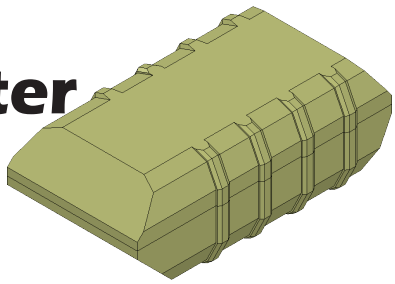
Chapter 33

Tank

Multibody Transmitter



A. Extrude1.

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**.

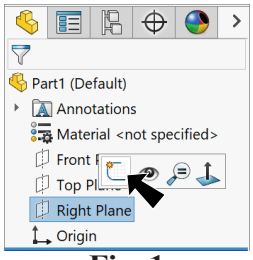


Fig. 1

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

Step 4. Sketch **horizontal midpoint line at Origin** and **vertical midpoint lines at both endpoints of horizontal line**, **Fig. 2**.

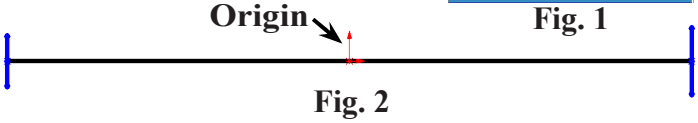


Fig. 2

Step 5. **Right click graphics area and click Select** from menu to unselect Line tool.

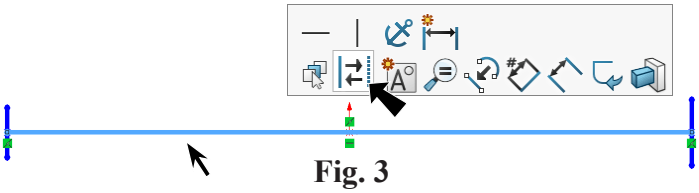


Fig. 3

Step 6. Click **horizontal line** and click **Construction Geometry** on the context toolbar, **Fig. 3**.

Step 7. **Ctrl click both vertical line** to select both. Release Ctrl key and click **Make Equal** on the context toolbar, **Fig. 4**.

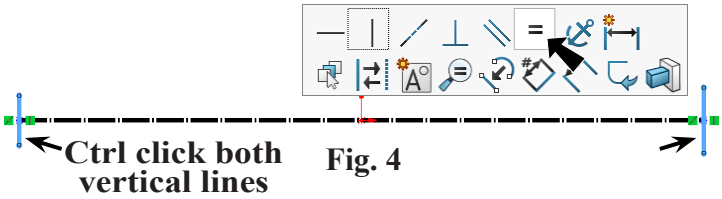


Fig. 4

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

Step 9. Sketch the **6 lines**, **Fig. 5**.

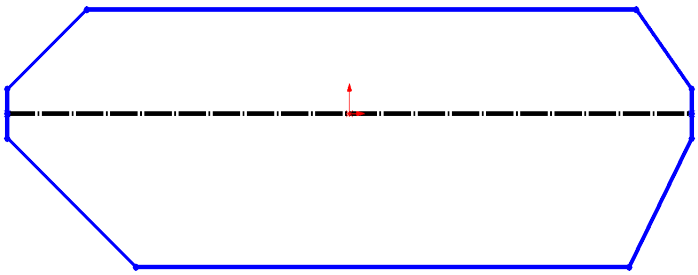


Fig. 5

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

Step 11. Add dimensions, **Fig. 6**. Dimension the angles last.

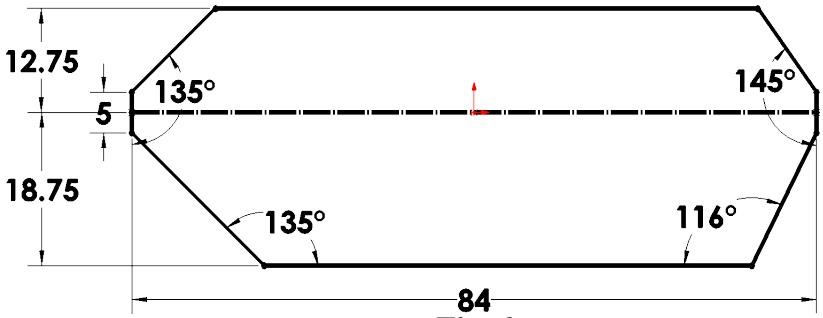


Fig. 6

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

Step 13. Click **Extruded Boss/Base** Extruded Boss/Base on the Features toolbar.

Step 14. In the Boss-Extrude Property Manager set:
under Direction 1, **Fig. 7**
End Condition **Mid Plane**
Depth D1 **43**
click OK .

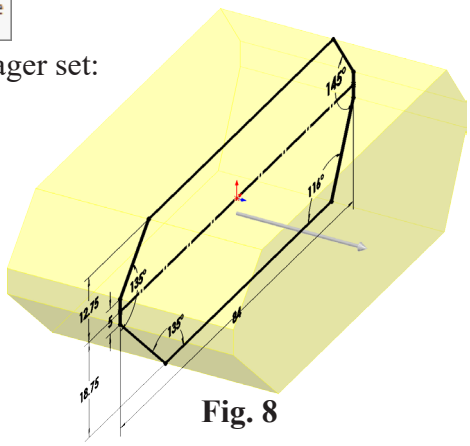


Fig. 8

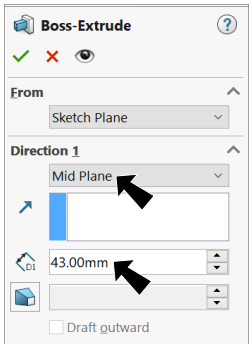


Fig. 7

B. Save as "MULTIBODY TRANSMITTER".

Step 1. Click File Menu > Save As.

Step 2. Key-in **MULTIBODY TRANSMITTER** for the filename and press ENTER.

C. Cut.

Step 1. Click **Front Plane** in the Feature Manager and click **Sketch** on the context toolbar, **Fig. 9**.

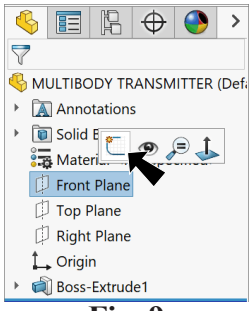


Fig. 9

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

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

Step 4. Sketch the **5 chained lines**, **Fig. 10**.

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

Step 6. Add dimensions, **Fig. 11**.

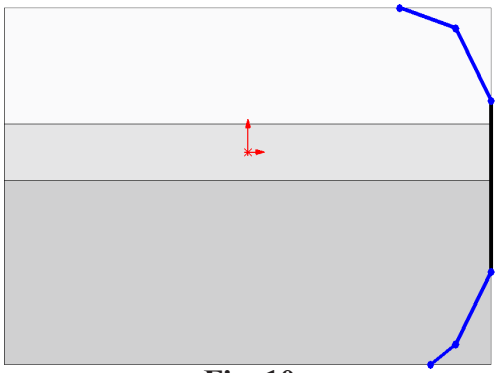


Fig. 10

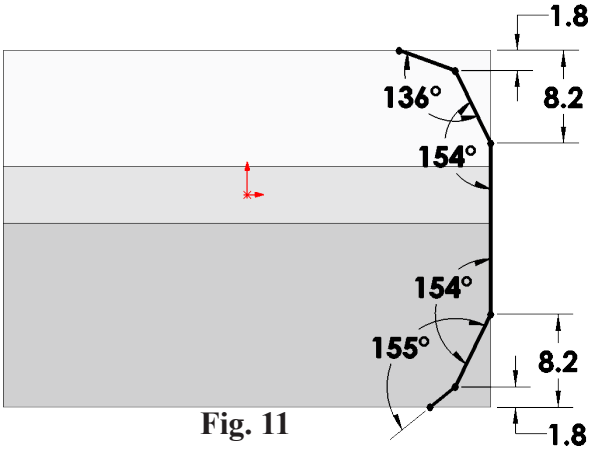


Fig. 11

Step 7. Click **Features** Features on the Command Manager toolbar.

Step 8. Click **Extruded Cut** Extruded Cut on the Features toolbar.

Step 9. In the Cut-Extrude Property Manager set:
under Direction 1, **Fig. 12**
End Condition **Through All - Both**
The **Direction arrow** should point towards area to be cut away (outside), **Fig. 13**. If arrow is pointing in wrong direction, uncheck Flip side to cut.

Click OK ✔.

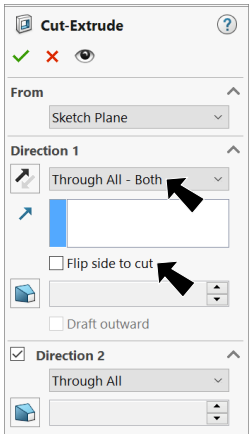


Fig. 12

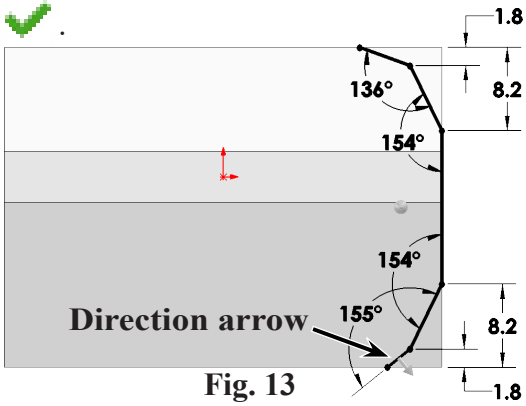


Fig. 13

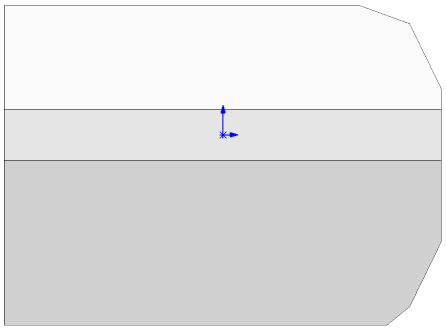


Fig. 14

D. Mirror1 Cut.

Step 1. Click **Trimetric** on the Standard Views toolbar.

Step 2. **Ctrl click Right Plane** and **Cut-Extrude1** in Feature Manager to select plane and feature, **Fig. 15**.

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

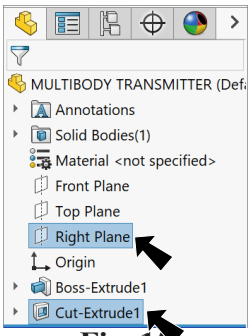


Fig. 15

Step 4. In the Mirror Property Manager click OK ✔, **Fig. 16**.

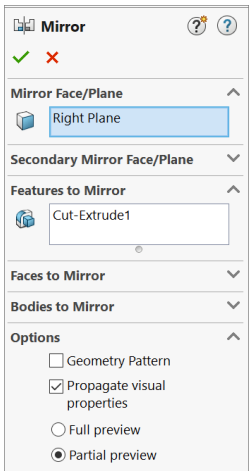


Fig. 16

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

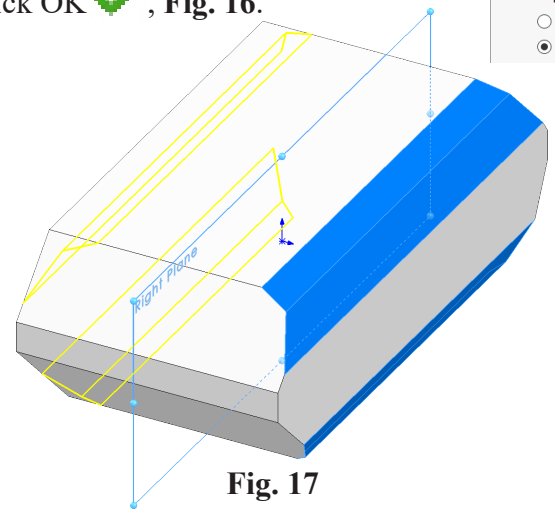
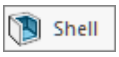


Fig. 17

E. Shell.

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

Step 2. In the Shell Property Manager set:
under Parameters, **Fig. 17**
Distance 1.5
click OK .

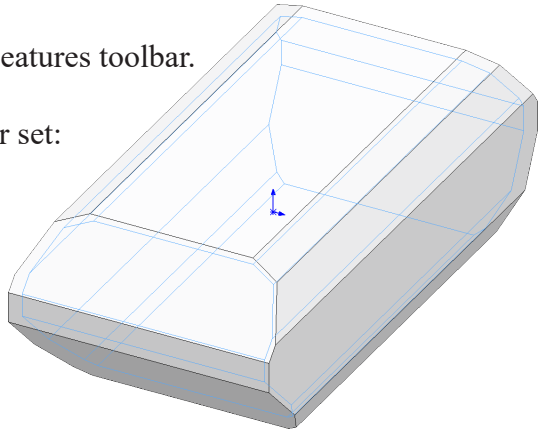


Fig. 19

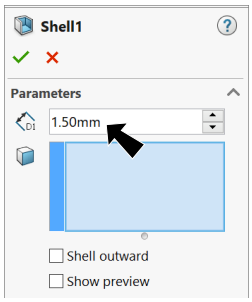


Fig. 18

F. Extrude2.

Step 1. Click **Front Plane** in the Feature Manager and click **Sketch** on the context toolbar, **Fig. 20**.

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

Step 3. Click **Convert Entities** on the Sketch toolbar.

Step 4. In the Convert Entities Property Manager:
under Entities to Convert, **Fig. 21**
click **edges on right side** (7), **Fig. 22**
click OK .

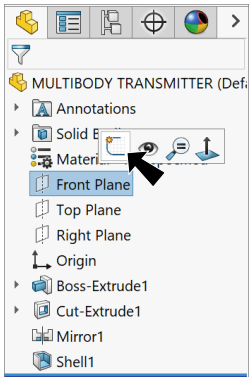


Fig. 20

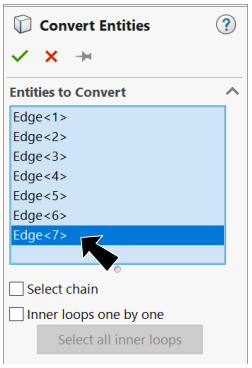


Fig. 21

Step 5. Click **Offset Entities** on the Sketch toolbar.

Step 6. In the Offset Entities Property Manager set:
under Parameters, **Fig. 23**
Distance 1
check **Reverse**
check **Select chain**
click **any geometry**, **Fig. 24**
offset should be on **outside**
click OK .

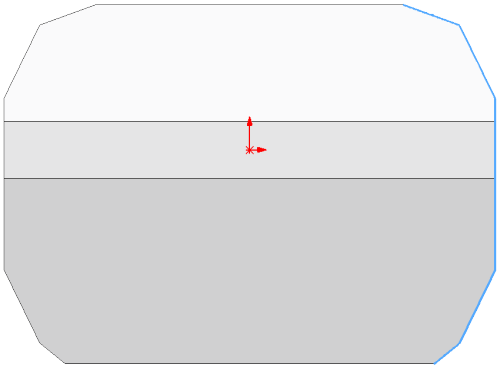


Fig. 22

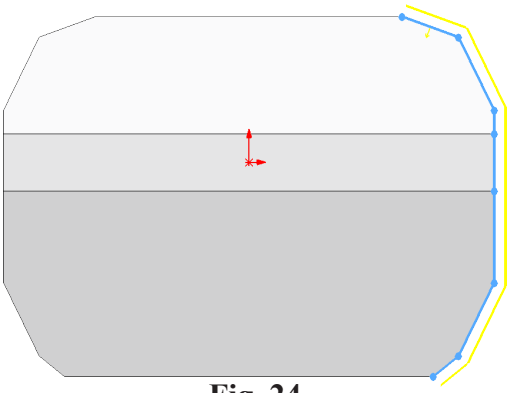


Fig. 24

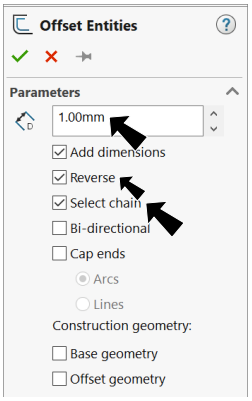


Fig. 23

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

Step 8. Sketch **horizontal line out from both end-points of converted edge lines**, **Fig. 25**.

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

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

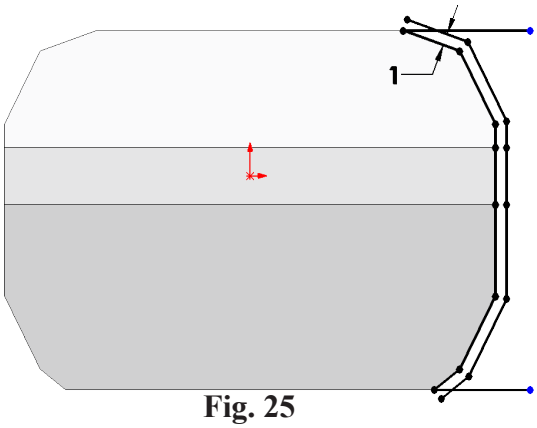
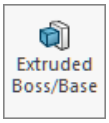


Fig. 25

Step 11. Click **Extruded Boss/Base**  on the Features toolbar.

Step 12. In the Boss-Extrude Property Manager set:
under Direction 1, **Fig. 26**
End Condition **Mid Plane**
Depth **4**
under Selected Contours
click the **contour**, **Fig. 27**
uncheck Thin Feature
click OK .

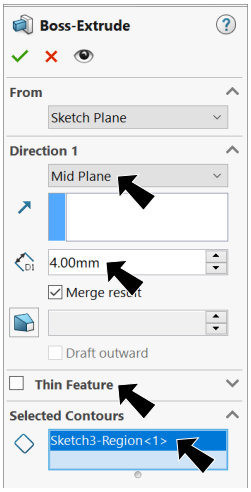


Fig. 26

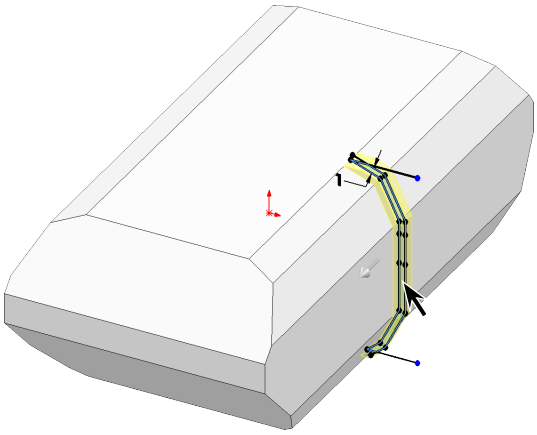


Fig. 27

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

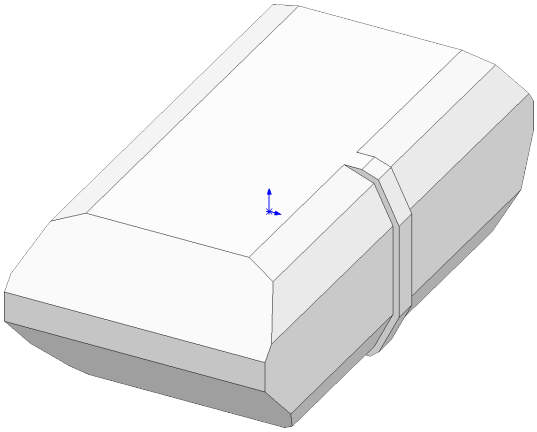


Fig. 28

G. Chamfer1.

Step 1. Rotate view to **view all edges of Exturde2, Fig. 30**. Use **Right** on the Standard Views toolbar (**Ctrl-4**), then **Right Arrow** **key once and Down Arrow** **key once**.

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

Step 3. In the Chamfer Property Manager set:

under Chamfer Type, **Fig. 29**

select **Angle Distance**

click **front angled edges along Extrude2 (5) - not horizontal edges, Fig. 30**

under Chamfer Parameters

Distance **1**

Angle **30º**

The Direction arrow should point to **rear.**

If arrow is pointing in wrong direction, click **arrow**.

Click OK .

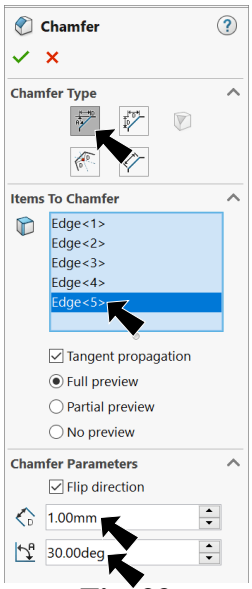


Fig. 29

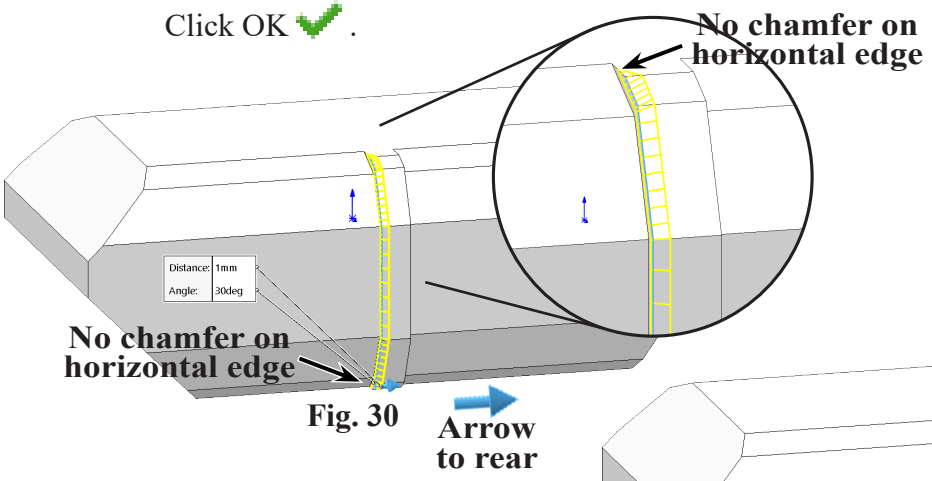


Fig. 30

Arrow to rear

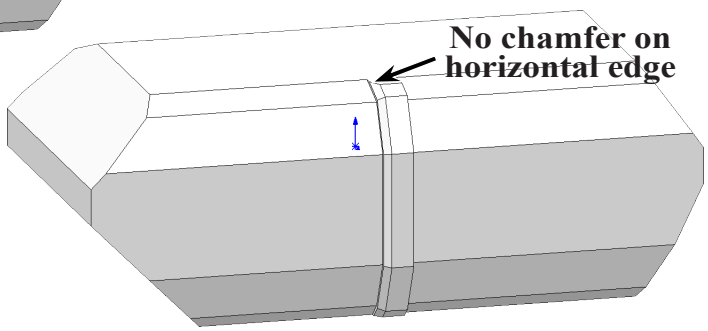


Fig. 31

H. Chamfer2.

Step 1. Click **Chamfer**

on the Features toolbar.

Step 2. In the Chamfer Property Manager set:

Tip: under Chamfer Options, **Fig. 32**
uncheck **Select through faces**
under Chamfer Type
select **Angle Distance**
click **rear angled edges along Extrude2 (5), Fig. 33**
under Chamfer Parameters
Distance **1**
Angle **30º**
check **Flip direction**.
The Direction arrow should point to **front.**
If arrow is pointing in wrong direction, check **arrow**.
Click OK .

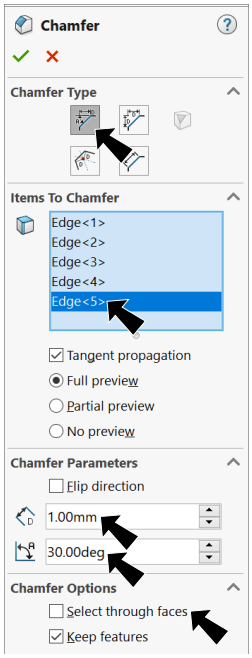


Fig. 32

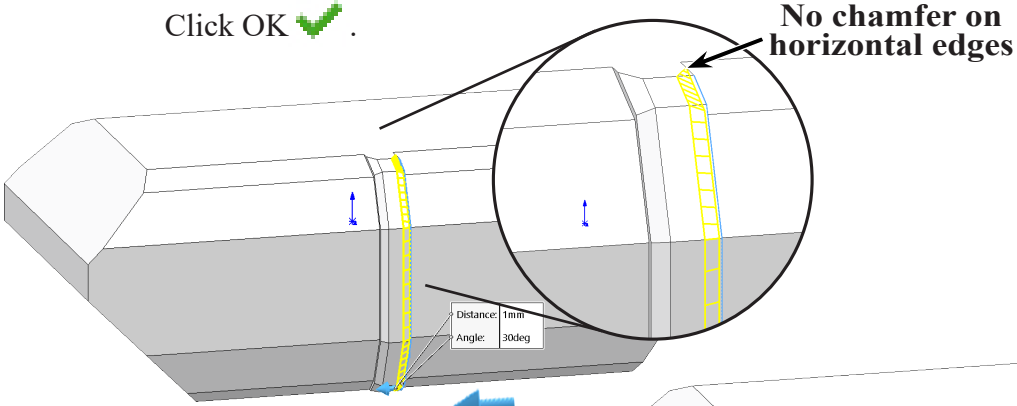


Fig. 33

Arrow to front

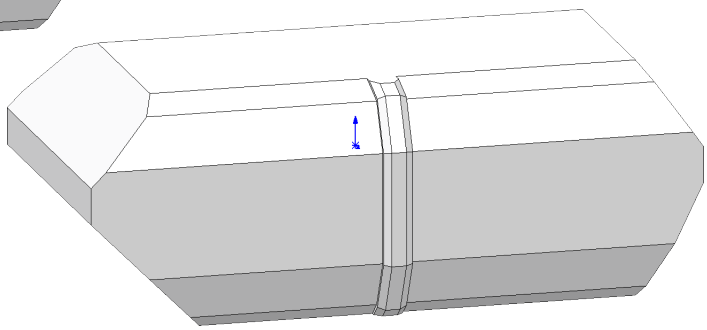


Fig. 34

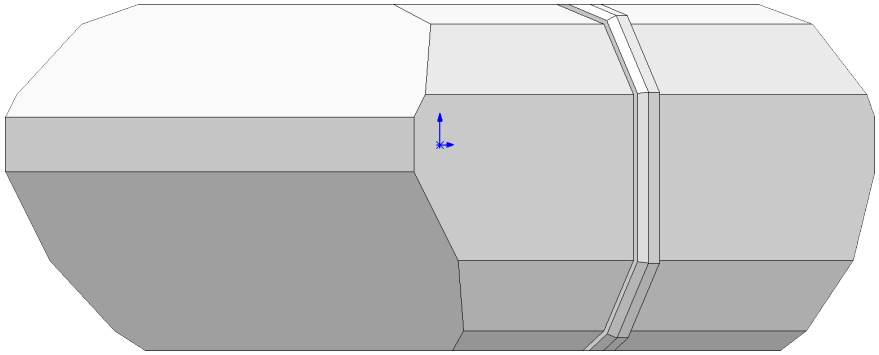


Fig. 35

I. Linear Pattern.

Step 1. Click **Linear Pattern**

on the Features toolbar.

Step 2. In the Linear Pattern Property Manager set:
under Direction 1, **Fig. 36**
click a **horizontal edge**, **Fig. 37**
Spacing **14**
Number of Instances **3**
under Direction 2
click **edge again**, **Fig. 12**
Reverse Direction
Spacing **14**
Number of Instances **2**
check **Pattern seed only**
under Features and Faces
click **Boss-Extrude2 and both Chamfers**
click OK ✔.

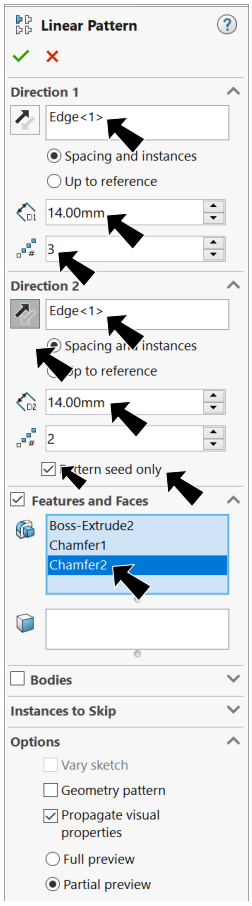


Fig. 36

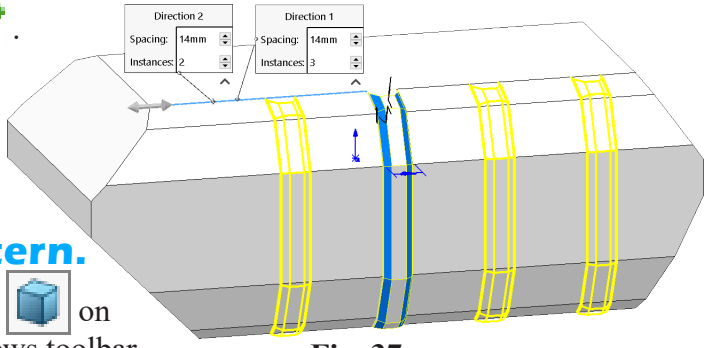


Fig. 37

J. Mirror2 LPattern.

Step 1. Click **Trimetric** on the Standard Views toolbar.

Step 2. **Ctrl click Right Plane**, **Boss-Extrude2, both Chamfers** and **LPattern1** in Feature Manager to select plane and features, **Fig. 38**.

Step 3. Click **Mirror**

on the Features toolbar.

Step 4. In the Mirror Property Manager click OK ✔, **Fig. 39**.

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

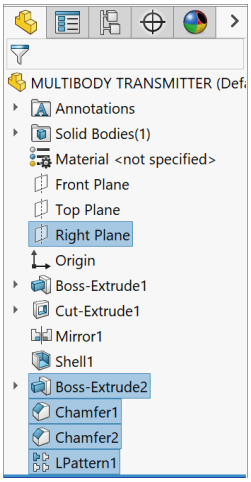


Fig. 38

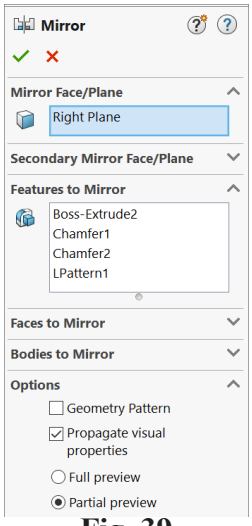


Fig. 39

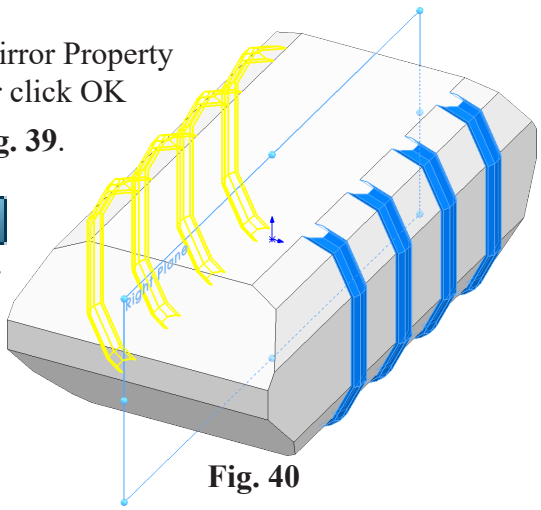


Fig. 40

K. Split.

Step 1. Click **Top Plane** in the Feature Manager to select plane, **Fig. 41**.

Step 2. Click Insert Menu > Features > Split.

Step 3. In the Split Property Manager:
under Trim Tools, **Fig. 42**
Top Plane was preselected
click **Cut Part** button
under Resulting Bodies
click **Select All**
uncheck **Consume cut bodies**
click OK .

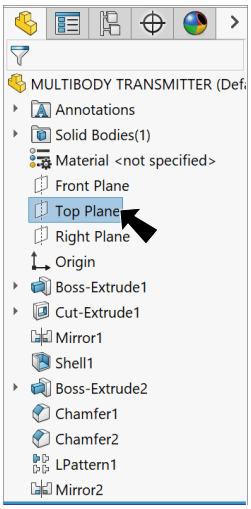


Fig. 41

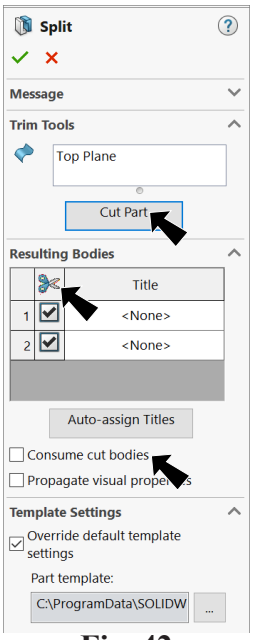


Fig. 42

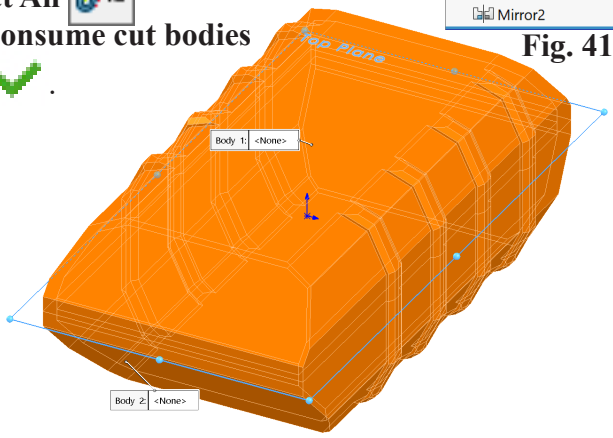


Fig. 43

L. Rename Bodies.

Step 1. **Rename bodies TOP TRANSMITTER and BOTTOM TRANSMITTER, Fig. 44**. To rename, expand **Solid Bodies** folder in the Feature Manager. Slowly click twice over Split1 and key-in **TOP TRANSMITTER** or use the F2 key. Repeat and rename **BOTTOM TRANSMITTER**.

Step 2. Save (**Ctrl-S**).

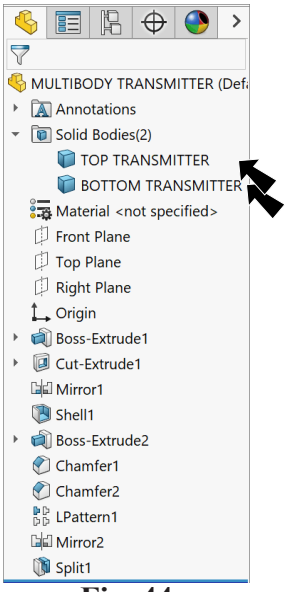


Fig. 44

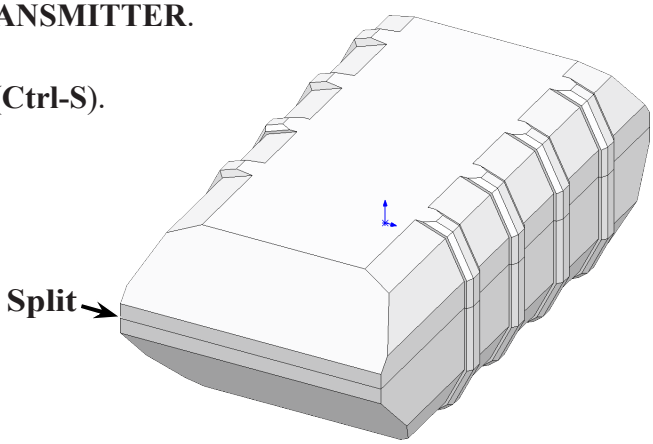


Fig. 45

M. Lip Groove.

Step 1. Click Insert Menu > Fastening Feature > Lip / Groove.

Step 2. In the Lip/Groove Property Manager:
under Body/Part Selection, **Fig. 46**

for Select body part/component on which to create groove
click **Top body**, **Fig. 47**

for Select body part/component on which to create lip
click **Bottom body**

for Select a plane to define direction of lip/groove
Expand the flyout Feature Manager design tree (click ▸) in the top left corner of the graphics area and click **Top Plane**, **Fig. 48**
under Groove Selection

Select face on which to create groove
Rotate Body to underside, **Fig. 49**
click **bottom horizontal face of Top at split**

Select inter or outer edge for groove remove material
click **inside edges of Top** (4) or **right click edge** and select **Loop**
under Lip Selection

Select face on which to create lip

Rotate view to **Previous View** (**Ctrl-Shift-Z**), **Fig. 50**
click **top horizontal face of Bottom at split**

Select inter outer edge for lip add material
click **inside edges of Bottom** (4)
under Parameters

- **A 1** Groove width
- **B .3** Spacing between lip and groove
- **C 3** Groove draft angle
- **D 0** Upper gap between lip and groove
- **E 1.9** Lip height
- **F .7** Lip width
- **H 0** Gap between lip and groove

click OK .

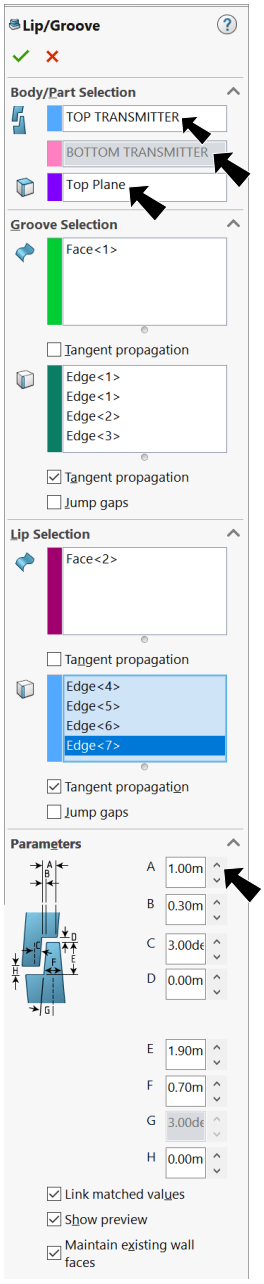


Fig. 46

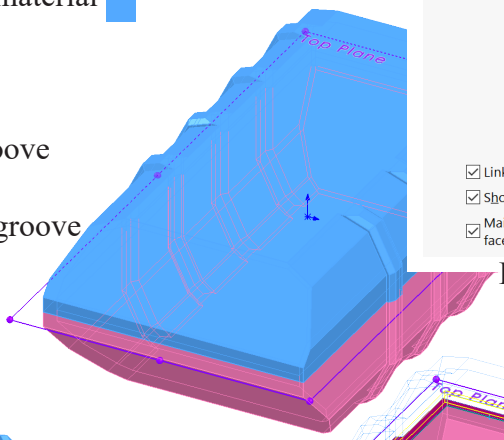


Fig. 47

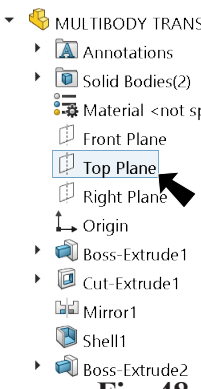


Fig. 48

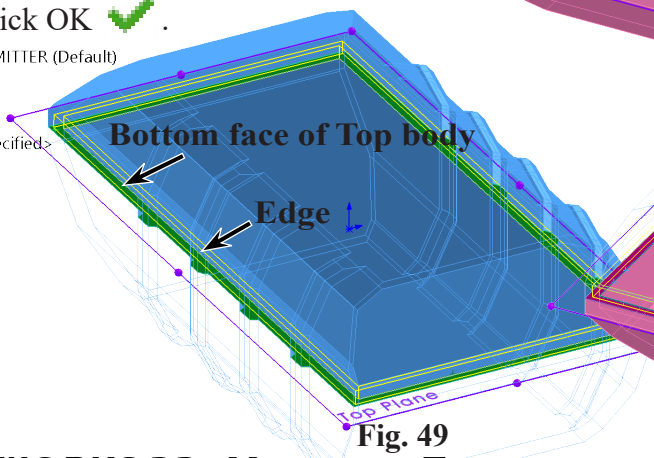


Fig. 49

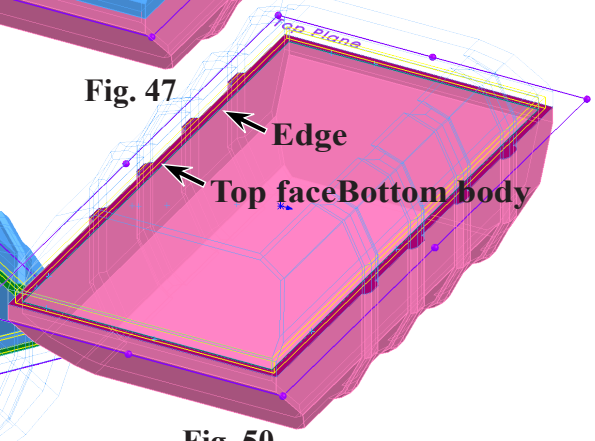


Fig. 50

N. Snap Hook 3D Sketch.

Step 1. **Hide Top** body. To hide, position cursor over the body in graphics area and press **Tab** key, **Fig. 51**. **Shift - Tab** to show.

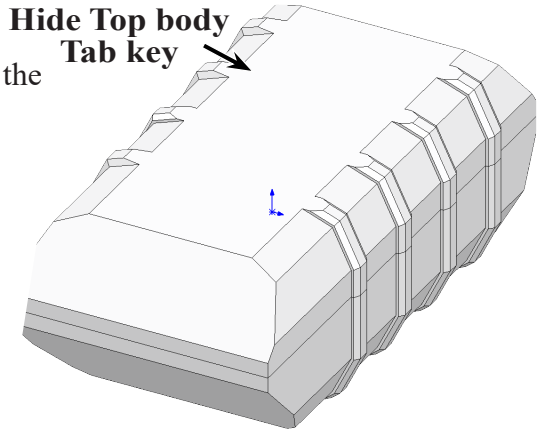


Fig. 51

Step 2. Click **Sketch** on the Command Manager toolbar.

Step 3. Click **3D Sketch** in the **Sketch fly-out** on the Sketch toolbar.

Step 4. Click **Point** **(S)** on the Sketch toolbar.

Step 5. Sketch a **point on inside starboard vertical face of Bottom body**, **Fig. 52**.

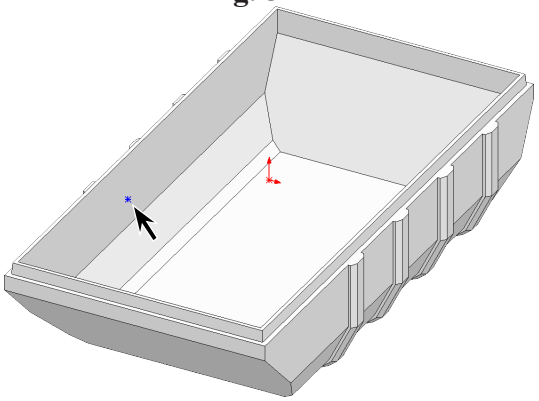


Fig. 52

Step 6. **Right click graphics area and click Select** from menu to unselect Point tool.

Step 7. **Ctrl click Front Plane** in the Feature Manager, **Fig. 53** and the **point** in graphic area to select both. Release Ctrl key and click **Make On Plane** on the context toolbar, **Fig. 54**.

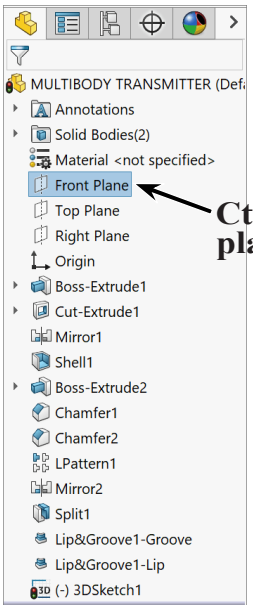


Fig. 53

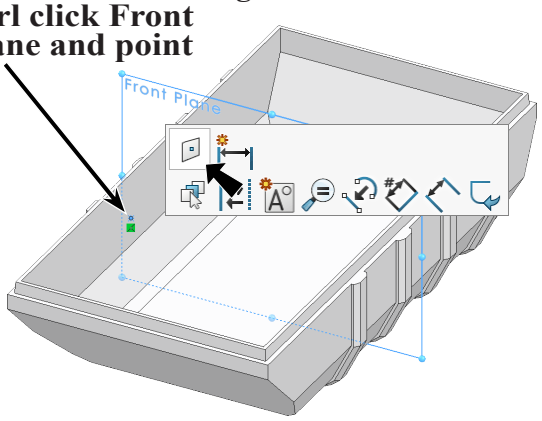


Fig. 54

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

Step 9. Dimension Point **16.3 on Y axis** to Origin , **Fig. 55**.

To dimension, click Origin and Point, then use the **Tab key** to change the sketch plane to Y and click. Key-in 16.3.

Step 10. Save **(Ctrl-S)**.

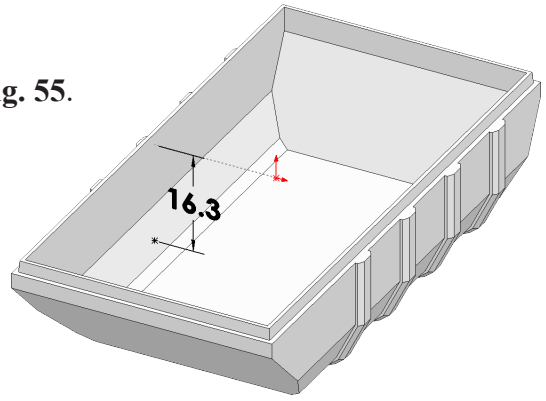


Fig. 55

O. Snap Hook.

Step 1. Click Insert Menu > Fastening Feature > Snap Hook.

Step 2. In the Snap Hook Property Manager: under Snap Hook Selections, **Fig. 56**

for Select a position click **3D Point**, **Fig. 57**

for Define vertical direction Expand flyout Feature Manager design tree and click **Top Plane**, **Fig. 58**

for Define the direction of the hook click **Right Plane** check **Reverse direction**

under Snap Hook Data

A	**2.5**	Depth at top of hook
B	**1.6**	Hook height
C	**0**	Hook lip height
D	**19.5**	Body height
E	**3.76**	Depth at base of hook
F	**1.4**	Hook overhang
G	**3.5**	Total width
H	**6**	Top draft angle

click OK .

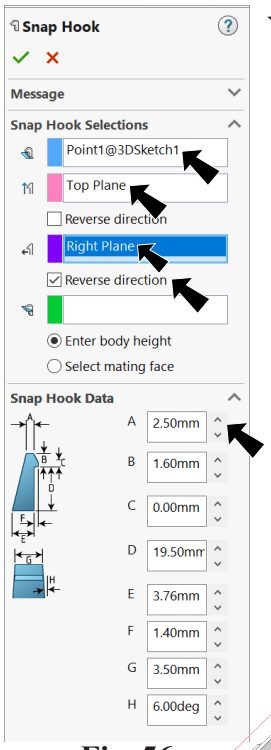


Fig. 56

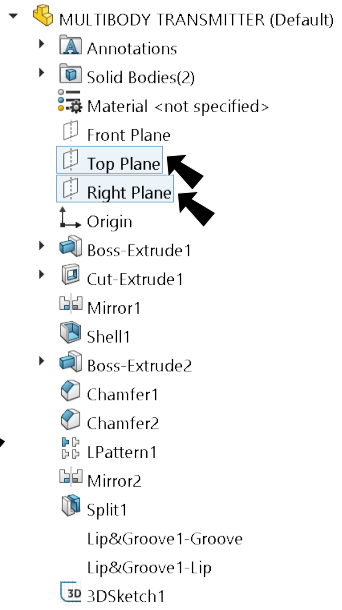


Fig. 58

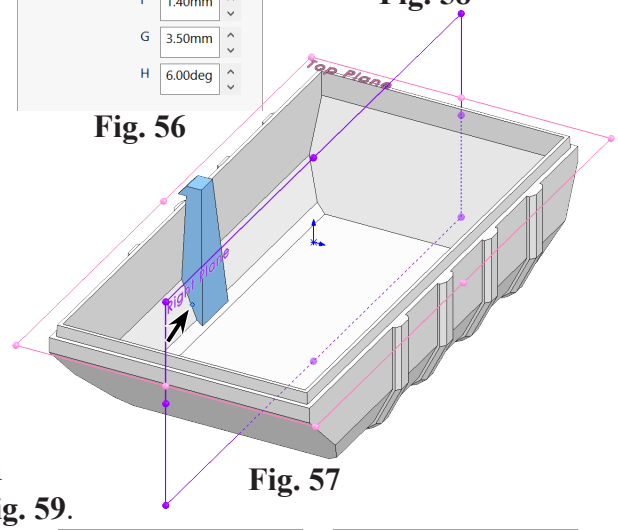


Fig. 57

P. Mirror3 Snap Hook.

Step 1. **Ctrl click Right Plane** and **Snap Hook1** in Feature Manager to select plane and feature, **Fig. 59**.

Step 2. Click **Features** Features on the Command Manager toolbar.

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

Step 4. In the Mirror Property Manager click OK , **Fig. 60**.

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

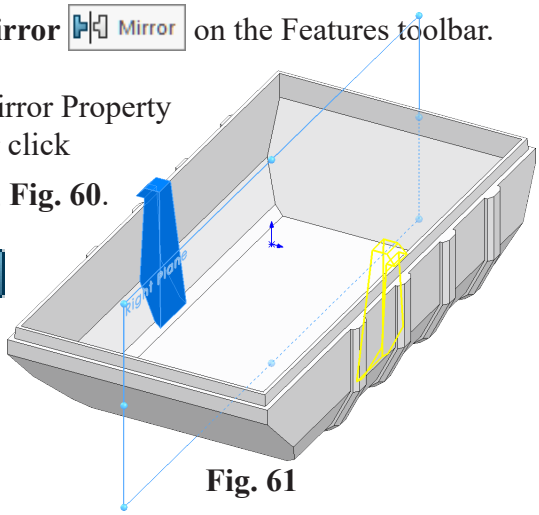


Fig. 61

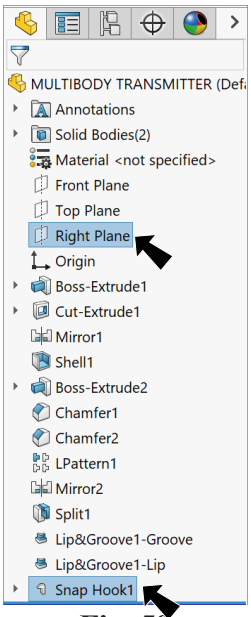


Fig. 59

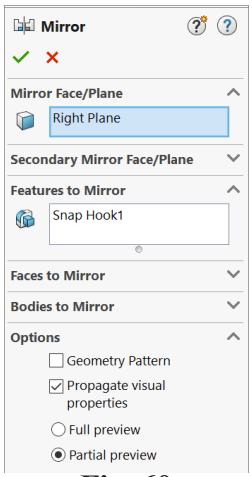


Fig. 60

Q. Snap Groove.

Step 1. **Show Top** body. To show, position cursor over where the body would be in graphics area and press **Shift - Tab** key, **Fig. 62**.

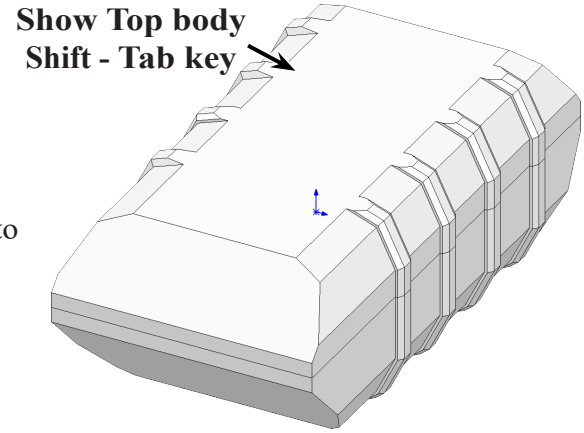


Fig. 62

Step 2. Click **Snap Hook1** in the Feature Manager to select feature, **Fig. 63**.

Step 3. Click Insert Menu > Fastening Feature > Snap Hook Groove.

Step 4. In the Snap Hook Groove Property Manager: under Feature and Body Selections, **Fig. 64**

for Select snap hook feature
Snap Hook1 was preselected

click in Select a body box
click **Top body**, **Fig. 65**

A	**0**	Offset height from snap hook
B	**-**	Hook height
C	**2**	Gap height
D	**0**	Groove clearance
E	**0**	Gap clearance
F	**0**	Offset width from snap hook

Use **Up Arrow key** to rotate view

click OK .

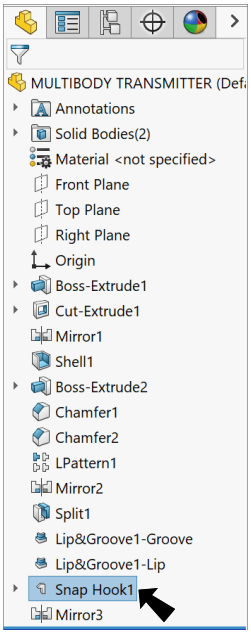


Fig. 63

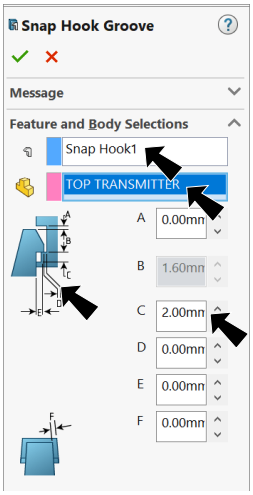


Fig. 64

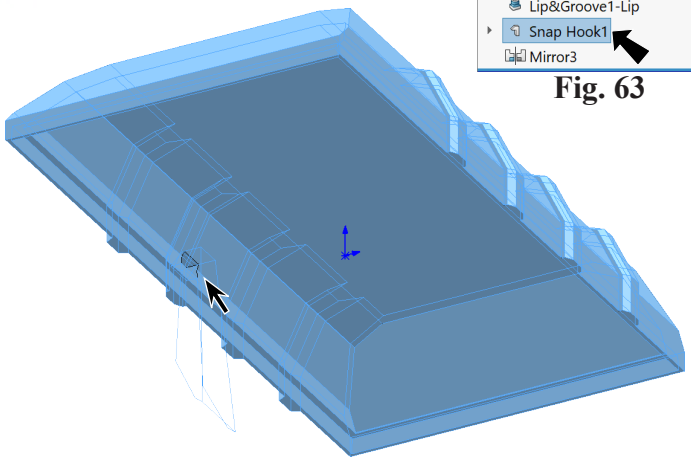


Fig. 65

R. Mirror4 Snap Groove.

Step 1. Click **Trimetric** on the Standard Views toolbar.

Step 2. **Ctrl click Right Plane** and **Snap Groove1** in Feature Manager to select plane and feature, **Fig. 66**.

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

Step 4. In the Mirror Property Manager click OK, **Fig. 67**.

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

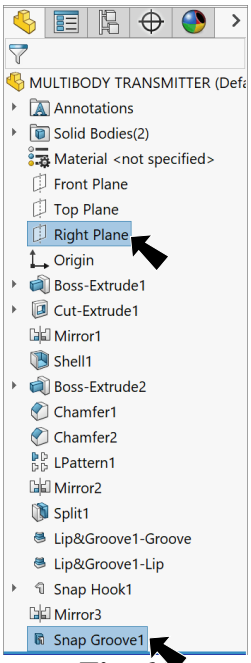


Fig. 66

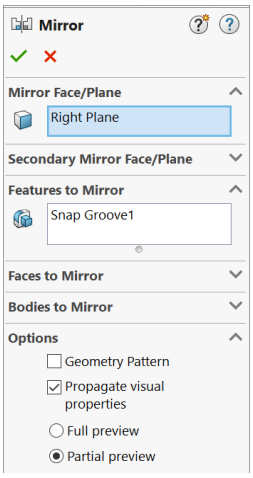


Fig. 67

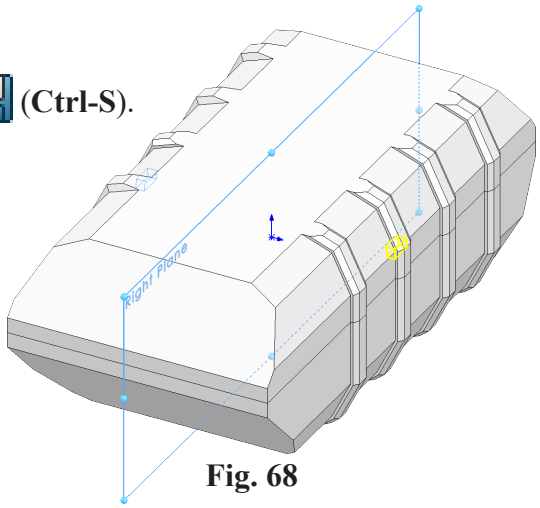


Fig. 68

S. Appearance: Green.

Step 1. Click part, click **Appearance Callout** on the context toolbar and click **MULITIB..**, **Fig. 69**.

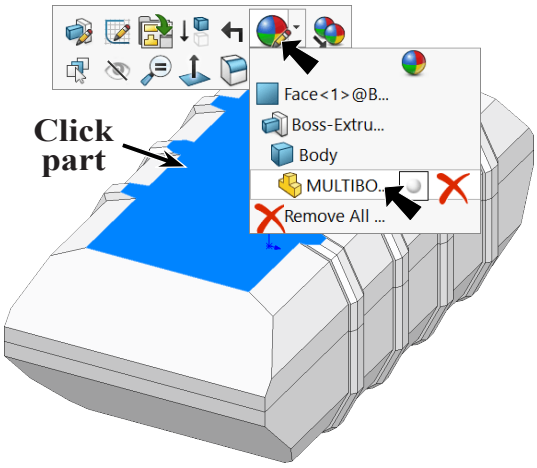


Fig. 69

Step 2. In the Appearances Task pane at the bottom select **tank** and in the lower pane select **army green high gloss plastic**, **Fig. 70**.

Step 3. Click **OK** in Property Manager.

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

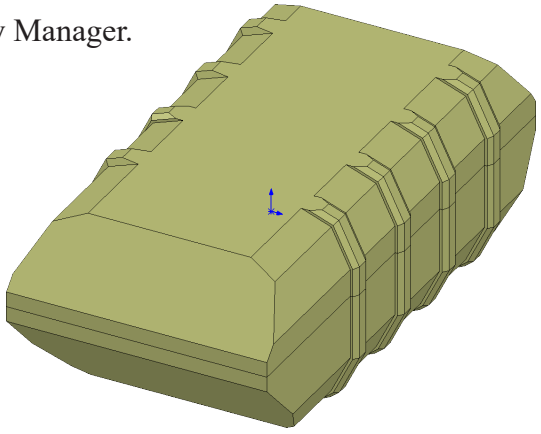


Fig. 71

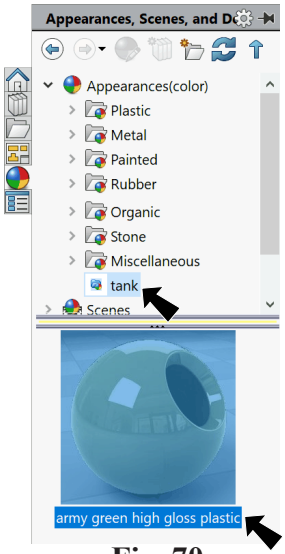


Fig. 70

T. Save Bodies.

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

Step 2. In the Save Bodies Property Manager:
under Create Assembly, **Fig. 72**
click **Browse** button
in the Save As dialog box:
key in **TRANSMITTER ASSEMBLY** for file name
navigate to **My Documents/Tech Ed 21-22/Tank** folder
click **Save**, **Fig. 73**
under Template Settings
set **Part and Assembly template preference to your Metric templates**
under Resulting Parts
click **Auto-assign Names** button
check **Propagate visual properties**
click OK .

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

Tip: When saving Assembly if you get an error message "The above file name is invalid" open up your template and save using the newer version of SOLIDWORKS.

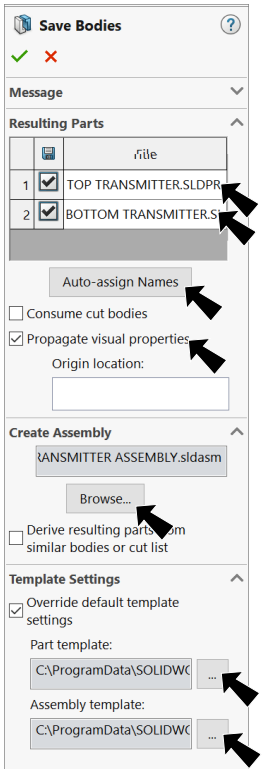


Fig. 72

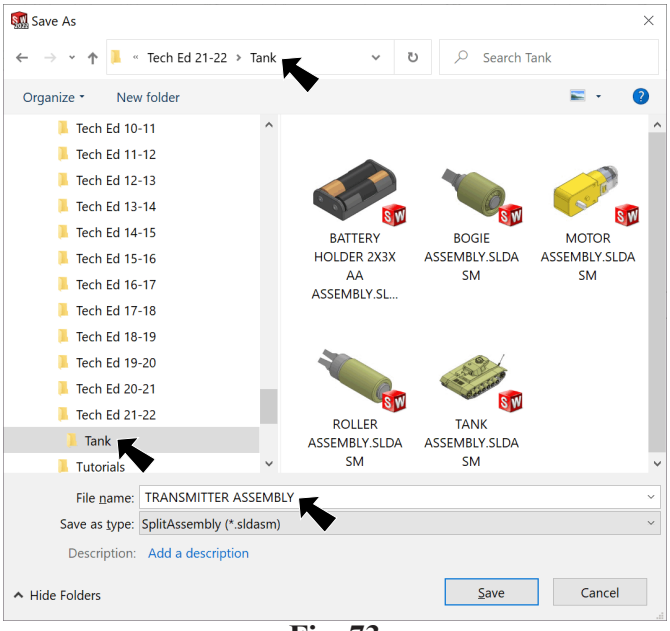


Fig. 73

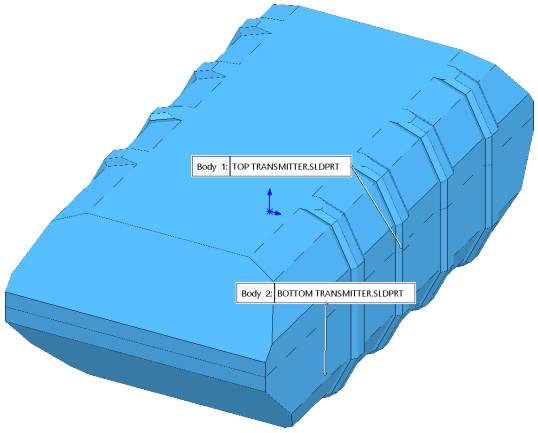


Fig. 74