**ME 481 – Pre-Feedback on dimensioning preferences**

* Remove sheet format (to save space)
* Draw the part at a scale where it looks much larger than the dimension lines and text
* If you like to show the 3D part, show it at smaller scale and as line drawings
* No shaded drawings
* Use 3rd angle projection
* All dimension text must read horizontally (ASME format)
* All dimension lines and text are to be black (not gray color)
* Avoid unnecessary or long dimension lines
* If there is space, use the dimension text between arrows format
* Align or stagger dim lines and text so it is easy to see what dim lines and text go together.
* Do not forget centerlines of cylinders, shafts, holes – in all views.
* For feature patterns, define pattern shape and anchor it to the rest of the geometry – keep anchor points close to each other and on the same view.
* Keep the dimensions that define a feature close together and close to the feature and preferably on one view. The reader should exert minimum effort locating all the dimensions related to a feature.
* Select a front view that best shows most of the features of the part. Define feature dimensions on the front view as much as possible. Use section views, detailed views, and cutout views when necessary.
* Remove unneeded views. When no dimensions on a view, the view may not be needed.
* Views must be aligned except for enlarged detail views.
* Remove hidden lines when not needed.
* Do not show tangent lines. Set up Solid Works not to draw the tangent lines.
* All cylindrical features should have centerlines and center marks.
* Do not show the shaft diameter when thread specification is provided.
* Show the depth of a keyway from the opposite face of the shaft or hub.
* Do not use THRU ALL, use THRU.
* Use of nX for linear dimensions is not common and confusing because there are so many linear dimensions in a drawing. Unless there is a clear pattern of many closely spaced lines, apply the dimensions directly.
* Do not use part’s visible lines or hidden lines as extension lines.
* Do not use extension lines as dimension lines.
* Align dimensions – it looks better and takes less space.
* While appearance of symmetry does not imply symmetry, if two (or more) features, such as holes or slots, share the same extended centerlines, they are considered to be centered to each other.
* Move all dimension text outside the view boundaries (the rectangle that encloses the part profile). Occasionally, the text can be placed within the box but outside the outer part boundary edges. Rarely, the text may be placed inside the part boundary.
* When applying parallel dimension lines, try to maintain a uniform spacing. The distance between dimension lines should be about the height of one character. The closest dimension line to the part has about 1.5 characters spacing.
* Remove trailing zeros
* Show the end of the small blind holes as a cone allowing drilling (Solid Works hole wizard). Rarely it is necessary to create a flat bottom for these holes. No need to dimension the drill cone.
* Place the dimensions on the view in which the feature shape is best understood.
* Use compact notation only for small features
* Avoid obvious over dimensioning
* Do not use dual dimensions unless necessary (in this class for reducing text)