

Modifying Objects

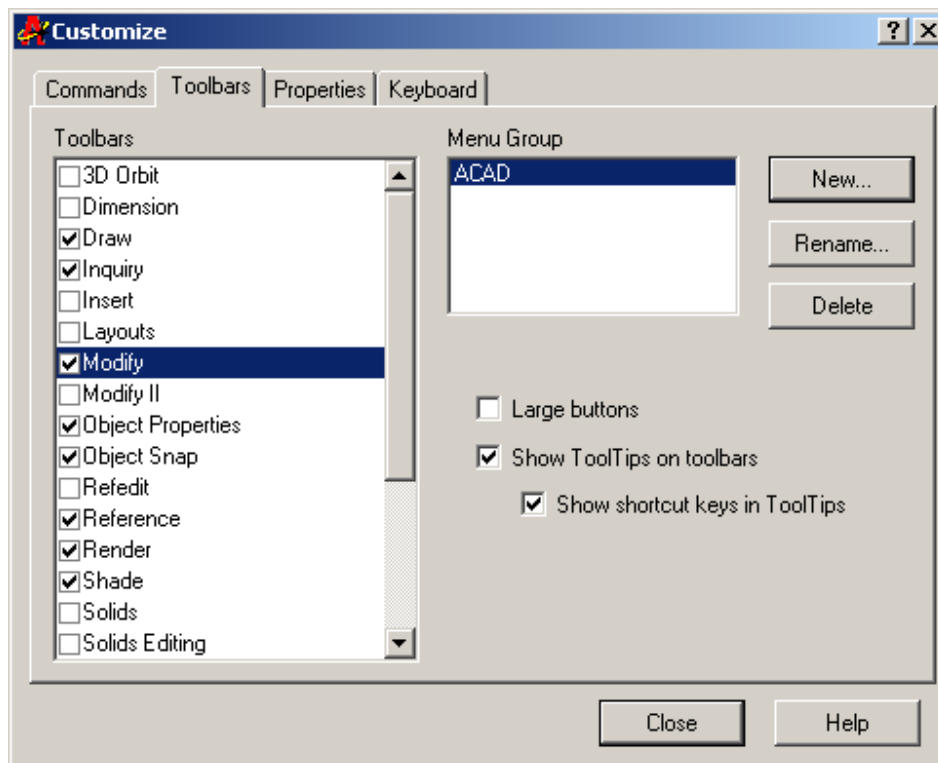
by David Watson

Introduction

AutoCAD drawings are rarely completed simply by drawing lines, circles etc. Most likely you will need to *Modify* these basic drawing objects in some way in order to create the image you need. AutoCAD provides a whole range of modify tools such as Move, Copy, Rotate and Mirror. As you can see, the command names are easily understandable. However, the way these commands work is not always obvious. This tutorial is designed to show you how all of the Modify commands work. If you just need information quickly, use the QuickFind toolbar below to go straight to the information you need or select a topic from the contents list above.



As is usual with AutoCAD, the Modify tools can be accessed in one of three ways, from the keyboard, from the pull-down menu and from the toolbar. All of the Modify tools are available from the Modify pull-down and the Modify toolbar. In each section below, the toolbar, pull-down and keyboard options are given. The method you choose is entirely up to you. Ultimately you will use the method that you feel most comfortable with or the one you find most efficient. AutoCAD allows great flexibility and there aren't any right or wrong ways of working. That said, it should be pointed out that the use of toolbars in AutoCAD is almost always quicker than any other method.




The Modify toolbar is usually displayed by default but if it is not already displayed, you can display it using the

TOOLBAR command, **View ► Toolbars...**


from the pull-down menu. When the Toolbar dialogue box (shown above) appears, simply check the box next to "Modify" in the toolbars list. Many AutoCAD users work with the Modify toolbar permanently *docked* on their screen because it gives one-click access to all of the commands, making the drawing process much more efficient.

The Erase Command

Toolbar Modify 

Pull-down **Modify ► Erase**


Keyboard **ERASE** short-cut **E**


The Erase command is one of the simplest AutoCAD commands and is one of the most used. The command erases (deletes) any selected object(s) from the drawing. Remember you can always get deleted objects back by typing **U** to undo,  from the Standard toolbar or by using the OOPS command.

Command Sequence

Command: **ERASE**

Select objects: (pick an object to erase)

Select objects:  (to end the selection and erase the object)

If you simply want to erase the last object you created you can type **L** at the "Select objects" prompt. The last object will be highlighted and you can then select more objects or  to end the command. See the "Object Selection" tutorial for more information on selecting objects.

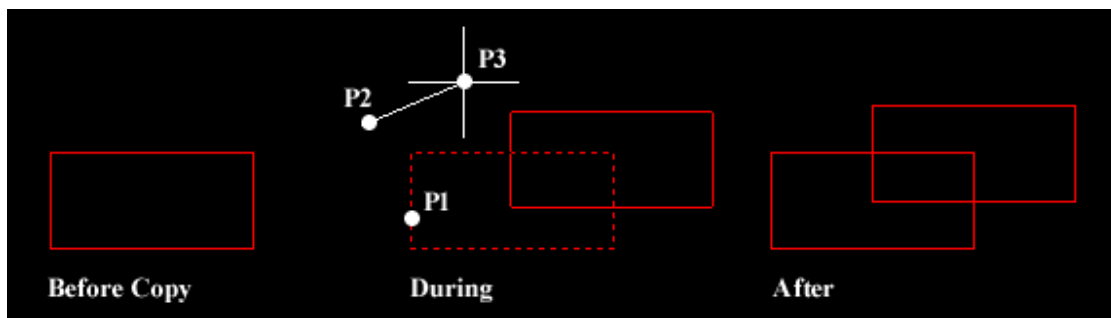
The Copy Command

Toolbar Modify 

Pull-down **Modify ► Copy**

Keyboard **COPY** short-cut **CP**


The Copy command can be used to create one or more duplicates of any drawing object or objects which you have previously created. Copy is a very useful and time-saving command because you can create very complex drawing elements and then simply copy them as many times as you like.



Command Sequence

Command: **COPY**

Select objects: (pick object to copy, P1)

Select objects:  (to end selection)


Specify base point or displacement, or [Multiple]: (pick P2 or M for multiple copies)

Specify second point of displacement or <use first point as displacement>: (pick P3)

The multiple option allows you to create additional copies of the selected object(s) by picking as many new points as you like. To end a multiple copy, just hit the **↵** key.

Notice that the "Base point", P2 and the "Second point", P3 do not have to be picked on or near the object. The two points are simply used to indicate the distance and direction of the copied object from the original object.

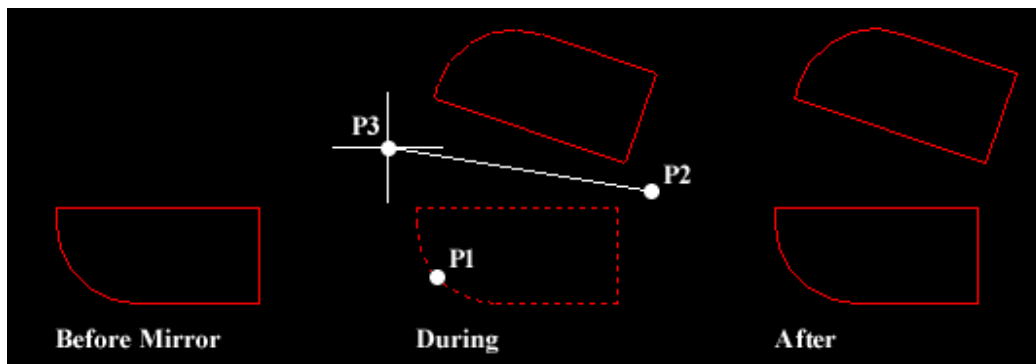
The Mirror Command

Toolbar Modify 

Pull-down **Modify** ► **Mirror**

Keyboard **MIRROR** short-cut **MI**

The Mirror command allows you to mirror selected objects in your drawing by picking them and then defining the position of an imaginary mirror line using two points.



Command Sequence

Command: **MIRROR**

Select objects: (pick object to mirror, P1)

Select objects: **↵** (to end selection)

Specify first point of mirror line: (pick P2)

Specify second point of mirror line: (pick P3)

Delete source objects? [Yes/No] <N>: **↵** (for No to keep the original object)

Notice that in the command sequence above, pressing the **↵** key when asked whether to "Delete source objects?" resulted in a "No" response. This is because "No" is the *default* option. AutoCAD always shows the default option within triangular brackets, in this case "<N>". Many AutoCAD commands offer default options. They are usually the most commonly used and you need only right-click and select "Enter" from the menu or press **↵** on the keyboard to accept them.

In order to create perfectly horizontal or vertical mirror lines use Ortho. For more information on the use of Ortho and other drawing aids, see the "Drawing Aids" tutorial.

The Offset Command

Toolbar Modify 

Pull-down **Modify ► Offset**Keyboard **OFFSET** short-cut **O**

Offset is probably one of the most useful commands for constructing drawings. The Offset command creates a new object parallel to or concentric with a selected object. The new object is drawn at a user defined distance (the offset) from the original and in a direction chosen by the user with a pick point. You can offset lines, arcs, circles, ellipses, 2D polylines, xlines, rays and planar splines.

Command Sequence

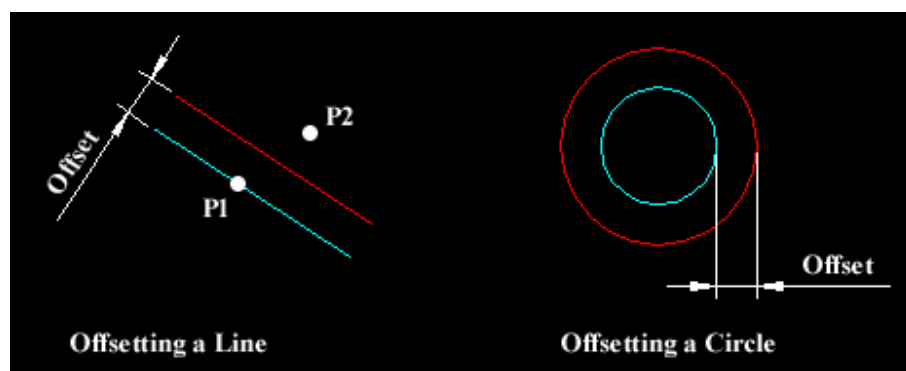
Command: **OFFSET**Specify offset distance or [Through] <1.0000>: **10** (specify distance)

Select object to offset or <exit>: (select object, P1)

Specify point on side to offset: (pick direction, P2)

Select object to offset or <exit>: **↵** (to end or select another object to offset)

In the illustration below, a line (cyan) has been offset to the right through a distance "Offset" by picking a point to the right of the original line. The result is a new line (red) to the right of the original.



Circles can be offset inside or outside of themselves to create a new circle which is concentric (has the same centre point) with the original circle. In the illustration, a new circle (red) has been created outside of the original by picking a point outside of the original circle. The radius of the new circle is the offset distance "Offset" plus the radius of the original circle.

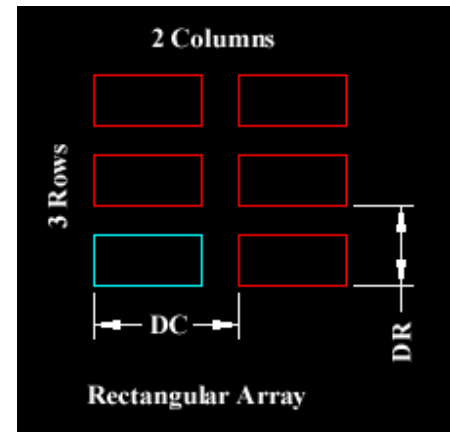
In the illustration above, the new objects and original objects are shown in different colours for clarity. In reality, offset objects inherit their *object properties* from the original object. See the "Object Properties" tutorial for details.

The Array Command

Toolbar **Modify** Pull-down **Modify ► Array**Keyboard **ARRAY** short-cut **AR**

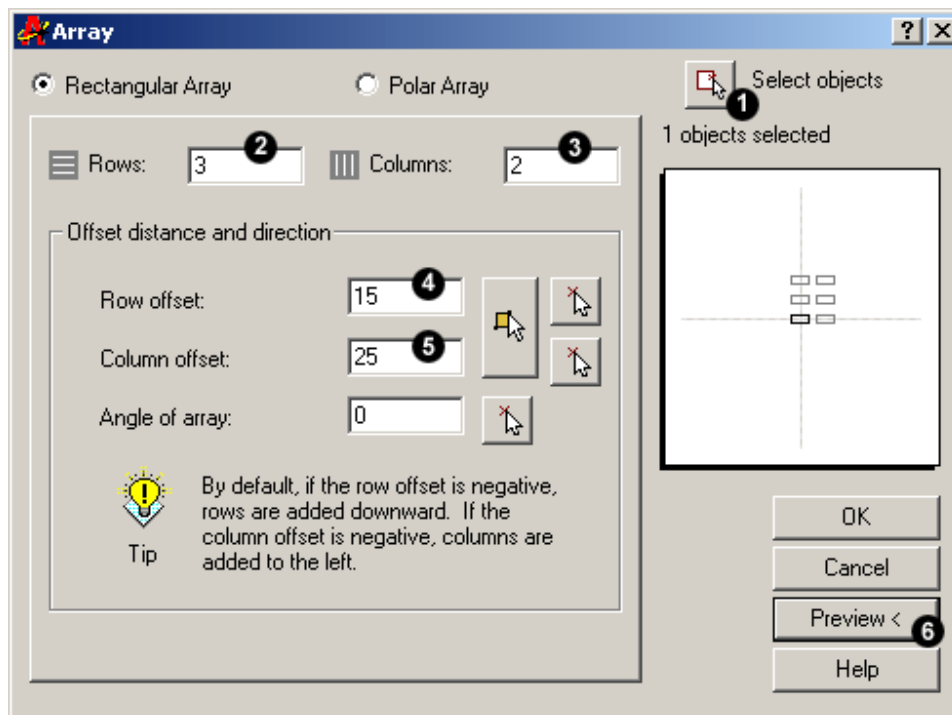
The Array command makes multiple copies of selected objects in a rectangular matrix (columns and rows) or a polar (circular) pattern. This command has been completely transformed in AutoCAD 2000i. It is now completely dialogue box driven with the option to see a preview of the array before it is created. You can also now create rectangular arrays at a user specified angle angle. This constitutes a major improvement in usability.

The Rectangular Array



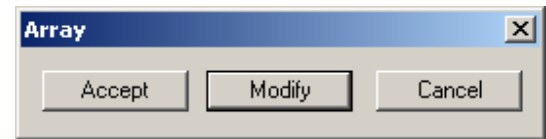
The illustration on the right shows the results of a rectangular array with two columns and three rows. The distance between rows is indicated with the dimension **DR** and between columns with **DC**. When creating rectangular arrays it is important to remember that new rows are created above the original object and new columns are created to the right of the original object. The resulting array is, therefore, always created with the original object in the bottom left hand position with respect to the current co-ordinate system. In the illustration, the original object is shown in cyan and the new objects in red. It is possible to create arrays which do not align with the *World Co-ordinate System* by setting a *User Co-ordinate System* first. See the UCS tutorial for details.

Try creating this array yourself. First, draw a rectangle 20 drawing units wide and 10 drawing units tall. To do this, start the rectangle command **Draw ► Rectangle** from the pull-down menu. When you are prompted to pick a point, pick somewhere in the lower left quarter of the drawing area. Then, when you are prompted for a second point, enter the relative cartesian co-ordinate **@20,10**. You have now drawn a rectangle at the right size. See the Using Co-ordinates tutorial for more information on drawing with co-ordinates.



Now start the Array command. First of all, make sure that the Rectangular Array radio button is selected and that you are looking at the Rectangular Array dialogue box and now follow the steps below:

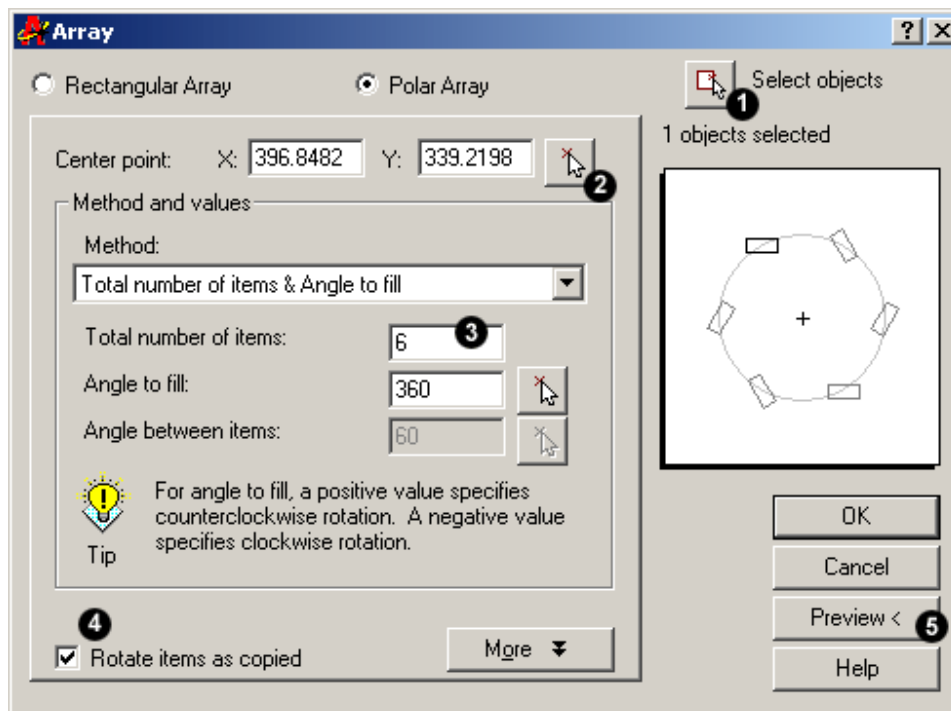
1. Click the Select objects button. The dialogue box will temporarily disappear enabling you to select the rectangle you just drew. Press the Return button on your keyboard to complete the selection. You are now returned to the dialogue box and the message immediately below the Select Objects button should read "1 objects selected".
2. Enter the number of rows required in the Rows edit box. For this example, enter the value "3". Notice that the schematic preview on the right hand side of the dialogue box updates to reflect the values you are entering.
3. Enter the number of columns required in the Columns edit box. Enter the value "2".
4. Enter the row offset in the Row Offset edit box. This is the distance DR in the illustration above. Note that tis is not the distance between rows. In this example, our rectangle is 10 units high and we will enter a row offset of 15. The result will be a 5 unit gap between rectangles.
5. Enter the column offset in the Column Offset edit box. The same parameters apply as for the row offset. Enter a value of 25 to give a 5 unit gap between our rectangles.
6. Click on the Preview button. Once again, the dialogue box disappears and the specified array is temporarily drawn so that we can preview it. We are now offered 3 choices. If the array isn't quite right, click the Modify button to return to the Array dialogue box. If you are happy with the array, click the Accept button, the array will be permanently drawn and the command is ended.



You should now have an array that looks similar to the one in the illustration above consisting of 6 rectangles arranged in 3 rows and 2 columns. As you can see, this command is very powerful and can save lots of time if used carefully.

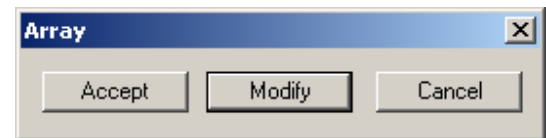
The Polar Array

The polar array works in a similar way to the rectangular array. The main difference is that rather than specifying the number and offset for rows and columns, you must specify a center point and the total number of objects in the array (including the original object).

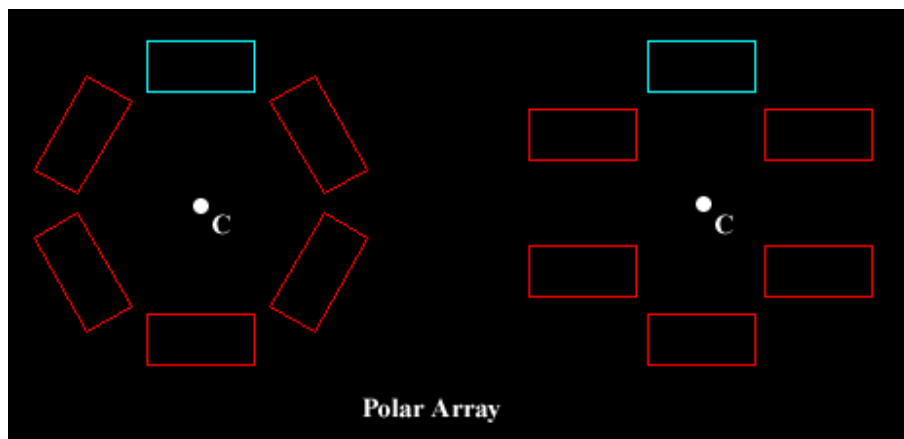


You can try this for yourself by drawing another rectangle as described above. This time, though, locate the rectangle in the centre top half of the drawing area. Now, start the Array command, click the Polar Array radio button and follow the steps below:

1. Click the Select objects button. The dialogue box will temporarily disappear enabling you to select the rectangle you just drew. Press the Return button on your keyboard to complete the selection. You are now returned to the dialogue box and the message immediately below the Select Objects button should read "1 objects selected".
2. Specify the center point for the array. This is the point C in the illustration below. You can do this by entering x and y co-ordinates into the appropriate edit boxes if you know what these values should be. However, this is rarely the case and most often you will want to click the Pick Center Point button to pick a point from the drawing area. Pick a point somewhere below the rectangle you have just drawn.
3. Enter a value for the total number of items. For this example, enter the value "6". Notice that once again, the schematic preview updates to reflect the values you have entered.
4. Make sure that the Rotate items as copied checkbox is checked.
5. Click on the Preview button. Once again, the dialogue box disappears and the specified array is temporarily drawn so that we can preview it. We are now offered 3 choices. If the array isn't quite right, click the Modify button to return to the Array dialogue box. If you are happy with the array, click the Accept button, the array will be permanently drawn and the command is ended.



The array you have just drawn should look something like the one illustrated below, left. Take some time to play around with the other options in the Array dialogue box to see what they can do.



The illustration above shows the results of a polar array using an original rectangle (cyan) copied six times about a centre point C, through an angle of 360 degrees (full circle). The only difference between the array on the left and the one on the right is that the Rotate items as copied checkbox was *not* checked in the right hand example.

All new objects created with the Array command will inherit the object properties of the original object. See the "Object Properties" tutorial for information on object properties.

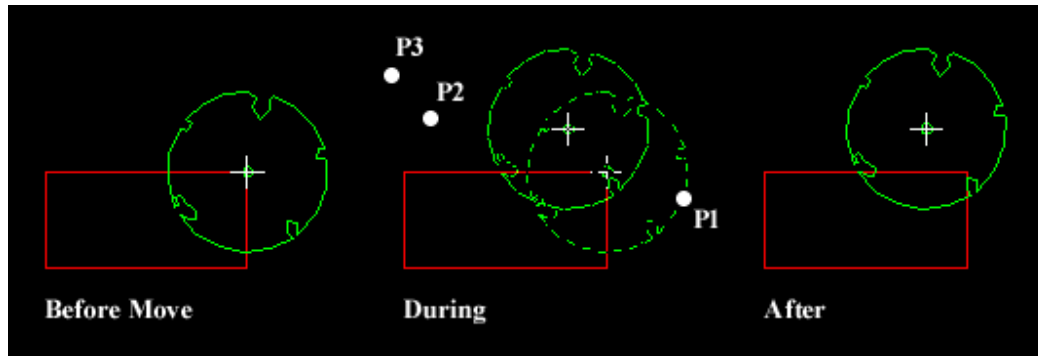
The Move Command

Toolbar Modify

Pull-down **Modify ► Move**

Keyboard **MOVE** short-cut **M**

The Move command works in a similar way to the Copy command except that no copy is made, the selected object(s) is simply moved from one location to another.



Command Sequence

Command: **MOVE**

Select objects: (pick object to move, P1)


Select objects: **↵** (to end selection)

Specify base point or displacement: (pick P2)

Specify second point of displacement or <use first point as displacement>: (pick P3)

Note that as with the Copy command, the two pick points, P2 and P3 are used only to indicate the distance and direction of movement.

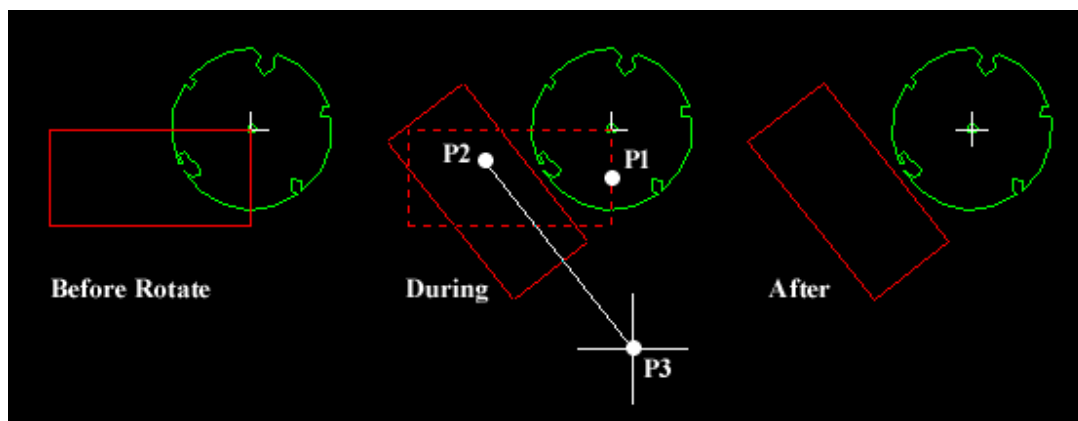
The Rotate Command

Toolbar **Modify** 

Pull-down **Modify ► Rotate**

Keyboard **ROTATE** short-cut **RO**

The Rotate command allows an object or objects to be rotated about a point selected by the user. AutoCAD prompts for a second rotation point or an angle which can be typed at the keyboard.




Command Sequence

Command: **ROTATE**

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

Select objects: (pick object to rotate, P1)

Select objects:  (to end selection)


Specify base point: (pick base point, P2)

Specify rotation angle or [Reference]: (pick second point, P3 or enter angle)

Remember, by default, AutoCAD angles start at 3 o'clock and increase in an anti-clockwise direction. The "ANGDIR" and "ANGBASE" variables remind you of this. If you want to rotate in a clockwise direction you can enter a negative angle by using a minus sign.

Note: You can change the angle direction and the base angle using the Units command, **Format ► Units...** from the pull-down menu. Click the "Clockwise" check box to change the direction and click the "Direction..." button to set the base angle.

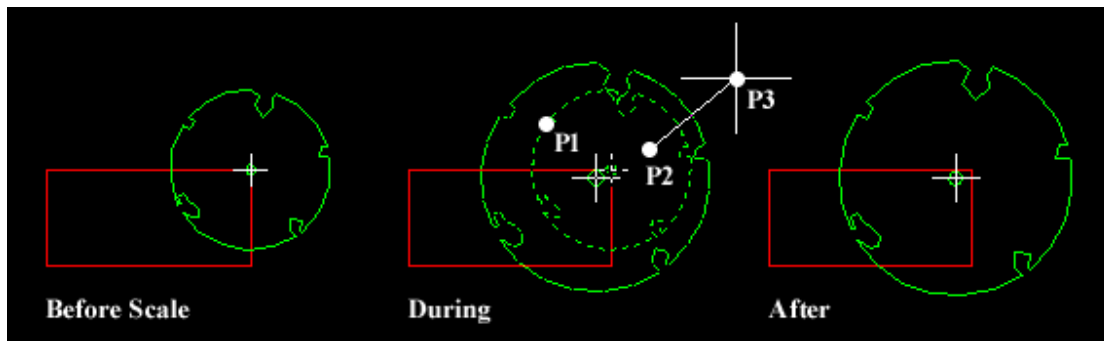
The Scale Command

Toolbar Modify 

Pull-down **Modify ► Scale**

Keyboard **SCALE** short-cut **SC**


The Scale command can be used to change the size of an object or group of objects. You are prompted for a pick point about which the selection set will be scaled. Scaling can then be completed by picking a second point (not always easy because it can sometimes be difficult to precisely control the scaling) or by entering a scale factor at the keyboard. For example a scale factor of 2 will double the size of the objects in the selection set and a factor of 0.5 will half them.



Command Sequence

Command: **SCALE**

Select objects: (pick objects to be scaled, P1)

Select objects:  (to end selection)

Specify base point: (pick base point, P2)

Specify scale factor or [Reference]: (pick second point, P3 or enter scale factor)

In the example shown above, the original tree symbol has been enlarged by dynamically scaling it using pick points to determine the change in scale. If you want to scale an object precisely, it is much easier to enter a scale factor using the keyboard.

Note that the position of the new sized tree symbol is determined by the location of the base point. The base point, P2 has been picked to the upper right of the centre of the tree which resulted in the centre of the tree shifting to a lower left position. If the base point had been picked in the centre of the tree symbol, the tree would have remained in the same position. In theory the base point can be any point in the drawing area but for ease of control it is best to choose a known point so that the results are obvious.

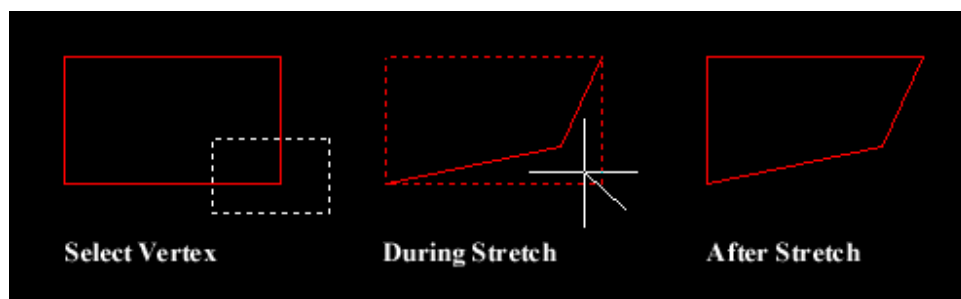
The Stretch Command

Toolbar Modify 

Pull-down **Modify** ► **Stretch**

Keyboard **STRETCH** short-cut **S**

The Stretch command can be used to move one or more vertices of an object whilst leaving the rest of the object unchanged. In the example below, a rectangle has been stretched by moving one vertex to create an irregular shape.




Command Sequence

Command: **STRETCH**

Select objects to stretch by crossing-window or crossing-polygon...

Select objects: (pick first point of crossing window)

Specify opposite corner: (pick second point of window)

Select objects:  (to end selection)

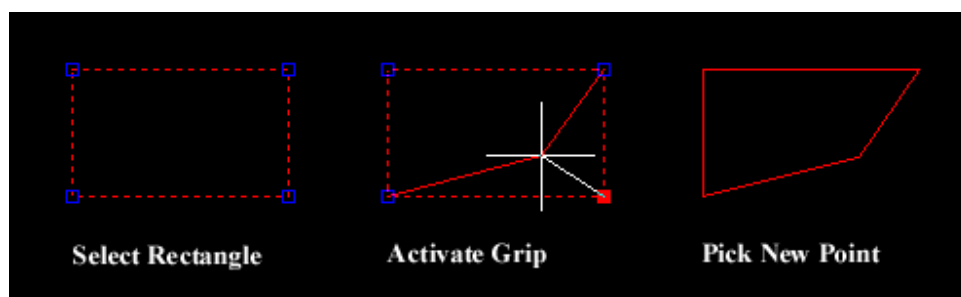
Specify base point or displacement: (pick base point)

Specify second point of displacement: (pick second point)

To select vertices to stretch, you must use a crossing window or polygon. See the "Object Selection" tutorial for details of these selection methods.


Stretching with Grips

Although the Stretch command can be very useful, it has largely been superseded by the use of "Grips" which allow this sort of modification to be made much more intuitively.



To stretch an object using grips, simply select the object by clicking on it (you can do this without starting a command). The object becomes highlighted and small square grips appear at each vertex and various snap points, depending upon the object type. Click a grip to activate it and click again to reposition it. When you have completed your modifications, use the Escape key (Esc) at the top left of your keyboard to deselect the object and release grips.

The Lengthen Command

Toolbar Modify 

Pull-down **Modify** ► **Lengthen**

Keyboard **LENGTHEN** short-cut **LEN**

The Lengthen command can often be used instead of either the Trim or Extend commands. Indeed the end result is exactly the same. The Lengthen command can be used to either lengthen or shorten Lines, Arcs, open Polylines, elliptical Arcs and open Splines without the use of cutting or boundary edges.


Command Sequence

Command: **LENGTHEN**

Select an object or [DElta/Percent/Total/DYnamic]: **DY**

Select an object to change or [Undo]: (select a line or arc)

Specify new end point: (pick new end point)

Select an object to change or [Undo]:  (to end)

The command sequence above demonstrates the use of the Dynamic Lengthen option which is probably the most useful for general purpose drafting. However, the other options are worth getting to know because they can save lots of time and effort.

The Total option allows you to change the total length of a line to any value that you specify. The Percent option allows you to change a line length using a percentage. For example, a value of 50 will result in a line one half the length of the original and a value of 200 will result in a line twice the length of the original. The Delta option can be used to extend or reduce the endpoint of a line by a given distance. The endpoint affected by the change is the one closest to the pick point when the object selection is made.

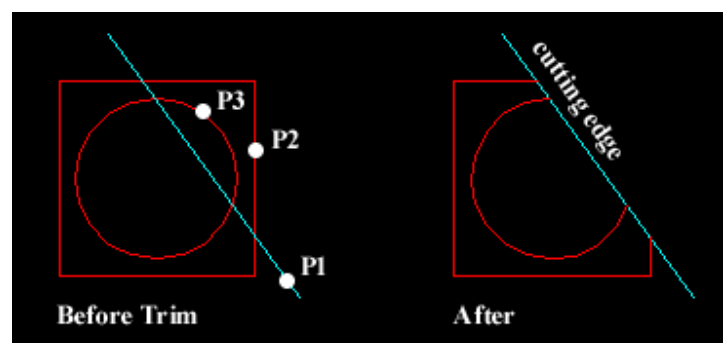
The Trim Command

Toolbar Modify 

Pull-down **Modify** ► **Trim**

Keyboard **TRIM** short-cut **TR**

The Trim command can be used to trim a part of an object. In order to trim an object you must draw a second object which forms the "cutting edge". Cutting edges can be lines, xlines, rays, polylines, circles, arcs or ellipses. Blocks and text cannot be trimmed or used as cutting edges. The illustration on the right shows the Trim command in action. The red square and circle have been drawn using the Polygon and Circle



commands respectively. In order to trim these objects, a line has been drawn (cyan in the illustration), this forms the cutting edge. The Trim command, unlike most other modify commands requires that two separate object selections are made. The cutting edges are selected first (there can be one or more) and then the objects to be trimmed are selected. In the example above, the line is selected first because it forms the cutting edge and then the square and circle are selected.

The Trim command is slightly more complicated than many other modify commands. To get a better understanding of how it works, draw a square, circle and line as illustrated above and then follow the command sequence below. Don't forget to watch the AutoCAD command line at each stage of the process.


Command Sequence

Command: **TRIM**

Current settings: Projection=UCS Edge=None


Select cutting edges ...

Select objects: (select the cutting edge, P1)

Select objects:  (to end cutting edge selection)

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (pick the part of the square which you want to trim, P2)


Select object to trim or shift-select to extend or [Project/Edge/Undo]: (pick the circle, P3)

Select object to trim or shift-select to extend or [Project/Edge/Undo]:  (to end)

Notice that at each trimming step you are given the option to undo the previous trim. This can be very useful if you inadvertently pick the wrong object.

In the above example, when the objects were trimmed, both pick points were made to the right of the cutting edge, resulting in the removal of that part of the objects to the right of the cutting edge. Obviously, the portion of square and circle to the left of the cutting edge could have been removed by picking to the left of the cutting edge. Also, you may not have noticed it, but by trimming a circle you have created an Arc object. This makes no visible difference but the object type has changed.

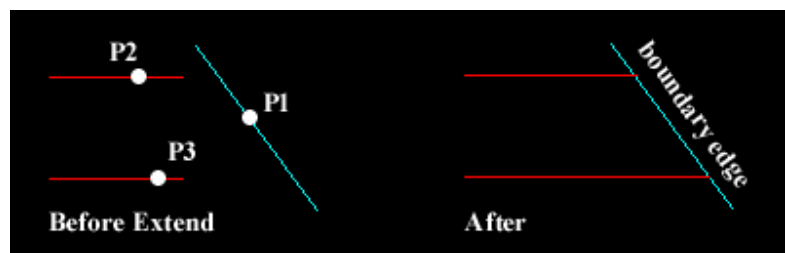
The Extend Command

Toolbar  Modify

Pull-down **Modify** ► **Extend**

Keyboard **EXTEND** short-cut **EX**

This command extends a line, polyline or arc to meet another drawing object (known as the boundary edge). In the illustration on the right, two lines (red) are extended to meet another line (cyan) which forms the boundary edge. This command works in a similar way to the Trim command, described above. Two selections are made, one for the boundary edge(s) and one for the object(s) to extend.



Midpoint

Lines and other objects can be extended in one of two directions. In the illustration on the right, the red line could be extended either to the right or to the left. You can tell AutoCAD in which direction to extend by picking a point to the right or left of the midpoint respectively. AutoCAD does not intuitively know where the boundary edge lies so you must explicitly indicate the direction of extension by picking either one side or other of the midpoint.

Draw the lines as shown in the illustration and follow the command sequence below.

Command Sequence

Command: **EXTEND**

Current settings: Projection=UCS Edge=None

Select boundary edges ...

Select objects: (select the boundary edge, P1)

Select objects: ↵ (to end boundary edge selection)

Select object to extend or shift-select to trim or [Project/Edge/Undo]: (pick the object which you want to be extended, P2)

Select object to extend or shift-select to trim or [Project/Edge/Undo]: (pick another object which you want to be extended, P3)

Select object to extend or shift-select to trim or [Project/Edge/Undo]: ↵ (to end)

Sometimes you may get the message "Object does not intersect an edge" or "No edge in that direction". If this happens it means that you are either picking the wrong end of the object or the object you are trying to extend will not meet the boundary edge. The solution is either to pick near the end you want to extend or to move the boundary edge so that the extended line will intersect with it.

Using Edgemode

If the line you are trying to extend does not intersect with the boundary line, you can use the "Edge" option to toggle *Edgemode*

to "Extend" (the default is "No Extend"). When the Extend command is set to Extend Mode, the objects being extended will extend to an imaginary line through the boundary edge, irrespective of whether the extended object actually intersects with the boundary edge. This is particularly useful and can save lots of time.



The illustration on the left shows the result of extending a line (red) to a boundary edge (cyan) with Edgemode set to "Extend". The same process would have resulted in an error message if Edgemode had been set to "No Extend". To get a better understanding of how this works, draw the two lines as shown in the illustration and try to extend them using the

default settings. When you have done that, follow the command sequence below.

Command Sequence (Edgemode)

Command: **EXTEND**

Select boundary edges: (Projmode = UCS, Edgemode = No extend)

Select objects: (select the boundary edge, P1)

Select objects: ↵ (to end boundary edge selection)

Select object to extend or shift-select to trim or [Project/Edge/Undo]: E (to use the Edge option)

Enter an implied edge extension mode [Extend/No extend] <No extend>: E (to set Edgemode to Extend)

Select object to extend or shift-select to trim or [Project/Edge/Undo]: (pick the object to be extended, P2)

Select object to extend or shift-select to trim or [Project/Edge/Undo]: ↵ (to end)

Notice that the current value of Edgemode is always displayed on the command line when you start the Extend command. The Edge option can also be used with the Trim command to enable trimming to cutting edges which do not actually intersect the object to trim. Edgemode is a *system variable*, so any change to its value will affect both the Trim and Extend commands.

See the Lengthen command for more ways to extend and trim objects.

Shift Selection with Trim & Extend

You may have noticed during the command sequences for the Trim and Extend commands that you have the option to "shift-select". This feature is new to AutoCAD 2000i and it enables you to extend while using the Trim command and to trim while using the Extend command. These two commands are very closely related and you often need to trim and extend objects at the same time. If you are a beginner with AutoCAD it may be a good idea to avoid this feature initially, the Trim and Extend commands can be tricky to get to grips with in any case. However, do remember this feature because it is a great time saver.

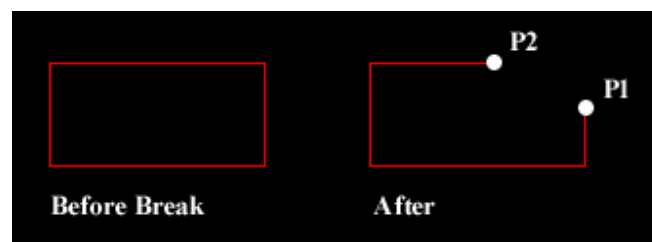
The Break Command

Toolbar Modify 

Pull-down **Modify** ► **Break**

Keyboard **BREAK** short-cut **BR**

The Break command enables you to break (remove part of) an object by defining two break points. In the illustration below, a corner of a rectangle has been removed. The Break command can be used with lines, polylines, circles, arcs ellipses, splines, xlines and rays. When you break an object, you can either select the object using the first break point and then pick the second break point, or you can select the object and then pick the two break points.



Command Sequence

Command: **BREAK**

Select objects: (select the object using the first break point, P1)

Specify second break point or [First point]: (pick the second break point, P2)

The section of the object is removed and the command ends.

Sometimes you may want to select the object first and then specify the two break points. If this is the case, use the "First

point" option to specify the first break point. By default, AutoCAD assumes that the point used to select the object is the first break point. This is often confusing for new users.

It may sometimes be necessary to break a line into two without removing any part of it. In this case, simply pick the first and second break points in the same position.

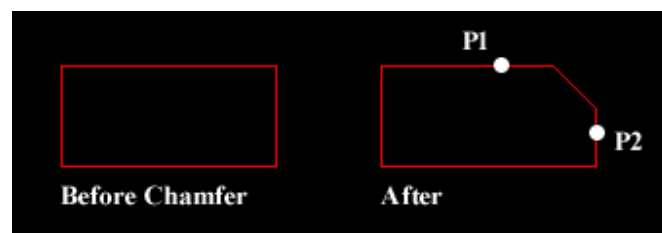
The Chamfer Command

Toolbar Modify 

Pull-down **Modify** ► **Chamfer**

Keyboard **CHAMFER** short-cut **CHA**

The Chamfer command enables you to create a chamfer between any two non-parallel lines as in the illustration below or any two adjacent polyline segments. Usually, the Chamfer command is used to set the chamfer distances before drawing the chamfer. Follow the command sequence below where the chamfer distances are changed to 20 before the chamfer is made.



Command Sequence

Command: **CHAMFER**

(TRIM mode) Current chamfer Dist1 = 10.0000, Dist2 = 10.0000

Select first line or [Polyline/Distance/Angle/Trim/Method]: **D** (to set distances)

Specify first chamfer distance <10.0000>: **20** (enter required distance)

Specify second chamfer distance <20.0000>: **↵** (first distance value or enter a different value)


Select first line or [Polyline/Distance/Angle/Trim/Method]: (pick P1)

Select second line: (pick P2)

The chamfer is made and the command ends.

Notice from the command sequence that there are a number of options which can be used to control the way the Chamfer command behaves. The Polyline option can be used to chamfer all vertexes of a polyline simultaneously. The Distance option allows you to specify the two chamfer distances. Angle allows the angle between the first line and the chamfer to be specified. Trim is used to control whether the original lines are trimmed to the chamfer or remain as they are. Finally, Method is used to toggle the command between Distance and Angle mode. When Angle mode is used, the chamfer is defined using one distance and an angle rather than two distances.

The Fillet Command

Toolbar Modify 

Pull-down **Modify** ► **Fillet**

Keyboard **FILLET** short-cut **F**

The Fillet command is a very useful tool which allows you to draw an arc between two intersecting lines or adjacent polyline segments. You first need to use the command to set the required radius and then a second time to select the two lines.

Command Sequence

Command: **FILLET**

Current settings: Mode = TRIM, Radius = 10.0000

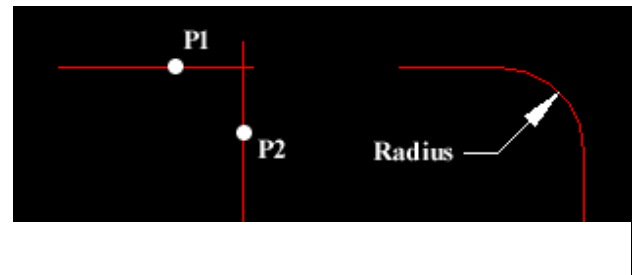
Select first object or [Polyline/Radius/Trim]: **R**

Specify fillet radius <10.000>: **25**

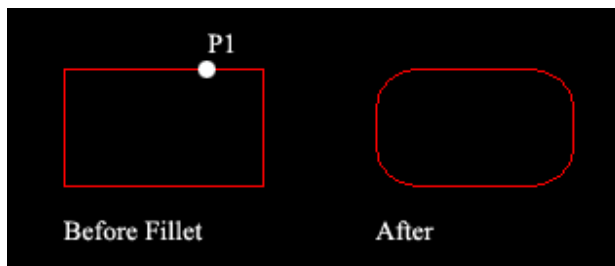
Select first object or [Polyline/Radius/Trim]: (pick P1)

Select second object : (pick P2)

The Fillet command can also be used to fillet arcs and circles. The "Polyline" option also allows you to fillet all vertices of a polyline with a single command. It's worth experimenting with this command, it can save you lots of time and enables you to construct shapes which otherwise would be quite difficult.



For



example, you can easily create the lozenge shape shown on the right from a simple rectangle. Since AutoCAD rectangles are just closed polylines, you can use the Polyline option of the Fillet command to fillet all polyline vertices simultaneously. Try this for yourself; draw a rectangle and then follow the command sequence below.

Command Sequence

Command: **FILLET**

Current settings: Mode = TRIM, Radius = 10.0000

Select first object or [Polyline/Radius/Trim]: **P**

Select 2D polyline: (pick P1)

4 lines were filleted

Tip: Make sure that the radius you specify will fit the objects you select, otherwise the fillet command will not work.

The Explode Command

Toolbar **Modify** 

Pull-down **Modify** ► **Explode**

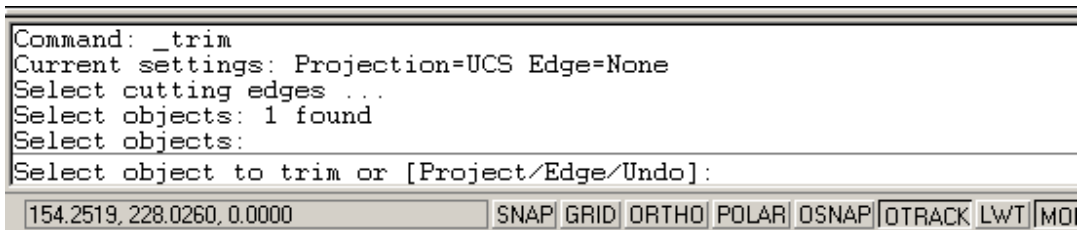
Keyboard **EXPLODE**


The Explode command is used to "explode" single objects back to their constituent parts. In other words, the command is used to return blocks, polylines etc. (which may be composed of a number of component objects) back to their individual

component parts. The change has no visible effect.

Tips & Tricks

- Always look at the command line for guidance when you are learning new AutoCAD commands. The command line will prompt you for information and this is the easiest way to find out how a new command works. This is especially true for the more complicated commands like Trim and Extend.
- The command line window displays 3 lines of text by default. You can change this by clicking and dragging the top of the window frame. When you are starting with AutoCAD you may like to see more than 3 lines. The illustration below shows the command line window increased in size to 6 lines.



- The command line can also be displayed as a floating text window. The AutoCAD text window is displayed by pressing the **F2** key on the keyboard. See Function Keys on the Drawing Aids tutorial for more details.
- You can control the way text is mirrored using the MIRRTEXT variable.
- You can use the Fillet command with a radius of zero to trim intersecting lines back to their intersection. Of course, you could also achieve this effect with the Trim command but if you have a number of operations to complete, the Fillet method is much quicker.
- When using the Trim and Extend commands, hitting  when prompted to "Select cutting edges" and "Select boundary edges" respectively will automatically select all valid cutting or boundary edges on the screen. This can be a real time saver if you have a complicated set of edges to select.

