UNIT C: Manufacturing Processes

Competency: D403.00

Demonstrate the basic concepts of the manufacturing processes.

Objective: D403.02

Explain the ANSI standards of applying annotations to a drawing that best describes the manufacturing process.

Introduction: The purpose of this unit is to generate student awareness of the use of detailed annotations needed to describe the manufacturing process in the development of a part.

Manufacturing Annotations: *R1 (292-328), R2 (229-275) R3 (193-223), R4 (332-385)*

A. Two Kinds of manufacturing notes:
   1. General Note – applies to general information about the part as a whole. Such as:
      FILLETS & ROUNDS ARE TO BE R.125.
      FINISH ALL OVER.
   2. Local Note – is a note that is connected to a leader pointing to the appropriate area applying to a specific machining operation.

B. Drilling Terminology and Callouts
   1. Drill - Is the process used to cut a cylindrical hole with a drill press and drill bit.
   2. Boring - Enlarges the hole slightly and making it rounder and straighter.
   3. Ream - To enlarge a hole to a more accurate size and surface quality.
   4. Machined holes by their profiles (see Figure 1):
      a. Through – a through hole is one that passes all the way through the object.
      b. Blind – a blind hole cuts into but does not pass completely through the object.
      c. Counterbore - To enlarge the end of a drill hole to a specific diameter and depth in order to recess a mating part.
      d. Countersink - To recess a hole with a conically (cone) shaped tool to provide a seat for flat head screws.
      e. Spotface - The cutting of a shallow counterbore, usually about .0625 deep (depth symbol is omitted). The spotface depth does not need to be specified. The spotface provides an accurate bearing surface for the underside of a bolt head.

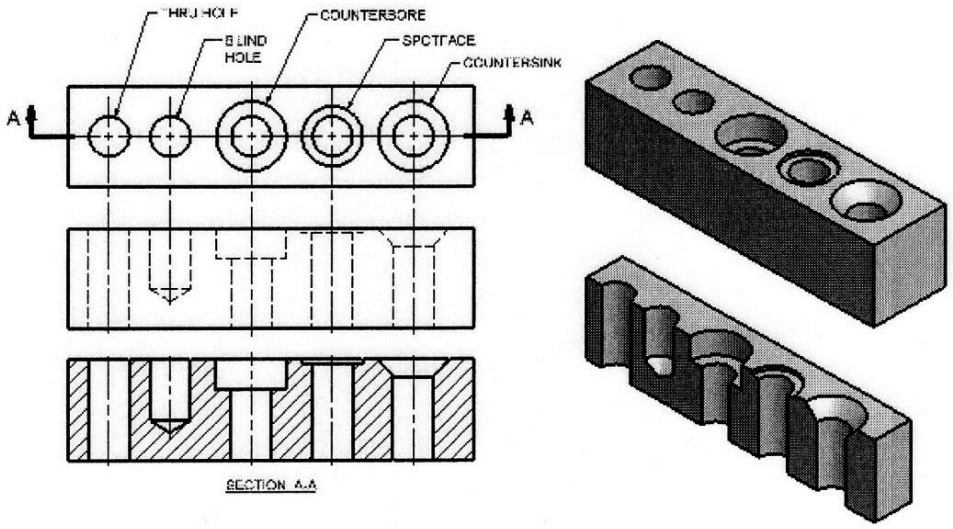

Figure 1. Types of Holes

5. Standards for annotations and symbols when identifying hole callouts.
   a. Notes should always be lettered horizontally on the drawing paper.
   b. Always attach leaders at the front of the first word of a note, or after the last word.
   c. When sizing a drill hole the arrowhead of the leader should point towards the center of the circle. When the circular view of the hole has two or more concentric circles, as for a counterbore, the arrowhead should touch the most outer circle.
   d. Fractional size drills are available drill sizes of 1/16" diameter to 3-1/2" diameter. It is common practice (*as recommended by ANSI*) to give the drill size in decimal-inch size for all diameters.
   e. For numbered or letter-size drills it is recommended that the decimal size be given in parentheses; thus,

      #25 (.1495) DRILL, E (.250) DRILL.

      *The word **DRILL** may be omitted from the note.*

   f. Metric drills are usually listed separately with a decimal-millimeter value.
   g. Repetitive drill holes with the same diameter are specified by the use of an **X** following the number of times the hole is required; thus, **4X n.375** *(four holes with a diameter of .375)*
   h. Holes equally spaced about a common center are located by giving the center of the holes and diameter of the bolt circle (BC). See Figure 2.

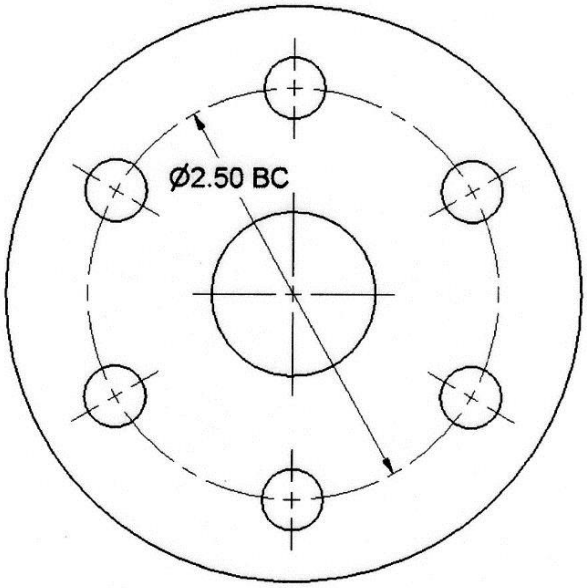

Figure 2. Holes Spaced Equally about a Center.

i. The order of the drill callout corresponds to the order of procedure in the shop in producing the hole. For example: The smaller drilled hole is given first, then the counterbore diameter, followed by the depth.

Counterbore Example:
Ø.375
⌴Ø.75 ↧.375

Countersink Example:
Ø.500
⌵Ø.75 X 82°

Spotface Example:
Ø.250
⌴Ø1.00

Note:

- The symbol is placed in front of the decimal size.
- The depth for spotface is commonly know to be .0625 and is not normally required with the hole callout.

C. Surface Texture Symbols:

1. Used to indicate that a surface is to be machined, or finished.
2. It is not necessary to show the finish marks for machining processes such as drilling, reaming, boring, etc.
3. The check mark symbol is the preferred symbol by ANSI. See Figure 3.

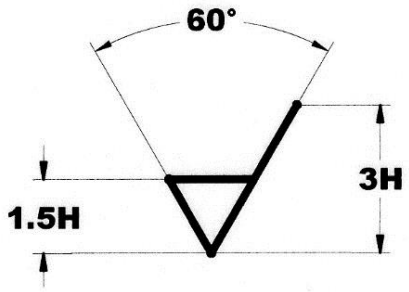

Figure 3. Surface Texture Symbol.

4. The point of the vertex of the finish mark should be directed inward toward the body of the part. Such as that of a cutting tool.
5. The finish mark symbol should be positioned to read from the bottom of the sheet or from the right side of the sheet, as shown in Figure 4.
6. The finish mark is only shown on the edge view of a finished surface and is repeated in any other view in which the surface appears as a line, even if the line is a hidden line.
7. If a part is to be finished all over, finish marks are omitted and a general note such as, **FINISH ALL OVER** or **FAO** should be placed in the lower portion of the sheet, next to the title block.

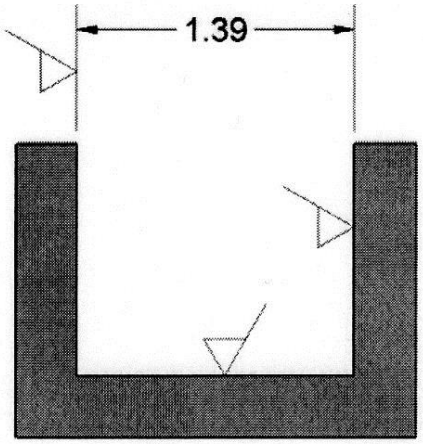

Figure 4. Orientation of Finish Marks to Machined Surfaces.

D. Knurling – is a roughened surface commonly found on thumbscrews and handles of various kinds to provide a better grip. Also can be created to provide a press fit between two parts (see Figure 5).
   1. Types of Knurls
      a. Diamond – crossing diagonal groves.
      b. Straight – parallel groves
   2. Dimensioning of Knurls.
      a. For handgrip knurls, only the pitch of the knurl (sometimes listed as fine, medium, or coarse), the type of knurl (diamond or straight), and the length of the knurl area are required.

b. For a press fit type knurl, the tolerance diameter of the class of fit is given before the actual knurling note. The most commonly used diametrical pitches (DP) are 64 DP (coarse), 96 DP (medium), 128 DP (fine), and 160 DP (extra fine).

c. A knurl symbol (hatching pattern) does not have to be shown on the drawing when a local note is applied.

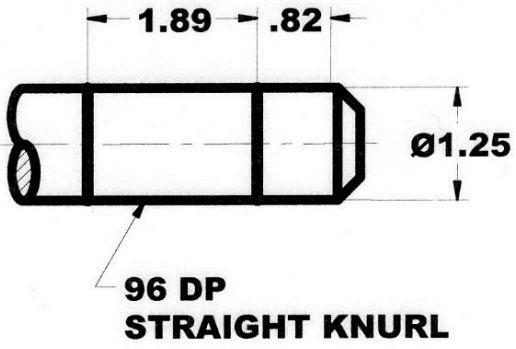

Figure 5. Dimensioning a Knurl.

E. Fillets and Rounds - are normally found on cast, forged, and plastic parts.

1. The purpose of fillets and rounds is to add strength and protection from sharp edges.
2. A rounded interior corner is called a fillet.
3. A rounded exterior corner is called a round.
4. The presence of the curved surfaces is indicated only where they appear as arcs.

F. Runouts – is the method of representing fillets in connection with plane surfaces tangent to cylinders. See Figure 6.

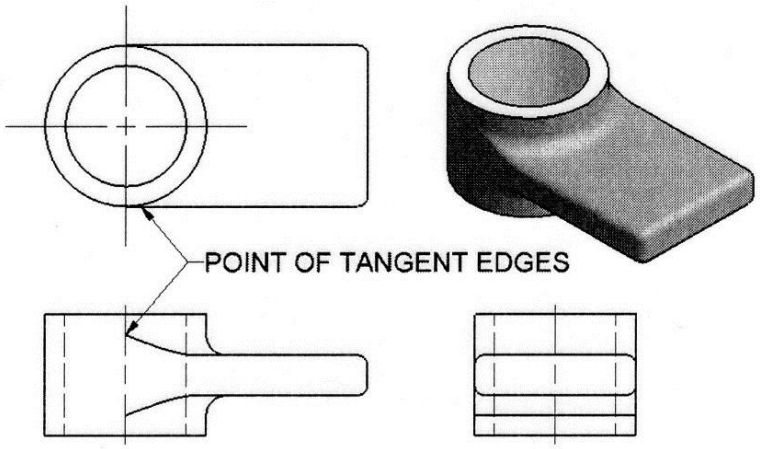

Figure 6. Conventional Fillets, Rounds, and Runouts.

G. Conventional Edges – rounded and filleted intersections eliminate sharp edges and sometimes make it difficult to present a clear shape description. True projection may actually be

misleading. Projection lines should project from the actual intersection of the surfaces as if the fillets and rounds were not present (see Figure 7).

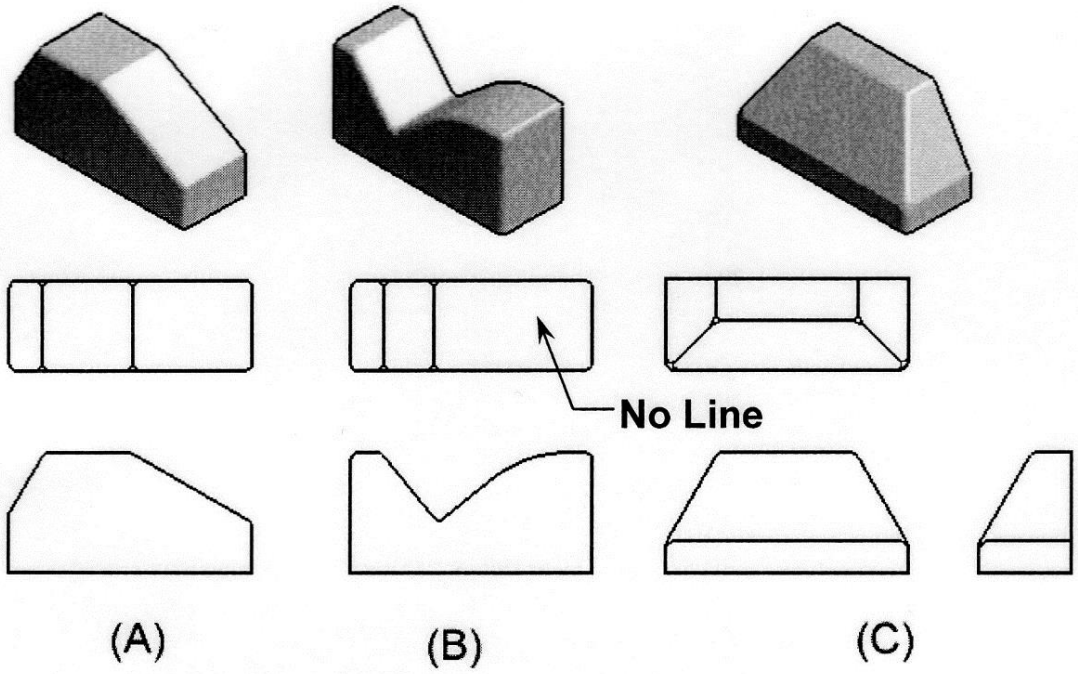

Figure 7. Conventional Edges.

H. Chamfer – is a beveled edge and it is dimensioned by giving the length of the offset and the angle (see Figure 8).

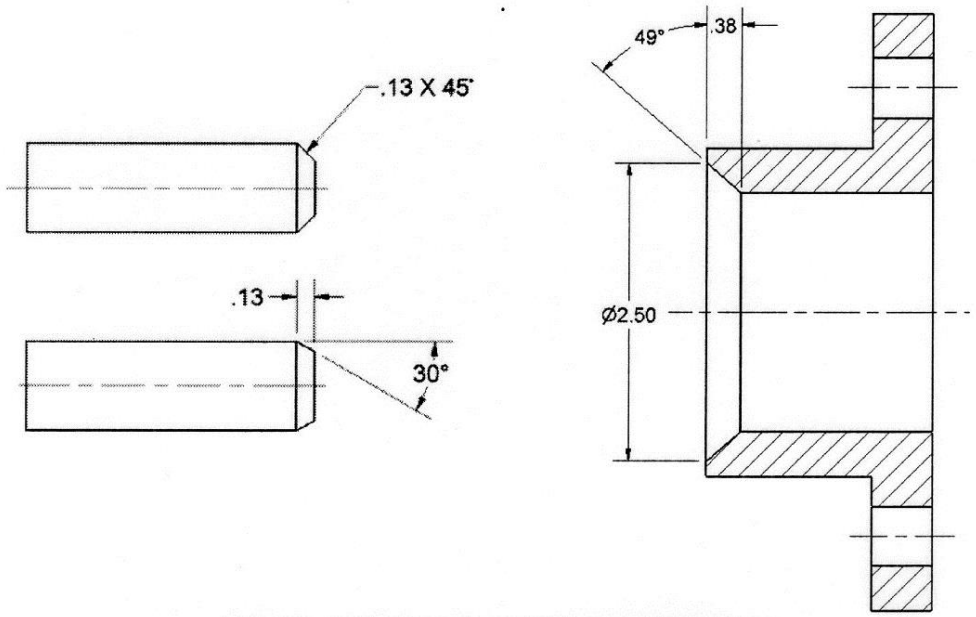

Figure 8. External and Internal Chamfers