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1.7 - POWERFUL MODIFYING COMMANDS





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Topics covered in this Lesson:

Rotate, Fillet, Chamfer, Array

Now it's time to learn a few more commands. Like all of the commands learnt so far, these too will be ones that you will use regularly.

Here are the commands that you will be learning in this lesson.

Command	Keystroke	Icon	Menu	Result
Rotate	Rotate / RO		<u>M</u> odify > <u>R</u> otate	Rotates objects to a certain angle
Fillet	Fillet / F		<u>M</u> odify > <u>F</u> illet	Creates a round corner between two lines
Chamfer	Chamfer / CHA		<u>M</u> odify > <u>C</u> hamfer	Creates an angled corner between two lines
Array	Array / AR		<u>M</u> odify > <u>A</u> rray	Creates a repeating pattern of the selected objects

Once again you will recreate a drawing. This one is called Assignment #5.


Click [here](#) to see the GIF format file.

Click [here](#) for the DWG file.

Follow the steps shown carefully. As these commands require a little more input, make sure that you **keep an eye on the command line**. You will be asked to provide information throughout the commands.

Start up AutoCAD and [load the acad.dwt](#) template like you have for the other lessons.

Start by drawing a horizontal 10" X 7" border with the bottom left corner at **0,0**

Draw  a rectangle 1" wide by 3" tall with the bottom left corner at **.75, .75**

You are now going to rotate this rectangle 90° clockwise.

Start the **ROTATE** command. AutoCAD asks you to select objects. Select all parts of the rectangle and press <ENTER>. Now you must indicate a '**base point**'. Think of this as a pivot point around which the rectangle will rotate. In this example, you want to select the bottom right corner (*remember to use your Osnap*). Once you've selected the base point, the command line shows **rotation angle or [Reference]**: This means that 'Rotation angle' is the default, so type in the angle you want to rotate the object. Think about how [AutoCAD measures angles](#). Looking at your rectangle and the one on the assignment sheet, you'll see that you want to rotate the rectangle clockwise or: **-90** degrees. Enter that number and press <ENTER>.

Command: **RO** <ENTER>

Current positive angle in UCS: ANGDIR=counterclockwise ANGBASE=0

Select objects: <Select the Rectangle> 1 found

Select objects: <ENTER>

Specify base point: <PICK BOTTOM RIGHT CORNER OF THE RECTANGLE>

Specify rotation angle or [Reference]: **-90** <ENTER>

The rectangle is now been rotated -90 degrees from its original position. Picking different base points will give you different results. **Undo** the last command. Try a few different combinations of base points and angles to see what results you get. When you are done practicing, get the rectangle back to the position it was at the end of previous step.

Make a **COPY** of the rectangle 2" above the first one (remember your relative co-ordinates).

Now you're going to modify the second rectangle so that it has rounded corners. Start the **FILLET** command. Look at the command line. It will look something like this:


Command: **F** <ENTER> **FILLET**

Current settings: Mode = TRIM, Radius = 0.0000

Select first object or [Undo/Polyline/Radius/Trim/Multiple]:

AutoCAD first shows you what the current fillet radius is (0.0000). This will be the last value that was used. Once it's changed, it will keep the new value in memory. The next line shows you what options you have in this command. Remember that the **Capitol** of each option selects that particular option. What you want to do is change the fillet radius to 3/8" (or .375). To do this you have to type **R** <ENTER>. When you type this AutoCAD will give the chance to enter a new fillet radius. At this point enter **.375** and press <ENTER>.

The fillet radius is now .375 (which is what you want). The default option is **Select first object**. Select the left side of the top rectangle (yes, the whole rectangle will highlight if you drew it as a rectangle). AutoCAD now asks you to select second object. Select the top line and AutoCAD will make a smooth round corner with a radius of .375. AutoCAD automatically ends the command at this point.

Restart the **FILLET** command  and do this to the remaining corners so that you have an object similar to the example.

NOTE: The Fillet command is commonly used with a Zero radius. This can sometimes be much quicker than trimming two lines that meet at endpoints. To practice this, fillet this lines you made with round corners using a Zero radius.

Copy the first rectangle to a point 4-1/2" above. Now you will use the chamfer command to give this rectangle sharp, angled corners.

Start the **CHAMFER** command. Look at the command line. It should look like this:

```
Command: CHA <ENTER> CHAMFER
(TRIM mode) Current chamfer Dist1 = 0.0000, Dist2 = 0.0000
Select first line or [Polyline/Distance/Angle/Trim/Method]: D <ENTER>
Specify first chamfer distance <0.5000>: .375 <ENTER>
Specify second chamfer distance <0.3750>: <ENTER>
Select first line or [Undo/Polyline/Distance/Angle/Trim/mEthod/Multiple]: <select
one side of the rectangle>
```

This is very similar to the fillet command. You have several options available. What you want is an even 45 degree angle 3/8" in from the corner. Like the fillet command, you first have to tell AutoCAD what distance you want. To do this, type **D** to select the Distance option. The command line now looks like this:

```
Specify first chamfer distance <0.5000>: .375 <ENTER> as your first distance.) The
command line now asks for the second distance. AutoCAD will automatically change the default of the
second distance to match the distance you entered for the first.
```

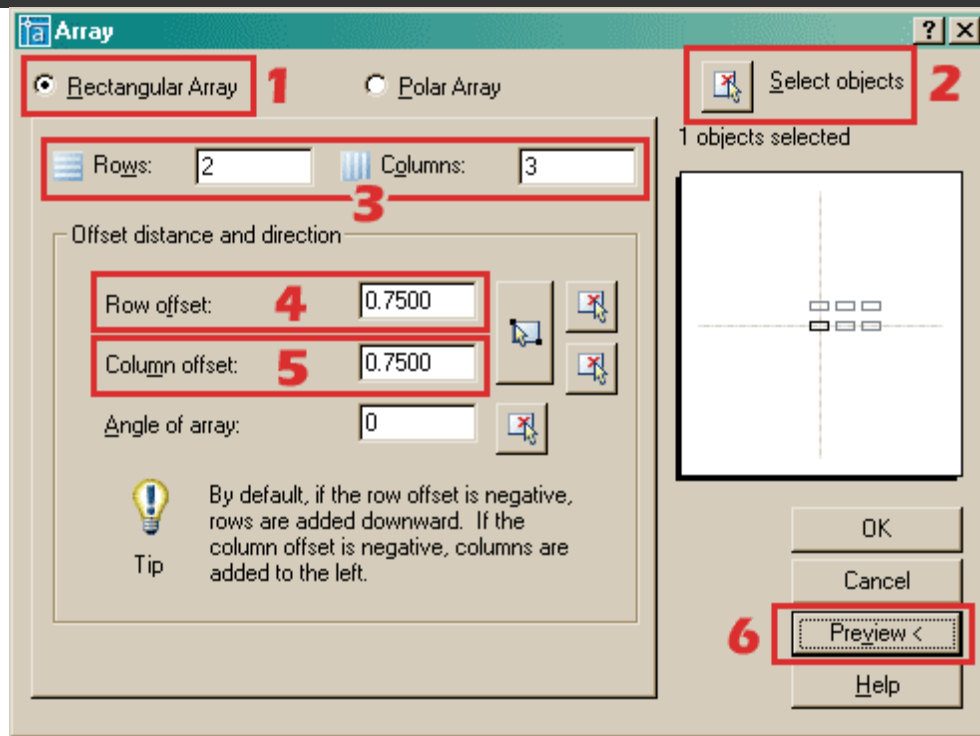
```
Specify second chamfer distance <0.3750>: (Press <ENTER> to accept this)
```

You will then be asked to **Select first line**. The chamfer command works just like the fillet command. Select the line on the left of the top rectangle. (Don't worry if the entire rectangle highlights.) When prompted to **Select second line**: select the top line. You will now have a perfect sharp corner at a 45 degree angle 3/8" in from the corner. Do this to the rest of the corners.

Now look at the assignment sheet and notice the group of six rectangles on the bottom right. You could draw each one individually, but AutoCAD has a command that will allow you to draw one, and it will make the others.

Create a rectangle that is 1/2" square with the bottom left corner at **6,1.5** (absolute points).

Start the **ARRAY** command. Look at the dialog box shown below:



When confronting a new dialog box, I recommend that you look for what is needed from the **TOP DOWN** to the bottom. This is a great example.

1. Choose the radio button for "Rectangular Array". This will array the object in a row/column arrangement.
2. Next select the object you want to array, by picking on the button in the top right corner. (Press enter when done)
3. Enter the number of rows (going across the page) and column (running up and down the page).
4. Enter the Row offset. This is this from the bottom left of the original rectangle, to the bottom left of where the first copy will go.
5. Enter the Column offset
6. Pick the Preview button to see the array before committing.

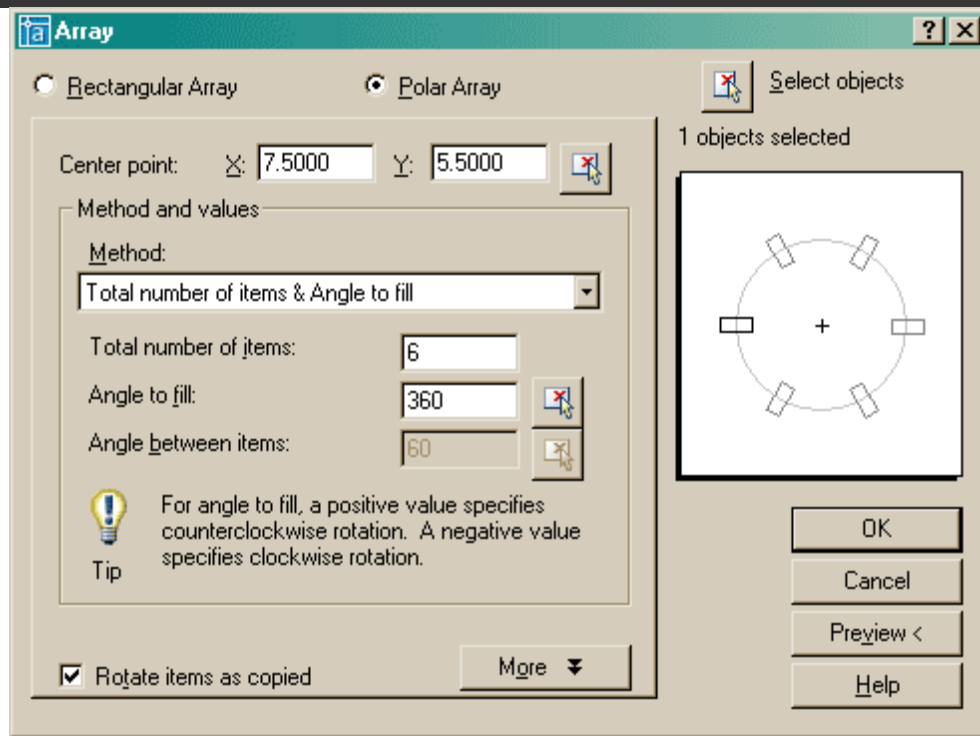


If the array is correct (check the sample drawing), press the "Accept" button. If you need to change anything, press the "Modify" button, make your changes in the dialog box and preview again.

Now you are going to use the **ARRAY** (polar) command to create the shape in the top right corner of the assignment.

Start by making a **CIRCLE** with a center point of **7.5, 5.5** and a diameter of **1.5** Next make a **LINE** from the center of the circle going 1" to the right (remember your relative input and Osnaps).

Start the **ARRAY** command. When asked to select objects, pick the line you just drew.



Examine the dialog box above. Remember to start from the TOP. In this case, you have to select your objects and select a Center Point for the array. (Select the center of the circle.)

NOTE: Sometimes the Array command can be quicker than the offset command. Think of creating lines for a ceiling grid. You 'could' offset 30 lines one at a time, or you 'should' use the array command to create all 30 lines at once.

Save and [print](#) your drawing.

 [View the video for this assignment .](#)

So far in seven lessons, you have learned many of the common commands in AutoCAD. It may not seem like a lot, but the idea is to become fluent in them. It should be second nature to run these commands, as these are the ones you will be using most often. With practice, you won't the command line to help you along. Think of the first time you drove a car with a manual transmission. After time, it got smoother, didn't it. I can't stress enough how important practice is during these early lessons.

Extra Practice: Copy this drawing - [extra_009.gif](#)

 [View the video for extra_009.](#)

Extra Practice: Copy this drawing - [extra_010.gif](#)

 [View the video for extra_010.](#)

Extra Practice: Copy this drawing using 2 methods. First create the lines using the offset command, then create the lines using the Array command. Compare which method should be easier, and when you might use one over the other - [extra_012.gif](#)

 [View the video for extra_012.](#)

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