

Toolbox LT 2009



Getting Started Guide Part 2

Contents

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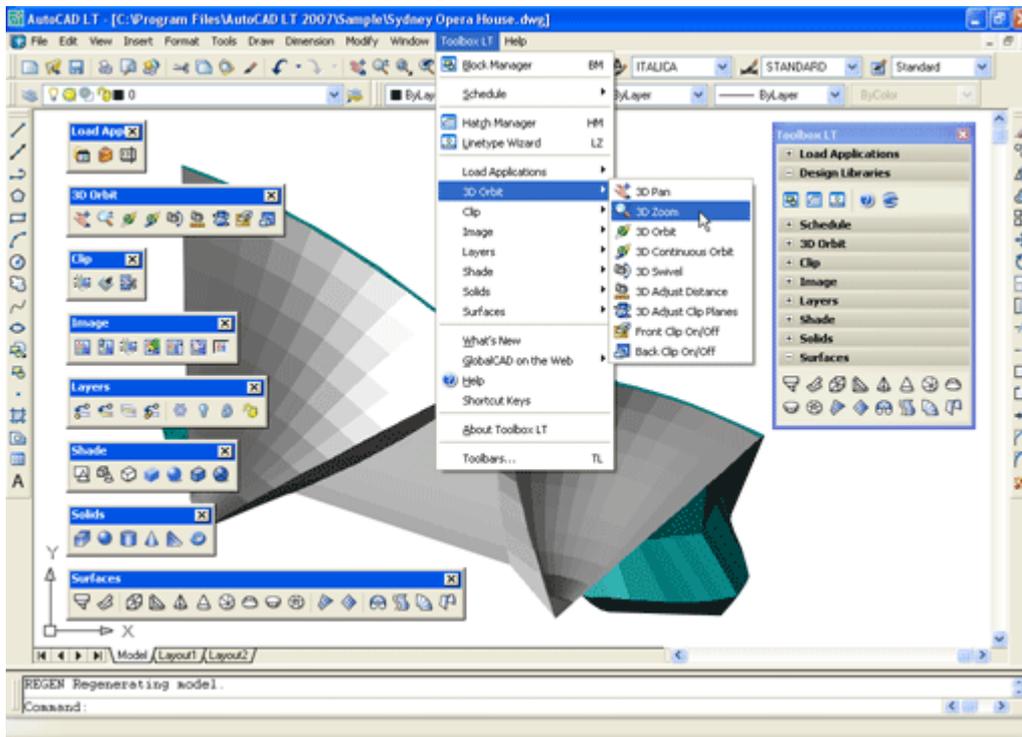
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GlobalCAD

Introduction

Welcome to Toolbox LT 2009, the most cost effective solution for extending your design and drafting capabilities with AutoCAD LT. Now you can design in 3D with solids and surface modelling, generate realistic shaded renders, import and manipulate graphic images and load and run Lisp utilities and ARX applications.



This user guide provides step-by-step instructions on getting started with the 'Design Libraries' and 'Schedule' bonus tools found within Toolbox LT.

Please refer to the Getting Started Guide Part 1 for the product's complete feature list, available from www.globalcad.com/support/documentation.htm

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Toolbox LT 2009 is compatible with AutoCAD LT 2000, 2000i, 2002, 2004, 2005, 2006, 2007, 2008 and 2009. The software can also be used in conjunction with other AutoCAD LT third-party applications. Multi-user licensing options are available and provide support for all major network platforms including Novell Netware and Windows NT Server.

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Creating and Managing 2D and 3D Symbol Libraries

Block Manager allows AutoCAD drawing blocks and details to be effectively stored within a powerful database, providing for a more controlled and managed working environment. This form of control is essential for the efficient and productive use of AutoCAD.

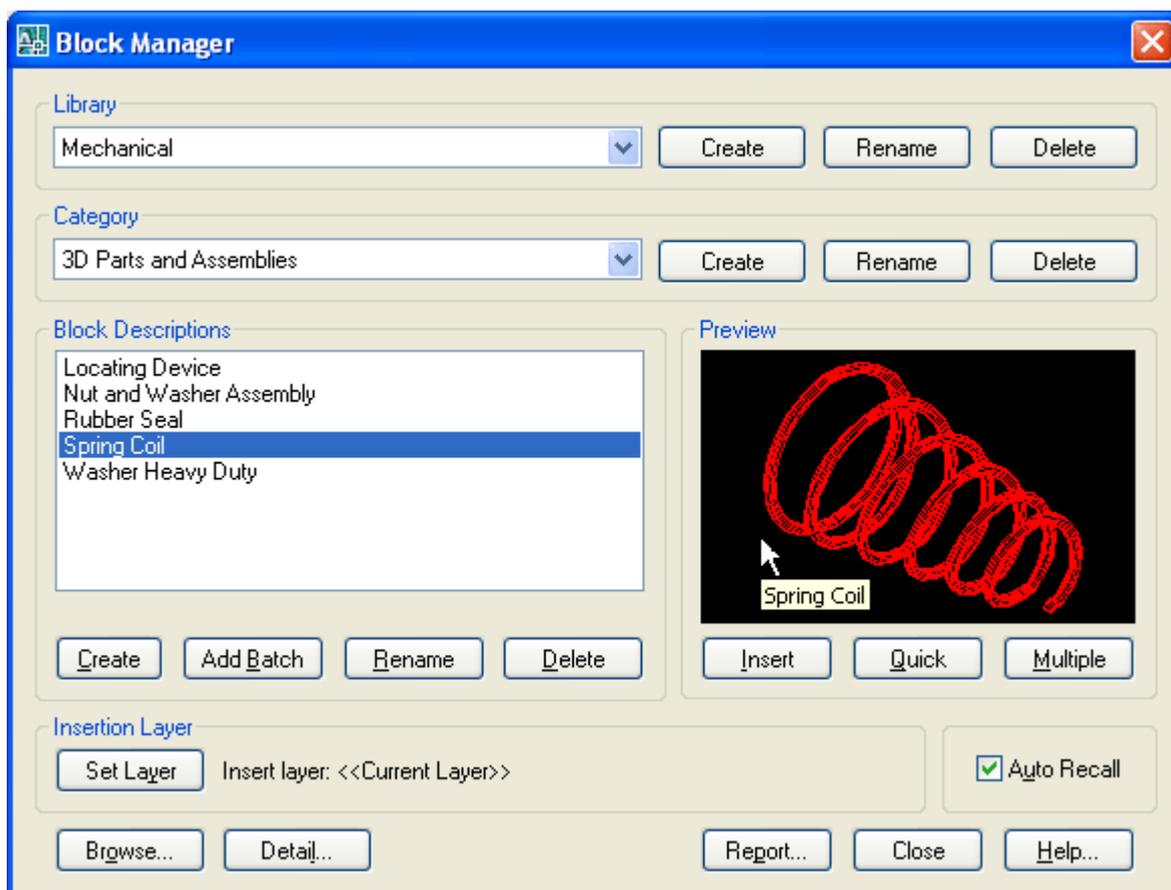
Now your entire collection of blocks can be sensibly organised within distinct libraries with advanced viewing capabilities. This means that blocks can be identified fast and are accessible right when you need them. Management tools ensure the database is always up to date, with the ability to view and print block details and other library information. Blocks can be contained on local drives or on a network.

With Block Manager you can:

- Create categories containing 2D, 3D and attributed blocks.
- Add multiple blocks in seconds using the unique batch-processing tools.
- Build 3D libraries with photo-realistic rendered thumbnails.
- View and compare blocks fast within libraries.
- Avoid unnecessary duplication of blocks with full network compatibility.
- Secure your work with full library password protection.
- Effectively manage and keep track of all your AutoCAD blocks.

Initial Steps

Block Manager's main dialog displays the contents of the blocks database. The current library and category is shown at the top with the relevant block descriptions listed below. Displayed right is the preview image for the block currently highlighted and below is shown the current layer for insertion. Libraries are stored within the GlobalCAD...\Libs folder.



Creating a working library and category is the first step in adding your blocks to the database. The library can be assigned a full and meaningful description, such as the working discipline e.g. 'architectural symbols' or maybe a project name.

Note when using the Add Batch feature to add multiple blocks: To successfully add multiple blocks under Block Manager, each block must be capable of being independently opened within the current AutoCAD session. This means that the associated menu, font files and any other dependent files are present and the block(s) open without error.

Creating a Library

Select the **Create** button under the Library heading from the main display.

- 1 Enter the proposed library description within the text box.
- 2 Next, enter your signature details. Note the default shown is the user's login name.
- 3 Choose **OK** to accept the descriptions and proceed.

To rename an existing library, pick the chosen description from the Library list box and click on the **Rename** button shown right. Next, enter the proposed new library heading within the text box and choose **OK** to accept this description.

Similarly, to delete an existing library, pick the chosen description from the Library list box and click on the **Delete** button shown right. Next, to confirm your wish to remove this library, click on the **OK** button shown or **Cancel** to exit.

Note that when a library is deleted, its corresponding contents (including blocks) are also deleted.

Creating a Category

Select the **Add** button under the Category heading from the main display.

- 1 Enter the proposed category description within the text box.
- 2 Choose **OK** to accept the description and proceed.

To rename an existing category, pick the chosen description from the Category list box and click on the **Rename** button shown right. Next, enter the proposed new category heading within the text box. Choose **OK** to accept this description.

Similarly, to delete an existing category, pick the chosen description from the Category list box and click on the **Delete** button shown right. Next, to confirm your wish to remove this category, click on the **OK** button shown or **Cancel** to exit.

Note that when a category is deleted, its corresponding contents (including blocks) are also deleted.

Creating a Block

Blocks can be added individually to the database, created from existing drawing objects in the normal way.

- 1 Open the drawing or block that contains the details to be added.
- 2 Select the **Create** button under the Block Descriptions heading from the main display.
- 3 Select the objects that will make up the new block.
- 4 Specify the insertion point for the block and Block Manager will automatically zoom in around the selected objects to create the preview image.
- 5 Enter the proposed description for the block within the text box and click on the **OK** button.

Adding Multiple Blocks

Blocks can also be added in multiple selections with the unique and powerful Add Batch feature. Use this method to create a complete category listing from a set of blocks stored anywhere on disk (local and network drives).

- 1 Select the **Add Batch** button under Block Descriptions from the main display.
- 2 From the Add Batch dialog the following options are available, according to preference:
 - Include sub-folders
 - Create categories from sub-folder names
 - Ignore duplicate-named blocks
 - Ignore illegally-named blocks

The Advanced section of the dialog also provides a range of options for creating 3D wireframe or photo-realistic libraries. These are available by picking the **Create 3D Library** tick box:

- Wireframe (hidden lines removed)
- Render Photo Real

- Define Viewpoint
- Utilise existing Bitmap files

Selecting the *Wireframe (hidden lines removed)* option will create wireframe preview images of all blocks (containing 3D objects) encountered during the Add Batch operation. The *Render Photo Real* option will create photo-realistic or realistically shaded images using the AutoCAD Render command. In both cases, the viewpoint location used to display each block will be the value specified within the *Define Viewpoint* text box.

Note in order to include materials (optional) for use during the *Render Photo Real* process: From the standard AutoCAD menu, select Tools, Preferences, Files and amend the Texture Maps Search Path to include the folder where the relevant drawings and associated material TGA files are located.

Choose the *Utilise existing Bitmap files* option if you prefer the Add Batch operation to use existing, manually created, Bitmap images. This enables you to define each 3D object's unique viewpoint and therefore obtain the very best rendered image. The BMP extension files must be located within the same folder, and share the same filename, as the associated DWG blocks. Use the standard AutoCAD Render command to create the Bitmaps, with the following settings:

Set the Render Type to Photo Real and the File Type to BMP.

The User Defined Size (image size) is X=216 and Y=137 with the default Aspect Ratio of 1.0.

The recommended Colour setting is 8 Bits (256 colours).

Note that, due to certain restrictions with the standard AutoCAD Render command, the *Render Photo Real* automated feature is only available under AutoCAD 14. In order to create rendered libraries with AutoCAD 2000, 2000i or 2002, you will need to use the *Utilise existing Bitmap files* option explained above. If you wish to build 3D libraries under AutoCAD LT, use the Wireframe option.

- 3 Next, click on the **Browse** button to select the folder containing the relevant blocks. Note, if the *Include sub-folders* option has been specified, blocks contained within any sub-folders will also be added. In addition, selecting the *Create categories from sub-folder names* option means that new categories will be created, containing the contents of each sub-folder.

You can also create photo-realistic rendered preview images (thumbnails) of your 3D blocks by selecting the *Render preview* option and defining the viewpoint setting to be applied.

- 4 Returning to the main Add Batch dialog, click on **OK** to proceed.

Please wait while the blocks are now processed and added to the database. Once this operation is complete, the main Block Manager dialog displays the current category and new block entries.

To rename an existing block description, pick the chosen description from the Block Descriptions box and click on the **Rename** button shown below. Next, enter the proposed new block title within the text box and choose **OK** to accept this description.

Similarly, to delete an existing block description, pick the chosen description from the Block Descriptions box and click on the **Delete** button shown below. Next, to confirm your wish to remove this description, click on the **OK** button shown or **Cancel** to exit.

Inserting Blocks

Blocks can be inserted by clicking on the **Insert**, **Quick** or **Multiple** buttons.

The **Insert** button functions in the same way as AutoCAD's standard block insert command i.e. it prompts for the block's insertion point, axis and rotation angle. **Quick** requests only the block's insertion point and rotation angle. **Multiple** requests the same parameters as the **Quick** button, but once specified, multiple copies of the block can be inserted in succession.

Select the block for insertion from the Block Descriptions text box, click on one of the insert buttons described above and follow the prompts.

Setting the Insertion Layer

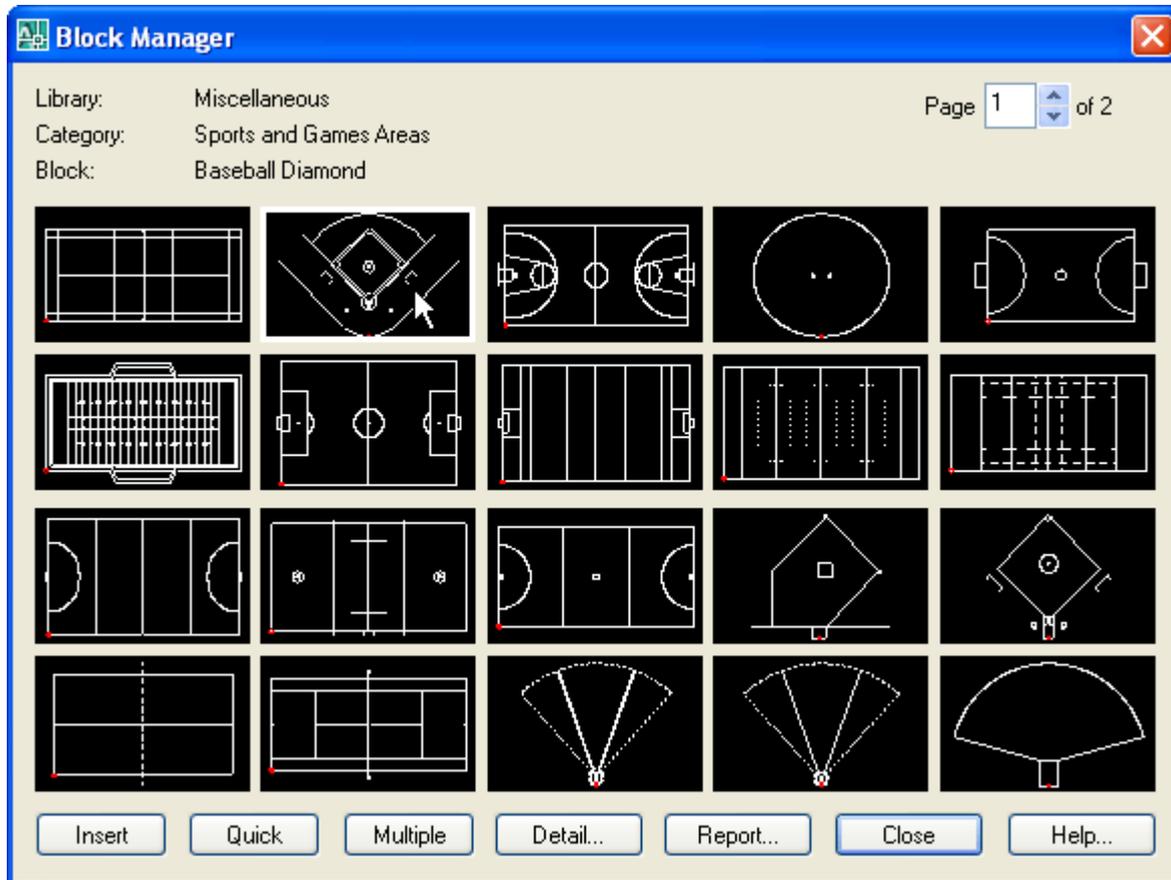
The layer upon which blocks will be inserted is shown near the bottom of the main dialog display, under the heading Insertion Layer. The default insertion layer is 0. In order to change this setting:

- 1 Select the **Set Layer** button. The existing drawing layers available are displayed in a text box.
- 2 Select the desired layer for block insertion and click on the **OK** button.

Note, to create a new layer, enter the new name in the Create New Layer text box below and click on the **OK** button. This will be created and made the current layer.

Browsing Libraries

Sometimes it is useful to view and compare blocks simultaneously, usually in order to make an informed choice from similar blocks within a library. Using Block Manager's Browse feature you can view blocks in this way, rapidly browsing an entire library if desired.



1 Select the **Browse** button from the main display.

The page scroll box is located in the top right-hand corner of the Browse dialog.

2 Click on the scroll box arrows to change the page number to view.

Current Library Details

This feature displays information concerning the current library.

Select the **Detail** button from the main display and click on **OK** when finished.

Information displayed includes the library filename and folder location, the library description, creator, current category and total number of blocks present within this category.

Library Protection

Once created, you may wish to protect specific libraries from accidental modification or deletion.

1 Select the **Detail** button from the main display.

2 Pick the **Library Protected** tick box at the base of the dialog display.

3 Enter a unique password and confirm this in the text box below.

All library maintenance functions (i.e. Add, Delete and Rename) will now be disabled until the tick box is de-selected. Note that the original password specified must be re-entered in order to unprotect the library. It is recommended that you make a note of this and store it in a safe place.

Viewing the Block Report

Click on the **Report** button to display the full list of blocks stored under the current category.

Blocks are listed in alphanumerical order.

Choose the Print Report button to obtain hardcopy records.

Working with Attribute Groups

An attribute is a label or tag that attaches data to a block. Examples of data that might be contained in an attribute are part numbers, prices, comments, and owners' names. The tag is equivalent to a column name in a database table. Attribute information extracted from a drawing can be used in a spreadsheet or database to produce a parts list or a bill of materials.

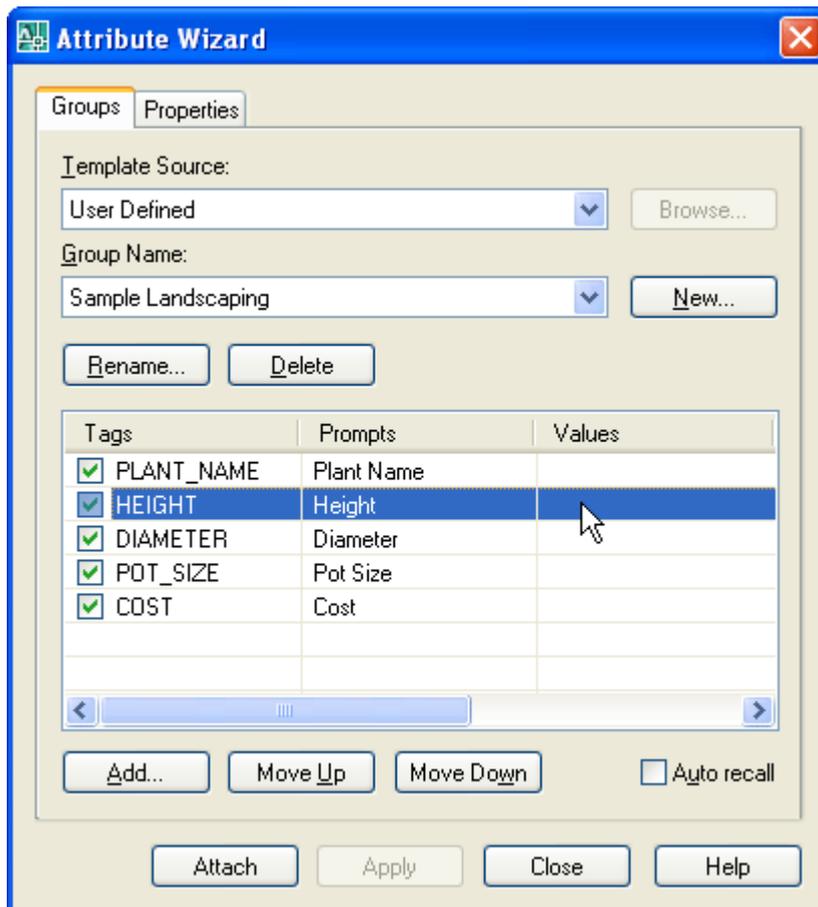
With Attribute Wizard, you can attach a group of attributes to drawing entities or blocks – all at the click of a button! Forget using the cumbersome AutoCAD ATTDEF and WBLOCK commands, now you can create and save attribute groups as templates for all your projects. Attribute Wizard provides full control over the properties of each attribute and lets you specify the order in which you wish them to appear.

With Attribute Wizard you can:

- Instantly attach a group of attributes to entities and blocks.
- View and compare attribute groups quickly and easily within libraries.
- Review and edit all tags, prompts and values in a single report.
- Control the display order of attributes within a group.
- Manage individual attribute properties.
- Share group attribute templates with full network compatibility.

Initial Steps

Attribute Wizard's main dialog displays the contents of the attribute group database. The **Groups** tab displays the current group name, followed by the associated attribute tags, prompts and values in the report view below. The **Properties** tab defines the standard AutoCAD properties and text options for the current group. Each group definition is stored within the GlobalCAD\...\Attributes folder.



Creating a working attribute group is the first step in adding your attributes to the database. The group can be assigned a full and meaningful description, such as the working discipline e.g. 'architectural finishes' or maybe a project name.

Creating an Attribute Group

From the Groups tab, select the **New** button next to the Group Name drop-down list.

- 1 Enter the proposed group name within the text box and click on the **OK** button.
- 2 Next, to add an individual attribute to the group, click the **Add** button below the report view.
- 3 In the Add Attribute dialog, enter the attribute tag, prompt and value (optional). You can also set the options for attribute mode i.e. Invisible, Constant, Verify and Preset.

Notes

- Attribute tag values are forced to uppercase and spaces are not permitted.
- Attribute modes: Invisible specifies that attribute values are not displayed or printed when you insert the block (ATTDISP overrides Invisible mode). Constant gives attributes a fixed value for block insertions. Verify prompts you to verify that the attribute value is correct when you insert the block. Preset sets the attribute to its default value when you insert a block containing a preset attribute.
- To edit an existing attribute's tag, prompt, value and/or mode see **Help** topic Editing Group Attributes.
- If an attribute group is deleted, its corresponding contents (attributes) are also deleted.

Editing Group Attributes

Highlight the attribute for editing in the report view and double-click it to display the Edit Attribute dialog.

Attributes can also be re-positioned within an attribute group by using the **Move Up** and **Move Down** buttons. This will affect the prompt sequence when the attribute group is next attached to a block.

To edit the current attribute group's standard AutoCAD properties and/or text options, select the **Properties** tab.

To delete an attribute from the current group, highlight the attribute and press the **Del** key.

Attaching Groups to Blocks

Attribute groups can be attached to a set of entities or an existing block.

- 1 Choose the relevant group and click on the **Attach** button.
- 2 Select the objects that will make up the new block.
- 3 Specify the insertion point for the first attribute within the group and, if not already defined, the text height.
- 4 Specify the insertion point for the next attribute or press **Return** to position it below the last attribute.
- 5 Repeat point 4 until all attributes are correctly positioned.
- 6 Specify the insertion point for the block.
- 7 Next, enter a description in the Create New Block dialog and click on the **OK** button.

Note the standard AutoCAD Edit Attributes dialog will then prompt for any vacant attribute values (optional).

Generating Schedules and Bill of Materials

The tools in GlobalCAD Schedule let you query drawings for block attribute data and save it to a variety of file formats. Unique formatting options mean you can pre-set the style for each spreadsheet and export only the data that's critical to your needs.

With GlobalCAD Schedule you can:

- Query block attribute values within multiple drawings.
- Process Xrefs and nested blocks.
- Set spreadsheet styles with unique formatting options.
- Merge tabled items based on user-defined criteria.
- Quantify blocks for Bill of Materials.
- Share block attribute templates with full network compatibility.
- Preview schedules fast with an intuitive graphical interface.
- Export data to Microsoft Excel, Access, CSV, XML and TXT.

Initial Steps

GlobalCAD Schedule allows the easy extraction of block attribute data to Microsoft Excel, Access and other formats. You can attach helpful alias names to blocks and attributes and publish data from multiple drawings and Xref attachments. In addition, you can save templates of selected blocks, attributes, and alias information for reuse within other drawings.

The advanced GUI interface employs tabs so you can easily jump to different stages in the export process or simply query drawings for attribute data at any time. Select the **Schedule** option from the left places bar to display the tabs.

Selecting Drawings

The Drawing List on the **Drawings** tab displays the drawings to query for block attribute data. The current drawing is pre-selected by default.

From the **Select Drawings** group box, choose one of the following methods to extract the required block information:

- **Select Objects.** Select this option followed by the pick button to create a selection set of blocks from the current drawing. When done, **right-click** the mouse or press **Enter** to return to Drawings tab.
- **Current Drawing.** Pre-selects all blocks defined in the current drawing.
- **Select Drawings.** Select this option followed by the pick button to process all blocks in one or more external drawings files. The Select Drawings dialog permits the selection of individual drawing files or multiple selections by using **Shift** or **Ctrl**.

From the **Drawing Options** group box, choose from the following preferences:

- **Include Xrefs.** The current Drawing List may contain drawings for extraction that have Xrefs (external references) attached. With this option enabled you can also extract block information from these associated Xrefs.
- **Include nested blocks.** Permits the extraction of data from nested blocks (blocks within blocks).
- **Include frozen layers.** Permits the extraction of block data located on layers that are frozen.
- **Current space only.** Extracts block data from the current space only (can be model space or paper space).

Notes

- Drawing List entries can be sorted into ascending or descending by simply clicking the desired column heading. This will display a small direction arrow indicating the sort direction. To reverse the sort order, click the column heading again.
- Clicking the **Apply** button will apply any changes you've made, such as Drawing Options, without closing the dialog.
- For increased performance, purge all drawings before exporting data.

Selecting Blocks and Attributes

The **Attributes** tab block name view displays the block names selected for extraction. Clicking on a block name displays the associated attribute names and values in the attributes view shown opposite.

You can check/uncheck blocks and attribute values depending on the data you want to export to file. See also notes on the **General** tab under **Formatting Options** help topic for further options.

Aliases can be assigned to block and/or attribute names by clicking in the relevant **Alias** column location in the **Block Name** or **Attributes** view and entering the chosen alias name.

- **Working with Block Attribute Templates**

You can save and retrieve template files containing block attribute settings and apply them to the current drawing selection. For example, you may want to automatically de-select blocks or attributes with a particular name or value. Templates can also store pre-defined aliases for block and/or attribute names.

To save a template based on the current Attribute tab state, select the **Templates** button, followed by **Save Template**. Save the template file to a suitable location. Templates are saved with the **.gst** file name extension.

To open a template and apply it to the current drawing selection, select the **Templates** button, followed by **Open Template** and double-click the template file name to open it.

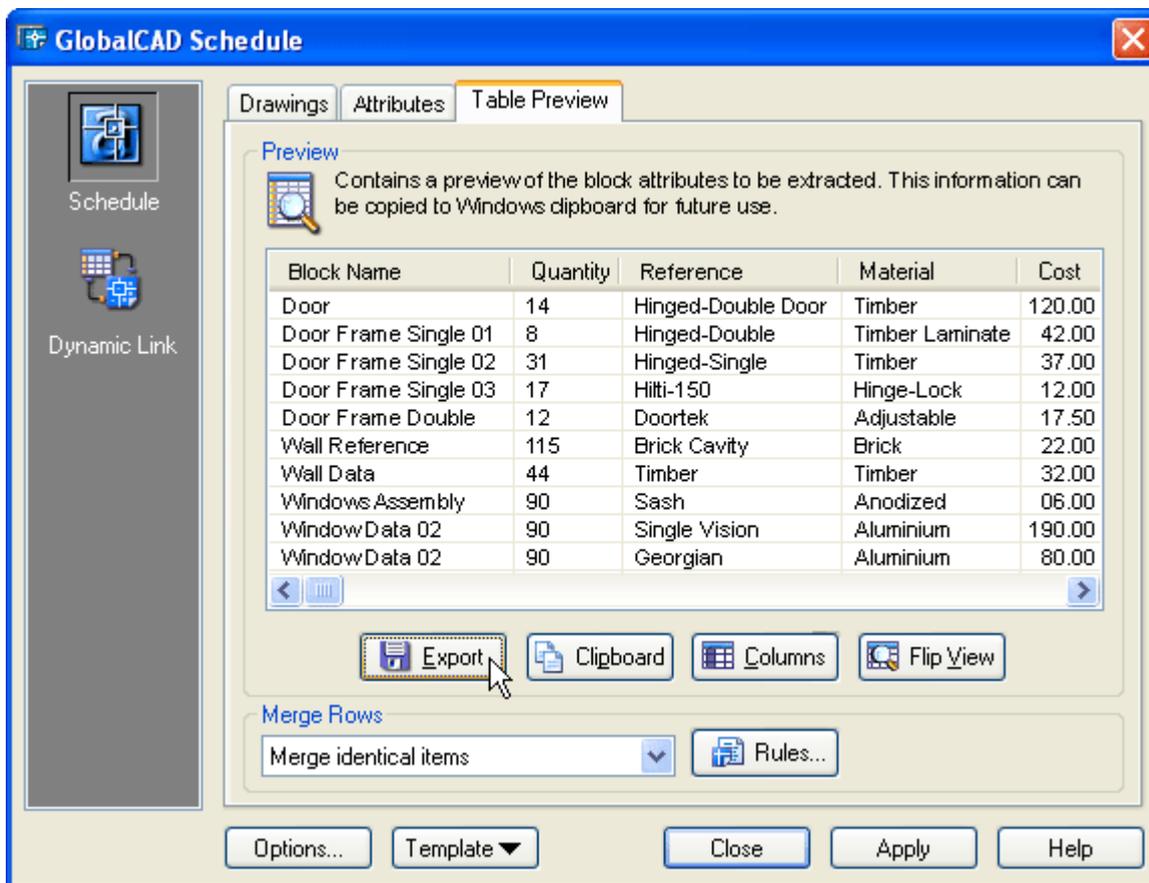
Notes

- **Block Name** and **Attribute** entries can be sorted into ascending or descending by simply clicking the desired column heading. This will display a small direction arrow indicating the sort direction. To reverse the sort order, click the column heading again.
- Clicking the **Apply** button will apply any changes you've made to the **Attributes** tab layout without closing the dialog.
- The current drawing units determine the number of decimal places used in the display of standard AutoCAD properties for a block, e.g. the X insertion point.

Exporting the Data

The **Table Preview** tab contains a preview of the blocks and block attributes to be exported to file. The **Flip View** feature provides 2 alternative views for the display of information and this affects how the data will be exported to file:

- **Landscape**. The Block Name column displays the names of the blocks selected for attribute extraction. Other columns display names of attributes associated with each block. The rows display the values for each attribute. Landscape is the default view.
- **Portrait**. The Block Name column displays the names of blocks selected for attribute extraction. The Attribute column displays the name of the attribute. The Attribute Value column displays the value of the attribute.



Clicking the **Clipboard** button copies all or selected portions of the table to the Windows clipboard.

Select the **Columns** button if you want to re-arrange the table columns into a particular order before exporting. The column view permits the selection of individual column headings or multiple selections by using **Shift** or **Ctrl**.

The **Merge Rows** group box contains a drop-down list of rules that define how rows within the table are merged, depending on user-defined criteria. Clicking the **Rules** button displays the Merge Rule Manager dialog – from here you can select an existing rule or create a completely new one based on your own personal preference.

When you are ready to save the data to file, click the **Export** button. From the **Export Table** dialog, choose the file format from those listed and enter the filename followed by **Save**. Depending on the amount of drawing data, this may take several moments.

- **Working with Multiple Drawings**

GlobalCAD Schedule renames blocks of the same name but that have different attributes that are found in multiple drawings or Xrefs. For each duplicate-named block after the first instance, with different attributes, GlobalCAD Schedule renames the block by appending to the block name a tilde character (~) and the path name and file in which the block was found.

For example, a block named WINDOW is in both C:\Drawings\Office Layout1.dwg and C:\Drawings\Office Layout2.dwg, but the block has different attributes in each file. Using GlobalCAD Schedule, the instance of WINDOW in the first file is shown with the block name WINDOW. The instance of WINDOW in the second file is shown with the block name WINDOW~C:\Drawings\Office Layout2.dwg.

Notes

- **Block Name** and **Attribute** entries can be sorted into ascending or descending by simply clicking the desired column heading. This will display a small direction arrow indicating the sort direction. To reverse the sort order, click the column heading again.
- Clicking the **Apply** button will apply any changes you've made to the **Table Preview** tab layout i.e. switching the layout with **Flip View**, without closing the dialog.

Formatting Options

Unique formatting options mean you can pre-set the style for each spreadsheet and export only the data that's critical to your needs.

From the main GlobalCAD Schedule display, select the **Options** button followed by the **General** tab. From here you can choose which standard AutoCAD property fields you want to include in exported files.

Pick the **Export quantity** option (set by default) if you want to include a quantity value with your exported data. This is useful to create a bill of materials and introduces a **Quantity** column listing block quantities, positioned to the right of the Block Name column. De-selecting this option means that all blocks selected for export will be exported to file independently of each other with a unique spreadsheet entry.

GlobalCAD Schedule allows for the export of block attribute data to the following file formats:

- **Microsoft Excel Workbook (XLS)**
- **Microsoft Access Database (MDB)**
- **Comma Delimited (CSV)**
- **Tab Delimited (TXT)**
- **Extensible Markup Language (XML)**

Note the file formats available depend on the applications currently installed. Microsoft Excel and/or Microsoft Access must be installed in order for the XLS and/or MDB file formats to be available. The CSV, TXT and XML formats are always available.

From the main GlobalCAD Schedule display, select the **Options** button. Format tabs are available for each file format and include a number of pre-formatting options:

- **Microsoft Excel Workbook (XLS)**

One block definition per sheet. Specifies that each individual block definition will be placed on a separate sheet.

Single sheet. Specifies that all block definitions will be placed on a single sheet.

Export to odd lines. Block attribute data is exported to odd lines only.

Start row. Determines the start row for exported data, when the **Single sheet** option is selected.

Include header row. Displays a header row with titles for each column.

Display header bold. Displays the header row bold.

Apply autofit to columns. Makes all column widths fit the contents of the exported data.

Autorun Microsoft Excel when done. Runs Excel and opens the spreadsheet when data is exported to the XLS format.

- **Microsoft Access Database (MDB)**

One block definition per table. Specifies that each individual block definition will be placed in a separate table.

Single table. Specifies that all block definitions will be placed in a single table.

Autorun Microsoft Access when done. Runs Access and opens the database when data is exported to the MDB format.

- **Comma Delimited (CSV)**

Include header row. Displays a header row with titles for each column.

Open CSV file when done. Opens the CSV file with the system's default CSV editor when data is exported.

- **Tab Delimited (TXT)**

Include header row. Displays a header row with titles for each column.

Open TXT file when done. Opens the TXT file with the system's default TXT editor when data is exported.

Notes

- The delimiter used in the comma-separated file format (CSV) is based on the locale – the list separator for the locale separates the exported data.
- XML is a standard, simple, self-describing way of encoding both text and data so that content can be processed with relatively little human intervention and exchanged across diverse hardware, operating system and applications.
- For attribute names that will be assigned as field names in files exported to Microsoft Access, GlobalCAD Schedule substitutes an underscore character (_) for the characters shown in the following table:

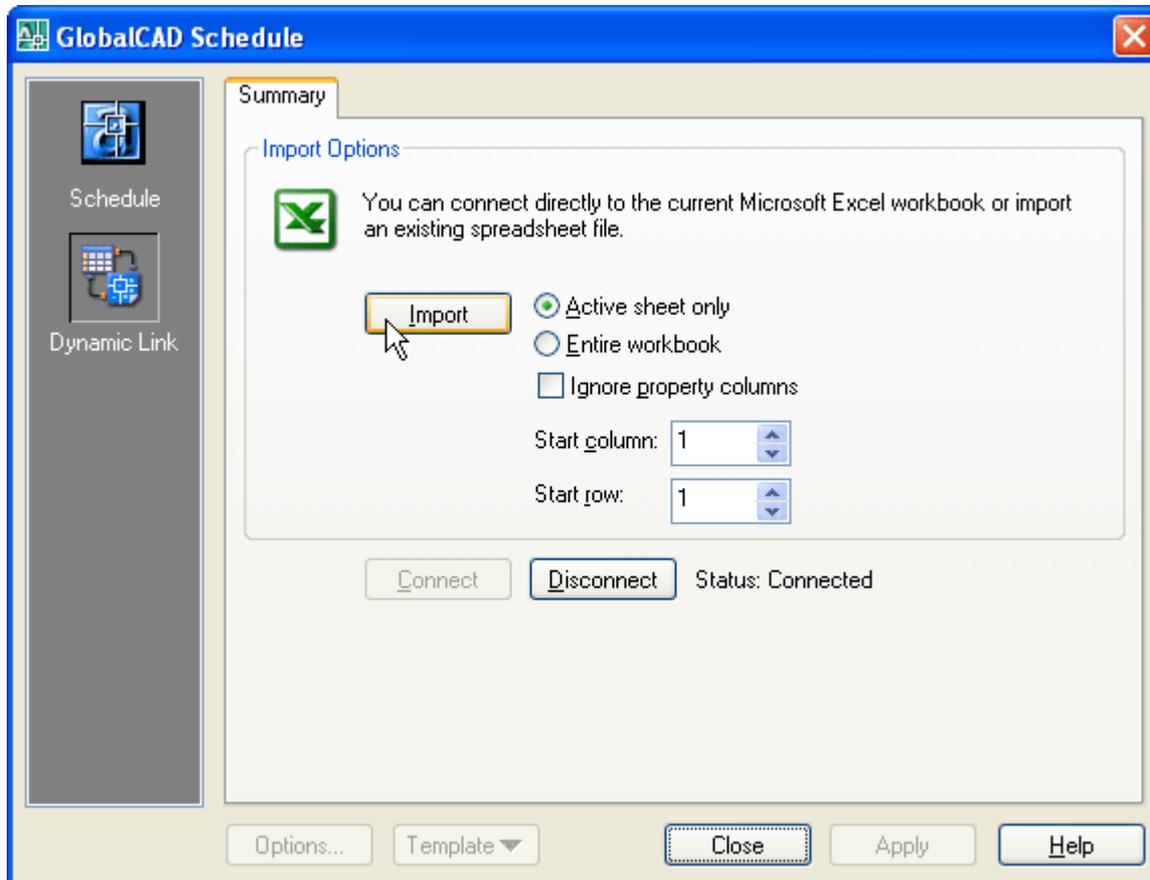
Restricted Characters in Microsoft Access Fields

Character	Character Name
.	Period
!	Exclamation mark
[Left square bracket
]	Right square bracket

For example, attribute tag names that appear as column headings will be field names in Microsoft Access and any instance of the restricted characters will be replaced by an underscore.

Linking MS Excel and AutoCAD

Working with cost estimates, schedules and bill of materials are a critical component of any project. Having created them with GlobalCAD Schedule, the challenge is to ensure they remain accurate, up-to-date and synchronized with your drawing data. With integrated support for Microsoft Excel, Dynamic Link provides the effective and reliable solution to all of these issues.



The tools in Dynamic Link enable you to bi-directionally link AutoCAD drawings with Microsoft Excel – simply edit the Excel spreadsheet and your drawing updates automatically!

With Dynamic Link you can:

- Hot-link AutoCAD drawings with Excel spreadsheets.
- Dynamically update block attribute values.
- Modify the standard AutoCAD properties of blocks.
- Connect 'live' to spreadsheet or import from spreadsheet file.
- Share the technology with full network support.
- Work with Microsoft Excel 97, 2000, XP and 2003.

Initial Steps

Dynamic Link analyzes the cells from the current Excel spreadsheet and dynamically updates the associated 'live' AutoCAD drawing with any changes as they occur.

Each block, with its own attribute values, is linked to a unique spreadsheet row by using the block's handle value (a unique hexadecimal value generated by AutoCAD at the time of block creation). The first cell in each row, located under the **Block Name** column, is dedicated to the handle value of the block to which it is linked.

Select the **Dynamic Link** option from the GlobalCAD Schedule left places bar.

Working with Microsoft Excel

Both of the methods described below assume you have already used **GlobalCAD Schedule** to export block attribute data, relating to the current drawing, to the Microsoft Excel **XLS** format.

- **Dynamic Update**

This option creates a connection (link) between the current AutoCAD drawing and Microsoft Excel spreadsheet.

The connection 'captures' the **SheetChange** event, issued by Excel whenever data in the spreadsheet is changed. When a change is detected, and the spreadsheet directly relates to the current drawing, Dynamic Link finds the index of changed row and column and performs the change in AutoCAD.

- 1 From the **Dynamic Link** display, select the **Connect** button and open the relevant Excel spreadsheet.

The spreadsheet is displayed in an Excel window in front of the current AutoCAD drawing.

- 2 Edit the Excel spreadsheet in the normal way, saving changes as necessary.

Individual block attribute values and selected standard AutoCAD properties (see table below) can be edited. Clicking outside the current cell or pressing **Enter** performs a 'live' visible update of the associated block properties in AutoCAD.

Warning! Do not edit values in the **Block Name** column as these cells contain unique block reference keys necessary for communication with AutoCAD.

Standard AutoCAD Properties (Editable)

X insertion point	X scale	X extrude
Y insertion point	Y scale	Y extrude
Z insertion point	Z scale	Z extrude
Orient		

- **Update from File**

This option imports an existing Microsoft Excel spreadsheet and automatically applies any updates to the current AutoCAD drawing.

Dynamic Link reads the cells from the selected spreadsheet and identifies the associated blocks in AutoCAD. Any changes to block attribute values and/or standard AutoCAD properties are then applied to the current drawing. If an attribute value in the Excel spreadsheet is not present in the AutoCAD drawing it is ignored.

- 1 From the **Dynamic Link** display, select the **Import** button and specify the Excel spreadsheet to open.

The spreadsheet is imported and the current drawing updated. Note the spreadsheet must share one or more unique block reference keys with the drawing for it to be correctly identified and accepted.

- 2 Close the **Dynamic Link** display and **Save** the current drawing when you are satisfied with the spreadsheet updates.

The following options are available when importing the Excel spreadsheet:

Active sheet only. Only the active Excel sheet is referred to. All other sheets are ignored.

Entire workbook. All Excel sheets are referred to when updating the drawing.

Ignore property columns. Any changes to standard AutoCAD properties are ignored.

Start column. Text box displays the starting column number (range 1-255).

Start row. Text box displays the starting row number (range 1-65536).

Creating and Managing Hatch Pattern Libraries

Hatch Manager provides the solution to all your hatch creation and management needs. You can build entire libraries from scratch, add existing PAT files in seconds or create your own unique patterns from existing drawing objects. The software also links directly into AutoCAD's Boundary Hatch (Bhatch) dialog, ensuring all your patterns remain fully featured with options such as associative and non-associative.

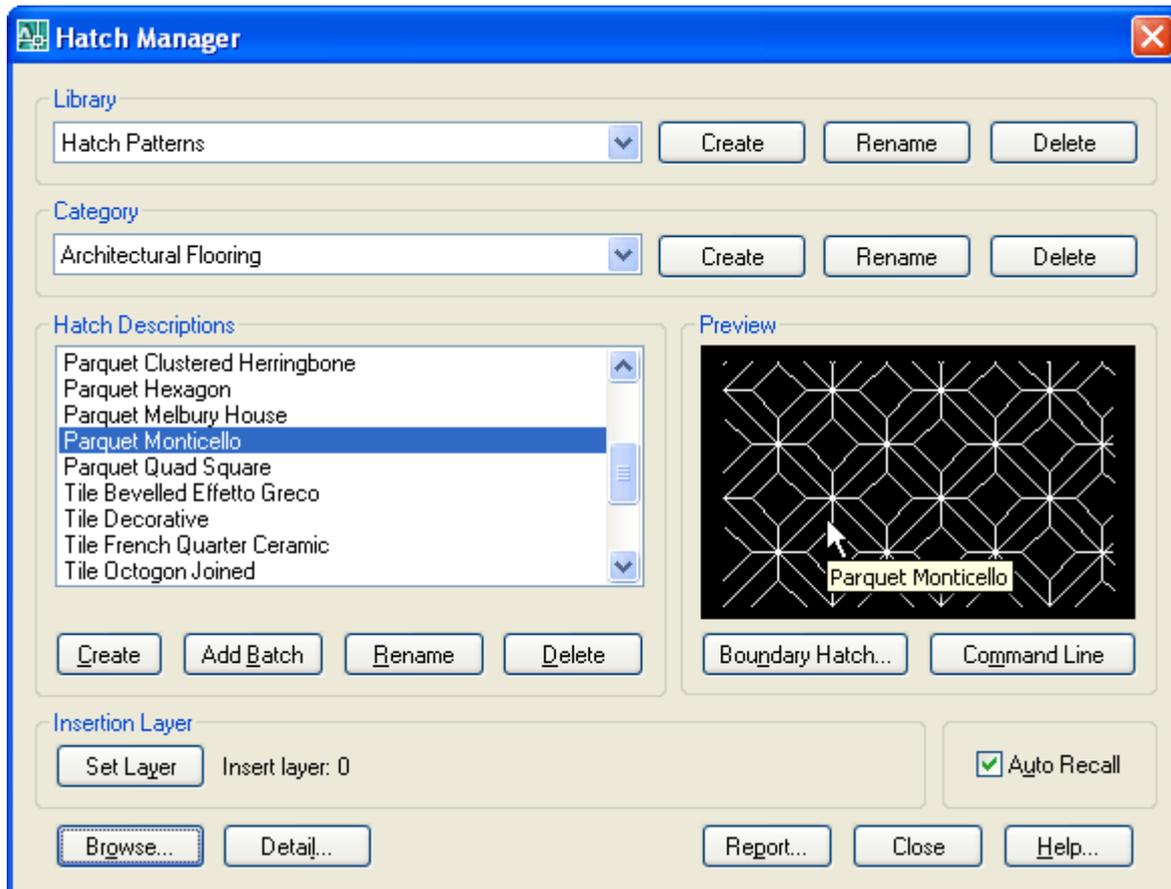
Patterns are organised within distinct libraries with advanced viewing capabilities. They can be identified fast and are accessible right when you need them. Management tools ensure the library database is always up to date, with the ability to view and print pattern details and other library information. Patterns can be contained on local drives or on a network.

With Hatch Manager you can:

- Convert existing drawing entities, blocks and logos into hatch patterns.
- Capture and extract patterns you find in any drawing.
- Add multiple PAT files in seconds using the unique batch-processing tools.
- View and compare patterns quickly and easily within libraries.
- Avoid unnecessary duplication of patterns with full network compatibility.
- Secure your work with full library password protection.
- Effectively manage and keep track of all your hatch patterns.

Initial Steps

Hatch Manager's main dialog displays the contents of the hatch pattern database. The current library and category is shown at the top with the relevant pattern descriptions listed below. Displayed right is the preview image for the pattern currently highlighted and below is shown the current layer for insertion. Libraries are stored within the GlobalCAD\...\Libs folder.



Creating a working library and category is the first step in adding your patterns to the database. The library can be assigned a full and meaningful description, such as the working discipline e.g. 'architectural hatch patterns' or maybe a project name.

Note when using the Add Batch feature to add multiple hatch patterns: To successfully add multiple patterns under Hatch Manager, each pattern must be capable of being independently generated within the current AutoCAD session. This means that the .PAT file(s) is valid and opens without error.

Creating a Library

Select the **Create** button under the Library heading from the main display.

- 1 Enter the proposed library description within the text box.
- 2 Next, enter your signature details. Note the default shown is the user's login name.
- 3 Choose **OK** to accept the descriptions and proceed.

To rename an existing library, pick the chosen description from the Library list box and click on the **Rename** button shown right. Next, enter the proposed new library heading within the text box and choose **OK** to accept this description.

Similarly, to delete an existing library, pick the chosen description from the Library list box and click on the **Delete** button shown right. Next, to confirm your wish to remove this library, click on the **OK** button shown or **Cancel** to exit.

Note that when a library is deleted, its corresponding contents (including patterns) are also deleted.

Creating a Category

Select the **Add** button under the Category heading from the main display.

- 1 Enter the proposed category description within the text box.
- 2 Choose **OK** to accept the description and proceed.

To rename an existing category, pick the chosen description from the Category list box and click on the **Rename** button shown right. Next, enter the proposed new category heading within the text box. Choose **OK** to accept this description.

Similarly, to delete an existing category, pick the chosen description from the Category list box and click on the **Delete** button shown right. Next, to confirm your wish to remove this category, click on the **OK** button shown or **Cancel** to exit.

Note that when a category is deleted, its corresponding contents (including patterns) are also deleted.

Creating a Hatch Pattern

Hatch patterns can be created from a wide range of existing drawing entities. In addition, advanced formatting options let you control the pattern resolution and spacing between each pattern repetition.

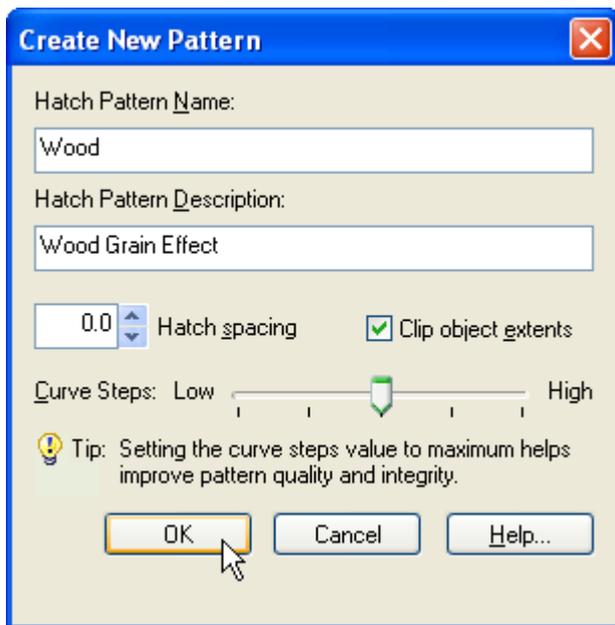
- 1 Draw an approximation of your pattern, using standard AutoCAD draw commands.

This can include Line, Pline, Point, Arc, Circle, Ellipse etc – even blocks!
- 2 Select the **Create** button under the Hatch Descriptions heading from the main display.
- 3 Next, specify the pattern **Lower Left Corner** and **Upper Right Corner**. This forms a crossing selection (a rectangular area drawn to select objects fully or partly within its borders) – also see explanation of **Clip Object Extents** option below.
- 4 From the Create New Pattern dialog, enter a name and description for the hatch pattern.

You can control the spacing between each repetition of the hatch pattern with **Hatch Spacing**. Setting the value to 1 will generate a space equal to the size of the pattern, applied to all four sides. The range is limited to 0-255 and is set to 0 by default (no spacing). Values within the range can also be set to one decimal point e.g. 0.5.

The **Clip Object Extents** option relates to the way entities you select are processed during hatch pattern generation. If this option is on, the selection rectangle you specified in point 3 above, will be re-sized (if necessary) to the precise extents of the entities selected. This simplifies entity selection and means you do not have to manually locate and snap to the entity extents yourself. Alternatively, with the Clip Object Extents option turned off, any free space between the selection rectangle you specify and the entities will be used to determine the hatch spacing between each pattern repetition.

You can also increase or reduce the **Curve Steps** by moving the slider – this option relates to the way in which curved entities such as arcs, circles and ellipses are represented within the hatch pattern. Increasing this value can create shapes that are more accurate representations of the selected curved entities.



- 5 Click on the **OK** button and wait for the hatch pattern definition to be generated – this may take a few seconds.
- 6 Next, a dialog is displayed prompting for the **Preview Image Scale**. If you are satisfied with the image click **Accept**. Otherwise, pick **Define Image Scale**, enter an alternative numerical value (non-zero) in the text box and click **Retry**. You can repeat this process until you are satisfied with the preview image.

Hatch Manager will automatically generate the pattern and zoom in around the selected objects to create the preview image.

Notes

- AutoCAD currently does not support Spline or Text entities for hatch pattern creation. Curved entities increase processing time, please be patient as this can take some moments.
- All entities selected for hatch pattern creation must lie in the current UCS plane. Blocks must be uniformly scaled.

Troubleshooting

- The file size of hatch patterns can increase substantially when derived from entities where circles and/or arcs are present. To reduce pattern file size, replace circles and/or arcs with short line segments where possible.
- When hatching boundaries that are located a long distance from the current UCS origin, patterns may appear to 'break apart'. This is a known issue within AutoCAD. In order to avoid this, temporarily set the UCS origin close to the proposed boundary by typing UCS followed by MOVE at the AutoCAD **Command line**. You can then set the UCS origin back to its default location once the pattern has been successfully generated.
- Occasionally, the default pattern scale of 1.0 in the AutoCAD Boundary Hatch dialog generates the error message 'Hatch spacing too dense, or dash size too small'. This can be remedied by changing the AutoCAD HPSCALE system variable from its default value of 1.0 to an alternative value such as 0.1.

Adding Multiple Patterns

Hatch patterns can also be added in multiple selections with the unique and powerful Add Batch feature. Use this method to create a complete category listing from a set of patterns stored anywhere on disk (local and network drives).

- 1 Select the **Add Batch** button under Hatch Descriptions from the main display.
- 2 From the Add Batch dialog the following options are available, according to preference:

Include sub-folders
 Create categories from sub-folder names
 Ignore duplicate-named hatch patterns
 Ignore illegally-named hatch patterns

The Advanced section of the dialog also provides an option to define the preview image scale for each new hatch pattern:

Define preview image scale

Choose the *Define preview image scale* option if you would like to manually specify how the hatch pattern will be displayed in the Hatch Manager preview image. Each time a hatch pattern is loaded, a Preview Image dialog will be displayed, requesting the image scale to use. This value must be numerical and non-zero.

- 3 Next, click on the **Browse** button to select the folder containing the relevant patterns. Note, if the *Include sub-folders* option has been specified, patterns contained within any sub-folders will also be added. In addition, selecting the *Create categories from sub-folder names* option means that new categories will be created, containing the contents of each sub-folder.
- 4 Returning to the main Add Batch dialog, click on **OK** to proceed.

Please wait while the patterns are now processed and added to the database. Once this operation is complete, the main Hatch Manager dialog displays the current category and new pattern entries.

To rename an existing pattern description, pick the chosen description from the Hatch Descriptions box and click on the **Rename** button shown below. Next, enter the proposed new pattern name within the text box and choose **OK** to accept this description.

Similarly, to delete an existing pattern description, pick the chosen description from the Hatch Descriptions box and click on the **Delete** button shown below. Next, to confirm your wish to remove this description, click on the **OK** button shown or **Cancel** to exit.

Hatching Objects

You can utilise the standard AutoCAD *Bhatch* or *Hatch* command to hatch objects in the current drawing by clicking on the **Boundary Hatch** or **Command Line** button.

The Boundary Hatch button displays the current hatch pattern in the standard AutoCAD boundary hatch dialog. The Command Line button functions in the same way as AutoCAD's standard hatch command i.e. it prompts for the scale, angle and objects to hatch at the AutoCAD **Command line**.

Setting the Insertion Layer

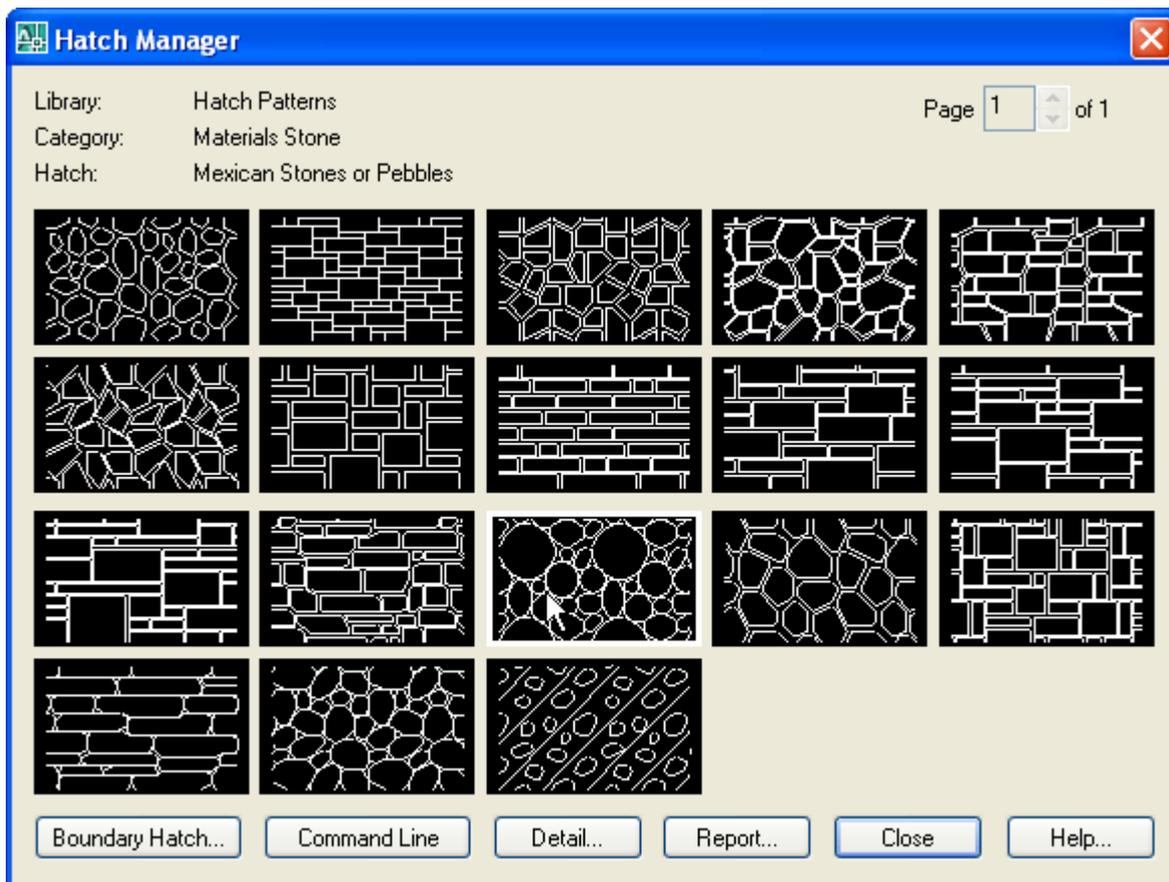
The layer upon which hatch patterns will be created is shown near the bottom of the main dialog display, under the heading Insertion Layer. The default insertion layer is 0. In order to change this setting:

- 1 Select the **Set Layer** button. The existing drawing layers available are displayed in a text box.
- 2 Select the desired layer for pattern creation and click on the **OK** button.

Note, to create a new layer, enter the new name in the Create New Layer text box below and click on the **OK** button. This will be created and made the current layer.

Browsing Libraries

Sometimes it is useful to view and compare hatch patterns simultaneously, usually in order to make an informed choice from similar patterns within a library. Using Hatch Manager's Browse feature you can view patterns in this way, rapidly browsing an entire library if desired.



- 1 Select the **Browse** button from the main display.

The page scroll box is located in the top right-hand corner of the Browse dialog.

- 2 Click on the scroll box arrows to change the page number to view.

Current Library Details

This feature displays information concerning the current library.

Select the **Detail** button from the main display and click on **OK** when finished.

Information displayed includes the library filename and folder location, the library description, creator, current category and total number of hatch patterns present within this category.

Library Protection

Once created, you may wish to protect specific libraries from accidental modification or deletion.

- 1 Select the **Detail** button from the main display.
- 2 Pick the **Library Protected** tick box at the base of the dialog display.
- 3 Enter a unique password and confirm this in the text box below.

All library maintenance functions (i.e. Add, Delete and Rename) will now be disabled until the tick box is de-selected. Note that the original password specified must be re-entered in order to unprotect the library. It is recommended that you make a note of this and store it in a safe place.

Viewing the Hatch Report

Click on the **Report** button to display the full list of hatch patterns stored under the current category.

Patterns are listed in alphanumerical order.

Choose the Print Report button to obtain hardcopy records.

Creating and Managing Complex Linetype Libraries

Linetype Wizard provides the solution to all your linetype creation and management needs. You can build entire libraries from scratch using standard drawing entities to create both simple and complex linetypes.

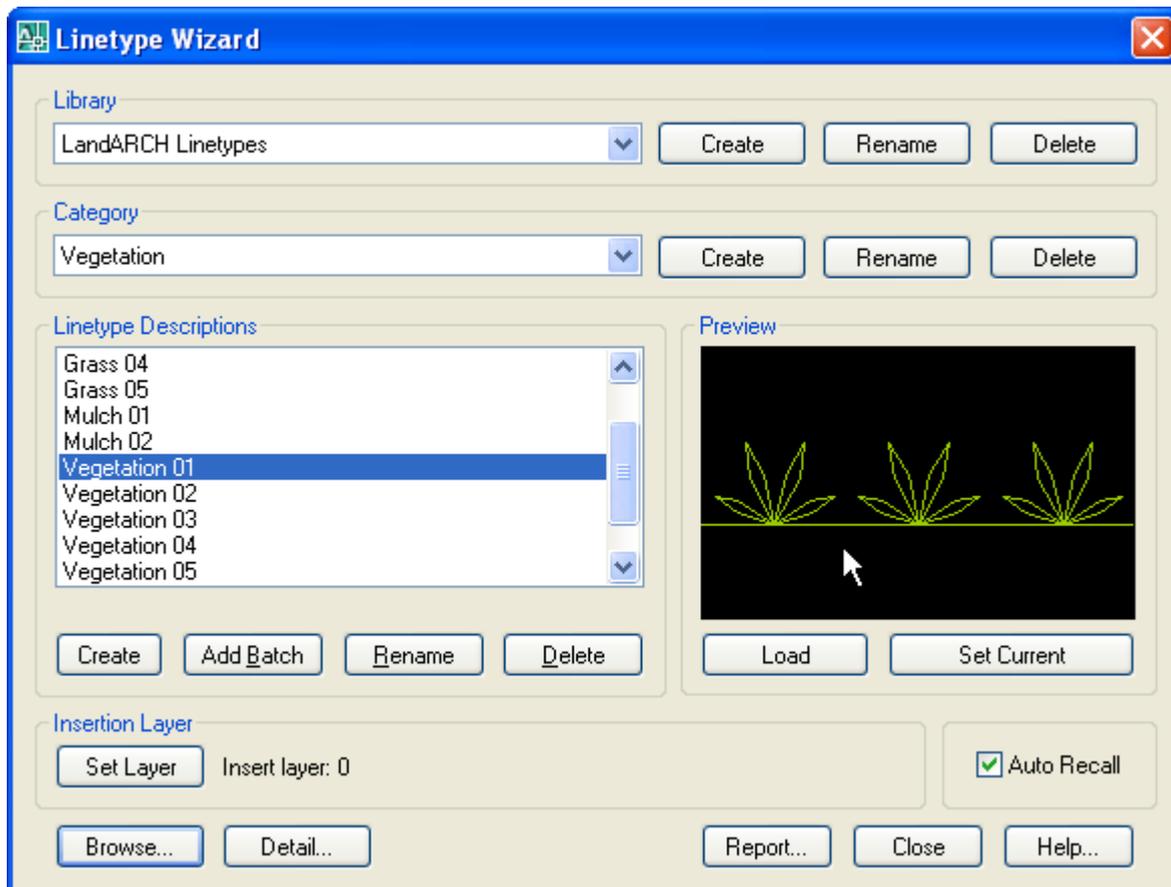
Furthermore, linetypes are organised within distinct libraries with advanced viewing capabilities. They can be identified fast and are accessible right when you need them. Management tools ensure the library database is always up to date, with the ability to view and print linetype details and other information.

With Linetype Wizard you can:

- Create linetypes from existing drawing entities, blocks and text.
- Capture and extract linetypes you find in any drawing.
- View and compare linetypes quickly and easily within libraries.
- Avoid unnecessary duplication of linetypes with full network compatibility.
- Secure your work with full library password protection.
- Effectively manage and keep track of all your AutoCAD linetypes.

Initial Steps

Linetype Wizard's main dialog displays the contents of the linetypes database. The current library and category is shown at the top with the relevant linetype descriptions listed below. Displayed right is the preview image for the linetype currently highlighted and below is shown the current layer for insertion. Libraries are stored within the GlobalCAD\...\Libs folder.



Creating a working library and category is the first step in adding your linetypes to the database. The library can be assigned a full and meaningful description, such as the working discipline e.g. 'architectural linetypes' or maybe a project name.

Creating a Library

Select the **Create** button under the Library heading from the main display.

- 1 Enter the proposed library description within the text box.

- 2 Next, enter your signature details. Note the default shown is the user's login name.
- 3 Choose **OK** to accept the descriptions and proceed.

To rename an existing library, pick the chosen description from the Library list box and click on the **Rename** button shown right. Next, enter the proposed new library heading within the text box and choose **OK** to accept this description.

Similarly, to delete an existing library, pick the chosen description from the Library list box and click on the **Delete** button shown right. Next, to confirm your wish to remove this library, click on the **OK** button shown or **Cancel** to exit.

Note that when a library is deleted, its corresponding contents (including linetypes) are also deleted.

Creating a Category

Select the **Add** button under the Category heading from the main display.

- 1 Enter the proposed category description within the text box.
- 2 Choose **OK** to accept the description and proceed.

To rename an existing category, pick the chosen description from the Category list box and click on the **Rename** button shown right. Next, enter the proposed new category heading within the text box. Choose **OK** to accept this description.

Similarly, to delete an existing category, pick the chosen description from the Category list box and click on the **Delete** button shown right. Next, to confirm your wish to remove this category, click on the **OK** button shown or **Cancel** to exit.

Note that when a category is deleted, its corresponding contents (including linetypes) are also deleted.

Creating a Linetype

Simple linetypes usually consist of a repeating series of dashes, dots and/or text objects. Complex linetypes can also include shapes created from existing drawing entities.

- 1 Draw an approximation of your linetype, using standard AutoCAD draw commands.

This can include Line, Pline, Point, Text, Arc, Circle, Ellipse etc – even blocks!

For best results, ensure you include a short horizontal line at the start and end of your selected entities and that these lines are exactly aligned on the same axis i.e. in a straight line. This will improve the 'joining' of linetype repetitions, especially when the new linetype is applied to polylines, splines etc.

- 2 Select the **Create** button under the Linetype Descriptions heading from the main display.
- 3 Using AutoCAD's object snap, specify the linetype **Start Point** and **End Point**.
- 4 Select the objects that will make up the new linetype.
- 5 From the **Create New Linetype** dialog, enter a name and description for the linetype.

You can also increase or reduce the **Curve Steps** by moving the slider – increasing this value can create shapes that are more accurate representations of the selected entities.

Select the **Define Preview** option if you want to control how the linetype will be scaled in the preview image. After clicking the **OK** button, a dialog will be displayed prompting for the **Preview Image Scale**. If you are satisfied with the image click **Accept**. Otherwise, pick **Define Image Scale**, enter an alternative numerical value (non-zero) in the text box and click **Retry**. You can repeat this process until you are satisfied with the preview image.

- 6 Click on the **OK** button. Linetype Wizard will automatically generate the linetype and zoom in around the selected objects to create the preview image.

Notes

- AutoCAD currently does not support Spline entities for linetype creation and will force text with a non-standard width factor to 1. It also does not support linetype definitions that begin with an empty segment. If you try to create a linetype that begins with an empty segment, the program will automatically add a dot to the beginning of the definition.
- Complex linetypes containing polylines or lightweight polylines may display better with PLINEMEN set to 1.
- All entities selected for linetype creation must lie in the current UCS plane. Blocks must be uniformly scaled.

Troubleshooting

- Sometimes linetypes that are applied to closed polylines (such as rectangles or polygons) are not uniformly applied. Instead, you see long continuous dashes at the juncture of vertices.

This display of linetypes reflects the default setting for Linetype Generation (disabled). When Linetype Generation is disabled, linetypes that would display a truncated section at the vertices will display a continuous dashed line instead of a portion of the repeated pattern that makes up the linetype. If you enable Linetype Generation in the Properties window, the linetype will continue to the vertices, displaying a truncated section of the linetype.

Use the Properties window to select the polylines that need to be changed.

On the Modify menu, click Properties.

Select the polyline(s) that you want to modify.

In the Properties window, select the field to the right of Linetype Generation, and then select Enable from the drop-down list.

The results are displayed on the screen immediately. Note some linetypes will display a 'gap' or overlapping linetype components at the vertices when Linetype Generation is enabled.

- When sending drawings with complex linetypes electronically to a third-party, use the AutoCAD eTransmit feature, located on the Files drop-down menu. This will create a zip or self-extracting exe transmittal set that includes all dependant support files.

Complex linetypes you create using Linetype Wizard may contain compiled shape files (SHX files) and AutoCAD needs to find these in order to display the drawing correctly. With eTransmit, all SHX files are included with the associated drawing ready for sending. The recipient can then simply extract the drawing and associated support files to a common folder for the drawing to display correctly when opened.

- Compiled shape files (SHX files) that reside in AutoCAD complex linetypes have a physical limit of 64 characters. This limit extends to the path so, for example, the shape file C:\Program Files\AutoCAD 2002\Support\Shapes.shx consists of 44 characters in total. Therefore, as a precaution when sending drawings with dependant complex linetypes, always recommend the recipient opens the eTransmit transmittal set to a short path location.

Loading Linetypes

Linetypes can be loaded into the current drawing by clicking the **Load** button, located below the preview image on the main dialog display. Alternatively, click the **Set Current** button to load the linetype and force it to become the current linetype.

Setting the Insertion Layer

The layer upon which linetypes will be created is shown near the bottom of the main dialog display, under the heading Insertion Layer. The default insertion layer is 0. In order to change this setting:

- 1 Select the **Set Layer** button. The existing drawing layers available are displayed in a text box.
- 2 Select the desired layer for linetype creation and click on the **OK** button.

Note, to create a new layer, enter the new name in the Create New Layer text box below and click on the **OK** button. This will be created and made the current layer.

Browsing Libraries

Sometimes it is useful to view and compare linetypes simultaneously, usually in order to make an informed choice from similar linetypes within a library. Using Linetype Wizard's Browse feature you can view linetypes in this way, rapidly browsing an entire library if desired.

- 1 Select the **Browse** button from the main display.

The page scroll box is located in the top right-hand corner of the Browse dialog.

- 2 Click on the scroll box arrows to change the page number to view.

Current Library Details

This feature displays information concerning the current library.

Select the **Detail** button from the main display and click on **OK** when finished.

Information displayed includes the library filename and folder location, the library description, creator, current category and total number of linetypes present within this category.

Library Protection

Once created, you may wish to protect specific libraries from accidental modification or deletion.

- 1 Select the **Detail** button from the main display.
- 2 Pick the **Library Protected** tick box at the base of the dialog display.
- 3 Enter a unique password and confirm this in the text box below.

All library maintenance functions (i.e. Add, Delete and Rename) will now be disabled until the tick box is de-selected. Note that the original password specified must be re-entered in order to unprotect the library. It is recommended that you make a note of this and store it in a safe place.

Viewing the Linetype Report

Click on the **Report** button to display the full list of linetypes stored under the current category.

Linetypes are listed in alphanumerical order.

Choose the Print Report button to obtain hardcopy records.