

Bolt Circle dims

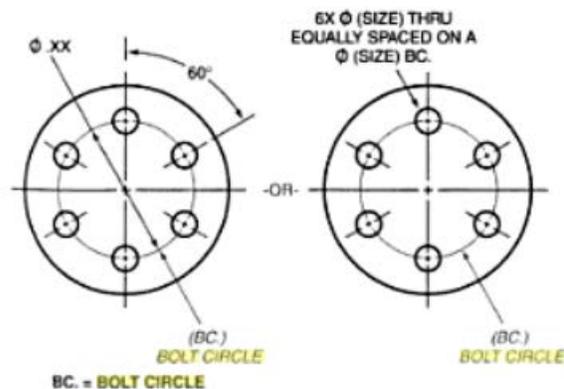


FIGURE 10-91 Dimensioning holes on a **bolt** center diameter

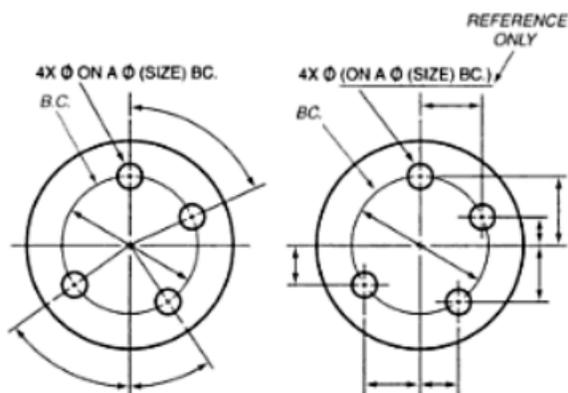
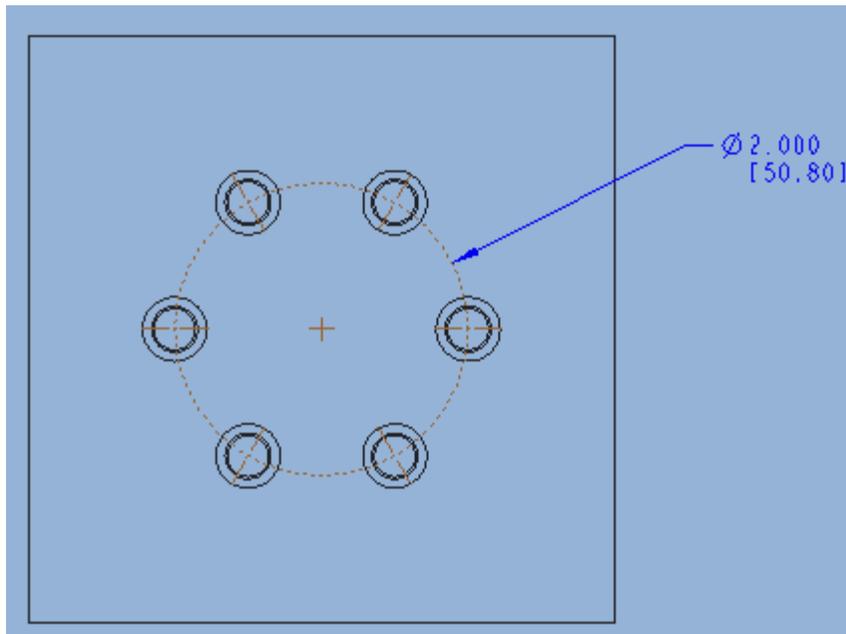


FIGURE 10-92 Dimensioning holes around a center

DIMENSIONING HOLES ON A **BOLT** CENTER DIAMETER

Figure 10-91 illustrates the proper methods used in dimensioning holes on a **bolt** center diameter. In the example on the left, the diameter of the bolt's **circle** is given and one angle is shown to indicate the number of degrees between the center lines of the six circles. This is sufficient information for the machinist to properly drill the holes. In the example on the right, a note is used to accomplish the same thing. Notice that the note specifies the number and size of the holes. It explains that the holes are drilled through the object and equally spaced on a **bolt** center. Either of these methods may be used for dimensioning holes around a center. Figure 10-92 shows two other methods that are used for dimensioning holes around a center. These methods are used when the holes around the **bolt** center are not equally spaced. The example on the left gives a dimension for the **bolt** center, a note containing the number of holes and their size, and angles turned from a reference center line for locating the center lines of the other three holes. The example on the right substitutes linear dimensions for angular dimensions.

When you create a hole from the Creo hole program and used the Diameter Hole creation, you need to specify an axis to measure the diameter from and then when you pattern the hole with a specified Axis pattern, doing this is why the B.C. axis would not show up. If you have defined a hole using either Diameter or Radial selection and want to pattern, with a hole circle, all you need to do is when you pattern just keep the pattern selection at dimension and select the **angle** dimension that you specified when you created the lead hole and the axis will show up on your drawing. Along with the Bolt Circle diameter dimension.

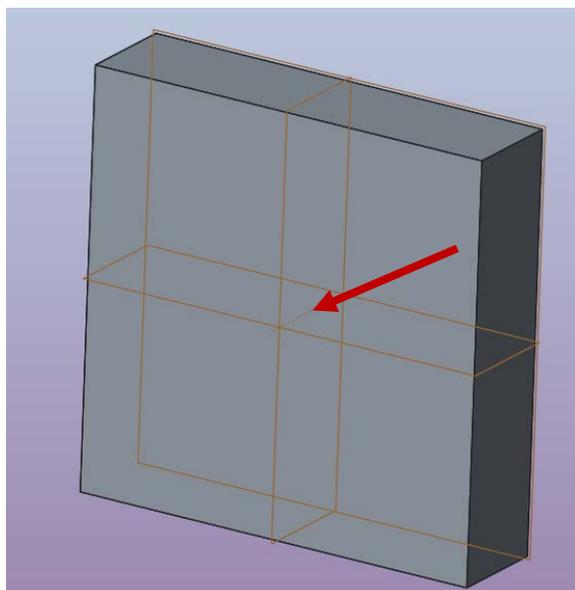


Open this part and drawing in Creo. BOLT_CIRCLE_SQUARE. To work this example.

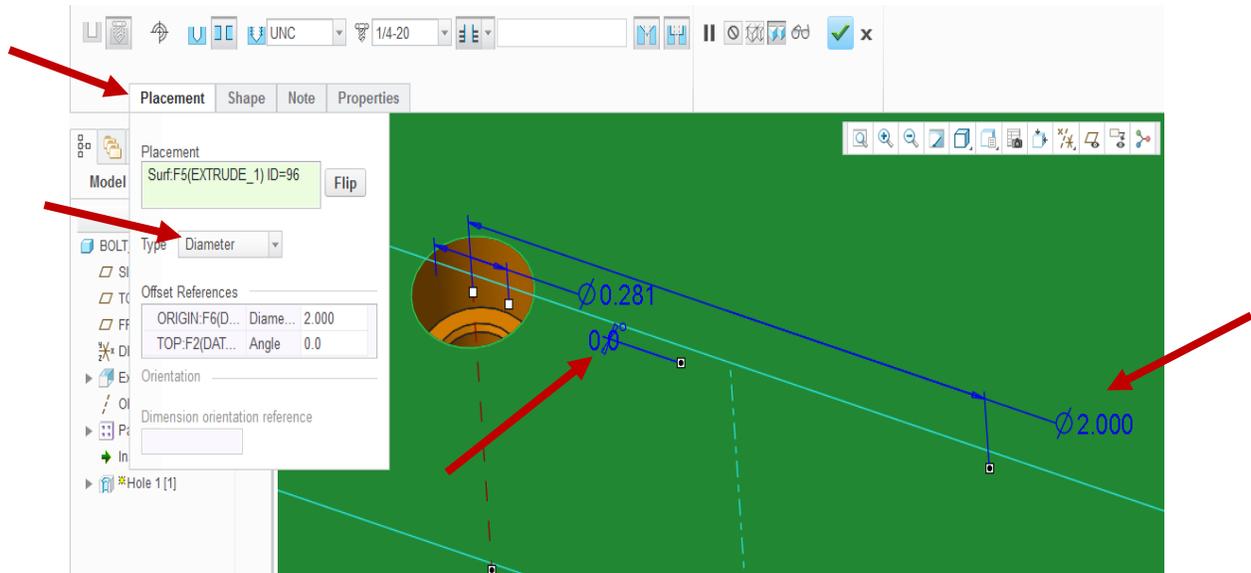
The Z-axis is regarded as the Beam center line.

The square part is centered on the Z-axis with the Side and Top datum planes intersecting at the origin.

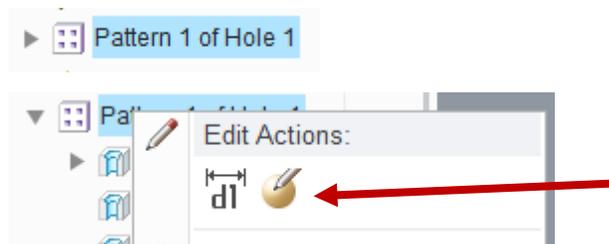
An axis datum is placed at the intersection and renamed ORIGIN.



1. Edit definition for the first hole, in the pattern. Select the **Placement tab**, Set Type, **Diameter**, not Linear.
2. For new creation, select the front datum. Input the hole requirements, move the highlighted little square drag handles. One on the origin axis and the other to the side or top datum plane.
3. Set angle to the hole location. Set it to 0 degrees. And modify the Bolt Circle diameter. Okay.

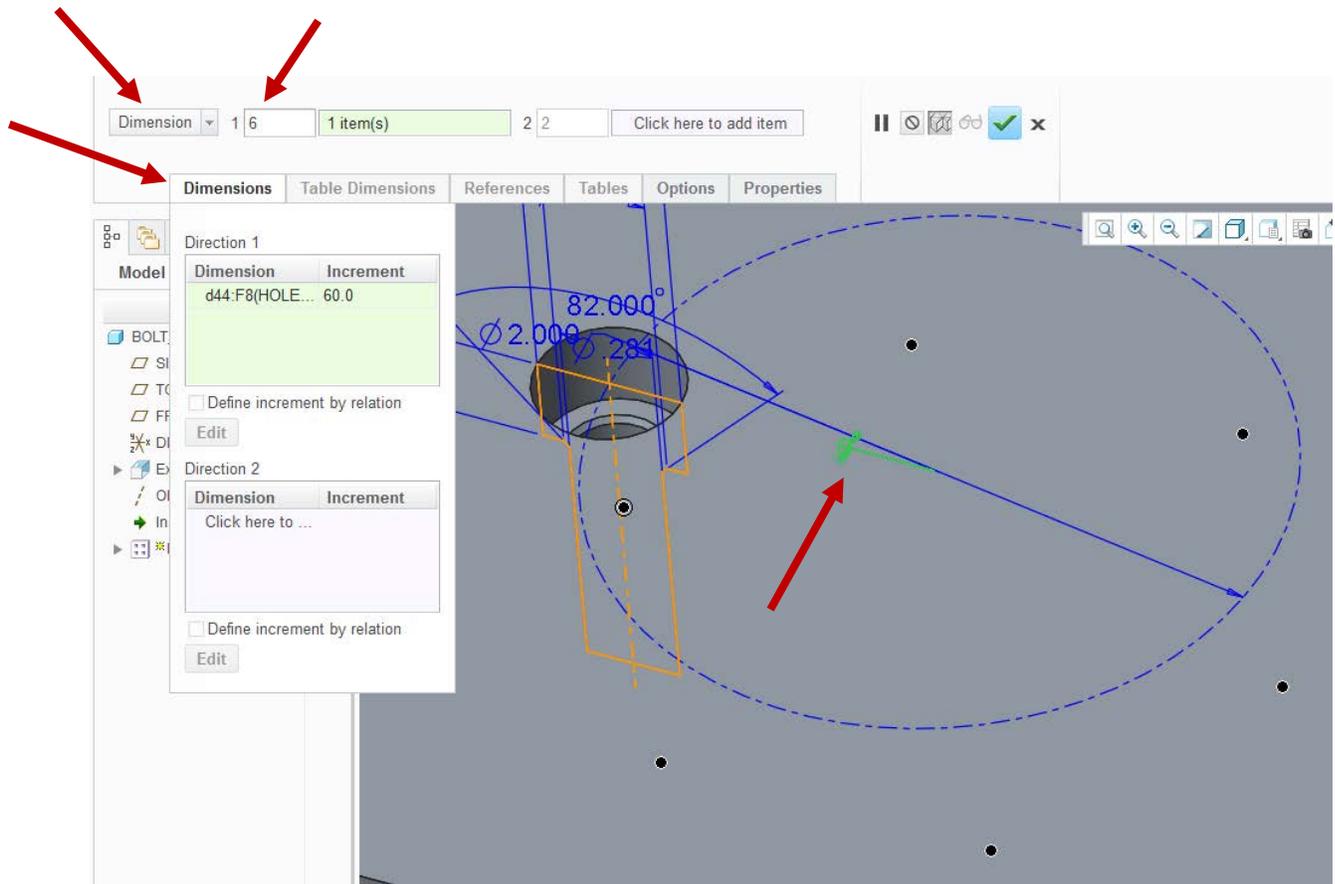


4. Then pattern, set the type of pattern to **Dimension**.



5. Open the Dimensions tab.
6. Select the angle dimension and set Direction 1 and Increment to the angle. 60.0 or what's needed.
7. Enter the number of pattern members in the first direction. 6 or what's needed.
8. Done, click on the check box.

Edit or create new, the definition of the hole pattern.



1. Open the Drawing.
2. Select the Annotate tab, click on Show Model Annotations.
3. Select one of the holes.
4. Click on Show the model datums. Select them all and OK. The Bolt Circle will be shown.

