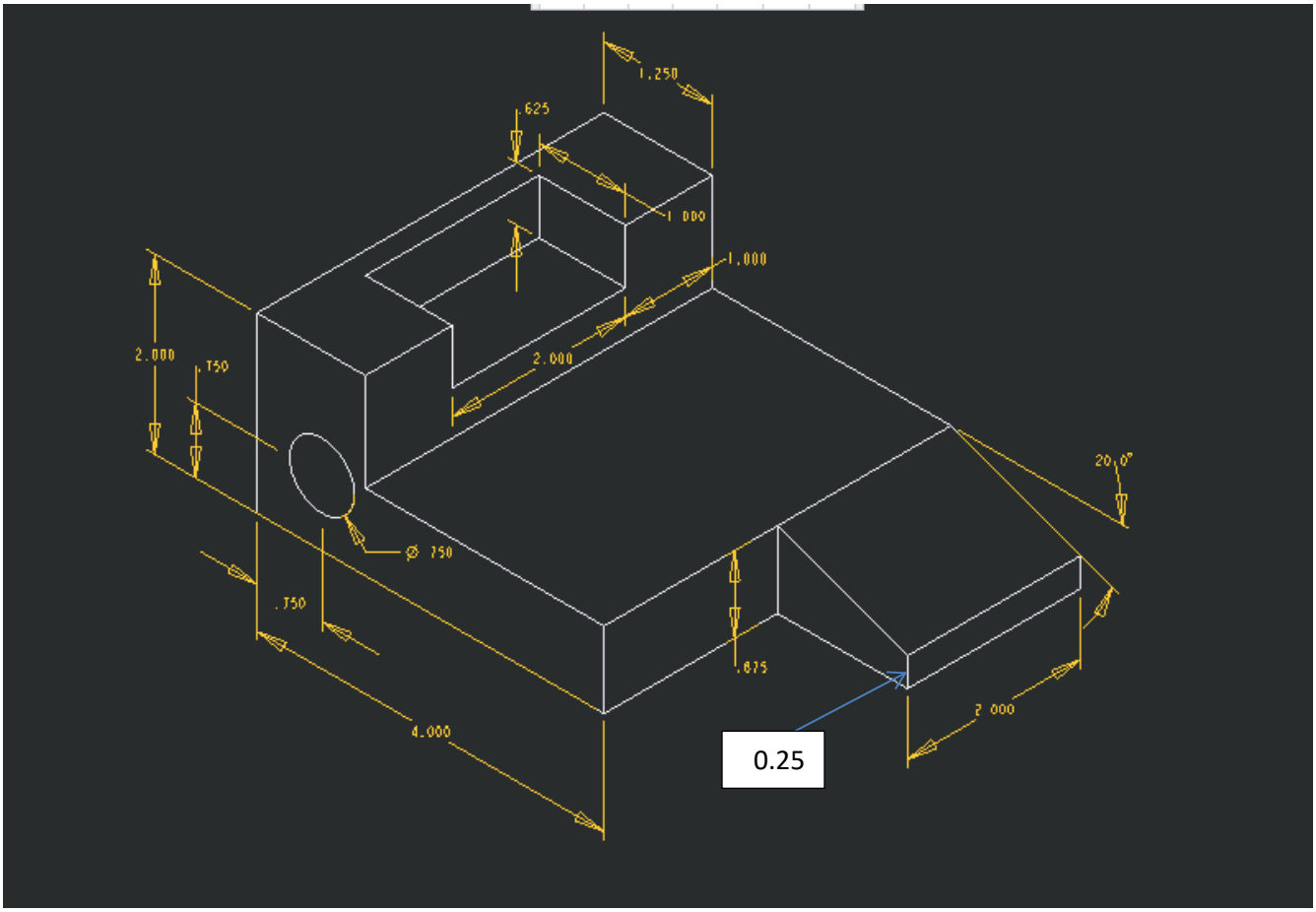
Creo 3D Annotate

By: Matthew Jourden

Brighton High School

3D Annotation allows the user to place notes, dimensions, industrial symbols, Etc. on a solid model as opposed to the 2D Engineering Drawing. The user can design and place all notes in the 3D Solid Model and layout onto a blank border and title block.

1. Design the following part



Placing 3D Dimensions onto Solid Model

This section will all be in the 3D Solid Model File of the Part

Note: Default Settings have been changed to accept 3D Annotations on an Engineering Drawing. If your settings are not correct change the following setting in your Drawing Options:

File >Prepare > Drawing Options > Detail Options > Change > These Options Controls Dimensions > allow_3d_dimensions > yes > add/change > ok

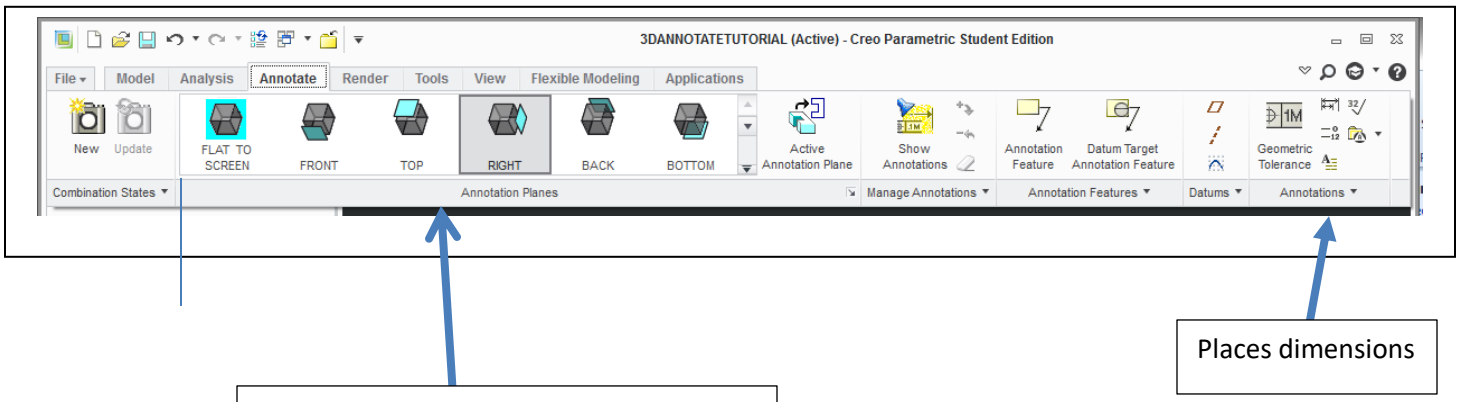
2. Placing Linear Dimensions

   Placing 3D dimensions is similar to placing 2D dimensions in where the user has two options of selection

   a. Left Click on 1 line > Middle Mouse Button to Place
   b. Left Click on 1 Line and then Left Click on a Parallel Line > Middle Mouse Button to Place

   NOTE: Do not worry about where the dimension is placed. User will organize the dimensions later

3. Go To Annotate Tab



Annotation Planes: Allows the user to choose orientation of dimension. Note the orientation plane needs to be the same as the view one would normally place a dimension I.E Depth Dimensions= Top or Right Plane

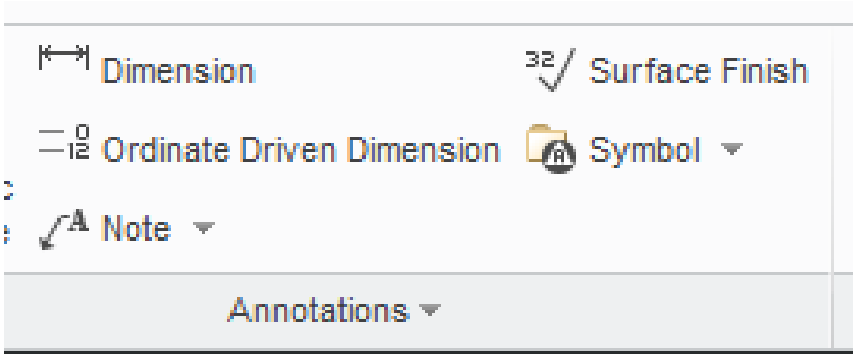
Places dimensions

4. Select Front Annotation Plane (The next few steps will stay on this plane. Do Not change until instructed)

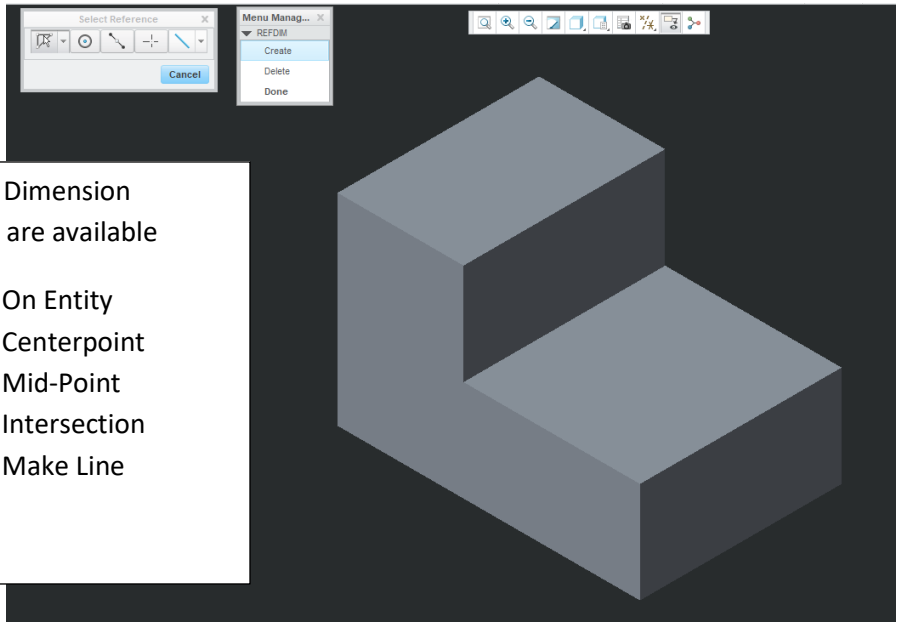
5. Click on Type of Annotation Feature Desired

Types of Annotations

1. Driven Dimension (Linear)
2. Reference Dimension
3. Ordinate Dimension
4. Ordinate Reference Dimension
5. Geometric Dimensioning & Tolerancing
6. Datum Tag
7. Surface Finish
8. Punch Model
9. Manufacturing
10. Note
11. Custom Symbol



6. Select Dimension Icon



Regular Dimension Options are available

1. On Entity
2. Centerpoint
3. Mid-Point
4. Intersection
5. Make Line

7. Placing Dimensions is the same as placing dimensions in the Engineering Drawing. Select Edge and Middle Mouse Button to Place or Select Two Edges and Middle Mouse Button to Place. Note the rules of dimensioning are slightly different from dimensioning orthographic views. When dimensioning a 3D part user should be aware of location of dimensions and place them where they will not interfere with other dimensions or features of the part. Dimensions should be grouped and placed on similar reference planes if possible (I.E. The majority of Length Dimensions will be placed on the Front or Top Reference Plane.)

Sample Dimensions in different Reference Planes

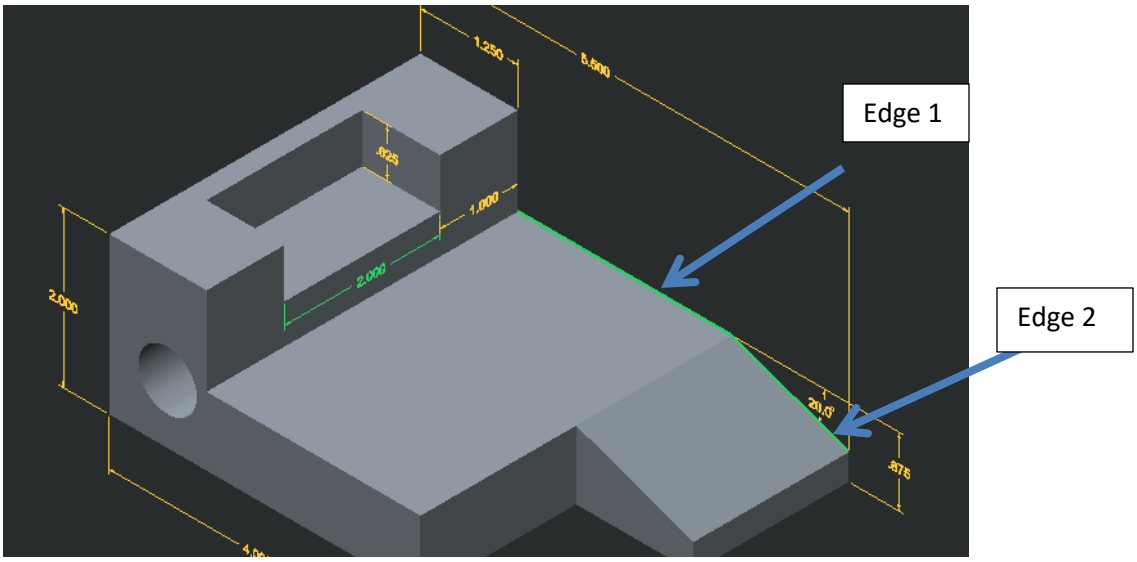
Length

Height

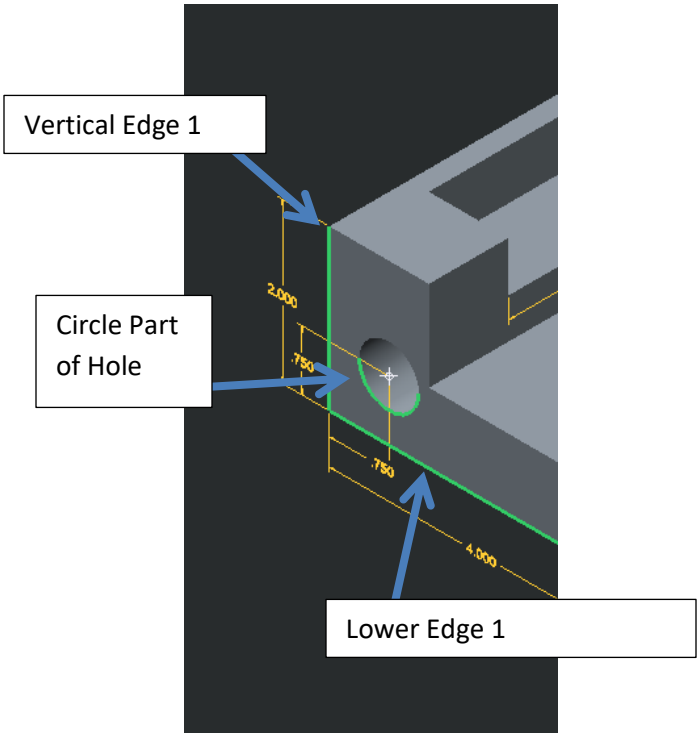
Depth

NOTE: Various issues may arise based upon the user selections. See below for a variety issues user may want to watch for when placing dimensions.

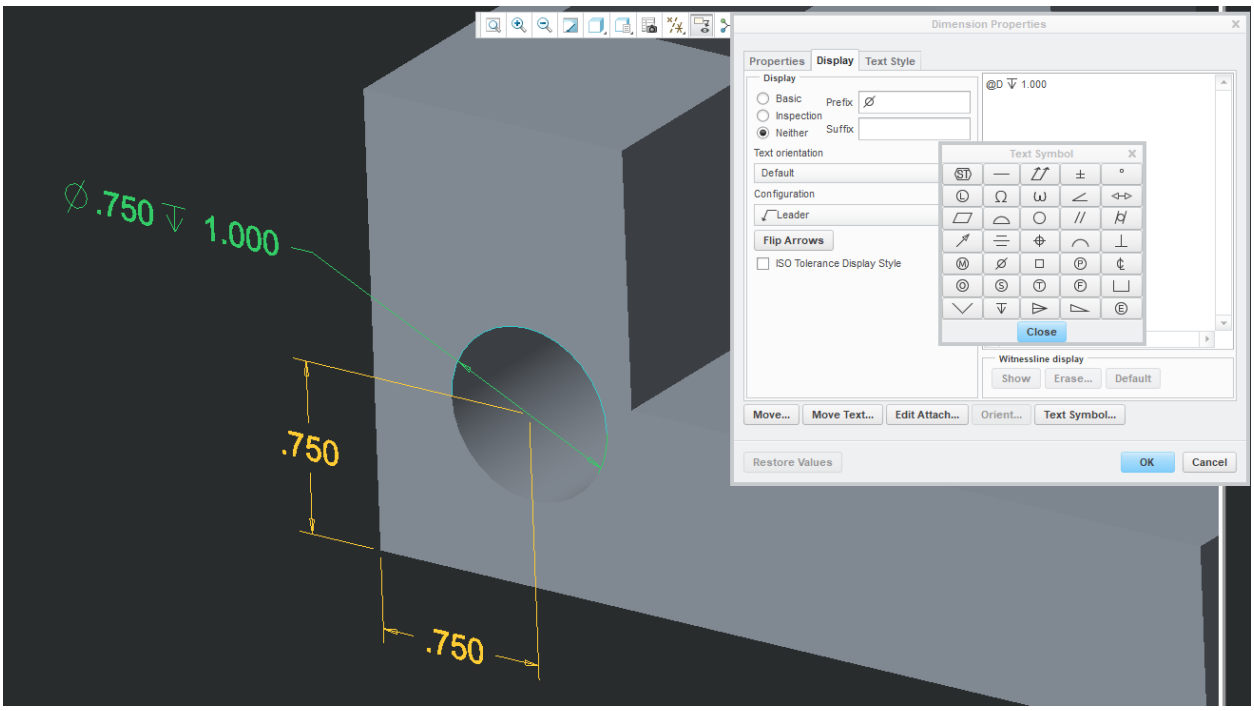
8. Angle Dimension:
    a. Change Annotation Plane to Front
    b. Select Dimension
    c. Select the following Edge 1 and Edge 2
    d. Middle Mouse Button to Place



9. Hole Dimension Locators
    a. Annotate Plane Front
    b. Select Dimension Icon
    c. Menu Manager= On Entity
    d. Left Click on Left Edge 1
    e. Menu Manager = Center
    f. Left Click on Circle Part of the Hole
    g. Middle Mouse Button to place
    h. Repeat for Vertical Locator (Change Edge to Lower Edge 2)(Look at Menu Manger to see what is chosen for selection Edge=On Entity, Hole=Center



10. Setting Size of Hole
    a. Annotate Plane Front
    b. Select Dimension Icon
    c. Left Click once on the Hole (If to Place dimension after this point user will get Radius)
    d. Left Click Second Time on the same part of the Hole
    e. Middle Mouse Button Diameter will be placed
    f. Add the Depth: Double Click on the Diameter > Loads Dimension Settings > Click on Display Tab > Place Depth Symbol after @D > Type in depth value.



11. Make sure the Part is Fully Dimensioned: Place missing Dimensions

Layout of 3D Annotated Part

1. Open New Drawing > Use A-Size Title Block for your Format
2. Place General Isometric View of the part. Center and scale part on sheet
3. Select View
4. Annotate Tab > Show Model Annotations > This will show all of the dimensions on your part. Including the Software Dimensions (Weak and Strong) and user placed Annotations
5. Scroll to the Bottom of the list > Select only the Dimensions that start with AD (These represent your dimensions.)
6. Organize dimensions. Use Snap Lines and Creator Snap Lines to organize dimensions. Remember that dimensions do not have to be grouped. We place dimensions wherever looks the best.

Save and Print Project