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Printing drawings

You can print a copy of your drawing exactly as you created it, or you can add formatting and specify print controls to change how your drawing looks when it is printed.

Sometimes you may require multiple printed drawings, each with a different look or layout. For example, you may need one printed drawing for a client presentation, along with several other variations for production contractors. For each type of printed drawing that you require, you can create a layout that defines its characteristics, including scale, area to print, print style tables, and more.

This section explains how to:

• Start printing right away.

• Set up a drawing to print multiple layouts from paper space on a Layout tab.

• Define how you want your drawing to look when it is printed.

• Print or plot your drawing.

Topics in this chapter

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Getting started printing

When you create a drawing, you do most of your work on the Model tab. At any time you can print your drawing to see how it looks on paper. It's easy to get started print- ing, and then later create layouts and custom print settings to enhance your printed output.

To start printing

1   From the Model tab, do one of the following:
    • Choose File > Print.

    • On the Standard toolbar, click the Print tool ( ).
    • Type *print* and then press Enter.
If you click the Print tool, the Print dialog box does not display. Your drawing will be sent directly to the selected printer.

2   Click Print.

There are many options that you can set before printing, such as the scale of the drawing, print area, print style tables, and more. For details, see "Customizing print options".

NOTE *Type* qprint *and then press Enter to print the current viewport. The Print dialog box is bypassed and the drawing is sent directly to the selected printer.*

Defining layouts for printing

You can print your drawing directly from the Model tab where you created it, or you can create custom layouts for printing on Layout tabs. When printing from the Model tab, you can print your drawing exactly the way it appears, or you can modify the drawing before printing by adding dimensions, a leg- end, or a title block.

You typically use the Layout tabs for printing if you require multiple print layouts. You may also want to use a Layout tab for printing even if you want your drawing printed only one way. For example, if you want to include large amounts of text on your printed drawing, you can add the text to a Layout tab so it does not clutter your drawing while you work on the Model tab.

Understanding layouts

When you create a drawing, you do most of your work on the Model tab. Each drawing that you create can contain numerous layouts that simulate the paper on which you will print a copy of the drawing. Each of these

layouts is created on a Layout tab.You can prepare a separate layout for each way you want to print your drawing. The layout allows you to organize different views to control which portion of your drawing prints and at what scale.

Before you print, you can also include additional entities and layout settings that control how your drawing prints. Additional items only appear on the Layout tab, not on the Model tab. For example, a layout can contain title blocks, legends, or keynote that print with your model, but do not clutter the screen when you work with your model on the Model tab.

Use these general steps to prepare your drawing for printing multiple layouts:

1   On the Model tab, create your drawing.

2   Create a new layout. You can use an existing Layout1 or Layout2 tab, or you can create a new Layout tab. For details about creating a new layout, see "Creating a new layout".

3   Create at least one layout viewport on the Layout tab. Use each viewport to help control that portion of the drawing prints and at what scale. For details, see "Working with layout viewports".

4   Include any additional items that may be required for the specific layout, such as dimensions, a legend, or a title block.

5   Specify additional settings for the layout, such as the scale of the drawing, print area, print style tables, and more. For details, see "Customizing print options".

6   Print or plot your drawing. For more details, see "Printing or plotting your drawing".

Understanding paper space and model space

When you start a drawing session, your initial working area is called model space. Model space is an area in which you create two-dimensional and three-dimensional entities based on either the World Coordinate System (WCS) or a user coordinate system (UCS). You view and work in model space while using the Model tab.

Your view of this area is a single viewport that fills the screen. You can create addi- tional views on the Model tab, called viewports, which can show the same or differ- ent two-dimensional or three-dimensional views, all of which are displayed in a tiled manner. You can work in only one of these viewports at a time on the Model tab, and you can print only the current viewport.
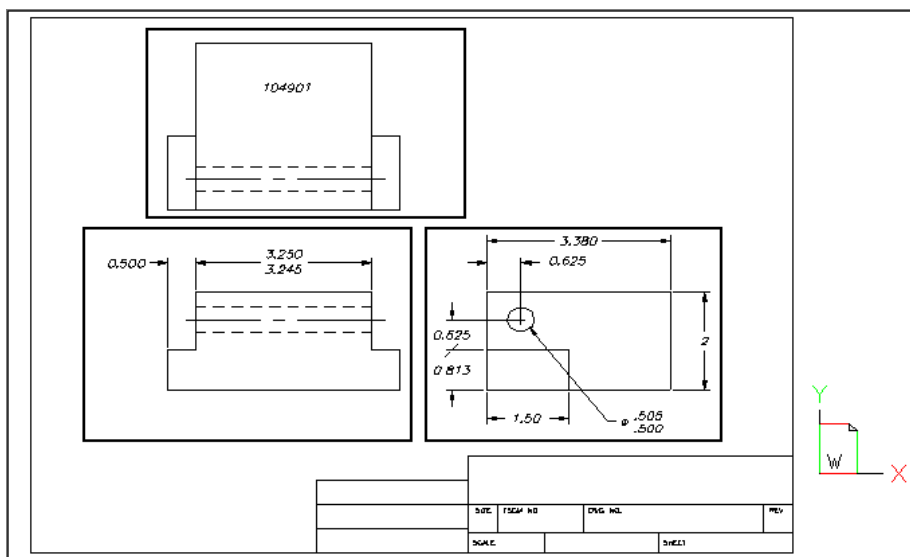

Model space with two viewports.
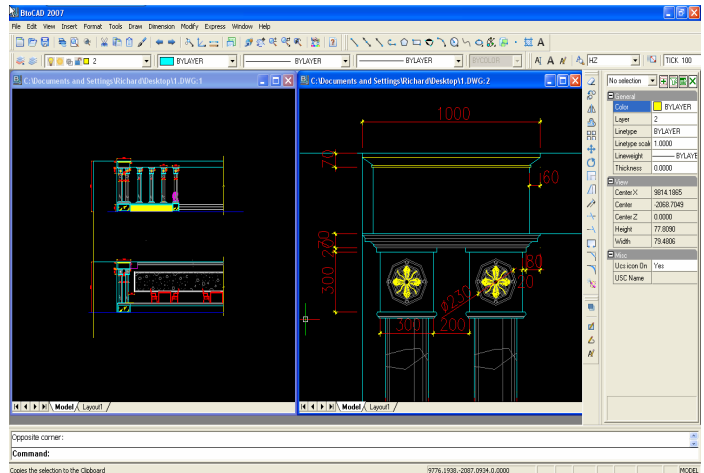
BtoCAD provides an additional work area, called paper space. The contents of paper space represent the paper layout of your drawing. In this work area, you can create and arrange different views of your model similar to the way you arrange detail drawings or orthogonal views of a model on a sheet of paper. You can also add keynotes, annotations, borders, title blocks, and other print-related entities in paper space, which reduces clutter when you work with your model in model space.

You view and work in paper space while using a Layout tab. Each view, or layout viewport, that you create in paper space provides a window of your drawing in model space. You can create one layout viewport or several. You can place layout viewports anywhere on the screen; their edges can be touching or not; and you can print them all at the same time.

You do not need to use paper space to print your drawing, but it offers several advantages:

•   Print the same drawing with different print settings that you save with each layout, for example, printer configuration files, print style tables, lineweight settings, drawing scale, and more.

•   Add print-related entities that are not essential to the model itself, such as key- notes or annotations, to reduce clutter when you work with your model in model space on the Model tab.

•   For a single layout, create multiple layout viewports that print the model at different views and scales.

Paper space with layout viewports.

Viewing drawings in paper space and model space

When you work in paper space on a Layout tab, you can still view your drawing in model space. First you need to create a layout viewport in paper space; this allows you to view your model space entities from paper space.

Within a layout viewport you can modify and snap to model space entities, but it is often more convenient to modify these entities on the Model tab. Zooming or panning the drawing in model space or paper space affects the entire drawing, unless you use multiple windows or viewports.

To view a drawing in model space on the Model tab

Do one of the following:

- Click the Model tab.

- Right-click the Model/Paper Space toggle in the status bar, and then choose Model.

To view a drawing in paper space on a Layout tab

Do one of the following:

- Click one of the Layout tabs.

- Type *layout* and then press Enter. In the prompt box, choose Set. Type a name for the layout you want to make current, and then press Enter.

- Double-click the Model/Paper Space toggle in the status bar. For example, double- click "Model" or "M:Tab Name" in the status bar to switch to paper space.

- While using a Layout tab, type *pspace* and then press Enter.

- While using a Layout tab, double-click outside of the layout viewport.

The first time you switch to a Layout tab, your drawing seems to disappear. This is normal. You must create at least one layout viewport to see your model. For details, see "Working with layout viewports".

To view a drawing in model space on a Layout tab

Do one of the following:

- Click the desired Layout tab, and then create and view a layout viewport. For more details, see "Working with layout viewports".

- Double-click the Model/Paper Space toggle in the status bar. For example, double- click "P:Tab Name" in the status bar to switch to model space on the current Lay- out tab.

- Type *mspace* and then press Enter.

- While using a Layout tab, double-click inside of the layout viewport.

Displaying the Model and Layout tabs

The Model and Layout tabs can be hidden, if desired. You may want to hide the tabs if you only work on the Model tab or if you use the command bar and status bar to switch between tabs.

To turn the Model and Layout tabs display on or off

Do one of the following:

• Choose View > Model and Layout tabs.

• Choose Tools > Options > Display tab, and choose Show Tabs.

Creating a new layout

In BtoCAD, you can create multiple layouts for a single drawing. Each layout represents a sheet of paper. For each layout you can specify the print area, print styles, print scale, lineweight scale, pen mappings, and add viewports, dimensions, a title block, and other geometry specific to the layout. The entities you add to a layout in paper space do not appear in model space.

Each layout requires at least one layout viewport. This viewport displays the drawing's model space entities. When you create a new drawing, the drawing automatically contains two default layouts: Layout1 and Layout2. You can start by using one of the default layouts, you can create your own, or your can create a new layout from another template (.dwt) file, drawing (.dwg) file, or drawing interchange (.dxf) file.

Each drawing can contain up to 255 layouts.

To create a new layout using the Layout1 or Layout2 tab

1   Click the Layout1 or Layout2 tab.

2   Set up at least one layout viewport. For details, see "Working with layout view- ports".

3   If desired, rename the layout. For details, see "To rename a layout".

To create a new layout using a new Layout tab

1   Do one of the following:
     • Choose Insert > Layout > New Layout.

     • On the Layouts toolbar, click the New Layout tool ( ).

     • Type *layout*, press Enter, and choose New.

2   Type a unique name for your layout and then press Enter.

The name can be up to 255 characters in length and can contain letters, numbers, the dollar sign ($), hyphen (-), and underscore (_), or any combination.

3   Set up at least one layout viewport. For details, see "Working with layout view- ports"

To create a new layout from an existing file

1   Do one of the following:

- Choose Insert > Layout > Layout from Template.
- On the Layouts toolbar, click the Layout from Template tool ( ).
- Type *layout*, press Enter, and choose Template.

2 Select the desired template, drawing, or drawing interchange file that contains the layout you want, and then click Open.

3 Select the layout(s), and then click OK. You can choose multiple layouts by hold- ing down Ctrl while selecting layout names.

Reusing layouts from other files

Save time by re-using layouts that you have already created. Within the same drawing, you may want to make a copy of a layout that contains most of the settings you want, and then make changes to the new copy. If you created layouts that you want to use again when you create new drawings, you can save the layouts as a drawing template.

To make a copy of a layout

1 Type *layout* and then press Enter.

2 In the prompt box, choose Copy.

3 Type the name of the layout you want to copy, and then press Enter.

4 Type a name for the new layout, and then press Enter.

To save a layout as a drawing template

1 Type *layout* and then press Enter.

2 In the prompt box, choose Save.

3 Type the name of the layout that you want to save, and then press Enter.

4 Specify the file name and location for the template, and then click Save.

After you save a layout as a template, you can use the template when you create new drawings. You can also import the template's layouts into another drawing.

Managing layouts in a drawing

You can rename layouts, delete layouts, and view a list of all layouts available in a drawing. You can also change the order in which the Layout tabs appear; the Model tab is always stationary.

If you want to rename, delete, or reorder a layout when the Layout tabs are hidden, you can type *layout* to make your changes or choose View > Model and Layout Tabs to display the tabs.

To rename a layout

1   Right-click the Layout tab to rename.

2   Type a new name for the layout.

3   Click OK.

The name can be up to 255 characters in length and can contain letters, numbers, the dollar sign ($), hyphen (-), and underscore (_), or any combination.

To delete a layout

1   Right-click the Layout tab to delete.

2   Click OK to confirm the deletion.

NOTE *You cannot delete the Model tab or the last remaining Layout tab. To delete all geometry from the Model tab or a Layout tab, first select all geometry and then use the Erase command.*

To reorder the Layout tabs

1   Right-click the Layout tab you want to move.

2   Do one of the following:

- Choose Move Right, and then choose a new location.

- Choose Move Left, and then choose a new location.

To view a list of all layouts

1   Type *layout* and then press Enter.

2   In the prompt box, choose ? to list all layouts.

3   Type *s* or press Enter to scroll through the layouts.

Working with layout viewports

When you begin working in a drawing on the Model tab, it consists of a single view of your model. You may have created additional views by dividing the drawing space into multiple windows; each window is a separate viewport on the Model tab.

On a Layout tab, you must create at least one layout viewport to see your model. However, you can create multiple layout viewports that display unique views of your model located in model space. Each layout viewport functions as a window into your model space drawing. You can separately control the view, scale, and contents of each layout viewport. Each layout viewport is created as a separate entity that you can move, copy, or delete.

Click any layout viewport to make it the current viewport, and then add or modify model space entities in that viewport, even while using entity snaps for model space entities. Any changes you make in one layout viewport are immediately visible in the other viewports (if the other layout viewports are displaying that portion of the drawing). Zooming or panning in the current viewport affects only that viewport.

This section focusses on working with layout viewports in paper space on a Layout tab. For additional information about viewports in model space, see "Dividing the current window into multiple views".
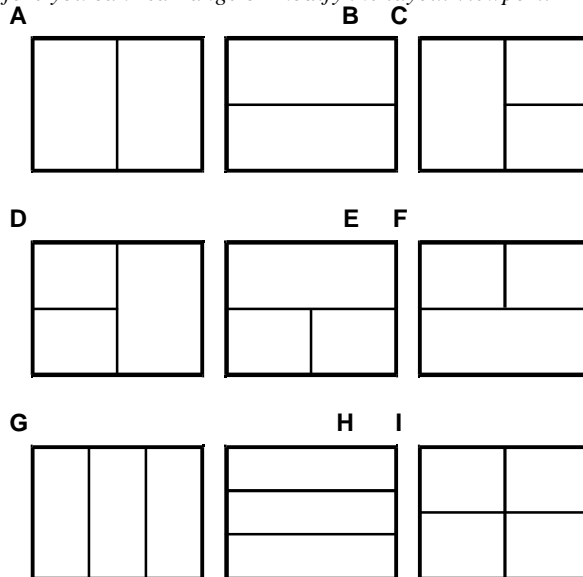
Creating layout viewports

The first time you switch to a Layout tab, all of your model space entities disappear. You must create at least one layout viewport on the Layout tab to see your work. You can create layout viewports anywhere inside the drawing area. You can control the number of viewports created and the arrangement of the viewports.

To create layout viewports

1 Do one of the following:

- Choose View > Layout Viewports.

- On the Views toolbar, click the Layout Viewports tool ( ).

- Type *mview* and then press Enter.

2 In the prompt box, choose Fit To View, Create 2 Viewports, Create 3 Viewports, or Create 4 Viewports, or specify two opposing corners to create a custom viewport.

3 In the prompt box, choose the viewport orientation.

4 Do one of the following:

- To arrange the viewports to fill the current graphic area, in the prompt box, choose Fit To Screen.

- To fit the viewports within a bounding rectangle, specify the corners of a rectangle.

TIP *When you create a layout viewport, the layout viewport border is created on the current layer. You can make layout viewport borders invisible by creating a new layer before you create layout viewports and then turning off that layer after you cre ate the layout viewports. To select a layout viewport's borders, you must turn that layer back on before you can rearrange or modify the layout viewport.*

You can create a single layout viewport, or you can divide the graphic area into two viewports arranged vertically (**A**) or horizontally (**B**); three viewports arranged left (**C**), right (**D**), above (**E**), below (**F**), vertically (**G**), or horizontally (**H**); or four viewports (**I**).

Viewing and scaling layout viewports

If you have created numerous layout viewports, your system performance may be affected. If necessary, you can turn a layout viewport on or off. Turning off a layout viewport does not delete the viewport or its contents; it simply turns off its display.

You can also change how you view items within a layout viewport by specifying a scale factor, which changes how large or small model space entities appear within the layout viewport.

To turn layout viewports on or off

1 Click the desired Layout tab.

2 Do one of the following:
- Choose View > Layout Viewports.
- On the View toolbar, click the Layout Viewports tool ( ).
- Type *mview* and then press Enter.

3 Choose On or Off.

4 Select the edge of the layout viewport to turn on or off, and then press Enter.

To change the layout viewport scale

1 Do one of the following:
- Choose Modify > Properties.
- On the Modify toolbar, click the Properties tool ( ).
- Type *entprop* and then press Enter.

2 Select the edge of the layout viewport.

3 In Scale, enter the scale at which you want to view model space entities from within the layout viewport.

4 Click OK.

To change the scale of model space entities relative to paper space

1 Click the Model tab.

2 Click a viewport to make it current.

3 Choose View > Zoom > Zoom.

4 Type the zoom scale factor relative to paper space by appending the suffix *xp* to the scale factor, and then press Enter. For example, to increase the scale of the entities in the viewport on the Model tab to twice the size of paper space units, type *2xp*. To decrease the scale to half the size of paper space units, type *.5xp*.

Modifying layout viewports

After you create layout viewports, you can modify them as needed. On the Layout tab, you can snap to the viewport borders using entity snaps. You can copy, delete, move, scale, and stretch layout viewports as you would any other drawing entity. You can also lock a layout viewport and assign a UCS to a layout viewport.

Modifying a layout viewport on a Layout tab does not affect the model space entities within the layout viewport.

To modify layout viewport properties

1   Click the desired Layout tab.

2   Do one of the following:
- Choose Modify > Properties.
- On the Modify toolbar, click the Properties tool
- Type *entprop* and then press Enter.

3   Select the edge of the layout viewport you want to modify.

4   Adjust the center point, width, or height of the viewport.

5   In Scale, enter the scale at which you want to view model space entities from within the layout viewport.

6   Mark Display Viewport to view model space entities from within the layout viewport.

7   Mark Lock Viewport to lock the viewport scale and view in model space when panning or zooming in the layout viewport.

8   Mark UCS per Viewport to specify a UCS for the layout viewport.

9   Click OK.

NOTE *You can select only layout viewports for modification. If you click a viewport on the Model tab, it makes that viewport active, not available for modification.*

Clipping layout viewports

You can clip layout viewports so that only a portion of the viewport contents are visi- ble. The visible portion can be in the shape of any entity.

Clipping can be turned on an off. If you turn off clipping for a layout viewport, all viewport contents are visible provided that the contents are on a layer that is on and thawed. The clipping information is retained however, and you can turn clipping back on at any time. If you delete clipping from a layout viewport, the clipping is removed permanently.

To clip a layout viewport

1   Click the desired Layout tab.

2   Do one of the following:
- Choose View > Layout Viewports.
- On the View toolbar, click the Layout Viewports tool (   ) .
- Type *mview* and then press Enter.

3   Choose Clip.

Customizing print options

Before you print, you can set up many aspects of printing:

- Specify paper size and orientation.

- Select and configure a printer or plotting device.

- Specify the view and scale of a printed drawing, including which portion of a drawing to print, the print scale, and the origin of the print area.

- Choose whether to print and scale lineweights.

- Choose whether to implement print style tables to control colors, pen widths, line- types, and lineweights.

- Open printer configuration (PCP) files; create and save PCP files.

Setting the paper size and orientation

You can specify a paper size and paper orientation for all drawings.

To select the paper size and orientation

1   Do one of the following:
- Choose File > Print Setup.

- On the Layouts toolbar, click the Print Setup tool ().

- Type *psetup* and then press Enter.
- Type *print* and press Enter, and then click Print Setup.

2   Select the paper size and paper orientation, and then click OK.

You can also adjust the orientation by printing a drawing upside down on the paper. Each layout in your drawing can specify whether to print upside down.

To print a drawing upside down

1   If necessary, click the desired Layout tab or the Model tab.

2   Do one of the following:

- Choose File > Print.

- Type *print* and then press Enter.

3   Click the Advanced tab.

4   Select Print Upside Down.

5   Select Save Changes to Layout, and then click Apply to save your changes.

Selecting a printer or plotter

You can specify a printer or plotter to be used when printing any drawing. You can print your drawing on any printer or plotter that is compatible with Windows, includ- ing raster printers.

To select a printer or plotter

1   Do one of the following:
   - Choose File > Print Setup.
   - On the Layouts toolbar, click the Print Setup tool ( ).
   - Type *psetup* and then press Enter.
   - Type *print* and press Enter, and then click Print Setup.

2   From the Printer Name list, select a printer or plotter, and then click OK.

Setting the scale and view

You can print or plot the entire drawing or a selected portion, depending on which options you select in the Print dialog box. You can choose to print what is visible on the screen, or you can specify to print an area of the drawing.

You can control the position of the drawing on the paper by specifying the origin of the print area, the location of the lower left corner of the print area, in relation to the lower left corner of the paper. The origin is normally set to 0,0, which places the lower left corner of the print area as close to the lower left corner of the paper as the printer or plotter will allow. You can specify a different origin, however, by specifying different coordinates.

When you create a drawing, you generally draw entities full-size. When you print the drawing, you can specify the scale of the resulting print or let the program adjust the size of the drawing to fit the paper. To print the drawing at a specific scale, specify the scale as a ratio of drawing units to printed units.

If you are printing from a Layout tab, the scale and view options you specify can be different for each layout that you create.

To automatically scale the drawing for printing

1   If necessary, click the desired Layout tab or the Model tab.

2   Do one of the following:
   - Choose File > Print.
   - Type *print* and then press Enter.

3   Click the Scale/View tab.

4   To scale the drawing to fit on one printed page, in Print Scale, select Scale to Fit.

5   Select Save Changes to Layout, and then click Apply to save your changes.

To specify the scale factor yourself

1   If necessary, click the desired Layout tab or the Model tab.

2   Do one of the following:

- Choose File > Print.

- Type *print* and then press Enter.

3   Click the Scale/View tab.

4   In Print Scale, do one of the following:

- Select a pre-defined scale. For example, choose 1:2 if you want 1 printed unit (inch or millimeter) to equal 2 drawing units.

- Select Custom, and then under User Defined Scale, type the ratio of printed units of measure (inches or millimeters) to drawing units.

5   To specify the printed units of measure, click Inches or Millimeters.

6   Select Save Changes to Layout, and then click Apply to save your changes.

To specify a portion of the drawing to print

1   If necessary, click the desired Layout tab or the Model tab.

2   Do one of the following:

- Choose File > Print.

- Type *print* and then press Enter.

3   Click the Scale/View tab.

4   Under Print Area, click one of the following:

- Current View – prints the view on the screen.

- Saved View – prints the selected saved view.

- Extents – prints the area that contains entities in the drawing.

- Limits – prints to the limits defined for the layout or drawing.

- Window – prints the portion of the drawing contained in the specified window, maintaining the aspect ratio of the windowed area to the drawing.

If you clicked Window, you must specify the window. Under Windowed Print Area, enter the diagonal x- and y-coordinates of the window, or select the area on the screen.

To print only the area in the window without considering whether there is extra space on the screen, select the Print Only Area Within Specified Window check box.

5   Under Entities To Print, click one of the following:

- All Entities Within Print Area – prints all drawing entities contained within the specified print area.

- Selected Entities Within Print Area – prints only the entities you select from within the specified

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print area.

6   Select Save Changes to Layout, and then click Apply to save your changes.

A   Displays either "Model" or the layout name to which the print settings apply.

B   Click to select the area of the drawing that you want to print.

C   Type the x- and y-coordinates of the two opposing corners of the rectangular area to print; or, to specify coordinates in the drawing window, click Select Print Area.

D   Select to print the area of the window while ignoring the aspect ratio to the remainder of the drawing.

E   Click to specify drawing units and paper size in millimeters or inches.
F   Specify the custom scale for the print area by typing the ratio of drawing units to printed inches or printed millimeters. (Available only if Custom is selected for Print Scale.)

G   Select to fit the specified print area to the current paper size, select a scale ratio, or select to specify your own custom scale in User Defined Scale.

H   Click to print only selected entities within the specified print area.

I   Click to print all entities within the specified print area.

J   Select to save the print settings for the model or layout.

K   Select to print using the current print settings or using the settings from the last time you printed.
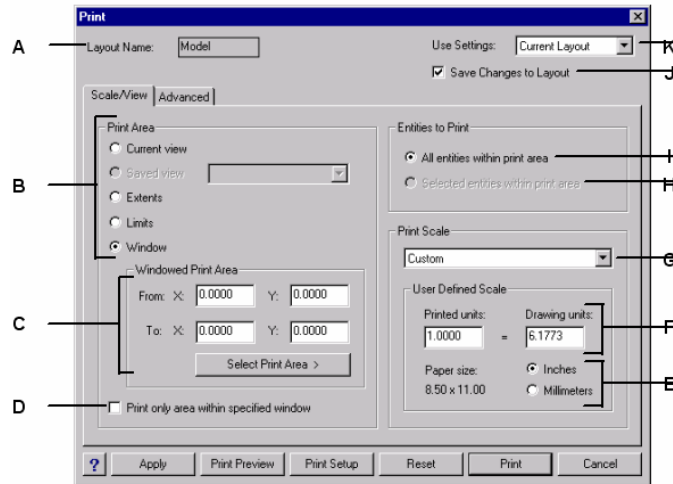
NOTE *BtoCAD saves your print settings each time you print. To restore the IntelliCAD default print settings, click Reset in the Print dialog box.*

To specify the print area origin

1   If necessary, click the desired Layout tab or the Model tab.

2   Do one of the following:

   •   Choose File > Print.

   •   Type *print* and then press Enter.

3   Click the Advanced tab.

4   Under Origin of Print Area, do one of the following:

- To center the specified print area on the printed page, select the Center on Page check box.

- To specify an origin for the print area, type the x- and y-coordinates, or click Select Origin and then specify a point on the drawing.

5   Select Save Changes to Layout, and then click Apply to save your changes.

Choosing how lineweights print

If entities are assigned lineweights, you can control whether they print with the assigned lineweights. If you turn off lineweight printing, entities print with a default outline. You can also control whether lineweights print in proportion to the scale you set on the Scale/View tab. Each layout in your drawing can specify whether to print and scale lineweights.

To set lineweight options

1   If necessary, click the desired Layout tab or the Model tab.

2   Do one of the following:

- Choose File > Print.

- Type *print* and press Enter.

3   Click the Advanced tab.

4   Choose how you want to print lineweights:

- Print Lineweights – prints entities with their assigned lineweights. If not selected, entities print with a default outline.

- Scale Lineweights – prints lineweights in proportion to the scale you set on the Scale/View tab. If not selected, lineweights print at their assigned size without adjustments for the print scale. (A Layout tab must be active to scale lineweights.)

5   Select Save Changes to Layout, and then click Apply to save your changes.

NOTE *Print styles can also affect how lineweights print. For details, see the next section.*

A   Displays either "Model" or the layout name to which the print settings apply.

B   Select a print style table.

C   Click to load print settings from a PCP file.

D   Click to save the current configuration as a PCP file.

E   Select to print the drawing upside down on your printer.

F   Select to print entities with their assigned lineweights.

G   Select to print lineweights in proportion to the scale you set on the Scale/View tab. (A Layout tab must be active.)

H   Click to specify the print area origin by selecting a point within the drawing.

I   Type x- and y-coordinates to specify the origin of the print area.

J   Select to center the print area on the page.

K   Click to make changes to the selected print style table.

L   Click to create a new print style table.

M   Select to save the print settings for the model or layout.

N   Select to load print settings from the current layout (or model) or the last time you printed.
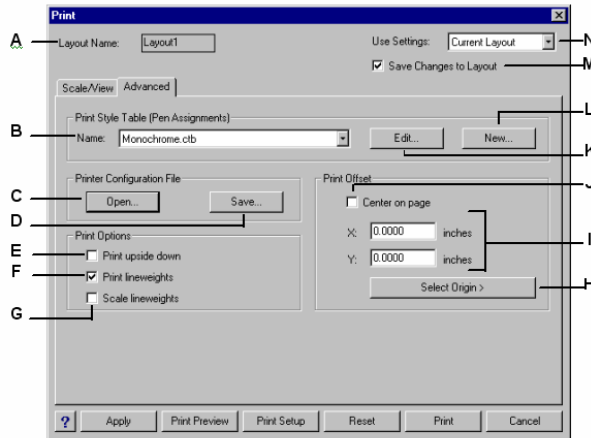
NOTE *BtoCAD saves your print settings each time you print. To restore the IntelliCAD default print settings, click Reset in the Print dialog box.*

Using print styles

BtoCAD uses print styles to change the appearance of your printed drawing without modifying the actual entities in your drawing. Assigning print styles allows you to customize the color, pen width, linetype, and lineweight that are used to print your drawing.

Print styles help you control what your drawing looks like when it is printed. Rather than describe what an entity looks like on your screen, print styles describe what an entity will look like when you print it. For example, you can map all yellow entities in your drawing to print in blue without modifying the actual entities. You could also map all yellow entities to print with whatever lineweight, linetype, or pen width that you specify.

Because print styles are saved in print style tables, which are files located on your computer, disk, or server, you can reuse them to help eliminate the need to reconfig- ure your print settings each time you print a drawing. For example, you may have multiple clients who have their own printing preferences. You can save print styles in a named file for each of your clients. You can even share the file with co-workers, or store the files on a network to ensure that everyone in your office uses the same stan- dards.

Understanding print style tables
A print style table is a collection of print styles that allows you to change the appearance of your printed

drawing without modifying the actual entities in your drawing. Each print style table is saved in a file that can be located on your computer, disk, or server.

A drawing can use one type of print style table at a time. There are two types of print style tables:

- **Color-dependent print style tables (CTB)** contain a collection of print styles based on each of the 255 index colors available in a drawing. True colors and color books are not applicable to color-dependent print style tables.

- **Named print style tables (STB)** contain a collection of print styles that you define. They can vary regardless of color.

With color-dependent print style tables, you cannot assign print styles to individual entities or layers. To use these print styles, you assign a specific color to an entity or layer. When you specify a color-dependent print style table at printing time, the entity colors and layer colors map to color-based print styles in the print style table that you specify.

With named print style tables, you can assign named print styles to individual entities and layers. Entities and layers assigned print styles are printed according to the print style table that you specify at printing time. If you specify a print style for a specific entity, that print style overrides any print style assigned to the layer on which the entity resides.

Sometimes a named print style assigned to an entity or layer is not located in the print style table that is assigned to a layout or drawing. This can happen if the print style has been deleted from the named print style table or if you assign a different named print style table to the drawing that does not contain the named print style. In this case, the entity is printed using its default properties, which is similar to assigning the Normal print style to an entity or layer. If you plan on interchanging named print style tables within the same drawing, it's a good idea to coordinate the tables to use the same print style names.

If a new drawing is based on a template, the new drawing uses the same type of print style table as the template. If a new drawing is created without a template, the type of print style table is specified in the New Drawing Wizard; by default, the new drawing uses the print style table type specified in Tools > Options on the Printing tab. Every drawing is designed to use print style tables, but you decide whether to implement them.

Implementing print style tables

Every drawing is designed to use print style tables, but you decide whether to implement them. Even if you use one of the default print style tables available with IntelliCAD, using print style tables requires planning ahead of time to ensure that your drawing prints as planned.

For example, a single drawing of a floor plan might require the printing of the following drawing sheets:

- **Main Floor Plan** Walls print with thick, black lines.
- **Electrical Plan** Walls print with normal gray lines, indicating that they are not the focus.
- **HVAC Plan** Walls print with normal gray lines, indicating that they are not the focus.
- **Roof Plan** Walls print with thin, gray lines and a hidden linetype, indicating that they are hidden under the roof in a plan view.

In this example, you can create four named print style tables, each containing a print style named "WallPstyle". Each print style table contains its own settings for "WallPstyle" to control how the walls print. Assign WallPstyle to either the wall entities or to a wall layer. Then, assign a different named print style table each time you print, or create four layouts and assign a different print style table to each layout.

Assigning print style tables

Select a print style table before printing if you want to change how your drawing appears when you print it. Print style tables can modify how colors, pen widths, line- types, and lineweights look when they are printed.

You can assign print style tables globally for all layouts (including the Model tab), or individually for the Model tab or a Layout tab. Assigning a print style table to an individual layout allows you to further customize the layouts you use to print a drawing.

However, assigning different named print style tables to various layouts may result in mismatched print style names; a named print style assigned to an entity or layer may not be located in the assigned print style table at print time. In this case, entities are printed using their default properties, which is similar to assigning the Normal print style to an entity or layer.

To assign print style tables

1   If necessary, click the desired Layout tab, or click the Model tab.

2   Do one of the following:

•   Choose File > Print.

•   Type *print* and then press Enter.

3   Click the Advanced tab.

4   Under Print Style Table (Pen Assignments), select a print style table that you cre- ated or one of the following:

•   **None** — Applies no print style table. Entities print according to their own properties.

•   **Icad** — Uses the default print style table and its color assignments.

•   **Monochrome** — Prints all colors as black.

5   At the prompt, choose Yes to assign the print style table to all layouts in the draw- ing, including the Model tab, or choose No to assign the print style table only to
the individual layout listed in Layout Name on the Print dialog box.

6   Select Save Changes to Layout, and then click Apply to save your changes.

NOTE *When your drawing was first created it was set up to use either color-dependent or named print style tables. For details on converting a drawing to use a different type of print style table, see "Understanding print style tables".*

Creating new print style tables

BtoCAD offers several print style tables to help you get started. If you want to cus- tomize your print output further, you can create your own print style table. You can create a new print style table entirely from scratch, based on BtoCAD registry set- tings, or by importing a printer configuration file (PCP file).

To create new print style tables

1 Do one of the following:

- Choose File > Print Styles Manager.

- Type *stylesmanager* and then press Enter.

2 Double-click Create a Print Style Table.

3 Complete the setup wizard. On the last wizard page, you can click Print Style Table Editor to set up the print styles for the table. For more details about the Print Style Table Editor options, see the next section.

Modifying print style tables

When your drawing was created, it was set up to use color-dependent or named print style tables.

Color-dependent print style tables (.ctb files) You can modify individual print styles within the table, but you cannot add, rename, or delete print styles. Color-dependent print style tables always have 255 print styles, each named for a specific color. Your changes affect all entities and layers assigned that color.

Named print style tables (.stb files) You can add, modify, rename, and delete individual print styles within the table. However, you cannot modify, rename, or delete the Nor- mal print style. Your changes affect all entities and layers that are assigned that print style name.

TIP *If you can't remember what type of print style table is assigned to your drawing, use the PSTYLEMODE system variable to determine the print style table type.*
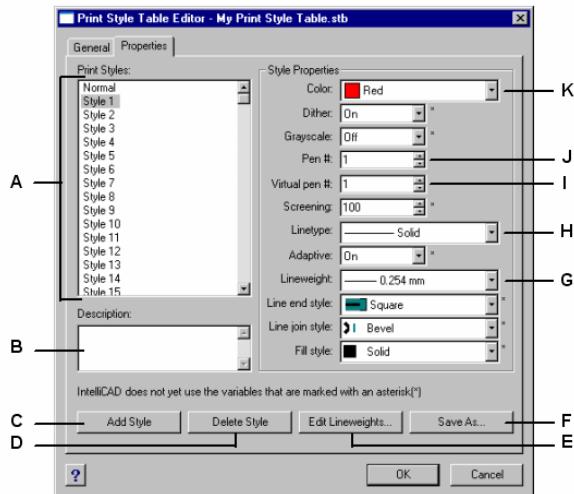
Each print style within a print style table specifies a color, pen numbers, linetype, and lineweight. BtoCAD recognizes additional characteristics for compatibility with AutoCAD only, including: dither, grayscale, screening, adaptive, line end style, line join style, and fill style. When specifying print style characteristics, be sure to consider the limitations of your output device.

NOTE *It is recommended that you only modify print style tables that you have created. If you modify a default print style table that came with BtoCAD, you overwrite the original information which is then lost.*

To modify print style tables

1 Do one of the following:

- Choose File > Print Styles Manager.

- Type *stylesmanager* and then press Enter.

2 Double-click the print style table you want to modify.

3 Click the General tab, and then do any of the following:

- Enter a new table description.

- Select Apply Global Scale Factor to Non-ISO Linetypes to apply the scale factor to non-ISO linetypes used for any print style in the current print style table. This also applies to fill patterns, which are not used in BtoCAD, but are recognized for compatibility with AutoCAD.

- Enter a scale factor to apply to non-ISO linetypes used for any print style in the current print style table.

4 Click the Properties tab, and then do any of the following:

- Make format changes to a print style by selecting it in the Print Styles list, then make color, pen map, linetype, or lineweight changes for the print style. Your changes are saved automatically for the selected print style.

- Add a new print style by clicking Add Style. Enter a new name, and then click OK. Select the options for the print style. (Available for named print styles only.)

- Rename a print style by selecting it in the Print Styles list. Single-click the print style again, and then enter a new name. (Available for named print styles only.)

- Delete a print style by selecting it in the Print Styles list. Click Delete Style.
(Available for named print styles only.)

5 Click OK.

A Select a print style to modify it.

B Enter a description for the selected print style.

C Click to create a new print style. (Named print style tables only.)

D Click to delete the selected print style. (Named print style tables only.)

E Click to modify the list of available lineweights for the current print style table.

F Click to save the print style table with a new name or in a new location.

G Choose a lineweight for the selected print style.

H Choose a linetype for the selected print style.

I Type or scroll to the width of the virtual pen for the selected print style (for printers that don't have physical pens, such as laser or inkjet printers).

J Type or scroll to the width of the physical pen for the selected print style.

K Choose a color for the selected print style.

Copy, rename, or delete print style tables

Copy, rename, or delete a print style table just as you would any other file on your computer. Regardless of which print style table type your drawing uses, you can use the Print Style Manager to manage both color-dependent and named print style tables.

To copy, rename, or delete print style tables

1   Do one of the following:

- Choose File > Print Styles Manager.

- Type *stylesmanager* and then press Enter.

2   Select a print style table. Color-dependent print style tables are .ctb files and named print style tables are .stb files.

3   Copy, rename, or delete the print style table just as you would any other file on your computer.

Changing a drawing's print style table type

When your drawing was created, it was set up to use color-dependent print style tables (.ctb files) or named print style tables (.stb files). A drawing can use one type of print style table at a time. If necessary, after a drawing is created you can convert the drawing to use the other type of print style table.

If you can't remember what type of print style table is assigned to your drawing, use the PSTYLEMODE system variable to determine the print style table type.

To change a drawing to use named print style tables

1   If you want to reuse any of the existing print style information, convert your color- dependent print style tables to named print style tables. For details, see "To convert a color-dependent print style table to a named print style table".

2   Open the drawing that uses color-dependent print style tables (.ctb files).

3   Type *convertpstyles*, and then press Enter.

4   If you have already converted your individual color-dependent print style tables to named tables, click OK in the prompt that displays.

If you have not converted the tables, click Cancel. First use *convertctb* to convert your color-dependent print style tables to named tables. If you do not, all of the print style information you specified in your drawing will be lost.

5   Select a named print style table (.stb file) that you want to use with the drawing.

6   Click Open.

NOTE *If you have not converted a color-dependent print style table to a named print style table, you will be warned that the table you have selected does not contain color mapping and the drawing cannot be converted.*

To change a drawing to use color-dependent print style tables

1   Open the drawing that uses named print style tables (.stb files).

2   Type *convertpstyles*, and then press Enter.

3   If you are sure you want to convert the drawing and lose all print style assignments, click OK in the prompt that displays.If you do not want to lose the print style assignments, click Cancel.

NOTE *Converting a drawing to use color-dependent print style tables will remove all of the named print style information from entities and layers. However, the named print style tables are not deleted from your computer.*

Converting print style tables

You can convert a color-dependent print style table to a named print style table. You cannot convert a named print style table to a color-dependent print style table because color-dependent tables contain only print styles that are named after the 255 colors to which they map. Converting a color-dependent print style table to a named print style table can be helpful in the following situations:

• You don't want to create a named print style table from scratch.

• You want to create a named print style table that has the same settings as a color- dependent print style table, but with some new print styles or other custom set- tings.

• You want to convert a drawing to use named print style tables and you want to reuse most of print styles already defined in a color-dependent print style table.

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To convert a color-dependent print style table to a named print style table

1   Type *convertctb*, and then press Enter.

2   Select the color-dependent print style table (.ctb file) that you want to convert.

3   Click Open.

4   Enter a name for the new named print style table (.stb file).

5   Click Save.

The print styles in the new table are named Style 1, Style 2, and so on. If want to use different print style names, rename the print styles before you assign them to entities and layers in your drawing. If you rename the print styles after assigning them, they will not match when you print your drawing. For information about renaming print styles, see "Modifying print style tables".

Turning off print style tables

When you turn off print style tables, entities print according to their own properties. However, all of the print style information is saved so you can easily turn on print styles again. Actual print style table files are not deleted, and for drawings that use named print style tables, entities and layers retain their assigned print styles.

To turn off print style tables

1   If necessary, click the desired Layout tab, or click the Model tab.

2   Do one of the following:

   •   Choose File > Print.

   •   Type *print* and then press Enter.

3   Click the Advanced tab.

4   Under Print Style Table (Pen Assignments), select None.

5   At the prompt, choose Yes to turn off print style tables for all layouts in the drawing, including the Model tab, or choose No to turn off print style tables only for the individual layout listed in Layout Name on the Print dialog box.

6   Select Save Changes to Layout, and then click Apply to save your changes.

Reusing print settings

After you set up print settings for a drawing or layout, you can save them and reuse them again using printer configuration files.

Printer configuration files store the printer information you create for specific drawings or layouts, which eliminates the need to completely reconfigure your print settings each time you print a drawing. Printer configuration files also allow you to share and reuse print settings between different drawings and layouts.

BtoCAD supports the printer configuration (PCP) file format used by AutoCAD. This feature makes it possible to use existing PCP files saved in AutoCAD, as well as to save your BtoCAD print configuration settings to a PCP format.

TIP *You can convert an AutoCAD PC2 file to PCP format using the Device And Default selection feature in the AutoCAD Print dialog box.*

The printer configuration files that you specify can be different for each layout that you create.

To save print settings in a PCP file

1   If necessary, click the desired Layout tab or the Model tab.

2   Do one of the following:

- Choose File > Print.

- Type *print* and then press Enter.

3   Click the Advanced tab.

4   Specify the desired print settings.

5   Under Printer Configuration File, click Save.

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6   Name the file, and then click Save.

To assign a PCP file

1   If necessary, click the desired Layout tab or the Model tab.

2   Do one of the following:

- Choose File > Print.

- Type *print* and then press Enter.

3   Click the Advanced tab.

4   Under Printer Configuration File, click Open.

5   Locate and select the PCP file, and then click Open.

6   Select Save Changes to Layout, and then click Apply to save your changes.

PCP files created before BtoCAD 6 contain obsolete PenMap/Width settings that you can convert to use with print style tables. Import the PCP file to create a new color-dependent print style table. For details, see "Creating new print style tables".

TIP *It is easy to reuse print settings from the last time you printed. In the Print dialog box under Use Settings, choose Previous Print.*

Printing or plotting your drawing

After you have configured your drawing and any layouts for printing, you are ready to print. If desired, you can preview your page before printing.

Previewing a drawing before printing

Viewing a drawing before printing gives you a preview of what your drawing will look like when it is printed. This helps you see if there are any changes you want to make before actually printing the drawing. If you are using print style tables, the preview shows how your drawing will print with the assigned print styles. For example, the preview may display different colors or lineweights than those used in the drawing because of assigned print styles.

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To preview a drawing before printing

1   If necessary, click the desired Layout tab or the Model tab.

2   Do one of the following:

      • Choose File > Print Preview.

      • On the Standard toolbar, click the Print Preview tool ().

      • Type *ppreview* and then press Enter.

3   After checking the preview image, do one of the following:

      • To print the drawing, click Print Settings to display the Print dialog box.

      • To return to the drawing, click Close.

Printing a drawing

The Print dialog box is organized by tabs into two functional areas: scaling and viewing, and advanced

printing options. The print setting options available under each tab were described in the previous sections.

NOTE *You cannot print a rendered image directly to a printer. To print a rendered image, you must first save the drawing to a different format and then print it from another graphics program.*

To print a drawing

1   If necessary, click the desired Layout tab or the Model tab.

2   Do one of the following:

      • Choose File > Print.

      • On the Standard toolbar, click the Print tool

If you click the Print tool, the Print dialog box does not display. Your drawing will be sent directly to the selected printer.

      • Type *print* and then press Enter.

3   From the Print dialog box, make any adjustments to the settings.

4   Click Print.

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NOTE *Instead of using the print settings you saved with your layout, you can select Previous Print in the Use Settings list to print according to the settings used the last time you printed. If necessary, you can click Reset to restore the BtoCAD default print settings.*