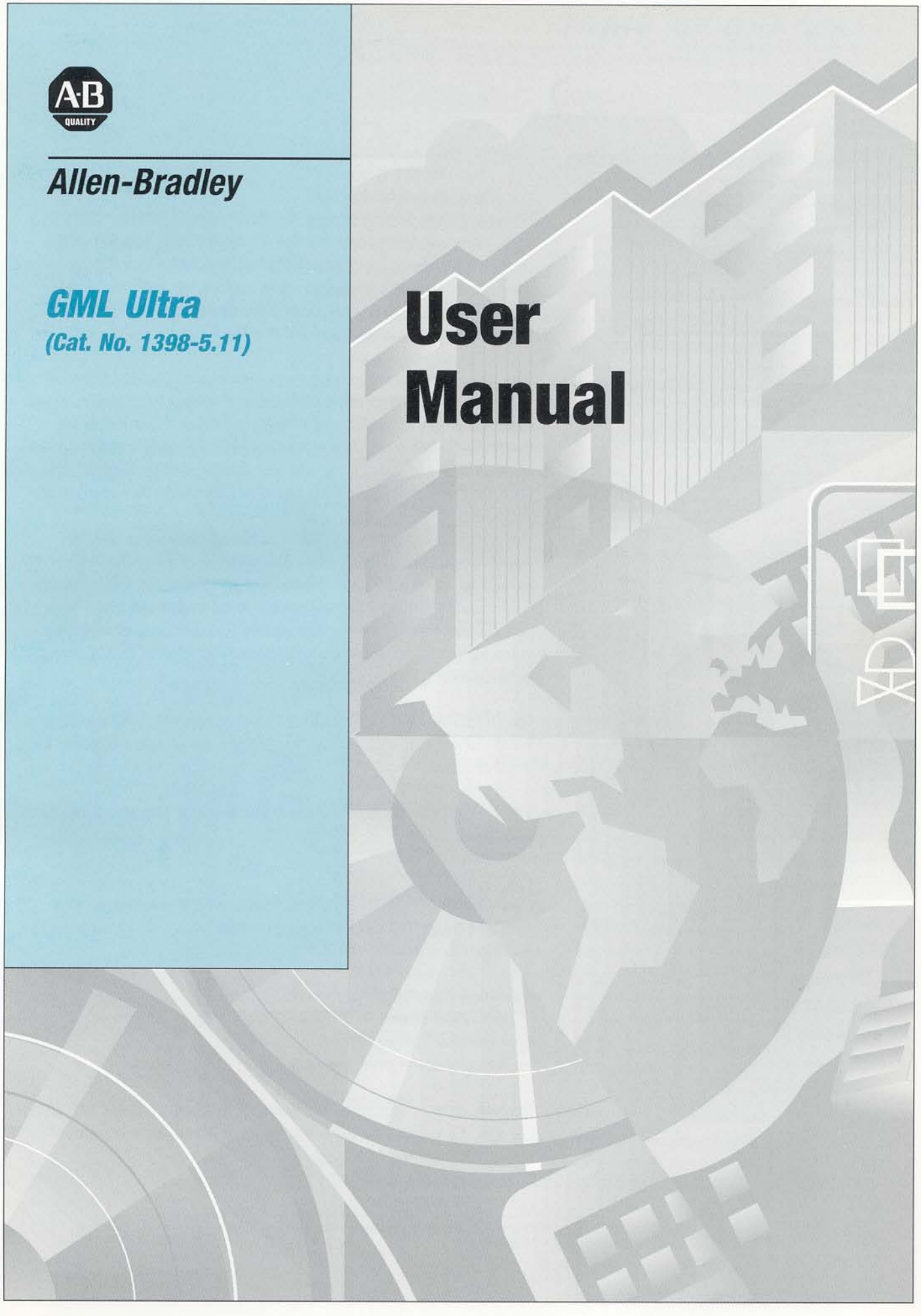
Allen-Bradley

GML Ultra

(Cat. No. 1398-5.11)

User Manual



Important User Information

Because of the variety of uses for the products described in this publication, those responsible for the application and use of this control equipment must satisfy themselves that all necessary steps have been taken to assure that each application and use meets all performance and safety requirements, including any applicable laws, regulations, codes, and standards.

The illustrations, charts, sample programs and layout examples shown in this manual are intended for purposes of example. Since there are many variables and requirements associated with any particular installation, Allen-Bradley office does not assume responsibility or liability (to include intellectual property liability) for actual use based upon the examples shown in this publication.

Allen-Bradley publication SGI-1.1, *Safety Guidelines for the Application, Installation, and Maintenance of Solid-State Control* (available from your local Allen-Bradley office), describes some important differences between solid-state equipment and electromechanical devices that should be taken into consideration when applying products such as those described in this publication.

Throughout this manual we use notes to make you aware of safety considerations:

⚠ **ATTENTION:** Identifies information about practices or circumstances that can lead to a personal injury or death, property damage or economic loss.

Attention statements help you to:
- Identify a hazard
- Avoid the hazard
- Recognize the consequences

Important: Identifies information that is critical for successful application and understanding of the product.

Preface

GML Ultra Overview — Chapter 1

Before you Begin — Chapter 2

General Procedures — Chapter 3

Working With Blocks Chapter 4

Working with Diagrams — Chapter 5

Working With Modules — Chapter 6

Working with Scripts — Chapter 7

Going Online — Chapter 8

Understanding Blocks

Chapter 9

Preface

Read this preface to familiar yourself with this manual. This preface covers the following topics.

- Who should use this manual
- The purpose of this manual
- The contents of this manual
- Related documentation
- Where to find help
- Technical support
- Common techniques used in this manual

Who Should Use this Manual

Use this manual if you are responsible for designing, testing, or debugging GML Ultra® diagram programs used with ULTRA Plus® controllers. GML Ultra is a member of the GML™ family.

Purpose of this Manual

This manual is a user guide for the GML Ultra programming tool. It describes the procedures you use to design a diagram; define the parameters of each block function; and download, test, and debug the diagram, all using the graphical interface unique to GML.

Contents of this Manual

Chapter	Title	Contents
1	*Overview of GML Ultra*	This chapter provides an introduction to GML Ultra.
2	*Before You Begin*	This chapter describes what you need to know about GML Ultra before you build your first diagram. It includes: Descriptions of the GML Ultra screen; what it looks like, how it works, what options are available to make your diagram easy to build, easy to copy, easy to find, and easy to troubleshoot. Descriptions of GML Ultra menus and toolbars. Descriptions of function block libraries.

Chapter	Title	Contents
3	*General Procedures*	This chapter contains step-by-step procedures you use to create and edit diagrams and scripts.
4	*Working with Blocks*	This chapter covers the physical aspect of blocks, such as selecting, moving, and using blocks to define functions for your diagram. It also describes how to use the Expression Builder feature.
5	*Working with Diagrams*	This chapter provides procedures for creating, testing, and documenting your diagram.
6	*Working with Modules*	This chapter includes procedures for creating, viewing, and documenting modules.
7	*Working with Scripts*	This chapter provides procedures for using the Script Editor, translating a diagram to script, and printing the script.
8	*Going Online*	This chapter includes procedures for downloading a diagram, debugging and fine tuning, and uploading controller options.
9	*Understanding Blocks*	This chapter gives a brief explanation of each block's function and directions for using each block. For detailed information about blocks, refer to the *GML Ultra Reference Manual.*
	Index	

Where to Find Help

GML Ultra provides two types of help:

- A set of user manuals.
- Online help.

Using the Manual Set

This manual is part of a documentation set for GML Ultra:

- *GML Ultra Getting Started* (publication 1398-5.10).
- *GML Ultra User Manual* (publication 1398-5.11).
- *GML Ultra Reference Manual* (publication 1398-5.12).

Use this manual	To find information on this topic
GML Ultra Getting Started	• Installing the software • Starting the software • Setting up the system • Connecting the hardware
GML Ultra User Manual	• Basic features and functions • Screen, menu, and toolbar functions • Mechanics of working with blocks, modules, and diagrams • Basic block definitions
GML Ultra Reference Manual	• Setup details • Expression Builder details • Block function details

Using Online Help

You have access several types of online help:

To use this	Do this	Description
GML help	Select **Help** from the menu bar.	• Detailed descriptions of all menus and screens. • Creating a diagram • Error messages • Troubleshooting information • Getting the ULTRA Plus started • PSM diagnostics • Post command language
Block descriptions	Double-click on a block to open its dialog box. Double-click the Library block to open the Block Library.	A brief description of a block's function is found in: • The block's dialog box • The block library
Definition descriptions	From the Definitions menu, select a setup procedure.	Each setup dialog box contains a brief description of the definition you are configuring.

Related Documentation

The following documents contain additional information concerning related Allen-Bradley products. To obtain a copy, contact your local Allen-Bradley office or distributor.

For	Read this Document	Publication Number
Information regarding ULTRA Plus hardware	*ULTRA Plus Installation and Setup Manual*	1398 - 5.1
An overview of the ULTRA Series family	*ULTRA Series Digital Servo Drives*	1398 - 1.0
An overview of ULTRA Plus Positioning Servo Drives	*ULTRA Plus Positioning Servo Drives*	1398 - 1.1
Descriptions and specifications for the ULTRA family	*ULTRA Series Product Data*	1398 - 2.0
An article on wire sizes and types for grounding electrical equipment	*National Electrical Code*	Published by the National Fire Protection Association of Boston, MA.
A complete listing of current Allen-Bradley documentation, including ordering instructions. Also indicates whether the documents are available on CD-ROM or in multi-languages	*Allen-Bradley Publication Index*	SD499
A glossary of industrial automation terms and abbreviations	*Allen-Bradley Industrial Automation Glossary*	AG - 7.1

Rockwell Automation Support

Rockwell Automation offers support services worldwide.

Local Product Support

Contact your local Allen-Bradley representative for:

- Sales and order support
- Product technical training
- Warranty support
- Support service agreements

Technical Product Assistance

If you need technical assistance, first review the information in the *Troubleshooting* chapter of the *GML Ultra User Manual*. If you need more information, call your local Allen-Bradley representative.

For the quickest possible response, we recommend that you have the catalog numbers of your products available when you call. Refer to *Where to Find Help* for the publication numbers related to this product.

The Rockwell Automation Technical Support number is **(216) 646-6800.**

Common Techniques Used in this Manual

The following conventions are used throughout this manual:

- Bulleted lists provide information, not procedural steps.
- Numbered lists provide sequential steps.
- Words that you type or select and keys that you press appear in **bold.**
- Field names and references appear in *italics*.
- Warnings appear with the following symbol:

ATTENTION: This warning identifies information about practices or circumstances that can lead to personal injury or death, property damage, or economic loss. This symbol indicates a situation that requires immediate attention for personnel safety or for preventing harm to machinery .

- **IMPORTANT:** Identifies information that is critical for successful application and understanding of the product.
- The term “select” means that you use your mouse cursor to point to the value, then click-and-release the left mouse button to mark your choice. Depending on the field, you can select one or more options for a field. The options appear in various formats:
  - Sometimes you must browse through a list to find the value you want. Clicking an option in the list highlights your selection.
  - Sometimes you can select several values in one area. A check in a box  is used when you can select more than one option.
  - Sometimes only one value is allowed. A radio button  is used when you can select only one option.
  - When you select a block in your diagram, the block is highlighted. You can now:

- Move the block by clicking and dragging, rather than clicking and releasing.
- Open the dialog box by double-clicking.

In all cases, the term indicates your choice to GML Ultra.

GML Ultra Overview

GML, the exclusive Graphical Motion Control Language from Allen-Bradley, provides a graphical method of programming your motion controller. This revolutionary tool reduces the time you need to learn motion control programming and makes problem-solving easier. GML integrates software programming and debugging to solve your motion control problems. GML Ultra, a member of the GML family, is a Microsoft® Windows®-based interface to the ULTRA Plus series of controllers.

How it Works

GML Ultra uses a flowchart approach to motion control programming. To produce a diagram (motion program), you place function blocks (representing the specific actions) on the screen, then connect them in the proper order of operations.

Later you enter motion and process limits, using a fill-in-the-form approach. Each block has its own form, which you select with a few clicks of the mouse. You can enter and change any block at any time, minimizing the need to consult a manual or understand any specialized syntax.

After you complete the diagram, you download it to a controller, where it is translated into a program or script in the native language of the motion controller.

You need a PC for program development but you do not need it for the final turn-key application.

Encapsulating, which combines two or more blocks into a module, is a unique function of the GML Ultra graphical interface. You can duplicate the module, use it again in the same diagram, use it in another program, or save it as a diagram file in a library folder for use in subsequent diagrams.

Motion Controllers that Use GML Ultra

GML Ultra is designed for the Allen-Bradley ULTRA Plus series of controllers, a family of 1½ axis digital servo and positioning servo drives designed for a wide range of applications.

The ULTRA Plus series Positioning Drive Module (PDM) combines a drive and a controller to create a powerful, self-contained, 1 ½ axis programmable motion controller.

Personality Module for ULTRA Plus Controllers

The Personality Module (PM) is a nonvolatile memory device that stores the information necessary to customize an ULTRA Plus controller for a specific application. The PM holds parameters to match the motor and the controller, as well as user programs and parameters. A Personality Module can be physically removed and transferred to another ULTRA Plus controller if the replacement of a controller is necessary to simplify servicing the machine. The Personality Module data can also be saved in a computer file and loaded into the ULTRA Plus controller using the upload options in Online Manager.

Communications for ULTRA Plus Controllers

The following basic communication links are pre-installed in all ULTRA Plus controllers.

- RS-232
- RS-422
- Multi-drop

RS-232

RS-232 is an industry-defined electrical interface standard for serial communication.

RS-422

RS-422 is a hardware standard that defines a method for transmitting high-speed serial data over very long distances using a balanced, twisted-pair transmission line.

Multi-drop mode

All ULTRA Plus controllers can be configured to operate in multi-drop mode. This mode allows up to ten controls to share a single RS-422 communication link by using two special non-echoed commands to activate individual units. The individual units then respond to commands issued by the operator interface device or by the host computer.

Before you Begin

Before you begin using GML Ultra, read this chapter to become familiar with:

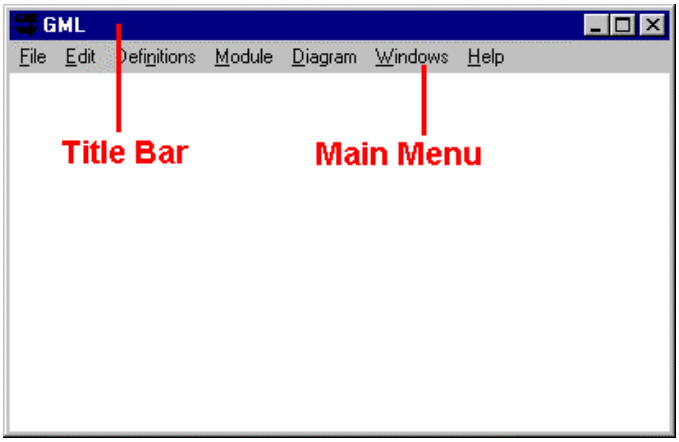
- Your work area
- Your tools

Understanding Your Work Area

GML Ultra provides several work areas made up of windows. Some windows are active only when you are online. Sometimes a window, or a portion of a window, changes function, depending on other options, such as your operating mode.

GML Ultra Application Window

The application window, illustrated below, is the basic GML Ultra window. It provides the main menu for accessing the GML Ultra functions and features. Refer to the *Understanding Your Tools* section in this chapter for descriptions of the title bar and the main menu.



This window is always visible when GML Ultra is open, but you never see it alone. When you open the application, a New Diagram window appears immediately after the GML Ultra application window opens. When you are online, the Online Manager window also appears here.

GML Ultra Diagram Window

The GML Ultra diagram window provides the workspace for using the Diagram Editor to create, edit, and test programs. It also provides access to:

- all the menus and toolbars
- the block palette, definitions menus, setting breakpoints, translating diagrams to script function, and the Online Manager

The illustration below shows a diagram window on top of the application window

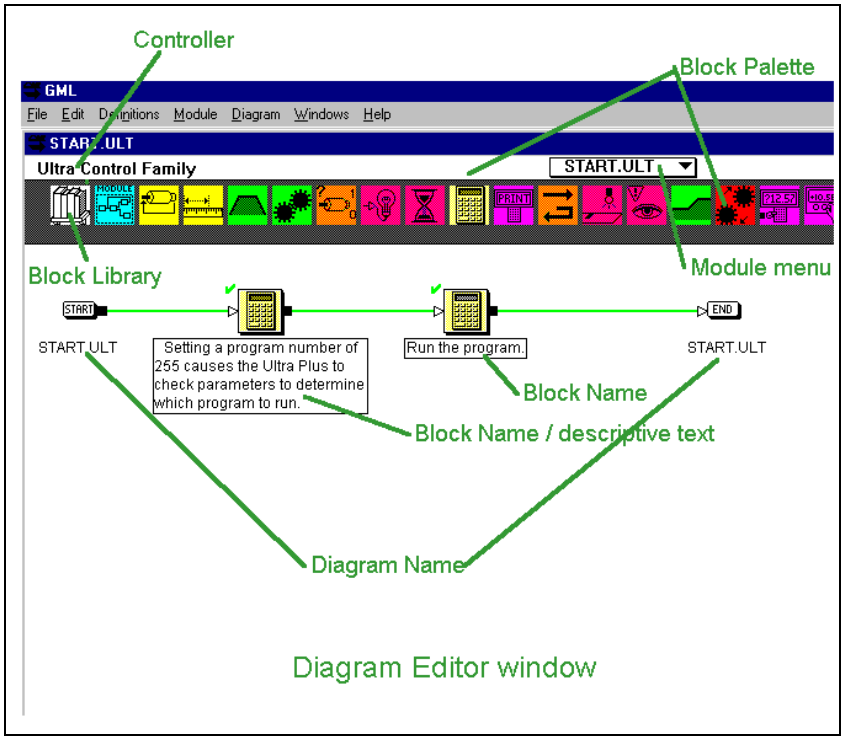


Diagram Editor Window

A new Diagram Editor window, illustrated above, appears each time you open GML Ultra and each time you open a new diagram.

You use the Diagram Editor to create, edit, translate, and test your GML Ultra application programs. You can create or edit programs in this graphical environment, then transparently translate them into their native language.

Although several Diagram Editor windows can be open simultaneously, only one can be active at one time.

Online Manager Window

Use the Online Manager to communicate with your motion controller. The Online Manager window provides real-time communication with the motion controller through the scripted program. It allows you to exchange information and control processing. It also provides a venue to monitor and supervise the execution of commands and partial or total programs. Tools (such as breakpoints) are provided for graphical testing and debugging.

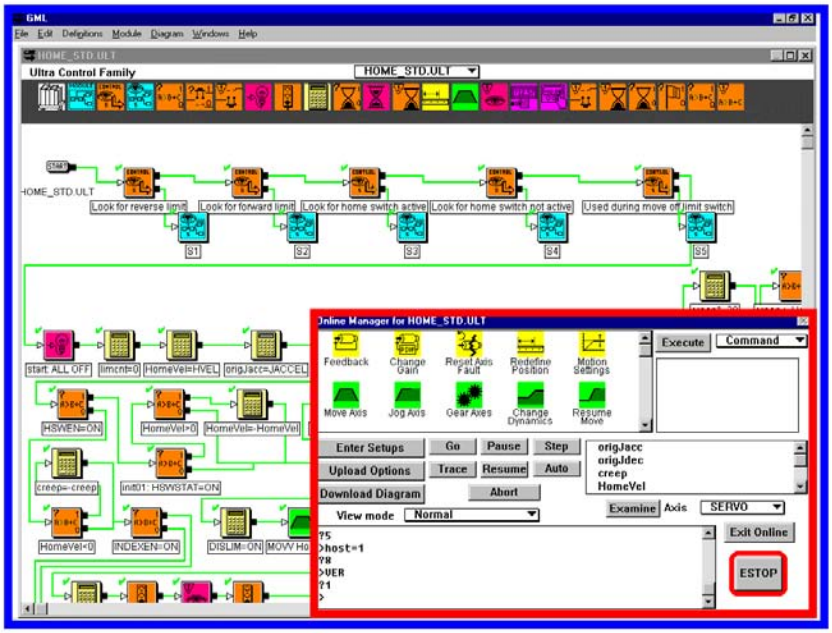
Two additional windows are available when you are online; the Trace window and the Watch window

- In the Trace window, you can visually follow program execution. GML Ultra highlights each block or module as the program in the motion controller executes that block. For more information, refer to the Trace Window section in this chapter.
- In the Watch window, you can watch a command function as it executes in a program. For more information, refer to the *Watch Window* section in this chapter.

Accessing the Online Manager

To communicate with your motion controller:

1. From the menu bar, select **Diagram**. The Diagram menu appears.
2. Select **Online**. The Online Manager window similar to the following appears; communication with the motion controller is enabled.



Moving the Online Manager Window

You can move the Online Manager window to a more convenient place on your screen.

To move the Online Manager window:

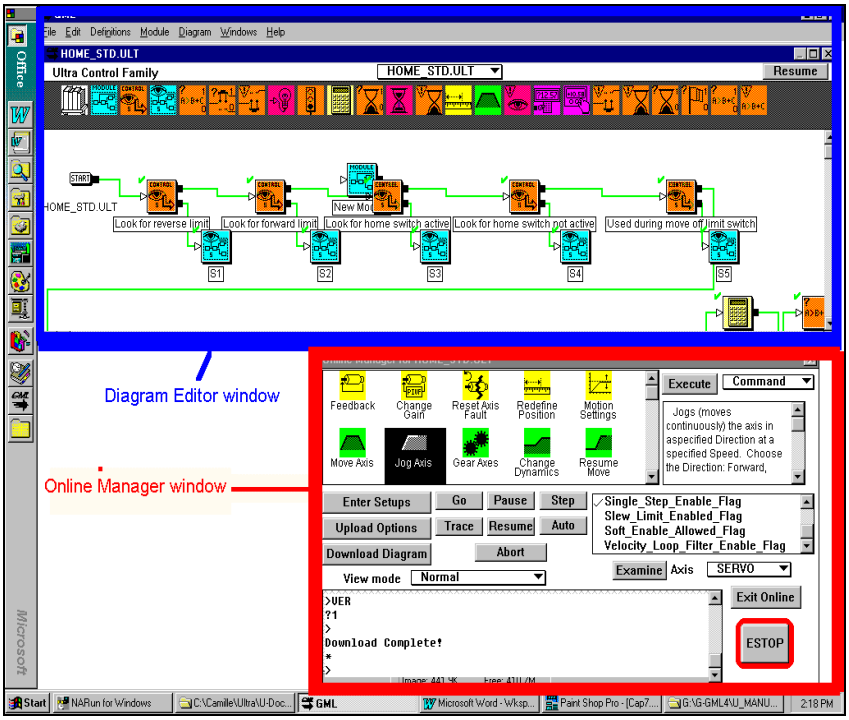
1. Click in the title bar and hold the mouse button down as you drag the window to a new position.

2. Release the mouse button.

Note: You cannot resize the Online Manager window.

Viewing Both Diagram Editor and Online Manager

You can move easily between the Diagram Editor window and the Online Manager window. If you click on your diagram while the Online Manager window is open, the diagram moves to the front and the Online Manager window moves to the back. We recommend that you resize your diagram window so both windows are visible at the same time, similar to the following:



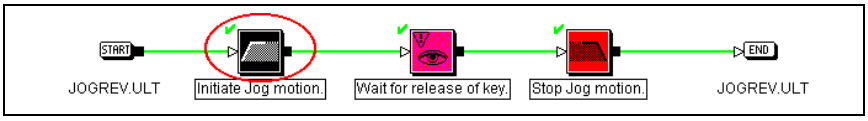
For information on using the Online Manager to communicate with your controller, refer to the *Going Online* chapter.

Trace Window

The Trace window looks similar to the Diagram Editor window, which it replaces when you are online. In the Trace window, however, each block or module is highlighted as the program (partial or total) in the motion controller executes that block, allowing you to visually follow program execution. Tools such as breakpoints are provided for graphical debugging.

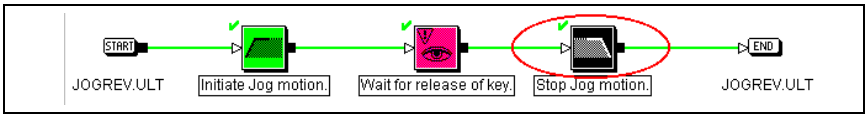
The following graphic shows the Trace window as:

The first block executes:

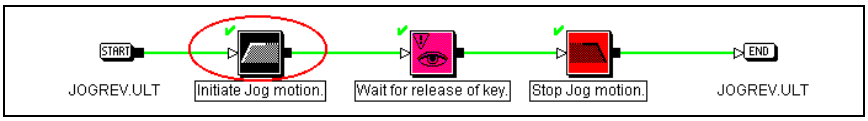


The second block executes:

The third block executes:

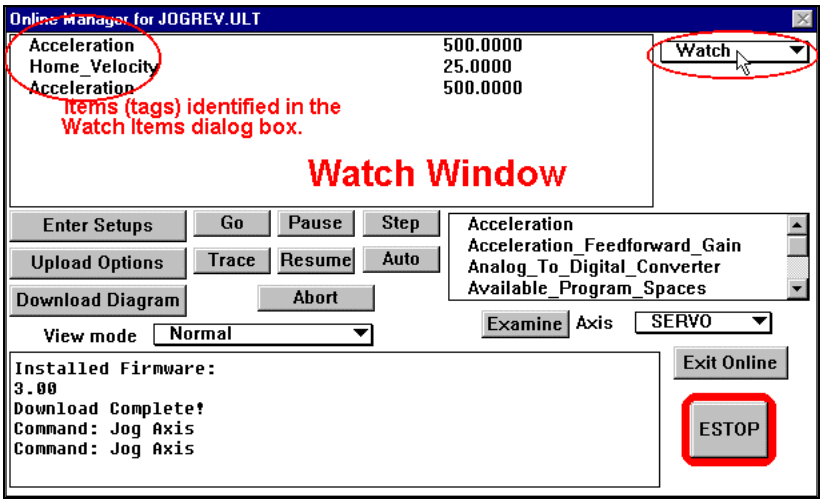


The first block executes again:

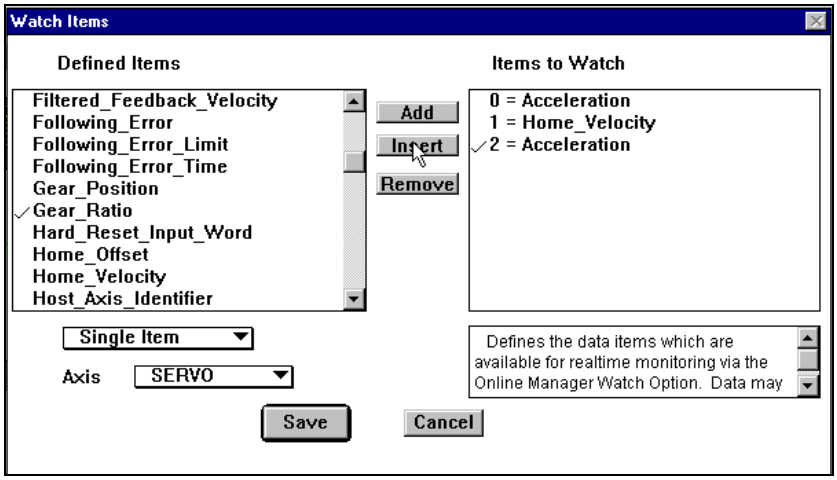


Watch Window

When working online, you use the Watch window to monitor a variable as it executes in the program.



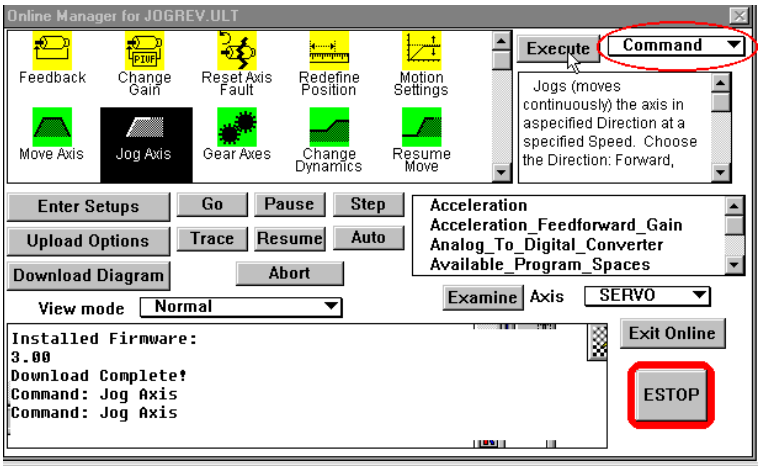
If you have not already identified the items you want to watch, a dialog box alerts you to do so. The following Watch Items dialog box is similar to the one you use to select the tags you want to monitor.



Command Window

When you are working online, use the Command window to:

- View the command and/or response links with the controller.
- Execute block functions.



Script Editor Window

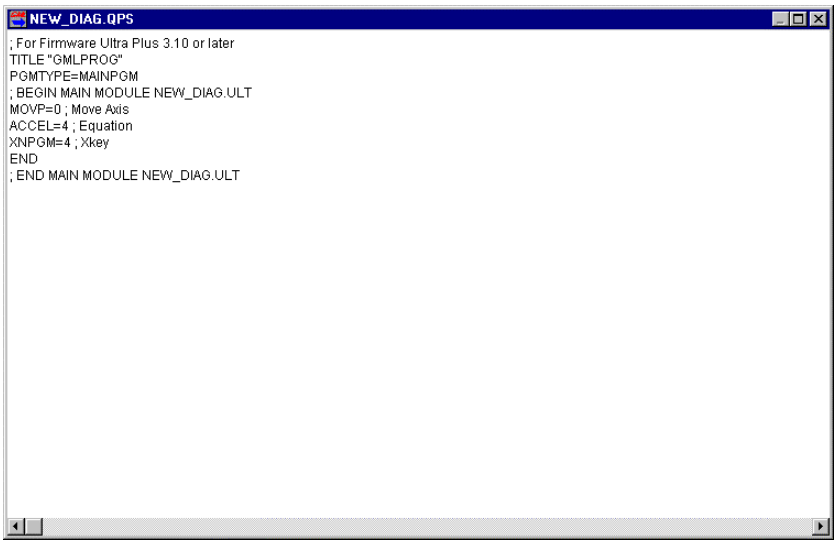
Some of you are comfortable identifying problems from the script version of the diagram. Use the Script Editor to view and edit text files, including script documents generated from GML diagram documents.

Note: Your computer must be online to download a script.

ATTENTION: If you make changes to the script, they are not converted back to the graphical diagram. Once you make a change to a program's script, you cannot go back to working in GML mode. We suggest that you use script only for viewing, not editing.

To access the Script Editor window:

1. Make sure the diagram you want to translate is active.
2. From the menu bar, select **Diagram**. The Diagram menu appears.
3. Select **Translate to Script**. Upon successful translation, a script file similar to the following appears in the Script Editor window.



For more information on using the Script Editor to view and fine tune your program, refer *Translating to Script*.

Quick Reference to GML Ultra Windows

Work area	Purpose	Available Off line	Available Online
Application Window	Access to the main menu for GML Ultra functions and features.	✓	✓
Diagram Window	Workspace for using the Diagram Editor to create and edit diagrams.	✓	✓
Diagram Editor Window	Provides functions and features to create and edit diagrams.	✓	
Online Manager Window	To communicate in real-time with your motion controller.		✓
Trace Window	Online, to monitor the sequence of blocks executing in a program. For debugging.		✓
Watch Window	Online, to monitor a variable as it executes in a program.		✓
Command Window	Online, to monitor a command and/or response link with the controller.		✓
Script Editor window	To view and edit text files, including script documents generated from GML documents.	✓	✓

Understanding Your Tools

Title Bar

The GML Ultra title bar shows the name of your diagram along with the size and close controls. Use the Title Bar to locate a diagram in the window. If you have not yet named or saved your diagram, the diagram is temporarily named New Diagram. Subsequent unnamed diagrams are also named New Diagram. For example, when you open a new diagram, you see the following:

Refer to *Naming Your Diagram*, in the *Working With Diagrams* section.

After you save a diagram, the name changes, similar to the following:

Note: GML Ultra automatically adds the **.**ULT suffix to the name of your diagram.

Main Menu

The main menu is at the top of the window. It provides access to the primary GML Ultra functions.

Refer to *Using the Diagram Menu* for a description of each menu option.

Block Palettes and Libraries

The colored building blocks are the graphical elements you use to create a diagram.

For instructions on selecting and placing blocks in a diagram, refer to the *Working with Blocks* chapter.

For a brief description of each block, refer to the *Understanding Blocks* chapter.

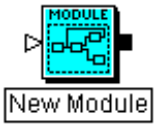
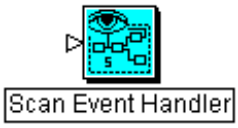
For detailed information on blocks, refer to the *GML Ultra Reference* manual.

Block Palette

The on-screen palette shows the most recently used blocks.

Block Libraries

The Block Library varies depending on the type of diagram you are building. There are three block libraries.

This specialized module	Does this	Contains this set of blocks
	Places a new blank module on the diagram. You can use this module in two ways: When you use this module once, it acts as the container for the entire diagram. The control code it produces is placed in the main program if you do not define modules in the call table. For example, a new diagram is really a New Module block. When you encapsulate or add a New Module block to your diagram, a sub-routine is created and a call to that sub-routine occurs with each use.	All GML Ultra blocks.
	Places a new blank Scan Event Handler module in the diagram. Scan Event Handlers monitor the condition of scan events. If the condition is true and the scan event is enabled when scanned, the actions are performed in parallel with program execution. You can enable and disable scan events in the application.	Blocks that are relevant to scan events.
	Places a new blank Xkey Handler module in the diagram. Xkey Handlers perform their actions in parallel with program execution. The keys on the Operator Terminal marked X1 through X4 are special purpose keys that, when pressed, cause the execution of an Xkey Handler.	Blocks that are relevant to Xkey functions.

Color Code for Blocks

GML uses color to group blocks by function.

Color	Function
Green	Initiate or change motion
Red	Stop motion
Yellow	Change setting or configuration
Brown	Affect program flow control
Pink	Provide I/O control
Violet	Provide operator interface functions
Light Green	Provide communications facilities
Light Yellow	Provide calculations, tables, and other similar functions
White	Provide miscellaneous functions that are not available in the categories above

General Procedures

This chapter contains step-by-step procedures for tasks you use frequently while programming in GML Ultra. These tasks include the following:

- Starting GML Ultra
- Creating new diagrams and scripts
- Opening diagrams and scripts
- Saving and naming a diagram
- Naming blocks and modules
- Printing diagrams and scripts
- Closing a file
- Exiting GML Ultra

Starting GML Ultra

To start GML Ultra, double-click on the GML Ultra icon.

The GML Ultra application window appears with a new, active, unnamed diagram window.

Creating New Files

One of GML Ultra's time-saving features is its ability to use existing diagrams to create new ones, and to save diagrams or parts of diagrams for this purpose. Also, GML Ultra supplies a library of generic diagrams/modules that are designed for use in your diagrams.

You can create a new file in two ways:

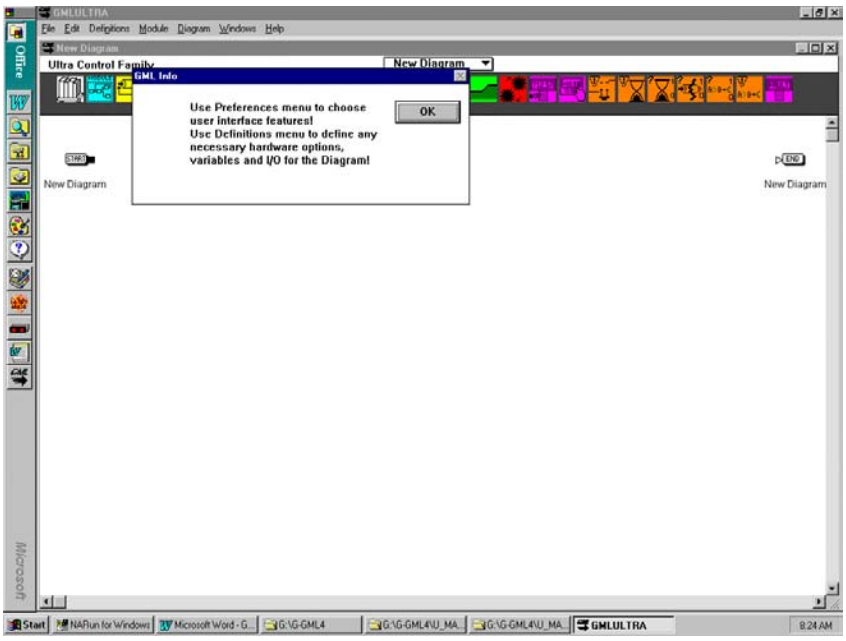
- Select New Diagram or New Script from the File menu.
- Save an existing diagram or script with a new name using Save As from the File menu.

Creating a New Diagram

When you have a diagram open, you can start to work on another. Your application is already active — you just need a new Diagram Editor window to build a new diagram. Use this option to open a new GML Ultra Diagram Editor window in which you build a new diagram.

1. From the menu bar, select **File**. The File Menu appears.

2. Select **New Diagram**. A new GML Ultra diagram window similar to the following appears:



For a detailed explanation of this window, refer to the *Before You Begin* chapter.

Using the Diagram Menu

The diagram menu located at the top of the window provides access to the primary GML Ultra functions.

This menu	Gives you access to
File	Standard Windows functions (such as open, save, print and close) that apply to GML Ultra diagrams and files. In addition it has one important GML Ultra function, Preferences. Use Preferences to set general program parameters that remain constant in your application environment. These parameters do not change unless you elect to change them. You need not address this option if the settings for your current diagram are no different from your previous diagram.
Edit	Options that pertain to the Diagram Editor and the development of your diagram. Use this menu to make normal editing changes (such as cut, copy, paste) to your diagram as you develop it. You can also make other types of edits, such as moving blocks.

This menu	Gives you access to
Definitions	Dialog boxes that you use to tell your computer about your motion controller. The GML Ultra menus are customized based on the information provided.
Module	Options for encapsulating blocks into modules and accessing module details.
Diagram	Statistical information such as number of modules and blocks in the diagram. Access to diagram functionality, such as testing, online connection, translating to script, and the find function.
Windows	All open GML Ultra windows from which you can select the one you want to view.
Help	GML Ultra's online help functions.

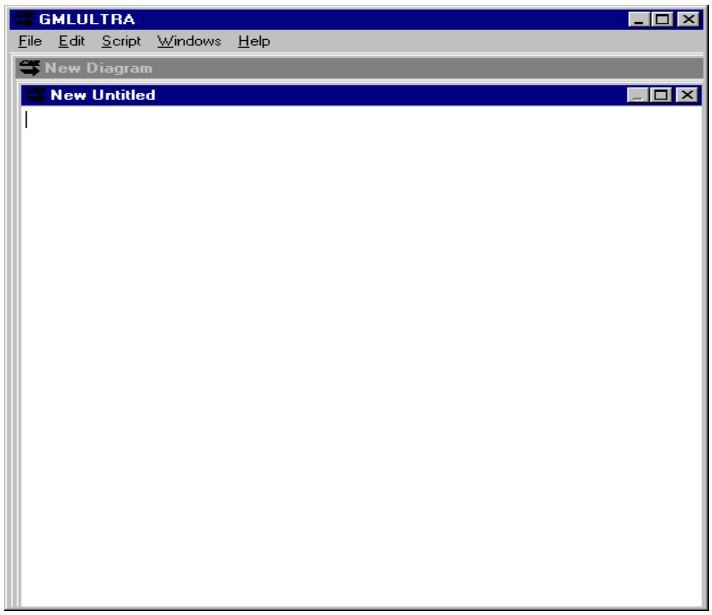
Creating a New Script

A script is the text equivalent of a graphic-based diagram. Use the New Script option to write a program using script instead of using the graphical interface.

IMPORTANT: We recommend that you use the New Diagram option to develop new programs.

To create a new script document:

1. From the menu bar, select **File**. The File menu appears.
2. Select **New Script.** A GML script window appears.



Using the Script Menu

The Script window displays menu options pertaining to writing, importing, and downloading scripts.

This menu option	Provides access to these features
File	Open new and existing diagrams and scripts, print functions, and general GML Ultra system settings.
Edit	Standard text editing functions.
Script	Find/Replace.
Windows	A list of active diagrams and scripts.
Help	GML Ultra online help.

Opening Existing Files

Opening a Diagram

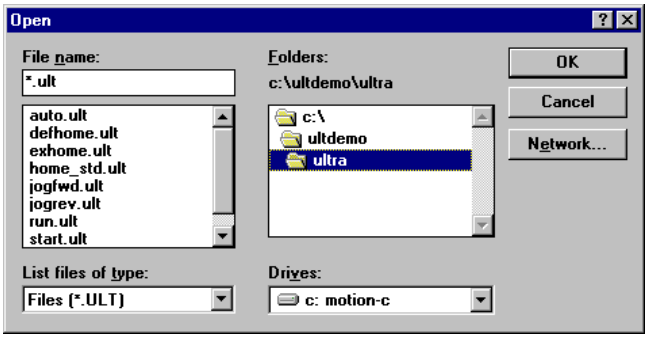
You can access previously written and saved functional modules (saved as diagrams) and entire diagrams to build your new diagram. When the diagram opens, you can:

- Edit or further develop a previously saved diagram.
- Copy formatted blocks, modules, or larger portions of the diagram for use in another diagram (a primary benefit of GML's graphical design structure.)

To access a diagram:

1. From the menu bar, select **File**. The File menu appears.

2. Select **Open Diagram.** The Open dialog box appears.



3. Locate the diagram file you want to open.

   Note: The file must have a **.**ULT extension.

4. Select the diagram you want by double-clicking on the name of the file. The file opens and the Diagram Editor window appears.

   You can now:
   - Make changes to the diagram.
   - Copy portions or blocks.
   - Convert the diagram to script.
   - Download the diagram to your controller.

Opening a Script

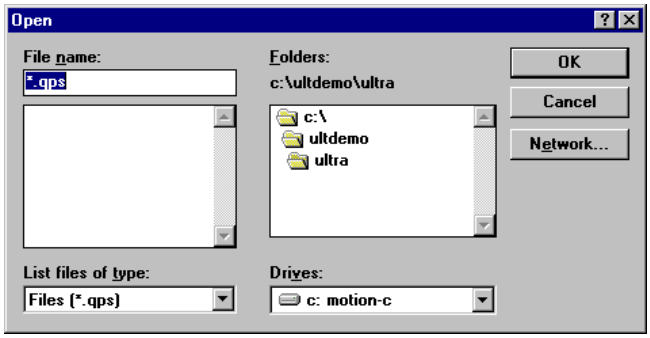
Use the Open Script option to open a saved script. You can then copy all or portions of the script as you build your new program. When the existing script opens, you can:
- Edit or further develop the existing script.
- Copy portions of the script for reuse in another script.

To access an existing diagram:

IMPORTANT: Once you convert a diagram to script, the procedure cannot be reversed. The script text is the actual language used for the motion controller program.

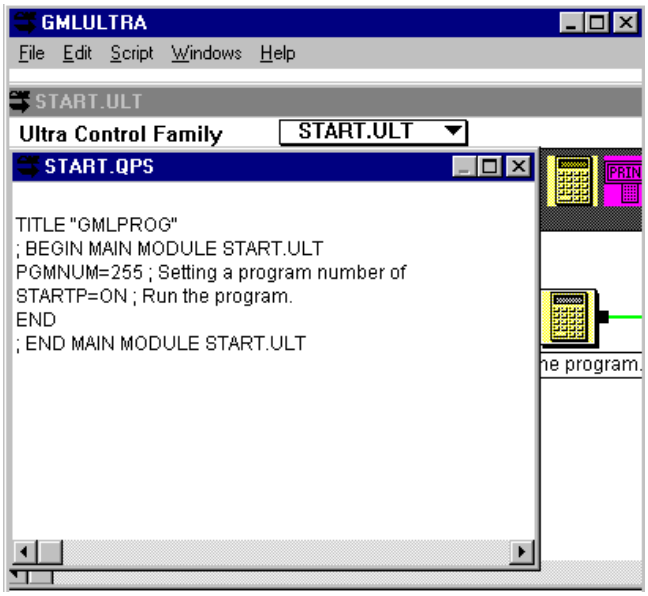
1. From the menu bar, select **File**. The File Menu appears.

2. Select **Open Script.** The Open dialog box appears with ***.**qps text files identified the *List files of type* field, similar to the illustration below.



3. Select the file you want.

4. Select **Open.** The script file opens in the window similar to the illustration below.



You can now:

- Make changes to the script
- Select and copy portions of the script to another file
- Download the script

Editing GML Ultra Files

It is common to make changes as you work on a diagram. Use the following options on the Edit menu to make the changes.:

- Cutting blocks
- Clearing information
- Deleting blocks
- Copying information
- Pasting information
- Selecting a diagram
- Duplicating part of a diagram
- Undoing an action
- Redoing an action
- Editing a script

Making Changes to Your Diagram

All the tools you need to make changes to your diagram are available through the Edit menu.

Cut, Clear, and Delete are Not the Same

This command	Allows undo	Allows redo	Allows paste
Cut	✗	✗	✗
Clear	✗	✗	
Delete	✗		

Cutting Blocks from Your Diagram

To cut, or remove, selected information from your diagram:

1. Select the information you want to remove.
2. From the menu bar, select **Edit**. The Edit menu appears.
3. Select **Cut**. The information is removed from your diagram and stored temporarily on a clipboard.

At this point you can either:

- Ignore it and consider the information permanently deleted from your diagram.
- Paste the information into another diagram, or a different place in the same diagram.
- Undo the delete action to restore the selected parts.

Clearing Information from Your Diagram

Clear deletes the selected blocks, but does not put them on the clipboard so it is not available for pasting.

To clear selected information from your diagram:

1. Select the block or blocks you want to clear. The blocks are highlighted.
2. From the menu bar, select **Edit**. The Edit menu appears.
3. Select **Clear**. The information is deleted from your diagram.

At this point you can either:

- Ignore it and consider the information permanently deleted from your diagram.
- Undo the delete action to restore the selected parts.

Deleting a Block from a Diagram

If you place a block in the Diagram Editor window and then change your mind, you can delete it.

To delete a block:

1. Select the block.

2. Press **Delete**.

3. At this point you can either:
- Ignore it and consider the information permanently deleted from your diagram.
- Undo the delete action to restore the selected parts.

Copying Blocks

You can copy any block or group of blocks along with their associated parameters. Copying has no effect on the original diagram, leaving the selected blocks exactly as they were. When you copy a module, the copied information includes all the blocks, connections, parameters, and other modules contained in the module.

The primary purpose of Copy and Paste is to allow you to repeat information in another location easily.

 **ATTENTION:** Not all the information is transferred.
- Axis name is not transferred.
- Use of variables is not transferred.

You can copy information from one diagram and paste it into another:
- Diagram.
- Location in the same diagram.

To copy:

1. Select the information you want to copy.

2. From the menu bar, select **Edit.** The Edit menu appears.

3. Select **Copy**. The information is stored on a clipboard.

   Note: The information remains on the clipboard until you perform another Copy or Cut.

Pasting Information into Your Diagram

The information that you want to paste can originate from either a Cut or a Copy. GML Ultra allows you to paste the information multiple times.

To paste the contents of the clipboard in the same diagram window that you cut or copied from:

1. Place your cursor at the location at which you want to put the information.
2. Click the left mouse button.
3. From the menu bar, select **Edit**. The Edit menu appears.
4. Select **Paste**. The copied information appears in the diagram.

   Note: The information on the clipboard remains there until you cut or copy something else, which then replaces the previous information. The clipboard only holds one item at a time.

Selecting a Whole Diagram at Once

This option selects all the elements in the Diagram Editor window. Select All is most often used in combination with other editing changes to your diagram, such as aligning and spacing.

To select the entire diagram:

1. From the menu bar select **Edit**. The Edit menu appears.
2. Select **Select All**. Every block, module, and connection is highlighted.

Note: At this point the information is NOT copied.

Duplicating Part of Your Diagram

Duplicate creates an identical block or set of blocks in the same diagram. Those blocks can then be moved to a different location in the diagram.

To create an identical block or group of blocks:

1. Select the information to be duplicated. The block(s) are highlighted.
2. From the menu bar, select **Edit**. The Edit menu appears.
3. Select **Duplicate**. A second block or set of blocks appears in the diagram. The new set is highlighted.
4. Drag the new blocks to the new location.

Undoing the Most Recent Action

You can undo an action if you make a mistake or change your mind. You only have one chance to undo and you must perform the undo *immediately* after the action you want to undo.

To undo your most recent action.

1. From the menu bar , select **Edit**. The Edit menu appears.
2. Select **Undo**. The diagram returns to the way it was immediately before your action.

   Note: You can undo only one action.

Redoing Your Last Action

To redo your last action:

1. From the menu bar, select **Edit**. The Edit menu appears.
2. Select **Redo**. The last action you performed on your diagram is repeated.

Note: If you select Redo after an undo, the action you undid is redone.

Note: Undo and Redo go backwards one step.

Editing Your Script

Use the Script Editor window to view and edit text files, including script documents generated from GML Ultra diagram documents. Script Editor provides standard cut, copy, paste, delete, and find/replace functions.

 **ATTENTION:** Changes to the script cannot be converted back to the graphical diagram.

For more information on using scripts, refer to the *Working with Scripts* chapter.

Saving and Naming a New File

Use the following procedures for saving files:

- Saving and naming a diagram

- Saving changes to a diagram
- Changing a Diagrams name and location

Saving and Naming a New Diagram

IMPORTANT: Save and name your new diagram, even though it is empty.

The first time you save your diagram, you are required to:
- Name the diagram.
- Place the diagram in a specific folder, in a specific location.

Saving Changes to a Diagram

We recommend that you save any changes to the diagram (new blocks, new modules, new connections, new parameter settings.)

To save changes to a diagram.

1. From the menu bar, select **File**.

2. Select **Save**. All changes since the last save are saved.

Changing a Diagram's Name and Location

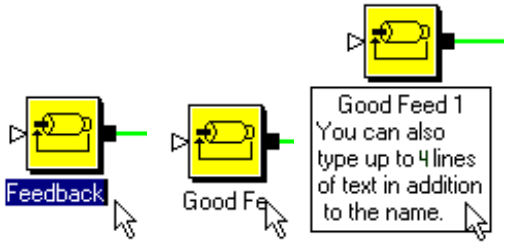
You can save the current diagram under a different name and/or in a different location.

Naming a Block

When you first create a block, the description below it is generic. You probably want to rename it using a descriptive name reflecting the block's function within this program. When you place the block, the description below it is highlighted.

To name a block:

1. With the left mouse button, click in the name area below the block.



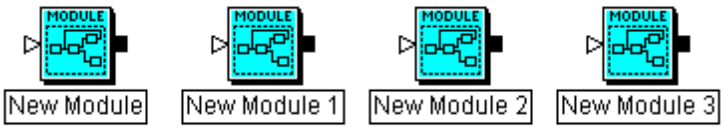
2. Type a descriptive block name.

Note: You can type a name and up to four additional lines describing the block. There is no word wrap, so press ENTER after each line.

3. When you are finished, click any blank area in the diagram. The block is released and the new name appears.

Naming a Module

When you create a module, GML Ultra assigns it the name *New Module x*.

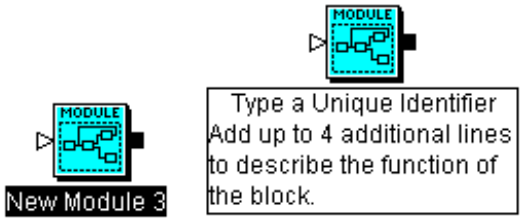


Note: The first module, New Module, is counted as zero.

To make your diagram easier to identify, give the module a more descriptive name:

1. Select the box below the icon that contains the name. The box is highlighted.

2. Type a descriptive name reflecting the combined functions represented by the module.



ATTENTION: Because GML Ultra uses the first line to identify modules, it must contain a unique identifier. For modules the first line can be up to 32 characters long. For blocks, the first line can be up to 64 characters long.

3. To document the functionality of the module within the diagram, type up to four lines of additional information in this box.

Note: Press ENTER after each line.

4. Click on an area outside the module to deselect the module.

Printing Files

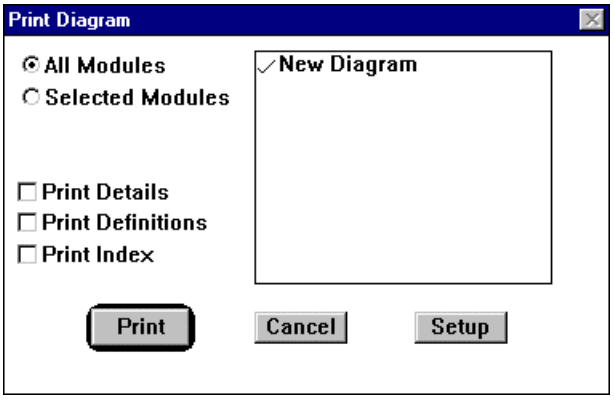
Use the following procedures for printing:

- Setting up the printer
- Printing a diagram
- Printing a picture of all diagram modules
- Printing a picture of select diagram modules
- Printing a script

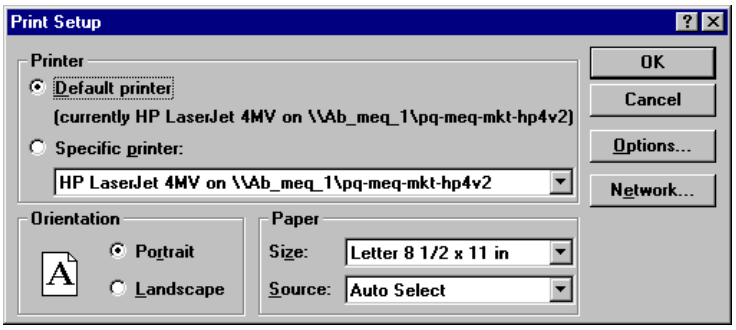
Setting Up Your Printer

To set up your printer:

1. From the File menu, select **Print**. A Print Diagram dialog box like the following appears.



2. From the Print Diagram dialog box, select **Setup**. A dialog box similar to the one below appears. Complete the fields based on your printer, your version of Windows, and other criteria.



3. Select **OK**. The Print Diagram dialog box appears.

4. Select **Print**. The diagram and/or parts of the diagram print.

Printing a Diagram

For a printed record of all or parts of your diagram, you can print:

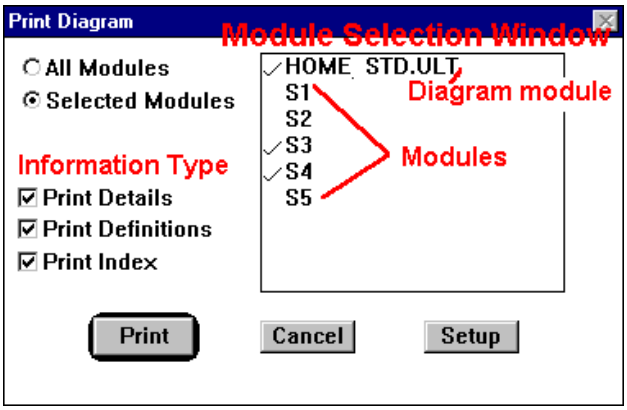
- Only the diagram
- The diagram and all the modules
- The diagram and some of the modules
- Only the modules
- Only some of the modules

For each selection, you can choose to print:

- Details
- Descriptions
- An index

To print your diagram:

1. From the menu bar, select **File.** The File menu appears.
2. Select **Print**. The Print Diagram dialog box appears.



Printing a Picture of All Diagram Modules

To print a picture of all diagram modules

1. Select **All Modules**. All modules, including the top level module (the diagram itself) are printed.

   Modules are printed exactly as they appear on the computer screen and each module is printed on a separate page.

2. Select the type of information you want to print about each module. For a description of information types, refer to the table in the next procedure.

   Note: You can print only four pages for each diagram.

3. Select the **Print** button.

Printing a Picture of Select Diagram Modules

To print a picture of select modules:

1. Select **Selected Modules**. All the modules in the diagram are listed in the window (including the diagram itself).
2. From the module list, select the modules you want to print. A check (✓) appears in front of the selected modules.

   Modules are printed exactly as they appear on the computer screen and each module is printed on a separate page.

3. Select one or more information types you want to print about each module. Use the information in the following table.

Field	Description
Print Details	To print a page describing the parameter values for each block. Note: Each block is assigned a numeric identifier and is printed on a separate page.
Print Definitions	To print a page describing the module definitions for each selected module.
Print Index	To print an index containing a page reference for each module that has been printed.

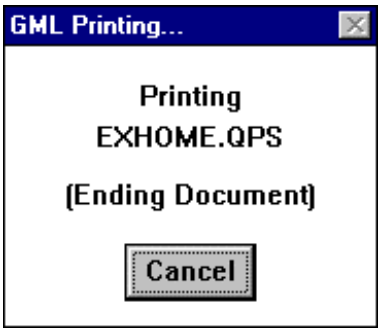
4. Select **Print**.

Printing a Script

For a hard copy of your program script to edit or keep on file, print a script as follows:

1. Convert your diagram to script.

2. From the File menu, select **Print**. The print function begins. Status dialog boxes similar to the following appear. The script prints.



Closing a File

You can close a file at any time. You need not be finished with the diagram. The system saves whatever you have completed. Use the following procedures:

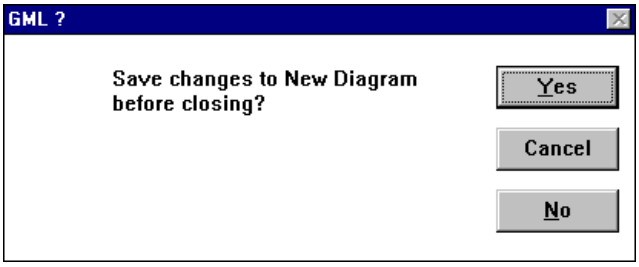
- Closing a file before saving it.
- Closing a file that is saved and named

Closing a File Before Saving It

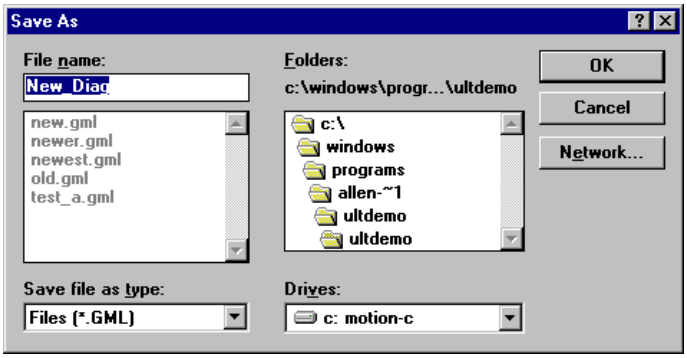
A diagram is considered new until it is saved. Usually, you name a diagram when you first save it. If you close a file without saving it, your work is lost.

To close a diagram:

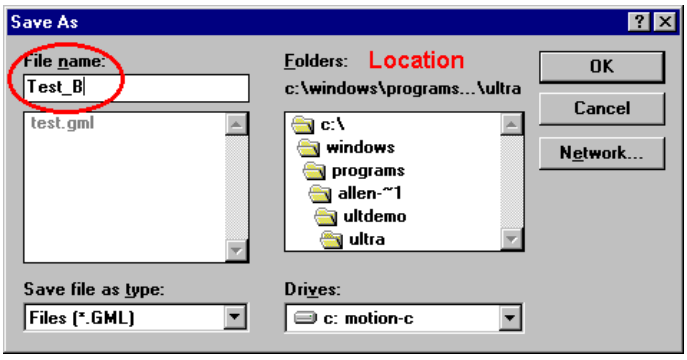
1. From the menu bar, select **File.**
2. Select **Close**. The following dialog box appears.



3. Select **Yes**. The Save As dialog box appears. The name is highlighted in the *File name* field.

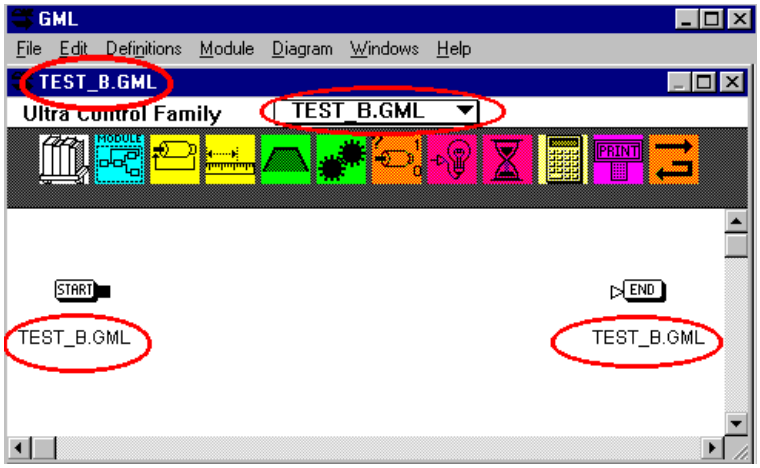


4. In the *File name* field, type a new file name.



5. Confirm the path where the file is saved.

6. Select **OK.** The file is named, saved, and closed. The GML Ultra window appears with the new name in four places: the diagram Title Bar, the module list, and beneath the START and END blocks.



If other files are open in GML Ultra, they appear when the current file closes.

Closing a File Already Saved and Named

Normally, you have named your diagram, and saved it periodically during the diagram development process.

To close a file.

1. From the menu bar, select **File.** The File menu appears.
2. Select **Close**. The file closes.

If other GML Ultra files are	This happens
Open	They appear when the current file is closed.
Not open	The window is blank.

Exiting GML Ultra

1. From the menu bar, select **File**. The File Menu appears.
2. Select **Exit.** GML Ultra checks all open documents for changes and asks if each changed document should be saved before quitting.
3. The GML Ultra application closes.

Note: To close only a file within the application, select Close, not Exit, from the File Menu.

Working With Blocks

To create a program or diagram in GML Ultra, you place blocks representing program functions between the START and END blocks. These two blocks are always present in the diagram and cannot be deleted. You generally add blocks left to right in the Diagram Editor window. You then connect the blocks to show the flow of program functions. You can move blocks around and change their connections, which changes program flow. You can simplify complex diagrams by grouping (encapsulating) several blocks together, then giving the module a name that describes the group's function.

To make working with blocks easier, we recommend that you:

- Try to have block connections flow from left to right or up to down.
- Limit the number of blocks in the work space to ten or less.
- Try to keep your diagram on one screen so that you don't need to scroll to see parts of it.

This chapter covers:

- The physical aspect of blocks, such as selecting, moving, and naming.
- Using blocks to define functions for your diagram and using the Expression Builder.

Selecting and Positioning Blocks on the Diagram Editor Window

There are two ways to select and position blocks in the Diagram Editor window:

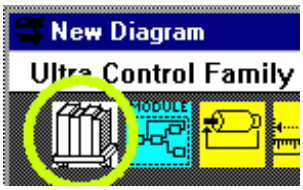
- Selecting a block from the Diagram Library.
- Picking and placing a block from the block palette.

Using the Diagram Library

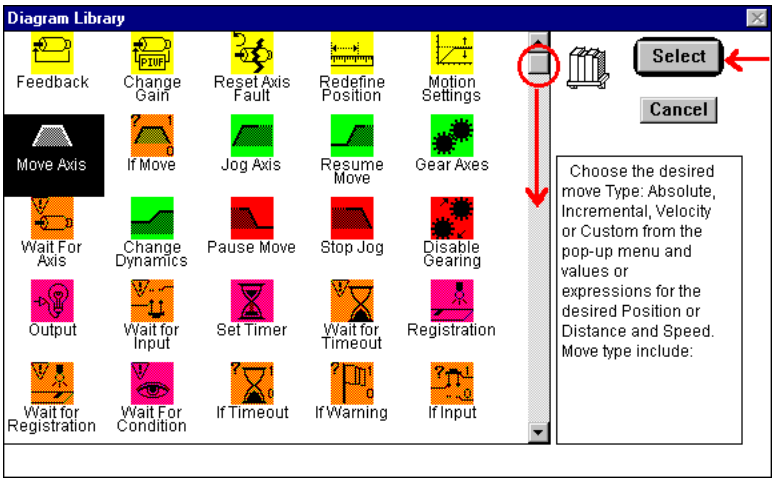
The Diagram Library contains all available blocks for your motion controller.

To access blocks from the Diagram Library:

1. At the far left of the block palette, select the Library Browser block.



The Diagram Library, similar to the following, appears:



Note: More blocks are available by sliding the scroll bar down.

2. Select a block. The block is highlighted and a description of the block's function appears in the text area on the right.

   Note: In the example above, the Move Axis block is selected.

3. Select the **Select** button. The Diagram Editor window appears and the cursor changes to a finger.

4. Position the cursor and click the mouse button. The selected block appears in the Diagram Editor window.

Using the Block Palette

Recently used blocks appear in the block palette — the string of colored function blocks you see across the top of the screen.

This palette is useful when you need to reuse a block repeatedly. The most recently selected block is positioned in the next block that has not been used, starting after the New Module block.

To select a block from the toolbar and position it using pick and place:

1. Hold the pointer over the desired block.

2. Select the block.

Note: Do not hold down the mouse button.

3. Place the cursor into position in the Diagram Editor window.

4. Click and release the mouse button. The block appears in the diagram.

Manipulating Blocks

Most blocks and modules, including the START and END blocks, have input and output nodes. The diagram must start at the START block. It also must have connections drawn from an output node of one block to the input node of another. The diagram usually ends at the END block, but does not need to.

Using the Cursor

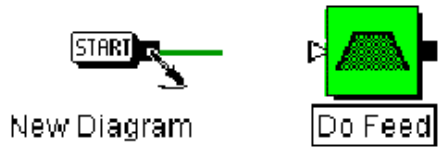
When you build a diagram, the cursor acts as three separate tools:

- A pointer allows you to select and move blocks.
- A soldering iron allows you to connect blocks.
- A wire cutter allows you to remove connections.

Connecting Blocks

To connect blocks and modules:

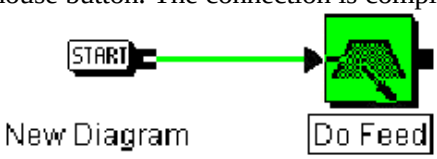
1. Move the cursor to the desired output node. The cursor changes from a pointer to a soldering iron.



2. Press and hold the mouse button.

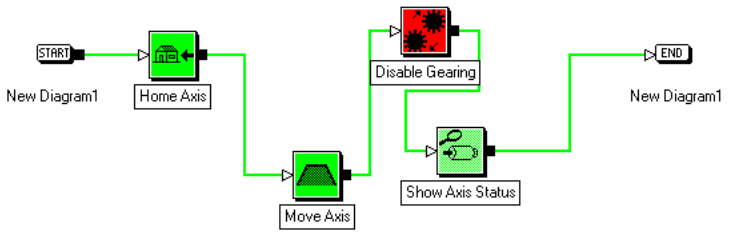
3. Drag the soldering iron to the desired input node or anywhere inside the icon. As you drag, a green line representing the connection is drawn.

4. When the soldering iron is in the appropriate location, release the mouse button. The connection is complete.



Moving Blocks

You can drag blocks, including the START and END blocks, anywhere in the diagram. Their connections remain no matter where you move them.



Showing the Connection Lines

Use the Diagram Drawing option (select Preferences from the File menu) to show connection lines:

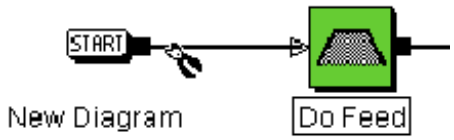
- On top of blocks to help keep track of connections in the foreground..
- Underneath blocks to show blocks in the foreground.

Disconnecting Blocks

To cut a connection:

1. Move the pointer to a connecting line. The cursor changes to a wire cutter.

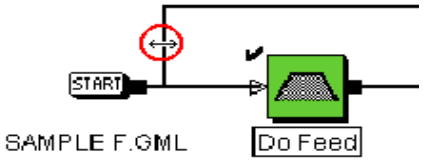
   Note: You can only cut horizontal connecting lines.



2. Click your left mouse button. The connection is cut.

In some cases, when you have a connection that loops back to an earlier block in the diagram, you could want to move a vertical connection to avoid other blocks. To move lines:

1. Place the pointer on the vertical connecting line. A double arrow replaces the pointer.



2. Press the left mouse button, drag the connection to the desired position and release.

Making Changes to Your Diagram

All Edit menu options pertain to the Diagram Editor window. You use this window to make standard editing changes to your diagram, such as cut, copy, paste. Descriptions of these options are in the *General Procedures* chapter.

You also use this menu to select editing features that are specific to GML Ultra, such as moving blocks and accessing block parameters. These options are described in this section.

Selecting All

Use the Select All option to select all the elements in the Diagram Editor window. This option is often used with other editing changes to your diagram, such as aligning and spacing.

To select the entire diagram, select **Select All** from the Edit menu. Every block, module, connection, property, and parameter is highlighted.

Duplicating a Block

Use Duplicate on Edit menu to make a copy of an existing block in a diagram.

To duplicate a block:

1. Select the block you want to duplicate.
2. From the File menu, select **Duplicate.**
3. A copy of the selected block appears next to the original block.

Adding a Block

You add a new block to a diagram by selecting it from the Diagram Library. You can display the Diagram Library in two ways:

1. Click on the Library Browser — the first block on the left.
2. From the File menu, select **Add**.
3. Move the cursor when you want to add a block.

4. Click the left mouse button. The block is added.

Replacing a Block

You can quickly replace one block on the diagram with another block. To replace a block:

1. On the diagram, select the block you wish to replace.
2. From the File menu, select **Change Block**. The Diagram Library appears.
3. Select the replacement block; in the diagram, it replaces the original block.

Inserting a Block

Use the Insert Between option to insert a new block between two existing blocks.

To insert a block:

1. On the diagram, select the two existing blocks.
2. From the File menu, select **Insert Between**. The Diagram Library appears.

Note: The Insert Between option is disabled (gray) until you select the two blocks.

3. From the Diagram Library, select the block you wish to insert. The new block appears between the two selected blocks.

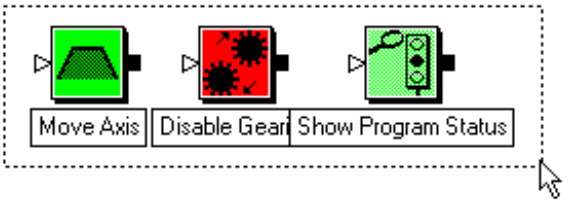
Grouping Blocks

You can treat a group of blocks as a unit by placing a temporary box around them. They can then be manipulated as a group.

To group blocks:

1. Place the blocks so you can draw a box around them without including other blocks.
2. Place your cursor at the upper left of the group.

3. Press and hold the left mouse button and drag it to the lower right of the group. A box is drawn around the group.



4. Release the mouse button. The blocks are highlighted.

You can now work with the block group as a unit. For example, you can move (as shown above), cut, or encapsulate the blocks as a group.

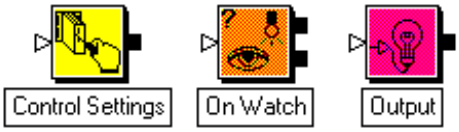
For more information about encapsulating blocks, refer to the Encapsulating Blocks section.

Swapping Blocks

Use the Swap Blocks option to swap the location and connections of two selected blocks. The blocks need not be connected.

In the example that follows, the Control Settings block is swapped with the Output block.

This is the current position of the blocks.



To swap two selected block positions in a diagram:

1. Select one of the blocks to be swapped by selecting it with your left mouse button. It is highlighted.

2. Select the second block to be swapped by holding down the Shift key and clicking the second block with your left mouse button. You should now have two selected blocks.

3. From the Edit menu select **Swap Blocks**, The blocks switch positions.

4. Click your left mouse button in another area of the pane to clear the selection.

Aligning Blocks

Use the Align Blocks option to align blocks on a vertical or a horizontal grid.

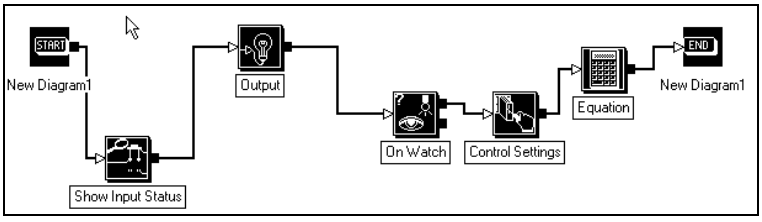
The Align Blocks option affects either:

- The entire Diagram Editor window pane (you do not need to select anything). This is the default.
- Only the portion of blocks that you select.

To align the entire diagram:

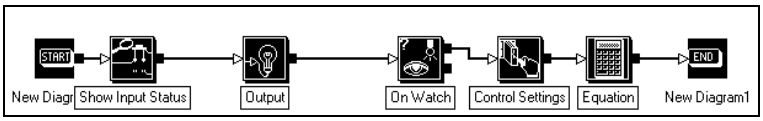
1. From the menu bar, select **Edit**. The Edit menu appears.

2. Select **Select All**. All the blocks and modules in the diagram are selected.



3. From the menu bar, select **Edit**. The Edit menu appears.

4. Select **Align Blocks**. The selected blocks align.

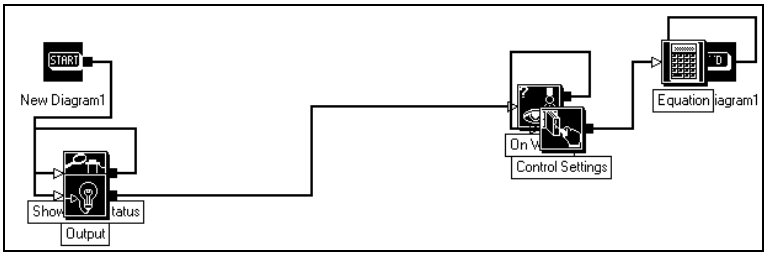


Spacing Blocks

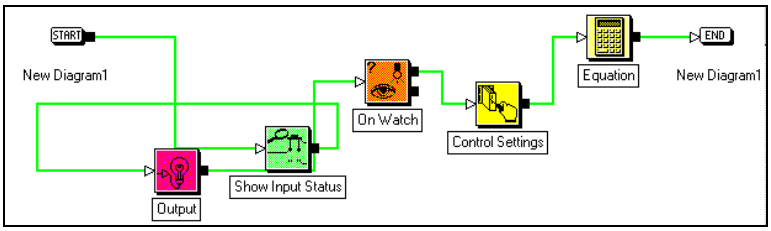
Blocks that are evenly spaced are easier to read. When a diagram becomes cluttered and blocks overlap, it is time to space the blocks and modules so you can keep track of what you are doing.

To space blocks for readability:

1. From the menu bar, select **Edit**. The Edit menu appears.
2. Select **Select All.** The entire diagram is selected.



3. From the menu bar, select **Edit**. The Edit menu appears.
4. Select **Space Blocks**.
5. Click your left mouse button in another area. The diagram selection is cleared and the blocks are uniformly spaced.



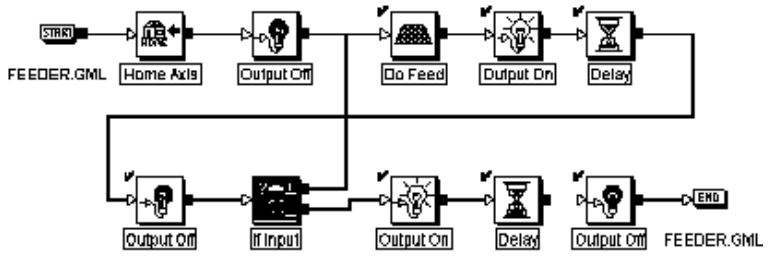
Note: You can Align Blocks at this time for optimum readability.

Snapping to the Grid

You define how you place blocks in the diagram by using the Snap To Grid option in the Preferences dialog box. GML Ultra divides the diagram into invisible horizontal and vertical lines. The horizontal grid is set for $\frac{1}{4}$ the size of a block and the vertical spacing is set for $\frac{1}{2}$ the size of a block.

You can specify that blocks be aligned to the nearest horizontal and/or vertical grid line when placed in the diagram.

Below is an example of a diagram aligned with horizontal grid snap. Note that functions and operations can flow to an additional row (or rows) of blocks if required.



Accessing Block Information

You can access information specific to a block in two ways.

To access block information from a menu:

1. Select a block.
2. From the menu bar, select **Edit**. The Edit menu appears.
3. Select **Block Information**. The dialog box for that block appears.
4. Examine or change the parameters that have been set for that function block.

To access block information from the diagram:

1. Place your cursor over a block.
2. Double-click your left mouse button. The dialog box pertaining to that block appears.
3. Examine or change the parameters for that function block.

Setting Block Function Parameters

Most block parameters are defined during the setup procedure. Once a block has been added to a diagram, you can enter or change parameters specific to that block function and that diagram through the block's dialog box. For example, you often access a block's dialog box during the tuning and testing procedures.

To define function parameters in a block:

1. Double-click on a block. A dialog box appears.

2. Select or define the parameters that apply to your diagram.

3. Select **OK**. The defined parameters are saved. The dialog box closes and the Dialog Editor window appears.

Using Mathematical Expressions as Values

Expressions are formulas of variables, constants and functions that can be used in place of a numeric parameter.

Although you can type a number, expression, or variable directly into a field, you can also use the Expression Builder to build an expression. The Expression Builder includes a calculator keypad, a list of special operators, and easy access to lists of all variables, constants, and functions that can be used in expressions.

This section describes the process for entering values in fields. For a detailed description of the Expression Builder, refer to the Expression Builder chapter in the *GML Ultra Reference* manual (Publication 1398-5.12).

Understanding Expressions

Sometimes, you need a numerical value or expression to define a parameter for a block. A value can be a single constant, a variable, an array reference, a function, or any combination in the form of a mathematical calculation.

You can either enter an expression string directly in the field or build an expression by selecting items from the lists in the lower left corner of the Expression Builder. In GML Ultra, all operators are of equal precedence and compute in left to right order. Use parenthesis to change precedence. You can nest operators up to 25 levels deep.

Arrays or indirect variable references can be incorporated in any expression. You can access an array through the Indirect Variable system function. The pointer or index to an array or indirect variable reference can be any valid expression.

Using the Expression Builder

The Expression Builder is a specialized calculator for constructing mathematical expressions (function value parameters) in GML Ultra blocks. You create your value or calculation in the Expression Builder dialog box, and then automatically fill in a data entry field with that value by selecting OK.

The Expression Builder consists of:

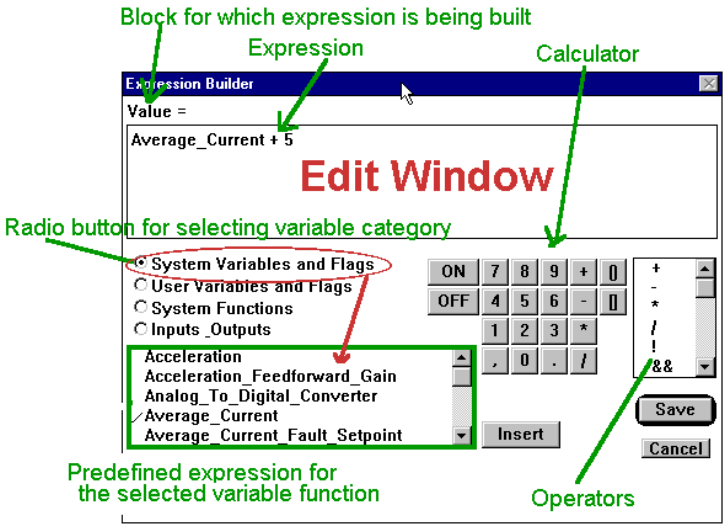
- Constants
- System functions
- A calculator key pad
- A list of expression operators
- Internal (system) variables
- User-defined variables
- An edit box for building the expression

To use the Expression Builder:

1. In any field requiring a numeric value, highlight the value or click in the field.

2. From the menu bar, select **Edit**. The Edit menu appears.

3. Select **Build Expression**. The Expression Builder dialog box appears. If the Expression Builder dialog box does not appear, you cannot build an expression in the field you selected.



Building an Expression

Use the procedures below to select appropriate items from the lists to build an expression.

Adding System Variables and Flags

1. Select **System Variables and Flags**. The radio button is highlighted and the predefined elements for that category appear in the list.
2. Select the expression. Scroll to see all the variables.
3. Select **Insert.** The variable is inserted into the expression.

Adding User Variables and Flags

1. Select **User Variables and Flags**. The radio button is highlighted and the predefined elements for that category appear in the list.
2. Select the expression. Scroll to see all the variables.
3. Select **Insert**. The variable is inserted into the expression.

Note: User Variables and Flags must be predefined. Refer to the *GML Ultra Getting Started* manual (Publication 1398-5.10).

Adding System Functions

1. Select **System Functions**. The radio button is highlighted and the predefined elements for that category appear in the list.
2. Select the expression. Scroll to see all the variables.
3. Select **Insert**. The variable is inserted into the expression.

Adding Inputs/Outputs

1. Select **Inputs/Outputs**. The radio button is highlighted and the predefined elements for that category appear in the list.
2. Select the expression. Scroll to see all the variables.
3. Select **Insert**. The variable is inserted into the expression.

Note: Inputs/Outputs must be predefined. Refer to the *GML Ultra Getting Started* manual.

4. Select **Save**. The expression is copied to the block's field.

Adding Operators

Select an expression operator from the list. The operator is inserted automatically into the expression.

Adding Numerals

Select a number from the calculator. The number is inserted automatically into the expression.

Using Variables, Constants, and I/O User Variables

You can either build expressions from scratch or you can use predefined values for the expressions you build in the Expression Builder. There are two types of predefined values:

- User-defined variables
- System-defined variables

User-Defined Variables

A user-defined variable (such as a constant, a discrete input or output), must be defined before it appears as a selection in the lower window. You usually define these as needed.

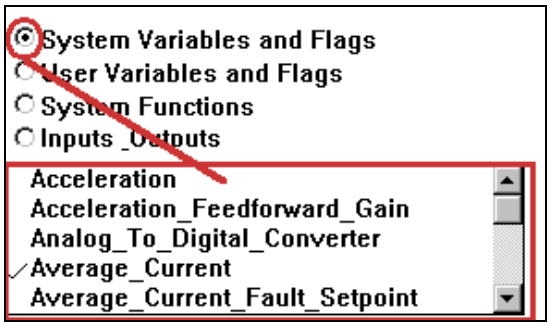
Refer to the *GML Ultra Getting Started* manual for more information on defining variables.

System-Defined Variables

The Expression Builder offers numerous system variables and system functions based on your system preferences and controller configuration. You can select the type of variable you want by selecting the appropriate radio button. The related system variables then appear in the list window below.

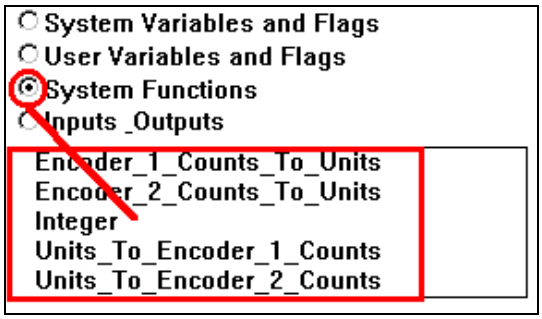
Example 1: System Variables

When you select System Variables, the predefined system variables appear by name in the list window. Use the scroll bar to view all the variables.



Example 2: System Functions

The example below shows the System Functions that are defined by GML Ultra.

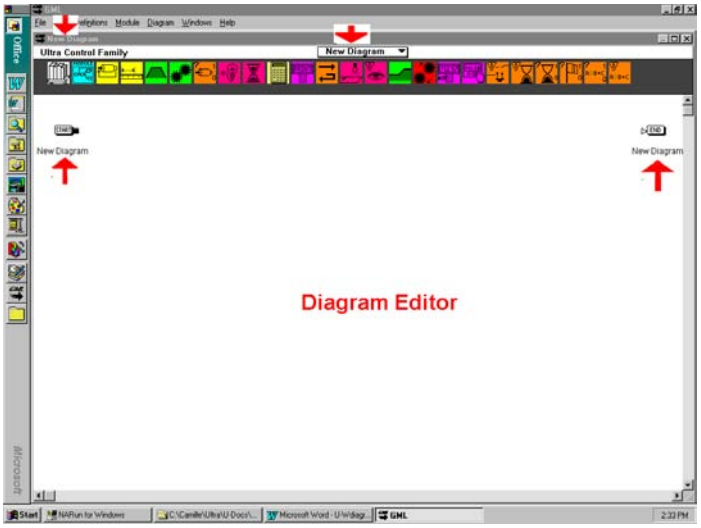


Working with Diagrams

Diagrams are the graphical equivalent of scripted programs. You can create original programs or copy and paste parts of existing diagrams into your new diagram.

Creating New Diagrams

Each time you open GML Ultra, the GML Ultra window appears, ready for you to begin developing a new diagram.



Creating a New Diagram

To create a new diagram, select **New Diagram** from the File menu. A new window appears with a temporary name. Rename it as you wish.

Note: When GML Ultra is open, do not start a new diagram by selecting the icon from the program window. This would result in two GML Ultra applications running at the same time.

Editing Your Diagram as you Build It

All the options in the Edit menu pertain to the Diagram Editor window.

For information on standard editing features - such as cut, copy, and paste - refer to the *General Procedures* chapter.

For information on block editing features that are specific to GML Ultra, refer to the *Working with Blocks* chapter.

Working with Finished Diagrams

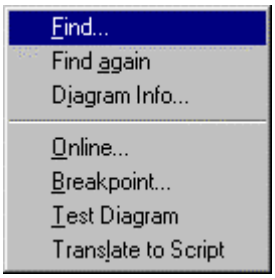
Once you have designed a diagram, the Diagram menu provides access to:

- The Find feature.
- Diagram statistics.
- The Online Manager window.
- Diagram testing.
- Translation from graphical design to script.

Accessing the Diagram Menu

Depending on previous options and current selections, some options can be inactive and appear gray. Only the active options appear black.

From the menu bar select **Diagram**. The Diagram menu appears.

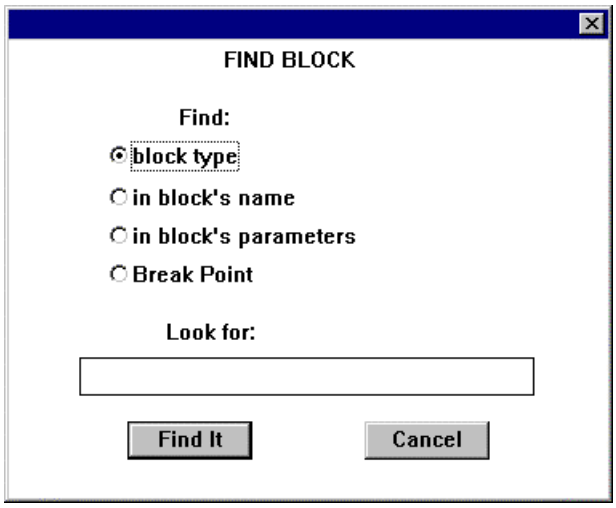


Finding a Specific Block or Parameter

The Find option on the Diagram menu allows you to search for text in the current diagram or script file.

To search for text:

1. From the Diagram menu, select **Find**. A dialog box similar to the following appears.



2. In the *Find* area, select the type of search.

3. In the *Look for*: field, enter the text you are looking for. You can enter up to 255 characters in this field.

4. Select the **Find It** button. GML Ultra searches for the text and displays it if it is located.

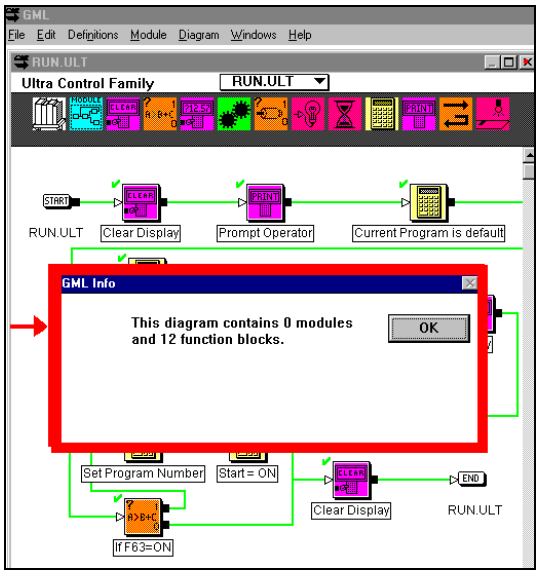
Finding the Same Block or Parameter Again

Use the Find Again option on the Edit menu to repeat a search you just completed using the Find option.

Diagram Information

Use the Diagram Info option to display the number of blocks and modules in the current diagram.

To display this information about the current diagram from the Definitions menu, select **Diagram Info**. A GML Info dialog box similar to the following appears, displaying the information.



Testing Diagrams

This option allows you to test your diagram for valid translation into a script for the selected motion controller, including tests for valid block parameters and block connections. If GML Ultra finds an error, a window appears, describing the error and identifying the block where the error occurred.

Note: Testing diagrams is not the same as testing programs.

To test your diagram for valid translation to script:

1. From the Diagram menu, select **Test Diagram**. If one or more errors is present, the first of the warning messages appears, similar to the following:

2. Select **OK**. GML Ultra automatically highlights the offending block. If the error is a parameter within a block, GML Ultra also automatically opens the block and presents the dialog box with the parameter fields.

3. Fix the error.

4. Retest by repeating steps 1 through 4. When no more error messages appear, you have completed a successful test.

IMPORTANT: A successful test indicates that the diagram translates to a valid program, not that the motion controller operates your machine flawlessly.

Accessing the Online Manager

Select the Online option to access the Online Manager.

For more information about the Online Manager, refer to the *Going Online* chapter.

Inserting a Breakpoint

Use the Breakpoint option on the Diagram menu to set or clear breakpoints. Breakpoints are set and cleared from the Diagram menu but you must be online to set a breakpoint.

Breakpoints can only be set after the diagram is downloaded. The diagram must be downloaded every time a change is made to the diagram.

1. From the menu bar, select **Diagram**. The Diagram menu appears.
2. Select **Online**. The Online Manager window appears.
3. To display the Breakpoint Control dialog box, select the block where you want to set a breakpoint.
4. Download the current diagram.
5. Select **Breakpoint** from the Diagram menu.
6. Set or clear the breakpoints by selecting the appropriate radio button.

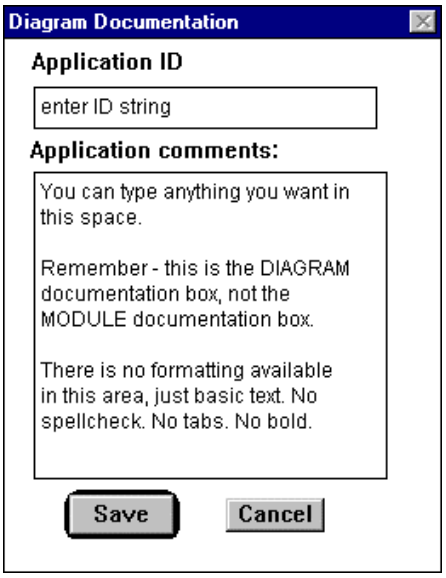
A block with a breakpoint set has a black validation check mark.

Documenting Your Diagram

Documentation provides a place to record information about a specific diagram.

To document information about your diagram:

1. Select the desired diagram from the **Windows** menu.
2. From the Definitions menu, select **Documentation**. The Diagram Documentation dialog box appears.



3. Type your information.

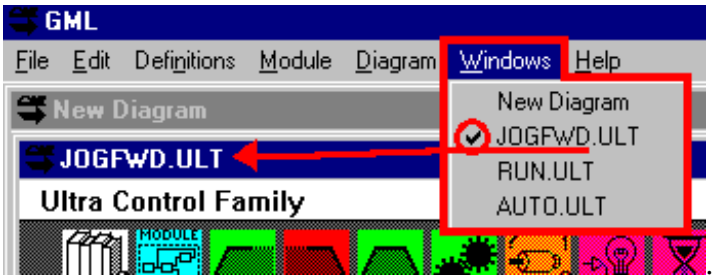
4. Select **Save**. The information is saved and the dialog box disappears.

Accessing Previous Documentation

When working with a diagram, or a module from another diagram, you can look at notes or comments the author made. You can also add your comments.

To view or document information about this diagram:

1. From the menu bar, select **Windows**. The Window menu appears, listing the open diagrams. The active diagram is checked.



2. Select the diagram you want to access. That diagram is active.

3. From the Definitions menu, select **Documentation**. The Diagram Documentation dialog box appears.

4. Change, add, delete, or copy the text in the dialog box.

5. Select **Save**. The information is saved and the Dialog Editor window appears.

Using Preexisting Modules to Create Diagrams

Sometimes a diagram, or part of a diagram, is very similar to one already created. You can recycle part of a diagram (usually a module) for use in your new diagram.

IMPORTANT: In GML Ultra, you can use files with the **.**ULT extension. However, you cannot use files with the **.**GML extension.

Modules are especially convenient tools to recycle commonly used configurations. Modules can be very complex and their creation can be time-consuming.

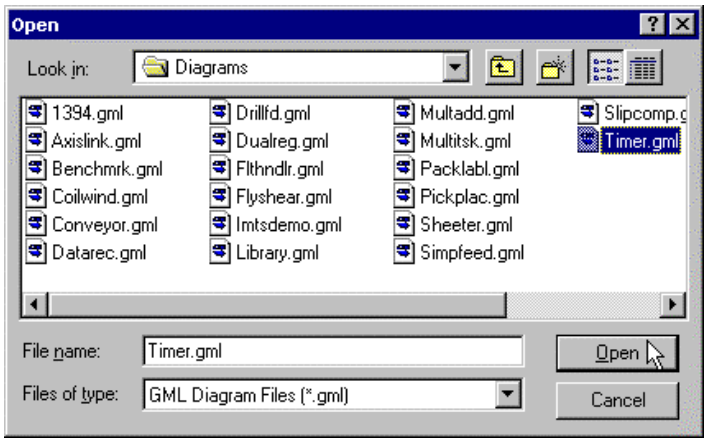
The process for using preexisting modules, or parts of diagrams, involves copying and pasting.

Using a Preexisting Module

To find, copy, and paste a preexisting module:

1. From the menu bar, select **File**. The File menu appears.

2. Select **Open Diagram**. An Open dialog box appears, similar to the following:



3. Highlight the diagram you want to use. Select **Open**. The diagram opens and becomes active.

   Note: A diagram can be an entire program or it can be just a module.

4. Select the module you want to copy. The module is highlighted.

   Note: It is not necessary to open a module to copy and use it.

5. From the menu bar, select **Edit**. The Edit menu appears.

6. Select **Copy**. The module and all its blocks, connections, and block parameters are copied.

7. From the menu bar, select **Windows**. A list of open diagrams appears.

8. Select your new diagram by name or number. That new diagram becomes active.

9. In the Diagram Editor window of the new diagram, place your cursor where you want to insert the copied module.

10. From the menu bar, select **Edit**. The Edit menu appears.

11. Select **Paste**. The module is pasted into the new diagram.

    Note: You can change settings now, later, or keep the module as is.

12. Continue adding blocks and modules (new or existing) to complete your diagram.

Using a Preexisting Diagram

You can copy and paste an entire preexisting diagram (usually a module) into a new diagram. You can then use it as is, or modify parts of it.

To Copy an Entire Diagram

1. From the menu bar, select **Edit**. The Edit menu appears.

2. Select **Select All**. The entire diagram is highlighted.

3. From the menu bar, select **Edit**. The Edit menu appears.

4. Select **Copy**. The diagram is copied to the clipboard.

5. From the menu bar, select **Windows**. A list of open diagrams appears.

6. Select your new diagram by name or number. That new diagram becomes active.

7. In the Diagram Editor window of the new diagram, place the cursor where you want to insert the copied module.

8. From the menu bar, select **Edit**. The Edit menu appears.

9. Select **Paste.** The module is pasted into the new diagram.

   Note: You can change settings now or later, or keep the module as is.

10. Continue adding blocks and modules (new or existing) to complete your diagram.

To Copy Part of a Diagram

1. From the menu bar, select **File**. The File menu appears.
2. In the Diagram Editor window, place your cursor at the upper left corner of the area you want to copy.
3. Press and hold the left mouse button.
4. Drag the cursor to the lower right corner of the area you want to copy. A dotted rectangle surrounds the area.
5. Release the mouse button. The selected portion of the diagram is highlighted.
6. From the menu bar, select **Edit**. The Edit menu appears.
7. Select **Copy**. The selected portion is copied to the clipboard.
8. From the menu bar, select **Windows**. A list of open diagrams appears.
9. Select your new diagram by name. That new diagram becomes active.
10. Place your cursor in the Diagram Editor window of the new diagram at the place where you want to insert the copied module.
11. From the menu bar, select **Edit**. The Edit menu appears.
12. Select **Paste.** The module is pasted into the new diagram.

    Note: You can change settings now, later, or keep the module as is.

13. Continue adding blocks and modules (new or existing) to complete your diagram.

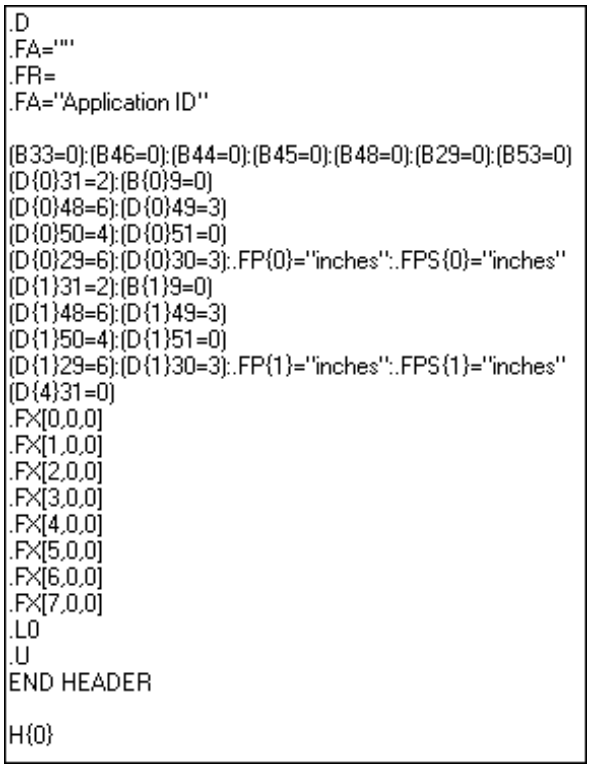
Converting a Diagram to a Script

Before you perform a download, you could view a script to identify problems.

IMPORTANT: Changes made to a script do not convert to changes in the corresponding diagram.

To translate your diagram:

1. Make sure the diagram you want to view is active.
2. From the menu bar, select **Diagram.**
3. Select **Translate to Script.** The script appears in a window similar to the following.



```
.D
.FA=""
.FR=
.FA="Application ID"

(B33=0):(B46=0):(B44=0):(B45=0):(B48=0):(B29=0):(B53=0)
(D{0}31=2):(B{0}9=0)
(D{0}48=6):(D{0}49=3)
(D{0}50=4):(D{0}51=0)
(D{0}29=6):(D{0}30=3):.FP{0}="inches":.FPS{0}="inches"
(D{1}31=2):(B{1}9=0)
(D{1}48=6):(D{1}49=3)
(D{1}50=4):(D{1}51=0)
(D{1}29=6):(D{1}30=3):.FP{1}="inches":.FPS{1}="inches"
(D{4}31=0)
.FX[0,0,0]
.FX[1,0,0]
.FX[2,0,0]
.FX[3,0,0]
.FX[4,0,0]
.FX[5,0,0]
.FX[6,0,0]
.FX[7,0,0]
.L0
.U
END HEADER

H{0}
```

You can easily switch between the script window and the diagram window to make changes in the diagram because they are both open.

Working With Modules

A module is one block made up of several related blocks. These blocks together comprise either a larger function or related functions. Using modules allows you to hide detail and save space. You can duplicate, copy, move, connect, and encapsulate a module just like other blocks in the diagram.

Creating a New Module

There are four ways to create a module:

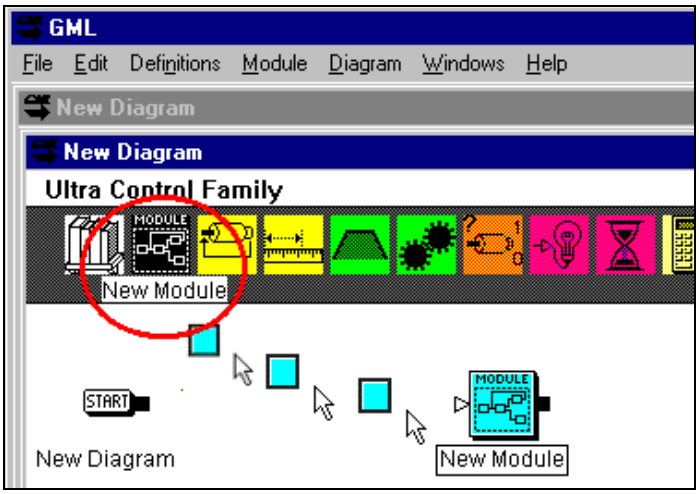
- Select the New Module block from the main Block Palette.
- Add a module from the Block Library.
- Add a module from the Edit menu.
- Encapsulate several blocks.

Using the Block Palette

Use this method when you want to create a diagram by starting with a high level idea. For example, each module might be a major program function, such as startup or setup. A new module is empty when selected from the palette. You can add blocks, block parameters, and connections later.

To create a new module:

1. From the Block Palette, select (click and release) the New Module block. The block attaches itself to the cursor until you place it in the Diagram Editor.



2. Place the New Module block in the Diagram Editor window by clicking again.

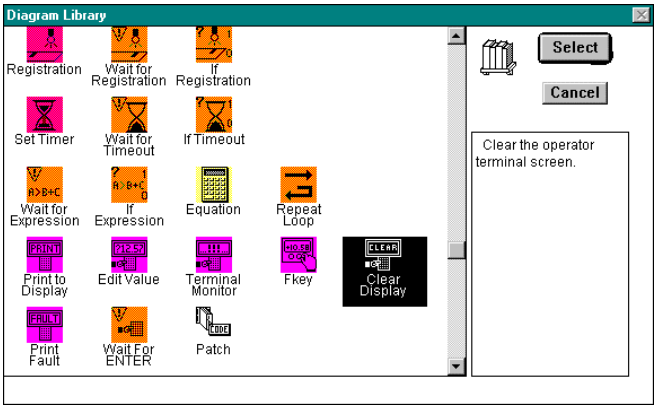
3. Name the block. Add appropriate blocks and connections later.

Using the Diagram Library

The first icon in the block palette is the Library Browser. Select this icon to display all available blocks for the ULTRA Plus controller.

To access the Diagram Library:

1. Select the  icon. A Diagram Library similar to the one below appears.



2. Use the scroll bar to find the New Module block.

3. Select the **New Module** block. It is highlighted.

4. In the Diagram Library window, select **Select**. The Diagram Editor window appears.

5. Point to the location where you want to place the block.

6. Click and release your cursor. The New Module block appears.

Using the Edit Menu

To add a new module block using the Edit menu:

1. From the Edit menu, select **Add**. The Block Library appears.

2. Scroll to find the New Module block.

3. Select the New Module block. It is highlighted.

4. Select **Select**. The Diagram Editor appears with the cursor looking like a pointing hand.

5. Point to the location where you want to place the New Module block.

6. Click and release your cursor. The New Module block appears.

Encapsulating Blocks or Modules

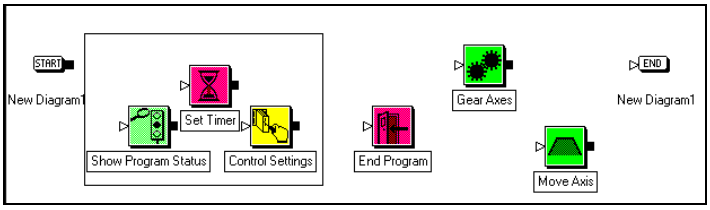
You can simplify the structure of your diagram by combining (or encapsulating) several blocks or modules into a single module. You can easily reverse this process by unencapsulating the module. Refer to the *Unencapsulating a Module* section.

To create one new module by encapsulating several blocks and/or modules:

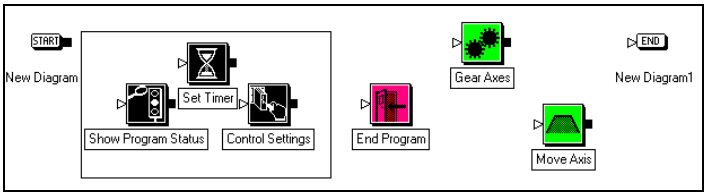
1. Position the blocks that you want to include in the new module so that:
   - They are next to each other, and
   - Unwanted blocks are excluded when you select the desired blocks.

2. Hold down the mouse button.

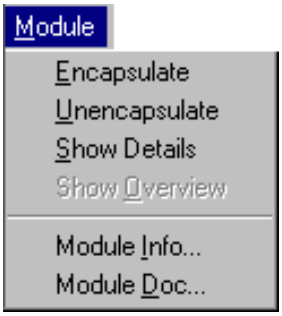
3. Drag a rectangle around the files you want to select.



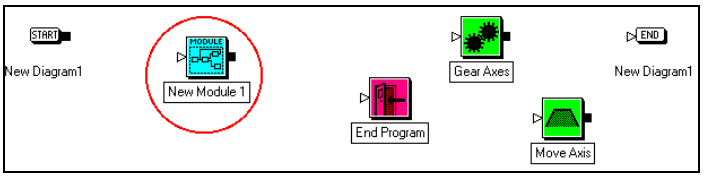
4. Release the mouse button. The targeted blocks are highlighted.



5. From the menu bar, select **Module.** A menu appears.



6. Select **Encapsulate**. The selected blocks are encapsulated into a New Module.



Using an Existing Module

Once you have created a module, you can use it in other diagrams.

The best way to store a collection of prebuilt modules is to save each module as a separate diagram, with a name that makes its function evident. Create a library folder of modules.

ATTENTION: All user variables and any defined I/O is copied and could conflict with ones already in the diagram. Also, none of the axis information is transferred when you paste from one diagram to another.

To use a module from your library:

1. From the menu bar, select **File**. The File menu appears.
2. Select **Open Diagram**. The Open dialog box appears.
3. Locate the directory and module that you want to use.
4. Open the diagram.
5. Select the module. It is highlighted.
6. From the menu bar, select **Edit**. The Edit menu appears.

7. Select **Copy**. The module and its contents is copied to the clipboard.

8. Select your original diagram to make it active.

9. From the Edit menu, select **Paste**. The module appears in the diagram receiving the diagram.

10. Name the new module.

Unencapsulating a Module

If you've encapsulated several blocks or modules into one module, you can reverse the process by unencapsulating the module.

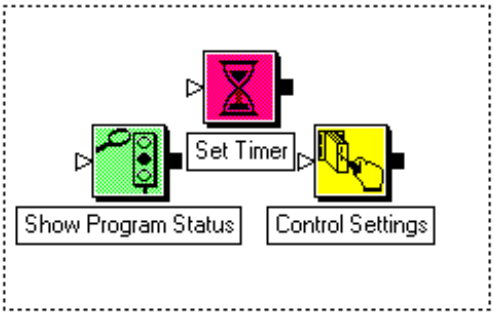
ATTENTION: If you unencapsulate a module that contains several blocks into a diagram that also contains several blocks, sorting the old from the new can be difficult.

To unencapsulate a module:

1. Select the module. The module is highlighted.

2. From the menu bar, select **Module**. The Module menu appears.

3. Select **Unencapsulate**. The module separates into its component parts.



Note: If you want to view the blocks in the module without unencapsulating, refer to the *Viewing Module Details* section.

Viewing Modules

When designing a very complex diagram, you can find it easier to work with if you can simplify it visually. GML Ultra lets you do this without actually changing the structure of the diagram, as you do when you encapsulate a module. You use the Show Overview option to visually simplify a module. Alternately, you can view module detail by using the Show Details option.

Viewing Module Details

You can access individual blocks and their details within the module while preserving the module.

To access individual blocks while preserving the module:

1. Select the module. The module is highlighted.
2. From the menu bar, select **Module**. The Module menu appears.
3. Select **Show Details**.

Diagram with blocks highlighted.

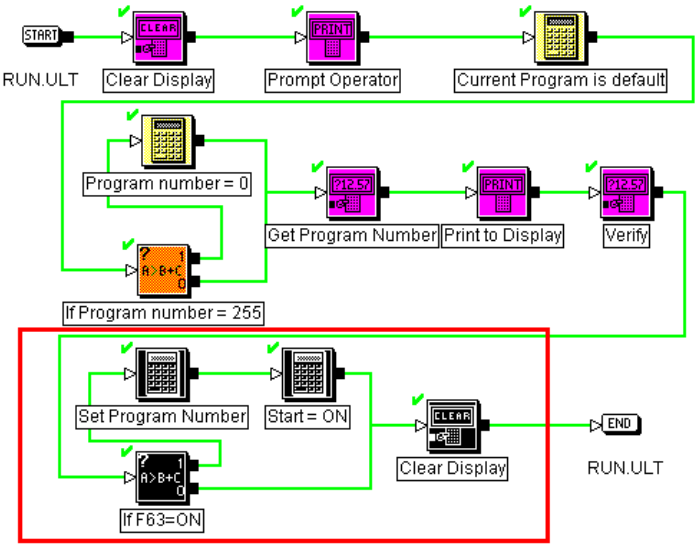
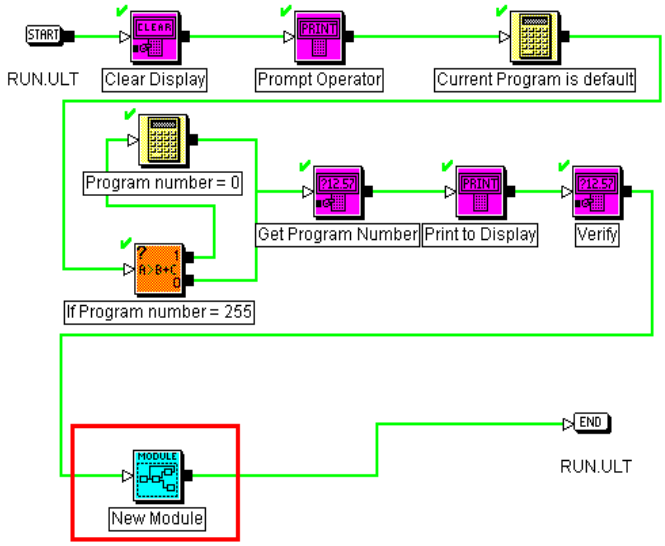
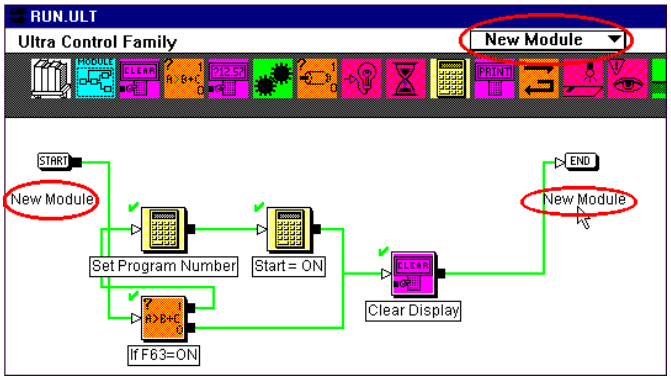


Diagram with highlighted blocks encapsulated into New Module [0].



The resulting view is exactly like the diagram window except that you are looking at a different level. The Module List shows the module level in the Diagram Editor. The module name appears under the START and END blocks. Only the blocks encapsulated in the module appear.

New Module showing details.



Viewing a Module Overview

Use the Show Overview option to show the module level above the current level in the Diagram Editor pane. For example, if there is a module in the diagram view, the module level above the current module level is the diagram.

To view a higher module level, select **Module** from the menu bar. The information in the Diagram Editor shows the next higher level of information.

Diagram1: Level [0]

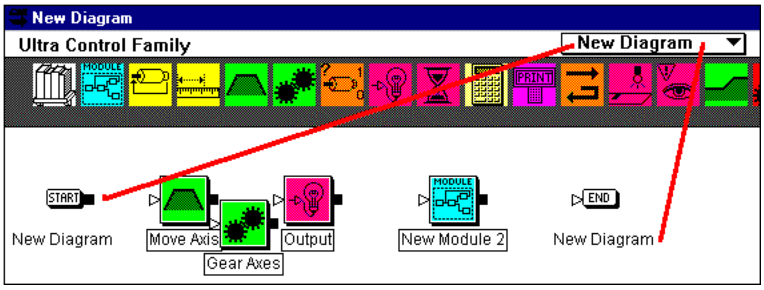


Diagram1: Level 1

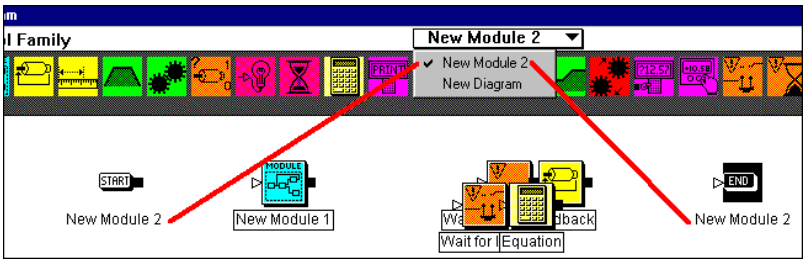


Diagram1: Level 2

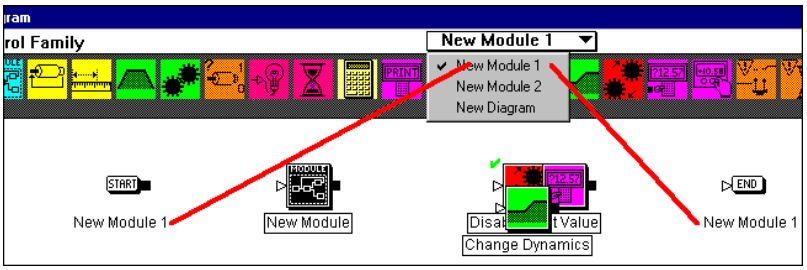
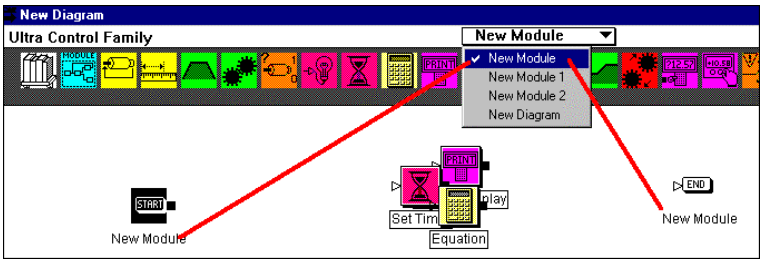


Diagram1: Level 3

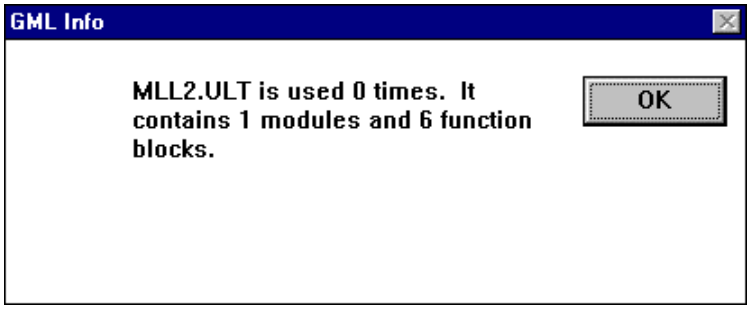


You can easily identify the current module because it is always highlighted in the Module List at the top center of the diagram. You can also check the name of the START and END blocks in the Module Editor window.

Displaying Module Information

To determine the number of blocks and modules at any level:

1. Select the level that you want details about. The level is highlighted.
2. From the menu bar, select **Module**. The Module menu appears.
3. Select **Module Info**. The GML Info dialog box appears, showing the number of blocks and the number of modules in the selected diagram.

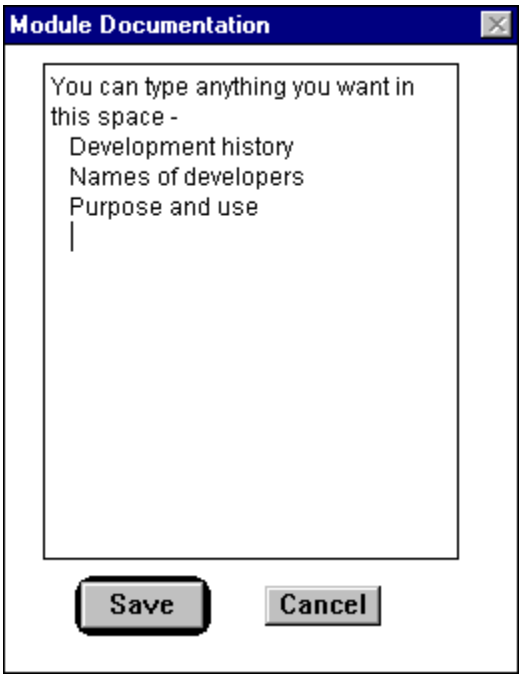


Documenting a Module

In a special dialog box, you can enter information about the module that might be helpful to you and future users.

To document a module:

1. From the Diagram Editor window, select the module you want to document.
2. From the menu bar, select **Module**. The Module menu appears.
3. Select **Show Details** (or double-click the module) to open the module.
4. Select **Documentation**. The Module Documentation dialog box appears.



5. Type new information or change the existing information.

   Note: The text has no formatting or word wrap capabilities.

6. Select **Save**. The information is saved. The Diagram Editor appears.

Working with Scripts

A script is a text-based program. The ULTRA Plus controller requires programs in this format, specifically in a programming language called iCODE. Because a GML Ultra diagram is not in iCODE/script format, it requires translation to it. The Script Editor in GML Ultra is precisely for this purpose—to translate your graphical-based program into a script that the controller can use.

In addition, you can use the Script Editor to create, edit, and print script programs. For example, script files reside on your controller. You can upload these to your PC and save them as scripts.

This chapter describes these two topics:

- Translating a diagram to script
- Creating and editing a script

Translating a Diagram to Script

When you download your diagram to the controller, GML Ultra translates your diagram to iCODE, the native language of the ULTRA Plus controller. You can access the script to view or fine tune the program.

ULTRA Plus controllers can store up to 32 programs. In turn, you can upload these programs to a script file on your PC and store them there.

You can translate a diagram to script before downloading it, but changes to the script cannot be converted back to the graphical diagram. If changes to the script are permanent, that script becomes the program for the download.

This option automatically includes a testing function. The translation process is very similar to the testing process. When a problem is identified, a dialog box informs you of the problem and its location. It automatically goes to that block and opens the block's dialog box so you have ready access to fix the problem.

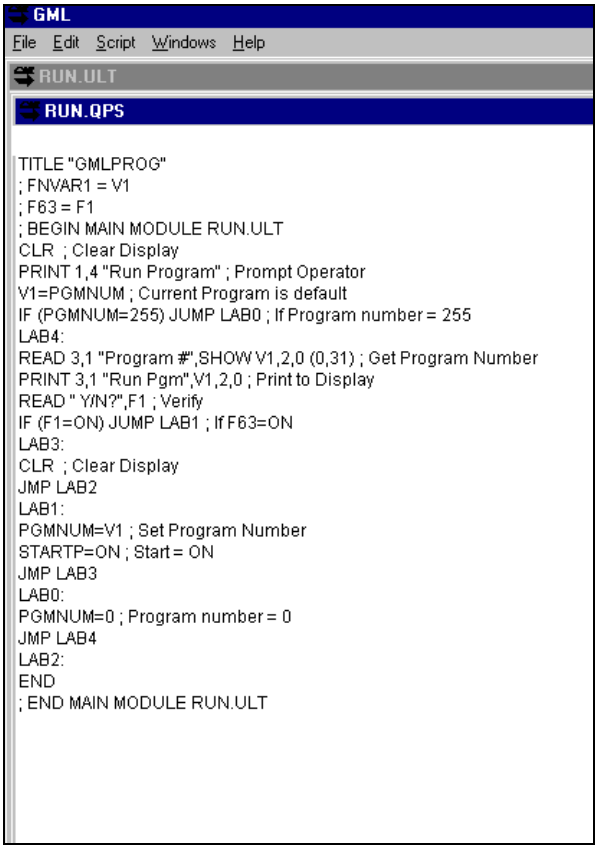
To translate a graphical GML Ultra diagram to a native language script:

1. Make sure the diagram you want to translate is active.
2. From the menu bar, select **Diagram**. The Diagram menu appears.

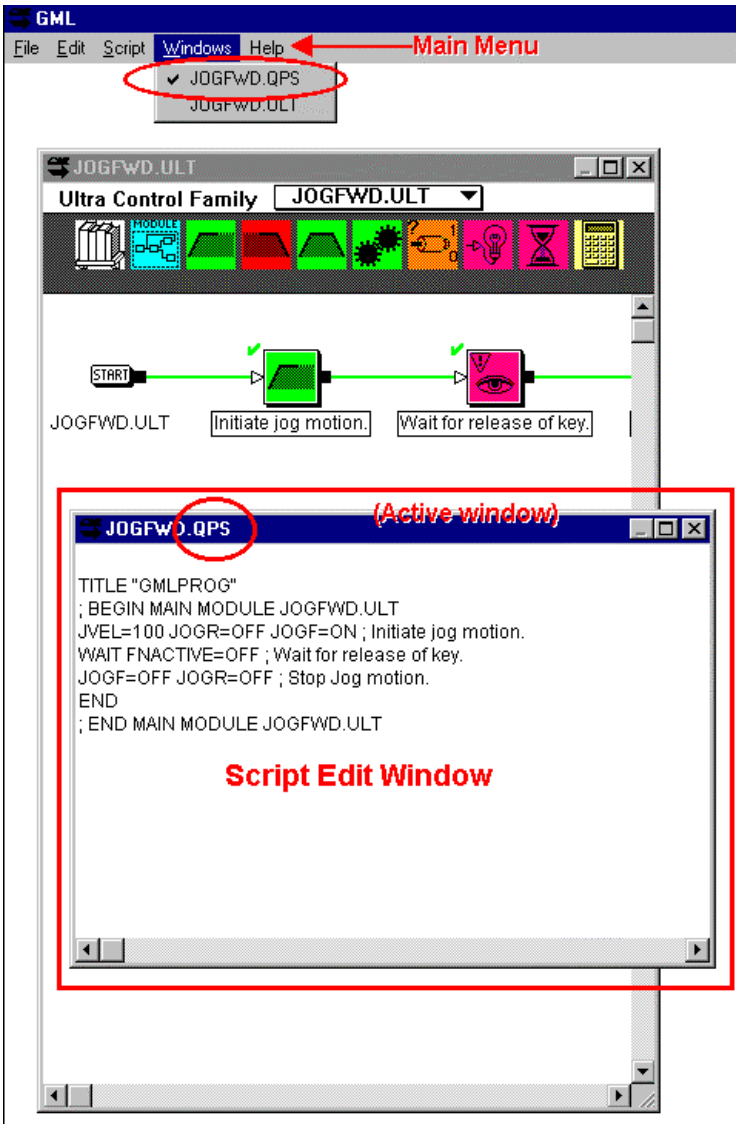
3. Select **Translate to Script**. As GML Ultra translates the diagram into a script, a dialog box appears to inform you of the translation progress.

Successful Translation to Script

When the process completes successfully, the Diagram Editor window changes to a Script Edit window. The script document appears, similar to the example below:

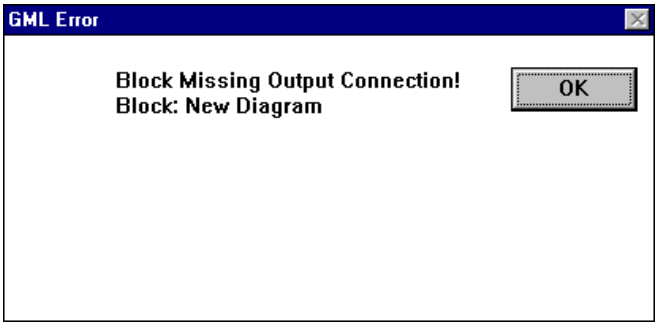


In the above example, both the diagram file (*xxx*.ULT) and the script file (*xxx*.QPS) are open but only the script file is active. The menu bar changes to reflects the script file.



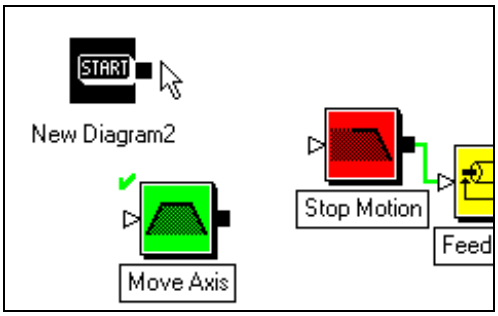
Handling Unsuccessful Translations

The GML Ultra translator checks for errors or potential errors. If the translation is not successful, a dialog box similar to the following appears, identifying the problem and its location.



To correct a problem in the diagram:

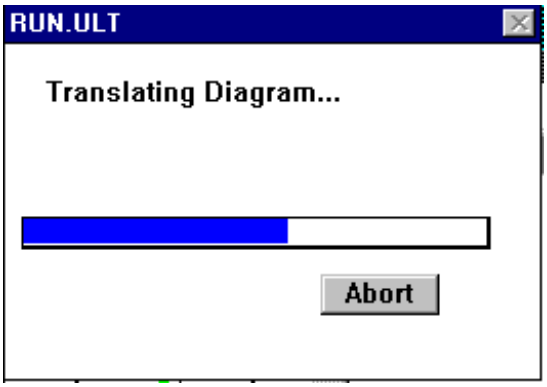
1. From the GML Error dialog box, select **OK**. GML Ultra automatically takes you to the part of the diagram or the block that has a problem, as shown below.



2. Fix the problem.

3. From the menu bar, select **Diagram**. The Diagram menu appears.

4. Select **Translate to Script**. As GML Ultra translates the diagram into a script, a dialog box appears to inform you of the translation progress.



5. Repeat Steps 1 through 3 until the program translates successfully.

Creating a Script

Use the Script Editor to create a new script, view and edit text files. Examples include:

- Script documents generated from GML Ultra diagram documents.
- Script files that reside on your controller. You can upload these to your PC and save them as scripts.

The Script Editor provides standard cut, copy, paste, delete, and find/replace functions.

ATTENTION: Changes to the script cannot be converted back to the graphical diagram.

To create a new script that uses iCODE:

1. From the menu bar, select **File**. The File menu appears.
2. Select **New Script.** The menu changes to one specifically for working with scripts.

3. Enter program code (iCODE) to create your new script program. Use the editing features described next to edit your program text.

Editing a Script

Using the Edit Menu Functions

Use the Edit menu to access standard edit functions:

Inserting Text

To insert text at a specific location, do one of the following:

- Press any of the four arrow keys on the keyboard to reach the location you want.
- Click the mouse once with the I-beam cursor at the desired location.

When the Script Editor window is active, the text insert position is indicated by a blinking vertical bar. Characters typed from the

keyboard appear at this location. While the cursor is inside the Script Editor window, it appears as an I-beam.

Deleting One Character of Text

To delete one character to the right of the text insertion point:

1. Delete the character to the left of the text insert location.
2. Place the cursor to the immediate left of the character to be removed.
3. Press the **Delete** key. The character is deleted.

Deleting a Section of Text

To delete a section of text:

1. Cut or delete a section of text.
2. Click and hold the mouse button at the beginning of the text to be removed.
3. Drag the cursor to the end of the text to be removed.
4. Release the mouse button. The text is highlighted.
5. From the menu bar, select **Edit**. The Edit menu appears.
6. Select **Cut**. The selection is deleted without being copied to the clipboard.

Copying a Section of Text

To copy a section of text:

1. Click and hold the mouse button at the beginning of the text to be copied.
2. Drag the cursor to the end of the text to be copied.
3. Release the mouse button. The text is highlighted.
4. From the menu bar, select **Edit**. The Edit menu appears.
5. Select **Copy**. The selection is copied to the clipboard.

Pasting Text

To paste text that has been cut or copied to the clipboard:

1. Place the cursor into the script at the new location.
2. From the menu bar, select **Edit**. The Edit menu appears.
3. Select **Paste**. The selection is pasted into the script at that location.

Using the Script Menu Functions

Use the Script menu to access these functions:

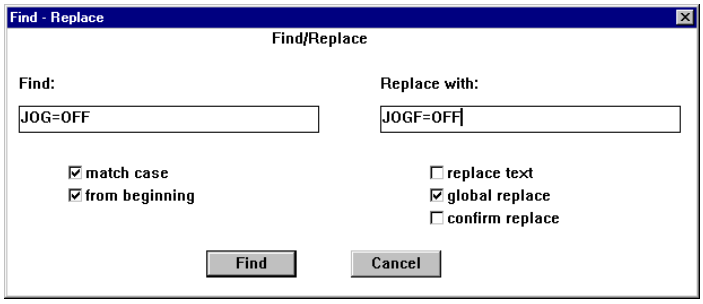
- Find and Replace text strings
- Find again

Finding/Replacing Text

Find allows efficient searches for known script patterns. Replace allows you to automatically make global changes. These tools are especially effective for multiple searches in long documents

To find and/or replace script patterns:

1. Convert your diagram to script. Refer to the *Translating a Diagram to Script* section.
2. From the menu bar of the active script window, select **Script**. The Script menu appears.
3. Select **Find-Replace**. The Find-Replace dialog box appears.



4. In the *Find* field, type the text you want to search for in the current text file exactly as it appears in the script file.

   Note: Pay special attention to dots and spaces. The search is not case sensitive unless you select that option. Refer to Step 6.

5. In the *Replace with* field, type the text you want to replace in the *Find* field.

6. Complete the check boxes for the following fields:

Field	Description
match case	Select the box to match upper and lower case to exactly what you typed in the *Find* and *Replace with* fields.
from beginning	Select the box to search the entire file from the beginning, instead of starting where the cursor rests.
replace text	Select the box to search and replace one find at a time.
global replace	Select the box to search and replace all the finds in the file at one time.
confirm replace	Select the box to perform a global search, but to confirm each occurrence before you replace it with the new text.

7. Select **Find**. The search begins. When the text in the *Find* field is identified, the new text automatically replaces the old text, the new text is highlighted and the search stops.

Find/Replace Again

Use Find Again to find, in the current file, the next occurrence of the text you searched with Find/Replace.

Note: The Find-Replace dialog box does not change even when you activate a new diagram or a new script. Therefore, you can search for the same text in different files without typing new text in the *Find* and *Replace With* fields.

To find previously defined text:

1. Make sure the script file you want to search is active.

2. From the Script Main Menu, select **Script**. The Script menu appears.

3. Select **Find Again**. The next occurrence of the search text is highlighted.

Printing Your Script

For information about printing a script, see Printing Files in the *General Procedures* chapter.

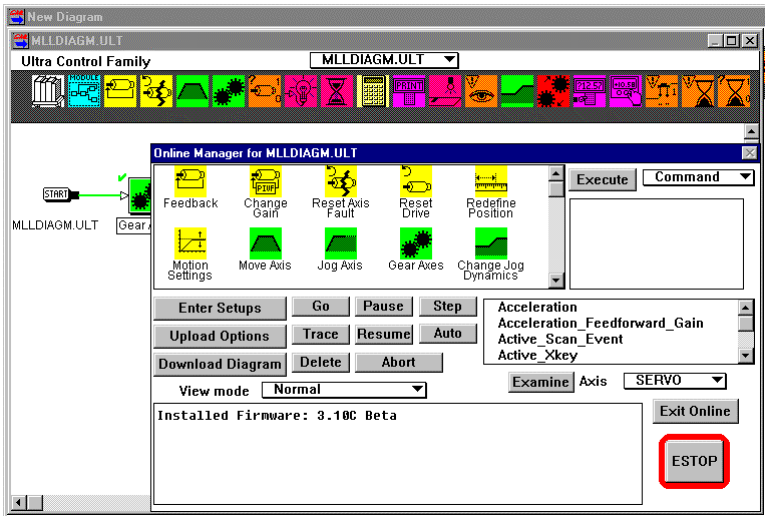
Going Online

You can build and edit your diagram, create variables, and test connections without connecting to your ULTRA Plus controller, but you must go online to communicate with the controller.

1. Turn on your ULTRA Plus.

2. From the menu bar, select **Diagram**. The Diagram menu appears.

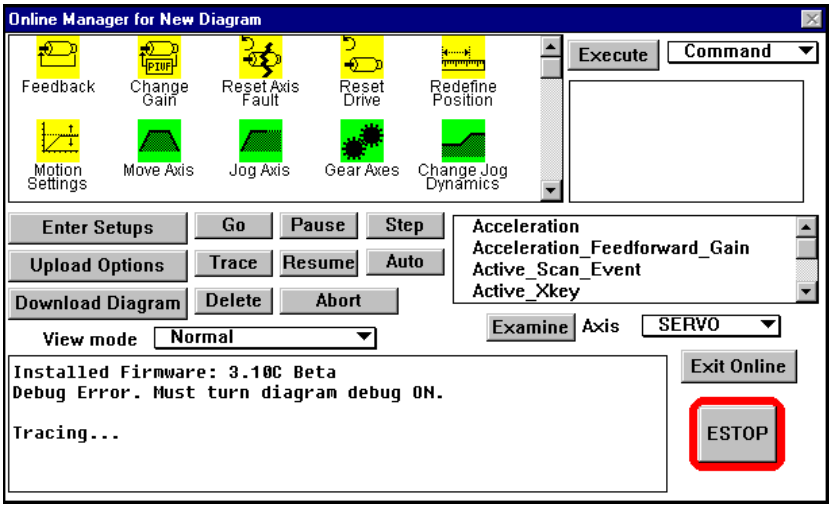
3. Select **Online**. The Online Manager window appears in the lower right corner of the screen and communication with your ULTRA Plus is enabled.



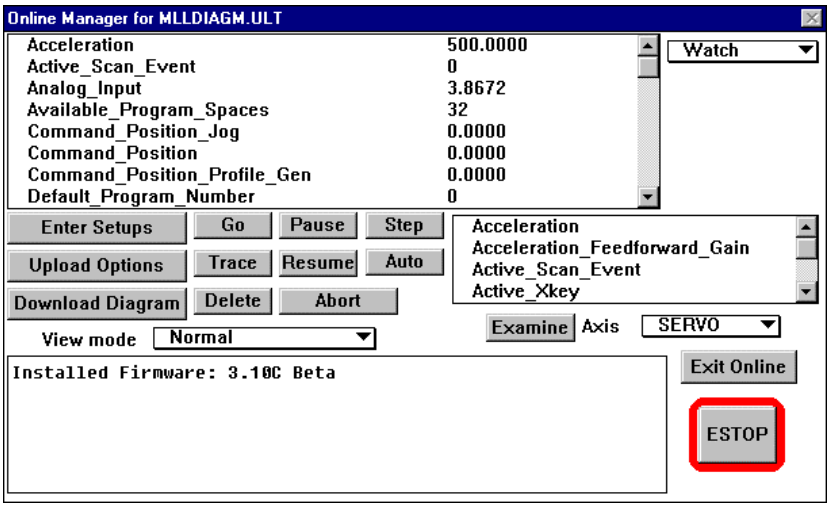
Using the Online Manager Window

The Online Manager window provides direct communication to the ULTRA Plus. This window has two primary functions: to execute a direct motion command to the ULTRA Plus and to display watch item variables. You select the function you want active by selecting Command/Watch in the top right corner of the window. Regardless of the option you select, the available options remain the same with one exception—the content of the box in the top left block changes to reflect your selection.

If you select Command, the box contains function blocks to execute.



If you select Watch, it displays watch item variables for monitoring.



How the Window is Organized

The Online Manager window has three areas: upper, center, and lower. The organization of the window is summarized below.

Window Area	Element	Purpose	For More Information Refer to this section
Upper	Command/Watch button (at top right)	To select the primary function of the window: • To execute a direct motion command to the ULTRA Plus (Command) • To display Watch Item variables (Watch)	*Selecting a Communication Function* *Executing a Direct Motion Command* *Monitoring Watch Items*
	Execute button	To execute the command (function block) you selected in the large box to the left.	*Executing a Direct Motion Command*
	Large box	• If you select Command as the primary function of the window, this box contains function blocks. • If you select Watch as the primary function of the window, you can monitor what your controller and motor are doing and how your program is running.	N/A
Center	Group of Buttons	Use to control online functions.	*Controlling Online Functions*
	Variable List Box	List of variables you can examine.	*Examining Variable Values*
	Examine Button	After selecting a variable, press this button to display the contents of the variable.	*Examining Variable Values*
	View Mode	Select a format for the information in the box in the lower portion of the window.	*Choosing the Information View Mode*
	Axis	Select the Servo or Master Axis.	N/A
Lower	Large box	Display the status of the online operation in progress.	N/A
	Exit Online button	Select to exit online mode.	N/A
	ESTOP button	Select to kill motion, stop feedback execution, and turn off Feedback.	*Stopping a Program and Killing Motion*

Choosing the Information View Mode

In the *View mode* field, select:	To view information in:
Normal	English. Information is independent of the controller language. Use this view for full graphical tracing of program execution in the Diagram Editor window.
Full Size terminal	iCODE. Information is shown in the native language of the ULTRA Plus.
2 x 40 terminal	A smaller display window.

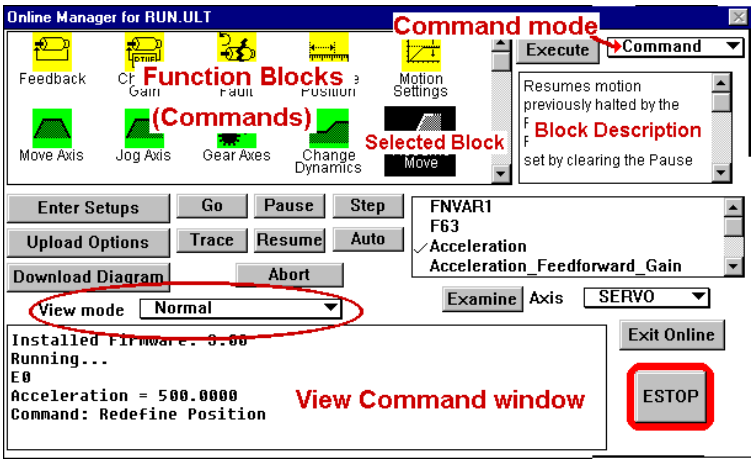
Selecting a Communication Function

The Online Manager window can function as a command window or as a watch window. The Command window allows you to communicate with your controller. The Watch window allows you to monitor what your controller and the motor are doing and how your program is running.

Sending Commands Directly to an ULTRA Plus

To send commands directly to an ULTRA Plus:

1. In the *Command/Watch* field in the Online Manager window, select **Command**. The Command window appears.



2. Select the block command from the graphical block command browser.

3. In the Online Manager window, select **Execute**. The command is sent to your controller.

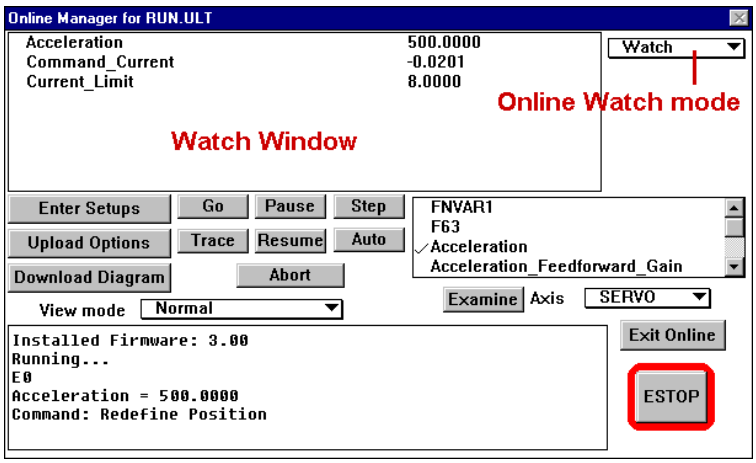
Note: The Online Manager window opens in Command mode.

Showing Watch Items

GML Ultra allows you to watch your program as it executes by using a watch window.

To show the watch items:

1. In the *Command/Watch* field in the Online Manager window, select **Watch**. The Watch window appears.



2. Select what you want to watch from the Watch window.

Closing the Watch Window

To close the Watch Items window, select **Command** from the Command/Watch list box.

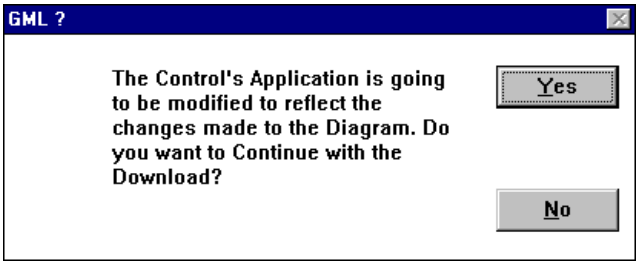
Note: The Watch window must be closed before you can download, upload, or delete a file.

Downloading Your Diagram

The download process translates the diagram to script and downloads the script to the controller. If you make functional changes to your diagram, you must download it again before you run it. You do not need to download if the changes are non-functional changes, such as dragging blocks or connections.

To download your diagram:

1. In the Online Manager window, select **Download Diagram**. The download process begins. A dialog box appears requesting confirmation to download.



2. Select **Yes**. The download continues.

Downloading Axis and Drive Setup Data

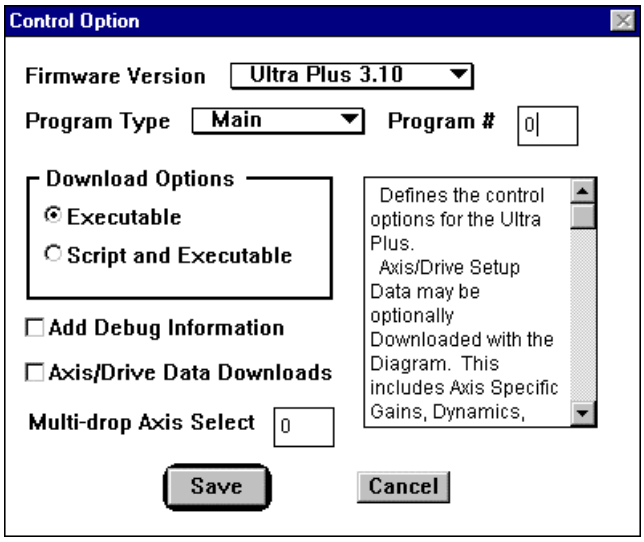
You can download axis/drive setup data along with the diagram. This includes axis specific gains, dynamics, drive data, encoder data, motion profiles, tolerances, homing, overtravel and fault configurations. If you have changed the GML Ultra default setup options and want to override the setup configuration resident in the controller, you must select the *Axis/Drive Data Download* field in the Control Options dialog box.

ATTENTION: Be sure you have valid axis/drive data in the Axis Use dialog box before you download your diagram if you have selected *Axis/Drive Data Download*.

To download the saved axis/drive data with your diagram:

1. From the menu bar, select **Definitions**. The Definitions menu appears.

2. Select **Control Options**. The Control Options dialog box appears.



3. Select **Axis/Drive Download Data**. A checkmark appears in the box.

4. Select **Save**. The configuration you defined in the GML Ultra Definitions menu is downloaded to the controller. The new settings override the settings resident in the controller.

Executing a Direct Motion Command

To execute a direct motion command:

1. In the *Command/Watch* field in the Online Manager window, select **Command**. The Command window appears.

2. In the Direct Command window, select the appropriate function block. The block is highlighted.

3. Select **Execute**. The function executes and the script for the function displays in the display field.

Deleting a Diagram

To delete an executable diagram (and script file) residing in the ULTRA Plus.

1. In the ULTRA Plus Directory dialog box, select the number or name of the file to delete.

IMPORTANT: Before you delete an executable diagram or script file, make sure you have a backup of it.

2. Select the radio button for the type of file you want to delete—the executable diagram or both script and executable diagram.

Controlling Online Functions

You can control most online functions by using the buttons that appear in the center of the Online Manager window. These functions are of two types:

- for managing program files
- for managing program execution

Functions for Managing Program Files

Use this button:	To do this:	For more information refer to:
Delete	Delete a diagram (and script) residing in an ULTRA Plus.	*Deleting a Diagram*
Download Diagram	Translate and send the current diagram to the ULTRA Plus. The program type and program # selected in Control Options (on the Definitions menu) determine what type of program is downloaded and where it is stored in the ULTRA Plus. If you change the program you must download it before trying to run it.	*Downloading Your Diagram*
Enter Setups	Display the Axis Setups dialog box. From here you can: • Select and run diagnostic tests. • Tune motors and drives. • Monitor I/O status. • Initialize the Personality Module. • Display version information of the ULTRA Plus firmware.	
Upload Options	Display the Transfer Control dialog box. From here you can: • Transfer a script file from the ULTRA Plus to the PC • Send or receive an entire NVRam image from the PC to the ULTRA Plus. • Transfer the Fault History from the ULTRA Plus to the PC. • Send or receive an executable file from the PC to the ULTRA Plus.	*Upload Options*

Functions for Managing Program Execution

Use this button:	To do this:	For more information refer to this section:
Abort	Stop program execution.	*Stopping a Program*
Auto	Automatically step through an application program one instruction at a time.	*Using Auto Trace*
ESTOP	Kill any motion, stop program execution, and turn the Feedback off.	*Stopping a Program and Killing Motion*
Examine	Examine the value of any defined variable. Select the variable from the variable list, then select the Examine button. To monitor a variable continuously in real time, select Watch from the *Command/Watch* field.	
Go	Execute an application program residing in an ULTRA Plus. The program to be run is determined by the program type and program # selected in Control Options (on the Definitions menu and must have been downloaded to the ULTRA Plus).	
Resume	Resume program execution. When motion resumes, the position target is the same as before the interruption.	*Resuming a Suspended program*
Step	Manually step through an application program one instruction at a time.	*Using Step*
Trace	Show program flow while running the program at full speed.	*Using Trace*

Starting and Stopping Programs

GML Ultra allows you to:

- start a downloaded program
- pause a program
- suspend a program
- resume a suspended program
- stop a program
- stop a program and kill motion

Starting a Downloaded Program

Note: Before you start the program, you must download the diagram to the controller. Once it is downloaded, the control-resident application program is synchronized with the diagram.

Note: If you select Go while in Trace mode, Trace mode is aborted.

To start an application program after it has been downloaded to your ULTRA Plus, in the Online Manager window, select **Go**. The program runs.

Pausing a Program

To pause the program at a specific point to monitor machine function, set a breakpoint. Breakpoints can only be set after the diagram is downloaded. The diagram must be downloaded every time a change is made to it. You can set up to twenty-four breakpoints:

1. After your PC is online with the controller and your program is downloaded, select the block in the location at which you want to pause the program.
2. From the Diagram menu, select **Breakpoint**. The checkmark turns black.
3. Select **Go**. The program executes until the breakpoint is identified.
4. The program pauses.
5. To restart the program, select **Resume**. The program executes from the point where it paused.

Suspending a Program

To suspend program execution, in the Online Manager window, select **Pause**. The program suspends execution. It can be resumed without a new download.

Resuming a Suspended Program

To resume a suspended program, from the point of suspension in the current run mode in the Online Manager window, select **Resume**. The program restarts from the point of suspension in the current run mode.

Stopping a Program

To stop a program, in the Online Manager window, select **Abort**. The program stops.

Stopping a Program and Killing Motion

To stop a program and kill motion, press **ESTOP**. The enabled relay is opened.

To recover from this condition, either execute a Feedback on the block or cycle power to the controller.

Monitoring Program Flow

GML Ultra provides several options for monitoring program flow:

Option	Description
Trace	Displays program flow, with the executing block highlighted. Runs at full speed.
Auto	Displays program flow, with the executing block highlighted. Runs at less than full speed
Step	Displays program flow. You control the process by pressing a button to step through the program one block at a time.
Breakpoints	Set a specific point to pause a program so you can check it.
Watch	To display variables you want to monitor.

Using Trace

Use Trace to periodically show program flow while running at full program speed.

Note: Because trace is fast, use Step to monitor the program at a suitable pace.

In the Online Manager window, select **Trace**. The Diagram Editor window changes to Trace mode. Blocks in the Diagram Editor window are highlighted as they are executed in the program.

Selecting a View Mode

Select Normal view to run Trace, Auto, and Step.

Tracing Tasks within a Diagram

To trace tasks within a diagram:

1. In the Online Manager window, select **Download Diagram**. The program is sent to the controller.
2. In Normal view mode, select **Trace**. The program runs at full speed and highlights blocks as they are executed.

Terminating Trace Mode

To terminate Trace mode and continue program execution without aborting the program, select **Go**.

Using Auto

Use Auto to automatically step through program blocks. Auto shows program flow at slower than normal program speed. In the Online Manager window, in Normal view mode, select Auto. When execution of that block is complete, the next block in the diagram is highlighted.

Selecting a View Mode

Select Normal view to run Trace, Auto, and Step.

Using Step

Use Step to manually step through a program one block at a time:

1. In the Online Manager window, select **Normal View**.
2. Select **Step**.

Selecting a View Mode

Select Normal view to run Trace, Auto, and Step.

Setting a Breakpoint

You can set and clear breakpoints from the Diagram menu but you must be online to set a breakpoint.

You can set the breakpoint only after the diagram is downloaded. A block with a breakpoint set has its validation check mark highlighted in black instead of green. Up to 24 breakpoints are supported. When a breakpoint is reached, the block with the breakpoint is highlighted.

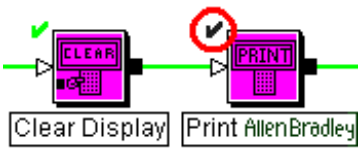
You can pause program execution when it reaches a breakpoint so you can check your motor function or fine tune your program.

To set a breakpoint in your diagram:

1. In the Online Manager window, download the program.
2. In the diagram, select the block at the location at which you want the program to stop.
3. From the menu bar, select **Diagram**. The Diagram menu appears.
4. Select **Breakpoint**. The Breakpoint Control dialog box appears.
5. Select **Set Breakpoint**.
6.

To:	Do this:
Cause the program to stop when it reaches a specified block	Select **Stop Program When Reached**.
Allow the program to continue without pausing	Go to step 7.

7. Select **OK**. The menu closes and the green checkmark at the upper left corner of the block changes to black. The breakpoint is set.



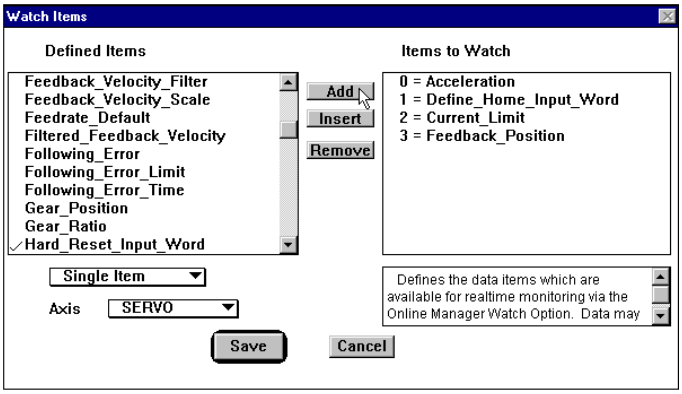
Defining Watch Items

You can monitor specific functions within your diagram as the program executes.

To identify the items you want to watch during Trace:

1. From the menu bar, select **Definitions**. The Definitions menu appears.

2. Select **Watch Items**. The Watch Items dialog box appears.



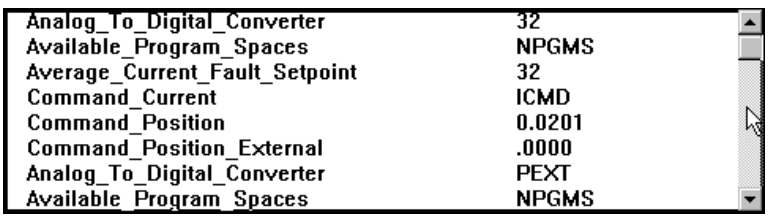
3. In the *Defined Items* field in the Watch Items dialog box, select the variables you want to track. A check mark ✓ appears in front of the selected variable.

4. Select **Add**. The variable appears in the *Items to Watch* field.

5. Repeat steps 3 and 4 to add more items.

6. Select **Save**. The dialog box closes. The items are placed in the Variable View window on the Online Manager window.

7. In the *Command/Watch* field in the upper right corner of the Online Manager window, select **Watch.** The Watch window appears. The selected items appear in the Watch Window in the Online Manager window.



8. Scroll the Watch window to the find the variable(s) you want to monitor.

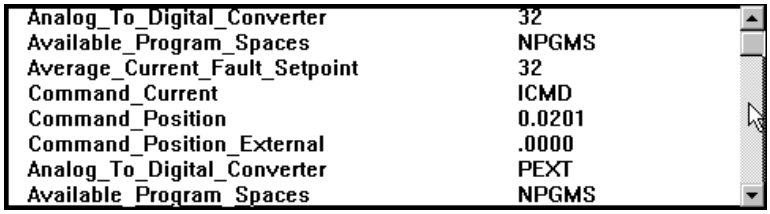
Note: Refer to *Watch Items* in *The Definitions Menu* section of this manual for more information on defining the items to watch.

Monitoring Watch Items

To monitor pre-defined watch items in real time:

1. In the *Command/Watch* field in the upper right corner of the Online Manager window, select **Watch.** The Watch window appears.

2. Scroll the Watch window to the find the variable(s) you want to monitor.



Examining Variable Values

To display a description of a variable and its value:

1. From the Variable List in the Online Manager window, select a variable. A checkmark appears in front of the variable.

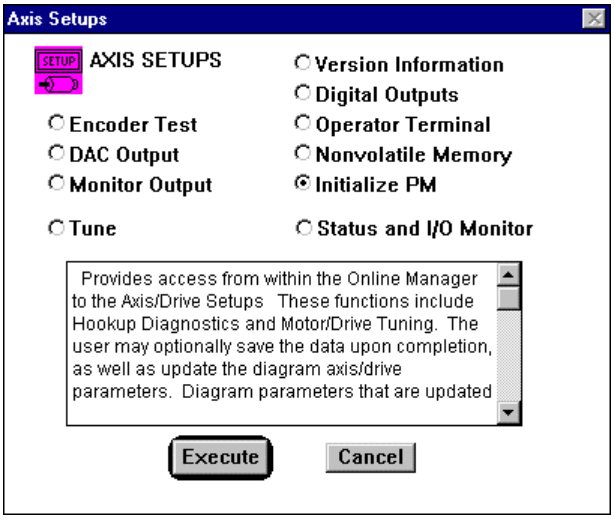
2. Select **Examine**. An English description of the variable and its value appear in the View window.

Selecting Axis Setups Options

Use the Axis Setups dialog box for several tasks:

- To view version information. Refer to *Viewing Version Information*.
- To initialize the Personality Module. Refer to *Initializing the Personality Module*.
- To select and run diagnostic tests. Refer to *Selecting and Running Diagnostic Tests.*
- To tune motors and drives. Refer to *Tuning Motors and Drives*.
- To monitor I/O status. Refer to *Monitoring I/O Status*.

To display the Axis Setups dialog box, select **Enter Setups** in the Online Manager window.



Viewing Version Information

To view the ULTRA Plus firmware version:

1. In the Online Manager window, select **Enter Setups**. The Axis Setups window appears.

2. Select **Version Information**.

3. Select **Execute**. The current firmware version appears in the window on the bottom of the Online Manager window.

Initializing the Personality Modules

Before you can initialize the personality module, your ULTRA Plus must be disabled.

To initialize and load the appropriate personality module files for PDM and motor combination:

1. Apply input power and check the LEDs:
   - The green DC BUS LED should turn on to indicate power is applied.
   - The bicolor status LED should turn green.
   - If the status light is red, the PDM is in a fault condition. Continue with the following instructions and clear faults. To clear faults, use the Status and I/O Monitor option on the Axis Setups dialog box. Refer to the *ULTRA Plus Installation Manual* (Publication 1398.51) for more information.
2. From the menu bar, select **Diagram**. The Diagram menu appears.
3. Select **Online**. The Online Manager window appears.
4. Select **Enter Setups**. The Axis Setups dialog box appears.
5. Select **Initialize PM.**
6. Select **Execute**. The Initialize Personality Module dialog box appears.
7. Make entries in the following fields:

Field	Description
Drive Type	Select the appropriate PDM type.
Motor Type	Select the appropriate motor type.
Motor/Encoder Size	Select the appropriate motor encoder size.

8. Select **Update PM**.
9. Select **OK**. The Personality Module initialization begins. A message appears showing the percent of the update that is complete.
10. Select **Close**. The Initialize Personality Module window closes.
11. Select **Cancel**. The Axis Setups dialog box closes.

Selecting and Running Diagnostic Tests

The following table identifies the purpose of each test. Step-by-step instructions for completing each test follow.

IMPORTANT: To run these tests:

- Make sure the ULTRA Plus is disabled.
- Make sure diagrams are not running.

This test:	Does this:
Encoder	Checks both Encoder 1 and Encoder 2.
DAC Output	Exercises the Digital and Analog Converter (DAC1) output.
Monitor Output	Exercises the Monitor Output.
Digital Outputs	Allows digital outputs to be forced ON or OFF to test the outputs and system wiring.
Operator Terminal	Verifies communication between the ULTRA Plus and the Operator Terminal.
Nonvolatile Memory	Does a non-destructive test of the ULTRA Plus nonvolatile RAM (NVRam.)
I/O Status Monitor	Continuously monitors machine status and the status of the digital inputs and outputs. Can be done anytime — even if a program is running.

Testing the Encoder

The Encoder test checks both Encoder 1 and Encoder 2.

To test and/or check the size of an encoder:

1. From the menu bar, select **Diagram**. The Diagram menu appears.
2. Select **Online**. The Online Manager window appears.
3. Select **Enter Setups**. The Axis Setups dialog box appears.
4. Select **Encoder Test**.
5. Select **Execute**. The Encoder dialog box appears.

6.

To:	Do this:
Verify that encoder counts are being received	1. Rotate the encoder. As the counts are received, the count display is incremented or decremented depending on the direction the encoder is rotated. 2. Select **Zero Counts** to clear the count display.
Check the size of the encoder	1. Select **Zero Counts** to clear the count display. 2. Rotate the encoder slowly until the size appears in the *Size* field. The size displayed is in encoder counts (number of encoder lines multiplied by 4).

Testing the DAC Output

The DAC1 output is a 12 bit Digital to Analog Converter (DAC) that is used to provide an analog voltage signal. The range of the output is ±10 volts which provides a resolution of about 5 mV.

The DAC Output test exercises the DAC1 output. The test outputs either a triangular waveform between ±10 volts or a fixed voltage on the DAC1 output. To test the DAC1 output:

1. Connect a meter or oscilloscope to the DAC1 output (P3-6) and to analog common (P3-4).

2. In the Axis Setups dialog box, select **DAC Output**.

3. Select **Execute**.

4.

To:	Do this:
Output a triangular waveform with a range of ±10 volts	1. Select Triangular Waveform. 2. Select Start. A Waveform Running message appears.
Output a fixed voltage	1. Select Set **DAC Voltage**. 2. Type the desired voltage (between ±10 volts. 3. Select **Start**. A Voltage Set message appears.

5. Check the voltage signal on the meter or oscilloscope.

6.

To:	Do this:
Stop a waveform or the fixed voltage signal	1. Select Set DAC Voltage. 2. Type a value of 0 volts. 3. Select Start. A Voltage Set message appears.
Stop a waveform or the fixed voltage signal and close the dialog box	Select Close.

7. Once the test is running, to change from one type of waveform to the other, select the corresponding radio button.

8. To perform more DAC output tests, go to step 4.

Testing the Monitor Output

The Monitor Output test exercises the Monitor Output. The Monitor Output is an 8 bit Digital to Analog Converter (DAC) that can be used to monitor a variable. The range of the output is ±10 volts. The test outputs either a triangular waveform between ±10 volts or a fixed voltage on the Monitor output.

To test the Monitor Output:

1. Connect a meter or oscilloscope to the Monitor output (P3-7) and to analog common (P3-4).

2. In the Axis Setups dialog box, select **Monitor Output**.

3. Select **Execute**.

4.

To:	Do this:
Output a triangular waveform with a range of ±10 volts	1. Select **Triangular Waveform**. 2. Select **Start**. 3. Select **Start**. A Voltage Set message appears.
Output a fixed voltage	1. Select Set DAC Voltage. 2. Type the desired voltage (between ±10 volts. 3. Select **Start**.

5. Check the voltage signal on the meter or oscilloscope.

6.

To:	Do this:
Stop the waveform or the fixed voltage signal	1. Select **Set DAC Voltage**. 2. Type a value of 0 volts.
Stop the waveform or the fixed voltage signal and close the dialog box	Select Close.

7. Once the test is running, to change from one type of waveform to the other, select the corresponding radio button.

8. To perform more monitor output tests, go to step 4.

Testing the Digital Outputs

The Digital Outputs test allows digital outputs to be forced on or off to test the outputs and system wiring. The relay outputs, Ready and Enabled, can also be turned on and off in this test.

To test the Digital Outputs:

1. In the Axis Setups dialog box, select **Digital Outputs**.

2. Select **Execute**. The Digital Outputs dialog box appears.

3.

To:	Do this:
Turn on an output	Select the check box corresponding to the output number.
Turn on all outputs	Select **All On**.
Turn the Ready or Enabled relay on	Select the appropriate check box.
Turn off individual outputs	Clear the corresponding check box.
Turn all outputs off	Select **All Off**.
Turn the Ready or Enabled relay off	Clear the appropriate check box.

The outputs are restored to their previous state when you select Close.

Testing the Operator Terminal

The Operator Terminal test verifies communication between the ULTRA Plus and the operator terminal. This test sends a string of characters to the operator terminal and displays the characters sent, so you can visually check the operator terminal. Any operator terminal keys that are pressed while in this test are displayed on the screen.

To test the operator terminal:

1. In the Axis Setups dialog box, select **Operator Terminal**.
2. Select **Execute**. The Operator Terminal dialog box appears and the test starts immediately.
3. Select **Close**. The test stops.

Characters sent to the operator terminal are also sent to the dialog box for verification. Any keypress action at the operator terminal (on or off) is also shown in the dialog box.

Testing the Nonvolatile Memory

The Nonvolatile Memory test does a non-destructive test of the ULTRA Plus nonvolatile RAM (NVRam). The number of passes completed and the current block being tested are shown. If there is a failure, a message box appears showing the address where the failure occurred. As long as there are no failures, the test continues until you select Close.

To test the NVRam:

1. In the Axis Setups dialog box, select **Nonvolatile Memory**.
2. Select **Execute**. The Nonvolatile Memory dialog box appears and the test starts immediately.
3. Select **Close**. The test stops.

Tuning Motors and Drives

Two tune modes are available:
- Auto Tune
- Manual Tune

Understanding Auto Tune

Auto Tune mode provides a method for tuning the servo amplifier connected to a machine without any special equipment other than the serial terminal. It allows a reasonable set of tuning parameters to be

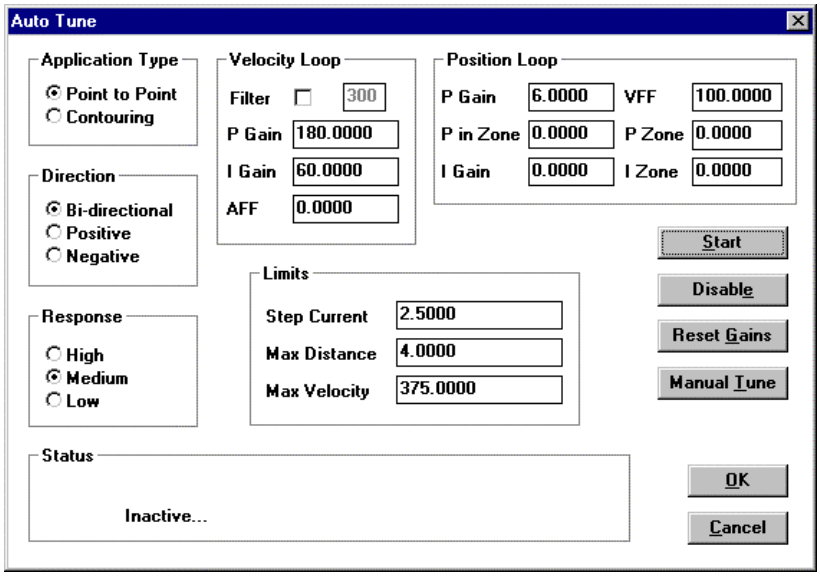
developed quickly for a particular machine. For many applications, running Auto Tune generates tuning parameters that are adequate for machine performance. However, if you require very high performance, or special conditions exist (such as changing loads or large inertia mismatches), you can use Manual Tune to fine-tune the tuning parameters after you run Auto Tune.

Auto Tune implements a simple self-tuning algorithm that adjusts the tuning parameters by computing the total inertia, consisting of the motor inertia plus the load inertia at the motor shaft. Auto Tune operates by commanding a constant current to the motor, producing a constant motor torque. The acceleration of the motor is measured and used to compute the inertia of the system. Once the actual inertia of the system is measured, the values of the tuning parameters are automatically adjusted to achieve the desired performance.

Using Auto Tune

To tune the system using Auto Tune:

1. Disable the ULTRA Plus.
2. In the Online Manager window, select **Enter Setups**. Axis Setups dialog box appears.
3. Select **Tune**.
4. Select **Execute**. The Auto Tune dialog box appears.



Note: The fields in the *Application Type* and *Response* areas are used to calculate the system tuning gains after the system inertia estimation is complete.

5. In the *Application Type* area, make entries in the following fields:

Field	Description
Point to Point	Select this option if your application is point to point.
Contouring	Select this option if your application is contouring.

6. In the *Direction* area, make entries in the following fields:

Field	Description
Bi-directional	Select this option to rotate the motor shaft in both directions.
Positive	Select this option to rotate the motor shaft in a clockwise direction when looking at the motor shaft end.
Negative	Select this option to rotate the motor shaft in a counter-clockwise direction when looking at the motor shaft end.

Note: Max Distance is the maximum distance moved during each acceleration and deceleration cycle of the Auto Tune process.

7. In the *Response* area, make entries in the following fields:

Field	Description	
High	Select this option if the bandwidth is:	And if you selected:
	10 Hz	Position loop
	30 Hz	Velocity loop
Medium	Select this option if the bandwidth is:	And if you selected:
	5 Hz	Position loop
	15 Hz	Velocity loop
	Select this option if you are not sure which response suits your application.	
Low	Select this option if the bandwidth is:	And if you selected:
	2 Hz	Position loop
	7 Hz	Velocity loop

8. In the *Velocity Loop* area, the system makes entries in the following fields:

Field	Description
Filter	Select this field to enable the low pass filter on the output of the velocity regulator. The value is the bandwidth in Hz of the low pass filter on the output of the velocity regulator. Reducing the value of the filter smooths the torque command, which reduces noise from high frequency torque pulsations. The maximum value is 300 Hz. The system turns the filter off.
P Gain	Proportional gain of the velocity loop. Increase proportional gain to reduce dynamic velocity errors and to increase the velocity loop bandwidth.
I Gain	Integral gain of the velocity loop. Integral gain is used to improve the stiffness of the velocity loop and to reduce the effects of load disturbances. Excessive integral gain results in velocity overshoot and could cause instability. Auto tune sets the velocity loop integral gain (I) to zero for point to point. For contouring, integral gain is set to a non-zero value that results in about a 15% velocity step overshoot.
AFF	Acceleration feedforward gain of the position loop in percent. The system sets acceleration feedforward gain (AFF)=0%.

9. In the *Limits* area, the system makes entries in the following fields:

Field	Description
Step Current	Type the step current in user units per the Timebase for *Max Velocity* field. The default value should provide good results for most systems. Reduce the value if the torque exceeds machine specifications.
Max Distance	Type the maximum distance in user units. The default value should provide good results for most systems.
Max Velocity	Type the maximum velocity in user units per the *Timebase for Max Velocity* field. The default value should provide good results for most systems.

10. In the *Position Loop* area, the system makes entries in the following fields:

Field	Description
P Gain	Proportional gain adjusts the bandwidth of the position loop. The higher the value of the proportional gain, the stiffer the system response. Proportional gain appears in inches per minute per mil (i.e., meters per minute per millimeter).
P in Zone	Position loop proportional gain in zone is used when the system is within the region of the commanded position defined by the *PZone* field.
I Gain	Position loop Integral gain is used to bring the system into the desired position more quickly and increase the stiffness of the positioner. The *IZone* field determines the region around the commanded position where integral gain is active. The system sets position loop integral gain (I Gain)=0.
VFF	Velocity feedforward gain adjusts the following error of the position loop. Velocity feedforward gain is entered in percent. The system sets velocity feedforward gain=100%.
PZone	Proportional zone is the region around the commanded position where the position loop proportional gain is changed to the gain set by the *P in Zone* field. The proportional zone value appears in user units as defined in the *Transducer Counts/Unit* field. The system sets proportional zone (Pzone)=0.
IZone	Integral zone is the region around the commanded position where the integral gain is active. The integral zone value appears in user units as defined in the *Transducer Counts/Unit* field. The system sets integral zone (Izone)=0.

11. Select **Enable**. The ULTRA Plus is enabled.

12. Select **Start**. The Auto Tune process begins. The motor shaft oscillates back and forth.

Note: Enable and Start are toggle buttons. When you select Enable, for example, it changes to Disable and when you select Start, it changes to Stop.

After about 5 seconds, the motion ceases and the calculated load inertia to motor inertia ratio and the new calculated gains appear.

Under certain conditions, an incorrect value can be computed for the system inertia that results in high gain settings. This could cause instability when returned to the closed-loop mode of operation. The conditions are:

Condition	Description	Solution
Max Distance or Max Velocity is set too low	Either of these conditions can cause very rapid reversal of the servo motor and audible vibration.	Increase Max Distance or Max Velocity.
Step current is set too low	The servo motor does not move.	Increase Step Current.

The maximum gain values are position loop

- Position Loop PGain=14.9 inches/minute/mil
- Velocity Loop PGain=32,767
- Velocity Loop IGain=32,767

13. If the gains are set to the maximum value, repeat the Auto Tune process with higher values for Step Current, Max Distance, and Max Velocity.

14. After the Auto Tune process is complete:

If:	Do this:
You are satisfied with the new gain settings	Select **OK**. Your information is saved.
You are not satisfied with the new gain settings	Select **Reset Gains**. The gains are restored to the previous settings.
Further fine-tuning of the gains is required	Go to *Understanding Manual Tune.*

15. Select **Cancel**. The Auto Tune dialog box closes.

Understanding Manual Tune

If you require very high performance, or special conditions exist (such as changing loads or large inertia mismatches), you can use Manual Tune to fine-tune the tuning parameters after you run Auto Tune.

Manual Tune provides a means of tuning the velocity and position control loops independently. In Manual Tune mode, the tuning parameters are manually adjusted.

Using Manual Tune

Use Manual Tune to tune the velocity and position control loops independently. The inner velocity loop must be tuned first, as this tuning affects the position loop response. This procedure assumes that the system, including the machine the motor is connected to, can tolerate small-signal step velocity changes.

To tune the velocity loop, you change the proportional gain, integral gain, and filter. Proportional Gain is the proportional gain of the velocity loop. Increasing Proportional Gain reduces the time required to reach the commanded velocity.

Integral gain is the integral gain of the velocity loop. Integration in the velocity loop forces the motor velocity to precisely follow the commanded velocity with no error under steady state conditions (no changes in velocity, command, or load). Increasing integral gain increases the stiffness, or the ability to reject load disturbances. Increasing integral gain also, however, increases the amount of velocity overshoot when responding to a step change in velocity. Too much integral gain can cause the system to go unstable. To reduce stress on the mechanical parts of the machine, you should set proportional gain and integral gain as low as possible while still maintaining the desired performance.

The table below shows some general rules for tuning:

To:	Do This:
Increase bandwidth	Increase Velocity Loop Proportional Gain (PGain).
Increase stiffness	Increase Velocity Loop Proportional Gain (PGain) or Velocity Loop Integral Gain (IGain).
Reduce overshoot	increase Velocity Loop Proportional Gain (PGain) or reduce Velocity Loop Integral Gain (IGain).
Reduce rise time	Increase Velocity Loop Proportional Gain (PGain) or Velocity Loop Integral Gain (IGain).
Reduce resonance	Reduce the Filter value.
Reduce motor rattle	Reduce Velocity Loop Proportional Gain (PGain), Velocity Loop Integral Gain (IGain), or Filter value.

Once the velocity loop has been tuned, the position loop can be tuned.

The parameters used to tune the position loop are:
- Position Loop Proportional Gain (PGain)
- Integral Gain (IGain)
- Integral Zone (Izone)

- Velocity Feedforward Gain (VFF)
- Proportional Zone (PZone)
- Proportional Gain (P in Zone).

Changing the position loop proportional gain changes the position loop bandwidth. When you use an integral gain in the position loop you reduce the effects of friction and allow zero error when holding position.

Integral gain (IGain) is used in conjunction with the integral zone (Izone). The integral zone (Izone) is the area around the commanded position where the integral gain (IGain) is active.

The velocity feedforward gain is used in the position loop to minimize the following error when the system is moving.

The proportional zone parameter (PZone) sets a zone around the commanded position where the position loop proportional gain is changed to the gain set by the proportional gain in zone (P in Zone) parameter. Tuning using the step response allows adjustment of the position loop proportional gain (PGain), integral gain (IGain), integral zone (IZone), proportional zone (PZone), and proportional gain in zone (P in Zone).

Tuning the Velocity Loop

To tune the velocity loop using Manual Tune:

1. Follow the Auto Tune procedure to provide a near-optimal tuning starting point. Refer to *Using Auto Tune*.

2. Disable the ULTRA Plus.

3. Set the Monitor Output to Feedback_Velocity for Encoder 1 (default setting).

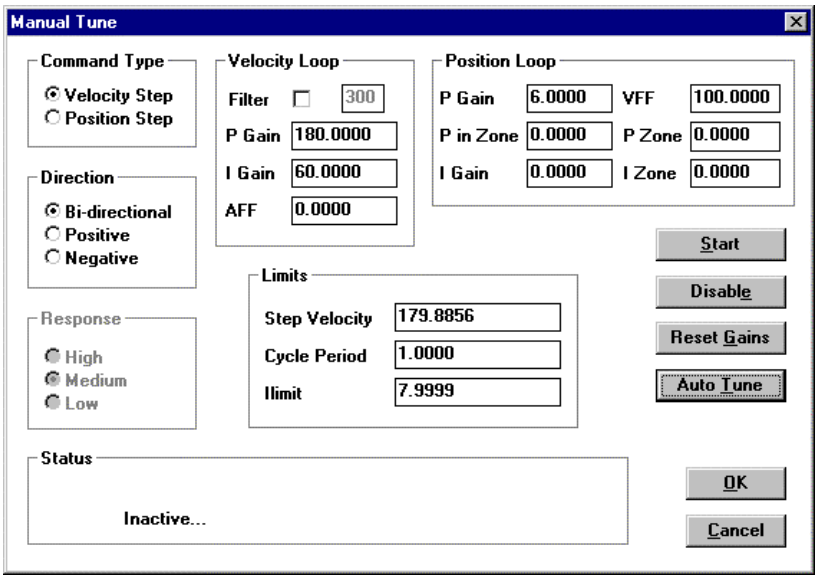
4. Connect an oscilloscope to the Monitor output (P3-7) and to analog common (P3-4).

5. In the Online Manager window, select **Enter Setups**. The Axis Setups dialog box appears.

6. Select **Tune**.

7. Select **Execute**. The Auto Tune dialog box appears.

8. Select **Manual Tune**. The Manual Tune dialog box appears.



9. In the *Command Type* area, select **Velocity Step**:

IMPORTANT: You should always tune the velocity loop before tuning the position loop as the velocity tuning affects the position loop response.

10. In the *Direction* area, make entries in the following fields:

Field	Description
Bi-directional	Select this option to rotate the motor shaft in both directions.
Positive	Select this option to rotate the motor shaft in a clockwise direction when looking at the motor shaft end.
Negative	Select this option to rotate the motor shaft in a counter-clockwise direction when looking at the motor shaft end.

11. In the *Limits* area, make entries in the following fields:

Field	Description
Step Velocity	Type the step velocity of the internal square wave generator that you want. (A good value is between 100 and 500 RPM.) Note: An excessive Step Velocity or a low peak current limit setting can cause the servo amplifier to enter current limit, which should be avoided while tuning the ULTRA Plus. If the ULTRA Plus reaches current limit, The message *In Peak Current* appears.
Cycle Period	Type the time in seconds to complete one cycle of the command. If the Direction is set to Bi-directional, one cycle is a move forward and back. If the Direction is either Positive or Negative, one cycle is the move in one direction only. Do not set the cycle period to a high value (one that would allow the system to reach the end of travel) or to a value less than 0.02 seconds.
Ilimit	Type the current limit to be used during manual tuning. When you first select Manual Tune, this is equal to the Peak Current Limit parameter. You can change Ilimit without changing the Peak Current Limit parameter.

12. Clear the *Filter* field.

13. Select **Enable**. The ULTRA Plus is enabled.

14. Select **Start**. The Manual Tune process begins. The motor shaft oscillates back and forth.

If the Step Velocity and/or Cycle Period:	Do this:
Need to be changed	**1.** Select **Stop**. **2.** Adjust the values as necessary. **3.** Select **Start**.
Do not need to be changed	Go to step 15.

15. In the *Velocity Loop* area, set the Integral Gain (IGain) to a low value (no noticeable overshoot).

16. If you haven't already done so, set the Monitor Output to Feedback_Velocity for Encoder 1.

17. While watching the Feedback_Velocity signal on the Monitor Output with the oscilloscope, in the *Velocity Loop* area increase Proportional Gain (PGain) until the desired rise time is reached.

18. In the *Velocity Loop* area, increase Integral Gain (IGain) until the acceptable limit for the amount of overshoot is reached.

19. Select the *Filter* field.

20. Reduce the value in the *Filter* field until the overshoot begins to increase.

Tuning the Position Loop

To tune the position loop using Manual Tune:

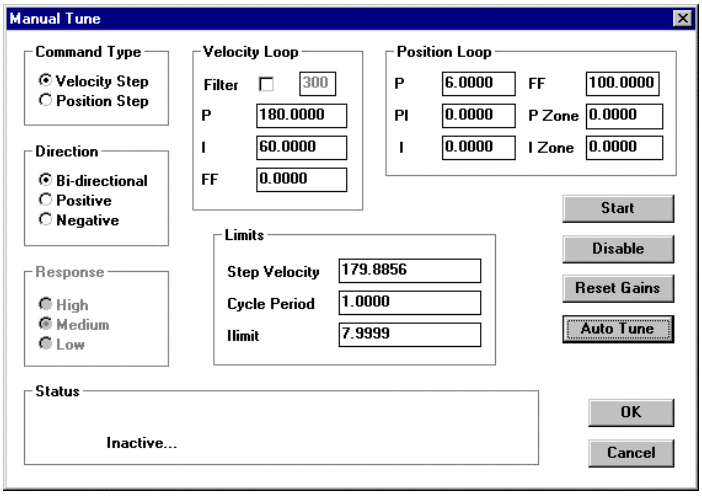
1. In the Online Manager window, use an Equation block to set the Following_Error_Limit and Following_Error_Time parameters (from the System Variable/Flags list) to the maximum allowed to avoid excess following error faults while tuning.

2. In the Online Manager window, select **Enter Setups**. The Axis Setups dialog box appears.

3. Select **Tune**.

4. Select **Execute**. The Auto Tune dialog box appears.

5. Select **Manual Tune**. The Manual Tune dialog box appears.



6. In the *Command Type* area, select **Position Step**:

7. In the *Direction* area, make entries in the following fields:

Field	Description
Bi-directional	Select this option to rotate the motor shaft in both directions.
Positive	Select this option to rotate the motor shaft in a clockwise direction when looking at the motor shaft end.
Negative	Select this option to rotate the motor shaft in a counter-clockwise direction when looking at the motor shaft end.

8. In the *Limits* area, make entries in the following fields:

Field	Description
Step Position	Type the step of the desired position change. (A good value is 0.25.)
Cycle Period	Type the time in seconds to complete one cycle of the command. If the Direction is set to Bi-directional, one cycle is a move forward and back. If the Direction is either Positive or Negative, one cycle is the move in one direction only. Do not set the cycle period to a high value (one that would allow the system to reach the end of travel) or to a value less than 0.02 seconds.
Ilimit	Type the current limit to be used during manual tuning. When you first select Manual Tune, this is equal to the Peak Current Limit parameter. You can change Ilimit without changing the Peak Current Limit parameter.

9. In the *Position Loop* area, make entries in the following fields:

Field	Description
PGain	Proportional gain adjusts the bandwidth of the position loop. The higher the value of the proportional gain, the stiffer the system response. Proportional gain appears in inches per minute per mil (i.e., meters per minute per millimeter).
P in Zone	Type **0**.

Field	Description
	Position loop proportional gain in zone is used when the system is within the region of the commanded position defined by the *PZone* field.
IGain	Position loop Integral gain is used to bring the system into the desired position more quickly and increase the stiffness of the positioner. The *IZone* field determines the region around the commanded position where integral gain is active. The system sets position loop integral gain (IGain)=0.
VFF	Type **0**. Velocity feedforward gain adjusts the following error of the position loop. Velocity feedforward gain is entered in percent. The system sets velocity feedforward gain=100%.
P in Zone	Proportional zone is the region around the commanded position where the position loop proportional gain is changed to the gain set by the *P in Zone* field. The proportional zone value appears in user units as defined in the *Transducer Counts/Unit* field. The system sets proportional zone (Pzone)=0.
IZone	Integral zone is the region around the commanded position where the integral gain is active. The integral zone value appears in user units as defined in the *Transducer Counts/Unit* field. The system sets integral zone (Izone)=0.

10. Select **Enable**. The ULTRA Plus is enabled.

11. Select **Start**. The Manual Tune process begins. The motor moves back and forth.

12.

If motor response is:	Do this:
Sluggish	Type a slightly higher value in the *PGain* field in the *Position Loop* area.
Too stiff	Type a slightly lower value in the *PGain* field in the *Position Loop* area.

13. While watching the Feedback_Velocity signal on the Monitor output with the oscilloscope, adjust the *PGain* field in *the Position Loop* area for the quickest response with minimum overshoot. It can be helpful to connect an LED and 2.2k ohm resistor in series to the In-Position output to check the In-Position output during the tuning process.

14. Adjust the *IGain* field in the *Position Loop* area.

15. Type a value in the *Izone* field in the Position Loop area (about twice the In-Position Window is a reasonable value).

16. Slowly increase the value in the *IGain* field in the *Position Loop* area from 0 while watching the Feedback_Velocity signal on the oscilloscope. As Position Loop IGain is increased, the system begins to overshoot.

17. Adjust the *IGain* field in the *Position Loop* area to achieve the fastest possible time to come into position with minimum overshoot.

18. When you are satisfied that the gains have been set to give the correct system response, select OK. The information is saved and the window closes.

19. Use an Equation block to set the Following_Error_Limit and Following_Error_Time parameters back down to reasonable values.

20.

To restore:	Do this:
Gains to the previous values	Select **Reset Gains**.
Gains to the previous values and close the dialog box	Select **Cancel**.

Monitoring I/O Status

The Status and I/O Monitor is used to continuously monitor machine status and the status of the digital inputs and outputs at any time, even if a program is running.

To get machine status:

1. In the Online Manager window, select **Enter Setup**. The Axis Setups dialog box appears.

2. Select **Status and I/O Monitor**.

3. Select **Execute**. The Status and I/O Monitor dialog box appears.

The box contains the status of all the inputs and outputs along with certain dedicated inputs and outputs, Ready and Enabled relay status, and machine faults (if any).

A check box shows the status of each input/output. If the box is selected, the corresponding input/output is active. The dedicated inputs and outputs are shown in bold if active, and grayed out if not active.

The dedicated outputs shown are:
- Program Running
- At Home
- Home Sequence Complete
- In-Position
- Error
- Ready
- Enabled

The dedicated inputs shown are:
- Forward Limit
- Reverse Limit
- Pause

Upload Options

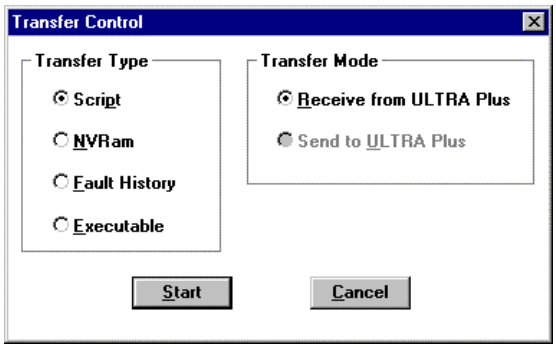
Use the Transfer Control dialog box to transfer information between your PC and the ULTRA Plus. In the Transfer Type dialog box, select the type of information you want to transfer. In the Transfer Mode dialog box, select the direction of the transfer. You can select the Send to ULTRA Plus option only when you've selected NVRam or Executable in the Transfer Type box.

Transferring a Script File

From the ULTRA Plus to the PC

To transfer a script file from the ULTRA Plus to the PC:

1. In the Online Manager window, select **Upload Options**. The Transfer Control dialog box appears.



2. Select **Script**.
3. Select **Start.** The Save As file dialog box appears.
4. Select a destination PC file name.
5. Select the program number to be transferred from the ULTRA Plus Directory dialog box.
6. Select **OK** to start the transfer.

Transferring a NVRam Image

From the ULTRA Plus to the PC

To transfer an entire NVRam image (all of the memory in the ULTRA Plus) from the ULTRA Plus to the PC:

1. In the Online Manager window, select **Upload Options**. The Transfer Control dialog box appears.
2. Select **NVRam**.
3. Select **Receive from ULTRA Plus**.
4. Select **Start**. The Save As file dialog box appears.
5. Select a destination PC file name.
6. Select **OK**. The transfer begins.

From the PC to the ULTRA Plus

To transfer an entire NVRam image (all of the memory in the ULTRA Plus) from the PC to the ULTRA Plus:

1. In the Online Manager window, select **Upload Options**. The Transfer Control dialog box appears.
2. Select **NVRam**.
3. Select **Send to ULTRA Plus**.
4. Select **Start**. The Open file dialog box appears.
5. Select the file source.
6. Select **OK**. The transfer begins.

Transferring the Fault History

From the ULTRA Plus to the PC

The ULTRA Plus stores the last 46 faults and the running time when the fault occurred. (The running time is based on the first time the ULTRA Plus was powered up.) The Fault History is saved in NVRam so it is not lost when power is removed from the ULTRA Plus.

To transfer the Fault History from the ULTRA Plus to the PC:

1. In the Online Manager window, select **Upload Options**. The Transfer Control dialog box appears.
2. Select **Fault History**.
3. Select **Start**. The Save As file dialog box appears.
4. Select a destination PC file name.
5. Select **OK**. The transfer begins.

Transferring an Executable File

From the ULTRA Plus to the PC

To transfer an executable file from the ULTRA Plus to the PC:

1. In the Online Manager window, select **Upload Options**. The Transfer Control dialog box appears.
2. Select **Executable**.
3. Select **Receive from ULTRA Plus**.
4. Select **Start.**
5. In the Save As file dialog, select a destination PC file name.
6. Select **OK** The transfer begins.

From the PC to the ULTRA Plus

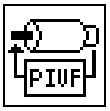
To transfer an executable file from the PC to the ULTRA Plus:

1. In the Online Manager window, select **Upload Options**. The Transfer Control dialog box appears.
2. Select **Executable**.
3. Select **Send to ULTRA Plus**.
4. Select **Start**.
5. In the Open file dialog box, Select the file source.
6. Select **OK**. The transfer begins.

Understanding Blocks

This section gives a brief explanation of each block's function and directions for using each block. The blocks names are arranged in alphabetic order.

You can find more information about function blocks in the *GML Ultra Reference Manual* (Publication 1398-5.12).



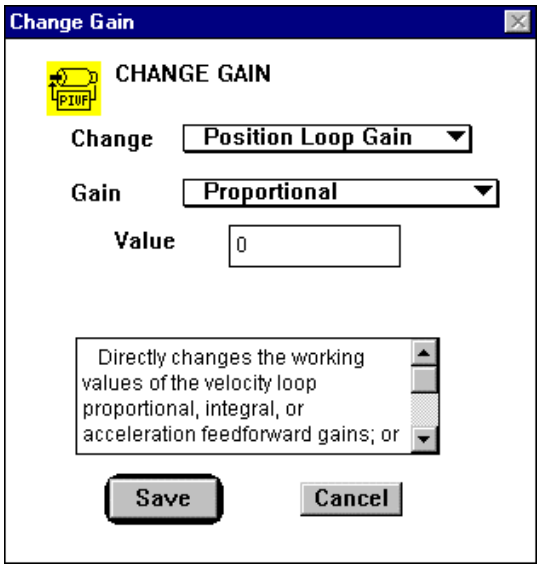
Change Gain

The Change Gain block allows you to change—on-the-fly—the working values of the gain setting of the servo axis.

Note: You can use this block to change this setting at any time. It has no effect on the stored power-up values.

To change a gain setting:

1. Double-click on the Change Gain block. A dialog box similar to the following appears:



2. Make entries in the following fields:

Field	Description
Change	Select one of the following: Position Loop Gain — To adjust a position loop gain. Velocity Loop Gain — To adjust a velocity loop gain.
Gain	Select one of the following: Proportional — To set a proportional gain. If the *Change* field is Velocity Loop Gain, increase the value to reduce dynamic velocity error and to increase the velocity loop bandwidth. Excessive proportional gain could cause instability. Proportional (in zone) — Use this when the system is in the proportional zone. If the *Change* field is Position Loop Gain, increase the bandwidth of the position loop. The higher the value, the stiffer the system response. Feedforward — To set a feedforward gain. If the *Change* field is Position Loop Gain, use this field to adjust the following error of the position loop. Enter in percent. If the *Change* field is Velocity Loop Gain, use this field to adjust the following error of the velocity loop. Enter in percent.

Field	Description
	Integral (in zone). To set an integral gain. If the *Change* field is Position Loop Gain, use this field to bring the system into the specified position more quickly and increase the stiffness of the position loop. The zone determines the region around the commanded position where the position loop gain is active. If the *Change* field is Velocity Loop Gain, use this field to improve the stiffness of the velocity regulator and to reduce the effects of load disturbances. Excessive integral gain results in velocity overshoot and could cause instability.
Value	Type the positive number that represents the proportion of the gain Note: You can use the Expression Builder to enter this value. (Select **Edit** from the menu bar.)

3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

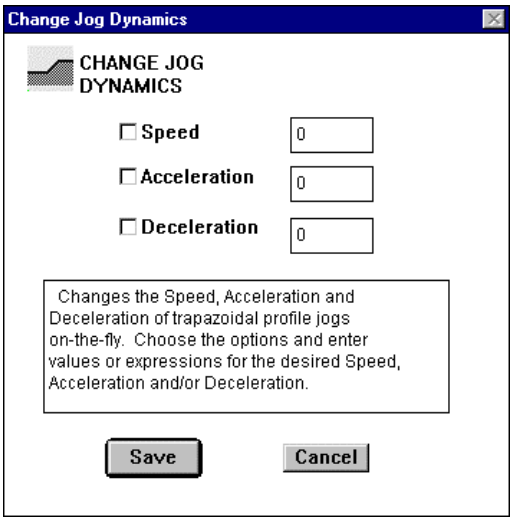
Change Jog Dynamics

The Change Jog Dynamics block allows you to change—on-the-fly—the speed, acceleration, and deceleration of trapezoidal profile jogs.

Once a jog has been stopped by a Change Dynamics to Zero speed, you can either Change Jog Dynamics to a nonzero speed to resume the jog in the same direction, or use a Jog Axis block to start a new jog.

To define the speed-related changes for your diagram:

1. Double-click on the Change Jog Dynamics block. A dialog box similar to the following appears:



2. Make entries in the following fields:

Field	Description
Speed	Type the speed at which the move or jog occurs.
Acceleration	Type the acceleration at which the move or jog occurs in axis position units per second squared.
Deceleration	Type the deceleration at which the move or jog occurs in axis position units per second squared. Note: If this value is changed without changing the speed, the new value is used for any subsequent jog motion.

3. Select **Save**. The diagram appears with the block checked to indicate that the parameters are set.

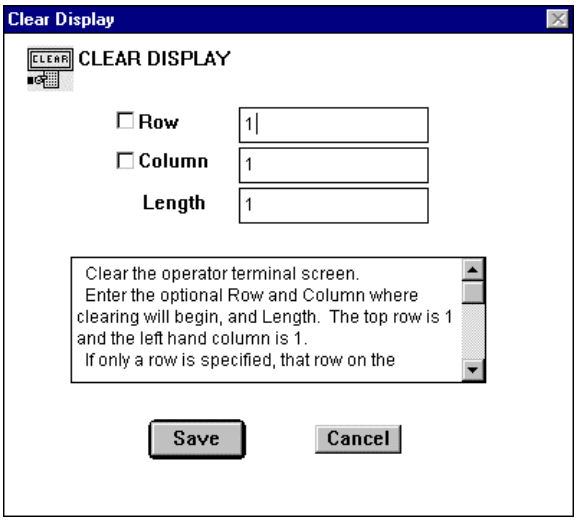
Clear Display

The Clear Display block allows you to clear the operator terminal screen.

To clear the entire screen, don't make entries in this dialog box.

To clear specific parts of the terminal display screen:

1. Double-click on the Clear Display block. A dialog box similar to the following appears:



2. Make entries in the following fields:

Field	Description
Row	To clear an entire row on the operator terminal screen, type the number of that row. To clear an area within a row: 1. Type the number of the row at which you want to begin clearing. 2. Enter column and length values.
Column	Type the number of the column at which you want to begin clearing. To clear a number of characters beginning at a location, you also have to enter a length in the *Length* field.
Length	Type the number of characters to be cleared starting at the row and column specified.

3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

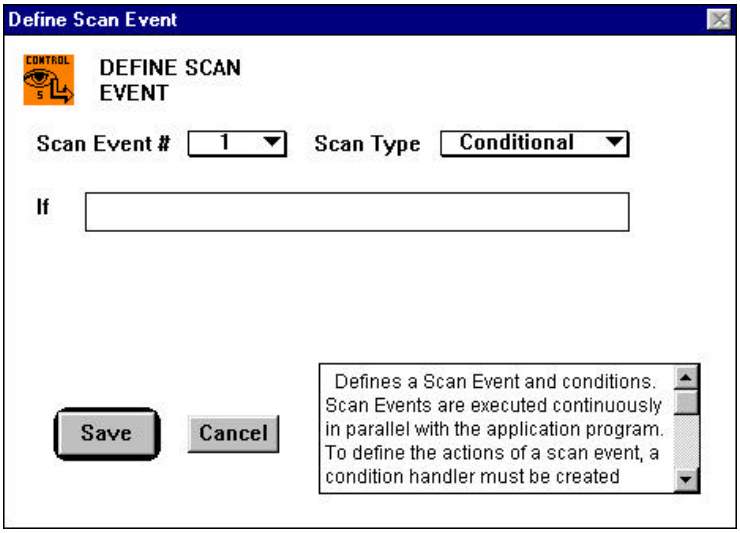
Define Scan Event

The Define Scan Event block allows you to define a scan event and conditions. Scan events are executed continuously in parallel with the application program. To define the actions of a scan event, you need to create a condition handler using the Scan Event Handler block. You must enable each scan event using the Set Condition block.

IMPORTANT: You must connect this block to the Start block or another Define Scan Event block of the Main Module. You cannot encapsulate the Define Event Scan block.

To define the scan event:

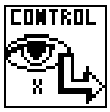
1. Double-click on the Define Scan Event block. A dialog box similar to the following appears:



2. Make entries in the following fields:

Field	Description
Scan Event #	Select a number from 1 to 8 to identify this scan event.
Scan Type	Select one of the following: Conditional — To test for a condition you specify. Timer — To use a timer to determine the elapsed time between two events, or delay an action.
If	If the *Scan Type* field is Conditional, type an expression or event.
Timer #	If the *Scan Type* field is Timer, select the number that you want to assign to this timer.
Timer Type	If the *Scan Type* field is Timer, select one of the following: Action After Timeout — To delay the action by the value of the time once the condition is met. Elapsed Time — To determine the elapsed time between two events.
Start Timer	If the *Scan Type* field is Timer, type the condition that starts the time.
Stop Timer Condition	If the *Timer Type* field is elapsed time, type the condition that stops the timer. Note: The present timer value is available in the TIMER_1 or TIMER_2 system variable.

3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.



Define Xkey

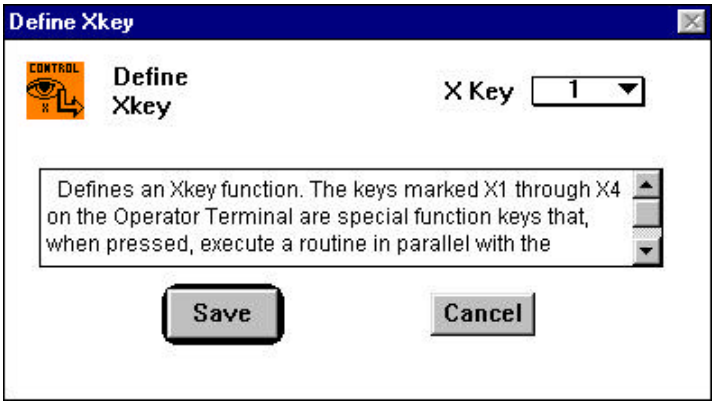
The Define Xkey block allows you to define the function of the keys marked X1 through X4 on the operator terminal. These are function keys that, when pressed, execute a routine in parallel with the application program. To define the actions performed when the key is pressed, you need to use the Xkey Handler block to create a condition handler. You must enable each Xkey using the Set Condition block.

IMPORTANT: You must connect this to the Start block, a Define Scan Event block, or another Define Xkey block of the Main Module.

IMPORTANT: You cannot encapsulate the Define Xkey block.

To define the Define Xkey block:

1. Double-click on the Define Xkey block. A dialog box similar to the following appears:



2. Make an entry in the following field:

Field	Description
Xkey	Select the number of the function key (1 to 4) that you want to define.

3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

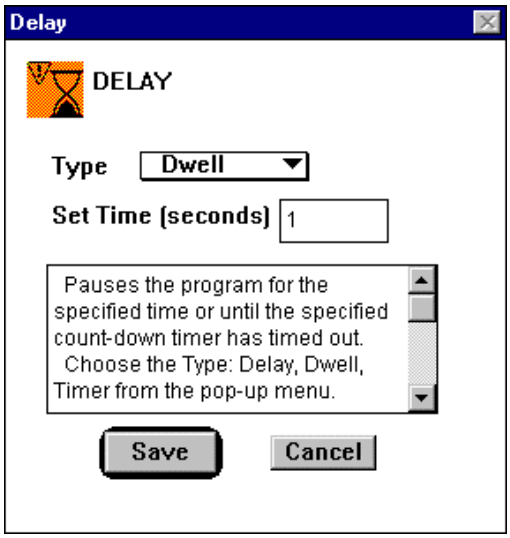
4. Create an Xkey routine using the Xkey Handler block. Refer to the *Xkey Handler* section for more information.

Delay

The Delay block allows you to pause the program for the specified time or until the count-down timer you specify times out.

To define the type of timeout you want:

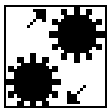
1. Double-click on the Delay block. A dialog box similar to the following appears:



2. Make entries in the following fields:

Field	Description
Type	Select one of the following: Dwell To pause the program for a specified time that is affected by the feedrate. Delay To pause the program for a specified time that is not affected by the feedrate. Timer To wait for a timer to expire.
Set Time (seconds)	If you selected Dwell or Delay in the *Type* field, type the amount of delay in seconds.
Timer	If you selected Timer in the *Type* field, type the number of the timer (1 or 2).

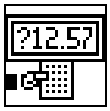
3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.



Disable Gearing

The Disable Gearing block allows you to stop the electronic gearing motion axis immediately. If gearing is the only motion in progress on the axis, the axis stops.

The Disable Gearing block does not require you to enter any information.

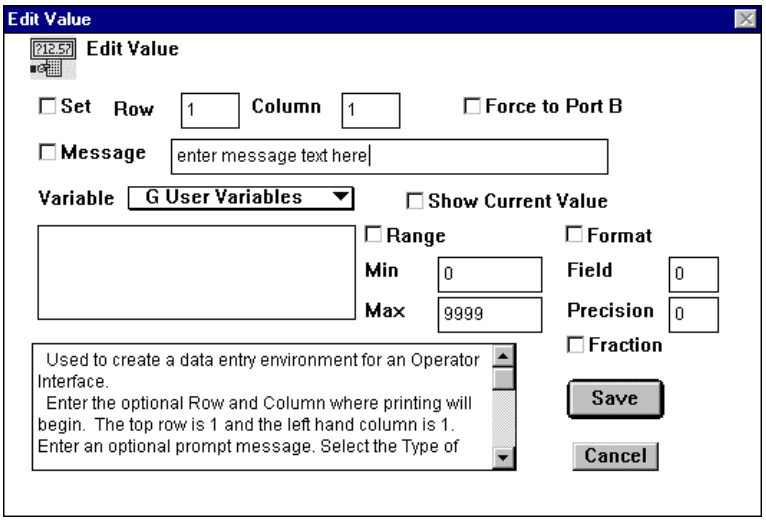


Edit Value

The Edit Value block allows you to create a data entry environment for an operator interface. This block allows you to enter a value on the operator interface stored in a user variable.

To define the Edit/Value block:

1. Double-click on the Edit Value block. A dialog box similar to the following appears:



2. Make entries in the following fields:

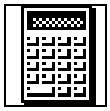
Field	Description
Set	Select this check box to define row/column.
Row	If you selected the *Set* check box, type the number of the row in which the display begins.

Field	Description
Column	If you selected the *Set* check box, type the number of the column in which the display begins.
Force to Port	Select the serial port that you want to use. The default is Port A, the operator terminal port.
Message	Select this check box to enter a prompt message. In the adjacent box, type the prompt message.
Variable	Select the type of variable that you want to read.
Show Current Value	Select this field if you want the current value of the variable to appear. This allows the operator to see the current value and simply press **Enter** to keep it.
Range	Select this check box if you want to specify a minimum/maximum range. This option is only available if you selected G or V User Variables.
Min	If you selected the *Range* check box, specify the minimum value the operator is allowed to enter. This option is only available if you selected G or V User Variables.
Max	If you selected the *Range* check box, specify the minimum value the operator is allowed to enter. This option is only available if you selected G or V User Variables.
Format	Select this check box if you want to specify the field and precision of the displayed variable or if you want to display it as a fraction. This option is only available if you selected G or V User Variables.
Field	If you selected the *Format* check box, type the number of characters that you want to allow for the variable, including the decimal point. This option is only available if you selected G or V User Variables.
Precision	If you selected the *Format* check box, type the number of places that you want to display after the decimal point. This option is only available if you selected G or V User Variables.

Field	Description
Fraction	If you selected the *Format* check box, select this field if you want the variable to appear as a fraction. This option is only available if you selected G or V User Variables.

3. Select the variable that you want to display from the scrolling list.

4. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

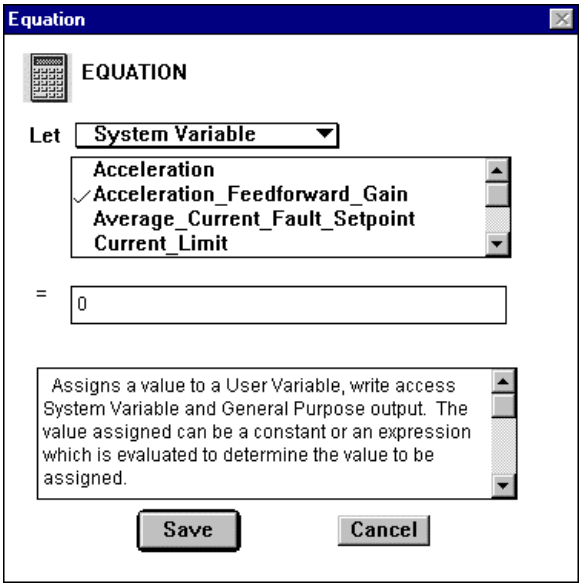


Equation

The Equation block allows you to assign a value to a user variable, a system variable, or a general purpose output. You can assign a constant or an expression to determine the value that is to be assigned.

To define a value:

1. Double-click on the Equation block. A dialog box similar to the following appears:

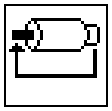


2. Make entries in the following fields:

Field	Description
Let	Select the type of variable that you want to use in the equation. A list of those variables appears in the window.
=	Type the value or expression that defines the value for the variable.

3. Select the variable in the window.

4. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.



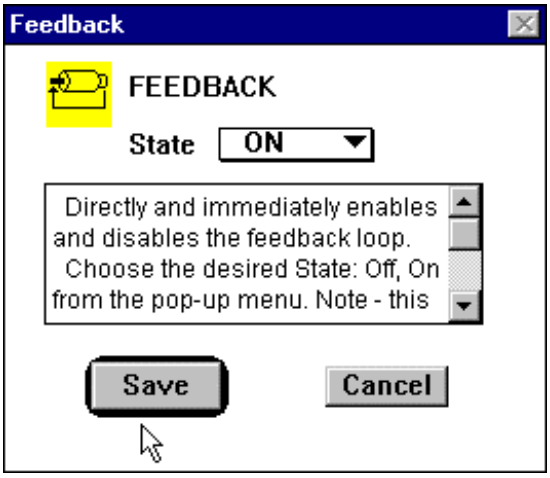
Feedback

The Feedback block allows you to directly and immediately enable or disable the feedback loop on the servo axis.

You can use the Feedback block anywhere in a GML Ultra diagram. However, because it causes an abrupt, uncontrolled stop, you should not use it while the axis is moving.

To set feedback condition state:

1. Double-click on the Equation block. A dialog box similar to the following appears:



2. Make an entry in the following field:

Field	Description
State	Select one of the following: ON To immediately enable the feedback loop on the servo axis. OFF To immediately disable the feedback loop on the servo axis.

3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

To use feedback when you must move the axis by hand:

1. Select the appropriate axis.
2. In the Feedback *State* field, select OFF. The position continues to be tracked even with feedback OFF.
3. Move the axis by hand as needed.
4. Select **ON**. When feedback is turned on again, the axis is again under closed-loop control, but at the new position.
5. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

Fkey

This block allows you to simulate pressing an Fkey from within the program.

The operator terminal has four function keys (Fkeys F1 through F4) you can program to perform different functions. Once these key are programmed, you can assign text to them so their functions appear on the operator terminal screen. Enable these displays by connecting the operator terminal to serial port A (P7).

There are two places to set the Fkey mode displayed on the operator terminal:

- From a program.
- By using the mode key.

To set the Fkey mode from a program:

Use a Print to Display block with a “^Cn” in the message ('n' corresponds to the Fkey mode number.) The number 'n' is zero-based—that is, 0 corresponds to Fkey Mode #1, 1 corresponds to Fkey Mode #2, and so on.

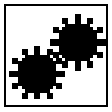
To set an Fkey mode using the MODE key:

1. Press the MODE key to step through as many modes as are defined in the FKey Set Up dialog box. If Fkey functions are not assigned to a mode, the operator terminal pass over that mode and moves to the next mode with defined Fkeys.

2. Press the MODE key. The Fkey labels appear on the screen.

To clear the labels from the terminal screen:

Press either the CLEAR key or the STATUS key, which displays one of the status displays.



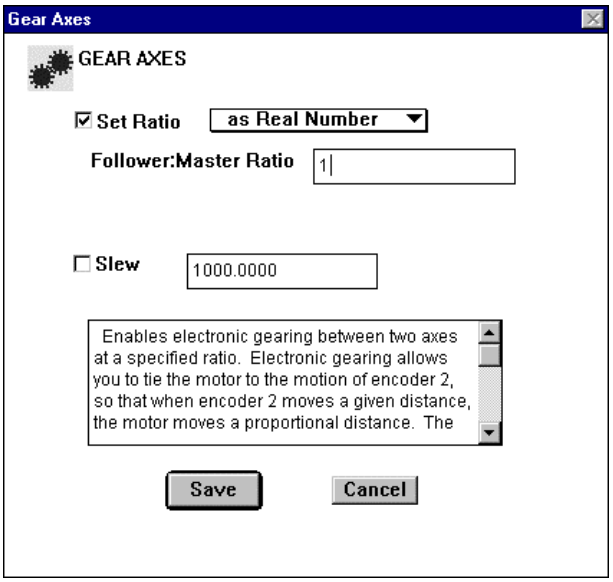
Gear Axes

The Gear Axes blocks allows you to enable electronic gearing between the servo and master axes at a specified ratio.

Electronic gearing remains active through any subsequent Jog Axis or Move Axis. This allows electronic gearing motions to be superimposed on the jog or move motions to create complex motions and synchronization.

To define the gear ratio:

Double-click on the Gear Axes block. A dialog box similar to the following appears:

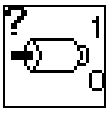


1. Make entries in the following fields:

Field	Description
Set Ratio	Select one of the following: as a Real Number — To allow the gear ratio to be specified as a real number or expression representing the ratio of follower axis counts to master axis counts. as a Fraction — To allow the gear ratio to be specified as a pair of integer numbers or expressions representing the ratio between the number of follower axis feedback counts and the number of master axis feedback counts. Note: A negative gear ratio causes the motor to move in the direction opposite that of encoder 2.

Field	Description
Follower: Master Ratio	If your ratio is a real number, type the real number or expression that represents the ratio of follower axis counts to master counts.
Follower Counts Master Counts	If your ratio is a fraction, type the pair of integer numbers or expressions that represent the ratio between the number of follower axis feedback counts and the number of master axis feedback counts.
Slew	Select the *Slew* check box and type the value that defines the maximum acceleration of the follower axis.

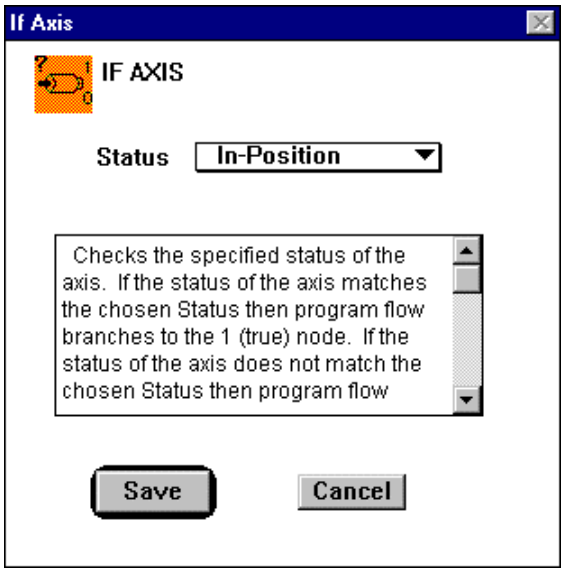
2. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.



If Axis

The If Axis block allows you to check the status of an axis. If the status of the axis matches the chosen status, program flow branches to the 1 (true) node. If the status of the axis does not match the chosen status, program flow continues to the 0 (false) node.

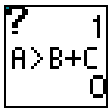
1. Double-click on the If Axis block. A dialog box similar to the following appears:



2. Make an entry in the following field:

Field	Description	
Status	Select one of the following:	
	In Position	To check for an *in position* status. When the axis locks onto its command position, the program branches to the 1 (true) node (In_Position_Flag=ON).
	Jogging Done	To check for a *jogging done* status. When the axis is no longer being commanded to jog, it branches to the 1 (true) node (Jog_Active_Flag= OFF).
	Moving Done	To check for a *moving done* status. When the axis is no longer being commanded to move, the program branches to the 1 (true) node (Move_Complete_Flag=ON).
	Homing Done	To check for a *homing done* status. When the axis has been homed, the program branches to the 1 (true) node (Home_Sequence_Complete_Flag=ON).
	Gearing Done	To check for a *gearing done* status. When the axis is no longer being commanded to move from the electronic gearing function, the program branches to the 1 (true) node Gearing_Active_Flag=OFF).

3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

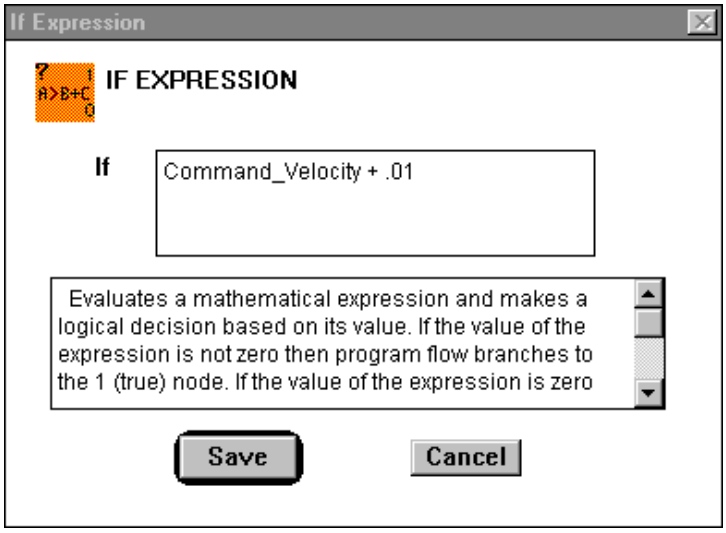


If Expression

The If Expression block allows you to evaluate a mathematical expression and make a logical decision based on its value. If the value of the expression is true, program flow branches to the 1 (true) node. If the value of the expression is false, program flow continues to the 0 (false) node.

To use the If Expression block to make a decision based on an expression:

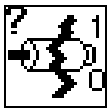
1. Double-click on the If Expression block. A dialog box similar to the following appears:



2. Make an entry in the following field:

Field	Description
If	Type or use the Expression Builder to create a logical expression.

3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

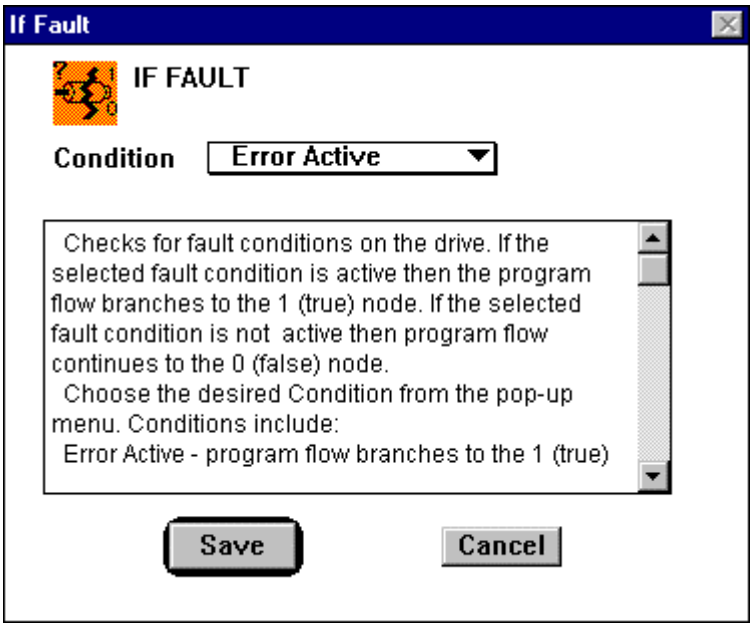


If Fault

The If Fault block allows you to check for fault conditions on the drive. If the selected fault condition is active, the program flow branches to the 1 (true) node. If the selected fault condition is not active, the program flow continues to the 0 (false) node.

To check for a fault condition on the drive:

1. Double-click on the If Fault block. A dialog box similar to the following appears:



2. Make an entry in the following field:

Field	Description	
Condition	Select one of the following:	
	Error Active	To detect a condition where program flow branches to the 1 (true) node if an error has occurred.
	Warning Active	To detect a condition where program flow branches to the 1 (true) node if a warning has occurred.

3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

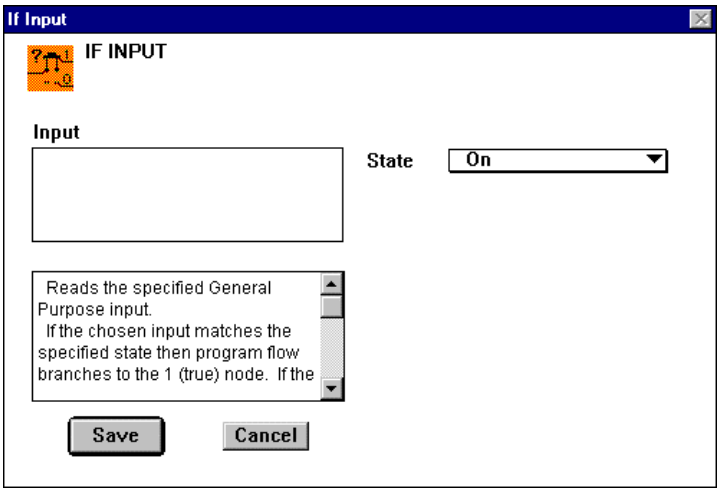
If Input

The If Input block allows you to read the specified general purpose input.

If the input you selected matches the specified state, program flow branches to the 1 (true) node. If the input you selected does not match the specified state program flow continues to the 0 (false) node.

To read a general purpose input:

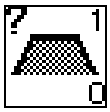
1. Double-click on the If Input block. A dialog box similar to the following appears:



2. Make an entry in the following field:

Field	Description
Input	Select the input that you want to read.
State	Select one of the following: On — To have the program flow branch to the 1 (true) node if the input is ON. Off — To have the program flow branch to the 0 (false) node if the input is OFF. Flag — To have the program flow branch to the 1 (true) node if the state of the input matches the state of the specified flag variable.

3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.



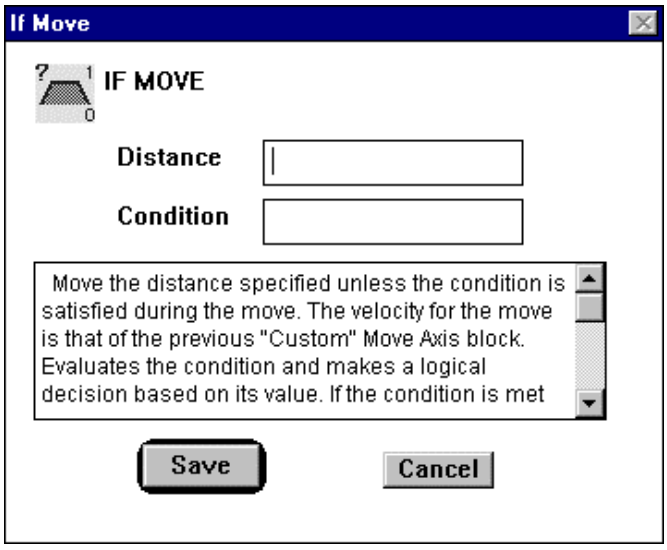
If Move

The If Move block allows you to move a specified distance unless the condition is satisfied during the move. The velocity for the move is that of the previous Segment Move Axis block. This block evaluates the condition and makes a logical decision based on its value. If the condition is met, then program flow branches to the 1 (true) node. If the condition is not met, then program flow continues to the 0 (False) node.

IMPORTANT: This block must follow a Segment Move Axis block. Also, the final block of a complex If Move profile must be a Segment Move Axis block.

To define a move condition:

1. Double-click on the If Move block. A dialog box similar to the following appears:



2. Make entries in the following fields:

Field	Description
Distance	Type a value or expression that represents the maximum distance of the move.
Condition	Type a value or expression that represents the condition that must be met to exit the move. • If the condition is met at any time during the move, program flow branches to the 1 (true) node. • If the condition is not met, program flow pauses on this block until the specified distance is reached, then branches to the 0 (false) node.

3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

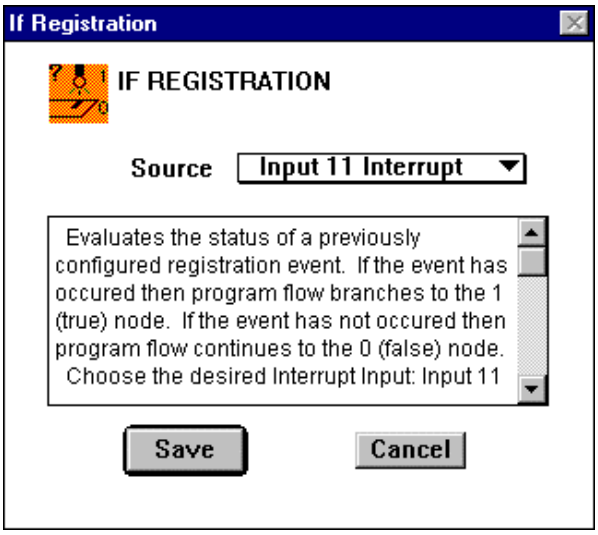
If Registration

The If Registration block allows you to evaluate the previously configured registration event. If the event occurs, program flow branches to the 1 (true) node. If the event has not occurred, program flow continues to the 0 (false) node.

You need to have set up a registration for the specified axis previously in the diagram using a Registration block.

To evaluate a registration event:

1. Double-click on the If Registration block. A dialog box similar to the following appears:



2. Make an entry in the following field:

Field	Description
Source	Select one of the following: Input 11 Interrupt — To select hardware Interrupt 1. Input 12 Interrupt — To select hardware Interrupt 2.

If the event:	Program flow branches to the:
Does occur	1 (true) node.
Does not occur	0 (false) node.

3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

If Timeout

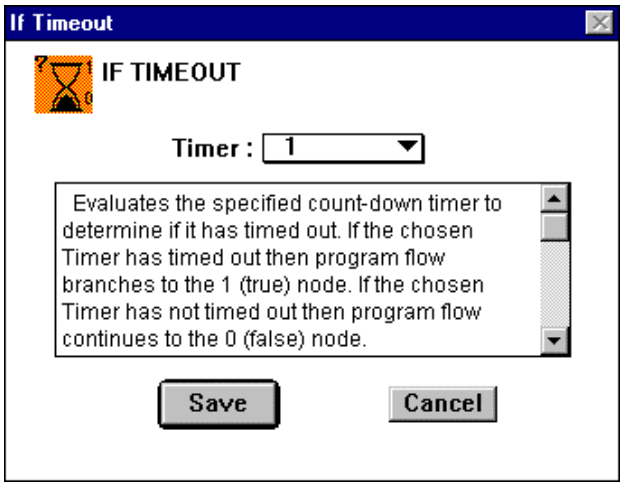
The If Timeout block allows you to evaluate the count-down timer you specify to determine if it has timed out. If the timer has timed out, program flow branches to the 1 (true) node. If the chosen timer has not timed out, program flow continues to the 0 (false) node.

The following three events must be complete before you insert the If Timeout block:

- A Define Scan Event block was used to define the use of the timer.
- A Set Timer block was used to set the value of the timer.
- A Set Condition block was used to activate the Scan event.

To use a specified preset timer to decide program flow:

1. Double-click on the If Timeout block. A dialog box similar to the following appears:



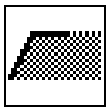
2. Make an entry in the following field:

Field	Description
Timer	Select the preset timer that you want to use: 1 or 2.

If the chosen timer has:	Then program flow branches to the:
Timed out	1 (true) node.
Has not timed out	0 (false) node.

3. Make sure you have set the parameters in the specified Set Timer, Define Scan Event, and Set Condition blocks.

4. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

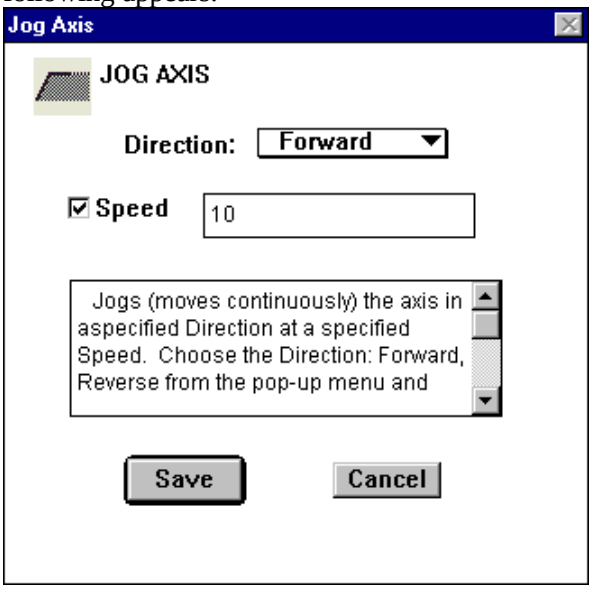


Jog Axis

The Jog Axis block allows you to jog (move continuously) the servo axis a specific direction at a specified speed. You can set the jog and location rates by using a Change Jog Dynamics block or in Jogging under Axis Configuration.

To define a jog:

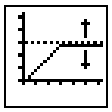
1. Double-click on the Jog Axis block. A dialog box similar to the following appears:



2. Make entries in the following fields:

Field	Description
Direction	Select one of the following: Forward — To jog the motor forward. Reverse — To jog the motor reverse.
Speed	Type the speed at which the motor jogs.

3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.



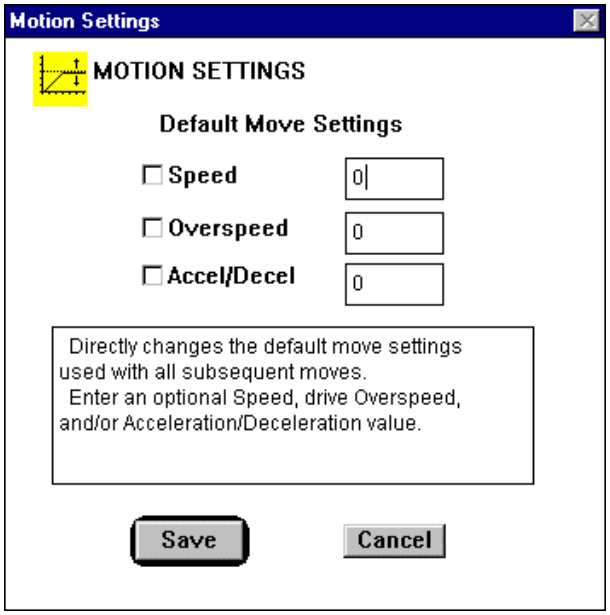
Motion Settings

The Motion Settings block allows you to change the working profile values used with all subsequent moves.

Note: This block does not change the power-up profiles that are defined in setup. It only changes the current working profile and subsequent motion for this axis.

To change the value of all moves following this block:

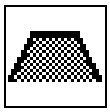
1. Double-click on the Motion Settings block. A dialog box similar to the following appears:



2. Make entries in the following fields:

Field	Description
Speed	Type the speed for the move or jog. This changes the working velocity value used with subsequent Move blocks.
Overspeed	Type the overspeed fault set point.
Accel/Decel	Type the acceleration and deceleration value.

3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

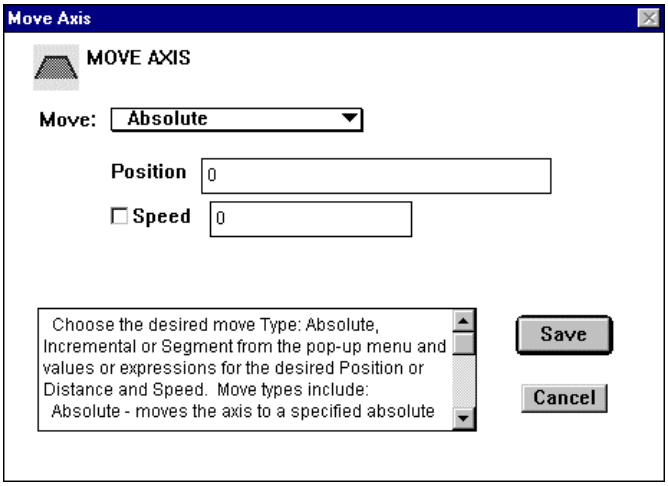


Move Axis

The Move Axis block allows you to move the servo axis to a specified absolute position or to a specified incremental distance at a specified speed. The Move Axis block can also generate custom move segments.

To define a move:

1. Double-click on the Move Axis block. A dialog box similar to the following appears:

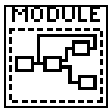


2. Make entries in the following fields:

Field	Description
Move:	Select one of the following: Absolute — To move the axis to an absolute position. Incremental — To move the axis by an incremental distance. Segment — To move the axis by an incremental distance representing a portion of a complex profile.
Position	If you selected Absolute in the *Move* field, type the value or expression that defines the specific position.
Distance	If you selected Incremental or Segment in the *Move* field, type the value or expression that defines the incremental distance.
Speed	Type a value or expression that represents the speed of the move.

Field	Description
Override Profiles	If you selected Segment in the *Move* field, select one of the following: S-Curve — S-curve acceleration and deceleration (controlled jerk). Trapezoidal — Linear acceleration and deceleration.

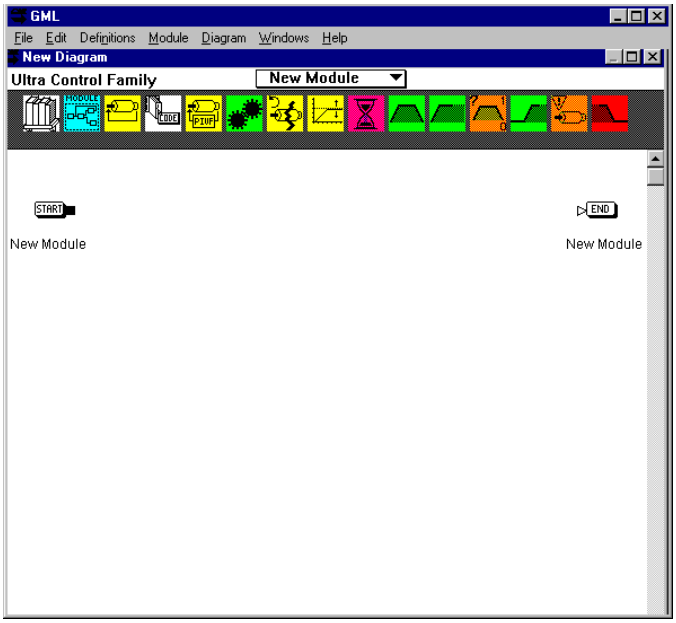
3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

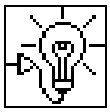


New Module

The New Module block allows you to place a new blank module on the Diagram Editor window.

Double-click on the New Module block. A dialog box similar to the following appears:



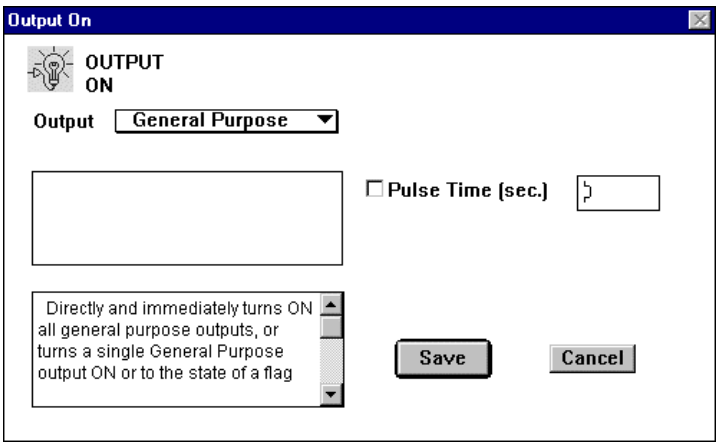


Output On

The Output On block allows you to turn ON a single output or to set a group of outputs to the binary state of an expression.

To define a single output as ON:

1. Double-click on the Output On block. A dialog box similar to the following appears:



2. Make entries in the following fields:

Field	Description
Output	Select one of the following: General Purpose — To specify a general purpose output. Binary — To specify a group of outputs to set to the binary state of an expression. All — To set all general purpose outputs ON.
Start Bit	If the *Output* field is Binary, type a value from 1 to 8 that represents the least significant bit of the group of binary outputs.
Set To	If the *Output* field is Binary, type a value or expression that represents the binary value.
Number of Bits	If the *Output* field is Binary, type the number of outputs included in this group.
Pulse Time (sec.)	If the *Output* field is General Purpose, use this field to set an optional pulse time for the output. Program flow continues to the next block and the output turns OFF after the specified time.

3. Select the output that you want to turn on if set to General Purpose.

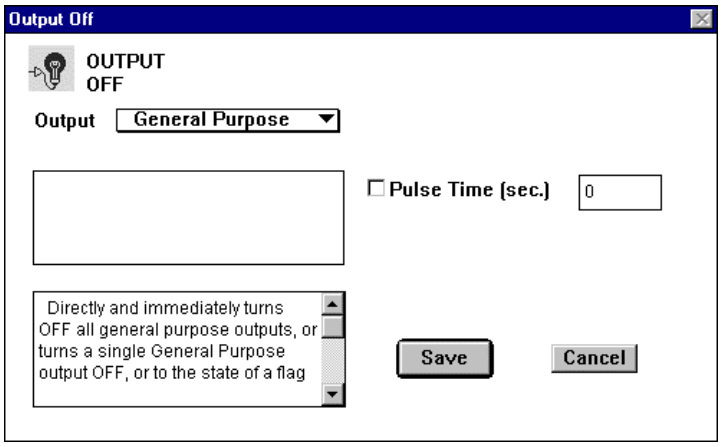
4. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

Output Off

The Output Off block allows you to immediately turn a single output OFF or to the state of the flag variable, or set a group of outputs to the binary state of a variable.

To define a single output as OFF:

1. Double-click on the Output Off block. A dialog box similar to the following appears:



2. Make entries in the following fields:

Field	Description
Output	Select one of the following: General Purpose — To specify a general purpose output. Binary — To specify a group of outputs to set to he binary state of an expression. All — To set all general purpose outputs OFF.
Start Bit	If the *Output* field is Binary, type the output number from 1 to 8 that represents the least significant bit of the group of binary outputs.

Field	Description
Set To	If the *Output* field is Binary, type a value or expression that represents the binary value.
Number of Bits	If the *Output* field is Binary, type the number of outputs included in this group.
Pulse Time (sec.)	If the *Output* field is General Purpose, use this field to set an optional pulse time for the output. Program flow continues to the next block and the output turns ON after the specified time.

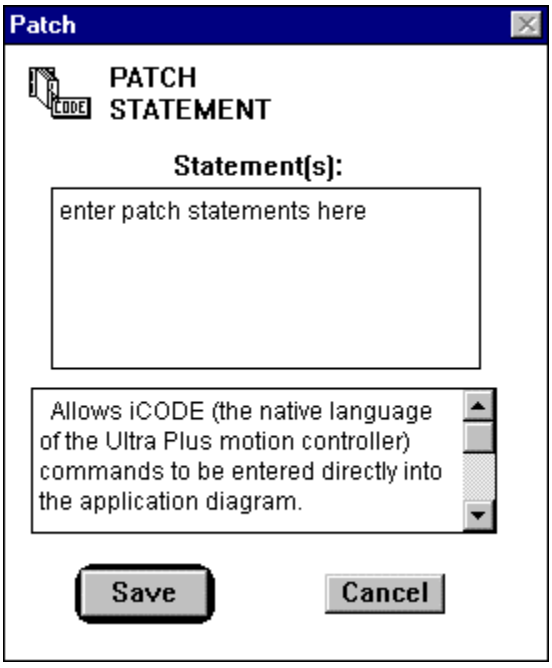
3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

Patch Statement

The Patch Statement block allows you to enter iCODE (the native language of the ULTRA Plus motion controllers) commands directly into the application diagram.

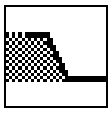
IMPORTANT: Use this block only if you are experienced with iCODE.

1. Double-click on the Patch block. A dialog box similar to the following appears:



2. Type the iCODE commands in the scrolling Statement(s) window.

3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.



Pause Move

The Pause Move block allows you to ramp motion to a stop and pause. You can also set the Pause input to perform this function in setup.

The Pause block does not require you to enter information.

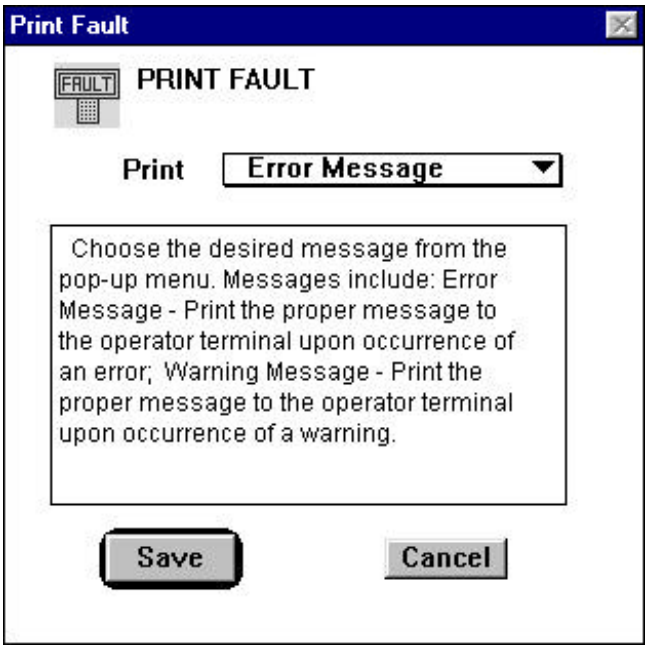
Print Fault

The Print Fault block allows you to print messages on the operator terminal.

Note: This block is usually used within the Error program.

To select the type of message that displays on the operator terminal when a fault occurs:

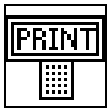
1. Double-click on the Print Fault block. A dialog box similar to the following appears:



2. Make an entry in the following field:

Field	Description
Print	Select one of the following: Error Message — To display an error message when a fault occurs. Warning Message — To display a warning message when a fault occurs.

3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

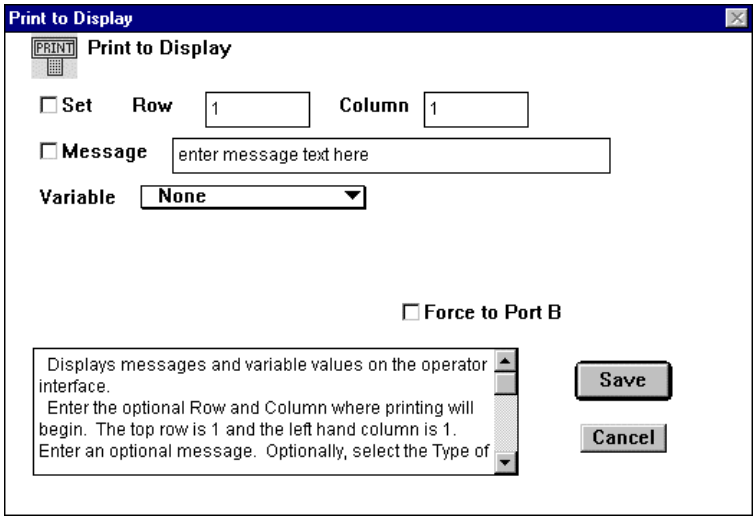


Print to Display

The Print to Display block allows you to display messages and variable values on the operator interface.

To define a message and position it on the operator interface:

1. Double-click on the Print to Display block. A dialog box similar to the following appears:

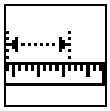


2. Make entries in the following fields:

Field	Description
Set Row	Type the row at which the message begins.
Set Column	Type the column at which the message begins.
Message	Type the message that you want to appear.
Variable	Select the type of variable that you want to display.
Force to Port B	Select the Force to Port B check box to print to Port B. The default is Port A, the operator terminal port.

3. Select the variable you want to display from the scrolling list.

4. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

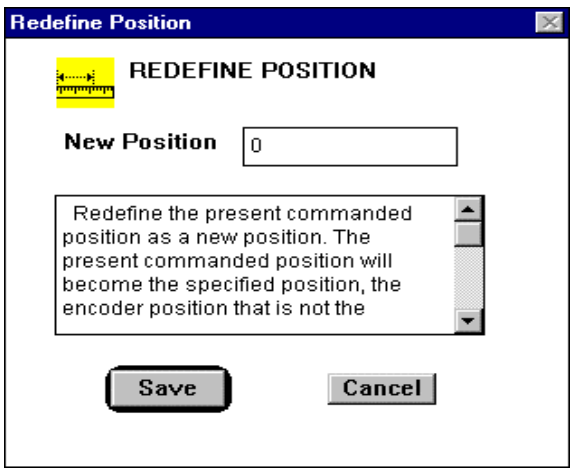


Redefine Position

The Redefine Position block allows you to set the actual or command position of the axis to a specific absolute or relative position. You can use the Redefine Position block when the axis is moving or at rest. This block does not cause motion; it simply redefines the current axis position. In addition, the master axis position is set to zero.

To redefine the position of the axis:

1. Double-click on the Redefine Position block. A dialog box similar to the following appears:



2. Make an entry in the following field:

Field	Description
New Position	Type the value or expression that defines the new command position of the axis.

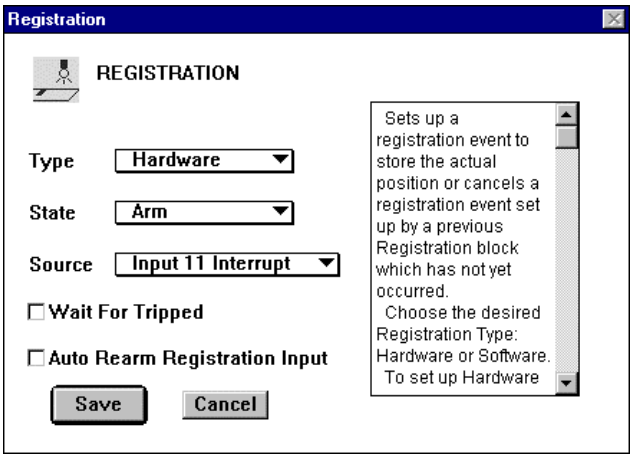
3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

Registration

The Registration block allows you to set up a registration event to store the actual positions of the servo and master axes. Also use this block to cancel a registration event set up by a Registration block that has not been executed.

To set up a registration event:

1. Double-click on the Registration block. A dialog box similar to the following appears:

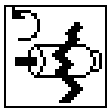


2. Make entries in the following fields:

Field	Description	
Type	Select one of the following:	
	Hardware	To set up hardware registration.
	Software	To activate a software simulation at an Input 12 interrupt.

Field	Description	
State	Select one of the following:	
	Arm	To monitor the specified axis until a transition of the registration source you specified in the *Source* field occurs.
	Disarm	Disengage a previously set registration event.
Source	Select one of the following:	
	Input 11 Interrupt	To select Hardware Interrupt 1.
	Input 12 Interrupt	To select Hardware Interrupt 2.
Wait For Tripped	Select this to pause the program until the registration event occurs.	
Auto Rearm Registration Input	Select this to activate the registration event continuously.	

3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

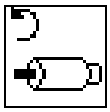


Reset Axis Fault

The Reset Axis Fault block allows you to directly reset the ULTRA Plus controller.

It resets or clears the specified fault status on the specified axis by setting the value of the appropriate fault variable to zero.

The Reset Axis Fault block does not require you to enter information.

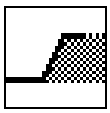


Reset Drive

The Reset Drive block allows you to perform a hardware reset.

Note: This block causes the axis to lose home complete status.

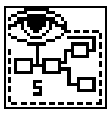
This block does not require you to enter information.



Resume Move

The Resume Move block allows you to resume motion previously halted by the Pause Move block. You can also set a Resume Move by clearing the Pause input.

The Resume Move block does not require you to enter information.



Scan Event Handler

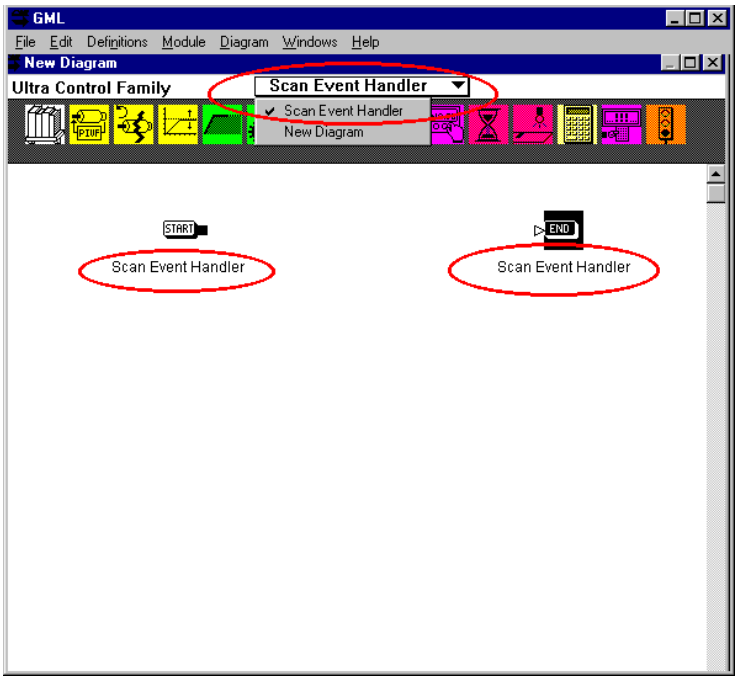
The Scan Event Handler block allows you to place a new blank module on the diagram. Scan event handlers monitor the condition of an event. When the scan event is enabled, if the event is scanned and the condition is true, the actions are performed in parallel with program execution. Scanned events can be enabled and disabled within the application.

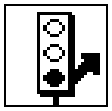
IMPORTANT: You need to connect the input node from node 2 (the lower output node) of a Define Scan Event block.

IMPORTANT: You cannot encapsulate this block.

IMPORTANT: You cannot connect this block back into the diagram.

Only blocks relevant to this module are available in the block library.



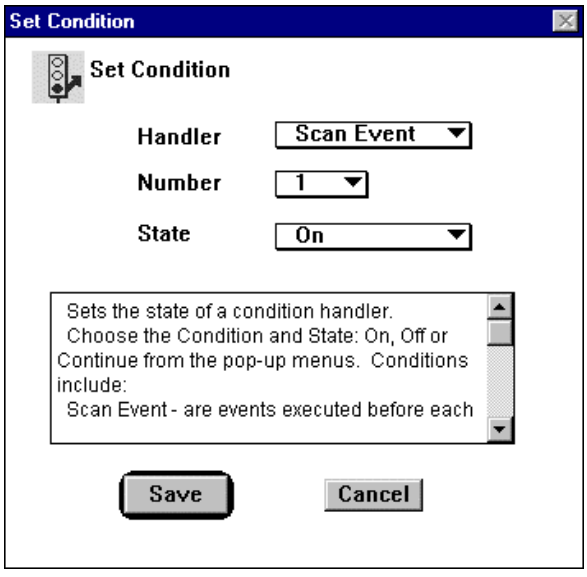


Set Condition

The Set Condition block allows you to set the state of a condition handler.

To set the state of a condition handler:

1. Double-click on the Set Condition block. A dialog box similar to the following appears:



2. Make entries in the following fields:

Field	Description
Handler	Select one of the following: Scan Event — To activate a scan event. Note: You need to use a Define Scan Event Handler block to create a condition handler for a Scan Event routine. Xkey — To activate an Xkey routine. Note: You need to use a Define Xkey block and an Xkey Handler block to create a condition handler for a Xkey routine.
Number	Select a number between 1 and 8 (1 and 4 for Xkey) to define the handler that you selected in the *Handler* field.
State	Select one of the following: On — To activate the condition handler for a single event. Off — To deactivate the condition handler. Continue — To allow for continuous monitoring of the condition handler.

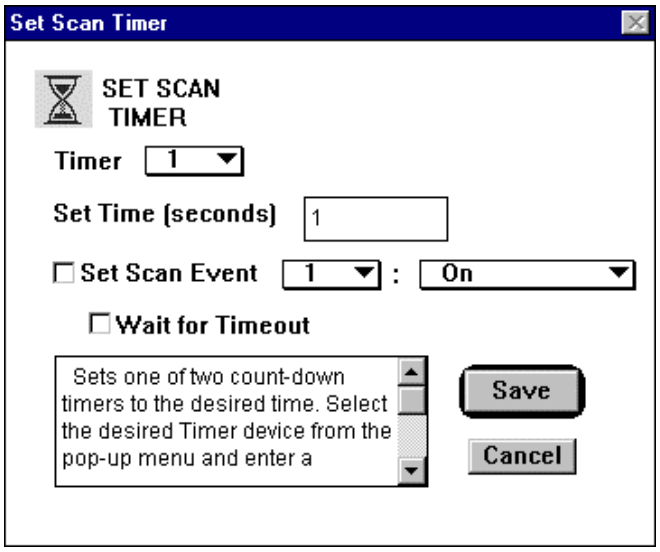
3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

Set Scan Timer

The Set Scan Timer block allows you to set the value of one of two count-down timers to the time.

To set a Scan Event timer:

1. Double-click on the Set Scan Timer block. A dialog box similar to the following appears:

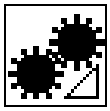


2. Make entries in the following fields:

Field	Description
Timer	Select the number that represents the timer device that you use (1 or 2).
Set Time (seconds)	Type the value or expression in seconds to indicate the amount of time before the timer expires.
Set Scan Event	Type the number that represents the event that is scanned for.
:	Select one of the following: On — To activate the scan event for a single event. Continuous — To allow for continuous monitoring of the scan event handler.

Field	Description
Wait for Timeout	Select this field to pause the program flow until the timer you selected in the *Timer* field times out.

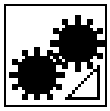
3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.



Slew Disable

The Slew Disable (red) block allows you to disable the rate of change (accel/decel) limit for the electronic gearing function.

The Slew Disable block does not require you to enter information.



Slew Enable

The Slew Enable (green) block allows you to set and enable the rate of change (acceleration/deceleration) limit for the electronic gearing function. Slew is the maximum acceleration permitted as a result of a signal received from the master axis.

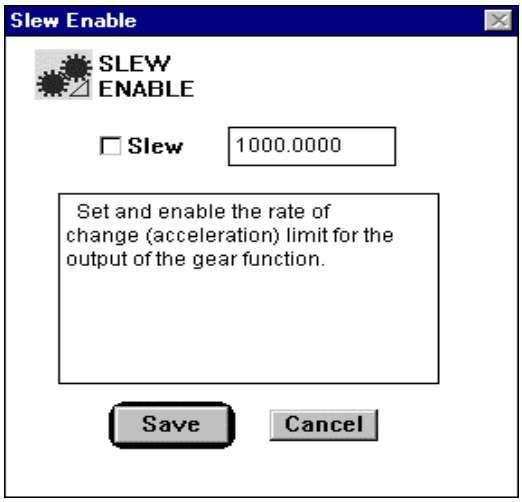
The slew limit is frequently used when the follower starts following a master that is already moving. If the master is moving and a gear is enabled with no slew limit set, the follower uses maximum acceleration to come up to speed. This can cause excessive wear on most machines and potentially damage equipment.

If you set a slew limit, the follower uses this acceleration rate to match the speed of the master.

Note: The slew rate can be set in either the Slew Enable block or the Enable Gear block.

To enter a slew limit:

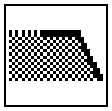
1. Double-click on the Slew Enable block. A dialog box similar to the following appears:



2. Make an entry in the following field:

Field	Description
Slew	Type the value in user units per second that represents the maximum acceleration to match speed with the master.

3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.



Stop Motion

The Stop Motion block allows you to disable motion in progress and bring the axis to a decelerated stop.

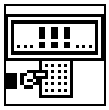
To enter a stop motion:

1. Double-click on the Stop Motion block. A dialog box similar to the following appears:

2. Make entries in the following fields:

Field	Description	
Mode	Select one of the following:	
	Stop All Motion	All motion caused by any previous Move Axis, Jog Axis, and Gear Axis blocks is stopped simultaneously, bringing the axis to rest.
	Stop All Gearing	Stop all Gear Axis blocks.
	Stop All Jog	Stop all Jog Axis blocks.
	Stop All Move	Stop all Move Axis blocks.
Decelerate Move	Select this check box to use the default acceleration to stop motion. If you do not select this check box, no deceleration is used.	

3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

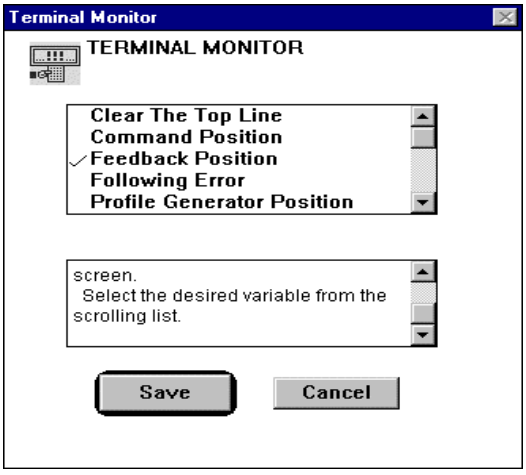


Terminal Monitor

The Terminal Monitor block allows you to display monitor variable values on the top line of the operator terminal screen.

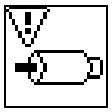
To select the variable that you want to appear:

1. Double-click on the Terminal Monitor block. A dialog box similar to the following appears:



2. Select the variable that you want to monitor on the operator terminal. A check mark appears in front of the variable.

3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

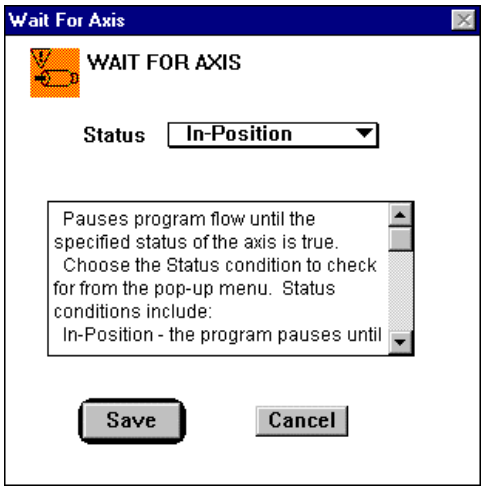


Wait For Axis

The Wait For Axis block allows you to pause program flow until the status that you specify for the axis is true.

To define the status condition you want the axis to wait for:

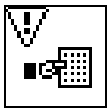
1. Double-click on the Wait For Axis block. A dialog box similar to the following appears:



2. Make an entry in the following fields:

Field	Description	
Status	Select one of the following:	
	In Position	To make the axis wait for a locked condition.
	Jogging Done	To make the axis wait for jogging to finish.
	Moving Done	To make the axis wait for moving to finish.
	Homing Done	To make the axis wait for homing to be complete.
	Gearing Done	To make the axis wait for gearing motion to finish.

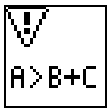
3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.



Wait For ENTER

The Wait For ENTER block allows you to pause program flow until you press ENTER on the operator terminal.

The Wait For Enter block does not require you to enter information.

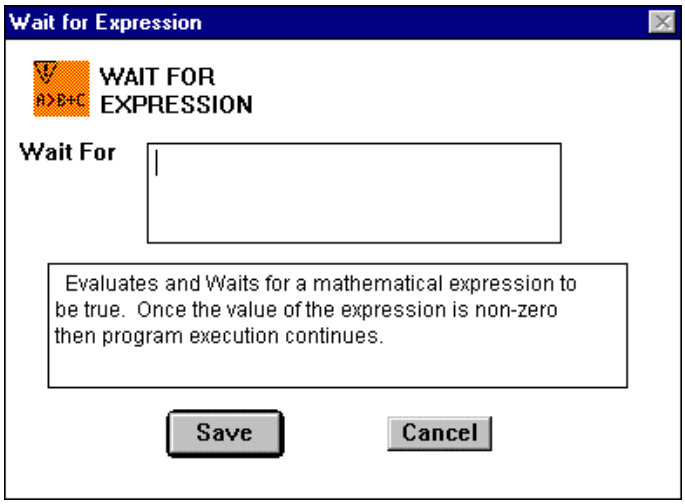


Wait For Expression

The Wait For Expression block allows you to evaluate and wait for a mathematical expression to be true. When the value of the expression is true, then program execution continues.

To pause a program until a value or expression is met:

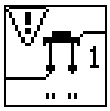
1. Double-click on the Wait For Expression block. A dialog box similar to the following appears:



2. Make an entry in the following field:

Field	Description
Wait For	Type or use the Expression Builder to create the mathematical expression, which when true, allows the program to continue.

3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

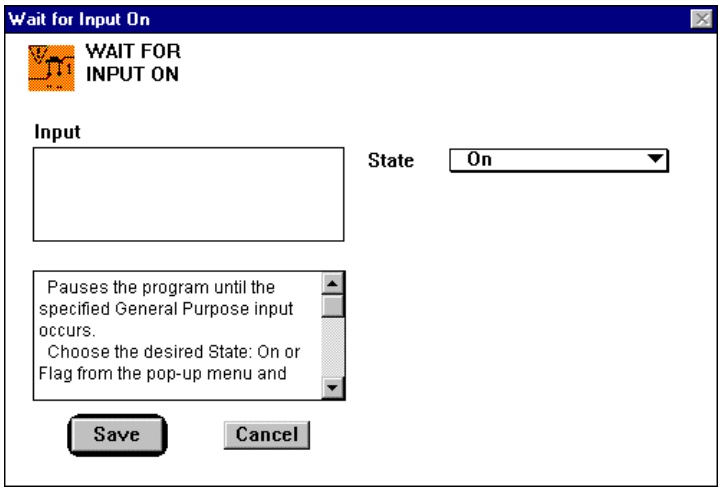


Wait For Input On

The Wait For Input On block allows you to pause program flow until a condition is met for a general purpose input that you specify.

To specify the condition that pauses your program:

1. Double-click on the Wait For Input On block. A dialog box similar to the following appears:



2. Make entries in the following fields:

Field	Description
Input	Select the input that you want to use to pause the program.
State	Select one of the following: On — To wait for the input to be ON. Flag — To wait for the state of the input to be equal to the specified flag variable.

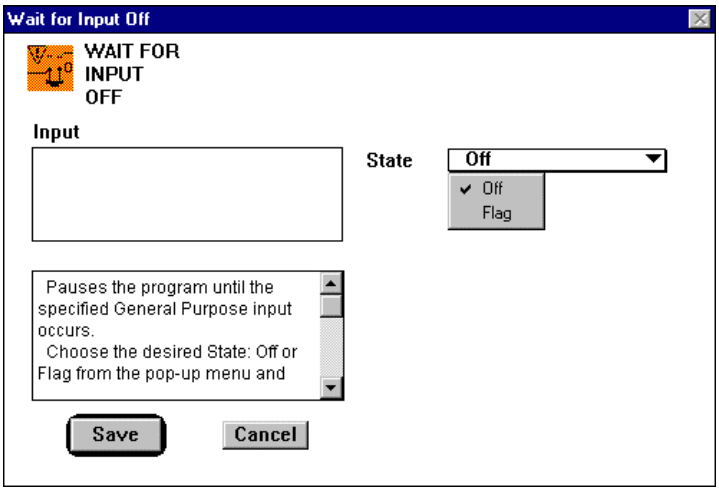
3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

Wait For Input Off

The Wait For Input Off block allows you to pause program flow until a condition is met for the general purpose input that you specify occurs.

To specify the condition that pauses your program:

1. Double-click on the Wait For Input Off block. A dialog box similar to the following appears:



2. Make entries in the following fields:

Field	Description
Input	Select the input that you want to use to pause the program.
State	Select one of the following: Off — To wait for the input to be OFF. Flag — To wait for the state of the input to equal the state of the specified flag variable.

3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

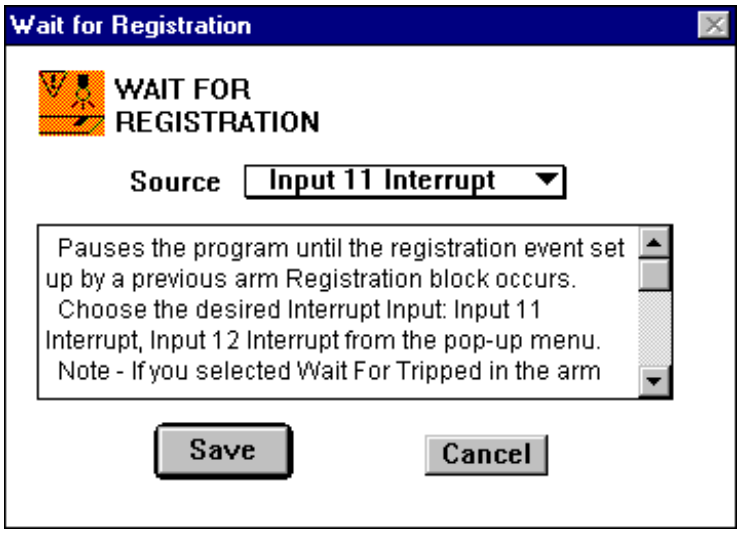
Wait For Registration

The Wait For Registration block allows you to pause the program until the registration event set up by a previous Registration block occurs.

IMPORTANT: If you selected Wait For Tripped in the Registration block, do not use a Wait For Registration block. If you do, the program waits for the same event twice.

To specify the condition that pauses your program:

1. Double-click on the Wait For Registration block. A dialog box similar to the following appears:



2. Make an entry in the following field:

Field	Description
Source	Select one of the following: Input 11 Interrupt — To wait for Interrupt 1 to occur. Input 12 Interrupt — To wait for Interrupt 2 to occur.

3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

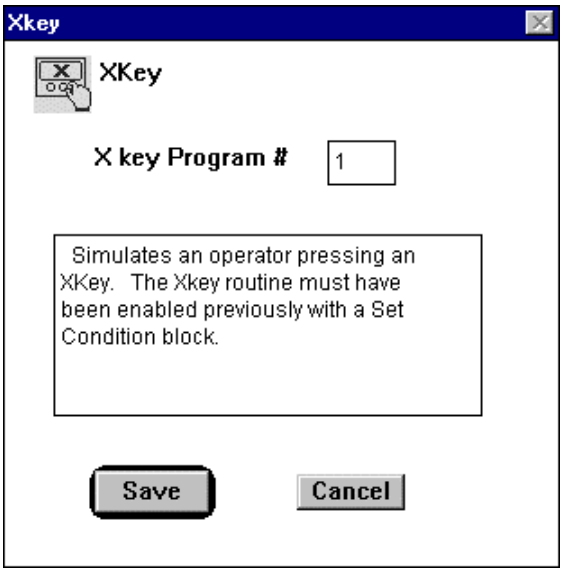
Xkey

The Xkey block allows you to simulate an operator pressing an Xkey. The keys marked X1 through X4 on the operator terminal are special function keys (Xkeys) that, when pressed, execute a routine in parallel with the application program. To enable an Xkey routine, you need to use the Xkey Handler block to create a condition.

IMPORTANT: Only one Xkey routine can be running at a time. If an Xkey routine is running when the Xkey block is executed, then the current Xkey routine ends and the new Xkey routine is started.

To simulate pressing an Xkey:

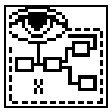
1. Double-click on the Xkey block. A dialog box similar to the following appears:



2. Make an entry in the following field:

Field	Description
X key Program #	Type the number (1-4) that represents the Xkey routine defined by a previous Define Xkey block and Xkey Handler, and activated by a previous Set Condition block.

3. Select **Save**. The dialog box closes. The diagram appears with the block checked to indicate that the parameters are set.

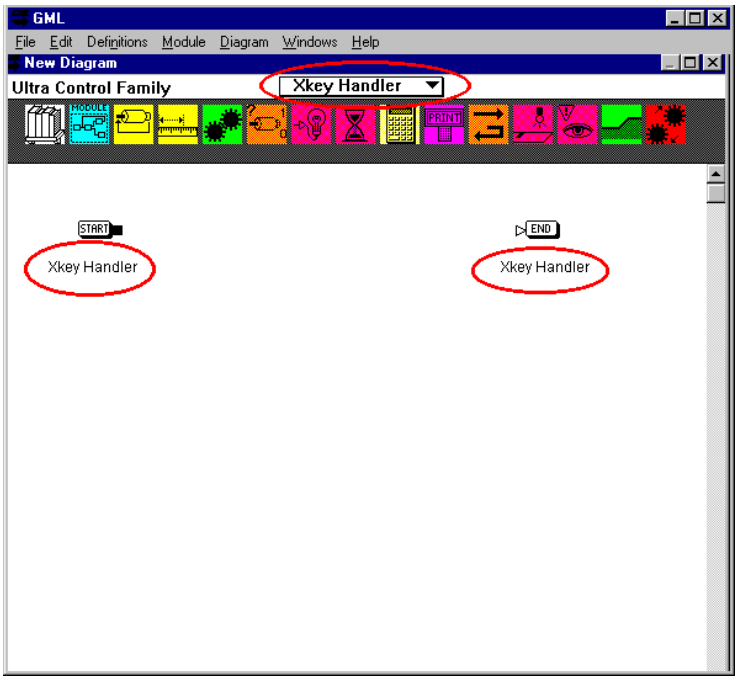


Xkey Handler

The Xkey Handler allows you to place a new blank module on the diagram. Xkey handlers perform their actions in parallel with program execution. The keys on the operator terminal marked X1 through X4 are special purpose keys (Xkeys) that, when pressed, cause the execution of an Xkey Handler.

Only blocks relevant to this module are available in the block library.

IMPORTANT: You must connect the input node from node 2 (the lower output node) of a Define Xkey block and you cannot encapsulate it. You cannot connect back to the diagram.



A

B

C

D

E

F

G

H

I

J

L

M

N

O

P

Q

R

S

T

U

V

W

X

Rockwell *Automation*

Allen-Bradley

Allen-Bradley, a Rockwell automation business, has been helping its customers improve productivity and quality for more than 90 years. We design, manufacture and support a broad range of automation products worldwide. They include logic processors, power and motion control devices, operator interfaces, sensors and a variety of software. Rockwell is one of the world's leading technology companies.

Worldwide representation.

Argentina • Australia • Austria • Bahrain • Belgium • Brazil • Bulgaria • Canada • Chile • China, PRC • Colombia • Costa Rica • Croatia • Cyprus • Czech Republic • Denmark Ecuador • Egypt • El Salvador • Finland • France • Germany • Greece • Guatemala • Honduras • Hong Kong • Hungary • Iceland • India • Indonesia • Ireland • Israel • Italy Jamaica • Japan • Jordan • Korea • Kuwait • Lebanon • Malaysia • Mexico • Netherlands • New Zealand • Norway • Pakistan • Peru • Philippines • Poland • Portugal Puerto Rico • Qatar • Romania • Russia–CIS • Saudi Arabia • Singapore • Slovakia • Slovenia • South Africa, Republic • Spain • Sweden • Switzerland • Taiwan • Thailand Turkey • United Arab Emirates • United Kingdom • United States • Uruguay • Venezuela • Yugoslavia

Allen-Bradley Headquarters, 1201 South Second Street, Milwaukee, WI 53204 USA, Tel: (1) 414 382-2000 Fax: (1) 414 382-4444

Publication 1398-5.11 – February 1997

P/N74102-245-01