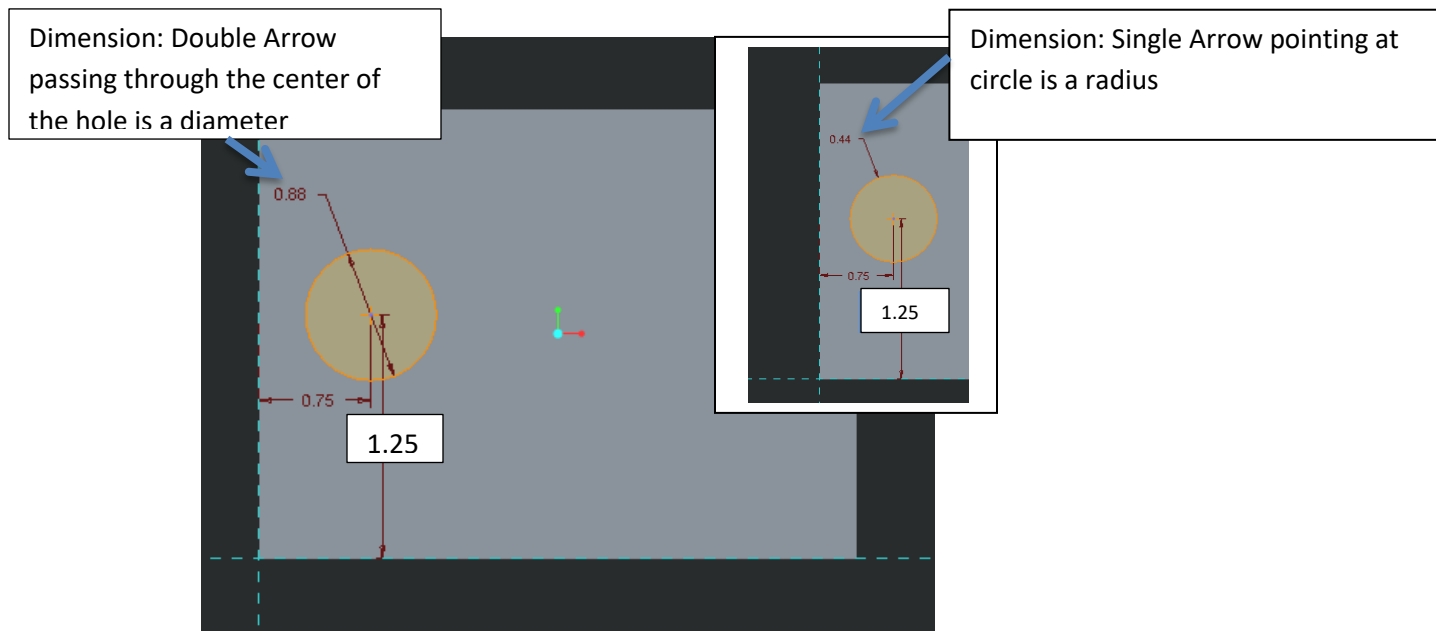


Creo: Hole Tutorial

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Brighton High School

1. Extrude a rectangle that measures Length 4" Height 3" on the Front Datum.
Extrude Distance 1.00
2. There are two ways to extrude a hole
 - a. Extrusion- Standard taking a 2D profile and sending the profile in a particular direction
 - b. Hole Tool- Allows the user design various types of holes in one standard tool (I.E Simple, Counterbore, Countersink, Threads, etc.)
3. Holes and Dimensions
Holes have 3 2D Dimensions and 1 Depth Dimensioned to define them.
All holes are measure in diameter and arcs are measured in radius
 - a. 1 Diameter Dimension
 - b. 2 Linear Locator Dimensions- measuring from an edge to the center of a hole or center to center of holes. (Holes are never measured to a tangent edge)
 - c. Depth Dimension
4. Extrusion
 - a. Select the Extrude Icon > Right Mouse Button > Define Internal Sketch > Select the front face of the part
 - b. Use the Circle Tool and draw a circle at this location



- c. Adjust the dimensions as shown above. To create a new dimension select Normal Icon > Select the edge to be measured from and click the center point of the circle > Middle Mouse Button to place the dimension. NOTE: CHECK which type of arrow head is shown to set the proper dimension whether Diameter or Radius (Radius= 1/2Diameter). Our worksheets will ALWAYS show Holes as diameters.

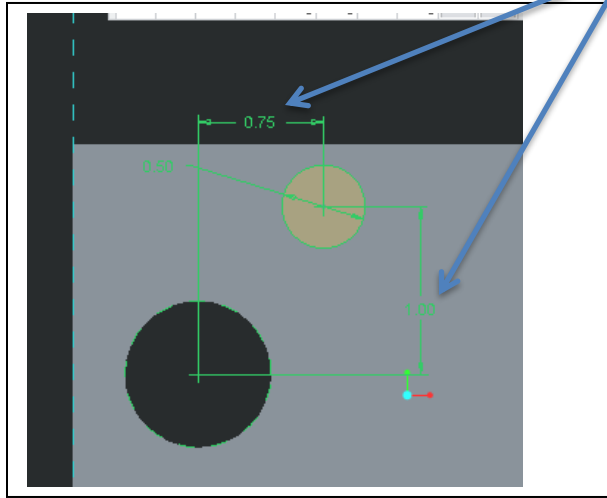
- d. Changing the size of the hole. Notice the following two types of arrows. One shows radius and the other show diameter.

Remember the ratio that

- a. $2 \times \text{radius} = \text{Diameter}$ or
- b. $\frac{1}{2} \text{Diameter} = \text{Radius}$

- e. Create a second hole using a new extrude. Set the hole size and locators to the following sizes.

- i. Diameter 0.50
- ii. Hole Depth 0.25



NOTE: How the Length Locator is measuring from center to center of holes. To place this dimension left click on the center of one hole, then left click on the center second hole then middle mouse button in the area the tutorial has the dimensions placed. You may get a dimension that goes directly point to point on an angle we do not want this dimension, simply delete the dimension and try again OR Measure from a known edge and add the values.

5. Hole Tool

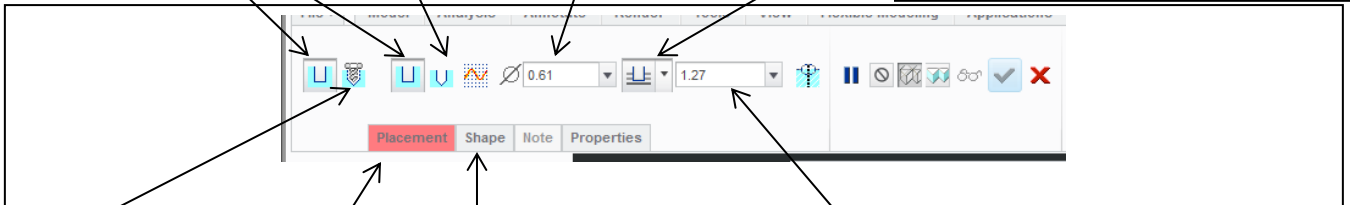
- a. Select the Hole Tool in the Model Tab
- b. The Info Tool Box will show up



Type of Hole Cut

1. Blind
2. Symmetric about Datum
3. Drill up to Next Surface
4. Drill Through All
5. Drill to Intersection of Surface
6. Drill to Selected Surface

Creates Simple
Counterbore and Countersink
Diameter of Hole



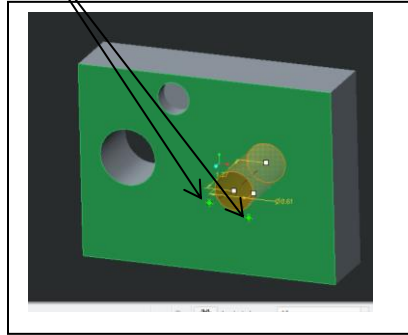
Creates Threaded Holes

Shape: Shows 2D Representation of the hole to help set the values

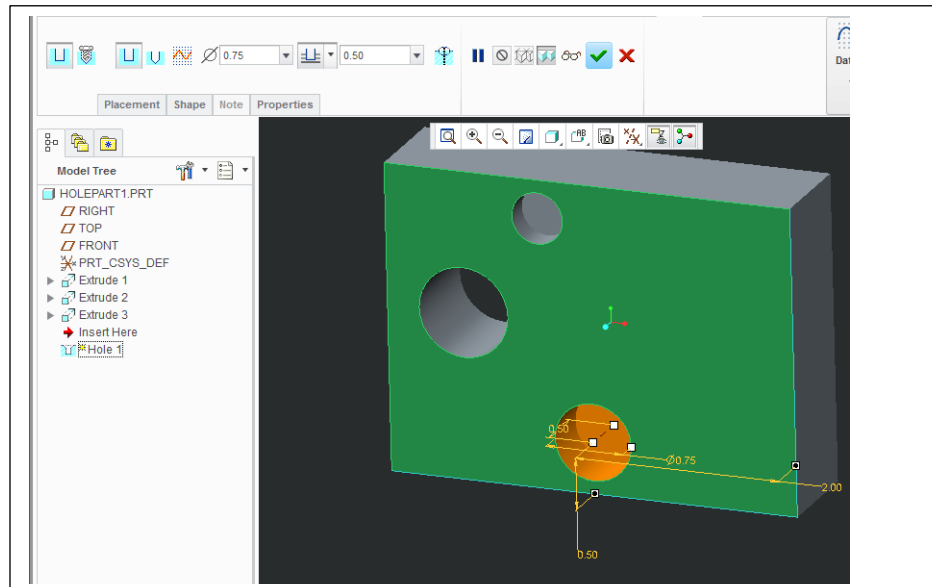
Depth of Cut in a Blind Cut

Placement: Adjusts locaters of the hole

- c. Select the front face of the part. A preview of the hole will show up with its dimensions. Dimensions the Green Handholds are your locators, Diameter and the depth of the hole.

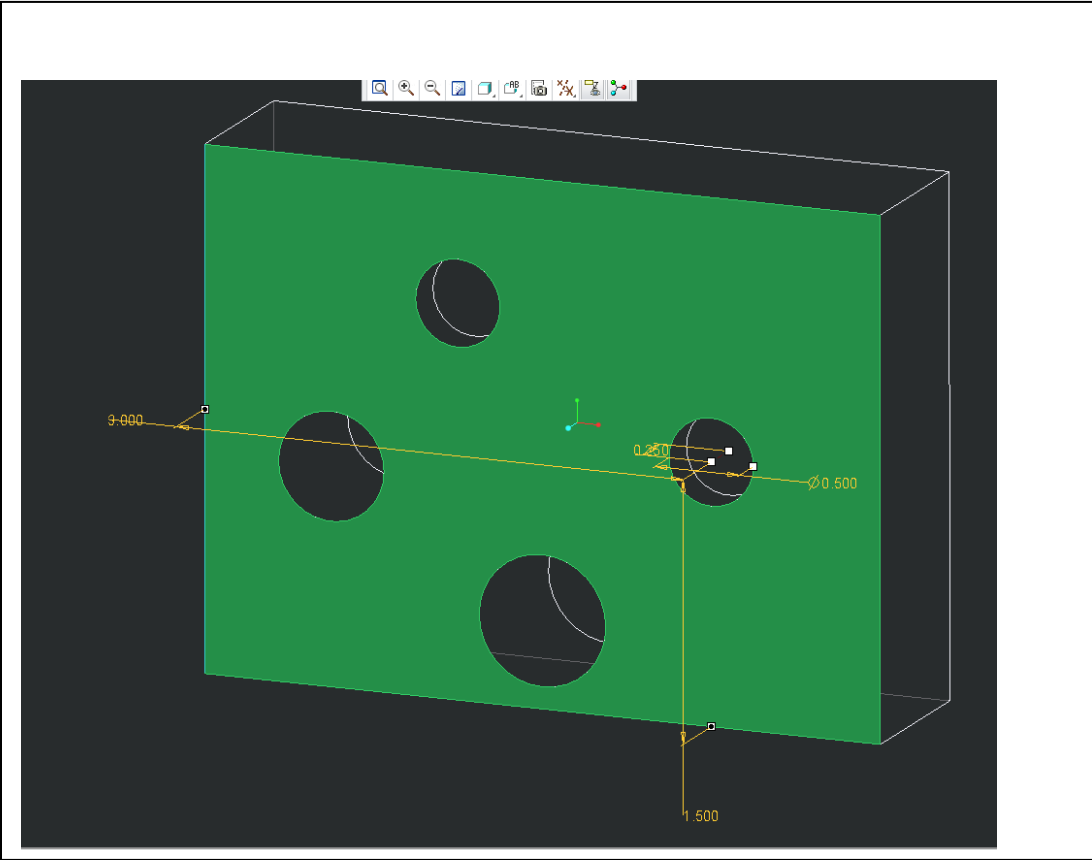


- d. Place your cursors on the Green Handhold then hold your left mouse key down on the Green Handhold and drag and drop on the following edge. Repeat the step with the other Green Handhold and place on the following edge. (Note these Green Handholds can be placed on edges or Datums)



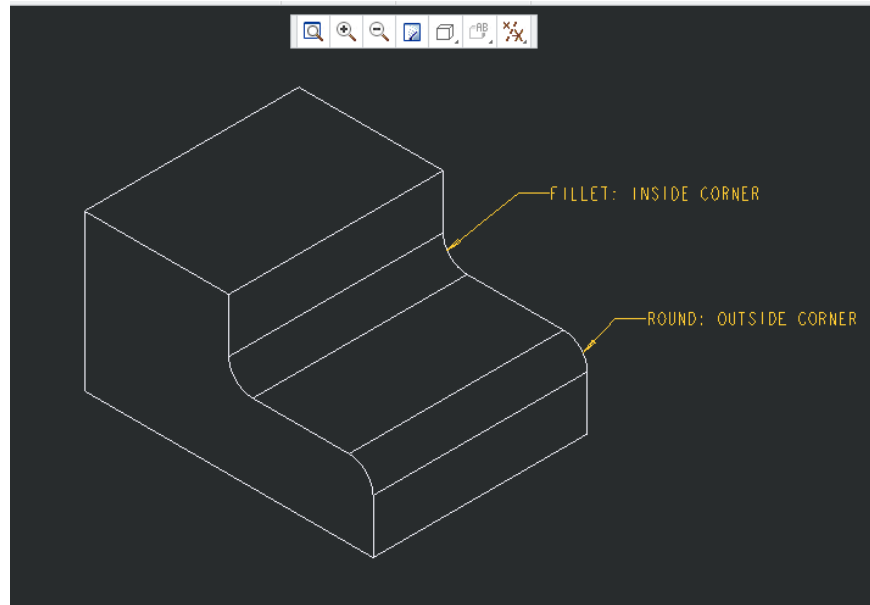
- e. Change the values in Hole Tool Info Box
i. Diameter 0.75
ii. Hole Depth: Through All
f. Green Check to finish the feature

g. Repeat the steps for the next hole. Set the following dimensions

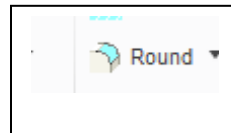


Filletlets and Rounds

1. Rounded Edges allow for the user to design a part without any sharp corners.
 - a. There are two different types of rounded edges. These edges are by-products of the manufacturing process. They allow for parts to be removed from a mold or cast easier. They also provide safety breaking sharp corners so there are no sharp edges.
 - i. Rounds- Are found on the outside corner of a part and subtracts material from the part
 - ii. Fillets are on the inside corner of a part and adds material.



2. Select the Round Edge Tool
This tool will Round and Fillet Edges
3. Round Edges Tool Bar

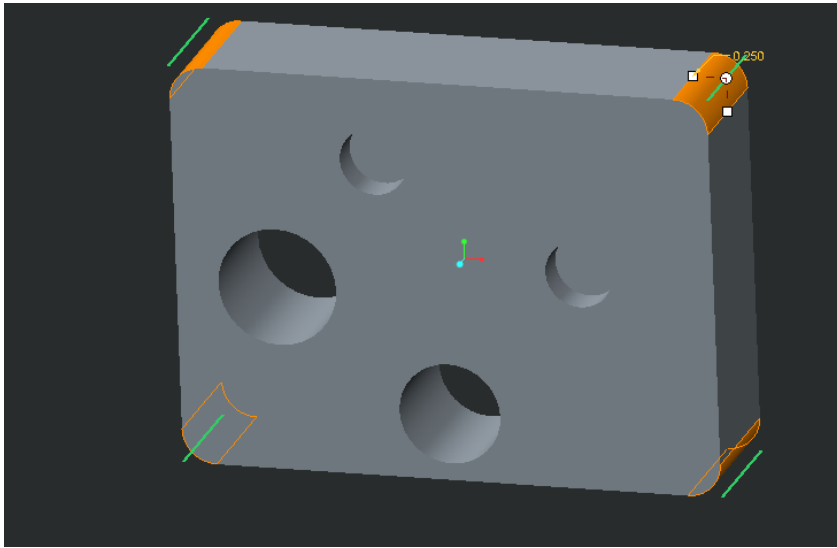


Tangent Round: Allows the user to select multiple edges to round

Sets the Radius of the Round or Fillet

Sets: Allow the user to select different edges and change the Radial size of different edges in one solid feature as an opposed to multiple.

4. To set all edges the same size at one time. Hold CTRL Key and select the following 4 edges. If the user does not hold the CTRL Key then then the user will have to set each Radius by opening the Set Tab and modifying each Set's Radius



5. Set the Radius of the Rounds to .25
6. Green Check to Finish the feature

SAVE PART

Design DWG 11, 12 and 13

Do not Start the Dimensioning until Demonstrated

Layout of a Part with Holes

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

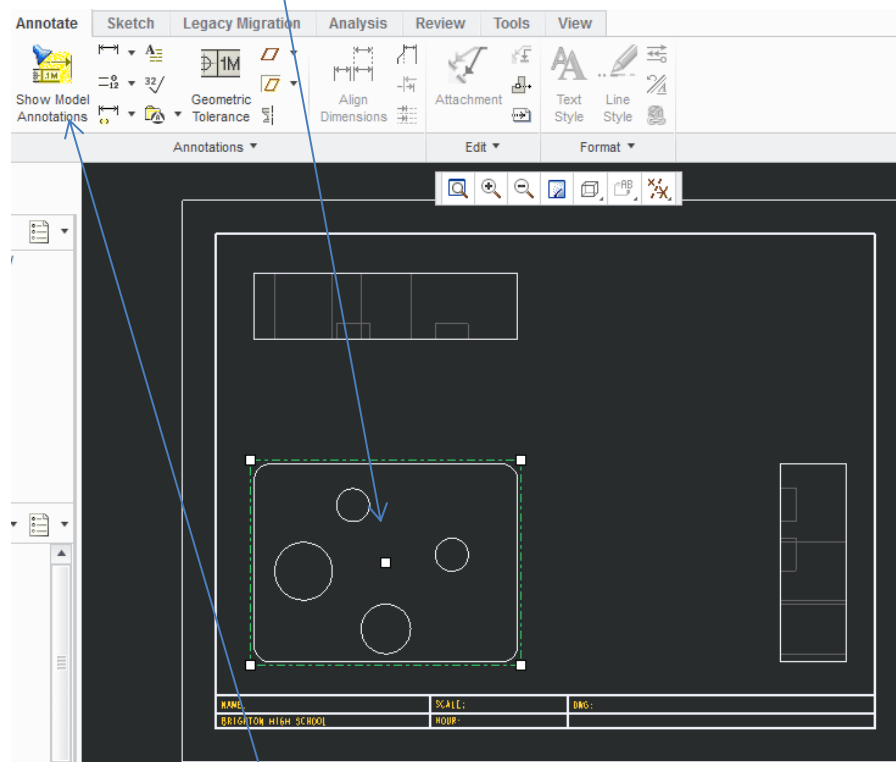
2. Place the front, top and right side views with hidden lines and the isometric without hidden lines.

Scale: FULL

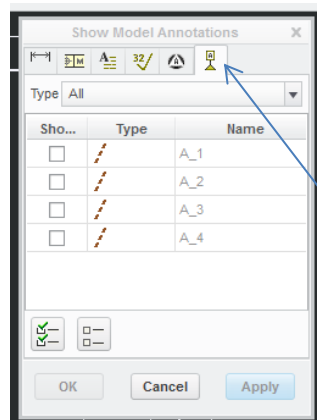
Be Sure to go to View Properties > View Display > Tangent Edges > None

3. Placing Centerlines

a. Select a view



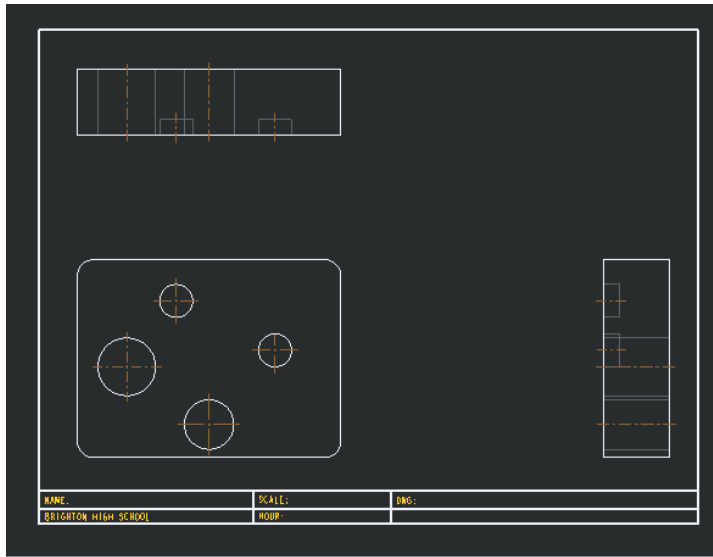
b. Select Show Annotations from the Annotate Tab > Select the last tab on the pop up menu



Tabs

1. Auto Dimension
2. GD&T Dimension: For Machining
3. 3D Annotation Tool
4. Surface Finish
5. Custom Symbols
6. Centerline

- c. Select all of the Centerlines Tab (Last Tab on the Pop-Up Menu) and click OK
- d. Repeat steps for the other two views. Finished Drawing should look as follows



4. Placing Dimensions

Holes need 3 Dimensions

- a. Diameter and Depth (If not all the way through)
- b. 2 Locators

Holes should always be dimensioned where you see the circle profile of the hole. Never the hidden lines

5. Placing Locators

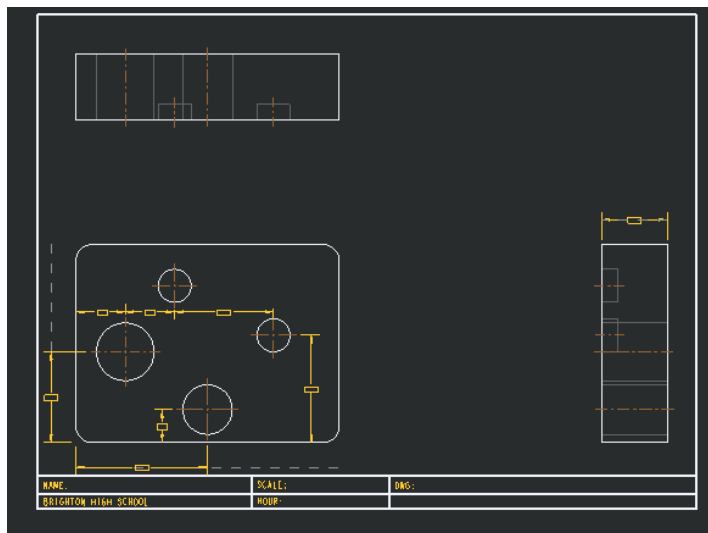
Setting Locators is the same as setting a linear dimension on edges.

Select the dimension tool and select either an 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 has 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 may be 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 spaced 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

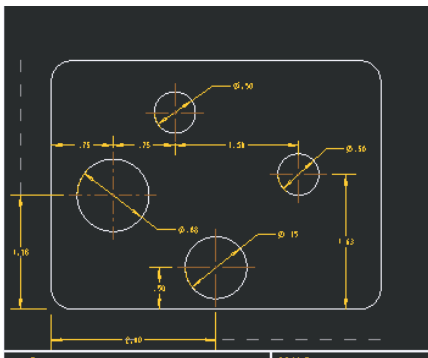


6. Placing Leader Line Dimensions

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

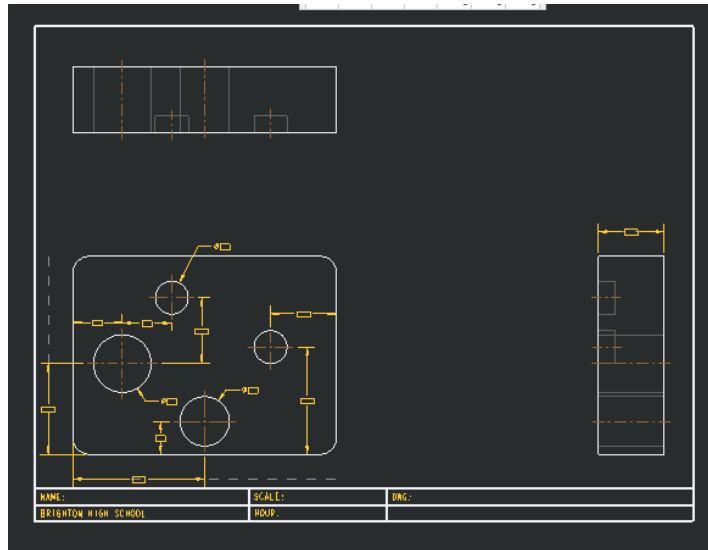
- To get Radius, use your linear dimension tool > Left Click once on the edge of the circle > Middle Mouse Button to Place
- To get Diameter, use your linear dimension tool > Left Click Twice on the edge of the circle > Middle Mouse Button to Place

7. Place the following Diameters for your Holes




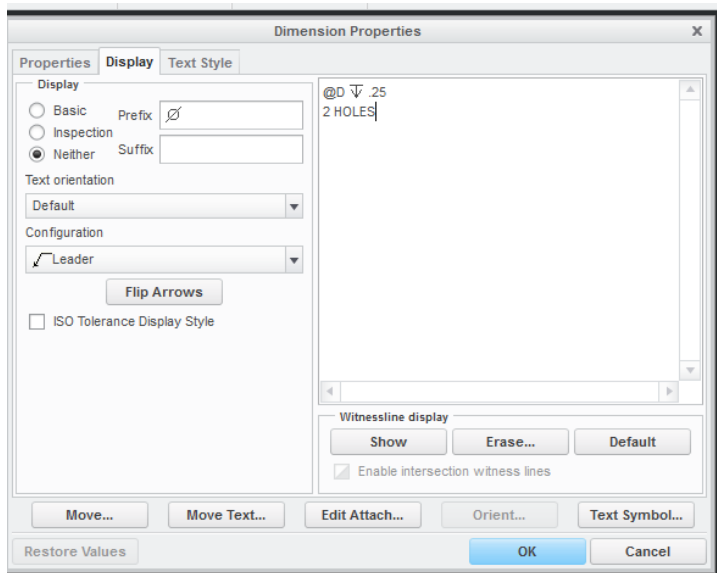
8. Adjust Arrow Style

- 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



9. 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. Double click on the Dimension with a Hole Diameter of 0.50
- c. This will take the user into the properties of the dimension
- d. Select the middle tab Display. This menu allows the user to add to the dimension note.
- e. 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 like 
- f. Type .25 after the symbol > Hit the Enter Key > Type on the second line 2 HOLES > Press OK
- g. This means the hole is a diameter of .50 a depth of .25 and there are 2 Holes that share these measurements.

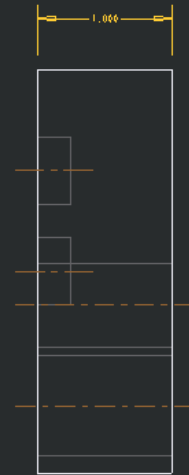
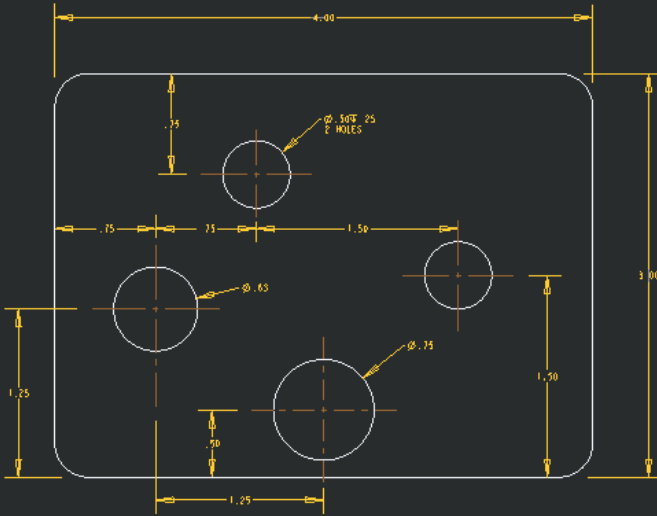
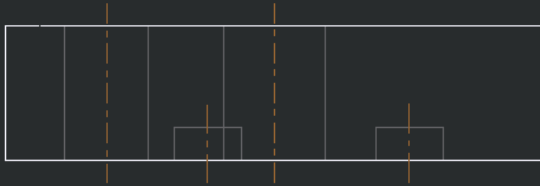


10. Dimension Fillets or Rounds

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



NOTE: ROUNDS R .25