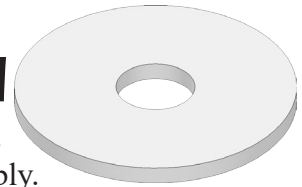


Wheel Alignment Tool



Note: This part is used as a spacer when assembling your 3D printed wheel components and is not used in SOLIDWORKS assembly.

A. Extrude.

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

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

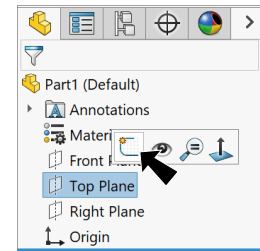


Fig. 1

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

Step 4. Sketch two circles at **Origin** , **Fig. 2**.

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

Step 6. Dimension **diameters 26.8 and 7.5**, **Fig. 2**.

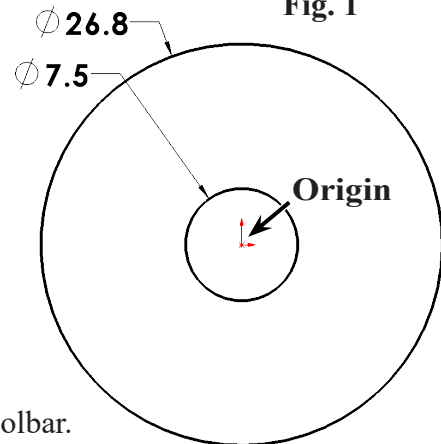


Fig. 2

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

Step 8. Click **Extruded Boss/Base**  on the Features toolbar.

Step 9. In the Boss-Extrude Property Manager set:

under Direction 1, **Fig. 3**

End Condition **Blind**

Depth  **1.75**

click OK .

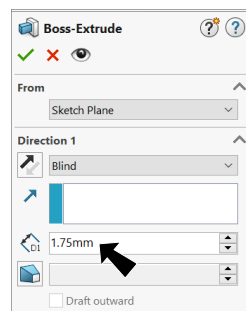


Fig. 3

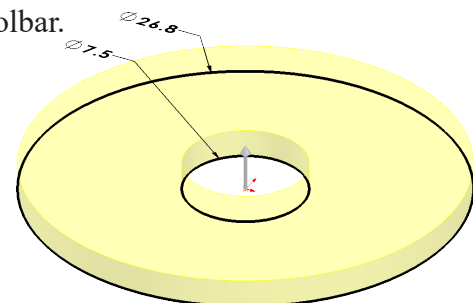


Fig. 4

B. Save as "WHEEL ALIGNMENT TOOL".

Step 1. Click File Menu > Save As.

Step 2. Key-in **WHEEL ALIGNMENT TOOL** for the filename and press ENTER.

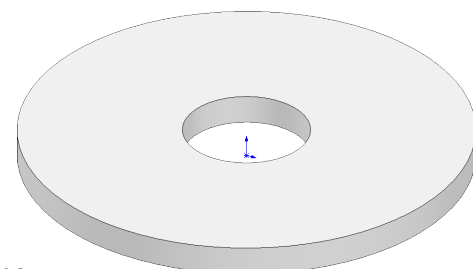


Fig. 5

Note: This part is not a component of the assembled project. The 3D printed part is used as a spacer when assembling the 3D printed wheel parts.