

# Engineering Graphics

## Technical Sketching and AutoCAD 2009

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St. Louis, Missouri

### Sketching

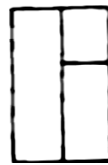
PICTORIAL DRAWING



ORTHOGRAPHIC PROJECTION



TOP VIEW

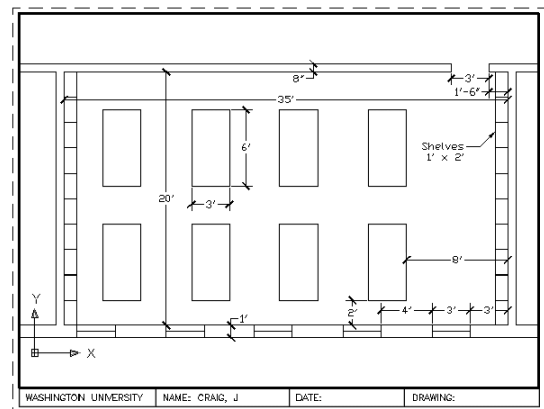


FRONT VIEW

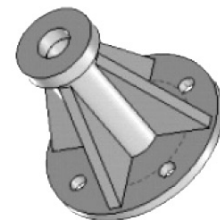
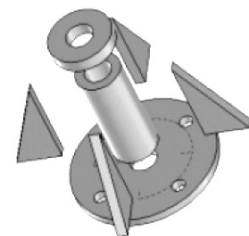


R SIDE VIEW

### 2D AutoCAD



### 3D AutoCAD Solids Modeling



**SDC**  
PUBLICATIONS

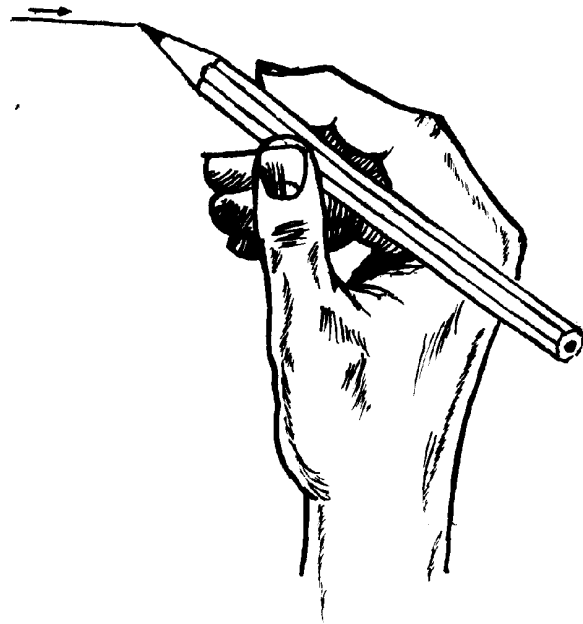
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## Introduction to Freehand Sketching

Sketching is a very important technique for technical communication. Sketches can transfer ideas, instructions and information in a clear, concise form. "Thinking with a pencil" is a practice designers use to bring ideas and mental pictures to reality. Freehand sketches are often the first view of new designs. Long before formal drawings are made or computer models are created, a series of detailed sketches are analyzed and approved.

***Sketching is a personal skill which everyone can improve.***



Sketching tools include:

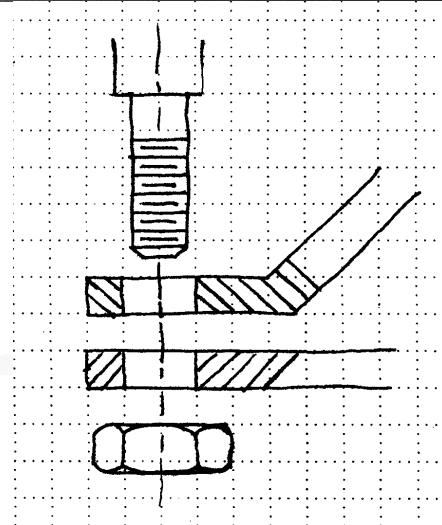
- \_\_ Pencil or leadholder.
- \_\_ Leads: "H", "F" or "HB" hardness.
- \_\_ Eraser: pink type preferred or white.
- \_\_ Sketching paper: rectangular grid, isometric grid, vellum overlay, newsprint, etc.
- \_\_ Straightedge.
- \_\_ Circle and ellipse templates.
- \_\_ Plastic triangles.
- \_\_ Gadgets like tiny drafting machines, roller parallel tools, etc.

Designers may use a variety of these tools when many sketches are to be made. Since sketches may be needed quickly, it is best to rely only on pencil, paper and eraser.



Printed grids or papers with lightly ruled lines are often used for sketching. The grids are used to keep freehand lines straight. Grids may also be used to create accurately scaled sketches, keeping the drawing in correct proportion.

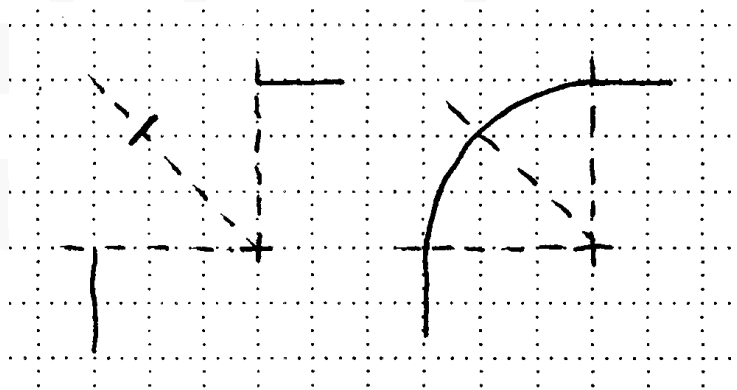
***Grids act as guides for sketching. They help keep lines straight and in correct proportion.***



### Using Sketching Grids

Grids may be used to keep features in correct X - Y scale. The rounded corner is measured three units horizontal, three units vertical and a construction point is measured half-way around the arc.

*Sketches must be very neat and proportional to the real objects. Lines must be very black to scan or fax clearly.*



Engineering and architectural forms are available with many guide line spacings. 1/4 inch forms are common. Sketches of objects may be scaled down (or up) to fit the sheet size. The examples show several possible scale factors.

*Accurately scaled sketches are needed to convey proportions.*

$$\frac{1}{4}'' = 1'-0'' \quad \left| \begin{array}{c} 7'-6'' \\ \hline \end{array} \right.$$

$$\frac{1}{4}'' = 100' \quad \left| \begin{array}{c} 525' \\ \hline \end{array} \right.$$

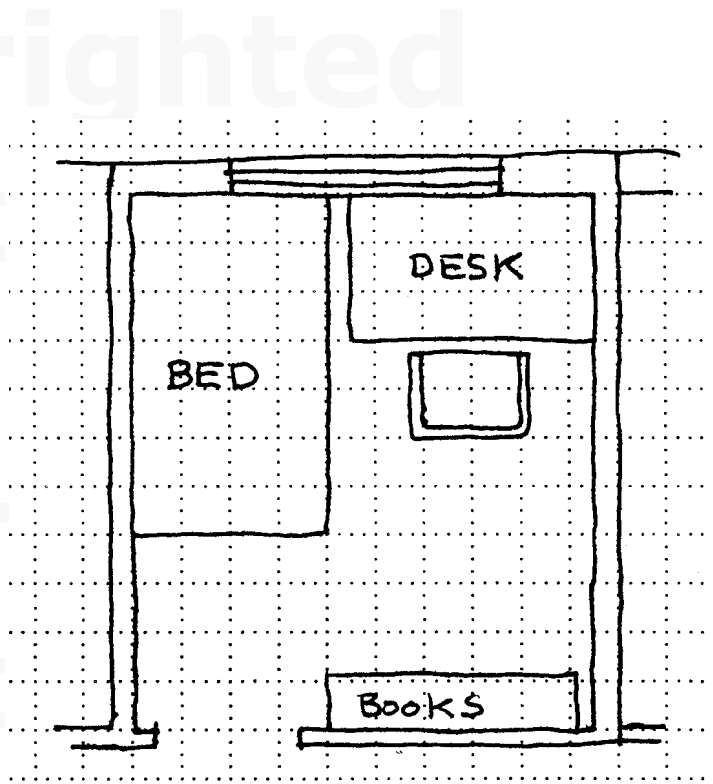
$$1 \text{ GRID} = 20 \text{ UNITS} \quad \left| \begin{array}{c} 90 \\ \hline \text{UNITS} \end{array} \right.$$

This sketch of a small room is drawn to a scale of 1/4" = 1'-0". It indicates the room is about 9'-6" wide by 11'-0" deep. The bookcase is 14" deep and 5' wide. Other furnishings are sketched to scale.

*Scaled sketches are often the first step in working out the sizes and proportions for a design.*

Sketches may be created anywhere and at any time with a minimum of tools. Neat sketches convey accurate information. Professionals take pride in the appearance and accuracy of their sketches.

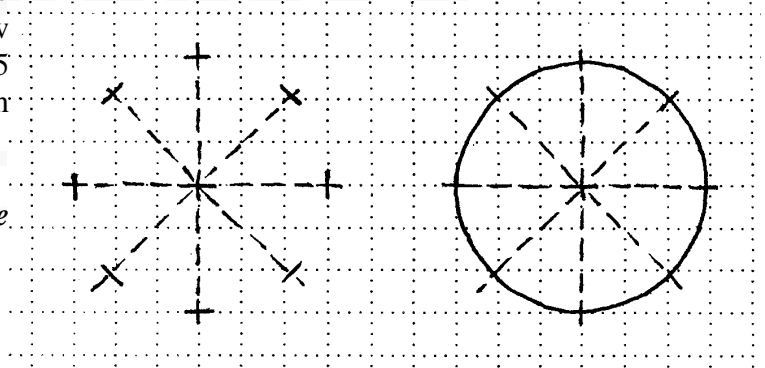
**The quality of the work projects the integrity of the designer.**



## Sketching Arcs and Circles

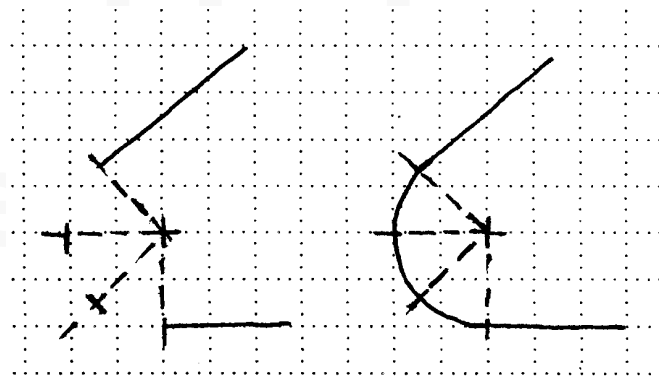
Construction lines and points will improve the appearance of freehand circles. It takes only a few seconds to block in horizontal, vertical and 45 degree points shown. These points help when sketching the arcs for the circle.

*The edge of a piece of paper may be used as a gage for measuring points on a circle.*



Tangent points define the extent of an arc. Tangent points are located by sketching a line from the center of the arc or circle perpendicular to the tangent line. These points show where the arc ends and the straight line begins.

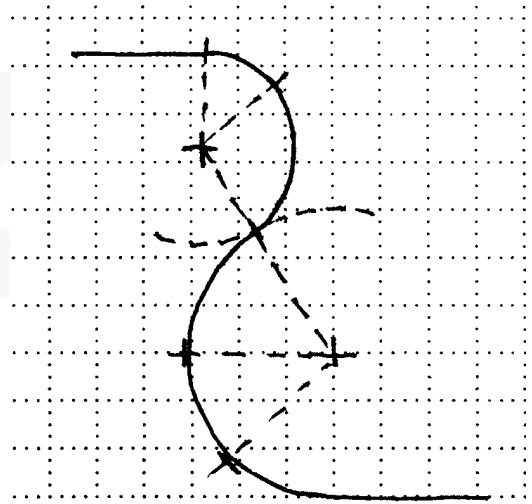
*Locating tangent points will improve the appearance of sketches.*



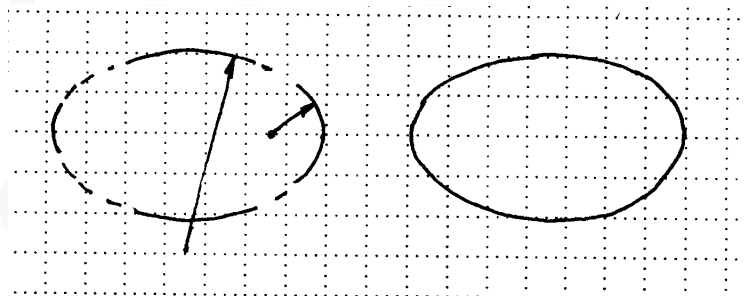
When two circles are tangent, the tangent points are on the line connecting the centers. This point shows where one arc ends and the other begins.

*There are only two rules to remember when locating tangent points:*

- 1. The point of tangency between a line and a circle is on the line perpendicular through the center of the circle.*
- 2. The point of tangency between two circles is on the line connecting the centers of the circles.*



Ellipses may be sketched using four arcs. Sketch a small arc at each side and a longer arc across the top and bottom.

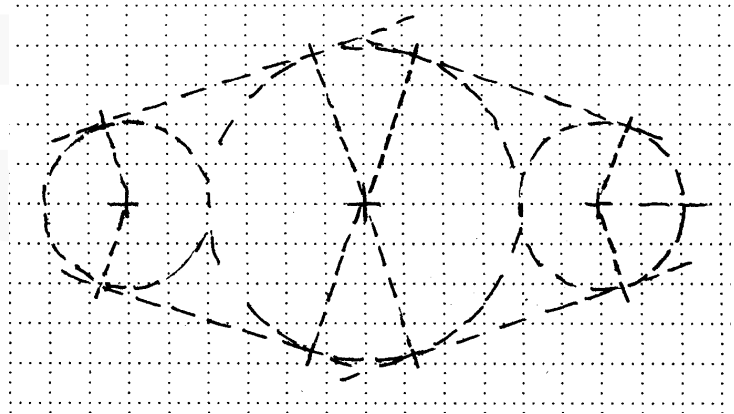


### Sketching Techniques

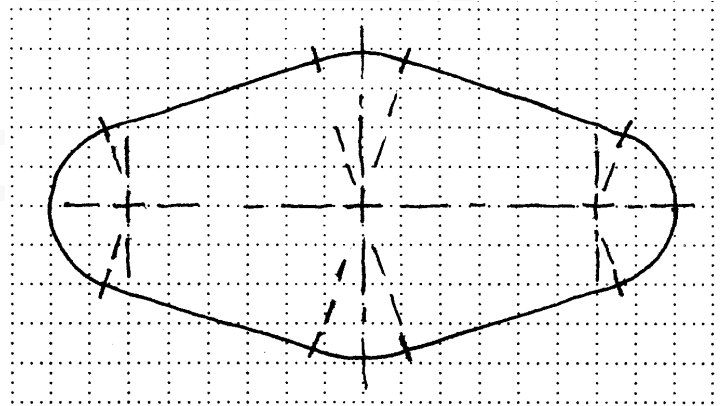
Block in the entire circle even if only a part of the circle will be needed. This helps space the object and verify proportions.

- \_\_ Sketch the three circles.
- \_\_ Sketch the lines tangent to the circles.
- \_\_ Sketch each radius perpendicular to each tangent line.

These constructions will locate points needed for darkening-in the final shape.

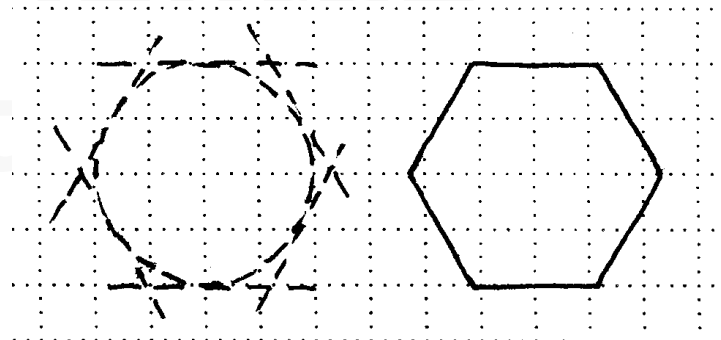


*The final shape is sketched by pressing down hard on the pencil to create a smooth black outline.*



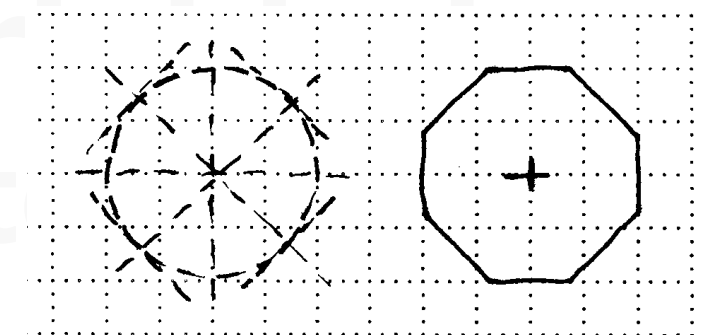
Geometric shapes like this hexagon are based on first sketching a circle. Draw the hexagon using the circle as a control surface. Construct tangent lines at approximately 60 degrees (in this case).

Blacken the lines for the final shape.



An octagon has eight sides.

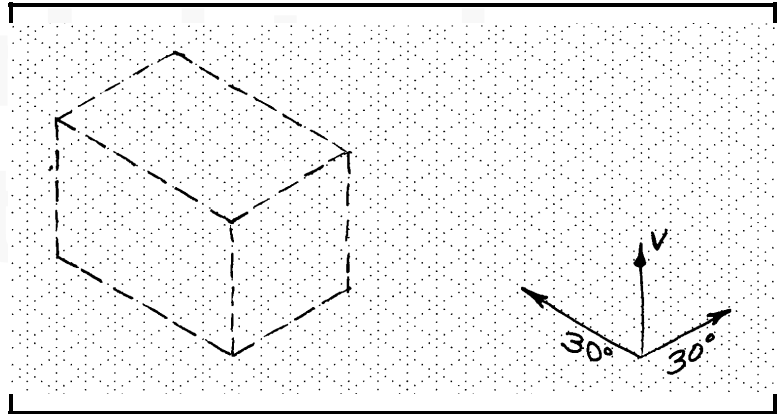
- \_\_ Sketch a circle.
- \_\_ Sketch horizontal and vertical lines tangent to the circle.
- \_\_ Sketch lines from the center of the circle at 45 degrees. (Use the diagonals of the grid boxes.)
- \_\_ Sketch lines perpendicular at each diagonal.
- \_\_ Blacken-in the octagonal shape.



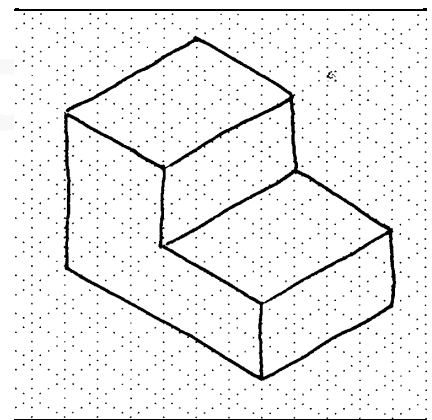
### Pictorial Sketches

Pictorial sketches are easier if an isometric sketching grid is used. By following the angles of the guide lines, the shape of the front, top and side faces may be sketched.

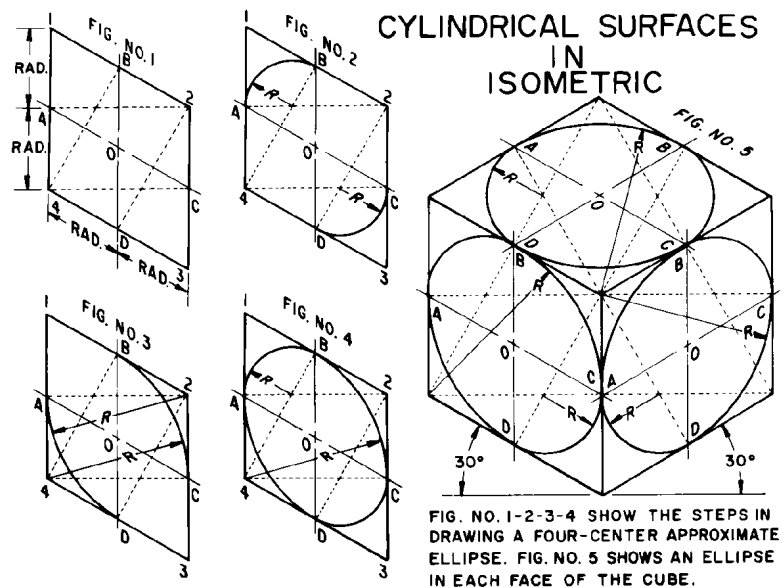
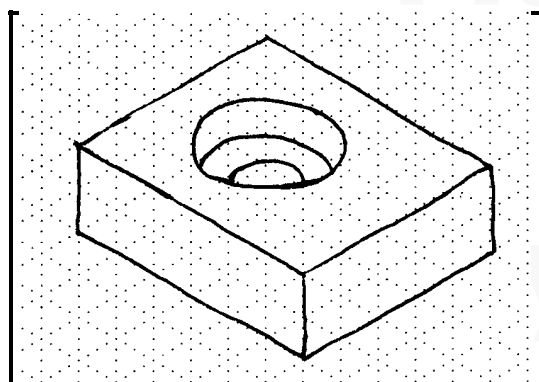
*Isometric axes are 30 degrees upward to the left (width), 30 degrees upward to the right (depth) and vertical (height).*



Pictorial sketches show three faces of a three dimensional object. This type of sketch helps designers and viewers portray three dimensional shapes quickly.

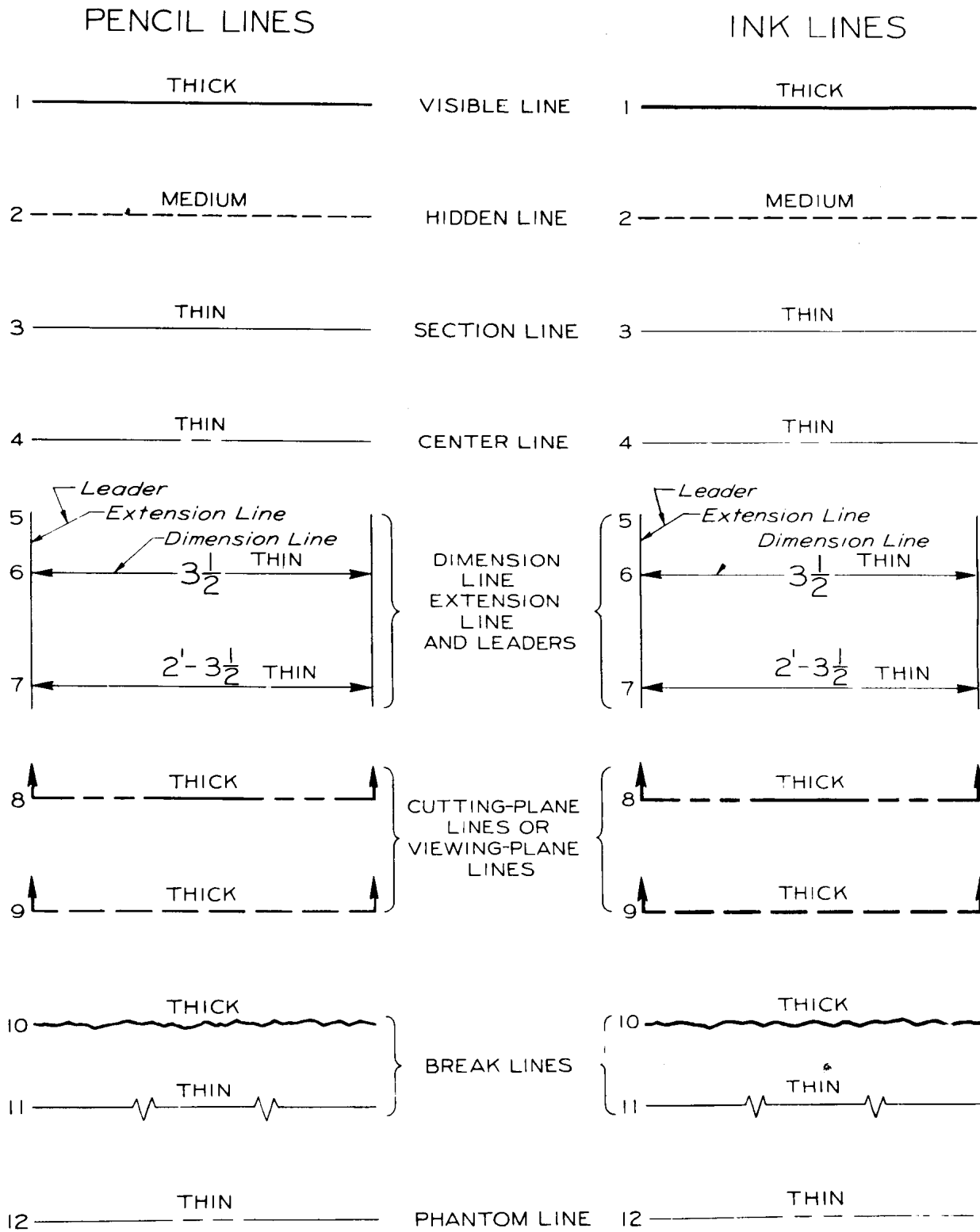


Circular shapes appear as ellipses on pictorial sketches. 4-center ellipses are often sketched or drawn. Centers are located by construction lines.



Pictorial sketch of a counter bored hole.

### ANSI Linetypes



WIDTH AND CHARACTER OF LINES  
FIGURE 1

Extracted from American National Standards Institute Y14.2

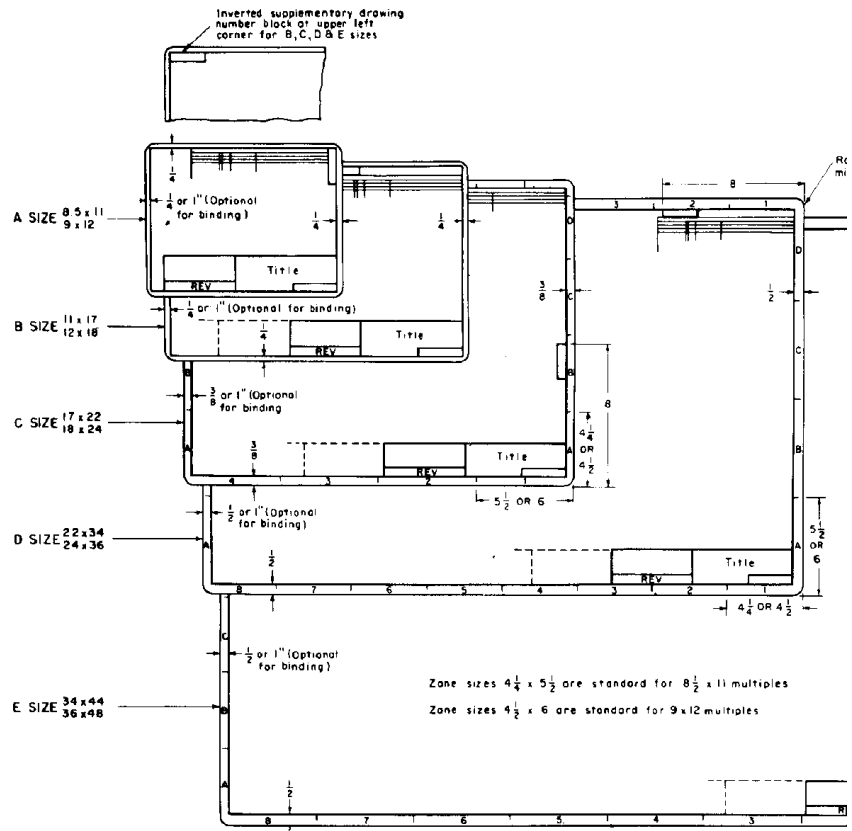
## Drawing Sizes - Copying Large Drawings

Drawings may require very large sheet sizes. Standard sizes are labeled:

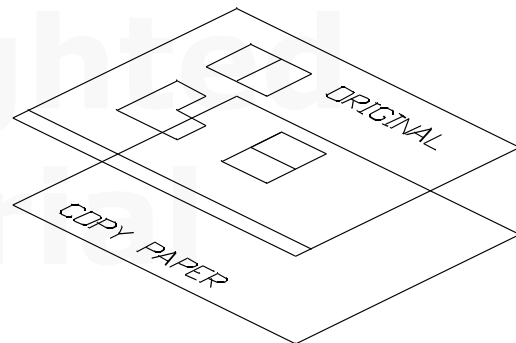
- \_\_\_ A = 8 1/2 x 11
- \_\_\_ B = 11 x 17
- \_\_\_ C = 17 x 22 etc.

(Figure extracted from A.N.S.I Y14.1).

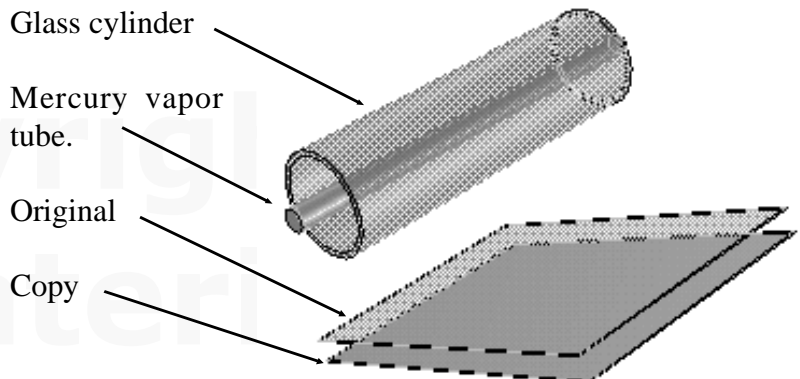
Drawings are created on transparent paper or plastic film. The reason the material is transparent relates to the copy process used to duplicate drawings. Many drawings are copied using the diazo process. This process allows large sheets to be duplicated at relatively low cost.



Prints are made by copying through the original. The copy paper is placed underneath. A bright light shines through from the top. Lines on the original must be very black to prevent light from passing through.



In commercial machines a mercury-vapor light is placed inside a glass cylinder. The original and copy paper roll around the cylinder during exposure. Very long drawings may be duplicated this way.



Hot ammonia gas is used to develop the dry copy.

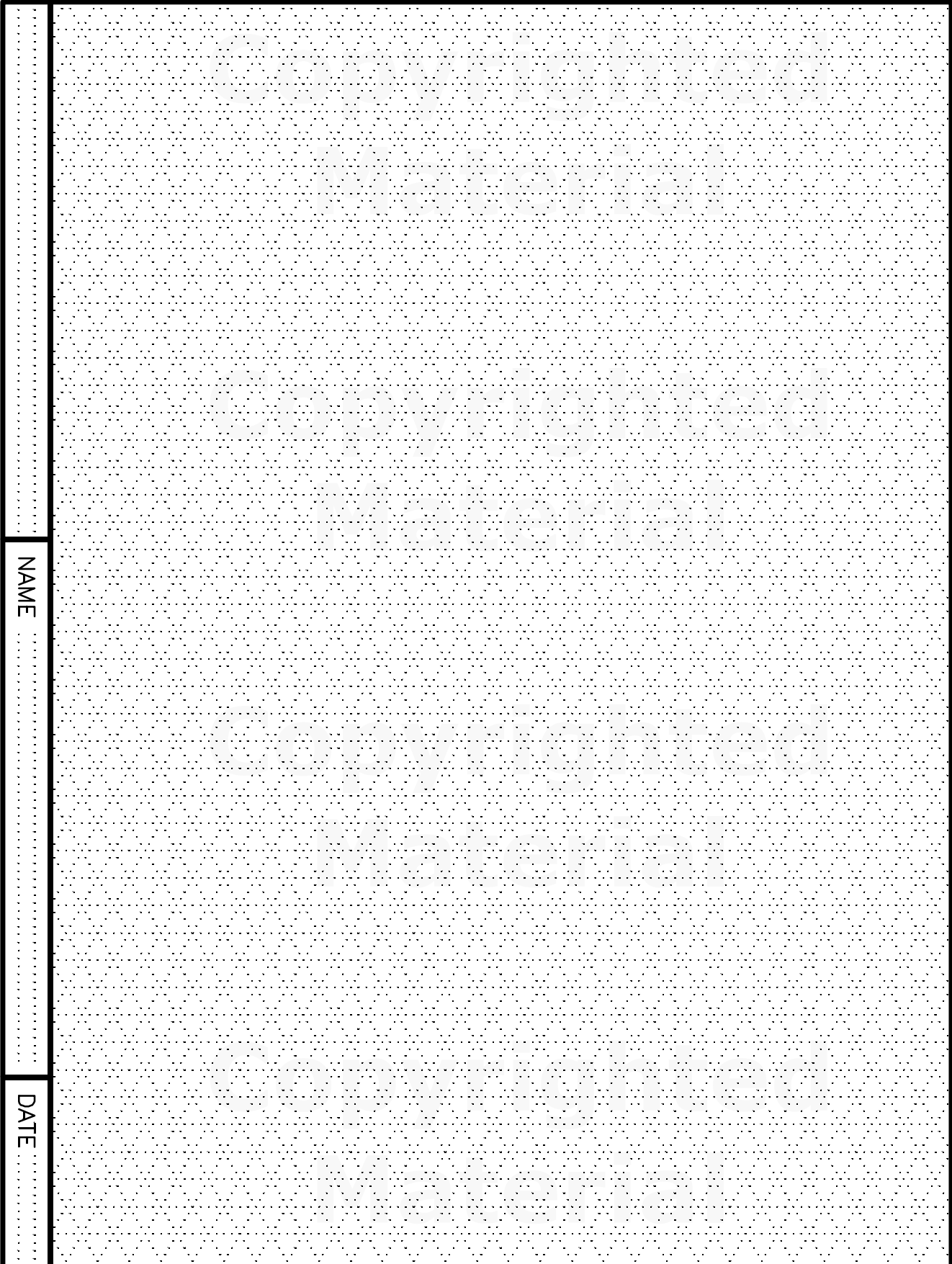


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NOTE: Use this grid format  
in horizontal direction.

FILE NUMBER .....

GRADE .....



NAME .....

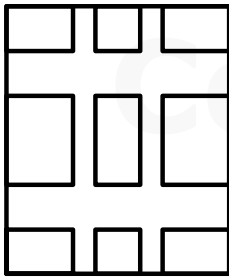
DATE .....

Copyright 2007  
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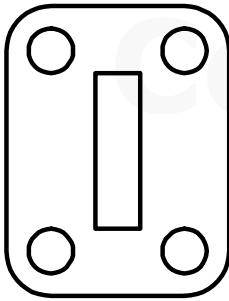
LINES MUST BE BLACK

FILE NUMBER .....

GRADE .....



SKETCH THE  
FIGURE SHOWN  
2X LARGER.  
CENTER THE  
SKETCH ON THE GRID.



SKETCH THE  
FIGURE SHOWN  
2X LARGER.  
CENTER THE  
SKETCH ON THE GRID.

C-101 FREEHAND SKETCHING

NAME .....

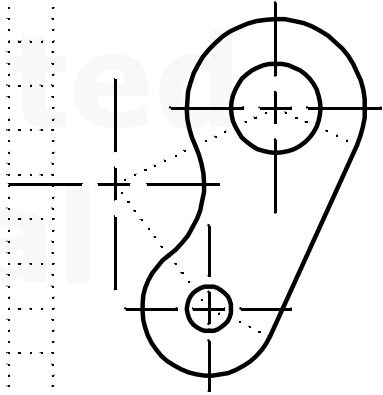
DATE .....

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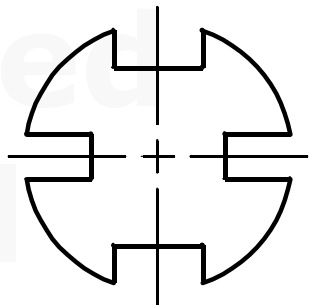
MAKE LINES BLACK

FILE NUMBER .....

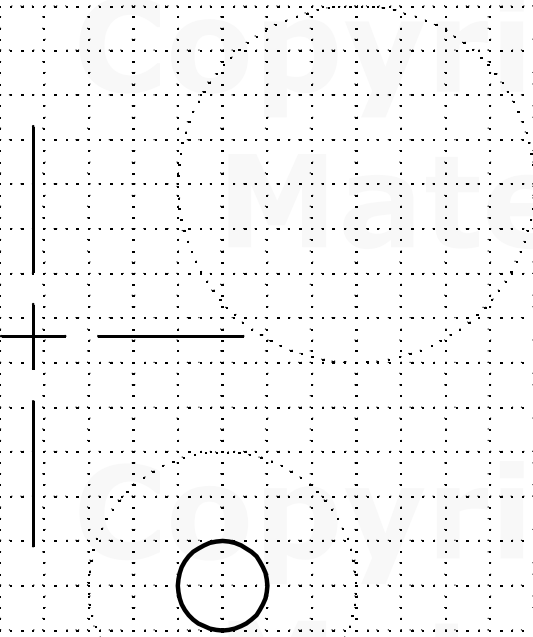
GRADE



SKETCH THE FIGURE  
SHOWN 2X LARGER  
ON THE GRID.  
CENTER THE SKETCH.  
SHOW CENTER LINES  
AND TANGENT LINES.



SKETCH THE FIGURE  
SHOWN 2X LARGER  
ON THE GRID.  
CENTER THE SKETCH.  
SHOW CENTER LINES.



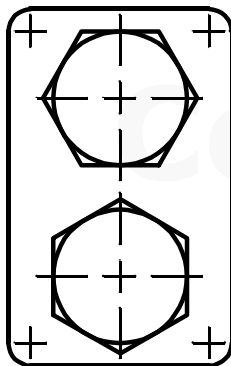
C-102 FREEHAND SKETCHING

NAME .....

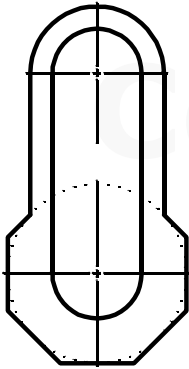
DATE .....

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FILE NUMBER ..... GRADE .....



SKETCH THE FIGURE  
SHOWN 2X LARGER  
ON THE GRID.  
CENTER THE SKETCH.  
SHOW CENTER LINES.



SKETCH THE FIGURE  
SHOWN 2X LARGER  
ON THE GRID.  
CENTER THE SKETCH.  
SHOW CENTER LINES.

Large grid area for sketching the figures shown in the left panels, scaled 2x larger.

C-103 SKETCH FREEHAND

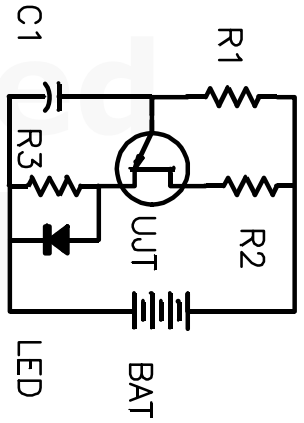
NAME .....

DATE .....

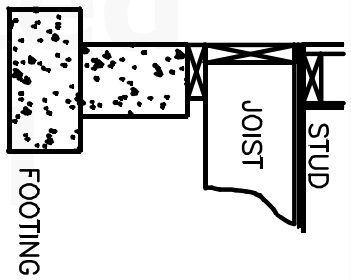
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FILE NUMBER .....

GRADE .....



SKETCH THE ELE-  
TRONIC SCHEMATIC  
ON THE GRID.  
ENLARGE 3X.  
LABEL AS SHOWN



SKETCH THE CON-  
STRUCTION DETAIL  
ON THE GRID.  
MAKE THE FOOTING  
8 UNITS WIDE.

C-104 SKETCH FREEHAND

NAME .....

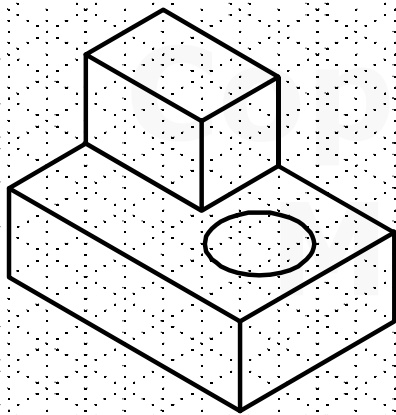
DATE .....

GRADE .....

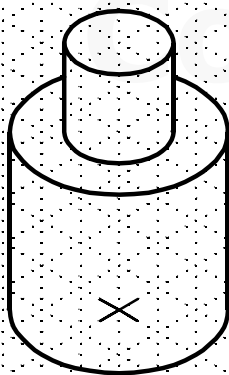
FILE NUMBER .....

NOTE: TURN PAGE VERTICALLY FOR USE.

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SKETCH OBJECT 2X SIZE.  
START AT CORNER SHOWN.



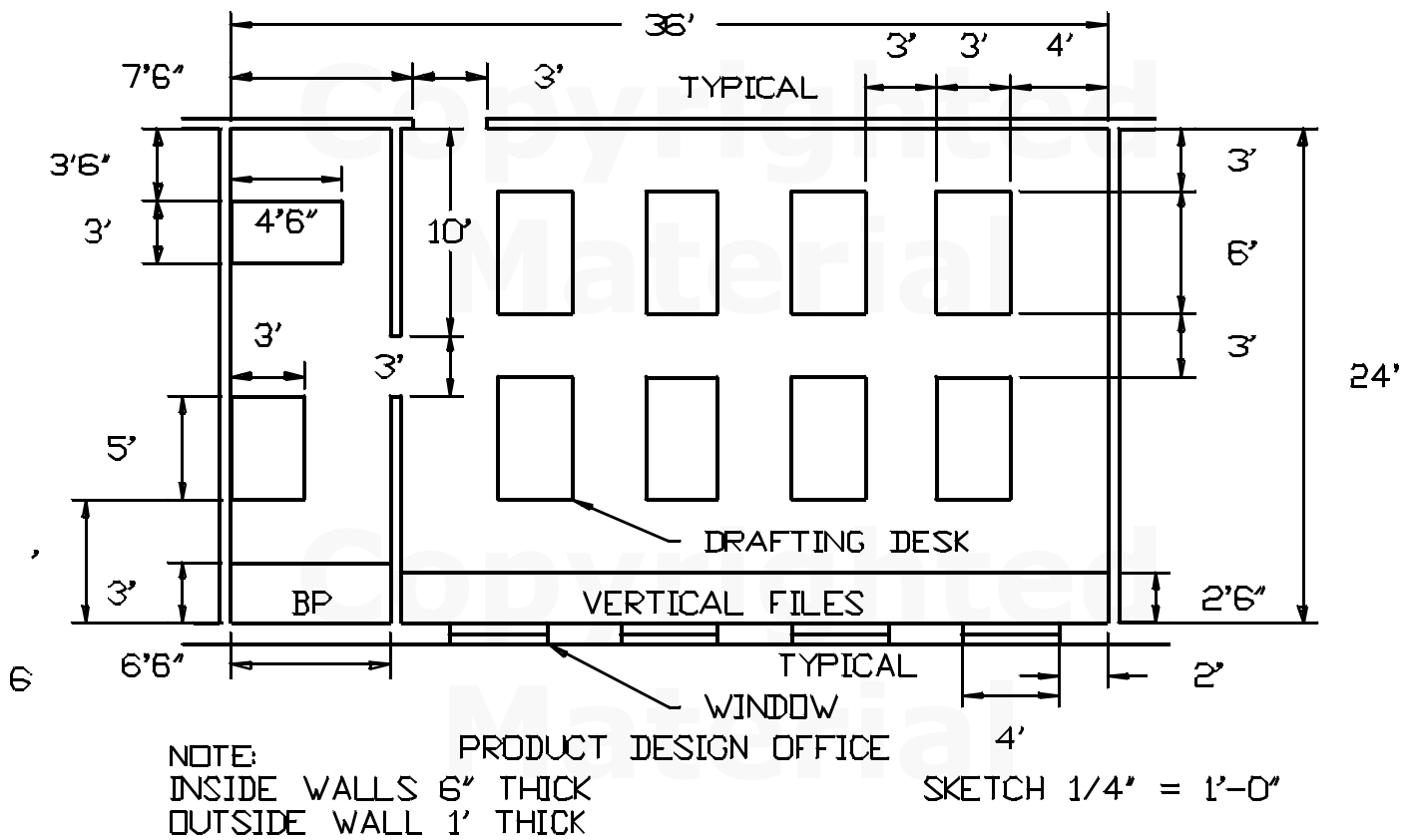
SKETCH OBJECT  
2X SIZE.  
START BOTTOM  
CENTER AT  
POINT SHOWN.



DATE .....

NAME .....

C-105 SKETCH FREEHAND



C-206. Sketch the design office layout on rectangular grid. Use 1/4" = 1'-0" scale.

(One grid = 1 foot). Use rectangular grids printed on the back of other pages in this chapter. Dimension as shown.

Some dimensions may have to be moved inside the walls to fit the page.

*Do not place dimensions outside the title block border lines.*

Note size and shape of arrows. Arrows should be 1/8" long and 1/3 as wide as long.

Omit text under NOTE:

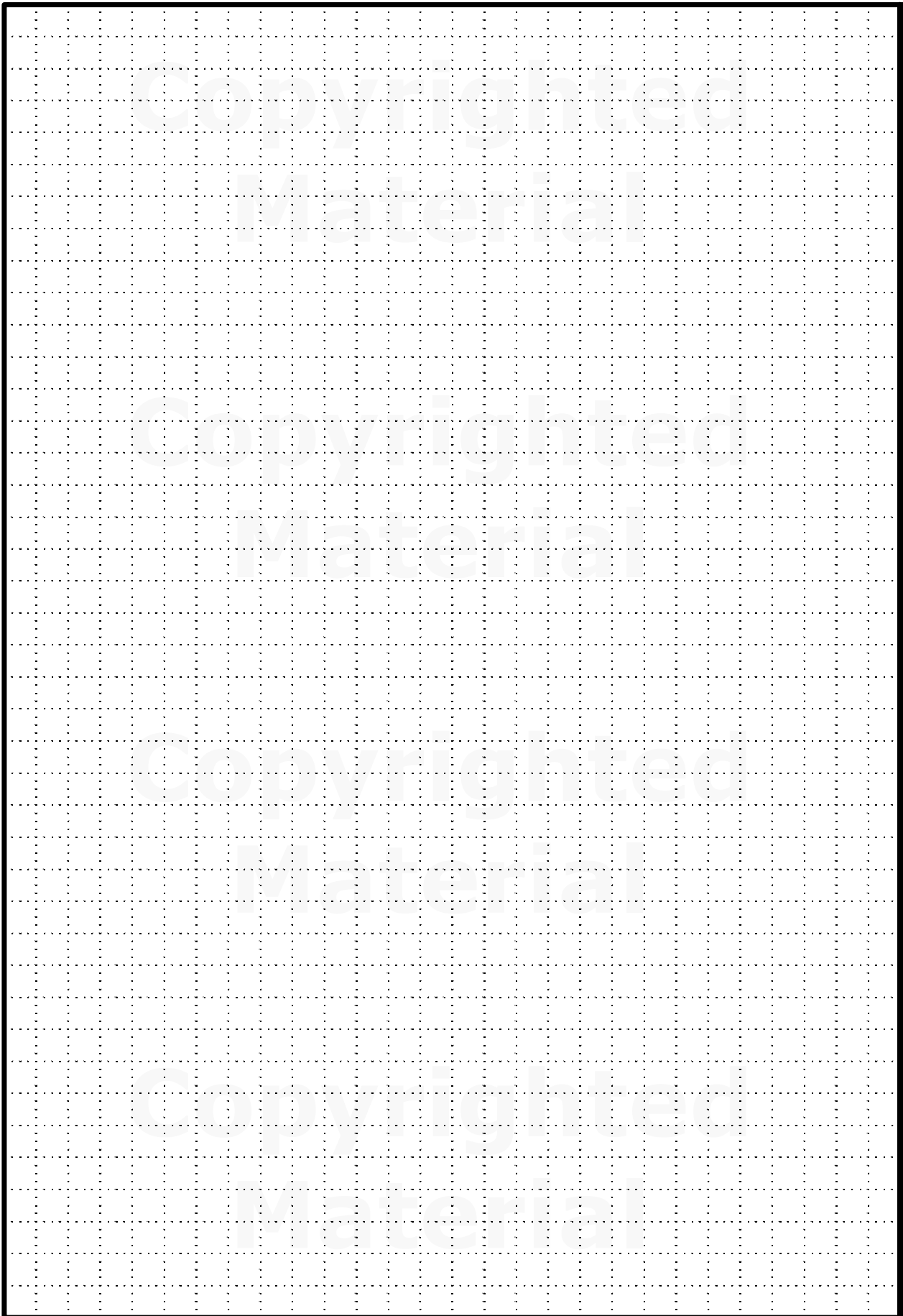
GRADE .....

FILE NUMBER .....

DATE .....

NAME .....

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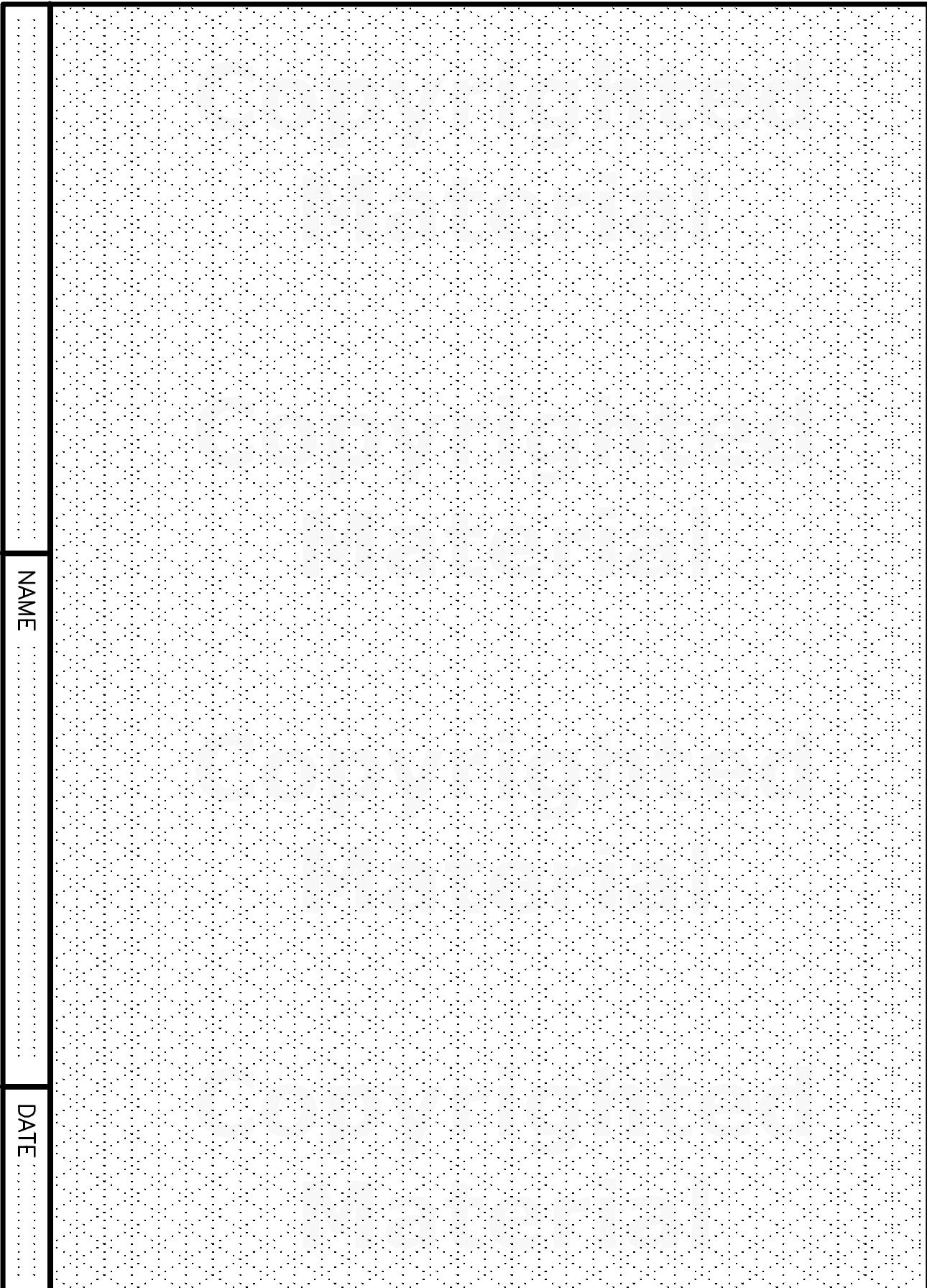


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NOTE: Turn the page  
vertically for use.

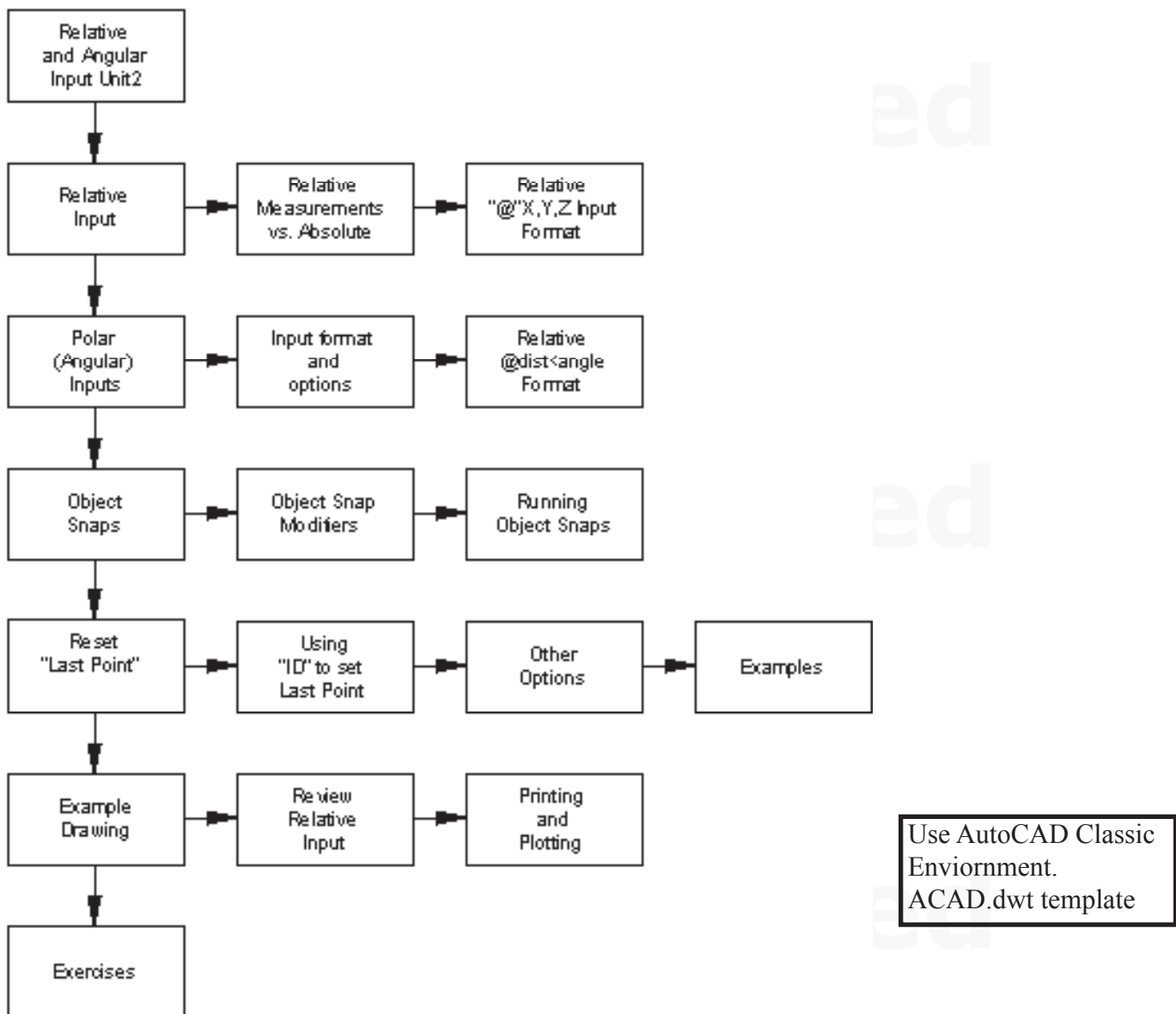
FILE NUMBER .....

GRADE



NAME .....

DATE .....



Most drawings are created using relative coordinates. This means that the next point is set from the last point drawn. The last point drawn is stored as “temporary 0,0”. AutoCAD uses the “@” (shift 2) as a prefix to the X,Y coordinates to denote relative input mode. Any measurements after the “@” are referenced from the last point drawn rather than from the world space 0,0.

Absolute input: X,Y

Relative input: @X,Y

Polar (angular) input: @dist<angle

DYNAMIC input is a new form of input recently introduced. We will work with dynamic input at a later time. For now, be sure dynamic input is turned off.

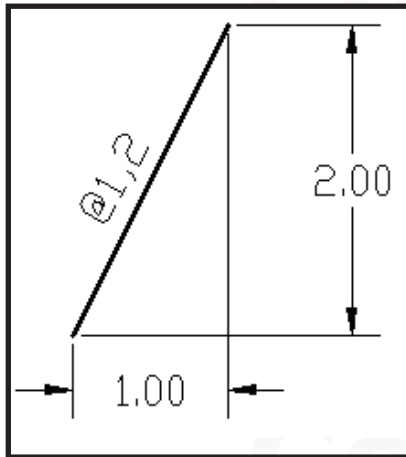
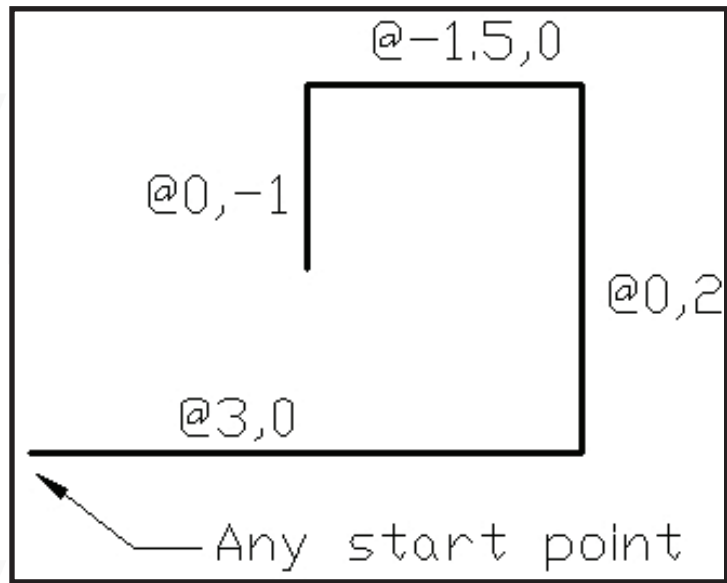


Turn Dynamic input OFF

Relative coordinates have two basic formats: One (@X,Y) when the endpoints of lines are known, and Two (@dist<angle) when the length of a line and the angle is known.

The example shows the sequence for drawing horizontal and vertical lines. Two inputs are needed.

- Any point to the right is positive.
- Any point to the left is negative.
- Any point up is positive.
- Any point down is negative.



```

Command: 1
LINE Specify first point:
Specify next point or [Undo]: @3,0

Specify next point or [Undo]: @0,2

Specify next point or [Close/Undo]: @-1.5,0

Specify next point or [Close/Undo]: @0,-1

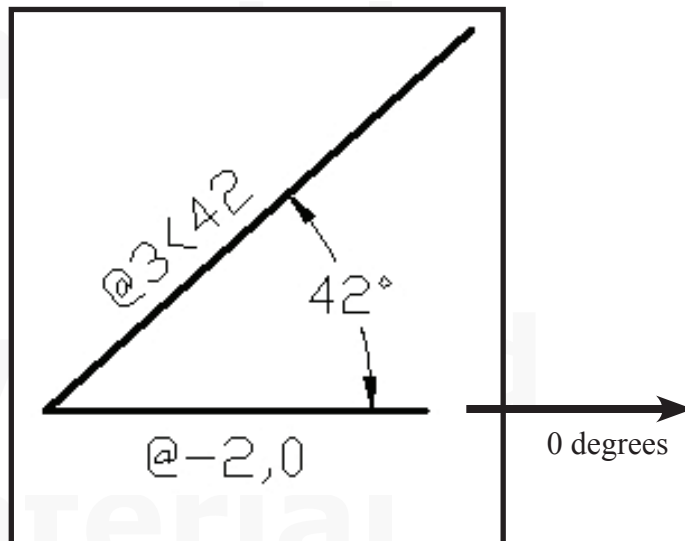
Specify next point or [Close/Undo]:
    
```

Above, an **offset** input is shown. Lines may be drawn at angles by giving the X distance over and the Y distance up from the start point. Positive or negative offsets may be specified.

**Angular** (polar) input specifies the length along the line (hypotenuse) and the angle from zero.

Zero defaults to East or the X-axis but may be set to a different direction.

Positive angles are measured counter-clockwise.

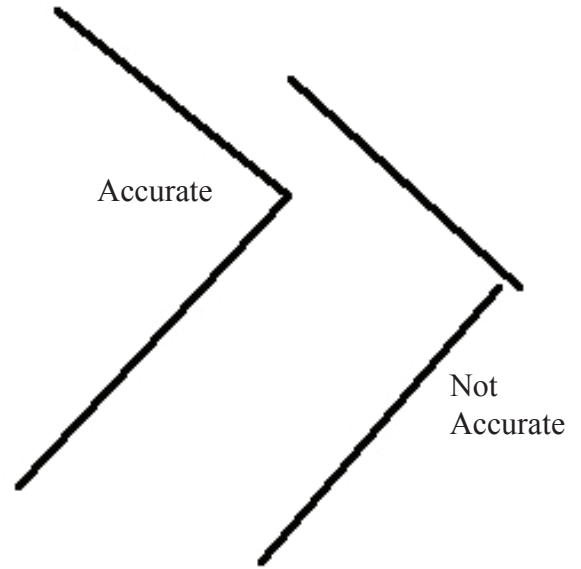


**Object Snaps.**

Drawings require very accurate connections. Information from drawings may be used in many different ways. Approximate (“eyeball”) connections may cause inaccurate results or failure of extended uses. Exporting drawing data to machine tools is a common process. Machine tools need a series of connected lines to calculate tool paths. If the lines are not connected, the machine will stop.

AutoCAD provides a number of modifiers (**Object Snaps**) which assure accurate connections. Object snaps may be invoked as the drawing is created or a series of **Running Object Snaps** may be set.

Basic format: Start a line (you can use an object snap to start the line accurately) the end the line (you can add an object snap to end the line accurately).



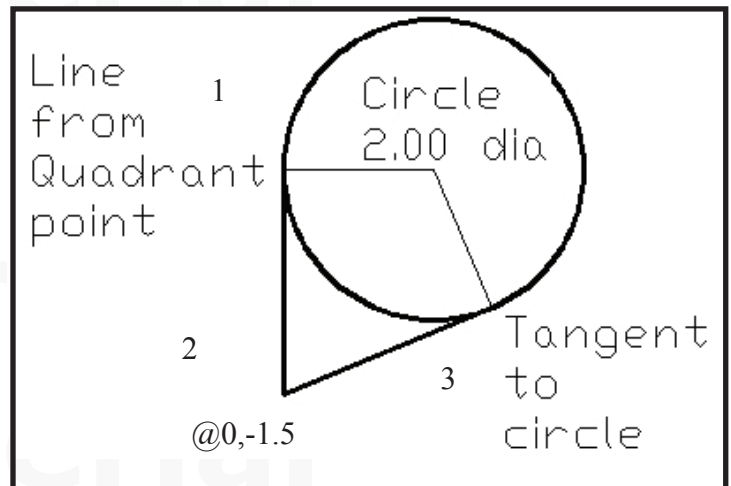
A circle with a 2.00 inch diameter was drawn.

1. The **Line** command was started then the object snap **Quadrant** was invoked.

Line ↵ Qua ↵ {point at left side of circle the endpoint was @0,-1.5

The next point was object snap **Tangent**.

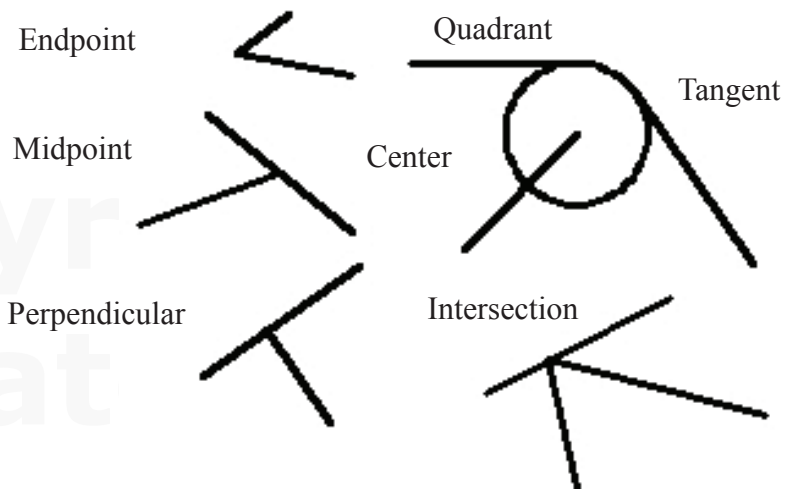
Tan ↵



**Common Object Snaps** are:

- Endpoint**
- Intersection**
- Tangent**
- Center**
- Quadrant**
- Perpendicular**
- Nearest**
- Midpoint**

Use the first three letters. Some drafters prefer to invoke each object snap by typing each command as the drawing is created.



Running Object Snaps.

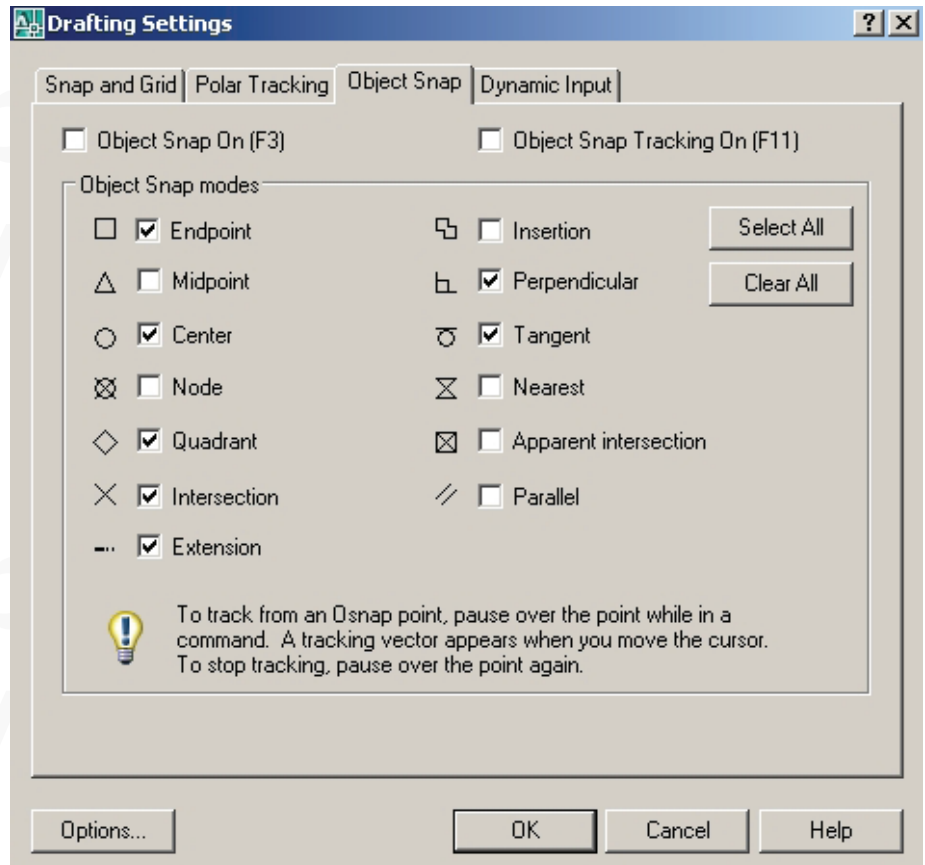
Pop down **Tools** .....  
**Drafting Settings**

Toggle **on** the Object Snaps you wish to use. (All on at one time can cause confusing results.)

Study the icons for each object snap. These appear as you work. Click when you see the icon to set the connection

**F-3** function key turns the running object snaps off/on.

Click **OSNAP** on the bottom toolbar to turn running object snaps on/off.

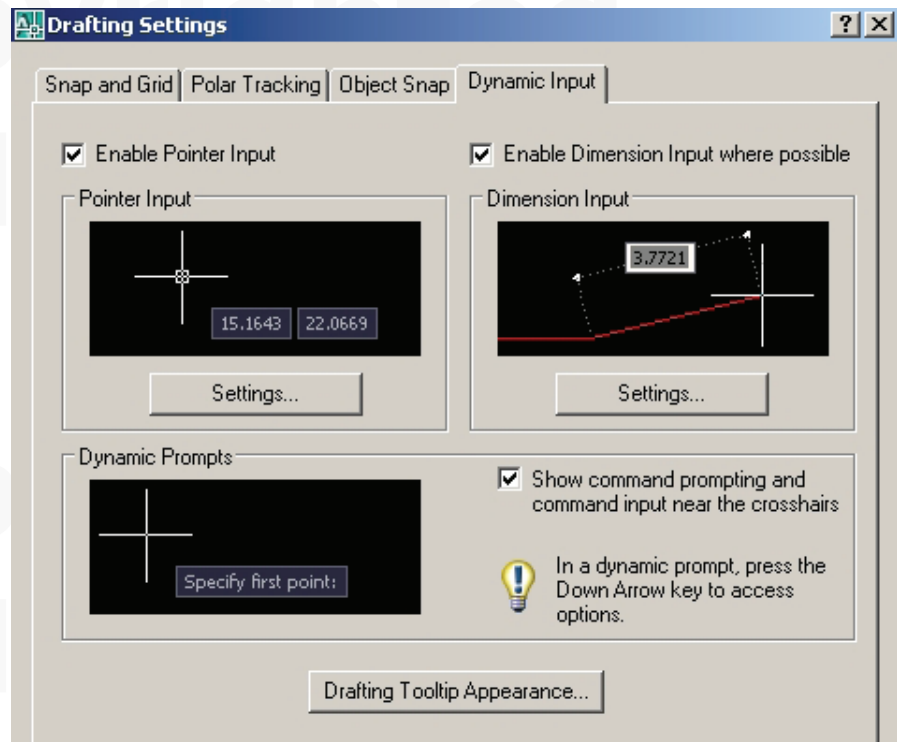


Dynamic Input may be set from the next tab over.

To use Dynamic Input, turn on the DYN button. (see above).

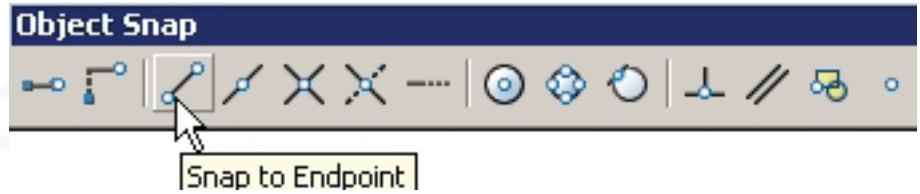
Experiment with these available tools. See what you prefer to use as you draw.

AutoCAD will configure menus, toolbars and input options to work as you like to work. After several months, designers begin to customize their workspace.



### Adding the Object Snap Toolbar to your drawing.

- \_\_ right-click on any toolbar.
- \_\_ select **Object Snap**

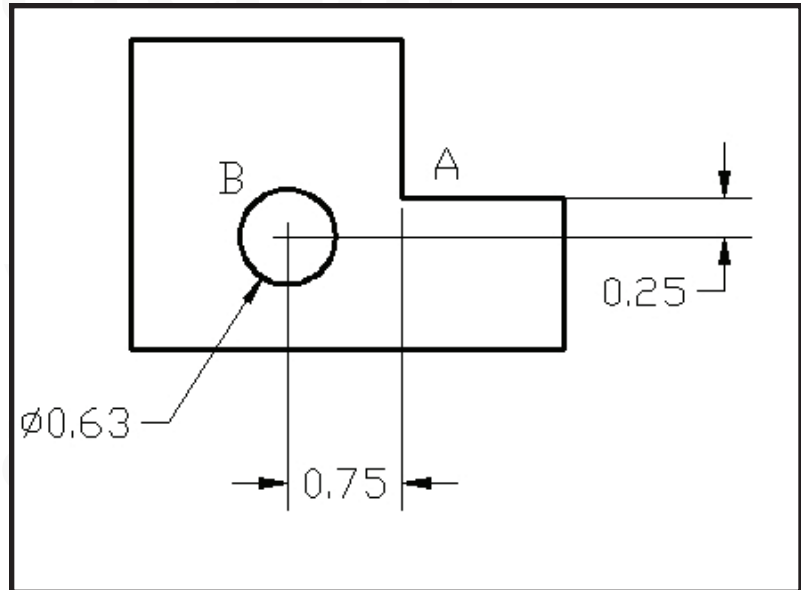


### Picking a reference point - (Resetting the temporary 0,0 point.)

Situations occur while creating drawings where an object must be drawn in reference to an existing point or intersection.

The circle B must be drawn in reference to the intersection at A.

- \_\_ 1st method. ID the intersection at "A". Type ID ↵ (identify) ... Int ↵ and point at the intersection at "A"
- This resets the temporary "last point" 0,0 to "A".



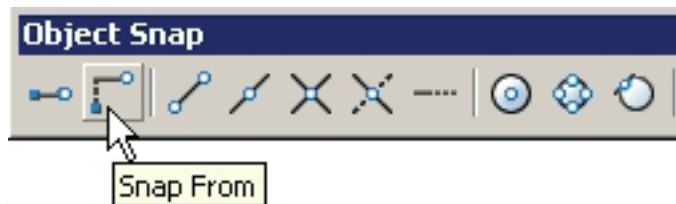
- Type C (circle) ↵ and key in @-.75,-.25 ↵ to set the center.
- Key in D ↵ and .625 ↵ to set the size.

- \_\_ 2nd method. Start the circle command.
- C ↵ (type) **from** ↵ (type) int ↵ and point at "A".
- key in @-.75,-.25 ↵
- Key in D ↵ and .625 to set the size.

- \_\_ 3rd method. Use the From option on the **Object Snap Toolbar**.

Start the circle command then click the **Snap From** icon.

- Select the point at "A" then key in the offsets: @-.75,-.25.
- Type in the circle diameter.



*Newer versions of AutoCAD added the "from" command to shorten the sequence of locating one object from another.*

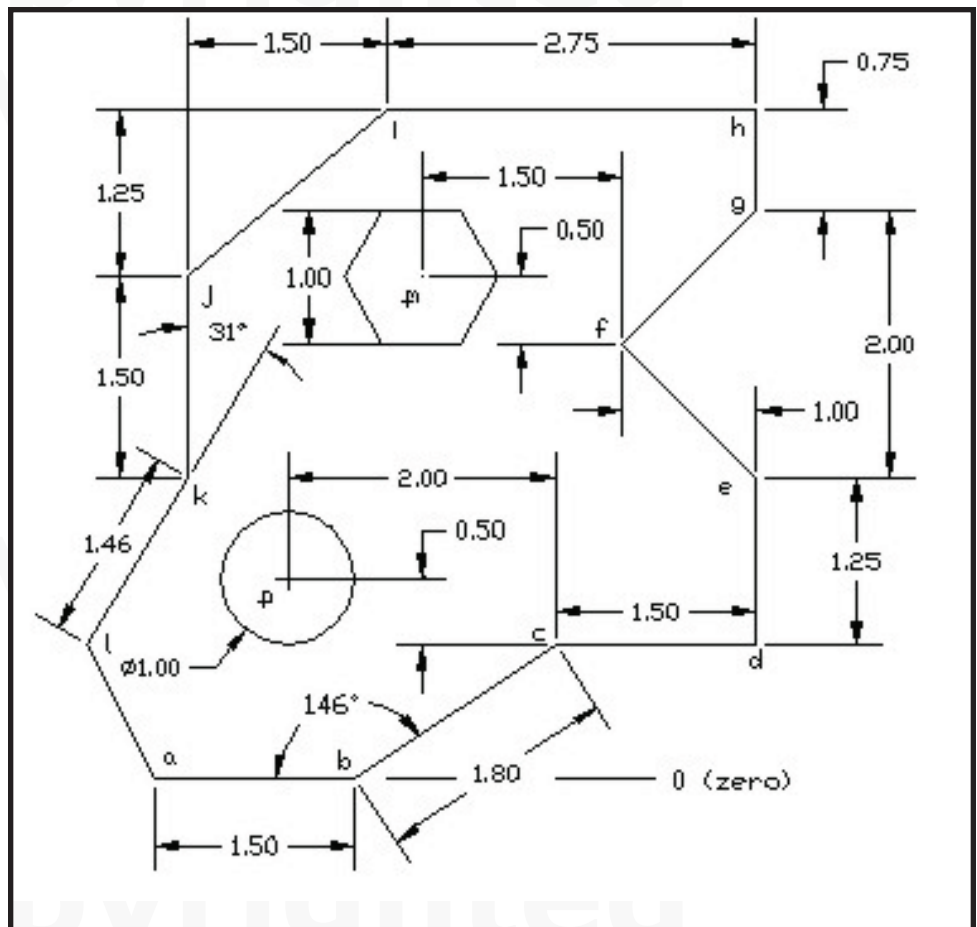
**Tutorial - Relative and Polar mode drawing.**

The drawing shown below combines a number of relative (linear) and polar (angular) drawing inputs. To start the drawing, start the **line** command and pick any point for “A”.

Command: line ↵  
 From point: pick any point  
 To point: @1.50,0 (a to b)  
 To point: @1.80<34 (angle from zero - b to c)  
 To point: @1.50,0 (c to d)  
 To point: @0,1.25 (d↵ to e)  
 To point: @-1,1 (e to f)  
 To point: @1,1 (f to g)  
 To point: @0,.75 (g to h)  
 To point: @-2.75,0 (h to i)  
 To point: @-1.5,-1.25 (i to j)  
 To point: @0,-1.5 (j to k)  
 To point: @1.46<239 (angle from zero — k to l)  
 c↵ to close. ( Or, osnap endp at “a”)

Note: Key in this example. Verify the steps, math and reasoning.

Below: (Some numbers will vary.)  
**Sequence to draw the circle and hexagon:**



```
Command: id      Draw the circle: first ID point "c"
Point: int
of X = 3.8307   Y = 1.9197   Z = 0.0000   (These numbers will vary)

Command: circle
3P/2P/TTR/<Center point>: @-2,.5   Key in the offset from "c" and draw the circle
Diameter/<Radius>: d
Diameter: 1

Command: id
Point: int
of X = 4.3307   Y = 4.1697   Z = 0.0000   Draw the polygon: ID point "f"

Command: polygon
Number of sides <4>: 6

Edge/<Center of polygon>: @-1.50,.5

Inscribed in circle/Circumscribed about circle (I/C) <I>: c

Radius of circle: .5

Command:
```



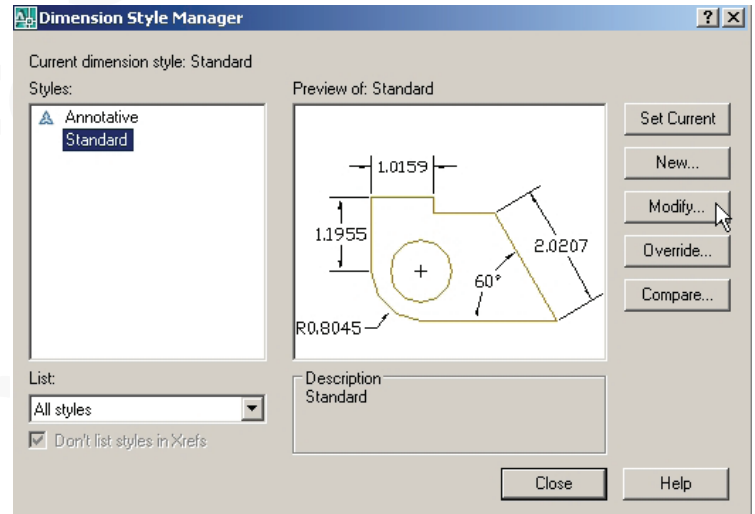
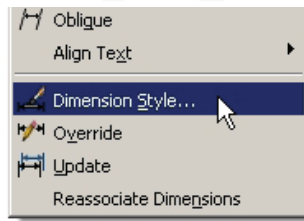
Save the drawing. Make a habit of saving drawings each 15 to 20 minutes. A computer crash can wipe out a lot of work.

Dimension the drawing approximately as shown. Some dimension settings need to be changed:

Pop down Dimension ....

Select Dimension Style

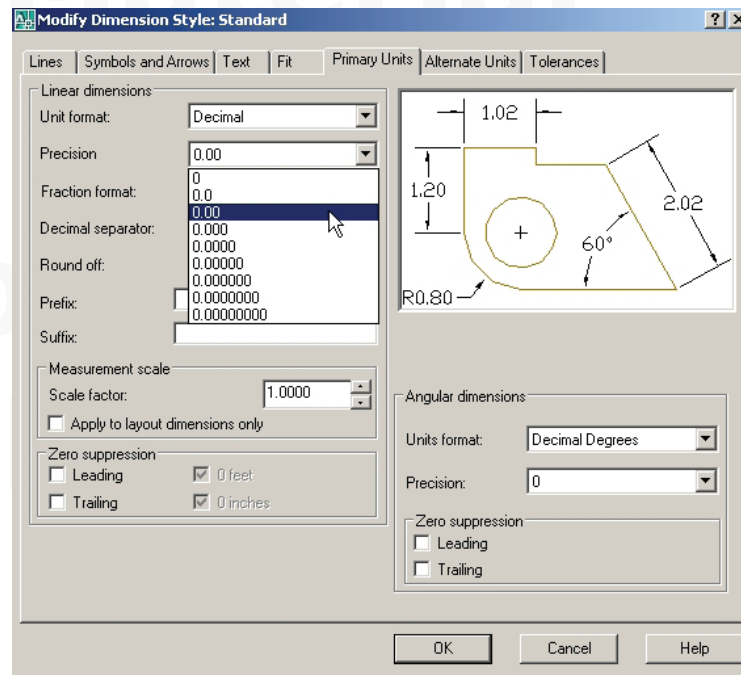
Select Modify on the Dimension Style Manager dialog box.



Select the Primary Units Tab.

Set the number of decimal places as shown to 2 places.

*Note: The number of decimal places on a dimension is often used on drawings to indicate the required accuracy of the dimensions. No dimension can be perfect so all dimensions on a drawing must have a Tolerance. A Tolerance Block is placed on drawings to show the required accuracy.*



**TOLERANCES**  
 .X = ± .1 IN  
 .XX = ± .03 IN  
 .XXX = ± .010 IN.

.X = +/- .1 inches is a very large tolerance - rarely used.

.XX = +/- .03 inches. Common shop tolerance for non-critical features. .06 inches is the thickness of a penny. Preferred.

.XXX = +/- .010 inches. About the thickness of 5 pieces of paper. Closer tolerance. More expensive to produce.



The dimension pop down menu is shown.

\_\_ Use **Object Snaps Endpoint, Intersection, Center, etc.** when placing dimensions to be sure you are connecting to the correct points.

\_\_ Use **Linear** dimensions to place horizontal and vertical dimensions.

\_\_ Use **Aligned** dimensions to place dimensions on an angle.

\_\_ Always dimension the **Diameter** of full circles and dimension the **Radius** of arcs.

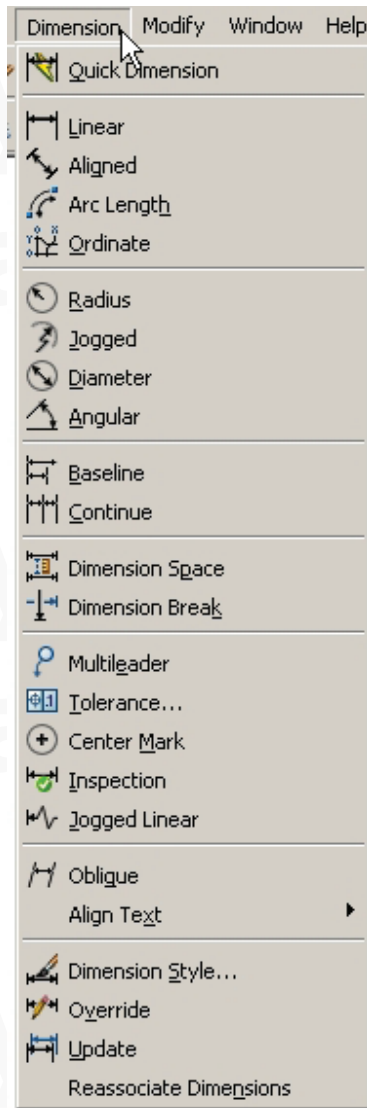
*The easiest way to edit a dimension is to erase and re-dimension.*

\_\_ You do not need to place the letters (a,b,c,d,e,f etc.) on the drawing.

\_\_ Be sure to put your name on the drawing.

\_\_ **Save** the drawing and **plot** the drawing as we did in the previous chapter.

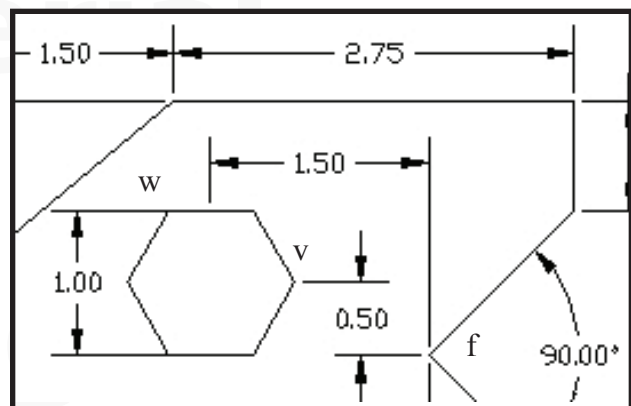
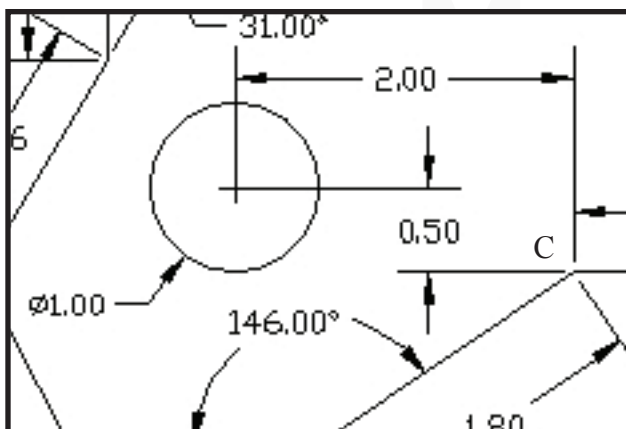
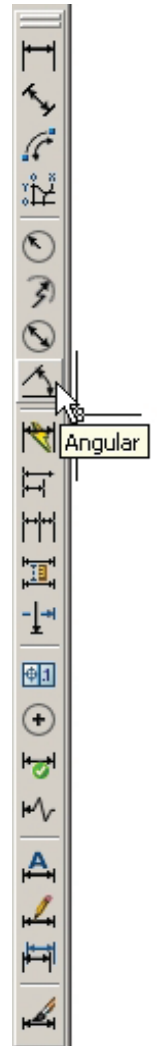
**End of Tutorial**



Right click any toolbar and select the **DIMENSION** toolbar.

Drag anyplace on the window.

This is a quick way to select and place dimensions.



Use object snap to **center** to dimension from “c” to the center of the circle. Vertical and horizontal dimensions are needed.

There is no center for the polygon. For now, dimension from “f” to the **Intersection** at “v” and “f” to the **Midpoint** at “w”.

**Using the List command to verify objects.**

\_\_ Type List and select an object. A dialog box will appear that tabulates all the information about the object. This is a tool which allows you to verify starting points, lengths, angles, etc. It is very helpful when troubleshooting a drawing.

The hexagon from the tutorial was selected. This is the data that the list command provided.

```

Select objects:

                LWPOLYLINE Layer: "0"
                        Space: Model space
                Handle = 11b
                Closed
                Constant width    0.00
                area    0.87
                perimeter  3.46

                at point X=    4.78 Y=    6.61 Z=    0.00
                at point X=    4.49 Y=    6.11 Z=    0.00
                at point X=    4.78 Y=    5.61 Z=    0.00
                at point X=    5.35 Y=    5.61 Z=    0.00
                at point X=    5.64 Y=    6.11 Z=    0.00
                at point X=    5.35 Y=    6.61 Z=    0.00
    
```

**Exercise C-101.**

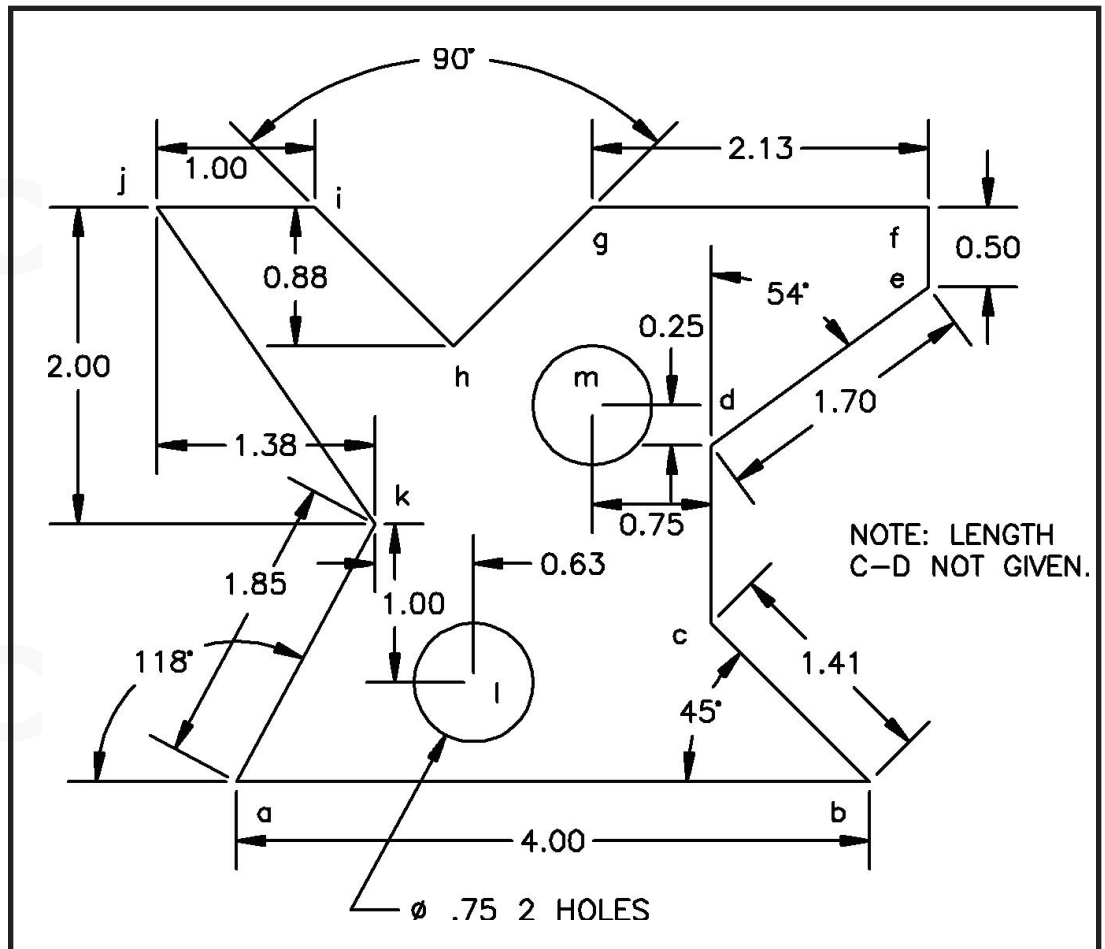
Start a new drawing and draw the object shown.

You will use many different input formats for the layout.

Pick any point to start at "a". Draw to b and to c. Then go back to a and draw to k, j, i etc.

Length c-d is not given. When you finish, c-d should be a vertical line. Save often.

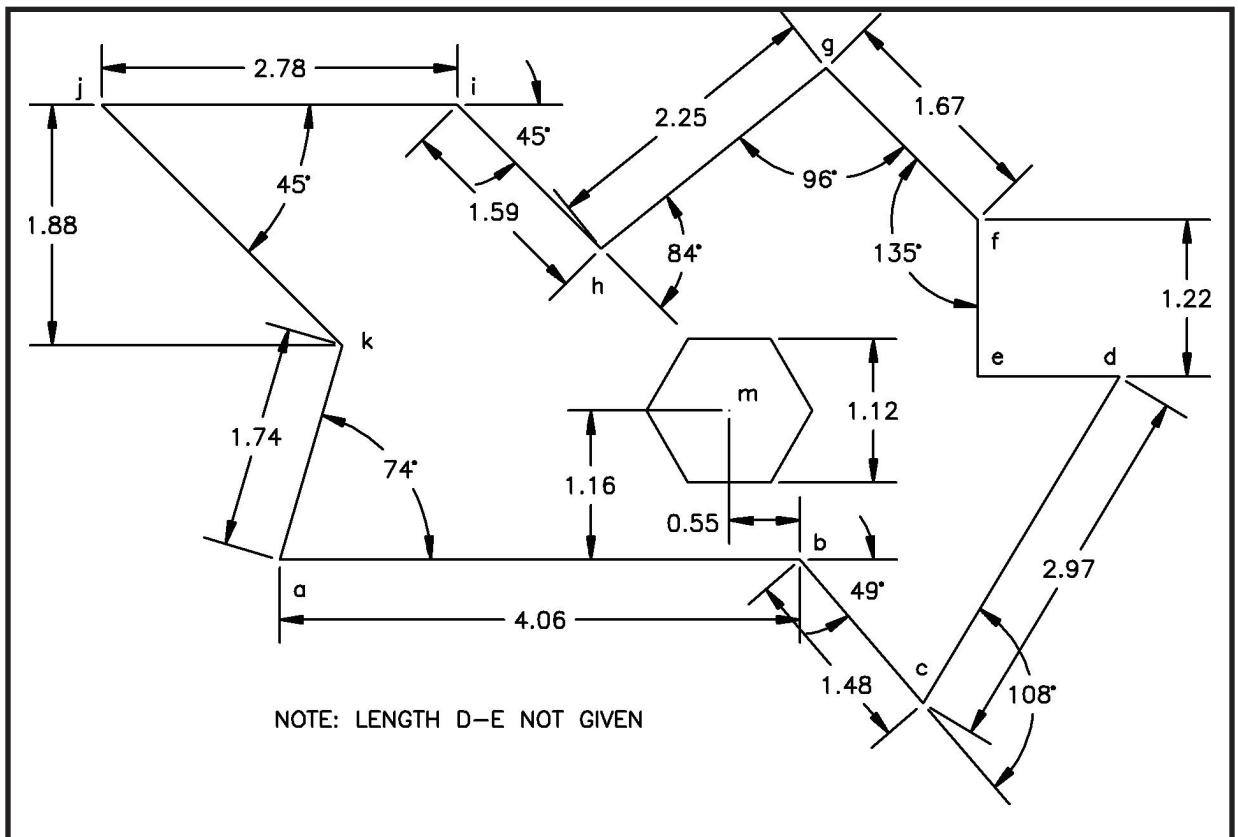
- \_\_ Put your name on the drawing.
- \_\_ Dimension the drawing.
- \_\_ Plot the drawing



**Exercise C-102.**

Layout the drawing. Be sure to put your name on and dimension the layout as shown.

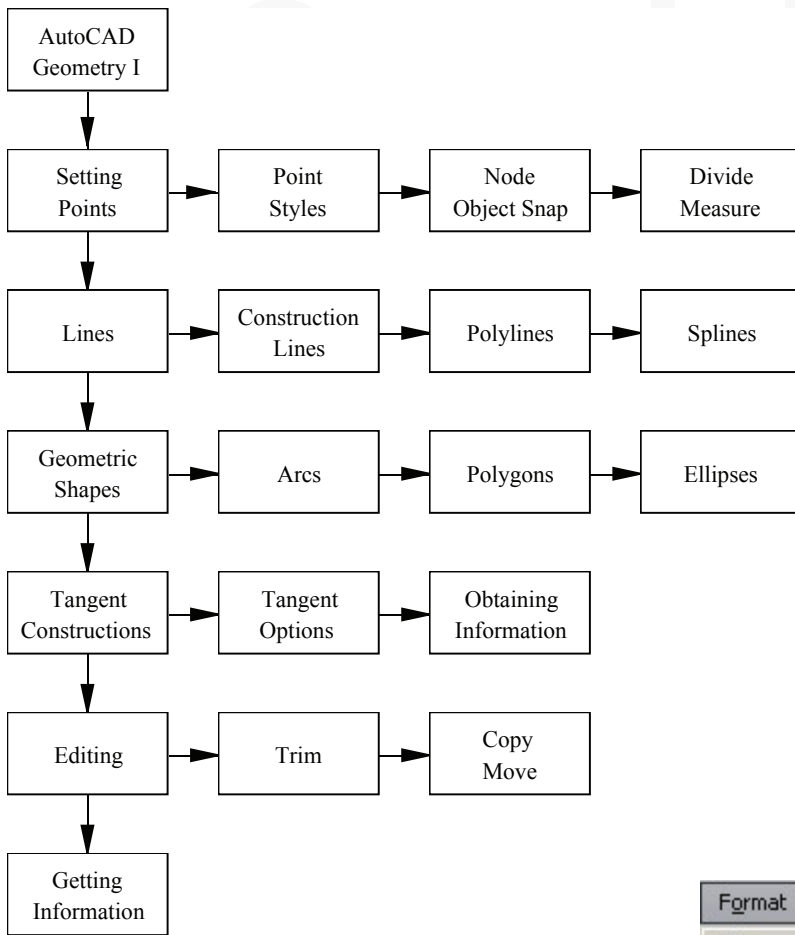
\_\_ Save  
\_\_ plot.



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Material

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Material

### AutoCAD Geometry

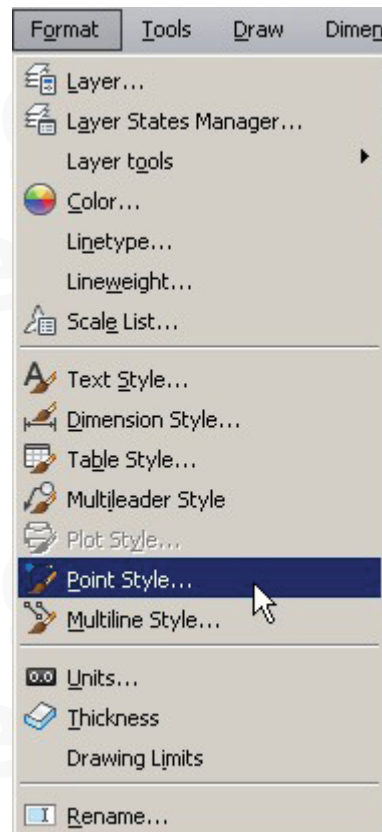


Use AutoCAD Classic Environment.  
ACAD.dwt template

**Points** are the most basic element of a drawing. Lines are defined by two points. A circle is defined by a center point and a point on the radius. Setting points at key locations on a layout is a very good drawing technique and helps assure accuracy.

A point is a single pixel on the drawing screen and by being so small may be very hard to see. AutoCAD provides special point marker icons which make points more visible on drawings. At plot time there is an option to display or not display the points on a drawing.

\_\_ Pop down **Format** menu and select **Point Style**.



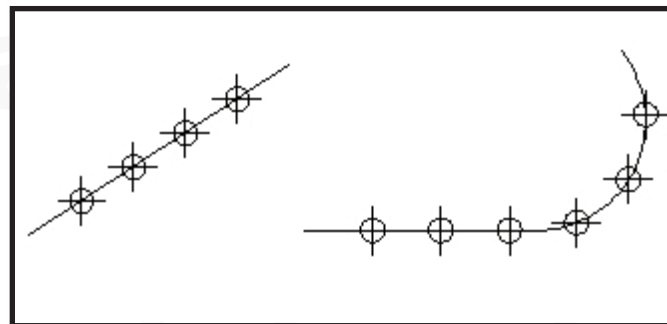
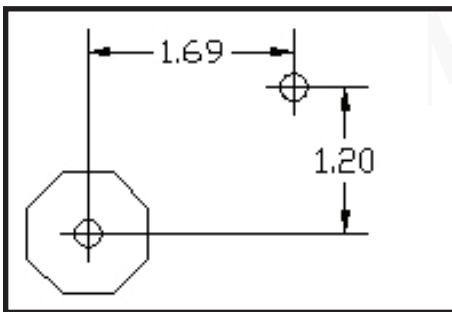
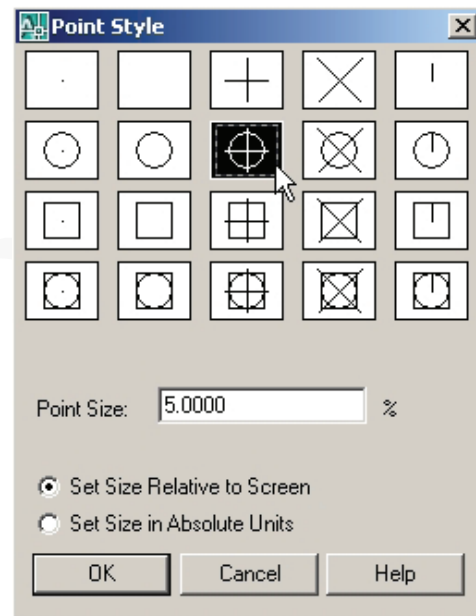
Select a marker shape from the chart. Note the options for a single dot and no marker in the upper left.

Earlier versions of AutoCAD set point styles using the command: **Pdmode**. (Point description mode).  
The marker highlighted is Pmode 34.

The command to draw a point is: **Point** ↵

To snap to a point use **Object Snap: Node**.

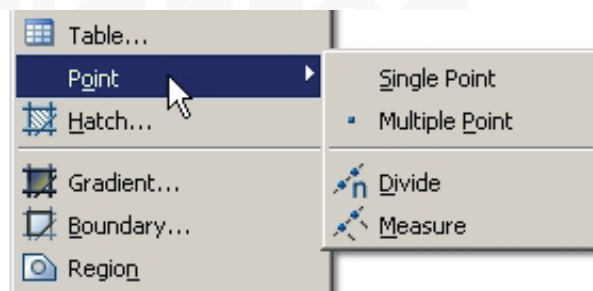
**Tip:** Since polygons do not have a center point, set a point first then draw the polygon. Dimensions to the center of the polygon can reference the Node.



**Divide** and **Measure** are on a flyout from the **Point** command.

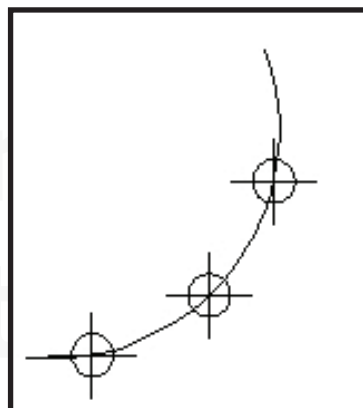
**Draw .... Point .... Measure or Divide**

These commands set points at intervals along lines, polylines, circles, arcs, splines, etc.



**Divide** will place points at equal intervals.

**Measure** will place points at the distance specified. Pick the end of the object to start measuring from.

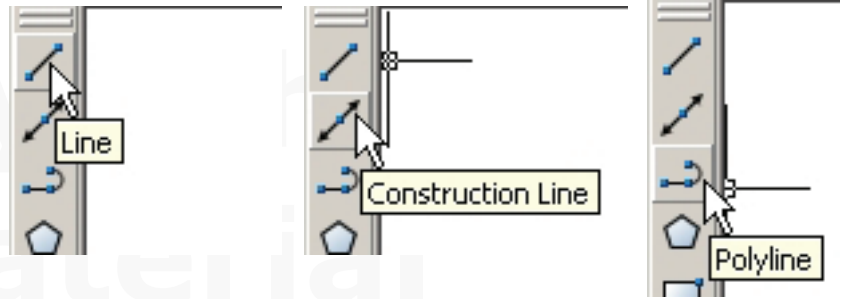


Measure along an arc.

**Lines.** Several types of lines are available.

**Standard lines** are individual entities used to draw shapes.

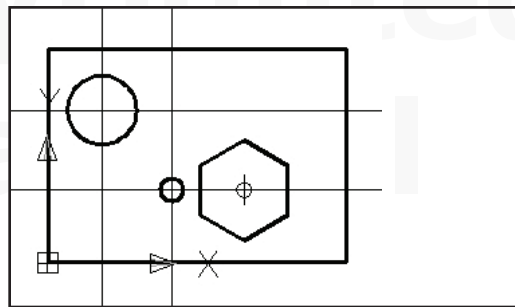
**Construction lines** are infinite in length and do not print on drawings.



**Polylines** are a series of connected lines and arcs. They are considered one entity. Polylines were originally added to define tool paths for machining. Polylines are also important for defining shapes to be extruded onto solids. Polylines should be used for drawing shapes whenever possible.

```
Command: _xline Specify a point or [Hor/Ver/Ang/Bisect/Offset]:
Specify through point: @3<30
```

**Construction lines - xlines** are defined as shown. They are used to quickly “block in” key locations on a layout.



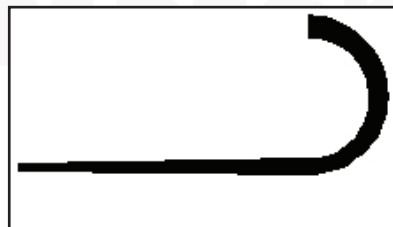
**Polylines** may be returned to individual lines and arcs using the **Explode** command.

Individual lines and arcs may be joined into polylines using the **Pedit .... Join** commands.

```
Command: pedit
Select polyline or [Multiple]:
Object selected is not a polyline
Do you want to turn it into one? <Y>

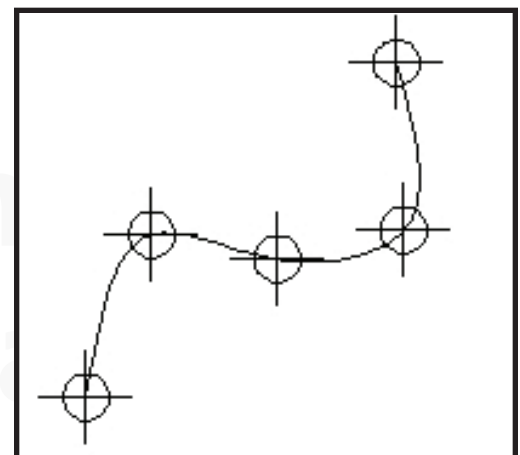
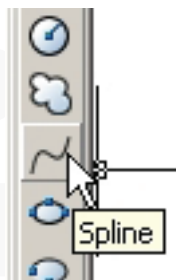
Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype
gen/Undo]: j
```

Polylines may be assigned thickness and variable thickness.



**Spline** is a special type of mathematical curve. It is used to define curved shapes that are more complex than simple arcs.

A spline curve was placed through several points.



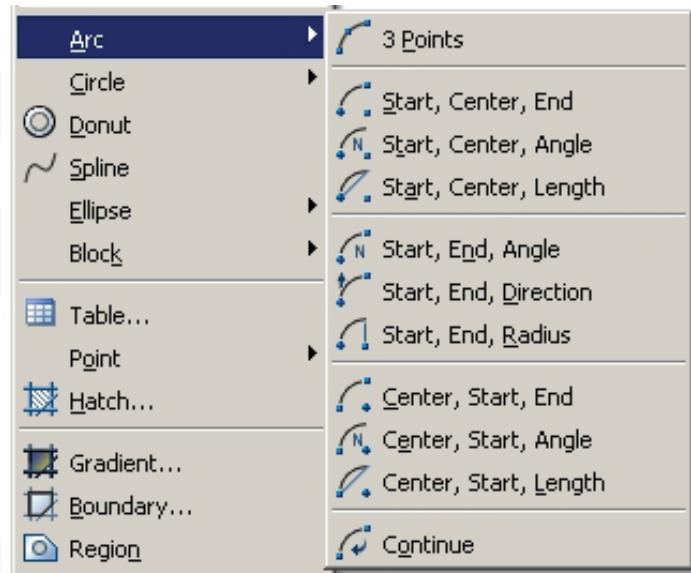
**Arcs.**

Many options are available for drawing arcs.

In many drawings it is better to draw a circle then trim the circle to display the arc segment needed. (Examples will be shown later.)

Circles require 2 points to draw - Arcs require 3 points.

Polygons were drawn on manual drawings by first drawing a construction circle then inscribing the polygon or circumscribing the polygon. The circle acted as a “control surface” to assure accuracy.



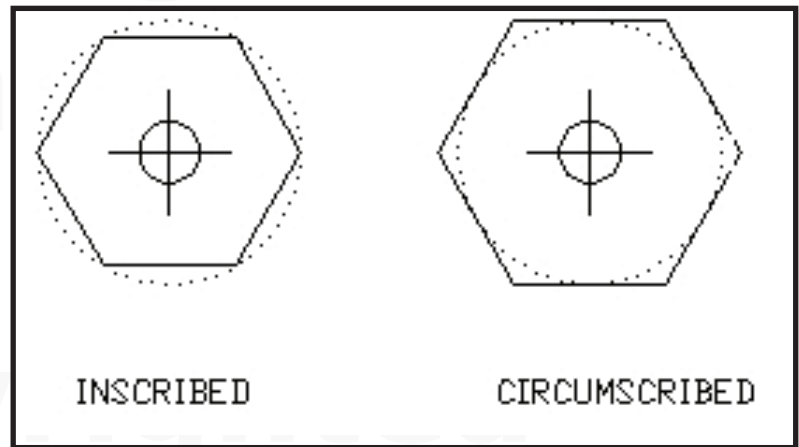
AutoCAD used this analogy when prompting for polygon construction data.

*The circles are not drawn but the prompt is based on the circle construction.*

**Polygons.** AutoCAD will draw regular polygons from 3 sides up.

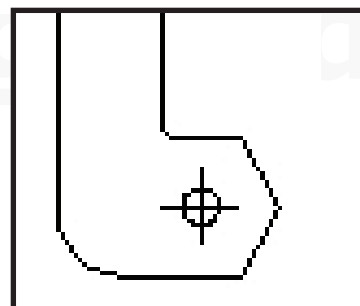
Procedure:

1. Set a point at the center of the polygon.
2. Click Polygon
3. Input number of sides.
4. Select the center point.
5. Set Inscribed or Circumscribed. (I or C)
6. Input “Radius of circle.”



```
Command: _polygon Enter number of sides <8>: 6
Specify center of polygon or [Edge]:
Enter an option [Inscribed in circle/Circumscribed about circle] <C>: c
Specify radius of circle: 1
```

Polygons are constructed using polylines. Polygons may be exploded to edit individual elements.

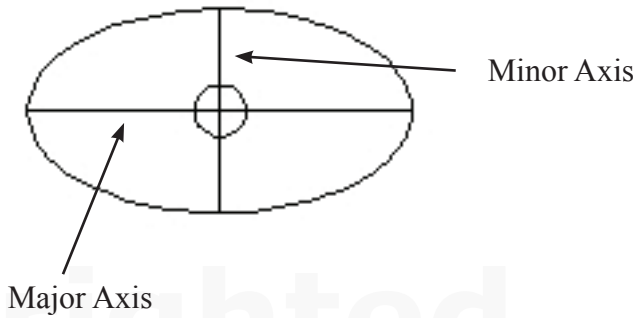


**Ellipses** are generated by two methods:

1. Specify the ends of the first axis and one end of the second axis. (default method).
2. Specify the center, end of one axis, end of the other axis.

```
Command: _ellipse
Specify axis endpoint of ellipse or [Arc/Center]:
Specify other endpoint of axis:
Specify distance to other axis or [Rotation]:
```

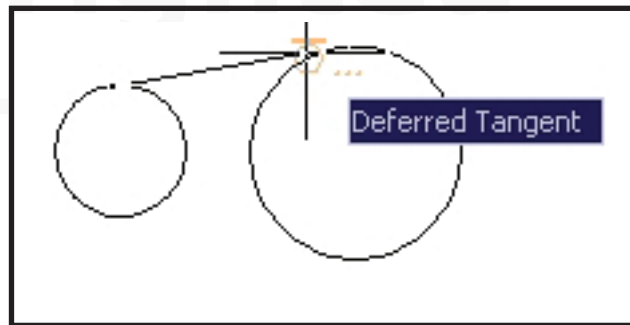
A **Ellipse Arc** command is also available.



**Tangent** constructions are needed for many shapes.

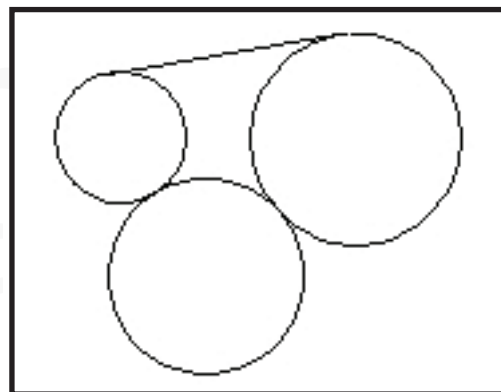
**Line Tangent to two circles.**

- \_\_ Start the Line command.
- \_\_ Invoke Tangent Object Snap to start the line.
- \_\_ Pick the 1st circle.
- \_\_ Invoke Tangent Object Snap to finish the line.
- \_\_ Click the 2nd circle.



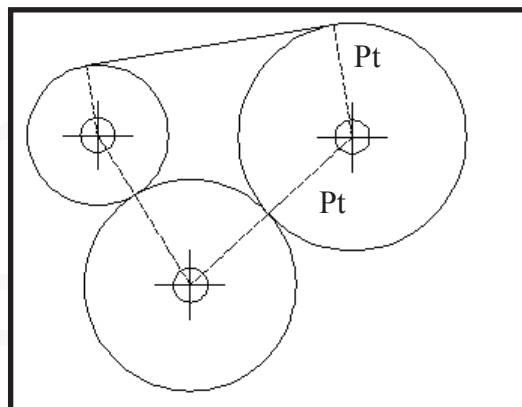
**Circle Tangent to two circles.**

- \_\_ Start the circle command.
- \_\_ Type TTR (Two tangents and a Radius).
- \_\_ Click the first circle
- \_\_ Click the 2nd circle
- \_\_ Type in the Radius.



*Note: AutoCAD infers the circle connection and direction based on how you select the existing circles.*

Tangent lines show points of tangency in the lower figure.





## Editing Geometry

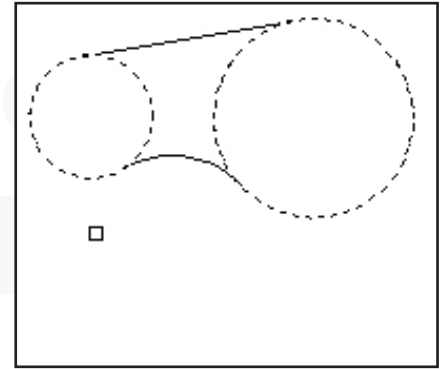
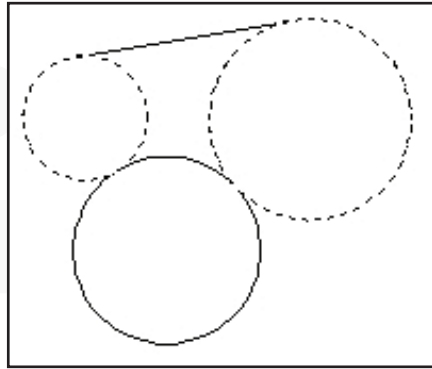
**Trim.** This command requires a bit of practice as it works differently than you might expect.

*First, select the cutting elements.*

*Press Enter*

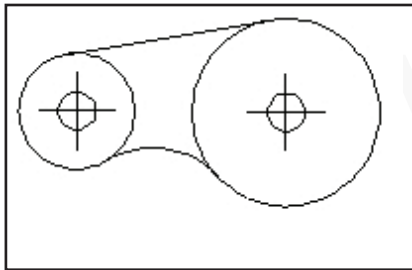
*Second, select the object to be cut.  
(Select the line to remove).*

*Press Enter*

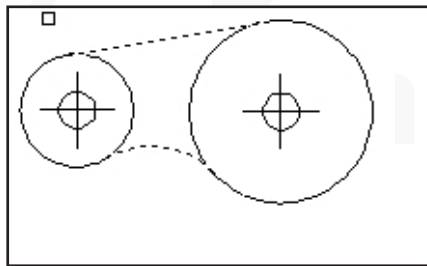


\_\_ Start Trim  
 \_\_ Select the left and right circles.  
 \_\_ Press Enter ↵

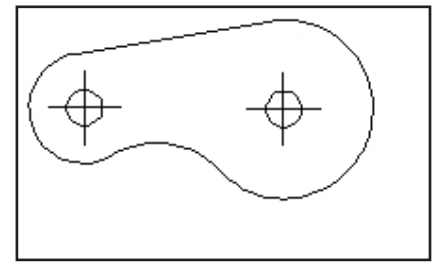
\_\_ Select the bottom of the middle circle.  
 \_\_ Press Enter ↵



\_\_ Start the Trim command.



\_\_ Select the top line and the bottom arc.  
 \_\_ press Enter ↵



\_\_ Click the inner arcs.  
 \_\_ press Enter ↵

**Copy and Move** work in a similar way.

1st select the objects.

Press Enter.

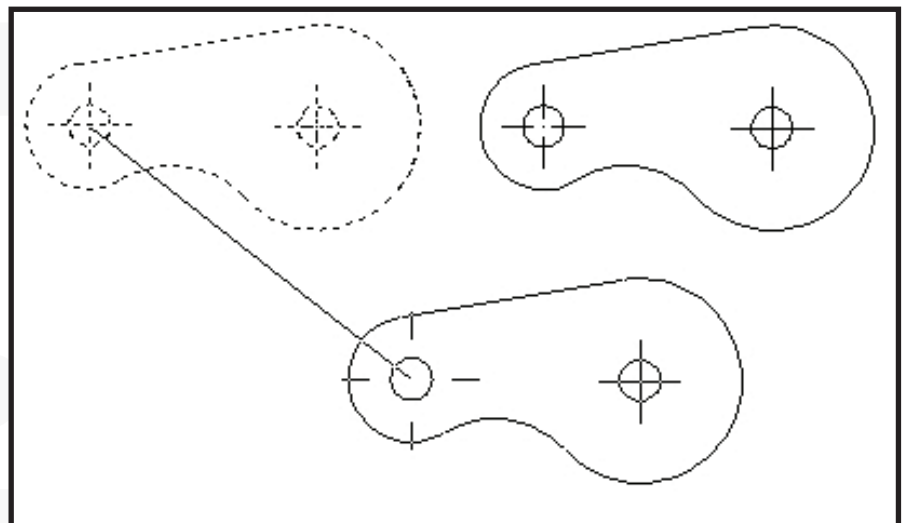
2nd Show where the objects are now.

Press Enter.

3rd Show where the object is to be copied/moved to. Click to set.

Copy will continue placing more copies.

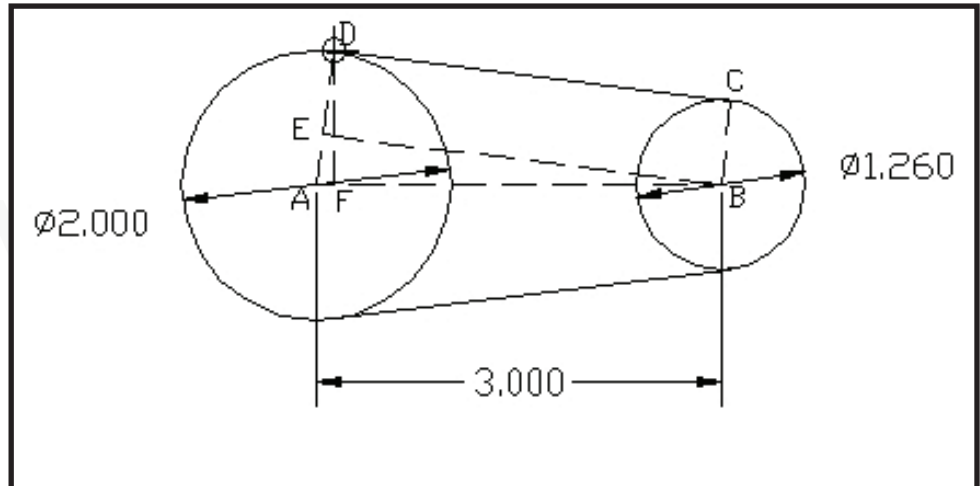
Press ESC.



*Practice with the commands shown. be sure you are familiar with the way each command works and what responses are needed at each step.*

**Getting Information.** The internal accuracy of AutoCAD drawings is 17 decimal places. If the drawing is accurate the very accurate data can be obtained.

The X,Y location of the tangent point at D is needed for machining or surveying. Calculating the tangent point involves solving several right triangles.



AutoCAD will display the X,Y coordinates using the ID command.

1. Reset the UCS to the center point at "A"
2. Type ID and point at the tangent point at "D". The X,Y data is displayed.

```
Specify new origin point <0,0,0>: end
of
Command: id
Specify point: end
of X = 0.1233      Y = 0.9924      Z = 0.0000
```

**Exercise C-201.**

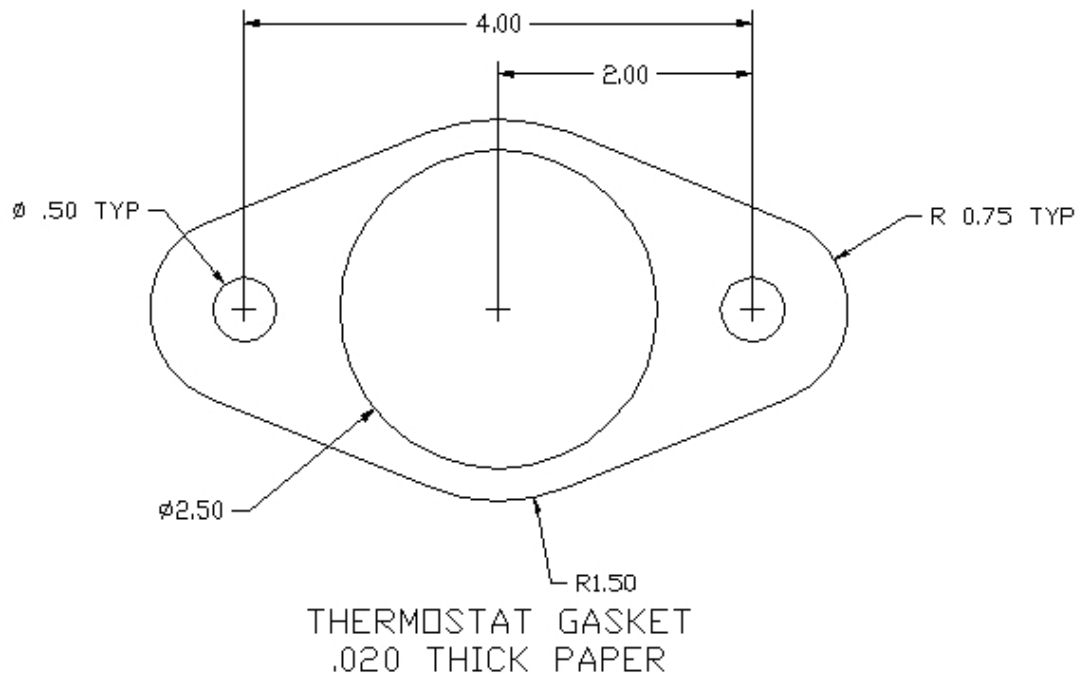
Draw the part shown.

\_\_ Dimension as shown.

\_\_ Put your name on the drawing.

\_\_ Save.

\_\_ Plot.



Material

**Exercise C-202**

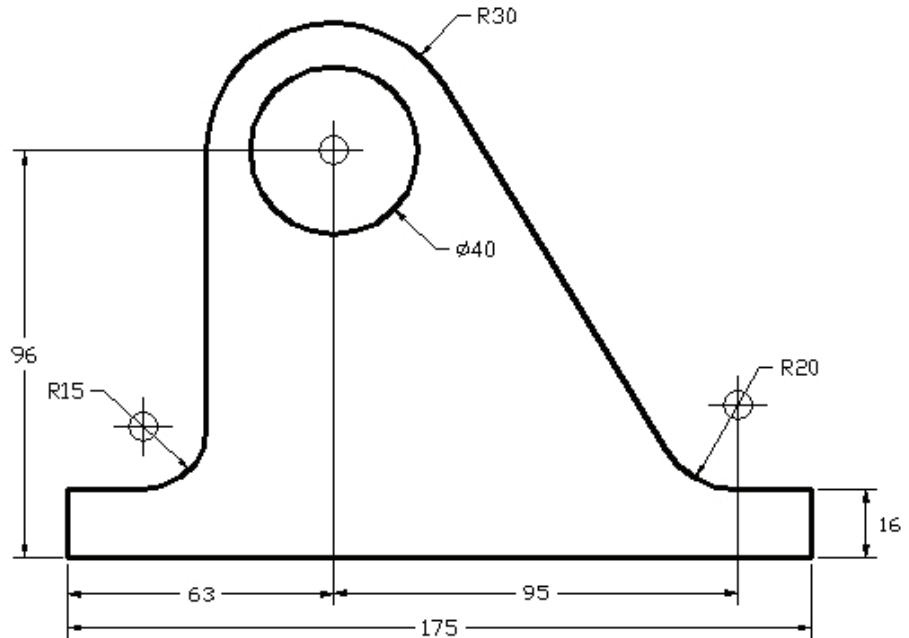
Draw the part shown.

\_\_ Dimension as shown.

\_\_ Put your name on the drawing.

\_\_ Save.

\_\_ Plot.



SUPPORT - RH  
1 REQD, MILD STEEL  
METRIC DIMENSIONS

**Exercise C-203.**

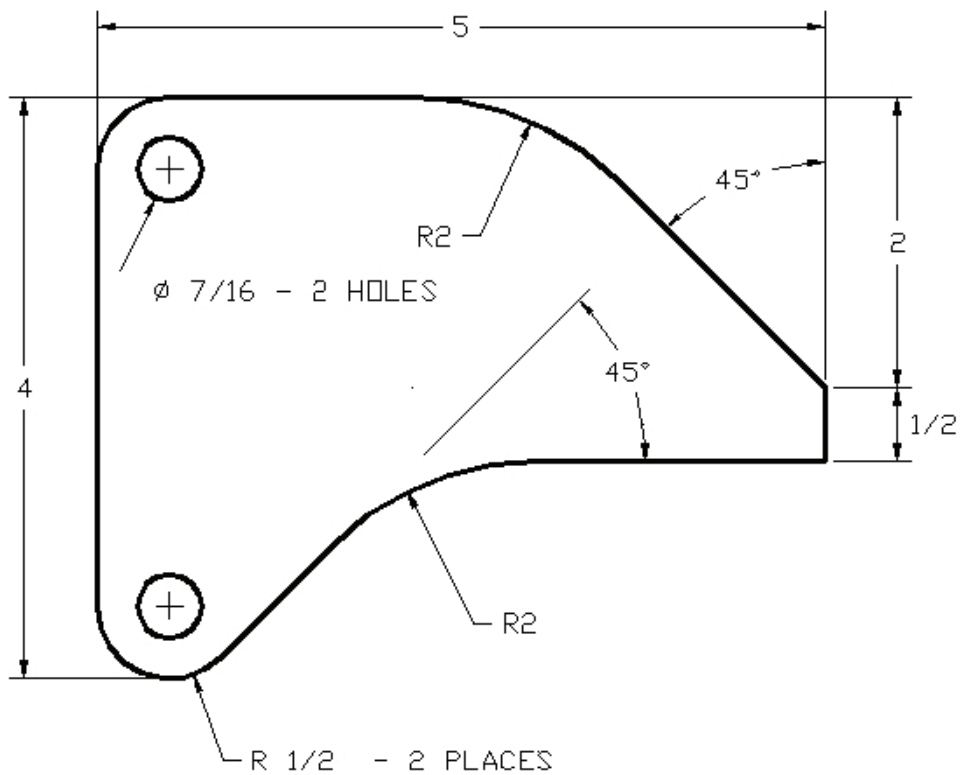
Draw the part shown.

\_\_ Dimension as shown.

\_\_ Put your name on the drawing.

\_\_ Save.

\_\_ Plot.



SCRAPER PLATE  
1 REQD, ALUMINUM