



**ACR-MotionMax**  
**User Interface Manual**

**Version 1.0, April 2004**

# IMPORTANT

## User Information



### Warning!



ACR Series products are used to control electrical and mechanical components of motion control systems. You should test your motion system for safety under all potential conditions. Failure to do so can result in damage to equipment and/or serious injury to personnel.

ACR series products and the information in this guide are the proprietary property of Parker Hannifin Corporation or its licensors, and may not be copied, disclosed, or used for any purpose not expressly authorized by the owner thereof.

Since Parker Hannifin constantly strives to improve all of its products, we reserve the right to change this guide, and software and hardware mentioned therein, at any time without notice.

In no event will the provider of the equipment be liable for any incidental, consequential, or special damages of any kind or nature whatsoever, including but not limited to lost profits arising from or in any way connected with the use of the equipment or this guide.

© 2004 Parker Hannifin Corporation  
All Rights Reserved

### Technical Assistance

Contact your local automation technology center (ATC) or distributor.

#### North America and Asia

Parker Hannifin  
5500 Business Park Drive  
Rohnert Park, CA 94928  
Telephone: (800) 358-9070 or (707) 584-7558  
Fax: (707) 584-3793  
Email: [emn\\_support@parker.com](mailto:emn_support@parker.com)  
Internet: <http://www.parkermotion.com>

#### Germany, Austria, Switzerland

Parker Hannifin  
Postfach: 77607-1720  
Robert-Bosch-Str. 22  
D-77656 Offenburg  
Telephone: +49 (0) 781 509-0  
Fax: +49 (0) 781 509-176  
Email: [sales.hauser@parker.com](mailto:sales.hauser@parker.com)

#### Europe (non-German speaking)

Parker Hannifin  
21 Balena Close  
Poole, Dorset  
England BH17 7DX  
Telephone: +44 (0) 1202 69 9000  
Fax: +44 (0) 1202 69 5750  
Email: [EMDTech.Help@parker.com](mailto:EMDTech.Help@parker.com)

#### Italy

Parker Hannifin  
20092 Cinisello Balsamo  
Milan, Italy via Gounod, 1  
Telephone: +49 (0) 781 509-0  
Fax: +49 (0) 781 509-176  
Email: [sales.sbc@parker.com](mailto:sales.sbc@parker.com)



Automation

Technical Support E-mail

[emn\\_support@parker.com](mailto:emn_support@parker.com)

# Table of Contents

Main Screen	6
<b>Mode Buttons</b> .....	7
InCycle Toolbar Functions	8
Cycle Start Button .....	8
Feed Hold Button .....	8
Reset Button .....	8
Auto / Step Button.....	8
<b>InCycle Information Area</b> .....	9
System Status Bar	10
CNC_MODE.....	10
ESTOP .....	10
FEEDHOLD.....	10
OVER TRAVEL .....	10
CYCLE TIME.....	10
Graphics Features	11
Part Program Verification.....	11
To Zoom In or Out.....	11
Program Editor	12
Program Error Messages.....	12
Part File Locations .....	12
Auto Loading a Part File.....	12
Coordinate Display Features	14
Readout Groups.....	14
Overrides	16
Feedrate .....	16
Rapid.....	16
Spindle .....	16
Spindle Features	17
Spindle Pause Resume.....	17
Coolant Pause Resume .....	17
Dry Run Option	19
DryRun Mode .....	19
M01 Enable Option	20
Virtual Estop Button	21
<b>File Menu</b> .....	23
<b>Edit Menu</b> .....	24
<b>View Menu</b> .....	25
<b>Setup Menu</b> .....	26
<b>Tools Menu</b> .....	27
<b>Help Menu</b> .....	28

Jogging and Homing	<b>29</b>
To Jog an Axis .....	29
To Home a Single Axis.....	29
To Home All Axis.....	29
Homing All Sequence.....	29
How the Software Homing Works:.....	29
Homing Subroutine Addresses in Prog0.8k.....	29
Virtual Pendant	<b>31</b>
Using MDI Mode	<b>32</b>
Using Tool Offsets	<b>33</b>
To Set Tool Diameter.....	33
To Set Tool Length.....	33
To add or subtract value from Current Offset Value.....	33
Using Work Offsets	<b>34</b>
Tool Changer Menu	<b>35</b>
Alternate Method to Load Tools in the Magazine via the Spindle .....	35
To Manually Load tools in Magazine using editor.....	36
System Diagnostics Screen	<b>37</b>
Parameter Editor	<b>39</b>
<b>CNC Config</b> .....	40
<b>Software Options</b> .....	41
<b>Spindle Options</b> .....	42
<b>Tool Changer</b> .....	43
<b>Homing Offsets</b> .....	44
<b>Tang Options</b> .....	45
<b>Speeds and Feeds</b> .....	46
<b>System Resolution (PPU)</b> .....	47
<b>In Position Bands</b> .....	48
<b>BackLash Compensation</b> .....	49
<b>Gains</b> .....	50
<b>Auxiliary Mcodes</b> .....	52
<b>Mcodes Validated</b> .....	53
The Gains Editor	<b>54</b>
Tuning a single axis .....	54
Tuning 2 Axis Simultaneously.....	54

G Code Modal Status Screen	55
Validating MCodes	56
How to Configure User Defined MCodes	57
Configuring User Defined Error Messages	58
How to Setup the ToolChanger	60
Virtual Pendant Setup	62
HandWheel Setup	63
Configuring a Tangential Axis	65
Using the GCode Optimizer	68
Machine Control Signal Bits 128 to 255	70
M Code Bits 1920-2047	71
Input and Output Map	72
Inputs Map .....	72
Outputs Map.....	73
System Log Files	74
Technical Support	75

Main Screen

Click a screen section to see it's associated Help.

The screenshot displays a CNC control software interface. At the top, the title bar shows the current file: "Current File: C:\CNC\_PartFiles\19z-141-1 Lifting Adapter.tap". Below the title bar is a menu bar with "File", "Edit", "View", "Setup", "Tools", and "Help".

The main interface is divided into several sections:

- Program Line:** Shows "ProgLine: 1" and "Cycle Complete". Below this, the current line of code is displayed: "T = 2 H4: -27.2711 D104.0 W.1".
- Program Code:** A scrollable area containing G-code: "O1000(19Z-141-1 LIFTING ADAPTER)", "G91 G0 G28 G80 Z0", "T1 M6 ( 2-1/2 FACE MILL )", "S850 M3", "G90 G0 G43X1.75Y-6.5Z1.0H1 M8", "Z0.0", "M98 P1001", "G90 G0 Z1.0", "M5", "M9", "G691 G0 G28 G80 Z0", "T6 M6 (#5 CENTER DRILL)", "S1200 M3".
- Coordinate Display:** Shows X, Y, and Z coordinates. X and Y are at 00.0000, and Z is at 27.2711. There are also "Distance to Go" fields for each axis, all showing 00.0000.
- Control Panel:** Includes buttons for "F2 Edit", "F3 Run", "F4 Jog", "F5 MDI", "F6 Offset", "Start", "F-Hold", "Reset", "Auto", and "Step".
- Spindle and Coolant Controls:** Shows "Feedrate 100%", "Rapid 100%", "Current Feedrate 0000.00", "Actual RPM: 00000.00", "Program RPM: 03377.00", and "Spindle Load 0%". There are also "Resume" and "Off" buttons for both Spindle and Coolant.
- Viewing Options:** Buttons for "Clear", "Top", "Front", "Side", "NW", "NE", "SW", "SE", and "Zoom".
- Status Bar:** At the bottom, it shows "CNC\_Mode: RUN", "EStop = OFF", "FeedHold = OFF", "OverTravel = OFF", and "Cycle Time: 0:00:06".

## Mode Buttons



### Edit Mode F2

When Edit Mode is selected the Editor Screen will change to Full Screen Mode for Editing.

If an Open File is changed, a Save Button will appear on upper right side of the Editor to remind you to save your work.

An edited file can't be run until it is saved with No Errors, because the system Pre-Process the G Code Files prior to Runtime.

The File PullDown Menu items (Open Save and New) will be enabled, allowing user File Operations.

Note: Cycle Start Functions are disabled in Edit Mode

### Run Mode F3

File functions are locked out in Run Mode.

When Run Mode is selected the Editor will shrink to half size and the bottom half will display the Graphics Screen

When Cycle Start is activated the Editor Background will turn Gray and the Current G Code line will be highlighted.

When the Cycle completes, the Editor Background Color will return to White and line 1 will be highlighted.

Runtime Background colors can be changed using the Screen Setup Menu.

### Jog Mode F4

Select Jog Mode to Jog or Home the Machine Axis.

Note: Not available when the control is in cycle. [See Jog Mode Screen](#)

### MDI Mode F5

Enter any Valid M or G Code and Press Execute or Enter to Execute one line Commands.

Note: Not available when InCycle or in DRYRUN Mode. [See MDI Screen](#)

### Offsets Mode F6

Use this Mode to Edit & Display the Tool and Work Offsets.

[See ToolOffsets](#)

[See WorkOffsets](#)

## InCycle Toolbar Functions



### Cycle Start Button

- ❑ Press to initiate the Cycle, or to advance one line at a time if in Step Mode.
- ❑ A program may be started from a line other than the first line.
- ❑ When Cycle Start is clicked it looks at the Editor Cursor position if it's anywhere but on the first line a popup box will ask you if you want to run from the selected line... click ok to continue or cancel to abort.

Note: Gcode Program must be loaded with no Errors and in Run Mode or this button to be disabled

### Feed Hold Button

- ❑ Pressing Feed hold will pause the Program but not end the cycle, release Feed Hold to continue.

Note: Feedhold may not always stop the program where it's at, depending on what type of Gcode is executing such as a canned cycle like Rigid Tapping or and MCode, the control will complete if started executing.

### Reset Button

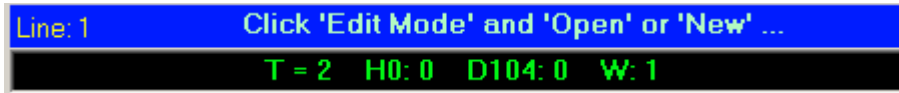
- ❑ Reset will stop the current running program and reset to the program start ... line 1. Reset will also terminate a currently executing MDI command.

### Auto / Step Button

- ❑ This button sets the Control Run Mode.
- ❑ Select Auto to run the program from start to finish, Step to run the program one line at a time.



## InCycle Information Area



Line 1 ... Displays the Current Gcode Line, and any User Prompt or File Load Error Status Messages.

Line 2 ... Displays Current Tool and Work Offset information as Follows:

- T = Tool Number
- H = Tool Length Offset Number
- D = Tool Diameter Offset Number
- W = Current Work Offset Number

## System Status Bar

CNC_Mode: RUN	EStop = OFF	FeedHold = OFF	OverTravel = OFF	Cycle Time: 0:00:01
---------------	-------------	----------------	------------------	---------------------

### CNC\_MODE

- Displays the Control Mode Run / Edit / Jog / MDI / Offsets.

Run = Bit 134    Edit = Bit 135    Jog = Bit 136    MDI = Bit 137    Offsets = Bit 138

### ESTOP

- Flashes On/Off when the Estop Signal (Bit 129) is on.

### FEEDHOLD

- Flashes On/Off when the Feed Hold Signal (Bit 131) is on.

### OVER TRAVEL

- Flashes On/Off when the OverTravel Signal Bit (192) is on.

### CYCLE TIME

- Displays the Gcode Program Running Time in Hours/Min/ Seconds.

Note: This Display is reset to Zero upon a Cycle Start.

## Graphics Features



The Zoom Toolbar allows you to View the program from different View Angles.

### Part Program Verification

If the control is in Dry Run or in Offline Mode you can run the program to verify its movements.

Graphics are only available in Run Mode.



The Zoom Button will allow the user to zoom into the current view, or zoom back out.

### To Zoom In or Out

Click the Zoom button and a crosshair will appear when you click the mouse, then drag the window to size and release the mouse button to end.

Select Zoom All to automatically return the graphics to its original un-zoomed state.

Note: The Current View Zoom level is saved if zoomed in until followed by a zoom all or a new program is loaded. What this means is you may have your NE View Zoomed in close and the other full scale.

## Program Editor

The Editor is only available in Edit Mode.

### Error Checking

All files must be saved with No Errors, for the Cycle Start Button to Enable in Run Mode.

Full Text Editing is provided such as Copy Paste Find Undo Print etc.

If the user changes the file a Save Button" will popup signaling the user to save the file.

To Open (Load) a file:

1. Click F2 Edit Mode.
2. Click the File PullDown Menu and Select Open.
3. Using the Browser Select the Desired File.
4. Click OK and the Control will Read in the program and check it for Errors and unknown commands.
5. After loading is complete the Status Box above the editor will display the load status. See below.
6. Click F3 to set Control in Run Mode
7. Press Cycle Start to initiate program running if it loaded successfully.
8. If the Program has Errors they must be corrected and saved to run a program

## Program Error Messages

The Status Box above the editor will display:

Program Errors found" if an Error is found in the Program.

Program Loaded Ok" upon successfully loading.

## Part File Locations

Part Files are initially set to a Sub-Folder of where the program installed called Partfiles.

You may change this path by browsing to another location and the program will remember your last Path and filename.

OpenProgWithLastFile=0

## Auto Loading a Part File

The System will automatically load the last program ran when the Software first starts if the Parameter OpenProgWithLastFile is set =1.

This Parameter OpenProgWithLastFile is in the CNC\_Config Section of the Parameters.

## Coordinate Display Features

Valid Axis Letters are X Y Z for Axis 0 1 2 and A B or C for the 4<sup>th</sup> Axis.

4<sup>th</sup> Axis letters Must Designate Rotary Table Orientation.

A/B = Rotary Axis is Vertical, C = Rotary Axis is Horizontal (Facing Spindle Tool)

## Readout Groups

The Control System Displays 2 Groups of Coordinate Options



	Program	Distance to Go
X	00.0000	00.0000
Y	00.0000	00.0000
Z	00.0000	00.0000
C	000.000	000.000

Group 1:

Machine = ABS Position.

Program = ABS Position + Tool Offsets.



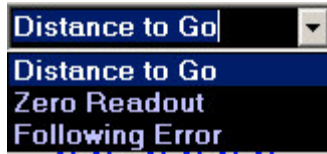
Group 2:

Distance to Go = Distance remaining from Target Position

Zero Readout = Zeros the Right Side Readouts Group to be ABS from the Current position.

Note: This does not zero where the machine thinks it is in any way. This feature is used to zero on some position of reference and to move off from that position without having to decode the Program or Machine Position Coordinates.

Following Error = Shows the Axis Following Error Typically used for [Axis Tuning](#) .

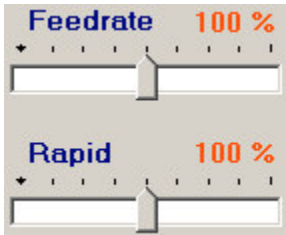


To Change Coordinate Colors use the View->Screen Colors PullDown Menu.

The Axis Displays may be turned On/Off via the View->AxisReadout PullDown Menu.

Important Note: Even if you reassign an Axis Letter for Display you must still write your programs using XYZ. We are working on this feature but at this time it's not ready for release.

## Overrides



Feedrate ... Scales Current Feedrate

Rapid ... Scales Current Rapid Feedrate

Spindle ... Scales Current Spindle Speed

Override Parameters are contained in the Section = SoftwareOptions

To Configure for Type 98 Override (Using the Screen Sliders)

1. UseADCC ChanelForOverrides = 0
2. Set all 3 Override Types to = 98
3. Set all 3Max Override Value Parameters
4. Set all 3 OverridePotMaxVoltage Parameters

To Configure for Type 1 Override (Using Digital 4 Bit Pots on Control Panel)

1. UseADCC ChanelForOverrides = 0
2. Set All 3 Override Types to = 1
3. Set Max Override Value Parameters
4. Write code in AcroBasic that Sets Digital\_OVR bits of the control Signals.

To Configure for Type 98 Override (Using VB Screen sliders Exclusively for Overrides)

UseADCC ChanelForOverrides = 0

Set All 3 Override Types to = 98

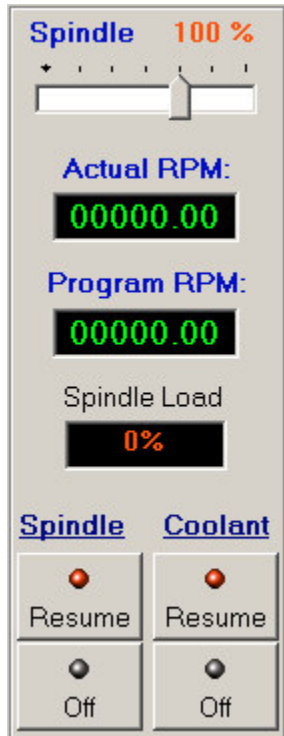
Set Max Override Value Parameters

To Configure for Type 99 Override (Hardcode Overrides to Max Values)

1. UseADCC ChanelForOverrides = 0
2. Set All 3 Override Types to = 99
3. Set Max Override Value Parameters ( These Values will Be Constant for the Overrides)



## Spindle Features



Spindle Override	Adjusts the Actual Spindle Speed.
Actual RPM	Displays the Actual Spindle RPM.
Program RPM	Displays the Programmed RPM.
Spindle Load	Displays the Analog Feedback from the Spindle Load Output.

## Spindle Pause|Resume

Pause -> Stops Spindle if Running (Actually Executes an M5)

Resume = Restarts Spindle from Pause

Restarts the Spindle if it was in Pause, and restarts in the direction it was previously running.

## Coolant Pause|Resume

Pause -> Stops Coolant ... Executes an M08

Resume = Restarts Coolant ... Executes a M09

The above 2 features are typically used while in cycle to check the part or change cutter inserts.

#### Dry Run Option



**Dry Run**

**M01 Enable**

#### DryRun Mode

Uses bit 132 = 1 when active.

Select DryRun to run program and verify Program Graphically with out Motion.

Dry run will not execute any Axis Moves or M Codes.

If this program is run without a motion controller in the PC, or the Card is not found due to computer problems the System is automatically set to DryRun mode. This allows the software to programmed offline.

## M01 Enable Option

 **Dry Run**  
 **M01 Enable**

Select M01 enable to Enable M01's in the program.

Typically a programmer would write a program with M01's included at key points, to stop execution to check sizes or change parts. Selecting this option will cause the Gcode program until Cycle Start is pressed.

If unchecked all M01's will be ignored.

Note: This option must be selected before program execution starts.

## Virtual Estop Button



The Virtual Estop Function Button is to allow the user to have Estop if the machine isn't equipped with an external Emergency Stop Button.

Virtual Estop must be enabled in the parameters by setting UseVirtualEstop=1 to enable this feature.

Setting this parameter to 1 will set bit 193.

### Function:

#### Virtual Estop Pressed (down)

Bit 522 Kill All Moves is set to 1.

Bit 194 sets the Virtual Estop Active bit to =1.

Bit 129 sets the program reset bit.

#### Virtual Estop Released (up)

Bit 522 Kill All Moves is set to 0.

Bit 194 clears the Virtual Estop Active bit to = 0.

Bit 129 the program reset bit is cleared = 0.

### Related Parameters:

SectionName CNC\_Config      ParameterName = UseVirtualEstop

### Related Bits used

Bit#	ParameterName	Description
193	EnableVirtualEstop	Enables Virtual Estop to be used
194	VirtualEstopActive	This bit is 1 when V-Estop is on
195	VirtualEstopSealbit	Mirror of bit 194 for status updates

If your application needs to do more then you may use these bits to call a subroutine in prog1 line 1000 of the AcroBasic.

## File Menu



Items -> Open / Save / New / Print are only available in F2 Edit Mode.

Open	... Opens Explorer Type File Browser.
Save	... Saves the Current File or Renames file.
New	... Creates a New file.
Print	... Prints Current G-Code File.
Printer Setup	... Configure System Printer.
Exit	... Ends the Program

### Additional Information:

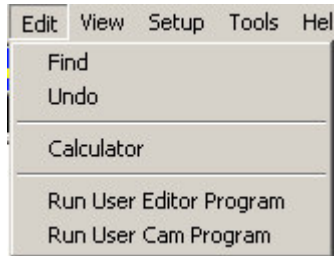
When Edit Mode is selected the Editor Screen will change to Full Screen Mode for Editing.

If an Open File is changed, a Save Button will appear on upper right side of the Editor to remind you to save your work.

An edited file can't be run until it is saved with No Errors, as the system Pre-Process G Code Files prior to Runtime.

Note: When Control is in Edit Mode Cycle Start Functions are Disabled, Switch to Run Mode to Run Program.

## Edit Menu



Find... finds or replaces a search string in the program.

Undo ... reverts to the last edit.

Calculator ... this will open the Operating System Calculator.

Run User Editor ... Open the user definable external text editor such as notepad.

Run User Cam ... Open the user definable external Cam program.

To Set up User Editor:

Go to the Machine Setup Screen.

Select CNC\_Config" Section and expand its Branch.

Edit the Item Named ConversationalProgPath" giving it the full path to the program you want to run.

Example: UserEditorPath =notepad.exe

To Set up User Cam Program:

Go to the Machine Setup Screen.

Select CNC\_Config" Section and expand its Branch.

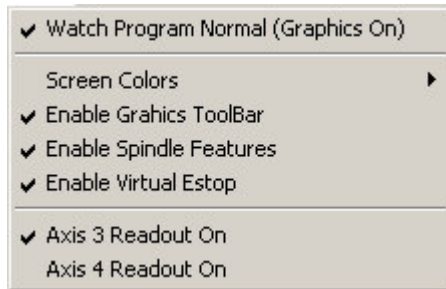
Edit the Item Named ConversationalProgPath" giving it the full path to the program you want to run.

Example: ConversationalProgPath = C:\MY\_Cam.exe

Note: You may edit this and any other Parameter directly by editing the Parameters.Cfg file in the Parfiles Sub-Folder that you installed to.



## View Menu



Watch Program Normal (Graphics On): This feature if selected disables the graphics output so you just see the Gcode program running.

Screen Colors: Enables the Screen colors editor options fly-out menu.

Enable Graphics ToolBar

Enables the Full Graphics Toolbar or shows just the Zoom Button.

Enable Spindle Features.

Enables the Spindle displays for milling type machines. This is typically turned off for Laser/Water Jet Type Machines.

Enable Virtual Estop

Use if your machine has no external Estop button.

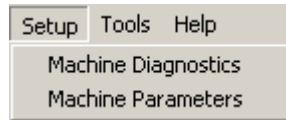
Axis 3 Readout On

Enables/Disables the 3<sup>rd</sup> axis readout.

Axis 4 Readout On

Enables/disables the 4<sup>th</sup> axis readout.

## Setup Menu

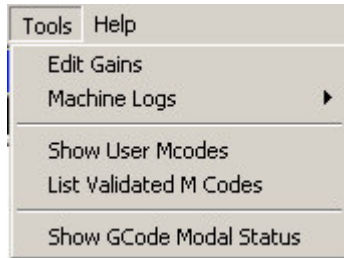


Machine Diagnostics: Opens the Systems Diagnostics Screen and is password protected.

Machine Parameters: Opens the System Parameters Editor and is password protected.

Machine Diagnostics are available when the control InCycle, Parameters are not.

## Tools Menu



### Edit Gains

Shows the Live Gains editor Screen. Use this Screen and the Following Error to fine-tune the system while its running a program. [See using the Gains Editor](#)

### Machine Logs

Displays the following Log Fly-out Menu items.

Show Move Log... Displays the moved Coordinates of the current program.

Show Machine Operation Log ... Displays a secession log of all moves the user made.

Note: These log files are used for trouble shooting, and may be requested by technical support to solve Gcode Program Anomalies and Control System Errors.

### Show User MCodes

Displays the 10 User Definable MCodes for the user to use.

[How to Configure User Defined Error Messages](#)

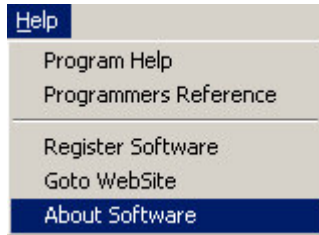
### List Validated MCodes

Displays a screen listing the Validated MCodes for the System to use.

### Show Gcode Modal Status

Displays a Runtime Gcode Modal Status list in the Graphics Area of the program.

## Help Menu



### Program Help

Loads this Help System

### Programmers Reference

Loads the Programmers Reference Help

### Register Software

Shows the Program Registration Screen with a Code derived from the PC to allow the user to Register or Enable Updates to higher levels of software features based on a Unlock code from our company.

### Goto our WebSite

Will open the users Internet and Goto our WebSite for online downloads or support.

### About Software

Displays this Programs Version and Pathing information.

## Jogging and Homing

### To Jog an Axis

Select XYZ or 4<sup>th</sup> axis letter of Axis to Jog, the Selected Axis will display in top of Jog Screen.

Press + or – and the Axis will Jog until you let up.

(Select Fast to Jog in Rapid) ... Jog Fast/Slow Parameters are set in the Parameters (Jog Speeds) Section.

### To Home a Single Axis

Select Axis and select Home, the axis will perform the Homing Action.

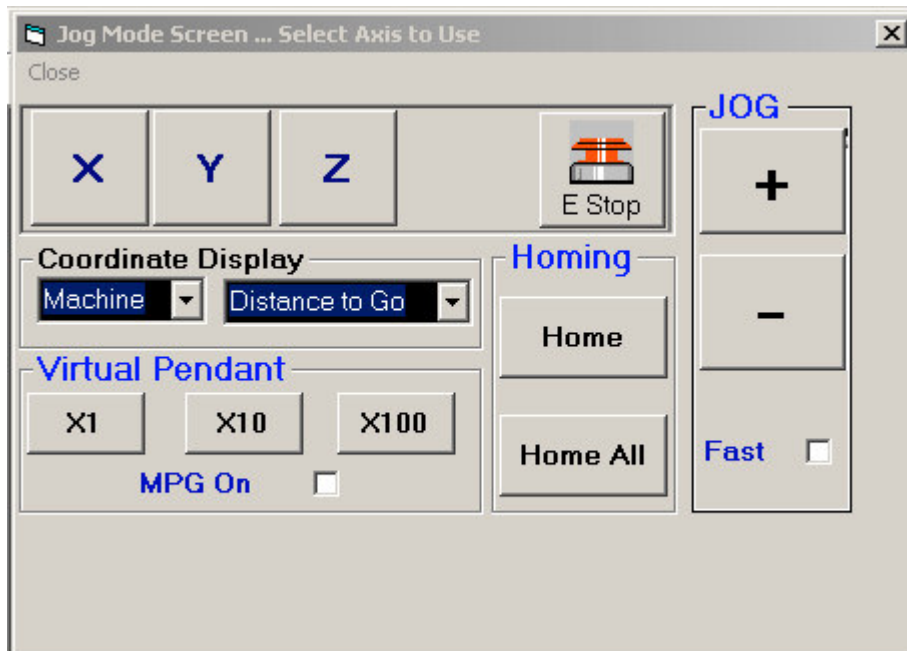
### To Home All Axis

Select Home All and the axis will perform the Homing according to the HomingSequence Parameter.

### Homing All Sequence

The Homing sequence is programmable and is set in the Homing Sequence Parameter.

The Default Homing Sequence is ZYX. Parameter Section = CNC\_Config / Parameter Name = HomingSequence.



### How the Software Homing Works:

The software initiates homing by calling predefined homing subroutines contained in the AcroBasic Prog0.8k the file is in a (Subfolder Named AcroBasicFiles) shipped with this product. You may edit the subroutines to fit your application.

### Homing Subroutine Addresses in Prog0.8k

X homing subroutine is at line 100 to 199

Y homing subroutine is at line 200 to 299

Z homing subroutine is at line 300 to 399

A B C homing subroutine is at line 400 to 499

The subroutine functions work like this:

100 set homing active bit 144. This informs the system that homing is active.

Enter your homing code here.

199 Set the X homing Done Bit (170) and clear bit 144 when homing is done and exit the sub.

Home Offsets

The Homing Offsets are stored the Parameters file section HomeOffsets. You may use these offsets to move an additional distance after homing is complete. The offsets are stored in the AcroBasic Global P Variables P15/16/17/18.

## Virtual Pendant

If Parameter HandwheelInstalled=0 " then the system will enable this section for Machines not equipped with a remote pendant.

Parameter Used:

Section Name = CNC\_Config"      Parameter Name HandWheelInstalled"

To Use:

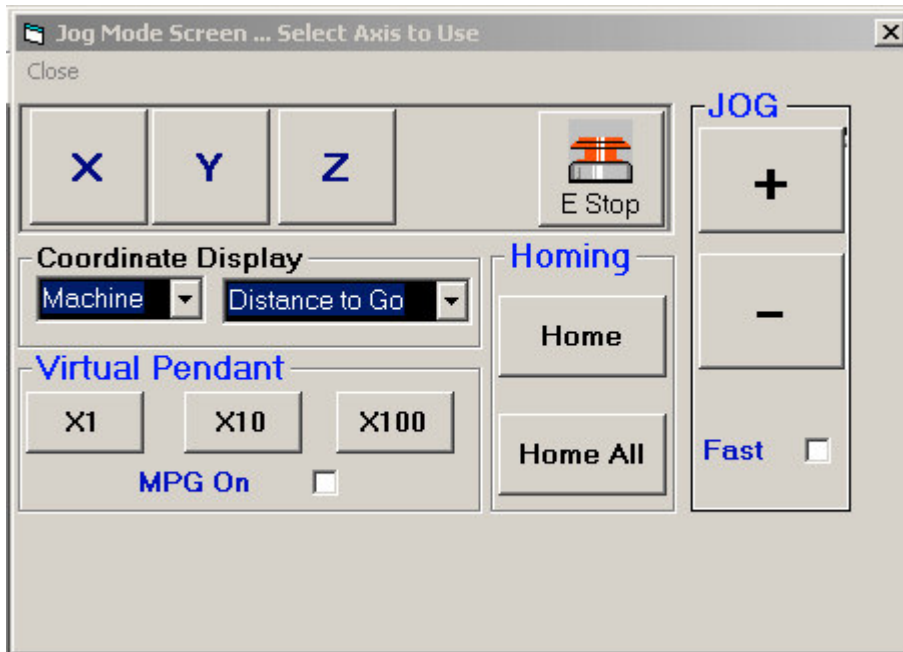
Switch to Jog or Offsets Mode

Select MPG ON" to Enable.

Select the Axis Letter

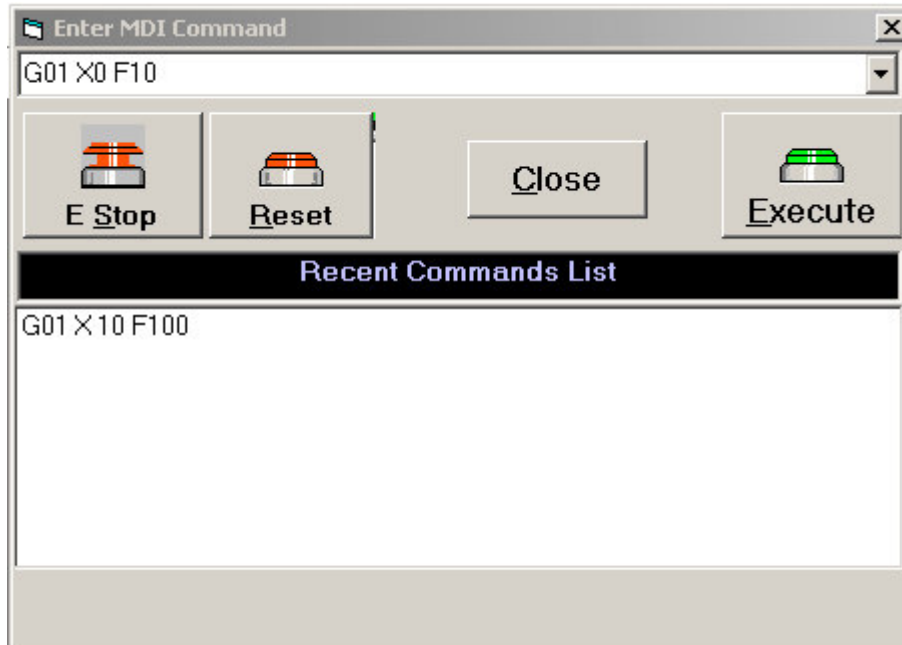
Select (X1) (X10) (X100) for .001/ .010 / .100 increment moves on the Selected Axis.

Press & Release the + or – buttons to move in the Selected Move Increment.



## Using MDI Mode

Enter any Valid M or G Code and Press Execute or Enter to Execute one line Commands.



### Features:

Previous commands are Stored in the Recent Commands List, so you can Repeat a Previous Command.

Select by clicking The Recent commands List or using the arrow key and selecting a previous command.

Estop and Reset is provided to assist the operator in case of problems.



## Using Tool Offsets

The Control System provides 200 Tool Offsets.

Jogging and HandWheel functions are provided to touch off Tooling and set offsets.

### To Set Tool Diameter

Click on the desired Diameter (D) and Manually Edit the Offset or Click the Set X or Set Y buttons to directly set the offset from the Machine Coordinates.

### To Set Tool Length

Click on the desired ToolLength, and Manually Edit the Offset or Click Length". This will use the Machine Readouts to set the ToolLength.

Special Feature:

To add or subtract value from Current Offset Value.

Example:

Click in desired Cell, move cursor to end of dimension and type + .125, and press Enter.

The .125 value will be added to the current value. Valid operators are + or –.

Tool Offsets				
Description	T#	Pkt	H	D
▶ 2.5 Ingersoll	1	1	0	0
.75 Ingersoll	2	2	0	0
Blank	3	3	0	0
.75 X 45Deg	4	4	0	0.75
1.0 Ingersoll	5	5	-24.7011	1.002
#6 CDrill	6	6	-26.0491	0.22
#6 Drill	7	7	-24.3925	0
13/32 Drill	8	8	-23.4076	0
3/8 CBore	9	9	-23.0021	0
1/8 CEndmill	10	10	-28.2761	0
Blank	11	11	0	0
17/32 Drill	12	12	0	0
31/32 Drill	13	13	-24.9286	0
Blank	14	14	0	0
5 CEndmill	15	15	-26.49	0.4954

## Using Work Offsets

The Control System provides 100 Work Offsets.

Jogging and HandWheel functions are provided to touch off Tooling and set offsets.

Work Offset # Column is Read-only, all others are Editable

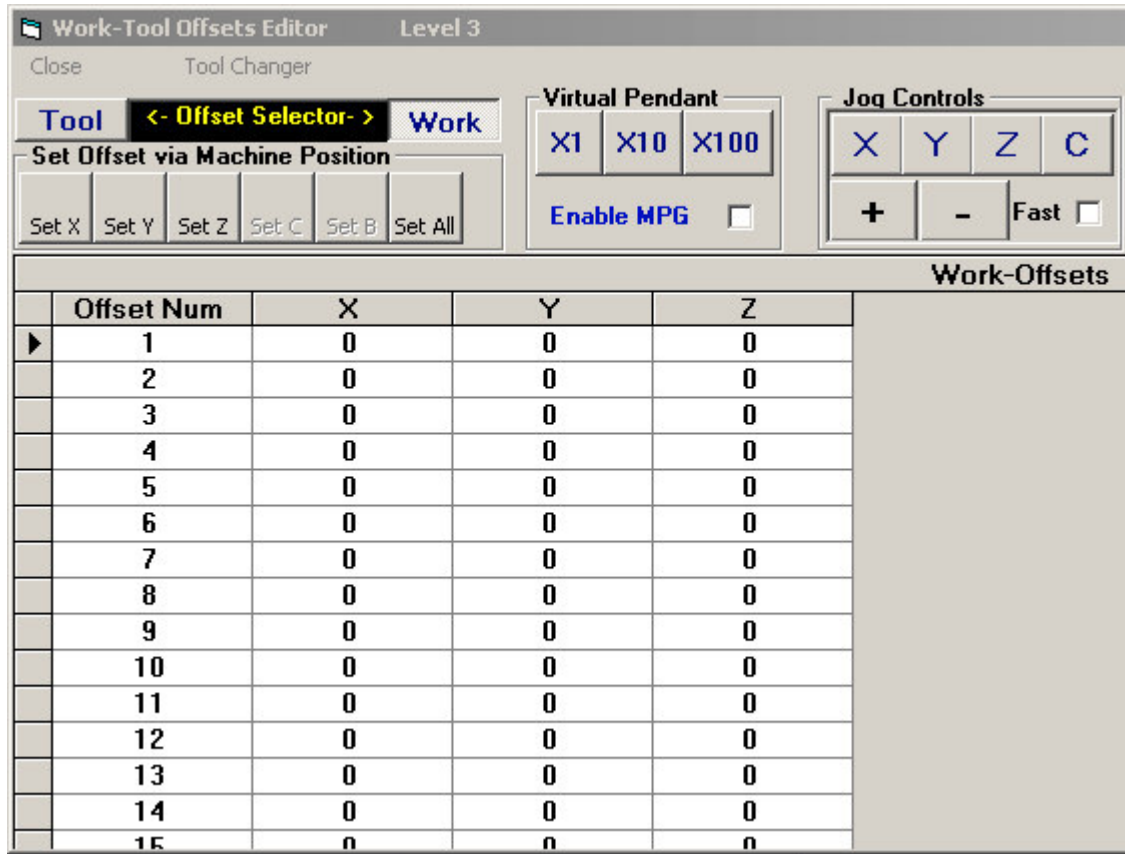
Use the Set X, Y, Z, and B buttons to set that WorkOffset from the Machine" Readouts Position.

Use Set All To set all of the Axis's WorkOffsets from the Current CNC Readouts, at the same time.

Add or Subtract value from Current Offset Value.

Example: Click in desired Cell, move cursor to end of dimension and type + .125, and press Enter.

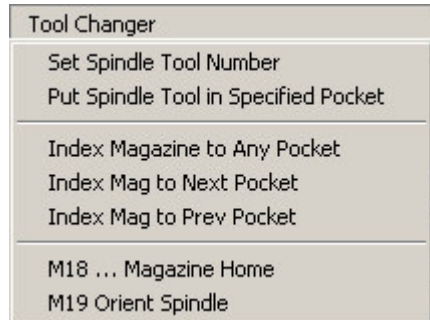
The .125 value will be added to the current value. Valid operators are + or -.



The screenshot displays the 'Work-Tool Offsets Editor' window at 'Level 3'. The interface includes a 'Close' button, a 'Tool Changer' section with 'Tool' and 'Work' buttons, and a 'Virtual Pendant' section with 'X1', 'X10', 'X100' buttons and an 'Enable MPG' checkbox. A 'Jog Controls' section contains 'X', 'Y', 'Z', 'C' buttons and '+', '-', 'Fast' controls. Below these controls is a table titled 'Work-Offsets' with columns for 'Offset Num', 'X', 'Y', and 'Z'. The table contains 15 rows, all with a value of '0' in the X, Y, and Z columns. A mouse cursor is positioned over the first cell of the table.

Offset Num	X	Y	Z
1	0	0	0
2	0	0	0
3	0	0	0
4	0	0	0
5	0	0	0
6	0	0	0
7	0	0	0
8	0	0	0
9	0	0	0
10	0	0	0
11	0	0	0
12	0	0	0
13	0	0	0
14	0	0	0
15	0	0	0

## Tool Changer Menu



Used to Load and Move the Tool Magazine via the Spindle

Set Spindle Tool Number... Makes Requested Tool in Spindle Current (User Prompted for T number to Set to)

Put Spindle Tool in Specified Pocket ... Puts Spindle Tool in magazine (User Prompted for Pocket Number)

Index Magazine to Any Pocket ... Moves Magazine to Requested Pocket (User Prompted for Pocket Number to Set to)

Index Magazine to Next Pocket ... Moves Magazine to the Next open Pocket.

Index Magazine to Previous Pocket ... Moves Magazine Backwards to the next open Pocket.

M18 ... Orients Magazine

M19 ... Orients Spindle

These Functions are not available when Control is in Cycle

To load tools in the Magazine via the Spindle.

Load the tool into the Spindle Manually.

Click the ToolChanger Menu and select "Set Spindle Tool Number" and enter a tool number.

Click "Put Spindle tool in Specified Pocket".

The System will Index the Magazine, and place the Spindle tool in the Desired Pocket, and update the Tool Table automatically when Tool change completes.

### Alternate Method to Load Tools in the Magazine via the Spindle

1. Select "Set the Spindle Tool Number" as above.
2. Switch to MDI Mode and execute a M6T0, The System will simply lookup the next empty pocket found in the ToolTable and place the tool in that Pocket and update the ToolTable.

To Manually Load tools in Magazine using editor

1. Place tools in magazine pockets manually.
2. Goto (F6) Offsets Screen ... click Tool" section.
3. Enter Tool Number information directly in the Desired Pocket of Column 3.

Tool (T#) Column 2 ... is Read-only, all others are Editable.

A Zero in a Pocket Means the Pocket is currently empty or it's in the Spindle.

## System Diagnostics Screen

The Diagnostics Screen Displays all system I/O, and allows the user to turn Control Signals On or Off.

The System I/O Map is split into 5 Sections

Signal Description	ACR Bit Address	Description Editable
Inputs 0 to 63	0 to 31 256 to 287	YES
Outputs 0 to 64	32 to 63 288 to 319	YES
Control System Signals	128 to 255	NO
Master Flags	512 to 543	NO
MCodes M00 to M127	1920 to 2047	NO

To Turn a Signal On/Off

Click its Cell and The Signal Name Will Display in the Signal Name Box in the top center this Screen.

2. Click the On or Off Button as desired.

Also: If you know the Explicit Bit Address of the Signal Bit, you may type it in the box and press on or off.

To Edit an Input or Output Description

DbI-Click or place Cursor in the desired cell and press Enter, the Cell background color will turn Yellow.

Edit item, then press Enter to Save. Click any other Cell and edits will not be saved.

Inputs can't be Turned On/Off as they are the result from Hard-wired Sources.

**Diagnostics Screen**  
Close Help

Bit Number			Description
0	Off	On	signal name
Inputs		Outputs	
Inp / Bit	Signal Name	Out / Bit	Signal Name
0 / 0	Reset PB	0 / 32	Pc On Lamp
1 / 1	FeedHold PB	1 / 33	Cycle Start Lamp
2 / 2	Cycle Start PB	2 / 34	Feed Hold Lamp
3 / 3	Estop PB	3 / 35	Reset Lamp
4 / 4	Pendant Enable	4 / 36	SPINDLE Drive Fault Rese
5 / 5	X Select	5 / 37	Winding Selection
6 / 6	Y Select	6 / 38	L Gear Select
7 / 7	Z Select	7 / 39	M Gear Select
8 / 8	4th Select	8 / 40	Spindle Stop

System Signals
Bit / Signal Name
128 / CYCLESTART
129 / ESTOP
130 / RESET
131 / FEEDHOLD
132 / DRYRUNMODE
133 / AUTO / Step Mode [Auto
134 / RunMode
135 / JOGMODE
136 / MDIMODFACTIVE

Close

Master Signals	M Codes
Bit / Signal Name	Bit / M Code
512 / Accelerating	1920 / M 0
513 / Decelerating	1921 / M 1
514 / Stopping	1922 / M 2
515 / Jerking	1923 / M 3
516 / In Motion	1924 / M 4
517 / Move Buffered	1925 / M 5
518 / Feed Holding	1926 / M 6
519 / In Feedhold	1927 / M 7
520 / Feedhold Req.	1928 / M 8

## Parameter Editor

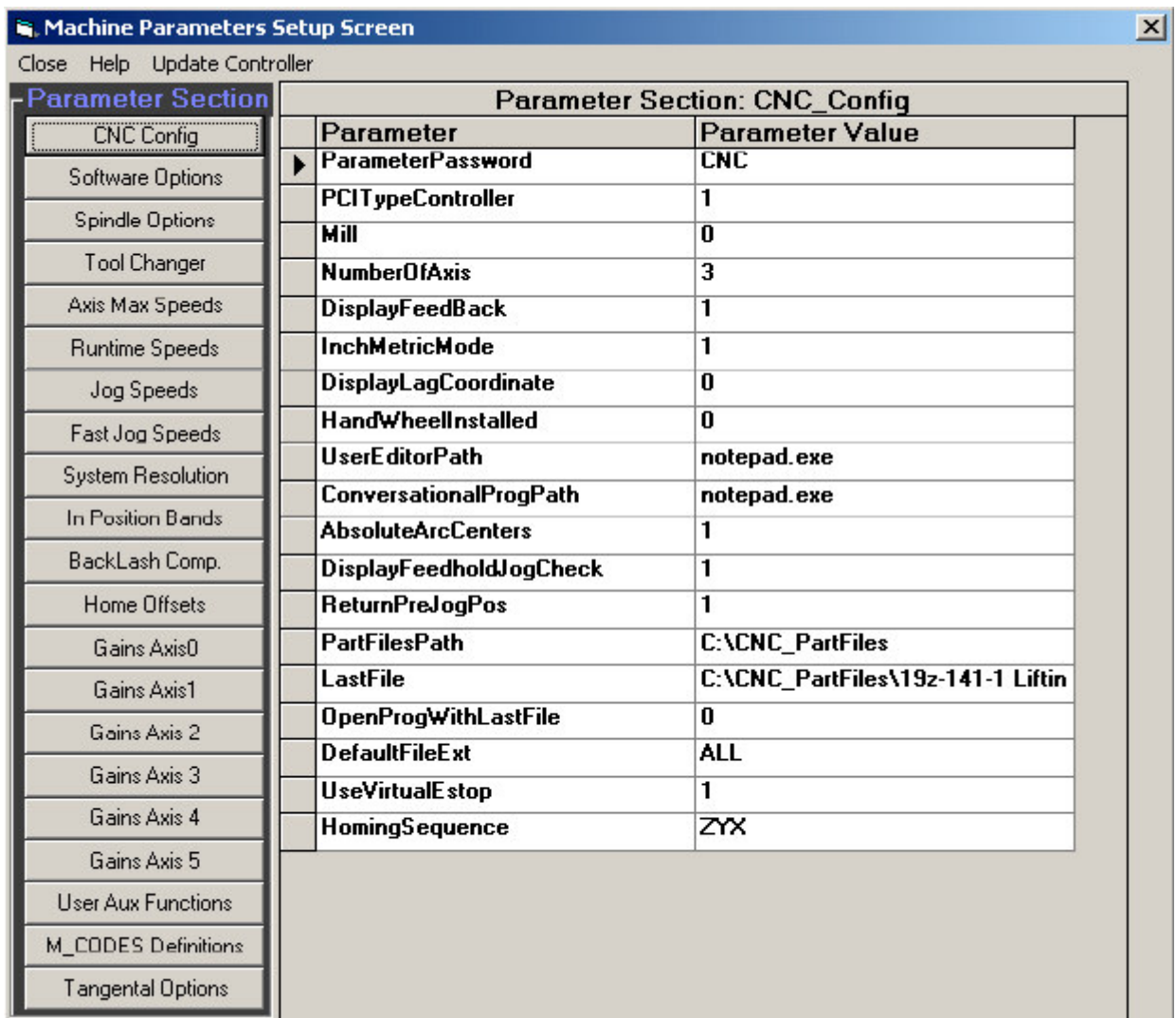
Machine Parameter Sections are shown on the left and are derived from the Parameters.Cfg file.

To Edit a Parameter ... Select a Parameter Section Button on the left, and click on the grid item to edit its value.

To Save a Parameter ... Press Enter after editing to save changes.

Update Controller ... Updates the Control System instantly from the current parameters such as the Gains.

Otherwise changes won't take effect till the Setup Screen closes, which updates whenever a parameter has changed.



The screenshot shows a window titled "Machine Parameters Setup Screen" with a menu bar containing "Close", "Help", and "Update Controller". On the left is a "Parameter Section" list with buttons for various sections. The "CNC Config" section is selected. The main area displays a table for the "Parameter Section: CNC\_Config" with columns for "Parameter" and "Parameter Value".

Parameter	Parameter Value
ParameterPassword	CNC
PCITypeController	1
Mill	0
NumberOfAxis	3
DisplayFeedBack	1
InchMetricMode	1
DisplayLagCoordinate	0
HandWheelInstalled	0
UserEditorPath	notepad.exe
ConversationalProgPath	notepad.exe
AbsoluteArcCenters	1
DisplayFeedholdJogCheck	1
ReturnPreJogPos	1
PartFilesPath	C:\CNC_PartFiles
LastFile	C:\CNC_PartFiles\19z-141-1 Liftin
OpenProgWithLastFile	0
DefaultFileExt	ALL
UseVirtualEstop	1
HomingSequence	ZYX

CNC Config

<b>[CNC CONFIG]</b>	<b>Description</b>	<b>Default</b>
PciTypeController	Tells the system the Type of Controller being used. Set to 1 for all PCI Type 0 for All ISA types.	<b>1</b>
ParameterPassword	Used for Parameter & Diagnostics Password	<b>CNC</b>
Mill	Set to 1 for Mill or 0 for Lathe	<b>1</b>
NumberOfAxis	Sets the Number of Axis is to Control (Program)	<b>1</b>
InchMetricMode	Set to 1 Inch or 0 for Metric	<b>1</b>
HandWheellInstalled	Set to 1 in using Remote Pendant	<b>0</b>
UserEditorPath	Pointer to the User configurable Editor Path	
ConversationalProgPath	Pointer to the User configurable Cam Program Path	
AbsoluteArcCenters	Set to 1 if using ABS Arc Centers else set to 1 for incremental	<b>1</b>
PartFilesPath	Pointer to Partfiles Folder	
LastFile	Pointer to the Last File Opened	
OpenProgWithLastFile	Set to 1 to Force the system to Auto Load the last opened program at start up	<b>0</b>
DefaultFileExt	Sets the Default file extension to use in Browser window	<b>ALL</b>
UseVirtualEstop=0	Turns on the Virtual Estop Button on the screen... Set to 1 to use in case you don't have and Estop button on your Machine	<b>0</b>
HomingSequence	Tells the system Homing Sequence ZYX is default. Valid Letters for Axis's 0/1/2 are XYZ. Valid Letters for the 4th axis are ABC. The Sequence is Read from Left to Right. Ex: ZYX will home Z first then Y then X. for 2 Axis Applications the sequence could be XY.	<b>ZYX</b>



Software Options

<b>[SoftwareOptions]</b>	<b>Description</b>	<b>Default</b>
AxisDisplayOn	Set to 1 to Make Axis coordinate visible in GUI Readouts	<b>1,1,1,0,0</b>
AxisDesignation0	Axis Letter Designation	<b>X</b>
AxisDesignation1	Axis Letter Designation	<b>Y</b>
AxisDesignation2	Axis Letter Designation	<b>Z</b>
AxisDesignation3	Axis Letter Designation	<b>B</b>
AxisDesignation4	Axis Letter (not used)	
AxisDesignation5	Axis Letter (not used)	
ExactStopRequired	Forces Control to Stop Motion at End of Every Rapid Move	<b>0</b>
UseADCChannelForOverrides	Specifies to Control System that Overrides are Derived From the ADC channels when =1	<b>0</b>
RapidOverrideMax	Sets the Max that the Override can be set to	<b>100</b>
FeedrateOverrideMax		<b>200</b>
SpindleSpeedOverrideMax		<b>150</b>
RapidOverridePotVoltage	Specifies to Control Max DC Voltage Derived From the ADC	<b>5</b>
FeedrateOverridePotVoltage		<b>5</b>
SpindleSpeedOverridePotVoltage		<b>5</b>
RemoveRapidRotMov	Removes Rapid Rotary Moves ( Makes All Rapids Feed moves @ max feed)	<b>0</b>
Axis4RadiusCalcMethod	Sets plane to Calculate 4th axis Feedrate on ... 0=XZ 1=YZ 2=XY	<b>1</b>
FeedOverrideType	Specifies the Type of Override used 0=Analog POT 1=4bit POT 98=Use Screen Sliders Only 99=Hardcode to Overrides Max	<b>98</b>
RapidOverrideType		<b>98</b>
SpindleOverrideType		<b>98</b>
MachineXLength	Set X Length of your Machine Travel	<b>60</b>
MachineYLength	Set Y Length of your Machine Travel	<b>30</b>
MachineZLength	Set Z Length of your Machine Travel	<b>30</b>
UseMachWindow	NOT USED (obsolete)	<b>1</b>
DefaultViewPort	Sets Graphics to Initial View Plane 1= Top view	<b>1</b>
CannedCycleInitialPlaneMode	Set to G98	<b>98</b>
OptimizeGCode	Turn on Arc Optimization if =1	<b>0</b>
OptimizationDeviation	If Arc Optimization is on, how close does the arcs follow the original code	<b>0.007</b>
CorneringCheck	Turns On Corner velocity checker. This will look for sharp corners to apply a final velocity to the move.	<b>0</b>
CorneringAngle	If Corner checking is on, this will make the final velocity 10% of the original if over this angle	<b>25</b>
CorneringStopAngle	If Corner checking is on, this will make the final velocity=CornerStopVelocityif over this angle	<b>50</b>
CornerStopVelocity	If Corner checking is on, this will be the final velocity if over the CornerStopAngle	<b>10</b>
CornerAccelerationMult	If Corner checking is on, this multiplier is given to the default acceleration and used in its	<b>0.1</b>
CornerAccelerationDist	NOT USED	<b>0.125</b>
CoreCheckSWK	For API Versions & Dll use only	<b>0</b>
CircleCheck	Works with a G2/G3 command to enable full circle arcs with the same start/end point as the ACR card wont do full circle arcs so this feature is handled in the software to create 2 180 degree arcs to form the circle. Checking is enabled when this parameter =1	<b>0</b>

## Spindle Options

<b>[SpindleOptions]</b>	<b>Description</b>	<b>Default</b>
EnableSpindleFeatures	Enables the spindle Displays on the Screen if = 1	<b>1</b>
SpindleAxis	Points to the Spindle Axis number as wired into the Machine if machine is using a spindle. This parameter set the spindle Dac out and the encode input to the programmed axis.	<b>5</b>
SpindleEncoderInstalled	Set to 1 If you have an Spindle with Encoder on your Machine	<b>1</b>
SpindleDAMaxVoltage	Max Voltage for controller to Output to spindle Drive	<b>9</b>
RigidTappingInstalled	Set to 1 if Machine Has Spindle Encoder for Rigid Tapping	<b>0</b>
OrientBeforeTap	Set to 1 to Force an M19 on every Tap Cycle	<b>1</b>
SpindleGears	Set to The Number of Gears on the Spindle	<b>2</b>
SpindleMaxRPMGearRange0	Sets Max. Speed of Gear Range	<b>1000</b>
SpindleMaxRPMGearRange1	Sets Max. Speed of Gear Range	<b>4000</b>
SpindleMaxRPMGearRange2	Sets Max. Speed of Gear Range	<b>4000</b>
SpindleMaxRPMGearRange3	Sets Max. Speed of Gear Range	<b>4000</b>
SpindleFwdVoltage	Specifies to Invert Spindle voltage for M03 if = 0	<b>1</b>
SpindleRevVoltage	Specifies to Invert Spindle voltage for M04 if = 0	<b>1</b>
SpindleActualSpeedMult	Specifies a Ratio of the Analog Spindle speed feed back to the On screen Display	<b>1</b>
ReverseTappingDirection	Reverses the Spindle tapping direction if the spindle encoder is phased backwards.	<b>0</b>

Tool Changer

<b>[ToolChanger]</b>	<b>Description</b>	<b>Default</b>
ChangerInstalled	Specifies that Machine is Equipped with a ToolChanger	<b>0</b>
NumberOfToolPockets	Specifies Total Number of Pockets in Magazine	<b>24</b>
BiDirectionalMagazine	Set to 1 if Magazine is Bi-directional ... Used for Shortest Path to Tool Lookup	<b>0</b>
VirtualPockets	Set to 1 Virtual Tool Pocket Lookup Strategy is Desired	<b>0</b>
CheckCurrentPocket	Tells the DLL to check a tool change to see if its already in that tool so that it ignores the tool change instruction. This is a Special Parameter Created for a Customer. Normal users will never change this setting	<b>1</b>

Homing Offsets

<b>[HomeOffsets]</b>	<b>Description</b>	<b>Default</b>
0AxisHomeOffset=0.0	Writes the home offset to the ACR Cards global variable P15. This is the Axis0 Move offset distance.	<b>0</b>
1AxisHomeOffset=0.0	Writes the home offset to the ACR Cards global variable P16. This is the Axis1 Move offset distance.	<b>0</b>
2AxisHomeOffset=0.0	Writes the home offset to the ACR Cards global variable P17. This is the Axis2 Move offset distance.	<b>0</b>
3AxisHomeOffset=0.0	Writes the home offset to the ACR Cards global variable P18. This is the Axis3 Move offset distance.	<b>0</b>
4AxisHomeOffset=0.0	Not Used at this Time	<b>0</b>
SpindleOrientOffset=0	Not Used at this Time	<b>0</b>

Tang Options

<b>[TangOption]</b>	<b>Description</b>	<b>Default</b>
TangOn	Enables the software Tangential Feature when = 1	<b>0</b>
TangAxis	Pointer to the Tang Axis	<b>2</b>
TangPathAxis1	Pointer to the first of 2 axis's that tang is used with	<b>0</b>
TangPathAxis2	Pointer to the other axis that tang is used with	<b>1</b>
TangAng	Homed Angle of the Tangential Axis	<b>0</b>
TangMaxVel	Maximum velocity of the Tangential Axis	<b>1000</b>
TangVel	Standard Velocity of the Tangential Axis	<b>1000</b>
TangAcc	Acceleration of the Tangential Axis	<b>10</b>
TangDec	Deceleration of the Tangential Axis	<b>10</b>
TangMaxLimit	Max Orbital Degree of the Axis	<b>0</b>
TangMinLimit	Min Orbital Degree of the Axis	<b>0</b>
TangRewindAng	Angle to goto to Rewind the Axis	<b>0</b>

Speeds and Feeds

<b>[Speeds]</b>	<b>Description</b>	<b>Default</b>
0AxisMaxFeedrate	Not Used at this Time (Uses Velocity)	
1AxisMaxFeedrate	Not Used at this Time	
2AxisMaxFeedrate	Not Used at this Time	
3AxisMaxFeedrate	Not Used at this Time	
4AxisMaxFeedrate	Not Used at this Time	
Acceleration	Sets Machine Acceleration	<b>10</b>
Deceleration	Sets Machine Deceleration ... usually set same as STP	<b>10</b>
Velocity	Sets Machine Max Speed in inches per min.	<b>400</b>
Stp	Sets Machine Deceleration ... usually set same as Acceleration	<b>10</b>
Jerk	Enables the Jerk / S Curve trajectory if =1	<b>0</b>
<b>[JogSpeeds]</b>	<b>Description</b>	<b>Default</b>
Acceleration	Not Used	<b>1</b>
Deceleration	Not Used	<b>1</b>
Velocity	Sets Normal Jog Speeds	<b>100</b>
Stp	Sets Machine Deceleration ... usually set same as Acceleration	<b>1</b>
<b>[FastJogSpeeds]</b>	<b>Description</b>	<b>Default</b>
Acceleration	Not Used	<b>1</b>
Deceleration	Not Used	<b>1</b>
Velocity	Sets Fast Jog Speeds	<b>200</b>
Stp	Sets Machine Deceleration ... usually set same as Acceleration	<b>1</b>

System Resolution (PPU)

<b>[SystemResolution]</b>	<b>Description</b>	<b>Default</b>
0PulsesPerUnit	Encoder Pulses Per Inch of Travel	<b>1000</b>
1PulsesPerUnit	Encoder Pulses Per Inch of Travel	<b>1000</b>
2PulsesPerUnit	Encoder Pulses Per Inch of Travel	<b>1000</b>
3PulsesPerUnit	Encoder Pulses Per Degree of Travel (Rotary Axis)	<b>1000</b>
4PulsesPerUnit	Encoder Pulses Per Inch of Travel	<b>0</b>
5PulsesPerUnit	Encoder Pulses Per Inch of Travel	<b>0</b>
6PulsesPerUnit	Encoder Pulses Per Inch of Travel	<b>0</b>
7PulsesPerUnit	Encoder Pulses Per Inch of Travel	<b>0</b>

In Position Bands

<b>[InPositionBand]</b>	<b>Description</b>	<b>Default</b>
0AxisPB	Sets the tolerance(In Position Band) of a Completed move before another starts	<b>0.01</b>
1AxisPB	Same as above	<b>0.01</b>
2AxisPB	Same as above	<b>0.01</b>
3AxisPB	Same as above	<b>1</b>
4AxisPB	Same as above	<b>1</b>
5AxisPB	Same as above	<b>1</b>
6AxisPB	Same as above	<b>1</b>
7AxisPB	Same as above	<b>1</b>



BackLash Compensation

<b>[BackLashComp]</b>	<b>Description</b>	<b>Default</b>
0AxisBackLash=0.0	Sets Backlash Compensation for the axis	<b>0</b>
1AxisBackLash=0.0	Sets Backlash Compensation for the axis	<b>0</b>
2AxisBackLash=0.0	Sets Backlash Compensation for the axis	<b>0</b>
3AxisBackLash=0.0	Sets Backlash Compensation for the axis	<b>0</b>
4AxisBackLash=0	Sets Backlash Compensation for the axis	<b>0</b>
5AxisBackLash=0	Sets Backlash Compensation for the axis	<b>0</b>
6AxisBackLash=0	Sets Backlash Compensation for the axis	<b>0</b>
7AxisBackLash=0	Sets Backlash Compensation for the axis	<b>0</b>

Gains

<b>[Gains0]</b>	<b>Description</b>	<b>Default</b>
PGAIN=0.002441406	Proportional Gain	<b>0.002</b>
IGAIN=0.001	Integral Gain	<b>0</b>
ILIMIT=0.001	I Limit Delay	<b>0</b>
IDELAY=00.000000000	I Delay	<b>0</b>
DGAIN=00.000000000	Derivative Gain	<b>0</b>
DWIDTH=00.000000000	Derivative Gain Bandwidth	<b>0</b>
FFVEL=00.000000000	Feed Forward Velocity	<b>0</b>
FFACC=00.000000000	Feed Forward Acceleration	<b>0</b>
TLM=10.000000000	Torque Limit in Volts	<b>10</b>

<b>[Gains1]</b>	<b>Description</b>	<b>Default</b>
PGAIN=0.002441406	Proportional Gain	<b>0.002</b>
IGAIN=0.001	Integral Gain	<b>0</b>
ILIMIT=0.001	I Limit Delay	<b>0</b>
IDELAY=00.000000000	I Delay	<b>0</b>
DGAIN=00.000000000	Derivative Gain	<b>0</b>
DWIDTH=00.000000000	Derivative Gain Bandwidth	<b>0</b>
FFVEL=00.000000000	Feed Forward Velocity	<b>0</b>
FFACC=00.000000000	Feed Forward Acceleration	<b>0</b>
TLM=10.000000000	Torque Limit in Volts	<b>10</b>

<b>[Gains2]</b>	<b>Description</b>	<b>Default</b>
PGAIN=0.002441406	Proportional Gain	<b>0.002</b>
IGAIN=0.001	Integral Gain	<b>0</b>
ILIMIT=0.001	I Limit Delay	<b>0</b>
IDELAY=00.000000000	I Delay	<b>0</b>
DGAIN=00.000000000	Derivative Gain	<b>0</b>
DWIDTH=00.000000000	Derivative Gain Bandwidth	<b>0</b>
FFVEL=00.000000000	Feed Forward Velocity	<b>0</b>
FFACC=00.000000000	Feed Forward Acceleration	<b>0</b>
TLM=10.000000000	Torque Limit in Volts	<b>10</b>

<b>[Gains3]</b>	<b>Description</b>	<b>Default</b>
PGAIN=0.002441406	Proportional Gain	<b>0.002</b>
IGAIN=0.001	Integral Gain	<b>0</b>
ILIMIT=0.001	I Limit Delay	<b>0</b>
IDELAY=00.000000000	I Delay	<b>0</b>
DGAIN=00.000000000	Derivative Gain	<b>0</b>
DWIDTH=00.000000000	Derivative Gain Bandwidth	<b>0</b>
FFVEL=00.000000000	Feed Forward Velocity	<b>0</b>
FFACC=00.000000000	Feed Forward Acceleration	<b>0</b>
TLM=10.000000000	Torque Limit in Volts	<b>10</b>

<b>[Gains4]</b>	<b>Description</b>	<b>Default</b>
PGAIN=00.002441406	Proportional Gain	<b>0.002</b>
IGAIN=00.000000000	Integral Gain	<b>0</b>
ILIMIT=00.000000000	I Limit Delay	<b>0</b>
IDELAY=00.000000000	I Delay	<b>0</b>
DGAIN=00.000000000	Derivative Gain	<b>0</b>
DWIDTH=00.000000000	Derivative Gain Bandwidth	<b>0</b>
FFVEL=00.000000000	Feed Forward Velocity	<b>0</b>
FFACC=00.000000000	Feed Forward Acceleration	<b>0</b>
TLM=10.000000000	Torque Limit in Volts	<b>10</b>

<b>Spindle Axis [Gains5]</b>	<b>Description</b>	<b>Default</b>
PGAIN=0.0004	Proportional Gain	<b>0.002</b>
IGAIN=2.0	Integral Gain	<b>0</b>
ILIMIT=0.009	I Limit Delay	<b>0</b>
IDELAY=0	I Delay	<b>0</b>
DGAIN=0.00005	Derivative Gain	<b>0</b>
DWIDTH=0.0001	Derivative Gain Bandwidth	<b>0</b>
FFVEL=0	Feed Forward Velocity	<b>0</b>
FFACC=0	Feed Forward Acceleration	<b>0</b>
TLM=10.000000000	Torque Limit in Volts	<b>10</b>

Auxiliary Mcodes

<b>[Aux Functions]</b>	<b>Description</b>	<b>Default</b>
Aux1Text=[M101] HP/WashDown On	User Aux Function Description Text	
Aux1Mcode=2021	User Aux Function MCode Bit	<b>2021</b>
Aux2Text=[M102] HP/WashDown Off		
Aux2Mcode=2022		<b>2022</b>
Aux3Text=[M103] Chip Conveyer On		
Aux3Mcode=2023		<b>2023</b>
Aux4Text=[M104] Chip Conveyer Off		
Aux4Mcode=2024		<b>2024</b>
Aux5Text=User5		
Aux5Mcode=0		
Aux6Text=User6		
Aux6Mcode=0		
Aux7Text=User7		
Aux7Mcode=0		
Aux8Text=User8		
Aux8Mcode=0		
Aux9Text=User9		
Aux9Mcode=0		
Aux10Text=User10		
Aux10Mcode=0		

Mcodes Validated

<b>[MCODES]</b>	<b>Description ( only 11 of 128 shown )</b>	<b>Default</b>
M0=1	Enable MCode MXX (1=Enabled 0=Disabled)	<b>1</b>
DESC0=OPTIONAL PROGRAM STOP	MCode Description Text	
M1=1		<b>1</b>
DESC1=PROGRAM STOP		
M2=1		<b>1</b>
DESC2=PROGRAM STOP		
M3=1		<b>1</b>
DESC3=SPINDLE FWD		
M4=1		<b>1</b>
DESC4=SPINDLE REV		
M5=1		<b>1</b>
DESC5=SPINDLE STOP		
M6=1		
DESC6=TOOL CHANGE		
M7=0		
DESC7=		
M8=1		
DESC8=FLOOD COOLANT ON		
M9=1		
DESC9=ALL COOLANTS OFF		
M10=1		
DESC10=LOCK 4TH AXIS		
M11=1		
DESC11=UNLOCK 4TH AXIS		

## The Gains Editor

The Gains editor allows the user to tune the axis while a program is running.

It also has a Combined Tuning Function that sets the Gains of 2 Axis's at the same time.

Typically the user will have the Following Error displayed in the Readouts while tuning.

### Tuning a single axis

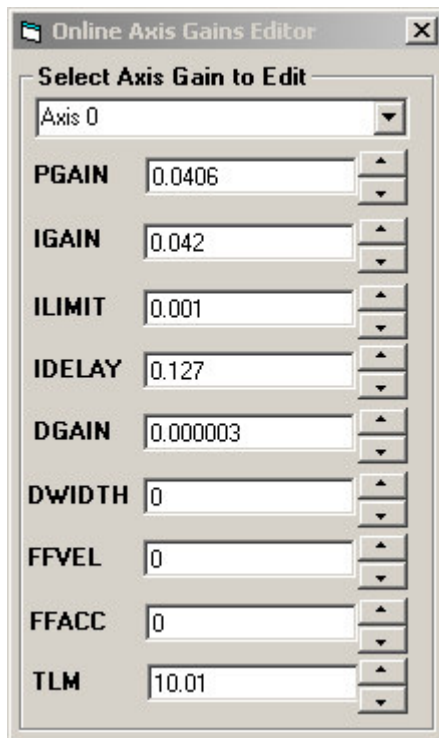
Select the single Axis to Tune from the drop down list.

Manually edit and press Enter or raise or lower Gain Value with Up/Down Pointer.

### Tuning 2 Axis Simultaneously

Select the combined Axis to Tune from the drop down list.

Manually edit and press Enter or raise or lower Gain Value with Up/Down Pointer.



## G Code Modal Status Screen

This screen displays the G Code processor Modal Status as a Gcode Program is running.

<b>G1</b>	<b>Movement Modes</b>	<b>Close</b>
<b>G17</b>	<b>Arc Plane Selection</b>	
<b>G90</b>	<b>Coordinate Mode</b>	
<b>G94</b>	<b>Feed Type</b>	
<b>G20</b>	<b>Units...Inches</b>	
<b>G40</b>	<b>CutterComp</b>	
<b>G49</b>	<b>Tool Length Compensation</b>	
<b>G80</b>	<b>Canned cycles...G7x - G8x</b>	
<b>0</b>	<b>Reserved</b>	
<b>G50</b>	<b>Scaling</b>	
<b>G97</b>	<b>Spindle Mode</b>	
<b>0</b>	<b>Reserved</b>	
<b>G64</b>	<b>Contouring Mode</b>	
<b>G69</b>	<b>Coordinate Rotations</b>	

## Validating MCodes

This Screen Displays all Valid MCodes Programmed in the AcroBasic Software.

The Purpose of this List is to: Assure that any MCode encountered in the Gcode Program has been Validated by the Machine Builder.

If a MCode is Not Found in this list at runtime the Program will Error the User that a Invalid MCode was encountered" and Stop.

The Machine Builder will Edit the Machine Parameter Section MCodes as Shown Below. To Enable any of the possible 128 MCodes that the System recognizes, Edit via the Parameters Section MCodes"

Example Entries.

- A Mxx=0 entry Disables the MCode.
- A MXX=1 entry Enables the MCode.
- DescX Relates to the Corresponding MCode.
- MCode Description can be Anything.

Actual MCodes" Section Excerpt

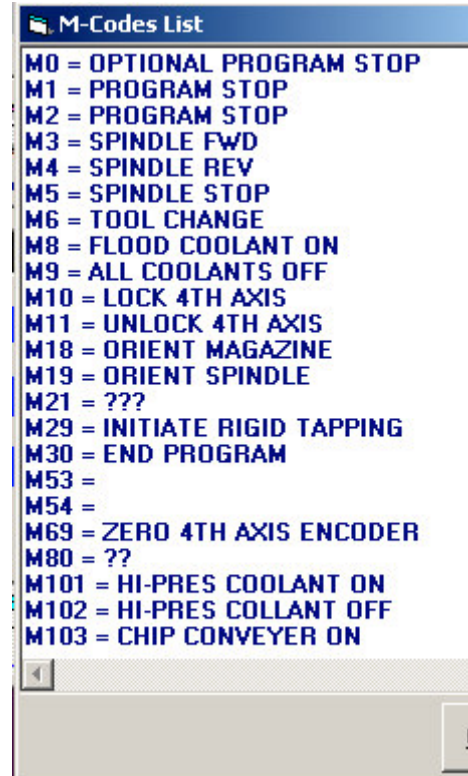
[MCODES]

M0=1

DESC0=OPTIONAL PROGRAM STOP

M1=1

DESC1=PROGRAM STOP





## How to Configure User Defined MCodes

User Definable MCodes are 10 User Definable Buttons on the Screen that Execute Mcode subroutines in AcroBasic that can be activated at runtime or anytime in a cycle.

For example, your InCycle and you want to activate your Chip conveyer but you don't want to stop the Cycle. You simply bring up this menu and click 1 of the 10 buttons as shown on the right.

The Mcode functions themselves are in prog0, these buttons cause the control to execute the Mcode function you told it to as shown below.

[M101] HP/WashDown On
[M102] HP/WashDown Off
[M103] Chip Conveyer On
[M104] Chip Conveyer Off
User5
User6
User7
User8
User9
User10

### Defining a User Definable MCode Button

Example: Setting Aux1 function.

- 1) Assign the MCode Bit in Aux1Mcode = 2021 ... this sets the MCode to Execute a M101
- 2) Assign the Button Text by setting the Aux1Text = [M101] HP/WashDown On
- 3) That's all there is to it when you restart the program and click on the First button it will Execute a M101 and set the M Strobe Signal (Bit 150) Automatically.
- 4) Don't forget to place your code handler in the AcroBasic M101 Section to actually do the Work!

Parameters.Cfg File Excerpt [Aux\_Functions]:

[Aux\_Functions]

Aux1Text=[M101] HP/WashDown On

Aux1Mcode=2021

Aux2Text=[M102] HP/WashDown Off

Aux2Mcode=2022

Aux3Text=User 3 Chip Conveyer On

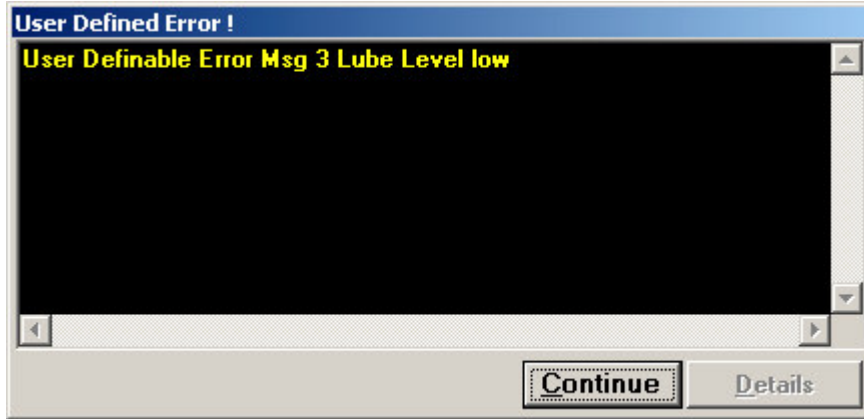
Aux3Mcode=2023

Aux4Text=User 4 Chip Conveyer Off

Aux4Mcode=2024

## Configuring User Defined Error Messages

This Software System Contains a Text File called UserDefined.err that Contains Error Messages to be sent to the Control Display. These Messages are triggered in the AcroBasic and displayed on the Software Display.



The User Error System Works by Setting the Desired User Error Code Binary Bit Code, and then Setting Bit (250) the User Error Request bit, to Signal the Control System to Decode the message and display it.

When the system finds Bit 250 =1 it will decode the 5 User Error bits and Display the appropriate Error Message on the Screen.

Example: To set a lube Alarm User Error Message.

The Lube Alarm Message is at Position 3 in the UserDefined.err file, so we need to set the Error Bit Code mask to Binary 3 (0011). The program will load the string in position 3 of the 0-based indexed file for display.

### UserDefined.err File Sample

User Definable Error MSG0	Position 0
User Definable Error MSG1	Position 1
User Definable Error MSG2	Position 2
User Definable Error MSG3	Position 3
User Definable Error MSG4	Position 4
User Definable Error MSG5 to 31 not shown	

This procedure to actually trigger the Error as follows:

Set Bit 251&252 = 0011 or binary 3

Then Set Bit 250 ... this triggers the message to be displayed on the screen.

The user will click Ok and the Error Display will go away.

Note: If the source of the Error still exists the message will popup again.

Error Code Binary Mask from the Control Signals Bits sample.

Bit 250	User Error Request	Triggers Error to display
Bit 251	User Error bit 0	Triggers Error to display
Bit 252	User Error bit 1	Triggers Error to display
Bit 253	User Error bit 2	Triggers Error to display
Bit 254	User Error bit 3	Triggers Error to display
Bit 255	User Error bit 4	Triggers Error to display

## How to Setup the ToolChanger

Parameters to be set by Machine Builder:

[ToolChanger]

ChangerInstalled=1	Set to 1 if CNC has a Tool Changer
NumberOfToolPockets=24 Magazine	Set to Number of Tool Positions of the CNC Tool Changer
BiDirectionalMagazine=0	Set to 1 if Magazine can Move forward and Backward
VirtualPockets=1	Set to 1 on Machines capable of using Virtual Tool Pockets

Tool Changer Types:

Type 1 (CNC with No Tool Changer)

If CNC has no Tool Changer assign your tools normally, and the system will prompt the User to put the new Tool in the Spindle” on a M6. Next the Message Click ok to Continue” will be posted allowing the user to change the tool manually and Clicking OK to continue running Gcode program.

Type 2 (Standard Tool Changer Configured)

If using a standard Tool Changer simply match the pocket with the same tool number in the Tool Offsets Screen.

Example:

Tool 1 = Pocket 1

Tool 2 = Pocket 2

Type 3 (Virtual Tool Pocket Lookup Changer Configured)

Virtual Tool Pocket Tool Change Sequence:

Example: Tool 10 currently in Spindle.

M06 T5 is executed.

The System will find which pocket Tool 5 is in, and index the magazine to that pocket.

Next the M6 will swap the Spindle Tool (T10) with T5, and assign Tool 10 in Tool 5's pocket location.

A Zero in a Pocket means it has no Tool Assigned to it.

You can't assign a Tool in a Pocket Greater Than the # of Actual tool Pocket Positions assigned in the Machine Parameters Section (ToolChanger).

## Virtual Pendant Setup

If Parameter HandwheelInstalled=0 ” then the system will enable this section for Machines not equipped with a remote pendant.

Parameter Used:

Section Name = CNC\_Config”      Parameter Name HandWheelInstalled”

To Use:

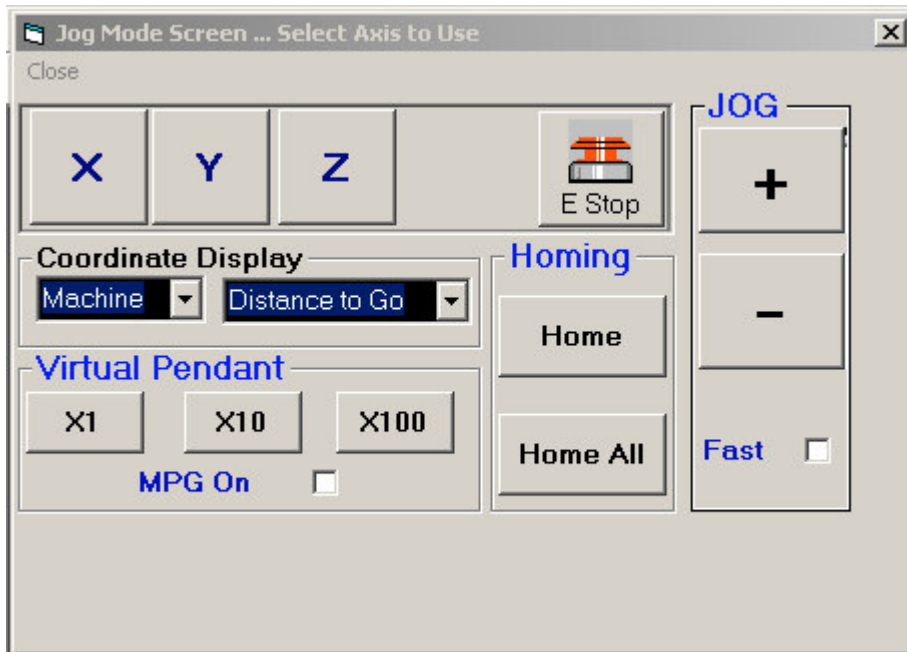
Switch to Jog or Offsets Mode

Select MPG ON” to Enable.

Select the Axis Letter.

Select (X1) (X10) (X100) for .001/ .010 / .100 increment moves on the Selected Axis.

Press & Release the + or – buttons to move in the Selected Move Increment.



## HandWheel Setup

How to Set the Remote Pendants Mpg (Manual Pulse Generator) Resolutions for X1 X10 X100 Modes.

### Background Information:

When the User clicks the Axis Mode Selector Switch of the Pendant the system will call 1 of 3 Subroutines in program 0 to set the Hand Wheel Gearing Ratio.

Gearing Ratio is as follows:

line 50 for X1 ( Move .0001 per Handwheel click)

line 60 for X10 ( Move .001 per Handwheel click)

line 70 for X100 ( Move .010 per Handwheel click)

To set MPG resolution Example:

1. Open the AcroBasic program 0 using AcroView.

2. Edit Program 0 as shown

Target = .0001 per Click

Calculate Gearing Ratio as follows:

Axis X PPU/Target Distance

Axis X PPU = 15673 ... Target move Distance = .0001

Axis X PPU/Target Distance = 1.5673

Plug the 1.5673 Value into program 0 @ lines 50,51,52 of prog0 as the excerpt code below shows.

Next repeat Step 2 for X10 @ Starting on line 60

And repeat for X100 @ Starting on line 70

```
REM SET FOR .0001 RESOLUTION
```

```
50 GEAR RATIO X1.5673 :REM *** [ SET HANDWHEEL PPU Ratio FOR X Axis ]
```

```
51 GEAR RATIO Y1.5673 : REM *** [ SET HANDWHEEL PPU Ratio FOR Y Axis ]
```

```
52 GEAR RATIO Z1.5673 : REM *** [ SET HANDWHEEL PPU Ratio FOR Z Axis ]
```

```
53: REM SAVE FOR 4TH AXIS FUTURE USE
```

54: REM SAVE FOR 5TH AXIS FUTRUE USE

55 GOTO 99



## Configuring a Tangential Axis

This option activates the axis tangent feature. TANG is useful for positioning an axis relative to the motion path. When an axis is put in TANG mode, it will automatically move as the motion path changes.

Because the TANG axis is in degrees instead of the linear axis, it must have its own Acceleration, Deceleration and Velocity.

Before each movement, both linear and circular, the TANG axis is rotated in the direction of movement. If a circular movement is inacted, the TANG axis will rotate with the linear motion in a coordinated movement, keeping the TANG axis pointing in the direction of the circular motion.

If the TANG axis cannot rotate infinitely, the Max and Min limits are observed...if a limit is going to be exceeded, then the rewind angle is first issued. If the TangMaxLimit and the TangMinLimit are both set to zero, no limit is used.

See [Tang Options](#)

[TangOption]

TangOn=0

TangAxis=2

TangPathAxis1=0

TangPathAxis2=1

TangAng=0

TangMaxVel=60000

TangVel=60000

TangAcc=100000

TangDec=100000

TangMaxLimit=0

TangMinLimit=0

TangRewindAng=0

## Configuring a Cammed Axis

Camming is typically done with a Laser or WaterJet type Machine. The Point is to run the laser power proportional the Combined total velocity of the coordinated axis movements of Axis X & Y. This will make the laser output proportional to the Axis Velocity.

The following is and Actual working example of how to accomplish Camming a Laser to the Vector velocity.

Step 1. Add the following code to the top of program 1 in AcroBasic as shown.

Rem :Program 1 Example of Camming a Laser wired into the DAC of Axis 3 to the Vector Velocity.

ATTACH MASTER1

ATTACH SLAVE0 AXIS3 "L"

1 PBOOT : REM Set Program AutoStart on Boot

2 PON : REM TURNS ON PLC

3 GOSUB 50000 : REM INITIALIZE CAMING PARAMETERS

Step 2. Add the following Subroutine to the bottom of Program 1.

```
REM *****  
REM ** SETUP FOR CAMMING OF LASER TO VELOCITY **  
REM *****  
50000 DIM LA (2)  
50010 DIM LA0 (2)  
50020 LA0 (0) = 0  
50030 LA0 (1) = 400  
50040 DIM LA1 (11)  
50050 LA1 (0) = 400: REM initial value above 25/1016  
50060 LA1 (1) = 1000: REM value at 20320/10 counts or 2" second  
50070 LA1 (2) = 4000: REM value at 4" second  
50080 LA1 (3) = 6000: REM value at 6" second  
50090 LA1 (4) = 8000: REM value at 8" second  
50100 LA1 (5) = 10000: REM value at 10" second  
50110 LA1 (6) = 10000
```

50120 LA1 (7) = 10000  
50130 LA1 (8) = 10000  
50140 LA1 (9) = 10000  
50150 LA1 (10) =0  
50160 CAM DIM L2  
50170 CAM SEG L (0,1000,LA0)  
50180 CAM SEG L (1,20320,LA1)  
50190 CAM SRC L P8215: SET CAM SOURCE FROM MASTER VELOCITY SCALER IN PROG 2  
LINE 10  
50195 CAM SHIFT L 0  
50200 RES L  
50210 CAM SCALE L -1.0: REM reverse the voltage output because couldn't switch wire because  
of grounding problems  
50220 SET 51: REM turn of nitrogen  
50230 CLR 50: REM reset bed shift commands  
50240 CLR 49: REM reset bed shift commands  
50260 RETURN

Step 3. PASTE THE FOLLOWING INTO PROGRAM 2.

REM STORE MASTER 0 VECTOR VELOCITY TO DUMMY ADDRESS 8215  
REM FOR CAMMING CALCULATION SCALE  
1 PBOOT  
10 P8215=P8193\*1016.0  
20 GOTO 10

## Using the GCode Optimizer

OptimizeGCode

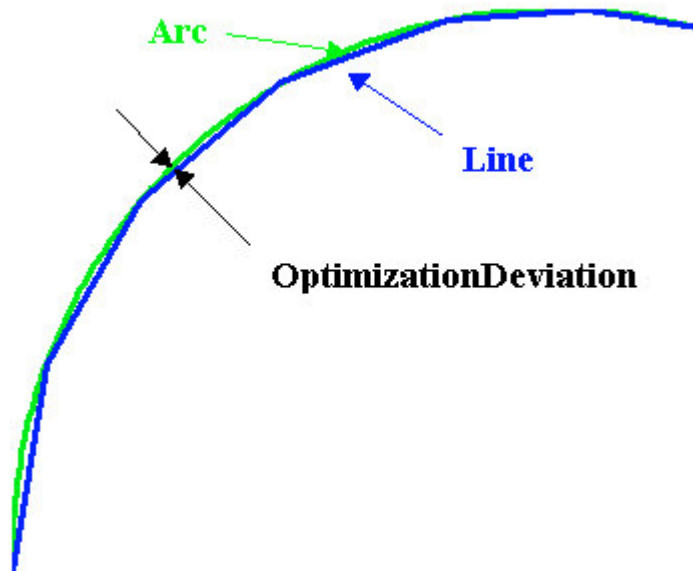
[SoftwareOptions]

OptimizeGCode=0

OptimizationDeviation=0.003

OptimizationDeviation

This option in the [SoftwareOptions] attempts to convert small line segments into smooth Arcs. This is primarily done to reduce the number of blocks per second required during the execution of the G Code program to the ACR motion card. The OptimizationDeviation is the amount the new ARC can deviate +/- from the original line segments.



Corner Checking

[SoftwareOptions]

CornerRadius=0

CorneringAngle=10

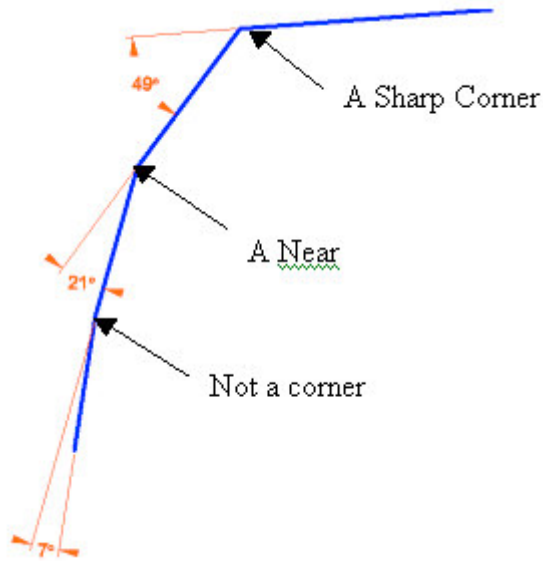
CorneringStopAngle=40

CornerStopVelocity=10.0

NearCornerStopVelocity=30.0

This option in the [SoftwareOptions] looks ahead in the path of motion to determine a final velocity (FVEL) of a movement. Unlike most look ahead features, this feature looks ahead through the entire file at the time it is loaded. Therefore, there is no limit to the look ahead.

This feature is turned on by setting the `CorneringCheck=1`. Any change in direction of motion in excess of the `CorneringAngle` is evaluated. If the angle of change is greater than `CorneringAngle` but less than the `CorneringStopAngle`, the final velocity is set as the `NearCornerVelocity`. Only when the change in direction exceeds the `CorneringStopAngle` will the final velocity be set at the `CornerStopVelocity`.



Machine Control Signal Bits 128 to 255

These signals are the core of the Control System, they are Bi-directional from the software and AcroBasic.

Do not use any of these signals in your AcroBasic programs without fully understanding what they do.

The Software uses these bits to Control and Monitor All Machine functions.

ACR Bit	API Index	Description	ACR Bit	API Index	Description	ACR Bit	API Index	Description	ACR Bit	API Index	Description
128	0	CYCLESTART	160	32	MPG ENABLED	192	64	OverTravel Active	224	96	User Scratch Bit Area
129	1	ESTOP	161	33	MPG X SELECTED	193	65	Enable Virtual Estop	225	97	User Scratch Bit Area
130	2	RESET	162	34	MPG Y SELECTED	194	66	VirtualEstopActive	226	98	User Scratch Bit Area
131	3	FEEDHOLD	163	35	MPG Z SELECTED	195	67	VirtualEstopSealbit	227	99	User Scratch Bit Area
132	4	DRYRUN MODE	164	36	MPG 4 SELECTED	196	68	TangOn/Off 1=On	228	100	User Scratch Bit Area
133	5	Auto/Step Mode [Auto=1]	165	37	MPG 5 SELECTED	197	69	Open for Future Use	229	101	User Scratch Bit Area
134	6	RunMode	166	38	X.1 MODE SELECTED	198	70	Open for Future Use	230	102	User Scratch Bit Area
135	7	JOG MODE	167	39	X1 MODE SELECTED	199	71	Open for Future Use	231	103	User Scratch Bit Area
136	8	MDI MODE	168	40	X10 MODE SELECTED	200	72	Open for Future Use	232	104	User Scratch Bit Area
137	9	Offsets Mode	169	41	X100 MODE SELECTED	201	73	Open for Future Use	233	105	User Scratch Bit Area
138	10	INCYCLE	170	42	X Home Done	202	74	Program Running	234	106	User Scratch Bit Area
139	11	Edit Mode Active	171	43	Y Home Done	203	75	Program Modified	235	107	User Scratch Bit Area
140	12	Jogged in Cycle	172	44	Z Home Done	204	76	Open for Future Use	236	108	User Scratch Bit Area
141	13	Return to PreJog Pos Active	173	45	4th Axis Home	205	77	Open for Future Use	237	109	User Scratch Bit Area
142	14	Optional Stop Active	174	46	5th Axis Home	206	78	Open for Future Use	238	110	User Scratch Bit Area
143	15	Control Ready Signal bit	175	47	MAGREFDONE	207	79	Open for Future Use	239	111	User Scratch Bit Area
144	16	HOMING ACTIVE	176	48	GEARCHANGEACTIVE	208	80	Open for Future Use	240	112	User Scratch Bit Area
145	17	Control Initialized Bit	177	49	SGEARRANGE0	209	81	Open for Future Use	241	113	User Scratch Bit Area
146	18	SpindleEncoderInstalled	178	50	SGEARRANGE1	210	82	Open for Future Use	242	114	User Scratch Bit Area
147	19	MDONE	179	51	SGEARRANGE2	211	83	Open for Future Use	243	115	User Scratch Bit Area
148	20	SDONE	180	52	Spindle Direction Bit 0-1	212	84	Open for Future Use	244	116	User Scratch Bit Area
149	21	TDONE	181	53	ZeroSpeedArrival	213	85	Digital_Fov Bit 0	245	117	User Scratch Bit Area
150	22	M STROBE	182	54	SpeedArrival	214	86	Digital_Fov Bit 1	246	118	User Scratch Bit Area
151	23	S STROBE	183	55	OrientComplete	215	87	Digital_Fov Bit 2	247	119	User Scratch Bit Area
152	24	T STROBE	184	56	Spindle Enable	216	88	Digital_Fov Bit 3	248	120	User Scratch Bit Area
153	25	TOOLCHANGEACTIVE	185	57	Rigid Tapping Active	217	89	Digital_Rov Bit 0	249	121	ERROR MSG PENDING
154	26	TOOLCHANGEDONE	186	58	Tapping Active	218	90	Digital_Rov Bit 1	250	122	User Error Request
155	27	TOOLATCMDPOS	187	59	4thAxisPresent	219	91	Digital_Rov Bit 2	251	123	User Error Bit1
156	28	TOOLSEEKACTIVE	188	60	4thAxisClamped	220	92	Digital_Rov Bit 3	252	124	User Error Bit2
157	29	TOOLSEEKDONE	189	61	Contouring	221	93	Digital_Sov Bit 0	253	125	User Error Bit3
158	30	TOOLSEEKDIR	190	62	SpindleRunning	222	94	Digital_Sov Bit 1	254	126	User Error Bit4
159	31	TOOL Change Request	191	63	OrientBeforeTap	223	95	Digital_Sov Bit 2	255	127	User Error Bit5

M Code Bits 1920-2047

These signals are set when a M Code is Executed by the Control System. When an M Code is encountered in a G code Program, The Appropriate MCode bit is set=1 along with the MStrobe Bit 150. The Parser function in prog1 looks at which MCode bit is = 1 and calls the associated M code handle Function.

Example: an M08 is executed.

The system will set bit 1928 and then Bit 150, when prog1 sees 150 go high it will look to see which MCode bit is high and then call the M08 MCode function and return out clearing bit 1928 and 150.

ACR Bit	API Index	Description	ACR Bit	API Index	Description	ACR Bit	API Index	Description	ACR Bit	API Index	Description
1920	0	M00	1952	32	M32	1984	64	M64	2016	96	M96
1921	1	M01	1953	33	M33	1985	65	M65	2017	97	M97
1922	2	M02	1954	34	M34	1986	66	M66	2018	98	M98
1923	3	M03	1955	35	M35	1987	67	M67	2019	99	M99
1924	4	M04	1956	36	M36	1988	68	M68	2020	100	M100
1925	5	M05	1957	37	M37	1989	69	M69	2021	101	M101
1926	6	M06	1958	38	M38	1990	70	M70	2022	102	M102
1927	7	M07	1959	39	M39	1991	71	M71	2023	103	M103
1928	8	M08	1960	40	M40	1992	72	M72	2024	104	M104
1929	9	M09	1961	41	M41	1993	73	M73	2025	105	M105
1930	10	M10	1962	42	M42	1994	74	M74	2026	106	M106
1931	11	M11	1963	43	M43	1995	75	M75	2027	107	M107
1932	12	M12	1964	44	M44	1996	76	M76	2028	108	M108
1933	13	M13	1965	45	M45	1997	77	M77	2029	109	M109
1934	14	M14	1966	46	M46	1998	78	M78	2030	110	M110
1935	15	M15	1967	47	M47	1999	79	M79	2031	111	M111
1936	16	M16	1968	48	M48	2000	80	M80	2032	112	M112
1937	17	M17	1969	49	M49	2001	81	M81	2033	113	M113
1938	18	M18	1970	50	M50	2002	82	M82	2034	114	M114
1939	19	M19	1971	51	M51	2003	83	M83	2035	115	M115
1940	20	M20	1972	52	M52	2004	84	M84	2036	116	M116
1941	21	M21	1973	53	M53	2005	85	M85	2037	117	M117
1942	22	M22	1974	54	M54	2006	86	M86	2038	118	M118
1943	23	M23	1975	55	M55	2007	87	M87	2039	119	M119
1944	24	M24	1976	56	M56	2008	88	M88	2040	120	M120
1945	25	M25	1977	57	M57	2009	89	M89	2041	121	M121
1946	26	M26	1978	58	M58	2010	90	M90	2042	122	M122
1947	27	M27	1979	59	M59	2011	91	M91	2043	123	M123
1948	28	M28	1980	60	M60	2012	92	M92	2044	124	M124
1949	29	M29	1981	61	M61	2013	93	M93	2045	125	M125
1950	30	M30	1982	62	M62	2014	94	M94	2046	126	M126
1951	31	M31	1983	63	M63	2015	95	M95	2047	127	M127

Input and Output Map

Important! If you're using an ISA type Card only Inputs 0-15 and outputs 0 to 15 are available.

This is because cards like the ACR-2000 on have 16 inputs and 16 outputs.

All PCI type controllers have the base 32 inputs and 32 outputs.

Inputs Map

ACR Bit	API Index	DEFAULT I/O MODULE Description	ACR Bit	API Index	I/O MODULE 1 Description
0	0		256	32	
1	1		257	33	
2	2		258	34	
3	3		259	35	
4	4		260	36	
5	5		261	37	
6	6		262	38	
7	7		263	39	
8	8		264	40	
9	9		265	41	
10	10		266	42	
11	11		267	43	
12	12		268	44	
13	13		269	45	
14	14		270	46	
15	15		271	47	
16	16		272	48	
17	17		273	49	
18	18		274	50	
19	19		275	51	
20	20		276	52	
21	21		277	53	
22	22		278	54	
23	23		279	55	
24	24		280	56	
25	25		281	57	
26	26		282	58	
27	27		283	59	
28	28		284	60	
29	29		285	61	
30	30		286	62	
31	31		287	63	

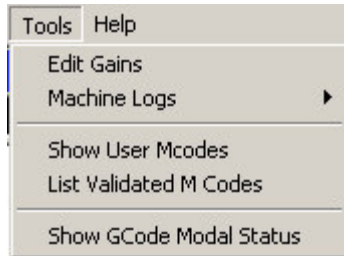


Outputs Map

ACR	API	DEFAULT I/O MODULE	ACR	API	I/O MODULE 1
Bit	Index	Description	Bit	Index	Description
32	0		288	32	
33	1		289	33	
34	2		290	34	
35	3		291	35	
36	4		292	36	
37	5		293	37	
38	6		294	38	
39	7		295	39	
40	8		296	40	
41	9		297	41	
42	10		298	42	
43	11		299	43	
44	12		300	44	
45	13		301	45	
46	14		302	46	
47	15		303	47	
48	16		304	48	
49	17		305	49	
50	18		306	50	
51	19		307	51	
52	20		308	52	
53	21		309	53	
54	22		310	54	
55	23		311	55	
56	24		312	56	
57	25		313	57	
58	26		314	58	
59	27		315	59	
60	28		316	60	
61	29		317	61	
62	30		318	62	
63	31		319	63	

## System Log Files

The control system is equipped with a logging system that records what the user has done while running the software to help in precise trouble shooting of problems. The user may be asked to email these log files to Tec-support to quickly solve problems as they arise. The 2 Main log files are named Movelog.txt and user.txt they are located in the Parfiles” subfolder where the software is installed.



The Tool Menu Displays the following Log Fly-out Menu items.

Show Move Log... Displays the moved coordinates of the current G code program.

Show Machine Operation Log ... Displays a log of all actions the user made in the software.

## System Diagnostics Screen

The diagnostics Screen may be open while in cycle to assist you in trouble shooting what is occurring in the I/O System/Control Signals/ Master flags/ Mcode bits. Tech Support may have you open this screen to assist in solving the problem on the phone.

Key files to aid in troubleshooting. Tec Support may ask you to email any or all of these files.

These files are all in the Parfiles subfolder of the program.

Movelog.txt ... shows us what the controller set to the motion card in moves and M/S/T codes.

User.txt ... shows a log of what the user clicked and any errors sent to the user.

The AcroBasic Programs Prog0.8k Prog1.8k Sys.8k Gains.8k

The Parameters file (Parameters.Cfg) ... this file contains all the Machine Parameters.

## Technical Support

Parker Automation

ACR-MOTIONMAX Technical Support

5500 Business Park Drive  
Rohnert Park, CA 94928  
800-358-9070  
[emn\\_support@parker.com](mailto:emn_support@parker.com)

Key files to aid in troubleshooting.

Technical Support may ask you to email any or all of these files.

These files are all in the Parfiles subfolder of the program.

Movelog.txt ... shows us what the controller set to the motion card in moves and M/S/T codes.

User.txt ... shows a log of what the user clicked and any errors sent to the user.

The AcroBasic Programs Prog0.8k Prog1.8k Sys.8k Gains.8k

The Parameters file (Parameters.Cfg) ... this file contains all the Machine Parameters.

