Creo Dimensioning Tutorial

Step 1: Layout Views
   Already done for Intro CADD DWG 1-4

Step 2: Placing Dimensions

1. 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
2. Select Linear Dimension Tool
There are two ways to place dimensions
   a. **Left Clicking** on the line to be measured > **Middle Mouse Button** to place
   OR
   b. Holding **CTRL** Select **two parallel lines** > **Middle Mouse Button** to place
   (NOTE: Select two vertical lines gets a horizontal measure and selecting two
t   vertical gets a vertical measure)

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.

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.
3. Changing Arrowhead direction
   If you have a dimension that you only can see the arrowheads or the values you will need to flip the arrows on the outside of the extension lines. In this sample this would not have to be done, but we will experiment with it.
   a. Select any dimension
   b. Right Mouse button on the value
   c. Select Flip Arrowheads
   d. The arrows should flip on the outside (See Before and After Below)(You may have to do this on DWG 2-4).
   e. Flip the arrows back internally

![Before](image1.png) ![After](image2.png)
4. **Cleanup Dimensions**
   a. Select all of your dimensions by holding the CTRL key and Left Click on the dimensions. The dimensions should turn green.
   b. Select **Cleanup Dimensions** icon in the **Annotate Tab**. This tool will allow us to evenly space our dimensions.
   c. A pop up window will appear.
      i. Space out dimensions
         1. Offset (Distance from edge/view outline) = 0.375
         2. Increment (Distance from another layer of dimension) = 0.25
      ii. Offset Reference
         1. View Outline: Sets dimensions based on the green outline of the view
         2. Baseline: Allows the user to select an edge to snap off of.
   d. 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.
   e. Select Close
5. **Final Drawing should look as follows**

6. **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: TODAY'S DATE
      ii. SCALE: FULL (NOTE for DWG 1-4 the Scale is FULL or 1.00=1.00)
      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

7. **Print**
   a. Autoscale first by pressing the **A Key**
   b. **File > Print > Print**
   c. Click **OK**
   d. Click **Ok (Second Time)**

**NOTE:** Printer knows the paper size and orientation. These menus allow you to change and alter the paper selection of needed.

**NOTE:** If the Print Preview Window Appears find the Print Button on the Print Tab at the Top of the screen and press print > this will load the printer selection window.
The Default Printer should be B32 HP 5200 (unless otherwise instructed) > **Click Print**
8. Save the Drawing


   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.