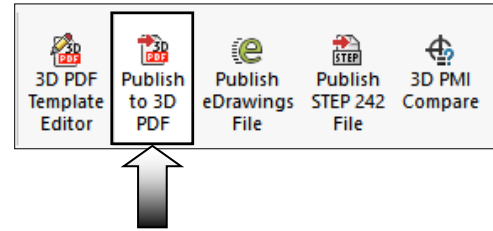


Publishing PDF and eDrawing Files

(Use the same part document from the previous lesson for this exercise.)

1. Publishing to 3D PDF:

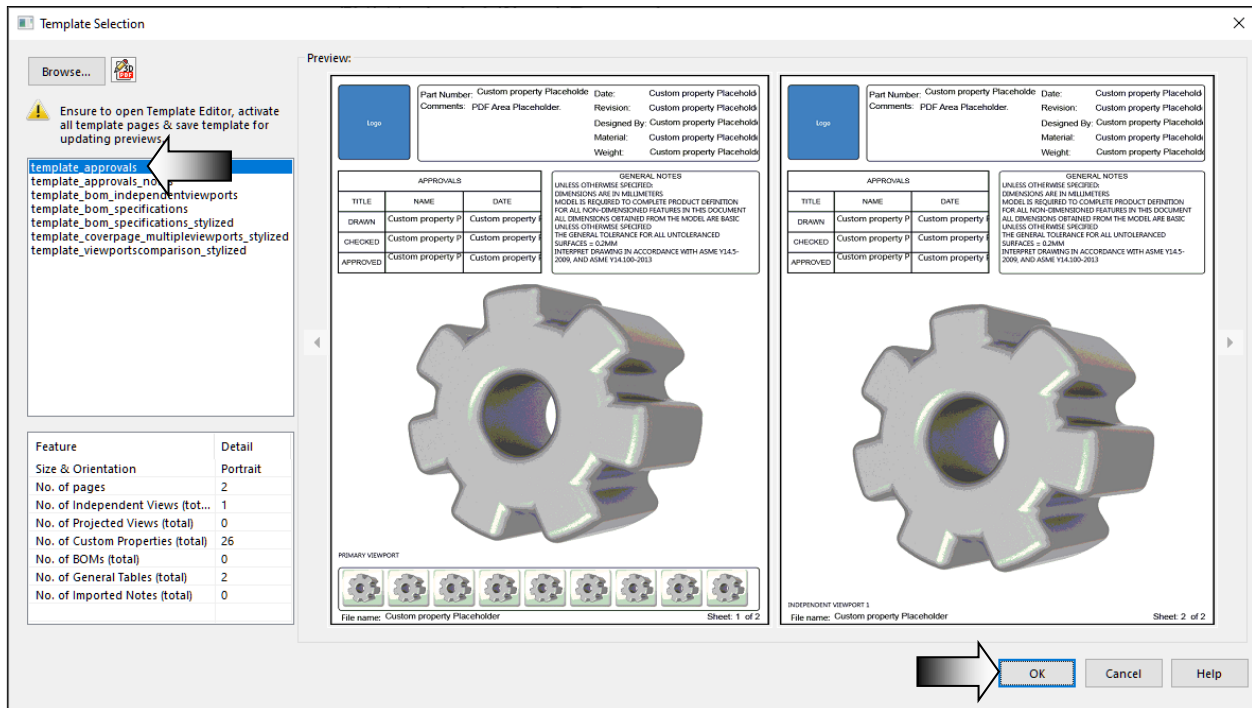
- Publishing to 3D PDF or eDrawing is the last step in using SOLIDWORKS MBD.



- Publishing takes a few steps to complete. The user has to define the template, the selection of the views and output details that include text and property.

- Click **Publish to 3D PDF** (either on the MBD toolbar or at the lower left corner).

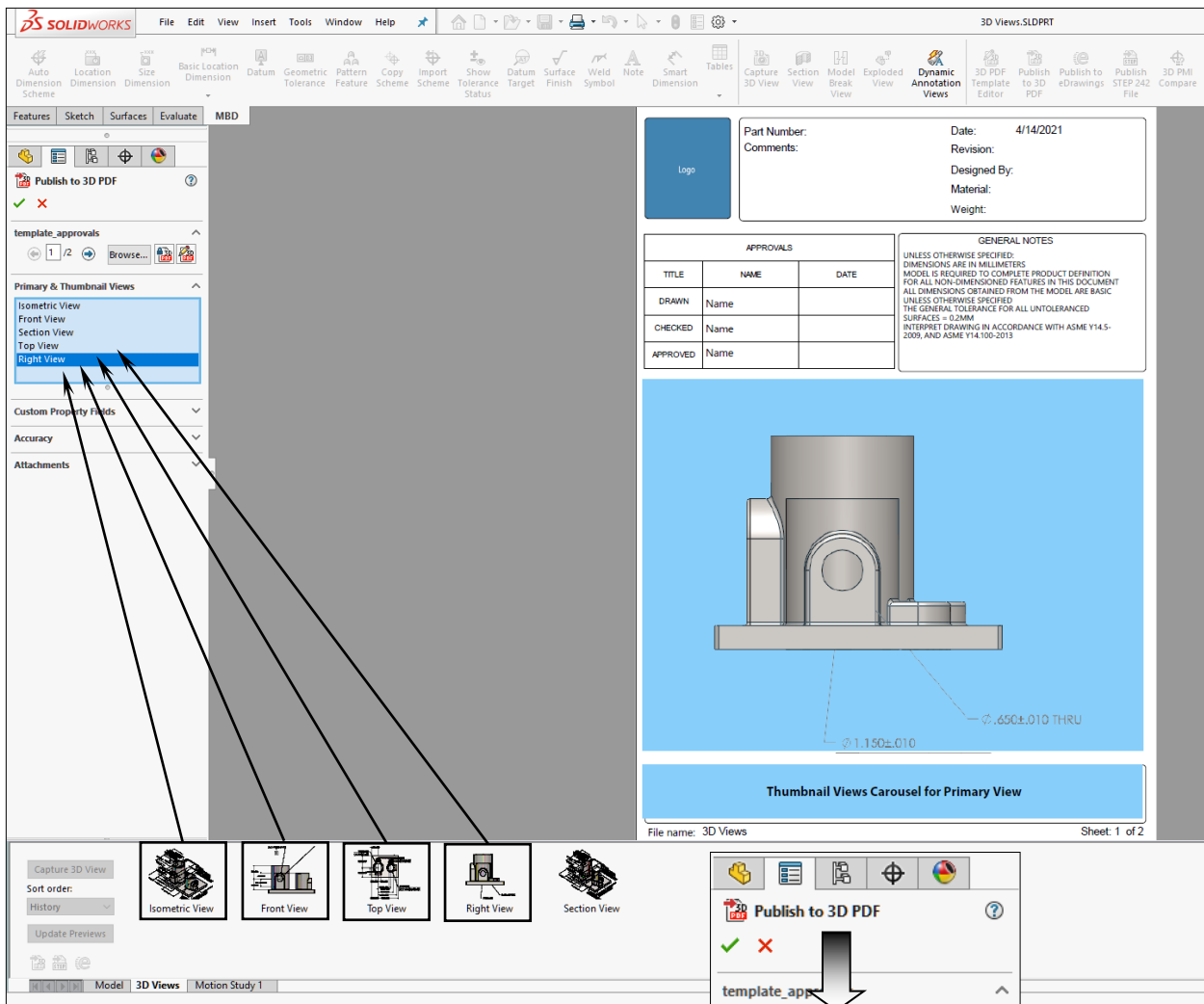
- For Theme, select **Template_Approval** (arrow). This template comes with 2 sheets by default.



- Click **OK**.

SOLIDWORKS 2019 | Intermediate Skills | SOLIDWORKS MBD

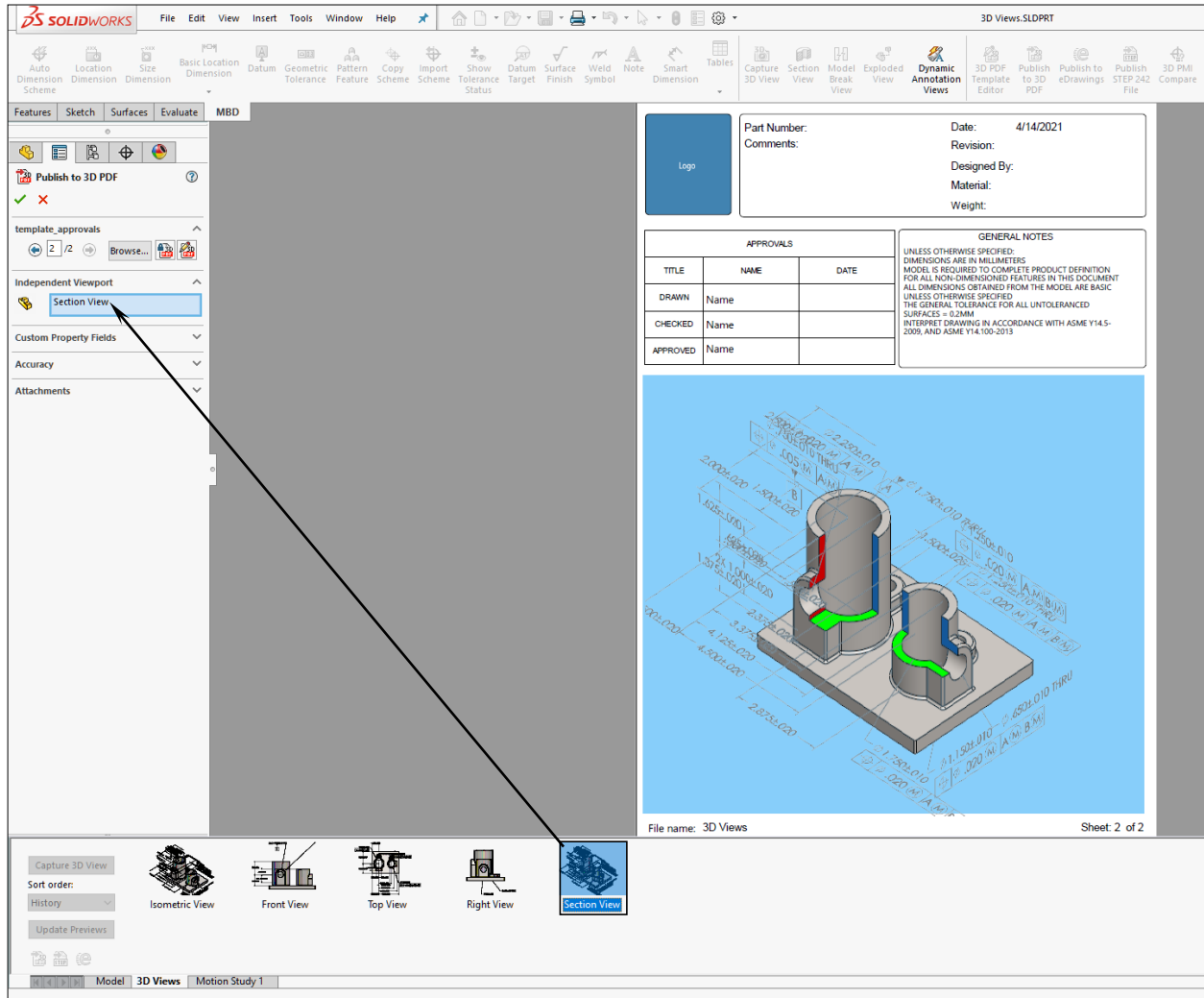
- The **Publish to 3D PDF** screen appears displaying the details of the 2 sheets.
- Click inside the box under Primary & Thumbnail View and select the **Front View**, **Top View**, and **Right View** in the bottom pane.
- The 1st view in the list (the Isometric View) will be used as the Primary View, and the other views will be added to the Thumbnail Carousel in the PDF file.



- Click **Next**  to switch to **Sheet 2 of 2**.

SOLIDWORKS 2019 | Intermediate Skills | SOLIDWORKS MBD

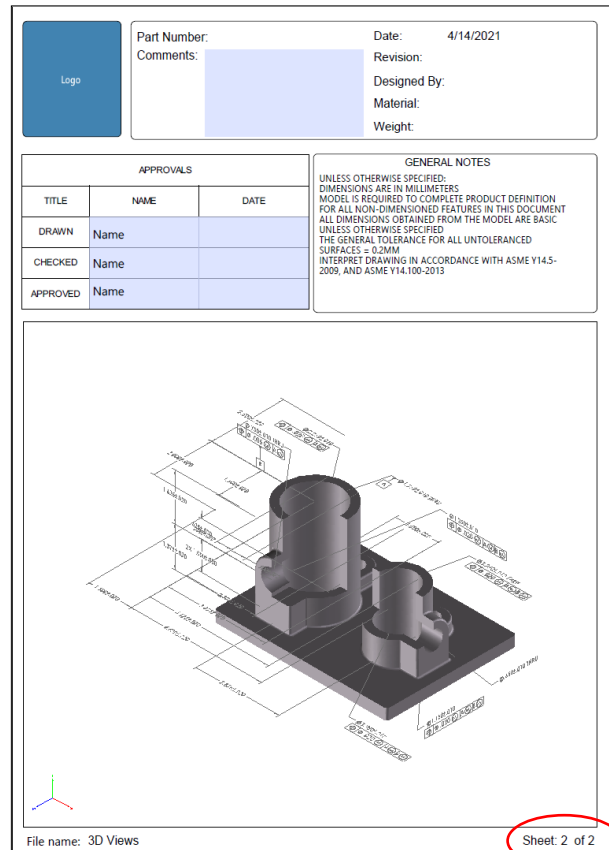
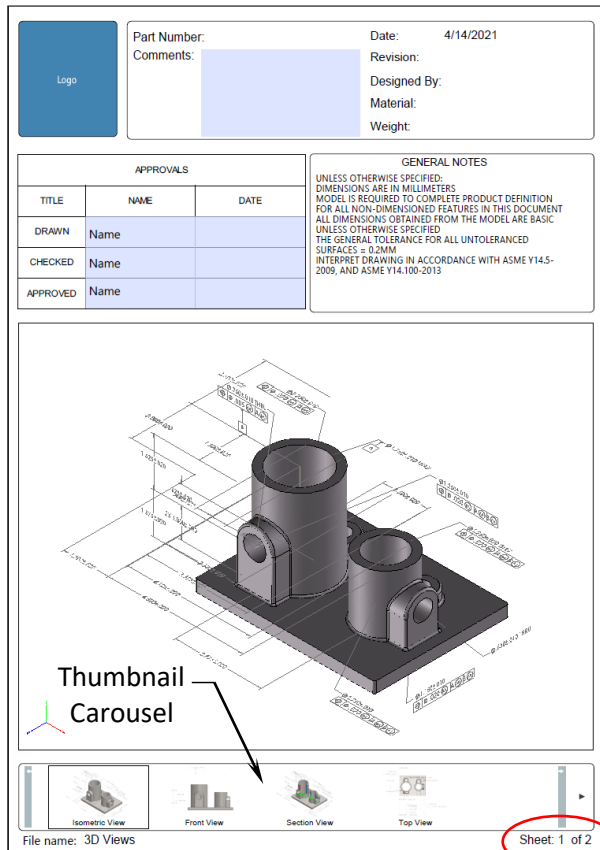
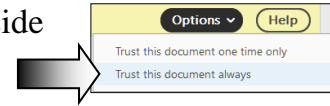
- Expand the **Independent Viewport** section, click in the box and then select the **Section View** from the 3D Views tab in the bottom pane.



- The Section View will be displayed in sheet 2 of the PDF file.
- Click **OK**.
- The Publish to 3D PDF PropertyManager reads the specify details of the active model and saves them in a 3D PDF file.

2. Viewing the 3D PDF file:

- Open the PDF file manually if the PDF application is not launched automatically.
- If prompted, expand the **Options** section at the top right side of the screen and select: **Trust this document always**
- Select the **Isometric View** from the Thumbnail Carousel to activate it. Click any other view to display it in the graphics area.



- * Hover the mouse cursor over the model area and roll the scroll-wheel up and down to zoom in and out.
- * Press and hold the scroll-wheel to rotate the view.
- * Press and hold the control key + the left mouse button to pan.
- * Press and hold the shift key + the left mouse button to zoom in/out.

3. Saving the PDF document:

- Save, close and return to the previous SOLIDWORKS document.