Thank you for installing the AutoCAD Express Tools!

The AutoCAD Express Tools are a library of productivity tools designed to help you extend the power of AutoCAD. They are easy to use, easily integrated into your menus and toolbars, and cover a wide range of AutoCAD functions, including dimensioning, drawing, object selection, object modification, and much more.

The AutoCAD Express Tools are unsupported. However, if you have questions or comments about an Express Tool, you can visit the AutoCAD Express Tools newsgroup. In general, these utilities are provided as a courtesy to you by Autodesk and Autodesk does not assume any responsibility for their successful operation.
To activate the AutoCAD Express Tools, use the **EXPRESSTOOLS** command.

The library files for the AutoCAD Express Tools are not loaded at AutoCAD startup by default. This was done to reduce the time needed to load AutoCAD. The libraries load the first time that an Express Tools command is issued, creating a delay the first time you use an Express Tool in an AutoCAD session. You can use the EXPRESSTOOLS command to force the AutoCAD Express Tools libraries to be loaded at the beginning of the AutoCAD session.

If the AutoCAD Express Tool commands are available on the command line but the Express menu is not displayed, use the **EXPRESSMENU** command to display the menu.
Uses a block (or entities within a block reference) as a boundary to extend objects in a drawing.

Command: BEXTEND
Select edges for extend: *Select block to use as the extending edge*
Specify object to extend or shift-select to trim or
[Fence/Crossing/Project/Edge/eRase/Undo]: *Select object to extend*

See also: **BTRIM**
Uses a block (or entities within a block reference) as a boundary to trim objects in a drawing.

Command: BTRIM
Select cutting edge: *Select block to use as the cutting edge*
Specify object to trim or shift-select to extend or
[Fence/Crossing/Project/Edge/eRase/Undo]: *Select object to trim*

See also: BEXTEND
Command: BURST
Select objects: Use an object selection method

**Usage Example:**
Explode attributes and convert them to text objects.

*Select a block containing two attributes:*

![Block diagram](image1)

*Block exploded and attribute values converted to text:*

![Block diagram with attributes](image2)
Select nested objects to copy: *Use an object selection method*
Specify base point or displacement, or [Multiple]: *Specify a point, or enter M for multiple copies*
Second point of displacement: *Specify a point*
Command: XLIST
Select nested xref or block object to list: *Use an object selection method*

**Usage Example:**
Select an object as a nested block or xref file to display the Object and Block Name, Layer, Color, and Linetype properties.

*The pick box identifies the plant as the selected object:*

Selecting the object opens the Xref/Block Nested Object List dialog box with the property information as shown:
Dialog Box Options

**Usage Example:** Draw an arc, and then use ARCTEXT to place text along the arc's curve.
Command: RTEDIT
Select objects: Select the Rtext object
Style: MYSTYLE Height: 0.1000, Rotation: 0
Enter an option [Style/Height/Rotation/Edit]: Specify an option
Options

To create Rtext objects use the RTEXT command.
Reactive text (RText) objects are displayed the same way normal Text or MText objects are displayed, but the source for the text is either an external text file or the value of a DIESEL expression. You can edit an RText object with the RTEDIT command.

Command: RTEXT
Current text style: STANDARD  Text height: 0.2000  Text rotation: 0
Enter an option [Style/Height/Rotation/File/Diesel] <Diesel>: Specif
Options

You can use an RTxt object as a file reference to display text, such as a sheet note or a legal disclaimer, that is common to several drawings. You can also use it to display larger bodies of text such as specifications or assembly instructions.
Using DIESEL Expressions

DIESEL provides options previously accessible only with custom programs. Here are some examples of how RTText objects with DIESEL expressions can be used in your drawings. You can paste these examples into the DIESEL Expression dialog box.

To display the drawing name:

Drawing file: $(getvar, "dwgname")

Output:

Drawing file: 102-fp12.dwg

To include the directory path with the file name:

Drawing name: $(getvar, "dwgprefix")$(getvar, "dwgname")

Output:

Drawing file: C:\Projects\97-102\Arch\102-fp12.dwg

When you plot a drawing, you may want the hard copy to show the date and the time that the plot was created. The following DIESEL expression displays this information in your drawing:

$(edtime, 0, MON DD"," YYYY - H:MMam/pm)

Output:

Mar 22, 2000 - 11:12pm

The $(getprop) DIESEL function

RTText supports Drawing Properties through a locally defined $(getprop) DIESEL function. With $(getprop), values from Drawing Properties dialog box tabs can be extracted and displayed in RTText objects.

The syntax for the $(getprop) DIESEL function is:

$(getprop, property name)
where *property name*; can be any of the following fields from the DWGPROPS command:

- Title
- Subject
- Author
- Comments
- Keywords
- LastSavedBy
- Revno
- Custom Property  the name of a custom property

For example, if the Drawing Properties for a drawing contains the text "Excavated Site" as the Subject, the DIESEL expression:

```
Subject: $(getprop, subject)
```

in an RText object will display as:

```
Subject: Excavated Site
```

If you have a custom property named "Project Name", the following DIESEL expression will display the value:

```
Project: $(getprop, %PROJECT_NAME)
```

**The $(xrefs) DIESEL function**

RText supports listing Xref files attached to a drawing through the $(xrefs) DIESEL function.

The syntax for the $(xrefs) DIESEL function is:

```
$(xrefs [, flags [, leader [, trailer]]])
```

**Example:** The RText DIESEL expression:

```
$(xrefs, 3)
```

displays a list of Xrefs in the following format:

```
B-ELEC  [c:\proj-14\b-elec.dwg]
M-ELEC  [c:\proj-14\m-elec.dwg]
R-ELEC  [c:\proj-14\r-elec.dwg]
```
while the expression:

$(xrefs,2,Includes: )

will list the Xrefs as:

Includes: B-ELEC  
Includes: M-ELEC  
Includes: R-ELEC  
Includes: F-ELEC

**The $(images) DIESEL function**

RText supports listing images attached to the drawing through the $(images) DIESEL function.

The syntax for the $(images) DIESEL function is:

$(images [, flags [, leader [, trailer]]])

Behavior is similar to the $(xrefs) function.

**The $(getrec) DIESEL function**

RText supports displaying Xrecord data through the $(getrec) DIESEL function.

The syntax for the $(getrec) DIESEL function is:

$(getrec, key, code)

The $(getrec) function extracts a value from an Xrecord by looking in the Named Object Dictionary for *key*, then for a data value associated with *code*. Currently, only group codes in the ranges 1-9 (string), 40-59 (real), 60-79 (integer) and 300-309 (string) are supported.
Command: TEXTFIT
Select Text to stretch or shrink: Use an object selection method
Specify end point or [Start point]: Specify a new end point, or enter S to specify a new start point
Pick new starting point: Specify a new start point
ending point: Specify a new endpoint

Usage Example:
Fit labels within enclosed borders by setting a new text width.

Select text:

```
CONFERENCE ROOM
ROOM 2027
```

Select a new endpoint:

```
CONFERENCE ROOM
ROOM 2027
```

Text size shrinks:

```
CONFERENCE ROOM
ROOM 2027
```
Command: TEXTMASK
Current settings: Offset factor = <default>, Mask type = <default>
Select text objects to mask or [Masktype/Offset]: Select an option or text or press ENTER to exit

Text objects are grouped together with the mask objects so that they moved, copied, and erased together. To update the mask after editing the text, run TEXTMASK again, and select the text item to be updated. The previous mask object is erased and a new one is created. A companion command, TEXTUNMASK, removes masks from text.

**Masktype Options**

Command: TEXTMASK
Current settings: Offset factor = <default>, Mask type = <default>
Select text objects to mask or [Masktype/Offset]: Masktype

Mask type currently set to <default>
Specify object type to use for mask [Wipeout/3dface/Solid] <default>

**Offset Option**

Command: TEXTMASK
Current settings: Offset factor = <default>, Mask type = <default>
Select text objects to mask or [Masktype/Offset]: Offset

Mask offset currently set to <default>
Enter offset factor relative to text height <default>: Specify a value, or select points to set the mask size

Initial default values are Offset factor = 0.35 and Masktype = Wipeout. Subsequent default values are the last specified values.

Objects in paper space used to mask objects in model space are not displayed correctly in plots. The display may look correct but your plot will not. At plot time, AutoCAD processes the model space viewports before processing paper space objects. This makes it impossible for anything in paper space to be "on top of" anything in model space.
**Wipeout**

A Wipeout is a custom object created by the WIPEOUT tool. Since the Wipeout object is like an image, it has the same requirements for plotting. You need a raster-capable plotter with an ADI 4.3 raster-capable driver or the system printer driver. Masking does not plot properly if the Hide Lines option is used while plotting. One advantage of using a Wipeout mask is that what you see on screen is what you get on your plot.

**3Dface**

The 3Dface created by TEXTMASK has all of its edges set to invisible. This option is useful if you do not have a raster-capable plotter. The Hide Lines option must be used while plotting for the mask effect to work. With this option, masked text does not look masked on screen.

**Solid**

If you are using an HP plotter, use the HPCONFIG command and make sure that the In Color option is selected in Print Colors. To make the mask invisible (that is, to hide all objects behind it,) press the pens button in HPCONFIG, and create a pen that uses a shaded fill. Then set the shade percentage to 0%. With this option, masked text appears masked on screen. Also, masks can be colored to highlight text. Masking does not plot properly if the Hide Lines option is used while plotting.

**Usage Example:**

*Text over lines:*

![Image of text over lines]

*Masked text over lines:*

![Image of masked text over lines]
Command: TEXTUNMASK
Select text or MText object from which mask is to be removed.
Select objects: Use an object selection method
**TXT2MTXT** will work on any text objects created with the TEXT or DTEXT commands. If the original set of text objects was created with DTEXT or with successive text commands, enabling the **Sort top-down** option and disabling the **Create word-wrap MText** options will produce the most visually similar results.

The selected text objects will be removed from the drawing and a new MText object will be inserted.

This command attempts to honor text size, font and color changes between text objects.

Command: TXT2MTXT  
Select text objects or press Enter to set Options:  
Select objects: *Use an object selection method to select text*

**Usage Example:**  
Select three text objects and convert them to mtext objects.

*Select text items:*

*Normal text to be converted to Mtext*  

*Text converted to mtext:*

*Normal text to be converted to Mtext*
Options dialog
Command: TXTEXP
Select text to be EXPLODED: Use an object selection method

Usage Example:
Draw text, and then use TXTEXP. The text objects explode into lines and arcs.

Text:

EXPLODING TEXT
TO LINES

Exploded text turns into lines and arcs:

EXPLODING TEXT
TO LINES
Command: ALIGNSPACE
FIRST alignment point in MODEL space: Specify a point
SECOND alignment point in MODEL space: Specify a point
FIRST alignment point in PAPER space: Specify a point
SECOND alignment point in PAPER space: Specify a point

Usage Example:
Align a circle in model space with a dimension in paper space.

Specify two points in model space:

Specify two points in paper:

The circle is aligned with the dimension in paper space:
Command: VPScale
Select edge of viewport: Select a viewport

The scale is displayed in one of three ways:

- If your current units are architectural, the result looks like this:

  \[
  \text{PS:MS} == 1:96 \\
  \text{Viewport Scale: } 1/8" = 1'-0"
  \]

- If your current units are engineering, the result looks like this:

  \[
  \text{PS:MS} == 1:96 \\
  \text{Viewport Scale: } 1" = 8'
  \]

- For all other units, the scale looks like this:

  \[
  \text{PS:MS} == 1:96
  \]
Command: VPSYNC
Select master viewport: Select a viewport
Select viewports to be aligned to master viewport.
Select objects: Select one or more viewports

Usage Example:
Synchronize two viewports so that they look like one.

Select master viewport and child viewport:

Viewports are now synchronized:
Dimension Style Export

Export Filename
Drawing1.dim Browse...

Available Dimension Styles
Standard

Text Style Options
- Text Style Name Only
- Full Text Style Information
DIMEX Dialog Box Options
Dimension Style Import

- Import Filename
  - Drawing1.dim
  - Browse...

- Import Options
  - Keep Existing Style
  - Overwrite Existing Style

- OK
- Cancel
DIMIM Dialog Box Options
Command: QLATTACH
Select Leader: Select a leader
Select Annotation: Select the mtext, tolerance, or block reference object for the annotation

Usage Example:
Select a leader line, and then specify the mtext object to attach.

Separate leader line and mtext objects:

At the Select Leader: prompt, you will be prompted until you select a leader object or abort the command. At the Select Annotation: prompt, you will be prompted until you select an MText, Tolerance, or Block Reference object for the annotation.

If attachment is successful, the tail of the leader will jump to the default attachment point of MText or Tolerance annotation. If the angle is appropriate a hook line will appear.

The leader and annotation objects must be on the same plane, otherwise they will not attach.

If you wish to attach to a Block Reference, you should move the leader endpoint to where you want it before you execute qlattach. The leader will not move, and you will see no immediate visual feedback that the leader was successfully attached.
AutoCAD R13 offered the Leader object, but not attachment to the leader annotation. This command lets you process the leaders in a Release 13 drawing, to attach them to their annotation objects for more productive editing.

QLATTACHSET will report the number of Leaders found, along with the number successfully attached.

This command is intended primarily for leaders created by R13.

Command: QLATTACHSET
Select objects: Select one or multiple leader lines and annotation objects

**Usage Example:**
Select leader line(s), and mtext objects to attach.

Select objects: Select one or multiple leader lines and annotation objects
Number of Leaders = 2
Number with annotation detached = 2.

*Leader with detached mtext object:*

![Diagram of Leader with detached mtext object]

*Leader attached to mtext object:*

![Diagram of Leader attached to mtext object]
All leaders in the selection set will be detached from their annotation objects. The only visual feedback will be the loss of a hook line.

Command: QLDETACHSET
Select objects: Select leader line

Usage Example:
Select a leader line to detach from an mtext object.

Select objects: Select leader line
Number of Leaders = 1
Number with annotation detached = 1

Leader with mtext object attached:

Leader detached from the mtext object, which can then be moved:
The behavior of FASTSELECT can be controlled with the command FSMODE. If FSMODE is set to ON (the default), FASTSELECT selects only those objects that are touching the selected object. When FSMODE is OFF, FASTSELECT selects all the objects that are touching the selected object as well as any objects that are touching those objects. The command continues down the chain of unselected objects that are touching the selected object until no more unselected objects are visible on screen.

Command: 'FS
Select touching object: Select an object

Command: 'FSMODE
FASTSEL chain selection <OFF>: Enter ON or OFF

Usage Example:

*FASTSELECT used to select objects with FSMODE turned off:*

*FASTSELECT used to select objects with FSMODE turned on:*
Command: GETSEL
Select an object on the Source layer <*>: *Use a selection set method or press ENTER for all layers*
Select an object of the Type you want <*>: *Use a selection set method or press ENTER for all object types*

**Usage Example:**
Draw objects on different layers, and select lines from LAYER1

Select an object on the Source layer <*>: *Select an object on LAYER1*
Select an object of the Type you want <*>: *Select a LINE object*
Collecting all LINE objects on layer LAYER1...
1 object has been placed in the active selection set.
Allows curved clipping; isolates specified portions of blocks, xref drawings, images, and Wipeouts, by clipping or removing the display of all other objects with a polyline, circle, arc, ellipse, text, mtext, or attribute definition.

With CLIPIT you can use ARCS, CIRCLES, or POLYLINES to define clipping boundaries for BLOCKS, XREFS, IMAGES, and WIPEOUTS. Draw arcs, circles, and/or polylines to define area(s) to be clipped.

Command: CLIPIT
Pick a POLYLINE, CIRCLE, ARC, ELLIPSE, or TEXT object for clipping edge...
Select objects: Use an object selection method
Pick an IMAGE, a WIPEOUT, or an XREF/BLOCK to clip: Select the object to clip
Enter max error distance for resolution of arcs <0.0200>: Specify a resolution value or press ENTER

CLIPIT makes clipping to curved boundaries possible by traversing arc segments with a series of short straight segments. CLIPIT lets you to control the precision with which it traverses arc segments by allowing you to specify the maximum distance between the midpoint of any given segment and the arc.

Maximum error distance for resolution of arcs:

![Diagram of curved clipping boundary with error distance indicated.](image)

Entering a large error value results in fewer segments and faster AutoCAD performance, but arcs appear less smooth. Conversely, entering a very small value results in smooth clipping along arcs, but places more of a processing burden on AutoCAD during regeneration of a drawing.

**Usage Example:**
Draw a polyline, a circle, or an arc, and clip a portion of an externally referenced floor plan to isolate it.

Select a circle as the clipping boundary:
Select xref:

Xref clipped with a circle as the clipping boundary:
Command: MOCORO
Select objects: Use an object selection method
Base point: Specify a base point
[Move/Copy/Rotate/Scale/Base/Undo]<eXit>: Specify an option
Options

Usage Example:
Use the move, copy, rotate, and scale options.

Select an object (chair):

Specify a base point, and then enter the option to move the object:

Specify the new location for the object:

Copy the object:

Rotate the object:
Scale the object:
Command: MSTRETCH
Define crossing windows or crossing polygons...
Options: Crossing Polygon or Crossing first point
Specify an option [CP/C] <Crossing first point>: Use a crossing window or a crossing polygon selection method
Specify an option [CP/C/Undo] <Crossing first point>: Specify an option, or press ENTER to continue
Specify an option [Remove objects] <Base point>: Specify a base point, press ENTER or enter R to remove object(s)
Second base point: Specify a second point or press ENTER
Option

Usage Example: 
Select multiple objects and expand their size with a crossing window and a crossing polygon.

Rectangular shapes:

Stretching shapes:

Shapes enlarged:
All instances of the shape in the drawing are replaced with insertions of the newly defined block.

Command: SHP2BLK
Select shape entity to convert: Select a shape object
Enter the name of the block to create <default>: Enter a name for the block

Usage Example:
Convert a ground shape into a block.

Command: SHP2BLK
Select shape entity to convert: Select the ground shape object Enter the name of the block to create <GROUND>: Press ENTER to accept the default
Deleting block WMF0.
1 block deleted.
The shape GROUND has been replaced with block GROUND.
SUPERHATCH Dialog Box Options

For images:

Insertion point <0,0>: *Specify an insertion point*
Base image size: Width: 1.00, Height: 0.24 <Unitless>
Scale factor <1>: *Specify a scale factor*

For blocks and xrefs:

Insertion point: *Specify an insertion point*
X scale factor <1> / Corner / XYZ: *Specify the X scale factor*
Y scale factor (default=X): *Specify the Y scale factor*
Rotation angle <0>: *Specify the rotation angle*
Select a window around the block to define column and row tile distances.
Specify block [Extents] First corner <magenta rectang>: *Specify a frame for tiling, and press ENTER*

After the preceding steps have been completed:

Selecting visible objects for boundary detection...Done.
Specify an option [Advanced options] <Internal point>: *Specify an internal point for hatch boundary, or enter A for advanced options, and press ENTER*
Preparing hatch objects for display...
Done.

Use TFRAMES to toggle object frames on and off. (for images and Wipeouts only)

This command is very versatile and opens up many possibilities for you to be creative. You can use the Block option to create hatch patterns on-the-fly; you can created blocks with different layers, colors, and linetypes to create hatch patterns that react to layer states and plot with a variety of line weights. You can use hatch images on surfaces to get a rendered effect. Use 3D blocks as hatch patterns to place repeating items such as bricks, roof tiles, or fence fills in 3D drawings.
Error tolerance for curves

SUPERHATCH makes hatching to curved boundaries possible by traversing arc segments with a series of short straight segments. SUPERHATCH lets you to control the precision with which it traverses arc segments by allowing you to specify the maximum distance between the midpoint of any given segment and the arc. (See the Curve Error Tolerance option described in the Dialog Box Options above.)

Maximum error distance for resolution of arcs:

![Diagram of error distance for arc resolution]

Entering a large error value results in fewer segments and faster AutoCAD performance, but arcs appear less smooth. Conversely, entering a very small value results in smooth hatching along arcs but places more of a processing burden on AutoCAD during regeneration of a drawing.

Usage Example:
Use a block as a hatch pattern.

Place the block:

![Diagram showing placement of block]

Specify the framing around the block:
Specify the boundary:

Finished hatch:

1) Start the group command.
Command: group

2) Make sure the Include Unnamed check box is selected.

3) Select the group. Note that "superhatch" appears in the description field for groups created by the SUPERHATCH command.

4) Click Selectable.

5) Choose OK.

The individual objects can now be modified separately.
Command: IMAGEEDIT
Use the IMAGEAPP command to specify a non-system-default editor.

See also: IMAGEAPP
Once you choose a directory, the drawing's BAK file is placed in the named directory after every Save.

Enter new value for MOVEBAK, or . for none <current setting>: 
Enter a . (period) for none or type the directory name

If you enter ".", the MOVEBAK directory name is cleared and BAK files will be left in the original directory.
This command will attempt to close all open drawings, offering to save any modified drawings, then exit AutoCAD. If a drawing has not been named yet, QQUIT will prompt for a file name.
Command: REDIR
Current REDIRMODE: Styles,Xrefs,Images,Rtext
Find and replace directory names
Enter old directory (use '* for all), or ? <options>: Specify the directory to be replaced or press ENTER for options (see REDIRMODE for option descriptions)
Replace "*" with: Specify a new directory or press ENTER to specify an empty string (see tip below.)

Usage Example 1:
List the current path associations and then change them:

Command: REDIR
Current REDIRMODE: Styles,Xrefs,Images,Rtext
Find and replace directory names
Enter old directory (use '* for all), or ? <options>: Enter ?
File references to list <*>: Press ENTER

<table>
<thead>
<tr>
<th>TYPE</th>
<th>NAME</th>
<th>FILE</th>
</tr>
</thead>
<tbody>
<tr>
<td>STYLE</td>
<td>STANDARD</td>
<td>TXT</td>
</tr>
<tr>
<td>STYLE</td>
<td>TTF</td>
<td>ARIAL.TTF</td>
</tr>
<tr>
<td>STYLE</td>
<td>XREFA</td>
<td>STANDARD</td>
</tr>
<tr>
<td>STYLE</td>
<td>XREFA</td>
<td>TTF</td>
</tr>
<tr>
<td>SHAPE</td>
<td>I:\WORK\TEST\ES.SHX</td>
<td></td>
</tr>
<tr>
<td>XREF</td>
<td>XREFA</td>
<td>I:\WORK\TEST\XREFA.DWG</td>
</tr>
<tr>
<td>IMAGE</td>
<td>THATCH</td>
<td>I:\work\TEST\brick2.bmp</td>
</tr>
<tr>
<td>IMAGE</td>
<td>XREFA</td>
<td>THATCH</td>
</tr>
<tr>
<td>RTEXT</td>
<td>I:\work\TEST\57072_b.txt</td>
<td></td>
</tr>
</tbody>
</table>

Enter old directory (use '* for all), or ? <options>: I:\WORK\TEST\nReplace "I:\WORK\TEST\" with: K:\TEST\n
Searching for old dir: I:\WORK\TEST\ in order to replace it with: K:\TEST\n
<table>
<thead>
<tr>
<th>TYPE</th>
<th>NAME</th>
<th>FILE</th>
</tr>
</thead>
<tbody>
<tr>
<td>XREF</td>
<td>XREFA</td>
<td>I:\WORK\TEST\XREFA.DWG -&gt; K:\TEST\XREFA.DWG</td>
</tr>
<tr>
<td>SHAPE</td>
<td>I:\WORK\TEST\ES.SHX -&gt; K:\TEST\ES.SHX</td>
<td></td>
</tr>
<tr>
<td>IMAGE</td>
<td>THATCH</td>
<td>I:\WORK\TEST\BRICK2.BMP -&gt; K:\TEST\BRICK2.BMP</td>
</tr>
<tr>
<td>RTEXT</td>
<td>I:\work\TEST\57072_b.txt -&gt; K:\TEST\57072_B.TXT</td>
<td></td>
</tr>
</tbody>
</table>

1 style/shape records modified.
1 image references modified.
1 xrefs modified.
1 rtext objects modified.
Changes to some externally referenced objects may be temporary.

**Usage Example 2:**
Strip all hard-coded paths

Command: REDIR
Current REDIRMODE: Styles,Xrefs,Images,Rtext
Find and replace directory names
Enter old directory (use '*' for all), or ? <options>: Enter *

Replace "**" with: Press ENTER

Searching for old dir: *
in order to replace it with:

```
XREF   XREFA            K:\TEST\XREFA.DWG -> XREFA.DWG
SHAPE  K:\TEST\ES.SHX  -> ES.SHX
IMAGE  THATCH          K:\TEST\BRICK2.BMP -> BRICK2.BMP
RTEXT  K:\TEST\57072_B.TXT -> 57072_B.TXT
```

1 style/shape records modified.
1 image references modified.
1 xrefs modified.
1 rtext objects modified.
Changes to some externally referenced objects may be temporary.
Command: PLT2DWG
Enter the plot file <Drawing>: Specify the file name or ~ for File d
REDIRMODE Dialog Box Options

Command: -REDIRMODE
Current REDIRMODE: Styles,Xrefs,Images,Rtext
Replace directories in Xrefs,Styles,Images,Rtext. <current>: Specify the object type(s)

Usage Example:
Set REDIRMODE so that the REDIR command acts only on xrefs and styles:

Command: REDIRMODE
Current REDIRMODE: Styles,Xrefs,Images,Rtext
Replace directories in Xrefs,Styles,Images,Rtext. <current>: xref,st
Current REDIRMODE: Styles,Xrefs


Type SHOWURLS or select the menu item to display the Showurls dialog box. Highlight a single URL to show URL, or one or more URL(s) to edit or replace.
SHOWURLS Dialog Box Options
You can create aliases for AutoCAD commands or DOS executable commands.
Options

There are two tabbed sections: Command Aliases and Shell Commands. Options common to both sections include

DOS Command: alias
Command: MKLTYPE
Select a ".LIN" file: <C:\Express\Drawings\acad.lin>: Enter a linetype file name and directory path
Enter linetype name: Specify a linetype name
Linetype description: Specify a linetype description
Starting point for line definition: Specify a starting point
Ending point for line definition: Specify an ending point
Select objects: Select a line, polyline, lwpolyline, point, shape, and/or text object

A linetype is a repeating series of dashes, dots, shapes, and text objects. With the standard AutoCAD PLINE, LINE, POINT, TEXT, and SHAPE commands, draw an approximation of your linetype; then select the objects at the Select Objects prompt in MKLTYPE.

Usage Example:
Create a dashed cold water linetype.

After specifying the file name, linetype name and description, select points to show the start points and end points for line definition:

Specify the line and text objects to be used to define the linetype:

The linetype is loaded and is ready to use:
Command: MKSHAPE
Select a ".SHP" file: <drawing.shp>: Enter a shape file name where you would like to save your shape
Reading shape file: drawing.shp...Done.
Enter the name of the shape: Specify the shape name
Enter resolution <124>: Enter a number for the resolution value (see Resolution Values below.)
Insertion base point: Specify a point
Select objects: Select line, polyline, lwpolyline, arc, circle, and/or ellipse objects
(for definition purposes, the selected objects are treated as though they are of a continuous linetype)

Determining geometry extents...Done.
Building coord lists...Done.
Formatting coords...Done.
Writing new shape...Done.

Compiling shape/font description file
Compilation successful. Output file C:\Express\drawings\drawing.shx contains 152 bytes.

Use the SHAPE command to place shapes in your drawing.

Shapes display relatively simple geometry quickly. The size of a single shape is limited to 2200 bytes of data. If you create a shape that exceeds that limit, MKSHAPE creates complex geometry shapes to form a single shape. In the SHAPE command, if you use the ? option and enter *, those "sub-shapes" are listed on screen.

Neither points nor zero-length line segments are used by MKSHAPE. However, you can get the effect of a point by drawing a very short line segment and using a resolution that is high enough to prevent the segment from being converted to a 0 length. If the short segment does not appear in the shape that results, try again, using a higher resolution value or a longer line segment.

Resolution Values

Large resolution values create shapes that are more accurate representations of
the selected geometry. However, large resolution values tend to create shapes that contain more coordinate data and place a higher processing burden on AutoCAD during regens. Conversely, small resolution values create less accurate shapes that are more quickly processed by AutoCAD during regens. You can experiment with different resolution values to find the balance of performance and accuracy that best fits your drafting needs. Note that the resolution value is rounded off to the nearest factor of 8 and cannot exceed 32,767. The following graphic shows how the resolution factor affects the resulting shape's appearance:

![Resolution Factor Example](image)

**Usage Example:**
Create a switch shape.

*After specifying the file name and shape name, select the insertion point for the shape:*

![Insertion Point](image)

*Next, select the objects to be used to define the shape:*

![Object Selection](image)
The shape can now be placed using the SHAPE command:
Sets the display size for the aperture, in pixels. The aperture is the selection tool used in drawing commands.
SYSVDLG Dialog Box Options

No changes are saved to the drawing until you choose OK. All pending changes can be discarded with the Cancel button. SYSVDLG will validate new system variable values as they are changed.
Command: XDATA
Select object: *Use an object selection method*
Application name: *Specify application name*
[3Real(DIR/DISP/DIST/Hand/Int/LAyer/LOng/Pos/Real/SCale/STr/eXit]
<eXit>: *Enter an option*
Options

**Usage Example:**
Attach piping data to a line

Select object/<None>: *Select a line*
Enter Application name: *Enter Myapp555-1212*
MYAPP555-1212 new application.
3Real/DIR/DISP/DIST/Hand/Int/Layer/Long/Pos/Real/Scale/Str/<eXit>:
*Enter Str*
ASCII string: *Enter Sewer*
3Real/DIR/DISP/DIST/Hand/Int/Layer/Long/Pos/Real/Scale/Str/<eXit>:
*Enter Int*
16-bit integer: *Enter 4*
3Real/DIR/DISP/DIST/Hand/Int/Layer/Long/Pos/Real/Scale/Str/<eXit>:
<ENTER> *Press ENTER*
New xdata appended.
Select object: *Use an object selection method*
Application name <**: Specify an application name or press ENTER for all applications**
Usage Example:

1. On the command line, type acadinfo.
2. In the Select File Name dialog box, save acadinfo.txt to a location on your hard drive.
3. View acadinfo.txt to find information about the setup and installation of AutoCAD. For example, you can view:
   - General Information (user name, computer name, and platform)
   - File loading information (such as a list of currently loaded ARX applications. Useful for troubleshooting systems)
   - System variable settings (displays a list of system variables. Useful for comparing data on two systems)
Command: BCOUNT
Press Enter to select all or...
Select objects: Use an object selection method for single or multiple blocks

**Sample output:**

<table>
<thead>
<tr>
<th>Block</th>
<th>Count</th>
</tr>
</thead>
<tbody>
<tr>
<td>CHAIR</td>
<td>10</td>
</tr>
<tr>
<td>TABLE</td>
<td>9</td>
</tr>
<tr>
<td>DESK</td>
<td>4</td>
</tr>
</tbody>
</table>
Command: BLOCK?
Enter block name <Return to select>: \textit{Press ENTER to select an object}
Select a block: \textit{Use an object selection method}
Enter an entity type <Return for all>: \textit{Press ENTER to list all block definition objects}

\textbf{Usage Example:}

Enter a block name or select an inserted block, and then specify an object type (such as line or text) to search for. Or accept the default to list all objects in the definition.

Block name/<Return to select>: Enter \textit{furn01}
Enter an entity type <Return for all>: Enter \textit{line}
Press ESC to exit or any key to continue.

\begin{verbatim}
((0 . "BLOCK") (2 . "FURN01") (70 . 0) (10 0.0 0.0 0.0) (-2 . <Entity name: 1f30c10>) ((-1 . <Entity name: 1f30c28>) (0 . "LINE") (5 . "F5") (100 . "AcDbEntity") (67 . 0) (8 . "0") (100 . "AcDbLine") (10 20.0346 22.2133 0.0) (11 4.19456 22.2133 0.0) (210 0.0 0.0 1.0))
\end{verbatim}
DWGLOG is useful in network situations where people are contending for the same drawing files. If you try to open a drawing that someone else is using, AutoCAD indicates that someone else has the file open, but does not identify the person. If all users on your network have logging turned on, you can look at the DWH file to find out who has the file open.

Command: DWGLOG
If you use DWGLOG when logging is turned off, the following message appears:

Logging for drawing files has been turned on.
If you use DWGLOG when logging is on, the following prompt appears:

You are about to turn off logging for drawing files.
Do you wish to continue? [Yes/No] :y
Logging for drawing files has been turned off.
Type 'dwglog' to turn logging back on.

When logging is on and someone accesses a drawing file, that file's drawing history (DWH) file is updated. If a DWH file does not exist for that drawing, DWGLOG creates one in the directory where the drawing file resides. Actions that are logged include opening a drawing, closing a drawing, and attaching a drawing as an xref. Following is a sample DWH file:

Opened by Penelope on MACHINE1 at 18:14:09 on 7/10/1997
Closed by Penelope on MACHINE1 at 18:18:44 on 7/10/1997
Attached by Bartholomew on MACHINE2 at 07:59:58 on 7/11/1997
The timer can be turned on and off and can be reset. A timeout option suspends timing after a specified period of inactivity.

Command: EDITTIME
EditTime settings: Status=OFF Timeout=10 min.
Active=0.0000 hr.
Enter an option [Reset/Timeout/ON/OFF]: Enter desired option
EDITTIME Options
LISP access

EDITTIME-related data is stored per drawing, in the Named Object Dictionary. The:

(acet-getvar ...)

and

(acet-setvar ...)

functions (available in the acetutil.fas module) provide access to the BNS_EDITTIME_TOTAL "profile variable". If EDITTIME has been enabled on a drawing, you can use:

(acet-getvar '("BNS_EDITTIME_TOTAL"))

to extract the elapsed time (up to the opening of the current editing session) from the current drawing as a Julian time value (a REAL value, containing the number of 24-hour days the drawing has been in "active" use).

The following LISP functions are exported by edittime.arx:
Command: EXPRESSMENU
Perform a full rebuild of the Express Tools menu before loading it? : Enter Yes or No
EXPRESSTOOLS places the *Express* directory on the search path, and loads and places the Express menu on the menu bar. You can use this command to make the AutoCAD Express Tools available in the current profile if they are not yet available. The AutoCAD Express Tools can be installed with AutoCAD, or you can add them later using Add or Remove Programs on the Control Panel.

Once the EXPRESSTOOLS command is used, the AutoCAD Express Tools libraries are loaded when AutoCAD is launched. This increases the load time for AutoCAD but reduces the delay when you use one of the tools for the first time.
Command: EXTRIM
Pick a POLYLINE, LINE, CIRCLE, ARC, ELLIPSE, IMAGE or TEXT for cutting edge...
Select objects: Select an object to define the cutting edge
Pick the side to trim on: Select a point inside or outside the object (objects are trimmed to objects cutting edge)

**Usage Example:**
Draw several overlapping circles with intersecting lines. Specify an internal point for location of the trim.

*Select a circle to define the cutting edge:*

*Specify a point inside the circle:*

*Objects trimmed to the edge of the circle:***
Command: GATTE
Select block or attribute [Block name]: Use an object selection method, or enter B for the block name
Known tag names for block: MDLN.
Select attribute or type attribute name: Use an object selection method or enter the attribute tag name
Number of inserts in drawing = 2 Process all of them? [Yes/No] <Yes>: If no, select the block inserts to process

**Usage Example:**
Insert several of the same attributed blocks into a drawing. Specifying a new value for one of the attributes replaces all values with the same tag name.

*Blocks with the same attributes:*

```
1 A-1

Select a single attribute to edit:
```

```
1 A-1

Value for all attributes edited globally:
```

```
1 B-2
```
Command: IMAGEAPP
Raster editing application or . for system default <.>: Enter the file name

See also: IMAGEEDIT
Command: DATE
Sun 1997/3/9 10:36:23.349
AutoLISP functions included in JULIAN.LSP:

Usage Example:
Use the DATE command to display the current date/time in this format: Day YYYY/M/D HH:MM:SS.msec. DATE uses (JTOD) to convert to calendar format, and edits the date/time from there. The day of the week is calculated with (JTOW).
Command: LSP
Enter an option [?/Commands/Functions/Variables/Load]: Specify an option
Options
(defun torus (/ cen1 cem2 r r2) (setq cem2 (point "Specify center point of torus: ")) (setq r2 (read 0.1)) (let ((cen1 (torus1 cem2))) (setq cen2 (+ cen1 r2)) (when (point-in-circle cem2 cen1 r2) (setq r2 (point "Specify radius of torus or [Diameter]: ")) (if (equal (read) 1) (setq r2 (point "Specify diameter of torus: "))) (when (point-in-circle cen2 cen1 r2) (display "Diameter cannot exceed torus radius: "))))
LSPSURF Dialog Box Options
Command: MPEDIT
Select objects: Use an object selection method
Convert Lines and Arcs to polylines? [Yes/No] <Yes>: If no, selection set ignores lines and arcs.
Enter an option [Open/Close/Join/Width/Fit/Spline/Decurve/Ltype gen/eXit]
<eXit>: Enter an option

Usage Example:
Draw a circle with overlapping lines and polylines with varying widths; then edit their widths.

Plines with varying widths:

```
circle and lines

polyline (width = 0)

polyline (width = 0.025)
```

Plines with the same width:

```

```
Either type SSX at the Command prompt to create a "previous" selection set or type (SSX) in response to any Select Objects prompt.

SSX returns a selection set that is either exactly like a selected object or, if you adjust the filter list, very similar to that object.

Command: SSX
Select object/<None>: Use an object selection method to select a template object, or press ENTER for none
>>Block name/Color/Entity/Flag/Layer/LType/Pick/Style/Thickness/Vector:
Filter Options

Usage Example:
Select a line to use as a template object, but remove the layer filter.

Command: SSX
Select object/<None>: Select a line
Current Filter: ((0 . "LINE") (8 . "0") (39 . 2.0) (62 . 1) (210 0.0 0.0 1.0))
Enter filter option [Block name/Color/Entity/Flag/Layer/LineStyle/Pick/Style/Thickness/Vector]: layer
>>Layer name to add/<RETURN to remove>: Press ENTER to remove layer filter
Current Filter: ((0 . "LINE") (39 . 2.0) (62 . 1) (210 0.0 0.0 1.0))
Enter filter option [Block name/Color/Entity/Flag/Layer/LineStyle/Pick/Style/Thickness/Vector]: Press ENTER
To select objects that match the filter, enter "previous" at the next Select Objects prompt.
Command: TFRAMES
IMAGE/WIPEOUT frames are toggled OFF.
or
IMAGE/WIPEOUT frames are toggled ON.

**Usage Example:**
Use TFRAMES to hide frames of a Wipeout and an image.

*Frames for image and Wipeout turned on:*

![Image with frames on]

*Frames turned off:*

![Image with frames off]