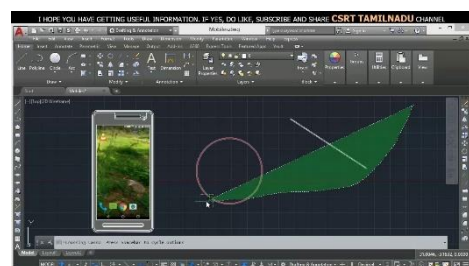
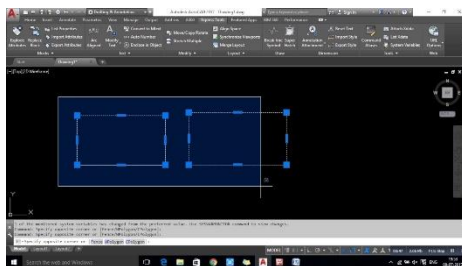


CAD Notes: The commands will be typed on the Command line located near the bottom of the workspace, and then press the Spacebar or Enter. You can also use the right-click key on the mouse to enter a command. If you right-click and go down to Recent Input, you can activate a previous command used.

1. **Line (L)** – This command is used to draw simple, single lines. Type (L) > Enter > Click once on the screen to set the first point > Then click a second time in order to draw the line segment. If you know the length of the line, then point in the direction you desire the line segment to be drawn, and type the length of the line.
2. **Polyline (PL)** – This command is used to draw line and arc segments. The difference is that the Polyline can be a continuous line segment made up of several line segments, arcs, or a mixture of the two. Type (PL) > Enter > Click once on the screen to set the first point > Then begin drawing the line. If you desire to draw an arc, type (A), then press enter. If you desire to return to drawing straight lines, type (L) then press enter. If you wish to close the polyline, then type the words (close) and you will close the polyline command with a final segment.
3. **Arc (A)** – This command is used to draw an arc. Type (A) > Enter > Click once on the screen to set the initial base point of the arc. If you know the radius of the arc, click on the word [Center] along the command bar > Type in the Radius of the arc > Then click a final time on the screen.
4. **Circle (C)** – This command is used to draw circles. Type (C) > Enter > Click on the screen to set the center of the circle > Then type the radius of the circle.
5. **Rectangle (Rec)** – This command is used to draw rectangles or squares. Type (REC) > Click once on the screen to set the base point of the rectangle > Type the “@” symbol, followed by the x (width) and y (height) coordinates of the rectangle. For example, if the rectangle is 100’-0”x250’-0”, then you would type @100’,250’, then press enter.
6. **Move (M)** – This command is used to move objects. Type (M) > Enter > Select the object you desire to move by its endpoint, midpoint, or center > Point the object in the direction you desire to move it, then type the distance. Or snap the object you’ve selected by its endpoint, midpoint, or center, to the endpoint, midpoint, center, or perpendicular of another object.
7. **Extend (EX)** – This command is used to lengthen the line segment to meet the boundary of another segment or object. Type (EX) > Enter > Select all objects > Enter > Click on the ends of the segments that you wish to extend.
8. **Trim (TR)** – This command is used to trim a segment or portion of an object that overlaps another. Type (TR) > Enter > Select all of the objects you desire to trim, and every object that touches the segments you desire to trim > Enter > Now click only on the segments or portions that you wish to trim away.
9. **Erase (E)** – This command is used to erase segments or objects drawn in AutoCAD. Type (E) > Enter > Select the objects you desire to erase > Press Enter. If you click to select dragging from left to right, you will see a Blue Window. Everything that is contained within the window will only be affected and selected. If you click to select dragging from right to left, you will see a Green Fence. Everything that the fence touches will be affected and selected.



10. **Offset (O)** – This command is used to offset a line segment or an object a set distance from its original position, without deleting the original object. Type (O) > Enter > Type the distance that you desire to offset the object > Click the segment or the object > Then click the side in which you desire to offset the object from its origin.
11. **Stretch (S)** – This command is used to stretch the size or location of a segment or object. Type (S) > Enter > Using the fence selection method (green) from note 9, select the portion of the object you wish to stretch, point it in the direction you desire to stretch the object or segment > Type the numerical distance and press Enter.
12. **Fillet** – This command is used to join a vertical and horizontal segment together using a radius. Type (Fillet) > Enter > Type the letter R > Enter > Enter the radius you desire to connect the two segments with > Enter > Click segment 1, then click segment 2.
13. **Mirror (MI)** – This command is used to create an identical copy of a segment or an object, by using a horizontal or a vertical axis point. Type (MI) > Enter > Select the object or segment > Enter > Click the first endpoint of your axis, then click the second endpoint of your axis line > Type Y for Yes if you desire to erase the original object or N for No on the command line if you desire to keep the original object > Enter.
14. **Array (AR)** – This command is used to make an identical copy of line segments or objects, but be able to control the number of copies and distance between the objects, horizontally and vertically. There are three methods that you can array an object: Rectangular, Path or Polar. Type (AR) > Enter > Select the segment or the object > Enter > Select the method you desire to array: Rectangular, Path or Polar > Modify the number and distances.
15. **Hatch (H)** – This command is used to hatch an object with a closed boundary. Type (H) > Enter > There are two methods to hatch an object: Pick points or Select the Object. Choose which method best fits your needs and begin hatching the object. A suggestion is to use the solid hatch initially and edit the hatch after the selection is hatched.
16. **Hatch Edit (HE)** – This command is used to modify a hatch by changing the scale of the hatch, the pattern, color or direction of the hatch. Type (HE) > Enter > Click on the hatched area > The Hatch Edit dialogue box will appear > Make the modifications necessary and click OK.
17. **RegenAuto (REA)** – This command is used to refresh the AutoCAD screen and correct any glitches that might affect arc, circles and lines from showing the proper way.
18. **Polygon** – This command is used to draw multi-sided objects or polygons. Type (POLYGON) > Enter > Type the number of sides > Enter > Click on the screen to set the origin of the object > Enter > Type the radius > Enter.
19. **Rotate (RO)** – This command is used to rotate segments or objects based on the number of degrees you desire to have it rotated. Type (RO) > Enter > Select the object > Enter > Click the anchor point of the segment or the object in which you desire to have the rotation made from > Type the degree you desire to have the object rotated. Negative degrees if you desire to rotate the object clockwise. Positive degrees if you desire to rotate the object counterclockwise.
20. **Zoom (Z)** – This command is used to zoom into any area of your drawing. You can use the scroll wheel on your mouse in order to do this as well. Type (Z) > Enter > Type (E) and press Enter if you desire to Zoom the extents of what is drawn on the screen. Type (W) and press Enter if you desire to Zoom using a window, in order to zoom into a section of your drawing screen.
21. **Distance (DI)** – This command is used to check the length, width and height of an object or line segment. Type (DI) > Enter > Click on the endpoint on a segment, then click another point and then read the command line for the Distance of the object.