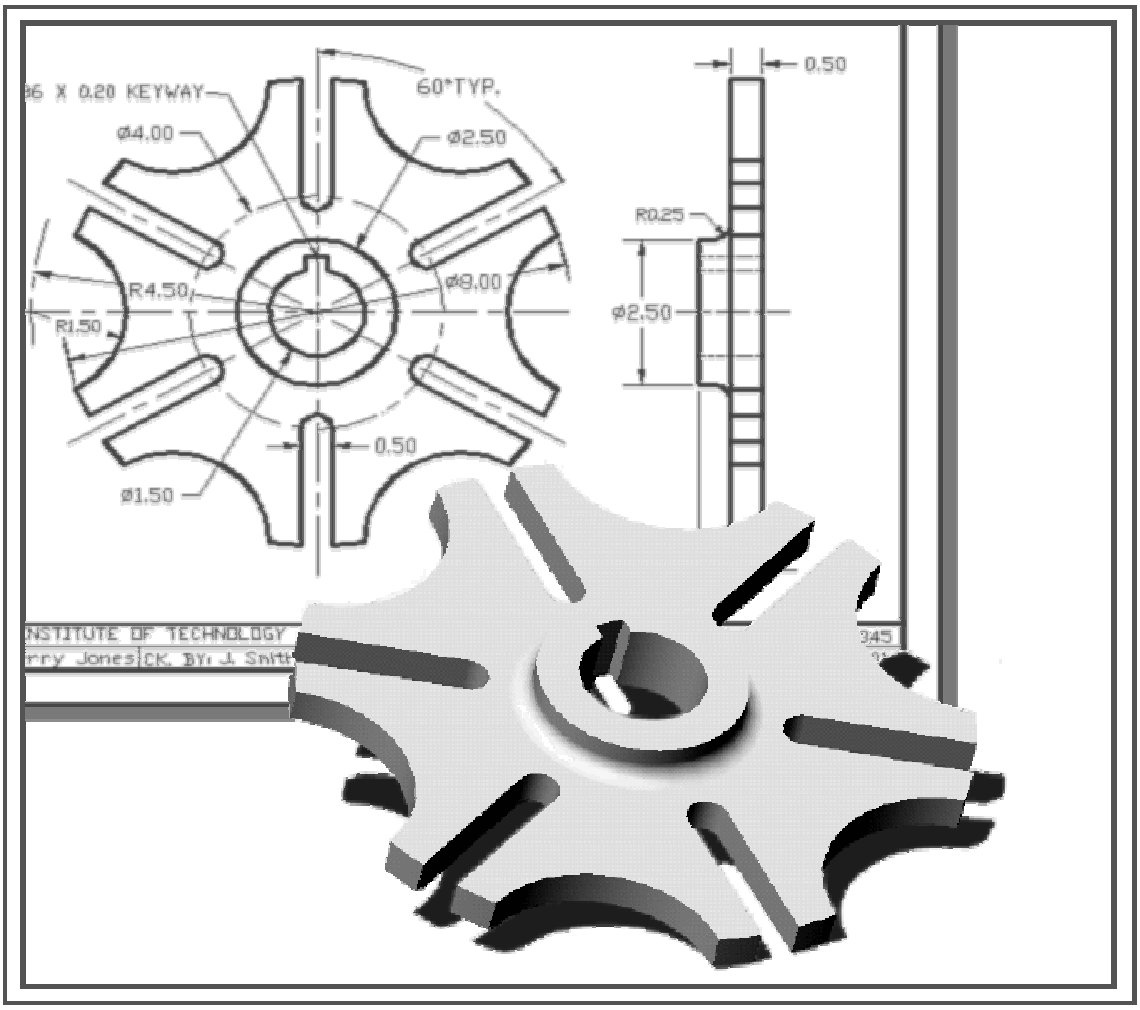
AutoCAD® LT 2000 MultiMedia Tutorial



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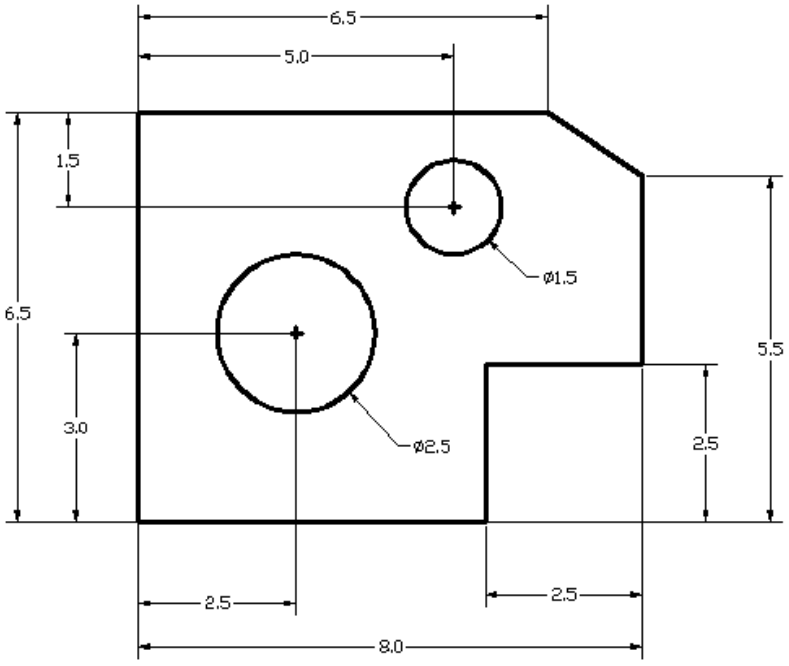
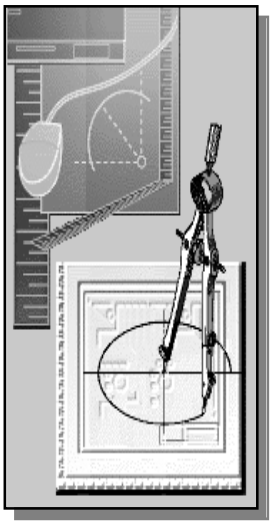
Lesson 1
Geometric Construction Basics





Learning Objectives

When you have completed this lesson, you will be able to:

- **Create an AutoCAD drawing file.**
- **Use Visual reference commands.**
- **Draw, using the Line and Circle commands.**
- **Use the ERASE command.**
- **Define Positions using the Basic Entry methods.**
- **Use the Pan Realtime command.**
- **Save an AutoCAD drawing File.**

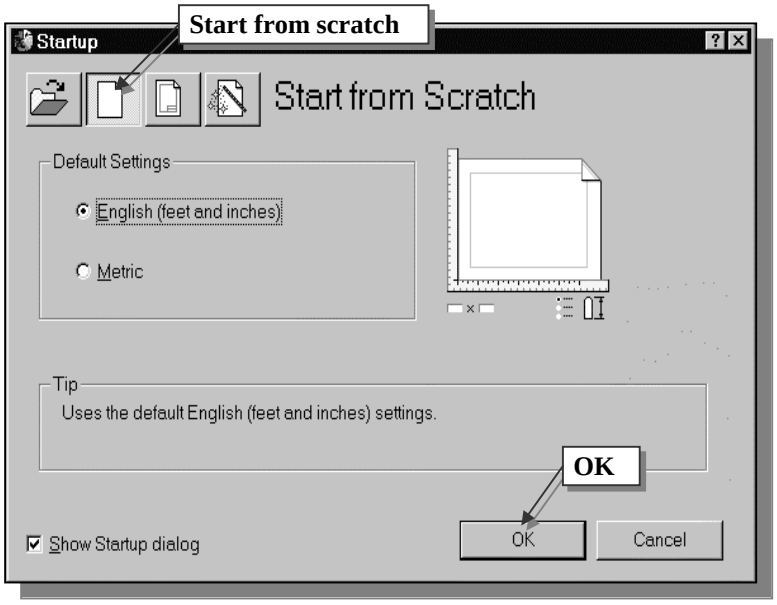
Introduction

Learning to use a CAD system is similar to learning a new language. We need to begin with the basic alphabets and learn how to use them correctly and effectively through practice. This will require learning some new concepts and skills as well as learning a different vocabulary. All CAD systems create designs using basic geometric entities. Many of the constructions used in technical designs are based upon two-dimensional planar geometry. The method and number of operations that are required to accomplish the construction are different from one system to another.

In order to become effective in using a CAD system, we must learn to create geometric entities quickly and accurately. In learning to use a CAD system, **Lines** and **Circles** are the first two, and perhaps the most important two, geometric entities that we need to master the skills in creating and modifying. Straight lines and circles are used in almost all of the technical designs. In examining the different types of planar geometric entities, we can see that triangles and polygons are planar figures bounded by straight lines. Ellipses and Splines can be constructed by connecting arcs with different radii. As we gain some experience in creating lines and circles, the similar procedures can be applied to create other geometric entities. In this lesson, we will examine the different ways of creating lines and circles in AutoCAD® LT 2000.

Starting Up AutoCAD® LT 2000

1. Select the AutoCAD® LT 2000 option on the program menu or select the AutoCAD® LT 2000 icon on the desktop. Once the program is loaded into memory, the AutoCAD® LT 2000 *Drawing Screen* and the *Startup dialog box* will appear on the screen.

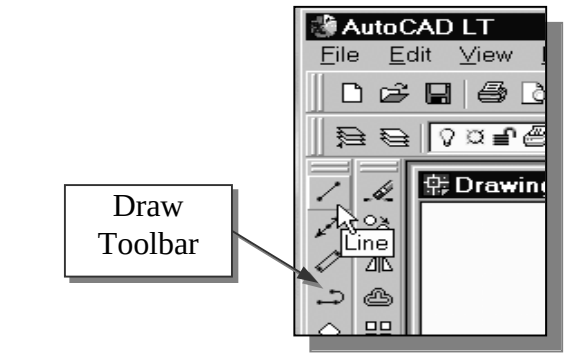


2. In the *Startup dialog box*, select the ***Start from Scratch*** *icon* with a single click of the left-mouse-button.

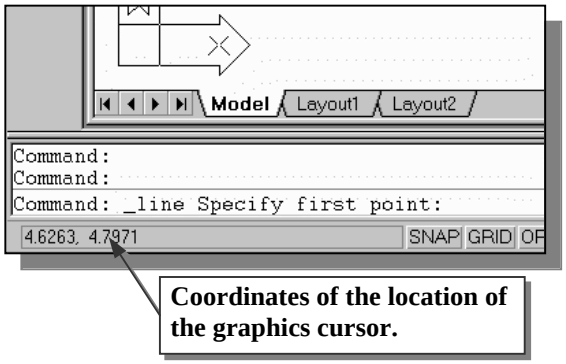
3. In the *Default Settings* section, Pick **English (Feet and Inches)** as the drawing units.

4. Pick **OK** in the *Startup dialog box* to accept the selected settings.

Using the *LINE* command

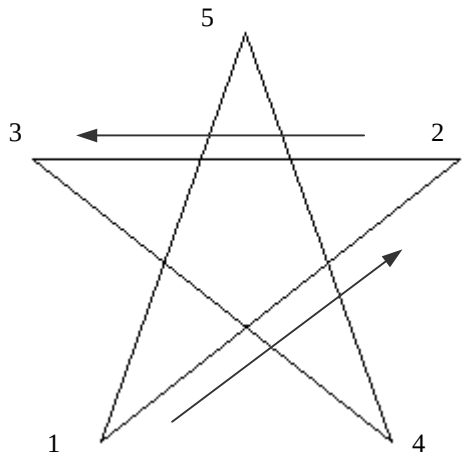


1. Move the *graphics cursor* to the first icon in the *Draw toolbar*. This icon is the **LINE** icon. A *Help-tip box* appears next to the cursor and a brief description of the icon is displayed at the bottom of the AutoCAD *Drawing Screen*: "*Creates Straight line segments: LINE.*"



2. Select the icon by clicking once with the **left-mouse-button**; this will activate the *LINE* command. In the *command prompt area*, near the bottom of the AutoCAD *Drawing Screen*, the message *"_line Specify first point:"* is displayed. AutoCAD expects us to identify the starting location of a straight line.

3. Move the *graphics cursor* inside the *Graphics window* and watch the display of the coordinates of the graphics cursor at the bottom of the AutoCAD LT *Drawing Screen.* The three numbers represent the location of the cursor in the X and Y directions. We can treat the *Graphics window* as if it is a piece of paper and we are using the graphics cursor as if it is a pencil with which to draw.

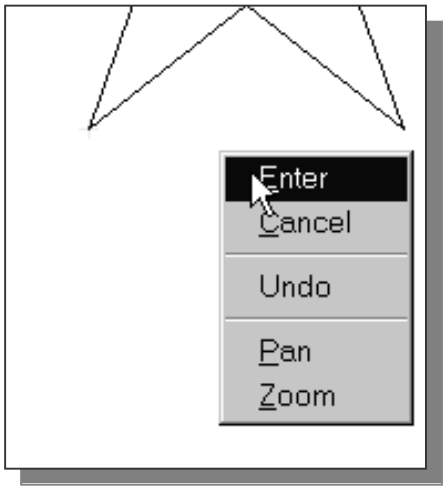


4. We will create a freehand sketch of a five-point star using the *LINE* command. Create the sketch near the center of the *Drawing window*. Do not be overly concerned with the size or the accuracy of your freehand sketch. This exercise is to give you a feel for the AutoCAD® LT 2000 user interface.

5. Start at a location about one-third from the bottom of the *Graphics window*, then left-click once to position the starting point of our first line. This will be *point 1* of our sketch. Next move the cursor upward and toward the right side of *point 1*. Notice the rubber-band line that follows the *graphics cursor* in the *Graphics window*. **Left-click** again and we have created our first line of the sketch.

6. Move the cursor to the left of *point 2* and create a horizontal line about the same length as the first line on the screen.

7. Repeat *step 5* and complete the freehand sketch by adding three more lines (from *point 3* to *point 4*, then connect to *point 5* and back to *point 1*).



8. Notice that the *LINE* command remains activated even after we connected the last segment of the line to the starting point *(point 1)* of our sketch. Inside the *Graphics window*, **click once with the right-mouse-button** and a popup menu appears on the screen.

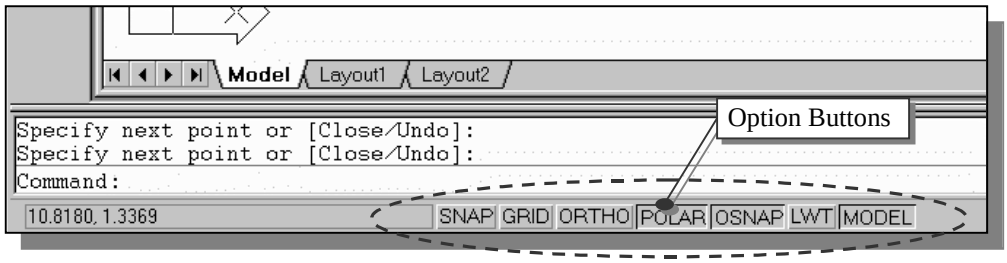
9. Select **Enter** with the left-mouse-button to end the *LINE* command. (This is equivalent to hitting the [**Enter**] key on the keyboard. The *right-mouse-click* brings up more options and we should get used to using the *option menu*.)

10. Move the cursor near *point 2* and *point 3*, and estimate the length of the horizontal line by watching the displayed coordinates for each point at the bottom of the screen.

Visual reference

The method we just used to create the freehand sketch is known as the **interactive method**, where we use the cursor to specify locations on the screen. This method is perhaps the fastest way to specify locations on the screen. However, it is rather difficult to try to create a line of a specific length by watching the displayed coordinates. It would be helpful to know what one-inch or one-meter looks like on the screen while we are creating entities. AutoCAD LT 2000 provides us with many tools to aid the construction of our designs. We will use the ***GRID*** and ***SNAP*** options to get a visual reference as to the size of objects and learn to restrict the movement of the cursor to a set increment on the screen.

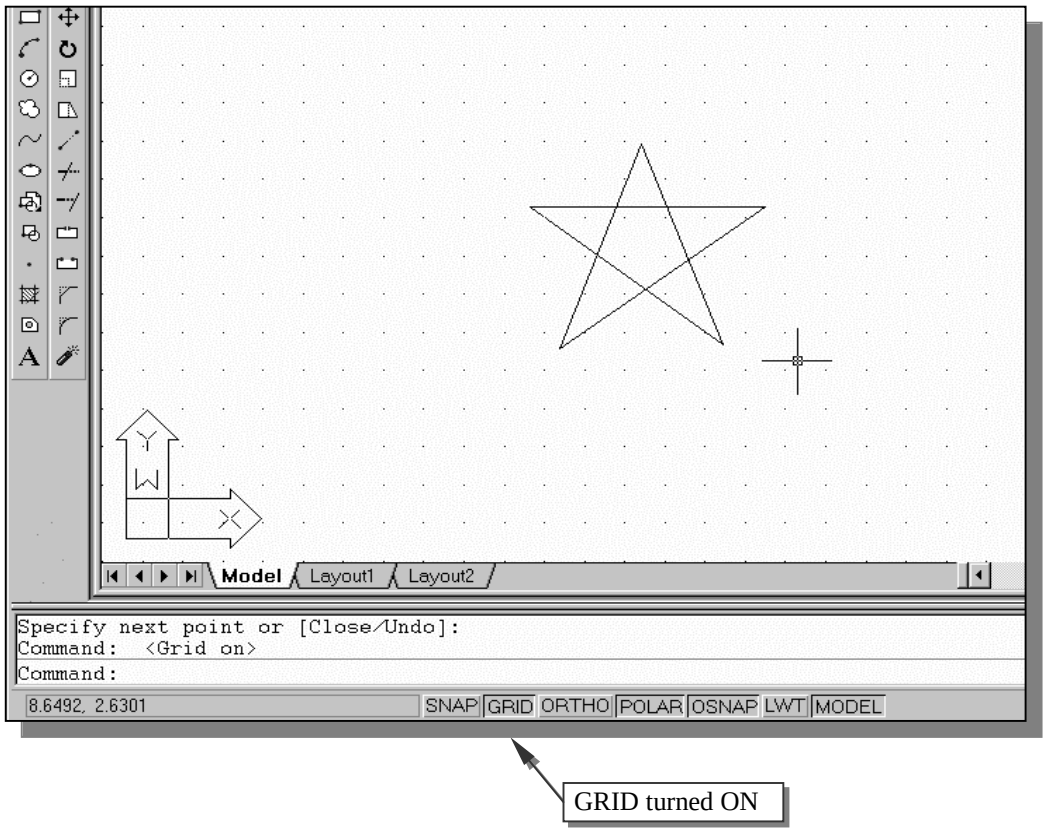
The *Status Bar* area is located at the bottom of the AutoCAD LT *Drawing Screen.* The words SNAP, GRID, ORTHO, POLAR, OSNAP, LWT and MODEL appearing to the right of the coordinates are buttons that we can left-click to turn these special options *ON* and *OFF*. When the corresponding button is *highlighted*, the specific option is turned on. These buttons act as toggle switches; each click of the button will toggle the option on or off. Using the buttons is a quick and easy way to make changes to these *Drawing Aid* options. We can toggle the options on and off in the middle of another command.



GRID ON

1. Left-click the ***GRID* button** in the *Status Bar* to turn ***ON*** the *GRID* option. (Notice in the *command prompt area*, the message *"<Grid on>"* is also displayed.)

2. Move the cursor inside the *Graphics window*, and estimate the distance in between the grid points by watching the coordinates display at the bottom of the screen.



❖ The *GRID* option creates a pattern of dots that extends over an area on the screen. Using the grid is similar to placing a sheet of grid paper under a drawing. The grid helps you align objects and visualize the distance between them. The grid is not displayed in the plotted drawing. The default grid spacing, which means the distance in between two dots on the screen, is 0.5 inches. We can see that the sketched horizontal line in the above sketch is about 2.5 inches long.

SNAP ON

1. Left-click the ***SNAP* button** in the *Status Bar* to turn ***ON*** the *SNAP* option.

2. Move the cursor inside the graphics window, and move the cursor diagonally on the screen. Observe the movement of the cursor and watch the *coordinates display* at the bottom of the screen.

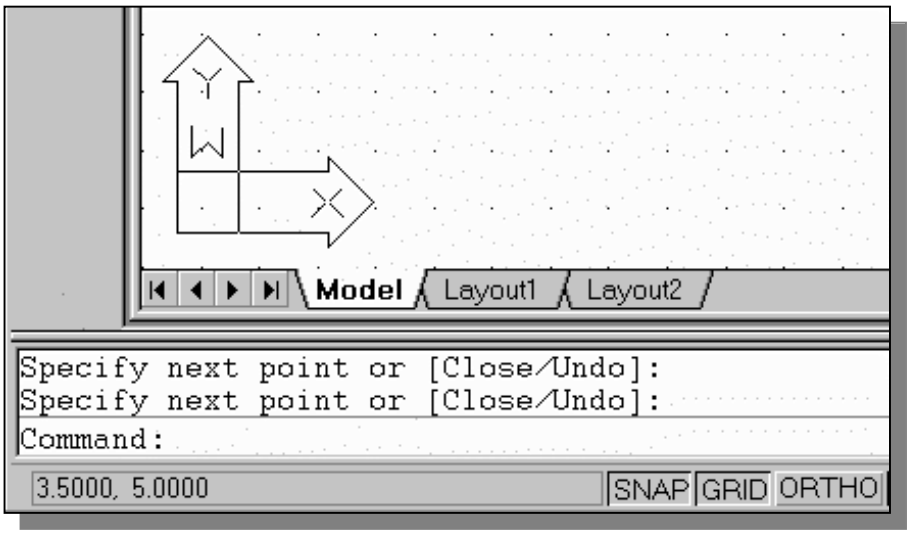
The *SNAP* option controls an invisible rectangular grid that restricts cursor movement to specified intervals. When *SNAP* mode is on, the screen cursor and all input coordinates are snapped to the nearest point on the grid. The default snap interval is 0.5 inches, and aligned to the grid points on the screen.

3. Click on the **LINE** icon in the *Draw toolbar*. In the *command prompt area*, the message *"_line Specify first point:"* is displayed.

4. Create another sketch of the star with the *GRID* and *SNAP* options switched on.

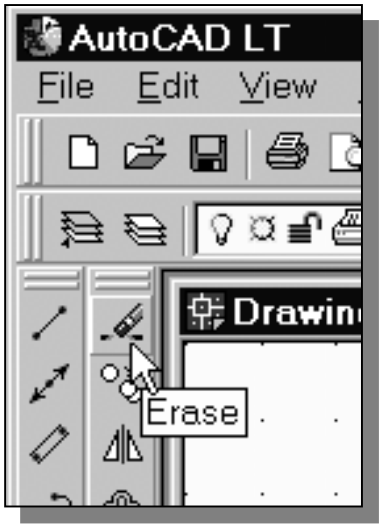
5. Use the **right-mouse-button** and select **Enter** in the popup menu to end the *LINE* command if you haven't done so.

6. In the *command prompt area*, notice that "**Command:**" is displayed. This indicates that AutoCAD LT is waiting for us to activate the next desired command.



Using the *ERASER*

One of the advantages of using a CAD system is the ability to remove entities without leaving any marks. We will erase two of the lines using the *ERASE* command.

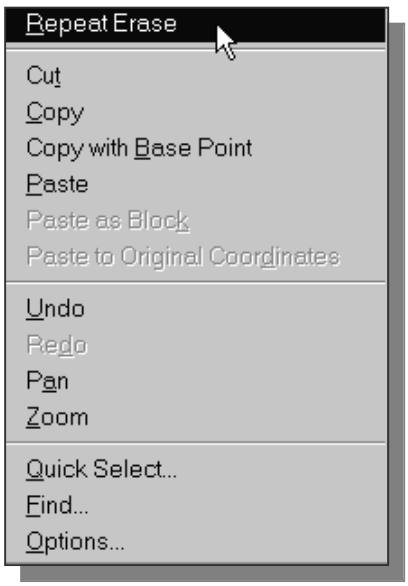


1. Pick ***Erase*** in the *Modify toolbar*. (The icon is the first icon in the *Modify toolbar*. The icon is a picture of an eraser at the end of a pencil.) The message *"Select objects"* is displayed in the *command prompt area* and AutoCAD awaits us to select the objects to erase.

2. **Left-click** the ***SNAP*** button on the *Status Bar* to turn off the *SNAP* option so that we can more easily move the cursor on top of objects. We can toggle the *Status Bar* options on and off in the middle of another command.

3. Select any two lines on the screen, and *right-mouse-click* once to accept the selections. The selected two lines are erased.

Repeat the last command

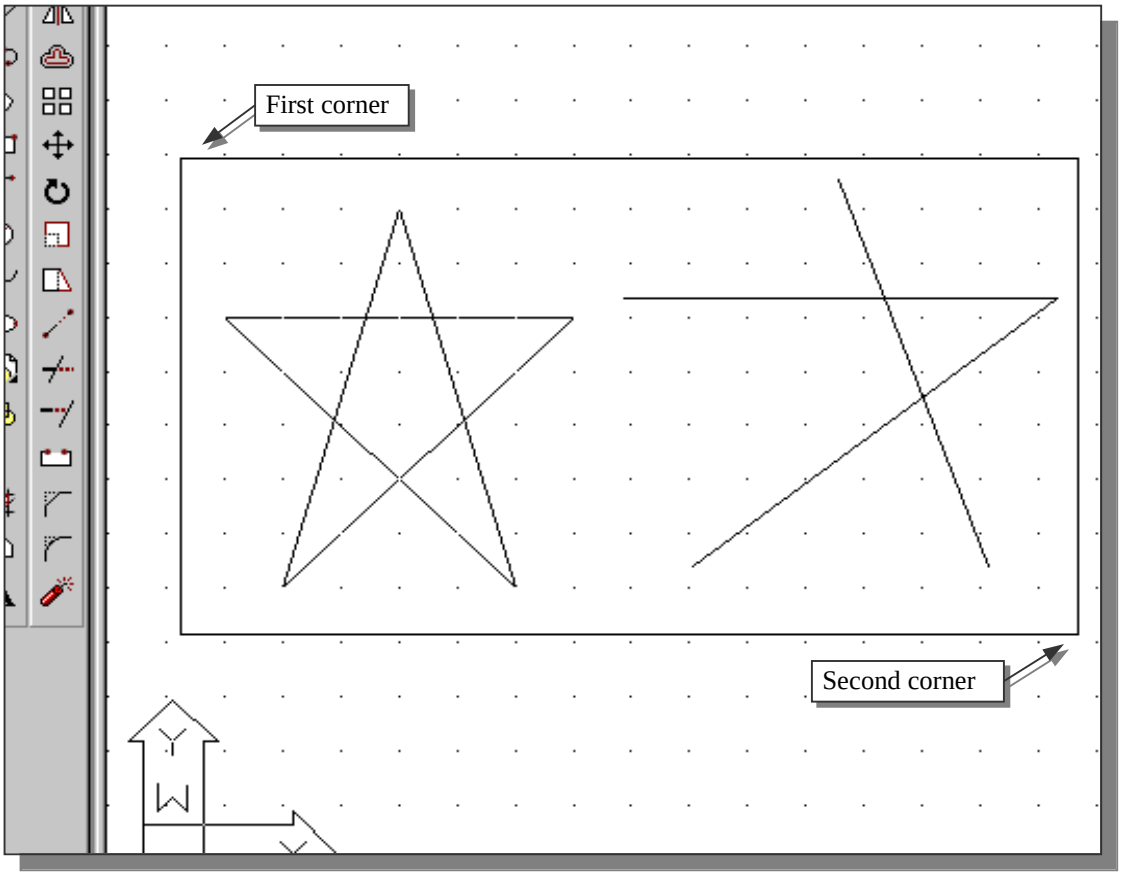


1. Inside the *Graphics window*, **click once with the right-mouse-button** to bring up the popup *option menu*.

2. Pick ***Repeat Erase***, with the left-mouse-button, in the *popup menu* to repeat the last command. Notice the other options available in the popup menu.

- AutoCAD LT 2000 offers many options to accomplish the same task. Throughout this text, we will emphasize the use of the new **AutoCAD Heads-up Design™** interface, which means focus on the screen, not on the keyboard.

3. Move the cursor to a location that is above and toward the left side of the entities on the screen. **Left-mouse-click** once to start a corner of a rubber-band window.



4. Move the cursor toward the right and below the entities, then **left-mouse-click** to enclose all the entities inside the **selection window**. Notice all entities that are inside the window are selected.

5. Inside the *Graphics window*, **right-mouse-click** to proceed with erasing the selected entities.

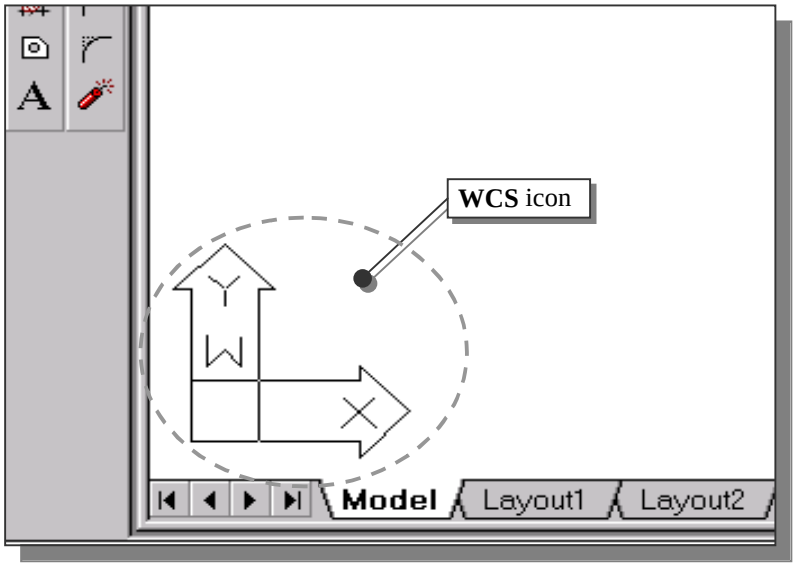
➢ On your own, create a sketch of your choice using the ***LINE*** command. Experiment with using the different commands we have discussed so far, such as switching the *GRID* and *SNAP* options on and off in the middle of a command.

❖ Do not be in a hurry to rush through the tutorials. Build up your CAD skills by familiarizing yourself with the commands and options demonstrated, along with the concepts discussed in the lessons. Feel free to repeat, at any time, any of the lessons.

The CAD Database

Designs and drawings created in a CAD system are usually defined and stored using sets of points in what is called **world space**. In most CAD systems, the world space is defined using the three-dimensional *Cartesian coordinate system*. Three mutually perpendicular axes, usually referred to as the X, Y, and Z axes, define this system. The intersection of the three coordinate axes forms a point called the **origin**. Any point in *world space* can then be defined as the distance from the origin in the X, Y and Z directions. In most CAD systems, the direction of the arrows shown on the axes identify the positive sides of the coordinates.

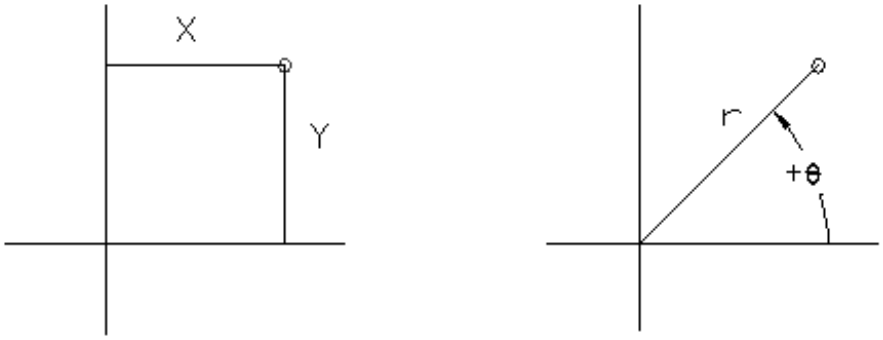
A CAD file, which is the electronic version of the design, contains data that describe the entities created in the CAD system. Information such as the coordinate values in world space for all endpoints, center points, etc., along with the descriptions of the types of entities are all stored in the file. Knowing that AutoCAD stores designs by keeping coordinate data helps us understand the inputs required to create entities.



The icon near the bottom left corner of the default AutoCAD *Graphics Window* shows the positive X-direction and positive Y-direction of the coordinate system that is active. The letter W appearing in the icon indicates that the *world coordinate system* (**WCS**) is active. The **world coordinate system** is a coordinate system used by AutoCAD as the basis for defining all objects and other coordinate systems. We can think of the **origin** of the **world coordinate system** as a fixed point being used as a reference for all measurements. The default orientation of the Z-axis can be considered as positive values in front of the monitor and negative values inside the monitor.

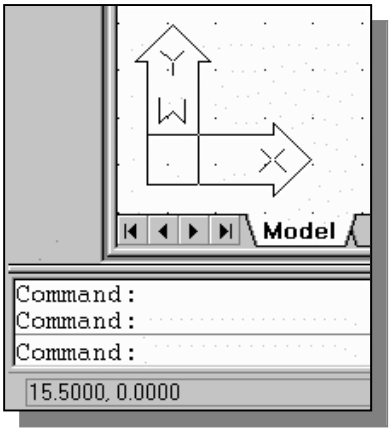
Cartesian and Polar Coordinate Systems

In a two-dimensional space, a point can be represented using different coordinate systems. The point can be located, using *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 θ 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 θ between the direction of r and the positive X axis. The default system for measuring angles in AutoCAD LT 2000 defines positive angular values as counter-clockwise from the positive X-axis.

Absolute and Relative Coordinates



AutoCAD LT 2000 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.

- 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 LT 2000, the *absolute coordinates* and the *Relative coordinates* can be used in conjunction to 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.

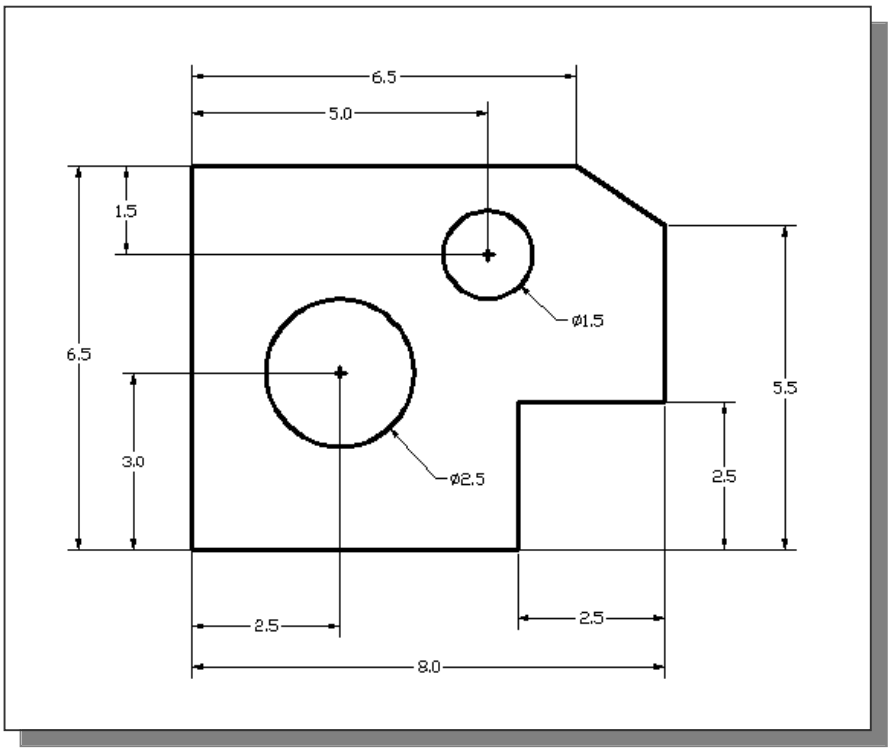
Defining Positions

In AutoCAD, there are five methods to specify 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.

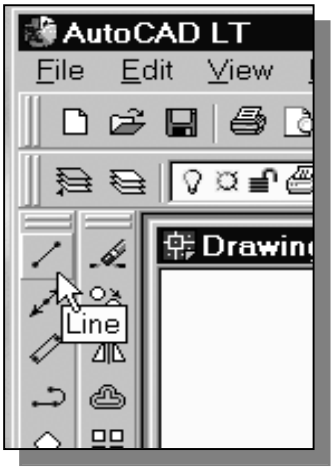
The Guide Plate

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 **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 systems. This allows the designers and CAD operators to concentrate their attentions on the more important issues – the design.



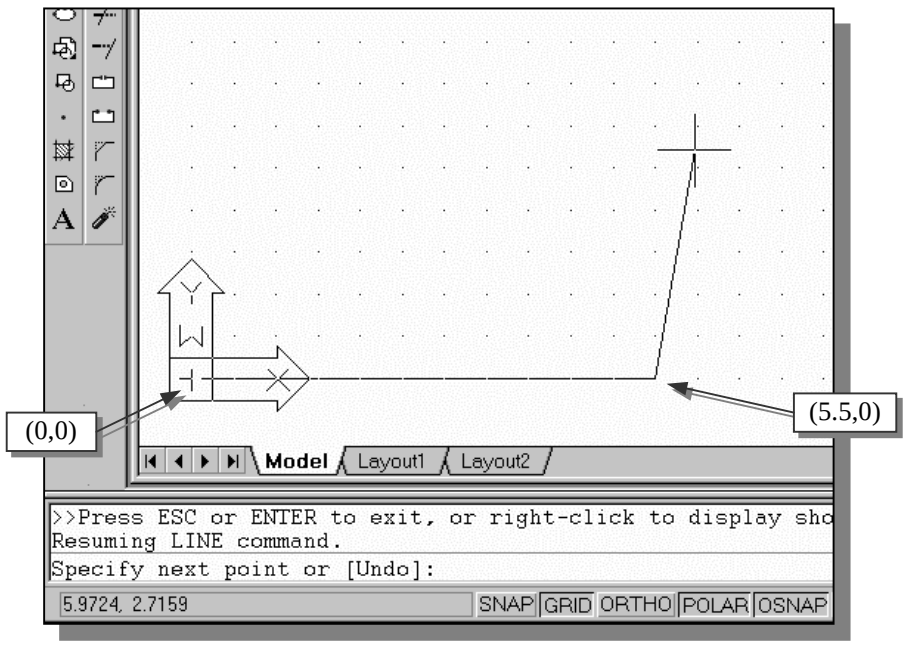
1. Select the ***LINE** command icon* in the *Draw toolbar*. In the *command prompt area*, near the bottom of the AutoCAD *Graphics window*, the message *"_line Specify first point:"* is displayed. AutoCAD expects us to identify the starting location of a straight line.

2. In the *command prompt area*, we will locate the starting point of our design at the origin of the *world coordinate system*.

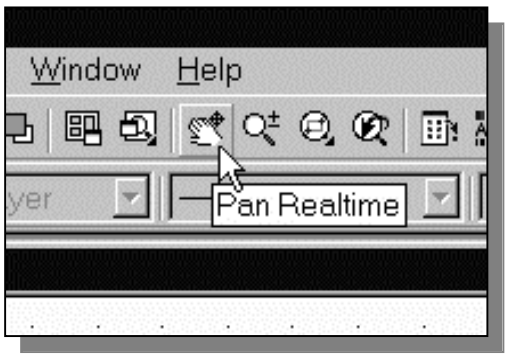
   Command: _line Specify first point: **0,0 [Enter]**
   (Type **0,0** in the *command prompt area* and press the [**Enter**] key once.)

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 viewing 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.

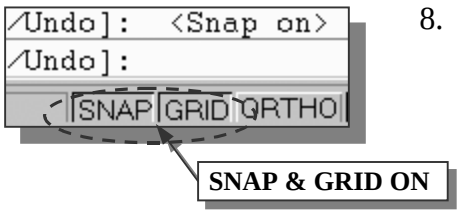
   The *PAN* command enables us to move the view to a different position. This function acts as if you are using a video camera.

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]**

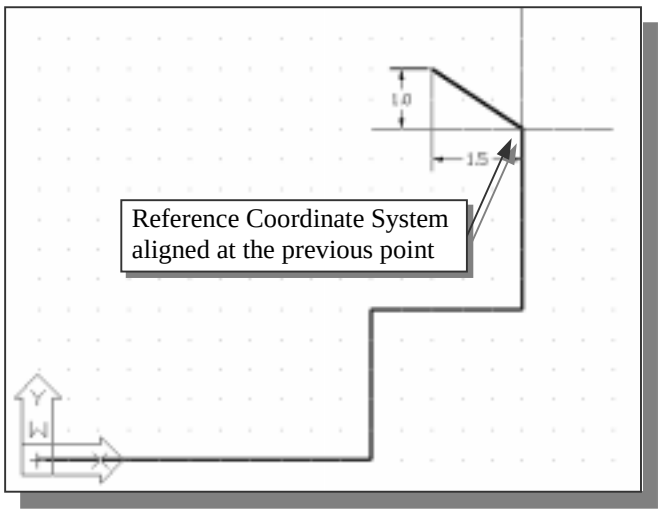


8. We can mix any of the entry methods in positioning the locations of the endpoints. Move the cursor to the *Status Bar* area, and turn on the *GRID* and *SNAP* options.

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 and an angle of **90** degrees)



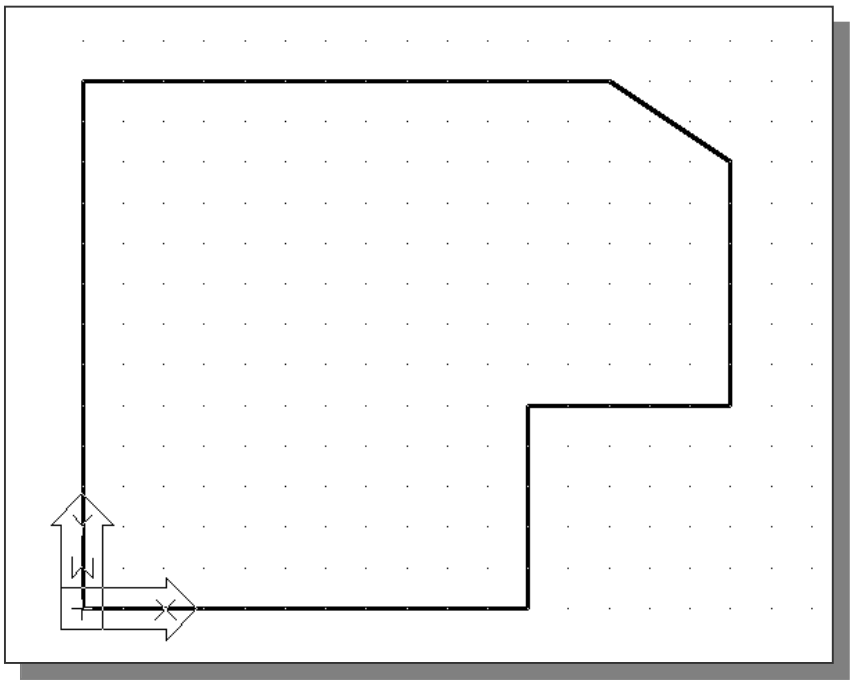
11. Using the *relative rectangular coordinates entry method* to create the next line, we can imagine a *reference coordinate system* aligned at the previous point. Coordinates are measured along the two reference axes.

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.

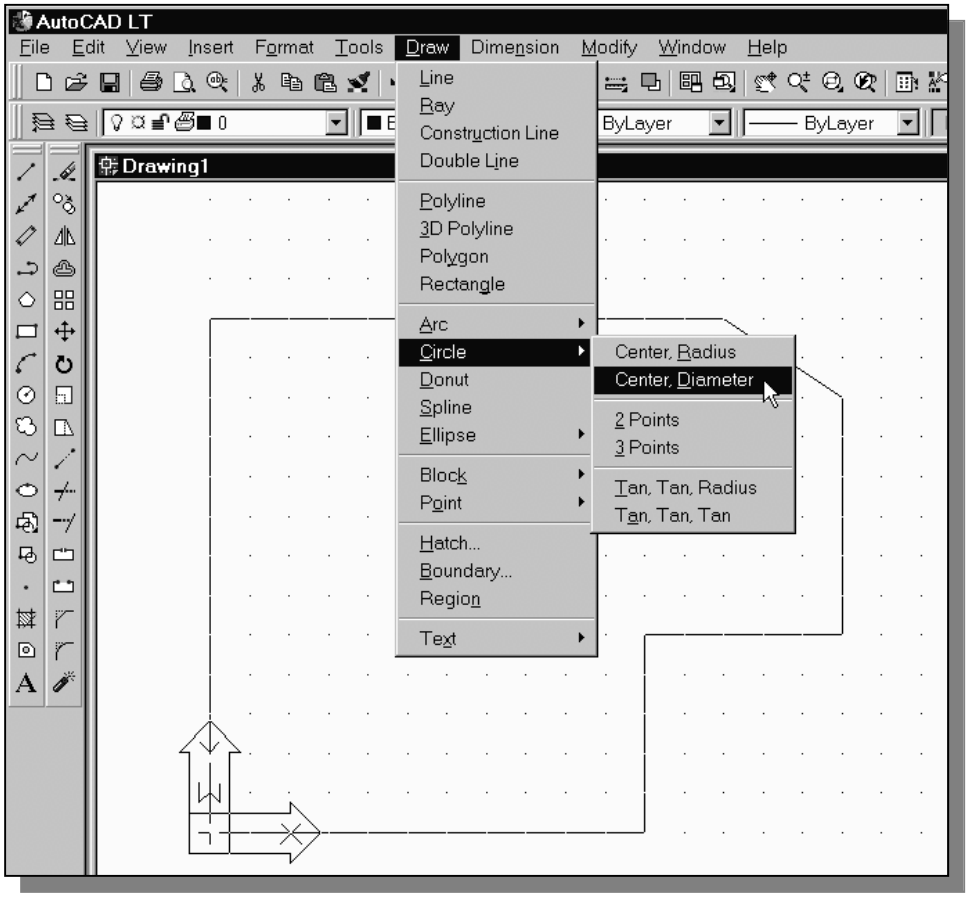


Creating *Circles*

The menus and toolbars in AutoCAD LT 2000 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:

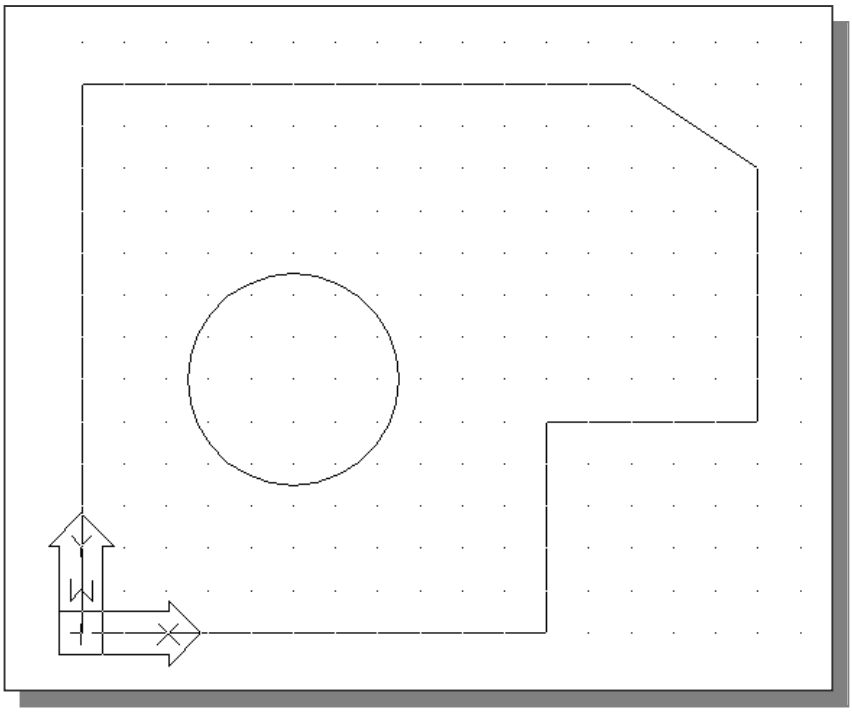
- **Center Point**: Draws a circle based on a center point and a diameter or a radius.
- **3 Points**: Draws a circle based on three points on the circumference.
- **2 Points**: Draws a circle based on two endpoints of the diameter.
- **TTR—Tangent, Tangent**, **Radius**: Draws a circle with a specified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

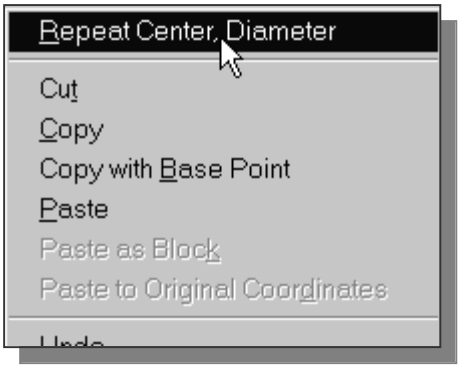
2. In the *command prompt area*, the message *"Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:"* is displayed. AutoCAD expects us to identify the location of a point or enter an option. We can use any of the four coordinate entry methods to identify the desired location. We will enter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

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

3. In the *command prompt area*, the message *"Specify diameter of circle:"* is displayed.

   Specify diameter of circle: **2.5 [Enter]**





4. Inside the *Graphics window*, **right-mouse-click** to bring up the *popup option menu*.

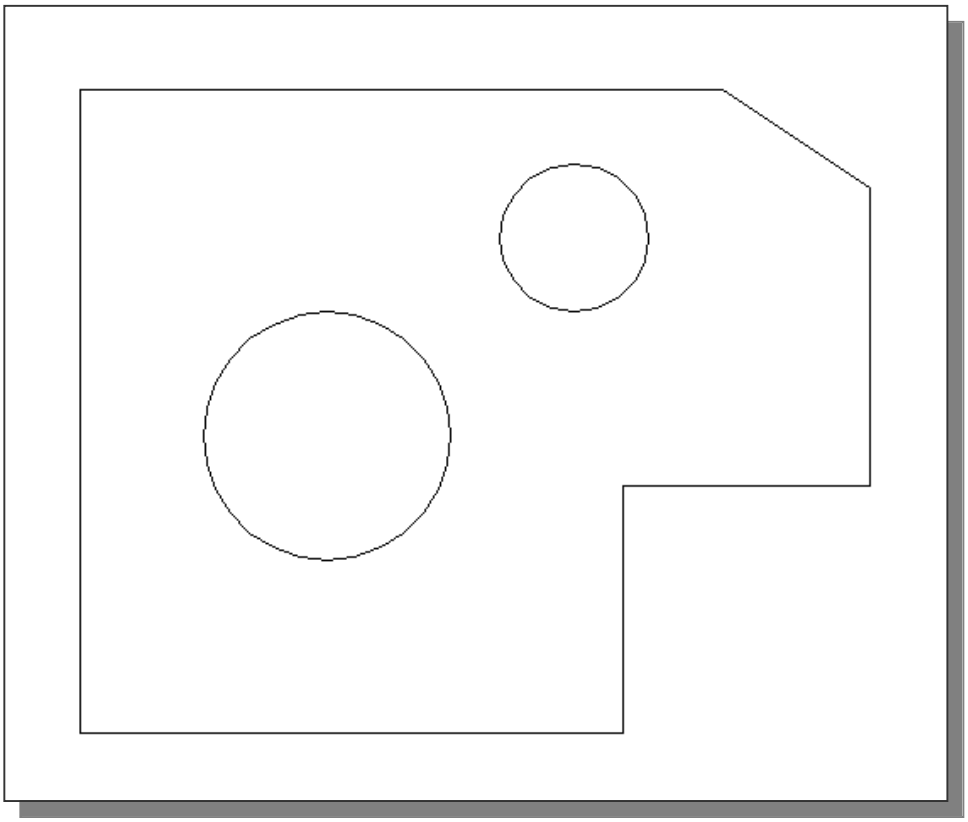
5. Pick **Repeat Center, Diameter** with the left-mouse-button in the *popup menu* to repeat the last command.

6. 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)***.

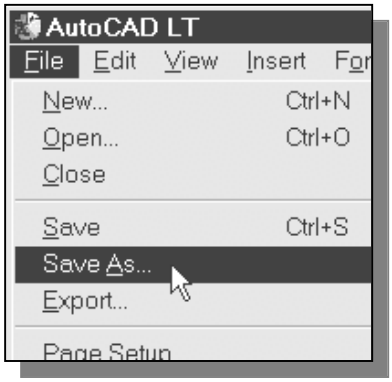
   Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **@2.5,2 [Enter]**

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]**



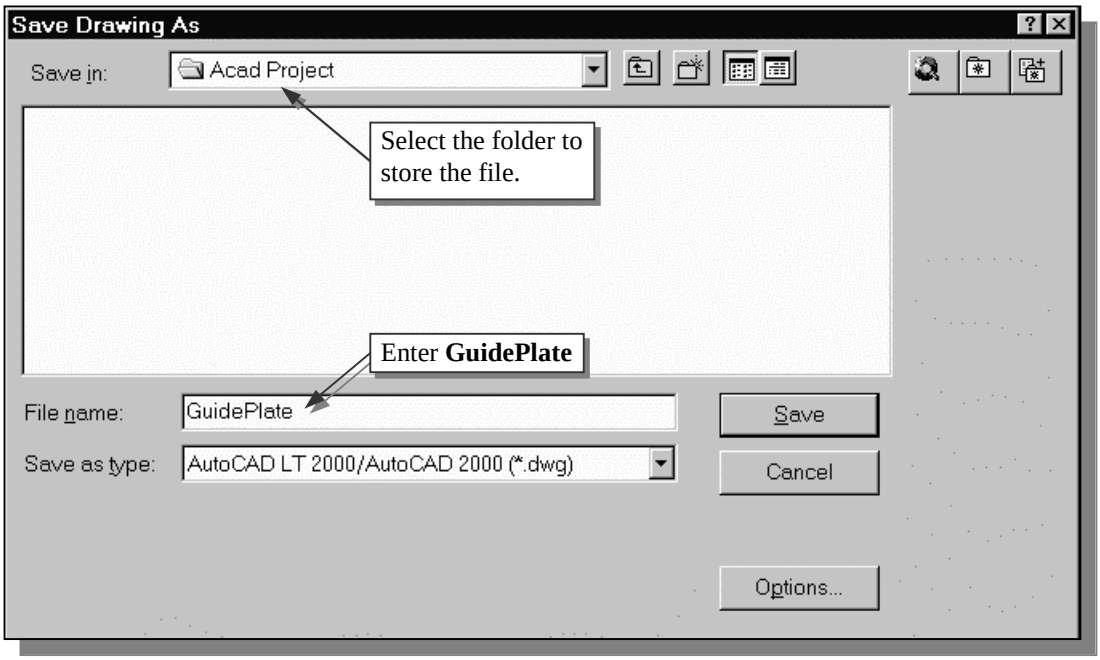
Saving the CAD file



1. In the *pull-down menus*, select:

   [File] >> [Save As]

2. In the *Save Drawing As* dialog box, select the folder in which you want to store the CAD file and enter **GuidePlate** in the *File name* box.



3. Pick **SAVE** in the *Save Drawing As* dialog box to accept the selections and save the file.

Exit AutoCAD LT

- To exit AutoCAD® LT 2000, select **File** then choose **Exit** from the pull-down menu or type **QUIT** at the command prompt.

Questions:

1. What is the first thing we should consider when starting a new model?

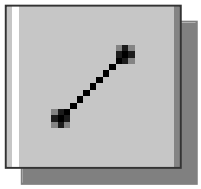
2. How do the *GRID* and *SNAP* options assist us in sketching?

3. List and describe the different **coordinate entry methods** available in AutoCAD LT?

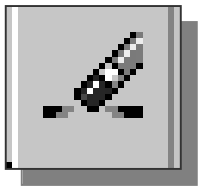
4. List and describe two types of coordinate systems commonly used for planar geometry.

5. Identify the following commands:

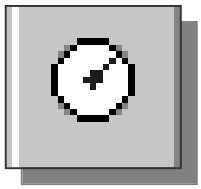
(a)



(b)



(c)

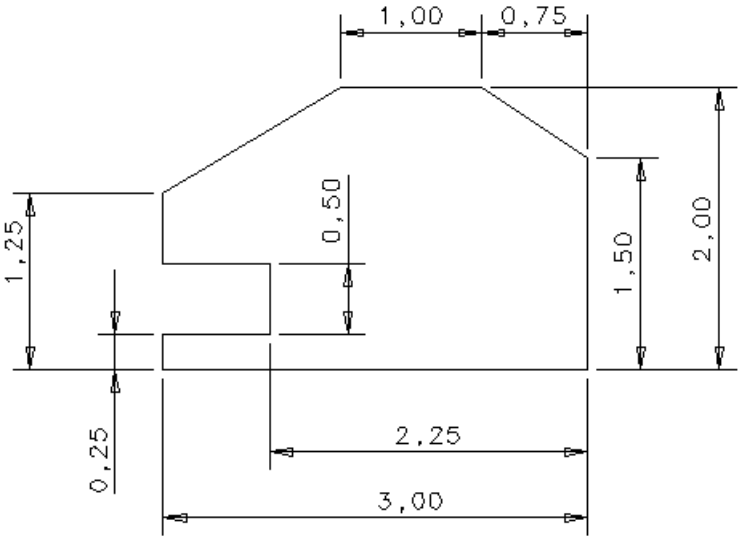


(d)

Tan, Tan, Radius

Exercises:

1.



2.