Engr 210 AA – Engineering Graphics
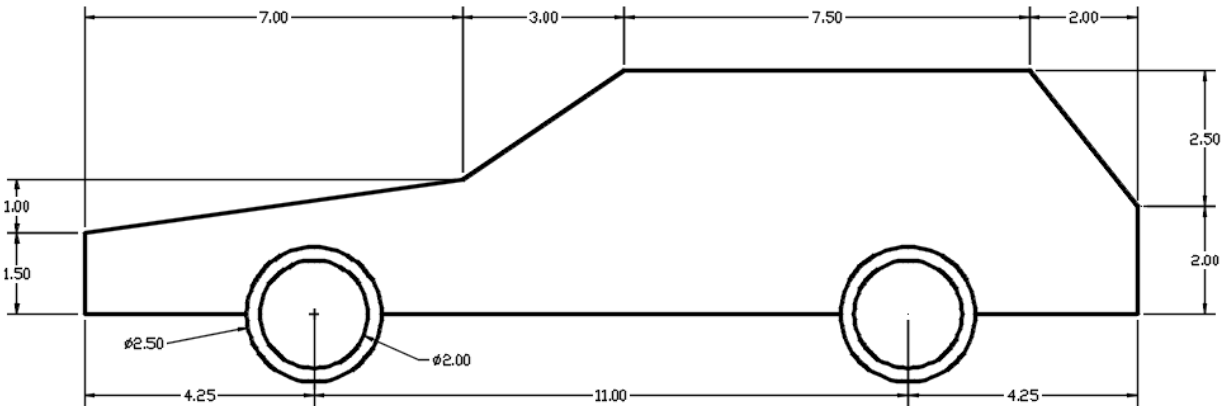
**Lab #1 -Basic Construction and Editing**

Part 1: Basic 2-D Construction

In this exercise basic construction and edit commands will be learned including: creating lines, circles, arcs, and basic dimensioning to create the drawing shown below:


1. Start with a new AutoCAD drawing by selecting the *Qnew* command icon in the *Quick Access* toolbar. The *Select Template* dialog box will appear.
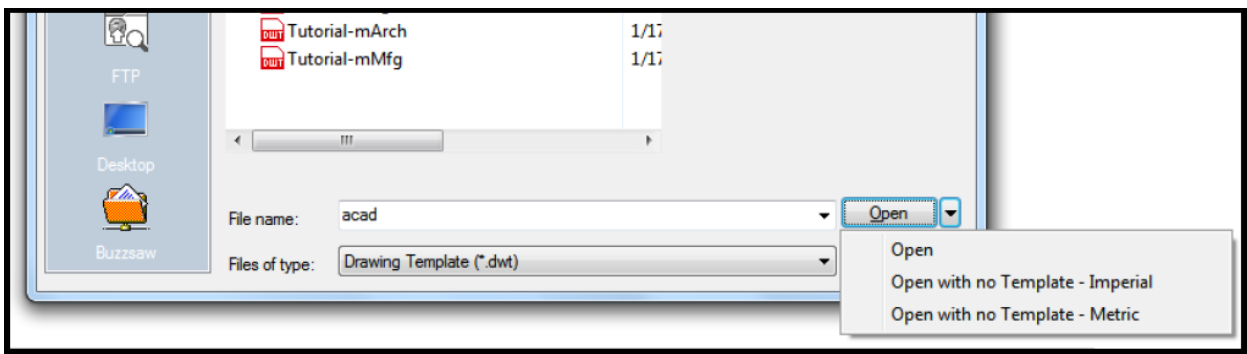
2. In the Select Template manager, **left-mouse-click** on the *triangle* button to the right of the *Open* button, and select *Open with no Template – Imperial*. This sets the default units to inches.
3. Select the *Circle* command icon in the Draw toolbar.
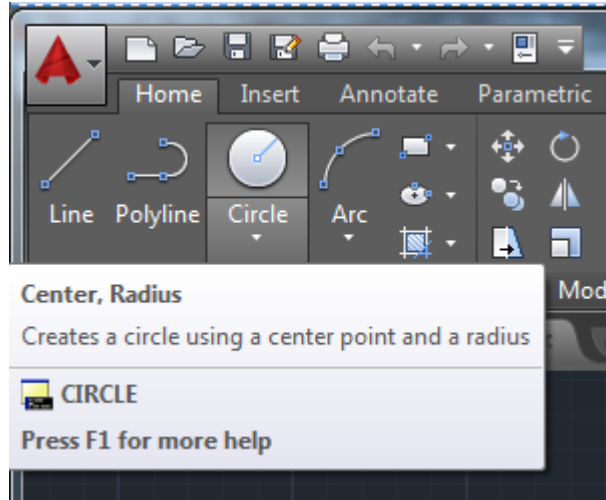
4. In the command prompt area near the bottom of the screen, AutoCAD prompts you to identify the starting point of the circle.
   Command: Specify center point for circle: **0,0 [ENTER]**
   (Type the coordinates 0,0 and press the [ENTER] key).

5. To create the bottom circle, enter the absolute coordinates of the second point. *Specify radius of circle or [Diameter]:* **1.25 [ENTER]**

6. To create the inner circle, identify the starting point of the circle. *Command: Specify center point for circle:* **0,0 [ENTER]**

7. To finish creating the inner circle, enter the radius of the circle. *Specify radius of circle or [Diameter]:* **1 [ENTER]**

8. Now we will create the second set of circles that make up the second wheel. Select the **Circle** command in the Draw tool again. This time the starting point will be on a different point. *Command: Specify center point of circle:* **11,0 [ENTER]**

9. Complete the circle with a radius of 1.25. *Specify radius of circle or [Diameter]:* **1.25 [ENTER]**

10. Now make the inner circle with the same specifications as the previous one. *Command: Specify center point for circle:* **11,0 [ENTER]**

11. Create a circle with a radius of 1. *Specify radius of circle or [Diameter]:* **1 [ENTER]**

12. Now we will create the body of the car. Begin by selecting the ***Line*** command icon in the Draw toolbar.
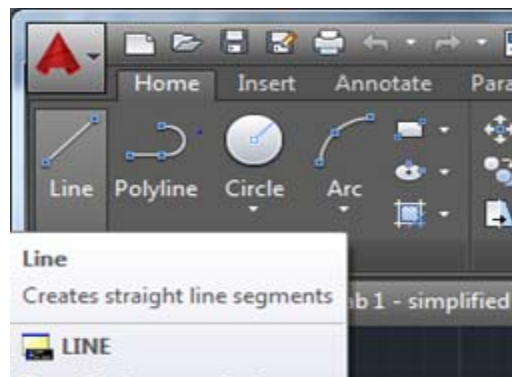

13. In the command prompt area near the bottom of the screen, AutoCAD prompts you to identify the starting point of the line. *Command: _line Specify the first point:* **-1.25,0 [ENTER]**

14. To create the first horizontal line, type the following. *Specify next point or [Undo]*: **-4.25,0 [ENTER]**

15. To create the first vertical line we will use the *relative coordinate* by typing the @ symbol preceding the coordinates *relative to the previous point. Specify next point or [Undo]*: **@0,1.5 [ENTER]**

16. To create the diagonal line feature, without knowing the angle of elevation, use the *relative coordinates* by typing the @ symbol preceding the coordinates *relative to the previous point. Specify next point or [Undo]*: **@7,1 [ENTER]**

17. For the next point, type the following relative coordinates. *Specify next point or [Undo]*: **@3,2 [ENTER]**

18. To create the horizontal line, type the following relative coordinates. *Specify next point or [Undo]*: **@7.5,0 [ENTER]**

19. For the point, type the following relative coordinates. *Specify next point or [Undo]*: **@2,-2.5 [ENTER]**

20. To create the vertical line, type the following relative coordinates. *Specify next point or [Undo]*: **@0,-2 [ENTER]**

21. To create the following horizontal line, use *relative **polar** coordinates*. Use the @ symbol for relative coordinates preceding the distance, and the < symbol preceding the angle. Type the following relative coordinates. *Specify next points or [Undo]*: **@3<180 [ENTER]**

22. To finish off the drawing we need to create the final horizontal line that connects the two wheels together. Select the **Line** tool in the *Draw toolbar*. Then move the mouse over to origin. The line we have to create will have to be along the x-axis, without going through the wheel. To accomplish this we hover the mouse to the right, from the origin until a dotted green line appears to follow. This feature is the *Snap* feature. Left mouse click when a green cross is over the outer circle. Then specify the second point by going along the x-axis until you reach the second wheel. Left mouse click to finish it off. Then click [ENTER]
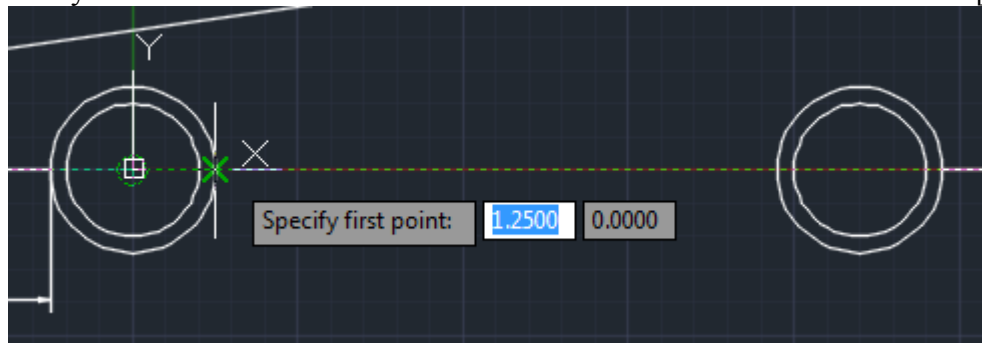
23. To zoom in on the drawing, select the zoom window icon on the Navigation Bar under the View tab. If the Navigation Bar isn't visible, add it to your screen by selecting it under the View tab and selecting Navigation Bar.
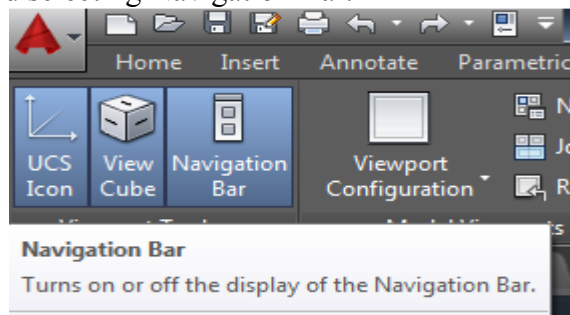
24. To create a window around the drawing, left-mouse-click on one corner of the window when prompted Specify corner of window.
25. **Left-mouse-click** the opposite corner of the window when prompted.
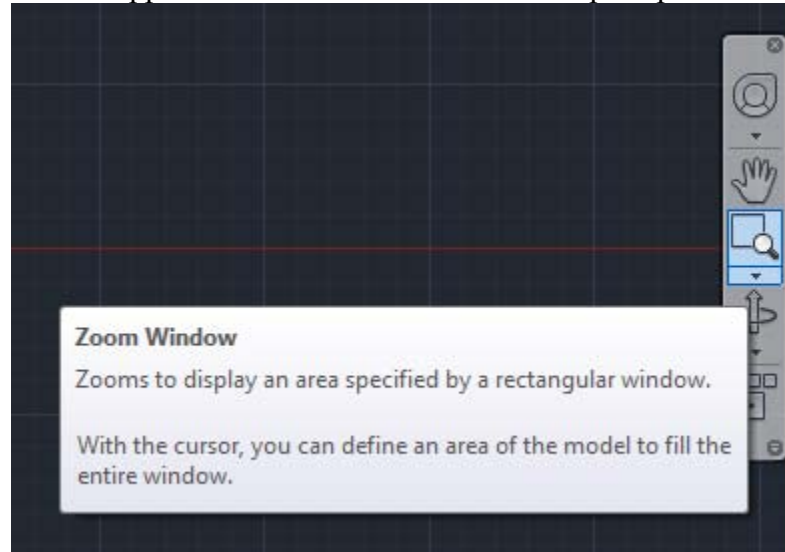

Next dimensions of the features in the drawing will be added using AutoCAD's many dimensioning commands.

26. Before adding dimensions, the default accuracy of dimensions has to be changed. Other dimension settings such as text, arrows, and dimension line properties can be also changed. Click Home tab → Annotation panel → Dimension Style Manager
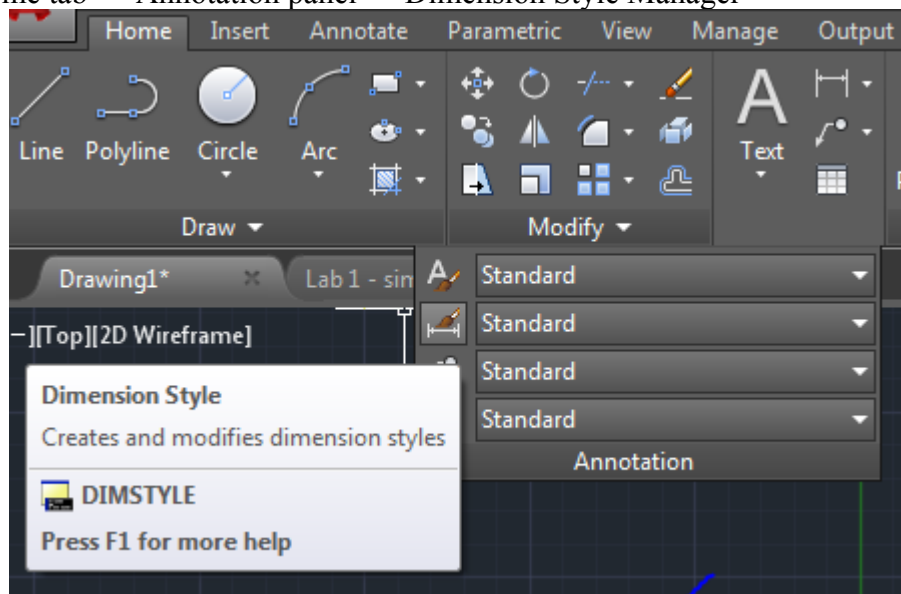
27. The *Dimension Style Manager* dialog box appears on the screen.
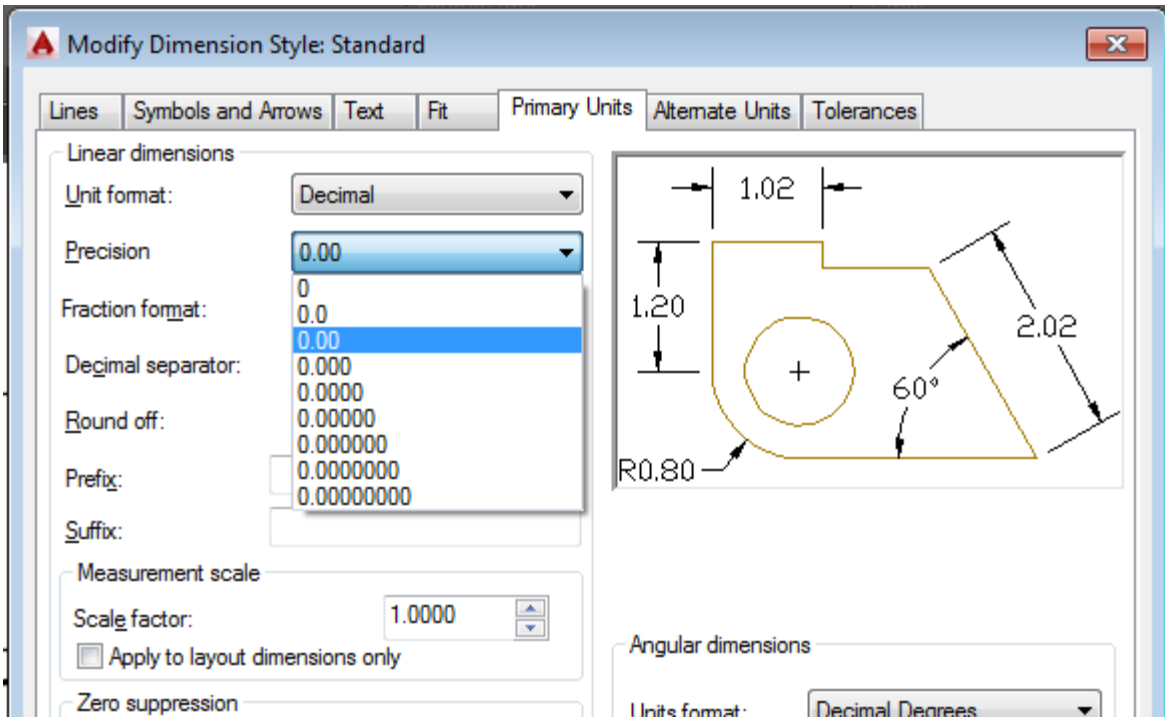
28. Click on **Modify** to bring up the *Modify Dimension Style* dialog box
29. Click on the **Primary Units** tab.
    Select Precision 0.00 to set the default accuracy of the dimensions to be displayed
    to two decimal places. All dimensions added from this point on will be displayed
    with this accuracy.
30. Click on the **OK** button to accept the settings and close the dialog box.
31. Click on the **CLOSE** button to close the *Dimension Style Manager* dialog box.
32. Be sure the *Dimension toolbar* is displayed. To find the *Dimension toolbar*, click
    on the **Annotate tab** on the *Standard toolbar*.
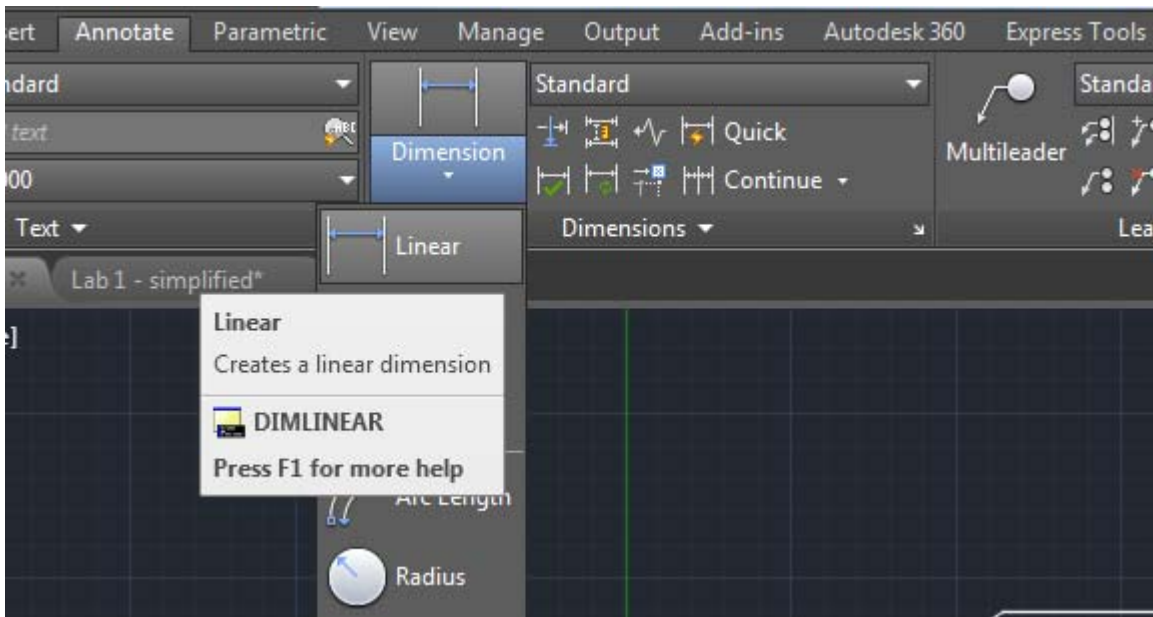
33. From the *Dimension toolbar* **select** *Linear Dimension*.

34. Pick the **origin of a line** when prompted to specify the first extension of the line origin.
35. Pick the **ending of the line** when prompted to specify the second line origin.
36. Pick a point that is about 1.0 inch away from the line to place the dimension text.
37. Repeat the **Linear Dimension** command to add the remaining dimensions.
38. To add the diameter dimensions, from the *Dimension toolbar* select *Diameter Dimension*. Dimension the four circles as shown in the figure on the first page of this lab.
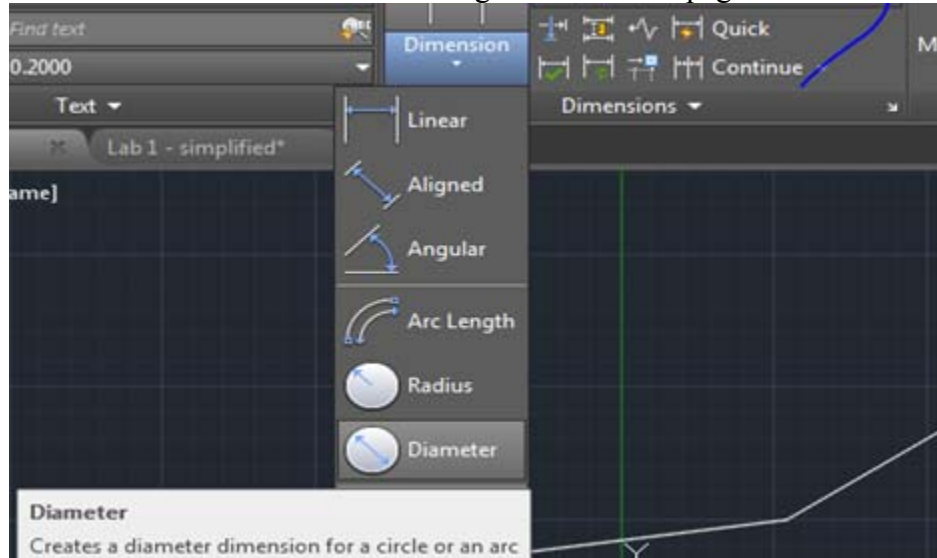

39. In the pull-down menus, select: **File → [Save As]**
40. Save the drawing as **lab1-1**. The file will be saves as an AutoCAD drawing with a default file extension of ".dwg".

Part 2: Drafting Aids

In this exercise, drafting aids such as grid, snap, and object snaps will be used to create the drawing shown below.
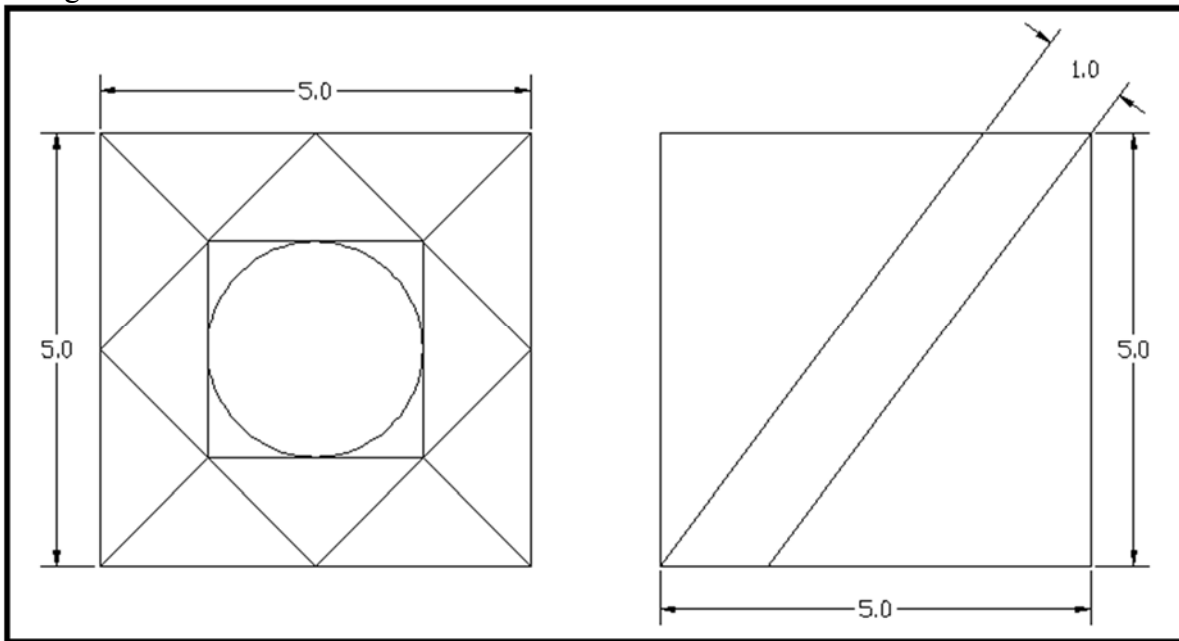
1. Start with a new AutoCAD drawing by selecting the *Qnew* command icon in the *Quick Access* toolbar. The *Select Template* dialog box will appear.
2. In the dialog box, select **Open with no Template – Imperial** to set the default units to inches.
3. Left click the **GRID** button in the *Status Bar* to turn **ON** the *GRID option*. This creates a pattern of dots on the screen. The default spacing between grid points is 0.5 inches.
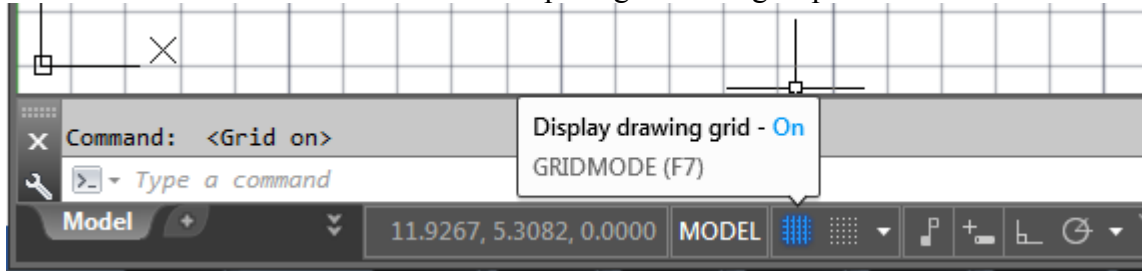

4. Left-click the **SNAP** button in the *Status Bar* to turn **ON** the *SNAP option*. The *SNAP* option controls the cursor movements to specified intervals. The default spacing between snap points is 0.5 inches.
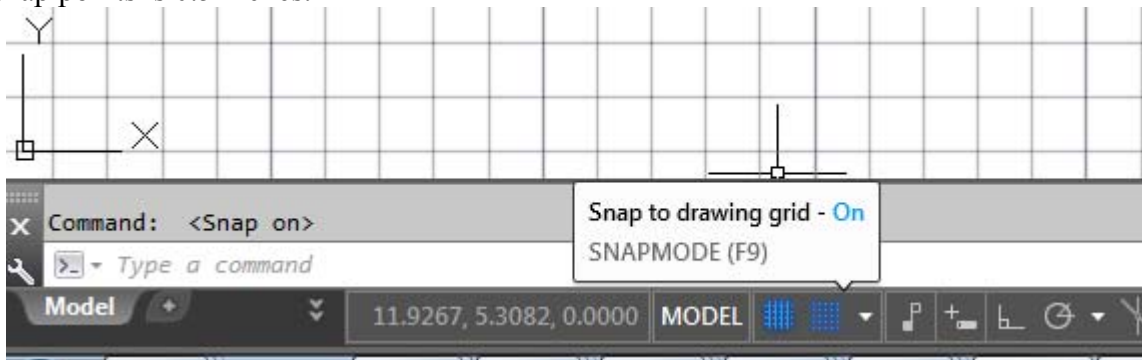
5. Create the two 5-inch squares using the **RECTANGLE** command button in the *Draw* toolbar. Use the cursor to locate the opposite corners of each rectangle while watching the coordinates display at the bottom of the screen. Make the second rectangle about 1.5 inches to the right of the first one.
6. Right-click the **OSNAP** button in the *Status Bar*. Left-click on **Object Snap Settings** to display the *Object Snap Settings* dialog box.
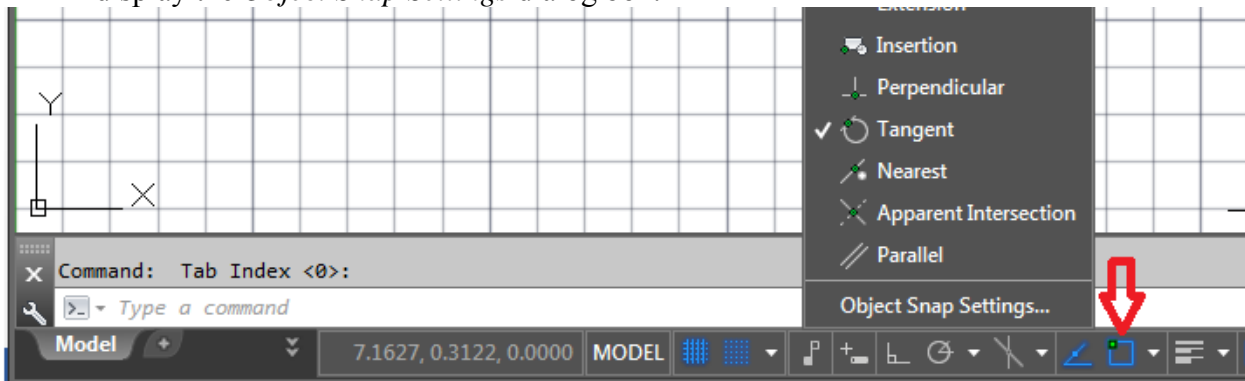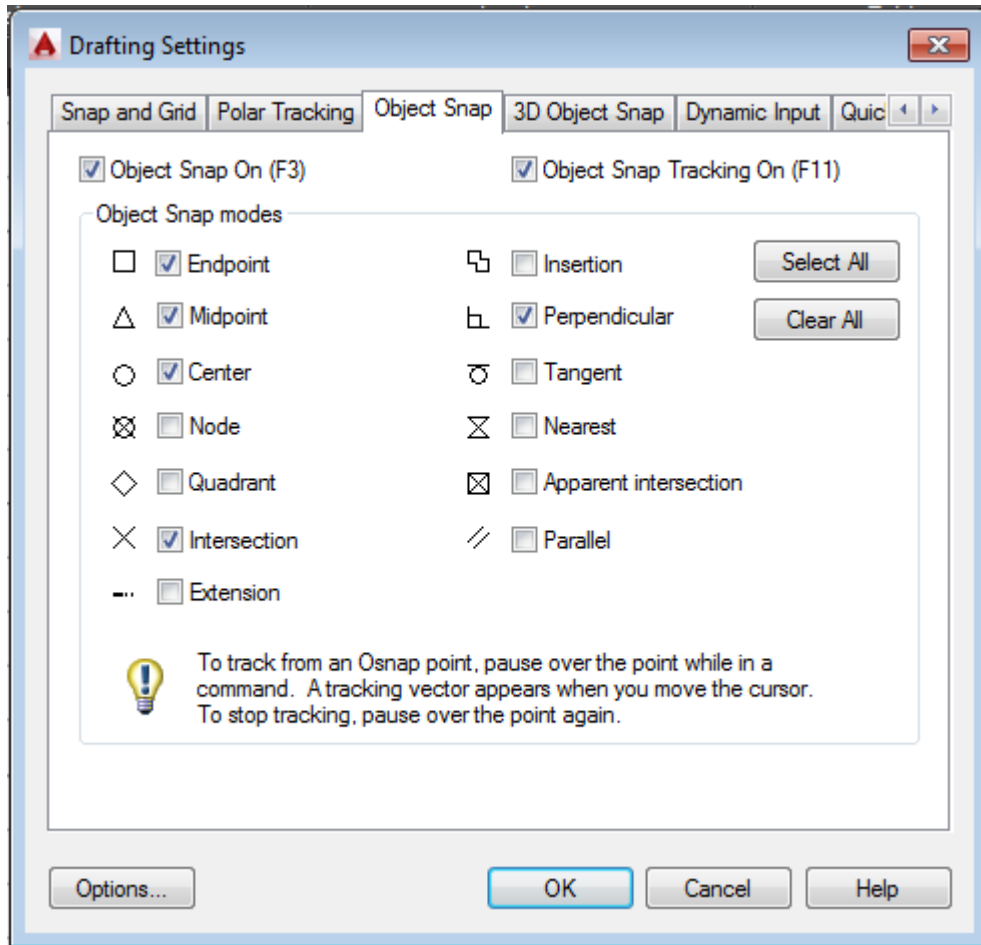
7. **Object Snap** is a powerful construction tool that allows accurate location of the cursor to points on objects such as endpoints, midpoints, centers, and intersections. **Check** the following *Object Snap modes*: *Endpoint, Midpoint, Center, Intersection* and *Perpendicular*.

8. Turn on the *Object Snap*.
   The *Drafting Settings* dialog box can also be used to change the settings of the snap and grid parameters. Select the *Snap and Grid* tabs in the dialog box to check the current settings of the snap and grid parameters.

9. **Left-click** on *OK* to close the dialog box.

10. For the left object in the drawing,
    a. Construct the rotated square using the *LINE* command. To locate the vertices of this square, position the cursor near the midpoint of each line. The *Snap to Midpoint* option of the **Object Snap** will locate the exact midpoint of the lines.
    b. Draw the lines from the corners of the big squares to the corners of the small square. The *Snap to Endpoint* and the *Snap Midpoint* options will locate exact positions of points.
    c. Draw the small square using the points found in Step (b).
    d. In the *Draw* tool bar, select the *Circle* command.
    e. In command prompt area, select *Circle by 3 Points* by typing **3p**.
       Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **3p [ENTER]**

      f.   Click on the midpoints of the sides of the small square as the points on the circle. (You might need to turn the *SNAP* option of by left-clicking on the *SNAP* button in the *Status Bar*.

11. For the right object, locating the parallel lines is tricky. Use concepts of geometric construction as described below to locate their positions.
      a.   Draw a circle of radius 1.0 centered at the lower-left corner of the square.
      b.   Draw a line from the upper-right corner of the square, tangent to the circle drawn in Step (a) above.
      c.   You may repeat the above procedure for the other line, or simply construct a parallel line.

12. Add dimensions to the drawings.
13. Save the drawing as **lab1-2**.