


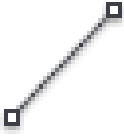




LECTURE # 2




Drawing Tools in AutoCAD

| Command | Description | Option |
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| DRAWING TOOLS: | | |
| 1. Open | Opens an existing file | Application menu → Open |
| 2. Units | Allows you to set the unit type (Architectural or Decimal), angle display format (degrees-minutes-seconds, or decimal degrees), and precision of the units, | Application menu → Utilities → Drawing setup |
| 3. Limits | Changes the imaginary boundaries of the drawing, and controls whether drawing can be made outside of the boundaries. If Limit- checking is ON, drawing cannot be done outside of limits. | It sets lower left corner and upper right corner. Type limits on the command bar. |
| 4. Dimension style  | Creates and modifies dimension styles | ---- |
| 5. Properties | Controls properties of existing object | Select the and right click, the various properties of the object will be displayed |






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| <p>6. Line</p>  | <p>Used to draw straight line, between two specified points</p> | <ul style="list-style-type: none">• At command line just type L and press enter• On the ribbon click on the line icon. <p>Any of the above options can be used to draw a line. AUTOCAD will ask you to specify the first point, click on a point and move the mouse. AutoCAD will now ask ‘specify the next point or [undo]’. Respond by clicking at another point. AutoCAD will keep asking ‘specify the next point or [undo]’, until you finish the command by pressing Enter.</p> |
| <p>7. Construction line</p>  | <p>Draws a line to full page at any angle.</p> | <ul style="list-style-type: none">• At command line just type CL and press enter or• On the ribbon click on the construction line icon. |
| <p>8. Multiline</p>  | <p>Allows you to draw 1 to 6 lines parallel to each other, by specifying the distance between the lines.</p> | <ul style="list-style-type: none">• On command line type ‘ml’ |





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| <p>9. Poly-line</p>  | <p>Creates a closed polyline of a boundary by picking an open area within an area enclosed by lines, arcs or circles.</p> <p>Unlike the standard line they can have width and they can follow curved path. To draw poly-line you need to give start and end point. Other options are to be selected only after the first point chosen.</p> | <ul style="list-style-type: none">• H sets the half-width of the polyline segments• U Undoes previous segment• W sets the width of poly-line segments |
| <p>10. Spline</p>  | <p>Allows you draw a curve. You need to specify the different points on the screen and curve passing through all the points will be drawn.</p> | <ul style="list-style-type: none">• On command line type spline. |
| <p>11. Polygon</p>  | <p>Draws regular polygons with a specified number of sides. Polygons are Polyline entities</p> | <ul style="list-style-type: none">• E Specifies size and rotation of polygon by picking endpoints of one edge• C Circumscribes polygon around a circle• I Inscribes polygon within a circle |






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| <p>12. Arc</p>  | <p>Draws an arc. The default method of drawing arcs is selecting three points (so-called "3-point arc"), which are the two endpoints of the arc and some other point along its locus. Other methods of drawing an arc can be specified by three letters, such as SEA, which means "Start Point, End Point, and included angle.</p> | <ul style="list-style-type: none"> • An Included angle • C Center point of arc • D Direction angle of a line tangent to the arc • E Endpoint of arc • L Length of chord passing thru both endpoints of the arc • R Radius • S Start point of arc |
| <p>13. Circle</p>  | <p>Draws a circle of any size. The default method is to pick a center point and pick a point on the radius or type the radius dimension, but other methods can be selected</p> | <ul style="list-style-type: none"> • 2P Specifies circle by picking 2 points on the diameter • 3P Specifies circle by picking 3 points through which the circle will pass • D Allows entering the diameter dimension instead of radius dimension • TTR Specifies circle by picking two lines, arcs or circles for the circle to be tangent to, and entering the dimension of the radius • <RET> Enters radius of circle (the default) |
| <p>14. Rectangle</p>  | <p>Draws a rectangle. You can select rectangle, AutoCAD will ask you to</p> | <ul style="list-style-type: none"> • 'A' on command line will give you rectangle of specified area. AutoCAD will ask you to specify the area. When you enter area it will |



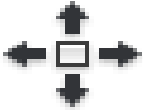




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| | <p>‘specify the first corner’, click on a point.</p> <p>AutoCAD will ask you to ‘specify the opposite corner’, click on another point.</p> | <p>ask you specify the length.</p> <ul style="list-style-type: none"> • ‘D’ on command line will give rectangle with specified dimensions. AutoCAD will ask you specify the length and then width. |
| <p>15. Ellipses</p>  | <p>This command is used to draw ellipse. The first two points of the ellipse determine the location and length of the first axis.</p> <p>The third point determines the distance between the center of the ellipse and the end point of the second axis.</p> | <ul style="list-style-type: none"> • C Allows specification of Center point of ellipse rather than first axis endpoint • R Allows specification of eccentricity rotation rather than length of second axis |
| <p>16. Donut</p>  | <p>Draws filled rings with specified inside and outside diameters</p> | <ul style="list-style-type: none"> • Select donut option from menu • It will ask you for inner diameter, specify it. • Next it will ask for outer diameter specify it. |








MODIFYING TOOLS

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| <p>17. Erase</p>  | <p>Erases selected entity from the drawing.</p> | <p>Type E on the keyboard and select the object you want to delete</p> |
| <p>18. Copy</p>  | <p>Draws a copy of selected objects using two methods -- "base point" method, or "displacement" method.</p> | <p>M Allows multiple copies to be made of an object (in drawing which have similar object occurring many times such as doors window will have to be drawn a number of times, the COPY command can be used repeatable to save time.)</p> |
| <p>19. Mirror</p>  | <p>Reflects or makes mirror image of existing or designated entities about a user-specified axis.</p> | <p>Type MI on the keyboard and select the object you want to mirror</p> |





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| <p>20. Offset</p>  | <p>Creates a new line, polyline, arc or circle parallel to the entity and at a specified distance from it.</p> | <p><number> specifies offset distance T "Through" allows specification of a point through which the offset line, polyline, arc or circle is to pass</p> |
| <p>21. Array</p>  | <p>Makes multiple copies of selected objects in a rectangular (parallel with the snap rotation) or circular pattern. Each resulting object can be manipulated independently.</p> | <p>The operation of array varies depending on type of array you choose. In rectangular pattern AutoCAD asks for the number of rows (Horizontal) and columns (Vertical) to be constructed.</p> |
| <p>22. Move</p>  | <p>Moves designated entities from their present location to another location without changing their size and orientation.</p> | <p>Type M on the keyboard and select the object you want to move</p> |
| <p>23. Rotate</p>  | <p>Rotates the object around a central rotation point.</p> | <ul style="list-style-type: none"> • Select the objects to rotate, then specify the base point about which the rotation is to occur • Specify the angle through which the rotation should occur |
| <p>24. Stretch</p>  | <p>Allows moving a portion of a drawing while retaining their connections to other parts of the drawing. You cannot stretch Blocks, Hatch patterns, or Text entities, however.</p> | <ul style="list-style-type: none"> • Allows lengthening or shortens the object. • Crossing window or polygon must cross the objects you want to stretch. • Any object lies completely within the selection window is moved. • You need to enter the base point and amount of stretch. |




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| <p>25. Lengthening</p>  | <p>Increases or decreases the length of objects</p> | <ul style="list-style-type: none"> Type len on the keyboard and select the object you want to lengthen and by how much. |
| <p>26. Trim</p>  | <p>Erases a portion of selected entities crossing the "cutting edge" specified by you.</p> | <ul style="list-style-type: none"> AutoCAD will ask you for cutting edge. Once the cutting edge is selected it will ask you for object that you want to get rid off. Select the object you will get rid off it. |
| <p>27. Extend</p>  | <p>Extends the length of a line, arc, or polyline to meet a specified "boundary"</p> | <p>----</p> |
| <p>28. Break</p>  | <p>Erases part of a line, arc or circle, or splits it into two lines or arcs</p> | <p>-----</p> |
| <p>29. Chamfer</p>  | <p>Creates a chamfer (a angled line connection) at the intersection of two lines</p> | <p>D Sets chamfer distances P Chamfers an entire polyline</p> |
| <p>30. Fillet</p>  | <p>Constructs an arc of specified radius between two lines, arcs, circles, or will create arcs of the specified radius at the vertices of a polyline. Radius of the arc to be constructed may be set to 0, which will make a sharp corner</p> | <p>P Fillets an entire Polyline at the vertices R Allows setting of the fillet radius. Default value is 0. Radius remains set until changed again</p> |
| <p>31. Explode</p>  | <p>Separates a block, dimension or hatch pattern into its constituent entities or makes a polyline into a</p> | <p>-----</p> |







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| | series of straight lines | |
| <p>32. Hatch</p>  | Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill. | <pattern-name> uses hatch pattern name from library file to fill designated area with a pseudo- block hatch entity -- can be reduced to individual lines and points by Exploding it. |
| <p>33. Pedit</p>  | Allows editing of polylines which are already drawn | <p>C Closes an open polyline</p> <p>D curves, or returns a Spline curve to its control frame or series of connected straight lines</p> <p>E Edit vertices</p> <p>F Fits curve to a polyline -- makes a series of straight lines into a curve which will pass through the vertices</p> <p>J Joins a line or arc or another polyline to an open polyline</p> <p>L Toggles line type generation to be either a continuous pattern of dashes passing through the vertices, or a pattern which starts and ends at each vertex</p> <p>O Opens a closed polyline</p> <p>S Uses the polyline vertices as a frame for a Spline Curve -- type of Spline Curve is set by the Variable "Spline type."</p> |







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| <p>34. Osnap</p>  | <p>Enables points to be precisely located on reference points of existing objects. This is the so-called "Running Mode" of OSNAP, which sets selection method to run continuously until set to NONE (none) or until overridden by selecting another "Interrupt Mode" OSNAP method from the cursor menu. Combinations of OSNAP methods can be used by selecting a series of options separated by commas. For instance, if you want to always pick either endpoints or intersection points when locating endpoints of lines, you would issue the command as follows: OSNAP <RET> END, INT <RET></p> | <p>can be transparent CEN Centre of arc or circle END closest Endpoint of arc or line INS Insertion point of Text or Block INT Intersection of line, arc, or circle MID Midpoint of line, arc, rectangle side, or polygon side NEA Nearest point selected by aperture on line, polyline, arc, or circle NOD Node (another name for a Point) NON None -- used when a "Running OSNAP" is on to temporarily turn off OSNAP selection PER Perpendicular point to line, arc or circle -- when used with an arc or circle it will draw a line to the surface of the arc or circle heading toward the center point QUA Quadrant point of arc or circle (top, bottom, right or left side) QUI Quick mode -- this is a modifier to one of the other OSNAP options -- it will find the first point that meets the requirements, not necessarily the closest point to the aperture. TAN Tangent point to arc or circle</p> |
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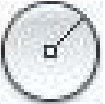


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| <p>35. Ortho</p>  | <p>Constrains drawing so that only lines aligned with the grid can be drawn, usually means only horizontal or vertical lines, however, if the crosshairs are rotated through the "Snap" "Rotate" command sequence, the lines drawn are constrained to being parallel with the crosshair rotation. Constraint can be overridden by snapping to a point or by entering exact coordinates for endpoints</p> | <p>-----</p> |
| <p>36. Measure</p>  | <p>Places points (or, optionally, Blocks) at intervals along a selected line, polyline, arc or circle. The interval distance is given by the user. If points are used as the marker to be placed along the entity, they are not visible unless the Point type is set to type 3 with the PDMODE command.</p> | <p>B specifies that a defined Block is to be used as a marker instead of a point.</p> |
| <p>37. Scale</p>  | <p>Alters the size of existing objects</p> | <p>R Resizes with respect to a reference size</p> |
| <p>38. Text</p>  | <p>Draws text characters of any size with selected styles</p> | <ul style="list-style-type: none">• Text is very important in precision drawings such as those produced by AUTOCAD. Text is an object on drawing, just like line or circle. This is open to same editing commands like scale, move and erase. AutoCAD gives you two options• Single line.• Multiline or |



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| 39. Divide | Places points along a line, polyline, arc, or circle, dividing it into the specified number of equal parts | <ul style="list-style-type: none">• Select the object.• AutoCAD will ask you to specify the number of divisions or parts. Enter it you get that object divided into specified number of segments. |
| 40. Leader  | Draws a line or series of lines with an arrow head (commonly called a "leader") to point to an object to notate it. This command will also prompt you for the note at the end of the leader line, but it will allow only one line of text. The most useful method of using this command is to simply draw leaders between the object to be notated and text that you create with the DTEXT command | ----- |
| 41. Linear dimension  | Draws the dimension of a line linearly | ----- |
| 42. Aligned dimension  | Draws a linear dimension with the dimension line parallel to the selected dimension origin points. This lets you align a dimension with an angled line that is not either exactly horizontal or vertical | ----- |
| 43. Angular  | Draws an arc and calculates the angle between two non-parallel lines, and draws the text and arrowheads. Arrowheads are the standard AutoCAD filled triangles, regardless of your defined Arrow Block | ----- |



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| <p>44. Base-line dimension</p>  | <p>Continues a linear dimension from the baseline (the so-called "first extension line") of the previous or selected dimension. This is not frequently used in architectural dimensions</p> | <p>-----</p> |
| <p>45. Centre</p>  | <p>Draws a pair of crossed lines at the center of a circle or arc on the current layer</p> | <p>-----</p> |
| <p>46. Diameter</p>  | <p>Draws a dimension through the center of a circle or arc, calculating the diameter, with one of the arrow heads located at the point on the circle or arc which is picked</p> | <p>-----</p> |
| <p>47. Radius</p>  | <p>Draws the radius dimension of an arc or circle from the center to the point on its locus where the arc or circle was selected.</p> | <p>-----</p> |