Residential Design

using

AutoCAD 2007
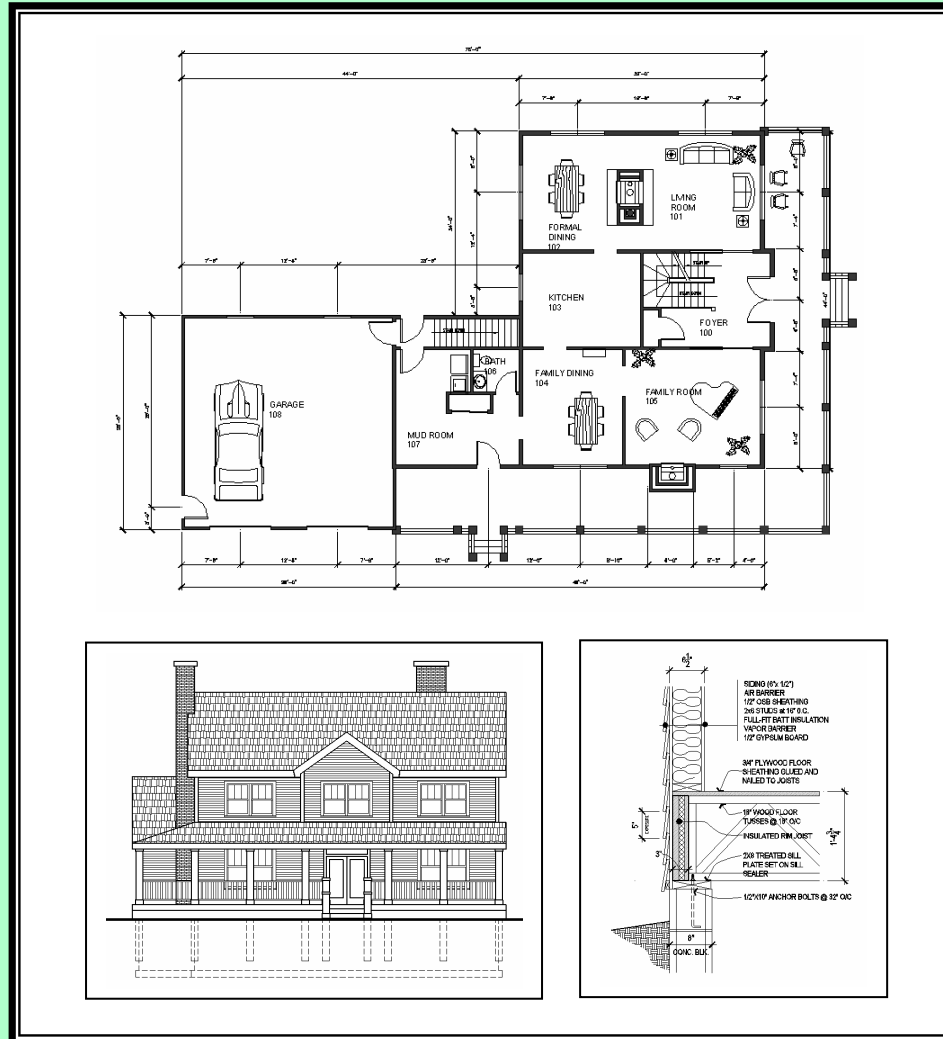
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Lesson 2
Crash Course Introduction (the Basics)

This lesson is meant to give you a brief introduction to a few of the most used commands. You will be walked through each drawing, in this lesson, step-by-step. The intent is for the user to get some of the basic concepts in mind before going through subsequent chapters.

Exercise 2-1:
Lines and Shapes

Drawing Lines:

The number one command in AutoCAD is the Line command. Lines in AutoCAD are extremely precise drawing elements. This means you can create very accurate drawings. Lines, or any drawn object, can be as precise as eight decimal places (i.e., 24.999999999) or 1/256.

AutoCAD is a vector based program. That means each drawn object is stored in a numerical database. For example, a line is stored in the drawing's database as the X, Y, Z coordinates for its starting point and the X,Y,Z, coordinates for its endpoint. These coordinates are all relative to the drawings 0, 0, 0 position; called the **Origin**.

When a line needs to be displayed on the screen, AutoCAD reads the line's coordinates from the drawing database and displays a line between those two points on the screen. This means that the line will be very accurate at any scale or zoom magnification.

A raster based program, in contrast to vector based, is comprised of dots that infill a grid. The grid can vary in density, and is typically referred to as resolution (e.g., 600x800, 1600x1200, etc.). This file type is used by graphics programs that typically deal with photographs; such as Adobe Photoshop. There are two reasons this type of file is not appropriate for CAD programs:

1. A raster based line is composed of many dots on a grid (which also represents the line's width). When you zoom in (or magnify) the line, it becomes pixilated and you actually start to see each dot in the grid. In a vector file you can "infinitely" zoom in on a line and it will never become pixilated because the program recalculates the line each time you zoom in.

2. A CAD program, such as AutoCAD, only needs to store the starting point and end point coordinates for each line; the dots needed to draw the line are calculated on the fly for the current screen resolution. Whereas a raster file has to store each dot that represents the full length and width of the line. This can vary from a few hundred dots to several thousand dots, depending on the resolution, for the same line.

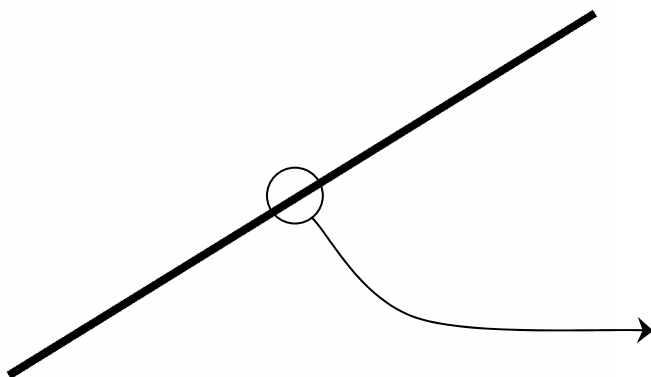The following graphic illustrates this point:

Vector vs. Raster lines

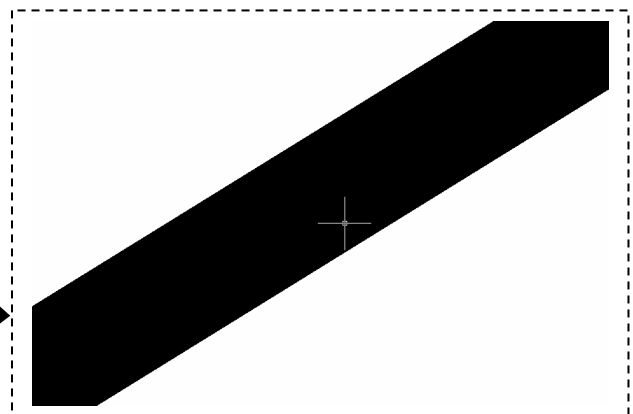**Figure 2-1.1** Vector Line Example
File Size: approx. 33kb

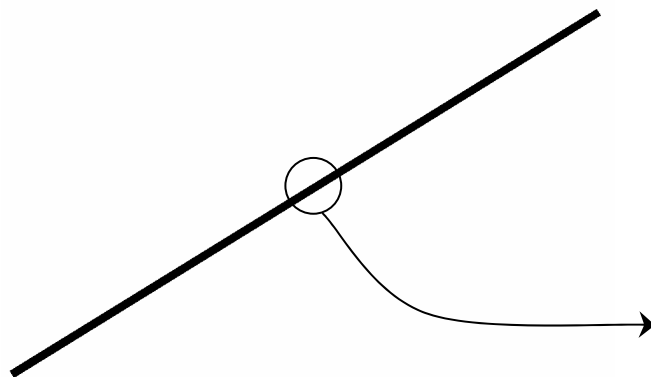**Figure 2-1.1a** Vector Line Enlarged 1600%

Figure 2-1.2 Raster Line Example
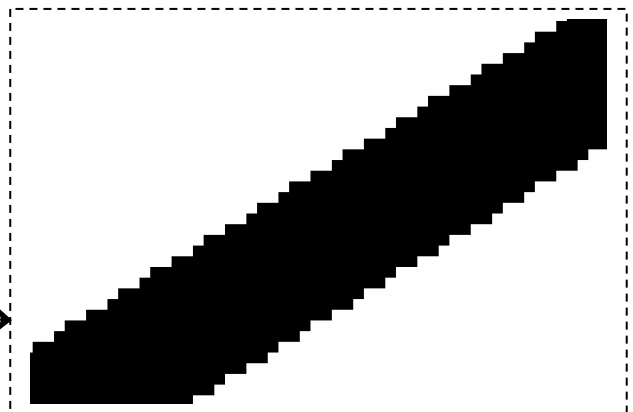*File Size: approx. 4.4MB*

Figure 2-1.2a Raster Line Enlarged 1600%

The Line Command:

You will now study the Line command.

1.   **Open** AutoCAD.

2.   **Close** the empty drawing automatically opened by AutoCAD.

Warning:

The empty startup drawing that is opened by default when you start AutoCAD, does not have the various variables preset for architectural CAD use.

So you should make sure you start with the correct template file whenever instructed to start a new drawing file in this book.

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3.   Select **New** from the *File* menu.

4.   Use the following Template file to start your new drawing from: **Architectural Imperial.dwt** *(in the SheetSets sub-folder).*

5.   Click on the **Model Tab** to switch to *Model Space* (which is where all the drawing is done as previously mentioned).
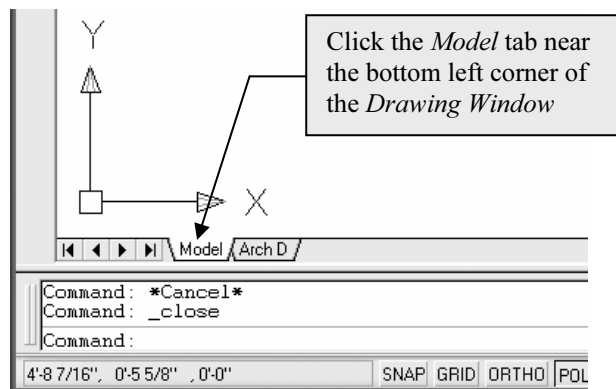
6.   Select the **Line** command from the *Draw Toolbar.*

7.   **Draw a line** from the lower left corner of the screen to the upper right corner of the screen (by simply clicking two points on the screen)

     NOTE: Do not drag/hold your mouse button down; just click. (Figure 2-1.5)

After clicking your second point, you should notice by looking at the *Command Window*, that the *Line* command is still active. You could continue to draw lines; with the last point picked becoming the first point for the next line segment. In this case you will end the line command per the following instructions.

8. After clicking your second point (i.e., the endpoint), **right-click** anywhere on the screen and select **Enter** from the pop-up menu. (Figure 2-1.6)
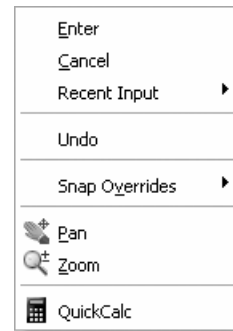
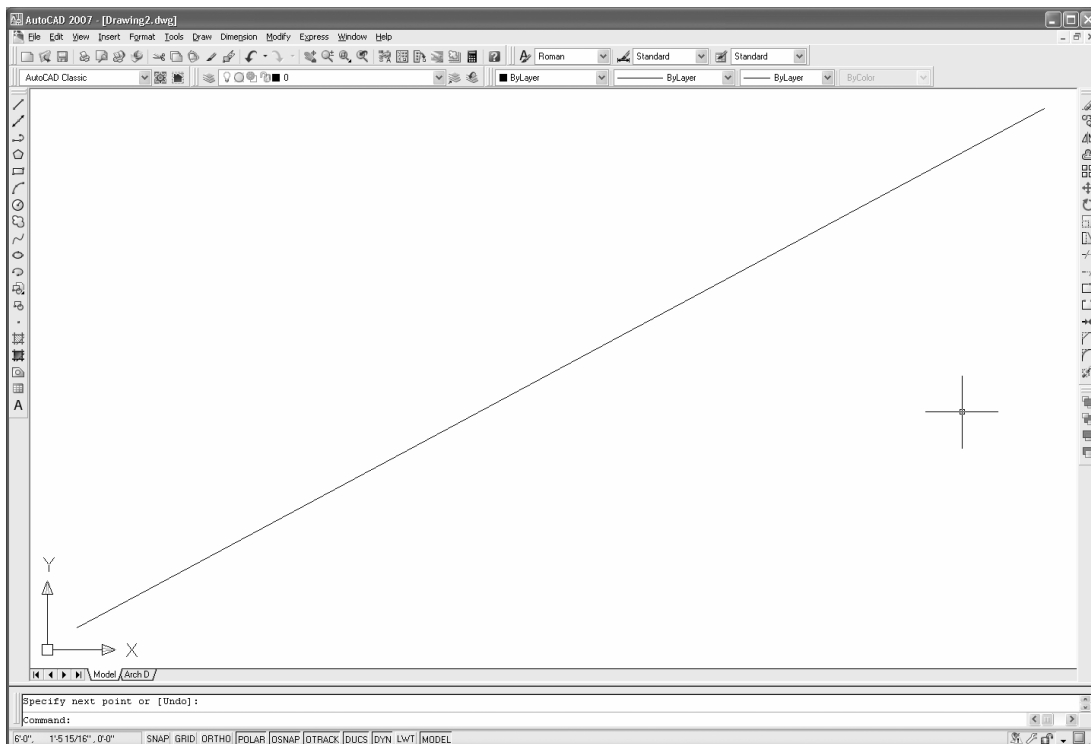**Figure 2-1.6** Right-click Menu; line command active

Figure 2-1.5 Your first line!

This constitutes your first line! However as you are probably aware, it is not a very accurate line as far as length and angle are concerned.

The Properties Palette:

Typically you need to provide information such as length and angle when drawing a line; you rarely pick arbitrary points on the screen like you did in the previous steps. Most of the time you also need to accurately pick your starting point. (For example: how far one line is from another or picking the exact middle point on another line.)

Did you make a mistake?

Whenever you make a mistake in AutoCAD you can use the **UNDO** command to revert to a previous drawing state. You can perform multiple UNDOs all the way to your previous *Save* (and then some).

Similarly, if you press *Undo* a few too many times, you can use **REDO**.

Having said that, however, the line you just drew still has precise numbers associated with its length and angle. In the next step you will use the *Properties Palette* to view the line's length and angle.

9.   Select the **Properties Icon** from the *Standard Toolbar* (or press Ctrl + 1).

You now have the *Properties Palette* open on your screen. Notice that the text-box at the top says "No selection" - this is because no objects are selected yet. (Figure 2-1.7)

Tip:

If the Properties Palette is too narrow on the screen, some information may not be completely visible. You can easily re-size any Palette by dragging one of the corners (opposite the palette title bar).

Now that the *Properties Palette* is open, you can select the line to display its properties.

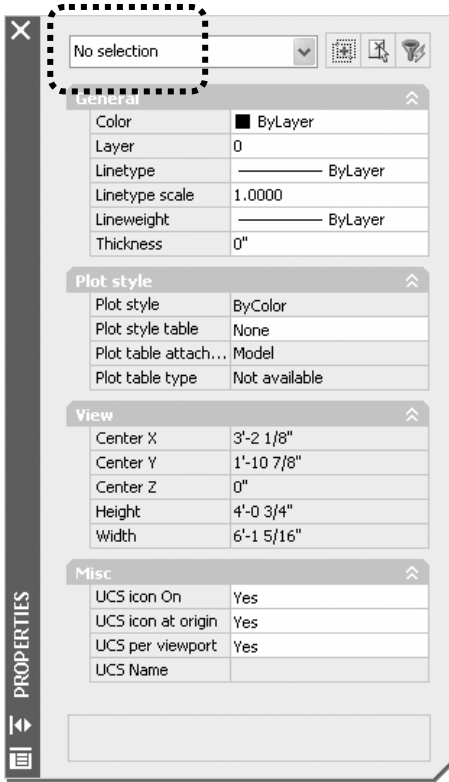10.   **Select the Line** - hover the cursor over the line & click on it.

No selection		
General		
Color	■ ByLayer	
Layer	0	
Linetype	——— ByLayer	
Linetype scale	1.0000	
Lineweight	——— ByLayer	
Thickness	0"	
Plot style		
Plot style	ByColor	
Plot style table	None	
Plot table attach...	Model	
Plot table type	Not available	
View		
Center X	3'-2 1/8"	
Center Y	1'-10 7/8"	
Center Z	0"	
Height	4'-0 3/4"	
Width	6'-1 5/16"	
Misc		
UCS icon On	Yes	
UCS icon at origin	Yes	
UCS per viewport	Yes	
UCS Name		

Figure 2-1.7 Properties Palette –
No objects selected

> **FYI:**
>
> Notice the line highlights before you select it. This helps to select the correct line the first time. Objects highlighting when the cursor is hovering over them is a feature new to AutoCAD 2007.

You should now see the line's properties. Notice the text-box at the top now indicates what type of object is selected. (Figure 2-1.8)

> **Are your units and drawing window size correct?**
>
> If the properties on your computer are displaying the line's data in decimals instead of feet and inches (as shown in Figure 2-1.8), that means you did not start using the correct template file.
>
> Similarly, if you're drawing window is not approx. 5'-10" x 3'-9" then you either started with the wrong template or your template file has been altered; if the later is the case, which is less likely, zoom in (or out) to correct the screen display.
>
> At this point you should close the drawing and start this lesson over.
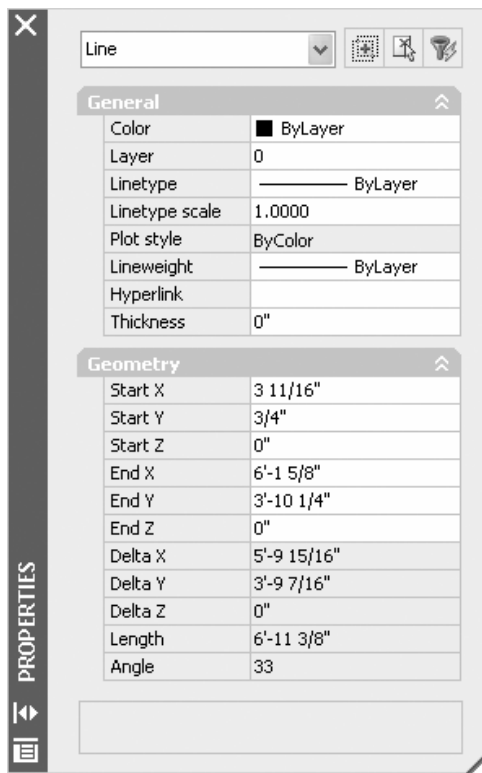
Notice the information displayed for the selected line:

- o The X, Y and Z coordinates are listed, relative to the drawings origin (0,0,0), for both the start and end points.
- o The line's Length and Angle are listed.
- o Delta X and Delta Y; the Delta X is the horizontal distance between the start point and the end point.

The white areas are editable and the grey areas are not. You cannot edit the length, for example, because AutoCAD needs more info, like what direction to make the line longer.

Figure 2-1.8
Properties Palette – Line selected

You should make one final observation about the line's properties and the current view: the Delta X and Delta Y represent the horizontal and vertical distances between the line's start and end points. This tells us that the drawing area, currently visible on the screen, is about 5'-10" across (horizontally) and 3'-9" up and down (vertically). That means that the largest item we can draw (and still see what we are doing) is something that is 5'-10" wide x 3'-9" tall. (See Figure 2-1.9.)

> **RECAP:** At any given time you could draw a diagonal line, from one corner of the drawing window to the other, list that lines Properties, observe the Delta X and Delta Y values and infer, from that information, the current drawing area that you are zoomed into.
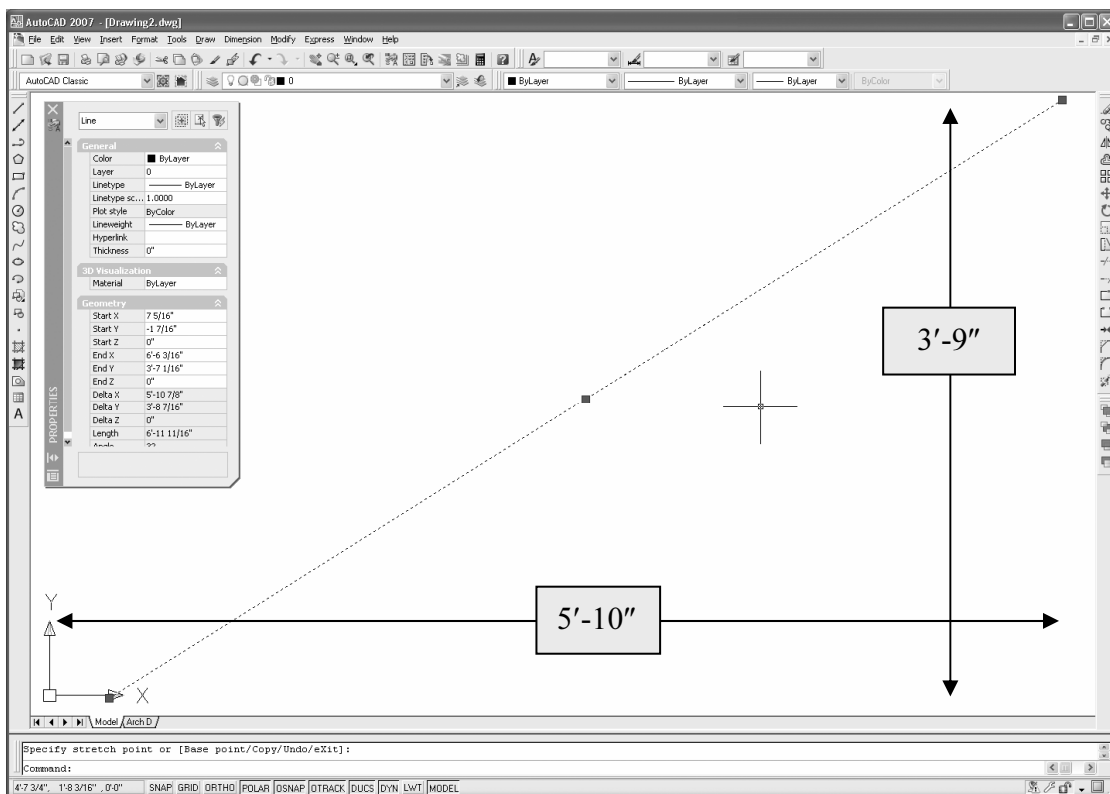

Figure 2-1.9 Visible area of current drawing

Everything is Drawn Full-Scale:

Even though your current view is showing a relatively small area, *Model Space* (where you do all your drafting and design) is actually an infinite drawing board.

In architectural CAD drafting, everything is drawn "real-world" size (or full-scale) ALWAYS! If you are drawing a building, you might draw a line for the exterior wall that is 600'-0" long. If you are drawing a window detail, you might draw a line that is 1/8" long. In either case you are entering that line's actual length.

You could, of course, have a 1/8" line and a 600'-0" line in the same drawing. Either line would be difficult to see at the current drawing magnification (i.e., 3'-2"x2'-11" area). So you would have to zoom in to see the 1/8" line and zoom out to see the 600'-0" line. You will try this next:

Draw a 1/8" Line:

11. For now, **Close** the *Properties Palette* by clicking the **X** in the upper corner of the palette

The next steps will walk you through drawing a 1/8" horizontal line. AutoCAD provides more than one way to do this. You will try one of them now.

You should understand that it is virtually impossible to draw a perfectly horizontal or vertical (or any precise angle) line by visually picking points on the screen. You need AutoCAD's help!

12. On the *Status Bar*, make sure **Polar Tracking** is turned on. (Figure 2-1.10). *It should look like a button pushed in (i.e., ON).*
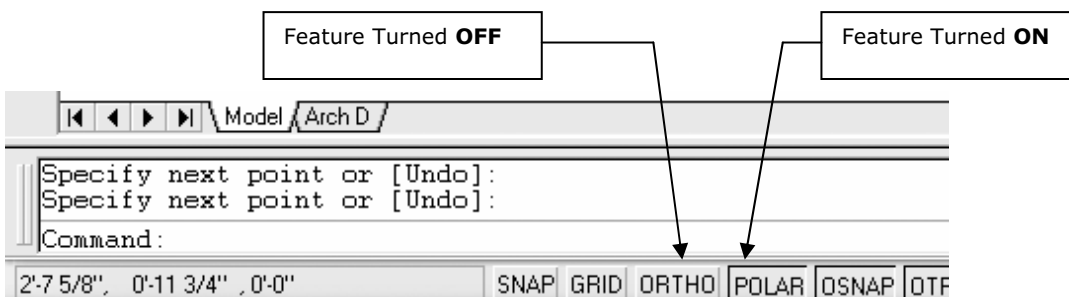

Figure 2-1.10   Status Bar – shown with Polar (i.e., Polar Tracking) turned on
TIP: You can adjust the Polar settings by right-clicking on the button.

Polar Tracking:

This drawing aid feature allows you to draw lines at precise angles. When you are in the process of drawing a line, *Polar Tracking* will "snap" to certain angles in the *Drawing Window* whenever your cursor comes close to that angle.

A dashed line will appear on the screen giving you a visual tip that the line is "snapped" to one of the predefined angles set within the Polar Tracking settings. In addition to the dashed line, the tool-tip (near your cursor) displays the word Polar and the locked angle.

Clicking the mouse or entering a length, while the dashed line is visible, ensures the line's proper angle.

13. Using the **Line** Command, pick a point somewhere in the upper left corner of the *Drawing Window.*

14. Start moving the mouse towards the right, and generally horizontal until you see a dashed reference line (extending off the screen in the direction you want to draw the line) appear on the screen as in Figure 2-1.8.
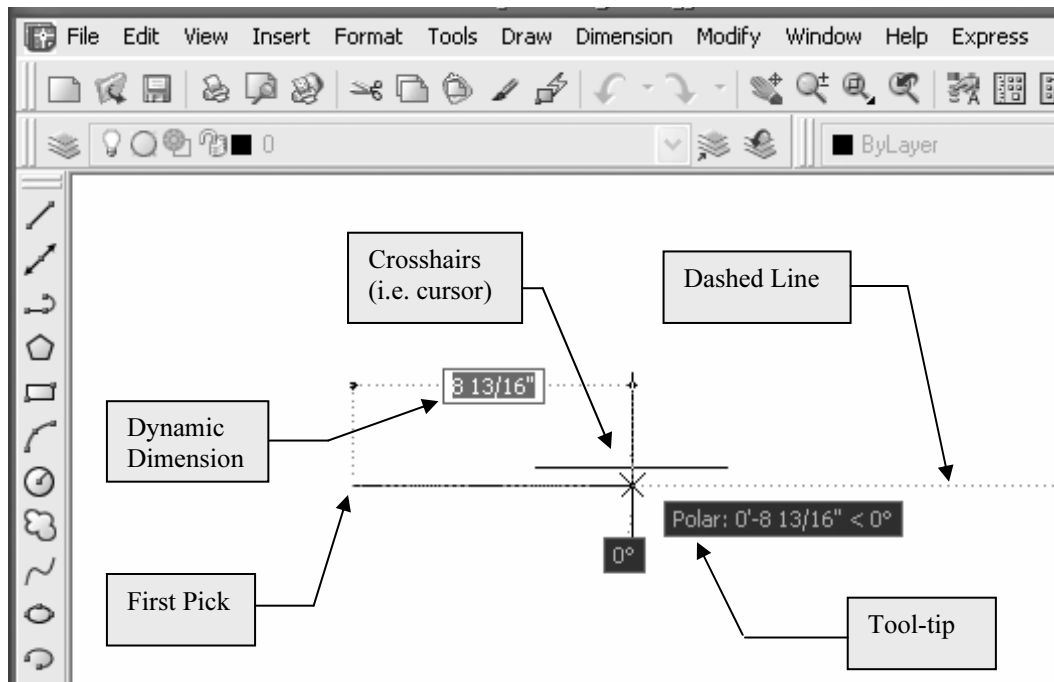

Figure 2-1.11   Drawing a line with the help of Polar Tracking
               TIP: Polar Tracking is active when you see the dashed line extending off the screen

15. With the dashed line and *Tool-tip* visible on the screen, take your hand off the mouse (so you don't accidentally move it), type **1/8** and then press **Enter**.

   TIP: Notice you did not have to type the inch symbol; AutoCAD always assumes you mean inches unless you specify otherwise. A future lesson will review the input options in a little more detail.

You have just drawn a line with a precise length and angle!

16. Use the **Properties Palette** to verify it was drawn correctly (similar to steps 9 and 10).

17. Use the **Zoom Window** tool to enlarge the view of the 1/8" line.

18. Now use the **Zoom Extents** or **Zoom Previous** tools so that both lines are visible again.

Draw a 600' line:

19. Select the **Line** icon and pick a point in the lower right corner of the *Drawing Window.*

20. Move the crosshairs (i.e., mouse) straight up from your first point so that *Polar Tracking* activates. (Figure 2-1.12)

21. With the dashed line and tool-tip visible on the screen, take your hand off the mouse (so you don't accidentally move it), and type **600'** and then press **Enter**.
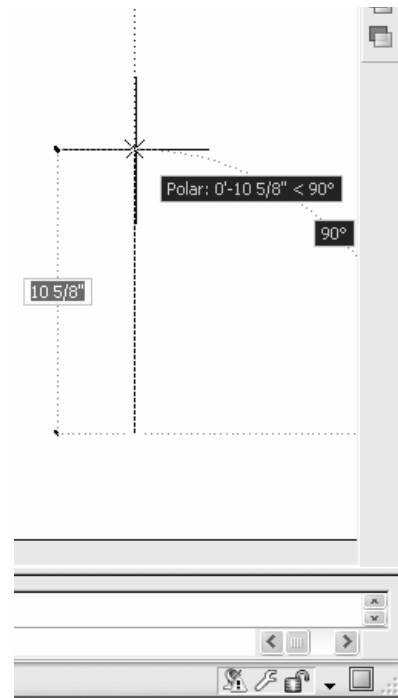
Figure 2-1.12   Drawing another line with the help of Polar Tracking

   TIP: Notice this time you had to type the foot symbol (´) or AutoCAD would have assumed you meant 600 inches, which is only 50'.
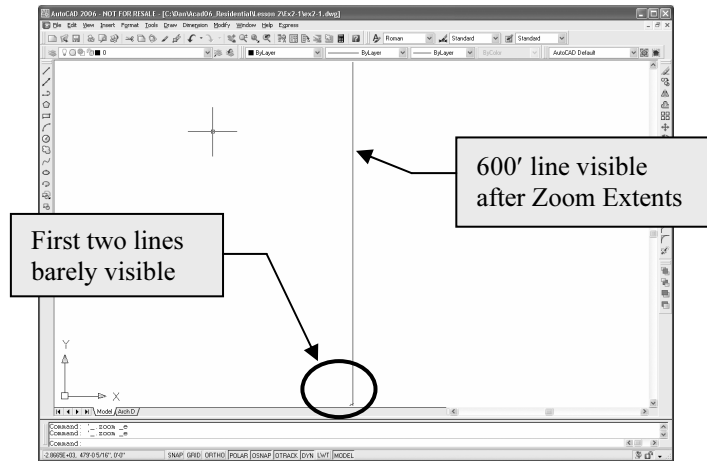
Figure 2-1.13   Drawing with three lines

22. End the Line command *(right-click and select Enter)*.

Because the visible portion of the drawing area is only 3'-9" tall (Figure 2-1.9), you obviously would not expect to see a 600' line. You need the change the drawings magnification (i.e., zoom out) to see it.

23. Use **Zoom Extents** to see the entire drawing.
     (Figure 2-1.13)

Drawing Shapes:

AutoCAD gives you the commands to draw common shapes like square/rectangles, circles, ellipses, polygons. These commands can be found on the *Draw Toolbar*. You will take a look at *Rectangle* and *Circle* now.

> **MOVING TOOLBARS**: You only see a toolbar's title (like the one above) if it is floating; i.e. not docked. You can move a floating toolbar by dragging it by the title bar, and a docked toolbar can only be moved by dragging the two bars at the left (or the top for vertical toolbars). Remember, toolbars cannot be moved when the padlock in the lower right corner of the AutoCAD window is in the locked position; click the padlock to toggle between locked and unlocked.

Drawing a Rectangle:

24. Use **Zoom Previous** (or Zoom Window) to get back to the original view.

25. Select the **Rectangle** tool from the *Draw Toolbar.*

    Notice the *dynamic tool-tip* prompt near your cursor:

    Specify first corner point or [1'-2 1/8"] [2'-10 7/16"]
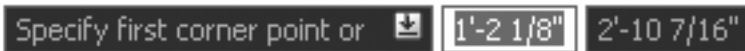
    TIP: Pay attention to these prompts while you are learning AutoCAD; they tell you what information is expected/requested. Similar prompts are also displayed in the *Command Window*.

26. Pick your "**first corner point**" somewhere near the middle-bottom of the *Drawing Window*. (Figure 2-1.14)

    Notice the *dynamic tool-tip* prompt near your cursor:

    Specify other corner point or [2 11/16"] [6"]
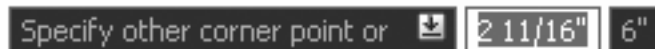
At this point you can pick a point on the screen, type a X,Y,Z coordinate, or enter the dimensions on-screen via the dynamic tool-tip feature near you cursor; you will use the latter method as it is easiest.

27. While being prompted for the "other corner", simply start typing (AutoCAD defaults to the *Width* tool-tip for dynamic input). Type **6.5** (this equals 6 1/2") and then press the **Tab** key on the keyboard - do not press Enter.

Pressing *Tab* toggles you over to the *Height* tool-tip for dynamic input (see prompt image below).

Notice the Width just entered has a padlock symbol next to it, which means that value is locked in. So, even though you will enter a numeric value for the Height, you could pick a point on the screen and no mater where you picked the Width would be 6.5".

> **Tip:**
> When drawing a rectangle, typing negative values allows you to dictate which direction the rectangle is drawn (relative to your first pick point). When both numbers are positive, as in the previous example, the rectangle is drawn up and to the right of your first pick point.

Notice the current on-screen prompt:

Specify other corner point or   6.5   7 9/16"

28. Type **14** and then press **Enter** (this equals 1'-2").
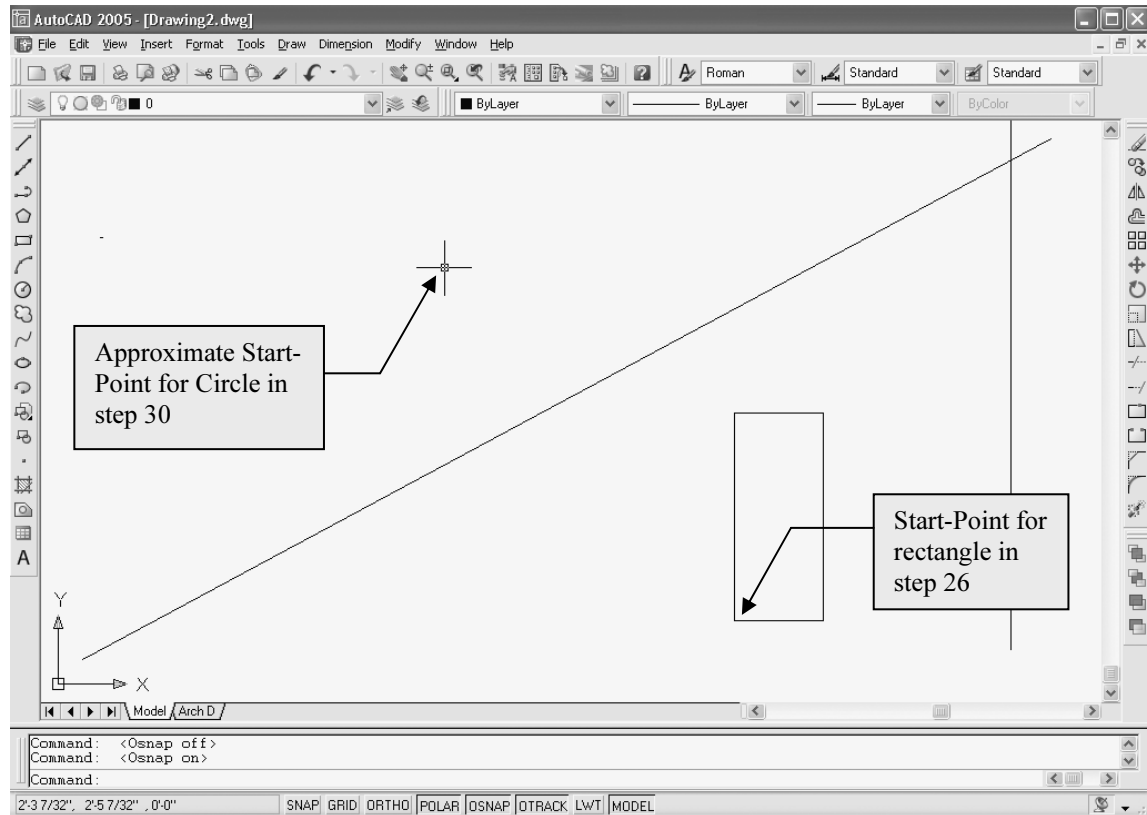

Figure 2-1.14   Drawing with Rectangle added

Drawing a Circle:

29. Select the **Circle** tool from the *Draw Toolbar.*

30. You are now prompted to pick the center point for the circle;
     pick a point approximately as shown in Figure 2-1.14.

     You should now see the following on-screen prompt:

Specify center point for circle or   1'-0 3/16"   2'-9 5/16"

2-13

Tool-tip menus:

Whenever you see a down-arrow icon within the on-screen tool-tip (like the one above), you can press the down-arrow key on the keyboard a list of options pop up below the tool-tip (see example to right). The options related to the active command (the active command being the circle command in this example).
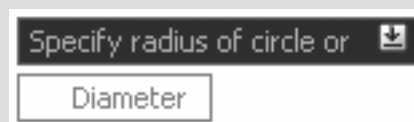
Right-click menus:

Another option is to right-click the mouse to see the same options as above plus a few standard options; this is called the context menu. If you clicked Diameter, the context menu goes away and AutoCAD expects you to enter the diameter rather than the radius.
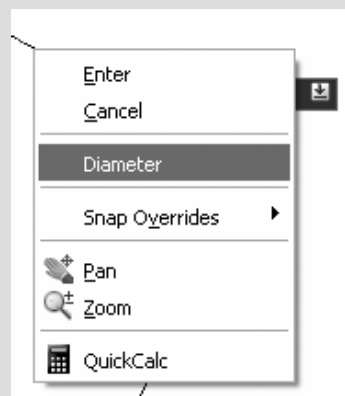
Move your mouse around to see that you could arbitrarily pick a point on the screen to create a quick circle if needed, then proceed to the next step where you will draw a circle with a radius of 6 5/8".

31. Type **6-5/8** and then press **Enter** (this equals 6 5/8"). (Figure 2-1.15)

32. View the Properties for both the Circle and Rectangle.

    NOTE: See Figure 2-1.16 for the Circle's Properties; you can actually modify the Circle by entering a new Circumference or Area.
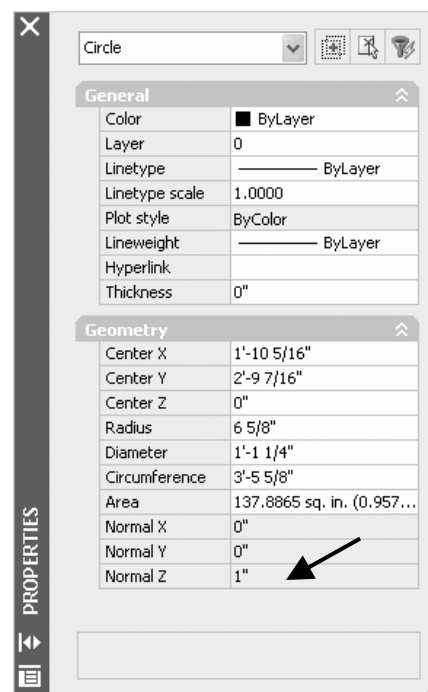
33. **Save** your drawing as **ex2-1.dwg**.

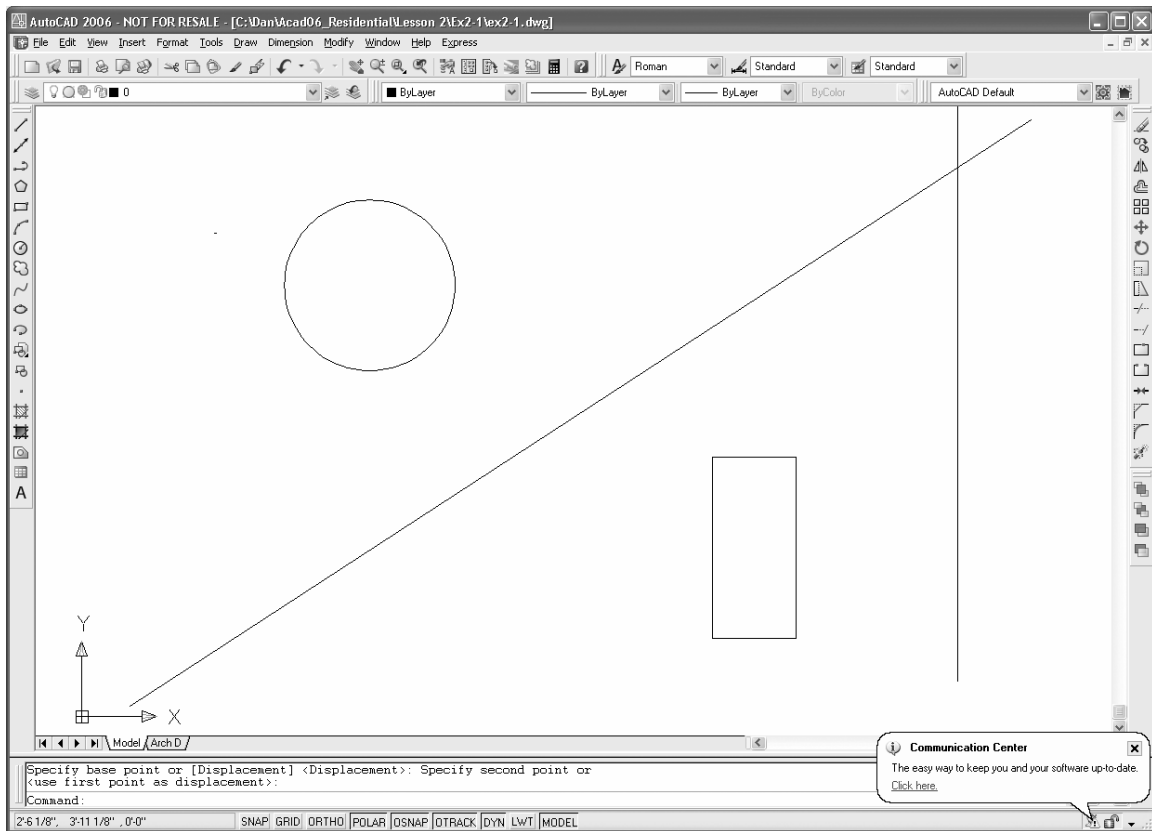**Figure 2-1.16**   Properties for circle

Figure 2-1.15   Drawing with Circle added

Where should I save my files?

You should create a folder to save these files into; typically a sub-folder in the My Documents folder. Your instructor may specify a location for your files.

Exercise 2-2:
Object Snaps

Object Snap is a tool that allows you to accurately pick a point on an object. For example, when drawing a line you can use Object Snap (osnap) to select, as the start-point, the endpoint or midpoint of another line.

This feature is absolutely critical to drawing accurate technical drawings. Using this feature allows you to be confident you are creating perfect intersections, corners, etc. (Figure 2-2.1)

Object Snaps Options:

You can use *Object Snaps* in one of two ways:

Figure 2-2.1 Typical problem when Object Snap is not used

o Using the *Running OSNAP* mode
o On an individual pick-point basis

Running *Osnap* mode is a feature that, when turned on, constantly scans the area near your crosshairs when AutoCAD is looking for user input. You can configure which types of points to look for.

Using an *Object Snap* for individual pick-points allows you to quickly select a particular point on an object. This option will also override the Running *Osnap* setting, which means you tell AutoCAD to just look for the endpoint on an object (for the next pick-point only) rather than the three or four types being scanned for by the Running *Osnap* feature.

Enabling Running Object Snaps:

To toggle *Running Object Snap* on and off you click the **OSNAP** button on the *Status Bar*.
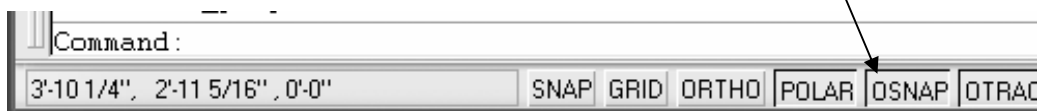
OSNAP turned **ON**

Command:

3'-10 1/4", 2'-11 5/16" , 0'-0"   SNAP GRID ORTHO POLAR OSNAP OTRAC

**Figure 2-2.2** Status Bar – shown with OSNAP (Object Snap) turned on

Snap Symbols:

By now you should be well aware of the Object Snap Symbols that AutoCAD displays as you move your cursor about the drawing window (while you are in a command like Line and AutoCAD is awaiting your input or pick-point).

If you hold your cursor still for a moment, while a snap symbol is displayed, a tool tip will appear on the screen. However, when you become familiar with the snap symbols you can pick sooner (rather than waiting for the tool tip to display). (Figure 2-2.3)

Symbol	Position	Keyboard Shortcut
✕	Intersection	INT
☐	Endpoint	END
△	Midpoint	MID
◇	Quadrant	QUA
○	Center	CEN
⊠	Nearest	NEA
⌐	Perpendicular	PER
○	Tangent	TAN

Figure 2-2.3 OSNAP symbols that are displayed on the screen when selecting a point.
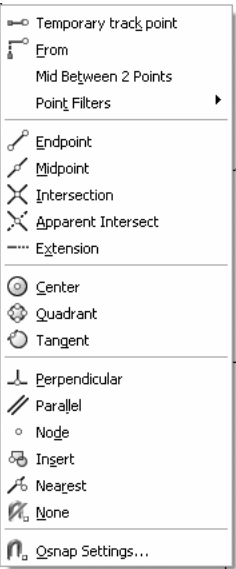
The TAB key cycles through the available snaps near your cursor.

The keyboard shortcut turns off the other snaps for one pick. For example, if you type END on the keyboard while in the line command, AutoCAD will only look for an Endpoint for the next pick.

Finally, if you need a particular Snap for just one pick, you can hold the Shift key and right-click for the Osnap context menu (see image to left). If you pick Center, AutoCAD will only look for a Center to Snap to for the next pick and then revert back to the previous settings.

Setting Object Snaps:

You can set AutoCAD to have just one or all Object Snaps running at the same time. Let's say you have Endpoint and Midpoint set to be running. While using the Line command, move your cursor near an

existing line. When the cursor is near the end of the line, you will see the Endpoint symbol show up. When you move the cursor towards the middle of the line, you will see the Midpoint symbol show up.

The next step shows you how to tell AutoCAD which Object Snaps you want it to look for.

First you need to **Open** the drawing from the previous Lesson.

1. **Open** drawing **ex2-1.dwg** (if not currently open).

2. Next, do a **Save-As** and name the drawing **ex2-2.dwg**.

3. Right-click on the *OSNAP* button on the *Status Bar* and select **Settings** from the pop-up menu. (Figure 2-2.4)
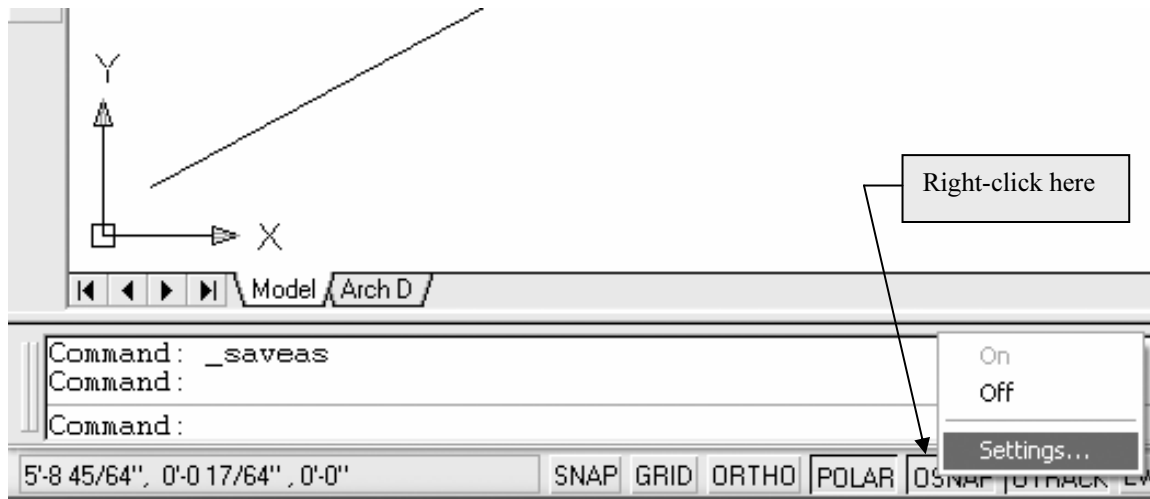

Figure 2-2.4 Right-click on OSNAP button located on the Status Bar

You are now in the **Drafting Settings** dialog box on the *Object Snaps* tab, compare to **Figure 2-2.5**.

4. Make sure only the following *Object Snaps* are checked:
   a. Endpoint
   b. Midpoint
   c. Center
   d. Intersection
   e. Perpendicular

For more information…

For more on using Object Snaps, search AutoCAD's Help System for **Drafting Settings**.

Then double-click the **Drafting Settings Dialog Box** and then select the **Object Snap** link within the article.
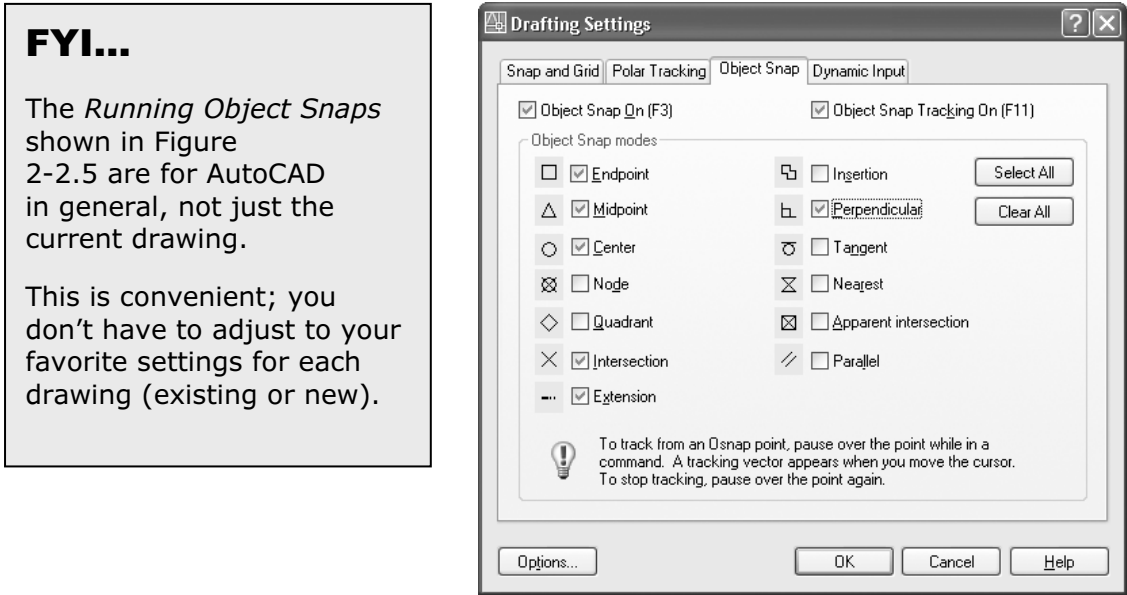
5. Click **OK** to Close the dialog box.

<table>
<tr>
<td>

FYI...

The *Running Object Snaps* shown in Figure 2-2.5 are for AutoCAD in general, not just the current drawing.

This is convenient; you don't have to adjust to your favorite settings for each drawing (existing or new).

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</tr>
</table>

Figure 2-2.5 Object Snap tab on Drafting Settings dialog

Now that you have the *Running Object Snaps* set, you will give this feature a try.

6. Using the **Line** command, move your cursor to the lower-left portion of the diagonal line. (Figure 2-2.6)

7. Hover the cursor over the line's endpoint (without picking), when you see the *Endpoint Symbol* you can click to select that point.
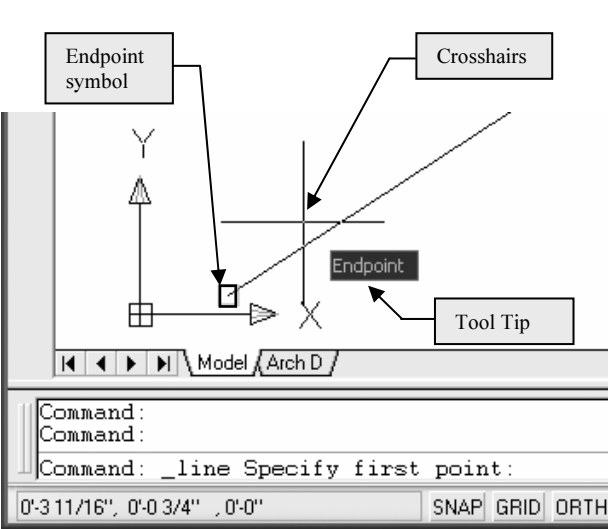

Figure 2-2.6 Endpoint OSNAP symbol visible

Just so you know...

It is important that you see the *OSNAP* symbol before clicking. Also, once you see the symbol you should be careful not to move the mouse too much.

These steps will help to ensure accurate corners.

While still in the Line command, you will draw additional lines using *OSNAP* to accurately select the line's *start point and endpoint*.

8. Draw the additional lines shown in **Figure 2-2.7** using the appropriate *Object Snap* (changing the Running Snap as required to select the required point).

> **TIP #1:** When using the Line command you can draw several line segments without the need to terminate the Line command and then restart it for the next line segment. After picking the start and end points for a line, the end point automatically becomes the first point for the next line segment; this continues until your finish the Line command (i.e., right-click and select Enter).

> **TIP #2:** At any point, while the Line command is active, you can right-click on the OSNAP button (on the Status Bar) and adjust its settings, or press Shift+Right-click; the line command will not cancel.
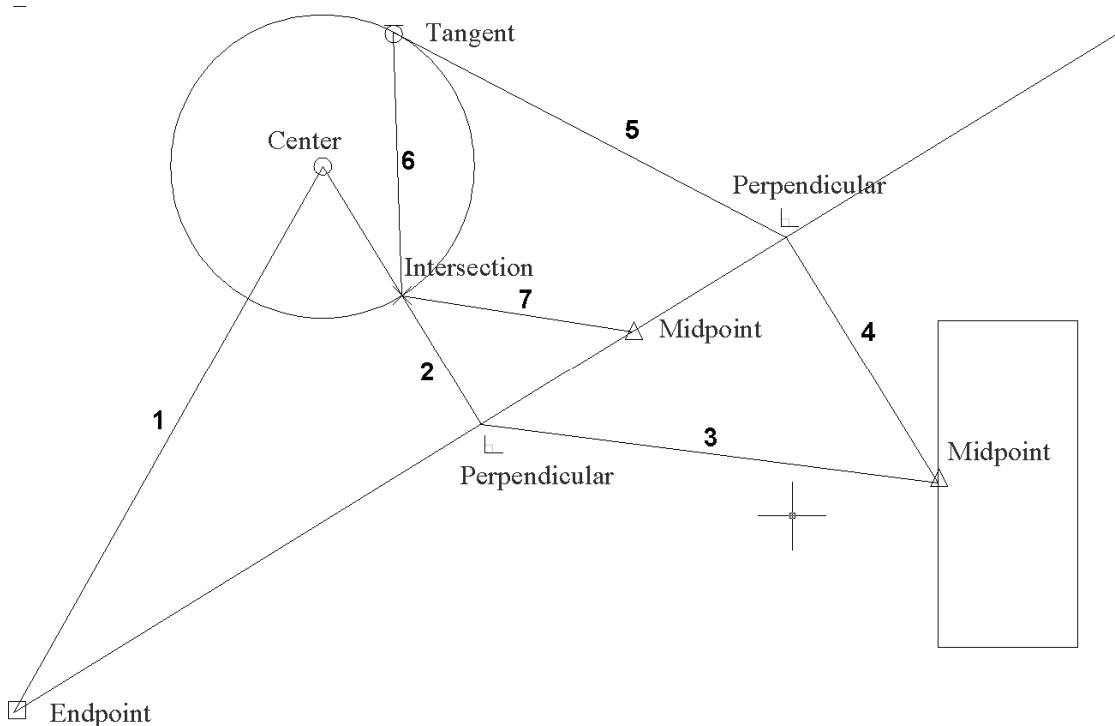

Figure 2-2.7 Several lines added using running OSNAPS.
The numbers indicate the order the lines are to be drawn in.

9. **Save** your drawing.

> # This is an architectural example?
>
> OK, this is not really an architectural example yet. However, the point here is to focus on the fundamental concepts and not architecture just yet.

Exercise 2-3:
Modify Tools

The Modify Tools are the most used in AutoCAD. Much time is spent tweaking designs and making code related revisions.

Example: Use the Draw tools (e.g., Lines, Circles, Rectangles) to initially draw the walls, doors, windows and furniture. Then use the Modify tools to *Stretch* a room so it is larger, *Mirror* a door so it swings in the direction of egress, and *Move* the furniture per the owner's instructions.

You will usually access the various Modify tools from the Modify Toolbar; you can also access them from the Modify Menu shown in Figure 2-3.1. (You can probably visualize what most of the commands do by their name.)
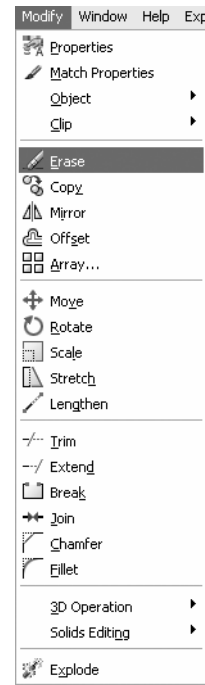
Figure 2-3.1
Modify menu

In this exercise you will get a brief overview of a few of the Modify tools, manipulating the tangled web of lines you have previously drawn.

1. **Open** drawing **ex2-2.dwg** from the previous lesson.

2. **Save-As ex2-3.dwg**.

   FYI: You will notice in this book that instructions or commands that have already been covered will have less "step-by-step" instruction.

Erase command:

It is no surprise that the Erase tool is a necessity, things change and mistakes are made. You can erase one entity at a time or several. Erasing entities is very easy; you select the Erase Icon, pick the objects to erase and then right-click (which equals pressing Enter on the keyboard). You will try this on two lines in your drawing.

3. Select **Erase.**

4. [Notice the *Command Line* is prompting: **Select objects:**]
    Use the mouse to select the lines identified in **Figure 2-3.2**.

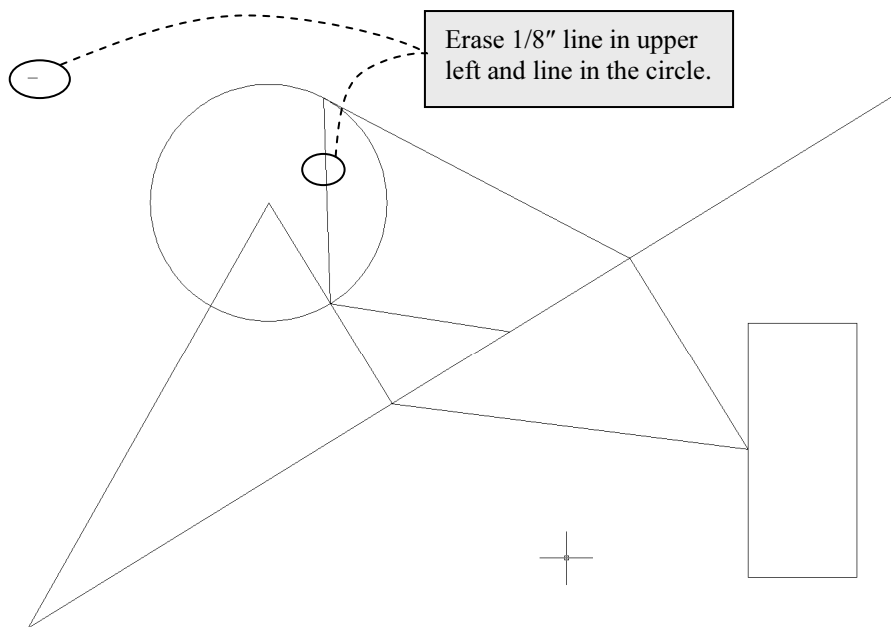   **TIP:** See the section below on Selecting Entities.

> Erase 1/8″ line in upper left and line in the circle.

Figure 2-3.2 Lines to be erased

5. **Right-click** to complete the task.

Selecting entities:

At this time we will digress and take a quick look at the various techniques for selecting entities in AutoCAD. Most commands work the same when it comes to selecting entities. As mentioned before, you need to keep your eye on the *prompts* so you know when AutoCAD is ready for you to select entities or provide other user input.

When selecting entities, you have two primary ways to select them:
  o   Individually select entities one at a time
  o   Select several entities at a time with a window
You can use one, or a combination of both, methods to select entities (when using the Copy command for example).

<u>Individual Selections</u>:

When prompted to select entities (to copy or erase, for example), simply move the cursor over the object and click. With most commands you repeat this process until you have selected all the entities you need – then you typically press Enter or right-click to tell AutoCAD you are done selecting.

<u>Window Selections</u>:

Similarly, when prompted to select entities, you can pick a Window around several entities to select them all at once. To select a window, rather than selecting an object as previously described, you select one corner of the window you wish to select (that is, you pick a point in "space"). Now as you move the mouse you will see a rectangle on the screen that represents the window you are selecting. When the Window encompasses the entities you wish to select – click the mouse.

You actually have two types of Windows you can select. One is called a **Window** and the other is called a **Crossing Window**.

Window:
This option allows you to select only the entities that are completely within the Window. Any lines that extend out of the Window are not selected.

Crossing Window:
This option allows you to select all the entities that are completely within the Window and any that extend outside the Window.

Using Window versus Crossing Window:
To select a *Window* you simply pick your two points from left to right (Figure 2-3.3a).

Conversely, to select a Crossing Window, you pick the two diagonal points of the window from right to left (Figure 2-3.3b).
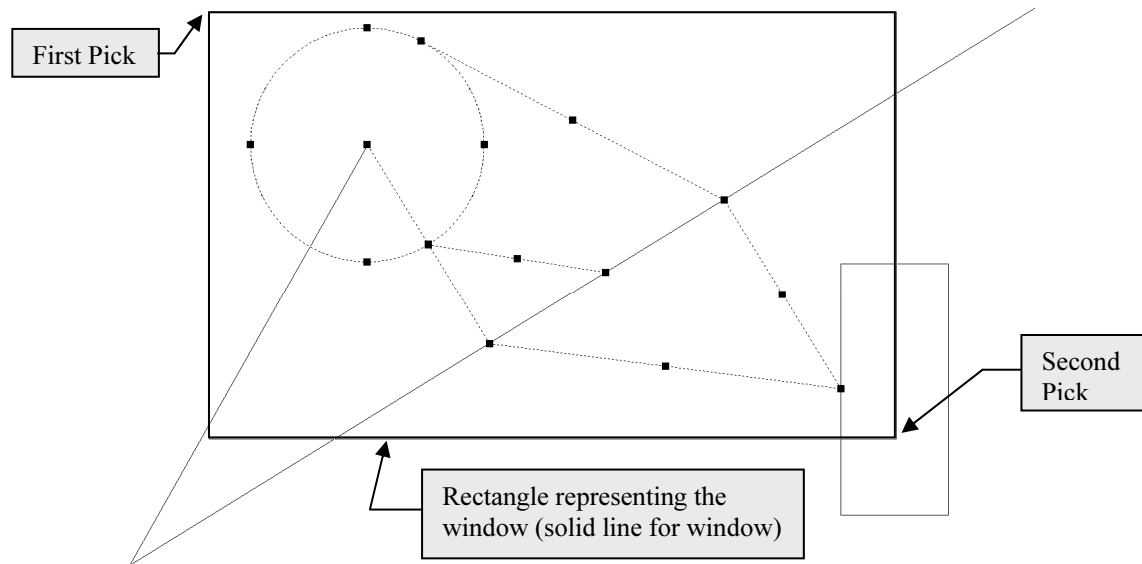
First Pick

Second
Pick

Rectangle representing the
window (solid line for window)

Figure 2-3.3a Lines selected using *Window;* only the lines within the window are selected.

First Pick

Second
Pick

Rectangle representing the
crossing window (dashed
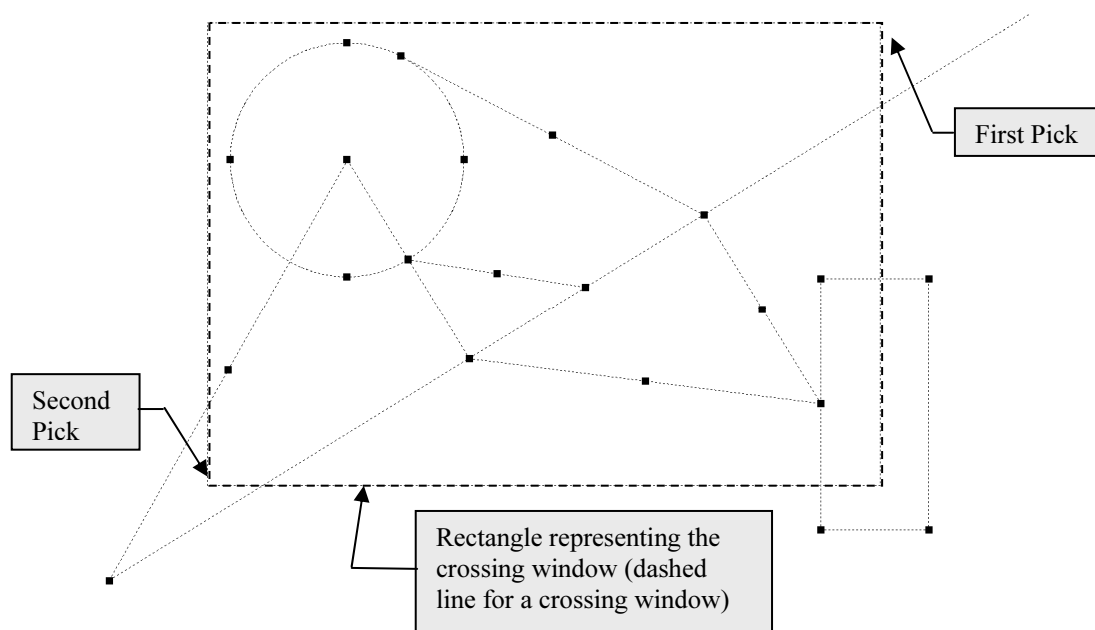line for a crossing window)

Figure 2-3.3b Lines selected using a *Crossing Window;* all lines are selected.

Copy command:

The Copy tool allows you to accurately duplicate an entity(s). You select the items you want to Copy and then pick two points that represent an imaginary vector (which provides both length and angle) defining the path used to copy the entity to; you can also type in the length and angle if there are no convenient points to pick in the drawing. You will try both methods next.

6. Select **Copy**.

7. **Select the circle** and then **right-click** - *this tells AutoCAD that you are done selecting entities to copy*.

   Notice the prompt: Specify base point or   2'-7 1/4" 3'-6 1/2"

8. Pick the **Center** of the **Circle** as the base point. (Figure 2-3.4)

   FYI: *You actually have three different OSNAPS you can use here: Center, Endpoint and Intersection. All occur at the exact same point.*

   Specify second point or <use first point as displacement>:   0'-11" < 51°

9. Pick the **Endpoint** of the angled line in the lower left corner. (See Figure 2-3.5.)

   FYI: *At this point you could continue copying the circle all over your drawing; AutoCAD will remain in copy mode until you press Esc or Enter.*
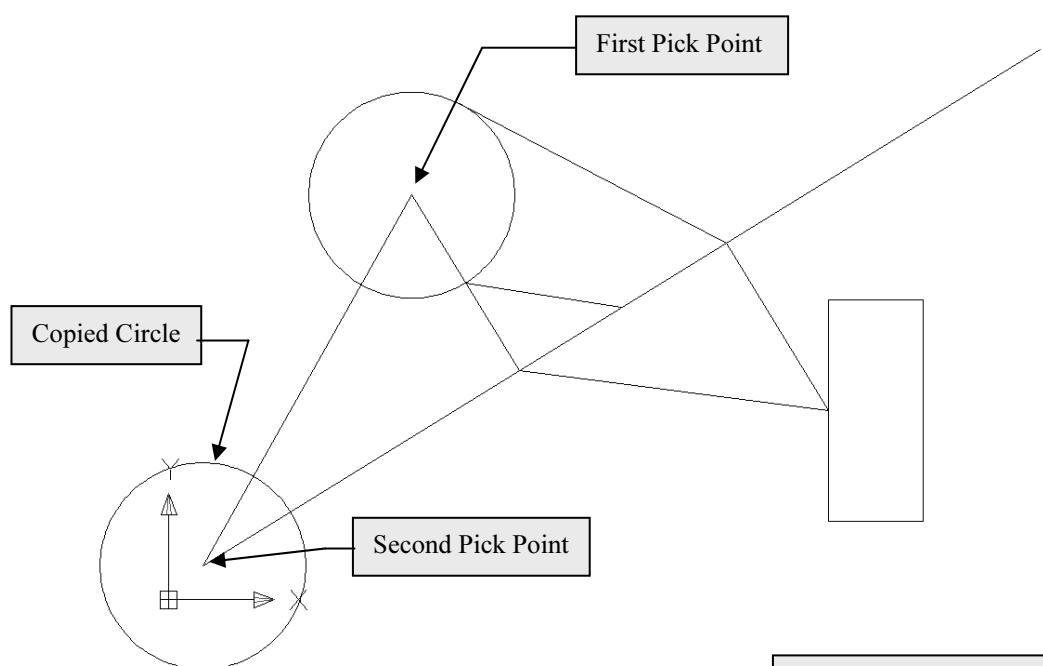
10. Press **Enter**.

First Pick Point

Copied Circle

Second Pick Point

Figure 2-3.4
Copied circle; also indicates points selected

Tip:
You may need to zoom out to see all of the circle just copied.

11. Select **Copy**.

12. **Select the rectangle** and then **right–click**.

13. Pick an arbitrary point on the screen. *(In this scenario, it makes no difference where you pick; you will see why in a moment.)*

At this point you will type in the displacement data (i.e., distance and angle), rather than picking a second point on the screen.

14. *The following will be entered via the on-screen dynamic input*: Type **18**, press **Tab** and then type **45** and press **Enter**. (See Figure 2-3.5 and on-screen prompts below.)

Specify second point or <use first point as displacement>: 18 < 212°

Specify second point or <use first point as displacement>: 18 🔒 45

What does 18<45 mean?

When you need to Move or Copy something to a very specific location, you can type in the distance and angle from the original entity (i.e. use the first point as displacement). This is very useful when you do not have any entities drawn that represent this distance and angle (e.g., as per the steps when you copied the circle, Figure 2-3.4).

The **18** represents the distance, in this case 18 inches. Remember, if you do not specify feet or inches <u>AutoCAD assumes inches</u>. So in this example you would not have to press the extra key for the inch symbol; although 18" would work.

Here are some other valid distances:

1'4	*equals 1'-4"*	16	*equals 1'-4"*
3'	*equals 3'-0"*	36	*equals 3'-0"*
0-3/4	*equals 3/4"*	.75	*equals 3/4"*
3-5/8	*equals 3 5/8"*	1'3-5/8"	*equals 1'-3 5/8"*

The **<** symbol tells you the number after it is the angle.

The 45 represents the angle (or direction) the entities will be Moved or Copied; the angle is based off of the positive x-axis in a counter-clockwise direction.

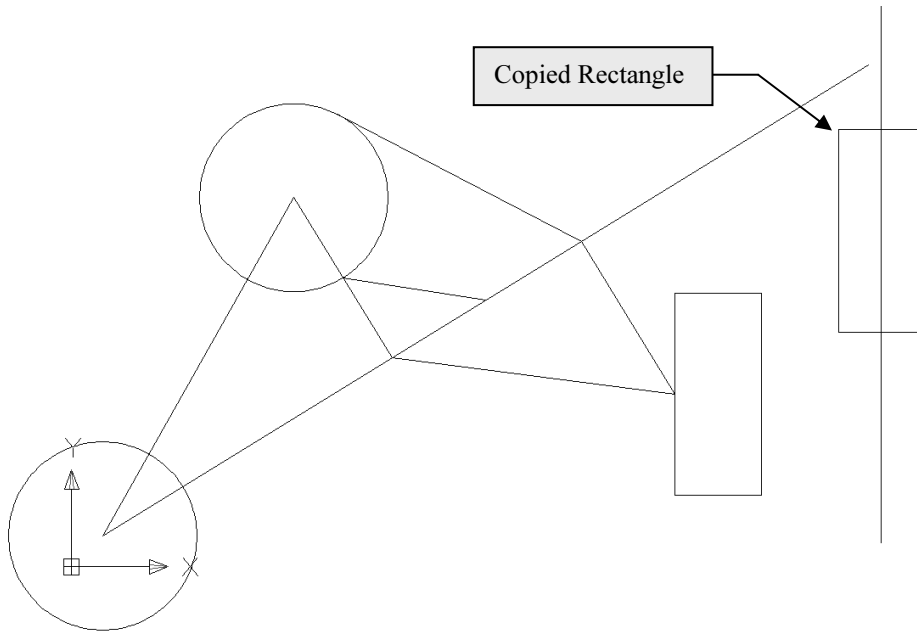*Here are some angles and a graphical representation of there direction:*
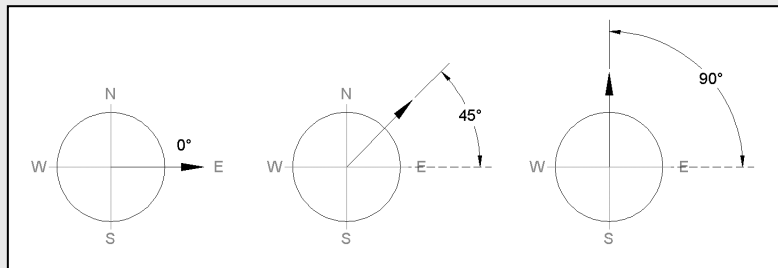
Figure 2-3.5 Copied rectangle. Dashed line represents the distance and angle of displacement.

> **NOTE:** Your drawing may look a little different than this one because your drawing is not totally to scale.

Move command:

The *Move* command works exactly like the *Copy* command except, of course, you move the entities rather than copy them. Given the similarity to the previous command covered, the *Move* command will not be covered here. You are encouraged to try it yourself. Use the *Undo* command to get back to the drawing shown in Figure 2-3.5 before proceeding.

Rotate command:

With the *Rotate* command, you can arbitrarily or accurately rotate one or more entities in your drawing. When you need to rotate accurately, you can pick points that define the angle (assuming points exist in the drawing to pick) or you can type a specific angle. The architectural template you are using in this book has the degrees "type" set to decimal. Other types are available such as Deg/Min/Sec and Radians, but architects typically use decimal angles unless drawing a site plan from a surveyor's drawing that has angles listed in another format.

15. Select **Rotate**.

16. **Select the rectangle** you just copied and then **right-click**. *This tells AutoCAD that you are done selecting entities to rotate.*

    You are now prompted: `Specify base point:` `5'-3 3/8"` `2'-2 9/16"`

This is the point about which the rotation will occur.

17. Pick the *Midpoint* of the vertical line on the left side of the rectangle. (**Figure 2-3.6**)
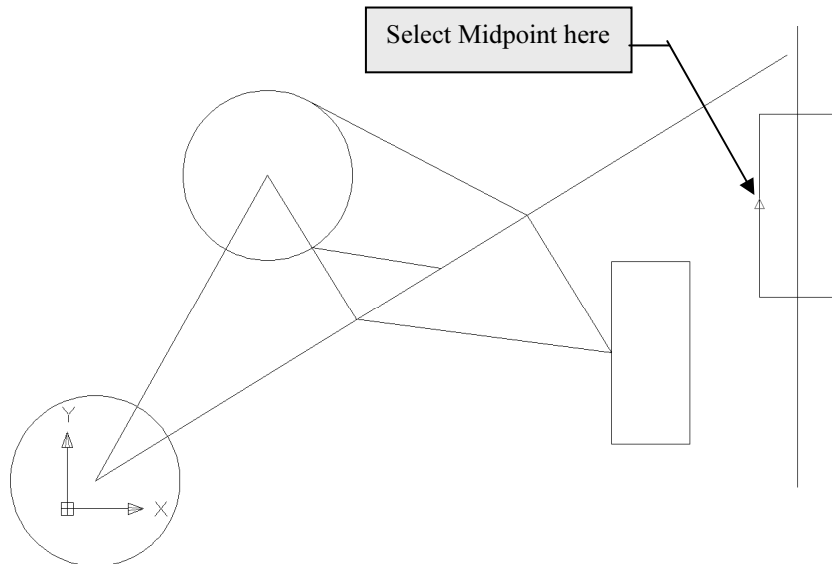

Select Midpoint here

Figure 2-3.6 Rotate command. First Step - Select midpoint of vertical line on left.

    You are now prompted: `Specify rotation angle or` `90`

18. Type **90** and then press **Enter**.

Similar to the rotation angle examples recently discussed, the previous step just rotated the rectangle 90 degrees counter-clockwise. (Figure 2-3.7)
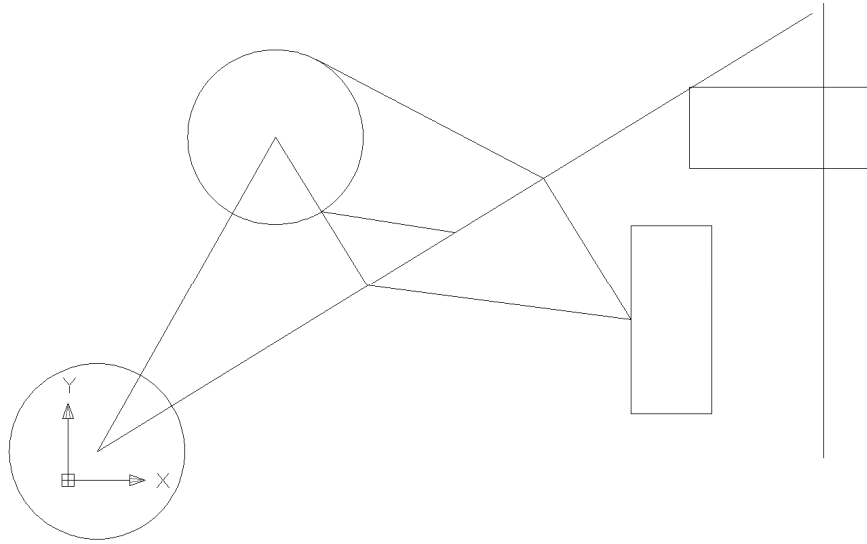
19. Select the **Undo** command.

Figure 2-3.7 Rotate command – Rectangle rotated

Now you will do the same thing, except with a different angle. A negative angle rotates the entities in a clockwise direction.

20. After selecting *Undo* in the previous step, use the **Rotate** command, **select the rectangle** and then **pick the same Midpoint** as in step 17.

21. This time, for the *Rotation Angle,* type **-45** and **Enter**.

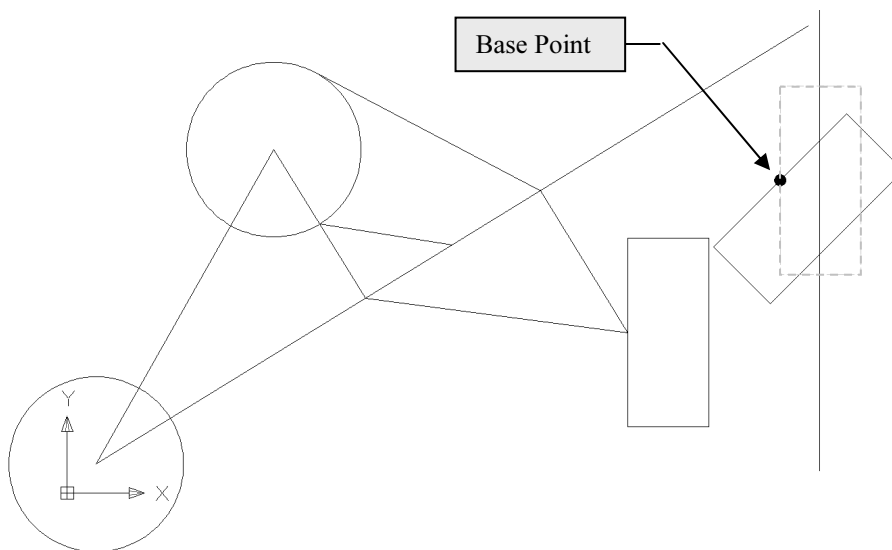Notice, now, that the rectangle rotated 45 degrees in the clockwise direction. (Figure 2-3.8)

Base Point

Figure 2-3.8 Rectangle rotated. Dashed rectangle represents original position.

Scale command:

The *Scale* command has steps similar to the *Rotate* command. First, select what you want to scale and then specify the base point. Then, instead of entering a rotation angle, enter a scale factor (e.g., 2 or .5; where 2 would be twice the original size and .5 would be half the original size).

Next you will use the *Scale* command to adjust the size of the circle near the bottom.

Before you scale the circle, you should use the **Properties Dialog** box to note the **diameter** of the circle. After scaling the circle, you will refer back to the Properties Dialog to note the change. This step is meant to teach you how to verify the accuracy and dimensions of entities in AutoCAD.

> **TIP:** You can double-click on most objects to view its properties.

22. Select **Scale**.

23. **Select the circle** (the one at the bottom) and then **right-click**.

You are now prompted: Specify base point: -1'-9" 1'-7"

> **TIP:** The numbers shown in the prompt above are X & Y values from the Origin.

This is the point about which the entity (or entities) will be scaled.

24. Pick the center of the circle. *(In this case you can use one of three Osnaps: Center, Endpoint or Intersection.)*

You are now prompted: Specify scale factor or .5

25. Type **.5** (as shown in previous prompt) and press **Enter**.

Now use the **Properties Dialog** to note the change in the size of the circle, from 6 5/8" to 3 5/16" radius. A Scale Factor of .5 reduces the entities to half their original scale. (Figure 2-3.9)

Selecting the correct base point:

To get the results desired, you need to select the appropriate Base Point, for both the *Scale* and *Rotate* commands. A few examples are shown in Figure 2-3.10. The dashed line indicates the original position of the entity being modified. The black dot indicates the base point selected.
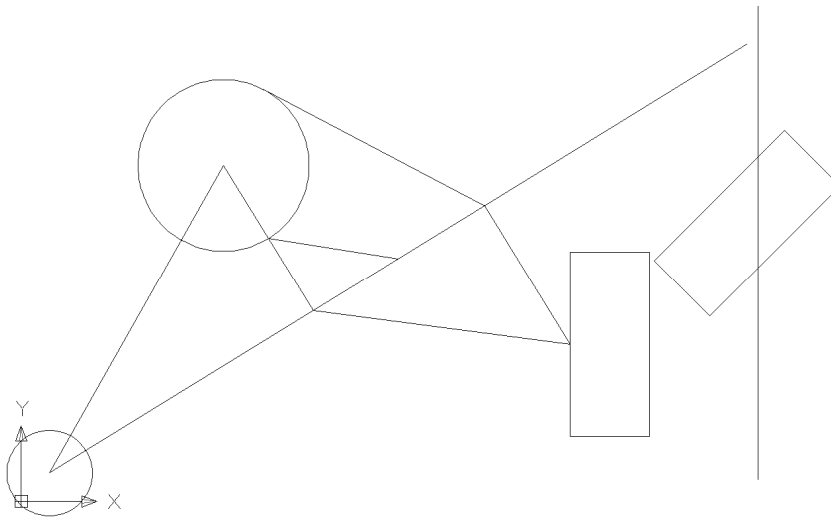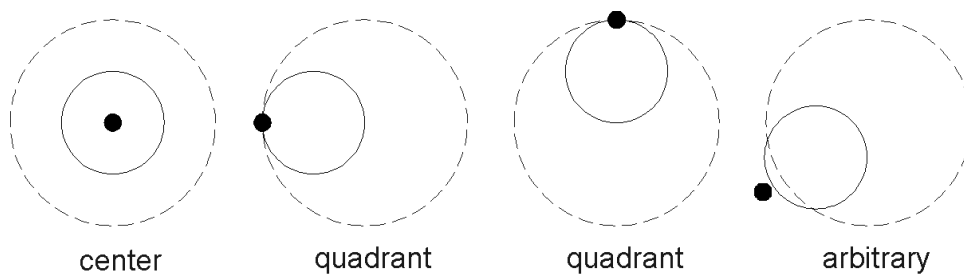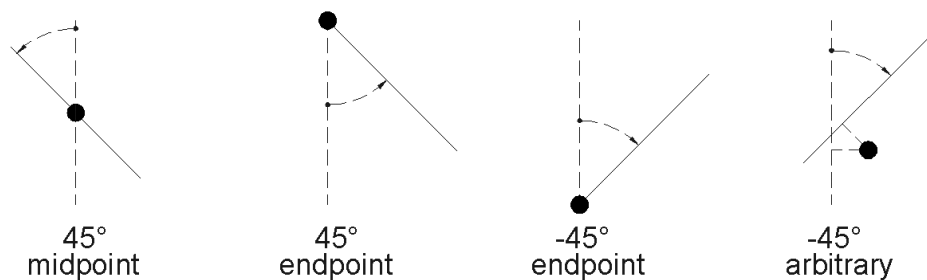
Figure 2-3.9 Scaled circle. Notice it is half the size of the other circle.

center   quadrant   quadrant   arbitrary
Base Point Options: Scale Command

45°   45°   -45°   -45°
midpoint   endpoint   endpoint   arbitrary
Base Point Options: Rotate Command

Figure 2-3.10 Base Point Options. The various results when different base points are selected.

This will conclude the brief tour of the Modify toolbar.

As you surely noticed, the Modify Toolbar has a few other icons that have not been investigated yet. Most of these will be cover later in this book.

26. **Save** your drawing (your drawing should already be named: **ex2-3.dwg** per step number 2 above).

Images courtesy of LHB
www.LHBcorp.com

Carter Residence

Exercise 2-4:
Annotations

Annotations (text; notes) allow designers and drafters to accurately describe the drawing. You will take a quick look at this feature now.

Annotations:

Adding annotations to a drawing can be as essential as the drawing itself. Often the notes (i.e., annotations) describe something about a part of the drawings that would be difficult to discern from the drawing alone.

For example, a wall section drawing showing a bolt may not indicate, graphically, how many bolts are needed or at what spacing (because the spacing of the bolts are perpendicular to the view). The note might say "*5/8" anchor bolt at 24" O.C.*"

Next you will add text to your drawing.

1.   **Open** drawing **ex2-3.dwg** from the previous lesson.

2.   **Save-As ex2-4.dwg**.

3.   **Zoom** out, if required, your screen should look like Figure 2-4.2.

4.   Select the **Multiline Text** (referred to as *Mtext*).

Notice the current *Command Prompt*; you are given the current *Text Style* (covered more thoroughly later in the book) and the current *Text Height*. You are also prompted to **specify first corner**. Basically you are specifying a window (or box) that will contain the text, which is easily adjusted later as well. (Figure 2-4.1)
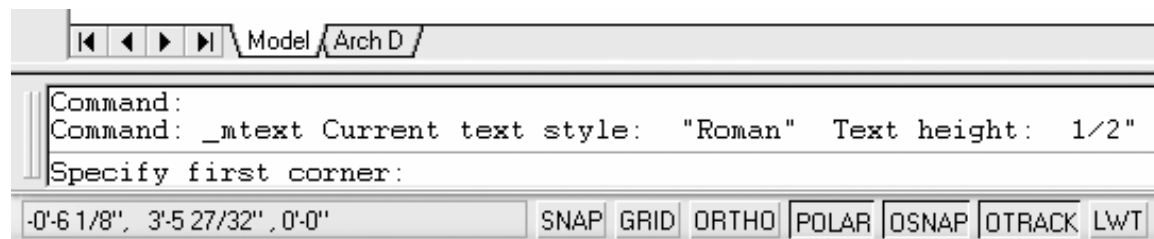

Figure 2-4.1 Command prompt for Mtext command

5. Pick your first point and then your second point to define a box as shown in **Figure 2-4.2**.

    TIP: You rarely click and drag your mouse button in AutoCAD; mostly you click and release the mouse button and then position the cursor at the second location and then click again.
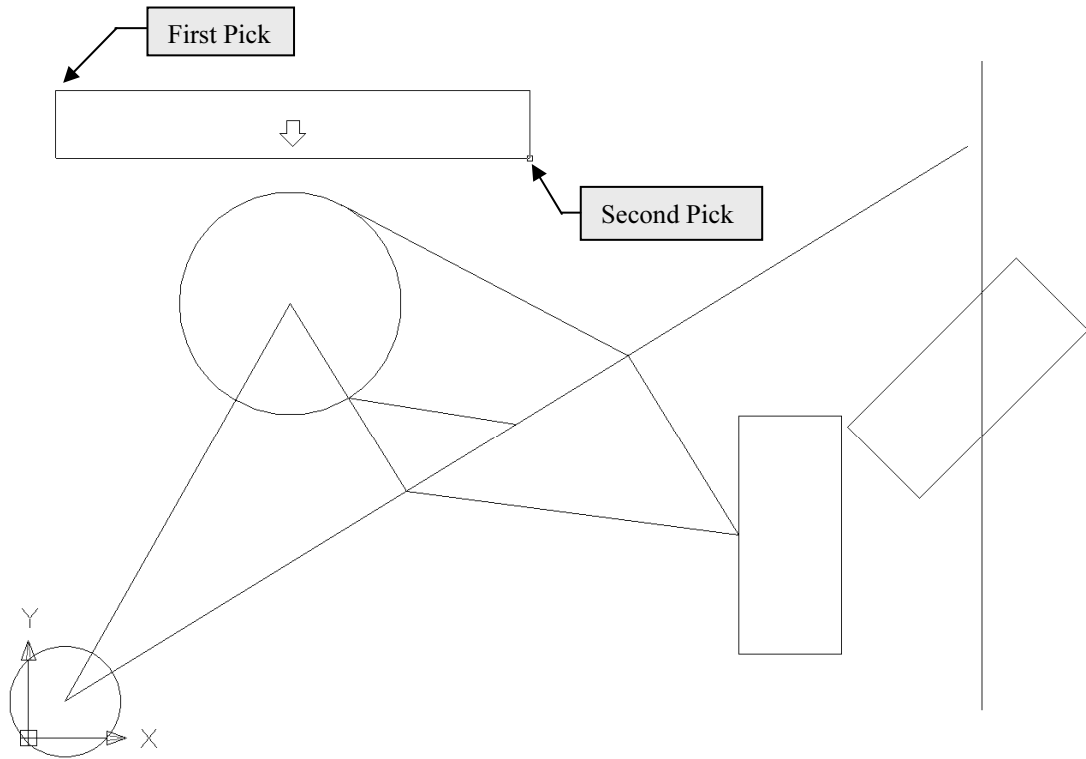

Figure 2-4.2 Mtext; defining the window that contains the text (first point picked; picking second)

When picking the Mtext window, the width is all that really matters. The width of the selected window is what controls the text wrapping feature. AutoCAD will never hide text; so as you type, if the line of text becomes longer than the width of the Mtext window you specified, the text will return to the next line (just like a word processor). AutoCAD will create as many lines as are required to show all the text you typed, regardless of the initial height of the Mtext window.

TIP: In the *Mtext* editor you can right-click to see a pop-up menu that gives you access to several tools like Select All and Find and Replace. See image to the right for an example.

You can now enter text in the *Mtext* window. This window will not be visible when you are done typing. The *Mtext* tool is similar to text editors like MS Word or WordPerfect; you can make text bold, italic, centered, underlined, etc.

Next you will enter some text.

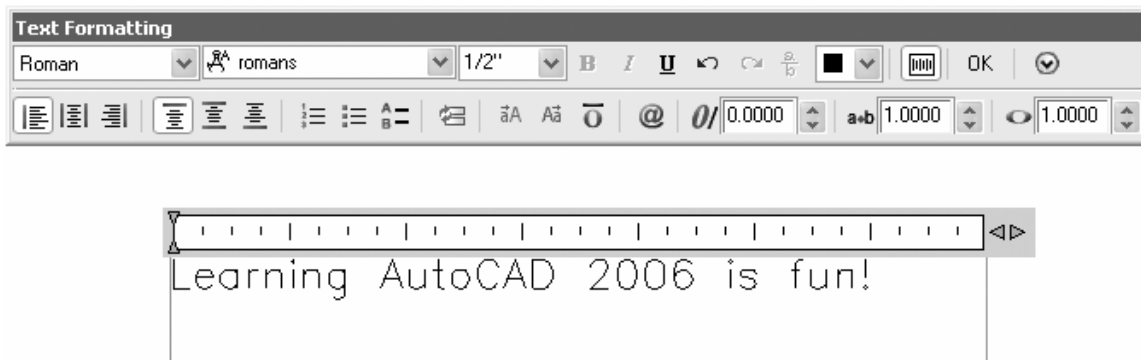6. In the *Mtext Editor* you will type the following text (Figure 2-4.3): **Learning AutoCAD 2007 is fun!**

Figure 2-4.3 Mtext; text typed in text editor window, Text Formatting toolbar displayed

7. When finished typing text, click the **OK** button on the floating *Text Formatting* toolbar shown in Figure 2-4.3 above.

    TIP: If you ever want to cancel the Mtext window without typing text, you can simply press the Escape key on the keyboard.

Your text should look similar to Figure 2-4.3. Text can be Moved and Rotated with the tools on the *Modify Toolbar*.

> **Tip:**
>
> Whenever you select an Mtext odject, AutoCAD displays a grip in each corner to represent the Mtext window. A grip can be moved to increase or decrease the width of the window. The height will automatically adjust accordingly. Try it.

Whenever you want to edit text, you double-click on it.

8. Double-click on the text, position the cursor at the end of the sentence, press *Enter* and then type your name.
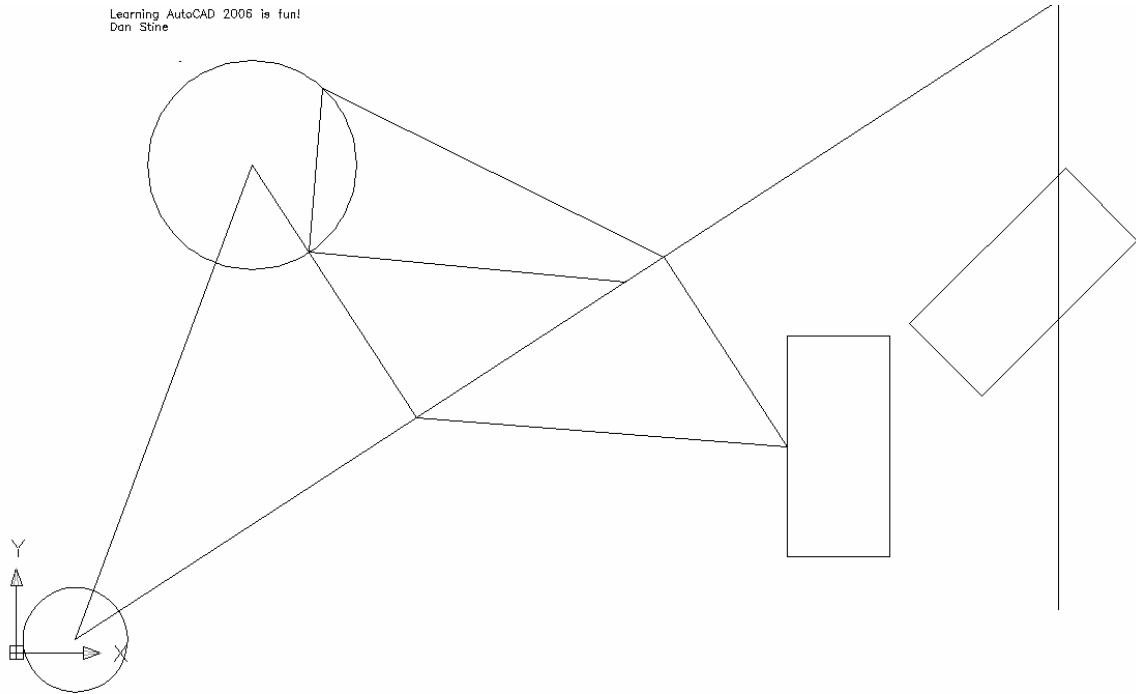
9. **Save** your drawing.

Learning AutoCAD 2006 is fun!
Dan Stine

Figure 2-4.4 Mtext; final results of text entered

Tip:

With the Mtext Editor open (Fig. 2-4.3), you can Copy/Paste text in from other programs such as MS Word. Most of the formatting (e.g. font, bold, underline, etc.) will be preserved.

Exercise 2-5:
Printing

The final exercise in this lesson will cover another essential feature in CAD, which is printing. Printing takes some time to fully understand. Printing will be covered more, later in this book (Lesson 10).

Print Room in a large architectural firm

Image courtesy of BWBR Architects
www.bwbr.com

In this exercise you will print to an 8 1/2" x 11" piece of paper with the drawing filling the screen (i.e. not to scale).

1.   Open drawing **ex2-4.dwg**.

2.   Use the Zoom tools if required to display the portion of drawing shown in Figure 2-4.4.

3.   Select the **Plot** icon (called print in other programs).

You are now in the *Plot Dialog* box. This is where you tell AutoCAD what printer or plotter to use, what scale and what size paper - to name a few examples. (Figure 2-5.1)
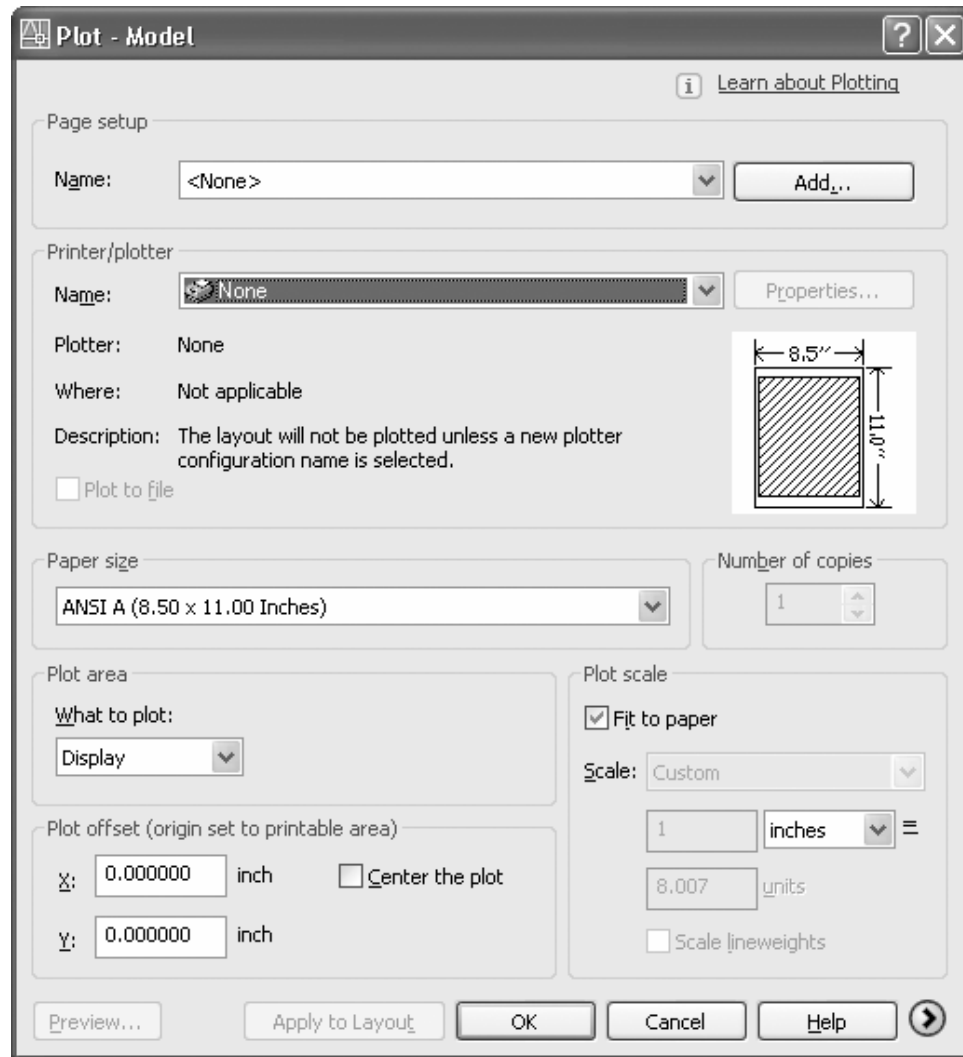
Figure 2-5.1 Plot Dialog Box; basic options

4. Make the following adjustments: (Figure 2-5.1)
    a. Select a **Printer** you have access to.
    b. Paper size: **Letter** (8.5" x 11")
    c. What to plot: **Display**
    d. Center the plot: *checked*
    e. Plot scale: **Fit to paper** *checked*

    FYI: The Display option (item 4c above) will only plot what is visible on screen; that is why you need to zoom in or out per step 2.

Additional settings are available, but are hidden because they are not used as often. You will take a quick look at this before plotting.

5. Click the "more options" arrow in the lower right corner of the *Plot* dialog box. (Figure 2-5.1)

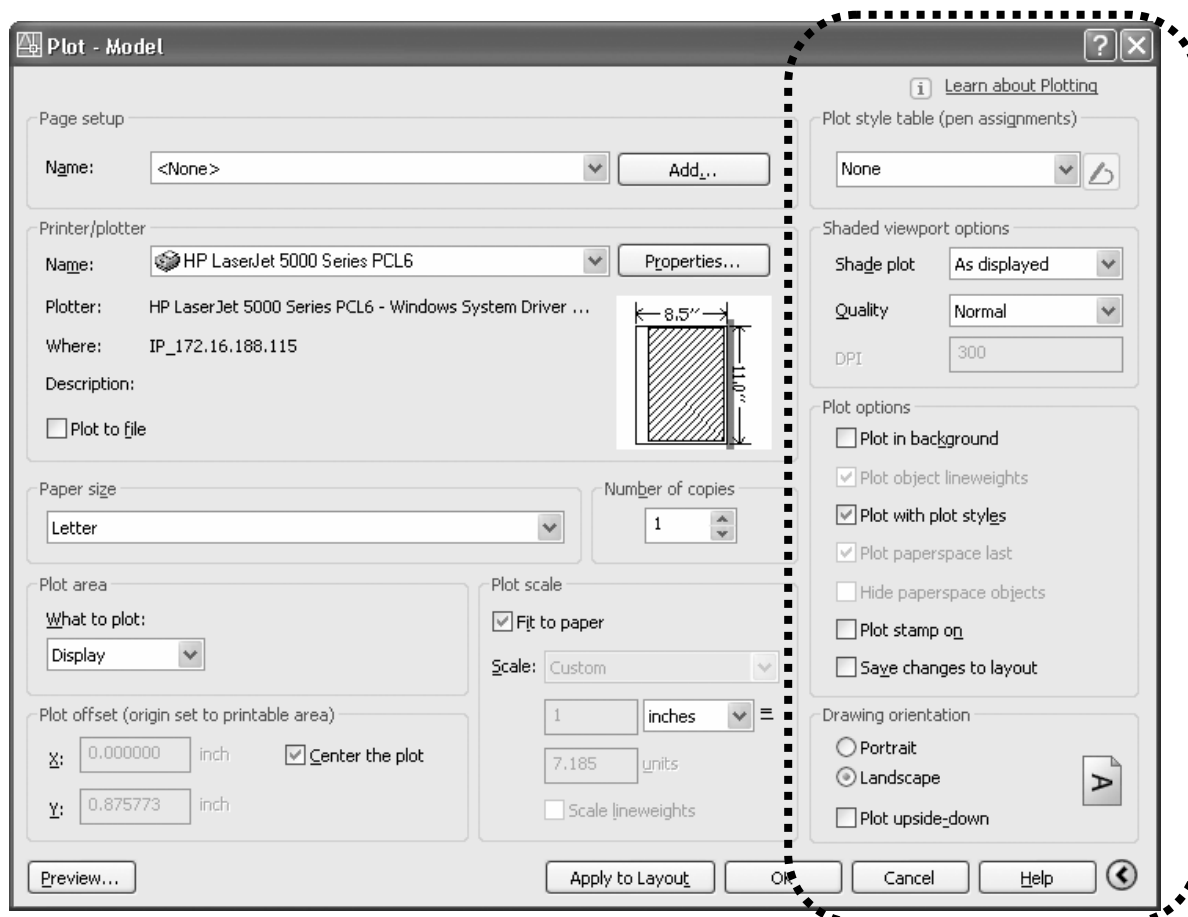6. The only setting you need to verify is that *Drawing Orientation* is set to **Landscape**.

Figure 2-5.2 Plot Dialog Box; basic + more options shown

AutoCAD can print to any printer/plotter installed on your computer.

A Printer is an output device that uses smaller paper (e.g. 8 1/2"x11" or 11"x17"). A Plotter is an output device that uses larger paper; plotters typically have one or more rolls or paper ranging in size from 18" wide to 36" wide. A roll feed plotter has a built-in cutter that can – for example – cut paper from a 36" wide roll to make 24"x36" sheet.

Plotter w/ three paper rolls

Color **printer** / copier

7. Click **OK** to send the plot (print).

Using PREVIEW...

Before you click OK, it is a good idea to click the *Preview* button first. This will show how your drawing will look on the size paper selected.

Doing a *Preview* first will save time and paper.

Lesson 2 recap:

In this lesson, *Crash Course Introduction (the Basics)*, you learned how to draw accurate lines and circles, use the *Object Snap* feature for precise pick points, use a handful of *Modify* tools (e.g. *Copy*, *Move*, *Rotate* & *Scale*). You also had a brief intro to creating text and plotting. You should have a well balanced amount of base knowledge to begin creating more meaningful objects, which is what you will do in the next lesson!

Self-Exam:

The following questions can be used as a way to check your knowledge of this lesson. The answers can be found at the bottom of this page.

1. AutoCAD is only accurate to 3 decimal places. (T/F)

2. The diamond shaped Osnap symbol represents the Midpoint snap. (T/F)

3. "Full Preview" in the Plot Dialog WILL save you time and paper. (T/F)

4. Use the _____ command to duplicate objects.

5. When selecting entities, use the _____ _____ to select all the entities in the selection window and all entities that extend through it.

Review Questions:

The following questions may be assigned by your instructor as a way to assess your knowledge of this section. Your instructor has the answers to the review questions.

1. AutoCAD is a raster based program. (T/F)

2. The Base Point you select for Rotate and Scale is not important. (T/F)

3. Entering 16 for a distance, actually means 16'-0" to AutoCAD. (T/F)

4. Use the _____ command to create squares.

5. You can change the height of the text while in the Mtext editor. (T/F)

6. Selecting this icon ( ) allows you to _____ one or more entities in a drawing.

7. What is the name of the Template file you are to use when creating a new Drawing in this book? _____

8. With the Rotate command, typing 45 for the degrees prompt, rotates the entities in the _____ direction; whereas -45 rotates the entities in the _____ direction.

9. List all the Osnap points available on a circle (ex. Line: endpoint, midpoint, nearest) _____.

10. The OSNAP button on the Status Bar must be "pushed in" (i.e., on) to automatically and accurately select snap points while drawing. (T/F)