

Working with drawings

CAD drawings help you organize information for greater efficiency. With BtoCAD, you can draw entities representing different types of information on various layers and use those layers to control color, linetype, and visibility. BtoCAD also includes drawing aids that help you draw accurately.

This section explains setting up drawings and using built-in drawing aids, including how to:

- Create new drawings, open existing drawings, and save changes to drawings.
- Use aids such as the grid, snap, and orthogonal settings to draw accurately.

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Creating a new drawing

When you start BtoCAD, the program automatically creates a new drawing based on a template drawing, `icad.dwt`. This template includes predefined settings such as drawing units, text size, print style table type, and drawing area. You can either use these settings or change them according to your needs. There is nothing unique about a template drawing. You can use any drawing as a template for future drawings.

You can save many steps by basing a new drawing on an existing template (.dwt file). By doing this, a new drawing will contain all the settings and entities you need. When you open a new drawing from your custom template, you can modify existing settings and delete any entities that you don't need. When you save a drawing that was created using a template, you do not change the template.

To create a new drawing based on a template

- 1 Do one of the following:
 - Choose File > New.
 - Type *newwiz* and then press Enter.
- 2 Click Use a Template Drawing, and then click Next.
- 3 To display the Open Template dialog box, click Browse.
- 4 Select the template (.dwt) file that you want, and then click Open. You can also choose any drawing (.dwg) file to use as a template.
- 5 Click Finish.

Opening a drawing

You can open drawing (.dwg) files, Drawing Exchange Format (.dxf) files, Design Web Format™ (.dwf) files, and drawing template (.dwt) files. You can also open and check drawings that you suspect are damaged.

Opening an existing drawing


You can open any of these drawing files:

- Standard drawing files with a .dwg extension.
In addition to your own drawing files, you can open and use one of the sample drawings that are included with BtoCAD.
- Drawing Exchange Format files with a .dxf file extension.
- Design Web Format files with a .dwf file extension.

- Drawing templates with a .dwt file extension.

To open an existing drawing

1 Use one of the following methods:

- Choose File > Open.
- On the Standard toolbar, click the Open tool 
- Type *open* and then press Enter.

2 In Files of Type, choose the type of file you want to open.

3 Choose the folder containing the desired file.

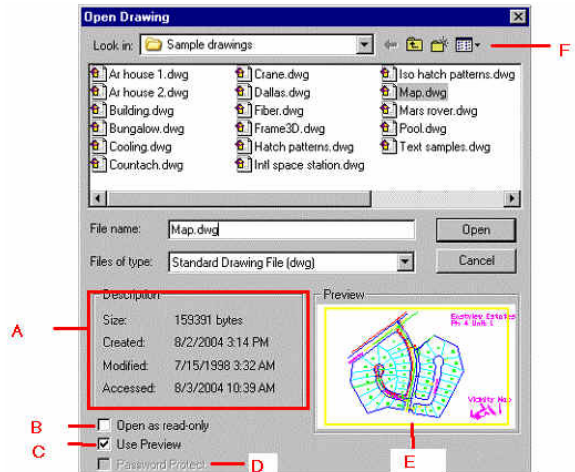
4 Do one of the following:

- Choose the drawing you want to open, and then click Open.
- Double-click the drawing you want to open.

If the drawing requires a password, enter password, click OK to verify, and then click Open again.

TIP You can also open drawings while browsing files on your computer using programs that came with your operating system, such as Windows File Explorer or My Computer. Simply double click the file to open it. Easily find the drawing you want by viewing thumbnail images of the drawing files as you browse them.

- A Displays a description of the file size, creation date, and other information about the drawing.
- B Opens the drawing as read-only to prevent making changes to the file.
- C Turns the drawing preview on or off.
- D Unavailable when opening drawings; available only when saving drawings.
- E If a thumbnail image exists in the selected drawing, displays an image of the drawing before you open it.
- F Defines how drawings display in the list, including file details and thumbnail images.



TIP To quickly open a drawing file that you recently used, choose File > <file name>. The program tracks the last four drawings.

Opening damaged files

Files can become damaged for many reasons. For example, if you are working on a drawing during a power outage, a system crash, or a hardware failure, your drawing file may become damaged. BtoCAD allows you to open and check damaged files to attempt file recovery.

Recovering a file attempts to open one of the following file types:

- Standard drawing files with a .dwg extension.
- Drawing Exchange Format files with a .dxf file extension.
- Design Web Format files with a .dwt file extension.
- Drawing templates with a .dwt file extension.

You can also audit any open file to check it for errors. You specify whether you want BtoCAD to fix any errors that are found automatically. BtoCAD fixes as many errors as possible and any errors that cannot be fixed are reported as “Ignored” in the Prompt History window.

To open a damaged file

- 1 Use one of the following methods:
 - Choose File > Drawing Utilities > Recover.
 - Type *recover* and then press Enter.
- 2 In Files of Type, choose the type of file you want to recover.
- 3 Choose the directory containing the damaged file.
- 4 Choose the damaged file you want to recover.
- 5 Click Open.

If you want to check all drawings for errors automatically when you open them, choose Tools > Options > General tab and mark the check box for Open Drawings using Recover.

To check a drawing file for errors

- 1 With the drawing open that you want to check, do one of the following:
 - Choose File > Drawing Utilities > Audit.
 - Type *audit* and then press Enter.
- 2 Choose whether you want BtoCAD to fix any found errors automatically, and then press Enter.

NOTE If the *AUDITCTL* system variable is set to *On* and errors are found during a file recovery or audit, an *ASCII* file is created that describes the audit. The *ASCII* file is saved in the same folder as the audited drawing and has the same name as the drawing file, but with an .adt extension.


Setting up a drawing

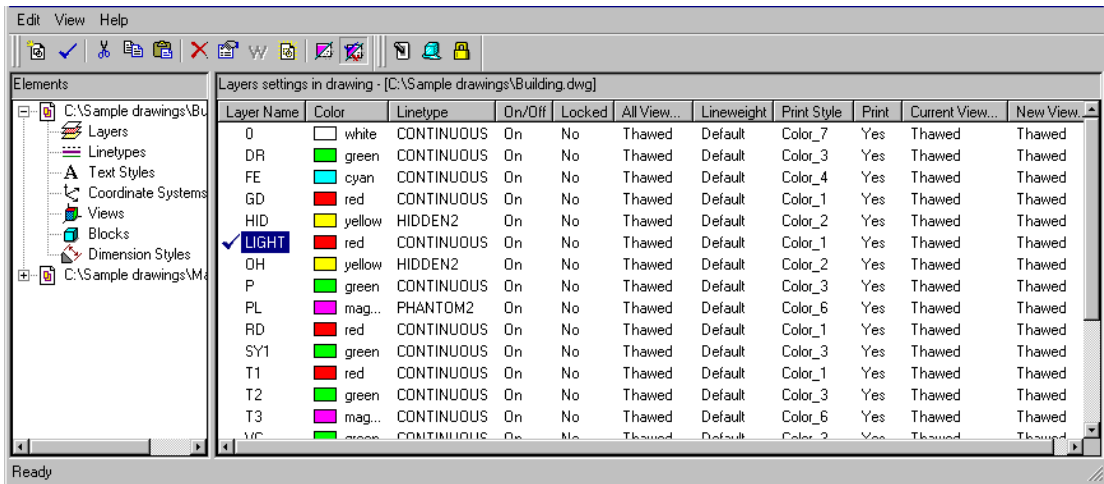
You can specify individual settings when you create a new drawing or when you modify settings in a drawing created from a template.

Setting the current layer

Layers are like the overlays that you use in manual drafting. You use layers to organize different types of drawing information. Every drawing has at least one layer, the default layer, named “0.” Your drawing can also contain an unlimited number of additional layers. When you create an entity, it is created on the current layer.

To set the current layer

- 1 Do one of the following:
 - Choose Tools > Explore Coordinate Systems....
 - On the Settings toolbar, click the Set Layer by Entity tool 
 - Type *explayers* and then press Enter.
 - Type *la* and then press Enter.
- 2 Double-click the layer name that you want to make current.
- 3 Close the BtoCAD Explorer window.



Double-click the layer name that you want to make current.

TIP On the status bar, right-click on the current layer control, and from the list, select the layer you want to make current.


Setting the current entity color

An entity's color determines how it is displayed and, if you are using a color printer, how it prints. Entities are created in the current color.

When you open a new drawing, entities are created in the color **BYLAYER**, which adopts the color of the current layer. Initially, layer 0 is both the only layer and the current layer. Its default color is white, so your entities appear as white.

There are index colors, which contain two additional color properties that are often referred to as colors, true colors, and color book colors. The two additional color properties are **BYLAYER** and **BYBLOCK**. These color properties cause an entity to adopt the color either of the layer or of the block in which it is a member.

To set the current entity color

- 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 Click Select Color.
- 4 In the Color dialog box, click one of the following tabs and select a color:
 - **Index Color** — Click **BYBLOCK**, **BYLAYER**, or one of the 255 index colors. You can also type the color number in the Index box.
 - **True Color** — Click a basic color, click a color in the color palette, enter the Hue, Saturation, and Luminance (HSL) values, or enter the Red, Green, Blue (RGB) values. There are more than 16 million true colors from which you can choose.
 - **Color Books** — Select a color book from the list, then click a color. You can select Show Only Color Book Colors Used in Drawing to limit the selection to only those color book colors that are used in the current drawing.
- 5 Click OK.
- 6 Click OK again.

TIP On the status bar, right-click on the current color, and select from the list the color you want to use for new entities. You can also click Select Color to choose additional colors.

For more details about using color in the many aspects of your drawing, see “Working with colors”.

Setting the current linetype


Linetypes help convey information. You use different linetypes to differentiate the purpose of one line from another. A linetype consists of a repeating pattern of dots, dashes, or blank spaces. Linetypes determine the appearance of entities both on the screen and when printed. By default, every drawing has at least three linetypes: CONTINUOUS, BYLAYER, and BYBLOCK. Your drawing may also contain an unlimited number of additional linetypes.

When you create an entity, it is created using the current linetype. By default, the current linetype is BYLAYER. AutoCAD indicates that the entity linetype is determined by the current layer's linetype by assigning the BYLAYER property as the default linetype setting. When you assign BYLAYER, changing a layer's linetype changes the linetype of all the entities assigned that layer (if they were created using the linetype BYLAYER).

You can also select a specific linetype as the current linetype, which overrides the layer's linetype setting. Entities are then created using that linetype, and changing the layer linetype has no effect on them.

As a third option, you can use the linetype BYBLOCK property, in which case new entities are drawn using the CONTINUOUS linetype until you group them into a block. The entities then inherit the block's linetype setting when you insert the block into a drawing.

To set the current linetype

- 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 In the Entity Linetype list, choose the linetype that you want to make current.
- 4 Click OK.

TIP On the status bar, right-click the word *BYLAYER* for the current linetype, click *Properties*, and then choose the linetype that you want to make current.

Setting the linetype scale


You can specify the linetype scale. The smaller the scale, the more repetitions of the linetype pattern are generated per drawing unit. For example, a linetype pattern is defined as a sequence of dashed lines and open spaces, each 0.25 units long. The linetype scale uses the drawing scale factor to determine the length. A scale factor of 0.5 would reduce the length of each line and space to 0.125 units; a scale factor of 2 would increase

the length of each to 0.5 units.


NOTE *Setting the linetype scale too large or too small may result in a line pattern looking like a solid line, depending on what the scale view is or at what scale the drawing is printed.*

You can control a new entity's individual linetype scale factor as well as the overall or global scale factor applied to all the entities in the drawing.

To set the current individual linetype scale

- 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 In the Linetype Scale field, type or choose the linetype scale that you want to make current.
- 4 Click OK.

To change the global linetype scale

- 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 In the Global Linetype Scale field, type or choose the global linetype scale that you want to change.
- 4 Click OK.

Setting the current lineweight

Lineweights help differentiate the purpose of one line from another. Lineweights determine how thick or thin entities appear both on the screen and when printed. Every drawing has these lineweights: DEFAULT, BYLAYER, BYBLOCK, and many additional lineweights in millimeters (or you can use inches).

When you create an entity, it is created using the current lineweight. By default, the current lineweight for a new entity is BYLAYER. This means that the entity line- weight is determined by the current layer. When you assign BYLAYER, changing a layer's lineweight changes the lineweight of all the entities assigned that layer (if they were created using the lineweight BYLAYER).

You can also select a specific lineweight (or DEFAULT) as the current lineweight, which overrides the layer's lineweight setting. Entities are then created using that lin- eweight (or the DEFAULT lineweight), and

changing the layer lineweight has no effect on them.


As a third option, you can use the lineweight BYBLOCK property, in which case new entities are drawn using the DEFAULT lineweight until you group them into a block. The entities then inherit the block's lineweight setting when you insert the block into a drawing.

If you choose a lineweight that is less than .025 millimeter, it displays as one pixel when you create your drawing. When you print your drawing, it prints at the thinnest lineweight that is available for your printer.

You cannot assign linewidths to planes, points, TrueType fonts, or raster images (if supported in your version of BtoCAD).

To set the current lineweight

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 In the Lineweight list, choose the lineweight that you want to make current.

4 Click OK.

TIP On the status bar, right-click the word *BYLAYER* for the current lineweight, and then choose the current lineweight. You can also double-click the word *LWT* to toggle the display of linewidths on and off.

NOTE To see linewidths in your drawing, you may need to turn on linewidths. For details, see see “Controlling the display of linewidths”.

Setting the current print style

Print styles are used to change the appearance of an entity when it prints, without actually changing the entity in the drawing.


If your drawing uses named print style tables, you can specify a print style for any entity. Named print style tables contain print styles that you set up. If your drawing uses color-dependent print style tables, the print style is BYCOLOR, which cannot be changed. These types of print style tables determine printing requirements by the color assigned to an entity or layer. For details about converting a drawing that uses color-dependent print style tables to use named print style tables, see “Changing a drawing's print style table type”.

When you create an entity in a drawing that uses named print style tables, the entity is created using the current print style. By default, the current print style is BYLAYER. When you assign BYLAYER, changing a layer's print style changes the print style of all the entities assigned that layer if they were created using the print style BYLAYER.

You can also select a specific print style as the current print style, which overrides the layer's print style setting. Entities are then created using that print style, and changing the layer print style has no effect on them.

As a third option, you can use the print style **BYBLOCK**, in which case new entities use the Normal print style until you group them into a block. The entities then inherit the block's print style setting when you insert the block into a drawing.

To set the current print style in a drawing that uses named print style tables

- 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 In the Print Style list, choose the print style that you want to make current. If necessary, choose Other and then select a print style.
- 4 Click OK.


TIP On the status bar, right-click the word **BYLAYER** for the current print style, click *Other*, and then choose the print style that you want to make current. Or, type *printstyle* to choose the current print style.

Setting drawing units

With BtoCAD, you typically draw at full-size (1:1 scale), and then set a scale factor when you print or plot your drawing. Before you begin drawing, however, you need to determine the relationship between drawing units and real-world units.

For example, you can decide whether one linear drawing unit represents an inch, a foot, a meter, or a mile. In addition, you can specify the way the program measures angles. For both linear and angular units, you can also set the degree of display precision, such as the number of decimal places or smallest denominator used when displaying fractions. The precision settings affect only the *display* of distances, angles, and coordinates. BtoCAD always stores distances, angles, and coordinates using floating-point accuracy.

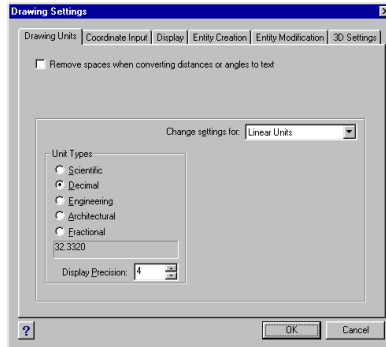
To set the linear drawing units

- 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 Drawing Units tab.

- 3 Under Change Settings For, choose Linear Units.
- 4 Under Unit Types, select a unit type.
- 5 Under Display Precision, type the display precision according to the number of decimal places you want, or click the arrows to select it.

The field above this setting shows an example of the linear unit type at the current precision.


- 6 Click OK.



A Select the type of linear units.

B Choose the display precision for linear units.

To set the angular drawing units

- 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 Drawing Units tab.
- 3 Under Change Settings For, choose Angular Units.
- 4 Under Unit Types, select a unit type.
- 5 Under Display Precision, type the display precision according to the number of decimal places you want, or click the arrows to select it. The field above this setting shows an example of the angular unit type at the current precision.
- 6 Under Angle Direction, select the direction in which angles increase when you specify a positive angle value.
- 7 Under Angle Base, specify the compass location for the zero angle. For example, by default, angle 0 is at the “three o’clock” or “east” position. The adjacent icon shows the current location of the angle base.
- 8 Click OK.

Understanding scale factors

Instead of drawing to a particular scale, you draw everything in the program full-size. Although it’s a good idea to keep your scale factor in mind when setting up a drawing, you don’t need to set the scale until you

print it. For example, when you draw a mechanical part 40 inches in length with BtoCAD, you actually draw it as 40 inches, rather than applying a scale factor as you draw. When you print your drawing, you can assign the scale at which the drawing is to print.

Scale, however, does affect the way a few elements such as text, arrows, or linetypes print or plot in your drawing. For these, you can make adjustments when you first set up your drawing so that they print or plot at the correct size. For example, when you draw text, you need to determine the text size so that when you print it later at a particular scale, the text height is correct.

After you determine the eventual scale of your finished drawing, you can calculate the scale factor for the drawing as a ratio of one drawing unit to the actual scale unit represented by each drawing unit. For example, if you plan to print your drawing at $1/8" = 1'-0"$, your scale factor ratio is 1:96 ($1/8" = 12"$ is the same as $1 = 96$). If you want your printed scale to be 1 inch = 100 feet, your scale factor ratio is 1:1200.

The following table shows some standard architectural and engineering scale ratios and equivalent text heights required to create text that measures 1/8-inch high when you print the drawing at the specified scale.

Standard scale ratios and equivalent text heights

Scale	Scale factor	Text height
$1/16" = 1'-0"$	192	24"
$1/8" = 1'-0"$	96	12"
$3/16" = 1'-0"$	64	8"
$1/4" = 1'-0"$	48	6"
$3/8" = 1'-0"$	32	4"
$1/2" = 1'-0"$	24	3"
$3/4" = 1'-0"$	16	2"
$1" = 1'-0"$	12	1.5"
$1\ 1/2" = 1'-0"$	8	1"
$3" = 1'-0"$	4	0.5"
$1" = 10'$	120	15"
$1" = 20'$	240	30"
$1" = 30'$	360	45"
$1" = 40'$	480	60"
$1" = 50'$	600	75"
$1" = 60'$	720	90"
$1" = 100'$	1200	150"

You can use these scale factors to predetermine the size of your drawing to make sure that it fits on a specific size paper when you print it. You control the size of your drawing by the drawing limits. To calculate the drawing limits to match the size of your paper, multiply the dimensions of your paper size by your scale factor.

For example, if the paper you use to print measures 36 inches x 24 inches and you print your drawing at $1/8" = 1'-0"$ (in other words, using a scale factor of 96), the size of your drawing measured in drawing units is 36 x 96 (or 3,456 units) wide and 24 x 96 (or 2,304 units) high.



Keep in mind that you can print the finished drawing at any scale, regardless of the scale factor you calculate. You can also print on paper of a different size and use the Layout tabs to create different views of your drawing and to position and scale those views differently. The scaling factor is not related to the size of the entities you draw; it simply provides a preliminary guide to help you establish the text height and drawing limits when you begin your drawing. You can change the text height and drawing limits at any time.

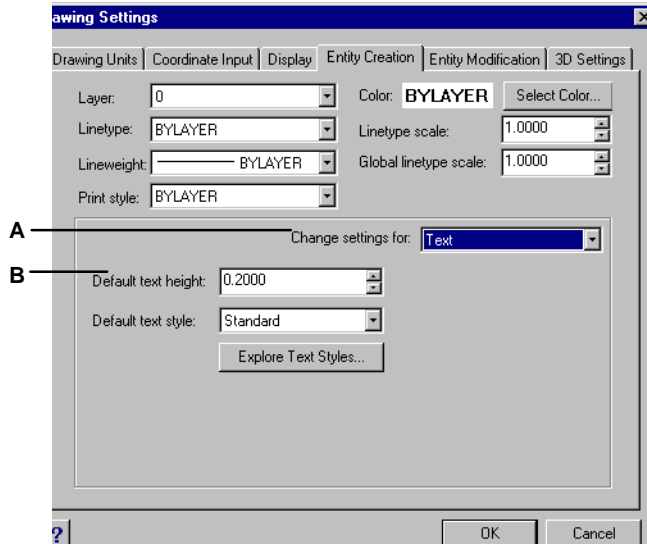
Setting the text height

The text height setting controls the height of text, measured in drawing units. Set this value initially so that text used for your most common annotations, when scaled to the size at which you will print a drawing, measures 1/8-inch high on the printed drawing.

For example, if you plan to print your drawing at $1/8" = 1'-0"$ and you want your text to be 1/8-inch high in the final drawing, create that text 1 foot high (in your real-world drawing units) so that when you print it, it appears 1/8-inch high on the paper. You must create text 4 feet high that you want to print 1/2-inch high.

To set the text height

- 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, choose Text.
- 4 In the Default Text Height field, select the text height or type the text height value that you want.
- 5 Click OK.




- A Choose Text
B Specify the text height in drawing units.

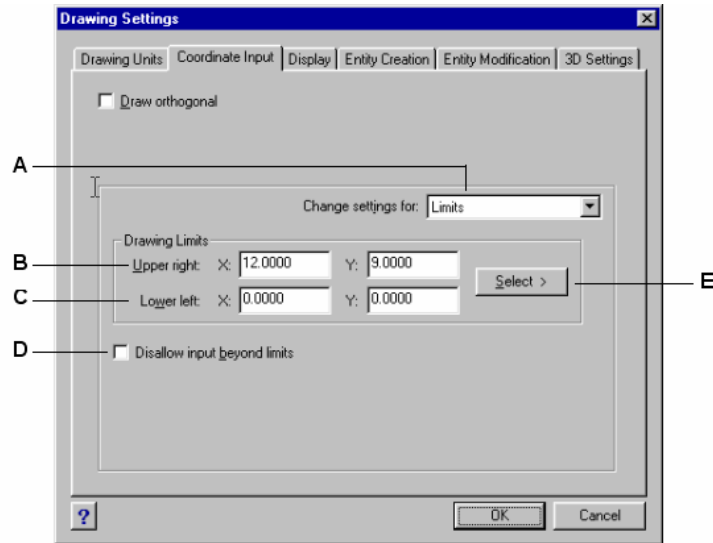
Setting the drawing limits

You can specify the drawing limits that form an invisible boundary around your drawing. You can use the drawing limits to make sure that you do not create a drawing larger than can fit on a specific sheet of paper when printed at a specific scale.

For example, if you plan to print your drawing at $1/8" = 1'-0"$ (in other words, using a scale factor of 96) on a sheet of paper measuring 36 inches x 24 inches, you can set drawing limits to 3,264 units wide (that is, 34×96) and 2,112 units high (22×96), which allows a 1-inch margin around the edges of the printed image.

To set the drawing limits

- 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 either the Coordinate Input tab or the Display tab.
- 3 Under Change Settings For, choose Limits.
- 4 Specify the x-coordinate and y-coordinate of the upper right drawing limit and the lower left drawing limit. You can also click Select to specify the drawing limits by selecting points in the drawing.
- 5 To constrain your drawing to these drawing limits, click the Disallow Input Beyond Limits check box.
- 6 Click OK.



- A Choose limits.
- B Specify the x-coordinate and y-coordinate of the upper right drawing limit.
- C Specify the x-coordinate and y-coordinate of the lower left drawing limit.
- D When you click this check box, the program prevents you from drawing outside the drawing limits.
- E Specify the drawing limits by selecting points in the drawing.

Working with colors

An entity's color determines how it is displayed and, if you are using a color printer, how it prints. Entities are created in the current color specified for the drawing.

Layers can also control the color of entities. When you open a new drawing, entities are created in the color BYLAYER, which adopts the color of the current layer. Initially, layer 0 is both the only layer and the current layer. Its default color is white, so your entities appear as white.

For entities and layers in BtoCAD, there are three different types of colors:

- Index colors
- True colors
- Color book colors

True colors and color books are unavailable in some cases, such as dimension entities and cursor display.

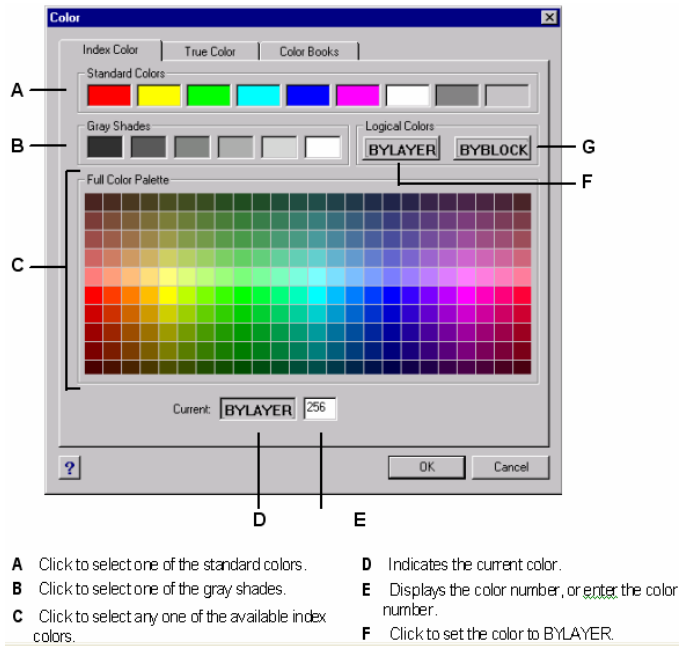
You can choose colors by selecting them from the Color dialog box. In the command bar or in some dialog boxes, you specify a color either by name or by number.

Using index colors

There are 255 standard index colors and two additional color properties that are often referred to as colors (BYLAYER and BYBLOCK). You can use seven of the 255 standard index colors by name: red, yellow, green, cyan, blue, magenta, and white. (Numbers eight and nine are not named.) Each index color has a unique number from 1 to 255. The two additional color properties are BYLAYER and BYBLOCK. These color properties cause an entity to adopt the color either of the layer or of the block in which it is a member. BYLAYER is color number 256, and BYBLOCK is color number 0. In all commands for which you would use a color, you can indicate BYLAYER and BYBLOCK as well as by numbers 256 and 0, respectively.

To select an index color

- 1 Click Select Color in the desired dialog box, such as Layers, Drawing Settings, Properties, or Multiline Text. The Color dialog box opens.
- 2 Click the Index Color tab.
- 3 Do one of the following:
 - Click BYBLOCK.
 - Click BYLAYER.
 - Click the color of your choice, or type the color number in the Current box.
- 4 Click OK.



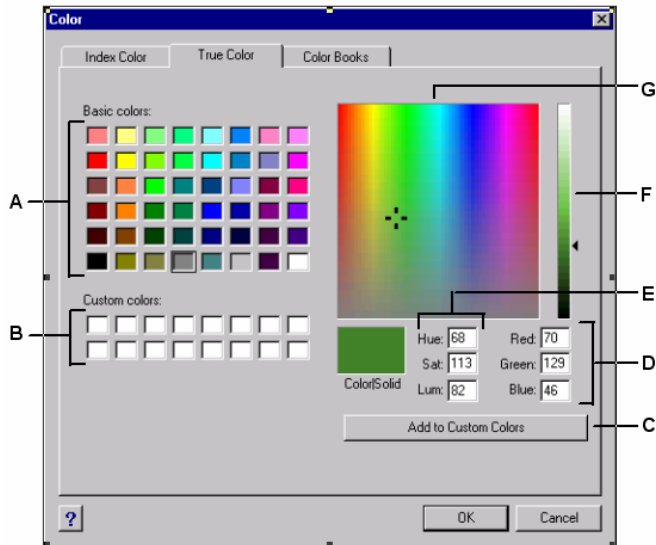
Using true colors

There are more than 16 million true colors from which you can choose. True colors are defined using 24-bit color. Even with so many colors available, you can quickly choose a color from the display of basic colors or by clicking the color palette. Alternatively, if you know the values used to define the desired color, you can enter the Hue, Saturation, and Luminance (HSL) values, or you can enter the Red, Green, Blue (RGB) values.

To select a true color

- 1 Click Select Color in the desired dialog box, such as Layers, Drawing Settings, Properties, or Multiline Text. The Color dialog box opens.
- 2 Click the True Color tab.
- 3 Do one of the following:
 - Click a basic true color.
 - Click a true color in the color palette.
 - Enter HSL values for the desired true color.
 - Enter RGB values for the desired true color.

- 4 Click OK.



- A Click to select one of the basic true colors.
- B Click to select one of the custom true colors, if any are defined.
- C Click to add the selected color to the custom color area.
- D Enter (or view) the amount of red, green, and blue of the desired true color.
- E Enter (or view) the amount of hue, saturation, and luminance of the desired true color.
- F Slide or click to adjust the color luminance.
- G Click to select a true color in the color palette.

Using color books

BtoCAD uses color books to store collections of colors. For example, you can store a unique color scheme for a client in a color book and then use colors only from that color book for the client's drawings.

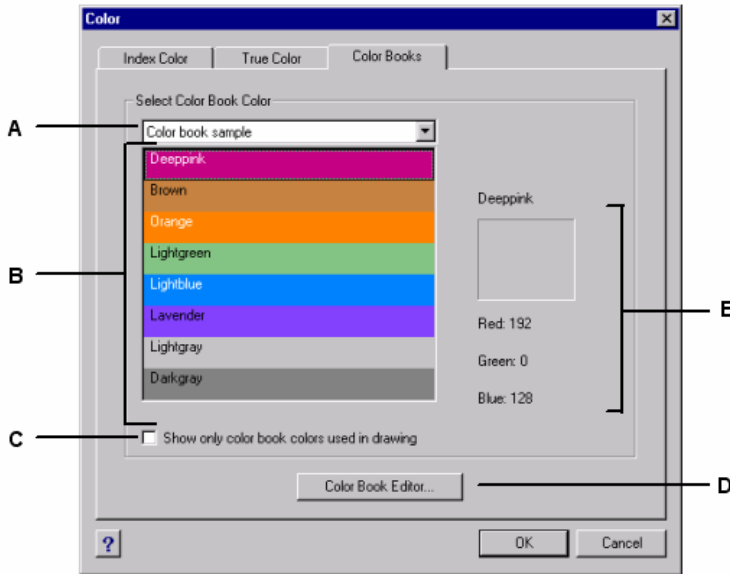
To select a color book color

- 1 Click Select Color in the desired dialog box, such as Layers, Drawing Settings, Properties, or Multiline Text. The Color dialog box opens.
- 2 Click the Color Books tab.
- 3 Select a color book from the list.
- 4 If you want to narrow your color search, do one of the following:

- In the list of colors, select a color book page, if one is available. Pages are particularly helpful in large color books — they help group colors so you can find them quickly.
- Select Show Only Color Book Colors Used in Drawing. Only those color book colors that are used in the current drawing will display in the list.

5 Select the desired color.

6 Click OK.



- A Select a color book.
- B Click to select a color book color.
- C Select to list only those color book colors that are used in the current drawing.
- D Click to create and modify color books.
- E Displays the selected color and its RGB values.

Creating color books

You may have a color book given to you by a client, developed by a third-party, or you can create your own. Each color book has an .acb extension and is saved automatically in Extensible Markup Language (XML) format.


To create a color book

1 Click Select Color in the desired dialog box, such as Layers, Drawing Settings, Properties, or Multiline Text.

The Color dialog box opens.

2 Click the Color Books tab.

3 Click Color Book Editor.

4 In the Color Book Editor, click the New tool ().

5 In Color Book Name, enter the name of the color book. This name appears in the list of color books on the Color Books tab in the Color dialog box.

6 Do the following to define organizational pages in the color book:

- In the contents of the color book, click an existing page or color where you want to add a page.

- Define a color in the palette.
- Enter the name of the page, and then click Add Page.

7 Do the following to define colors in the color book:



- In the contents of the color book, click an existing page or color where you want to add a color.

- Define a color in the palette.
- Enter the name of the color, and then click Add Color.

8 Do any of the following to change existing pages and colors in the color book:

- Modify pages and colors — Select a page or color in the color book, define its new color settings in the palette, enter any changes to its name, and then click Modify.

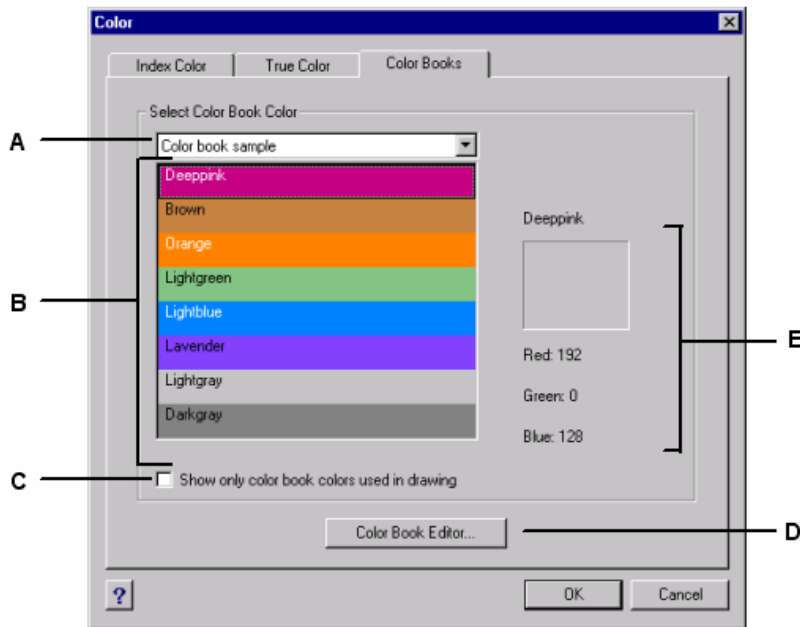
- Delete pages and colors — Select a page or color in the color book, and then click Delete.

- Rearrange pages and colors — Select a page or color in the color book, and then click the up arrow () or down arrow ().

9 In the Color Book Editor, click the Save tool ().

10 Enter a filename for the color book, and then click Save.

By default, the file is saved in the default folder where BtoCAD searches for color books.



- A** Click New to create a color book; click Open to open a color book; click Save to save the color book; click Save As to save the color book with a new filename.
- B** Select a page or color to modify, delete, or reorder it, or to view its settings.
- C** Click to create a new page in the color book using the current color settings.
- D** Click to create a new color in the color book using the current color settings.
- E** Click to delete the selected page or color from the color book.
- F** Click to redefine the selected page or color to use the current color settings.
- G** Click to move the selected page or color up or down one position in the color book.
- H** Enter the name of the page or color as you want it to appear in the color book.
- I** Enter (or view) the amount of red, green, and blue of the desired color.
- J** Enter (or view) the amount of hue, saturation, and luminance of the desired color.
- K** Slide or click to adjust the color luminance.
- L** Click to select a color.
- M** Enter the name of the color book.

Modifying color books

You can modify your own color books and the color books that came with Intelli-CAD. If you modify a color book that came with BtoCAD, it is recommended that you save it with a new filename first so the original color book is not overwritten.

To modify a color book

1 Click Select Color in the desired dialog box, such as Layers, Drawing Settings, Properties, or Multiline Text.

The Color dialog box opens.



2 Click the Color Books tab.

3 Select the color book you want to modify.


4 Click Color Book Editor.

5 In Color Book Name, enter any changes to the color book name. This name appears in the list of color books on the Color Books tab in the Color dialog box.

6 Do any of the following to modify pages or colors in the color book:

- Add pages — In the contents of the color book, click an existing page or color where you want to add a page. Define a color in the palette, enter the name of the page, and then click Add.
- Add colors — In the contents of the color book, click an existing page or color where you want to add a color. Define a color in the palette, enter the name of the color, and then click Add.
- Modify pages and colors — Select a page or color in the color book, define its new color settings in the palette, enter any changes to its name, and then click Modify.
- Delete pages and colors — Select a page or color in the color book, and then click Delete.
- Rearrange pages and colors — Select a page or color in the color book, and then click the up arrow () or down arrow ().

7 Do one of the following to save the color book:

- To save the color book with the same filename, click the Save tool () in the Color Book Editor.
- To save the color book with a new filename or in a new location, click the Save

As tool in the Color Book Editor.

Loading color books

If you have a color book given to you by a client or developed by a third-party, simply save it on your computer in a folder where BtoCAD can find it. By default, color books are stored in the same folder where you installed BtoCAD.

To load a color book

1 Save the color book in the folder where BtoCAD searches for color books. By default, this is the same folder where you installed BtoCAD. To verify the folder location, choose Tools > Options, click the Paths/Files tab, and find the Color Book folder in the Paths list.


Setting and changing the grid and snap alignment

Grid and snap settings are effective tools to use in your drawing to ensure accuracy. Although many users find it convenient to match grid points to snap settings, they are independent of each other and should not be confused. Grid points are for visual reference only; they do not affect your drawing and they do not print. Snap points are, by themselves, not visible; however, when set, they constrain the creation of new entities.

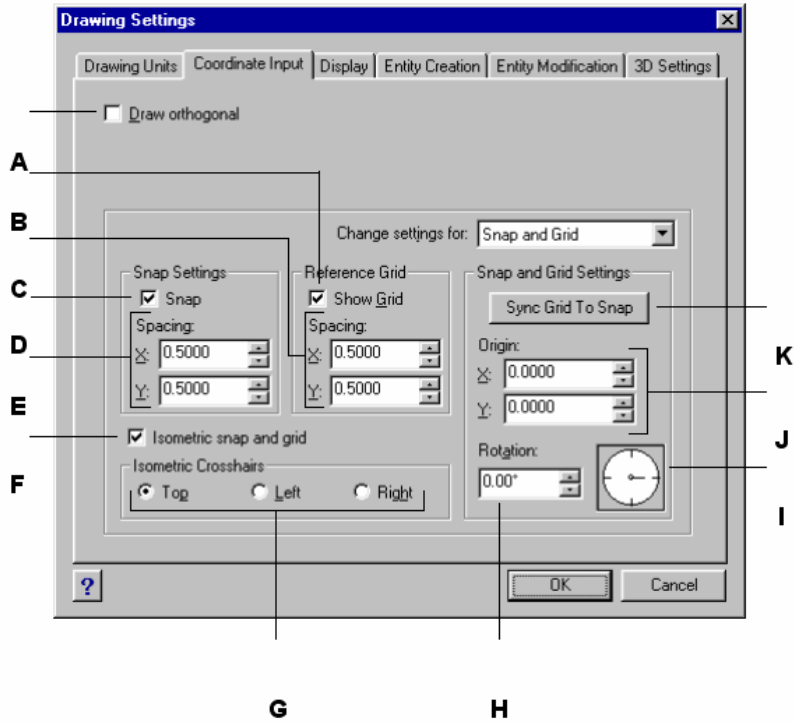
Setting a reference grid

A reference grid displays as a pattern of regularly spaced dots. You can turn the display on and off, and you can specify how far apart the dots are spaced. The reference grid extends only to the limits of the drawing, helping you to visualize the boundary of your drawing and to align entities and visualize distances between entities. You can turn the grid on and off as needed. You can also change the spacing of the grid at any time.

To turn the grid on and set the grid spacing

- 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 Coordinate Input tab.
- 3 Under Change Settings For, choose Snap and Grid.
- 4 Under Reference Grid, click the Show Grid check box.
- 5 Under Reference Grid Spacing, in the X field, choose the horizontal grid spacing.
- 6 Under Reference Grid Spacing, in the Y field, choose the vertical grid spacing.
- 7 Click OK.

TIP To toggle the grid display on and off at any time, double-click the **GRID** setting on the status bar, click the **Reference Grid** too on the **Settings** toolbar, or press **F7**.




- | | |
|---|---|
| A Click the check box to enable orthogonal mode. | G Click the current isometric plane. |
| B Click the check box to display the reference grid. | H Specify the grid rotation angle. |
| C Specify the x and y grid spacing. | I Indicates the current grid rotation angle. |
| D Click the check box to enable snap mode. | J Specify the x- and y-coordinates of the snap origin. |
| E Specify the x and y snap spacing. | K Click to match the grid spacing to the snap spacing. |
| F Click the check box to use an isometric snap and grid. | |

Setting snap spacing

Another way to ensure drawing accuracy is to turn on and set snap spacing. When snap is turned on, the program restricts the selection points to predetermined snap intervals. Although it is often helpful to match the snap spacing to some interval of the grid spacing or another related setting, the settings do not have to match.

To turn snap settings on and set snap spacing

- 1 Do one of the following:
 - Choose Settings > Drawing Settings.
 - On the Settings toolbar, click the Drawing Settings tool 
 - Type *settings* and then press Enter.
- 2 Click the Coordinate Input tab.
- 3 Under Change Settings For, choose Snap And Grid.
- 4 Under Snap Settings, click the Snap check box to turn Snap on.
- 5 Under Snap Settings Spacing, in the X field, choose the horizontal snap spacing.
- 6 Under Snap Settings Spacing, in the Y field, choose the vertical snap spacing.
- 7 Click OK.


TIP To toggle snap settings on and off at any time, double-click the *SNAP* setting on the status bar, click the *Snap* tool on the Settings toolbar, or press *F9*.

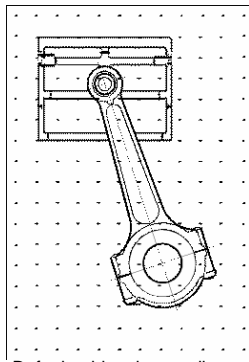
In addition to setting the snap spacing, you can change the snap and grid orientation. You can also rotate the alignment of the grid or set it to create isometric drawings.

Changing the snap and grid angle and base point

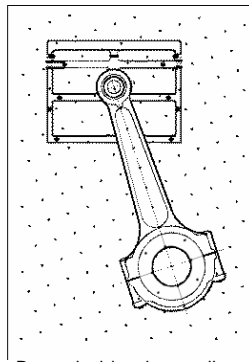
The snap and grid are both normally based on the drawing origin, the 0,0 coordinate in the World Coordinate System (WCS). You can relocate the snap and grid origin, however, to help you draw entities in relation to a different location. You can also rotate the grid to a different angle to realign the crosshairs to the new grid angle. If the grid is on, and the grid spacing is 0,0, then the grid defaults to the snap spacing.

To change the snap angle and base point

- 1 Do one of the following:
 - Click Settings > Drawing Settings.
 - On the Settings toolbar, click the Drawing Settings tool .
 - Type *settings* and then press Enter.
- 2 Click the Coordinate Input tab.
- 3 Under Change Settings For, select Snap And Grid.
- 4 Under Snap Settings, click the Snap check box to turn Snap on.
- 5 Under Reference Grid, click the Show Grid check box to turn Show Grid on.
- 6 Under Snap And Grid Settings, in the X Origin field, type the x-coordinate of the new snap origin.
- 7 Under Snap And Grid Settings, in the Y Origin field, type the y-coordinate of the new snap origin.
- 8 Under Snap And Grid Settings, in the Rotation field, type the grid rotation angle.
- 9 Click OK.



Default grid and snap alignment.



Rotated grid and snap alignment.

Using isometric snap and grid

You can use the Isometric Snap And Grid option to create two-dimensional isometric drawings. With the isometric option, you are simply drawing a simulated three-dimensional view on a two-dimensional plane, much the same as you might draw on a piece of paper. Do not confuse isometric drawings with three-dimensional drawings. You create three-dimensional drawings in three-dimensional space.

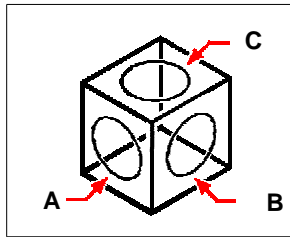
The isometric option always uses three preset planes, which are denoted as left, right, and top. You cannot alter the arrangement of these planes. If the snap angle is 0, the three isometric axes are 30 degrees, 90 degrees, and 150 degrees.

When you check the Isometric Snap And Grid option and select an isometric plane, the snap intervals, grid, and crosshairs align with the current plane. The grid is always shown as isometric and uses y-coordinates to calculate the grid spacing. If you click the Draw Orthogonal check box, the program restricts the drawing of entities to the current isometric plane.

TIP To toggle between isometric planes, press F5.

To turn the Isometric Snap And Grid option on

- 1 Do one of the following:
 - Choose Settings > Drawing Settings.
 - On the Settings toolbar, click the Drawing Settings tool
 - Type *settings* and then press Enter.
- 2 Click the Coordinate Input tab.
- 3 Under Change Settings For, select Snap And Grid.
- 4 Click the Isometric Snap And Grid check box.
- 5 Under Isometric Crosshairs, click the option for the isometric plane you want (Top, Left, or Right).
- 6 Click OK.



Isometric planes left (A), right (B), top (C).

Using the Draw Orthogonal option


You can restrict cursor movement to the current horizontal and vertical axes so that you can draw at right angles, or orthogonally. For example, with the default 0-degree orientation (angle 0 at the “three o’clock” or “east” position), when the Draw Orthogonal option is enabled, lines are restricted to 0 degrees, 90 degrees, 180 degrees, or 270 degrees. As you draw lines, the rubber-banding line follows either the horizontal or vertical axis, depending on which axis is farthest from the cursor. When you enable the isometric snap and grid, cursor movement is restricted to orthogonal equivalents within the current isometric plane.

BioCAD ignores orthogonal drawing when you type coordinates in the command bar or use entity snaps.

To enable orthogonal drawing

- 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 Coordinate Input tab.
- 3 Click the Draw Orthogonal check box.
- 4 Click OK.

TIP To toggle orthogonal drawing on and off at any time, double-click the *ORTHO* setting on the status bar, click the Draw Orthogonal tool on the Settings toolbar, or press F8.

Using entity snaps

Entity snaps enable you to quickly select exact geometric points on existing entities without having to know the exact coordinates of those points. With entity snaps, you can select the endpoint of a line or arc, the center point of a circle, the intersection of any two entities, or any other geometrically significant position. You can also use entity snaps to draw entities that are tangent or perpendicular to an existing entity.

You can use entity snaps any time the program prompts you to specify a point—for example, if you are drawing a line or other entity. You can work with entity snaps in one of two ways:


- Enable a running entity snap that remains in effect until you turn it off by choosing an entity snap when no other command is active.
- Enable a one-time entity snap for a single selection by choosing an entity snap when another command is active. You can also use a one-time entity snap to over-ride a running entity snap.

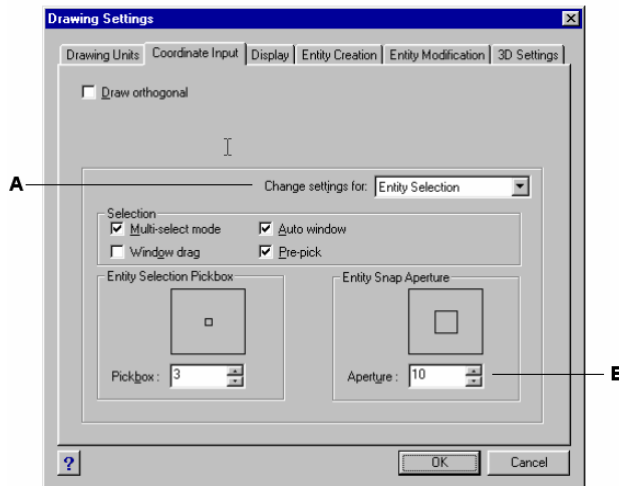
TIP If you type the name of the entity snap, you need to type only the first three letters.

When using entity snaps, the program recognizes only visible entities or visible portions of entities. You cannot snap to entities on layers that have been turned off or to the blank portions of dashed lines. When you specify one or more entity snaps, an entity snap target box is added to the crosshairs. In addition, an icon appears adjacent to the crosshairs indicating the active entity snap. When you select an entity, the program snaps to the snap point closest to the center of the target box.

TIP Type *m2p* or *mtp* to enable a one-time snap to the midpoint of two points that you specify, such as two points selected using entity snaps.

To change the size of the entity snap target box

- 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 Coordinate Input tab.
- 3 Under Change Settings For, choose Entity Selection.
- 4 Under Entity Snap Aperture, change the value in the Aperture field.
- 5 Click OK.



- A** Choose Entity Selection. **B** Type or select the entity snap aperture size.

Setting entity snaps

You can set entity snaps using any of the following methods:

- On the Object Snaps toolbar, click one of the entity snap tools.
- In the command bar, type an entity snap command.
- In the status bar, right-click OSNAP and choose Settings.
- Press and hold down the Shift key while right-clicking anywhere within the drawing window to display the entity snap shortcut menu, and then choose the entity snap you want to set.


TIP When you select an entity snap, a check mark appears next to the entity snap in the menu, the associated tool if the *Entity Snaps* toolbar is active, and the corresponding box is checked in the *Coordinate* tab of the *Drawing Settings* dialog box.

Nearest Snap tool

Use the Nearest Snap tool to snap to the nearest point of another entity. You can snap to the nearest point on an arc, circle, ellipse, elliptical arc, line, point, polyline segment, ray, spline, infinite line, or hatch pattern that is visually closest to the cursor.

To set the Nearest Snap

Do one of the following:


- On the Object Snaps toolbar, click the Set Nearest Snap tool 
- Type *nearest* and then press Enter.

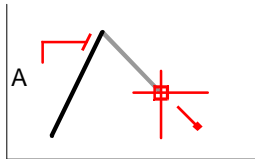
Endpoint Snap tool

Use the Endpoint Snap tool to snap to the endpoint of another entity. You can snap to the closest endpoint of an arc, line, polyline segment, ray, hatch pattern, plane, or three-dimensional face. If an entity has thickness, the Endpoint Snap also snaps to the endpoints of the edges of the entity.

To set the Endpoint Snap

Do one of the following:

- On the Object Snaps toolbar, click the Set Endpoint Snap tool 
- Type *endpoint* and then press Enter.



To snap to the endpoint, select anywhere on the entity near its endpoint (A).


Midpoint Snap tool

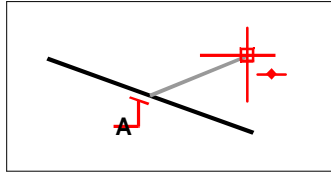
Use the Midpoint Snap tool to snap to the midpoint of another entity. You can snap to the midpoint of an arc,

ellipse, line, polyline segment, plane, infinite line, spline, or hatch pattern. In the case of infinite lines, the midpoint snaps to the first defined point. If an entity has thickness, the midpoint entity snap also snaps to the midpoint of the edges of the entity.

To set the Midpoint Snap

Do one of the following:

- On the Object Snaps toolbar, click the Set Midpoint Snap tool 
- Type *midpoint* and then press Enter.




To snap to the midpoint, select anywhere on the entity near its midpoint (A).

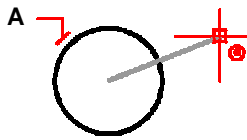
Center Snap tool

Use the Center Snap tool to snap to the center point of another entity. You can snap to the center of an arc, circle, polygon, ellipse, or elliptical arc. To snap to the center, you must select a visible portion of the entity.

To set the Center Snap

Do one of the following:

- On the Object Snaps toolbar, click the Set Center Snap tool 
- Type *center* and then press Enter.



To snap to the center, select anywhere on the visible portion of the entity (A).


Perpendicular Snap tool

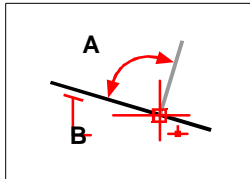
Use the Perpendicular Snap tool to snap to the perpendicular point of another entity. You can snap to an arc, circle, ellipse, line, polyline, infinite line, ray, spline, hatch pattern, or edge of a plane to form a

perpendicular alignment with that entity or with an imaginary extension of that entity.

To set the Perpendicular Snap

Do one of the following:

- On the Object Snaps toolbar, click the Set Perpendicular Snap tool 
- Type *perpendicular* and then press Enter.




To form a perpendicular angle (A) to an entity (B), select anywhere on the entity.

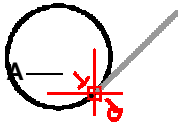
Tangent Snap tool

Use the Tangent Snap tool to snap to the tangent point of another entity. You can snap to the point on an arc, ellipse, spline, or circle that, when connected to the previous point, forms a line tangent to that entity.

To set the Tangent Snap

Do one of the following:

- On the Object Snaps toolbar, click the Set Tangent Snap tool 
- Type *tangent* and then press Enter.




To snap to a tangent, select the entity near the tangent point (A).

Quadrant Snap tool

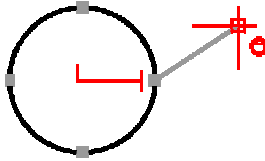
Use the Quadrant Snap tool to snap to the quadrant point of another entity. You can snap to the closest quadrant of an arc, circle, ellipse, or elliptical arc.

To set the Quadrant Snap

Do one of the following:

- On the Object Snaps toolbar, click the Set Quadrant Snap tool 
- Type *quadrant* and then press Enter.

A




To snap to a quadrant, select the entity near the quadrant point (A).

Insertion Point Snap tool

Use the Insertion Point Snap tool to snap to the insertion point of an attribute, block, or text entity.

To set the Insertion Point Snap

Do one of the following:


- On the Object Snaps toolbar, click the Set Insertion Point Snap tool 
- Type *insertion* and then press Enter.

Point Snap tool

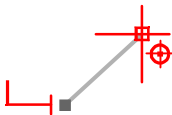
Use the Point Snap tool to snap to a point entity.

To set the Point Snap

Do one of the following:

- On the Object Snaps toolbar, click the Set Point Snap tool 
- Type *node* and then press Enter.

A



To snap to a point entity, select the entity (A).

Intersection Snap tool


The Intersection Snap tool snaps to the actual intersection in three-dimensional space of any combination of entities. You can snap to the combination of an arc, circle, line, infinite line, polyline, ray, ellipse, elliptical arc, spline, hatch pattern, polygon mesh, or polyface mesh. You can also snap to an intersection point within a single entity, including a polyline or spline.

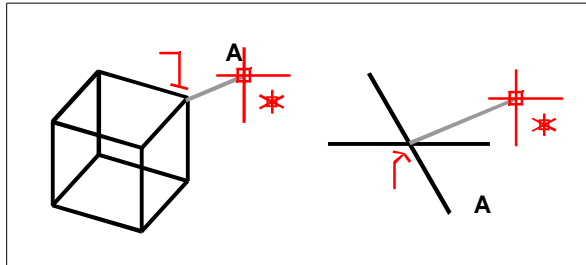
The Extended Intersection Snap option snaps to the logical location where two entities (lines, arcs, or elliptical segments) would intersect if they were of infinite length. BtoCAD automatically uses the extension option only when you type *int* in the command bar (not the full *intersection* command name) after selecting a command, such as Line or Circle. Dashed extension lines are drawn to help show the extended intersection.

NOTE *There are two types of intersection snaps. You can set the Intersection Snap or Apparent Intersection Snap, but you cannot use both at the same time.*

To set the Intersection Snap

Do one of the following:

- On the Entity Snaps toolbar, click the Set Intersection Snap tool 
- Type *intersection* and then press Enter.



To snap to an intersection, select the intersection (A).

To snap to an extended intersection point

- 1 Choose a command, for example, *line*.
- 2 Type *int* and then press Enter.
- 3 Select an extended intersection point.

The Extended Intersection Snap turns off automatically after you select a point.

Quick Snap command

Normally, an entity snap searches all the entities crossing the target and selects the one closest to the center of the target. Use the Quick Snap command to modify the current entity snap so that the program stops

searching for the snap point as soon as it finds one entity with at least one point of the current entity type.

To set the Quick Snap


- Type *quick* and then press Enter.

Clear Entity Snaps tool

Use the Clear Entity Snaps tool to turn off all entity snap settings, regardless of how they were set: by menu, tool, command, or in the Drawing Settings dialog box.

To set Clear Entity Snaps

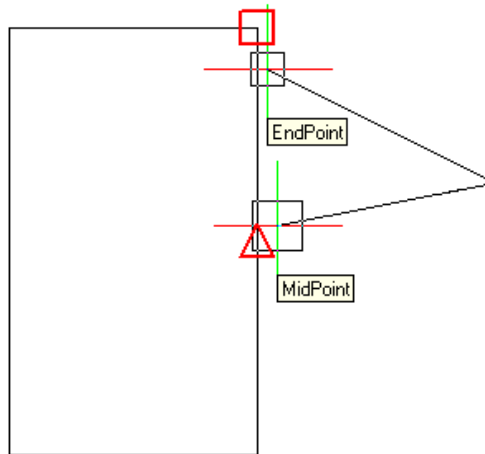
Do one of the following:

- On the Object Snaps toolbar, click the Clear Entity Snaps tool 
- Type *none* and then press Enter.


Using fly-over snapping

Fly-over snapping is a visual aid to help you see and use entity snaps more efficiently. When fly-over snapping is turned on, BtoCAD displays a colored marker at matching entity snap points as you move the crosshairs around the drawing.

Each entity snap has its own marker. When fly-over snapping is enabled and multiple entity snaps are on, you can press Tab to cycle through the available entity snap points of the entities covered by the target box. For example, when the Endpoint and Midpoint Snaps are on and the aperture box is located on a line, press Tab to cycle between the line's closest endpoint and midpoint.



To set the fly-over snapping options

- 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 Coordinate Input tab.
- 3 Click Fly-Over. The Options dialog box opens to the Snapping tab.
- 4 Mark Enable Fly-Over Snapping to turn on fly-over snapping.
- 5 Set the fly-over options, including the color, size, and thickness of the snap marker.
- 6 Click OK.
- 7 Click OK again.

Saving your drawing

When you save a drawing, your work is saved in a drawing (.dwg) file. After you have saved your drawing for the first time, you can save it with a new name. In addition to drawing (.dwg) files, you can also save a drawing in a Drawing Exchange Format (.dxf) file or a drawing template (.dwt) file.

If you created your drawing using a template, saving the drawing does not alter the original template. If you need an added level of security for your drawing files, you can save the drawing with a password so only you and those who know the password can open the drawing.

Saving a drawing

To save a drawing, choose any of the following methods:

- I Choose File > Save.
- I On the Standard toolbar, click Save
- I Type *save* and then press Enter.
- I Type *qsave* and then press Enter.

TIP When you save a drawing the first time, the program displays the Save Drawing As dialog box so that you can choose a directory and type a name for the drawing. You can use any name when you first save the drawing.

Saving a drawing with a new name or file format

To save a drawing with a new name or file format

- 1 Do one of the following:
 - Choose File > Save As.
 - Type *saveas* and then press Enter.
- 2 In the Save Drawing As dialog box, under Save As Type, choose the file format.
- 3 Specify the name of the file you want to create.
- 4 Click Save.

Saving a drawing with a password

To save a drawing with a password

- 1 Do one of the following:
 - Choose File > Save As.
 - Type *saveas* and then press Enter.
- 2 In the Save Drawing As dialog box, under Save As Type, choose AutoCAD 2004 (dwg).
- 3 Click Password Protect.
- 4 Choose a folder where you want to save the drawing.
- 5 In File Name, specify the name of the file you want to create.
- 6 Click Save.
- 7 In the Password dialog box, enter a password.
- 8 If desired, record and store the password in a safe place. If you forget the drawing's password, the drawing cannot be opened or recovered.
- 9 Click OK.