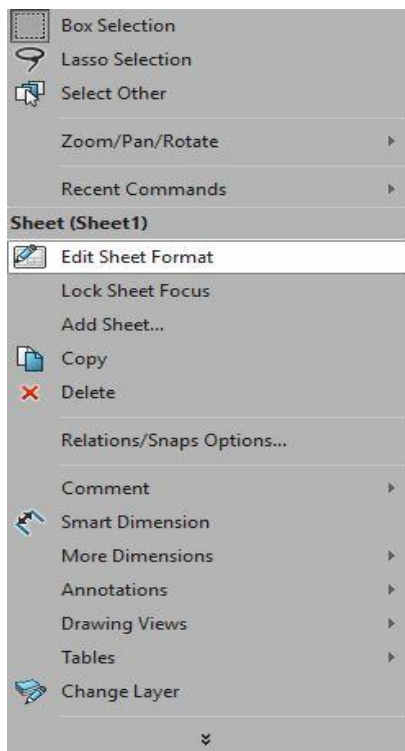


WATERMARK IN SOLIDWORKS DRAWINGS

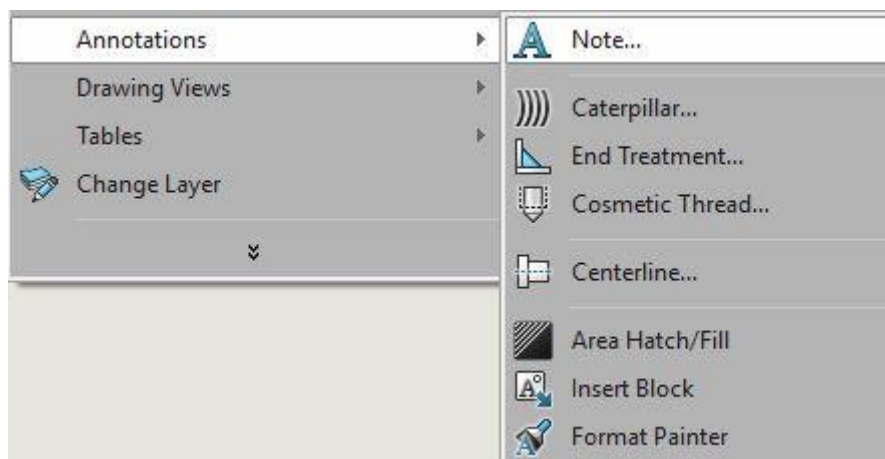
You can add watermarks in SOLIDWORKS drawings to show the status of your drawings like "APPROVED"," CONFIDENTIAL"," REVIEW" etc.

Here is how you can do it.

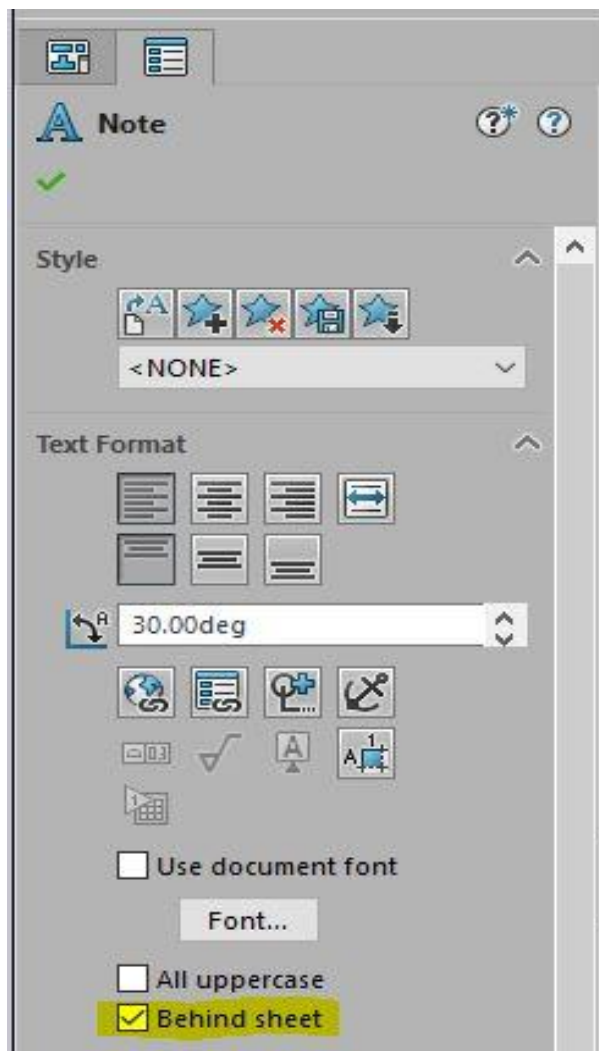
1. Right click on drawing sheet, select "Edit Sheet Format"



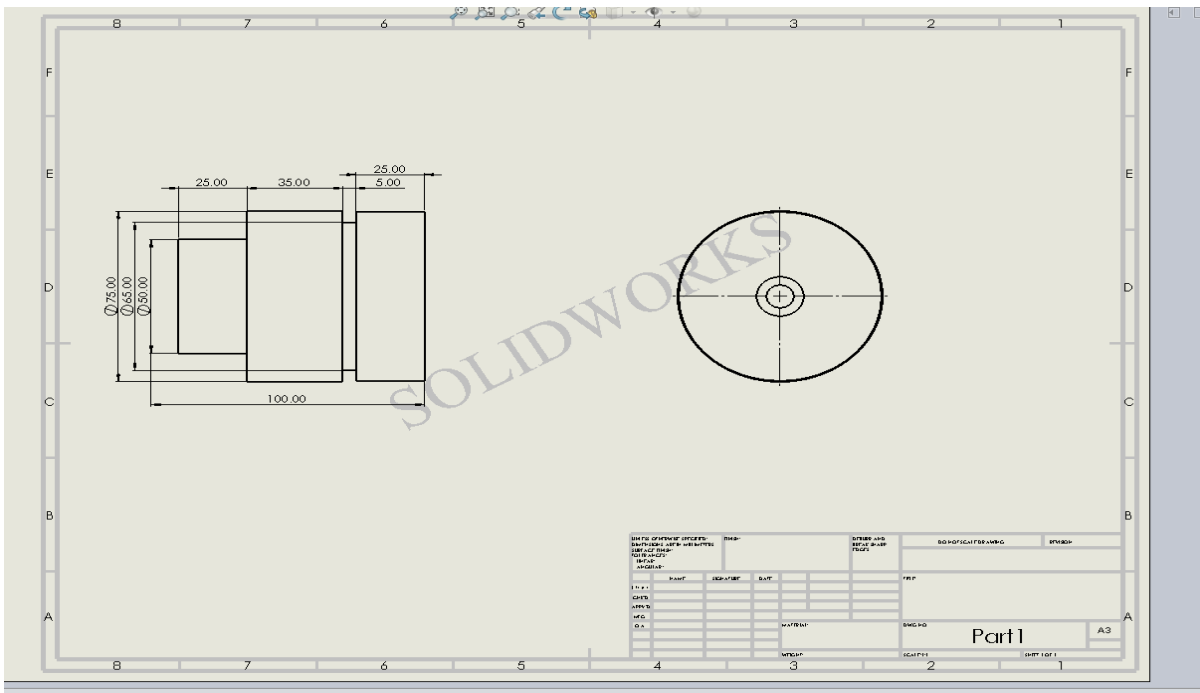
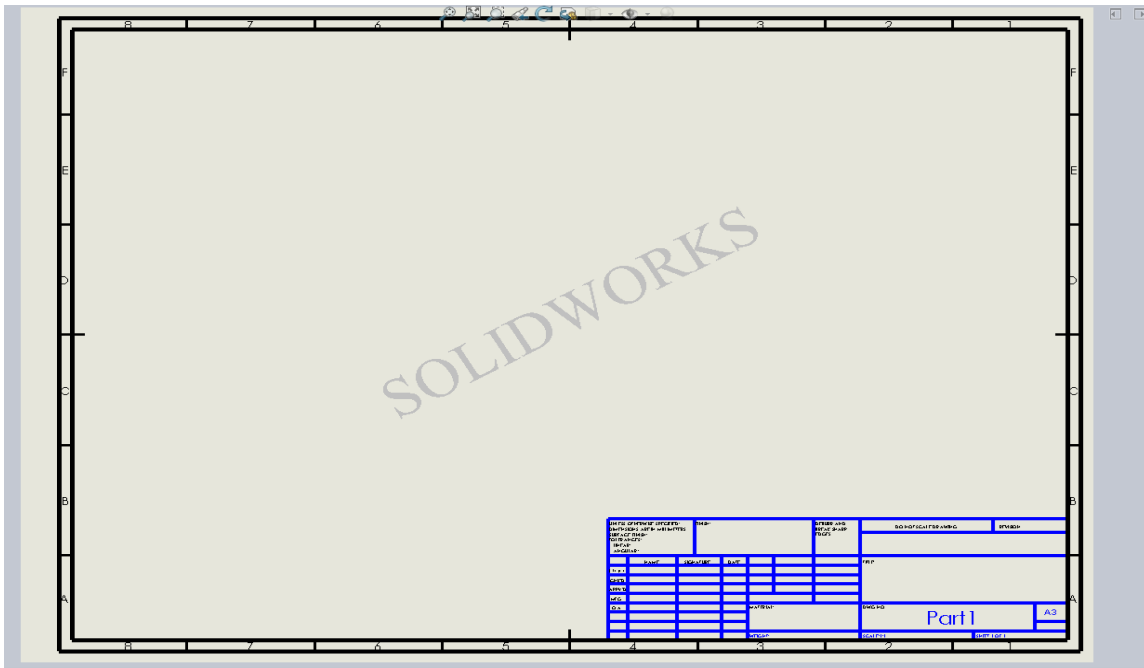
2. Right click on sheet and select Annotations, then select Note



3. Create any note and any font and color you want. I will create SOLIDWORKS note for my example.
4. After your note is created, you can use the angle value to place the note inclined.
5. We need to do one more IMPORTANT step so the note will display behind geometry of any views we create.
6. Click on the note and in the note display manager under Text Format, check the box called Behind sheet.



7. Now right click and select Edit Sheet to get out of Editing Sheet Format mode.



8. Save this as a Sheet format/template so that you can use this for future drawing & Detailing.
