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Creating simple entities

With BtoCAD, simple entities include lines (both finite and infinite), circles, arcs, ellipses, elliptical arcs, points, and rays. In addition, BtoCAD includes a freehand sketch tool. Entities drawn freehand are also considered to be simple entities.

This section explains several methods for creating simple entities, including how to:

• Use menu commands on the Insert menu.

• Use the tools on the Draw 2D toolbar.

• Type commands in the command bar.

In some cases, there are a number of different ways to create an entity. For the most part, one or two are given here. Refer to the online Help to learn how to create entities using other methods.

When you use a tool or a drawing command, the program prompts you to enter coordinate points, such as endpoints or insertion points. You can enter the points or distances either using a mouse or by typing coordinate values in the command bar. As you draw, BtoCAD also displays a context-sensitive prompt box with appropriate additional options for the type of entity you are drawing.

After you create entities, you can modify them using the entity modification tools.

Topics in this chapter

Drawing lines

A line consists of two points: a start point and an endpoint. You can connect a series of lines, but each line segment is considered a separate line entity.

To draw a line

1 Do one of the following:

  • Choose Draw > Line.

  • On the Draw toolbar, click the Line tool

  • Type *line* and then press Enter.

2 Specify the start point.

3 Specify the endpoint.

4 In the prompt box, choose Done to complete the command.
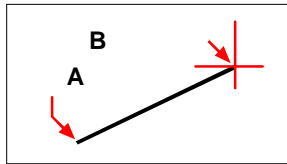
Start point (**A**) and endpoint (**B**).

The prompt box provides several options as you draw. For example, when you draw the first line segment, you can specify its length or orientation angle. After you draw at least one line segment, you can click Undo to remove the previous line segment. You can click Done to end the line command. After you draw two or more line segments, you can click Close to complete the line command by drawing a line segment that connects to the start point of the first line segment you drew. If the last entity you drew was an arc, you can also draw a line tangent to and starting from the endpoint of the arc.

To draw a line as a continuation from the end of an arc

1 Do one of the following:

  • Choose Draw > Line.

  • On the Draw toolbar, click the Line tool

  • Type *line* and then press Enter.

2 In the prompt box, choose Follow.

3 Specify the length of the line.
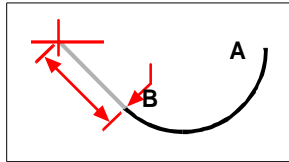
Endpoint of previous arc (**A**) and length of the line (**B**).

Drawing circles

The default method for drawing a circle is to specify a center point and radius. You can draw circles using any of the following methods:

- Center-Radius ( )
- Center-Diameter ( )
- Two points ( )
- Three points ( )
- Radius-Tangent-Tangent ( )
- Convert Arc to Circle ( )

**To draw a circle by specifying its center and radius**

1 Do one of the following:

- Choose Draw > Circle > Circle Radius.

- On the Draw toolbar, click the Circle Center-Radius tool

- Type *circle* and then press Enter.

2 Specify the center point.

3 Specify the radius of the circle.
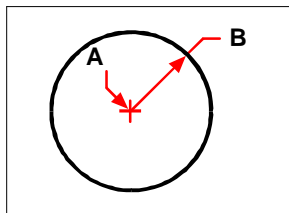
Center point (**A**) and radius (**B)**

**To draw a circle tangent to existing entities**

1   Do one of the following:

- Choose Insert > Circle > Tan,Tan, Radius.

- On the Draw toolbar, click the Circle Radius-Tangent tool ( ).   Go to step 3.

- Type *circle* and then press Enter.

2   In the prompt box, choose Radius-Tangent-Tangent.

3   Specify the radius of the circle.

4   Select the first entity to which to draw the circle tangent.

5   Select the second entity to which to draw the circle tangent.
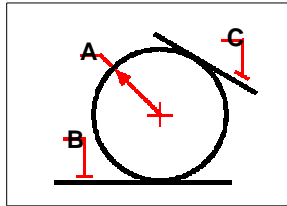
Radius of circle (**A**) and tangent lines (**B**) and (**C**).

To convert an arc to a circle

1   Do one of the following:

- On the Draw toolbar, click the Convert Arc to Circle tool ( ). Go to step 3.

- Type *circle* and then press Enter.

2   In the prompt box, choose Turn Arc Into Circle.

3   Select the arc you want to convert to a circle.
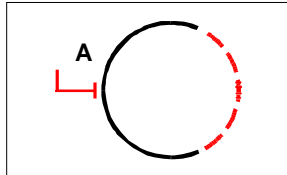
Select an arc (**A**) to convert to a circle.

Drawing arcs

An arc is a portion of a circle. The default method for drawing an arc is to specify three points—the start point,

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a second point, and the endpoint. You can draw arcs using any of the following methods:

- Three points on an arc ( )

- Start point-center-endpoint ( ), or Start point-endpoint-center ( ), or Center- start point-endpoint ( ).

- Start point-center-included angle ( ), or Start point-included angle-center
( ), or Center-start point-included angle ( ).

- Start point-center- length ( ), or Center-start point-length ( ).

- Start point-endpoint-radius ( ), or Start point-radius-endpoint ( ).

- Start point-endpoint-included angle ( ), or Start point-included angle-endpoint ( )

- Start point-endpoint-starting direction ( ), or Start point-starting direction-end- point ( ).

- Start point-radius-angle( )

- As a tangent continuation of the previous arc or line ( )

To draw an arc by specifying three points

1   Do one of the following:
- Choose Draw > Arc > 3 Points.

- On the Draw toolbar, click the 3-Point Arc tool ( ).
- Type *arc* and then press Enter.

2   Specify the start point.
3   Specify a second point.
4   Specify the endpoint.
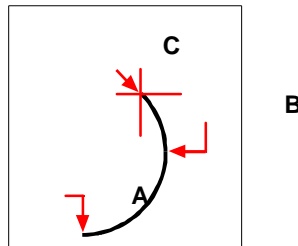
Start point (**A**), second point (**B**), and endpoint (**C**).

The prompt box provides additional options for drawing arcs. For example, after you specify the start point of an arc, you can choose Angle, Center, Direction, Endpoint, or Radius. You can select the options in a different order as well. For instance, you can draw an arc by specifying its start point, endpoint, and radius, or you can specify the start point, radius, and then endpoint.

To draw an arc by specifying its start point, center point, and endpoint

1   Do one of the following:
- Choose Draw > Arc >Start, Center, End.
- On the Draw toolbar, click the Arc Start-Center-End tool ( ).
- Type *arc* and then press Enter.

2   Specify the start point.

3   In the prompt box, choose Center.

4   Specify the center point.

5   Specify the endpoint.
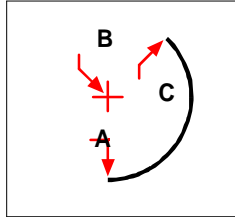
Start point (**A**), center point (**B**), and endpoint (**C**).

To draw an arc by specifying two points and an included angle

1   Do one of the following:
- Choose Insert > Arc > Start, End, Angle.
- On the Draw toolbar, click the Arc Start-End-Angle tool ( ).
- Type *arc* and then press Enter.

2   Specify the start point.

3   In the prompt box, choose Angle or type *angle*.

4   To draw an arc in a counterclockwise direction, enter a positive value for the included angle. To draw an arc in a clockwise direction, enter a negative value for the included angle.

5   Specify the endpoint.
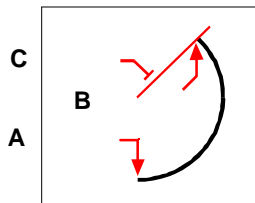
Start point (**A**), endpoint (**B**), and included angle (**C**).

If the last entity you drew was an arc or a line, you can also draw an arc tangent to and starting from the endpoint of the arc or line.

To draw an arc tangent to an arc or line

1   Do one of the following:

   • On the Draw toolbar, click the Tangent Arc tool ( ). Go to step 3.
   • Type *arc* and then press Enter.

2   In the prompt box, choose Follow.

3   Specify the endpoint.
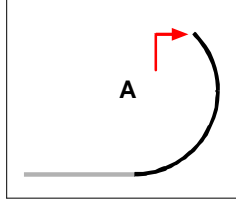
Endpoint (**A**).

TIP *To convert an arc to a circle, on the Draw 2D toolbar, click the Convert Arc To Circle flyout tool ( )*

Drawing ellipses

The default method for drawing an ellipse is to specify the endpoints of one axis of the ellipse, and then specify a distance representing half the length of the second axis. The endpoints of the first axis determine the orientation of the ellipse. The longer axis of the ellipse is called the major axis, and the shorter one is the minor axis. The order in which you define the axes does not matter. The program determines the major and minor axes based on their relative lengths. You can draw ellipses using any of the fol- lowing methods:

• Axis-axis ( )

• Axis-rotation ( )

• Center-axis ( )

• Center-rotation ( )

**To draw an ellipse by specifying the axis endpoints**

1   Do one of the following:
   • Choose Draw > Ellipse > Center.
   • On the Draw toolbar, click the Ellipse Axis-Axis tool ( ).
   • Type *ellipse* and then press Enter.

2   Specify the first endpoint.

3   Specify the second endpoint.

4   Specify the half-length of the other axis.

First axis endpoint (**A**), second axis endpoint (**B**), and half-length of other axis (**C**).

Drawing elliptical arcs

An elliptical arc is a portion of an ellipse. The default method for drawing an ellipti- cal arc is to specify the endpoints of one axis of the ellipse, and then specify a dis- tance representing half the length of the second axis. Then you specify the start and end angles for the arc, measured from the center of the ellipse in relation to its major axis. You can draw elliptical arcs using any of the following methods:
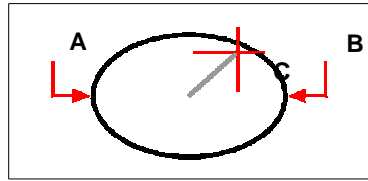
• Axis-axis ( )

• Axis-rotation ( )

• Center-axis ( )

• Center-rotation ( )

To draw an elliptical arc by specifying the axis endpoints

1   Do one of the following:

• Choose Draw > Ellipse > Elliptical Arc.

• On the Draw toolbar, click the Elliptical Arc Axis-Axis tool ( ).

• Type *ellipse* and then press Enter, and then type *a* (for Arc) and press Enter.

2   Specify the first endpoint.

3   Specify the second endpoint.

4   Specify the half-length of the other axis.

5   Specify the start angle of the arc.

6   Specify the end angle.

NOTE   *BtoCAD draws elliptical arcs in the direction you specify. Go to Settings > Drawing Settings > Drawing Units tab. Under Change Settings For, select Angular Units. The default setting is counterclockwise.*
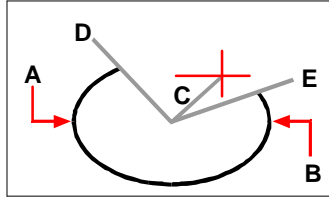
First axis endpoint (**A**), second axis endpoint (**B**), half-length of other axis (**C**), start angle of arc (**D**), and end angle (**E**).

Creating point entities

You can draw a point entity formatted as either a single dot or as one of 19 other possible display styles.

To draw a point

1   Do one of the following:

- Choose Draw > Draw Point > Single Point.
- On the Draw toolbar, click the Point tool).
- Type *point* and then press Enter.

2   Specify the location of the point.

To draw several points

1   Do one of the following:

- Choose Draw > Draw Point > Multiple Point.
- On the Draw toolbar, click the Point tool ( ).
- Type *point* and then press Enter.

2   In the prompt box, choose Multiple Points.

3   Specify the location of each point.

4   In the prompt box, choose Done to complete the command.

Changing the size and appearance of point entities

Changing the size and appearance of point entities affects all point entities already in the drawing, as well as all points that you subsequently draw. Positive values repre- sent the absolute size of the point entity measured in drawing units. Negative values represent a percentage relative to the drawing screen, so that points retain their visual size as you use the Zoom command to change the magnification of the drawing.

To change the size and appearance of point entities

1   Do one of the following:

- Choose Tools > Drawing Settings.

  - On the Settings toolbar, click the Drawing Settings tool (🖼️).
  - Type *settings* and then press Enter.

2 Click the Entity Creation tab.

3 Under Change Settings For, click Points.

4 Under Point Display Type, select the style you want.

5 Under Point Size, select the point size, or choose one of the options.

6 Click OK.

When you regenerate the drawing, all point entities change to reflect the new size and appearance settings.
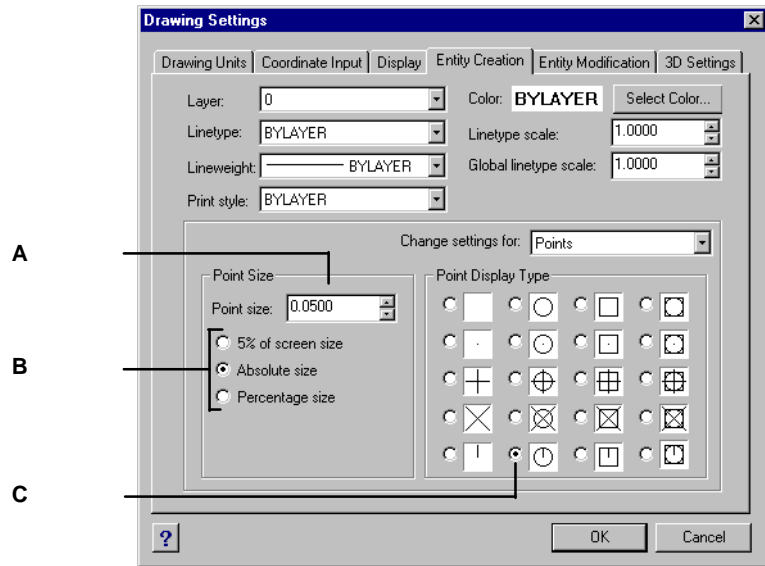
A To increase or decrease the point size, type or select a value.

B To use one of the preset point size options, click the one that you want.

C Select the button for the Point Display Type that you want.

Drawing rays

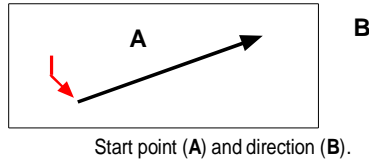A ray is a line in three-dimensional space that starts at a point and extends to infinity. Because rays extend to infinity, they are not calculated as part of the drawing extents. The default method for drawing a ray is to select

the start point of the ray and then specify its direction. You can draw a ray in any of the following ways:

- Horizontal draws the ray parallel to the x-axis of the current user coordinate sys- tem (UCS).

- Vertical draws the ray parallel to the y-axis of the current UCS.

- Angle draws the ray parallel to a specified angle.

- Bisect draws the ray perpendicular to an existing entity.

- Parallel draws the ray parallel to an existing entity.

To draw a ray

1   Do one of the following:
    - Choose Draw > Ray.

    - On the Draw toolbar, click the Ray tool ( ).
    - Type *ray* and then press Enter.

2   Specify the start point.

3   Specify the direction.

4   To complete the command, press Enter.

Start point (**A**) and direction (**B**).

Drawing infinite lines

Infinite lines are sometimes referred to as construction lines. An infinite line is a line through a given point, oriented at a specified angle in three-dimensional space and extending to infinity in both directions. Because infinite lines extend to infinity, they are not calculated as part of the drawing extents.

The default method for drawing an infinite line is to select a point along the line and then specify the direction of the line. You can draw an infinite line in any of the fol- lowing ways:

- Horizontal draws the infinite line parallel to the x-axis of the current UCS.

- Vertical draws the infinite line parallel to the y-axis of the current UCS.

- Angle draws the infinite line parallel to a specified angle.

- Bisect draws the infinite line perpendicular to an existing entity.

- Parallel draws the infinite line parallel to an existing entity.

To draw an infinite line

1   Do one of the following:
- Choose Draw > Infinite Line.

- On the Draw toolbar, click the Infinite Line tool (⬉).
- Type *infline* and then press Enter.

2   Specify a point along the line.

3   Specify the direction.

4   To complete the command, press Enter.
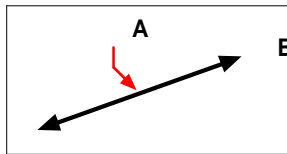
Point along the infinite line (**A**) and the direction (**B**).

You can also draw infinite lines at a specific angle or at an angle relative to an existing entity.

To draw an infinite line at a specified angle relative to another entity

1   Do one of the following:
- Choose Insert > Infinite Line.

- On the Draw toolbar, click the Infinite Line tool (⬉).
- Type *infline* and then press Enter.

2   In the prompt box, choose Angle.

3   In the prompt box, choose Reference.

4   Select the reference entity.

5   Specify the angle of the infinite line in relation to the selected entity.

6   Specify the location of the infinite line.

7   To complete the command, press Enter.
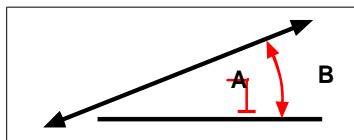
Reference entity (**A**) and angle in relation to entity (**B**).

Setting the sketch method and accuracy

Using polylines for freehand sketches makes it easier to go back and edit sketches. You control whether to create freehand sketches using line segments or polylines in the Drawing Settings dialog box. You can also control the length of sketch segments in this dialog box.

To specify lines or polylines when sketching

1   Do one of the following:
   - Choose Tools > Drawing Settings.
   - On the Settings toolbar, click the Drawing Settings tool ( ).
   - Type *settings* and then press Enter.

2   Choose the Entity Creation tab.

3   Under Change Settings For, click Freehand Sketches.

4   Under Freehand Sketching Methods, click either Freehand Command Creates Lines or Freehand Command Creates A Polyline.

5   Click OK.
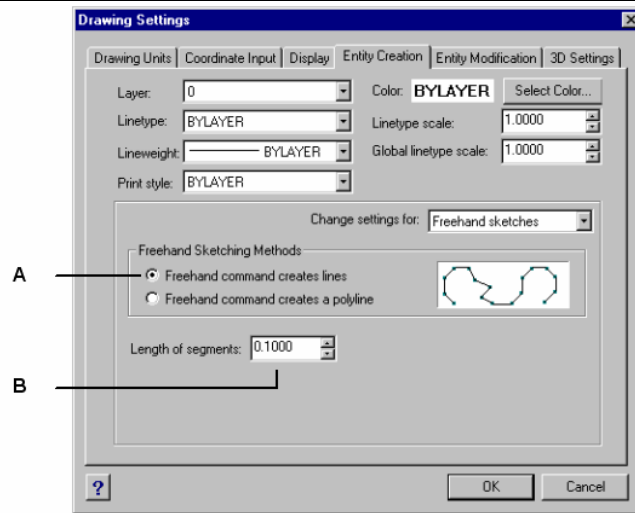
A Select the sketch method.　　　**B** Specify the default length of sketch segments.