

14.9.3 Dividing an Object into Equal Segments

Divide Command

- Divides an entity into equal segments
 - *Select entity to divide*
 - *Enter the number of equal segments desired*
 - *Set Point style to another style other than default dot*
 - *Use OSNAP node to select points*

14.9.4 Setting off Equal Distances

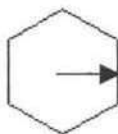
Measure command

- Locates points based on the input distance
 - *Points are started from the closest endpoint that is used to select the line*
 - *Set Point style to another style other than default dot*
 - *Use OSNAP node to select points*

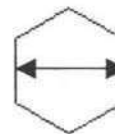
14.9.5 Polyline Command

Creates a multisided closed shape

- Located by the center and radius
or
- Located by edges of polygon



Center - radius



Edge points

Fig. 14.13

- Polygon may be inscribed or circumscribed about a circle (Remember inscribed is inside!)
- (a) Circumscribed Polygon is outside like a bolt head radius is to the middle of segment
- (b) Inscribed Polygon is inside radius is to the corner

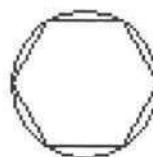
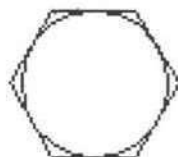


Fig. 14.14

14.9.6 Ray Command

- Creates a line that is fixed on one end and infinite in one direction
 - When a ray is broken, one segment turns into a line and the other remains a Ray, which is infinite in one direction

14.9.7 Rectangle Command

- Creates a rectangular object based on 2 opposite corner points
- Use relative coordinates: @dist<ang or @X,Y for second point
 - Rectangle options
 - Fillet - fillets the corners based on input radius
 - Chamfer - chamfers the ends based on chamfer distances
 - Width - changes width of lines

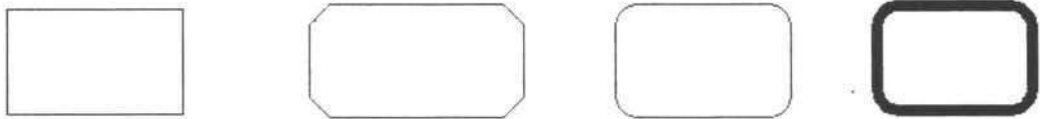


Fig. 14.15

14.9.8 Arc Command

- Arcs are created in the counter-clockwise rotation
- The 3 point Arc may be created in either direction

Arc	3 Points
Circle	Start, Center, End
Donut	Start, Center, Angle
Spline	Start, Center, Length
Ellipse	Start, End, Angle
Block	Start, End, Direction
Point	Start, End, Radius
Hatch...	Center, Start, End
Boundary...	Center, Start, Angle
Region	Center, Start, Length
Text	Continue

Fig. 14.16

3 point Arc

- 3 points are selected on the circumference of the circle



Fig. 14.17

Start, Center, End

Selection begins with a Start point on the circumference.

- Then select the center point of the arc
- Finally, select the End point on the circumference counter-clockwise from the start point

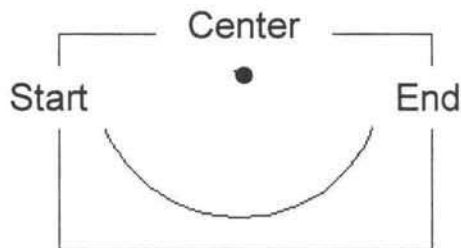


Fig. 14.18

Start, Center, Angle

- Select the Start point and Center locations.
- Type the value for the Angle in for counter-clockwise arc creation

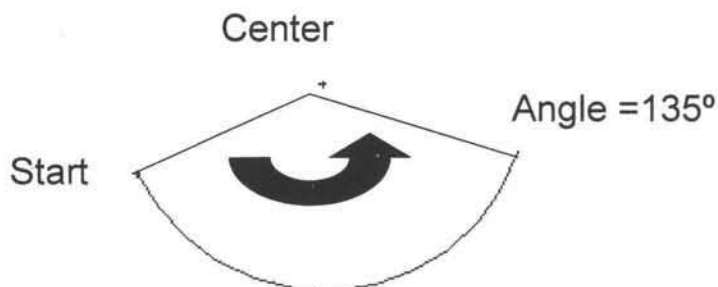


Fig. 14.19

Start, Center, Length of Chord

- Select the Start and Center points
- The length of chord is a linear distance from the Start point to the End point based on the arc's radius

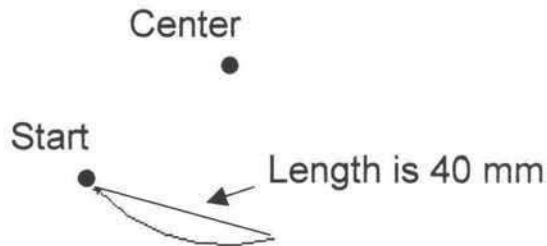


Fig. 14.20

Arc Command**Start, End, Angle**

- Select the Start Point and then the End point of the Arc
- The Angle value is then input for counter-clockwise Arc

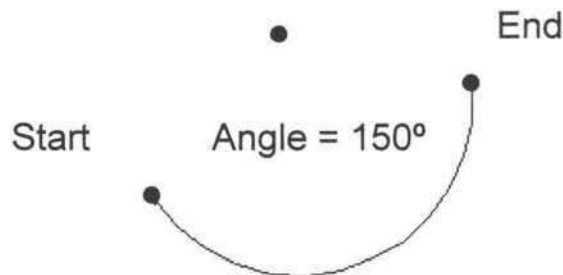


Fig. 14.21

Start, End, Direction

- Select Start and End Points
- Drag for the direction of Arc
 - Caution : Radius of Arc is not known while dragging

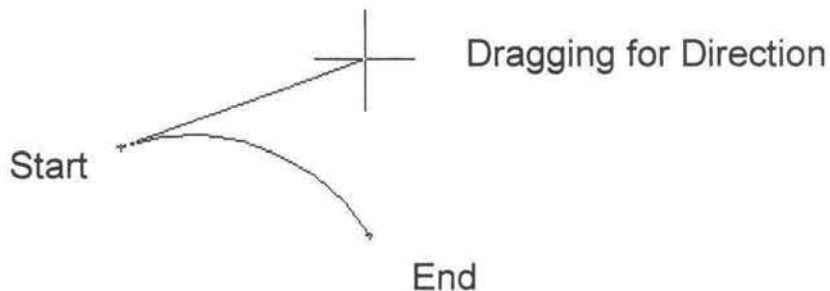


Fig. 14.22

Start, End, Radius

- Select Start point and End point of Arc
- Radius is then input
 - $+ \text{radius} = \text{Minor Arc}$
 - $- \text{radius} = \text{Major Arc}$

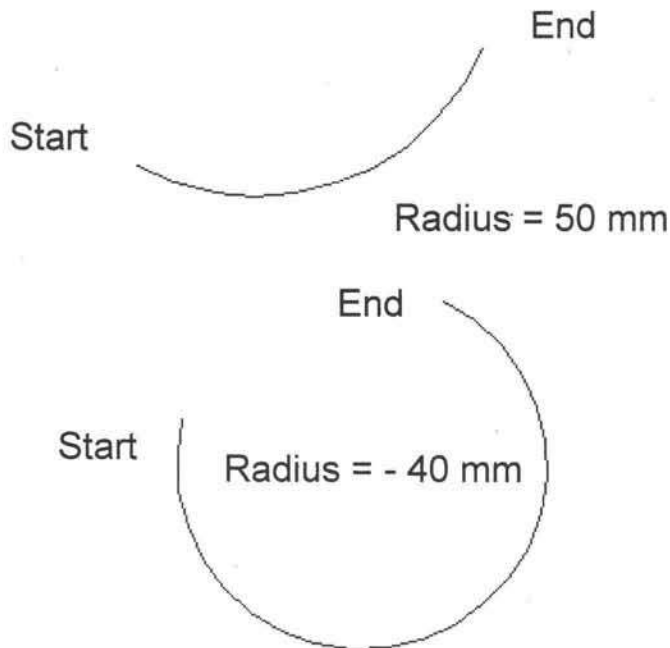


Fig. 14.23

14.9.9 Circle Command

Activate from Draw pull-down

Multiple options to create a circle

Center, Radius

Center, Diameter

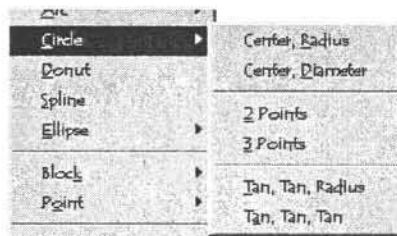


Fig. 14.24

Other options

(2 point, 3 point, Tangent) covered in Geometric Constructions Unit

A minimum of 2 points needed to create circle.

Select the center point

Type the Radius or Diameter distance at the next prompt

2 point & 3 point (**Fig. 14.25**)

- Create a circle by selecting 2 or 3 points on the circumference of the circle.



Fig. 14.25

14.9.10 Ellipse Command

- Center and Radius
- Create by selecting the center, then the 2 radius distance for the major and minor Axis
- Axis Endpoints
- Select the endpoints of the major axis first, and then select the radius distance for the minor axis

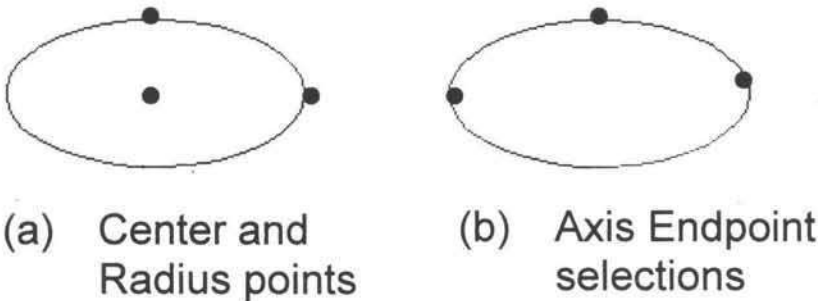


Fig. 14.26

Rotated Ellipse (Fig. 14.27)

- Created by selecting endpoints
- The first axis defined is now used as an axis of rotation that rotates the ellipse into a third dimension

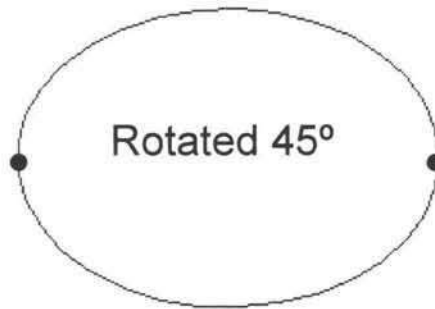


Fig. 14.27

14.10 The Drawing Tools of CADD

The following are the basic drawing tools found in a CADD program:

- Line types
- Multiple parallel lines
- Flexible curves
- Arcs and circles
- Ellipses and elliptical arcs
- Text
- Dimensions
- Hatch patterns
- Polygons
- Arrows

14.10.1 Using Line Types

There are a number of line types available in CADD that can be used to enhance drawings. There are continuous lines, dotted lines, center lines, construction lines, etc. CADD enables you to follow both geometrical and engineering drawing standards. You can use line types to represent different annotations in a drawing. For example, an engineer can use line types to differentiate between engineering services in a building plan. One line type can be used to show power supply lines, while the others to show telephone lines, water supply lines and plumbing lines.

The drawing tools

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CADD is preset to draw continuous lines. When you enter the line command and indicate a starting point and end point, a continuous line is drawn. If you want to draw with another line type, you need to set that line type as the current line type. Thereafter, all the lines are drawn with the newly selected line type.

14.10.2 Drawing Multiple Parallel Lines

CADD allows you draw parallel lines simultaneously just by indicating a starting point and an end point. These lines can be used to draw something with heavy lines or double lines. For example, they can be used to draw the walls of a building plan, roads of a site map, or for any other presentation that requires parallel lines.

Most programs allow you to define a style for multiple parallel lines. You can specify how many parallel lines you need, at what distance and if they are to be filled with a pattern or solid fill. A number of add-on programs use multiple lines to represent specific drawing features. For example, an architectural program has a special function called “wall”. When you use this option, it automatically draws parallel lines representing walls of specified style and thickness.

Note:

Multiple lines are a unified entity. Even though double lines are drawn, they are treated as one line. You cannot erase or edit one line separately. However, there are functions available that can break the entities apart.

14.10.3 Drawing Flexible Curves

CADD allows you to draw flexible curves (often called splines) that can be used to draw almost any shape. They can be used to create the smooth curves of a sculpture, contours of a landscape plan or roads and boundaries of a map. To draw a flexible curve, you need to indicate the points through which the curve will pass. A uniform curve is drawn passing through the indicated points. The sharpness of the curves, the roughness of the lines and the thickness can be controlled through the use of related commands.

Drawing Arcs and Circles

CADD provides many ways to draw arcs and circles. There are a number of advanced techniques available for drawing arcs and circles, which can simplify many geometrical drawing problems. You can draw an arc by specifying circumference and radius, radius and rotation angle, chord length and radius, etc.

Arcs are drawn so accurately that a number of engineering problems can be solved graphically rather than mathematically. Suppose you need to measure the circumference of an arc, just select that arc and the exact value is displayed.

The following are basic methods for drawing arcs and circles:

(These are essentially the same methods you learn in a geometry class.

However, when drawing with CADD the approach is a little different.)

- Center point and radius
- 3 points
- Angle and radius

- 2 points
- 2 tangents and a point
- 3 tangents

14.10.4 Drawing Ellipses and Elliptical Arcs

Ellipses are much easier to draw with CADD than on a drawing board. On a drawing board, you need to find the right size template or draw a series of arcs individually to draw an ellipse. With CADD, all you need to do is specify the size of the ellipse.

The following are two basic methods for drawing ellipses:

- Length and width
- Axis and rotation angle

Note: The above topics are illustrated with Figs. in CADD PRIMER.

Adding Text to Drawings

CADD allows you to add fine lettering to your drawings. You can use text to write notes, specifications and to describe the components of a drawing. Text created with CADD is neat, stylish and can be easily edited. Typing skills are helpful if you intend to write a lot of text.

Writing text with CADD is as simple as typing it on the keyboard. You can locate it anywhere on the drawing, write it as big or as small as you like and choose from a number of available fonts.

Note:

The drawing tools

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When large amounts of text are added to drawings, it slows down the screen displays. Many programs provide options to temporarily turn off text or to display text outlines only. This feature helps save computer memory and speeds up the display of screen images. The text can be turned back on whenever needed.

The following are the basic factors that control the appearance of text:

(The exact terms and procedures used vary from one program to another.)

- Text height
- Height to width ratio and inclination of letters
- Special effects
- Alignment of text (justification)
- Text fonts

Note: The above topics are illustrated with Figs. in CADD PRIMER.

Defining a Text Style

As discussed, there are a number of factors that control the appearance of text. It is time-consuming to specify every parameter each time you need to write text. CADD allows you to define text styles that contain all the text information such as size, justification and font. When you need to write text, simply select a particular style and all the text thereafter is written with that style. CADD offers a number of ready-made text styles as well.

Important Tip

There are a number of add-on programs available that can make working with text faster and easier. These programs provide basic word-processing capabilities that can be used to write reports and make charts. They provide access to a dictionary and thesaurus database that can be used to check spelling and to search for alternative words.

Drawing Dimensions

CADD's dimensioning functions provide a fast and accurate means for drawing dimensions. To draw a dimension, all you need to do is to indicate the points that need to be dimensioned. CADD automatically calculates the dimension value and draws all the necessary annotations.

The annotations that form a dimension are: dimension line, dimension text, dimension terminators and extension lines (see fig.). You can control the appearance of each of these elements by changing the dimensioning defaults.

The following are the common methods for drawing dimensions:

- Drawing horizontal and vertical dimensions
- Dimensioning from a base line
- Dimensioning arcs and circles

CADD PRIMER: <http://www.caddprimer.com>

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- Drawing dimensions parallel to an object
- Dimensioning angles

Note: The above topics are illustrated with Figs. in CADD PRIMER.

Adding Hatch Patterns to Drawings

The look of CADD drawings can be enhanced with the hatch patterns available in CADD. The patterns can be used to emphasize portions of the drawing and to represent various materials, finishes, and spaces. Several ready-made patterns are available in CADD that can be instantly added to drawings.

Hatch patterns are quite easy to draw. You don't need to draw each element of a pattern one by one. You just need to specify an area where the pattern is to be drawn by selecting all the

drawing objects that surround the area. The selected objects must enclose the area completely, like a closed polygon. When the area is enclosed, a list of available patterns is displayed. Select a pattern, and the specified area is filled. (Illustrated with Fig. In CADD PRIMER)

Drawing Symbols

Symbols provide a convenient way to draw geometrical shapes. You may compare this function with the multi-purpose templates commonly used on a drawing board. To draw a geometrical shape, such as a pentagon or hexagon, select an appropriate symbol from the menu, specify the size of the symbol, and it is drawn at the indicated point. (Illustrated with Fig. In CADD PRIMER)

Drawing Arrows

Arrows (or pointers) in a drawing are commonly used to indicate which note or specification relates to which portion of the drawing, or to specify a direction for any reason. There are several arrow styles available in CADD programs. You can choose from simple two-point arrows to arrows passing through a number of points, and from simple to fancy arrow styles. To draw an arrow, you need to indicate the points through which the arrow will pass. (Illustrated with Fig)

The Command Line Box

- There are two basic ways to input a command:
 - The command line
 - Clicking on a command icon. The command icons will execute the appropriate text based command on the command line
- Additionally some commands have a keyboard shortcut option that normally involves the “cntrl” or “alt” keys on the keyboard
- The command line box size can be changed to show more or fewer command lines

Basic Commands

- The “Draw” Toolbar
 - Lines
 - Polylines
 - Circles
 - Arcs
- The “Modify” Toolbar
 - Erase
 - Copy
 - Move
 - Offset
 - Fillet

- Array
- Trim
- Extend

Using the Help function

- AutoCAD has a good command reference in its help function.
- This presentation will not duplicate that reference. You should frequently refer to the command reference as you learn the various commands.
- Some practical pointers are added here that may not be easily encountered in the command reference.

Drawing Lines

- Either type "line" on the command line or click on the line icon in the draw toolbar.
- Lines can be drawn by point and click.
 - Can keep an eye on the coordinates display to make sure that you get what you want.
- Lines can be specified by their end point coordinates.
 - Type in the coordinates on the command line
- Lines can be specified by their first point coordinates, then by an distance and angle.
 - Select starting point, type in "@distance<angle"
 - Example: @5<45 would go 5 units at a 45 degree angle
- Click the line icon (or enter "line" on the command line)
- Draw a horizontal line whose left end coordinate is 0,.5 and is 5 inches long.
- Continue the line so that the second segment starts at the end of the first line and goes vertically up 1.5 inches

The Copy Command

- The copy command is used to make a single duplicate of an entity or group of entities.
- Click on copy icon in the modify menu and follow the instructions on the command line.
- Note that copy offsets may be independent of the actual line

Selecting Objects

- AutoCAD has several ways to select objects.
 - Click on each object that you want to select.
 - Make a window that encloses all the objects that you want to select.
- Click on the lower or upper LEFT corner of desired window area
- Click on the opposite corner of the window area
 - Make a boundary that selects every thing that is within the boundary and that CROSSES the boundary.
- Click on the lower or upper RIGHT corner of desired window area
- Click on the opposite corner of the window area

Methods of selecting draw commands

Pull-down menu

Tool Bar

Command name typed at command line

Command alias typed at command line

C = circle command

L = line command

A = arc command

<u>D</u> raw	<u>D</u> imension	<u>M</u> odify
<u>L</u> ine		
<u>R</u> ay		
<u>C</u> onstruction Line		
<u>M</u> ultiline		
<u>P</u> olyline		
<u>3D</u> Polyline		
<u>P</u> olygon		
<u>R</u> ectangle		
<u>A</u> rc		▶
<u>C</u> ircle		▶
<u>D</u> onut		
<u>S</u> pline		
<u>E</u> llipse		▶
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WORKSHEETS

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