4) Enter your WCS .

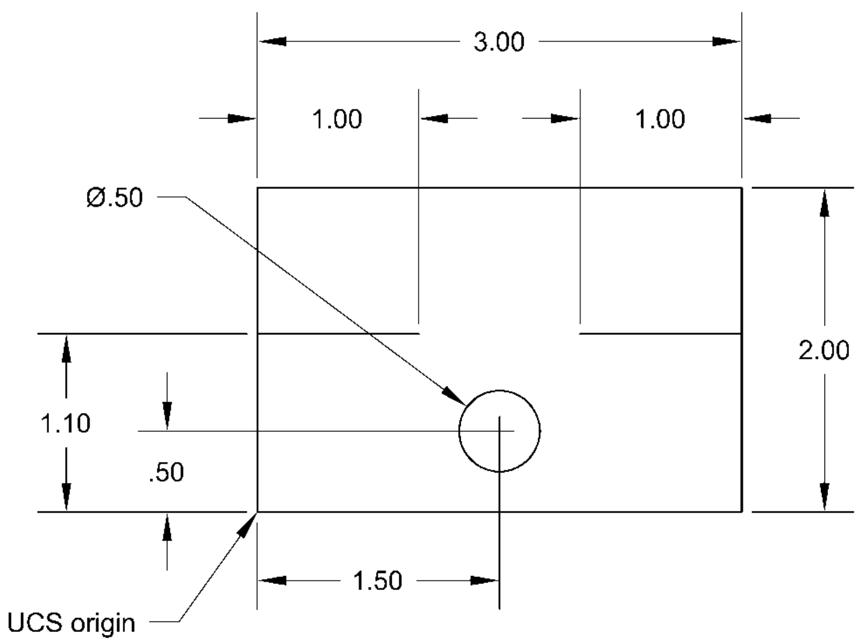
5) If you are using *Dynamic Input*, set the *Pointer Input Format* to **Absolute coordinates**.

6) In the ***Viewport*** layer, draw a **RECtangle** that indicates the edges of your limits/paper (11x8.5).

7) **Zoom All**

8) In your ***Visible*** layer, draw the visible lines of the front view.
   a) Draw a **RECtangle** that is **3** inches long and **2** inches wide near the bottom left corner of your drawing area.
   b) Set your **UCS** origin  to the bottom left corner of the front view.
   c) Draw the 2 **Lines** within the rectangle.
   d) Draw the **Circle**. (Note: ∅ = diameter)

Note: Some visible features are missing, but this is all we can do for now.



4.11.2) Drawing the right side view

1) Turn your ***Object Snap*** on and set the following object snaps to be automatically detected (Endpoint, Midpoint, Center, Quadrant, Intersection, Nearest, Perpendicular, Extension).

The dimensioning standards presented in this chapter are in accordance with the ASME Y14.5-2009 standard. This standard was created to establish a uniform way of dimensioning an engineering drawing. This also minimizes errors that could occur while reading or interpreting an engineering drawing. When a drawing conforms to the standard, it is noted in the tolerance block as shown in Figure 7.2-1. Other common sense practices will also be presented.

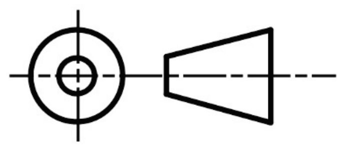
UNLESS OTHERWISE SPECIFIED DIM ARE IN INCHES TOL ON ANGLE ±.XX° 2 PL ±.XX 3 PL ±.XXX INTERPRET DIM AND TOL PER ASME Y14.5 - 2009
THIRD ANGLE PROJECTION


Figure 7.2-1: ASME Y14.5 – 2009 designation

7.3) DIMENSION APPEARANCE

7.3.1) Lines Used in Dimensioning

Dimensions require the use of *dimension*, *extension* and *leader lines*. **All lines used in dimensioning are drawn thin so that the print reader does not confuse them with visible lines.** Thin lines should be drawn at approximately 0.016 inch (0.3 mm). Figure 7.3-1 illustrates the different features of a dimension and Figure 7.3-2 illustrates different leader line configurations.

- Dimension line: A dimension line is a thin solid line terminated by arrowheads, which indicates the direction and extent of a dimension. A number is placed near the midpoint to specify the feature's size.
- Extension line: An extension line is a thin solid line that extends from a point on the drawing to which the dimension refers. There should be a visible gap between the extension line and the object, and long extension lines should be avoided.
- Leader line: A leader line is a straight inclined thin solid line that is usually terminated by an arrowhead. It is used to direct a dimension, note, symbol, item number, or part number to the intended feature on a drawing. Leader lines should not be drawn vertical or horizontal, except for a short horizontal portion extending to the first or last letter of the note. The horizontal part should not underline the note and may be omitted entirely. Leader lines may be terminated by an arrow if it ends on the outline of an object or without an arrowhead or with a dot (∅1.5 mm, minimum) if it ends within the outline of an object (see Figure 7.3-2). You should avoid creating long leaders, crossing leaders, or leaders that are parallel to features on the drawing.

Dimensioning symbols replace text and are used to minimize language barriers. Many companies produce parts all over the world. A print made in the U.S.A. may have to be read in several different countries. The goal of using dimensioning symbols is to eliminate the need for language translation. Table 7.3-4 shows some commonly used dimensioning symbols. These symbols will be used and explained throughout the chapter.

Term	Symbol	Term	Symbol
Diameter	∅	Depth / Deep	↧
Spherical diameter	S∅	Dimension not to scale	$\underline{10}$
Radius	R	Square (Shape)	□
Spherical radius	SR	Arc length	$\overset{\frown}{5}$
Reference dimension	(8)	Conical Taper	⌲
Counterbore / Spotface	⌴	Slope	⌳
Countersink	⌵	Symmetry	⌯ or =
Number of places	4X		

Table 7.3-4: Dimensioning symbols

7.4) FEATURE DIMENSIONS

The following section illustrates the standard ways common features are dimensioned.

a) A circle is dimensioned by its diameter and an arc by its radius using a leader line and a note or linear dimension. Diameter dimensions of solid parts, such as a cylinder, is given as a linear dimension. A diameter dimension is preceded by the symbol "∅", and a radial dimension is preceded by the symbol "R". On older drawings you may see the abbreviation "DIA" placed after a diameter dimension and the abbreviation "R" following a radial dimension. Figure 7.4-1 illustrates the diameter and radius dimensions.

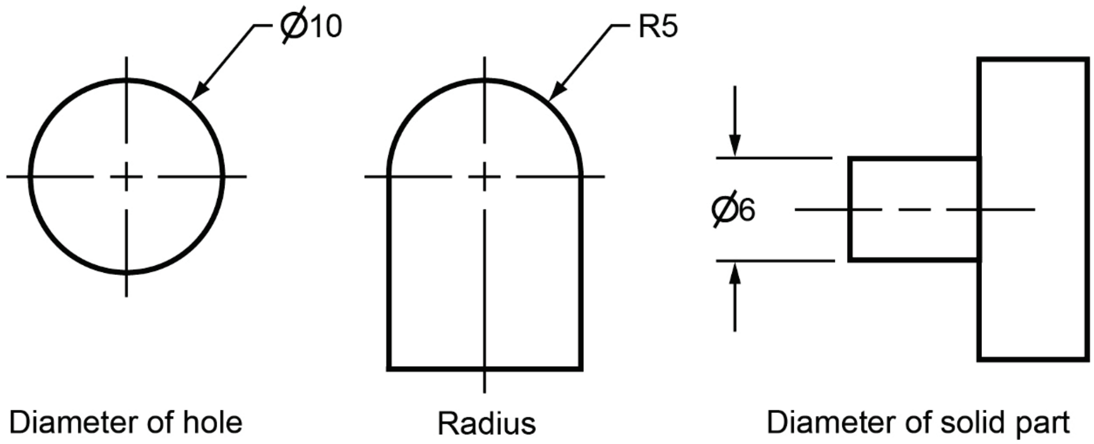


Figure 7.4-1: Diameter and radius dimensions

b) The depth (↧) of a blind hole is specified under the hole's diameter dimension and is the depth of the full diameter from the surface of the object. Figure 7.4-2 illustrates how a dimension for a blind hole (i.e. a hole that does not pass completely through the object) is given.

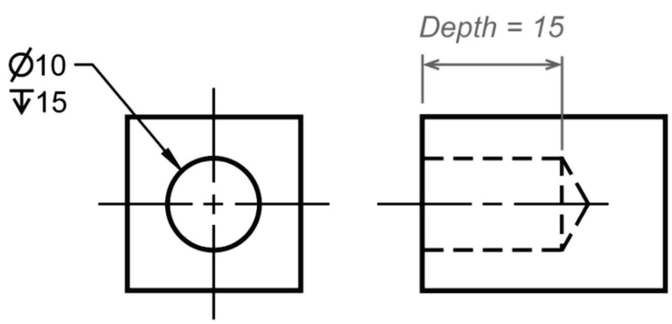


Figure 7.4-2: Dimensioning a blind hole.

c) If a hole goes completely through the feature and it is not clearly shown on the drawing, the abbreviation "THRU" follows the dimension.

d) If a dimension is given to the center of a radius, a small cross is drawn at the center. Where the center location of the radius is unimportant, the drawing must clearly show that the arc location is controlled by other dimensioned features such as tangent surfaces. Figure 7.4-3 shows several different radius configurations.

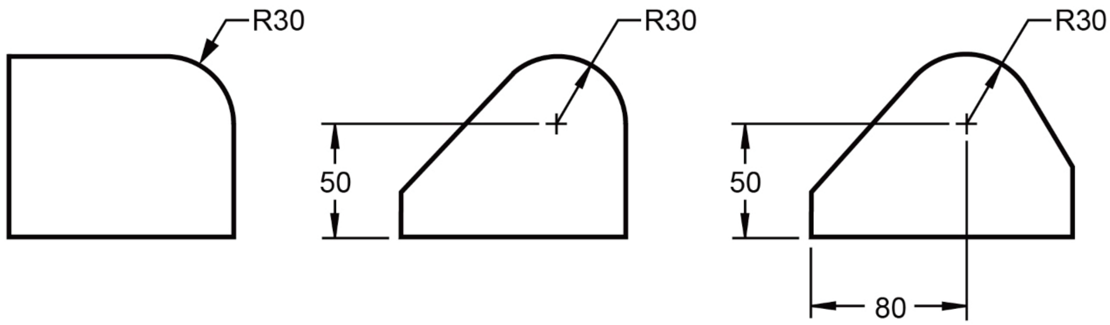


Figure 7.4-3: Dimensioning radial features.

e) A complete sphere is dimensioned by its diameter and an incomplete sphere by its radius. A spherical diameter is indicated by using the symbol "S∅" and a spherical radius by the symbol "SR". Figure 7.4-4 illustrates the spherical diameter and spherical radius dimensions.

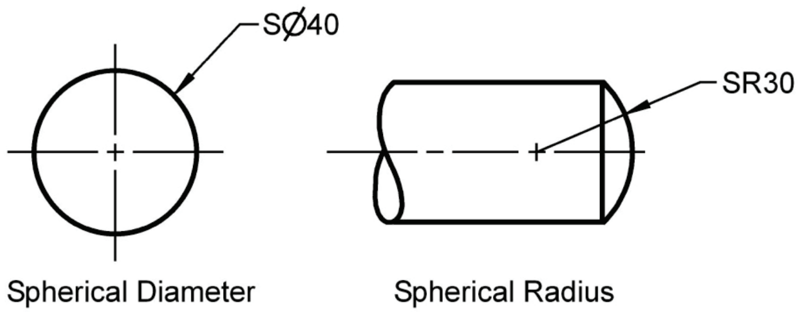


Figure 7.4-4: Dimensioning spherical features.

f) Holes are dimensioned by giving their diameter and location in the circular view. Cylinders are dimensioned by giving their diameter and length in the rectangular view, and are located in the circular view. By giving the diameter of a cylinder in the rectangular view, it is less likely to be confused with a hole.

Try Exercise 7.4-2

g) Repetitive features or dimensions are specified by using the symbol "X" along with the number of times the feature is repeated. There is no space between the number of times the feature is repeated and the "X" symbol; however, there is a space between the symbol "X" and the dimension (i.e. 8X ∅10).

h) Equally spaced features are specified by giving the number of spaces followed by the repeated feature symbol "X", a space, and then the dimension value of the space as shown in Figure 7.4-5. The total distance may be given in parentheses after the dimension and one spacing may be dimensioned and given as a reference value.

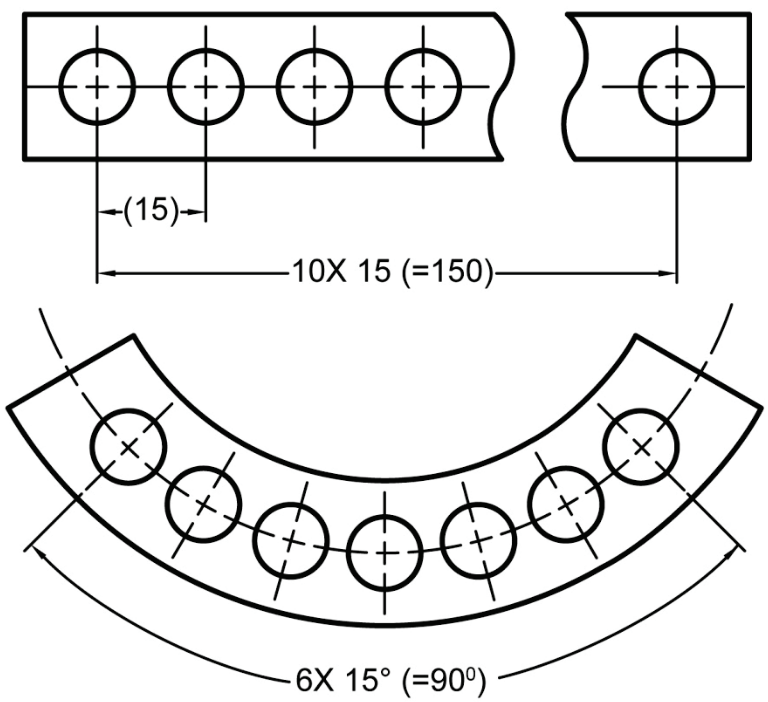


Figure 7.4-5: Equally spaced features

Try Exercise 7.4-3

m) If a part is symmetric, dimensions on one side of the center line of symmetry may only be given. The center line of symmetry is indicated by using the symbol "⌯" or "=". On older drawings you might see the symbol "℄" used instead. Figure 7.4-10 illustrates the use of the symmetry symbol.

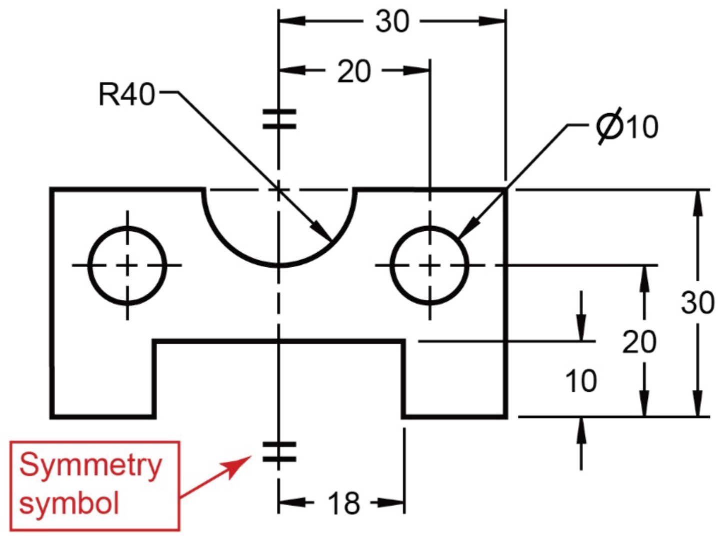


Figure 7.4-10: Center line of symmetry.

n) Counterbored holes are specified by giving the diameter (∅) of the drill (and depth if appropriate), the diameter (∅) of the counterbore (⌴), and the depth (↧) of the counterbore in a note as shown in Figure 7.4-11. If the thickness of the material below the counterbore is significant, this thickness rather than the counterbore depth is given.

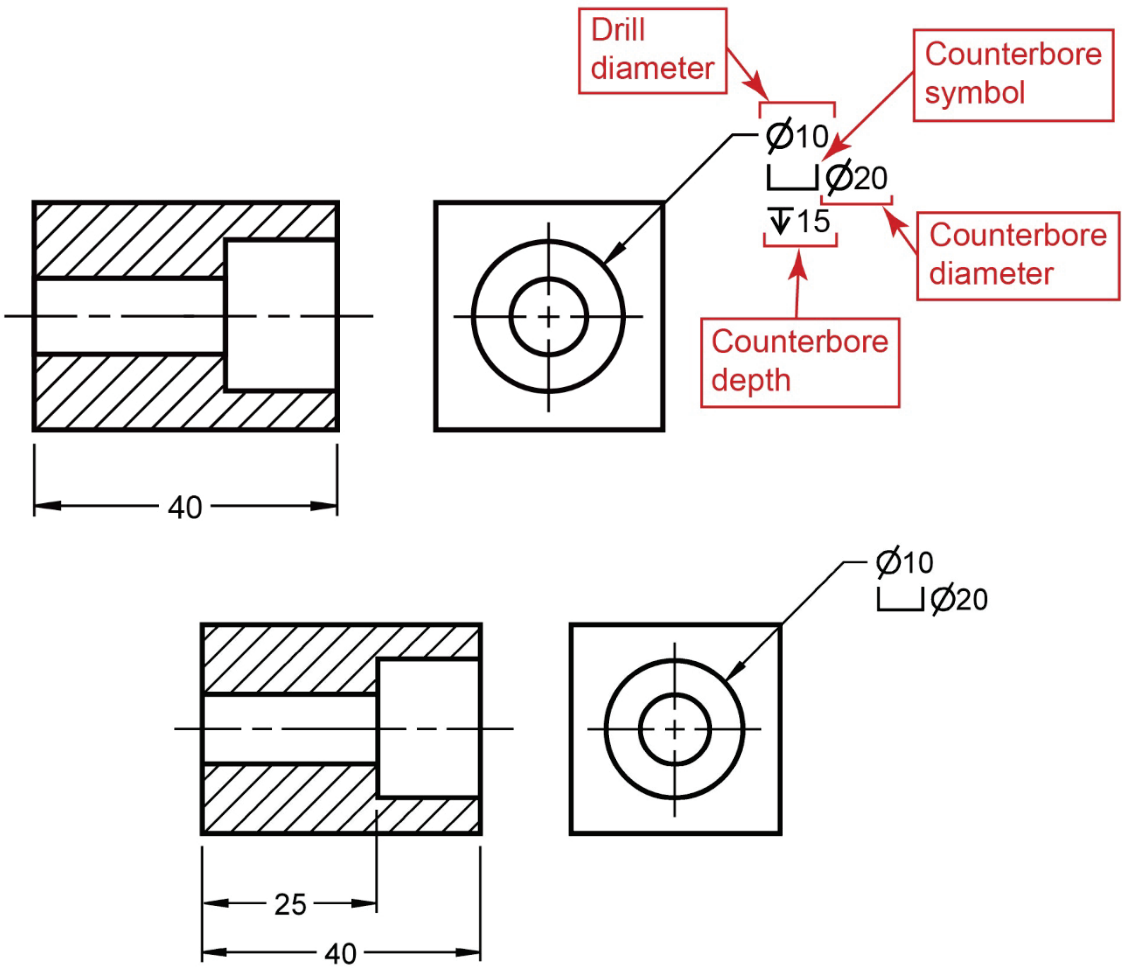


Figure 7.4-11: Counterbored holes.

Application Question 7.4-1

What is the purpose of a counterbored hole? (See Figure 7.4-11.)

o) Spotfaced holes are similar to counterbored holes. The difference is that the machining operation occurs on a curved surface. Therefore, the depth of the counterbore drill is not given in the note. It must be specified in the rectangular view as shown in Figure 7.4-12.

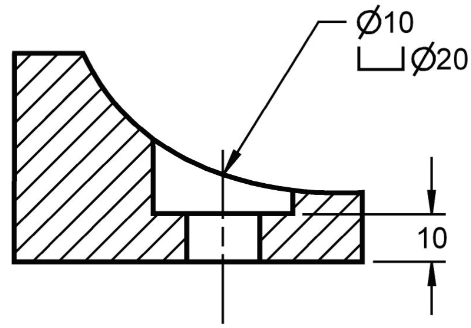


Figure 7.4-12: Spotfaced holes.

p) Countersunk holes are specified by giving the diameter (∅) of the drill (and depth if appropriate), the diameter (∅) of the countersink (⌵), and the angle of the countersink in a note as shown in Figure 7.4-13.

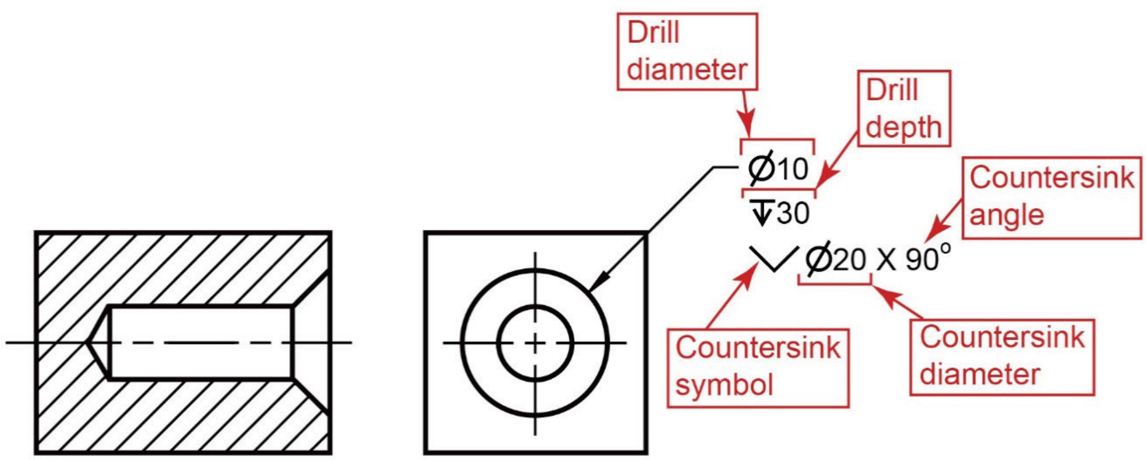


Figure 7.4-13: Countersunk holes.

7.5.3) Manufacturing

Rule 5) Don't specify manufacturing processes with your dimension. For example, words such as "DRILL", "REAM", and "PUNCH" should not be placed with the feature size. On old prints, you may see a hole dimension specified as ".12 DRILL". This dimension specifies a manufacturing process. The dimension should read "∅.12". The exception is when the process is essential to the definition of the engineering requirements.

Rule 6) Parts identified by gage or code numbers are dimensioned using their actual decimal size. The gage or code number may be shown in parentheses next to the dimension. Sometimes you may encounter a drawing that specifies standard drills, broaches, and the like by a number or letter.

Rule 7) All dimensions are applicable at 20ºC unless otherwise specified. If the material has a high thermal expansion coefficient, compensations may be made.

Rule 8) Part sizes prior to processing may be specified on a drawing. In general, final part dimensions are specified on a drawing; however, non-mandatory processing dimensions may also be present. These dimensions are identified using the note "NONMANDATORY (MFG DATA)". Non-mandatory processing dimensions specify part sizes prior to processing such as finishing the part or part shrinkage.

Your choice of dimensions will directly influence the method used to manufacture a part. However, your **choice of dimensions should depend on the function and the mating relationship of the part**, and then on manufacturing. Learning the topics in this section and the upcoming sections will guide you when choosing your dimension units, decimal places and the dimension's starting point. Even though dimensions influence how the part is made, the manufacturing process is not specifically stated on the drawing. Listed are a few examples of how dimension placement and dimension text influence how the part gets manufactured.

- ✓ Increasing the number of decimal places (e.g. 1.00 goes to 1.000) will increase the cost of manufacturing. Some manufacturing processes are not as accurate as other processes (e.g. *casting* is not very accurate; *grinding* is more accurate.) More accurate processes are generally more expensive.
- ✓ Identifying a datum feature (i.e. a surface from which most dimensions originate) will influence the surfaces used in the manufacturing process. This topic is covered in section 7.5.4.
- ✓ Dimension placement also influences error build up. This is how much error is allowed during the manufacturing process. This topic is covered in section 7.5.5.

READING DIMENSIONS QUESTIONS

Name: ________________________________ Date: _______________

Dimension appearance

Q7-1) A detailed drawing is an orthographic projection with .... (Circle all that apply.)

a) dimensions.
b) notes.
c) manufacturing specifications.
d) everything necessary to manufacture and inspect the part as intended by the designer.

Q7-2) Dimensions generally take the form of ... (Circle all that apply.)

a) linear dimensions
b) extension lines
c) angular dimensions
d) notes

Q7-3) Which line type does not ever have arrowheads? (dimension, extension, leader)

Dimensioning and locating features

Q7-4) What is this symbol? ↧

a) repeated feature
b) depth
c) counterbore
d) countersink
e) symmetry

Q7-5) What is this symbol? ⌵

a) repeated feature
b) depth
c) counterbore
d) countersink
e) symmetry

Q7-6) What is this symbol? ⌯ or =

a) repeated feature
b) depth
c) counterbore
d) countersink
e) symmetry

Q7-7) X is the symbol used for repeated features. What else is this symbol used for?

a) chamfer
b) concentricity
c) counterbore
d) symmetry

Q7-8) A reference dimension is given within ....

a) brackets.
b) double quotes.
c) parentheses.
d) single quotes.

Q7-9) Is the following dimension a repeated feature dimension or a chamfer dimension?

2X 45º

Dimensioning/manufacturing

Q7-10) What unit of measure is most commonly used on English drawing?

Q7-11) What unit of measure is most commonly used on metric drawing?

8.4.6) Alternative units

- **DIMALT**: Adds an additional dimension text in an alternative unit.
    - 0 = Off
    - 1 = On
- **DIMALTD**: Controls the number of decimal places in the alternative unit.
- **DIMALTF**: Controls the conversion factor of the alternative unit. For example, the conversion factor from inches to millimeters is 25.4.

8.4.7) Toleranced dimensions

- **DIMLIM**: Presents dimensions in limit form.
- **DIMTOL**: Presents dimensions in tolerance form.
- **DIMTM**: Sets the negative tolerance value.
- **DIMTP**: Sets the positive tolerance value.

8.4.8) Miscellaneous

- **DIMARCSYM:** Controls whether or not an arc symbol will be placed above an arc length dimension.
    - 0 = Before the dimension text.
    - 1 = Above the dimension text.
    - 2 = Will not display the arc length symbol.

8.5) ASSOCIATIVE DIMENSIONS

Associative dimensions are dimensions that are associated with a geometric object or a particular feature of your part. This means that if the feature is changed, the associated dimension value will change. For example, if the diameter of a circle is 10 mm then the diameter dimension value will read ∅10. If you subsequently change the diameter of the circle to 20 mm within the *Properties* window, the dimension value will automatically change to ∅20. Associativity is broken if you manually type in the dimension text, replace the dimension text or EXPLODE the dimension. Leader dimensions are not associative. The dimension commands that are related to associativity are:

- **DIMDISASSOCIATE:** Removes associativity from a selected dimension.
- **DIMREASSOCIATE:** Associates a selected dimension to geometric objects.
- **DIMREGEN:** Updates the locations of all associative dimensions.
- **DIMASSOC:** Controls the associativity of dimensions and whether dimensions are exploded.

8.6) ANNOTATIVE OBJECTS

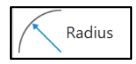
Annotative objects are objects that can support multiple viewport scales. Consider the following situation. I start a metric drawing and set my text height to 3 mm. If I print at a 1:1 scale, my text height will measure 3 mm on the printed page. However, if I print at a 1:2 scale my text height will only measure 1.5 mm. Annotative text adjusts its height so that no matter what viewport scale you select the text will always measure 3 mm on the printed page. Figure 8.6-1 shows an example of regular text and annotative text at three different viewport scales. Notice that the regular text height increases or decreases depending on the viewport scale, and the annotative text height never changes.

3) Space the linear dimensions evenly.

   a) <u>Command:</u> **dimspace** or <u>*Dimensions* panel:</u> 
   b) `Select base dimension:` Select the 5 mm dimension.
   c) `Select dimensions to space:` Select the 35 mm and then the 55 mm dimension.
   d) `Select dimensions to space:` **`Enter`**
   e) `Enter value or [Auto] <Auto>:` **a**
   f) Notice that the AUTO option spaces the dimensions too close.
   g) Repeat the above process using **8** mm as the spacing.
   h) Evenly space the vertical linear dimensions.

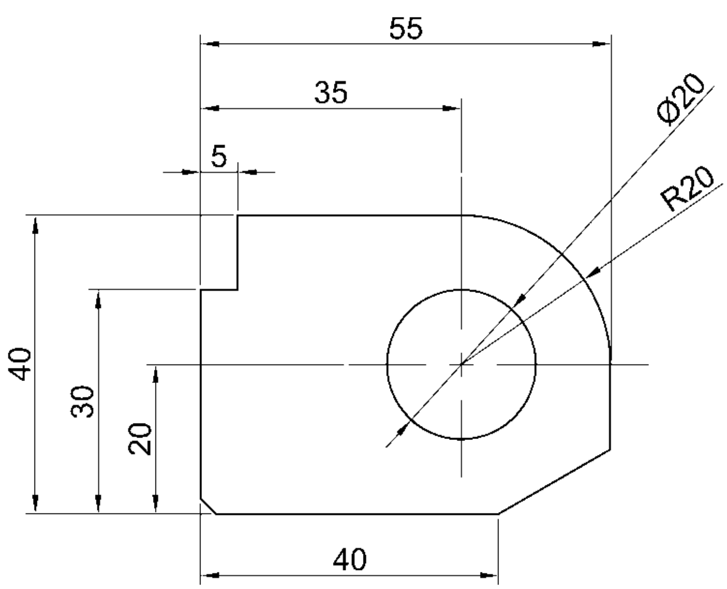
8.7.3) Drawing diameter, radius, and angular dimensions

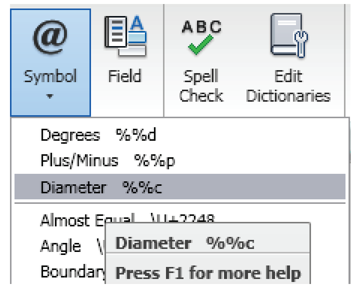
1) Dimension the radius and diameter.

   a) <u>Command:</u> **dimradius** or <u>*Dimensions* panel:</u> 

   b) `Select arc or circle:` Select the circle.
   c) `Specify radius dimension location or [Mtext, Text, Angle]:` Pull the dimension out and away from the object and left click when the dimension is in the approximate position shown. Notice that the radius symbol "R" is automatically placed in front of the dimension value.
   d) Repeat the process to add the ∅20 diameter dimension (**DIMDIAMETER**, 

).





3) Add a diameter symbol to the 1.00 diameter cylinder dimension.
   a) Command: **ddedit**
   b) `Select an annotation object or [Undo]:` Select the 1.00 dimension text of the cylinder. Move the cursor to the front of the text.
   c) *Text Editor* tab:
      i. The symbols menu is available by selecting the **@** icon. Select ***Diameter*** from the menu.
      ii. 

   d) The dimension text should now read ∅**1.00**.

4) On your own, change the chamfer dimension from 4X .12 X .12 to **4X .12 X 45°**. Insert a degree symbol in the same way that you inserted a diameter symbol.

5) Add the repeated feature and depth text to the ∅.50 dimension text.
   a) Command: **ddedit**
   b) `Select an annotation object or [Undo]:` Select the ∅.50 dimension text. Type **2X** and a space. Then use the **`Right arrow`** to position your cursor at the end of the dimension text. Press **`Enter`** to start a new line of text and type **x.50**. Make sure that the **x** is in lower case.
   c) *Text Editor* tab:
      i. Highlight the lower case **x** and change its font to ***gdt***.
      ii. Align the text to the right.
      iii. 

6) On your own, change the countersink dimension text. The following is a list of useful *gdt* font symbols.

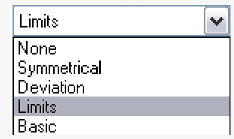
   i = ⌯ n = ∅ v = ⌴ w = ⌵ x = ↧

7) Space the dimension evenly.

8) 

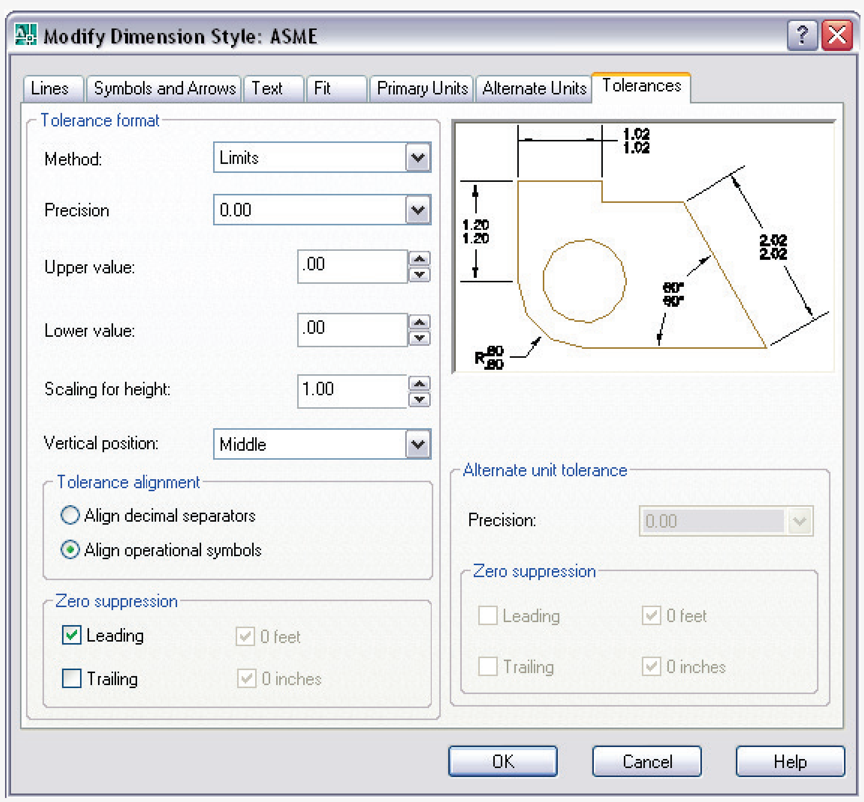
Setting tolerance parameters

1) Command: **dimstyle** or *Dimensions* panel: 
2) *Dimension Styles Manager* window: ***Override…*** (Select ***New…*** if you want to create a *Tolerance* dimension style.)
3) *Override Current Style: ASME* window – *Tolerance* tab - *Tolerance format* area:
   a) *Method* field: Select the tolerance form.



   - None = No tolerance
   - Symmetric = A plus-minus tolerance that is symmetric.
   - Deviation = A plus-minus tolerance.
   - Limits = A limit tolerance
   - Basic = Places a box around the dimension indicating a basic (non-toleranced) dimension.

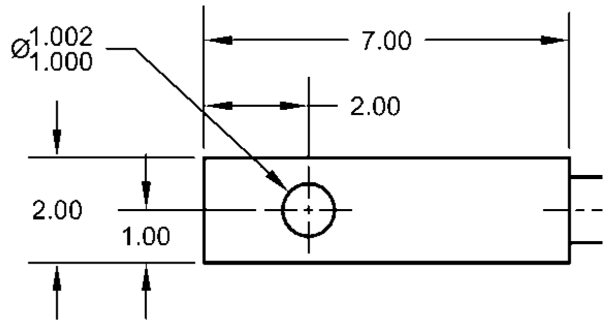
   b) *Precision* field: Select the number of decimal places that your dimension will have.
   c) *Upper value* field: Enter the value that will be added to the basic or nominal size.
   d) *Lower value* field: Enter the value that will be subtracted from the basic or nominal size. (Note: If a negative number is placed in this box, the value will be added to the basic size.)
   e) *Vertical position field:* Select the position of the dimension symbols such as ∅ and R.
   f) ***OK***
4) *Dimension Styles Manager* window: ***Close***
5) Newly created dimensions will have the override properties. To change an existing dimension, use the *update* icon  located in the *Dimensions* panel.



e) _Dimensions_ panel: 

f) `Select objects:` Select the ∅1.00 dimension of the hole.

g) `Select objects:` **Enter**



2) Change the diameter of the shaft to a limit form dimension using a RC3 clearance fit. Looking at the fit tables, the amount that is added to and subtracted from the basic size of the shaft (1.25) is -0.0010 and 0.0016. Remember to add the diameter symbol if you have not done so already. Don't worry; your dimensions will look much smaller.

a) Command: **dimstyle** or _Dimensions_ panel: 

b) _Dimension Styles Manager_ window: ***Override…***

c) _Override Current Style: ASME_ window – _Tolerance_ tab – _Tolerance format_ area:

   i. _Method_ field: ***Limits***

   ii. _Precision_ field: ***0.0000***

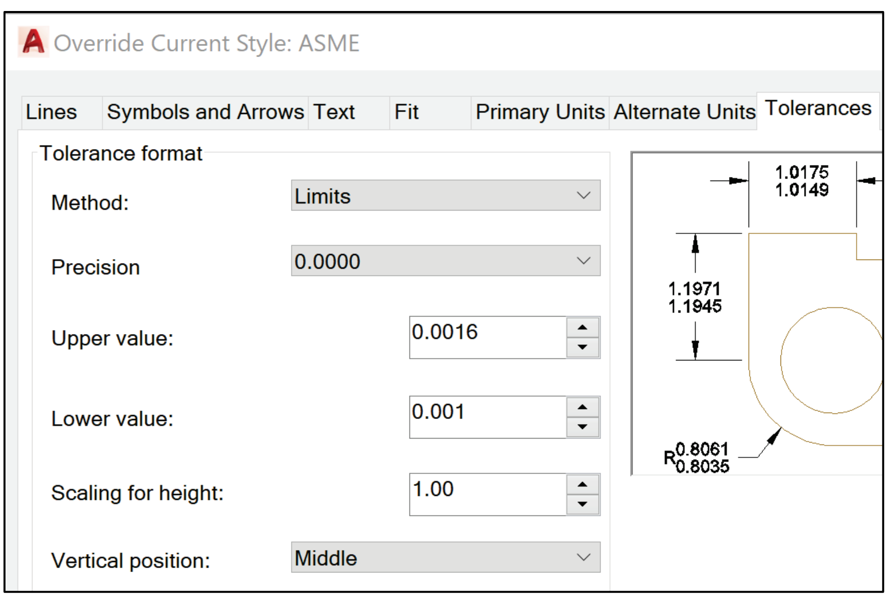
   iii. _Upper value_ field: **0.0016**

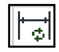
   iv. _Lower value_ field: **0.001**

   v. _Vertical position_ field: ***Middle***

   vi. ***OK***

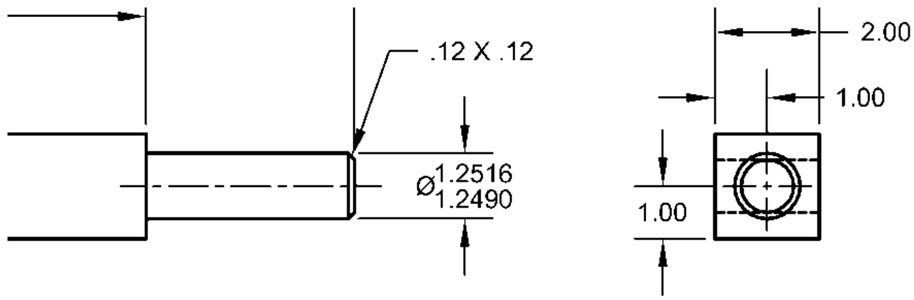
d) _Dimension Styles Manager_ window: ***Close***



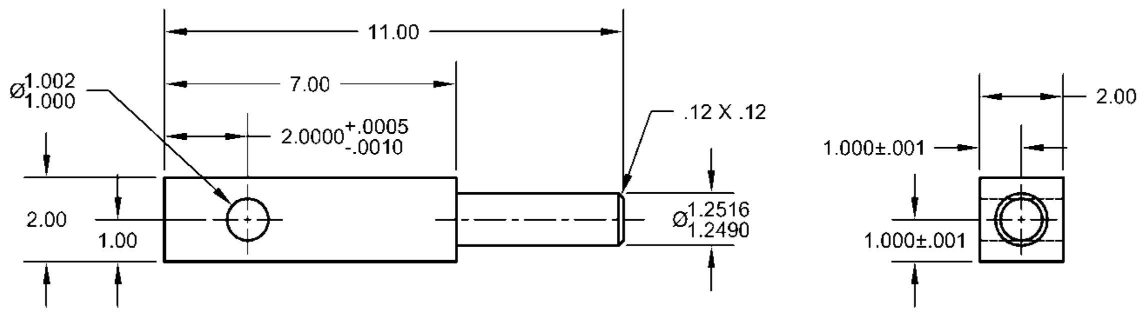
e) _Dimensions_ panel: 

f) `Select objects:` Select the ∅1.25 dimension of the hole.

g) `Select objects:` **`Enter`**



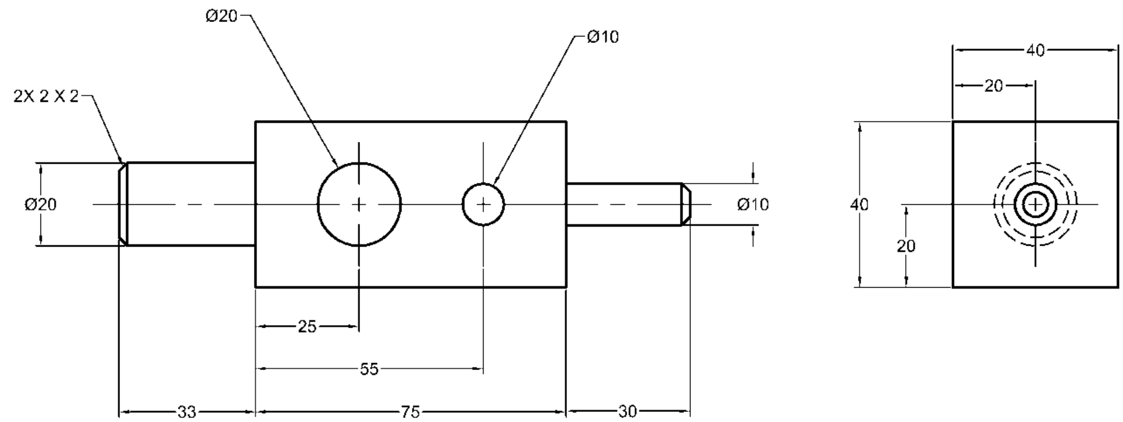
3) In a similar fashion, apply a ±0.001 ***symmetric*** tolerance to the dimensions that locate the shaft and a +0.0005 and -0.001 ***deviation*** tolerance to the 2.00 hole location dimension. The basic size and tolerance values should have the same number of decimal places and the leading zero on the tolerance values should be suppressed.



4) Save your drawing, enter ***Layout*** space, **insert** your title block and ***print*** your drawing using a 1:3 scale. By default, there is no 1:3 scale. You will have to add a custom scale and then add that scale support to your annotative objects.

TOLERANCING IN AUTOCAD PROBLEMS

P14-1) Draw the following object including dimensions. Apply a H7/g6 Sliding clearance fit to the ∅20 hole and shaft. Apply a U7/h6 force fit to the ∅10 hole and shaft. Insert your title block and print.



P14-2) Draw and dimension the *Drive Pulley* using proper dimensioning techniques. This *Drive Pulley* is part of the *Pulley Assembly* given in the Assembly chapter problem section. Notice that the dimensioned isometric drawing does not always use the correct symbols or dimensioning techniques.

- Part name = Drive Pulley
- Part No. = 2
- Material = Steel
- Required = 1

P14-3) Draw and dimension the *Follower Pulley* using proper dimensioning techniques. This *Follower Pulley* is part of the *Pulley Assembly* given in the Assembly chapter problem section. Notice that the dimensioned isometric drawing does not always use the correct symbols or dimensioning techniques.

- Part name = Follower Pulley
- Part No. = 3
- Material = Steel
- Required = 1

P14-4) Draw and dimension the *Shaft* using proper dimensioning techniques. This *Shaft* is part of the *Pulley Assembly* given in the Assembly chapter problem section. Notice that the dimensioned isometric drawing does not always use the correct symbols or dimensioning techniques.

- Part name = Shaft
- Part No. = 4
- Material = Hardened Steel
- Required = 1

- Tolerance Position: The tolerance position specifies the amount of allowance and is indicated by a letter. Uppercase letters are used for internal threads and lower-case letters for external threads. The letter "e" is used for large allowances, "g" and "G" are used for small allowances, and "h" and "H" are used for no allowance.

5. Right handed or left handed thread: Right handed threads are indicated by the symbol "RH" and left handed threads are indicated by the symbol "LH." Right handed threads are assumed if none is stated.
6. Depth of thread: The thread depth is given at the end of the thread note and indicates the thread depth for internal threads, not the tap drill depth.

Exercise 15.7-1: Metric thread note components

Identify the different components of the following Metric thread notes.

M10 x 1.5 – 4h6h – RH

M	
10	
1.5	
4h	
6h	
Internal or External	
RH	

15.7.2) Metric thread tables

Standard screw thread tables are available in order to look up the major diameter, threads per inch, tap drill size, and minor diameter for a particular thread. These thread tables are given in the ASME B1.13M-2001 standard and are given in Appendix B.

Exercise 15.7-2: Metric thread tables

For a ∅16 internal Metric thread, what are the two available pitches and the required tap drill diameter and the corresponding minor diameter for the mating external thread?

Pitch	Tap drill size	Minor DIA

Which has the finer thread?

The finer thread is M16 x (   )

CREATING THREADS AND FASTENERS PROBLEMS

Name: ________________________________ Date: _______________

P15-1) Write the thread notes for the following external threads. Also, what are the minor diameter and the pitch? Thread class = 2.

	(a)	(b)	(c)	(d)	(e)	(f)	(g)
Major ∅	1/4	7/8	3/4	1/2	1	3/8	5/16
Series	Fine	Coarse	Fine	Coarse	Fine	Extra Fine	Coarse

Thread Note	
Minor diameter	
Pitch	

Thread Note	
Minor diameter	
Pitch	

Thread Note	
Minor diameter	
Pitch	

P15-2) Write the thread notes for the following internal threads. Also, what are the tap drill size and/or diameter and the pitch? Thread class = 3.

	(a)	(b)	(c)	(d)	(e)	(f)	(g)
Major ∅	7/16	1/4	5/8	1 1/4	3/8	1/2	1
Series	Fine	Coarse	Fine	Coarse	Fine	Extra Fine	Coarse

Thread Note	
Tap drill size and/or diameter	
Pitch	

Thread Note	
Tap drill size and/or diameter	
Pitch	

Thread Note	
Tap drill size and/or diameter	
Pitch	

P15-3) Write the thread notes for the following threads. Also, what is the major diameter in inches?

	(a)	(b)	(c)	(d)	(e)	(f)	(g)
Major ∅	#0	#2	#4	#5	#6	#8	#10
Series	Fine	Coarse	Fine	Coarse	Fine	Fine	Coarse

Thread note	
Major diameter	

Thread note	
Major diameter	

Thread note	
Major diameter	

Name: ______________________________ Date: ______________

P15-4) Write the thread notes for the following external threads. Also, what are the minor diameter and the number of threads per mm?

	(a)	(b)	(c)	(d)	(e)	(f)	(g)
Major ∅	M3	M4	M8	M10	M12	M20	M24
Series	Coarse	Coarse	Fine	Coarse	Fine	Fine	Coarse

Thread Note	
Minor diameter	
# of threads per mm	

Thread Note	
Minor diameter	
# of threads per mm	

Thread Note	
Minor diameter	
# of threads per mm	

P15-5) Write the thread notes for the following internal threads. Also, what are the tap drill size and/or diameter and the number of threads per mm?

	(a)	(b)	(c)	(d)	(e)	(f)	(g)
Major ∅	M1.6	M5	M6	M12	M18	M22	M27
Series	Coarse	Coarse	Coarse	Coarse	Fine	Fine	Coarse

Thread Note	
Tap drill size and/or diameter	
# of threads per mm	

Thread Note	
Tap drill size and/or diameter	
# of threads per mm	

Thread Note	
Tap drill size and/or diameter	
# of threads per mm	

Name: ________________________________ Date: _______________

15-6) Fill in the given table for a hex head bolt with the following major diameters.

	(a)	(b)	(c)	(d)	(e)	(f)	(g)	(h)
Major ∅	1/4	5/16	1/2	7/8	1	9/16	3/8	7/16

Major diameter	
Width across flats	
Max. width across corners	
Head height	
Normal clearance hole	

Major diameter	
Width across flats	
Max. width across corners	
Head height	
Normal clearance hole	

Major diameter	
Width across flats	
Max. width across corners	
Head height	
Normal clearance hole	

P15-7) Fill in the given table for a hexagon (socket) head cap screw with the following major diameters.

	(a)	(b)	(c)	(d)	(e)	(f)	(g)	(h)
Major ∅	1/4	5/16	1/2	#8	#5	9/16	3/8	#10

Major diameter	
Max. head diameter	
Max. head height	
Normal clearance hole	
Counterbore diameter	
Counterbore depth	

Major diameter	
Max. head diameter	
Max. head height	
Normal clearance hole	
Counterbore diameter	
Counterbore depth	

15-8) Fill in the given table for a slotted flat countersunk head cap screw with the following major diameters.

	(a)	(b)	(c)	(d)	(e)	(f)	(g)	(h)
Major ∅	1/4	5/16	3/8	7/16	1/2	9/16	5/8	3/4

Major diameter	
Max. head diameter	
Max. head height	
Normal clearance hole	
Countersink diameter	
Countersink angle	

Major diameter	
Max. head diameter	
Max. head height	
Normal clearance hole	
Countersink diameter	
Countersink angle	

Name: ________________________________ Date: _______________

P15-9) Fill in the given table for a hex head bolt with the following major diameters.

	(a)	(b)	(c)	(d)	(e)	(f)	(g)	(h)
Major ∅	M5	M12	M20	M30	M36	M48	M14	M24

Major diameter	
Max. width across flats	
Max width across corners	
Max. head height	
Thread length for a screw that is shorter than 125 mm	
Normal clearance hole	

Major diameter	
Max. width across flats	
Max. width across corners	
Max. head height	
Thread length for a screw that is shorter than 125 mm	
Normal clearance hole	

P15-10) Fill in the given table for a socket head cap screw with the following major diameters.

	(a)	(b)	(c)	(d)	(e)	(f)	(g)	(h)
Major ∅	M1.6	M2.5	M4	M6	M12	M16	M24	M42

Major diameter	
Max. head diameter	
Max. head height	
Normal clearance hole	
Counterbore diameter	
Counterbore depth	

Major diameter	
Max. head diameter	
Max. head height	
Normal clearance hole	
Counterbore diameter	
Counterbore depth	

Name: ________________________________ Date: _______________

P15-11) Fill in the given table for a flat countersunk head cap screw with the following major diameters.

	(a)	(b)	(c)	(d)	(e)	(f)	(g)	(h)
Major ∅	M16	M3	M12	M5	M6	M8	M10	M4

Major diameter	
Head diameter	
Head height	
Normal clearance hole	
Countersink diameter	
Countersink angle	

Major diameter	
Head diameter	
Head height	
Normal clearance hole	
Countersink diameter	
Countersink angle	

SP15-1) Write the thread notes for the following external threads. Also, what are the minor diameter and the pitch? Thread class = 2. The answer to this problem is given on the *Independent Learning Content*.

Major ∅	7/16
Series	Coarse
Thread Note	
Minor diameter	
Pitch	

SP15-2) Write the thread notes for the following internal threads. Also, what are the tap drill size and/or diameter and the pitch? Thread class = 3. The answer to this problem is given on the *Independent Learning Content*.

Major ∅	9/16
Series	Fine
Thread Note	
Tap drill size and/or diameter	
Pitch	

SP15-3) Write the thread notes for the following threads. Also, what is the major diameter in inches? The answer to this problem is given on the *Independent Learning Content*.

Major ∅	#3
Series	Coarse
Thread note	
Major diameter	

SP15-4) Write the thread notes for the following external threads. Also, what are the minor diameter and the number of threads per mm? The answer to this problem is given on the *Independent Learning Content*.

Major ∅	M33
Series	Fine
Thread Note	
Minor diameter	
# of threads per mm	

SP15-5) Write the thread notes for the following internal threads. Also, what are the tap drill size and/or diameter and the number of threads per mm? The answer to this problem is given on the *Independent Learning Content*.

Major ∅	M24
Series	Coarse
Thread Note	
Tap drill size and/or diameter	
# of threads per mm	

SP15-6) Fill in the given table for a hex head bolt with the following major diameters. The answer to this problem is given on the *Independent Learning Content*.

Major ∅	3/4
Width across flats	
Max. width across corners	
Head height	
Normal clearance hole	

SP15-7) Fill in the given table for a hexagon (socket) head cap screw with the following major diameters. The answer to this problem is given on the *Independent Learning Content*.

Major ∅	7/8
Max. head diameter	
Max. head height	
Normal clearance hole	
Counterbore diameter	
Counterbore depth	

SP15-8) Fill in the given table for a slotted flat countersunk head cap screw with the following major diameters. The answer to this problem is given on the *Independent Learning Content*.

Major ∅	1
Max. head diameter	
Max. head height	
Normal clearance hole	
Countersink diameter	
Countersink angle	

SP15-9) Fill in the given table for a hex head bolt with the following major diameters. The answer to this problem is given on the *Independent Learning Content*.

Major ∅	M8
Max. width across flats	
Max. width across corners	
Max. head height	
Thread length for a screw that is shorter than 125 mm	
Normal clearance hole	

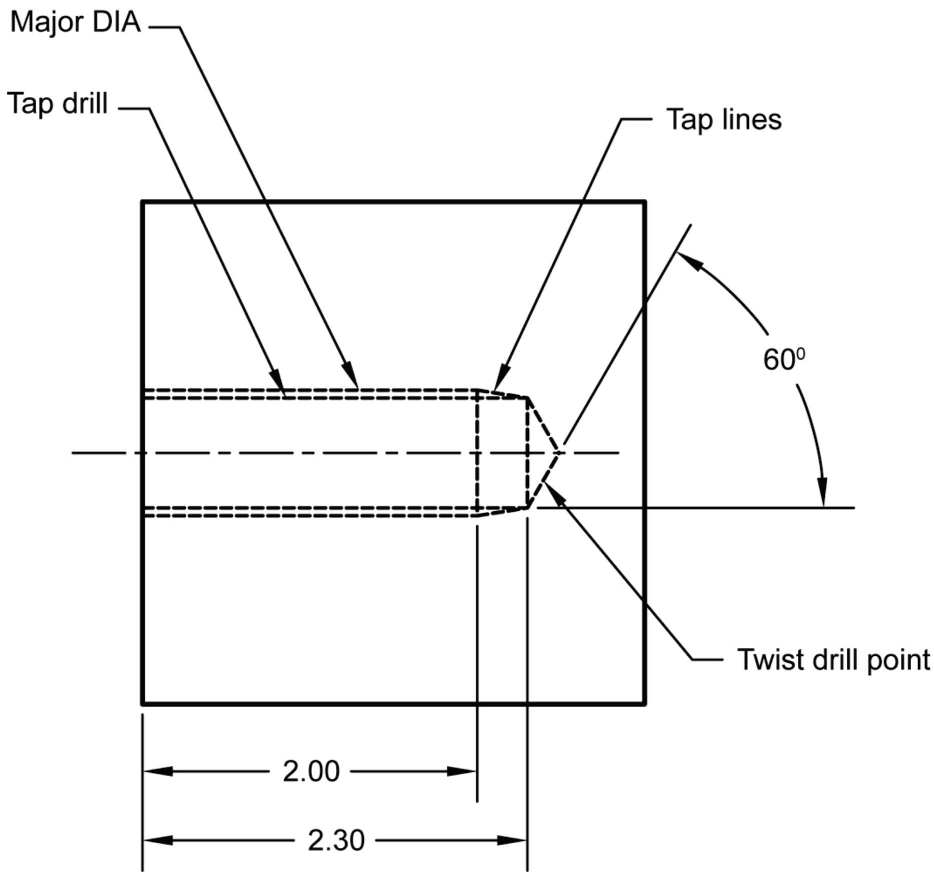
SP15-10) Fill in the given table for a socket head cap screw with the following major diameters. The answer to this problem is given on the *Independent Learning Content*.

Major ∅	M14
Max. head diameter	
Max. head height	
Normal clearance hole	
Counterbore diameter	
Counterbore depth	

SP15-11) Fill in the given table for a flat countersunk head cap screw with the following major diameters. The answer to this problem is given on the *Independent Learning Content*.

Major ∅	M20
Head diameter	
Head height	
Normal clearance hole	
Countersink diameter	
Countersink angle	

4) Draw the 3/4 - 10 UNC – ⩛2.00 internal threads in the right side view.
   a) In your ***Hidden*** layer, draw the rectangular view of the major diameter at a depth of 2 inches.
   b) Draw the rectangular view of the tap drill at a depth equal to the thread depth plus three times the pitch (2 + 3P = 2+ 3(1/10) = 2.3).
   c) Draw the 30° twist drill point at the end of the tap drill and the tap lines that connect the major diameter to the end of the tap drill.
   d) Add the appropriate centerline.



<u>Part#3: Top Plate</u> If you are just studying the basics and have not covered threads and fasteners, replace the 16X 10 – 24 UNC dimension with a 16x ∅.19 dimension.