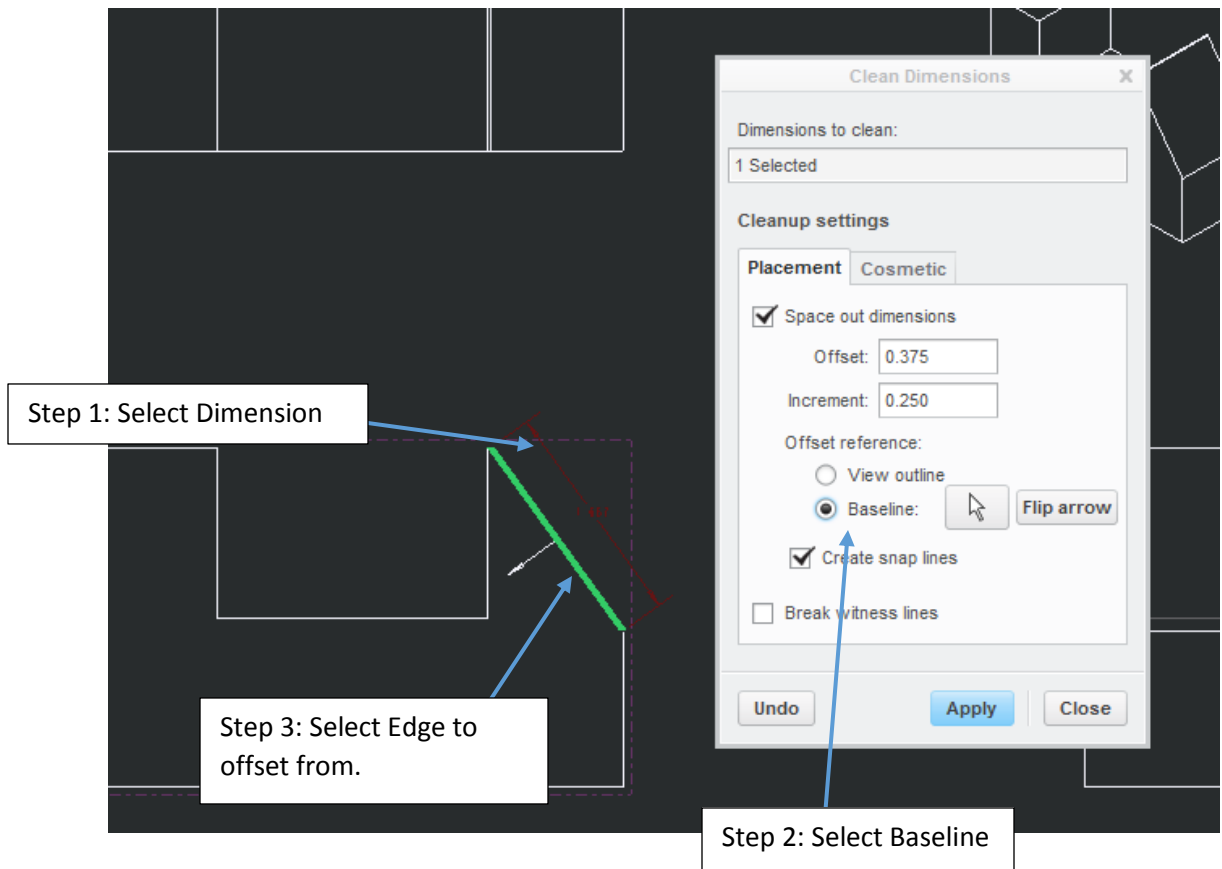


## Creating Independent Snap Lines

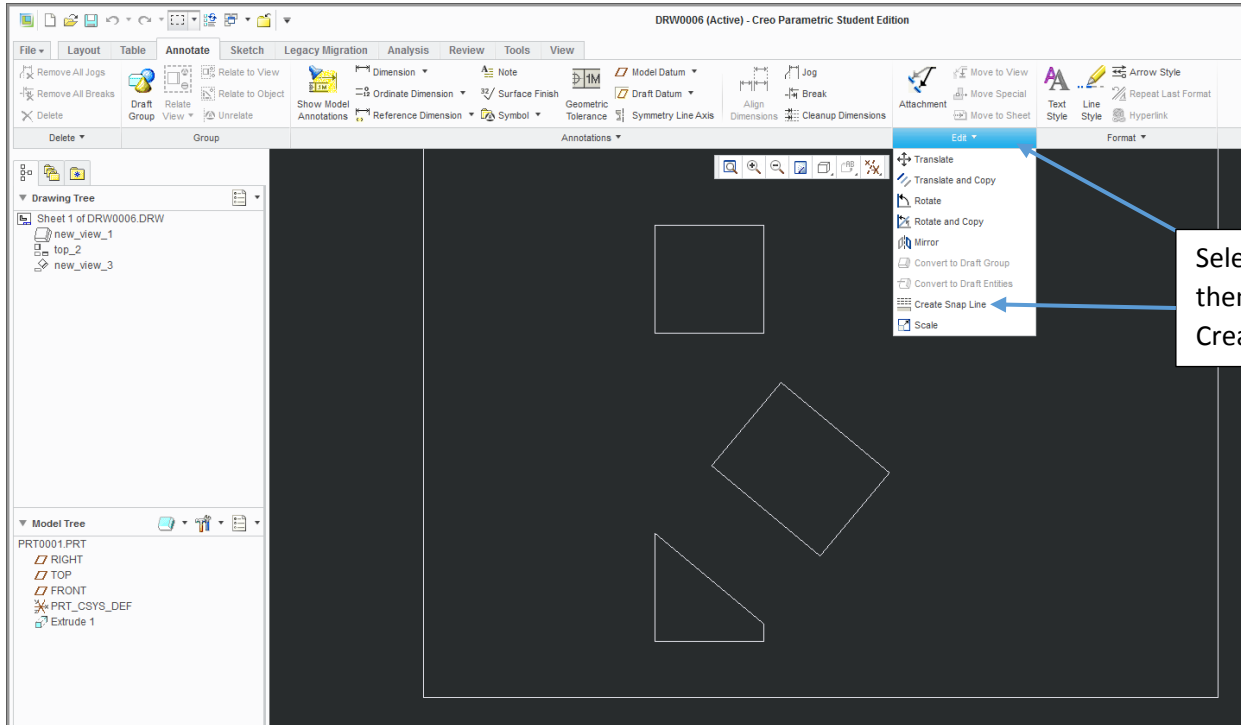
### Option 1: Cleanup Dimensions

1. Place Dimensions
2. Highlight Dimensions to evenly space
3. Select Cleanup Dimensions
  - a. Select Baseline > Select Edge on part to offset from > Default distance should be set under offset/increment > Click Apply



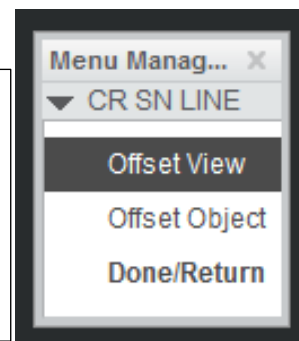
### Option 2: Create Snap Lines on Angle Surfaces

1. Layout your part
2. Annotate Tab > Select Black Arrow next to the Edit Menu > Select Create Snap Line > Menu Manager will appear and give two options

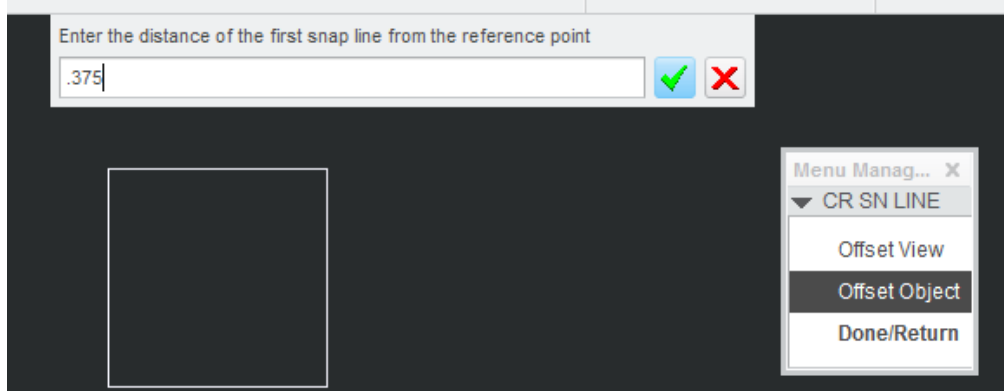


### Types of Snap Lines

- a. Offset View: Places snap lines parallel to the view's edges (green outline when view is selected)
- b. Offset Object: Places snap lines parallel to the edge that the user selects



3. Select Offset Object
4. Select the Line you wish to have a Snap Line Parallel to > Middle Mouse Button in the area you wish to place the snap line
5. Pop up menu at the top of the screen will ask to Enter the distance of the first snap line from the reference point > Enter .375 > Left Click on the Green Check or Press Enter Key



6. Next the software will ask Enter how snap line to be created > Enter the number of snap lines you wish to create

- a. If the user enters any number other than one 2,3, 4 the software will ask for the distance between the given snap lines, which will always be .25 I.E

Press enter and enter the appropriate information based upon the number of snap lines you wish to create.

7. Finished part should have snap lines parallel to the given edge. NOTE: User can still change the length of the snap line by selecting it and dragging the end points as needed.

