Learning AutoCAD® 2005

A SELF-DIRECTED APPROACH
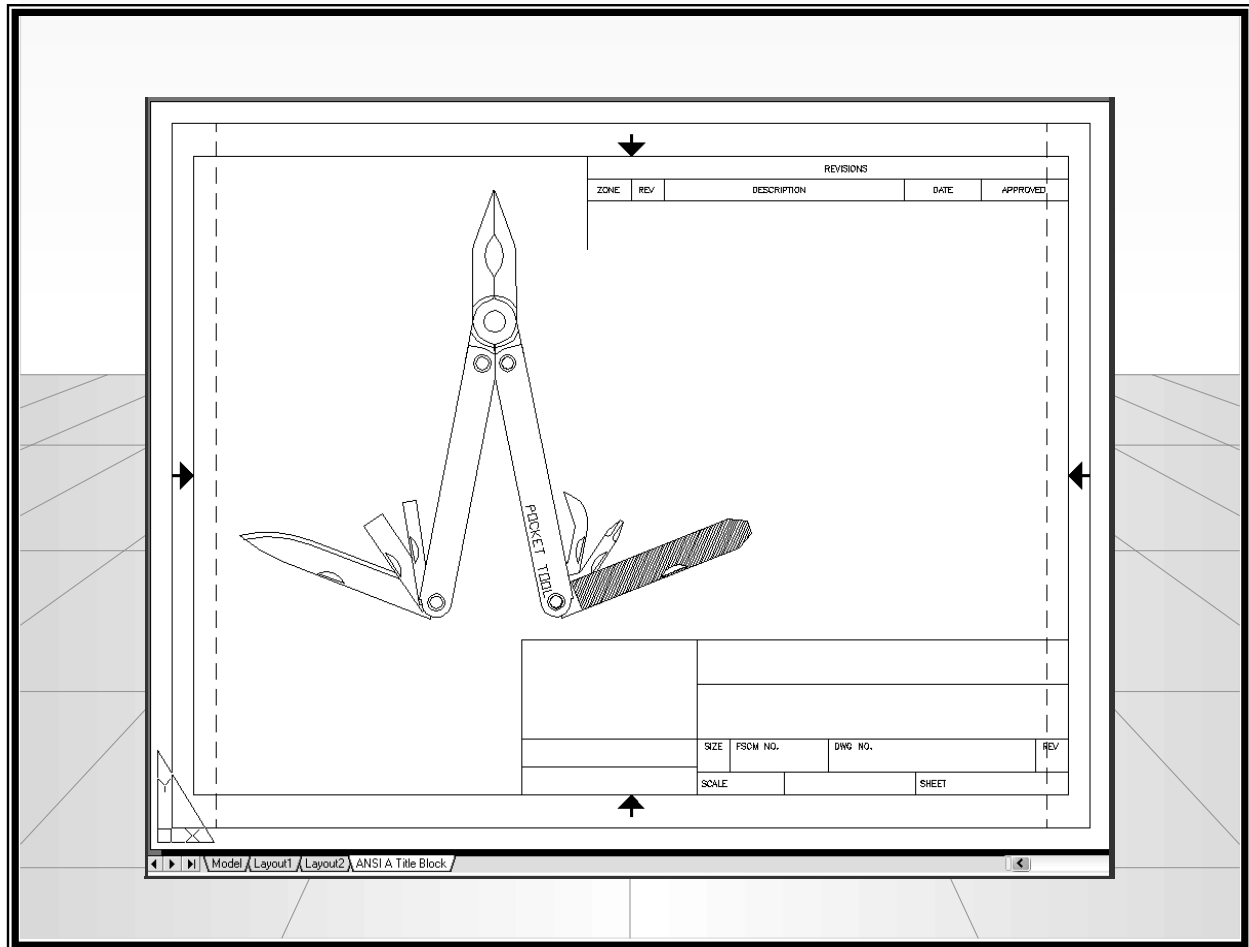
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Chapter 3 - Creating Entities

Objectives

1. Explore the Draw Toolbar
2. Draw circles, arcs and ellipses
3. Draw rectangles, polygons and points

What are "Entities?"

You already created a few entities in Chapter 2. A line is an entity. So are circles, arcs, polygons, rectangles (when created using the rectangle command) and so on. There will be more on this subject later in Chapter 4, since you'll need to be able to pick entities in order to edit them.

The Draw Toolbar

Start a new AutoCAD drawing using the acad template. Left-click on the **Zoom All** icon. Make sure the **Draw** toolbar is on (if you don't see it, pick it from the list that appears when you right-click on an existing toolbar). Here's what it looks like, together with the commands we'll be using in this chapter:
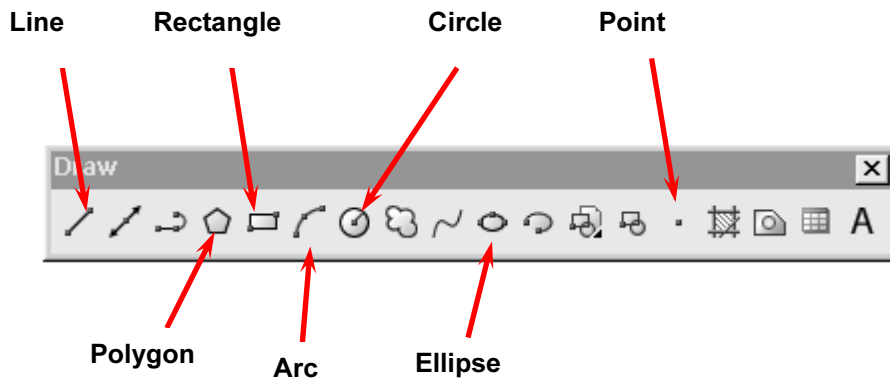

Drawing Circles

1. Select the **Circle** command. What does the command prompt ask for?

2. Notice you have many options, but for now left-click near the center of the screen to specify the center point of the circle.

3. Move the mouse around, What happens? Notice the size of the circle grows and shrinks depending on the position of the mouse on the screen.

4. At this point, you may either enter a number for the **radius** of the circle (or you can enter the **diameter** by typing "**d**" first), or simply left-click when the circle is looks about the right size.

5. On your own, create at least two more circles using the **3P** and **2P** options. What does **3P** and **2P** mean?

6. There is one more very useful option called **TTR**, or tangent-tangent radius. Let's try it. Draw two circles of 4 units in diameter, one on each side of the center of the drawing area as shown below:
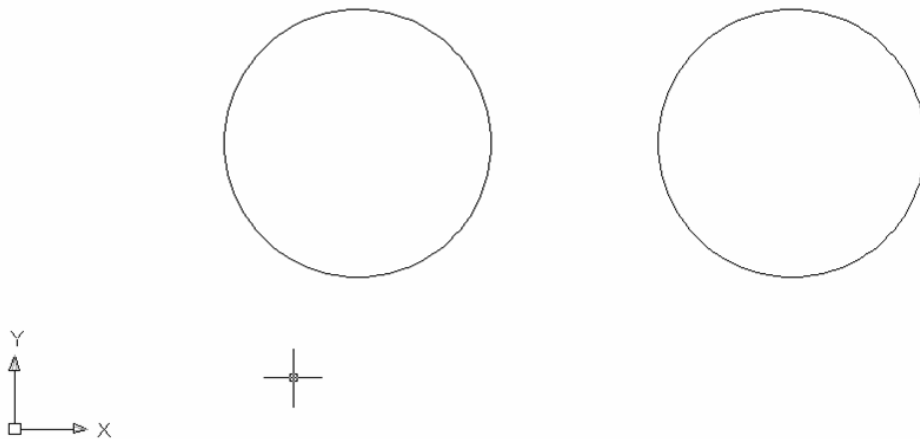
7. Next, start the **Circle** command again by selecting the **Circle** icon and type "**t**" at the command prompt. What does AutoCAD ask for?

8. Position the crosshairs near the upper-right area of the circle on the left. What is a deferred tangent?
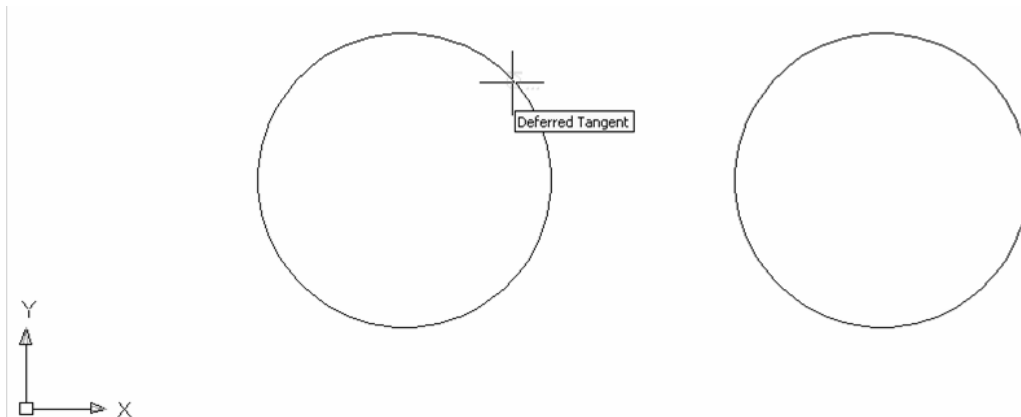
9. Using the crosshairs as a guide, left-click on the upper-right portion of the left circle, and the upper-left portion of the right circle.

10. When prompted, enter a value of "**2**" at the command prompt. Note that the last value that you entered is bracketed by <>. If the number in the brackets happens to be the value you want, just press enter to accept it.
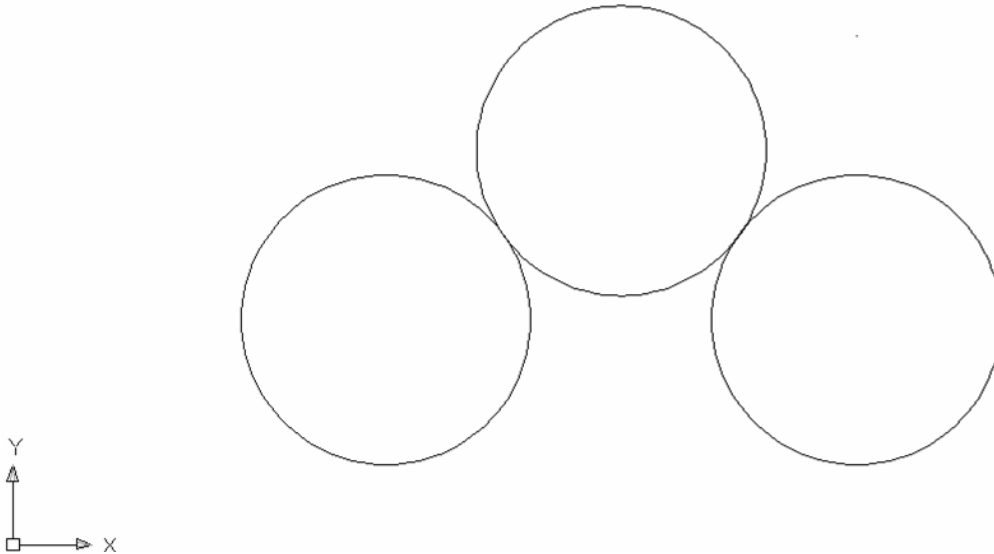
11. You should get a third circle that just touches (tangent) the other two (if you get an error that reads "circle does not exist" try it again with a larger value for the diameter).

12. On your own, place a fourth circle opposite of the last one and tangent to the first two, so your drawing looks something like this:
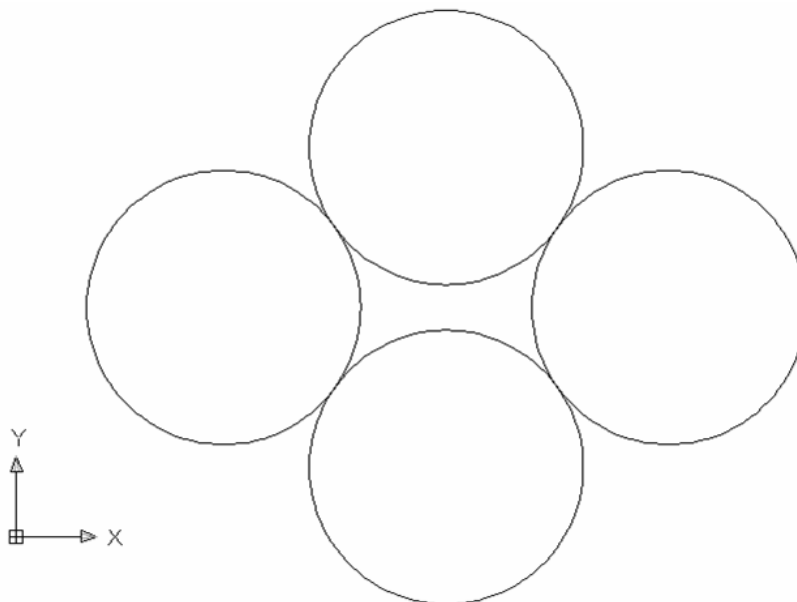
Creating Arcs

Arcs are just portions of circles. In most cases you'll find it more helpful to draw a circle and change it into an arc using AutoCAD's modify commands discussed later. However, we will try a few ways to create Arcs here:

1. Start with a clean drawing area either by starting a **New** drawing or deleting the circles done previously in this chapter. A quick way to do this is to type "**e**" at the command prompt and press **Enter**. This invokes the **Erase** command. Now type "**all**" at the command prompt and press the **Enter** key. Everything should be erased now.

2. Locate and left-click on the **Arc** icon in the **Draw** toolbar. What does AutoCAD ask you to do next?

3. Specify a start point near the center of the drawing area. Then click two more places on the drawing area and observe how the arc is created. This is a 3-point method, similar to the **3P** option in the **Circle** command.

4. Now, let's try the **Arc Center** method. The important thing to consider here is that AutoCAD creates arcs counter-clockwise. Left-click on the **Arc** icon and enter "**c**" at the command prompt followed by the **Enter** key.

5. Once more, position the crosshairs in the center of the screen and left-click.

6. For the start point of the arc, left-click to the right of the center point you just defined in step 4. For the end point move the crosshairs to the left of the center point and left-click. Note how the arc "sweeps" counterclockwise, over the top of the center point.

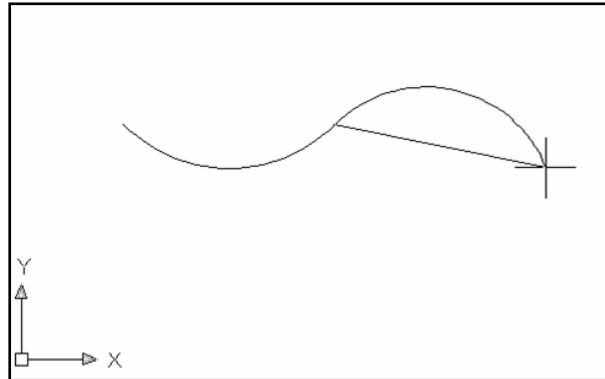
2-Start

3-End   **1- Center**
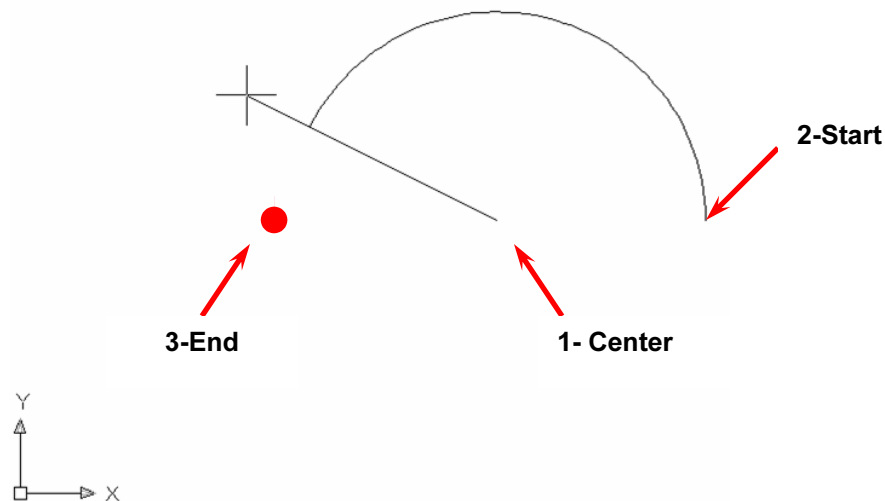
7. On your own, create another arc opposite of the first one using the center option, so your drawing area looks like this:
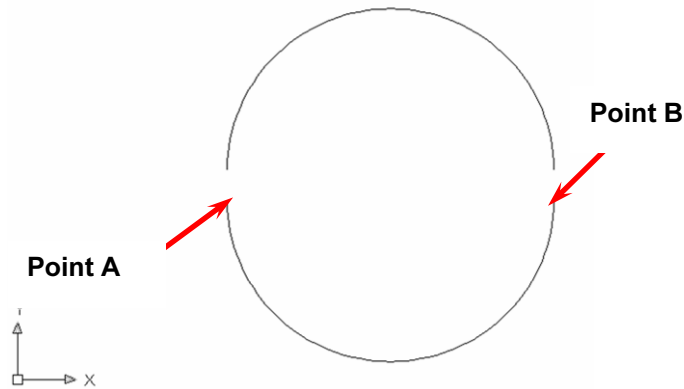
8. Which point did you have to specify first? Point A or B?

9. Last, try the **continue arc** method. Create a small, somewhat open arc in the lower left-hand corner of the screen using the three point (default) option.

10. After specifying the last point, press the **spacebar** on your computer keyboard twice. What happens?

11. The **Arc** command should start again and pick up where you left off, tangent to the last arc. Continue this method until you create a drawing similar to the one above and to the right.
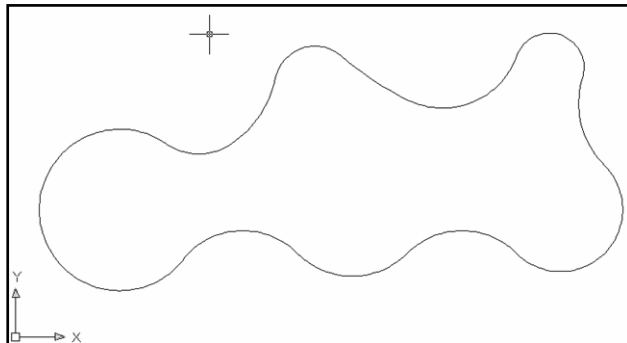

Tip: Pressing the **spacebar** is just like hitting the **Enter** key. AutoCAD will re-issue the last command. Pressing it again will specify the last point picked. This works with all drawing commands, not just Arc.
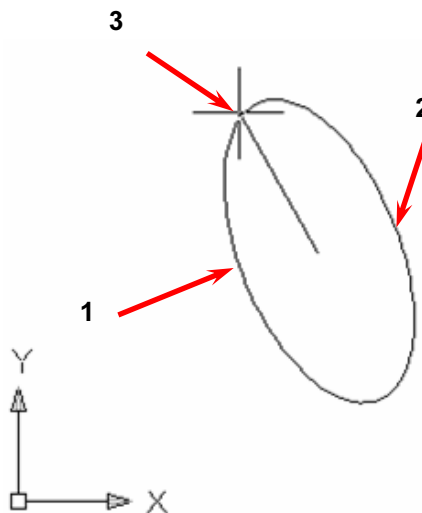
Creating Ellipses

An ellipse is an ob-round circle. There are three major features to an ellipse: the **center**, **minor axis** and **major axis**. Ellipses are useful for many things, as will be shown in the learning activities at the end of this chapter. To create an ellipse:
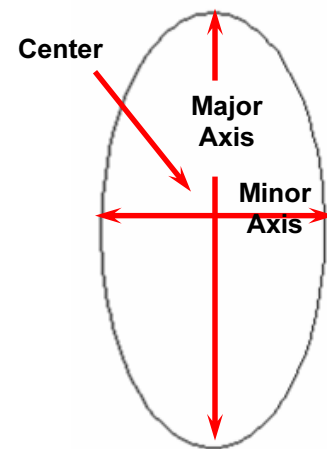
1. Locate and left-click on the **Ellipse** icon in the **Draw** toolbar. AutoCAD requests an **axis endpoint**. This is the default three-point method for creating an ellipse. Position the crosshairs on the left side of the drawing area and left-click.

2. Next, AutoCAD asks for the **other endpoint**. Go ahead and pick any point.

3. Finally, AutoCAD asks for the distance to the other axis. Pick yet another point and left-click. Note that the major and minor axes are up to you to decide, depending on which three points you select.

4. Try drawing another ellipse with the **ORTHO** option turned on in the **status bar**. What is the result?

5. Now try drawing ellipses using the other two options available. Entering "**a**" at the command prompt (after initiating the ellipse command) will draw an **elliptical arc**. A "**c**" changes the method to picking the **center** of the ellipse and the two axis endpoints.

Drawing Rectangles

The Rectangle command allows the user to quickly create a rectangle (or square) by specifying the opposite corners of the rectangle.

1. Locate and left-click on the **Rectangle** icon in the **Draw** toolbar. Pick two points on the screen for the opposing corners.
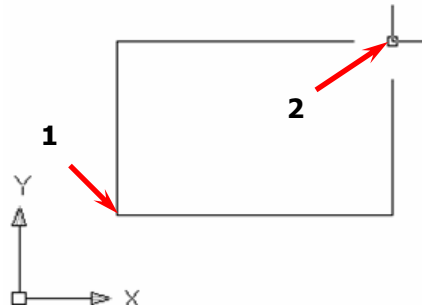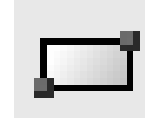
2. Try it again. Note before you pick the second point that there is another option, "**d**" for dimensions. Try it! This option lets you specify a rectangle of a specific size.

3. On your own, attempt some of the options presented to you when you start the **Rectangle** command. The most useful ones are **Chamfer**, **Fillet** and **Width**. A chamfer "breaks" the corners, fillets (pronounced "fill-its") rounds corners, and the width will thicken the lines of the rectangle. On your own try to duplicate the two rectangles shown below (one inside the other). Use the **Chamfer**, **Fillet** and **Width** options.
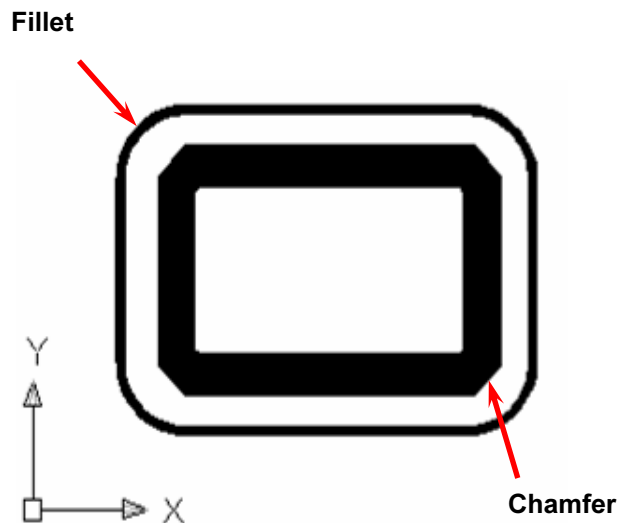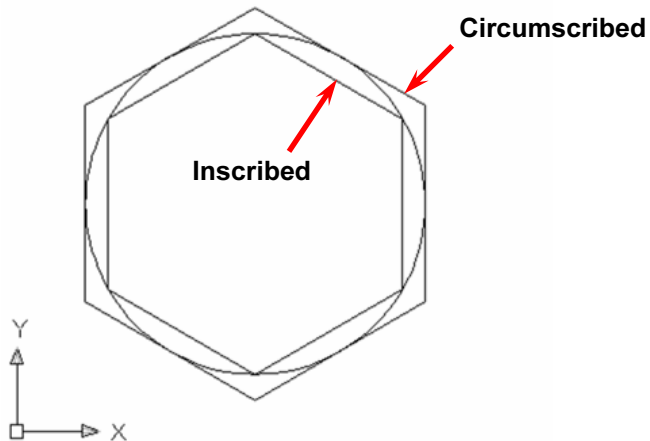
Creating Polygons

Polygons are many sided objects such as triangles, squares, pentagons, hexagons, etc. AutoCAD will either create them inscribed within a circle (circle through the points), or circumscribed about a circle (circle tangent to the sides). The method you chose depends on the situation at hand.

1. For this exercise, create a large **Circle** near the center of the drawing area.

2. Locate and left-click on the **Polygon** icon in the **Draw** toolbar. The first parameter is the number of sides. Enter "**6**."

3. Left-click near the center of the circle you drew in step one to specify the center of the polygon, then type "**i**" for and **inscribed polygon**.

4. Click on the edge of the circle. The polygon should exist more or less inside of the circle. Try creating another hexagon in the same place, but this time using "**c**" for a **circumscribed polygon**. Try to make the resulting hexagons and circle look like the ones below.


5. On your own, try the **Edge** (rather than center) option. Type "**e**" and press enter after entering the **Polygon** command. What is the difference between edge and center?

Creating Points

AutoCAD has the capability of creating points at locations specified by the user in the drawing area. This command is used by drafters of various CAD disciplines. The size and style of the points are specified by two variables that can be entered at the command prompt: **PDMODE** and **PDSIZE**.
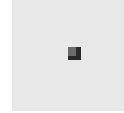
1. To set the point style, enter "**PDMODE**" at the command prompt. You may choose the numbers 0 through 4, but only 2, 3 and 4 are really visible.
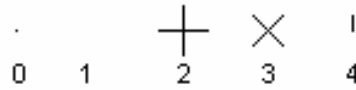
2. Additionally, you may specify any of the numbers instead found in the graphic below to specify a shape around the point.

3. "**PDSIZE**" is entered at the command prompt to specify the size of the point in relationship to the drawing area. The value you enter is equivalent to a percentage of the drawing area.
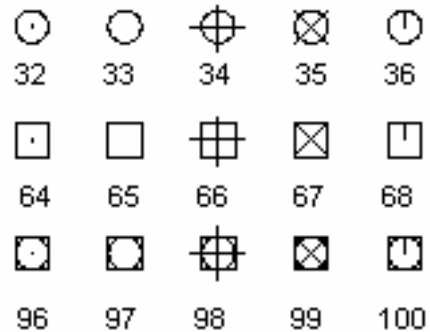
4. On your own, attempt to create the representation of the Big Dipper shown below. **PDMODE= 34**, **PDSIZE=.5**
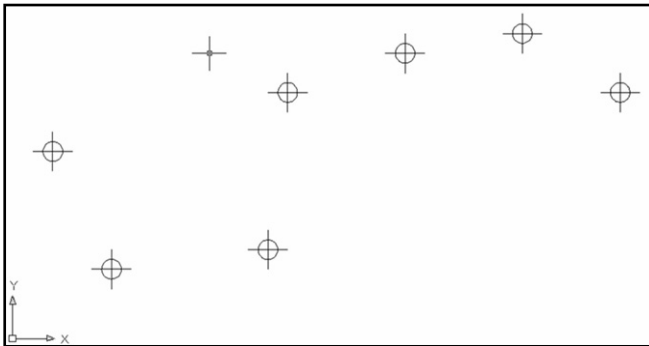
    ♦ Note: Changing the **PDMODE** and **PDSIZE** variables affect **all** of the points of the drawing globally, even the ones you've drawn previously!

5. Finally from the **Draw** menu, locate the **Point** cascading menu. Notice that you may use the **Point** command to divide-up another entity (**Divide**), and even use it to distribute objects along a path (**Measure**).
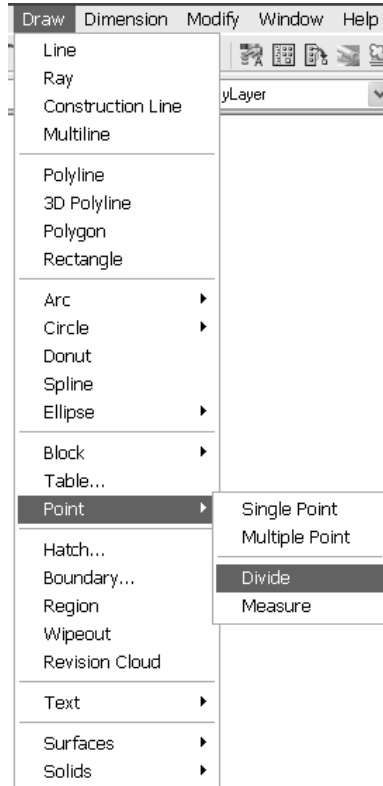
Chapter 3 Exercises

Based on your chosen drafting emphasis, pick from the following exercises.

Mechanical

Start new drawings for each of the sketches that follow. Use the commands and command options discussed in this chapter along with the **Line** command and **ORTHO** option. **Save** your drawings with the drawing name (e.g., lamp.dwg). Don't worry about things lining up perfectly; we'll cover that in a later chapter.
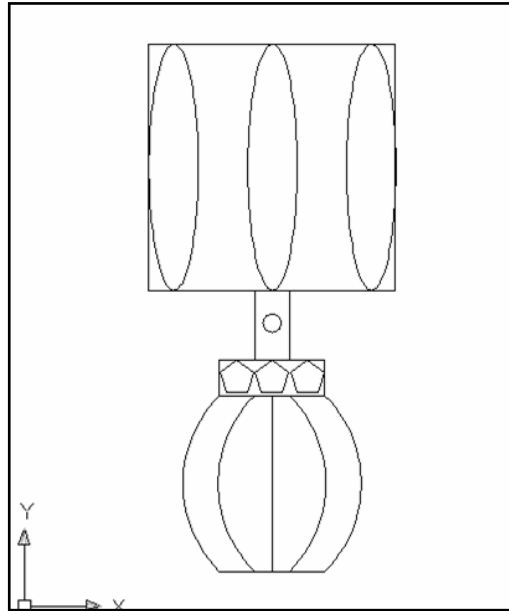
1. Schematic:
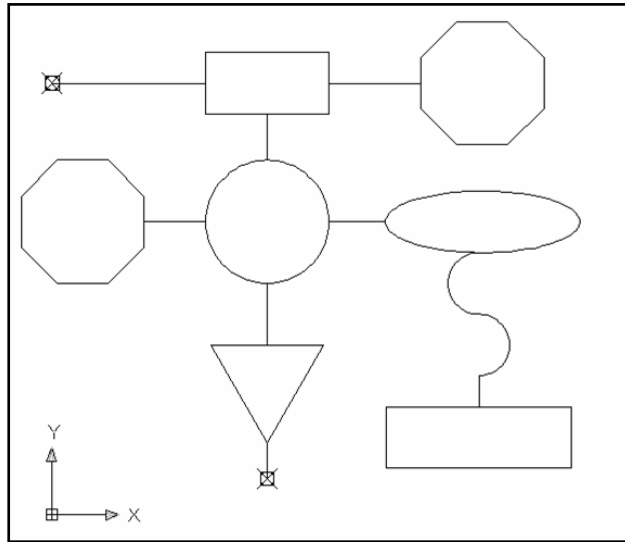
2. End Block (use a series of short lines to make the hidden & center lines):
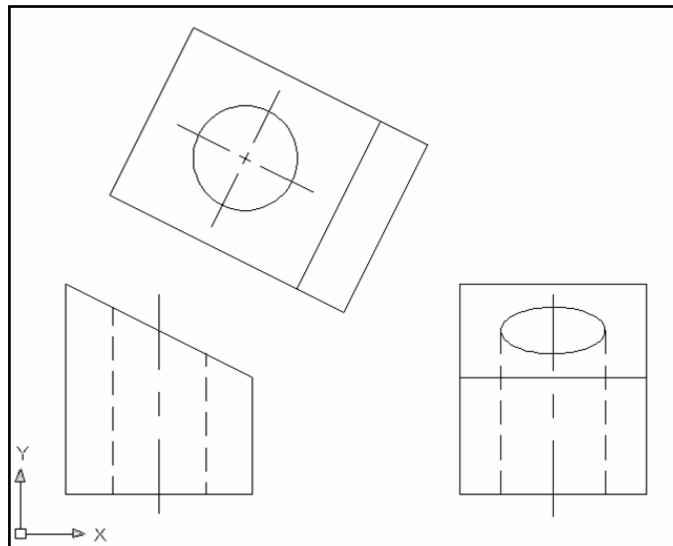
3. Lamp:

Architectural

Using the commands discussed in this chapter along with **Line** and **ORHTO** mode, sketch a landscape plan like the one shown in the drawing below. Save it as **landscape.dwg**.