ME 424/426 Drawing Guidelines

Mechanical drawings are primarily used to relay design information about a device to manufacturing. Through these drawings, all of the information about various features in a device is conveyed. The key to having "good" drawings is conveying this information in a concise and clear manner.

There are several conventions used in drawings to reduce the number of dimensions and notes. Geometric Dimensioning and Tolerancing (GD&T) is currently used in industry and is an established standard by ASME. This standard makes use of symbols and tolerancing schemes to simplify and standardize drawings. The proper use of this standard is beyond the scope of this course at this time. However, if you are interested in learning more about GD&T, the standard is ASME Y14.5M-1994. In addition, there are numerous books, manuals, and training programs on GD&T.

This drawing guidelines packet will provide several conventions to use in simplifying drawings for projects. In addition to these conventions, there are numerous standards published by ASME on drawings such as formats. Corporations have created their own drawing standards based upon the products and processes in which they are involved.

Drawing Formats

There are 5 basic US sheet form sizes, A, B, C, D, and E, where E is the largest. Each increment in size has twice the area of the preceding sheet. The following lists the sheet sizes.

Sheet	A	B	C	D	E
Size	8.5 x 11	11 x 17	17 x 22	22 x 34	36 x 48

For class, any of these sizes may be used. However, A and B are the most convenient to include in reports, notebooks, etc. In ME 426, a drawing needs the following information, which is typically included in the title block.

- Name of drawing or part
- Scale
- Unit of measure
- Drawn by
- Material
- Drawing number (create your own number format, makes referring to parts easier)
- Team Name

Dimensioning

Rules of Thumb

- Do not dimension to hidden lines/features.
- Place all of the dimensions for one feature in one view if possible. For instance, place the hole diameter and position dimension on the same view.
- Combine callouts. For countersunk holes, combine the through hole and countersinking callouts into one callout.
- If possible, match units to machine units.
- Do not cross leader lines.

Symbols

The following is a list of selected symbols from ASME Y14.5M-1994.

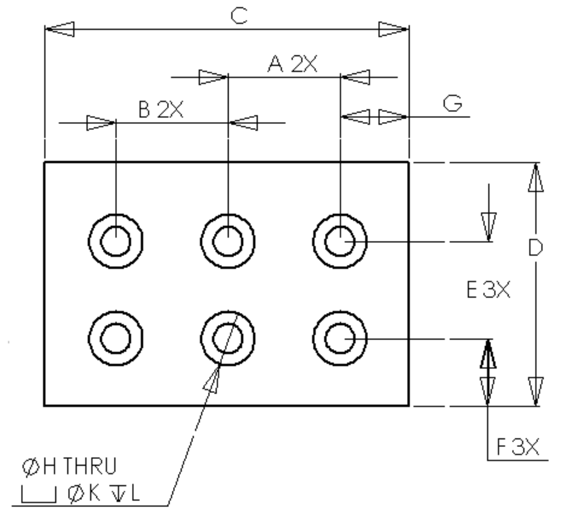
ASME Y14.5M Symbol	Description
⌴	Counterbore or Spotface
⌵	Countersink
↧	Depth: typically used with holes
⌀	Diameter
±	Plus-minus Symmetric tolerance
R	Radius
SR	Spherical Radius
S⌀	Spherical Diameter
(X.XX)	Reference Dimension
$\boxed{X.XX}$	Basic Dimension: theoretcial
16 √	Surface Finish: 16 microinch with machining
3X	Number of Places: a number followed by X indicates the number of instances

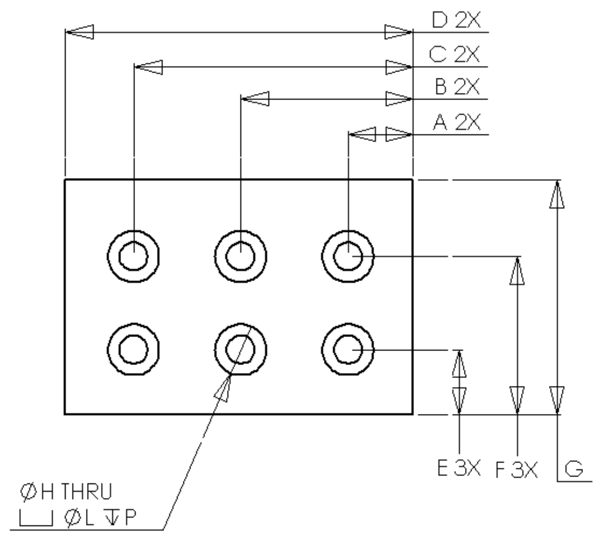
Baselines

Defining baselines to dimensions typically keeps dimensions clean and organized. It also avoids unwanted tolerance stack-ups as shown in the following example. If possible, keeping leader lines off of the part reduces confusing lines.

For both examples, the holes must be precisely located to within 2 thousandths of the nominal position with respect to the right hand edge. While both drawings do show hole location dimensions, the second drawing has less stackup error. Stackup is an analysis of the tolerances. In the first drawing, if dimensions G, A, and B had a symmetric tolerance of 0.002, the upper right hole would be acceptable if the true G dimension was within G±0.002. However, as the dimensions are presented, the upper left hole would be acceptable if its position was within G + B + A ±0.006. The 6 thousandths is the tolerance stackup of that hole. The stackup is merely the sum of the tolerances. There are three dimensions with ±0.002 tolerances.

The second drawing has all of the dimensions referenced from one feature, or baseline (the lower right hand corner). This feature could have just as easily been one of the holes. Now each hole has just one dimension in each axis, and one tolerance, ±0.002. Also note how all of the dimensions are neatly arranged along the baseline rather than placed randomly around the view.

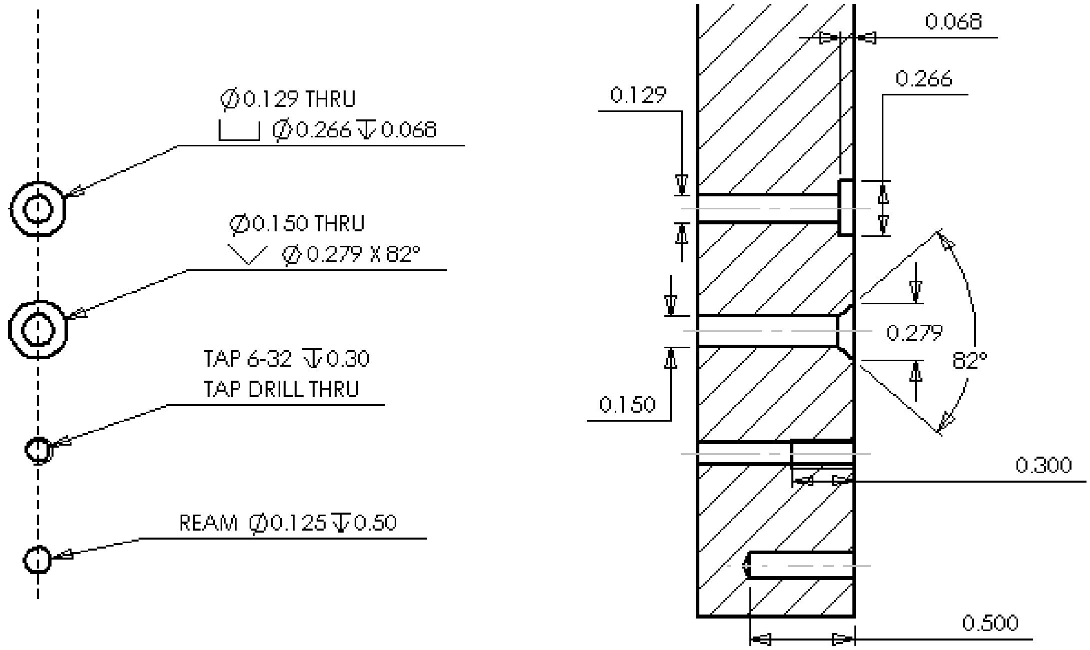




Holes/Threading

Typically the diameter rather than the radius of a complete hole is dimensioned. The following shows dimensioning conventions commonly used for hole call outs. The Machinery's handbook lists various bolt related hole sizes including counterbore diameters and clearance hole diameters under the Cap Screw sections.

These hole callouts mean: these cross-sections



Tolerances

Many drawings will globally specify tolerances in the title block based upon the number of digits in the dimension. For instance, X.XXX: ±0.005 is a common tolerance for three decimal places in inches. Typically X.XX is ±0.01. These tolerances are relatively easy to achieve with the machines in our shop for most part features.

Three forms of specifying tolerances are widely used industry. One is the symmetric tolerance and is designated by the ± symbol. This tolerance is symmetric about the nominal dimension. The second form is the bilateral tolerance, which is asymmetric about the nominal dimension. The third tolerance is the limit tolerance where the nominal dimension is replaced by the maximum and minimum permissible dimensions.

Fits

There are three general classifications of fits: running, locational, and force. ANSI standardized these fits into the following and are listed in the Machinery's Handbook under "Allowances and Tolerances."

Fit	Description	Use
RC	Running Clearance Fit	Sliding fits with accurate location
LC	Locational Clearance	For stationary parts that may be easily disassembled
LT	Locational Translation	For stationary parts, disassembly may require force
LN	Locational Interference	For stationary parts where location is important, disassembly requires force
FN	Forced Interference	Force or shrink fits

Example Drawing

The following is an example drawing showing baseline dimensioning and multiple uses of views.