

Creating complex entities

With BtoCAD, complex entities include polylines (including rectangles and polygons), spline curves, donuts, and planes. In addition, BtoCAD includes tools for adding hatching to your drawings.

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

- Use menu commands on the Insert menu.
- Use the tools on the Draw toolbar.
- Type commands in the command bar.

When you type a drawing command or select a tool, the program prompts you to enter coordinate points, such as endpoints or insertion points. As you draw, the program displays a context-sensitive prompt box with appropriate additional options for the type of entity you are drawing.

After you create complex entities, you can modify them using the entity-modification tools. Several of these entity types require special editing commands.


Topics in this chapter

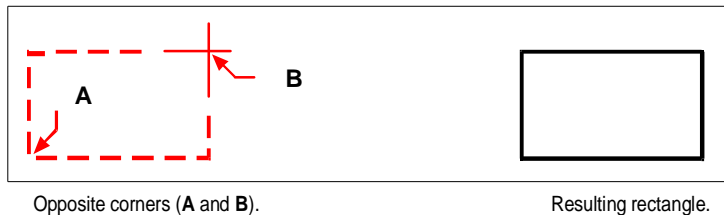
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Drawing rectangles

With BtoCAD, rectangles are closed polylines with four sides. You draw a rectangle by specifying its opposite corners. The rectangle is normally aligned parallel to the current snap and grid alignment, but you can use the Rotated option to align the rectangle to any angle.

To draw a rectangle


- 1 Do one of the following:
 - Choose Draw > Rectangle.
 - On the Draw toolbar, click the Rectangle tool ().
 - Type *rectangle* and then press Enter.
- 2 Specify one corner of the rectangle.
- 3 Specify the opposite corner of the rectangle.



You can edit each side of a rectangle individually using the Edit Polyline tool () on the Modify II toolbar. You can convert the sides into individual line entities using the Explode tool () on the Modify toolbar. You can control whether wide rectangle lines are shown filled or as outlines using the Fill tool () on the Settings toolbar.

You can also use the Rectangle tool to draw a square. Instead of specifying the opposite corners, you specify the length of one side and the alignment of the square.

To draw a square

- 1 Do one of the following:
 - Choose Draw > Rectangle.
 - On the Draw toolbar, click the Rectangle tool ().
 - Type *rectangle* and then press Enter.
- 2 In the prompt box, choose Square.
- 3 Specify one corner of the square.
- 4 Specify the length of one side of the square by selecting its other end.

The point you specify determines both the size and alignment of the square.



Endpoints of one side of the square (A and B).

Resulting square.


TIP To control the line width of the rectangle, choose the *Width Of Line* option in the prompt box. After you change the width of the line, the new width setting remains in effect for subsequent rectangles until you change it again.

Drawing polygons

Polygons are closed polylines composed of a minimum of three and a maximum of 1,024 equal-length sides. The default method for drawing a polygon is to specify the center of the polygon and the distance from the center to each vertex. You can draw a polygon using any of the following methods:

The vertex polygon drawing method creates an equal-sided polygon defined by its center point and the distance to its vertices. You specify the number of sides, the center point, and the location of one vertex, which determines both the size and orientation of the polygon.

To draw a polygon by vertex

- 1 Do one of the following:
 - Choose Draw > Polygon.
 - On the Draw toolbar, click the Polygon tool (.
 - Type *polygon* and then press Enter.
- 2 Type 5 to specify five sides for the polygon.
- 3 Specify the center of the polygon.
- 4 Specify the vertex of the polygon.



The center (A) and vertex (B).

Resulting polygon.

Drawing polylines

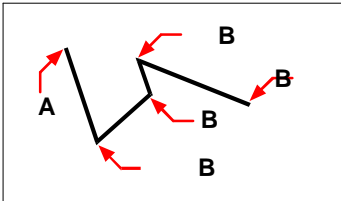
With BtoCAD, a polyline is a connected sequence of arcs and lines that is treated as a single entity. You can draw a polyline with any linetype and using a width that either remains constant or tapers over the length of any segment. When editing a polyline, you can modify the entire polyline or change individual segments.

After you specify the start point of a polyline, a prompt box provides several options as you draw, such as Distance, Halfwidth, and Width. You can specify different starting and ending widths to create a tapered polyline segment.

After you draw at least one polyline segment, you can use the Undo tool (↶) to remove the previous segment. After you draw two or more polyline segments, you can use the Close option to complete the command by drawing a segment that ends at the start point of the first polyline segment you drew. Choose Done to complete the command without closing the polyline.

To draw a polyline with straight segments

- 1 Do one of the following:
 - Choose Draw > Polyline.
 - On the Draw toolbar, click the Polyline tool (↶↷).
 - Type *polyline* and then press Enter.
- 2 Specify the start point.
- 3 Specify the endpoint of each segment.
- 4 To complete the command, choose Close or Done.




Polyline start point (A) and segment endpoints (B).

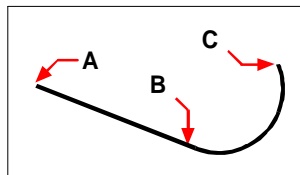
With the Draw Arcs option, you can continually draw arc segments until you select the Draw Lines option to go back to drawing line segments. When you draw arc segments, the first point of the arc is the endpoint of the previous segment. By default, you draw arc segments by specifying the endpoint of each segment. Each successive arc segment is drawn tangent to the previous arc or line segment. If you choose Close while in the Draw Arcs option, the closing segment is created as an arc.

You can also specify the arc using any of the following methods:




- Start point, included angle, center point
- Start point, included angle, radius
- Start point, center point, endpoint
- Start point, included angle, endpoint
- Start point, center point, included angle
- Start point, center point, chord length
- Start point, direction, endpoint
- Start point, radius, included angle
- Start point, second point, endpoint

To draw a line segment followed by an arc polyline segment

- 1 Do one of the following:
 - Choose Draw > Polyline.
 - On the Draw toolbar, click the Polyline tool ().
 - Type *polyline* and then press Enter.
- 2 Specify the start point.
- 3 Specify the endpoint.
- 4 In the prompt box, choose Draw Arcs.
- 5 Specify the endpoint of the arc segment.
- 6 To complete the command, choose Done.




Polyline start point (A), line endpoint/arc start point (B), and arc endpoint (C).

You can edit entire polylines and individual segments using the Edit Polyline tool () on the Modify II toolbar. You can convert polylines into arc and line entities using the Explode tool () on the Modify toolbar. You can control whether wide polylines are shown filled or as outlines using the Fill tool () on the Settings toolbar.

Drawing splines

A spline is a smooth curve defined by a set of points. You can use splines to create sculptured shapes such as the cross section of a turbine blade or an airplane wing.

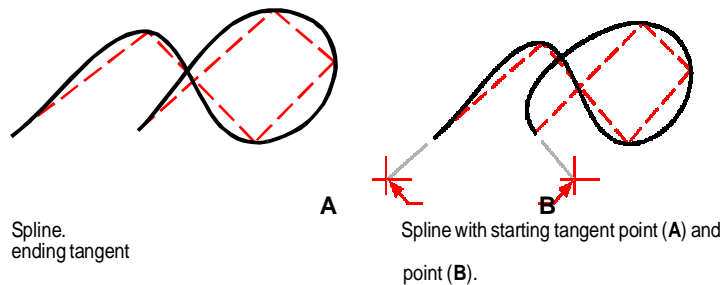
To draw a spline

- 1 Do one of the following:
 - Choose Draw > Spline.
 - On the Draw toolbar, click the Spline tool (.
 - Type *spline* and then press Enter.
- 2 Specify the first point of the spline.
- 3 Specify the second point of the spline.
- 4 Specify as many more points as you want.
- 5 When you have finished, press Enter.

You can add tangents to the spline, which are lines that give it shape.

To add tangents to a spline


- 1 Specify the starting tangent point.
- 2 Specify the ending tangent point.



Specifying fit tolerance

By default, a spline passes through all of the control points. When you draw a spline, you can change this by specifying the fit tolerance. The fit tolerance value determines how closely the spline fits the set of points you specify. For example, a spline fit tolerance value of 0 causes the spline to pass through the control points. A value of 0.01 creates a spline that passes through the start and endpoints and within 0.01 units of the intermediate control points.


To specify the fit tolerance

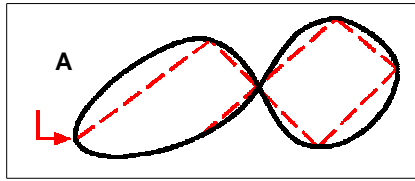
- 1 Do one of the following:
 - Choose Draw > Spline.
 - On the Draw toolbar, click the Spline tool ().
 - Type *spline* and then press Enter.
- 2 Specify the first point of the spline.
- 3 Specify the second point of the spline.
- 4 In the prompt box, choose Fit Tolerance.
- 5 To accept the default of 0.0000, press Enter. To specify a different fit tolerance, type the number, and then press Enter.
- 6 Specify the additional points you need to draw a spline or a closed spline.

Drawing a closed spline

You can draw a closed spline, which is a spline for which the start point and endpoint are the same. Because the spline is closed, you specify only one tangent.

To draw a closed spline

- 1 Do one of the following:
 - Choose Draw > Spline.
 - On the Draw toolbar, click the Spline tool ().
 - Type *spline* and then press Enter.
- 2 Specify the first point of the spline.
- 3 Specify the second point of the spline.
- 4 Specify as many more points as you want.
- 5 When you have finished, in the prompt box, choose Close.
- 6 To complete the command, specify the tangent point.



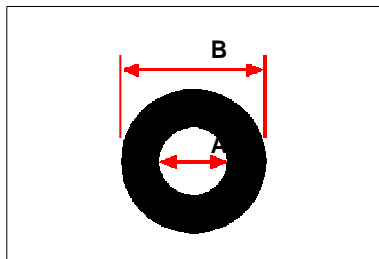
The start point and endpoint (A) of a closed spline.

Drawing donuts

Donuts are solid, filled circles or rings created as closed, wide polylines. You can draw a donut using one of several methods. The default method is to specify the inside and outside diameters of the donut, and then specify its center. You can then create multiple copies of the same donut by specifying different center points until you press Enter to complete the command.

To draw a donut

- 1 Do one of the following:
 - Choose Draw > Donut.
 - Type *donut* and then press Enter.
- 2 Specify the inside diameter of the donut.
- 3 Specify the outside diameter of the donut.
- 4 Specify the center of the donut.
- 5 Specify the center point to draw another donut, or choose Done to complete the command.



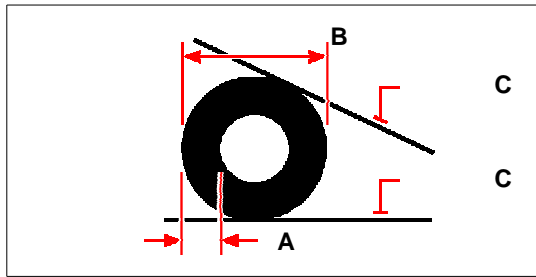
Inside diameter (A) and outside diameter (B).

The prompt box provides additional options for drawing donuts. For example, you can specify the width of the donut and two points on the diameter of the donut, or you can specify the width and three points on the donut. You can also draw a donut tangent to existing entities.




TIP A donut can be a completely filled circle by specifying an inside diameter of zero.

To draw a donut tangent to existing entities

- 1 Do one of the following:
 - Choose Draw > Donut.
 - Type *donut* and then press Enter.
- 2 In the prompt box, choose Radius Tangent Tangent.
- 3 Specify the width of the donut.
- 4 Specify the diameter of the donut.
- 5 Select the first tangent entity to which to draw the donut.
- 6 Select the second tangent entity to which to draw the donut.



Width (A) and diameter (B) of the donut and tangent entities (C).

You can edit donuts using the Edit Polyline tool () on the Modify II toolbar. You can convert donuts into arc entities using the Explode tool () on the Modify toolbar. You can control whether donuts are shown filled or as outlines using the Fill tool () on the Settings toolbar.

TIP You can control the default outside and inside diameter of donuts by choosing *Settings > Drawing Settings*, and then clicking the *Entity Creation* tab and choosing the options you want

Drawing wipeouts

Wipeouts are unique entities that can help you hide areas of your drawing. They display with the current background color, so the details behind the wipeout do not display or print. Wipeouts are similar to other entities — you can copy, mirror, array, erase, rotate, and scale them, and they can be used in both model space and paper space.

If you want to print the wipeout entities located in a drawing, you must print to a raster-capable printer. Note that in some cases you may have unexpected results when printing drawings that contain wipeout entities, for example, if printing on colored paper.


NOTE If your version of BtoCAD does not include raster image capability (that is, does not include an Image

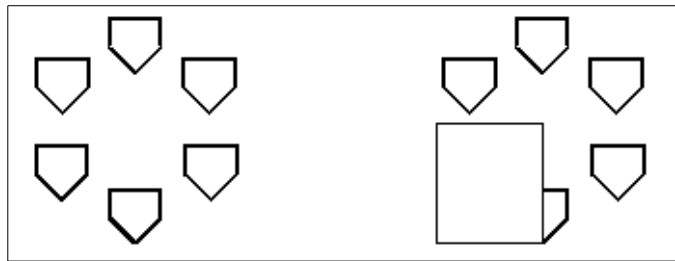
menu), wipeout entities display, but the details behind the wipeouts also display.

Drawing wipeouts using polygons and polylines

Wipeouts are created using existing polygons, closed zero-width polylines made up of only line segments, or new polylines that you draw while using the Wipeout command.

To draw a wipeout


- 1 Do one of the following:
 - Choose Draw > Wipeout.
 - On the Draw toolbar, click the Wipeout tool (.
 - Type *wipeout* and then press Enter.
- 2 Specify the start point.
- 3 Specify the endpoint of each segment.
- 4 After specifying the last endpoint, choose Done or press Enter.



Drawing without wipeout.

Drawing with wipeout.

To create a wipeout using an existing polygon or polyline


- 1 Do one of the following:
 - Choose Draw > Wipeout.
 - On the Draw toolbar, click the Wipeout tool (.
 - Type *wipeout* and then press Enter.
- 2 Choose Polyline.
- 3 Select the closed polyline to use for the wipeout.
- 4 Choose one of the following:
 - **Yes** — Creates the wipeout and deletes the polyline used to create the wipeout.
 - **No** — Creates the wipeout and keeps the polyline used to create the wipeout.

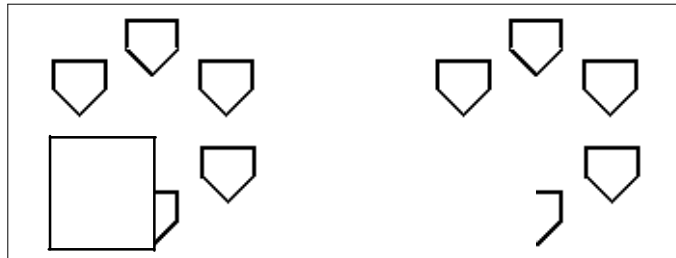
Turning wipeout frames on or off

Each wipeout has a frame along its boundary. Wipeout frames can be turned on or off for any drawing. When

wipeout frames are on, you can select and modify wipeouts. You may want to turn off wipeout frames when it's time to print.

To turn wipeout frames on or off

- 1 Do one of the following:
 - Choose Draw > Wipeout.
 - On the Draw toolbar, click the Wipeout tool (.
 - Type *wipeout* and then press Enter.
- 2 Choose Frames.
- 3 Choose On or Off.



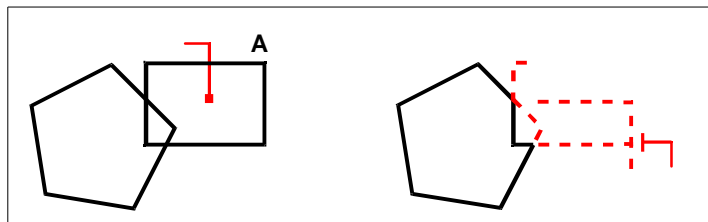
Wipeout frames turned on.

Wipeout frames turned off.

Creating boundary polylines

With the Boundary command, you can designate a specific area of a drawing for operations such as hatching and dimensioning. You create a boundary polyline by selecting an area inside a closed loop. The area you select can be bounded by a single closed entity or by multiple entities that intersect.

In cases where entities intersect, BtoCAD interprets the boundary as the closed loop closest to the point specifying the area. In the following figure, for example, the area point selected in the rectangle results in a boundary consisting of the closed loop nearest the point selection, as opposed to the closed loop formed by the rectangle itself.



Selected point (A).

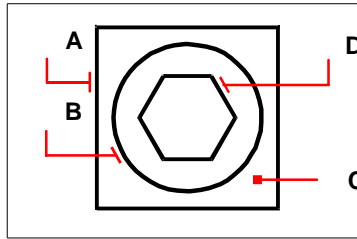
Resulting boundary (B).

To make boundaries more specific, you can create a boundary set. A boundary set specifies which entities are considered in determining the boundary path. This can make creating the boundary polyline faster if you are working with a complex drawing.

Using islands and island detection

Islands are closed loops that reside inside other closed loops. BtoCAD provides island-detection options so that you can specify which islands, if any, should be considered in the area selection for a boundary.

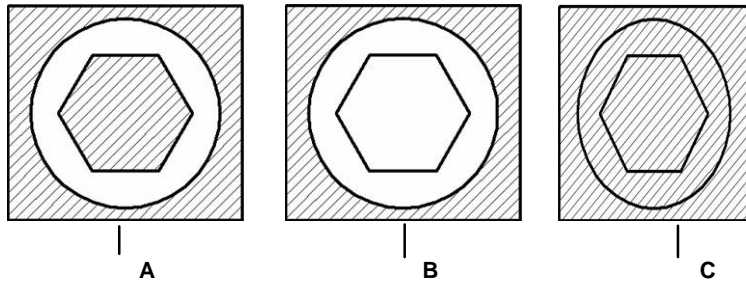
The following figure shows a rectangular polyline with two islands. The circle is referred to as the outer island, and the octagon is referred to as a nested island.



Rectangular polyline (A) with outer island (B), with point (C) specifying the area selection, and nested island (D).

You can choose from three island-detection methods.

- **Nested Islands** The outer entity and all its islands are considered for the polyline.
- **Outer Only** Only the outer entity and its outer island are considered for the polyline.
- **Ignore Islands** Only the outer entity is considered for the polyline.



Nested islands (A), with outer island (B), and with ignore islands (C).

To draw a boundary polyline

Advanced experience level

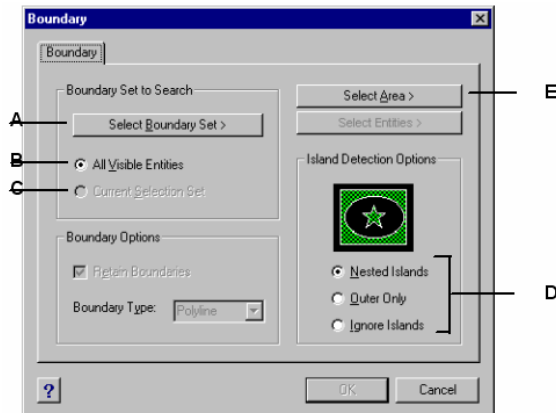
- 1 Do one of the following:
 - Choose Insert > Boundary Polyline.
 - Type *boundary* and then press Enter.

2 Specify the entities that you want made available for the boundary polyline by doing one of the following:

- **All entities** Select All Visible Entities to have all entities in the drawing considered when creating the boundary polyline.
- **Custom selection** Define only specific areas to be considered when creating the boundary polyline, which can improve system performance if you are working with a complex drawing. Click Select Boundary Set. In the drawing, select the entities individually or by choosing a selection method from the prompt box, and then press Enter. The Current Selection Set option becomes selected automatically, which indicates that the entities you selected with the Select Boundary Set button will be considered when creating the boundary polyline.

NOTE You can alternate between All Visible Entities and Current Selection Set without having to select entities again using the Select Boundary Set button. The Current Selection Set option uses the last set of entities you selected with the Select Boundary Set button.

- 3 Choose an island-detection option.
- 4 Click Select Area.
- 5 In the drawing, click inside the area whose closed perimeter forms the boundary, not on the polyline itself. If desired, continue clicking inside additional closed perimeters.
- 6 To complete the selection, press Enter.
- 7 In the Boundary dialog box, click OK.



- A Opens the drawing area for selection of entities to be considered when creating the boundary polyline.
- B Choose to consider all visible entities when creating the boundary polyline.
- C Choose to use the entities you selected for the boundary set. (Becomes available after you click the Select Boundary Set button.)
- D Select an island-detection option.
- E Opens the drawing area for selection of the enclosed areas used to create new boundary polylines.

Adding hatching


When you add hatching to a drawing, BtoCAD fills entities or enclosed areas with a pattern. You can choose a predefined hatch pattern, or you can create your own hatch pattern.

First you specify the hatch pattern and other options, and then you choose which entities or enclosed areas that you want to hatch.

NOTE *Hatch patterns are memory intensive and can take a considerable amount of time to draw and display. To improve performance, add hatching as one of the last steps when you create a drawing, or insert hatches on a separate layer that you can freeze as you continue to work on your drawing.*

To open the Boundary Hatch dialog box

Do one of the following:

- Choose Draw > Hatch.
- On the Draw toolbar, click the Boundary Hatch tool ()
- Type *bhatch* and then press Enter.


Specifying a hatch pattern

A hatch pattern consists of a repeating pattern of lines, dashes, and dots. You can select a hatch pattern from a set of predefined patterns, or you can define a pattern of your own. The hatch pattern you used most recently is the default pattern the next time you add hatching.

The program supplies predefined standard hatch patterns, which are stored in the *icad.pat* and *icadiso.pat* hatch pattern library files. You can use other external hatch pattern libraries, such as an office standard library, customized patterns, and libraries available from vendors or standards organizations.

NOTE *The icad.pat hatch pattern library files are ANSI (American National Standards Institute)-compliant patterns; the icadiso.pat hatch pattern library files are ISO (International Standards Organization)-compliant.*

To specify a predefined hatch pattern

- 1 Do one of the following:
 - Choose Draw > Hatch.
 - On the Draw toolbar, click the Boundary Hatch tool ()
 - Type *bhatch* and then press Enter.
- 2 From the Boundary Hatch dialog box, click the Pattern Properties tab.
- 3 In the Pattern Type list, click Predefined. With Predefined, you can apply a scale factor to make the pattern larger or smaller than the default size.
- 4 For Scale, enter the scale factor as a percentage of the default.
- 5 For Angle, enter the angle of the pattern in degrees (1-360).

The default angle is clockwise; you can change the angle of any hatch pattern by entering a numerical value.

6 For ISO Pen Width, enter the pen width. If you choose a predefined, ISO standard pattern, you can scale the pattern based on the ISO pen width.

7 To copy the pattern properties from an existing hatch, choose Copy Hatch Properties and select the hatch.


8 To associate the hatch pattern to its boundary entities, under Hatch Attributes, select the Associative check box. An associative hatch updates automatically if you move any of its boundaries.

9 To continue, add a hatch by doing one of the following:

- Select the entities you want to add a hatch. For details, see “Selecting entities for hatching”.
- Select an area or boundary you want to add a hatch. For details, see “Selecting areas for hatching” .

To specify a user-defined hatch pattern

1 Do one of the following:

- Choose Draw > Hatch.
- On the Draw toolbar, click the Boundary Hatch tool (.
- Type *bhatch* and then press Enter.

2 From the Boundary Hatch dialog box, click the Pattern Properties tab.

3 In the Pattern Type list, click User Defined.

4 For Spacing, enter the line spacing for the pattern.

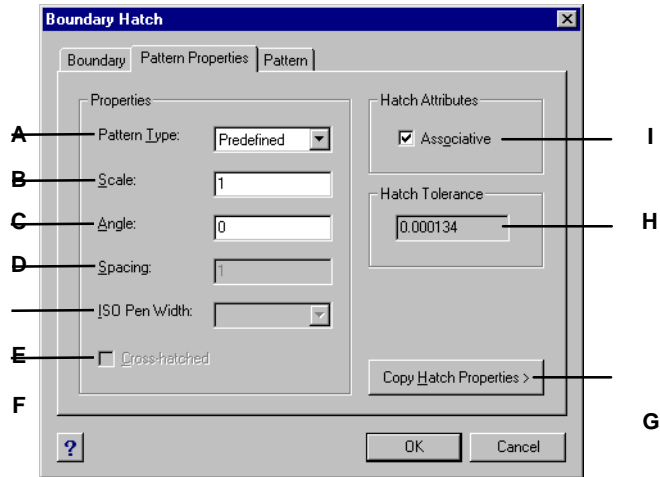
5 To crosshatch the pattern, select the Cross-Hatched check box. You can choose to crosshatch the pattern. Cross-hatching imposes a copy of the specified user-defined pattern at a 90-degree angle over the first pattern you defined.

6 To copy the pattern properties from an existing hatch, choose Copy Hatch Properties, and select a hatch pattern from a hatched entity in the drawing.

7 To associate the hatch pattern to its boundary entities, under Hatch Attributes, select the Associative check box. An associative hatch updates automatically if you move any of its boundaries.


8 To continue, add a hatch by doing one of the following:

- Select the entities you want to add a hatch. For details, see “Selecting entities for hatching”.
- Select an area or boundary you want to add a hatch. For details, see “Selecting areas for hatching”.

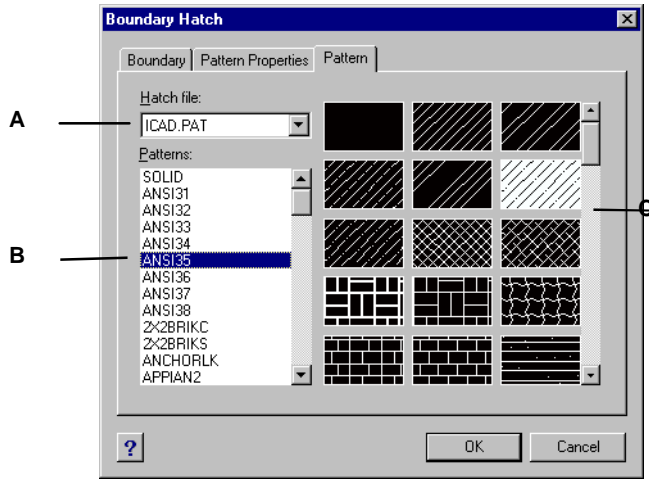


- A Determines how the hatch will be defined.
- B Determines density of hatching for Predefined hatch patterns.
- C Sets angle of hatch in relation to entity (User Defined only).
- D Determines density of hatching for User Defined hatch patterns.
- E Sets pen width for ISO-standard pattern.
- F Imposes another copy of the specified pattern at a 90-degree angle over the first.
- G Closes dialog to allow selection and copying of existing hatch pattern properties.
- H Displays the tolerance that non-touching entities can be within and still be used to create the hatch pattern boundary.
- I Select to update the hatch automatically if you move any of its boundaries.

To use a predefined library pattern

- 1 Do one of the following:
 - Choose Draw > Hatch.
 - On the Draw toolbar, click the Boundary Hatch tool (.
 - Type *bhatch* and then press Enter.
- 2 From the Boundary Hatch dialog box, click the Pattern tab.
- 3 For Hatch File, select the *icad.pat* or *icadiso.pat* hatch pattern library file.
- 4 To select a predefined pattern, do one of the following:
 - In the Patterns list, click the pattern name.
 - Click the graphical representation of the hatch pattern.
- 5 To continue, add a hatch by doing one of the following:

- Select the entities you want to add hatching. For details, see “Selecting entities for hatching”.
- Select an area or boundary you want to add hatching. For details, see “Selecting areas for hatching”.




- A Hatch pattern library files.
 B Hatch patterns listed by name.
 C Hatch patterns shown graphically.

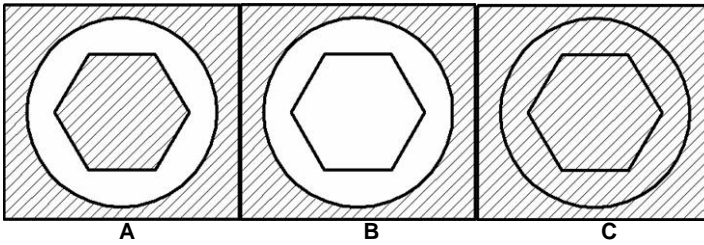
NOTE Hatch patterns are memory intensive and can take a considerable amount of time to draw and display. To improve performance, add hatching as one of the last steps when you create a drawing, or insert hatches on a separate layer that you can freeze as you continue to work on your drawing.

Selecting entities for hatching

You can add hatching to any entity that forms a closed boundary, for example, a circle or rectangle. You can assign hatching to a single entity or several entities at the same time.

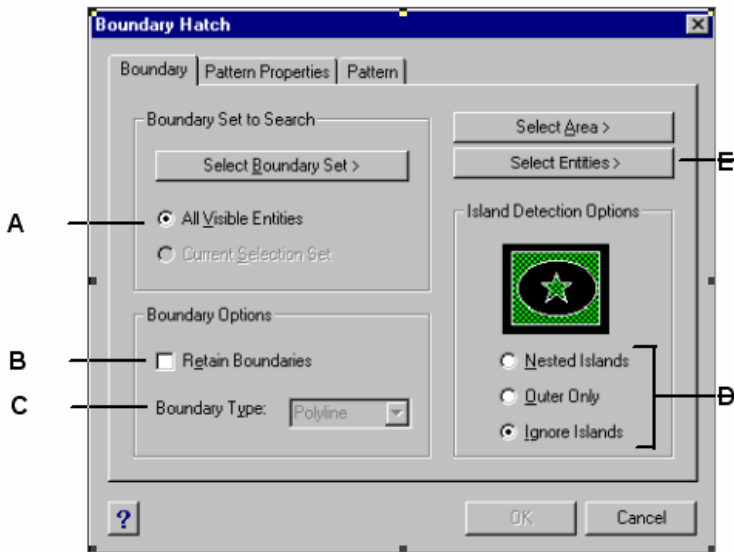
To select entities for hatching

- 1 Do one of the following:
 - Choose Draw > Hatch.
 - On the Draw toolbar, click the Boundary Hatch tool (.
 - Type *bhatch* and then press Enter.
- 2 From the Boundary Hatch dialog box, click the Boundary tab.
- 3 Under Island Detection Options, choose one of the following:
 - **Nested Islands** The outer entity and all its islands are considered for hatching.
 - **Outer Only** Only the outer entity and its outer island are considered for hatching.
 - **Ignore Islands** Only the outer entity is considered for hatching.



Nested islands (A), with outer island (B), and with ignore islands (C).

- 4 To keep any new entities that are created for drawing the boundary hatch, select the Retain Boundaries check box. Existing entities are always retained.
- 5 Click Select Entities.
- 6 In the drawing, click the entities to be hatched individually or by choosing a selection method from the prompt box, and then press Enter when done.
- 7 In the Boundary Hatch dialog box, click OK.




- A Choose to consider all visible entities when creating the boundary hatch.
- B Mark the check box to keep any new entities that are created to draw the boundary hatch. Existing entities are always retained.
- C (Display only) Indicates the boundary is created as a polyline.
- D Determines how hatching interacts with islands.
- E Opens the drawing area for selection of entities to be hatched.

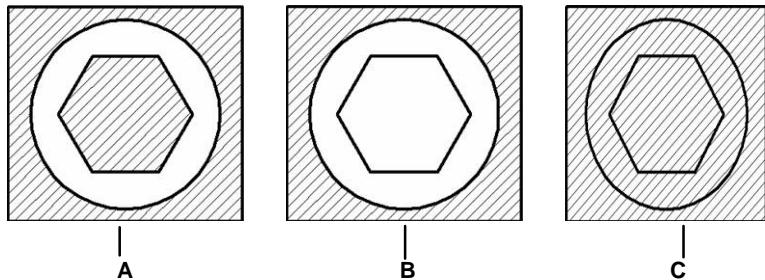
Selecting areas for hatching

You can add hatching to an area enclosed by selected entities to form the hatch boundary. The hatch is

formed in the enclosed area, not the entities themselves. After BtoCAD draws the hatch, the entire hatch is treated as a single entity and it is either associative or

To select an area for hatching

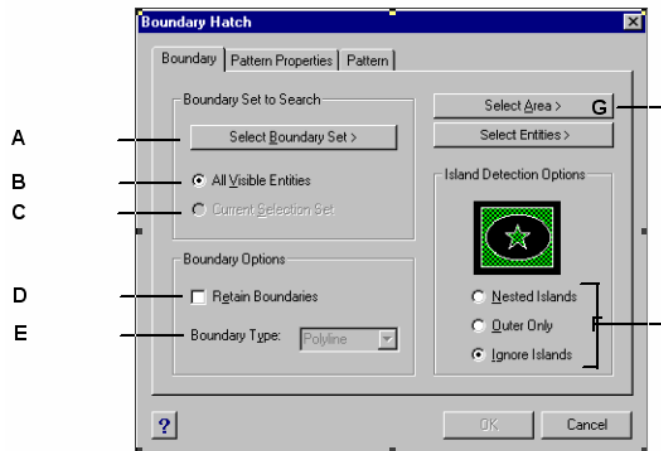
- 1 Do one of the following:
 - Choose Draw > Hatch.
 - On the Draw toolbar, click the Boundary Hatch tool (.
 - Type *bhatch* and then press Enter.
- 2 From the Boundary Hatch dialog box, click the Boundary tab.
- 3 Under Island Detection Options, choose one of the following:
 - **Nested Islands** The outer entity and all its islands are considered for hatching.
 - **Outer Only** Only the outer entity and its outer island are considered for hatching.
 - **Ignore Islands** Only the outer entity is considered for hatching.



Nested islands (A), with outer island (B), and with ignore islands (C).

- 4 To keep any new entities that are created for drawing the boundary hatch, select the Retain Boundaries check box. Existing entities are always retained.
- 5 Specify the entities that you want made available for boundary hatching by doing one of the following:
 - **All entities** Select All Visible Entities to have all entities in the drawing considered when creating the boundary hatch.
 - **Custom selection** Define only specific areas to be considered when creating the boundary hatch, which can improve system performance if you are working with a complex drawing. Click Select Boundary Set. In the drawing, select the entities individually or by choosing a selection method from the prompt box, and then press Enter. The Current Selection Set option becomes selected automatically, which indicates that the entities you selected with the Select Boundary Set button will be considered when creating the boundary hatch.

- 6 In the Boundary Hatch dialog box, click Select Area.
- 7 In the drawing, click inside the closed perimeter of a boundary, not on the boundary itself. If desired, continue clicking inside additional closed perimeters.
- 8 To complete the selection, press Enter.
- 9 In the Boundary Hatch dialog box, click OK.



- A Opens the drawing area for selection of entities to be considered when creating the boundary hatch.
- B Choose to consider all visible entities when creating the boundary hatch.
- C Choose to use the entities you selected for the boundary set. (Becomes available after you click the Select Boundary Set button.)
- D Mark the check box to keep any new entities that are created to draw the boundary hatch. Existing entities are always retained.
- E (Display only) Indicates the boundary is created as a polyline.
- F Determines how hatching interacts with islands.
- G Opens the drawing area for selection of enclosed areas to be hatched.