

Viewing your drawing

BtoCAD provides many ways to display and view your drawing. You can also change various display settings to speed up the display or printing of a drawing. This section explains how to:

- Navigate within a drawing by scrolling, panning, and rotating the view.
- Change the magnification of a drawing by zooming in and out.
- Work with multiple windows or views of a drawing.
- Control the display of elements to optimize performance when working with large or complex drawings.

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Redrawing and regenerating a drawing

As you work on a drawing, visual elements may remain after the completion of a command. You can remove these elements by refreshing, or redrawing, the display.

To redraw (refresh) the current window display

Do one of the following:

- Choose View > Redraw.
- Type *redraw* and then press Enter.

Information about drawing entities is stored in a database as floating point values, ensuring a high level of precision. Sometimes a drawing must be recalculated, or regenerated, from the floating-point database to convert those values to the appropriate screen coordinates. This occurs automatically. You can also manually initiate a regeneration. When the drawing is regenerated, it is also redrawn.

To regenerate the current window, type *regen* in the command bar. If more than one window is displayed, type *regenall* to regenerate all the windows.

Moving around within a drawing

You can move the view of a drawing displayed in the current viewport by scrolling, panning, or rotating the view. Doing this changes the portion of the drawing you are viewing without changing the current magnification. Scrolling lets you move around in the drawing horizontally and vertically. Panning lets you move the drawing in any direction. Rotating lets you view your drawing from any angle.

NOTE You can also move to a different view using the *Model* and *Layout* tabs. For more details, see “Viewing drawings in paper space and model space”.

Using scroll bars

To assist you in navigating within a drawing, horizontal and vertical scroll bars are available in each drawing window. The size of the scroll box in relation to the scroll bar indicates the current level of drawing magnification. The position of the scroll box in relation to the scroll bar indicates the location of the center of the drawing in relation to the extents of the drawing (the smallest rectangle containing all the entities in the drawing).


To turn scroll bars on or off

Do one of the following:

- Choose View > Scroll Bars.
- Choose Tools > Options > Display tab, and select Show Scroll Bars.
- Type *scrollbar*, press Enter, and then select On, Off, or Toggle.

Using the Pan command



You can move the drawing in any direction using the Pan tool () on the View toolbar. Panning shifts or slides the view of the drawing horizontally, vertically, or diagonally. The magnification of the drawing remains the same, as does its orientation in space. The only change is the portion of the drawing displayed.

To pan, you can use any of the following methods:

- For precise panning, specify two points defining the magnitude and direction of the pan. The first point, or base point, indicates the starting point of the pan. The second point indicates the amount of pan displacement relative to the first point.

- To pan in real time, press Ctrl + Shift and use the right mouse button, or use the Real-Time Pan tool




() on the View toolbar.

- If you have a mouse with a wheel, press and hold the wheel, and then move the mouse.
- To pan in small increments, use the arrow keys (if the arrow keys were not selected for command history navigation in Tools > Options > Display tab).

To pan in real time

- 1 Do one of the following:

Choose View > pan > Real-Time Pan.

On the View toolbar, click the Real-Time Pan tool ()

Type rtpan and then press Enter.

- 2 Click and hold the left mouse button.
- 3 Move the cursor in the direction you want to pan.
- 4 To stop panning, release the mouse button.

TIP Hold down the right mouse button while simultaneously pressing and holding Ctrl + Shift to pan in real-time.

NOTE If you are working with a large drawing file, it may be helpful to set the RTSKIPCOUNT system variable to a higher number to reduce the number of entities that display. For example, if the value is set to 10, only the 10th entity will display when panning and rotating in real-time.

To pan using a mouse with a wheel

- Press and hold the wheel, and then move the mouse in the direction you want to pan. (The MBUTTONPAN system variable controls this feature.)

To pan using the arrow keys

- Press the up, down, right, or left arrow keys.

NOTE You can pan using the arrow keys if Use Up/Down Arrows for Command History Navigation is not marked on the Display tab in Tools > Options.

Rotating the view in real time

BtoCAD allows you to rotate your view of a drawing in real time. This allows you to view your model from any angle while in model space. You cannot rotate the view while in paper space. If desired, you can continue the view rotation after you release your mouse. Choose Tools > Options to enable continuous internal motion in real time. For more information, see “Changing the options on the Display tab”.

To rotate the view in real time

- 1 Do one of the following:
 - Choose View > Real-Time Sphere.
 - Type *rtrot* and then press Enter.
 - Press and hold Ctrl.
- 2 Click and drag the left mouse button. The view rotates according to the movement of your mouse.
- 3 To stop rotating, release the mouse button.
- 4 If the view continues to rotate, press Enter or right-click the drawing when finished.

TIP You can use the *Real-Time X*, *Real-Time Y*, and *Real-Time Z* commands to lock the rotation in the corresponding axis. You can also press Ctrl and use the right mouse button to rotate the view about the z-axis.

NOTE If you are working with a large drawing file, it may be helpful to set the *RTSKIPCOUNT* system variable to a higher number to reduce the number of entities that display. For example, if the value is set to 10, only the 10th entity will display when panning and rotating in real-time.

Changing the magnification of your drawing


You can change the magnification of your drawing at any time by zooming. The cursor changes to a magnifying glass (🔍) when a zoom tool is active. Zoom out to reduce the magnification so you can see more of the drawing, or zoom in to increase the magnification so you can see a portion of the drawing in greater detail. Changing the magnification of the drawing affects only the way the drawing is displayed; it has no effect on the dimensions of the entities in your drawing.

Zooming in and out


One of the easiest ways to change the magnification of the drawing is to zoom in or out by a preset increment. On the View toolbar, the Zoom In tool (🔍+) doubles the current magnification of the drawing. The Zoom Out tool (🔍-) reduces the magnification of the drawing by half. The portion of the drawing located at the center of the current viewport remains centered on the screen as you zoom in and out.

Zooming methods


To zoom, you can use any of the following methods:

- To define the portion of the drawing to zoom, create a window.
- To zoom in real time, press Ctrl + Shift and use the left mouse button, or use the Real-Time Zoom tool () on the View toolbar.
- If you have a mouse with a wheel, rotate the wheel to zoom in and out.

To zoom in to an area using a window

- 1 Do one of the following:
 - Choose View > Zoom > Window.
 - On the View toolbar, click the Zoom Window tool ()
 - Type *zoom* and then press Enter.
- 2 Select one corner of the window around the area you want to magnify.
- 3 Specify the opposite corner of the window around the area you want to magnify.


To zoom in real time

- 1 Do one of the following:
 - Choose View > Zoom > Real-Time Zoom.
 - On the View toolbar, click the Real-Time Zoom tool ()
 - Type *rtzoom* and then press Enter.
 - Simultaneously press and hold Ctrl + Shift.
- 2 Click and hold the left mouse button.
- 3 To zoom in, move the cursor up the screen; to zoom out, move the cursor down the screen.
- 4 To stop zooming, release the mouse button.

To zoom using a mouse with a wheel

- Rotate the wheel away from you to zoom in or toward you to zoom out. Each rotation of the wheel away from you zooms out .8 times; each rotation toward you zooms in 1.25 times.

Displaying the previous view of a drawing

After you zoom in or pan to view a portion of your drawing in greater detail, you may want to zoom back out to see the entire drawing. On the View menu, the Zoom Previous tool () lets you restore the previous view. Selecting this tool repeatedly steps back through up to 25 successive zoomed or panned views.

Zooming to a specific scale


You can increase or decrease the magnification of your view by a precise scale factor measured relative to the overall size of the drawing or in relation to the current display. When you change the magnification factor, the portion of the drawing located at the center of the current viewport remains centered on the screen.

To change the magnification of the view relative to the overall size of the drawing, type a number

representing the magnification scale factor. For example, if you type a scale factor of 2, the drawing appears at twice its original size. If you type a magnification factor of .5, the drawing appears at half its original size.


You can also change the magnification of the drawing relative to its current magnification by adding an *x* after the magnification scale factor. For example, if you type a scale factor of 2*x*, the drawing changes to twice its current size. If you type a magnification factor of .5*x*, the drawing changes to half its current size.

To zoom to a specific scale relative to the current display


- 1 Do one of the following:
 - Choose View > Zoom > Zoom In.
 - On the View toolbar, click the Zoom In tool .
 - Type *zoom* and then press Enter.
- 2 Type the scale factor, followed by an *x* (such as 2*x*).
- 3 Press Enter.

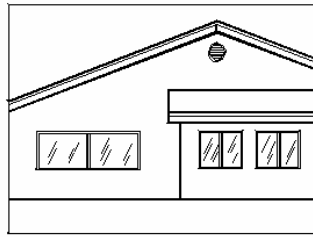
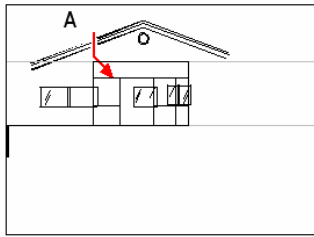
Combining zooming and panning

You can specify the point you want at the center of the view when you change the drawing magnification. You can specify the point you want at the lower left of the view when you change the magnification of the drawing with the Zoom Left tool

() on the View toolbar. With the exception of the Zoom Window tool, the other zoom tools zoom in or out from the center of the current view.


To change the center of the current view


- 1 Do one of the following:
 - Choose View > Zoom > Center.
 - On the Zoom toolbar, click the Zoom Center tool .
 - Type *zoom*, press Enter, and then in the prompt box, choose Center.
- 2 Select the point you want located at the center of the new view.
- 3 Specify the zoom scale factor or the height of the drawing in drawing units.

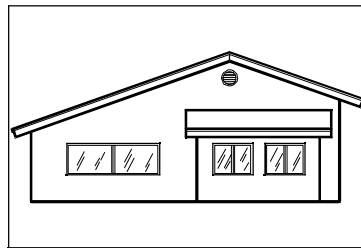


Current view showing the point to be centered in the new view (A), and the new view zoomed using a scale factor of 2x.

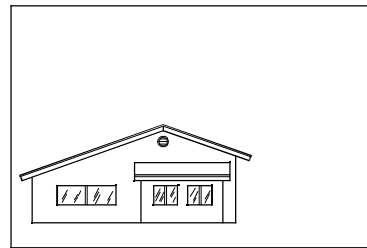
Displaying the entire drawing

You can use the Zoom All tool () on the View toolbar to display an entire drawing. If you have drawn any entities outside the defined limits of the drawing, the extents of the drawing are displayed. If you drew all entities within the limits of the drawing, the drawing is displayed all the way to the drawing limits.

The Zoom Extents tool () on the View toolbar displays the drawing to its extents, making the image fill the display to the greatest possible magnification.



Zoom extents (displays all entities).



Zoom all (displays to drawing limits).

Displaying multiple views

When you begin a new drawing, it is displayed in a single window. You can view the drawing in a second window, or you can divide one window into multiple windows. You can also open and display multiple drawings.

Working with multiple views of a single drawing

You can open and work with several views of the same drawing simultaneously. There are two methods for dividing the current drawing into multiple views:

- Open a new window of the open drawing.

- Divide the current window into multiple views.

After you divide a single window into multiple windows, you can control each window separately. For example, you can zoom or pan in one window without affecting the display in any of the other windows. You can control the grid, snap, and view orientation separately for each window. You can restore named views in individual windows, draw from one window to another, and name window configurations individually so you can reuse them later.

As you draw, any changes you make in one window are immediately visible in the others. You can switch from one window to another at any time, even in the middle of a command, by clicking the window's title bar.

Opening a new window of the same drawing

You can open additional windows to create more than one view of a drawing. To open a new window, choose Window > New Window. After you open a new window, you can change its display without affecting any of the other windows.

When you open more than one window for a single drawing, each window is assigned a unique number (for example, mydrawing:1, mydrawing:2, and so on). If your current window is maximized, you can switch to another open window by selecting its name from the bottom of the Window menu.

The names of other open drawings appear at the bottom of the Window menu. You can also use the Cascade, Tile Horizontally, and Tile Vertically commands to arrange all the open windows and drawings. To arrange all the windows and drawings into a stack of identically sized windows, choose Window > Cascade. Arranging windows and drawings in this way makes it easy to see the title bar for each window.

To arrange all the windows and drawings horizontally so they are placed in order from top to bottom, choose Window > Tile Horizontally. Arranging windows and drawings in this way displays each open window. The windows are resized to fit within the available space.

To arrange all the windows and drawings vertically so they are placed side by side, choose Window > Tile Vertically. Arranging windows and drawings in this way displays each open window. The windows are resized to fit within the available space.

BtoCAD uses the commands in the following table to control its windows.

BtoCAD window-control commands

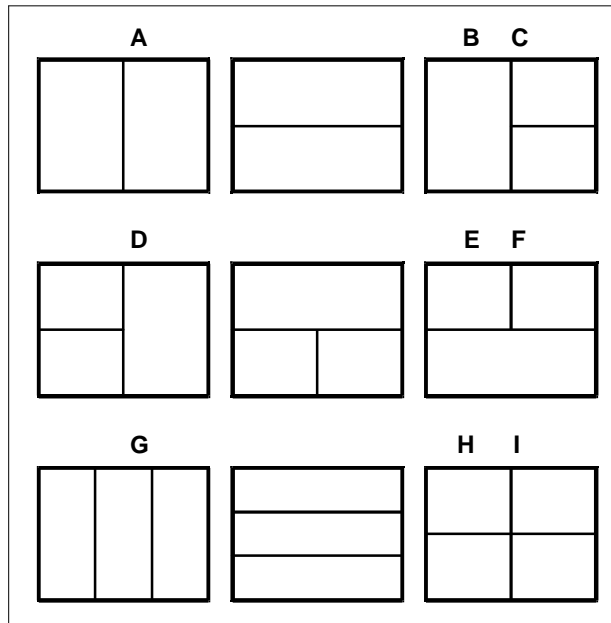
Command	Result
<i>vports</i>	Splits the current window into two, three, or four tiled windows.
<i>wcascade</i>	Cascades (overlaps) all open windows.
<i>wclose</i>	Closes the current window.
<i>wcloseall</i>	Closes all windows; also closes all drawings.
<i>whtile</i>	Tiles all windows horizontally.
<i>wiarrange</i>	Arranges window icons.
<i>wopen</i>	Opens another window of the current drawing.
<i>wvtile</i>	Tiles all windows vertically.

Dividing the current window into multiple views

You can divide a single drawing window into multiple tiled windows (called view-ports) on the Model tab. You can control the number of windows created and the arrangement of the windows. You can also save and restore named window configurations and display a list of the current and saved window configurations.

To create multiple views

- 1 Do one of the following:
 - Choose View > Viewports.
 - Type *viewports* and then press Enter.
- 2 In the prompt box, choose Create 2 Viewports, Create 3 Viewports, or Create 4 Viewports.
- 3 In the prompt box, choose the viewport orientation.



You can divide a drawing window into two windows arranged vertically (A) or horizontally (B); three windows arranged left (C), right (D), above (E), below (F), vertically (G), or horizontally (H); or four tiled windows (I).

To join two views

- 1 Do one of the following:
 - Choose View > Viewports.
 - Type *viewports* and then press Enter.

- 2 In the prompt box, choose Join.
- 3 Click anywhere inside the window you want to keep.
- 4 Click anywhere inside the adjacent window you want to join to the first window.
- 5 Press Enter.

Saving window configurations

If you have divided the drawing window into multiple views, you can save the current window arrangement so that you can recall it to the screen later. The number and placement of the windows are saved exactly as they are currently displayed. The settings for each window are also saved.

To name and save a window configuration

- 1 Do one of the following:
 - Choose View > Viewports.
 - Type *viewports* and then press Enter.

- 2 In the prompt box, choose Save.
- 3 Type a configuration name, and then press Enter.

The name can be up to 31 characters in length and can contain letters, numbers, the dollar sign (\$), hyphen (-), and underscore (_), or any combination.

To restore a named window configuration

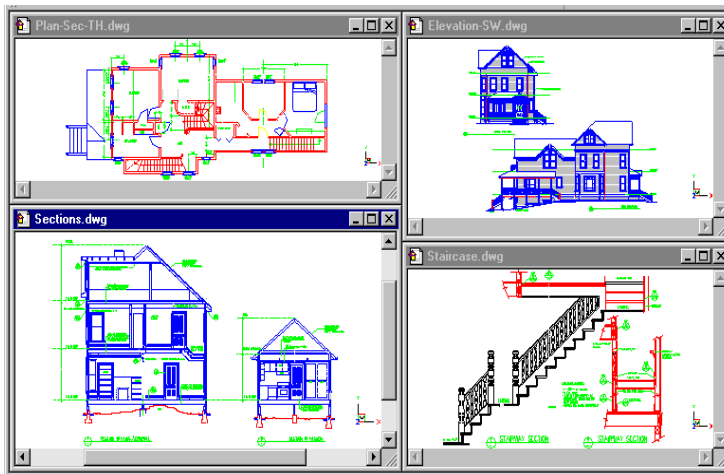
- 1 Do one of the following:
 - Choose View > Viewports.
 - Type *viewports* and then press Enter.
- 2 In the prompt box, choose Restore.
- 3 Type the name of the window configuration you want to restore.

Working with multiple drawings

With the multiple-document interface (MDI) feature, you can open more than one drawing inside of BtoCAD. Because you can open and work on several drawings at one time, you can copy, cut, or paste an entity from one drawing to another.

Each drawing appears in a drawing window, which has the following advantages:

- You can see two or more drawings side by side.
- You can easily copy entities from one drawing to another.
- Like viewports on the Model tab, you can tile or overlap drawing windows; unlike viewports on the Model tab, drawing windows maximize or reduce to an icon.



One session of BtoCAD with four drawings open.

Each drawing window that you open and work on retains in the Prompt History log all the commands that you perform, but the command line does not indicate when you have switched windows.

When you work with more than one drawing open in its own window, you can easily move, cut, copy, and paste in between drawings. If you move an entity from one window to another and then want to undo this action, you must undo it in both drawings for it to take effect. If you copy an entity from one window to another and then want to undo that action, you must undo it from the drawing into which you copied the entity. If you cut and paste an entity and then want to undo that action, you must undo it in both drawings.

Controlling visual elements



The number of entities in your drawing and the complexity of the drawing affect how quickly BtoCAD can process commands and display your drawing. You can improve overall program performance by turning off the display of certain visual elements, such as solid fills and text, while you work on the drawing. When you are ready to print your drawing, turn on the display of these elements so your drawing prints the way you want. You can also improve performance by turning off entity-selection highlighting, turning off the display of marker blips created when you select locations in the drawing, and by turning off the display of lineweights.

Turning Fill on and off

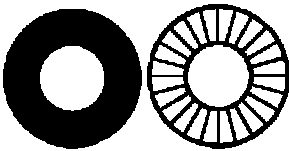
You can reduce the time it takes to display or print a drawing by turning off the display of solid fill. When Fill is turned off, all filled entities, such as wide polylines and planes, display and print as outlines. When you turn Fill on or off, you must redraw the drawing before the change is displayed.

To turn Fill on or off

1 Do one of the following:

- On the Settings toolbar, click the Fill tool  
- Type *fill* and then press Enter.

2 Choose View > Redraw.



Fill on.


Fill off.


TIP On the Settings menu, a check mark appears next to the Fill command when it is turned on and the Fill tool on the Settings toolbar is activated.

Turning Quick Text on and off

Text entities require a considerable amount of time to display and print. You can reduce the time it takes to display or print a drawing by enabling Quick Text. For example, if you're doing a preliminary check print of a drawing, you may want to turn Quick Text on to speed up printing. When Quick Text is enabled, text entities are replaced by rectangular boxes that indicate the outline of the area occupied by the text. When you turn Quick Text on or off, you must regenerate the drawing before the change is displayed.

To turn Quick Text on and off


- 1 Type *qtext*, press Enter, and then in the prompt box, choose On or Off.
- 2 Click Tool  Display tab.

- 3 Under Change Settings For, click Display.
 - 4 Select or clear the Enable Quick Text check box.
 - 5 Click OK.
 - 6 To regenerate your drawing, do one of the following:
 - Choose View > Regen.
 - On the View toolbar, click the Regen tool ()
 - Type *regen* and then press Enter.
-

Turning highlighting on and off

You can improve overall program performance by turning highlighting off. When you select entities to modify, the program highlights them using a dashed linetype. This highlight disappears when you finish modifying the entities or when the entities are cleared. Sometimes highlighting entities can take a considerable amount of time.


To turn highlighting on and off

- 1 Do one of the following:
 - Choose Settings > Drawing Settings.
 - On the Settings toolbar, click the Drawing Settings tool 
 - Type *highlight*, press Enter, and then in the prompt box, choose On or Off.
- 2 Click the Display tab.
- 3 Under Change Settings For, click Display.
- 4 Select or clear the Highlight Item When Selected check box.
- 5 Click OK.


Turning Blips on and off

You can turn Blips off. They are the temporary markers that appear on the screen when you select an entity or location. Blips are visible only until you redraw the drawing. You cannot select Blips; they are used only for reference and never print.

To turn Blips on and off

- 1 Do one of the following:
 - Choose Settings > Drawing Settings.
 - On the Settings toolbar, click the Drawing Settings tool 
 - Type *blipmode*, press Enter, and then in the prompt box, choose On or Off.
- 2 Click the Display tab.
- 3 Under Change Settings For, click Display.
- 4 Select or clear the Show Marker Blips check box.
- 5 Click OK.

Controlling the display of lineweights**To control the display of lineweights**

- 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.
 - Type *lweight* and then press Enter.
- 2 Click the Display tab.
- 3 Under Change Settings For, click Lineweights.
- 4 Select or clear Display Lineweights.
- 5 In Units for Listing, choose Millimeters or Inches.
- 6 In Default, select the lineweight assigned to layers and entities that use the Default lineweight.
- 7 In Adjust Display Scale, move the slider to the scale you want. By default, the slider begins at 1.00.
- 8 Click OK.

TIP On the status bar, double-click the word *LWT* to turn the display of lineweights on or off.

You can turn lineweights on or off when you print. For details, see “Choosing how lineweights print”.