Contents

Introduction

Basic Concepts

A GammaCAD drawing is composed of different drawing elements. *Drawing elements* are lines, arcs, circles, points, text, symbols, polylines and filled areas. To create one of these drawing elements, you select it from the *Draw* menu or select its icon from the command tool bar. You must then specify where the drawing element is to be placed in the drawing area. See the Parts of the Program Window section to locate the drawing area, command tool bar and other parts of the GammaCAD window. When specifying where a drawing element is to be located, you use drawing coordinates. See the Drawing Coordinate System section for more information. For a description of the different ways in which you can specify these coordinates within GammaCAD, see the section on Specifying Positions .

Getting Started

If you are not familiar with Windows, you should go through the "Getting Started" section of your Windows manual first.   There is also an online tutorial in the *Main* group of Program Manager.   Double click on the *Windows Tutorial* icon to run it.   If you are not familiar with how to use this help facility, select the *How To Use Help* option on the *Help* menu above.

If you have never used a CAD program, you should go through the tutorials first.   Run the GammaCAD program and select the *Help* menu.   Click on the *Tutorials* option and go through each tutorial in order.   These tutorials will teach you the basics of how to use the program.

If you are familiar with CAD and Windows, you can probably jump right in and start using the program and use the Command Reference section, as needed, to learn about specific commands.

If you are upgrading from a previous version of this program, you will probably want to start with the section, What's New in this Release.

See also:
Help Menu

Creating Drawing Elements

The commands on the *Draw* menu are used to create elements in your drawing.   The following drawing elements can be created in GammaCAD:

Lines
Arcs
Circles
Points
Text
Symbols
Dimensioning
Polylines
Rectangles
Fill / Hatch

Stop

Using Element Snaps

Element snaps allow you to specify a position in the drawing area when a command prompts you to enter a point. This method allows you to specify positions based on elements already in the drawing. For example, if you want a line to start at the exact same position as the end of another line in your drawing, you would use the *End* element snap. To use element snaps, simply choose the desired mode using the *Snaps* pull down menu.

The *Snaps* pull down menu or the snaps tool bar is used to set the current snap mode. That mode remains in effect until another is set. The current snap mode is denoted on the snaps menu by a check mark (*None* in the figure below)
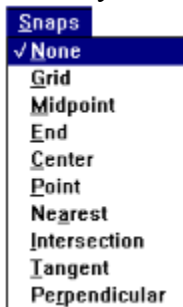
Snaps Menu

and it is denoted by the depressed button on the snap tool bar (*NONE* below).

Snaps Tool Bar

When the program is not prompting you for a point, the cursor is the standard Windows arrow cursor. If the current snap mode is *None* or *Grid* and the program is prompting you for a point, the cursor is a plus ($+$) in the drawing area. Otherwise, the program will display a box cursor (□) when it is prompting you for a point.

You can choose an element snap before or during a command. When a position is selected and a snap mode other than *None* or *Grid* is set, the program will look to see if an element is within the cursor box. If one is, the program will determine the proper coordinates to use for the point from this element using the current snap mode. The program tries to snap to the first element it finds within the box. If it cannot snap to this element it uses the center of the box as the input point. When you are using element snaps, you should make sure only one element lies within the cursor box, so you are certain you are snapping to the proper element. Use the *Zoom*

Window command to enlarge the area around the element you are snapping to, if necessary, to avoid getting other elements in your cursor box.

For example, if the box cursor is placed as in the figure below and the current snap mode is set to *End*, the program will use the coordinates of the closest end of the line as input.

Element Snap Pick

In this case the program snapped to the top of the line and that point is used as the start of a new line as shown below.
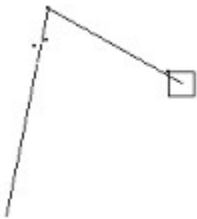
Element Snap Result

The *Grid* snap mode is different than the other modes in that it does not snap to an element in the drawing.   When the program is in grid snap mode and you pick a point using the mouse, the program will snap to the closest grid point and use that for the coordinates of the point you are inputting.   The program will snap to the nearest grid point even if the grid is not currently displayed.   Use the *Options/Grid snap parameters* command to turn the display of the grid on and off and control the spacing of the grid points.

The element snaps which are available within this program are:

NONE None   no element snaps are used.

Grid   snap to the closest grid point.

Midpoint   snap the the middle of a line or arc.

End   snap to the center of an arc or circle.

Point   snap to the position of a point element.

Nearest   snap to the closest position on a line, arc or circle.

Intersection   snap to the intersection of two lines or a line and an arc or circle.

Tangent   snap to the tangent point on an arc or circle from the last point entered.

Perpendicular   snap to the perpendicular point on a line, arc or circle from the last point entered.

The table below summarizes which element snap modes can be used for each drawing element. If an element does not support a particular snap mode, there is no *X* in the column for the element. If you try to snap to an element using a nonsupported snap mode, no snap position is determined and the position of the center of the cursor box is used.

	Line	Arc	Circle	Point	Text	Insert	Pline	Fill
Grid	x	x	x	x	x	x	x	x
Midpoint	x	x					x	
End	x	x					x	
Center		x	x					
Point				x				
Nearest	x	x	x				x	
Intersection	x	x	x					
Tangent		x	x					
Perpendicular	x	x	x				x	

Element Snaps

See also:
Specifying Positions

Setting Drawing Properties

The following properties are stored in every drawing.   When a new drawing is created, these properties are set to a default value.   You can modify this default value by creating a prototype drawing.   See the section on creating a prototype drawing for more information.

Line Properties
Text Properties
Point Properties
Fill / Hatch Properties
Dimension Properties
Layers
Drawing Base Point
Grid Snap Parameters
Number Format
Display Page
Display DXF Elements

The color, type and size buttons are short cuts for setting the color, type and size for the current drawing command.   For example, if you choose the *Draw/Line* command and click on the color button , the color dialog will be displayed and you can enter the color for the next line.   See the following sections for more information.

Color Button
Type Button
Size Button

File Operations

Using DXF Files

Drawing Interchange Files (*DXF*) are used to transfer graphical information from one program to another.   This format is defined by Autodesk and is used by AutoCAD to import and export drawing information.   GammaCAD can also read (import) and write (export) DXF files.   This means you can transfer drawings to GammaCAD from AutoCAD and any other program which supports DXF output.   It also means you can transfer drawings from GammaCAD to AutoCAD and any other program which supports DXF input.

The GammaCAD DXF transfer is designed to be a *loss less* transfer.   This means all the information in the DXF file is stored in the GammaCAD drawing on import and then written to the DXF file on export.   In other words, if you import a DXF file into GammaCAD and then immediately export the drawing to a new DXF file, the new DXF file will contain all the information present in the original (although not necessarily in the same order).   Not all programs which support DXF transfer will preserve your drawing in this manner and the DXF import / export process will introduce unwanted changes in your drawing.   For example, a line type might be different, a color changed or the z-coordinates lost.

What this means to you is GammaCAD can be used to edit drawings created in another program (e.g. AutoCAD) and then imported back into that program without introducing any unwanted changes in the drawing.   Of course, any editing you do to the drawing in GammaCAD will be present when you import the drawing back into the original program.   This allows you to use GammaCAD to supplement a more expensive CAD system.   What this does not mean, is that GammaCAD will display the drawing exactly as it was in the original program.   GammaCAD does not support all the types of entities and entity properties possible in a DXF file but it does preserve them in the GammaCAD drawing.

The GammaCAD DXF transfer is designed for AutoCAD release 11 or later.

See Also:
Importing DXF Files
Exporting DXF Files

Printing / Plotting a Drawing

The *File/Print/plot* command allows you to make a hard copy of your drawing.   In GammaCAD, printing and plotting refer to the same thing, sending the current drawing to a Windows print device.   The device may be a printer or a plotter.   Before plotting the drawing, you should position the drawing on the printer page.   The *File/Page setup* commands are used for this.   After setting up the page, choose the *File/Print/plot* command.   This will bring up the print dialog which allows you to set certain parameters for this printout.
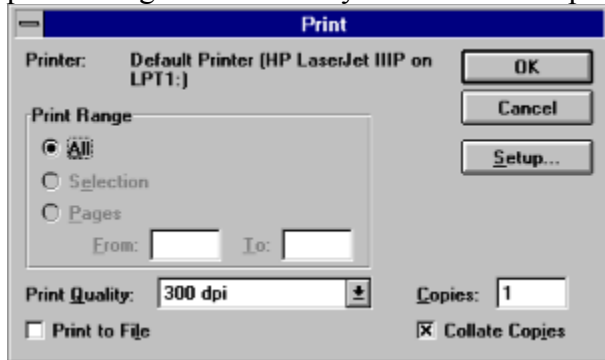
Print Dialog

Not all printers support all the options in this dialog.   If the printer does not support an option then it will be grayed out.   The *Print Range* will always be set to *All* since GammaCAD prints the drawing to one page only.   It will not divide a drawing into multiple pages if it is too large to fit on one.   You cannot specify a page range.   Pressing the *Setup* button brings up the print setup dialog.   This dialog allows you to select a printer and set the parameters for that printer. Pressing the *OK* or *Cancel* button in the print setup dialog returns you to the print dialog.

After the *OK* button is pressed in the print dialog, another dialog box is displayed.   This dialog is displayed while the plot is being created.   This dialog has one button, *Cancel*.   Pressing this button will cause the plot operation to be canceled.   If you want to create a plot, do **not** click on this button or press Enter.   If this button is not pressed, the plot is created and sent to the Print Manager.   For more information on the Print Manager see your Windows manual.   Print Manager handles the print job from this point on, sending it to the designated printer when it is ready.
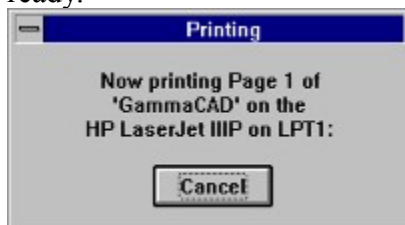
Print Cancel Dialog

Editing Drawing Elements

Once you have placed drawing elements in the drawing, you can modify them using the commands on the *Edit* pull down menu.   Some of the commands on the *Edit* menu operate on the currently selected drawing elements.   These commands are grouped together on the menu with the *Select* command.   The commands are:

> Cut
> Copy
> Delete
> Rotate
> Scale
> Change
> List

If no elements are selected in the drawing, these commands are grayed out and unselectable. For more information on selecting elements for these commands see the section on Selecting Drawing Elements.

The following edit operations can be performed on the current drawing element selection.
Moving Drawing Elements
Copying Drawing Elements
Moving / Copying Drawing Elements to Another Drawing
Deleting Drawing Elements
Rotating Drawing Elements
Scaling Drawing Elements
Changing Drawing Element Properties
Listing Drawing Element Properties

These edit commands do not require or use the current drawing element selection.
Trimming Lines, Arcs and Circles
Extending Lines and Arcs
Filleting Two Lines
Converting Polylines to Line Segments (so they can be broken, extended and filleted)
Changing Text Strings
Open/Close a Polyline
Modifying Symbols
Modifying the Dimension Arrow Head
Placing Drawings in Documents in Other Programs

After editing a drawing element, you can undo the changes with the *Undo* command.
Undoing Changes to the Drawing

Changing the View of the Drawing

The view commands are used to display different parts of your drawing.   They can be used to look at an area in detail or the drawing as a whole.   They do not change the actual coordinates or dimensions of the drawing elements.   They only allow you to view them in different ways.

Zooming In on an Area of the Drawing
Displaying the Entire Drawing
Restoring a Previous View
Zooming In and Out
Displaying the Page View
Displaying the Page and Entire Drawing
Moving the Drawing Right, Left, Up and Down (panning)
Redraw

Program Configuration

The *Options/Program configuration* commands allow you to set certain properties for the program as a whole and are not tied to a specific drawing. In other words, these properties remain the same for all drawings. The program configuration options are:

Selection Set Color
Cursor Increment
Arc/Circle Step Factor
Monochrome Print/Plot
Grid Dot Size
Grid Density

If you make changes to these options, they are stored in the file GAMMACAD.INI in your WINDOWS directory. Initially, the program uses default parameters for these options and does not use this file. To return to the default settings for these options, you can delete the appropriate line in the GAMMACAD.INI file.

Command Reference

Move the cursor over the desired menu item and click the left mouse button or select the menu from the list below.

| File | Edit | Draw | View | Snaps | Options | Help |

[File](#)
[Edit](#)
[Draw](#)
[View](#)
[Snaps](#)
[Options](#)
[Help](#)

Dimension Menu

Move the cursor over the desired menu item and click the left mouse button or select the item from the list below.

Leader
Vertical
Horizontal
Parallel
Radial

Leader
Vertical
Horizontal
Parallel
Radial

Using the Keyboard

Quick keys can be used as a short-cut instead of choosing the corresponding commands off of the menu.   The table below summarizes the quick keys which can be used in this program and what they do.   These are also referred to as keyboard accelerators or accelerator keys.

Key	Description
Ctrl+S	Save the drawing to the current file.
Ctrl+Z	Undo the last edit operation or draw operation.
Ctrl+E	Select drawing elements.
Ctrl+X	Cut the current selection to the clipboard.
Ctrl+C	Copy the current selection to the clipboard.
Ctrl+V	Paste the contents of the clipboard.
Delete	Delete the current selection
Ctrl+H	Change properties of the current selection.
Ctrl+L	Draw a line.
Ctrl+A	Draw an arc.
Ctrl+I	Draw a circle.
Ctrl+P	Draw a point.
Ctrl+T	Draw a text string.
Ctrl+O	Load symbol library.
Ctrl+W	Zoom window.
Ctrl+R	Zoom previous.
F2	Brings up the coordinate input dialog when the program is prompting for a position.
F1	Brings up the Windows help facility and moves to the section on the current command (if any).
Esc	End the current command.
+	Set/unset *Add* selection mode.
-	Set/unset *Remove* selection mode.

To choose a command from the menu, press the Alt key and the underlined character of the main menu containing the command (e.g. Alt+F for the *File* menu).   Then press the underlined character of the desired command or highlight the command using the up and down arrow keys and press Enter.

Key	Description
Alt+F	Pulls down the file menu.
Alt+E	Pulls down the edit menu.
Alt+D	Pulls down the draw menu.
Alt+V	Pulls down the view menu
Alt+S	Pulls down the snaps menu
Alt+O	Pulls down the options menu
Alt+H	Pulls down the help menu

The arrow keys can be used to move the cursor around in the drawing area and the Enter key to select a point.

The following keys can be used in dialogs.

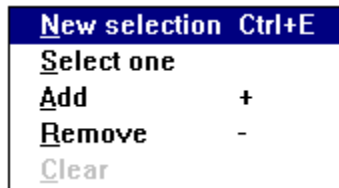| Key	Description
Tab	Move to the next field.
Shift+Tab	Move to the previous field.
Space Bar	To select a radio button or check box.
Enter	To press the highlighted button.

Other sections in this document describe any special key input required for a particular command or operation.   See your Windows manual for more information on Windows keyboard input.

Glossary

Accelerator Key	A short-cut key sequence which can be used instead of choosing a command from a menu.
CAD	Computer Aided Design.
Command	Any of the selections available on the pull down menu or iconic tool bars.
Current Selection	All the drawing elements which have been selected using the *Select* commands.
Cursor Increment	The number of pixels the cursor moves when a keyboard arrow key is pressed.
Cursor Step	Same as *Cursor Increment*.
Dialog Box	A Windows pop up menu which contains input fields, check boxes, buttons and so on.
Drawing Area Extents	The range of coordinates displayed in the drawing area.
Drawing Extents	The range of coordinates covered by all the drawing elements.
Drawing Element	One of the basic parts of a drawing (e.g. lines, arcs, circles, points, text,...).
Drawing Units	The dimensions used in the drawing area. They can be anything (e.g. feet,meters,etc.)
Font	A particular style of text.
K (Kilobyte)	1024 bytes of memory or disk space.
M (Meg or Megabyte)	1024 K bytes of memory or disk space.
Origin	The (0,0) point of an xy coordinate system.
Pan	To shift the view of the drawing within the drawing area left, right, up or down.
Plot Units	The dimensions used to specify sizes on the plot (inches or centimeters).
Quick Key	A short-cut key sequence which can be used instead of choosing a command from a menu.
Raster	In this manual, it denotes graphics defined by a series of points as opposed to *Vector*. For example, a line defined by a series of points.
Selected Elements	Same as *Current Selection*.
Selection Set	Same as *Current Selection*.
Text Angle	The rotation, counter-clockwise, of the baseline of a text string.
Text Justification	How the text is aligned. For example, at the left top corner.
Text Orientation	The rotation, counter-clockwise, of the individual characters in a text string.
Vector	In this manual, it denotes graphics defined by their end points as opposed to *Raster*. For example, a line is defined by 2 points.
Zooming	Enlarging or decreasing the view of the drawing.

Select Menu

Move the cursor over the desired menu item and click the left mouse button or select the item from the list below.

New selection	Ctrl+E
Select one	
Add	+
Remove	-
Clear	

New selection
Select one
Add   ＋
Remove   ▬
Clear

Program Description

GammaCAD is a full featured CAD (Computer Aided Design) program for Windows.   It provides a wide range of functionality in an easy to use package.

GammaCAD allows you to create drawings of real world objects in two dimensions using their actual measurements.   They can then be plotted at any scale you choose on any of the vast number of printers and plotters supported by Windows.   The biggest advantage of using a computer to create these drawings over conventional hand methods is the ease with which a computer based drawing can be edited and replotted at different scales.

CAD is used in many fields including: architecture, cartography, electronics and engineering.   It can be used to create house plans, maps of all kinds, printed circuit diagrams, and any kind of graph, chart, diagram or illustration.

This program allows you to create drawings using the following drawing elements: lines, arcs, circles, points, polylines, rectangles, filled and hatched areas .   You may enter exact coordinates for these elements, pick a position on the display or *snap* to a position on another element in the drawing.   For example, you can draw a line which starts exactly at the end of an existing line. GammaCAD also has predefined symbols you can place in your drawing for such things as floor plans, furniture, polygons, electronics, landscaping, computer equipment and civil engineering. The program also allows you to place dimensions on your finished drawing.

Once you have placed elements in your drawing, you can edit them to your liking.   They can be edited individually or as a group.   The elements can be deleted, moved, copied, scaled, rotated and their properties can be changed.   Certain elements can also be trimmed, extended and filleted.   These operations can be undone.   This allows you to experiment with the drawing.

Many properties of the drawing elements can be specified, such as line type (e.g. solid, dashed, dotted, etc.), line color and line width. Text properties include: font, size, color, angle and justification.   Point type, color and size can also be set.   Drawing elements can also be placed on different layers which can be selectively turned on and off.

The drawing can be zoomed and panned to look at different parts in more detail.   Zoom functions include: window, extents, previous and factor.

The drawings you create in GammaCAD can be copied to the Windows clipboard as a bitmap and pasted into your favorite word processor, paint program, or any program which lets you paste a bitmap.   This means you can place technical drawings in your documents.   GammaCAD also supports DXF input and output which allows you to transfer drawings to an from other CAD programs.

GammaCAD is designed to be powerful and easy to use.   The program contains on-line tutorials, examples, and context-sensitive help, to get you up and running as quickly as possible. A prompt area is displayed at the bottom of the window which instructs you how to perform each command.   A status area displays pertinent program information at a glance.   The program also employs easy to use visual dialog boxes.

Getting Help (Technical Support)

See the <u>Getting Started</u> section for information on how to get started with GammaCAD.

If you have a problem with this program and cannot find the answer in the documentation, help may be obtained by calling:

(970) 490-2928   voice/fax (start fax immediately after dialing)

or via E-mail at:

CompuServe: 73737,1721
AOL: GammaSoft
Internet: GammaSoft@aol.com
MSN: GammaSoft

or by writing to:

Gamma Software
P.O. Box 8191
Fort Collins, CO 80526
U.S.A.

We also have support areas at the following locations:
WWW: http://execpc.com/~gamma
MSN: "Go to" GammaSoft to visit our support area.

Please describe the problem in sufficient detail so it can be reproduced by us.

We are open 8am-5pm Mountain time (Mon-Fri) excluding holidays.

Support is currently free and unlimited.   We do reserve the right to change these policies in the future.

System Requirements

Windows 3.1 or later and a system capable of running it.
Hard disk with at least 3 megabytes of free disk space.
Floppy drive (if installing from floppy disk).
A Windows supported printer, mouse and color display are highly recommended.

Note - this program will **NOT** work with Windows 3.0 or previous versions of Windows.

Conventions Used in this Document

When two keys need to be pressed simultaneously to perform a given operation they are separated by a plus (+).   For example, Shift+Tab means to press the Shift key and hold it down while the Tab key is pressed and then releasing both.

When a menu option or other text from the program is mentioned it is *italicized*.   For example, the text *File/Open* instructs you to select the *File* pull down menu and then select the *Open* command on that menu.

Values which must be input by the user are enclosed in quotes.   Enter them without the quotes. For example, if the manual instructs you to enter "45" in a field within the program, enter the number without the quote marks.

The terms print and plot are interchangeable in this documentation.   They both refer to outputting a drawing to a Windows print device.   This device may be a printer or a plotter.

What's New in this Release

The following topics cover features which are new or have been modified in this release of the program.

Polyline Menu

Move the cursor over the desired menu item and click the left mouse button or select the item from the list below.
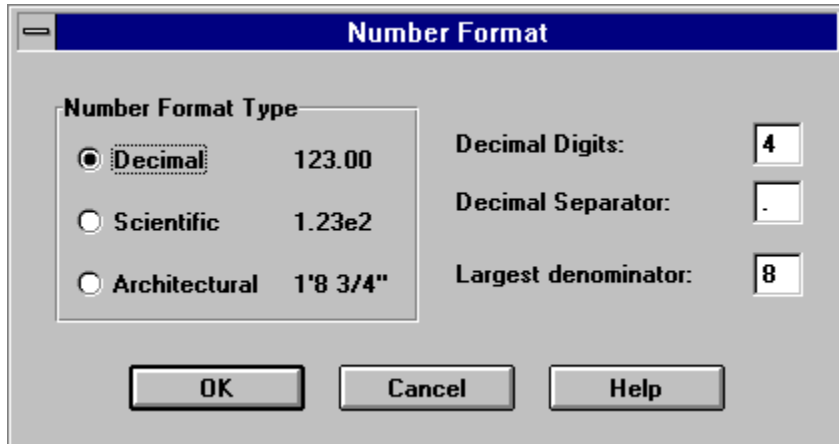

Open polyline
Closed polyline

Number Format

To specify how you want numbers displayed and how you want to enter numbers in the drawing, use the *Options/Number format* command.   The following dialog is displayed when you choose this command.

Number Format Dialog

For a description of the fields in this dialog, click on the desired field.   For example, for a description of the *Number Format Type* click the mouse anywhere inside the *Number Format Type* box.

When you create a new drawing, it is initially set to use decimal numbers.   The initial values of the *Decimal Digits* field and *Decimal Separator* field are taken from the settings in the international section (*intl*) of the Windows WIN.INI file.   These parameters can be configured using the *International* option in the Control Panel.   Click on the *Change* button in the *Number Format* section and set the *Decimal Separator* and *Decimal Digits* fields.   The values you set in the Control Panel will be used the next time a new drawing is created.

Once a drawing is created, the changes you make in the Control Panel have no affect.   This is because the values input in the number format dialog are saved with the drawing and are restored when you open the drawing.   This means if you have a prototype drawing, it will effectively override you Control Panel settings.

Only coordinate values and distances in drawing units are formatted using the format type specified in the dialog above.   Scale factors and angles are always displayed in decimal format. They must be entered in decimal format as well.   Very large scale factors are displayed in scientific format and can be entered that way as well.   Architectural format can never be used for scale factors or angles.
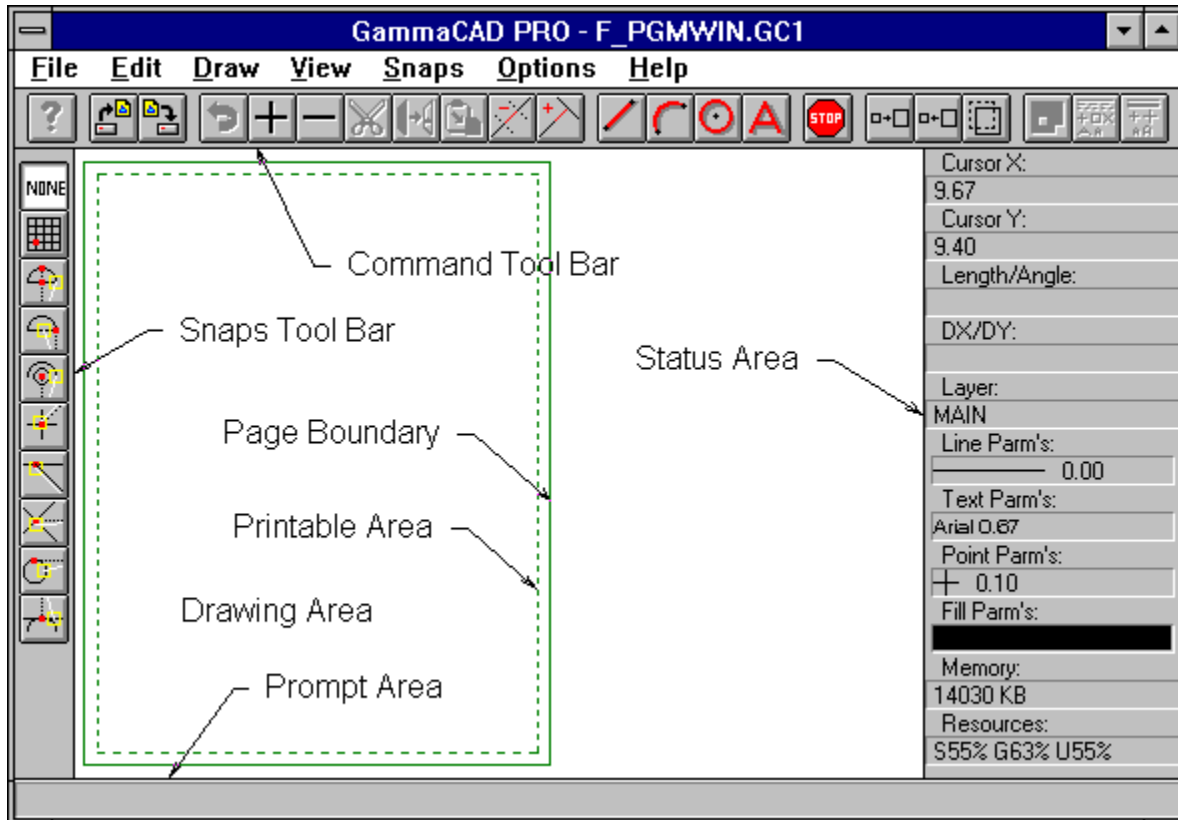
See Also:
Entering Decimal Numbers
Entering Scientific Numbers
Entering Architectural Numbers

Parts of the Program Window

The figure below shows the different areas within the program window.   These will be referenced throughout this manual.


Main Window

The main or pull down menu contains the program commands.   The command tool bar, directly below the main menu, provides a quick way of accessing certain commands.   The prompt area displays what the program currently wants you to enter.   The status area displays the current cursor xy coordinates and other pertinent program information.   The drawing area is where drawing elements are displayed and drawn.   It is the entire area in white in the above figure. The snap tool bar provides a quick way of changing the current snap mode.   The snap mode can also be set using the *Snaps* menu.   The page boundary and print area boundary display the position of the page based on the current printer and page setup.

Drawing Coordinate System

This program uses a standard 2 dimensional cartesian coordinate system.   This consists of 2 axes, x and y, which are perpendicular to each other.   The x axis increases to the right and the y axis increases upward.

Y

X
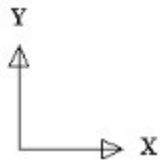
XY Coordinate System

Positions within this coordinate system are specified by giving their x and y distance from the origin.   The *origin* is where the x and y axes intersect at coordinates (0,0).   Coordinates are denoted by x and y values in parentheses within this manual (x,y).   For example, (10,20) designates a position 10 units to the right (x distance) and 20 units up (y distance) from the origin.

Y

(10,20)

20
15
10
5

X

(0,0)   5 10 15
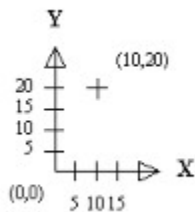
Coordinate Example

These coordinates can represent anything.   For example, they can be feet, meters, inches, centimeters, miles or kilometers.   They can even be in units other than distance.   For example, you may want the x axis to represent years and the y axis to represent sales in a graph.   **You** decide what the units are.   When you create your drawing, simply use those units when entering the coordinates for drawing elements.   Within this program these units are referred to as *drawing units*.   Before you print your drawing, you need to specify a <u>plot scale</u> which relates the drawing units you are using to inches or centimeters on the printer.

There are some limitations on the coordinates you can use in your drawing.   Values larger than 1e100 (10 to the 100th power) and smaller than -1e100 cannot be used.   Values within the range -1e-100 and 1e-100 also cannot be used.   The coordinate system must be *isotropic*.   This means distances in the x and y directions are equal.   In other words, a distance of 10 in the x direction is the same as a distance of 10 in the y direction.

The drawing area can be thought of as a window which displays a portion of the entire coordinate range.   Drawing elements which lie outside the current window are not displayed but are still part of the drawing.   The *drawing area extents* are the lower-left and upper-right coordinates of the current drawing area.
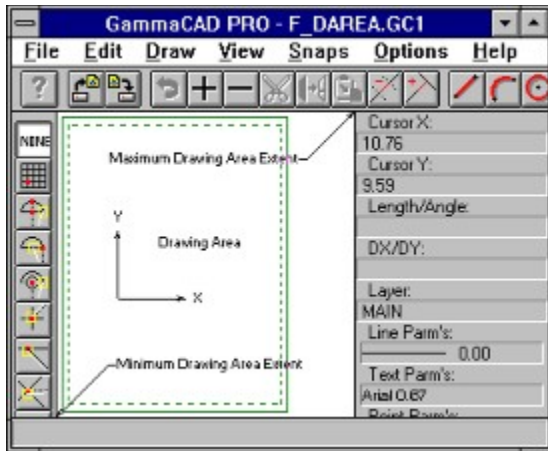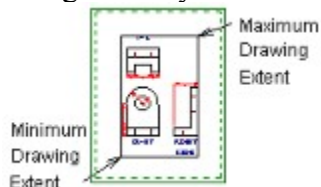
Drawing Area Extents

The view commands allow you to view different ranges of drawing coordinates in the drawing area by zooming in, zooming out, and panning, similar to the view finder of a camera. These commands change the drawing area extents and thus give you a different view of the drawing and display a different window onto the entire coordinate range. The view commands only change your view of the drawing. They do not change the coordinates of the drawing elements. You must use the edit commands to change the actual coordinates of drawing elements.

By default, when a new drawing is first loaded, the origin is at the lower-left corner of the page boundary. This can be changed using the page setup commands.
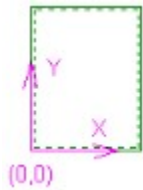
Coordinate System Origin

The *drawing extents* are the minimum and maximum xy coordinates of all the drawing elements. Note, this is different than the drawing area extents which is the extents of the rectangular drawing area in the program window. The drawing area extents are changed using the view commands. The drawing extents are determined by the elements in the drawing and only change when you add or edit drawing elements.

Drawing Extents

The *X* and *Y* fields of the status area give you the current coordinates of the cursor within the drawing area. The cursor can be moved with the mouse or keyboard arrow keys and the xy position in the status area will be updated as it is moved.

Specifying Positions

You need to specify positions (points) for many of the commands in this program.   A position is the (x,y) coordinates designating a location in the drawing.

There are three methods for specifying positions within this program:

1. Pick a position in the drawing area using the cursor.
2. Enter exact coordinates.
3. Snap to an existing drawing element.

Coordinate Input Examples

XY Input

This example illustrates how to draw a line from xy coordinates (1,1) to (4,1).   Select the *Draw/Line* command from the menu.   Click your right mouse button or press the F2 key.   This will bring up the coordinate input dialog.   Make sure the *XY input* radio button is chosen on the left.   Select the *X* field and enter "1."   Select the *Y* field and enter "1."   Press the *OK* button.   Click your right mouse button again or press F2.   Enter "4" in the *X* field and "1" in the *Y* field and press *OK*.   You have now drawn a horizontal line which starts at (1,1) and ends at (4,1).   If you do not see it, you may need to use the <u>view</u> commands to change the view so the line falls within the drawing area.   Choose the *View/Zoom extents* command and then the *View/Zoom factor* command and enter a factor of "0.5".

Polar Input

This example illustrates how to draw a vertical line with length of 4 which starts at the xy coordinate (4,1).   Select the *Draw/Line* command.   Click your right mouse button or press F2.   Make sure the *XY input* radio button is chosen to the left.   Select the *X* field and enter "4."   Select the *Y* field and enter "1."   Press the *OK* button.   Click your right mouse button or press F2 again.   Choose the *Polar input* radio button.   The prompts next to the two fields on the right will change to *Distance* and *Angle*.   Enter "4" in the *Distance* field and "90" in the *Angle* field and press *OK*.   You have now drawn a line which starts at (4,1) and has a length of 4 drawing units at an angle of 90 degrees.   Use the view commands, if necessary, to display the line in the drawing area.

XY Offset Input

This example illustrates how to draw a horizontal line which starts at (4,4) and extends 4 drawing units to the left and 0 drawing units up.   Choose the *Draw/Line* command.   Click your right mouse button or press F2.   Make sure the *XY input* radio button is selected to the left.   Select the *X* field and enter "4."   Select the *Y* field and enter "4."   Press the *OK* button.   Click your right mouse button or press F2 again.   Choose the *XY offset input* radio button.   The prompts next to the two fields on the right will change to *DX* and *DY*.   Enter "-4" in the *DX* field and "0" in the *DY* field and press *OK*.   You have now drawn a line which starts at (4,4) and extends left 4 drawing units and 0 drawing units up or down.   Use the view commands, if necessary, to display the line in the drawing area.

Lines

Lines are drawn by entering their start and end points.

Line Input

To Draw a line, all you have to do is choose *Line* from the *Draw* pull down menu or you can press Ctrl+L or press the draw line button on the tool bar.   The prompt area will display:

> Select first end point of line

After entering the first end point, the prompt area changes to:

> Select second end point of line

As you move the cursor, a line is drawn from the first point to the current cursor position.   This represents the line which would be added to the drawing if the current cursor position is used for the second end point.   When the second end point is specified, a line with the designated start and end points is added to the drawing.   It is displayed using the current line properties in effect.

The program continues to prompt you for a second point.   If you wish to draw another line which starts at the end of the previous one, select the second end point of the new line.

Choosing another operation from the main menu (except those on the *Snaps*, *Options*, or *Help* menus) ends line input.   You can also end line input by clicking on the stop button on the command tool bar or by pressing the Esc key.

See the section on Specifying Positions for more information on entering the line end points. See the section on Line Properties for more information on the different types of lines which can be drawn.

Arcs

Arcs are drawn by specifying their start, center and end points. They are always drawn in a counterclockwise direction.
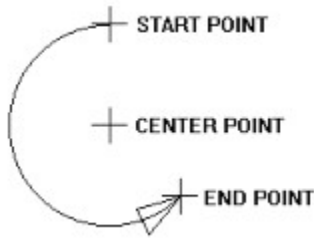
Arc Input

To Draw an arc, all you have to do is choose *Arc* from the *Draw* pull down menu or you can press Ctrl+A or press the draw arc button on the tool bar. The prompt area will display:

> Select first end point of arc

After entering the first end point, the prompt area changes to:

> Select center point of arc

After entering the center point, the prompt area changes to:

> Select second end point of arc

As you move the cursor, an arc is drawn from the first point to the current cursor position around the given center point in a counterclockwise direction. This represents the arc which would be added to the drawing if the current cursor position is used for the second end point. When the second end point is specified, an arc with the designated start and end points is added to the drawing. It is displayed using the current line properties in effect.

The program continues to prompt you for the first point of an arc. If you wish to draw another arc, you can enter it at this point.

Choosing another operation from the main menu (except those on the *Snaps*, *Options*, or *Help* menus) ends arc input. You can also end arc input by clicking on the stop button on the command tool bar or by pressing the Esc key.

Arcs are always drawn in a counterclockwise direction. This means the order in which you input the arc end points determines the interior angle of the arc. To get the complimentary arc to a given arc, simply input the end points in reverse order.
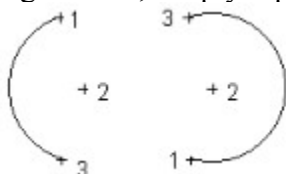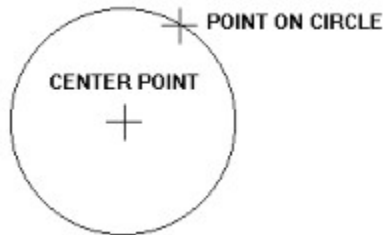
Arc Point Entry

See the section on <u>Specifying Positions</u> for more information on entering the arc points.   See the section on <u>Line Properties</u> for more information on the different types of arcs which can be drawn.

Circles

Circles are drawn by specifying their center point and a point anywhere on the circle.

Circle Input

To Draw a circle, all you have to do is choose *Circle* from the *Draw* pull down menu or you can press Ctrl+I or press the draw circle button on the tool bar. The prompt area will display:

Select center point of circle

After entering the center point, the prompt area changes to:

Select point on circle

As you move the cursor, a circle is drawn around the center point and through the current cursor position. This represents the circle which would be added to the drawing if the current cursor position were used. When the point on the circle is specified, a circle with the designated center and circumference point is added to the drawing. It is displayed using the current line properties in effect.

The program continues to prompt you for the center point of the next circle. If you wish to draw another circle, you can enter it at this point.

Choosing another operation from the main menu (except those on the *Snaps*, *Options*, or *Help* menus) ends circle input. You can also end circle input by clicking on the stop button on the command tool bar.

See the section on Specifying Positions for more information on entering the circle points. See the section on Line Properties for more information on the different types of circles which can be drawn.

Points

Points are drawn by specifying their position.

To Draw a point, all you have to do is choose *Point* from the *Draw* pull down menu or you can press Ctrl+P.   The prompt area will display:

      Select point

When a position is specified, a point is drawn using the current point properties in effect.

The program continues to prompt you for a point position.   If you wish to draw more points you can do so at this time.

Choosing another operation from the main menu (except those on the *Snaps*, *Options*, or *Help* menus) ends point input.   You can also end point input by clicking on the stop button ![stop] on the command tool bar or by pressing the Esc key.

See the section on Specifying Positions for more information on entering the point positions. See the section on Point Properties for more information on the different types of points which can be drawn.

Text

Text is drawn by specifying the text alignment point and entering the desired text string.
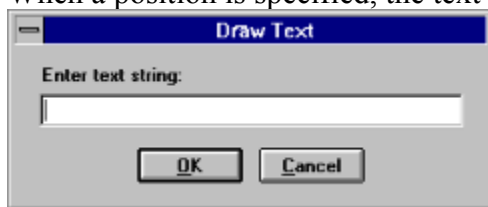
ALIGNMENT POINT

TEXT STRING

Text Input (Top-left justification)

To Draw text, all you have to do is choose *Text* from the *Draw* pull down menu or you can press Ctrl+T or press the draw text button **A** on the tool bar.   The prompt area will display:

Select text alignment point

When a position is specified, the text string input dialog is displayed.

Text Input Dialog

Enter the desired text and press the *OK* button.   The text is displayed using the current text properties in effect.

The program continues to prompt you for a text alignment position.   If you want to enter more text strings, do so at this time.

Choosing another operation from the main menu (except those on the *Snaps*, *Options*, or *Help* menus) ends text input.   You can also end text input by clicking on the stop button on the command tool bar or by pressing the Esc key.

See the section on Specifying Positions for more information on entering the text alignment points.   See the section on Text Properties for more information on the different types of text which can be drawn.

Symbols

The *Draw/Insert symbol* command is used to insert a GammaCAD drawing as a symbol in the current drawing.   Any GammaCAD drawing can be placed in another drawing as a symbol.   The insertion point, angle and scale factor must be specified.   There is no difference between a symbol file and any other GammaCAD drawing file.   Symbol files should be given a SYM extension instead of GC1.   This will help you separate the files you use as symbols from your drawing files.   Also, the *Insert symbol* and *Redefine symbol* commands default to listing files with a SYM extension.   If you use this extension, the files will be automatically listed for those commands.

To create your own symbols, choose the *New* option from the *File* menu, and draw the symbol like any other drawing.   To set the base point of the symbol, use the *Set Base Point* command on the *Options* menu.   This point will be matched up with the insertion point you select later when you insert the symbol in another drawing.   When you save the drawing, make sure you give it a SYM extension.   For example, if you are creating a symbol you want to name WHEEL, enter WHEEL.SYM as the file name.

When you want to place the symbol in another drawing, choose *Insert symbol* from the *Draw* pull down menu.   The *Open* dialog will be displayed and it will list all the symbol files (*.SYM) in the current directory.   Double-click on the symbol you wish to place in the drawing.   You can select any drawing file even if you did not give it a SYM extension.   To do this, type in the name of the desired file in the file name box.   You can list the drawing files with a GC1 extension by clicking on the down arrow next to the file types box and choosing *Drawing Files (*.GC1)*.

The prompt area will display:

> Select symbol insertion point

The base point of the symbol is aligned with the insertion point you enter.

When a position is specified, the rotation angle dialog is displayed.   Enter the desired rotation angle for the symbol in degrees counterclockwise and press the *OK* button.   Enter "0" for no rotation.   The scale dialog is displayed next.   Enter any value other than 0.   A value less than 1 will cause the symbol to be displayed smaller than its actual size (e.g. 0.5 will cause it to be half size).   A value greater than 1 will cause the symbol to be larger than its actual size (e.g. 2. will cause it to be double size).   A value less than 0 will cause the symbol to be mirrored across x and y axes centered at the insertion point.

After the angle and scale have been entered, the program will draw the symbol using the parameters you specified.   It then prompts you for the insertion point again.   You can place any number of the same symbol in the drawing, each with its own angle and scale.   Choosing another operation from the main menu (except those on the *Snaps*, *Options*, or *Help* menu) ends symbol input.   You can also end symbol input by clicking on the stop button on the command tool bar or by pressing the Esc key.

See the section on <u>Specifying Positions</u> for more information on entering the symbol insertion points.

<u>General Notes About Symbols</u>
When a symbol is inserted into a drawing for the first time, it is read from the file which contains the drawing elements making up the symbol.   These drawing elements are stored in the drawing only once.   They are defined separately from the main drawing and also from other symbols. Each symbol is given a name in the current drawing.   The name is the same as the file name from which it was read (except the file name extension is removed).   The next time you insert the same symbol into the drawing, the program will see it is already defined and it will not reread the symbol file.   Each insertion of a symbol in the drawing contains only the name of the symbol, its position, rotation angle and scale factor.
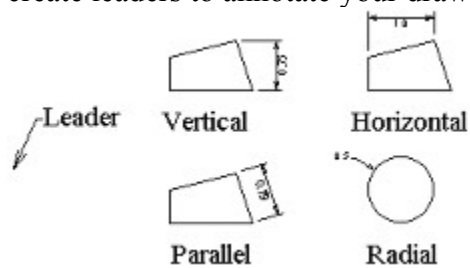
See the section <u>Modifying Symbols</u> for information on how to change drawing symbols.

See also:
<u>Symbol Libraries</u>

Dimensioning

Vertical, horizontal, parallel and radial dimensions can be created by this program.   You can also create leaders to annotate your drawing.

Dimension Types

The dimensioning commands use the current text font, so you must set the font you want to use for the dimension text before choosing the dimension command.   Use the *Options/Text properties/Text font* command to do this.   You should choose a font which allows baseline rotation (e.g. True Type fonts), since vertical and parallel dimension text is rotated.   The dimension commands also use the current line properties for the dimension and extension lines. Use the *Options/Line properties* commands to set the desired line settings before creating the dimension.   You should also set the desired dimension arrow size and dimension offset before dimensioning.

To create a dimension, choose *Dimension* from the *Draw* menu.   The dimension type sub menu will be displayed, select the type of dimension you want to create.

Leader
Vertical Dimension
Horizontal Dimension
Parallel Dimension
Radial Dimension

Polylines

Polylines are drawn just like regular lines by entering their start and end points.   The only difference is that all the line segments making up a polyline are treated as a unit.   When you select one segment of the polyline all the segments are selected.   GammaCAD allows you to draw two types of polylines: open and closed.   An open polyline starts and ends at two different positions.   A closed polyline ends at the same point as it started.

Open Polyline   Closed Polyline
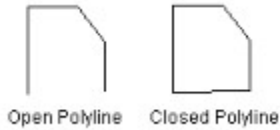
Open/Closed Polylines

If you specifically close a polyline by snapping to the start point when drawing the last segment, then the open/close state of the polyline has no effect; the polyline is always closed.   If you are drawing a closed polyline and the last point you enter is not the same as the start point, a segment is automatically drawn from the last point to the start point.

To Draw an open polyline choose the *Draw/Polyline/Open polyline* command.   To draw a closed polyline choose the *Draw/Polyline/Closed polyline* command.

Rectangles

To Draw a rectangle, all you have to do is choose *Rectangle* from the *Draw* pull down menu. The prompt area will display:

>   Select first corner of rectangle

After specifying the first end point, a box is drawn from that point to the current cursor position as you move the cursor.   This represents the rectangle which would be added to the drawing if the current cursor position was used for the second corner.   The prompt area changes to:

>   Select second corner of rectangle

When the second corner is specified, a rectangle with the designated corner points is added to the drawing.   It is displayed using the current line properties in effect.   The rectangle is actually a polyline element with four segments.   The program then prompts you for the first corner of the next rectangle.   You can add another rectangle to the drawing if you desire.

Choosing another operation from the main menu (except those on the *Snaps*, *Options*, or *Help* menus) ends rectangle input.   You can also end rectangle input by clicking on the stop button on the command tool bar or by pressing the Esc key.

See the section on Specifying Positions for more information on entering the rectangle end points.   See the section on Line Properties for more information on the different types of rectangles which can be drawn.

Fill / Hatch

To create a solid fill or cross-hatched area, you must first define the boundary of the area with a polyline.   Next, choose *Fill / Hatch* from the *Draw* pull down menu.   The mouse cursor will change to a box and the prompt area will display:

> Select a polyline to fill / hatch

Move the box cursor so one segment of the desired polyline passes through it and click the left mouse button or press Enter.   The area bounded by the polyline is filled using the current fill / hatch properties.   The program continues to prompt you to select a polyline to fill / hatch.   If you want to fill / hatch another area you may do so at this point.

Selecting another operation from the main menu (except those on the *Snaps*, *Options*, or *Help* menus) ends fill / hatch input.   You can also end fill / hatch input by clicking on the stop button on the command tool bar or by pressing the Esc key.

Line Width

To specify a line width, choose *Options/Line Properties/Line width* from the menu.   You can also select a line width using the size button on the tool bar ![STOP]. It is active only after you have chosen the line drawing command.   Enter the desired line width.   This value is specified in drawing units and its displayed width will vary depending on the current drawing area extents. Specifying a width of 0 will cause the line to be drawn as thin as possible (1 pixel wide) no matter what the drawing area extents.
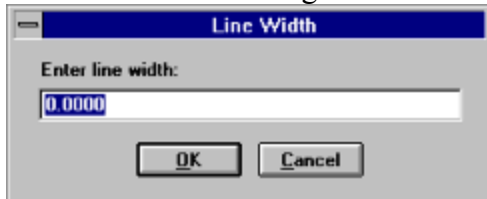
Line Width Dialog

One important limitation is only solid lines can have a width of more than one pixel.   If a line width is specified the line type will always be solid.

See also:
Line Properties

Line Type

To specify a line type, choose the *Options/Line properties/Line type* command from the menu.

You can also select a line type using the type button on the tool bar ![STOP].   It is active only after you have chosen the line drawing command.   To select a line type, click on the desired type (or use the arrow keys).   Click on the *OK* button (or press Enter) to use the selected line type.

Line Type Dialog

Only the solid line type can have a width.   The line width must be 0 to display any line type other than solid.

See also:
Line Properties

Set Base Point

The *Options/Set base point* command allows you to set the base point for any drawing.   After choosing this option, the prompt area will display:

Select base point for drawing

The base point of a drawing is initially set to (0,0).   You input this point like any other (see the section on <u>specifying positions</u> for more information).

The base point of a drawing is only used if the drawing is used as a symbol.   The base point you select will be aligned with the insertion point you specify when the drawing is inserted into another drawing as a symbol.   See the <u>*Draw/Insert symbol*</u> command.

Symbol Libraries

The *Draw/Load symbol library* command is used to load one of the GammaCAD symbol libraries.   You can choose the command from the *Draw* menu or press the Ctrl+O accelerator key.   The symbol library selection dialog is displayed and it lists the symbol libraries (.DLL) in the GammaCAD directory.   Choose the desired library by double-clicking on it or click on it once and press the *OK* button.   You can also use the keyboard up and down arrow keys to choose the desired library and then press Enter.
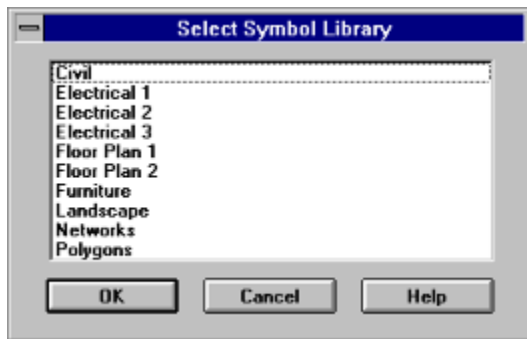
Symbol Library Selection Dialog

After selecting the desired library, the symbols in this library are displayed in the next dialog. Click on the symbol you want or use the right arrow key to highlight the desired symbol and press Enter.

The prompt area will display:

    Select symbol insertion point

The base point of the symbol is aligned with the insertion point you enter.   The base point of the symbol is denoted by the red dot on the button for the symbol in the symbol dialog.

Entering the rotation angle and scale factor are accomplished the same way as in the *Draw/Insert symbol* command.   You can continue to insert additional copies of this symbol by specifying their insertion point, rotation and scale factor.

To select another symbol from the symbol library dialog, position the mouse cursor over it and click the left mouse button.   To select another symbol using the keyboard, press the Tab key to move back to the current library dialog and highlight the desired symbol using the right arrow key and press Enter.   You will then be prompted for the insertion point, rotation and scale, as before.   To switch to a different symbol library using the mouse you can simply click on it. Using the keyboard, choose the *Draw/Load symbol library* command and select the desired symbol library.   If it is already loaded, it will be made the current symbol library.   If it is not loaded, it will be loaded and made the current symbol library.

To position the symbol library dialog with the mouse, click the left mouse button on the title bar and hold it down.   Drag the dialog to the desired position and release the left mouse button.   To position the dialog with the keyboard, first make sure it is the active window (i.e. the caption bar

is painted the active color).   Press the Tab key to activate it if it is not active.   Next, press the Alt key to bring up the system menu and choose *Move* from the menu.   Now press the arrow keys to move the dialog and press Enter when it is in the desired location.

To close a symbol dialog, double click on the system menu button or click on it once and choose *Close*.   Using the keyboard, first activate the symbol dialog and then press the Alt key and select *Close* from the menu.   You may also press Alt+F4 to close the dialog while it is active.

When using the keyboard and a symbol dialog is active, you must first press Tab to return to the main GammaCAD window before you can use any of the GammaCAD keys.   For example, to bring up a GammaCAD menu using an Alt key combination or to use one of the GammaCAD accelerator keys you must first press Tab if a symbol dialog is currently active.

Modifying Symbols

The drawing elements making up a symbol are treated as a unit once a symbol is inserted into a drawing. When you select one drawing element in a symbol, the entire symbol is selected. This is due to the fact the elements making up a symbol are stored in the drawing only once. Each insertion of a symbol only contains the symbol name, position, scale and rotation.

The *Change* command on the *Edit* menu has no affect on symbols. To change a symbol, you must load the symbol file into GammaCAD using the *File/Open* command. Symbol files are exactly the same as other drawing files. They are given a SYM extension merely as a file keeping aid. From the *Open* dialog, click on the arrow button to the right of the *List Files of Type* field (or press Alt+T and then the down arrow key). Choose *Symbol File (*.SYM)* from the list. The dialog will list the symbol files in the current directory. Select the desired symbol from the list. You may need to change to the GammaCAD directory.

Once the symbol file is loaded, you can use any of the program commands to add, change or delete drawing elements. Choose the *Save* command from the *File* menu when you are done editing the symbol. The next time you insert the symbol into a new drawing the new symbol will be used.

The changes will not affect drawings created previously containing the symbol unless you use the *Redefine Symbol* command on the *Edit* menu. When you choose this option, the *Open* dialog will be displayed. The symbol files in the current directory are displayed. Select the file which you modified and the program will delete the old definition of the symbol and read the new definition from the symbol file you selected.

The program will display the message: *Cannot find selected figure* if the symbol you selected does not exist in the drawing. The symbol name in the drawing is the name of the file it was originally inserted from. If you insert a symbol into a drawing, change the name of the symbol file, and then try to redefine it using the new name, the program will not be able to find it.

The *Redefine Symbol* command cannot be undone so you may want to save your drawing prior to redefining a symbol, then you can exit without saving the drawing if the command does not have the desired effect.

Text Font

To specify a font, choose *Options/Text properties/Text font* from the menu.   You can also select a text font using the type button on the tool bar **[STOP]**.   It is active only after you have selected the text drawing command.

The font name is the main property affecting the appearance of a text string.   It determines how different characters in the string are drawn.   Windows comes with several built in fonts.   With version 3.1 of Windows, True Type fonts were introduced.   It is recommended you use these fonts exclusively.   The True Type fonts work much better with this program because they are more scalable.   Raster fonts (e.g. Fixedsys) are not as scalable and may not appear at the proper size when you zoom in and out in your drawing or when you specify different plot scales.   There are also a couple of drawbacks to using the vector fonts which are not True Type.   One is the *Zoom Extents* command only uses the alignment point of vector fonts in its calculations.   This means part or all of a text string which uses a vector font may lay outside the drawing area even after a *Zoom Extents* command.   The second drawback is the alignment point must be included in your selection area in order to select a string using a vector font which is not True Type.

When you choose the *Text font* command the following dialog is displayed.
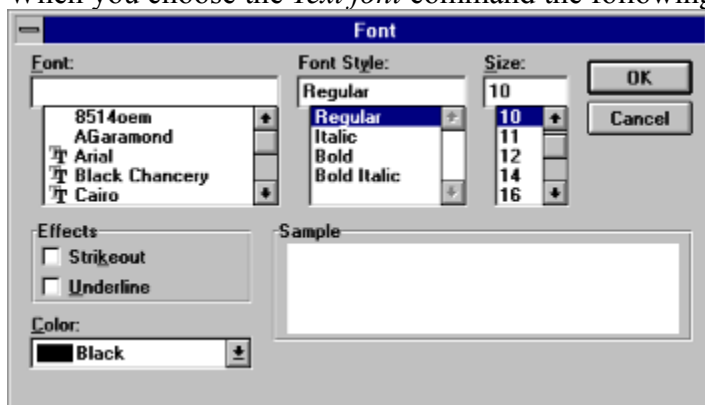

Font Dialog

To choose the font name, click on the desired font name from the list.   You can scroll the list up and down by clicking on the scroll arrows if the list is longer than the number displayed.   To select a font style, click on the desired style.   The font size, effects and color can be specified in the same way.   A sample of the currently chosen font is displayed in the lower right of the dialog.   When you have input the desired text properties, click on the *OK* button.

The parameters in this dialog are pretty self-explanatory with the possible exception of the *Size* parameter.   These sizes are in points.   A point is 1/72 of an inch.   This specifies the size of the text when plotted.   The size which appears on the display will vary depending on the currently specified plot scale and the current drawing area extents.   When you specify a text size, it is converted to the units you are using in the drawing area using the currently specified plot scale. The next time text is drawn, it will be drawn at that size.

For example, if you specified a 10 point font and your current plot scale is 1 inch equals 100 drawing units then the height, in drawing units, for the text would be 13.9 (10 X 1/72 X 100).   If

the y extent of the drawing area is 100, the text will be 13.9% as tall as the y extent of the drawing area. If the drawing is zoomed in so the y extent is 50, the text will be 27.8% of the drawing area height.
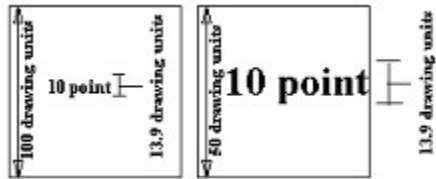
Text Example

This figure demonstrates how the current zoom factor (size of the drawing area) affects the size of the text. The size of the text is not changing (it remains 13.9) but the size relative to the size of the current drawing area is changing. It is important to remember the view commands do not change the drawing elements in any way, they simply change the view of the drawing elements. It is similar to the zoom on a camera. If you zoom in on an object with your camera it appears to be larger, but of course, the actual size of the object remains the same.

When you print/plot the text, it will always be plotted with a height of 13.9 drawing units no matter what the current drawing extents. The actual height of the text in plot units (inches/centimeters) on the page is determined by the print/plot scale.

To sum up, there are two factors which affect the size of the text on the display when you select a point size in the font dialog above. First, it is scaled by the current print/plot scale. That value is stored as the text height for the text element in the drawing. The second factor is the current zoom factor of the drawing area (drawing area size). The text is drawn at the stored height relative to the size of the current drawing area. There is only one factor affecting the size of the text when it is printed/plotted and that is the plot scale.

If you have trouble setting the text size using the text font dialog then you may want to use the *Options/Text properties/Text height* command. With this command you can specify the text height directly in drawing units and not have to take the current plot scale into account.

See also:
Text Properties

Line Properties

The *Options/Line properties* commands allow you to specify how lines, arcs, circles and polylines are drawn.   All lines have the following three properties:

      <u>Line Color</u>
   <u>Line Type</u>
      <u>Line Width</u>

These properties can be specified before or during the input of a drawing element.   Each layer in the drawing has its own set of line properties.   When you choose one of these options, you are changing the property for the current layer only.   The selected properties remain in effect for the layer until another selection is made.   Whenever a line, arc or circle is created, it is displayed using the current line color, type and width set for the current layer.   They will always be displayed this way unless the properties are changed.   See the section <u>Changing Drawing Element Properties</u> for more information.

Line Color

Lines can only be displayed in solid colors.   In other words, they cannot use *dithered* colors which use a pattern of different solid colors to produce the look of another color.   The number of solid colors available is dependent on the display device.
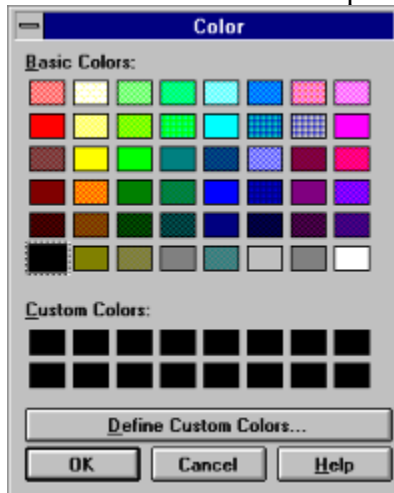
Line Color Dialog

To specify a line color, choose *Options/Line properties/Line color* from the menu.   You can also select a line color using the color button on the tool bar .   It is active only after you have chosen the line drawing command.   To make a selection from the color input dialog, simply click on the desired color (or highlight it using the arrow keys and press the spacebar).   Then press the *OK* button.   If a nonsolid color is chosen, the closest solid color is used instead.   You must also be careful not to select colors which map to the same color as the background since they will not be visible on the background.

See also:
Line Properties

Leader

To create a leader, you must select the position the leader will point to, then the end of the leader line, and finally enter the leader text.   The end of the leader arrow will fall exactly on the first point you select.   The second point marks the end of the leader line.   An extension line will be drawn from that point with a length equal to the amount specified by the *Dimension offset* command.   The leader text is placed at the end of this line.   It is always drawn horizontally.
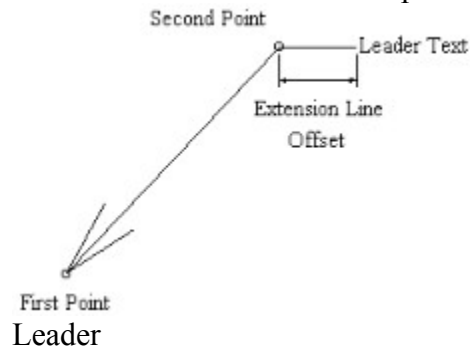
Leader

After choosing the leader option from the dimension menu, the program prompts you for the first end point.   Next, the program prompts for the second end point.   See the section on specifying positions for more information on ways of entering these two points.   The last input required to create a leader is the leader text.   Enter the text you wish to place at the end of the leader line and click on the *OK* button or press Enter.

See also:
Dimensioning

Text Height

To specify the height of text, choose *Options/Text properties/Text height* from the menu.   You can also set the text height using the size button on the tool bar ![STOP].   It is active only after you have selected the text drawing command.   Enter the desired text height in the dialog box which appears.   This value is specified in drawing units and the displayed height of the text will vary depending on the current drawing area extents.


Text Height Dialog

See also:
Text Properties

Linear Dimension

To create horizontal, vertical, or parallel dimensions, you must select the two extension line positions and the dimension line position.   Optionally, you can change the dimension text.   The extension lines will be offset from the points you pick by the extension line offset specified by the *Dimension Offset* command.   The dimension line will pass through the position you specify for the dimension line.
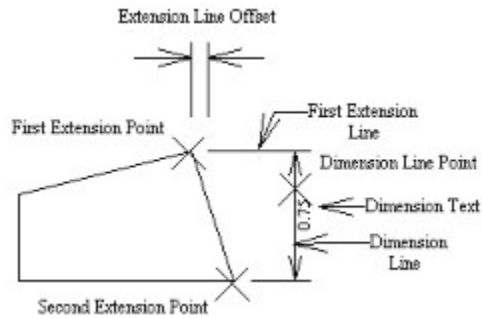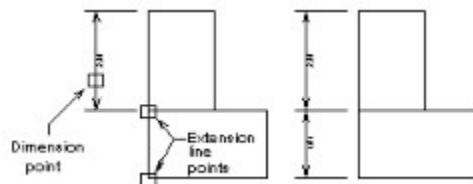

Vertical Dimension

After choosing the vertical, horizontal or parallel option from the dimension menu, the program prompts you for the first extension line point.   Next, the program prompts for the second extension line point.   Lastly, you are prompted for the dimension line position.   See the section on specifying positions for more information on ways of entering these three points.   The last input required to create these dimensions is to input the dimension text.   The string defaults to the distance between the two extension lines in drawing units.   It is formatted according to the number format set by the *Options/Number format* command.   You can change the text any way you want.   For example, you may want to add the proper units to the end of the string (e.g. feet or meters).   You may want to change the text entirely.   Once you have entered the desired text, click on the *OK* button or press Enter.

The program will prompt you for the first extension line point again.   You can create another dimension of the same type or end the command.

To line up consecutive dimensions, use the *Nearest* element snap when selecting the dimension line point and snap to the previous dimension line.
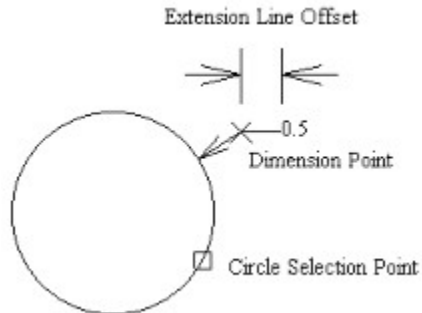

See also:
Dimensioning

Radial Dimension

To create a radial dimension, you must select an arc or circle and then the dimension position and optionally change the dimension text.   An extension line is drawn horizontally from the dimension point with a length equal to the amount specified by the *Dimension offset* command.
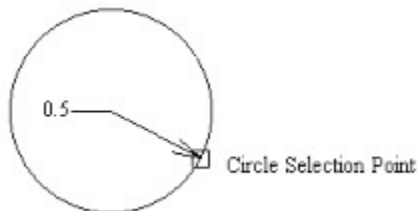
Radial Dimension Example 1

After choosing the radial option from the dimension menu, the program prompts you to select an arc or circle.   Place the box cursor so the arc or circle you want to dimension passes through it. Next, the program prompts for the dimension line point.   See the section on specifying positions for more information on ways of entering these two points.   The last input required to create these dimensions is to input the dimension text.   The string defaults to the radius of the arc or circle in drawing units.   You can change the text any way you want.   For example, you may want to add the proper units to the end of the string (e.g. feet or meters).   You may want to change the text entirely.   Once you have entered the desired text, click on the *OK* button or press Enter.

The program will prompt you to select a circle or arc again.   You can create another radial dimension or end the command.

You can create different kinds of radial dimensions depending on how you select the points for the dimension.   In the example below, the dimension point was selected by using the *Center element snap*.   The position of the dimension arrow is determined by the point you pick when selecting the arc or circle.

Radial Dimension Example 2

Arcs or circles which are part of an inserted figure cannot be dimensioned.   If you try to select an arc or circle which is part of an inserted figure, the program will display the message: *Element is not an arc or circle*.   You will also get this message if the first drawing element the program finds within the selection box is not an arc or circle.   When selecting an arc or circle for radial dimensioning, make sure it is the only one in the selection box.

See also:
[Dimensioning](#)

Text Angle

To specify the angle of text, choose *Options/Text properties/Text angle* from the menu.   Enter the desired text angle in the dialog box which appears.   The text angle specifies the rotation of the baseline of the text string in degrees counter-clockwise from the x axis.

Text Angle

Windows raster fonts (e.g. Courier, Fixedsys, MS Sans Serif, MS Serif, Small Fonts, System and Terminal) will not rotate.

See also:
Text Properties

Text Orientation

To specify the orientation of text, choose *Options/Text properties/Text orientation* from the menu. Enter the desired text orientation in the dialog box which appears. The text orientation specifies the rotation of the individual characters in a text string in degrees counter-clockwise. Support for text orientation differs from font to font and from device driver to device driver. The ones which do support text orientation may only support certain angles (e.g. 90, 180, 270).

See also:
Text Properties

Text Justification

To specify the alignment of text, choose *Options/Text properties/Text justification* from the menu. When you create a text element you are asked to input the alignment point. This point can represent different positions within the string depending on the current text justification setting. The text justification is initially set to *Top* and *Left* when the program is first loaded. The text justification dialog allows you to specify one of three horizontal justification points (Left, Center and Right) and one of three vertical justification points (Top, Baseline and Bottom). To select one, click on its radio button. When you have specified the horizontal and vertical justification, click on *OK*.

Text Justification Dialog

The figure below shows all of the possible text justification points.
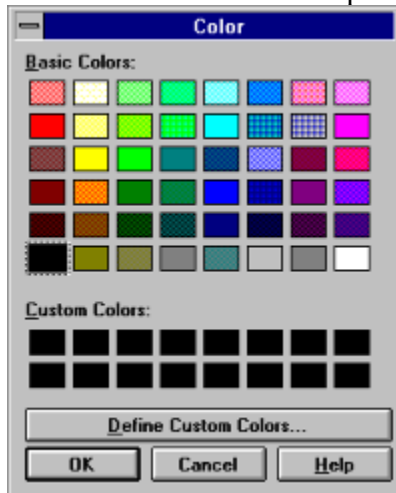

Text Justification

See also:
<u>Text Properties</u>

Point Color

Points can only be displayed in solid colors.   In other words, they cannot use *dithered* colors which use a pattern of different solid colors to produce the look of another color.   The number of solid colors available is dependent on the display device.

Point Color Dialog

To specify a point color, choose *Options/Point properties/Point color* from the menu.   You can also select a point color using the color button on the tool bar .   It is active only after you have chosen the point drawing command.   To make a selection from the color input dialog, simply click on the desired color (or highlight it using the arrow keys and press the spacebar). Then press the *OK* button.   If a nonsolid color is selected, the closest solid color is used instead. You must also be careful not to select colors which map to the same color as the background since they will not be visible on the background.

See also:
Point Properties

Point Type

To specify a point type, choose the *Options/Point properties/Point type* command from the menu.   You can also select a point type using the type button on the tool bar ![STOP].   It is active only after you have chosen the point drawing command.   To select a point type, click on the desired type (or use the arrow keys).   Click on the *OK* button (or press Enter) to use the selected point type.


Point Type Dialog

See also:
Point Properties

Point Size

To specify a point size, choose *Options/Point Properties/Point size* from the menu.   You can also select a point size using the size button on the tool bar .   It is active only after you have chosen the point drawing command.   Enter the desired point size.   This value is specified in drawing units and its displayed size will vary depending on the current drawing area extents.
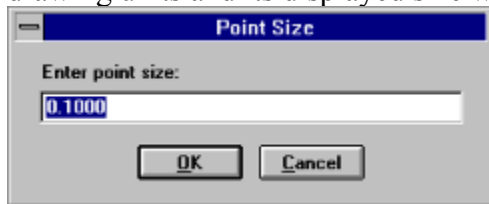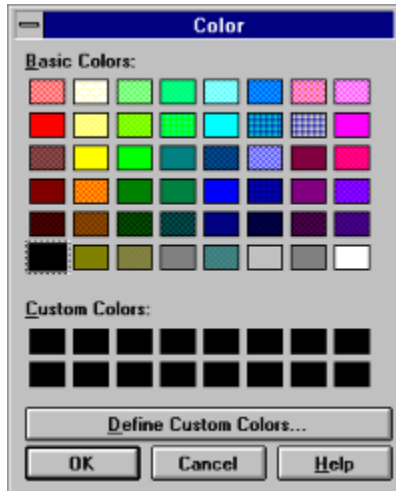

Point Size Dialog

See also:
Point Properties

Fill / Hatch Color

Hatched areas can only be displayed in solid colors.   In other words, they cannot use *dithered* colors which use a pattern of different solid colors to produce the look of another color.   The number of solid colors available is dependent on the display device.   Filled areas can use dithered colors.


Fill / Hatch Color Dialog

To specify a fill / hatch color, choose *Options/Fill / Hatch properties/Fill / Hatch color* from the menu.   You can also select a fill / hatch color using the color button on the tool bar .   It is active only after you have chosen the fill / hatch drawing command.   To make a selection from the color input dialog, simply click on the desired color (or highlight it using the arrow keys and press the spacebar).   Then press the *OK* button.   If a nonsolid color is chosen for a hatched area, the closest solid color is used instead.   In this case, you must be careful not to select colors which map to the same color as the background since they will not be visible on the background. You may use any color for filled areas.

See also:
Fill / Hatch Properties

Fill / Hatch Type

To specify a fill / hatch type, choose the *Options/Fill / Hatch properties/Fill / Hatch type* command from the menu.   You can also select a fill / hatch type using the type button on the tool bar .   It is active only after you have chosen the fill / hatch drawing command.   To select a fill / hatch type, click on the desired type (or use the arrow keys).   Click on the *OK* button (or press Enter) to use the selected fill / hatch type.
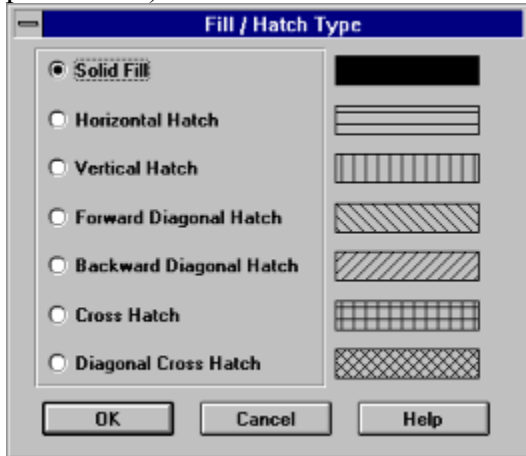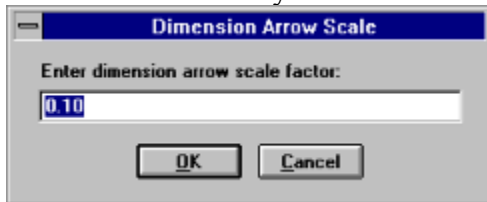

Fill / Hatch Type Dialog

See also:
Fill / Hatch Properties

Dimension Arrow Scale

To specify the dimension arrow size, select *Options/Dimension Properties/Dimension arrow scale* from the menu.   Enter the desired dimension arrow scale in the dialog which appears. This value is the scale factor used for the arrow head symbol when it is inserted.   If the arrow head symbol is drawn with a length of one (as the default one is), the value you enter in this dialog also represents the length of the arrow head in drawing units.   You must enter this value in decimal, even if you have architectural number format specified for the drawing.   Click on the *OK* button when you have entered the desired scale.

Dimension Arrow Scale Dialog

See also:
Dimension Properties

Dimension Offset

To specify the dimension offset, select *Options/Dimension Properties/Dimension offset* from the menu.   Enter the desired offset in the dialog which appears.   This value is in drawing units. Click on the *OK* button when you have entered the desired scale.

Dimension Offset Dialog

The dimension offset specifies different things for the different kinds of dimensions.   For leaders and radial dimensions, it specifies the length of the line which extends horizontally from the second end point you select for the leader (see Leader).   For linear dimensions (horizontal, vertical and parallel), it specifies the offset of the extension lines from the first two dimension points you select and the distance the extension lines extend beyond the dimension line (see Linear Dimension).

See also:
Dimension Properties

This button locks the selected layer(s) in the layer list above this button.   When a layer is locked, the elements on the layer cannot be selected or edited.

Layers

The *Options/Layer* command allows you to create layers, rename layers, set the current layer and set layer properties.   Every element in the drawing is on a particular layer.   When you enter a new drawing, the layer *MAIN* is automatically created and set as the current layer.   Any drawing elements you create are placed on this layer until you specify a different layer.   Layers provide a means of grouping drawing elements together.   For example, if you were drawing a floor plan you may want to place different floors on different layers (e.g. "BASEMENT" and "FLOOR1") and thus have one drawing with the entire floor plan instead of placing the floors in separate drawings.   Or you might create separate layers for the electrical wiring, plumbing and so on.

Layers can be turned on and off.   When they are off, the elements on the layer are not drawn and cannot be edited.   Layers can be locked, which means the elements on the layer are displayed but they cannot be edited.   The layers can be marked as not printable, which means they are not printed or plotted.   This can be used to create templates and construction lines which help guide your drawing but should not be plotted.   These properties can be set independently for each layer.   One layer is always set as the current layer and any drawing elements created are placed on the current layer.   All of these properties can be set in the layer dialog.   This dialog is also used to create new layers and rename existing layers.

Each layer also has the following set of properties associated with it.
> Line Type
> Line Color
> Line Width
> Text Font
> Text Height
> Text Angle
> Text Orientation
> Text Justification
> Point Color
> Point Type
> Point Size
> Fill / Hatch Color
> Fill / Hatch Type
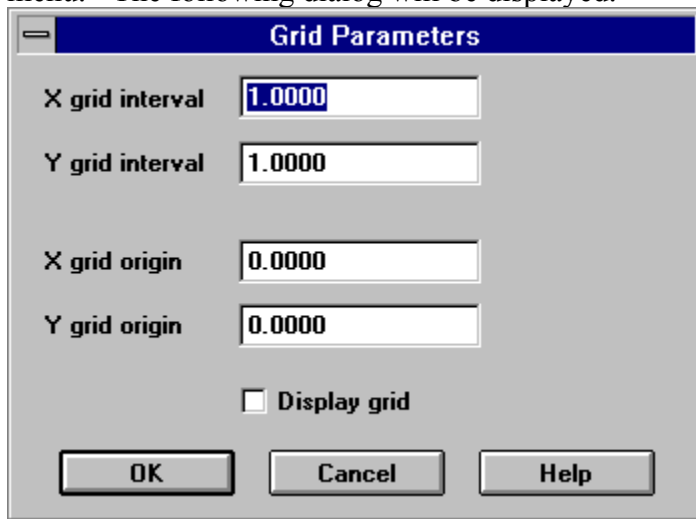> Dimension Arrow Size
> Dimension Offset

Whenever a new drawing element is created which contains one of these properties, the setting for the property on the current layer is used.   For example, when you create a new line, the color is determined by the line color set on the current layer.   Once set, however, the color of the line remains the same even if you change the line color setting for the current layer.   The layer properties only affect new elements.   You must use the *Edit/Change* command to change the color of the line.   This is because the line color for the element is stored in the line element itself.   Its initial value is set using the settings for the current layer but after this the color is not tied to the layer settings.

You may create as many layers as you like.   There is no limit to the number of layers a drawing may contain.   The layers and their properties are stored with the drawing so each layer makes

your drawing file larger and also increases the amount of memory needed to edit the drawing. There is no limit to the number of drawing elements on each layer except the amount of available memory.

Grid Snap Parameters

To set the grid snap parameters choose the *Options/Grid snap parameters* command from the menu. The following dialog will be displayed.

Grid Parameters	
X grid interval	1.0000
Y grid interval	1.0000
X grid origin	0.0000
Y grid origin	0.0000
☐ Display grid	
OK Cancel Help	

Grid Snap Dialog

This dialog allows you to specify the grid which is used when the element snap mode is set to *Grid*. For a description of the different parts in the above dialog, click on the part. For example, for a description of the *X grid interval* click on its edit box.

See also:
Grid Dot Size
Grid Density

Changing Drawing Element Properties

To change the properties of drawing elements, select the elements to change (see the section on Selecting Drawing Elements).   Then, choose *Change* from the *Edit* menu or press Ctrl+H.   This will bring up a dialog which allows you to select what property to change.   You can only change one property at a time.   To choose a property, click on its radio button and click on the *OK* button.
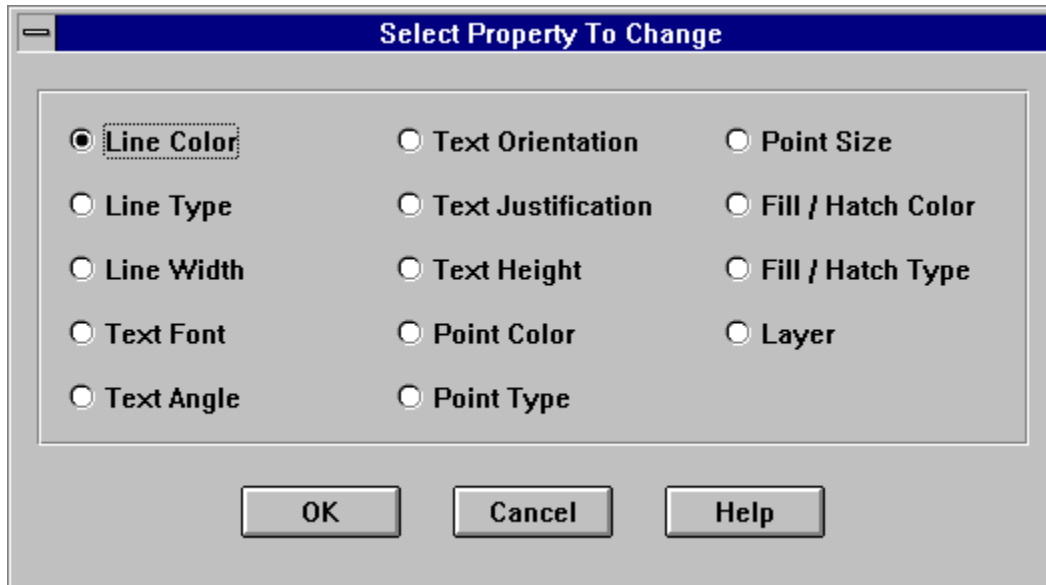
Change Dialog

The program will bring up a dialog which allows you to specify the new property.   These dialogs are the same ones used for the corresponding commands on the *Options* menu.   See the appropriate section under Setting Drawing Properties in this document for more information on using these dialogs.   The only exception to this is the *Layer* option.   See the section Changing the Layer of a Drawing Element for more information.

When the new property is specified, the selected elements are changed.   You will have to clear the selection set (see Clear Selection) in order to see color changes since the current selection set is always drawn in the selection set color.

Text Properties

The *Options/Text properties* commands allow you to specify how text is drawn.   Text strings have the following properties:

<u>Text Font</u>
<u>Text Height</u>
<u>Text Angle</u>
<u>Text Orientation</u>
<u>Text Justification</u>

These properties can be specified before or during the input of a text element.   Each layer in the drawing has its own set of text properties.   When you choose one of these options, you are changing the property for the current layer only.   The selected properties remain in effect for the layer until another selection is made.   Whenever a text element is created, it is displayed using the current text properties set for the current layer.   It will always be displayed this way unless the properties are changed.   See the section <u>Changing Drawing Element Properties</u> for more information.

Point Properties

The *Options/Point properties* commands allow you to specify how points are drawn.   Points have the following three properties:

Point Color
Point Type
Point Size

These properties can be specified before or during the input of a point element.   Each layer in the drawing has its own set of point properties.   When you choose one of these options, you are changing the property for the current layer only.   The selected properties remain in effect for the layer until another selection is made.   Whenever a point is created, it is displayed using the current point color, type and size set for the current layer.   They will always be displayed this way unless the properties are changed.   See the section Changing Drawing Element Properties for more information.

Fill / Hatch Properties

The *Options/Fill/Hatch properties* commands allow you to specify how filled areas are drawn. The following two properties can be specified for filled / hatched areas.

        Fill / Hatch Color
        Fill / Hatch Type

These properties can be specified before or during the input of a fill / hatch area.   Each layer in the drawing has its own set of fill / hatch properties.   When you choose one of these options, you are changing the property for the current layer only.   The selected properties remain in effect for the layer until another selection is made.   Whenever a filled or hatched area is created, it is displayed using the current type and color set for the current layer.   It will always be displayed this way unless the properties are changed.   See the section Changing Drawing Element Properties for more information.

Dimension Properties

The *Options/Dimension properties* commands allow you to specify how dimensions are drawn. The following two properties affect how dimensions are created.

        <u>Dimension Arrow Scale</u>
   <u>Dimension Offset</u>

These properties can be specified before or during the input of a dimension.   Each layer in the drawing has its own set of dimension properties.   When you choose one of these options, you are changing the property for the current layer only.   The selected properties remain in effect for the layer until another selection is made.   Whenever a dimension is created, it is displayed using the current arrow size and offset set for the current layer.   The dimension commands create line, text and insert (the arrow head) drawing elements.   Once created, they can be edited like any line, text or insert element.

The current <u>line properties</u> affect how the lines are drawn and the current <u>text properties</u> affect how the text is drawn.   The arrow head is an insertion of the symbol stored in the file D_ARROW.SYM.   You can modify this symbol like any other if you desire a different arrow head (see <u>Modifying Symbols</u>).

Layer Dialog

When you choose the *Options/Layers* command from the menu, the following dialog is displayed.
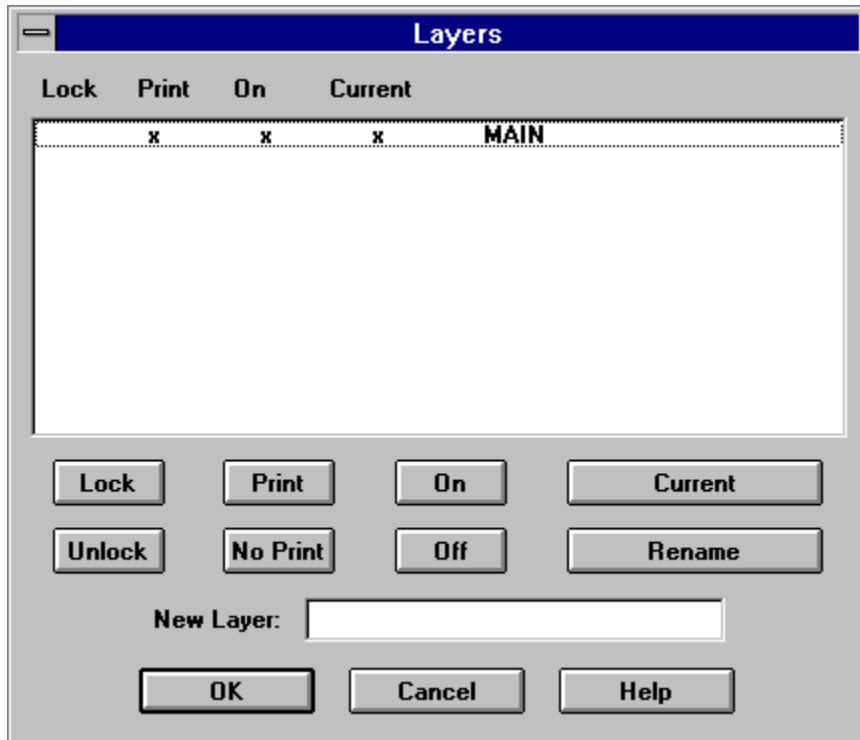
Layer Dialog

For a description of the different parts in the above dialog, click on the part.   For example, for a description of the *Lock* button click on the *Lock* button.

The layer dialog is used to set certain properties for each layer, create new layers, set the current layer and rename layers.   To set the lock/unlock, print/no print and on/off status of a layer or group of layers, you must first select the layers you wish to change.   To select a layer with the mouse, simply click on the desired layer.   To select a layer using the keyboard, press Tab to move to the box containing the list of layers.   Use the up and down arrow keys to move through the list and the space bar to select a layer.   Selecting a layer again unselects it (i.e. selecting a selected layer unselects it).   When you have the desired layer(s) selected, click on the desired button to change its state or use the Tab key to highlight the desired button and press Enter.   For example, clicking on the *Off* button will turn the selected layer(s) off.

To rename a layer or layers, select the layers you want to rename and click on the *Rename* button.   For each layer you selected, the program will display a dialog with the current name in it.   Type in the new name and click on *OK*.   If you do not wish to change the name of the layer, simply click on *OK* without changing the name.   Clicking on *Cancel* will end the command without changing the name of the layer currently listed and return you to the layer dialog.   Any layers you renamed previously will still be changed.

To set the current layer, select one layer and click on the *Current* button.   If more than one layer is selected, the program will display a warning message and the current layer will not be changed.   Before clicking on the *Current* button make sure only one layer is selected.

To add a new layer, click on the *New Layer* edit box (or press Tab until a cursor appears in the box) and type in the name of the layer you wish to create.   Press Enter or click on *OK*.   The new layer is added to the layer list.   The layer dialog remains active so you can type in more new layers if you desire.   Press *OK* again to exit the dialog.

The changes you make in this dialog do not take affect until you press the *OK* button and exit the layer dialog.   Pressing *Cancel* exits the dialog without making the changes.

See also:
<u>Layers</u>

This button unlocks the selected layer(s) in the layer list above this button.   When a layer is unlocked, the elements on the layer can be selected and edited.

This button sets the selected layer(s) in the layer list above this button to printable.   When a layer is printable, the elements on the layer are drawn on the output device when the drawing is printed / plotted.

This button sets the selected layer(s) in the layer list above this button to not printable. When a layer is not printable, the elements on the layer are not drawn on the output device when the drawing is printed / plotted.

This button turns on the selected layer(s) in the layer list above this button.   When a layer is on, the elements on the layer are drawn on the display and printer / plotter.

This button turns off the selected layer(s) in the layer list above this button.   When a layer is off, the elements on the layer are not drawn on the display or printer / plotter.

To use this button, select one and only one layer in the layer list and then click on this button. The selected layer will be set as the current layer.   When a new drawing element is created, it is placed on the current layer.

This button allows you to rename the selected layer(s) in the layer list above this button.   After pressing this button, the program will display a dialog with the current name for each layer you selected in the layer list.   After typing in the name you desire, press the *OK* button.   After all the layers you selected have been renamed, you are returned to the layer dialog.   Pressing *Cancel* ends the command without renaming the currently displayed layer but any layers previously renamed remain renamed.

You cannot change the name of a layer to the same name as an existing layer.   The layer name cannot contain any blanks.   Any blanks you enter are stripped out of the name.   The name is also converted to all uppercase.

This edit box is used to create a new layer. Type in the name of the layer you wish to create and press Enter or click on the *OK* button. If the layer does not already exist then it will be created. The layer name cannot contain any blanks. Any blanks you enter are stripped out of the name. The name is also converted to all uppercase.

This area lists the layers in the drawing and their current settings.   The layer name is displayed on the right and the settings for the layer are displayed to the left in the same row.   An x in the *Lock* column indicates the layer is locked, no x indicates the layer is unlocked.   An x in the *Print* column means the layer is printable, no x means it is not.   An x in the *On* column indicates the layer is on, no x indicates it is off.   The layer with an x in the *Current* column is the current layer.

This is also the area where you select which layer(s) you want to operate on.   Click on the row containing the layer to select it (or use the up and down arrow keys to highlight the desired layer and press the space bar to select it).   You may select any number of layers.   To unselect a layer simply select it again.   If the list is too long to display, a scroll bar will be displayed on the right. The buttons below the layer list operate on the selected layers only.

Selection Set Color

The program displays selected elements in a certain color.   The *Options/Program configuration/Selection set color* command allows you to set the color in which selected elements are drawn.   This command will bring up the color selection dialog.   Choose the new color from this dialog and click on the *OK* button.   The selection set will now be displayed in the chosen color.

See also:
Program Configuration
Selecting Drawing Elements

Cursor Increment

The cursor moves a certain number of pixels on the display when an arrow key is pressed. The cursor increment, or step size, specifies how many pixels it moves. This value is initially set to 1. Larger values cause the cursor to move more quickly across the display but make it more difficult to position the cursor at desired coordinates. Smaller values have the opposite effect.

To change this value, choose *Cursor Increment* off of the *Options/Program configuration* menu. Enter the desired cursor increment (greater than 0) and click on the *OK* button. The cursor will move by the input number of pixels when an arrow key is pressed and the cursor is in the drawing area.

See also:
Program Configuration

Arc/Circle Step Factor

At certain times it becomes necessary to draw arcs and circles as a series of line segments instead of using the standard arc and circle Windows API functions.   This occurs when it is impossible to specify the position of the arc or circle within the integer coordinate space used by the Windows GDI functions.   This can happen if you zoom in very close on a portion of an arc or circle, or if you specify an arc or circle with a large radius relative to your current drawing area extents.   In these situations, GammaCAD displays the arc or circle as a series of line segments.

The *Options/Program configuration/Arc/circle step factor* command allows you to adjust the length of the line segments used to display the arc/circle.   The program calculates a segment size which is approximately equal to the resolution of the device.   It then multiplies that value by the specified step factor.   The default step factor is one.   The step factor must be greater than zero. The smaller the value the smoother the arc/circle will appear but it will take longer to display. Again, this value is used only if the arc/circle cannot be displayed using the standard Windows API arc and circle functions.

See also:
Program Configuration

Creating a New Drawing

If you are currently editing a drawing and would like to start a new drawing, choose *New* from the *File* menu.   The contents of the drawing area are cleared and you are placed in a blank drawing.

If you select the *New* item from the *File* menu and the drawing you are currently working on has not been saved, a message box will appear and ask if you want to save the current drawing. If *No* is chosen, any changes made to the current drawing will be lost.   Choosing *Cancel*, ends the *New* command and returns you to the current drawing.   If you select *Yes*, the drawing will be saved to the file name you specified previously.   If no previous name was specified, you will be prompted for a name.   After saving the file, you are placed in a new drawing.

The default drawing properties (e.g. line type, line color, text font, etc.) for the new drawing are read from the prototype drawing, if any.

When you first run the GammaCAD program, you are placed in a new drawing.

Exporting DXF Files

To create a DXF file choose the *File/Dxf export* command.   After selecting this command, the file *Save As* dialog is displayed.   Enter the name of the file you want the drawing written to in DXF format.   A DXF extension will be added if you do not specify one.   To bring the drawing into another program, use the DXF import command in that program   For example, in AutoCAD you use the *dxfin* command.   See the documentation for that program for more information on importing a DXF file.

If the drawing you are exporting was originally imported from a DXF file, then any DXF properties contained in the drawing are also written to the DXF file.   See the section on importing DXF files for more information.

Certain element properties in a GammaCAD drawing need to be converted before being written to the DXF file.   For example, DXF files do not use the same method of specifying colors as used in Windows.   Colors in the GammaCAD drawing are converted to a DXF color on DXF export and may not appear the same as in the original drawing.   The following properties are handled in this manner: color, line type, point type, text style and text justification.   If a DXF property is present for the element, it is used and no conversion is done.

A text style is created in the DXF file for each font you have defined (and used) in the GammaCAD drawing.   The name of the style in the DXF file is the same as the name of the font in GammaCAD.   If the file was originally input from a DXF file and a style with the font name exists then those parameters are output.   Text elements using the default *System* font in GammaCAD are output in the DXF file with no text style name which means they use the *Standard* style.   When a text element is output to the DXF file, if it contains a DXF style (as input from a DXF file) then it is given that style name.   If it has no DXF style name then the font name is used as the style name.

The line types used by GammaCAD are defined in the TABLES section of the DXF file.   These line types are used when a line, arc or circle is output unless a   DXF line type (as input from a DXF file) is present.

Filled and hatched areas created in GammaCAD are not output to the DXF file since there is no equivalent DXF entity.

Hints on Bringing DXF Files into AutoCAD

In AutoCAD use the *style* command to change the parameters associated with a text style.   Text orientation angles in GammaCAD are ignored when creating a DXF file.

Use the *ltscale* command to adjust the dash and dot spacing of lines, arcs and circles.

AutoCAD only allows one point type and one point size.   In other words, you cannot have points represented by a plus (+) and points represented by an x (X).   To set the point type and size, use the *setvar* command to set the variables PDMODE and PDSIZE, respectively.

GammaCAD symbols are defined as blocks in the DXF file.

The AutoCAD *dxfin* command functions differently depending on whether you are using it in a new or existing drawing.   In an existing drawing, AutoCAD will ignore everything but the ENTITIES section of the DXF file.   This means the text styles, line types and blocks defined in the DXF file are ignored by AutoCAD.   You must make sure these are defined prior to executing the *dxfin* command in this case.   Use the *style*, *linetype* and *block* commands in AutoCAD. When you use *dxfin* to bring in a DXF file created by GammaCAD in a new drawing, everything will be defined for you.

See also:
Using DXF Files

Opening an Existing Drawing

If you want to edit a drawing created previously, choose the *Open* command from the *File* menu or press the open button on the command tool bar.   This brings up the file selection dialog.   Select the proper drive and directory from the drive and directory lists.   Then choose one of the files from the file name list and click on the *OK* button.   The drawing in the given file will be displayed in the drawing area.   You must select an existing file with this option.   You can now edit the elements in the displayed drawing or add more elements if desired.
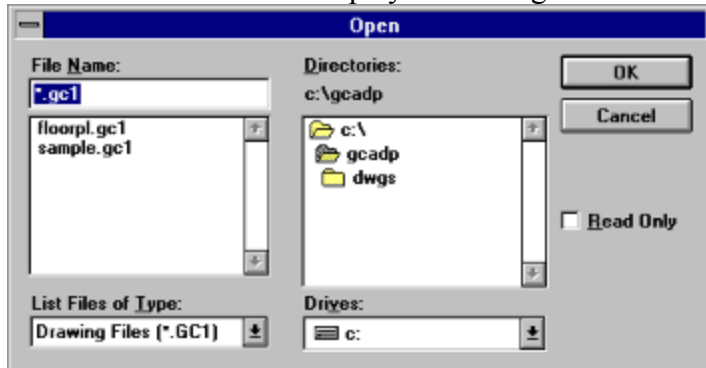
Open Dialog

If you choose a file from this dialog and the drawing you are currently working on has not been saved, a message box will appear and ask if you want to save the current drawing. If *No* is selected, any changes made to the current drawing will be lost.   Selecting *Cancel*, ends the *Open* command and returns you to the current drawing.   If you choose *Yes*, the drawing will be saved to the file name you specified previously.   If no previous name was specified, you will be prompted for a name.   After saving the file, the file open dialog will be displayed.

Print/Plot Setup

The *File/Print/plot setup* command brings up the print setup dialog. When you press the *Setup* button on the print dialog, which is displayed during the *File/Print/plot* command, the print setup dialog is also displayed. The print setup dialog allows you to select a printer from the printers you have installed in Windows. To install a printer in Windows, use the Control Panel (see your Windows manual). The print setup dialog also lets you set certain parameters for the selected printer. The *Options* button brings up another dialog which lets you set parameters specific to the selected printer. When you press *OK* or *Cancel* in that dialog you are returned to the print setup dialog.
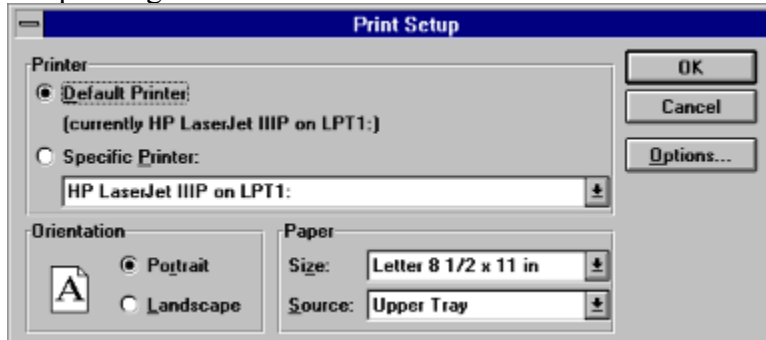
Print Setup Dialog

Press *OK* to save the changes you have made to the printer setup or press *Cancel* to exit the dialog without saving them.

See also:
Page Setup

Importing DXF Files

To bring a drawing you created in another program into GammaCAD, you must first use the other program to create a DXF file.   For example, in AutoCAD you use the *dxfout* command.   See the documentation for that program for more information on creating a DXF file.   Once the DXF file is created, you can bring it into GammaCAD using the *File/Dxf import* command.   After selecting this command, the file open dialog is displayed and all of the files with a DXF extension in the current directory are listed.   Select the file you wish to import.   The program reads the file and displays it.   You may need to use the *View/Zoom extents* command to see the drawing if it does not fall within the current window.

GammaCAD does not support every possible drawing entity which can be present in a DXF file.   It does preserve all the information present in the DXF file, however.   When GammaCAD encounters an entity in the DXF file which it does not support, the parameters which describe the entity are stored and the entity is drawn with the following symbol ⊞.   If the drawing is later exported to a DXF file, the entity is written to the DXF file with the same parameters it had when imported.   This allows you to edit drawings in GammaCAD created by another program without losing information in the DXF import / export process.   The HEADER and TABLES section of the DXF file are also preserved.

The following DXF entities are supported by GammaCAD: line, arc, circle, point, text, insert, block, endblk, polyline, vertex and seqend.   Trace and solid entities are converted to a fill element on DXF import and converted back on export.   Attdef entities are displayed as text elements.   Attrib entities are not displayed but are preserved within the insert element.   3Dface entities are displayed as 2d polylines.   Shapes and associative dimensions are not supported at all and are drawn with the following symbol ⊞.   If you want associative dimensions to be displayed in GammaCAD, you must use the *explode* command in AutoCAD to explode them.   Of course, they will no longer be associative dimensions.   If you use the AutoCAD *setvar* command and set the DIMASO variable to 0, then non-associative dimensions are created by default.

Certain entity properties in a DXF file are either not supported by GammaCAD or need to be converted before they can be used.   In either case, the original value of the property is preserved in the drawing and used when the drawing is exported to a DXF file.   If the property is edited, the value from the DXF file is removed.   The following properties are handled in this manner: color, line type, point type, text style and text justification.

For example, DXF files do not use the same method of specifying colors as used in Windows.   Colors in the DXF file are converted to a Windows color on DXF input and may not appear the same as in the original drawing.   The original value is stored, however, and if the drawing is exported to a DXF file then the original color is used.   If the color of the drawing element is edited in GammaCAD, the DXF color is removed.   In this case, the Windows color will be converted to a DXF color value when the element is output to a DXF file.

In general, if the property is something you can not do in GammaCAD then it is not supported.   For example, you can not scale an inserted figure independently in the x and y directions in GammaCAD and GammaCAD ignores the y scale factor for an insert entity in a DXF file.   The

DXF properties which are not supported by GammaCAD include: elevation, thickness, extrusion direction, point angle, text x-scale factor, text obliquing angle, text generation flags, some text justifications, block type flags, xrefs, insert attributes, insert y/z scale, insert row/column counts/spacing, all polyline flags except closed, polylines with different start/end widths, curved polyline segments and invisible edges in a 3d face.

If you import more than one DXF file into a drawing, the header section of the second DXF file is ignored.   Any tables or block definitions which were defined in the first DXF file are skipped in the second.   This can result in missing table entries if you perform a DXF export on the drawing.   For example, suppose you import a DXF file which contains a line type table.   Then you import a second DXF file into the same drawing.   The second drawing also contains a line type table but it is ignored because the drawing already contains a line type table from the first DXF file.   If the second DXF line type table contains line type definitions which were not in the first, they will be missing when you later export the drawing to a DXF file.   It is up to you to avoid this situation by making sure the first DXF file you import contains all the necessary table entries for both DXF files.   Or, define all required table entries in the other program before importing the DXF file exported from GammaCAD.

When you cut or copy drawing elements which contain DXF information to the Windows clipboard, the DXF information is transferred as well.   Information from the DXF header is **NOT** transferred.   When you paste these elements into another drawing, only those tables which are not defined in the drawing are created.   In other words, DXF table entries for the clipboard elements are not merged with the DXF table entries in the drawing to which you are pasting. This is similar to importing more than one DXF file into the same drawing as described in the previous paragraph.

See also:
Exporting DXF Files
Using DXF Files
Display DXF Elements

Page Setup

The *File/Page setup* commands allow you to position your drawing on the printer / plotter page and set the size of the drawing on the page.   The page setup commands are:

> Fit Drawing to Page
> Print/Plot Scale
> Position Drawing on Page

GammaCAD queries the current print driver to get the page size and printable area.   The *File/Print/plot setup* command is used to set the current printer and its page parameters.   If the print driver returns the page size, GammaCAD draws the page on the display using the current plot scale and page origin.   The lower left corner of the page is placed at the page origin.   The page origin is initially set to (0,0) in a new drawing.

Printer Page Display

The color of the page display will depend on the background color of the window.   The Control Panel is used to set the window background color.   The solid line represents the boundaries of the page.   The dashed line represents the printable area on the page.   Most printers do not allow you to print all the way to the edge of the page.

The current plot scale determines the size of the displayed page.   The *File/Page setup/Print/plot scale* command allows you to set the plot scale.   The position of the page is determined by the page origin.   The *File/Page setup/Position drawing on page* command allows you to set the page origin.   Using the scale command, you can adjust the size of the page drawn in the display so the drawing fits within the printable area.   Then you can position the drawing on the page with the position command.   The *File/Page setup/Fit drawing to page* command calculates the plot scale and page origin automatically.   It is probably the easiest way to position the drawing on the printer page.   It makes the drawing as large as possible on the page and centers it on the page.

See also:
Display Page

Exit

The *File/Exit* command ends the program.   If you have any unsaved data the program will prompt you to save it before exiting.

See also:
Saving the Drawing

Fit Drawing to Page

The *File/Page setup/Fit drawing to page* command automatically sets the plot scale and page origin for the current drawing.   The plot scale determines the size of the drawing on the printer / plotter page.   The page origin determines the position of the drawing on the page, it is the position of the lower left corner of the page in the drawing.

After choosing this command, the program retrieves the current drawing extents.   If this is a new drawing and nothing has been drawn, the following dialog is displayed.
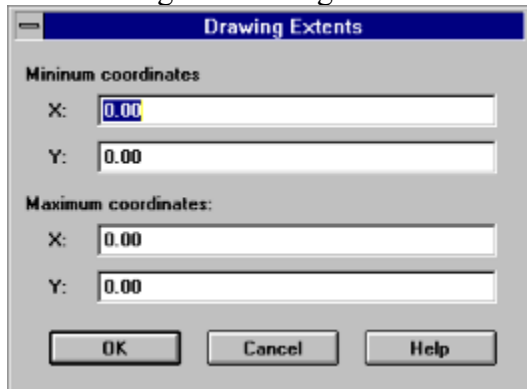
Drawing Extents Dialog

Enter the minimum and maximum coordinates you plan on using in your drawing.   For example, if you want your drawing to start at (0,0) and your drawing is 20 units wide and 15 units high, enter (0,0) for the minimum and (20,15) for the maximum coordinates.   This dialog allows you to set up the page for your drawing before drawing it.   You can, of course, rerun this command at any time after creating all or part of your drawing to refit the drawing to the page. Again, the above dialog is not displayed if any drawing elements have been added to the drawing.   The extents are taken directly from the drawing elements in this case.

After the drawing extents have been determined, this command calculates the scale which makes the drawing as large as possible on the page.   It then displays the scale it calculated in the plot scale dialog.   To use the calculated scale press the *OK* button.   If you would like to use an even number for the scale you can enter it in this dialog.   You should round up and not down.   If you specify a scale value less than the displayed value, the drawing will not completely fit on the page.   After the plot scale has been entered, this command calculates the page origin which centers the drawing on the page.   If you have moved or deleted drawing elements and the drawing has not been redrawn since these operations were performed, the calculated scale may not be correct.   The drawing extents are set each time the drawing is redrawn (e.g. when a view command is used).   If after running this command the drawing is not fit to the page, rerun this command again.

A dialog is then displayed which asks whether you would like to zoom to the new page.   If you choose *Yes*, the program performs a *Zoom page*.   If you choose *No*, the current view is not changed but the plot scale and page origin are changed.

See also:

<u>Page Setup</u>

Print/Plot Scale

The *File/Page setup/Print/plot scale* and *File/Page setup/Fit drawing to page* commands set the plot scale for the current drawing.   The plot scale determines the size of the drawing on the printer / plotter page.   Increasing the plot scale decreases the size of the drawing on the page. Decreasing the scale increases the size of the drawing on the page.   If you prefer to think in terms of fitting the page drawn on the display to the drawing, then increasing the plot scale increases the size of the page relative to the drawing.   Decreasing the scale decreases the size of the page.   Of course, the actual size of the page does not change, its size relative to the drawing is changing when you change the plot scale.

The following dialog is displayed when the *File/Page setup/Print/plot scale* command is chosen and also during the *File/Page setup/Fit drawing to page* command.   Enter the desired scale and press the *OK* button to set the plot scale.   Pressing the *Cancel* button closes the dialog without changing the plot scale.
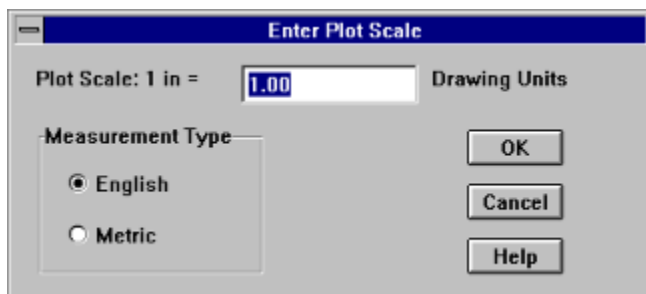
Plot Scale Dialog

This dialog sets the number of drawing units which equal 1 plot unit.   The plot units are either inches or centimeters.   Select the *English* radio button to use inches and the *Metric* radio button to use centimeters.   When a new drawing is created, the plot units are set according to the setting of the *Measurement* field in the international section (*intl*) of the Windows WIN.INI file. These parameters can be configured using the *International* option in the Control Panel.   Once a drawing has been created the plot unit setting is saved within the drawing.   Changes to the *Measurement* field in the Control Panel have no affect.   You must use the *File/Page setup/Print/plot scale* command to change the plot units.   The advantage to this is the drawing remembers the plot units you used and they are restored when you open the drawing.   Using a prototype drawing can override this default behavior.

Drawing units can be anything (e.g. inches, feet, miles, centimeters, meters, kilometers, etc.) the program does not care what they are.   The plot scale sets the relationship between the drawing units you use in the drawing and distances on the plotter.   This is all the program needs to know. The only exception to this is if you have chosen architectural numbers using the *Options/Number format* command.   In this case, the program assumes the drawing is in inches.   For a further description of drawing coordinates see the Drawing Coordinate System section.

See also:
Page Setup

Position Drawing on Page

The *File/Page setup/Position drawing on page* command sets the page origin for the current drawing.   The page origin determines the position of the drawing on the printer / plotter page, it is the position of the lower left corner of the page in the drawing.   You may want to use the *View/Zoom page and extents* command to set the view before choosing this command.   After selecting the *Position drawing on page* command, you are prompted to enter the page origin in drawing units.   You can use the mouse or arrow keys to position the page (as drawn on the display) relative to the drawing.   Press the left mouse button or press Enter to select a position. Alternatively, press the right mouse button or the F2 key to bring up the coordinate input dialog. The initial values displayed in this dialog are the previous page origin coordinates.   Input the desired page origin and press Enter or click on *OK*.

To position a drawing using a printout you created earlier, measure the distance (in inches) you want to move the drawing right or left and up or down.   Convert the values you measured from plot units (inches) to drawing units.   To do this, simply multiply the values you measured by the current plot scale.   To determine the current plot scale, choose the *File/Page setup/Print/plot scale* command.   Use the value to the right of the equal sign in the edit box.   Press *Cancel* in the plot scale dialog.   Next, select the *File/Page setup/Position drawing on page* command and click the right mouse button or press F2.   Choose the *XY Offset Input* radio button in the box on the left in the coordinate input dialog.   If you want to move the drawing right, enter the distance you calculated previously in the *DX* field as a negative number.   To move it left, enter the distance in the *DX* field as a positive number.   To move the drawing up, enter the distance in the *DY* field as a negative number.   To move the drawing down, enter the distance as a positive number.   Press the *OK* button in the coordinate input dialog.   The page displayed in the drawing area will be shifted and the next time you print the drawing it will be moved by the desired amounts.

Examples

You printed out your drawing and you want to move it right 0.1" and up 0.2" on the page.   Your plot scale is 1"=1 drawing unit.   This means you need to shift the current page origin left by 0.1 and down by 0.2.   Remember, you are shifting the page relative to the drawing so the movements have to be opposite to the directions you want to move the drawing on the page. Choose the *File/Page setup/Position drawing on page* command and press the right mouse button.   Select the *XY Offset Input* radio button.   Enter -0.1 in the *DX* field and -0.2 in the *DY* field and press the *OK* button on the coordinate input dialog.   Replot the drawing.   The drawing will be 0.1" further to the right and 0.2 inches higher on the page.

You printed out your drawing and you want to move it left 0.15" and down 0.02" on the page. Your plot scale is 1"=10 drawing units.   This means you need to shift the current page origin right by 1.5 (0.15x10) and up by 0.2 (0.02x10).   Enter 1.5 in the *DX* field and 0.2 in the *DY* field.

See also:
Page Setup

The Y grid interval specifies the distance between grid points in the y direction.   The value is in drawing units.

The X grid interval specifies the distance between grid points in the x direction.   The value is in drawing units.

The X grid origin specifies the x coordinate of the first grid point.   You can use this value to shift the grid right and left without changing the distance between grid points.   The value is in drawing units.

The Y grid origin specifies the y coordinate of the first grid point.   You can use this value to shift the grid up and down without changing the distance between grid points.   The value is in drawing units.

Click on this box to turn the display of the grid on and off. When the box is checked, the grid is displayed. The grid does not have to be displayed to use the grid snap feature. If grid snap is on, the program snaps to the closest grid point regardless of whether the grid is displayed or not.

The *Number Format Type* field specifies whether floating point numbers are displayed in decimal, scientific, or architectural format.   When *Architectural* format is selected, the program assumes your drawing is in inches.

Set the *Decimal Digits* field to the number of decimal places you want the program to display to the right of the decimal point.   This causes the program to display that number of decimal places everywhere it displays a real number.

The *Decimal Separator* specifies the character the program uses for the decimal point in a floating point number.

The *Largest Denominator* field sets the largest denominator to use in a fraction. This value is used only if *Architectural* is selected as the number format type.

Saving the Drawing

There are 2 commands for saving the drawing, *Save* and *Save As*, both are on the *File* pull down menu.   The *Save As* command allows you to specify a file name.   Once a file name has been specified for a drawing, you save it to the same file by choosing *Save* or by pressing CTRL+S or by pressing the save button 🖫 on the tool bar.   There are no further prompts when using the *Save* command.   If the *Save* command is selected before a file name has been specified, the *Save As* command is used instead of *Save*.

The *Save As* command brings up a file selection dialog box.   To specify a file name, choose a drive from the drive list and a directory from the directories list.   Type in the name of the file in the *File Name* box.   You do not have to specify an extension for the file name.   If you do not specify one, the extension GC1 will be appended to the name you enter.   It is recommended you use a GC1 extension on all your drawing files so they will appear in the list of file names in the file selection dialog.   If you enter an existing file name, the program will warn you and ask if you want to replace it.   Choosing *Yes* will completely replace the contents of the file with the current drawing.   The previous file will be renamed with a GCB extension.   The file name dialog will not allow you to enter an invalid path or file name.
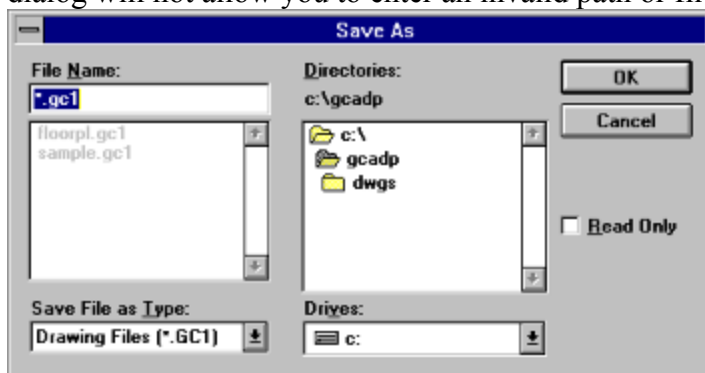
Save As Dialog

When the desired drive, path and file name have been entered, press the *OK* button and the current drawing will be saved.   The previous drawing file (if any) with the input name will be renamed with a GCB extension as a backup.

When a drawing is saved all the undo levels are removed.   This means previous operations cannot be undone after a save has been performed.   This is done to free up the memory used by the undo feature.

Selecting Drawing Elements

The *Cut, Copy, Delete, Rotate, Scale, Change* and *List* commands require one or more drawing elements be selected first.   They are grayed out and unselectable if no drawing elements are selected.   The *Edit/Select* commands allow you to create a new selection of drawing elements, add elements to the current selection, remove elements from the current selection and clear the selection.   There is only one selection of drawing elements maintained by GammaCAD.   The elements in that selection (if any) are displayed in the selection color.   The selection color is initially set to light gray but it can be changed using the *Options/Program configuration/Selection set color* command.

When selecting drawing elements, you input opposite corners of a rectangular area and all elements which fall within this area or cross it will be selected.   You may pick any corner of the area first.   To choose the first corner, click the left mouse button at the desired position and hold the left mouse button down.   Drag the mouse cursor to the opposite corner of the desired selection area.   After choosing the first corner, as you move the cursor, a box is displayed with one corner at your first selection point and the opposite corner at your current cursor position.   This box represents the selection area.   When the desired selection box is displayed, release the left mouse button.   This sets the second corner of the selection area.   Element snaps do not affect the points you pick during drag selection.
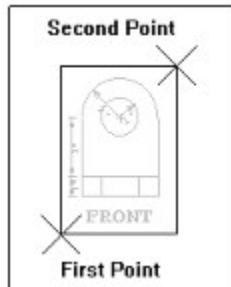
Selecting Elements

To select elements using the keyboard, use the arrow keys to position the cursor at the first corner of the selection area.   Press and hold down the Enter key.   Move the cursor to the second corner off the selection area using the arrow keys and release the Enter key.

When the second point is chosen, the program either creates a new selection with the elements in the selection box, adds them to the current selection, or removes them from the current selection. It depends on the current *selection mode*.

The selection mode can be set using the menu or command tool bar.   The selection mode can be either *new*, *add* or *remove*.   In *new* mode, when you select drawing elements, the old selection is removed and only the newly selected elements are selected.   In *add* mode, the selected elements are added to any previous selection.   In *remove* mode, the selected elements are removed from the previous selection.   When the *Add* item on the *Edit/Select* menu has a check mark next to it, the program is in *add* mode.   When *Remove* is checked, the program is in *remove* mode.   When neither is checked, the program is in *new* mode.

The add tool bar button corresponds to the check mark on the *Edit/Select/Add* menu

command.   When the add button is pressed, the *Add* menu option is checked and vice versa. The remove tool bar button
 corresponds to the *Edit/Select/Remove* menu command in the same way.   When neither tool bar button is pressed, the program is in *new* selection mode.   The plus (+) and minus (-) keys on the numeric keypad can also be used to set the *add* and *remove* modes, respectively.

To remove the current selection of drawing elements, use the *Edit/Select/Clear* command.   After choosing this command, no drawing elements are selected.

When selecting lines, polylines, arcs and circles with a width, you must make sure the center line of the element passes through the selection area to select it.

The following sections give a step by step description on:
Creating a New Selection
Selecting Elements One at a Time
Adding Elements to the Current Selection
Removing Elements from the Current Selection
Clearing the Current Selection

Creating a New Selection

There are two ways to create a new selection.

<u>One</u>

Choose the *Edit/Select/New selection* command.   The program displays the following prompt:

> Select first corner of selection area.

Move the cursor to the desired position and click and release the left mouse button or press Enter. The program will then display the prompt:

> Select second corner of selection area.

Move the cursor to the opposite corner of the desired selection area and click and release the left mouse button or press Enter.   Any previous selection is removed and all of the drawing elements within or crossing the selected area will be selected.

<u>Two</u>

Make sure neither the *Add* or *Remove* item is checked on the *Edit/Select* menu.   Click on a checked item to uncheck it.   Or, use the command tool bar to accomplish the same thing.   If either the add 🛑 or remove 🛑 button is pressed, click on it to unpress it.   Next, click on the first corner of the desired selection area and drag the cursor to the opposite corner and release the left mouse button.   Any previous selection is removed and all of the drawing elements within or crossing the selected area will be selected.

See also:
<u>Selecting Drawing Elements</u>

Adding Elements to the Current Selection

To add elements to the current selection, make sure the *Edit/Select/Add* menu item is checked. To check the item, choose it on the menu.   Or, to accomplish the same thing using the command tool bar, make sure the add button is pressed.   Click on it with the cursor if it is not pressed. Next, click on the first corner of the desired selection area and drag the cursor to the opposite corner and release the left mouse button.   All of the drawing elements within or crossing the selected area will be added to the current selection.

See also:
Selecting Drawing Elements

Removing Elements from the Current Selection

To remove elements from the current selection, make sure the *Edit/Select/Remove* menu item is checked.   To check the item, select it on the menu.   Or, to accomplish the same thing using the command tool bar, make sure the remove button is pressed.   Click on it with the cursor if it is not pressed.   Next, click on the first corner of the desired selection area and drag the cursor to the opposite corner and release the left mouse button.   All of the drawing elements within or crossing the selected area will be removed from the current selection.

See also:
Selecting Drawing Elements

Moving Drawing Elements

Elements within the drawing can be moved from one position to another by using a combination of the *Cut* and *Paste* commands on the *Edit* menu.   To move a drawing element, first select the element(s) you want to move.   See the section on <u>selecting drawing elements</u> for information on how to do this.

After selecting the desired drawing elements, choose the *Edit/Cut* command from the menu or press the cut button on the tool bar ✄.   The prompt area will display:

> Select cut reference point

The point you pick will correspond to the alignment point you select when you perform the paste operation.   Once the reference point has been entered, the selected elements will be erased from the display and deleted from the drawing.   If the selection is larger than 64k it cannot be cut to the clipboard.   You will get the message: *Selection too large!*

Next, choose *Paste* from the *Edit* menu.   The prompt line will display:

> Select paste alignment point

As mentioned previously, the paste alignment point corresponds to the cut reference point. When an alignment point is selected, the elements cut previously are drawn at the new position and added to the drawing.
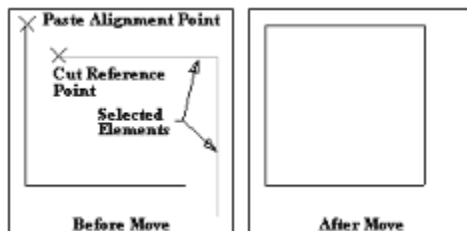
Moving Drawing Elements

The elements may be pasted as many times as desired at any location in the drawing area.   To end the paste command, choose another command from the menu or click the stop 🛑 button on the tool bar.

The cut operation copies the selected elements to the Windows clipboard. The paste operation retrieves them from the clipboard and inserts them into the drawing.   The *Paste* command is grayed out and unselectable if no data has been cut or copied to the clipboard.

This program uses a private data format when storing data in the clipboard.   This is because none of the currently defined clipboard formats provide enough information for drawing elements to be cut or copied to the clipboard and pasted back without loss of information.   This means drawing elements cut or copied to the clipboard cannot be pasted into other programs unless they specifically support the GammaCAD clipboard format.   Also, the GammaCAD *Paste* command cannot paste things from the clipboard which were cut/copied by another

program.   See the *Edit/Copy bitmap* command for information on how to copy a bitmap from GammaCAD to another program.

See also:
Copying Drawing Elements
Moving / Copying Drawing Elements to Another Drawing

Copy Bitmap

The *Copy Bitmap* command on the *Edit* pull down menu allows you to select a rectangular area in the drawing area to be copied to the Windows clipboard as a bitmap. The bitmap can be pasted into any program which allows you to paste a bitmap. After choosing this command, the prompt area will display:

Select first corner of bitmap area

When selecting the area, you input opposite corners of a rectangular area. You may pick any corner of the area first. After choosing the first corner, the first line of the prompt area becomes:

Select second corner of bitmap area

Now you must enter the diagonally opposite corner of the bitmap area rectangle. As you move the cursor, a box is displayed with one corner at your first selection point and the opposite corner at your current cursor position. This represents the area which would be used for the selection if the current cursor position were used for the second point.

When the second point is chosen, the program copies the selected area to the clipboard as a bitmap. To paste it into another program, change to that program and select *Paste* from its menu. Not all programs will paste a bitmap. See the documentation for the program you want to paste a drawing into for more information.

GammaCAD will not paste a bitmap from the clipboard. The GammaCAD *Paste* command will only paste drawing elements placed in the clipboard using the *Copy* or *Cut* options on the GammaCAD *Edit* menu.

Copying Drawing Elements

Elements within the drawing can be copied to another position by using a combination of the *Copy* and *Paste* commands on the *Edit* menu. To copy a drawing element, first select the element(s) you want to copy. See the section on <u>selecting drawing elements</u> for information on how to do this.

After selecting the desired drawing elements, choose the *Edit/Copy* command from the menu or press the copy button ▣→▣ on the tool bar. The prompt area will display:

> Select copy reference point

The point you pick will correspond to the alignment point you select when you perform the paste operation. Unlike the *Cut* command, the *Copy* command does not delete the selected elements. They remain in the drawing. If the selection is larger than 64k it cannot be copied to the clipboard. You will get the message: *Selection too large!*

Next, choose *Paste* from the *Edit* menu or press the paste button ▣ on the tool bar. The prompt line will display:

> Select paste alignment point

As mentioned previously, the paste alignment point corresponds to the copy reference point. When an alignment point is selected, the elements copied previously are drawn at the new position and added to the drawing.
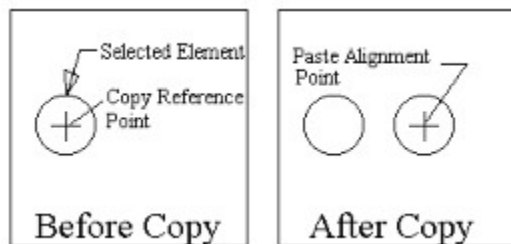
Copying Drawing Elements

The elements may be pasted as many times as desired at any location in the drawing area. To end the paste command, select another command from the menu or click on the stop button 🛑 on the command tool bar.

The copy operation copies the selected elements to the Windows clipboard. The paste operation retrieves them from the clipboard and inserts them into the drawing. The *Paste* command is grayed out and unselectable if no data has been cut or copied to the clipboard.

This program uses a private data format when storing data in the clipboard. This is because none of the currently defined clipboard formats provide enough information for drawing elements to be cut or copied to the clipboard and pasted back without loss of information. This means drawing elements cut or copied to the clipboard cannot be pasted into other programs

unless they specifically support the GammaCAD clipboard format.   Also, the GammaCAD *Paste* command cannot paste things from the clipboard which were cut/copied by another program.   See the *Edit/Copy bitmap* command for information on how to copy a bitmap from GammaCAD to another program.

See also:
Moving Drawing Elements
Moving / Copying Drawing Elements to Another Drawing

Moving / Copying Drawing Elements to Another Drawing

Drawing elements can be moved and copied between drawings as well as within the same drawing.   To do this, perform the cut or copy operation just as you do for moving/copying elements within the same drawing.   Then run another copy of the GammaCAD program and open the drawing to which you wish to move/copy the elements.   Paste the elements into that drawing.   Alternatively, instead of loading another copy of GammaCAD, you can open the drawing you want to paste the elements into within the current instance of the program.   The first method requires more memory but is much more convenient, especially if you need to perform multiple copy operations.

See also:
Moving Drawing Elements
Copying Drawing Elements

Deleting Drawing Elements

To delete drawing elements, first select the elements to delete (see the section on <u>Selecting Drawing Elements</u>).   Then choose *Delete* from the *Edit* menu or press the Delete key.   The selected elements are deleted from the drawing and erased from the drawing area.

Rotating Drawing Elements

To rotate drawing elements, select the elements to rotate (see the section on Selecting Drawing Elements).   Then, choose *Rotate* from the *Edit* menu.   The prompt area will display:

> Select rotation reference point

After picking a position, a dialog is displayed which allows you to enter the desired rotation angle.   Enter the angle in degrees counterclockwise and click on the *OK* button.   The selected elements will be rotated about the given center point by the given angle.
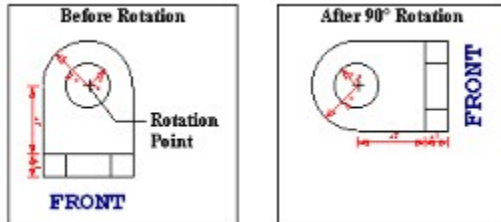
Rotating Drawing Elements

The program continues to prompt you for a rotation center point.   You can pick another point and rotate the elements again or end the command by clicking on the stop button on the command tool bar.

Scaling Drawing Elements

This command is used to increase or decrease the size of drawing elements.   First, you need to select the elements to scale (see the section on Selecting Drawing Elements).   Then, choose *Scale* from the *Edit* menu.   The prompt area will display:

Select scale reference point

After picking a position, a dialog is displayed which allows you to enter the desired scale factor. A factor greater than one increases the size of the selected elements and a factor greater than 0 but less than 1 reduces the size of the elements.   For example, a scale factor of 2 doubles the size of the elements.   A scale factor of 0.5 halves the size of the elements.   A scale factor less than 0 will cause the symbol to be mirrored across x and y axes centered at the reference point. When the desired scale factor has been input, click on the *OK* button.   The selected elements will be scaled about the given reference point by the input scale factor.
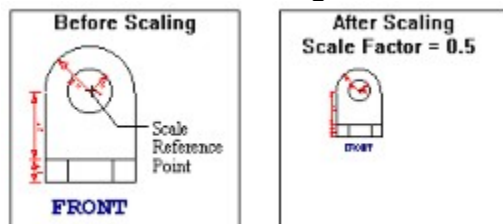
Scaling Drawing Elements

The program continues to prompt you for a scale reference point.   You can pick another point and scale the elements again or end the command by clicking on the stop button on the command tool bar.

Clearing the Current Selection

To remove the current selection of drawing elements use the *Edit/Select/Clear* command.   After choosing this command, no drawing elements are selected.

See also:
Selecting Drawing Elements

Listing Drawing Element Properties

With the *List* option on the *Edit* pull down menu you can display the properties of the selected elements (see <u>Selecting Drawing Elements</u>).   First, select the elements to list and then pick *List* from the *Edit* menu.   A dialog with the properties for the first element in the selection set is displayed.   Press *OK* to display the next element's properties or *Cancel* to end the command.

Trimming Lines, Arcs and Circles

The *Edit/Break end* command allows you to remove a section from the end of a line or arc.   The *Edit/Break middle* command allows you to remove a section from the middle of a line, arc or circle.

Break End

The *Edit/Break end* command is used to remove a section from the end of a line or arc.   When you choose this command, the prompt area displays:

   Select end of line or arc to break

The cursor is changed to the box cursor.   Move the cursor so the line or arc you want to break passes through the box.   The cursor should be closest to the end you want to remove.   Click the left mouse button or press Enter.   The prompt area displays:

   Select break point

Move the cursor to the position where you want the line to end and click the left mouse button or press Enter.
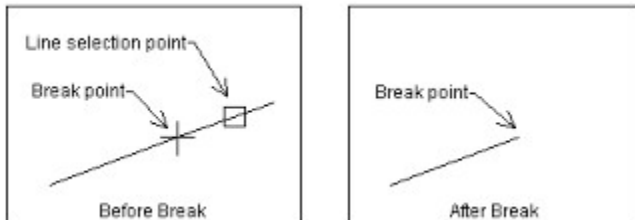
Break End Example

The program prompts you to select another line or arc to break.   You can select another or click on the stop button on the command tool bar to end the command.

Using the *Intersection* element snap to enter the break point, you can trim a line to another line.   Choose the *Edit/Break end* command and select the end of the line you want to trim.   Select the intersection snap mode before entering the break point and move the box cursor so it contains the intersection of the line you are trimming and the trim line.   You can also trim a line to an arc or circle using this method, as well as trim an arc to a line.
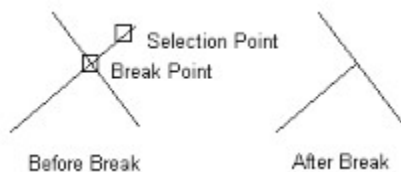
Trimming One Line to Another

Break Middle

The *Edit/Break middle* command is used to remove a section from the middle of a line, arc or circle.   You can choose this command by selecting it from the menu or by pressing the break button ![break button] on the command tool bar.   When you choose this command, the prompt area displays:

> Select a line, arc or circle to break

The cursor is changed to the box cursor.   Move the cursor so the line, arc or circle you want to break passes through the box.   Click the left mouse button or press Enter.   The prompt area displays:

> Select first break point

Move the cursor to the first end of the section you want to remove from the line, arc or circle and click the left mouse button or press Enter.   The program displays the prompt:

> Select second break point

Move the cursor to the other end of the section you want to remove and click the left mouse button or press Enter.
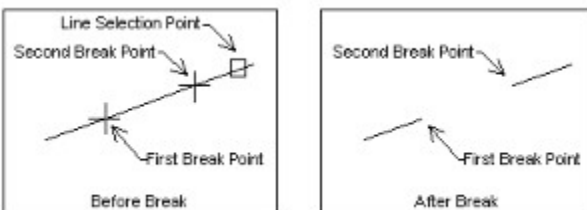
Break Middle Example

The program prompts you to select another line, arc or circle to break.   You can select another or click on the stop button ![stop button] on the command tool bar to end the command.

Using the *Intersection* <u>element snap</u> to enter the break points, you can trim a line to another line.   Choose the *Edit/Break middle* command and select the end of the line you want to trim.   Select the intersection snap mode before entering the break points and move the box cursor so it contains the intersection of the line you are trimming and the trim line.   You can also trim a line to an arc or circle using this method, as well as trim an arc to a line.
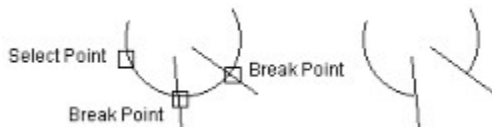
Trimming an Arc to Two Lines

When you break a circle, the order in which you choose the 2 break points determines whether the resulting arc has an interior angle greater than or less than 180 degrees.
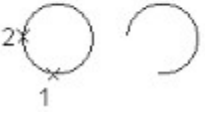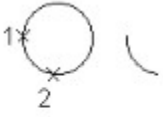
Breaking Circles

Extending Lines and Arcs

The *Edit/Extend* command is used to extend a line or arc.   A line can be extended to another line, arc or circle.   An arc can be extended to a line.   You can choose this command by selecting it from the menu or by pressing the extend button on the command tool bar.   When you choose this command, the prompt area displays:

> Select line, arc or circle boundary for extend

The cursor is changed to the box cursor.   Move the cursor over the line, arc or circle you want to use as the extend boundary and click the left mouse button or press Enter.   This is the element to which other elements will be extended.   If you selected a line for the extend boundary, the prompt area displays:

> Select line or arc to extend to the input boundary

If you selected an arc or circle as the extend boundary, the prompt displays:

> Select line to extend to the input boundary

Move the box cursor so the drawing element you want to extend passes through it.   Click the left mouse button or press Enter.
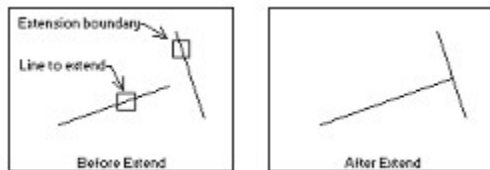
Extend Example

The element is then extended to the input boundary.   The program prompts you to select another drawing element to extend.   You can select another or click on the stop button on the command tool bar to end the command.

When extending a line to an arc or circle it can often be extended to two positions on the arc or circle.   The position closest to your selection point is used.   You cannot extend an arc to a circle.   If you select an invalid drawing element for the extension boundary or for an element to extend, the program will display a warning message.   If the drawing element which is selected for extension does not intersect the extension boundary, the element is not extended.

Filleting Two Lines

The *Edit/Fillet* command is used to fillet two lines.   Filleting adds a tangent arc of the given radius between the two selected lines.   The lines are trimmed to the ends of the arc.   After choosing this command, the following dialog is displayed
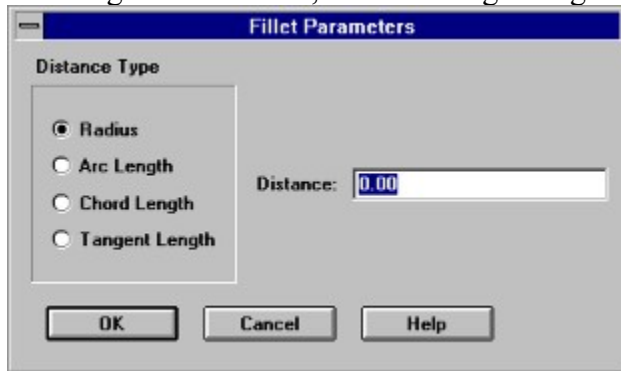
Fillet Parameters Dialog

This dialog sets the size of the fillet arc.   You may enter its radius, arc length, chord length or tangent length.   The *Distance Type* field sets what type of distance you are entering in the *Distance* field.   If you enter a value of zero for the distance, the fillet lines will be extended or trimmed as needed so they meet and no fillet arc is created.   Press the *OK* button when the desired parameters have been entered.

The program then displays the following prompt:

> Select first line to fillet

The cursor is changed to the box cursor.   Move the cursor so the line you want to fillet passes through the box.   Click the left mouse button or press Enter.   The prompt area then displays:

> Select second line to fillet

Select the second line in the same way as the first.

Fillet Example

The two lines are then filleted.   The program prompts you to select two more lines to fillet.

You can perform another fillet or click on the stop button on the command tool bar to end the command.   If the selected lines cannot be filleted using the input arc parameters (e.g. the arc is too large to fit within the lines), the program will display a warning message.

The positions you use to select the 2 fillet lines determines how the lines are filleted.   They

delineate the interior angle of the fillet.

Fillet Example 2

Converting Polylines to Line Segments

The break, fillet and extend commands will not work with polylines.   If you want to use one of these commands on two polyline segments, you can *Explode* the polyline.   This will convert each polyline segment to a line element.   The polyline element is removed from the drawing. The *Edit/Explode* command converts a polyline to line elements.   After choosing this command, the prompt area displays:

>   Select a polyline to explode

The cursor is changed to the box cursor.   Move the cursor so the polyline you want to explode passes through the box.   Click the left mouse button or press Enter.   The program will display a message box telling you the polyline has been converted to line elements.   If the element you chose is not a polyline, a warning message will be displayed.   The program continues to prompt you to select another polyline to explode.   You can explode another polyline or click on the stop button on the command tool bar to end the command.

Undoing Changes to the Drawing

The *Undo* command on the *Edit* pull down menu can be used to undo edit operations.   If there are no edit operations which can be undone, the *Undo* command is grayed and unselectable.

You can also choose the undo command by pressing the undo button on the command tool bar.

All of the commands on the *Edit* menu can be undone with the exception of the *Select*, *Copy*, *List, Redefine symbol* and *Copy bitmap* commands.   You also cannot undo an *Undo* command. The commands on the *Draw* menu can also all be undone.

Modifying the Dimension Arrow Head

The dimension commands insert the symbol contained in the file D_ARROW.SYM as the arrow head(s) on the dimension line.   You can change the arrow head like any other symbol.   See the section on <u>modifying symbols</u>.

Zooming In on an Area of the Drawing

To view an area of the drawing in more detail, use the *Zoom Window* command on the *View* menu.   With this command, you input a rectangular area within the drawing area.   That area is enlarged to cover the entire drawing area.
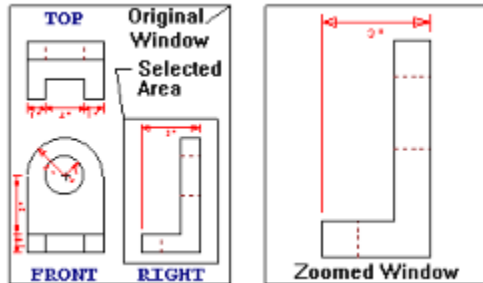
Zoom Window

When you choose *Zoom Window* from the *View* menu, press Ctrl+W or press the zoom window button ◻→◻ on the command tool bar, the prompt area changes to:

> Select first corner of zoom window

You may select any corner of the area you want to enlarge.   Once you have selected the first corner the prompt changes to:

> Select second corner of zoom window

As you move the cursor, a box is drawn from the first point entered to the current cursor position. This box represents the area which will be enlarged if the current cursor position is selected. When the second point is selected, the input area is enlarged as much as possible while still displaying the entire area selected.   If the area you input is of a different shape than the shape of the current drawing area, you may get some elements in the drawing area which were not in the area selected and the enlargement may not be as much as expected.   Keep in mind, this command displays the entire area selected at a minimum.   It will display more in one direction, if necessary, to display the entire area in the other direction.   The aspect ratio of the drawing elements is not changed.
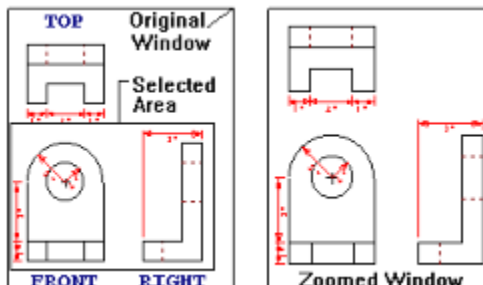

Effects of Zoom Window Shape

Displaying the Entire Drawing

To display all the elements in the drawing within the drawing area, use the *Zoom Extents* command on the *View* menu.   This command will display all the drawing elements as large as possible within the drawing area.   You may need to use this command more than once to get the drawing zoomed as large as possible if drawing elements have been moved or deleted since the last time the drawing was redrawn.

Restoring a Previous View

To display the previous view, choose *Zoom Previous* from the *View* menu, press Ctrl+R, or press the zoom previous button on the command tool bar.   If there is no previous view to restore, the *Zoom Previous* option is grayed and unselectable.   Up to 10 previous views are saved and can be restored.

Zooming In and Out

The drawing may be zoomed in (enlarged) and out (shrunk) using the *Zoom Factor* option on the *View* menu.   When you choose this option, a dialog box appears which prompts you for a zoom factor.   To zoom out, specify a number greater than 0 but less than 1.   To zoom in, specify a number greater than 1.   For example, specifying a value of 2 causes the drawing to be displayed at twice its current size.   Specifying a value of 0.5 causes the drawing to be displayed at half its current size.

The current center point of the display area remains in the center after the zoom.
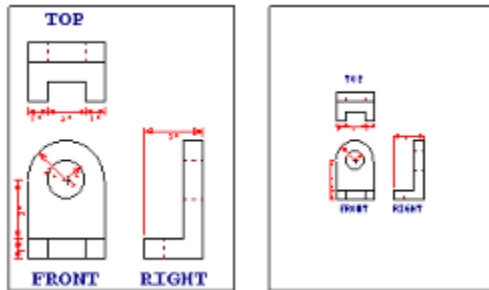
Zoom Factor (factor = 0.5)

Displaying the Page View

To change the view of your drawing to one in which the printer / plotter page fills the drawing area, choose *Zoom page* from the *View* menu.   See the section on page setup for a further description of the printer / plotter page and its size and position relative to your drawing.

Displaying the Page and Entire Drawing

To change the view of your drawing to one in which the printer / plotter page and the entire drawing is displayed in the drawing area, choose *Zoom page and extents* from the *View* menu. See the section on page setup for a further description of the printer / plotter page and its size and position relative to your drawing.

Pan

The drawing elements may be moved right, left, up and down in the drawing area while still maintaining the current zoom factor by using the *Pan* option on the *View* menu.   You can choose this option by selecting it from the menu or by pressing the pan button ⬚ on the command tool bar.   Choosing this option changes the prompt area to:

> Select first point defining pan movement

Pick a position within the drawing area you want to move elsewhere. The first line of the prompt area changes to:

> Select second point defining pan movement

Pick the position you would like the first point to be moved to within the drawing.   The drawing area will be redrawn with the drawing elements shifted by the specified amount.
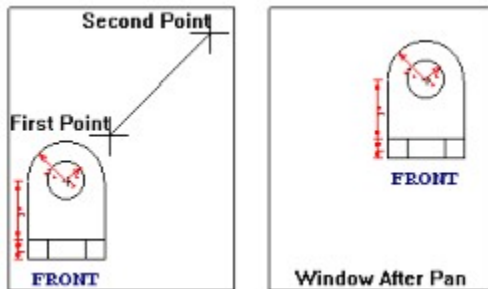
Pan

Redraw

During the course of drawing and editing you may get extra tic marks and such within the drawing area which are not actually part of the drawing.   To erase the drawing area and redraw only the elements actually in the drawing, choose *Redraw* from the *View* menu.

Entering Decimal Numbers

Many options in this program require you to enter coordinates or distances in drawing units. The format in which you enter these numbers is set by the *Options/Number format* command. When the format is set to *Decimal*, you must enter numbers in decimal format.   To enter a number in decimal format, you specify the number of whole drawing units, followed by the decimal character, followed by the fractional part (e.g. 1.234).   You may enter any number of decimal places you wish.   The program will read them all no matter what you have set the number of decimal places to using the *Options/Number format* command.   When the number is redisplayed, it will only contain the number of decimal places set by the *Options/Number format* command, although internally it will still contain all the decimal places.   If the number is being displayed in an editable field in a dialog and you press the *OK* button in the dialog, the value will be changed to the displayed value.   Any extra decimal places will be lost.   If you just want to view the values in a dialog and not change them, you should press the *Cancel* button to exit the dialog.

For example, if you enter a plot scale of 1.234 using the *File/Page setup/Print/plot scale* command the plot scale will be set to 1.234.   If you currently have the number of decimal places set to 2 and you choose the plot scale command again, it will display a scale of 1.23.   The program rounds the number to 2 decimal places in the dialog.   It is still stored as 1.234 but if you press the *OK* button in the plot scale dialog then the value is changed to 1.23.   In this case it might be best to set the number of decimal places to 3.

Entering Scientific Numbers

Many options in this program require you to enter coordinates or distances in drawing units. The format in which you enter these numbers is set by the *Options/Number format* command. When the format is set to *Scientific*, you must enter numbers in scientific notation.   To enter a number in scientific format, you specify the mantissa, followed by the letter "e" and then the exponent which is the power of 10 by which to multiply the mantissa.   The number of decimal places in the mantissa is set in the *Options/Number format* command.   For example, "1.23e2" represents 1.23x10x10 or 123.   The value 1.23 is the mantissa in this example and it is multiplied by 10 to the second power.   The coordinates are limited to the range -1e100 to 1e100. Coordinates within the range -1e-100 to 1e-100 cannot be entered.

Examples:
1.23e2   123.
1.23e-2   0.0123
-1.23e2"   -123.

Entering Architectural Numbers

Many options in this program require you to enter coordinates or distances in drawing units. The format in which you enter these numbers is set by the *Options/Number format* command. When the format is set to *Architectural*, you must enter numbers in architectural format. The program assumes your drawing is in inches when you select this mode. To enter a number in architectural format, you specify the number of feet (if any), followed by an apostrophe ('). Then you enter the number of whole inches (if any) followed by a space. Next enter the fractional number of inches. Separate the numerator and denominator by a forward slash (/). Do not use any spaces in the fraction. You may include double quotes (") at the end if you desire, but they are not required.

There must be a space between the whole number of inches and the fractional number of inches. For example, 2 and 3/4 inches must have a space between the 2 and the 3 (e.g. 2 3/4). There can be no spaces in the fraction (i.e. no spaces in 3/4). There must be an apostrophe following the number of feet. (e.g. 5').

Examples:

1/4"	1/4 inch
1/4	1/4 inch
2"	2 inches
2	2 inches
2 1/4"	2 and 1/4 inches
2 1/4	2 and 1/4 inches
1'	1 foot
1'1"	1 foot and 1 inch
1'1	1 foot and 1 inch
1'1 1/4"	1 foot and 1 and 1/4 inches
1'1 1/4	1 foot and 1 and 1/4 inches

File Menu

Move the cursor over the desired menu item and click the left mouse button or select the item from the list below.

New
Open
Save
Save as
Print/plot setup
Page setup
Print/plot
Dxf import
Dxf export
Exit

Draw Menu

Move the cursor over the desired menu item and click the left mouse button or select the item from the list below.

Draw	
Line	Ctrl+L
Arc	Ctrl+A
Circle	Ctrl+I
Point	Ctrl+P
Text	Ctrl+T
Insert symbol...	
Load symbol library...	Ctrl+O
Dimension	▶
Polyline	▶
Rectangle	
Fill / Hatch	
Stop	Esc

Line

Arc

Circle

Point

Text

Insert symbol

Load symbol library

Dimension

Polyline

Rectangle

Fill / Hatch

Stop

Edit Menu

Move the cursor over the desired menu item and click the left mouse button or select the item from the list below.

Edit	
Undo	Ctrl+Z
Select	▶
Cut	Ctrl+X
Copy	Ctrl+C
Paste	Ctrl+V
Delete	Del
Rotate...	
Scale...	
Change...	Ctrl+H
List...	
Break end	
Break middle	
Extend	
Fillet...	
Explode a polyline	
Change text	
Open/Close a polyline	
Redefine symbol...	
Copy bitmap	

Undo [STOP]

Select

Cut [STOP]

Copy [STOP]

Paste [STOP]

Delete

Rotate

Scale

Change

List

Break end

Break middle [STOP]

Extend [STOP]

Fillet

Explode a polyline

Change text

Open/Close a polyline

Redefine symbol

Copy bitmap

View Menu

Move the cursor over the desired menu item and click the left mouse button or select the item from the list below.

View	
Zoom window	**Ctrl+W**
Zoom extents	
Zoom previous	Ctrl+R
Zoom factor...	
Zoom page	
Zoom page and extents	
Pan	
Redraw	

Zoom window

Zoom extents

Zoom previous

Zoom factor

Zoom page

Zoom page and extents

Pan

Redraw

Options Menu

Move the cursor over the desired menu item and click the left mouse button or select the item from the list below.

Line properties
Text properties
Point properties
Fill / Hatch properties
Dimension properties
Layers
Set base point
Grid snap parameters
Number format
Display page
Display DXF elements
Program configuration
Enter coordinates

Color button
Type button
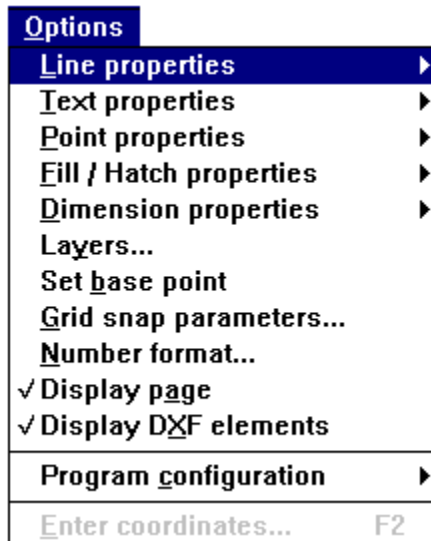Size button

Help Menu

Move the cursor over the desired menu item and click the left mouse button or select the item from the list below.

```
┌─────────────────────────────┐
│ Help                        │
├─────────────────────────────┤
│ Contents                    │
│ On current command  F1      │
│ Keyword search              │
│ Keyboard                    │
│ Commands                    │
│ Tutorials              ▶    │
│ Examples               ▶    │
│ Using help                  │
├─────────────────────────────┤
│ About...                    │
└─────────────────────────────┘
```

Contents

On current command

Keyword search

Keyboard

Commands

Tutorials

Examples

Using help

About

Ending a Command

Many of the commands in GammaCAD repeat.   In other words, after performing the operation once, the program prompts you to perform the same operation again until you end the command. Choosing a different command from the menu (except for the *Snaps*, *Options*, and *Help* commands) ends the current command.   You can end the current command by selecting *Draw/Stop*, pressing the escape key (Esc), or clicking on the stop button on the command tool bar.

Pick a Position

This is the easiest method of specifying a position in the drawing area when a command prompts you to enter a point.   All you have to do is position the cursor at the desired point in the drawing area using the mouse or keyboard arrow keys.   Then click the left mouse button or press Enter. The disadvantage to using this method is you cannot specify the exact coordinates of the point.

For example, if you wanted to pick a point at exactly (10,10), you could not do it using this method.   This is due to the fact the cursor moves in steps.   This method is best used when you do not care about the exact positions and dimensions of what you are drawing.

See also:
Specifying Positions

Coordinate Input Dialog

The coordinate input dialog allows you to specify a position in the drawing area when a command prompts you to enter a point.   It allows you to specify an exact position.   To use this method, click the right mouse button or press F2 while the cursor is in the drawing area (or choose *Enter Coordinates* from the *Options* pull down menu).   This will bring up a coordinate input dialog.   You use this dialog to enter the exact (x,y) coordinates of a drawing position. This method is used when you have the exact coordinates or dimensions of the object you are drawing.   The dialog is only displayed if the program is currently prompting you to enter a point.   In other words, you must first select a command.   When the program prompts you to enter a point, click the right mouse button to bring up the coordinate input dialog.
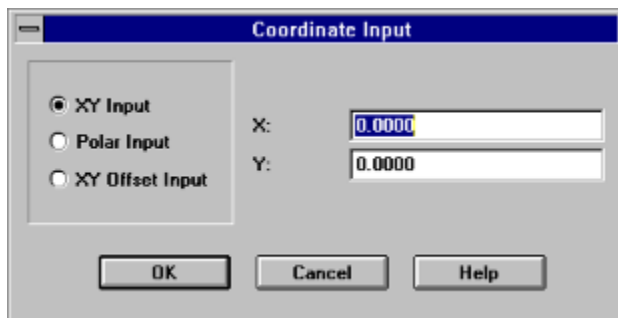
Coordinate Input Dialog

You may enter the xy coordinates of the point in this dialog.   You may also enter a distance and angle (*polar coordinates*) from the last point entered, or an xy offset from the last point entered. To choose the desired mode, click on the appropriate radio button in the left box.   The prompts before the two fields on the right will change depending on the mode you selected.   The next time this dialog is brought up it will be in the same mode.   After choosing the desired input mode, enter the two values on the right.   The program calculates the proper position and uses it as the next point for whatever you are currently drawing.   When entering the two values on the right, you must use the current number format as set by the *Options/Number format* command. See the Coordinate Input Examples section for examples on how to use this dialog.

See also:
Specifying Positions
Entering Decimal Numbers
Entering Scientific Numbers
Entering Architectural Numbers

Color Button

The color button on the command tool bar allows you to set the color for the drawing element you are currently creating.   You must choose an option from the *Draw* menu before this button becomes active.   For example, after selecting the *Draw/Line* command the color button is activated.   Clicking on this button will bring up the color input dialog and allow you to specify the color of the line you are about to create.   With this button you can specify the color of lines, arcs, circles, points, text, dimension lines, polylines, rectangles and filled areas.   The element you are setting the color for depends on the currently active draw command.

See also:
Line Color
Text Font
Point Color
Fill / Hatch Color

Type Button

The type button ⬛ on the command tool bar allows you to set the type for the drawing element you are currently creating.   You must choose an option from the *Draw* menu before this button becomes active.   For example, after selecting the *Draw/Line* command the type button is activated.   Clicking on this button will bring up the line type dialog and allow you to specify the type of the line you are about to create.   With this button you can specify the type of lines, arcs, circles, points, text, dimension lines, polylines, rectangles and filled areas.   The element you are setting the type for depends on the currently active draw command.

See also:
Line Type
Text Font
Point Type
Fill / Hatch Type

Help Contents

The *Help/Contents* command brings up the contents page of this help document.

Size Button

The size button on the command tool bar allows you to set the size of the drawing element you are currently creating.   You must choose an option from the *Draw* menu before this button becomes active.   For example, after selecting the *Draw/Line* command the size button is activated.   Clicking on this button will bring up the line width dialog and allow you to specify the width of the line you are about to create.   With this button you can specify the width of lines, size of points and text height.   The element you are setting the size for depends on the currently active draw command.

See also:
Line Width
Text Height
Point Size

Help on the Current Command

To get help on the current command select *Help/On the current command* from the menu, press the F1 key or click on the help button on the command tool bar.   This option is available only if a command is currently active.   For example, to get help on the line command, choose *Draw/Line* from the menu and then choose *Help/On the current command*.   The Windows help facility is loaded and you are positioned at the section describing the line command.

Help Keyword Search

The *Help/Keyword search* command brings up the windows help search facility and lets you search for a topic by keyword.

Help Using Help

The *Help/Help using help* command brings up the help file which describes using the windows help facility.

Help About

The *Help/About* command brings up a dialog which displays the program name, version and copyright information.

Changing Text Strings

The *Edit/Change text* command allows you to modify an existing text string. Only the text is changed, the font, justification, angle and orientation remain the same. After choosing this command, the prompt area displays:

Select a text string to edit

The cursor is changed to the box cursor. Move the cursor so the text string you want to edit passes through the box. Click the left mouse button or press Enter. The program will display a dialog box containing the current text string. Change the text to whatever you like and press the *OK* button. If the element you selected is not a text string a warning message will be displayed. The program continues to prompt you to select another text string to edit. You can edit another text string or click on the stop button on the command tool bar to end the command.

Open/Close a Polyline

The *Edit/Open/Close a polyline* command changes open polylines to closed and closed polylines to open.   After choosing this command, the prompt area displays:

>   Select a polyline to open/close

The cursor is changed to the box cursor.   Move the cursor so the polyline you want to open/close passes through the box.   Click the left mouse button or press Enter.   The program will close the polyline if it is open and open it if it is closed.   If the element you selected is not a polyline then a warning message will be displayed.   The program continues to prompt you to select another polyline to open/close.   You can open/close another polyline or click on the stop button on the command tool bar to end the command.

See the *Draw/Polyline* section for an explanation of open and closed polylines.

Open Polyline

To Draw an open polyline, all you have to do is choose the *Draw/Polyline/Open polyline* command from the pull down menu. The prompt area will display:

> Select first end point of open polyline

After specifying the first end point, a line is drawn from that point to the current cursor position as you move the cursor. This represents the first segment of the polyline which would be added to the drawing if the current cursor position was used for the second end point. The prompt area changes to:

> Select next end point of polyline

When the second end point is specified, a polyline segment with the designated start and end points is added to the drawing. It is displayed using the current line properties in effect. The program continues to prompt you for the end point of the next segment of the polyline. If you wish to add another segment which starts at the end of the previous one, select its end point.

Choosing another operation from the main menu (except those on the *Snaps*, *Options,* or *Help* menus) ends polyline input. You can also end polyline input by clicking on the stop button on the command tool bar or by pressing the Esc key.

See the section on Specifying Positions for more information on entering the polyline end points. See the section on Line Properties for more information on the different types of polylines which can be drawn. See the Polylines section for more information on polylines.

Display Page

To turn the display of the printer/plotter page on and off in the drawing area, choose the *Options/Display page* command.   If this item is checked, the page is displayed.   It it is not checked, the page is not displayed.

See also:
Page Setup

Closed Polyline

To Draw a closed polyline, all you have to do is choose the *Draw/Polyline/Closed polyline* command from the pull down menu. The prompt area will display:

>   Select first end point of closed polyline

After specifying the first end point, a line is drawn from that point to the current cursor position as you move the cursor. This represents the first segment of the polyline which would be added to the drawing if the current cursor position was used for the second end point. The prompt area changes to:

>   Select next end point of polyline

When the second end point is specified, a polyline segment with the designated start and end points is added to the drawing. It is displayed using the current line properties in effect. The program continues to prompt you for the end point of the next segment of the polyline. If you wish to add another segment which starts at the end of the previous one, select its end point.

Choosing another operation from the main menu (except those on the *Snaps*, *Options*, or *Help* menus) ends polyline input. You can also end polyline input by clicking on the stop button on the command tool bar or by pressing the Esc key. When the *Closed polyline* command is ended, a segment is drawn from the last point entered to the first point entered, closing the polyline.

See the section on Specifying Positions for more information on entering the polyline end points. See the section on Line Properties for more information on the different types of polylines which can be drawn. See the Polylines section for more information on polylines.

Display DXF Elements

A DXF file can contain certain entities which cannot be accurately displayed in GammaCAD. These entities are displayed with this symbol ⊞.   To turn the display of these DXF elements on and off in the drawing area, choose the *Options/Display DXF elements* command.   If this item is checked, the elements are displayed.   If it is not checked, these elements are not displayed.   You will have to redraw the drawing using the *View/Redraw* command to see the effects of this command.

See also:
Importing DXF Files

Monochrome Print/Plot

The *Options/Program configuration/Monochrome print/plot* command allows you to force all drawings to be plotted as monochrome.   When this option is checked and a drawing is printed/plotted, all colors of the drawing elements are changed to black.   The only exception to this is white drawing elements are still plotted as white.   The color of the drawing elements on the display, and as stored in the drawing file, are not changed.   This mapping of all colors to black is effective only during the print/plot process.   If the *Monochrome print/plot* option is not checked, the colors of the drawing elements are not changed and are sent to the printer/plotter with the same values as when they are drawn on the display.

Why would you want to use this option?   Some print/plot drivers will map colors to the closest color which they can display.   This can lead to some colors being mapped to white and thus they are not printed on the page.   If you are having trouble with drawing elements not showing up on the printout and you know they fall on the page, try checking the *Monochrome print/plot* option and reprinting the drawing.

See also:
Program Configuration

Grid Dot Size

The *Options/Program configuration/Grid dot size* command allows you to specify the size of the grid dots when a grid is displayed.   When you choose this command, the following dialog is displayed:
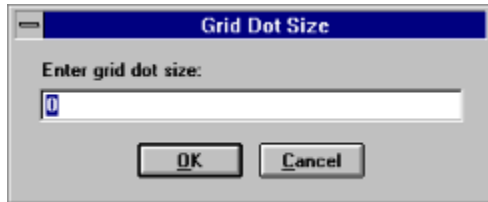
Grid Dot Size Dialog

This value must be greater than or equal to zero.   The larger the value, the larger the grid dot. When the desired value has been entered, press the *OK* button.   To exit this dialog without changing the current value, press the *Cancel* button.
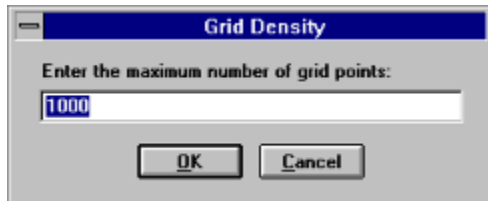
See also:
Program Configuration
Grid Snap Parameters
Grid Density

Grid Density

The *Options/Program configuration/Grid density* command allows you to specify the maximum number of the grid dots displayed in the drawing area.   This prevents the program from displaying the grid when the number of dots it would have to draw on the screen exceeds the input maximum.   The value which is best for you depends on how fast the grid dots are drawn on your system and at what point the grid dots tend to obscure the drawing area.   When the value is exceeded, the following message is displayed: *Grid too dense to display*.   This only affects the display of the grid.   You can still use the grid snap feature even though the grid is not visible.   If at some later point the grid can be displayed (e.g. you zoom in on the drawing), it will be redrawn.   To keep from getting this message, either increase the grid density value using this command or turn the grid display off.

When the *Grid density* command is chosen, the following dialog is displayed:

Grid Density Dialog

This value must be greater than or equal to zero.   The larger the value, the more grid dots the program will display in a grid.   When the desired value has been entered, press the *OK* button. To exit this dialog without changing the current value, press the *Cancel* button.   Do not enter a value of zero to turn off the grid display, instead turn the grid display off using the *Options/Grid snap parameters* command.
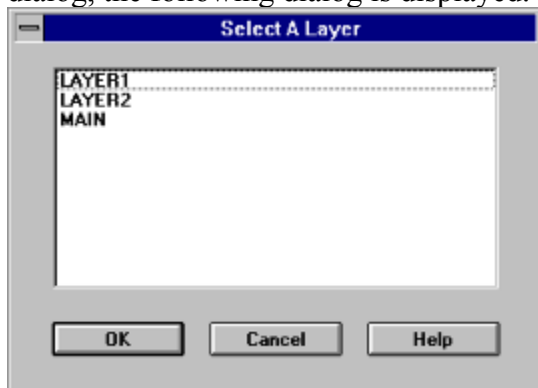
See also:
Program Configuration
Grid Snap Parameters
Grid Dot Size

Changing the Layer of a Drawing Element

When you select the *Edit/Change* command and then choose the *Layer* option in the change dialog, the following dialog is displayed.

```
┌─────────────────────────────────────┐
│ ▬        Select A Layer             │
├─────────────────────────────────────┤
│  ┌───────────────────────────────┐  │
│  │ LAYER1 ........................│  │
│  │ LAYER2                        │  │
│  │ MAIN                          │  │
│  │                               │  │
│  │                               │  │
│  │                               │  │
│  │                               │  │
│  └───────────────────────────────┘  │
│                                     │
│  ┌────────┐  ┌────────┐  ┌────────┐ │
│  │   OK   │  │ Cancel │  │  Help  │ │
│  └────────┘  └────────┘  └────────┘ │
└─────────────────────────────────────┘
```

Change Layer Dialog

This dialog is used to change the layer for all the elements which are currently selected in the drawing.   Double click on the layer on which you wish the selected elements to be placed.   You may also click on the layer once and press the *OK* button.   Use the up and down arrow keys to select it with the keyboard and then press the Enter key.   Press the Cancel button to end this command without changing the layer for the currently selected drawing elements.

Prototype Drawing

Whenever GammaCAD creates a new drawing, it looks in the GammaCAD directory for a file with the name GCAD.GC1.   This is the prototype drawing.   If the file is found, it is opened and read like any other drawing.   The properties in this drawing will thus be set in the new drawing. The name of the drawing after choosing the *File/New* command will still be *Untitled* whether or not a prototype drawing exists.   When you save it, you will be prompted for a drawing name.

The prototype drawing can be used to set a default value for any property which is saved with a drawing.   All of the properties on the *Options* menu are set with the exception of the *Program Configuration* options.   Printer and page setup are also set by the prototype drawing.

To create or modify a prototype drawing, select *File/New* from the menu.   If any previous prototype drawing has been created, its defaults will be used for the new drawing.   Next, use the *Options*, *File/Print/plot setup* and *File/Page setup* commands to set the desired values for the prototype drawing.   Then save the drawing to the file GCAD.GC1 in the GammaCAD directory. The next time you create a new drawing, the new prototype drawing properties will be used.

Selecting Elements One at a Time

The *Edit/Select/Select one* command allows you to select elements one at a time.   When you choose this command, the prompt area displays:

> Select a single element

The cursor is changed to a box cursor.   Move the cursor so the element you wish to select passes through the box cursor and click the left mouse button.   The first drawing element the program finds is selected.   This option only selects one element, so if two or more elements pass through the box cursor, only one is selected.   This is usually the one which was drawn first.   The program continues to prompt you to select another element.   You can select another element or end the command by clicking on the stop button ![stop] on the command tool bar.   Element snaps do not affect the selection points you pick with this command.

The current selection mode also affects this command.   This means this command can be used to add elements to the current selection one at a time.   It can also be used to remove elements from the current selection one at a time.   See the section on selecting drawing elements for a description of setting the selection mode.

This command can be used to select one element when two elements fall directly on top of each other.   For example, suppose you drew a series of 3 connected line segments.   At some later point you decide you would prefer they were a polyline.   You draw a polyline directly on top of the line segments using the *end* element snap.   Then you remove the lines by checking the *Edit/Select/Add* option and choosing the *Edit/Select/Select one* command.   Next, select each line segment.   Since you are using the *Select one* command which selects only the first element drawn, the line segments and not the polyline are selected.   When all segments are selected, press the delete key to delete them.   The program erases the line segments which also erases the polyline.   Choose *View/Redraw* to see the polyline is still there.

See also:
Selecting Drawing Elements

Line Properties Menu

Move the cursor over the desired menu item and click the left mouse button or select the item from the list below.
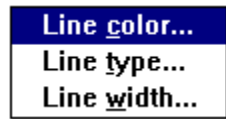
Line color
Line type
Line width

Page Setup Menu

Move the cursor over the desired menu item and click the left mouse button or select the item from the list below.

Fit drawing to page
Print/plot scale
Position drawing on page

Text Properties Menu

Move the cursor over the desired menu item and click the left mouse button or select the item from the list below.
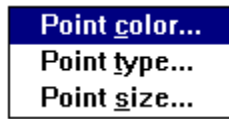

Text font
Text height
Text angle
Text orientation
Text justification

Point Properties Menu

Move the cursor over the desired menu item and click the left mouse button or select the item from the list below.

Point color...
Point type...
Point size...

[Point color](#)
[Point type](#)
[Point size](#)

Fill / Hatch Properties Menu

Move the cursor over the desired menu item and click the left mouse button or select the item from the list below.

Fill / Hatch color

Fill / Hatch type

Dimension Properties Menu

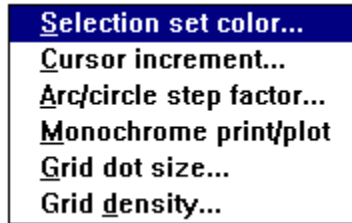Move the cursor over the desired menu item and click the left mouse button or select the item from the list below.

Dimension arrow scale...
Dimension offset...

[Dimension arrow scale](#)
[Dimension offset](#)

Program Configuration Menu

Move the cursor over the desired menu item and click the left mouse button or select the item from the list below.

```
Selection set color...
Cursor increment...
Arc/circle step factor...
Monochrome print/plot
Grid dot size...
Grid density...
```

<u>Selection set color</u>
<u>Cursor increment</u>
<u>Arc/circle step factor</u>
<u>Monochrome print/plot</u>
<u>Grid dot size</u>
<u>Grid density</u>

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Graphical Index

To use this index move the cursor over the part of the program window in which you are interested.   If the cursor changes to a pointing hand, then you can click the left mouse button and move to that topic within this document.
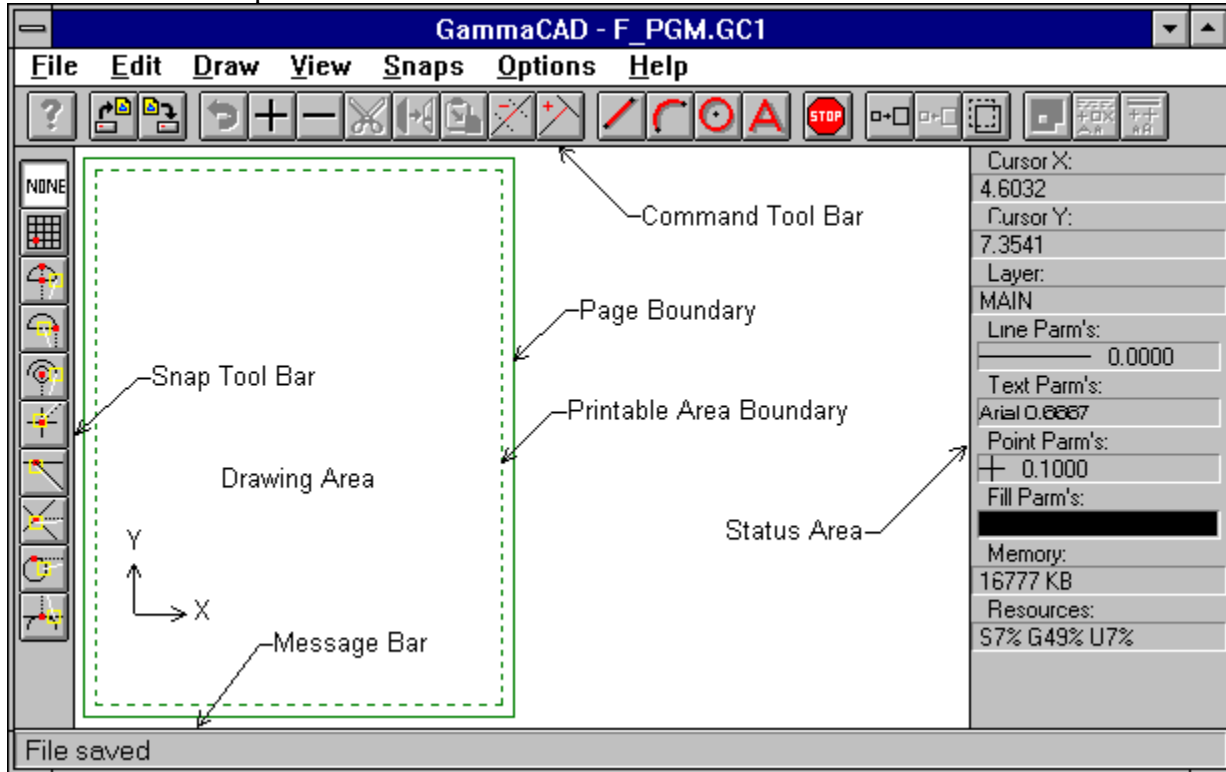
This is the prompt or message area where the program prompts you for input or displays messages.

This is the status area where the program displays the current xy coordinates and the line, point, text and fill properties for the current layer.

Help Tutorials

This command brings up the tutorials sub menu.

Help Examples

This command brings up the examples sub menu.