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1.3 - MODIFYING COMMANDS AND OSNAPS



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Topics covered in this Lesson:

Rectangle | Multi Line | Extend | Offset | Trim | [Introduction to Object Snaps](#)

The previous lesson dealt with drawing commands. This lesson will introduce the common modifying commands. In AutoCAD, you may actually use modifying commands more often than drawing commands. Now that you know that basics, here's some more commands to add to your collection. Three commands, Trim, Extend and Offset are used standard AutoCAD work.

Command	Keystroke	Icon	Menu	Result
Rectangle	RECTANGLE / REC		<u>D</u> raw > Rectangle	Draws a rectangle after you enter one corner and then the second.
Multi Lines	MLINE / ML	No Icon	<u>D</u> raw > <u>M</u> ultiline	Draw parallel lines based on the parameters you define.
Trim	TRIM / TR		<u>M</u> odify > <u>T</u> rim	Trims objects to a selected cutting edge.
Extend	EXTEND / EX		<u>M</u> odify > <u>E</u> xtend	Extends objects to a selected boundary edge.

Offset	OFFSET / O		<u>M</u> odify > Off <u>s</u> et	Offsets an object (parallel) by a set distance.
Object Snaps	OSNAP / OS / F3	CLICK 	<u>T</u> ools > Object S <u>n</u> ap Settings	Brings up the OSNAP dialog box.

Assignment #2 - Modifying Commands

The purpose of this assignment is to use the commands learned in the previous lesson and learn some new ones.

Duplicate the drawing called [Assign 2](#).

Click [HERE](#) to download the DWG file.

Once again, do not worry about title blocks, text or dimensions, draw only what is in yellow.

Start AutoCAD and begin the the drawing by opening up the template file like you did in [Lesson 1-2](#).

Draw a **LINE** from **1,2** to **3,2** to **3,4** to **1,4** (*Remember to watch the command line as you do this.) For the last line, you can either type in **1,2** or **C** to close the line back to the first point you entered. These are **absolute co-ordinates**. Make sure you understand what the points your just entered represent.

Draw the next square using the **RECTANGLE** command. A rectangle is created by specifying 2 points to represent the opposite corners. Enter the first point as **4.5,2** and then make the opposite corner 2 inches over and 2 inches up **@2,2** using **relative co-ordinates**. This is much faster and also makes the square **one object** and not 4 separate lines.

ERASE the rectangle. You will see that all of it is gone with **one pick**. Redraw it and continue.

Draw the third box using the **MULTILINE** command. This box is 1-1/2" square. The following steps are what you will see on the command line. Before drawing the lines, you have to set the SCALE to 0.15"(distance between the lines) by typing 'S'. Then set the JUSTIFICATION (offset origin) by typing 'J'.

Command: **ml** <enter>

MLINE

Current settings: Justification = Top, Scale = 1.00, Style = STANDARD

Specify start point or [Justification/Scale/STyle]: **S** <enter>

Enter mline scale <1.00>: **.15**

Current settings: Justification = Top, Scale = 0.15, Style = STANDARD

Specify start point or [Justification/Scale/STyle]: **8,2** <enter>

Specify next point: **@1.5,0** <enter>

Specify next point or [Undo]: **@0,1.5** <enter>

Specify next point or [Close/Undo]: **@-1.5,0** <enter>

Specify next point or [Close/Undo]: **@0,-1.5** <enter>

Specify next point or [Close/Undo]: <enter>

To review what you just did, you started the command, then set the scale (distance between the lines) to **.15** units. Then you just drew the rectangle using relative co-ordinates.

Erase the multiline rectangle you just drew. Notice that when you hover over the rectangle, it will turn to a dotted line. This means that it is highlighted as the object that will be selected if you pick at that moment. In highly detailed drawings, this is very handy.

Draw it again as shown below and note the subtle difference from the first one you drew.

Command: **ML** <enter>

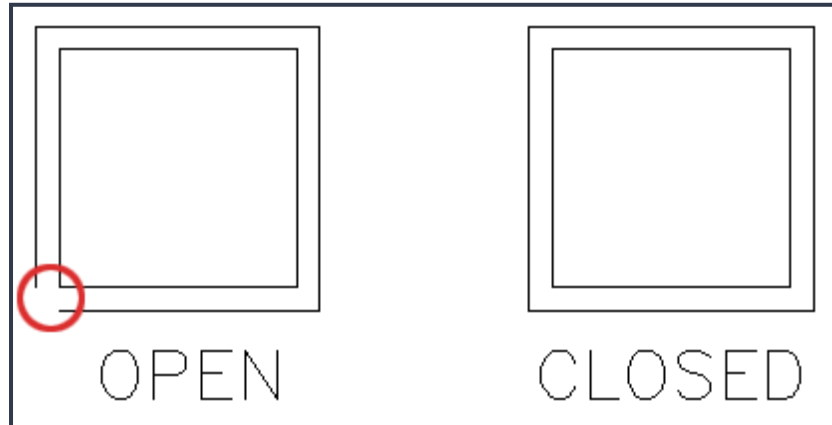
MLINE

```

Current settings: Justification = Top, Scale = 1.00, Style = STANDARD
Specify start point or [Justification/Scale/STyle]: S <enter>
Enter mline scale <1.00>: .15
Current settings: Justification = Top, Scale = 0.15, Style = STANDARD
Specify start point or [Justification/Scale/STyle]: 8,2 <enter>
Specify next point: @1.5,0 <enter>
Specify next point or [Undo]: @0,1.5 <enter>
Specify next point or [Close/Undo]: @-1.5,0 <enter>
Specify next point or [Close/Undo]: C <enter>

```

The difference is that instead of drawing to the last point, you used the C (close) option to complete the rectangle. The difference is shown in the drawing below.



This is an example of why you have to look at the command line as you work. As soon as you start the command, you have choices available. Whenever you see this come up, if you want to change anything, you just type the **CAPITAL** letter of the option. For example, if you want to undo the last point, you would type **u** at this point. For this assignment you only changed the scale. After you have completed the assignment, try different settings for this command. Use your AutoCAD **help** option to see what these options control. (NOTE: I don't normally use Multilines while drawing as I don't have complete control over both the inner and outer lines. On the other hand, Multilines can be great for drawing quick offset lines. Also, they cannot be trimmed (you have to explode them first).

Draw a line from **2,5** to **2,6.5** Draw another line from **1,6** to **3,6** You should now have two perpendicular lines. What you want to do is trim off the top of the vertical line and create a T.

Start the **TRIM** command. It will first ask for a cutting edge. Select the horizontal line and press <ENTER>. It will now ask for the object to be trimmed. Select the vertical line anywhere above the horizontal (cutting) line and press <ENTER> to finish the command.

This is what you saw on the command line:

```

Command: TR <enter> TRIM
Current settings: Projection=UCS, Edge=None
Select cutting edges ...
Select objects: <Select the Horizontal line>1 found
Select objects: <enter>
Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/
eRase/Undo]: <Select the vertical line>
Select object to trim or shift-select to extend or [Project/Edge/Undo]: <enter>

```

Once again, it is important to keep your eye on the command line as it will guide through most commands.

Draw a **LINE** from **4,6.5** to **6,6.5** Draw another line from **5,5** to **5,6** What you want to do now is

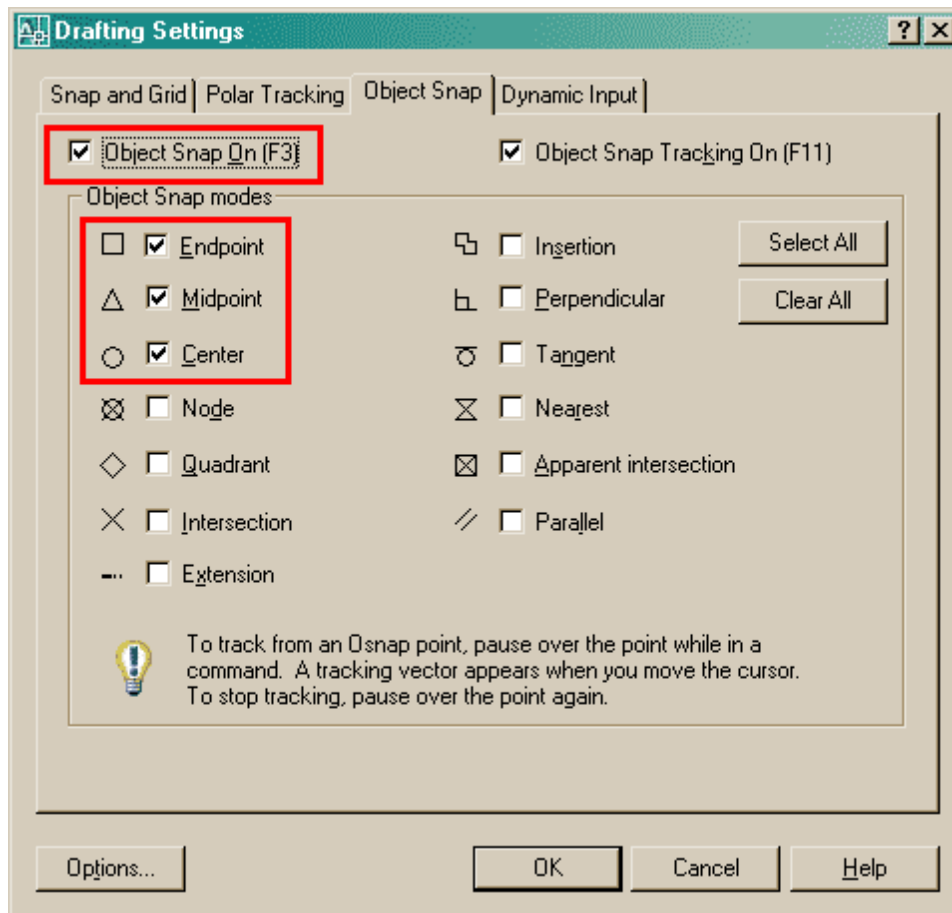
extend the vertical line up to the meet horizontal line. Start the **EXTEND** command. AutoCAD asks for a boundary edge; select the horizontal line press <ENTER>. It then asks for an object to extend; select somewhere in the top half of the vertical line. Press <ENTER> to end the command. Your command line history should match what is printed below.

```
Command: EX <enter> EXTEND
Current settings: Projection=UCS, Edge=None
Select boundary edges ...
Select objects: <Select the horizontal line> 1 found
Select objects: <enter>
Select object to extend or shift-select to trim or [Fence/Crossing/Project/Edge/Undo]: <Select the top half of the vertical line>
Select object to extend or shift-select to trim or [Project/Edge/Undo]: <enter>
```

Draw a **CIRCLE** with a center point of **7.5,5.5** with a radius of **.5** Now you will use to offset command to make another circle 1/4" larger. Start the **OFFSET** command (watch the command line) and enter **.125** as the offset distance. Now select the circle and pick anywhere **outside** the circle. Press <ENTER> to end the command.


















Object Snaps

Suppose you want to draw a line from the center of the circle to the middle of the vertical line you extended earlier. AutoCAD has a feature that makes this very easy. These are the Object Snaps (or Osnaps "Oh-Snaps"). Type **OS** <ENTER> . You will see this dialog box appear.



You may also see a toolbar with these Osnaps as shown below.



ICON	SETTING		ICON	SETTING
	Endpoint			Perpendicular
	Midpoint			Tangent
	Center			Nearest
	Node			Apparent Intersection
	Quadrant			Parallel
	Intersection			None
	Extension			Osnap Settings
	Insertion Point			Temporary Tracking Point
M2P	Midpoint between 2 points			Snap From

You may select whichever points you want to 'snap' on an object. Here is a list of your options. Followed by the command entry to invoke the needed Osnap.

Endpoint - snaps to either the beginning or the end of an object such as a line - [END](#)

Midpoint - snaps to the exact middle of a line or an arc - [MID](#)

Center - snaps to the center-point of a circle or arc - [CEN](#)

Node - snaps to 'nodes' (not covered in this course) - [NOD](#)

Quadrant - snaps to any of the four quadrants of a circle - [QUA](#)

Intersection - snaps to the point where two object cross - [INT](#)

Extension - Snaps to the phantom extension of an arc or line - [EXT](#)

Insertion - snaps to the insertion point of an object (such as a block or text) - [INS](#)

Perpendicular - will snap so that the result is perpendicular to line selected - [PER](#)

Tangent - snaps to create a line tangent to a circle or arc - [TAN](#)

Nearest - will find the closest point an object and snap to that point - [NEA](#)

Parallel -Snaps parallel to a specified line - [PAR](#)

None - temporarily turns off all Osnaps. (Pressing your F3 Key is quicker) - **NON**

Osnap settings - opens the Osnap dialog box.

Temporary Tracking - Creates a temporary tracking point (see Object Tracking).

From - Allows you to select a point, then denote a new location 'from' that point using relative co-ordinates. This can save you the time of drawing (and erasing) construction lines.

M2P - This isn't technically an 'Object Snap' as you are not snapping to an object, but it allows you to select 2 points and it will calculate the midpoint between those 2 points. This is a very handy option to have.

Note: Beside each checkbox is a symbol. That symbol will show up on the screen when you have found a valid snap point. (An endpoint will show a small square). If you select the "Options" button, you can change the aperture size and the color of the Osnaps. Depending upon the background you are drawing on, this may be needed.

- a. Check off the boxes as shown in the dialog box above (Object Snaps On, Endpoint, Midpoint, Center) and press **OK**.
- b. Begin the **L**INE command. Move your cursor around the screen and you'll see that as you get close to an object, it will 'snap' to one of the points that you had checked off in the dialog box. Place your cursor on the circle (not the middle of the circle) until you see a small purple circle appear at its center. Left-click to make this the start point of the line. Move the cursor towards the middle of the vertical line until you see a small triangle appear. (*Remember this is the symbol for 'midpoint'*). When you see it left-click to accept this as your endpoint. Press <ENTER> to end the line command.
- c. **Save** your drawing.
- d. **Print** your drawing with the same settings as in [Assignment #1](#).



[View the video for Assignment 2.](#)

TIP: Before you select the Osnap you want, you can press the TAB key on your keyboard to cycle through the available Osnaps in the area of your cursor.

CAUTION: Although it may seem tempting to turn 'all' the Osnaps on when drawing, you can have too much of a good thing. For example, in shorter lines, Midpoint, Nearest and Perpendicular could all be very close to each other, and you could select the wrong point.

When you have finished the assignment, continue practicing with the commands until they are mastered. These are common commands that you will use in everyday drafting.

Extra Practice: Copy this drawing, using lines, mlines, offset, osnaps - [extra_003.gif](#)



[View the video for Exercise 3.](#)

Extra Practice: Copy this drawing, it could a little to figure out, but still uses the commands you have learned so far. - [extra_004.gif](#)



[View the video for Exercise 4.](#)

Extra Practice: Have a look at [this drawing](#) of a simple reflected ceiling plan (RCF) and see how the offset, trim and extend commands might be used. If you know how to change units to Architectural, try drawing it.



[View the video for Bonus 1.](#)



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