Insert these standard views into a drawing sheet. We can use model view to insert, or open the file and select make drawing from part. Notice the completed title block.

Format the different views to make them easy to see and dimension.
To add views you can use projected views or auxiliary views in the view layout tab. You can also add section views to get a better view of the inside of a part. It is acceptable to dimension to a section view. Notice the difference in clarity between a top view with hidden lines, and a bottom view. The bottom view is clear to a reader.
Difference between a crop view and another section view. If part is symmetrical a crop out view may be acceptable.

Use baseline dimension to dimension the piston. Notice how it links the dimensions together. This can also be accomplished with smart dimension. Add centerlines with the centerline button in the annotation toolbar. Change dimension appearance in the options toolbar. Add notes to make dimensioning easier and less redundant.
Add a subassembly to the drawing package. You can auto bubble the subassembly for the bill of materials. Change around the way the auto bubbles appear in the property manager. Select a custom item number, rather than a default. Add a bill of materials under the table’s icon in the annotations toolbar. Change around the column headers, add headers and resize this box.