Modifying entities

BtoCAD provides many editing tools for modifying a drawing. You can easily move, rotate, or stretch drawing entities, or change their scale. When you want to remove an entity, you can delete it with a few clicks of the mouse. You can also make multiple copies of any entity and copy entities from one drawing to another.

You can modify most entities using general-purpose editing commands, many of which are located on the Modify toolbar and the Modify menu. Some complex entities require special commands. This section explains how to:

- Select entities using entity-selection methods and grips.
- Change the properties of entities.
- Rearrange entities by moving, rotating, or changing the display order.
- Resize entities by stretching, scaling, extending, trimming, or editing their lengths.
- Break and join entities.
- Group entities.
- Edit polylines.
- Explode entities.
- Create chamfers and fillets.

Topics in this chapter

Selecting entities

You can create a selection set that consists of one or more entities before you modify them. Use any of the following methods to create a selection set:

- Choose a command or tool first, and then select entities.
- Select entities first, and then choose a command or tool (most entities).
- Select entities by pointing, and then use grips to modify them.

Displaying selected entities highlighted

You can specify whether to display selected entities highlighted, which makes the selection set easier to see. By default, the highlighting feature is turned on.

To turn the highlighting feature on or off

1. Do one of the following:
- On the Settings toolbar, click the Drawing Settings tool

- Type *settings* and then press Enter.

2   Click the Display tab.

3   In the Change Settings For box, select Display.

4   Select or clear the Highlight Item When Selected check box.

5   Click OK.

Entity-selection methods

When you choose a command that requires you to select entities (when you're deleting or changing entity properties, for example), you can use any of the following selection methods by choosing them in the prompt box or entering them in the command bar:
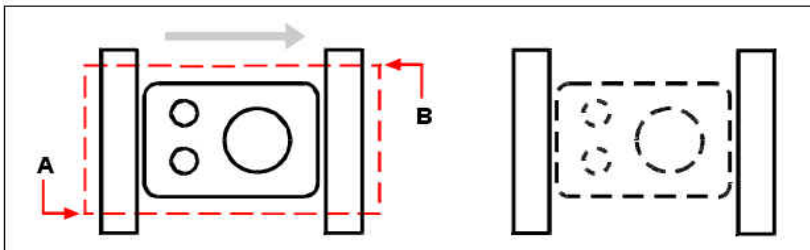
- **Select all entities (ALL)** Selects all entities in the current drawing.
- **Add to set (+ or A)** Adds one or more entities to the selection set.
- **Subtract from set (- or R)** Removes one or more entities from the selection set.
- **Previous selection (P)** Selects entities included in the previous selection set.
- **Last entity in drawing (L)** Selects the entity most recently added to the drawing.
- **Window-Inside (W)** Selects entities contained entirely within a rectangular selection window.
- **Crossing window (C)** Selects entities contained within or crossing the boundary of a rectangular selection window.

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- **Outside window (O)** Selects entities falling completely outside a rectangular selection window.
- **Window polygon (WP)** Selects entities contained entirely within a polygon selection window.
- **Crossing polygon (CP)** Selects entities contained within or crossing the boundary of a polygon selection window.
- **Outside polygon (OP)** Selects entities falling completely outside a polygon selection window.
- **Window circle (WC)** Selects entities contained entirely within a circular selection window.
- **Crossing circle (CC)** Selects entities contained within or crossing the boundary of a circular selection window.
- **Outside circle (OC)** Selects entities falling completely outside a circular selection window.
- **Point (PO)** Selects any closed entities that surround the selected point.
- **Fence (F)** Selects entities crossing a line or line segments.
- **Select by Properties (PRO)** Selects entities that match a particular set of proper- ties—for example, all entities on a particular layer or drawn in a certain color.

You can also use a few selection methods automatically, without displaying the prompt box. For example, you can simply click to select entities, or you can use a Window-Inside or Crossing Window by defining the opposite corners of a rectangular selection window. The direction in which you define the points of the rectangle
(left-to-right or right-to-left) determines which type of window you create.

To create a Window-Inside

1   Click to select a point in the drawing.

2   Click to the right of the first point to select a second point in the drawing.
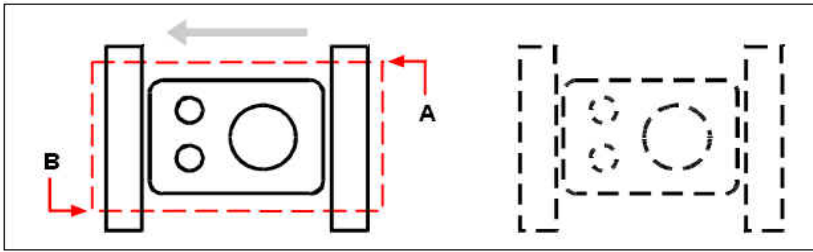

Window-Inside by selecting the first (**A**) and second (**B**)   Resulting selection.
points.

This method is most commonly referred to as simply a *window* or *selection window*.

To create a Crossing Window

1   Click to select a point in the drawing.

2   Click to the left of the first point to select a second point in the drawing.
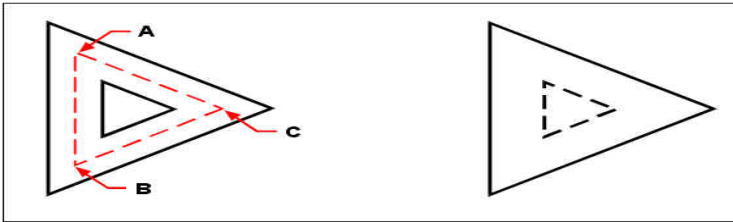
Crossing Window by selecting the first (**A**) and second
(**B**) points.

Resulting selection.

In addition to a rectangular window, you can define a selection window using other shapes such as a polygon, circle, or fence (a multisegmented line that selects entities it crosses).

To select entities using Window Polygon

1   Activate an entity-modification command.

2   In the prompt box, choose Window Polygon.
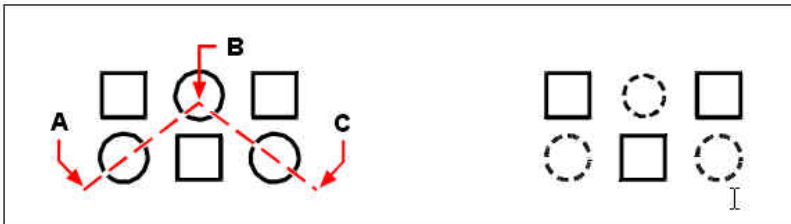
3   Specify the vertices of the polygon.

Window Polygon by specifying the vertices of the
polygon (**A**, **B**, and **C**).

Resulting selection.

4   To complete the selection polygon, press Enter.

To select entities using Fence

1   Activate an entity-modification command.

2   In the prompt box, choose Fence.

3   Specify the endpoints of the Fence segments.

4   To complete the Fence, press Enter.

Fence by specifying the endpoints of the fence segments
(**A**, **B**, and **C**).

Resulting selection.
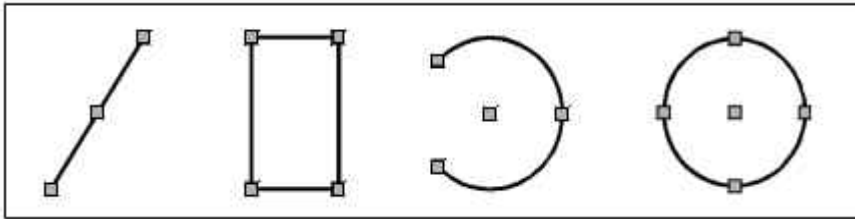
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Choosing the command first

When you choose an entity-modification tool or command, the program prompts you to select entities and displays a prompt box from which you can choose a selection method. You can select individual entities or use other techniques such as selection windows to select multiple entities.

When you select entities, you add them to the selection set. After you select at least one entity, you can remove entities from the selection set. To finish adding entities to the selection set, press Enter. Most entity-modification command

Selecting entities first

You can select entities first and then choose how to modify them. As you select each entity, it is highlighted with small squares called grips, which appear at strategic points on the entity.

The locations of the grips depend on the type of entity selected. For example, grips appear at the endpoints and midpoint of a line, at the quadrant points and center point of a circle, and at the endpoints, midpoint, and center of an arc.

Examples of grip locations.

After you select one or more entities, you can choose an entity-modification command, such as Copy or Move, from the Modify menu or toolbar. You can also click the right mouse button to display a shortcut menu containing the entity-modification commands appropriate for the selected entities, and then choose the command from the menu.

When you select entities and then issue a command, the program immediately acts on the entities you've selected. In many cases, a command-specific prompt box provides additional options for that editing operation. If you want to modify the selection set at that point, right-click to display the prompt box with the selection options and choose the option you want. To redisplay the command-specific prompt box, right-click again.

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Deselecting entities

If an entity is no longer needed in a selection set, you can deselect it to remove it from the selection set.

To remove an entity from the selection set

1   Press Shift, and then select the entity again.

NOTE: *Pressing Shift while selecting entities using a crossing window removes all entities from the specified selection set.*

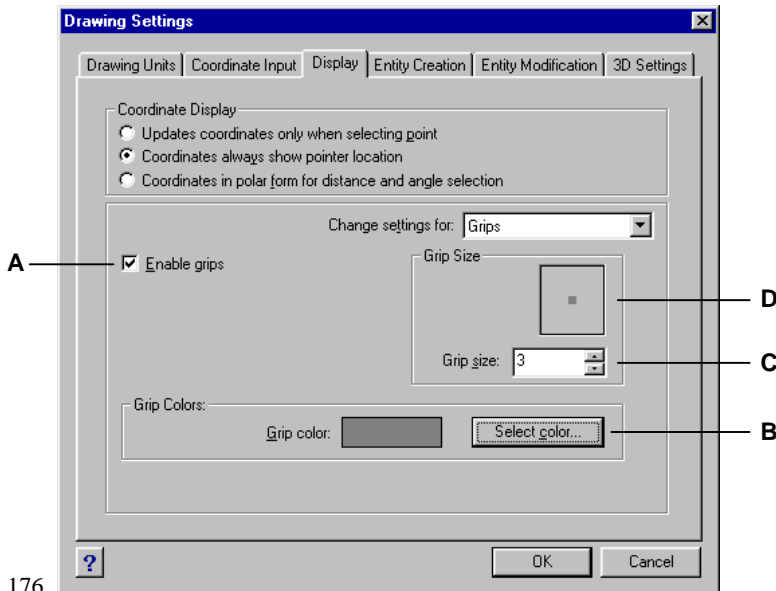**To remove all entities from the selection set**

  Press Escape.

Turning grips on and off

You can turn the use of grips on and off and control the size and color of grips.

To change grip settings

1   Do one of the following:

-
  On the Settings toolbar, click the Drawing Settings tool 🖳🗷

- Type *settings* and then press Enter.

2   In the Drawing Settings dialog box, click the Display tab.

3   In the Change Settings For list, click Grips.

4   Change the desired settings, and then click OK.

| **A** | Click to activate grips for all selected entities. | **C** | Specify the grip size. |
| **B** | Click to assign the color for grips. | **D** | Displays the current grip size. |

Editing with grips

To use grips for editing, you select an entity to display the grips, and then click a grip to make it active. The grip you select depends on the type of entity you're modifying and the editing operation you're performing. For example, to move a line entity, drag it by its midpoint grip. To stretch the line, drag one of the endpoint grips. You do not need to enter a command when using grips.

Modifying the properties of entities

You can change the layer, thickness, linetype, color, and linetype scale of one or more entities. Depending on the type of entity or entities you select, you can also change other properties, such as the start point and endpoint of lines, the center point and radius of circles, and the vertices of polylines.

You can modify all the properties of all entities simultaneously. For example, using the *entprop* command, select all entities on a particular layer, and then move the entities to another layer by simply selecting a name from the Layer text box.

In the Entity Properties dialog box, changes that you make in the Layer, Color, Thick- ness, Lineweight, Linetype, Linetype Scale, and Print Style (if using named print style tables) fields affect all selected entities. If you select several entities that all have different properties, the default value initially is *Varies*. Changes that you make in the entity tabs, such as Circle, Line, and Arc, affect all selected entities of that type. You can select the entities to be changed using any entity-selection method.

To modify properties of entities

1   Do one of the following:
   • Choose Modify > Properties.

   •
     On the Modify toolbar, click the Properties tool

   • Type *entprop* and then press Enter.

2   Select the entities, and then press Enter.

3   Make changes to the desired properties, and then click OK.

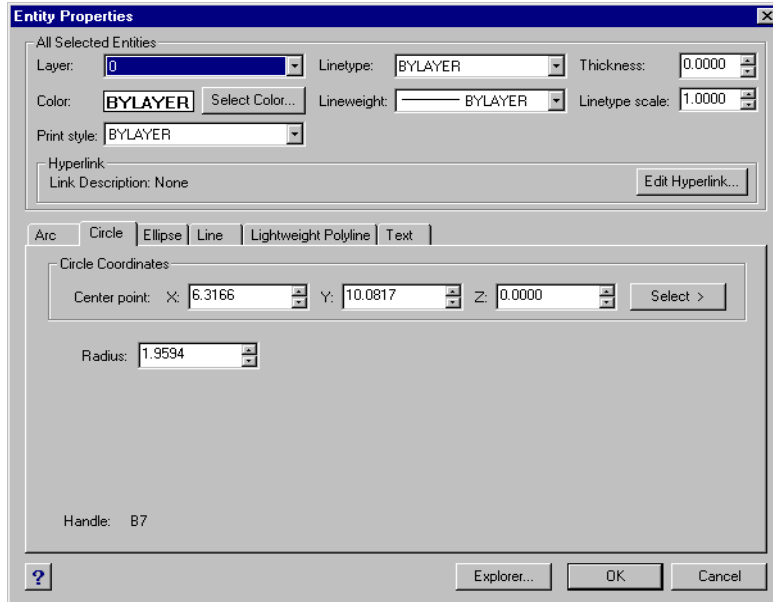**TIP** *Use the Entity Properties toolbar to change the properties of selected entities.*
Note that the Entity Properties toolbar settings that display when no entities are selected determine the

properties of new entities when you draw them.

The Entity Properties dialog box, shown in the following illustration, has two sections. The All Selected Entities section (the upper part of the dialog box) displays properties common to all selected entities, such as Layer, Color, Lineweight, Line- type, Linetype Scale, Thickness, and Print Style (if using named print style tables). This portion of the dialog box is equivalent to the dialog box displayed by the AutoCAD *ddchprop* command.

The lower section of the Entity Properties dialog box contains options specific to the selected entities. Each tab displays the properties appropriate for the entity. This feature is equivalent to repeatedly applying the AutoCAD *ddmodify* command to one entity at a time

The BtoCAD Entity Properties dialog box.

Deleting entities

You can remove entities from a drawing. You can delete entities using any of the entity-selection methods.

To delete a selection set

1   Do one of the following:
> • Choose Edit > Delete.
>
> • On the Standard toolbar, click the Delete tool (✕).
>
> • Type *delete* and then press Enter.

2   Select the entities, and then press Enter.

TIP *Typing the Undelete command restores the most recently deleted selection set. If you have made additional modifications since deleting the entities, use Undelete rather than Undo to restore those entities without reversing those modifications.*

Copying entities

You can copy one or more entities, making one copy or multiple copies within the current drawing. You can also copy entities between drawings.

Use any of the following methods to copy entities within the current drawing:

•   Create a copy at a location referenced from the original.

•   Create a copy aligned parallel to the original.

•   Create a copy as a mirror image of the original.

•   Create several copies in a rectangular or circular pattern.
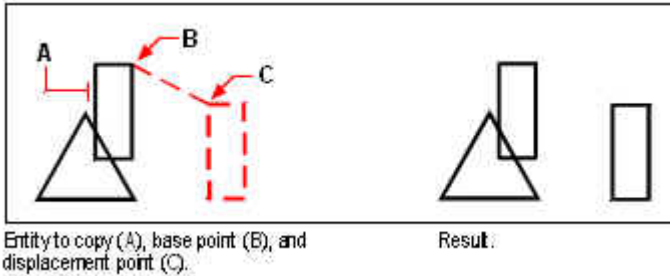
Copying entities within a drawing

You can duplicate entities within the current drawing. The default method is to create a selection set and then specify a starting point, or base point, and an endpoint, or displacement point, for the copy. You can also make multiple copies or copy the selection set to a location you specify, using a direction vector.

To copy a selection set once

1   Do one of the following:
> • Choose Modify > Copy.
>
> •
>
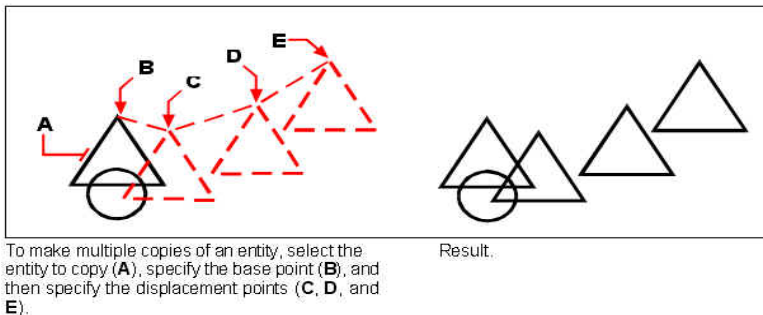> On the Modify toolbar, click the Copy tool

 • Type *copy* and then press Enter.

2   Select the entities, and then press Enter.

3   Specify the base point.

4   Specify the displacement point.

Entity to copy (A), base point (B), and
displacement point (C).   Result.

To make multiple copies of a selection set

1   Do one of the following:
 • Choose Modify > Copy.

 •
 On the Modify toolbar, click the Copy tool

 • Type *copy* and then press Enter.

2   Select the entities, and then press Enter.

3   In the prompt box, choose Multiple Copies.

4   Specify the base point.

5   Specify the displacement point of the first copy.

6   Specify the displacement point of the next copy.

7   Continue specifying displacement points to place additional copies.

8   To complete the command, press Enter.

To make multiple copies of an entity, select the
entity to copy (**A**), specify the base point (**B**), and
then specify the displacement points (**C**, **D**, and
E).   Result.

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Copying between drawings

You can use the Clipboard to cut or copy entities from one drawing to another. Cutting removes the selected entities from a drawing and stores them on the Clipboard. Copying duplicates the selected entities from a drawing and places them on the Clipboard.

To cut entities to the Clipboard

1   Select the entities you want to cut.

2   Do one of the following:
     • Choose Edit > Cut.

     •
       On the Standard toolbar, click the Cut tool

     • Type *cutclip* and then press Enter.

To copy entities to the Clipboard

1   Select the entities you want to copy.

2   Do one of the following:

       Choose Modify > Copy.

     •
       On the Modify toolbar, click the Copy tool

     • Type *copy* and then press Enter.

Anything that you can copy to the Clipboard can be pasted into a drawing. The format in which the program adds the Clipboard contents to the drawing depends on the type of information in the Clipboard. For example, if you copy BtoCAD drawing entities to the Clipboard, the program pastes them into the drawing as BtoCAD entities. If you copy items to the Clipboard from other programs, they are pasted into the current drawing as embedded ActiveX® objects

To paste entities from the Clipboard

1   Do one of the following:
     • Choose Edit > Paste.

     •
       On the Standard toolbar, click the Paste tool

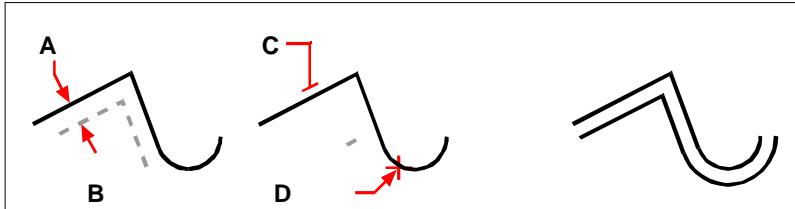   • Type *pasteclip* and then press Enter.

Making parallel copies

You can use the parallel feature to copy selected entities and align them parallel to the original entities at a specified distance. You can make parallel entities using arcs, circles, ellipses, elliptical arcs, lines, two-dimensional polylines, rays, and infinite lines.

Making parallel copies of curved entities creates larger or smaller curves, depending on which side of the original entity you place the copy. For example, placing a parallel copy of a circle outside the circle creates a larger concentric circle; positioning the copy inside the circle creates a smaller concentric circle.
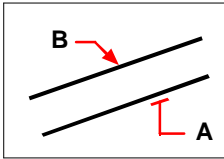
To make a parallel copy by specifying the distance

1   Do one of the following:
     • Choose Modify > Parallel.

     • Type *parallel* and then press Enter.

2   Specify the distance by selecting two points or by entering a distance.

3   Select the entity to copy.

4   Specify on which side of the entity to place the parallel copy.

5   Select another entity to copy, or press Enter to complete the command.

To make a parallel copy, specify the distance between copies by entering a distance or selecting two points (**A** and **B**), select the entity to copy (**C**), and specify on which side to place the copy (**D**).

To make a parallel copy passing through a point

1   Do one of the following:
     • Choose Modify > Parallel.

     • Type *parallel* and then press Enter.

2   In the prompt box, choose Through Point.

3   Select the entity to copy.

4   Specify the point for the entity to pass through.

5   Repeat steps 3 and 4, or press Enter to complete the command.

To make a parallel copy passing through a point, select the entity to copy (**A**) and then specify the through point (**B**).

Mirroring entities

You can create a mirror image of an entity. You mirror the entity about a mirror line, which you define by specifying two points in a drawing. You can delete or retain the original entities.

To mirror entities
1   Do one of the following:

• Choose Modify > Mirror.

•
On the Modify toolbar, click the Mirror tool

• Type *mirror* and then press Enter.

2   Select the entity, and then press Enter.

3   Specify the first point of the mirror line.

4   Specify the second point of the mirror line.

5   In the prompt box, choose one of the following:

•   Yes, Delete Entities – deletes the original entities.

•   No, Keep Entities – retains the original entities.

To mirror an entity, select it (A), and then specify the
first point (B) and second point (C) of the mirror line.
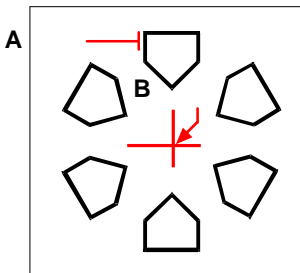
Arraying entities

You can copy an entity in a rectangular or polar (circular) pattern, creating an array. For a rectangular array, you control the number of copies in the array by specifying the number of rows and columns. You also specify the distance between each row

and column. For a polar array, you control the number of copies that compose the array and whether to rotate the copies.

To create a polar array

1   Do one of the following:
   • Choose Modify > Array.

   •
     On the Modify toolbar, click the Array tool 🔲
   • Type *array* and then press Enter.

2   Select the entities, and then press Enter.

3   In the prompt box, choose Polar.

4   Specify the center point of the array.

5   Specify the number of items to array, including the original selection set.

6   Specify the angle the array is to fill, from 0 to 360 degrees.

The default setting for the angle is 360 degrees. Positive values create the array in a counterclockwise direction; negative values create the array in a clockwise direction.

7   In the prompt box, choose one of the following:

•   Yes, Rotate Entities – rotates entities as they are arrayed.

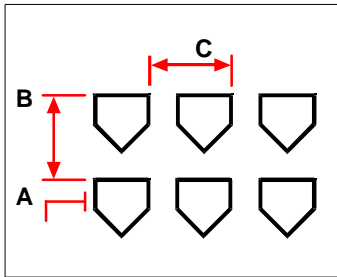•   No, Do Not Rotate – retains the original orientation of each copy as it is arrayed.

To create a polar array, select the entity to copy (**A**), specify the center point of the array (**B**), and then specify the number of items to array, the angle the array is to fill, and whether to rotate the items.

To create a rectangular array

1   Do one of the following:
   • Choose Modify > Array.

- On the Modify toolbar, click the Array tool

- Type *array* and then press Enter.

2   Select the entities, and then press Enter.

3   In the prompt box, choose Rectangular.

4   Type the number of rows.

5   Type the number of columns.

6   Specify the distance between the rows.

7   Specify the distance between the columns.

To create a rectangular array, select the entity to copy (**A**), type the number of rows and columns, and then specify the distance between each row (**B**) and column (**C**).

Rearranging entities

You can move one or more entities, and you can also rotate entities about a specified point.If you have entities that overlap, you can also change the display order.
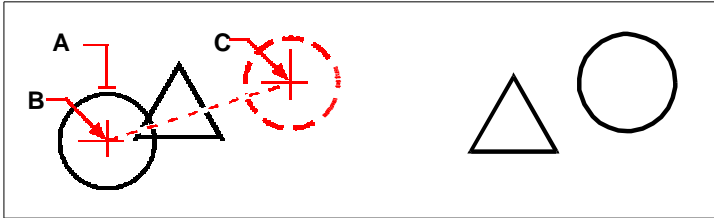
Moving entities

You can move entities around within the current drawing or from one drawing to another. The default method is to create a selection set and then specify a starting point, or base point, and an endpoint, or displacement point, to define the relocation of the entities. You can also relocate the entities using a direction vector.

To move a selection set

1   Do one of the following:
- Choose Modify > Move.

- On the Modify toolbar, click the Move tool

- Type *move* and then press Enter.

2   Select the entities, and then press Enter.

3   Specify the base point.

4   Specify the displacement point.

To move an entity, select it (**A**), and then specify the base point (**B**) and the displacement point (**C**).

You can also move entities using grips. To move an entity using grips, select the entity to display its grips, and then click a grip and drag it. The grip you select depends on the type of entity you're modifying. For example, to move a line entity, select the midpoint grip. To move a curved entity, such as an arc, circle, or ellipse, select the center point grip. Not all entities can be moved using grips.
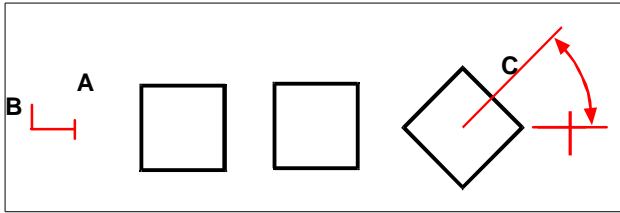
To move an entity using grips

1   Select the entity.

2   Click a grip to select it.

3   Drag the entity to where you want to relocate it.

4   Click to release.

Rotating entities

You can rotate entities about a specified point at a specified rotation angle or by an angle referenced to a base angle. The default method rotates the entities using a relative rotation angle from their current orientation.

To rotate a selection set

1   Do one of the following:
   • Choose Modify > Rotate.

   •
     On the Modify toolbar, click the Rotate tool

   • Type *rotate* and then press Enter.

2   Select the entities, and then press Enter.

3   Specify the rotation point.

4   Specify the rotation angle.

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To rotate an entity, select the entity to rotate (**A**), and then specify the rotation point (**B**) and the rotation angle (**C**).

To rotate a selection set in reference to a base angle

1   Do one of the following:

- Choose Modify > Rotate.

- On the Modify toolbar, click the Rotate tool
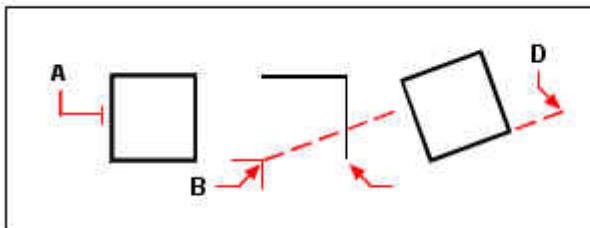
- Type *rotate* and then press Enter.

2   Select the entities, and then press Enter.

3   Specify the rotation point.

4   In the prompt box, choose Base Angle.

5   Specify the base angle.

6   Specify the new angle.

To rotate an entity in reference to a base angle, select the entity (A), specify the rotation point (B), select the base angle and pick point (B) again (or type the @ symbol), specify the second point (C), and then specify the point representing the new angle (D).

Reordering entities

When multiple entities overlap, you can change the order in which they are displayed and printed. You can

move entities to the front, back, or on top or below of another entity.

To reorder entities:

1   Type *draworder* and then press Enter.

2   Select the entity you want to reorder, and then press Enter.

3   In the prompt box, specify the new drawing order, and then press Enter.

4   If you are reordering above or under, select the entity you want the first entity to be above or below, and then press Enter.

NOTE   *The SORTENTS system variable automatically turns on, which may affect system performance.*
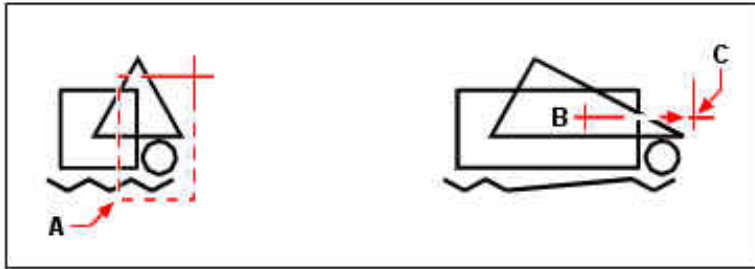
Resizing entities

You can change the size of an entity or set of entities by stretching, scaling, extending, trimming, or editing their lengths.

Stretching entities

You can change the size of entities by stretching them. When you stretch entities, you must select the entities using either a crossing window or a crossing polygon. You then specify a displacement distance or select a base point and a displacement point. Entities that cross the window or polygon boundary are stretched; those completely within the crossing window or crossing polygon are simply moved.

To stretch an entity

1   Do one of the following:
   • Choose Modify > Stretch.

   •
     On the Modify toolbar, click the Stretch tool
   • Type *stretch* and then press Enter.

2   In the prompt box, choose Crossing Window or Crossing Polygon.

3   Select the entities, and then press Enter.

4   Specify the base point.

5   Specify the second point of displacement.

To stretch entities, select them using a crossing window (A) or crossing polygon, and then specify the base point (B) and displacement point (C).

To stretch an entity using grips, you select it to display its grips and then select a grip to make it the active grip. This becomes the base point. Then you move the active grip to a new location. The grip you select depends on the type of entity you're modifying. For example, to stretch one corner of a rectangle, select the corner point grip. To stretch a line, select an endpoint grip. Not all entities can be stretched using grips.

To stretch an entity using grips

1 Select the entity.

2 Click a grip to activate it.

3 Drag the grip.

4 Click to release.

To stretch an entity using grips, select the entity (A), select a grip (B), and drag the grip to its new location (C).

Result.

Scaling entities

You can change the size of a selected entity by scaling it in relation to a base point. You can change the size of an entity by specifying a base point and a length, which is used as a scale factor based on the current drawing units, or by specifying a scale factor. You can also use a scale factor referenced to a base scale factor, for example, by specifying the current length and a new length for the entity.

To scale a selection set by a scale factor

1 Do one of the following:
   - Choose Modify > Scale.
   - On the Modify toolbar, click the Scale tool
   - Type *scale* and then press Enter.

2 Select the entities, and then press Enter.

3 Specify the base point.

4 Specify the scale factor.

Extending entities
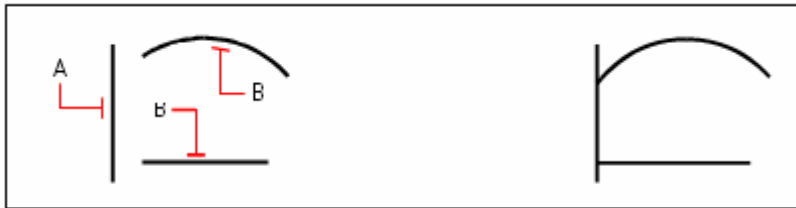

Select the boundary edge (A), and then select the entities to extend (B).   Result.

You can extend entities so that they end at a boundary defined by other entities. You can also extend entities to the point at which they would intersect an implied boundary edge. When extending entities, you first select the boundary edges and then specify the entities to extend, selecting them either one at a time or using the fence selection method.

You can extend arcs, lines, two-dimensional polylines, and rays. Arcs, circles, ellipses, lines, splines, polylines, rays, infinite lines, and viewports on a Layout tab can act as boundary edges.

To extend entities, select the boundary edge (A), and then   Result.
select the entities to extend (B).

To extend an entity

1   Do one of the following:
   • Choose Modify > Extend.

   •
      On the Modify toolbar, click the Extend tool

   • Type *extend* and then press Enter.

2   Select one or more entities as boundary edges, and then press Enter.

3   Select the entity to extend.

4   Select another entity to extend, or press Enter to complete the command.

Select the boundary edge (A), and then specify the   Result.
first point (B) and second point (C) of the fence.

When you extend a wide polyline, its centerline intersects the boundary edge. Because the end of the polyline is always cut at a 90-degree angle, part of the polyline may extend past the boundary edge. A tapered polyline continues to taper until it intersects the boundary edge. If this would result in a negative polyline width, the ending width changes to 0.

A tapered polyline (A) continues to taper until
it intersects the boundary edge (B).

Trimming entities

You can clip, or trim, entities so they end at one or more implied cutting edges defined by other entities. You can also trim entities to the point at which they would intersect an implied cutting edge. When trimming entities, you first select the cutting edges and then specify the entities to trim, selecting them either one at a time or using the fence selection method.

You can trim arcs, circles, lines, open two-dimensional and three-dimensional polylines, and rays. Arcs, circles, lines, polylines, rays, infinite lines, and viewports on a Layout tab can act as cutting edges. An entity can be both a cutting edge and one of the entities being trimmed.
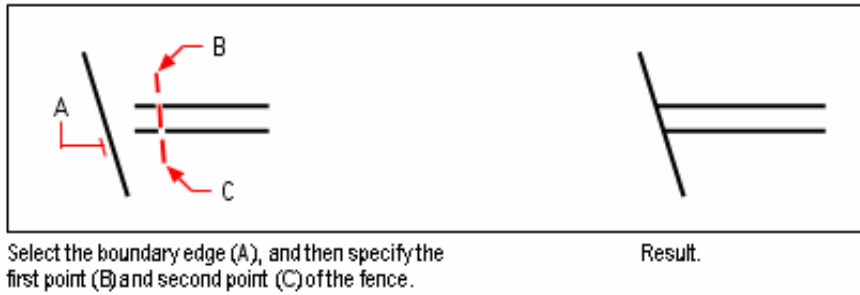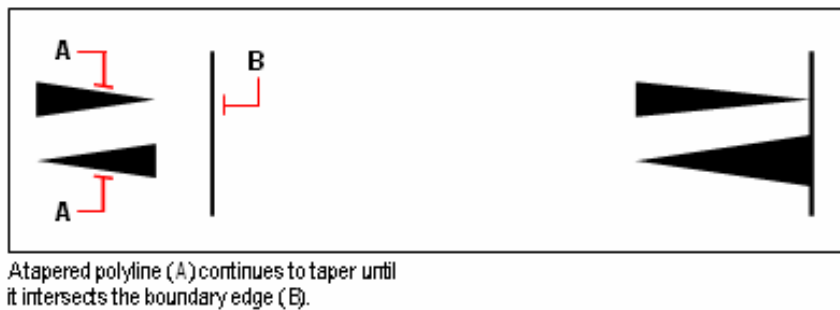
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To trim an entity

1   Do one of the following:
   • Choose Modify > Trim.

   •
   On the Modify toolbar, click the Trim tool
   • Type *trim* and then press Enter.

2 Select one or more cutting edges, and then press Enter.

**3** Select the entity to trim.

4 Select another entity to trim, or press Enter to complete the command.

To trim entities, select the cutting edge (A), and then select
the entities to trim (B).   Result.

To trim an entity to an implied boundary

1 Do one of the following:
- Choose Modify > Trim.
-
  On the Modify toolbar, click the Trim tool
- Type *trim* and then press Enter.

2 Select one or more cutting edges, and then press Enter.

3 In the prompt box, choose Edge Mode.

4 In the prompt box, choose Extend.

5 Select the entity to trim.

6 Select another entity to trim, or press Enter to complete the command.

To trim several entities using the fence selection method

1 Do one of the following:
- Choose Modify > Trim.
-
  On the Modify toolbar, click the Trim tool
- Type *trim* and then press Enter.

2 Select one or more cutting edges, and then press Enter.

3 In the prompt box, choose Fence.

4 Specify the first point of the fence.

5 Specify the second point of the fence.

6   Specify the next fence point, or press Enter to complete the command.

Select the boundary edge (A), and then specify the first   Result.
point (B) and second point (C) of the fence.

Editing the length of entities

You can change the length of entities or the included angle of arcs. Use any of the following methods to change the length of an entity:

- Dynamically drag the endpoint or angle.

- Specify an incremental length or angle measured from an endpoint.

- Specify the new length as a percentage of the total length or angle.

- Specify a new length or included angle.

You can change the length of arcs, lines, and open polylines.

To change the length of an entity by dragging

1   Do one of the following:
- Choose Modify > Edit Length.

- On the Modify toolbar, click the Edit Length tool

- Type *editlen* and then press Enter.

2   In the prompt box, choose Dynamic.

3   Select the entity you want to change.

4   Specify the new endpoint or included angle.

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Select the entity (A), and then select the new
endpoint (B).

Breaking and joining entities

You can break an entity into two parts, removing a portion of the entity in the process. You can also join two entities into a single entity.

Breaking entities

You can break arcs, circles, ellipses, lines, polylines, rays, and infinite lines. When breaking entities, you must specify two points for the break. By default, the point you use to select the entity becomes the first break point; however, you can use the First option to select a break point different from the one that selects the entity.

To break an entity

1   Do one of the following:
    • Choose Modify > Break.
    •
      On the Modify toolbar, click the Break tool
    • Type *break* and then press Enter.
2   Select the entity.
3   Specify the second break point.

Select the entity (A), and then specify the second break
point (B).

To select an entity and then specify the two break points

1   Do one of the following:
    • Choose Modify > Break.

    •
    On the Modify toolbar, click the Break tool

    • Type *break* and then press Enter.

2   Select the entity.

3   In the prompt box, choose First.

4   Specify the first break point.

5   Specify the second break point.

Select the entity (A), and then specify the first (B) and   Result.
second (C) break points.

TIP   *To break an entity in two without removing a portion of the entity, specify the same point for the first and second break points by typing the at sign (@) and pressing Enter instead of specifying the second break point.*

Joining entities

You can join two entities into a single entity. You can join either two lines or two arcs. The two lines must be parallel; the two arcs must share the same center point and radius.

When you join two lines, the farthest endpoints remain at their existing locations; the program draws a new line between these points. Arcs are joined counterclockwise, from the first arc you select to the second.

To join two entities

1   Type *join* and then press Enter.
2   Select the first arc or line.
3   Select the second arc or line.

Grouping entities

A group is a collection of entities saved together as one unit. After you select the entities that belong in the group, you can later add more entities, remove entities, and reorder the entities. If necessary, you can also ungroup the entities at any time to work with the entities separately.

The Group dialog box controls the settings for all groups in a drawing.

Creating groups

When you create a group, you enter a group name and description, and then select the entities for the group.

To create a group

1   Type *group* and then press Enter.
2   Under Create New Group, enter the name and description of the group.

3   Click Selectable if you want all entities in the group to be selected when you select one entity of the group in the drawing.

4   Click Select Entities and Create Group.

5   Select the entities for the group, and then press Enter.

6   In the Group dialog box, click OK.

TIP *You can enter the name of a group in the command bar when selecting entities.*

Modifying groups

To modify a group and its entities

1   Type *group* and then press Enter.

2   Select the group you want to modify.

3   Under Modify Selected Group, do one or more of the following:

•   Enter a new name, and then click Rename Group.

•   Enter a new description, and then click Change Group Description.

•   Select whether you want the group to be selectable in the drawing.

•   Click Add Entities to Group, select the entities to add to the group, and then press Enter.

•   Click Remove Entities from Group, select the entities to remove from the group, and then press Enter.

4   In the Group dialog box, click OK.

To change the order of entities in a group

1   Type *group* and then press Enter.

2   Under Modify Selected Group, click Reorder Entities.

3   In the Reorder Grouped Entities dialog box, select the group you want to reorder.

4   To see the order of entities in the group, click Highlight. Follow the prompts that display to view the entities one by one.

5   To reverse the order of all entities in the group, click Reverse Order.

6   To change the order of specific entities or a range of entities:

•   In Remove from Position, enter the current position of the entity.

•   In Place to Position, enter the new position of the entity.

•   In Number of Entities, enter the number of entities or range of entities to reorder. For example, if you are changing the order of only one entity, enter 1.

•   Click Reorder.

NOTE *The entities in a group are numbered 0, 1, 2, 3, and so on.*

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7   Click OK, and then click OK again.

Ungrouping entities

When you ungroup entities, the entities remain in the drawing but the group is deleted from the drawing.

To ungroup entities

1   Type *group* and then press Enter.

2   Select the group to delete.

3   Under Modify Selected Group, click Ungroup Entities.

4   Click OK.

Editing polylines

You can modify any type of two-dimensional or three-dimensional polyline. Entities such as rectangles, polygons, and donuts, as well as three-dimensional entities such as pyramids, cylinders, and spheres, are all variations of polylines that you can edit.

You can edit a polyline by opening or closing it, by changing its overall width or the widths of individual segments, and by converting a polyline with straight line segments into a flowing curve or an approximation of a spline. In addition, you can use the Edit Polyline tool to edit individual vertices, adding, removing, or moving verti- ces. You can also add new segments to an existing polyline, change the linetypes of a polyline, and reverse the direction or order of the vertices.

To modify a polyline, you first select the polyline, and then select a polyline editing option. The available options vary depending on whether the selected polyline is a two-dimensional or three-dimensional entity. If the selected entity is not a polyline, the Edit Polyline tool provides the option of turning it into one. You can convert only arcs and lines into polylines. If several arcs or lines are joined endpoint to endpoint, they can all be selected and turned into one polyline.

To convert an entity into a polyline

1   Do one of the following:
- Choose Modify > Edit Polyline.

- Type *editpline* and then press Enter.

2   Select the entity.

3   In the prompt box, choose Yes-Turn Into Polyline.

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

Opening and closing polylines

When you close a polyline, the program draws a straight polyline segment from the last vertex of the polyline to the first vertex. Opening a polyline removes the closing segment. When you select a polyline for editing, the prompt box displays either the Open or Close option, depending on whether the polyline you select is closed or open.
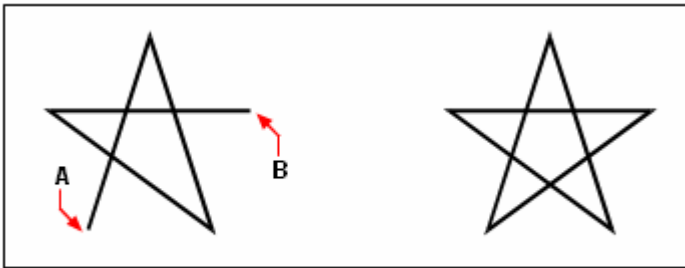
To close an open polyline

1   Do one of the following:
    • Choose Modify > Edit Polyline.

    • Type *editpline* and then press Enter.

2   Select the polyline.

3   In the prompt box, choose Close.

Closing an open polyline adds a straight   Result.
polyline segment between the first (A) and last
(B) vertices.

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

Curving and decurving polylines

You can convert a multisegment polyline into a smooth curve using either the Fit or Spline option. The Fit option creates a smooth curve connecting all the vertices. The Spline option computes a smooth curve that is pulled toward the vertices but passes through only the first and last vertices. The Decurve option removes Fit or Spline curves and arcs, leaving straight segments between the vertices.

To fit a curve to a polyline

1   Do one of the following:

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> • Choose Modify > Edit Polyline.

> • Type *editpline* and then press Enter.

2 Select the polyline.

3 In the prompt box, choose Fit.

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

Original polyline.   After applying Fit curve.   After applying Spline.

Joining polylines

You can add an arc, line, or polyline entity to an existing open polyline, forming one continuous polyline entity. To join an entity to a polyline, that entity must already share an endpoint with an end vertex of the selected polyline.

When you join an entity to a polyline, the width of the new polyline segment depends on the width of the original polyline and the type of entity you are joining to it:

• A line or an arc assumes the same width as the polyline segment for the end vertex
to which it is joined.

• A polyline joined to a tapered polyline retains its own width values.

• A polyline joined to a uniform-width polyline assumes the width of the polyline to which it is joined.

To join an arc, line, or polyline to an existing polyline

1 Do one of the following:

> • Choose Modify > Edit Polyline.

> • Type *editpline* and then press Enter.

2 Select the polyline.

3   In the prompt box, choose Join.

4   Select the arc, line, or polyline to join.

5   In the prompt box, choose another option, or choose Done to complete the command.

Changing the polyline width

You can change the width of an entire polyline, applying a uniform width to the entire entity or tapering the polyline uniformly along its entire length.

To apply a uniform width to an entire polyline

1   Do one of the following:

> • Choose Modify > Edit Polyline.
>
> • Type *editpline* and then press Enter.

2   Select the polyline.

3   In the prompt box, choose Width.

4   Specify the new polyline width.

5   In the prompt box, choose another option, or choose Done to complete the command.

To taper a polyline uniformly along its length

1   Do one of the following:

> • Choose Modify > Edit Polyline.
>
> • Type *editpline* and then press Enter.

2   Select the polyline.

3   In the prompt box, choose Taper.

4   Specify the starting width.

5   Specify the ending width.

6   In the prompt box, choose another option, or choose Done to complete the command.

Editing polyline vertices

You can use the Edit Vertices option to modify individual polyline vertices. When you select this option, the program switches into a special vertex editing mode and places an x on the first vertex. The x indicates the vertex you are editing. The Next and Previous options move the x to the next or previous vertex. You can edit only one vertex at a time.

When editing vertices, you can modify the polyline in the following ways:

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- Convert a polyline segment into a curve by specifying a new tangent angle.

- Break a polyline into two separate polylines.

- Insert a new vertex after the current vertex.

- Move the current vertex.

- Straighten the polyline segment between two vertices.

- Change the width of the polyline segment between two vertices.

To move a polyline vertex

1   Do one of the following:
    • Choose Modify > Edit Polyline.

    • Type *editpline* and then press Enter.

2   Select the polyline.

3   In the prompt box, choose Edit Vertices.

4   In the prompt box, choose Next Vertex.

Repeat until the *x* reaches the vertex you want to move.

5   In the prompt box, choose Move.

6   Specify the new location for the vertex.

7   In the prompt box, choose another option, or choose Exit to stop editing vertices.

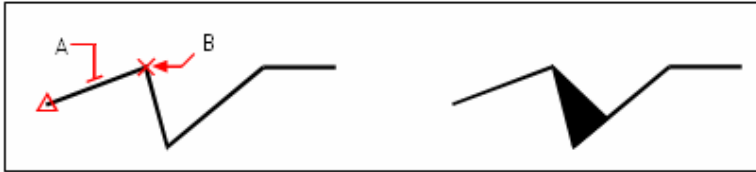8   In the prompt box, choose another option, or choose Done to complete the command.

Select the polyline (A), move the current vertex
marker to the vertex you want to move (B), and then
specify the new vertex location (C).

To taper the width of an individual polyline segment

1   Do one of the following:
    • Choose Modify > Edit Polyline.

    • Type *editpline* and then press Enter.

2   Select the polyline.

3   In the prompt box, choose Edit Vertices.

4   In the prompt box, choose Next Vertex. Repeat until the *x* reaches the first vertex of the segment you want to taper.

5   In the prompt box, choose Width.

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6   Specify the starting width.

7   Specify the ending width.

8   In the prompt box, choose another option, or choose Exit to stop editing vertices and update the display.

9   In the prompt box, choose another option, or choose Done to complete the command.

Select the polyline (A), move the current vertex
marker to the first vertex of the segment you want to
taper (B), and then specify the new starting and
ending widths for that segment.

Result.

Exploding entities

You can convert a complex entity, such as a block or polyline, from a single entity into its component parts. Exploding a polyline, rectangle, donut, polygon, dimension, or leader reduces it to a collection of individual line and arc entities that you can then modify individually. Blocks are converted to the individual entities, possibly including other, nested blocks that composed the original entity.

With the following exceptions, exploding an entity usually has no visible effect on a drawing:
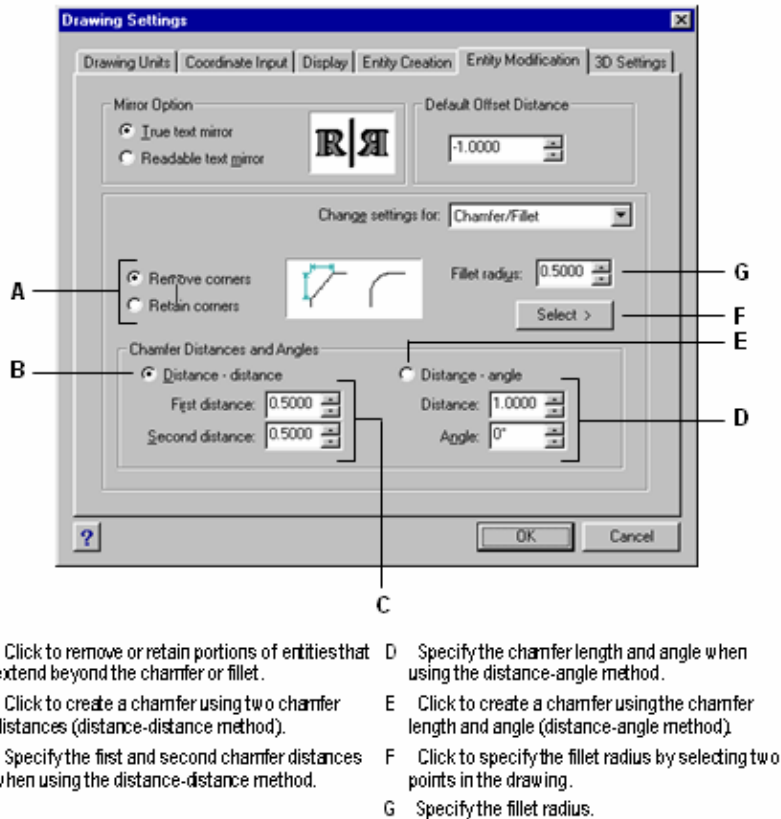
•   If the original polyline had a width, the width information is lost when you explode it. The resulting lines and arcs follow the centerline of the original polyline.

•   If you explode a block containing attributes, the attributes are lost, but the original attribute definitions remain.

•   Colors, linetypes, lineweights, and print styles assigned BYBLOCK may be different after exploding an entity, because they will adopt the default color, linetype, lineweight, and print style until inserted into another block.

To explode an entity

1   Do one of the following:
   • Choose Modify > Explode.

   •
     On the Modify toolbar, click the Explode tool

   • Type *explode* and then press Enter.
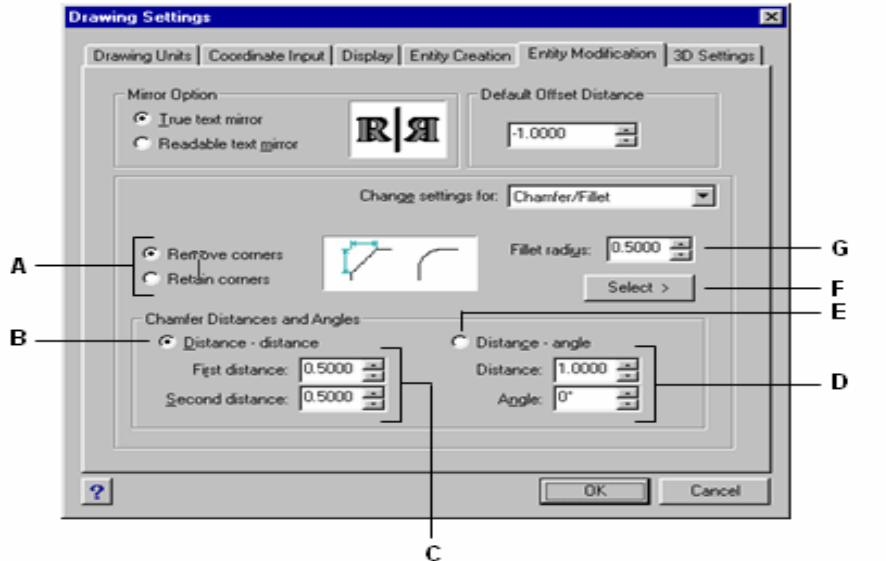
2   Select the entities to explode.

A   Click to remove or retain portions of entities that extend beyond the chamfer or fillet.

B   Click to create a chamfer using two chamfer distances (distance-distance method).

C   Specify the first and second chamfer distances when using the distance-distance method.

D   Specify the chamfer length and angle when using the distance-angle method.

E   Click to create a chamfer using the chamfer length and angle (distance-angle method).

F   Click to specify the fillet radius by selecting two points in the drawing.

G   Specify the fillet radius.

3   Press Enter.

Chamfering and filleting entities

You can chamfer or fillet entities. A chamfer connects two nonparallel entities with a line to create a beveled edge. A fillet connects two entities with an arc of a specified radius to create a rounded edge. If both entities you are working with are on the same layer, the chamfer or fillet is drawn on that layer. If they are on different layers, the chamfer or fillet is drawn on the current layer.

The Chamfer/Fillet settings in the Drawing Settings dialog box control the chamfer and fillet settings. The portions of the entities that extend beyond the chamfer or fillet are normally deleted when you create the chamfer or fillet. You can retain these original entities, however, by changing the settings in the dialog box.

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A   Click to remove or retain portions of entities that   D   Specify the chamfer length and angle when
     extend beyond the chamfer or fillet.   using the distance-angle method.
B   Click to create a chamfer using two chamfer   E   Click to create a chamfer using the chamfer
     distances (distance-distance method).   length and angle (distance-angle method).
C   Specify the first and second chamfer distances   F   Click to specify the fillet radius by selecting two
     when using the distance-distance method.   points in the drawing.
                                                            G   Specify the fillet radius.

Chamfering entities

You can connect two nonparallel entities by extending or trimming them and then joining them with a line to create a beveled edge. You can chamfer lines, polylines, rays, and infinite lines. When creating a chamfer, you can specify how far to trim the entities back from their intersection (distance-distance method), or you can specify the length of the chamfer and the angle it forms along the first entity (distance-angle method).

When chamfering a polyline, you can chamfer multiple segments between two selected polyline egments, or you can chamfer the entire polyline.

To chamfer two entities using the distance-distance method

1   Do one of the following:
   • Choose Modify > Chamfer.
   •
     On the Modify toolbar, click the Chamfer tool

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• Type *chamfer* and then press Enter.

2 In the prompt box, choose Chamfer Settings.

3 In the Drawing Settings dialog box, click the Entity Modification tab.

4 Under Chamfer Distances And Angles, click Distance-Distance.

5 Under Chamfer Distances And Angles, specify the first and second chamfer distances.
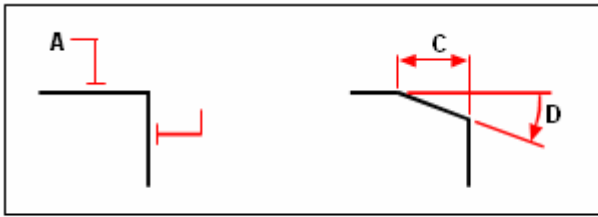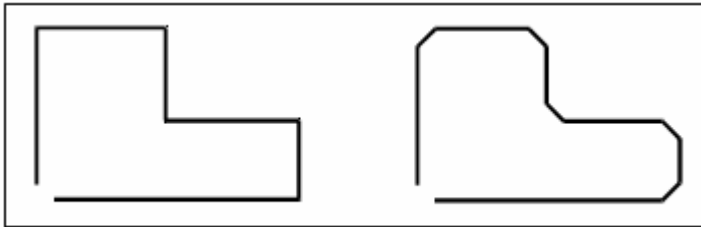
6 Click OK.

7 Select the first entity.

Select the first (A) and second (B) entities. The chamfer is drawn, based on the first (C) and second (D) chamfer distances.

8 Select the second entity.

To chamfer two entities using the distance-angle method

1 Do one of the following:

• Choose Modify > Chamfer.

•
On the Modify toolbar, click the Chamfer tool

• Type *chamfer* and then press Enter.

2 In the prompt box, choose Chamfer Settings.

3 In the Drawing Settings dialog box, click the Entity Modification tab.

4 Under Chamfer Distances And Angles, click Distance-Angle.

5 Under Chamfer Distances And Angles, specify the chamfer distance and angle.

6 Click OK.

7 Select the first entity.

8 Select the second entity.

Select the first (A) and second (B) entities. The chamfer is drawn, based on the distance measured along the first entity (C) and the angle (D) formed with the first entity.

To chamfer all vertices in a polyline

1   Do one of the following:

- Choose Modify > Chamfer.

-
   On the Modify toolbar, click the Chamfer tool

- Type *chamfer* and then press Enter.

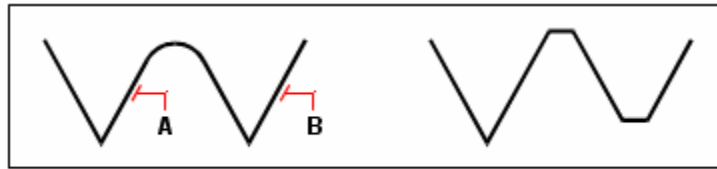2   In the prompt box, choose Polyline.

3   Select the polyline.

Original polyline.   Result after chamfering.

To chamfer selected vertices in a polyline

1   Do one of the following:

- Choose Modify > Chamfer.

-
   On the Modify toolbar, click the Chamfer tool

- Type *chamfer* and then press Enter.

2   Select the polyline along the segment where you want to begin the chamfer.

3   Select the polyline along the segment where you want to end the chamfer.

Select the polyline along the (A) and (B) segments.
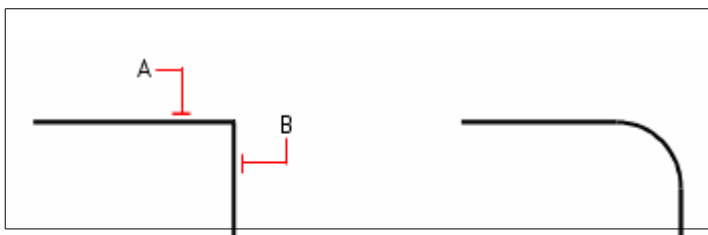
Result after chamfering.

Filleting entities

You can connect two entities with an arc of a specified radius to create a rounded edge. You can fillet pairs of line segments, straight polyline segments, arcs, circles, rays, and infinite lines. You can also fillet parallel lines, rays, and infinite lines. When filleting a polyline, you can fillet multiple segments between two selected segments,
or you can fillet the entire polyline.

To fillet two entities

1   Do one of the following:
    • Choose Modify > Fillet.
    •
      On the Modify toolbar, click the Fillet tool
    • Type *fillet* and then press Enter.

2   In the prompt box, choose Fillet Settings.
3   In the Drawing Settings dialog box, specify the fillet radius.
4   Click OK.
5   Select the first entity.
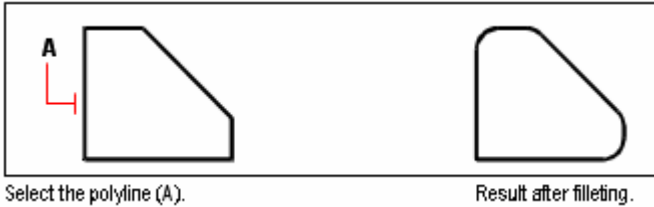6   Select the second entity.

Select the first (A) and second (B) entities.

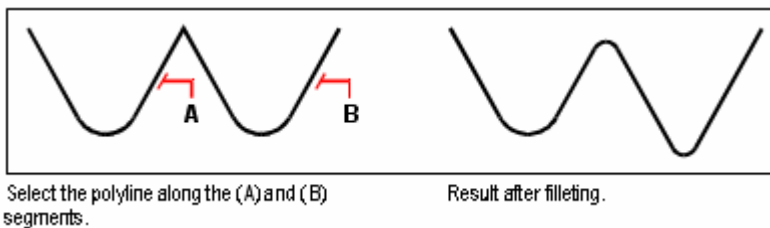Result after filleting.

To fillet an entire polyline

1   Do one of the following:

- Choose Modify > Fillet.

-
     On the Modify toolbar, click the Fillet tool

- Type *fillet* and then press Enter.
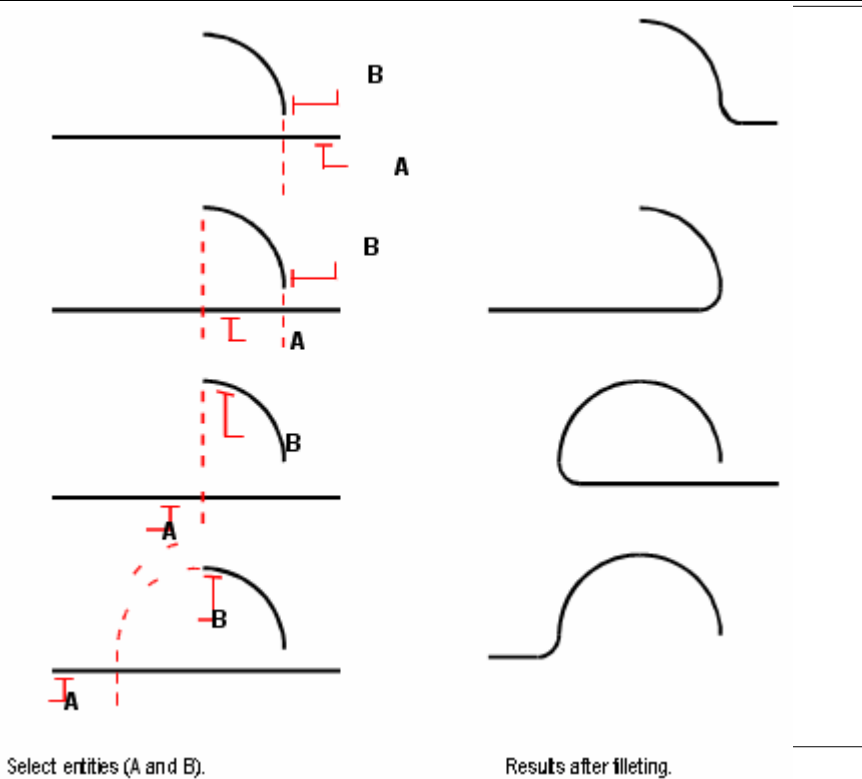
2   In the prompt box, choose Polyline.

Select the polyline (A).   Result after filleting.

3   Select the polyline.

To fillet selected vertices in a polyline

1   Do one of the following:

- Choose Modify > Fillet.

-
     On the Modify toolbar, click the Fillet tool

- Type *fillet* and then press Enter.

2   Select the polyline along the segment where you want to begin the fillet.

3   Select the polyline along the segment where you want to end the fillet.

Select the polyline along the (A) and (B)   Result after filleting.
segments.

When you fillet circles and arcs, more than one fillet can exist between the entities.

The point at which you select the entities determines the fillet.

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Select entities (A and B).   Results after filleting.

You can fillet parallel lines, rays, and infinite lines. The first entity must be a line or ray; the second entity can be a line, ray, or infinite line. The diameter of the fillet arc is always equal to the distance between the parallel entities. The current fillet radius is ignored.

Properties

Do one of the following

  Choose Modify > Properties

  On Modify toolbar, please click

The properties tab is used for editing color, layer, linetype, scale, line width, width. It shows you the

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completed properties of selected object.

No selection	
General	
Color	BYLAYER
Layer	A-WALLS-HATCH
Linetype	CONTINUOUS
Linetype scale	0'-1"
Lineweight	BYLAYER
Thickness	0"
View	
Center X	0'-8 3/4"
Center	0'-10 1/16"
Center Z	0"
Height	0'-11 1/16"
Width	1'-9 7/8"
Misc	
Ucs icon On	Yes
USC Name	