

Creo Dimensioning Tutorial

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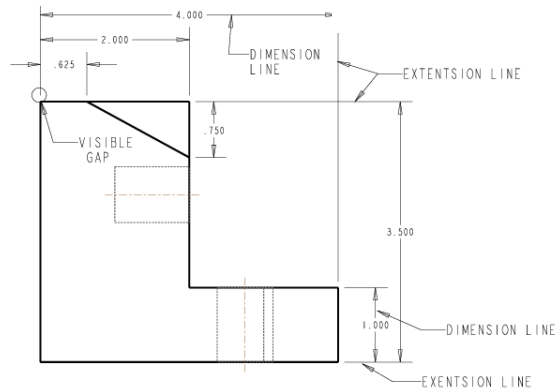
Step 1: Layout Views

Already done for Engineering Design Technology 1

DWG 1-4 Views (Front, Top, Right Side) Scale is 1.000 = 1.000 (Full); Isometric Scale does not matter as long as it does not interfere with the Front, Top and Right Side Views

Step 2: Placing Dimensions

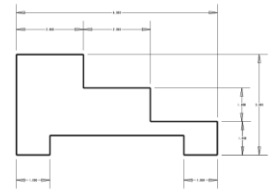
Parts to a Dimension



Dimensioning Schemes

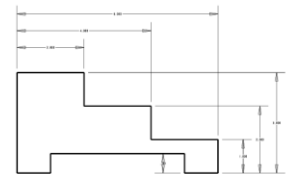
IN-SERIES DIMENSIONING

IN-SERIES DIMENSIONING HAS THE DIMENSIONING LINES UP TO ONE ANOTHER ASSUMING THE DIMENSIONING LINES TO NOT INTERSECT AN EXTENSION LINE



STAIR STEP (DATUM) DIMENSIONING

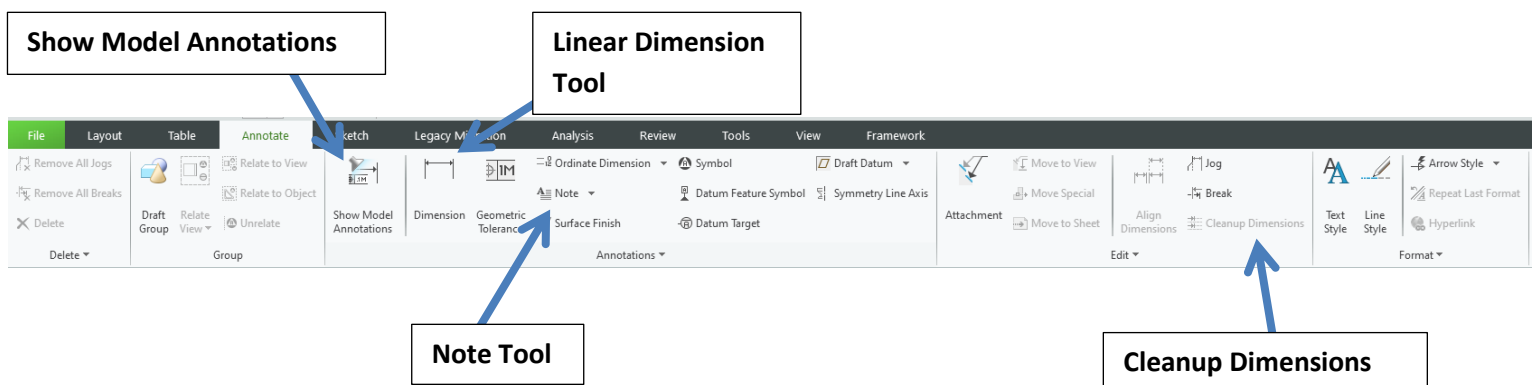
STAIR STEP DIMENSIONING SCHEME HAS THE USER USE A COMMON DATUM (EDGE) TO MEASURE FROM FOR EACH OF THE DIMENSIONS. THE DATUM SHOULD BE A COMMON EDGE THAT THE DESIGNER WILL USE AS A KNOWN EDGE (DATUM) THAT WILL BE SQUARE TO THE JIG OR FIXTURE OR MOLD THAT WILL HOLD THE PART DURING MANUFACTURING



1. Open the Drawing Layout File for Part 1
2. Turn Off Datums (D Key). If datums are **NOT** turned off they will printout
3. Go to Annotate Tab

The annotate tab will allow a user to dimension parts in various ways. We are only going to use a couple of buttons here.

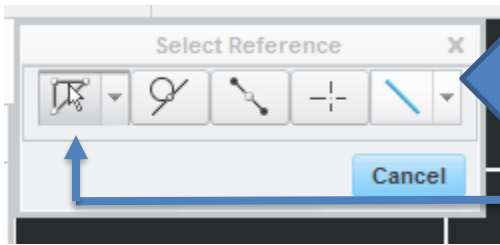
- a. **Linear Dimension Tool**: Places dimensions on the part
- b. **Note Tool**: Places text notes on the part
- c. **Show Model Annotations**: Auto dimensions, finishes, and centerlines
- d. **Cleanup Dimensions**: Organizes the dimensions by spacing and using snap lines



4. Select Linear Dimension Tool

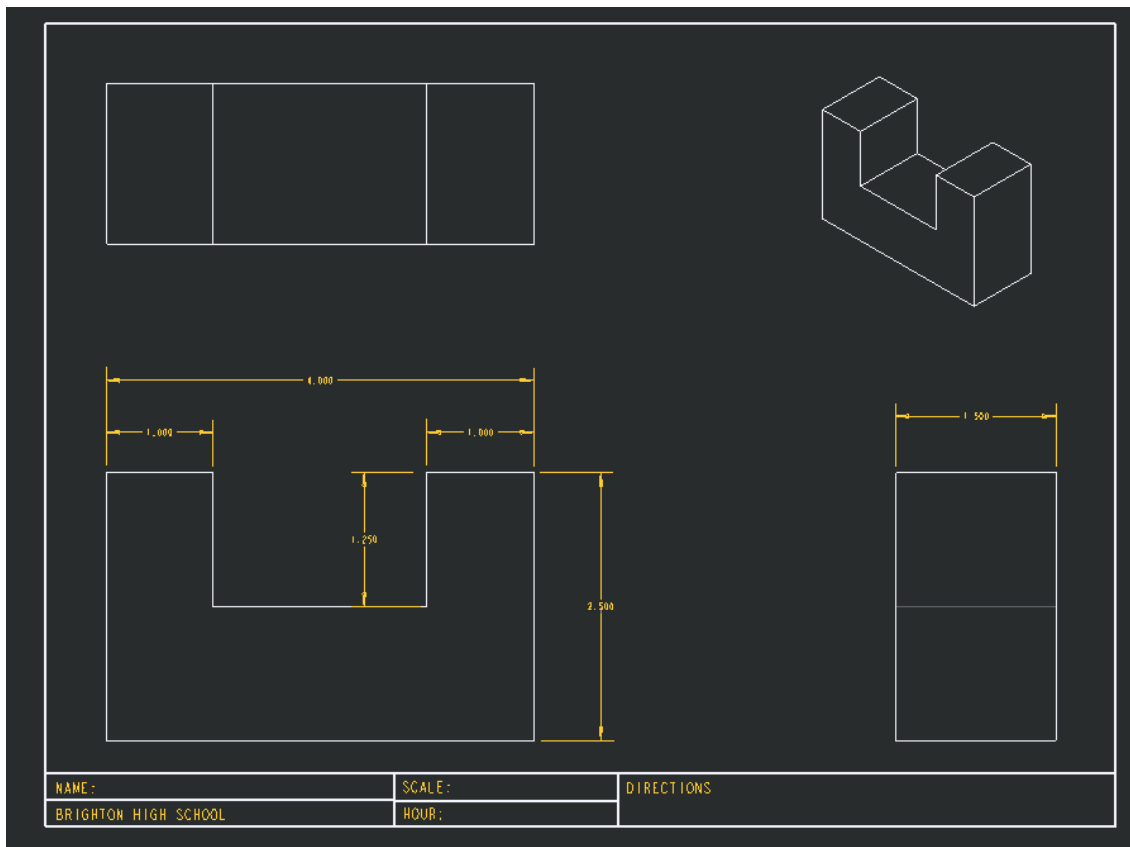
There are two ways to place dimensions

- Left Clicking** on the line to be measured > **Middle Mouse Button** to place
OR
- Holding **CTRL** Select **two parallel lines** > **Middle Mouse Button** to place (NOTE: Select two vertical lines gets a horizontal measure and selecting two vertical gets a vertical measure)



After selecting the Linear Dimension Tool the Menu Manager will appear. This allows the user to select various entities or find known points. We will just keep the option selected for **On Entity** for right now.

Select edges and place the following dimensions. Try each method. Do not be concerned about where you place the dimensions. We will organize the dimensions later. Note place smaller dimensions closer to the part.



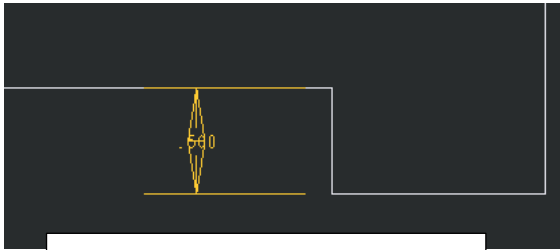
5. Changing Arrowhead direction

The Arrowheads will need to be flipped externally if

- a. that only the arrowheads are seen
- b. The dimension value overlaps the Extension Lines

In this sample this would not have to be done, but we will experiment with it. (NOTE This dimension **DOES NOT** need external arrowheads)

Example

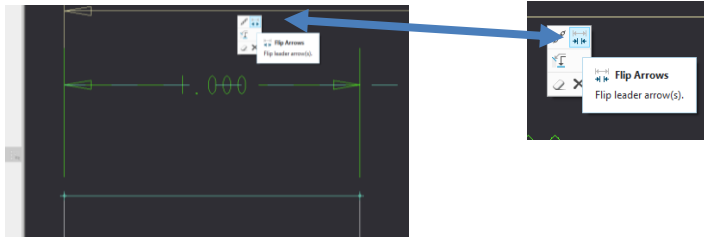


Notice Dimensional Value cannot be easily read because of the arrowheads



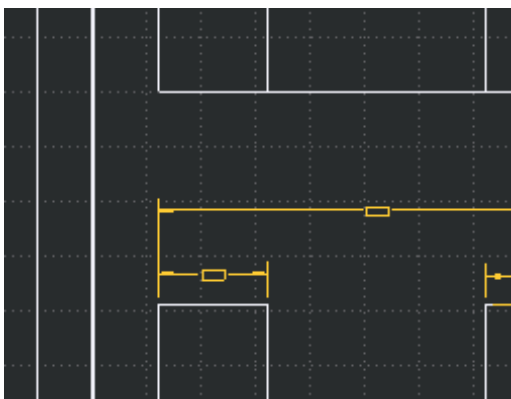
Notice Dimensional Value cannot be easily read now that the arrowheads are external

- c. Select any **dimension**
- d. Select **Flip Arrowheads**

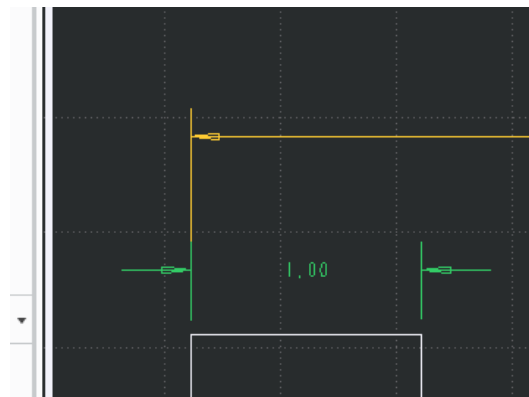


- e. Flip the arrows back internally

Before



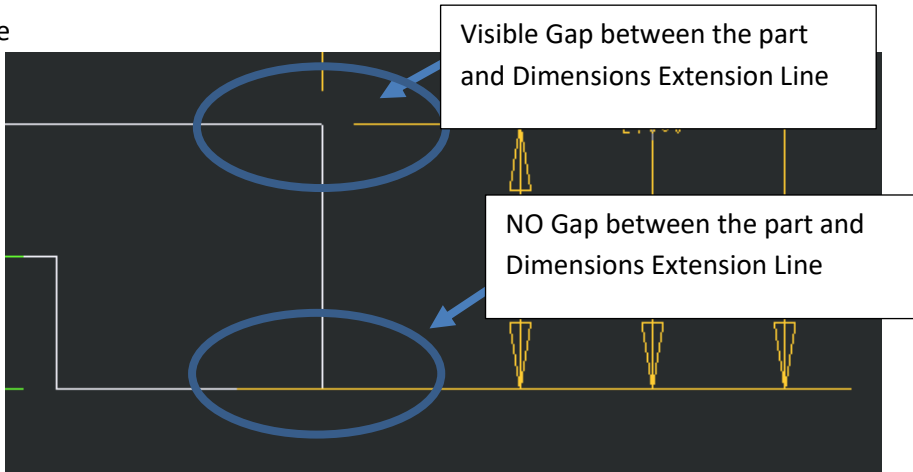
After



6. Modify Extension Lines

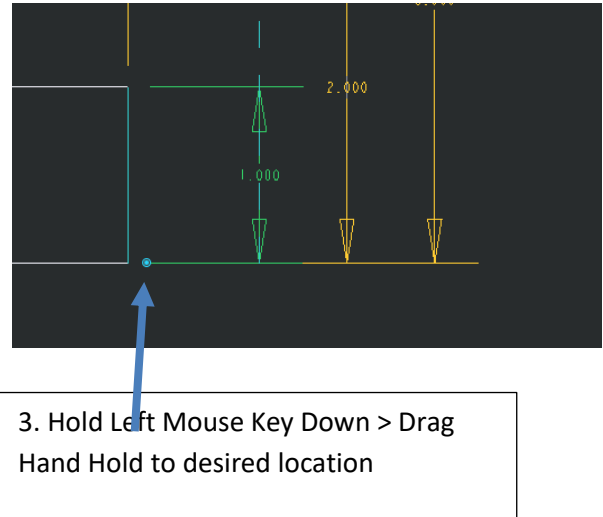
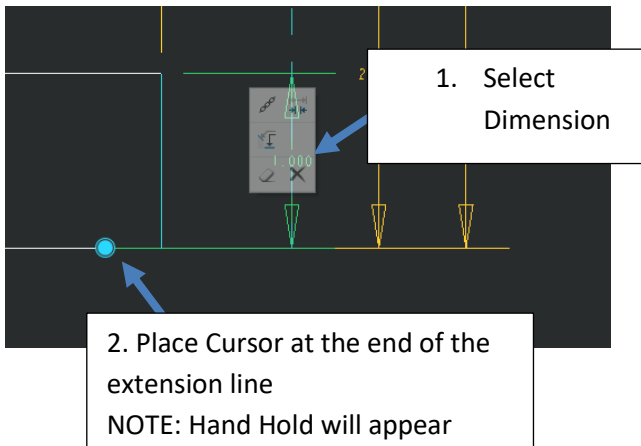
7. Extension Lines **WILL** have a gap between the point of the part that is being measured and the start of the Extension Line.

8. Example



i. To Modify

1. Select Dimension to modify: Left Click on the Dimensional Value Place Cursor at the end of the extension line until a hand hold appears
2. Hold Left Mouse Key down on the hand hold
3. Drag Extension Line to desired location

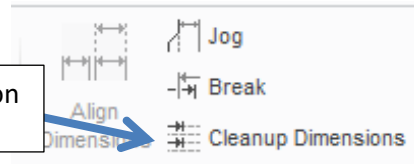


9. Cleanup Dimensions

a. View Outline

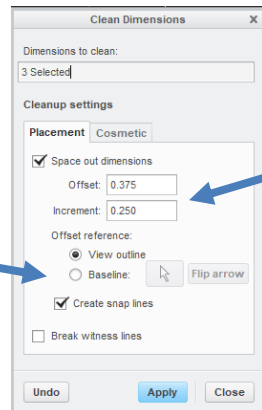
- i. Select **Layout Tab**
- ii. Select all of your dimensions (Except the 1.25 Dimension inside of the cut) by holding the CTRL key and Left Click on the dimensions or draw a window around the dimensions. The dimensions should turn green when selected
- iii. > Select **Cleanup Dimensions** Icon in the **Annotate Tab**. This tool will allow us to evenly space our dimensions

Select Layout Tab > Select Cleanup Dimension



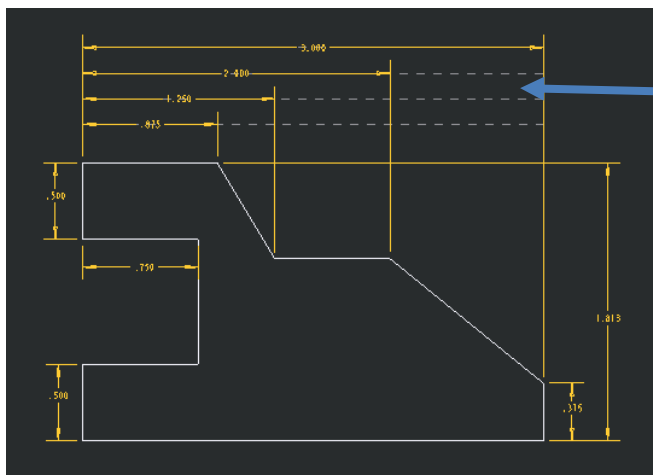
- iv. A pop up window will appear.
 1. Space out dimensions
 - a. Offset (Distance from edge/view outline) = 0.375
 - b. Increment (Distance from between layers of dimensions) = 0.25
 2. Offset Reference
 - a. View Outline: Sets dimensions based on the green outline of the view
 - b. Baseline: Allows the user to select an edge to snap off of.

Offset Reference: Allows the user to use the View Outline (Green Box) or Baseline (Selected Edge) to snap off of. NOTE: Baseline is used for dimensions inside the part. See Mr Jourden for Baseline



Space of Dimensions

- v. Select Apply. Dashed Lines will Appear under the dimensions. These are snap lines. The user can drag a dimension over these lines and the dimension will snap to it. The user may also create their own snap lines, this will be discussed later.
- vi. Select Close



Snap Lines: Link the Dimension to a set Distance.
 NOTE: Dimension can move by selecting > Hold Left Mouse Key > Drag Cursor

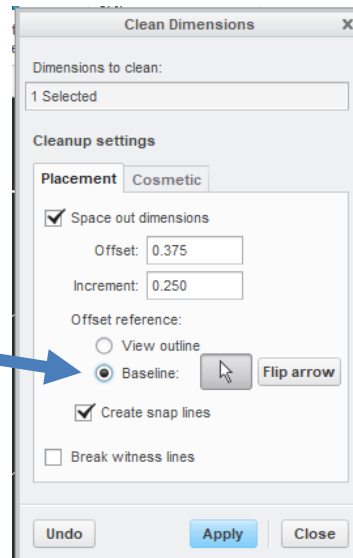
Snap Line Lengths: Can be adjusted Select the Snap Line > Place Cursor on an End Point > Hold Left Mouse Key > Drag Cursor to desired location

NOTE: Snap Lines DO NOT Print or Export, So DO NOT delete after placing

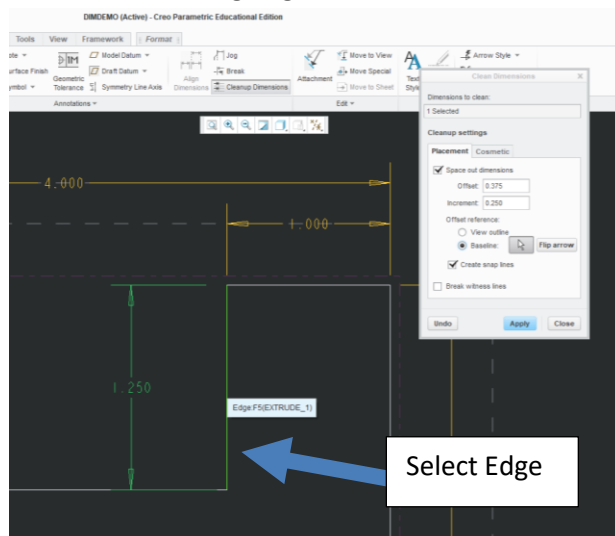
b. Baseline

- i. Select the 1.250 measurement in the front view
- ii. Select Cleanup dimensions
- iii. Select Baseline

Select Baseline

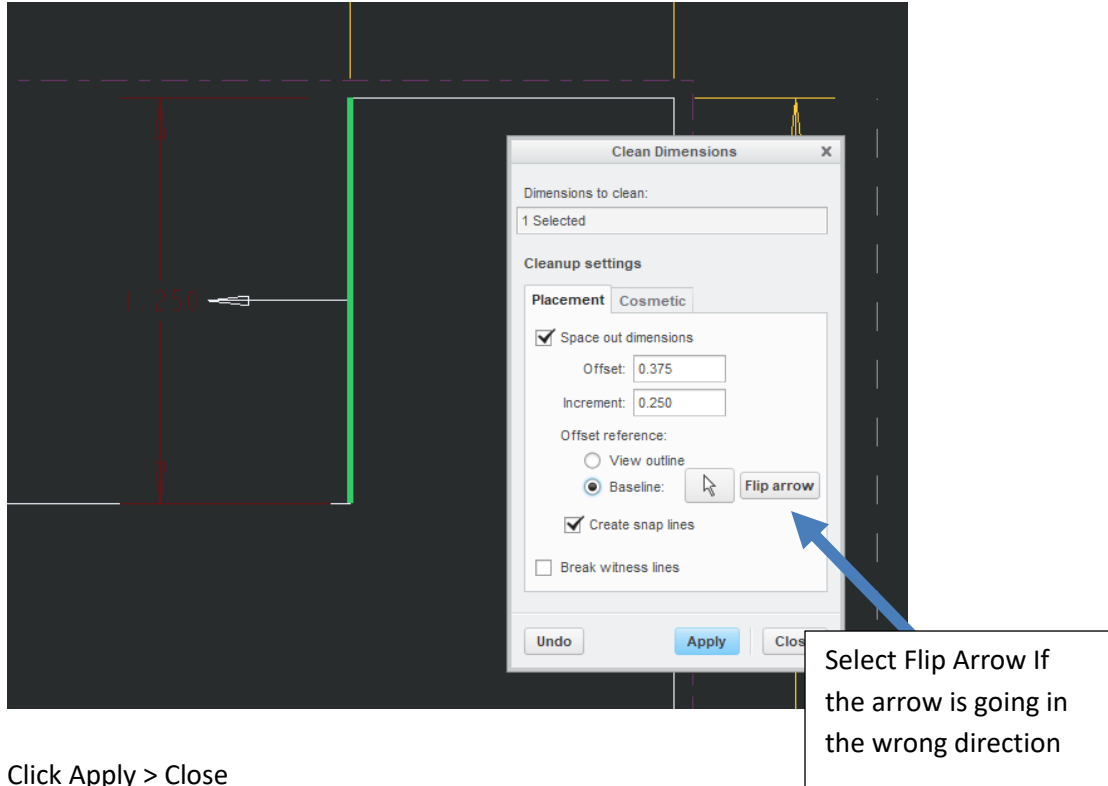


iv. Select the following edge



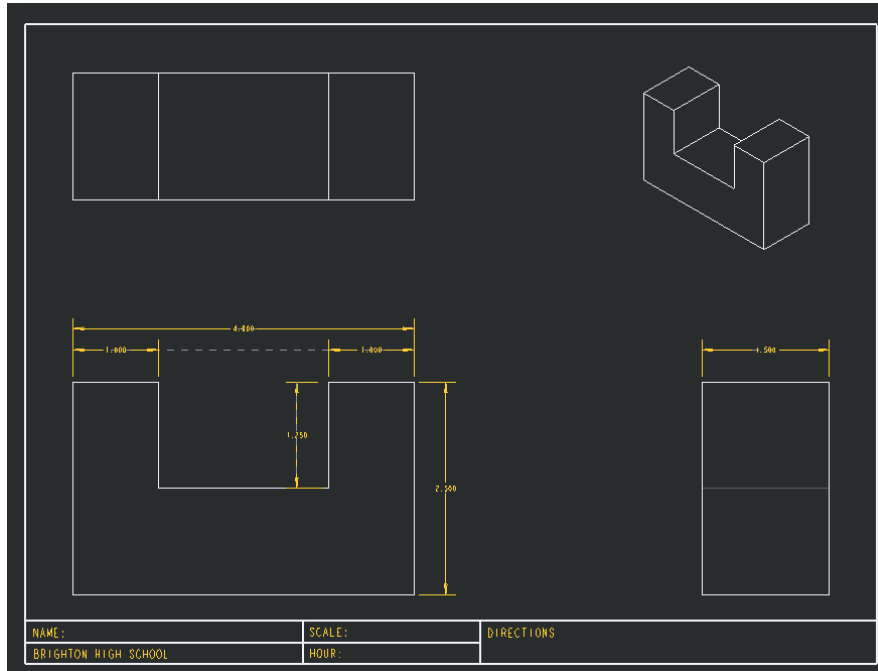
Select Edge

- v. Arrow head represents the direction the snap line will be sent parallel to the edge. Be sure the Arrow is pointing towards the cut. If it does not click the Flip Arrow icon next to the Baseline Option in the Cleanup Dimensions Settings Window



- vi. Click Apply > Close

10. Final Drawing should look as follows

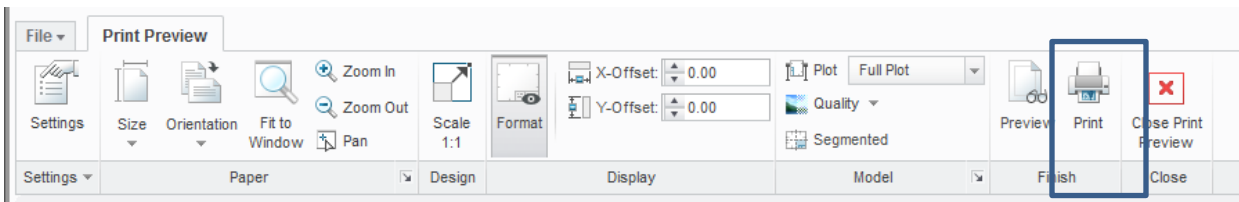


11. Fill out the Title Block

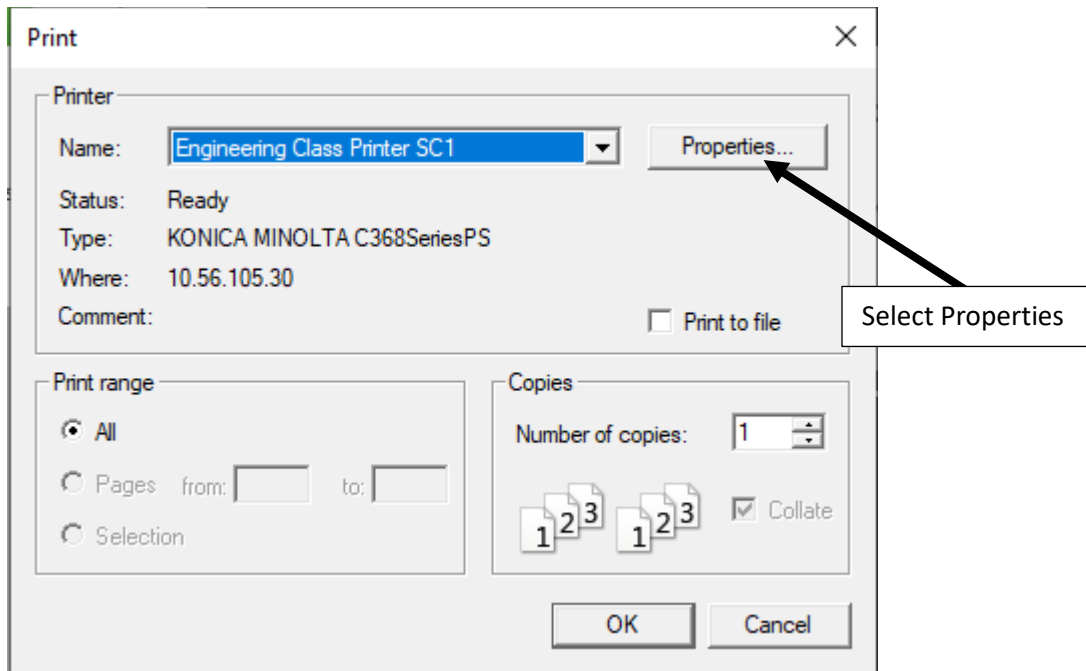
- a. Use the text tool in the Annotate Tab to fill in the following, Be sure to CHANGE TEXT HEIGHT to 0.1.
 - i. DATE: TODAYS DATE
 - ii. SCALE: FULL (NOTE for DWG 1-4 the Scale is FULL or 1.000 = 1.000)
 - iii. DWG: UBLOCK
 - iv. SHEET: 1 OF 1 (NOTE all of Intro CADD Drawings will be 1 OF 1; may change on the format to make it permanent)
 - v. Align Text to the appropriate boxes

12. Print

- a. Autoscale first by pressing the **A** Key
- b. **File > Print > Select Print**
- c. Click **OK**

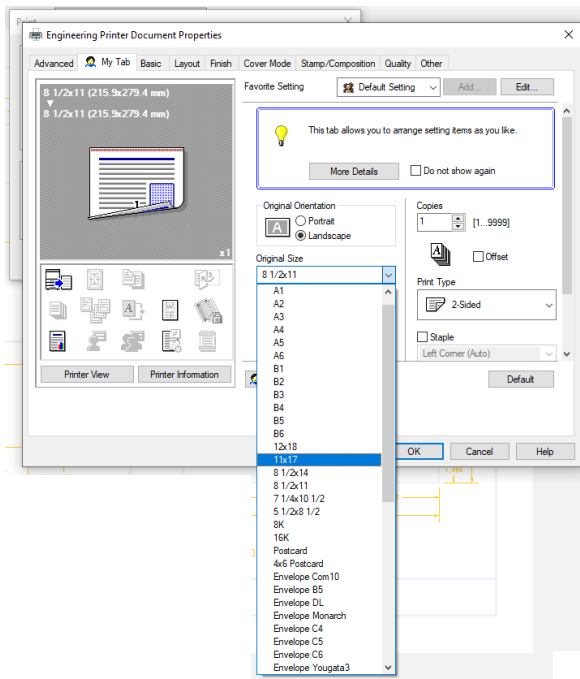


- a. Choose the Engineering Printer or Engineering Class Printer > Select Properties

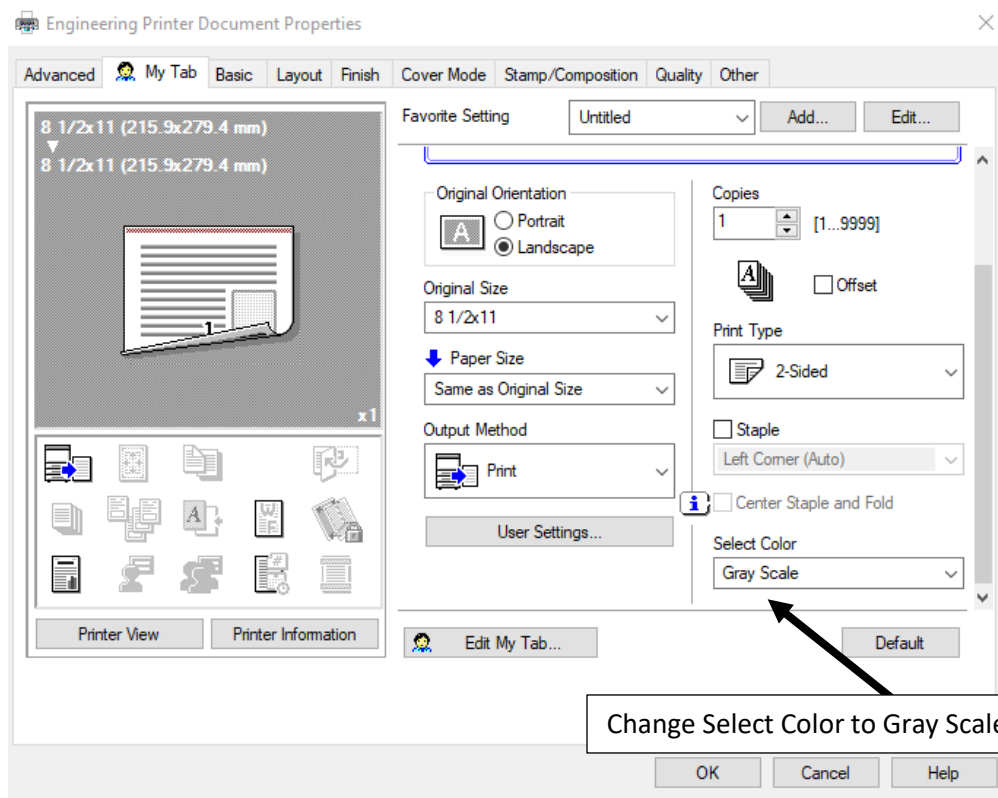


- b. Tab: My Tab

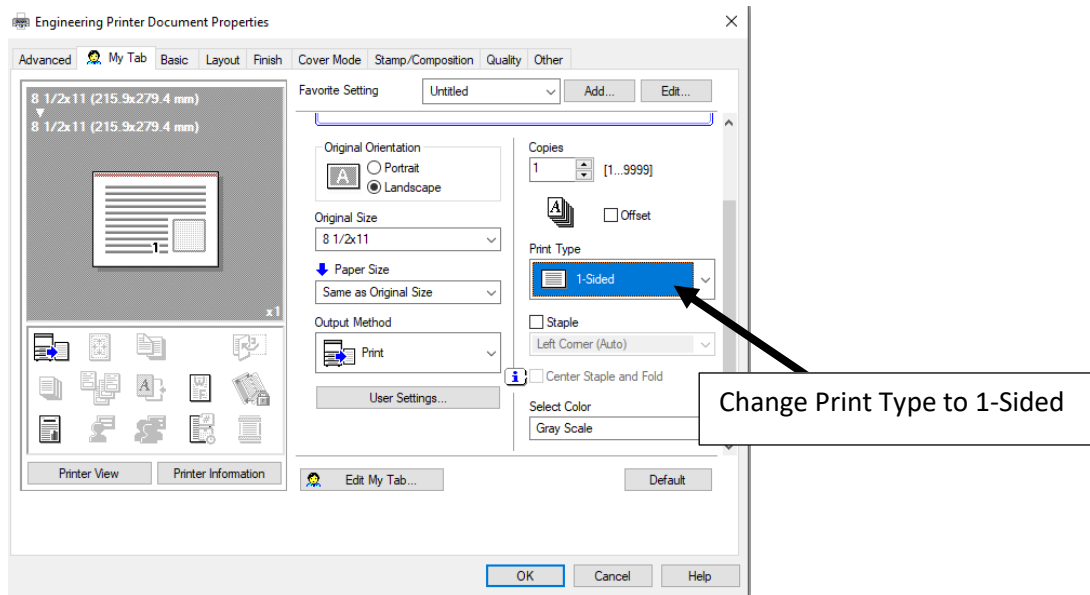
- i. Change Paper Size as needed 8 1/2" x 11" or 11" x 17"
- ii. For Parts 1-14 Choose 8 1/2" x 11"



iii. Change Select Color (Bottom Right Option) to Gray Scale



iv. Change Print Type to 1-Sided



v. Final Menu > Select OK > Select Ok to send to the printer

13. Save the Drawing

14. Repeat Steps of DWG 2-4.

NOTE all dimensions are placed on the profile given on the worksheet, except for the extrude distance. Place the extrude distance on one of the other views other than the profile view. Dimension exactly as shown on the worksheet.