

2. Unit 2 Basic Drawing Commands

1.1. Line command and its options

Create a series of contiguous line segments. Each segment is a line object that can be edited separately.

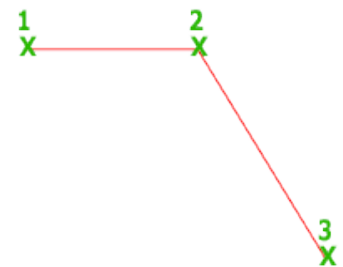
Symbol:



The following prompts are displayed.

Specify first point

Sets the starting point for the line. Click a point location. With object snaps or grid snap turned on, the points will be placed precisely. You can also enter coordinates. If instead, you press Enter at the prompt, a new line starts from the endpoint of the most recently created line, polyline, or arc. If the most recently created object is an arc, its endpoint defines the starting point of the line. The line is tangent to the arc.

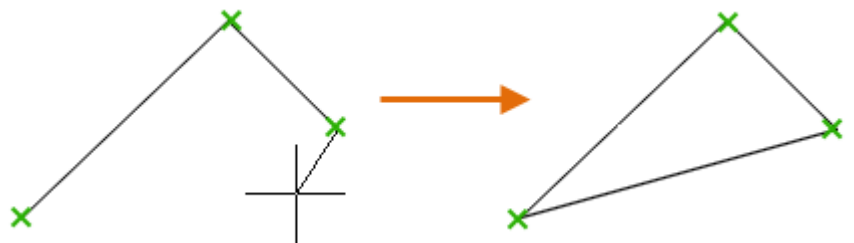


Specify next point

Specifies the endpoint of the line segment. You can also use polar and object snap tracking together with direct distance entry.

Close

Connects the first and last segments.




Undo

Removes the most recent segment of a line sequence.

1.2. Point command

Creates a point object.

Symbol: 

Point objects can act as nodes to which you can snap objects. You can specify 2D and 3D locations for a point. The current elevation is assumed if you omit the Z coordinate value. The PDMODE and PDSIZE system variables control the appearance of point objects.

You can use MEASURE and DIVIDE to create points along an object. Use PTYPE to specify point size and styles.

The following prompt is displayed.

Specify a point

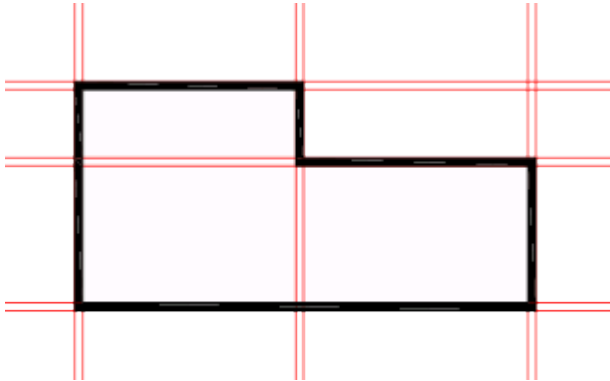
Specifies the location of the point object.

1.3. Xline command and its options

Creates a construction line of infinite length.



Xlines are useful for creating construction and reference lines, and for trimming boundaries.

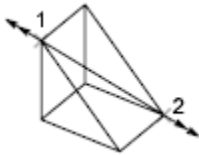


The following prompts are displayed.

Point

Specifies the location of the infinite line using two points through which it passes.

The xline is created through the specified point.



Hor

Creates a horizontal xline passing through a specified point.

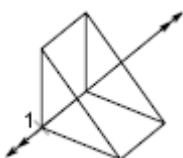
The xline is created parallel to the X axis.



Ver

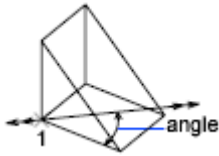
Creates a vertical xline passing through a specified point.

The xline is created parallel to the Y axis.



Ang

Creates an xline at a specified angle.



Angle of Xline

Specifies the angle at which to place the line.

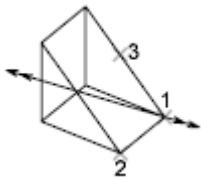
Reference

Specifies the angle from a selected reference line. The angle is measured counterclockwise from the reference line.

Bisect

Creates an xline that passes through the selected angle vertex and bisects the angle between the first and second line.

The xline lies in the plane determined by the three points.



Offset

Creates an xline parallel to another object.

Offset Distance

Specifies the distance the xline is offset from the selected object.

Through

Creates an xline offset from a line and passing through a specified point.

NOTE:

AutoCAD has two commands to draw lines of infinite length that can work as construction references for drawings. They are very useful to draw elevations or sections based on a plan. They are both very similar.

XLINE: Draws a line of infinite length. We can draw it free or set it to be horizontal, vertical, follow a certain angle, be the bisector of an angle or be parallel to certain line. If we draw the line directly after entering the command, we will draw it freely, if we check the command line, we will find the different options mentioned before.

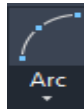
RAY: Is like a simplified version of XLINE. RAY draws an infinite length line, but from a start point (a line with a beginning but without end). It has no options like the ones available for XLINE, simply select a initial point and a second one will define de angle.

1.4. Arc command and its options

The arc is defined as the smooth curve formed by joining two or more points. The arc of a circle is defined as the portion of the circumference of a circle. By default, the arc is drawn counter-clockwise.

Note: We are required to press the Ctrl key to change the direction of the arc in the clockwise.

To create an arc, you can specify combinations of center, endpoint, start point, radius, angle, chord length, and direction values. Arcs are drawn in a counterclockwise direction by default. Hold down the Ctrl key as you drag to draw in a clockwise direction. The icon of Arc on the ribbon panel looks like the below image:



The following are the options available

- **3 points:**
- **Start, Center, End**
- **Start, Center, Angle**
- **Start, Center, Length**
- **Start, End , Angle**
- **Start, End, Radius**
- **Center, Start, End**
- **Center, Start, Angle**
- **Center, Start, Length**
- **Continue**

Start point

Draws an arc using three specified points on the arc's circumference. The first point is the start point (1).

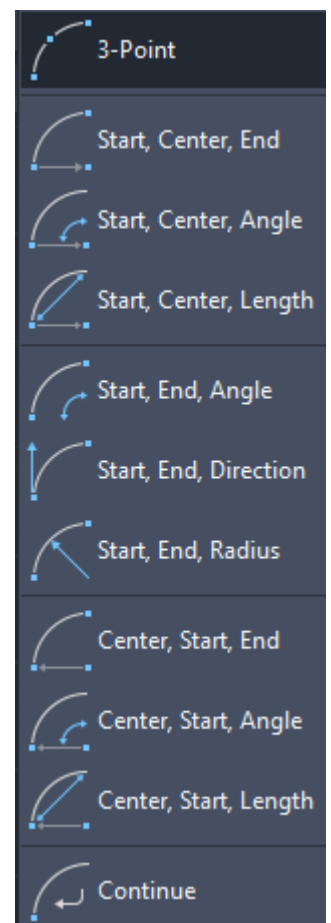
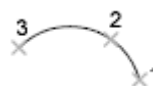
Note: If you press ENTER without specifying a point, the endpoint of the last drawn line or arc is used and you are immediately prompted to specify the endpoint of the new arc. This creates an arc tangent to the last drawn line, arc, or polyline.

Second point

Specify the second point (2) is a point on the circumference of the arc.

End point

Specify the final point (3) on the arc.



You can specify a three-point arc either clockwise or counterclockwise.

Center

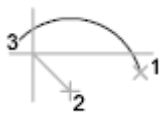
Starts by specifying the center of the circle of which the arc is a part.

Start point

Specify the start point of the arc.

End point

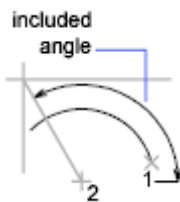
Using the center point (2), draws an arc counterclockwise from the start point (1) to an endpoint that falls on an imaginary ray drawn from the center point through the third point (3).



The arc does not necessarily pass through this third point, as shown in the illustration.

Angle

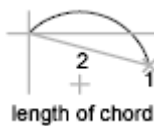
Draws an arc counterclockwise from the start point (1) using a center point (2) with a specified included angle. If the angle is negative, a clockwise arc is drawn.



Chord length

Draws either a minor or a major arc based on the distance of a straight line between the start point and endpoint.

If the chord length is positive, the minor arc is drawn counterclockwise from the start point. If the chord length is negative, the major arc is drawn counterclockwise.

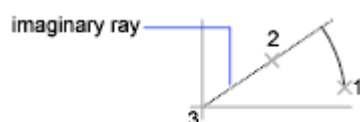


End

Starts by specifying the endpoint of the arc.

Center point

Draws an arc counterclockwise from the start point (1) to an endpoint that falls on an imaginary ray drawn from the center point (3) through the second point specified (2).



Angle

Draws an arc counterclockwise from the start point (1) to an endpoint (2), with a specified included angle. If the angle is negative, a clockwise arc is drawn.

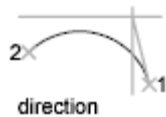
Included angle

Enter an angle in degrees or specify an angle by moving the pointing device counterclockwise.



Direction

Begins the arc tangent to a specified direction. It creates any arc, major or minor, clockwise or counterclockwise, beginning with the start point (1), and ending at an endpoint (2). The direction is determined from the start point.

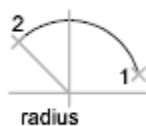


Radius

Draws the minor arc counterclockwise from the start point (1) to the endpoint (2). If the radius is negative, the major arc is drawn.

Center

Specifies the center of the circle of which the arc is a part.

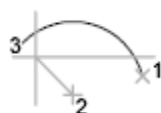


Start point

Specify start point of arc.

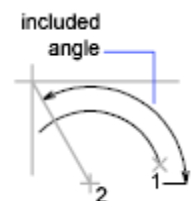
End point

Draws an arc counterclockwise from the start point (1) to an endpoint that falls on an imaginary ray drawn from the center point (2) through a specified point (3).



Angle

Draws an arc counterclockwise from the start point (1) using a center point (2) with a specified included angle. If the angle is negative, a clockwise arc is drawn.

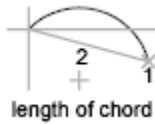


Chord length

Draws either a minor or a major arc based on the distance of a straight line between the start point and endpoint.

If the chord length is positive, the minor arc is drawn counterclockwise from the start point.

If the chord length is negative, the major arc is drawn counterclockwise.



Tangent to last line, arc, or polyline

Draws an arc tangent to the last line, arc, or polyline drawn when you press ENTER at the first prompt.



End point of arc

Specify a point (1).

1.5. Circle command and its options

Creates a circle.



Find

The following prompts are displayed.

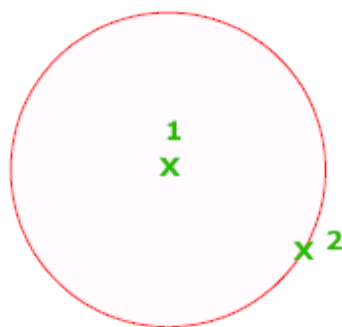
Center Point

Creates a circle based on a center point and a radius or diameter value.

Radius

Enter a value, or specify a point.

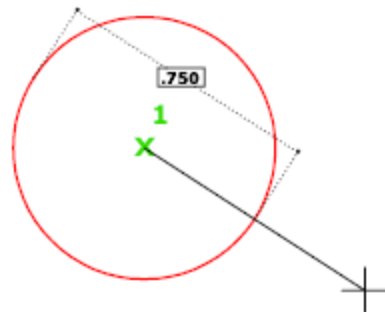
For example:



Diameter

Enter a value, or specify a second point.

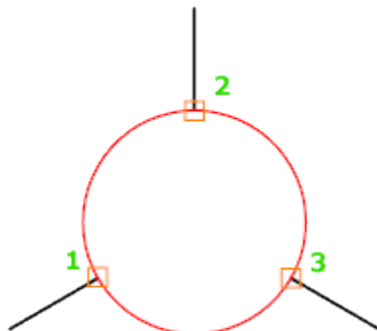
For example:



3P (Three Points)

Creates a circle based on three points on the circumference.

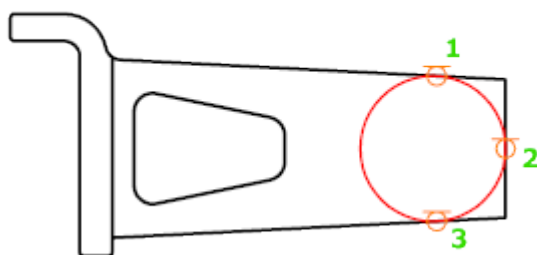
For example:



Tan, Tan, Tan

Creates a circle tangent to three objects.

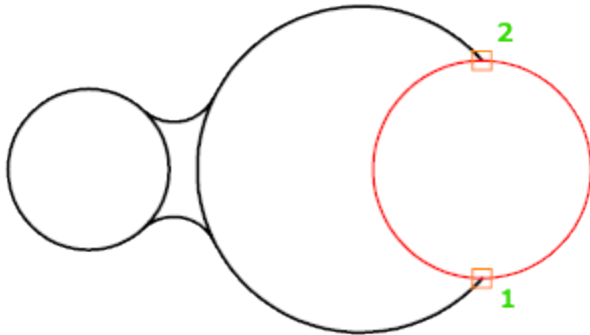
For example:



2P (Two Points)

Creates a circle based on two endpoints of the diameter.

For example:

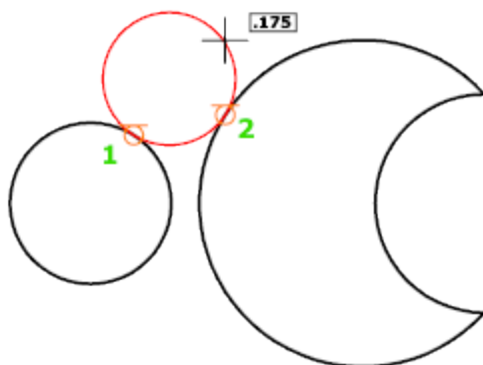


TTR (Tangent, Tangent, Radius)


Creates a circle with a specified radius and tangent to two objects.

Sometimes more than one circle matches the specified criteria. The program draws the circle of the specified radius whose tangent points are closest to the selected points.

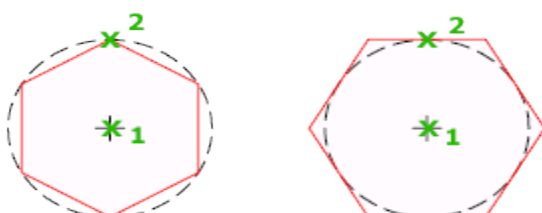
For example:



1.6. Polygon command and its options

Creates an equilateral closed polyline. Find 

You specify the number of sides of the polygon and whether it is inscribed or circumscribed.

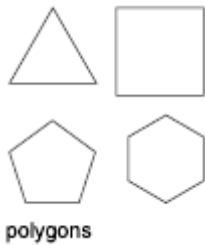


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The following prompts are displayed.

Number of sides

Specifies the number of sides in the polygon (3-1024).

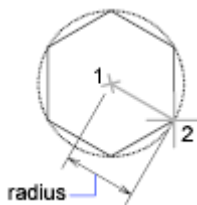


Center of polygon

Specifies the location of the center of the polygon and whether the new object is inscribed or circumscribed.

Inscribed in circle

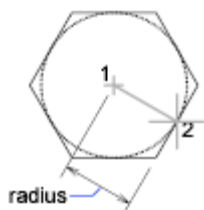
Specifies the radius of a circle on which all vertices of the polygon lie.



Specifying the radius with your pointing device determines the rotation and size of the polygon. Specifying the radius with a value draws the bottom edge of the polygon at the current snap rotation angle.

Circumscribed about circle

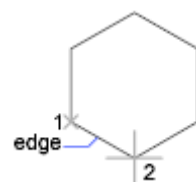
Specifies the distance from the center of the polygon to the midpoints of the edges of the polygon.



Specifying the radius with your pointing device determines the rotation and size of the polygon. Specifying the radius with a value draws the bottom edge of the polygon at the current snap rotation angle.

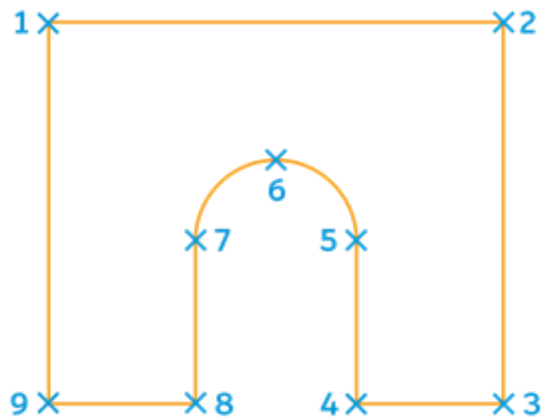
Edge

Defines a polygon by specifying the endpoints of the first edge.



1.7. Pline command and its options

Creates a 2D polyline, a single object that is composed of line and arc segments.



The PLINE command differs from the LINE command in that the segments of the PLINE are connected. When using the LINE command, each segment is its own object. When using PLINE, all line segments are one object.

To Draw a Polyline

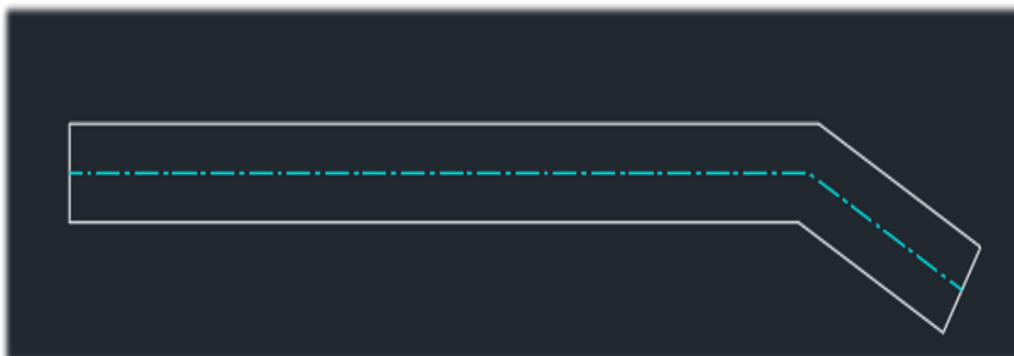
1. Click PLINE.
2. Specify the first point of the polyline.
3. Specify the endpoint of the first segment.
4. Continue specifying segments as needed. Polylines can be composed of straight lines or arcs. You can switch between Line and Arc by choosing the desired option in the command line.
5. Press Enter to end the command, or choose Close Shape to close the polyline to its original start point.

5.1. Mline command and its options

The MLINE command gives you the possibility of drawing parallel lines of pre-defined properties at set distances in one single command.

One of the most common uses for this is to draw a side view of an axle with solid lines defining the extents of the axle and a dash-dot line down the middle representing the center of the axle.

To edit and/or create new Multiline styles type MLSTYLE, here you can define the number of lines, their color, line type and spacing.



Start point

Specifies the next vertex of the multiline.

If you create a multiline with two or more segments, the prompt includes the Close option.



Next point

Draws a multiline segment to the specified point using the current multiline style and continues to prompt for points.



Undo

Undoes the last vertex point on the multiline.

Close

Closes the multiline by joining the last segments with the first segments.



Justification

Determines how the multiline is drawn between the points you specify.

- Top
- Zero
- Bottom

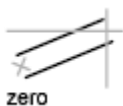
Top

Draws the multiline below the cursor, so that the line with the most positive offset is at the specified points.



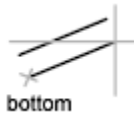
Zero

Draws the multiline with its origin centered at the cursor, so that the MLSTYLE Element Properties offset of 0.0 is at the specified points.



Bottom

Draws the multiline above the cursor, so that the line with the most negative offset is at the specified points.



Scale

Controls the overall width of the multiline. This scale does not affect linetype scale.

The scale factor is based on the width established in the multiline style definition. A scale factor of 2 produces a multiline twice as wide as the style definition. A negative scale factor flips the order of the offset line—the smallest on top when the multiline is drawn from left to right. A negative scale value also alters the scale by the absolute value. A scale factor of 0 collapses the multiline into a single line.



Style

Specifies a style to use for the multiline.

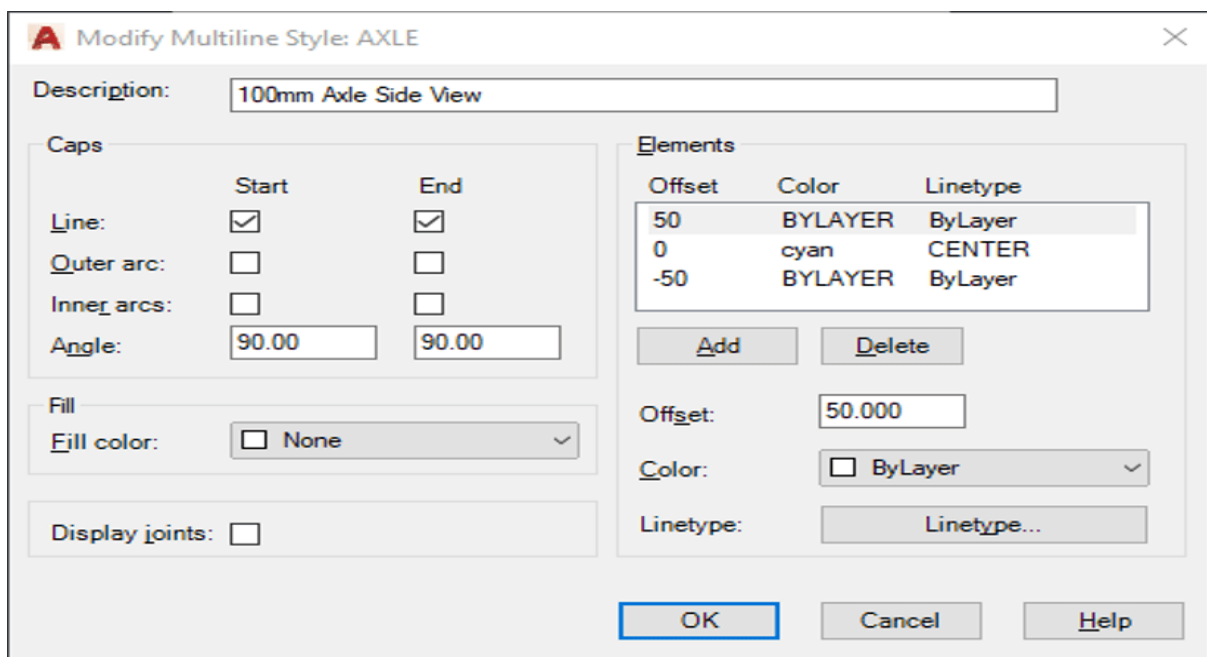
Style name

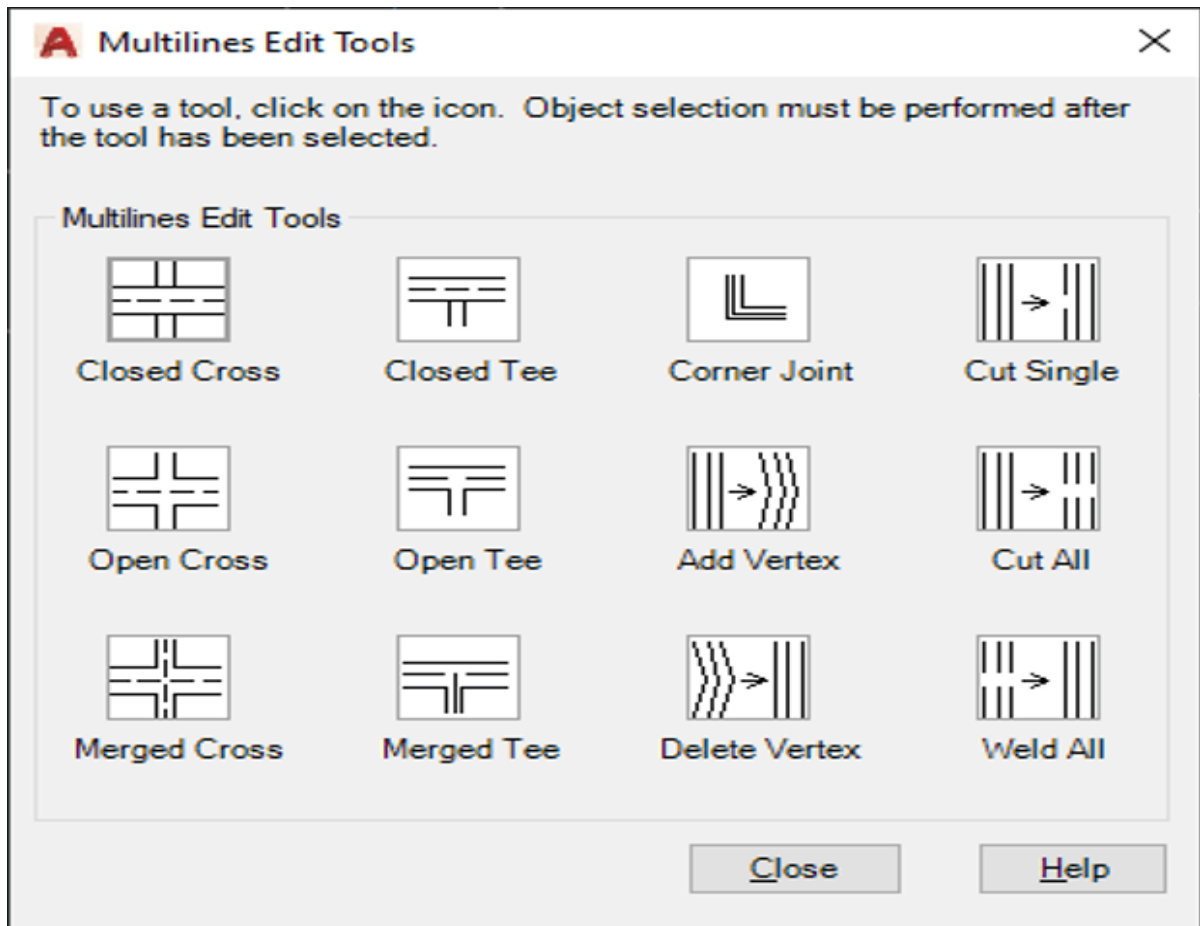
Specifies the name of a style that has already been loaded or that has been defined in a multiline library (MLN) file that you have created.

?—List styles

Lists the loaded multiline styles.

Modify Multiline Style: There are many different options for editing Multiline once placed in the model. The MLEDIT command will open the editing tools available.





5.2. Spline command and its options

The spline command in AutoCAD is used to create a smooth curve, which passes through a set of predefined points.

- It creates a non-uniform curve passing through the points.
- Thus, spline can be created by defining fit points or Control Vertices (CV) points.
- The control vertices define a control frame, which is used to control the shape of the spline, while fit points coincided with the spline.

Example 1: Using Fit points

The steps to create spline using fit points are listed below:

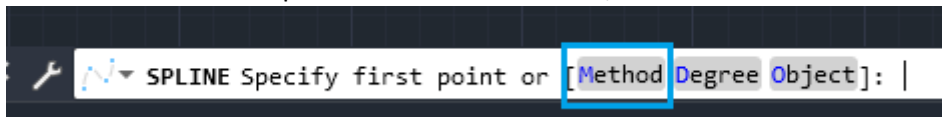
1. Select the Spline icon under the Draw interface from the ribbon panel, as shown below



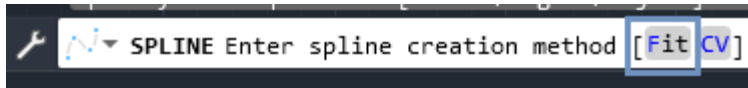
Or

Type SPL on the command line or command prompt and press Enter.

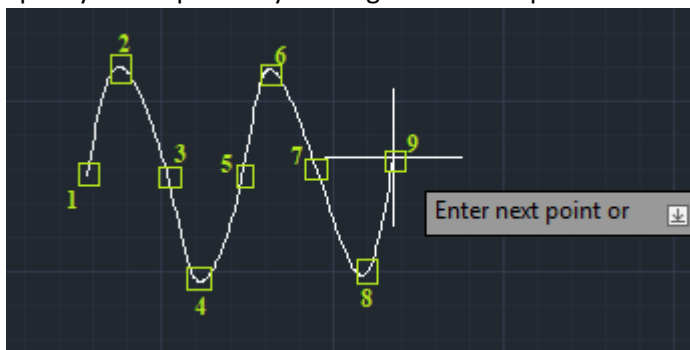
2. Click on the 'Method' option on the command line, as shown below:



3. Click on the 'Fit' option, as shown below:

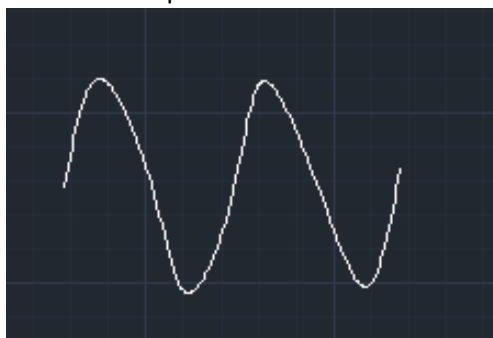


4. Specify the fit points by clicking with the help of the mouse. It is shown below:



Continue specifying points and press Enter or Esc to exit.

The created spline is shown below:



Example 2: Using CV

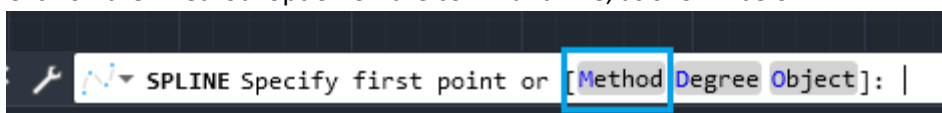
The steps to create spline using Control Vertices (CV) are listed below:

1. Select the Spline icon under the Draw interface from the ribbon panel.

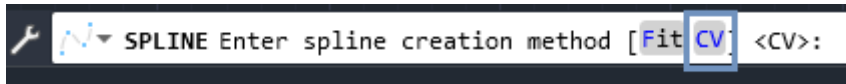
Or

Type SPL on the command line or command prompt and press Enter.

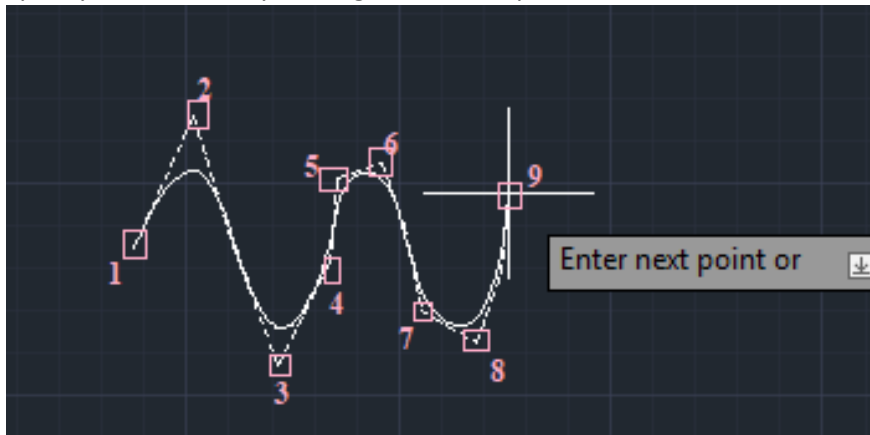
2. Click on the 'Method' option on the command line, as shown below:



3. Click on the 'CV' option, as shown below:

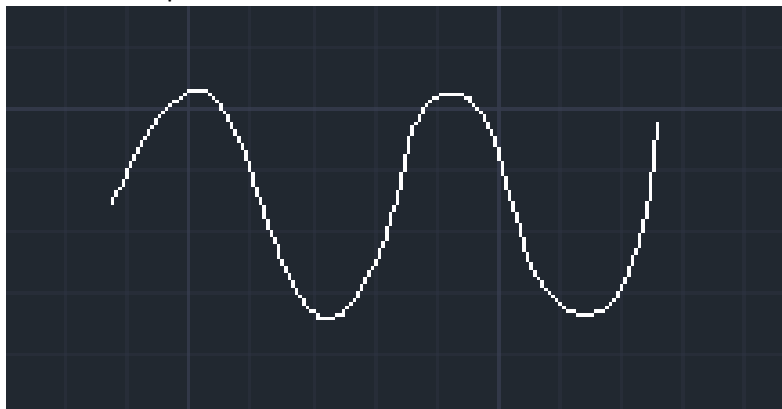


4. Specify the vertices by clicking with the help of the mouse. It is shown below:



Continue specifying points and press Enter or Esc to exit.

The created spline is shown below:



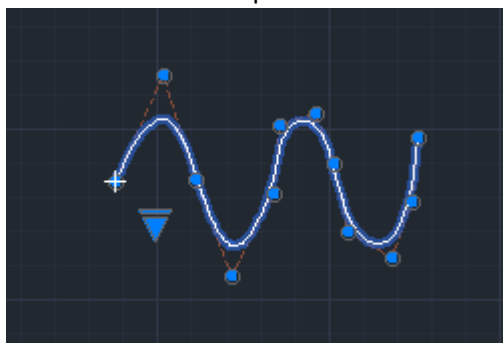
Edit Control Vertices

Let's consider the above spline.

The steps are listed below:

1. Click on the spline.

The vertices on the spline will look like the below image:



2. Click on any particular vertex. The operations such as Stretch, Add, Refine, and Remove can be performed on different vertices, according to the requirements.

3. Press the Ctrl button to switch between different operations.

Spline with degree

We can create a spline with any degree.

The steps to create such spline are listed below:

- a. Select the **Spline** icon under the Draw interface from the **ribbon panel**.

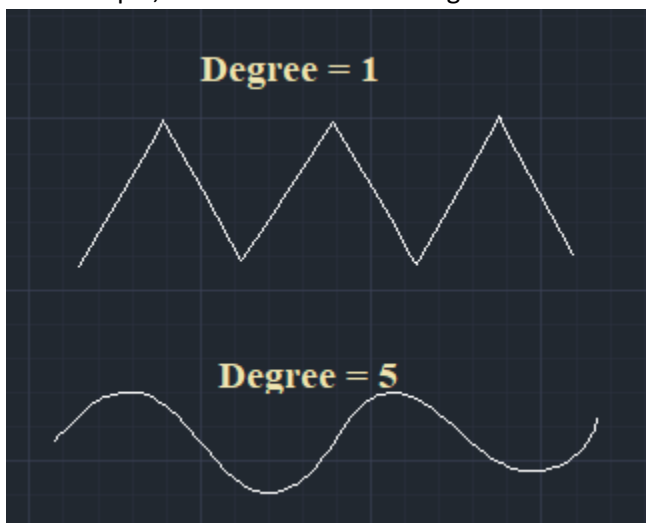
Or

Type SPL on the command line or command prompt and press Enter.

- b. Click on the '**Degree**' option on the command line, as shown below:
- c. Specify the degree of a spline.
- d. Press Enter.
- e. Specify the points by clicking with the help of the cursor.

The spline with a specified degree will be created.

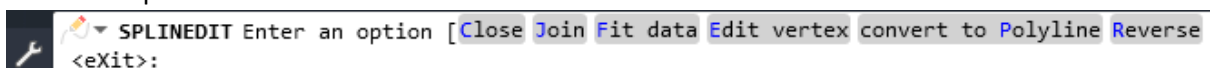
For example, consider the below image:



To convert spline into a polyline

The steps to convert a spline into a polyline are listed below:

1. Type SPLINEDIT on the command line.
2. Press Enter.
3. Select the spline to convert.



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The command line will look like the below image:

4. Type P and press Enter.
5. Specify the precision value between 0 and 99.
6. Press Enter.

The spline will be converted to the polyline.

