NX Product and Manufacturing (PMI) (3D Annotation/Dimensioning)

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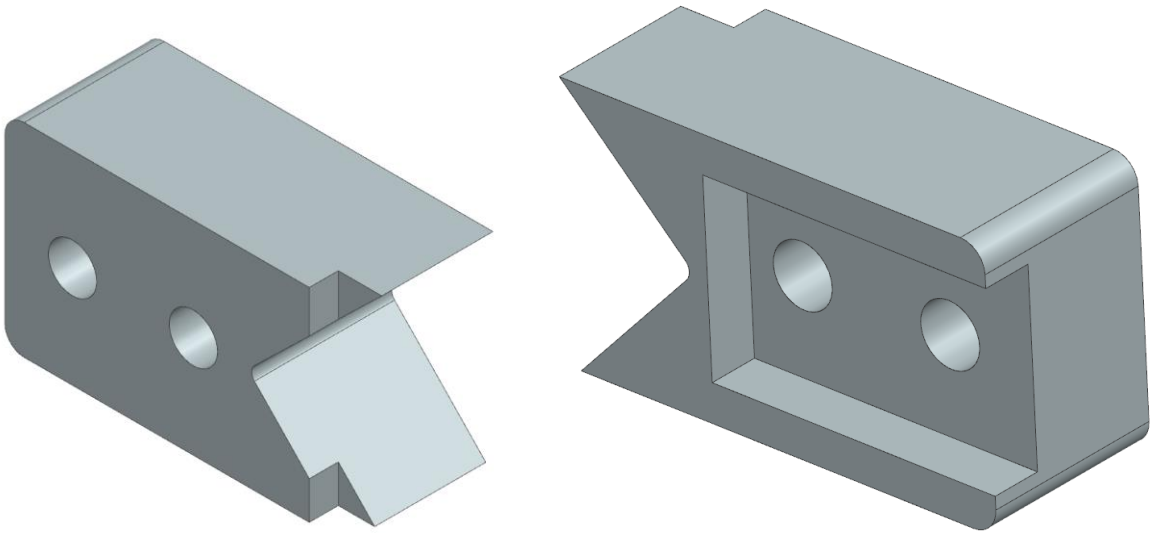
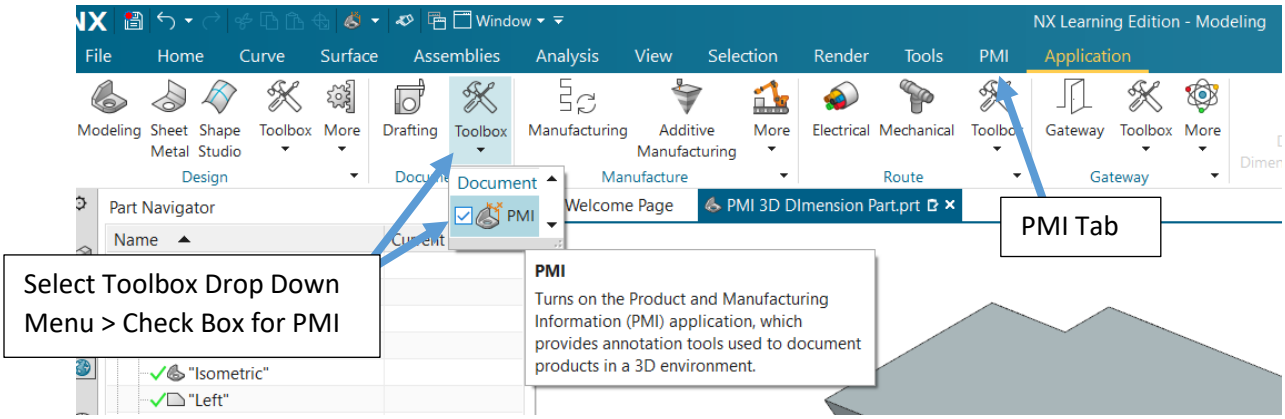
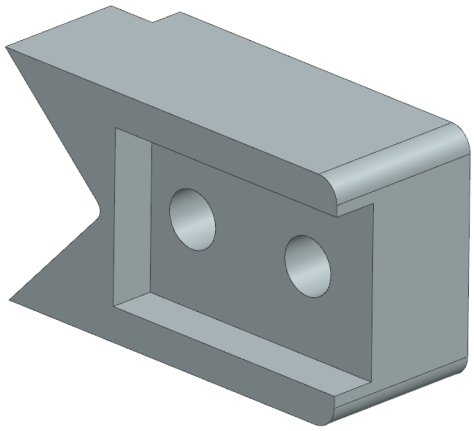
1. Open NX > Open the Design Part Shown Below from Extrude Tutorial and Assignments



2. Activate PMI Tool: Select Application Tab > Select Toolbox Drop Down Menu > Check Box next to PMI > This will add PMI Tab to workspace

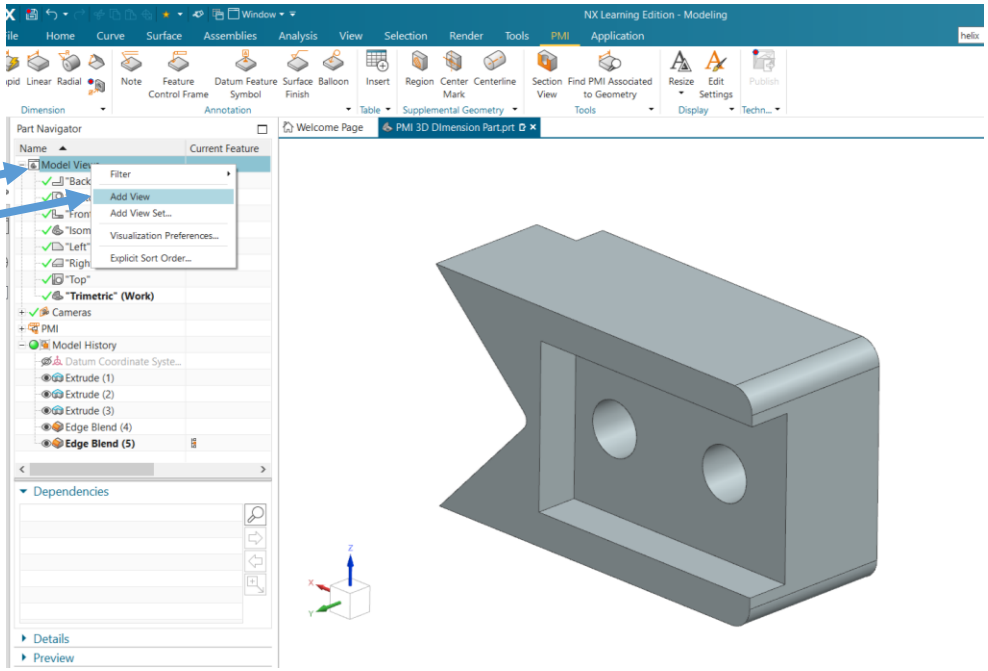


3. Creating Working View
    a. Rotate the Model to approximately the following Orientation



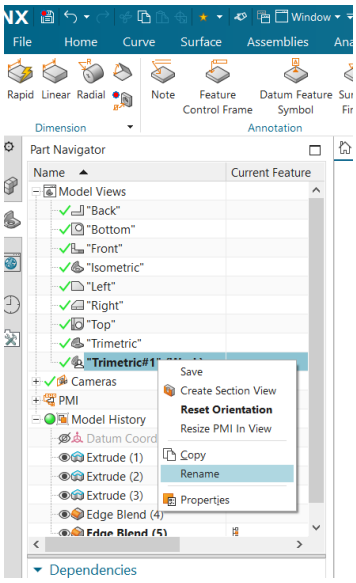
    b. In the Model Tree: Open Model Views by pressing the + Sign > Right Click on Model Views > Select Add View

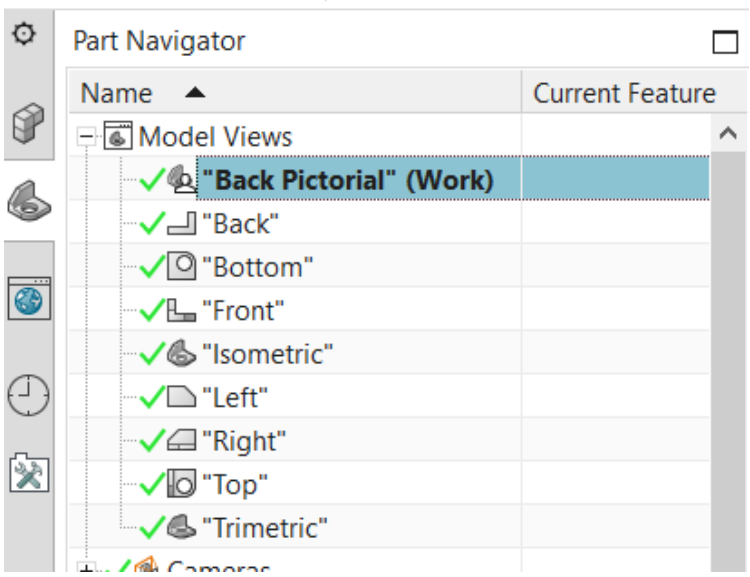
Right Click on Model Views > Select Add View



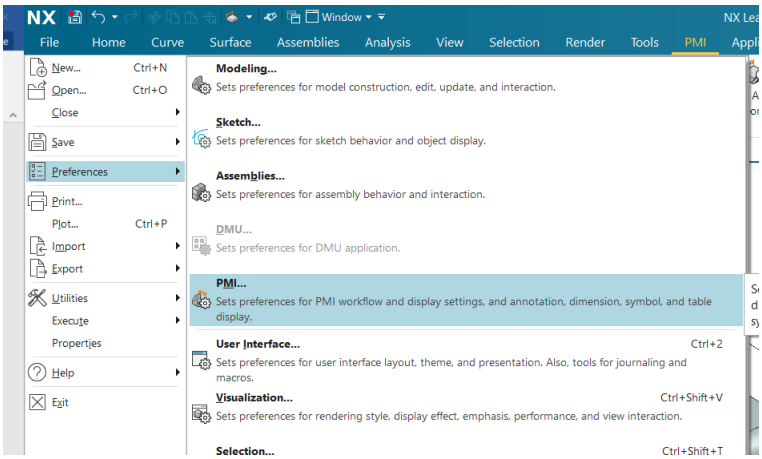
Notice View Name is the same as the active working view. In this case the active view was Trimetric.

    c. Rename View: Select Trimetric#1 (View just Created) > Right Click > Select Rename > Rename to Back Pictorial



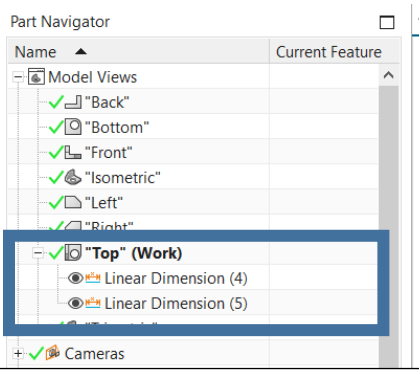


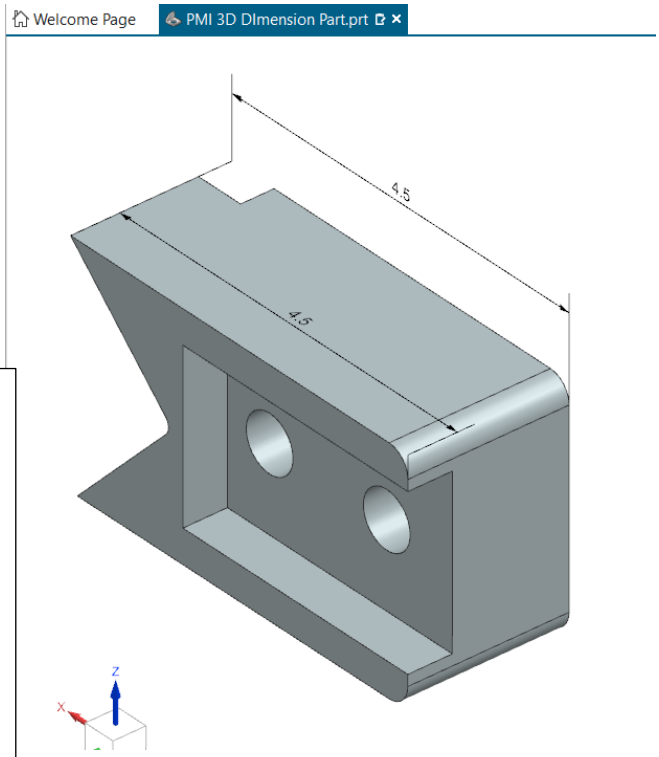
4. PMI Settings (Text Height, Arrow Size, etc.)
    i. Select File Tab > Preferences > PMI



    ii. Set the following Settings
        1. Common > Lettering > Height = .0625
        2. Common > Line/Arrow > Arrowhead > Length = .09
        3. Common > Extension Line > Gap > .125
        4. Common > Extension Line > Format > Extension Line Overhang = .09
        5. Dimension > Text > Units > Check Show Trailing Zeros
        6. Dimension > Text > Orientation and Location > Orientation > Text Aligned to Dimension
        7. Dimension > Text > Dimension Text > Height = .0625

5. Select PMI Tab
    a. Considerations
        i. Dimensions: When dimensioning on a part the user can determine what face a datum/face the dimension will be placed on. The placed PMI Dimensions can transfer over into a standard layout. The dimensions placed will be attached to a specific Model View (Front, R.S, Top, Isometric, etc.)





Notice

1. Both Dimensions are attached to Top Datum which is the Working (active datum)
2. The placement of the Overall Length of the part.

Placed

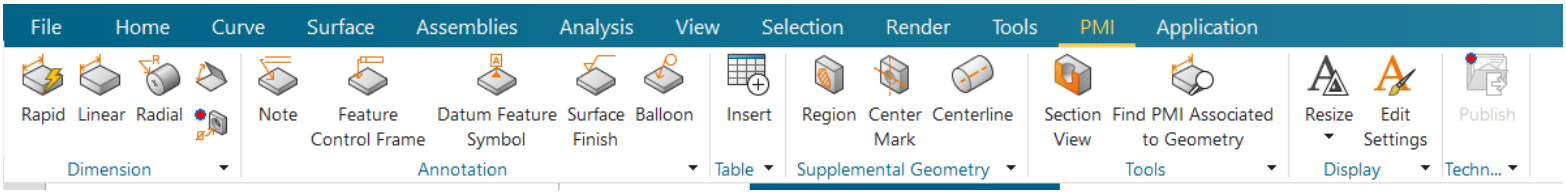
1 is located on XZ Plane (Front)

1 is located on XY Plane (Top)

PMI Dimension Uses

1. Traditional Drawing: If the user is planning to use these dimensions in a traditional layout planning of what view each dimension is placed on will be important.
2. 3D Solid Model: Quick view of Dimensions without having to Edit Features to Find Sizes.

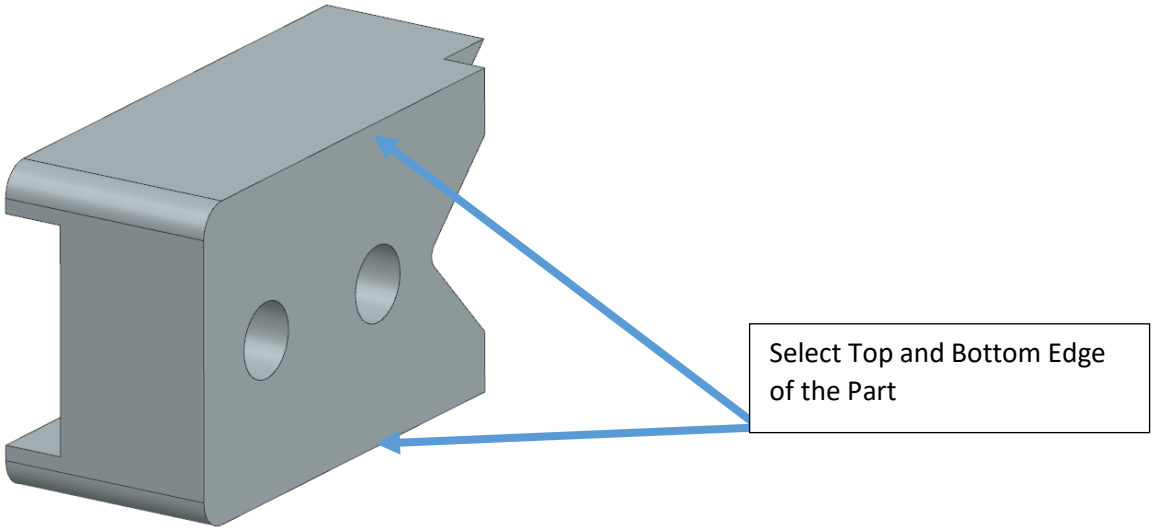
b. Menu: Tools are similar to standard annotate (dimension) tools found in a layout

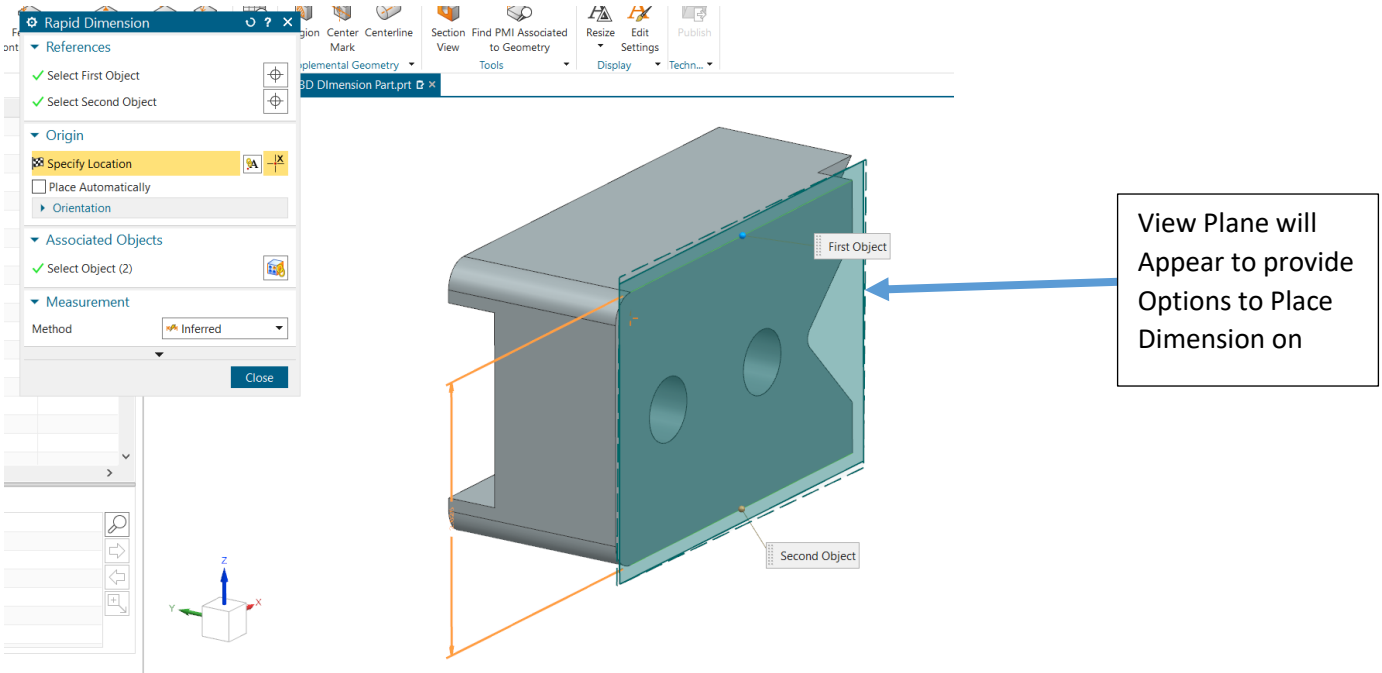


c. Linear Dimension

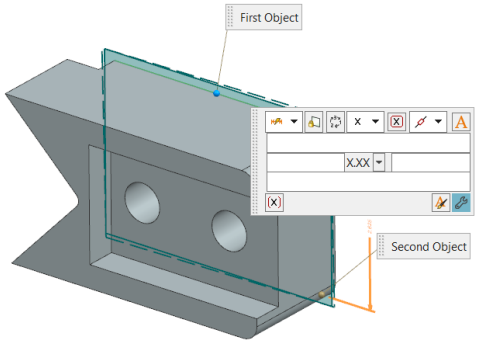
i. Overall Height

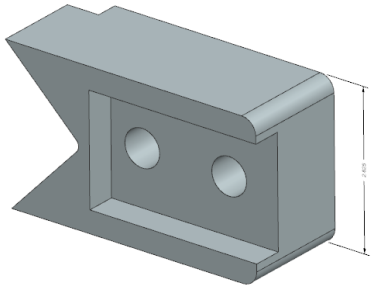
1. Activated Model View > Back Pictorial (Double Click to Activate)
2. Select Rapid Tool or Linear Tool > Select the Following Edges
   NOTE: May have to rotate view to select Edges





3. Rotate View Approximately Back to Back Pictorial Perspective to provide rough placement distance from the part > Left Click to Place the Dimension



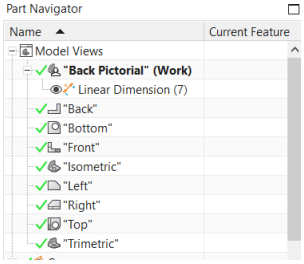


Approximately Rotate to Back Pictorial Angle

Left Click to Place Dimension

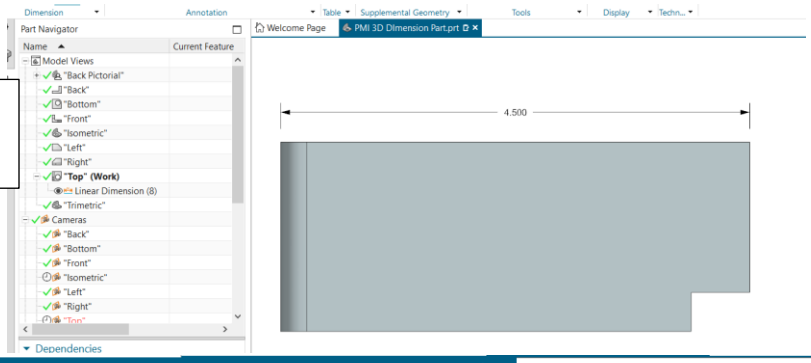
NOTE: To move Dimension: Left Click On Dimension > Hold Left Mouse Key Down > Drag Mouse to Desired Location

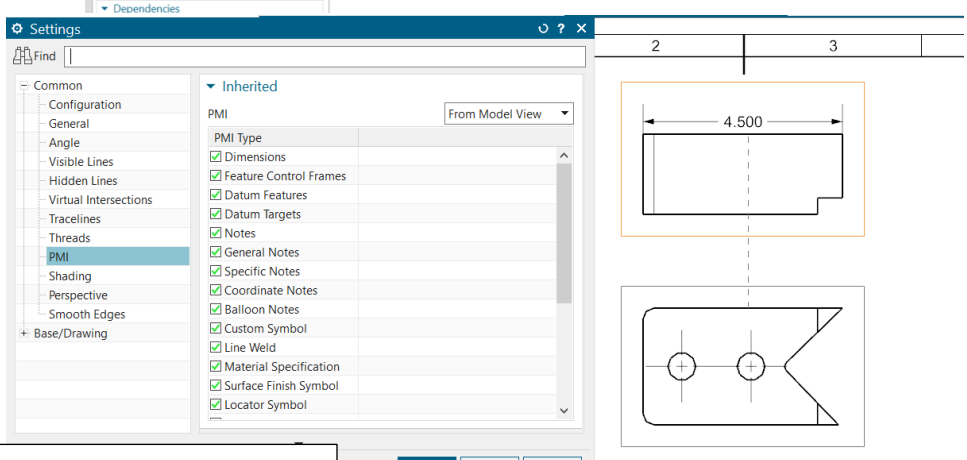
4. The New Dimension is added to the Working View



NOTE: If the Dimension is placed on a standard Orthographic View the Dimension will appear in the working drawing layout

Overall Length Dimension Placed on Top Model View

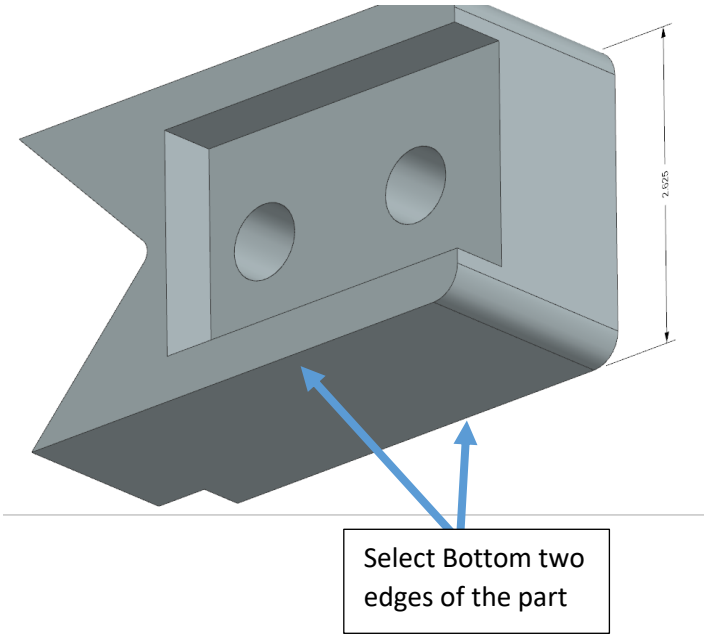




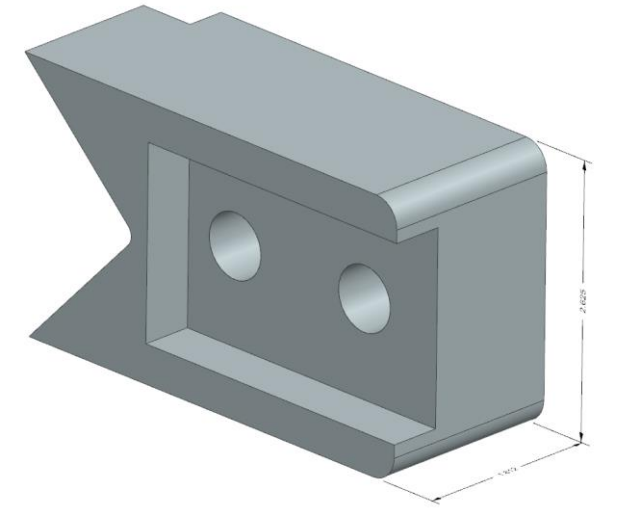
NOTE: PMI Dimensions needs to be activated via Settings Menu. Once Activated they will appear on the View that they were attached too

ii. Overall Depth

1. Back Pictorial Activate
2. Select Rapid Tool or Linear Dimension Tool
3. Select the Following 2 Edges (Note: May have to rotate to see edges
4. Rotate Image Approximately to the Back Pictorial Angle
5. Place Dimension so arrows are aligned with Overall Height Dimension

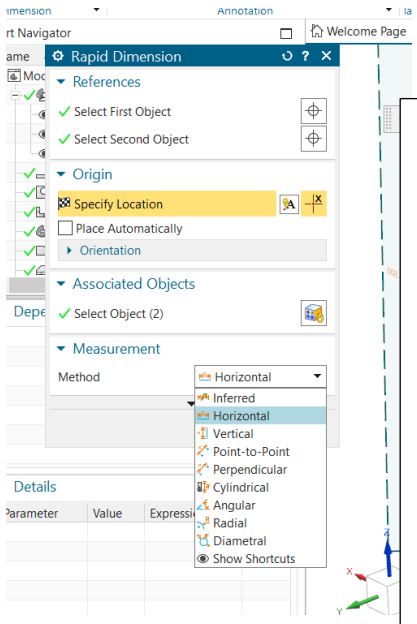


Select Bottom two edges of the part



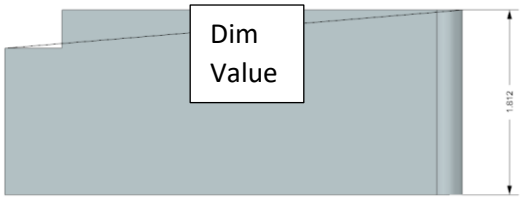
iii. Overall Length

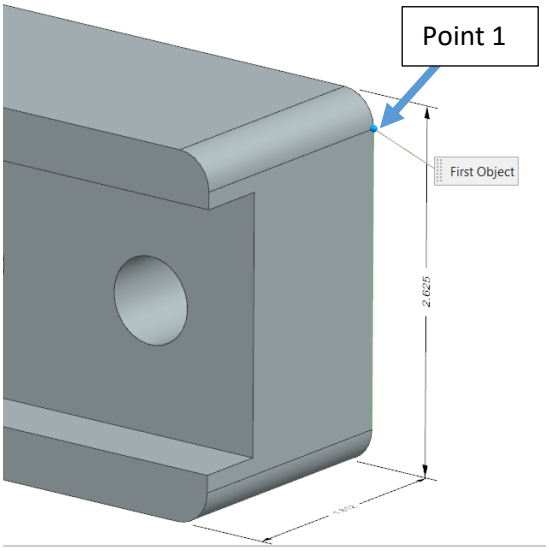
1. Select Rapid Dimension Tool or Linear Dimension Tool > Change Measurement Method = Horizontal (This will make the dimension aligned to selected plane and not measuring shortest distance between points) Select the Following 2 Points.
   NOTE Order Matters in the Selection. The first selection point will determine where the view plane will be placed. Desired Location of View plane for this dimension is flush with the back side of the part

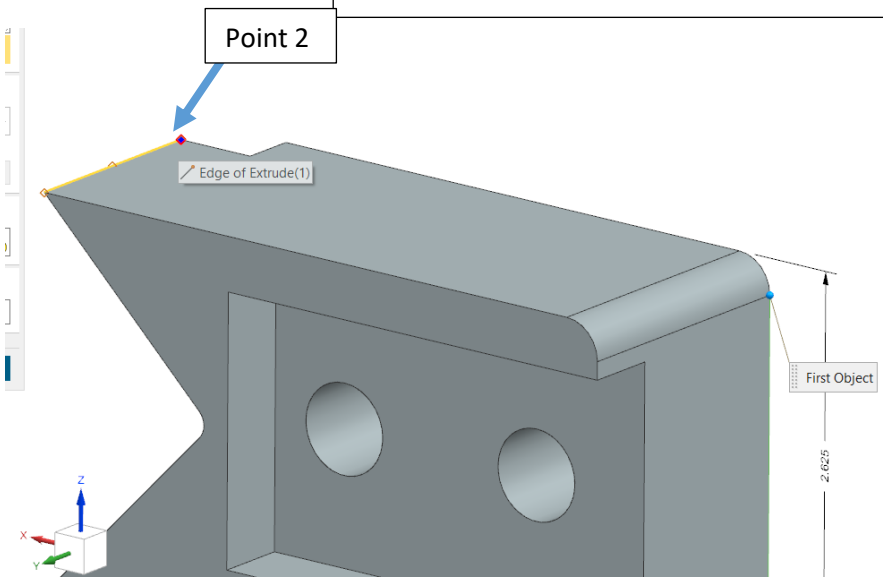


Change Measurement Method to Horizontal

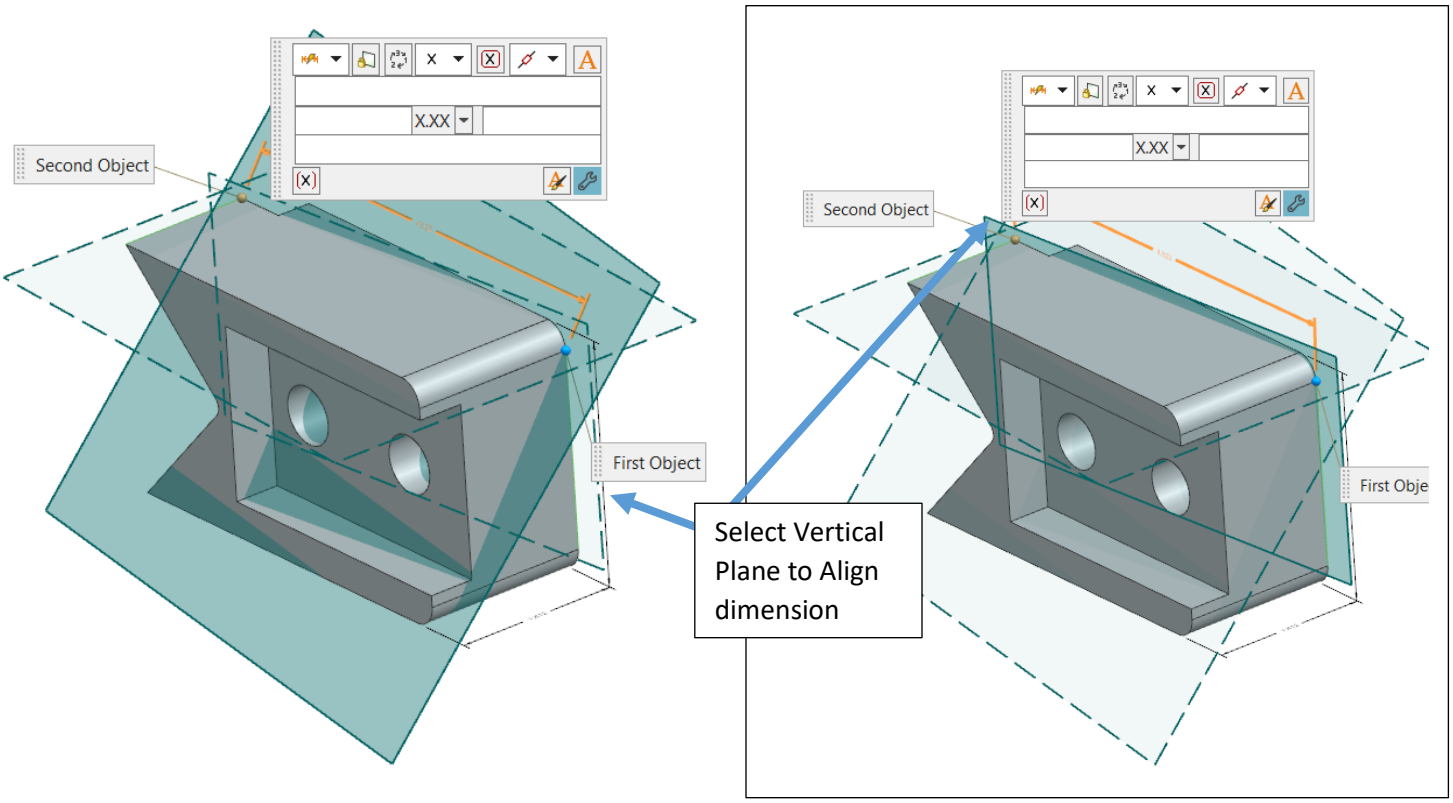
i.e. Without Change Dimension Measures shortest distance between points



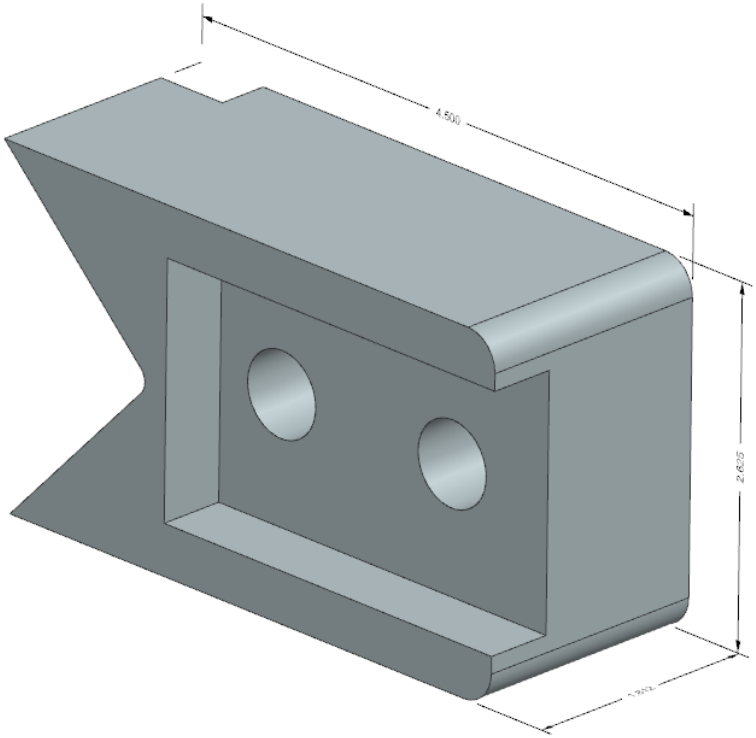




2. Planes will Appear to align the dimension to > Select the Vertical Plane that is Flush with the Back Surface of the Part > Left Click to Place Dimension

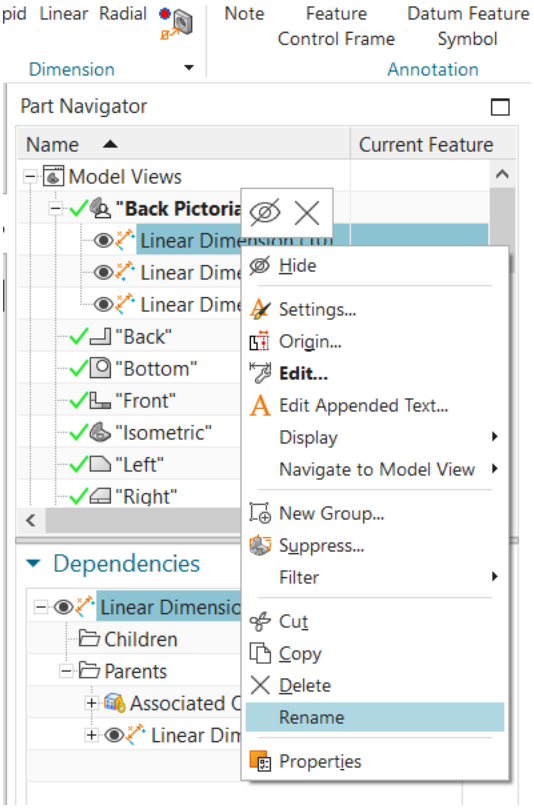


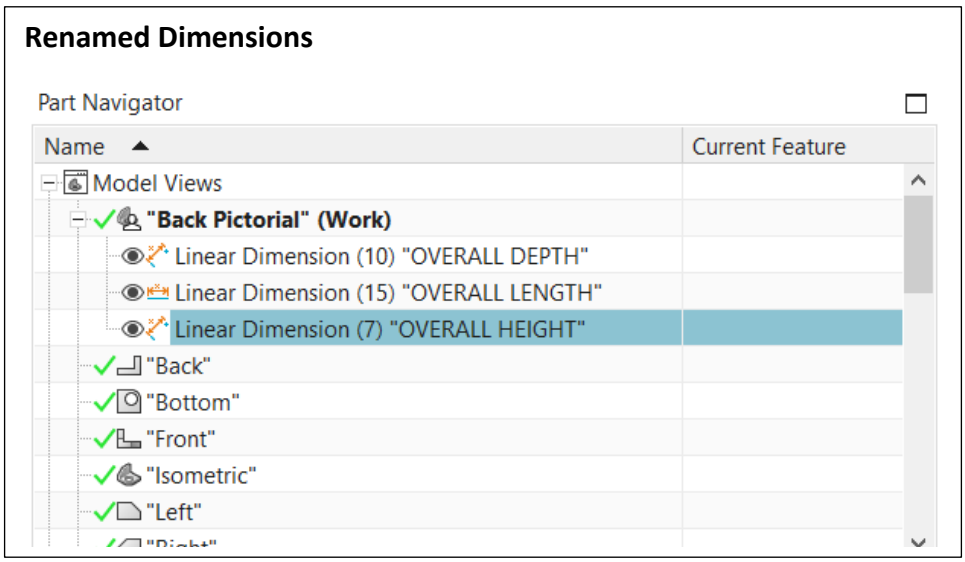
Overall Dimension Placements



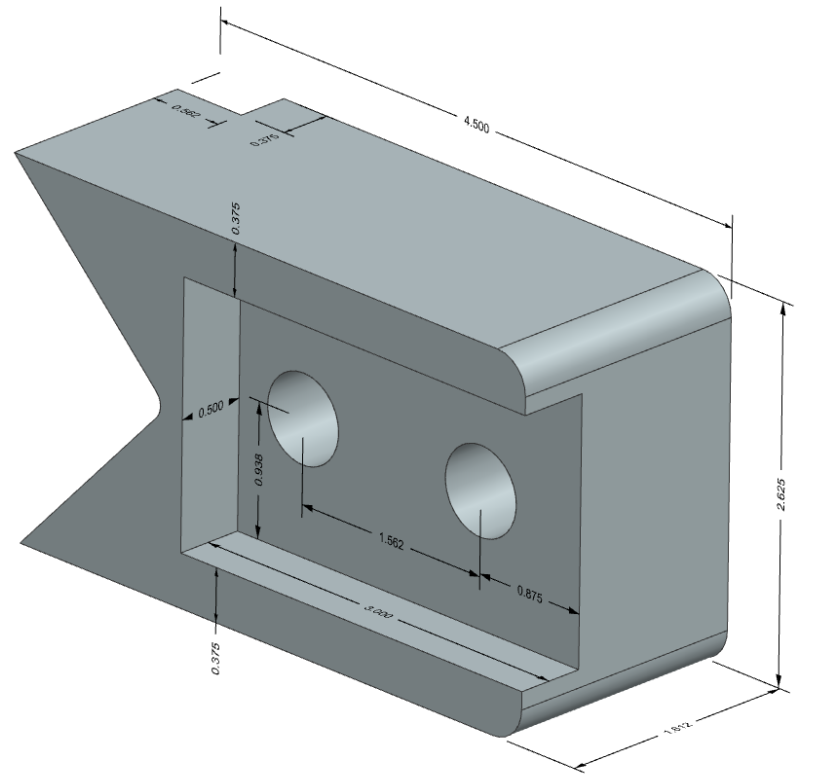
iv. Rename Dimensions in Model Tree

1. Select Dimension > Right Click > Select Rename > Rename Dimensions to
    a. Overall Height
    b. Overall Depth
    c. Overall Length

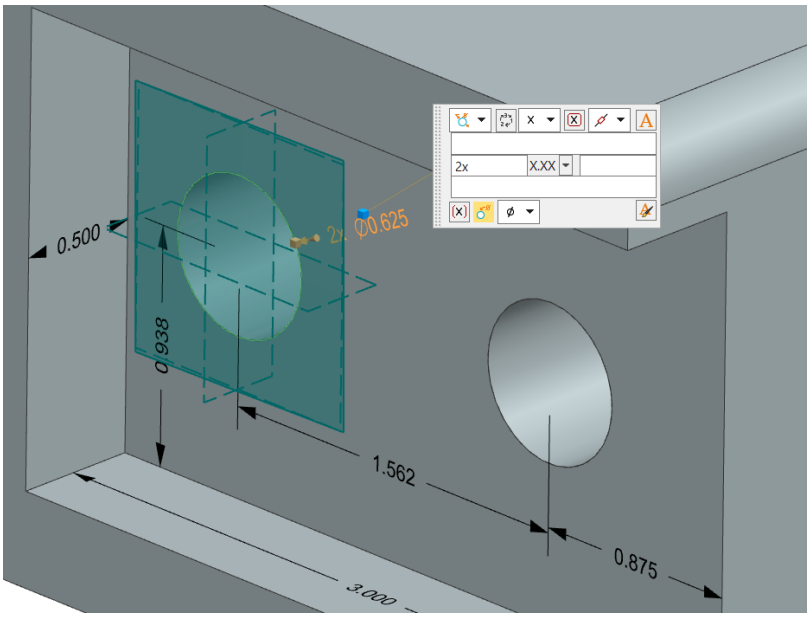


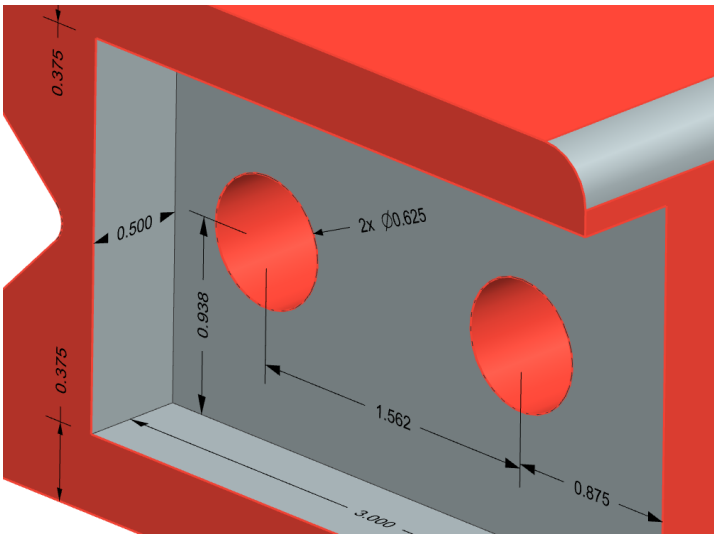


v. Place the Following Dimensions
NOTE: To place locators for the holes select the Center Point of the hole

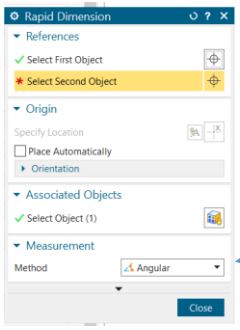


vi. Diameter Dimension: Select Rapid Dimension Tool > Select the Edge of the Circle > Leader Line will Appear > Place Dimension > Adding a Note: Select the Diameter Dimension Place the Following Note Before the Diameter



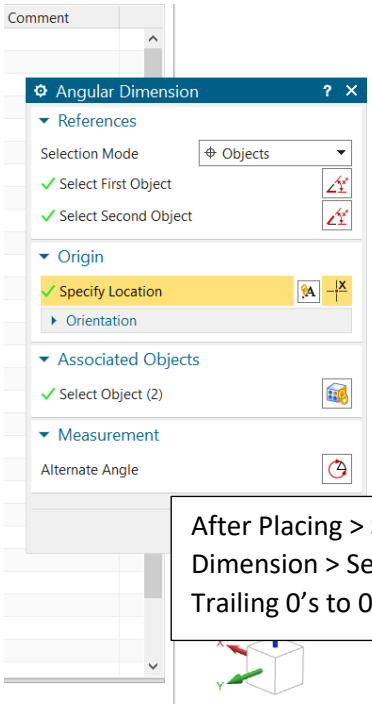


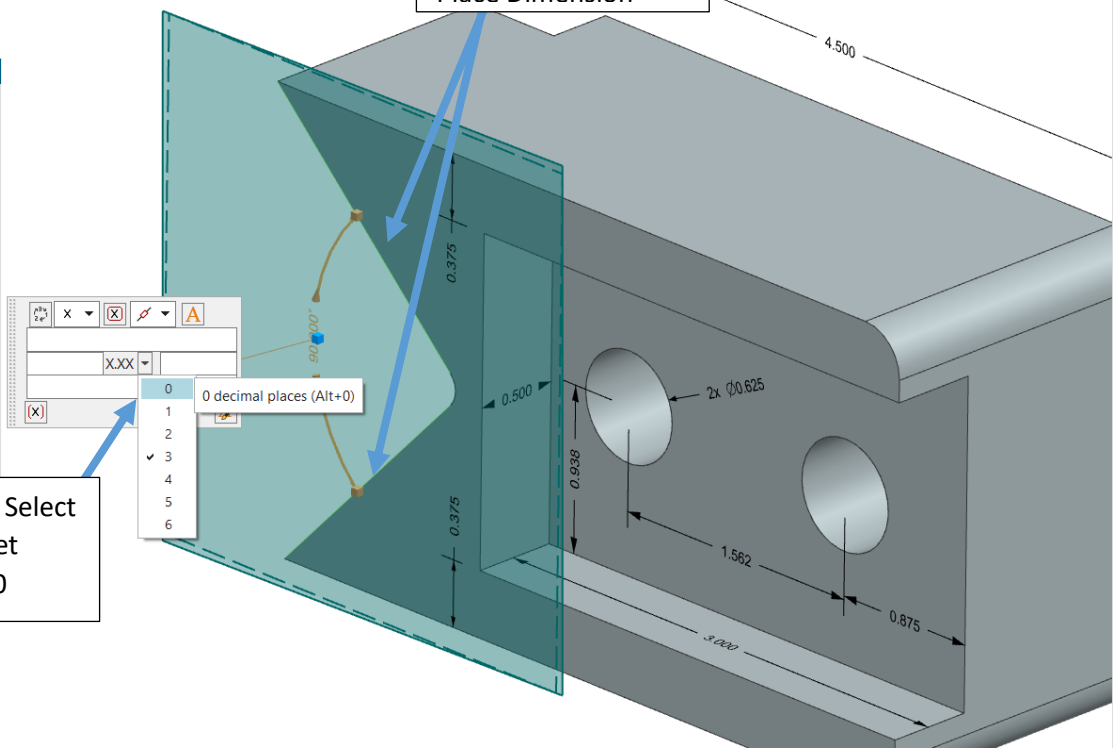
vii. Setting Angle: Select Rapid Dimension Tool > Set Measurement Method to Angular > Select the 2 angle Edges of the Part > Place Dimension > Select Dimension > Remove the 0's from the Dimension



Change Measurement Method to Angular

Select 2 Edges > Place Dimension

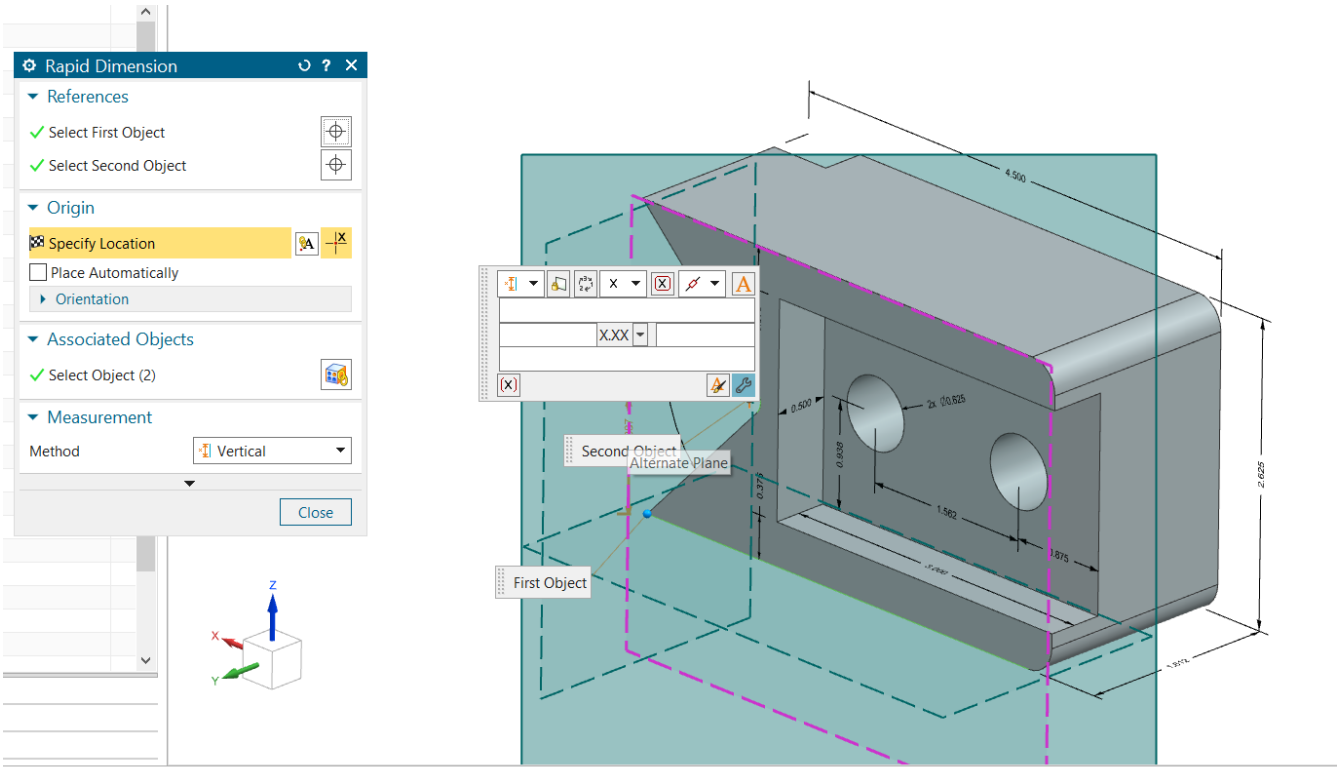




After Placing > Select Dimension > Set Trailing 0's to 0

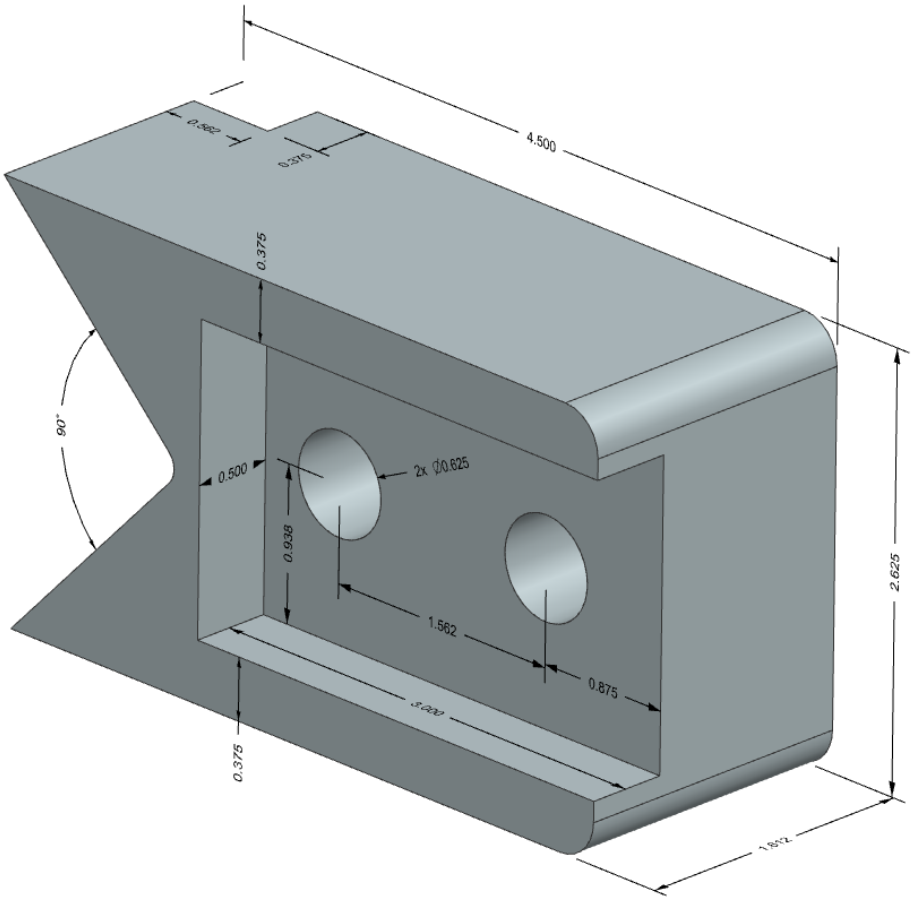
viii. Angular Dimension Placement

1. Select Rapid Dimension Tool > Set Measurement Method to Vertical Select Bottom Edge of Part > Select Center Point of Fillet > Select Vertical Plane to Place on > Place Dimension > Adjust Dimension placement so linear and Angular Dimension DO NOT cross

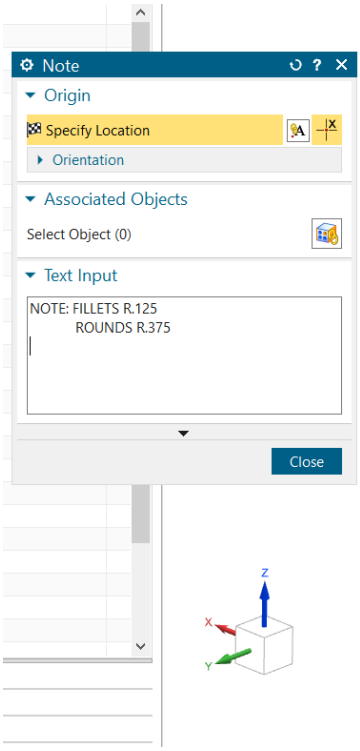


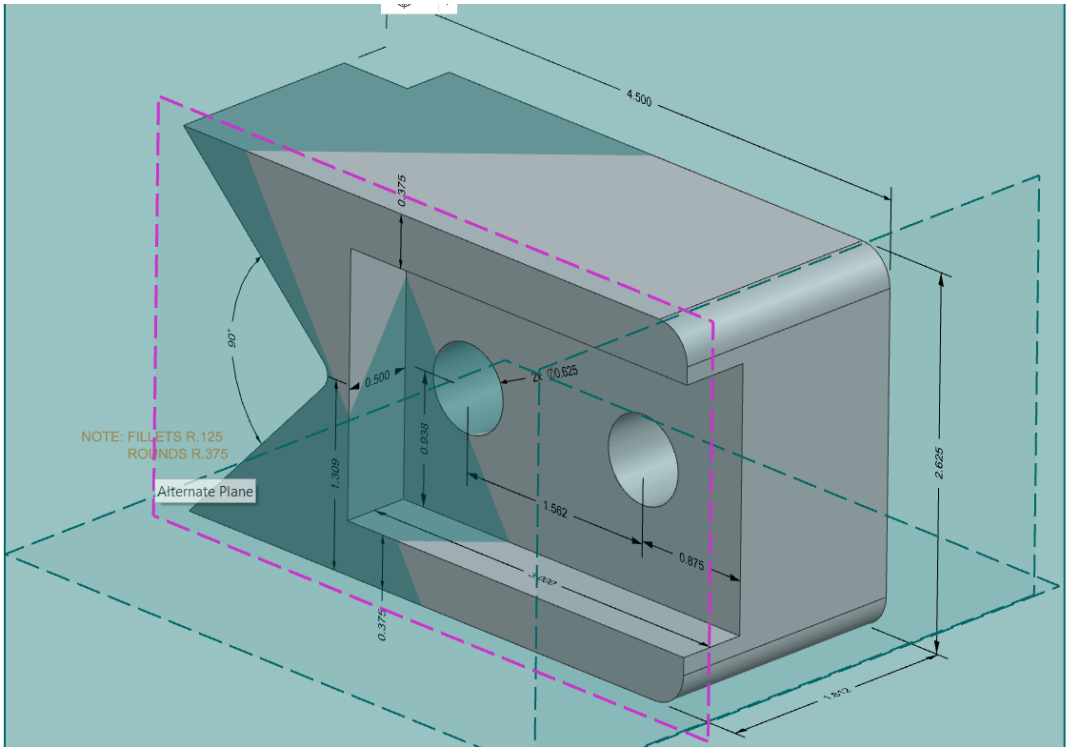
Alternate Plane

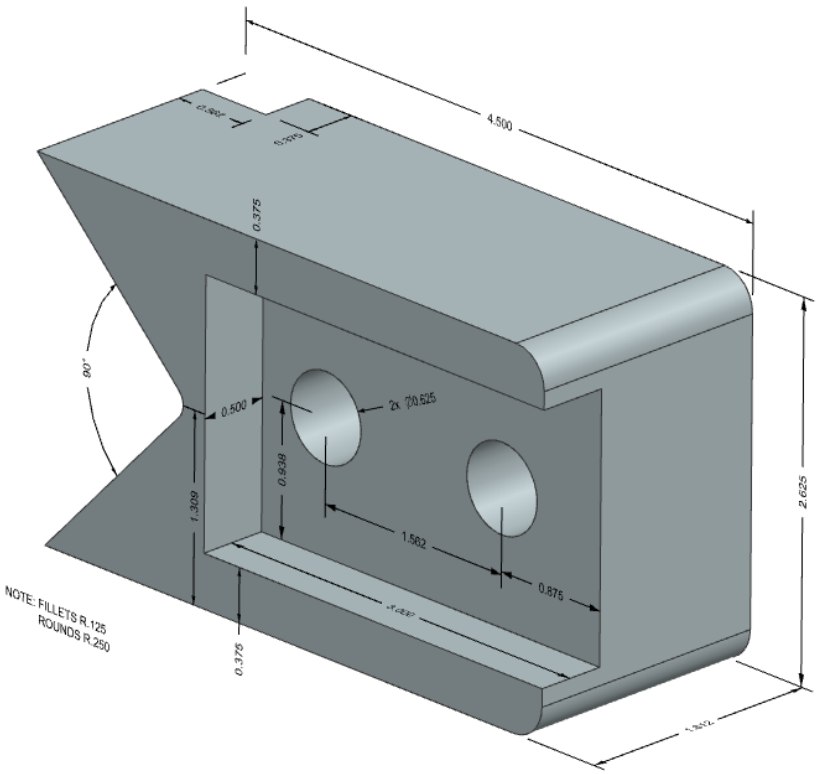
Sample Placement



ix. Shop Note: Select Note Tool > Select Plane to Place Shop Note on > Type the following note
NOTE: FILLETS R.125
ROUNDS R.250
Place Note in Desired Location

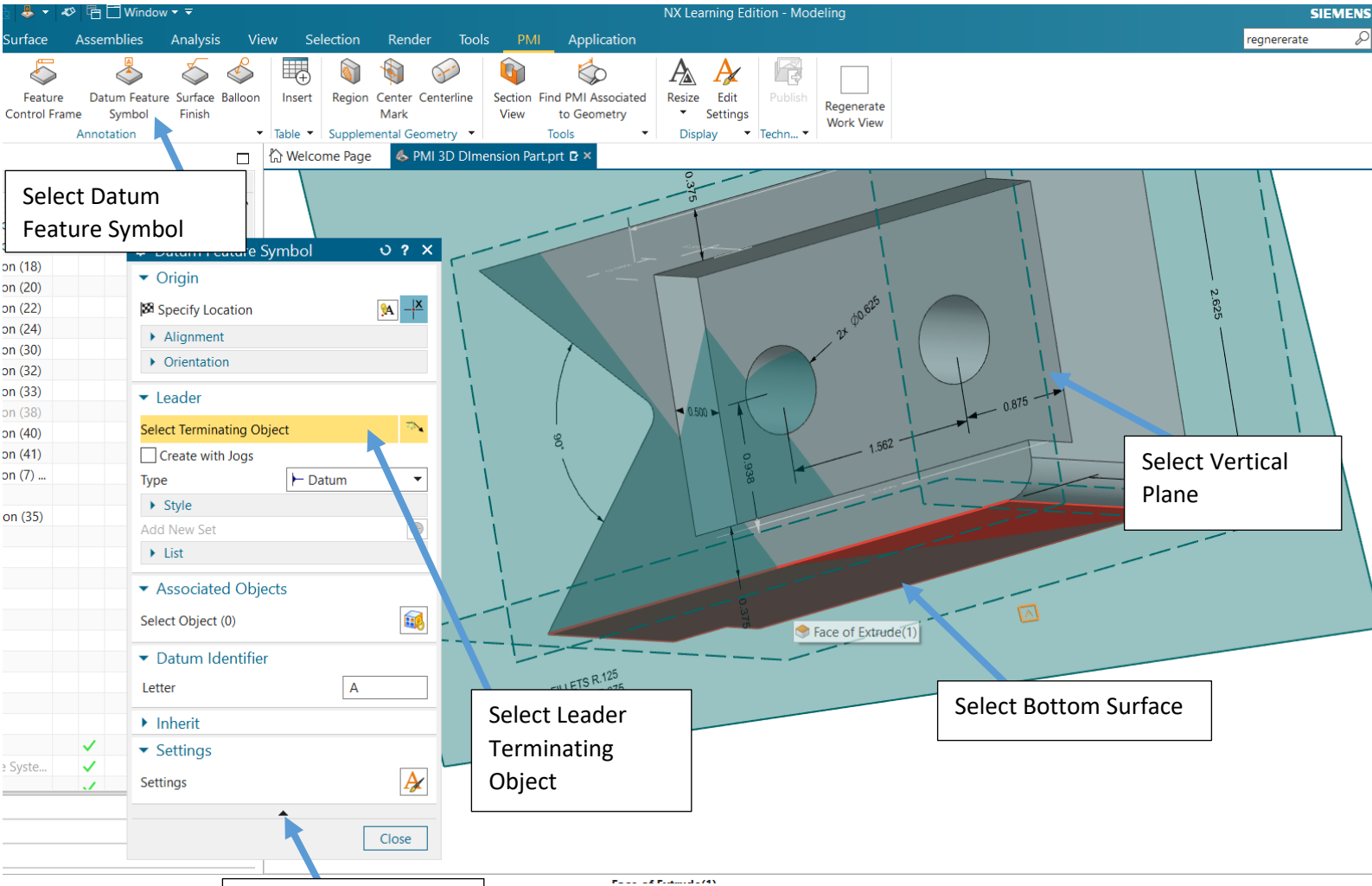


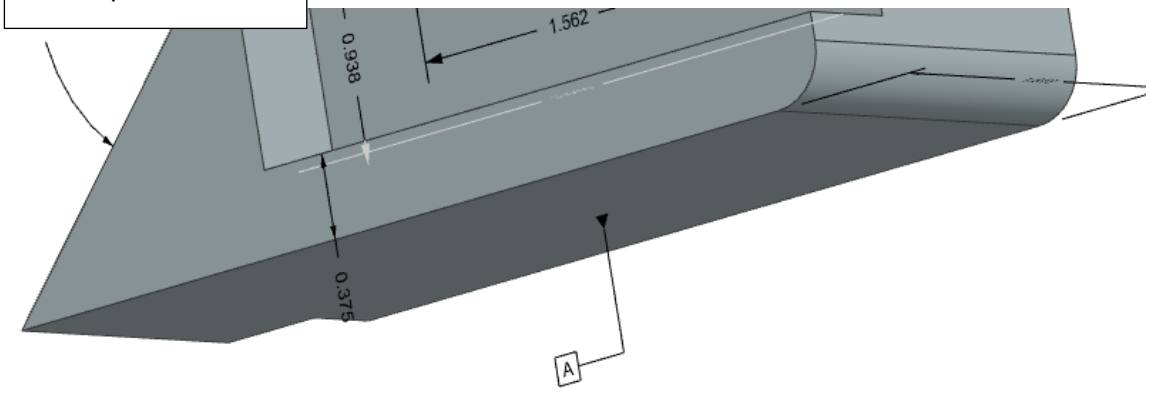




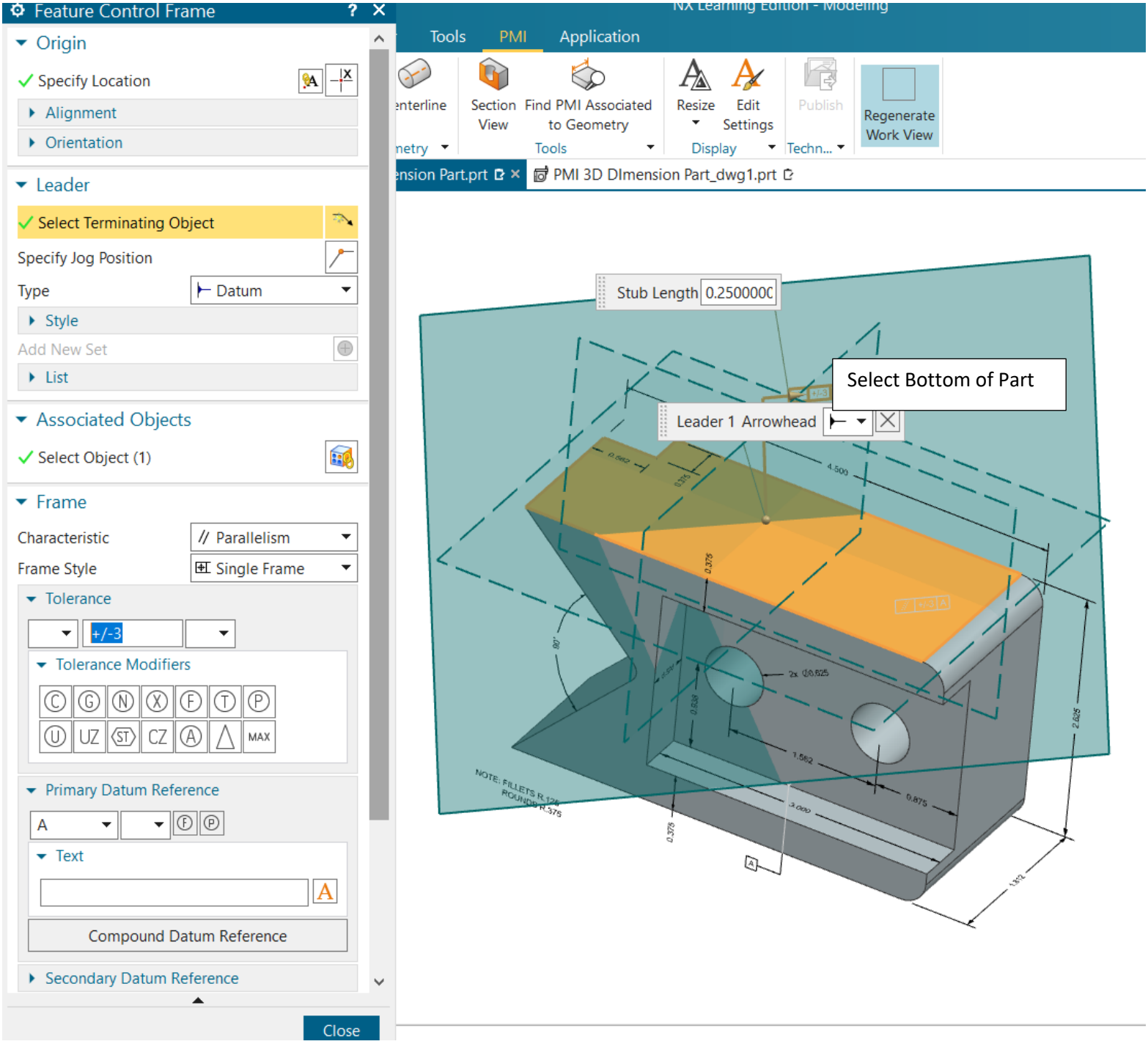
x. Geometric Dimensioning & Tolerance

1. Place Datum Marker: Bottom Surface of Part will be Primary Control Surface for Reference Select Datum Feature Symbol > Select Vertical Placement Plane > Press the Down Arrow to open more Options in the Dialogue Box > Select Leader Terminating Object Option in Dialogue Box > Select Bottom Surface of the Part

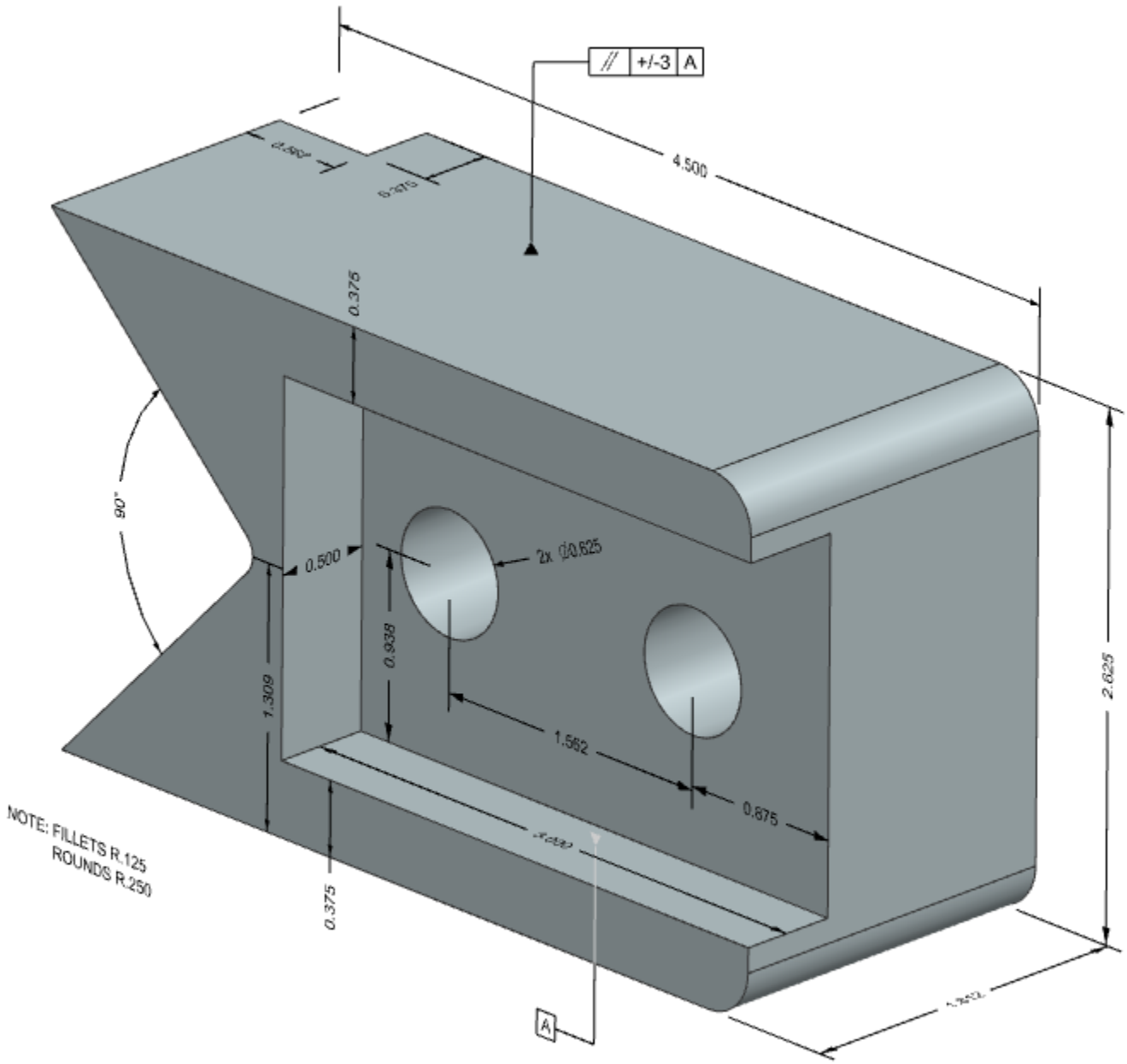




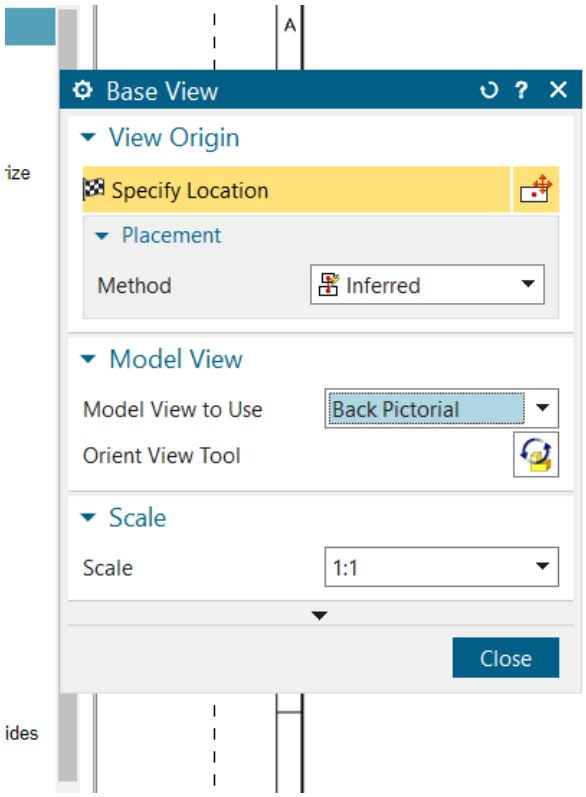
2. Control Box: Control Box allows the user to setup Tolerance Parameters when compared to a Datum Surface. The For this exercise we will be placing a tolerance for the top surface being parallel to Datum A
    a. Select Feature Control Frame Tool > Change Leader Type to Datum >Select Leader Terminating Object > Select Object > Select Top Surface > Set the Tolerance Dimension Values as shown below.
    The Dimension is reading Parallel Symbol to Datum A +/- 3 Degrees

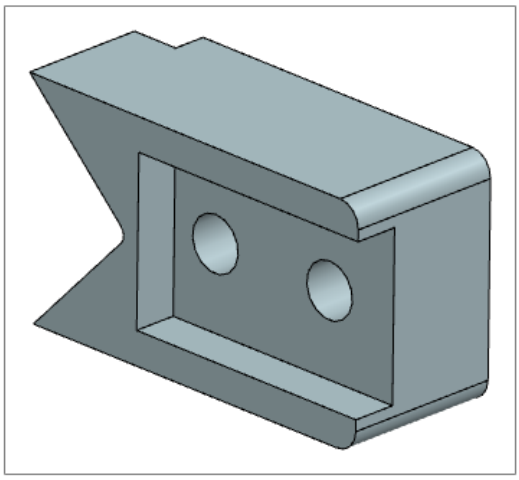


Final Part


// +/-3 A
4.500
0.375
90°
0.500
2x ⌀0.625
0.938
1.309
1.562
0.675
2.625
0.375
A
NOTE: FILLETS R.125
ROUNDS R.250

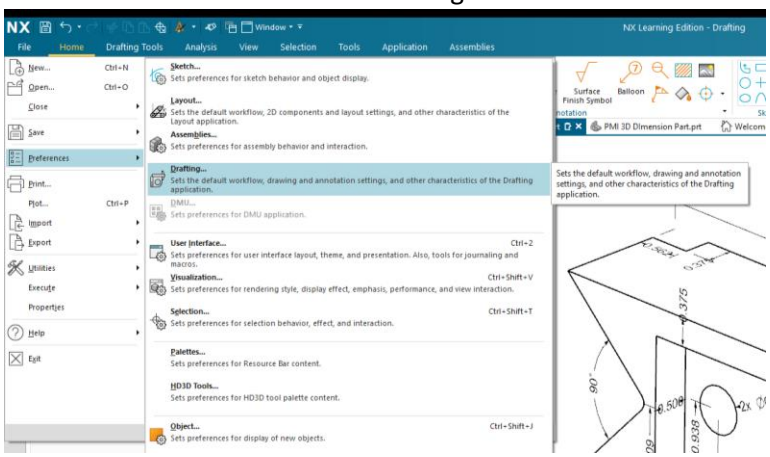
6. Place on Border and Title Block
    a. Select New > Select Drawing Tab > Select B-Size Title Block > Select Back Pictorial View > Place View > Select Close





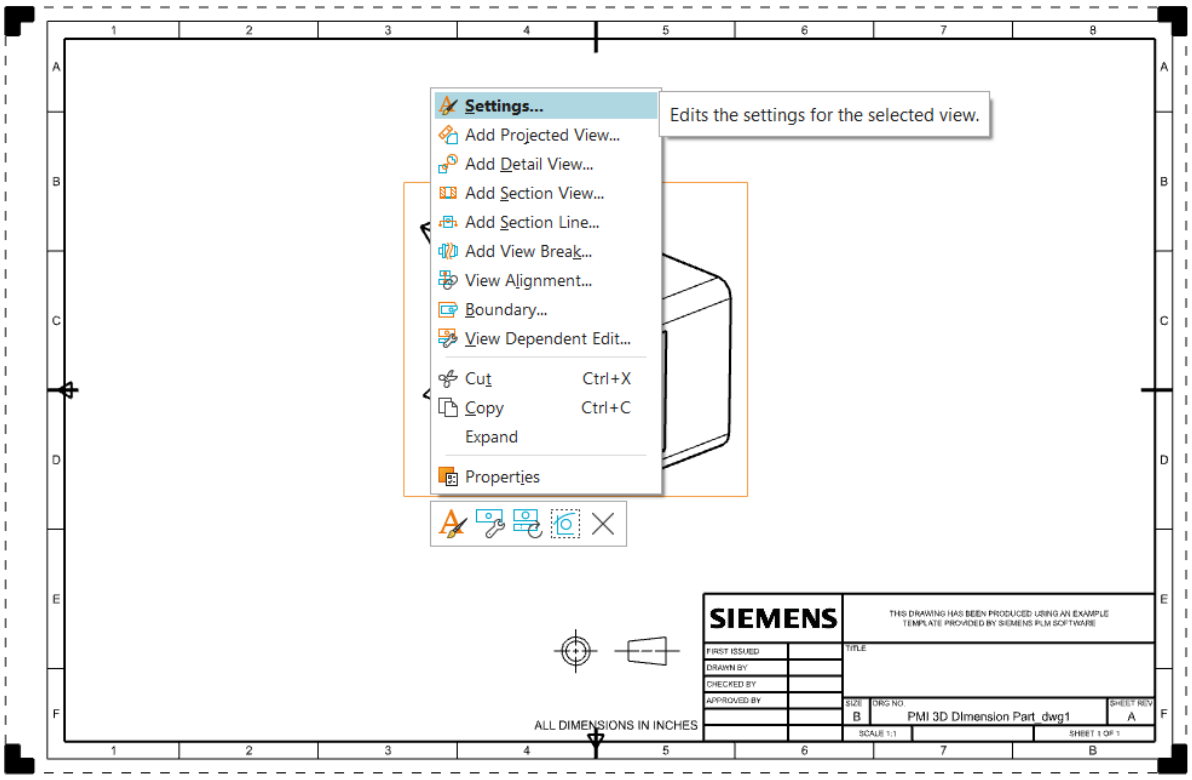
    b. Dimensioning Settings

        i. Select File > Select Preferences > Select Drafting

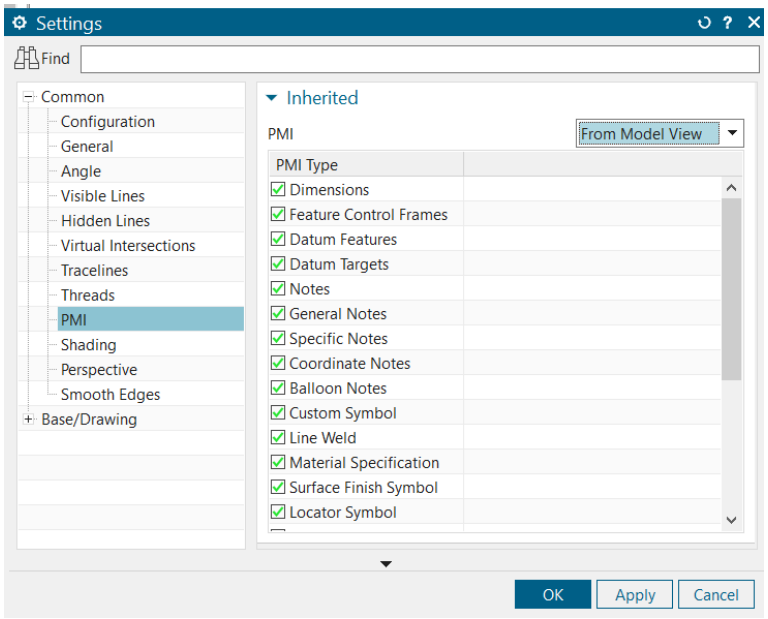


        ii. Change the following
            1. Common > Lettering > Height = .0625
            2. Common > Line/Arrow > Arrowhead > Length = .09
            3. Common > Extension Line > Gap > .125
            4. Common > Extension Line > Format > Extension Line Overhang = .09
            5. Text > Units > Check Show Trailing Zeros
            6. Text > Orientation and Location > Orientation > Text Aligned to Dimension
            7. Text > Dimension Text > Height = .0625
            8. Drawing View > Workflow > Border > Uncheck Display

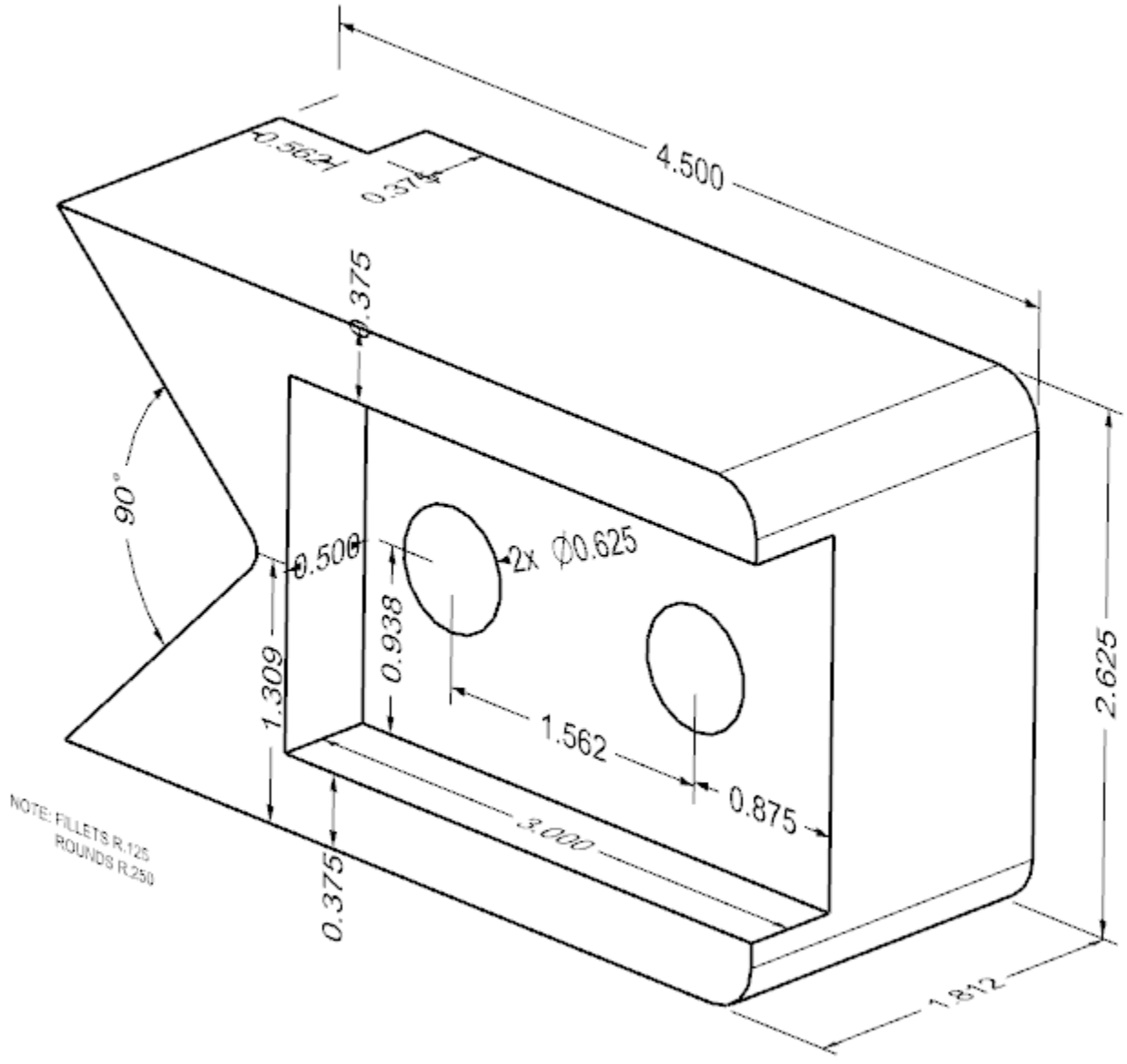
c. Select View > Right Mouse Button > Select Settings



d. Select PMI > Change PMI to From Model View > Select OK
NOTE: User can turn on/off certain types of dimensions if desired

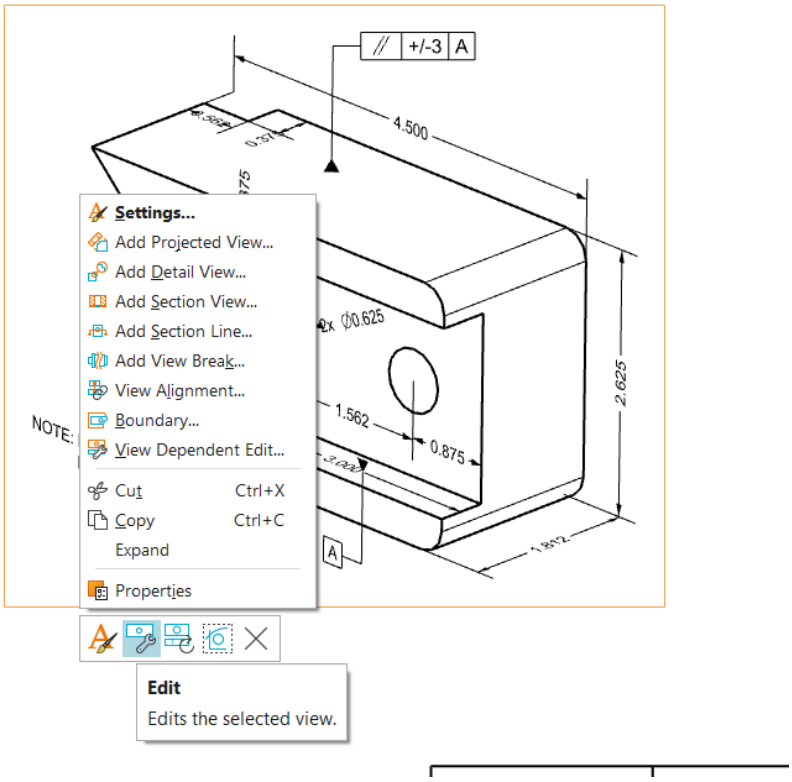


e. Dimensions Appear

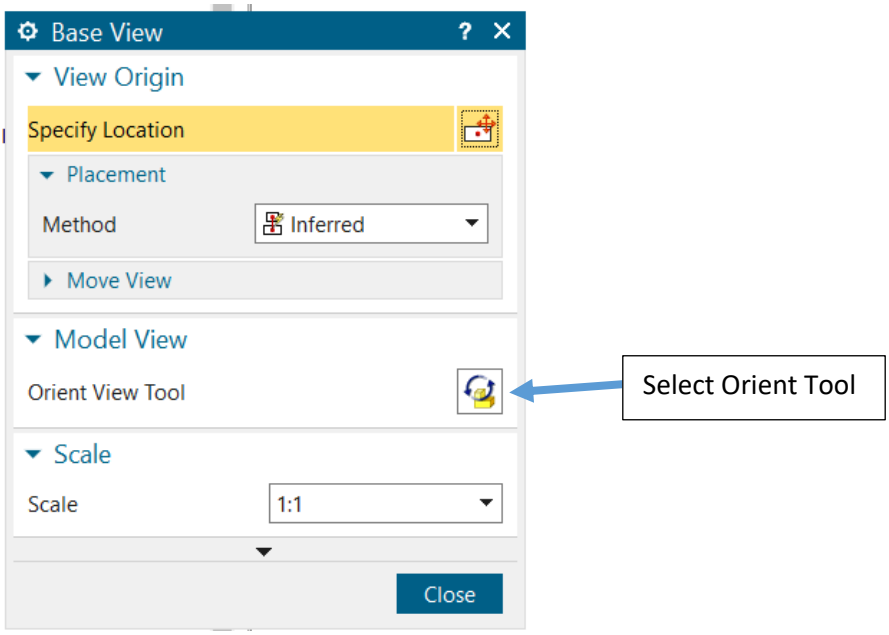


f. Rotating View to Orthographic

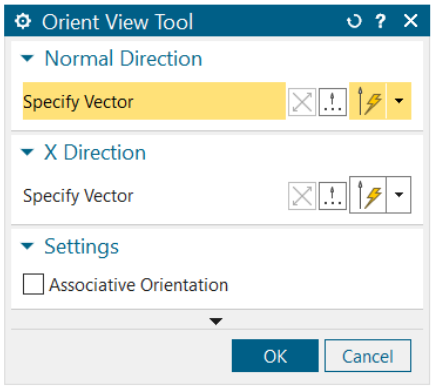
i. Select View Outline > Right Click > Select Edit

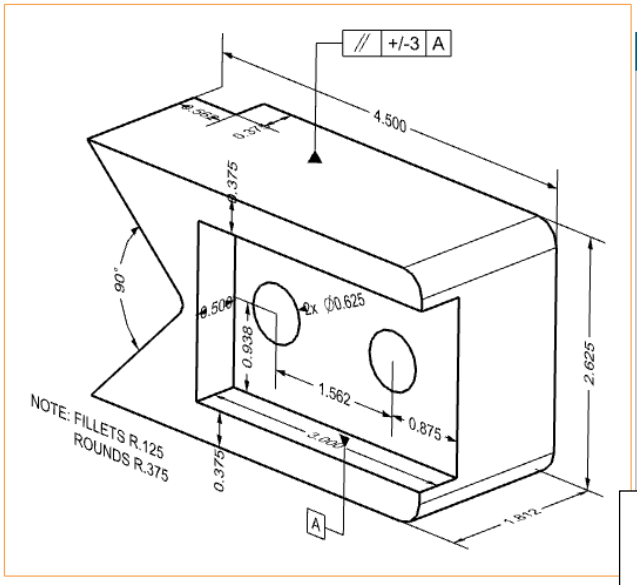


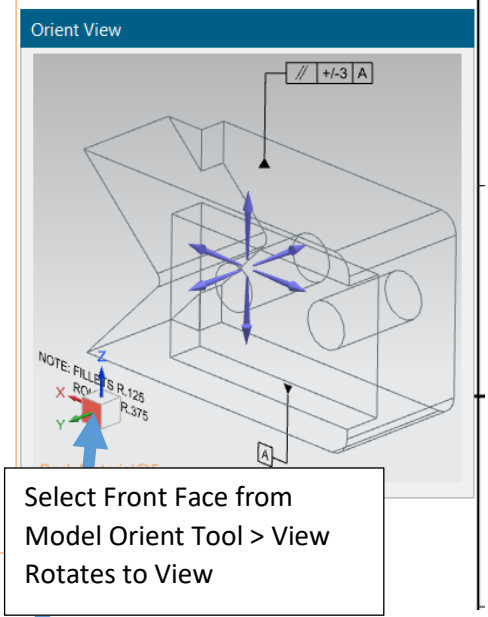
ii. Select Orient Tool



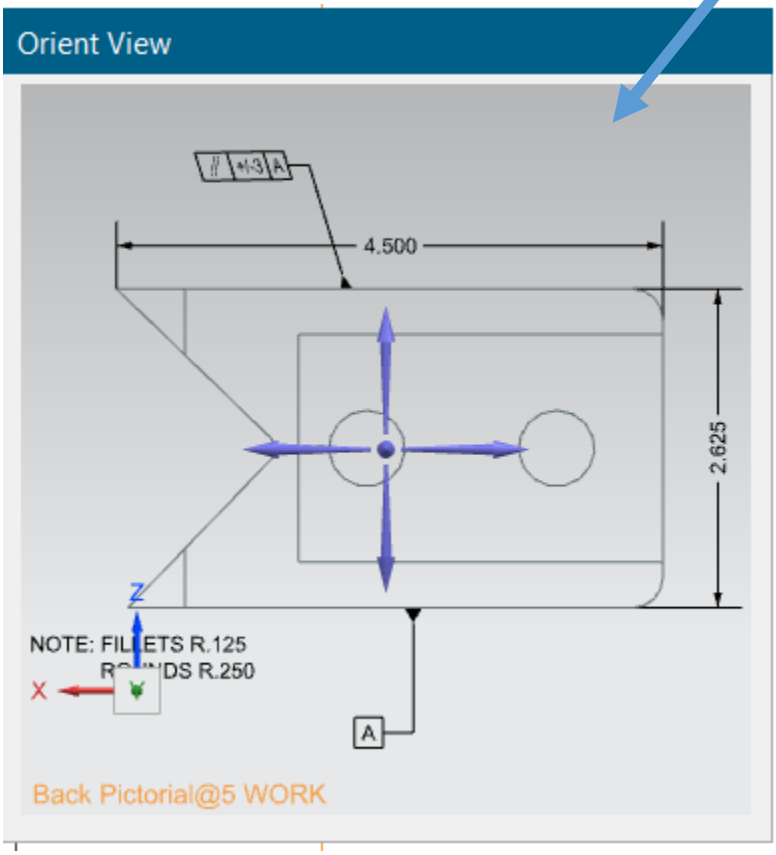
iii. Orient View Pop-Up Menu Appears > Set the View orientation by rotation or selecting a surface from the orient icon to set view > press Ok







Select Front Face from Model Orient Tool > View Rotates to View



Final Rotated View

NOTE: Dimensions can be selected and moved if needed to cleanup a view.

NOTE: Dimensional Settings can also be adjusted then updated on the view as needed. (To update: Select View Right Click > Select Update)