Layout of a Part with Holes

1. Open a blank drawing with your border and title block

2. Place the front, top and right side views with hidden lines and the isometric without hidden lines. Scale: .500 = 1.000

NOTE: Set for Each View except Isometric as the following: View Properties > View Display > Tangent Edges > None (SEE IMAGES Below with and without Tangent edges)
3. Placing Centerlines

a. Select a view > Select Show Annotations from the Annotate Tab > Select the last tab on the pop up menu

b. Select View

c. Select all of the Centerlines Tab (Last Tab on the Pop-Up Menu) and click OK

NOTE: Centerlines will only appear if the feature is a circle creation (Full 360 degrees). Features like Rounds, Fillets, Arcs that are not cutting away material will not have centerlines. Arcs that cut material away will have centerlines.

d. Repeat steps for the other two views. Finished Drawing should look as follows

NOTE: This centerline is not needed. Delete or uncheck from Centerline menu. Centerline is present

4. Option selects all centerlines or user can select one centerline at a time

DO NOT Place these Centerlines
**Placing Dimensions**

Holes need 3 Dimensions
a. Diameter and Depth (If not all the way through)
b. 2 Locators

Holes and Arcs should always be dimensioned where the circle profile is seen of the hole/arc. Never the hidden lines

4. **Placing Locators**

Setting Locators is the same as setting a linear dimension on edges.
Select the dimension tool and select either and edge or centerline or circle to dimension start from and then select a parallel edge or centerline or hole to dimension to > Middle Mouse Button to place. NOTE: if you select a circle Creo have a Menu Manager pop-up and it will ask you do you want to dimension to the center of the hole or tangent; you should always select center.

Depending on where the hole is located on the part, placing dimensions on the part maybe acceptable as is on our tutorial part. The Cleanup Dimension Spacing will not be in effect if a dimension is on the part. The user simply places evenly spaces it.

There should be a gap between the extension line and centerline. Be sure to stretch the centerline or extension line appropriately.

Place the following Hole Locators

![Dimension to this end because it is vertical surface. The far left vertical edge is the tangent line to the arc. Designers should avoid dimensioning tangent to an arc or hole](image)

Notice The Extension Lines appear to be part of the object lines. Drag the lines back to form a gap

Modify Arrow Length

File > Prepare > Drawing Properties > Detail Options > Change > draw_arrow_length > 0.09 > Add/Change > apply > Close
5. **Placing Leader Line Dimensions**

There are two types of Leader Line Dimensions: Radius and Diameter

A. To get Radius, use your linear dimension tool > Left Click once on the edge of the circle > Middle Mouse Button to Place

B. To get Diameter, use your linear dimension tool > Left Click Twice on the edge of the circle > Middle Mouse Button to Place

1. Place the following Diameters for your Holes
2. **Adjust Arrow Style**

   a. There are various ways to show the arrows. The default is a more traditional way, but can get confusing and in the way. We will change this to a more modern style that gives the engineer more flexibility in dimensioning.

   b. Select one of the Diameters > Right Mouse Button on it > Select Flip Arrows > Repeat Steps for the other Diameters

   Your Dimension should look as follows

3. Do the same for the other hole in the top view
4. Modifying Text
   a. The user might need to modify the text of a diameter if
      i. A hole does not go all the way through the part
      ii. There is one or more size holes of the same diameter and depth
   b. Select Dimension to add information too > Dimension Tab will appear at the top that looks as follows > Select Display Icon
      - Select Dimension to modify > Select Dimension Text Icon
      - This will take the user into the properties of the dimension
d. Place your cursor to the right of the @D (This is programming language referencing the solid model at that point) > Left Click > Select Text Symbols (Bottom Right Screen of the pop up menu)> Select the Depth Symbol that looks 

Type .500 after the symbol > Hit the Enter Key > Type on the second line 2 HOLES > Press OK
This means the hole is a diameter of 1.000 a depth of .500 and there are 2 Holes that share these measurements.

Repeat the above steps for the third hole
5. Dimension Fillets or Rounds

There are two ways to dimension Rounds and Fillets.

a. Dimension using a leader. Set the leader as Radius by Left Clicking once on the Round or Fillet > Middle Mouse Button to Place. If more than one Round or Fillet of the same size write TYP. Next to the Radius Value. If there are multiple different Rounds or Fillets than each should be dimensioned.

b. Shop Note: A Shop Note is used when basic information (Rounds, Fillets, Chamfers, Thickness are common throughout the part. I.E. NOTE: ALL ROUNDS R .250

NOTE: ALL UNMARKED ROUNDS AND FILLETS R .500

i. The Text Height for this text would be .1 tall. This NOTE should be placed outside the views. Usually placed below the Right Side View or above the Isometric

c. Choose either option when dimensioning the tutorial part.
Finished Part

Clean-Up Dimensions and Complete the dimensioning scheme shown

NOTE: Overall Length is not stated because the left extension line would be dimensioning tangent to an Arc. Designer should never dimension tangent to an arc or hole; only to the center

Save File

Print and Turn In Drawing