ANILAM

6000M
CNC Setup Utility
Manual

www.anilam.com

Warranty

ANILAM warrants its products to be free from defects in material and workmanship for one (1) year from date of installation. At our option, we will repair or replace any defective product upon prepaid return to our factory.

This warranty applies to all products when used in a normal industrial environment. Any unauthorized tampering, misuse or neglect will make this warranty null and void.

Under no circumstances will ANILAM, any affiliate, or related company assume any liability for loss of use or for any direct or consequential damages.

The foregoing warranties are in lieu of all other warranties expressed or implied, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose.

The information in this manual has been thoroughly reviewed and is believed to be accurate. ANILAM reserves the right to make changes to improve reliability, function, or design without notice. ANILAM assumes no liability arising out of the application or use of the product described herein.

Section 1 - Setup Utility Concepts

Section 2 - Machine Constants

Section 3 - Other Setup Options

Section 4 - Configuration Utilities

Section 5 - Tuning the Current, Velocity, and Position Controller

Section 6 - Setup Utility Maps

Section 1 - Setup Utility Concepts

Introduction

The Setup Utility is used to configure the Computer Numerical Control (CNC) and optimize the system. The machine builder performs most of the initial machine setup at the time of the installation. This manual documents all parameters and the procedure to change them. All changes are made using the Setup Utility. The parameter settings are saved in a configuration file in the CNC's memory. The name of the configuration file is P6MCFG.CFG.

Effectivity Notation

Some sections of this manual apply only to specific ANILAM CNC product(s). In these sections, icons in the left margin identify the product(s) to which the information applies. **Table 1-1** lists the icons for each CNC product and the number of axes supported by each product.

Table 1-1, CNC Effectivity Icon Description

Icon	Product	Axes Supported
6000M-3X	6000M-3X Systems	3
6000M-4X	6000M-4X Systems	4

NOTE: All systems also support one spindle axis.

The main difference between the products is the number of axes supported. Generally, this manual describes the 6000M-3X systems. The 6000M-4X operates exactly as the 6000M-3X system except for features that include the additional axes.

There are many parameters that are defined per axis. In these cases, this manual will document the primary axes (that is, XYZ). The parameters for the auxiliary axis (that is, U) are entered in the same way as those for the primary axes. Some parameters can also be specified for the Spindle axis (that is, S).

Software Version Information

To facilitate verification of software version information, a text file is added to all CNC machine and offline software disks. The file lists the version and the CNC type. The software version contained on the disk is coded into the filename using the following format: 0xxxx.txt. For example, software version 4.14A is formatted as **0414A.txt**. Therefore, a disk containing software version **4.14A** contains a file named **0414A.txt**.

Navigating Through the Setup Utility

The Setup Utility provides access to parameter settings through menus and submenus. Each menu contains a list and a highlight. Highlight one of the choices listed. Press **ENTER** to activate the highlighted choice. Each menu provides access to parameter settings or another menu.

Press **ENTER** to toggle settings **On** or **Off**. Type a specific value where required. Press **ENTER** or **Exit** (**F10**) to save settings when prompted by the software. Press **Exit** (**F10**) to close a menu and return to the previous menu.

Refer to "Section 6 - Setup Utility Maps" for all maps referenced in "Sections 1 – 4." Use these maps to locate parameter settings. The maps also serve as a quick reference guide.

NOTE: All dimensions, numbers, assigned values, and defaults provided in this manual are subject to change without notice depending upon individual manufacturing considerations and industry standards.

Default Settings

The Setup Utility has default settings pre-loaded in the configuration file. These settings remain active unless you change them. In this manual, default settings are specified as: [Default: **Setting**].

Keypad Keys

In this manual, the names **ARROWS**, **CLEAR**, **SHIFT**, and **SPACE** are used for the corresponding keypad keys. See **Table 1-2** for their identifying key faces.

Additionally, the alphanumeric characters, (**A** – **Z**) and (**0** – **9**), are used to reference corresponding alphanumeric keys.

Table 1-2, Keypad Keys

Name	Key Face
ARROWS	
CLEAR	
SHIFT	
SPACE	

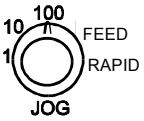
Axis Keys

Some parameters require that you specify an axis. Use the **X**, **Y**, **Z**, or **U** key to specify the axis.

Console Switches/Manual Panel Keys

Console switches and Manual Panel Keys are referred to as shown in **Table 1-3**.

Table 1-3, Console Switches/ Manual Panel Keys

Name	Switch/Key
Axis Selector Switch	
Jog Selector Switch	
Feedrate OVERRIDE Switch	
Spindle OVERRIDE Switch	
E-Stop Key	
Jog Plus Key	
Jog Minus Key	
Servo Reset Key	
Start Key	
Hold Key	

(Continued…)

Table 1-3, Console Switches/ Manual Panel Keys (Continued)

Name	Switch/Key
Spindle Reverse Key	SPDL REV
Spindle Forward Key	SPDL FWD
Spindle Off Key	SPDL OFF

ENTER Key

Press **ENTER** to enter parameters into the system.

Highlighting Menu Options

Press **Up Arrow** (**F3**) and **Down Arrow** (**F4**) to highlight menu selections in the Setup Utility. The corresponding arrow keys can also be used.

Exiting a Screen

Press **Exit** (**F10**) to return to the previous screen.

Password Restricted Parameters

Some machine parameters are protected by passwords. The CNC provides four access levels of passwords. Operators are assigned limited access, which allows them to set parameters used in normal machine operations. Service and factory technicians require a higher level of access. The Programmable I/O Interface requires a separate password. See **Table 1-4** for default machine passwords.

Table 1-4, Default Machine Passwords

Access Level	Password Level
Limited – Operator	159
Service Technician	Z48
Factory Technician	Reserved for factory use
Programmable Logic Controller	IPI

NOTE: Service supersedes Limited. Factory level is the highest and supersedes all, except IPI, which is independent of the other passwords.

Changing Protected Parameters

To change protected parameters, enter a password when the CNC displays the password prompt.

NOTE: You are only required to type a password once during Setup. However, when you exit the Setup Utility and re-enter, you will again be prompted for a password.

Saving Changes to Setup Parameters

When you exit the Setup Utility menu after you have changed any parameters, the CNC displays the prompt "**Save Changes?**".

Select one of the following:

- **Yes** (**F1**) to save the changes.
- **No** (**F2**) to cancel the changes.
- **Cancel** (**F9**) to return to the **Setup Utility Menu**.

NOTE: When **No** (**F2**) is pressed, all parameters revert to the settings prior to changes.

All configuration parameters are saved in a configuration file, (P6MCFG.CFG). Every time a parameter is changed, the configuration file is saved; the CNC automatically creates a backup file, (P6MCFG.BAK). The CNC provides utilities to manage the configuration file. Refer to "Section 4 - Configuration Utilities" for detailed information.

Setting Parameters in Setup Utility

To set parameters in the Setup Utility, do the following:

1. Highlight the menu in which the parameter appears and press **ENTER**.

Change the parameter by following one of the steps mentioned below:

- In some cases a parameter can only have two selections. Pressing **ENTER** changes from one value to the other.
- In some cases, a parameter may have more than two selections and pressing **ENTER** will display a pop-up menu with the list of selections. Highlight the desired selection and press **ENTER**.
- In other cases, the CNC will highlight an entry field and you will be allowed to type the value for the parameter. Type the desired value, or setting, and press **ENTER**.

Using Valid Parameter Ranges

All parameters entered in an entry field must be within the valid range for the parameter. If the value entered is not within the valid range, an error message is displayed. The error message shows the valid range for the parameter. Pressing **F10** or **CLEAR** can clear the error message. Once the error message is cleared, you can enter another value. The previous value can be restored by pressing **UP ARROW** and then **ENTER**.

Accessing Setup Utility

To access the Setup Utility menus, do the following:

1. Turn on the CNC.

   When the CNC is turned on, the CNC software starts automatically. The CNC displays messages to indicate the status of the startup. When the CNC software has successfully started, the CNC displays ANILAM Company information and the software version number.

2. Press (**F10**) to continue.

   The CNC displays the **Software Options** screen.

3. Use the **ARROW** keys to highlight **Setup Utility**. Press **ENTER**.

If already in Manual mode, access the Software Options screen by pressing **SHIFT** + **F10**. The servos must be off or the CNC will not allow you to exit Manual mode.

In either case, the CNC displays the **Setup Options** Menu. Refer to Map 1**, Menu A**. This menu allows you to access the setup parameters.

Units of Measurement

The Units of Measurement parameter specifies the units used to enter dimensional data. If you are using mixed data, input data in one format (inch or mm) first. Change the format (inch or mm) and enter the rest of the data. You can change the units as many times as you need to. By using the proper units you do not need to convert values, but can enter data precisely (that is, no rounding during conversion). See **MC_1002: Default units**. [Default: **Inch**]

All dimensional data will be displayed according to the units specified in this parameter.

6000M-4X

The only exception to this rule is the dimensional parameter corresponding to rotary axes. If the auxiliary axis (that is, U) is configured as a rotary axis, then the unit is always in degrees or degrees per minute (that is, deg/min).

Section 2 - Machine Constants

The Machine Constants configures the settings for the CNC.

Machine Constants Group Assignments

Refer to **Table 2-1** for range assignments.

6000M-4X

The Setup Utility displays Machine Constants for axes X, Y, Z, and U. Machine Constants for the U-axis need to be set when the U-axis is used in a 6000M-4X.

Table 2-1, Machine Constant Group Assignments

Setup Parameter Group	MC Range
Control Software	MC_1000 – MC_1099
Draw Mode	MC_1100 – MC_1199
User Definable Variables	MC_1120 – MC_1149
Tool Probe Variables	MC_1150 – MC_1199
Editor Mode	MC_1200 – MC_1299
Program Directory	MC_1300 – MC_1349
RS-232 Communication	MC_1350 – MC_1374
Printer	MC_1375 – MC_1399
X-axis Setup	MC_2000 – MC_2099
Y-axis Setup	MC_2100 – MC_2199
Z-axis Setup	MC_2200 – MC_2299
U-axis Setup	MC_2300 – MC_2399
Spindle axis Setup	MC_2900 – MC_2999
Linear Correction Compensation	MC_3000 – MC_3014
Skew Error Compensation	MC_3015 – MC_3029
Backlash Compensation	MC_3030 – MC_3049
Ballscrew Compensation	MC_3050 – MC_3099
Software Limits	MC_4000 – MC_4019
Continuous Path	MC_4020 – MC_4029
Position Error Check (PEC)	MC_4030 – MC_4049
Jog Return Position	MC_4050 – MC_4059
Direct Numeric Control (DNC)	MC_4060 – MC_4065
Handwheel	MC_4100 – MC_4149
Home	MC_4200 – MC_4249
Miscellaneous	MC_4300 – MC_4399
M-Code Macro Call	MC_4400 – MC_4419
Tool Management	MC_5000 – MC_5099
Interface	MC_5100 – MC_5149
More Parameters	MC_5200 – MC_5299

Machine Constants Setup

Refer to **Table 2-2** for the parameter descriptions and setting information. The table has subheadings to help you identify the parameters; these subheadings do not display in the software or the Off-line. The default value in **Table 2-2** is bold where there are multiple selections available.

NOTE: Press **ENTER** to toggle the available settings.

Table 2-2, Machine Constants Setup

Machine Constant Parameter	Function	Settings
Control Software Setup Parameters		
MC_1000: **Default axis display**	Switches the default axis display between large and normal.	Large - Configures the axis display to show enlarged X, Y, Z, and U Program position display only. **Normal** - Configures the axis display to show Machine, Program, Target, and Distance To Go displays. [Default]
MC_1001: **Default plane**	A plane defines movement along two axes, excluding a third. Thus, planar movement is two-dimensional. Circular moves and tool diameter compensation are confined to the plane chosen by the user. (Linear moves can occur in all three axes simultaneously.).	**XY** - (top view) displays program in X and Y. [Default] XZ - displays program in X and Z. YZ - displays program in Y and Z.
MC_1002: **Default units**	Switches the default measurement units (Inch/MM Modes).	**Inch** – Activates Inch Mode as default. [Default] MM – Activates MM (millimeter) Mode as default.
MC_1003: **Default axis values**	Switches Absolute/ Incremental default mode (determines how axis values for arcs, lines, and other moves are measured).	**Absolute** – Makes every move in reference to an Absolute Zero position (Program Zero or Part Zero). [Default] Incremental – Makes each move in reference to the last programmed endpoint.

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_1004: **Circle adjustments**	Specifies whether circle centers or endpoints will be adjusted. Circle centers require adjustment when the CNC encounters incorrect circle center or end-point coordinates.	Center - Adjusts the position of the circle center when the CNC encounters incorrect coordinates for either a circle center or endpoint. **End-Point** - Adjusts the position of the circle endpoint when the CNC encounters incorrect coordinates for either a circle center or end-point. [Default]
MC_1005: **Circle centers**	Switches the default mode for programmed circle center coordinates.	Absolute - CNC interprets programmed circle center coordinates as Absolute values. **Incremental** - CNC interprets programmed circle center coordinates as Incremental values. [Default] Modal - CNC interprets programmed circle center coordinates based on current Incremental or Absolute setting.
MC_1006: **Maximum arc correction**	Specifies the maximum amount of correction the CNC will apply to an arc block before declaring an error.	Range (0.000000–1.000000) 0.005000 [Default]
MC_1007: **Internal accuracy**	Maximum accuracy available (system resolution).	Range (0.00000001–0.00100000) 0.00000100 [Default]
MC_1008: **External accuracy**	Specifies the maximum system accuracy obtainable on a given machine (machine resolution).	Range (0.00000001–0.00100000) 0.00010000 [Default]
MC_1009: **Compensation cutoff angle**	Minimizes wasted travel on acute angle. Refer to Figure 2-1, Compensation Cutoff Angle.	Range (1.0–90.0) 15.0 (degrees) [Default]
MC_1011: **User macro file**	Specifies macro filename created by user.	USERCANN.G [Default]
MC_1012: **Load user macro file**	Specifies whether to load user macro at system startup.	**No** - Does not automatically load user macro at startup. [Default] Yes - Automatically loads user macro at startup.

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_1013: **Disk access marker**	Activates/deactivates the Disk access marker.	**On** - Activates the Disk access marker. When the CNC is reading/writing information from/to a disk the Disk access marker is displayed in the upper-left corner of the screen. The Disk access marker looks like a small arrow. [Default] Off - Deactivates Disk access marker.
MC_1014: **Max. memory allocated (in MB-bytes)**	Used only with off-line software. Limits the amount of memory available to the software. This parameter is used to limit the amount of memory available in multitasking environments that provide virtual memory.	Range (2–128) 4 (MB) [Default]
MC_1015: **Force simulation mode**	In Simulation Mode, the CNC does not generate motor and I/O signals. The CNC starts in Simulation Mode. Moves can be commanded and displayed, but no actual machine movements occur.	**Yes** - Enable [Default] No - Disable
MC_1016: **Screen blanking delay (minutes**)	Specifies the screen blanking delay period, in minutes. The delay will be the time between a detected screen idle condition and the activation of the screen saver. To reactivate, press any key.	Range (0–20160) 5 (minutes) [Default]
MC_1032: **Enable radius compensation error checking**	Activates the tool radius compensation error checking. The error checking is designed to eliminate simple gouges caused by overcompensation.	Yes - Enable **No** – Disable [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
Draw Mode Setup Parameters		
MC_1100: **Restore Draw to previous session**	Sets the CNC to re-activate the last active session when you re-enter Draw.	**Yes** - CNC re-activates last session when Draw activated. [Default] No - CNC ignores parameter.
MC_1101: **Default program block mode**	Sets default mode in Draw.	**Auto** [Default] S.Step Motion
MC_1102: **Display program text**	Determines whether program text is displayed in Draw Mode.	**Yes** - Shows program text. [Default] No - Does not show program text.
MC_1103: **Grid**	Activates/deactivates grid as a dotted or solid line.	**None** - Deactivates grid. [Default] Solid - Activates solid line grid. Dotted - Activates dotted line grid.
MC_1104: **Grid size**	Determines the size of the grid (in the active Inch or MM Mode). **NOTE:** The CNC converts the set grid value if the measurement unit is changed. For example: if the Grid Size is set for 1 in Inch Mode and you switch to MM Mode, the CNC changes the Grid Size to 25.4 mm (equal to 1 inch).	Range (0.0–25,400.0) 1.0 [Default] (If the CNC is in Inch Mode, each square in the grid will be one square inch in size for this setting.)
MC_1105: **Tool display**	Turns the tool display On and Off.	**On** - The tool (as defined by the Tool Location Code and Radius in the Tool Page) will be displayed as it cuts the workpiece. [Default] Off - No tool is displayed.
MC_1106: **Default tool type**	Determines shape of displayed tool.	None - No tool shown **Flat** - Flat-end tool shown [Default] Ball - Ball-end tool shown

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_1107: **Cutter compensation in Draw**	Activates/deactivates cutter compensation in Draw Modes.	Ignore - CNC will not show compensated moves (if any) used in the program. Use - CNC shows compensated and non-compensated programmed moves. **Both** - CNC runs the program twice. First, the program is run without compensated moves. Second, the program is run showing compensated moves. This provides a comparison of the two paths to determine programming errors related to compensation. [Default]
MC_1108: **Draw view**	Determines perspective of Draw view.	**XY** - (top view) displays program in X and Y. [Default] XZ - displays program in X and Z. YZ - displays program in Y and Z. ISO - displays program in X, Y, and Z.
MC_1109: **Aspect ratio correction factor**	Corrects circularity problems with display of circles and drawings in general. In cases where a circle may look distorted (that is, like an egg), this parameter can be used to make the circle look like a TRUE circle. Increasing the number will make the circle taller and skinnier; decreasing the number will make the circle shorter and fatter.	Range (0.01–10.00) 1.47 [Default]
MC_1110: **Save/Restore Draw image when using Edit**	Saves draw image when user switches to Edit Mode. In Draw Mode, when the **Edit** (**F2**) soft key is pressed, the CNC switches to Edit Mode. The user later re-enters the Draw Mode when you exit Edit Mode. If this option is enabled (Yes), the CNC restores the image on the screen prior to entering Edit. This image will correspond to the part program drawing.	**Yes** - Saves draw image. [Default] No - Does not save draw image.

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
User Definable Variables Setup Parameters		
MC_1120: **User definable variable #1120**	Variable defined by user to be used in general CNC programming. Integer value: 0 to 99999	Range (-99999–99999) 0 [Default]
MC_1121: **User definable variable #1121**	Variable defined by user to be used in general CNC programming. Integer value: 0 to 99999	Range (-99999–99999) 0 [Default]
MC_1122: **User definable variable #1122**	Variable defined by user to be used in general CNC programming. Integer value: 0 to 99999	Range (-99999–99999) 0 [Default]
MC_1123: **User definable variable #1123**	Variable defined by user to be used in general CNC programming. Integer value: 0 to 99999	Range (-99999–99999) 0 [Default]
MC_1124: **User definable variable #1124**	Variable defined by user to be used in general CNC programming. Integer value: 0 to 99999	Range (-99999–99999) 0 [Default]
MC_1125: **User definable variable #1125**	Variable defined by user to be used in general CNC programming. Integer value: 0 to 99999	Range (-99999–99999) 0 [Default]
MC_1126: **User definable variable #1126**	Variable defined by user to be used in general CNC programming. Integer value: 0 to 99999	Range (-99999–99999) 0 [Default]
MC_1127: **User definable variable #1127**	Variable defined by user to be used in general CNC programming. Integer value: 0 to 99999	Range (-99999–99999) 0 [Default]
MC_1128: **User definable variable #1128**	Variable defined by user to be used in general CNC programming. Integer value: 0 to 99999	Range (-99999–99999) 0 [Default]
MC_1129: **User definable variable #1129**	Variable defined by user to be used in general CNC programming. Integer value: 0 to 99999	Range (-99999–99999) 0 [Default]
MC_1130: **User definable variable #1130**	Variable defined by user to be used in general CNC programming. Unit based (Inch or MM).	Range (-25000.0000–25000.0000) 0.0000 [Default]
MC_1131: **User definable variable #1131**	Variable defined by user to be used in general CNC programming. Unit based (Inch or MM).	Range (-25000.0000–25000.0000) 0.0000 [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_1132: **User definable variable #1132**	Variable defined by user to be used in general CNC programming. Unit based (Inch or MM).	Range (-25000.0000–25000.0000) 0.0000 [Default]
MC_1133: **User definable variable #1133**	Variable defined by user to be used in general CNC programming. Unit based (Inch or MM).	Range (-25000.0000–25000.0000) 0.0000 [Default]
MC_1134: **User definable variable #1134**	Variable defined by user to be used in general CNC programming. Unit based (Inch or MM).	Range (-25000.0000–25000.0000) 0.0000 [Default]
MC_1135: **User definable variable #1135**	Variable defined by user to be used in general CNC programming. Unit based (Inch or MM).	Range (-25000.0000–25000.0000) 0.0000 [Default]
MC_1136: **User definable variable #1136**	Variable defined by user to be used in general CNC programming. No units assigned.	Range (-99999.0000–99999.0000) 0.0000 [Default]
MC_1137: **User definable variable #1137**	Variable defined by user to be used in general CNC programming. No units assigned.	Range (-99999.0000–99999.0000) 0.0000 [Default]
MC_1138: **User definable variable #1138**	Variable defined by user to be used in general CNC programming. No units assigned.	Range (-99999.0000–99999.0000) 0.0000 [Default]
MC_1139: **User definable variable #1139**	Variable defined by user to be used in general CNC programming. No units assigned.	Range (-99999.0000–99999.0000) 0.0000 [Default]
MC_1140: **User definable variable #1140**	Variable defined by user to be used in general CNC programming. Unit based (Inch or MM).	Range (-25000.0000–25000.0000) 0.0000 [Default]
MC_1141: **User definable variable #1141**	Variable defined by user to be used in general CNC programming. Unit based (Inch or MM).	Range (-25000.0000–25000.0000) 0.0000 [Default]
MC_1142: **User definable variable #1142**	Variable defined by user to be used in general CNC programming. Unit based (Inch or MM).	Range (-25000.0000–25000.0000) 0.0000 [Default]
MC_1143: **User definable variable #1143**	Variable defined by user to be used in general CNC programming. Unit based (Inch or MM).	Range (-25000.0000–25000.0000) 0.0000 [Default]
MC_1144: **User definable variable #1144**	Variable defined by user to be used in general CNC programming. Unit based (Inch or MM).	Range (-25000.0000–25000.0000) 0.0000 [Default]

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_1145: **User definable variable #1145**	Variable defined by user to be used in general CNC programming. Unit based (Inch or MM).	Range (-25000.0000–25000.0000) 0.0000 [Default]
MC_1146: **User definable variable #1146**	Variable defined by user to be used in general CNC programming. Unit based (Inch or MM).	Range (-25000.0000–25000.0000) 0.0000 [Default]
MC_1147: **User definable variable #1147**	Variable defined by user to be used in general CNC programming. Unit based (Inch or MM).	Range (-25000.0000–25000.0000) 0.0000 [Default]
MC_1148: **User definable variable #1148**	Variable defined by user to be used in general CNC programming. Unit based (Inch or MM).	Range (-25000.0000–25000.0000) 0.0000 [Default]
MC_1149: **User definable variable #1149**	Variable defined by user to be used in general CNC programming. Unit based (Inch or MM).	Range (-25000.0000–25000.0000) 0.0000 [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
Tool Probe Variables Setup Parameters		
MC_1150: **3-D probe type**	Transmission type used for the installed 3-D touch probe.	**Corded** [Default] Cordless
MC_1151: **Nominal probe stylus diameter**	The overall nominal probe stylus diameter.	Range (0.0000–51.0000) 0.0000 [Default]
MC_1152: **Maximum stroke from home for first pick**	The distance from machine Z home with the shortest tool or the spindle face to just below the probe stylus top as the maximum stroke for the initial probe pick.	Range (0.0000–999.0000) 0.0000 [Default]
MC_1153: **RPM for calibration and tool measurement**	The spindle spin RPM for tool touch.	Range (100–1000) 0 (rev/min) [Default]
MC_1154: **Probe orientation**	Sets the probe orientation.	Range (-2–2) 1 Probe is pointing to the right as you are facing the machine in the +X direction. -1 Probe is pointing to the left of the machine in the -X direction. **0** [Default] 2 Probe is pointing away from you, toward the back of the machine in the +Y direction. -2 Probe is pointing toward you, toward the front of the machine in the –Y direction.
MC_1155: **Z first pick, FAST feed-rate**	Sets user definable FAST feed-rate.	Range (2.5–2540.0) 0.0 [Default]
MC_1156: **Z first pick, MEDIUM feed-rate**	Sets user definable MEDIUM feed-rate.	Range (2.5–508.0) 0.0 [Default]
MC_1157: **Z first pick, SLOW feed-rate**	Sets user definable SLOW feed-rate.	Range (0.1–254.0) 0.0 [Default]
MC_1158: **Z retract amount**	Sets user definable distance the tool will back away on the Z-axis after it touches the probe.	Range (0.0100–25.400) 0.0000 [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_1159: **XY retract amount**	Sets user definable distance the tool will back away on the X-axis or Y-axis after it touches the probe.	Range (0.0100–25.400) 0.0000 [Default]
MC_1160: **Z rapid to start position from home**	Set the longest tool in the spindle and bring the Z-axis to machine home. With a tape measure, measure the distance from the tool tip to within 0.5" (12.7 mm) above the top of the probe stylus and enter that number. When using G151, this will cause the tool to rapid to this position in the Z-axis before starting the initial probe touch in the Z-axis.	Range (0.0000–999.0000) 0.0000 [Default]
MC_1161: **Diameter of tool probe gauge**	Sets the probe calibration standard diameter.	Range (0.1000–508.0000) 0.0000 [Default]
MC_1162: **Positioning feedrate normally**	Feedrate used for positioning the probe in protected mode. Typical value: 200 inches/minute (IPM).	Range (0.1–25400.0) 0.0 [Default]
MC_1163: **First touch feedrate**	Feedrate used for positioning for the initial pick. Typical value: 50 inches/minute (IPM).	Range (0.1–2540.0) 0.0 [Default]
MC_1164: **Nominal probe stylus ball radius**	Diameter of the probe stylus divided by 2.	Range (0.0100–25.4000) 0.0000 [Default]
MC_1165: **Diameter of spindle probe gauge**	The exact diameter of the ring gauge used for probe calibration.	Range (0.1000–508.0000) 0.0000 [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
Editor Mode Setup Parameters		
MC_1200: **Restore Editor to previous session**	If Yes (enabled), when user exits a program in Edit Mode, the CNC marks the position where the last edit was made. The next time the program is opened, the cursor will be located at that spot.	**Yes** - restores to previous session. [Default] No - does not restore to previous session.
MC_1201: **Show top line**	Determines whether an optional "top line" will be displayed in the Edit Mode. The top line contains the active mode information and first block of the open program.	**Yes** - Displays top line. [Default] No - Does not display top line.
MC_1202: **Default insert mode**	Switches On/Off Default Insert Mode. Insert Mode inserts new text without overwriting existing text.	**On** - Automatically sets Insert Mode as default. [Default] Off - Does not automatically set Insert Mode as default.
MC_1203: **Auto tab to previous line's position**	This option is available only with off-line systems or systems with attached keyboards. When a line is indented, the CNC uses the indented position as the first tab position of the following line. For example, the user indents one line by four spaces and then moves to the beginning of the next line by pressing **ENTER**. When you press **TAB**, the cursor now advances four spaces.	**Yes** - Enables auto tab to previous line's position. [Default] No - Disables auto tab to previous line's position.
MC_1205: **Default tab width**	This option is available only with offline systems or systems with keyboards attached. Sets default tab width. Range is 2 to 16 spaces. When you press **TAB**, the cursor advances by the specified number of spaces.	Range (2–16) 4 (spaces) [Default]
MC_1206: **Create backup program**	A backup program is created when an edit is made. Each time the program is edited, the CNC updates the backup file. The backup program will not contain an edit until a new edit is made.	Yes - Backup program is created and maintained. **No** - No backup programs are created. [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_1208: **Case sensitive Find**	Determines whether the Find feature will search for uppercase and lowercase letters to determine a match.	Yes - Find search parameter looks for words that exactly match the entered word specific to capitalization and style. **No** - Find search parameter looks for the entered word, regardless of capitalization and style. [Default]
MC_1209: **Memory reserved from editor (in K-bytes)**	Specifies the amount of memory that the editor will not allocate (that is, leaving free).	Range (16–32000) 300 (KB) [Default]
Program Directory Setup Parameters		
MC_1300: **Program directory pattern**	Type of programs displayed. Plus sign '+' can be used to specify multiple types.	*.G+*.M [Default]
MC_1301: **Program directory display mode**	Specifies what program information will be displayed in the Program Directory.	**Short** - Filename and extension only [Default] Long - Detailed program information, including file size, etc.
MC_1302: **Program directory sort order**	Specifies the order in which programs are listed in the Program Directory.	Ignore - CNC ignores parameter. **Name** - Alphanumeric order by filename [Default] Extension - Alphanumeric order by file extension Size - By file size Date - By date program was created
MC_1303: **Automatically check disk at startup**	For machines equipped with hard drives, specifies whether and how often CNC will check the hard drive.	Always Daily Weekly **Monthly** [Default] Never
MC_1304: **Delete backup files during optimize**	For machines equipped with hard drives, specifies whether backup files will be deleted during hard drive optimization.	**Yes** - Backup files deleted during optimization process [Default] No - Backup files maintained during optimization process
MC_1305: **Directory for user program**	CNC will store user programs in specified directory.	C:\USER [Default] Enter user directory location

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
RS-232 Communication Setup Parameters		
MC_1350: Port	Selects a communications port or disables. Must enable for remote communications.	COM1, COM2, **Disabled** [Default: Disabled]
MC_1351: Baud	Selects a baud.	110, 150, 300, 600, 1200, 2400, 4800, **9600** [Default], or 19200
MC_1352: Parity	Selects parity.	NONE, ODD, or **EVEN** [Default]
MC_1353: **Data bits**	Enters number of data bits.	**7** [Default] 8
MC_1354: **Stop bits**	Enters number of stop bits.	0 **1** [Default]
MC_1355: **Software**	Refers to Xon or Xoff or "handshaking" (transmission/ receipt of data via RS-232) in communications packages.	**On** - Enables handshaking [Default] Off - Disables handshaking
Printer Setup Parameters		
MC_1375: **Default output device**	Specifies where file will be printed.	PRN [Default] To print to another file, enter drive, path, and filename with extension. If the filename entered is not an existing file, the CNC creates the file and transfers the data to the file. If the filename entered is an existing file, the CNC overwrites the data in the file with the print file data.
MC_1376: **Lines per page**	Number of lines to be printed per page (8.5 X 11").	Range (1–66) 55 [Default] Enter desired value
MC_1377: **Page heading**	Prints a page heading including filename, date and time, and page number.	**Yes** - Prints heading. [Default] No - Does not print heading
MC_1378: **Line numbers**	Prints line numbers on hard copy of file.	Yes - Prints line numbers **No** - Select No if no line numbers are desired [Default]
MC_1381: **Wrap text**	Wraps text to the next line if program is greater than 80 characters.	**Yes** - Wraps text [Default] No - Truncates text

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
X-axis Setup Parameters		
MC_2000: **X Motor Encoder Connector**	The connection to which the motor encoder for the X-axis is connected.	**X15** [Default] X16 X17 X18 X19
MC_2001: **X PWM Output Connector**	Defines the X-axis Pulse with Modulation (PWM) output connector.	X55 **X51** [Default] X52 X53 X54
MC_2002: **X Inverter Type**	The inverter type identifies the X-axis inverter being used. Inputting the wrong inverter type can result in undesired axis behavior or inverter damage.	SA 411A SA 201A SA 301C **SA 411C** [Default] PM 107 PM 115A PM 123A PM 132A PM 148A PM 207 PM 215A PM 223A
MC_2003: **X Motor Type**	The X-axis motor type is identified by the motor name. Inputting the wrong motor number can result in undesired axis behavior or motor damage.	NONE AM 820A AM 820AB AM 1150A AM 1150AB AM 1400C AM 1400CB AM **1400A** [Default] AM 1400AB AM 960A AM 960AB AM 1160A AM 1160AB AM 1160C AM 1160CB AM 1160E AM 1160EB AM 1550C AM 1550CB AM 1550E AM 1550EB AM 1550G AM 1550GB AM 1150C AM 1420C

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2004: **X Linear Encoder Connector**	Selects the measuring system input for the X-axis linear encoder.	**NONE** [Default] X1 X2 X3 X4 X6
MC_2005: **X Linear Encoder Sinewave Period**	Provides the number of encoder periods corresponding to the X-axis displacement as entered in MC_2006. The sine signal of the encoder is interpolated to obtain 1024* the nominal resolution. The input frequency of the encoder to the CNC may not exceed: 350 kHz for an encoder with 1Vpp signal **NOTE:** Both MC_2005 and MC_2006 may be multiplied by the same factor to obtain integer values. Also, division by the same factor is possible as long as the result is an integer. See "Encoder Resolution Examples" for sample calculations.	Range (0–20160) 1 [Default]
MC_2006: **X Linear Encoder µm per Sinewave**	X-axis grating pitch.	Range (1–100) 20 [Default]
MC_2007: **X Linear Encoder Type**	Defines the X-axis encoder type.	**Lin Enc** [Default] EverTrack
MC_2008: **X Linear Encoder Signal Type**	Defines the signal type for the X-axis encoder.	1Vpp [Default]
MC_2009: **X Linear Encoder Phase**	Moving the X-axis in a positive direction should result in a positive count on the axis display. Likewise, moving an axis in a negative direction should result in a negative count on the axis display. If an axis display does not count in the appropriate direction, adjust the Encoder Phase settings to correct the problem. **NOTE:** This is the only way to change the direction of the count without making hardware changes.	**Not invert** [Default] Invert

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2010: **X Ballscrew Pitch**	Pitch is the linear distance traveled per revolution of the X-axis ballscrew. **NOTE:** This parameter applies only to rotary encoders. Do not use if the axis is using a linear encoder for feedback.	Range (0.00000–30.00000) 0.47244 (inch) [Default]
MC_2011: **X Number of Teeth Motor**	Gearing on the X-axis motor side. Set at 1 if there is no gear train.	Range (1–200000) 1 [Default]
MC_2012: **X Number of Teeth Ballscrew**	Gearing on the X-axis spindle side. Set at 1 if there is no gear train.	Range (1–200000) 1 (no gear train) [Default]
MC_2013: **X Motor Encoder Phase**	Invert X-axis motor and encoder count direction.	**Invert** [Default] Not invert
MC_2014: **X DC Bus Voltage**	Standard X-axis value 560 (VDC). This value can be changed is the supply voltage deviates from the standard voltage 3*400 (VAC).	Range (100–800) 560 (VDC) [Default]
MC_2015: **X I2t Guarding**	The square of the actual current is integrated to monitor the actual power. The integration lasts for 10 seconds with feed motors and 150 seconds for main spindle motors. For the limit value, the nominal motor current is used, multiplied by the factor from MC_2015. Standard value is 100%.	Range (0–800) 0 Off [Default]
MC_2016: **X Commutation Offset Speed (rpm)**	The X-axis field angle offset (MC_2017) operates from this speed.	Range (0–40000) 0 (rev/min) [Default]
MC_2017: **X Commutation Offset Angle (deg)**	The X-axis field angle offset is interpolated between the value zero at MC_2016 speed and the MC_2017 value at Nmax (maximum) speed (velocity).	Range (0–360) 0 (degrees) [Default]
MC_2018: **X Velocity Filter**	The X-axis velocity filter is suitable for damping high-frequency spurious oscillations (>600 Hz).	**0** No filter [Default] 1 1st order filter (spurious oscillations less than (<) approximately 700 Hz) 2 2nd order filter (spurious oscillations greater than (>) approximately 700 Hz)

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2020: **X Current Control Gain N<Nom (mV/A)**	The X-axis current control (PI) gain is determined with MC_2020. Both P (Proportional) and I (Integral) components can be determined with just one machine constant.	Range (16–999999) 60000 (mV/A) [Default]
MC_2021: **X Current Control Gain N>Nom (mV/A)**	The X-axis current gain control usually has to be increased for revolutions above Nnom. The gain of MC_2021 is defined at Nmax. When MC_2021 is set at zero, the gain of MC_2020 is applied for the whole speed range. The gain between Nnom and Nmax is increased linearly.	Range (0–999999) 0 (mV/A) [Default]
MC_2022: **X Vel. Control Prop. Gain (mAs/rev)**	The X-axis proportional gain of the velocity control loop is set using the velocity loop gain Kvel. The overall loop gain depends on the machine constant value, the motor torque constant, and the equivalent mass moment of inertia (related to the motor).	Range (0.1–10000.0) 2.0 (mAs/rev) [Default]
MC_2023: **X Vel. Control Integral Timecons (.1ms)**	X-axis velocity control integral time constant in tenths of milliseconds.	Range (0.000–100.000) 0.100 (0.1ms) [Default]
MC_2024: **X Vel. Control Integral Limit (ms)**	If the X-axis “limit cycling” effect occurs during rest, limiting the integral buffer can compensate it. This compensation is switched off when MC_2024 = 0. Realistic input values are between 100–200 milliseconds.	Range (0.00–1000.00) 0.00 (ms) [Default]
MC_2025: **X Vel. Control Diff. Gain (.1mAs2/rev)**	Normally the X-axis differential gain is not used in the speed controller. The differential gain reduces oscillations in the low frequency range (<200 Hz), but it destabilizes the controller in the higher frequency range. Do not use this constant for machine axis if the motor is coupled to the spindle via a timing belt.	Range (0.00–1000.00) 0.00 ($0.1mAs^2$/rev) [Default]
MC_2026: **X Pos. Control Prop. Gain (1/min)**	Sets the X-axis positional control proportional gain. The positional control gain determines the dynamic servo error for an axis without fast feed. MC_2026 = 2000 [1/min], feed = 2000 [mm/min], the dynamic servo error is feed/MC_2024 = 1 [min].	Range (0.10–100.00) 40.00 (1/min) [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2027: **X Pos. Control Output Limit (rpm)**	Limits the X-axis output of the positional controller and the standard speed value.	Range (0.00–10000.00) 0.00 (rev/min) [Default]
MC_2028: **X Velocity FeedFwd. Gain**	X-axis velocity feed forward gain.	**No** [Default] Yes
MC_2029: **X Acceleration FeedFwd. Gain (.1mAs^2/rev)**	X-axis acceleration feed forward gain. In practice, this value is 2 to 3 times the motor inertia.	Range (0.0–10000.00) 0.0 (0.1mAs^2/rev) [Default]
MC_2030: **X Coulomb Friction FeedFwd. Gain (mA)**	X-axis torque compensation gain for friction at low rotational speed. Produces an offset according to direction of travel.	Range (-10000.0–10000.0) 0.00 (mA) [Default]
MC_2031: **X Torque Offset (mA)**	X-axis constant torque to offset the compensation (for example, for gravitational force in a vertical axis).	Range (-10000.00–10000.00) 0.00 (mA) [Default]
MC_2032: **X Friction FeedFwd. Timecons (.1ms)**	X-axis delay of the friction compensation to prevent overcompensating when changes in direction occur at high speeds. Typical value: 150 (0.1 msec).	Range (0.00–10000.00) 0.00 (0.1 ms) [Default]
MC_2033: **X Damping FeedFwd. at Nnom (mA)**	X-axis damping compensation at standard speeds. Used for heavy machines.	Range (-10000.00–10000.00) 0.00 (mA) [Default]
MC_2034: **X Torque LP Filter Timecons (.1ms)**	X-axis torque lowpass filter time constant is used when there is insufficient dumping of the axis. Standard value is zero. Realistic input values 3–20 (0.1msec)	Range (0.00–10000.00) 0.00 (0.1ms) [Default]
MC_2035: **X Torque Notch Filter Freq. (.1Hz)**	Vibrations can occur on critical axes and at the spindle in a frequency range, which cannot be compensated either with the differential factor (MC_2025) or with the MC_2034.	Range (0.00–10000.00) 0.00 (0.1Hz) [Default]
MC_2036: **X Torque Notch Filter Damp. (.1dB**)	X-axis damping values of the torque band-stop filter. Damping should not be set unnecessarily high which would restrict the operation of the control loop. Realistic input values are 30–90 (0.1dB).	Range (0.00–10000.00) 0.00 (0.1dB) [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2050: **X Display Resolution**	X-axis – See "Setting the Display Resolution."	.5 Micron **1 Micron** [Default] 2 Micron 5 Micron 10 Micron
MC_2051: **X In-position Tolerance Range**	X-axis –See "Setting In-Position Tolerance" to determine in-position range.	Range (0.0000–9.0000) 0.0004 (inch) [Default]
MC_2052: **X Default Feed Rate**	Setting the X-axis Default Feed Rate establishes a default feedrate for the X-axis, wherever a feedrate has not been programmed. This applies to programmed blocks or MDI commands. Jog moves in feed (that is, from a manual panel) can have a different feedrate.	Range (1.–50800.) 10. (inch/min) [Default]
MC_2053: **X Default Rapid Rate**	X-axis – See "Setting Default Rapid Rate."	Range (1.–50800.) 500. (inch/min) [Default]
Y-axis Setup Parameters		
MC_2100: **Y Motor Encoder Connector**	The connection to which the motor encoder for the Y-axis is connected.	X15 **X16** [Default] X17 X18 X19
MC_2101: **Y PWM Output Connector**	Defines the Y-axis PWM output connector.	X55 X51 **X52** [Default] X53 X54
MC_2102: **Y Inverter Type**	The inverter type identifies the Y-axis inverter being used. Inputting the wrong inverter type can result in undesired axis behavior or inverter damage.	**COMPACT** [Default] PM 107 PM 115A PM 123A PM 132A PM 148A PM 207 PM 215A PM 223A

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2103: **Y Motor Type**	The Y-axis motor type is identified by the motor name. Inputting the wrong motor number can result in undesired axis behavior or motor damage.	**AM 1400A** [Default] (See MC_2003 for a complete setting list)
MC_2104: **Y Linear Encoder Connector**	Selects the measuring system input for the Y-axis linear encoder connector.	**NONE** [Default] X1 X2 X3 X4 X6
MC_2105: **Y Linear Encoder Sinewave Period**	Provides the number of encoder periods corresponding to the Y-axis displacement as entered in MC_2106. The sine signal of the encoder is interpolated to obtain 1024* the nominal resolution. The input frequency of the encoder to the CNC may not exceed: 350 kHz for an encoder with 1Vpp signal **NOTE:** Both MC_2105 and MC_2106 may be multiplied by the same factor to obtain integer values. Also, division by the same factor is possible as long as the result is an integer. See "Encoder Resolution Examples" for sample calculations.	Range (0–20160) 1 [Default]
MC_2106: **Y Linear Encoder µm per Sinewave**	Y-axis grating pitch.	Range (1–100) 20 [Default]
MC_2107: **Y Linear Encoder Type**	Defines the Y-axis encoder type.	**Lin Enc** [Default] EverTrack
MC_2108: **Y Linear Encoder Signal Type**	Defines the signal type for the Y-axis encoder.	1Vpp [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2109: **Y Linear Encoder Phase**	Moving the Y-axis in a positive direction should result in a positive count on the axis display. Likewise, moving an axis in a negative direction should result in a negative count on the axis display. If an axis display does not count in the appropriate direction, adjust the Encoder Phase settings to correct the problem. **NOTE:** This is the only way to change the direction of the count without making hardware changes.	**Not invert** [Default] Invert
MC_2110: **Y Ballscrew Pitch**	Pitch is the linear distance traveled per revolution of the Y-axis ballscrew. **NOTE:** This parameter applies only to rotary encoders. Do not use if the axis is using a linear encoder for feedback.	Range (0.00000–30.00000) 0.47244 (inch) [Default]
MC_2111: **Y Number of Teeth Motor**	Gearing on the Y-axis motor side. Set at 1 if there is no gear train.	Range (1–200000) 1 [Default]
MC_2112: **Y Number of Teeth Ballscrew**	Gearing on the Y-axis spindle side. Set at 1 if there is no gear train.	Range (1–200000) 1 [Default]
MC_2113: **Y Motor Encoder Phase**	Invert Y-axis motor and encoder count direction.	**Invert** [Default] Not invert
MC_2114: **Y DC Bus Voltage**	Standard Y-axis value 560 (VDC). This value can be changed is the supply voltage deviates from the standard voltage 3*400 (VAC).	Range (100–800) 560 (VDC) [Default]
MC_2115: **Y I2t Guarding**	The square of the actual current is integrated to monitor the actual power. The integration lasts for 10 seconds with feed motors and 150 seconds for main spindle motors. For the limit value, the nominal motor current is used, multiplied by the factor from MC_2115. Standard value is 100%.	Range (0–800) 0 Off [Default]
MC_2116: **Y Commutation Offset Speed (rpm)**	The Y-axis field angle offset (MC_2117) operates from this speed.	Range (0–40000) 0 (rev/min) [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2117: **Y Commutation Offset Angle (deg)**	The Y-axis field angle offset is interpolated between the value zero at MC_2116 speed and the MC_2117 value at Nmax (maximum) speed (velocity).	Range (0–360) 0 (degrees) [Default]
MC_2118: **Y Velocity Filter**	The Y-axis velocity filter is suitable for damping high-frequency spurious oscillations (>600 Hz).	**0** No filter [Default] 1 1st order filter (spurious oscillations less than (<) approximately 700 Hz) 2 2nd order filter (spurious oscillations greater than (>) approximately 700 Hz)
MC_2120: **Y Current Control Gain N<Nom (mV/A)**	The Y-axis current control (PI) gain is determined with MC_2120. Both P (Proportional) and I (Integral) components can be determined with just one machine constant.	Range (16–999999) 60000 (mV/A) [Default]
MC_2121: **Y Current Control Gain N>Nom (mV/A)**	The Y-axis current gain control usually has to be increased for revolutions above Nnom. The gain of MC_2121 is defined at Nmax. When MC_2121 is set at zero, the gain of MC_2120 is applied for the whole speed range. The gain between Nnom and Nmax is increased linearly	Range (0–999999) 0 (mV/A) [Default]
MC_2122: **Y Vel. Control Prop. Gain (mAs/rev)**	The Y-axis proportional gain of the velocity control loop is set using the velocity loop gain Kvel. The overall loop gain depends on the machine constant value, the motor torque constant, and the equivalent mass moment of inertia (related to the motor).	Range (0.1–10000.0) 2.0 (mAs/rev) [Default]
MC_2123: **Y Vel. Control Integral Timecons (.1ms)**	Y-axis velocity control integral time constant in tenths of milliseconds.	Range (0.000–100.000) 0.100 (0.1ms) [Default]
MC_2124: **Y Vel. Control Integral Limit (ms)**	If the Y-axis “limit cycling” effect occurs during rest, limiting the integral buffer can compensate it. This compensation is switched off when MC_2124 = 0. Realistic input values are between 100–200 milliseconds.	Range (0.00–1000.00) 0.00 (ms) [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2125: **Y Vel. Control Diff. Gain (.1mAs2/rev)**	Normally the Y-axis differential gain is not used in the speed controller. The differential gain reduces oscillations in the low frequency range (<200 Hz), but it destabilizes the controller in the higher frequency range. Do not use this constant for machine axis if the motor is coupled to the spindle via a timing belt.	Range (0.00–1000.00) 0.00 (0.1mAs^2/rev) [Default]
MC_2126: **Y Pos. Control Prop. Gain (1/min)**	Sets the Y-axis positional control proportional gain. The positional control gain determines the dynamic servo error for an axis without fast feed. MC_2126 = 2000 [1/min], feed = 2000 [mm/min], the dynamic servo error is feed/MC_2124 = 1 [min].	Range (0–100000 1/min) 40.00 (1/min) [Default]
MC_2127: **Y Pos. Control Output Limit (rpm)**	Limits the Y-axis output of the positional controller and the standard speed value.	Range (0.00–10000.00) 0.00 (rev/min) [Default]
MC_2128: **Y Velocity FeedFwd. Gain**	Y-axis velocity feed forward gain.	**No** [Default] Yes
MC_2129: **Y Acceleration FeedFwd. Gain (.1mAs^2/rev)**	Y-axis acceleration feed forward gain. In practice, this value is 2 to 3 times the motor inertia.	Range (0.0–10000.00) 0.0 (0.1mAs^2/rev) [Default]
MC_2130: **Y Coulomb Friction FeedFwd. Gain (mA)**	Y-axis torque compensation gain for friction at low rotational speed. Produces an offset according to direction of travel.	Range (-10000.0–10000.0) 0.00 (mA) [Default]
MC_2131: **Y Torque Offset (mA)**	Y-axis constant torque to offset the compensation (for example, for gravitational force in a vertical axis).	Range (-10000.00–10000.00) 0.00 (mA) [Default]
MC_2132: **Y Friction FeedFwd. Timecons (.1ms)**	Y-axis delay of the friction compensation to prevent overcompensating when changes in direction occur at high speeds. Typical value: 150 (0.1 msec).	Range (0.00–10000.00) 0.00 (0.1 ms) [Default]
MC_2133: **Y Damping FeedFwd. at Nnom (mA)**	Y-axis damping compensation at standard speeds. Used for heavy machines.	Range (-10000.00–10000.00) 0.00 (mA) [Default]

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2134: **Y Torque LP Filter Timecons (.1ms)**	Y-axis torque lowpass filter time constant is used when there is insufficient dumping of the axis. Standard value is zero. Realistic input values 3–20 (0.1msec)	Range (0.00–10000.00) 0.00 (0.1ms) [Default]
MC_2135: **Y Torque Notch Filter Freq. (.1Hz)**	Vibrations can occur on critical axes and at the spindle in a frequency range, which cannot be compensated either with the differential factor (MC_2125) or with the MC_2134.	Range (0.00–10000.00) 0.00 (0.1Hz) [Default]
MC_2136: **Y Torque Notch Filter Damp. (.1dB)**	Y-axis damping values of the torque band-stop filter. Damping should not be set unnecessarily high which would restrict the operation of the control loop. Realistic input values are 30–90 (0.1dB).	Range (0.00–10000.00) 0.00 (0.1dB) [Default]
MC_2150: Y Display Resolution	Y-axis – See "Setting the Display Resolution."	.5 Micron **1 Micron** [Default] 2 Micron 5 Micron 10 Micron
MC_2151: **Y In-position Tolerance Range**	Y-axis –See "Setting In-Position Tolerance" to determine in-position range.	Range (0.0000–9.0000) 0.0004 (inch) [Default]
MC_2152: **Y Default Feed Rate**	Setting the Y-axis Default Feed Rate establishes a default feedrate for the Y-axis, wherever a feedrate has not been programmed. This applies to programmed blocks or MDI commands. Jog moves in feed (that is, from a manual panel) can have a different feedrate.	Range (1.–50800.) 10. (inch/min) [Default]
MC_2153: **Y Default Rapid Rate**	Y-axis –See "Setting Default Rapid Rate."	Range (1.–50800.) 500. (inch/min) [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
Z-axis Setup Parameters		
MC_2200: **Z Motor Encoder Connector**	The connection to which the motor encoder for the Z-axis is connected.	X15 X16 **X17** [Default] X18 X19
MC_2201: **Z PWM Output Connector**	Defines the Z-axis PWM output drive.	X55 X51 X52 **X53** [Default] X54
MC_2202: **Z Inverter Type**	The inverter type identifies the Z-axis inverter being used. Inputting the wrong inverter type can result in undesired axis behavior or inverter damage.	**COMPACT** [Default] PM 107 PM 115A PM 123A PM 132A PM 148A PM 207 PM 215A PM 223A
MC_2203: **Z Motor Type**	The Z-axis motor type is identified by the motor name. Inputting the wrong motor number can result in undesired axis behavior or motor damage.	**AM 1400AB** [Default] (See MC_2003 for a complete setting list)
MC_2204: **Z Linear Encoder Connector**	Selects the measuring system input for the Z-axis linear encoder.	**NONE** (none) [Default] X1 X2 X3 X4 X6

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2205: **Z Linear Encoder Sinewave Period**	Provides the number of encoder periods corresponding to the Z-axis displacement as entered in MC_2206. The sine signal of the encoder is interpolated to obtain 1024* the nominal resolution. The input frequency of the encoder to the CNC may not exceed: 350 kHz for an encoder with 1Vpp signal **NOTE:** Both MC_2205 and MC_2206 may be multiplied by the same factor to obtain integer values. Also, division by the same factor is possible as long as the result is an integer. See "Encoder Resolution Examples" for sample calculations.	Range (0–20160) 1 [Default]
MC_2206: **Z Linear Encoder µm per Sinewave**	Z-axis grating pitch.	Range (1–100) 20 [Default]
MC_2207: **Z Linear Encoder Type**	Defines the Z-axis encoder type.	**Lin Enc** [Default] EverTrack
MC_2208: **Z Linear Encoder Signal Type**	Defines the signal type for the Z-axis encoder.	1Vpp [Default]
MC_2209: **Z Linear Encoder Phase**	Moving the Z-axis in a positive direction should result in a positive count on the axis display. Likewise, moving an axis in a negative direction should result in a negative count on the axis display. If an axis display does not count in the appropriate direction, adjust the Encoder Phase settings to correct the problem. **NOTE:** This is the only way to change the direction of the count without making hardware changes.	**Not invert** [Default] Invert
MC_2210: **Z Ballscrew Pitch**	Pitch is the linear distance traveled per revolution of the Z-axis ballscrew. **NOTE:** This parameter applies only to rotary encoders. Do not use if the axis is using a linear encoder for feedback.	Range (0.00000–30.00000) 0.47244 (inch) [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2211: **Z Number of Teeth Motor**	Gearing on the Z-axis motor side. Set at 1 if there is no gear train.	Range (1–200000) 1 [Default]
MC_2212: **Z Number of Teeth Ballscrew**	Gearing on the Z-axis spindle side. Set at 1 if there is no gear train.	Range (1–200000) 1 [Default]
MC_2213: **Z Motor Encoder Phase**	Invert Z-axis motor and encoder count direction.	**Invert** [Default] Not invert
MC_2214: **Z DC Bus Voltage**	Standard Z-axis value 560 (VDC). This value can be changed is the supply voltage deviates from the standard voltage 3*400 (VAC).	Range (100–800) 560 (VDC) [Default]
MC_2215: **Z I2t Guarding**	The square of the actual current is integrated to monitor the actual power. The integration lasts for 10 seconds with feed motors and 150 seconds for main spindle motors. For the limit value, the nominal motor current is used, multiplied by the factor from MC_2215. Standard value is 100%.	Range (0–800) 0 Off [Default]
MC_2216: **Z Commutation Offset Speed (rpm)**	The Z-axis field angle offset (MC_2217) operates from this speed.	Range (0–40000) 0 (revolutions/minute) [Default]
MC_2217: **Z Commutation Offset Angle (deg)**	The Z-axis field angle offset is interpolated between the value zero at MC_2216 speed and the MC_2217 value at Nmax (maximum) speed (velocity).	Range (0–360) 0 (degrees) [Default]
MC_2218: **Z Velocity Filter**	The Z-axis velocity filter is suitable for damping high-frequency spurious oscillations (>600 Hz).	**0** No filter [Default] 1 1st order filter (spurious oscillations less than (<) approximately 700 Hz) 2 2nd order filter (spurious oscillations greater than (>) approximately 700 Hz)
MC_2220: **Z Current Control Gain N<Nom (mV/A)**	The Z-axis current control (PI) gain is determined with MC_2220. Both P (Proportional) and I (Integral) components can be determined with just one machine constant.	Range (16–999999) 60000 (mV/A) [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2221: **Z Current Control Gain N>Nom (mV/A)**	The Z-axis current gain control usually has to be increased for revolutions above Nnom. The gain of MC_2221 is defined at Nmax. When MC_2221 is set at zero, the gain of MC_2220 is applied for the whole speed range. The gain between Nnom and Nmax is increased linearly	Range (0–999999) 0 (mV/A) [Default]
MC_2222: **Z Vel. Control Prop. Gain (mAs/rev)**	The Z-axis proportional gain of the velocity control loop is set using the velocity loop gain Kvel. The overall loop gain depends on the machine constant value, the motor torque constant, and the equivalent mass moment of inertia (related to the motor).	Range (0.1–10000.0) 2.0 (mAs/rev) [Default]
MC_2223: **Z Vel. Control Integral Timecons (.1ms)**	Z-axis velocity control integral time constant in tenths of milliseconds.	Range (0.000–100.000) 0.100 (0.1ms) [Default]
MC_2224: **Z Vel. Control Integral Limit (ms)**	If the Z-axis "limit cycling" effect occurs during rest, limiting the integral buffer can compensate it. This compensation is switched off when MC_2224 = 0. Realistic input values are between 100–200 milliseconds.	Range (0.00–1000.00) 0.00 (ms) [Default]
MC_2225: **Z Vel. Control Diff. Gain (.1mAs2/rev)**	Normally the Z-axis differential gain is not used in the speed controller. The differential gain reduces oscillations in the low frequency range (<200 Hz), but it destabilizes the controller in the higher frequency range. Do not use this constant for machine axis if the motor is coupled to the spindle via a timing belt.	Range (0.00–1000.00) 0.00 ($0.1mAs^2$/rev) [Default]
MC_2226: **Z Pos. Control Prop. Gain (1/min)**	Sets the Z-axis positional control proportional gain. The positional control gain determines the dynamic servo error for an axis without fast feed. MC_2226 = 2000 [1/min], feed = 2000 [mm/min], the dynamic servo error is feed/MC_2224 = 1 [min].	Range (0–100000 1/min) 40.00 (1/min) [Default]
MC_2227: **Z Pos. Control Output Limit (rpm)**	Limits the Z-axis output of the positional controller and the standard speed value.	Range (0.00–10000.00) 0.00 (rev/min) [Default]
MC_2228: **Z Velocity FeedFwd. Gain**	Z-axis velocity feed forward gain.	**No** [Default] Yes

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2229: **Z Acceleration FeedFwd. Gain (.1mAs^2/rev)**	Z-axis acceleration feed forward gain. In practice, this value is 2 to 3 times the motor inertia.	Range (0.0–10000.00) 0.0 (0.1mAs^2/rev) [Default]
MC_2230: **Z Coulomb Friction FeedFwd. Gain (mA)**	Z-axis torque compensation gain for friction at low rotational speed. Produces an offset according to direction of travel.	Range (-10000.0–10000.0) 0.00 (mA) [Default]
MC_2231: **Z Torque Offset (mA)**	Z-axis constant torque to offset the compensation (for example, for gravitational force in a vertical axis).	Range (-10000.00–10000.00) 0.00 (mA) [Default]
MC_2232: **Z Friction FeedFwd. Timecons (.1ms)**	Z-axis delay of the friction compensation to prevent overcompensating when changes in direction occur at high speeds. Typical value: 150 (0.1 msec).	Range (0.00–10000.00) 0.00 (0.1 ms) [Default]
MC_2233: **Z Damping FeedFwd. at Nnom (mA)**	Z-axis damping compensation at standard speeds. Used for heavy machines.	Range (-10000.00–10000.00) 0.00 (mA) [Default]
MC_2234: **Z Torque LP Filter Timecons (.1ms)**	Z-axis torque lowpass filter time constant is used when there is insufficient dumping of the axis. Standard value is zero. Realistic input values 3–20 (0.1msec)	Range (0.00–10000.00) 0.00 (0.1ms) [Default]
MC_2235: **Z Torque Notch Filter Freq. (.1Hz)**	Vibrations can occur on critical axes and at the spindle in a frequency range, which cannot be compensated either with the differential factor (MC_2225) or with the MC_2234.	Range (0.00–10000.00) 0.00 (0.1Hz) [Default]
MC_2236: **Z Torque Notch Filter Damp. (.1dB)**	Z-axis damping values of the torque band-stop filter. Damping should not be set unnecessarily high which would restrict the operation of the control loop. Realistic input values are 30–90 (0.1dB).	Range (0.00–10000.00) 0.00 (0.1dB) [Default]
MC_2237: **G33 Gain Table Enable**	Enables/disables a special gain table for a rigid tapping cycle.	Yes (enable) **No** (disable) [Default]
MC_2238: **G33 Vel. Control Prop. Gain (mAs/rev)**	Velocity controller proportional gain for G33.	Range (1.0–10000.0) 2.0 (mAs/rev) [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2239: **G33 Vel. Control Integral Timecons (.1ms)**	Velocity controller integral time constant for G33.	Range (0.000–100.000) 0.100 (0.1 ms) [Default]
MC_2240: **G33 Pos. Control Prop. Gain (1/min)**	Position controller proportional gain for G33.	Range (0.10–100.00) 40.00 (1/min) [Default]
MC_2241: **G33 Acceleration FeedFwd. Gain (.1mAs2/rev)**	Acceleration feed forward gain for G33.	Range (0.0–1000.0) 0.0 (0.1mAs^2/rev) [Default]
MC_2242: **G33 Colomb Friction FeedFwd. Gain (mA)**	Colomb friction feed forward gain for G33.	Range (-1000.00–10000.00) 0.00 (mA) [Default]
MC_2250: **Z Display Resolution**	Z-axis – See "Setting the Display Resolution."	.5 Micron **1 Micron** 2 Micron [Default] 5 Micron 10 Micron
MC_2251: **Z In-position Tolerance Range**	Z-axis –See "Setting In-Position Tolerance" to determine in-position range.	Range (0.0000–9.0000) 0.0004 (inch) [Default]
MC_2252: **Z Default Feed Rate**	Setting the Z-axis Default Feed Rate establishes a default feedrate for the Z-axis, wherever a feedrate has not been programmed. This applies to programmed blocks or MDI commands. Jog moves in feed (that is, from a manual panel) can have a different feedrate.	Range (1.–50800.) 10. (inch/min) [Default]
MC_2253: **Z Default Rapid Rate**	Z-axis – See "Setting Default Rapid Rate."	Range (1.–50800.) 500. (inch/min) [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
U-axis Setup Parameters		
MC_2300: **U Motor Encoder Connector**	The connection to which the motor encoder for the U-axis is connected.	X15 X16 X17 **X18** [Default] X19
MC_2301: **U PWM Output Connector**	Defines the U-axis PWM output connection.	X55 X51 X52 X53 **X54** [Default]
MC_2302: **U Inverter Type**	The inverter type identifies the U-axis inverter being used. Inputting the wrong inverter type can result in undesired axis behavior or inverter damage.	**COMPACT** [Default] PM 107 PM 115A PM 123A PM 132A PM 148A PM 207 PM 215A PM 223A
MC_2303: **U Motor Type**	The U-axis motor type is identified by the motor name. Inputting the wrong motor number can result in undesired axis behavior or motor damage.	**NONE** [Default] (See MC_2003 for a complete setting list)
MC_2304: **U Linear Encoder Connector**	Selects the measuring system input for the U-axis linear encoder.	**NONE** [Default] X1 X2 X3 X4 X6

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2305: **U Linear Encoder Sinewave Period**	Provides the number of encoder periods corresponding to the U-axis displacement as entered in MC_2306. The sine signal of the encoder is interpolated to obtain 1024* the nominal resolution. The input frequency of the encoder to the CNC may not exceed: 350 kHz for an encoder with 1Vpp signal **NOTE:** Both MC_2305 and MC_2306 may be multiplied by the same factor to obtain integer values. Also, division by the same factor is possible as long as the result is an integer. See "Encoder Resolution Examples" for sample calculations.	Range (0–20160) 1 [Default]
MC_2306: **U Linear Encoder µm per Sinewave**	U-axis grating pitch.	Range (1–100) 20 [Default]
MC_2307: **U Linear Encoder Type**	Defines the U-axis encoder type.	**Lin Enc** [Default] EverTrack
MC_2308: **U Linear Encoder Signal Type**	Defines the signal type for the U-axis encoder.	1Vpp [Default]
MC_2309: **U Linear Encoder Phase**	Moving the U-axis in a positive direction should result in a positive count on the axis display. Likewise, moving an axis in a negative direction should result in a negative count on the axis display. If an axis display does not count in the appropriate direction, adjust the Encoder Phase settings to correct the problem. **NOTE:** This is the only way to change the direction of the count without making hardware changes.	**Not invert** [Default] Invert
MC_2310: **U Ballscrew Pitch**	Pitch is the linear distance traveled per revolution of the U-axis ballscrew. **NOTE:** This parameter applies only to rotary encoders. Do not use if the axis is using a linear encoder for feedback.	Range (0.00000–30.00000) 0.47244 (inch) [Default]

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2311: **U Number of Teeth Motor**	Gearing on the U-axis motor side. Set at 1 if there is no gear train.	Range (1–200000) 1 [Default]
MC_2312: **U Number of Teeth Ballscrew**	Gearing on the U-axis spindle side. Set at 1 if there is no gear train.	Range (1–200000) 1 [Default]
MC_2313: **U Motor Encoder Phase**	Invert U-axis motor and encoder count direction.	**Invert** [Default] Not invert
MC_2314: **U DC Bus Voltage**	Standard U-axis value 560 (VDC). This value can be changed is the supply voltage deviates from the standard voltage 3*400 (VAC).	Range (100–800) 560 (VDC) [Default]
MC_2315: **U I2t Guarding**	The square of the actual current is integrated to monitor the actual power. The integration lasts for 10 seconds with feed motors and 150 seconds for main spindle motors. For the limit value, the nominal motor current is used, multiplied by the factor from MC_2315. Standard value is 100%.	Range (0–800) 0 Off [Default]
MC_2316: **U Commutation Offset Speed (rpm)**	The U-axis field angle offset (MC_2317) operates from this speed.	Range (0–40000) 0 (revolutions/minute) [Default]
MC_2317: **U Commutation Offset Angle (deg)**	The U-axis field angle offset is interpolated between the value zero at MC_2316 speed and the MC_2317 value at Nmax (maximum) speed (velocity).	Range (0–360) 0 (degrees) [Default]
MC_2318: **U Velocity Filter**	The U-axis velocity filter is suitable for damping high-frequency spurious oscillations (>600 Hz).	**0** No filter [Default] 1 1st order filter (spurious oscillations less than (<) approximately 700 Hz) 2 2nd order filter (spurious oscillations greater than (>) approximately 700 Hz)
MC_2320: **U Current Control Gain N<Nom (mV/A)**	The U-axis current control (PI) gain is determined with MC_2320. Both P (Proportional) and I (Integral) components can be determined with just one machine constant.	Range (16–999999) 60000 (mV/A) [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2321: **U Current Control Gain N>Nom (mV/A)**	The U-axis current gain control usually has to be increased for revolutions above Nnom. The gain of MC_2321 is defined at Nmax. When MC_2321 is set at zero, the gain of MC_2320 is applied for the whole speed range. The gain between Nnom and Nmax is increased linearly	Range (0–999999) 0 (mV/A) [Default]
MC_2322: **U Vel. Control Prop. Gain (mAs/rev)**	The U-axis proportional gain of the velocity control loop is set using the velocity loop gain Kvel. The overall loop gain depends on the machine constant value, the motor torque constant, and the equivalent mass moment of inertia (related to the motor).	Range (0.1–10000.0) 2.0 (mAs/rev) [Default]
MC_2323: **U Vel. Control Integral Timecons (.1ms)**	U-axis velocity control integral time constant in tenths of milliseconds.	Range (0.000–100.000) 0.100 (0.1ms) [Default]
MC_2324: **U Vel. Control Integral Limit (ms)**	If the U-axis "limit cycling" effect occurs during rest, limiting the integral buffer can compensate it. This compensation is switched off when MC_2324 = 0. Realistic input values are between 100–200 milliseconds.	Range (0.00–1000.00) 0.00 (ms) [Default]
MC_2325: **U Vel. Control Diff. Gain (.1mAs2/rev)**	Normally the U-axis differential gain is not used in the speed controller. The differential gain reduces oscillations in the low frequency range (<200 Hz), but it destabilizes the controller in the higher frequency range. Do not use this constant for machine axis if the motor is coupled to the spindle via a timing belt.	Range (0.00–1000.00) 0.00 ($0.1mAs^2$/rev) [Default]
MC_2326: **U Pos. Control Prop. Gain (1/min)**	Sets the U-axis positional control proportional gain. The positional control gain determines the dynamic servo error for an axis without fast feed. MC_2326 = 2000 [1/min], feed = 2000 [mm/min], the dynamic servo error is feed/MC_2324 = 1 [min].	Range (0–100000 1/min) 40.00 (1/min) [Default]
MC_2327: **U Pos. Control Output Limit (rpm)**	Limits the U-axis output of the positional controller and the standard speed value.	Range (0.00–10000.00) 0.00 (rev/min) [Default]
MC_2328: **U Velocity FeedFwd. Gain**	U-axis velocity feed forward gain.	**No** [Default] Yes

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2329: **U Acceleration FeedFwd. Gain (.1mAs^2/rev)**	U-axis acceleration feed forward gain. In practice, this value is 2 to 3 times the motor inertia.	Range (0.0–10000.00) 0.0 (0.1mAs^2/rev) [Default]
MC_2330: **U Coulomb Friction FeedFwd. Gain (mA)**	U-axis torque compensation for friction at low rotational speed. Produces an offset according to direction of travel.	Range (-10000.0–10000.0) 0.00 (mA) [Default]
MC_2331: **U Torque Offset (mA)**	U-axis constant torque to offset the compensation (for example, for gravitational force in a vertical axis).	Range (-10000.00–10000.00) 0.00 (mA) [Default]
MC_2332: **U Friction FeedFwd. Timecons (.1ms)**	U-axis delay of the friction compensation to prevent overcompensating when changes in direction occur at high speeds. Typical value: 150 (0.1 msec).	Range (0.00–10000.00) 0.00 (0.1 ms) [Default]
MC_2333: **U Damping FeedFwd. at Nnom (mA)**	U-axis damping compensation at standard speeds. Used for heavy machines.	Range (-10000.00–10000.00) 0.00 (mA) [Default]
MC_2334: **U Torque LP Filter Timecons (.1ms)**	U-axis torque lowpass filter time constant is used when there is insufficient dumping of the axis. Standard value is zero. Realistic input values 3–20 (0.1msec)	Range (0.00–10000.00) 0.00 (0.1ms) [Default]
MC_2335: **U Torque Notch Filter Freq. (.1Hz)**	Vibrations can occur on critical axes and at the spindle in a frequency range, which cannot be compensated either with the differential factor (MC_2325) or with the MC_2334.	0.00 (0.1Hz) [Default]
MC_2336: **U Torque Notch Filter Damp. (.1dB)**	U-axis damping values of the torque band-stop filter. Damping should not be set unnecessarily high which would restrict the operation of the control loop. Realistic input values are 30–90 (0.1dB).	Range (0.00–10000.00) 0.00 (0.1dB) [Default]
MC_2350: **U Display Resolution**	U-axis – See "Setting the Display Resolution."	.5 Micron **1 Micron** [Default] 2 Micron 5 Micron 10 Micron
MC_2351: **U In-position Tolerance Range**	U-axis –See "Setting In-Position Tolerance" to determine in-position range.	Range (0.0000–9.0000) 0.0004 (inch) [Default]

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2352: **U Default Feed Rate**	Setting the U-axis Default Feed Rate establishes a default feedrate for the U-axis, wherever a feedrate has not been programmed. This applies to programmed blocks or MDI commands. Jog moves in feed (that is, from a manual panel) can have a different feedrate.	Range (1.–50800.) 10. (inch/min) [Default]
MC_2353: **U Default Rapid Rate**	U-axis – See "Setting Default Rapid Rate."	Range (1.–50800.) 500. (inch/min) [Default]
MC_2354: **U Axis Type**	Defines the type of U-axis	**Linear** [Default] Rotary
MC_2355: **U Reset Rotary at 360**	When U-axis is rotary, the position display can be automatically reset to 0 (zero) when the axis reaches 360 degrees.	**Yes** [Default] No
MC_2356: **Synchronized to XYZ**	Synchronizes U-axis to XYZ.	**Yes** [Default] No
MC_2357: **U M11/M10 Clamp Enable**	Enables clamping on U-axis using M-codes M11/M10.	**No** [Default] Yes
Spindle-axis Setup Parameters		
MC_2900: **Spindle Motor Encoder Connector**	The connection to which the motor encoder for the spindle-axis is connected.	X15 X16 X17 X18 **X19** [Default]
MC_2901: **Spindle PWM Output Connector**	Defines the spindle-axis PWM output connection.	**X55** [Default] X51 X52 X53 X54
MC_2902: **Spindle Inverter Type**	The inverter type identifies the spindle-axis inverter being used. Inputting the wrong inverter type can result in undesired spindle behavior or inverter damage.	**COMPACT** [Default] PM 107 PM 115A PM 123A PM 132A PM 148A PM 207 PM 215A PM 223A

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2903: **Spindle Motor Type**	The spindle-axis motor type is identified by the motor name. Inputting the wrong motor number can result in undesired axis behavior or motor damage.	None [Default] Analog SM 055A **SM 075A** SM 100A SM 120A SM 055C SM 055D SM 075C SM 075D SM 100C SM 100D
MC_2904: **Spindle Position Encoder Connection**	Selects the measuring system input for the spindle-axis position encoder connection.	**NONE** [Default] X1 X2 X3 X4 X6
MC_2905: **Spindle Position Encoder Sinewave Period**	Provides the number of encoder periods corresponding to the spindle-axis position displacement as entered in MC_2906. The sine signal of the encoder is interpolated to obtain 1024* the nominal resolution. The input frequency of the encoder to the CNC may not exceed: 350 kHz for an encoder with 1Vpp signal **NOTE:** Both MC_2905 and MC_2906 may be multiplied by the same factor to obtain integer values. Also, division by the same factor is possible as long as the result is an integer. See "Encoder Resolution Examples" for sample calculations.	Range (0–20160) 1 [Default]
MC_2906: **Spindle Position Encoder Line Count**	Spindle-axis grating pitch.	Range (1–100) 20 [Default]
MC_2908: **Spindle Position Encoder Signal Type**	Defines the signal type for the spindle-axis encoder.	1Vpp [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2909: **Spindle Position Encoder Phase**	Moving the spindle-axis in a positive direction should result in a positive count on the axis display. Likewise, moving an axis in a negative direction should result in a negative count on the axis display. If an axis display does not count in the appropriate direction, adjust the Encoder Phase settings to correct the problem. **NOTE:** This is the only way to change the direction of the count without making hardware changes.	**Not invert** [Default] Invert
MC_2913: **Spindle Motor Encoder Phase**	Invert spindle-axis motor and encoder count direction.	**Invert** [Default] Not invert
MC_2914: **Spindle DC Bus Voltage**	Standard spindle-axis value 560 (VDC). This value can be changed is the supply voltage deviates from the standard voltage 3*400 (VAC).	Range (100–800) 560 (VDC) [Default]
MC_2915: **Spindle I2t Guarding**	The square of the actual current is integrated to monitor the actual power. The integration lasts for 10 seconds with feed motors and 150 seconds for main spindle motors. For the limit value, the nominal motor current is used, multiplied by the factor from MC_2915. Standard value is 100%.	Range (0–800) 0 Off [Default]
MC_2916: **Spindle Commutation Offset Speed (rpm)**	The U-axis field angle offset (MC_2917) operates from this speed.	Range (0–40000) 0 (rev/min) [Default]
MC_2917: **Spindle Commutation Offset Angle (deg)**	The spindle-axis field angle offset is interpolated between the value zero at MC_2916 speed and the MC_2917 value at Nmax (maximum) speed (velocity).	Range (0–360) 0 (degrees) [Default]
MC_2918: **Spindle Velocity Filter**	The spindle-axis velocity filter is suitable for damping high-frequency spurious oscillations (>600 Hz).	**0** No filter [Default] 1 1st order filter (spurious oscillations less than (<) approximately 700 Hz) 2 2nd order filter (spurious oscillations greater than (>) approximately 700 Hz)

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2920: **Spindle Current Control Gain N<Nom (mV/A)**	The U-axis current control (PI) gain is determined with MC_2920. Both P (Proportional) and I (Integral) components can be determined with just one machine constant.	Range (16–999999) 10000 (mV/A) [Default]
MC_2921: **Spindle Current Control Gain N>Nom (mV/A)**	The spindle-axis current gain control usually has to be increased for revolutions above Nnom. The gain of MC_2921 is defined at Nmax. When MC_2921 is set at zero, the gain of MC_2920 is applied for the whole speed range. The gain between Nnom and Nmax is increased linearly	Range (0–999999) 0 (mV/A) [Default]
MC_2922: **Spindle Vel. Control Prop. Gain (mAs/rev)**	The spindle-axis proportional gain of the velocity control loop is set using the velocity loop gain Kvel. The overall loop gain depends on the machine constant value, the motor torque constant, and the equivalent mass moment of inertia (related to the motor).	Range (0.1–10000.0) 2.0 (mAs/rev) [Default]
MC_2923: **Spindle Vel. Control Integral Timecons (.1ms)**	Spindle-axis velocity control integral time constant in tenths of milliseconds.	Range (0.000–100.000) 0.100 (0.1ms) [Default]
MC_2924: **Spindle Vel. Control Integral Limit (ms)**	If the spindle-axis “limit cycling” effect occurs during rest, limiting the integral buffer can compensate it. This compensation is switched off when MC_2924 = 0. Realistic input values are between 100–200 milliseconds.	Range (0.00–1000.00) 0.00 (ms) [Default]
MC_2925: **Spindle Vel. Control Diff. Gain (.1mAs^2/rev)**	Normally the spindle-axis differential gain is not used in the speed controller. The differential gain reduces oscillations in the low frequency range (<200 Hz), but it destabilizes the controller in the higher frequency range. Do not use this constant for machine axis if the motor is coupled to the spindle via a timing belt.	Range (0.00–1000.00) 0.00 (0.1mAs^2/rev) [Default]
MC_2926: **Spindle Pos. Control Prop. Gain (1/min)**	Sets the spindle-axis positional control proportional gain. The positional control gain determines the dynamic servo error for an axis without fast feed. MC_2926 = 2000 [1/min], feed = 2000 [mm/min], the dynamic servo error is feed/MC_2924 = 1 [min].	Range (0–100000) (1/min) 40.00 (1/min) [Default]

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2927: **Spindle Pos. Control Output Limit (rpm)**	Limits the spindle-axis output of the positional controller and the standard speed value.	Range (0.00–10000.00) 0.00 (rev/min) [Default]
MC_2928: **Spindle Velocity FeedFwd. Gain**	Spindle-axis velocity feed forward gain.	**No** [Default] Yes
MC_2929: **Spindle Acceleration FeedFwd. Gain (.1mAs^2/rev)**	Spindle-axis acceleration feed forward gain. In practice, this value is 2 to 3 times the motor inertia.	Range (0.0–10000.00) 0.0 (0.1mAs^2/rev) [Default]
MC_2930: **Spindle Coulomb Friction FeedFwd. Gain (mA)**	Spindle-axis torque compensation gain for friction at low rotational speed. Produces an offset according to direction of travel.	Range (-10000.0–10000.0) 0.00 (mA) [Default]
MC_2931: **Spindle Torque Offset (mA)**	Spindle-axis constant torque to offset the compensation (for example, for gravitational force in a vertical axis).	Range (-10000.00–10000.00) 0.00 (mA) [Default]
MC_2932: **Spindle Friction FeedFwd. Timecons (.1ms)**	Spindle-axis delay of the friction compensation to prevent overcompensating when changes in direction occur at high speeds. Typical value: 150 (0.1 msec).	Range (0.00–10000.00) 0.00 (0.1 ms) [Default]
MC_2933: **Spindle Damping FeedFwd. at Nnom (mA)**	Spindle-axis damping compensation at standard speeds. Used for heavy machines.	Range (-10000.00–10000.00) 0.00 (mA) [Default]
MC_2934: **Spindle Torque LP Filter Timecons (.1ms)**	Spindle-axis torque lowpass filter time constant is used when there is insufficient dumping of the axis. Standard value is zero. Realistic input values 3–20 (0.1msec)	Range (0.00–10000.00) 0.00 (0.1ms) [Default]
MC_2935: **Spindle Torque Notch Filter Freq. (.1Hz)**	Vibrations can occur on critical axes and at the spindle in a frequency range, which cannot be compensated either with the differential factor (MC_2925) or with the MC_2934.	Range (0.00–10000.00) 0.00 (0.1Hz) [Default]
MC_2936: **Spindle Torque Notch Filter Damp. (.1dB)**	Spindle-axis damping values of the torque band-stop filter. Damping should not be set unnecessarily high which would restrict the operation of the control loop. Realistic input values are 30–90 (0.1dB).	Range (0.00–10000.00) 0.00 (0.1dB) [Default]

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2937: **Spindle Flux Reduction Begin (rpm)**	For spindle 1 only. To reduce the turning-noise of the spindle motor in a no-load situation, or to achieve a greater voltage-span at the beginning of a brake situation, it is possible to reduce the "no-load' magnetization from a certain revolution. This applies for asynchronous motors only. The value of MC_2937 describes the number of revolutions, from which the no-load magnetization is reduced.	Range (0–40000) 0 (rev/min) [Default]
MC_2938: **Spindle Flux Reduction End (rpm)**	For spindle 1 only. See also MC_2937. The revolutions for which the no-load magnetization is reduced.	Range (0–40000) 0 (rev/min) [Default]
MC_2939: **Spindle Flux Change Factor (%)**	For spindle 1 only. See also MC_2937. Factor by which the no-load magnetization is reduced (maximum 50%, 0=no reduction).	Range (0–100) 0 Off (percent) [Default]
MC_2940: **Spindle Slip Change Timecons. (ms)**	Spindle slip change time constant is for spindle 1 only. If a value not equal to 0 is entered, the motor slip at the start of braking is increased (by a factor of 2). This reduces the power fed back into the converter. The MC-value provides the time constant with which this slip increase is returned to normal during the braking process. This action in the case of overcurrent (successive reduction and increase of the torque current setpoint in relation to the actual current), which can lead to substantial noise, especially with large machines (lathes), is deactivated if MC-valve >0.	Range (0–5000) 0 (ms) [Default]
MC_2941: **Spindle ramp (ms)**	Time in milliseconds required for the spindle to go from zero rpm to programmed rpm, and from programmed rpm to zero, when a spindle CW/CCW or spindle stop command is issued.	Range (180.0–10000.0) 900.0 (ms) [Default]
MC_2942: **Analog Spindle Maximum motor speed (rpm)**	Defines the maximum constant speed specified by the manufacturer of the motor.	Range (10.0–99999.0) 1000.0 (rev/min) [Default]

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2943: **Analog Spindle motor phase**	Defines the motor phase to change the direction of the count.	Invert **Not invert** [Default]
MC_2944: **Analog Spindle Pos. Control Integral Gain**	Defines the integral gain of the position controller. This value applies a long-term accumulation of error over time.	Range (0–99999) 0 [Default]
MC_2945: **Analog Spindle Pos. Control Integral Limit**	Defines the maximum correction of the integral portion of the position controller.	Range (0–99999) 0 [Default]
MC_2961: **Low setting for M40 gear range**	Spindle low setting for M40 gear. See "Setting Spindle Gear Ranges."	Range (0–64000) 50 (revolutions/minute) [Default]
MC_2962: **High setting for M40 gear range**	Spindle high setting for M40 gear. See "Setting Spindle Gear Ranges."	Range (0–64000) 6,000 (rev/min) [Default]
MC_2963: **Low setting for M41 gear range**	Spindle low setting for M41 gear. See "Setting Spindle Gear Ranges."	Range (0–64000) 50 (rev/min) [Default]
MC_2964: **High setting for M41 gear range**	Spindle low setting for M41 gear. See "Setting Spindle Gear Ranges."	Range (0–64000) 6,000 (rev/min) [Default]
MC_2965: **Low setting for M42 gear range**	Spindle low setting for M42 gear. See "Setting Spindle Gear Ranges."	Range (0–64000) 165 (rev/min) [Default]
MC_2966: **High setting for M42 gear range**	Spindle high setting for M42 gear. See "Setting Spindle Gear Ranges."	Range (0–64000) 501 (rev/min) [Default]
MC_2967: **Low setting for M43 gear range**	Spindle low setting for M43 gear. See "Setting Spindle Gear Ranges."	Range (0–64000) 500 (rev/min) [Default]
MC_2968: **High setting for M43 gear range**	Spindle high setting for M43 gear. See "Setting Spindle Gear Ranges."	Range (0–64000) 1,471 (rev/min) [Default]
MC_2969: **Low setting for M44 gear range**	Spindle low setting for M44 gear. See "Setting Spindle Gear Ranges."	Range (0–64000) 1,470 (rev/min) [Default]
MC_2970: **High setting for M44 gear range**	Spindle high setting for M44 gear. See "Setting Spindle Gear Ranges."	Range (0–64000) 4,640 (rev/min) [Default]

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2971: **Gear change RPM**	Spindle gear change RPM specifies the spindle speed at which a gear change is performed.	Range (0–50) 10 (rev/min) [Default]
MC_2972: **Gear ranges used**	To set up for only one gear range, set to Single-M40 [Default]. To set up for multiple gear ranges, select **Multiple**. See "Setting Spindle Gear Ranges."	**Single-M40** [Default] Multiple
MC_2973: **RPM Display**	Spindle RPM Display allows you to configure the CNC to display feedback from a spindle encoder (Feedback) or from a programmed RPM. **NOTE:** This parameter affects only the displayed RPM value. It does not affect RPM or voltage output to the spindle.	**Feedback** [Default] Program
MC_2974: **Check spindle during gear change**	Checking the spindle during gear change allows you to either stop the spindle before you can change a gear (for example, change from M41 to M42) or enables you to change gears without stopping the spindle.	No **Yes** [Default] If this parameter is set to **Yes** and a gear change code is executed with the spindle running, the CNC will generate an error message. If parameter is set to **No**, a gear change code may be executed with the spindle running.
MC_2975: **Stop program on gear change**	Stopping the program on gear change allows you to configure the CNC to stop program execution during a gear change.	**No** [Default] Yes If you specify **Yes**, the CNC checks for spindle movement (RPM). If there is spindle movement, the CNC displays an error message and stops the machine. If you specify **No**, the CNC does not stop the machine for a gear change.
MC_2976: **Check RPM to be within gear range**	Checking RPM to be Within Gear Range prevents you from designating a RPM outside the active gear range.	No **Yes** [Default] If you specify **Yes**, the CNC checks the RPM. If the entered RPM is not within the active gear range, the CNC displays an error message. If you specify **No**, the CNC does not check the entered RPM against the active gear range.

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2977: **Stop/Start spindle during Hold/Start**	During Automatic operations (that is, Auto or Single Step), the CNC has the capability of automatically stopping the spindle when the **Hold** key is pressed and re-starting the spindle when the **Start** key is pressed. The spindle is re-started only if it was previously running. **NOTE:** This feature also applies to External Hold and External Start input functions.	**No** [Default] Yes If you specify **Yes**, this feature is enabled. Sending the appropriate M-codes through the interface does the stopping and starting of the spindle. That is, M5 is sent for stopping, and M3 or M4 (based on which one was active when the spindle was stopped) is sent for starting.
MC_2978: **Number of motor teeth for M40 gear range**	Gearing on the M40 spindle-axis motor side. Set at 1 if there is no gear train.	Range (0–64000) 1 [Default]
MC_2979: **Number of spindle teeth for M40 gear range**	Gearing on the M40 spindle-axis spindle side. Set at 1 if there is no gear train.	Range (0–64000) 1 [Default]
MC_2980: **Invert gear direction for M40**	Change the M40 spindle-axis motor direction.	**No** [Default] Yes
MC_2981: **Number of motor teeth for M41 gear range**	Gearing on the M41 spindle-axis motor side. Set at 1 if there is no gear train.	Range (0–64000) 1 [Default]
MC_2982: **Number of spindle teeth for M41 gear range**	Gearing on the M41 spindle-axis spindle side. Set at 1 if there is no gear train.	Range (0–64000) 1 [Default]
MC_2983: **Invert gear direction for M41**	Change the M41 spindle-axis motor direction.	**No** [Default] Yes
MC_2984: **Number of motor teeth for M42 gear range**	Gearing on the M42 spindle-axis motor side. Set at 1 if there is no gear train.	Range (0–64000) 1 [Default]
MC_2985: **Number of spindle teeth for M42 gear range**	Gearing on the M42 spindle-axis spindle side. Set at 1 if there is no gear train.	Range (0–64000) 1 [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_2986: **Invert gear direction for M42**	Change the M42 spindle-axis motor direction.	**No** [Default] Yes
MC_2987: **Number of motor teeth for M43 gear range**	Gearing on the M43 spindle-axis motor side. Set at 1 if there is no gear train.	Range (0–64000) 1 [Default]
MC_2988: **Number of spindle teeth for M43 gear range**	Gearing on the M43 spindle-axis spindle side. Set at 1 if there is no gear train.	Range (0–64000) 1 [Default]
MC_2989: **Invert gear direction for M43**	Change the M43 spindle-axis motor direction.	**No** [Default] Yes
MC_2990: **Number of motor teeth for M44 gear range**	Gearing on the M44 spindle-axis motor side. Set at 1 if there is no gear train.	Range (0–64000) 1 [Default]
MC_2991: **Number of spindle teeth for M44 gear range**	Gearing on the M44 spindle-axis spindle side. Set at 1 if there is no gear train.	Range (0–64000) 1 [Default]
MC_2992: **Invert gear direction for M44**	Change the M44 spindle-axis motor direction.	**No** [Default] Yes

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
Linear Correction Compensation Setup Parameters		
MC_3000: **Linear correction compensation**	Linear correction compensation corrects for detected mechanical errors (in the ballscrew or elsewhere) that affect the indicated distance displayed by the CNC. To determine the amount of correction required, measure the error with a calibration device. When linear correction is activated, the CNC multiplies the commanded move by the compensation value. If you do not require linear compensation, disable this feature. When enabled, you can specify a different correction value for each axis. Correction = Distance Read by CNC ÷ Distance Actually Traveled Enter any appropriate correction factor from 0.300000 to 3.000000.	On (enabled) **Off** (disabled) [Default]
MC_3001: **X Linear correction compensation**	To determine the amount of X-axis correction required, measure the error with a calibration device. When linear correction is activated, the CNC multiplies the commanded move by the compensation value. Correction = Distance Read by CNC ÷ Distance Actually Traveled	Range (0.300000–3.000000) 1.000000 [Default]
MC_3002: **Y Linear correction compensation**	To determine the amount of Y-axis correction required, measure the error with a calibration device. When linear correction is activated, the CNC multiplies the commanded move by the compensation value. Correction = Distance Read by CNC ÷ Distance Actually Traveled	Range (0.300000–3.000000) 1.000000 [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_3003: **Z Linear correction compensation**	To determine the amount of Z-axis correction required, measure the error with a calibration device. When linear correction is activated, the CNC multiplies the commanded move by the compensation value. Correction = Distance Read by CNC ÷ Distance Actually Traveled	Range (0.300000–3.000000) 1.000000 [Default]
MC_3004: **U Linear correction compensation**	To determine the amount of U-axis correction required, measure the error with a calibration device. When linear correction is activated, the CNC multiplies the commanded move by the compensation value. Correction = Distance Read by CNC ÷ Distance Actually Traveled	Range (0.300000–3.000000) 1.000000 [Default]
Skew Error Compensation Setup Parameters		
MC_3015: **Skew error compensation**	Skew error compensation corrects for orthogonal machine errors. Skew error is the amount of error, in radians, an axis deviates from the parallel. Use a calibration device to determine the amount of correction required.	On (enabled) **Off** (disabled) [Default] Switch **On** to activate compensation entered. The CNC activates skew compensation for all affected axes.
MC_3016: **X Skew error compensation**	Use a calibration device to determine the amount of X-axis correction required.	Range (-10.0000–10.0000) 0.0000 [Default]
MC_3017: **Y Skew error compensation**	Use a calibration device to determine the amount of Y-axis correction required.	Range (-10.0000–10.0000) 0.0000 [Default]
MC_3018: **Z Skew error compensation**	Use a calibration device to determine the amount of Z-axis correction required.	Range (-10.0000–10.0000) 0.0000 [Default]
MC_3019: **U Skew error compensation**	Use a calibration device to determine the amount of U-axis correction required.	Range (-10.0000–10.0000) 0.0000 [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
Backlash Compensation Setup Parameters		
MC_3030: **Backlash compensation**	Backlash is the loss of motion that occurs when the encoder reverses direction and begins to record motion before the table actually moves. Backlash compensation takes this lag into account and corrects the move. All systems that move mass under control exhibit backlash. Some of the causes are structural component flexing, bearing end thrust, and wind-up of the ballscrew that drive the slide.	On (enabled) **Off** (disabled) [Default] Switch **On** to activate compensation entered. The CNC activates the backlash compensation for all selected axes.
MC_3031: **X Backlash compensation**	Measure X-axis backlash and type the value. When backlash compensation activates, the CNC automatically calculates the necessary motion corrections.	Range (0.0000–9.0000) 0.0000 [Default]
MC_3032: **Y Backlash compensation**	Measure Y-axis backlash and type the value. When backlash compensation activates, the CNC automatically calculates the necessary motion corrections.	Range (0.0000–9.0000) 0.0000 [Default]
MC_3033: **Z Backlash compensation**	Measure Z-axis backlash and type the value. When backlash compensation activates, the CNC automatically calculates the necessary motion corrections.	Range (0.0000–9.0000) 0.0000 [Default]
MC_3034: **U Backlash compensation**	Measure U-axis backlash and type the value. When backlash compensation activates, the CNC automatically calculates the necessary motion corrections.	Range (0.0000–9.0000) 0.0000 [Default]
Ballscrew Compensation Setup Parameters		
MC_3050: **Ballscrew compensation**	The CNC can compensate for inaccuracies along the ballscrew. This ensures a high degree of precision in the finished workpiece. The machine builder can specify as many as 128 equally sized segments per axis for calibration.	On (enabled) **Off** (disabled) [Default] **NOTE:** Use Ballscrew Compensation with the Automatic File Loader. Perform a Machine Home sequence before you enable ballscrew compensation.

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_3051: **X Number of segments**	To determine the number of X-axis segments required, consider that the number of segments multiplied by the segment size should equal the entire range of travel for the axis. Type the number of segments desired for the X-axis.	Range (1–128) 1 [Default]
MC_3052: **Y Number of segments**	To determine the number of Y-axis segments required, consider that the number of segments multiplied by the segment size should equal the entire range of travel for the axis. Type the number of segments desired for the Y-axis.	Range (1–128) 1 [Default]
MC_3053: **Z Number of segments**	To determine the number of Z-axis segments required, consider that the number of segments multiplied by the segment size should equal the entire range of travel for the axis. Type the number of segments desired for the Z-axis.	Range (0–128) 1 [Default]
MC_3054: **U Number of segments**	To determine the number of U-axis segments required, consider that the number of segments multiplied by the segment size should equal the entire range of travel for the axis. Type the number of segments desired for the U-axis.	Range (0–128) 1 [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_3056: **X Offset**	Both the Offset and Zero Cross parameters enable you to specify a starting point for ballscrew compensation. Both values are measured from Machine Home. These values include distance and direction (positive or negative) from Machine Home. The CNC adds the two values to determine the starting point. For example, if the assigned offset is -0.01mm and the Zero cross is -6.00mm, then the CNC begins the compensated (lasered) area -6.01mm from Machine Home along the axis. Typically, Machine Home (0.0000) is the Zero Cross parameter and the Offset is just off the limit switch. However, any point along the range of travel can be selected for the Zero Cross or Offset. Enter the appropriate Ballscrew Offset for the X-axis. If the Offset location is Machine Home, enter **0.0000**. The ballscrew offset is measured from Machine Home.	Range (-999.00000–2032.00000) 0.00000 [Default]
MC_3057: **Y Offset**	Enter the appropriate Ballscrew Offset for the Y-axis. If the Offset location is Machine Home, enter **0.0000**. The ballscrew offset is measured from Machine Home. (See MC_3056: X Offset for details.)	Range (-999.00000–2032.00000) 0.00000 [Default]
MC_3058: **Z Offset**	Enter the appropriate Ballscrew Offset for the Z-axis. If the Offset location is Machine Home, enter **0.0000**. The ballscrew offset is measured from Machine Home. (See MC_3056: X Offset for details.)	Range (-999.00000–2032.00000) 0.00000 [Default]
MC_3059: **U Offset**	Enter the appropriate Ballscrew Offset for the U-axis. If the Offset location is Machine Home, enter **0.0000**. The ballscrew offset is measured from Machine Home. (See MC_3056: X Offset for details.)	Range (-999.00000–2032.00000) 0.00000 [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_3061: **X Zero cross**	Enter the appropriate Zero Cross parameter for the X-axis. If the Zero Cross parameter is at Machine Home, enter **0.0000**. All entered values are referenced to Machine Home. (See MC_3056: X Offset for details.)	Range (-999.00000–2032.00000) 0.00000 [Default]
MC_3062: **Y Zero cross**	Enter the appropriate Zero Cross parameter for the Y-axis. If the Zero Cross parameter is at Machine Home, enter **0.0000**. All entered values are referenced to Machine Home. See MC_3056: X Offset for details.)	Range (-999.00000–2032.00000) 0.00000 [Default]
MC_3063: **Z Zero cross**	Enter the appropriate Zero Cross parameter for the Z-axis. If the Zero Cross parameter is at Machine Home, enter **0.0000**. All entered values are referenced to Machine Home. (See MC_3056: X Offset for details.)	Range (-999.00000–2032.00000) 0.00000 [Default]
MC_3064: **U Zero cross**	Enter the appropriate Zero Cross parameter for the U-axis. If the Zero Cross parameter is at Machine Home, enter **0.0000**. All entered values are referenced to Machine Home. (See MC_3056: X Offset for details.)	Range (-999.00000–2032.00000) 0.00000 [Default]
MC_3066: **X Segment length**	In Ballscrew Compensation, the length of each lasered segment must be the same. The CNC counts off the segments from the beginning of the compensated area, as determined by the sum of the Offset and Zero Cross values assigned. The entered value should represent the segment length for each axis and the direction (positive or negative) of travel along the axis. Enter the desired segment length for the X-axis. (This value is a negative number for the negative travel direction with respect to the Machine Home position.)	Range (-999.00000–2032.00000) 0.00000 [Default]
MC_3067: **Y Segment length**	Enter the desired segment length for the Y-axis. (This value is a negative number for the negative travel direction with respect to the Machine Home position.)	Range (-999.00000–2032.00000) 0.00000 [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_3068: **Z Segment length**	Enter the desired segment length for the Z-axis. (This value is a negative number for the negative travel direction with respect to the Machine Home position.)	Range (-999.00000–2032.00000) 0.00000 [Default]
MC_3069: **U Segment length**	Enter the desired segment length for the U-axis. (This value is a negative number for the negative travel direction with respect to the Machine Home position.)	Range (-999.00000–2032.00000) 0.00000 [Default]
MC_3072: **Edit Ballscrew Table**	See "Laser File Data File Format."	Table Entries – Setup X 0.00000 Y 0.00000 Z 0.00000 U 0.00000
Software Limits Setup Parameters		
MC_4001: **X+ Software Limit**	Type the positive X-axis software limit. See "Setting Software Limits."	Range (0.0000–12192.0000) 0.0000 [Default]
MC_4002: **Y+ Software Limit**	Type the positive Y-axis software limit. See "Setting Software Limits."	Range (0.0000–12192.0000) 0.0000 [Default]
MC_4003: **Z+ Software Limit**	Type the positive Z-axis software limit. See "Setting Software Limits."	Range (0.0000–12192.0000) 0.0000 [Default]
MC_4004: **U+ Software Limit**	Type the positive U-axis software limit. See "Setting Software Limits."	Range (0.0000–12192.0000) 0.0000 [Default]
MC_4006: **X- Software Limit**	Type the negative X-axis software limit. See "Setting Software Limits."	Range (-12192.0000–0.0000) 0.0000 [Default]
MC_4007: **Y- Software Limit**	Type the negative Y-axis software limit. See "Setting Software Limits."	Range (-12192.0000–0.0000) 0.0000 [Default]
MC_4008: **Z- Software Limit**	Type the negative Z-axis software limit. See "Setting Software Limits."	Range (-12192.0000–0.0000) 0.0000 [Default]
MC_4009: **U- Software Limit**	Type the negative U-axis software limit. See "Setting Software Limits."	Range (-12192.0000–0.0000) 0.0000 [Default]
MC_4011: **X Software Limit Enable**	See "Setting Software Limits."	**Off** (disabled) [Default] On (enabled)
MC_4012: **Y Software Limit Enable**	See "Setting Software Limits."	**Off** (disabled) [Default] On (enabled)

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_4013: **Z Software Limit Enable**	See "Setting Software Limits."	**Off** (disabled) [Default] On (enabled)
MC_4014: **U Software Limit Enable**	See "Setting Software Limits."	**Off** (disabled) [Default] On (enabled)
Continuous Path Setup Parameters		
MC_4020: **Continuous Path Range**	With Continuous Path Mode active, the CNC blends one move into another, without a complete stop between moves. The Continuous Path Mode activates at power On and is used for feed moves. The CNC approaches the target position and comes within the continuous path tolerance of the target. Then, the CNC begins to calculate the next programmed move. It does not make an in-position check before it executes the next move. This results in a smoothly contoured profile or surface. The CNC activates the **Continuous Path Mode**(s) for all selected axes.	**On** (enabled) [Default] Off (disabled)
MC_4021: **X Continuous Path Range**	Type the desired X-axis range, or accept the default. **NOTE:** MC_2051: X In-position Tolerance Range must be smaller than MC_4021: X Continuous Path Range.	Range (0.0000–9.0000) 0.0700 (inches) [Default]
MC_4022: **Y Continuous Path Range**	Type the desired Y-axis range, or accept the default. **NOTE:** MC_2151: Y In-position Tolerance Range must be smaller than MC_4022: Y Continuous Path Range.	Range (0.0000–9.0000) 0.0700 (inches) [Default]
MC_4023: **Z Continuous Path Range**	Type the desired Z-axis range, or accept the default. **NOTE:** MC_2252: Z In-position Tolerance Range must be smaller than MC_4023: Z Continuous Path Range.	Range (0.0000–9.0000) 0.0700 (inches) [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_4024: **U Continuous Path Range**	Type the desired U-axis range, or accept the default. **NOTE:** MC_2351: U In-position Tolerance Range must be smaller than MC_4024: U Continuous Path Range.	Range (0.0000–9.0000) 0.0700 (inches) [Default]
Position Error Check (PEC) Setup Parameters		
MC_4030: **Position Error Checking**	The CNC detects a loss of motion and declares an error via the Position Error Check (PEC) algorithm. You can configure the parameters for these calculations. **WARNING: The Position Error Check parameter must be enabled for the CNC system to be able to declare a servo fault and shut down the system in an emergency.** If the PEC algorithm detects a fault, the servos shut off and the CNC displays one of the following messages: **ERROR: (AXIS) LAG OVER MAX!** **ERROR: LOST (AXIS) FEEDBACK!**	**Yes** (enabled) [Default] No (disabled)
MC_4031: **PEC Check idle time (ms)**	The amount of time, in milliseconds, allowed between the internal command for a move and the input of counts from the feedback device, indicating motion.	Range (25.00–5000.00) 100.00 (ms) [Default]
MC_4032: **PEC Maximum lag error**	The error distance allowed at rest or low feedrates, before declaring a fault.	Range (0.0000–2.5400) 0.0200 (mm) [Default]
Jog Return Position Setup Parameters		
MC_4050: **X Jog position**	X jog position defines a point on the machine X-axis in reference to Machine Home to which the CNC will return when the Jog and Return function is activated. In Auto and S.Step Modes, use the Jog and Return function to remove the tool from the part without aborting the program. It can be used to change a tool or inspect a critical dimension during normal operation.	Range (-9999.0000–25400.0000) 0.0000 [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_4051: **Y Jog position**	Y jog position defines a point on the machine Y-axis in reference to Machine Home to which the CNC will return when the Jog and Return function is activated.	Range (-9999.0000–25400.0000) 0.0000 [Default]
MC_4052: **Z Jog position**	Z jog position defines a point on the machine Z-axis in reference to Machine Home to which the CNC will return when the Jog and Return function is activated.	Range (-9999.0000–25400.0000) 0.0000 [Default]
MC_4053: **U Jog position**	U jog position defines a point on the machine U-axis in reference to Machine Home to which the CNC will return when the Jog and Return function is activated.	Range (-9999.0000–25400.0000) 0.0000 [Default]
Direct Numeric Control (DNC) Setup Parameters		
MC_4060: **DNC Execution Mode**	See "Selecting a DNC Execution Mode."	**Buffered** [Default] Drip Feed
MC_4061: **DNC Buffer Size (in buff. mode)**	See "Setting the Buffer Size."	16K **32K** [Default] 64K 128K Max
MC_4062: **DNC Use DNC Macro at end of block**	Select to use a specified DNC Macro at the end of each block.	**No** (disabled) [Default] Yes (enabled)
MC_4063: **DNC Macro Number**	Enter the DNC Macro Number to use at the end of each block.	Range (100–32000) 100 [Default]
MC_4064 **DNC Ignore G41/2, G59, and Blueprint**	The operator can enable/disable processing for G41 (left of path), G42 (right of path), G59 (Corner Rounding), and Blueprint programming (Canned Cycles). While in DNC, these features should be disabled unless absolutely required. If these features are enabled, even if a program does not use them, the CNC will waste processing time.	**Yes** [Default] No Choose **Yes** to disable Tool Comp and CornerRad. Choose **No** to enable Tool Comp and CornerRad during DNC.
MC_4065: **DNC Wait for Start**	See "Setting Wait for Start Before Execution."	No **First** [Default] Every

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
Handwheel Setup Parameters		
MC_4100: **Handwheel Active**	The counter port is configured for handwheel use.	**Off** (not active) [Default] On (active)
MC_4101: **Handwheel Resolution**	Selects the axis resolution for the handwheel attached to the port. Set the resolution to MP Switch to select a resolution from the jog selector on the manual panel.	**MP Switch** [Default] – This setting is available for Handwheel #1. The resolution is determined by the setting on the CNC's Manual Panel resolution selector rotary switch. SK Switch – This setting is only available for Handwheel #2. The resolution is determined by a setting selected via soft keys in Manual Mode. See P/N 70000487, *6000M CNC Programming and Operations Manual* for more details. Fixed 1 – Corresponds to resolution x1, x10 and x100 respectively. Fixed 10 Fixed 100 3-HW Adapter – Resolution selected by a switch on a 3-handwheel adapter.
MC_4102: **Handwheel Axis**	Selects the axis controlled by the handwheel on this port. Set the axis to MP Switch to select an axis from the rotary switch on the manual panel or assign the handwheel to a dedicated axis.	**MP Switch** [Default] – This setting is only available for Handwheel #1. The axis is determined by the setting on the CNC's Manual Panel axis selector rotary switch. SK Switch – This setting is only available for Handwheel #2. The axis is determined by a setting selected via soft keys in Manual Mode. See P/N 70000487, *6000M CNC Programming and Operations Manual* for more details. Fixed X – The handwheel axis can be fixed to a specific primary axis. Fixed Y Fixed Z 3-HW Adapter– Axis selected by a switch on a 3-handwheel adapter.
MC_4103: **Handwheel Scaling Factor**	Changes the sensitivity of the handwheel. A higher number will make the axis run faster. A lower number will make the axis run slower.	Range (0.0001–1.0000) 1.0000 [Default]

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
Home Setup Parameters		
MC_4104: **Handwheel Phase**	The counting direction of the handwheel measuring system is changed.	**Not invert** [Default] Invert
MC_4200: **Home required**	Home Required specifies whether the CNC requires a Home Sequence prior to performing any other operations at system power up. Switch the option On (**Yes**) to enable the Homing Feature or Off (**No**) to disable the Homing Feature. If you set the Home Required to **Yes**, you must perform the home sequence before you resume normal operations in the control software.	**No** [Default] Yes
MC_4201: **Home preset**	You can automatically preset Machine Zero to any coordinates. When the machine completes the homing sequence, the CNC sets the display to the preset values entered for all axes.	**Off** (disabled) [Default] On (enabled)
MC_4202: **X Home sequence**	Sets the order in which axes are homed. This sets the X-axis homing sequence. [Default homing order: Third, Second, First, and Fourth corresponding to axes X, Y, Z, and U)]	Range (0–4) 2 [Default]
MC_4203: **Y Home sequence**	Sets the order in which axes are homed. This sets the Y-axis homing sequence. [Default homing order: Third, Second, First, and Fourth corresponding to axes X, Y, Z, and U)]	Range (0–4) 1 [Default]
MC_4204: **Z Home sequence**	Sets the order in which axes are homed. This sets the Z-axis homing sequence. [Default homing order: Third, Second, First, and Fourth corresponding to axes X, Y, Z, and U)]	Range (0–4) 0 [Default]
MC_4205: **U Home sequence**	Sets the order in which axes are homed. This sets the U-axis homing sequence. [Default homing order: Third, Second, First, and Fourth corresponding to axes X, Y, Z, and U)]	Range (0–4) 3 [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_4207: **X Home type**	Allows you to specify the types of homing for the X-axis. Use Home Type to set the direction of travel for the Homing feature. Positive/Negative refers to the direction that the axis will travel during Homing, in reference to Machine Home. Positive/Negative Index Limit The CNC moves the selected axis in the positive/negative direction until it detects an index pulse from the linear encoder or rotary encoder. This method requires that the Machine Home position be known and physically marked on the axis, to ensure repeatability. Positive/Negative Index & Vector Limit When you specify homing, the CNC travels in the specified positive/negative direction along the axis being homed until it trips the home switch. The CNC then reverses direction until it detects an Index pulse. The CNC sets Machine Home for that axis where it detects the Index pulse.	**No Homing** [Default] Disables the homing function. + IL +IL & VL – IL – IL & VL
MC_4208: **Y Home type**	Allows you to specify the types of homing for the Y-axis. See MC_4207 for details.	**No Homing** [Default] Disables the homing function. + IL +IL & VL – IL – IL & VL
MC_4209: **Z Home type**	Allows you to specify the types of homing for the Z-axis. See MC_4207 for details.	**No Homing** [Default] Disables the homing function. + IL +IL & VL – IL – IL & VL
MC_4210: **U Home type**	Allows you to specify the types of homing for the U-axis. See MC_4207 for details.	**No Homing** [Default] Disables the homing function. + IL +IL & VL – IL – IL & VL

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_4212: **X Home search feed**	To set the speed at which the machine travels the X-axis during Homing.	Range (1.0–12700.0) 10.0 (inches/minute) [Default]
MC_4213: **Y Home search feed**	Sets the speed at which the machine travels the Y-axis during Homing.	Range (1.0–12700.0) 10.0 (inches/minute) [Default]
MC_4214: **Z Home search feed**	Sets the speed at which the machine travels the Z-axis during Homing.	Range (1.0–12700.0) 10.0 (inches/minute) [Default]
MC_4215: **U Home search feed**	Sets the speed at which the machine travels the U-axis during Homing.	Range (1.0–12700.0) 10.0 (inches/minute) [Default]
MC_4217: **X Home preset**	You can automatically preset Machine Zero to any coordinates. When the machine completes the homing sequence, the CNC sets the display to the preset values entered the X-axis.	Range (-25400.0000–25400.0000) 0.0000 [Default]
MC_4218: **Y Home preset**	Sets home preset value for the Y-axis.	Range (-25400.0000–25400.0000) 0.0000 [Default]
MC_4219: **Z Home preset**	Sets home preset value for the Z-axis.	Range (-25400.0000–25400.0000) 0.0000 [Default]
MC_4220: **U Home preset**	Sets home preset value for the U-axis.	Range (-25400.0000–25400.0000) 0.0000 (mm) [Default]
Miscellaneous Setup Parameters		
MC_4301: **Max. programmed linear axis feedrate**	The maximum-programmed linear axis feedrate sets a limit on how fast the CNC allows the machine to travel a linear axis in feedrate. **NOTE:** You can override the maximum-programmed feedrate with the **FEEDRATE OVERRIDE** switch. The range of the switch is 0 to 120% of the maximum-programmed feedrate. The switch varies the feedrate in increments of 10%.	Range (0.0–25400.0) 200.0 (inches/min) [Default]
MC_4302: **Linear axis dry run feedrate**	See "Linear and Rotary Axis Dry Run Feedrate."	Range (254.0–5080.0) 60.0 (inches/min) [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_4303: **Linear axis jog feedrate**	Set up the feedrate for linear axes at which the machine travels in Jog Mode. This defines the machine's default jog speed. **NOTE:** The **FEEDRATE OVERRIDE** switch allows you to override the Jog Feedrate. The range of the switch is 0 to 120% of the maximum programmable feedrate; or 0 to 100% of the maximum-programmed Rapidrate. The switch varies the feedrate in increments of 10%.	Range (0.0–12700.0) 40.0 (inches/min) [Default]
MC_4304: **Linear axis jog rapidrate**	Set up the rapidrate for linear axes at which the machine travels in Jog Mode. This defines the machine's default jog speed.	Range (0.0–12700.0) 300.0 (inches/min) [Default]
MC_4305: **Max. programmed rotary axis feedrate**	Max programmed rotary axis feedrate sets the maximum speed, in degrees per minute, that a rotary axis may be programmed **NOTE:** The Default Jog Feedrate can be overridden with the **FEEDRATE OVERRIDE** switch . The range of the switch is 0 to 120% of the maximum programmable feedrate. The switch varies the feedrate in increments of 10%. This menu selection defines the rotary feedrate at 100%.	Range (0.0–12700.0) 3,000.0 (degree per minute) [Default]
MC_4306: **Rotary axis dry run feedrate**	When a program is run in Dry Run Mode, the machine's rotary axis moves through the program without cutting into the work. The CNC activates Coolant Off and the work may or may not be placed on the table.	Range (254.0–2540.0) 1,000.0 (degree per minute) [Default]
MC_4307: **Rotary axis jog feedrate**	Set up the feedrate for rotary axes at which the machine travels in Jog Mode. This defines the machine's default jog speed for rotary axis. **NOTE:** The Default Jog Feedrate can be overridden with the **FEEDRATE OVERRIDE** switch. The range of the switch is 0 to 120% of the maximum programmable feedrate. The switch varies the feedrate in increments of 10%. This menu selection defines the jog rotary feedrate at 100%.	Range (0.0–12700.0) 1,016.0 (mm/degrees per minute) [Default]

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_4308: **Rotary axis jog rapidrate**	Set up the rapidrate for rotary axes at which the machine travels in Jog Mode.	Range (0.0–12700.0) 3,000.0 (mm/degree per minute) [Default]
MC_4309: **Servo up delay**	You can program a delay to allow the servos to stabilize before the CNC commands a move.	Range (0–999) 2 (second) [Default]
MC_4310: **Automatic feedrate override on arcs**	When this feature is activated, the CNC modifies the feedrate of arc moves in Cutter Compensation Mode. It ensures that the tool cuts at the programmed feedrate at the point where the edge of the tool contacts the workpiece. The CNC slows down the feedrate on inside arc moves and speeds up the feedrate on outside arc moves. The compensated feedrate assigned by the CNC depends on the active Cutter Compensation Mode (G41 Left of Path or G42 Right of Path), the active tool nose radius and the programmed arc radius.	**No** [Default} Yes (When you change the setting to **Yes**, the CNC activates automatic override feedrate for arc moves made in Cutter Compensation Mode. Switch the setting to **No** to deactivate the feature.)
MC_4311: **Rapid moves are free (unsynchronized)**	The rapid moves determined to be free or synchronized.	**No** [Default] Yes
MC_4312: **Display Resolution**	The Display Resolution parameter allows you to specify the display resolution of the system. Machine systems should use SVGA; while off-line systems can use VGA or SVGA.	**VGA** (640x480) SVGA (800x600) [Default]
MC_4314: **Rapid Ramp Size in ms**	Acceleration ramp time for rapid rate.	Range (0–270) 108 (ms) [Default]
MC_4315: **Feed Ramp Size in ms**	Acceleration ramp time for feed rate.	Range (0–270) 108 (ms) [Default]
MC_4316: **CNC Startup mode**	Acceleration ramp time for feed rate.	**Sfwr Options** – Software stops at the main menu after the introduction screen is displayed. Ctrl Software – Software goes to the Control software section with stopping at the main menu.

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_4390: **Show Introduction Screen**	Displays the introduction splash screen.	**Yes** – Enable screen display. [Default] No – Skip the introduction screen. The "System Options" main menu is displayed.
M-Code Macro Call Parameters		
MC_4400: **Mcode for macro call #1**	M-Code number you assign to call the macro in MC_4401.	Range (0–99999) 0 [Default]
MC_4401: **Macro called for Mcode #1**	The macro number that is called when the M-Code in MC_4400 is executed.	Range (0–99999) 0 [Default]
MC_4402: **Mcode for macro call #2**	M-Code number you assign to call the macro in MC_4403.	Range (0–99999) 0 [Default]
MC_4403: **Macro called for Mcode #2**	The macro number that is called when the M-Code in MC_4402 is executed.	Range (0–99999) 0 [Default]
MC_4404: **Mcode for macro call #3**	M-Code number you assign to call the macro in MC_4405.	Range (0–99999) 0 [Default]
MC_4405: **Macro called for Mcode #3**	The macro number that is called when the M-Code in MC_4404 is executed.	Range (0–99999) 0 [Default]
MC_4406: **Mcode for macro call #4**	M-Code number you assign to call the macro in MC_4407.	Range (0–99999) 0 [Default]
MC_4407: **Macro called for Mcode #4**	The macro number that is called when the M-Code in MC_4406 is executed.	Range (0–99999) 0 [Default]
MC_4408: **Mcode for macro call #5**	M-Code number you assign to call the macro in MC_4409.	Range (0–99999) 0 [Default]
MC_4409: **Macro called for Mcode #5**	The macro number that is called when the M-Code in MC_4408 is executed.	Range (0–99999) 0 [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_4410: **Mcode for macro call #6**	M-Code number you assign to call the macro in MC_4411.	Range (0–99999) 0 [Default]
MC_4411: **Macro called for Mcode #6**	The macro number that is called when the M-Code in MC_4410 is executed.	Range (0–99999) 0 [Default]
MC_4412: **Mcode for macro call #7**	M-Code number you assign to call the macro in MC_4413.	Range (0–99999) 0 [Default]
MC_4413: **Macro called for Mcode #7**	The macro number that is called when the M-Code in MC_4412 is executed.	Range (0–99999) 0 [Default]
MC_4414: **Mcode for macro call #8**	M-Code number you assign to call the macro in MC_4415.	Range (0–99999) 0 [Default]
MC_4415: **Macro called for Mcode #8**	The macro number that is called when the M-Code in MC_4414 is executed.	Range (0–99999) 0 [Default]
MC_4416: **Mcode for macro call #9**	M-Code number you assign to call the macro in MC_4417.	Range (0–99999) 0 [Default]
MC_4417: **Macro called for Mcode #9**	The macro number that is called when the M-Code in MC_4416 is executed.	Range (0–99999) 0 [Default]
MC_4418: **Mcode for macro call #10**	M-Code number you assign to call the macro in MC_4419.	Range (0–99999) 0 [Default]
MC_4419: **Macro called for Mcode #10**	The macro number that is called when the M-Code in MC_4418 is executed.	Range (0–99999) 0 [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
Tool Management Setup Parameters		
MC_5000: **Activate tool length offset**	See "Activation Options."	No - Function is not used. On Tn - Function activates only when a tool is activated (T-Word). On M6 - Function activates only when Tool Changer M-function (M6) activated. **Both** [Default] - Function activates when a tool number or M6 is activated.
MC_5001: **Output signal**	See "Activation Options."	**No** [Default] On Tn On M6 Both
MC_5002: **Orient spindle**	See "Activation Options."	**No** [Default] On Tn On M6 Both
MC_5003: **Default spindle orientation angle**	Selects an angle of orientation beyond the marker pulse. The range is 0.1 to 360 degrees. This feature eliminates the need for exact mechanical positioning of the spindle encoder. The spindle orientation angle is programmable via CNC software. Enter angle value in degrees.	Range (0.0–360 degrees) 0.0 [Default]
MC_5004: **Spindle orientation RPM**	Specifies the orientation RPM of the spindle. Consult relevant spindle drive documentation for proper spindle speed encoding and appropriate orientation speeds. Maximum programmable orientation spindle speed is 50 RPM. Refer to "Enabling M19 Commands." Enter orientation RPM.	Range (1–250) 10 (rev/min) [Default]
MC_5006: **Stop program execution**	Halts the running program until given a Cycle Start from the Manual Panel. For manual tool change operations using a Programmable Controller, set this selection to **No** (disabled). For manual tool change operations without a Programmable Controller, set to **On Tn**. The CNC will hold program run and display a message. Press **START** to resume program run.	**No** [Default] On Tn On M6 Both

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_5007: **Tool changer installed**	Refer to "Enabling M19 Commands."	None – Disables M19 **Fixed Rep.** [Default] – Fixed Replacement; active tool always returned to the same bin after use. Random Rep. – Random Replacement; active tool can be placed in the next available bin. CNC indexes the tool changer and keeps track of the bin number corresponding to each tool number. Adds Fix and Bin columns to the Tool Page. **BIN** specifies BIN in which a tool is initially located. **FIX** specifies if the corresponding tool needs to go back into the same BIN from which it was taken.
MC_5008: **Use tool change macro**	See "Guidelines for Setting Tool Change Macro Parameters."	**No** [Default] On Tn On M6 Both
MC_5009: **Tool change macro program**	Filename of the macro to be used during tool change operations. Enter Macro filename. See "Guidelines for Setting Tool Change Macro Parameters."	TC.G [Default]
MC_5010: **Tool change macro number**	Macro Number to be used during tool change operations. Macro files can contain more than one macro (number). Enter Macro Number. See "Guidelines for Setting Tool Change Macro Parameters."	Range (8000–9000) O8000 [Default]
MC_5011: **Number of digits in T word**	A 4-digit word will allow selection of from 1 to 99 tools. A 6-digit word will allow selection of from 1 to 999 tools. See "Guidelines for Setting the Number of Digits for T-Words."	Range (4 or 6) 4 [Default] T-word Format
MC_5012: **Number of bins in tool changer**	Enter number of bins.	Range (0–99) 0 [Default]
MC_5013: **Number of Tools to display in table**	Limits the number of tools displayed in the tool table. Enter maximum number of tools.	Range (25–255) 99 [Default]
MC_5014: **Default tool-table file**	Enter filename.	P6MTOOL.DAT

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_5015: **Force spindle off during tool change**	The CNC will generate an error message if tool change is attempted with spindle running. With an automatic tool changer, the tool change would normally handle turning off the spindle.	**Yes** [Default] No
MC_5018: **Restore TLO after power-up or home**	Restore tool-length offset (TLO) after power-up or home sequence.	**No** [Default] Yes
MC_5019: **Spindle orientation type**		**Standard** – Uses the marker pulse from the motor. Used only when there is a 1-to-1 ratio in the spindle. [Default] X30-Marker – Uses the marker pulse and the proximity switch. Used when ratio is not 1-to-1. X30 Only – Uses the proximity switch. Used on spindles with more than one gear.
MC_5020: **Logic of device used for X30 input**	Matches proximity switch logic type used for spindle orientation via X30.	**Norm. Closed** [Default] Norm. Open
MC_5021: **Spindle orientation new method selected**	The only spindle orientation types affected by this parameter are: Standard and X30 Only. If set to No, spindle orientation stops at the marker pulse and continues to the specified angle. If set to Yes, spindle orientation is performed without stopping at the marker pulse.	**No** [Default] Yes
MC_5022: **Spindle orientation with axis move**	If set to No, spindle orientation is completed then any axis move is initiated. If set to Yes, spindle orientation is performed along with any axis move that is commanded in the same block (simultaneous moves). The next block is executed when both M19 and axis move are finished.	**No** [Default] Yes
MC_5023: **Spindle orientation tolerance (deg.)**	The M19Flag changes from 1 to 2 only if the spindle error is within the tolerance set by MC_5023. MC_5023 setting 0.0000 (default) is designed to be the Off setting.	Range (0.0000–5.0000) 0.0000 [Off, Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
Interface Setup Parameters		
MC_5100: **Interface Timeout**	When an output port configured for a finish pulse is activated, the CNC is put on hold until an external finish pulse is received. While on hold, the CNC will not run program blocks (in Programmed Mode) and will not respond to keypad inputs (in Manual Mode). The finish pulse is a signal from a machine device. It informs the CNC that the requested operation is completed. If no finish pulse is received by the end of the timeout period, the CNC will display an **Error** message. **NOTE:** Press **E-STOP** and **SERVO RESET** to regain control of a CNC holding for a finish pulse.	Range: (0–600,000 milliseconds) A zero (0) entry causes an indefinite hold. 10,000 (milliseconds) [Default] (10 sec)
MC_5101: **Issue SpStop on SvoFlt or Estop**	Issuing a spindle stop allows you to stop the spindle when there is a servo fault or emergency stop.	**Yes** (on) [Default] No (off)
MC_5102: **Display internal interface message**	Displaying the internal interface messages allows you to override the internal interface messages (that is, feedhold, external hold, etc.) so that the IPI can generate a different message.	**Yes** (on) [Default] No (off)
MC_5103: **Gauge 1 Active**	Gauges allow you to monitor analog inputs that vary from zero to 5 VDC (for example, DC outputs for drive controllers, adjustment potentiometers, etc.).	**No** (deactivate) [Default] Yes (activate)
MC_5104: **Gauge 1 Name**	Assign a name (e.g., spindle load) to the gauge in the entry field.	Blank [Default]
MC_5105: **Gauge 1 Type**	Assign a value of 0 to the gauge in the entry field. This allows monitoring of the spindle load on the particular gauge. Other types of monitoring will be provided in the future. Gauges are displayed in the bottom of the CNC's Manual, Auto, S.Step, and MST screens.	Range (0–5) **0** [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
MC_5106: **Gauge 2 Active**	Gauges allow you to monitor analog inputs that vary from zero to 5 VDC (for example, DC outputs for drive controllers, adjustment potentiometers, etc.).	**No** (deactivate) [Default] Yes (activate)
MC_5107: **Gauge 2 Name**	Assign a name (e.g., spindle load) to the gauge in the entry field.	Blank [Default]
MC_5108: **Gauge 2 Type**	Assign a value of 0 to the gauge in the entry field. This allows monitoring of the spindle load on the particular gauge. Other types of monitoring will be provided in the future.	Range (0–5) 0 [Default]
MC_5109: **Gauge 3 Active**	Gauges allow you to monitor analog inputs that vary from zero to 5 VDC (for example, DC outputs for drive controllers, adjustment potentiometers, etc.).	**No** (deactivate) [Default] Yes (activate)
MC_5110: **Gauge 3 Name**	Assign a name (e.g., spindle load) to the gauge in the entry field.	Blank [Default]
MC_5111: **Gauge 3 Type**	Assign a value of 0 to the gauge in the entry field. This allows monitoring of the spindle load on the particular gauge. Other types of monitoring will be provided in the future.	Range (0–5) 0 [Default]

(Continued…)

Table 2-2, Machine Constants Setup (Continued)

Machine Constant Parameter	Function	Settings
More Parameters		
MC_5200: **Units**	The Units parameter specifies the units used to enter dimensional data. If you are using mixed data, input data in one format (inch or mm) first. Change the format (inch or mm) and enter the rest of the data. You can change the units as many times as you need to. By using the proper units you do not need to convert values, but can enter data precisely (that is, no rounding during conversion). The only exception to this rule is the dimensional parameter corresponding to rotary axes. If the auxiliary axis (that is, U) is configured as a rotary axis, then the unit is always in degrees or degrees per minute (that is, deg/min).	**Inch** [Default] MM
MC_5201: **Language**	You can order a system that displays messages and other text in languages other than English. If you attempt to set up the option for a language and the CNC cannot find the associated text file, it displays an error message.	**English** [Default] Spanish French German Italian Swedish
MC_5202: **MC_2X22 tuning increment value**	Sets the value that MC_2*x*22 (velocity control proportional gain (mAs/rev)) is increased or decreased using manual tuning mode.	Range (0.1–20.0) 1.00 (mAs/rev) [Default]
MC_5203: **MC_2X23 tuning increment value**	Sets the value that MC_2*x*23 (velocity control integral time constant in tenths of milliseconds (ms)) is increased or decreased using manual tuning mode.	Range (0.00001–0.5000) **0.00100** (0.1ms) [Default]
MC_5204: **MC_2X26 tuning increment value**	Sets the value that MC_2*x*26 (positional control proportional gain) is increased or decreased using manual tuning mode.	Range (0.100–50.00) **6.00** (1/min) [Default]
MC_5205: **MC_2X29 tuning increment value**	Sets the value that MC_2*x*29 (acceleration feed forward gain (.1mAs^2/rev) is increased or decreased using manual tuning mode.	Range (0.10–200.00) **9.00** (0.1mAs^2/rev) [Default]
MC_5210: **Vertical axis selection**	Defines the axis that is the vertical position. It is used to check the status of the motor brake for the axis that is vertical.	X Y **Z** [Default] U

Control Software Parameters

The Control Software Setup Parameter Group ranges from MC_1000 through MC_1099.

Compensation Cutoff

The **MC_1009: Compensation cutoff angle** parameter minimizes wasted travel on acute angle. Figure 2-1, Compensation Cutoff Angle illustrates two Compensation Cutoff scenarios. Assume all programmed moves are made with Tool Diameter Compensation active. The diagram describes two cases:

Diagram A shows the tool path that results when no Compensation Cutoff angle is used. The tool path travels beyond the part diameter to a point where compensated Moves 1 and 2 intersect, before the CNC executes Move 2.

Diagram B shows the tool path that results when a Compensation Cutoff angle is used. The CNC introduces an arc move, equal to the radius of the cutter, between Programmed Moves 1 and 2. This arc is not programmed, but is a function of the active Compensation Cutoff Angle and alters the tool path, decreasing the amount of travel necessary to complete the programmed moves.

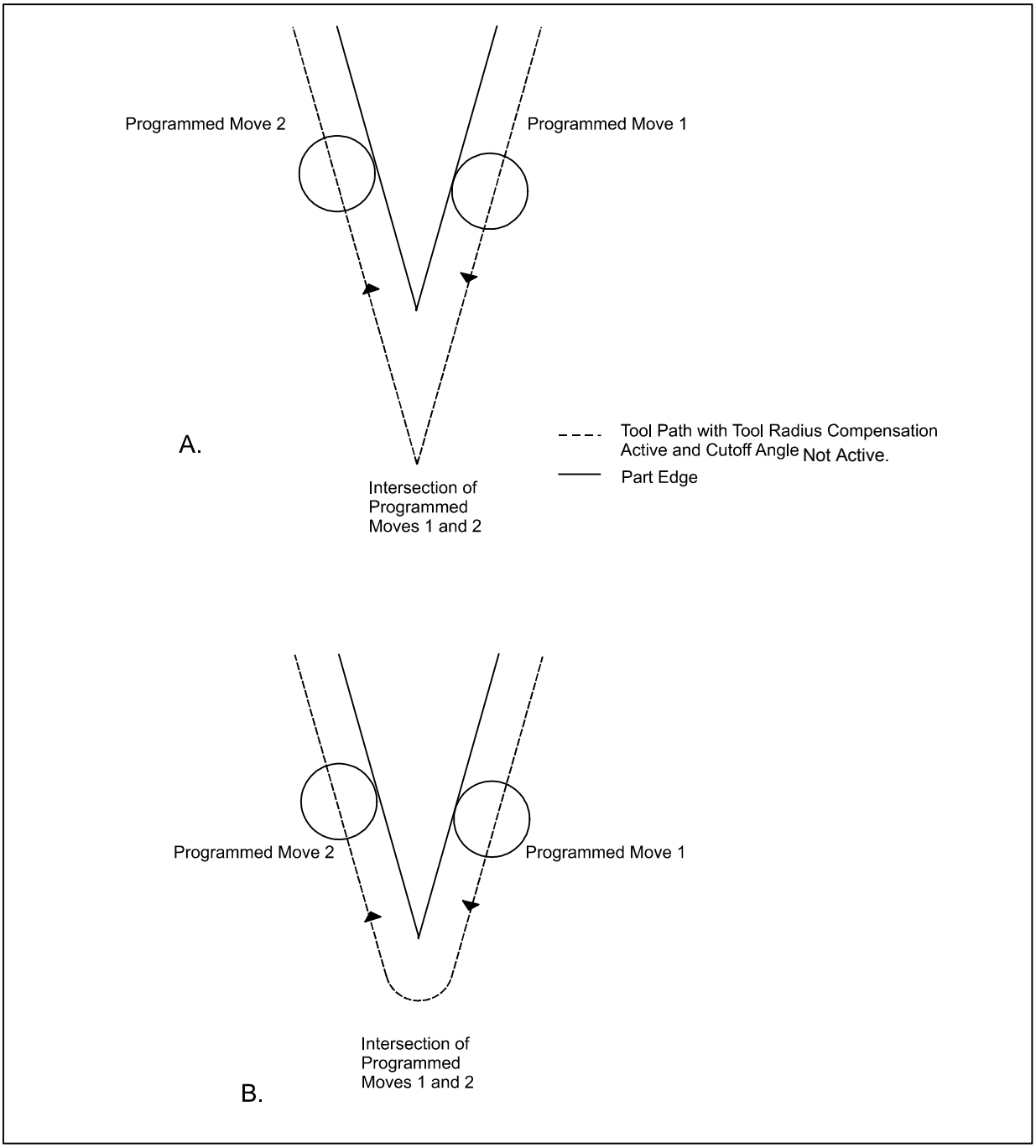


Figure 2-1, Compensation Cutoff Angle

User Definable Variables

User definable variables are defined via machine constants MC_1130 through MC_1139. These machine constants correspond directly to system variables #1130 through #1139. Thus, setting MC_1130 through MC_1139 can set the values of #1130 through #1139. MC_1130 through MC_1134 are unit based; which means, these are assigned the units specified for machine constants (Inch or MM). You can assign MC_1135 to MC_1139 only number values.

A typical usage of these variables would be to define the tool-changer height in a tool-change macro. By using a user definable variable, the height of the tool-changer can be adjusted without editing the macro itself. See "Tool Changer Macro Example."

Program Directory Parameters

The Program Directory Setup Parameter Group ranges from MC_1300 through MC_1349. These parameters specify the following:

The way program information is displayed in the Program Directory

Whether to delete backup files during optimization

Whether and how often the disk is checked via software

RS-232 Communication Parameters

The RS-232 Communication Setup Parameter Group ranges from MC_1350 through MC_1399.

NOTE: Both sending and receiving devices must have the same baud, parity, data bits, stop bits, and software parameter settings.

Encoder Resolution Examples

The following examples illustrate the calculations:

A rotary encoder has 1000 sine periods per revolution. When the encoder is coupled with a 1:1 ratio to a ball screw with 10-mm pitch, there is 10,000 μm.

A linear scale has a sine period of 20 μm. There is 1 period per 20 μm.

Thus:
MC_2004=10000 periods per
MC_3122=10,000μm.

The internal encoder resolution is 1000 * 1024 = 1,024,000 increments per revolution.

The axis resolution is 10,000 μm / 1,024,000 inc = 0.01 μm/increment. The smallest programming unit is always 0.001 mm.

Thus:
MC_2005=1 period per
MC_2006=20 μm.

The internal resolution of the linear scale is 1 * 1024 = 1024 increments per 20 μm.

The axis resolution is 20 μm / 1024 inc = 0.02 μm/increment.
The smallest programming unit is always 0.001 mm.

Axes Parameters

The Axes Setup Parameter Groups ranges from:

- MC_2000 – MC_2099 for **X-axis Setup Parameters**
- MC_2100 – MC_2199 for **Y-axis Setup Parameters**
- MC_2200 – MC_2299 for **Z-axis Setup Parameters**
- MC_2300 – MC_2399 for **U-axis Setup Parameters**
- MC_2900 – MC_2999 for **Spindle-axis Setup Parameters**

A detailed description of some specific axes parameters follows.

Setting the Display Resolution

Reference machine constant parameters:

- **MC_2050: X Display Resolution**
- **MC_2150: Y Display Resolution**
- **MC_2250: Z Display Resolution**
- **MC_2350: U Display Resolution**.

Display Resolution allows you to set the resolution of the axis display. The display resolution must be equal to or coarser than the actual resolution of the installed linear encoder or rotary encoder. Changing the Display Resolution does not affect the accuracy of the machine. Always select resolution in microns, whether the CNC is in Inch Mode or MM Mode. Ensure that resolution settings match the installed equipment. The CNC displays a pop-up window with the choices: 0.5, 1, 2, 5, or 10 Microns. The axis display will show movement at the selected resolution. [Default: **1 Micron**] (0.001mm/0.00005in.)

Refer to **Table 2-3** for conversion values.

Table 2-3, Micron to Inch Conversion

0.5 Micron	0.0005 mm	0.00002”
1 Micron	0.001mm	0.00005”
2 Microns	0.002mm	0.0001”
5 Microns	0.005mm	0.0002”
10 Microns	0.010mm	0.0005”

Setting In-Position Tolerance

Reference machine constant parameters:

MC_2051: X In-position Tolerance Range

MC_2151: Y In-position Tolerance Range

MC_2251: Z In-position Tolerance Range

MC_2351: U In-position Tolerance Range

NOTE: Rapid moves always execute in **In-Position Mode**.

When the CNC has positioned the tool within the in-position tolerance of the target, the CNC processes the next programmed move. At this time, the CNC displays the in-position indicator. Specify the in-position tolerance for each enabled axis in the Setup Utility.
[Default: **0.0100 mm**]

When determining in-position tolerance:

For rotary encoders, tolerance is usually four times the machine resolution (e.g., If machine resolution is 0.0002 in., the in-position tolerance is 0.0008 in.). Use this as a benchmark from which to adjust this value.

For linear encoders, tolerance equals the resolution of the linear encoder.

NOTE: In-position tolerance must be smaller than Continuous path tolerance.

Setting Default Rapid Rate

Reference machine constant parameters:

MC_2053: X Default Rapid Rate

MC_2153: Y Default Rapid Rate

MC_2253: Z Default Rapid Rate

MC_2353: U Default Rapid Rate

Default Rapid Rate sets the speed at which an axis operates in Rapid Mode. This applies to programmed blocks or MDI commands. Jog moves in rapid (that is, from a manual panel) can have a different rapid rate. The machine builder sets the maximum rapid rate according to the physical constraints of the machine. These factors include:

- Available motor torque
- Available servo drive output
- Ballscrew pitch
- Mass to be moved
- Any mechanical advantage gained by pulleys or gears

To override the default rapid rate, adjust the **FEEDRATE OVERRIDE** switch. This switch varies the rapid speed from 0 to 100% and does not affect the maximum rapid rate set.

Spindle Parameters

Refer to **MC_2900–MC_2999, Spindle-axis Setup Parameters**. Use these machine constants to configure spindle settings and gear ranges.

Reference machine constant parameter:

MC_2960: Spindle output.

Spindle output refers to the type of DC drive output provided by the control, as required for the spindle drive in use.

Unipolar – Output varies linearly, depending on the spindle speed the user selects. The range is 0 to +10VDC. Direction must be selected by other means such as reversing contactors.

Bipolar – Output ranges from –10 to +10VDC. A voltage of 0 VDC represents a commanded 0-revolutions/minute (RPM) spindle speed.

The system outputs a negative DC voltage for Spindle Reverse (M4) commands and a positive DC voltage for Spindle Forward (M3) commands. The DC voltage is linear with respect to the RPM of the spindle speed command. Consequently, required voltage (0 VDC to ±10

VDC) increases as spindle speed increases (in reverse or forward directions). The maximum voltage, ±10 VDC, is output at the highest RPM value of the gear range.

Setting Spindle Gear Ranges

Reference machine constant parameter:

MC_2972: Gear ranges used.

NOTE:	The DC output is a linear value based on the high setting for the M40 gear range.

Depending on the mechanical considerations of the system, the spindle drive may not require gearing and belt drive arrangements to provide the required spindle speeds and torque.

You can use the Setup Utility to set either one gear range or up to four separate gear ranges.

To set up for only one gear range, switch MC_2972 to **Single-M40** [Default]. To set up for multiple gear ranges, switch MC_2972 to **Multiple**.

When you set only one gear range, a programmed gear range is not required during spindle operation. For example, a command to activate a DC spindle drive at 1500 RPM reverse direction would be programmed as **S1500 M04.**

When you set multiple gear ranges, the CNC assumes DC spindle operation. You can program up to four separate gear ranges (**M41**, **M42**, **M43**, and **M44**). Each gear range specifies a minimum and a maximum speed for the range. The CNC program requires three entries for spindle operation commands: **gear range**, **speed**, and **direction**, as follows:

Gear range and speed	M42 S1500
Direction	M03

At the highest RPM in the range, the system outputs the maximum DC voltage, +10 VDC. 0 RPM always represents 0 VDC. The lowest RPM voltage is a ratio of the highest speed to the lowest speed. For example, if M41 has a range of 1,000 RPM to 10,000 RPM, then 10,000 RPM results in 10 VDC and 1000 RPM results in 1 VDC.

Defaults are as follows:

MC_2961: Low setting for M40 gear range [Default: **50** rpm]

MC_2962: High setting for M40 gear range [Default: **6,000** rpm]

MC_2963: Low setting for M41 gear range [Default: **50** rpm]

MC_2964: High setting for M41 gear range [Default: **6,000** rpm]

MC_2965: Low setting for M42 gear range [Default: **165** rpm]

MC_2966: High setting for M42 gear range [Default: **501** rpm]

MC_2967: Low setting for M43 gear range [Default: **500** rpm]

MC_2968: High setting for M43 gear range [Default: **0** rpm]

MC_2969: Low setting for M44 gear range [Default: **1,470** rpm]

MC_2970: High setting for M44 gear range [Default: **4,640** rpm]

Ballscrew Compensation Parameters

The Ballscrew Compensation Parameter Group ranges from MC_3050 through MC_3099.

Automatic File Loader

Reference machine constant parameter:

MC_3072: Edit Ballscrew Table.

This feature automatically loads a properly formatted laser data file into the Table Entries Setup Menu.

NOTE:	The File Loader does not change the way segment length ballscrew compensation is set. However, you must enter additional information. Some editing of the laser file will be necessary.

To load the laser file:

Select **On**. Press **Ldfile** (**F8**). Enter the appropriate password, if required by the system.

The CNC displays the Leadscrew Compensation File Loader menu. See Figure 2-2, Ballscrew Compensation Loader Menu. Refer to Table 2-4, Ballscrew File Loader Parameters for a description of the Ballscrew File Loader Parameters.

Highlight **Starting Segment**. Type the segment number of the first table entry, and press **ENTER**.

Highlight **Ending Segment**. Type the segment number for the last table entry, and press **ENTER**.

Highlight **Axis**, and press **ENTER**.

The CNC displays a pop-up menu with the following choices: **X** or **Z**.

5. Highlight the desired axis, and press **ENTER**.

The CNC returns to the Leadscrew Compensation File Loader menu.

6. Highlight **Action**, and press **ENTER.**
7. Highlight an option in the pop-up menu, and press **ENTER**.
8. Press **Ldfile** (**F8**) again to load the file.

A successful load shows the new entries in the table.

9. Repeat the procedure for the other axes.

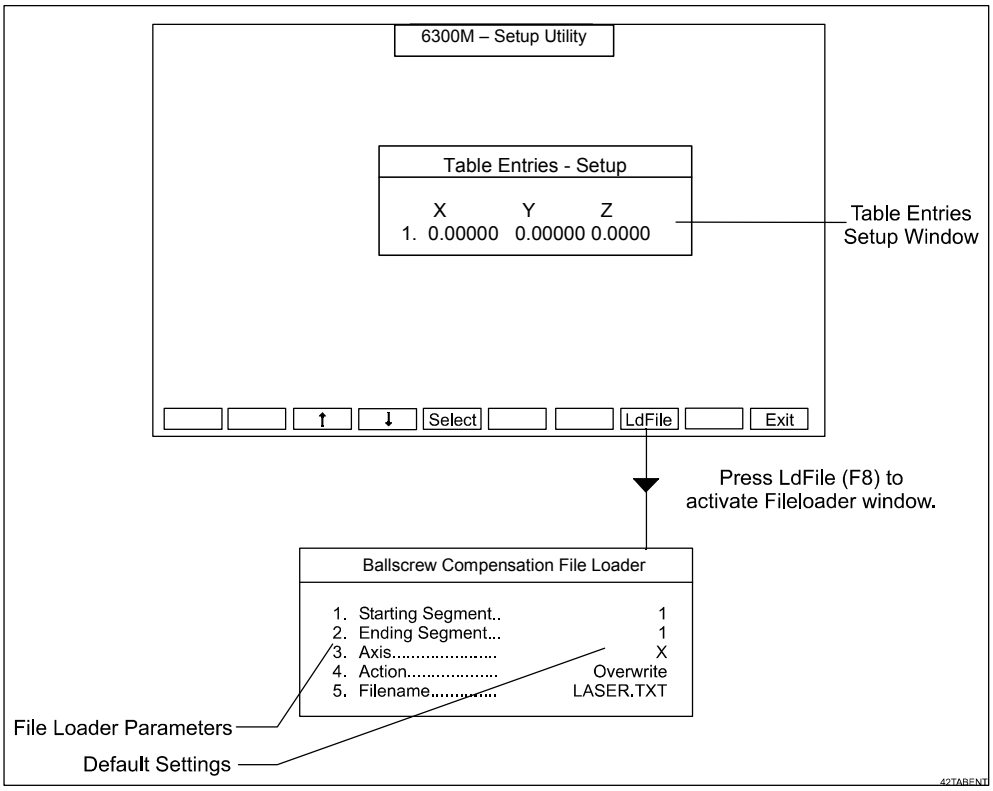


Figure 2-2, Ballscrew Compensation File Loader Menu

Table 2-4, Ballscrew File Loader Parameters

Parameter	Description
Starting Segment	Determines which segment will be the first for data transfer. If a value greater than 128 (maximum number of segments allowed) is entered, an **Error** message results. If a value greater than the Ending Segment value is entered, an **Error** message will be displayed.
Ending Segment	Determines which row in the ballscrew compensation table will be the last to receive data from the laser file. If the segment limit on the table for the axis is exceeded, data will not be entered beyond the limit.
Axis	Determines to which axis data will be applied.
Action	Three types of actions during data load can occur: • Replacing the existing data in the table • Adding to the existing data • Data is averaged with the existing data **Overwrite** will clear any values in the table beyond the segment limit for the axis. **Add** and **Average** only replace the old values. **Action** enables the user to fine-tune ballscrew compensation values from multiple passes of laser readings.
Filename:	Enter the DOS filename of the laser file, including the path, if different from the default.

Laser File Data File Format

Reference machine constant parameter:

MC_3072: Edit Ballscrew Table.

The laser file data must be in the following format for the File Loader Utility:

n1, n2

where:

n1 is the commanded position

, is the delimiter

n2 is the actual position as measured by the laser

Most laser data files have header information, which should be removed.

An example of an acceptable file format is as follows:

0 ,-1.05300568384907E-03

-1 ,-1.00202340866009

-2 ,-2.00227380774995

-3 ,-3.00247420656991

.....

-27 ,-27.0068997761763

-28 ,-28.0070941749639

The delimiter must be a comma (**,**).

Most text editors support Find/Change or Search/Replace commands that facilitate such changes. The first number (**0**, **-1**, **-2**,…) represents the commanded position; the second number represents the actual position measured by the laser.

For example, in the sample data file displayed above, a commanded move to -2.000 in. actually went to -2.00227380774995 in.

NOTE:	Include the 0 value. It is used to calculate the first segment value for the ballscrew compensation table.

Generating Ballscrew Compensation Values from Laser Files

This section describes how the CNC automatically interprets the laser data file. In the sample laser data file above, the following conditions apply:

The segment length is 1 inch.

The 0-inch value (commanded) from the laser data (measured) is approximately -0.00105 in.

The 1-inch value (commanded) from the laser data (measured) is approximately -1.00202 in.

The values are negative, indicating negative machine movement.

The CNC compares the two values by subtracting the 1-inch value from the 0-inch value, then subtracts the segment length from the result, and reverses the sign of the final result for positive travel values.

Method:

1. [(Current Position) - (Previous Position)] **–** (Segment Length) = Directed Error
2. - ((Sign of Segment) (Directed Error)) = Correction Entry

Example:

1. ((-1.00202) **–** (-0.00105)) **–** (-1) = -0.00097
2. -(-(-0.00097)) = -0.00097

This technique is used to find all ballscrew compensation table values. The File Loader automatically enters all compensation values into the Table Entries Setup Menu.

File Loader Error Messages

The File Loader allows up to 128 table entries. If more than 128 entries are loaded, the CNC displays the warning, "**Data from file truncated!**" The message will appear following the data transfer.

Set the segment limit to the proper limit before you attempt the laser file load.

Ensure that the segment length setting matches the displacements of the laser readings. Otherwise, the ballscrew compensation table will contain invalid data. The laser data provided above, for example, shows displacements of one inch per segment. To avoid data error, the operator must enter this value (1") as the segment length before loading the laser readings.

The positive/negative sign of the segment size during ballscrew compensation file loading must match the direction of machine travel used for the laser readings. This also applies to the laser values.

The zero value in the laser file can be positive or negative, regardless of the direction of travel. Otherwise, a negative travel laser file must contain all negative values (with the possible exception of the zero value). The segment size must be negative as well. For positive travel, substitute "positive" for "negative" in all cases.

Software Limits Setup Parameters

The Software Limits Setup Parameter Group ranges from MC_4000 through MC_4019.

Setting Software Limits

Enable the software limit for the specific axis. [Default: **Off** (Disabled)]

MC_4011: X Software Limit Enable

MC_4012: Y Software Limit Enable

MC_4013: Z Software Limit Enable

MC_4014: U Software Limit Enable

You can set positive and negative software limits to restrict travel range. Enter positive and negative software limits separately for each axis. [Default: **0.0000**]

MC_4001: X+ Software Limit

MC_4002: Y+ Software Limit

MC_4003: Z+ Software Limit

MC_4004: U+ Software Limit

MC_4006: X- Software Limit

MC_4007: Y- Software Limit

MC_4008: Z- Software Limit

MC_4009: U- Software Limit

NOTE:	The machine must have the Machine Home function enabled to use software limits.

Reference this physical limit to Machine Zero. If you change the Machine Zero position, the software limits will shift accordingly.

Use the software limits in conjunction with the home limit switches and a Homing cycle command (**G28**) to ensure that the software limits are reliably referenced to an absolute machine position each time the CNC is turned on.

If you do not use homing limit switches, use another method to determine an absolute machine position (e.g., an indicator.)

Direct Numeric Control Setup Parameters

The Direct Numeric Control Setup Parameter Group ranges from MC_4060 through MC_4065.

The Direct Numeric Control (DNC) feature allows the operator to run a program not stored in the CNC's memory. Programs that are larger than the CNC's memory, usually generated from CAD or CAM software, can be run. The program is sent via RS-232 from a computer, another CNC or any other device capable of RS-232 communications.

NOTE: For optimal performance and fewest limitations, transfer the program to the CNC via RS-232 and then run it in Auto Mode, rather than DNC.

Selecting a DNC Execution Mode

The **MC_4060: DNC Execution Mode** parameter tells the CNC to run the transmitted data in Drip Feed or Buffered Mode. [Default: Buffered]

In **Buffered Mode**, the CNC stores incoming data in a buffer (Received Buffer) until the buffer is full. Then, the data is transferred to the Execution Buffer and the CNC runs the transferred blocks. While the CNC runs the Execution Buffer data, it stores more data in the Received Buffer. When all the data in the Execution Buffer have been run, the CNC transfers the contents of the Received Buffer into the Execution Buffer and continues to run the program. The Received Buffer fills up with new data. The process continues until the entire program has been transmitted and run.

In **Drip Feed Mode**, the program is transmitted via RS-232, one block at a time. Blocks are run as soon as they are received. There is no initial delay, but transmission and run times are slower.

Setting the Buffer Size

The **4061: DNC Buffer Size (in buff. mode)** parameter enables the user to specify the amount of memory to be reserved for DNC in Buffered Mode. The choices are:

16K

32K [Default]

64K

128K

Max

"Max" indicates that the control will intelligently estimate the maximum memory allocation. Depending on the size of the program and the amount of available RAM available on the CNC, Max might allow the entire program to be transmitted before a run begins. [Default: **32K**]

Setting Wait For Start Before Execution

Reference the **MC_4065: DNC Wait for Start** machine constant parameter. Refer to **Table 2-5**. Use the **Wait for Start** parameter to specify whether the control will hold data transmission until **START** is pressed. [Default: **First**]

Table 2-5, Wait for Start Parameter Choices

	No Parameter	First Parameter	Every Parameter
Drip Feed	Runs DNC data as soon as it is available.	Must press **START** before running the first block.	Must press **START** before running every block.
Buffered	Runs DNC data as soon as it is available.	Must press **START** before the run of the first block. [Default]	Do not use. To run program block-by-block, switch to Single Step Mode.

The choices are:

No

First [Default]

Every

Miscellaneous Setup Parameters

The Miscellaneous Setup Parameter Group ranges from MC_4300 through MC_4399.

The Miscellaneous Setup parameters enable you to configure various CNC functions not addressed by other setup option menus. These functions are detailed in the following subsections.

Linear and Rotary Axis Dry Run Feedrate

MC_4302: Linear axis dry run feedrate

MC_4306: Rotary axis dry run feedrate

When a program is run in Dry Run Mode, the machine's linear axes (X, Y, and optionally Z), and rotary axis move through the program without cutting into the work. The CNC activates Coolant Off and the work may or may not be placed on the table.

Dry Run Mode is activated by M-codes M105 and M106 and deactivated by M107. Refer to Table 2-6, Dry Run Mode M-Codes for a list of Dry Run related M-codes. Dry run feed rates are set in the Setup Utility. They are often faster than conventional feed rates, but can be set at any rate.

Table 2-6, Dry Run Mode M-Codes

M Code	Function	Description
M105	Dry run on.	Enable machine Dry Run Mode. Program runs at dry run feedrates specified in the Setup.
M106	Dry run, No Z	Enable machine Dry Run Mode, No Z. Program runs at dry run feedrates specified in the Setup, without moving Z-axis.
M107	Dry run off	Cancels active Dry Run Mode.

M-Code for Macro Call and Macro Called for M-Code

The M-Code for macro call and Macro called for M-Code Setup Parameter Group ranges from MC_4400 through MC_4419. Up to 10 M-codes can be assigned to call macros. The M-code number must be greater than zero. The associated macro number must be specified after the M-code. The macro number must also be greater than zero. The macro must be in a macro file that is loaded by the CNC at start-up. This will allow the M-code to be used from manual or in a program. M-Codes that are assigned macros are not passed to the IPI. If the macro number is not assigned, then the M-code is passed to the IPI. For example, for MC_4400 enter 35; for MC_4401 enter 900. This means that whenever M35 is executed the system will run macro O900, if macro is not found then an error is generated.

Tool Management

The Tool Management Setup Parameter Group ranges from MC_5000 through MC_5099.

Activation Options

The following Tool Setup parameters require you to specify a type of activation:

MC_5000: Activate tool-length offset [Default: On Tn]

MC_5001: Output signal [Default: On Tn]

MC_5002: Orient spindle [Default: No]

MC_5006: Stop program execution [Default: No]

MC_5008: Use tool change macro [Default: No]

The available activation options are listed in **Table 2-7**.

Table 2-7, Tool Setup Activation Options

Setting	Description
No	Function is not used.
On Tn	Function activates only when a tool is activated (T-Word).
On M6	Function activates only when Tool Changer M-function (M6) activated.
Both	Function activates when a tool number or M6 is activated.

Manual Tool Change Operation

For manual tool change operations (that is, when a tool changer is not being used), use the settings specified in **Table 2-8**.

Table 2-8, Manual Tool Change Settings

Manual Tool Change Parameters	Required Setting	Description
MC_5000: Activate tool-length offset	On Tn	Tool-Length Offset activates upon completion of a T-word command to the Programmable Controller.
MC_5001: Output signal	On Tn	Refers to T-code data being sent to the Programmable Interface. Select **On Tn** to enable the output signal when the T-code activates.
MC_5006: Stop program execution	No or On Tn	Halts the running program until given a Cycle Start from the Manual Panel. For manual tool change operations using a Programmable Controller, set this selection to **No** (disabled). For manual tool change operations without a Programmable Controller, set to **On Tn**. The CNC will hold program run and display a message. Press **START** to resume program run.
MC_5007: Tool changer installed	None	Set to **None** (disabled). Use this selection to enable closed loop orientation of the spindle during tool changer operation.
MC_5011: Number of digits in T word	4	Format of T-words (Txxxx).
MC_5015: Force spindle off during tool change	Yes	Forces spindle to be off before processing a tool change command. If spindle On, the CNC will generate an error message.

Automatic Tool Change Operation

Program specialized Macro program modules to facilitate the use of any tool changer. Select and edit these Macros via the Tool Management Menu. Use the M6 command during automatic tool changer operations. Refer to **Table 2-9**.

Table 2-9, Automatic Tool Changer Settings

Automatic Tool Change Parameters	Required Setting	Description
MC_5000: Activate tool length offset	On M6	Tool-Length Offset activates upon an M6 command.
MC_5001: Output signal	On M6	"Signal" refers to T-code data being sent to the Programmable Interface. This selection enables that output, upon completion of an M6 command to the Programmable Interface. **On M6**, T-Code data is output by the CNC to the Programmable Interface until a Finish pulse is sent. The M6 data is sent by the CNC and a second Finish pulse is required from the Programmable Interface. If no Tool number is programmed on the same line as the M6 in the CNC program, then the currently active Tool offset data remains unchanged.
MC_5002: Orient spindle	No	Sets up a CNC-controlled, closed loop orientation cycle. If the machine has a mechanically controlled or spindle drive controlled orientation cycle, then the setting for this parameter should be **No** and orientation can be activated via M-code.
MC_5003: Default spindle orientation angle	Enter angle value in degrees.	Selects an angle of orientation beyond the marker pulse. The range is 0.1 to 360 degrees. This feature eliminates the need for exact mechanical positioning of the spindle encoder. The spindle orientation angle is programmable via CNC software.
MC_5004: Spindle orientation RPM	Enter orientation RPM	Specifies the orientation RPM of the spindle. Consult relevant spindle drive documentation for proper spindle speed encoding and appropriate orientation speeds. Maximum programmable orientation spindle speed is 50 RPM. Refer to "Enabling M19 Commands."
MC_5006: Stop program execution	No	When enabled (**Yes**), halts program run until **START** is pressed. For automatic tool changer operations with a Programmable Controller, set to **No** (disabled).

(Continued…)

Table 2-9, Automatic Tool Changer Settings (Continued)

Automatic Tool Change Parameters	Required Setting	Description
MC_5007: Tool changer installed	None ***– or –*** Fixed Replacement ***– or –*** Random Replacement	Enables/disables M19 command. Refer to "Enabling M19 Commands." Choose one of the following parameters: **None:** Disables M19 **Fixed Replacement:** Enables M19; active tool always returned to the same bin after use. **Random Replacement:** Enables M19; active tool can be placed in the next available bin. CNC indexes the tool changer and keeps track of the bin number corresponding to each tool number. Adds Fix and Bin columns to the Tool Page. **BIN** specifies BIN in which a tool is initially located. **FIX** specifies if the corresponding tool needs to go back into the same BIN from which it was taken. [Default]
MC_5008: Use tool change macro	On M6	Sets function to activate on M6 command.
MC_5009: Tool change macro program	Macro filename	Filename of the macro to be used during tool change operations.
MC_5010: Tool change macro number	Macro Number	Macro Number to be used during tool change operations. Macro files can contain more than one macro (number).
MC_5011: Number of digits in T word	T-word Format (4 or 6 digits)	A 4-digit word will allow selection of from 1 to 99 tools. A 6-digit word will allow selection of from 1 to 999 tools.
MC_5012: Number of bins in tool changer	Enter number.	Number of bins. A tool with a bin number greater than this number is considered a manual tool.
MC_5013: Number of Tools to display in table	Enter maximum # of tools.	Limits the number of tools displayed in the tool table. [Default: 99]
MC_5014: Default tool-table file	Enter filename.	Default tool table file: P6MTOOL.DAT
MC_5015: Force spindle off during tool change	No	The CNC will generate an error message if tool change is attempted with spindle running. With an automatic tool changer the tool change would normally handle turning off the spindle.

Enabling M19 Commands

Format: M19

Enable/disable the M19 command during tool changer operations through the **MC_5007: Tool changer installed** parameter in the Tool Management Setup. The M19 command orients the spindle to a given angle via the **MC_5004: Spindle orientation RPM**. The orientation is performed in Closed Loop Mode.

An M19 commands the orientation. The spindle will remain held in closed loop control, even after the Programmable Controller transmits the Finish signal for the M19, until a Tool Change Finish signal is next transmitted.

The M19 Code may be output at any time during the M6 automatic tool change cycle. (For example: A positive spindle position is necessary until the tool is removed from the draw bar or other tool holding mechanism.)

The **MC_5007: Tool changer installed** parameter configures the replacement of tools into the tool changer. Set the parameter to **Fixed Replacement** if a tool should be replaced in the bin from which it was taken. Set the parameter to **Random Replacement** to return the tool to the first available bin.

NOTE: The Random Replacement setting adds two columns to the Tool Page, FIX and BIN. In the BIN column, enter the initial Bins for all tools in the Tool Page. Enter a Y in the FIX column for tools that must always be returned to their original bins. See P/N 70000487, *6000M CNC Programming and Operations Manual* for more details.

Guidelines for Setting the Number of Digits for T-Words

Use the **MC_5011: Number of digits in T word** parameter to configure the number of available T-words. In the CNC program, a four-digit selection is necessary to enable four-digit tool codes:

"T 30 04"

The first two digits specify the tool "pot" or carousel position of the tool to be used (30). The second two digits specify the tool offset being used (T 04).

If a two-digit code is programmed, the offset number being used and the tool being used will be the same:

"T 04" - calls offset number 4 and tool pot number 4

A six-digit selection will permit the programmer to use six-digit tool codes:

"T 130 104"

The first three digits specify the tool "pot" or carousel position of the tool to be used (130). The second three digits specify the tool offset being

used (T 104). If a three-digit code is programmed, then the offset number being used and the tool being used will be the same:

"T 104" - calls offset number 104 and tool pot number 104

Guidelines for Setting Tool Change Macro Parameters

A tool change macro is a subprogram that prepares the machine axes and initiates necessary auxiliary functions prior to automatic tool changer operation.

The Setup Utility contains parameters to create, call and edit the tool change macro filename and macro number. To enable the tool change macro, set the **MC_5008: Use tool change macro** parameter to **On Tn**, **On M6**, or **Both**.

To call a tool change macro in the Setup Utility, specify the filename and macro number. Use the **MC_5009: Tool change macro program** parameter to specify the tool changer macro filename. Use the **MC_5010: Tool change macro number** to specify the appropriate macro number within the program.

NOTE: The macro file is stored in the C:\P6M directory.

The tool change macro is created and edited from the Setup Utility. Press **Edit** (**F8**) to activate the Edit Mode for the macro file and number specified in the menu.

Tool Changer Macro Example

Refer **Table 2-10**. This macro will stop the spindle and send all axes to a safe absolute position. It is a generalized version of an actual macro.

Table 2-10, Tool Change Macro Example

M2	* THIS COMMAND IS NOT OUTPUT TO THE * PROGRAMMABLE CONTROLLER
O 40000	* CREATES G8000
M5	* STOP SPINDLE
G28 Z	* HOME Z AXIS
G00 Z&P-0.6	* SAFE ABSOLUTE METRIC POSITION
G0 X&P0 Y&P0	* MOVE TO SAFE X AND Y POSITION
M99	* END OF MACRO

M2 is required in the first block of the tool change macro file.

Use the relevant G-code to call macros at any time during CNC operation. The macros, created by the macro file, are numbered in the range of G8000 through G8999. Use the O(n) Address Word, followed by the appropriate value, to program a macro G-code. Add 32,000 to the desired G-code number (n). For example, the O40000 program command would create a G8000 Code; O40002 would create G8002, and so forth.

Note usage of the ‘P’ modifier, which is used to provide an absolute position in metric free of any fixture or tool offsets.

Consider, also, using user definable variables (MC_1130 – MC_1139) to specify values that might need adjustment. For example, with MC_1130 set to –0.6 mm, you could substitute G00 Z&-0.6 with G00 Z&P(#1130). Future changes to the to tool changer height would only require changing MC_1130.

Section 3 - Other Setup Options

The following Setup Options are illustrated on Map 1:

- Builder Messages
- Programmable I/O Interface
- Display
- Software Update
- Security

The Configuration Utilities are described in next section.

Builder Messages

IPI can display customized messages to indicate machine status or possible error conditions. These messages are set up in the Setup Utility and displayed in the message area of the CNC screen. To use custom messages, you must create an appropriate conditional logic program that will initialize the proper IPI register.

IPI can send 256 different messages to the CNC, numbered from 0 to 255. The messages are grouped into the following types:

- **Error** codes — The CNC displays an Error message and stops the program run.
- **Warning** codes — The CNC displays an Error message, but allows the program run to continue.

Each message can be a maximum of 49 characters. Use Edit Error Messages to enter or edit Error messages. Refer to **Table 3-1** for message-code ranges and message types.

Table 3-1, Message Code Ranges and Types

Message Codes	Message Types
0	None
1 to 127	Error
127 to 255	Warning

You can create and edit these Builder Messages using the Builder Messages parameter.

Enabling Builder Messages

To enable Builder Messages:

1. See Map 1, **Menu B**. Highlight **Use Custom Messages**.
2. Press **ENTER** to toggle the setting to **No** (disabled) or **Yes** (enabled)].

   [Default: **Yes**]
3. Save the changes before you exit the Setup Utilities.

   The CNC creates the builder messages file **MBENG.TXT**.

Editing Error Messages

To edit Error messages:

1. See Map1**, Menu B**. Highlight **Edit Error Messages**.
2. Press **ENTER**.

   The Prog. Interface Error Messages screen activates. Refer to **Figure 3-1**.

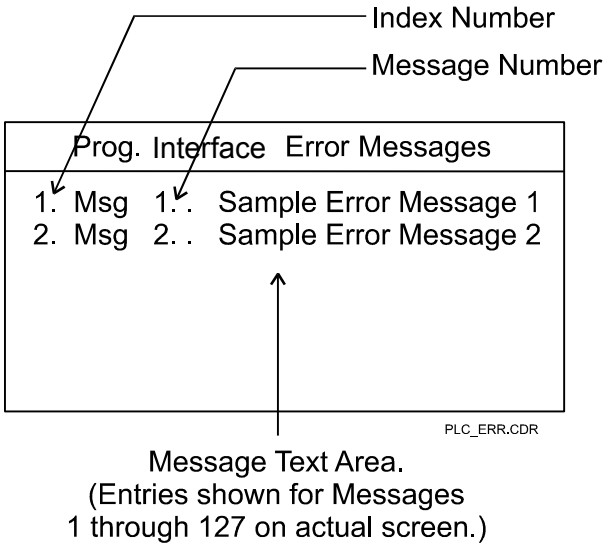


Figure 3-1, Builder Messages - Error Message Window

3. Highlight the message to be entered or edited. Press **ENTER**.

   The CNC activates the message text box.
4. Type the message text in the box. Press **ENTER**.

   The CNC assigns an index number and a message number to each message.

Editing Warning Messages

To edit Warning messages:

1. See Map 1, **Menu B**. Highlight **Edit Warning Messages**.
2. Press **ENTER**.

   The CNC displays the Prog. Interface Warning Messages window. See **Figure 3-2**.

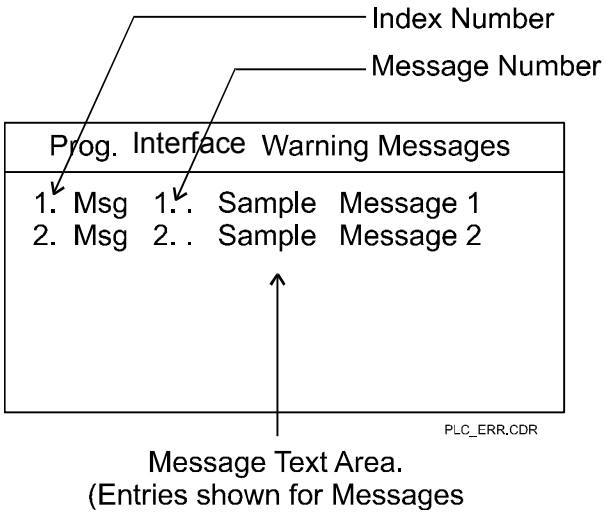


Figure 3-2, Builder Messages - Warning Message Window

3. Highlight the message to be entered or edited. Press **ENTER**.

   The message text box activates.
4. Type the message text in the box. Press **ENTER**.

   The CNC assigns an index number and a message number to each message.

Programmable I/O Interface Setup

The CNC has an integrated programmable I/O interface tool known as Integral Programmable Intelligence (IPI). For information on IPI, refer to the *6000M Integral Programmable Intelligence User's Guide*, P/N 70000488.

Display Settings

Refer to Map 1, **Menu D** for the available Display parameters. The listed parameters control how text and graphics are displayed on the screen. Also included are parameters for controlling display of background, rapid, rapid style, comp rapid, feed, feed style, comp feed, axes, tool, drill marker, and soft keys.

Software Updates

To install an updated version of the CNC software:

1. Insert the disk containing the updated version of the CNC software into the floppy drive.
2. See Map 1, **Menu E**. Highlight **Software Update**. Press **ENTER**.

   Follow the prompts on the screen to complete the installation. The installation process takes less than five minutes. If the system cannot install the new version, it displays an **Error** message. Otherwise, the system displays messages when it has completed each stage of the installation procedure (Extracting Control Software, and so forth)
3. Answer the prompts to complete the installation.

   The system displays a message when the installation is complete and then restarts automatically.

NOTE: Make a copy of the configuration file prior to any software update. Refer to "Section 4 - Configuration Utilities" for how to backup and restore a configuration file.

Security

The Security screen allows you to change passwords. For default passwords, refer to Table 1-4, Default Machine Passwords.

CAUTION: ANILAM urges you to take particular caution if you change the passwords that control access to the Setup Utility. If the password is lost, the operator must erase the current configuration file and reinstall the software (thus restoring the default password) or restore the configuration file from a previous back-up. Make a printed copy before erasing the configuration file. Settings must be input manually after software installation.

To change the password:

1. See Map 1, **Menu E**. Highlight the level of the password to be changed. Press **ENTER**.

   The CNC prompts for the old password.
2. Type the old password, and press **ENTER**.

   The CNC prompts for the new password.
3. Type the new password, and press **ENTER**.

   The CNC prompts for confirmation of the new password.
4. Re-type the new password, and press **ENTER**.

   The CNC activates the new password.

Section 4 - Configuration Utilities

Use the Utilities to manage the configuration file saved in the Setup Utility.

CAUTION:	**Always maintain an updated hardcopy of the configuration file. If you accidentally erase the file, you must enter the settings manually after you reinstall the software.**

Save Configuration

This feature "force saves" a configuration file, regardless of whether any changes were made to the existing file in the Setup Utility.

NOTE:	ANILAM recommends that you save your file before you use any other Configuration Utilities option.

1. See Map 2**, Menu B**. Highlight **Save Configuration**. Press **ENTER**. The system prompts the user for a password.
2. Press **ENTER**. The configuration will be saved. The backup filename is P6MCFG.BAK

Copy Configuration

This feature enables the user to make copies of the configuration and save the copies to various locations using new filenames, if desired.

1. See Map 2**, Menu B**. Highlight **Copy Configuration**. Press **ENTER**. **Menu C** displays.
2. Highlight **A:** to copy the configuration to a floppy diskette. The configuration will be saved as **A:\P6MCFG.CFG**.
3. Highlight **Other..** to save the configuration to another drive or under another filename.

   Type in the drive to which you wish to save the configuration and the new filename. For example: C:\HOME\FILE_1.CFG

NOTE:	If you choose a filename that already exists, the system will warn you that a file already exists. Unless you change the new filename, the system will overwrite the existing file.

Restore from Copy

Use this feature to restore a copy of the configuration from the A:-drive (A:\P6MCFG.GFG) and save it as the new configuration file.

NOTE:	If you restore your configuration file from a copy or a backup, you will need to reboot when the system prompts you to do so. You will be prompted for an automatic reboot.

1. See Map 2**, Menu B**. Highlight **Restore from Copy**. Press **ENTER**. The system prompts for a password.
2. Type the limited access password. (Refer to Table 1-4, Default Machine Passwords.) Press **ENTER**. **Menu D** displays.
3. Highlight **A:** to restore the configuration from a floppy diskette. The configuration will be restored from **A:\P6MCFG.CFG.**
4. Highlight **Other..** to restore the configuration from another drive or another filename. Type in the drive from which you wish to restore the configuration and the new filename (for example, C:\HOME\FILE_1.CFG).

Restore from Backup

When the configuration is saved, the system creates a backup file automatically. Use this feature to “swap” the backup file with the current file.

See Map 2**, Menu B**. Highlight **Restore from Backup**. Press **ENTER**. The system automatically swaps the current file with the backup file.

Compare Configuration

Use this feature to determine if your current configuration file is the same as another file, either on the A:-drive, or elsewhere.

1. See Map 2**, Menu B**. Highlight **Compare Configuration**. Press **ENTER**. **Menu E** displays.
2. Highlight **A:** to compare the current file with P6MCFG.CFG on the A:-drive.
3. Highlight **Other..** to compare the configuration with another drive and another filename.. Type the directory with which you wish to compare files. For example: C:\HOME\FILE_1.

Print Configuration

Use this feature to print the configuration file to a printer.

NOTE:	If a printer is not connected to your parallel port, an Error message is displayed.

1. See Map 2**, Menu B**. Highlight **Print Configuration**. Press **ENTER**.
2. Options Setup **Menu F** is displayed:
   - Highlight **Printer** and press **ENTER**. Press **Yes** (**F1**) to print to your printer. Press **No** (**F2**) to return to **Menu B**.

     – or –

   - Highlight **Text File (A:)** and press **ENTER**. P6MCFG.TXT will be the filename. Press **Yes** (**F1**) to print to the A:-drive. Press **No** (**F2**) to return to **Menu B**.

     – or –

   - Highlight **Text File (Other)** and press **ENTER**. Type the directory and filename to which you wish to print.

Section 5 - Tuning the Current, Velocity, and Position Controller

The CNC has features to automatically (auto-tuning) or manually tune: current, velocity, and position control loops. Auto tuning must be performed in the following sequence for best results: current, frequency, velocity, and position. The auto-tuning features will work in the majority of the systems. For systems in which the auto tuning does not work, guidelines are provided for manually tuning the system. In all cases, before starting a test, energize the servos and move the axis to, approximately, the middle of its travel. Cancel each test by pressing **Manual** (**F4**) before starting the next test.

Tuning Modes

From the Software Options menu, select **Motion Setup/Testing** (MST). From the MST screen, select **TuneMod** (**F6**) to display a pop-up menu with the tuning modes: Automatic or Manual. Select the tuning mode you want and press **ENTER**.

Automatic tuning – requires only an initial start to begin the test and no parameter entry from the user. The test is performed to completion without user intervention.

Manual tuning – requires you to modify parameters and initiate an iteration of the test by pressing **Start**. After an iteration, you can modify the parameters again if desired and continue with a new iteration.

From the MST screen, select an axis displayed on the MST screen [type **X**, **Y**, **Z**, **U**, or **S** (spindle)]. The axis you selected is displayed in the top-right of the MST screen in **Active Axis:**. Press **Tuning** (**F7**) to display a pop-up menu with the tuning mode tests:

Automatic tuning: Current Tune, Frequency Tune, Velocity Tune, and Position Tune.

Manual tuning: Current Tune, Frequency Tune, Velocity Proportional (Tune), Velocity Integral (Tune), Position Proportional (Tune), and (Acceleration) Feedforward Tune.

Automatic Tuning

Automatic tuning provides:

- Current Tune
- Frequency Tune
- Velocity Tune
- Position Tune

A description for each of these auto-tuning tests follows.

Current Controller Auto-tuning Test

The system provides an auto-tuning feature for the current controller. The velocity and position loops are open (that is, switched off) for this test.

To auto-tune the current controller

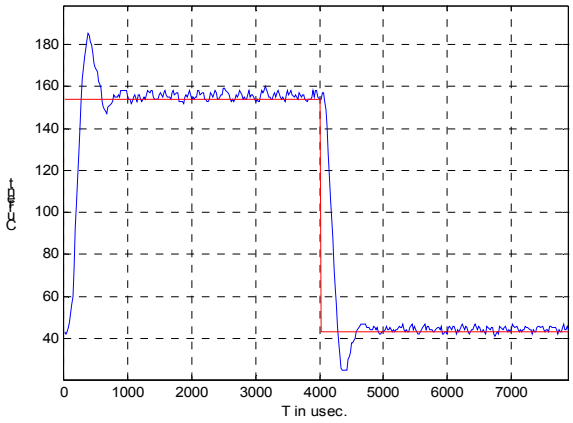
1. On the Software Options menu, select **Motion Setup/Testing** (MST) (option 3) and press **ENTER**.
2. Press **TuneMod** (**F6**). Select **Automatic** and press **ENTER**.
3. On the MST screen, select an axis [**X**, **Y**, **Z**, **U**, or **S** (spindle)]. This activates the axis.
4. Press **Tuning** (**F7**) to display a pop-up menu with the tuning mode tests. Select **Current Tune** and press **ENTER**.

   The test loads a new proportional gain value on each test iteration (it is increased by 1000). No initial parameter is needed. The initial gain is started at 1000, and is incremented until it produces an over-damped response. The feedback and setpoint current are stored and analyzed. While the test is running, the test window displays the loaded current proportional gain and the calculated compensation time. See Figure 5-1, Current Auto-tune Plot.
5. The test is stopped with an optimal proportional gain. That is, when the step response compensation time t_{aus} is less than 600 μsec. When the test stops, the display shows a message at top of the screen, and the optimal value is displayed at bottom of the screen. Another message asks, “Do you want to save the new parameter(s)?”
6. Press **Yes** (**F1**) to load and store the displayed test value in the configuration file. Press **No** (**F2**) to load and store the current value.
7. Repeat Steps 1–6 for each axis [**X**, **Y**, **Z**, **U**, or **S** (spindle)]. Then perform the Frequency Auto-tuning Test.

NOTE: Cycle **E-STOP** after each Current test.

Selecting **Yes** in Step 6 changes the following Machine Constants (depending on the axis selected):

- **MC_2020: X Current Control Gain N<Nom (mV/A)**
- **MC_2120: Y Current Control Gain N<Nom (mV/A)**
- **MC_2220: Z Current Control Gain N<Nom (mV/A)**
- **MC_2320: U Current Control Gain N<Nom (mV/A)**
- **MC_2920: Spindle Current Control Gain N<Nom (mV/A)**



Time in μ seconds

Figure 5-1, Current Auto-tune Plot

If the test does not appear to be producing reasonable results, then shut down the system, restart, and begin the test again. If the test still does not produce reasonable results: stop the test and use Manual Tuning, or use the following tables to enter a safe value. See **Table 5-1** and Table 5-2, Spindle Motors.

Table 5-1, Axes Motors

Motor	MC_2*x*20:	MC_2*x*21:
AM 820A	45,000	0
AM 960A	50,000	0
AM 1150A	45,000	0
AM 1160A	150,000	0
AM 1160C	100,000	0
AM 1160E	50,000	0
AM 1550C	50,000	0
AM 1550E	30,000	0
AM 1550G	30,000	0
AM 1400C	70,000	0
AM 1400A	55,000	0

Auto-tune sets **MC_2921** when it finishes. Both **MC_2920** and **MC_2921** should be entered when manually modified.
In general, 2 times **MC_2920** equals **MC_2921**.

Table 5-2, Spindle Motors

Motor	MC_2920:	MC_2921:
SM 055A	15,000 to 20,000	45,000 to 60,000
SM 075A	10,000	50,000 to 70,000
SM 100A	10,000	30,000 to 45,000
SM 120A	25,000	50,000
SM 055C/D	15,000	40,000
SM 075C/D	10,000	40,000
SM 100C/D	5,000	35,000

Frequency Auto-tuning Test

The system provides a frequency auto-tuning feature for the velocity controller.

To auto-tune using the frequency test

1. On the Software Options menu, select **Motion Setup/Testing** (option 3) and press **ENTER**.
2. Press **TuneMod** (**F6**). Select **Automatic** and press **ENTER**.
3. On the MST screen, select an axis [**X**, **Y**, **Z**, **U**, or **S** (spindle)]. This activates the axis.
4. Press **Tuning** (**F7**) to display a pop-up menu with the tuning mode tests. Select **Frequency Tune** and press **ENTER**.

NOTE: Frequency tune should be done before Velocity tune.

Frequency tune will calculate the oscillating frequency of the axis and set the following Machine Constants:

- **MC_2*x*18: *x* Velocity Filter (FIR filter)**
- **MC_2*x*35: *x* Torque Notch Filter Freq. (.1 Hz)**
- **MC_2*x*36: *x* Torque Notch Filter Damp. (.1 dB)**

The velocity proportional gain is increased by 200 per iteration. A step of 800 mm/min is sent to the velocity controller. After gathering 128 samples (step response), an inverted step is sent again to return motor to its initial position. The frequency spectrum of the step response is calculated through a Fast Fourier Transform (FFT). Then, the power spectral density (PSD) is calculated from the FFT. The total signal energy is calculated from the PSD and also the energy content in the frequency band from 180 Hz. to 540 Hz. (system is expected to resonate within this band). The energy percentage of this band with respect to the total energy is calculated, and if the percentage is more than 50% (system near point of oscillation), this phase is stopped and depending on the oscillation frequency, some filters are activated.

If the oscillating frequency is between 100 and 600 Hz., the notch filter frequency is set to the oscillating one, the damping to 6 dB and the FIR filter is set to 1.

If the frequency is between 600 and 700 Hz., only the FIR is used with a gain of 1.

If the frequency is between 700 and 833 Hz., the FIR filter is used with a gain of 2.

Velocity Controller Auto-tuning Test

The system provides an auto-tuning feature for the velocity controller. The position loop must be open for this test.

To auto-tune the velocity controller

1. The Software Options menu, select **Motion Setup/Testing** (option 3) and press **ENTER**.
2. Press **TuneMod** (**F6**). Select **Automatic** and press **ENTER**.
3. On the MST screen, select an axis [**X**, **Y**, **Z**, **U**, or **S** (spindle)]. This activates the axis.
4. Press **Tuning** (**F7**) to display a pop-up menu with the tuning mode tests. Select **Velocity Tune**, and press **ENTER**.

   No initial parameter is needed. The test always starts from low gain values to avoid motor resonance.

   The first phase of the test is proportional gain tuning. The gain is increased by 200 per test iteration. A step of 600 mm/min is sent to the velocity controller. After 128 samples (step response), an inverted step is sent again to return the motor to its original position. The frequency spectrum of the step response is calculated through a Fast Fourier Transform (FFT).

   Then the power spectral density (PSD) is calculated from the FFT. The total signal energy is calculated from the PSD, and also the energy content in the frequency band from 180 Hz. to 540 Hz.

   The energy percentage of this band with respect to the total energy is calculated. If the percentage is more than 50% (system near point of oscillation), this phase is stopped and half of the proportional gain is chosen as the optimal. At all moments during this phase, the velocity proportional gain and the energy percentage are shown in the test window. After an energy percentage of 20%, there is a check for increasing motor oscillations (motor becomes unstable). If this happens, the test is stopped and an error message is issued.

Figure 5-2 shows the setpoint and feedback after the first phase has finished:

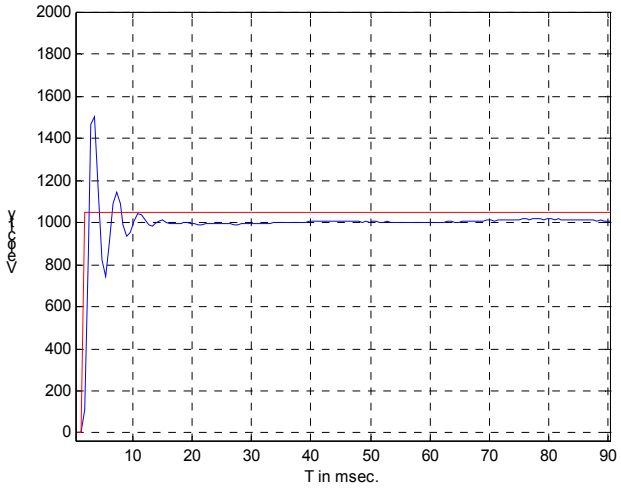


Figure 5-2, Velocity Auto-tune Display (first phase)

The second phase is the integral gain tuning. With the calculated proportional gain loaded, the integral test sends the same step as from the previous phase. It starts increasing its gain until the step response steady state gets within a small setpoint range (10 counts). In this phase, the loaded velocity proportional gain, integral gain, and the count difference between setpoint and feedback are displayed in the test window. Figure 5-3, Velocity Auto-tune Display (second phase) shows the setpoint and feedback after the second phase has finished.

5. When the test stops, the display shows a message at top of the screen, and the two optimal values are displayed at bottom of the screen. Another message asks, “Do you want to save the new parameter(s)?”

6. Press **Yes** (**F1**) to load and store the displayed test value in the configuration file. Press **No** (**F2**) to load and store the current value.

7. Repeat Steps 1–5 for each axis [**X**, **Y**, **Z**, **U**, or **S** (spindle)]. Then perform the Position Controller Auto-tuning Test.

Selecting **Yes** in Step 6 changes the following Machine Constants (depending on the axis selected):

- **MC_2022: X Vel. Control Prop. Gain (mAs/rev)**
- **MC_2023: X Vel. Control Integral Timecons (.1ms)**
- **MC_2122: Y Vel. Control Prop. Gain (mAs/rev)**
- **MC_2123: Y Vel. Control Integral Timecons (.1ms)**
- **MC_2222: Z Vel. Control Prop. Gain (mAs/rev)**
- **MC_2223: Z Vel. Control Integral Timecons (.1ms)**

- **MC_2322: U Vel. Control Prop. Gain (mAs/rev)**
- **MC_2323: U Vel. Control Integral Timecons (.1ms)**
- **MC_2922: Spindle Vel. Control Prop. Gain (mAs/rev)**
- **MC_2923: Spindle Vel. Control Integral Timecons (.1ms)**

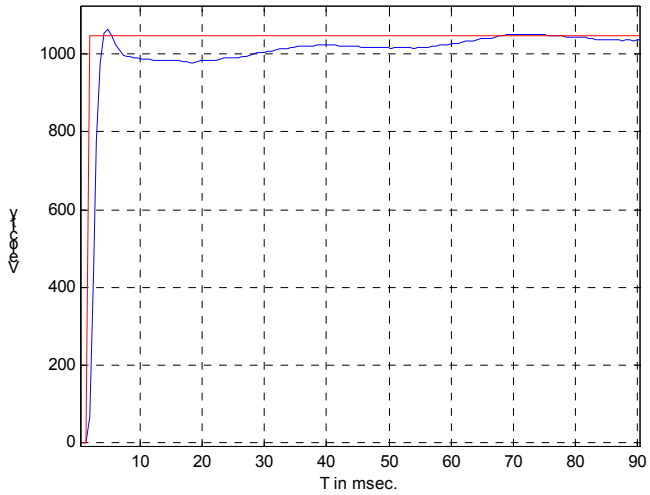


Figure 5-3, Velocity Auto-tune Display (second phase)

If the test fails, repeat to obtain a reasonable value. If the test does not complete, look at the plot and enter 50% of the value at which the plot starts to get distorted for **MC_2*x*22**. For **MC_2*x*23**, use 0.02. If the motor resonates, increase **MC_2x23** by 0.01 until resonance stops. See "Velocity Integral – Manual Tuning Test."

Guidelines to Fine-Tune the Velocity Controller

If the axis move is jerky or tends to overshoot, you can fine-tune the velocity controller by decreasing the proportional gain and increasing the integral time constant for the axis you are adjusting:

- Decrease in steps of 0.5 the Machine Constant (depending on the axis selected):
  - **MC_2022: X Vel. Control Prop. Gain (mAs/rev)**
  - **MC_2122: Y Vel. Control Prop. Gain (mAs/rev)**
  - **MC_2222: Z Vel. Control Prop. Gain (mAs/rev)**
  - **MC_2322: U Vel. Control Prop. Gain (mAs/rev)**
  - **MC_2922: Spindle Vel. Control Prop. Gain (mAs/rev)**
- Increase in steps of 0.005 the Machine Constant (depending on the axis selected):
  - **MC_2023: X Vel. Control Integral Timecons (.1ms)**
  - **MC_2123: Y Vel. Control Integral Timecons (.1ms)**
  - **MC_2223: Z Vel. Control Integral Timecons (.1ms)**

- **MC_2323: U Vel. Control Integral Timecons (.1ms)**
- **MC_2923: Spindle Vel. Control Integral Timecons (.1ms)**

For example, if you are adjusting the X-axis, then you change the **MC_2022** and **MC_2023** pair for the X-axis adjustment. Each axis has a Machine Constant pair for the fine-tuning adjustment.

See "Velocity Proportional – Manual Tuning Test" and "Velocity Integral – Manual Tuning Test."

Position Controller Auto-tuning Test

The system provides an auto-tuning feature for the position controller.

To auto-tune the position controller

1. On the Software Options menu, select **Motion Setup/Testing** (option 3) and press **ENTER**.
2. On the MST screen, select an axis [**X**, **Y**, **Z**, **U**, or **S** (spindle)]. This activates the axis.
3. Press **PosTune** (**F8**). Press the **Start** key to execute the command on the active axis. (For Off-line, press **Alt**+**S**.)

   The proportional gain is increased starting with a value of 10 until the risetime is less than 50 times the sample frequency or the overshoot is larger than 4%. The velocity setpoint, feedback, and the position proportional gain are shown in the test window.

   If velocity feedforward is active, the system also calculates friction feedforward, **MC_2*x*30** and **MC_2231**, torque offset (only for vertical axes). Friction feedforward is used to reduce reversal spikes. If adjustments are necessary, adjust manually. If spike is overcompensated decrease by 10; otherwise, increase by 10. Torque offset is only used for vertical axes. It compensates for any gravitational force in an unbalanced vertical axis.

   If velocity feedforward is activated, the gain for acceleration feedforward, **MC_2*x*29**, is calculated. The gain is increased until the peaks that the lag presents when accelerating and decelerating are reduced to 10 microns.

   If the lag cannot reach 10 microns or if the graph starts to degenerate, increase **MC_2*x*22** by 0.5 and decrease **MC_2*x*23** by 0.005, and repeat the test. If the test still fails, look at the plot and determine at which point the plot starts to degenerate. Enter this value for **MC_2*x*29**.

See "Position Proportional – Manual Tuning Test" and "Acceleration Feedforward – Manual Tuning Test."

Figure 5-4 shows the setpoint and feedback after the optimal proportional gain is found.

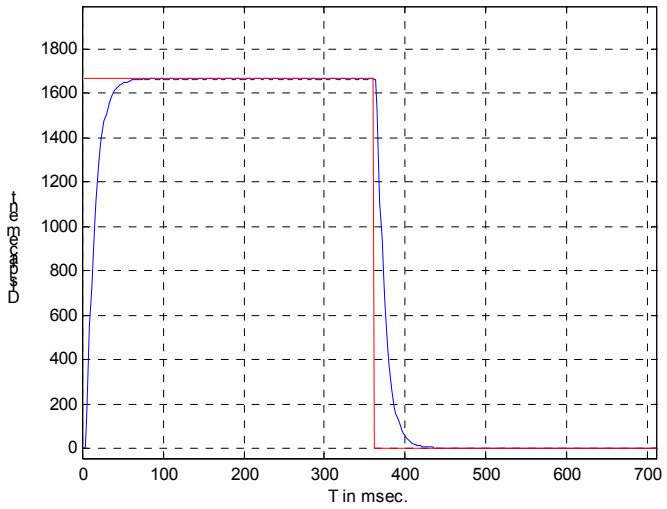


Figure 5-4, Position Auto-tune Display

4. When the test stops, the display shows a message at top of the screen, and:

   - If velocity feedforward is not activated, a single optimal value is displayed at bottom of the screen.
   - If velocity feedforward is activated, two optimal values are displayed at bottom of the screen.

   Another message asks, “Do you want to save the new parameter(s)?”

5. Press **Yes** (**F1**) to load and store the displayed value in the configuration file. Press **No** (**F2**) to load and store the current value.

6. Repeat Steps 1–5 for each axis [**X**, **Y**, **Z**, **U**, or **S** (spindle)]. Then exit the MST display.

Selecting **Yes** in Step 5 changes the following Machine Constants (depending on the axis selected):

- **MC_2026: X Pos. Control Prop. Gain (1/min)**
  When **MC_2028: X Velocity FeedFwd. Gain** is **Yes**, then also displayed is:
  - **MC_2030: X Coulomb Friction FeedFwd. Gain (mA)**
  - **MC_2029: X Acceleration FeedFwd. Gain (.1mAs2/rev)**

- **MC_2126: Y Pos. Control Prop. Gain (1/min)**
  When **MC_2128: Y Velocity FeedFwd. Gain** is **Yes**, then also displayed is:
  - **MC_2130: Y Coulomb Friction FeedFwd. Gain (mA)**
  - **MC_2129: Y Acceleration FeedFwd. Gain (.1mAs2/rev)**

- **MC_2226: Z Pos. Control Prop. Gain (1/min)**
  When **MC_2228: Z Velocity FeedFwd. Gain** is **Yes**, then also displayed is:
  - **MC_2230: Z Coulomb Friction FeedFwd. Gain (mA)**
  - **MC_2229: Z Acceleration FeedFwd. (.1mAs2/rev)**

- **MC_2326: U Pos. Control Prop. Gain (1/min)**
  When **MC_2328: U Velocity FeedFwd. Gain** is **Yes**, then also displayed is:
  - **MC_2330: U Coulomb Friction FeedFwd. Gain (mA)**
  - **MC_2329: U Acceleration FeedFwd. Gain (.1mAs2/rev)**

- **MC_2926: Spindle Control Prop. Cont. Gain (1/min)**
  When **MC_2928: Spindle Velocity FeedFwd. Gain** is **Yes**, then also displayed is:
  - **MC_2930: Spindle Coulomb Friction FeedFwd. Gain (mA)**
  - **MC_2929: Spindle Acceleration FeedFwd. Gain (.1mAs^2/rev)**

Manual Tuning

This test mode allows specific parameter manipulation by the user in order to perform the test. The parameter entries can be increased by pressing the letter "**I**" or decreased by pressing the letter "**D**". The increment amount for manual tuning for **MC_2*x*22**, **MC_2*x*23**, **MC_2*x*26**, and **MC_2*x*29** can be set with setup parameters **MC_5202**, **MC_5203**, **MC_5204**, and **MC_5205** respectively. The suggested default initial values for these parameters are:

MC_5202: MC_2*x*22 tuning increment value	1 (mAs/rev)
MC_5203: MC_2*x*23 tuning increment value	0.001 (0.1ms)
MC_5204: MC_2*x*26 tuning increment value	1 (1/min)
MC_5205: MC_2*x*29 tuning increment value	10 (0.1mAs^2/rev)

Current Tune – Manual Test

The Current Controller manual-tuning test performs exactly as the one for automatic tuning. See "Current Controller Auto-tuning Test." Use Table 5-1, Axes Motors and Table 5-2, Spindle Motors for an initial start value.

Frequency Tune – Manual Test

This test finds the critical frequency at which an axis starts oscillating when its velocity proportional gain is increased. Depending on this frequency, **MC_2*x*18** (Fir filter), **MC_2*x*35** (notch filter center frequency) and **MC_2*x*36** (notch filter damping) are set.

When the test is started, 4 parameter entries are displayed on the screen:

- **MC_2*x*18: *x* Velocity Filter (FIR filter)**
- **MC_2*x*22: *x* Vel. Control Prop. Gain (mAs/rev)**
- **MC_2*x*35: *x* Torque Notch Filter Freq. (.1 Hz)**
- **MC_2*x*36: *x* Torque Notch Filter Damp. (.1 dB)**

The initial values for all the parameters are taken from the values stored in the setup. In order to find the oscillation frequency, the entries for **MC_2*x*18**, **MC_2*x*35**, and **MC_2*x*36** should be set to zero to eliminate any kind of filtering. Every time the **Start** key is pressed, the following are displayed:

- A PSD plot is generated
- Energy content of the 180 to 700 Hz. frequency band
- Oscillation frequency

MC_2*x*22 should be increased little by little (by pressing **I**) and a step response should be commanded (by pressing **Start**) after each parameter change to check the energy percentage and the oscillation frequency results. When the energy percentage gets close to 100% (axis will start to make a loud noise), the test should be stopped.

If the oscillation frequency is less than 600Hz.:

- **MC_2*x*18** should be set to 1
- **MC_2*x*35** should be set to the oscillation frequency multiplied by 10
- **MC_2*x*36** should be set to 60.

If the oscillation frequency is between 600 and 700 Hz.:

- Only **MC_2*x*18** should be set to 1

If the oscillation frequency is more than 700Hz.:

- **MC_2*x*18** should be set to 2.

To save the frequency values

After entering the values, press **Save** (**F10**). If the test is cancelled, the parameters keep their original values.

Velocity Proportional – Manual Tuning Test

This test finds the optimal value for **MC_2*x*22** (velocity proportional gain). This is accomplished by increasing it (suggested initial value 5, increment value 1) until the energy percentage gets close to 100% (axis will start to make a loud noise).

At this time, press **Save** (**F10**). The found value times 0.5 is automatically stored and loaded in the control.

Every time the **Start** key is pressed, a plot of the step response, the loaded **MC_2*x*22** value, the energy percentage and the oscillation frequency are shown on the MST test window.

Velocity Integral – Manual Tuning Test

This test finds the optimal value for **MC_2*x*23** (velocity integral time constant). This is accomplished by decreasing it (by pressing **D**) in small steps (suggested initial value 0.1, increment value 0.001) until the Count Diff. (difference between commanded and feedback velocity at steady state) value is zero or is very close to zero (less than 2 counts).

At this time, press **Save** (**F10**). The values for **MC_2*x*22** and **MC_2*x*23** are automatically stored and loaded in the control.

Every time the **Start** key is pressed, a plot of the step response, the loaded **MC_2*x*22** and **MC_2*x*23** values, the count difference and the rise time are shown on the MST screen.

Position Proportional – Manual Tuning Test

This test finds the optimal value for **MC_2*x*26** (position proportional gain). This is accomplished by increasing it in small steps (suggested initial value 10, increment value 1) until a little overshoot is present in the step response.

At this time, start decreasing **MC_2*x*26** until the overshoot disappears. Press **Save** (**F10**). The value for **MC_2*x*26** is automatically stored and loaded in the control.

Every time the **Start** key is pressed, a plot of the position step response, the loaded **MC_2*x*26** and the overshoot are shown on the MST screen.

Acceleration Feedforward – Manual Tuning Test

This test finds the optimal value for **MC_2*x*29** (acceleration feedforward gain). This is accomplished by increasing it in small steps (suggested initial value 0, increment value 10) until the point in which the maximum lag value stops decreasing and starts to increase (overcompensation starts).

At this time, press **Save** (**F10**). The value for **MC_2*x*29** is automatically stored and loaded in the control.

Every time the **Start** key is pressed, a plot of the lag, the loaded **MC_2*x*29**, **MC_2*x*30**, and **MC_2231** (only for vertical axes) and the maximum lag are shown on the MST screen.

NOTE: If the value of **MC_2*x*30** (coulomb friction feedforward gain) or the value of **MC_2231** (torque offset, only vertical axes) is zero, their optimal value is calculated the first time the **Start** key is pressed. To achieve very small lag values during acceleration / deceleration (~4 microns) on some systems, it is recommended that **MC_2*x*23** be adjusted to a little smaller value than the one calculated in velocity integral tuning.

Miscellaneous Tests

On the MST screen, press **Misc** (**F8**) to display a pop-up menu that provides the selection of 3 tests:

- Current vs. Distance Plot
- Overall System Performance
- I/O monitor

Current vs. Distance Plot

This test displays a plot of the nominal current against the traveled distance commanded through Manual Data Input (MDI). The test detects the differences in friction along the ballscrew length. Thus, this test can be used to find ballscrew-related problems.

To activate Current vs. Distance Plot test

1. On the Software Options menu, select **Motion Setup/Testing** (MST) (option 3) and press **ENTER**.
2. On the MST screen, select an axis [**X**, **Y**, **Z**, or **U**]. This activates the axis.
3. Move the selected axis to one end of its travel.
4. Press **Misc** (**F8**). Select **Current vs. Distance Plot** and press **ENTER**.
5. Press **MDI** (**F5**) and command a move to the opposite end of travel. This displays a plot of the current, and the maximum, minimum, average, and standard deviation of the current are displayed in the MST test window. The plot should be generally flat to indicate no excessive friction along the ballscrews' travel.

NOTE: It is necessary to re-activate the test before each move you want to plot.

Offsets in the main vertical axis (typically, Z-axis) can be detected by using this test. To calculate this offset, the integral component of velocity must be disabled. (Temporarily load 0.5 in **MC_2223**.) After activating it, a move is commanded along the whole vertical axis travel at a slow speed (~5 inches/minute). After the move finishes, you record the average current (I1 in mA.). The process is repeated again in the opposite direction, and you record the average current (I2 in mA.). See **Figure 5-6**.

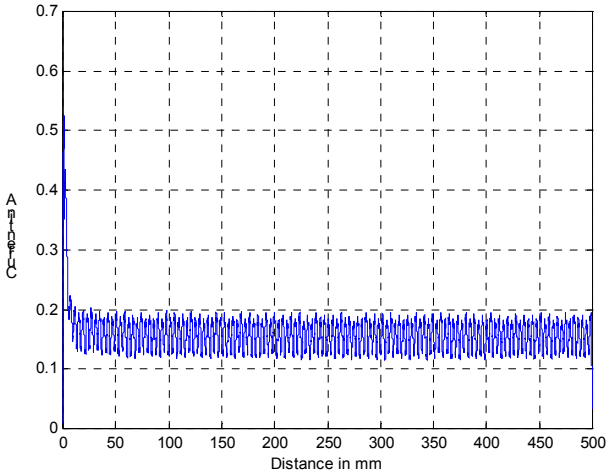


Figure 5-6, Current vs. Distance Plot

You make the following calculation: Ioff = (I1 + I2) / 2. **MC_2231** (Torque offset for Z-axis) is loaded with Ioff in the setup utility to balance the axis. After the calculation is complete, **MC_2223** must be reloaded with its original value.

Overall System Performance

This test displays a step response of the active axis with all loops, all compensations, and all ramps active. The test provides a performance measure of an axis that is following all commanded moves. **MC_4314** (Rapid ramp size in ms.) and **MC_4315** (feed ramp size in ms) may be modified in this test. See Figure 5-7, Overall System Performance Plot.

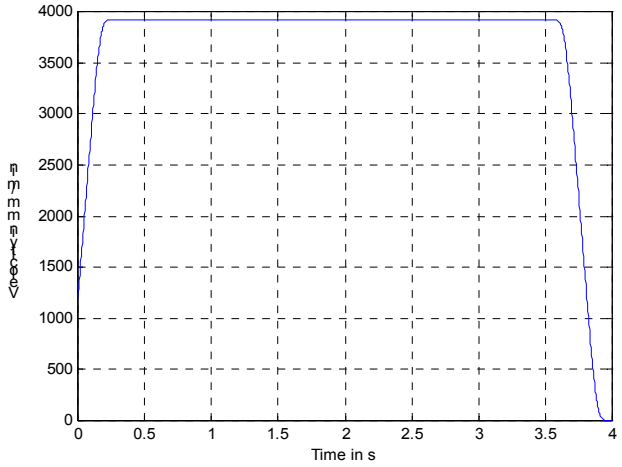


Figure 5-7, Overall System Performance Plot

Use an initial value for **MC_4314** and **MC_4315** of 200 ms. Decrease **MC_4314** by 10 ms. until an overshoot is displayed when a rapid move is commanded. When this occurs, increase **MC_4314** by 10 ms. until the overshoot disappears. Repeat this for **MC_4315** when a feed move is commanded. After the test has been run for all axes, use the highest value obtained from all of the axes tested for the **MC_4314** and **MC_4315** values.

Use **MDI** (**F5**) to command moves. The step response and the resulting overshoot percentage are displayed after the move ends. After **MC_4314** and **MC_4315** are adjusted, use **Save** (**F10**) to store the result.

I/O Monitor

This test displays manual panel inputs and PLC inputs and outputs to be monitored while they are actually changed. PLC I/O is divided in modules:

- Module 0 is always present and represents the internal I/O (32 inputs and 32 outputs).
- Module 1 and so on represent external I/O modules (64 input, 32 outputs per module). They are automatically detected.

The MST screen displays:

I/O Monitor> X30: 0

3-D Probe: 0

Manual Panel T Probe: 0

IN: 0123456789 0123456789 0123456789 01

Module 0

IN: 0123456789 0123456789 0123456789 01

OUT: 0123456789 0123456789 0123

Any active high (1) input or output will be displayed with its number highlighted and vice versa.

X30 Displays the state of the spindle reference input. Spindle proximity switch is connected to X30. This determined by the definition set for **MC_5020** (Logic of device used for X30 input).
Values: [0 (open) or 1 (closed) when **MC_5020** is Normal closed and vice versa when Normal open]

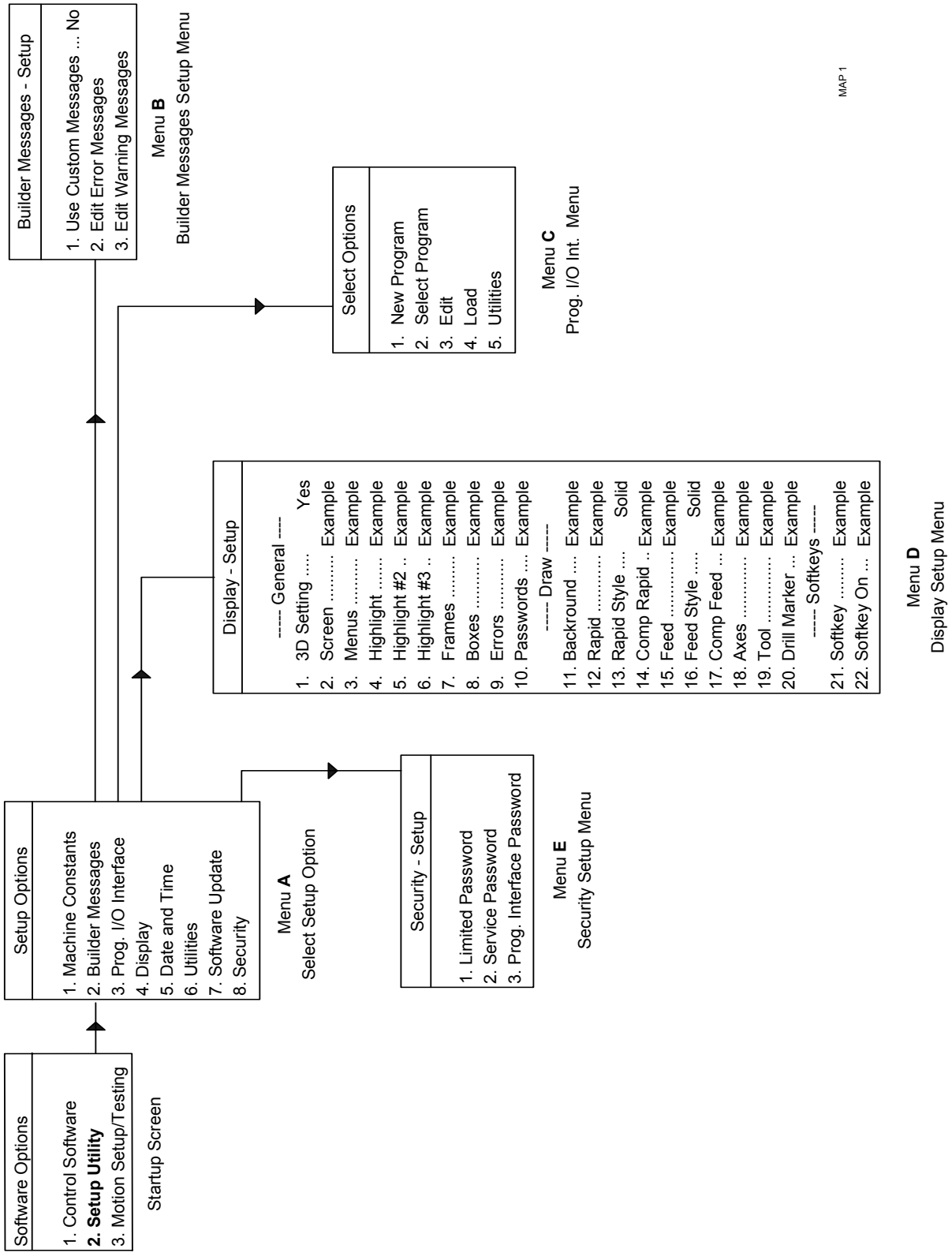
3-D Probe Displays the state of the trigger signals (deflected or not) in the X12 connector. Connector X12 corresponds to the 3-D Probe.
Values: [0 (probe deflected) or 1 (not deflected)]

T Probe Displays the state of the trigger signals (deflected or not) in the X13 connector. Connector X13 corresponds to the Touch Probe.
Values: [0 (probe deflected) or 1 (not deflected)]

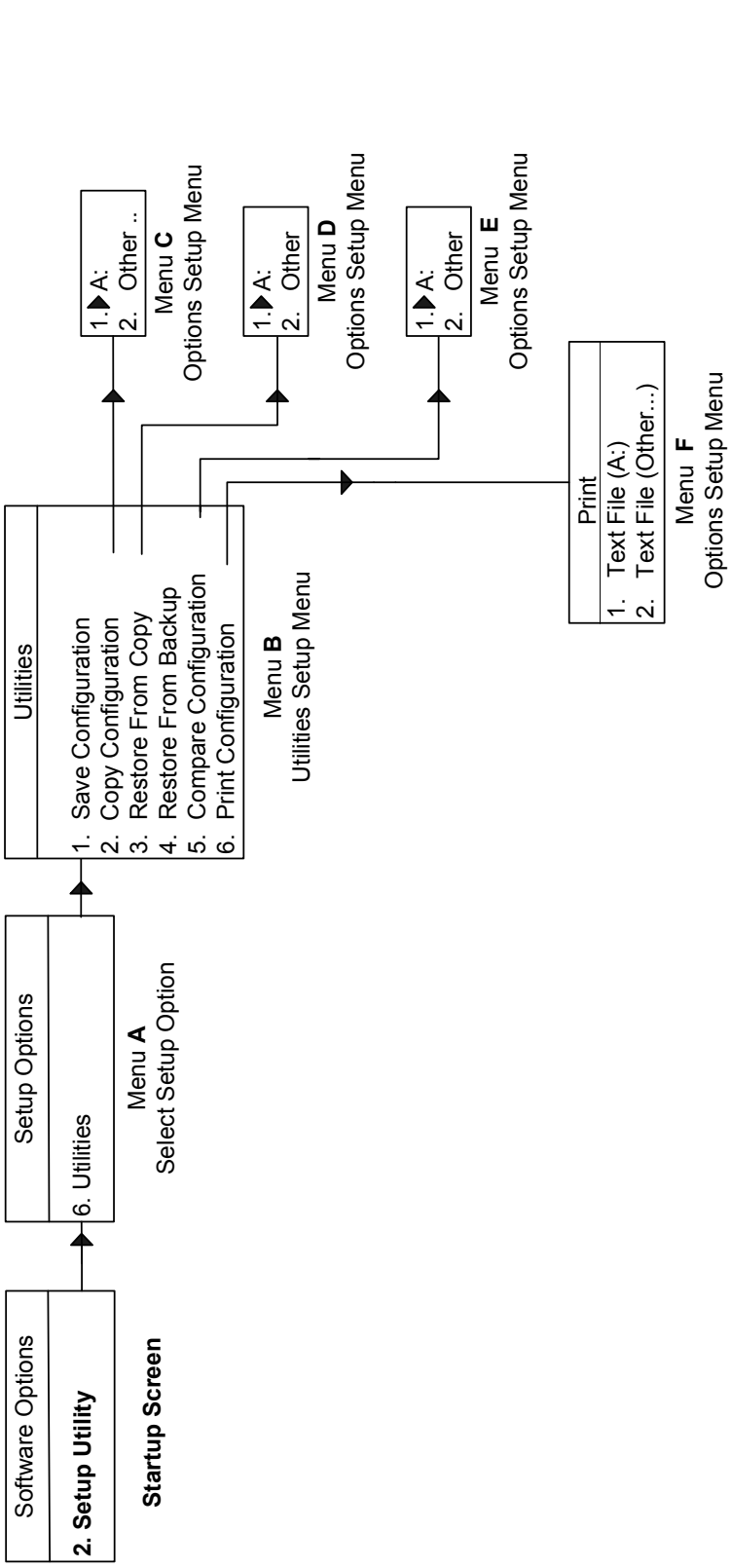
This test may be activated at the same time that the Index Pulse test **IndexP** (**F3**). This provides the marker pulse of the spindle motor and displays the state of the spindle proximity switch (X30). In order to have both tests active, the Index Pulse test must be activated first.

Section 6 - Setup Utility Maps

This section contains maps referenced in “Sections 1–4.” Refer to the Sections for instructions on how to use the software and maps.



Map 1



Map 2

A

B

C

D

E

F

G

H

V

Z

ANILAM

U.S.A.
ANILAM
One Precision Way
Jamestown, NY 14701
☎ (716) 661-1899
FAX (716) 661-1884
✉ anilaminc@anilam.com

ANILAM, CA
16312 Garfield Ave., Unit B
Paramount, CA 90723
☎ (562) 408-3334
FAX (562) 634-5459
✉ anilamla@anilam.com

Dial "011" before each number when calling from the U.S.A.

France
ANILAM S.A.R.L.
2 Ave de la Cristallerie
B.P. 68-92316
Serves Cedex, France
☎ +33-1-46290061
FAX +33-1-45072402
✉ courrier@acu-rite.fr

Germany
ANILAM GmbH
Fraunhoferstrasse 1
D-83301 Traunreut
Germany
☎ +49 8669 856110
FAX +49 8669 850930
✉ info@anilam.de

Italy
ANILAM Elettronica s.r.l.
10043 Orbassano
Strada Borgaretto 38
Torino, Italy
☎ +39 011 900 2606
FAX +39 011 900 2466
✉ info@anilam.it

Taiwan
ANILAM, TW
No. 246 Chau-Fu Road
Taichung City 407
Taiwan, ROC
☎ +886-4 225 87222
FAX +886-4 225 87260
✉ anilamtw@anilam.com

United Kingdom
ACI (UK) Limited
16 Plover Close, Interchange Park
Newport Pagnell
Buckinghamshire, MK16 9PS
England
☎ +44 (0) 1908 514 500
FAX +44 (0) 1908 610 111
✉ sales@aciuk.co.uk

China
Acu-Rite Companies Inc.(Shanghai Representative Office)
Room 1986, Tower B
City Center of Shanghai
No. 100 Zunyi Lu Road
Chang Ning District
200051 Shanghai P.R.C.
☎ +86 21 62370398
FAX +86 21 62372320
✉ china@anilam.com

P/N 70000490C
10-December-04