C h a p t e r

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 Command Line and the Tray**
- **The Line command to draw a closed polygon**
- **The Line command to draw single line segments**
- **Object Snaps (Osnaps) for accurate graphical selecting**
- **Checking your drawing time**
- **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 a radius or diameter**
- **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**

The Ribbon

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The rectangular problem will familiarize you with the basic commands that an AutoCAD user will utilize every day. If you are familiar with previous editions of AutoCAD, you are probably familiar with toolbars, which house groups of similar tools together. AutoCAD 2009 introduced a feature which groups the tools at the top of the screen and uses tabs to cycle through them. AutoCAD 2012 continues to utilize the ribbon feature, but has an expanded amount of default tabs. They are Home, Insert, Annotate, Parametric, View, Manage, Output, Plug-ins, Online, and Express Tools.
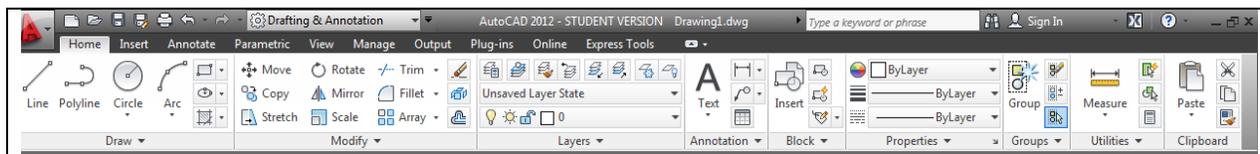
Figure 2.1 – The ribbon docked at the top of the screen

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 standard file commands such as open, save, save as, 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 options.

The Command Line and the Tray

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You will see a text field below the drawing area. This is the command line. All of the actions that you do while drawing will be recorded here. You can also enter text here to execute a command or answer a command prompt, as happens very often during drafting. You can hit Enter or the spacebar to execute a command at the command line.
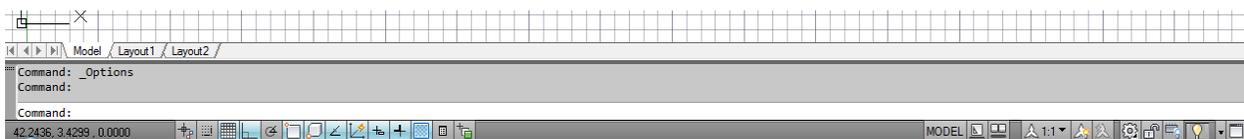
Figure 2.2 – The command line and the tray

The tray is located below the command line (Figure 2.2). It contains a number of options that you can toggle on or off by selecting the appropriate icon. For example, you will notice that the drawing area by default may have a grid displayed. If you wish to turn the grid off, just select the Grid Display icon. At any time you can hold your cursor over a tool or icon for a moment and the name of the tool will appear. If you continue to hold your cursor over some tools a more specific explanation will appear to help you understand what the tool does.

If you have Dynamic Input toggled on, the commands you enter with the keyboard will appear under the cursor rather than at the command line. However, once you execute the command by hitting Enter it will appear at the command line as if you typed it there. You can toggle

dynamic input on or off by select its icon on the tray. The pictures in all of these lessons will display the use of dynamic input.

Drawing your first lines

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. While in the Line command, you can toggle the orthographic mode on by hitting F8 on your keyboard to keep the lines perfectly horizontal or vertical when drawing. Move your cursor to the right, type "6", and hit ENTER on the keyboard (Figure 2.3). To draw the vertical line, move the cursor upward, directing the line and type "4" and ENTER (Figure 2.4). To draw the third line, move the mouse to the left, dragging the line and type "6" and ENTER (Figure 2.5). To close the rectangle, just type "c" and ENTER (Figure 2.7). Figure 2.6 displays the command line as it would appear throughout these actions.
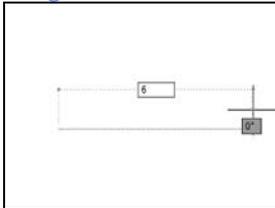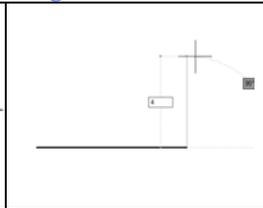
Figure 2.3	Figure 2.4	Figure 2.5	**Figure 2.6 – Command line**
			Command: _line Specify first point Specify next point or [Undo]: 6 Specify next point or [Undo]: 4 Specify next point or [Close/Undo]: 6 Specify next point or [Close/Undo]: c
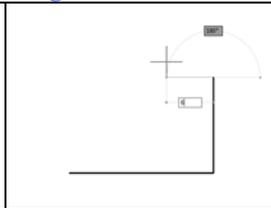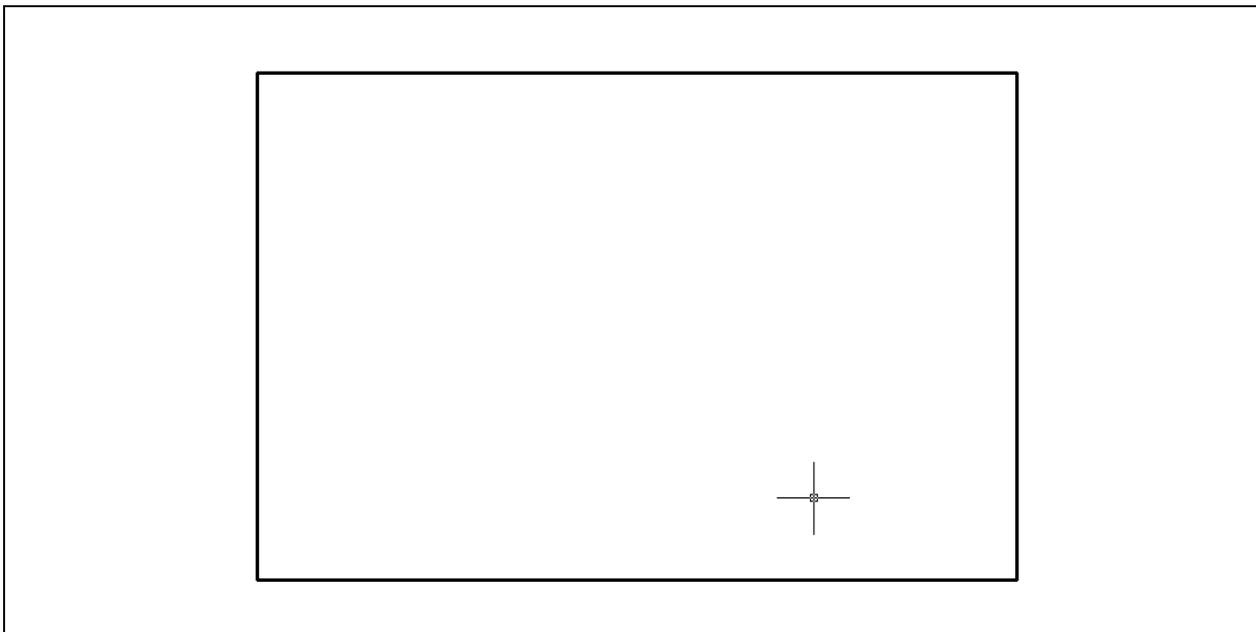
Figure 2.7 – The finished 6 x 4 rectangle.

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
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Setting Object Snaps allows you to select precise points that relate to specific points of other entities. To set Object Snaps, right click on the Object Snap icon on the tray 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 2.8).
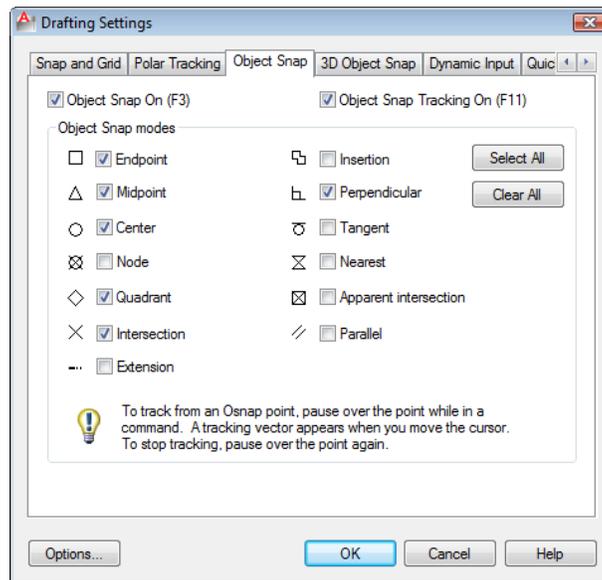
Figure 2.8 – The Object Snap settings tab on the Drafting Settings properties window
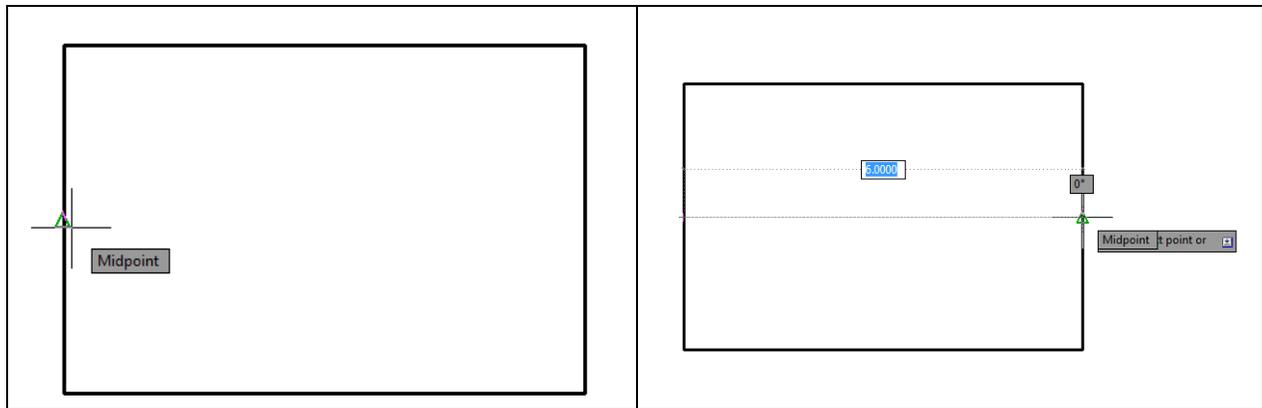
Figure 2.9 – Selecting the midpoint of a line   Figure 2.10 – Selecting another midpoint

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 (triangle) should appear. Select it by clicking the left mouse button (Figure 2.9). 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 2.10). This time, you will only draw one line segment, so hit ENTER on the keyboard to exit the line command. Now 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.
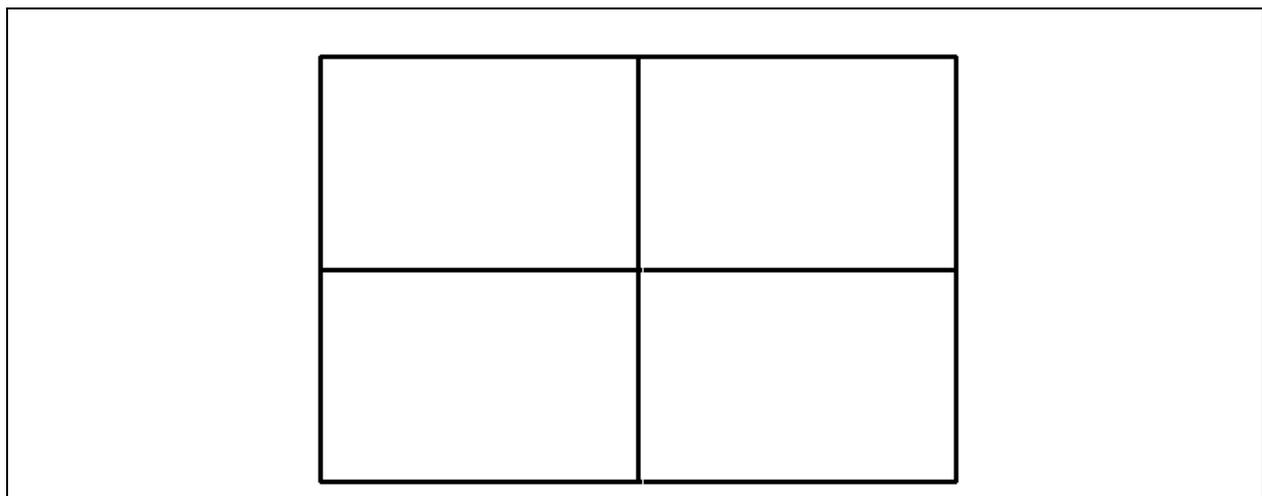
Figure 2.11 – The 6 x 4 rectangle with two centerlines placed exactly in the middle

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 prompt, which in this case is "0.25" (Figure 2.18). Then select the line you wish to offset. Start with the top horizontal line of the rectangle. After selecting the line (Figure 2.12), pick the direction that you want to offset the new line. For this first offset, select anywhere below the horizontal line you initially selected. 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 lines (Figure 2.13). Hit ENTER to leave the Offset command after placing the four new lines.
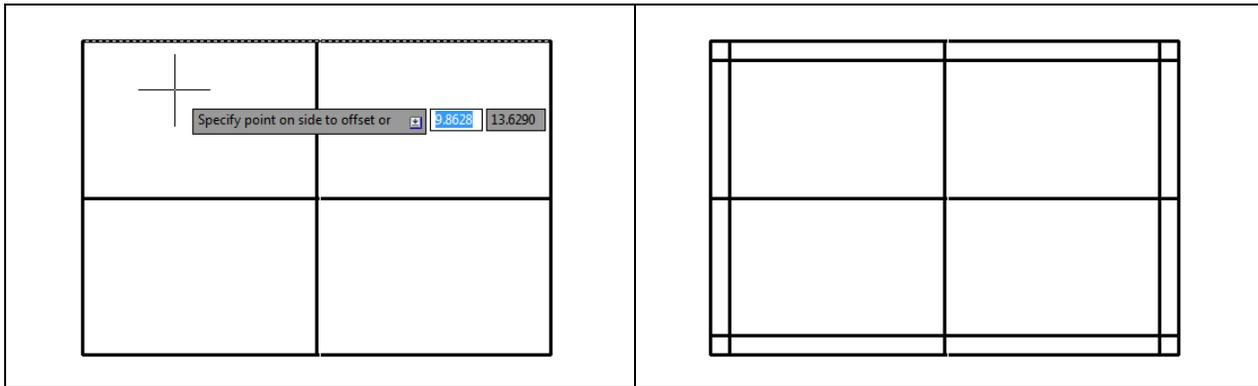
Figure 2.12 – Selecting the offset direction   **Figure 2.13 – Finishing the 0.25 offsets**
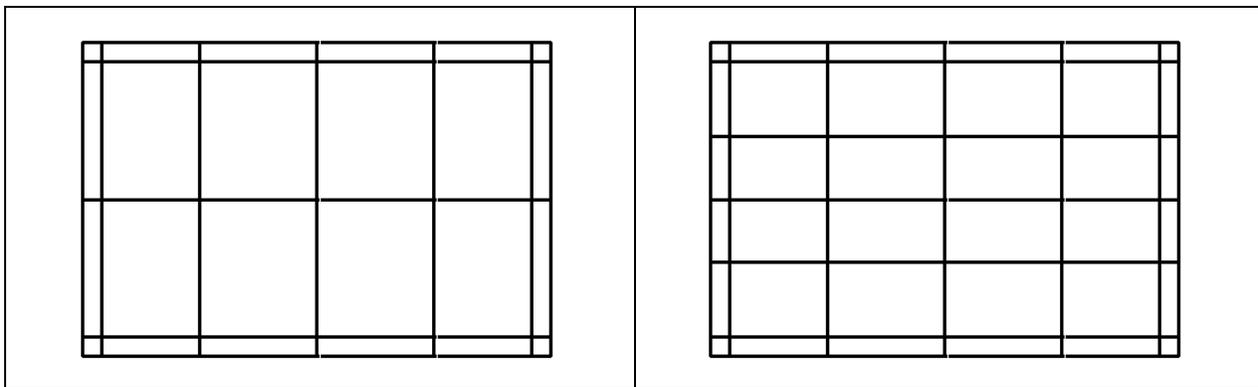
Figure 2.14 – Finishing the 1.5 offsets   **Figure 2.15 – Finishing the 0.8 offsets**

The next line to offset is the vertical centerline. Select the Offset tool from the Modify panel and type 1.5 and ENTER (Figure 2.17). Select the vertical centerline and then select to the right of the line to create a new line 1.5 to the right. Select the vertical centerline again and then to the left of it to create a new line 1.5 to the left. Hit ENTER to leave the Offset command after placing the two new lines (Figure 2.14).

The final line to offset is the horizontal centerline. Select the Offset tool from the Modify panel and type 0.8 and ENTER (Figure 2.18). Select the horizontal centerline and then select above the line to create a new line 0.8 above it. Select below the horizontal centerline to create a new line 0.8 below. Hit ENTER to leave the Offset command after placing the two new lines (Figure 2.15).

Figure 2.16   Command: _offset
Specify offset distance or [Through] <1.0000> 0.25

Figure 2.17   Command: _offset
Specify offset distance or [Through] <1.0000> 1.5

Figure 2.18   Command: _offset
Specify offset distance or [Through] <1.0000> 0.8
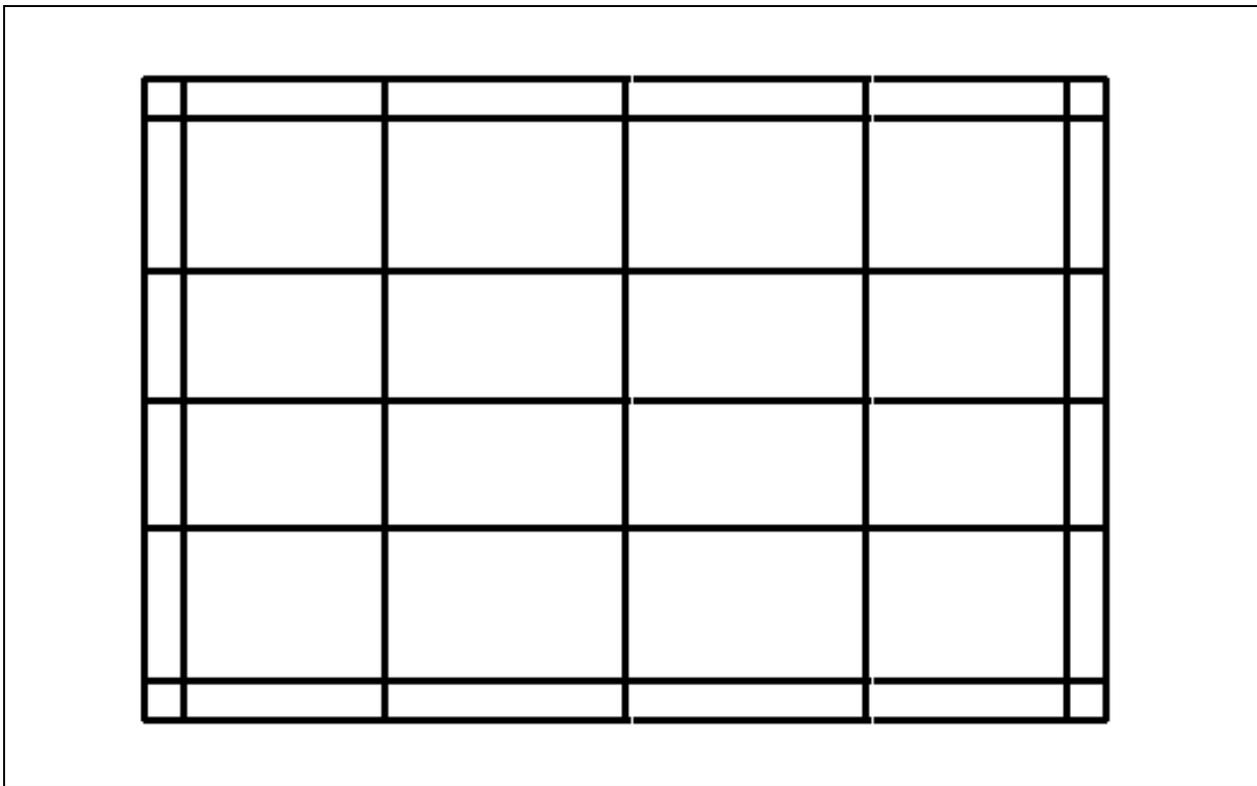
Figure 2.19 – The 4 x 6 rectangle with two centerlines and eight offset lines

* 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   –/--- Trim
of an existing line. Think of this command function just as you would a pair of scissors.
Select the Trim command on the Modify panel under the Home tab and hit 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 2.20). Notice how other
entities on the screen act as boundaries to each segment you select to trim. Continue to
select the two middle line segments on each side of the perimeter as shown in Figure
2.21. Finally, select four line segments as shown in Figure 2.22 to prepare for the Fillet
command.
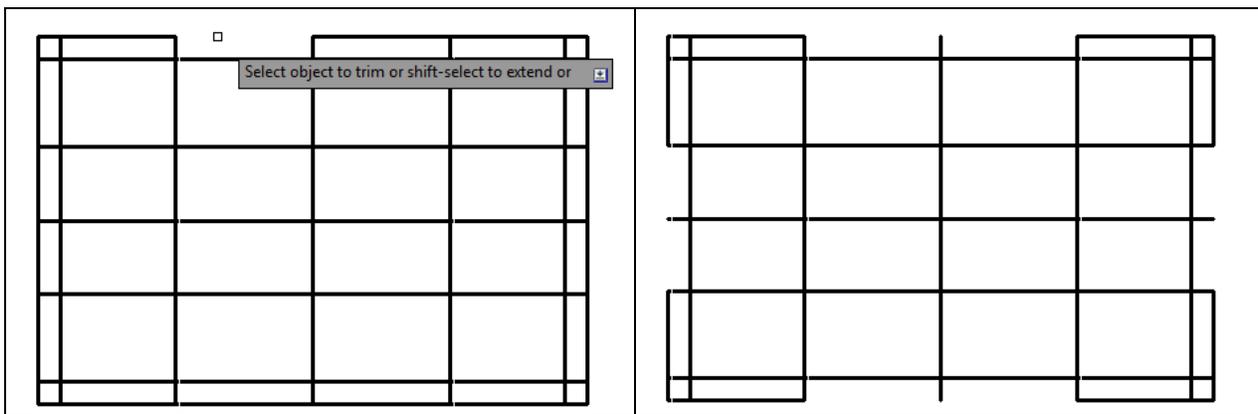
Figure 2.20 – Your first trimmed line   **Figure 2.21 – The perimeter trimmed**
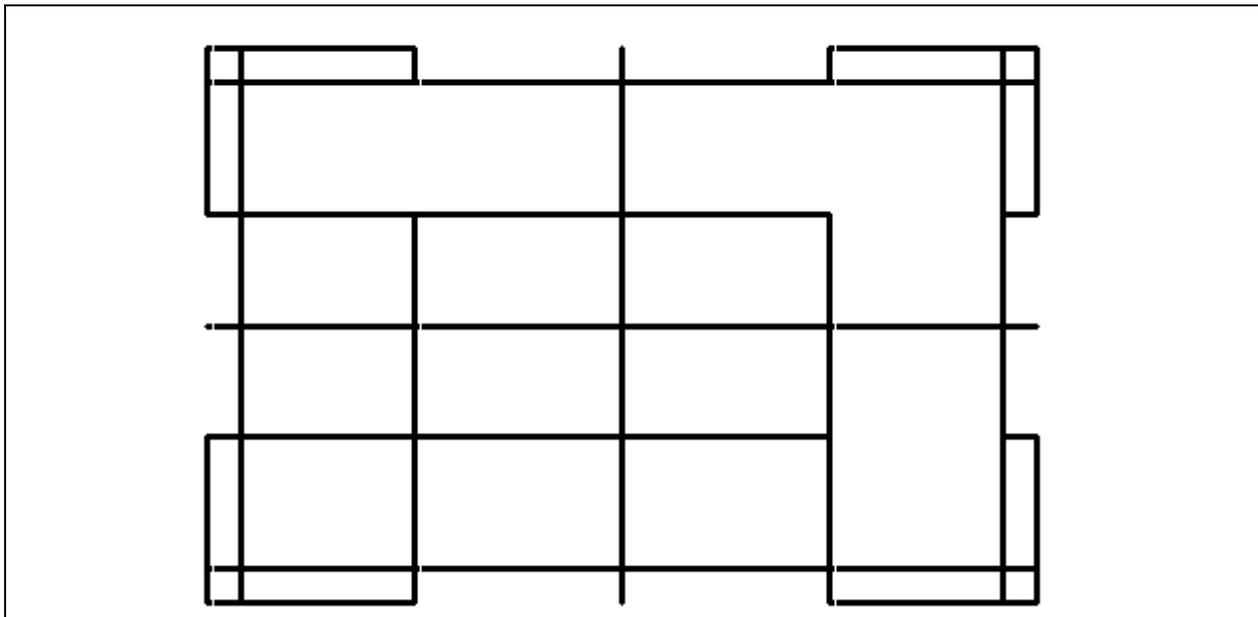
Figure 2.22 – Finished drawing before filleting

Using the Fillet Command

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The Fillet command allows you to add sharp corners (radius of 0) or arcs to the
intersection of two lines. First select the Fillet command from the Modify panel and
then type "R" for radius 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. Finally, type "T" and ENTER to change the trim mode. You will
need to type "T" and ENTER again to toggle the trim mode from No Trim to Trim.
Now, select the two lines between which you want to add a 1/16 radius, as shown in
Figure 2.23. 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 the one shown
in Figure 2.24. You will have placed 8 fillets around the perimeter of the part, 2 on each
side of the rectangle. Hit ENTER to exit the command.
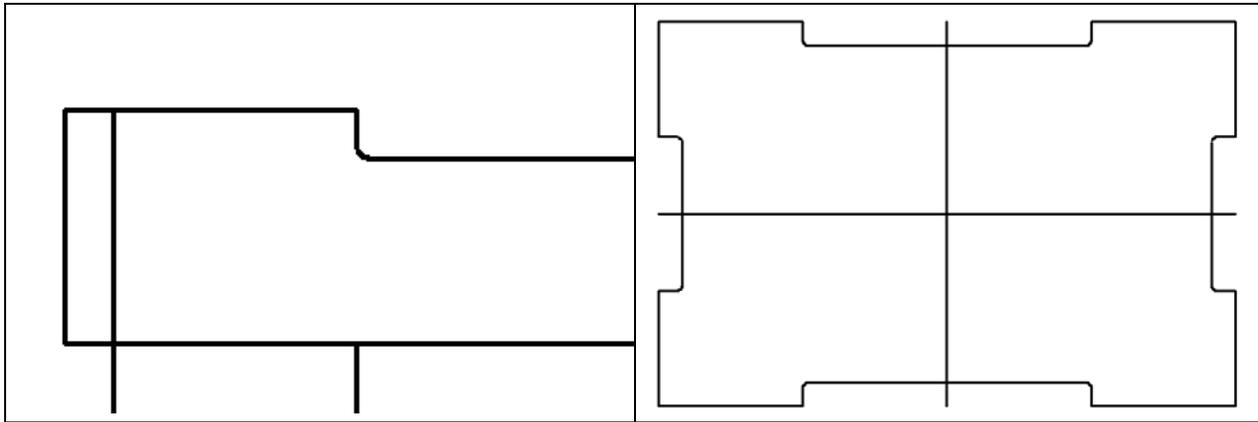
Figure 2.23 – A 1/16" arc   **Figure 2.24 – All 1/16" arcs added**
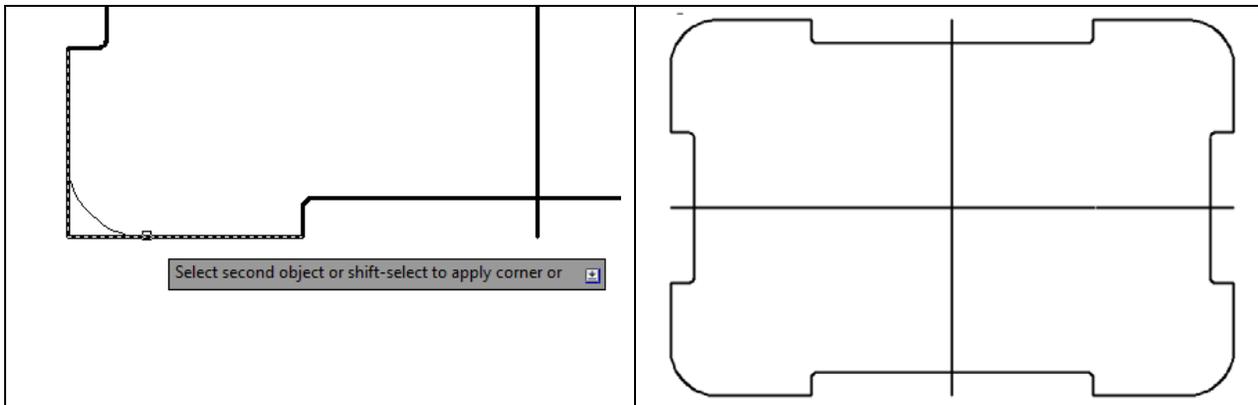
Figure 2.25 – A ½" arc   **Figure 2.26 – All ½" arcs added**

Now the drawing is really starting to take shape. Hit ENTER to repeat the Fillet command.
Type "R" and hit ENTER. Type "0.5" and hit ENTER to set the system for a ½" radius arc.
Again, type "M" and hit ENTER to fillet multiple corners without having to reset the command.
You do not need to reset the trim mode because it should remain toggled on. Select the two

lines that make the corners of the rectangle as shown in Figure 2.25. After selecting the two lines, a ½" radius arc will replace the original corner. Continue to repeat the process until your drawing looks like the one displayed in Figure 2.26. 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. These are valuable skills because 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.
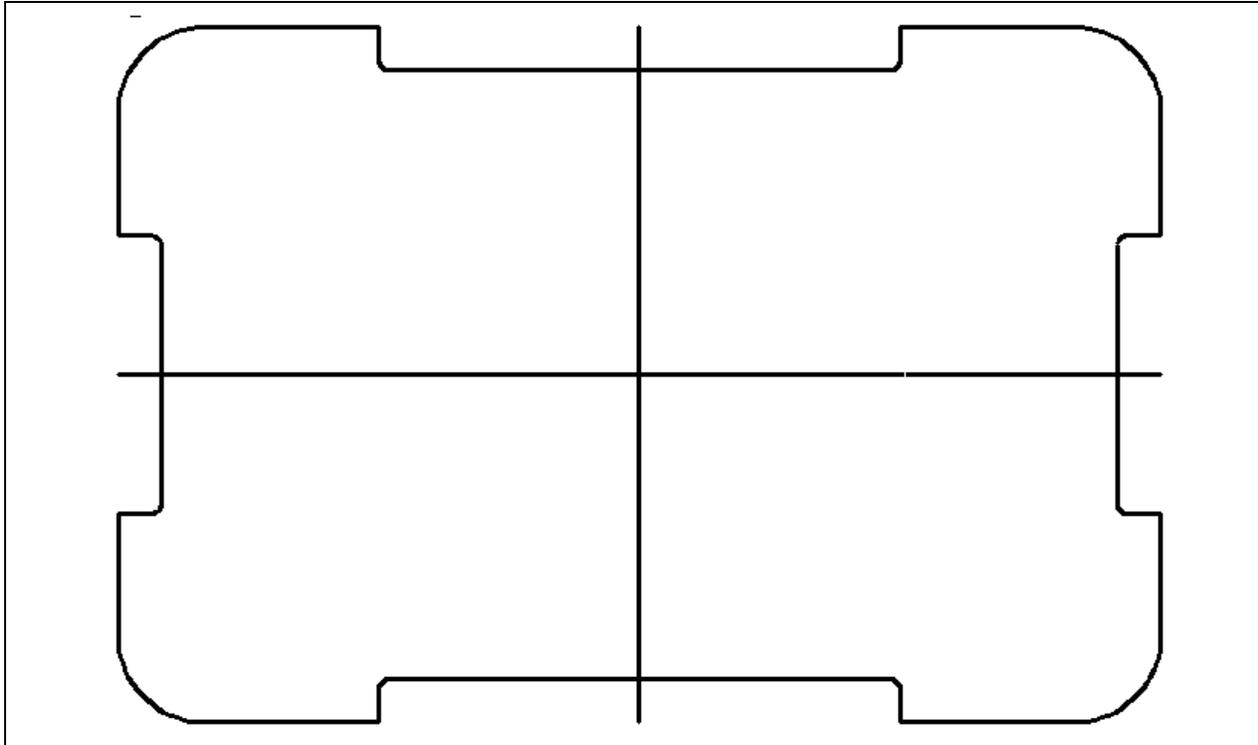
Figure 2.27 – The finished perimeter of the part with two centerlines

* 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

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 (green triangle) appears when the aperture cursor is near the middle of one of the centerlines. Select the Midpoint with the

left mouse button (Figure 2.28). 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 hitting ENTER. For this problem, type 1 and ENTER. A 1 unit radius circle will appear on the drawing (Figure 2.29). Hit 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 hit ENTER. The command line will ask for you to "specify diameter of circle." Type .8 and hit ENTER. A 0.8-unit diameter will appear inside the first circle (Figure 2.30).

Command: _circle Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
Specify radius of circle or [Diameter]: 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>: .8
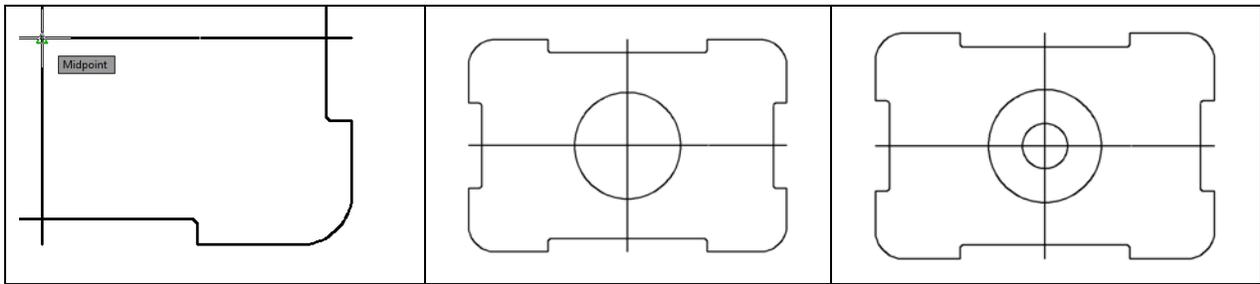
Figure 2.28   **Figure 2.29**   **Figure 2.30**

Hit 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 (green circle) will appear at the center point of the arc (Figure 2.31). Select the point with the left mouse button. Type "d" for diameter and hit ENTER, and then type ".5" and ENTER to create a 0.5 circle (Figure 2.32).
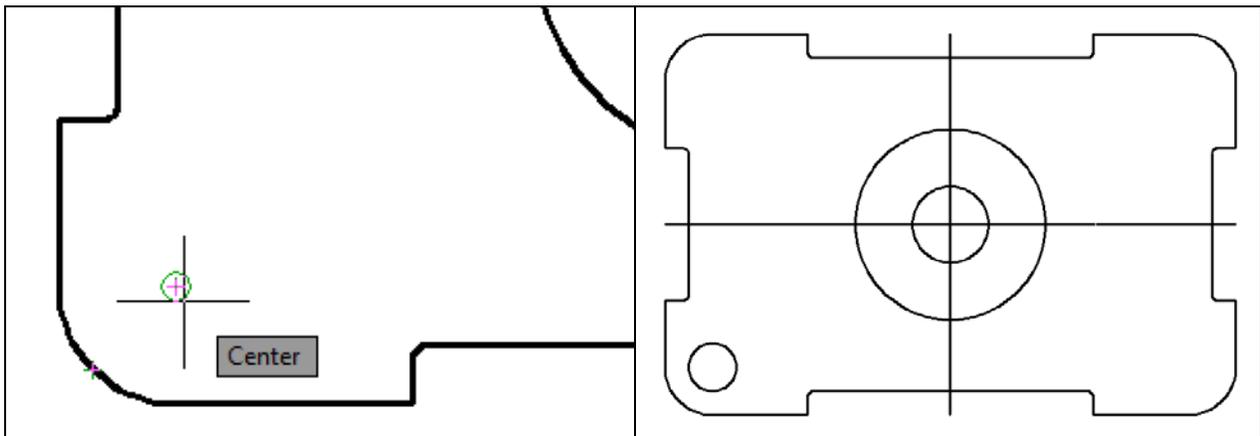
Figure 2.31 – Selecting the center   **Figure 2.32 – The 0.5 circle**

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. ✛ Move These commands are separated into two parts: the first is the selection process and the second is 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 2.33). You can do this by selecting the circle with the pick box that now has replaced the aperture cursor (Figure 2.34). The command line will respond with "1 found," so hit 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 2.35, 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 desired direction of movement as shown in Figure 2.35. 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 2.36)

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 2.33 – The command line sequence for your first Move command
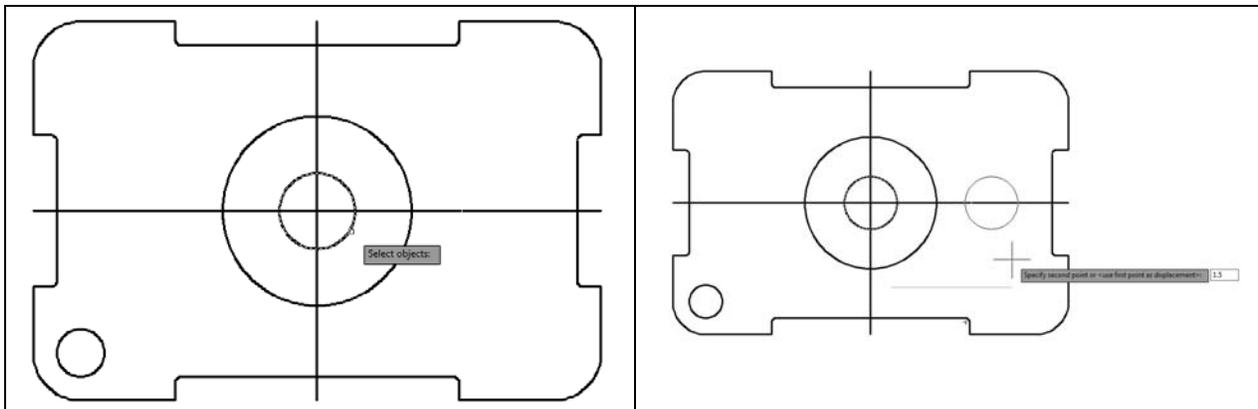
Figure 2.34 – Selecting the circle to move   **Figure 2.35 – The direction of movement**
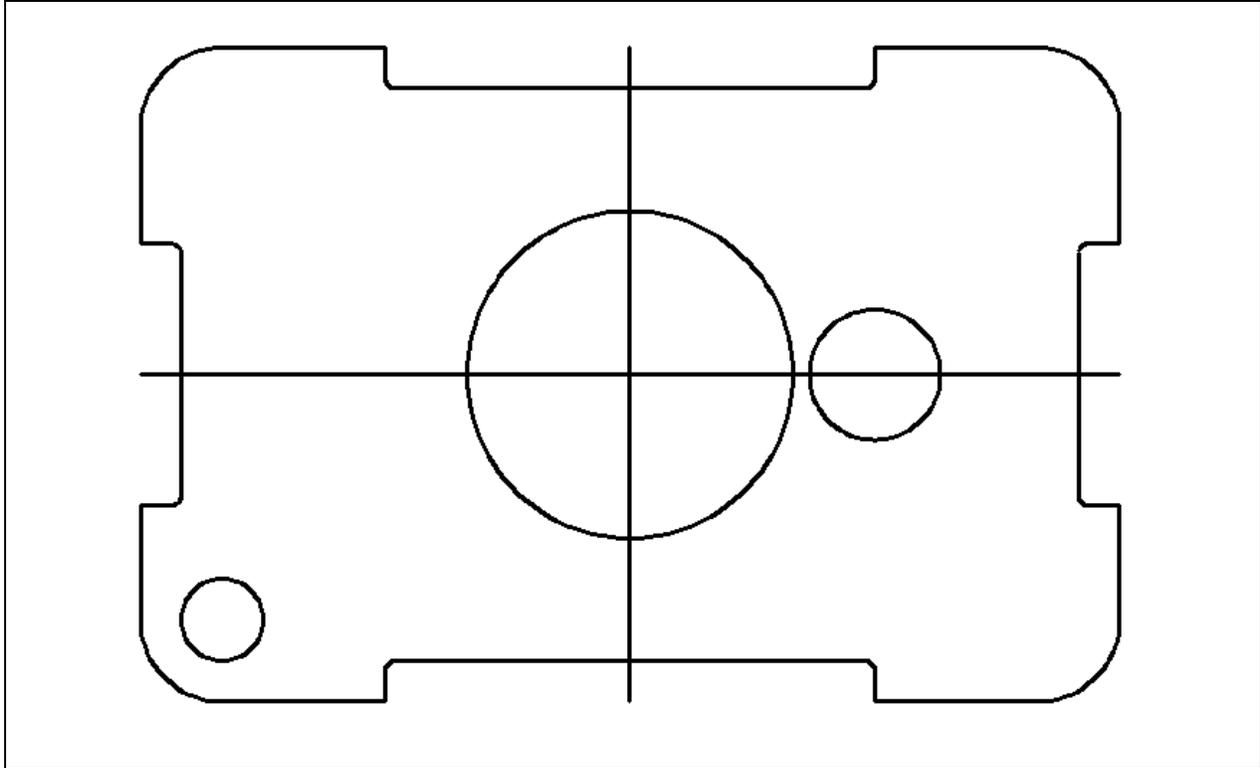
Figure 2.36 – Your first movement of an entity completed

Using the Copy Command

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To copy an AutoCAD entity, such as the 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 2.37). You can do this by selecting the circle with the pick box that now has replaced the aperture cursor (Figure 2.38). The command line will respond with "1 found." Hit 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 2.39 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 2.39 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 2.40)

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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
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Figure 2.37 – The command line sequence for your first Copy command
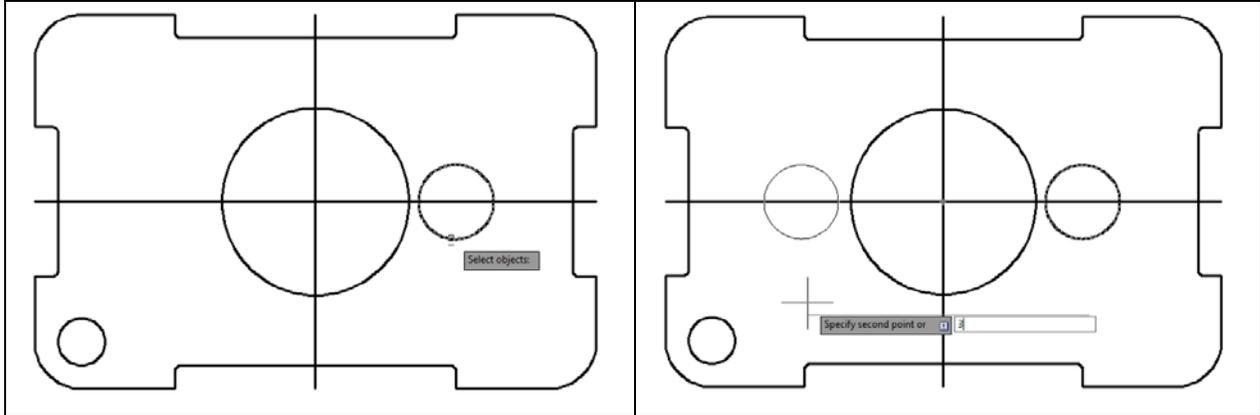
Figure 2.38 – Selecting the circle to copy   **Figure 2.39 - The direction of movement**

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.
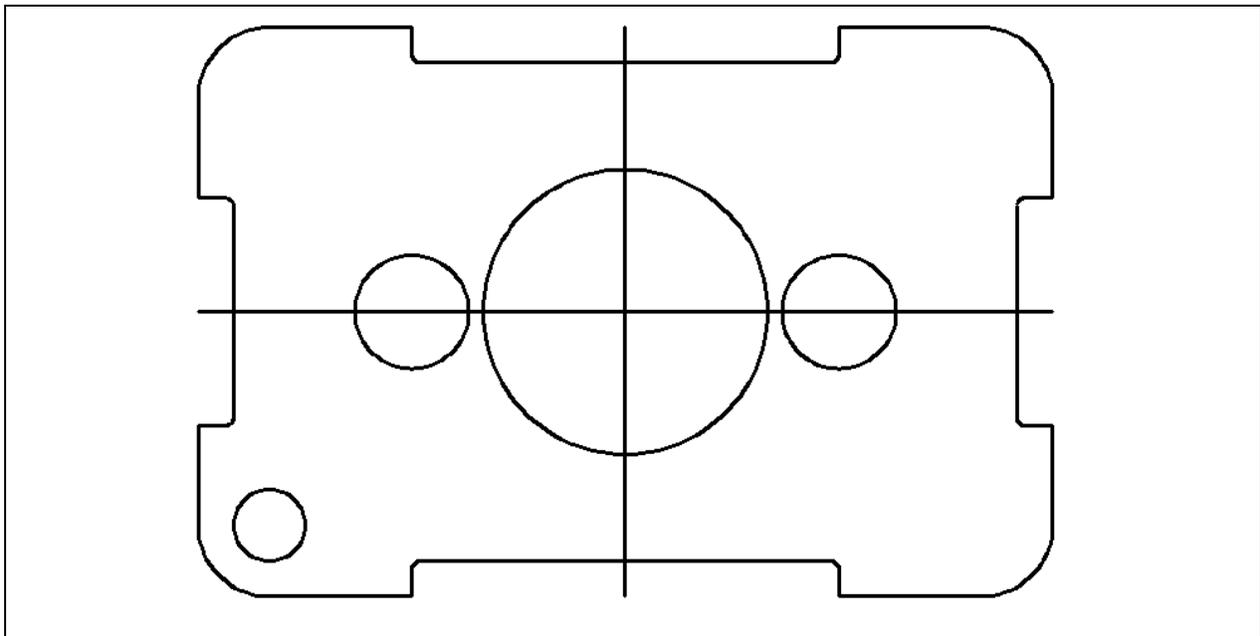
Figure 2.40 - The Copy command completed, resulting in two circles

* 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 0.8 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

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You need to add two lines connecting the upper and lower quadrants of each 0.8 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 (green diamond) will appear (Figure 2.41). Select the quadrant with the left button of the mouse and then select the north quadrant of the right circle (Figure 2.42). Hit ENTER to leave the command. You now have a single line segment connecting the north quadrants of both circles. Hit ENTER again to repeat the Line command and draw a second line connecting the south quadrants of the two 0.8 diameter circles. Your drawing should look like the one shown in Figure 2.43.
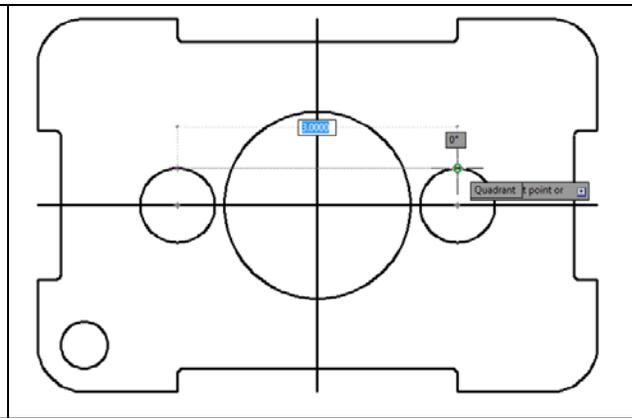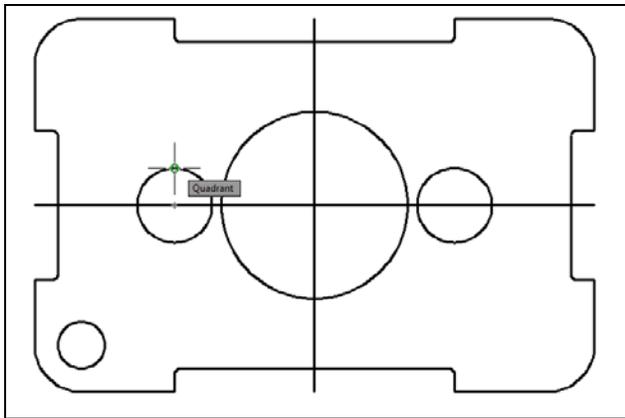
Figure 2.41– Selecting the north quadrant

Figure 2.42 – Connecting the circles
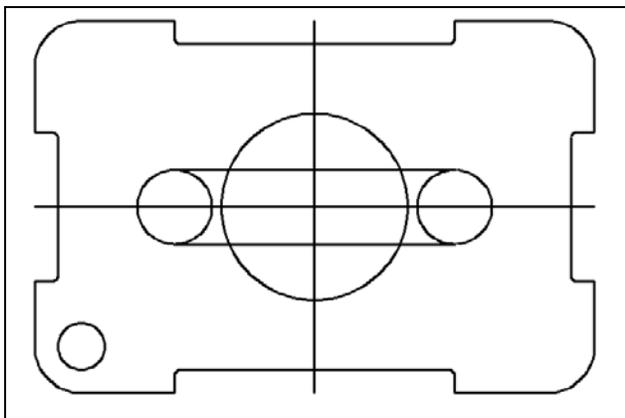
Figure 2.43 – The lower line drawn

The Quadrant Osnap is a very powerful tool that creates four reference points on a circle, compared to the single reference point offered 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 multiple endpoints, midpoints, centers, and quadrants.

Reinforcing the Trim Command

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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. Selection a point inside the upper left-hand corner of the rectangle, select once to create a selection window. Pull the window down to the right so that it completely encases all three circles and the two horizontal lines you just drew (Figure 2.44). Click the left mouse button to finalize the selection window. The selected entities will be dotted as shown in Figure 2.45, 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 2.46). The 6 red dots in Figure 2.46 show the line or arc segments to remove. Finally, our last complex detail is complete as shown in Figure 2.47.
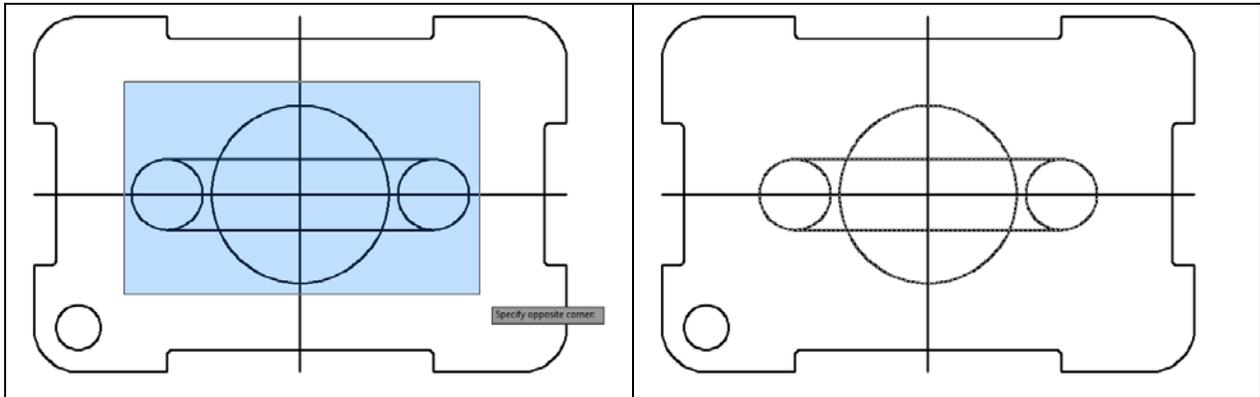
| Figure 2.44 – Selecting by windowing | Figure 2.45 – The selected objects |

| Figure 2.46 – 6 points to trim | Figure 2.47 – The trimming completed |
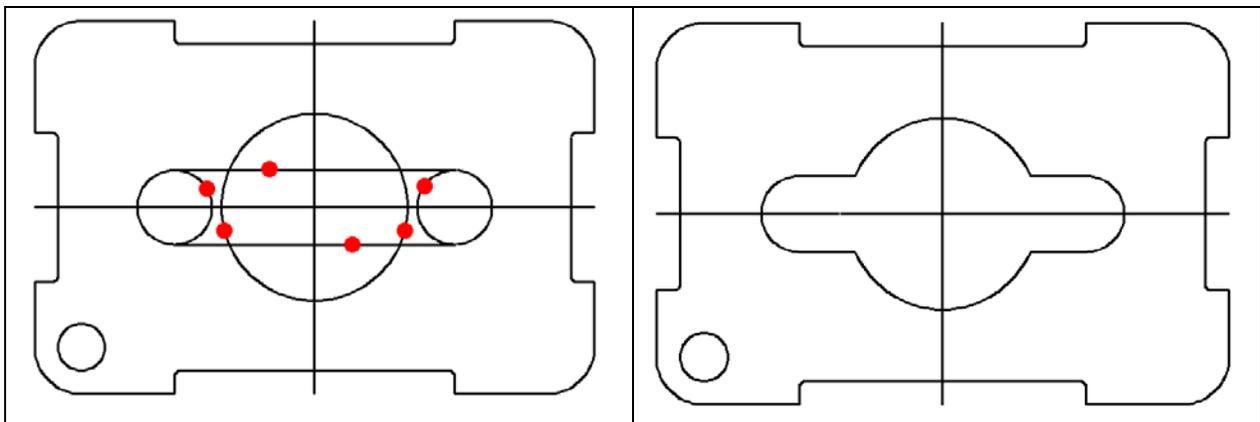
Using the Rotate Command

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The Rotate command will allow you to rotate entities around a base or center point. ⟳ Rotate
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 2.48).
The selected entities will be dotted as in Figure 2.49. To continue with the command,
hit ENTER. Select the Midpoint of the centerline with the left mouse button (Figure
2.50). Then type "45" and ENTER for the degrees of counter clockwise rotation to
finish this detail (Figure 2.51).
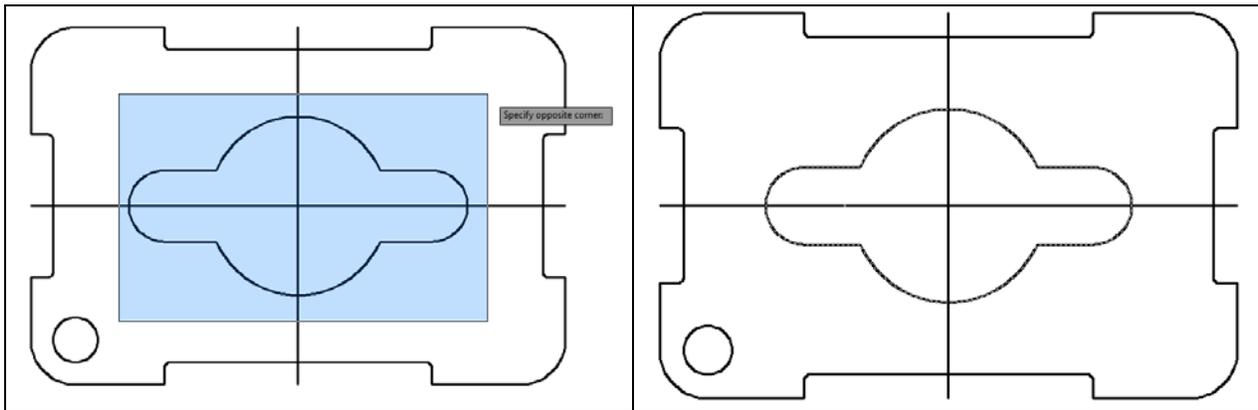
Figure 2.48 – Selecting by windowing   **Figure 2.49 – The selected detail**
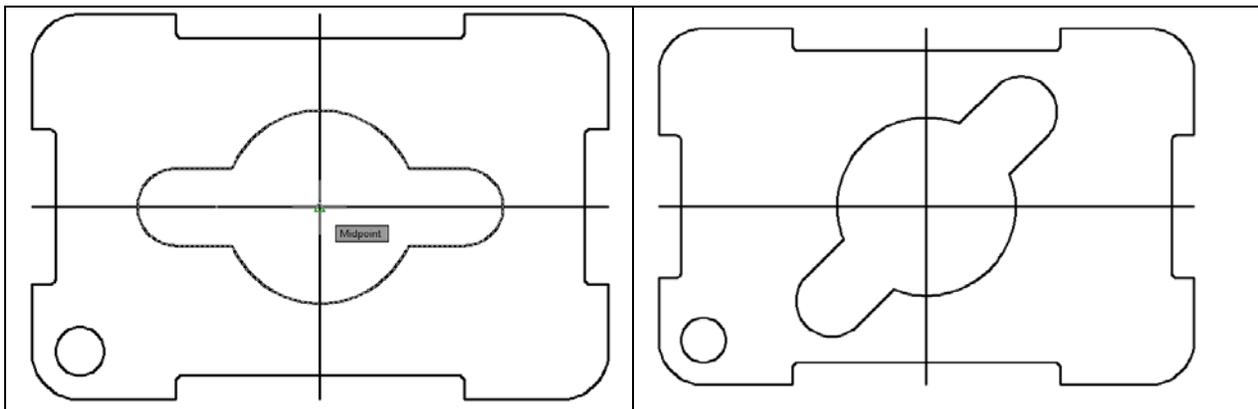
Figure 2.50 – Selecting the midpoint   **Figure 2.51 – A 45 degree rotation**

*** 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 0.8 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

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The Array command allows you to create a rectangular pattern of entities from selected   ⊞ Array ▾
objects, in this case the 0.5 unit circle. The array command has also been revised for the
2012 edition of AutoCAD. In previous versions of the software, the array command was driven
by a dialogue window which contained a number of options and settings you would need to set
up, and then you could preview the array before finalizing it. However, the new version of the
array command does not take you to another window. You can complete the command quickly
and easily with just a few selections on the graphical display. However, before you get started
you should toggle the Ortho Mode off by hitting F8 at the keyboard or selecting the Ortho
Mode icon on the tray.

Select the Array command from the Modify panel under the home tab. Next, select the 0.5-
diameter circle in the bottom left-hand corner of the rectangle as the object you want to array
(Figure 2.52). Hit ENTER to continue on to the next step of the command. Now you will be
prompted to select a corner to specify the number of items in the array. Move your cursor up
and to the right so that three more 0.5 circles appear in the graphical display and select with the
left mouse button (Figure 2.53). Now move the top right circle to the center of the arc in the top
right corner of the rectangle as shown in Figure 2.54. You will see that each circle in the array
will be properly spaced (Figure 2.55). Hit ENTER to accept the array you have created and to
exit the array command.
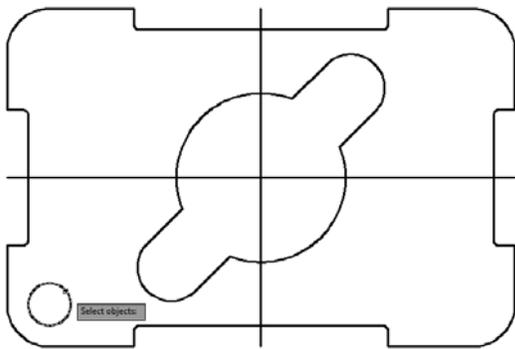
Figure 2.52 – Selecting the Circle
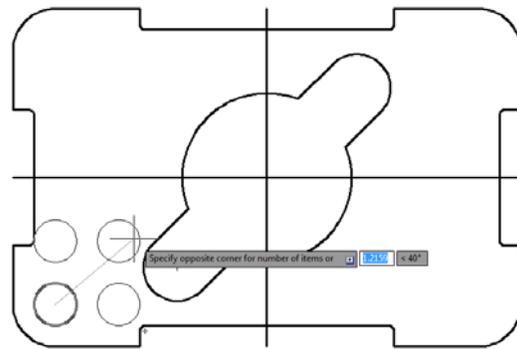
Figure 2.53 – Creating 4 circles
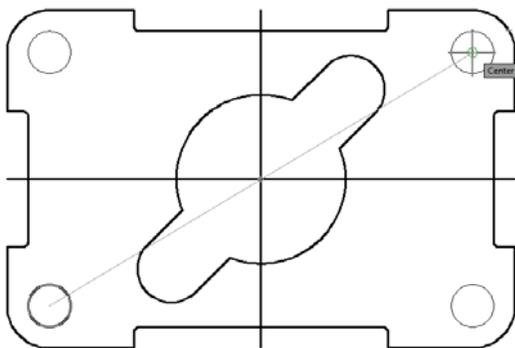
Figure 2.54 – Spacing the array
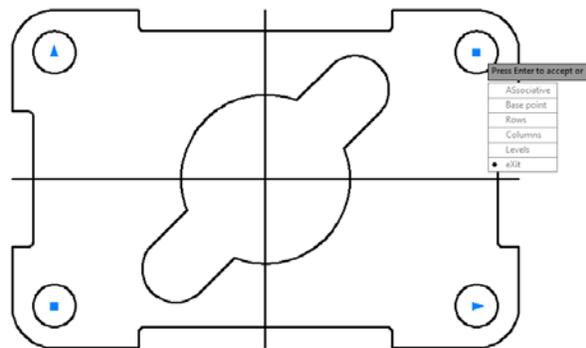
Figure 2.55 – Approving the array

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.
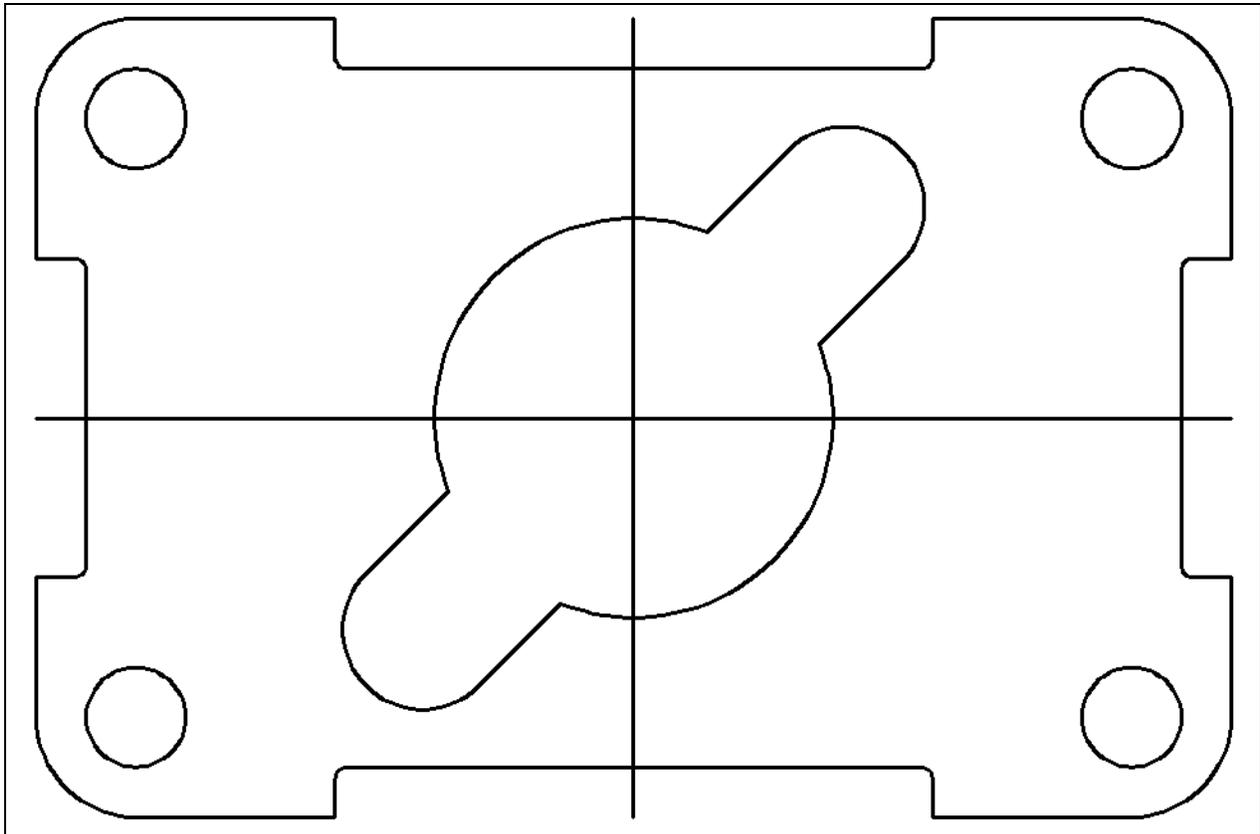
Figure 2.56 – The finished 2D part prior to dimensioning

* 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 0.8 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.