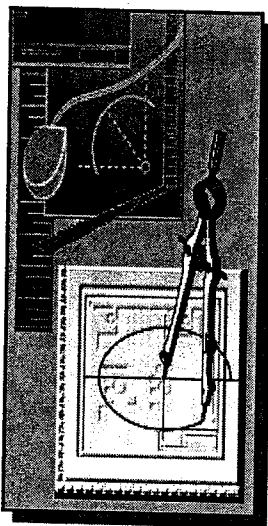
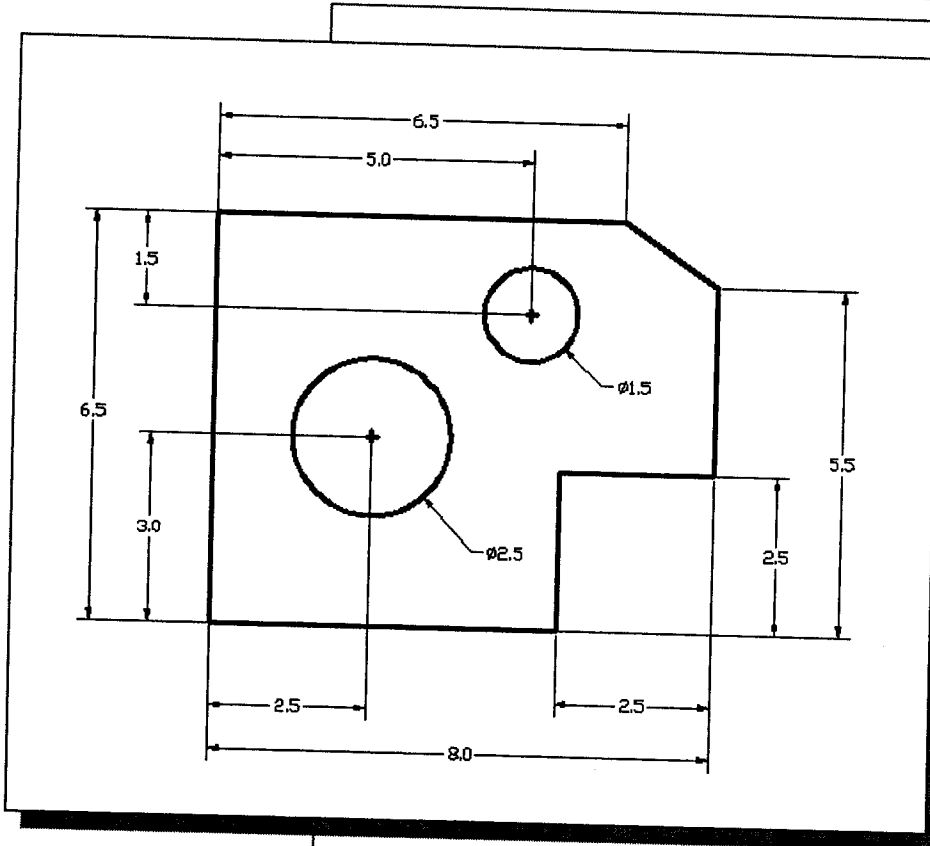


## Lesson 2

# Geometric Construction Basics

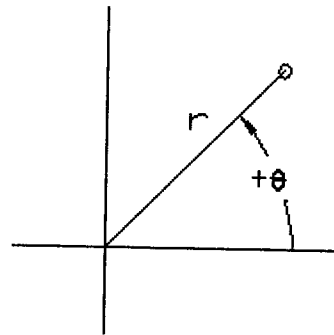
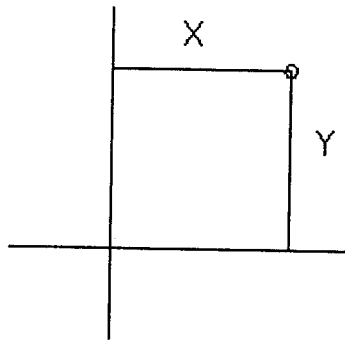


## Learning Objectives

- ◆ Create and Save AutoCAD drawing files.
- ◆ Use the AutoCAD visual reference commands.
- ◆ Draw, using the LINE and CIRCLE commands.
- ◆ Use the ERASE command.
- ◆ Define Positions using the Basic Entry methods.
- ◆ Use the AutoCAD Pan Realtime option.

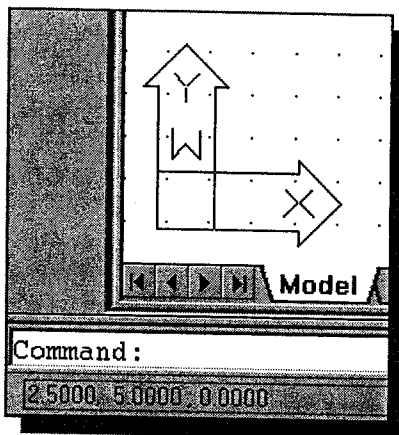
## Cartesian and Polar Coordinate Systems

In a two-dimensional space, a point can be represented using different coordinate systems. The point can be located, using a *Cartesian coordinate system*, as X and Y units away from the origin. The same point can also be located using the *polar coordinate system*, as r and  $\theta$  units away from the origin.



For planar geometry, the polar coordinate system is very useful for certain applications. In the polar coordinate system, points are defined in terms of a radial distance,  $r$ , from the origin and an angle  $\theta$  between the direction of  $r$  and the positive X axis. The default system for measuring angles in **AutoCAD® 2004** defines positive angular values as counter-clockwise from the positive X-axis.

## Absolute and Relative Coordinates



- **AutoCAD® 2004** also allows us to use *absolute* and *relative coordinates* to quickly construct objects. **Absolute coordinate values** are measured from the current coordinate system's origin point. **Relative coordinate values** are specified in relation to previous coordinates.
- Note that the *coordinate display area* can also be used as a toggle switch; each left-mouse-click will toggle the coordinate display *on* or *off*.

In **AutoCAD® 2004**, the *absolute* coordinates and the *relative* coordinates can be used in conjunction with the *Cartesian* and *polar* coordinate systems. By default, AutoCAD expects us to enter values in *absolute Cartesian coordinates*, distances measured from the current coordinate system's origin point. We can switch to using the *relative coordinates* by using the @ symbol. The @ symbol is used as the *relative coordinates specifier*, which means that we can specify the position of a point in relation to the previous point.

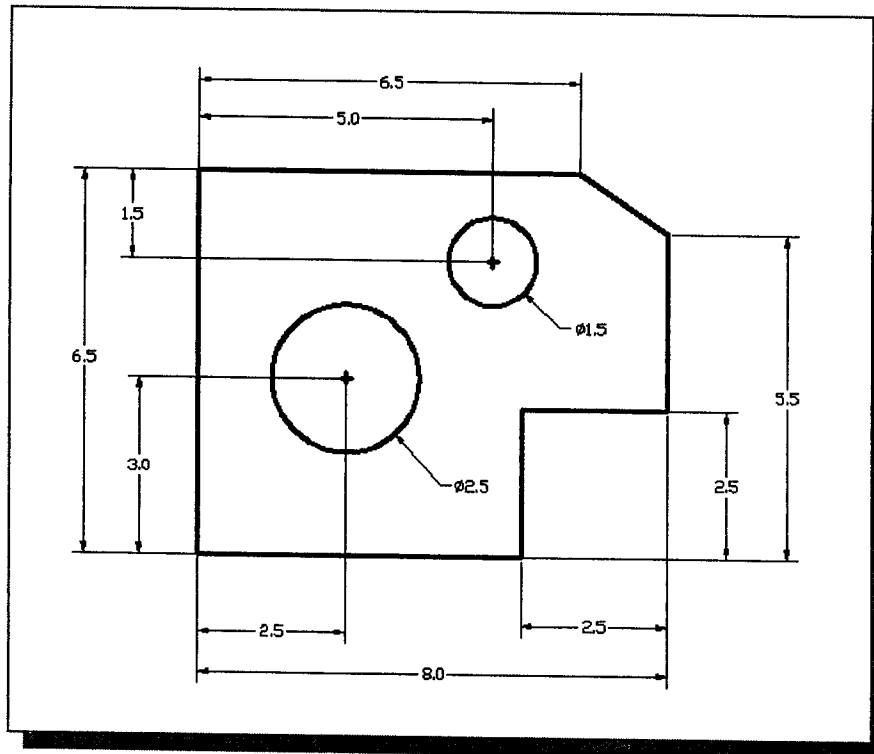
## Defining Positions

In AutoCAD, there are five methods for specifying the locations of points when we create planar geometric entities.

- **Interactive method:** Use the cursor to select on the screen.
- **Absolute coordinates (Format: X,Y):** Type the X and Y coordinates to locate the point on the current coordinate system relative to the origin.
- **Relative rectangular coordinates (Format: @X,Y):** Type the X and Y coordinates relative to the last point.
- **Relative polar coordinates (Format: @Distance<angle):** Type a distance and angle relative to the last point.
- **Direct Distance entry technique:** Specify a second point by first moving the cursor to indicate direction and then entering a distance.

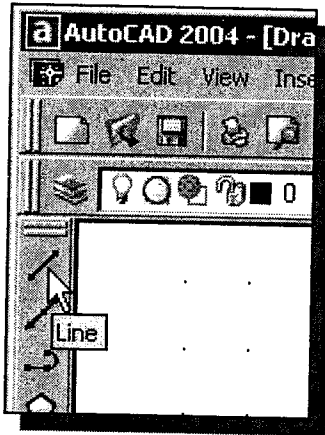
## The Guide Plate

We will next create a mechanical design using the different coordinate entry methods.



- Use the Erase command and erase all entities on the screen before proceeding to the next section.

- ❖ The rule for creating CAD designs and drawings is that they should be created at **full size** using real-world units. The CAD database contains all the definitions of the geometric entities and the design is considered as a virtual, full-sized object. Only when a printer or plotter transfers the CAD design to paper is the design scaled to fit on a sheet. The tedious task of determining a scale factor so that the design will fit on a sheet of paper is taken care of by the CAD system. This allows the designers and CAD operators to concentrate their attention on the more important issues – the design.

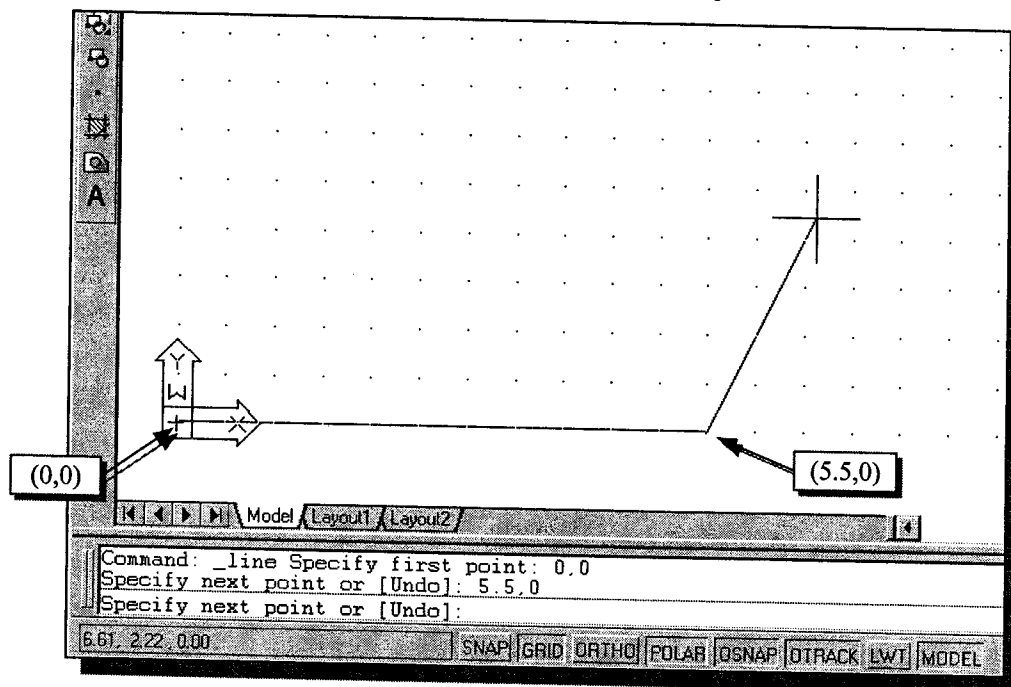


1. Select the **Line** command icon in the *Draw* toolbar. In the command prompt area, near the bottom of the AutoCAD graphics window, the message “*\_line Specify first point:*” is displayed. AutoCAD expects us to identify the starting location of a straight line.
2. In the command prompt area, we will locate the starting point of our design at the origin of the *world coordinate system*.

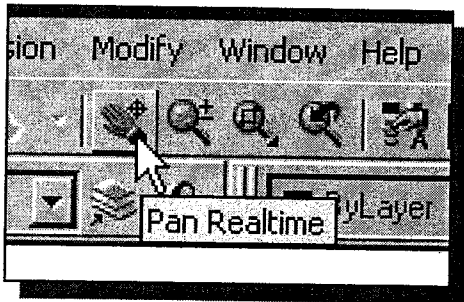
*Command: \_line Specify first point: 0,0 [ENTER]*  
 (Type 0,0 in the command prompt area and press the [ENTER] key once.)

3. We will create a horizontal line by entering the absolute coordinates of the second point.

*Specify next point or [Undo]: 5.5,0 [ENTER]*



- The line we created is aligned to the bottom edge of the drawing window. Let us adjust the view of the line by using the Pan Realtime command.

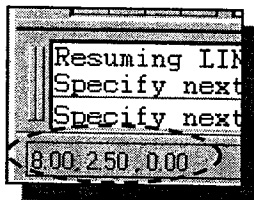
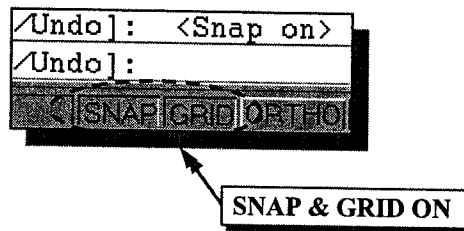


4. Click on the **Pan Realtime** icon in the *Standard* toolbar area. The icon is the picture of a hand with four arrows.
  - ❖ The Pan command enables us to move the view to a different position. This function acts as if you are using a video camera.

5. Move the cursor, which appears as a hand inside the graphics window, near the center of the drawing window, then push down the left-mouse-button and drag the display toward the right and top side until we can see the sketched line. (Notice the scroll bars can also be used to adjust viewing of the display.)
6. Press the [Esc] key to exit the Pan command. Notice that AutoCAD goes back to the Line command.
7. We will create a vertical line by using the *relative rectangular coordinates entry method*, relative to the last point we specified:

*Specify next point or [Close/Undo]: @0,2.5 [ENTER]*

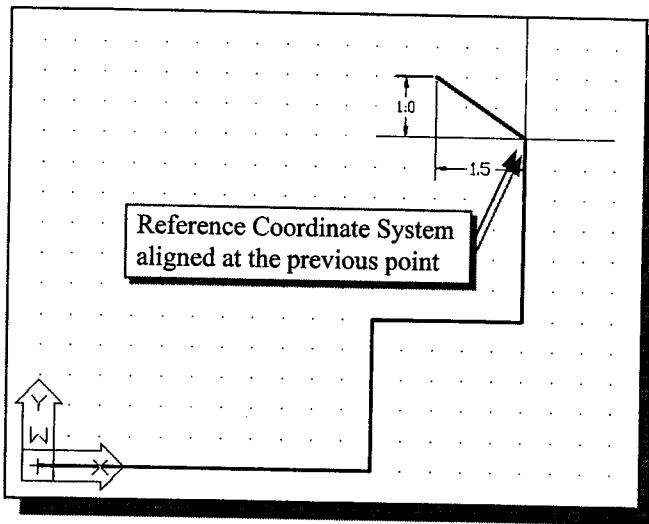
8. We can mix any of the entry methods in positioning the locations of the endpoints. Move the cursor to the *Status Bar* area, and turn on the *GRID* and *SNAP* options.



9. Create the next line by picking the location, world coordinates (8,2.5), on the screen.

10. We will next use the *relative polar coordinates entry method*, relative to the last point we specified:

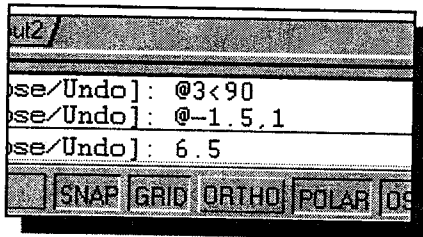
*Specify next point or [Close/Undo]: @3<90 [ENTER]*  
 (Distance is 3 inches with an angle of 90 degrees)



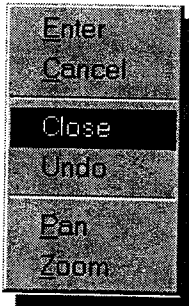
11. Using the *relative rectangular coordinates entry method* to create the next line, we can imagine a *reference coordinate system* aligned at the previous point. Coordinates are measured along the two reference axes.

Specify next point or [Close/Undo]:  
 @-1.5,1 [ENTER]

(-1.5 and 1 inches are measured relative to the reference point.)

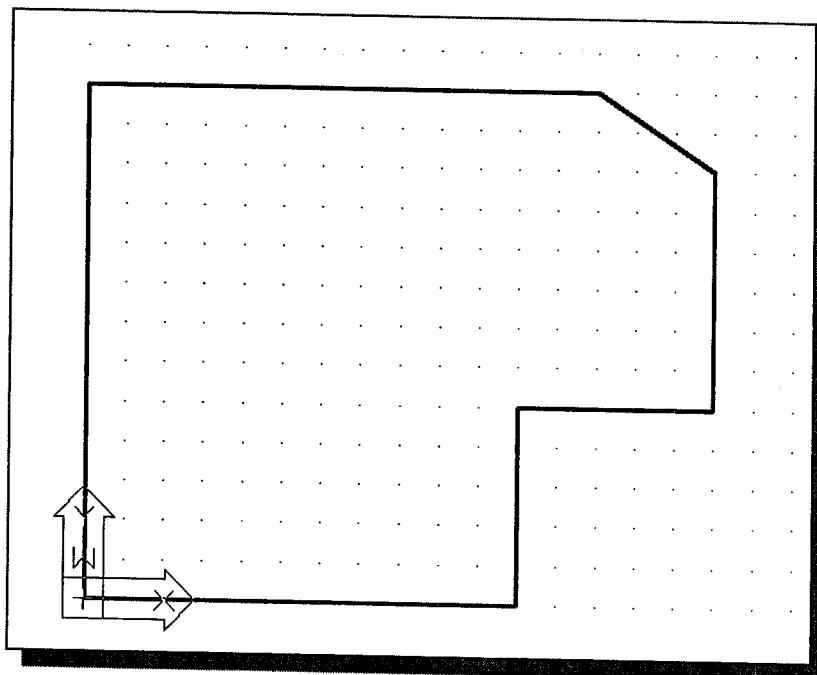


12. Move the cursor directly to the left of the last point and use the *direct distance entry technique* by entering 6.5 [ENTER].



13. For the last segment of the sketch, we can use the **Close** option to connect back to the starting point. Inside the graphics window, right-mouse-click and a popup menu appears on the screen.

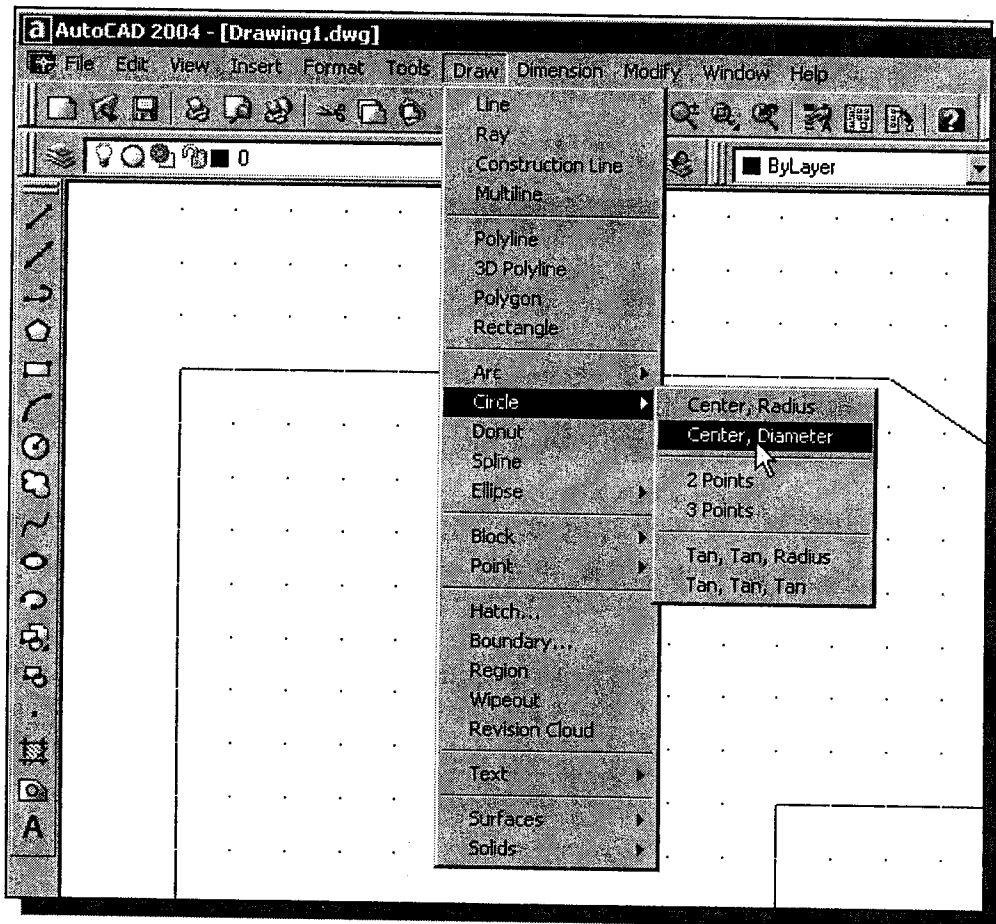
14. Select **Close** with the left-mouse-button to connect back to the starting point and end the Line command.



## Creating Circles

The menus and toolbars in AutoCAD® 2004 are designed to allow the CAD operators to quickly activate the desired commands. Besides using the *Draw* toolbar, we can also select the different *Draw* commands through the pull-down menus.

1. In the pull-down menus, select:  
**[Draw] → [Circle] → [Center, Diameter]**



Notice the different options available under the circle submenu:

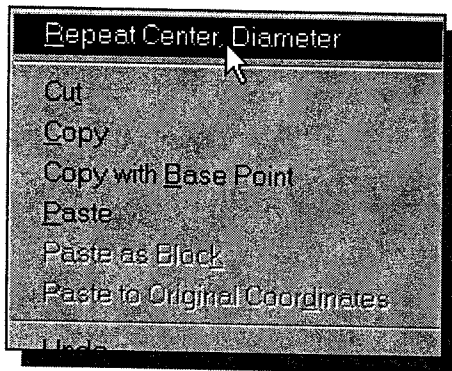
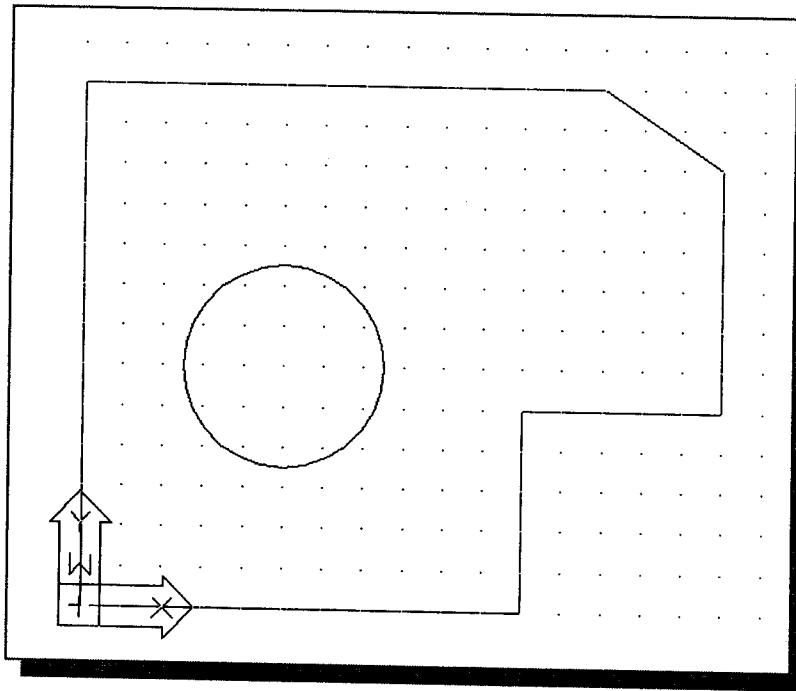
- **Center Point:** Draws a circle based on a center point and a diameter or a radius.
- **3 Points:** Draws a circle based on three points on the circumference.
- **2 Points:** Draws a circle based on two endpoints of the diameter.
- **TTR–Tangent, Tangent, Radius:** Draws a circle with a specified radius tangent to two objects.
- **TTT–Tangent, Tangent, Tangent:** Draws a circle tangent to three objects.

- In the command prompt area, the message “Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:” is displayed. AutoCAD expects us to identify the location of a point or enter an option. We can use any of the four coordinate entry methods to identify the desired location. We will enter the **world coordinates (2.5,3)** as the center point for the first circle.

*Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:* 2.5,3 [ENTER]

- In the command prompt area, the message “Specify diameter of circle:” is displayed.

*Specify diameter of circle:* 2.5 [ENTER]



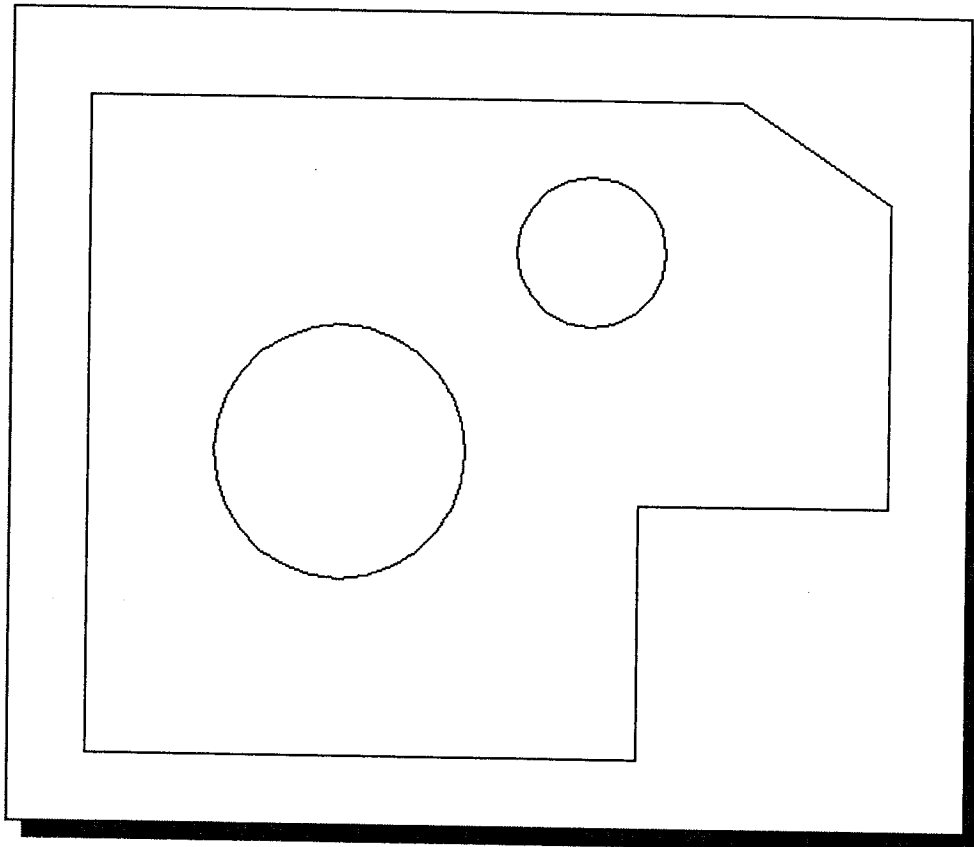
- Inside the graphics window, right-mouse-click to bring up the popup option menu.
- Pick **Repeat Center, Diameter** with the left-mouse-button in the popup menu to repeat the last command.
- Using the *relative rectangular coordinates entry method*, relative to the center-point coordinates of the first circle, we specify the location as (2.5,2).

*Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:* @2.5,2 [ENTER]

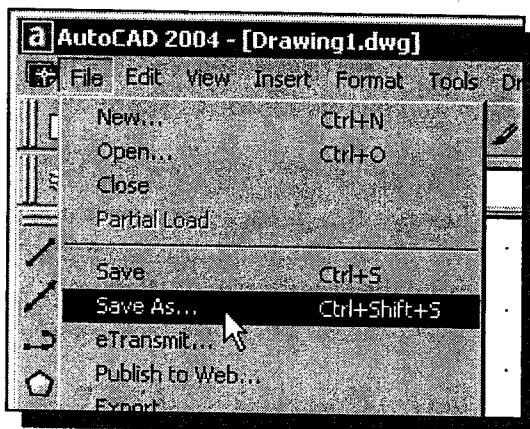


7. In the command prompt area, the message “*Specify Diameter of circle:* <2.50>” is displayed. The default option for the **Circle** command in AutoCAD is to specify the *radius* and the last radius used is also displayed in brackets.

*Specify Diameter of circle*<2.50>: 1.5 [ENTER]



## Saving the CAD Design

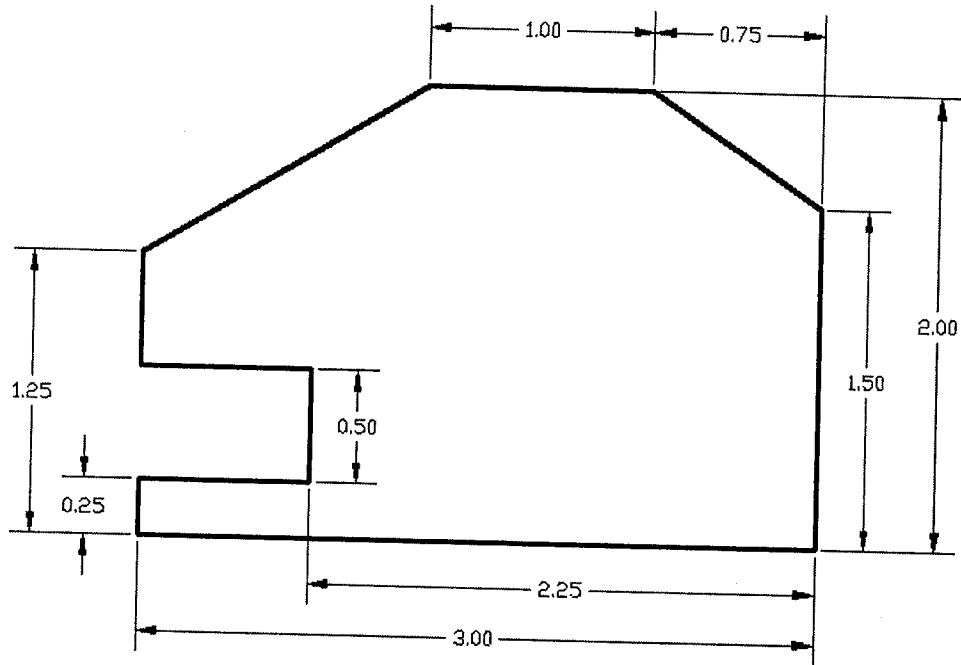


1. In the pull-down menus, select:

**[File] → [Save As]**

**Exercises:** (All dimensions are in inches.)

1.



2.

