

Chapter

2

The Rectangular Problem

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

1. Using the tools and toolbars for simple 2D Computer Aided Drafting (CAD)
2. Using the Line command to draw a closed polygon
3. Using the Line command to draw single line segments
4. Using Entity Snaps (Esnaps) for accurate graphical selecting
5. Checking your time to create a drawing
6. Using the Offset command to create new and accurate lines
7. Using the Trim command to divide lines into two, change line lengths and create arcs from circles.
8. Using the Fillet command to trim lines and add arcs
9. Using the Circle command to make circles with a radius or diameter
10. Using the Move command to change the location of an entity
11. Using the Copy command to create a new entity at another location on the drawing
12. Using the Rotate command to rotate a group of entities
13. Using the Array command to create a rectangular pattern
14. Achieving a high industry standard to be carried throughout your training

Working with toolbars

The rectangular problem was designed to give new CAD students an easy way to learn the most basic drawing techniques and commands. There are ten toolbars which you will need to use for the Rectangle problem. After starting progeCAD 2009, select View from the menu bar and select Toolbars from the bottom of the drop menu. The Select Toolbars window will appear. From here you can select which toolbars to use. The rectangle problem requires the Text, Draw, Object Snaps, Entity Properties, Standard, Dimension, and Modify toolbars. Place these toolbars around the outside your screen.

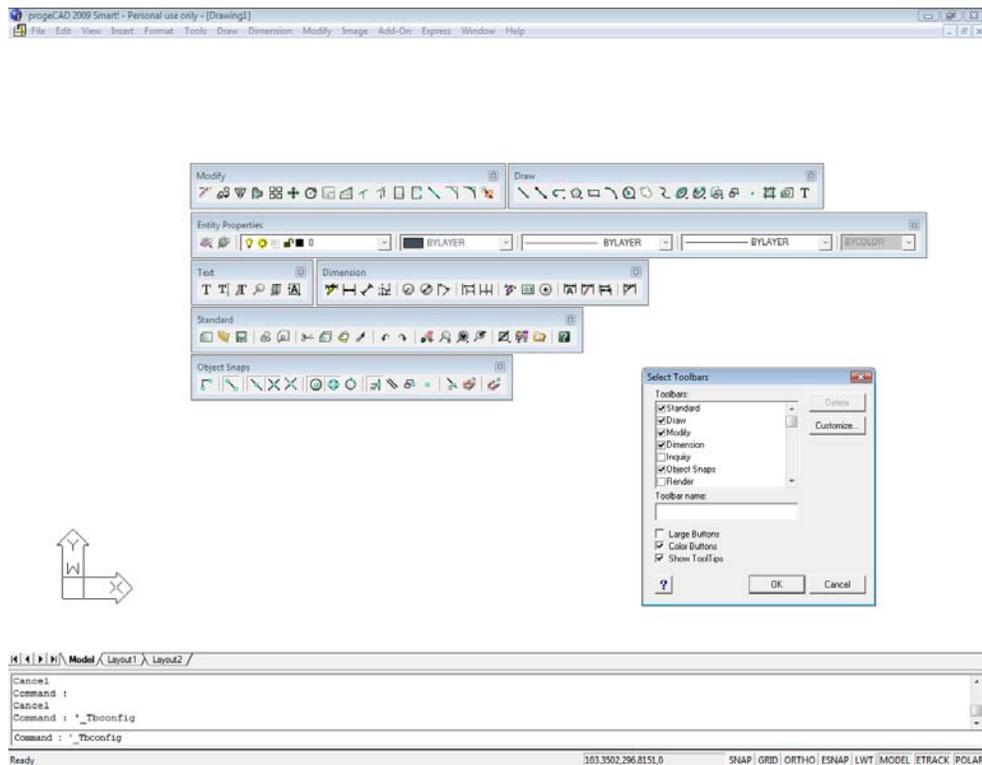


Figure 1.1 Move the toolbars by placing the mouse pointer over the blue section of the toolbar, clicking and holding the left mouse button and moving the toolbar to the desired position

Parking the toolbars is relatively easy by placing the Draw, Modify, , and Text toolbars on the left side of the view screen, the Standard and Entity Properties toolbars and the top of the view screen and the Dimension and Object Snap toolbars on the bottom or right side of the view screen; the choice is up to you.

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. Select the Line tool from the Draw toolbar. 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 1.2). To draw the vertical line, move the mouse upward, dragging the line and type “4” and ENTER (Figure 1.3). Drawing the third line, move the mouse to the left, dragging the line and type “6” and ENTER (Figure 1.4). To close the rectangle, just type “c” and ENTER.



Figure 1.2

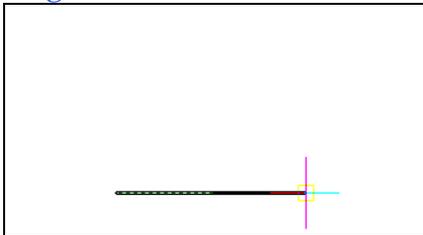


Figure 1.3

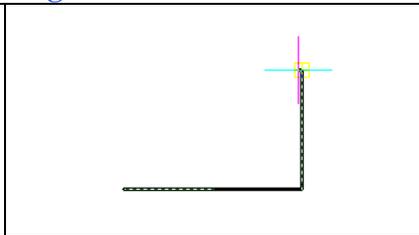


Figure 1.4

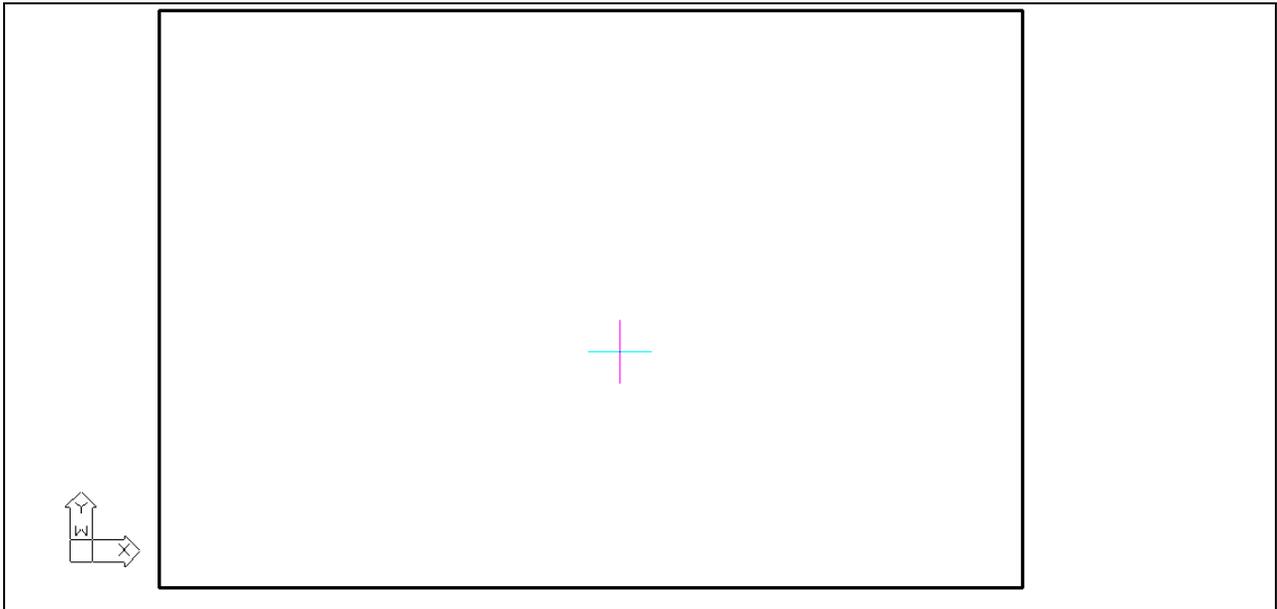
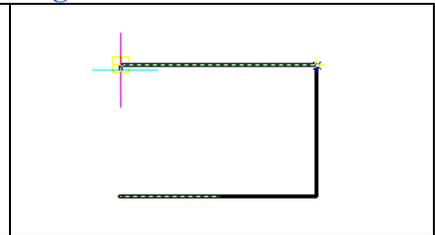


Figure 1.5 The finished 6 x 4 rectangle.

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.

Command : `_LINE`
 ENTER to use last point/Follow/<Start of line>:
 Angle/Length/<End point>:
 Angle/Length/Follow/Undo/<End point>: **6**
 Angle/Length/Follow/Close/Undo/<End point>: **4**
 Angle/Length/Follow/Close/Undo/<End point>: **6**
 Angle/Length/Follow/Close/Undo/<End point>: **c**

Figure 1.6 The Command Line Prompts from the Line Command

*** World Class CAD Challenge 100-01* – close this file by selecting File at the pull down menu 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 Entity Snaps

Setting Entity Snaps allows you to select the point precisely. Pick the ESnap Setting tool on the Object Snap toolbar (the magnet). In the rectangular problem, you need to select the following Entity 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), 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 90° from your first selection point (Figure 1.7).

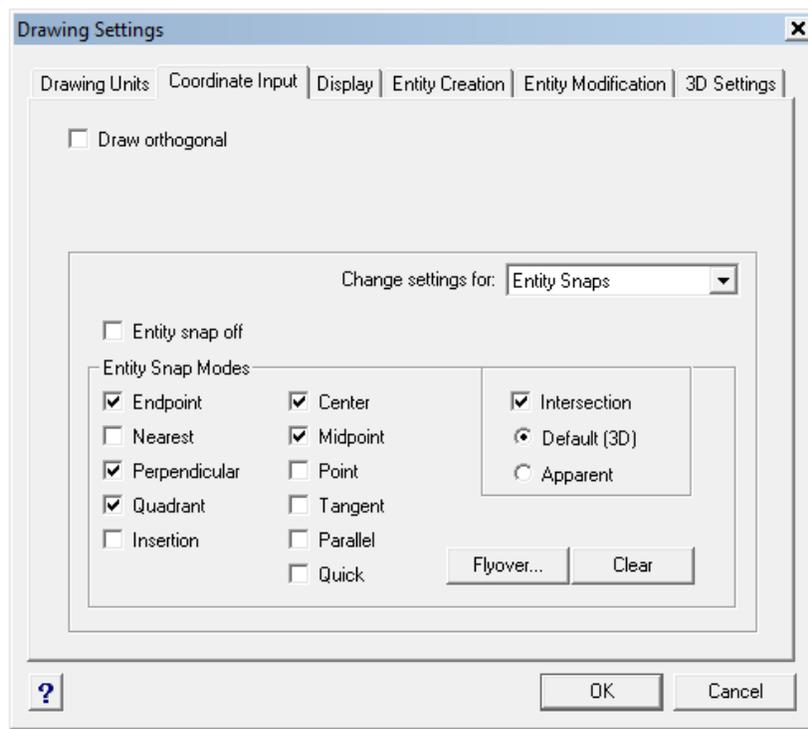


Figure 1.7 The Entity Snap Settings Area on the Drawing Settings Window

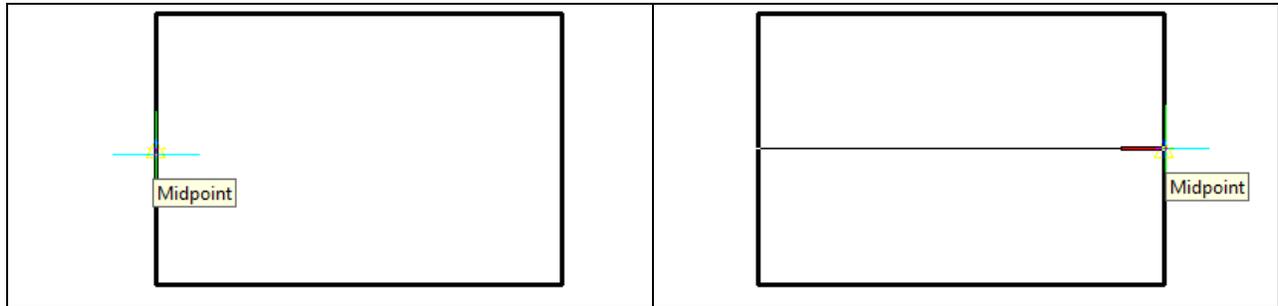


Figure 1.8 – Selecting the midpoint of a line **Figure 1.9 – Selecting another midpoint**

The next two lines you will draw are the two centerlines dividing the rectangle. Now that the Entity Snap Mode (referred from now on as ESNAP) 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 ESNAP symbol should appear. Select it by clicking the left mouse button (Figure 1.8). Move your mouse to the center of the right vertical line and when the triangle ESNAP symbol appears, select the midpoint by clicking the left mouse button (Figure 1.9). 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 ESNAP 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 ESNAP symbol to appear. Select the line with the left mouse button.

Step 3. Press ENTER to exit the Line command.

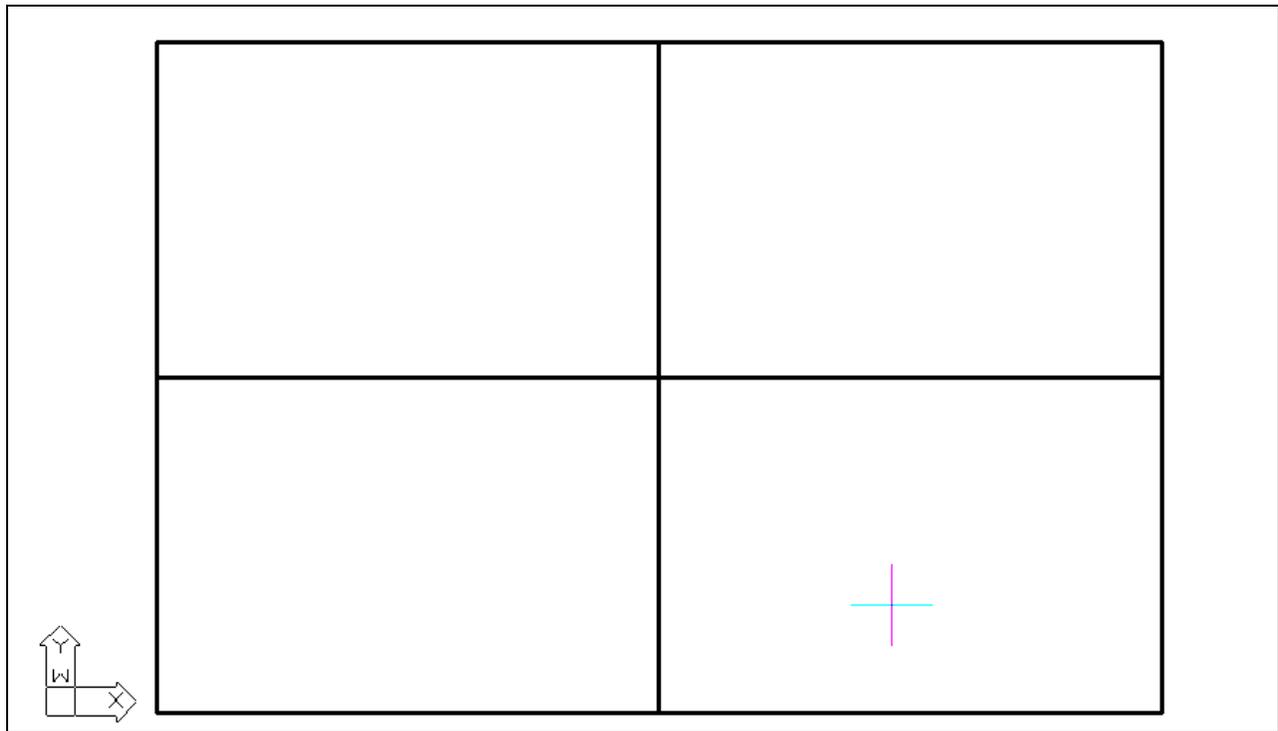


Figure 1.10 The 6 x 4 rectangle with two centerlines exactly in the middle.

*** World Class CAD Challenge 100-02*** - 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

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 toolbar. Next, answer the offset distance query on the command line, which in your problem is “0.25” (Figure 1.15). Then select the line you wish to offset, in this case the top horizontal line. After selecting the line (Figure 1.11), 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 1.12). Hit ENTER to leave the Offset command after placing the four new lines.



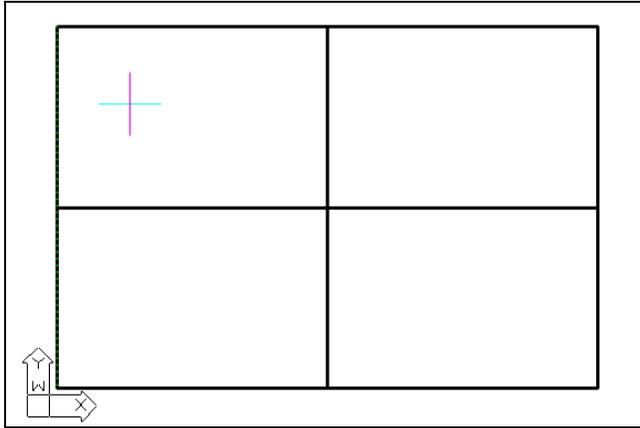


Figure 1.11 – Selecting the offset direction

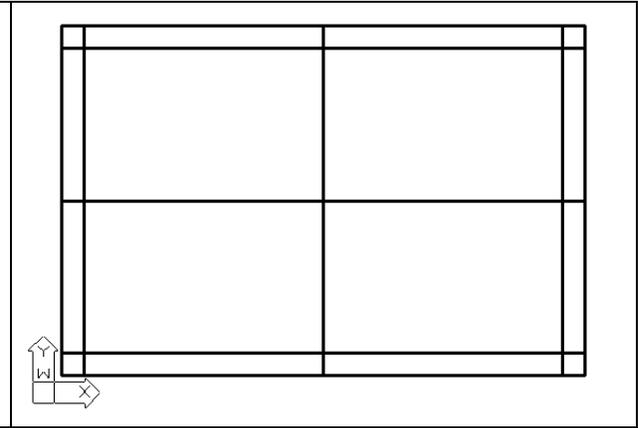


Figure 1.12 – Finishing the 0.25 offsets

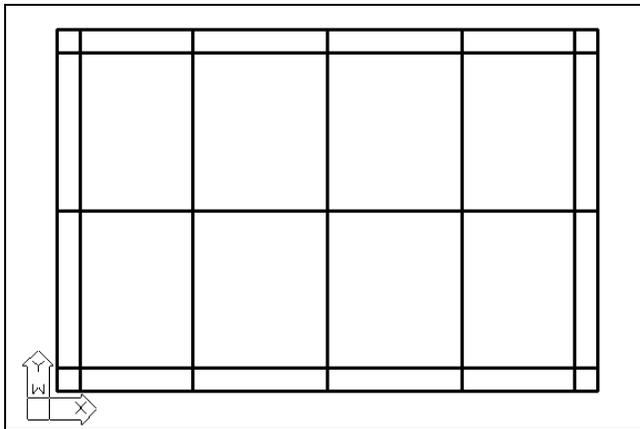


Figure 1.13 – Finishing the 1.5 offsets

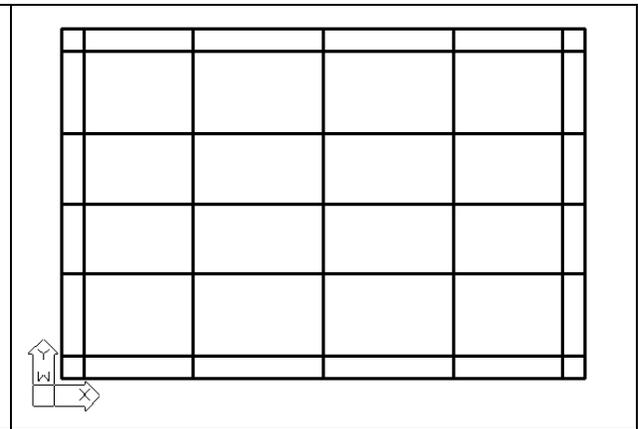


Figure 1.14 – Finishing the 0.8 offsets

The next line to offset is the vertical centerline. Select the Offset tool from the Modify toolbar and type 1.5 and ENTER (Figure 1.16). 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 1.3).



The final line to offset is the horizontal centerline. Select the Offset tool from the Modify toolbar and type 0.8 and ENTER (Figure 1.17). 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 1.14).

Figure 1.15 Command : `_PARALLEL`
Parallel: Through point/<Distance> <0.25>: .25

Figure 1.16 Command : `_PARALLEL`
Parallel: Through point/<Distance> <0.25>: 1.5

Figure 1.17 Command : `_PARALLEL`
Parallel: Through point/<Distance> <1.5>: .8

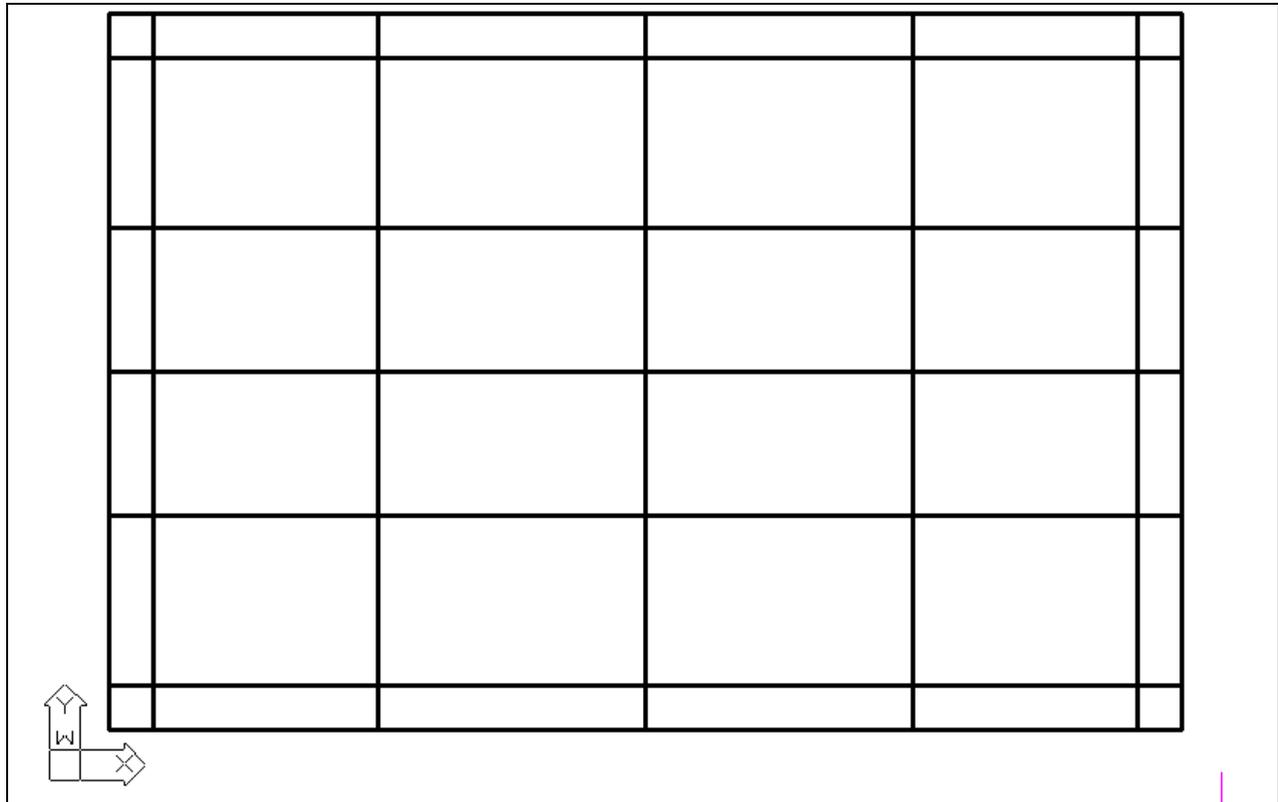


Figure 1.18 The 4 x 6 rectangle with two centerlines and eight offset lines

*** World Class CAD Challenge 100-03 * - 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

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 1.19). 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 1.20. Finally, select four line segments as shown in Figure 1.21 to prepare for the Fillet command.



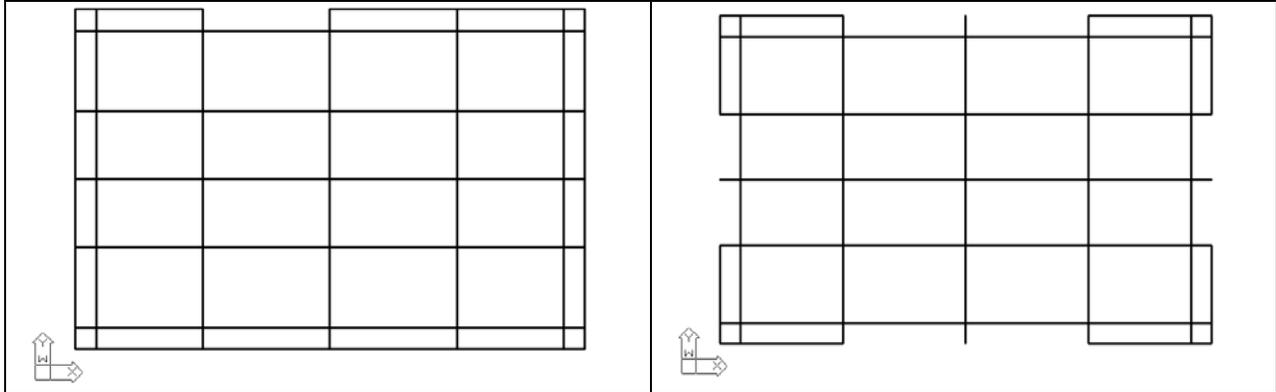


Figure 1.19 – Your first trimmed line

Figure 1.20 – The perimeter trimmed

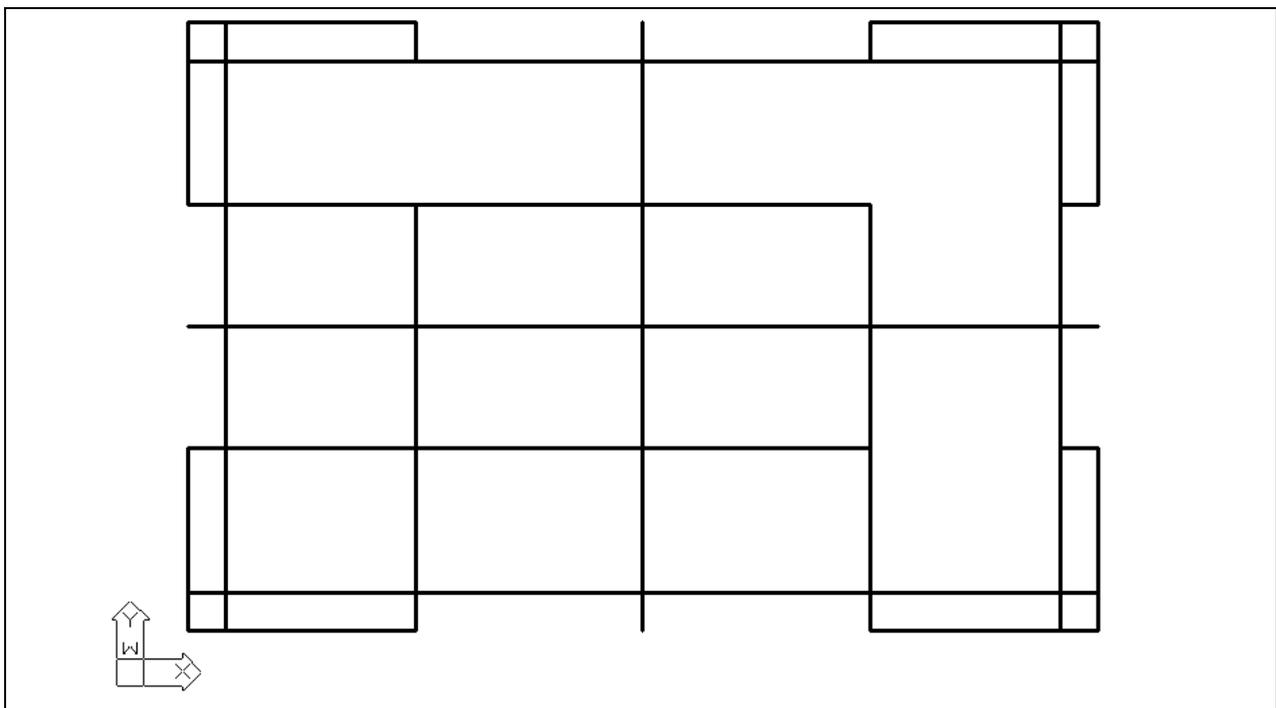


Figure 1.21 Finished drawing before filleting

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. Select the two lines that to which you want to add a 1/16 radius, as shown in Figure 1.22. After selecting the two lines, the lines will shorten and a 1/16 radius arc will appear. Press ENTER to repeat the command and select the next two lines. Continue to repeat the process until your drawing looks like ours in Figure 1.23.



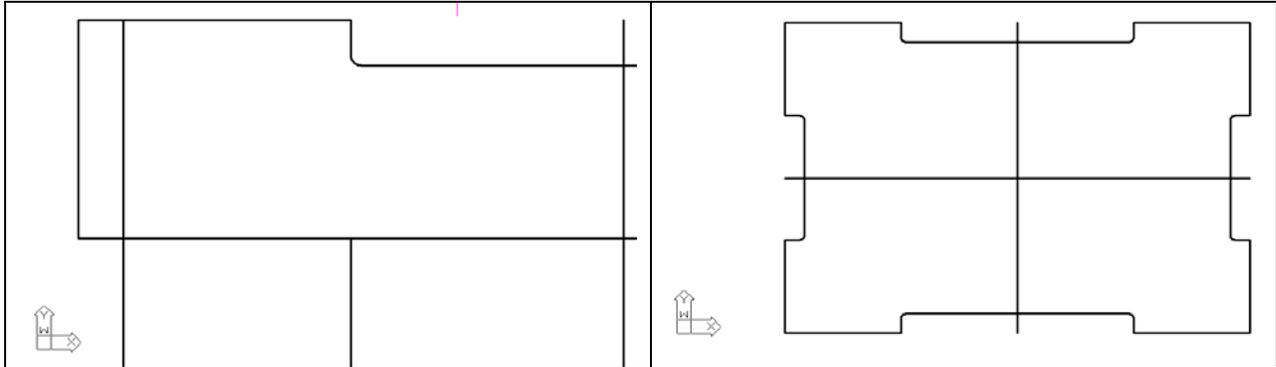


Figure 1.22 – Select the two lines to remain

Figure 1.23 – All 1/16 arcs added

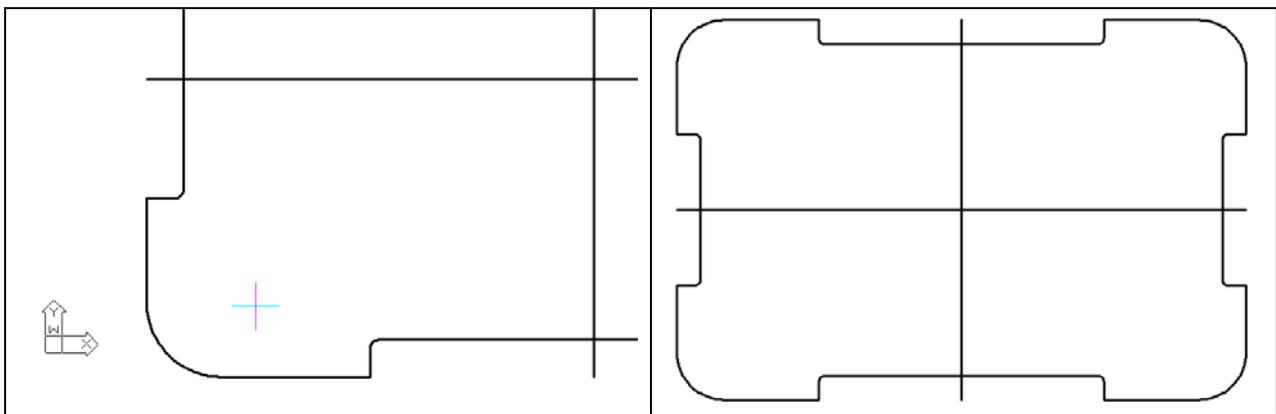


Figure 1.24 – One 0.5 Radius

Figure 1.25 – The finished perimeter

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. Select the two lines that make the corners of the rectangle, as in Figure 1.24. After selecting the two lines, the lines will shorten and a 1/2 radius arc will appear. Press ENTER to repeat the command and select the next two lines. Continue to repeat the process until your drawing looks like Figure 1.25.

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. In World Class CAD II, you will find that associate World Class CAD users follow these same processes to create any residential floor plan in less than one hour, hours ahead of the standard industry rate.

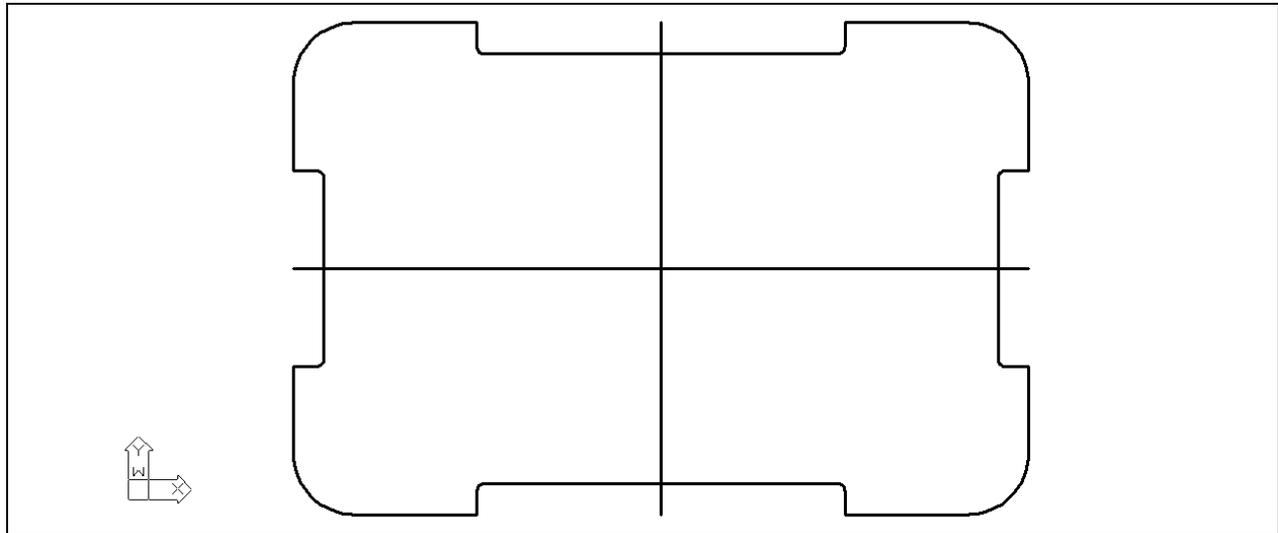


Figure 1.26 The finished perimeter of your part with two centerlines

*** World Class CAD Challenge 100-04*** - 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 toolbar. With your mouse, select the Midpoint of one of your centerlines. Notice that while your Esnaps 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 2.27). On the command line, a statement asking you to specify the diameter or radius of the circle 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 2.28). Press ENTER to repeat the Circle command and select the Midpoint of the centerline again. When the statement 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 2.29).



```
Command : _CIRCLE
2Point/3Point/RadTanTan/TTT/Arc/Multiple/<Center of circle>:
Diameter/<Radius> <0.375>: 1

2Point/3Point/RadTanTan/TTT/Arc/Multiple/<Center of circle>:
Diameter/<Radius> <1>: d
Diameter of circle <2>: .75
```

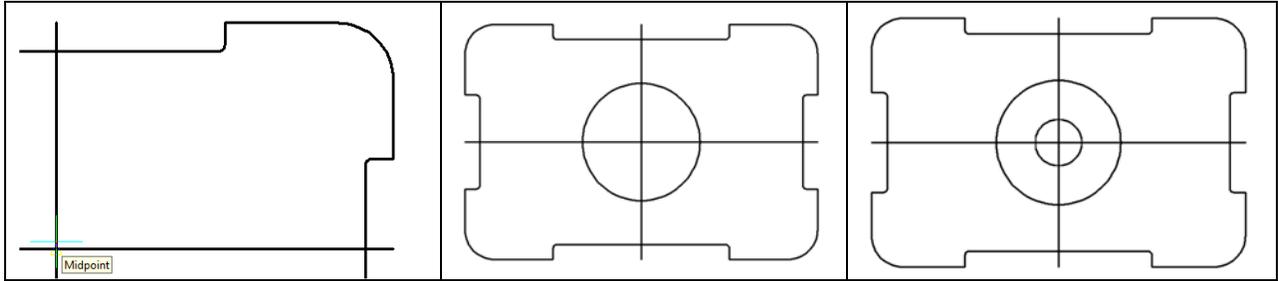


Figure 2.27

Figure 2.28

Figure 2.29

Press ENTER to repeat the Circle command. Again, the Esnap settings will aid you in finding the center of the radius that was drawn previously in the corner of the rectangle. When you specify the center point for the circle, use your mouse to place the aperture cursor over the arc in the lower left hand corner of the drawing (Figure 2.29). The Center Esnap symbol (orange circle) will appear, at the center point of the arc. Select the Center with the left mouse button. Type “d” for diameter and hit Enter, and type “.5” to create a 0.5 circle.

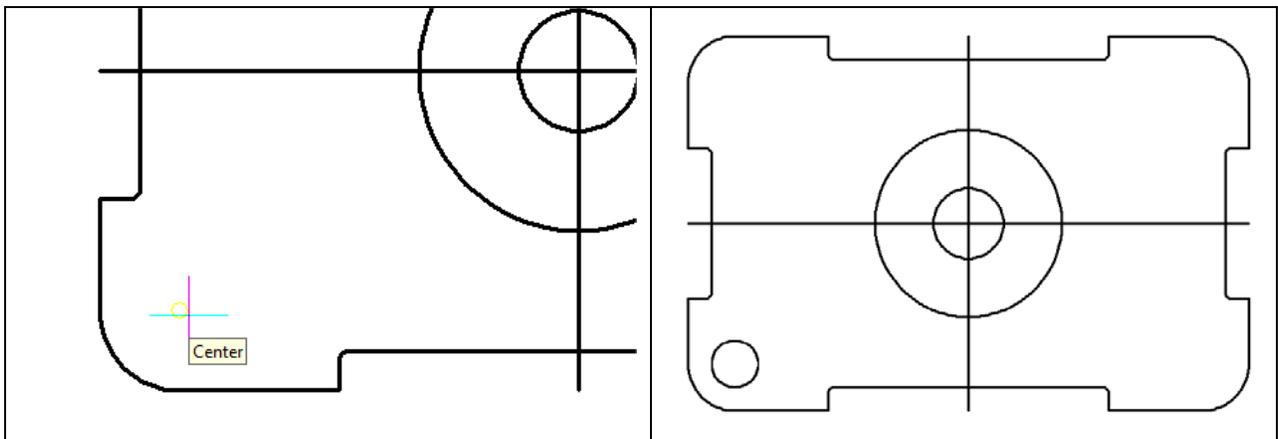


Figure 2.30

Figure 2.31

```
Command: _circle Specify center point for circle or [3P/2P/Tr (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 toolbar. 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 a progeCAD entity, such as a circle, select the Move command on the Modify toolbar. The command line will prompt you to “Select entities to move” (Figure 2.32). You can do this by selecting the circle with the pick box that now has replaced the aperture cursor (Figure 2.33). The command line will respond with “Entities in set: 1”, so press ENTER to go the second



part of the command. Now you need to specify a base point on the graphical display. Pick any point away from any entities so that your Esnaps do not interfere with the movement. In Figure 2.34, it shows a base point to the lower right hand corner 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 2.34. The Ortho setting is still on, so the movement will be at a perfect 0 degrees to the right. Type 1.5 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 2.35)

```
Command : _MOVE  
Select entities to move:  
Entities in set: 1  
Select entities to move:  
Vector/<Base point>:  
Displacement point: 1.5
```

Figure 2.32 – The command line sequence for your first Move command

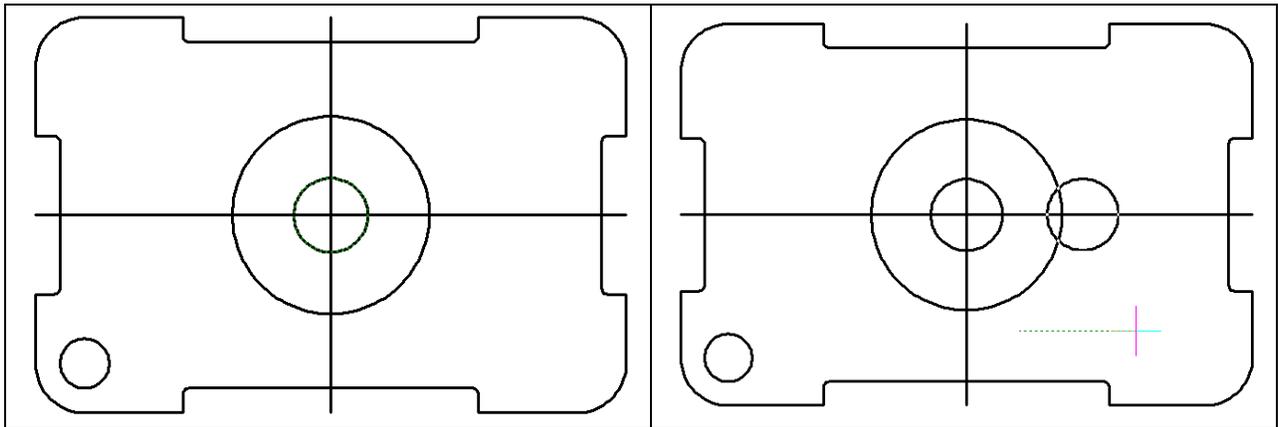


Figure 2.33 – Selecting the circle to move

Figure 2.34 – The direction of movement

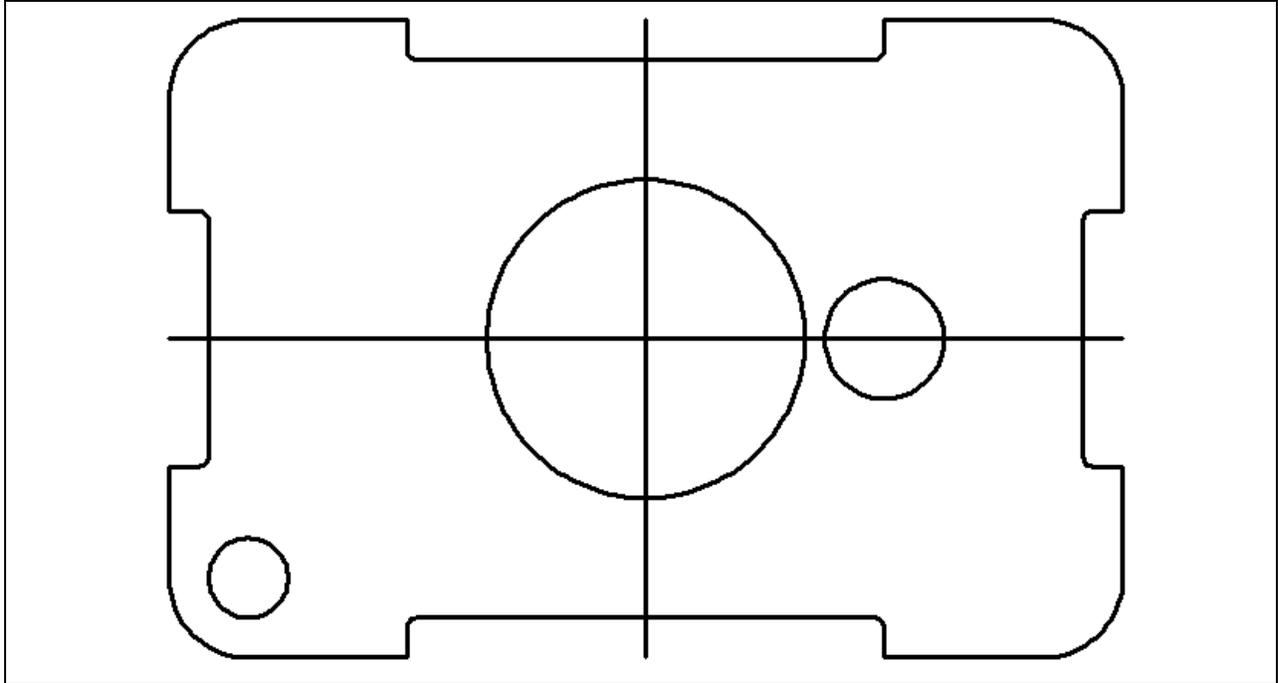


Figure 2.35 – Your first movement of an entity completed

Using the Copy Command

To copy an progeCAD entity, such as the same circle you just moved, select the Copy command on the Modify toolbar. The command line will prompt you to “Select entities to copy:” (Figure 2.36). You can do this by selecting the circle with the pick box that now has replaced the aperture cursor (Figure 2.37). The command line will respond with “Entities in set: 1.” Press ENTER to go the second part of the command. Now you need to specify a base point on the graphical display. Pick your base point away from any entities so that your Esnaps do not interfere with the movement. Figure 2.38 shows a base point selection to the lower right hand corner of the circles. After you pick a base point with the left mouse button, drag the circle to the left as shown in Figure 2.38 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 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 2.39)



```
Command : _COPY
Select entities to copy:
Entities in set: 1
Select entities to copy:
Vector/mOde/<Base point>:
Displacement point: 3
```

Figure 2.36 – The command line sequence for your first Copy command

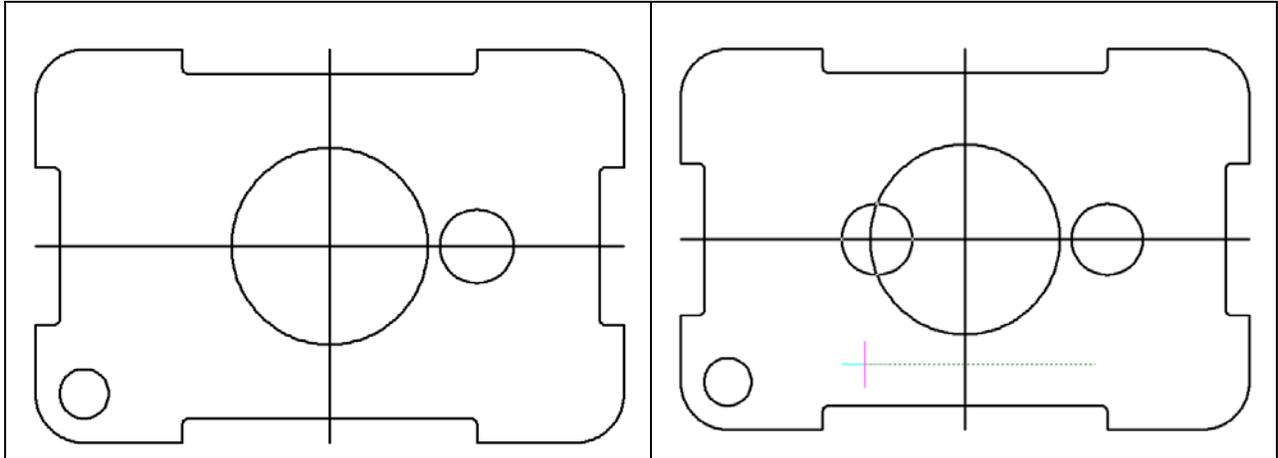


Figure 2.37 – Selecting the circle to copy

Figure 2.38 - 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 progeCAD, these are more powerful than the original Line or Circle commands when creating new entities in your drawings.

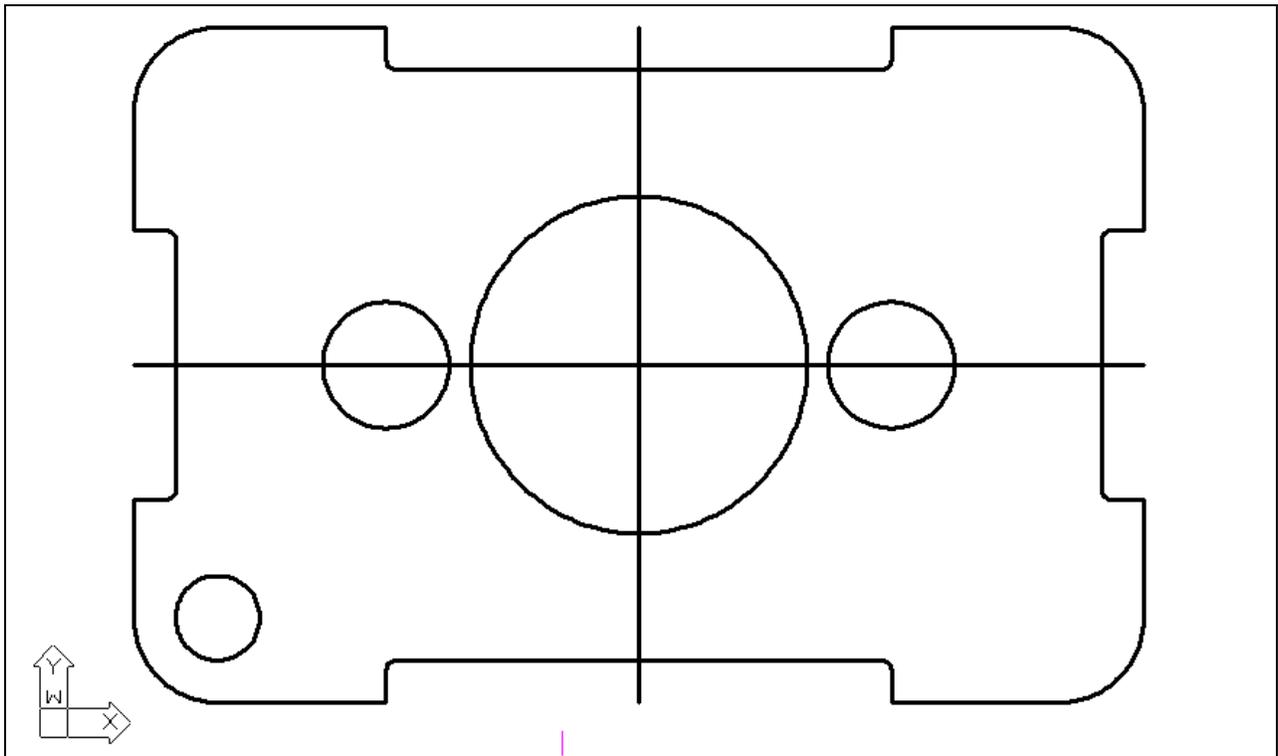


Figure 2.39 - The Copy command completed, resulting in two circles

*** World Class CAD Challenge 100-05 * - 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 Esnap

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 toolbar. Each circle is broken into four quadrants: north, south, east, and west. Place the aperture cursor over the north quadrant of the left circle and the Quadrant Esnap (orange diamond) will appear (Figure 2.40). Select the quadrant with the left button of the mouse and then select the north quadrant of the right circle (Figure 2.41). Press ENTER to leave the command. You now have a single line segment connecting the north quadrants of both circles. Press 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 2.42.

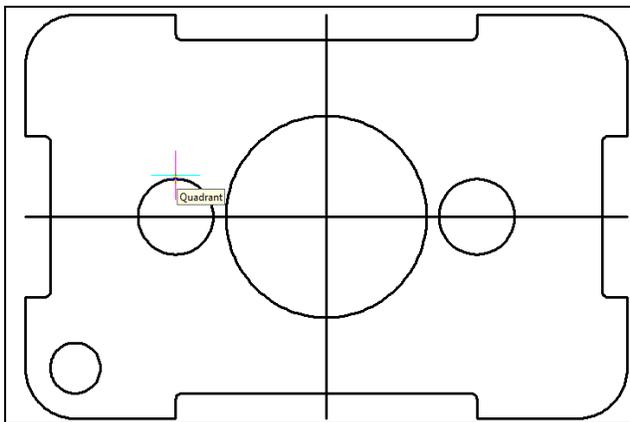


Figure 2.40 – Selecting the north quadrant

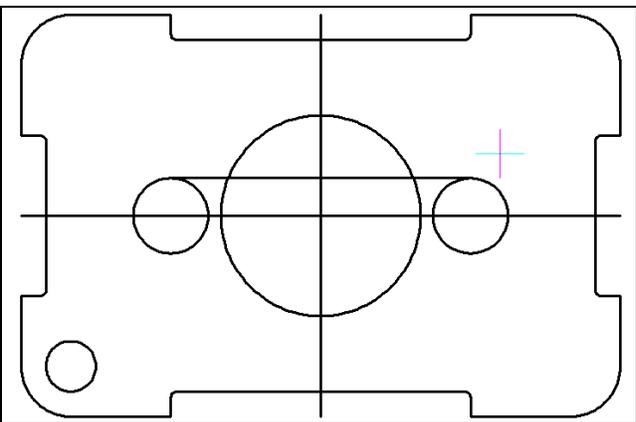


Figure 2.41 – Connecting the circles

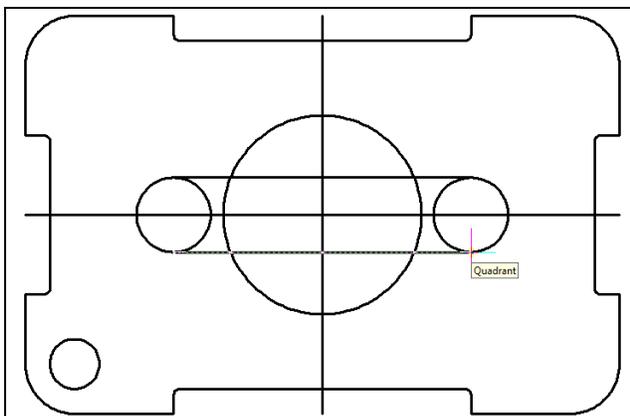


Figure 2.42 – Drawing the lower line

The Quadrant Esnap is a very powerful tool that creates four reference points to a circle, compared to the single reference point obtained by the Center Esnap. 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. World Class trained CAD operators, which include Architects, Designers, and Engineers can put their calculators away.

Reinforcing using 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 toolbar and place a window around the interior detail of three circles and two lines. Select 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 2.43). The selected entities will be dotted, as in Figure 2.44, and every edge of this detail will be a cutting line once you press ENTER. Proceed to pick each line segment that you would like to remove (Figure 2.45). The 6 red dots in Figure 2.45 show the line or arc segments to remove. Finally, our last complex detail is complete (Figure 2.46).

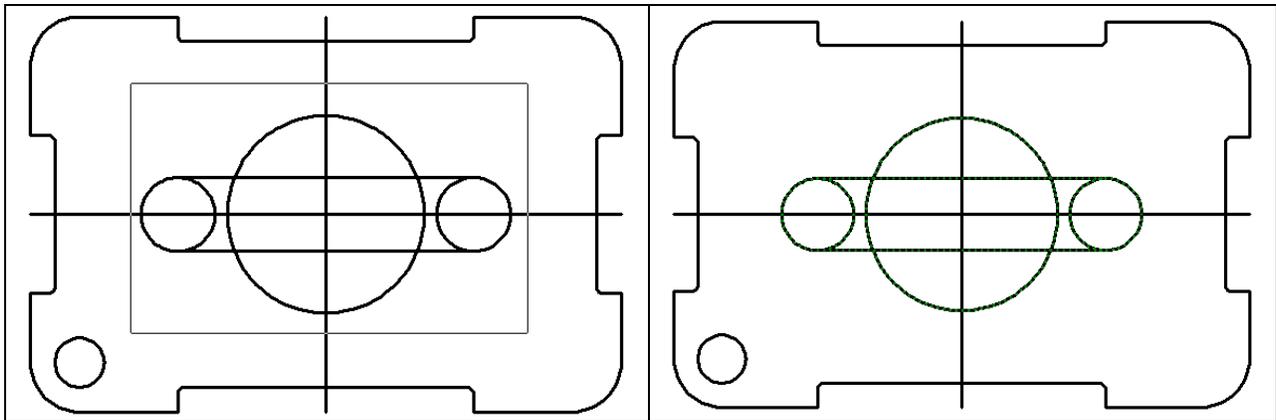


Figure 2.43 – Selecting by windowing

Figure 2.44 – The selected objects

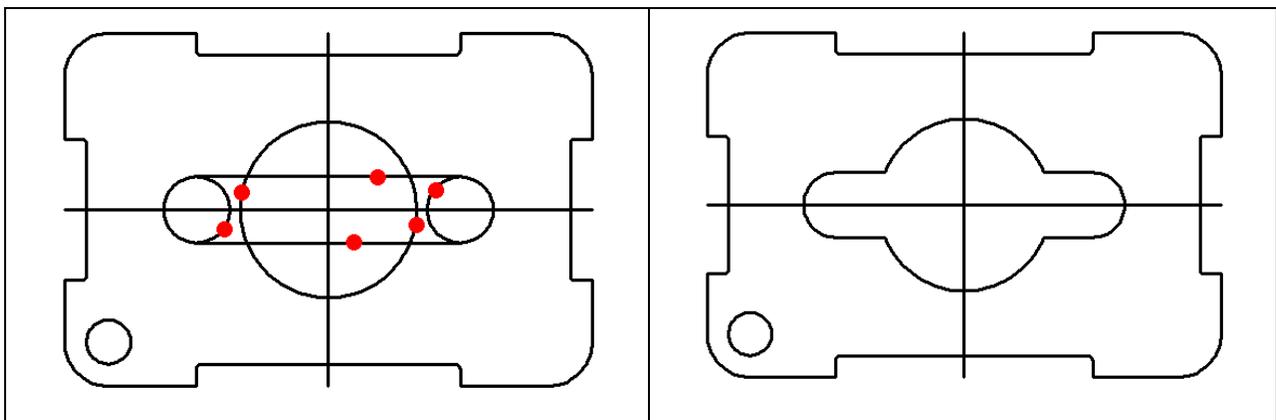


Figure 2.45 – 6 points to trim

Figure 2.46 – Finished with the trimming

Using the Rotate Command

The Rotate command will allow you to rotate entities around a base or center point. Select the Rotate command on the Modify toolbar. 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 2.47). The selected entities will be dotted as in Figure 2.48. To continue with the command, press ENTER. Select the Midpoint of the centerline with the left mouse button (Figure 2.49). Then type 45 for the degrees of counter clockwise rotation to finish this detail (Figure 2.50).

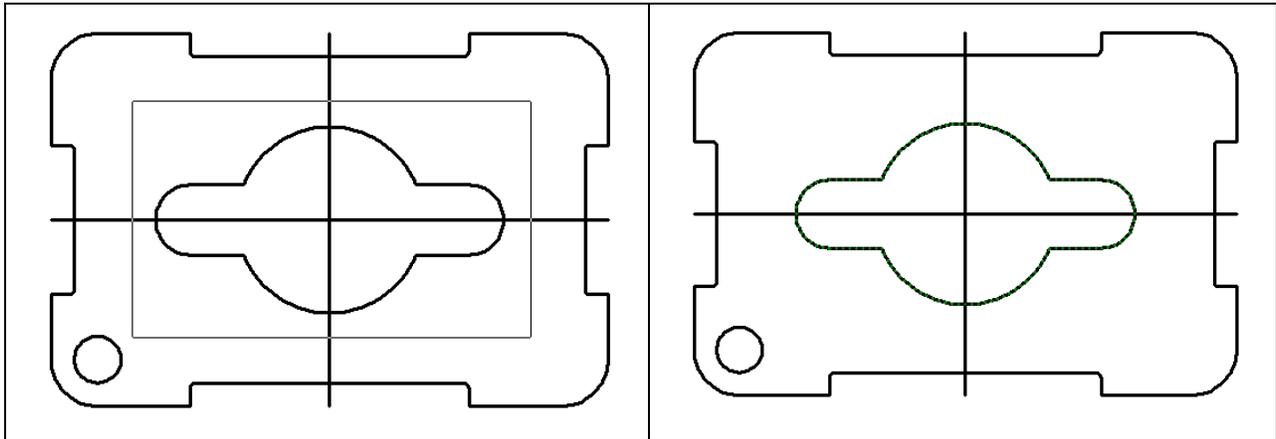


Figure 2.47 – Selecting by windowing

Figure 2.48 – The selected detail

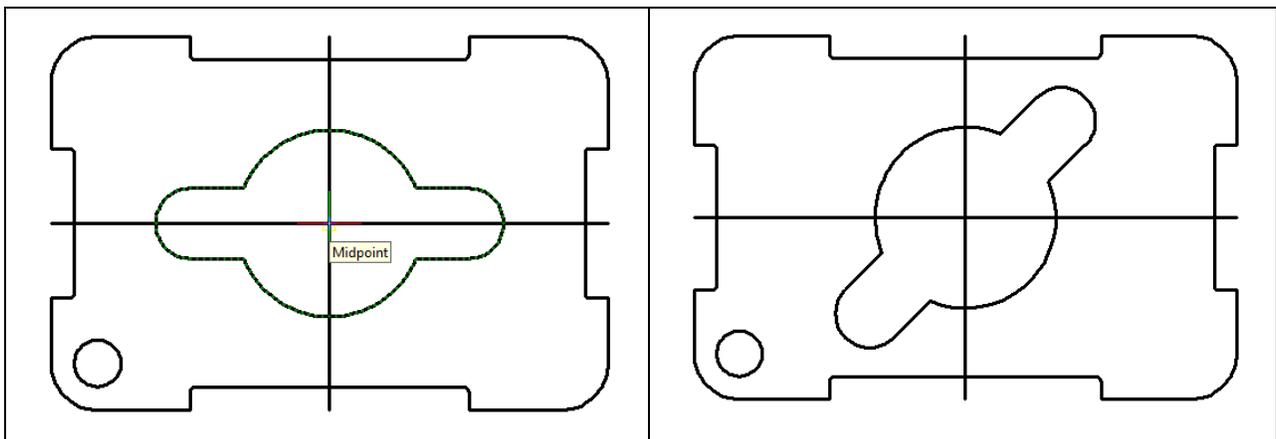


Figure 2.49 – Selecting the midpoint

Figure 2.50 – A 45 degree rotation

*** World Class CAD Challenge 100-06 * - 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 toolbar. The Array window will appear in the graphical screen area of your display. Stay on the Rectangular Array setting and type 2, each in the Rows and Columns textboxes. Type 3 in the Rows distance textbox and 5 in the Columns distance textbox (Figure 2.51). Select the “0 objects selected” button in the Array Window. The Array window will disappear and you will be able to select objects on the drawing.

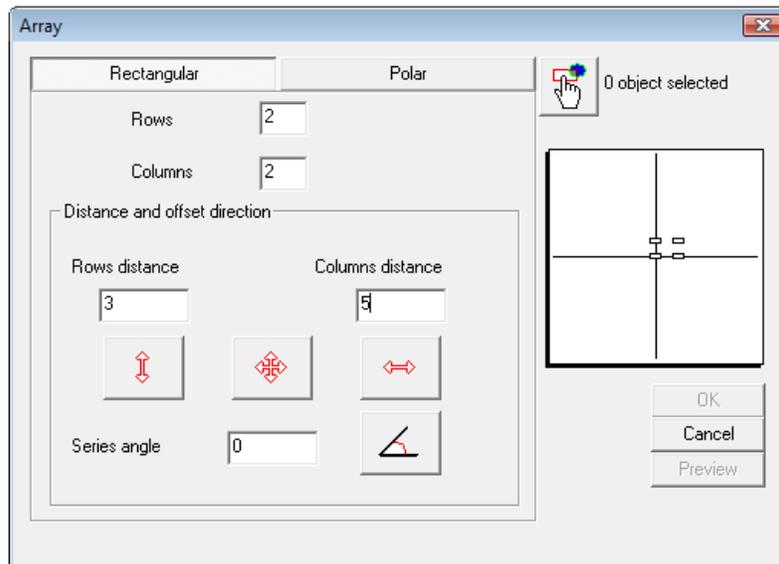
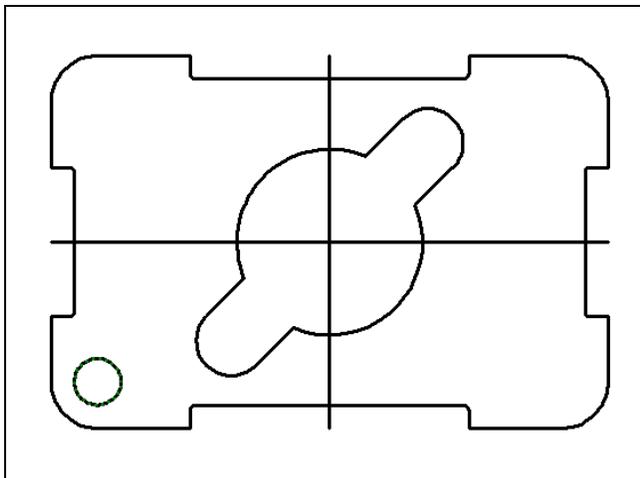


Figure 2.51 – The Array Window



Place the pick box over the 0.5 circle and press the left mouse button to select the object (Figure 2.52). 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 2.53. The typical CAD student needs 10 to 12 hours of training and practice to achieve World Class standards.

Figure 2.52 – Selecting the 0.5 circle

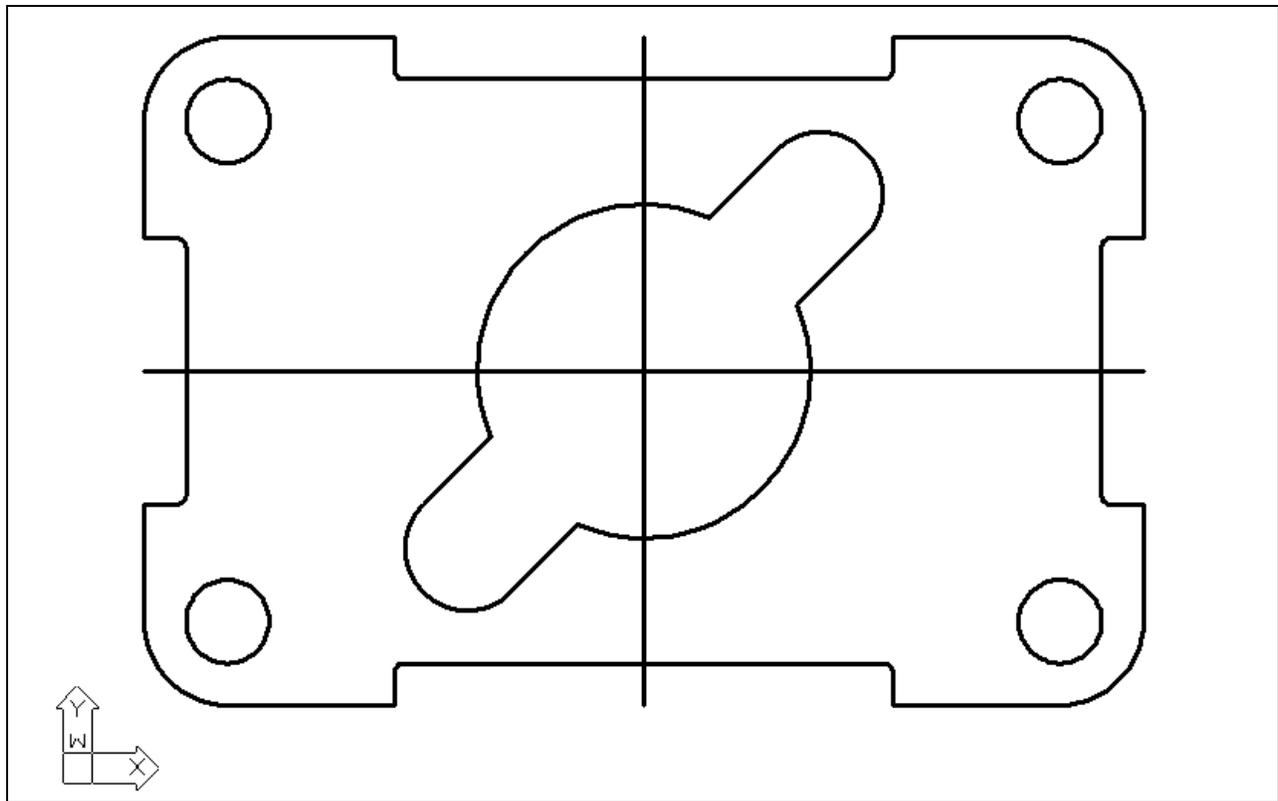


Figure 2.53 – The finished 2D part prior to dimensioning

*** World Class CAD Challenge 100-07 * - 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.**