Chapter

2

The Rectangular Problem

In this chapter, you will cover the following to World Class standards:

- **The tools for simple 2D Computer Aided Drafting (CAD)**
- **The Line command to draw a closed polygon**
- **The Line command to draw single line segments**
- **The Object Snaps (Osnaps) for accurate graphical selecting**
- **Checking your time to create a drawing**
- **The Offset command to create new and accurate lines**
- **The Trim command to divide lines into two, change line lengths and create arcs from circles.**
- **The Fillet command to trim lines and add arcs**
- **The Circle command to make circles with radius or diameters**
- **The Move command to change the location of an entity**
- **The Copy command to create a new entity at another location on the drawing**
- **The Rotate command to rotate a group of entities**
- **The Array command to create a rectangular pattern**
- **Achieving a high industry standard to be carried throughout your training**

The Ribbon

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The rectangular problem will familiarize you with the basic commands that an AutoCAD user will utilize everyday. If you are familiar with previous editions of AutoCAD, you are probably familiar with toolbars, which house groups of similar tools together. AutoCAD 2009 has a feature that groups the tools at the top of the screen and uses tabs to cycle through them. The six tabs are Home, Blocks & References, Annotate, Tools, View, and Output.
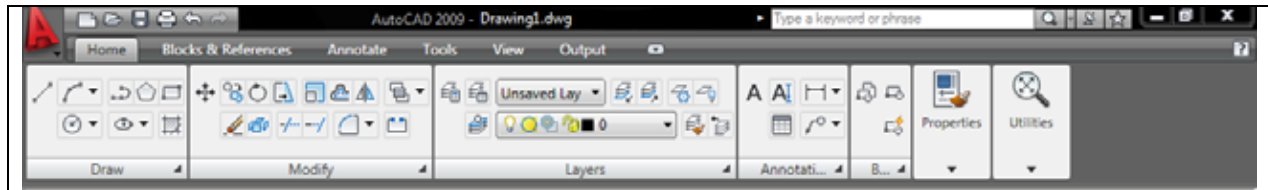
Figure 2.1 – The ribbon docked at the top of the screen

The Ribbon will be docked on the top of the screen as a default. Each ribbon tab contains panels that work like toolbars in that they further group similar tools. To change between different groups of panels, select the tab that corresponds to the panels and tools you need. To access the Menu Browser commands, such as File, Edit, View, Tools, etc., click the red A in the upper left hand corner of the screen and a drop-down menu will appear containing all of these selections.

Drawing your first lines

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Your first experience with the line command will be very simple. You will just be drawing a 6 by 4 rectangle. From the Ribbon, select the Line tool from the Draw panel under the Home tab. With your mouse, select a point at the lower left side of the graphic screen. Even while in the Line command, select orthographic mode (F8) on your keyboard to keep the lines perfectly horizontal or vertical when drawing. Move the line being drawn on the graphic screen to the right and type "6" and hit ENTER on the keyboard (Figure 2.2). To draw the vertical line, move the mouse upward, directing the line and type "4" and ENTER (Figure 2.3). Drawing the third line, move the mouse to the left, dragging the line and type "6" and ENTER (Figure 2.4). To close the rectangle, just type "c" and ENTER.
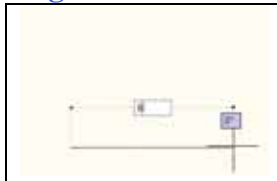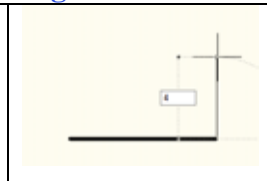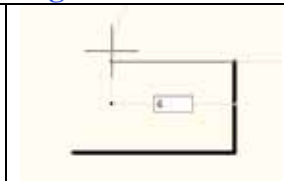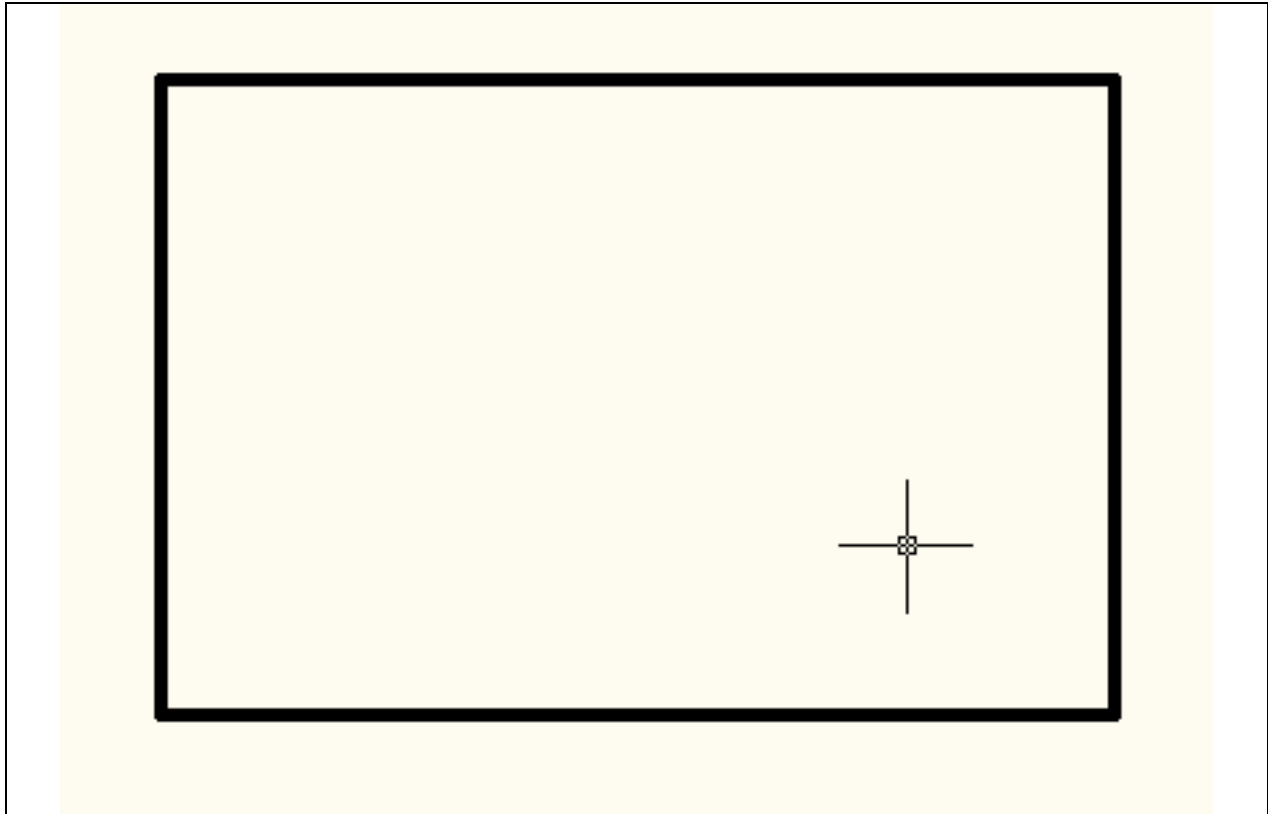
Figure 2.2	**Figure 2.3**	**Figure 2.4**	**Figure 2.5 – Command line**
			Command: _line Specify first point Specify next point or [Undo]: 6 Specify next point or [Undo]: 4 Specify next point or [Undo]: 6 Specify next point or [Undo]: c

**Figure 2.5 – The finished 6 x 4 rectangle.**

Type "Time" at the Command Line and read your total time. When opening a new file you will train so that you can draw four lines in less than twenty seconds.

* World Class CAD Challenge 02-01* – close this file by selecting File from the Menu Browser and then Close. Start a new file by selecting File and New. Draw the 6 x 4 rectangle in less than 20 seconds.

Drawing lines using Object Snaps

Setting Object Snaps allows you to select points precisely. To set Object Snaps, right click on the Object Snap button at the bottom of the Screen (shown at right) and select Settings from the menu. In the rectangular problem, you need to select the following Object Snap Modes: Endpoint for grabbing the ends of lines (recognized by a square when in the zone), Midpoint for grabbing the midpoint of a line (a triangle symbol), Center for grabbing the center of a circle or arc, Quadrant for snapping to the four points of a circle, north, east, south or west (a diamond symbol), Intersection to grab onto any intersection on the graphic screen (shown by an "x"), and Perpendicular for snapping to any object that is 90º from your first selection point (Figure 2.6).
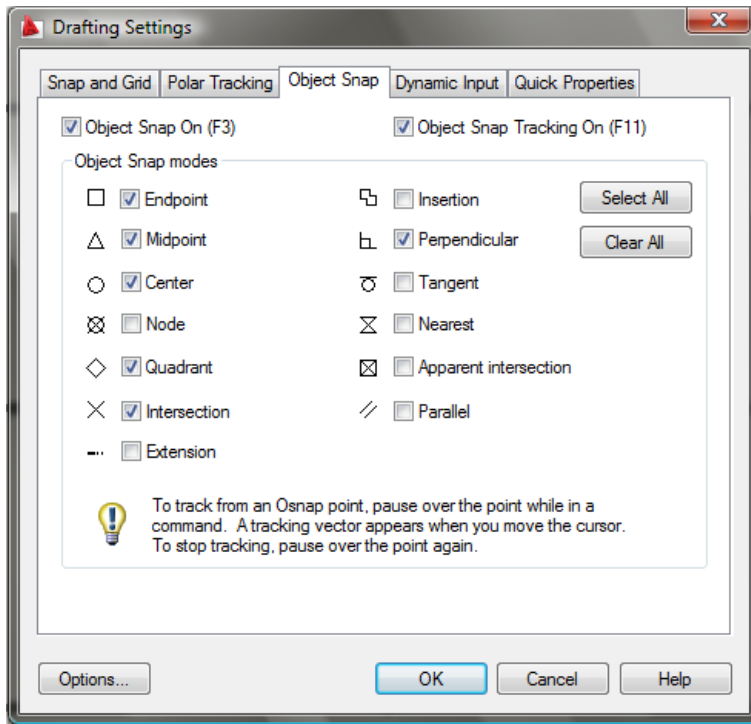
Figure 2.6 – The Object Snap settings tab on the Drafting Settings properties window
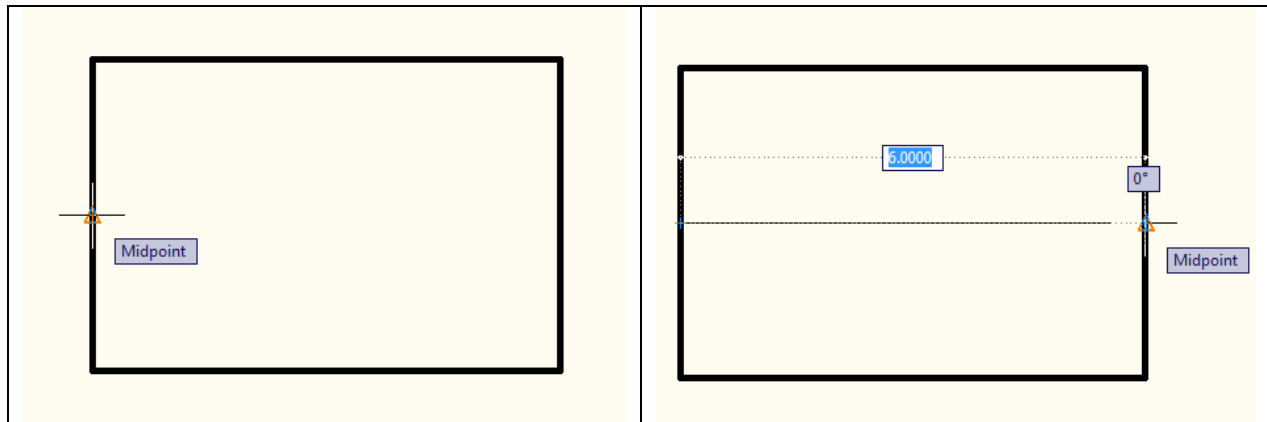
Figure 2.7 – Selecting the midpoint of a line   **Figure 2.8 – Selecting another midpoint**

The next two lines you will draw are the two centerlines dividing the rectangle. Now that the Object Snap Mode (referred from now on as OSNAP) is automatically set and turned on, select the line command a second time. Place the crosshairs, or aperture cursor, next to the middle of the left vertical line, and the midpoint OSNAP symbol should appear. Select it by clicking the left mouse button (Figure 2.7). Move your mouse to the center of the right vertical line and when the triangle OSNAP symbol appears, select the midpoint by clicking the left mouse button (Figure 2.8). This time, you will only draw one line segment, so press ENTER on the keyboard to exit the line command. Draw a second centerline, by repeating the sequence again.

Step 1. Move the aperture cursor next to the middle of the lower horizontal line and wait for the triangle midpoint OSNAP symbol to appear. Select the line with the left mouse button.

Step 2. Move the aperture cursor next to the middle of the upper horizontal line and wait for the triangle midpoint OSNAP symbol to appear. Select the line with the left mouse button.

Step 3. Press ENTER to exit the Line command.
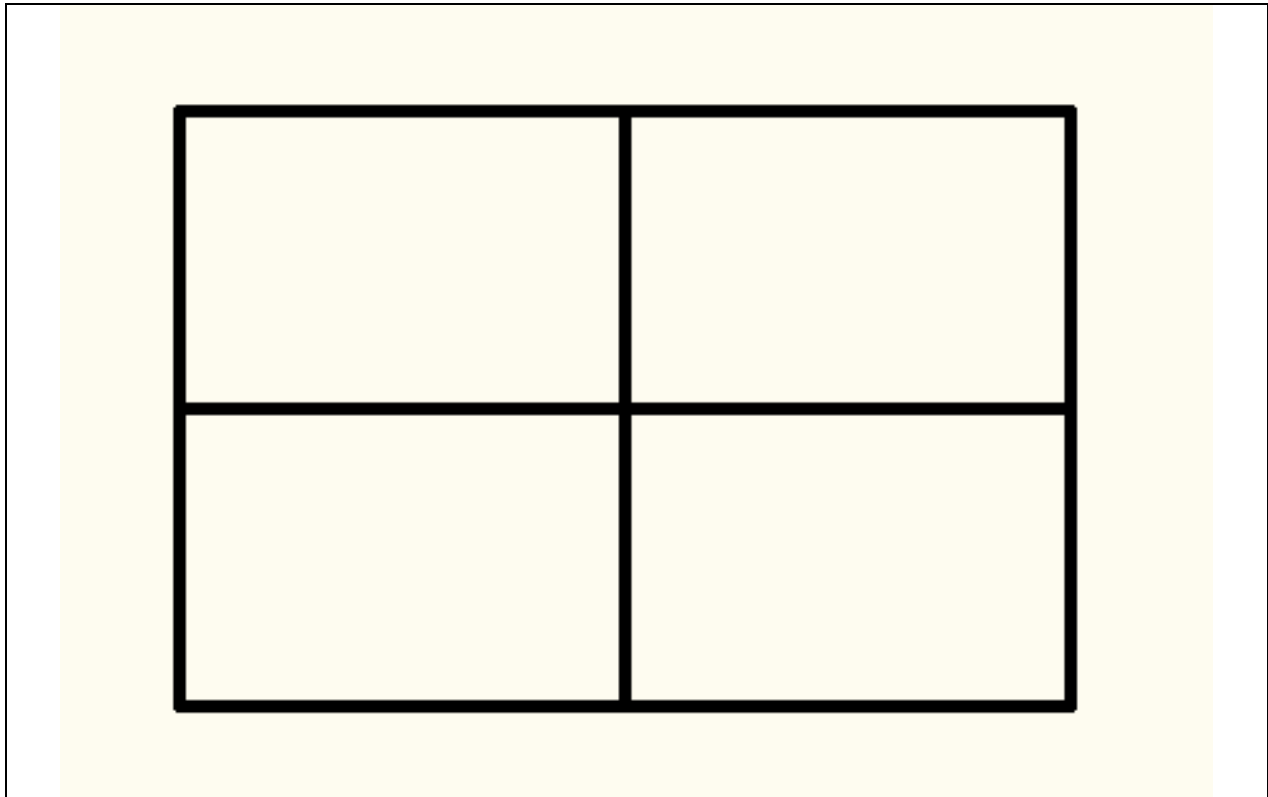
**Figure 2.9 – The 6 x 4 rectangle with two centerlines exactly in the middle**

* World Class CAD Challenge 02-02* - Close this drawing file. Create a New file and draw the six lines, four for the 6 x 4 rectangle box and two centerlines in less than 30 seconds. Continue this drill four times, each time completing the drawing under 30 seconds to maintain your World Class ranking.

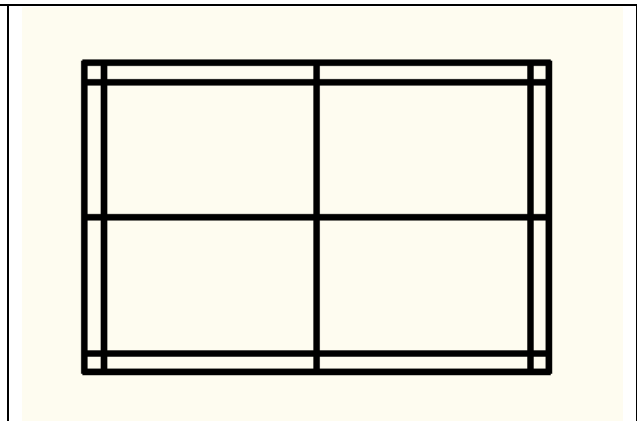## Using the Offset Command
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The Offset command is used to add lines to a drawing without selecting points to define the starting and end points of the line.   In your problem, you will choose the Offset tool from the Modify panel under the Home tab. Next, answer the offset distance query on the command line, which in your problem is "0.25" (Figure 2.14). Then select the line you wish to offset, in this case the top horizontal line. After selecting the line (Figure

2.10), pick the direction to offset the new line, which is below the top horizontal line. A new line will appear 0.25 below the top line. Continue to select the outside perimeter lines in a clockwise or counter-clockwise direction, first picking the existing line and then inside the perimeter to add the new 0.25 offset line (Figure 2.11). Hit ENTER to leave the Offset command after placing the four new lines.
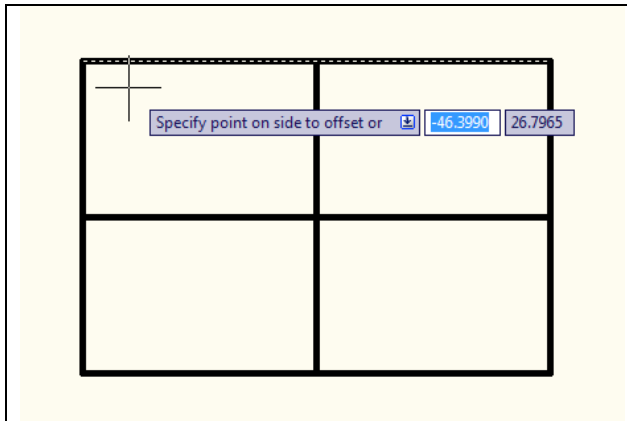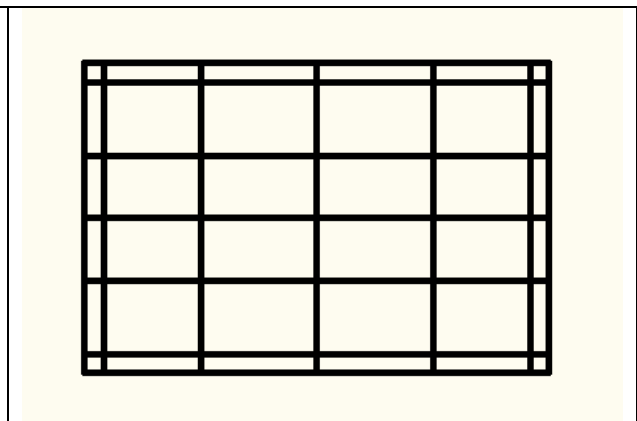

Figure 2.10 – Selecting the offset direction   **Figure 2.11 – Finishing the 0.25 offsets**
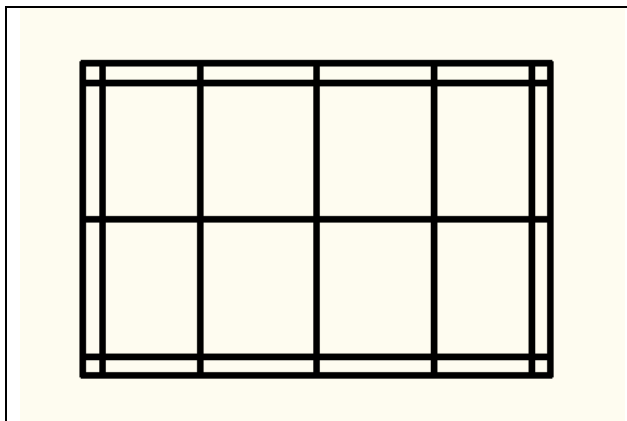
Figure 2.12 – Finishing the 1.5 offsets   **Figure 2.13 – Finishing the 0.8 offsets**

The next line to offset is the vertical centerline. Select the Offset tool from the Modify panel and type 1.5 and ENTER (Figure 2.15). Select the vertical centerline and then select to the right of the line to create a new line 1.5 to the right. Select to the left of the vertical centerline to create a new line 1.5 to the left. Hit ENTER to leave the Offset command after placing the two new lines (Figure 2.12).

The final line to offset is the horizontal centerline. Select the Offset tool from the Modify panel and type 0.8 and ENTER (Figure 2.16). Select the horizontal centerline and then select to the top of the line to create a new line 0.8 to the top. Select to the bottom of the horizontal centerline to create a new line 0.8 to the bottom. Hit ENTER to leave the Offset command after placing the two new lines (Figure 2.13).

Figure 2.14   Command: _offset
                      Specify offset distance or [Through] <1.0000> 0.25

Figure 2.15   Command: _offset
                      Specify offset distance or [Through] <1.0000> 1.5

Figure 2.16   Command: _offset
                      Specify offset distance or [Through] <1.0000> 0.8
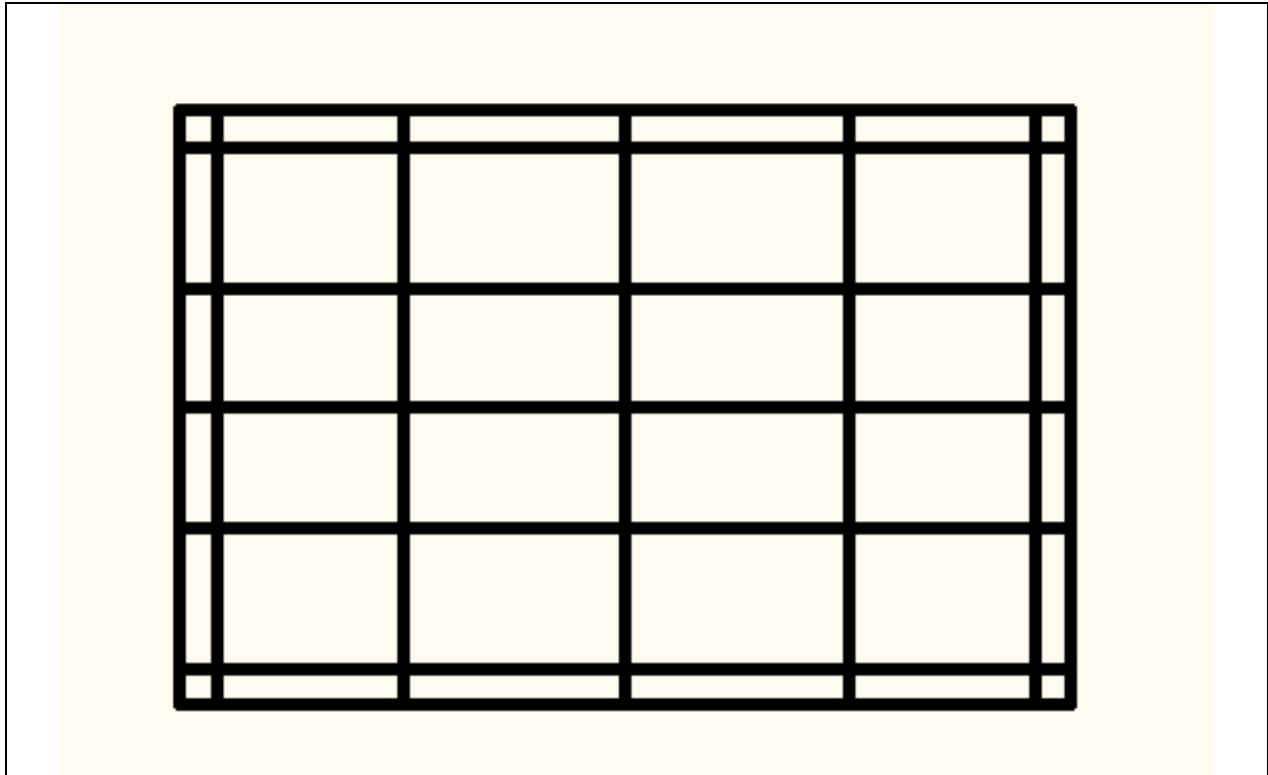
Figure 2.17 – The 4 x 6 rectangle with two centerlines and eight offset lines

* World Class CAD Challenge 02-03 * - Close this drawing file. Create a New file and draw the fourteen lines, four for the 6 x 4 rectangle box, two centerlines and eight offset lines in less than 70 seconds. Continue this drill four times, each time completing the drawing under 70 seconds to maintain your World Class ranking.

Using the Trim Command

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Your first use of the Trim command allows you to divide and remove unwanted portions of an existing line. Think of this command function just as you would a pair of scissors. Select the Trim command on the Modify toolbar and press ENTER to automatically select all lines on the graphical screen as cutting lines. Then proceed to pick each line segment that you would like to remove (Figure 2.18). Notice the line selected will be removed between two intersecting lines. Continue to select the two middle line segments on each side of the perimeter as shown in Figure 2.19. Finally, select four line segments as shown in Figure 2.20 to prepare for the Fillet command.
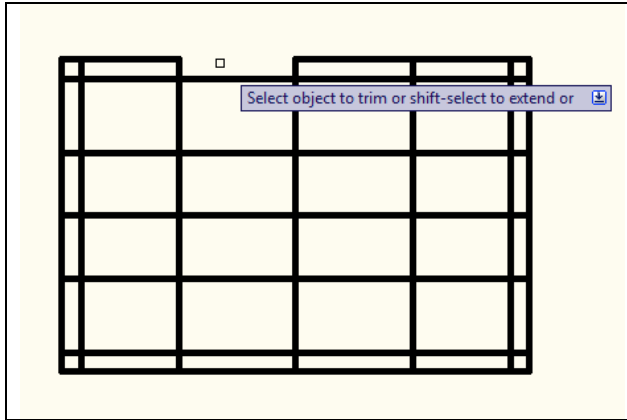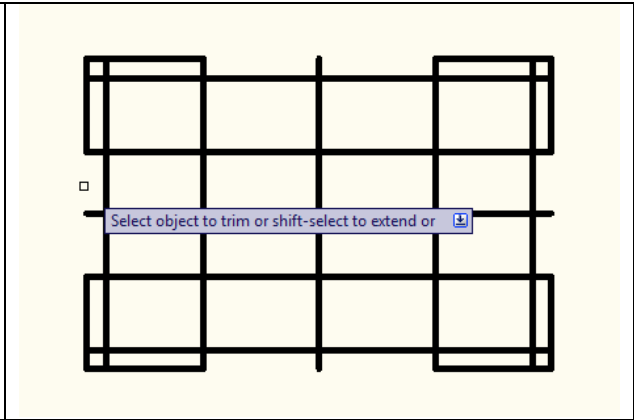
Figure 2.18 – Your first trimmed line
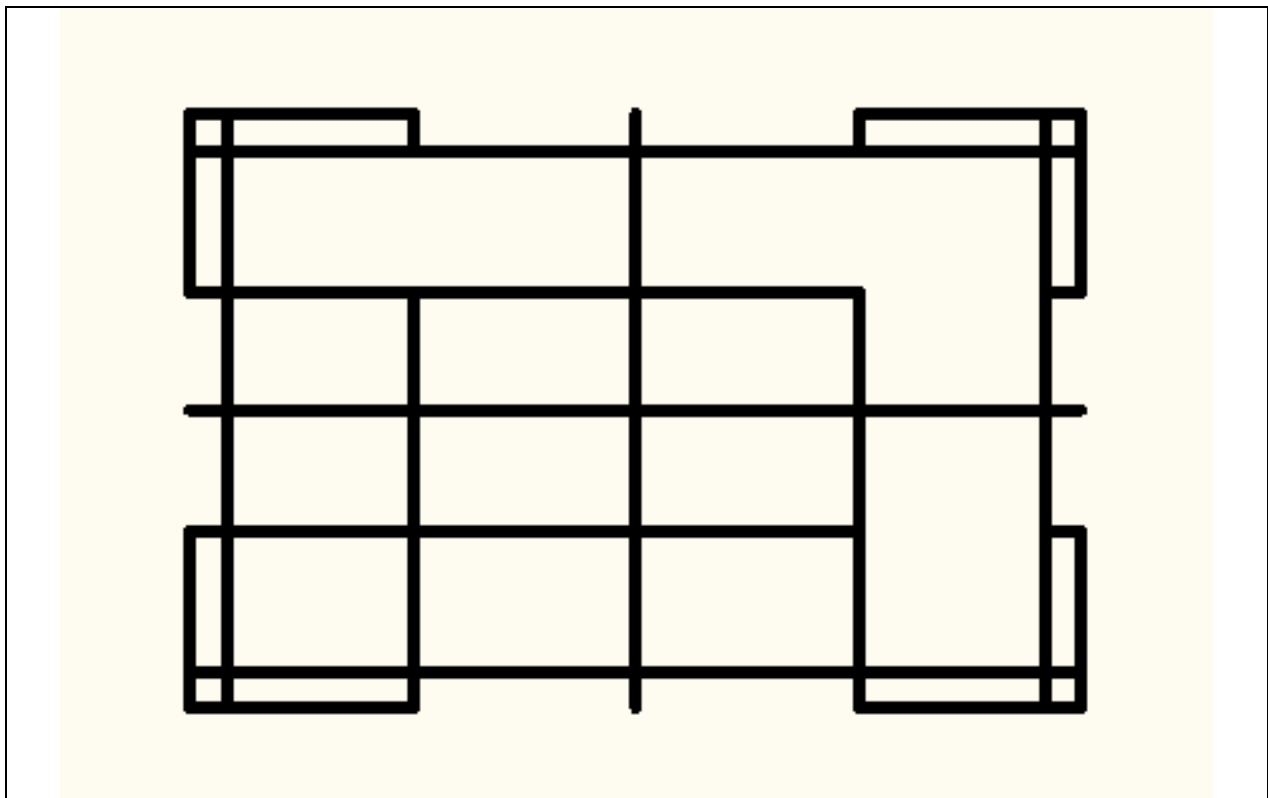
Figure 2.19 – The perimeter trimmed


Figure 2.20 – Finished drawing before filleting

Using the Fillet Command

The Fillet command allows you to trim and add sharp corners (radius of 0) or arcs to the intersection of two lines. First select the Fillet command from the Modify toolbar and then type "R" and ENTER. Type "0.06" and ENTER to set the system for a 1/16 radius arc. Also, type "M" and hit ENTER because you will be filleting multiple corners 1/16 corners. Select the two lines to which you want to add a 1/16 radius, as shown in Figure
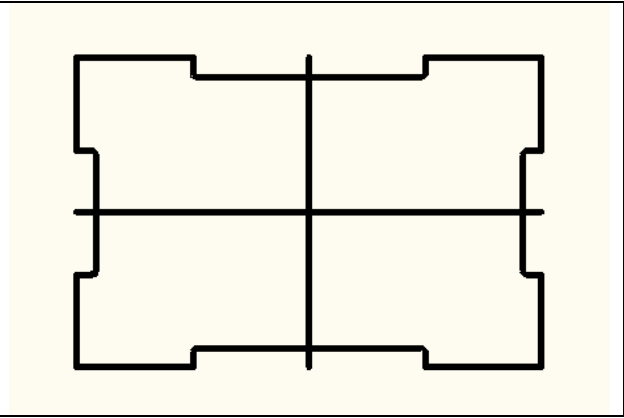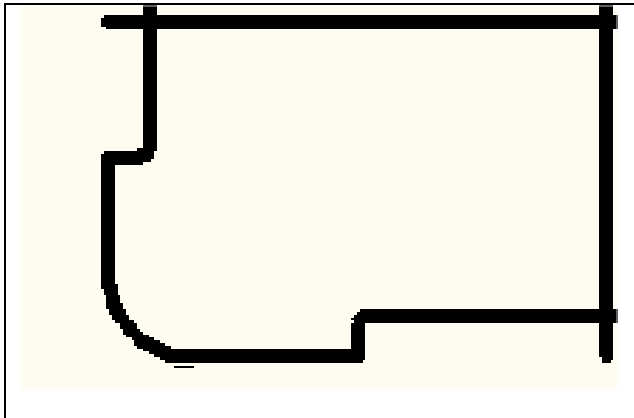
2.21. After selecting the two lines, the lines will shorten and a 1/16 radius arc will appear. Continue to repeat the process until your drawing looks like ours in Figure 2.22. Hit ENTER to exit the command.
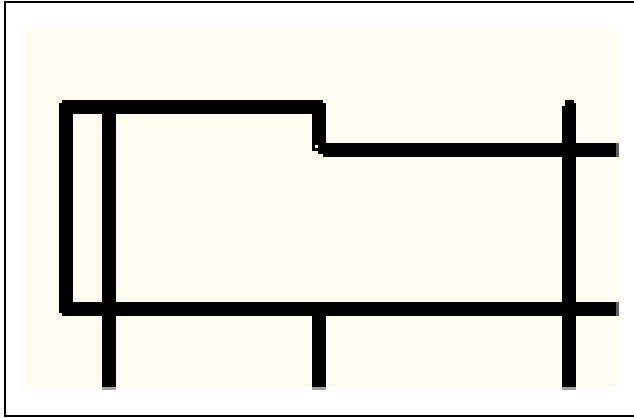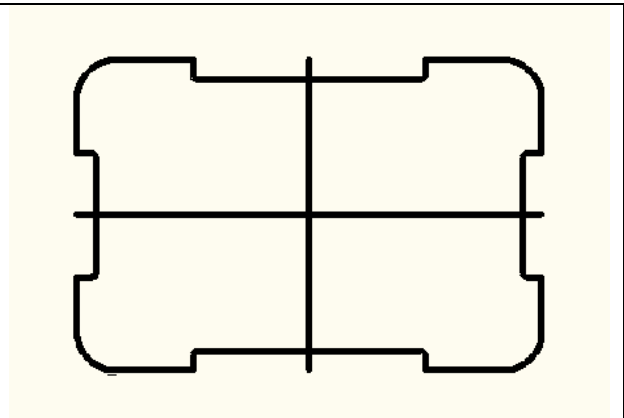

Figure 2.21 – A 1/16 arc   **Figure 2.22 – All 1/16 arcs added**


Figure 2.23 – A 1/2 arc   **Figure 2.24 – All 1/2 arcs added**

Now the drawing is really starting to take shape. Press ENTER to return to the Fillet command. Type "R" and hit ENTER. Type "0.5" and hit ENTER to set the system for a 1/2 radius arc. Again, type "M" and hit ENTER to fillet multiple corners without having to reset the command. Select the two lines that make the corners of the rectangle, as in Figure 2.23. After selecting the two lines, the lines will shorten and a 1/2 radius arc will appear. Continue to repeat the process until your drawing looks like Figure 2.24. Hit ENTER to exit the command.

You are totally finished with the perimeter of your first problem. You have used the Line, Offset, Trim and Fillet commands and have gained speed and efficiency through repetition. Many projects that you will do throughout your entire CAD experience will revolve around the creation of a perimeter that is functional and meets the needs of your project.
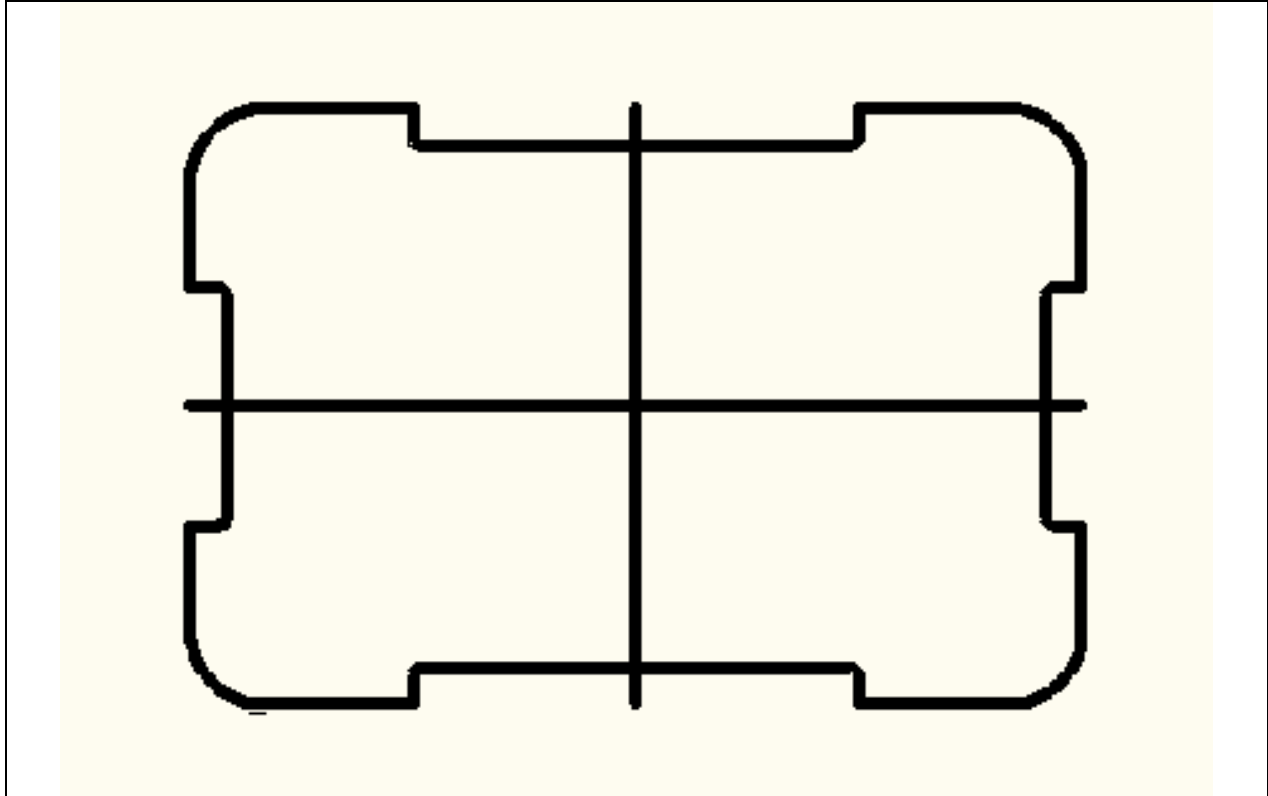
Figure 2.25 – The finished perimeter of your part with two centerlines

* World Class CAD Challenge 02-04* - Close this drawing file. Create a New file and draw the fourteen lines, four for the 6 x 4 rectangular box, two centerlines, and eight offset lines, trim the drawing and fillet the 12 corners in less than 2 minutes and 50 seconds. Continue this drill four times, each time completing the drawing under 2 minutes and 50 seconds to maintain your World Class ranking.

Drawing your first circles

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To add circles to your drawing, select the Circle command on the Draw panel under the Home tab. With your mouse, select the Midpoint of one of your centerlines. Notice that while your Osnaps are on, the Midpoint symbol (orange triangle) appears when the aperture cursor is near the middle of one of the centerlines. Select the Midpoint with the left mouse button (Figure 2.26). On the command line, a statement asking you to "specify radius of circle or [Diameter]" will appear. Any number in the greater/less than brackets (<>) is the default and you can select that number by pressing ENTER. For this problem, type 1 and ENTER. A 1 unit radius circle will appear on the drawing (Figure 2.27). Press ENTER to repeat the Circle command and select the Midpoint of the centerline again. When the prompt appears on the command line, type "d" for diameter and press ENTER. The command line will ask for you to "specify diameter of circle." Type .75 and press ENTER. A 0.75-unit diameter will appear inside the first circle (Figure 2.28).

Command: _circle Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
Specify radius of circle or [Diameter] <13.0000>: 1

Command: _circle Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
Specify radius of circle or [Diameter] <1.0000>: d
Specify diameter of circle <2.0000>: .75
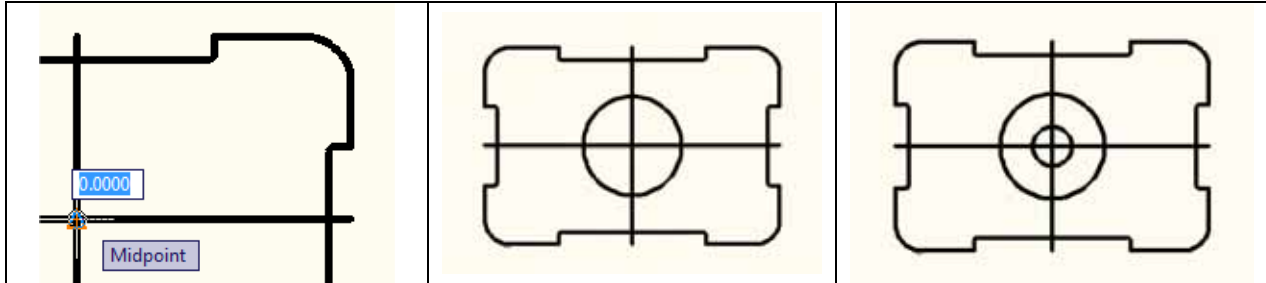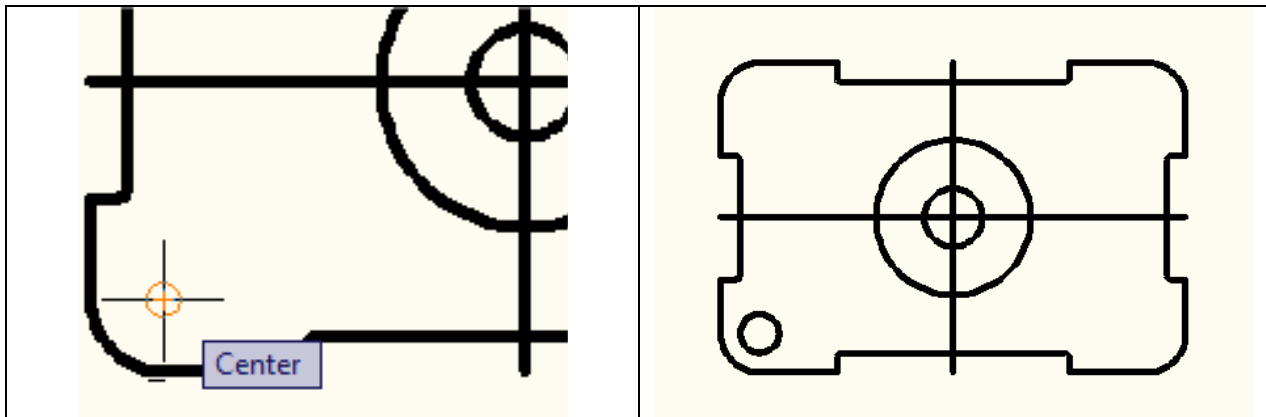

Figure 2.26   **Figure 2.27**   **Figure 2.28**

Press ENTER to repeat the Circle command. This time select the center of the arc in the
bottom left corner of the rectangle—the center Osnap will again help you find it easily.
The Center Osnap symbol (orange circle) will appear, at the center point of the arc
(Figure 2.29). Select the point with the left mouse button. Type "d" for diameter and
hit ENTER, and ".5" and ENTER to create a 0.5 circle.

Figure 2.29 – Selecting the center   **Figure 2.30 – The 0.5 circle**

Command: _circle Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: _cen of
Specify radius of circle or [Diameter] <0.4000>: d
Specify diameter of circle <0.8000>: .5

Using the Move and Copy Commands

The Move and Copy commands are located on the Modify panel under the Home tab.
These commands are separated into two parts, the first being the selection process and
the second being the function of the command itself, in this case Move. To move an
AutoCAD entity, such as a circle, select the Move command on the Modify panel. The
command line will prompt you to "Select Objects" (Figure 2.31). You can do this by

selecting the circle with the pick box that now has replaced the aperture cursor (Figure 2.32). The command line will respond with "1 found", so press ENTER to go the second part of the command. Now you need to "specify base point or displacement" on the graphical display. Pick any point away from any entities so that your Osnaps do not interfere with the movement. In Figure 2.33, it shows a base point to the lower right of the circles. Once you pick a base point with the left mouse button, drag the circle to the right to show the direction of movement as shown in Figure 2.33. The Ortho setting is still on, so the movement will be at a perfect 0 degrees to the right. Type "1.5" and ENTER at the command line for the second point of displacement and the circle will move 1.5 units to the right of its previous position (Figure 2.34)
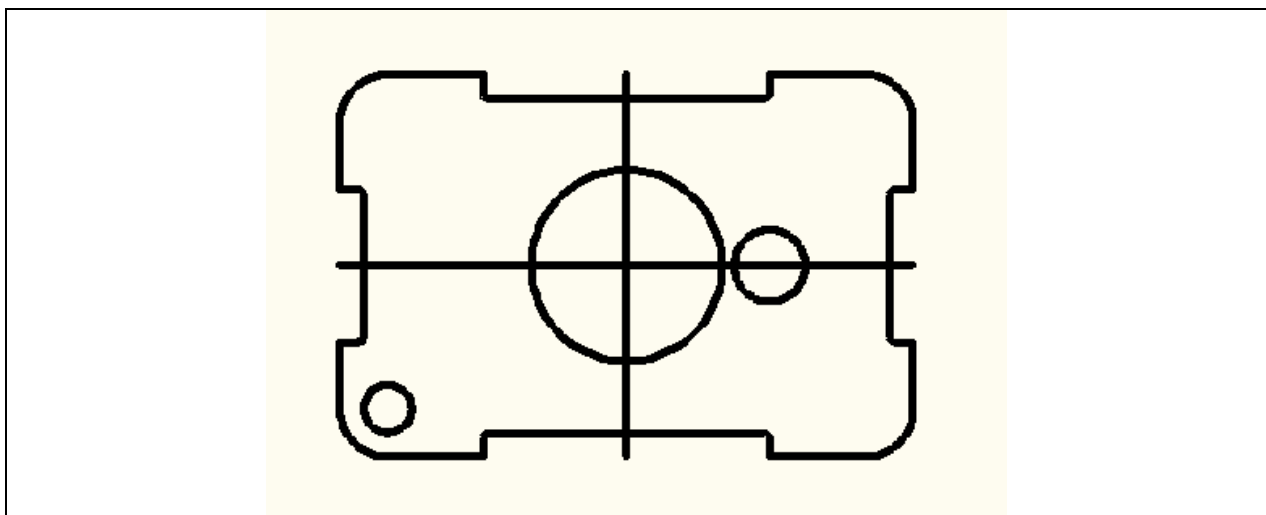
```
Command: _move
Select objects: 1 found
Select objects:
Specify base point or displacement:
Specify second point of displacement or <use first point as displacement> 1.5
```

Figure 2.31 – The command line sequence for your first Move command
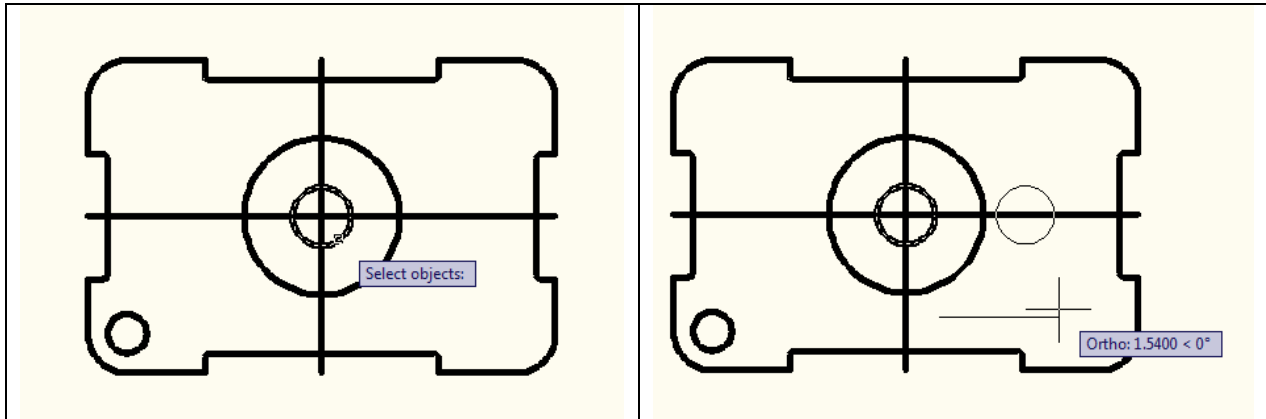
Figure 2.32 – Selecting the circle to move　　**Figure 2.33 – The direction of movement**

**Figure 2.34 – Your first movement of an entity completed**

Using the Copy Command

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To copy an AutoCAD entity, such as the same circle you just moved, select the Copy command on the Modify panel under the Home tab. The command line will prompt you to "Select Objects" (Figure 2.35). You can do this by selecting the circle with the pick box that now has replaced the aperture cursor (Figure 2.36). The command line will respond with "1 found." Press ENTER to go the second part of the command. Now you need to "specify base point or displacement" on the graphical display. Pick your base point away from any entities so that your Osnaps do not interfere with the movement. Figure 2.37 shows a base point selection to the lower right of the circles. After you pick a base point with the left mouse button, move the circle to the left as shown in Figure 2.37 to show direction of movement. The Ortho setting is still on, so the movement will be at a perfect 180 degrees to the left. Type "3" and ENTER at the command line for the second point of displacement and the circle will move 3 units to the left of its previous position (Figure 2.38)

```
Command: _copy
Select objects: 1 found
Select objects:
Specify base point or displacement:
Specify second point of displacement or <use first point as displacement> 3
```

Figure 2.35 – The command line sequence for your first Copy command
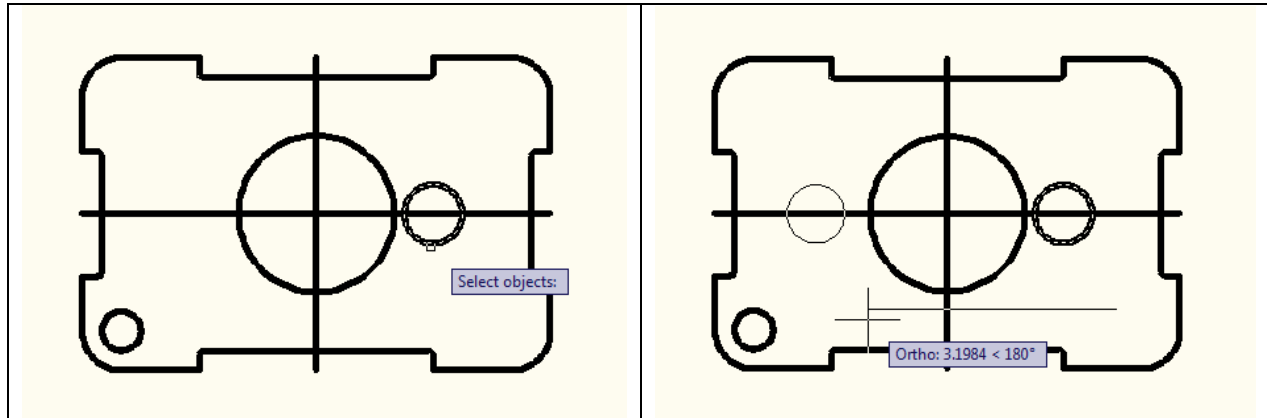
Figure 2.36 – Selecting the circle to copy　　　**Figure 2.37 - The direction of movement**

Notice that the procedures for the Move and Copy commands are identical, with the only difference being that for the first command, Move, the entity is given a new location in the drawing. The second command, Copy, left the initial entity in place and copied a second entity to the position specified by you. As in all modifying commands in AutoCAD, these are more powerful than the original Line or Circle commands when creating new entities in your drawings.
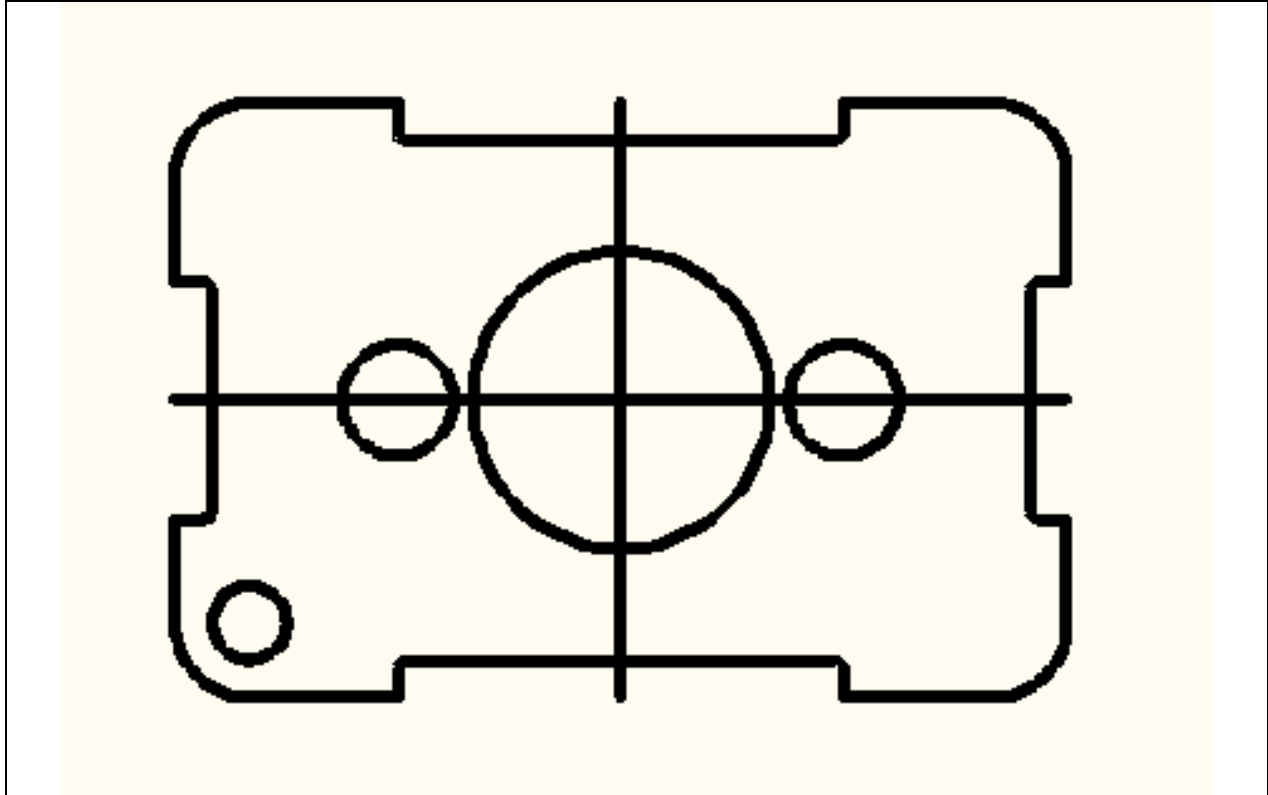
Figure 2.38 - The Copy command completed, resulting in two circles

* World Class CAD Challenge 02-05 * - Close this drawing file. Create a New file and draw the fourteen lines, four for the 6 x 4 rectangle, two centerlines and eight offset lines, trim the drawing, fillet the 12 corners, draw the 3 circles and move and copy the 0.75 circle in less than 3 minutes and 10 seconds. Continue this drill four times, each time completing the drawing under 3 minutes and 10 seconds to maintain your World Class ranking.

Using the Line Command with the Quadrant Osnap

You need to add two lines connecting the upper and lower quadrants of each 0.75 diameter circle. Select the Line command on the Draw panel. Each circle is broken into for quadrants: north, south, east, and west. Place the aperture cursor over the north quadrant of the left circle and the Quadrant Osnap (orange diamond) will appear (Figure 2.39). Select the quadrant with the left button of the mouse and then select the north quadrant of the right circle (Figure 2.40). Press ENTER to leave the command. You now have a single line segment connecting the north quadrants of both circles. Hit ENTER to repeat the Line command and draw a second line connecting the south quadrants of the two 0.75 diameter circles. Your drawing should look like Figure 2.41.
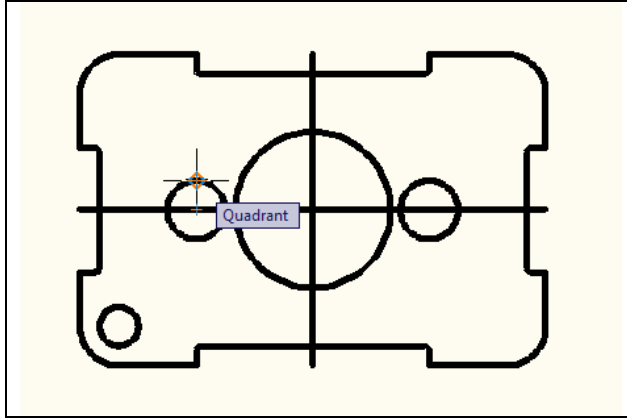
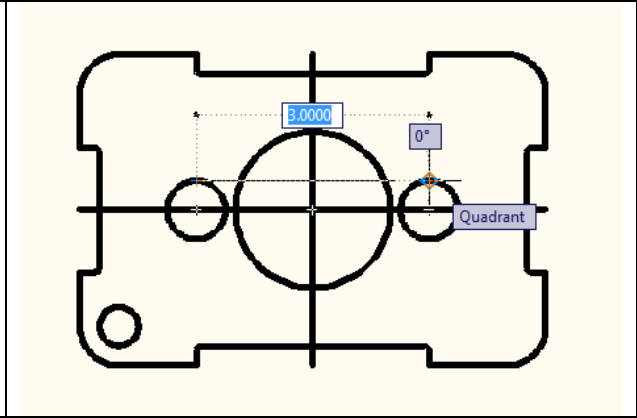**Figure 2.39 – Selecting the north quadrant**

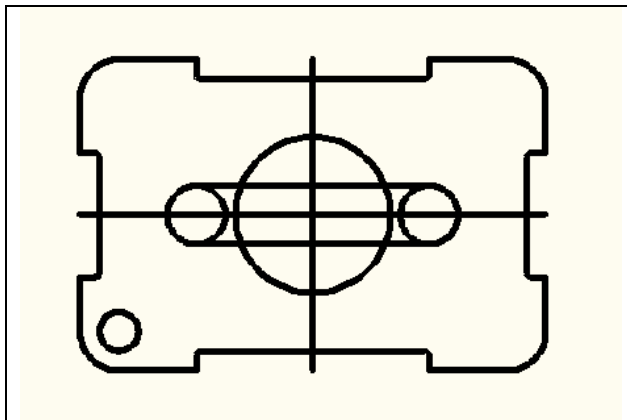**Figure 2.40 – Connecting the circles**

Figure 2.41 – The lower line drawn

The Quadrant Osnap is a very powerful tool that creates four reference points to a circle, compared to the single reference point obtained by the Center Osnap. Notice the complexity of your part is increasing and we have allowed the computer to compute the complicated geometry for you by referencing off Endpoints, Midpoints, Centers, and Quadrants.

Reinforcing the Trim Command

The second use of the Trim command allows you divide a line or remove unwanted portions of an existing line or circle. Again, think of using this command function just as you would a pair of scissors. Select the Trim command on the Modify panel under the Home tab and place a window around the interior detail of three circles and two lines. Select first on the upper left hand corner of the detail and then drag the selection window to the lower right hand corner of the detail (Figure 2.42). The selected entities will be dotted, as in Figure 2.43, and every edge of this detail will be a cutting line once you hit ENTER. Proceed to pick each line segment that you would like to remove (Figure 2.44). The 6 red dots in Figure 2.44 show the line or arc segments to remove. Finally, our last complex detail is complete as shown in Figure 2.45.
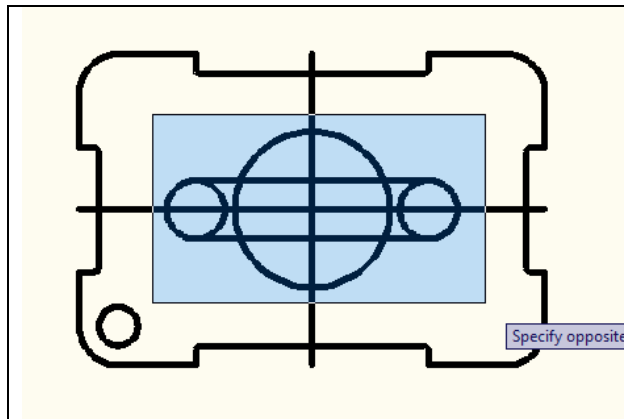
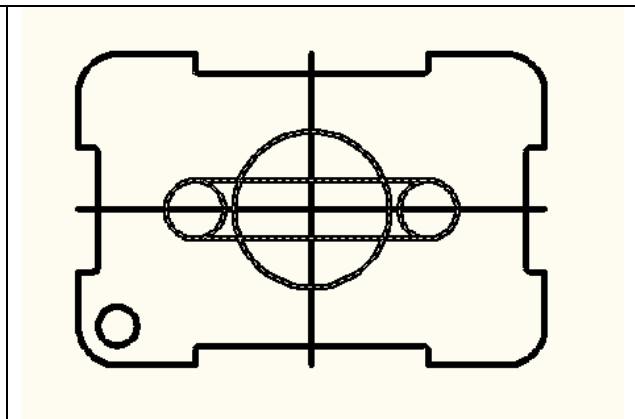**Figure 2.42 – Selecting by windowing**
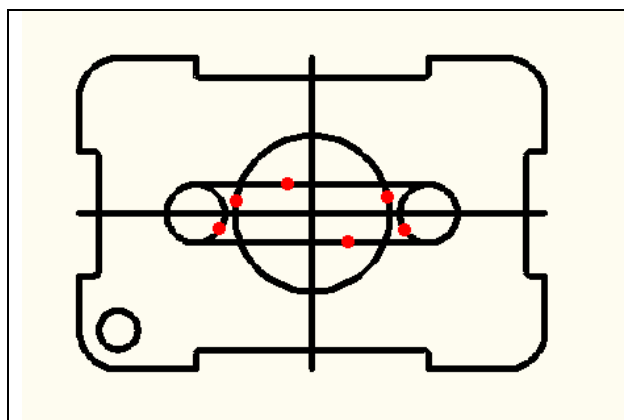

Figure 2.43 – The selected objects

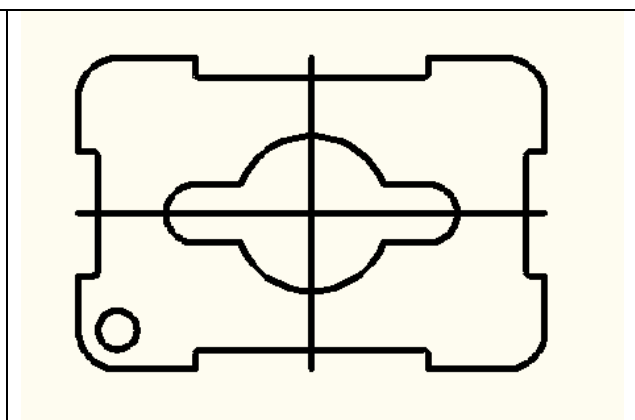**Figure 2.44 – 6 points to trim**

Figure 2.45 – The trimming completed

Using the Rotate Command

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The Rotate command will allow you to rotate entities around a base or center point. Select the Rotate command from the Modify panel under the Home tab. Place a window around the interior detail by selecting first on the lower left hand corner of the detail and then drag the selection window to the upper right hand corner of the detail (Figure 2.46). The selected entities will be dotted as in Figure 2.47. To continue with the command, press ENTER. Select the Midpoint of the centerline with the left mouse button (Figure 2.48). Then type "45" and ENTER for the degrees of counter clockwise rotation to finish this detail (Figure 2.49).
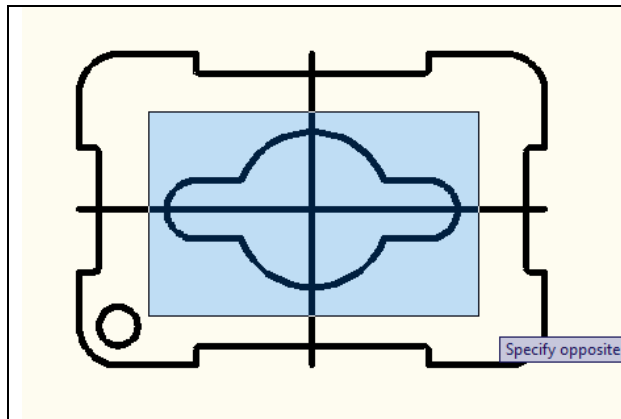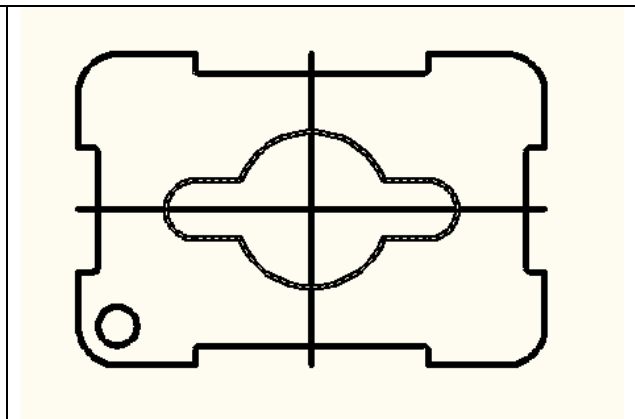
Figure 2.46 – Selecting by windowing   **Figure 2.47 – The selected detail**
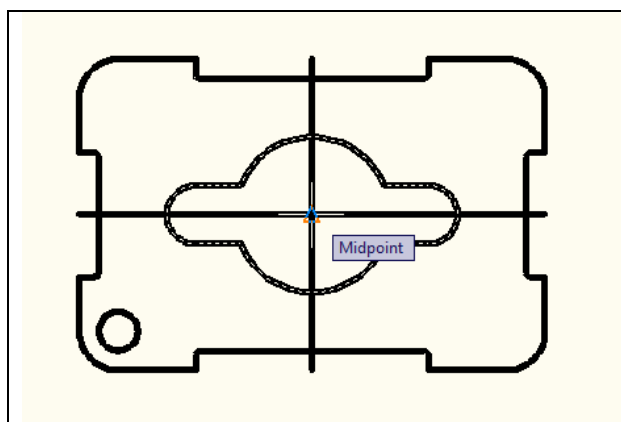

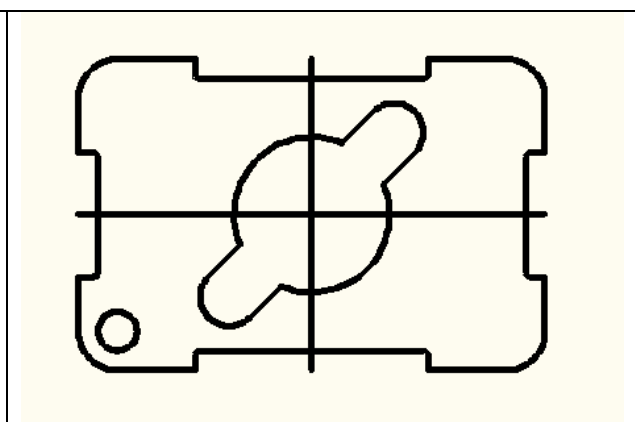**Figure 2.48 – Selecting the midpoint**   **Figure 2.49 – A 45 degree rotation**

* World Class CAD Challenge 02-06 * - Close this drawing file. Create a New file and draw the fourteen lines, four for the 6 x 4 rectangle, two centerlines and eight offset lines, trim the drawing, fillet the 12 corners, draw the 3 circles, move and copy the 0.75 circle, draw the lines connecting quadrants, trim the detail and rotate it 45 degrees in less than 3 minutes and 30 seconds. Continue this drill four times, each time completing the drawing under 3 minutes and 30 seconds to maintain your World Class ranking.

Using the Array Command to Create Rectangular Patterns

The Array command allows you to create a rectangular pattern of entities from selected objects, in this case the 0.5 unit circle. Select the Array command on the Modify panel under the Home tab. You may have to expand the Modify panel by clicking the small black triangle in its bottom right hand corner. The Array window will appear in the graphical screen area of your display. Make sure the Rectangular Array radial button is checked and type "2" in both the Rows and Columns textboxes. Type 3 in the Row Offset textbox and 5 in the Column Offset textbox (Figure 2.50). Select the "Selected Objects" button in the Array Window. The Array Window will disappear and you will be able to select objects on the drawing.
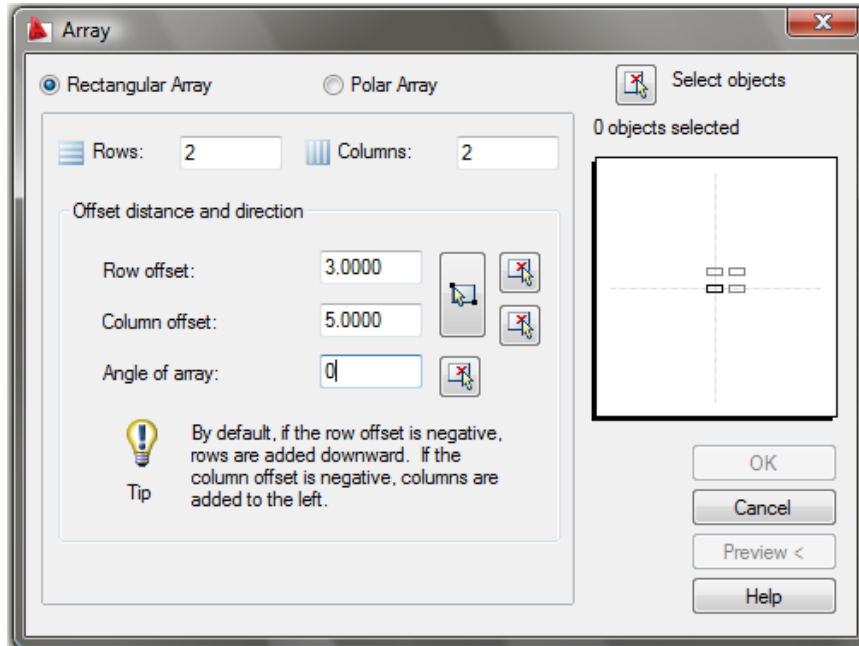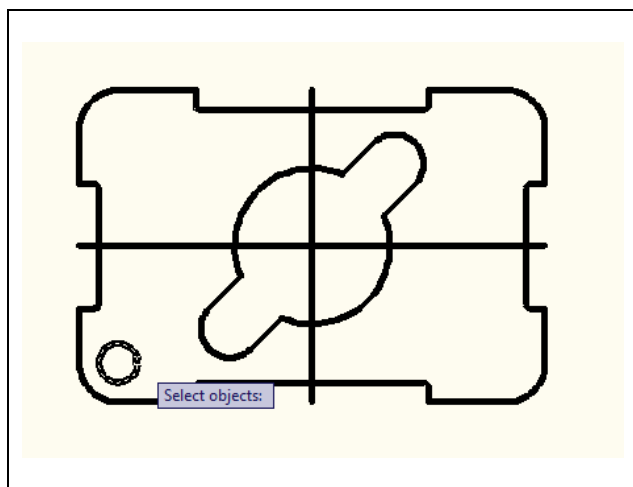
Figure 2.50 – The Array window

Place the pick box over the 0.5 circle and press the left mouse button to select the object (Figure 2.51). Then press ENTER to return to the Array window. You can see the rectangular pattern in the preview image. You can press the preview button to check the pattern on the part or press the OK button to finish the design of your first CAD drawing. The finished drawing, prior to dimensioning and notes, should look like Figure 2.52. The typical CAD student needs 10 to 12 hours of training and practice to achieve World Class standards.

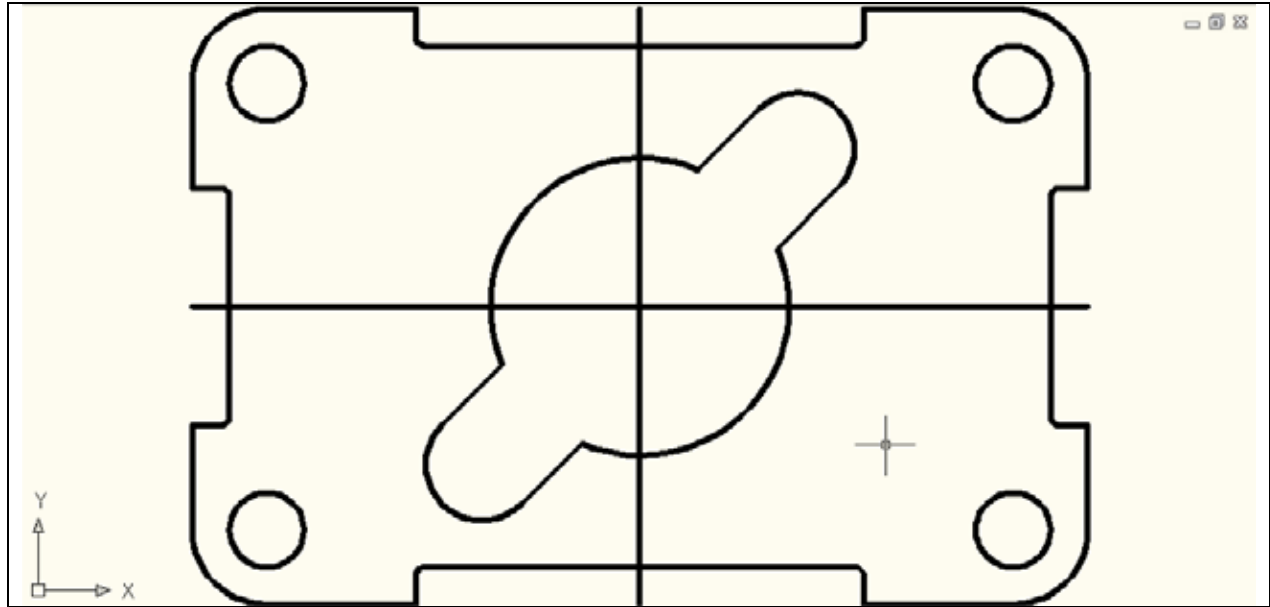**Figure 2.51 – Selecting the 0.5 circle**

Figure 2.52 – The finished 2D part prior to dimensioning

* World Class CAD Challenge 02-07 * - Close this drawing file. Create a New file and draw the fourteen lines, four for the 6 x 4 rectangle, two centerlines, and eight offset lines, trim the drawing and fillet the 12 corners. Draw the 3 circles, move and copy the 0.75 circle, draw the lines connecting quadrants, trim the detail, rotate it 45 degrees and array the 0.5 circle in less than 3 minutes and 50 seconds. Continue this drill four times, each time completing the drawing under 3 minutes and 50 seconds to maintain your World Class ranking.