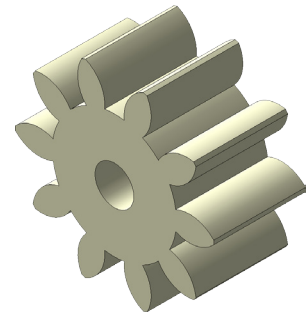







Solar Sidewinder Gear 10T



A. Toolbox New Part Spur Gear 10T.

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

If necessary activate Toolbox add-in. Click the **flyout of Options**  on the Standard toolbar and click **Add-Ins**. In the dialog box find **SOLIDWORKS Toolbox Library** and place a check in the check box under **Active Add-Ins** and **Start-Up**. Click OK.

Step 2. Expand the **Toolbox**  Toolbox
 Expand **ANSI Metric** folder  ANSI Metric
 Expand **Power Transmission** folder  Power Transmission
 Click **Gears** folder  Gears

Step 3. In the lower pane, **right click Spur Gear** and click **Create Part** from menu, **Fig. 1**.

Step 4. In the Component Property Manager set:

- under Properties, **Fig. 2**
- Module: .9**
- Number of Teeth: 10**
- Pressure Angle: 20**
- Face Width: 5**
- Hub Style: None**
- Nominal Shaft Diameter: 2**
- Keyway: None**

click OK .

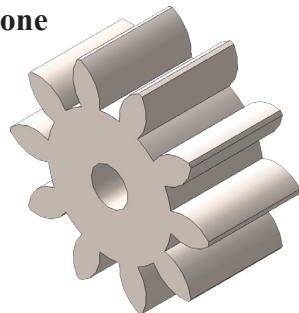


Fig. 3

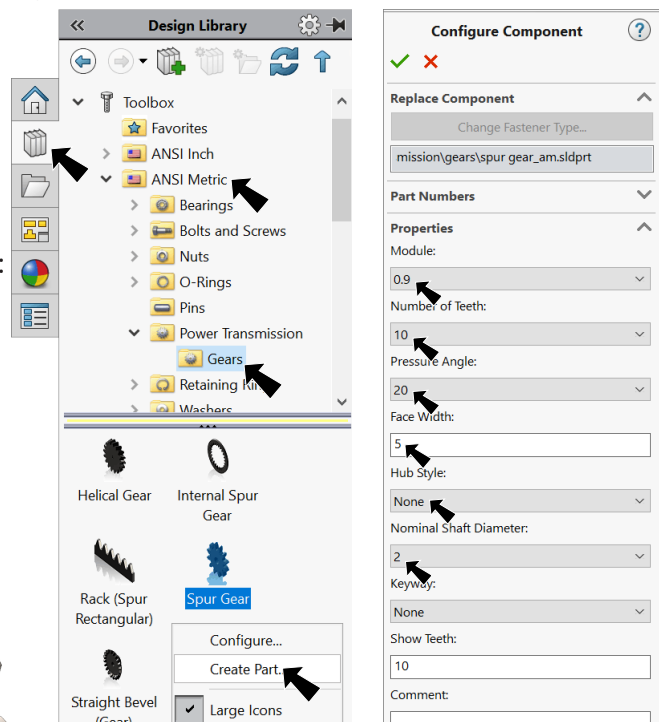


Fig. 1

Fig. 2

B. Save as "GEAR 10T".

Step 1. Click File Menu > Save As.

Step 2. In the Save As dialog box:
 key in **GEAR 10T** for file name.

Navigate to your **My Documents/ Tech Ed 24-25/Solar Sidewinder** folder
 click **Save**, **Fig. 4**.

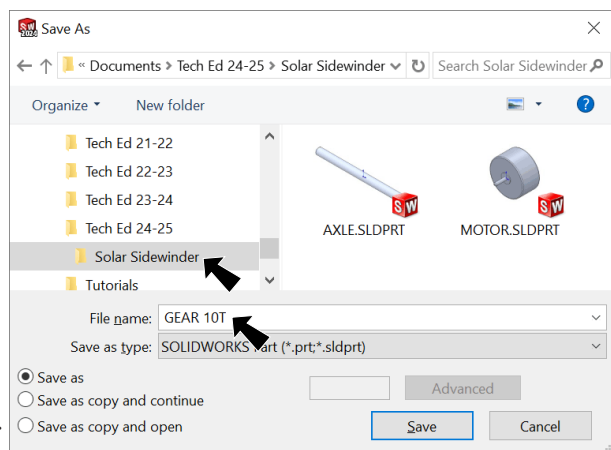


Fig. 4

C. Add Bore Shaft Clearance and Draft.

Step 1. Click **Plane3**  in the Feature Manager and click **Sketch**  on the context toolbar, **Fig. 1**.

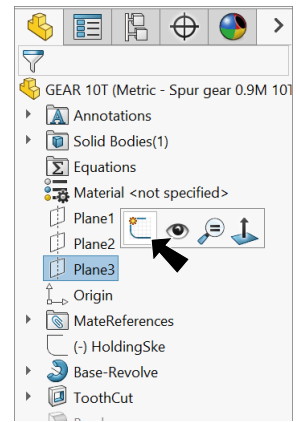






Fig. 5

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

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

Step 4. In the Offset Entities Property Manager set:
under Parameters, **Fig. 6**

Distance  **.1**
(clearance of Motor Shaft)
click edge of bore, **Fig. 7**
Yellow offset circle on outside
click OK .

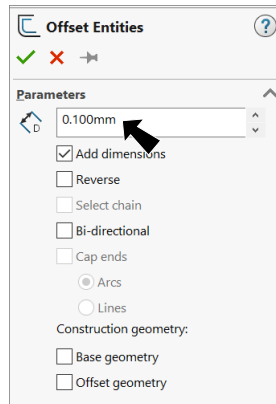


Fig. 6

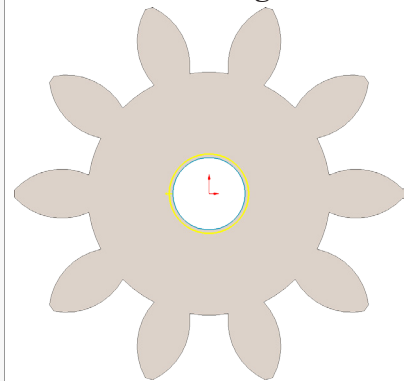

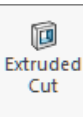




Fig. 7

Step 5. Click **Isometric**  on the Standard Views toolbar. (**Ctrl-7**)

Step 6. Click **Features**  on the Command Manager toolbar.

Step 7. Click **Extruded Cut**  on the Features toolbar.

Step 8. In the Cut-Extrude Property Manager set:
under Direction 1, **Fig. 12**
End Condition **Through All**

Reverse Direction 
click **Draft**  **.5°**
click OK .

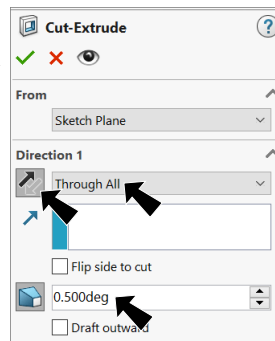


Fig. 9

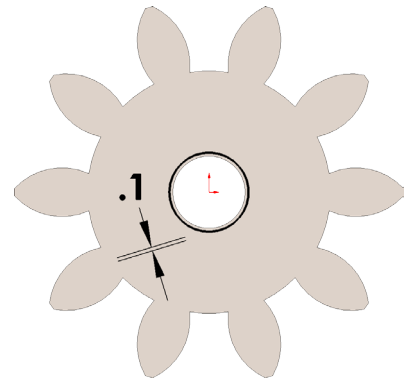


Fig. 8

Step 9. Save  (**Ctrl-S**).

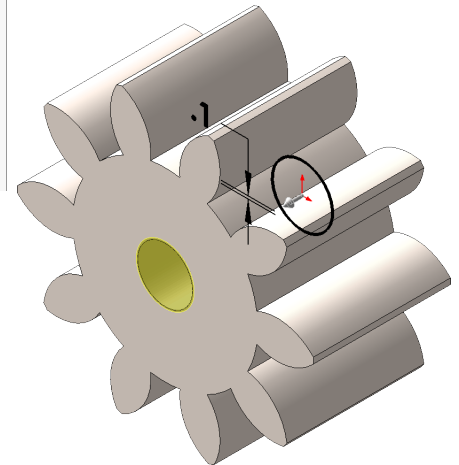


Fig. 10

D. Mate Reference.

Step 2. Expand **MateReferences** folder in the Feature Manager, **right click MateReference** and click **Edit Definition** from menu, **Fig. 11**.

Step 3. In the Mate Reference Manager:
under Primary Reference Entity, **Fig. 12**

click in the box and click a **cylindrical face**, **Fig. 13**
click OK .

Step 4. Rebuild . To Rebuild, click Rebuild on the Standard toolbar or **Ctrl-B**.

Step 5. Save (Ctrl-S).

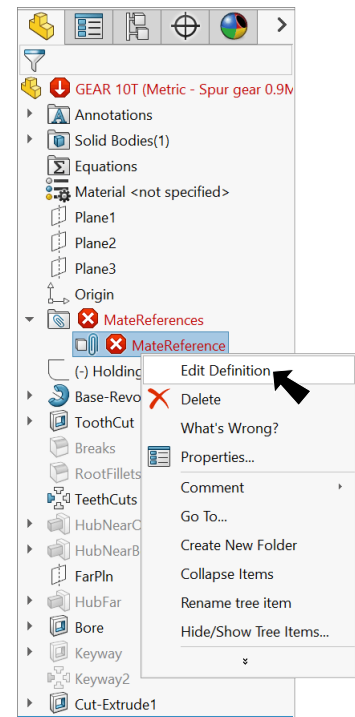


Fig. 11

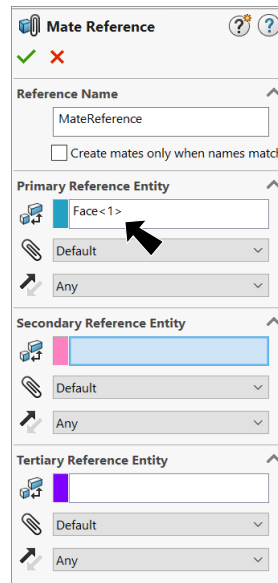


Fig. 12

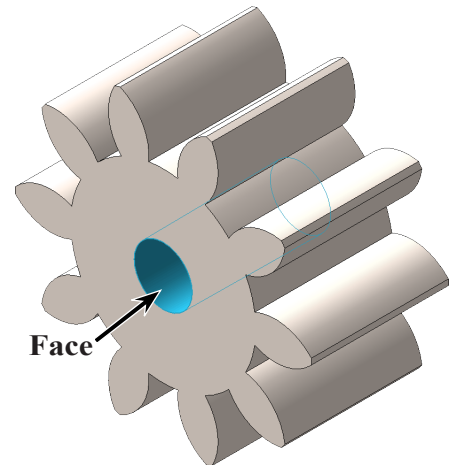


Fig. 13

E. Appearance Yellow Plastic.

Step 1. Click part, click **Appearance Callout**  on the context toolbar and click **GEAR 10T** , Fig. 12.

Step 2. In the Appearances Task pane, expand **Plastic**, click **High Gloss** and in the lower pane select **yellow high gloss plastic**, Fig. 13.

Step 3. In the Appearances Property Manager set:

under Color, Fig. 14

set RGB values

R 213

G 213

B 180

click OK .

Step 4. Save  (Ctrl-S).

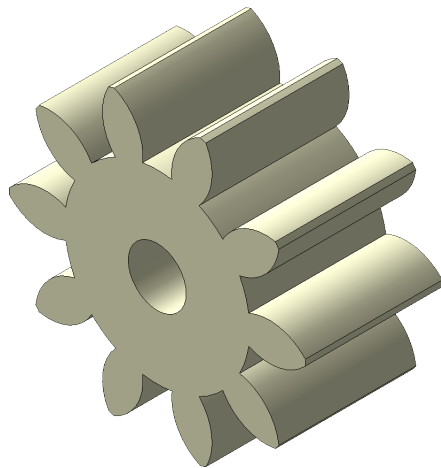


Fig. 17

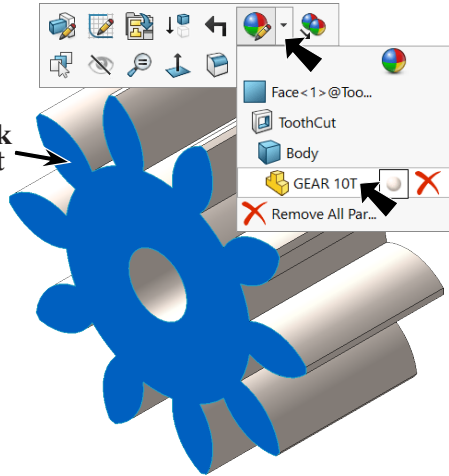


Fig. 14

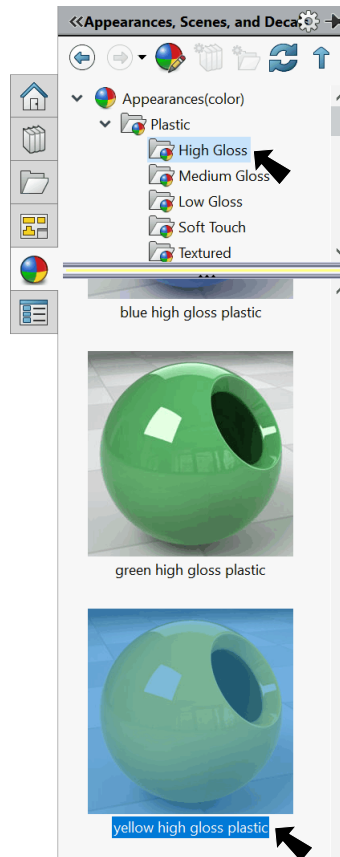


Fig. 15

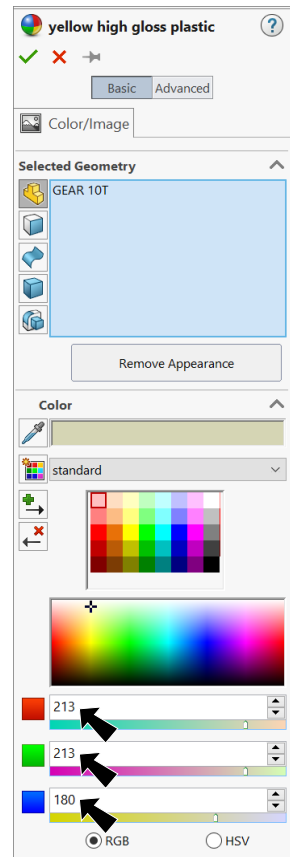


Fig. 16