

## Working with blocks, attributes and external references

Blocks, attributes, and external references provide mechanisms for managing entities in your drawings and for including additional information with the standard drawing entities. With blocks, you can combine numerous entities into a single entity, and then reuse it, inserting multiple copies. With attributes, you can associate text, such as part numbers or prices, with blocks, and then extract the text-attribute information to a separate file, such as a database, for further analysis. With external references, you can link separate reference drawing files to a drawing to combine information without adding the contents of the reference drawings to the current drawing. If you make changes to the referenced file, all references are updated automatically.

This section explains how to:

- Create, insert, and redefine blocks.
- Create, edit, and insert attributes.
- Extract attribute data to a separate file.
- Attach and work with external references.

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## Working with blocks

Usually, blocks are several entities combined into one that you can insert into a drawing and manipulate as a single entity. A block can consist of visible entities such as lines, arcs, and circles, as well as visible or invisible data called attributes. Blocks are stored as part of the drawing file.

Blocks can help you better organize your work, quickly create and revise drawings, and reduce drawing file size. Using blocks, you can create a library of frequently used symbols. Then you can insert a symbol as a block rather than redraw the symbol from scratch.

After you create a block from multiple entities, you save it once, which also saves disk space. You insert only multiple references to a single block definition. You can change the block definition to quickly revise a drawing, and then update all instances of the block.

If you insert a block that contains entities originally drawn on layer 0 and assigned color and linetype `BYLAYER`, it is placed on the current layer and assumes the color and linetype of that layer. If you insert a block that contains entities originally drawn on other layers or with explicitly specified colors or linetypes, the block retains the original settings.

If you insert a block that contains entities originally assigned color and linetype `BYBLOCK`, and the block itself has the color and linetype `BYLAYER`, those entities adopt the color and linetype of the layer onto which they are inserted. If the block is assigned an explicit color or linetype, such as red or dashed, those entities adopt those qualities.

A procedure called nesting occurs when you include other blocks in a new block that you are creating. Nesting is useful when you want to combine and include small components, such as nuts and bolts, into a larger assembly and you need to insert multiple instances of that assembly into an even larger drawing.

### Creating blocks

The tools and commands for creating blocks appear on the Tools toolbar and the Tools menu, respectively, when you set the program to the Advanced experience level. You can also use the BtoCAD Explorer to create blocks.


You can create blocks in two ways:

- By saving a block for use within the current drawing only.
- By saving the block as a separate drawing file that you can insert into other drawings.

When you create a block, you specify its name, its insertion point, and the entities that compose the block. The insertion point is the base point for the block and serves as the reference point when you later insert the block into a drawing.

**To create a block for use within a current drawing**

Advanced experience level

- 1 Do one of the following:
  - Choose Draw >Block> Make Block.
  - On the Tools toolbar, click the Create Block tool (  )
  - Type *block* and then press Enter.
- 2 Type a name for the block, and then press Enter.
- 3 Specify the insertion point for the block.
- 4 Select the entities that you want in the block, and then press Enter.

The block is created and exists only in the current drawing. The entities you select are removed from the display, because they are now part of the block.

- 5 To restore the original entities to the drawing while retaining the new block, type *undelete* or *oops*.

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**NOTE** If you frequently restore original entities after you define a block, you can customize the program to add the *Undelete* command to a menu or toolbar.

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You can create a block as a separate drawing file that you can insert into other drawings.

**To save a block as a separate drawing file**

Advanced experience level

- 1 Do one of the following:
  - Choose Draw > Block>Make block.
  - On the Tools toolbar, click the Save Block To Disk tool (  ).
  - Type *wblock* and then press Enter.
- 2 In the File Name field, type the name of the drawing file you want to create.
- 3 Click Save.
- 4 In the prompt box, choose one of the following:
  - **Multiple Blocks** This option saves one or more existing block entities to a separate drawing file. When prompted, type the name of the block(s).
  - **All Entities** This option immediately saves the entire drawing to a separate drawing file.
  - **Select Entities** This option saves those entities you select to a separate drawing file. When prompted, specify the insertion point for the block, select the entities for the block, and then press Enter.

**NOTE** *When saving multiple blocks or saving the entire drawing as a separate drawing file, the program assigns the 0,0,0 coordinate as the insertion base point. You can change the base point by opening the drawing and redefining the block.*

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## Inserting blocks

You can insert blocks and other drawings into the current drawing. When you insert a block, it is treated as a single entity. When you insert a drawing, it is added to the current drawing as a block. You can then insert multiple instances of the block without reloading the original drawing file. If you change the original drawing file, those changes have no effect on the current drawing unless you redefine the block by reinserting the changed drawing.

You can also insert blocks from another drawing into the current drawing, using the BtoCAD Explorer. Both drawings must be open at the same time to do this. see Chapter 8, “Working with the BtoCAD Explorer.”

When you insert a block or drawing, you must specify the insertion point, scale, and rotation angle. The block’s insertion point is the reference point specified when you created the block. When you insert a drawing as a block, the program takes the specified insertion point as the block insertion point. You can change the insertion point, however, by first opening the original drawing and redefining the block.

### To insert a block

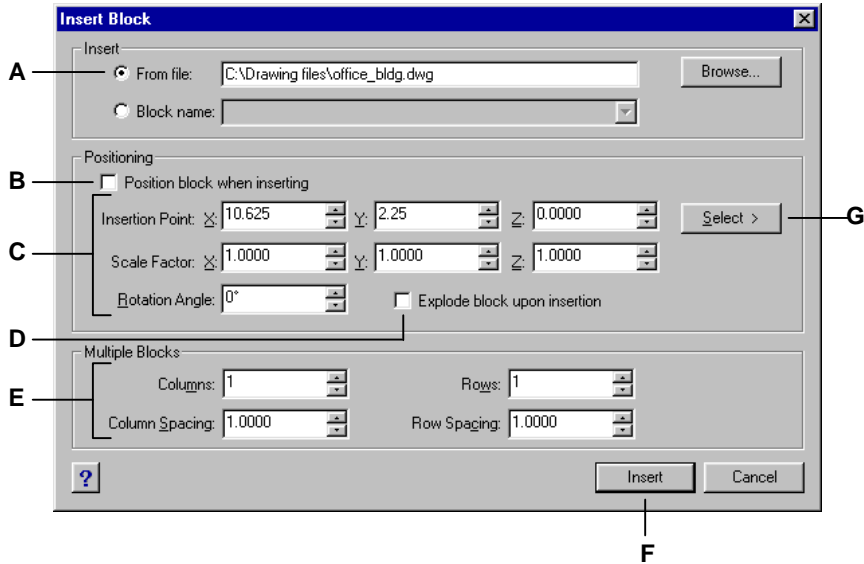
- 1 Display the Insert Block dialog box by doing one of the following:
  - Choose Insert > Insert Block.
  - Type *ddinsert* and then press Enter.
- 2 In the Insert Block dialog box, under Insert, click Block Name.
- 3 In the Block Name box, select the name of the block you want to insert.
- 4 Click Insert.
- 5 Specify the insertion point for the block.
- 6 Specify the x, y, and z scale factors and the rotation angle, or press Enter to accept the default values.

### To insert an entire drawing into the current drawing

- 1 Do one of the following:
  - Choose Insert > Insert Block.
  - Type *ddinsert* and then press Enter.
- 2 In the Insert Block dialog box, under Insert, click From File.
- 3 Type the path and the drawing file name, or click Browse to specify the file from the Insert Drawing dialog box, and click Open.
- 4 Click Insert.
- 5 Specify the insertion point for the block.

**6** Specify the x, y, and z scale factors and the rotation angle, or press Enter to accept the default values.

**NOTE** You can specify the insertion point, scale factors, and rotation angle in the *Insert Block* dialog box before inserting the block. You can also control whether the block is exploded back into its original component entities after insertion. Under *Positioning*, clear the *Position Block When Inserting* check box, and specify the appropriate coordinates. If you want to explode the block immediately, select the *Explode Upon Insertion* check box.



- A** Click and then enter the path and drawing file name to insert the entire drawing file as a block.
- B** Click to position the block when inserting.
- C** Specify the insertion point, scale factors, and rotation angle before you insert the block (available only when *Position Block When Inserting* is cleared).
- D** Click to explode the block on insertion.
- E** Specify columns, column spacing, rows, and row spacing.
- F** Click to insert the block.
- G** Click to select the block insertion point in the drawing before inserting the block (available only when the *Position Block When Inserting* check box is cleared).


## Redefining blocks

You can redefine all instances of a block within the current drawing. To redefine a block that was created in the current drawing, you create a new block using the same name. You can update all the blocks in the current drawing by redefining the block. If the block was inserted from a separate drawing file that was subsequently

updated, reinsert that block to update all other instances in the current drawing.

### To redefine a block in the current drawing

Advanced experience level

- 1 Do one of the following:
  - Choose Draw > Block>Make Block.
  - On the Tools toolbar, click the Create Block tool ().
  - Type *block* and then press Enter.
- 2 Type the name of the block you want to redefine, and then press Enter.
- 3 In the prompt box, choose Yes-Redefine Block.
- 4 Specify the insertion point for the block.
- 5 Select the entities for the block, and then press Enter.

The block is immediately redefined, and all instances of the block in the drawing are updated. The entities you select for inclusion in the block are removed from the drawing, because they are now part of the block.

**TIP** *To restore the original entities to the drawing while retaining the new block, type undelete or oops.*


You can update all instances of a block inserted from a separate drawing by reinsert- ing the drawing.

### Exploding blocks

You can explode an inserted block to its original component entities. When you explode a block, only that single instance of the block is affected. The original block definition remains in the drawing, and you can still insert additional copies of the original block. If you explode a block that contains attributes, the attributes are lost, but the original attribute definitions remain.

Exploding dissociates component entities to their next simplest level of complexity; blocks or polylines in a block become blocks or polylines again

#### To explode a block

- 1 Do one of the following:
  - Choose Modify > Explode.
  - On the Modify toolbar, click the Explode tool ().
  - Type *explode* and then press Enter.
- 2 Select the block.
- 3 Press Enter.

## Working with attributes

An attribute is a particular entity that you can save as part of a block definition. Attributes consist of text-based data. You can use attributes to track such things as part numbers and prices. Attributes have either fixed or variable values. When you insert a block containing attributes, the program adds the fixed values to the drawing along with the block, and you are prompted to supply any variable values.

After you insert blocks containing attributes, you can extract the attribute information to a separate file and then use that information in a spreadsheet or database to produce a parts list or bill of materials. You can also use attribute information to track the number of times a particular block is inserted into a drawing.


Attributes can be visible or hidden. Hidden attributes are neither displayed nor printed, but the information is still stored in the drawing and written to a file when you extract it.

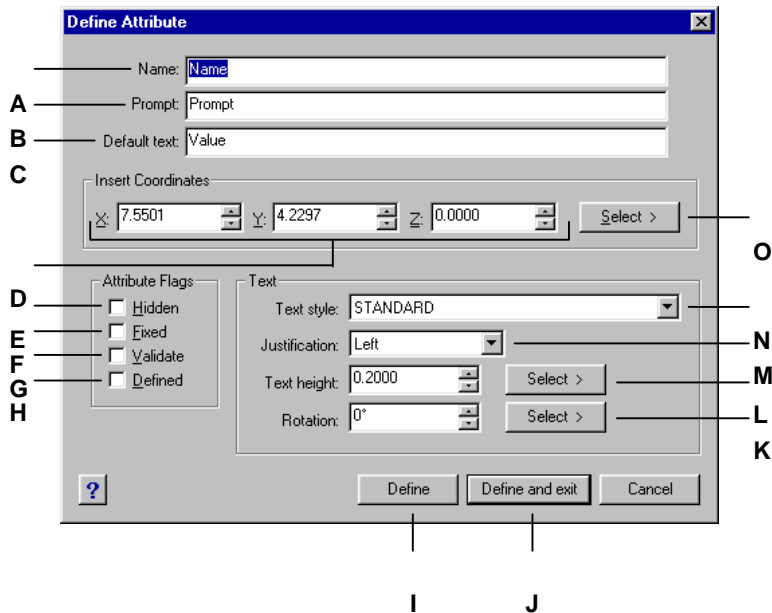
### Defining attributes

You add an attribute to a drawing by first defining it and then saving it as part of a block definition. To define an attribute, you specify the characteristics of the attribute, including its name, prompt, and default value; the location and text formatting; and optional modes (hidden, fixed, validate, and predefined).

#### To define an attribute

Advanced experience level

- 1 Do one of the following:
  - On the Tools toolbar, click the Define Attributes tool ().
  - Type *ddattdef* and then press Enter.
- 2 In the Define Attribute dialog box, type the name, prompt, and default value.
- 3 Under Insert Coordinates, specify the location of the attribute, or click Select to select a point in the drawing.
- 4 Under Attribute Flags, select the optional attribute modes.
- 5 Under Text, specify the text characteristics.
- 6 To add the attribute to the drawing, do one of the following:
  - Click Define to add the attribute and keep the dialog box active so you can define another attribute.
  - Click Define And Exit to add the attribute and end the command.



- A Type the name you want to assign to the attribute.
- B Enter the identifying prompt information displayed when you insert a block containing the attribute.
- C Enter the default or constant value. For variable attributes, the default value is replaced by the actual value when you later insert a block containing the attribute.
- D Specify the x-, y-, and z-coordinates for the attribute insertion point.
- E Select to create a hidden attribute.
- F Select to create a fixed-value attribute.
- G Select to create an attribute whose value must be validated when you later insert a block containing the attribute.
- H Select to create an attribute whose value is defined and not requested when you later insert a block containing the attribute, but that you can edit after the block is inserted.
- I Click to add the attribute and keep the dialog box active so you can define another attribute.
- J Click to add the attribute and end the command.
- K Specify the text rotation angle, or click to specify the rotation angle by selecting two points in the drawing.
- L Specify the text height, or click to specify the height by selecting two points in the drawing.
- M Choose the text justification.
- N Choose the text style from those styles already defined in the drawing.
- O Click to specify the attribute insertion point by selecting a point in the drawing.




## Editing attribute definitions

You can edit an attribute definition before you associate it with a block and before it is saved as part of a block definition.

### To edit an attribute definition

Advanced experience level

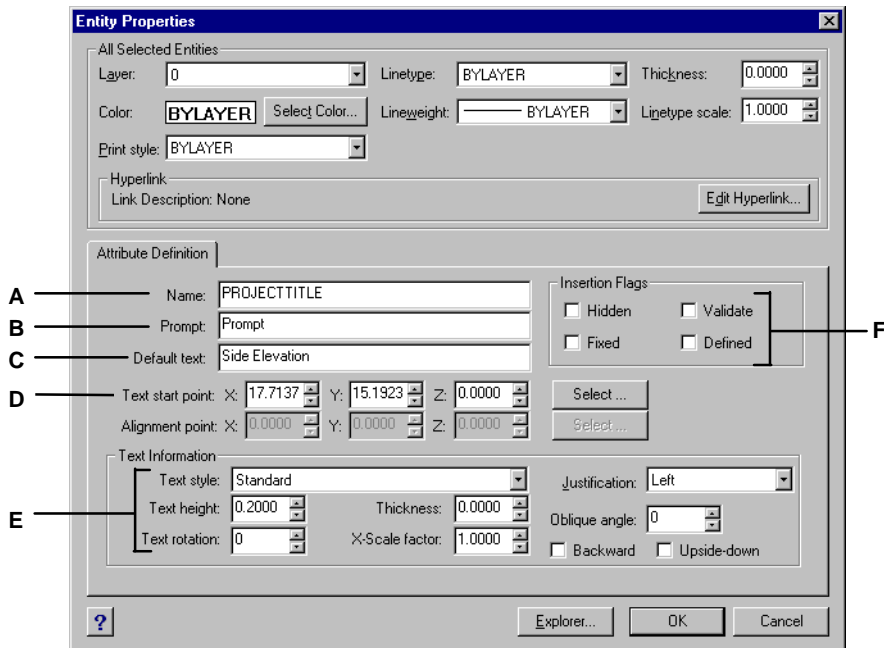
1 Do one of the following:

- Choose Modify > Edit Text.
- On the Modify toolbar, click the Edit Text tool (  ).
- Type *ddedit* and then press Enter.

2 Select the attribute definition text to edit.

3 Modify the attribute name, prompt string, default value, and other attribute definition characteristics.

4 Click OK.



- A** Modify the name assigned to the attribute.
- B** Modify the prompt that displays when you insert the attribute into the drawing.
- C** Modify the identifying prompt information displayed when you insert a block containing the attribute. Modify the attribute insertion point.
- E** Modify the attribute text style and appearance.
- F** Modify the attribute insertion flags to create hidden, fixed-value, validated, or defined attributes.

### Attaching attributes to blocks


You can attach attributes to a block after you define it and select it as one of the entities to include. Include the attributes when the program prompts you for the entities to include in the selection set for a block. After the attribute is incorporated into a block, the program prompts you each time you insert the block, so you can specify different values for the attributes each time you insert it into a new drawing.

### Editing attributes attached to blocks

You can edit the attribute values of a block that has been inserted into a drawing.

#### To edit an attribute attached to a block

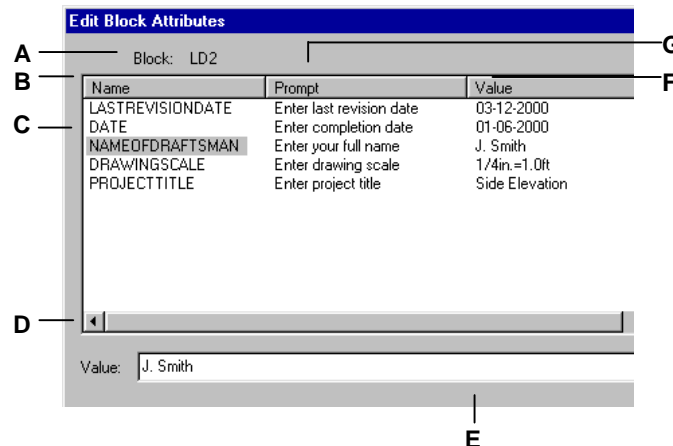
Advanced experience level

- 1 Do one of the following:
  - Choose Modify > Attribute > Edit Block Attributes.
  - On the Tools toolbar, click the Edit Block Attributes tool ().
  - Type *ddate* and then press Enter.

- 2 Select the block to edit.

The Edit Block Attributes dialog box displays all the attributes attached to the block you select.

- 3 Edit the attribute values as necessary.
- 4 Click OK.



- A Identifies the name of the block.
- B Displays the names of all attributes attached to the block.
- C Click to select attribute.
- D Type the new value for the selected attribute.
- E Click to update the attribute values and exit.
- F Displays the value of each attribute attached to the block.
- G Displays the prompt for each attribute attached to the block.

### Extracting attribute information

You can extract attribute information from a drawing and save it to a separate text file for use with a database program. You can save the file in any of the following formats:

- **Comma Delimited Format (CDF)** Contains one line for each instance of a block, with individual attribute fields separated by commas. Character string fields are enclosed with single quotation marks. You must specify a template file when extracting to a CDF file.
- **Space Delimited Format (SDF)** Contains one line for each instance of a block. Each attribute field has a fixed length; there are no separators or character string delimiters. You must specify a template file when extracting to a SDF file.
- **Drawing Exchange Format (DXF)** Creates a subset of a standard DXF file (a \*.dxf file) containing all the information about each block, including the insertion points, rotation angles, and attribute values. No template file is required.

Before extracting attributes to a CDF or SDF file, you must create a template file. The template file is an ASCII text file that specifies the attribute data fields to be written in the extract file. Each line of the template file specifies one attribute field. BtoCAD recognizes 15 different fields, which contain elements such as the block name, the x-, y-, and z-coordinates of its insertion point, the layer on which it is inserted, and so on. You can include any of these fields. The template file must include at least one attribute name.

Each line in the template file must start with the field name. Block name and insertion-point values must begin with *BL:*. The next nonblank character must be either a *C* (indicating a character string field) or an *N* (indicating a numeric field). This character is then followed by three digits indicating the width of the field (in characters). The final three digits indicate the number of decimal places (for numeric fields). In the case of character fields, the last three digits must be zeros (000). A typical template file is similar to the one shown here:


	A				G
B	BL:DESK				
	BL:X				F
C	BL:Y				E
	TYPE				
	MANUFACTURER				
D	MODELNUMBER				
	COST				

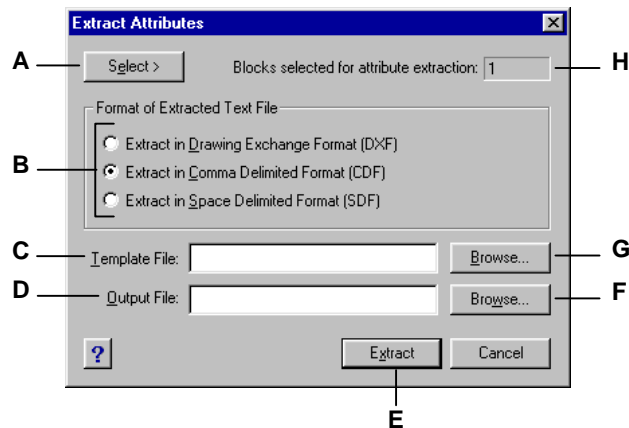
- A Field name.
- B Block name. Block names must begin with *BL*.
- C Insertion point. Insertion-point values must begin with *BL*.
- D Attribute tags.
- E Number of decimal places for numeric fields or *000* for character fields.
- F Field width for character or numeric fields.
- G C for character fields; N for numeric fields.

#### To create a template file

- 1 Create a template file using any ASCII text editor (such as Microsoft® Notepad or Microsoft® WordPad) or a word-processing program such as Microsoft® Word.
- 2 Include the necessary fields in the template file.
- 3 Save the template file in ASCII text format.

#### To extract attribute information

- 1 Do one of the following:
  - Choose Tools > Extract Attributes.
  - On the Tools toolbar, click the Extract Attributes tool (  ).
  - Type *ddatttext* and then press Enter.
- 2 Click Select, specify the entities from which to extract attributes, and then press Enter.
- 3 Specify the format of the extracted file.
- 4 For CDF and SDF formats, specify the template file.
- 5 Specify the extract output file.
- 6 Click Extract.



- A** Click to select blocks in the drawing containing attributes you want to extract.
- B** Click to specify the format of the extracted file.
- C** Specify the template file for CDF and SDF extracts.
- D** Specify the extract output file.
- E** Click to extract attributes.
- F** Click to specify the output file using a file dialog box.
- G** Click to specify the template file using a file dialog box.
- H** Indicates the number of blocks with attributes selected for extraction.

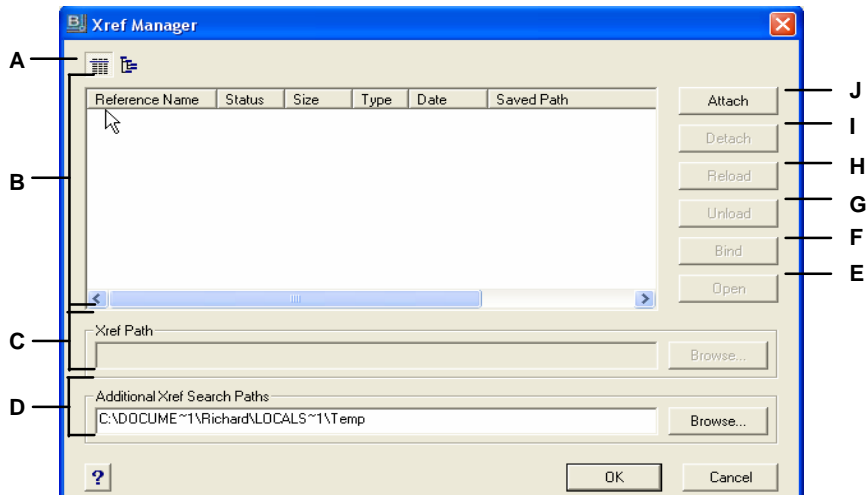
## Working with external references

You can link entire drawings to the current drawing as external references. Unlike inserting a drawing as a block, in which you add all the entities from the separate drawing into the current drawing, external references attach a pointer to the external file. The entities in the external reference appear in the current drawing, but the entities themselves are not added to the drawing. Thus, attaching an external reference does not significantly increase the size of the current drawing file.

External references provide additional capabilities not available when you insert a drawing as a block. When you insert a drawing as a block, the entities are stored in the drawing. Any changes you make to the original drawing are not reflected in the drawing in which you inserted it. When you attach an external reference, however, any changes you make to the original drawing file are reflected in the drawings that reference it. These changes appear automatically each time you open the drawing containing the external reference. If you know that the original drawing was modified, you can reload the external reference anytime you're working on the drawing.

External references are useful for assembling master drawings from component drawings. Use external references to coordinate your work with others in a group. External references help reduce drawing file size and ensure that you are always working with the most recent version of a drawing.

The Xref Manager helps you easily attach and work with external references.



- A Click to display a detailed list or hierarchical tree.
- B Select an external reference to modify its attachment.
- C Type or click Browse to specify the external reference location.
- D Type or click Browse to specify other search directories where external references may be located.
- E Click to open the source drawing for the external reference.
- F Click to make the external reference a permanent part of the drawing.
- G Click to remove the external reference, but keep elements and path information for easy reloading.
- H Click to update with changes from the external reference.
- I Click to completely remove the external reference.
- J Click to link a drawing.

## Attaching external references

Attaching a separate drawing to the current one creates an external reference. The external reference appears in the drawing as a block definition, but the drawing entities are linked rather than added to the current drawing. If you modify the linked drawing, the current drawing that contains the external reference is updated automatically when you open it, or you can reload the external reference manually so it reflects the latest version of the external reference.


When you attach an external reference, its layers, linetypes, text styles, and other elements are not added to the current drawing. Rather, these elements are also linked from the referenced file.

There are two ways you can attach an external reference:

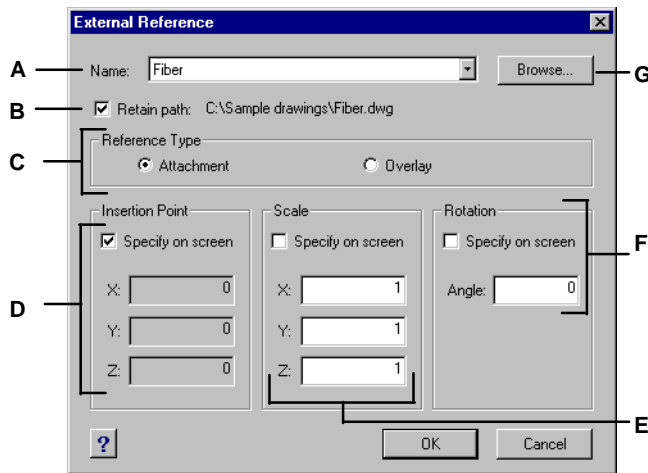
- An attachment is an inserted drawing that contains a link to the original file. Attachments can themselves contain other, nested reference files. When you attach an external reference, any nested references contained in the file also appear in the current drawing.
- An overlay is an inserted drawing that contains a link to the original file. Overlays allow you to lay a drawing on top of another drawing, similar to the way you work manually with transparencies. When a drawing that contains overlaid external references is itself attached or overlaid as an external reference in another drawing, the overlays do not appear as part of the external reference. Use overlaying when you want to see reference geometry in a drawing but you do not need to include that geometry in drawings that will be used by others (nested external references).

You can attach as many copies of an external reference file as you want. Each copy can have a different position, scale, and rotation angle.

### To attach an external reference

- 1 Do one of the following:
  - Choose Insert > Xref Manager.
  - On the Tools toolbar, click the Xref Manager tool ()
  - Type *xrm* and then press Enter.
- 2 Click Attach.
- 3 Specify the drawing file to attach as an external reference, and then click Open.
- 4 In Reference Type, choose how you want to insert the drawing:
  - Attachment – inserts a copy of the drawing and includes any other drawings that are externally referenced within the referenced drawing.
  - Overlay – lays a copy of a drawing over your original drawing; it does not include any nested external references from the externally referenced drawing.

- 5 Make any additional selections.
- 6 Click OK.
- 7 If you marked Specify On-Screen for any items, follow the prompts to attach the external reference.



- A Displays the external reference to attach, or select one from the list.
- B Select to save the folder location of the referenced drawing. If not selected, the referenced drawing must be located in the same folder as the current drawing.
- C Click Attachment to link a drawing, including any of its own external references. Click Overlay to link a drawing, omitting any of its own nested external references.
- D Choose to specify the insertion point in the drawing, or enter x-, y-, and z-coordinates.
- E Choose to specify the scale in the drawing, or enter x-, y-, and z-scale factors.
- F Choose to specify the rotation angle in the drawing, or enter a rotation angle.
- G Click to locate and select a different external reference.






## Viewing the list of external references

You can view a list of the external references that are linked to the current drawing two different ways using the Xref Manager:

- **List View** displays the external references in a list, which allows you to sort the list of references by name, status, size, type, date, or saved path.
- **Tree View** displays a hierarchical representation of the external references and the relationships between them. The tree view shows the level of nesting relationships of the attached external references.


### To view a list of external references

- 1 Do one of the following:
  - Choose Insert > Xref Manager.
  - On the Tools toolbar, click the Xref Manager tool ()
  - Type *xrm* and then press Enter.
- 2 Click List View () or Tree View ()

## Opening external references

From the Xref Manager you can quickly open the source drawing for any external reference. This is especially helpful if you are working with nested external references, which you cannot bind or detach. From the Xref Manager, open the source drawing, make changes, and then save and close the source drawing. When the Xref Manager displays again, simply reload the external reference.

### To open an external reference

- 1 Do one of the following:
  - Choose Insert > Xref Manager.
  - On the Tools toolbar, click the Xref Manager tool ()
  - Type *xrm* and then press Enter.
- 2 Select the external reference to open.
- 3 Click Open.


## Removing external references

Removing external references from the current drawing is easy with the Xref Manager. You can unload an external reference, which keeps some information about the external reference in the current drawing for easy


reloading later, or you can detach the external reference entirely.

When you unload an external reference, you remove it from the current drawing. However, its elements, such as layers and linetypes, remain in the drawing and it is still listed in the Xref Manager. By detaching an external reference you remove it and all of its elements from the current drawing, and it is no longer listed in the Xref Manager.

#### To unload an external reference

- 1 Do one of the following:
  - Choose Insert > Xref Manager.
  - On the Tools toolbar, click the Xref Manager tool ().
  - Type *xrm* and then press Enter.
- 2 Select the external reference to unload.
- 3 Click Unload.

#### To detach an external reference

- 1 Do one of the following:
  - Choose Insert > Xref Manager.
  - On the Tools toolbar, click the Xref Manager tool ().
  - Type *xrm* and then press Enter.
- 2 Select the external reference to detach.
- 3 Click Detach.


**NOTE** Only the external references that are attached directly to the current drawing can be detached; nested external references cannot be detached.

### Reloading external references

When you open or print a drawing, any external references in the drawing are updated automatically. If a drawing is already open and a referenced drawing is modified, you can update the current drawing manually to display the latest version of the referenced drawing.

You may also want to reload an external reference that has been unloaded temporarily.

#### To reload an external reference

- 1 Do one of the following:
  - Choose Insert > Xref Manager.
  - On the Tools toolbar, click the Xref Manager tool ().
  - Type *xrm* and then press Enter.


- 2 Select the external reference to reload.
- 3 Click Reload.

### Changing the path for external references

If the file associated with an external reference is moved to a different directory or renamed, the program displays a message indicating that it cannot load the external reference. You can re-establish the link to the file by doing any of the following:


- Change the path for the external reference.
- Specify additional directories for BtoCAD to search. This is especially helpful, if you have several external references that have moved to a new directory.

#### To change the path for a single external reference

- 1 Do one of the following:
  - Choose Insert > Xref Manager.
  - On the Tools toolbar, click the Xref Manager tool ().
  - Type *xrm* and then press Enter.
- 2 Select the external reference whose path you want to change.
- 3 In Xref Path, do one of the following:
  - Enter a new filename or location.
  - Click Browse to locate and select the referenced drawing. BtoCAD reloads the specified external reference automatically.

**NOTE** You cannot recursively reference a drawing from the same original drawing.

#### To change the search paths for all external references in the drawing

- 1 Do one of the following:
  - Choose Insert > Xref Manager.
  - On the Tools toolbar, click the Xref Manager tool ().
  - Type *xrm* and then press Enter.
- 2 In Additional Xref Search Paths, do one of the following:
  - Enter a new directory and its path. Separate multiple paths with a semicolon, for example, c:\My Drawings; d:\My Drawings\Backup.
  - Click Browse to locate and select a directory.

BtoCAD searches the specified directories; any found external references are reloaded automatically.


## Binding external references to drawings

External references are not part of the drawing. Rather, they are links to an externally referenced file. To provide a copy of a drawing containing external references to someone else, you must also provide all the external reference files. In addition, the person receiving the drawings must either re-create the same paths you used when linking the external references or change the paths for the external references.

To provide a copy of a drawing that contains external references, it is often easier to first bind the external references to the drawing. Binding the external references makes them a permanent part of the drawing, which is similar to inserting a separate drawing as a block.

You can bind external references that are attached directly to the current drawing; you cannot bind nested external references.

### To bind an existing external reference to a drawing

- 1 Do one of the following:
  - Choose Insert > Xref Manager.
  - On the Tools toolbar, click the Xref Manager tool ()
  - Type *xrm* and then press Enter.
- 2 Select the external reference to bind.
- 3 Click Bind.
- 4 Choose one of the following:
  - **Bind** Binds the external reference and creates a unique name for each named entity, such as a layer or block, that is located in the external reference. For example, a layer named Electric in the external reference will be named Xref\$0\$Electric in the current drawing. If the current drawing already has a layer or block with the same name, the name is changed incrementally, for example, Xref\$1\$Electric.
  - **Insert** Binds the external reference, but does not change the names of any named entities in the external reference. For example, a layer named Electric in the external reference will have the same name, Electric, in the current drawing. If the current drawing has a layer or block with the same name, the named entity in the external reference takes on the properties of the named entity in the current drawing.
- 5 Click OK.

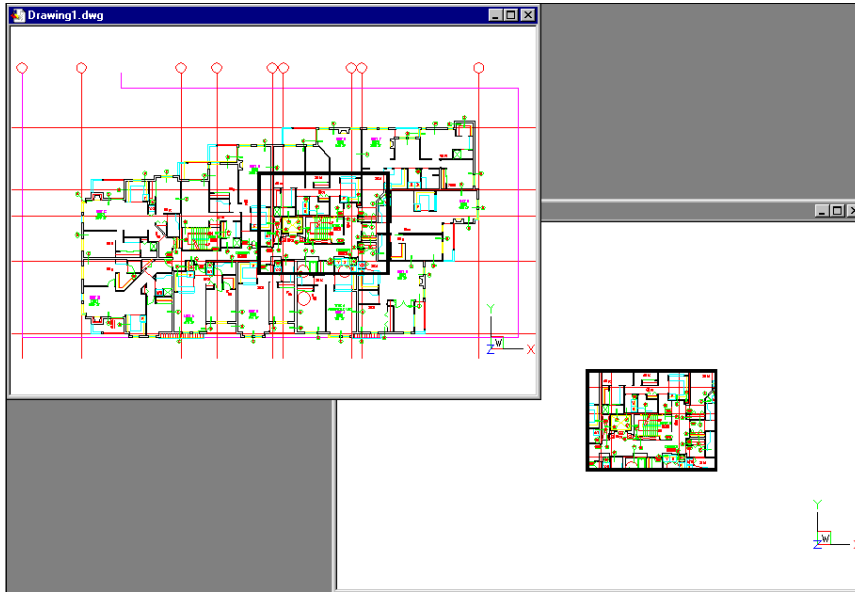
## Clipping external references

When you attach a drawing as an external reference, all of the referenced drawing displays in the current drawing. However, after you attach an external reference, you can define a clipping boundary that determines

which portion of the referenced drawing is visible or hidden.

You can edit, move, or copy clipped external references the same way you modify unclipped external references. The boundary moves with the reference. If an external reference contains nested clipped external references, they also appear clipped in the drawing.

**TIP** *In addition to clipping external references, you can also partially hide blocks using clipping boundaries.*



Example of an external reference clipped using a clipping boundary. The clipping boundary is the rectangle in the top window.

### *Adding a clipping boundary*

When you create a clipping boundary, it affects only the display of the referenced drawing; it does not affect the original referenced drawing or any referenced geometry. The portion of the external reference within the clipping boundary is visible and the remainder of the external reference becomes hidden.

### **To define a rectangular clipping boundary**

- 1 Do one of the following:
  - Choose Modify > Xref Clip.
  - Type *xclip* and then press Enter.
- 2 Select the external references to clip. If desired, you can also select blocks.
- 3 Press Enter.

- 4 Press Enter to create a new clipping boundary.
- 5 If prompted, press Enter to delete any existing boundaries.
- 6 Choose Rectangular.
- 7 Define the first corner of the clipping rectangle.
- 8 Define the second corner of the clipping rectangle.

The selected external references are clipped by the rectangle.

**TIP** You can first select all external references, right-click the selection, and then select *Xref Clip* from the shortcut menu.

#### **To define a clipping boundary using a polyline**

- 1 Draw a polyline where you want to clip external references.
- 2 Do one of the following:
  - Choose Modify > Xref Clip.
  - Type *xclip* and then press Enter.
- 3 Select the external references to clip. If desired, you can also select blocks.
- 4 Press Enter.
- 5 Press Enter to create a new clipping boundary.
- 6 If prompted, press Enter to delete any existing boundaries.
- 7 Choose Select Polyline.
- 8 Select the polyline to use as clipping boundary.

#### ***Turning clipping boundaries on and off***

You can turn xref clipping on or off. When a clipping boundary is turned off, the boundary does not display and the entire external reference is visible, provided that the geometry is on a layer that is on and thawed. When a clipping boundary is turned off, it still exists and can be turned on. However, deleting a clipping boundary is permanent.

#### **To turn clipping boundaries on and off**

- 1 Do one of the following:
  - Choose Modify > Xref Clip.
  - Type *xclip* and then press Enter.
- 2 Select the desired external references.
- 3 Press Enter.
- 4 To turn off clipping boundaries, choose Off. To turn on existing clipping boundaries, choose On.
- 5 Press Enter.

If you are turning off a clipping boundary, click the clipped portion of the external reference to view the previously hidden portion of the referenced drawing.

**TIP** When the `XCLIPFRAME` system variable is on (set to 1), you can select and print the clipping boundary frame.

### *Deleting a clipping boundary*

If you no longer need a clipping boundary for an external reference, you can delete it.


#### **To delete a clipping boundary**

- 1 Do one of the following:
  - Choose Modify > Xref Clip.
  - Type `xclip` and then press Enter.
- 2 Select the desired external references.
- 3 Press Enter.
- 4 Choose Delete, and then press Enter.
- 5 Click the clipped portion of the external reference.

The previously hidden portion of the referenced drawing displays.

## **Attach raster image**

#### **To joint raster image, do one of the following:**

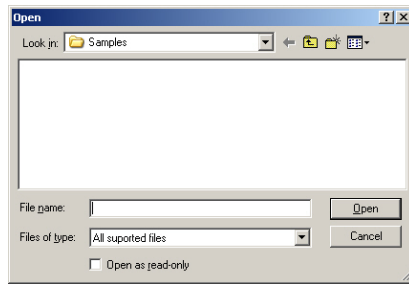
1. Choose insert menu > Attach raster image
2. On image tool bar, click attach raster image 

#### Command function

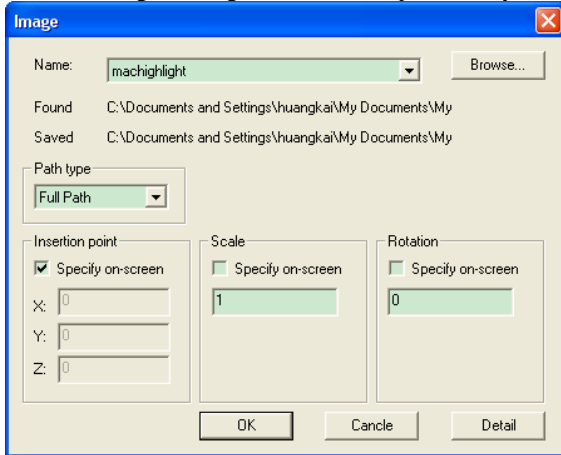
Attach raster image can refer to the image and put image into the image file while as the situation of xref, it is not a actual part of the image file. The image links to the image file through the path. And path can be changed or canceled. It can be attached again to the image like block for times. Any of the insert images has its owned setting for trim, lightness, contrast, fade and transparency.

#### Parameter instruction

After running the IMAGEATTACH command, system will popup a opened box for choosing the right attach image, like picture shown as below:



After choosing the image file, click on Open icon, system will popup another dialogue box.



Name: Identify the attached image

Browse: Open 'image file choosing' box

Found: display the path of chose image file

Saved: display the identified path for chose image file attaching to current image.

Path type: identify one of the three file folder path information and save with the attached image: full path, relative path and no path.

- Full path: appoint complete path for image file. Complete path is to set the complete identify level structure for file folder of xref location. Complete path includes number of local hard disk driver or number of internet server driver. This is the definite choice while lack of flexibility.
- Relative path: appoint relative path for image file.
- No path: only appoint the image file. The image file should be in the same folder as the current drawing file.

Insertion point: appoint the insert point of the selected image. The default setting is "specify on screen", and default insert point will be defined as (0,0).

Specify on screen(X/Y/Z): key in figure in the command bar or locate point on working area.

Scale: specify the scale of the appointed image. If select "specify on screen", one can directly type in the command bar or locate points on the working area. If not, please enter the number of scale. The default scale is 1.



Rotation: specify the rotation angle of the appointed image. If select “specify on screen”, one can directly rotate the object or type in scale in the command bar after quitting the dialog box. If not, one need to key in figure in the sale.

## Image Management

**To open image management, do one of the following:**

1. Choose insert menu > Image Management



2. On image tool bar, click Image Management

### Command Function:

List all image files that had been attached to the current drawing, and view the detailed information of the selected image, such as pixel width, color and resolution. New image file can also be attached and loaded attached image can be detached, reloaded, relocated or unloaded.

### Parameters:

After executing “image”command, system will pop up a image management dialog box, shown as below:

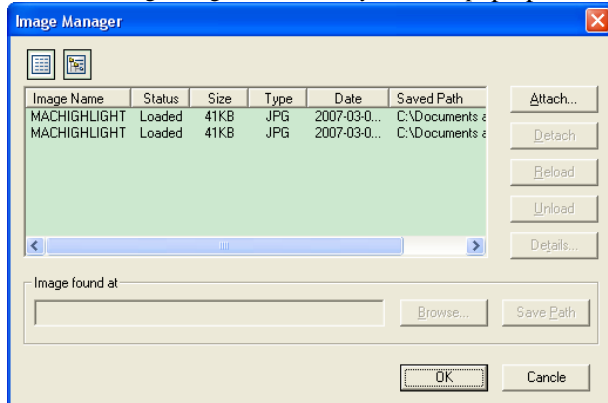


Image listings: show all the file names of the attached images. Click the two buttons on the top left corner to switch between list and tree category.

Attach: open “attach image” dialog box.

Detach: delete the defined image from the drawing database, and delete all related image object from the drawing and its display.

Reload: load the newest version of the image, or reload image that has been unloaded. Reload can not control whether to display the image or not, but it can assure the displayed image is the newest.

Unload: unload image data from the working CPU, but it do not delete the image object from the drawing. It is recommended that user unloads images needn't to be edited to improve the performance. The unloaded image can

not be displayed or printed.

Details: open the “image file details” dialog box to show the image name, save path, time of creation, file size, file type, color, pixel width and height, resolution, default size and preview image.

Image found at: display the path of the selected image file. If multiple images are selected, this bar will be blank.

Browse: open “select image file” dialog box. The selected path will display at “image found at” bar.

Save path: save new path information. If you don’t click “save path” after editing, you will go to the original image path next time you want to load image.

### **Tips:**

#### **List**

List the defined image attached to the drawing. No matter how many time you attach(insert) the image, the name of each image is only display once. You can list the image according to its name, status (loaded, unloaded or not found), type(like tiff), date, size or save path. In default situation, image will be listed according to the alphabet.

- Press SHIFT or CTRL to select multiple images
- Click the specific row title to list accordingly.
- To change the width of a row, please drag the separate lines between adjacent rows.

#### **Tree**

Display layers of the defined image and outer reference. The top layer of the tree displays show the name of the image that attached to the drawing, the reference image attached and that the drawing of outer reference. The name attached for the attached outer reference is display in the lower layer of drawing in the tree.

- To attach duplicate of the attached image, please select it and click “attach”.
- Tree only show the image name (not file name), and show only once no matter how many time it has been attached.