

# *Introduction to Computer Aided Drafting*

## **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.

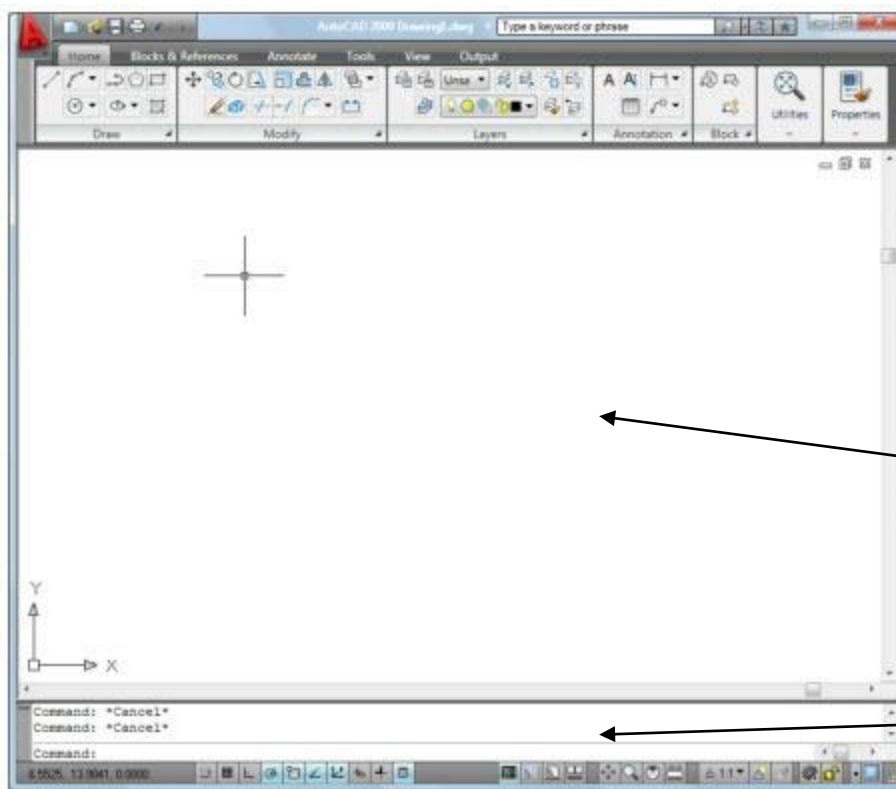
**Let's get started.....**

# Introduction to Computer Aided Drafting

## Activity 1 Introduction to AutoCAD LT



1. On your computer desktop, you will see the AutoCAD LT 2009 icon.
2. Double click on the icon to access the AutoCAD LT 2009 program.
3. The AutoCAD LT 2009 default “2D Drafting and Annotation” workspace will now appear on your screen.



The **horizontal ribbon** displays at the top of the drawing window by default when you create or open a drawing. The ribbon can also be placed at the bottom of the drawing window.

Drawing Window

Command Window

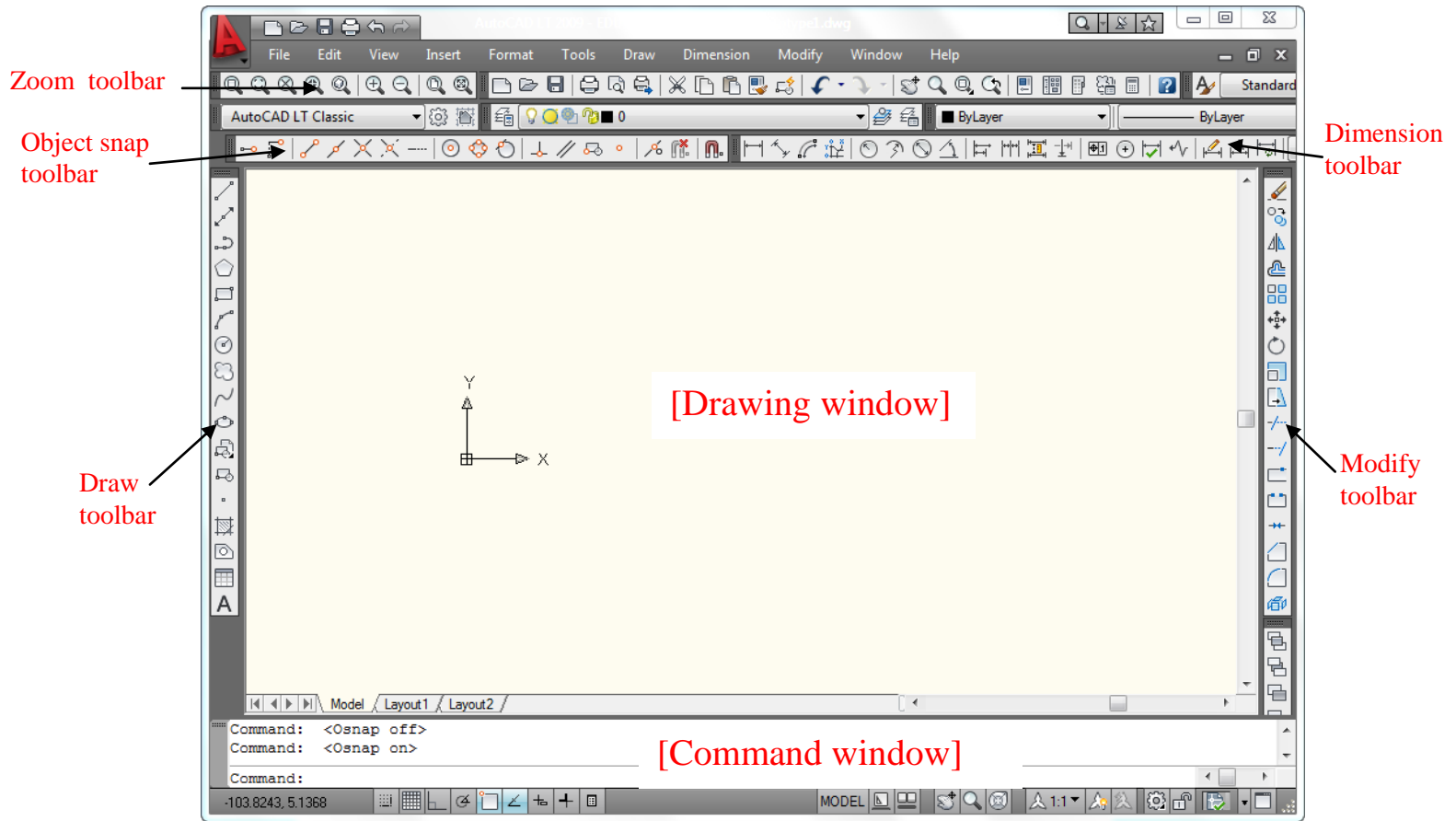
**IMPORTANT!:** We won't be using this workspace. We will use the **AutoCAD LT Classic** workspace which conforms more closely to these instructions. The AutoCAD LT Classic workspace is shown on the next page of these instructions.



Watch [this video](#) to see how to switch to the correct workspace for this module.

# Introduction to Computer Aided Drafting

The video that you watched (linked on the previous page) showed you how to get the AutoCAD Classic workspace which is shown in the image below. We will use this workspace throughout this module:



You may arrange your toolbars any way that you wish; just make sure that you know how to identify each of the commonly used toolbars.

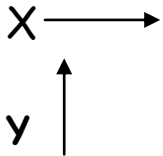
# *Introduction to Computer Aided Drafting*

The **Modify Toolbar** contains tools that will modify objects that already appear on your screen. You will use the Modify toolbar or 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 2009 is not open, double click on the icon to start the program.
2. Click on File and then click on Open.
3. Go to your AutoCAD Module folder on U:\ drive and open the folder.
4. Double click on the Student Files folder.
5. 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.**

# *Introduction to Computer Aided Drafting*

## 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 2009. 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**.
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).



# Introduction to Computer Aided Drafting

## Practice Exercise 2

1. Using absolute coordinates, let's 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. Also, you can click the **Zoom Extents** tool in the **Zoom** toolbar.

**Really Important**

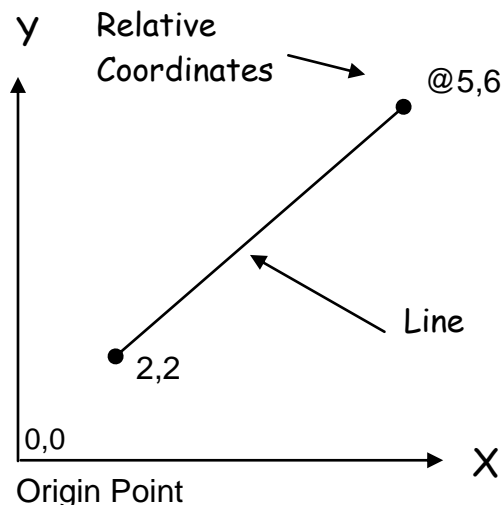
**Note:** Remember the video on using Coordinate Entry. The name of the video is **Relative** and was shown to the class by the teacher.

**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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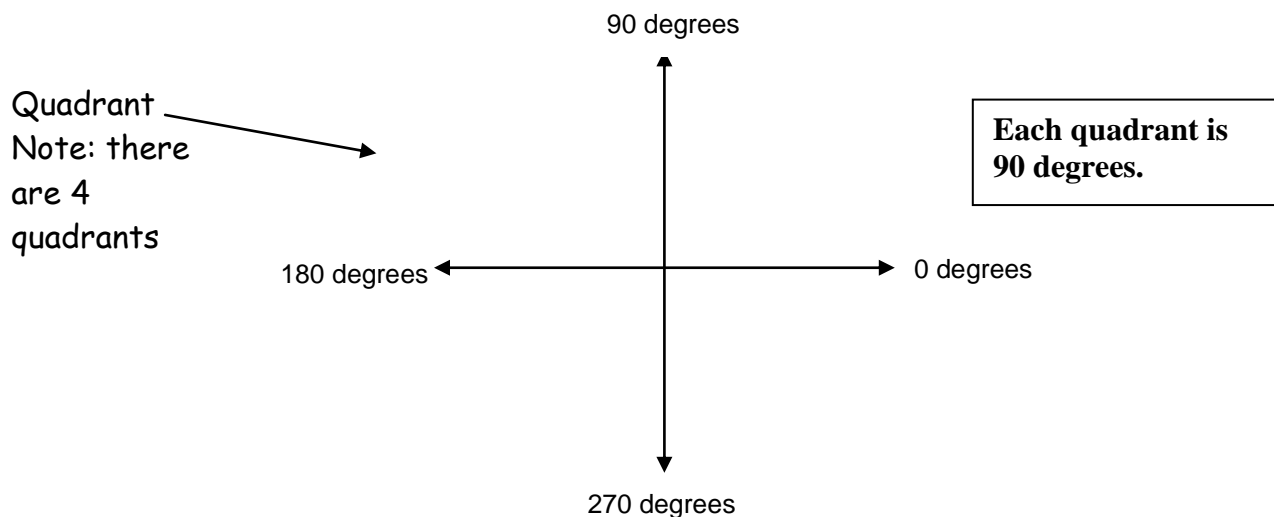
## Practice Exercise 3

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:** Remember the second video on using Coordinate Entry at this time. The name of the video is **Polar** and was shown to the class by the teacher.

**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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## **Practice Exercise 4**

1. Open **Prototype 1.dwg** (from the Student Files folder).
2. Click on the Line Tool in the Draw toolbar.
3. In the Command Window it will say Command: \_line specify first point
4. Place your mouse anywhere on the screen and click the mouse button.
5. Enter the following coordinates: @100<0 (this is the end point of our line using Polar Coordinates) and press Enter
6. Enter coordinates @50<90, press Enter
7. Enter coordinates @100<180, press Enter
8. Enter coordinates @50<270, press Enter
9. Press Enter again or right click your mouse to finish the drawing.
10. Place dimensions on your drawing.
11. Save the drawing in your **AutoCAD Assignments** folder on U:\ drive using filename: **act1\_pr\_ex4**
12. Close the drawing (don't close the AutoCAD program).

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. Open **Prototype 1.dwg** (from the Student Files folder).
2. Click on the Rectangle Tool in the Draw toolbar
3. Click your mouse anywhere on the drawing screen.
4. Now in the command line it asks for "other corner".
5. Type @100,50 and press Enter.
6. Place dimensions on your drawing.
7. Save the drawing in your **AutoCAD Assignments** folder on U:\ drive, filename: **act1\_pr\_ex5**
8. Close the drawing (don't close the AutoCAD program).

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).



# Introduction to Computer Aided Drafting

## **Practice Exercise 6**

1. Open **Prototype 1.dwg** (from the Student Files folder).
2. Click on the Circle tool in the Draw toolbar. 
3. The Command Window will now ask you to define the center point of the circle.
4. Click your mouse in the center of the screen
5. 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).
6. Enter a radius coordinate of 40 in the command line then press Enter.
7. Try drawing another circle following the same procedure, but changing the radius to 50.
8. Place dimensions on your drawing.
9. Save the drawing in your **AutoCAD Assignments** folder on U:\ drive, filename:  
**act1\_pr\_ex6**
10. Close the drawing (don't close the AutoCAD program).


## 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.

Watch this video for a better understanding of using the Polygon tool.

[Video A](#): Polygons **inscribed in a circle**

## **Practice Exercise 7**

1. Open **Prototype 1.dwg** (from the Student Files folder).
2. Select the Polygon tool in the Draw toolbar. 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. Place dimensions on your drawing.
12. Save the drawing in your **AutoCAD Assignments** folder on U:\ drive, filename:  
**act1\_pr\_ex7**
13. Close the drawing (don't close the AutoCAD program).

[Video B](#): Polygons **circumscribed about in a circle**

[Video C](#): Implications for later activities

# Introduction to Computer Aided Drafting

**So far, you have learned:**

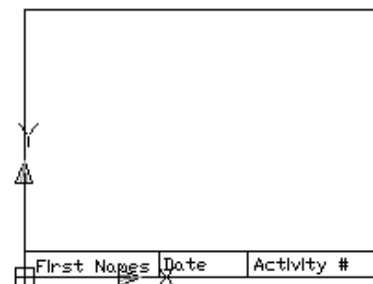
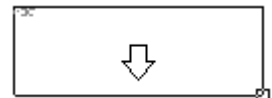
✓ What CAD is.	✓ The AutoCAD LT interface.
✓ The Drawing Tool Bar.	✓ The Modify Tool Bar (we will look at this more later).
✓ How to draw lines using coordinates.	✓ How to use the Command Window.
✓ How to draw Coordinate System (X,Y).	✓ rectangles using coordinates.
✓ How to draw circles using coordinates.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

## Activity 1 Exercises



### **Exercise 1** -- [Watch this Video](#)

1. Open **Prototype 1.dwg** (from the Student Files folder).
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 using Absolute Co-ordinates. 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 text window and a Text Formatting tool bar 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.
13. Place dimensions on your drawing.
14. Save your work, filename **Title**, into your **AutoCAD Assignments** folder.
15. Close the drawing (don't close the AutoCAD program).

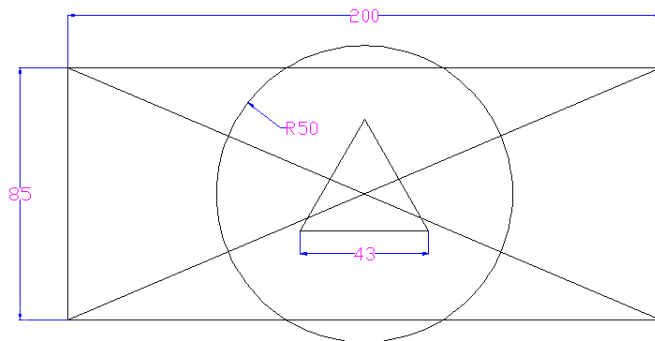


# Introduction to Computer Aided Drafting

## Exercise 2

1. Open Prototype1. Make sure that **OSNAP** is turned on at the bottom of the screen.
2. Click on the Rectangle tool and click anywhere on the screen to begin the drawing.
3. Draw a rectangle with the following coordinates: 200 long by 85 high (@200,85)
4. Click the line tool and move the mouse to the upper left corner of the rectangle.
5. When a small pink box appears on the corner of the rectangle click your mouse.
6. Move your mouse to the lower right corner until you get the same pink box and then click your mouse again.
7. Right click your mouse to shut off the line command.
8. Select the line tool again and draw another line from the opposite corners.
9. Select the circle tool.
10. The center point of the circle is the intersection of the 2 lines.
11. Move your mouse to where the lines intersect in the center and notice the pink x that appears.
12. Click your mouse when the pink x is visible.
13. Radius should be 50.
14. Select the polygon tool.
15. When the command window asks how many sides, choose 3.
16. The center point for the polygon is the intersection point of the 2 lines again.
17. The polygon is Inscribed.
18. Radius is 25.
19. Place dimensions on your drawing.
20. Save your activity in your **AutoCAD Assignments** folder, filename **act1\_ex2**.

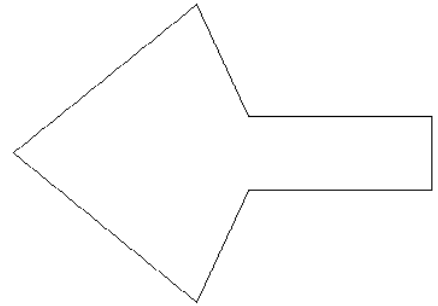
**Your finished CAD drawing should look like this:**



# Introduction to Computer Aided Drafting

## Exercise 3

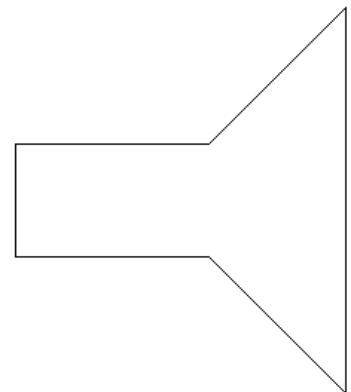
1. Open the document named **Prototype1**.
2. Select Line from the Draw toolbar.
3. Point 1 - Place your mouse on the screen and click anywhere.
  - a. Point 2 - @1.75, -1.5 Press Enter
  - b. Point 3 - @.5, 1.125 Press Enter
  - c. Point 4 - @1.75, 0 Press Enter
  - d. Point 5 - @0, .75 Press Enter
  - 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
4. Notice that the relative coordinates you entered are all based from the previous point entered.
5. Save your work as **act1\_ex3\_left\_arrow** in your **AutoCAD Assignments** folder.



## Exercise 4

1. Start the AutoCAD LT2009 software program if it is not open already.
2. Open the document named **Prototype1**.
3. Use the Line command with Relative and Polar coordinates and follow the directions below to complete the drawing.
4. Point 1 Place your mouse on the screen and click anywhere.
  - Point 2 @1, 0 (Relative)
  - Point 3 @1<315 (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 **AutoCAD assignments** folder, filename **act1\_ex4\_right\_speaker**.

**Note:** Angles are drawn using Polar Coordinates





### Activity 1 Review Questions

Now, you are going to go to an online testing site, find the quiz for the review questions for this first activity and answer those 5 questions.

- Go to <http://quizstar.4teachers.org/indexs.jsp> and login with the username and password given to you by your teacher (ignore the “Sign Up” option; you are already signed up by your teacher).
- Click on “Untaken Quizzes”.
- Find “**AutoCAD Activity 1 Review Questions**” in the list and click “Take” for that quiz.
- Complete the quiz.
- Logout and exit from the quiz site.
- Then move on to Activity 2 which starts on the next page of this instructions document.



# Introduction to Computer Aided Drafting

## Activity 2 Introduction to Object Snap

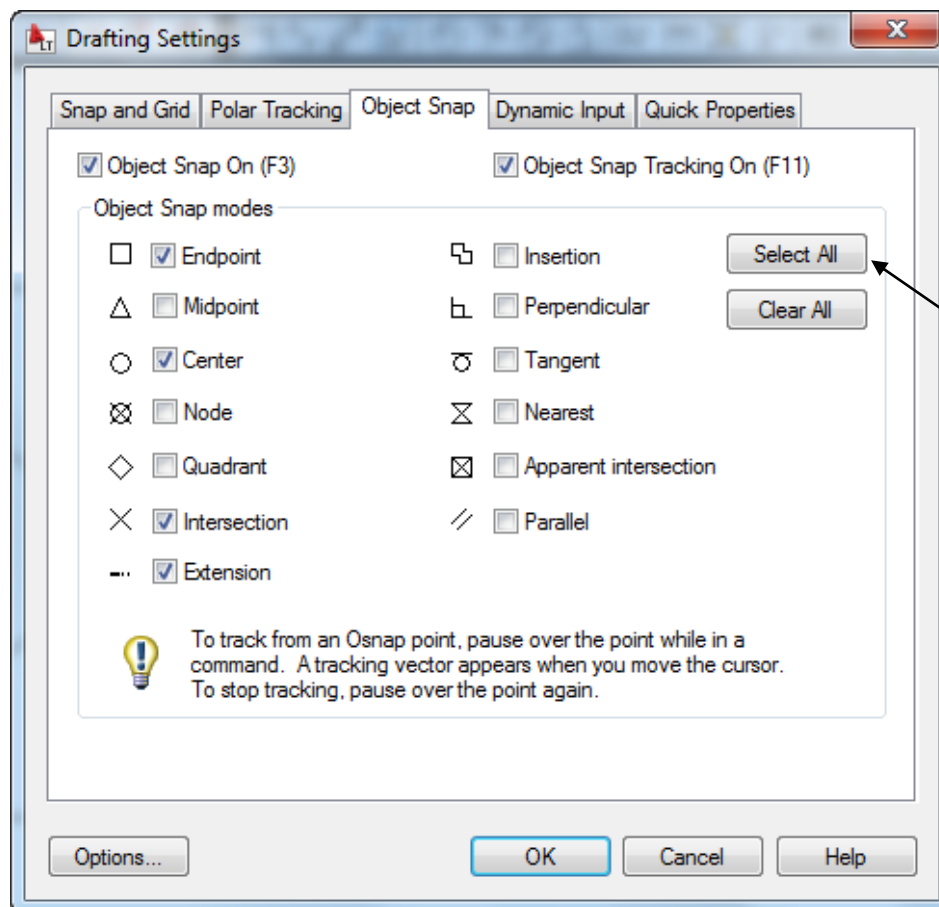
Object Snap is another feature that is available in CAD software packages. An important feature about Object Snap is that you can connect directly to a specific point in a drawing without knowing the coordinates. In this activity, we are going to learn how to use Object Snap, and we will learn about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

1. Before you get started, go to the **Object Snap Toolbar**



and click on the **Osnap Settings** button.

2. The Drafting Settings window will open with the **Object Snap** tab selected.
3. Click **Select All**.
4. All the Snap functions should now be selected.
5. Click **OK**.



Click  
Select All

# *Introduction to Computer Aided Drafting*

## **Object Snap Toolbar**

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

1. Click on the **Tools** menu.
2. Click on **Toolbars** and then on **AutoCAD LT**.
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 dark tab with two small dotted lines at one end of the toolbar, hold down the left mouse button and drag the toolbar to your desired docking location.



**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 using the rectangle tool.
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. Save the drawing in your AutoCAD Assignments folder on U:\ drive, filename:  
**act2\_pr\_ex1\_endpoint**

**Before we move to the next snap object, we are going to clear the screen. To do this, drag a box around the drawing and press the delete key on the keyboard.**



**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.

# Introduction to Computer Aided Drafting

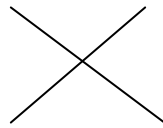
3. Click on the **Snap to Midpoint** button in the Snap Toolbar.
4. Place your mouse at the center of one of the rectangle lines. You will notice the center is highlighted with a pink triangle.
5. Click to accept the midpoint.
6. Click on the Snap to Midpoint button again.
7. Move your mouse to the center 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. Save the drawing in your **AutoCAD Assignments** folder on U:\ drive, filename:  
**act2\_pr\_ex2\_midpoint**



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

## Practice Exercise 3

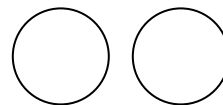
1. Draw two lines that intersect or cross each other.
2. Click on the Line tool and then the **Snap to Intersect** button.
3. Move your mouse to the point where the two lines intersect.
4. Click to snap to the intersect point.
5. Move your mouse down and click again to draw a straight line.
6. Right click or press enter to stop the line command.
7. Save the drawing in your **AutoCAD Assignments** folder on U:\ drive, filename:  
**act2\_pr\_ex3**



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

## Practice Exercise 4

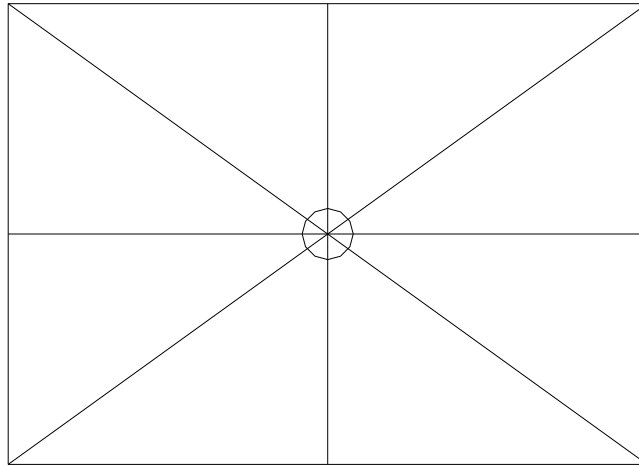
1. Draw two circles.
2. Click on the line tool.
3. Click on the Snap to Center button.
4. Move your mouse towards the center of one of the circles.
5. When the center is highlighted with a small pink circle, click to accept the point.
6. Click on the Snap to Center button again.
7. Connect the line to the center of the second circle.
8. Right click or press enter.
9. Save the drawing in your AutoCAD Assignments folder on U:\ drive, filename:  
**act2\_pr\_ex4**



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

# *Introduction to Computer Aided Drafting*

## **Activity 2 Exercises**



### **Exercise 1**

1. Open the file **Prototype1**.
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. Place dimensions on your drawing.
12. Save your drawing as **act2\_ex1**.



## Activity 2 Review Questions

Now, you are going to go to an online testing site, find the quiz for the review questions for this second activity and answer those 5 questions.

- Go to <http://quizstar.4teachers.org/indexs.jsp> and login with the username and password given to you by your teacher.
- Click on “Untaken Quizzes”.
- Find “**AutoCAD Activity 2 Review Questions**” in the list and click “Take” for that quiz.
- Complete the quiz.
- Logout and exit from the quiz site.
- Then move on to Activity 3 which starts on the next page of this instructions document



# Introduction to Computer Aided Drafting

## 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.

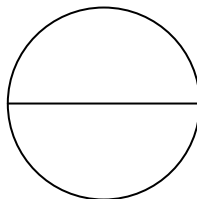
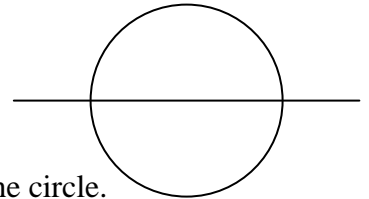
### Using the Trim Command



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**.
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 look like the one below.
13. Save the drawing in your AutoCAD Assignments folder on U:\ drive, filename:  
**act3\_pr\_ex1**




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



# Introduction to Computer Aided Drafting

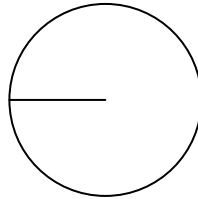
## Using The Extend Command



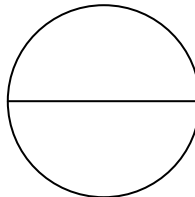
The **Extend** command is the opposite of the Trim command. The Extend is used when you want to extend a line to an object. The extend command is a two step process, just like the Trim command.

### Practice Exercise 2

1. Open the file **Prototype1**.
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:



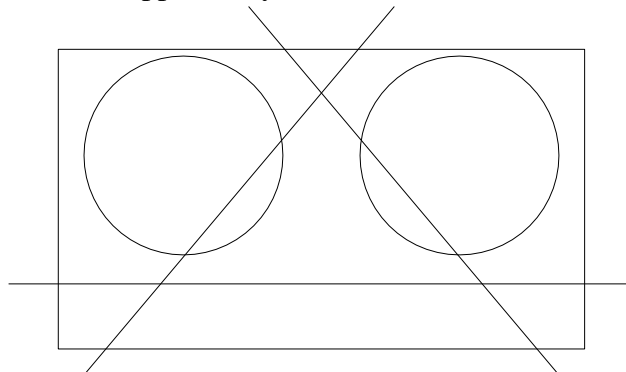
12. Save the drawing in your AutoCAD Assignments folder on U:\ drive, filename:  
**act3\_pr\_ex2**

# Introduction to Computer Aided Drafting

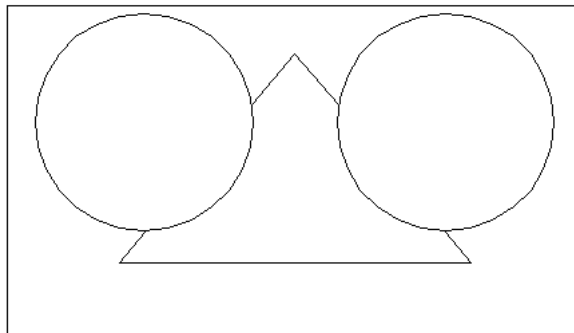
## Activity 3 Exercises

### Exercise 1

1. Make sure that the AutoCAD LT 2009 software is open.
2. Click on File and then click on Open.
3. Select **Activity3PartA.dwg** located in your AutoCAD Module\Student Files folder.
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.

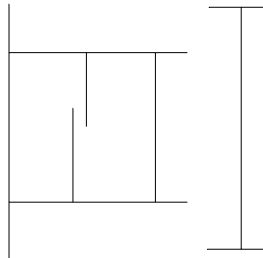


6. Save the drawing in your **AutoCAD Assignments** folder on U:\ drive, filename: **act3\_ex1**.

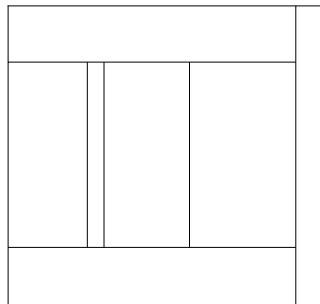
# *Introduction to Computer Aided Drafting*

## **Exercise 2**

1. Make sure that the AutoCAD LT 2009 software is open.
2. Click on File and then click on Open.
3. Select **Activity3PartB.dwg** located in your AutoCAD Module\Student Files folder.
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 the drawing in your AutoCAD Assignments folder on U:\ drive, filename: **act3\_ex2**



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



# *Introduction to Computer Aided Drafting*

## **Activity 3 Review Questions**

Now, you are going to go to an online testing site, find the quiz for the review questions for this third activity and answer those 5 questions.

- Go to <http://quizstar.4teachers.org/indexs.jsp> and login with the username and password given to you by your teacher.
- Click on “Untaken Quizzes”.
- Find “**AutoCAD Activity 3 Review Questions**” in the list and click “Take” for that quiz.
- Complete the quiz.
- Logout and exit from the quiz site.
- Then move on to Activity 4 which starts on the next page of this instructions document



# Introduction to Computer Aided Drafting

## 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. Open Prototype1.
2. Draw a rectangle on your screen with the following coordinates 150 long by 250 high.
3. Click on the **Move** button in the Modify toolbar.
4. You will be prompted to select an object.
5. Click on the rectangle and press Enter.
6. In the command window, you will see the command Base Point or Displacement.
7. **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).**
8. The top right hand corner of the rectangle will be the Base Point.
9. Place your mouse near the top right corner of the rectangle and click to select it.
10. Notice a line is drawn from the Base Point to the rectangle that you can now move.
11. Next, you will be prompted for a second point of displacement.
12. This is the location that you want to move the object to.
13. Enter the coordinates @40,0 and press Enter.
14. The rectangle moved 40 units to the right.
15. Move the rectangle again.
16. This time use the upper left corner as the base point and move the rectangle 50 units straight up. (@0,50)
17. Now, move it again, this time 200 units straight to the left using Polar.
18. Keep this file open, because you need to show it to your teacher before closing the file. But first, you will move on and do the next exercise.



#### 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.

# *Introduction to Computer Aided Drafting*

## **Practice Exercise 2**

1. Open the file **Prototype1**.
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. Now, copy the circle on the right 200 units straight down using polar.
12. Show your work for this and the previous activity to your teacher.

**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. Save the drawing in your AutoCAD Assignments folder on U:\ drive, filename:  
**act4\_pr\_ex3\_multiple\_copies**

## **Mirror Command**

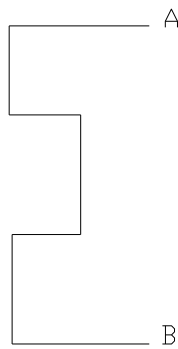


The **Mirror** command allows you to create a mirror image of an object. This is especially important when creating drawings that are symmetrical. The Mirror command allows you to copy and flip an object 180 degrees.

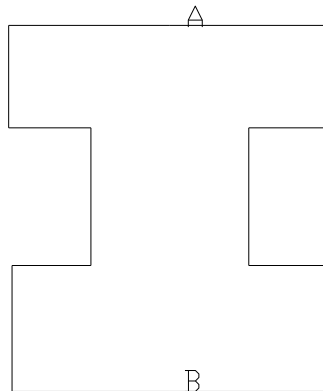
# Introduction to Computer Aided Drafting

## **Practice Exercise 4**

1. Open the **act4\_pr\_ex4\_mirror** file from your AutoCAD Module\Student Files folder.
2. 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.**



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. Place dimensions on your drawing.
13. Save the drawing in your AutoCAD Assignments folder on U:\ drive, filename:  
**act4\_pr\_ex4\_mirror**

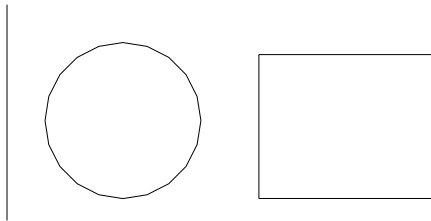
# Introduction to Computer Aided Drafting

## Try This!

### Using the Erase Command

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.**



This time select erase and click on the edge of the circle and rectangle.

7. Both items are selected.
8. Press enter and they both erase.
9. 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.

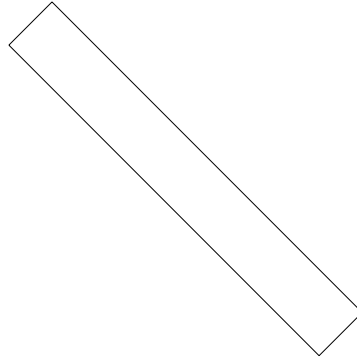
# Introduction to Computer Aided Drafting

11. Type 45 and press Enter (**this means 45 degrees**).
12. The rectangle should now appear as below in Figure 2.

**Figure 1**



**Figure 2**



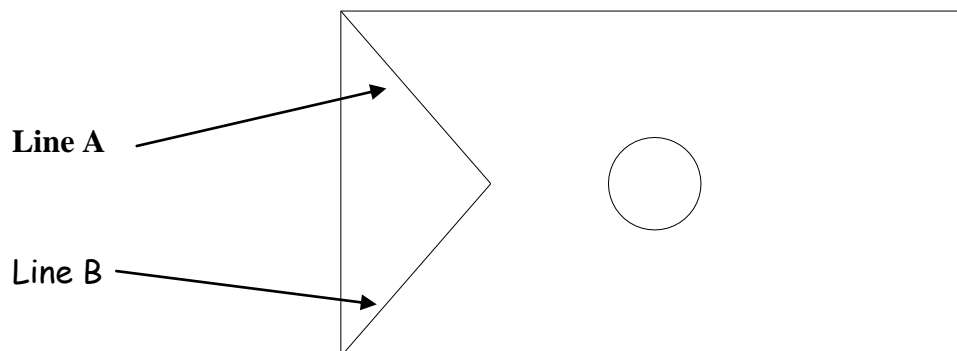
13. Save the drawing in your AutoCAD Assignments folder on U:\ drive, filename:  
**act4\_pr\_ex5**
14. Close the file and do not save the changes.

## 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 your AutoCAD Module\Student Files folder.

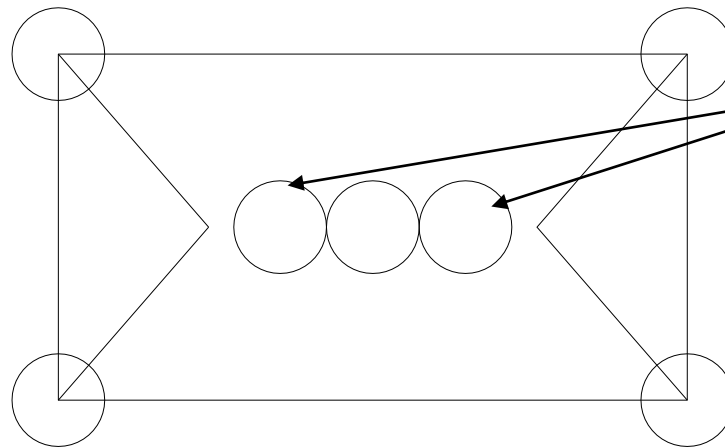


2. Using the **Mirror** command, create a mirror image of lines A and B shown above. They are to be relocated at the opposite end of the rectangle.



# Introduction to Computer Aided Drafting

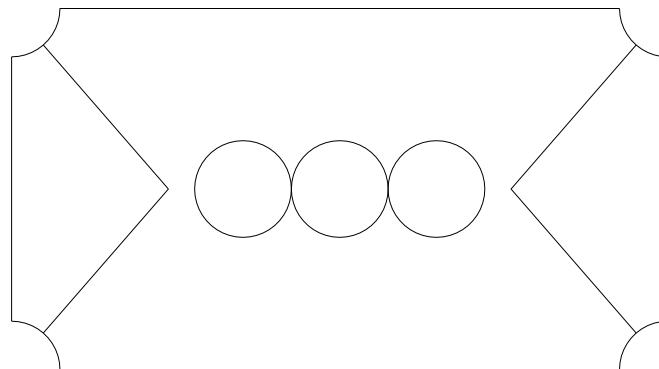
3. Using the **Copy** command, copy the circle in the center of the drawing and place it in six different locations.
4. In order to do this, you will need to use the Snap commands to define the correct locations of the circles (base point or displacement). Refer to the drawing below:



Place these two circles such that: (1) their edges are just touching -- not overlapping and no gap between their edges [zoom in to confirm], and (2) they are perfectly horizontal. [see the *Snap to Quadrant* option in the hint below]

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

5. Using the Trim command, trim the four circles in the corners so they become arcs as shown in the drawing below:
6. Save your drawing as **act4\_ex1**.



# *Introduction to Computer Aided Drafting*



## **Activity 4 Review Questions**

Now, you are going to go to an online testing site, find the quiz for the review questions for this fourth activity and answer those 4 questions.

- Go to <http://quizstar.4teachers.org/indexs.jsp> and login with the username and password given to you by your teacher.
- Click on “Untaken Quizzes”.
- Find “**AutoCAD Activity 4 Review Questions**” in the list and click “Take” for that quiz.
- Complete the quiz.
- Logout and exit from the quiz site.
- Then move on to Activity 5 which starts on the next page of this instructions document.

# Introduction to Computer Aided Drafting

## Activity 5 Using the Array Tool

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



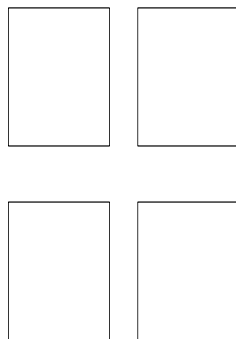
Imagine that you are designing a set of gears for a mountain bike. You have drawn a circle and one tooth of the gear. The Array command would let you copy that one tooth and place it in a pattern that you define. The result would be all of the teeth for your gear being drawn and placed in a circular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

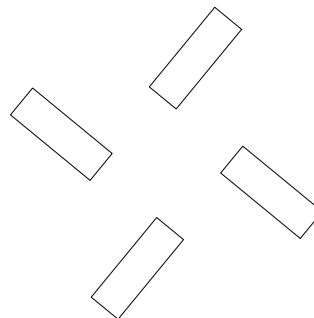
- Rectangular Array (pattern of rows and columns).
- Polar Array (circular pattern)

Let's try each method!

### Arrays



***Rectangular***



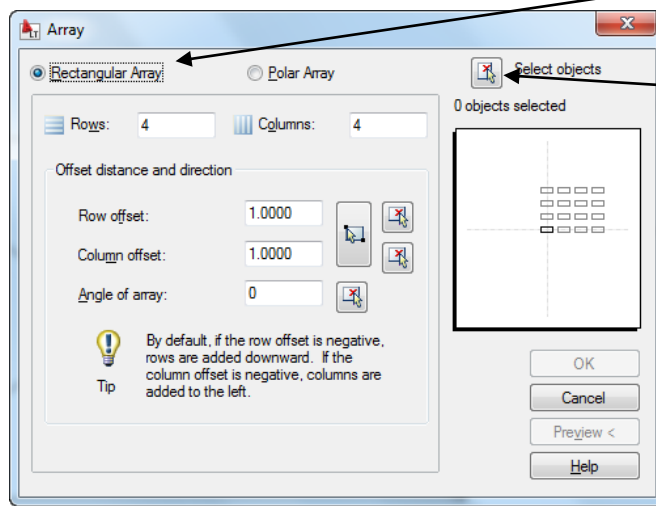
***Polar***

# Introduction to Computer Aided Drafting

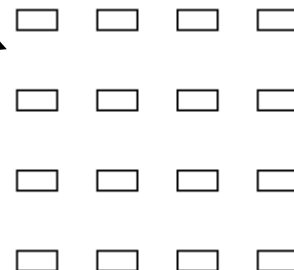
## Using the Rectangular Array Command

### Practice Exercise 1

1. Open Prototype1
2. Draw a rectangle 30 units long by 15 units 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 row offset (meaning the distance between similar points on objects in rows). Type in 60 for Row offset.
10. Enter 60 for Column Offset.
11. Click on OK.
12. Notice what happens. You have just created a Rectangular group of rectangles with 4 rows and 4 columns.  
(Do you understand why the gaps between rows are larger than the gaps between the columns? If not, ask your teacher.)
13. Save the drawing in your AutoCAD Assignments folder on U:\ drive, filename: **act5\_pr\_ex1\_rect\_array**

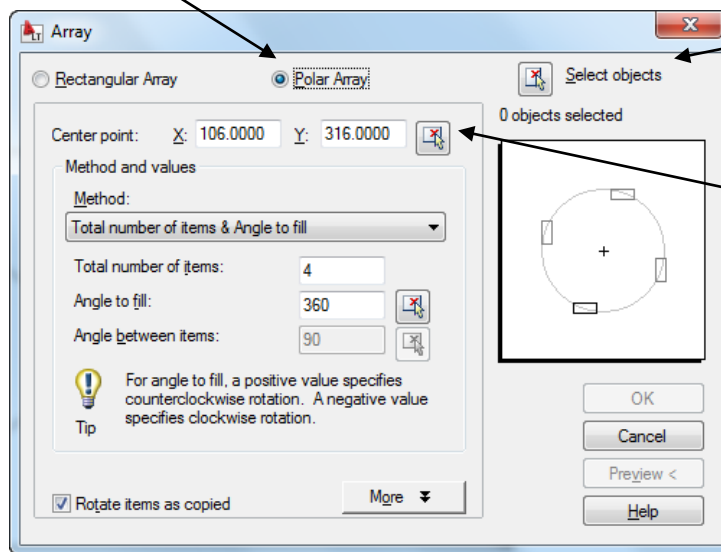


# Introduction to Computer Aided Drafting

## Using the Polar Array Command

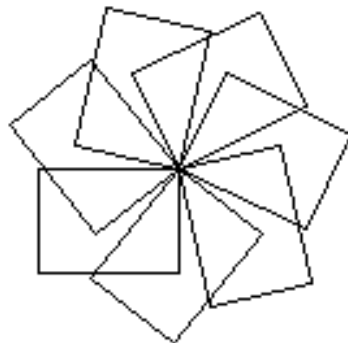
### Practice Exercise 2

1. Open Prototype1.
2. Draw a rectangle 40 units long by 30 units 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. Save the drawing in your **AutoCAD Assignments** folder on U:\ drive, filename:  
**act5\_pr\_ex2\_polar\_array**



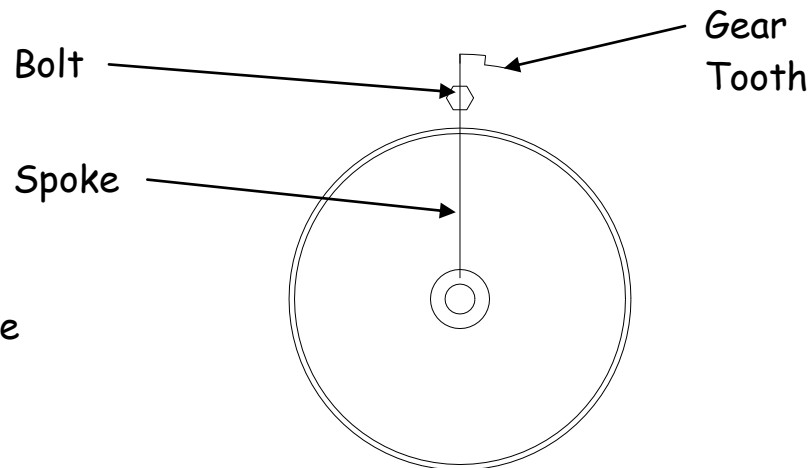


# Introduction to Computer Aided Drafting

## Activity 5 Exercise

### Exercise 1

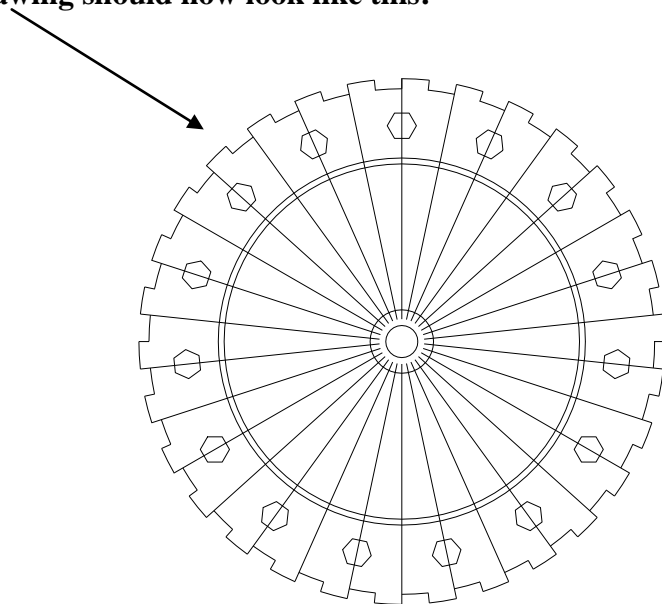
1. From your Autocad Module\Student Files folder, open the gear.dwg file.



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:**

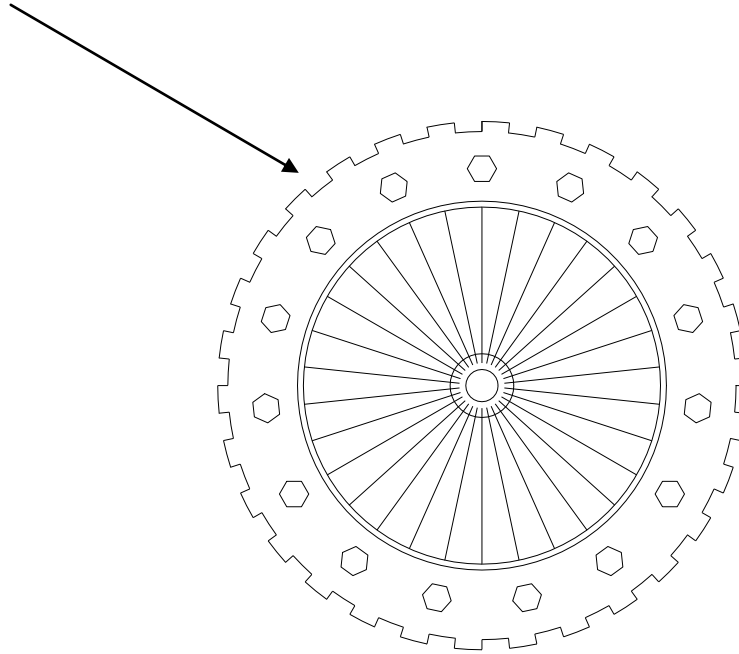


4. Using the Trim command, trim each spoke so that they only reach the inner circle on the gear.

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

# *Introduction to Computer Aided Drafting*

**Your drawing should look like this:**



5. Save your work as **act5\_ex1\_gear**.
6. Close the file.



### Activity 5 Review Questions

Now, you are going to go to an online testing site, find the quiz for the review questions for this fifth activity and answer those 5 questions.

- Go to <http://quizstar.4teachers.org/indexs.jsp> and login with the username and password given to you by your teacher.
- Click on “Untaken Quizzes”.
- Find “**AutoCAD Activity 5 Review Questions**” in the list and click “Take” for that quiz.
- Complete the quiz.
- Logout and exit from the quiz site.

Then move on to Activity 6 which starts on the next page of this instructions document

# *Introduction to Computer Aided Drafting*

## Activity 6 Career Research Project

For this project, you will watch the **AutoCAD career video** (double-click on the career video file in your AutoCAD “Student Files” folder). You will also use the Internet to research **two careers** associated with AutoCAD as a career and prepare a one page report (**half a page for each career**) in Microsoft Word as outlined below.

-- First career:

- Identify the career (for example: Civil Engineer, Architect, AutoCAD Technician, Mechanical Engineer, etc.)
- List the main duties (three or four main duties)
- List the education requirements
- Identify a **Canadian** salary range
- Identify the current employment potential (lots of jobs available? no jobs available?)
- Identify an **Atlantic Canadian college or university** which offers training for this career.
- Reference Internet sites used based on your research (place the web site credit with the related information in your report.

-- Repeat for second career.

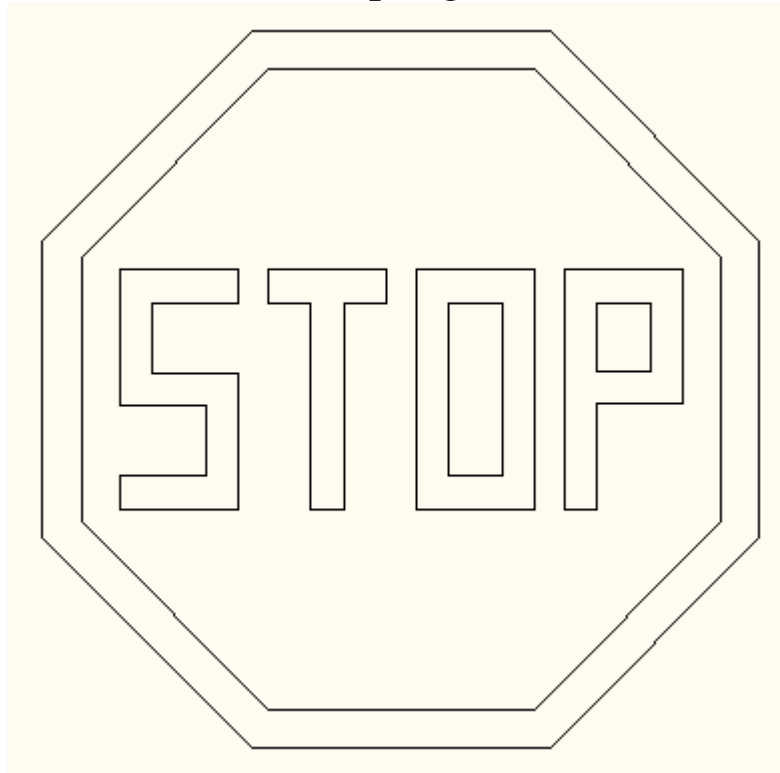
Save the one-page report in your **AutoCAD Assignments** folder on U:\ drive, filename: **act6\_careers.doc**

# *Introduction to Computer Aided Drafting*

## Activity 7

Using all the information that you have studied throughout this module create the following drawings shown below -- don't forget to open Prototype1 first. All coordinates that you will need are shown.

### Stop Sign



See the next page for the dimensions.



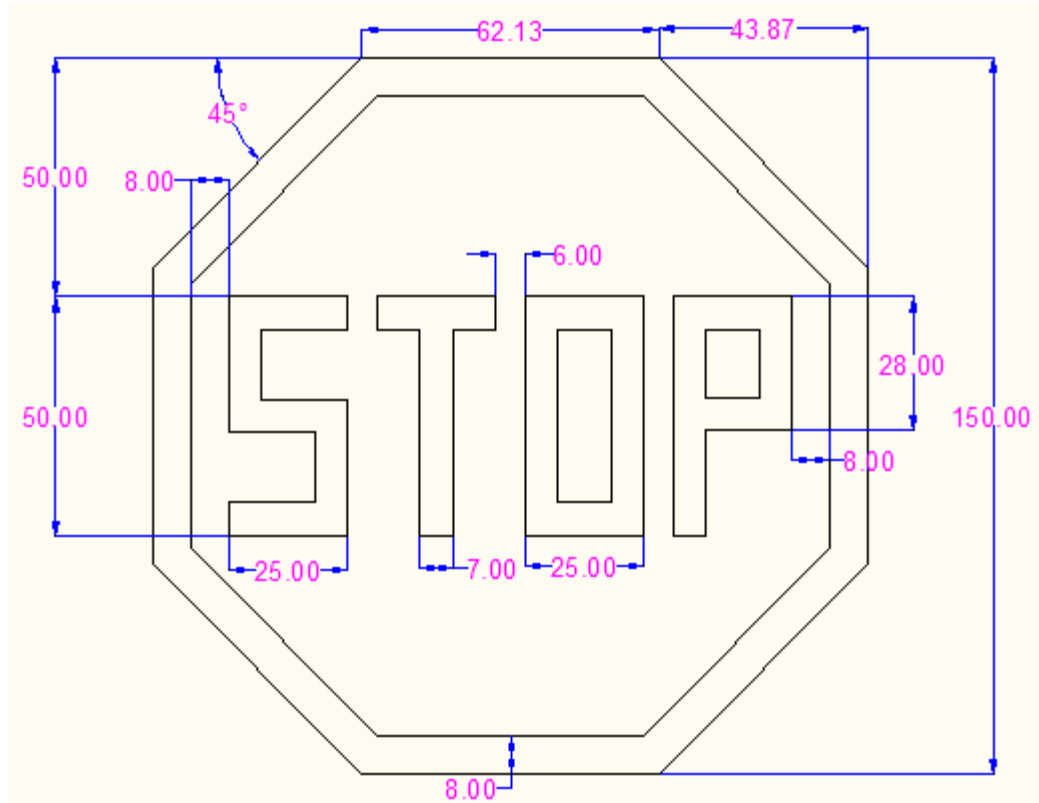
# Introduction to Computer Aided Drafting

Two concentric octagons.

All four letters are the same height and width.

The inside gaps between letters are 6 units.

The width of each line in each letter is seven units.

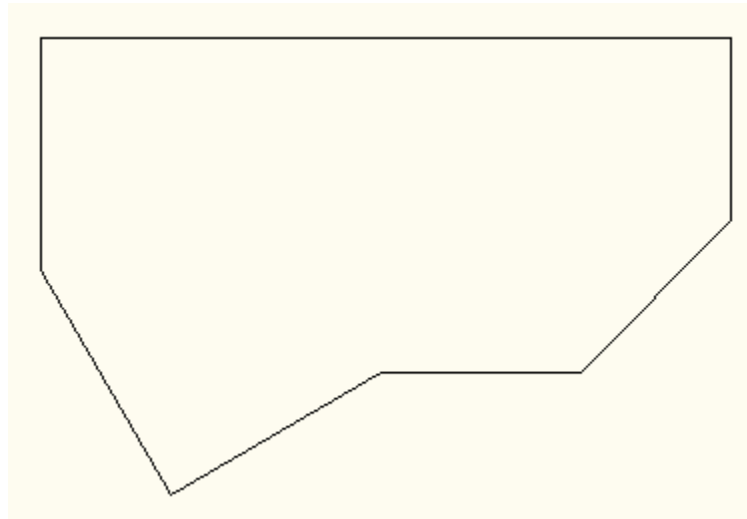


Dimension the drawing as shown in the diagram above.

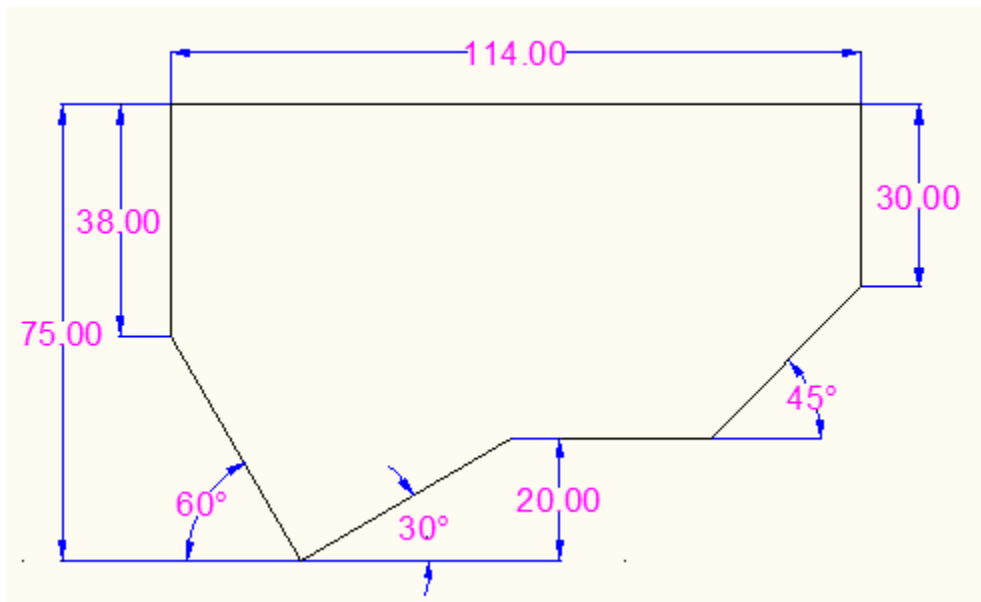
Save as **act7\_stop\_sign**

# *Introduction to Computer Aided Drafting*

## Activity 7 Drawing 2



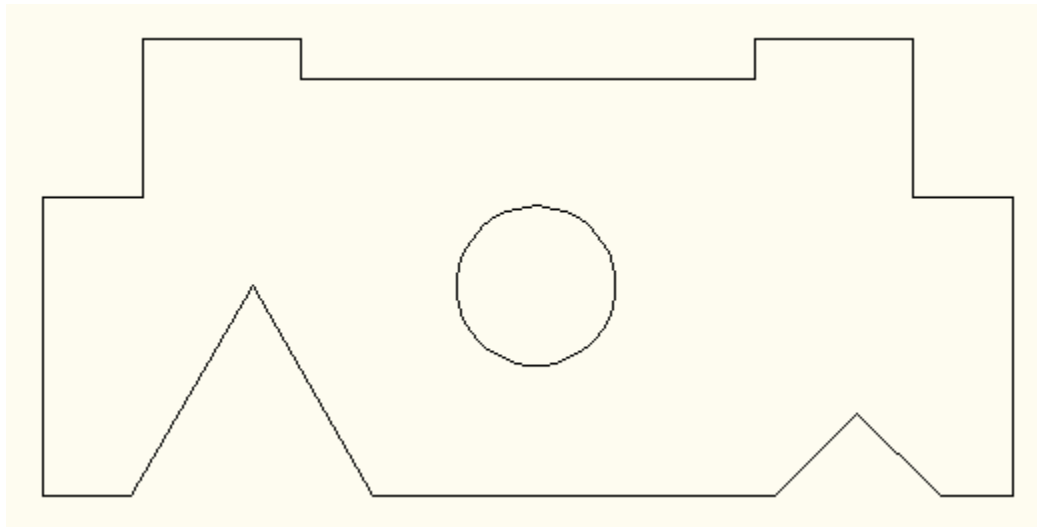
Dimension the drawing as shown in the diagram below:



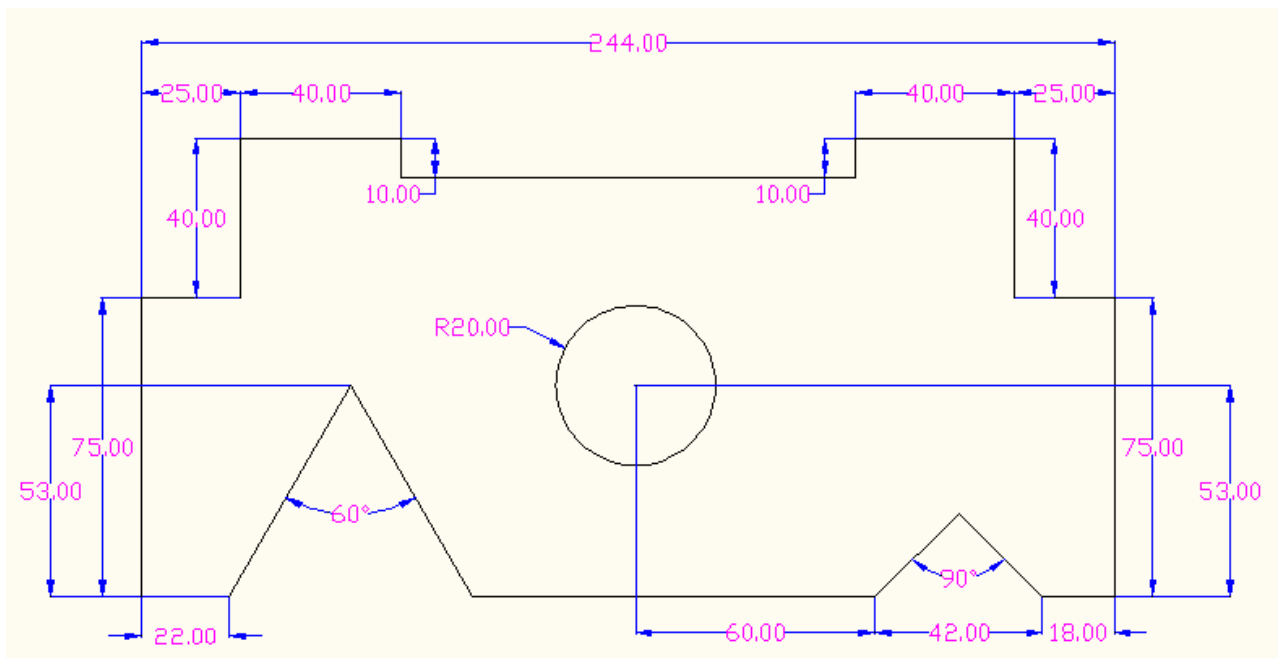
Save as **act7\_drawing\_2**

# *Introduction to Computer Aided Drafting*

## Activity 7 Drawing 3



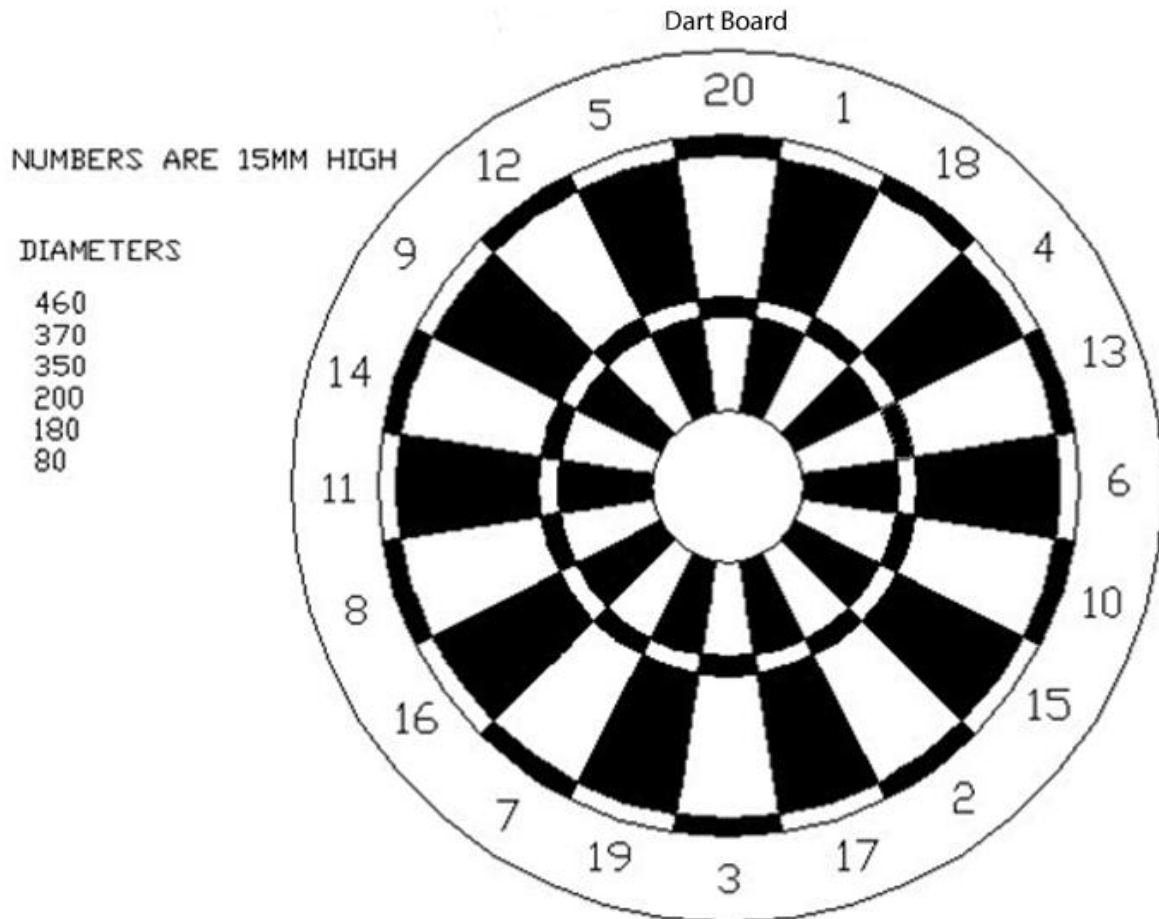
Dimension the drawing as shown in the diagram below:



Save as **act7\_drawing\_3**

# *Introduction to Computer Aided Drafting*

## Bonus Project



Use the Hatch tool to shade areas black (using a pattern of “Solid”)  
The wedges for 20 and 3 are centered vertically.  
When finished, place one diameter dimension on the largest (outer) circle only.

Save as **bonus\_dart\_board**

**When finished, go to the extra bonus opportunity at the bottom of the AutoCAD starting page linked from the BBT class web site.**