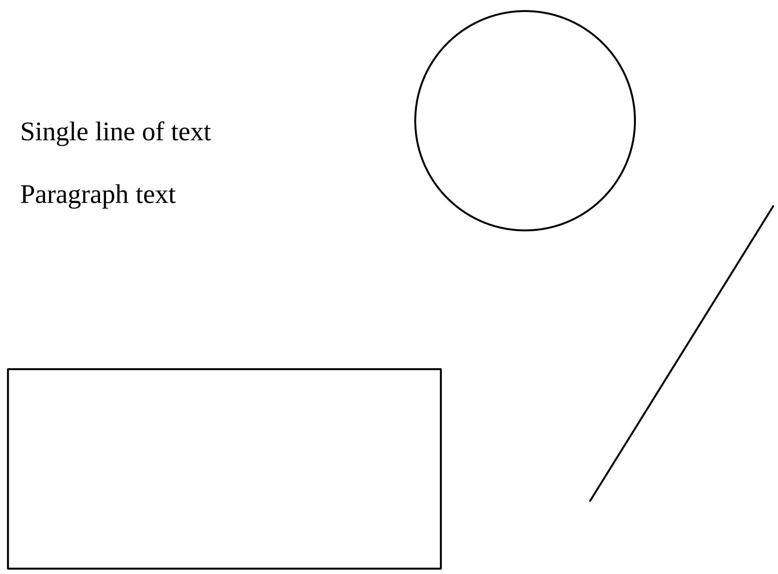
AutoCAD LT Drawing Tools

This section covers:

1. **2D Object creation commands**
2. **Text commands**



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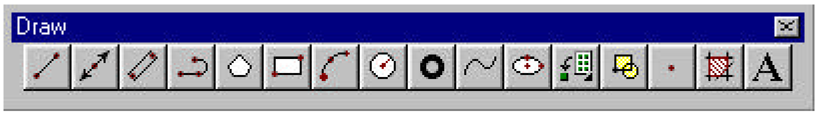
Drawing commands

"Without drawing objects would we have a drawing or a database?"

Drawing is the number one task that you will find yourself performing inside of AutoCAD LT. There are a variety of Drawing commands and Aids that we will be taking a look at during this section. Drawing commands range from the simplest, that being a Line to a Polyline which is a complex line that can have curves and multiple sections in it. We will be cover Drawing commands in two sub-sections. The first covering the basic drawing commands, and then we will be taking a look at some of the more Advanced commands.



Cartesian Coordinates

AutoCAD LT uses the Cartesian coordinates, (x,y) to indicate a location in the drawing. X represents points located along the horizon (or horizontal on the screen), and Y represents points located vertically in the drawing. All X and Y distances are measure from 0,0. Figure 7.1 shows where the X and Y values are positive and negative in the drawing. Figure 7.2 shows the direction of angles in the drawing.

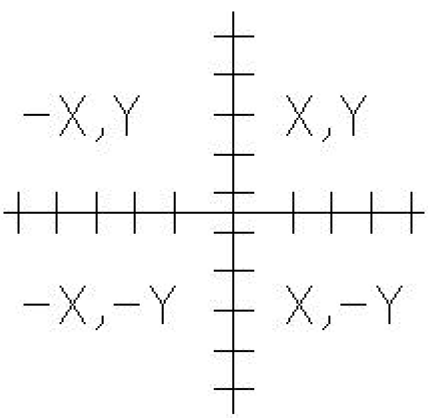


Fig. 7.1

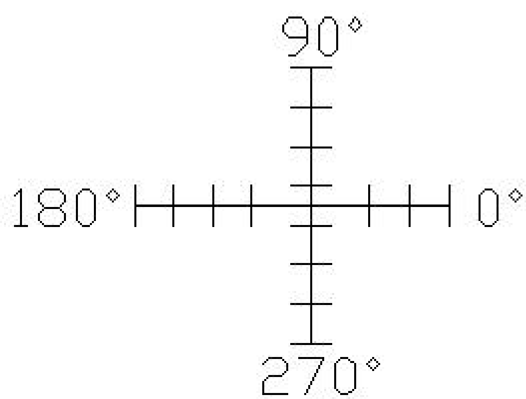


Fig. 7.2

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Point Entry Practice

During this practice you will be entering Absolute Coordinates with the Line command and entering points at the keyboard for the two points of the Line. Don't be to concerned about the how to use the Line command yet we will cover it a little bit later. 2 mins.

1. Start a new drawing from **Scratch** with **English** highlighted. This will ensure that all the settings are setup to AutoCAD LT standards.

2. Start up the **LINE** by typing in LINE at the command line.

3. At the **From point:** prompt type in **3,5** and press Enter to accept the value.

4. Now at the **To point:** prompt type in **6,8** and press Enter to accept the value. Press Enter one more time to exit the **LINE** command.

You should have created a Line that started at 3,5 and ended 6,8.

During this practice you will be entering Polar Coordinates with the Line command again and enter the first point and use direct distance for the second point. 2 mins.

1. Start a new drawing from **Scratch** with **English** highlighted. This will ensure that all the settings are setup to AutoCAD LT standards.

2. Start up the **LINE** command.

3. At the **From point:** prompt select a point in the drawing for the first point of the Line.

4. Now at the **To point:** prompt move the Crosshair below or above your original point and type-in **4** at the command line and press Enter to accept the value. Press Enter one more time to exit the **LINE** command.

You should have created a Line that is a total length of 4 from your original point. You could have also entered @4<90 or @4<270 instead of using your Crosshairs and tying in 4 at the command line.

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During this practice you will be entering Relative Coordinates with the Line command again and enter the first point and use direct distance for the second point. 2 mins.

1. Start a new drawing from **Scratch** with **English** highlighted. This will ensure that all the settings are setup to AutoCAD LT standards.

2. Start up the **LINE** command.

3. At the **From point:** prompt select a point in the drawing for the first point of the Line.

4. Now at the **To point:** prompt type in @**5,3** at the command line and press Enter to accept the value. Press Enter one more time to exit the **LINE** command.

You should have created a Line that started at your original point and then went 5 units in the X direction and 3 units in the Y direction from your point. The difference from Absolute and Relative is that Absolute goes to the coordinates that you enter and Relative uses the original point as the basis of the point to create the new point from.

During this practice you will be entering a combination Relative and Polar Coordinates with the Line command again and enter the first point and use direct distance for the second point. 2 mins.

1. Start a new drawing from **Scratch** with **English** highlighted. This will ensure that all the settings are setup to AutoCAD LT standards.

2. Start up the **LINE** command.

3. At the **From point:** prompt select a point in the drawing for the first point of the Line.

4. Now at the **To point:** prompt type in @**4<45** at the command line and press Enter to accept the value. Press Enter one more time to exit the **LINE** command.

You should have created a Line that is a total length of 4 at an angle of 45 degrees from your original point.

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Drawing Aids

Drawing Aids are a set of settings that help to speed up drawing objects in AutoCAD LT. Some of the settings are more valuable than others. Lets take a look at the Drawings Aids in AutoCAD LT and how they help in drawing. We will first take a look at the Drawing Aids dialog box (Fig. 7.4).

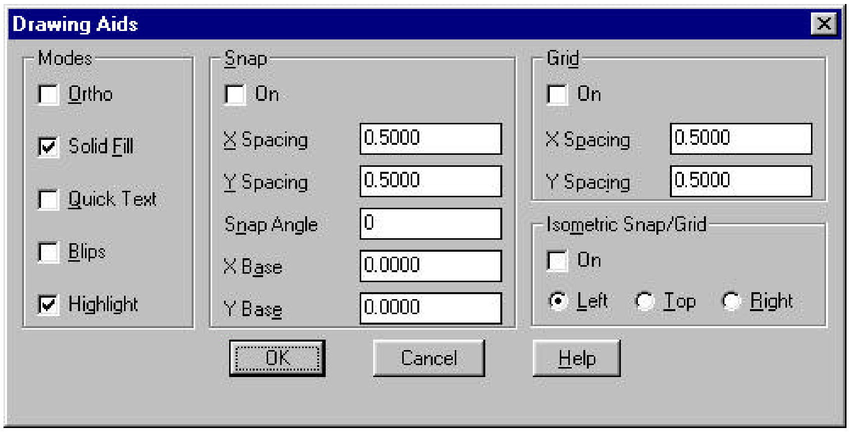

Fig. 7.3

Command Access	
Command Line	DDRMODES + Enter
Menubar	TOOLS>>Drawing Aids...

Drawing Aids - Modes

The drawing modes are a set of various tools that help in drawing and editing objects. The way that these tools are setup depends on what you are working with in your drawing and how you want to work.

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Drawing Aids - Modes - Ortho

Ortho (or Orthomode) is a setting that controls whether or not you are drawing your objects either parallel or perpendicular to your drawing plane. This setting aids in drawing straight lines and placing things at 0, 90, 180 , or 270 degrees to the current UCS (User Coordinate System). The Following Figures show how this setting affects the drawing of lines. Figure 7.4 has Ortho On (or checked), which will force a straight line to be drawn when selecting two points. Figure 7.5 has Ortho Off (or unchecked), which will allow the user to select any two points and draw the line along the give points.

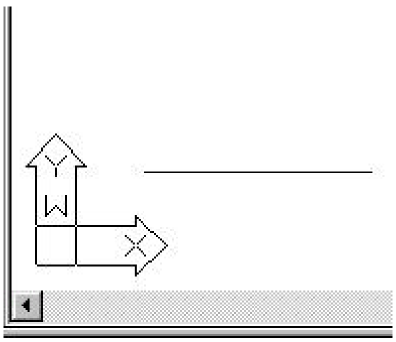

Fig.7.4

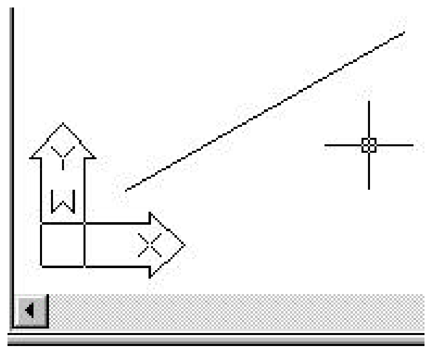

Fig. 7.5

Command Access	
Command Line	ORTHOMODE + Enter
Statusbar	Double Click ORTHO
Menubar	TOOLS>>Drawing Aids...
Accelerated Key	F8

Tip: You can use type ORTHOMODE at the command line with an apostrophe (') to use it transparently otherwise the other three options already provide that method automatically.

Note: For command line reference value of 0 is Off (or unchecked) and 1 is On (or checked).

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Drawing Aids - Modes - Solid Fill

Solid Fill (or Fillmode) is a setting that controls whether or not you see Donuts or Polyline widths filled in on the screen. This setting if turned Off (or unchecked) provides faster regenerations of the drawing if you have a lot of Donuts or Polylines with a width in your drawing. AutoCAD LT normally has this setting automatically turned On, because then you can tell what is filled in the drawing. Figure 7.6 has Solid Fill On (or checked), which will force the fill to appear in the drawing. Figure 7.7 has Solid Fill Off (or unchecked), which will force the filled object to show the outer lines of the filled areas, at times this could get very confusing on a very busy drawing.

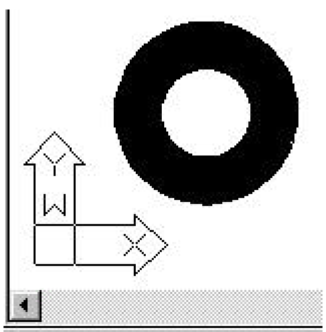

Fig.7.6

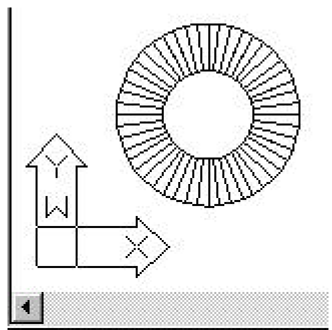

Fig. 7.7

Command Access	
Command Line	FILLMODE + Enter
Menubar	TOOLS>>Drawing Aids...

Tip: You can use type FILLMODE at the command line with an apostrophe (') to use it transparently otherwise the other option already provides that method automatically.

Note: For command line reference value of 0 is Off (or unchecked) and 1 is On (or checked).

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Drawing Aids - Modes - Quick Text

Quick Text (or Qtextmode) is a setting that controls whether or not you see Text or Attributes on the screen with the text value displayed or as a rectangular box that represents the size and location of the text. This setting if turned On (or checked) provides faster regenerations of the drawing if you have a lot of Text or Attributes with a fancy font associated with it in your drawing. AutoCAD LT normally has this setting automatically turned Off, because then you can see your Text or Attributes with there values in them. Figure 7.8 has Quick Text On (or checked), which will force the Text or Attributes in the drawing to appear looking like boxes. Figure 7.9 has Quick Text Off (or unchecked), which will force the Text or Attributes in the drawing to appear like normal with the actual values of the Text displayed.

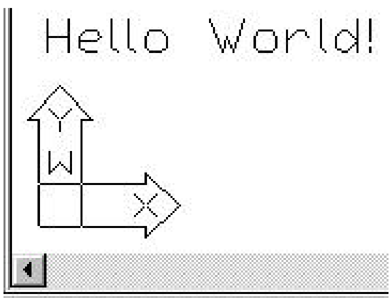

Fig.7.8

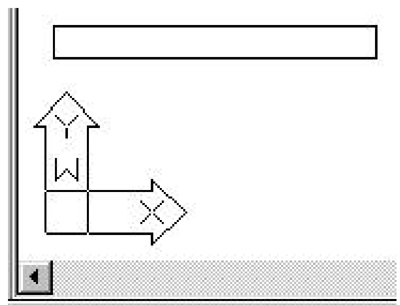

Fig. 7.9

Command Access	
Command Line	QTEXTMODE + Enter
Command Line	QTEXT + Enter
Menubar	TOOLS>>Drawing Aids...

Tip: You can use type QTEXTMODE at the command line with an apostrophe (') to use it transparently otherwise the other option already provides that method automatically.

Note: For command line reference value of 0 is Off (or unchecked) and 1 is On (or checked).

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Drawing Aids - Modes - Blips

Blips (or Blipmode) is a setting that controls whether or not you see a small point show up where you pick all your points on the screen. This setting really has no impact on speed or performance, of drawing anything. It is just a visual indicator that you selected a point on the screen. AutoCAD LT normally has this setting automatically turned Off, because most people find it really annoying to draw with them on. Figure 7.10 has Blips On (or checked), which will draw the little markers on the screen every time I select a point. Figure 7.11 has Blips Off (or unchecked), which will not draw the markers on the screen as I draw objects.

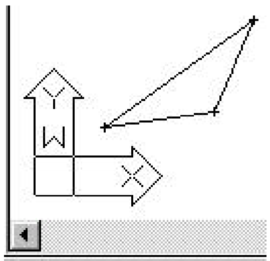

Fig.7.10

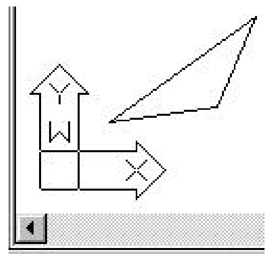

Fig. 7.11

Command Access	
Command Line	BLIPMODE + Enter
Menubar	TOOLS>>Drawing Aids...

Tip: You can use type BLIPMODE at the command line with an apostrophe (') to use it transparently otherwise the other option already provides that method automatically.

Tip: You can get rid of the Blips drawn on the screen by using the **REDRAW** command. This only refreshes the screen instead of the drawing like **REGEN** does so it is a lot faster to run.

Note: For command line reference value of 0 is Off (or unchecked) and 1 is On (or checked).

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Drawing Aids - Modes - Highlight

Highlight (or Highlight) is a setting that controls whether or not you see the objects that you select for the Editing commands become dotted so you know if you selected them for not. This setting really can impact speed and performance when you are trying to modify a lot of objects in the drawing at one single time. It is a very important visual indicator that lets you know if you missed selecting an object that you may have wanted to select. AutoCAD LT normally has this setting automatically turned On, because most people find it very useful to see a visual feed back of what they selected. Figure 7.12 has Highlight On (or checked), which will show selected objects as being dotted. Figure 7.13 has Highlight Off (or unchecked), which will not show selected objects as being dotted.

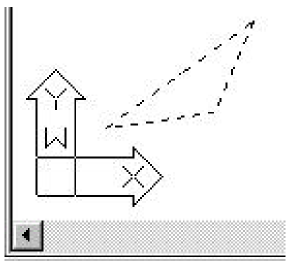

Fig.7.12

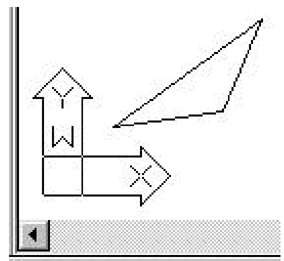

Fig. 7.13

Command Access	
Command Line	HIGHLIGHT + Enter
Menubar	TOOLS>>Drawing Aids...

Tip: You can use type HIGHLIGHT at the command line with an apostrophe (') to use it transparently otherwise the other option already provides that method automatically.

Note: For command line reference value of 0 is Off (or unchecked) and 1 is On (or checked).

Notes:

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Drawing Aids - Snap

The drawing snap is a set of settings that controls the spacing for snapping and also the angle in which the snapping grid goes. Snapping is used to create a fast and easy way to create symmetrical and non-symmetrical objects in even increments.

Drawing Aids - Snap - Snap Toggle

Snap Toggle is a setting that controls whether or not you use Snap while you are drawing your objects in your drawing. You can tell you Snap is On (or checked), because while you are drawing you will see that your Crosshairs moves in the drawing screen with a jumping motion. The jumping motion is create because of the spacing from on Snap point to the next Snap point.

Command Access	
Command Line	SNAPMODE + Enter
Statusbar	Double Click SNAP
Menubar	TOOLS>>Drawing Aids...
Accelerated Key	F9

Tip: You can use type SNAPMODE at the command line with an apostrophe (') to use it transparently otherwise the other three options already provide that method automatically.

Note: For command line reference value of 0 is Off (or unchecked) and 1 is On (or checked).

Drawing Aids - Snap - X Spacing

X Spacing is the distance in between each Snap going in the X direction of the current UCS.

Command Access	
Command Line	SNAPUNITS + Enter
Menubar	TOOLS>>Drawing Aids...

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Drawing Aids - Snap - Y Spacing

Y Spacing is the distance in between each Snap going in the Y direction of the current UCS.

Command Access	
Command Line	SNAPUNITS + Enter
Menubar	TOOLS>>Drawing Aids...

Drawing Aids - Snap - Snap Angle

Snap Angle is the angle in which Ortho reacts to. This setting doesn't affect how angles are input at the command line. Figure 7.14 shows what the Crosshair looks like at a Snap Angle of 45. Notice that it doesn't have an impact on the direction of the UCS. The Snap Angle can be very useful when needing to draw objects at a specific angle.

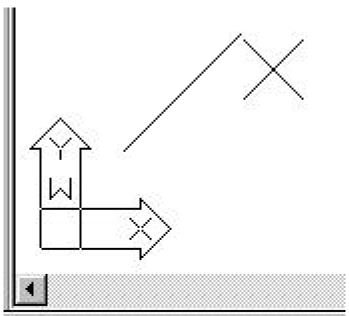

Fig. 7.14

Command Access	
Command Line	SNAPANG + Enter
Menubar	TOOLS>>Drawing Aids...

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Drawing Aids - Snap - X Base

X Base is the X value of the origin in which the Snap grid begins at.

Command Access	
Command Line	SNAPBASE + Enter
Menubar	TOOLS>>Drawing Aids...

Drawing Aids - Snap - Y Base

Y Base is the Y value of the origin in which the Snap grid begins at.

Command Access	
Command Line	SNAPBASE + Enter
Menubar	TOOLS>>Drawing Aids...

Tip: It is recommended that you leave the X and Y Base to both being 0. Moving the X and Y Base to a different location can sometimes lead to confusing drawings.

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Drawing Aids - Grid

The drawing grid is a set of settings that controls the spacing for Grid in which can be seen on the screen unlike the Snap Grid. This is one of the reasons why the Grid exists. The Grid can be either used with the Snap command or by itself as a visual representation of how much room something takes up by approximation.

Drawing Aids - Grid - Grid Toggle

Grid Toggle is a setting that controls whether or not you use Snap while you are drawing your objects in your drawing. You can tell you Snap is On (or checked), because while you are drawing you will see that your Crosshairs moves in the drawing screen with a jumping motion. The jumping motion is create because of the spacing from on Snap point to the next Snap point.

Command Access	
Command Line	GRIDMODE + Enter
Statusbar	Double Click GRID
Menubar	TOOLS>>Drawing Aids...
Accelerated Key	F7

Tip: You can use type GRIDMODE at the command line with an apostrophe (') to use it transparently otherwise the other three options already provide that method automatically.

Tip: Setting the X and Y Spacing of Grid to 0 will force the Grid to be the units specified by in the X and Y Spacing of Snap.

Note: The Grid will only be displayed up to your drawings limits (LIMITS command) size.

Note: For command line reference value of 0 is Off (or unchecked) and 1 is On (or checked).

Notes:

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Drawing Aids - Grid - X Spacing

X Spacing is the distance in between each Grid point going in the X direction of the current UCS.

Command Access	
Command Line	GRIDUNIT + Enter
Menubar	TOOLS>>Drawing Aids...

Drawing Aids - Grid - Y Spacing

Y Spacing is the distance in between each Grid point going in the Y direction of the current UCS.

Command Access	
Command Line	GRIDUNIT + Enter
Menubar	TOOLS>>Drawing Aids...

Tip: Setting the X and Y Spacing of Grid to 0 will force the Grid to be the units specified by in the X and Y Spacing of Snap.

Note: If you don't want to use the Units from Snap. You should at least make sure the grid points line up with the Snap Spacing some where, so it doesn't get confusing to you or someone else that might open your drawing.

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Drawing Aids - Isometric

The drawing isometric is a set of settings that controls whether or not you use Isometric Snapping and if so what drawing plane is it set to.

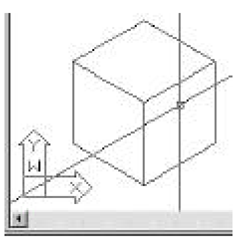
Drawing Aids - Isometric - Isometric Toggle

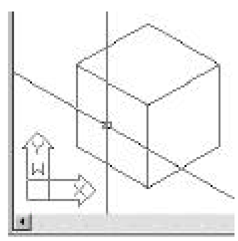
Isometric Toggle is a setting that controls whether or not you use Isometric Snap while you are drawing your objects in your drawing. You can tell if your Isometric Snap is On (or checked), because while you are drawing you will see that your Crosshair moves in the drawing screen with a jumping motion and your Crosshair is also rotated in a different orientation.

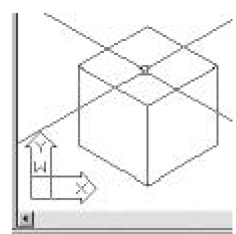
Command Access	
Command Line	SNAP>>STYLE + Enter
Menubar	TOOLS>>Drawing Aids...

Drawing Aids - Isometric - Isometric Plane

Isometric Plane is a setting that controls what plane the Crosshair is working in. You can toggle between three different options.


Right - Fig. 7.15


Left - Fig. 7.16


Top - Fig. 7.17

Command Access	
Command Line	ISOPLANE + Enter
Menubar	TOOLS>>Drawing Aids...
Accelerated Key	F5

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Object Snap (Osnap)

Object Snaps (Fig. 7.18) are used to control the way that you select points for drawing objects in AutoCAD LT. They help to ensure the accurate drawings are made by placing objects exactly where they need to be placed. There are a lot of different types of Object Snaps that AutoCAD LT offers for use during the creation and editing of Objects. Now lets take a look at the Object Snaps in AutoCAD LT and how they help in drawing.

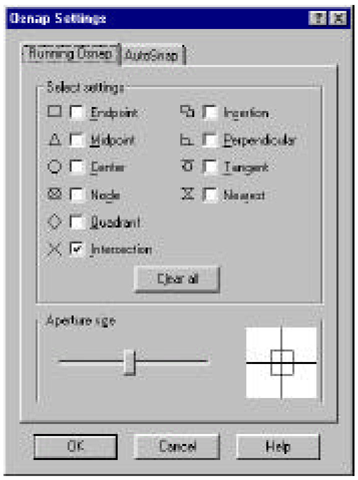

Fig. 7.18

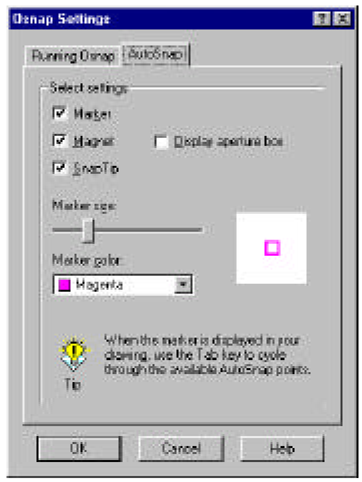

Fig. 7.19

Command Access	
Command Line	OSNAP + Enter
Toolbar	OBJECT SNAP
Statusbar	Double Click OSNAP
Menubar	TOOLS>>Object Snap Settings...
Accelerated Key	F3 or Ctrl + F

Tip: You can type DDOSNAP at the command line with an apostrophe (') to use it transparently, otherwise the other options already provide that method automatically.

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Object Snap (Osnap) - Settings

Object Snap		Some Objects Affected
Endpoint (END)	Used to select the Endpoint of an object.	Lines, Polylines, Ray, Double Lines, Arcs and anything else that has an ndpoint.
Midpoint (MID)	Used to select the Midpoint of an object.	Lines, Polylines, Ray, Xline, Double Line, and almost anything else.
Center (CEN)	Used to select the Center point of an object.	Arcs, Polylines, Splines, Circle, Donut, and Ellipse.
Node (NOD)	Used to select a Point object (or Node).	Points and Dimensions.
Quadrant (QUA)	Used to select the Quadrant of an object.	Circle, Arc, Donut, and Ellipse.
Intersection (INT)	Used to select the Intersection point of where two objects Bisect on the same plane.	Any objects that Bisect with themselves or with other objects.
Insertion (INS)	Used to select the Insertion point of an object.	Blocks, Text and Attributes.
Perpendicular (PER)	Used to select the Perpendicular point of where two objects would meet perpendicularly.	All objects have a perpendicular point in relation to any object.
Tangent (TAN)	Used to select the Tangent point of a circular based object.	Circle, Arc, Donut, and Ellipse.
Nearest (NEA)	Used to select any point on an object.	All objects have a nearest point in relation to itself.
None (NON)	Used to clear all Osnaps for just the current command prompt.	All objects are affected by it.

Object Snap (Osnap) - Clear All

Clear all is used to reset all the Object Snaps that are currently set.

Note: More than one Object Snap can be on at one time.

Notes:

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Object Snap (Osnap) - Aperture Size

Controls the size of the Aperture Box. The Aperture Box is used to calculate the working Object Snap. The working Object Snap is controlled only by the objects that are found in the Aperture Box.

> **Tip:** If more than one Object Snap falls inside the Aperture Box AutoCAD LT will try to set the correct working Object Snap. If the correct working Object Snap isn't set you can use the TAB key to cycle through the available Object Snaps for the location your Crosshairs are at.

Object Snap (Osnap) - AutoSnap Tab

The AutoSnap Tab (Fig. 7.20) controls the properties of the Object Snaps themselves. These properties control whether or not there are a visual recognition of what type of Object Snap you are currently trying to use.

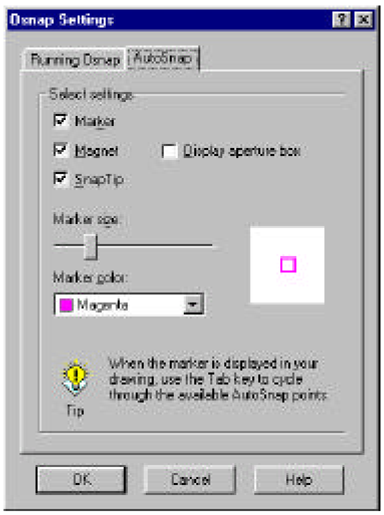

Fig. 7.20

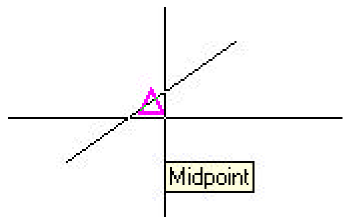

Fig. 7.21

AutoSnap Tab - Marker

The Marker (Fig. 7.21) is a colored indicator that appears at the point where the Object Snap will be activated. There is a different marker for each type of Object Snap.

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AutoSnap Tab - Magnet

The Magnet if turned on brings the Crosshairs to the snap point.

AutoSnap Tab - Tooltip

The Tooltip (Fig. 7.21) is a small label that appears on the screen to help you identify the Object Snap that is currently active. This helps you get used to the graphical marker for the Object Snaps.

AutoSnap Tab - Display Aperture

The Display Aperture controls the visibility of the Aperture Box. Figure 7.22 has the Aperture Box off and Figure 7.23 shows the Aperture Box on.

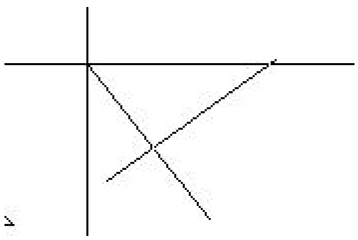

Fig. 7.22

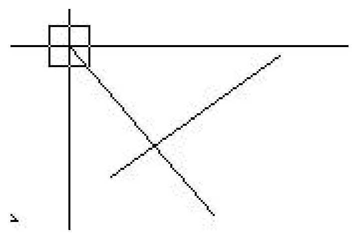

Fig. 7.23

AutoSnap Tab - Marker Size

Controls the size of the Object Snap marker.

AutoSnap Tab - Marker Color

Controls the color of the Object Snap marker.

Notes:

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PolarSnap Settings

PolarSnap (Fig. 7.18) is used to aid in creating 2D objects in AutoCAD LT. It is used to help in creating more accurate drawings and requiring less interaction by the user for typing values. Now lets take a look at the Polar Snaps in AutoCAD LT and how they help in drawing.

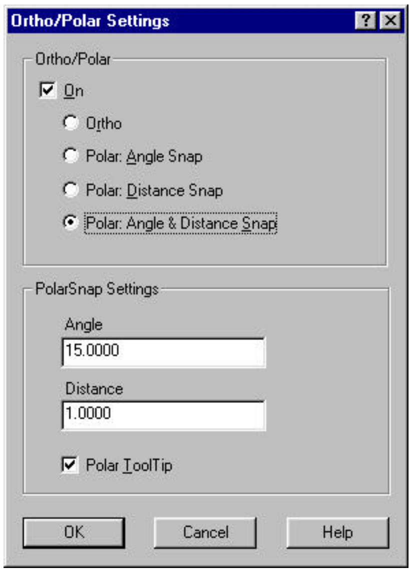

Fig. 7.18

Command Access	
Command Line	POLAR + Enter
Toolbar	_STANDARD 
Statusbar	Double Click ORTHO
Menubar	TOOLS>>PolarSnap Settings...

Tip: You can type-in POLAR at the command line with an apostrophe (') to use it transparently, otherwise the other options already provide that method automatically.

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PolarSnap - Ortho/Polar

The Ortho/Polar is used to determine what mode that Ortho and/or Polar are running in currently.

PolarSnap - Ortho/Polar - On

The On option toggles on the ability to use Ortho or PolarSnapping in the drawing.

PolarSnap - Ortho/Polar - Ortho

The Ortho option locks AutoCAD input to vertical and horizontal input from the mouse.

PolarSnap - Ortho/Polar - Polar: Angle Snap

The Polar: Angle Snap only allows angles that fall on the increment specified. The restriction starts from the last point that you just picked.

PolarSnap - Ortho/Polar - Polar: Distance Snap

The Polar: Distance Snap only allows distances that fall on the increment specified. The restriction starts from the last point that you just picked.

PolarSnap - Ortho/Polar - Polar: Angle & Distance Snap

The Polar: Angle & Distance Snap only allows angles and distances that fall on the increment specified. The restriction starts from the last point that you just picked.

PolarSnap - PolarSnap Settings - Angle

The Angle used for increments for of only the Polar:Angle Snap and Polar: Angle & Distance.

PolarSnap - PolarSnap Settings - Distance

The Distance used for increments for of only the Polar:Distance Snap and Polar: Angle & Distance.

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PolarSnap - PolarSnap Settings - Polar ToolTip

The Polar ToolTip is an active link and read out of your angle and distance you are currently at from the original point selected.

Note: PolarSnaps can help you to draw2D drawings faster and more accurately with out having to enter information at the command line.

Note: Never try to load an increment that is very small in size. If the size is to small it will become harder to use and therefore may not be as much of a return for as much time as you would need to investment in it.

Notes:

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PolarSnap Practice

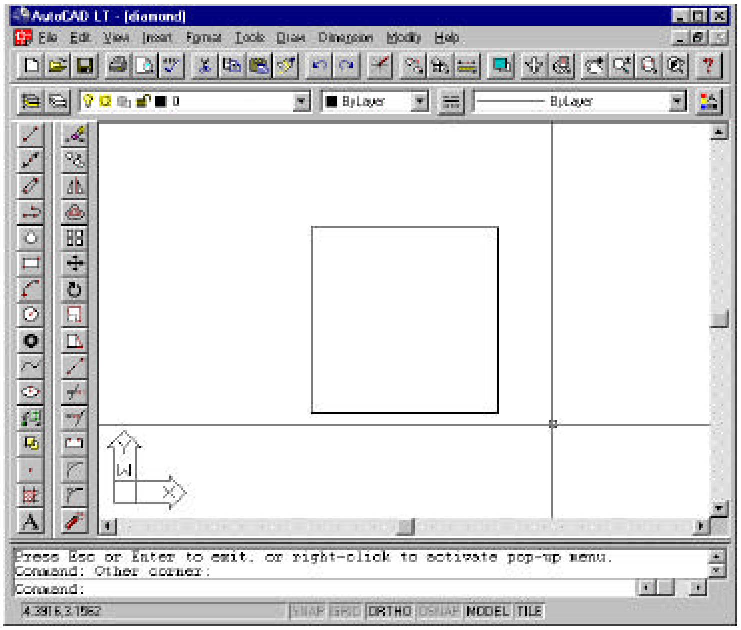

Fig. 7XX

During this practice you will be using PolarSnap with the Line command. Don't be to concerned about the how to use the Line command yet we will cover it a little bit later. 5 mins.

1. Start a new drawing from **Scratch** with **English** highlighted. This will ensure that all the settings are setup to AutoCAD LT standards.

2. Start up the **POLAR** command.

3. Set Ortho/Polar **On**.

4. Now with Ortho/Polar on select the option Polar: Angle & Direction to being the current setting.

5. Set the Angle to 15, and the Distance to 2.

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6. If the Polar ToolTip isn't checked, make it checked (On), and click the OK button to accept the changes.

7. Start up the **LINE** command.

8. Select your first point for the line. Notice after you select your first point that as you move the mouse the drag line gets shorted or lengthened as it rounds up or down for the distance and snaps on every angle that is based off of 0 and just add 15 every time.

9. Drag your mouse to the right of your original point. You should notice that it is creating a line of 2 inches and then adds 2 more and so on based on the distance you are from the original point. If you let your mouse sit long enough a **ToolTip** will appear and tell you your current distance and the angle you are at. **Draw** your first segment to the right of the original point a **distance of 2** and **angle of 0**.

10. Draw segment two up 2 units and at an angle of 90 degrees from the second point of segment one.

11. Draw segment three left 2 units and at an angle of 90 degrees from the second point of segment two.

12. Draw segment four up 2 units and at an angle of 90 degrees from the second point of segment three.

13. Type-in **C** at the command line to Close the Line and press Enter .

Try using other angles and distances to draw a different type of shape other than a box. Can you draw an Octagon easy enough?

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LINE Command

Command Access	
Command Line	LINE + Enter
Toolbar	DRAW 
Menubar	DRAW>>Line

The Line command is used to draw single lines or segments in a group of lines to form a shape. The Lines is the simplest of the 2D objects that can be drawn.

Command Options	
From Point:	First point of the Line.
To Point:	Second point of the Line.
Close	Allows the user to Close a Line that has two or more Line segments drawn.
Undo	Allows the user to Undo the last segment created.

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LINE Practice

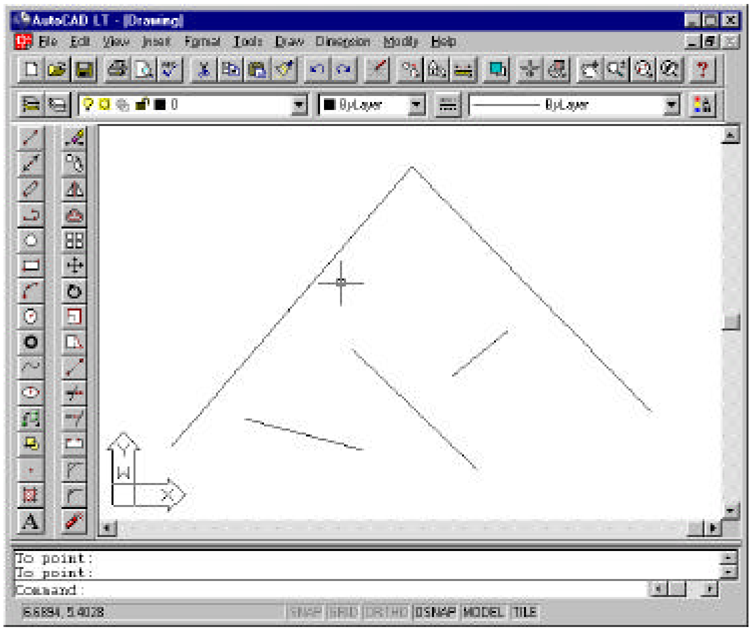

Fig. 7.25

During this practice you will be working with the Line command. 2 mins.

1. Start a new drawing from **Scratch** with **English** highlighted. This will ensure that all the settings are setup to AutoCAD LT standards.

2. Start up the **LINE** command.

3. Select the start point for your Line, and then select your endpoint.

4. Press Enter to end the Line command.

Take a little bit of time and draw some continued Line sections and use the close and undo command options to see how they work.

Notes:

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XLINE (or Construction Line) Command

Command Access	
Command Line	XLINE + Enter
Toolbar	DRAW
Menubar	DRAW>>Construction Line

The Xline command is used to draw a single line that is drawn in two directions from the point of origin. The Xline is a special type of Line that when you try to zoom out it will just increase in size to reach from edge to edge of the screen.

Command Options	
From Point:	First point of the Line.
Through Point:	Used to specify the direction of the Xline.
Hor (Horizontal)	Will only draw horizontal lines will active.
Ver (Vertical)	Will only draw vertical lines will active
Ang (Angle)	Will only draw lines at the specified angled will active
>>Reference/<Enter angle (0)>:	Specify an angle by either typing it in at the Command Line or selecting two points.
Bisect	Allows the user to specify a point in which you would like the Xline to go through.
>>Angle vertex point:	Point in which the Xline will Bisect with.
>>Angle start point:	Start point of reference for Bisecting Xline.
>>Angle end point:	Endpoint of reference for Bisecting Xline.
Offset	Allows the user to create an Xline from the offseting of an existing object in the drawing.
>>Offset distance of Through	The distance used for offseting from the existing object.
>>Select a line object:	The object used to offset from.
>>Side to offset?	Select the side of the object to create the offseted line.

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XLINE Practice

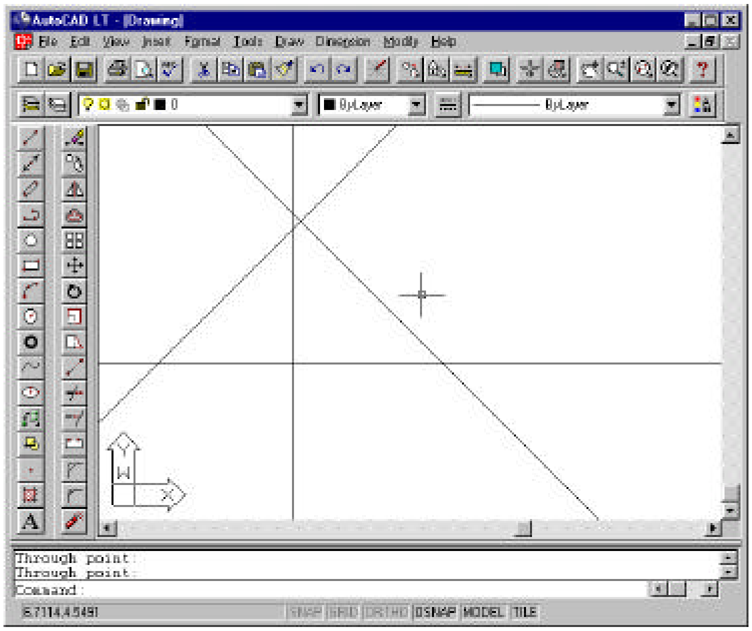

Fig. 7.26

During this practice you will be working with the Xline command. 2 mins.

1. Start a new drawing from **Scratch** with **English** highlighted. This will ensure that all the settings are setup to AutoCAD LT standards.

2. Start up the **XLINE** command.

3. Select the start point for your Xline, and then select your through point.

4. Press Enter to end the Xline command.

Take a little bit of time and draw some more Xlines with some of the other command options to see how they work.

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RAY Command

Command Access	
Command Line	RAY + Enter
Menubar	DRAW>>Ray

The Ray command is used to draw a single line that is drawn in a single direction from the point of origin. The Ray is a special type of Line that when you try to zoom out it will just increase in size to reach from origin to edge of the screen. You can think of a Ray as half of an Xline.

Command Options	
From Point:	First point of the Line.
Through Point:	Used to specify the direction of the Ray.

Notes:

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RAY Practice

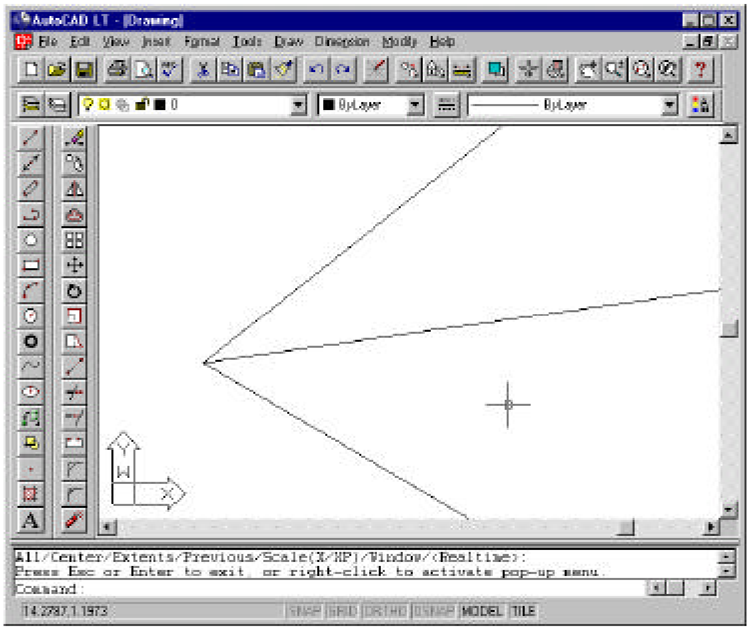

Fig. 7.27

During this practice you will be working with the Ray command. 2 mins.

1. Start a new drawing from **Scratch** with **English** highlighted. This will ensure that all the settings are setup to AutoCAD LT standards.

2. Start up the **RAY** command.

3. Select the start point for your Ray, and then select your through point.

4. Press Enter to end the Ray command.

Take a little bit of time and draw some additional Rays in your drawing.

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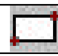
Rectangle Command

Command Access	
Command Line	REC + Enter
Toolbar	DRAW 
Menubar	DRAW>>Rectangle

The Rectangle command is used to draw single closed box. The box doesn't have to be unequal in size. You can have all four sides equal to draw a square, the name can be miss leading for this command.

Command Options	
First corner:	First corner of the Rectangle.
Other corner:	Opposite corner of the Rectangle.
Chamfer	Allows the user to enter a chamfer size for the corners of the Rectangle.
>>First chamfer distance for rectangles:	The size of the first side of the chamfer.
>>Second chamfer distance for rectangles:	The size of the second side of the chamfer.
Elevation	Controls the height at which the Rectangle is drawn at in the Z plane..
>>Elevation for rectangles:	Current Elevation height.
Fillet	Allows the user to enter a fillet size for the corners of the Rectangle.
>>Fillet radius for rectangles:	Current radius for Fillets.
Thickness	Allows the user to enter a thickness for all new Rectangles that are drawn.
>>Thickness for rectangles:	Current Thickness.
Width	Allows the user to specify the width of the lines that make up the outer lines of the Rectangle.
>>Width for rectangles:	Current Width.

Notes:

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Rectangle Practice

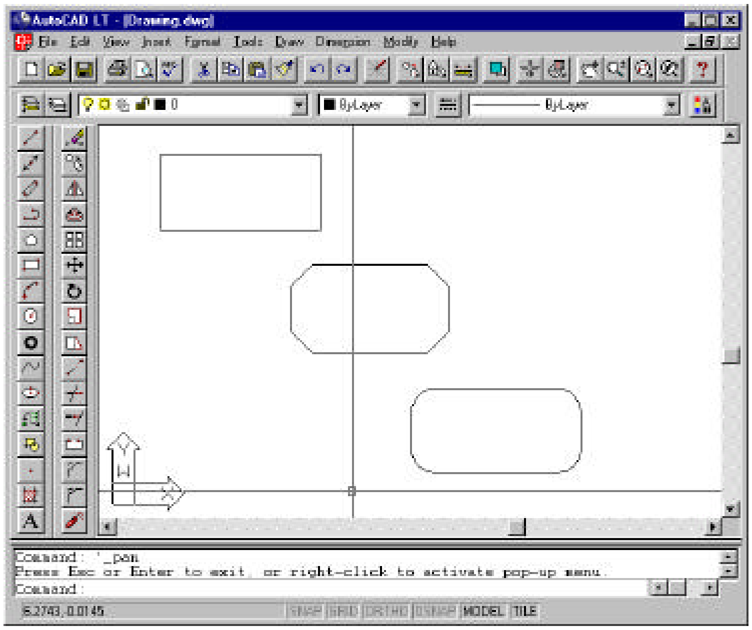

Fig. 7.28

During this practice you will be working with the Rectangle command. 2 mins.

1. Start a new drawing from **Scratch** with **English** highlighted. This will ensure that all the settings are setup to AutoCAD LT standards.

2. Start up the **RECTANGLE** command.

3. Select the first corner for your Rectangle, and then select your other corner.

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During this practice you will be working with the Rectangle command and setting the settings to create Chamfers on the corners of the Rectangle. 2 mins.

1. Start a new drawing from **Scratch** with **English** highlighted. This will ensure that all the settings are setup to AutoCAD LT standards.

2. Start up the **RECTANGLE** command.

3. At the prompt instead of picking your point, type-in **C** or **Chamfer**. This will bring you into the Chamfer settings.

4. Enter the **0.5** for both the First and Second chamfer distances.

5. Once back to the main options line draw a new Rectangle based on your new options that you just specified. You should now have a rectangle that has the corners cutoff at 45 degree angles.

During this practice you will be working with the Rectangle command and setting the settings to create Fillets on the corners of the Rectangle. 2 mins.

1. Start a new drawing from **Scratch** with **English** highlighted. This will ensure that all the settings are setup to AutoCAD LT standards.

2. Start up the **RECTANGLE** command.

3. At the prompt instead of picking your point, type-in **F** or **Fillet**. This will bring you into the Fillet settings.

4. Enter the **0.5** for the Radius of your Fillet.

5. Once back to the main options line draw a new Rectangle based on your new options that you just specified. You should now have a rectangle that has radius corners.

Take a little bit of time and try to figure out how to clear your Fillet setting, because all new Rectangles will be drawn with the Fillet setting.

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Polygon Command

Command Access	
Command Line	POL + 
Toolbar	DRAW 
Menubar	DRAW>>Polygon

The Polygon command is used to draw closed objects with a minimum of 3 sides and a maximum of 1024 sides on it. All sides of the Polygon are uniform in size and angles.

Command Options	
Number of sides:	Specify the number of equal sides the Polygon will contain.
Center of Polygon:	Allows the user to specify the center of the Polygon.
>>Inscribed	Specifies a circle, which determines the boundary for the points of the Polygon.
>>Circumscribed	Specifies a circle, which fits inside of the Polygon.
Edge:	Allows the user to specify two points for the first edge of the Polygon. This first edge is then used to generate the rest of the Polygon.

Note: Polygons are sometimes hard to work with in the fact that most people don't use them enough. Just take your time when working with Polygons and you should be just fine in getting them to turn out right.

Notes:

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Polygon Practice

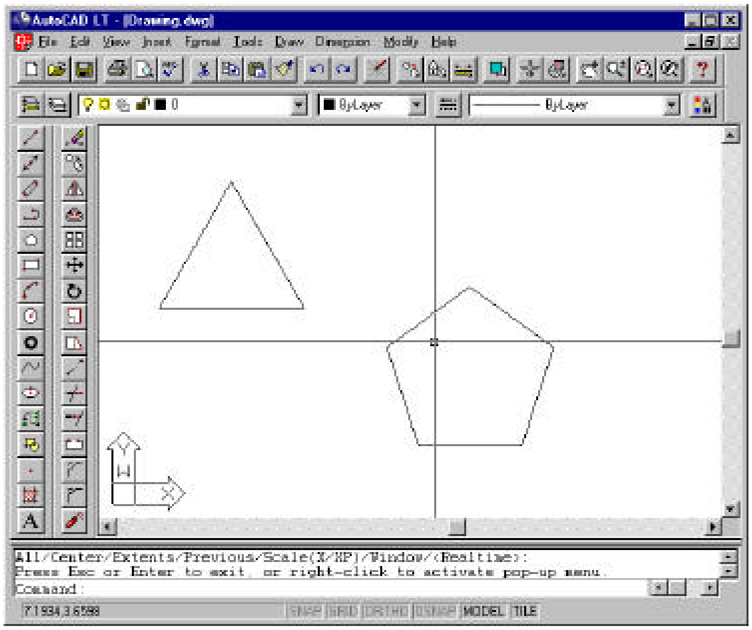

Fig. 7.29

During this practice you will be working with the Polygon command. 2 mins.

1. Start a new drawing from **Scratch** with **English** highlighted. This will ensure that all the settings are setup to AutoCAD LT standards.

2. Start up the **POLYGON** command.

3. Select **3** for the number of sides and press Enter to continue.

4. Select a point on the screen to use **Center of Polygon**.

5. Choose what ever method of Scribe you would like, (I for Inscribed or C for Circumscribed) and press Enter to continue. The select the size of your triangle by picking a point or entering it.

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During this practice you will be working with the Polygon command and using Edge to create the object. 2 mins.

1. Start a new drawing from **Scratch** with **English** highlighted. This will ensure that all the settings are setup to AutoCAD LT standards.

2. Start up the **POLYGON** command.

3. Select **5** for the number of sides and press Enter to continue.

4. Type in **E** for Edge and press Enter to continue.

5. Select a point on the screen to start the creation of your Polygon

6. The select the size of your pentagon by picking a point or entering a distance to create the size of your edges.

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Circle Command

Command Access	
Command Line	CIRCLE + Enter
Toolbar	DRAW
Menubar	DRAW>>Circle

The Circle command is used to draw a circular object.

Command Options	
Center point:	Specify the center point of the circle object to be created.
>>Radius	Allows the user to specify the radius of the circle to be created.
>>Diameter:	Allows the user to specify the diameter of the circle to be created.
3p or Three point:	Allows the user to specify a circle object based on three select points in the drawing.
>>First point:	First point of the circle object.
>>Second point:	Second point of the circle object.
>>Third point:	Third point of the circle object.
TTR or Tangent, Tangent, Radius:	Allows the user to specify a circle object based on two tangent points and a radius.
>>Enter Tangent spec:	First tangent point to be used for the circle.
>>Enter second Tangent spec:	Second tangent point to be used for the circle.
>>Radius:	The Radius to be used for the newly create circle.

Notes:

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Circle Practice

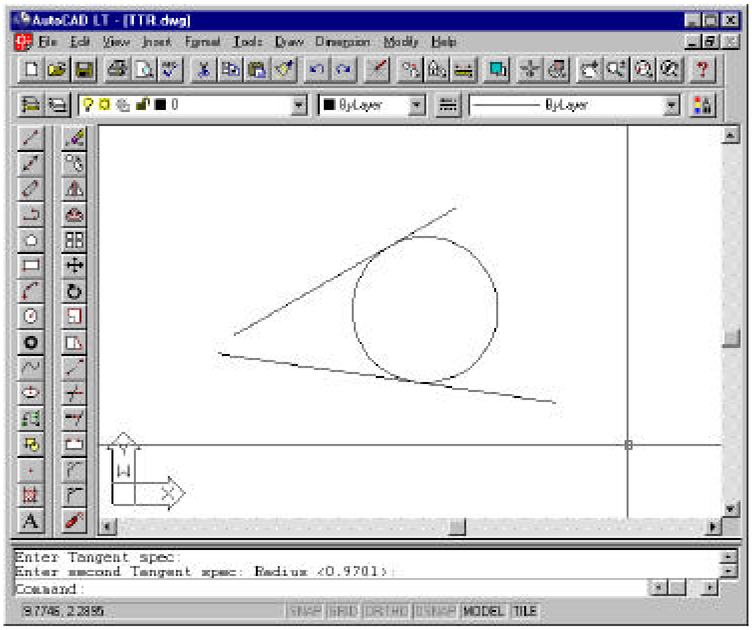

Fig. 7.30

During this practice you will be working with the Circle command. 2 mins.

1. Start a new drawing from **Scratch** with **English** highlighted. This will ensure that all the settings are setup to AutoCAD LT standards.

2. Start up the **CIRCLE** command.

3. Select a point on the screen for the **Center Point** of the Circle.

4. Select a point on the screen for the outer edge of the Circle or type in a number for the **Radius** of the Circle. If you want to you could type-in **D** and then enter a Diameter instead of a Radius.

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During this practice you will be working with the Circle command and the 3P option. 2 mins.

1. Start a new drawing from **Scratch** with **English** highlighted. This will ensure that all the settings are setup to AutoCAD LT standards.
2. Start up the **CIRCLE** command.
3. Type-in **3P** at the command line and press Enter to continue.
4. Select a **First** point on the screen for the Circle.
5. Select a **Second** point on the screen for the Circle.
6. Select the **Third**, and final first point on the screen for the Circle.

During this practice you will be working with the Circle command and the TTR option. 2 mins.

1. Open the drawing called **TTR.DWG**. This was a pre-setup drawing that contains two lines in it.
2. Start up the **CIRCLE** command.
3. Type-in **TTR** at the command line and press Enter to continue.
4. At the **Enter Tangent spec:** prompt select the top most Line. When you go above the Line you should see the Tangent Object Snap marker appear. It looks like a circle with a line next to it.
5. At the **Enter second Tangent spec:** prompt select the bottom most Line. When you go near it, the Tangent Object Snap should also appear.
6. At the **Radius** prompt, either accept the radius value or enter a new value and press Enter.

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Donut Command

Command Access	
Command Line	DONUT + Enter
Toolbar	DRAW
Menubar	DRAW>>Donut

The Donut command can be used for drawing a couple different looking objects. The donut is either a partially or a fully filled in Circle based on a couple settings.

Command Options	
Inside Diameter:	Specify the center point of the circle object to be created.
Outside Diameter:	Allows the user to specify the radius of the circle to be created.
Center of Donut:	Allows the user to specify the diameter of the circle to be created.

Tip: If you set the Inside Diameter to zero you can create a completely filled in circle.

Note: You can enter an Inside Diameter greater than the Outside Diameter, but AutoCAD LT will invert the values for you so it works correctly.

Notes:

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Donut Practice

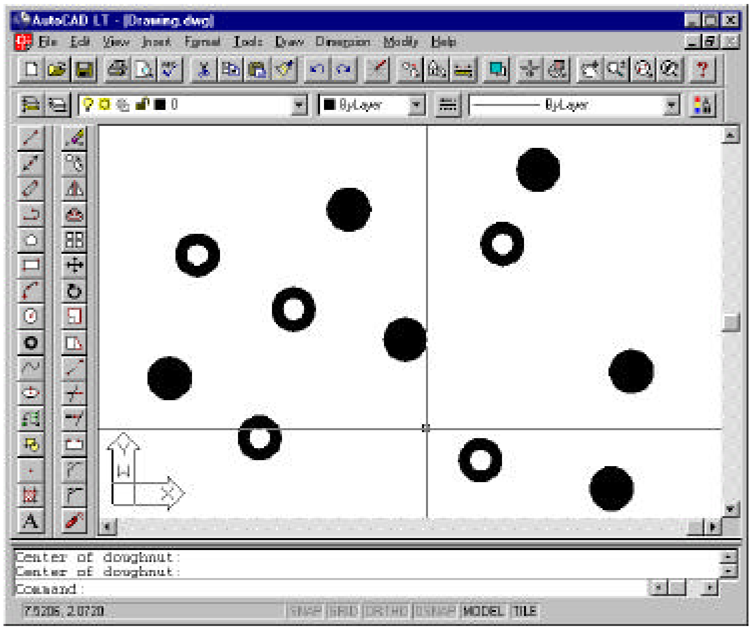

Fig. 7.31

During this practice you will be working with the Donut command. 2 mins.

1. Start a new drawing from **Scratch** with **English** highlighted. This will ensure that all the settings are setup to AutoCAD LT standards.

2. Start up the **DONUT** command.

3. At the **Inside Diameter:** prompt enter a value of **0.5000**.

4. At the **Outside Diameter:** prompt enter a value of **1.0000**.

5. Select a **Center point** for the Donut on the screen, and there you have it.

Notes:

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During this practice you will be working with the Donut command. 2 mins.

1. Start up the **DONUT** command.

2. At the **Inside Diameter:** prompt enter a value of **0.0000**.

3. At the **Outside Diameter:** prompt enter a value of **1.0000**.

4. Select a **Center point** for the Donut on the screen, and there you have it. Notice this time the donut is completely filled in.

5. Lets now turn **Solid Fill** Off. Remember Solid Fill is part of the **Drawing Aids**. After turning off Solid Fill remember to use the command **REGEN**. Did you see what happened to the Donut that had an Inside Diameter versus the Donut that had an Inside Diameter of zero.

6. Go back and turn **Solid Fill** back On, and perform another **REGEN**.

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Arc Command

Command Access	
Command Line	ARC + Enter
Toolbar	DRAW 
Menubar	DRAW>>Arc

The Arc command can be used for drawing a couple different looking objects. An Arc shares the same properties that of a circle, except that an Arc has a start point and an endpoint.

Command Options	
Start point:	Specify the Start point of the Arc object to be created.
>>Second point:	Specify the Second point of the Arc object to be created. This point is usually the mid point of the arc.
>>Center:	Specify the Center point of the Arc object to be created.
>>End or End point:	Specify the End point of the Arc object to be created.
>>Angle:	Specify the amount of angle the Arc has.
>>Include Angle:	Specify the angle in which the bulge of the Arc will be created with.
Center:	Specify the Center point of the Arc object to be created.

Note: There are a number of sub-option combinations that you can use to create an Arc with. To get used to the Arc command it is recommended to use the **DRAW>>Arc** from the menubar.

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Arc Practice

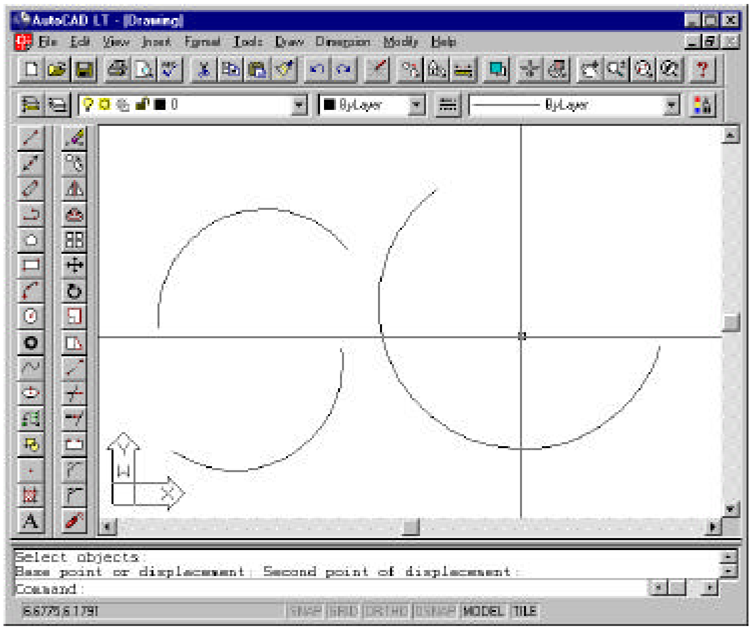

Fig. 7.32

During this practice you will be working with the Arc command and will be creating a 3-point arc. 2 mins.

1. Start a new drawing from **Scratch** with **English** highlighted.

2. Start up the **ARC** (**3 Points** from the menubar) command.

3. At the **Center/<Start point>:** prompt, select a point on the screen.

4. At the **Center/End/<Second point>:** prompt, select another point in the drawing.

5. At the **End point:** prompt, select the final point for the Arc in the drawing.

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During this practice you will be working with the Arc command and will be creating a Center, Start, End arc. 2 mins.

1. Start up the **ARC** (**Center, Start, End** from the menubar) command.
2. At the **Center/<Start point>: _c Center:** prompt, select a point on the screen for the center point.
3. At the **Start point:** prompt, select a point on the screen for the start point.
4. At the **Angle/<End point>:** prompt, select a point on the screen for the endpoint.

During this practice you will be working with the Arc command and will be creating a Start, Center, End arc. 2 mins.

1. Start up the **ARC** (**Start, Center, End** from the menubar) command.
2. At the **Center/<Start point>:** prompt, select a point on the screen for the start point.
3. At the **Center/End/<Second point>: _c Center:** prompt, select a point on the screen for the center point.
4. At the **Angle/<End point>:** prompt, select a point on the screen for the endpoint.

Take a couple of minutes and try some of the other Arc options from the DRAW>>Arc menubar that we haven't covered in the Examples. Find out what the Continue option is.

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Ellipse Command

Command Access	
Command Line	ELLIPSE + Enter
Toolbar	DRAW
Menubar	DRAW>>Ellipse

The Ellipse command is used to draw an ellipse or and elliptical arc object.

Command Options	
Arc:	Specify the center point of the circle object to be created.
Center:	Allows the user to specify the radius of the circle to be created.
Axis endpoint 1:	Specify the first end point of the ellipse object to be created.
>>Axis endpoint 2:	Specify the second end point of the ellipse object to be created.
>>Other axis distance:	Specify the distance of the secondary axis. This number is calculated from the center of the Ellipse.
>>Rotation:	Specify the major to minor axis ratio.
>>start angle:	Specify the start point/ angle of the Elliptical Arc.
>>end angle:	Specify the end point/ angle of the Elliptical Arc.
>>Include:	Specify an Angle of the Elliptical Arc versus selecting and endpoint.
>>Parameter	Uses the same input that Start Angle does, but produces a different result based on a mathematical formula.

Notes:

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Ellipse Practice

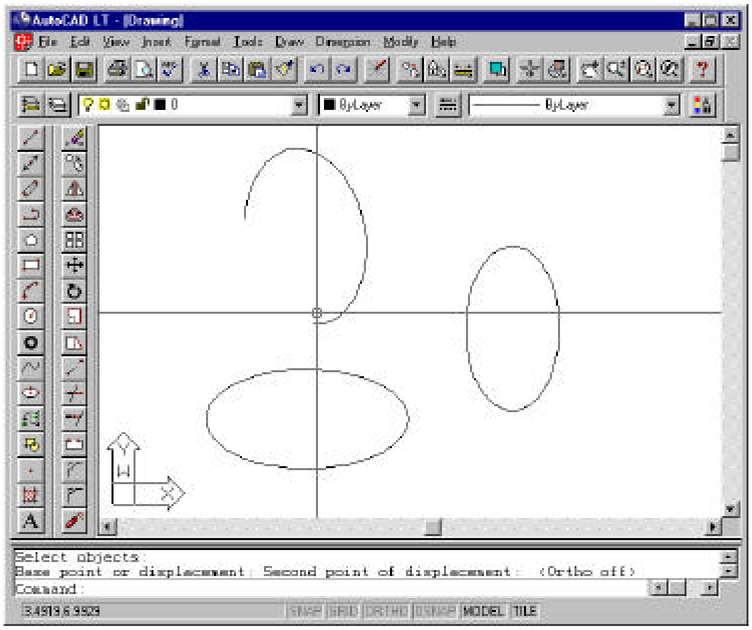
Fig. 7.33

During this practice you will be working with the Ellipse command and will be creating a Center Ellipse. 2 mins.

1. Start a new drawing from **Scratch** with **English** highlighted.

2. Start up the **ELLIPSE** (**Center** from the menubar) command.

3. At the **Center of ellipse:** prompt, select a point on the screen for the center of the Ellipse.

4. At the **Axis endpoint:** prompt, select another point in the drawing. This point will represent either the major or minor axis.

5. At the **<Other axis distance>/Rotation:** prompt, select the final point for the Ellipse. This point will represent either the major or minor axis based on the distance from the center of the Ellipse.

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During this practice you will be working with the Ellipse command and will be creating an Elliptal Arc. 2 mins.

1. Start up the **ELLIPSE** (**Arc** from the menubar) command.

2. At the **Center of ellipse:** prompt, select a point on the screen for the center of the Ellipse.

3. At the **Axis endpoint:** prompt, select another point in the drawing. This point will represent either the major or minor axis.

4. At the **<Other axis distance>/Rotation:** prompt, select the final point for the Ellipse. This point will represent either the major or minor axis based on the distance from the center of the Ellipse.

5. At the **Parameter/<start angle>:** prompt, enter an angle of 45. So type-in **45** at the command line and press Enter . The angle depends on how you select your axis points.

6. At the **Parameter/Included/<end angle>:** prompt, enter an angle of 180. So type-in **180** at the command line and press Enter . The send angle also depends on how you selected your axis points.

Take a couple of minutes and try the other Ellipse option from the DRAW>>Ellipse menubar.

Note: You will find yourself commonly drawing a true Arc over and Elliptical Arc, but you never know.

Tip: If you need to create an Ellipse that you can explode make sure the system variable PELLIPSE is set to 1 and not 0. You can't explode a standard Ellipse. To change PELLISPE simply type-in PELLISPE at the command line and give it a value of 1 and then press {ENTER}.

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Point Command

Command Access	
Command Line	POINT + Enter
Toolbar	DRAW
Menubar	DRAW>>Point

The Point command is used to add Point (or Node) object to the drawing. Most Point objects that are added to the drawing are simply used as a reference point. By far this is the simplest object that can be drawn in the drawing.

Command Options	
Point:	Origin point for the Point object to be drawn at.

Note: To change the style of the Point object you must use the DDptype command.

Note: If you change Point settings in the DDptype command you must perform a REGEN in order to see any changes.

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Point Practice

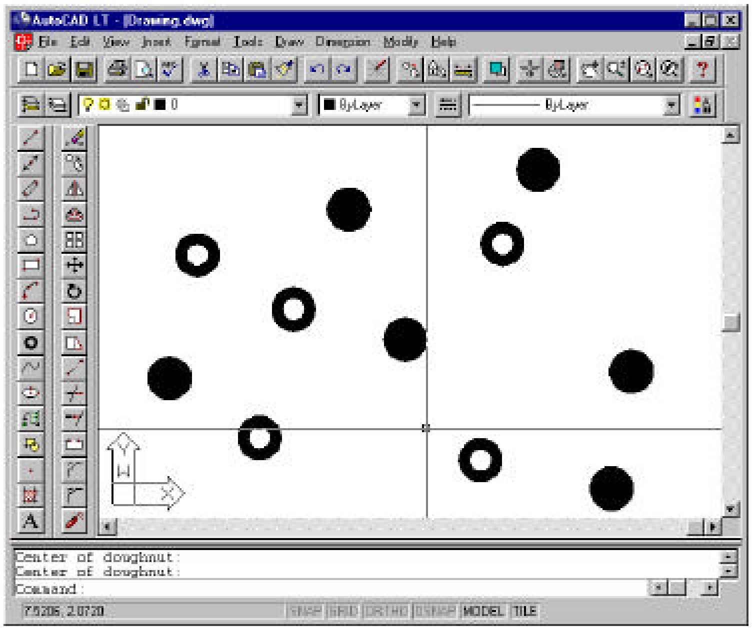

Fig. 7.35

During this practice you will be working with the Point and DDptype commands. 5 mins.

1. Start a new drawing from **Scratch** with **English** highlighted. This will ensure that all the settings are setup to AutoCAD LT standards.

2. Start up the **DDPtype** command.

3. Select the Fourth Point Style from the left side in the second row to be the active Point Style.

4. Set the **Point Size** to **1** and select the **Absolute Units** to being the active Point size.

5. Select the **OK** button to accept the changes.

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6. Run the **REGEN** command so it refreshes the drawing.

7. Start up the **POINT** command.

8. Select the point in the drawing where you would like to place a Point at.

9. Now that there is a **Point** in the drawing, use the **DDptypes** command again.

10. Select the First Point Style from the left side in the first row to be the active Point Style.

11. Keep the **Point Size** and **Absolute Unit** values the same.

12. Select the **OK** button to accept the changes, and run the REGEN command again so it refreshes the drawing. Notice that you didn't have to redraw the Point objects to modify their appearance.

Try changing the Point Style, Size and Units of the command DDptypes a couple more times and see what results you get.

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DText Command

Command Access	
Command Line	DTEXT + Enter
Menubar	DRAW>>Single Line Text

The DText command is used to create a single line of text in the drawing. The DText command is an improved version of the original Text Command because you can see what you are typing as you type your text string in the drawing.

Command Options	
Start point:	Specify the start point of the Text object to be created.
Justify:	Allows the user to specify the justification of the new Text object. (See Justification Chart)
Style:	Allows the user to specify the Text Style for the new Text object.
>>Style name (or ?) <STANDARD>:	Allows the user to enter a different Text Style. The Text style must first exist in the drawing.
>>Height:	Specify the height of the Text.
>>Rotation angle:	Specify the angle at which the Text will be created at.
>>Text:	Enter your text that you would like placed in the drawing here.

Note: When you are at the **Style name (or ?) <STANDARD>:** prompt, you can type-in a **?** and then press {ENTER}, and at the next prompt just press {ENTER} to have a list placed on the screen for you of the currently setup Text Styles.

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Justification	
Align	Both Text Height and Orientation by selecting the endpoints of the baseline for the Text.
Fit	Specifies that the Text will fit between the given two endpoints and the provided height.
Center	Creates the Text based from the Baseline Center of the Text.
Middle	Creates the Text based from the Middle of the Text.
Right	Creates the Text based from the Right on the Baseline of the Text.
TL	Creates the Text based from the Top Left of the Text.
TC	Creates the Text based from the Top Center of the Text.
TR	Creates the Text based from the Top Right of the Text.
ML	Creates the Text based from the Middle Left of the Text.
MC	Creates the Text based from the Middle Center of the Text.
MR	Creates the Text based from the Middle Right of the Text.
BL	Creates the Text based from the Bottom Left of the Text.
BC	Creates the Text based from the Bottom Center of the Text.
BR	Creates the Text based from the Bottom Right of the Text.

Note: By default the insertion point of Text is Baseline Left, which isn't an option under Justification. So if you don't want to make a change you have to exit the DText command and re-enter it.

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You can use control codes and features to your DText text strings. Below is a list of these control codes.

Control Codes	
%%o	Overscores the text
%%u	Underlines the text.
%%d	Places a Degree Symbol in its place.
%%p	Places a Plus/Minus (Tolerance) Symbol in its place.
%%c	Places a Diameter Symbol in its place.
%%%	Places a Percent Symbol in its place.

Note: You can use multiple control characters on a single piece of Text.

Tip: You can start and stop a special effect by starting the effect and ending the effect with the same control code in a string of text.

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DText Practice

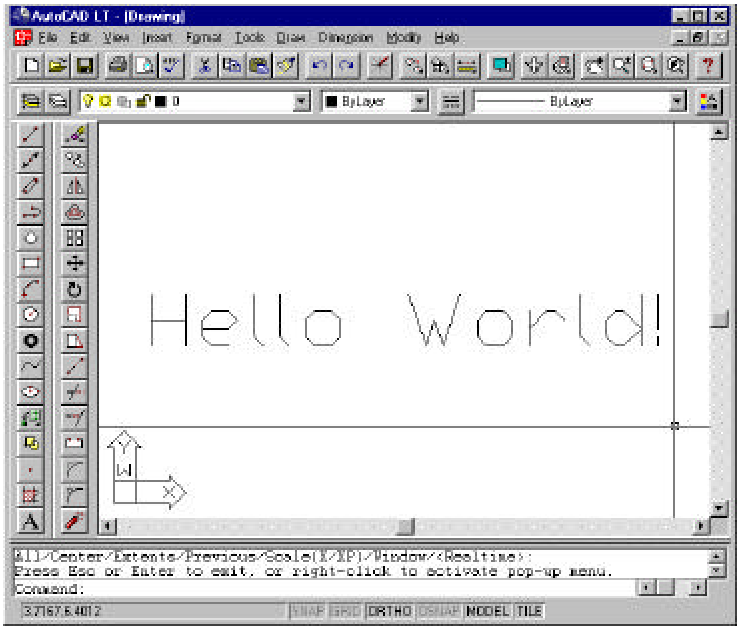

Fig. 7.36

During this practice you will be working with the DText command. 2 mins.

1. Start a new drawing from **Scratch** with **English** highlighted. This will ensure that all the settings are setup to AutoCAD LT standards.

2. Start up the **DText** command.

3. Select a point in the drawing for your starting point.

4. Press Enter twice to accept the defaults of both the Text Height and Rotation.

5. At the **Text:** prompt, type-in **Hello World!**, and press Enter twice to exit the DText command.

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During this practice you will be working with the DText command w/ a control codes. 2 mins.

1. Start up the **DText** command.
2. Select a point in the drawing for your starting point.
3. Press Enter twice to accept the defaults of both the Text Height and Rotation.
4. At the **Text:** prompt, type-in **%%uHello World!**, and press twice Enter to exit the DText command. Notice that all of Hello World! is underlined.
5. Repeat Steps 1 - 4 and use **%%uHello%%u World!** as the text string. Notice that this time only the Hello was underlined and not the rest of the string.

During this practice you will be working with the DText command and the Fit justification. 5 mins.

1. Start up the **DText** command.
2. At the **Justify/Style/<Start point>:** prompt, type-in **J** and press Enter to continue.
3. Select a point in the drawing for your first point of your baseline point.
4. When the Justification options come up type-in **F** or **Fit**, and press Enter to continue.
5. Select another point in the drawing for your second point of your baseline point.
6. Press Enter accept the default of the Text Height. The Rotation of the Text is already calculated by the points that you just selected.
7. At the **Text:** prompt, type-in **Hello World!**, and press Enter exit the DText command.
8. Repeat steps 1 - 7 again except pick two points that have a greater or smaller distance between them than your original points. After completing the step 6, did your Text get squashed together or stretched out of shape.

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MText Command

Command Access	
Command Line	MTEXT + Enter
Toolbar	DRAW 
Menubar	DRAW>>Paragraph Text

The MText command is used to create a Paragraph or block of Text. Unlike DText which can create Paragraph looking Text, but still acts like individual text lines, MText remains together as one group and can't be taken apart unless you Explode it. The MText (Fig. 7.37) command also has a special Text Editor to edit and make text changes in.

Command Options	
Specify first corner:	Specify the first corner of the area that you would like to added the Text to.
Specify opposite corner:	Specify the opposite corner from your original corner to finalize the Text area.
Height:	Specify the height of the Text.
Style:	Allows the user to enter a different Text Style. The Text style must first exist in the drawing.
Justify:	Specify the Justification of the Text.
Rotation angle:	Specify the angle at which the Text will be created at.
Width:	Specify the Width of the Paragraph Text object.

Note: When you are at the **Enter style name (or '?') <STANDARD>:** prompt, you can type-in a **?** and then press {ENTER}, and at the next prompt just press {ENTER} to have a list placed on the screen for you of the currently setup Text Styles.

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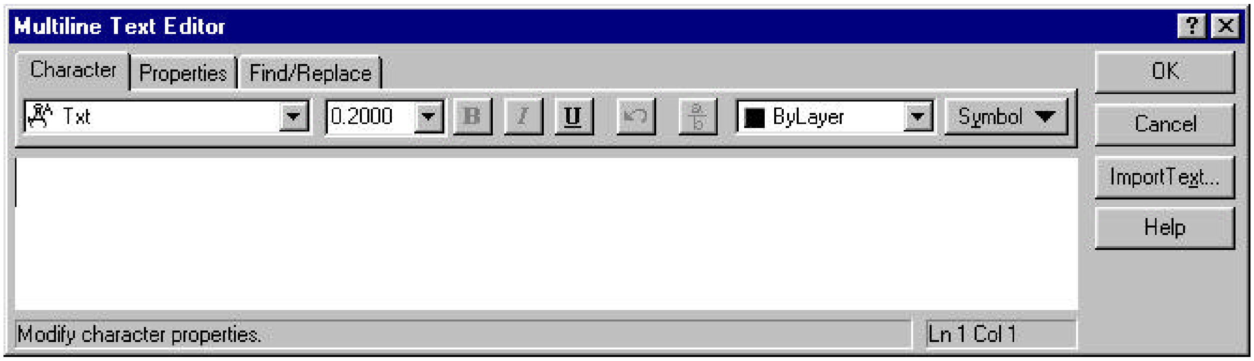

Fig. 7.37

MText - Edit Box

The Edit box is where you will be doing all you entering for your text strings and characters.

MText - Character Tab

There are a variety of controls under the Character Tab that affect the appearance properties of the text. We will start on the left and go to the right. The first control on the left is a dropdown box of all available fonts that you can mix inside of your text string. The fonts can only be TrueType fonts and not Postscript or DOS fonts. The next control is a combo box that allows you to enter a new Text height or select on that has been used previously.

The next three buttons control the text formats of Bold, Italics, and Underline. The control to the right of the U or Underline is an Undo button that will allow you to undo the last format change. Next to the Undo button is a number stack button that will allow you to stack two numbers between a Forward Slash ("/"). The control just past the number stack is a dropdown that is used to specify text with a different color.

The last control under the Characters tab controls the insertion of the control codes and other special symbols that can be inserted from a font style. The Other... option allows you to insert a Copyright symbol or anything else that you can find that is in a font style.

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MText - Properties Tab

There are a variety of controls under the Properties Tab that affect the size and the way the MText object is created in AutoCAD LT. We will start on the left and go to the right. The first control on the left is a dropdown box of all available Text Styles in AutoCAD LT. The next control is a dropdown box that allows you to select the Justification of the MText object after it is created. The Justification simple identifies where the starting point of the text is and how the text is generated from that point.

The third control is a combo box that allows you to specify the width of the MText object. This number will affect how the Text will look like in the Edit Box. This is also a special width called (no wrap) will get you a true what you see is what you will get Text string. The final control is a combo box for the rotation of the MText object when it is created.

MText - Find/Replace Tab

There are a variety of controls under the Find/Replace Tab that allow you to search through your text string for a word or phrase it and replace it with some other string of text. We will start on the left and go to the right. The first control on the left is a combo box of all past words that you searched for, and the current word that you are using for your search. The next control is a button that will trigger the search for the string or phrase. The third control is a combo box like the Find in the sense that it will store previously replaced words and the current word that you are using to replace with. The next control is a button that will trigger the replacement of the string or phrase.

The last two controls help to near down the searchable text even more. The first one will force you to search for a text string that has the Same cases as the one in the Find box. The second is used to have the search match only what is typed-in the Find search field.

MText - Import Text

The Import Text option allows you to import text from a Text Document (TXT) or Rich Text Format (RTF).

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MText Practice

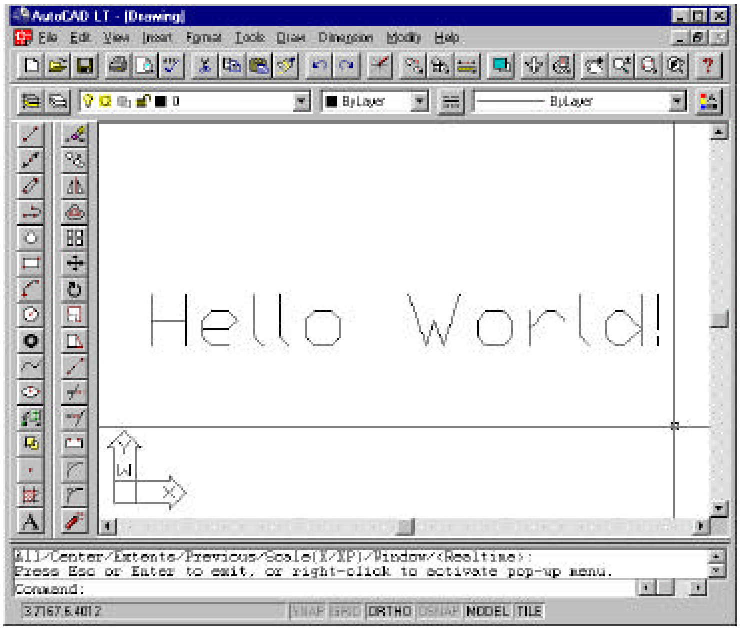

Fig. 7.36

During this practice you will be working with the MText command. 2 mins.

1. Start a new drawing from **Scratch** with **English** highlighted. This will ensure that all the settings are setup to AutoCAD LT standards.

2. Start up the **MText** command.

3. Select a point in the drawing for your first corner of your text.

4. Select another point in the drawing for your opposite corner of your text.

5. It will take a couple moments for the MText Editor to come up for the first time. In the edit box type-in **Hello World!**, and press **OK** to create the object.

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During this practice you will be working with the DText command w/ a control codes. 2 mins.

1. Start up the **MText** command.

2. Select a point in the drawing for your first corner of your text.

3. Select another point in the drawing for your opposite corner of your text.

4. Click on the **Import...** button. Once the browse dialog comes up go to the Section 7 folder and select the file called **mtext.text** and click the Open button.

5. Highlight all of the imported text and under the **Characters Tab** select the font **Times New Roman**.

6. Un-highlight the text and only highlight the first sentence of the imported text and make this **Bold**.

7. If both your sentences are not on separate lines go to the **Properties Tab** and adjust the **Width** until both are on separate lines.

8. After adjusting the width click the **OK** button to create the new text object.

After you have completed the practices go back and try some of the other settings to see how they affect the text.

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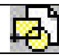
Insert Command

Command Access	
Command Line	DDINSERT + Enter
Toolbar	DRAW 
Menubar	INSERT>>Insert

The Insert command allows you to place a Block into the drawing that is either defined internally or externally out on a local or a network drive.

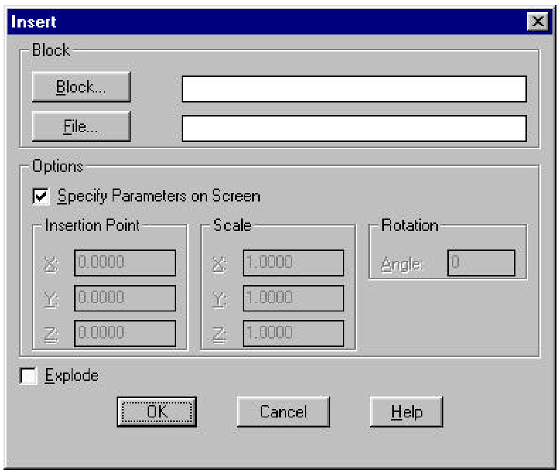


Fig. 7.38

Command Options	
Insertion point:	Specify the point in the drawing in which you would like to place the block.
X Scale:	Specify the scale factor in the X direction.
Y Scale:	Specify the scale factor in the Y direction.
Rotation:	Specify the amount of rotation for the block.

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Insert - Block

There are two methods to inserting a block. The first type of insert is by internal Block. The internal block is a block that already has been either inserted or created in the drawing. So you are only allowed to insert a block that already exists in the drawing. The second type of insert is by external file or block. The external block is stored as a drawing file, and is just brought into the current drawing. The external block allows you to reuse the block at a later date if you need it.

Insert - Options

The Options section of the dialog box is used to specify all of the Command Options for you if you want it to. If the Specify Parameters On Screen is checked that means you as the user have control of choosing the values for the Command Options.

Insert - Options - Insertion Point

The Insertion Point is the point at which the block is to be placed at in the drawing.

Insert - Options - Scale

The Scale is the stretch factor at which the block will be scaled and stretched to in the appropriate direction.

Insert - Options - Rotation

The Rotation is the angle at which the block will be placed at.

Insert - Explode Block

The Explode Block option allows you to automatically Explode the block after has been placed into the drawing..

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Insert Practice

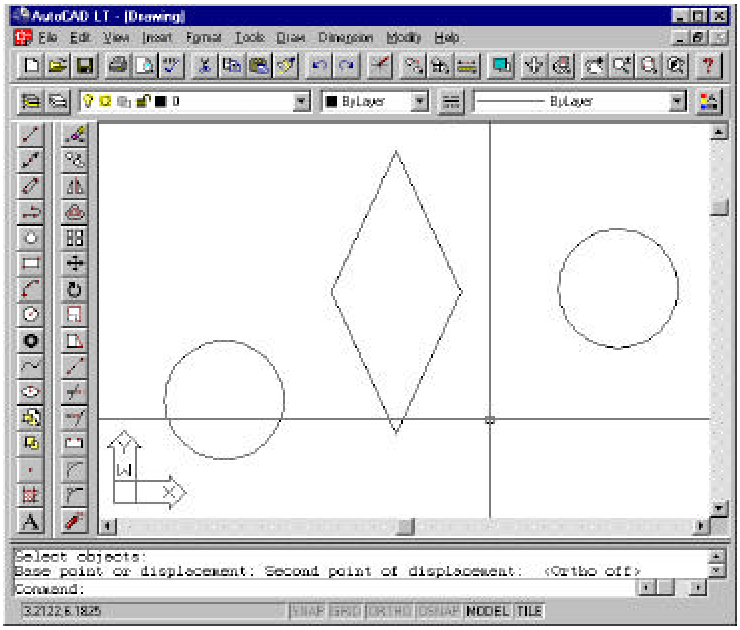

Fig. 7.39

During this practice you will be working with the Insert command to bring in an external block.
5 mins.

1. Start a new drawing from **Scratch** with **English** highlighted. This will ensure that all the settings are setup to AutoCAD LT standards.

2. Start up the **DDInsert** command.

3. Click on the File button so we can insert a block from an external location. We are going to browse to the Folder on the CD called **AutoCAD LT** and then in go to the **Section 7** folder to find the drawing called **Diamond**.

4. After you find the **Diamond** file select **Open**, and then **OK** to exit the Insert dialog box.

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5. At the **Insertion point:** prompt, select the point in the drawing in which you would like to add the Diamond at.

6. At the **X scale factor <1> / Corner / XYZ:** prompt, press Enter to accept the default X-Scale.

7. At the **Y scale factor (default=X):** prompt, press to Enter accept the default Y-Scale.

8. At the **Rotation angle <0>:** prompt, select or type-in the angle you wish to rotate the block at.

During this practice you will be working with the Insert command to bring in an internal block.
5 mins.

1. Start up the **DDInsert** command again.

2. Click on the **Block** button so we can insert a block from internally of the drawing. Clicking the Block button will bring up a list of Blocks that are currently available to use in the drawing.

3. Select the **Block** called **Circle** and click the **OK** button.

4. **Insert** the **Circle** block into the drawing. We are going to place the Circle block at one of the longer ends of the block, so lets use our Object Snaps to perform this task.

5. Type-in at the command line **'DDOSNAP** or select it from **TOOLS>>Object Snap Settings...** .

6. Once the dialog is up select the **INTersection** and **ENDpoint** Object Snaps and click the OK button.

7. Go to one of the longer points of the **Diamond**, you should notice that the Object Snap should tack affect and you should see the **Marker**.

8. Once you get the Marker select the point, and press Enter twice to accept the default of both the X and Y Scales.

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9. Select your rotation angle for the Circle or just press Enter to accept the default angle.

During this practice you will be working with the Insert command at the command line to bring in an internal block. 5 mins.

1. Start up the **Insert** command.

2. At the **Block name (or ?):** prompt, type-in **Circle** unless you also see **<Circle>** at the prompt line. If you see an item in Angle Brackets that will be the default block that will come-in if you press Enter right now. Either press Enter now or after you type-in **Circle**.

3. **Insert** the **Circle** block into the drawing. We are going to use the Object Snaps again. They should already be set since last time except we want to make sure that we use the **INTersection** Object Snap.

4. At the **Insertion point:** prompt, type-in **INT** and press Enter to continue. You should notice that the Object Snap should tack affect when you go by the point. Make sure that you select a different point on the **Diamond**.

5. Once you get the Marker select the point, and press Enter twice to accept the default of both the X and Y Scales.

6. Select your rotation angle for the Circle or just press Enter to accept the default angle.

7. Repeat the insertion process of the Circle until all points on the diamond have a **Circle** on them.

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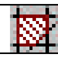
Hatch Command

Command Access	
Command Line	BHATCH + Enter
Toolbar	DRAW 
Menubar	DRAW>>Hatch...

The Hatch command allows you to fill an area with a repeating pattern of lines. AutoCAD LT comes with a set of 68 Hatch patterns, but if you need to you can either create your own hatch patterns, or purchase/ download them from third-party developers. Hatching areas on your drawing either can draw focus to that area or away from it based on how it is used in conjunction with the rest of your drawing.

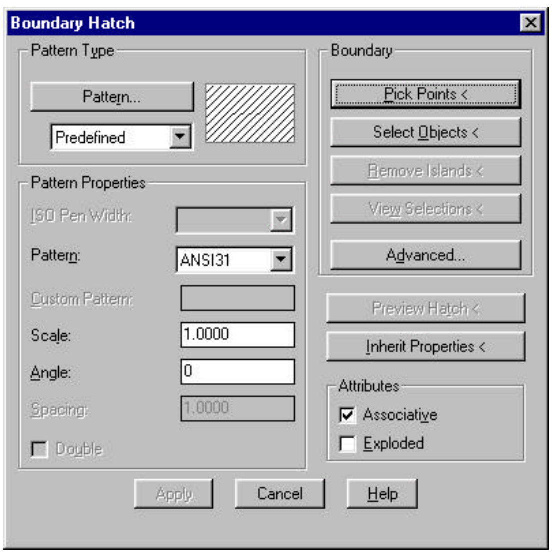


Fig. 7.38

Command Options	
Pattern...	Specify the Hatch pattern that you would like to use for hatching, and preview it.
ISO Pen Width:	Specify the pen width for any Hatch pattern that starts with ISO.
Pattern:	Specify the pattern name.

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Command Options	
Custom Pattern	Specify a custom Hatch pattern.
Scale:	Specify the scale at which the Hatch pattern will be applied to your drawing..
Angle:	Specify the angle at which the Hatch pattern will be applied at.
Spacing:	Specify the distance between the lines in a user-defined Hatch pattern.
Double:	Specify if a user-defined Hatch pattern will be double hatched.
Pick Points <	Specify a point inside of a closed in area or a boundary.
Select Objects <	Specify a group or an object that will be used to create a boundary. Objects (or Islands) in side of the boundary are must be selected individually.
Remove Islands <	Specify whether or not an Island inside of a boundary are will be filled over (if removed) or around (if left in). The Island must be located entirely in the boundary in order to be removed.
View Selections <	Allows the user to see the current boundary area that will be used for hatching.
Advanced...	Launches the Advanced dialog box.
Preview Hatch <	Allows the user to preview how the Hatch will look in the areas that they selected.
Inherit Properties <	Allows the user to select an existing piece of Hatch from the drawing to setup some of the parameters for the new Hatch pattern to apply.
Associative:	Specifies if the Hatch pattern stays with its boundary area or not.
Explode:	Specifies if the Hatch pattern will be exploded after it is applied.

Notes:

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