ANILAM

5000M 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. ACU-RITE Companies, Inc. reserves the right to make changes to improve reliability, function, or design without notice. ACU-RITE Companies, Inc. assumes no liability arising out of the application or use of the product described herein.

Section 1 - Setup Utility Concepts

Section 2 - Builder Setup

Section 3 - Operator Setup

Section 4 - Configuration Utilities

Section 5 - Fine-Tuning Systems with Linear Encoders

Section 6 - Setup Utility Maps

Section 1 - Setup Utility Concepts

Introduction

The Setup Utility is used to configure the 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 P5MCFG.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
5000M-3X	5000M Three Axes System	3
5000M-4X	5000M Four Axes System	4
5000M-5X	5000M Five Axes System	5

NOTE: All systems also support one spindle axis.

As far as the Setup Utility is concerned, everything that applies to 5000M Three Axes Systems also applies to 5000M Four Axes Systems. Similarly, everything that applies to 5000M Four Axes Systems also applies to 5000M Five Axes Systems. The main difference between the products is in the number of axes supported. Refer to **Table 1-1** for specification on number of axes per product. In addition, all systems support one spindle axis.

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

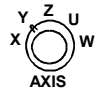
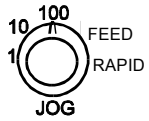
Axis Keys

Some parameters require that you specify an axis. Use the **X**, **Y**, **Z**, **U**, **W**, or **S** 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, (P5MCFG.CFG). Every time a parameter is changed, the configuration file is saved; the CNC automatically creates a backup file, (P5MCFG.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:

1. Highlight the menu in which the parameter is displayed, 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**, clears 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:

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.

Overview of Main Parameter Categories

There are, in general, two categories of parameters. One category of parameters corresponds to the type of parameters that the machine builders, or technicians, would normally be involved in specifying. These parameters are all under the Builder Setup menu entry. The second category of parameters corresponds to those that the CNC operator or programmer would be involved in specifying or customizing. These parameters are all under the Operator Setup menu entry. See Map 1**, Menu A**.

In general, Builder parameters require Service or Limited level password and Operator parameters do not require any password.

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).

To set the default measurement mode:

1. See Map 1**, Menu A**. Highlight **Units in Inch**.
2. Press **ENTER** to toggle between inch mode and millimeter mode. [Default: **Inch**]

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

5000M-4X
5000M-5X

The only exception to this rule are dimensional parameters corresponding to rotary axes. If any of the auxiliary axes (i.e., U or W) are configured as a rotary axis, then the unit is always in degrees or degrees per minute (i.e., deg/min).

Section 2 - Builder Setup

General Axis refers to the **X, Y**, and **Z**-axes. Enter the basic operating specifications for these axes into the CNC via the **Builder Setup – General Axis** menus. These menus allow you to configure the X, Y, and Z-axis in the CNC.

> **NOTE:** Some of the menus also provide a parameter for the S-axis (i.e., Spindle). It is only necessary to enter S-axis parameters if the machine is capable of operating the spindle as a controlled axis.

5000M-4X The menus for 5000M Four Axes will allow setting of parameters for axes X, Y, Z, and U.

5000M-5X The menus for 5000M Five Axes will allow setting of parameters for axes X, Y, Z, U, and W.

System Resolution

The CNC receives feedback from a linear encoder, rotary encoder, or an **EverTrack™ encoder and provides closed-loop positioning for the system. You can set resolution for the system (whether the feedback device is a rotary encoder or a linear encoder) via the **Resolution** menu.

Different types of feedback devices can be used in one system. For example, you can use a linear encoder on the X-axis and a rotary encoder on the Y-axis. Some of the parameters are specific to rotary encoders or linear encoders.

Setting Axes for Encoder Type

To set the axis encoder type: [Default: **Rotary Encoder**]

1. See the Resolution Setup, Menu G on "Map 1" in "Section 6 - Setup Utility Maps." For the remainder of the document, this is described as "See Map 1, **Resolution Setup, Menu G**."
2. Highlight **Type**.
3. Press **X**, **Y**, **Z**, **U**, or **W** for the axis being set, and press **ENTER**. A pop-up window displays the following selections:
   - **Linear Encoder**
   - **Rotary Encoder**
   - **EverTrack Encoder**
4. Highlight **Linear Encoder** for axes that use a linear encoder; highlight **Rotary Encoder** for axes that use a rotary encoder; highlight **EverTrack Encoder** for axes with linear encoders that have the EverTrack feature, and press **ENTER**. The CNC changes the encoder type to the selected option.

- - - - -

**EverTrack™ EverTrack™ is a trademark of ACU-RITE Companies, Inc.

Setting the Display Resolution

Display Resolution allows you to set the resolution of the axis display (**Display Res**). 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.

[Default: **2 Microns**] (0.002 mm/0.0001 inch)

To set display resolution:

1. See Map 1**, Menu G**. Highlight **Display Res**.
2. Press the appropriate axis key (i.e., **X**, **Y**, **Z**, **U**, or **W**).

   The CNC displays a pop-up window with the choices: 0.5, 1, 2, 5, or 10 Microns.
3. Highlight the desired display resolution, and press **ENTER**.

   The axis display will show movement at the selected resolution.

Setting the Linear Encoder Resolution

NOTE: This parameter applies only to linear encoders. Do not use if the axis is using a rotary encoder for feedback.

Enter the required resolution for each axis: X, Y, and Z. Always select resolution in microns, regardless of whether the CNC is in Inch Mode or MM Mode. [Default: **1 Micron**] (0.001 mm/0.00005 inch)

Refer to **Table 2-1** for ANILAM conversion values.

Table 2-1, Micron to Inch Conversion

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

NOTE: If resolution settings do not match those of the installed equipment, positioning errors will occur.

To set the linear encoder resolution:

1. See Map 1**, Menu G**.
2. Highlight **Linear Enc Res**
3. Press the appropriate axis key (i.e., **X**, **Y**, **Z**, **U**, or **W**). A pop-up window will display the following choices: 0.5, 1, 2, 5, or 10 Micron.
4. Highlight the appropriate linear encoder resolution, and press **ENTER**.

Setting Line Count for Rotary Encoder

> **NOTE:** This parameter applies only to rotary encoders. Do not use if the axis is using a linear encoder for feedback.

Enter the number of counts per revolution supported by the rotary encoder (**Rot Enc Lines**). The CNC accepts line counts of up to 10,000 lines per revolution.
[Defaults: **1000 lines** for X, Y, Z, U, or W]

[Defaults for AC Brushless systems only: **1024 lines** for X, Y, Z, U, or W]

To enter an encoder line count:

1. See Map 1**, Menu G**. Highlight **Rot Enc Lines**.
2. Press the appropriate axis key (i.e., **X**, **Y**, **Z**, **U**, or **W**).

   The CNC highlights the encoder line entry field for the axis.
3. Type the rotary encoder line count, and press **ENTER**.

Setting Ballscrew Pitch for the Rotary Encoder

> **NOTE:** This parameter applies only to rotary encoders. Do not use if the axis is using a linear encoder for feedback.

Pitch is the linear distance traveled per revolution of the ballscrew. Use the unit of measurement (inch or mm) to which the CNC defaults.

Set the pitch (**Bscrew Pitch**) of the ballscrew. [Default: **0.20 inch**]

To enter ballscrew pitch:

1. See Map 1**, Menu G**. Highlight **Bscrew Pitch**.
2. Press the appropriate axis key (i.e., **X**, **Y**, **Z**, **U**, or **W**). The CNC highlights the ballscrew pitch entry field for the axis.
3. Type the pitch of the ballscrew for that axis, and press **ENTER**.

Setting the Ratio Between the Ballscrew Pulley and the Motor/Encoder Pulley

The Ratio is the difference in the size of the pulleys, which represent the number of turns of the Encoder relative to the number of rotations of the Ballscrew.

Most encoders today are mounted to the shafts to the motors; therefore, the parameter for Motor Pulley represents the encoder pulley. If your encoder is not mounted to the motor shaft, then the correct entry for the Motor Pulley would be the actual encoder pulley.

For example, if the pulley on the ballscrew has 21 teeth, and the pulley on the motor has 14 teeth: the ratio is 1.5 to 1. You would enter 1.5 for the Ballscrew Pulley parameter, and 1 for the Motor Pulley parameter. If you do not know the actual ratio, you enter the number of teeth on the pulleys: 21 for Ballscrew Pulley, and 14 for the Motor Pulley.

Ballscrew Pulley Parameter

To enter the Ballscrew Pulley value:

1. See Map 1**, Menu G**. Highlight **Ratio (Bsc Ply)**.
2. Press the appropriate axis key (i.e., **X**, **Y**, **Z**, **U**, or **W**). The CNC highlights the value entry field for the axis.
3. Type the number of teeth on the Ballscrew pulley (or the Numerator of the ratio), and press **ENTER**.

[Defaults: X **1.50000**, Y **1.50000**, Z **1.80000**, U **1.00000**, and W-axis **1.00000**]

[Defaults for AC Brushless systems only: X **2.00000**, Y **2.00000**, Z **1.00000**, U **1.00000**, and W-axis **1.00000**]

NOTE: This parameter applies only to rotary encoders. Do not use it with a linear encoder.

Motor/Encoder Pulley Parameter

To enter the Motor/Encoder Pulley value:

1. See Map 1**, Menu G**. Highlight **Ratio (Mtr Ply)**.
2. Press the appropriate axis key (i.e., **X**, **Y**, **Z**, **U**, or **W**). The CNC highlights the value entry field for the axis.
3. Type the number of teeth on the Motor/Encoder pulley (or the Denominator of the ratio), and press **ENTER**.

[Defaults: X, Y, Z, U, and W-axis **1.00000**]

NOTE: This parameter applies only to rotary encoders. Do not use it with a linear encoder.

Setting the Starting Mark

> **NOTE:** This parameter applies only to EverTrack encoders. Do not use if the axis is using a rotary encoder for feedback.

The Starting Mark entry is the first mark from the right-most end of the encoder (as you look at the encoder). The entry is sign sensitive. If the right-most mark is at the positive end of the axis, then the Starting Mark must be positive. If the right-most mark is at the negative end of the axis, then the Starting Mark must be negative.
[Defaults: X, Y, Z, U, and W-axis **0**]

To determine the starting mark number, refer to "Starting Reference Mark" in *5000M CNC Motion Setup/Testing Utility*, P/N 70000636, for a description using Machine Setup & Testing (MST) to find the Starting Reference Mark.

If you know the starting mark number, use the following procedure to set the Starting Mark. To set the Starting Mark:

1. See Map 1**, Menu G**. Highlight **Starting Mark**.
2. Press the appropriate axis key (i.e., **X**, **Y**, **Z**, **U**, or **W**). The CNC highlights the entry field for the axis.
3. Type the starting mark for that axis, and press **ENTER**.

Setting 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.
[Default: **Off** (disabled)]

To set linear correction compensation:

1. See Map 1**, Menu F**. Highlight the menu selection that corresponds to the axis being set, and press **ENTER**.

   The CNC highlights the entry field for the selected axis.

2. Type the desired linear compensation correction, and press **ENTER**.
3. Highlight **Linear correction compensation**, and press **ENTER**.

   This selection activates/deactivates the option.

4. Press **ENTER** to toggle the selection On/Off to activate/deactivate the compensation value(s) entered.

Setting In-Position Check

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: **0.0004 in.** for X, Y, and Z; **0.0102 in.** for U and W]

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.

To define in-position tolerance:

1. See Map 1**, Menu E**. Highlight the menu selection that corresponds to the axis being set, and press **ENTER**.

   The CNC highlights the entry field for the axis.

2. Type the desired in-position tolerance, and press **ENTER**.

Setting Continuous Path

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. [Default: **0.0700 in.** for X, Y, and Z axes with Continuous Path turned On; **1.7780 in.** for U and W axes]

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

To activate and define the continuous path tolerance:

1. See Map 1**, Menu D**. Highlight the menu selection that corresponds to the axis being set, and press **ENTER**.

   The CNC highlights the entry field for the axis.

2. Type the desired tolerance, and press **ENTER**.

3. Highlight **Continuous path**, and press **ENTER**.

   This selection activates/deactivates the option.

4. Press **ENTER** to toggle the selection On/Off to activate/deactivate the Continuous Path Mode(s) entered.

   The CNC activates the **Continuous Path Mode**(s) for all selected axes.

Setting 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 (i.e., 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.
[Default: **X = 200 in/min**; **Y = 200 in/min**; **Z = 150in/min**; **U** and **W = 5,080 in/min**]

To set the default rapid speed:

1. See Map 2**, Menu E**. Highlight the menu selection that corresponds to the axis being set, and press **ENTER**.

   The CNC highlights the entry field for the axis.

2. Type the desired maximum default rapid rate, and press **ENTER**.

Setting Axis Default Feed Rate

Setting the Default Feed Rate establishes a default feedrate for each axis, wherever a feedrate has not been programmed. This applies to programmed blocks or MDI commands. Jog moves in feed (i.e., from a manual panel) can have a different feedrate.
[Default: **X, Y, Z= 10 in/min**, **U, W = 254.0 in/min**]

To set the axis default feed rate for an axis:

1. See Map 2**, Menu F**. Highlight the menu selection that corresponds to the axis being set, and press **ENTER**.

   The CNC highlights the entry field for the axis.

2. Type the desired axis default feedrate, and press **ENTER**.

Setting Software Limits

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

You can set positive and negative software limits to restrict travel range.

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.)

Enter positive and negative software limits separately for each axis. [Default: **Off** (disabled)]

Setting Positive Software Limits

To set Positive Software Limits:

1. See Map 2**, Menu C**. Highlight **Software limits**, and press **ENTER**.

   The CNC displays the Software Limit Setup Menu (**Menu G**).

2. Highlight **Positive software limit**, and press **ENTER**.

   The CNC displays the Positive Software Limit Setup Menu (**Menu H**).

3. Highlight the menu selection that corresponds to the axis being set, and press **ENTER**.

   The CNC highlights the entry field for the axis.

4. Type the desired positive software limit, and press **ENTER**.

Setting Negative Software Limits

To set Negative Software Limits:

1. See Map 2**, Menu C**. Highlight **Software limits**, and press **ENTER**.

   The CNC displays the Software Limit Setup Menu (**Menu G**).

2. Highlight **Negative software limit**, and press **ENTER**.

   The CNC displays the Negative Software Limit Setup Menu (**Menu I**).

3. Highlight the menu selection that corresponds to the axis being set, and press **ENTER**.

   The CNC highlights the entry field for the axis.

4. Type the desired negative software limit, and press **ENTER**.

Enabling Software Limits

To enable Software Limits:

1. In Map 2, **Menu G**, highlight **Software limits**.
2. Press **ENTER** to toggle the Software Limits **On**.

   The CNC enables software limits. [Default: **Off** (disabled)]

Enabling Vector Limits

Vector limit switches, also called directional limit switches, define the CNC's hardware travel limits. If installed, vector limits must be enabled in the Setup Utility for each axis that has a vector limit. Once you enable the vector limits for an axis, the CNC prohibits machine motion in that direction beyond the limit switch.
[Default: **Disable** for all axes]

Caution: Directional limit switches must be wired normally closed. No other configuration ensures proper and safe machine operation.

Vector Limit Switches restrictions follow:

- Must be normally closed switches.
- Must be on CAN Node 0
- Are hard-coded (See **Table 2-2** and Table 2-3, Vector Limit Inputs for 5000M 4-Axes and 5000M 5-Axes.)
- Cannot be used as general purpose I/O
- Both directions must be wired for assigned axes
- Can also be used as Home Switches

Only CAN Node 0 will accept vector limit inputs. A vector limit input port must not be used for another input function.

Refer to **Table 2-2** for vector limit input port assignments.

Table 2-2, CAN Bus Node 0 Vector Limit Input Port Assignments

Input Port	Pin	Assigned Vector Limit
0	0	X +
1	1	X -
2	2	Y +
3	3	Y -
4	4	Z +
5	5	Z -

5000M-4X
5000M-5X

Refer to **Table 2-3** for additional vector limits inputs in 5000M 4-Axes and 5000M 5-Axes.

Table 2-3, Vector Limit Inputs for 5000M 4-Axes and 5000M 5-Axes

Input Port	Pin	Assigned Vector Limit
6	6	U +
7	7	U -
8	8	W +
9	9	W -

To enable vector limits:

1. See Map 2**, Menu J**. Highlight the menu selection that corresponds to the axis being set.
2. Press **ENTER** to toggle the selection (**Enable**/**Disable**) to activate/deactivate the vector limits.

NOTE: If vector limits are enabled, a signal must be wired to both the positive and negative direction inputs assigned to the axis. Otherwise, the CNC will inhibit motion in the direction (positive or negative) of the unwired input.

When the vector limits for an axis are set, the assigned inputs cannot be used for input functions. Refer to Table 2-2, CAN Bus Node 0 Vector Limit Input Port Assignments and **Table 2-3** for vector limit input port assignments.

Vector/Home Limit Switch Connections

NOTE: Home switches are wired to the same ports as the vector limits. Wire the home switch to the input that corresponds to the direction that you select for each axis.

Home Switches restrictions follow:

- Must be normally closed switches.
- Must be on CAN Node 0
- Are hard-coded (See Table 2-2, CAN Bus Node 0 Vector Limit Input Port Assignments and **Table 2-3**.)
- Cannot be used as general purpose I/O
- Selected direction only must be wired for assigned axes
- Can also be used as Vector Limit Switches

Setting Encoder Phase to Correct Axis Direction Displayed

Moving an 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. [Default: **Not Invert**]

NOTE: This is the only way to change the direction of the count without making hardware changes.

To adjust the Encoder Phase Setting:

1. See Map 2**, Menu K**. Highlight the menu selection that corresponds to the axis being set.
2. Press **ENTER** to toggle the setting (Not Invert/ Invert). Change the Phase setting to invert the direction of count for the appropriate axis.

Setting 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.

Measure backlash and store the value in the Setup Utility. When backlash compensation activates, the CNC automatically calculates the necessary motion corrections. [Default: **Off** (disabled)]

To activate and define backlash compensation for an axis:

1. See Map 2**, Menu D**. Highlight the menu selection that corresponds to the axis being set, and press **ENTER**.

   The CNC highlights the entry field for the axis.
2. Type the desired backlash compensation, and press **ENTER**.
3. In **Menu D**, highlight **Backlash compensation**.

   This selection activates/deactivates backlash compensation.
4. Press **ENTER** to toggle the selection On/Off to activate/deactivate the backlash compensation.

   The CNC activates the backlash compensation for all selected axes.

Setting Ballscrew Compensation

NOTE: 1. Use ballscrew compensation with the Automatic File Loader.
2. Perform a Machine Home sequence before you enable ballscrew compensation

The CNC can compensate for inaccuracies along the ballscrew. This ensures a high degree of precision in the finished workpiece.

Ballscrew Compensation allows the ballscrew to be divided into as many as 128 segments per axis for calibration. Segment length is constant for all segments. [Default: **No** (Active parameter set to No)]

To set the Ballscrew Compensation:

1. See Map 3**, Menu C**. Highlight **Ballscrew compensation**, and press **ENTER**.

   The CNC displays Ballscrew Compensation Menu (Map 3**, Menu D**).

Setting Number of Segments

The machine builder can specify as many as 128 equally sized segments per axis. To determine the number of segments required, consider that the number of segments multiplied by the segment size should equal the entire range of travel for the axis being set.

To set the Number of Segments:

1. Highlight **Number of segments** in the Ballscrew Compensation Setup Menu (Map 3**, Menu D**), and press **ENTER**.

   The CNC displays Number of Segments Setup Menu (Map 3**, Menu E**).

2. Highlight the menu item pertaining to the axis being set, and press **ENTER**.

   The entry field for the selected axis highlights.

3. Type the number of segments desired for that axis, and press **ENTER**.

   Repeat the procedure for all axes being set.

Setting Table Entries

> **NOTE:** Refer to "Automatic File Loader" for details on how to enter values from a file.

Determine the amount of compensation required for each segment along an axis. Use a laser to make these measurements.

The compensation value is the difference between the desired positive or negative position commanded by the CNC and the actual position measured by the laser. Record the compensation required for each segment in the Table Entries Menu (**Menu F**).

The length of the table equals the largest number of entries assigned to any axis. If X requires 13 segments and Z requires 9 segments, then the table will be 13 lines long.

To enter Table entries manually:

1. See Map 3**, Menu F**. Press **X**, **Y**, **Z**, **U**, or **W** to set the appropriate axis.

   The CNC highlights the entry field for the selected axis.

2. Type the desired compensation for each segment assigned to the axis, and press **ENTER**.

   The CNC accepts the entered values.

Setting Offset and Zero Cross Parameters

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 –0.01 mm and the Zero cross is –6.00 mm, then the CNC begins the compensated (lasered) area –6.01 mm 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.

Offset

To set the Ballscrew Offset parameter:

1. See Map 3**, Menu G**. Highlight the menu selection corresponding to the axis being set, and press **ENTER**.

   The CNC highlights the entry field for that axis.

2. Type the appropriate Ballscrew Offset for that axis. If the Offset location is Machine Home, enter **0.00000**. The ballscrew offset is measured from Machine Home.

Zero Cross

To set the Ballscrew Zero Cross parameter:

1. See Map 3, **Menu H**. Highlight the menu selection corresponding to the axis being set, and press **ENTER**.

   The CNC highlights the entry field for that axis.

2. Type the appropriate Zero Cross parameter for that axis. If the Zero Cross parameter is at Machine Home, enter **0.00000**. All entered values are referenced to Machine Home.

Setting 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.

To set the Segment Length for Ballscrew Compensation:

1. See Map 3, **Menu I**. Highlight the menu selection corresponding to the axis being set, and press **ENTER**.

   The CNC highlights the entry field for that axis.

2. Enter the desired segment length for that axis. (This value is a negative number for the negative travel direction with respect to the Machine Home position.)

Activating Ballscrew Compensation

To activate Ballscrew Compensation:

1. See Map 3, **Menu D**. Highlight **Active**.

2. Press **ENTER** to switch between **Yes** and **No**. [Default: **No**]

Automatic File Loader

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:

1. See Map 3, **Menu F**. Press **LdFile** (**F8**). Type the appropriate password, if required by the system.

The CNC displays the **Ballscrew compensation file loader** menu. See **Figure 2-1**. Refer to Table 2-4, Ballscrew File Loader Parameters for a description of the Ballscrew File Loader Parameters.

2. Highlight **Starting segment**. Type the segment number of the first table entry, and press **ENTER**.
3. Highlight **Ending segment**. Type the segment number for the last table entry, and press **ENTER**.
4. Highlight **Axis**, and press **ENTER**.

   The CNC displays a pop-up menu with the following choices: **X**, **Y**, **Z**, **U**, **V**, or **W**.
5. Highlight the desired axis, and press **ENTER**.

   The CNC returns to the **Ballscrew compensation file loader** menu.
6. Highlight **Action**, and press **ENTER**.
7. Highlight an option in the pop-up menu with the following options: **Overwrite**, **Add**, or **Average**, 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.

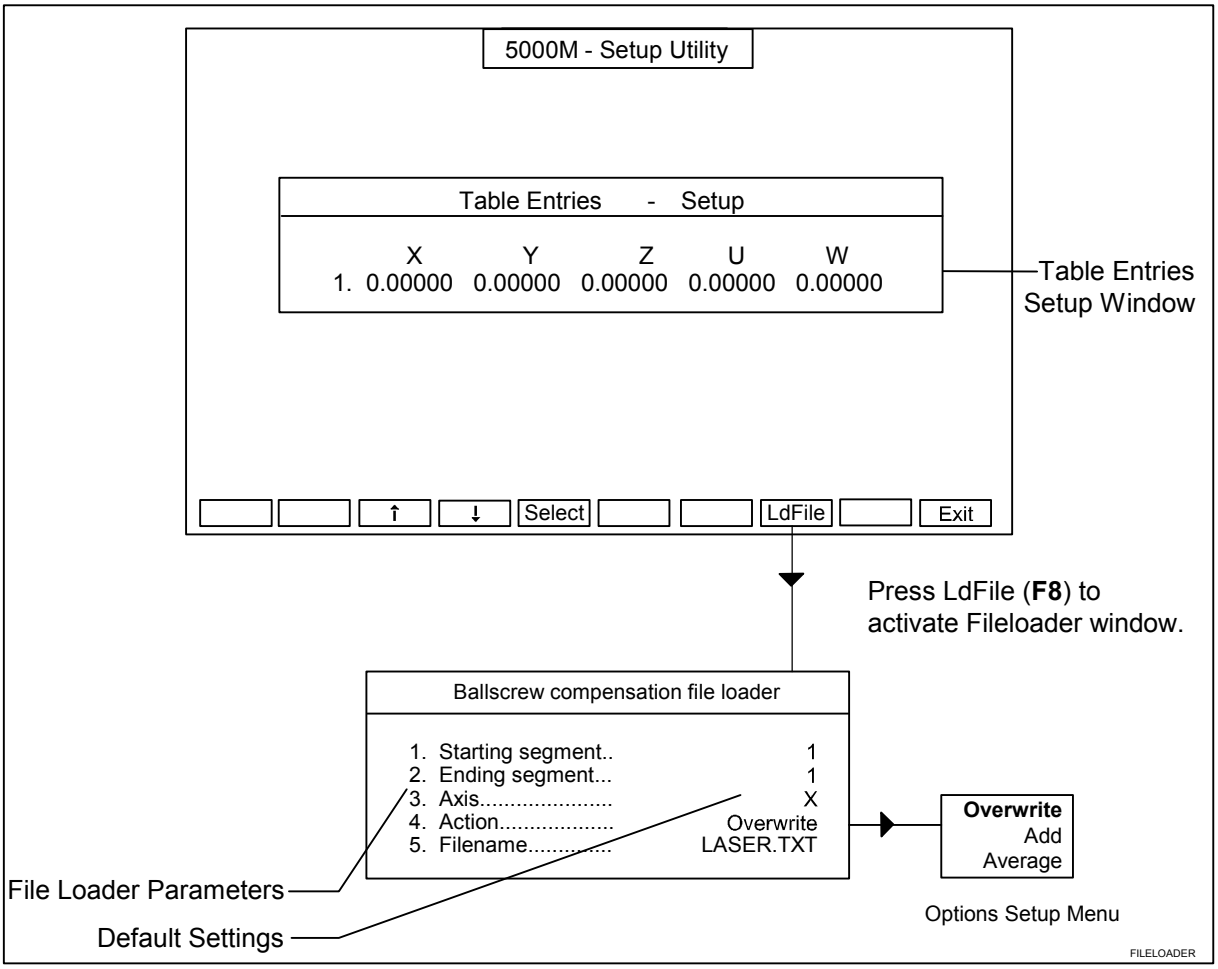


Figure 2-1, Ballscrew Compensation File Loader Menu

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 (max. 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. Valid range: (1 to 128)
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. Valid range: (1 to 128)
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 ❑ Be averaged with the existing data **Overwrite** will clear any values in the table beyond the segment limit for the axis. **Add** and **Average** replace only 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

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 (Map 3**, Menu F**).

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 (Refer to Map 3**, Menu D**) 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 type 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.

Jog Position

Jog position defines a point on the machine 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.

[Default: **0.0000**]

To set the Jog and Return Position in X, Y, Z, U, or W:

1. See Map 3**, Menu J**. Highlight the axis being set, and press **ENTER**.
2. In the highlighted entry field, enter the appropriate coordinate for the selected axis.

Axis Clamping

The control can clamp machine axes that are not in motion during machining.

Switch the **Clamping Allowed** option to enable (**Yes**) or disable (**No**) the clamping feature. [Default: disabled (**No**)]

Wait For Finish determines whether an external Finish signal is required upon the completion of clamping operations. Switch Wait for Finish to **Yes** to require a Finish pulse or to **No** if a Finish pulse is not required.

NOTE: Refer to the Table 2-12, CNC Input Functions for details on the external Finish signal.

Clamping must be programmed via M-Code in the user program. Refer to **Table 2-5**.

Table 2-5, Axis Clamping M-Code

M-Code	Format	Description
M9246	M9246 Xx Yy Zz Uu Vv Ww	Activates clamping for given axes.

When a machine axis is clamped, the servo drive output is disabled. The clamping mechanism is engaged and an error in position is incurred. Specify the maximum allowable error with the **Allowable Error** option.

To enter the Allowable Error:

1. Go to Map 4**, Menu Q**. Highlight **Allowable Error**, and press **ENTER**.
2. Type the Allowable Error.

Outputs available from the PC I/O Controller Board are used to disable the various servo drives during clamping. These outputs are specified with the **Axis Ports** option.

To disable an axis servo drive for clamping:

1. Refer to Map 4**, Menu R**. Highlight the axis for which outputs will be assigned, and press **ENTER**. The Axis Outputs Menu (**Menu S**) activates.
2. Choose the output required to disable the servo drive of the selected axis.

NOTE: The I/O Controller Board has eight (8) available auxiliary outputs. It is possible to assign more than one function to a single output accidentally. The user should verify that each output is assigned only one function.

Setting Axis Ports

This parameter allows you to assign an axis port on the OEM CNC chassis to a specific axis on the machine. Refer to Map 4, **Menu D**. Refer to **Table 2-6** for default axis port assignments. Normally, all axis ports are used. However, during machine setup or troubleshooting it might be necessary to disable an axis.

Table 2-6, Axis Ports for 5000M Three Axes

Port	Axis
0	X
1	Y
2	Z
3	S

To change the **Axis Ports** Parameters:

1. See Map 4, **Axis Ports, Menu D**.
2. Highlight the appropriate port, and press **ENTER**.
3. Highlight Disabled or the appropriate axis, and press **ENTER**. Refer to Map 4, **Menu E**.

Repeat this procedure for all axes.

5000M-4X

Refer to **Table 2-7** for axis port assignments on 5000M Four Axes.

Table 2-7, Axis Ports for 5000M Four Axes

Port	Axis
0	X
1	Y
2	Z
3	U
4	S

Refer to Map 5, **Menu D** and **Menu E**.

5000M-5X

Refer to **Table 2-8** for axis port assignments on 5000M Five Axes.

Table 2-8, Axis Ports for 5000M Five Axes

Port	Axis
0	X
1	Y
2	Z
3	U
4	S
5	W

Refer to Map 5**, Menu D** and **Menu E**.

Setting Feed, Rapid, and No-Motion Filter Parameters

These parameters enable tuning customized to the output of the combination of servos, motors and feedback devices on a specific installation.

Three Setup Utility menus affect the gain of each axis:

- Feed Filter Parameters Menu
- Rapid Filter Parameters Menu
- No Motion (or Holding) Filter Parameters Menu
- Rigid Tapping Filters Menu

These setup menus allow the operator to set a gain value for Feed moves, which require greater accuracy than Rapid moves. In Rapid Mode, machine inertia, available servo drive output power, and other mechanical factors must be considered. The No Motion gain values control the gain of the axes when the machine is holding position.

An understanding of motion control theory is required to change these values properly.

When the CNC commands a move, (n), the output from the system is a digital word, u' (n), representation of that move.

This digital word is derived from the output of the interpolators, which create a move-required value based upon current position of the axis, the geometric equations required to make the move, and the effect of any required compensation parameters. This move-required value is fed to the Feedforward Controller and the Digital Proportional, Integral, and Derivative Gain (PID) Filter, so that the output (digital word, u' (n)) can be defined by the following equation:

Output = Voltage Offset + (Kp + Ki + Kd)

Refer to Table 2-9, System Output Values and Definitions for an explanation of parameters.

To change the **Feed Filter** parameters:

1. See Map 6**, Feed Filter Parameters, Menu E**.
2. Highlight the PID parameter being set.
3. Press the axis key for the axis being set. The entry field for the axis highlights.
4. Type the appropriate value for the parameter, and press **ENTER**.
5. Repeat this procedure for all axes and parameters being set. [Defaults: Kp, default is **15.000**. Ki default is **0.000**. Kd default is **10.000**. Kf default is **0.000**. IL default is **0.000**. Ds default is **5**.]

To change the **Rapid Filter** parameters:

1. See Map 6**, Rapid Filter Parameters, Menu F**.
2. Highlight the PID parameter being set (**Kp**, **Ki**, **Kd**, **Kf**, **IL**, or **Ds**).
3. Press the axis key for the axis being set. The entry field for the axis highlights.
4. Type the appropriate value for the parameter, and press **ENTER**.
5. Repeat this procedure for all axes and parameters being set. [Defaults: Kp default is **10.000**. Ki default is **0.000**. Kd default is **10.000**. Kf default is **0.000**. IL default is **0.000**. Ds default is **2**.]

To change the **No Motion Filter** parameters:

1. See Map 6**, No Motion Filter Parameters, Menu G**.
2. Highlight the PID parameter being set (**Kp**, **Ki**, **Kd**, **Kf**, **IL**, or **Ds**).
3. Press the axis key for the axis being set. The entry field for the axis highlights.
4. Type the appropriate value for the parameter, and press **ENTER**.
5. Repeat this procedure for all axes and parameters that you set. [Defaults: For all axes, Kp default is **10.000**. Ki default is **5.000**. Kd default is **10.000**. Kf default is **0.000**. IL default is **10.000**. Ds default is **5**.]

To change the **Rigid Tapping Filters** parameters for the Z-axis:

1. See Map 6, **Rigid Tapping Filters, Menu H**.
2. Highlight the PID parameter being set (**Kp**, **Kd**, **Kf**, or **Ds**).
3. Type the appropriate value for the parameter, and press **ENTER**.
4. Repeat this procedure for all parameters that you wish to change.
5. For the Enable parameter, press **ENTER** to toggle between **Yes** and **No**. Toggle to **Yes** to use the table values; otherwise, the default values are used.
   [Defaults: For Z-axis, Kp default is **0.000**. Kd default is **0.000**. Kf default is **0.000**. Ds default is **0.** Enable is **No**.]
   Valid range Kp, Kd, Kf, Ds: 0.000 to 300.000)

Table 2-9, System Output Values and Definitions

Value	Definition
Kp	Proportional Gain. This value is derived by directly multiplying the Kp coefficient by the position error. It is designed to compensate for immediate changes in servo error position.
Ki	Integral Gain. This value applies a long-term accumulation of error correction over time. It is used to ensure that the static position error is zero: 0-position error at rest or at constant velocity. Multiplying the Ki coefficient by the position error and then adding it to the previously computed Integral Gain value derives it.
Kd	Derivative Gain. This value reacts to a change in error over time. The derivative value is calculated by multiplying the Kd coefficient by the current error minus the error calculated in the previous sample.
Kf	Feedforward Gain. Feedforward gain is used to reduce the amount of lag (following error) that an axis generates during constant velocity.
IL	Integral Limit. The total maximum amount of Ki correction permitted by the Digital Filter. Ki gain effect is held to a preset maximum, (the IL term) which is the total maximum amount of Ki correction permitted by the Digital Filter.
Ds	Derivative Sampling Time. The rate at which the derivative gain (Kd) is applied.

NOTE: Refer to *5000M CNC Motion Setup/Testing Utility*, P/N 70000636, manual for documentation on using features that allow the CNC to automatically determine filter parameters. The Rigid Tapping Filters are not set automatically.

Setting Position Error Check Parameters

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.

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. Refer to **Table 2-10** for definitions of the PEC parameters.

Table 2-10, Position Error Check Parameters

Position Error Check Parameter	Definition
Maximum 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. [Default: **100.00 msec**]
Maximum lag error	The error distance allowed at rest or low feedrates, before declaring a fault. [Default: **0.0100**]
Check rapidrate	Ensures that machine is reaching its full-programmed rapid rate. If it does not, an Error Message displays. [Default: **Yes**]
Check feedrate	Ensures that machine is reaching its full-programmed feed rate. If it does not, an Error Message displays. [Default: **Yes**]
Enable error checking	Allows you to configure or disable PEC option for troubleshooting or comparison. [Default: **Yes**] **CAUTION: You must enable the PEC parameter for the CNC 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!

To change a PEC parameter:

1. See Map 6**, Menu I**. Highlight the PEC parameter for the axis to be changed, and press **ENTER**.

   The entry field for that parameter highlights.

2. Type the desired value in the entry field, and press **ENTER**.
3. For the Enable Error Checking parameter, press **ENTER** to toggle between **Yes** and **No**.

   The CNC activates the selected choice.

Setting Amplifier Tuning Rapids

These parameters enable amplifier tuning rapid rate on specific axes. This is the maximum speed that a specific axis can operate. The Amplifier is tuned for this speed. The actual Rapid used is specified under Default Rapids and should be less than or equal to the Amplifier Tuning Rapids.
[Default: **0.**] Valid range: (0. to 2,000.)

To change an Amplifier Tuning Rapids parameter:

1. See Map 6**, Menu J**. Highlight the line for the axis to be changed, and press **ENTER**.

   The entry field for that parameter highlights.

2. Type the desired value in the entry field, and press **ENTER**.
3. For the Amplifier Tuning Rapid Enable parameter, press **ENTER** to toggle between **Yes** and **No**.

Setting Digital Amplifier

These parameters enable the Digital Amplifiers on specific axes. When an axis is selected to have Digital Amplifiers, the communication port will be open/close automatically when functions pertaining to the Digital Amplifiers are selected. Refer to *5000M CNC Motion Setup/Testing Utility*, P/N 70000636.

To set the digital amplifier:

1. See Map 8**, Menu D**. Refer to **Table 2-11** for a description of the Digital Amplifier Parameters. Highlight **Active digital amplifiers**, and press **ENTER** to display Map 8**, Menu E**.
2. Highlight the axis you want to change, and press **ENTER** to display Map 8**, Menu F**.
3. Press **ENTER** to toggle between **Disable** and **Enable**.

Table 2-11, Digital Amplifier Parameters

Digital Amplifier Parameter	Definition
Active digital amplifiers	Enables digital amplifier interface for a specific axis. [Default: **Disable**]
Balance adjustment (mV)	Used to increase/decrease the steps when running the Balance test (up/down arrows). [Default: **0.5** mV] Valid range: (0.3 to 100.0 mV)
Signal Gain adjustment (%)	Used to increase/decrease the Gain Adjustment step when running Signal Gain test (up/down arrows). [Default: **0.10** %] Valid range: (0.01 to 2.00 %)
Compensation adjustment (%)	Used to increase/decrease the Compensation Adjustment step when running Signal Gain test (right/left arrows). [Default: **0.02** %] Valid range: (0.01 to 2.00 %)

Setting Invert DAC Output

The Digital Analog Converter (DAC) establishes the direction of the spindle. A positive voltage is spindle forward. To reverse the direction of the spindle, change the option to Yes. See Map 8, **Menu G**.
[Default: X, Y, Z, U, W - Invert DAC Output **No**]

U Axis Setup

5000M-4X

In addition to the primary axis (i.e., XYZ), the 5000M Four Axes includes an Auxiliary axis, the U-axis. Most auxiliary axis parameters are included under the **General Axis Setup** menu in the same menus where the primary axis parameters are defined.

Refer to **Table 2-12** and Map 9, **Menu C**. Use this menu to configure auxiliary axis specific parameters.

Table 2-12, Auxiliary Axis Specific Parameters

Parameter	Definition
Type	None (disabled), Linear, or Rotary. [Default: **None**]
Display	Set to Yes display position value; otherwise, set to No. [Default: **Yes**]
Reset rotary at 360	If type is rotary this parameter can be set to force a reset (i.e., set to 0) when the axis reaches 360 degrees. [Default: **Yes**]
Synchronize to XYZ	Allows the axis to be synchronized to XYZ. [Default: **No**]

U and W Axes Setup

5000M-5X

The 5000M Five Axes includes two auxiliary axes; namely U and W. The W-axis parameters are setup in the same way as the U-axis parameters described above. Refer to Map 9, **Menu E**.

Spindle Axis Setup Menu

Refer to Map 10, **Menu C**. Use these parameters to configure spindle settings and gear ranges.

Setting Spindle DC Output

NOTE: Set the spindle drive to accept DC voltage.

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 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.

1. See Map 10**, Menu C**.
2. Highlight **Spindle output**. Press **ENTER** to toggle the setting to either **Bipolar** or **Unipolar**. [Default: **Unipolar**]

Setting Spindle Gear Ranges

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. Refer to Map 10**, Menu C**.

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

NOTE: If Spindle Forward is active, the voltage is positive. If Spindle Reverse is active, the voltage is negative.

Defaults are as follows:

- Low/High setting for **M40** gear range is **50** and **6,000** RPM
- Low/High setting for **M41** gear range is **50** and **6,000** RPM
- Low/High setting for **M42** gear range is **165** and **501** RPM
- Low/High setting for **M43** gear range is **500** and **1,471** RPM
- Low/High setting for **M44** gear range is **1,470** and **4,640** RPM

To set the Spindle Gear Range:

1. See Map 10**, Menu C**. Highlight **Gear ranges used**.
2. Press **ENTER** to toggle the setting between **Single-M40** and **Multiple**. [Default: **Single-M40**]
3. Highlight **Low Setting for**... (select appropriate RPM setting for M41 through M44), and press **ENTER**.
4. Type the desired Low range RPM. Press **ENTER** to store the setting.
5. Highlight **High Setting for**... (select appropriate RPM selection for M41 through M44), and press **ENTER**.
6. Type the desired High Range RPM. Press **ENTER** to store the setting.
7. Set up all necessary gear changes.

Invert DAC in M40–M44 Gear Range

The Digital Analog Converter (DAC) establishes the direction of the spindle. A positive voltage is spindle forward. To reverse the direction of the spindle, change the option to Yes.
[Default: **No**]

M40–M44 Ratio (Spindle Pulley)

The number of teeth for the M40–M44 spindle pulley.
[Default: **1.0**]

M40–M44 Ratio (Motor Pulley)

The number of teeth for the M40–M44 motor pulley.
[Default: **1.0**]

Gear Change RPM

Gear Change RPM specifies the spindle speed at which a gear change is performed. [Default: **10**]

To set the Gear change RPM:

1. See Map 10**, Menu C**. Highlight **Gear change RPM**, and press **ENTER**.

   The entry field for the parameter highlights.

2. Type the desired number, and press **ENTER**.

Setting the Spindle Encoder Lines

For machines fitted with spindle encoders, you must set the spindle encoder line count.

The spindle axis is normally operated in open-loop mode. It is not necessary for the spindle to be fitted with an encoder to allow DC spindle programming. [Default: **1,024** lines per revolution]

To set the number of spindle encoder lines:

1. See Map 10**, Menu C**. Highlight **Spindle encoder lines**, and press **ENTER**.

   The entry field for the parameter highlights.

2. Type the number of encoder lines, and press **ENTER**.

Setting Spindle RPM Display

NOTE: This parameter affects only the displayed RPM value. It does not affect RPM or voltage output to the spindle.

Spindle RPM Display allows you to configure the CNC to display feedback from a spindle encoder (Feedback) or from a programmed RPM. [Default: **Feedback**]

To set the Spindle RPM Display:

1. See Map 10**, Menu C**. Highlight **Spindle RPM Display**.
2. Press **ENTER** to toggle the setting to either **Feedback** or **Program**. Switch to **Feedback** to configure the spindle RPM display for rotary encoder feedback. Switch to **Program** to configure the display to show the programmed RPM.

Checking Spindle During Gear Change

Checking the spindle during gear change allows you to either stop the spindle before you can change a gear (e.g., change from M41 to M42) or enables you to change gears without stopping the spindle. [Default: **Yes**]

To check the spindle during a gear change:

1. Refer to Map 10**, Menu C**. Highlight **Check spindle during gear change**.
2. Press **ENTER** to toggle the setting On (**Yes**) or Off (**No**).

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.

Stopping Program on Gear Change

Stopping the program on gear change allows you to configure the CNC to stop program execution during a gear change. 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. [Default: **No**]

To stop the program during a gear change:

1. Refer to Map 10**, Menu C**. Highlight **Stop program on gear c4ange**.
2. Press **ENTER** to toggle the setting to On (**Yes**) or Off (**No**).

Checking RPM to be Within Gear Range

Checking RPM to be Within Gear Range prevents you from designating an RPM outside the active gear range. 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. [Default: **Yes**]

To check the RPM gear range:

1. Refer to Map 10**, Menu C**. Highlight **Check RPM to be within gear range**.
2. Press **ENTER** to toggle the setting to On (**Yes**) or Off (**No**).

Stop/Start Spindle when Hold/Start Pressed

During Automatic operations (i.e., 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. 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. [Default: **No**]

NOTE: This feature also applies to External Hold and External Start input functions.

To enable this feature:

1. Refer to Map 10**, Menu C**. Highlight **Stop/Start spindle when Hold/Start pressed**.
2. Press **ENTER** to toggle the setting to On (**Yes**) or Off (**No**).

Stop Program and Spindle on Gear Range Error

If set to Yes, and an error occurs during a gear range operation (for example, spindle speed out of range): the spindle stops, the program that is running is halted, and an error message is displayed.
[Default: **Yes**]

Encoder Mounted on Motor

Set to **Yes** when the spindle encoder feedback is mounted on the spindle motor; otherwise, set to **No**.
[Default: **No**]

Spindle Zero Speed RPM Tolerance

This parameter is the tolerance value used by IPI register M28-ZEROSPD. M28-ZEROSPD is set to TRUE when RPM Feedback is less than or equal to this parameter.
[Default: **1**] Valid range: (0 to 100)

Spindle at Speed Percent

This parameter is used to set the IPI register M29-ATSPD. M29-ATSPD is to TRUE when RPM Feedback is at the percentage specified by the parameter. (i.e. percent of actual versus command RPM)
[Default: **90**] Valid range: (50 to 100)

Setting Basic I/O Interface

This section describes how to setup basic CNC I/O functions.

The exchange of control signals between machine devices and the control is governed by the system I/O Controller. Input signal implementation is a standard feature.

The specific properties of the signals exchanged are configured via the software settings made in the Setup Utility.

The System I/O Controller in all configurations governs input signals from machine devices to the control.

Setting the I/O interface Type

The interface type specifies how the CNC exchanges input and output signals with other devices. The CAN I/O type can be used for simple installations. For more advanced capability, use the ANILAM Integral Programmable Interface (IPI) setting to create a logic program that runs on the CNC.

[Default: **CAN I/O**]

To activate the I/O interface:

1. See Map 10**, Menu D**. Highlight **Type**. This setting activates the CAN I/O, allowing the exchange of input and output signals between the control and CAN I/O boards.
2. Press **ENTER** to display a pop-up menu with the following selections:

   Disabled Disables all I/O. The CNC and the machine will not exchange I/O signals.

   CAN I/O Activates CAN I/O interface type. Inputs from the machine generate function signals that are sent to the CNC via the CAN-Bus channel. Function signals from the CNC are sent to an I/O board (via the CAN-Bus protocol) to activate the required outputs.

   ANILAM IPI Activates the ANILAM Integral Programmable Intelligence (IPI). IPI accesses CNC registers and system flags to create sophisticated programs that control many machine functions. For example, use IPI to control the turret on a turning center or a tool changer on a machining center. For details, refer to the *Integral Programmable Intelligence User's Guide*, P/N 70000416.

3. Highlight one selection, and press **ENTER**.
   [Default: **CAN I/O**]

Setting Finish Pulse Timeout

NOTE: Press **E-STOP** and **SERVO RESET** to regain control of a CNC holding for a finish pulse.

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.

To set the Timeout:

1. See Map 10**, Menu D**. Highlight **Timeout**, and press **ENTER**.

   The **Timeout** entry field highlights.

2. Type the number of milliseconds, and press **ENTER**.

   [Default: **10,000** (10 sec)] Valid range: (0 to 600,000 milliseconds) A zero (0) entry causes an indefinite hold.

Issuing Spindle Stop on Servo Fault or Emergency Stop

Issuing a spindle stop allows you to stop the spindle when there is a servo fault or emergency stop.

To issue a spindle stop:

1. Refer to Map 10**, Menu D**. Highlight **Issue SpStop on SvoFlt or Estop**.

2. Press **ENTER** to toggle the setting to either **No** (Off) or **Yes** (On). (Default: **Yes**)

Displaying Internal Interface Messages

Displaying the internal interface messages allows you to override the internal interface messages (i.e., feedhold, external hold, etc.) so that the IPI can generate a different message.

1. Refer to Map 10**, Menu D**. Highlight **Display internal interface messages**.

2. Press **ENTER** to toggle the setting ON (**Yes**) or Off (**No**). [Default: **Yes**]

Setting Up DSP^2 Node

To set up the DSP^2 Node:

1. See Map 10**, Menu H**. Highlight **DSP^2 Node**, and press **ENTER**.

   The CNC displays **Menu I**.

NOTE: The DSP^2 Node has three input bits, which can be assigned to any of three input functions.

2. To configure each bit, press **Input** (**F1**).

   The CNC displays **Menu K**.

3. Highlight **Input 0**, **Input 1**, or **Input 2**, and press **ENTER**.

   The CNC displays **Menu I**, ready for you to assign a function to each bit. [Default: **Bit 0**]

4. To assign bit functions in **Menu I**, highlight the desired input function, and press **ENTER**.

   The CNC displays **Menu J**.

5. Highlight **Off**, **Active Low**, or **Active High** for each function.

Setting Up CAN I/O Nodes

To set up a CAN I/O Node:

1. See Map 11**, Menu D**. Highlight **CAN Node #**, and press **ENTER**.
2. The CNC displays **Menu E**.
3. Press **ENTER** to configure the **Installed** parameter. Select **Yes** if the Can I/O board is installed; otherwise, select **No**.
4. Highlight **Type**. Press **ENTER** to select **All Digital** or **Digital/Analog**.

NOTE: CAN I/O nodes set to All Digital provide 10 digital inputs and 6 digital outputs. CAN I/O nodes set to Digital/Analog provide: one analog input, 10 digital inputs, and five digital outputs.

5. If **Digital/Analog** is selected in **Type** then the highlight **Analog input use**, press **ENTER** and Select Gauge.

NOTE: Refer to *OEM CNC Installation*, P/N 70000506, for additional hardware setup required using analog inputs.

Assigning Input Functions to an Input

NOTE: All CAN Nodes (0 through 5) operate identically. To assign input functions to a CAN Node's input ports, see Map 11. The **#** symbol indicates CAN Node **0** through CAN Node **5**, as determined in **Menu D**.

To configure CAN Input functions:

1. See Map 11**, Menu E**. Highlight **Inputs**, and press **ENTER**.

   The CNC displays **Menu F, CAN Input # Functions (Node #) – Setup**.

2. Press **Input (F1)** to select the input bit to be assigned to the input function.

   The CNC displays **Menu G**.

3. Highlight an input from **Input 0** through **Input 9**. [Default: **0**]. To select the input, press **ENTER**.

NOTE: The heading of **Menu F** changes to reflect the active bit selected in **Menu G**. Refer to Map 11. For example, if you select Input 4 in **Menu G** and press **ENTER**, the CNC displays the heading on **Menu F** as CAN Input 4 (Node #) **-** Setup.

4. Highlight one of the available input functions, and press **ENTER**.
5. The CNC displays **Menu H, Option Setup Menu**.
6. Highlight **Off**, **Active Low**, or **Active High** for each function.

CNC Input Functions

Each input function causes a specific action by the CNC. Refer to Map 11, **Menu F**. Refer to **Table 2-13** for a list of the available input functions.

Table 2-13, CNC Input Functions

Input Function	Definition
Tool guard	Holds the CNC program and stops the machine spindle. This input must be removed before the program can continue. To restart the spindle, press **START** once. To continue the program press **START** a second time. If the spindle was not running when the function was activated, press **START** once to continue the program. The tool guard function permits compliance with regulations that require an intact tool guard in place for the machine to run.
External finish pulse	An input signal from a machine device that informs the CNC that the requested operation was completed. When an output port configured for a finish pulse is activated, the CNC is put on hold until a finish pulse input is received. While on hold, the CNC will not continue to run program blocks (if in Programmed Mode) and will not respond to keypad inputs (if in Manual Mode).
Optional block skip (0–9)	Activating this input enables a block skip command (0–9). Block skip switches must be programmed into the part program.
External start	Performs the same function as **START** key on the CNC keypad.
External hold	Performs the same function as **HOLD** key on the CNC keypad.
External feed hold	Holds the program if the CNC attempts a feed. A Spindle Off condition during a feed move normally activates this function.
External manual select	This input enables manual hardware operation. Position display is generated from encoder inputs. In this mode, the CNC no longer commands axis motion. Thus, an operator can move the machine with the hand wheels and use the CNC as a digital read out. Use this function with the Auto/Manual switch included in the 5000M Kit Style CNCs.
Start reading keyboard	Allows the CNC to accept inputs from the CNC keypad (or keyboard).
Stop reading keyboard	Commands the CNC **NOT** to respond to inputs from the CNC keypad (or keyboard). Allows you to set a keyboard lockout system. When you lock out the keypad (or keyboard), all keys except **E-STOP** remain inoperative.
Initialize #999 to 0	When the input attached to **999** is activated, the value of the variable becomes **0**.
Increment #999	When activated, the value of register **999** increments by **1**.
Set 100 percent feed override	When activated, ignores settings of the **FEEDRATE OVERRIDE** switch and sets to **100%** (no feedrate override).

(Continued…)

Table 2-13, CNC Input Functions (Continued)

Input Function	Definition
Spindle CW (M3)	When activated, CNC initiates an **M3** output. Duplicates spindle **CW** function.
Spindle CCW (M4)	When activated, CNC initiates an **M4** output. Duplicates spindle **CCW** function.
Spindle Off (M5)	When activated, CNC initiates an **M5** output. Duplicates spindle **OFF** function.
General Error Input message	Stops the program and generates an Error message. You must correct the error condition before you can restart the program.
General Warning Input Message	Generates a Warning message at the CNC. The program continues to run.
Remote Jog +	Allows you to make positive jog moves from a remote manual panel. Works the same as the Jog + key on the manual panel, and follows the axis selector and resolution selector from the manual panel.
Remote Jog -	Allows you to make negative jog moves from a remote manual panel. Works the same as the Jog - key on the manual panel, and follows the axis selector and resolution selector from the manual panel.
Remote Resolution Selector	Allows you to select the axis resolution factor from a remote manual panel. Three input bits are used with each bit corresponding to x1, x10 and x100 resolution. The three input bits must be sequential. If used with an MPG, the Remote Resolution Selector applies to the MPG connected to CTR 0.
Remote Axis Selector	Allows you to select an axis from a remote manual panel. The number of bits required must correspond to the number of axes available. For example, three input bits are used with each bit corresponding to X, Y, and Z-axis on a 5000M Three Axes systems. Four bits are required on four axes systems, and five bits with five axes systems. The input bits must be sequential. If used with an MPG, the Remote Axis Selector applies to the MPG connected to CTR 0.
Clamping finish pulse	Acknowledge signal from clamp mechanism. Refer to "Axis Clamping."
Enable remote Z adjust	Remote Z Adjust is a custom feature. This feature allows the Z-axis to be moved during Auto, Single Step, or Manual mode. The input function "Remote Z Adjust" must be active (that is, switch pressed) to allow this special operating mode. This mode allows you to move the Z-axis during program execution. It is not necessary to press **HOLD**. If X and/or Y are moving, they will continue to move while the Z-axis is adjusted accordingly. The adjustments performed in this mode are tracked on the machine position (that is, program position does not change). Therefore, it is possible to build an offset (or shift) between the machine position and the program position. To cancel this offset, you must execute the M-code command M9204 X1. This cancels any offset introduced by using the Z Adjust operation mode.
Optional program stop	Corresponding hardware switch for M-code **M01** (Optional Program Stop). The status of the switch is reflected on this input function. When switch is ON, **M01** acts as **M00**. If switch is OFF, program will ignore **M01**. This input function is required for **M01**.

Output Function Setup

The CAN I/O board generates outputs to activate or deactivate various machine devices as commanded by the CNC. The CNC supports M-function outputs. M-function outputs are activated when programmed M-code blocks are executed and M-Function numbers (1 through 99) correspond to program M-Code numbers (1 through 99).

Run an M-code block to activate the outputs assigned to like-numbered M-functions.

To set an output port:

1. Configure the port to produce the type and logic of the signal required to control the device on the machine.
2. Assign the port to the functions that will activate and deactivate it.

   A port can be used by more than one function.

Configuring Output Ports

Default port settings provide a +24VDC common source when the port is active, and put the port in a high impedance state when the port is inactive. Each output port is rated for 500 mA. Usually, one function activates a port and another deactivates it (latched output). Sometimes, the CNC emits an output signal for a user-specified duration (pulsed output). The default port settings open and close relays that operate devices on the machine. Figure 2-2, Typical Inactive Port (Default Port Settings) illustrates typical Inactive port settings and Figure 2-3, Typical Active Port (Default Port Settings) illustrates typical active port settings.

Configure each port independently to generate a constant output or a single pulse (of definable width) with either high or low activation logic. Logical combinations of port setup options can be used together.

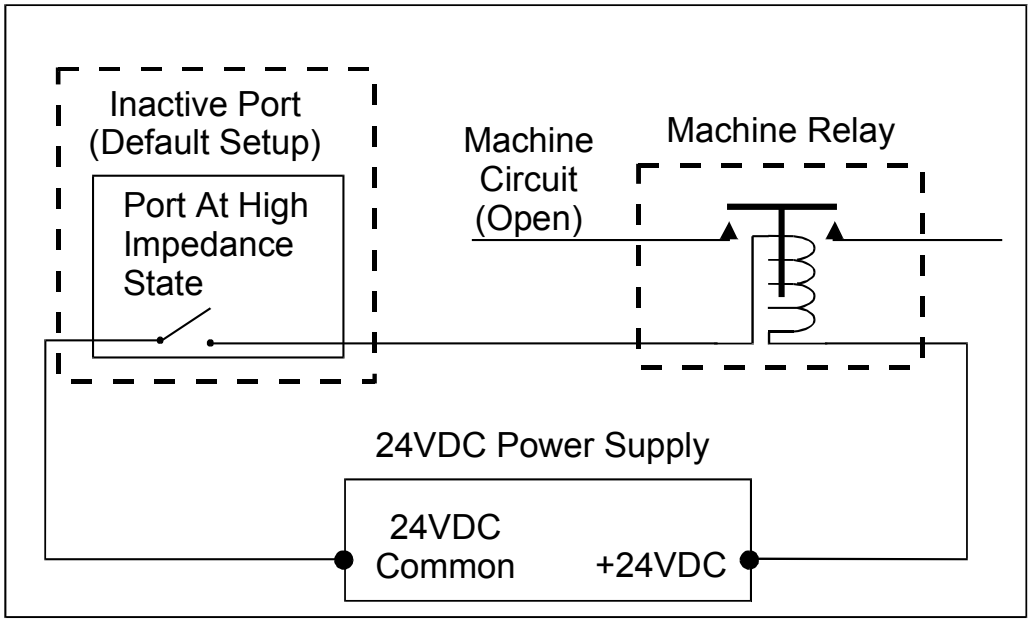


Figure 2-2, Typical Inactive Port (Default Port Settings)

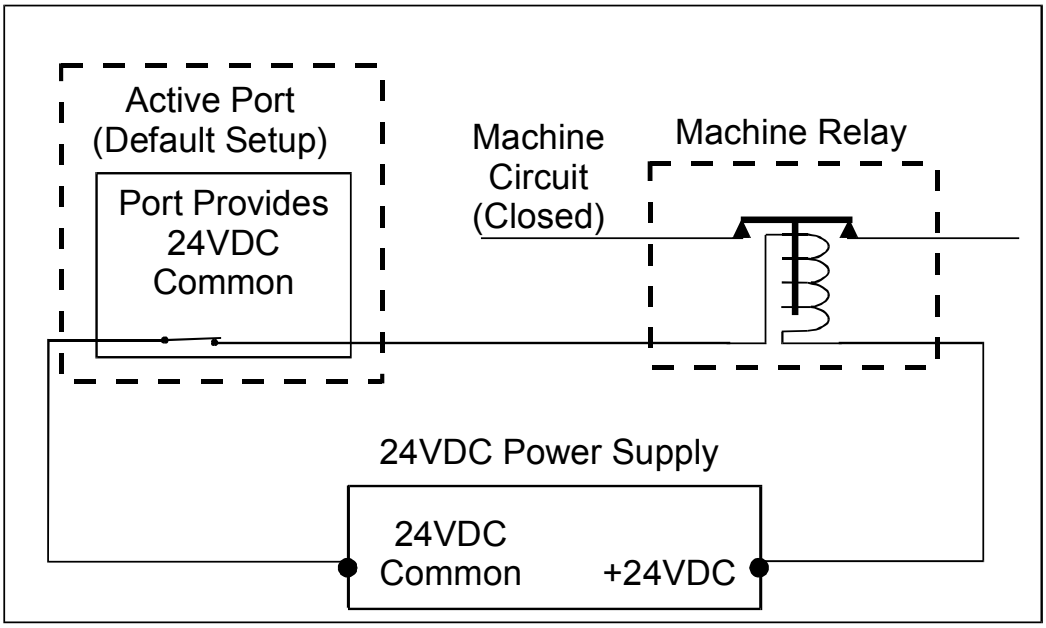


Figure 2-3, Typical Active Port (Default Port Settings)

Assigning M-Functions to Outputs

NOTE: CAN Node 0 and CAN Nodes 1 through 5 operate identically. To assign M-functions to a CAN Node's output ports, see Map 11. The **#** symbol indicates any CAN Node, as determined in **Menu E**.

The CNC can generate up to 99 M-functions (**M1** to **M99**). An M-function can activate or deactivate the ports assigned to it. An output can be assigned to more than one M-function.

1. See Map 11**, Menu D**. Highlight a **CAN Node** (**0–5**).

   The CNC displays **Menu E**.

2. Highlight **Outputs**, and press **ENTER**.

   The CNC displays **Menu I**.

Configure M-functions as follows:

1. Press **Output (F1)** to select the output bit to be assigned to an M-function.

   The CNC displays **Menu L**.

NOTE: The CNC changes the Heading in Menu I to reflect the active bit selected in Menu L. Refer to Map 11. For example, if you select Bit 4 in Menu K and press **ENTER**, the CNC displays the heading on Menu I as M-Functions (Output 4, Node #) - Setup.

2. Highlight the desired output bit, and press **ENTER**.
3. Highlight the M-function to be configured, and press **ENTER**.

   The CNC displays **Menu J**.

4. Highlight **Unused**, **Bit On**, or **Bit Off**, and press **ENTER**.
5. To configure the attributes for an output bit press **ATTR** (**F7**).

   The CNC displays **Menu K**.

6. Highlight a bit from 0 to 5, and press **ENTER**.

   The CNC displays **Menu M**.

Use Menu M to configure the following output bit attributes:

- **Pulse (msec) -** Maintains port output for the specified period, in milliseconds. Any value from 0 to 32,000 is allowed. If you specify 0-msec pulse, the CNC maintains output until the port is turned off. The CNC will delay running the program for the duration of a non-zero (msec) pulse.

- **Delay (msec) -** Holds program run for the specified period of time after completion of the M-function. Type the number of milliseconds. Any value from 0 to 32,000 is allowed. If the Delay (msec) is set to zero, then there is no delay.
- **Finish -** When a function activates an output port that is set for a finish pulse, the CNC delays running the program until an External Finish Pulse input function is received. If it does not receive a finish pulse by the end of the timeout period, the CNC displays an **Error** message and terminates the program.

  If you specify **Finish** for a port that has a non-zero (msec) pulse, the CNC acknowledges External finish pulse only after the pulse expires.
- **Active -** Sets output logic as Active High or Active Low. When set Active High, Sink CAN I/O boards provide a switched 24V common output and source CAN .I/O boards provide a switched 24VDC. When set Active Low, the port is in high impedance state when activated. See *OEM CNC Installation*, P/N 70000506, for more details.

NOTE: Output bit attributes can be set at any time after an output bit has been selected.

Setting Up Gauges

Gauges allow you to monitor analog inputs that vary from zero to 5 VDC (e.g., DC outputs for drive controllers, adjustment potentiometers, etc.).

NOTE: Before specifying a gauge, ensure that you have specified the Node Type for the CAN I/O as Digital/Analog. This will enable the gauge to be activated. See Map 11**, Menu E**.

To specify a gauge(s):

1. Refer to Map 10**, Menu D**. Highlight **Gauges**.

   A pop-up menu is displayed (Map 10**, Menu F**) listing the options: **Gauge #1**, **Gauge #2**, and **Gauge #3**.

2. Highlight the desired gauge, and press **ENTER**.

   A pop-up menu is displayed (Map 10**, Menu G)** listing options **Active**, **Name**, and **Input node**.

3. Highlight **Options Active**. Press **ENTER** to toggle the selection On/Off to activate/deactivate the gauge. (Default: **Yes**)

4. Highlight **Name**. Press **ENTER** and assign a name (e.g., spindle load) to the gauge in the entry field.

5. Highlight **Input node**. Press **ENTER** and assign a node (**0**–**5**) to the gauge in the entry field. This is the node from which the analog value is measured for the particular gauge.

Gauges are displayed in the bottom of the CNC's Manual, Auto, S.Step, and MST screens.

Programmable I/O Interface Setup

The CNC has an integrated programmable I/O interface tool known as Integral Programmable Intelligence or IPI. For information on IPI, refer to the *Integral Programmable Intelligence User's Guide*, P/N 70000416.

Handwheel/DRO Setup

The CNC supports counter ports. These ports can be configured as DRO (i.e., Digital ReadOut) or handwheels (i.e., MPGs). A system normally has two counter ports available. Any combination of DROs or handwheels that do not exceed the available counter ports can be used.

5000M-5X

The 5000M Five Axes has three counter ports. The third counter port is referred to as Handwheel/DRO #3. See Map 5**, Menu K**.

There are two parameters that apply to the counter port regardless of its use as a handwheel or DRO. Refer to **Table 2-14** for a definition of these parameters. See Map 4**, Menu G** and **Menu L**.

Table 2-14, Handwheel/DRO Common Parameters

Setting	Description
Type	Specifies the use of the counter port. Available choices are: None, Handwheel, Linear DRO, Rotary DRO, and Coupled DRO.
Phase	Switch this setting to invert one channel of the encoder output. An inversion of one channel changes the relationship of the A and B phases by 180° (positive direction becomes negative direction).

Setting of these parameters is described in the following topics.

Handwheel Parameters

Refer to **Table 2-15** for information on the parameters used to only when the counter port is configured for handwheel use. See Map 4**, Menu H** and **Menu N**.

Table 2-15, Handwheel Parameters

Setting	Description
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.
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.
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.

To configure Handwheel #1or #2:

NOTE: You may perform all of the steps below in sequence, or, if the handwheel is already configured, you may change one or more of the parameters.

1. See Map 4**, Menu B**. Highlight **Handwheel/DRO** in the **Builder Setup** menu.
2. Press **ENTER** to display the **Handwheel/DRO-Setup** menu. See Map 4**, Menu F**.
3. Press **ENTER** to display the **Handwheel/DRO #1 – Setup** menu. See Map 4**, Menu G**.
4. Press **ENTER** to select the **Type** for **Handwheel/DRO #1**. See Map 4**, Menu I**. [Default: **None**]
5. Highlight **Handwheel** and press **ENTER** to setup other Handwheel parameters.
6. Highlight **Phase**.
7. Press **ENTER** to toggle to the desired phase setting, **Invert** or **Not Invert**. [Default: **Not Invert**]
8. Highlight **Handwheel**, and press **ENTER**. This will allow you to configure other Handwheel specific parameters. See Map 4**, Menu H**.
9. Highlight **Resolution**, and press **ENTER**.
10. In the pop-up menu that is displayed, highlight one of the options: **MP Switch**, **Fixed 1**, **Fixed 10**, or **Fixed 100**. [Default: **MP Switch**]. Press **ENTER**. See Map 4**, Menu J**. Refer to Table 2-16, Handwheel Resolution Parameters for a description of these options.

Table 2-16, Handwheel Resolution Parameters

Setting	Description
MP Switch	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 *5000M CNC Programming and Operations Manual*, P/N 70000508, for more details.
Fixed 1, 10, 100	Corresponds to resolution x1, x10 and x100 respectively.

11. Highlight **Axis**, and press **ENTER**.
12. In the pop-up menu that is displayed, highlight one of the options: **MP Switch**, **Fixed X**, **Fixed Y**, or **Fixed Z**. [Default: **MP Switch**]. Press **ENTER**. See Map 4**, Menu K**. Refer to **Table 2-17** for a description of these options.

Table 2-17, Handwheel Axis Parameters

Setting	Description
MP Switch	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 *5000M CNC Programming and Operations Manual*, P/N 70000508, for more details.
Fixed X, Y, Z	The handwheel axis can be fixed to a specific primary axis.

13. Highlight Scaling Factor, and press **ENTER**.

    The entry field for the parameter highlights.

14. Type the desired value, and press **ENTER**. [Default: **1.00**]

Configure other handwheels in a same way as described above.

5000M-5X

The 5000M five axes configuration can support up to three handwheels. See Map 5**, Menu K** for the additional Handwheel/DRO entry. Thus, a machine can be configured with a handwheel dedicated to each of the primary axis (i.e., XYZ). In such a configuration, the Axis parameter should be set to Fixed X, Fixed Y, and Fixed Z for XYZ respectively.

DRO Parameters

Refer to **Table 2-18** for information on the settings used only when the counter port is configured for DRO use. See Map 5**, Menu H**.

Table 2-18, DRO Parameters

Setting	Description
Display Axis	Allows you to specify a label for the DRO. Available choices are: None, U, V, W, A, B, and C. See Map 5**, Menu H**. Do not use U or W as DRO labels if these correspond to controlled axis.
Type	Feedback device type (Linear Encoder or Rotary Encoder)
Reset Rotary at 360	If DRO type is Rotary DRO this parameters will force the CNC to initializes the DRO display to 0 when it reaches 360 degrees
Display Resolution	Define the display resolution to be used. Available choices are: 0.5 Micron, 1 Micron, 2 Micron, 5 Micron, and 10 Micron.
Resolution	Define the actual resolution of the feedback device. Available choices are: 0.5 Micron, 1 Micron, 2 Micron, 5 Micron, and 10 Micron.
Encoder Lines	Number of lines in rotary encoder. Use only when using a rotary encoder.
Pitch	Pitch of ballscrew.
Ratio	Ratio of ballscrew to encoder.

To configure DRO #1 or #2:

NOTE: You may perform all of the steps below in sequence, or, if the DRO is already configured, you may change one or more of the parameters.

1. See Map 4**, Menu B**. Highlight **Handwheel/DRO** in the **Builder Setup** menu.
2. Press **ENTER** to display the **Handwheel/DRO-Setup** menu. See Map 4**, Menu F**.
3. Press **ENTER** to display the **Handwheel/DRO #2 – Setup** menu. See Map 4**, Menu L**.
4. Press **ENTER** to select the **Type** for **Handwheel/DRO #2**. See Map 4**, Menu M**. [Default: **None**]
5. Highlight one of the DRO options, and press **ENTER**.
6. Highlight **Phase**.

7. Press **ENTER** to toggle to the desired phase setting, **Invert** or **Not Invert**. [Default: **Not Invert**]

8. Highlight **DRO**, and press **ENTER**. This will allow you to configure other DRO specific parameters. See Map 5**, Menu H**.

   Refer to Table 2-18, DRO Parameters, to setup DRO specific parameters.

5000M-4X
5000M-5X

NOTE: Do not use U or W as DRO labels if these correspond to controlled axis.

Coupled DRO Parameter

A coupled DRO axis is a DRO axis whose position feedback is summed (i.e., coupled) to the Z-axis position. The displayed Z-axis machine position is the combined display of the Z-axis program position and the DRO position. A DRO axis that is configured as a Coupled DRO is not displayed. Only one counter port can be configured as a Coupled DRO.

Coupled DRO Tolerance Parameter

Coupled DRO Tolerance is a parameter that applies to a DRO axis when the DRO axis is configured as Coupled DRO. This one tolerance parameter applies regardless of which counter port is configured as Coupled DRO. The parameter defines a distance that the CNC will allow the DRO axis to move (i.e., tolerance value) before generating an error. The error is only generated during Auto/S.Step.

To configure the Coupled DRO Tolerance:

1. See Map 5**, Menu K**. Highlight **Coupled DRO Tolerance**

2. Press **ENTER** and type the appropriate value.

Tool Management

Default Tool Table File

To enter the default tool table file:

1. See Map 12**, Menu C**. Highlight **Default tool-table file**, and press **ENTER**. In the highlighted entry field, type in the name of the file. [Default: **P5MTOOL.DAT**]

Activation Options

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

- Activate tool length offset [Default: **On Tn**]
- Output Signal [Default: **On Tn**]
- Orient Spindle [Default: **No**]
- Stop program execution [Default: **No**]
- Use tool change macro [Default: **No**]

The available activation options are listed in **Table 2-19**.

Table 2-19, 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 (i.e., when a tool changer is not being used), use the settings specified in **Table 2-20** and refer to Map 12**, Menu C**.

Table 2-20, Manual Tool Change Settings

Manual Tool Change Parameters	Required Setting	Description
Activate tool length offset	On Tn	Tool Length Offset activates upon completion of a T word command to the Programmable Controller.
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.
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.
Tool changer installed	None	Set to None (disabled). Use this selection to enable closed loop orientation of the spindle during tool changer operation.
Number of digits in T word	4	Format of T words (Txxxx).
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

Refer to Map 12**, Menu C**. 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 2-21**.

Table 2-21, Automatic Tool Changer Settings

Automatic Tool Change Parameters	Required Setting	Description
Activate tool length offset	On M6	Tool Length Offset activates upon an M6 command.
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.
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.
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.
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.
Spindle orientation in position (cts)	0 (counts) (No in position check) Range: (0 to 50)	At the end of a spindle orientation sequence, the system checks for the spindle position to be within this parameter range. If spindle position is within the counts (cts) range, the system completes the orientation sequence. If not within the count range, the system waits until the spindle is within the in position range to complete orientation.
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 2-21, Automatic Tool Changer Settings (Continued)

Automatic Tool Change Parameters	Required Setting	Description
Tool changer installed	None – or – Fixed Replacement – or – Random Replacement	Enables/disables M19 command. 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]
Use tool change macro	On M6	Sets function to activate on M6 command.
Tool change macro program	Macro filename	Filename of the macro to be used during tool change operations.
Tool change macro number	Macro Number	Macro Number to be used during tool change operations. Macro files can contain more than one macro (number).
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.
Number of bins in tool changer	Enter number.	Number of bins.
Number of Tools to display in table	Enter maximum # of tools.	Limits the number of tools displayed in the tool table. [Default: 99]
Default tool-table file	Enter filename.	[Default tool table file: P5MTOOL.DAT] .
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.
Restore TLO after Power-Up or Home	No	Restore tool length offset (TLO) after power-up or home sequence.
Send Tflag when T0 programmed on Fixed TC	No	When flag is set to No, T0 is not sent to the IPI. When set to Yes, whenever a T0 is executed, the Tflag is sent to IPI with a value 0.

Enabling M19 Commands

Format: M19

Enable/disable the M19 command during tool changer operations through the **Tool changer installed** parameter in the Tool Management Setup (refer to Map 12**, Menu C**). The **M19** command orients the spindle to a given angle via the **Spindle orientation RPM**, which is also set in **Menu C**. 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 **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 *5000M CNC Programming and Operations Manual*, P/N 70000508, for more details.

Guidelines for Setting the Number of Digits for T Words

Use the **Number of digits in T word** parameter (Map 12**, Menu C**) 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.

Refer to Map 12**, Menu C**. 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 **Use tool change macro** parameter to: **No**, **OnTn**, **OnM6** or **Both**.

To call a tool change macro in the Setup Utility, specify the filename and macro number. Use the **Tool change macro program** (Map 12**, Menu C**) parameter to specify the tool changer macro filename. Use the **Tool change macro number** to specify the appropriate macro number within the program.

NOTE: The macro file is stored in the C:\P5M 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 2-22**. 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 2-22, 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
IF (#1070 < 71) THEN        * VERIFY CNC CONTROL IS IN INCH MODE
G90  G0  Z-0.2362           * SAFE INCH POSITION FOR Z AXIS
ELSE                        * IF CNC CONTROL IS IN METRIC MODE
G90  G00  Z-0.6             * SAFE METRIC POSITION
ENDIF
G53  O0                     * CANCEL WORK CO-ORDINATES
G90  G0  X0  Y0             * 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 to 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, etc.

Miscellaneous Setup Parameters

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.

Manual Panel Port

For standard CNC systems, set the Manual panel port to **COM1**. **Disabled** is used for simulation systems or for Offline software. [Default: **COM 1**]

> **NOTE:** Manual panel operations are not allowed if the manual port is disabled.

To set the manual port:

1. See Map 6**, Menu C**. Highlight **Manual port**, and press **ENTER**.

   The CNC displays a pop-up menu. The menu lists the available RS-232 ports.

2. Highlight **COM 1**, and press **ENTER**.

Maximum 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. [Default: **80.0 in/min**]

To set the maximum-programmed feedrate:

1. See Map 6**, Menu C**. Highlight **Max programmed linear axis feedrate**, and press **ENTER**.
2. Type the maximum-programmed feedrate in the highlighted entry field, and press **ENTER**.

> **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%.

Linear Axis Dry Run Feedrate

When a program is run in Dry Run Mode, the machine's linear axes (X, Y, and optionally Z) 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 2-23** 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.
[Default: **40.0**]

Table 2-23, 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.

To set up the Dry Run Mode feedrate:

1. See Map 6**, Menu C**. Highlight **Linear axis dry run feedrate**.
2. Press **ENTER** to activate the entry field. Type the feedrate and press **ENTER** to activate it.

Linear Axis Jog Feedrate and Rapidrate

Set up the feedrate and rapidrate for linear axes at which the machine travels in Jog Mode. This defines the machine's default jog speed.

[Defaults: Feedrate **40.0**; Rapidrate **200.0**]

To set up the linear axis jog feedrate and rapidrate:

1. See Map 6**, Menu C**. Highlight **Linear axis jog feedrate** or **Linear axis jog rapidrate**, and press **ENTER**.
2. In the highlighted entry field, type the rate, and press **ENTER**.

> **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%.

Maximum Programmed Rotary Axis Feedrate

5000M-4X
5000M-5X

Max programmed rotary axis feedrate sets the maximum speed, in degrees per minute that a rotary axis may be programmed.
[Default: **3,000.0**]

To set the maximum rotary feedrate:

1. See Map 6**, Menu C**. Highlight **Max programmed rotary feedrate**.
2. Press **ENTER** to activate the entry field. Type the feedrate, and press **ENTER** to activate it.

> **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%.

Rotary Axis Dry Run Feedrate

5000M-4X
5000M-5X

Rotary axis dry run feedrate specifies the feedrate, in degrees per minute, at which rotary axes will travel during Dry Run Mode. Dry Run Mode is activated by M105 or M106 and canceled by M107, similar to the Dry Run Mode for linear axes (refer to **Dry Run Linear Feedrate**). [Default: **1,000.0**]

To set the maximum rotary feedrate:

1. See Map 6**, Menu C**. Highlight **Rotary axis dry run feedrate**.
2. Press **ENTER** to activate the entry field. Type the feedrate, and press **ENTER** to activate it.

Rotary Axis Jog Feedrate and Rapidrate

5000M-4X
5000M-5X

Set up the feedrate and rapidrate for rotary axes at which the machine travels in Jog Mode. This defines the machine's default jog speed for rotary axis. [Defaults: Feedrate **1000.0**; Rapidrate **3000.0**]

To set the default jog rotary feedrate and rapidrate:

1. See Map 6**, Menu C**. Highlight **Rotary axis jog feedrate** or **Rotary axis jog rapidrate**
2. Press **ENTER** to activate the entry field. Type the feedrate, and press **ENTER** to activate it.

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%.

Servo Up Delay

You can program a delay to allow the servos to stabilize before the CNC commands a move.
[Default: **1** sec]

To program a servo delay:

1. See Map 6**, Menu C**. Highlight **Servo up delay**, and press **ENTER**.
2. In the highlighted entry field, type servo delay.

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.
[Default: Disabled (**No**)]

To activate automatic feed override on arc moves:

1. See Map 6**, Menu C**. Highlight **Automatic feedrate override on arcs**.
2. Press **ENTER** to switch 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.

Rapid Moves are Free (Unsynchronized)

Rapid Moves Are Free (Unsynchronized) sets up rapid moves for synchronized or unsynchronized motion. Switch to Rapid Moves Are Free (Unsynchronized) to change how the CNC executes Rapid moves. If you select No, the CNC synchronizes Rapid moves and the resulting rapidrate is a vector rapid rate; if you select **Yes**, the CNC executes Rapid moves unsynchronized (free) and each axis moves at its specified rapid rate. [Default: **No**]

To set Rapid Moves Are Free:

1. See Map 6**, Menu C**. Highlight **Rapid moves are free (unsynchronized)**.
2. Press **ENTER** to select **Yes** or **No**.

Feed and Rapid Accel/Decel (ms)

Feed Accel/Decel and Rapid Accel/Decel define Feed and Rapid acceleration and deceleration ramp times (in milliseconds).
[Default: **140.00**]

To set Feed Accel/Decel and Rapid Accel/Decel:

1. See Map 6**, Menu C**. Highlight either **Feed Accel/Decel** or **Rapid Accel/Decel**, as applicable.

   The CNC highlights an entry field.
2. Type the desired value.

Check DSP^2 Integrity

Check DSP^2 Integrity enables (**Yes**) or disables (**No**) an integrity check of the DSP Motion Control Board and of all commands sent to it from the PC. [Default: **Yes**]

NOTE: The CNC performs the integrity check on the DSP^2 when you first turn it on. Integrity check on commands are performed on every command.

To set Check DSP^2 Integrity:

1. See Map 6, **Menu C**. Highlight **Check DSP Integrity**.
2. Press **ENTER** to select **No** or **Yes**.

Servo Loop Sample Time (ms)

Servo Loop Sample Time (ms) sets the rate at which the servo loop operates (in milliseconds). [Default: **0.4000**]

To set Servo Loop Sample Time:

1. See Map 6, **Menu C**. Highlight **Servo Loop Sample Time**.

   The CNC highlights an entry field.
2. Press **ENTER**.
3. Type the desired value.

Interpolator Rate Factor

The Interpolator Rate Factor defines the Interpolator Sample Rate.

Interpolator Sample Rate = (Servo Loop Sample Time) (Interpolator Rate Factor)

[Default: **12**]

To set Interpolator Rate Factor:

1. See Map 6, **Menu C**. Highlight **Interpolator rate factor**.

   The CNC highlights an entry field.
2. Press **ENTER**.
3. Type the desired value.

Enable Velocity Look Ahead

Velocity Look Ahead parameter is an optimization feature of the DSP motion control firmware. In most cases it should be left set to Yes (enabled). In applications that run at very slow feedrates and the slow feedrates are not being achieved the parameter should be set to No (disabled). [Default: **Yes**]

To set Enable Velocity Look:

1. See Map 6**, Menu C**. Highlight **Enable Velocity Look Ahead**.
2. Press **ENTER** to select **No** or **Yes**.

Display Resolution

The Display Resolution parameter allows you to specify the display resolution of the system. The choices available are: VGA (640x480) and SVGA (800x600). All CRT systems should use VGA, while all 12.1" flat panel systems should use SVGA. [Default: **VGA**]

To set the display resolution:

1. See Map 6**, Menu C**. Highlight **Display Resolution**.
2. Press **ENTER** to select **SVGA** or **VGA**.

The SVGA setting for flat panels applies to both flat panel based CNC console assemblies purchased from Anilam as well as flat panels in laptops for use with offline software.

Acceleration Ramp Type

Selects the Acceleration Ramp type or profile to be used. [Default: **Bell**]

To select the Acceleration Ramp Type:

1. See Map 6**, Menu C**.
2. Highlight **Acceleration Ramp type**. Press **ENTER** to select **S-Curve** or **Bell**.

Ramp Z axis during G33 primitive

Ramp Z-axis during G33 primitive enables (**Yes**) or disables (**No**) the Z-axis profile. [Default: **Yes**]

To set Ramp Z axis during G33 primitive:

1. See Map 6**, Menu C**.
2. Highlight **Ramp Z axis during G33 primitive**. Press **ENTER** to select **No** or **Yes**.

CNC Startup Mode

Set the acceleration ramp time for feed rate.
[Default: **Sfwr Options**]

To set the CNC startup mode:

1. See Map 7, **Miscellaneous Setup, Menu C**.
2. Highlight **CNC Startup mode**, and press **ENTER** to toggle between **Sfwr Options** or **Ctrl Software**.

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.

Show Introduction Screen

Enables the display of the introduction splash screen (**Yes**) or disables the display (**No**).
[Default: **Yes**]

To set the display of the introduction screen:

1. See Map 7, **Menu C**.
2. Highlight **Show Introduction Screen**, and press **ENTER** to select **No** or **Yes**.

User Definable Variables

User definable variables are defined via parameters **#1130** – **#1139** and **#1120** – **#1129** in Miscellaneous Setup (see Map 7, **Menu C**, Miscellaneous Setup Menu). These parameters correspond directly to system variables #1130 through #1139. Parameters **#1130** – **#1134** are unit based; which means, these are assigned the units specified for machine parameters (Inch or MM). You can assign parameters **#1135** – **#1139** and **#1120** – **#1129** 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."
#1130 – #1139 [Default: **0.0000**]
#1120 – #1129 [Default: **0**]

To set the user definable variables:

1. See Map 7, **Menu C**.
2. Highlight **User definable variable #1130** – **#1139** or **#1120** – **#1129** and press **ENTER**.
3. Type the value for the variable and press **ENTER**.

Tool Changer Macro Example

Refer **Table 2-24**. 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 2-24, 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
IF (#1070 < 71) THEN        * VERIFY CNC CONTROL IS IN INCH MODE
G90  G0  Z-0.2362           * SAFE INCH POSITION FOR Z AXIS
ELSE                        * IF CNC CONTROL IS IN METRIC MODE
G90  G00  Z-0.6             * SAFE METRIC POSITION
ENDIF
G53  O0                     * CANCEL WORK CO-ORDINATES
G90  G0  X0  Y0             * 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 to 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, etc.

Mcode for Macro Call #1 – #10

M-Code number you assign to call the macro in "Macro Called for Mcode #1 – #10."
[Default: **0**]

To set the Mcode for macro call:

1. See Map 7**, Menu C**.
2. Highlight **Mcode for macro call #1 – #10**, and press **ENTER** .
3. Type the value for the variable and press **ENTER**.

Macro Called for Mcode #1 – #10

The macro number that is called when the M-Code in **Mcode for macro call #1 – #10** is executed.
[Default: **0**]

To set the macro called for Mcode #1 – #10:

1. See Map 7**, Menu C**.
2. Highlight **Macro called for Mcode #1 – #10**, and press **ENTER**.
3. Type the value for the variable and press **ENTER**.

Machine Home

Most machines have an absolute reference, called Machine Home, defined by hardware. The CNC uses the zero marker pulse of the encoder and the home limit switch to define Machine Home.

You can also define Machine Home using only the zero marker of a rotary encoder or linear encoder.

Homing the Axes

NOTE: You cannot configure Retract Speed (the speed at which axis moves to index pulse) by using the Setup Utility.

After you define machine home, you can set the Machine Home feature to require a machine home when you turn the CNC on. Set the **Home required** parameter (Map 12**, Menu E**) to **Yes** to operate this way. You can perform Homing sequence at any time in the CNC mode, whether the Setup parameter requires it or not.

You can specify the order in which the machine homes the axes, and the speed at which the CNC moves to the Home Limit switch.

NOTE: If the Home Required parameter is On (**Yes**) and no home sequence has been performed, the axes can still be moved using the **JOG PLUS** (**+**) and **JOG MINUS** (**-**) keys on the Manual Panel.

Setting 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. [Default: **Yes**]

If you set the **Home required** to **Yes**, you must perform the home sequence before you resume normal operations in the control software.

To set up the **Home required** parameter:

1. See Map 12**, Menu E**. Highlight **Home required**.
2. Press **ENTER** to toggle the setting to **No** or **Yes**.

   Yes enables the Homing feature. Switching the setting to **No** deactivates the feature.

Setting Home Sequence

[Default homing order: **Third**, **Second**, or **First** corresponding to axes **X**, **Y**, and **Z**)]

To set the order in which axes are homed:

1. See Map 12**, Menu F**. Highlight **Home sequence**, and press **ENTER**.

   The CNC displays a pop-up menu with the options: **Home sequence for X… Third**, **Home sequence for Y… Second** and **Home sequence for Z … First**.

2. Select the axis whose homing sequence you want to specify, and press **ENTER**.

   The CNC displays a pop-up menu with the options: **First**, **Second**, and **Third**.

3. Highlight the desired choice, and press **ENTER**.

Home Type

The Home Type Setup Menu (Map 12**, Menu G**) allows you to specify the types of homing for each axis. Use **Home type** to set the direction of travel for the Homing feature. [Default: **No Homing**] No Homing disables the homing function.

Positive/**Negative** refers to the direction that the axis will travel during Homing, in reference to Machine Home.

With 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.

NOTE: The EverTrack encoder requires the **Home type** be setup as "With positive index limit" or "With negative index limit."

With Positive/Negative Index and 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.

To set up the direction of travel for Homing:

1. See Map 12**, Menu H**. Highlight **Home type**, and press **ENTER**.
2. Highlight the desired homing direction, and press **ENTER**.

NOTE: Home switches are wired to the same ports as the vector limits. Wire the home switch to the input that corresponds to the direction that you select for each axis. (See Table 2-2, CAN Bus Node 0 Vector Limit Input Port Assignments and Table 2-3, Vector Limit Inputs for 5000M 4-Axes and 5000M 5-Axes.)

Setting Datum Search Speed

[Default speed for X, Y, and Z-axes: **40 in/min**]

To set the speed at which the machine travels during Homing:

1. See Map 12**, Menu E**. Highlight **Datum search speed**, and press **ENTER**.

   The CNC displays a pop-up menu with options for entries in X, Y, and Z.

2. Highlight the desired axis. Press **ENTER** and type a value for the desired Datum Search Speed.

Setting 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.
[Default: **Off** (disabled)]

To set up Preset values for axes:

1. See Map 12**, Menu J**. Highlight the selection corresponding to the axis being preset, and press **ENTER**.
2. Type the preset value in the highlighted entry field.
3. Repeat for all preset axes.
4. Highlight **Home preset**, and press **ENTER** to toggle the selection **On** to enable it or **Off** to disable it.

Builder Text

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 2-25** for message-code ranges and message types.

Table 2-25, Message Code Ranges and Types

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

You can create and edit these builder texts using the Builder Text parameter.

Enabling Builder Text

To enable Builder Text:

1. See Map 13**, Menu C**. Highlight **Use builder text**.
2. Press **ENTER** to toggle the setting to **No** (disabled) or **Yes** (enabled)].

   [Default: **No]**
3. Save the changes before you exit the Setup Utilities.

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

Editing Error Messages

To edit Error messages:

1. Go to Map13**, Menu C**. Highlight **Edit Error Messages**, and press **ENTER**.

   The Prog. Interface Error Messages screen activates. Refer to **Figure 2-4**.

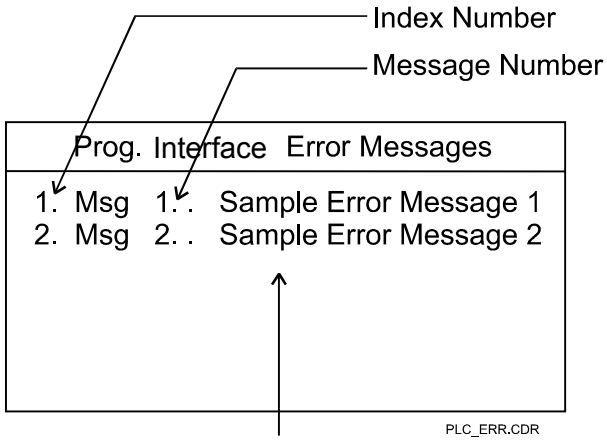


Figure 2-4, Builder Text - Error Message Window

2. Highlight the message to be typed or edited, and press **ENTER**.

   The CNC activates the message text box.

3. Type the message text in the box, and 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 13**, Menu C**. Highlight **Edit Warning Messages**, and press **ENTER**.

   The CNC displays the Prog. Interface Warning Messages window. See **Figure 2-5**.

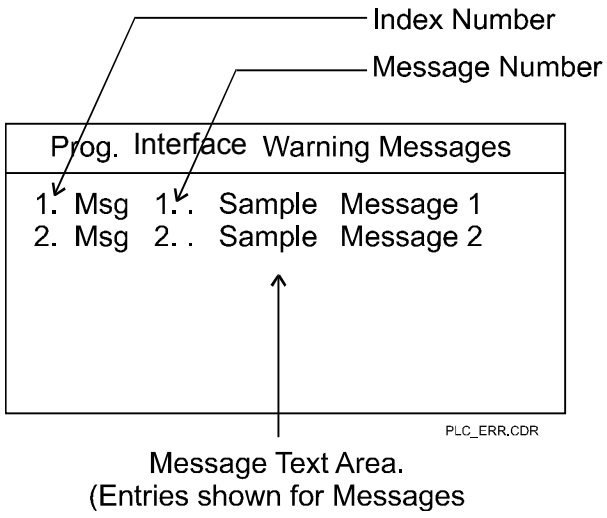


Figure 2-5, Builder Text - Warning Message Window

2. Highlight the message to be typed or edited, and press **ENTER**.

   The message text box activates.

3. Type the message text in the box, and press **ENTER**.

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

NOTE: You can also use a message's index number (including period) to access the message text box and edit the message.

With the Warning Message Window (**Figure 2-5**) activated, press the message index number, and press **ENTER**.

Edit the message.

Editing Soft Key Inputs

These are user definable soft key inputs. The soft key inputs are used in conjunction with IPI registers F1INPUT/R01 thru F9INPUT/R09. The text is defined in Builder Text. Nine (9) soft key inputs can be defined. **F10** is used for Exit. Up to seven (7) characters can be used for none-unicode label and up to three (3) characters for foreign Unicode label. The soft keys are accessed via (**SHIFT**–**F6**) in Manual, Auto, SingleStep, DNC, and Teach.

Labeled soft keys **F1** to **F10**, also called function keys, are located just below the monitor. Soft key functions are not hardwired; their functions change with changes in mode. Labels indicate the function of each soft key. Unlabeled soft keys are inactive.

To edit Soft Key Inputs:

1. See Map 13**, Menu C**. Highlight **Edit Soft Key Inputs**, and press **ENTER**.

   The CNC displays the Soft Key Inputs window. See **Figure 2-6**.

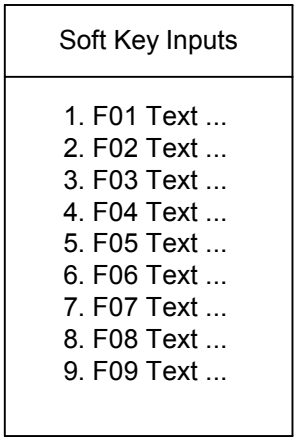


Figure 2-6, Builder Text – Soft Key Inputs Window

2. Highlight the soft key input to be typed or edited, and press **ENTER**.

   The message text box activates.

3. Type the text in the box, and press **ENTER**.

   The label you typed is displayed on the active soft key.

Languages

You can order a system that displays messages and other text in languages other than English. [Default: **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.

To set up the CNC to display text in a language other than English:

1. See Map 13**, Menu D**.
2. Highlight the desired language, and press **ENTER**.
3. The CNC loads the language file and restarts the system when you exit the Setup Utility. The CNC now displays Messages and other text in the selected language.

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 13**, Menu B**. Highlight **Software Update**, and press **ENTER**.
3. 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, etc.).
4. Answer the prompts to complete the installation.
5. 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 Configuration Utilities for how to backup and restore a configuration file.

Direct Numeric Control

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

See Map 13**, Menu F, DNC Setup Menu**. The **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.

To select Buffered/Drip Feed Mode:

1. See Map 13**, Menu F**.
2. Highlight **Execution Mode**. Press **ENTER** to switch the selection to **Drip Feed** or **Buffered**.

Setting the Buffer Size

This 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**.]

To set Buffer size:

1. See Map 13**, Menu F**.
2. Highlight **Buffer Size**, and press **ENTER**.
   The CNC displays **Menu G**.
3. Highlight the amount of memory to be devoted to the Buffer, and press **ENTER**.

Using a DNC Macro

To use a specified DNC Macro at the end of each block:

1. Refer to Map 13, **Menu F**. Highlight **Use DNC Macro at end of block**. Press **ENTER** to switch the setting to **Yes** (enabled). [Default: **No** (disabled)]
2. Highlight **DNC Macro Number**, and press **ENTER**.
3. Type the **DNC Macro Number**. [Default: **100**]

Enabling/Disabling 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.

To enable/disable G41/2 and G59:

1. See Map 13, **Menu F**.
2. Highlight **Ignore G41/2, G59, and Blueprint**
3. Press **ENTER** to switch the selection. Choose **Yes** to disable Tool Comp and CornerRad. Choose **No** to enable Tool Comp and CornerRad during DNC. [Default: **Yes**]

Setting Wait For Start Before Execution

Refer to **Table 2-26**. See Map 13, **Menu F**. Use the **Wait for Start** parameter to specify whether the control will hold data transmission until **START** is pressed.

Table 2-26, 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.

To set the **Wait for Start** parameter:

1. See Map 13, **Menu F**.
2. Highlight **Wait for Start**, and press **ENTER**. The Options pop-up menu is displayed (**Menu H**).
3. In **Menu H**, highlight an option, and press **ENTER** to activate it.

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 13**, Menu E**. Highlight the level of the password to be changed, and 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.

Probing

NOTE: While the Default for some of these settings may be zero, they must be changed to a valid number and in most cases, zero is not a valid number.

Setting the Spindle Probe Type

The transmission type used for the installed spindle touch probe is defined.

To set the probe type:

1. See Map 13, **Probing Setup, Menu I**.
2. Highlight **Spindle probe type**, and press **ENTER** to display **Menu J** that allows you to select between **Corded**, **Cordless**, **Cordless SG**, and **Cordless UD**. (Unidirectional)
3. Display the probe type that you want, and press **ENTER**.
   [Default: **Corded**]

Setting the Nominal Probe Stylus Diameter

The overall nominal probe stylus diameter is set.

To set the nominal probe stylus diameter:

1. See Map 13, **Probing Setup, Menu I**.
2. Highlight **Nominal probe stylus diameter**, and press **ENTER**.
3. Type the nominal probe stylus diameter, and press **ENTER**.
   [Default: **0.0000**] Valid range: (0.0000 to 51.0000)

Setting the 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 is set.

To set the maximum stroke from home for the first pick:

1. See Map 13, **Probing Setup, Menu I**.
2. Highlight **Maximum stroke from home for first pick**, and press **ENTER**.
3. Type the maximum stroke from home for the first pick, and press **ENTER**.
   [Default: **0.0000**] Valid range: (0.0000 to 999.0000)

Setting the RPM for Calibration and Tool Measurement

Sets the spindle spin RPM for tool touch.

To set the RPM for calibration and tool measurement:

1. See Map 13, **Probing Setup, Menu I**.

2. Highlight **RPM for calibration and tool measurement**, and press **ENTER**.
3. Type the spindle spin RPM for tool touch, and press **ENTER**.
   [Default: **0** (rev/min)] Valid range: (100 to 1000)

Setting the Probe Orientation

Sets the probe orientation using the following values:

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 – Not a valid value – must be changed]

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.

To set the probe orientation:

1. See Map 13**, Probing Setup, Menu I**.
2. Highlight **Probe orientation**, and press **ENTER** to display a pop-up window with the orientation values.
3. Select the orientation you want, and press **ENTER**.
   [Default: **0** – Not a valid value – must be changed]
   Valid range: (–2 to 2)

Setting the Z First Pick, FAST Feedrate

Sets user definable FAST feedrate.

To set the FAST feedrate:

1. See Map 13**, Probing Setup, Menu I**.
2. Highlight **Z first pick, FAST feedrate**, and press **ENTER**.
3. Type the FAST feedrate, and press **ENTER**.
   [Default: **0.0**] Valid range: (2.5 to 2540.0)

Setting the Z First Pick, MEDIUM Feedrate

Sets user definable MEDIUM feedrate.

To set the MEDIUM feedrate:

1. See Map 13**, Probing Setup, Menu I**.
2. Highlight **Z first pick, MEDIUM feedrate**, and press **ENTER**.
3. Type the MEDIUM feedrate, and press **ENTER**.
   [Default: **0.0**] Valid range: (2.5 to 508.0)

Setting the Z Final Pick, SLOW Feedrate

Sets user definable SLOW feedrate.

To set the SLOW feedrate:

1. See Map 13**, Probing Setup, Menu I**.
2. Highlight **Z final pick, SLOW feedrate**, and press **ENTER**.
3. Type the SLOW feedrate, and press **ENTER**.
   [Default: **0.0** – Not a valid value – must be changed]
   Valid range: (0.1 to 254.0)

Setting the Z Retract Amount

Sets user definable distance the tool will back away on the Z-axis after it touches the probe.

To set the Z retract amount distance:

1. See Map 13**, Probing Setup, Menu I**.
2. Highlight **Z retract amount**, and press **ENTER**.
3. Type the Z retract amount, and press **ENTER**.
   [Default: **0.0000**] Valid range: (0.0100 to 25.400)

Setting the XY Retract Amount

Sets user definable distance the tool will back away on the X-axis or Y-axis after it touches the probe.

To set the XY retract amount:

1. See Map 13**, Probing Setup, Menu I**.
2. Highlight **XY retract amount**, and press **ENTER**.
3. Type the XY retract amount, and press **ENTER**.
   [Default: **0.0000**] Valid range: (0.0100 to 25.400)

Setting the Z Start Position

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.

To set the XY retract amount:

1. See Map 13**, Probing Setup, Menu I**.
2. Highlight **Z start position**, and press **ENTER**.
3. Type the Z start position, and press **ENTER**.
   [Default: **0.0000**] Valid range (0.0000 to 999.0000)

Setting the Diameter of Tool Probe Gauge

Sets the probe calibration standard diameter.

To set the XY retract amount:

1. See Map 13**, Probing Setup, Menu I**.
2. Highlight **Diameter of tool probe gauge**, and press **ENTER**.
3. Type the diameter of tool prove gauge, and press **ENTER**.
   [Default: **0.0000**] Valid range: (0.1000 to 508.0000)

Setting the Positioning Feedrate Normally

Sets the feedrate used for positioning the probe in protected mode.

To set the positioning feedrate normally amount:

1. See Map 13**, Probing Setup, Menu I**.
2. Highlight **Positioning feedrate normally**, and press **ENTER**.
3. Type the positioning feedrate normally, and press **ENTER**.
   [Default: **0.0**] Valid range: (0.1 to 25400.0)
   Typical value: 200 inches/minute (IPM).

Setting the First Touch Feedrate

Sets the feedrate used for positioning for the initial pick.

To set the positioning feedrate normally amount:

1. See Map 13**, Probing Setup, Menu I**.
2. Highlight **First touch feedrate**, and press **ENTER**.
3. Type the first touch feedrate, and press **ENTER**.
   [Default: **0.0**] Valid range: (0.1 to 2540.0)
   Typical value: 50 inches/minute (IPM)

Setting the Nominal Probe Stylus Ball Diameter

Sets the diameter of the probe stylus divided by 2..

To set the nominal probe stylus ball radius amount:

1. See Map 13**, Probing Setup, Menu I**.
2. Highlight **Nominal probe stylus ball diameter**, and press **ENTER**.
3. Type the nominal probe stylus ball radius, and press **ENTER**.
   [Default: **0.0000**] Valid range: (0.0100 to 25.4000)

Setting the Diameter of Spindle Probe Gauge

Sets the exact diameter of the ring gauge used for probe calibration.

To set the diameter of spindle probe gauge amount:

1. See Map 13**, Probing Setup, Menu I**.
2. Highlight **Diameter of spindle probe gauge**, and press **ENTER**.
3. Type the diameter of spindle probe gauge, and press **ENTER**.
   [Default: **0.0000**] Valid range: (0.1000 to 508.0000)

Setting the Probe Logic

Sets the probe logic. If the signal from the probe is Normally Closed, use the default **No** setting. If the probe signal is Normally Open, press **ENTER** to toggle to **Yes**.

To set the probe logic:

1. See Map 13**, Probing Setup, Menu I**.
2. Highlight **Invert probe logic**, and press **ENTER** to toggle to the correct logic for your probe.
   [Default: **No**]

Section 3 - Operator Setup

The Operator Setup Utility (Refer to Map 14**, Menu B**) configures settings for the following:

- Control Software
- Communications
- Draw
- Editor
- Program
- Display
- Printer

Control Software Parameters

Refer to Map 14**, Menu C**. This menu defines settings that affect the control software. Refer to **Table 3-1** for control software parameter descriptions and setting information.

Table 3-1, Control Software Parameters

Control Software Parameter	Function	Settings
Default axis display	Switches the default axis display between large and normal.	**Normal** [Default] - Configures the axis display to show Machine, Program, Target, and Distance To Go displays. Large - Configures the axis display to show enlarged X, Y, Z, U and W Program position display only.
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.). Refer to Map 14, **Menu B.**	**XY** - [Default] XZ YZ
Default units	Switches the default measurement units (Inch/MM Modes).	**Inch** [Default] - Activates Inch Mode as default. MM - Activates MM Mode as default.

(Continued…)

Table 3-1, Control Software Parameters (Continued)

Control Software Parameter	Function	Settings
Default axis values	Switches Absolute/ Incremental default mode (determines how axis values for arcs, lines, and other moves are measured).	**Absolute** [Default] - Makes every move in reference to an Absolute Zero position (Program Zero or Part Zero). Incremental – Makes each move in reference to the last programmed endpoint.
Circle adjustments	Specifies whether circle centers or endpoints will be adjusted. Circle centers require adjustment when the CNC encounters incorrect circle center or endpoint coordinates.	Center - Adjusts the position of the circle center when the CNC encounters incorrect coordinates for either a circle center or endpoint **End-point** [Default] - Adjusts the position of the circle endpoint when the CNC encounters incorrect coordinates for either a circle center or endpoint.
Circle centers	Switches the default mode for programmed circle center coordinates.	Absolute - CNC interprets programmed circle center coordinates as Absolute values. **Incremental** [Default] - CNC interprets programmed circle center coordinates as Incremental values. Modal - CNC interprets programmed circle center coordinates based on current Incremental or Absolute setting.
Maximum arc correction	Specifies the maximum amount of correction the CNC will apply to an arc block before declaring an error.	0.005000 [Default] Valid range: (0.000000 to 0.039370)
Internal accuracy	Maximum accuracy available (system resolution).	0.00000100 [Default] Valid range: (0.00000001 to 0.00100000)
External accuracy	Specifies the maximum system accuracy obtainable on a given machine (machine resolution).	0.00010000 [Default] Valid range: (0.00000001 to 0.00100000)
Compensation cutoff angle	Minimizes wasted travel on acute angle. Refer to Figure 3-1, Compensation Cutoff Angle.	15.0 degrees [Default] Valid range: (1.0 to 90.0) degrees
User macro file	Specifies macro filename created by user.	USERCANN.G [Default]

(Continued…)

Table 3-1, Control Software Parameters (Continued)

Control Software Parameter	Function	Settings
Load user macro file	Specifies whether to load user macro at system startup.	**No** [Default] - Does automatically load user macro at startup. Yes - Automatically loads user macro at startup.
Disk access marker	Activates/deactivates the Disk access marker.	**On** [Default] - Activates the Disk access marker. When the CNC is reading/writing information from/to a disk the Disk access marker appears in the upper left corner of the screen. The Disk access marker looks like a small arrow. Off - Deactivates Disk access marker.
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.	[Default: 4 MB] Valid range: (2 to 128) MB
Force simulation mode	In Simulation Mode, the CNC does not generate DAC and I/O outputs. The CNC starts in Simulation Mode. Moves can be commanded and displayed, but no actual machine movements occur.	**Yes** – Enable [Default] No – Disable Press **ENTER** to toggle the option to Yes.
Enable radius compensation error checking	Activates the tool radius compensation error checking. The error checking is designed to eliminate simple gouges caused by overcompensation.	**No** – Disable [Default] Yes – Enable Press **ENTER** to toggle the option to Yes.
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.	[Default: 5 minutes] Valid range: (0 to 20160) minutes

Compensation Cutoff

Figure 3-1, 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.

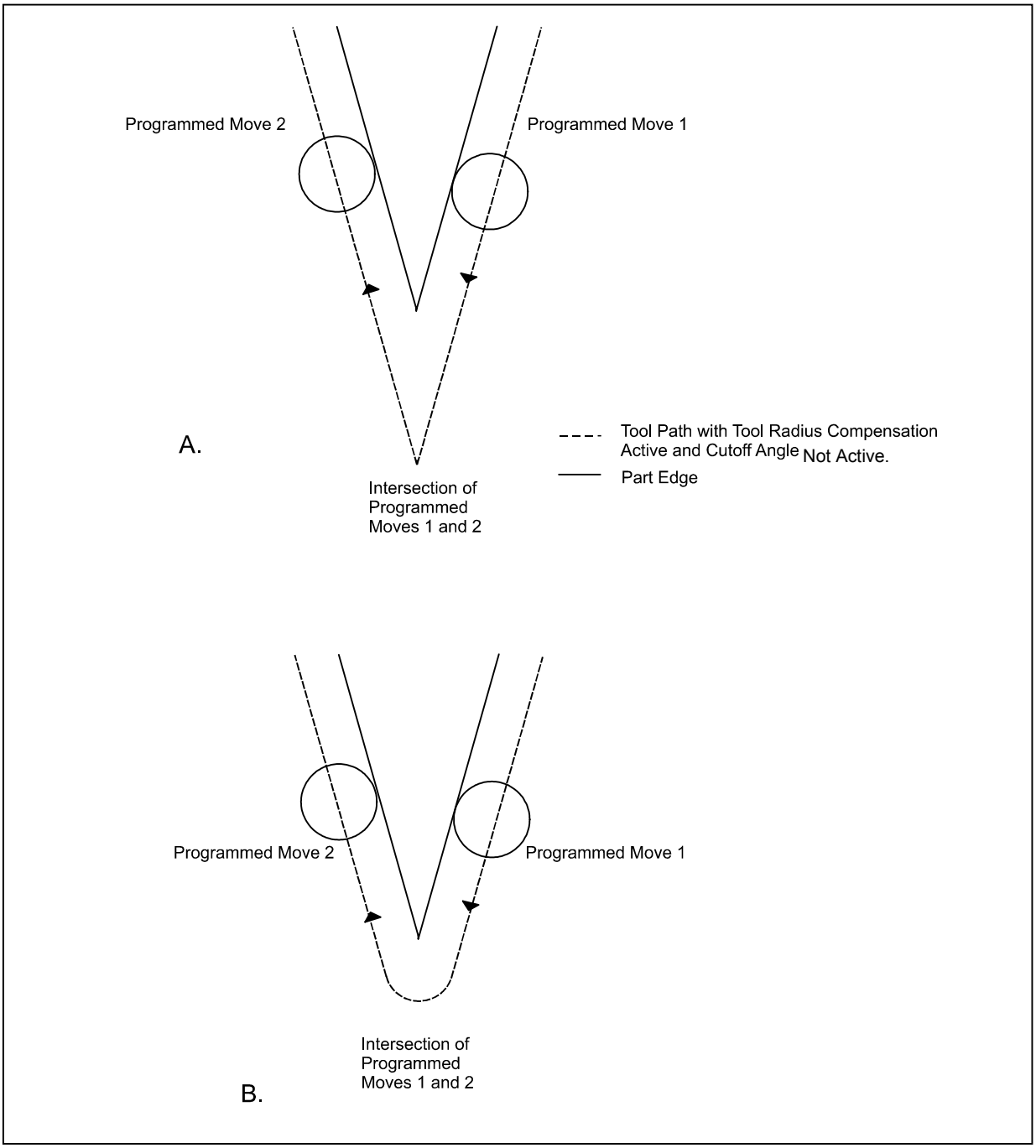


Figure 3-1, Compensation Cutoff Angle

Communication Setup Parameters

Refer to Map 15**, Menu C**. **Table 3-2** contains a description of each communications parameter and its settings.

Table 3-2, Communication Parameters

Parameter	Function	Settings
Port	Selects a communications port or disables. Must enable for remote communications.	COM1, **COM2**, Disabled [Default: COM2]
Baud	Selects a baud.	110, 150, 300, 600, 1200, 2400, 4800, **9600** [Default], or 19,200
Parity	Selects parity.	Odd, **Even** [Default], or None
Data bits	Enters number of data bits.	**7** [Default] or 8
Stop bits	Enters number of stop bits.	0 or **1** [Default]
Software	Refers to Xon or Xoff or "handshaking" (transmission/ receipt of data via RS-232) in communications packages.	**On** [Default] enables handshaking. Off disables handshaking.

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

To set up the communications parameters:

1. Refer to Map 15**, Menu C**. Highlight the desired parameter. Press **ENTER**.
2. Select one of the following options: **Port**, **Baud**, **Parity**, **Data bits**, **Stop bits**, or **Software**. Press **ENTER**.

   The CNC displays options for setting each parameter.
3. Select the desired option (see **Table 3-2**) and press **ENTER**.

NOTE: When you select **Data bits**, **Stop bits**, or **Software**, press **ENTER** to toggle the available settings.

Draw Mode Setup Parameters

Refer to Map 15, **Menu G** for the available Draw Mode Parameters. The parameters affect both Real-Time and Simulated Draw Modes. **Table 3-3** contains a description of each parameter and its settings.

Table 3-3, Draw Mode Setup Parameters

Parameter	Function	Settings
Restore 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.
Default program block mode	Sets default mode in Draw.	**Auto** [Default] S.Step Motion
Display program text	Determines whether program text appears in Draw Mode.	**Yes** - Shows program text. [Default] No - Does not show program text.
Grid	Activates/deactivates grid as a dotted or solid line.	**None** - Deactivates grid. [Default] Solid - Activates solid line grid. Dotted - Activates dotted line grid.
Grid size	Determines the size of the grid (in the active Inch or MM Mode).	Enter a value in the entry field. [Default: 1.0] Value range: (0.0 to 1000.0) If the CNC is in Inch Mode, each square in the grid will be one square inch in size for this setting.) **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).
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.
Default tool type	Determines shape of displayed tool.	None - No tool shown. **Flat** - Flat-end tool shown. [Default] Ball - Ball-end tool shown.

Table 3-3, Draw Mode Setup Parameters (Continued)

Parameter	Function	Settings
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]
Draw view	Determines perspective of Draw view.	XY (top view) - displays program in X and Y. XZ - displays program in X and Z. YZ - displays program in Y and Z. **ISO** - displays program in X, Y, and Z. [Default]
Aspect ratio correction factor	Corrects for distortion in displayed graphics (flattened circles, etc.).	1.47 [Default] Valid range: (0.01 to 10.00) **Caution: Only qualified technicians should adjust this setting.**
Save Draw image	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.

Editor Mode Setup Parameters

Refer to Map 15, **Menu L** for the available Editor Mode Parameters. **Table 3-4** contains a description of each parameter and its settings.

Table 3-4, Editor Parameters

Parameter	Function	Settings
Restore 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.
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.
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. **Off** - Does not automatically set Insert Mode as default. [Default]
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.
Update screen on macros	When non-programmed macros are used (Play, Record and Repeat options in the Control Software Edit Mode), this parameter defines how the CNC display will be updated.	**Yes** - Updates screen (prints macro results) line by line as macro is run. [Default] No - Updates screen (prints macro results) after macro is completed.
Default tab width	This option is available only with off-line 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.	Enter a value in the entry field. [Default: 4] Valid range: (2 to 16) spaces

Table 3-4, Editor Parameters (Continued)

Parameter	Function	Settings
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]
Delete internal file when program saved	When the program is saved, the CNC automatically deletes the existing internal file (*.S files) and replaces it with the saved file.	**Yes** - Deletes internal file when you save a file and replaces it with updated file. [Default] No - CNC does not delete internal file when you save a program.
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]
Memory reserved from Editor (in K-bytes)	Specifies the maximum memory allocation for the Editor.	Enter a value in the entry field. [Default: 300 K] Valid range: (16 to 32000) K

Program Directory Parameters

Refer to Map 16, **Menu C** for the available Program Parameters. 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

Table 3-5 contains a description of each parameter and its settings.

Table 3-5, Program Directory Parameters

Parameter	Function	Settings
Program directory pattern	Type of programs displayed.	*.G (program file extension)
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.
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 Extension. Size - By file size. Date - By date program was created.
Automatically check disk at startup	For machines equipped with hard drives, specifies whether and how often CNC will check the hard drive. **NOTE:** Disk Check is not available under any Windows operating system. If you select it, the CNC displays a message to inform you that the feature is disabled.	Always Daily **Weekly** [Default] Monthly Never
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.
Directory for user programs	CNC will store user programs in specified directory.	C:\USER [Default] Enter user directory location.

Display Settings

Refer to Map 16, **Menu F** 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.

Printer Settings

Refer to Map 16, **Menu G** for the available Printer Parameters. See **Table 3-6** for a description of each parameter and its settings.

Table 3-6, Printer Parameters and Selections

Printer Parameter	Function	Settings
Default output device	Specifies where file will be printed.	To send file to the printer, enter **PRN**. To print to another file, type: drive, path, and filename with extension. If the filename typed is not a current file, the CNC creates the file and transfers the data to the file. If the filename typed is an existing file, the CNC overwrites the data in the file with the print file data. [Default: PRN]. **NOTE:** The user directory for CNC systems is C:\USER
Lines per page	Number of lines to be printed per page (8.5 X 11”).	Enter a value in the entry field. [Default: 55] Valid range: (1 to 66) lines per page
Page heading	Prints a page heading including filename, date and time, and page number.	**Yes** - Prints heading. [Default] No - Does not print heading.
Line numbers	Prints line numbers on hard copy of file.	Yes - Prints line numbers. **No** - Select No if no line numbers are desired. [Default]
Print quality	Sets print quality. Generally, the lower the printer quality, the faster the file prints.	NLQ - Specifies near letter quality, highest quality, and lowest speed. Utility - Specifies middle quality, middle speed. High Speed - Specifies low quality, high speed. **Ignore** -Uses printer defaults. [Default]

(Continued…)

Table 3-6, Printer Parameters and Selections (Continued)

Printer Parameter	Function	Settings
Characters per inch	Sets the number of characters to be printed per inch. Select **Ignore** to print at the default value.	**10** - Specifies 10 characters per Inch. [Default] 12 - Specifies 12 characters per Inch. 17 - Specifies 17 characters per Inch. 20 - Specifies 20 characters per Inch. Ignore - Uses printer defaults.
Wrap text	Wraps text to the next line if program is greater than 80 characters.	**Yes** - Wraps text. [Default] No **-** Truncates text.

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 17**, 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 P5MCFG.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 17**, 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:\P5MCFG.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:\P5MCFG.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 17**, 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:\P5MCFG.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 17**, 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 17**, Menu B**. Highlight **Compare Configuration**. Press **ENTER**. **Menu E** displays.
2. Highlight **A:** to compare the current file with P5MCFG.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 17**, 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 - Fine-Tuning Systems with Linear Encoders

On systems equipped with linear encoders, make the following Setup Utility changes to minimize the effects of lost motion.

NOTE: This procedure requires you to move between the Setup Utility and the CNC's Manual screen. Whenever a change is made to the Setup Utility, make sure to save the changes when prompted.

Lost motion is the distance the ballscrew and/or motor moves before the table begins to move. It is the result of the mechanical characteristics of the motor and ballscrew.

Since a linear encoder measures table motion, movement corrections required by mechanical characteristics or servo drift can only be made after the CNC detects table movement.

The procedure compensates for the observed lost motion by fine-tuning some of the motion control setup parameters.

Make changes in the Setup Utility only after an axis is aligned and stable. Before you change the Setup, balance the servo cards, set the signal gain, adjust the PID filter gains and make sure the following error (lag) is the same on all axes.

1. In the CNC's Setup Utility, ensure that the CNC is set to **linear encoder**, with the proper encoder resolution and display resolution settings.
2. Use the AXIS Selector Switch to select the required axis for a manual move.
3. Use the JOG Selector Switch to set the CNC to Jog Mode 1. This sets the axis to move in increments of 1 times the machine resolution each time you press a **JOG** key.
4. Use the **PLUS** or **MINUS JOG** key to move the axis in a positive or negative direction.
5. Use the **PLUS** or **MINUS JOG** key to move the axis in the opposite direction.
6. Look at the ballscrew (or handwheel) of the axis. When lost motion occurs, the ballscrew moves a much greater distance than the system resolution (for example, 0.0005" on a 10-micron linear encoder). That distance is the lost motion on that axis.

NOTE: It is acceptable for the system to move a couple of times the lost motion amount to adjust itself, as long as the lost motion does not affect table movement or become a continuous oscillation (hunting).

7. If your system is oscillating (hunting) within that distance, make the following adjustments within the Setup screen:

   A. Check the Servo Loop Sample Time. With a 10-micron resolution linear encoder, set the servo sampling time to 0.800-msec. With 1, 2, or 5-micron resolution linear encoders, the value should remain at the default value, 0.400-msec. (Use increments of 0.050-msec when making minor adjustments to this value.)

   B. Eliminate the hunting created by the lost motion. While the axis is not moving, adjust the No-Motion Filter parameters to limit the reaction time of the servo's digital filter, as follows:

      - Eliminate any integral gain by setting the Ki (integral gain) and IL (integral limit) to 0. Integral gain accumulates over time; increasing the correction output and instigating oscillation due to lost motion.

      - Set the Kp (proportional gain) value to 1.50. If oscillation continues, reduce Kp in steps of 0.10, until oscillation stops.

NOTE: Lowering the Kp gain delays the correction time not only to lost motion, but also to drift offsets on the system.

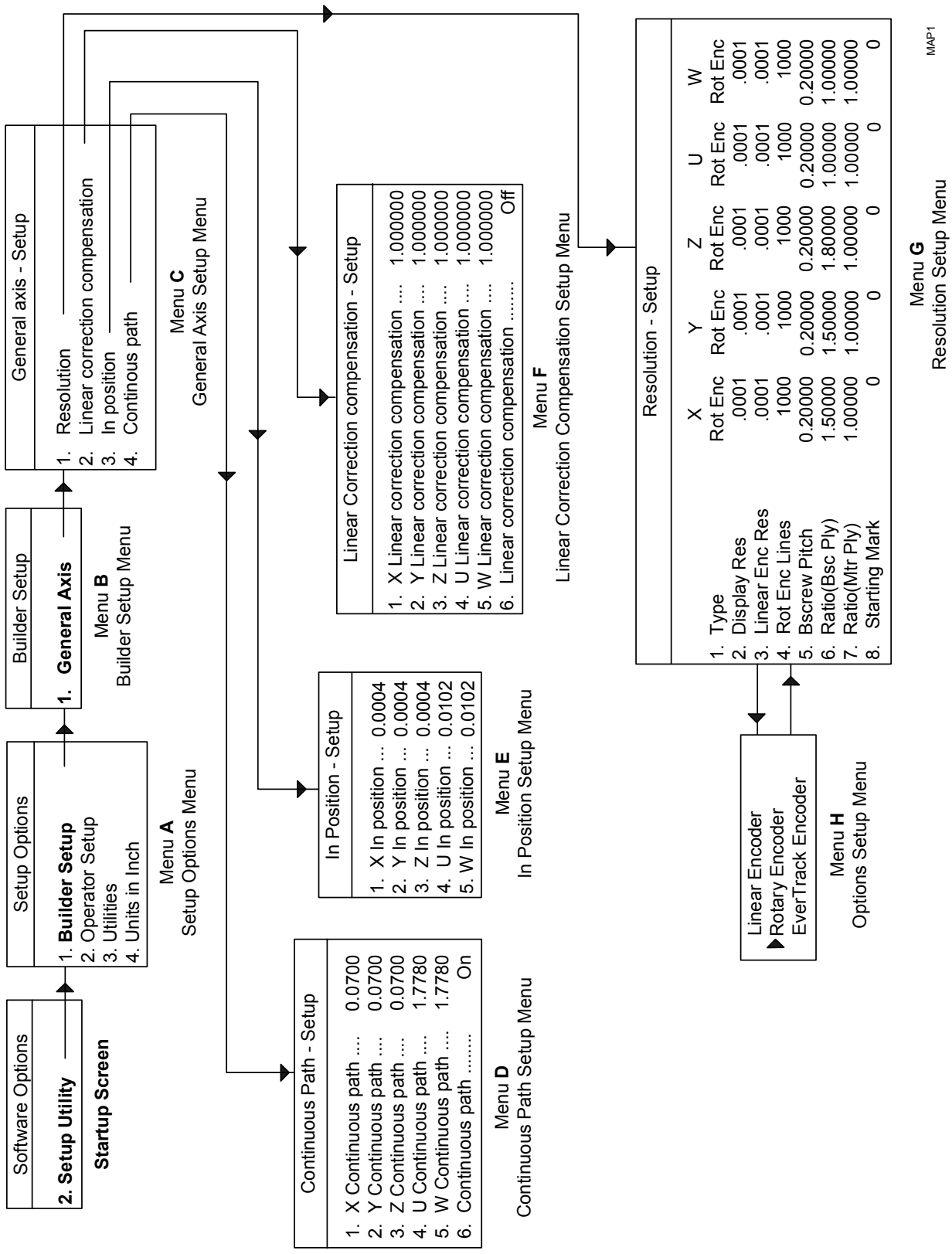
8. Repeat the procedure on all enabled axes.

9. From the CNC's Manual screen, use the **JOG** keys in Feed Mode to test the responsiveness of the axis. If an axis coasts when the **JOG** key is released, reduce the Feed Accel/Decel setting in the Setup screen. Use decrements of 10-msec when adjusting this parameter.

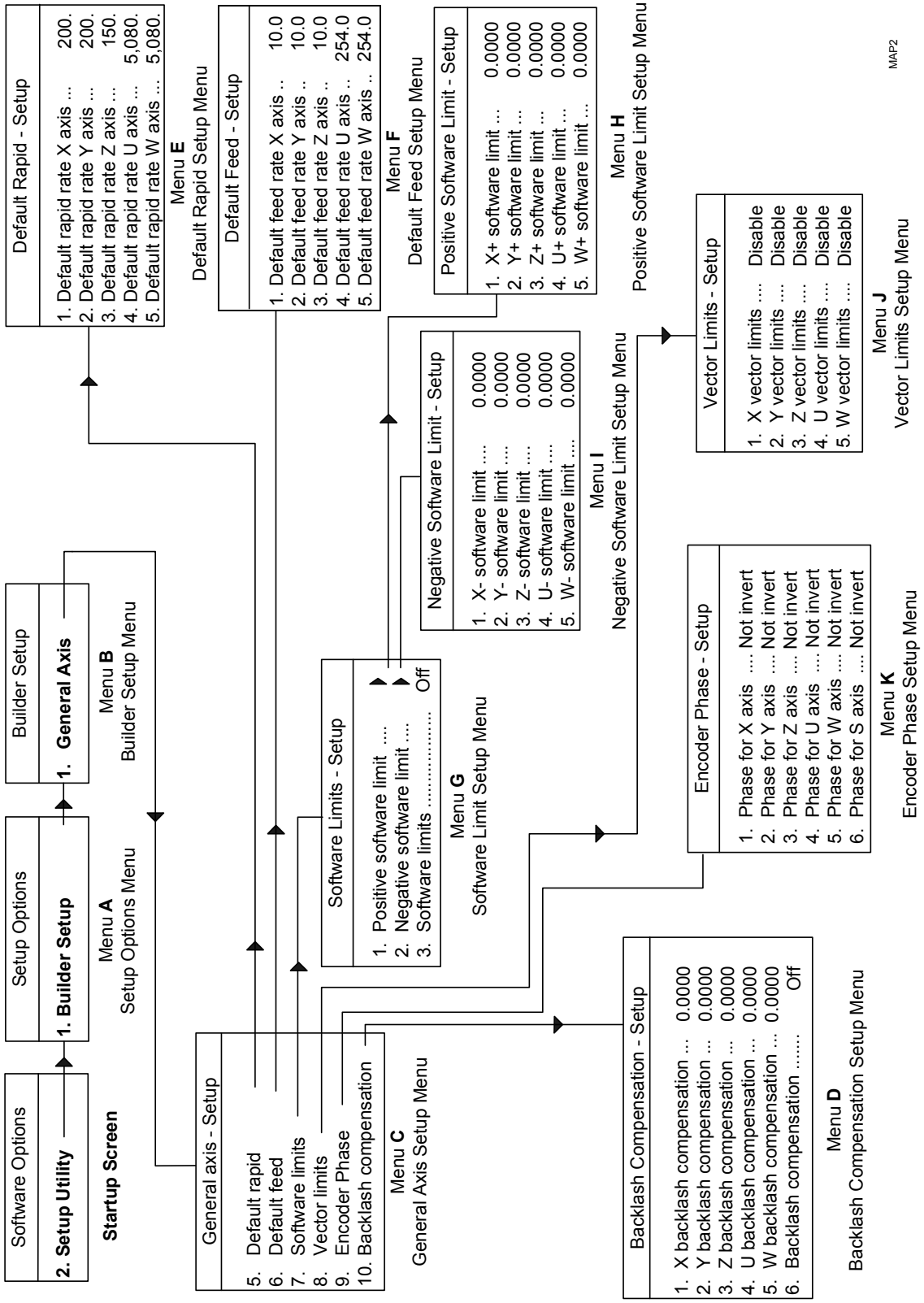
NOTE: Always make minor adjustments to Setup Utility parameters and check until the problem being corrected is fixed. If you make large changes to parameters, you may fix the problem at hand, but cause others.

Section 6 - Setup Utility Maps

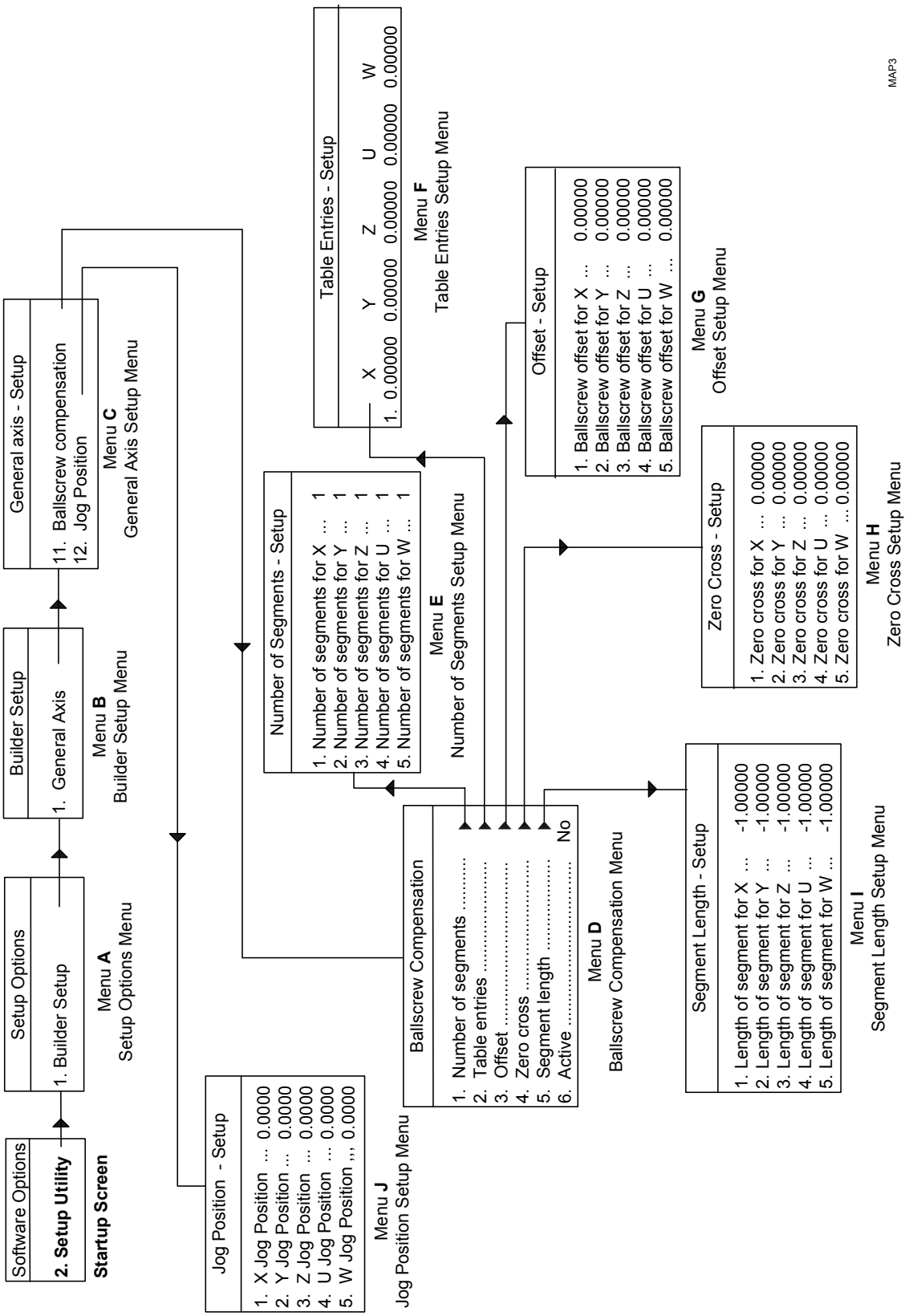
This section contains maps referenced in Sections 2 through 4. Refer to "Section 1 - Setup Utility Concepts" for instructions on how to use the software and maps.



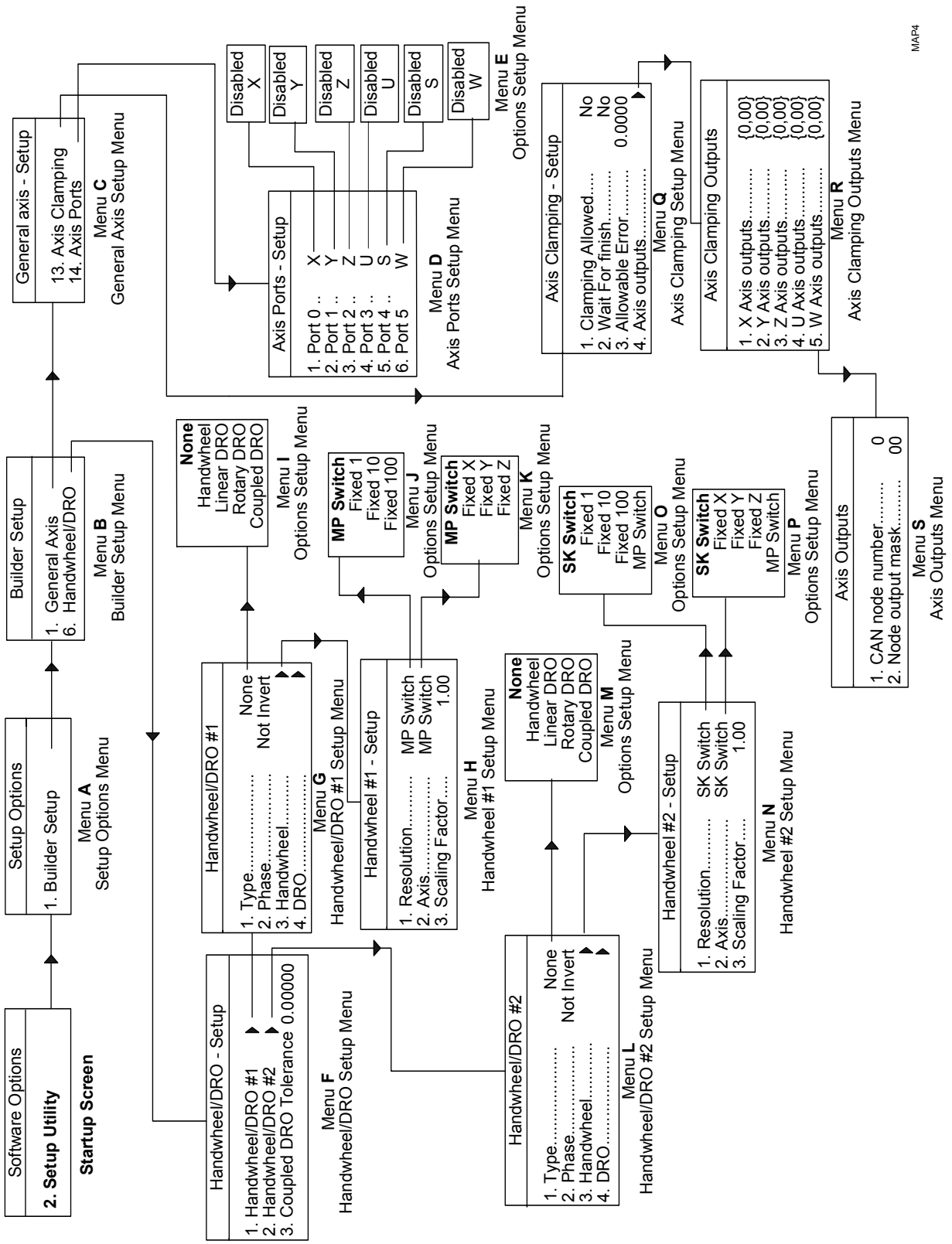
Map 1



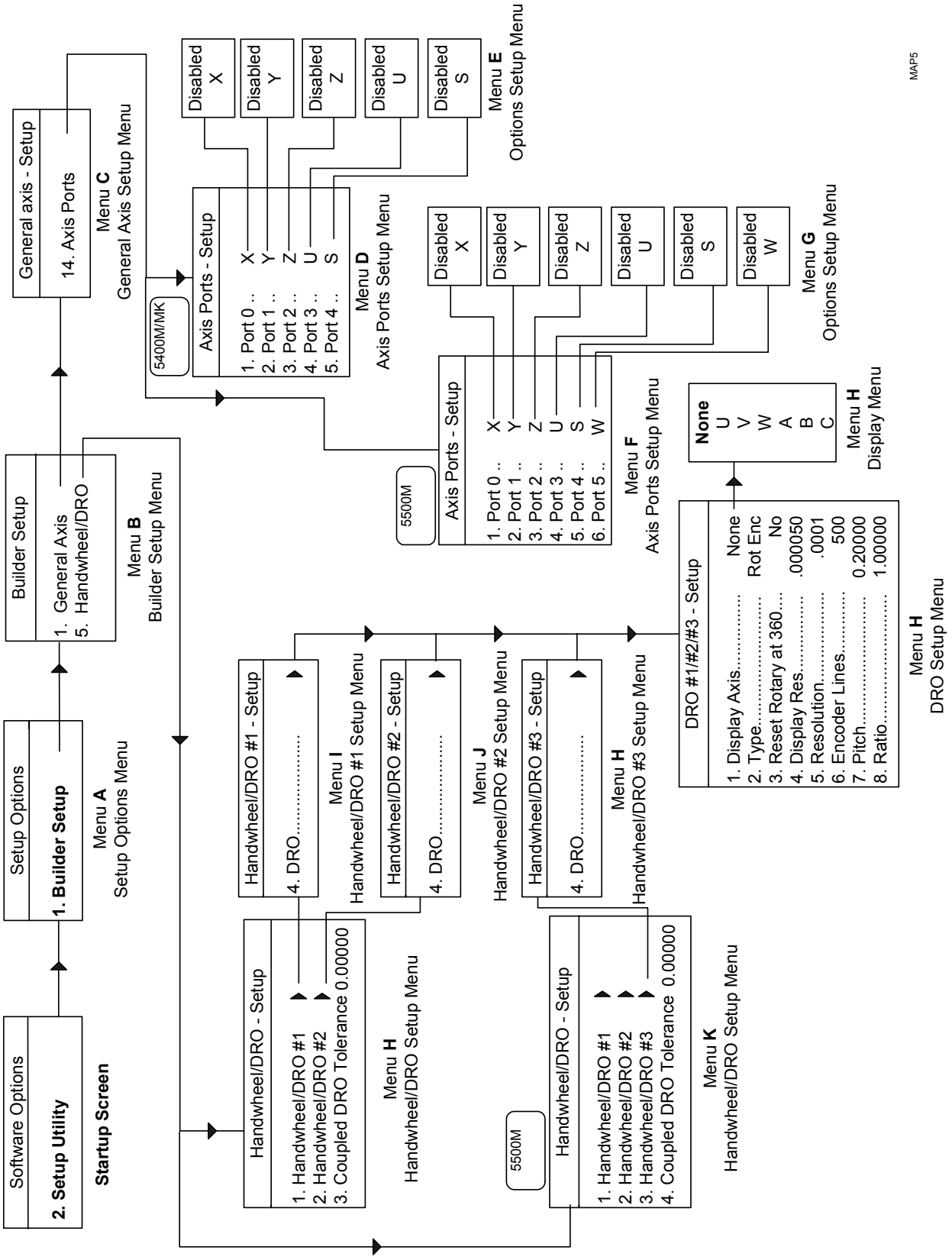
Map 2



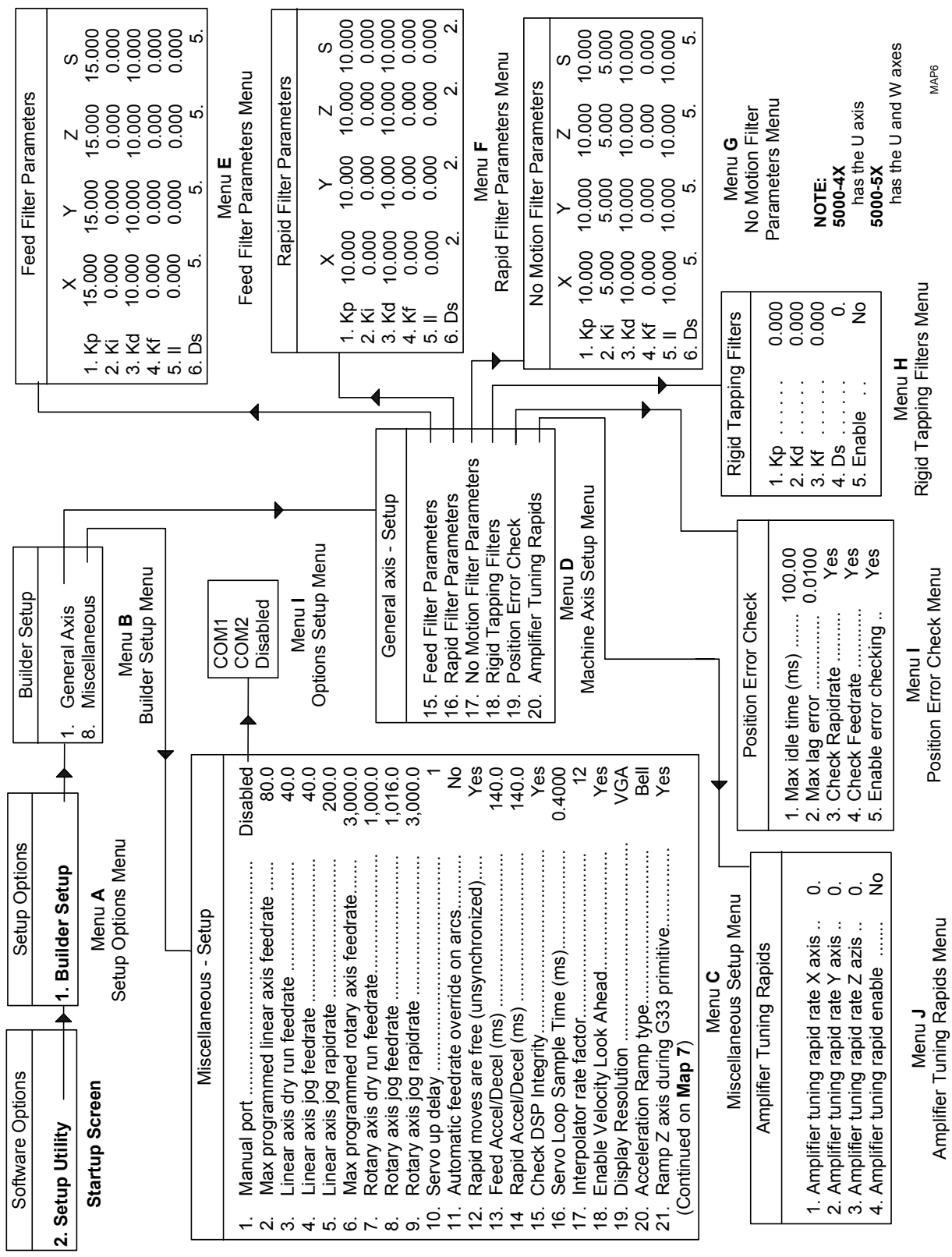
Map 3



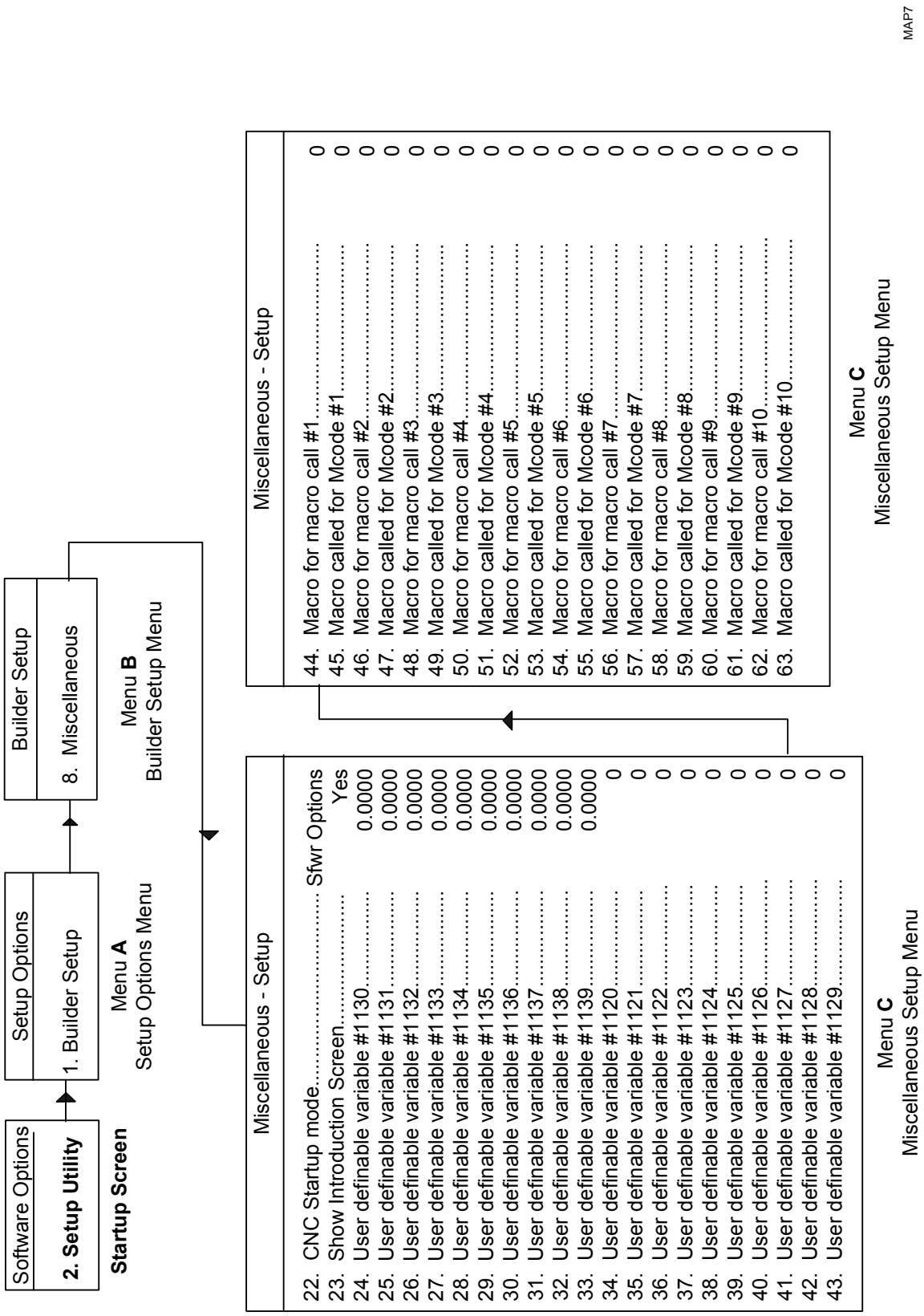
Map 4



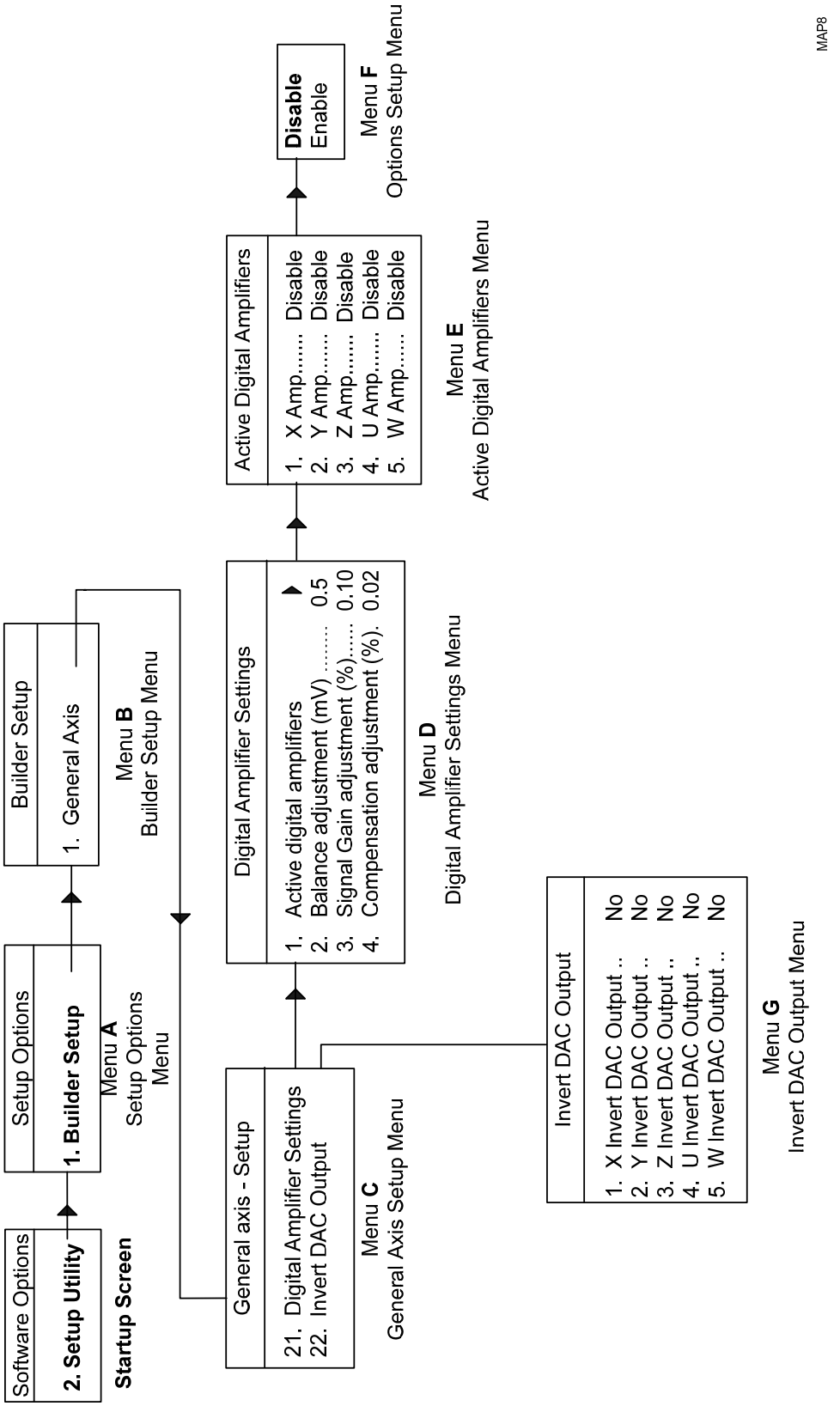
Map 5


Software Options
2. Setup Utility
Startup Screen
Setup Options
1. Builder Setup
Menu A
Setup Options Menu
Builder Setup
1. General Axis
8. Miscellaneous
Menu B
Builder Setup Menu
Miscellaneous - Setup
1. Manual port ..... Disabled
2. Max programmed linear axis feedrate 80.0
3. Linear axis dry run feedrate 40.0
4. Linear axis jog feedrate 40.0
5. Linear axis jog rapidrate 200.0
6. Max programmed rotary axis feedrate..... 3,000.0
7. Rotary axis dry run feedrate..... 1,000.0
8. Rotary axis jog feedrate 1,016.0
9. Rotary axis jog rapidrate 3,000.0
10. Servo up delay 1
11. Automatic feedrate override on arcs..... No
12. Rapid moves are free (unsynchronized).... Yes
13. Feed Accel/Decel (ms) 140.0
14 Rapid Accel/Decel (ms) 140.0
15. Check DSP Integrity..... Yes
16. Servo Loop Sample Time (ms)..... 0.4000
17. Interpolator rate factor..... 12
18. Enable Velocity Look Ahead..... Yes
19. Display Resolution ..... VGA
20. Acceleration Ramp type..... Bell
21. Ramp Z axis during G33 primitive..... Yes
(Continued on **Map 7**)
COM1
COM2
Disabled
Menu I
Options Setup Menu
General axis - Setup
15. Feed Filter Parameters
16. Rapid Filter Parameters
17. No Motion Filter Parameters
18. Rigid Tapping Filters
19. Position Error Check
20. Amplifier Tuning Rapids
Menu D
Machine Axis Setup Menu
Feed Filter Parameters
X Y Z S
1. Kp 15.000 15.000 15.000 15.000
2. Ki 0.000 0.000 0.000 0.000
3. Kd 10.000 10.000 10.000 10.000
4. Kf 0.000 0.000 0.000 0.000
5. II 0.000 0.000 0.000 0.000
6. Ds 5. 5. 5. 5.
Menu E
Feed Filter Parameters Menu
Rapid Filter Parameters
X Y Z S
1. Kp 10.000 10.000 10.000 10.000
2. Ki 0.000 0.000 0.000 0.000
3. Kd 10.000 10.000 10.000 10.000
4. Kf 0.000 0.000 0.000 0.000
5. II 0.000 0.000 0.000 0.000
6. Ds 2. 2. 2. 2.
Menu F
Rapid Filter Parameters Menu
No Motion Filter Parameters
X Y Z S
1. Kp 10.000 10.000 10.000 10.000
2. Ki 5.000 5.000 5.000 5.000
3. Kd 10.000 10.000 10.000 10.000
4. Kf 0.000 0.000 0.000 0.000
5. II 10.000 10.000 10.000 10.000
6. Ds 5. 5. 5. 5.
Menu G
No Motion Filter
Parameters Menu
Rigid Tapping Filters
1. Kp 0.000
2. Kd 0.000
3. Kf 0.000
4. Ds ..... 0.
5. Enable .. No
Menu H
Rigid Tapping Filters Menu
Position Error Check
1. Max idle time (ms) 100.00
2. Max lag error 0.0100
3. Check Rapidrate ..... Yes
4. Check Feedrate ..... Yes
5. Enable error checking .. Yes
Menu I
Position Error Check Menu
Miscellaneous Setup Menu
Menu C
Amplifier Tuning Rapids
1. Amplifier tuning rapid rate X axis .. 0.
2. Amplifier tuning rapid rate Y axis .. 0.
3. Amplifier tuning rapid rate Z azis .. 0.
4. Amplifier tuning rapid enable ..... No
Menu J
Amplifier Tuning Rapids Menu
NOTE:
5000-4X
has the U axis
5000-5X
has the U and W axes
MAP6

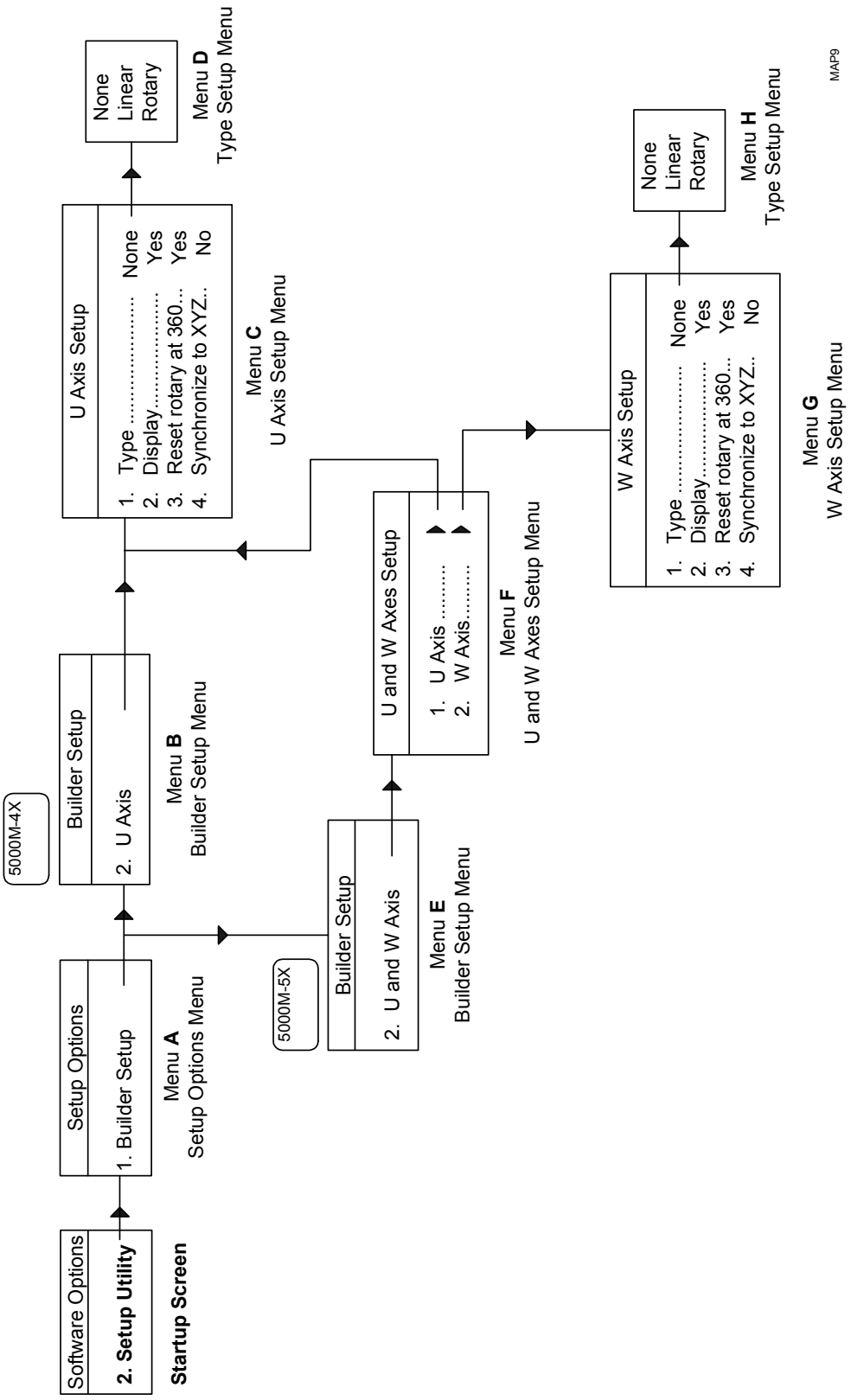
Map 6



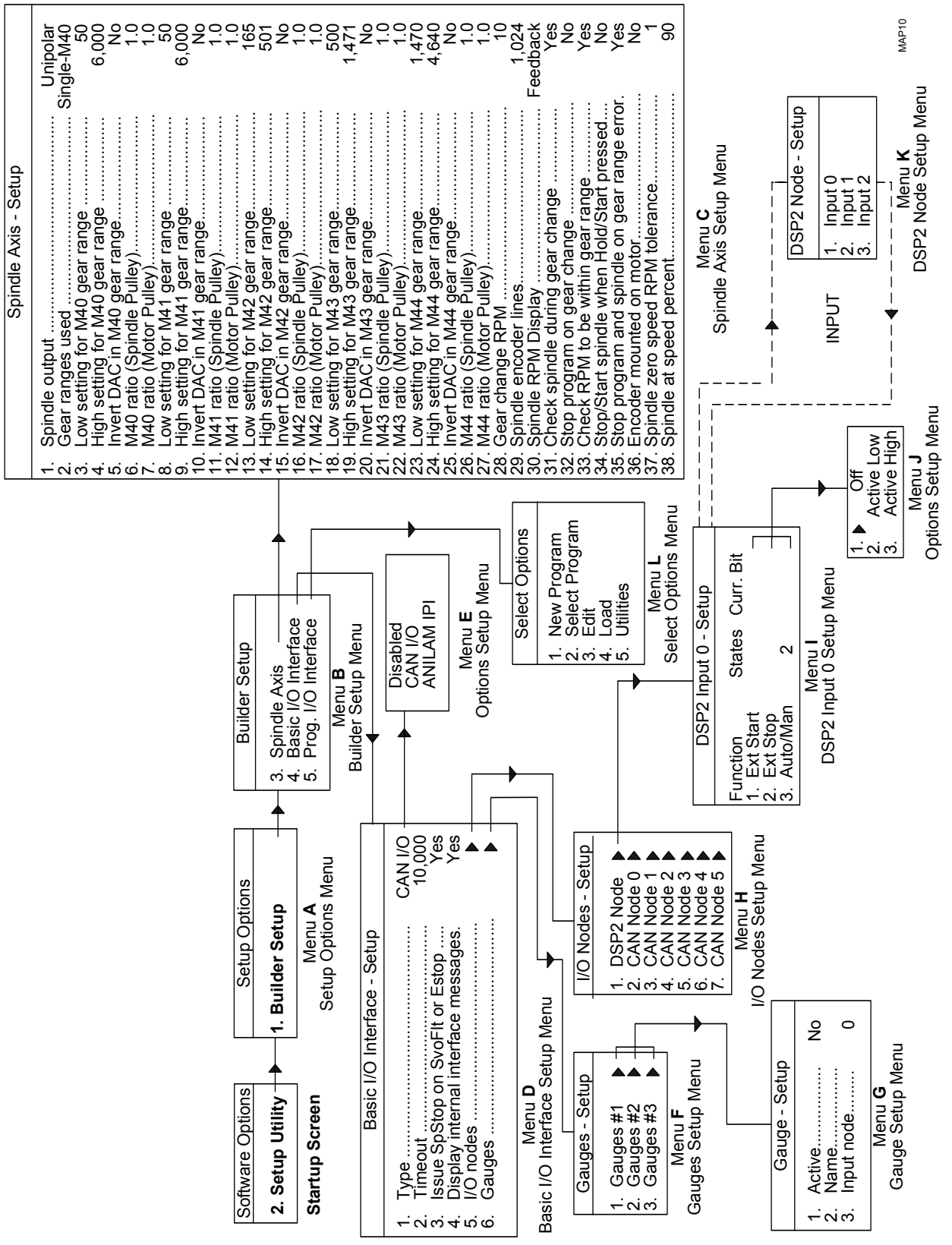
Map 7



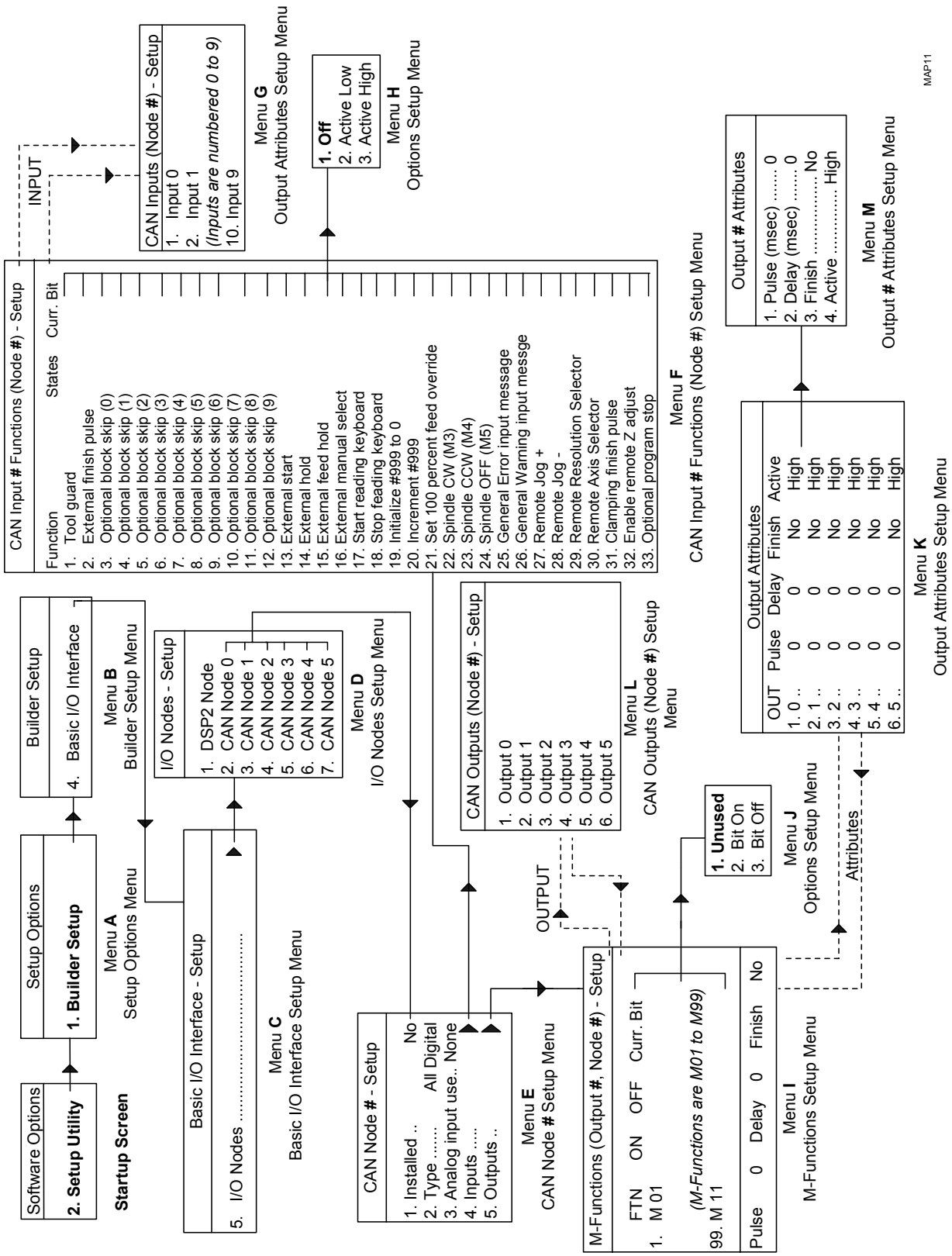
Map 8



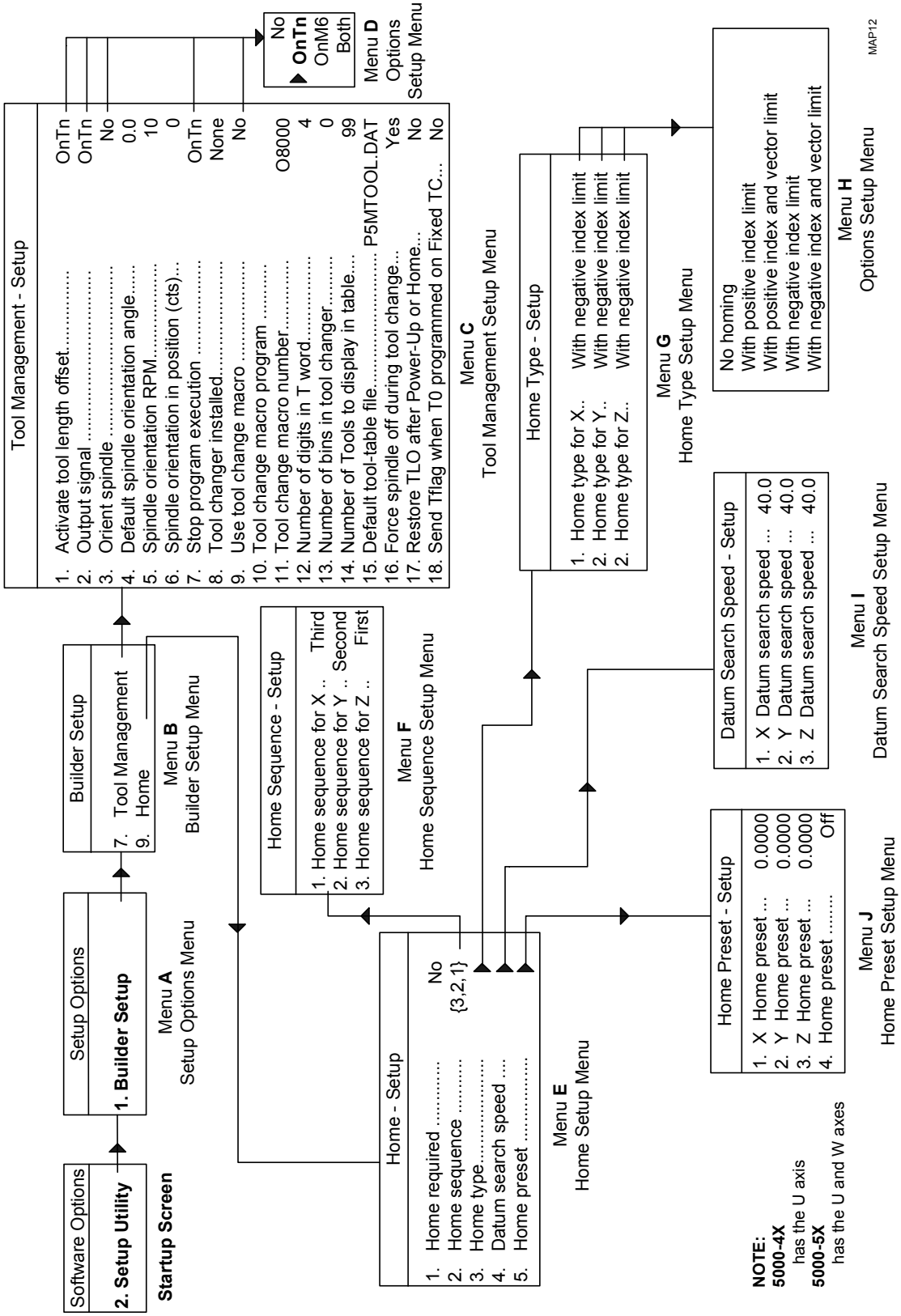
Map 9



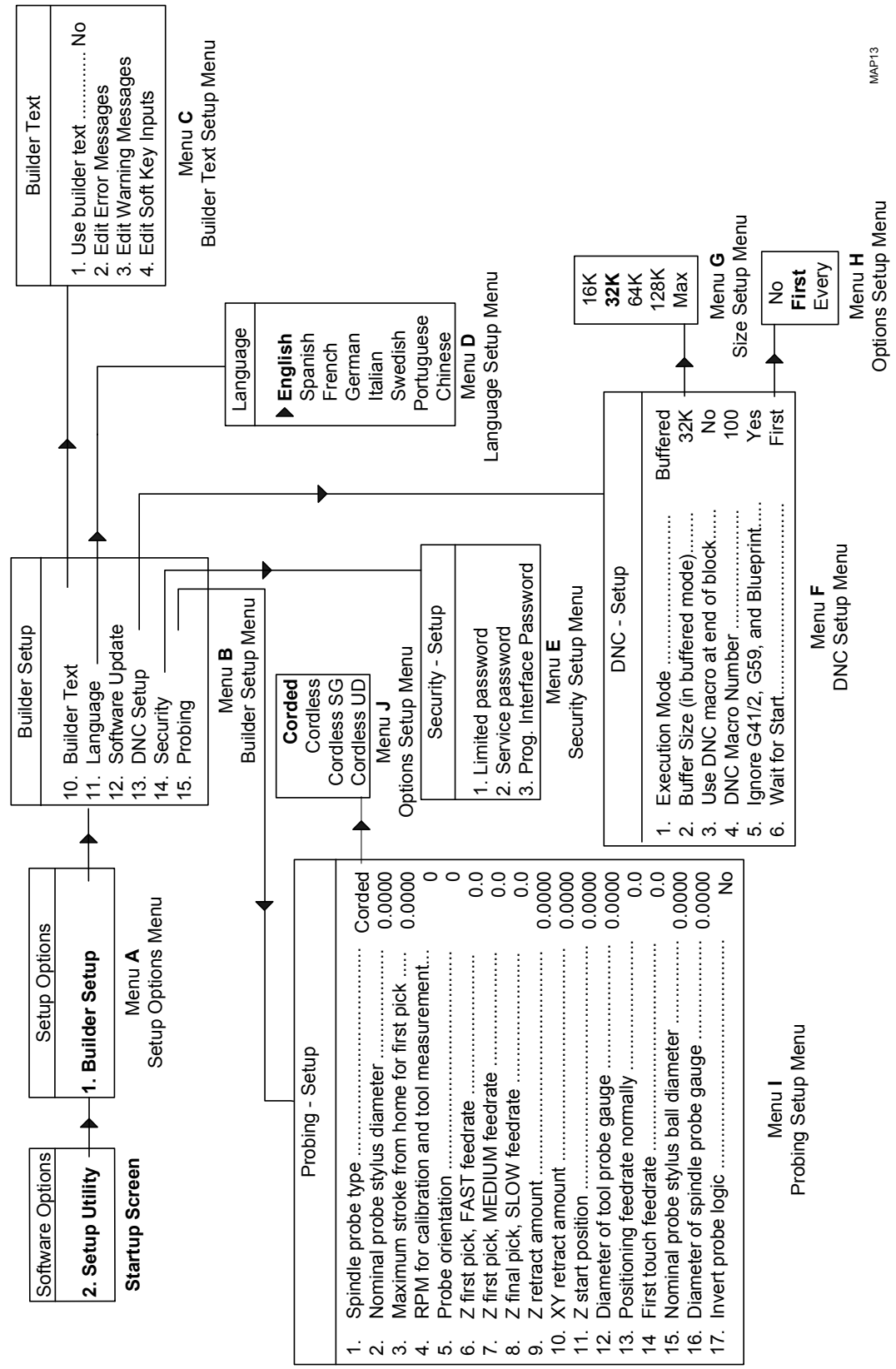
Map 10



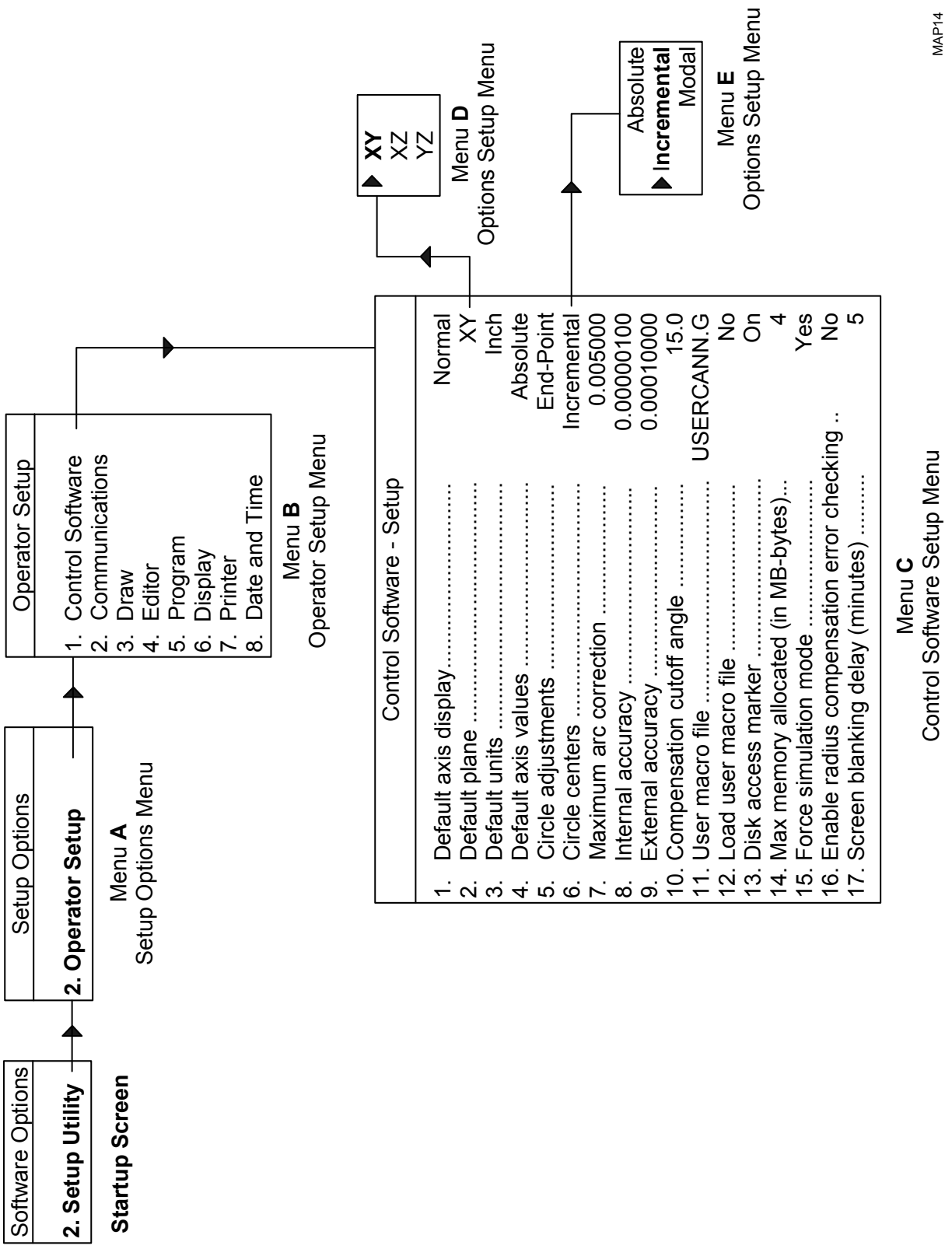
Map 11


Software Options
2. Setup Utility
Startup Screen
Setup Options
1. Builder Setup
Menu A
Setup Options Menu
Builder Setup
7. Tool Management
9. Home
Menu B
Builder Setup Menu
Tool Management - Setup
1. Activate tool length offset..... OnTn
2. Output signal ..... OnTn
3. Orient spindle ..... No
4. Default spindle orientation angle..... 0.0
5. Spindle orientation RPM..... 10
6. Spindle orientation in position (cts)... 0
7. Stop program execution ..... OnTn
8. Tool changer installed..... None
9. Use tool change macro ..... No
10. Tool change macro program O8000
11. Tool change macro number..... 4
12. Number of digits in T word..... 0
13. Number of bins in tool changer..... 99
14. Number of Tools to display in table...
15. Default tool-table file..... P5MTOOL.DAT
16. Force spindle off during tool change... Yes
17. Restore TLO after Power-Up or Home... No
18. Send Tflag when T0 programmed on Fixed TC... No
Menu C
Tool Management Setup Menu
No
OnTn
OnM6
Both
Menu D
Options
Setup Menu
Home Sequence - Setup
1. Home sequence for X .. Third
2. Home sequence for Y .. Second
3. Home sequence for Z .. First
Menu F
Home Sequence Setup Menu
Home - Setup
1. Home required ..... No
2. Home sequence ..... {3,2,1}
3. Home type.....
4. Datum search speed ....
5. Home preset .....
Menu E
Home Setup Menu
Home Type - Setup
1. Home type for X.. With negative index limit
2. Home type for Y.. With negative index limit
2. Home type for Z.. With negative index limit
Menu G
Home Type Setup Menu
No homing
With positive index limit
With positive index and vector limit
With negative index limit
With negative index and vector limit
Menu H
Options Setup Menu
Datum Search Speed - Setup
1. X Datum search speed ... 40.0
2. Y Datum search speed ... 40.0
3. Z Datum search speed ... 40.0
Menu I
Datum Search Speed Setup Menu
Home Preset - Setup
1. X Home preset ... 0.0000
2. Y Home preset ... 0.0000
3. Z Home preset ... 0.0000
4. Home preset ..... Off
Menu J
Home Preset Setup Menu
NOTE:
5000-4X
has the U axis
5000-5X
has the U and W axes
MAP12

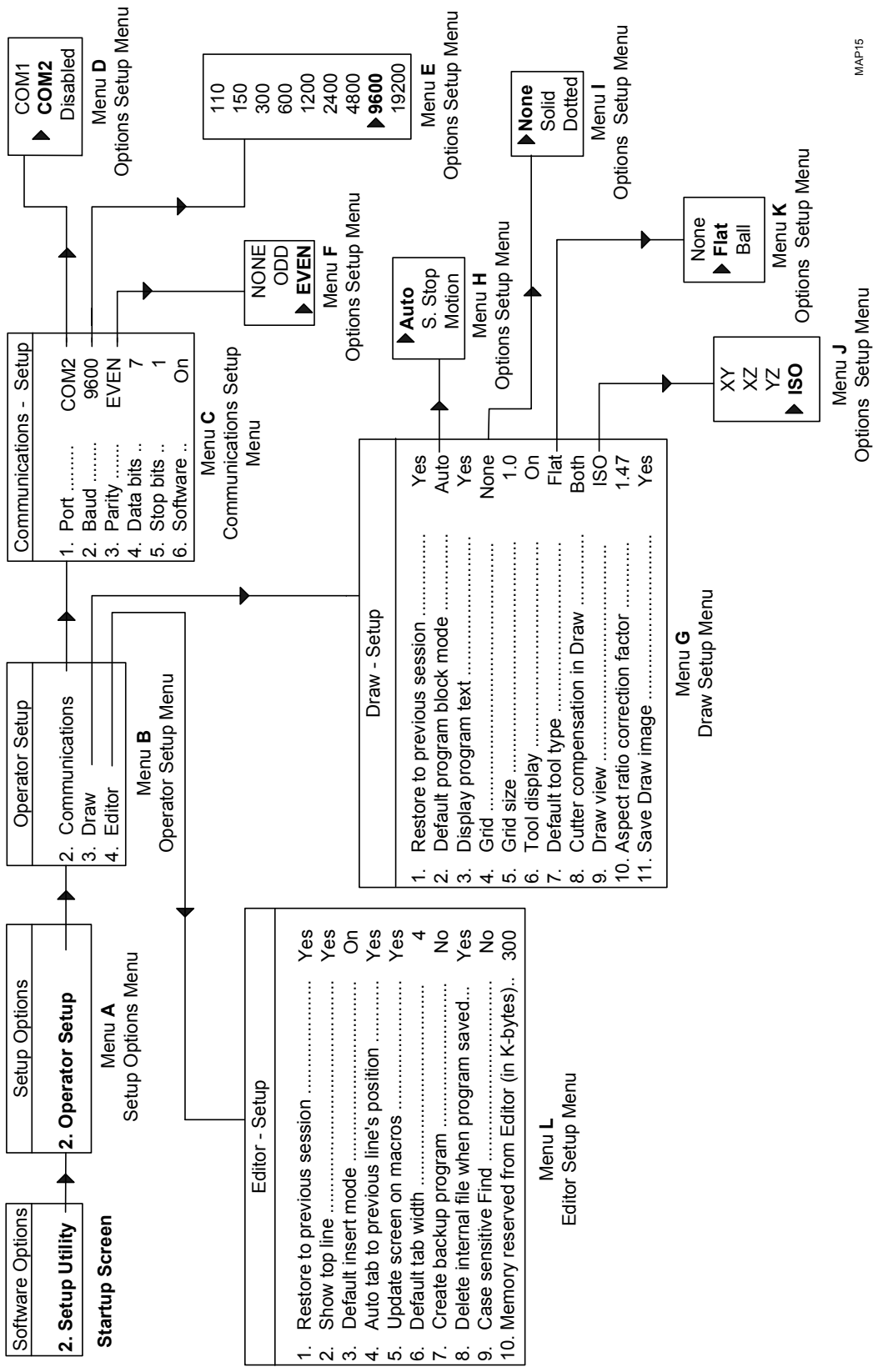
Map 12



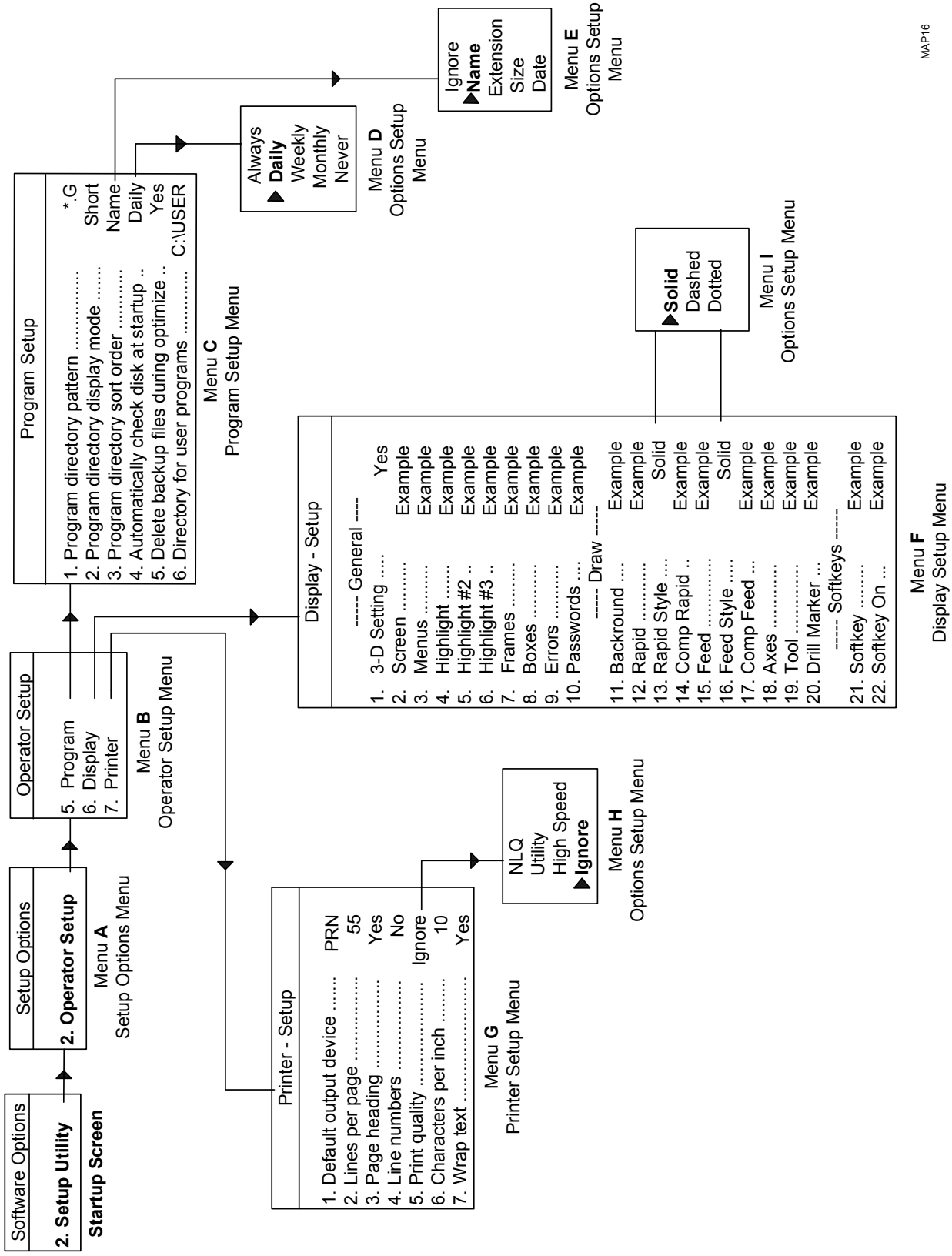
Map 13



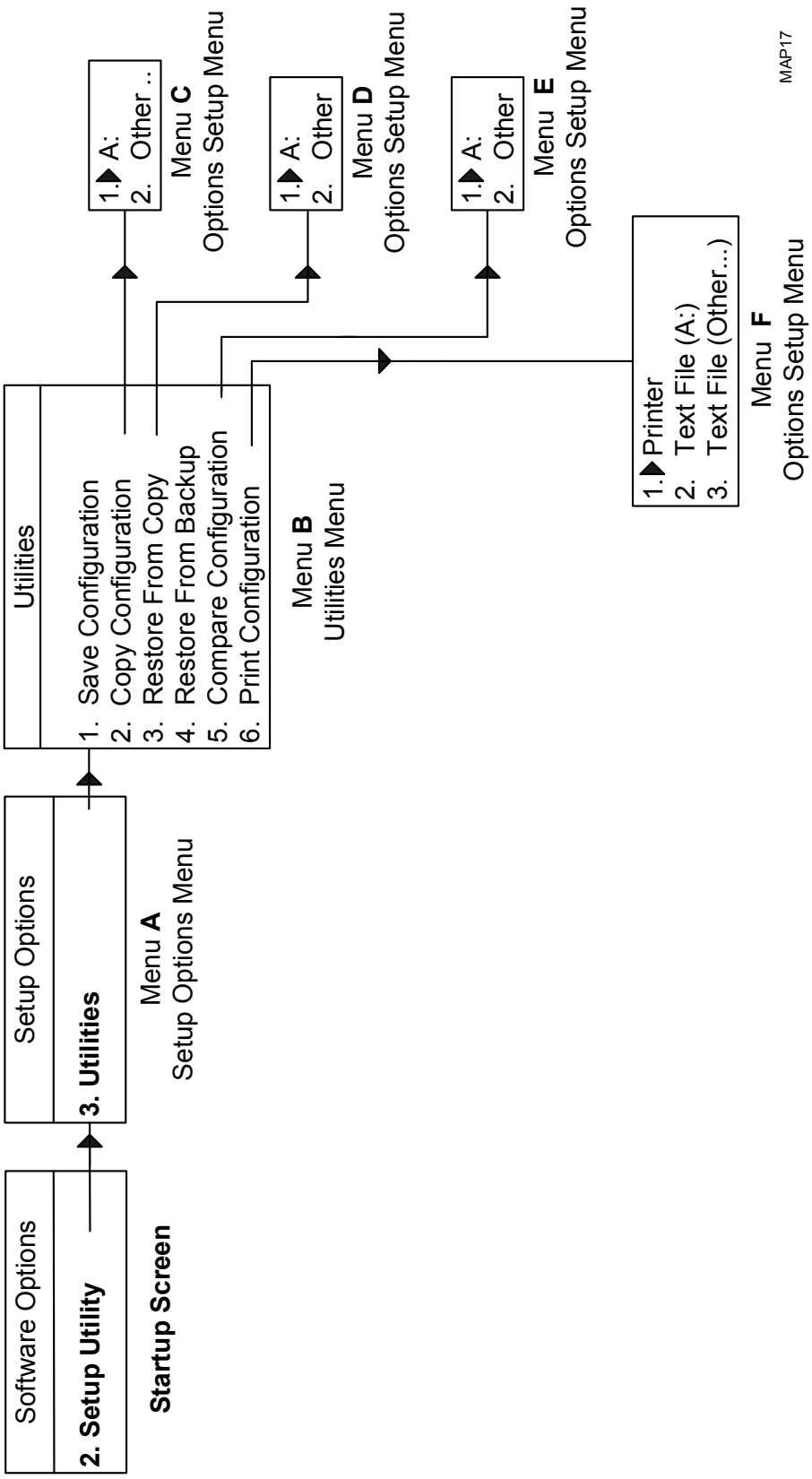
Map 14



Map 15



Map 16



Map 17

A

E

N

O

S

X

Y

Z

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.

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, R.O.C.
☎ +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