



Introduction to Computer Aided Drafting



Welcome to the AutoCAD LT BBT module! You probably have several questions that you would like answered before you start this module, for example:

- What is Computer Aided Drafting (CAD)?
- What is AutoCAD LT?
- What will I learn by doing this module?

Let's answer your questions:

What is Computer Aided Drafting (CAD)?

Computer Aided Drafting (referred to as CAD) is using a computer and software to draw and design objects. A CAD system can be used to:

- Draw and design parts for machines (mechanical drawing),
- Design houses and structures (architectural drawing),
- Design roads, highways and bridges (civil drawing),
- Draw and design electronic circuits (schematic drawing),
- Do two dimensional (2D) and three dimensional (3D) models.



What is AutoCAD LT?

AutoCAD LT is a CAD software package. It is one of the most widely used CAD packages in the world. AutoCAD LT is very user friendly and virtually anyone can learn how to use it. The AutoCAD LT interface is windows based which users will find easy to use.

What will I learn by completing this module?

By completing this module, you will learn:

- How CAD works.
- How to use basic CAD tools to complete drawings.
- How to manipulate drawings.
- The importance of CAD to industry.

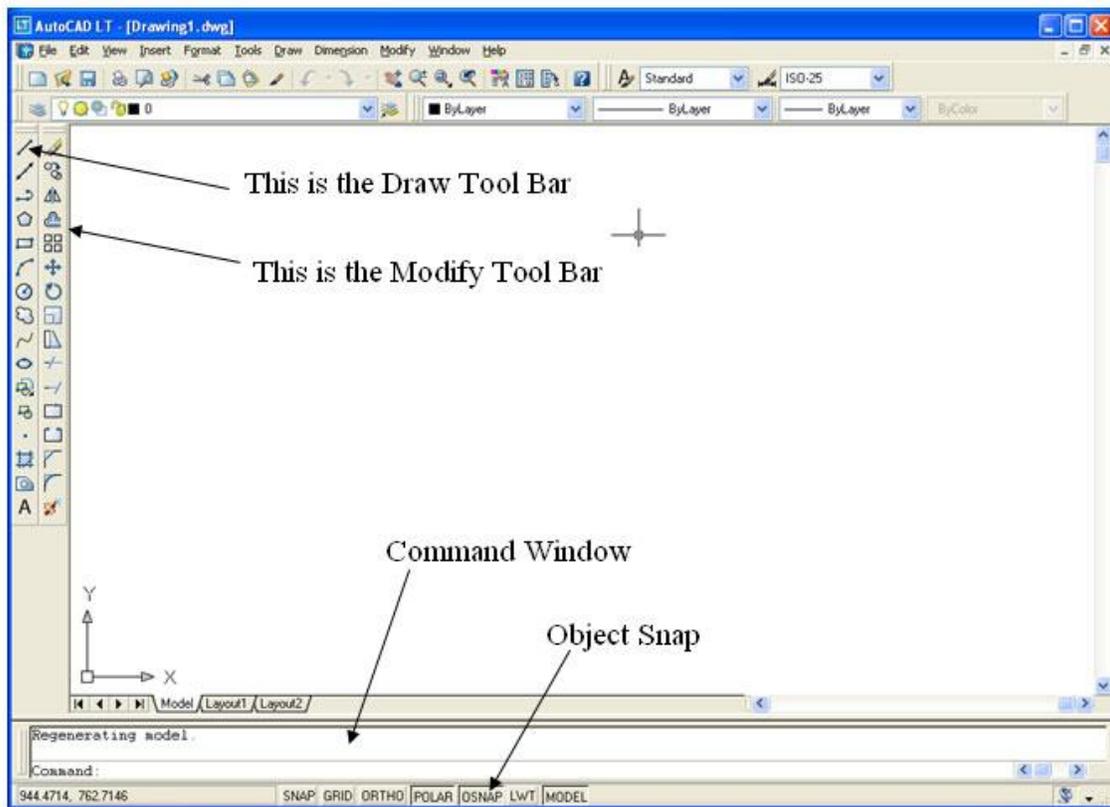
Let's get started.....



Activity 1

Introduction to AutoCAD LT

1. Power on your computer.
2. On your computer desktop, you will see the AutoCAD LT2004 icon.
3. Double click on the icon to access the AutoCAD LT 2004 program.
4. The AutoCAD LT interface will now appear on your screen, let's identify the main sections of the interface.



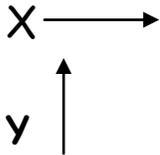
The **Modify Toolbar** contains tools that will modify objects that already appear on your screen. You will use this menu often throughout this module.



The Command Line

The Command Line – The command window is where commands are entered and executed in the AutoCAD LT program. We will use the command window throughout this module. Please become familiar with the Command Line, as it will prompt you on what you are to do next.

Coordinate System in AutoCAD



Coordinate System – is used to plot or identify points on a drawing. This means that you have to tell the program exactly where you want to put objects, lines, etc., in your drawing. All points in CAD drawings are identified by something called coordinates. For our purposes, we are going to look at two types of coordinates, the “X” coordinate and the “Y” coordinate. The point of origin is where the X and Y axes intersect (0,0) in the lower-left corner of the drawing.

The “X” coordinate is any point in the drawing along the horizontal plane (left to right).

(X, Y)

The “Y” coordinate is any point in the drawing along the vertical plane (up and down).

We have now identified the main areas of the drawing window so let’s begin to draw some different shapes.

1. If AutoCAD LT 2004 is not open, double click on the icon to start the program.
2. Click on File and then click on Open.
3. Find the student shared drive.
4. Double click on the Grade 10 BBT folder.
5. Double click on the AutoCAD LT 2004 folder.
6. Find the **Prototype 1** file and double click on it.

Important

Please note that each time a new drawing is started, the **Prototype1** drawing must be loaded. The screen will appear empty, but important settings have been saved for your convenience in this file.



Using the Draw Toolbar

The Draw Toolbar contains the tools necessary to draw and construct objects on your computer screen. **Hint:** If you take your mouse and put it over any of the buttons on the screen, it will tell you the name of each icon.

There are two methods that can be used for drawing objects within AutoCAD LT 2004. The first method is the free hand method. Using this method, you choose a drawing tool and place the object you have drawn on the screen. The problem with this method is that it is not very accurate. The second method is drawing by coordinate. This method is very accurate, as it tells the software exactly where you want to draw an object. This is the method we will be using.

Using the Line Tool



Practice Exercise 1

1. Open Prototype 1 and click on the Line Tool in the Draw Toolbar Menu.
2. Click anywhere on your drawing screen to start drawing a line.
3. Drag your mouse across the screen until you have the line the length that you want it, then click your left mouse button.
4. Right click or press Enter to stop the line command.
5. This is an example of **free hand drawing**. It is very difficult to create an accurate drawing using this method.
6. Click on the line that you have just drawn, it will now be selected. Delete the line by pressing the delete key on your keyboard.

Using Coordinates

Absolute Coordinate values are based on the origin (0,0), where the X and Y axes intersect. Use an absolute coordinate when you know the precise X and Y values of the point coordinate. For example, the coordinate 3,4 specifies a point 3 units along the X axis and 4 units along the Y axis from the origin.

- X represents the distance and direction along the horizontal axis from the origin (0,0).
- Y represents the distance and direction along the vertical axis from the origin (0,0).

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AutoCAD LT 2004

Practice Exercise 2

1. Using absolute coordinates lets draw some lines.
2. Click the line tool.
3. In the Command Window it will say Command: _line Specify first point
4. Type in 0,6 then press Enter
5. The command window will now say Specify next point.
6. Type in 80,6 and press Enter twice.
7. You have now drawn a straight line using absolute coordinates.

Important

NOTE: If any of your drawings are not clearly visible, or if it is small and off to the side, it can be zoomed to a new size. **Click on View, Zoom and then Extents.** This will zoom in or magnify your work area.

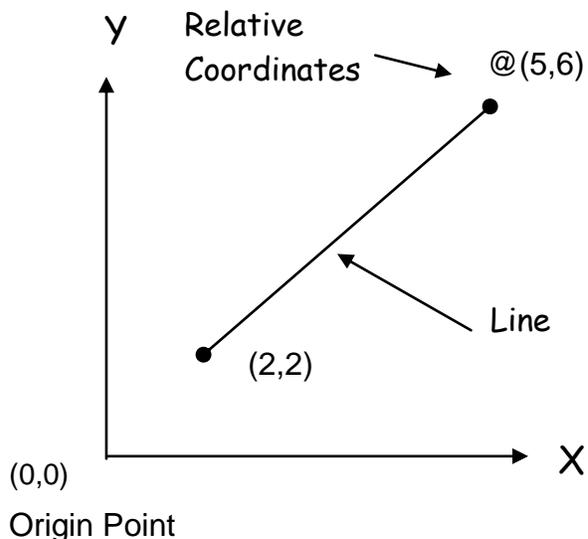
Note: Please watch the video on using Coordinate Entry at this time. The name of the video is **Relative** and is saved on the S:\StuShare in the AutoCAD folder.

Relative Coordinates – are used to locate points on the X axis and Y axis relative to a previous point. It's like entering the measurement of an object. For example, if you have a line that starts at (2,2) and you want it to finish at (7,8) you can use relative coordinates to draw the line. It would be done like this:

- (2,2) using absolute coordinates would be the starting point of the line.
- If you enter the @(5,6) (relative coordinates) your line would finish at (7,8)(absolute coordinates).
- Simply, the distance between (2,2) and (7,8) is 5 units in the X direction and 6 units in the Y direction, these are the relative coordinates.
- Refer to the diagram below:

(@X,Y)

The @ symbol is used when entering Relative Coordinates



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AutoCAD LT 2004

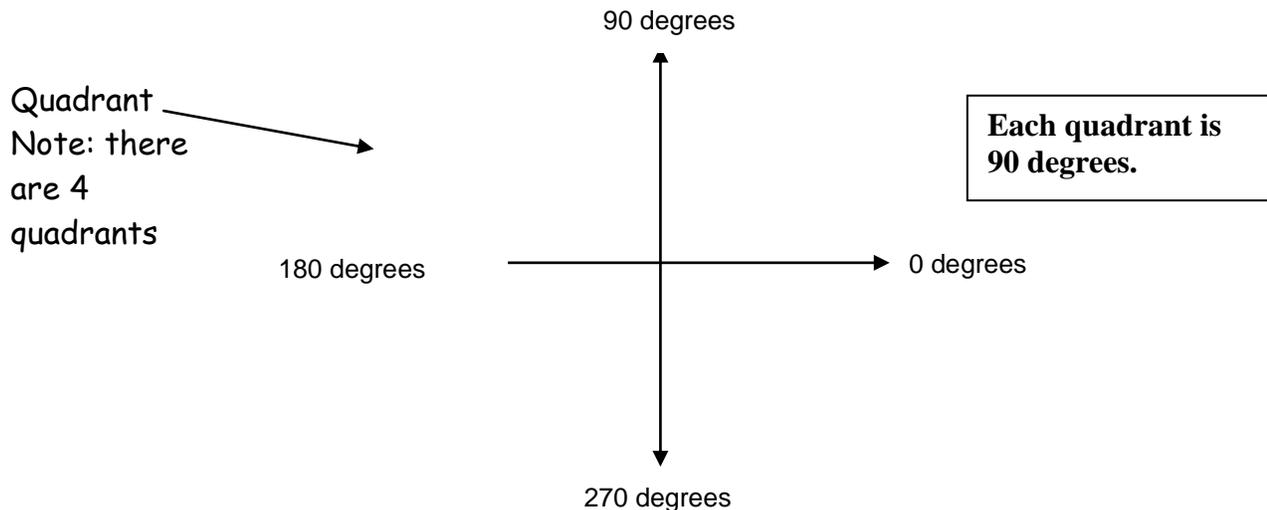
Practice Exercise 3

1. Click on the Line Tool.
2. In the Command Window (at the bottom of your screen) it will say Command: `_line`
Specify first point
3. Place your mouse anywhere on the screen and click the mouse.
4. The Command Window will then say: Specify next point.
5. Enter the following coordinates: `@100,0` then press enter (this is the end point of our line using Relative coordinates)
6. Enter coordinates `@0,50` then press enter
7. Enter coordinates `@-100,0` then press enter
8. Enter coordinates `@0,-50` then press enter
9. Press enter again or right click your mouse and you will exit the command
10. You just created a rectangle that is 100 long by 50 high.
11. Delete this rectangle by selecting the lines and pressing delete.

Important

Note: Please watch the video on using Coordinate Entry at this time. The name of the video is Polar and is saved on the S:\StuShare in the AutoCAD folder.

Polar Coordinates – are coordinates used when you want to define points based on an angle and distance from a previous point. The angles are based on 360 degrees (circle). The angles are broken into quadrants. Refer to the diagram below:



The symbol for using Polar Coordinates is `<` (less than sign). When entering Polar Coordinates, you use the first point followed by the angle that you want to create. For example, if you wanted to draw a line that is 10 units at a 30 degree angle, you would input `@10<30`.

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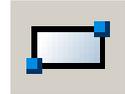
AutoCAD LT 2004

Practice Exercise 4

1. Click on the Line Tool.
2. In the Command Window it will say Command: `_line` specify first point
3. Place your mouse anywhere on the screen and click the mouse button.
4. Enter the following coordinates: `@100<0` (this is the end point of our line using Polar Coordinates) and press Enter
5. Enter coordinates `@50<90`, press Enter
6. Enter coordinates `@100<180`, press Enter
7. Enter coordinates `@50<270`, press Enter
8. Press Enter again or right click your mouse to finish the drawing.
9. Show your teacher your work and ask the teacher to sign your activity sheet.
10. Delete the rectangle.

You just created a rectangle that is 100 long by 50 high, identical to the previous drawing. So, you can see that by using either method you can achieve the same results. As you can see, the coordinate method is very accurate, it puts an object exactly where you tell it to.

Rectangle Tool



Practice Exercise 5

This time we will make a rectangle of the same size using the Rectangle Tool

1. Click on the Rectangle Tool in the toolbar
2. Click your mouse anywhere on the drawing screen.
3. Now in the command line it asks for “other corner”.
4. Type `@100,50` and press Enter.
5. Show your teacher your work and ask the teacher to sign your activity sheet.
6. Delete the rectangle.

A rectangle that is 100 long by 50 high is created. Using the rectangle tool is quicker and easier than using the line tool.

Circle Tool



Another drawing tool that is used often is the *Circle Tool*. There are four different ways you specify coordinates for drawing circles. The method which we will use is the default (the method the software automatically goes to first).

Practice Exercise 6

1. Click on the circle tool in the Draw Tool Bar menu.
2. The Command Window will now ask you to define the center point of the circle.
3. Click your mouse in the center of the screen
4. You have just defined the center point of the circle, now we have to define the radius (the distance from the center of the circle to the outside edge of the circle).

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AutoCAD LT 2004

5. Enter a radius coordinate of 40 in the command line then press Enter.
6. Try drawing another circle following the same procedure, but changing the radius to 50
7. Show your teacher your work and ask the teacher to sign your activity sheet.
8. Delete your drawings so that your screen is clear.

Polygon Tool

The last tool we are going to look at in this activity is the **Polygon Tool**. The Polygon Tool is used to create multi-side objects. The multi-sided object can have 3 equal sides (triangle), 4 equal sides (square), 8 equal sides (octagon), or as many sides as you need for your object.



Practice Exercise 7

1. Click on the Polygon Tool in the Draw Tool Bar menu.
2. In the Command Window, you will be asked to define the number of sides in the polygon (note: the default is 4).
3. Type 3 in the Command Window and press Enter.
4. Next, you will define the center of the polygon.
5. Place and click your mouse anywhere on the screen.
6. You will be asked if your polygon is Inscribed or Circumscribed.
7. Type in an I and press Enter.
8. You will be asked in the command line for the radius.
9. Type 50 and press Enter.
10. Now try drawing an octagon with the same radius as the triangle mentioned above.
11. Show your teacher your work and ask the teacher to sign your activity sheet.

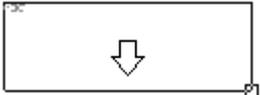
So far, you have learned:

- ✓ What CAD is.
- ✓ The AutoCAD LT interface.
- ✓ The Drawing Tool Bar.
- ✓ The Modify Tool Bar (we will look at this more later).
- ✓ Coordinate System (X,Y).
- ✓ How to use the Command Window.
- ✓ How to draw lines using coordinates.
- ✓ How to draw rectangles using coordinates.
- ✓ How to draw circles using coordinates.
- ✓ How to draw polygons using coordinates.

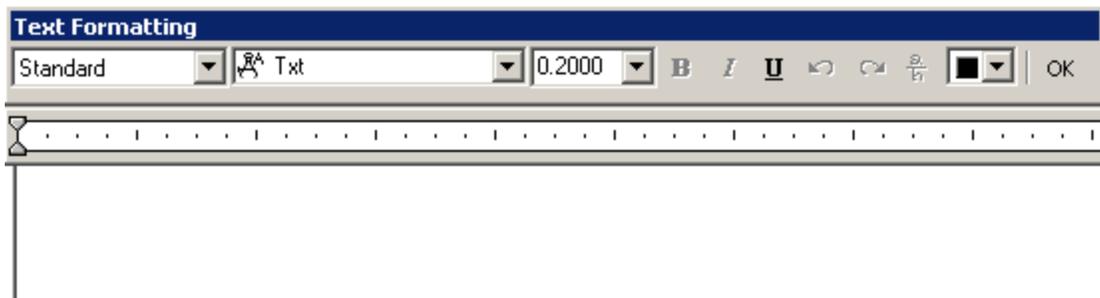


Activity 1 Exercises

Exercise 1

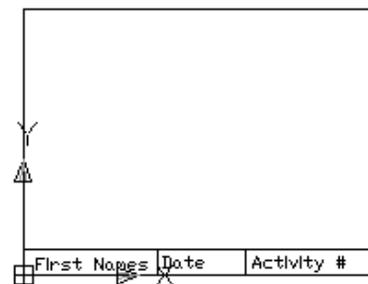
1. Open the Prototype1 file.
2. At the bottom of the window under the command line click on the OSNAP  button to turn it off.
3. You are now going to draw a title block. A title block is commonly used when presenting CAD drawings. To complete the title block complete the following steps.
4. Create a rectangle that has a first corner of 0,0 and the other corner at 80,60.
5. Draw a line from 0,6 (Press Enter) to 80,6 (Press Enter twice)
6. Draw a new line from 30,0 (Press Enter) to 30,6 (Press Enter twice)
7. Draw another new line from 50,0 (Press Enter) to 50,6 (Press Enter twice)
8. Click on the Text Tool in the Draw Toolbar.
9. Move the mouse to the first block that you drew in your title block, hold down the left mouse button and drag to create a box to type your name in. 

10. A window similar to the one shown below will appear.



11. Type in your first name and click OK.
12. Repeat this process until you have filled in the title block and it resembles the one shown below.

NOTE: You may need to adjust the size of the text for the name, date and activity.



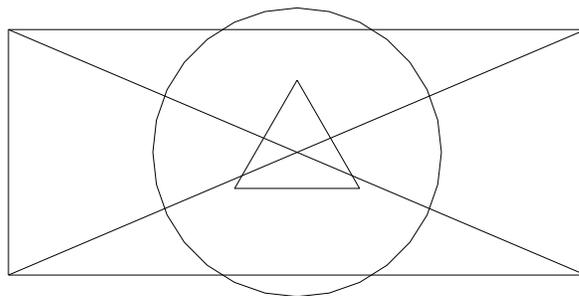
13. Save your work as Title.
14. Show the drawing to your teacher and ask them to sign your activity sheet.



Exercise 2

1. Start AutoCAD LT 2004 if it is not already open.
2. Open Prototype1. Make sure that **OSNAP** is turned on at the bottom of the screen.
3. Click on the rectangle tool and click anywhere on the screen to begin the drawing.
4. Draw a rectangle with the following coordinates: 200 long by 85 high (@200,85)
5. Click the line tool and move the mouse to the upper left corner of the rectangle.
6. When a small pink box appears on the corner of the rectangle click your mouse.
7. Move your mouse to the lower right corner until you get the same pink box and then click your mouse again.
8. Right click your mouse to shut off the line command.
9. Select the line tool again and draw another line from the opposite corners.
10. Select the circle tool.
11. The center point of the circle is the intersection of the 2 lines.
12. Move your mouse to where the lines intersect in the center and notice the pink x that appears.
13. Click your mouse when the pink x is visible.
14. Radius should be 50.
15. Select the polygon tool.
16. When the command window asks how many sides, choose 3.
17. The center point for the polygon is the intersection point of the 2 lines again.
18. The polygon is Inscribed.
19. Radius is 25.
20. Save your activity as **Act1 Ex2**.
21. Have your teacher check your work and sign your activity sheet.

Your finished CAD drawing should look like this:



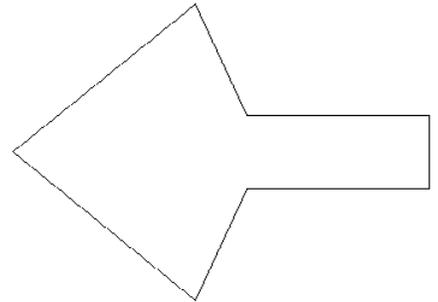
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Exercise 3

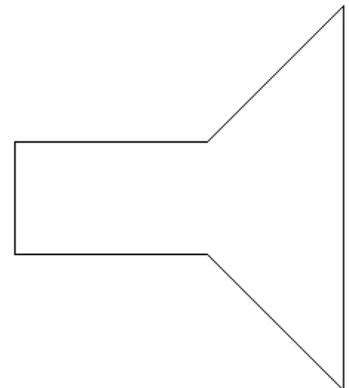
1. Open AutoCAD LT2004.
2. Open the document named **Prototype1** from the student shared drive. (BBT10/AutoCad)
3. Save the document as **Act1 Ex3** in your student folder.
4. Select Line from the Draw toolbar.
5. Point 1 - Place you mouse on the screen and click anywhere.
 - a. Point 2 - @1.75, -1.5 Press Enter
 - b. Point 3 - @.5, 1.125 Press Enter
 - c. Point 4 - @1.75, 0 Press Enter
 - d. Point 5 - @0, .75 Press Enter
 - e. Point 6 - @-1.75, 0 Press Enter
 - f. Point 7 - @-.5, 1.125 Press Enter
 - g. Point 8 - @-1.75, -1.5 Press Enter
 - h. Right click or press Enter to cancel the Line command
6. Notice that the relative coordinates you entered are all based from the previous point entered.
7. Save your work in your student folder as **Act1 Ex3**.
8. Have your teacher check your work and sign your activity sheet.



Exercise 4

1. Start the AutoCAD LT2004 software program.
2. Open the document named **Prototype1** which is found in the student shared folder.
3. Use the Line command with Polar Coordinates and follow the directions below to complete the drawing.
4. Point 1 Place you mouse on the screen and click anywhere.
 - Point 2 @1, 0 (Relative)
 - Point 3 @1<-45 (Polar)
 - Point 4 @0, 2 (Relative)
 - Point 5 @1 <225 (Polar)
 - Point 6 @1 <180 (Polar)
 - Point 7 @.5836<270 (Polar)Right click or press Enter to cancel the Line command
5. Save your drawing in your student folder as **Act1 Ex4**.
6. Have your teacher check your work and sign your activity sheet.

Note: Angles are drawn using Polar Coordinates and are based from the Origin Point (0,0)





Activity 1 Review Questions

1. The Modify Toolbar allows you to
 - A. Draw new objects.
 - B. Set coordinates.
 - C. To print.
 - D. Manipulate drawn objects.

2. Specific points in a CAD drawing are referred to as
 - A. Measurements.
 - B. Details.
 - C. Coordinates.
 - D. Specifications.

3. Any point along a horizontal plane is referred to as
 - A. Y.
 - B. X.
 - C. Z.
 - D. W.

4. The origin point in a CAD drawing is
 - A. (10,10).
 - B. (5,5).
 - C. (0,0).
 - D. (50,50).

5. This window is used for entering specific details for drawing.
 - A. Draw toolbar.
 - B. Command window.
 - C. Modify toolbar.
 - D. Snap window.

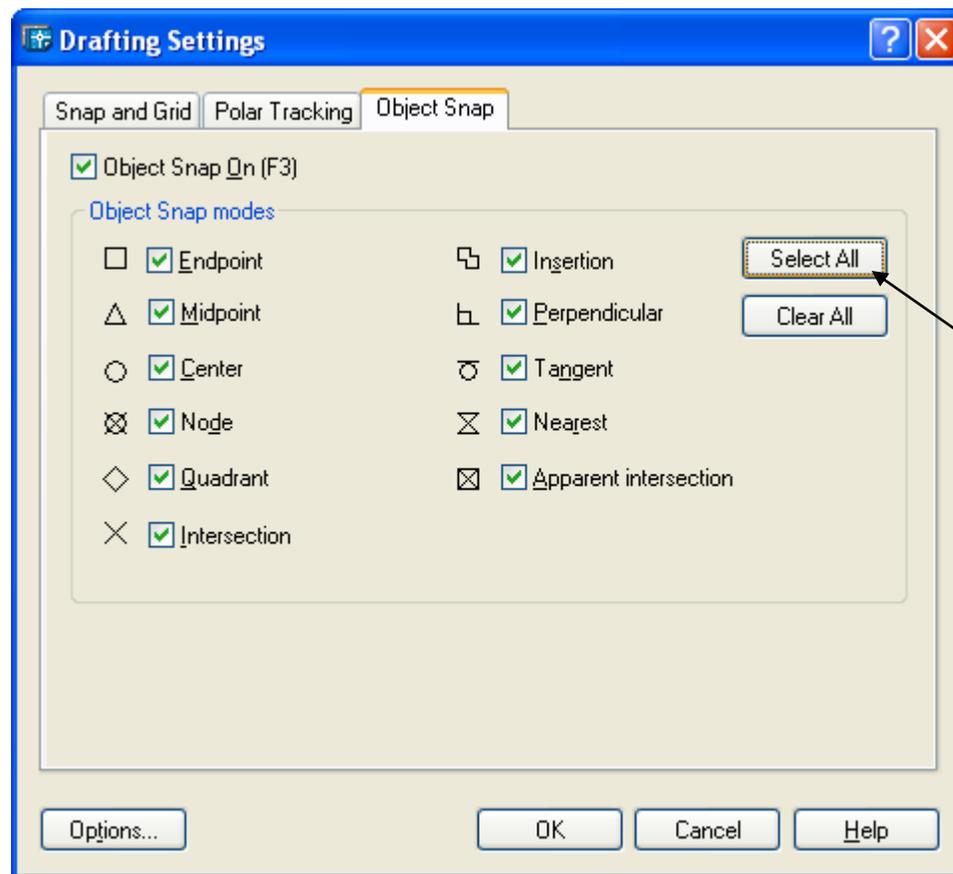


Activity 2

Introduction to Object Snap

Object Snap is another feature that is available in CAD software packages. An important feature about Object Snap is that you can connect directly to a specific point in a drawing without knowing the coordinates. In this activity, we are going to learn how to use Object Snap and its' importance.

1. Before you get started, go to the pull down menu at the top of the AutoCAD screen and select **Tools and then click on Drafting Settings**.
2. Click on the Object Snap Tab at the top of the window that just opened.
3. Click Select All.
4. All the Snap functions should now be selected.
5. Click OK.



Click
Select All



Object Snap Toolbar

If the Object Snap toolbar is not open on your screen do the following:

1. Click on the View menu.
2. Click on Toolbars.
3. In the Toolbar list that opened click in the check box in front of Object Snap to select it.
4. Click on Close.
5. To dock the toolbar with the other toolbars put your mouse in the top blue border, hold down the left mouse button and drag the toolbar to the left side of the screen. When you release the mouse button the toolbar should be docked along side the other toolbar.



Snap to Endpoint – This command allows you to snap (connect) directly to the nearest endpoint of a line, rectangle, or polygon.

Practice Exercise 1

1. Draw a freehand rectangle of any size.
2. Select the line tool.
3. Click on the **Snap to Endpoint** button in the Snap Toolbar.
4. Move your mouse towards the endpoints of the lines in the rectangle and notice the small pink box that appears. This shows the snap points for the line.
5. Using one of the corners (highlighted by the Snap to Endpoint button).
6. Click to start a line. You now have a line connected to one corner of the rectangle.
7. Click on the **Snap to Endpoint** button again and connect the line to the opposite corner (diagonally).
8. Right click to end the line command or press Enter.
9. Using the same procedure above, draw a line to the other corners of the rectangle.
10. Show your teacher your work and ask the teacher to sign your activity sheet.



Snap to Midpoint – This command allows you to snap directly to the midpoint of a line or line segment. This is very useful, especially when you want to start a new line at the midpoint of another line.

Practice Exercise 2

1. Use the same rectangle that you just constructed.
2. Select the line tool.
3. Click on the **Snap to Midpoint** button in the Snap Toolbar.
4. Place your mouse at the center of one of the rectangle lines. You will notice the center is highlighted with a pink triangle.
5. Click to accept the midpoint.

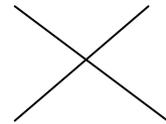


6. Click on the Snap to Midpoint button again.
7. Move your mouse to the center of the line on the opposite side of the rectangle.
8. Click to accept the midpoint.
9. Right click or press enter to end the line command.
10. Follow the same procedure to connect a line to the midpoints of the opposite rectangle sides.
11. Show your teacher your work and ask the teacher to sign your activity sheet.

Before we move to the next snap object, we are going to clear the screen. To do this, click on each line and press the delete key on the keyboard.



Snap to Intersection – This command allows you to snap to an intersection between two lines, arcs, or objects.



Practice Exercise 3

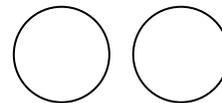
1. Draw two lines that intersect or cross each other.
2. Click on the Line tool and then the **Snap to Intersect** button.
3. Move your mouse to the point where the two lines intersect.
4. Click to snap to the intersect point.
5. Move your mouse down and click again to draw a straight line.
6. Right click or press enter to stop the line command.
7. Show your teacher your work and ask the teacher to sign your activity sheet.



Snap to Center – This command allows you to snap to the center of a circle.

Practice Exercise 4

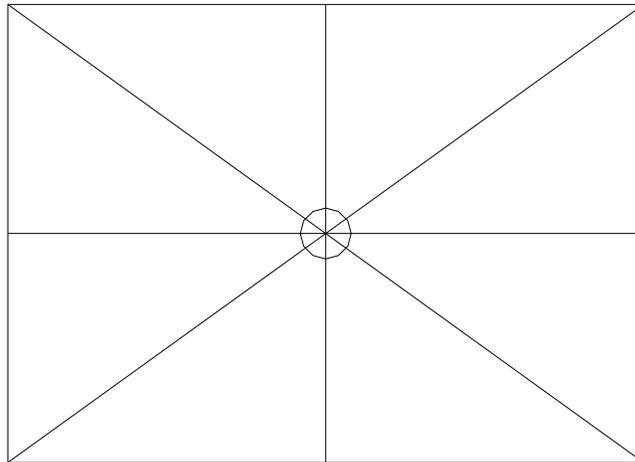
1. Draw two circles.
2. Click on the line tool.
3. Click on the Snap to Center button.
4. Move your mouse towards the center of one of the circles.
5. When the center is highlighted with a small pink circle, click to accept the point.
6. Click on the Snap to Center button again.
7. Connect the line to the center of the second circle.
8. Right click or press enter.



Now that you are familiar with some of the main snap commands, move on to complete the Activity 2 Exercise.



Activity 2 Exercises



Exercise 1

1. Open the file Prototype1 from the S:\StuShare drive.
2. Using the draw toolbar and the snap commands you have just learned, create a drawing similar to the one shown above.
3. The starting point for the rectangle can be anywhere.
4. The size of the rectangle is 250 long by 180 high.
5. All other lines are to be drawn using snap commands.
6. Can you figure out which snap commands to use?
7. The last step is to draw a circle.
8. Select the Circle tool and then Snap to Intersection.
9. Snap where the two centerlines intersect and then click.
10. The radius of the circle is 10.
11. Save your drawing as **Act2 Ex1**.
12. Show your work to your teacher and have the teacher sign your activity sheet.



Activity 2 Review Questions

1. The Snap toolbar can be found in the
 - A. File menu.
 - B. Insert menu.
 - C. View menu.
 - D. Modify menu.
2. Identify the Snap to Midpoint button.
 - A. 
 - B. 
 - C. 
3. The snap command is useful because
 - A. It determines drawing limits.
 - B. It automatically sets Z coordinates.
 - C. You can connect to a specific point in a drawing.
 - D. It always draws from (0,0).
4. An important feature about Object Snap is that you can connect directly to a specific point in a drawing providing you know the coordinates.
 - A. True
 - B. False
5. The snap commands can only be used when drawing straight lines.
 - A. True
 - B. False

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AutoCAD LT 2004

Activity 3 Using Trim and Extend

In this activity, you will learn how to use the Trim and Extend commands from the Modify toolbar. The trim and extend commands allow you to modify lines, arcs and other objects which intersect each other. This is a valuable tool and saves a lot of time when completing drawings.

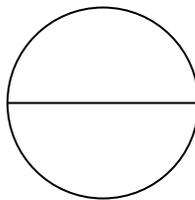
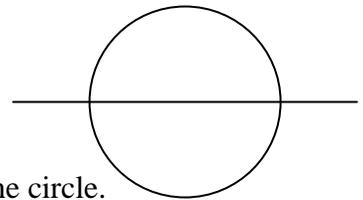
Using the Trim Command



The **Trim** command is used when you want to trim a line or arc that extends to the outer limits of a geometric shape.

Practice Exercise 1

1. Open the file Prototype1 from the student shared drive.
2. Draw a circle on your screen.
3. Draw a line through the circle similar to the diagram to the right.
4. We are now going to trim the parts of the line that extend outside the circle.
5. Click on the Trim button in the Modify toolbar.
6. You will then be asked to select the object that you want to trim to.
7. Click on the circle to select it.
8. Press Enter.
9. Now you have to select the object you want to trim. In this case it would be the lines that extend outside the circle.
10. Click on each part of the line that you want to trim and notice what happens.
11. Press Enter when you have finished trimming.
12. Your drawing should like the one below.
13. Show your teacher your drawing and ask the teacher to sign your activity sheet.



Neat Feature: Try using the functions of the Zoom command in the View Menu!

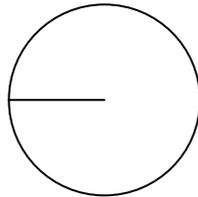


Using The Extend Command

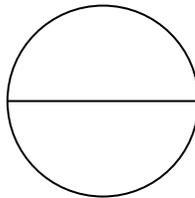
The **Extend** command is the opposite of the Trim command. The Extend is used when you want to extend a line to an object. The extend command is a two step process, just like the Trim command.

Practice Exercise 2

1. Open the file Prototype1 from the student shared drive.
2. Draw a circle on your screen.
3. Using the line tool and snap commands, draw a line from the center of the circle to an outside edge. It should look like the drawing below:



4. We are now going to use the extend command to extend the line in the center to the opposite side of the circle.
5. Click on the Extend button in Modify toolbar.
6. You will then be prompted to select the object that you want to extend the line to.
7. Click on the circle, then press Enter.
8. Next, you will select the object that you want to extend.
9. Click on the line.
10. Notice that the line automatically extends to the other side of the circle.
11. Press Enter when you are finished. Your drawing should now look like this:



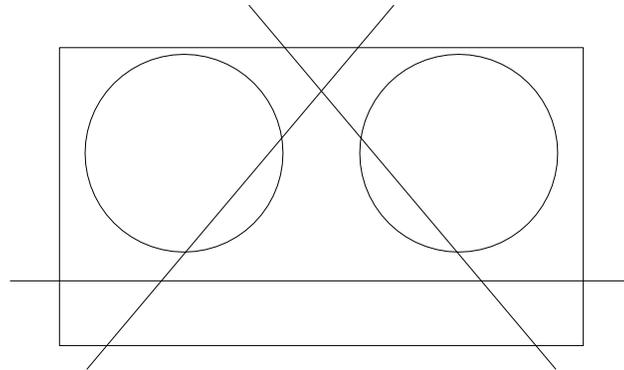
12. Show your teacher your drawing and ask the teacher to sign your activity sheet.



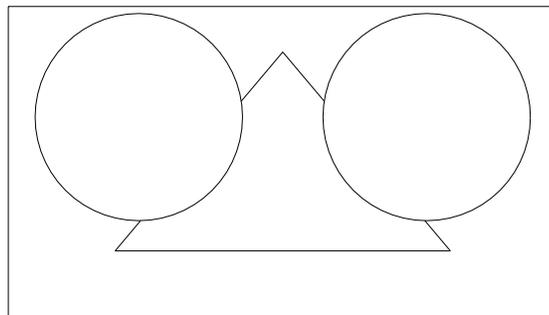
Activity 3 Exercises

Exercise 1

1. Make sure that the AutoCAD LT 2004 software is open.
2. Click on File and then click on Open.
3. Select **Activity3PartA.dwg** located in the S:/StuShare/Grade10 BBT/AutoCAD.
4. The drawing below should appear on your screen.



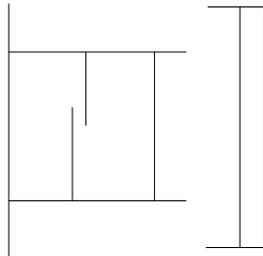
5. Using your Trim command from the modify menu, trim the lines in the drawing so that when you are finished, your drawing looks like the drawing below.
6. Save your work as **Act3 Ex1**.
7. Have your teacher check your work and sign your activity sheet.



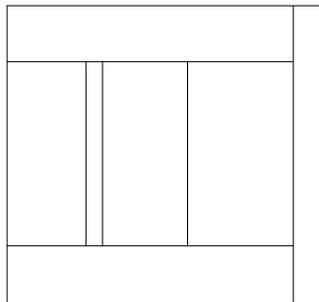


Exercise 2

1. Make sure that the AutoCAD LT 2004 software is open.
2. Click on File and then click on Open.
3. Select **Activity3PartB.dwg** located in the S:/StuShare/Grade10 BBT/AutoCAD.
4. The drawing below should appear on your screen.



5. Using the Extend command, modify the drawing on your screen so that all possible lines are extended. The finished drawing should look like this:
6. Save your drawing as **Act3 Ex2**.
7. Have your teacher check your work and sign your activity sheet.



Note: You will have to use the Trim command to trim the lines in the upper left hand and lower left hand corners.



Activity 3 Review Questions

1. Using the Extend command is a
 - A. Three step process.
 - B. Four step process.
 - C. Single step process.
 - D. Two step process.

2. The _____ command is used to remove unwanted parts of lines or arcs.
 - A. erase
 - B. extend
 - C. trim
 - D. delete

3. To Snap to the center of a line you must choose the _____ command
 - A. Endpoint
 - B. Center
 - C. Midpoint
 - D. Intersection

4. Polar coordinates are based on:
 - A. 6 quadrants
 - B. 4 quadrants
 - C. 2 quadrants
 - D. 8 quadrants

5. This type of coordinate entry works based on a previous point of entry.
 - A. Point
 - B. Absolute
 - C. Polar
 - D. Relative



Activity 4

Using the Move, Copy, Erase, Rotate and Mirror Commands

Move Command

The **Move** command allows you to move objects from a specific point on the screen to a designated point.

Practice Exercise 1

1. Start AutoCAD LT 2004.
2. Open Protoype1.
3. Draw a rectangle on your screen with the following coordinates 150 long by 250 high.
4. Click on the **Move** button in the Modify toolbar.
5. You will be prompted to select an object.
6. Click on the rectangle and press Enter.
7. In the command window, you will see the command Base Point or Displacement.
8. **This means that you need to define a point in the object, where you can pick the object up to move it (also called a handle point).**
9. The top right hand corner of the rectangle will be the Base Point.
10. Place your mouse near the top right corner of the rectangle and click to select it.
11. Notice a line is drawn from the Base Point to the rectangle that you can now move.
12. Next, you will be prompted for a second point of displacement.
13. This is the location that you want to move the object to.
14. Enter the coordinates @40,0 and press Enter.
15. The rectangle moved 40 units to the right.
16. Move the rectangle again.
17. This time use the upper left corner as the base point and move the rectangle 50 units straight up. (@0,50)
18. Show your teacher your work and ask the teacher to sign your activity sheet.
19. Close this file and do not save it.

Copy Command

The **Copy** command allows you to copy an existing object. This command saves time, especially when a drawing contains parts or objects which are identical. The Copy command works very similar to the Move command.

The major difference between the Copy and Move command is once you define your second point of displacement, a second object appears.

Note: the original object stays in the original location.



Practice Exercise 2

1. Open the file Prototype 1 from the student shared drive.
2. Draw a circle with a radius of 40 anywhere on the screen.
3. Click on the **Copy** button in the Modify toolbar.
4. You will then be prompted to select the object to copy.
5. Click on the circle to select it and then press Enter.
6. Next, you will be asked to define a Base point or displacement / Multiple.
7. Snap to the center of the circle to identify this as the base point.
8. In the command line you will be asked for the second point of displacement.
9. Type @100,0.
10. A second circle appears with the centers of the circles 100 units apart.
11. Show your work to your teacher and have the teacher sign your activity sheet.

Note: Snap commands can also be used to define base points or displacement

Practice Exercise 3

You can also copy an object several times using the **Multiple** command.

1. Click on the Copy command.
2. Next, select the circle you first created and press enter.
3. You will be prompted for a Base point or displacement / Multiple.
4. Push **M** on your keyboard. The base point will be the center of the circle.
5. Place 4 more circles anywhere on the screen.
6. Show your teacher your work and have your teacher sign your activity sheet.
7. Close this file and do not save your work.

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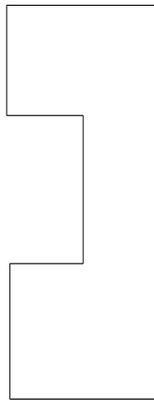
Mirror Command

The **Mirror** command allows you to create a mirror image of an object. This is especially important when creating drawings that are symmetrical. The Mirror command allows you to copy and flip an object 180 degrees.

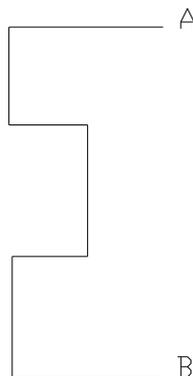


Practice Exercise 4

1. Open the Prototype1 file.
2. Use the Line tool to make a drawing similar to the one shown.



3. Click on the Mirror button in the Modify toolbar.
3. Click on the lines of the object to select them.
4. Press Enter.
5. You will be prompted for the first point of the mirror line.
6. To define this point, **Snap to Endpoint of point A.**

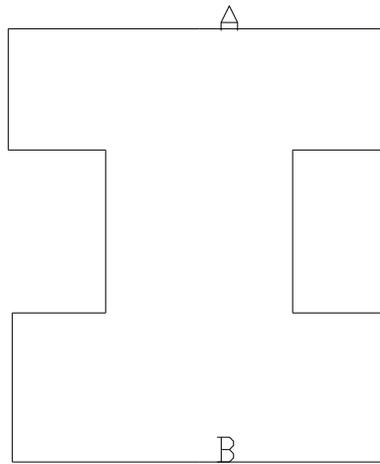


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8. The second point of the mirror line must now be defined.
9. To do this, use the **Snap to Endpoint** command and snap to the **endpoint B**.
10. Delete old objects will appear in the command window, press N and then Enter.
11. You just created a mirror image.
12. Show your teacher your work and have your activity sheet signed.
13. Close the file and do not save the changes.

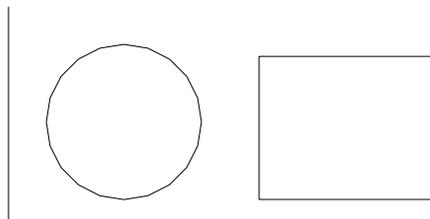


Using the Erase Command

Try This!

1. Open Prototype 1.
2. Create a line, a circle and a rectangle anywhere on your screen.
3. Select the Erase tool from the Modify toolbar.
4. In the command line it will ask you to “select objects”.
5. Position your mouse anywhere on the line and click. The line is now selected and highlighted.
6. Press enter and the line is erased.

If you make a mistake and erase the wrong item, simply type UNDO in the command line and press enter and the object reappears.





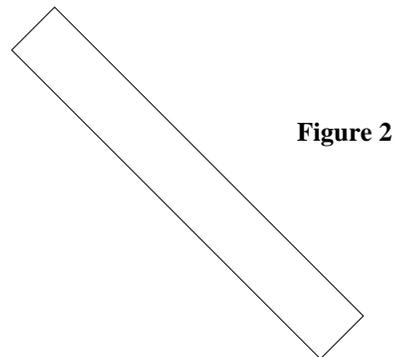
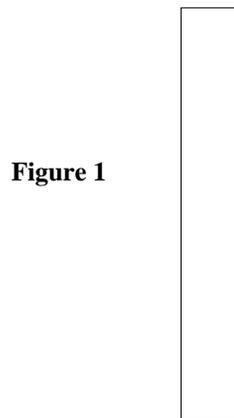
7. This time select erase and click on the edge of the circle and rectangle.
8. Both items are selected.
9. Press enter and they both erase.
10. Close this file and do not save the changes.

Note: You can select one object at a time or several objects. This can be done in most of the Modify command toolbar functions.

Using the Rotate Command

Practice Exercise 5

1. Open Prototype 1.
2. Create a rectangle similar to the one shown below in Figure 1.
3. In the Modify toolbar select the Rotate tool.
4. You will be asked to select objects.
5. Click on the edge of the rectangle.
6. Press enter.
7. You will be asked to select a base point.
8. Click on the lower left corner of the rectangle.
9. The base point or displacement is the last point defined when manipulating an object.
10. You will be asked to specify rotation angle.
11. Type 45 and press Enter (**this means 45 degrees**).
12. The rectangle should now appear as below in Figure 2.



13. Show your work to your teacher and have your activity sheet signed.
14. Close the file and do not save the changes.

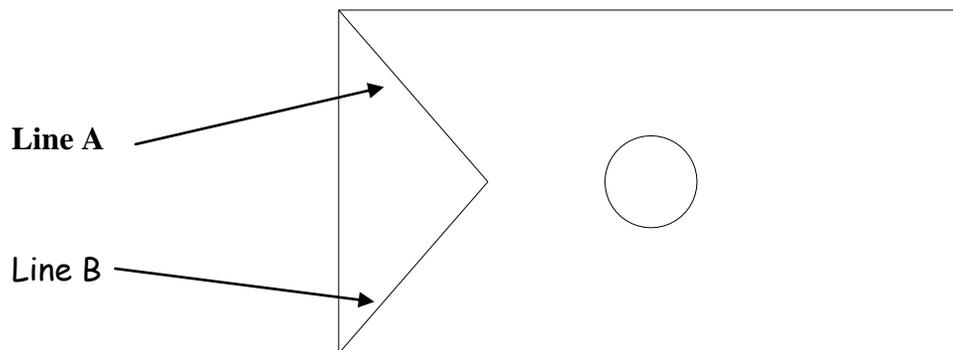


Activity 4 Exercise

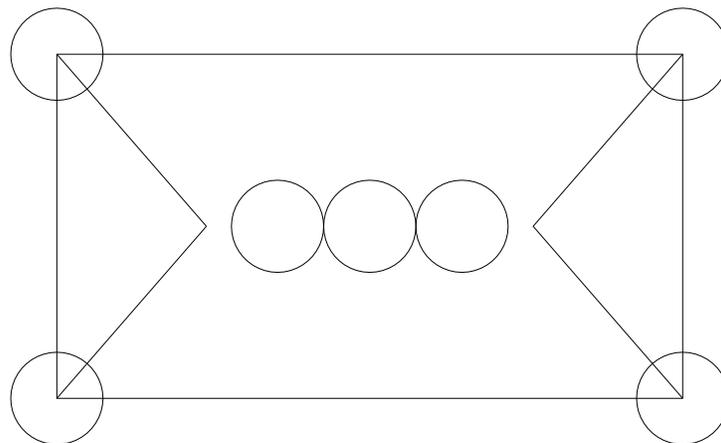
Exercise 1

In this activity, you will use the Move, Copy and Mirror commands to complete this drawing.

1. Open the file Activity4.dwg from the S:/StuShare drive.



2. Using the **Mirror** command, create a mirror image of lines A and B shown above. They are to be relocated at the opposite end of the rectangle.
3. Using the **Copy** command, copy the circle in the center of the drawing and place it in six different locations.
4. In order to do this, you will need to use the Snap commands to define the correct locations of the circles (base point or displacement). Refer to the drawing below:



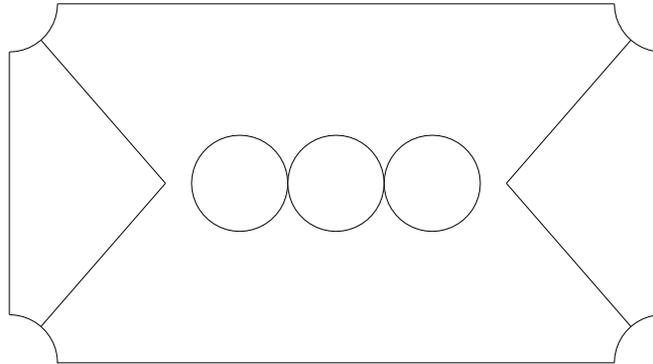
Hint: Use the Snap to Quadrant, Snap to Center and Snap to Midpoint commands.

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- Using the Trim command, trim the four circles in the corners so they become arcs as shown in the drawing below:



- Save your drawing as **Act4 Ex1**.
- Have your teacher check your work and sign your activity sheet.



Activity 4 Review Questions

1. This command allows you to place objects in a new location:
 - A. Scale
 - B. Mirror
 - C. Move
 - D. Rotate

2. You can create the other half of a symmetrical object with this command.
 - A. Array
 - B. Move
 - C. Mirror
 - D. Copy

3. This command will allow you to connect to the outside edge of a circle.
 - A. Snap to Radius
 - B. Snap to Center
 - C. Snap to Edge
 - D. Snap to Quadrant

4. When the Mirror command is used, how many points have to be defined?
 - A. Four
 - B. Three
 - C. Two
 - D. None of the above.



Activity 5

Using the Array Tool

The **Array** command allows you to copy existing objects and place them in a defined pattern.



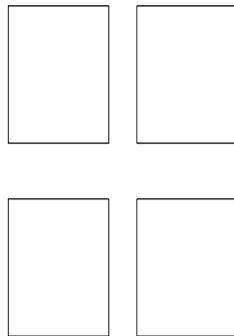
Imagine that you are designing a set of gears for a mountain bike. You have drawn a circle and one tooth of the gear. The Array command would let you copy that one tooth and place it in a pattern that you define. The result would be all of the teeth for your gear being drawn and placed in a circular pattern. Think of the time this would save!

The Array command can define a pattern in two different methods. Patterns can be defined in either a:

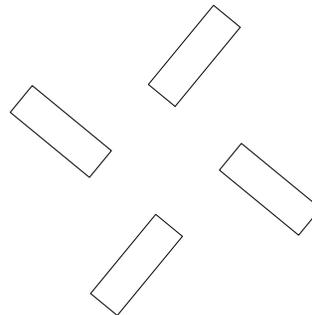
- Rectangular Array (pattern of rows and columns).
- Polar Array (circular pattern)

Let's try each method!

Arrays



Rectangular



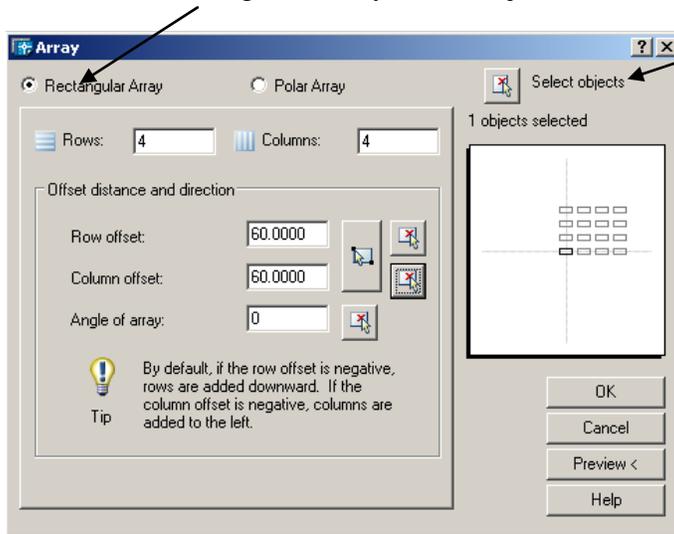
Polar



Using the Rectangular Array Command

Practice Exercise 1

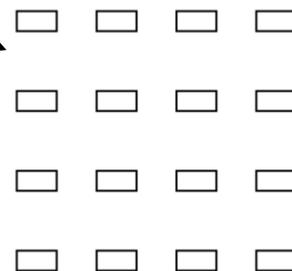
1. Open Prototype
2. Draw a rectangle with the following coordinates 30 long by 15 high.
3. Click on the **Array** button in the Modify toolbar and the Array window shown below should open.
4. Select the Rectangular Array as the object to be modified.



5. Click on the Select Objects button and select the rectangle and press enter to return to this window.

6. Input the following:
7. Number of rows: 4
8. Number of columns: 4
9. Next, enter the unit cell or distance between rows. Type in 60.
10. Enter the distance between columns as 60.
11. Click on OK.

12. Notice what happens. You have just created a Rectangular group of rectangles with 4 rows and 4 columns which are 20 units apart.
13. Show the drawing to your teacher and have them sign your activity sheet.
14. Close this file and do not save it.

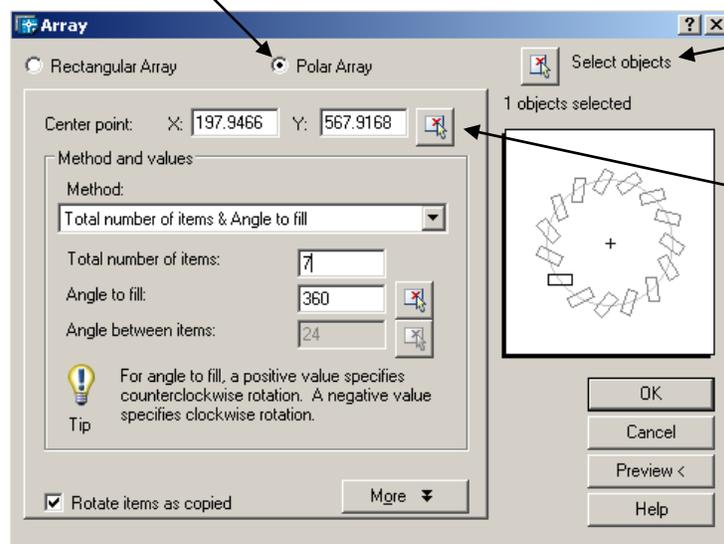




Using the Polar Array Command

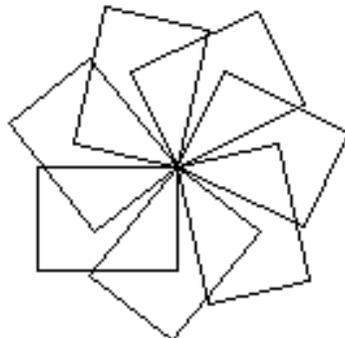
Practice Exercise 2

1. Open Prototype 1.
2. Draw a rectangle with the following coordinates 40 long by 30 high.
3. Click on the Array button in the modify toolbar and the Array window shown below should open.
4. Click on Polar Array.



5. Click the select objects button and select the rectangle and press enter to return to this window.
6. Next, the center point of the Array must be defined. Click on the Pick Center Point button.
7. Select the upper right corner of the rectangle by using the snap command and press enter.
8. The number of items in the Array will be entered next.
9. Input the number 7.

10. The default angle of rotation is 360 degrees.
11. The last option is to rotate the objects being copied, make sure this box is selected.
12. Notice what happens. The copied rectangles are rotated and placed in a 360-degree array.
13. Your finished drawing should look like the one below.
14. Show your drawing to your instructor and have them sign your activity sheet.
15. Close the file and do not save the changes.



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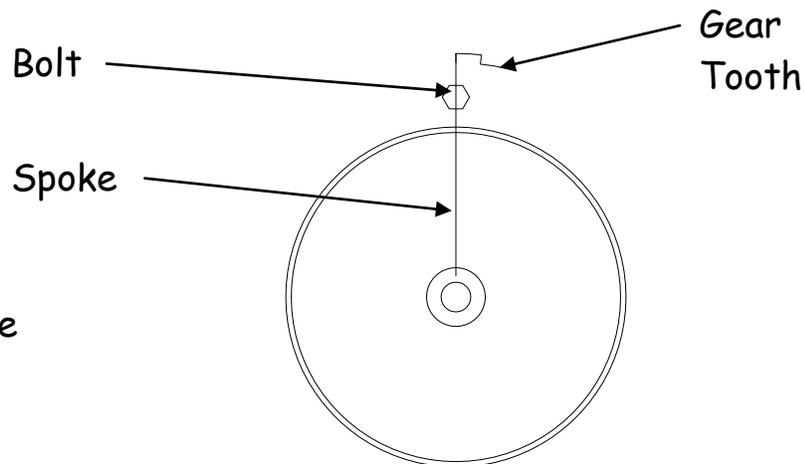


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Activity 5 Exercise

Exercise 1

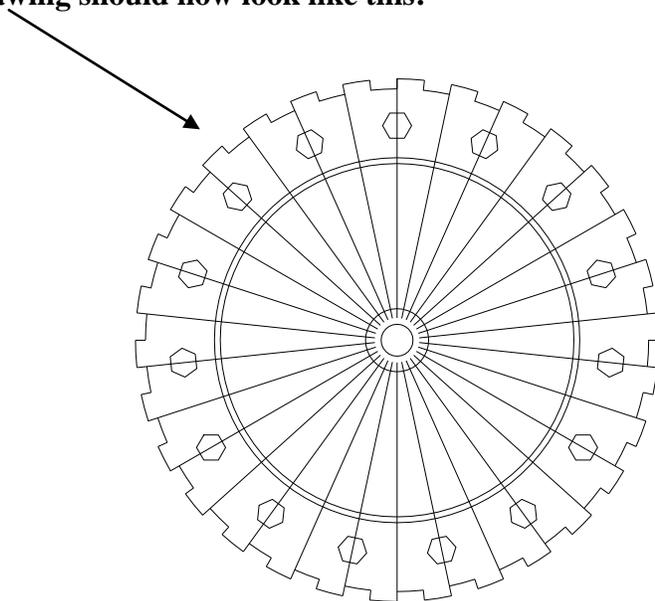
1. From the StuShare folder, open the gear.dwg file.



Your drawing
should look like
this!

2. Using the **Array** command, copy and array the gear teeth and spoke 30 times around the outside of the outer circle.
3. In addition, copy and array the six sided bolt 15 times.

Your drawing should now look like this:



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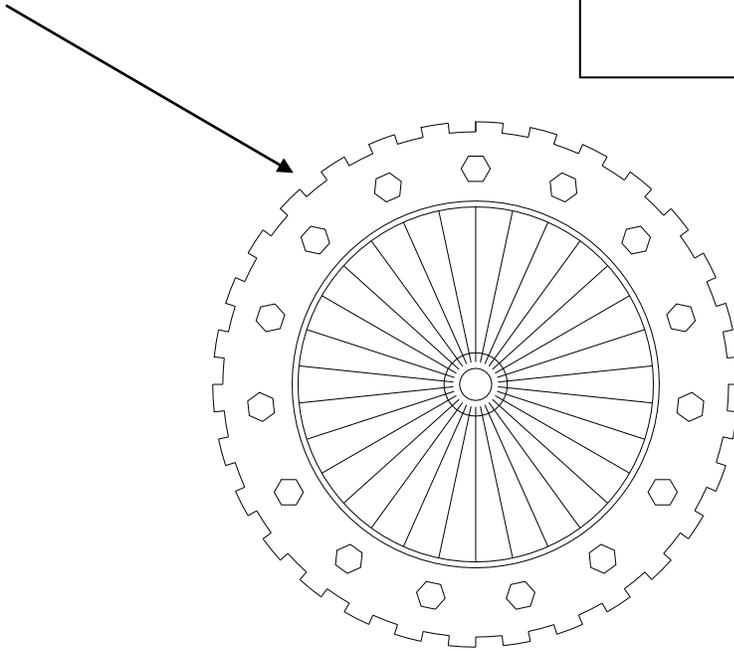


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- Using the Trim command, trim each spoke so that they only reach the inner circle on the gear.

Your drawing should look like this:

Refer back to Activity 3 for instructions on using the Trim Command.



- Save your work as **Act5 Ex1**.
- Have your teacher check your work and sign your activity sheet.
- Close the file.



Activity 5 Review Questions

1. The Array button is found in the
 - A. Draw toolbar.
 - B. Snap toolbar.
 - C. Modify toolbar.
 - D. Command toolbar.
2. There are _____ different types of patterns that can be used with the Array function.
 - A. three
 - B. one
 - C. four
 - D. two
3. The cell distance between rows and columns when using the Rectangular Array function is
 - A. The distance between objects.
 - B. The distance between objects plus the unit size of the object.
 - C. The combined default distance.
 - D. None of the above.
4. When using Polar Array, the first thing that has to be determined is
 - A. The width of rows and columns.
 - B. The number of items in the array.
 - C. The center point of the array.
 - D. The rotation angle of the array.
5. The default angle of rotation for the Polar Array command is
 - A. 90 degrees.
 - B. 360 degrees.
 - C. 180 degrees.
 - D. 270 degrees.



Activity 6 Career Research Project

For this project, you will watch the **AutoCAD career video** that is saved in the StuShare drive. You will also use the Internet to research two careers associated with AutoCAD as a career. Prepare a one page report as outlined below.

Project Name	Career Researched
Career Details	<p>Included in the report:</p> <ul style="list-style-type: none"> ■ Introduce the careers, explaining details of your chosen careers, including, what education requirements are needed to study in this career, the expected salary, potential employment, and any other information you may feel relevant. ■ Use descriptive vocabulary, images, career details, and effective presentation techniques to present your careers. ■ Express and explain your career plan. ■ Use the Internet to gather/research information on colleges offering training in AutoCAD. ■ Draw a conclusion based on your career information.
Materials and Resources	Reference Internet sites used based on your research and identify software applications used in your report/presentation.

Submit your final report to your teacher.

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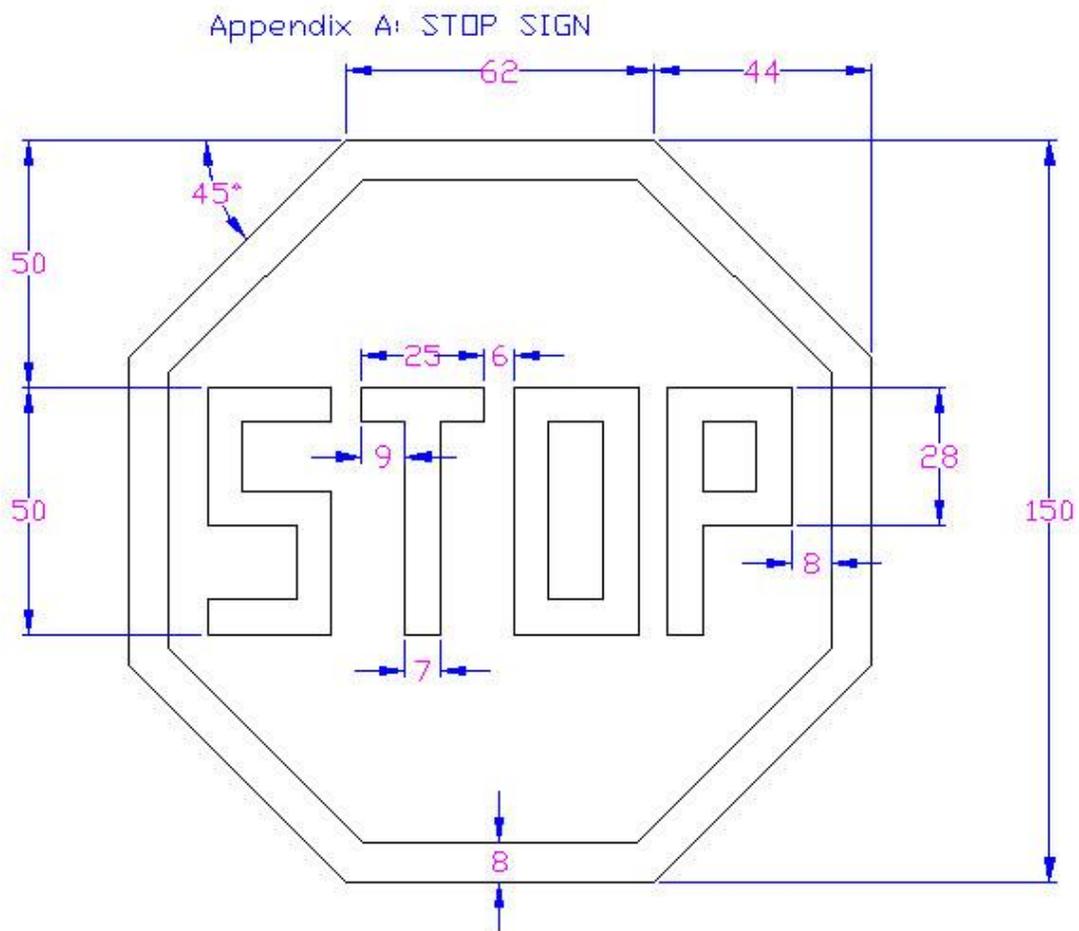


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Activity 7

Using all the information that you have studied throughout this module create the following drawings shown below. All coordinates that you will need are shown.

Note: Drawing Area is 250 X 200

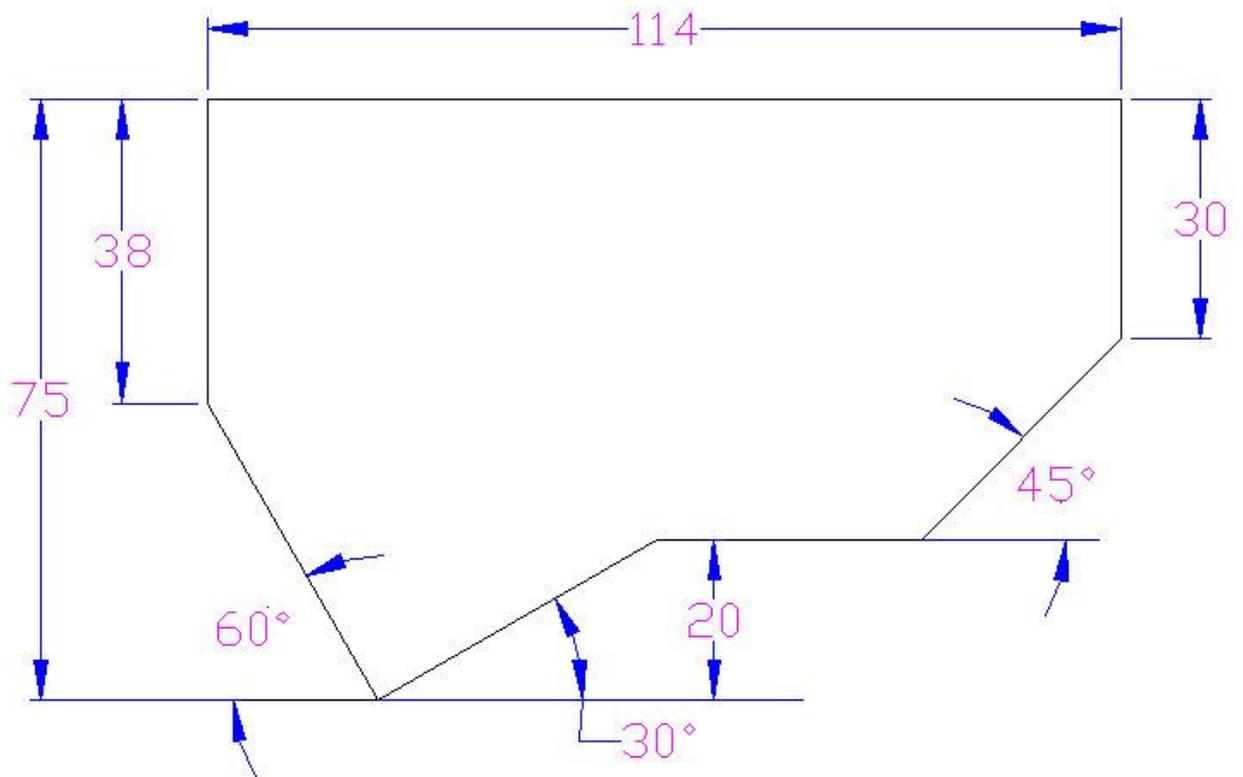


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Appendix A: Drawing 1

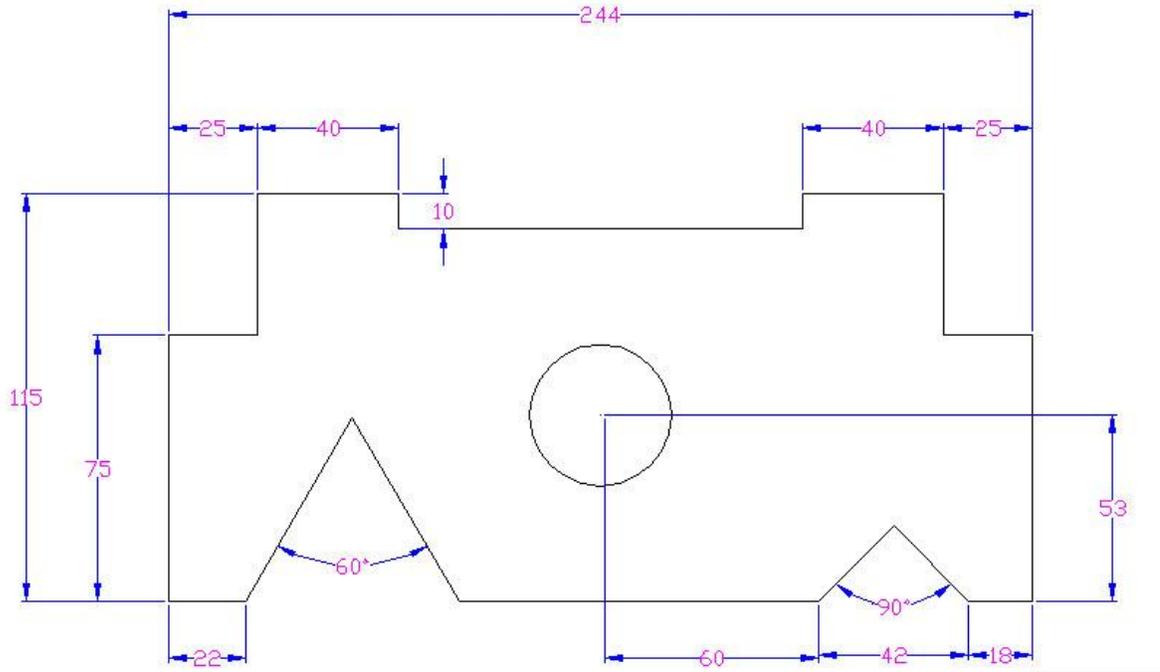


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Appendix A: Drawing 2



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