AutoCADD Tutorial
A survival guide

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October 9, 1997
Revised April 17, 1998

Contents

a) RETURN or click the right mouse button to repeat last command

b) RETURN or click the right mouse button to start next drawing element at last point selected on screen

(or designated by relative coordinates)

Introduction

AutoCADD is an extremely powerful drafting tool. It has enabled accuracy in drawing barely imaginable thirty years ago. AutoCADD has taken a lot of the guesswork out of locating routes for roads, locations for underwater foundations, and even space research. It is so accurate that it can give commands to an electronically controlled saw to cut wood to the thousandth of an inch. That's what we use it for in MIL; to cut out robot parts and copper circuit boards. The machine is called a t-tech machine, and presents specific problems that must be addressed in AutoCADD. Specifically, we have to leave little holes in the outlines of the things to be cut in order to anchor the object in place, and account for the width of the blade in cutting. First we'll learn about AutoCADD, then come back to learn the specific applications for the MIL Lab.

Relative Coordinates

Relative coordinates are extremely useful in drawing precisely. The relative coordinate command takes the last point to which you clicked and temporarily makes it the origin. It works for rectangular and polar coordinates (also for spherical, but 3-D applications are more complicated). After selecting a first point, type in " @X,Y" or " @L<θ". The @ sign is what makes it a relative coordinate. For absolute coordinates, leave off the @ symbol. (Note: absolute coordinates are harder to work with; few drawings begin at the origin.)

Example:

Draw a horizontal line 5 units long

1) type 'line' at the command prompt or select it from the draw toolbar

2) prompt now says '*select first point'*

3) click on any point on the screen

4) prompt now says *'next point:'*

5) type " @5,0 (rectangular coords) or type @5<0 (polar coords)"

6) prompt now says *'next point:'*

7) press RETURN or click the right mouse button to end line sequence

O-Snap

O-Snap stands for Object-snap and is also extremely useful in drawing precisely. It allows you to "snap" to the end, midpoint, center, tangent, intersection, and other key points of lines and other elements (circles, etc.) on the drawing.

Example:

Draw a circle of radius 2.5 units with the center of the circle at the endpoint of the line you drew before

1) type 'circle' at the command prompt or select it form the draw toolbar

2) prompt now says '*Center/3 point/2 point/. . . <center>"*

3) press RETURN or click the right mouse button (the command in brackets is the default command, which is what we want)

4) Select O-snap from the toolbar menu

5) select endpoint from the osnap toolbar

6) cursor changes to show a little box. You must get the endpoint of the line in the box ("target") for the computer to recognize the keypoint (in this case endpoint) desired.

7) prompt now says *'Radius/Diameter <Radius>:'*

8) type "2.5"

9) Command prompt returns

Grid

A grid is helpful if you are drawing fairly small things (relative to the drawing scale) that are whole unit distances. It basically turns the screen into Engineering Computation paper.

To turn grid on:

1) type "grid", or select grid from the settings menu

2) prompt now says on/off <off>

3) type "on"

Snap

Snap works with the grid to allow you to snap from grid point to grid point, eliminating the need for relative

coordinates and O-snap as long as you are working with whole units. (You can set grid and snap spacing for less than whole units, but it gets confusing)

Note: you can turn snap on without the grid on, but that gets **REALLY** confusing; I don't recommend it.

To turn snap on:

1) type "snap", or select snap from the settings menu
2) prompt now says *spacing/. . ./on/off<1.0>*
3) press RETURN or click the right mouse button (this sets your snap spacing to 1 unit)
4) press RETURN again
5) prompt now says *spacing/. . ./on/off<1.0>*
6) type "on"
7) move your cursor on the screen. It "jumps" to the grid points
8) to turn snap off, type or select snap, then type "off"

Line

The line command draws line elements. It will draw three dimensional lines as well, so be sure you stay in 2-D coordinates unless you want large O-snap problems and huge headaches.

Circle

The circle command draws circles. Shocking, isn't it?

P-line

P-line stands for Polyline, and enables you to draw and edit connected line segments as if they were all one thing. I recommend drawing all lines as plines for easy manipulation later.

Example: Draw a representation of a 3 X 4 rectangle using a p-line.

1) type "pline" or select it from the toolbar
2) pick any point on the screen
3) pick a point 3 units away horizontally using either the grid and snap or relative coordinates
4) pick a point 4 units away vertically using either the grid and snap or relative coordinates
5) continue until you close the rectangle
6) press RETURN or click the right mouse button to end the command

P-edit

P-edit stands for polyline edit, and enables you to manipulate and join polylines together. It is useful to change all your lines to polylines and join them before offsetting them to the outline and inline layers; it reduces editing later.

To join lines using pedit:

1) type "pedit" at the command prompt
2) prompt now says *join/stretch/. . ./<select polyline>*
3) pick the line (or pline) you wish to edit
4) if it was a regular line, command prompt says *do you wish to make this a pline? Yes/no*
5) type "y", RETURN RETURN
6) repeat this until all the lines are plines
7) type "pedit" (or click the right mouse button, or press RETURN)
8) select one line
9) type "j" for join
10) select all the lines to be joined to it
11) press return or click the right mouse button when finished

Text and D-Text

If you want to add text to your drawing, the best way is to use d-text. D-text stands for dynamic text, and that command allows you to see what you are typing as you type it. Text doesn't allow you to do that. The other difference between text and d-text is that d-text allows you to enter multiple lines of text just as you

would in a simple word processor; just put a return at the end of a line and keep typing. Text requires that you enter the command for each line of text separately. With that in mind, we'll learn how to use d-text.

To add text to a drawing:

1) type 'dtext" at the command prompt

2) the prompt now says *Style/. . .Justify/<enter first point>*

3) pick the point on the screen where you want text to be entered

4) prompt now says *rotation<0.00>*. Press enter for horizontal text

5) type text

6) press RETURN twice when finished

Offset

Offset is by far one of the most useful drawing commands you have. It copies a line on your screen a designated distance parallel to itself. It will do the same thing for circles and p-lines. It's extremely useful for locating centers of circles and drawing the cut lines for the t-tech machine from your actual shape.

To offset a shape:

1) type "offset" at the command prompt, or select it from the modify options on your toolbar

2) the prompt now says *Enter distance or<Through>*

3) type value of the perpendicular distance from the line/shape you have to the new line

4) prompt now says *select object*

5) pick the object you want to offset

6) prompt now says *Side to offset?*

7) pick the side of the screen where you want the new object to appear

8) press RETURN when finished offsetting the object; you con offset it more than once at t time

Trim

The trim command takes off the unwanted material from a line that is too long or a circle of which only part

is wanted. You will use the trim command to make the tabs for the t-tech machine, as well as to trim corners. To use the trim command, you must have an intersection at the point at which you want to trim the entity.

To trim an entity:

1) type "trim" at the command prompt or select it from the modify tool bar

2) the prompt now says *select cutting objects/undo*

3) select the object(s) against which you want to cut (the straightedge(s) along which you want to cut)

4) press RETURN or click the right mouse button.

5) select portion of object(s) that you want to disappear.

6) press RETURN or click the right mouse button when finished

Extend

Erase

Erase erases. (That's another shocker, isn't it?)

To erase an element:

1) type "erase" at the command prompt or select it from the modify toolbar

2) select the object(s) to erase

3) press RETURN or click the right mouse button when finished

Escape key

The escape key cancels any command and returns the command prompt

Selecting objects

There are two way to select objects: 1) clicking on the object, or 2) windowing the object. The former is fairly self-explanatory.

To window an object:

1) pick a point on one side of the object(s) to be selected

2) pick a point diagonally across the object(s)

3) selected objects are shown in dashed lines
4) to unselect an object, hold down the shift key and pick or window the object

Move and copy

The move and copy commands are very similar; the copy command is slightly more complicated. If you can copy, you can move.

To copy an object:

1) type "copy" at the command prompt or select it from the modify toolbar
2) prompt says *select object(s):* select the object(s) to be copied
3) press RETURN when finished
4) prompt says *<select first point> or Multiple:* select the point from which you want to copy (in other words, where do you want the "origin" to be while you're copying?) If you're going to copy the same thing over and over, press "m" for multiple, RETURN, then select the first point
5) prompt says *select second point* select the point to which you want to copy the object (note: use o-snap to move and copy accurately)

Zoom

The zoom command enables you to look at different things on the screen. You can zoom in, out, window, previous, extents, dynamic, and probably other ways I've forgotten because you never use them.

To zoom

1) type "z" at the command prompt
2) the command prompt now says *<window>extents/ dynamic/in/out/. . .*
3) choose the appropriate zoom command (window and previous are the most useful

Drawing setup for MIL applications

layers

The MIL t-tech machine requires the use of three layers (four for text):

1) The 0 layer
2) the outline layer
3) the inline layer
4) text

To set up the MIL layers:

1) select "layers" from the data pull-down menu
2) a dialog box pops up. Notice that the zero layer is already defined.
3) highlight the zero layer in the box
4) choose "set color"
5) set color to green
6) click "OK"
7) select the zero layer so that it is not highlighted
8) type "outline" in the layer name box at the bottom
9) select new
10) the outline layer appears in the box above. Highlight it
11) choose "set color"
12) set color to #84 (dark green)
13) repeat steps for the inline layer (color: red), and the text layer (??????)
14) click "OK" until the dialog box disappears

layer 0 (zero)

Layer zero is where you draw the actual shape of the piece you want to cut. It is your working drawing layer. You will draw all of your elements here unless told otherwise.

outline and inline layers

The outline and inline commands are used to account for the width of the cutting bit in the t-tech machine. The blade is 64 mils (.064") wide. If we were to cut directly on the outline of the shape we wanted, the resulting shape would be 64 mils smaller (for outline objects) or 64 mils larger (for cutouts) than desired. We account for this by using the offset command to enlarge or reduce the size of the object, the Extend or Trim commands to close in the shape, the Mlayer command to change the layer of the new objects.

Outline

The outline layer is used for objects that are to be outlined by the t-tech machine. The t-tech would make the objects too small unless we oversize them.

To create an outline for the t-tech machine:

1) set the layer to outline using the pull-down at the top of the screen
2) draw a line anywhere on this layer (it will be erased later)
3) offset all the outline elements out by 0.032 inches. Notice that they stay on layer zero even though you have your layer set to outline.
4) type "DDModify" at the command prompt
5) select all the outline elements
6) press RETURN or click the right mouse button
7) select the line you drew on layer outline
8) all of the elements change to layer outline
9) erase the line you drew on layer outline
10) use the fillet, extend, and trim commands as necessary to enlose the new lines

Inline

The inline layer is for objects inside the cutting surface (screw holes, access hatches, etc.) If we were to let the t-tech cut on the layer zero lines, the holes would be too big when they were cut. We must undersize the holes by 64 mils total.

To create an inline for the t-tech machine:

1) set the layer to inline using the pull-down at the top of the screen
2) draw a line anywhere on this layer (it will be erased later)
3) offset all the ouline elements in by 0.032 inches. Notice that they stay on layer zero even though you have your layer set to outline.
4) type "Mlayer" at the command prompt
5) select all the inline elements
6) press RETURN or click the right mouse button
7) select the line you drew on layer inline
8) all of the elements change to layer inline
9) erase the line you drew on layer inline
10) use the fillet, extend, and trim commands as necessary to enclose the new lines

anchor tabs

Anchor tabs are necessary to keep the wood outline together during cutting; if the t-tech cuts out a hole completely, the wood will fall out, and the blade will get stuck in the hole. From what I understand, the result is not pretty at all. By convention, the anchor tabs are created by trimming the cutting lines with little circles that are strategically placed to effectively anchor the piece. These little circles are 80mils (0.080") in diameter.

To locate an anchor tab:

1) select "circle" from the draw toolbar
2) pick "near" from the osnap toolbar
3) select the point on the element you wish to have an anchor tab
4) type "d" (for diameter)
5) type .080
6) press RETURN
7) select "trim" from the modify toolbar

8) select the circle as your cutting object
9) press RETURN
10) select the portion of the line inside the circle
11) press RETURN
12) You can create multiple tabs at a time by making many circles and invoking the trim command
with all the circles as cutting objects