AcroMill

Cutting Control

Version 2.00.00 (07/01/2000)

User's Guide

ACROLOOP MOTION CONTROL SYSTEM'S, INC.
3650 CHESTNUT STREET NORTH
CHASKA, MN 55318
PH: 952-448-9800 FAX: 952-448-9321

Table of Contents

Using This Manual

The purpose of the AcroMill *User's Guide* is to provide an easy to use supplement to the AcroMill software. It is recommended that the user take a few minutes reading the first four sections before installing AcroMill.

AcroMill software is composed of up to 2 layers of menu commands. The software is Windows based and provides a powerful data base of commands to fit your specific requirements. The software is designed to work with a standard keyboard, mouse and touch screen. To facilitate the touch screen, the menus are designed to have large pushbutton selections allowing easy finger control. Data entry follows standard Windows interface practices.

As you familiarize yourself with the software, use the *User's Guide* manual and the software HELP screens for assistance.

If you have questions about AcroMill commands, or would like to make suggestions on improving the *User's Guide*, please call or e-mail Acroloop at:

Phone: (952)448-9800 Fax: 952-448-9321
Web site: www.acroloop.com
Technical Support: techsupport@acroloop.com

-

AcroMill Overview

AcroMill is a universal software package designed for its flexibility and ease of use for controlling 2-8 axis of motion in CNC milling operations. It features a built-in EIA-RS-274D interface, 3D graphics for part verification, cutter radius compensation and High performance like NURBS, SPLINES, 3D ARCS and multi block lookahead for high speed machining. Not all of these features however are accessable with all Acroloop controllers. Some of the less expensive controllers like the ACR1500 are not able to do NURBS, or LOOKAHEAD. AcroMill comes equipped with on-board DXF interfaces, built-in Part Libraries, and a user configurable Alarm Screen. A high speed multitasking PLC can be set up to be used in conjunction with AcroMill and it runs simultaneously in the background with no sacrifice to performance. AcroMill also allows you to program most I/O functions by simply selecting or specifying the I/O. Being a Windows application, AcroMill also allows OEM users to run independent applications in parallel and have those applications share data with AcroMill via the Parameter Storage area hence forth referred to as “ SYSVAR storage”. The SYSVAR storage is a data base that can be simultaneously accessed by AcroMill and independent OEM applications.

AcroMill software pushes the technological envelope and allows you to change system parameters as your requirements dictate. For example, the software can be used to set the parameters of a servo loop such as integral and proportional gains to tune the motors. A handy four channel oscilloscope is built-in to the AcroMill software to allow *on-the-fly* diagnostics of the parameters.

Installing AcroMill

System requirement
Windows NT 4.0 Service Pack 4 and above.
Pentium II 450 or better CPU.
At least 64 Meg of RAM.
At least 1 Gbyte Harddrive.
Controller (ACR1500, ACR2000, ACR8010 or ACR8020) plugged into the PC bus and addressed as Card 0 with Firmware Version 1.18 or later.
At least 1 available PC Interrupt (Usually Int5).
800x600 SVGA PCI graphics controller.

Before installing AcroMill software, install the SDK drivers for the operating system. Part of the SDK installation will ask you for the type of controller installed. AcroMill NT will work with all PC based controllers like the ACR1500, ACR2000, ACR8010, ACR8020 as well as AcroWire IEEE 1394 based chassis containing standalone controllers like the ACR1200, ACR2000, ACR8010.

AcroMill NT software (v2.00) comes on one CD ROM disk . To install AcroMill simply insert the CD rom into the drive and it will AUTORUN. Follow the prompts. When asked what type of controller is installed, select “NONE”.
If the CD does not AUTORUN, run the SETUP.EXE file on the CD.

.

Before Starting AcroMill

This section is a preview of the AcroMill display screens when AcroMill is running. Each status window and dialog box are explained in detail. Become familiarized with the status windows and dialog prompts before running AcroMill. You will find this section is a handy reference for understanding and locating the status information when running programs.

What's on the Screen

Figure 1 shows a typical screen of AcroMill.

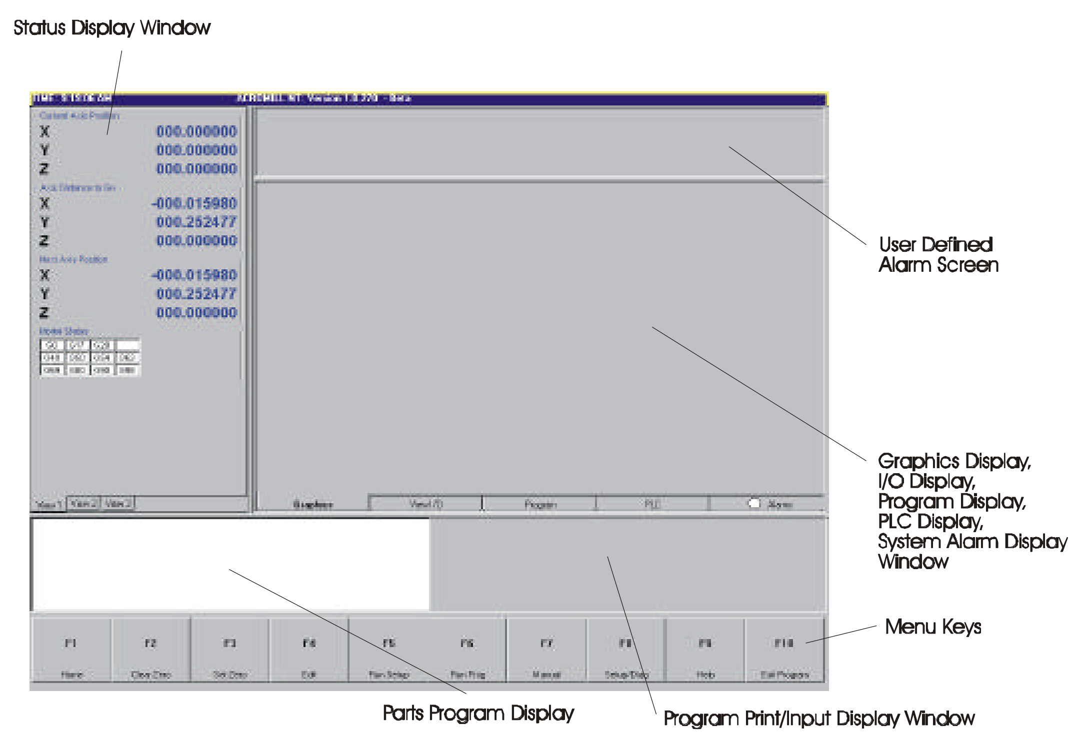


Figure 1. A Typical Screen of AcroMill

The Status Display Window shows one of three selectable VIEWS. The different views offer the following information.

CURRENT POSITION
DISTANCE TO GO
NEXT POSITION
MODAL STATUS
ESTOP
FEEDRATES
VIRTUAL PUSHBUTTON PANEL

These can be selected via a tabbed control at any time.

Program Pring/Input Display is used by AcroMill to prompt user for values of various parameters and also to show any print messages embedded into the RS274D NC program.

Menu Bar shows you the current active menus. A menu can be selected by hitting a function key (for example, F1) attached to it.

Display Window is used by AcroMill to display several different data selected via a tabbed control.
Graphics will be putup when a part is loaded.
Currently executing program can be put up for viewing only when the program is running.
A live I/O Status can be put up at any time.
A live PLC status can be put up ant any time.
System Alarms can be viewed at any time.

Selecting Commands

The most common way of selecting a command in AcroMill is to use the menu bar. You can select a command by simply hitting the function key attached to a menu of the menu bar. Also, a mouse or touch screen can be used to select the Run commands by clicking on the command. Performing this will either take you to the next menu or execute the command attached to the menu.

Entering Numbers

You enter data in AcroMill using standard Windows type dialog boxes. Additionally, the pushbuttons are large enough to be used with a finger when a touch screen is installed

Starting AcroMill

AcroMill will start normally like a windows program.

Once started, AcroMill verifies that the controller is communicating and responding. After a few seconds, the control is ready to accept commands and run programs.

A Sample Session

The chapter takes you through a "test drive" of AcroMill. It assumes you have already installed the AcroMill software on your machine.

First time use.

For first time use, in order to ensure AcroMill on a host computer is correctly connected to an Acroloop Controller, card, you should run the following diagnostics.

Note

If you are initializing AcroMill for the first time, make sure that your servo loops are not active. Default settings for parameters used by AcroMill may cause motors to runaway.

☞**To make sure that the Controller card is properly installed**

When AcroMill starts it checks if the controller is responding. If there is a problem, an error message saying "NO CARD FOUND" will be displayed.
If there are no errors found, the main AcroMill Screen will show up.
From the SETUP/DIAGNOSTIC selection from the main menu, several diagnostics can be run to verify that the controller is operating properly. See section on diagnostic on page 90.

Tuning Servos

Before running a program on AcroMill, make sure the servos are properly tuned.

Tuning should only be attempted by experienced service personnel. Please read Appendix C on tuning.

Note

Before attempting to use the tuning features make sure each axes being tuned is a closed loop system with good encoder and Digital to Analog Conversion (DAC) signals. In addition, the servo amplifier must be fully operational and tuned if it is to be controlled in the velocity mode. Check out Appendix C on complete information on how to tune the Servo Amplifiers. Additionally, the SYSTEM PARAMETERS must be set for each motor. See Page 60..62 for parameter setup.

Homing the Machine

The first step in running a machine is the Home. The HOME command is a reference point and may be the position the cutting tool resides when the machine is not running a program or when disabled. Home is typically defined by a physical input such as a limit switch. The sequence of commands shown below assumes home switches for each axis are already installed and tested. Also, all the parameters for the servo loops have been properly set (See AXES menu described on page 61 and SERVO menu described on page 62).

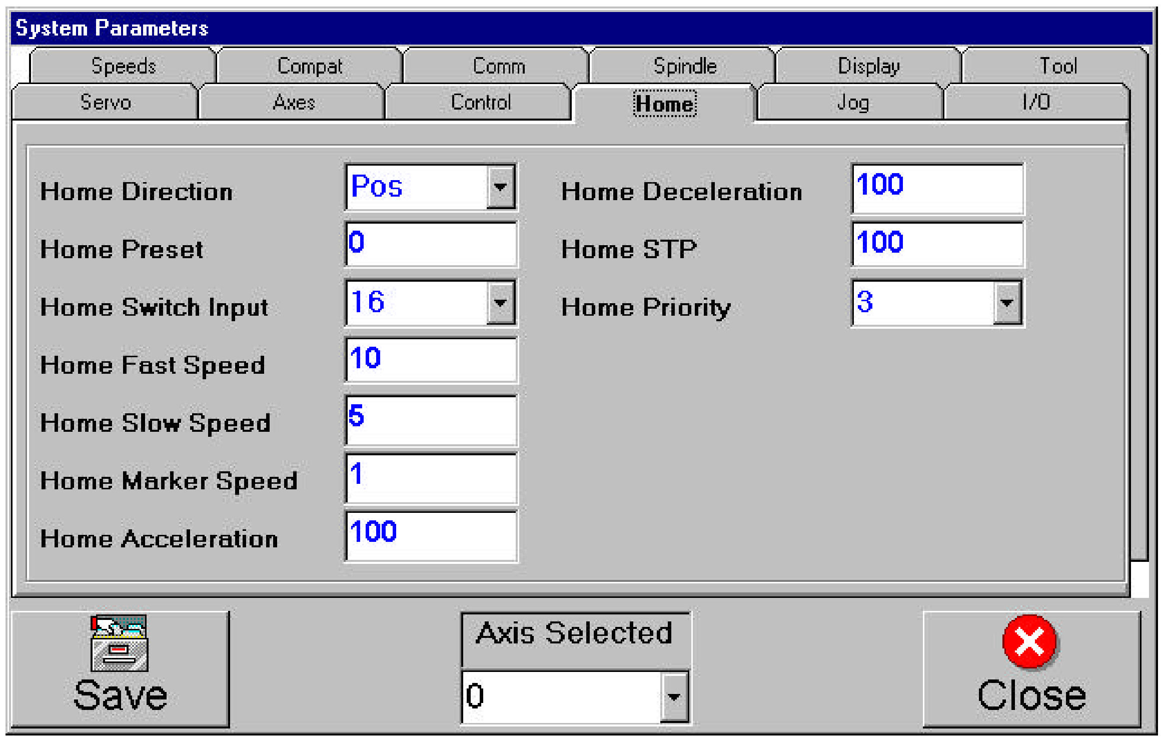
DO NOT PROCEED UNLESS THE SERVOS HAVE BEEN PROPERLY TUNED!

☞ **Setup Home Parameters**

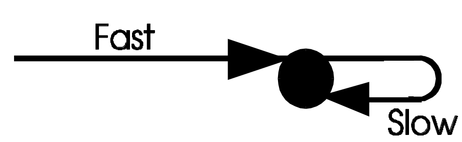
Note that this screen is only setup once by the machine builder and need not be accessed during day to day

HOME operation. During this setup make sure the SAVE key is pressed after any entries are changed or the changes will not take affect. All values are in Units.

1. Select **SET/DIAG** (F8) from the MAIN menu to access the SETUP/DIAGNOSTICS menu.
2. Select **SYS PARAM** (F1) under Level 2 to access the SYSTEM PARAMETERS menu.
3. Enter a password if required to get into this menu.



4. Select **HOME** tab to bring up the menu showing the HOME SETUP FOR AXIS 0.
5. Select the Home Direction as POS or NEG.
6. Enter the Home Preset as 0 or any other preferred value.
7. Enter the Home Switch Input number that the physical input is wired to (for example, Output 33).
8. Enter the faster speed (for example, 30 units/min) at which home switch on Axis 0 will be approached.
9. Enter the slower speed (for example, 5 units/min) at which home switch on Axis 0 will be re-approached.

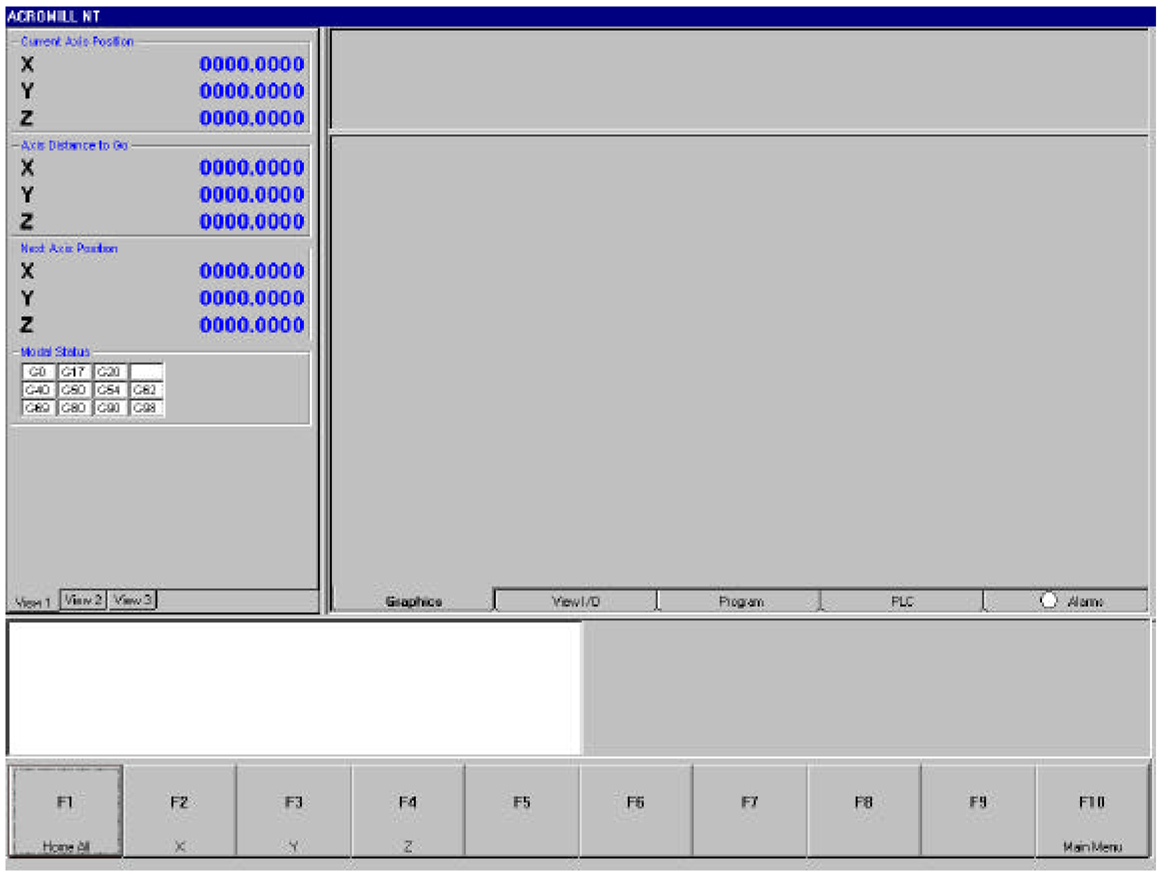


10. Enter the Marker Speed . This is the speed that the control will seek the marker on the feedback device.
11. Enter the HOME Acceleration in Units per sec per sec. This is the ramp to accelerate.
12. Enter the HOME Deceleration in Units per sec per sec. This is the ramp to decelerate.
13. Enter the HOME STP in Units per sec per sec. This is the ramp to stop.
14. Enter the Home Priority as 1 for Axis 0. This should be 0 if axis need not home, 1 if it is in the group or axes that Home first, 2 if it is in the group of axes that Home second…etc.
15. Make sure you select SAVE after entering the values, otherwise the entries will be discarded!
12. Repeat Steps 5-15 for each axis.
13. Select CLOSE to exit.

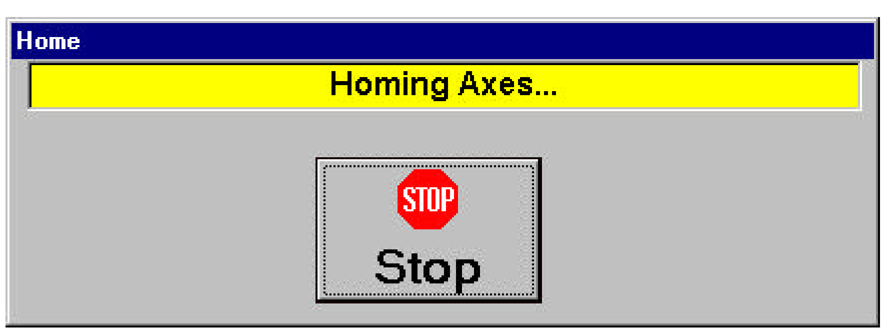
☞ **Execute the Home Cycle**

After the Home parameters have been set, you can start the Home cycle.

1. From MAIN menu select **HOME** (F1) to select the Home cycle.
2. The following screen allows you to select homing of all or individual axes.



After making axes selection, and selecting Start Home, the following screen will appear as the axes start the HOME routine.



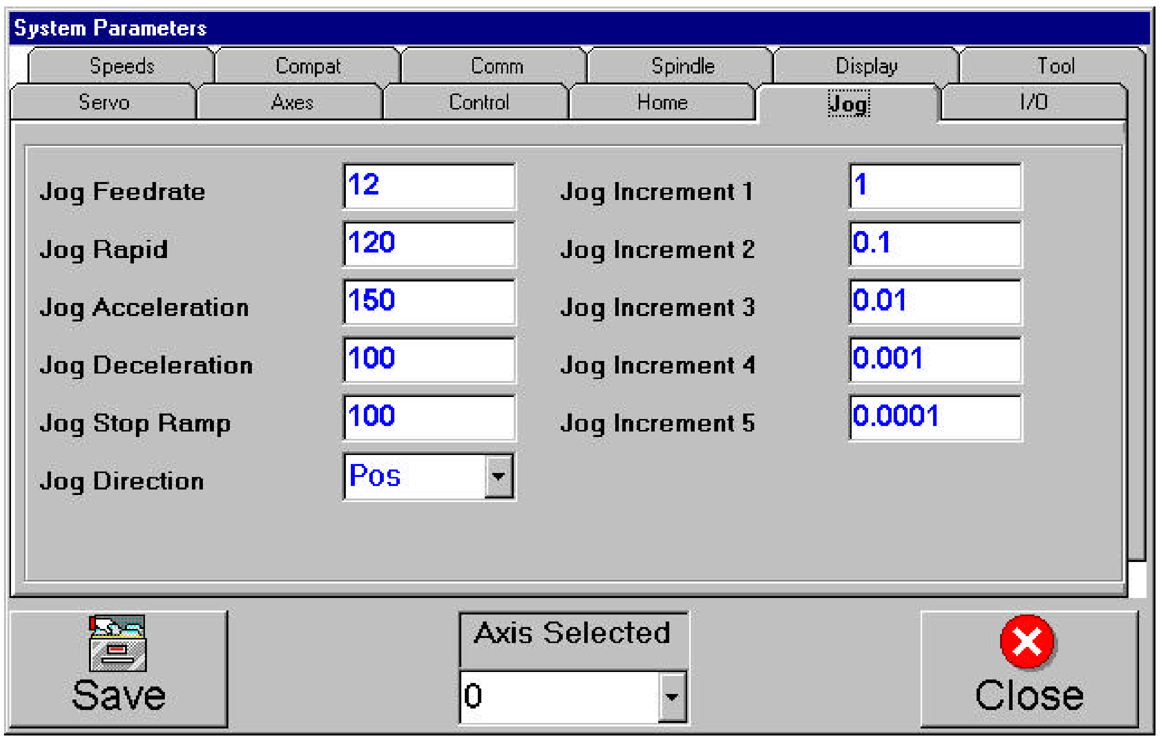
If for any reason, the HOME cycle needs to be aborted, hit the STOP button.

Jogging the Machine

Once the Home cycle has been run, you may want to jog the machine to a particular position before starting execution of a program.

☞ **Setup JOG Parameters**

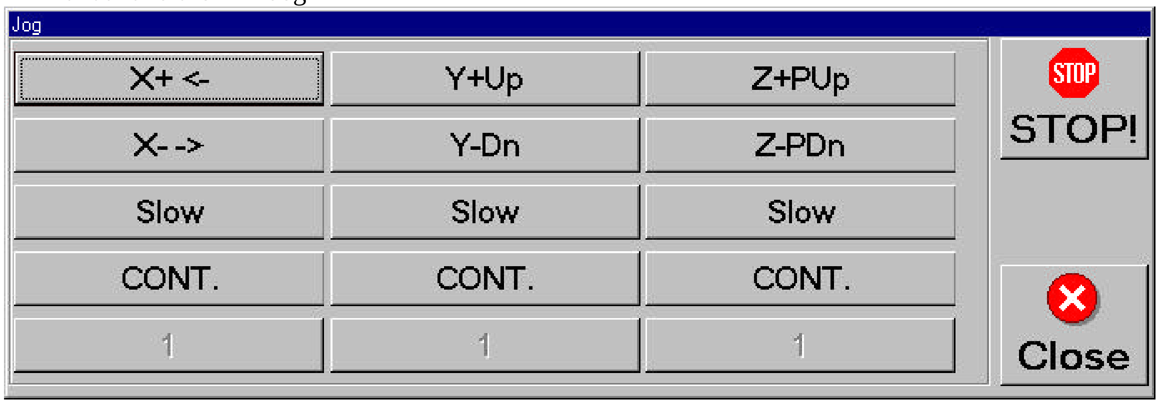
Jog parameters are setup in the System Parameger data base in the following screen. This screen is accessed from the main menu via F8-SetupDiagnostics and F1-SysParm.



Note that this screen is only setup once by the machine builder and need not be accessed during day to day Jogging operation. During this setup make sure the SAVE key is pressed after any entries are changed or the changes will not take affect. All values are in Units.

☞ **Jogging Axes**

Once the jog setup is done, the actual jogging screen can be accessed from the main menu by selecting F7-Manual and then F2-Jog



If a touchscreen is installed, directly touch the keys to make the machine move.
If a manual joystick is installed, it can be directly used to move the machine.
Otherwise, use the mouse to make the machine move. The machine will jog as long as the buttons are depressed. As soon as the button is released, motion will decelerate to a stop. As is the case with HOME, Jogging should not be attempted unless System parameters like Axis PPU, Soft Limits, Max federates…etc have be properly setup.

Using the Manual Data Input (MDI) Mode

AcroMill supports a set of the EIA RS-274D interface command set

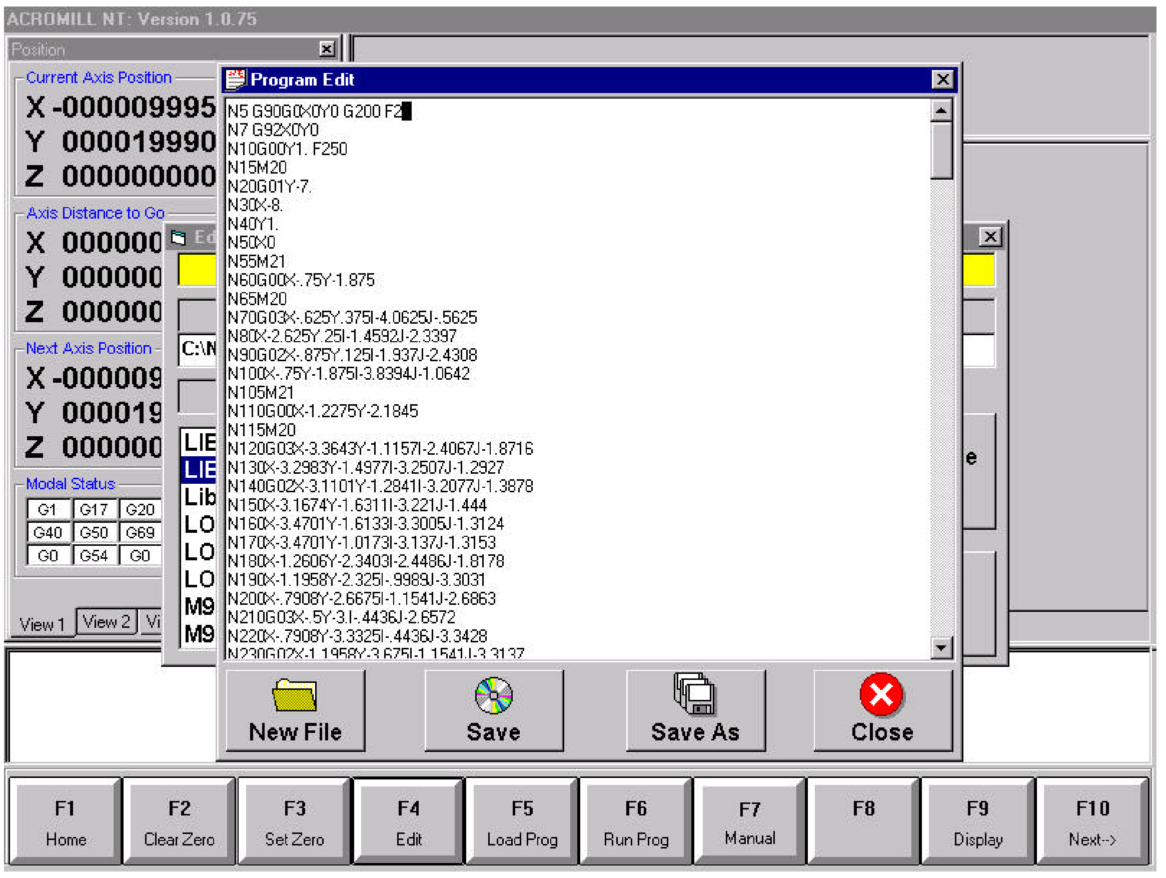
☞ **To Use MDI**

1. From MAIN menu select **F7-MANUAL** and then **F1-MDI** to go to MDI menu.

2. Type a MDI command such as G1 F50 X5.6 Y3.5. See the chapter on **RS-274D Format** (page 23) for additional MDI commands.
3. Hit EXECUTE button to start the move. Note that if the Buffer is turned on, the control will store the just executed line in the text buffer shown above. This way the user can “create” a program sequentially by executing each move. The text buffer can be edited in case a mistake is made.

Editing and Running a Program

AcroMill allows you to edit and run your own programs. This exercise assumes you have been able to successfully home (see **Homing the Machine** on page 11).



☞ **To edit a program on disk**

1. From the topmost main menu select **EDIT** (F4) to invoke the editor.

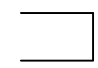
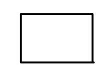
   AcroMill will bring up the FILE NAME (.TXT) dialog box.

2. Enter the file name **1111.txt** for a new file.

3. Enter the program shown below. The edited program will draw a complete square.

```
F100 G91
G1X1
G1Y1
G1X–1
```
➔ **G1Y–1**

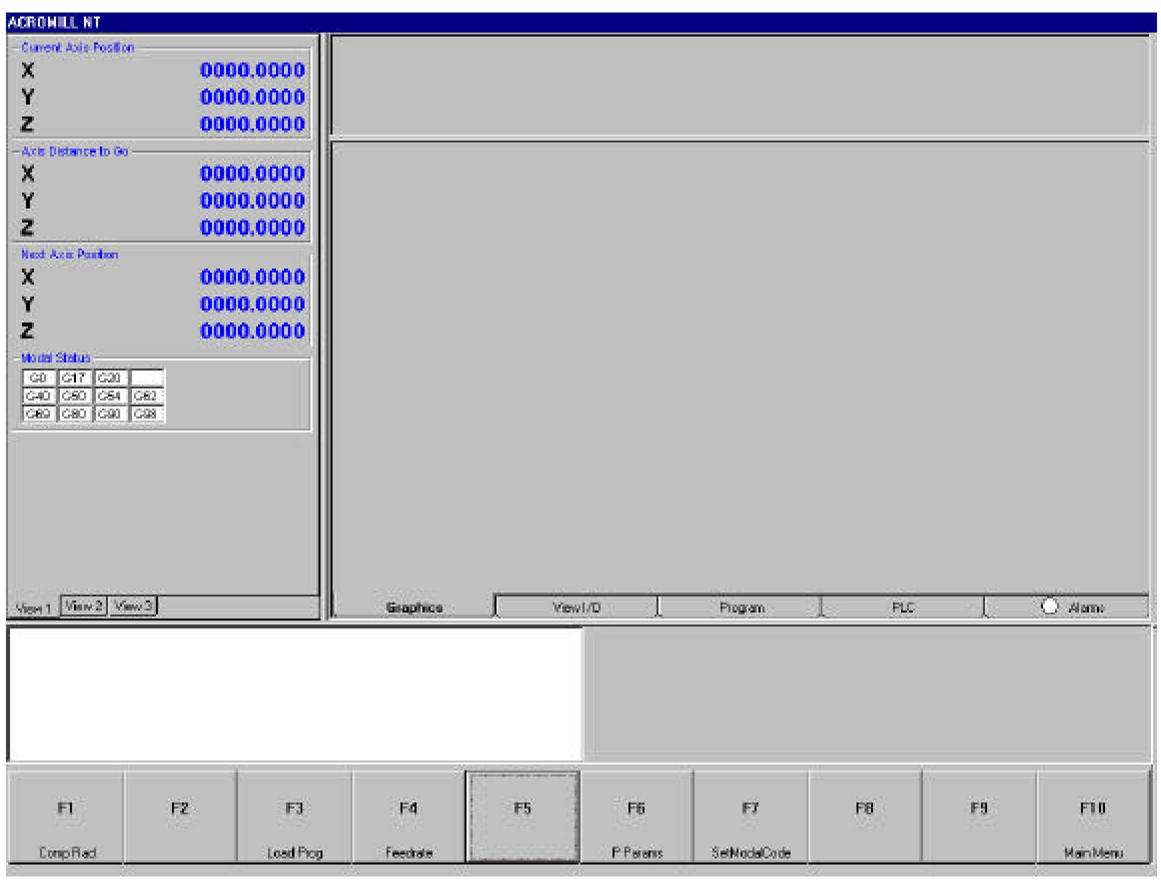
The following is a representation of the **1111.txt** program.

Program Statement	***Explanation***	**Action**
F100 G91	*Set Feedrate to 100, Incremental mode*	
G1X1	*In Feedmode, move X-axis by 1"*	
G1Y1	*In Feedmode, move Y-axis by 1"*	
G1X–1	*In Feedmode, move X-axis by -1"*	
G1Y–1	*In Feedmode, move Y-axis by -1"*	

After saving the file before exiting, the program is now ready to run.

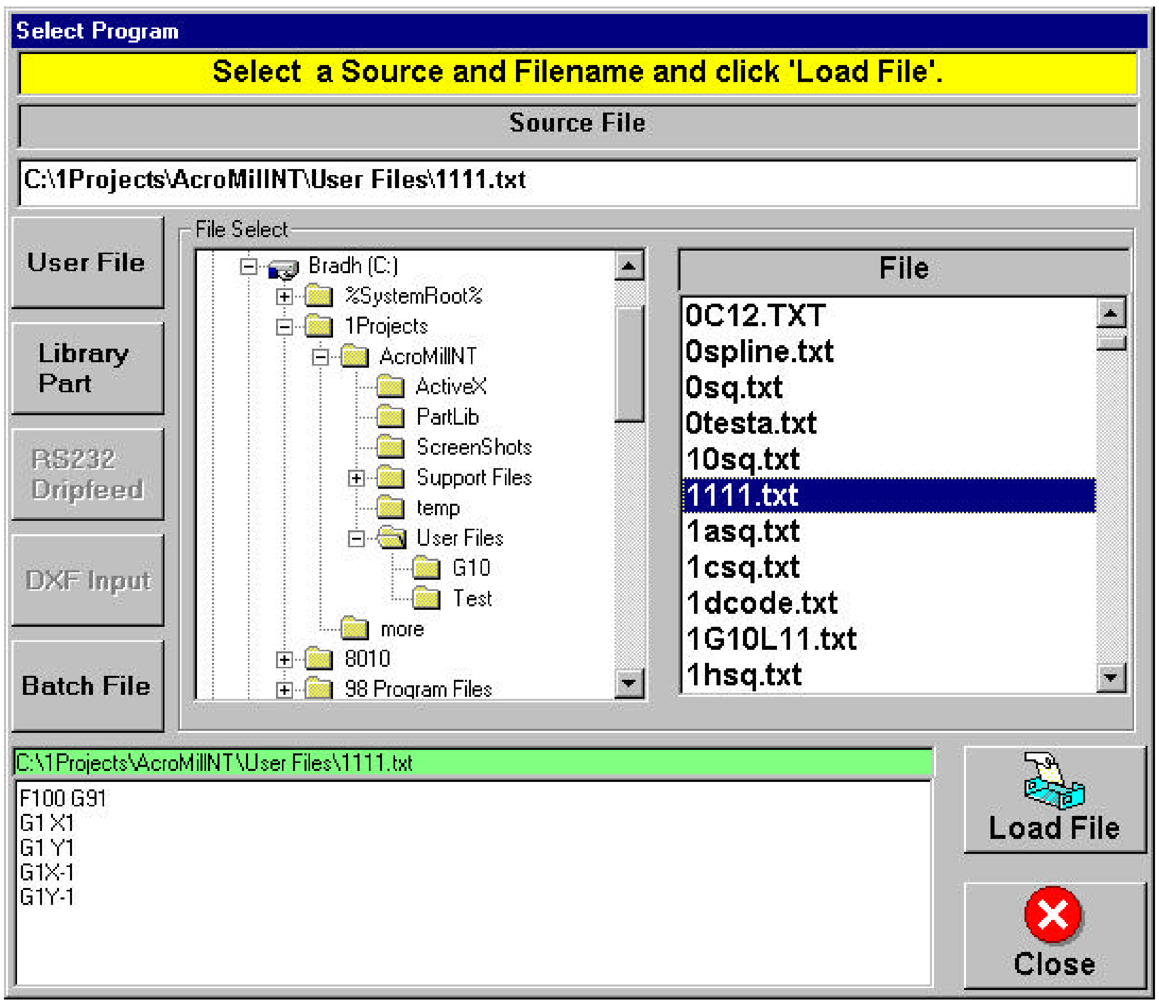
☞ **To load and run a program from disk**

1. **From the Main Menu, select PROGRAM SETUP(F3) to go to the LOAD PROGRAM MENU.**



Note that this screen allows entering Cutter Radius size, Loading program, Setting Feedrate, Setting P-Parameters and Setting MODAL G Codes.

If the program to be loaded has its own cutter radius selection and sets federates, P parameters and Modal g codes, the user can directly select **F3-LOAD PROG** to go to the load program menu.



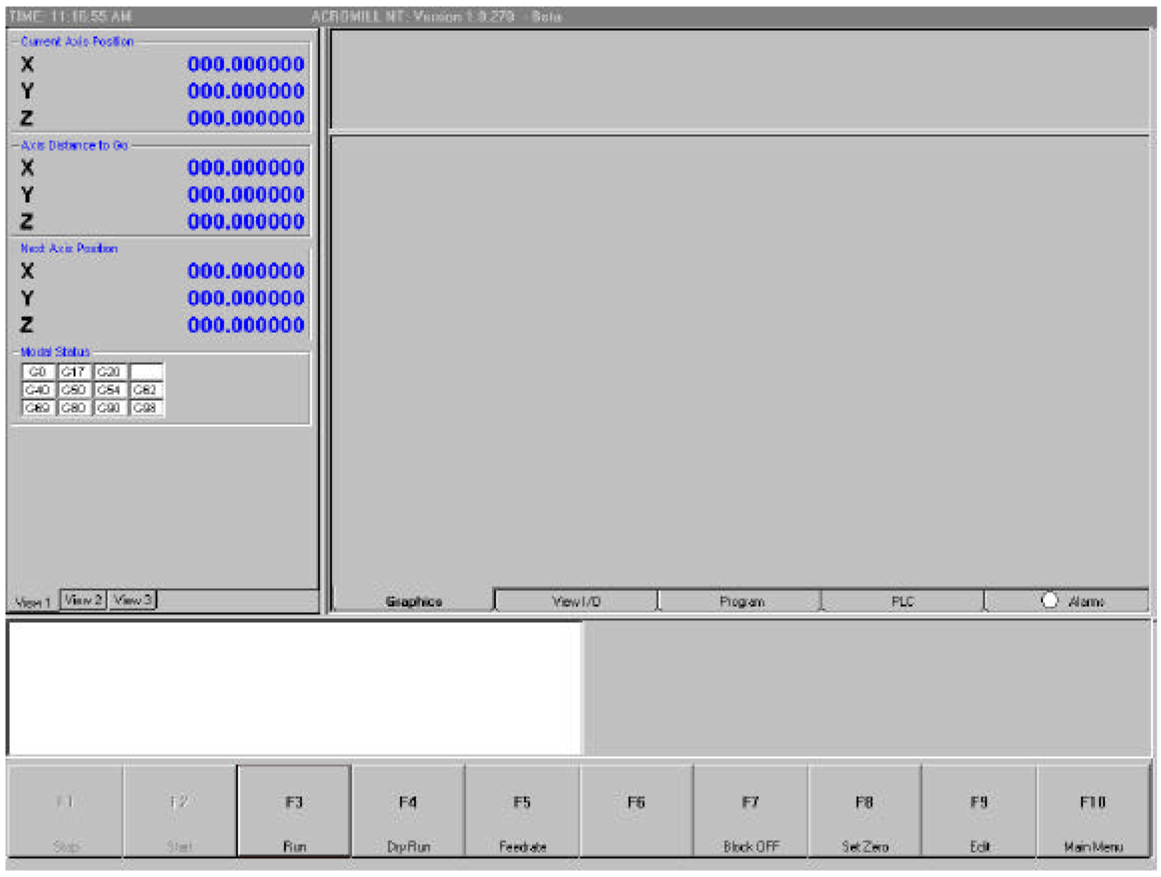
2. Here, you can either select:

USER FILE to indicate you want to load your own program.
LIBRARY PART to indicate you want to load an existing library part.
RS232 DRIP FEED This is not implemented in Initial versions of AcroMill.
DXF File This allows specifying a DXF file from AutoCad drawing.
BATCH FILE This is a batch file consisting of names of multiple NC text files.

3. Select 1111.txt as the file name in the PART PROGRAM (.TXT) dialog box

At this time, the file 1111.txt text should appear in the text box at the bottom of the above screen.
Now select LOAD FILE.

AcroMill will respond by going into the **RUN PROG** menu.

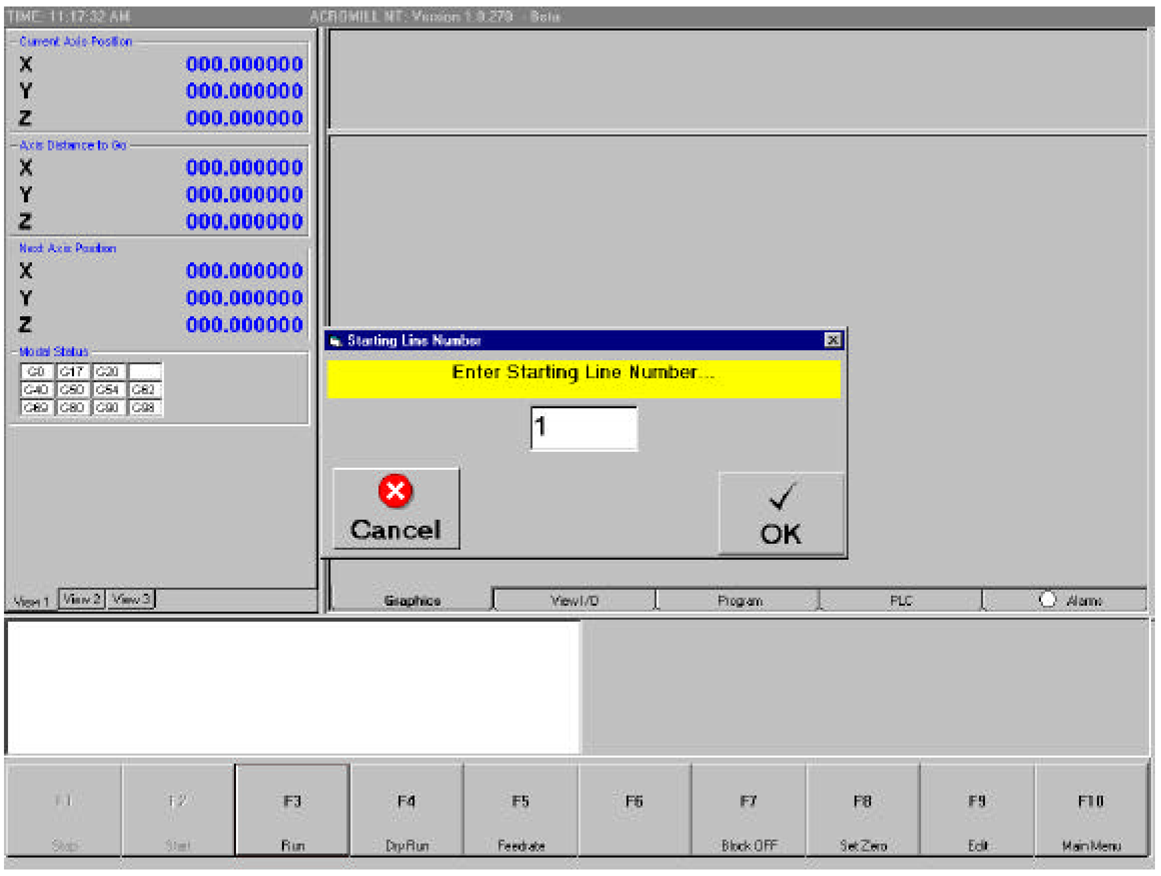


This screen allows selecting:
RUN- To Run the program.
DRY RUN- To Run the program without the cutting device coming on.
FEEDRATE- To select a specific feedrate in case there is no feedrate set in the program.
BLOCK- To either let the program RUN in CONTINUOUS or BLOCK mode.
SET ZERO- To set a floating zero.
EDIT- To Edit a different program than the one just loaded.

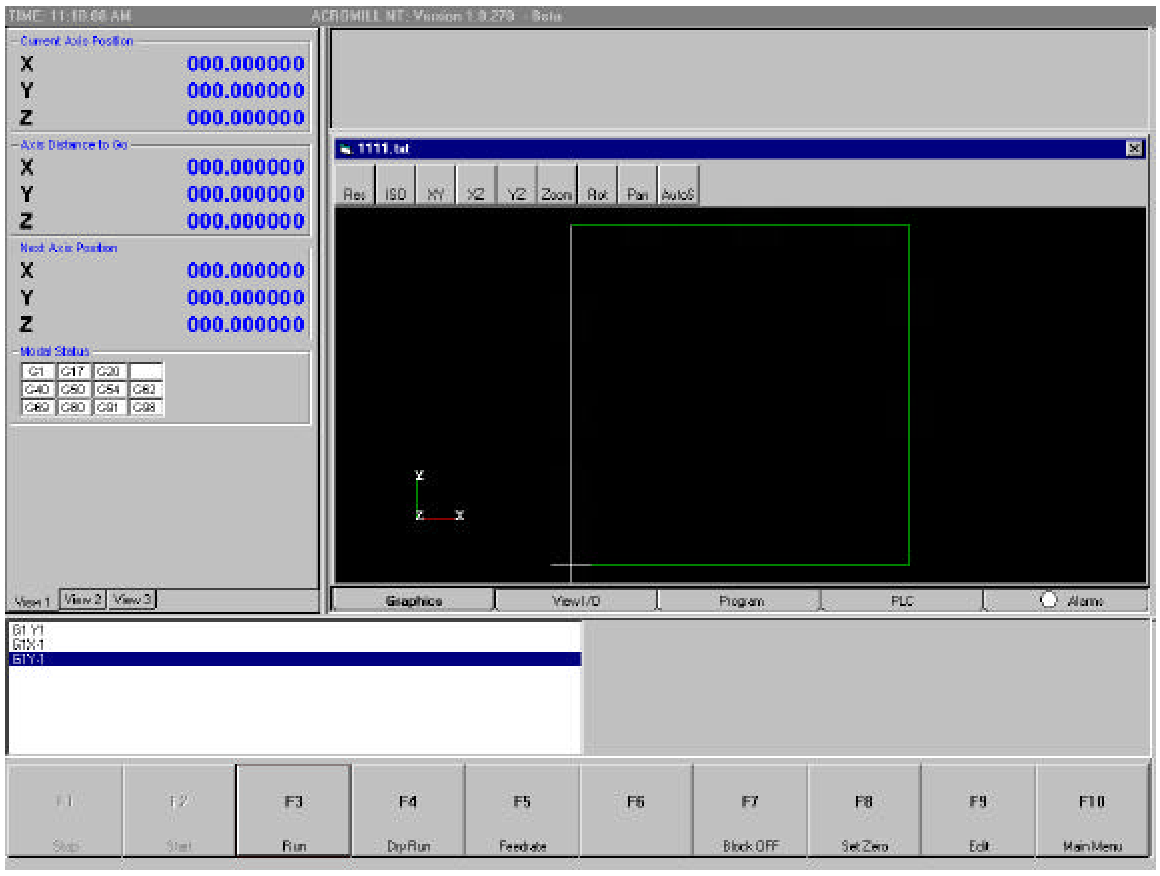
The screen also allows viewing GRAPHICS, I/O STATUS, CURRENTLY RUNNING PROGRAM, PLC and ALARMS via selection on the TAB control.

This screen also allows selection of VIEW1, VIEW2 or VIEW3 while the program is running.

In this case, for example, we can select RUN and the system will ask for a starting line number. The default is line 1.



After selecting OK to start from line 1, the following screen will appear.



This screen shows the cursor as the machine moves about the program and allows STOPPING and RESUMING the program as well as watching other status displays.

RS-274D Format

AcroMill uses Electronic Industries Association (EIA) standard RS-274D as the format for library parts, manual data commands, CAD converted files.

All RS-274D commands consists of a command letter followed by a numerical argument. These commands are divided into the following categories.

Command Category	Description
Axis Movement	Lines and Arcs
Setup	Feedrate, Circle Direction, Kerf On/Off
G Codes	Preparatory Function; Modal and One-shot
M Codes	Cutting Tool Controls- On/Off
Program Flow	Conditional/Unconditional, routine Jumps
Parametric Math	Complex Evaluations; N Command

The following tables summarize the Command Category listed above.

Axis Movement Commands

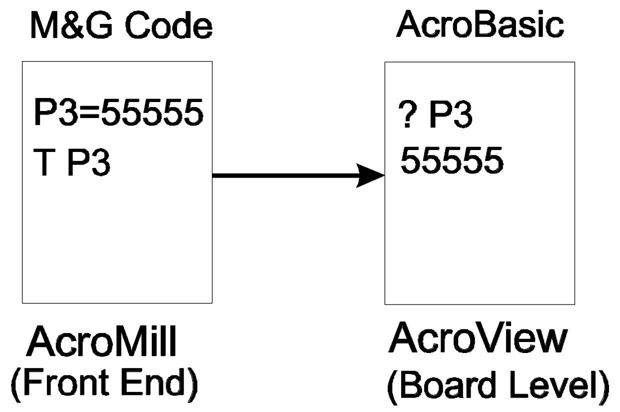
Command	Description
X	Specify X axis endpoint
Y	Specify Y axis endpoint
Z	Specify Z axis endpoint
A	Specify A axis endpoint
B	Specify B axis endpoint
C	Specify C axis endpoint
U	Specify U axis endpoint
V	Specify V axis endpoint
I	Specify X center in XY, XZ ARC move
J	Specify Y center in XY, YZ ARC move
K	Specify Z center in XZ, YZ ARC move

Setup Commands

Command	Description
F	Feedrate
P	Program Parameters 1..100
S	Spindle Speed
D	Select Cutter Radius, Wear
H	Select Offset, Tool Wear

T-Codes

T-Codes can be used along side of M and G-Codes to attach parameter values from AcroMill and transfer these values to the board level. In general, the illustration below depicts transfer.



An example of a T-Code program would look like the following steps procedure. In this example, program 5 (P05>) is used.

1. P05>10 P4=P0
   20 BIT60=NOT BIT60
   30 GOTO 10

   This program is on laptop computer and serially connected to AcroMill Controller.

2. On AcroMill in M.D.I. the general format is:

   P*x*=*xxxxx*
   T P*x*

Type in: **P3=55555**
T P3

3. Using AcroView (Acroview should be installed as part of the SDK installation) to communicate with the Controller , type

   ?P3

The value will come back as 55555.

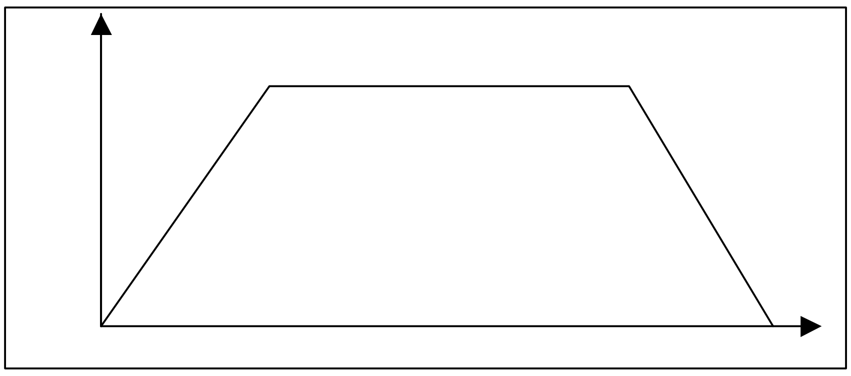
G-Codes

Command	Description
G0	Rapid Positioning
G1	Linear Interpolation
G2	Circular or Helical Interpolation CW
G3	Circular or Helical Interpolation CCW
G4 Fxx	Dwell (Fxx seconds) where x=.1 to 999.9
G9	Exact Stop
G10	Set Offsets
G17,G18,G19	XY,ZX,YZ Plane Selection
G20	Inch Units
G21	Metric Units
G27	Reference Point Return Check
G28	Reference Point Return
G29	Return from Reference point
G30	2nd,3rd and 4th Reference position return
G31	Probing Block Skip function
G40	Turn Off Kerf Compensation
G41	Turn On Left Kerf Compensation
G42	Turn On Right Kerf Compensation
G43	Tool length offset + direction
G44	Tool length offset -direction
G49	Tool length cancel
G50	Scaling OFF
G51	Scaling ON
G52	Local Co-ordinate setting
G53	Machine Co-ordinate system selection.
G54..G59	Machine Co-ordinate system 1..6 selection
G61	Exact Stop mode
G62	Automatic Corner Override
G63	Tapping Mode
G64	Cutting mode
G68	Coordinate System Rotation ON
G69	Coordinate System Rotation OFF
G70	Inch Mode
G71	Metric Mode
G73	High speed peck drilling
G74	Counter tapping
G76	Fine boring cycle
G80	Can Cycle cancel
G81	Drilling and spot drilling
G82	Drilling with dwell (spot drilling)
G83	Deep hole drilling
G84	Tapping
G85	Boring
G86	Boring
G87	Back Boring
G88	Boring
G89	Boring with dwell
G90	Absolute Mode
G91	Incremental Mode
G92	Floating Zero Preset
G98	Initial point return in Can Cycle
G99	Return to R-plane selection in Can Cycle
G100	Disable Fixture Offsets
G101-G199	Enable Fixture Offsets
G200 FXX	I/J Inversion (XX=0,1 or 2)

**Note G100 is the same as Hoffset (H0). See HOFFSETS
G101-199 is the same as Hoffsets (H1-32). See HOFFSETS
G90 and G91 is not recommended to be used in the same program.

G0 - Rapid

This command is a modal command and puts the Control into the RAPID profile mode. This mode is the mode of choice to position the tool tip to a particular point before cutting starts. Notice that the machine will always start from a stopped state, make the index at the RAPID speed previously defined in the System Parameters and decelerate to a full stop at the end of the move. G0 also cancels G1,G2,G3 modal modes.



RAPID PROFILE

The ACCEL and DECEL ramps are predefined in the System parameters. Note that RAPID moves can only be overridden via the RAPID FEED OVERRIDE switch. After encountering the G0 command, all subsequent moves use this mode. This mode can be cancelled by G1,G2,G3 commands.

G1 - FEED

This command is a modal command and puts the control into the FEED mode. G1 (lines), along with the G2 (CW arc) and G3 (CCW ARC) modes are the modes of choice for cutting and they cancel the G0 mode. Any move made with the G1 mode active will try to keep the feed at the programmed rate. Feedrate can be changed (using the F command) for each successive move and the machine will accelerate or decelerate the the program feed. This mode is also referred to as "continuous" mode. During the cutting process, the control must not hesitate between moves as this might leave "bumps" on the part. Therefore, the G1,G2,G3 modes are used whenever the tool is cutting, and the G0 mode is used to position the machine to each start of cut. Note that FEED moves can be overridden via the FEEDOVER RIDE switch

G2 - Clockwise ARC

This command is a modal command and causes the control to generate a Clockwise ARC. Modes G1 (lines), along with the G2 (CW arc) and G3 (CCW ARC) are the modes of choice for cutting and they cancel the G0 mode. Any move made with the G2 mode active will try to keep the feed at the programmed rate. Feedrate can be changed (using the F command) for each successive move and the machine will accelerate or decelerate the the program feed. This mode is also referred to as "continuous" mode. During the cutting process, the control must not hesitate between moves as this might leave "bumps" on the part. Therefore, the G1,G2,G3 modes are used when ever the tool is cutting, and the G0 mode is used to position the machine to each start of cut.

This command is used in conjunction with the PLANE selection commands G17, G18, and G19. The format for the Clockwise Arc command for all three planes is shown:

G2 G17 I_______J_____ X_____ Y______
G2 G18 I_______K_____X______ Z______
G2 G19 J______ K_____ Y______ Z______

A third axis can be specified to make a HELICAL interpolation move simultaneously to the ARC.

G3 - Counter Clockwise ARC

This command is a modal command and causes the control to generate a Counter Clockwise ARC. Modes G1 (lines), along with the G2 (CW arc) and G3 (CCW ARC) are the modes of choice for cutting and they cancel the G0 mode. Any move made with the G2 mode active will try to keep the feed at the programmed rate. Feedrate can be changed (using the F command) for each successive move and the machine will accelerate or decelerate the the program feed. This mode is also referred to as "continuous" mode. During the cutting process, the control must not hesitate between moves as this might leave "bumps" on the part. Therefore the G1,G2,G3 modes are used when ever the tool is cutting, and the G0 mode is used to position the machine to each start of cut.

G4 - Programmed Dwell

This command causes the AcroMill control to dwell for a programmed amount of time, in seconds. Feedhold will cause the countdown to stop until Cycle Start is pushed. Hitting the ABORT menu selection or hitting the Emergency Stop pushbutton will cancel this dwell.

G9 - Exact Stop

This command causes the machine to inhibit the next block until the axes have all reached their final destination. The final destination is achieved when all axes are theoretically at their target position AND their feedback devices are reading positions within the INPOSITION band set in the SYSTEM PARAMETERS. As an example, if the INPOSITION band is set to be 2 encoder pulses, G9 will cause the machine to wait until all axis are less or equal to 2 pulses of their commanded target position. This command is only active for the block that contains the G9.

G10 - Parameter Data Setting

G17 - XY Plane Select

This command sets the mode to generate CW or CCW arcs in the XY plane. This command is modal and is cancelled by G18 or G19.

G18 - ZX Plane Select

This command sets the mode to generate CW or CCW arcs in the ZX plane. This command is modal and is cancelled by G17 or G19.

G19 - YZ Plane Select

This command sets the mode to generate CW or CCW arcs in the YZ plane. This command is modal and is cancelled by G17 and G18.

G20 - Inch Units

This sets up the AcroMill control to interpret program data in English units. All values specified for axes position will be assumed to be in Inches. Feedrate is specifed in Inches/minute.

G21 - Metric Units

This sets up the AcroMill control to interpret program data in metric units. All values specified for axes positions will be assumed to be in millimeters. Feedrate is specified in mm/minute.

G27 - Reference Point Return Check

G27 X______Y______Z_____
Here X____Y____Z_____ is the COMMANDED POSITION.

This command positions the machine at RAPID FEED to the REFERENCE position.
At this point, an output (selected in the System parameters) is Energized. The control then checks in input (also set in the system parameters) to verify that the Reference position is indeed been reached . If the machine actually does not end up at the reference point, an alarm is reported and the program is stopped. If the reference position is indeed reached, the machine then proceeds at RAPID to the COMMANDED POSITION. Cancel offsets before commanding a G27, as the reference position will not be reached with any offset and an alarm will sound.

G28 - Reference Point Return

G28 X______Y______Z______
Here X___Y___Z___ is called the INTERMEDIATE POINT.

This command first positions at RAPID feedrate, to the intermediate point. Then the machine moves to the REFERENCE POINT. At this time, if the machine is not at the REFERENCE markers, an OUTPUT is turned on signaling a position error. In general, this command is used to position the machine to start an AUTOMATIC TOOL CHANGE. The REFERENCE POINT is also memorized by the control. So if on subsequent G28 commands, an axis position is not specified, the control will use the previous value for that axis. Cancel offsets before commanding a G28, as the reference position will not be reached with any offset and an alarm will sound.

G29 – Automatic Return from Reference point

G29 X______Y______Z______
Here X___Y___Z___ is called the COMMANDED POSITION.

This command first positions at RAPID feedrate, to the intermediate point (the last used intermediate point when a G28 was done). Then the machine moves to the COMMANDED POSITION again at RAPID. This command is usually used to go to a position right after a TOOL CHANGE sequence

G30 – Second Reference Point Return

G30 P <expression> X____Y____Z____

Where XYZ is the target point. It can be a single or multiple axes command.

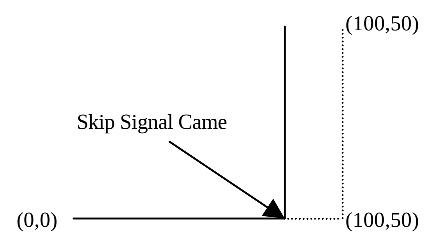
The P <expression> must equal to 2,3 or 4 and it specifies one of 4 reference points.
If no P is specified, the default is 2.
This command positions the machine to the second reference position by first going to the commanded Target point. The second reference position is set in the system parameters. This command only works if the machine was HOMED once after power up. G29 can still be used to return from the second reference point.

G31 - Skip function

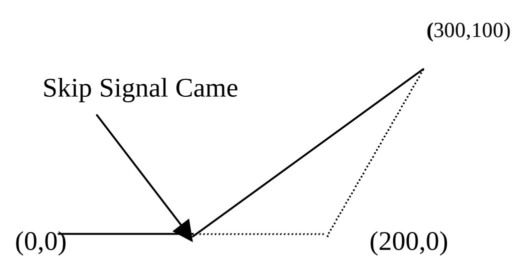
G 31 G91 X______Y____Z_____
Where XYZ is the target point.

This command will cause the machine to start moving towards the target point. Before the machine reaches that point, if the BLOCK SKIP signal arrives, the remainder of the move is discarded and the machine will decelerate to a halt. Then the machine backs up to the precise location that the BLOCK SKIP signal was received at. At this time, the control will ignore the remainder of the command line and go on to the next block.

G31 G91 X100
Y50



G31 G90 X200
X300 Y100



At the instant that the Skip Signal comes, the encoder positions are stored in parameters #101,#102,#103,#104,#105,#106,#107,#108 for axis X,Y,Z……

This command is useful for Constant feed in Grinding machines and Tool Measurement with tactile sensor probe input. This is also useful for Digitizing molds.

AcroMill, uses the INTCAP feature of the Acroloop Controller Cards. to implement the G31 probing cycle. System parameters in the AcroMill software allow the operator to specify what inputs the probe is wired to.

Since this feature is installed by the Machine builder, it is beyond the scope of this User guide to cover the hardware details.

REFER TO THE ACROLOOP CONTROLLERS USER GUIDE FOR DETAILS ON THE INTCAP COMMANDS.

The input settings are done in the System parameters. The triggering can be selected to happen either on the leading or the trailing edge.

G31 is a one shot command that is only active for one block. In this mode, the axis will try to complete the current command line. If the probe contact closes before the commands are complete, the control will immediately skip to the next line. At the instant that the probe hits, the control SAMPLES the encoder position of all the axes and stores it in GLOBAL paramaters #101, #102,#103,#104,#105,#106,#107,#108 for up to 8 Axes.

As an example the following program should be considered.

```
N100 F100 G1 X0Y0
        N110 G31 X1
        N120 Y1
        N130 X0
        N140 Y0
```

If the probe does not trigger, the above program will describe a SQUARE of 1 Unit on each side.

If, on the other hand, the probe hits when the axis is at 0.75 units in block N110, the axes will decelerate to a stop. Then the axes will move back to the exact point when the probe hit. (This compensates for the decelerating distance that the axes took to come to a halt). At this point, the rest of the move in block N110 will be abandoned and execution will go on to block N120. The net result is a RECTANGLE that is 0.75 units wide and 1 unit.

At the instant that the probe hits the part, the system stores the axes positions into parameters #101....#108. This allows the user to set offsets, or floating zeros, to compensate for tool wear or fixture positions.

G40 - Compensation OFF

This command turns off all cutter radius compensation . This command is modal and is cancelled by G41 or G42.

G41 - Compensation ON Left

This command turns on LEFT cutter compensation. This command is modal and is cancelled by G40 or G42.

G42 - Compensation ON Right

This command turns on RIGHT cutter compensation. This command is modal and is cancelled by G40 or G41.

G43 - Tool length Offset + Direction

This command turns on tool length offset in the positive direction. The offset itself is set with the G10 command or entered into the Tool table.

G44 – Tool length Offset - Direction

This command turns on tool length offset in the negative direction. The offset itself is set with the G10 command or entered into the Tool table.

G49 – Tool length Cancel

This command turns off tool length compensation

G50 – Scaling Cancel

This cancels the scaling command

G51 – Scaling.

G51 X__________ Y__________ Z__________ P_________

or

G51 X__________ Y__________ Z__________ I______ J______ K_________

Here XYZ is the scaling center and P is the scale factor for ALL axes. The I,J,K are the individual scale values. Both forms of the G51 command are modal and all subsequent moves will be scaled up or down around the scale center.

G52 - Local Co-ordinate Setting

G52 X______Y______Z_____
Here X___Y___Z___ represents the offset applied to the Zero position within G54..G59 to come up with a new ZERO position.
This command must be executed after a G54…G59 is executed.
To cancel this offset within the offsets (G54…G59), the following command must be executed.
G52 X0Y0Z0

G53 - Machine Co-ordinate System Selection

After the Machine goes through a HOME sequence, that point becomes its REFERENCE POINT. This point is fixed by the machine builder and is not changeable by the user. The co-ordinate system that reflects this ZERO point is called the MACHINE COORDINATE SYSTEM.
G53 selects this coordinate system after a different coordinate system (G54….G59).

G54 - Machine Co-ordinate System 1 Selection

A co-ordinate system other than the MACHINE CO-ORDINATE system is called a WORKPIECE co-ordinate system. AcroMill allows 6 such work piece co-ordinate systems.
These workpiece co-ordinate systems are set by one of the following methods.
1. Utilizing the G92 X_____Y______Z_____ command
2. Commanding G54….G59 commands.
All these commands are modal and will remain in effect until another command in the range of G53..G59 is commanded.
The preset offsets for this Co-ordinate system are entered in the SYSTEM PARAMETERS.
They can also be changed in the program by the following command
G10 L2 Px X______Y______Z______
Here x is a number between 1 and 6 to represent the workpiece offset being changed.
X____Y____Z_____ are the actual offsets with respect to the HOME (Machine Coordinate) position.

G55 - Machine Co-ordinate System 2 Selection

A co-ordinate system other than the MACHINE CO-ORDINATE system is called a WORKPIECE co-ordinate system. AcroMill allows 6 such work piece co-ordinate systems. These workpiece co-ordinate systems are set by one of the following methods.
1. Utilizing the G92 X_____Y______Z_____ command
2. Commanding G54….G59 commands.
All these commands are modal and will remain in effect until another command in the range of G53..G59 is commanded.
The preset offsets for this Co-ordinate system are entered in the SYSTEM PARAMETERS. They can also be changed in the program by the following command
G10 L2 Px X______Y______Z______
Here x is a number between 1 and 6 to represent the workpiece offset being changed.
X____Y____Z_____ are the actual offsets with respect to the HOME (Machine Coordinate) position.

G56 - Machine Co-ordinate System 3 Selection

A co-ordinate system other than the MACHINE CO-ORDINATE system is called a WORKPIECE co-ordinate system. AcroMill allows 6 such work piece co-ordinate systems. These workpiece co-ordinate systems are set by one of the following methods.
1. Utilizing the G92 X_____Y______Z_____ command
2. Commanding G54….G59 commands.
All these commands are modal and will remain in effect until another command in the range of G53..G59 is commanded.
The preset offsets for this Co-ordinate system are entered in the SYSTEM PARAMETERS. They can also be changed in the program by the following command
G10 L2 Px X______Y______Z______
Here x is a number between 1 and 6 to represent the work piece offset being changed.
X____Y____Z_____ are the actual offsets with respect to the HOME (Machine Coordinate) position.

G57 - Machine Co-ordinate System 4 Selection

A co-ordinate system other than the MACHINE CO-ORDINATE system is called a WORKPIECE co-ordinate system. AcroMill allows 6 such work piece co-ordinate systems. These workpiece co-ordinate systems are set by one of the following methods.
1. Utilizing the G92 X_____Y______Z_____ command
2. Commanding G54….G59 commands.
All these commands are modal and will remain in effect until another command in the range of G53..G59 is commanded.
The preset offsets for this Co-ordinate system are entered in the SYSTEM PARAMETERS. They can also be changed in the program by the following command
G10 L2 Px X______Y______Z______
Here x is a number between 1 and 6 to represent the work piece offset being changed.
X____Y____Z_____ are the actual offsets with respect to the HOME (Machine Coordinate) position.

G58 - Machine Co-ordinate System 5 Selection

A co-ordinate system other than the MACHINE CO-ORDINATE system is called a WORKPIECE co-ordinate system. AcroMill allows 6 such work piece co-ordinate systems. These workpiece co-ordinate systems are set by one of the following methods.
1. Utilizing the G92 X_____Y______Z_____ command
2. Commanding G54….G59 commands.

All these commands are modal and will remain in effect until another command in the range of G53..G59 is commanded.
The preset offsets for this Co-ordinate system are entered in the SYSTEM PARAMETERS. They can also be changed in the program by the following command
G10 L2 Px X______Y______Z______
Here x is a number between 1 and 6 to represent the workpiece offset being changed.
X____Y____Z_____ are the actual offsets with respect to the HOME (Machine Coordinate) position.

G59 - Machine Co-ordinate System 6 Selection

A co-ordinate system other than the MACHINE CO-ORDINATE system is called a WORKPIECE co-ordinate system. AcroMill allows 6 such work piece co-ordinate systems. These workpiece co-ordinate systems are set by one of the following methods.
1. Utilizing the G92 X_____Y______Z_____ command
2. Commanding G54….G59 commands.
All these commands are modal and will remain in effect until another command in the range of G53..G59 is commanded.
The preset offsets for this Co-ordinate system are entered in the SYSTEM PARAMETERS. They can also be changed in the program by the following command
G10 L2 Px X______Y______Z______
Here x is a number between 1 and 6 to represent the workpiece offset being changed.
X____Y____Z_____ are the actual offsets with respect to the HOME (Machine Coordinate) position.

G61 - Exact Stop Mode

This command causes machine to decelerate to a stop on all subsequent moves and wait for inposition signal from the controller. This command is modal and is cancelled by G62 or G64.

G62 - Automatic Corner Override

This command causes machine to decelerate on corner so as to perform complete cutting. This causes the corners to be cut more accurately. The slowdown is controlled via setting in the SYSTEM parameters. This is a modal command and is cancelled by G61, G63 or G64

G63 - Tapping mode

This command causes machine to ignore all FEEDRATE OVERRIDE setting and runs always at 100% of the programmed feedrate . This command is modal and is cancelled by G61,G62 or G64.

G64 - Cutting mode

This command causes machine not to decelerate at corners on all subsequent moves. This command is modal and is cancelled by G61 or G62.

G65 - Call Macro

G65 <sub_number> { L <repeat_count>} {<var_assignment>}*
where <var_assignment> ::= variable <expression>

This is a new command and offers the following type of call structure
G65 P1000 L2 X1 Y2 Z3 A4 B5 C6 D7 E8 F9 G10 H11 I12 J13 K14
In this above example, Subroutine N1000 will be called 2 times with the

following values loaded into the local variables.
#1 = 1
#2 = 2
#3 = 3
#4 = 4
#5 = 5
#6 = 6
#7 = 7
#8 = 8
#9 = 9
#10 = 10
#11 = 11
#12 = 12
#13 = 13
#14 = 14

G66 - Call Modal Macro

G66 <sub_number> { L <repeat_count>} {<var_assignment>}*
where <var_assignment> ::= variable <expression>

This command works identical to G65 with the exception that once the above setup is completed, the subroutine call is evoked after every subsequent move block until the mode is cancelled by using the G67 command. This is a very powerful tool for writing SPECIAL CAN CYCLES

G67 - Cancel Modal Macro

This command Cancels the MODAL MACRO mode.

G68 - Co-ordinate System rotation

G17 G68 X____Y____ R_____
G18 G68 Y____Z_____R_____
G19 G68 X____Z_____R_____

This command will rotate around the first two axis dimensions R degrees. Angle of rotation is PLUS for CCW and MINUS for CW rotation.

G69 - Co-ordinate System Rotate Cancel

This cancels the G68 command.

G70 - Inch Mode

This activates the Inch mode. All program dimension following this command are treated as Inches.

G71 - Metric Mode

This activates the Metric mode. All program dimension following this command are treated as Millimeters.

G79-G89. Caned Cycles- Overview.

The following Can Cycles are kept in a file called CYCLES.TXT. If the user wants to use these cycles, this file must be included in the part program by using the following command at the end of the patricular file.

$I CYCLES.TXT

The above command should be on a line by itself and must be AFTER the main body of the user program, past the M2 (end of program) command.

The Canned Cycles use labels in the range of N73000…… and higher. Therefore, the user programs must not use any labels higher than say 70000 for any user program.

Canned Cycles follow 6 steps generally.

Position major axes (Typically X,Y)
Rapid Z Axis to R-PLANE dimension.
Do the Hole machining (Drilling, Boring, Peck….etc).
Do some operation at the bottom of the hole (Delay….etc).
Retract Z to R-PLANE dimension.
Rapid to Initial Point. (Depending on G98 or G99 mode selection)

The Canned Cycles are Plane dependent and follow the following convention.

Gcode	Positioning Axes	Drilling/Boring Axis
G17	X,Y	Z
G18	Z,X	Y
G19	Y,Z	X

In conjunction with the Can Cycles, modal commands G98 and G99 selects whether to retract only to R-PLANE or all the way to the Initial Point.

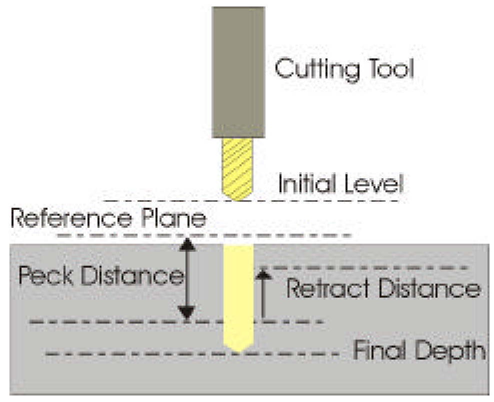
The general drilling format is given below.

GXX X_____Y_____ Z_____R_____Q______P_______F________ L_______

Here XX is the type of cycle 73 through 89.
X____Y____ is the location of the HOLE.
Z specifies the distance from the R-Plane to the bottom of the hole.
R specifies the distance from the R-PLANE dimension to the bottom of the hole.
Q specifies the PECK value G73,G83 and the SHIFT value with G76, G87.
P specifies the DWELL time at the bottom of the hole.
F specifies the feedrate.
L specifies the number of repeats. This is used when XY is incremental. This will result in multiple holes with one command.

G73 - High speed peck drilling

G90 G73 X_____Y______Z<final depth> R<reference plane> Q<peck distance> F_____

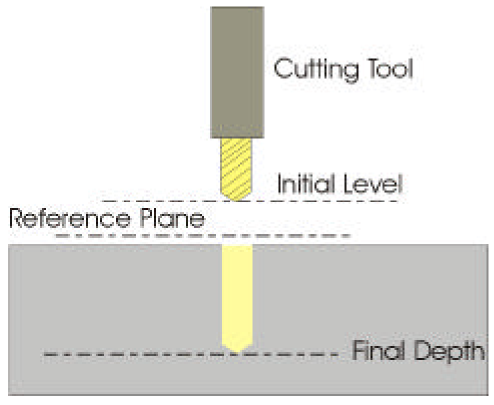


In the above diagram, the position of Z axis at the beginning of the G73 command is the initial level or position. The spindle must be turned on CW before commanding cycle. As the cycle starts,

1. Z will plunge at down at RAPID feed to the Reference (R-Plane).
2. Then it feeds down at the programmed feedrate an incremental programmed PECK distance.
3. Then Z will retract an incremental RETRACT distance at RAPID feed.
4. Steps 2 and 3 are repeated until final depth is reached.
5. Finally, if G98 mode is active, Z will retract at RAPID feed to the Initial Level. Otherwise Z will RAPID feed to the R-Plane.

G74 - Counter Tapping

G90 G74 X_____Y______Z<final depth> R<reference plane> F_____

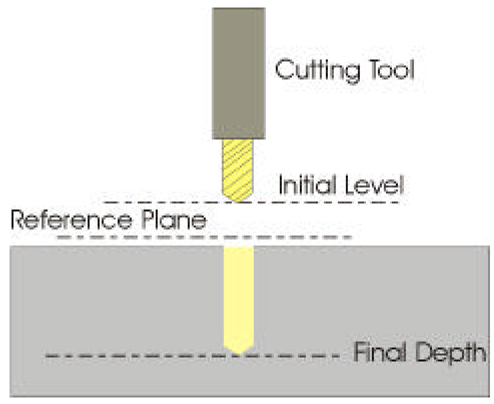


In the above diagram, the position of Z axis at the beginning of the G74 command is the initial level or position. The spindle must be turned on CCW before commanding cycle. As the cycle starts,

1. Z will plunge at down at RAPID feed to the Reference (R-Plane).
2. Spindle is CCW and Z feeds down at the programmed feedrate to the final Z depth.
3. At the bottom of the hole spindle is reversed to CW direction and Z feeds up to R-Plane.
4. Finally, if G98 mode is active, Z will retract at RAPID feed to the Initial Level. Otherwise, Z stays at R-Plane.

G76 - Fine Boring cycle

G90 G76 X_____Y______Z<final depth> R<reference plane> Q<shift distance> F_____



In the above diagram, the position of Z axis at the beginning of the G76 command is the initial level or position. The spindle must be turned on CW before commanding cycle. As the cycle starts,

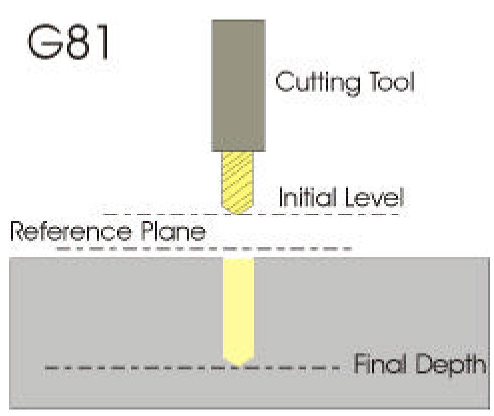
1. Z will plunge at down at RAPID feed to the Reference (R-Plane).
2. Z feeds down at the programmed feedrate to the final Z depth.
3. At the bottom of the hole spindle is stopped and goes through an ORIENTATION cycle.
4. Then the axis shifts incremental <shift distance> in X.
5. Finally, if G98 mode is active, Z will retract at RAPID feed to the Initial Level. Otherwise, Z RAPID feeds to the R-Plane.

G80 - Can Cycle cancel

This command cancels the previously selected Can Cycle.

G81 - Drilling and Spot Drilling

G90 G81 X_____Y______Z<final depth> R<reference plane> F_____

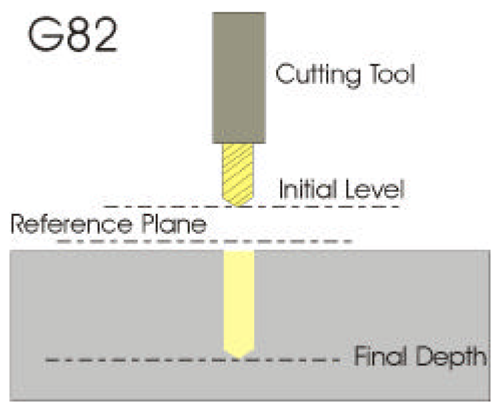


In the above diagram. the position of Z axis at the beginning of the G81 command is the initial level or position. The spindle must be turned on CW before commanding cycle. As the cycle starts,

1. Z will plunge at down at RAPID feed to the Reference (R-Plane).
2. Z feeds down at the programmed feedrate to the final Z depth.
3. Finally, if G98 mode is active, Z will retract at RAPID feed to the Initial Level. Otherwise, Z RAPID feeds to the R-Plane.

G82 - Drill with Dwell (Counter Boring)

G90 G82 X_____Y______Z<final depth> R<reference plane> P<dwell time> F_____

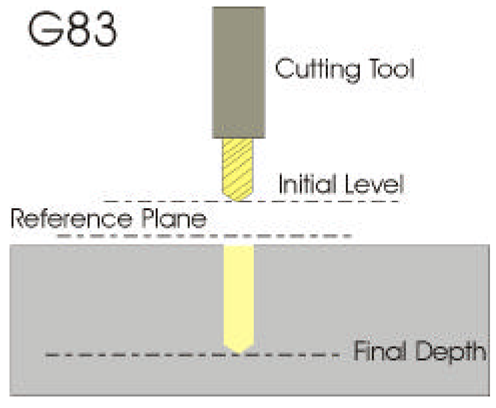


In the above diagram. the position of Z axis at the beginning of the G82 command is the initial level or position. The spindle must be turned on CW before commanding cycle. As the cycle starts,

4. Z will plunge at down at RAPID feed to the Reference (R-Plane).
5. Z feeds down at the programmed feedrate to the final Z depth.
6. Dwell is started for <dwell time> seconds.
7. Finally, if G98 mode is active, Z will retract at RAPID feed to the Initial Level. Otherwise, Z RAPID feeds to the R-Plane.

G83 - Deep Hole Drilling

G90 G83 X_____Y______Z<final depth> R<reference plane> Q<peck distance> F_____



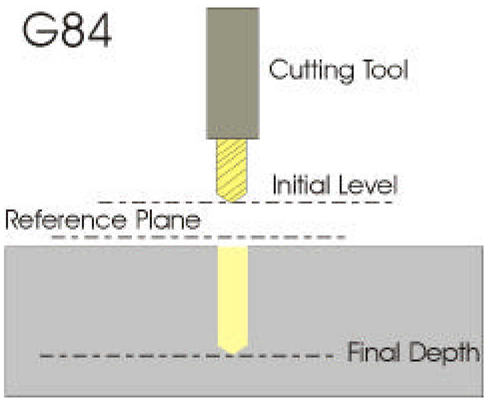
In the above diagram. the position of Z axis at the beginning of the G83 command is the initial level or position. Spindle must be turned on CW before commanding cycle. As the cycle starts,

8. Z will plunge at down at RAPID feed to the Reference (R-Plane).
9. Z feeds down at the programmed feedrate to in incremental <peck distance>.
10. Z then RAPID feeds up to R-PLANE.
11. Z then RAPID feeds down to within “D” distance of the previously drilled depth.
12. Z then FEEDS an additional <peck distance> + “D”.
13. Steps 10..12 are repeated until the final Depth is reached.
14. Finally, if G98 mode is active, Z will retract at RAPID feed to the Initial Level. Otherwise, Z RAPID feeds to the R-Plane.

In the above Cycle, the distance “D” is entered via SYSTEM PARAMETERS.

G84 - Tapping

G90 G84 X_____Y______Z<final depth> R<reference plane> P<dwell time> F_____

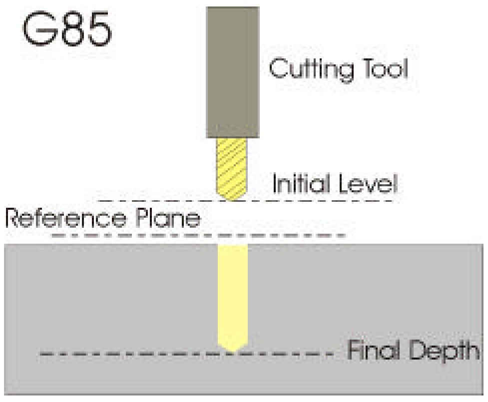


In the above diagram. the position of Z axis at the beginning of the G84 command is the initial level or position. The spindle must be turned on CW before commanding cycle. As the cycle starts,

1. Z will plunge at down at RAPID feed to the Reference (R-Plane).
5. Z feeds down at the programmed feedrate to the final Z depth.
6. At the bottom of the hole spindle is reversed to CCW direction.
7. Dwell is started for <dwell time> seconds.
8. Z feeds up to R-Plane.
9. Finally, if G98 mode is active Z will retract at RAPID feed to the Initial Level. Otherwise, Z stays at R-Plane.

G85 - Boring

G90 G85 X_____Y______Z<final depth> R<reference plane> F_____

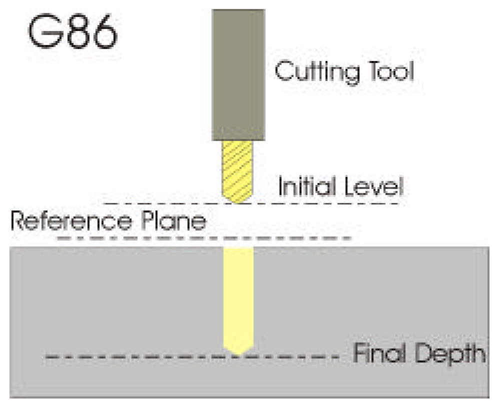


In the above diagram. the position of Z axis at the beginning of the G85 command is the initial level or position. The spindle must be turned on CW before commanding cycle. As the cycle starts,

1. Z will plunge at down at RAPID feed to the Reference (R-Plane).
2. Z feeds down at the programmed feedrate to the final Z depth.
3. Z feeds up to R-Plane.
4. Finally, if G98 mode is active, Z will retract at RAPID feed to the Initial Level. Otherwise, Z stays at R-Plane.

G86 - Boring

G90 G86 X_____Y______Z<final depth> R<reference plane> F_____

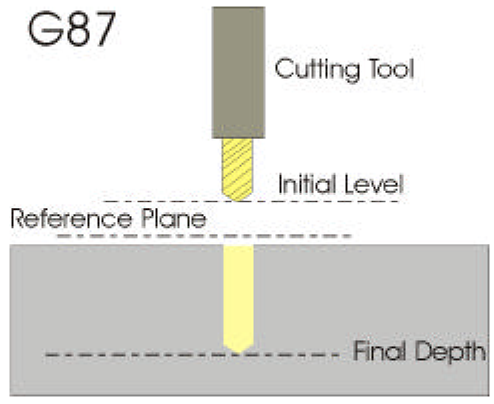


In the above diagram. the position of Z axis at the beginning of the G86 command is the initial level or position. The spindle must be turned on CW before commanding cycle. As the cycle starts,

5. Z will plunge at down at RAPID feed to the Reference (R-Plane).
6. Z feeds down at the programmed feedrate to the final Z depth.
7. Finally, if G98 mode is active, Z will retract at RAPID feed to the Initial Level. Otherwise, Z RAPID feeds to the R-Plane.

G87 - Back Boring

This CYCLE is not implemented.

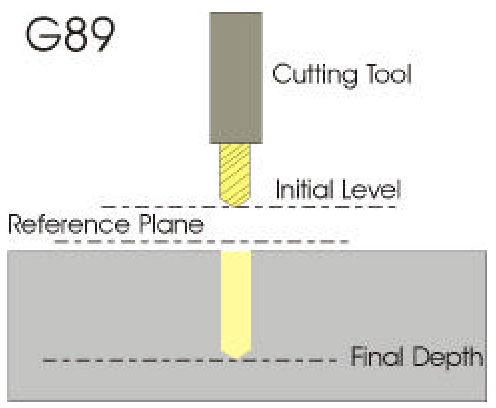


G88 - Boring

This CYCLE is not implemented.

G89 - Boring with Dwell

G90 G89 X_____Y______Z<final depth> R<reference plane>P<dwell time> F_____



In the above diagram. the position of Z axis at the beginning of the G89 command is the initial level or position. The spindle must be turned on CW before commanding cycle. As the cycle starts,

1. Z will plunge at down at RAPID feed to the Reference (R-Plane).
2. Z feeds down at the programmed feedrate to the final Z depth.
3. Dwell is started for <dwell time> seconds.
4. Finally, if G98 mode is active, Z will retract at RAPID feed to the Initial Level. Otherwise, Z RAPID feeds to the R-Plane.

G90 - Absolute

This selects the ABSOLUTE dimension mode. All Axis commands are interpreted as absolute values. This command is modal is is cancelled by the G91 mode.

G91 - Incremental

This selects the INCREMENTAL dimensional mode. All Axis commands are interpreted as incremental from the last axis value. This command is modal and is cancelled by the G90 command.

G98 - Initial point return

This is a modal command that directs Can Cycles to return to the Initial Z depth at the end of the cycle.

G99 - R-point return

This is a modal command that directs Can Cycles to return to the R-Plane at the end of the cycle.

G100 - Cancel Fixture Offsets

This cancels fixture offsets .

G101..199 - Enable Fixture offsets for tool 1…99

This enables the fixture offsets for too 1..99. The fixture offsets are entered via the G10 command or entered from Offsets screen.

G200 - IJ Inversion selection

G200 F0
G200 F1
G200 F2

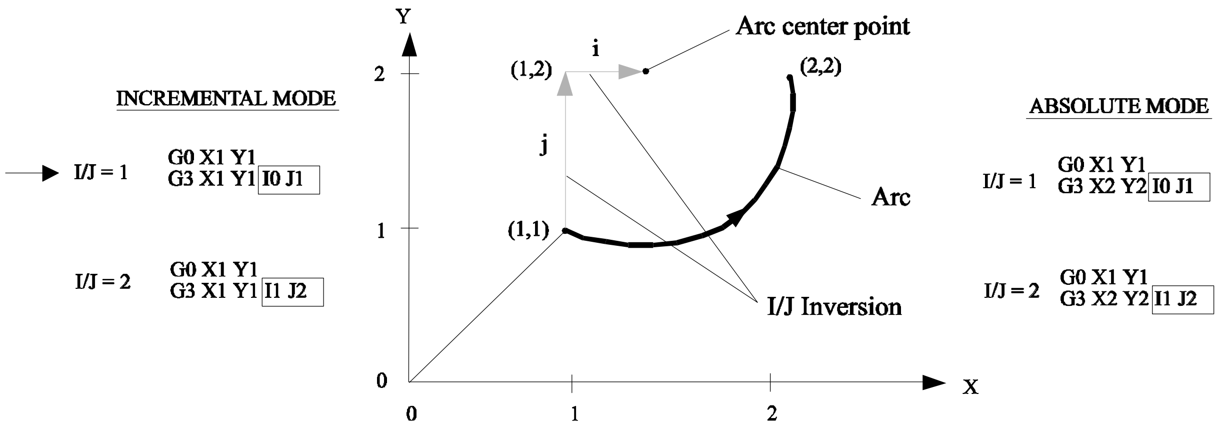
There are 3 different ways to interpret I and J commands in a typical Circle command shown below. The first command simply positions the machine to X1 Y1. Then a CW circle is executed.

X1 Y1
G2 X1 Y1 I0 J1

The I/J Inversion defines the inverse vector relationship for the start of the arc or the center point of the arc cut.

0 - means I/J is treated as the incremental negative value from the start of the arc.
1 - means I/J is treated as the incremental positive value from the start of the arc
2 - means I/J is treated as a *absolute* position.

The diagram below depicts the I/J Inversion setting. In this example, the settings are in the incremental mode. An arrow is positioned next to the correct code for the illustration. Note the G-Command code is determined by the I/J Inversion settings. In this case, I/J = 0. Also, note the I/J Inversion values do not change in either incremental or absolute modes.



*Disparate CNC formats interpret I,J definitions in a CIRCLE/ARC command differently. The ability to select this mode makes this RS274D interface flexible.

M Codes

The M-Codes must be setup with a corresponding output. These outputs can be any of the 32 outputs standard offered and are numbered 32-63. Note that the ACR2000 only has 16 Inputs and 16 Outputs, so that its I/O is numbered 00..15 for inputs and 32..47 for outputs.

M-Codes are universally accepted when the RUN command is entered. However, the DRY RUN command which is used in test verification applications will inhibit the standard M-Codes listed below. It should be noted that customized M-Codes will function under both RUN and DRY RUN commands.

Command	Description
M0	Program Stop For Inspection
M1	Optional Stop For Inspection
M2	Stop (End) Program
M3	Spindle On Clockwise
M4	Spindle On Counterclockwise
M5	Spindle Off
M6	Tool Change
M7	Coolant Mist
M8	Coolant Flood
M9	Coolant Off
M12–M24	Turn On Discrete Outputs
M25–M37	Turn Off Discrete Outputs
M595	Keeps axis tangential to the programmed path
M596	Terminates or cancels M95
M597	Keeps axis to the right of the programmed path
M598	Keeps axis to the left of the programmed path
M599	Terminates or cancels M597 and M598

M1XX Will wait until software relay#XX is energized.
M2XX Will wait until software relay#XX is de-energized

Acromill has provisions for installing Custom M-Codes. Only an authorized Acroloop VAR should attempt to install these M Codes.

Program Flow Commands

N Command Block Numbering

Format: Nxxxx

Example:
```
G91
F100
P5=0
N100
P5=P5+1
X1
Y1
IF P5<5 THEN GOTO 100
M2
```

Tip: If it was desired to run the same shape forever, remove the IF command line out and set the RepeatForEver parameter in the system parameter to one (1).

Example:
```
G91
F100
P5=0
N100
P5=5



X1
Y1
M2
```

The N command is the Block Number command

Unconditional Branch

Format: GOTO XXX

This command will transfer control to block number XXX.

Conditional Branch

Format: IF <Statement> THEN <GOTO Command>

This command specifies a conditional branch.

Examples:

If P1=5 THEN GOTO 100
This line will jump to Block 100 if P1=5

If #1 = 1 THEN GOTO 100

This line will jump to Block 100 if Input 1 is energized.

Subroutines

Format: GOSUB XXX

This command will cause a jump to a valid block number XXX

Format: RET

This command will cause a return from a routine. An error is generated if a RET is entered before a GO.

Up to 10 levels of GO and RET are supported.

Parametric Math Operations

Format: P*xx* = <Expression>
Or
#xx = <Expression>

In parametric programming, the arguments can be any P Parameters. These parameters are referenced by either the letter "P" or the symbol "#"
The parameters are defined in the following ranges.

P0—P33 Global variables hosted in ACRXXXX Controller No Persistence
P34—P99 Reserved Internal Variables hosted in ACRXXXX Controller No Persistence
P100—P199 AcroMill global variables hosted in SysVarStore. Persistent
P200—P233 UserVariables hosted in SysVarStore. Persistent
P234—P248 SystemVariables hosted in SysVarStore. Persistent
P249—P270 Canned Cycle variables hosted in SysVarStore. Persistent
P271—P399 Reference Position variables hosted in SysVarStore. Persistence
P400—P799 Available.
P800—P832 Floating Zero Variables hosted in SysVarStore. Persistent
P833—P899 Available.
P900—P999 Work Coordinate Variables hosted in SysVarStore. Persistent
P1000—P1400 Tool (D) Offsets hosted in SysVarStore. Persistent
P1401—P1499 Available.
P1500—P1900 Tool (D) Wear Offsets, hosted in SysVarStore. Persistent.
P1901—P1997 Available.
P1998 Current D Offset . Reset by AcroMill to default on startup.
P1999 Current D Offset Wear. Reset by AcroMill to default on startup
P2000—2999 Tool (H) Geometric Offsets hosted in SysVarStore. Persistent
P3000—P3999 Tool (H) Wear Offsets hosted in SysVarStore. Persistent.
P4000 Current H Geometric Offset reset by AcroMill to default on startup.
P4001 Current H Wear Offse reset by AcroMill to default on startup.
P4002—P4095 Available.
P4096—P20487 Range reserved for ACRXXXX Controller realtime parameters. Non Volatile

P100000—P1003424 SysVarStore System Parameters used by AcroMill

Axis commands can either have a literal number or an expression as argument.

Eg:

X(#1+#3) Y(P4+P6)

In the above command X Axis will move to the sum of what ever is in parameters 1 and 3. Y Axis will move to the sum of whatever is in parameters 4 and 6.

P40 = 1.555*Sin(45) + P6/P10

The above command will assign the value of the expression on the right side of the equation to parameter 40.

Math function supported:

+ , - , * , / ,

Logical functions supported.

> , < , = , >= , <= , AND, NOT,OR,XOR

Transcendental functions supported:

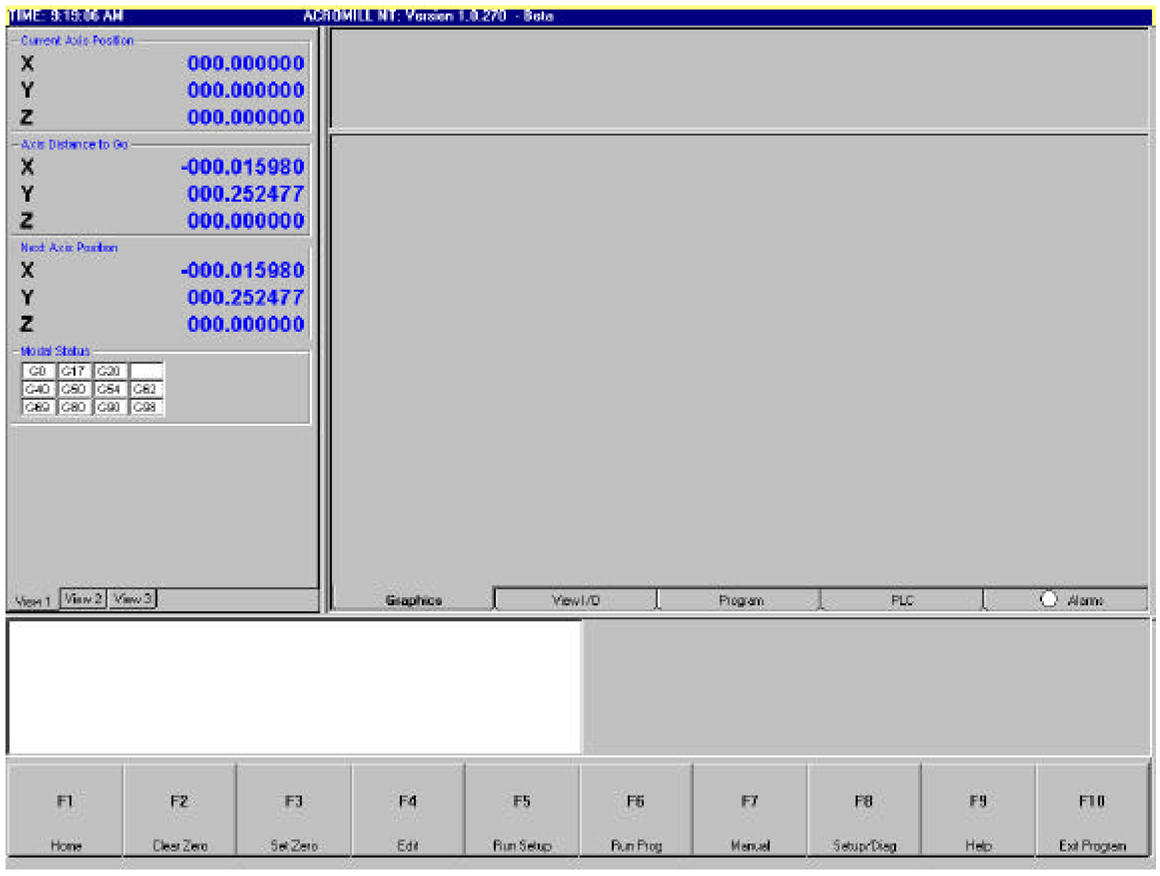
Sin, Cos, Tan, Atan,

Note the all parameters that are real time (parameters that reside on the ACRXXXX controller) will have a tendency to have certain access times associated with them. This is because as soon as an equation is executed that references a parameter on the board, the system cannot “look ahead” and read the parameter before the block with the reference is executed.

Menu Operation

MAIN MENU

This section describes the overall menu structure for the AcroMill program. The MAIN menu table below provides a brief description of the menu commands. See the following pages in this section for in depth menu descriptions and illustrations.



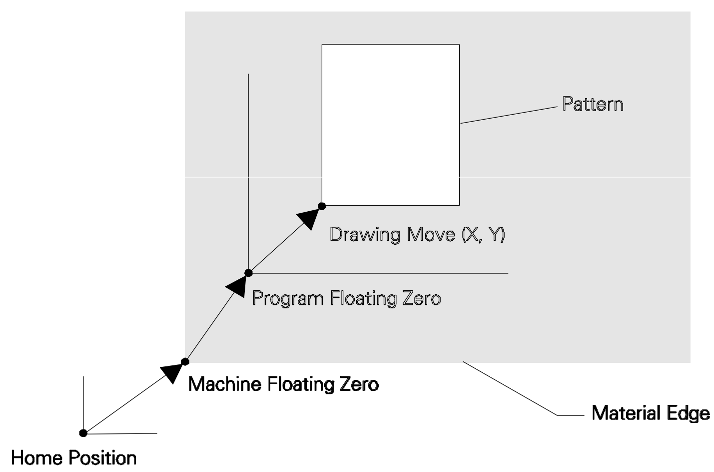
MAIN MENU Command	Parent Menu Description
HOME (F1)	Reference the machine
CLR ZERO (F2)	Not used
SET ZERO (F3)	Sets the axes to zero.
EDIT (F4)	Allows Editing a program.
RUN SETUP(F5)	Allows setting up a program or Library Part for running. Also allows setting modals codes and P Parameters.
RUN PROG (F6)	Initiates a menu to run a program
MANUAL (F7)	Initiates a menu to the manual interface (Jog, Handwheel MDI)
SET/DIAG (F8)	Initiates a menu to setup system parameters and run diagnostics
HELP (F9)	Brings up the HELP Screen
EXIT (F10)	Terminates the ACROMILL application.

Home

HOME (F1)

The HOME command is a physical reference an may be the position the cutting tool resides when the machine is not running a program or when disabled. Home is typically defined by a physical input such as a limit switch.   Below, is an example illustrating the HOME position.

Also see **Home** setup command described on page 65.



The HOME key activates the homing command cycle of the program for your machine. Before the machine can be jogged or any axis moved about, the HOME cycle must be executed. The setup of the HOME cycle is entered into system parameters via the HOME SETUP menu. The speeds, direction, and Home switch input number should be preset in this section (refer to HOMING THE MACHINE section).

Activate the Home sequence. A message in the dialog box lets you know the homing sequence is in progress.   The HOME cycle moves all axes to their reference limit switches.   After the switches are tripped, the motors will reverse direction *un-tripping* the switches. Then the motors will seek the marker on the encoder. When this execution is completed, all axes will display ZERO or the HOME PRESET position (HOME offset position is also entered in the SETUP screen).

To abort the homing sequence hit the STOP button.

Clr Zero

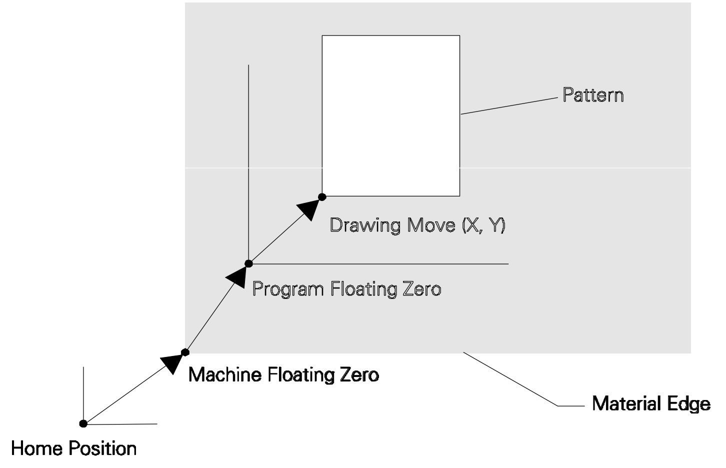
CLR ZERO (F2)

The CLR ZERO command is used to clear zero once the SET ZERO (F3)command is used. An example of how to use this command is jogging to a specific coordinate where X is 1.588 and Y is –2.12. Using the SET ZERO command will set the coordinates to zero. Using the CLR ZERO command will reset the coordinates to the values above.

Set Zero

SET ZERO (F3)

The SET ZERO command sets all or individual axes to zero from the front end. SET ZERO relates to the machine floating zero **only** that it sets all axes to zero. Below, is an example illustrating the SET ZERO command.

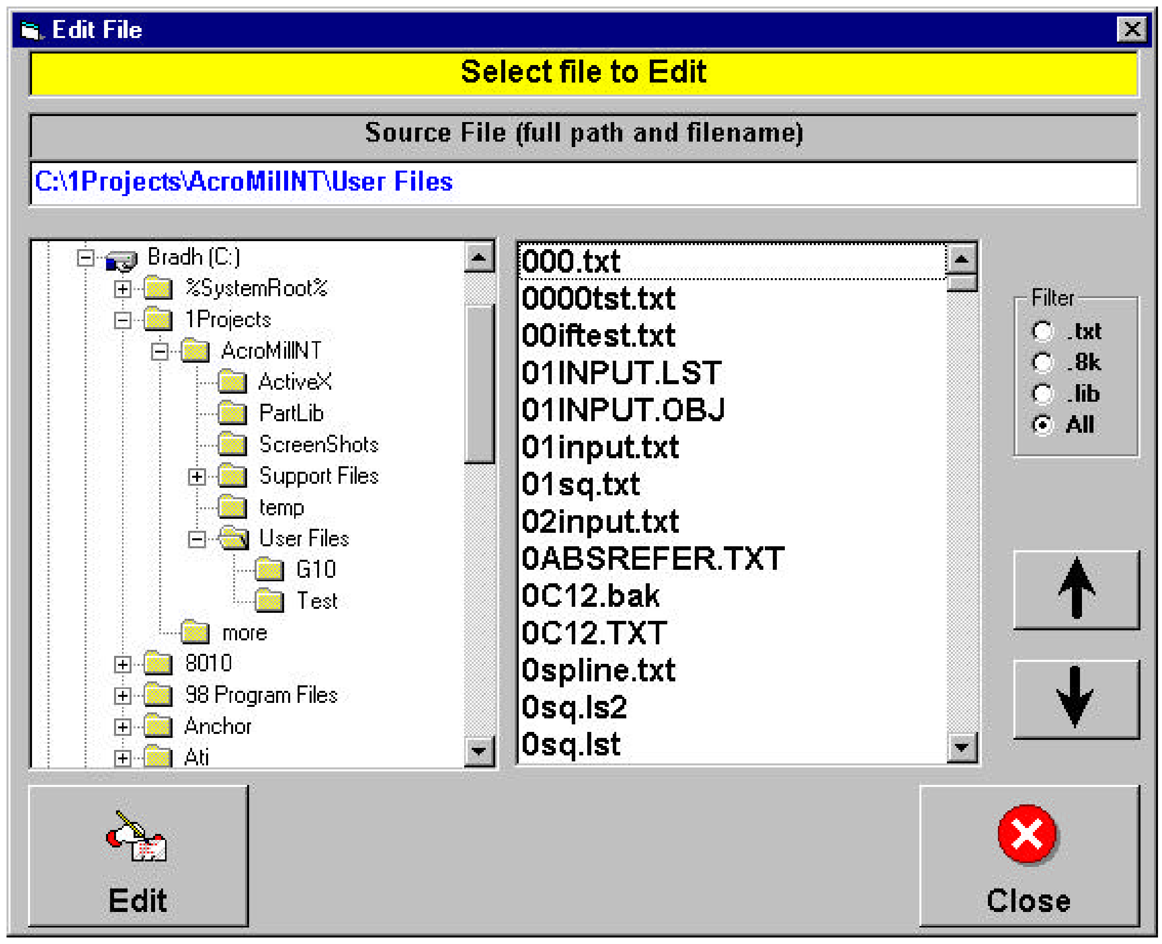


Note:

The SET ZERO command is similar to the FL ZERO command. However, it is for setting floating zeroes to a *zero* value. The SET ZERO command when activated sets **all** axes to a floating zero. The SET ZERO command can be located under RUN PROGRAM and MAIN menus.

Edit Prog

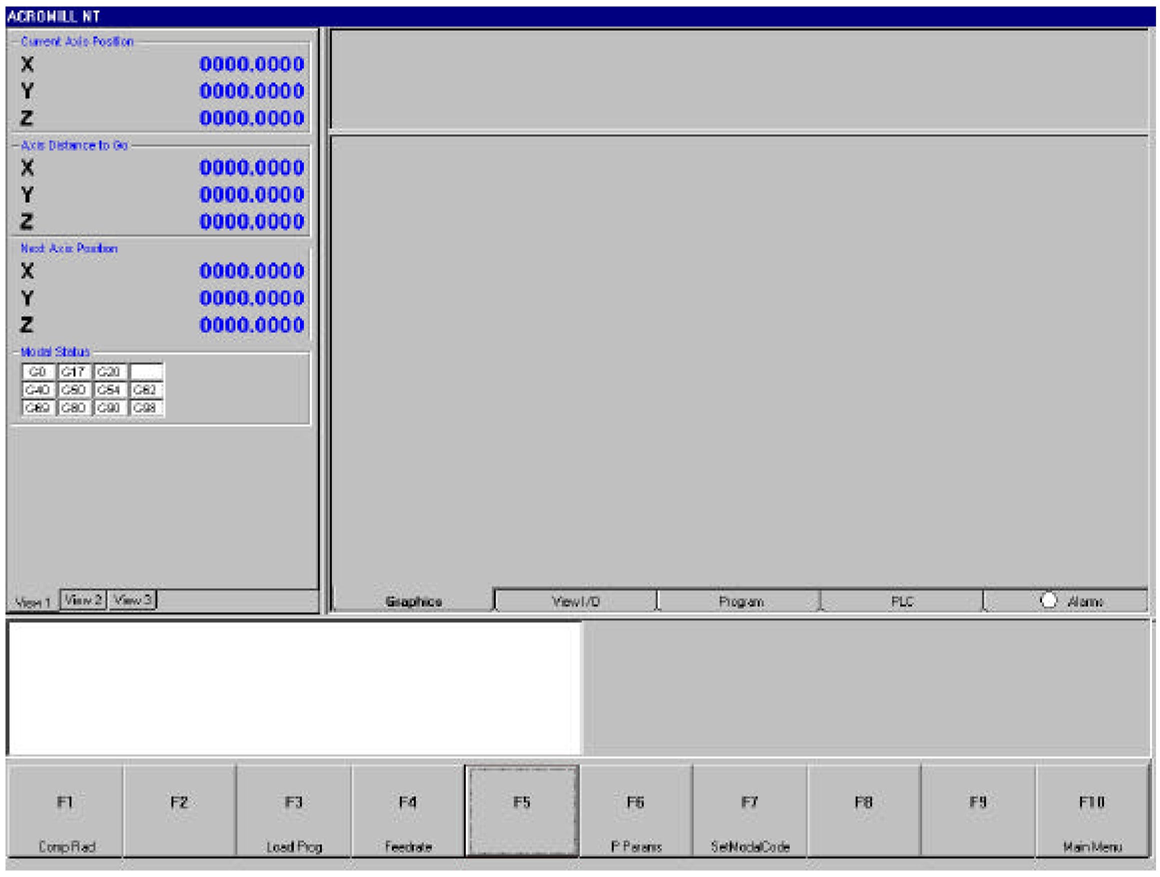
This Screen allows editing a program. The currently executing program cannot be edited. To edit the program that is currently running, its execution must be stopped. Any other program can be edited at any time.



The above menu allows selecting any given file and once the file is selected an edit box shows the file to be edited.

Run Setup

RUN SETUP MENU (F4)



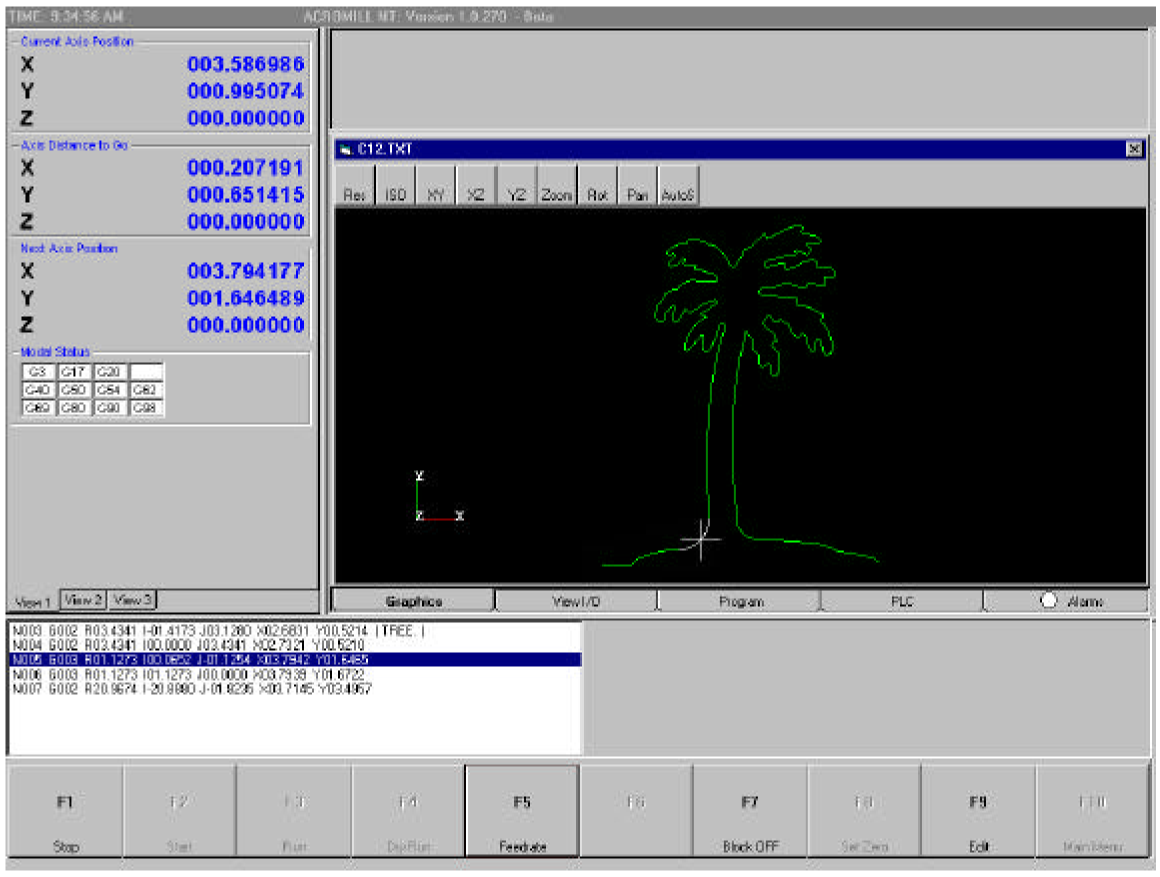
The RUN SETUP command shows the above menu. The menu allows activating LIBRARY PARTS or customer programs, and downloading parts from the serial port (LINK).

LOAD PROG Command	Parent: MAIN MENU Description
CMP RAD (F1)	Sets the Compensation Width of cutting tool
LOAD PROG (F3)	Load a user program, Library Part, DXF File, Batch file or Drip feed.
FEEDRATE (F4)	Sets up feedrate if no feedrate is specified in the program.
P PARM (F6)	Allows presetting P parameters.
SET MODALS (F7)	Allows setup of all modal G Codes.
TO MAIN MENU	Returns to the MAIN menu

Run Prog

RUN PROG→ RUN PROG MENU (F6)

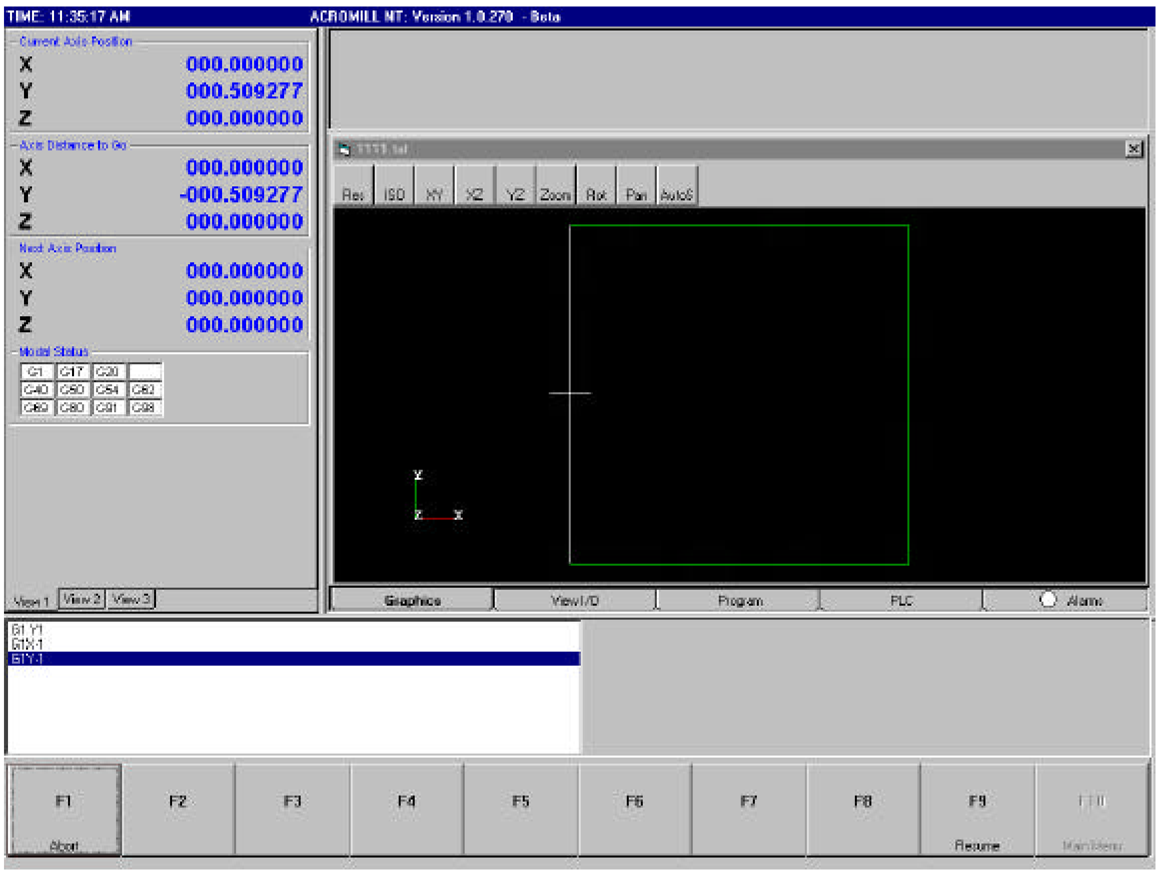
The RUN PROG command key asks for the starting line number and starts the loaded program.



RUN PROG **Command**	**Parent: MAIN MENU** **Description**
STOP (F1)	Stops the running program
START (F2)	Start running the loaded program
RUN (F3)	Run the loaded program
DRY RUN (F4)	Run the program at maximum feed
FEEDRATE (F5)	Manipulate Feedrate Parameters
BLOCK (F7)	Switch between Block and Auto Modes
SET ZERO (F8)	Sets all axes to a zero value
EDIT(F9)	Allows Editing a different program.
MAIN MENU (F10)	Returns to the Main Menu

Stop

RUN PROG MENU (F6) → STOP (F1)

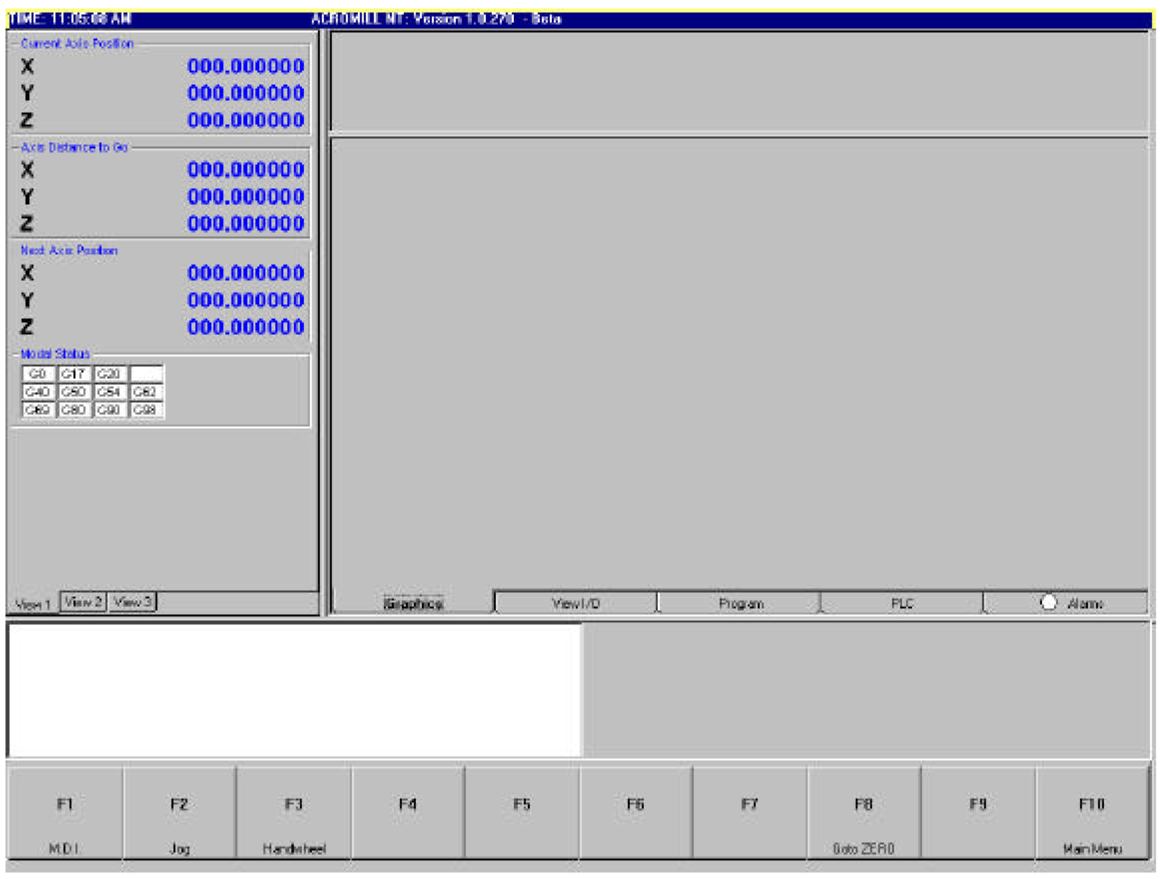


The STOP menu allows the machine operator to stop or pause the PROGRAM SEQUENCE. At the Status line in the Program Status Window, "----PAUSE----" is indicated when the STOP key is activated.
The following STOP menu table provides a brief description of the menu commands. See STOP menu under Level 3 for in depth menu descriptions and illustrations.

STOP MENU Command	Parent: RUN PROG MENU Description
ABORT (F1)	Resets the program to the start position
RESUME (F8)	Resume normal running of the program

Manual

MANUAL→ MANUAL MENU (F6)



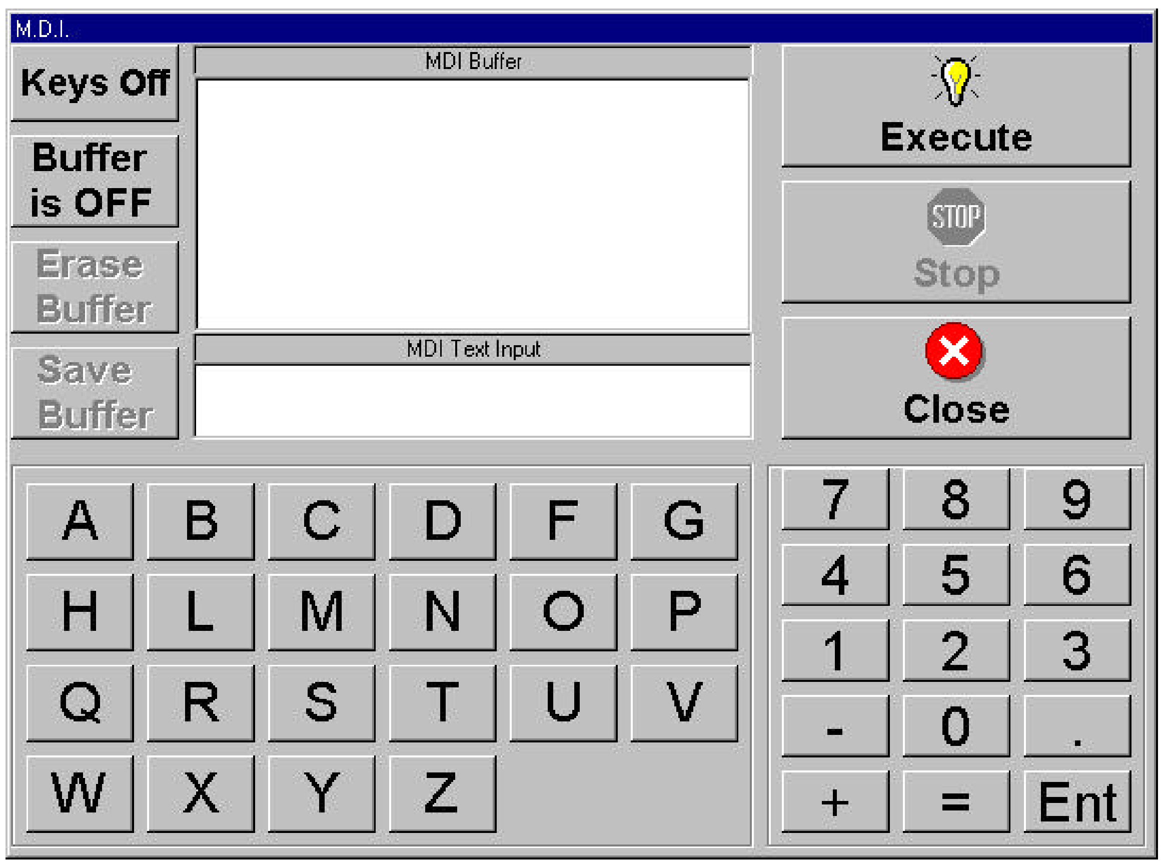
The above menu allows selection of various ways to move the machine manually.

The MANUAL menu allows manually moving the machine via Jogging, Handwheel, Manual Data Input (MDI) and Joystick. The following menu provides a brief description of the MANUAL menu

MANUAL MENU Command	Parent: MAIN MENU Description
MDI(F1)	Permits direct issuance of RS-274D commands from command line by interface keyboard.
JOG (F2)	Specifies fast jog in continuous or incremental motion.
HANDWHEEL (F3)	Permits movement of one or all axes by means of a handwheel
GOTO ZERO (F8)	Causes the machine to return to floating zero.

MDI

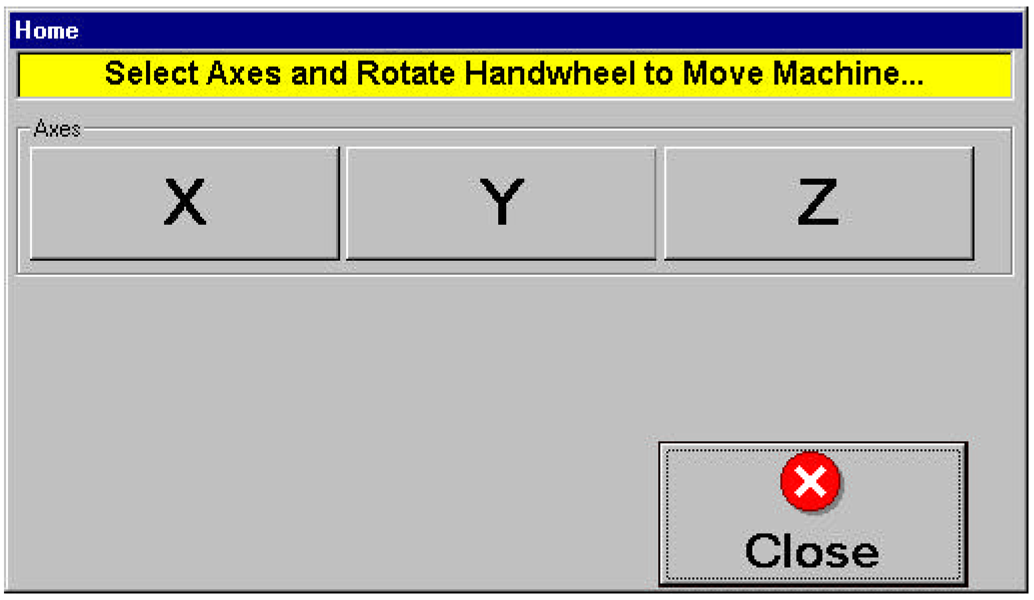
The MDI (Manual Data Input) screen will appear as follows.



This screen allows entering all valid RS274D commands and executing them as soon as the Execute button is pressed.
This screen also allows accumulating the MDI commands in the MDI buffer so that they can be transferred into a program.

Handwheel

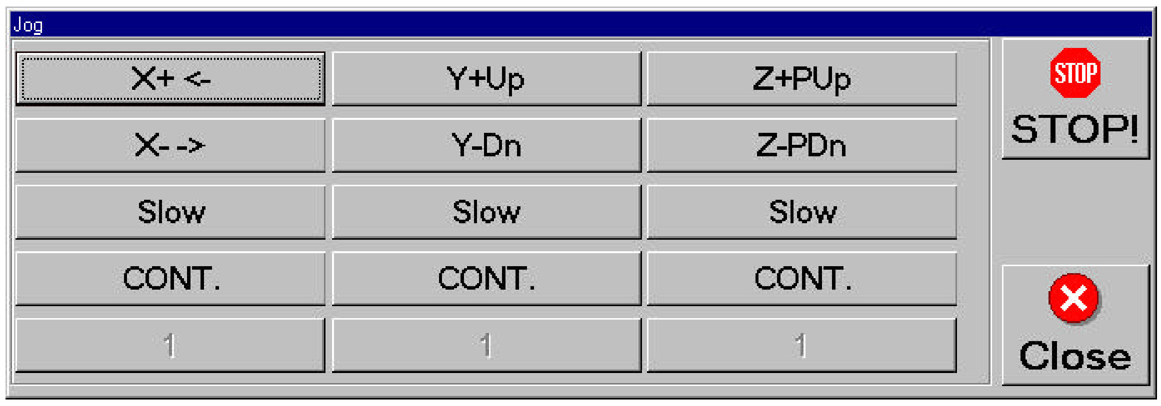
The HANDWHEEL screen appears as follows.



By selecting one or more axes, the manual pulse generator option (HANDWHEEL) will cause the axis to move.

Jog

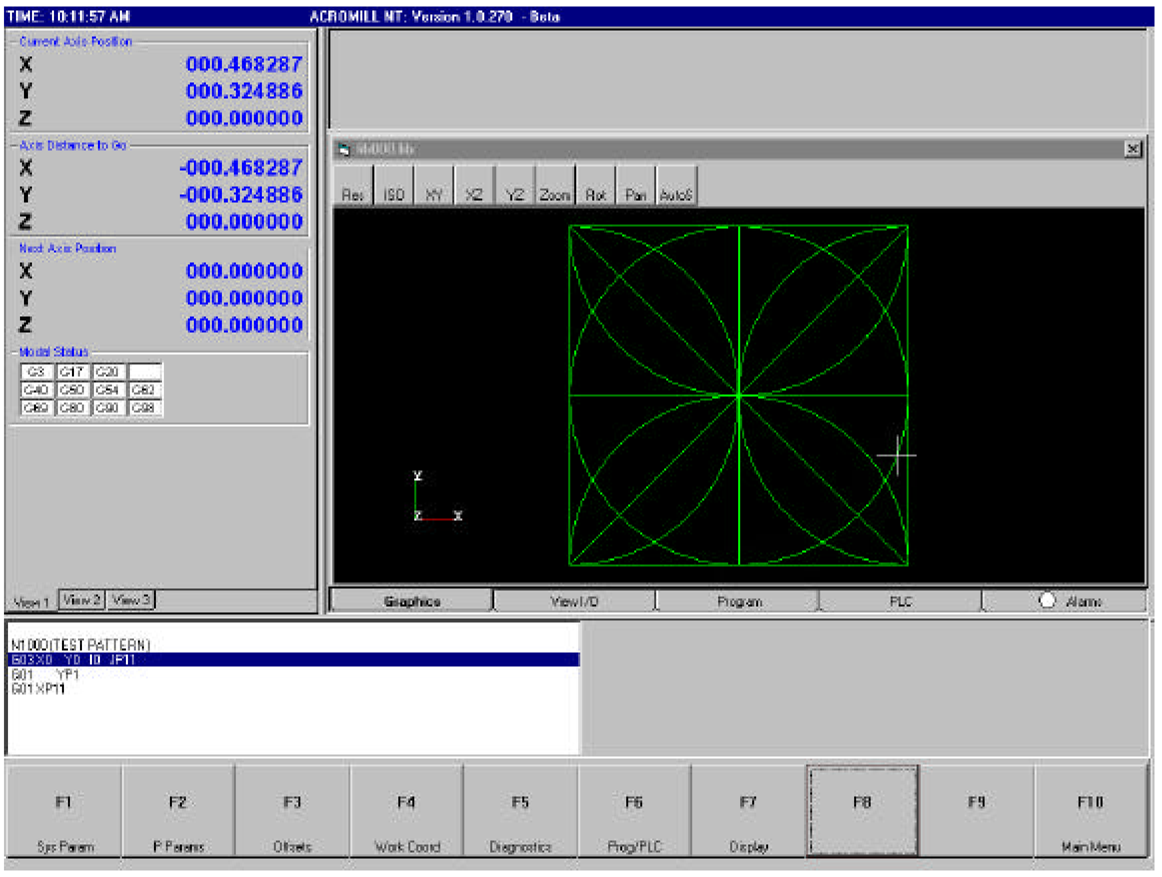
The Jog screen appears as follows.



All axis on the machine can be jogged either via JOYSTICK, MOUSE or TOUCHSCREEN options.

Set/Diag

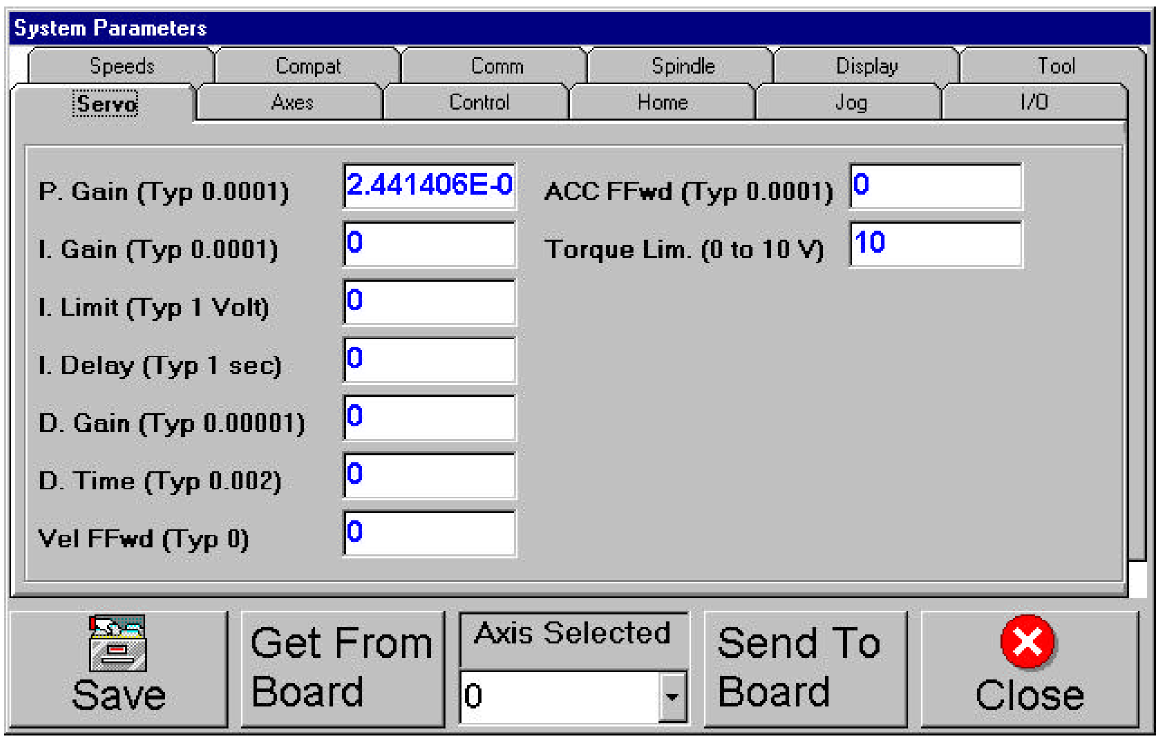
The SET/DIAG menu allows setting up the system parameters, Offsets,Work Coordinates, Diagnotics, Programs/Plc or Change Display Characteristics. The machine system parameters deal with defining I/O assignments, axis feeds, limits, pulses per inch, accelerations, decelerations, COMM port communications, etc.



Sys Params

The SYSTEM PARAMETER (F1) brings up the following tab controls for setting various types of settings.

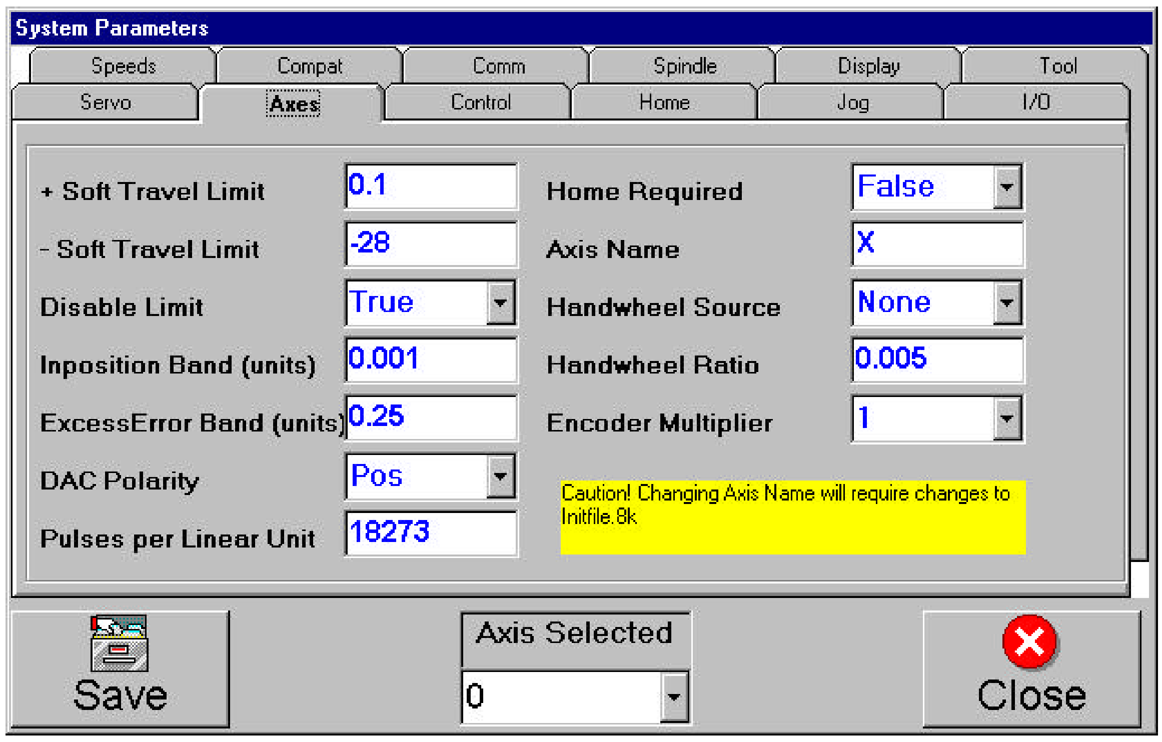
SERVO



The SERVO tab allows changing values of the following parameters for each axis. The table below shows the default values, units, and a descriptive summary.

Parameter	Typical Value	Default Value	Units	Description
P Gain	0.001	0.00244	volts/pulses of following error	Proportional Gain
I Gain	0.001	0.00000	volts/second/pulses of following error	Integral Gain
I Delay	1.0	0.00000	second	Integral Delay
I Limit	1.0	0.00000	volts	Integral Limit
D Gain	0.00001	0.00000	volts/pulses of following error/second	Derivative Gain
D Time	0.002	0.00000	second	Derivative Sample Time
ACC FFwd	0.0001	0.00000	volts/incoming setpoint pulses/sec^2	FeedForward Acceleration
Vel FFwd	0	0.00000	volts/incoming setpoint pulses/second	FeedForward Velocity
Torque Limits	0-10	10	volts	Torque Limits

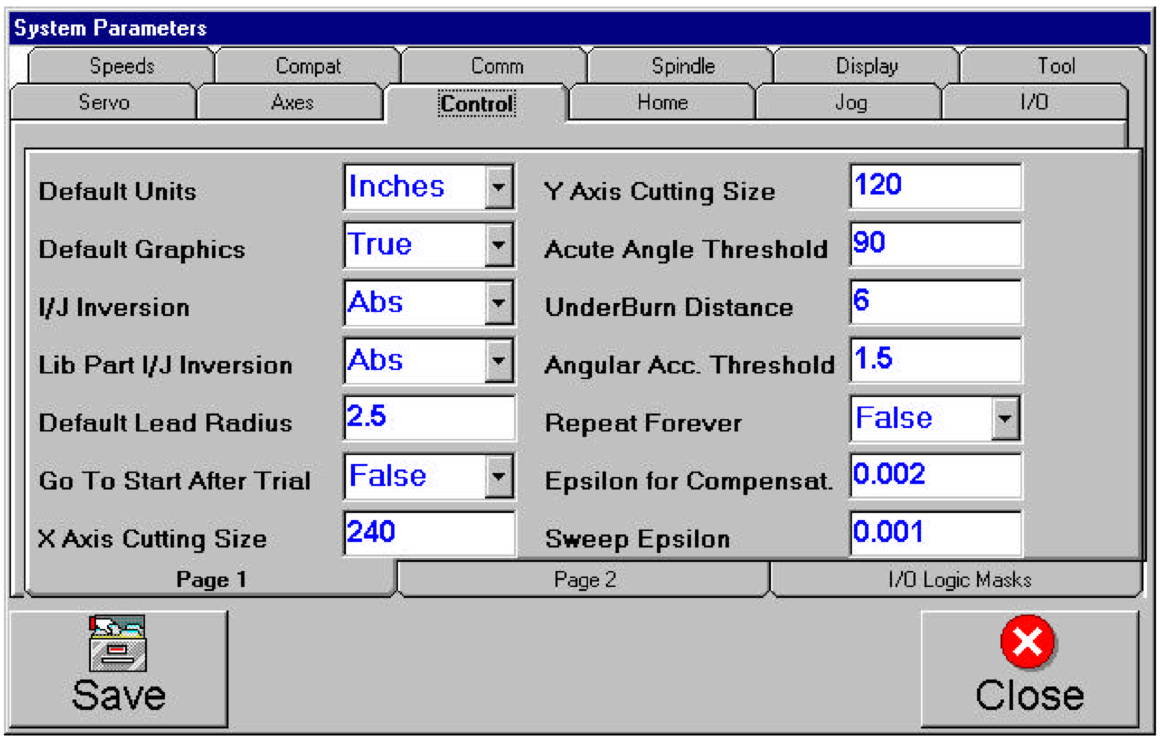
Axes



The AXES tab allows setting up of Soft Limits, Inpositio

Parameter	Typical Value	Default Value	Units	Description
+Soft Limit	Machine+	0	Inch or mm.	Legal +Travel from Home Position
-Soft Limit	Machine-	0	Inch or mm.	Legal –Travel from Home Position
Disable Limit	False	True	Boolean	Set True if NO Limits are required
Inposition Band	0.001	0.00000	Inch or mm.	Acceptable Position Error to call Axis Done moving.
ExcessErr Band	0.25	0.00000	Inch or mm	Maximum following error for which the machine is allowed to operate.
Dac Polarity	Pos	Pos	Sign	Sets direction of axis travel for +ve moves.
Pulses/Unit	10000	10000	Float	Pulses from Feedback device per Unit of travel
Home Reqd	True	False	Boolean	Set True if HOME is required for machine
Axis Name	X,Y….	X,Y…	NA	Typical names are X,Y,Z,A,B,C,U,V
Handwheel Src	Axis4	None	NA	Points to Handwheel Encoder input if installed
HandwheelRatio	0.005	0.005	Inch or mm	100 clicks of handwheel usually = 0.1 Inch.
Encoder Mult	4	4	Integer	Sets encoder +/-multiplier and direction of axis travel for +ve moves

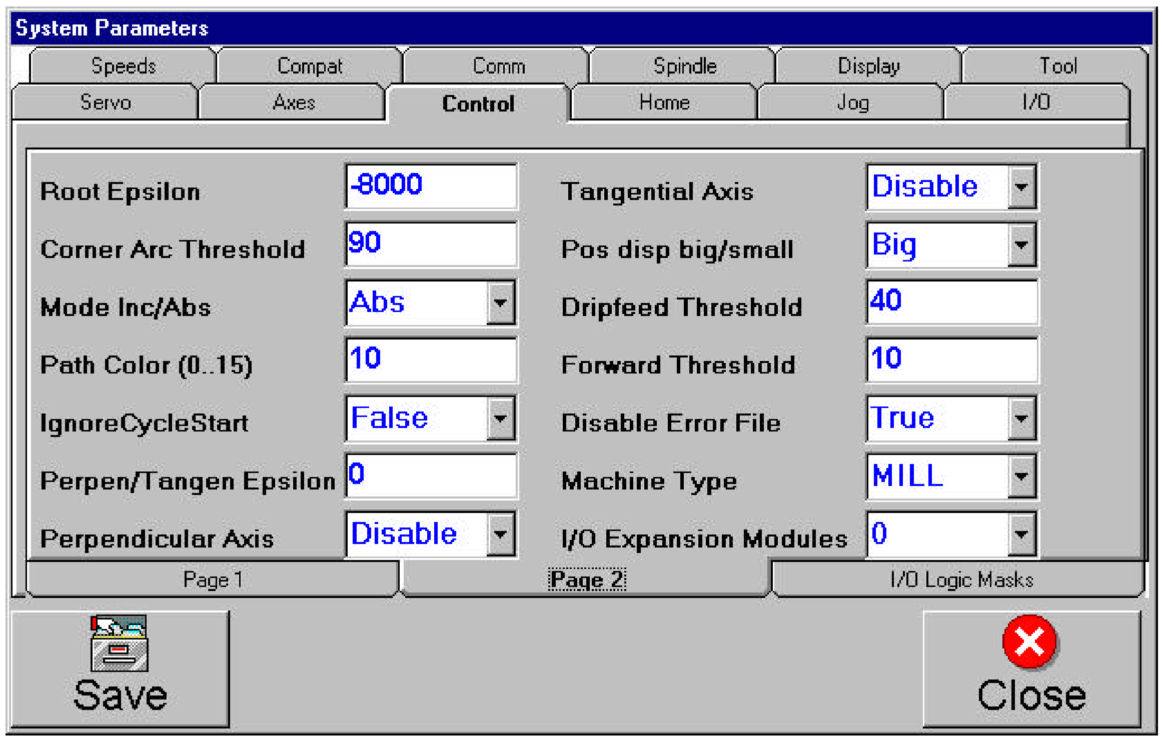
Control



Parameter	Typical Value	Default Value	Units	Description
Default Units	Inch	Inch	Inch or mm.	Sets the System for Inch or Metric Operation
Default Graphics	True	True	Boolean	Allows showing Graphical part.
I/J Inversion	ABS	ABS	NA	Sets one of three ways to interpret I/J in a circle command. See G200 explanation for more info.
Lib Part I/J Inversion	ABS	ABS	NA	Sets one of three ways to interpret I/J in a circle command in Library Parts. See G200 description for more information.
Default Lead in Radius	0.25	0.25	Inch or mm	For Library parts and OFF Path resuming, specifies Lead in radius.
Goto Start After Start	False	False	Boolean	After DRY RUN makes machine go to the initial starting position automatically.
X Axis Cutting Size	24	24	Inch or mm	Specifies Blank Material Size X Dimension
Y Axis Cutting Size	24	24	Inch or mm	Specifies Blank Material Size Y Dimension.
Z Axis Cutting Size	24	24	Inch or mm	Specifies Blank Material Size Z Dimension
Acute Angle1 threshold	0	0	Degrees	Specifies Corner Slow down vector angle to start slowing down.
Acute Angle2	0	0	Degrees	Specifies Corner Slow vector angle for compete stop
Under Burn Distance	0	0	Inch or mm	Specifies distance away from endpoint at which cutting device auto height sensing is turned off
Repeat Forever	False	False	Boolean	Allows running of loaded part in a forever

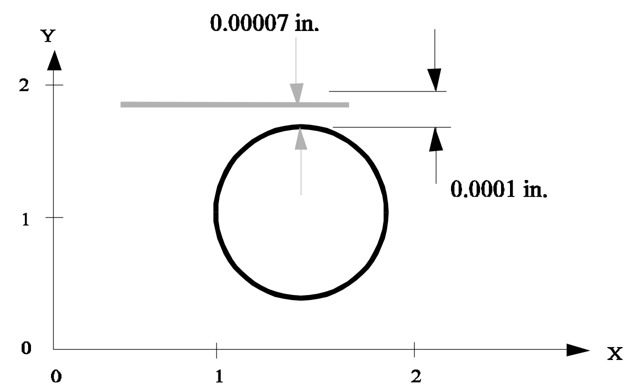
				loop.

Page 2 of the Controls menu offers the following choices.



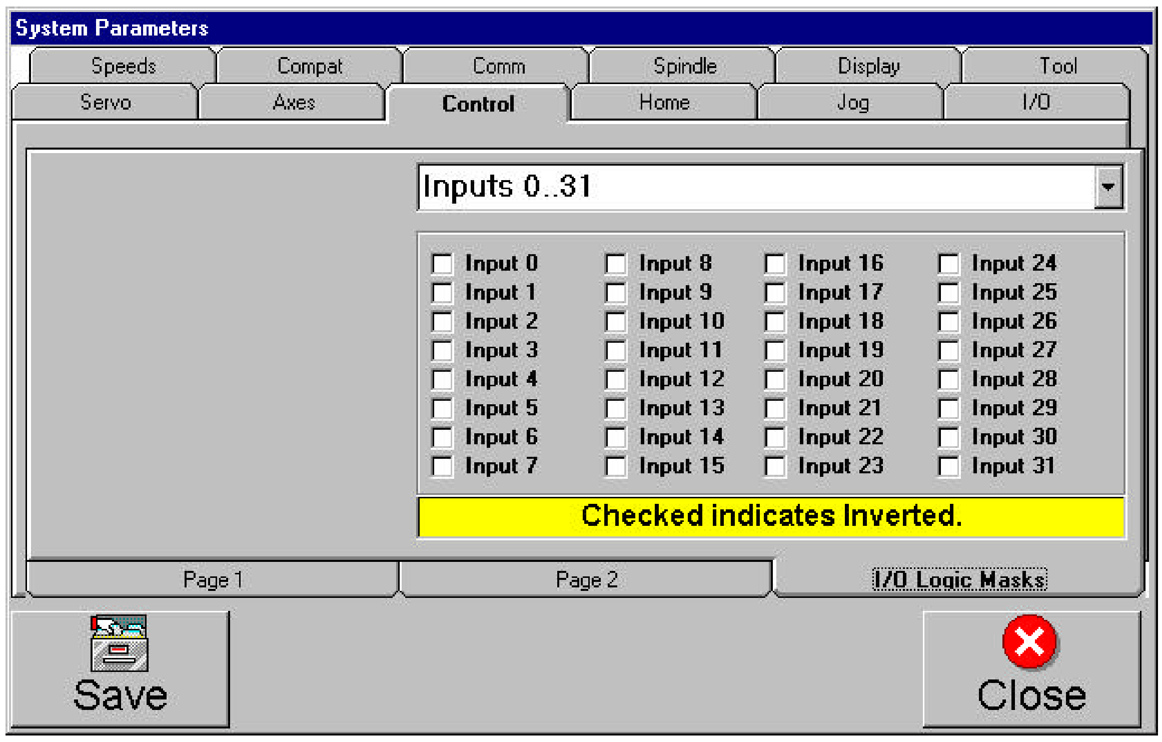
Parameter	Typical Value	Default Value	Units	Description
Epsilon for Compensation	0.002	0.002	Inch or mm.	Epsilon for Compensation sets the compensation for CAD drawings where lines do not meet. For example, if Epsilon for Compensation is set for 0.0001, all lines will be interpreted as joined within this threshold range. See Drawing below.
Sweep Epsilon	0.01	0.01	Radians.	The Sweep Epsilon defines in radians the smallest valid sweep of an arc below which the arc can be treated as a straight line. This is used in cutter compensation math. Sweep Epsilon defines an arc as two points. If the angle ϕ is small enough as it approaches zero (this depends on such variables as arc radius or distance between the start and stop points), the Sweep Epsilon will interpret the start and stop points as one point.
Root Epsilon	-4	-4	Integer	The circle to circle intersections require a square root calculation. If the circles do not meet, the square root argument becomes negative. The ROOT EPSILON allows the user to fudge this "GAP" between the arcs by telling the control to ignore the gap and force an intersection.
Power Up Incremental or Absolute Mode	ABS	ABS	NA	Forces Control to power up either in Absolute or Incremental mode.
Path Color	10	10	Integer	Selects color to show tool path color
Perpen/Tangent Epsilon	0	0	Degrees	Not Implemented as yet.
Tangential Axis	Disable	Disable	NA	Specifies one of the axes as being required to be tangential during Auto Tangential moves.
Position Display	Big	Big	NA	Specifies Position display to be big or small font size.
DripFeed Threshold	40	40	Integer	Specifies how many blocks get calculated before machine starts moving.
Forward Threshold	10	10	Integer	Specifies how many blocks to calculate forward.
Disable Error File	True	True	NA	Disable Error Logging file.
Machine Type	MILL	MILL	NA	Specifies type of Machine under control
I/O Expansion Modules	0	0	Integer	Specifies how many I/O Expansion Modules are attached to Controller

Drawing Explaining Epsilon for Compensation



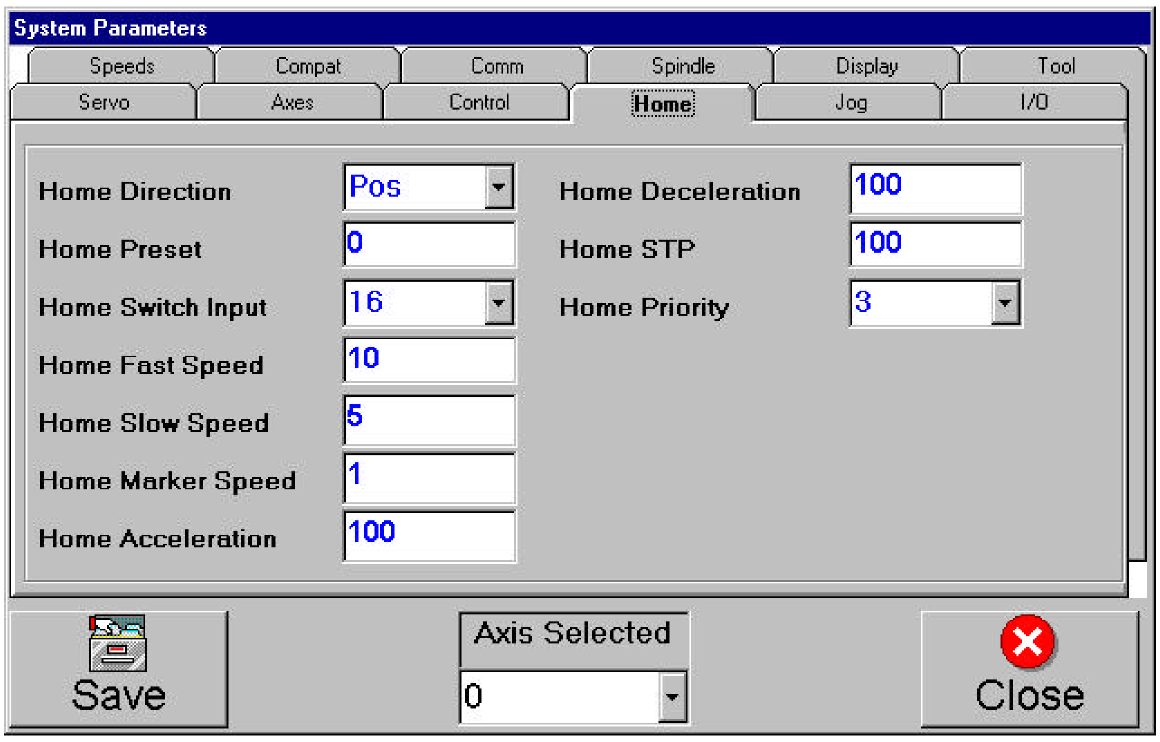
Assuming the Epsilon parameter is set for 0.0001", in the above drawing, the actual distance is 0.0007 inches which is less than the Epsilon compensation setting. In this example, AcroMill would interpret the line and circle as being tangential or joined.

Page three of the Control Screen shows the following parameters.



This tab allows selecting the polarity of inputs and outputs. Default is none of the Inputs and Outputs Inverted.

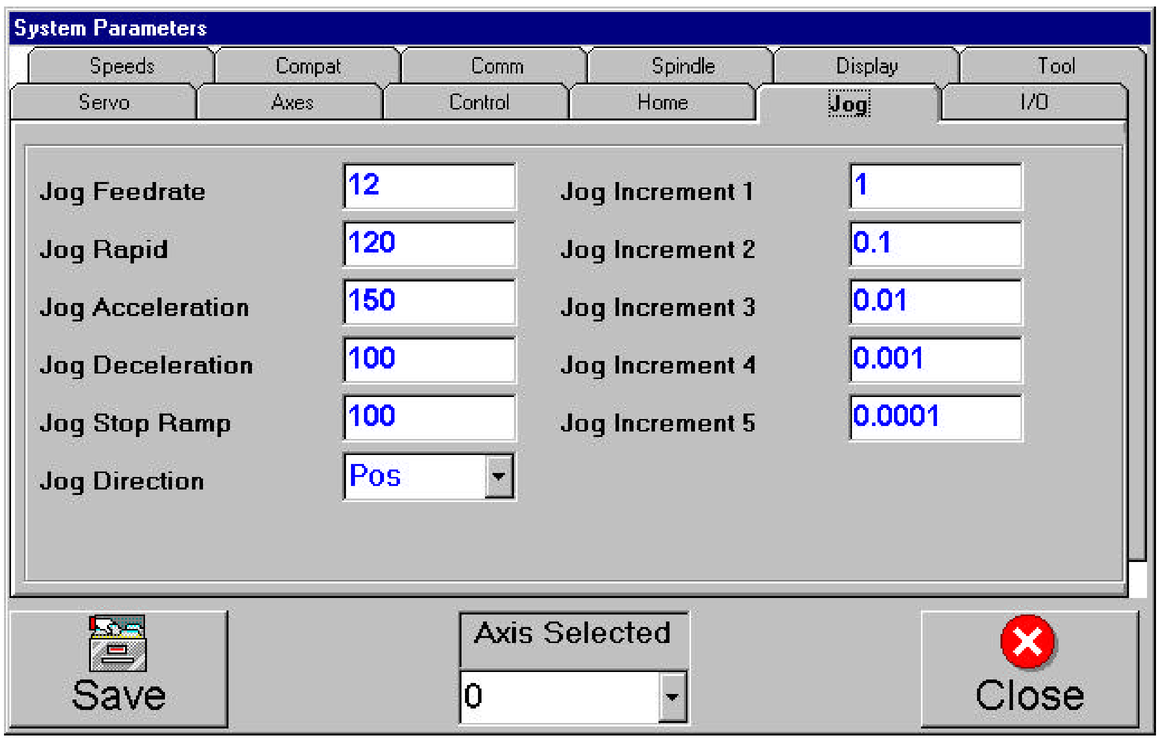
Home



Parameter	Typical Value	Default Value	Units	Description
Home Direction	POS	POS	NA	Select positive or negative direction for the axis to travel towards the Home switch.
Home Preset	0	0	Inch or mm.	If the Home position is required to be non zero, enter any value in this paramter.
Home Switch Input	16	16	Integer	This is the physical Input on the controller that the Home switch is wired to.
Home Fast Speed	100	10	Inch or mm per minute	Specifies speed to go towards the home switch.
Home Slow Speed	10	5	Inch or mm per minute	Specifies speed to go away from the home switch after contacting with the home switch.
Marker Speed	10	2	Inch or mm per minute	Specifies speed to search for the marker. Since the Acroloop controllers have Hardware Capture ckts, this speed can be very fast.
Home Acceleration	100	100	Inch or mm per minute	Specifies axeis Acceleration in units per sec per second.
Home Deceleration	100	100	Inch or mm per minute	Specifies axis Deceleration in units per sec per second.
Home STP	100	100	Inch or mm per minute	Specifies axis STOP Ramp in units per sec per second
Home Priority	1	1	Integer	Specifies what order the axis is going to Home. 0=Disable, 1=Home 1st, 2=Home Second….etc

On this and all menus, make sure the SAVE button is hit when making changes otherwise the changes will not be saved .

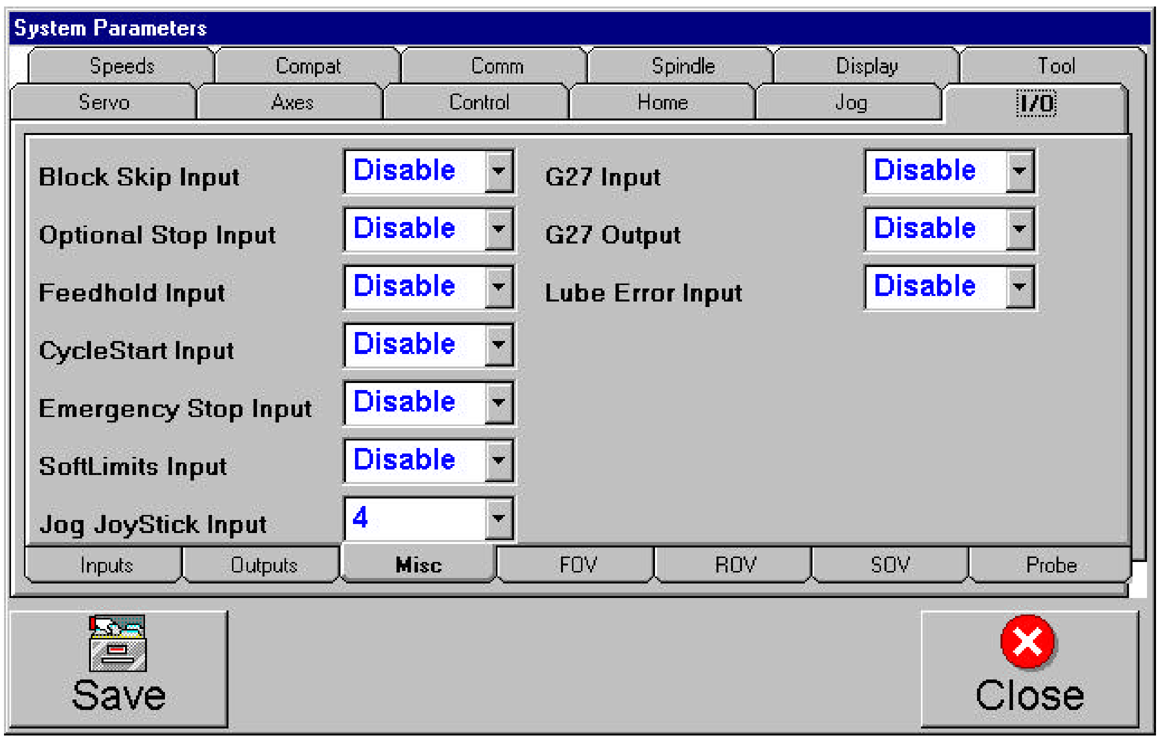
Jog



The Jog tab allows editing of the following parameters.

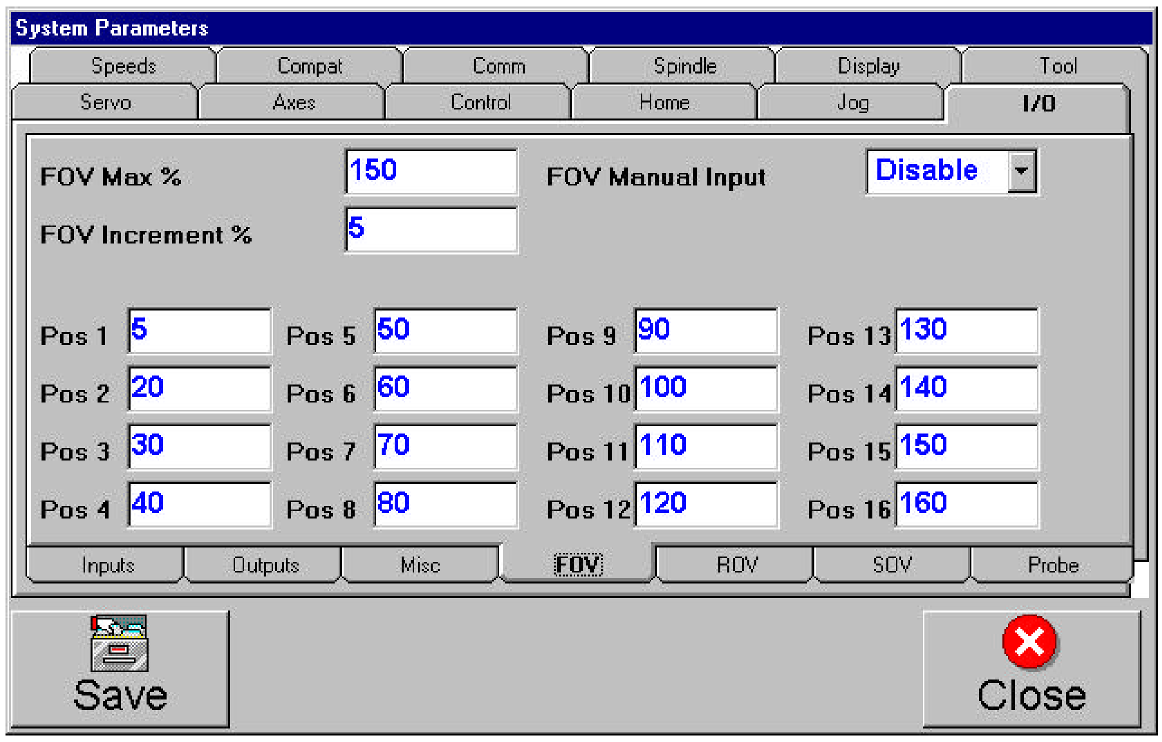
Parameter	Typical Value	Default Value	Units	Description
Jog Feedrate	10	10	Inch or mm per minute	Selects the Slow Jog Speed in Units per minute
Jog Rapid	100	100	Inch or mm per minute	Selects the Fast Jog Speed in Units per minute
Jog Acceleration	150	150	Inch or mm per sec per second	This is the Jog Acceleration in Units per sec per sec.
Jog Deceleration	100	10	Inch or mm per sec per second	This is the Jog Deceleration in Units per sec per sec.
Jog STP Ramp	100	100	Inch or mm per sec per second	This is the Jog Stop Ramp in Units per sec per sec.
Jog Direction	POS	POS	NA	Specifies Plus or Minus direction for particular key being hit.
Jog Increment1	1	1	Inch or mm	For Incremental Jog Mode, this parameter specifies distance moved when a particular Key is hit. There are a total of 5 Selections.
Jog Increment 2	0.1	0.1	Inch or mm	For Incremental Jog Mode, this parameter specifies distance moved when a particular Key is hit. There are a total of 5 Selections.
Jog Increment 3	0.01	0.01	Inch or mm	For Incremental Jog Mode, this parameter specifies distance moved when a particular Key is hit. There are a total of 5 Selections.
Jog Increment 4	0.001	0.001	Inch or mm	For Incremental Jog Mode, this parameter specifies distance moved when a particular Key is hit. There are a total of 5 Selections.
Jog Increment 5	0.0001	0.0001	Inch or mm	For Incremental Jog Mode, this parameter specifies distance moved when a particular Key is hit. There are a total of 5 Selections.

I/O



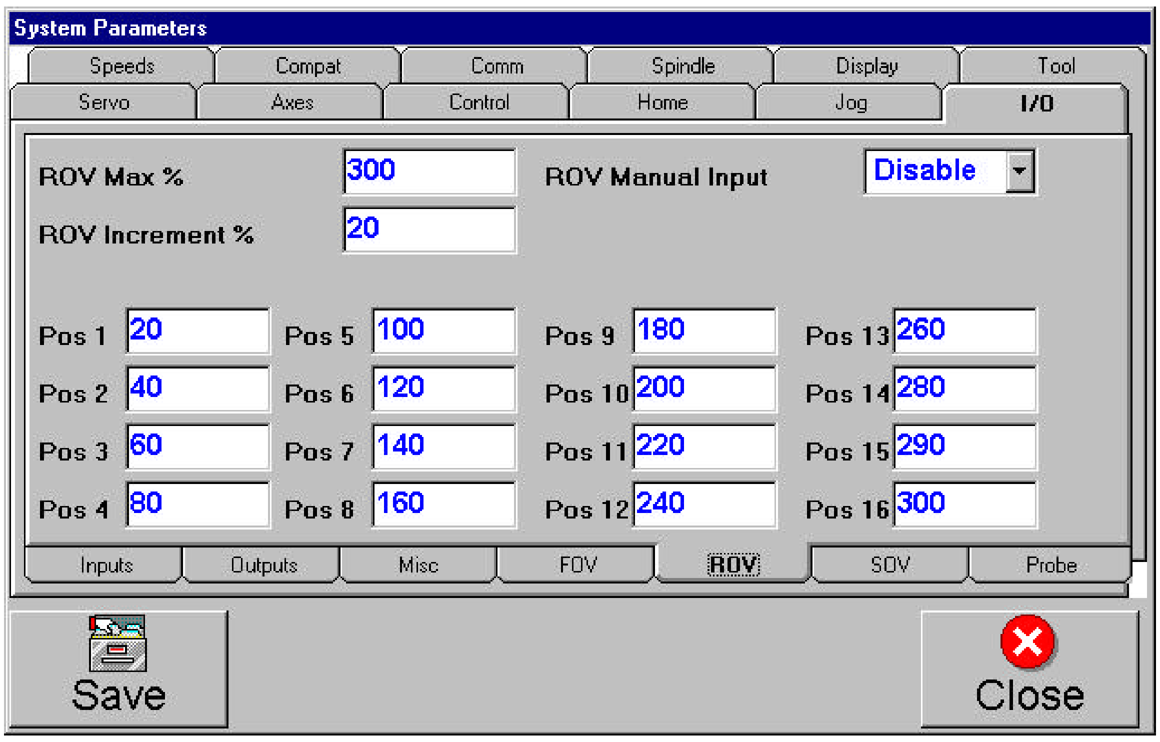
Parameter	Typical Value	Default Value	Units	Description
Block Skip	User Choice	Disable	Integer	Selects input wired to the BLK SKIP Switch
Optional Stop	User Choice	Disable	Integer	Selects input wired to the OPT STOP Switch
FeedHold Input	User Choice	Disable	Integer	Selects input wired to the FeedHold Switch
Cycle Start Input	User Choice	Disable	Integer	Selects input wired to the Cycle Start Switch
Emergency Stop	User Choice	Disable	Integer	Selects input wired to the E-Stop Relay
Soft Limits Input	User Choice	Disable	Integer	
Jog JoyStick Input	User Choice	Disable	Integer	Points to the start (lowest number) input of a consequtive bank of 4 inputs wired to a 2 axes Jog joystice.
G27 Input	User Choice	Disable	Integer	
G27 Output	User Choice	Disable	Integer	
Lubrication Error	User Choice	Disable	Integer	Selects input wired to the Lubrication fault contact.

FOV



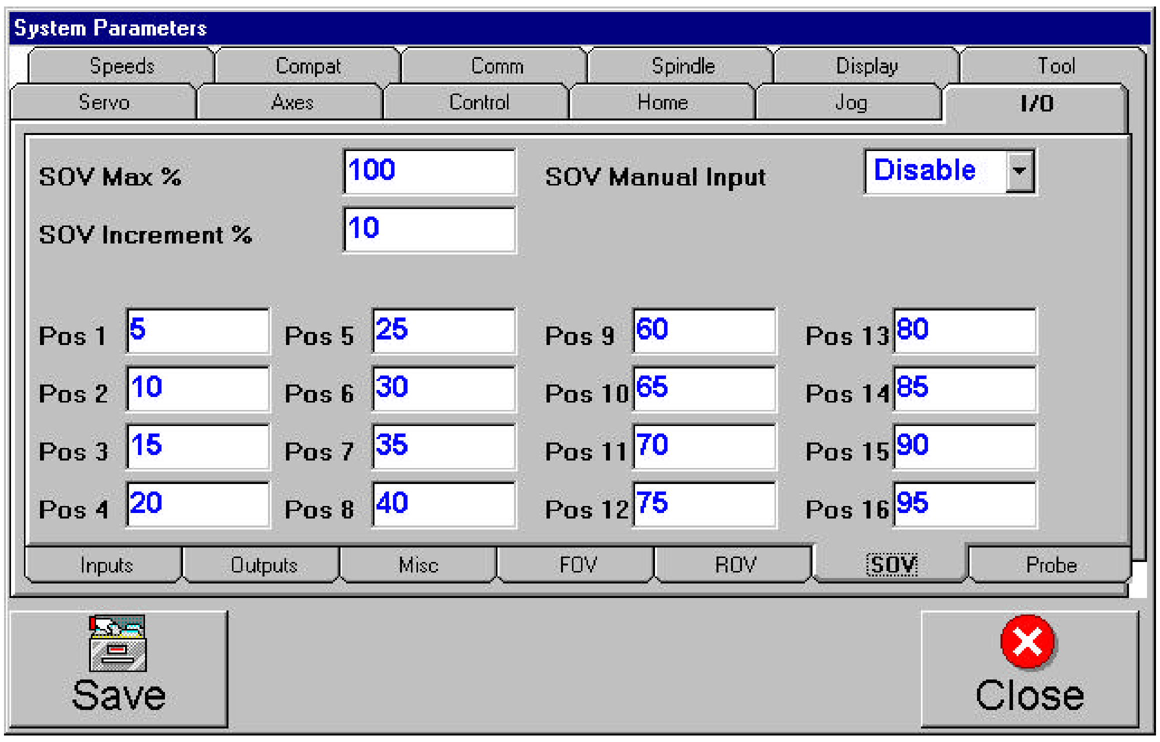
This tab allows associating a FEEDRATE OVERRIDE POT (OPTION) clicks to correspond to a particular feedrate percentage override.
In the standard override pot offered by Acroloop, there are 16 position. Each of these positions can be associated with an ever increasing percentage as the user desires.

ROV



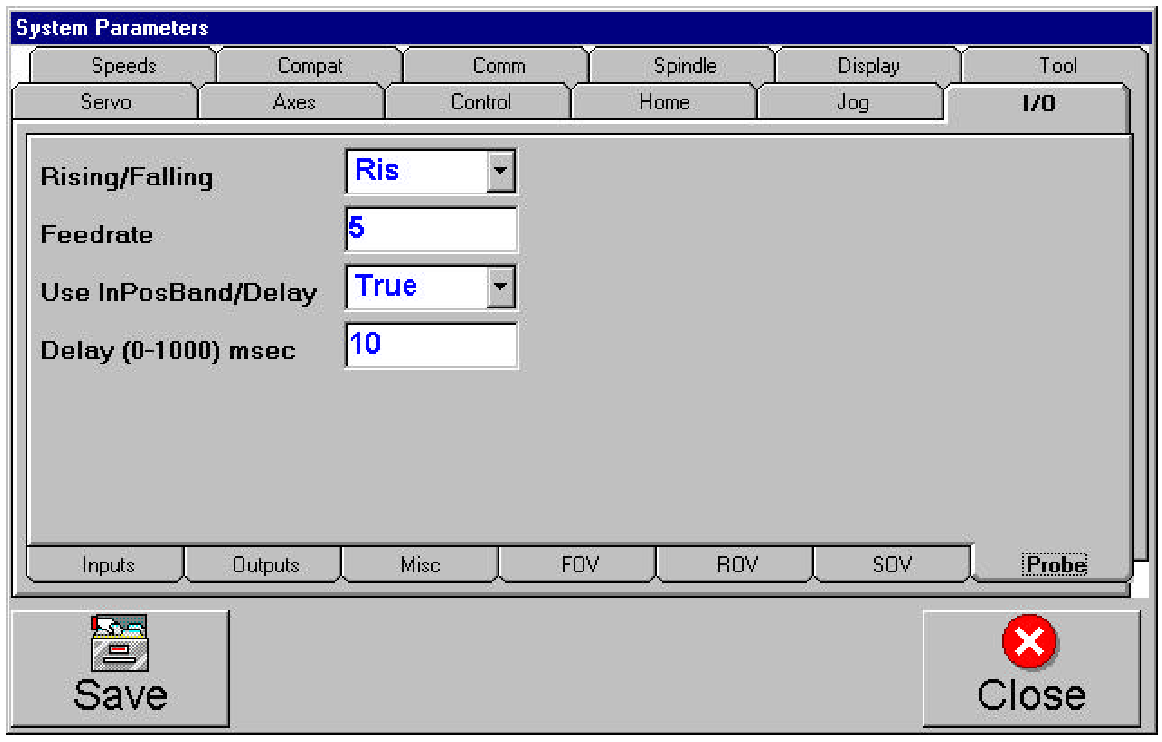
This tab allows associating a RAPID OVERRIDE POT (OPTION) clicks to correspond to a particular RAPID moves percentage override.
In the standard override pot offered by Acroloop, there are 16 position. Each of these positions can be associated with an ever increasing percentage as the user desires.

SOV



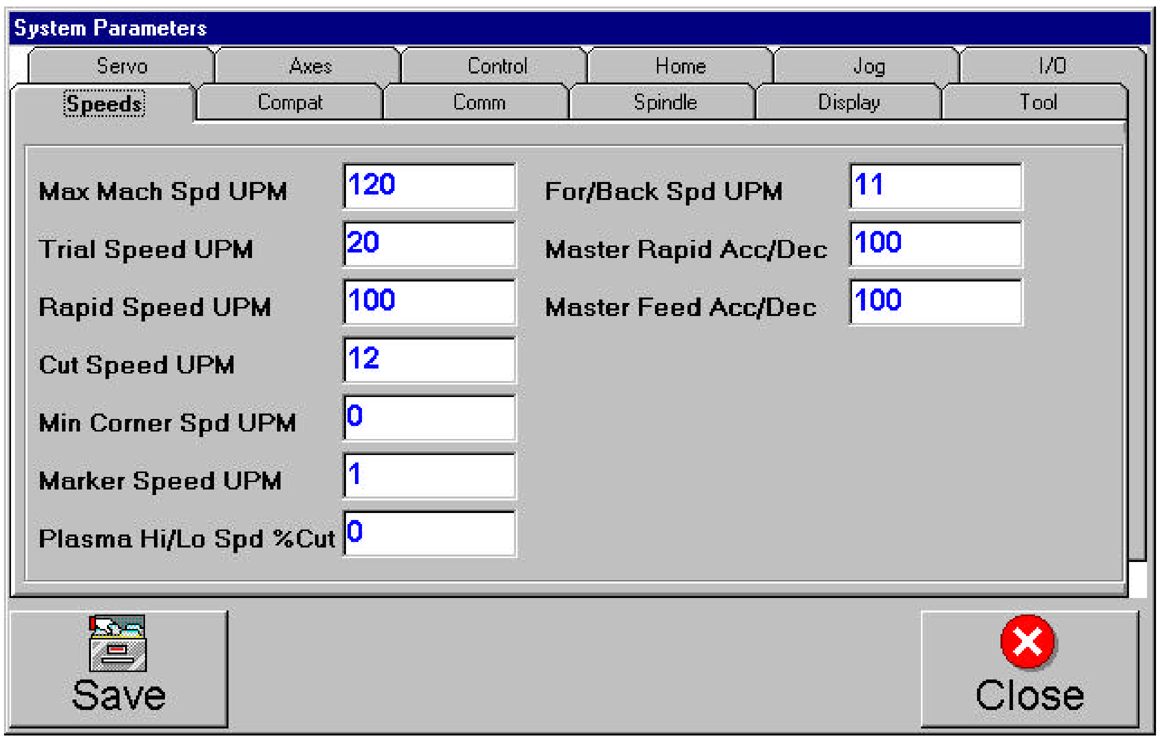
This tab allows associating a SPINDLE SPEED OVERRIDE POT (OPTION) clicks to correspond to a particular SPINDLE SPEED percentage override.
In the standard override pot offered by Acroloop, there are 16 position. Each of these positions can be associated with an ever increasing percentage as the user desires.

Probe



This tab sets up the PROBING command (G31) to work with a probe that typically hits a part surface and facilitates digitizing on the control.

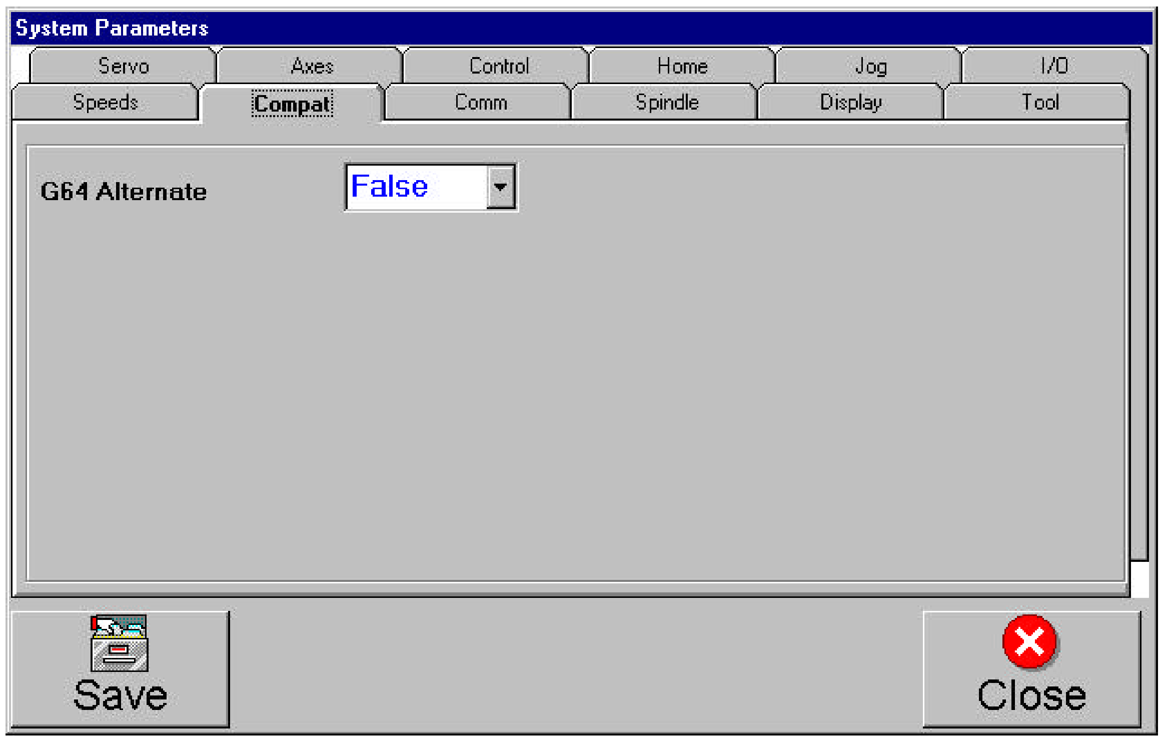
Speeds



This tab allows setting up various speed

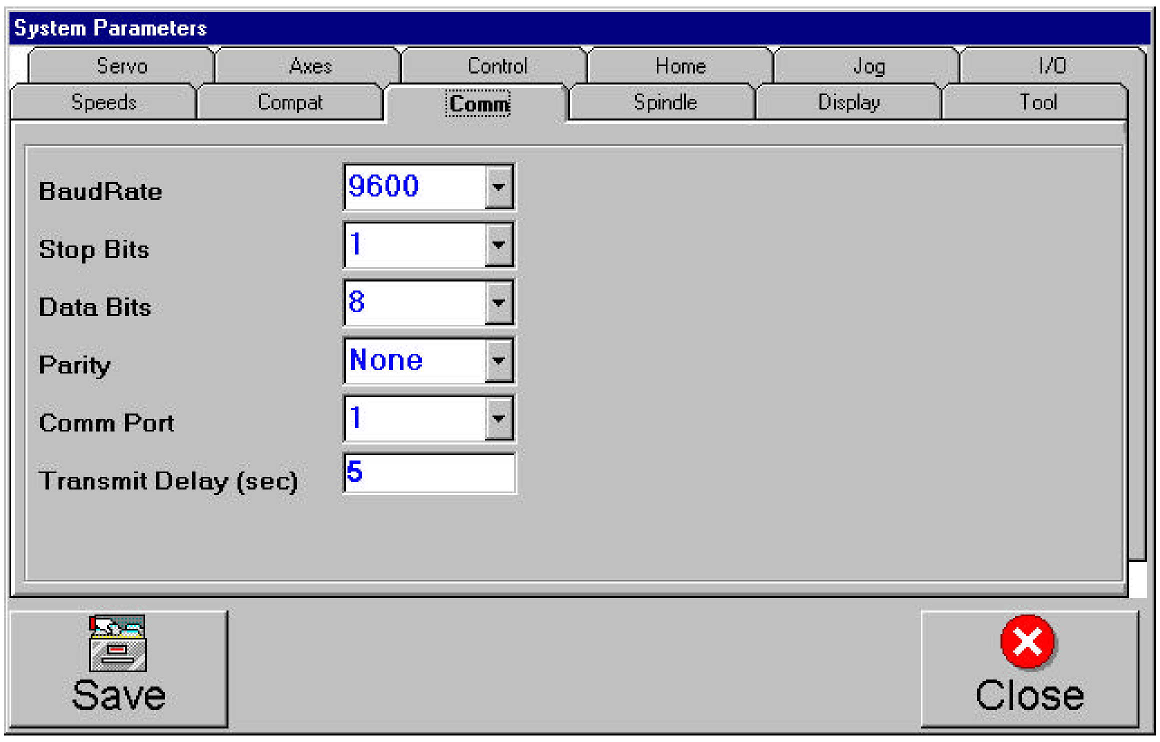
Parameter	Typical Value	Default Value	Units	Description
Max Machine Speed	200	200	Inch or mm per minute	Sets the Maximum Machine Speed allowed either in Feedrate or Rapid Mode.
Trial Speed	100	100	Inch or mm per minute	Selects speed to use in DRY RUN mode.
Cut Speed	10	10	Inch or mm per minute	Selects Cutting speed for RUN mode if there is no speed specified in the program.
Min Corner Speed	40	40	Inch or mm per minute	Sets the mimimum corner speed when Acute angle threshold parameter is set.
Marker Speed	NA	NA	Inch or mm per second	Not implemented
Plasma Hi/Lo Speed % Cut	NA	NA	NA	Not implemented
Forward Backward Speed	NA	NA	Inch or mm per minute	Sets the feedrate used to do PATH RETRACE and FORWARD TRACE moves Not Implemented
Master Rapid ACC/DEC	100	100	Inch or mm per sec per sec	Sets the accel and decel ramps used during RAPID feed moves.
Master Feed ACC/DEC	100	100	Inch or mm per sec per sec	Sets the accel and decel ramps used during cutting Feed moves.

Compatibility



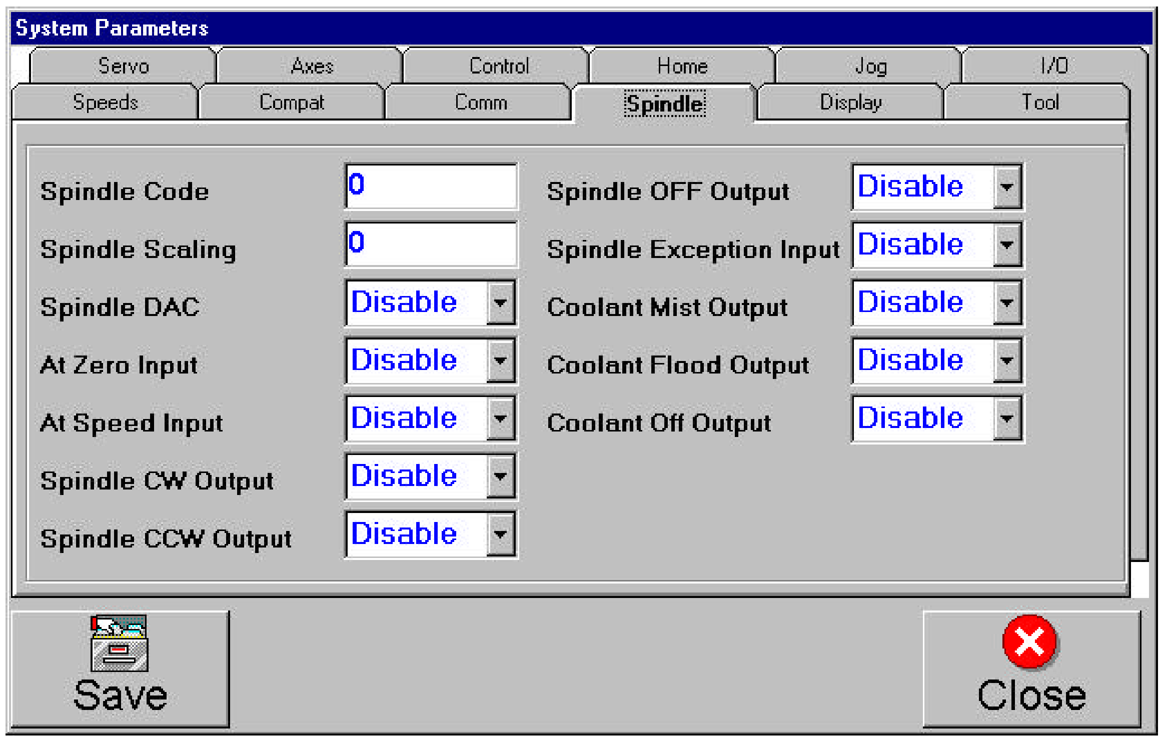
This selection must be left as FALSE in order for G64 to work as explained earlier. Setting it to TRUE disables the G64 mode and the system will always remain in the G62 mode.

Communications



This tab allows setting up Serial RS232 communication parameters.

Spindle



This tab allows setting up of Spindle related parameters.

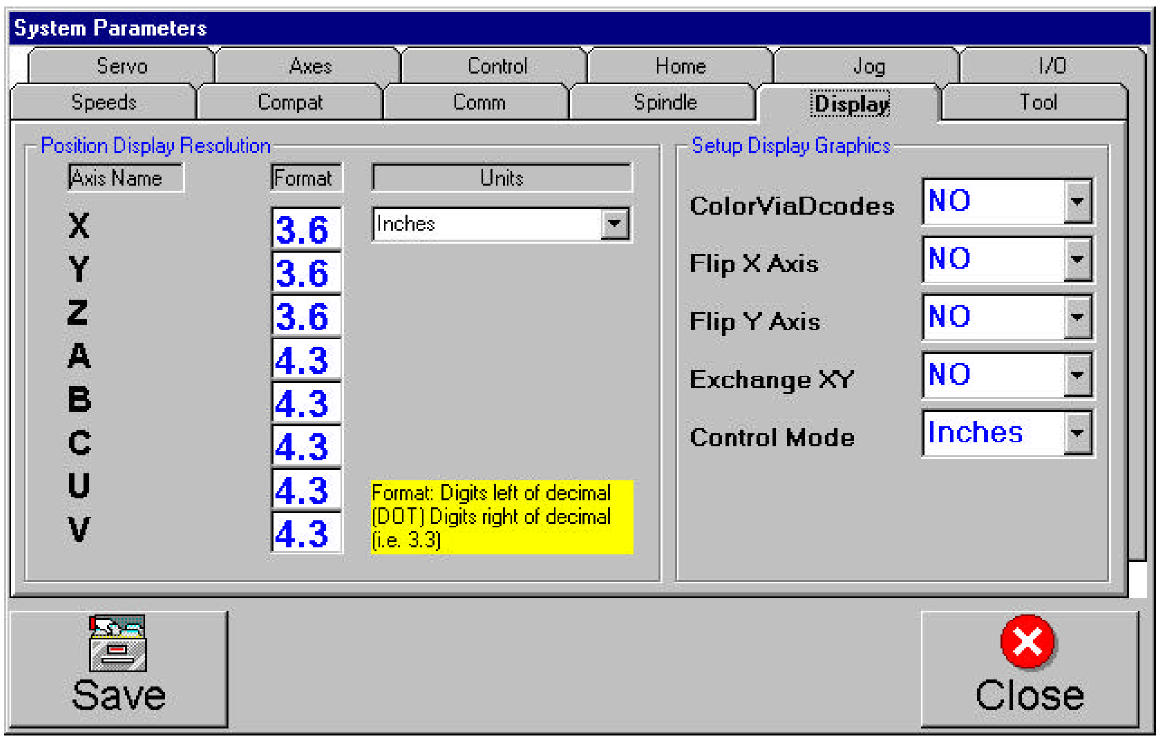
Parameter	Typical Value	Default Value	Units	Description
Spindle Code	6	0	Integer	This parmeter designates five different bit codes which allows the Spindle handshake operation to work properly. BIT4= 1 means LOOK for HI on ZERO SPEED. BIT3= 1 means LOOK for ZERO SPEED. BIT2= 1 means LOOK for HI on UPTOSPEED. BIT1= 1 means WAIT for UPTOSPEED. BIT0= 0 for an output between 0 and 10 volts and 1 for an output between −10 and 10 volts. The above information is invaluable for programmers in initial machine setup. It is recommended that the machine operator contact their machine builder before changing any of these values.
Spindle Scaling	0.00333	0	Float	Spindle Scaling multiplies all arguments to S commands by this factor before sending it to the DAC. The ratio of $\frac{Volts}{RPM}$ represents the units of measurement used in Spindle Scaling. If the maximum speed of the spindle motor is 3000 rpm, the corresponding voltage will be 10 volts which is the typical maximum analog signal output in Acroloop motion controllers. It follows that if the Spindle Scaling is set for 1500, the voltage is equal to a proportional 5 volts. Spindle scaling setup can be found under the Spindle Setup table on page 77.
Spindle DAC	3	-1	Integer	This specifies what DAC number (0-7) is attached to the spindle. See Note below.
At Zero Output	User Choice	-1	Integer	This specifies an input that signals that the spindle is at zero speed. If this entry is not **−1**, the control will wait until it gets a spindle At Zero speed acknowledge on the input number (based on the spindle code) on a SpindleOFF Out Command.
At Speed Input	User Choice	-1	Integer	This specifies an input that signals the spindle is at speed. If this entry is not **−1**, the control will wait until it gets a spindle At Speed acknowledge on the input number (based on the spindle code) on a SpindleCW or SpindleCCW Out Commands.
Spindle CW Output	User Choice	-1	Integer	This specifies the spindle ON output for clockwise direction.
Spindle CCW Output	User Choice	-1	Integer	This specifies the spindle ON output for counterclockwise direction.

Spindle OFF Output	User Choice	0	Integer	This specifies the spindleOFF output.
Spindle Exception Input	User Choice	-1	Integer	This is a user *defined* input for sensing spindle faults. Sensing this input will cause the maching to go int a FEEDHOLD cycle and program will terminate. An alarm will be given.
Coolant Mist Output	User Choice	-1	Integer	Specifies the Mist output
Coolant Flood Output	User Choice	-1	Integer	Specifies the Flood output.
Coolant OFF Output	User Choice	-1	Integer	Specifies MIST/FLOOD OFF Output

Note

If the DAC number for the spindle is changed, you ***must*** change the statement **P6480=P9993** in two files called the INITFILE.8K and RUNFILE.8K in AcroMill. This statement should be changed to the following: **Pxxxx=P9993**, where **xxxx** is the index for the DAC output.

Display



This allows customizing the Postion Display and the Graphics display.

POSITION DISPLAY RESOLUTION

First, the display format of each axis can be selected for both INCH and METRIC display. The format is specified as how many digits to the left of the decimal point followed be a period and how many digits to the right of the decimal points. So as an example entering 3.6 next to the X Axis space will result in the X axis position display shown with 3 digits to the left and 6 digits to the right of the decimal point.

Typically, for INCH mode, the display is 3.4 and for metric, it is 4.5.

SETUP DISPLAY GRAPHICS

Color VIA D Codes.
This allows showing tool path in different colors depending on the tool number.

Flip X Axis
This allows mirroring the GRAPHICS DISPLAY image on the screen around X Axis. Machines have different directions of movement for positive or negative moves.

Flip Y Axis
This allows mirroring the GRAPHICS DISPLAY image on the screen around Y Axis. Machines have different directions of movement for positive or negative moves.

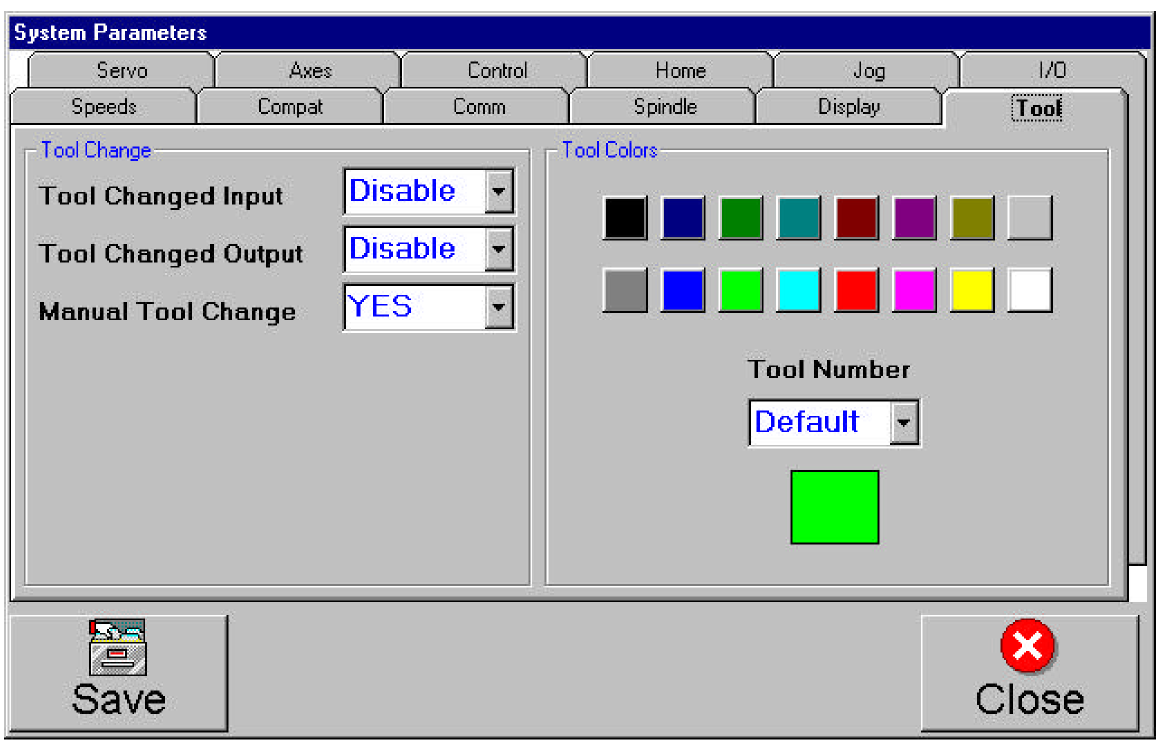
Exchange XY

This allows mirroring the GRAPHICS DISPLAY image on the screen around X and Y Axes. Machines have different axes describing LEFT and RIGHT and UP and DOWN movement on the Graphics screen.

Control Mode

This decides if the Control will power up in the INCH or the METRIC mode. In Europe, this parameter is usually left in METRIC and in the US, in INCH mode.

Tool



This screen allows programming Tool change mechanism and tool colors. There are 16 colors to choose from. Once this selection is made and in the previous tab (DISPLAY), the option is selected to show tool path color via D Codes, the Graphics will draw tool path in different colors depending on the currently active tool code.

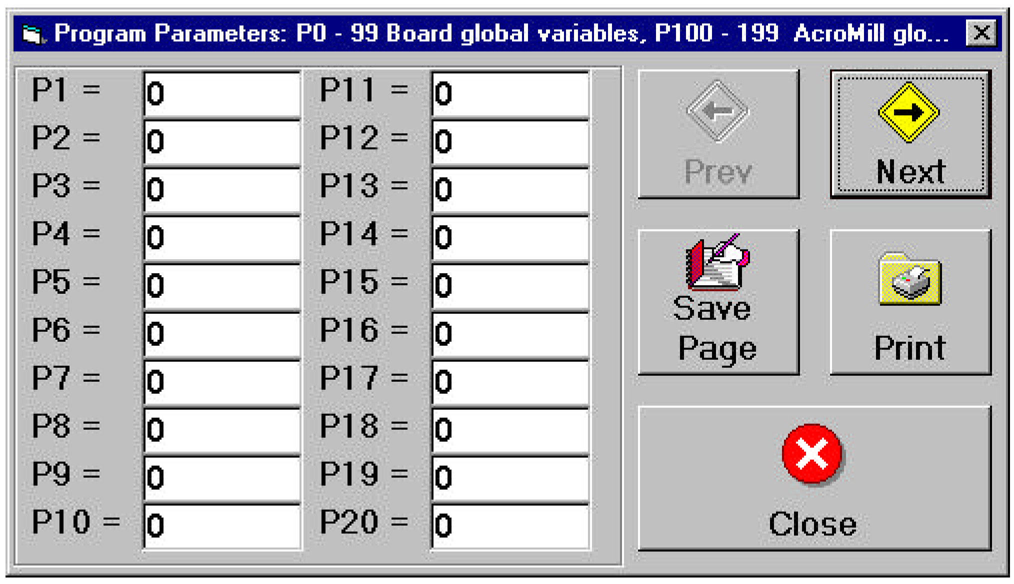
The tool change mechanism can either be manual or automatic.

Parameter	Typical Value	Default Value	Units	Description
Tool Changed Input	User Defined	Disable	Integer	This is the physical input that the Manual TOOL RESET pushbutton or the Automatic TOOL CHANGE COMPLETE signal is wired to.
Tool Change Output	User Defined	Disable	Integer	This is the physical output that signals the outside magnetics that the tool change has been requested.
Manual Tool Change	User Defined	YES	Boolean	This selects whether the Tool Change is Manual or Automatic.

P Parameters

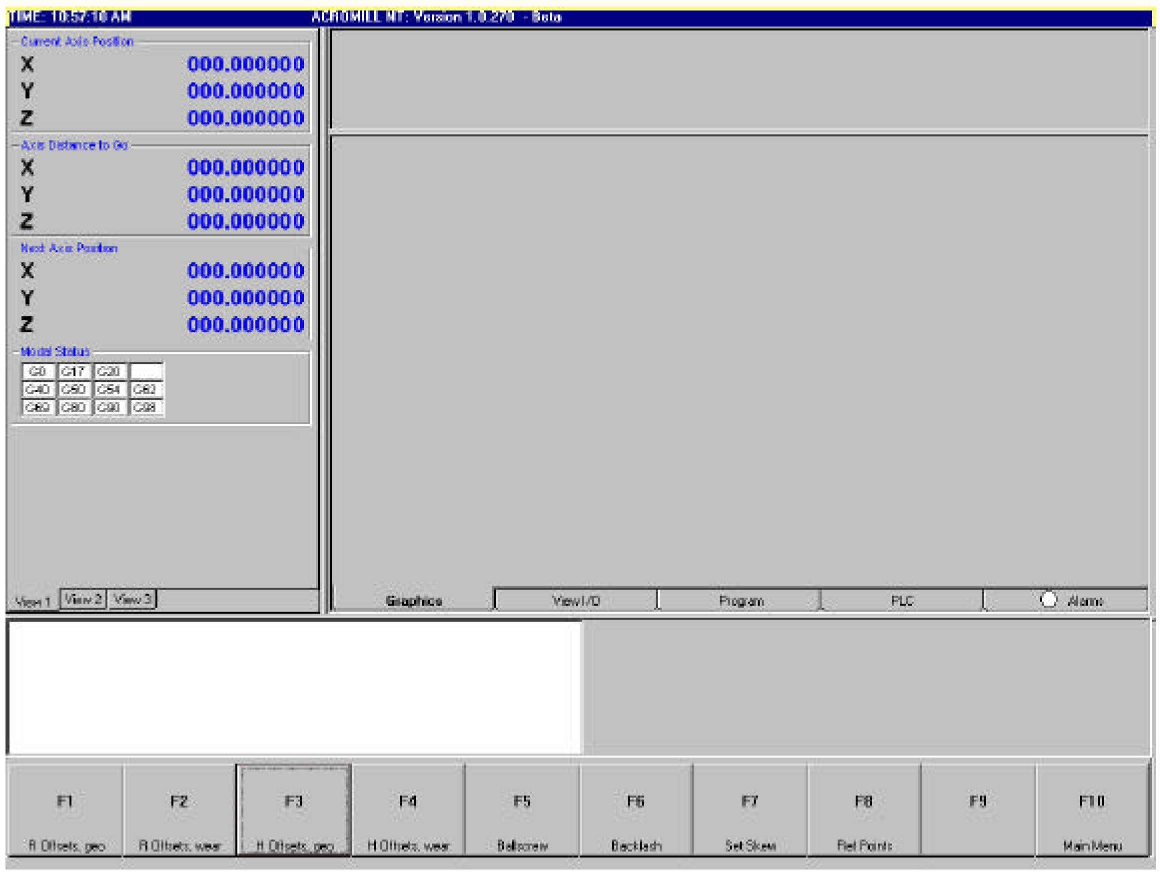
SET/DIAG (F8)→ P PARM MENU (F2)

This menu allows setup of P Parameters. The settable P Parameters are in the range of P0..P99 on the Controller, and P100…P199 in the AcroMill NT Software.



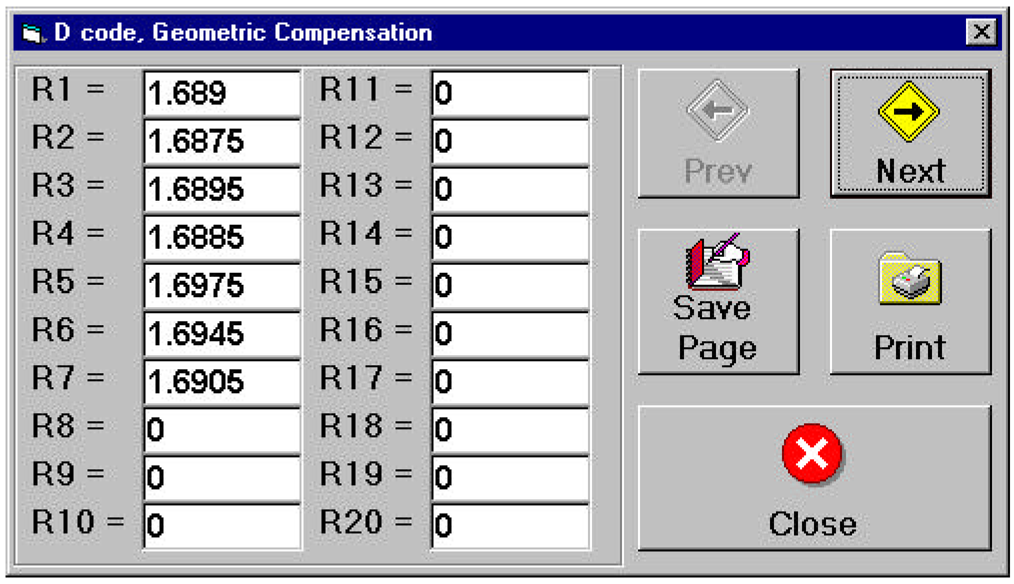
Offsets

SET/DIAG (F8)→ OFFSETS MENU (F3)



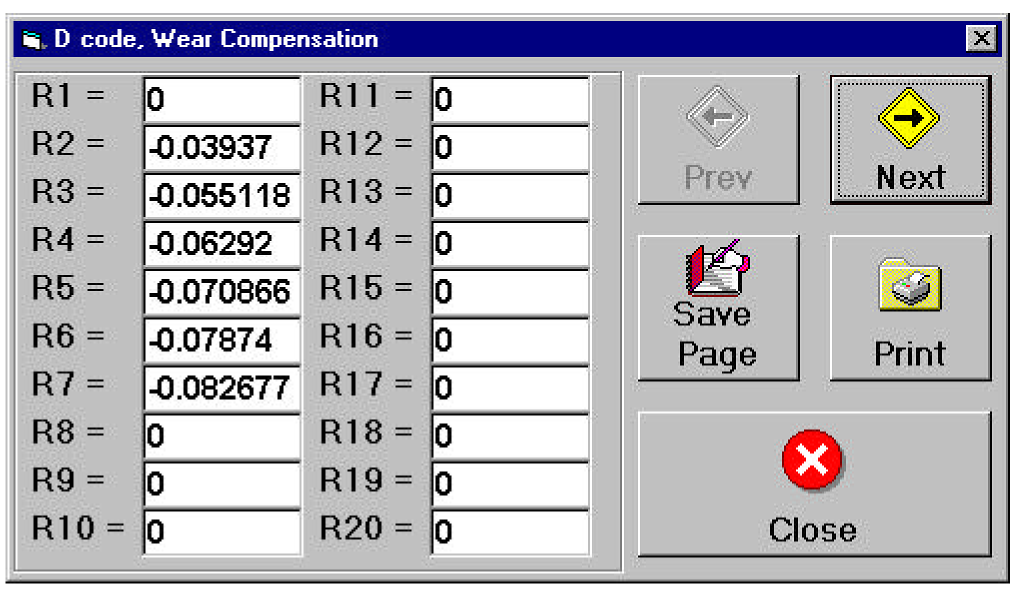
The Offset Menu allows editing of various offsets, Ballscrew, Backlash, and Reference points.

R Offsets (Geometric)



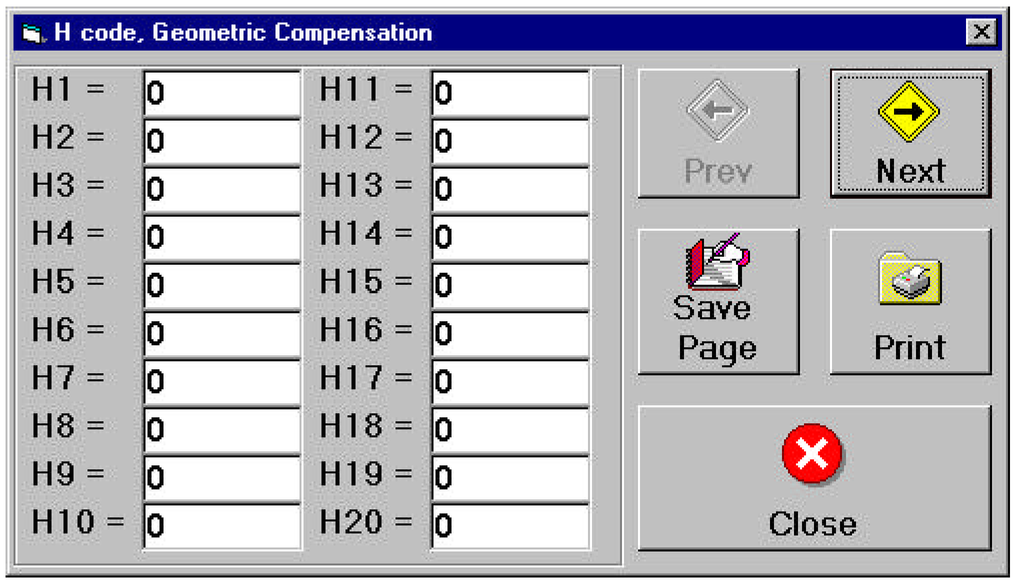
These are the Cutter Radius Offsets. A total of 400 offsets can be maintained. Like most edit screens in Acromill NT, make sure the SAVE PAGE button is hit after the entries are changed to save to disk.

R Offset (Wear)



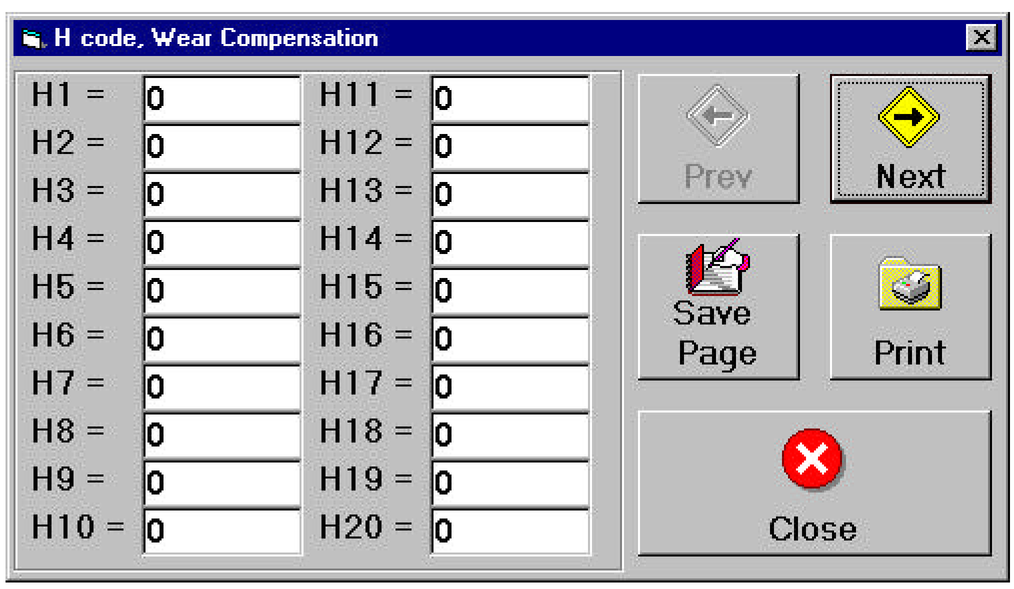
These offsets maintain “wear “ factor on the tool radius. The user fills the wear amount. The control will compensate for tool radius+tool wear always. By dynamically updating the Wear parameter, the user can compensate for tool wear on a continuous basis. A total of 400 wear offsets are available.

H Offsets (Geometric)



This allows the user to enter the TOOL Length offsets for up to 999 tools.

H Offsets (Wear)

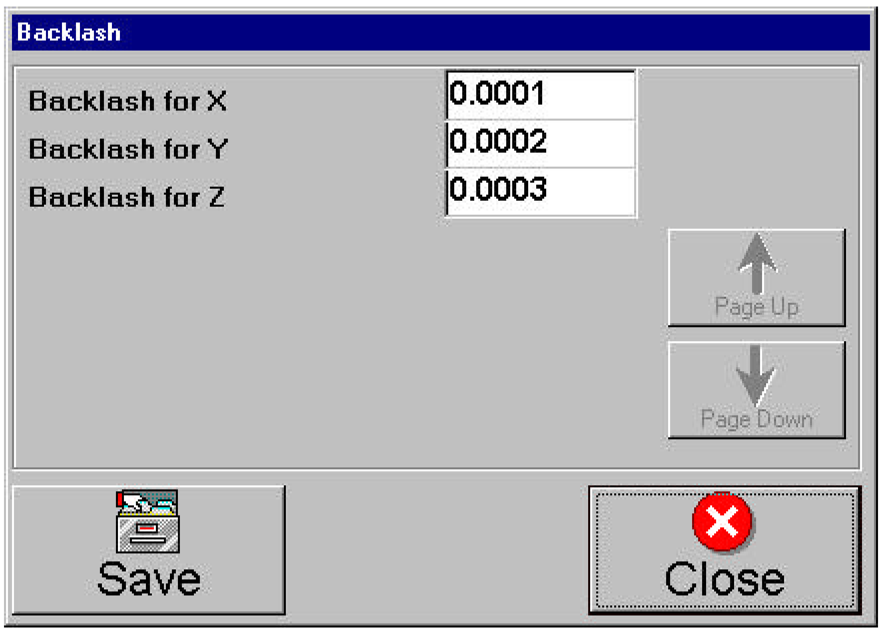


Like the Geometric H Offsets, there are 999 wear offsets available.

Ballscrew Compensation.

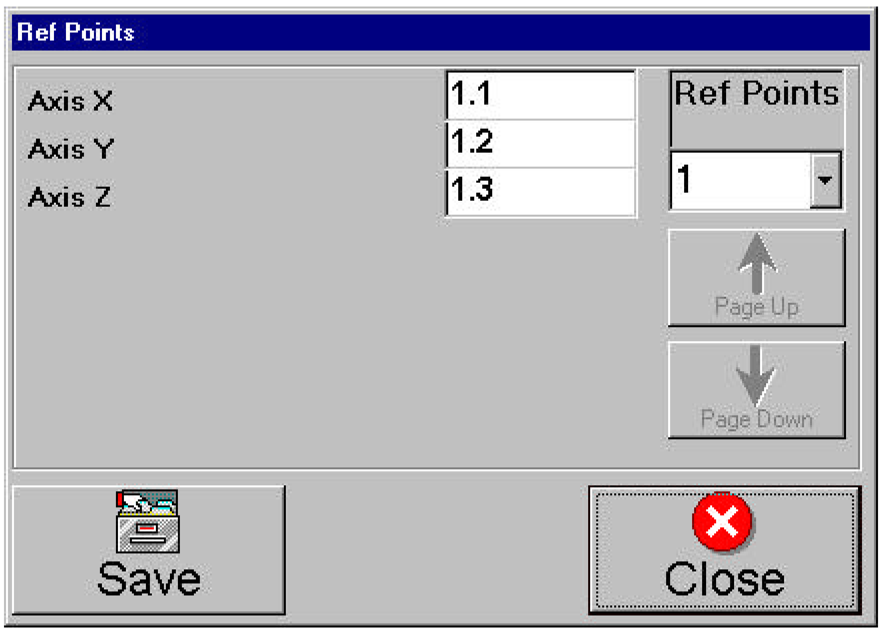
This is currently not implemented in AcroMill NT.

Backlash Compensation.



This type of compensation takes care of slop in the ballscrew mechanism. The motors will take up the slop in one direction making sure that the nut is always compressed when switching directions.

Reference Points.

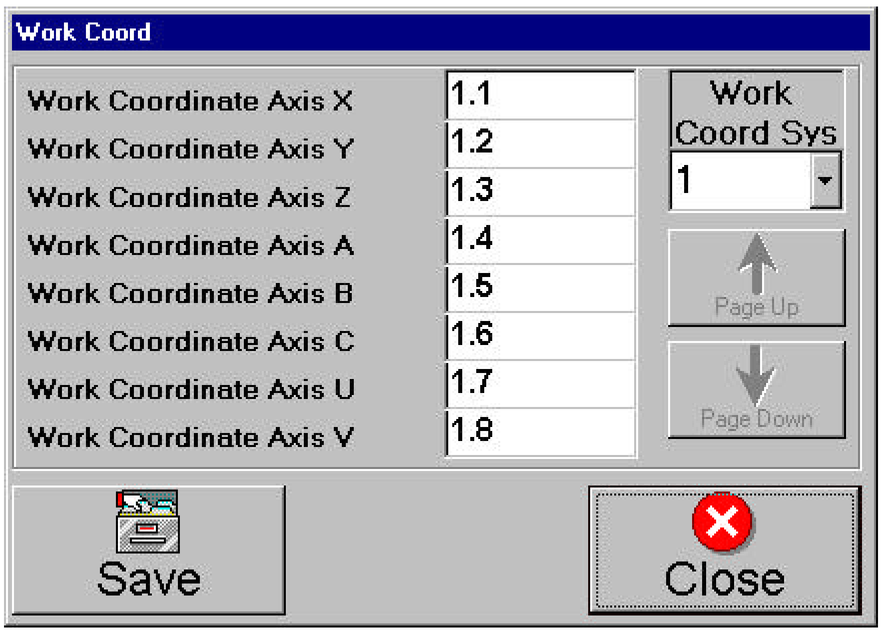


This screen allows setting up of REFERENCE POINTS to be used in conjunction with the G27, G28 and G29 commands.

Work Coordinates

SET/DIAG(F8)→ WORK COORDINATES MENU (F4)

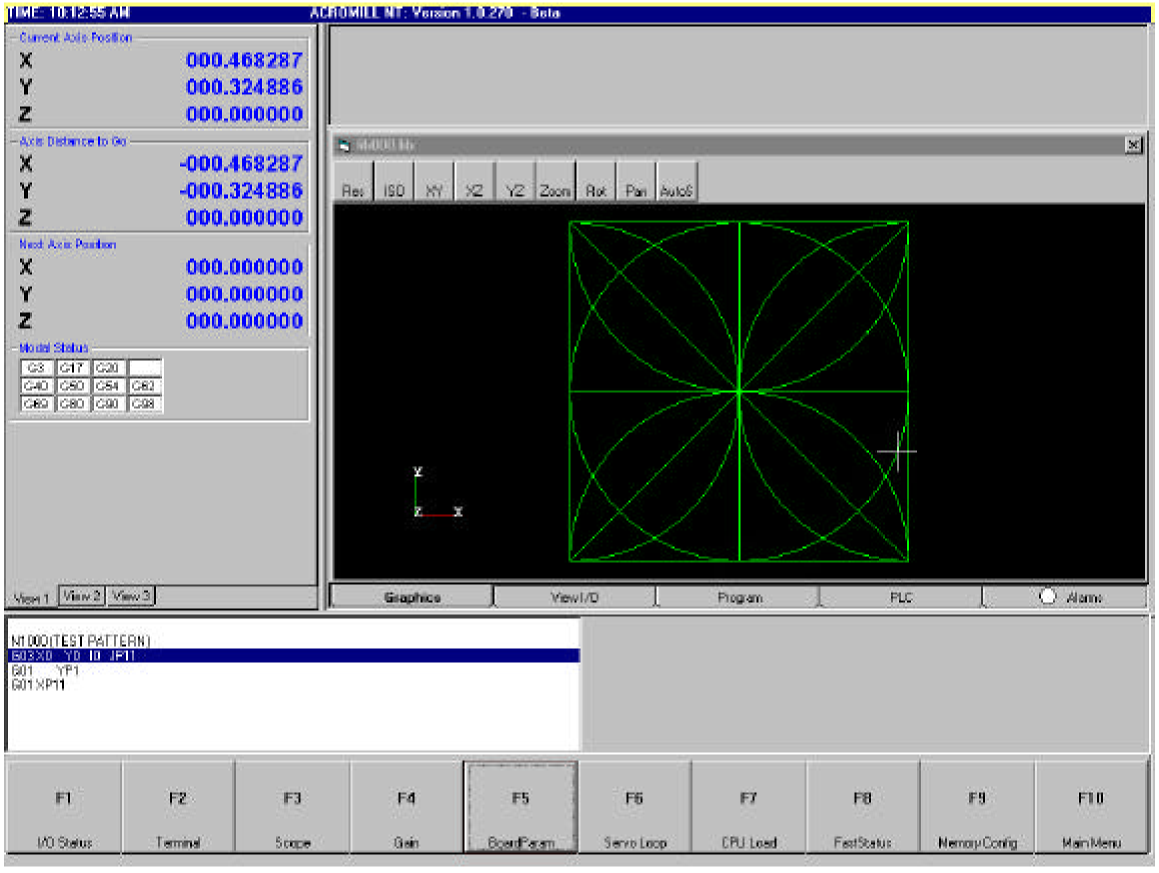
This menu allows setting up of work coordinates 1 through 8.



Diagnostics

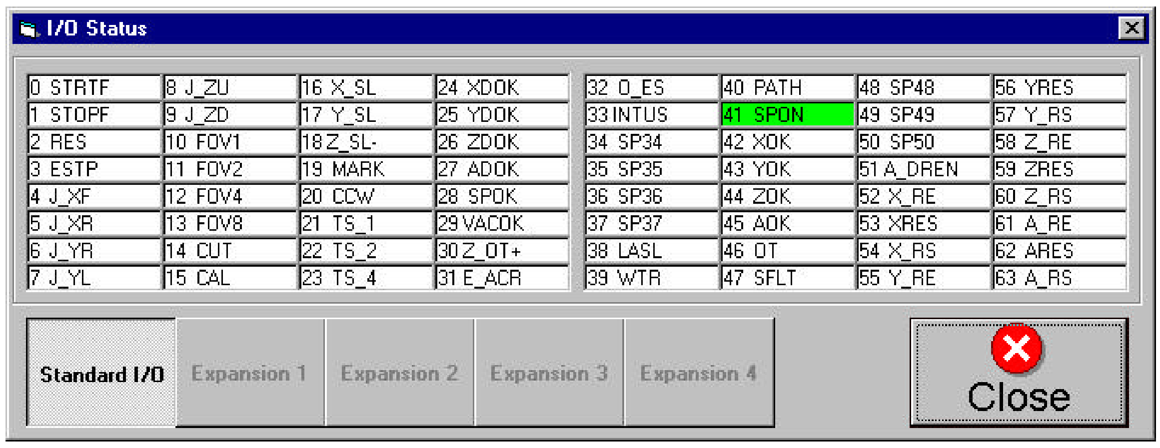
SET/DIAG (F8) → DIAGNOSTICS MENU (F5)

The main diagnostic screen below allows diagnosing major portions of the Controller.



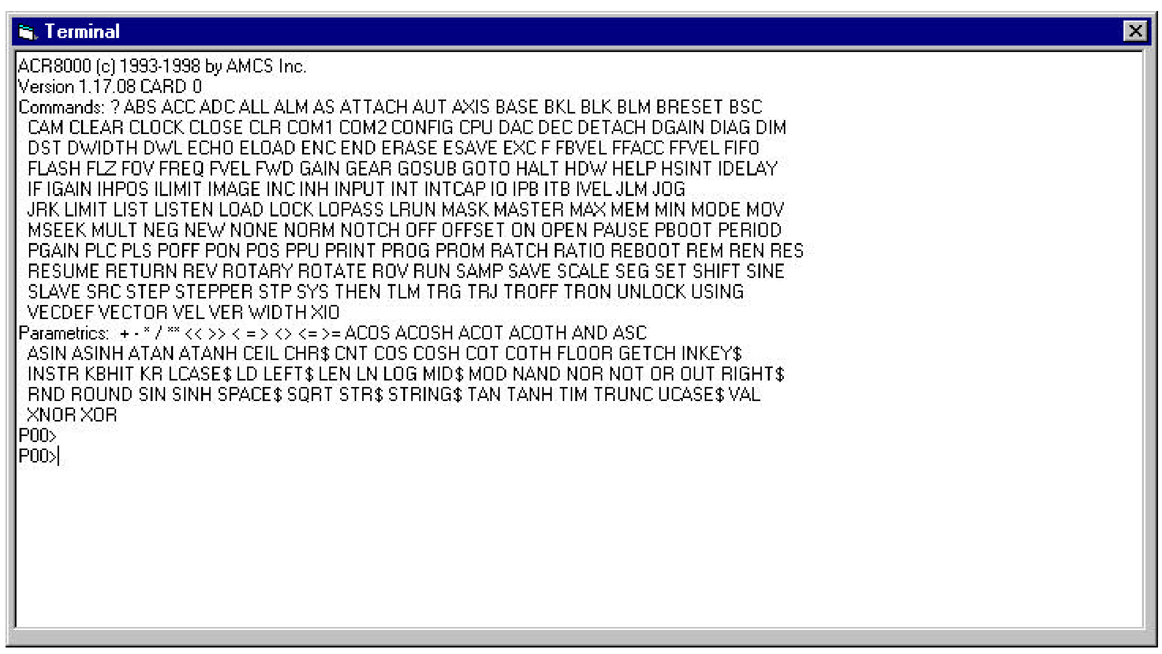
This screen allows diagnosing various aspects of the Controller interface.

I/O Status



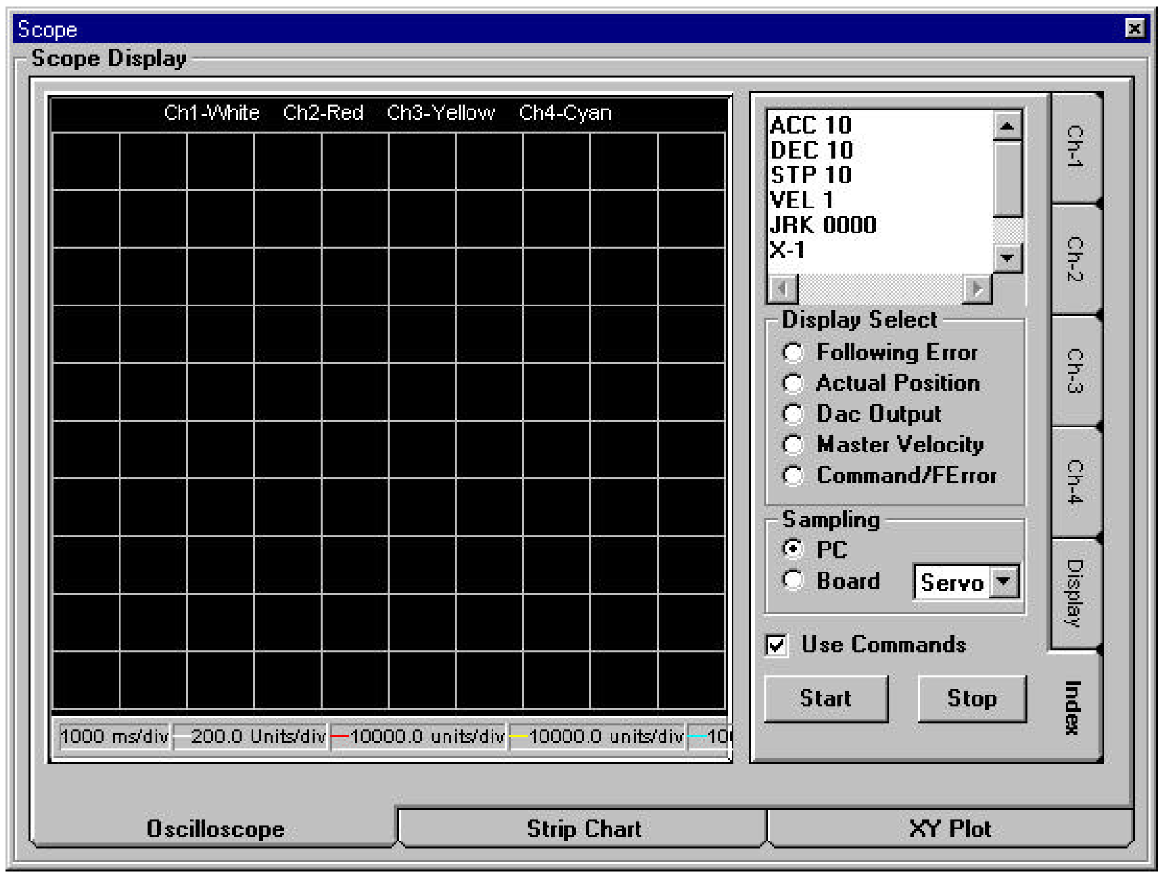
The I/O Status is a live display of as many inputs as the control has. The names of the inputs are user programmable.

TERMINAL



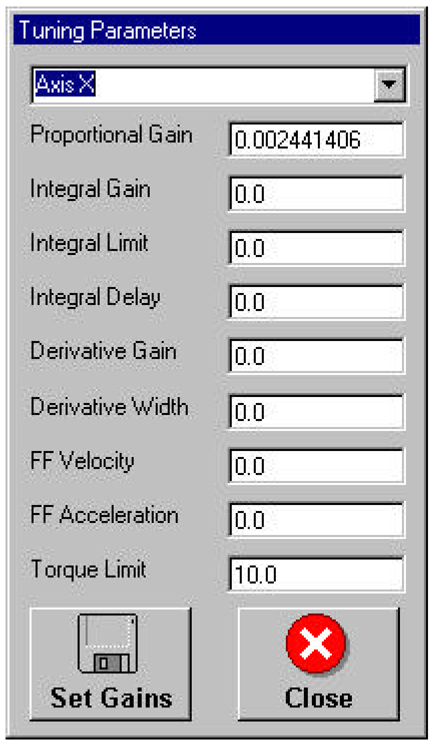
The terminal allows direct communications with the controller. Only experienced service personnel should attempt to user this terminal as sending the wrong command to the controller can cause unexpected operation.

Scope



This screen in addition to the GAIN screen allows TUNING the servos. Only experienced service personnel should attempt to use this screen. Sending the wrong commands can cause the machine to react unexpectedly and may cause damage. Refer to APPENDIX C on complete description on how to tune.

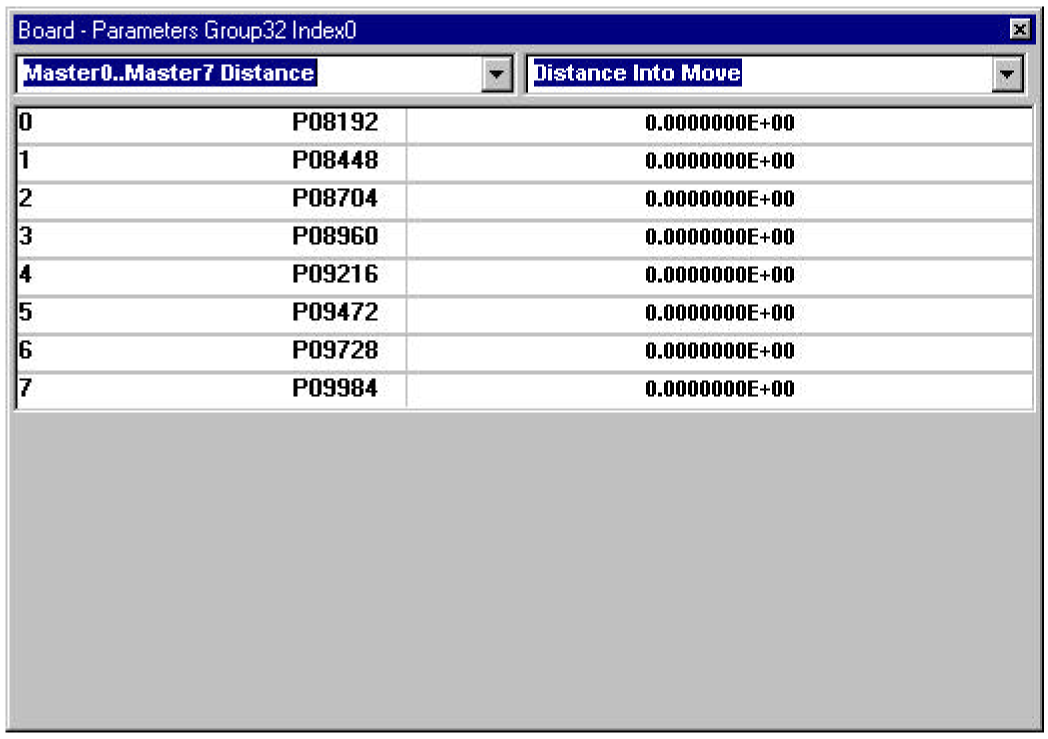
Gain



This screen allows changing gain parameters for each axis. After the gains are changed, the image is stored on to the hard disk.

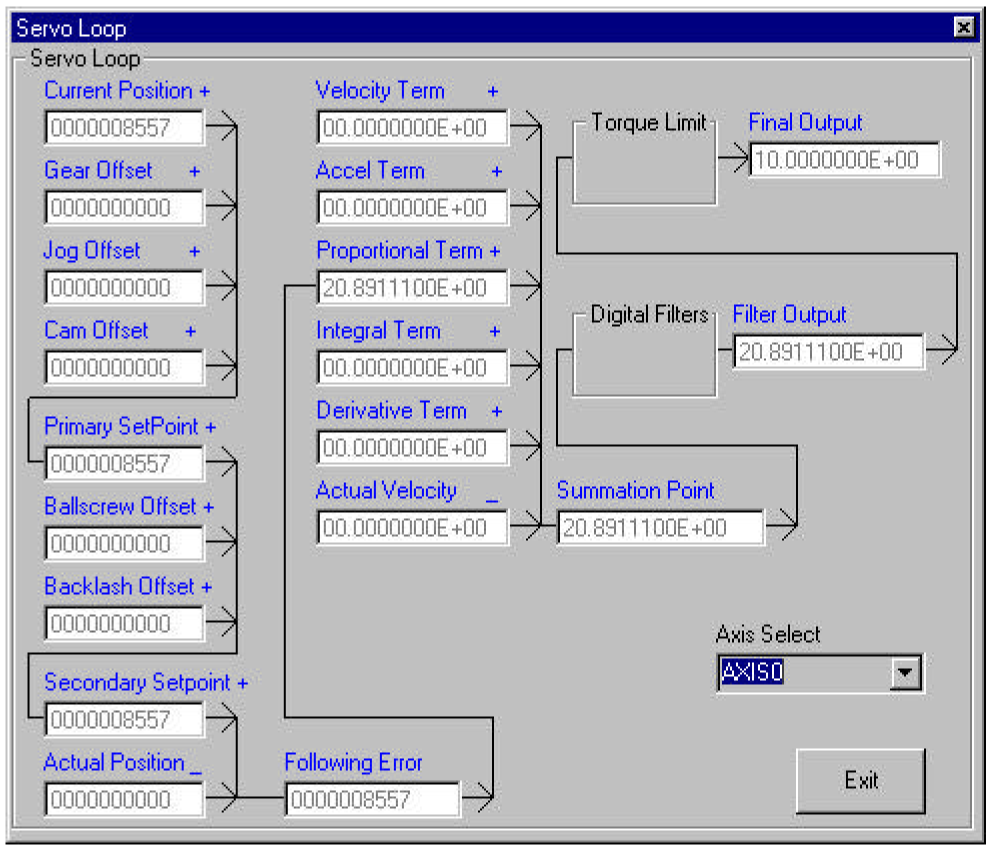
Only experienced service personnel should attempt to user this screen. Sending the wrong commands can cause the machine to react unexpectedly and may cause damage. Refer to APPENDIX C on complete description on how to tune.

Board Parameters



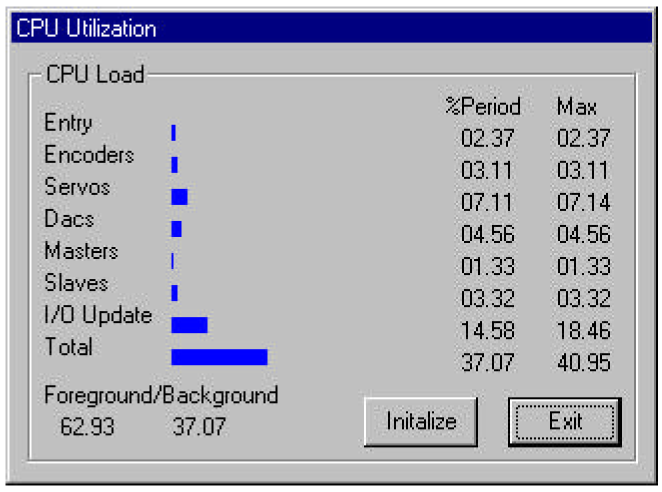
This screen allows viewing every parameter on the controller board. Only experienced service personnel should attempt to user this screen. Sending the wrong commands can cause the machine to react unexpectedly and may cause damage.

Servo Loop



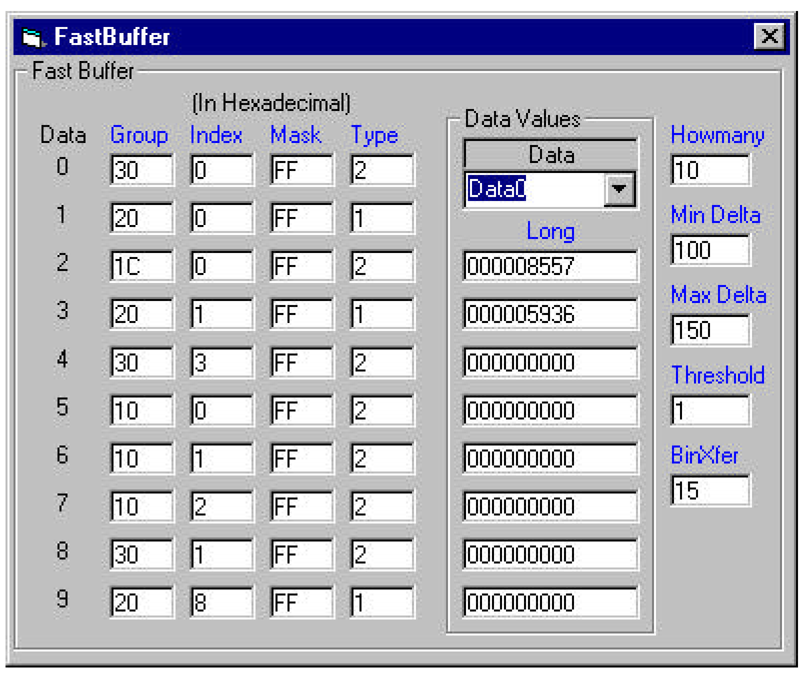
This screen allows viewing the entire servo loop for a particular axis.

CPU Load



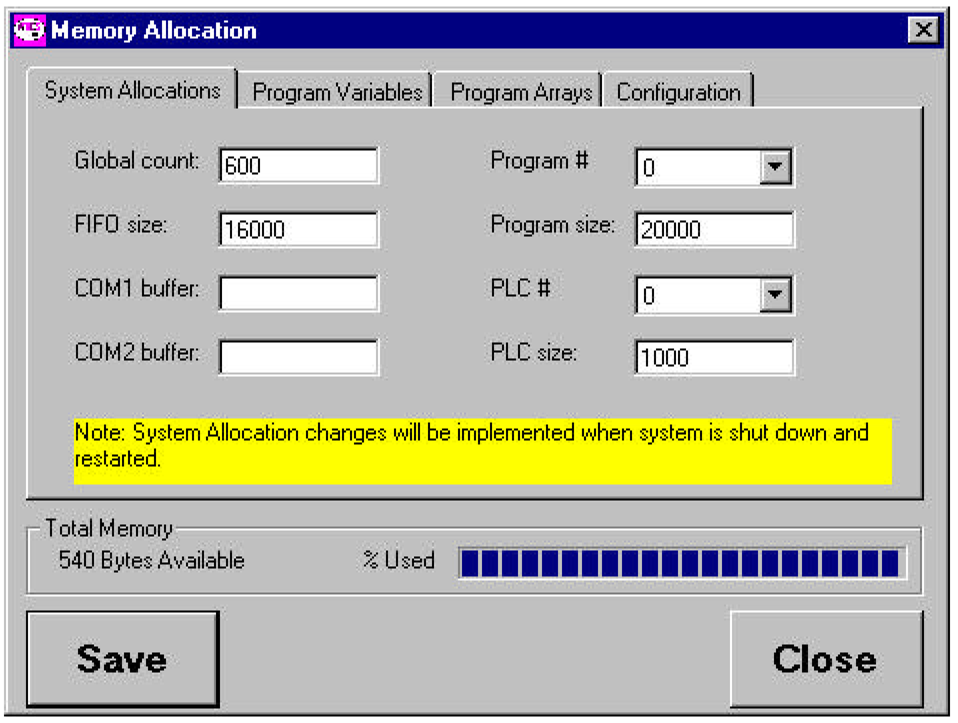
This screen allows viewing the current CPU load on the Acroloop Controller. Only experienced service personnel should attempt to use this screen.

Fast Status



This screen shows FAST STATUS information being gathered from the Acroloop Controller. Only experienced service personnel should attempt to user this screen. Sending the wrong commands can cause the machine to react unexpectedly and may cause damage.

Memory Config

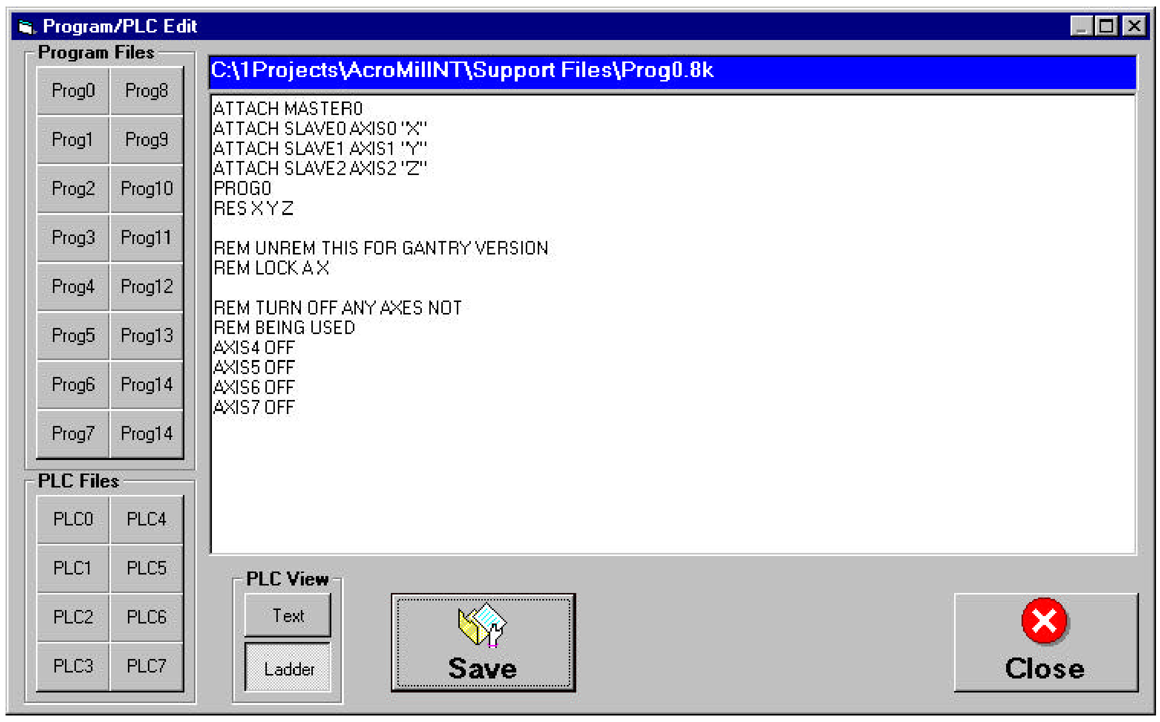


This screen allows configuring memory for programs and PLC's running on the Acroloop Controller . Only experienced service personnel should attempt to user this screen. Sending the wrong commands can cause the machine to react unexpectedly and may cause damage.

Prog/PLC

SET/DIAG→ PROGRAM/PLC MENU (F6)

This screen allows viewing and editing PLC's and Programs running on the controller.

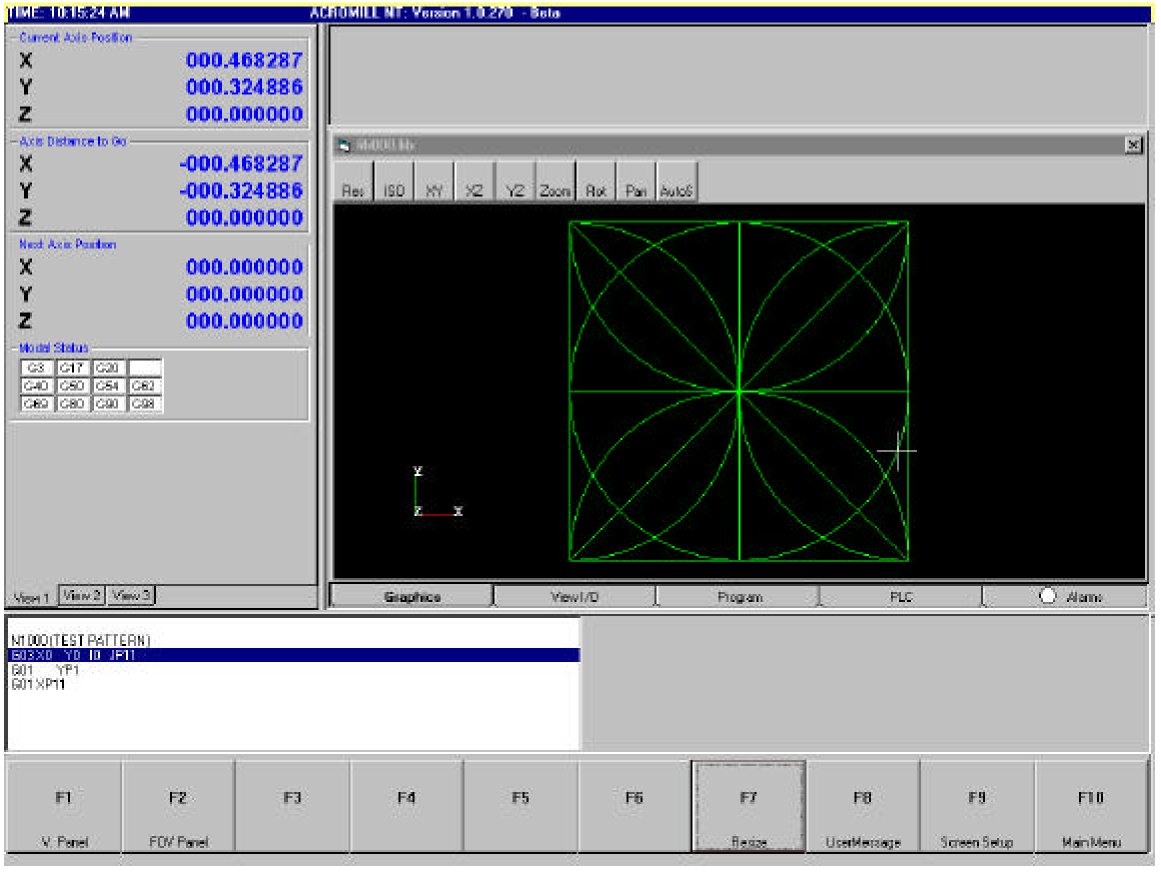


Display

SET/DIAG(F8)→ DISPLAY MENU (F7)

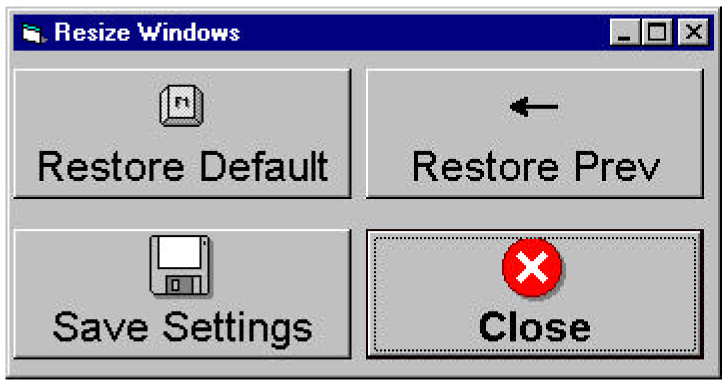
This menu allows customizing the following Display windows.

Virtual Machine Button Panel.
FOV Panel
Resizing Windows
Editing User Pop Up Messages
Screen Setup.



Resizing Windows

SET/DIAG(F8)→ DISPLAY MENU (F7) → RESIZE SCREEN (F7)

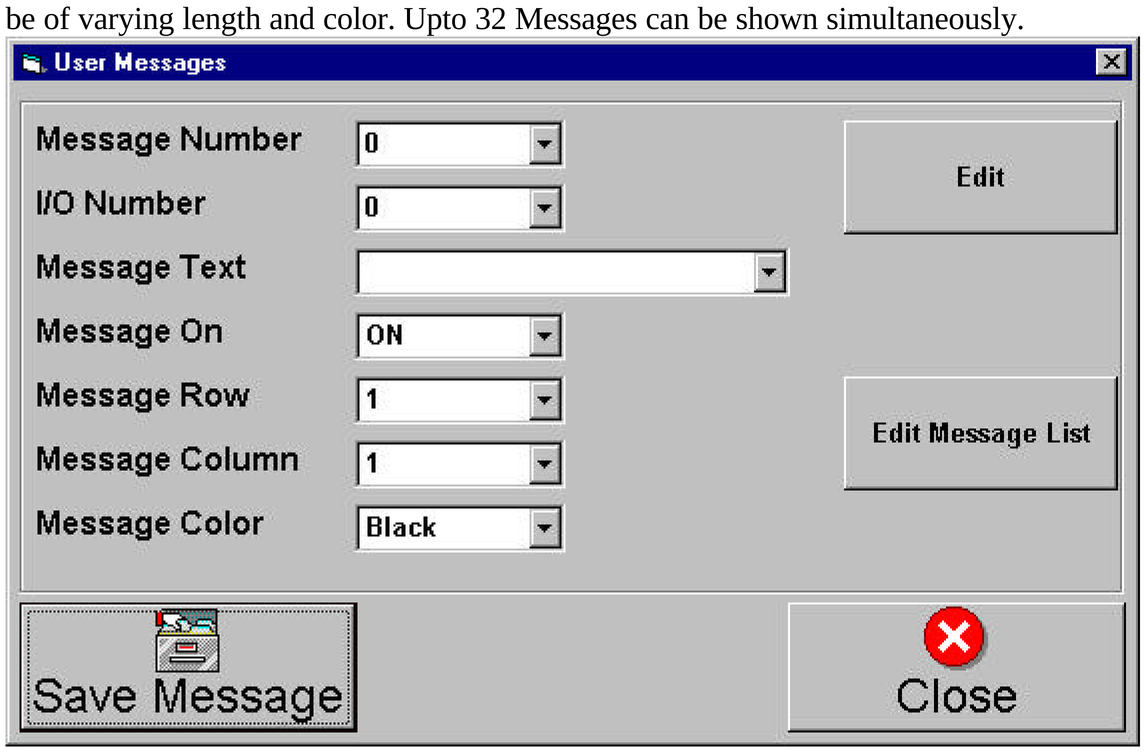


This allows restoring default settings on the Window sizes and appearances.

User Messages

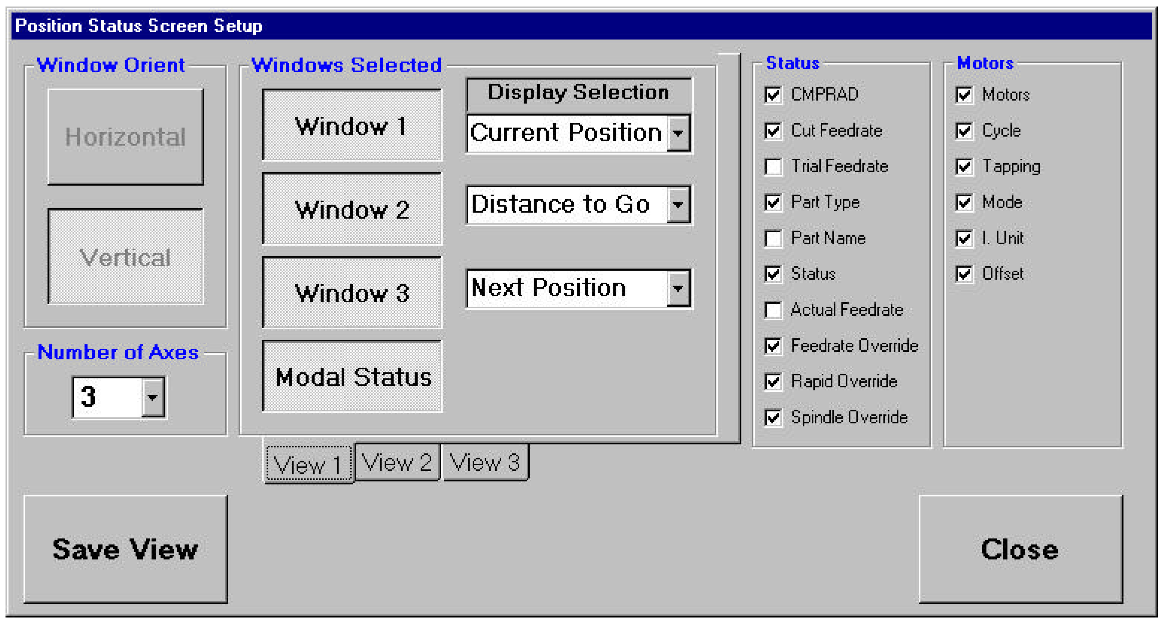
SET/DIAG(F8)→ DISPLAY MENU (F7) → USER MESSAGES(F8)

This screen allows entering user messages to appear triggered by input and output state. The messages will appear in the User Alarm screen on the top of the main screen and can be of varying length and color. Upto 32 Messages can be shown simultaneously.



Setup Screen

SET/DIAG(F8)→ DISPLAY MENU (F7) → SETUP SCREEN (F9)



This screen allows customizing of VIEW1, VIEW2 and VIEW3. Different types of data can be shown.

For Each of View1, View2 and View3 the user can select what kinds of information to put on the screen. Make sure that the SAVE VIEW button is hit after setting each View for it to take effect.

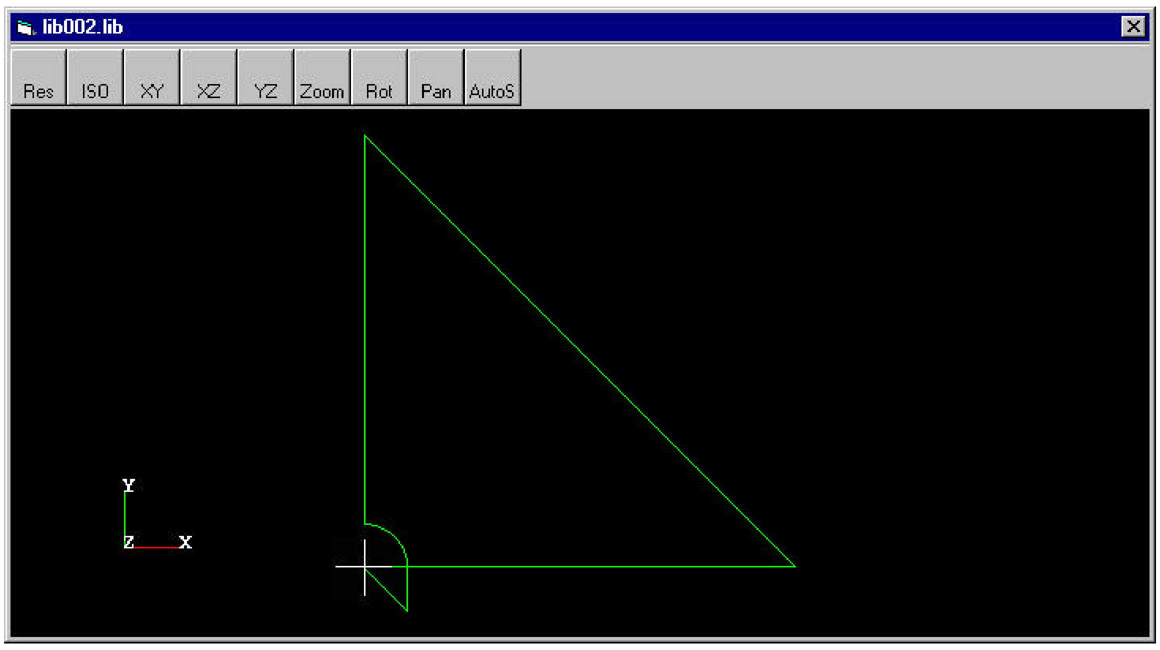
Cutter Radius Compensation

Initial Concepts

AcroMill NT offers G41 and G42 commands to offset the cutting path by the amount of the cutter radius. It is helpful to understand how the compensation works so some common pitfalls can be avoided.

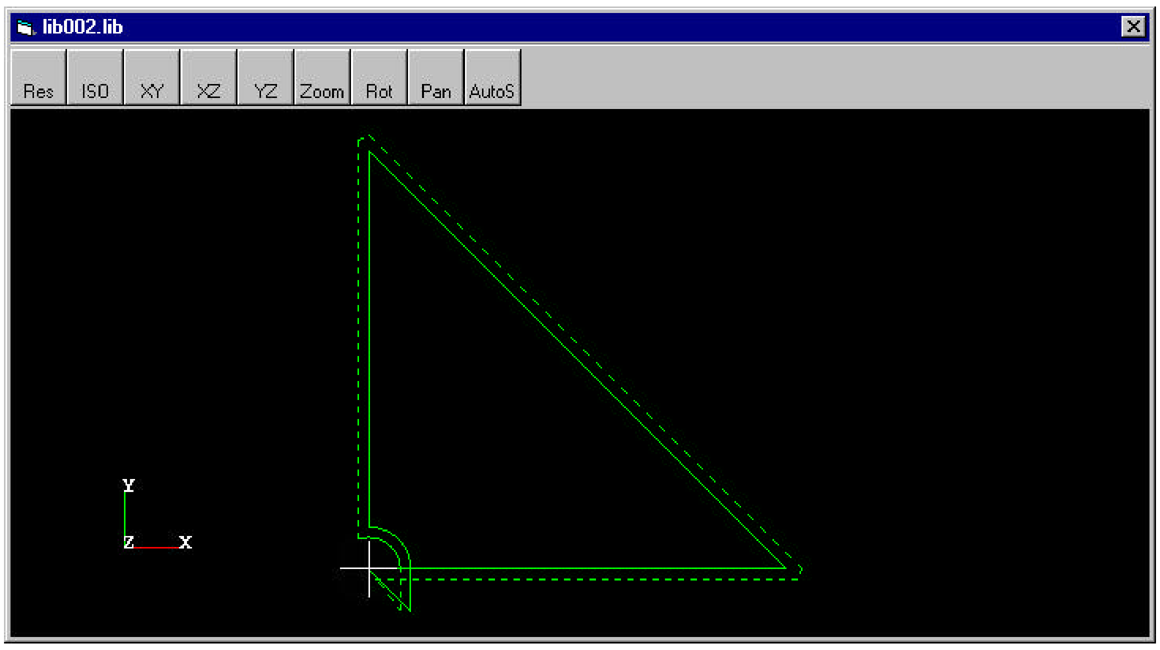
Physical geometry of various parts differs over a wide spectrum. In some cases, the geometry consists largely of straight edges and only requires LINE to LINE compensation. Some geometries are made up of all arcs. This requires ARC to ARC compensation. Most common geometries however consist of a mixture of arcs and lines. These kinds of geometries require LINE to LINE, ARC to ARC, LINE to ARC and ARC to LINE compensation.
There are instances where the geometry is such that for a zero cutter radius, the geometry makes sense as originally specified. Yet as the cutter radius increases, the geometry is such that intersections between successive arcs or lines and arcs disappear. In these cases, AcroMill NT's compensation module will actually "change" the user specified geometry slightly so that the intersections do indeed happen. In some instances.
Similarly, there are instances where increasing the cutter radius causes the tool movement to "reverse" direction and result in "GOUGING" the part. AcroMill NT will flag such points in the program giving the user the chance to fix these areas.
For feedrate control, AcroMill offers MULTIBLOCK LOOKAHEAD that will automatically adjust the feedrate so that axis do not come to an abrupt halt on sharp corners.

The following is an example of a Library part run with ZERO cutter radius.



The part is cut starting at where the Cross hair cursor is. Then the tool goes 45 degrees south east of the start position. Then goes straight up till it hits the bottom base of the triangle, cuts the CCW arc, goes straight up to the apex of the triangle then heads towards the bottom right corner. Last move brings the tool back to the starting point.

When cutter compensation is being utilized, care needs to be taken on where in relation to the part geometry, the actual compensation is turned on (G41 or G42).
In the above example the G41 (Left compensation is being used as the tool is offset to the left of the part) is being used. Notice that the programmed tool path starts and ends AWAY from the part surface. This is because the small moves into and out of the part give the control ample room to do the cutter offset moves without gouging the part surface.
These moves can also be refered to by LEAD IN and LEAD OUT moves
So with the cutter compensation turned ON with the cutter radius set to 0.25 inches, the same tool path shows up as a dashed line.



The same library part is now shown with 0.25" diameter cutter. Note that the part is shown via a "DASHED" line.

Note that at the top corner and the bottom right corner the tool path seems "rounded". This is because the AcroMill Compensation routines are smart enough to not go away from the corner too so as to be cutting 'thin air' and insert moves that keep the part accuracy up and minimize cutting time. These "inserted" moves will show up as lines with the word <ins> at the very beginning.

Common programming errors

If the programs are being generated via a CAD/CAM system, the geometry is usually cutter compensated right on the CAD/CAM system and what is received by the AcroMill control is usually a large CNC text file with lines and arcs. In this case the programs will usually run without modifications.
Even if the CAD/CAM does not to the cutter compensation, it can be done very successfully on the AcroMill control as it reads the CNC text file generate by the particular CAM/CAM Software.

However, of the parts are being generated via artist rendition on Corel Draw or Autocad and then exported through DXF conversion, there are potential problems to watch out for. Usually in these kinds of parts, the drawing are not always 'clean' in terms of circle to circle and line to circle intersections. This might result in the AcroMill control 'over fixing' the geometry by inserting lines. The end result might not be acceptable.

A solution for these kinds of parts is to design them taking the cutting device into account so that by the time AcroMill gets the part, it can be run AS-IS without any compensation modifications. The Acroloop controller equipped with Firmware Versions 1.18 and above are sophisticated enough to run any kind of geometry even if the start radius of the user specified arc is not the same as the end radius of the same arc. The problem comes when the cutter compensation routine gets to work on such geometries, it wants to 'fix' unequal start and end radii and if the geometry is too sloppy, too many errors are generated.

For parts generated by hand, such errors can be better controlled as programming from a blue print will not have the 'artistic' component of random shapes and surfaces.

There are specific parameters in the SYSTEM PARAMETERS that help the cutter compensation routine understand what the user wants to be done with the geometries that are contained in the typical user program.

Some industries are running Milling Machines and Machining Centers with AcroMill that require resolutions of 0.00001".
On the other hand, some industries are using AcroMill for doing wood working or Foam cutting that only require resolutions of 0.001"

Parts being generated by these industries will differ in their number of digits in accuracy.
AcroMill uses the following parameters to work on Low resolution or High Resolution geometries.

Epsilon for Compensation	0.002	0.002	Inch or mm.	Epsilon for Compensation sets the compensation for CAD drawings where lines do not meet. For example, if Epsilon for Compensation is set for 0.0001, all lines will be interpreted as joined within this threshold range.
Sweep Epsilon	0.01	0.01	Radians.	The Sweep Epsilon defines in radians the smallest valid sweep of an arc below which the arc can be treated as a straight line. This is used in cutter compensation math. Sweep Epsilon defines an arc as two points. If the angle ϕ is small enough as it approaches zero (this depends on such variables as arc radius or distance between the start and stop points), the Sweep Epsilon will interpret the start and stop points as one point.
Root Epsilon	-4	-4	Integer	The circle to circle intersections require a square root calculation. If the circles do not meet, the square root argument becomes negative. The ROOT EPSILON allows the user to fudge this "GAP" between the arcs by telling the control to ignore the gap and force an intersection.

In general never use a G40 (cutter compensation OFF), G41 (LEFT cutter compensation ON, G42 (RIGHT cutter compensation ON) in an ARC block.

Always use positive Cutter compensation. If negative compensation is used, the entry and exit points into the part have to be changed.

If programming by hand, make sure arc to arc, arc to line and line to arc intersections are specified as accurate to 0.0001 inches.

If CAD/CAM system is used, make sure the settings for numerical accuracy is set to at least 0.0001 inches.

APPENDIX A (Ballscrew Compensation)

BALLSCREW COMPENSATION SETUP PROCEDURE

Use the following illustrations for guidance in setting up Ballscrew compensation. A recommended diagnostic tool for Ballscrew compensation is the ACROVIEW software. Acroview allows monitoring the Ballscrew compensation for Axes 0 through 7 by observing the Axis Parameters. For axis 0 the Axis Parameters would be P12294 and P12294 (see the *User's Guide* for additional Axis Parameter information). The illustration below depicts Setpoint Summation. The Primary Setpoint (P12294) can be monitored against the Secondary Setpoint (P12295) by subtracting the two parameters. The resultant value will provide the Ballscrew compensation. Note the Secondary Setpoint is the summation of Primary Setpoint, Backlash and Ballscrew Offsets. If Backlash is used, this value must be subtracted to get the Ballscrew offset value (Ballscrew compensation).

AcroView will be installed during the Acroloop SDK installation.

The procedure for setting up ballscrew compensation requires comparative data from Primary Setpoint ballscrew travel data and actual travel data (see table below). These values are found from encoder information which are measured in control units (factory set in inches or metric units) and from ballscrew travel measuring devices such as laser interferometers. The following is the procedure for setting up ballscrew compensation.

1. Determine the Home position as it relates to the position of the ballscrew(s). The HOME command is a physical reference and may be the position cutting tool resides when the machine is not running a program or when disabled. Home is typically defined by a physical input such as a limit switch. See the two drawings depicted below. Home is defined in the drawings by positions *x*=2.000, *y*=2.000.

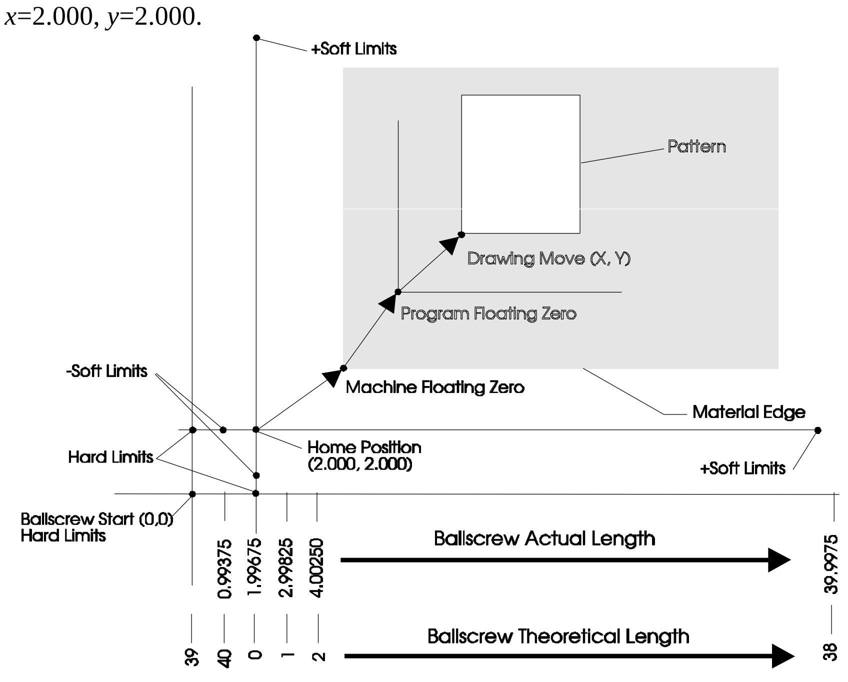


Figure A

The illustration below depicts a typical gantry style cutting table.

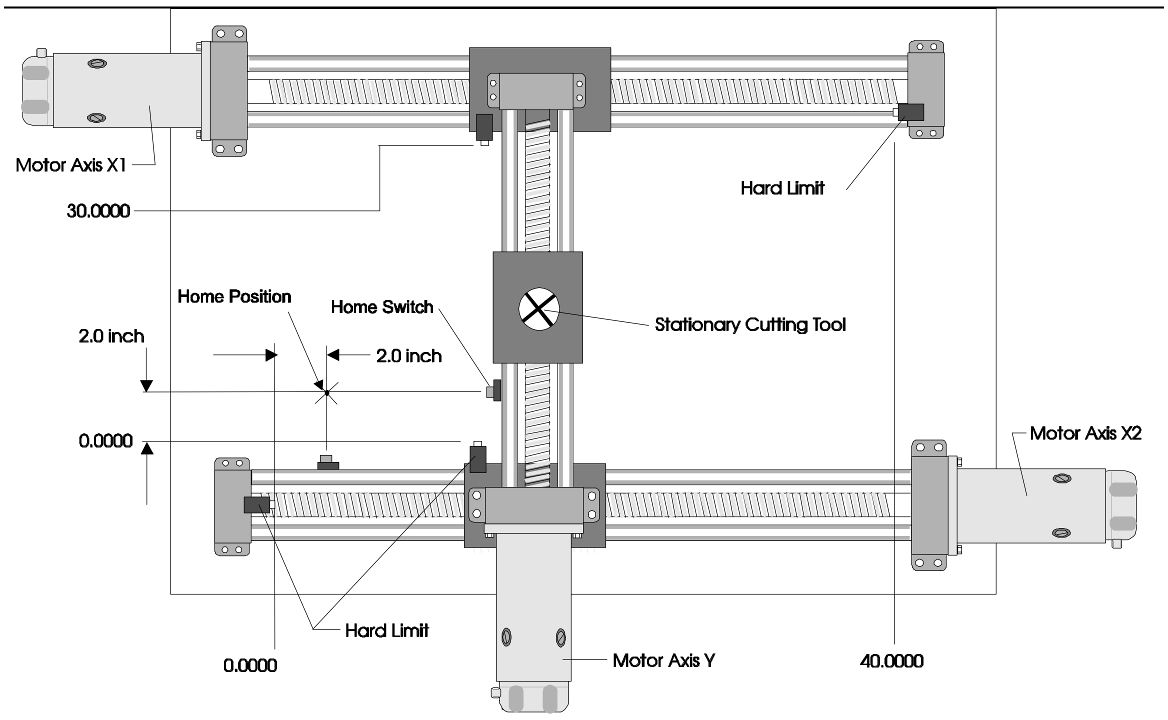


Figure B

2. Calculate the mean difference to get the ballscrew compensation required. A sample table is provided below. Note the Actual Mean Values in the table are shown in Figure A above.

Primary Setpoint (N)	Actual 1 (n1)	Actual 2 (n2)	Actual 3 (n3)	Actual 4 (n4)	Actual $_{Mean}$ (n1+n2+n3+n4)/Σn	Actual Difference (N–Actual$_{Mean}$)
0	0	0	0	0	0	0
1	0.979	1.000	0.997	0.999	0.99375	0.00625
2	1.996	1.998	1.995	1.998	1.99675	0.00325
3	2.997	3.000	2.997	2.999	2.99825	0.00175
4	4.004	4.001	4.004	4.001	4.00250	–0.00250
•	•	•	•	•	•	•
•	•	•	•	•	•	•
•	•	•	•	•	•	•
40	39.996	39.999	39.996	39.999	40.9975	0.00250

3. Since the Home position is defined at physical travel position (2.000,2.000), building the BSC input file will always start by interpreting Home position at zero (0). The BSC input file will look like the following:

X0E.00325
X1E.00175
X2E–.00250
•
•
•

X39E.00250
X40E.00000
X41E.00625

Note, lines X40 and X41 are the *wrap-around* compensation for where the Home position is located.

Be aware of static and dynamic criteria that may affect the ballscrew(s) functionality. Static factors are factors such as resultant accumulation of tolerances in pitch diameter (root and major), lead, and the groove radius of ball. Some dynamic factors are compression, tension, radial, and eccentric loading, and critical speed. All of these may be factors that affect ballscrew compensation.

APPENDIX B (DXF Import)

This feature is not yet released in AcroMill NT Beta.

PROCEDURE FOR IMPORTING DXF FILES

1. The procedure for importing a DXF file from diskette is described below. Starting from the MAIN menu, enter the FILES/EDIT menu by hitting the F7 key. The following display screen displays the DXF I/O menu. The DXF I/O menu allows importing and showing the directory of CAD files that have been converted to the DXF protocol.

   Error! Not a valid link.
2. Enter the DXF I/O menu by hitting the F3 key. The following table is the DXF <**--**> TEXT UTILITY dialog box that appears when DXF I/O is initiated. Choose number **1** for importing DXF files from your keypad.

   Error! Not a valid link.

3. The following table is initiated *on-the*-fly as soon as your number selection is entered. The operator can specify CONTINUOUS PATH or DRILL PATTERN by selecting the appropriate numbers as listed below. The CONTINUOUS PATH option interprets the data in the DXF file as a continuous path for cutting. The DRILL PATTERN option interprets the data in the DXF file as a sequence of holes to be drilled. In our example we will select CONTINUOUS PATH by entering number **1**.

   Error! Not a valid link.

4. The INPUT DXF FILE (.DXF) table is activated immediately after entering your number selection from the previous table. Use the arrow(↓) keys to scroll down to the appropriate floppy drive. In this example, the floppy drive is designated by A:\. Hit Carriage Return to enter your selection.

   Error! Not a valid link.

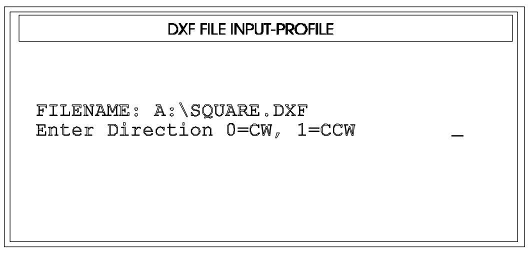
5. The INPUT DXF FILE (.DXF) will display the current drive as shown below.

   Error! Not a valid link.

6. Next, type in ***.DXF** to list the DXF file(s) you want to import. If you know the specific name(s) of the DXF file(s), enter the file name now and hit Carriage Return. Illustrated below in the INPUT DXF FILE (DXF) table is a sample DXF file that has been entered as described above. Hit Carriage Return to enter the next display table.

   Error! Not a valid link.

7. The DXF FILE INPUT **-**PROFILE table is activated and prompts you to enter the desired trace direction. Enter zero(**0**) for clockwise direction or one(**1**) for counterclockwise direction. Hit Carriage Return to enter your value.



8. Next, the same display table prompts you to whether you require entering a specific CAD layer name. Enter the CAD layer name and hit Carriage Return. Disregard this table by hitting Carriage Return if the CAD layer name is inconsequential.
   Error! Not a valid link.
9. For the OUTPUT FILE NAME (.TXT) table, type in the filename with a .TXT extension. As an example, you would type in the DXF file such as **SQUARE.TXT** with a .TXT extension. The OUTPUT FILE NAME (.TXT) table illustrated below shows the directory of .TXT files.
   Error! Not a valid link.
10. Once the DXF file has been converted (you will see a flash message indicating the conversion was successful), verify whether your program requires additional editing by entering the EDIT menu (F1). You have now successfully imported a DXF file to AcroMill.

APPENDIX C (Tuning)

QUICK STEP TORQUE MODE TUNING PROCEDURE.

This tuning procedure addresses Torque Mode Tuning. For Torque mode, the following parameters will have to be setup. In the torque mode, all of the following must be used to get good results. By the time we are finished, assuming the motor and amplifier is matched properly for the load, the tuning should yield maximum following error of less than +/- 5 encoder counts for the entire range of motor RPM. Additionally, the feedback encoder must yield at least 4000 counts per motor revolution at 4x multiplication for a suitable operation at slow speed.

Gain Parameters.

1.	PGAIN	(Volts per encoder pulse of following error).
2.	IGAIN	(Volts per second per pulse of following error)
3.	IDELAY	(Seconds after axis stops being commanded to move)
4.	ILIMIT	(Volts)
5.	DGAIN	(Volts per pulses per second)
6.	DWIDTH	(Seconds) Zero = Default is servo period.
7.	FFVEL	(Volts per pulse per second)
8.	FFACC	(Volts per pulses per second per second)
9.	TLM	(Volts)
10.	FBVEL	(Volts per pulses per second)

These gain parameters (except #10) are controlled and set via the “View” “GAIN” screen from the main menu in Acroview. The right click of the mouse on this screen will allow you to select “Always on Top” selection so that this screen will always remain visible as you do different things. Make sure you press the ENTER key each time you change any of the Gain values to actually update the controller.

If a particular motor/amplifier has never been tuned before, remove the load from the motor so that if the motor runs away during tuning, it is not going to damage anything.

If the above is not possible, start with the low TLM (torque limit) value so that if the motor does run away, no damage will occur. In any case, the machine must have HARDWARE OVERTRAVEL limit switches that will prevent any damage if any of the axes do take off towards any end of travel.

Among the10 gain parameters, the DWIDTH is one parameter that can be left at zero for most cases. This parameter determines how often the rate of change of following error is sampled. In the case where there are ample encoder pulses available per rev of the motor (at least 4096 after 4x multiplication), and the application is such that the motor is not going to be primarily used at very slow speeds, the DWIDTH parameter can be left at zero. This means that the rate of change of following error will be sampled every servo period (normally 500 microseconds). In cases where the operational speed is very slow, the DWIDTH can be changed to 2x 3x.. etc multiplier of the servo period in seconds. The FBVel parameter can also be left at ZERO for all cases that are not using DUAL FEEDBACK encoder (one encoder on the back of the motor for velocity feedback, second encoder on the load for position feedback).

Initial Considerations.

One can skip this section if the motor encoder count direction and servo amplifier dac polarity is in sync with the controller. If they are not in sync, the dac output from the controller will cause the motor shaft to turn in the opposite direction to what the controller wants and a possible runaway will occur.
As a safety issue, the user should set an EXCESS ERROR range of about 3 Motor Revs using the EXC command in the software. Then route the NOT EXCESS ERROR bits for all axes being tuned (Bits 769, 801, 833, 865, 897, 929, 961, 993) via the PLC to an output that shut motors power.

```
PROG0
PPU X4096 Y4096
EXC AXIS0 4 AXIS1 4 AXIS2 4 AXIS3 4 AXIS4 4 AXIS5 4 AXIS6 4 AXIS7 4

PLC0
HALT
NEW
10      LD 769
20      AND 801
30      AND 833
40      AND 865
50      AND 897
60      AND 929
70      AND 961
80      AND 993
90      OUT 32
RUN
```

(Note there is also a COMBINED NOT IN POSITION bit in the MASTER FLAGS for each master that could be used instead of individual axes NOT IN POSITION bits as shown above.)

In the above PLC, output 32 will go off if any one of the axis0…axis7 exceeds the following error limit that was set to be 4 revs

Assuming "X" is the motor axis being tuned, the following parameters are to be used to switch direction of encoder counting and polarity of DAC output.

```
MULT X4             (Times 4 multiplication in default direction)
MULT X-4            (Times 4 multiplication in opposite direction)
DAC0 GAIN 3276.8    (Dac0 normal polarity)
DAC0 GAIN –3276.8   (Dac0 opposite polarity)
```

Now one has to make sure that for the particular motor/amplifier/encoder combination, the Dac output polarity from the controller and its encoder count direction make sense.

With normal settings for the MULT command and DAC0 GAIN of 3276.8, the Dac output goes negative for a positive count on the encoder. This means that for a negative dac input voltage, the servo amplifier must turn the motor whatever direction it takes to make the encoder count in the positive direction in the controller. Monitoring the "Encoder Position" parameters P6144, P6160, P6176, P6192, P6208, P6224, P6240, P6256 and turning the motor shaft by hand can check this. With negative voltage input, the Servo Amplifier must turn the motor shaft in a direction that makes the Encoder to count up. If they are not in agreement, flip either the DAC gain, or the Encoder Multiplier (MULT command) or swap Motor Leads (this last option is not recommended for Brushless DC motors as it might throw off the commutation sequence relation ship with the HALLS). In any case, do this one at a time.

Start the Tuning procedure.

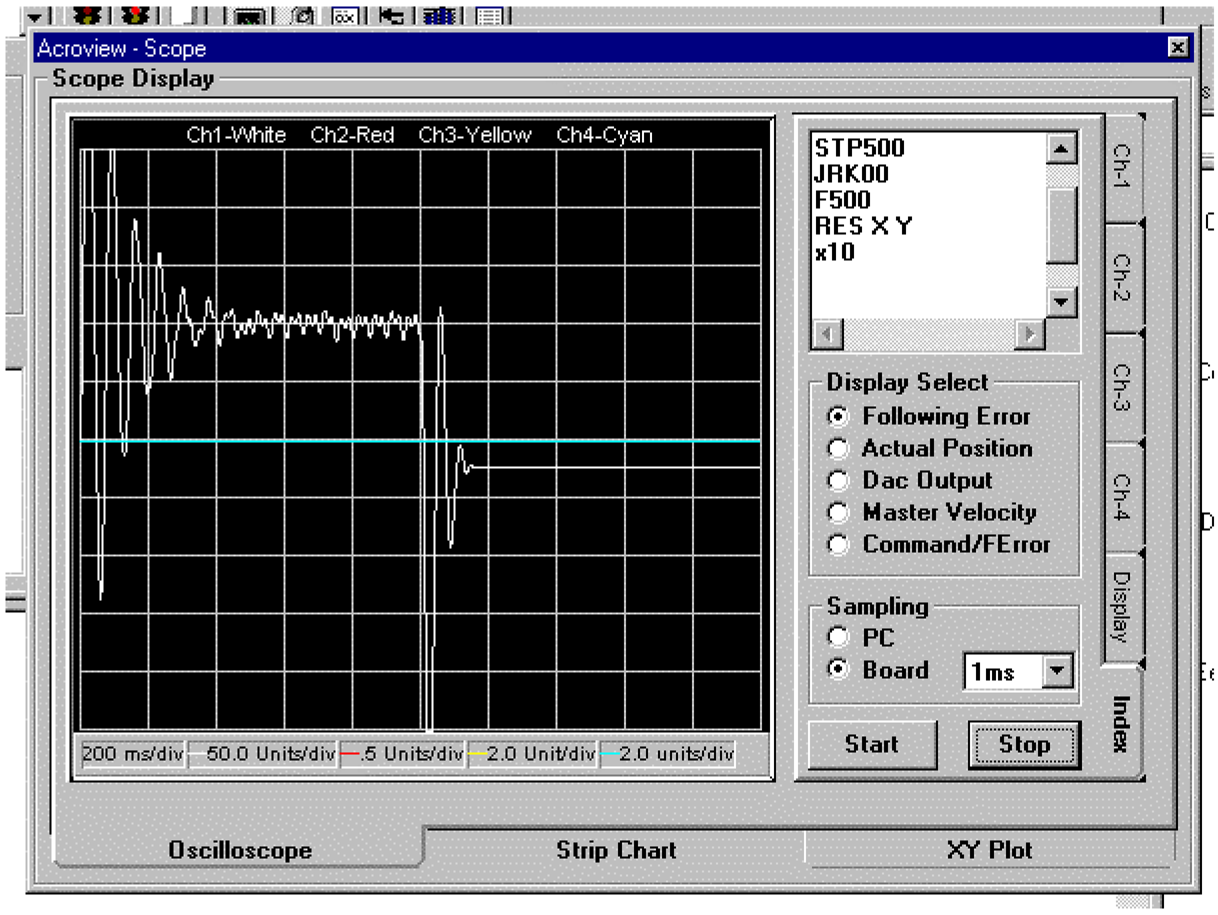
1. Make sure motor shaft is free to move about. If the motor is mounted on the machine load, position load axis in the middle of travel.
2. Tune 1 axis at a time. This means enable only the motor you are tuning.
3. With the motor disabled, ZERO out all the gain parameters #1...#9.
4. Assuming that the axis being tuned is X, set TORQUE LIMIT to 5% to 10% of amplifier/motor input range. Some amplifiers are setup for +/- 5 volts = full range, some for +/- 10 volts = full range. For example, if an amplifier puts out 15 amps continuous with a 10 volt input, but the motor is rated only 6 amps continuous, 5% to 10% means the TLM should be in the range of 5% of (10/15) * 6 to 10% of (10/15) * 6. In this case, the TLM command should be in the range of TLM X 2 to TLM X 4.
5. Going with 10% of full torque, do TLM X4
6. Do a RES X to ensure that the axis position, and therefore the Dac output, is zeroed out. Now energize the motor. Keep one hand on the EMERGENCY STOP pushbutton if motor runs away for whatever reason.
7. At this stage, the motor shaft will be limp and fairly free to move. If the motor is on the load, the load should not be moving.
8. Next, enter a very low value into the DGAIN. Low values for this parameter usually means 0.000001- assuming encoder is giving about 4000 pulses per motor revolution.
9. At this stage, the shaft of a no load motor will become a little stiff and resist, up to a small degree, any force applied to either direction. The motor is still in open loop so the shaft will not come back to null position, but it will add some resistance to forces applied to either direction by hand. Of course, if the machine load is bolted to the motor, this is not very evident. Increase DGAIN until the motor shaft loose most of its spongy feeling and resists displacement in both directions, even though it will never correct as it is running open loop
10. Next enter a starting value of around 0.001 to PGAIN.
11. Keep increasing PGAIN until the motor begins to buzz and back off until the buzzing stops.
12. At this stage, it should be possible to make close loop indexes and watch following error. Set up an index initially for maybe 10 or 15 revs at 500 RPM to start off, keep accel, decel and stp so that the system will come up to speed in the desired time for the application. As an example if one wants to run at 4000 rpm and accelerate 0 to 3000 RPM (50 RPS) in 0.1 seconds, then use formula a=v/t and come up with 50/0.1 = 500. This means that ACC500 DEC500 STP500 should do the trick.
13. In the SCOPE screen in Acroview, Select “Oscilloscope” and “Index”. The top profile text box should be changed to show the following profile.

    ACC 500
    DEC 500
    STP 500
    JRK 0
    F500
    RES X
    X15
14. Select the “Display”. Now setup the Scope Screen to see FOLLOWING ERROR. Setup the CH1 scale to about 50, OFFSET to 0 and source should be left to watch X-axis following error. Select source of sampling to CARD and frequency to 5 msec. Now press the “Display” tab to setup Horizontal Time Sweep Rate. We wish to capture the entire index during the ACCEL, AT SPEED and DECEL portion. There are a total of 10 vertical divisions on the scope. As a estimate, assuming ZERO acceleration the index should take

    15 revs * 60 seconds / 500 RPM = 1.8 seconds.

    This means that if we choose the Horiz msec/Div scale of about 200 Horiz msec/div, we will see the entire profile. Now press START. This will result in the motor go 15 revs in the forward direction and stop. If there is not enough storage space allocated to program 0 on the card, an error will be reported. If this happens, go into the Main Menu "VIEW" and "SYSTEM CONFIGURATION" and allocate more memory to PROGRAM 0.

15. Tweak the Channel 1 setup parameters to get a good look at the profile and execute the profile again. At this stage, the following error is going to probably be off scale. Now one must manipulate the PGAIN, the two DGAIN parameters, the three IGAIN parameters and the two Feed forward parameters to bring the gain under control. At this stage, the profile might look like the following. Note that the profile is using 10 revs instead of 15. The result is that the profile will not end up using the entire screen.

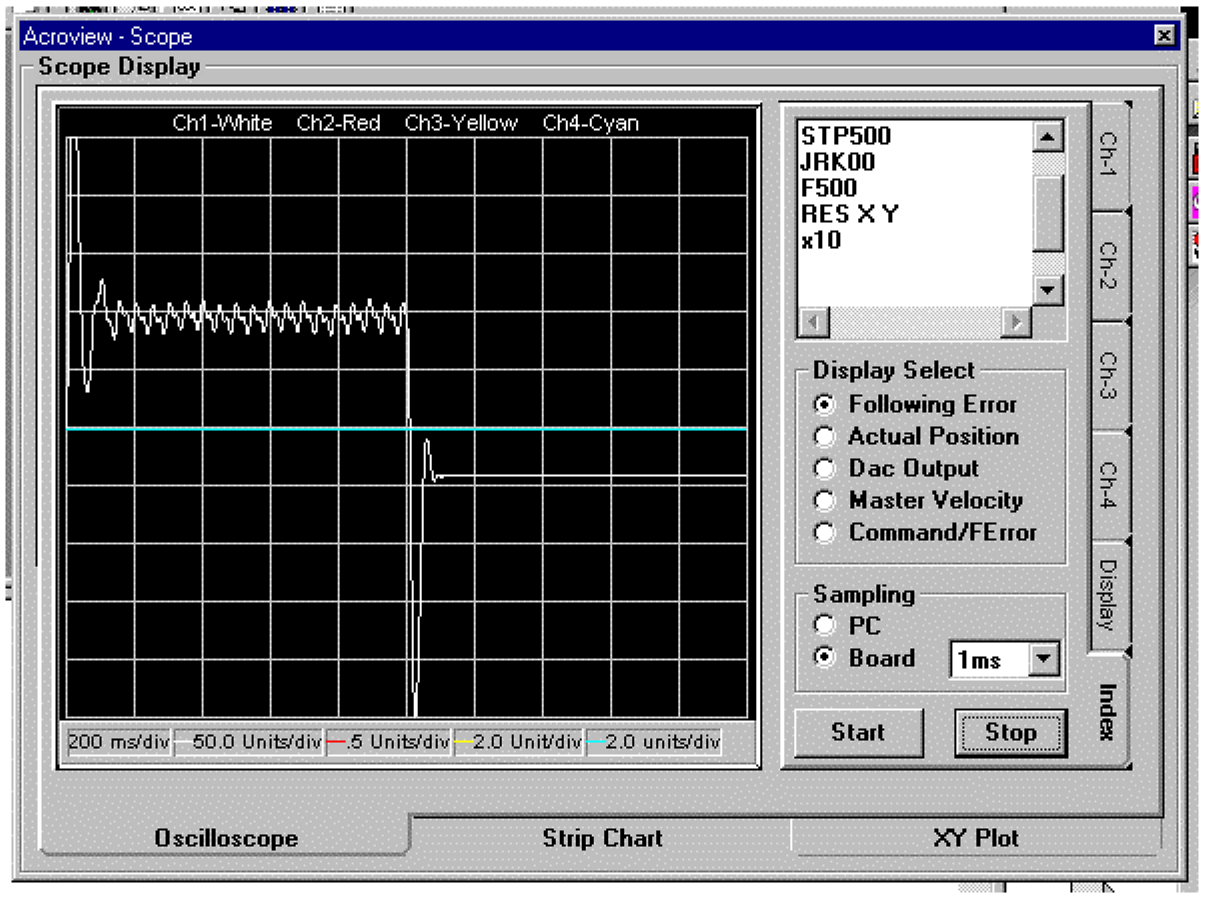


Note that there is a lot of ringing during the ACCEL portion of the profile and the DECEL portion at the end end. Note that the Horizontal scale is 200 msec/div and the vertical scale is 50 pulses per division. At this stage the following gain values are effective

PGAIN	0.001
IGAIN	0
IDELAY	0
ILIMIT	0
DWIDTH	0
DGAIN	0.000002
FFVEL	0
FFACC	0
TLM	4

Also verify that it is indeed taking approximately 100 msec to accel and decel in the profile. There is a steady error during the at speed portion of the profile. Additionally, there is steady state error after the profile is over and done with. All of these will be taken care of before we are done. Intuitively, note that PGAIN effects the entire portion of the profile. DGAIN effect portions where the following error is CHANGING. This is verified quickly by looking at the ringing, which seems very evident during the ACCEL and DECEL portions.

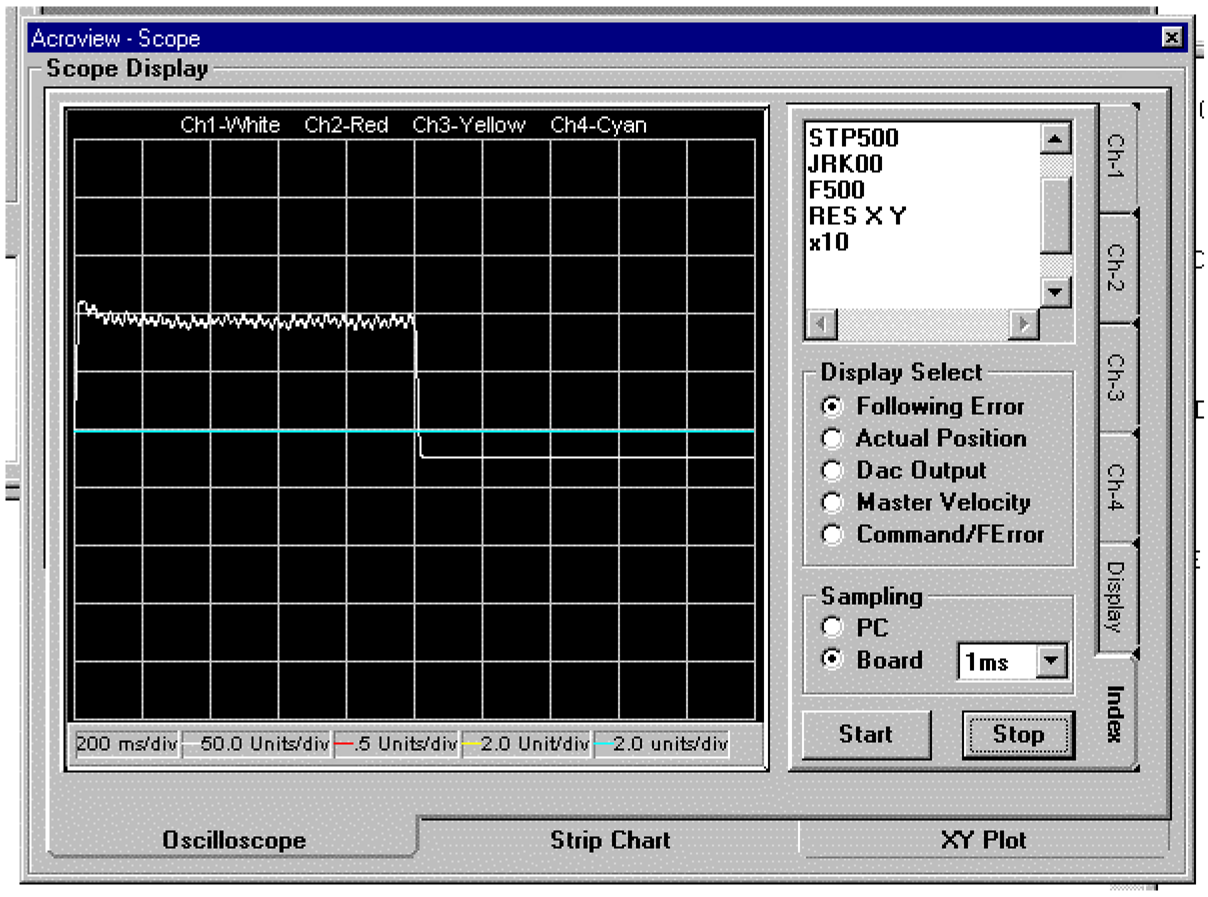
The ringing is due to not enough DGAIN in relation to the PGAIN. Note that the DGAIN and PGAIN kind of work opposite to each other in the sense that PGAIN adds and DGAIN subtracts from the signal. This next printout is when DGAIN is increased to 0.000008. Everything else remains the same.



Note that most of the ringing is gone. Note also that the ringing is less but the flat following error during the at speed portion of the profile has not changed. At this point the following values are active:

PGAIN	0.001
IGAIN	0
IDELAY	0
ILIMIT	0
DWIDTH	0
DGAIN	0.000008
FFVEL	0
FFACC	0
TLM	4

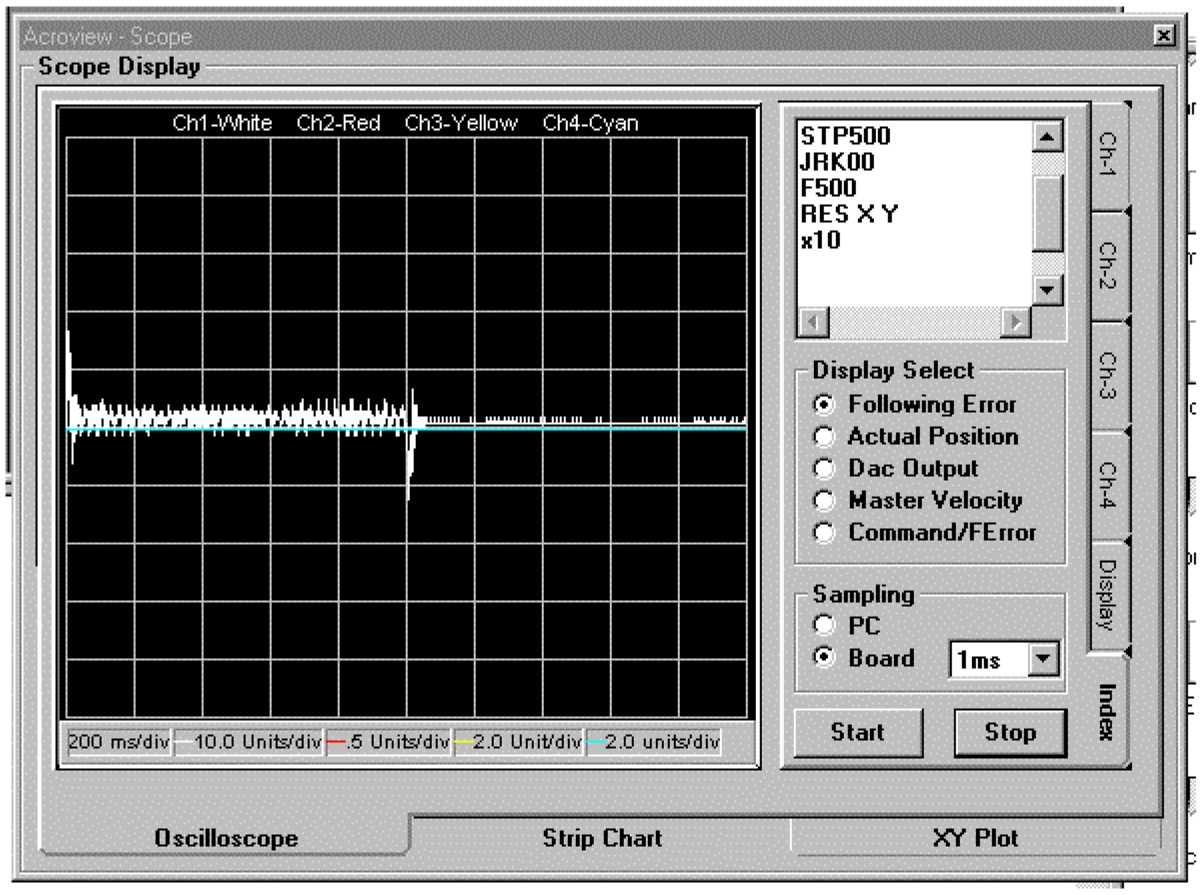
We must increase the DGAIN further to get rid of the last of the ringing. The next profile illustrates the point.



At this stage the following values are effective.

PGAIN	0.001
IGAIN	0
IDELAY	0
ILIMIT	0
DWIDTH	0
DGAIN	0.00005
FFVEL	0
FFACC	0
TLM	4

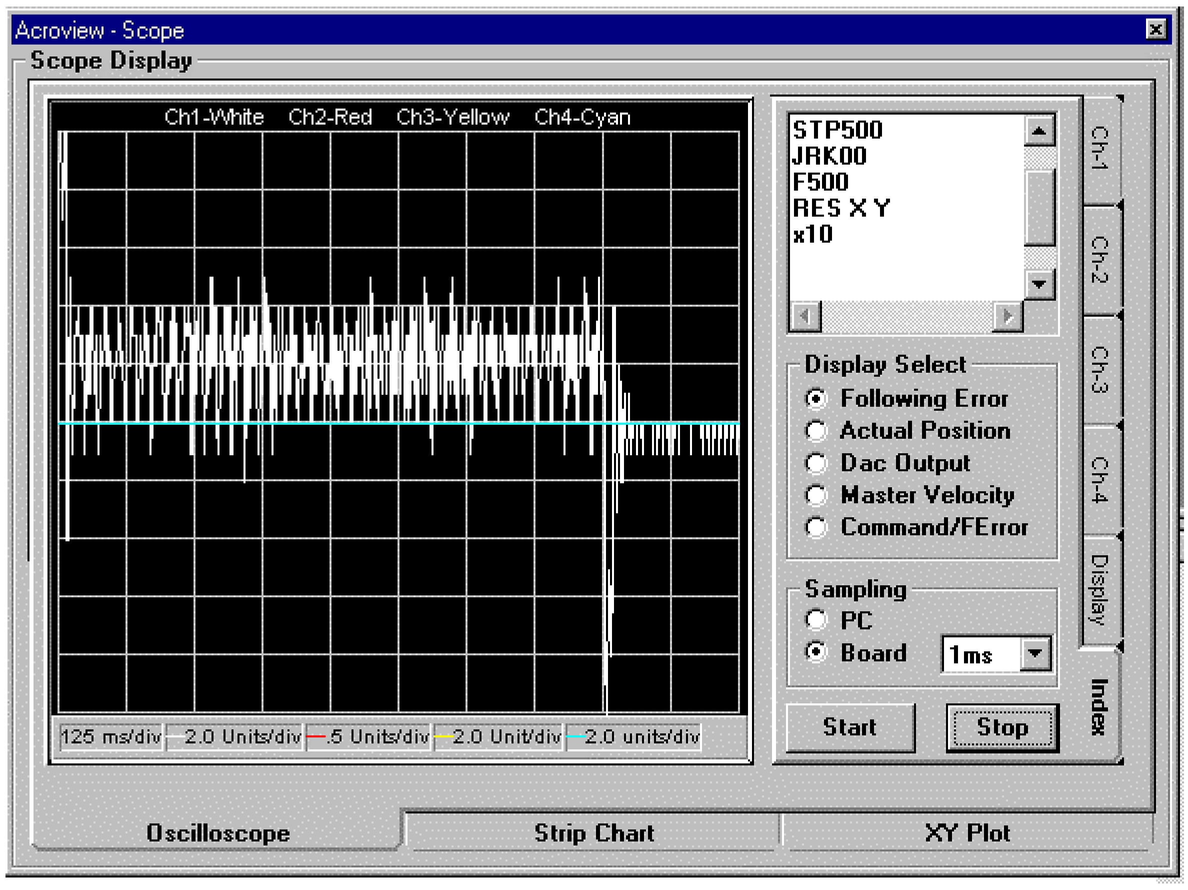
Note that the ringing is gone. Now, we can try to bring the AT SPEED following error down by increasing the PGAIN. Note that the following error is running at about 100 encoder pulses. The following trace is the result of just increasing PGAIN to 0.05



Note that with this gain, the following error has all but disappeared. To get the above trace, the Profile was run with vertical scale changed to 10 pulses per division. The following values are in effect at this stage.

PGAIN	0.05
IGAIN	0
IDELAY	0
ILIMIT	0
DWIDTH	0
DGAIN	0.00005
FFVEL	0
FFACC	0
TLM	4

Notice that the only problem we see is a blip in the ACCEL and the DECEL portion of the profile. This will be taken care of by changing the feedforward parameters. At this stage, we can take care of any steady state errors by bringing in some IGAIN. Also the vertical scale will be changed to 2 pulses per division.

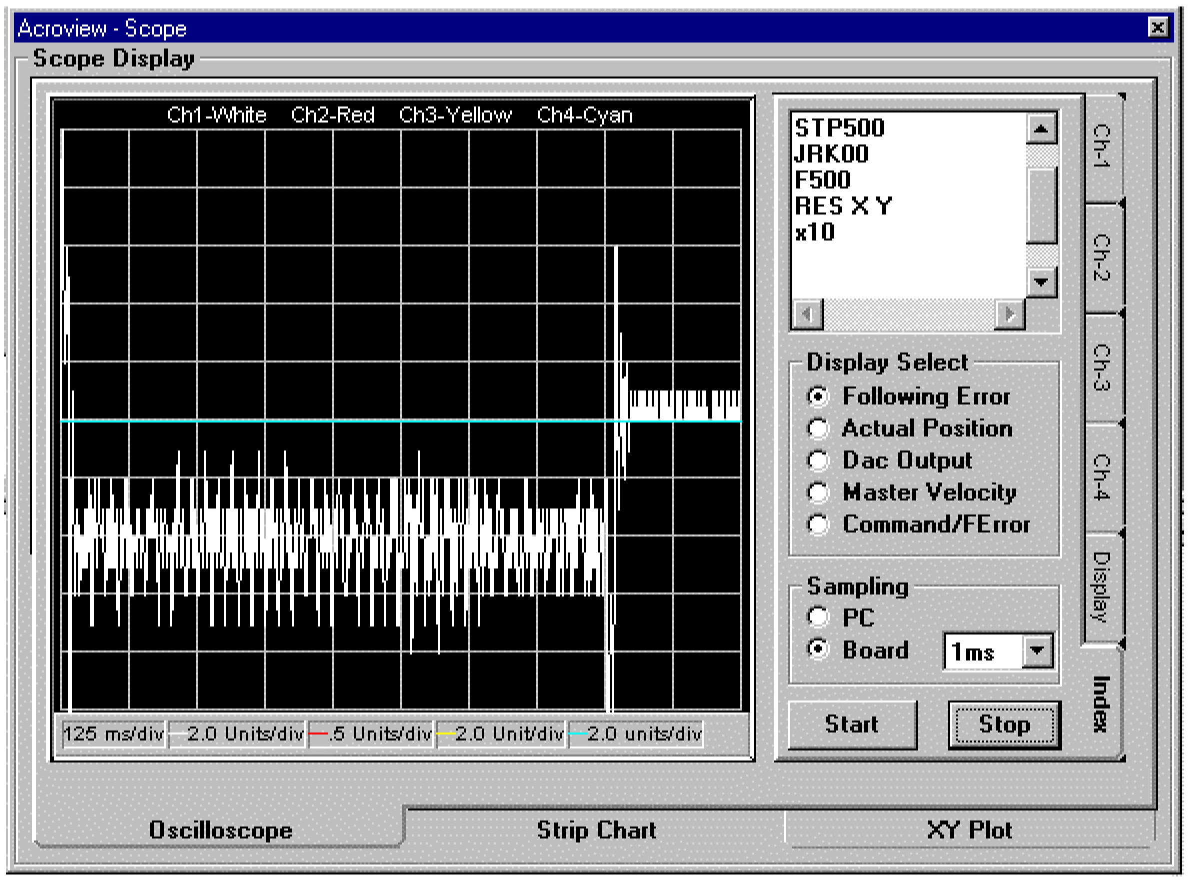


Note that the steady state error at the extreme right of the trace is showing 1 pulse error only! Also, there are error spikes during ACCEL and DECEL and some constant error during the AT SPEED portion of the profile. Note that the horizontal scale was also changed to be 125 msec per division to show the trace a little better. Also, the IGAIN is only a temporary value for now. The IDELAY is also setup for now such that the IGAIN only will start correcting 10 msec after the axis stops been commanded with a profile move.

At this stage the following parameters are in effect.

PGAIN	0.05
IGAIN	0.1
IDELAY	0.01
ILIMIT	1
DWIDTH	0
DGAIN	0.00005
FFVEL	0
FFACC	0
TLM	4

In the previous trace, the AT SPEED portion error will be fixed by the FFVelocity parameter. The ACC and DEC portion error will be fixed by the FFAcceleration parameters.
Next we will modify the FFVelocity parameter to bring the AT SPEED following error further down.



Notice that now the AT SPEED following error has FLIPPED and is now negative. This means that the FFVelocity parameter is too large. At this stage, the following parameters are active.

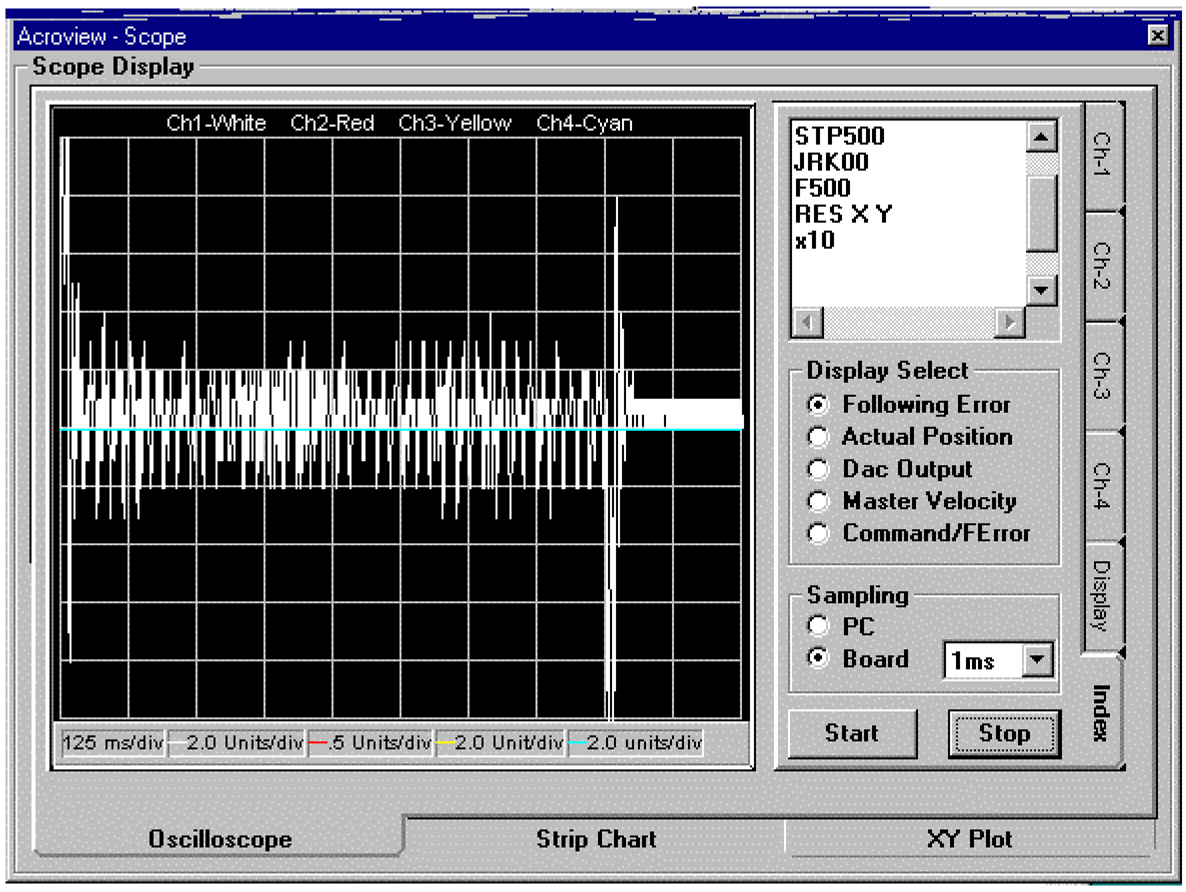
PGAIN	0.05
IGAIN	0.1
IDELAY	0.01
ILIMIT	1
DWIDTH	0
DGAIN	0.00005
FFVEL	0.00001
FFACC	0
TLM	4

At this time, it might be worth mentioning that the FFVel value can also be calculated from the following formula. Although for the purpose of this addendum, the manual way is being used for the purpose of showing how the numbers are derived and what effect they have on the overall response.

Run a trace with FFVel set to zero and note the At Speed, following error in pulses and also note the velocity in pulses per second. Then apply the following formula to come up with the correct FFVel number.

FFVel = (Pgain in Volts per pulse)* (Following Error In Pulses) / (Velocity In Pulses/Seconds)

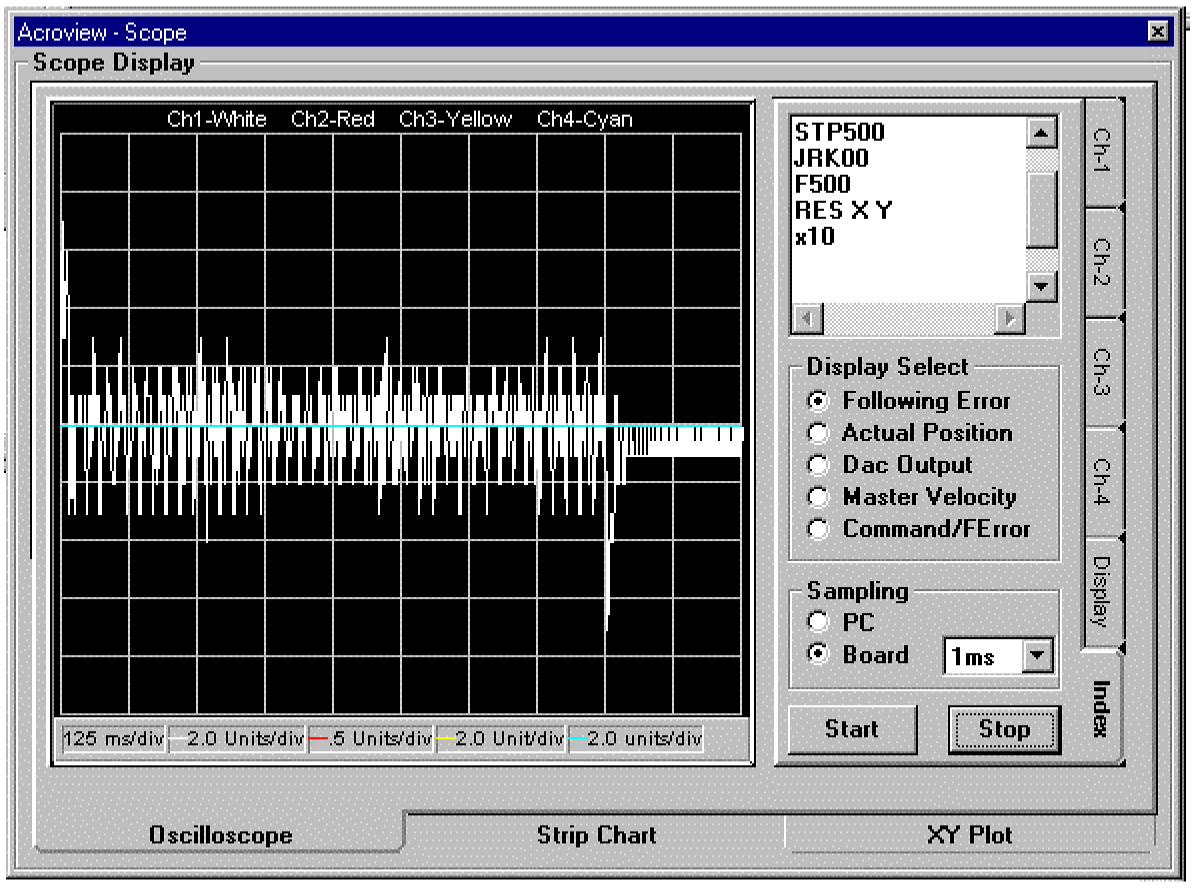
In any case doing it manually, trying a lower value the following trace is captured.



Note that this now looks pretty good except for the ACCEL and DECEL portions. At this stage, the following gain parameters are active.

PGAIN	0.05
IGAIN	0.1
IDELAY	0.01
ILIMIT	1
DWIDTH	0
DGAIN	0.00005
FFVEL	0.0000015
FFACC	0
TLM	4

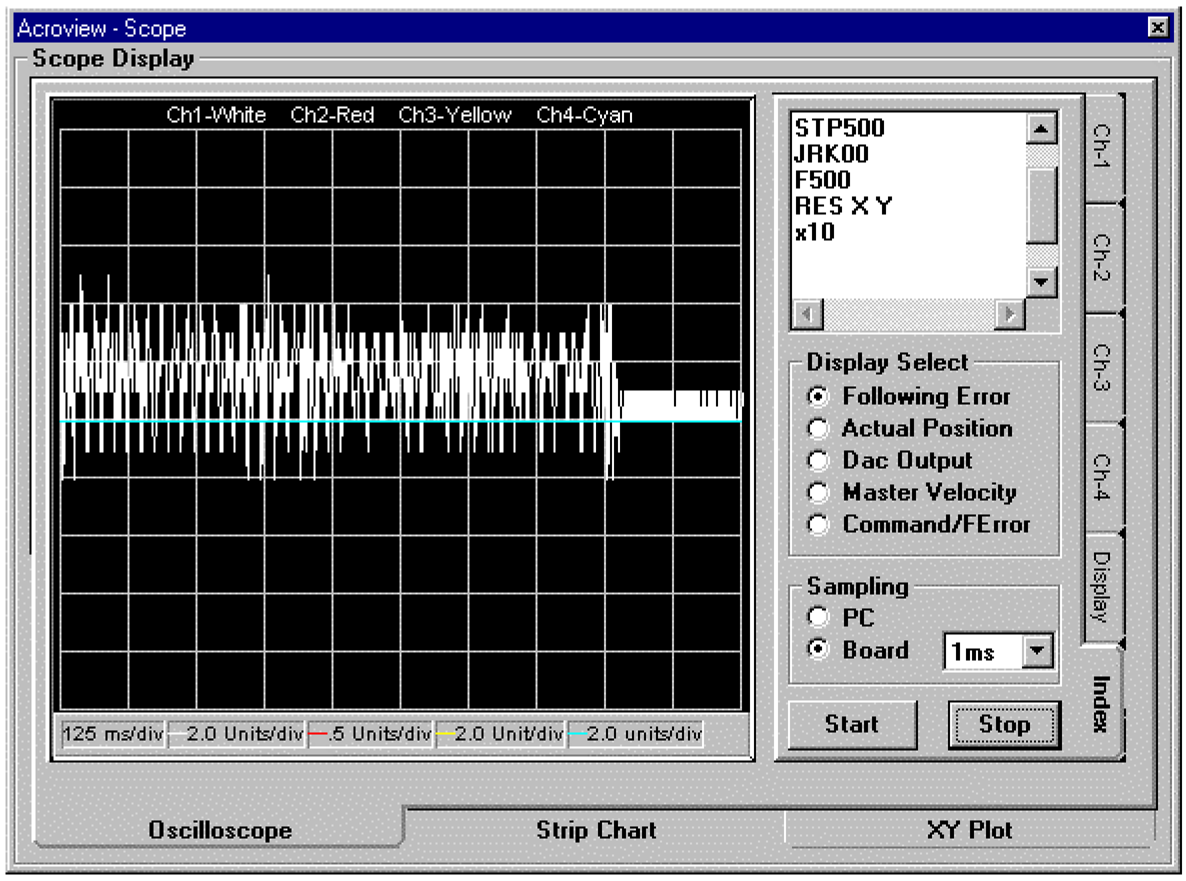
Now we can tackle the ACCEL and DECEL portions. Starting with the value of 0.000 000 1, the following trace is captured.



Note that this did improve the ACCEL and DECEL portions. There is still room for improvement. At this stage, the following gain parameters are active.

PGAIN	0.05
IGAIN	0.1
IDELAY	0.01
ILIMIT	1
DWIDTH	0
DGAIN	0.00005
FFVEL	0.0000015
FFACC	0.0000001
TLM	4

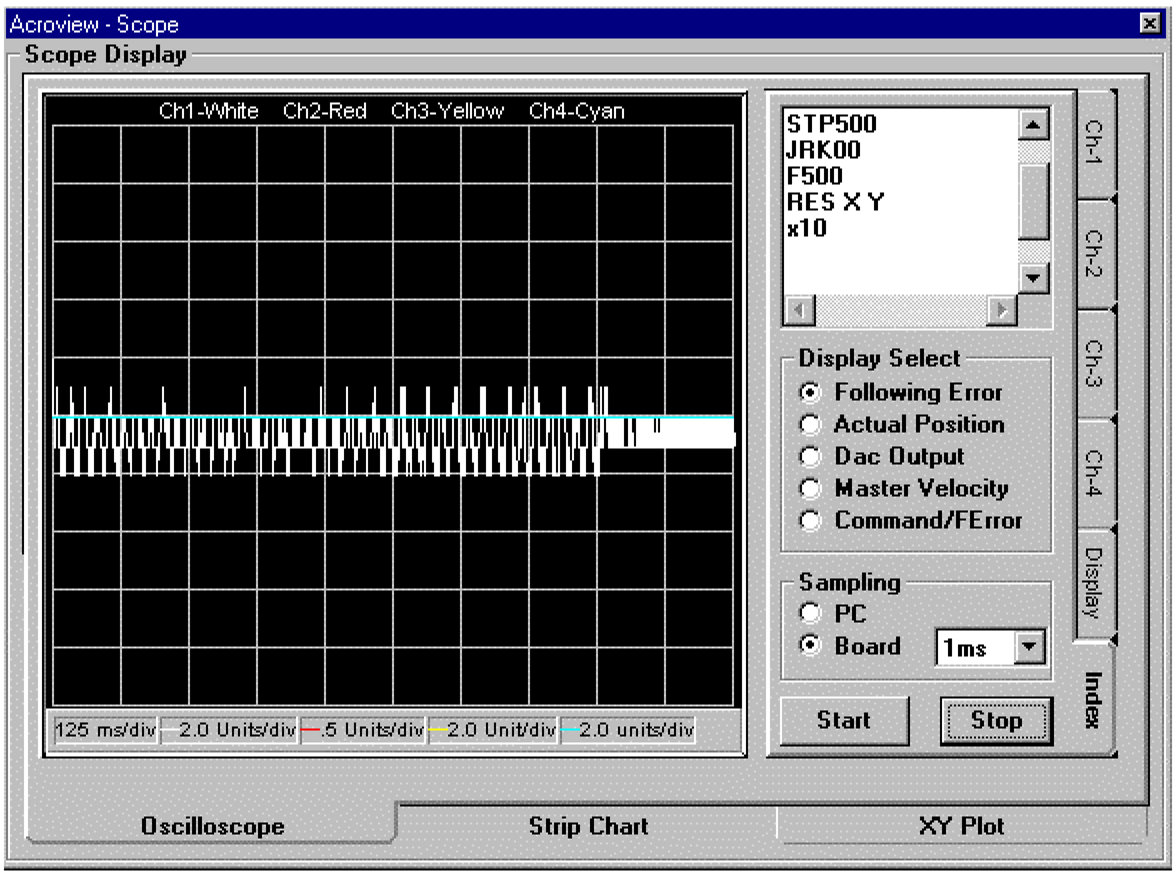
After a couple of tries the following trace was captured.



Note that the spikes are now gone and overall trace is looking very good. The following gain parameters are active at this time.

PGAIN	0.05
IGAIN	0.1
IDELAY	0.01
ILIMIT	1
DWIDTH	0
DGAIN	0.00005
FFVEL	0.0000015
FFACC	0.000000199
TLM	4

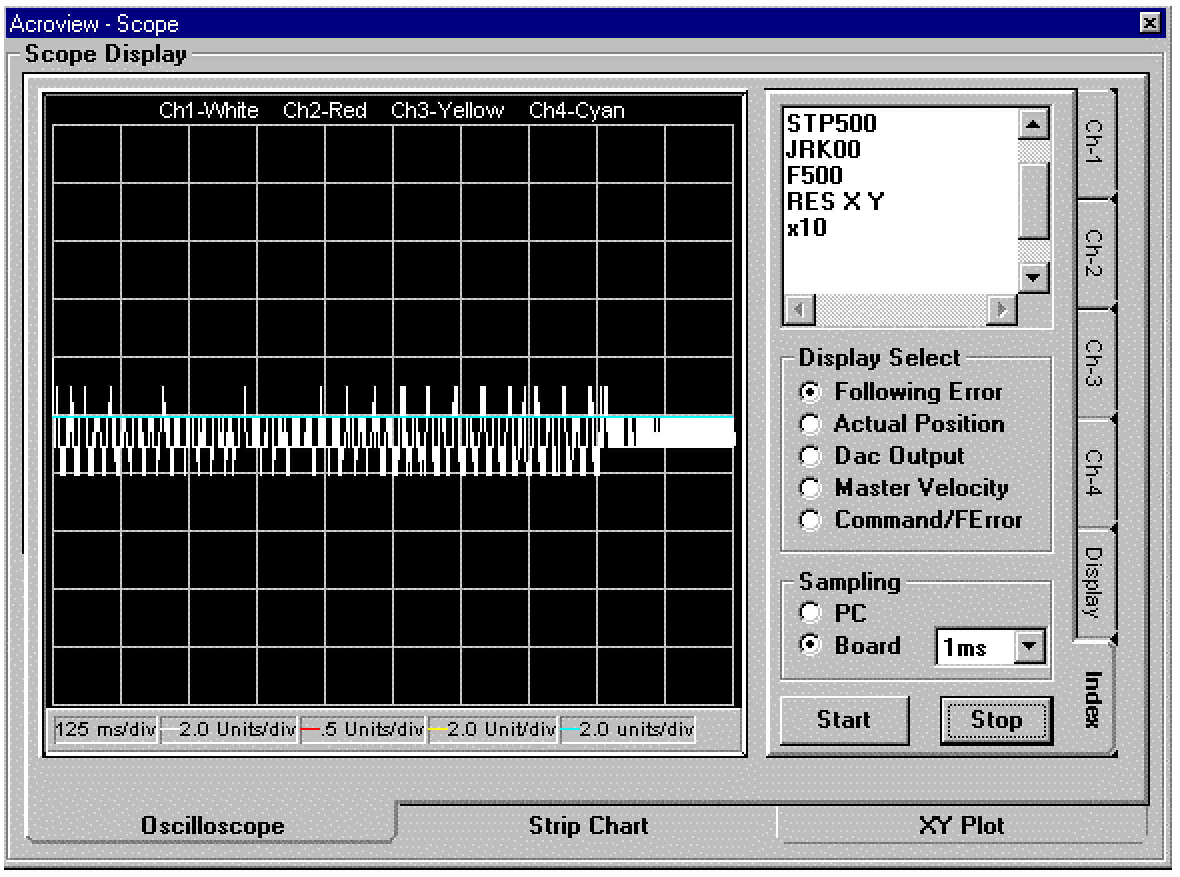
At this stage, we can still tweak the DGAIN and the PGAIN up and down a little to see if we can improve the trace still further. Lastly, we will look at IGAIN and enable it to work at all times.



This did seem to improve the trace quite a bit. At this stage, the following parameters are in effect. The DGAIN was increased to 0.00021 in two tries.

PGAIN	0.05
IGAIN	0.1
IDELAY	0.01
ILIMIT	1
DWIDTH	0
DGAIN	0.00021
FFVEL	0.0000015
FFACC	0.000000199
TLM	4

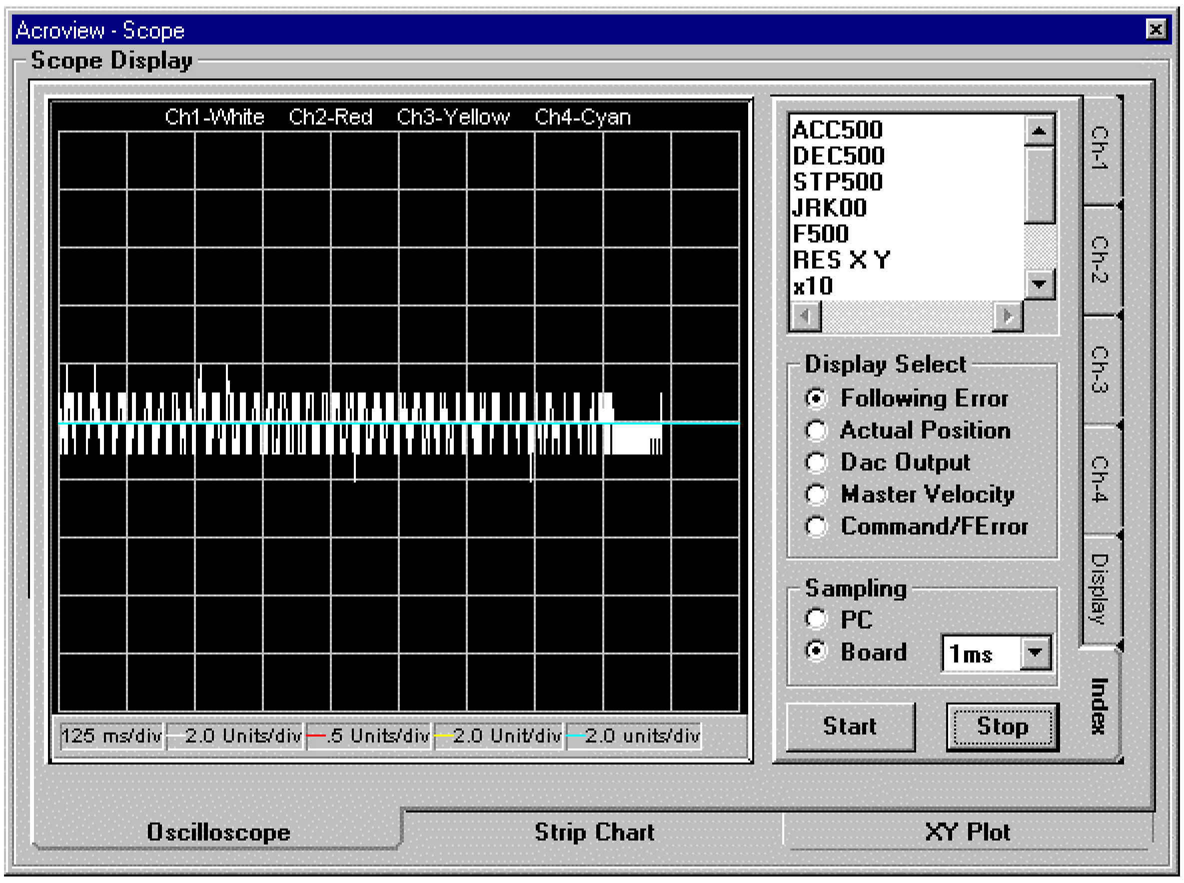
Next we can try to increase the PGAIN a little more. Raising it to 0.08 showed improvement. Raising it further caused instability. So at 0.08 the following trace was captured



No drastic improvement, but we will leave it here, as the higher gain will combat load disturbances better than the lower value, as long as the motor is not buzzing or unstable.
The following gain values are active.

PGAIN	0.08
IGAIN	0.1
IDELAY	0.01
ILIMIT	1
DWIDTH	0
DGAIN	0.00021
FFVEL	0.0000015
FFACC	0.000000199
TLM	4

Now we will take care of continuous load changes affecting the stability of the system. For this, the IGAIN will be enabled to work at all times, instead of after the Profile move stops. The following trace will captured using the IGAIN of 2 and ILIMIT of 2



For the above trace, the following gain parameters are in effect.

PGAIN	0.08
IGAIN	2
IDELAY	0
ILIMIT	2
DWIDTH	0
DGAIN	0.00021
FFVEL	0.0000015
FFACC	0.000000199
TLM	4

To filter or not to filter

On top of the main nine main parameters, we still have two separate 2nd Order BI-QUAD FILTERS that have not been used yet. These can be utilized to either filter out a band of undesirable frequency from the DAC output or implement a LOPASS filter to reduce the bandwidth of the DAC output.
Servo amplifiers from different manufacturers vary tremendously in their bandwidth. Some of the analog servo amplifiers might have a bandwidth as high as 500Khz while the digital ones might be as low as a few hundred Hz. All of this, with the machine mechanics, might have the bandwidth substantially lower than 100 Hz. As an example, a motor running a Disk drive in a PC might have a bandwidth on the order of 500Hz.
Depending on the above factors, one can implement a lopass filter at a 1000Hz to reduce some sharp peaks going to the amplifier. Most cases will not need the filter at all. In the case where a NOTCH filter is needed or a LOPASS filter is needed, the following commands can be implemented.

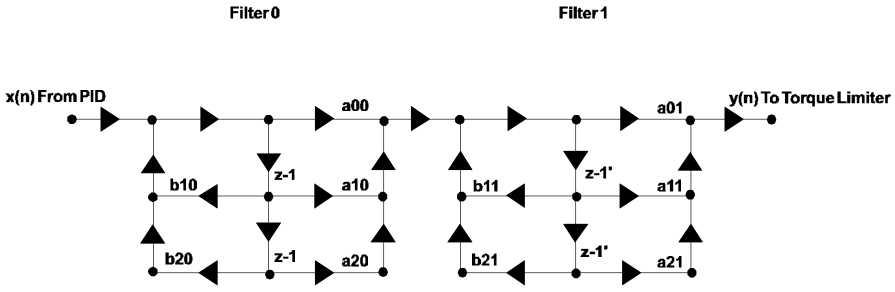
LOPASS X1000
REM This sets up one of the BiQuads as a LOPASS filter for X Axis with 1000Hz 3db rollover .

NOTCH X(500, 50)
REM This sets up one of the BiQuads as a NOTCH filter at 500Hz with 50Hz bandwidth.

The necessity of the above entirely depends on each application. The scope display will let the user know if either of these are helping or hurting the tuning effort. Playing with these must be the last step in the tuning process. The user can also not even use the LOPASS and the NOTCH command and choose to load the POLES and ZERO information directly into the CO-EFFECIENTS of these cascaded filters directly.

Filter Mathematics.

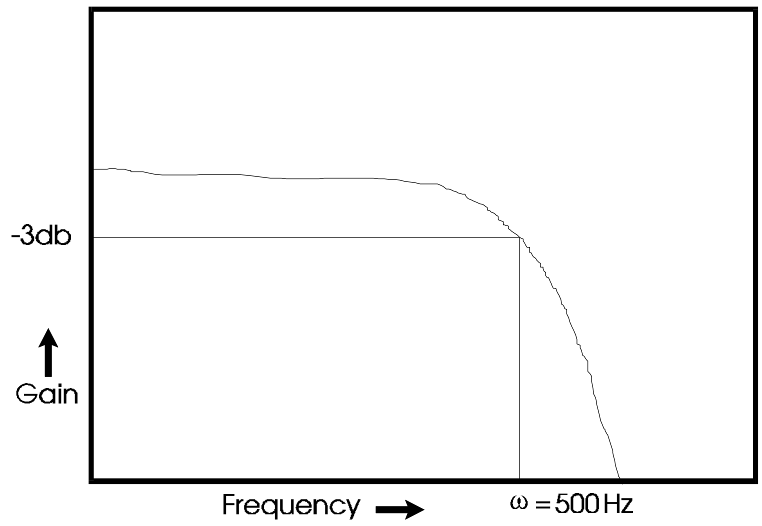
There are two separate filters that are cascaded and inserted between the output of the PID filter and the input of the Torque Limiter (TLM). When the filters are not in use, the output of the PID is fed directly into the input of the Torque Limiter.
The bottom line reason for the two filters is to suppress certain range of frequencies from showing up at the final DAC output. The NOTCH filter suppresses frequencies between a LOWER threshold and a HIGHER threshold. The LOPASS filter suppresses all frequencies higher than the threshold frequency. Here the word suppression is used loosely as the frequencies are not simply cut off, but rather the strength of the frequencies being suppressed is greatly attenuated. In the case where the machine has a certain frequency range that sets up oscillations, the control can ensure that it “skips” that range in its DAC output so as not to cause the machine to go unstable.



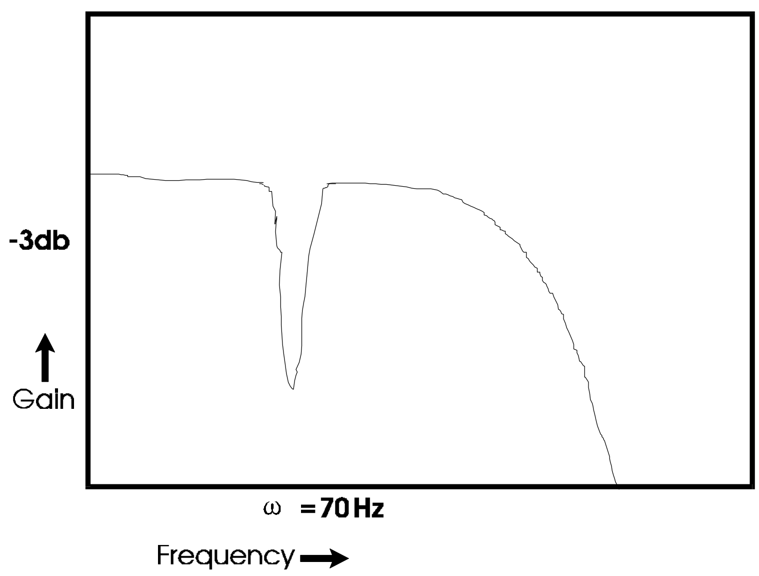
The above two filters have the following equation in the Z Domain.

$$H(z) = \left[\frac{a00 + a10z^{-1} + a20z^{-2}}{1 - b10\,z^{-1} - b20\,z^{-2}}\right]\left[\frac{a01 + a11z^{-1} + a21z^{-2}}{1 - b11\,z^{-1} - b21\,z^{-2}}\right]$$

In the current example, the servo period was running at 2500 Hz. Any filter frequencies approaching the servo period frequency will be useless. Trying frequencies in the range of 400Hz to 1000Hz, it was determined that the performance was not improving, so the filter was disabled. The following plot show a lopass filter at 500 Hz

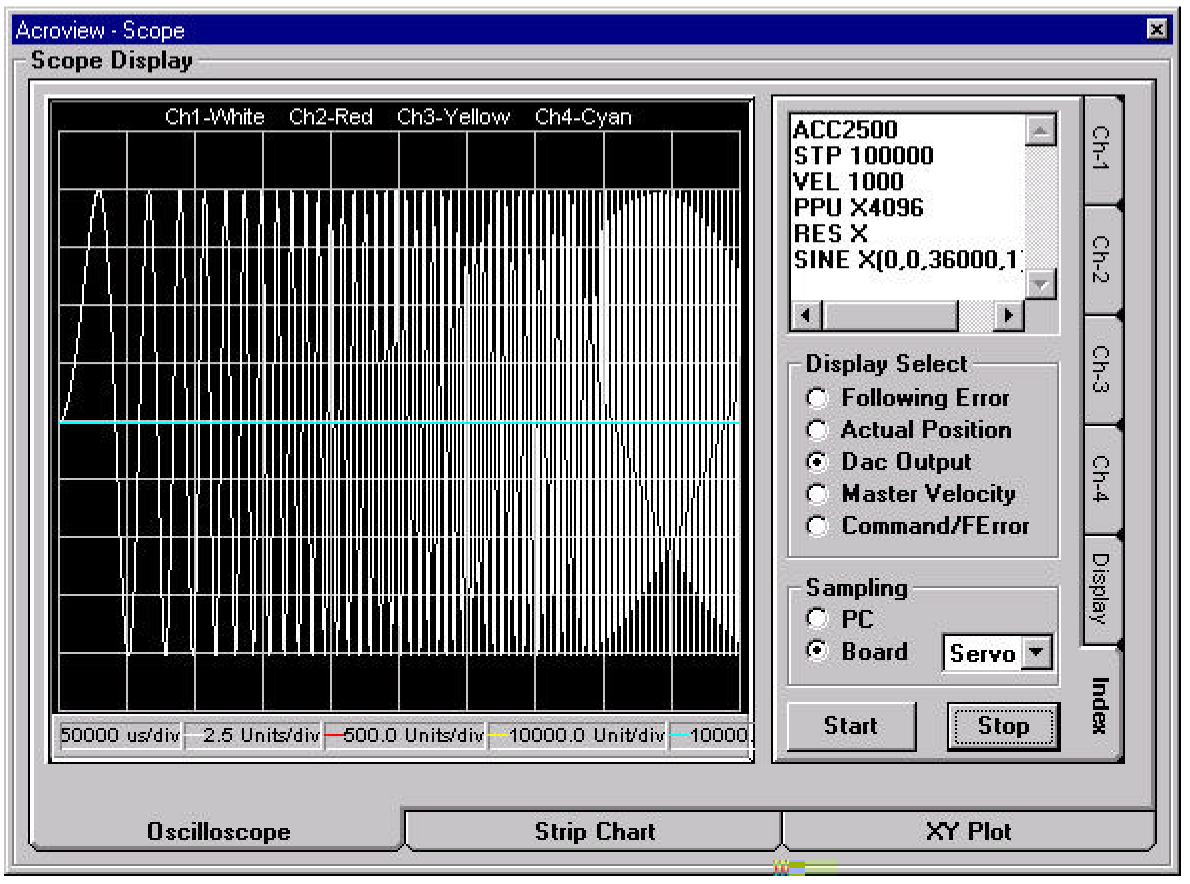


The following plot shows a NOTCH filter at 70 Hz.



In all probability, the tuning for this example was done on a test bed and therefore the motor was running under no load conditions. Any potential machine harmonics and other subtleties that would require us to use one or both filters were not present.

As an example, the controller will generate a pure sine wave pattern. The DAC output will be sampled and displayed in the Scope. For this test, the motor/amplifier will be disconnected. The Sine wave pattern is generated by commanding the SINE command with the final velocity set to 1000Hz and the accel set to 2500cycles per second per second and a total move distance of 100 revolutions (36000 degrees). In reality, this will result in the SINE waves starting at very low frequency and accelerating to 1KHz in about 400 msecs. So on the Scope display in Acroview, the left-hand side of the trace will represent low frequencies and the right side will show frequencies closing in towards 1Khz. On purpose the amplitude of the sine wave will be kept small enough as not to cause the DAC to go into saturation. This will allow us to view the SINE wave with a +/-10 volt swing. The following trace shows the raw unfiltered DAC output. The DAC channel is CH2 and is setup for showing 2.5volts per div. DAC gain is set so that 1 REV of following error gives 10 volts (PGAIN = 0.00244). With the motor disabled and the PPU set for 4096 pulses (1/2 of 16 bit DAC scale), Sine wave of 1 Unit will give us 10 volts on the DAC.



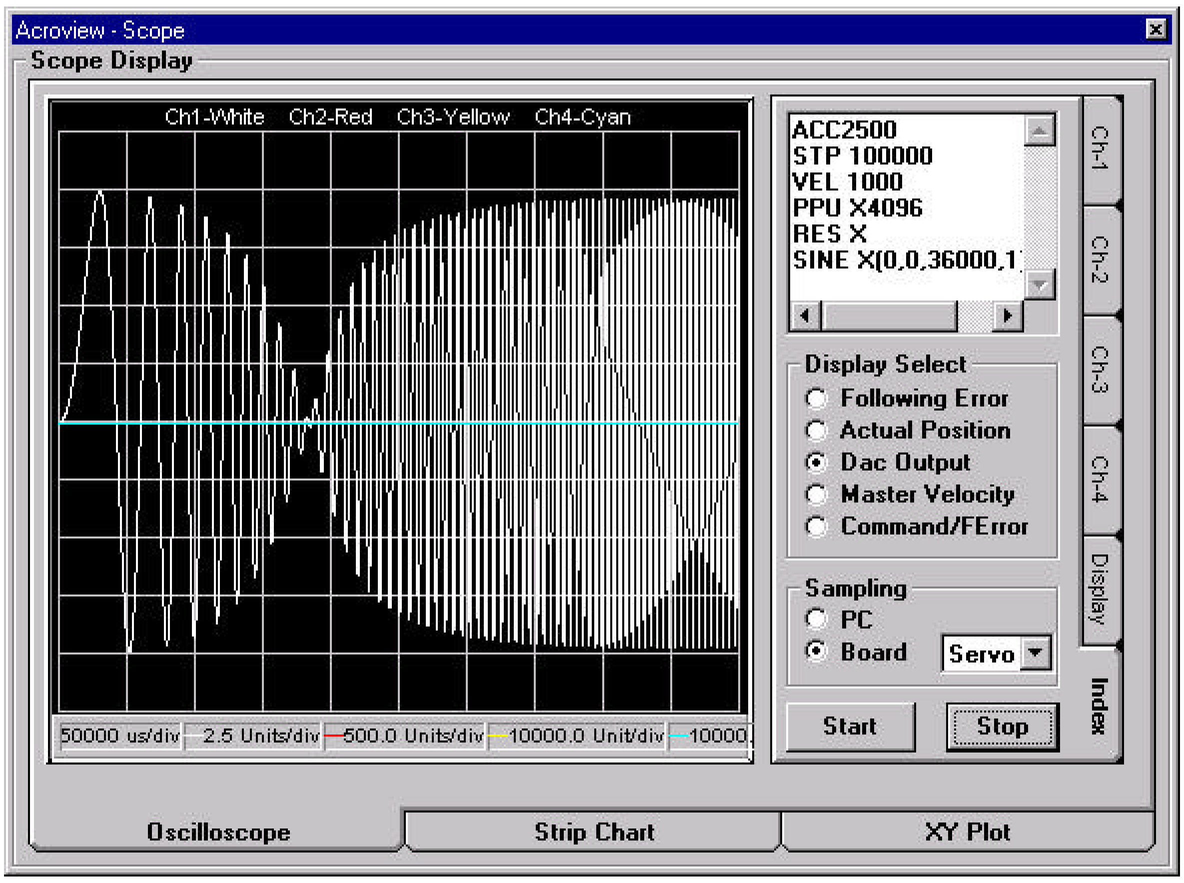
First we will observe the effects of a LOPASS filter on the DAC output, and then the effect of the NOTCH filter. The period for the controller will be changed to 5000 Hz. Then the NOTCH filter will be introduced.

NOTCH FILTERING.

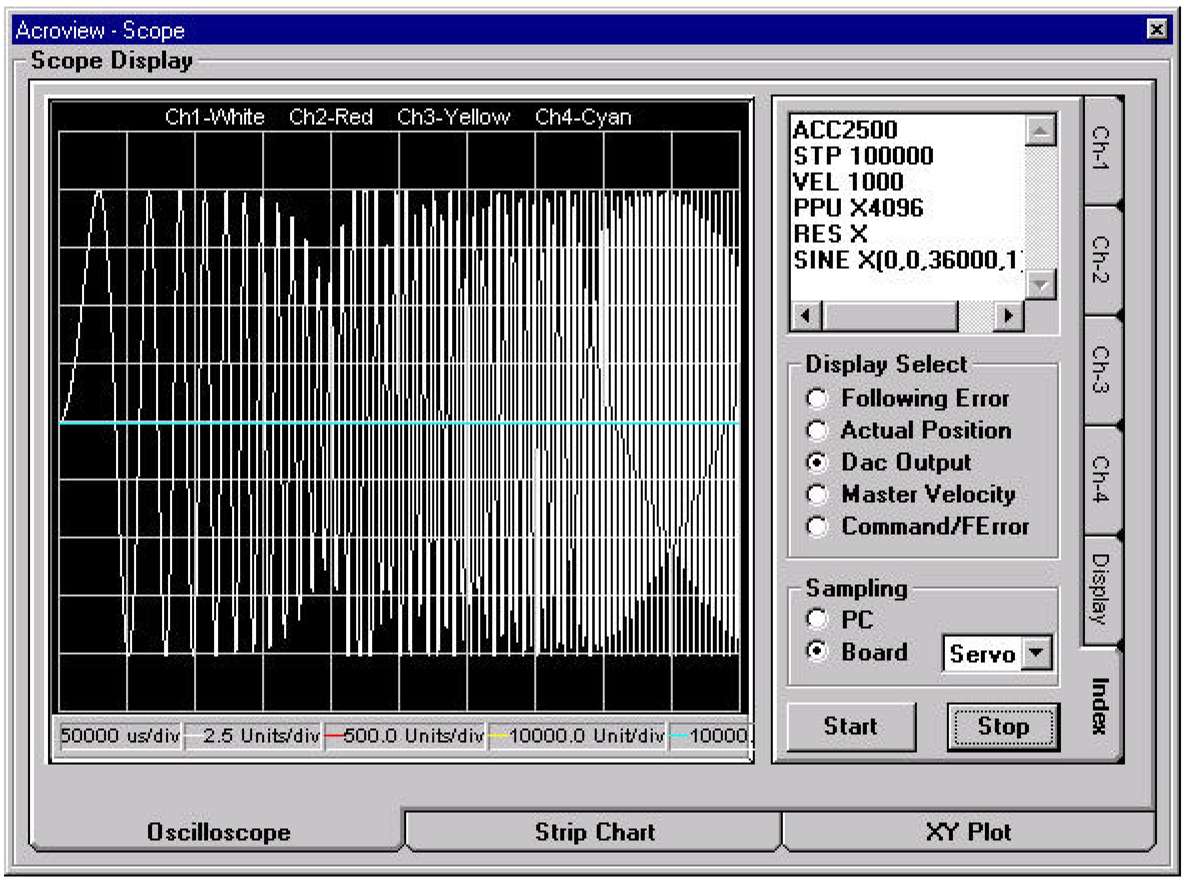
The NOTCH filter has two parameters. The first is the NOTCH frequency. This is the frequency that causes the particular mechanics to oscillate uncontrollably. This is the frequency we have to suppress from the DAC. The second parameter is the bandwidth of the notch. The larger the bandwidth the higher the attenuation that the filter will offer. On the flip side, the larger the bandwidth, the more range of frequencies that will get killed uselessly. The idea is to only reduce or attenuate the signal as much as needed to eliminate the excitation of the mechanics of the system. If the NOTCH width is too large, the entire system will become unstable and unusable. The width of the NOTCH can be specified as low as 1

Hertz. But at this value, the attenuation might be very little. If the width is made as wide as the NOTCH center frequency, the DAC output will completely go to ZERO at the undesirable frequency. The user should start at the low value of 1 and work upwards until the problem gets fixed. The following trace shows the effect of calling up a NOTCH filter at 100Hz with the NOTCH width of 100 Hz. The Servo Update was setup at 5000Hz. The following command was used

NOTCH X(100,100)



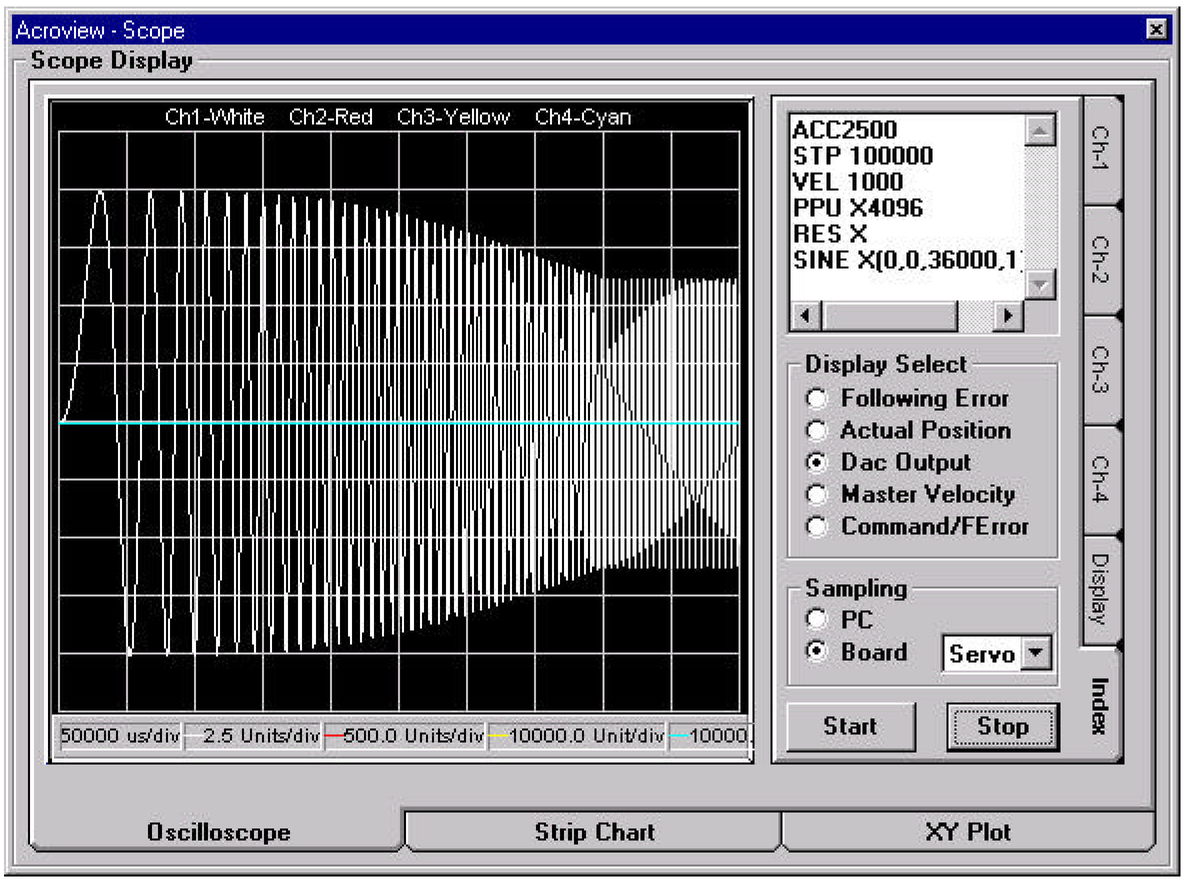
Notice that the DAC signal goes to ZERO at the NOTCH frequency. This is shown purely for the sake of understanding the command, as there is such a large gap in the output frequency that the machine would not be stable with such a large notch. The next trace shows a more reasonable choice of NOTCH width.



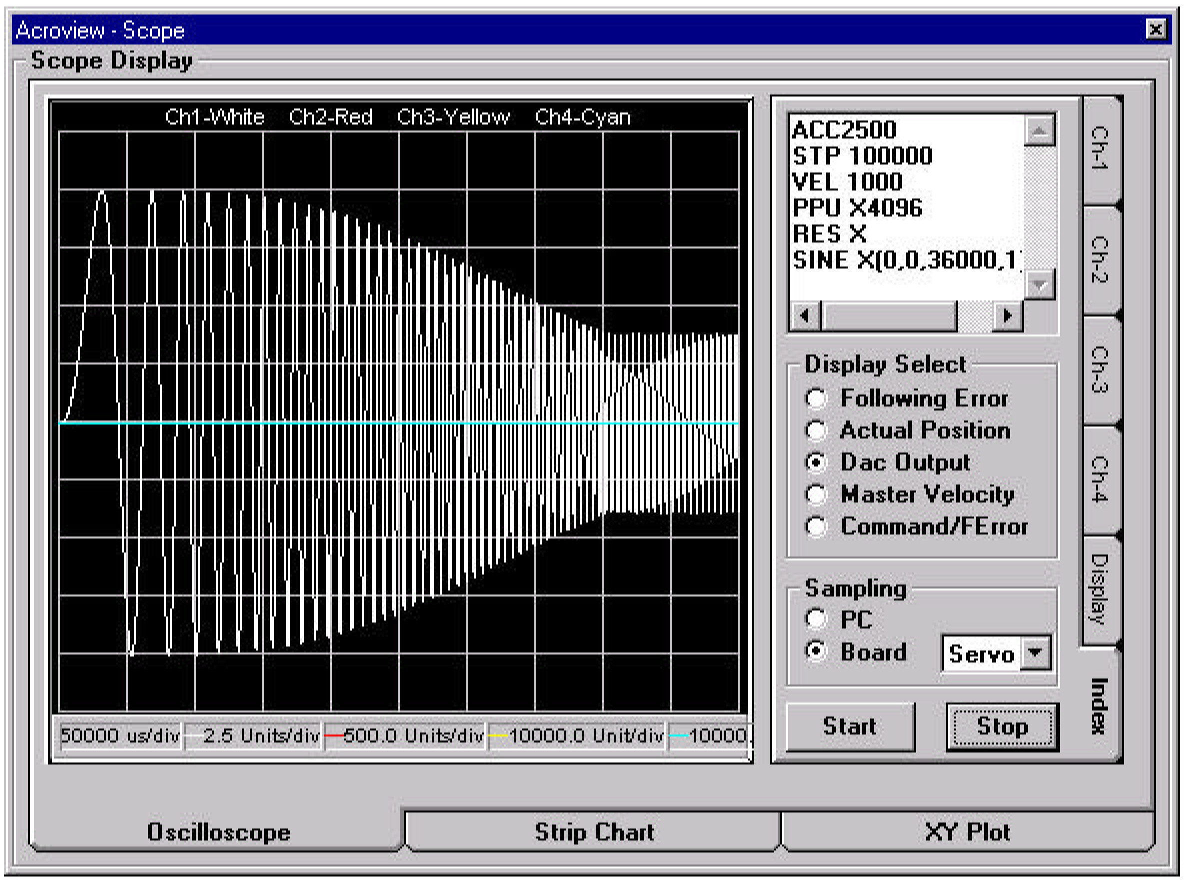
Here, the signal dips slightly at the undesirable frequency and comes right back up within a few cycles. The key is to determine whether the amount of attenuation is adequate to suppress the harmonics in the mechanics. This can be achieved via some experimenting. Usually most methods of excitation of the mechanics can tell the system designer not only what frequencies must be suppressed, but also how small or large the bandwidth of such frequencies is.

Next we will cover the LOPASS filter. Unlike the NOTCH filter, which needs careful investigation, the low pass filter frequency can be chosen such that it is well beyond the calculated bandwidth of the mechanics. The low pass frequency must not exceed 10% of the servo update frequency. The following trace shows a lopass filter set for 200Hz with the Servo Update set at 5000Hz. The following command was used.

LOPASS X200



In the above case, the lopass filter can be seen to be attenuating all frequencies starting at 200 Hz. Note that both the LOPASS command and the NOTCH command each use ONE bi-quad filter. If the NOTCH filter is not being used, adding a second lopass filter by duplicating the filter parameters from FILTER0 to FILTER1 can increase the slope of the LOPASS filter. The following trace was obtained by doing just that.



This was achieved by copying the filter0 parameters into filter1 parameters.

P12341=P12336
P12342=P12337
P12343=P12338
P12344=P12339
P12345=P12340

Note that the attenuation is now sharper than the previous trace.

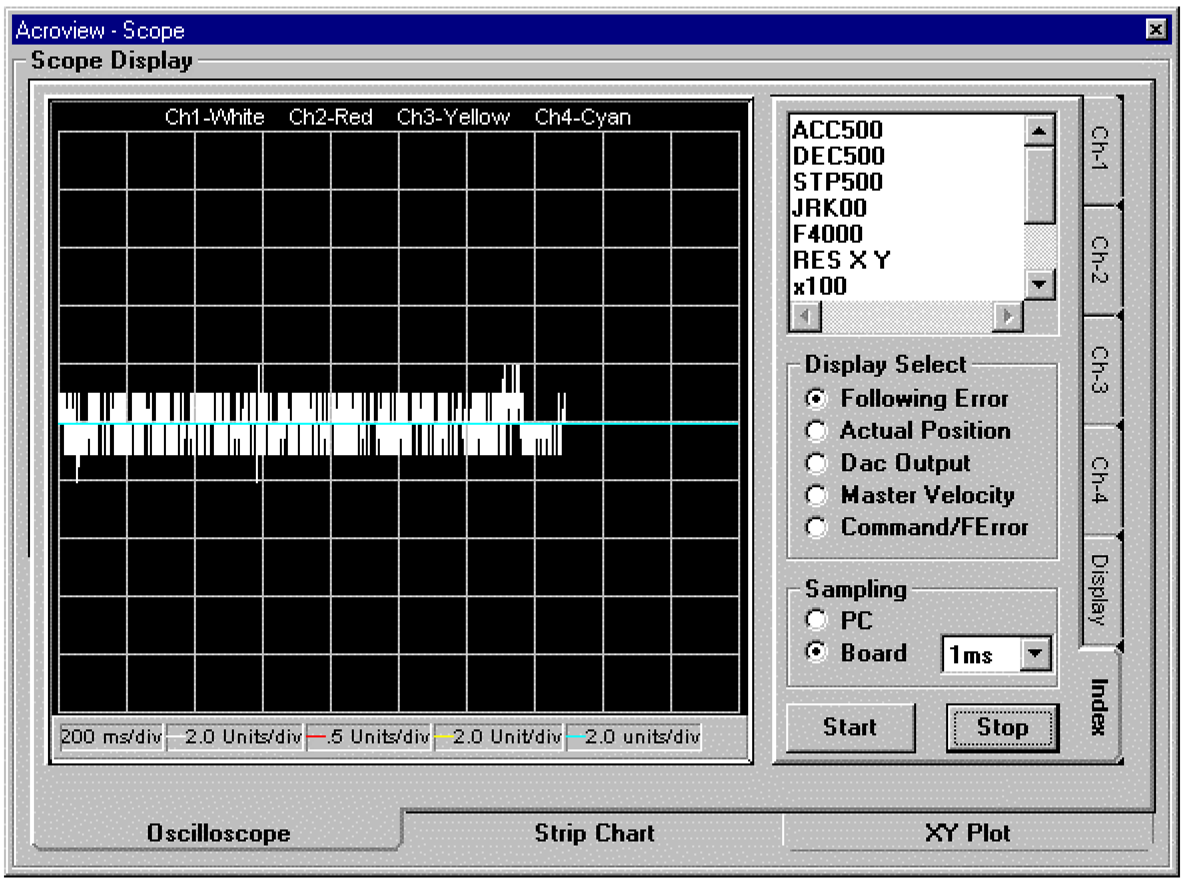
The user is free to design any type of filter by loading values into the filter co-efficients. The "BIQUAD FILTER ACTIVATE" bit flag controls the usage of the filters. The user must first disable the bit, then change the filter parameters as needed and reactivate the filter again.

Tuning with DUAL FEEDBACK encoder arrangement.

In some cases, the user will have two encoders for each axis. The purpose of the two encoder feedback is to allow the position feedback to be taken from the encoder mounted directly on the load. A second encoder is then mounted on the motor shaft. Doing this is entirely optional. The pros and cons can be argued either way. In any case, when there is a separate encoder feedback mounted on the motor and is different from the position feedback encoder mounted on the load, the FBVel allows the user to inject the motor velocity signal into the SUMMATION point that is fed into the dual bi-quad Digital filter stage. The signal can be thought of as a ‘software tachometer’ signal and is subtracted from the output of the PID loop. This is shown in the ACROVIEW-SERVO LOOP screen.
The FBVel parameter dampens the servo loop. It is to be used in addition to the the normal PID parameters, not instead of them. The optimization of this parameter must be done by viewing the SCOPE trace during the tuning process. Note that this parameter can not be changed from the GAIN screen in the current version of Acroview. Therefore, it can be either changed via the terminal or directly from the Index Text Box on the SCOPE screen.
Be aware that because the current Acroview version does not allow changing this parameter from the GAIN screen, it will not be saved automatically in the user project. Therefore, this parameter must be loaded with the desired value for all applicable axes in the user program(s).

Wrap Up.

This completes the TUNING effort. These parameters will serve for the entire range of RPM on this motor as long as the motor has enough power to push the load. The gain settings will work for all RPM and ACC and DEC ranges. For example the following trace was done for the same motor but at 4000 RPM and 100 REVS. The Horizontal scale was changed to 200msec per division to get a better picture.



Even though we tuned at a much slower speed, the gain parameters are flawlessly performing even at higher RPM's. Some might choose to tune at the fastest application speed and the fastest application acceleration, but there are issues of safety that get magnified at high speed. If one does the tuning properly at the fastest possible required acceleration and moderate speed, the parameters should hold true for the entire range of speeds assuming that the motor/amplifier combination have enough torque to drive the load at the required speed and acceleration.

With the above settings, even if the motor sees on the fly load changes, the following error should stay fairly close to zero.

CUSTOMIZING ACROMILL NT

DISCLAIMER

AcroMill allows registered VAR'S to enhance the power of the standard AcroMill software. Only experienced VARs should be attempting to customize this software. If an end user has a need to add or customize his machine, getting in touch with Acroloop will get him referrals to the nearest VAR that can efficiently deal with the requirement. Modifying critical portions of the AcroMill by an unauthorized technican or programmar will void warrantees and may lead to dangerous machine behaviour.