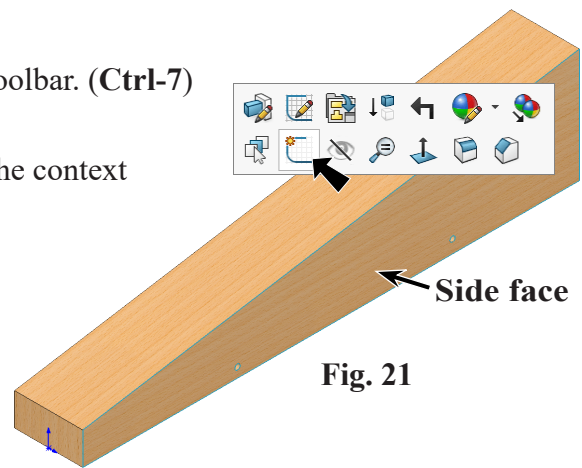


Fig. 21

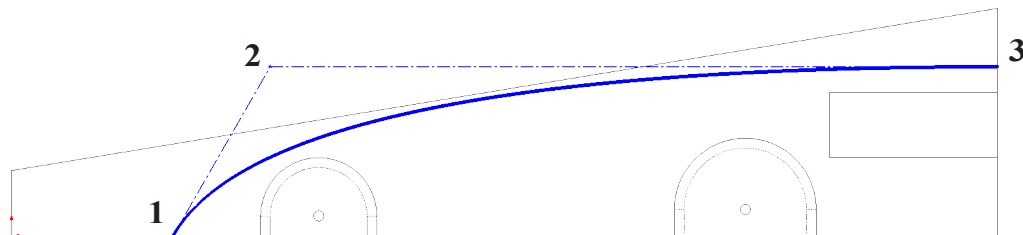


Fig. 22

Step 6. Sketch a **3 control vertex point Style Spline** between bottom edge of Body and back edge of Body, **Fig. 22**. To sketch Style Spline, click bottom edge of Body between front of Blank and Wheel shell for 1st control vertex point. Click above front Wheel shell for 2nd control vertex point. Then, click rear edge of Body above Cartridge hole for 3rd control vertex point. Press Escape to end spline.

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

Step 8. Add dimensions, **Fig. 23**.

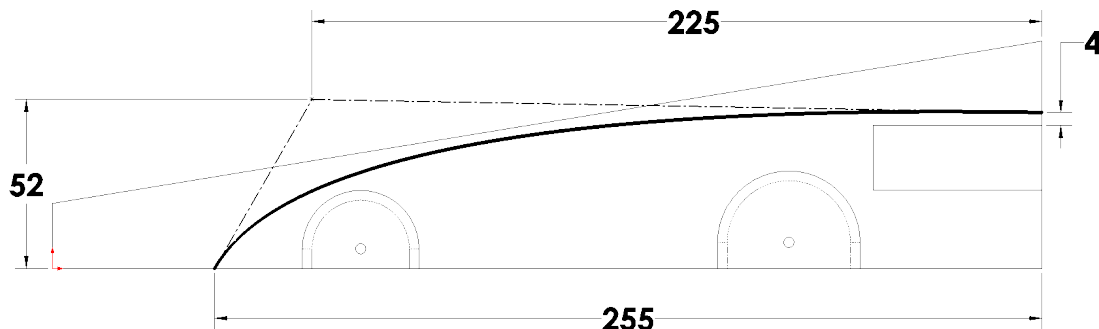


Fig. 23

Step 9. Click **Style Spline**  in the **Spline flyout**  on the Sketch toolbar.

Step 10. Sketch a **3 control vertex point Style Spline** under Cartridge hole, **Fig. 24**. To sketch Style Spline, click bottom edge of Body behind rear shell for 1st control vertex point. Click above and to the rear for 2nd control vertex point. Then, click rear edge of Body just below Cartridge hole for 3rd control vertex point. Press Esc to end spline.

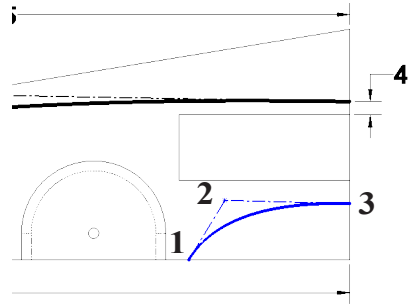


Fig. 24

Step 11. Click **Smart Dimension**  (S) on the Sketch toolbar.

Step 12. Add dimensions, **Fig. 25**.

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

Step 14. Sketch a line to **connect the ends of the splines**, **Fig. 26**.

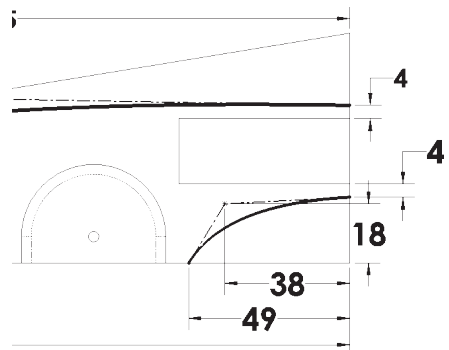
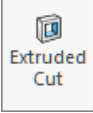


Fig. 25

Step 15. Click **Features**  on the Command Manager toolbar.

Step 16. Click **Extruded Cut**  on the Features toolbar.

Step 17. In the Cut-Extrude Property Manager:
under Direction 1, **Fig. 27**
End Condition **Through All**
The **Direction** arrow should point towards area to be cut away, **Fig. 28**. If arrow is pointing in wrong direction, check **Flip side to cut**, **Fig. 27**.
Click OK .

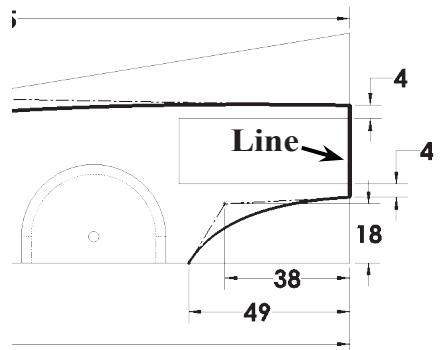


Fig. 26

Step 18. Click **Isometric**  on the Standard Views toolbar. (Ctrl-7)

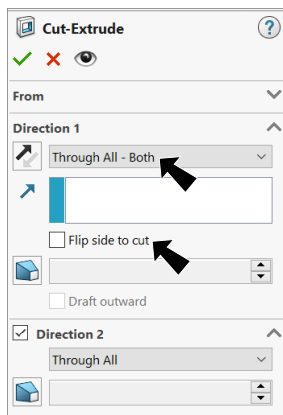


Fig. 27

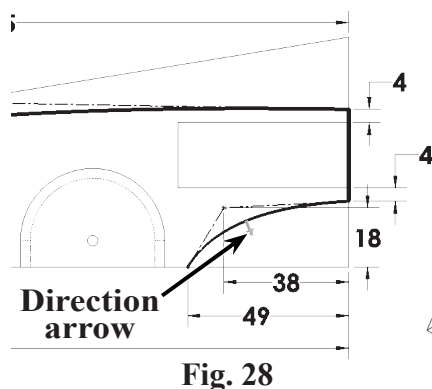


Fig. 28

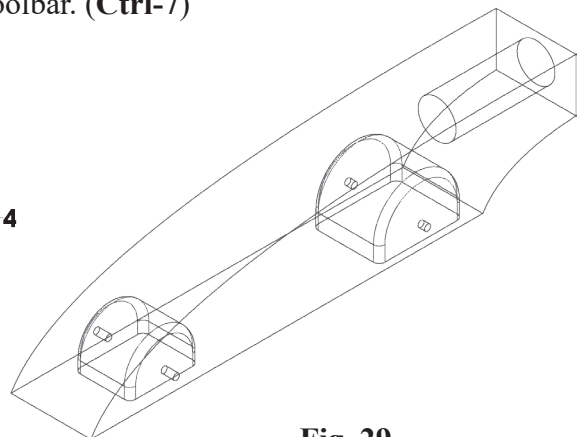


Fig. 29

G. Rename Cut-Extrude5 to SIDE CUT.

Step 1. **Rename Cut-Extrude5 to SIDE CUT** in the Feature Manager, **Fig. 30**. To rename, click **Cut-Extrude5** name in Feature Manager and press **F2** on keyboard. Key-in **SIDE CUT**.

H. Bottom Cut.

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

Step 2. Click the **bottom face** and click **Sketch**  on the context toolbar, **Fig. 31**.

Step 3. Click **Centerline**  in the **Line flyout**  on the Sketch toolbar.

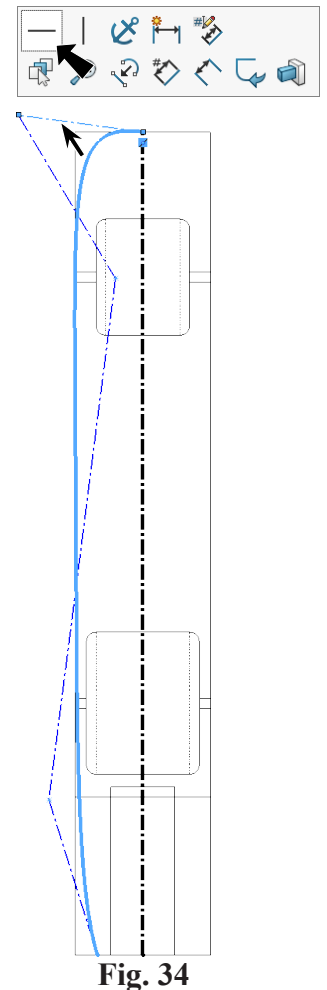
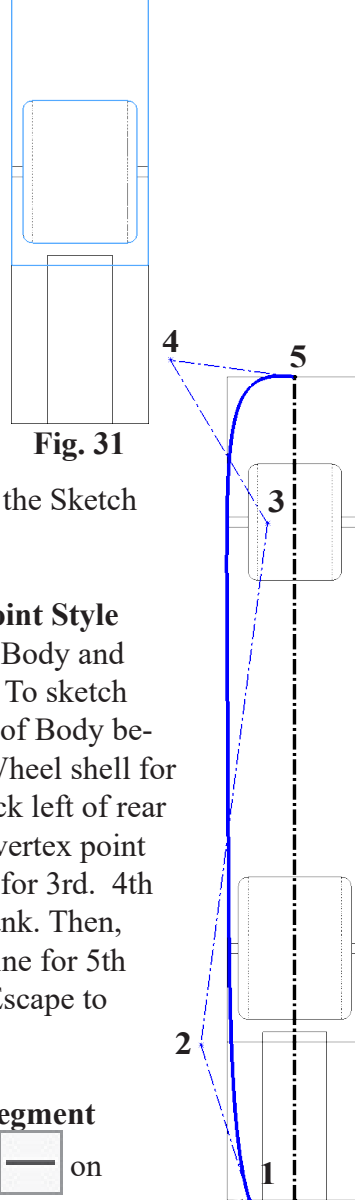
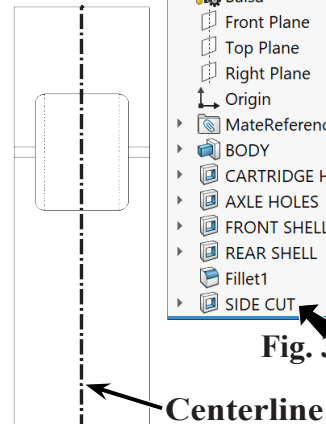
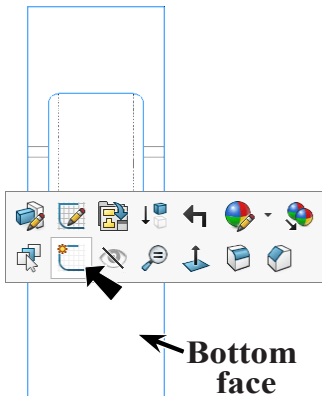
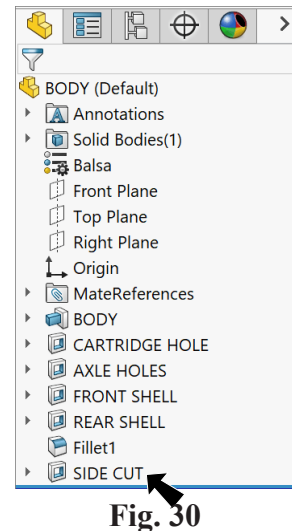
Step 4. Starting from midpoint  of the rear edge of body, sketch a centerline up through the body, **Fig. 32**.

Step 5. Click **Style Spline**  in the **Spline flyout**  on the Sketch toolbar.

Step 6. Sketch a **5 control vertex point Style Spline** between rear edge of Body and front edge of Body, **Fig. 33**. To sketch Style Spline, click rear edge of Body between corner of Blank and Wheel shell for 1st control vertex point. Click left of rear Wheel shell for 2nd control vertex point and inside front Wheel shell for 3rd. 4th is outside front corner of Blank. Then, click top endpoint of centerline for 5th control vertex point. Press Escape to end spline.

Step 7. Click **top control polygon segment** and click **Make Horizontal**  on the context toolbar, **Fig. 34**.

Keep spline point away from cartridge hole



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



Step 9. Add dimensions, **Fig. 35**. Dimensioning will thread the Style Spline between the edge of the Blank and the Wheel shells. If the spline is outside Blank SOLIDWORKS will not cut. If the spline is too close to the shell the shell will break through.

Step 10. **Unselect Smart Dimension**. To unselect, **right click graphics area and click Select** from menu.

Step 11. **Ctrl click spline and centerline** to select both, **Fig. 36**.

Step 12. Click **Mirror Entities** on the Sketch toolbar, **Fig. 37**.

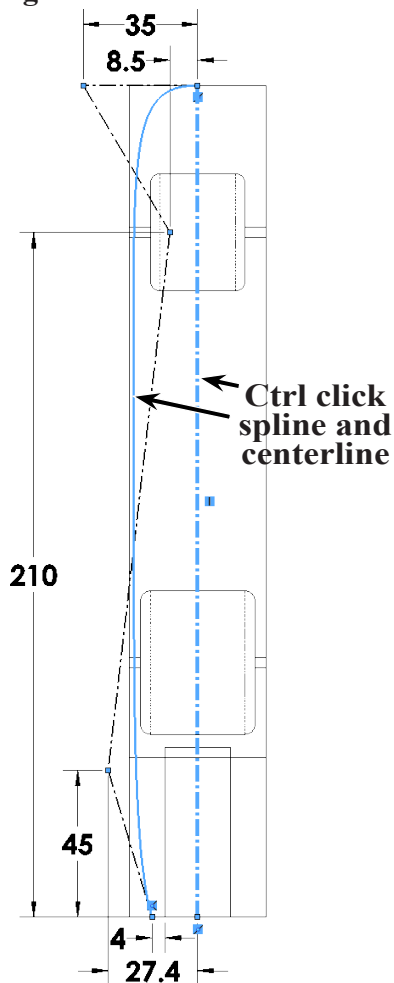
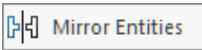


Fig. 36

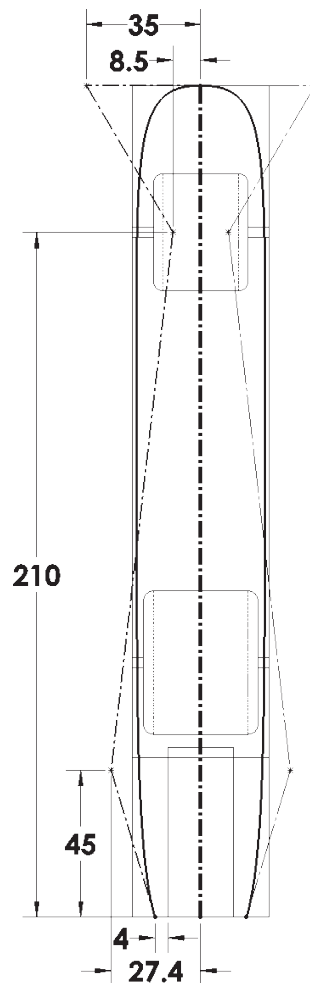


Fig. 37

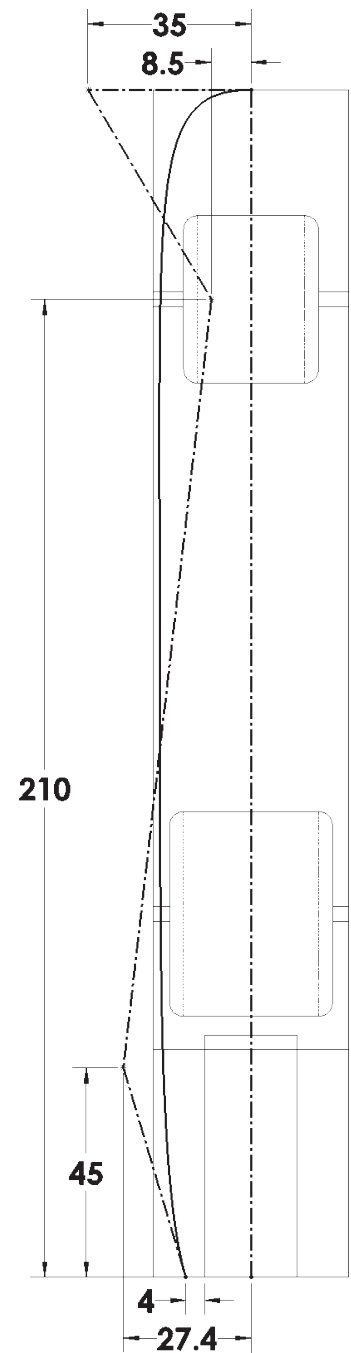
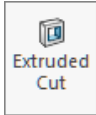


Fig. 35

Step 13. Click **Features**  on the Command Manager toolbar.

Step 14. Click **Extruded Cut**  on the Features toolbar.

Step 15. In the Cut-Extrude Property Manager:
under Direction 1, **Fig. 38**

End Condition
Through All

The **Direction arrow** should point towards area to be cut away, **Fig. 39**. If arrow is pointing in wrong direction, check **Flip side to cut**, **Fig. 38**.

Click OK .

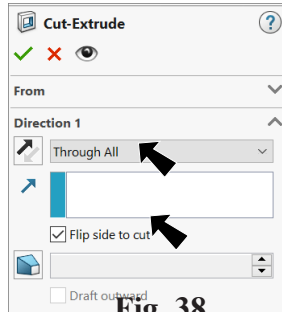


Fig. 38

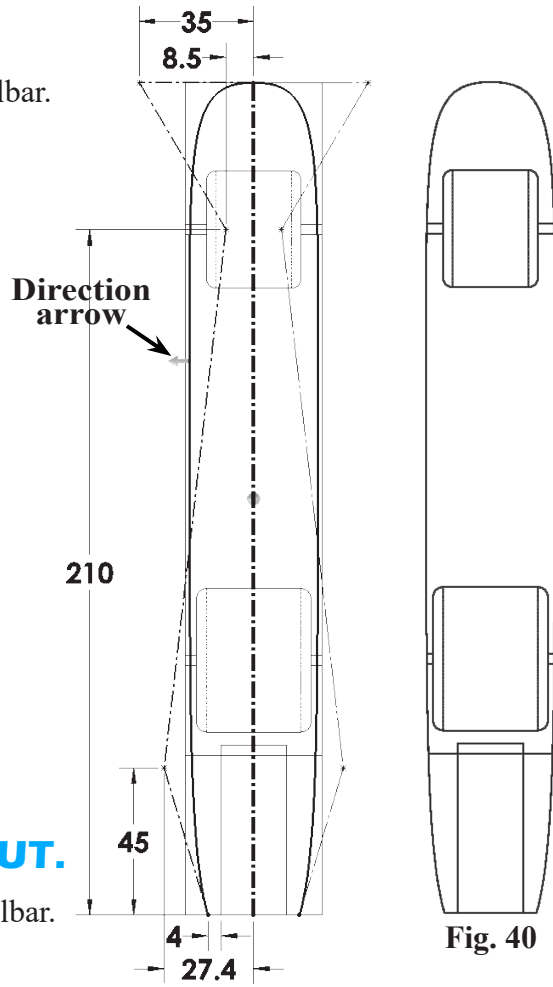


Fig. 40

Fig. 39

Step 16. Click **Shaded With Edges**  on the View toolbar.

I. Rename Cut-Extrude6 BOTTOM CUT.

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

Step 2. **Rename Cut-Extrude6 to BOTTOM CUT** in the Feature Manager, **Fig. 42**. To rename, click **Cut-Extrude4** name in Feature Manager and press **F2** on keyboard. Key-in **BOTTOM CUT**.

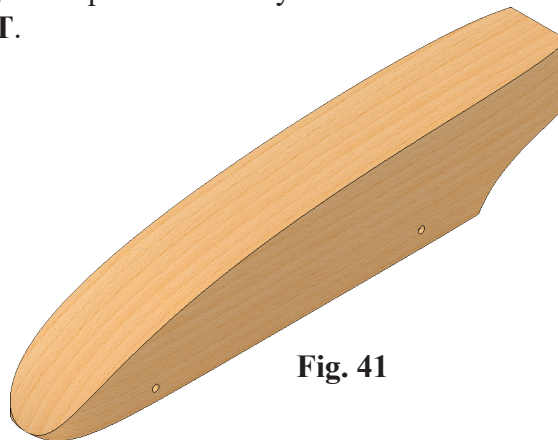


Fig. 41

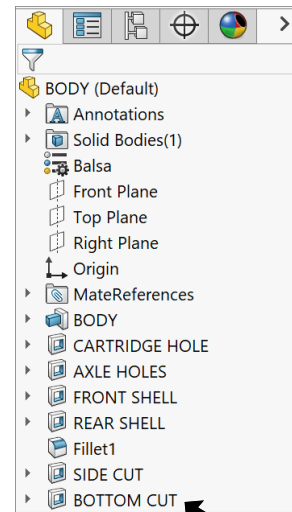


Fig. 42

J. Fillet Edges.

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

Step 2. In the Fillet Property Manager set:
select **FilletXpert**, Fig. 43

Radius  **12**
select **Full preview**

Click a **top edge** of body and selection will propagate to select the other edge, Fig. 44.

Click **Apply**.

Step 3. Rotate view to bottom rear, on the **Reference**

Triad  at bottom left corner of graphics area **Ctrl-Shift** click the **X axis** twice, Fig. 47.

Step 4. In the Fillet Property Manager:
select **Manual**, Fig. 46
under Fillet type

select **Variable Size Fillet** 
click **both edges** under Cartridge hole, Fig. 47
set **Variable radius 12** in rear and **6** in front. To set radius,
click and key-in radius.
Click OK .

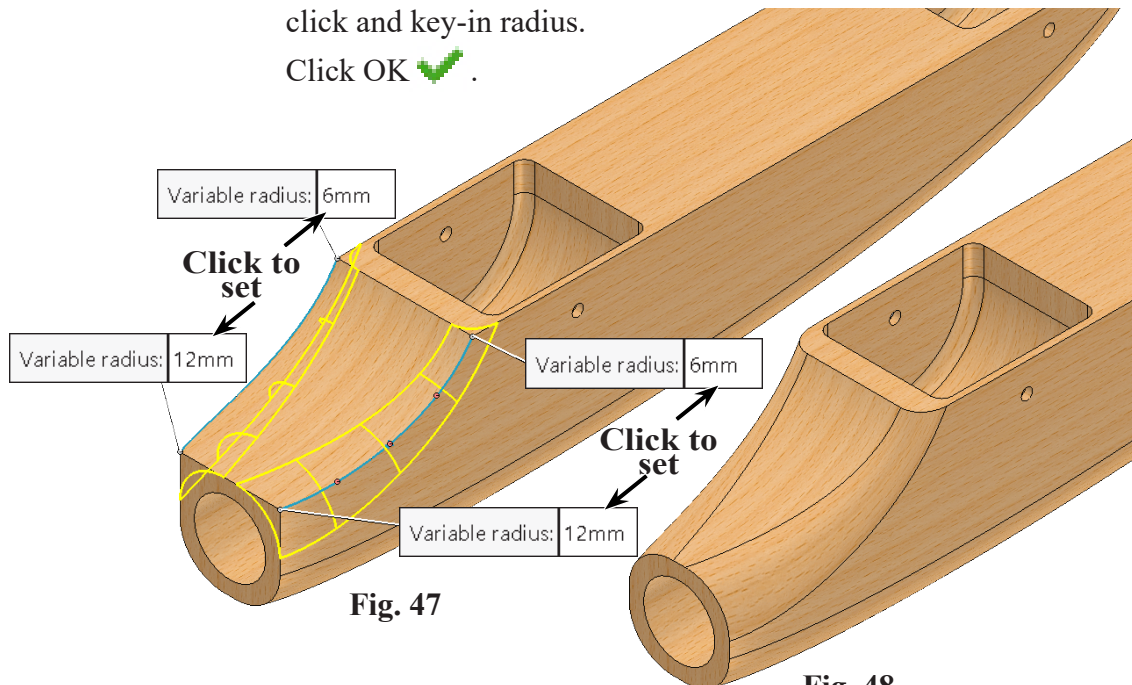


Fig. 47

Fig. 48

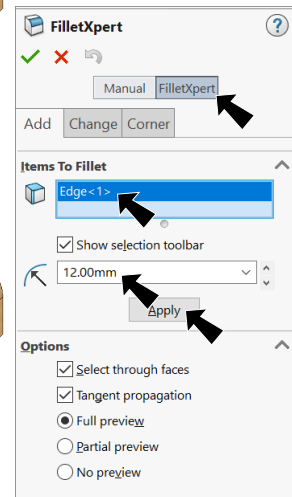


Fig. 43

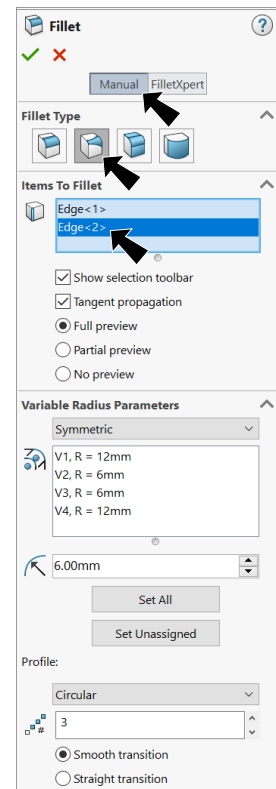


Fig. 46

K. Appearance - Polished Steel.

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

Step 2. Click the Body to select the part, click **Appearances Callout**  on the context toolbar and click **BODY** , Fig. 49.

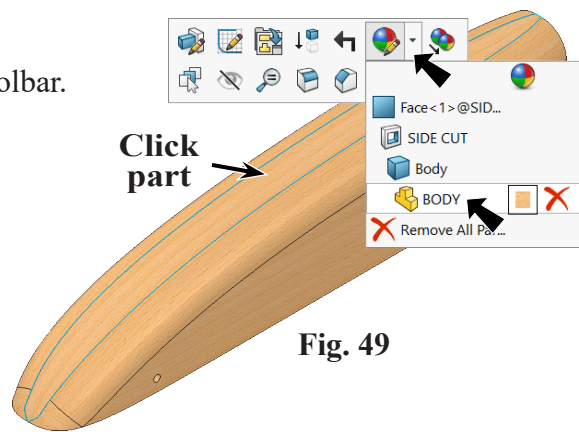


Fig. 49

Step 3. In the Appearances Task pane, expand **Metal**, click **Steel** and in the lower pane select **polished steel**, Fig. 50.

Step 4. In the Appearances Property Manager click OK .

Step 5. Save  (Ctrl-S).

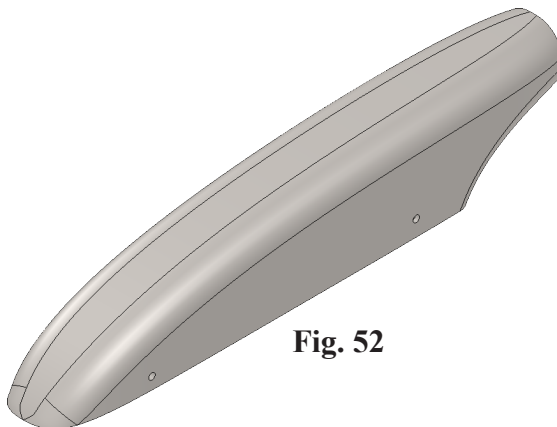


Fig. 52

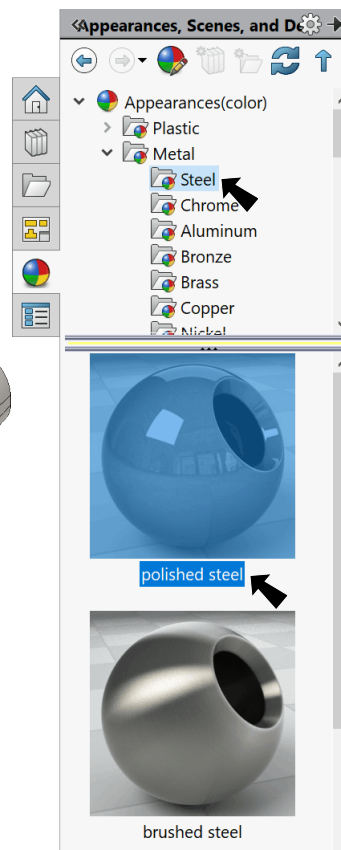


Fig. 50

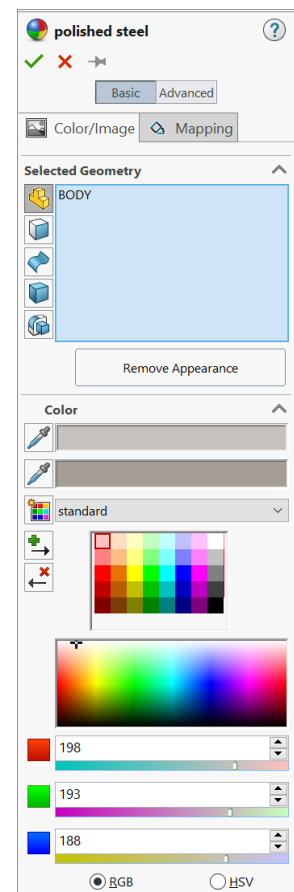


Fig. 51