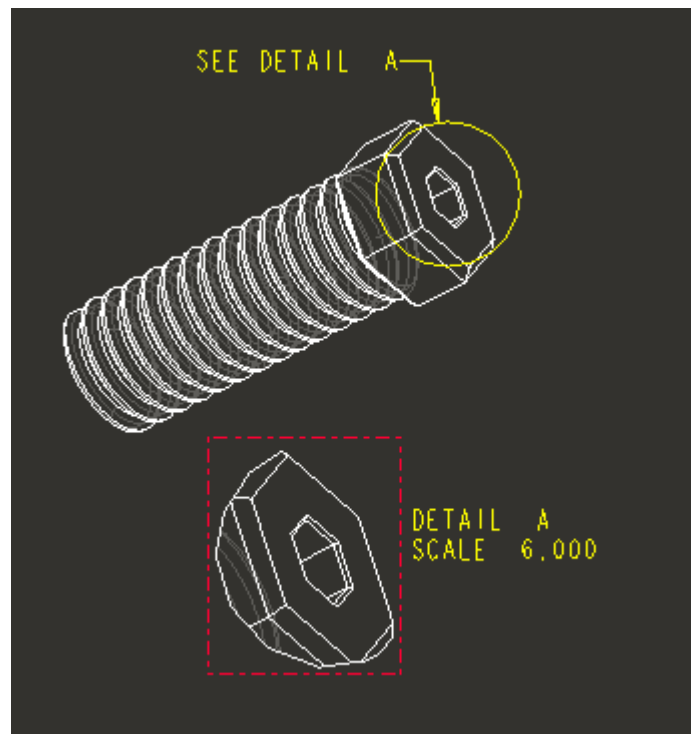


Technique for Modifying Detail view Circles to Splines or Ellipse

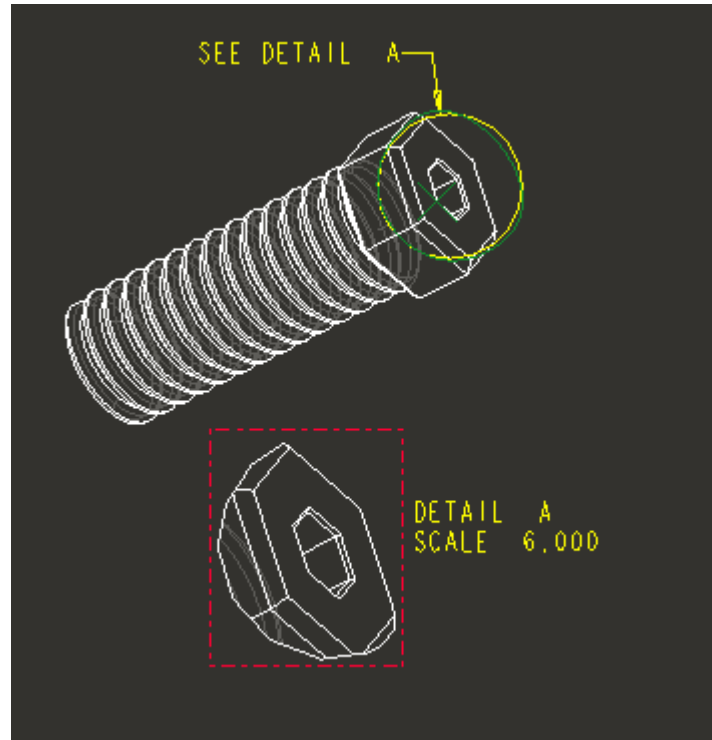
During the creation of a **Detailed** view in Drawing mode, a reference point is selected on an existing view and a spline is sketched around the model geometry to include in the **Detailed** view. This spline defines the diameter of the detail view circle (with the attached detail view note) shown on the parent view. The detail view circle can be defined as an ellipse or spline during view creation. The following steps illustrate the method for modifying an existing detail view circle to be in the shape of an ellipse or spline.

Procedure

To modify the shape of the detail view circle shown in Figure 1, Pick the detail view, **Edit > Properties** the Drawing View dialog box opens, select **> View Type** from the Categories list.



A red box highlights the **Detailed** view and the original spline sketched to create the view displays in green, as shown in Figure 2. The View Type menu appears with the options for the Boundary type on parent view, including **Circle**, **Ellipse**, **H/V Ellipse**, **ASME 94 Circ** and **Spline**. An **Ellipse** shape is formed from the longest diagonals of the sketched spline and an **H/V Ellipse** is shaped from the longest horizontal and vertical vertices of the sketched spline. The **ASME 94 Circ** option draws a circle in the parent view according to the ASME 1994 standard. The **Spline** option displays the detail view circle in the shape of the spline sketched around the reference point.



Select the desired shape for the detail view circle and choose **OK** from the Drawing View dialog box. The detail view circle will update to the specified shape, as displayed in Figure 3.

