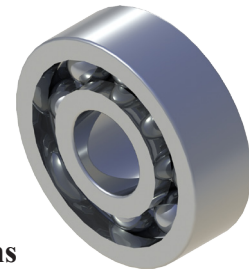


Hoverboard Bearing



A. Enable Toolbox Browser.

Step 1. If necessary, turn on Toolbox Browser, click the flyout of Options

on the Standard toolbar and click **Add-Ins**.

Step 2. Check **SOLIDWORKS Toolbox Library** to place a check in the both check boxes, then click OK, **Fig. 1**.

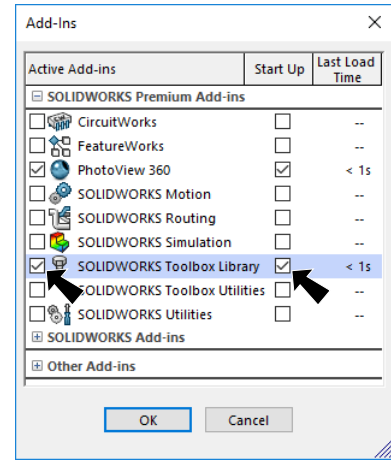


Fig. 1

B. Toolbox Bearing.

Step 1. Click **Design Library** tab in the Task Pane (right side of graphics area), **Fig. 2**.

Step 2. Expand the **Toolbox** **Toolbox**
 Expand **ANSI Metric** folder > **ANSI Metric**
 Expand **Bearings** folder > **Bearings**
 Click **Ball Bearings** folder **Ball Bearings**

Step 3. **Right click Instrument Ball Bearing...** and click **Create Part** from menu, **Fig. 2**.

Step 4. In the Component Property Manager set:
 under Properties, **Fig. 3**
 Size **0080-22**
 Display **Detailed**
 click OK .

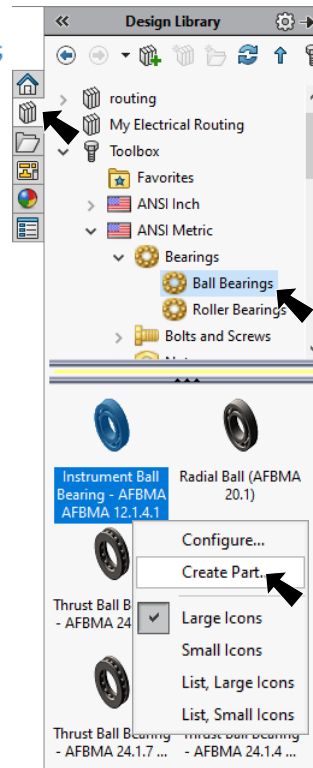


Fig. 2

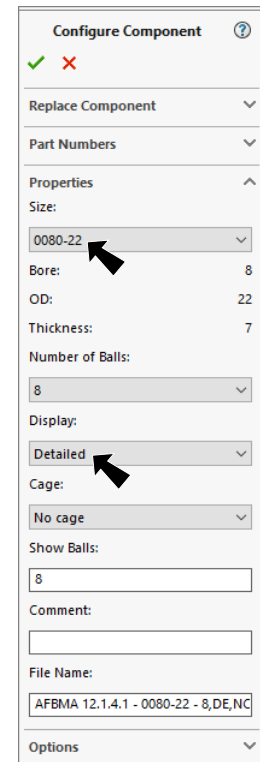


Fig. 3

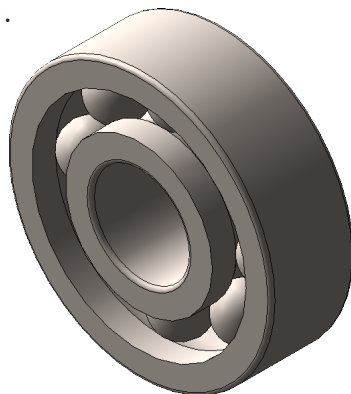


Fig. 4

C. Save As in Tech Ed folder.

Step 1. Click File Menu > Save As.

Step 2. In the Save As dialog box:
key in **BEARING 0080-22**
for file name

navigate to **My Documents/
Tech Ed 20-21/Hoverboard**
folder
click **Save**, Fig. 5.

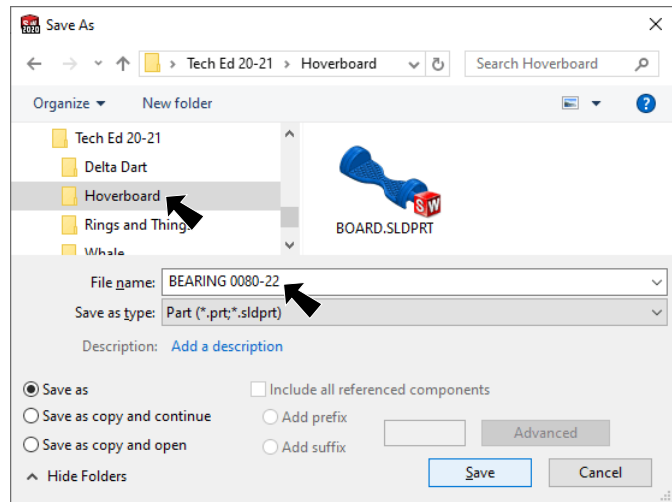




Fig. 5

D. Appearance Chrome.

Step 1. Click part to select, click **Appearance**

Callout  on the context toolbar and
click **BEARING...** , Fig. 6.

Step 2. In the Appearances Task pane, ex-
pand **Metal**, click **Chrome** and in
the lower pane select **chromium
plate**, Fig. 7.

Step 3. Click OK  in the Property Manager.

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

Step 5. Close File. Use **Ctrl-W**.

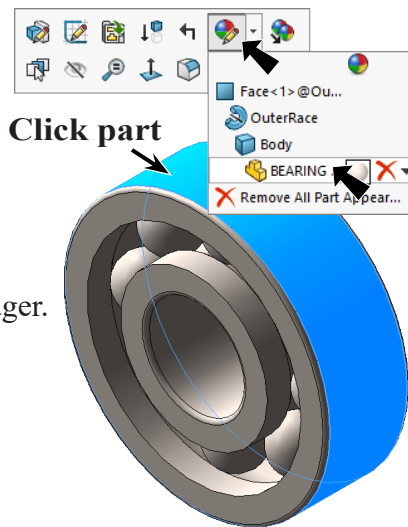


Fig. 6

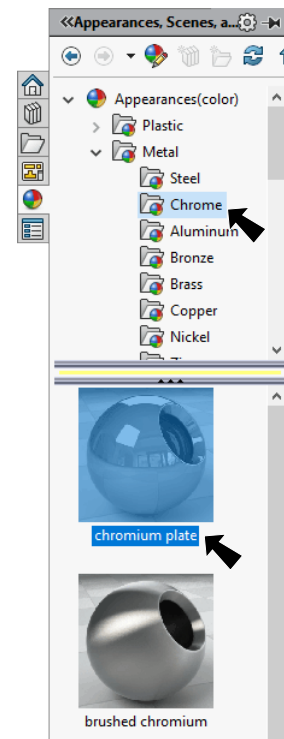
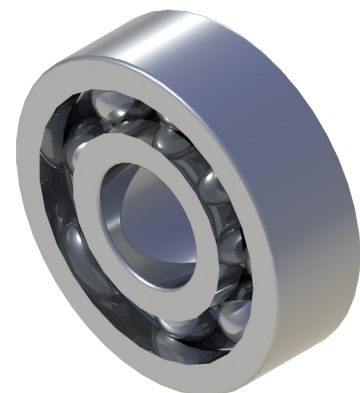


Fig. 7



Fig. 8



Rendered