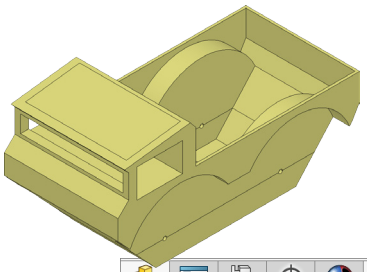
Chapter 25

Cluster Wheel Vehicle
Master Model



A. Extrude1 Sketch1 Body.

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

Step 2. Click **Right Plane** in the Feature Manager and click **Sketch** on the context toolbar, **Fig. 1**.

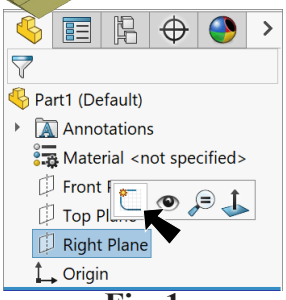


Fig. 1

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

Step 4. Starting above Origin **sketch the 4 lines, Fig. 2.** Use the inferencing line to create vertical relation and horizontal relation with Origin .

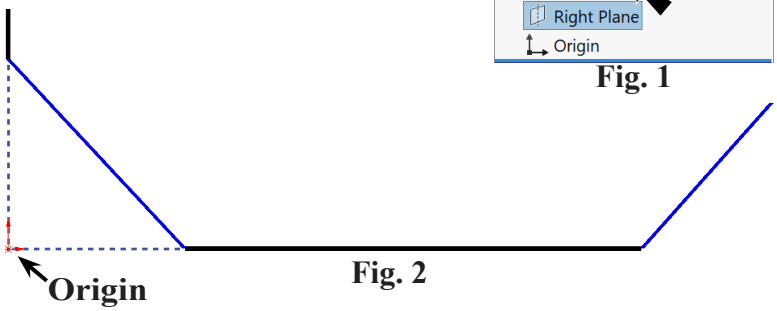


Fig. 2

Step 5. Continue and **sketch the 9 lines, Fig. 3.**

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

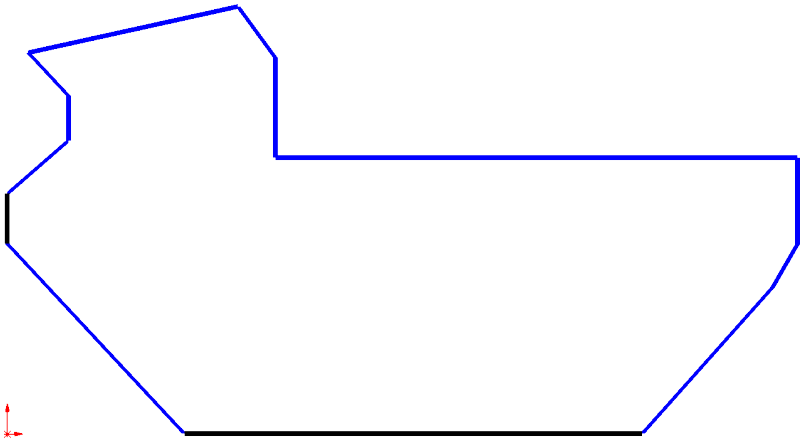


Fig. 3

Step 7. Add dimensions, **Fig. 4.**

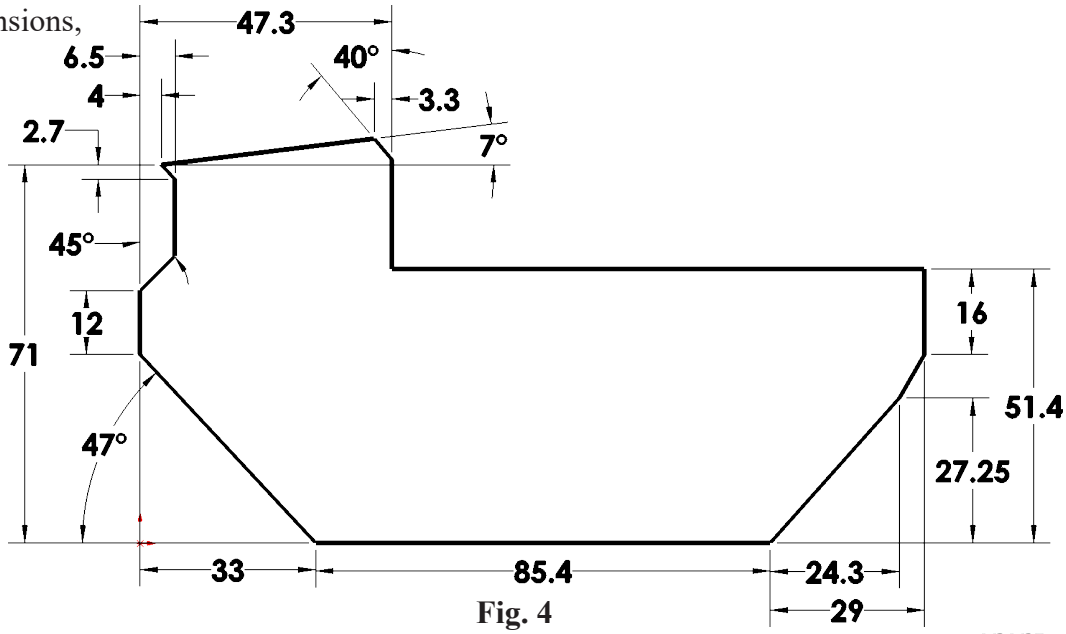


Fig. 4

1/31/25

Step 8. Click **Features** Features on the Command Manager toolbar.

Step 9. Click **Extruded Boss/Base** Extruded Boss/Base on the Features toolbar.

Step 10. In the Boss-Extrude Property Manager:
under Direction 1, **Fig. 5**
End Condition **Mid Plane**
Depth D1 **69.5**
click OK .

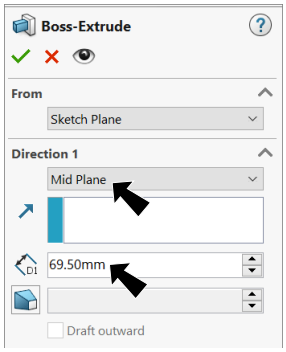


Fig. 5

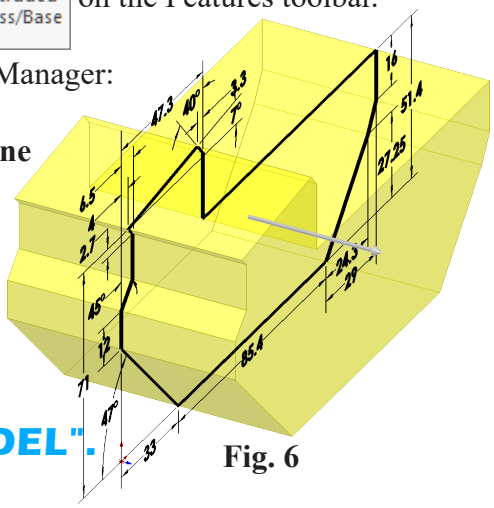


Fig. 6

B. Save as "MASTER MODEL".

Step 1. Click File Menu > Save As.

Step 2. Key-in **MASTER MODEL** for the filename and press ENTER.

C. Extruded Cut1 Sketch2 Wheel Wells.

Step 1. Click the **side face** and click **Sketch** on the context toolbar, **Fig. 7**.

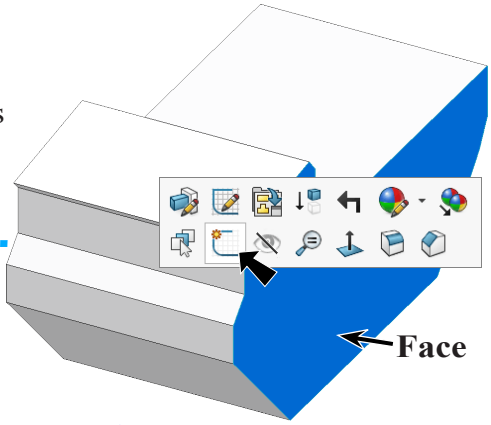


Fig. 7

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

Step 3. Sketch **two circles**, **Fig. 8**.

Step 4. **Unselect Circle tool**. To unselect, right click graphics area and click Select from menu.

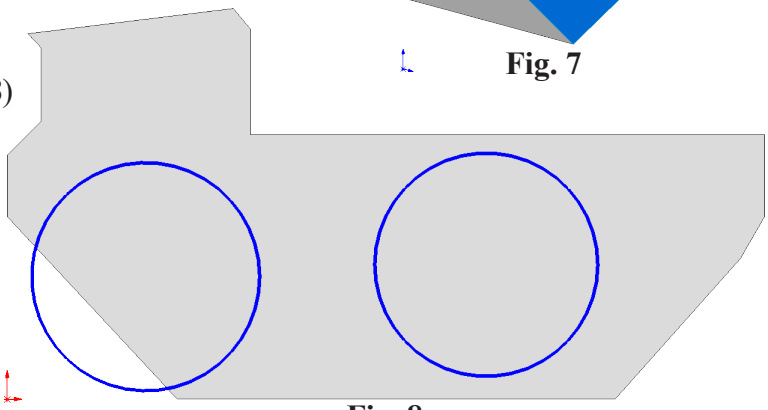


Fig. 8

Step 5. **Ctrl click both circles** to select both. Release Ctrl key and click **Make Equal** on the context toolbar, **Fig. 9**.

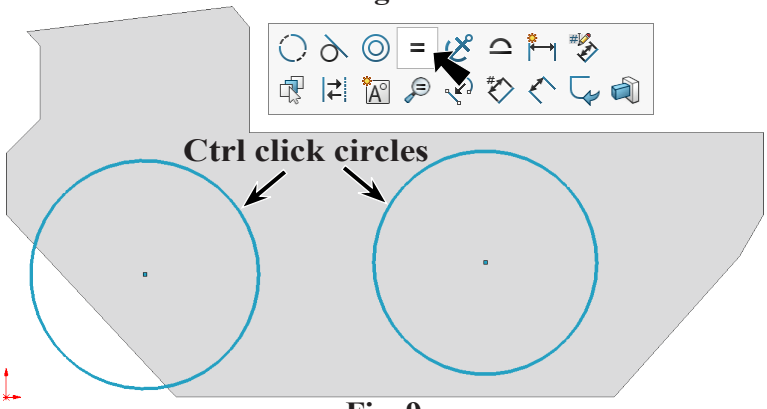


Fig. 9

Step 6. **Ctrl click both circles centerpoints** to select both. Release Ctrl key and click **Make Horizontal**  on the context toolbar, **Fig. 10**.

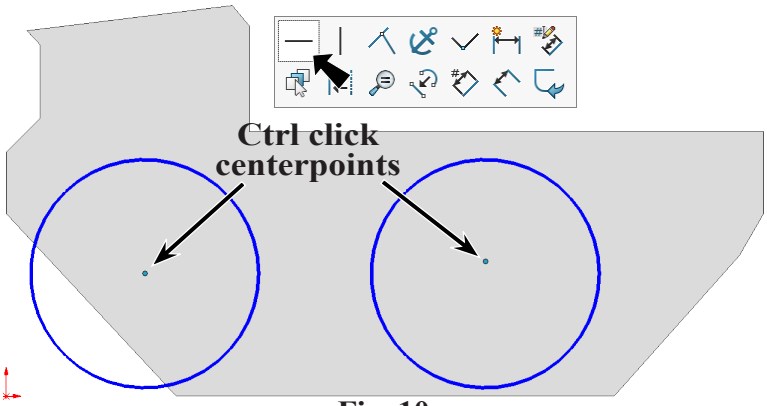


Fig. 10

Step 7. Click **Line**  (**L**) on the Sketch toolbar.

Step 8. Sketch **line horizontal line across circles**, **Fig. 11**. To sketch nontangent line to circle, **click circle and move cursor up very slightly** and then across to other circle.

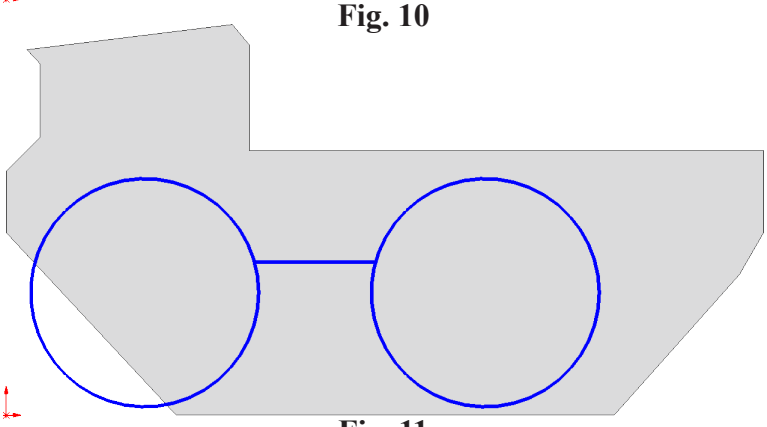


Fig. 11

Step 9. Click **Smart Dimension** 

(**S**) on the Sketch toolbar.

Step 10. Add dimensions, **Fig. 12**.

Step 11. Click **Isometric**  on the Standard Views toolbar. (**Ctrl-7**)

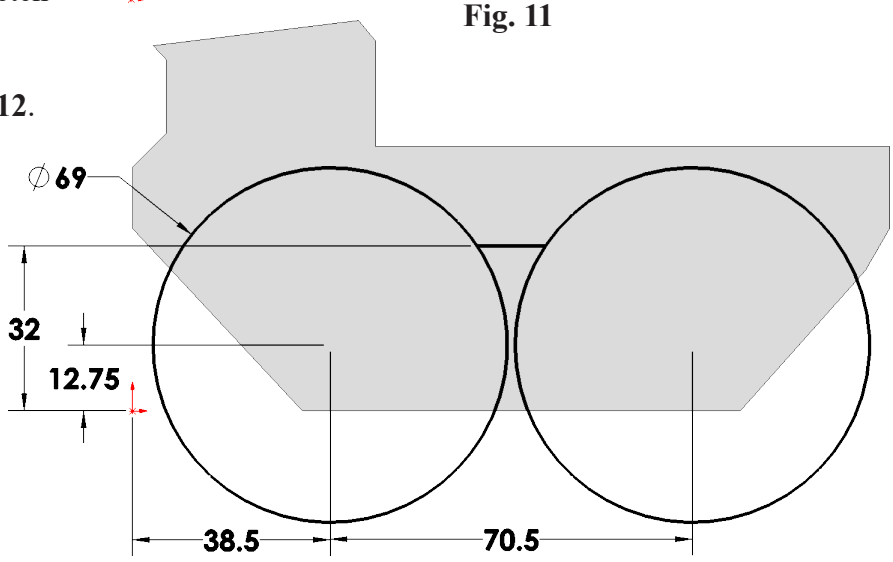


Fig. 12

Step 12. Click **Features** Features on the Command Manager toolbar.

Step 13. Click **Extruded Cut** Extruded Cut on the Features toolbar.

Step 14. In the Cut-Extrude Property Manager set:
under Direction 1, **Fig. 13**
End Condition **Blind**
Depth D1 **10**
under Selected Contours
click the **three contours**, **Fig. 14**
click OK ✔.

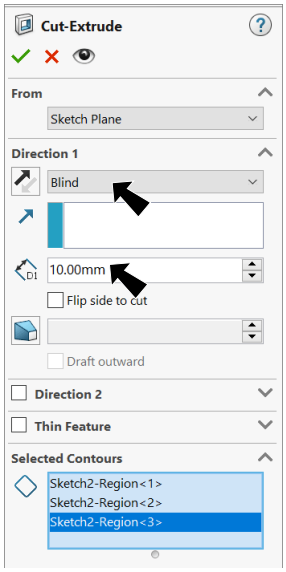


Fig. 13

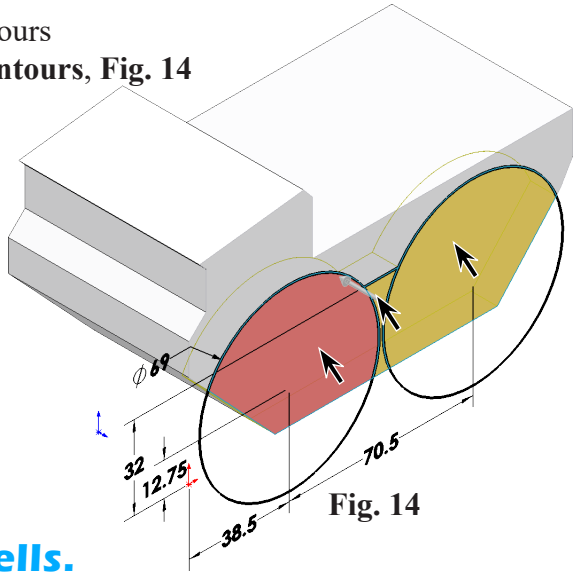


Fig. 14

D. Mirror1 Wheel Wells.

Step 1. **Ctrl click Right Plane** and **Cut-Extrude1** features to select Plane and feature, **Fig. 15**.

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

Step 3. In the Mirror Property Manager click OK ✔.

Step 4. Save (**Ctrl-S**).

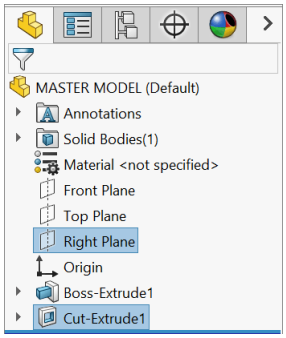


Fig. 15

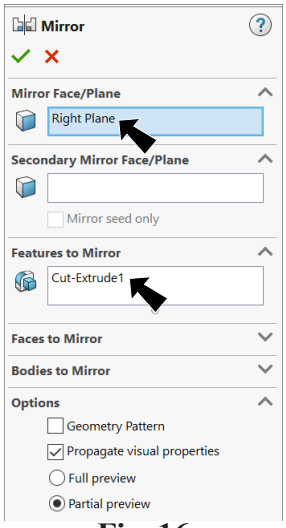


Fig. 16

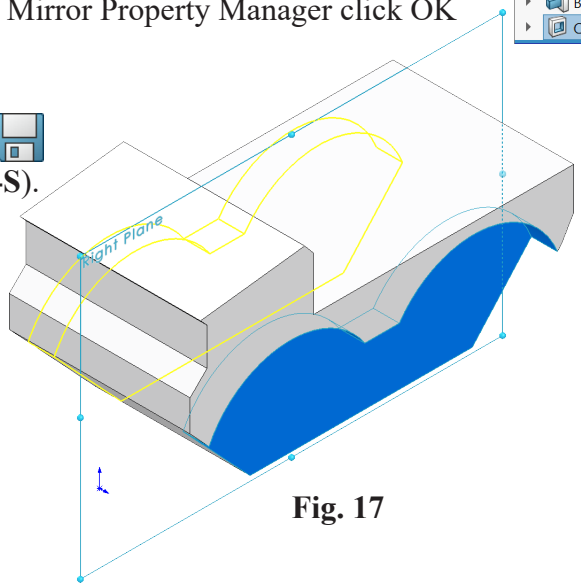


Fig. 17

E. Split1 Sketch3.

Step 1. Click **Right Plane** in the Feature Manager and click **Sketch** on the context toolbar, **Fig. 18**.

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

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

Step 4. Sketch **horizontal line out to left from bottom rear corner of cap and chained line down at angle to edge of body**, **Fig. 19**.

Step 5. Sketch **two lines at angle to horizontal line** to make a triangle, **Fig. 20**.

Step 6. **Unselect Line tool**. To unselect, right click graphics area and click Select from menu.

Step 7. **Drag a selection to left** to select both short lines and click **Construction Geometry** on the context toolbar, **Fig. 21**.

Step 8. **Drag a selection to left** to select both short lines and click **Make perpendicular** on the context toolbar, **Fig. 22**.

Step 9. **Ctrl click the short angle line and long angled line at intersection** to select both. Release Ctrl key and click **Make Collinear** on the context toolbar, **Fig. 23**.

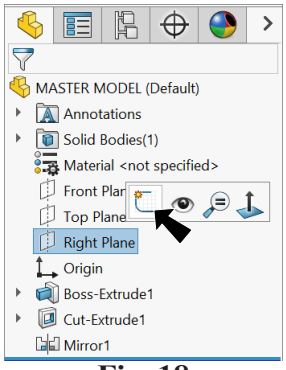


Fig. 18

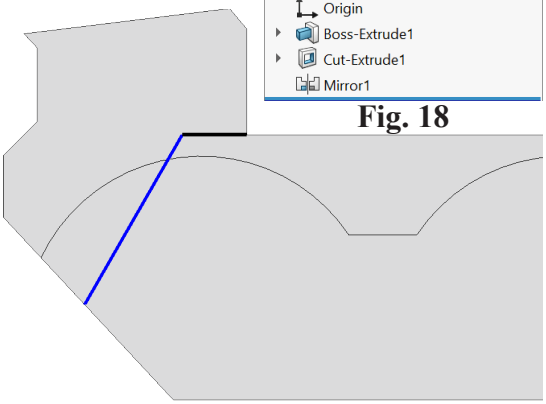


Fig. 19

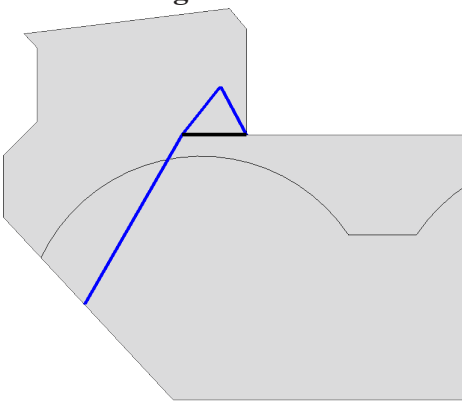


Fig. 20

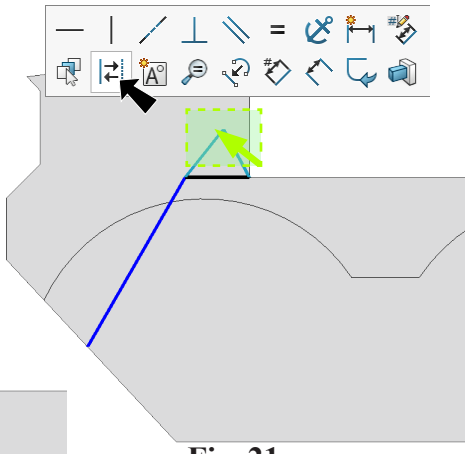


Fig. 21

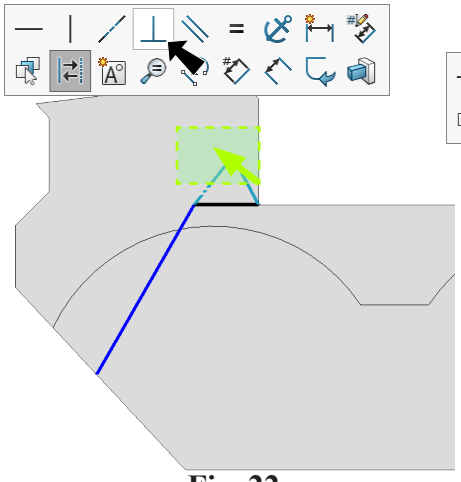


Fig. 22

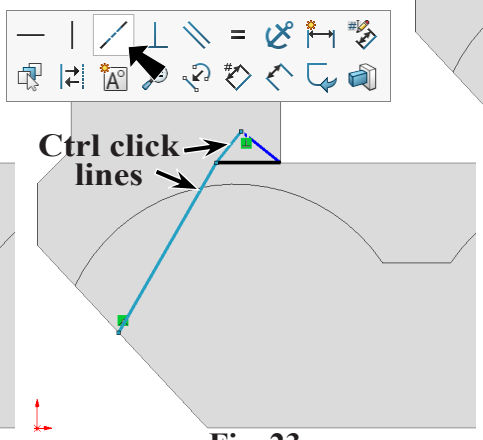


Fig. 23

Step 10. Click **Smart Dimension** 

(S) on the Sketch toolbar.

Step 11. Add dimensions, **Fig. 24**.

Step 12. Click Insert Menu > Features > Split.

Step 13. In the Split Property Manager:
under Trim Tools, **Fig. 25**
Sketch3 was preselected
click **Cut Part** 

button
under Resulting Bodies
click **Select All** 
uncheck **Consume cut bodies**
click OK  .

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

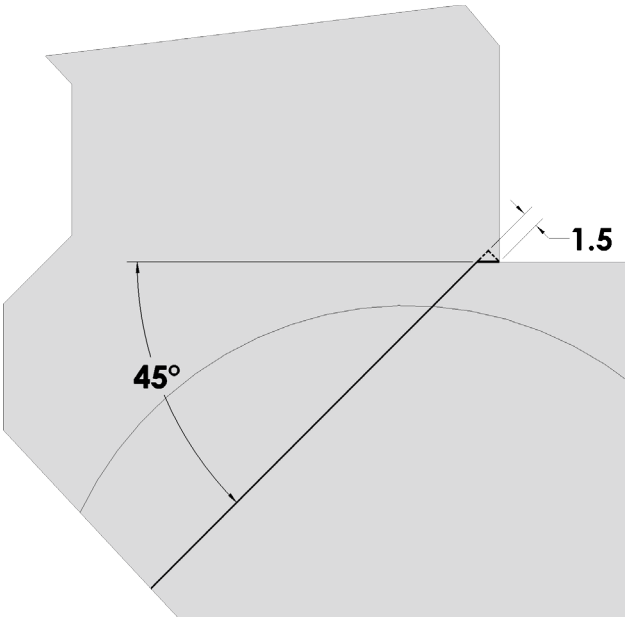


Fig. 24

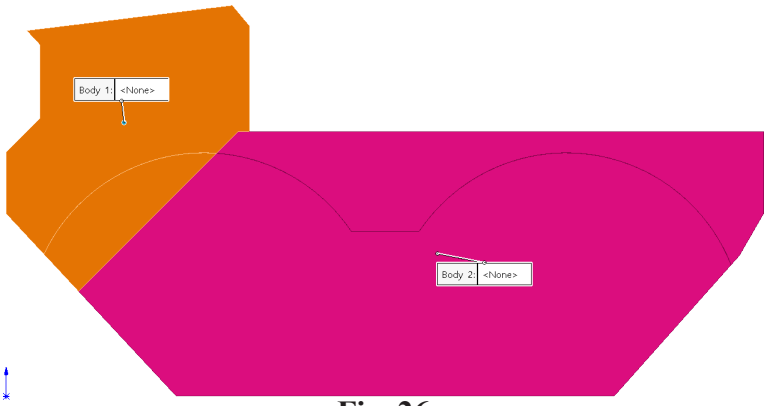


Fig. 26

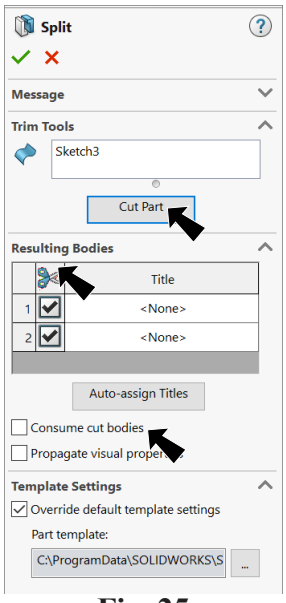


Fig. 25

F. Shell.

Step 1. **Hide Sketch3**. To hide, click **Sketch3** in the Feature Manager and **Hide** on the context toolbar, **Fig. 27**.

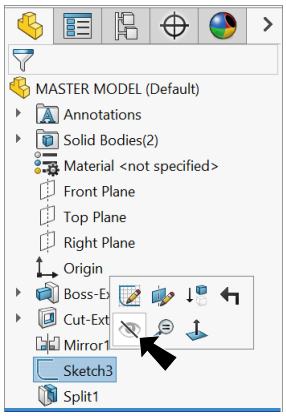


Fig. 27

Step 2. Click **Isometric** on the Standard Views toolbar. (**Ctrl-7**)

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

Step 4. In the Shell Property Manager set:
under Parameters, **Fig. 28**
Distance D1 **1.5**
Face to Remove
click **top face of bed**, **Fig. 29**
check **Show preview**
click OK .

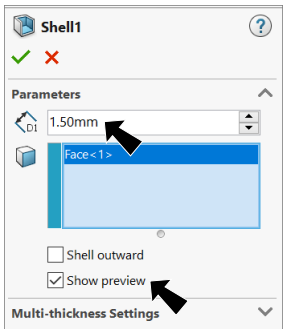


Fig. 28

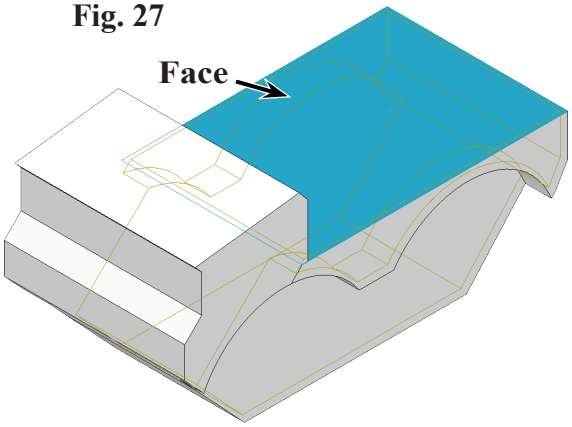


Fig. 29

Step 5. Save (**Ctrl-S**).

G. Combine Bodies1.

Step 1. Click Insert Menu > Features > Combine.

Step 2. In the Combine Property Manager:
under Operation Type, **Fig. 30**
select **Add**
select all Ctrl-A or **drag a selection**, **Fig. 31**
click **Show Preview** Show Preview
click OK .

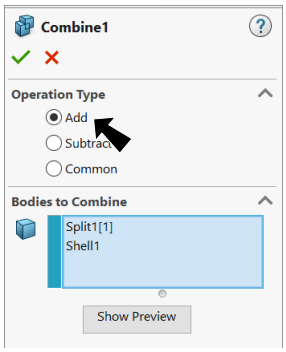


Fig. 30

Step 3. Save (**Ctrl-S**).

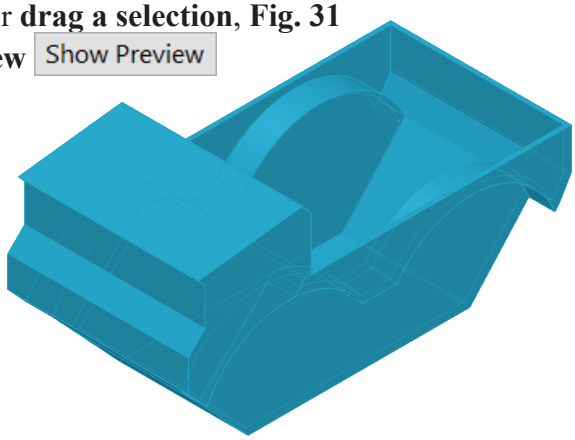


Fig. 31

H. Extruded Cut2 Sketch4 Side Window.

Step 1. Click **Right Plane** in the Feature Manager and click **Sketch** on the context toolbar, **Fig. 32**.

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

Step 3. Click **Corner Rectangle** in the **Rectangle flyout** on the Sketch toolbar.

Step 4. Sketch corner rectangle, **Fig. 33**.

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

Step 6. Add dimensions, **Fig. 34**.

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

Step 8. Click **Features** on the Command Manager toolbar.

Step 9. Click **Extruded Cut** on the Features toolbar.

Step 10. In Cut-Extrude Property Manager:
under Direction 1, **Fig. 35**
End Condition **Through All - Both**
click OK .

Step 11. Save **(Ctrl-S)**.

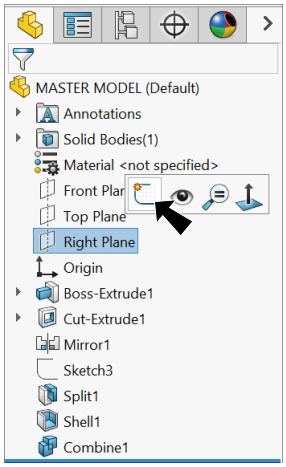


Fig. 32

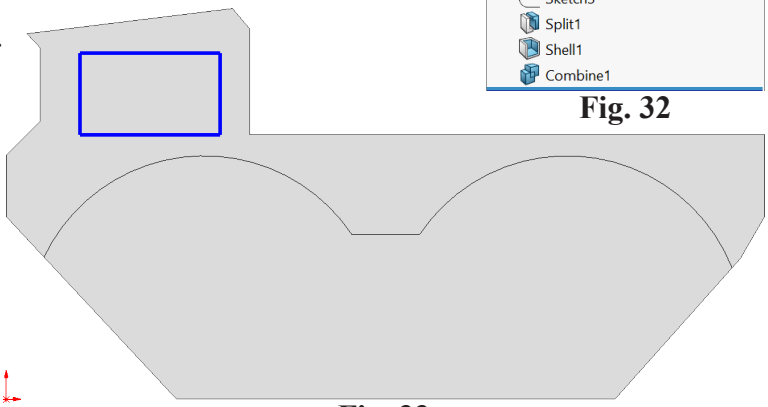


Fig. 33

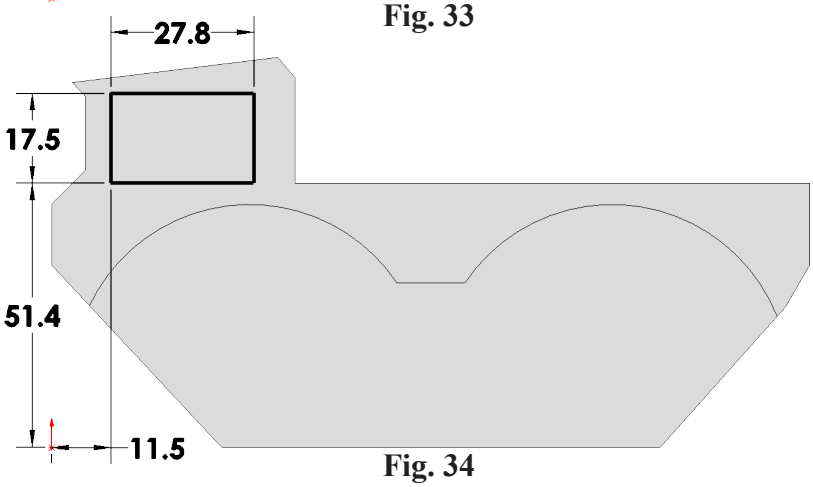


Fig. 34

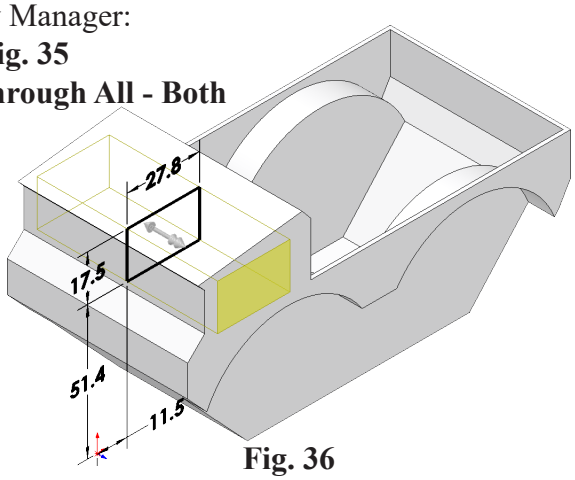


Fig. 36

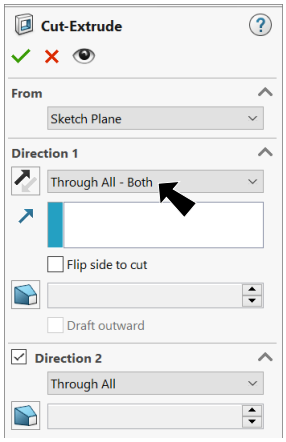


Fig. 35

I. Extruded Cut3 Sketch5 Windshield.

Step 1. Click **Front Plane** in the Feature Manager and click **Sketch** on the context toolbar, **Fig. 37**.

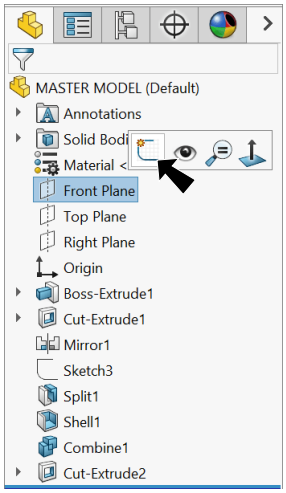


Fig. 37

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

Step 3. Click **Corner Rectangle** in the **Rectangle flyout** on the Sketch toolbar.

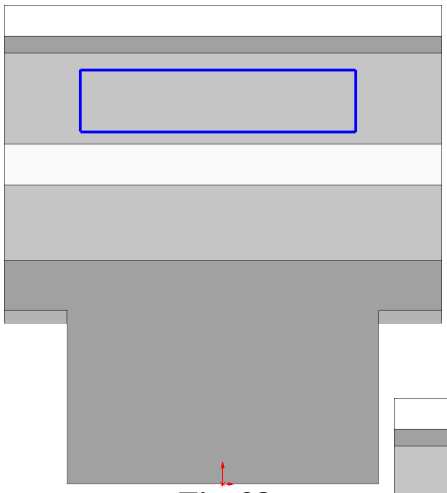


Fig. 38

Step 4. Sketch rectangle, **Fig. 38**.

Step 5. **Unselect Rectangle tool**.

Step 6. **Ctrl click midpoint of a horizontal line** and **Origin** to select both. Release Ctrl key and click **Make Vertical** on the context toolbar, **Fig. 39**.

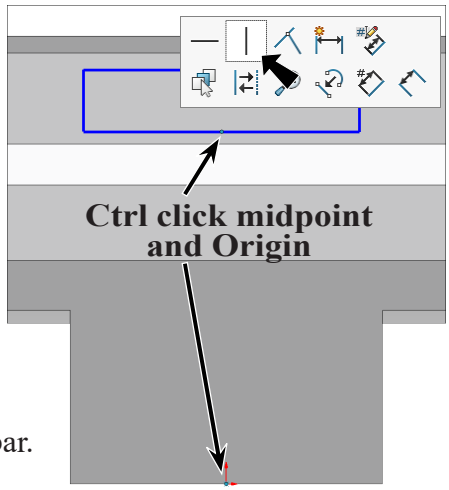


Fig. 39

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

Step 8. Add dimensions, **Fig. 40**.

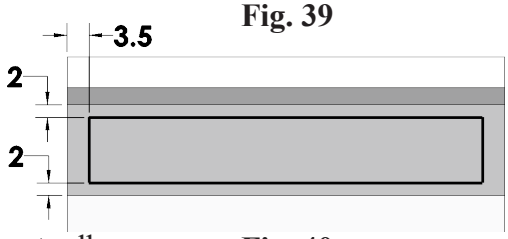


Fig. 40

Step 9. Click **Isometric** on the Standard Views toolbar.

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

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

Step 12. In Cut-Extrude Property Manager:
under Direction 1, **Fig. 41**
End Condition
Up To Next
click OK .

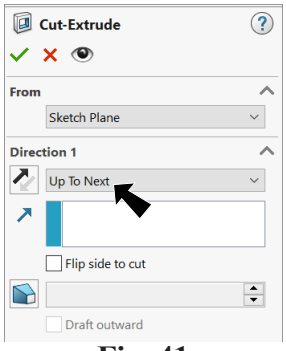


Fig. 41

Step 13. Save **(Ctrl-S)**.

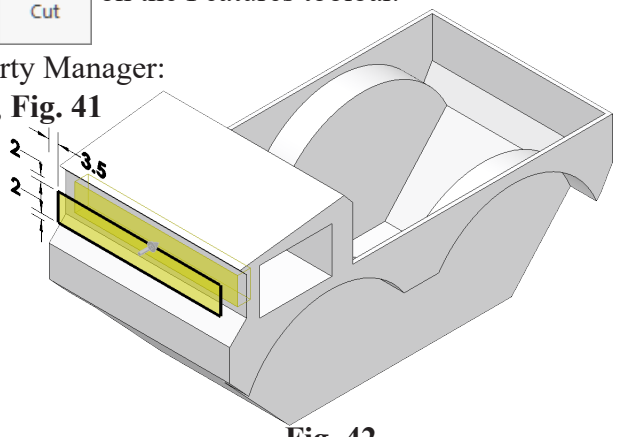


Fig. 42

J. Extruded Cut4 Sketch6 Axle Holes.

Step 1. Click **Right Plane** in the Feature Manager and click **Sketch** on the context toolbar, **Fig. 43**.

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

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

Step 4. Sketch **two circles**, **Fig. 44**.

Step 5. **Unselect Circle tool**. To unselect, right click graphics area and click Select from menu.

Step 6. **Ctrl click both circles** to select both. Release Ctrl key and click **Make Equal** on the context toolbar, **Fig. 45**.

Step 7. **Ctrl click a circle and circular edge of the wheel well** to select both. Release Ctrl key and click **Make Concentric** on the context toolbar, **Fig. 46**.

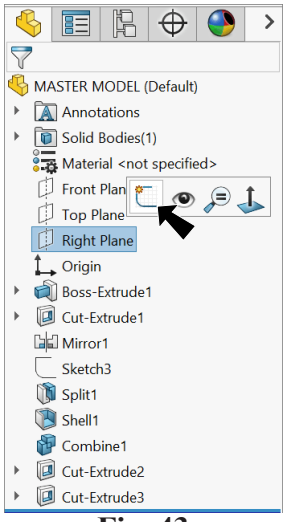


Fig. 43

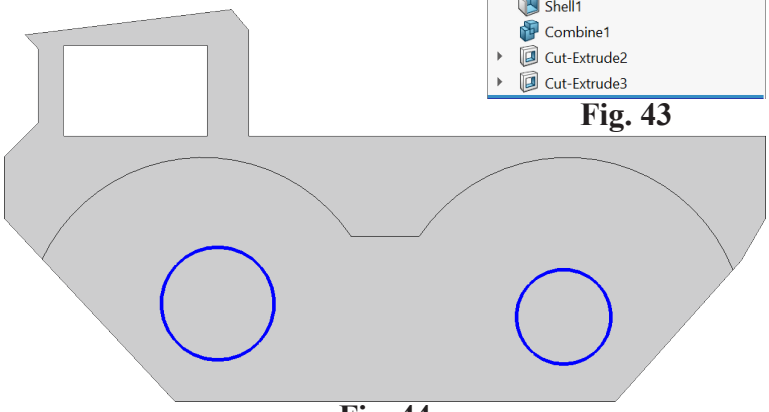


Fig. 44

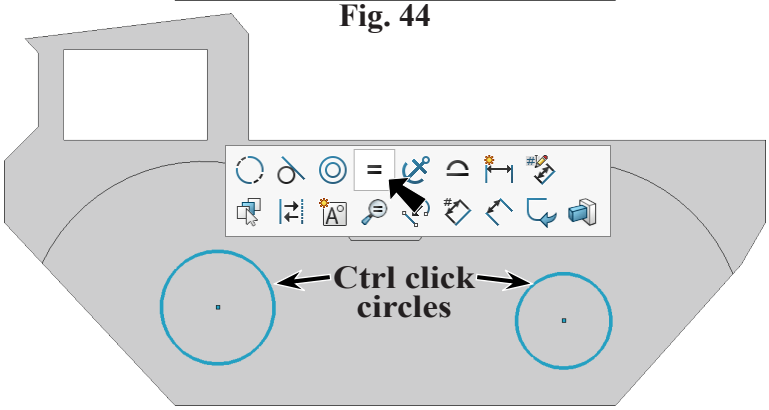


Fig. 45

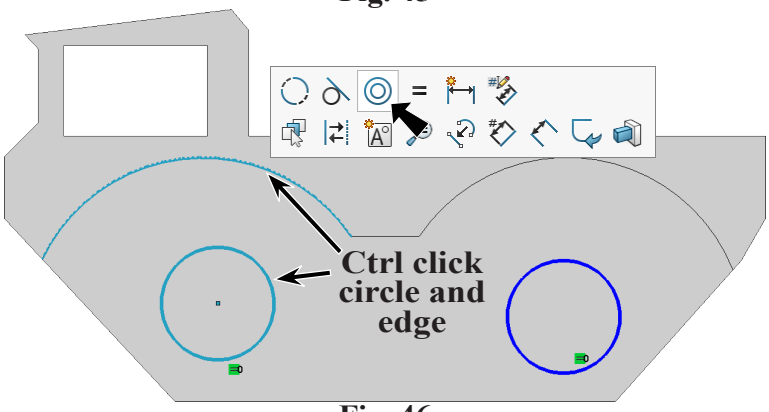


Fig. 46

Step 8. Repeat and add **Concentric** relation to other circle and it's wheel well, **Fig. 47**.

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

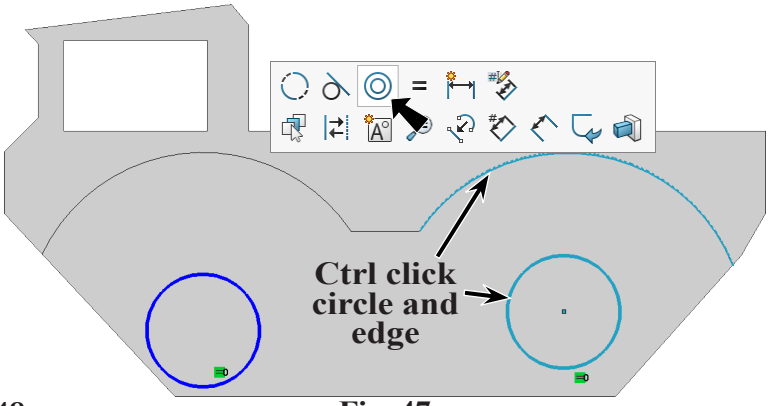


Fig. 47

Step 10. Dimension a diameter **2.3**, **Fig. 48**.

Step 11. Click **Isometric** on the Standard Views toolbar. (**Ctrl-7**

Step 12. Click **Features** on the Command Manager toolbar.

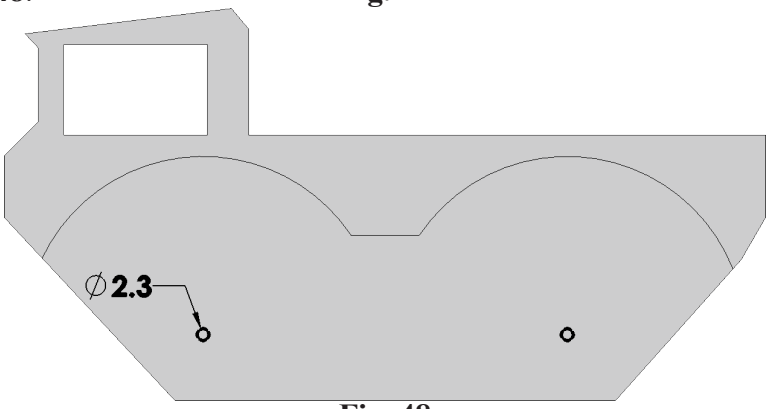


Fig. 48

Step 13. Click **Extruded Cut** on the Features toolbar.

Step 14. In the Cut-Extrude Property Manager set:
under Direction 1, **Fig. 49**
End Condition **Through All - Both**
click OK .

Step 15. Save (**Ctrl-S**).

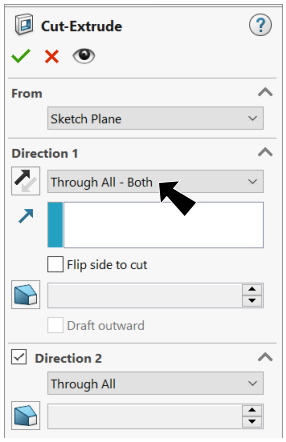


Fig. 49

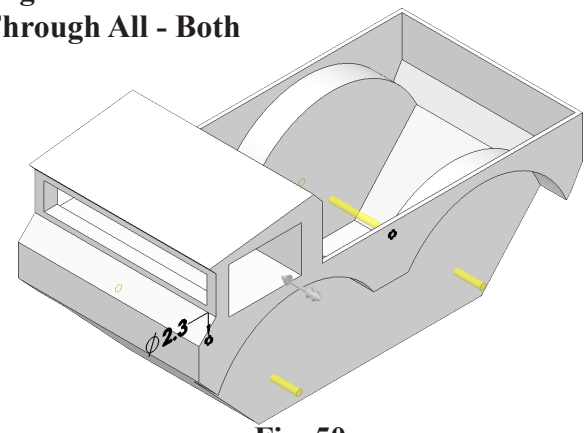


Fig. 50

K. Split2 Sketch7 Sunroof.

Step 1. Click **Right Plane** in the Feature Manager and click **Sketch** on the context toolbar, **Fig. 51**.

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

Step 3. Zoom in on **cabin**, **Fig. 52**.

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

Step 5. Sketch the **5 lines**, **Fig. 53**.

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

Step 7. Add dimensions, **Fig. 54**.

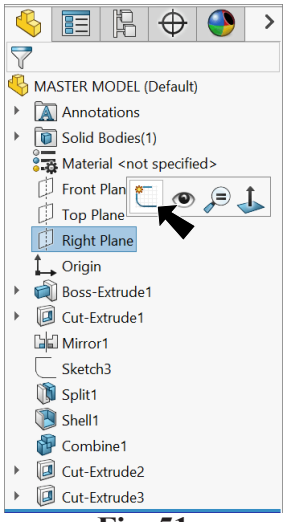


Fig. 51

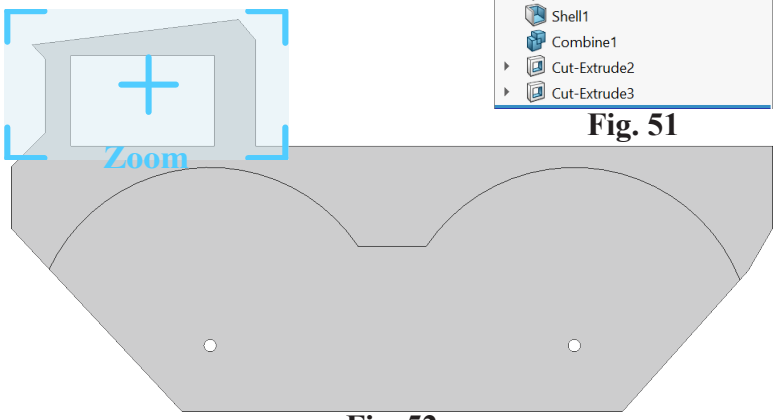


Fig. 52

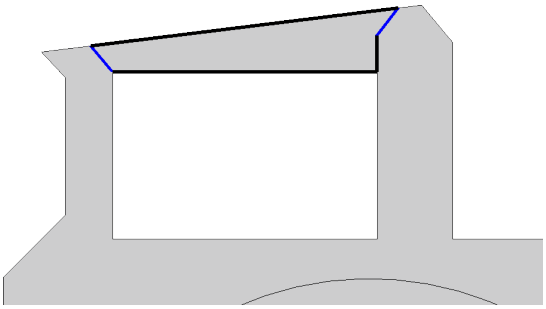


Fig. 53

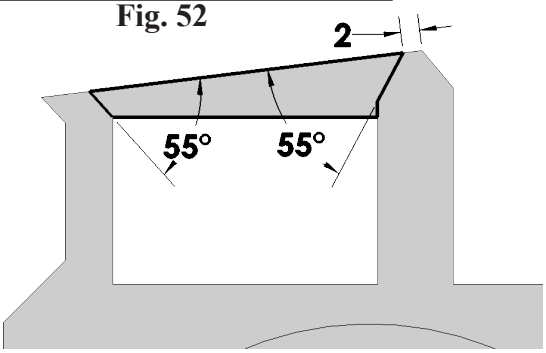


Fig. 54

Step 8. Click Insert Menu > Features > Split.

Step 9. In the Split Property Manager:
under Trim Tools, **Fig. 55**
Sketch7 was preselected
click **Cut Part** Cut Part button
under Resulting Bodies
click **Select All**
uncheck
Consume cut bodies
click OK .

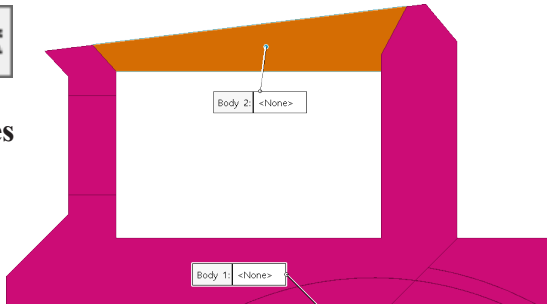


Fig. 56

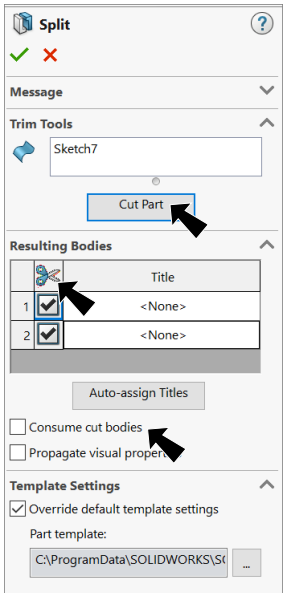


Fig. 55

L. Split3 Sketch8 Sunroof.

Step 1. **Hide Sketch7**. To hide, click **Sketch7** in the Feature Manager and **Hide** on the context toolbar, **Fig. 57**.

Step 2. Click **Top Plane** in the Feature Manager and click **Sketch** on the context toolbar, **Fig. 57**.

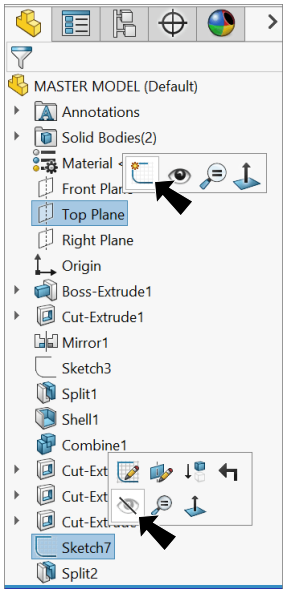


Fig. 57

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

Step 4. Click **Corner Rectangle** in the **Rectangle flyout** on the Sketch toolbar.

Step 5. Sketch **corner rectangle on top of cab coincident with front and rear edges of Split2**, **Fig. 58**.

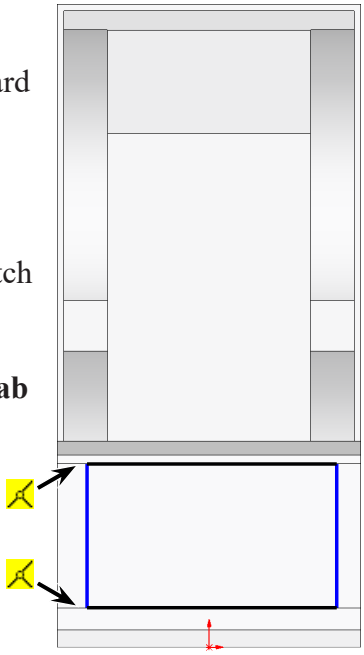


Fig. 58

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

Step 7. Add dimensions, **Fig. 59**.

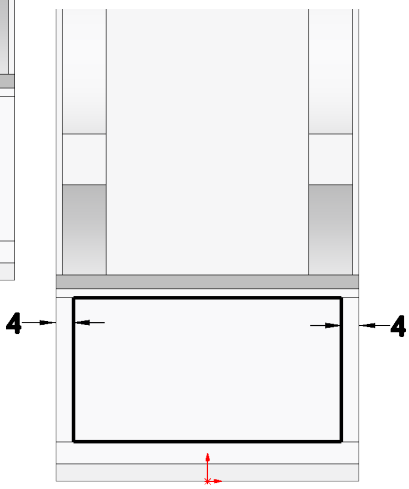


Fig. 59

Step 8. Click Insert Menu > Features > Split.

Step 9. In the Split Property Manager:
under Trim Tools, **Fig. 60**
Sketch8 was preselected
click **Cut Bodies** Cut Bodies
under Resulting Bodies
click **Select All**
uncheck **Consume cut bodies**
Note: It looks like **Body 6** is our Sunroof
click OK .

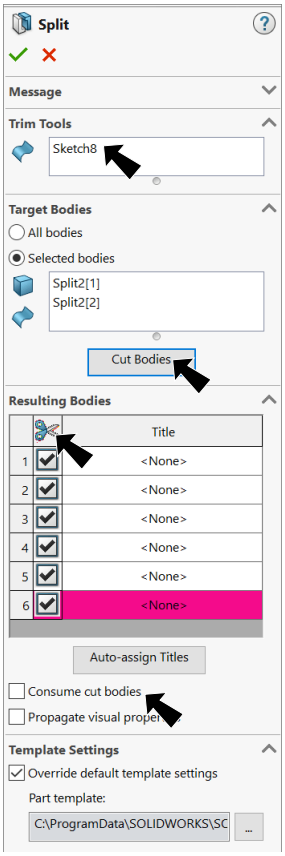


Fig. 60

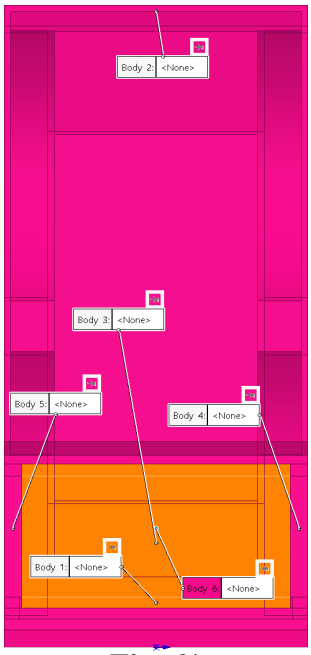


Fig. 61

M. Combine Bodies2.

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

Step 2. **Hide Sketch8**. To hide, click **Sketch8** in the Feature Manager and **Hide** on the context toolbar, **Fig. 62**.

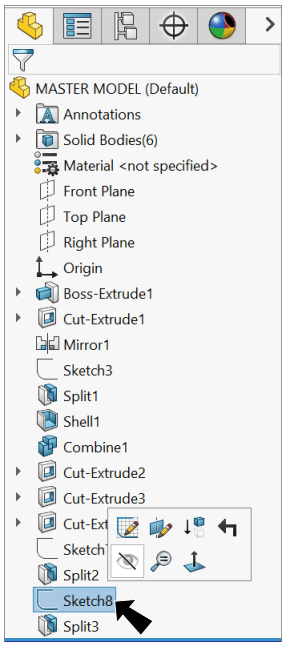


Fig. 62

Step 3. Select all bodies **except Sunroof body**. First, confirm which body is the Sunroof.
To confirm, expand **Solid Bodies** folder in the Feature Manager, **Fig. 63**.
Click Sunroof in the graphics area, **Fig. 64**.

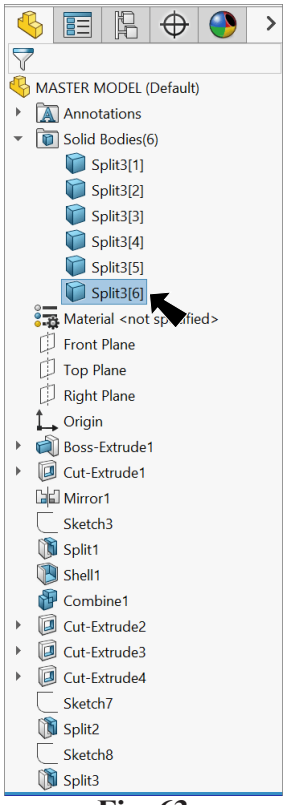


Fig. 63

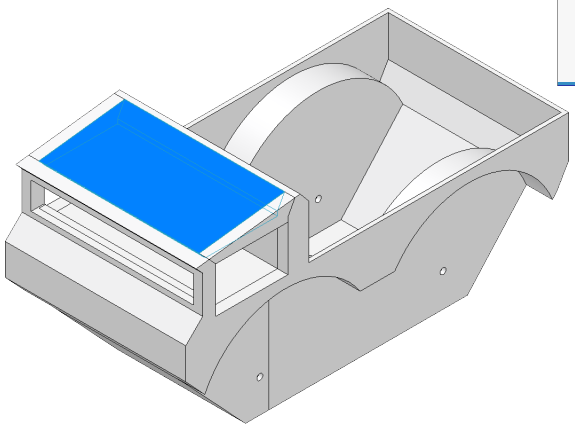


Fig. 64

Then, in the **Solid Bodies** folder, **Ctrl click all bodies except Sunroof body**, **Fig. 65**.

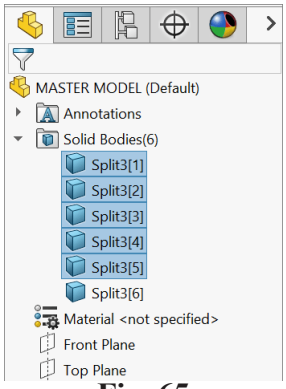


Fig. 65

Step 4. Click Insert Menu > Features > Combine.

Step 5. In the Combine Property Manager:
under Operation Type, **Fig. 66**
select **Add**
Bodies were preselected (not the Sunroof), **Fig. 67**
click OK .

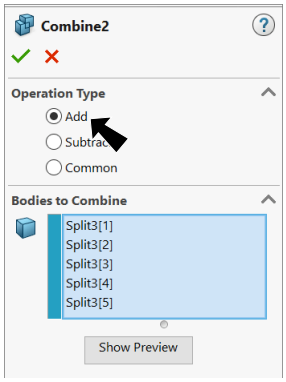


Fig. 66

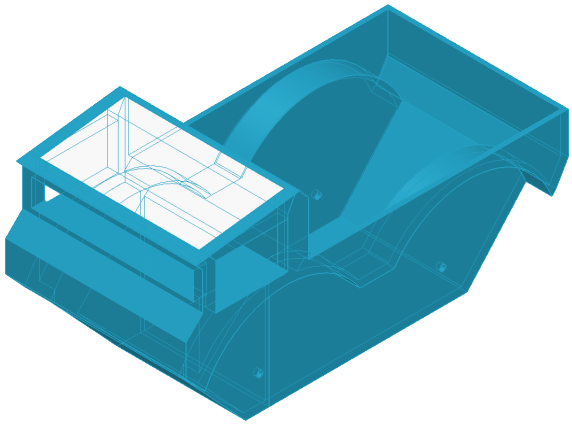


Fig. 67

N. Split4 Sketch9 for Hatch and Chassis Part.

Step 1. Click **Right Plane** in the Feature Manager and click **Sketch** on the context toolbar, **Fig. 68**.

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

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

Step 4. Sketch **horizontal line coincident with centerpoint of an Axle hole**, **Fig. 69**. To sketch line coincident, hover over edge of hole to wake up centerpoint. Do not click, but move cursor horizontally to maintain the relation and then click. Click the second endpoint. Drag out the endpoint beyond the body.

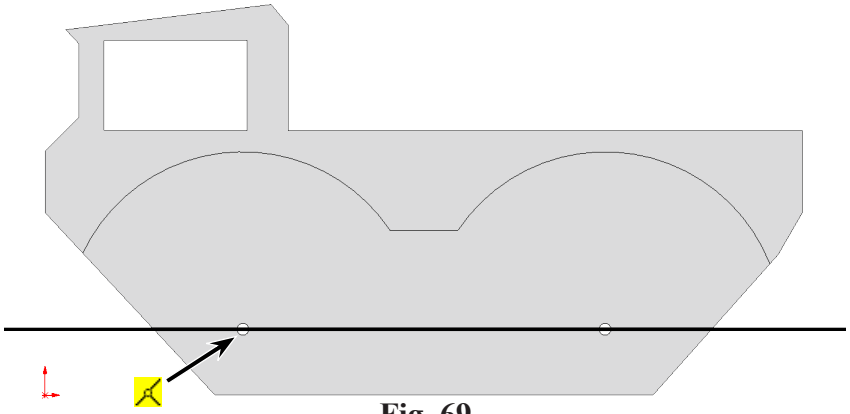


Fig. 69

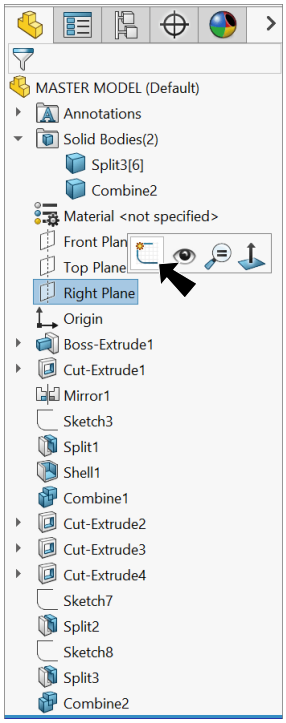


Fig. 68

Step 5. Click Insert Menu > Features > Split.

Step 6. In the Split Property Manager:
under Trim Tools, **Fig. 70**
Sketch9 was preselected
click **Cut Bodies** Cut Bodies button
under Resulting Bodies
click **Select All**
uncheck **Consume cut bodies**
click OK .

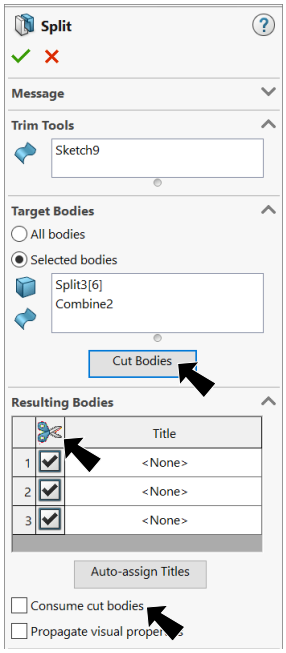


Fig. 70

Step 7. Save (**Ctrl-S**).

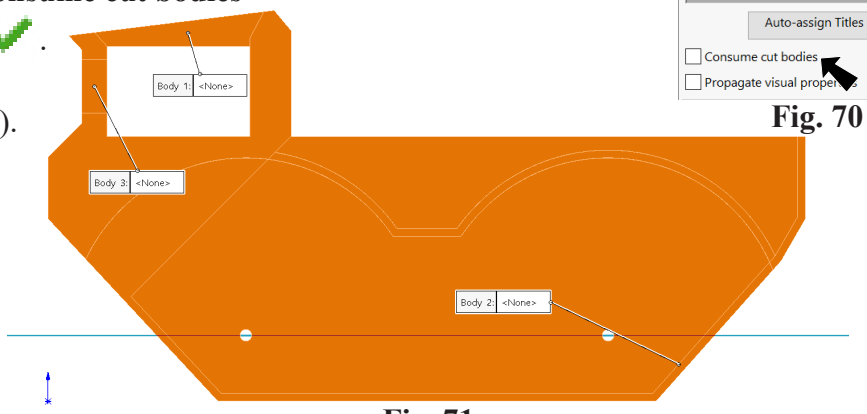


Fig. 71

O. Rename Bodies.

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

Step 2. **Hide Sketch9** . To hide, click **Sketch9** in the Feature Manager and **Hide** on the context toolbar, **Fig. 72**.

Step 3. **Rename bodies SUNROOF**, **CHASSIS and HATCH**, **Fig. 73**. To rename, expand **Solid Bodies** folder in the Feature Manager. Slowly click twice over Split4(1) and key-in **SUNROOF** or use the F2 key. Repeat and rename **CHASSIS** and **HATCH**.

Step 4. Save (**Ctrl-S**).

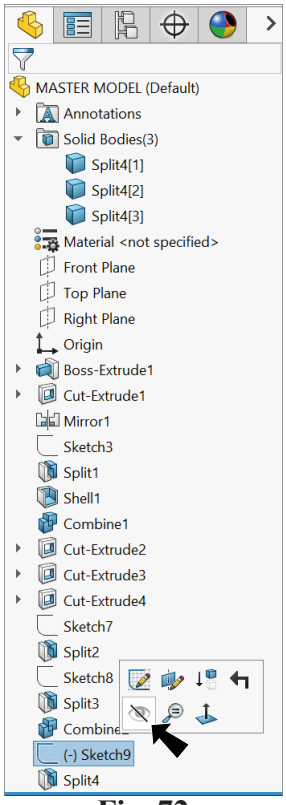


Fig. 72

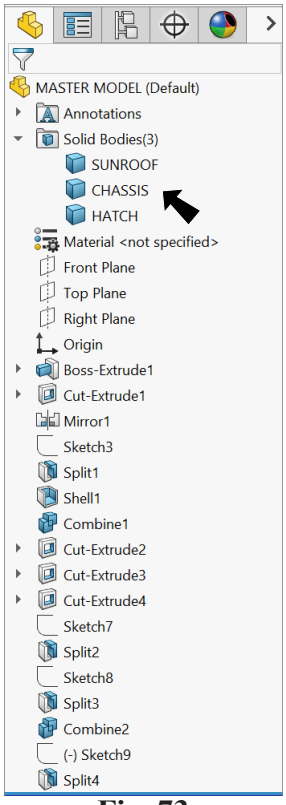


Fig. 73

P. Appearance: Green.

Step 1. Click part, click **Appearance Callout** on the context toolbar and click **MASTER MODEL**, **Fig. 74**.

Step 2. In the Appearances Task pane at the bottom select **CWV** and in the lower pane select **army green plastic**, **Fig. 75**.

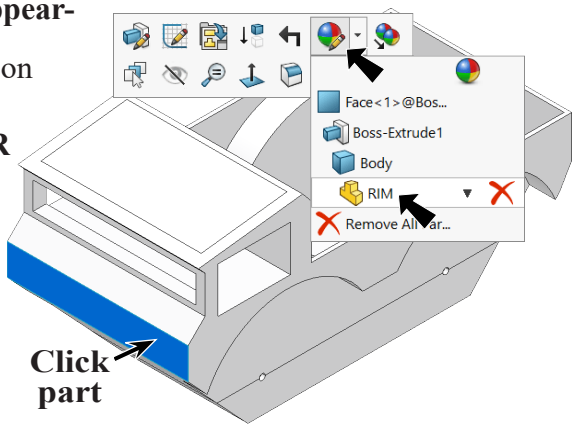


Fig. 74

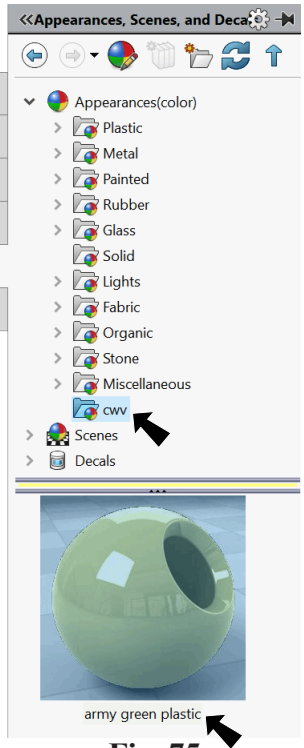


Fig. 75

Step 3. Click OK in Property Manager.

Step 4. Save (**Ctrl-S**).

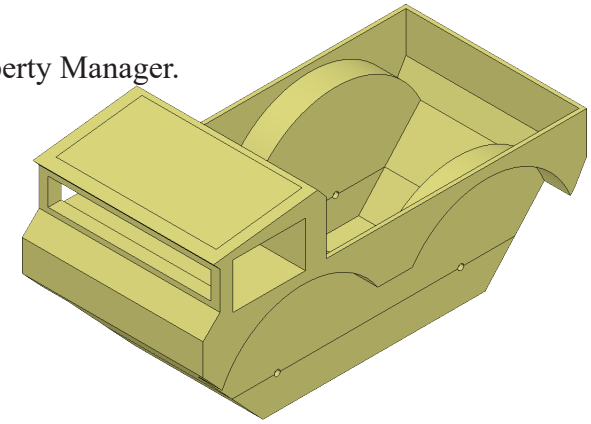


Fig. 76

Q. Save Bodies.

Step 1. Click Insert Menu > Features > Save Bodies.

Step 2. In the Save Bodies Property Manager:
under Create Assembly, **Fig. 77**
click **Browse** button
in the Save As dialog box, **Fig. 78**
key in **CWV ASSEMBLY** for file name
navigate to **My Documents/Tech Ed 24-25/CWV** folder
click **Save**
under Template Settings
set **Part and Assembly template preference to your Metric** templates
under Resulting Parts
click **Auto-assign Titles** Auto-assign Titles button
check **Propagate visual properties**
click OK .
SOLIDWORKS creates the **three Part files and the Assembly file**.

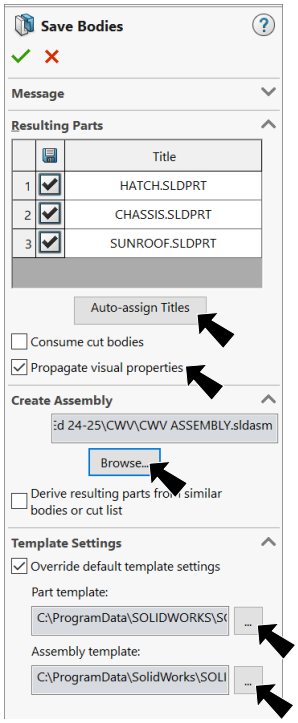


Fig. 77

Step 3. Save (**Ctrl-S**).

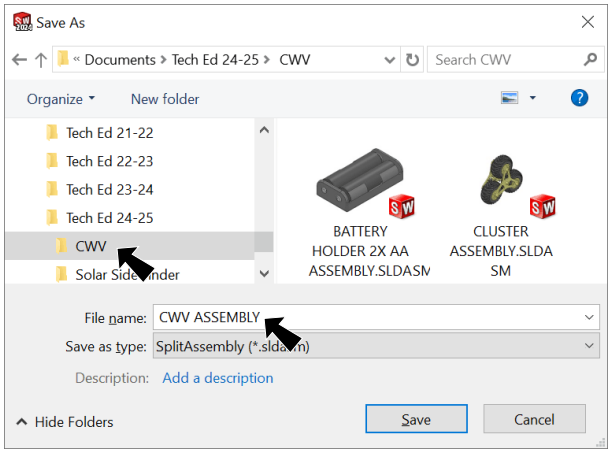


Fig. 78

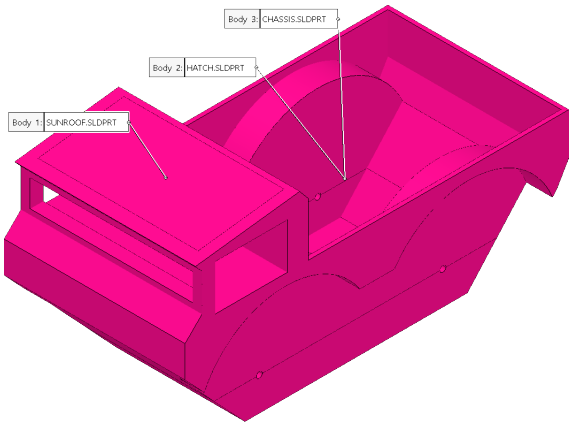


Fig. 79