ANILAM

3000M 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 - Machine Setup

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 - Machine Setup

Introduction

This manual provides instructions on how to set up and operate the ANILAM Setup Utility for the 3000M. The Setup Utility provides access to 3000M settings through a series of menus and submenus. Each menu provides access to configuration settings or another menu.

Effectivity Notation

Some information does not apply to all ANILAM CNC products discussed in this manual. Therefore, icons identify products to which the information applies. Refer to **Table 1-1**. There are many parameters that are defined per axis. In these cases, this manual will mostly document the primary axes (i.e., XYZ). The parameters for the auxiliary axis (i.e., U) 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).

Table 1-1, Effectivity Notation

Icon	Product
3000M-2X	3000M Two-Axis Systems
3000M-3X	3000M Three-Axis Systems
3000M-4X	3000M Four-Axis Systems

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

To navigate through the Setup Utility, use the keypad keys referenced in **Table 1-2**. See "Section 6 - Setup Utility Maps," for all maps referenced in this section. Use these maps to navigate through Setup Utility software features.

To select one of the items in each Setup Utility menu, highlight the item by using the arrow keys on the keypad. Press **ENTER** to activate the selected highlight.

Press **ENTER** to switch settings On or Off, or enter 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.

Keypad Keys

Refer to **Table 1-2** for a description of the keys on the CNC console keypad. In the text, the name of a key always is displayed in small capital letters.

Table 1-2, Setup Utility Keypad Keys

Key Name	Key Face
Enter	
Arrow	
Clear	
X Axis	
Y Axis	
Z Axis	
U Axis	
Feedrate Override	
E-stop	
Start	 (Green)
Hold	 (Red)

Specifying an Axis

Press the **X**, **Y**, **Z**, or **U** dimension key to specify an axis. For example, "Press **X**" prompts the operator to press the **X**-AXIS key.

Entering Parameters

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

Highlighting Menu Options

Press **Up Arrow** (**F3**) and **Down Arrow** (**F4**) to highlight menu options in the Setup Utility.

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 that 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 1-3** for default machine passwords.

Table 1-3, 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, type 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

If changes have been made to any setup parameters, a **"Save Changes?"** prompt is displayed when the user exits Setup Utility. Select **Yes** (**F1**) to save changes, **No** (**F2**) to cancel changes, and **Cancel** (**F9**) to return to Setup Utility.

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

All configuration parameters are saved in a configuration file (P3MCFG.CFG). Every time a parameter changes and the configuration file is saved, the CNC automatically creates a backup file (P3MCFG.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**, perform the following steps:

1. Highlight the menu in which the parameter is displayed, and press **ENTER**.
2. Use one of the following three methods to set parameters:
   - If a default selection is listed in an entry field, press **ENTER** to cycle through the available selections.
   - If a pop-up menu is displayed after the parameter is selected, use the **ARROW** keys to highlight the required option, and press **ENTER**.
   - If an entry field highlights after the parameter is selected, type the required value or setting, and press **ENTER**.

Accessing Setup Utility Menus

Refer to the Startup Screen on "Map 1" in "Section 6."

To access **Setup Utility** menus, perform the following steps:

1. Turn on the CNC.

   After the CNC is turned on, the software starts automatically. The CNC displays messages to indicate the status of the startup. When the CNC software has successfully started, it displays a screen with ANILAM company information and the software version number.

2. Press **F10** to continue. The **Software Options Menu** is displayed.
3. Highlight **Setup Utility**, and press **ENTER**. The **Setup Options Menu, Menu A**, is displayed.

Overview of Main Parameter Categories

There are, in general, two categories of parameters. The first category, located under the **Builder Setup** menu entry, corresponds to the type of parameters that the machine builders, or technicians, specify. The second category, located under the **Operator Setup** menu entry, corresponds to those that the CNC operator, or programmer, specify or customize. See Map 1, **Setup Options, Menu A**.

In general, Builder parameters require a 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 necessary. By using the proper units, you do not need to convert values, but you can enter data precisely (i.e., no rounding during conversion).

To set the default measurement mode, perform the following steps:

1. See Map 1**, Setup Options, Menu A**.
2. Highlight **Units in Inch**.
3. 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.

Section 2 - Builder Setup

The **Builder Setup Menu** allows access to the basic operating menus for the **X**, **Y**, **Z**, and **U**-axes. Configure each axis in the CNC through this menu.

Setting Axis Resolution Parameters

The CNC receives feedback from a linear encoder, rotary encoder, or an **EverTrack™ encoder and provides closed-loop positioning for the system. Each axis must be set for the type of feedback device used, either a linear encoder or a rotary encoder.

[Default: **Rotary Encoder**]

Setting Axes for Encoder Type

To set the axis encoder type:

1. See the Resolution Setup, Menu D on "Map 1" in "Section 6 - Setup Utility Maps." For the remainder of the document, this is described as "See Map 1**, Menu D**."
2. Highlight **Type**.
3. Press **X**, **Y**, **Z**, or **U** 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.

Setting the Display Resolution

> **NOTE:** You can display resolution in the Setup Utility in either MM Mode or Inch Mode. One micron equals 0.001mm.
> [Default: **Inch Mode**]

Enter the required resolution for each axis. Always select resolution in microns, regardless of whether the CNC is in Inch Mode or MM Mode. [Default: **2 Microns** (0.002 mm/0.0001")]

You can set the display resolution for each axis. The display resolution should be equal to or coarser than the actual resolution of the installed encoder. Changing the display resolution will not affect the accuracy of the machine.

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

Refer to **Table 2-1** for Micron-to-Inch conversion values.

Table 2-1, Micron-to-Inch Conversion

Micron	Millimeter	Inch
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”

To set display resolution:

1. See Map 1**, Resolution Setup, Menu D**.
2. Highlight **Display Res**, and press **ENTER**.
3. Press **X**, **Y**, **Z**, or **U** for the axis being set, and press **ENTER**. A pop-up window displays the following selections:
   - **0.5 Micron**
   - **1 Micron**
   - **2 Micron**
   - **5 Micron**
   - **10 Micron**
4. Highlight the appropriate resolution, and press **ENTER**.

Setting the Linear Encoder Resolution

NOTE: You can display resolution in the Setup Utility in either MM Mode or Inch Mode. One micron equals 0.001 mm.
[Default: **Inch Mode**]

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

Ensure that resolution settings match the installed equipment.

To set the linear encoder resolution:

1. See Map 1**, Resolution Setup, Menu D**.
2. Highlight **Linear Enc Res**.
3. Press **X**, **Y**, **Z**, or **U** for the axis being set. A pop-up window displays the following selections:
   - **0.5 Micron**
   - **1 Micron**
   - **2 Micron**
   - **5 Micron**
   - **10 Micron**
4. Highlight the appropriate linear encoder resolution, and press **ENTER**. The axis display will show movement at the selected resolution.

Setting Line Count for Rotary Encoder

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

Enter the number of counts per revolution specified by the rotary encoder. The software accepts line counts of up to 10,000 counts per revolution. [Defaults: **1000 lines** for X, Y, Z, and U]

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

To enter a rotary encoder line count:

1. See Map 1**, Resolution Setup, Menu D**.
2. Highlight **Rot Enc Lines**.
3. Press **X**, **Y**, **Z**, or **U** for the axis being set. The CNC highlights the encoder line entry field for the axis.
4. 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 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: **0.20000**]

To set ballscrew pitch:

1. See Map 1**, Resolution Setup, Menu D**.
2. Highlight **Bscrew Pitch**.
3. Press **X**, **Y**, **Z**, or **U** for the axis being set. The CNC highlights the pitch entry field for the axis.
4. 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 were 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 D**. Highlight **Ratio (Bsc Ply)**.
2. Press the appropriate axis key (i.e., **X**, **Y**, **Z**, or **U**). 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**]
[Defaults for AC Brushless systems only: X **2.00000**, Y **2.00000**, Z **1.00000**, U **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 D**. Highlight **Ratio (Mtr Ply)**.
2. Press the appropriate axis key (i.e., **X**, **Y**, **Z**, or **U**). 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, and U-axis **1.00000**]

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

Setting the Starting Mark

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, and U-axis **0**]

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

To determine the starting mark number, refer to "Starting Reference Mark" in *3000M CNC Motion Setup/Testing Utility*, P/N 70000635, 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**, Resolution Setup, Menu D**.
2. Highlight **Starting Mark**.
3. Press **X**, **Y**, **Z**, or **U** for the axis being set. The CNC highlights the entry field for the axis.
4. Type the starting mark for that axis, and press **ENTER**.

[Defaults: X **0**, Y **0**, Z **0**, U **0**]

Setting Linear Correction Compensation

Linear correction compensation corrects for linear errors due to linear encoders or ballscrews. To determine the amount of correction required, measure the error with a calibration device. When you activate linear correction, the CNC applies the linear correction to the active resolution of the axis.

If no linear correction is required, disable this feature. When enabled, you can specify a different correction value for each axis.

Enter any appropriate correction factor from 0.300000 to 3.000000. A value of 1.00 indicates no linear error correction for the selected axis.

[Default: **Disabled** (Off)]

To set linear correction compensation:

1. See Map 1**, Linear Correction Compensation Setup, Menu E**.
2. Highlight the menu selection that corresponds to the axis being set. (For example, highlight **X Linear correction compensation** to set the X-axis.) Press **ENTER** to highlight the entry field for each axis.
3. Type the appropriate linear compensation correction, and press **ENTER**.

[Defaults: X **1.000000”**, Y **1.000000”**, Z **1.000000”**, U **1.000000”**]

4. Highlight **Linear correction compensation**. This selection activates/deactivates the option.
5. Press **ENTER** to toggle the selection **On** or **Off** to activate/deactivate compensation values entered. The CNC activates linear compensation for all affected axes.

[Default: **Off**]

Setting In-Position Tolerance

When the CNC has positioned the tool within the in-position tolerance of the target, it calculates 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"**]

NOTE: The CNC always executes Rapid moves in In-Position Mode.

For linear encoders, as a rule of thumb, tolerance equals two times the resolution of the linear encoder.

When determining in-position tolerance for rotary encoders, tolerance is usually four times the machine resolution (for example, if machine resolution is 0.0002", in-position tolerance will be 0.0008"). Use this as a benchmark from which to adjust this value.

In-position tolerance must be smaller than continuous path tolerance.

To define in-position tolerance:

1. See Map 1**, In Position Setup, Menu F**.
2. Highlight the menu selection that corresponds to the axis being set. (For example, highlight **X In position** to set the X-axis.)
3. Press **ENTER** to highlight the entry field for the axis.
4. Type the appropriate in-position tolerance, and press **ENTER**.

Setting Continuous Path

Use the Continuous Path Mode for feed moves. With Continuous Path Mode active, the CNC blends one move into another without a complete stop between 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"** for all axes, with **Continuous path** turned On]

To activate and define the continuous path tolerance:

1. See Map 2**, Continuous Path Setup, Menu D**.
2. Highlight the menu selection that corresponds to the axis being set. (For example, highlight **X Continuous path** to set the X-axis.)
3. Press **ENTER** to highlight the entry field for the axis.
4. Type the appropriate tolerance, and press **ENTER**.
5. Highlight **Continuous path**. This selection activates/deactivates the option.
6. Press **ENTER** to toggle the selection **On** or **Off**. Select On to activate **Continuous Path Mode**.

Setting Default Rapidrate

Default Rapidrate sets the speed at which an axis operates in Rapid Mode. The machine builder sets the maximum rapidrate according to the physical constraints of the machine. Physical constraints are as follows:

- 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, adjust **FEEDRATE OVERRIDE**. The **FEEDRATE OVERRIDE** switch allows the operator to decrease only the default rapid rate of the machine. It does not allow the operator to exceed the maximum rapid rate.

[Defaults: X=**200** in/min; Y=**200** in/min; Z=**150** in/min; U=**200** in/min]

To set the default rapid speed:

1. See Map 2**, Default Rapid Setup, Menu E**.
2. Highlight the menu selection that corresponds to the axis being set. (For example, highlight **Default rapidrate X axis** to set the X-axis.)
3. Press **ENTER** to highlight the entry field for the axis.
4. Type the appropriate maximum default rapidrate, and press **ENTER**.

Setting Axis Default Feedrate

The Axis Default Feedrate establishes a default feedrate for each axis wherever a feedrate has not been programmed.

[Default: **10** inches per minute for the X, Y, Z, and U-axis]

To set the axis default feedrate for an axis:

1. See Map 2**, Default Feed Setup, Menu F**.
2. Highlight the menu selection that corresponds to the axis being set. (Example: Highlight **Default feedrate X axis** to set the X-axis.)
3. Press **ENTER** to highlight the entry field for the axis.
4. Type the appropriate axis default feedrate, and press **ENTER**.

Setting Software Limits

NOTE: The machine must have the Machine Home function enabled and turned on in order to use software limits properly.

The operator or programmer can set positive and negative software limits to restrict travel range.

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

If no vector limits are used, use another method to determine an absolute machine position (an indicator, for example).

[Default: **Off** (Disabled)]

To activate/deactivate software limits:

1. See Map 2**, Software Limits Setup, Menu G**.
2. Highlight **Software limits**. This selection activates/deactivates the option.
3. Press **ENTER** to toggle the selection **On** or **Off**. Select **On** to activate software limits.

Enter positive and negative software limits separately for each axis.

To enter positive software limits:

1. See Map 2**, Positive Software Limit Setup, Menu H**.
2. Highlight the menu selection that corresponds to the axis being set. (For example, highlight **X+ Software limit** to set the X-axis.)
3. Press **ENTER** to highlight the entry field for the axis.
4. Type the appropriate positive software limit, and press **ENTER**.

To enter negative software limits:

1. See Map 2**, Negative Software Limit Setup, Menu I**.
2. Highlight the menu selection that corresponds to the axis being set. (For example, highlight **X- Software limit** to set the X-axis.)
3. Press **ENTER** to highlight the entry field for the axis.
4. Type the appropriate negative software limit, and press **ENTER**.

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 for each axis in the Setup Utility. 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]

To enable the vector limits for an axis:

1. See Map 2**, Vector Limits Setup, Menu J**.
2. Highlight the menu selection that corresponds to the axis being set. (For example, highlight **X vector limits** to set the X-axis.)
3. Press **ENTER** to toggle the setting (**Enable**/**Disable**).

Setting Encoder Phases to Correct Axis Direction Displayed

Moving an axis in a positive direction results in a positive count on the axis display. Likewise, moving an axis in a negative direction results 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** for X, Y, and Z-axis; **Invert** for U and S-axis]

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

To adjust the Encoder Phase A Setting:

1. See Map 3**, Encoder Phase Setup, Menu D**.
2. Highlight the menu selection that corresponds to the axis being set. (For example, highlight **Phase for X axis** to set the X-axis.)
3. Press **ENTER** to toggle the setting (**Not Invert**/**Invert**). Change the Phase A setting to invert the direction of count for the adjusted axis.

Setting Backlash Compensation

Backlash is the loss motion that occurs when the encoder reverses direction and begins to record motion before the table actually moves.

Backlash compensation takes this loss motion into account and corrects the move. All systems that move mass under control exhibit backlash. Some causes include structural component flexion, bearing end thrust, and wind-up of the ballscrew that drives the slide.

Measure backlash, and store the value in Setup Utility. Once backlash compensation activates, the CNC automatically calculates the necessary motion corrections.
[Default: **0.0000"** for all axes, with **Backlash Compensation** turned Off (Disabled)]

To activate and define backlash compensation for an axis:

1. See Map 2**, Backlash Compensation Setup, Menu K**.
2. Highlight the menu selection that corresponds to the axis being set. (For example, highlight **X backlash compensation** to set the X-axis.)
3. Press **ENTER** to highlight the entry field for the axis.
4. Type the appropriate backlash compensation, and press **ENTER**.
5. Highlight **Backlash compensation**. This selection activates/deactivates the option.
6. Press **ENTER** to toggle the selection **On** or **Off**. Select **On** to activate backlash compensation.

Setting Ballscrew Compensation

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

NOTE: Perform a Machine Home sequence before enabling ballscrew compensation.

Setting Number of Segments

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**)]

1. See Map 3**, Ballscrew Compensation Setup, Menu E**.
2. Highlight **Number of segments**, and press **ENTER**. The machine builder can specify as many as 128 equally sized segments. 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:
   - See Map 3**, Number of Segments Setup, Menu F**.
   - Highlight the menu item pertaining to the axis being set. (For example, highlight **Number of segments for X** for the X-axis). Press **ENTER** to highlight the entry field for the selected axis.
3. Type the number of segments desired for that axis, and press **ENTER**.
4. Repeat the procedure for all axes being set.

Setting Table Entries

NOTE: Refer to "Using the 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 G**).

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 G**. Press **X**, **Y**, **Z**, or **U** 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, **Offset Setup Menu, Menu H**.
2. Highlight the menu selection corresponding to the axis being set. (For example, highlight **Ballscrew offset for X** to set the X-axis.)
3. Press **ENTER** to highlight the entry field for that axis.
4. Enter the appropriate ballscrew offset for that axis. If the offset location is Machine Home, enter **0.00000**. Measure the ballscrew offset from Machine Home.

Zero Cross

To set the Ballscrew Zero Cross parameter:

1. See Map 3, **Zero Cross Setup Menu, Menu I**.
2. Highlight the menu selection corresponding to the axis being set. (For example, highlight **Zero cross for X** to set the X-axis.)
3. Press **ENTER** to highlight the entry field for that axis.
4. Enter 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 Standard ballscrew compensation, the length of each lasered segment is the same. The CNC counts off the segments from the beginning of the compensated area, determined by the sum of the Offset and Zero cross values previously assigned.

The entered value should represent the segment length for each axis and the direction (positive or negative) of the compensation along the axis.

To set segment length:

1. See Map 3**, Segment Length Setup, Menu J**.
2. Highlight the menu selection corresponding to the axis being set. (For example, highlight **Length of segment for X** to set the X-axis.)
3. Press **ENTER** to highlight the entry field for that axis.
4. Enter the appropriate segment length for that axis. (This will be 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**, Ballscrew Compensation Setup, Menu E**.
2. Highlight **Active**, and press **ENTER**.
3. Press **ENTER** to toggle between **Yes** and **No**. The CNC activates ballscrew compensation as set by the user.
   [Default: **No**]

Using the Automatic File Loader

The Automatic File Loader automatically loads a properly formatted laser data file into the **Table Entries Setup Menu**. Refer to Map 3**, Menu G**.

> **NOTE:** The File Loader does not change the way you set standard, segment length ballscrew compensation. (Refer to previous section.) However, the user must type additional information. Some editing of the laser file will be necessary.

To load the laser file automatically:

1. See Map 3**, Table Entries Setup, Menu G**. Refer to **Figure 2-1**.
2. Press **LdFile** (**F8**). Enter the appropriate password, if required. The CNC displays the **Ballscrew compensation file loader** menu.

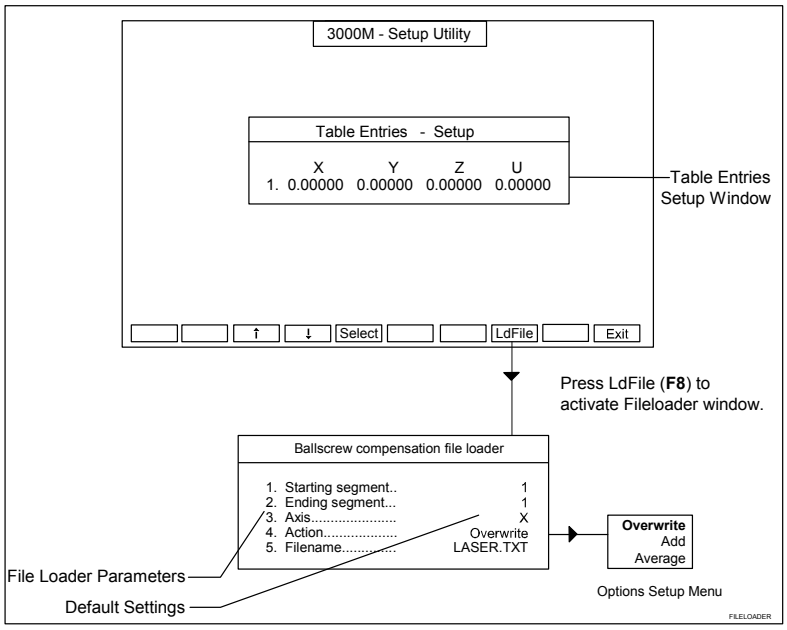


Figure 2-1, Displaying the File Loader Menu

For parameter descriptions refer to Table 2-2, Ballscrew File Loader Parameters.

3. Highlight **Starting Segment**. Type the segment number of the first table entry, and press **ENTER**.
4. Highlight **Ending Segment**. Type the segment number for the last table entry, and press **ENTER**.
5. Highlight **Axis**, and press **ENTER**. Highlight an axis (**X**, **Y**, **Z**, or **U**) in the pop-up menu that activates, and press **ENTER**.
6. Highlight **Action**, and press **ENTER**. Highlight an option in the pop-up menu that activates, and press **ENTER**.
7. Press **Ldfile** (**F8**) again to load the file. Repeat the procedure on the other axes. A successful load shows the new entries in the table after you exit the file loader.

Table 2-2, Ballscrew Compensation File Loader Parameters

Parameter	Description
Starting segment	Determines which segment will be the first for data transfer. If you type a value greater than 128 (maximum number of segments allowed) or a value greater than the Ending Segment value, an Error message is 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** clears any values in the table beyond the segment limit for the axis. **Add** and **Average** replace only the old values. **Action** allows you to fine-tune ballscrew compensation values from multiple passes of laser readings.
Filename	Type 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 and/or trailer information, which you should remove.

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 is the actual position measured by the laser.

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

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

The following information applies to the sample laser file in the preceding subsection:

- The segment length used for the laser data is 1".
- The difference between the commanded inch value and the 1" subtracted from the laser-measured travel is approximately -0.00105.
- The 1" value (commanded) from the laser data (measured) is approximately -1.00202.
- The values are negative, signifying negative machine movement.
- The CNC compares the two values by subtracting the 1" value from the 0" value: $|-0.00105| - |-1.00202| = -1.00097$
- The resulting value (-1.00097) must be negative, because machine movement is negative.
- Next, the CNC subtracts the segment length value. For example, the CNC subtracts the segment length, -1, from -1.00097. The result is the first ballscrew compensation table value: -0.00097. The CNC uses this technique to find all ballscrew compensation table values. The File Loader automatically inserts these values into the **Table Entries Setup Menu**. Refer to Map 3**, Menu G**.

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!** after the data transfer.

Set the segment limit (refer to Map 3**, Segment Length Setup, Menu J**) 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, show displacements of one inch per segment. To avoid data error, enter this value (1”) as the segment length before loading the laser readings.

The positive/negative sign of the segment size during ballscrew compensation file loading must match the direction of machine travel used for the laser readings. This applies also 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 a “positive” value for a “negative” value in all cases.

Setting Axis Ports

Use the **Axis Ports Setup Menu** to assign the active axes on the machine. Normally, axes X, Y, and Z are assigned to ports 0, 1, and 2, respectively. There are four axis ports available (0-3).

To set the axis ports:

1. See Map 4**, Axis Ports Setup Menu, Menu D**.
2. Highlight a port (0–4), and press **ENTER**.
3. A pop-up menu activates showing the available axes: X, Y, Z, U, or S. Typically, these axes can be assigned to the selected port. (Refer to Map 4**, Menu E**.) Highlight the appropriate axis, and press **ENTER**. The CNC assigns the selected axis to that port.
4. Repeat the procedure to set all active axes. For unused ports, select **Disabled** from the pop-up menu.

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.

The following 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 higher 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, the output from the system is a digital word representation of that move. The CNC derives this digital word from the output of the interpolators, which creates a move-required value that it feeds to the Digital Proportional, Integral, and Derivative Gain (PID) Filter, so that the following equation can define the output [digital word]:

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

Refer to Table 2-3, System Output Values and Definitions for a detailed explanation of parameters.

Table 2-3, System Output Values and Definitions

Value	Definition
Voltage Offset	A fixed voltage value always present at the output.
Kp + Ki + Kd	The Digital PID Filter Parameters.
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. It is derived by multiplying the Ki coefficient by the position error and then adding it to the previously computed Integral Gain value.
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 *3000M CNC Motion Setup/Testing Utility*, P/N 70000635, manual for documentation on using features that allow the CNC to automatically determine filter parameters. The Rigid Tapping Filters are not set automatically.

To change the **Feed Filter** parameters:

1. See Map 5**, Feed Filter Parameters, Menu E**.
2. Highlight the PID parameter being set (**Kp**, **Ki**, **Kd**, **Kf**, **Il**, or **Ds**).
3. Press **X**, **Y**, **Z**, **U**, or **S** 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**, **15.000** for **X axis** and **Y axis**, and **11.300** for **Z axis**. **Ki**, **0.000**, **Kd**, **10.000**, **Kf**, **0.000**, **Il, 0.000**, and **Ds**, **5]**

To change the **Rapid Filter** parameters:

1. See Map 5**, Rapid Filter Parameters, Menu F**.
2. Highlight the PID parameter being set (**Kp**, **Ki**, **Kd**, **Kf**, **Il**, or **Ds**).
3. Press **X**, **Y**, **Z**, **U**, or **S** 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: For all axes: **Kp**, **10.000**, **Ki**, **0.000**, **Kd**, **10.000**, **Kf**, **0.000**, **Il**, **0.000**, and **Ds**, **2**]

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

1. See Map 5**, No Motion Filter Parameters, Menu G**.
2. Highlight the PID parameter being set (**Kp**, **Ki**, **Kd**, **Kf**, **Il**, or **Ds**).
3. Press **X**, **Y**, **Z**, **U**, or **S** 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**, **10.000**, **Ki**, **5.000**, **Kd**, **10.000**, **Kf**, **0.000**, **Il**, **10.000**, and **Ds**, **5**]

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

1. See Map 5**, Rigid Tapping Filters Setup, Menu I**.
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**.]

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. The variables of these calculations are configurable. Refer to **Table 2-4** for definitions of these parameters.

If the PEC algorithm detects a fault, the servos shut off, and one of the following messages appears:

"ERROR: (AXIS) LAG OVER MAX!"

"ERROR: LOST (AXIS) FEEDBACK!"

To change a PEC parameter:

1. See Map 5**, Position Error Check, Menu H**.
2. Highlight the PEC parameter to be changed, and press **ENTER**. The corresponding entry field for that parameter highlights.
3. Press **ENTER** and type the password when prompted.
4. Enter the appropriate value for each of the parameters.

Table 2-4, Position Error Check Parameters

Position Error Check Parameter	Definition	Default
Max idle time (msec)	The amount of time, in milliseconds, allowed between the internal command for a move and the input of counts from the feedback device, signifying motion.	100.0 msec
Max lag error	The error distance allowed at rest or low feed rates, before declaring a fault.	0.0100
Check Rapidrate	Enables rapidrate test. This test checks the feedback during rapid moves to see if the axes are reaching the programmed rapid rate. Ensures that machine is reaching its full programmed rapid rate; if it does not, an Error Message displays.	Yes
Check Feedrate	Enables feedrate test. This test checks the feedback during feed moves to see if the axes are reaching the programmed feed rate. Ensures that machine is reaching its full programmed feed rate; if it does not, an Error Message displays.	Yes
Enable error checking	This setting is used to enable/disable Position Error Checking (PEC) for troubleshooting or comparison. **CAUTION: You must enable the PEC parameter for the CNC to declare a servo fault and shut down the system in an emergency.**	Yes

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 5**, Amplifier Tuning Rapids, 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 *3000M CNC Motion Setup/Testing Utility*, P/N 70000635.

To set the digital amplifier:

1. See Map 7**, Menu D**. Refer to **Table 2-5** for a description of the Digital Amplifier Parameters. Highlight **Active digital amplifiers**, and press **ENTER** to display Map 7**, Menu E**.

2. Highlight the axis you want to change, and press **ENTER** to display Map 7**, Menu F**.

3. Press **ENTER** to toggle between **Disable** and **Enable**.

Table 2-5, 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 7**, Menu G**.
[Default: X, Y, Z, U, W - Invert DAC Output **No**]

U Axis Setup

3000M-4X

In addition to the primary axis (i.e., XYZ), the 3000M 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 2-6** and Map 6**, Menu C**. Use this menu to configure auxiliary axis specific parameters.

Table 2-6, 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: **No**]
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: **No**]
Synchronize to XYZ	Allows the axis to be synchronized to XYZ. [Default: **No**].

Setting the Spindle Axis

Standard CNC systems use the Z-axis for the spindle.

To set the spindle axis:

1. See Map 8**, Spindle Axis Setup, Menu C**. Use this menu to set all setup parameters associated with the spindle axis.
2. Highlight the axis used by the spindle, and press **ENTER**. A pop-up menu activates showing the axes to which you can assign the spindle.
3. Highlight **Z** axis for standard CNC systems, and press **ENTER**. The CNC assigns the spindle to the Z-axis.

Setting Spindle 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 selected spindle speed. The range is 0 VDC to +10 VDC.

Bipolar Output ranges from –10 VDC to +10 VDC. A voltage of 0 VDC represents a commanded 0 RPM spindle speed.

The system outputs a negative DC voltage for Spindle Reverse commands and a positive DC voltage for Spindle Forward 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 system outputs the maximum voltage, +10 VDC, at the highest RPM value of the gear range.

To set the spindle output:

1. See Map 8**, Spindle Axis Setup, Menu C**.
2. Highlight **Spindle output** and press **ENTER** to activate a pop-up menu.
3. Highlight the appropriate spindle output (**Bipolar** or **Unipolar**), and press **ENTER**.

   [Default: **Unipolar**]

Setting Spindle Gear Ranges

Depending on the mechanical considerations of the system, the spindle drive may require gearing and belt drive arrangements to provide required spindle speeds and torque.

NOTE: The Analog Spindle Output option is required for controlled spindle operation.

You can use the Setup Utility to set one gear range for the spindle. In the following setup, you specify high and low settings for each gear range. Refer to Map 8**, 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 select **Single-M40** [Default], the CNC does not require a gear range entry during spindle operation. Use the Tool Page to program spindle direction and, for DC operation, spindle speed.

At the highest RPM in the range, the system outputs the maximum DC voltage, ±10 VDC. Zero RPM always represents 0 VDC. The lowest RPM voltage will be a ratio of the highest speed to the lowest speed.

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

Program the RPM and spindle direction (forward or reverse) in the Tool Page. With the tool activated, the corresponding spindle direction and speed take effect.

[Default: **Low/High** setting for the **M40** gear range is **50** and **6,000** RPM]

To set the spindle gear range:

1. See Map 8**, Spindle Axis Setup, Menu C**.

NOTE: If you select **M40-M41**, you must set up the input function **Double Gear Select**.

2. Highlight **Gear ranges used**, and press **ENTER** to toggle the setting between **Single-M40** and **M40-M41**. [Default: **Single-M40**]
3. Highlight **Low setting for M40 gear range**, and press **ENTER**. Type the appropriate Low range RPM, and press **ENTER** to store the setting. [Default: **50**]
4. Highlight **High setting for M40 gear range**, and press **ENTER**. Type the appropriate High range RPM, and press **ENTER** to store the setting. [Default: **6,000**]

NOTE: **Invert DAC in M40 gear range** inverts the polarity of the spindle DAC output while the **M40** gear is selected. The Spindle output must be set to **Bipolar**.

5. Highlight **Invert DAC in M40 gear range**, and press **ENTER**. Highlight **No**, and press **ENTER**. [Default: **No**]
6. Highlight **Low setting for M41 gear range**, and press **ENTER**. Type the appropriate Low range RPM, and press **ENTER** to store the setting. [Default: **50**]
7. Highlight **High setting for M41 gear range**, and press **ENTER**. Type the appropriate High range RPM, and press **ENTER** to store the setting. [Default: **6,000**]

> **NOTE:** **Invert DAC in M41 gear range** inverts the polarity of the spindle DAC output while the **M41** gear is selected. The Spindle output must be set to **Bipolar**.

8. Highlight **Invert DAC in M41 gear range**, and press **ENTER**. Highlight **No**, and press **ENTER**.
   [Default: **No**]

M40–M41 Ratio (Spindle Pulley)

The number of teeth for the M40–M41 spindle pulley.
[Default: **1.0**]

M40–M41 Ratio (Motor Pulley)

The number of teeth for the M40–M41 motor pulley.
[Default: **1.0**]

Setting the Number of Lines on the Spindle Encoder

For machines fitted with spindle encoders, you must set the spindle speed display to correspond to the spindle encoder specifications.

[Default: **1,024** lines per revolution]

To set the number of spindle encoder lines:

1. See Map 8**, Spindle Axis Setup, Menu C**.
2. Highlight **Spindle encoder lines**, and press **ENTER**.
3. The entry field for the parameter highlights. Type the number of encoder lines, and press **ENTER**.

Setting Spindle RPM Display

Configure the spindle RPM display to accept feedback from either a spindle encoder or a display-programmed RPM.
[Default: **Program**]

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

To set the RPM display:

1. See Map 8**, Spindle Axis Setup, Menu C**.
2. Highlight **Spindle RPM Display**, and press **ENTER** to toggle between **Feedback** and **Program**.
3. Select **Feedback** to configure the spindle RPM display for encoder feedback.

   – or –

   Select **Program** to configure the display to exhibit the programmed RPM.

Setting Stop/Start Spindle When Hold/Start is Pressed

The CNC can start and stop the spindle as a function of the program Start and Hold keys. To activate this function, highlight **Stop/Start spindle when Hold/Start is pressed**, and press **ENTER**. If the spindle is running and the **Hold** key is pressed, the spindle will be commanded to stop as well. When the **Start** key is pressed next, spindle operation will resume at the previous settings. If used, External Hold and External Stop CAN I/O inputs will function in the same manner. This feature is active only in AUTO mode. If MANUAL mode is selected and the spindle is then stopped, it will not restart automatically.
[Default: **No**]

Setting Stop Program and Spindle on Gear Range Error

On a Gear Error Condition (caused by programming error or use of override) the CNC will force a HOLD condition and issue an M5 to stop the spindle. Once you correct the Error Condition, you will need to restart the program and spindle manually. (In Auto, press **M3** or **M4** and **START**; or, go to Manual, correct Program, and rerun.)
[Default: **No**]

Setting Encoder Mounted on Motor

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

Setting 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 – 100

Setting 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 verses command RPM)
[Default: **90**] Valid range 50 – 100

Setting Z Axis Type

Use the following guidelines when setting the Z-axis type:

- If Z is a non-control, non-DRO axis, select **None**.
- If Z is a controlled axis with position display and feedback loop, select **Linear**.
- If Z is a DRO, with position display but no motion control, select **DRO**.

To set the Z-axis type:

1. See Map 4**, Z Axis Type Setup, Menu F**.
2. The CNC automatically highlights **Z Axis Type**.
3. Press **ENTER** to activate the pop-up selection menu, which lists the options **None**, **Linear**, and **DRO**.
4. Highlight the appropriate selection, and press **ENTER** to activate the selected option.
   [Default: **Linear** for 3000M, **DRO** for 3000M Two-Axis Systems]

Setting Basic I/O Interface

A Servo Control Board exchanges control signals between machine devices and the control. Input signal implementation is a standard feature. Installed security devices enable M-function output signals.

The specific properties of the signals exchanged are configured via the software settings in the Setup Utility. Refer to "Section 6 - Setup Utility Maps," to locate the setup screens required for I/O setup.

I/O ports are single-bit ports located at the board's P3 connector. There are ten inputs and six outputs. Refer to *Drawing Package*, P/Ns 70000595 (2-axis systems) and 70000596 (3-axis systems) for detailed explanation of the I/O setup in 3000M two-axis Systems and 3000M three-axis systems respectively.

Activating the I/O Interface

To activate the I/O Interface:

1. See Map 8, **I/O Interface Setup, Menu D**.
2. Highlight **Type** to activate the CAN I/O, allowing the exchange of input and output signals between the control and the Servo Control Board.
3. Press **ENTER** to activate a pop-up menu with the following selections:

   Disabled — Disables CAN I/O. The CNC and the machine do not exchange any I/O signals.

   CAN I/O — Activates system CAN I/O. Inputs from the machine generate function signals, which the machine sends to the CNC via CAN-Bus channel. The CNC sends Function signals to the CAN I/O (via CAN-Bus protocol) to activate the required output ports.

   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 details, refer to *Integral Programmable Intelligence User's Guide*, P/N 70000416.

4. Highlight the appropriate selection, and press **ENTER**.
   [Default: **CAN I/O**]

Output Function Setup

The CAN I/O 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 when triggered by specific “conversational language” events. M-function numbers (1–98) correspond to program M-code numbers (1–98).

Run an M-code block to activate the outputs assigned to like-numbered M-functions. **Table 2-7** lists M-functions generated by conversational language events.

Table 2-7, Functions Generated by Conversational Language Events

M-Function	Initiating Event	Standard Function Use (ref. EIA Standard RS 274-D)
M02	Run an EndMain program block.	Stops spindle and coolant.
M03	Activate a tool that lists a FWD spindle direction on the Tool Page.	Start CW spindle rotation.
M04	Activate a tool that lists a REV spindle direction on the Tool Page.	Start CCW spindle rotation.
M05	Activate a tool that contains a Tool Page spindle direction “OFF” listing.	Stop spindle in normal manner.
M08	Activate a tool with coolant ON listed on the Tool Page.	Turn coolant pump on.
M09	Activate a tool with coolant OFF listed on the Tool Page.	Turn coolant pump off.
M03, M05, M04	Run the Tapping Cycle.	Start CW spindle rotation, stop spindle then start CCW spindle rotation.
Au/Ma	Activates output when system is in Auto or Single Stop Mode or during Homing sequence.	Not a standard function.

To set an output port:

1. Configure the port to produce the required type and logic of the signal to control the device on the machine.
2. Assign the port to the functions that will activate and deactivate it. More than one function can use a port.

Configuring Output Ports

Default port settings provide a +24 VDC 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(s) and another deactivates it (latched output). Sometimes, the CNC emits an output signal for a user-specified duration (pulsed output).

Refer to **Figure 2-2** and **Figure 2-3**. The default port settings are designed to open and close relays that operate devices on the machine.

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.

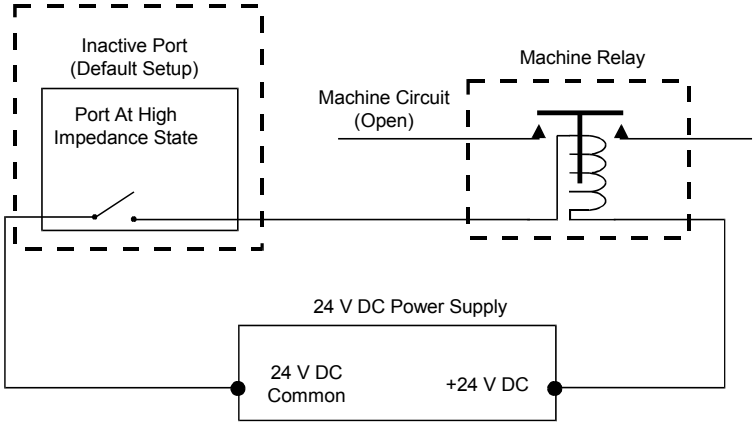


Figure 2-2, Typical Inactive Port (Default Port Settings)

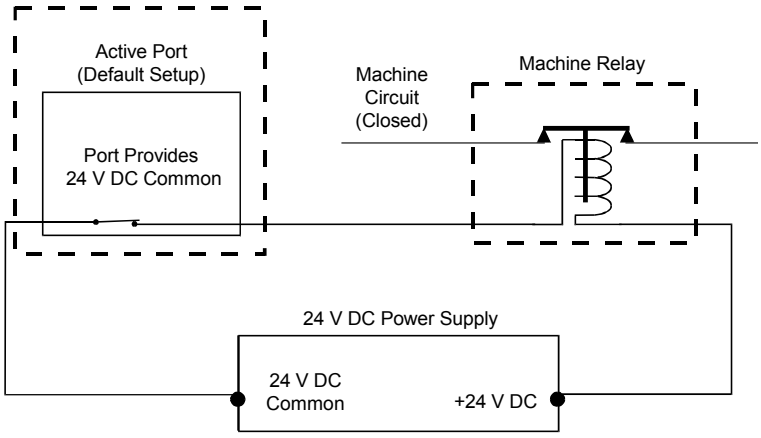


Figure 2-3, Typical Active Port (Default Port Settings)

Setting Up DSP^2 Nodes

To set the DSP^2 Nodes:

1. See Map 8**, I/O Nodes Setup, Menu E**.
2. Highlight **DSP2 Node**, and press **ENTER** to activate **Menu F**.
3. To configure each bit, press **Bit** (**F1**). **Menu H** activates.
4. Highlight **Input 0**, **Input 1**, or **Input 2**, and press **ENTER** to return to Menu **F**, ready for you to assign a function to each bit. [Default: **Bit 0**]
5. To assign bit functions in **Menu F**, press **ENTER**. **Menu G** activates. **Highlight Off**, **Active Low**, or **Active High** for each function.

Assigning Output Ports to M-Functions

NOTE: CAN Node 0 and CAN Node 1 operate identically. To assign M-functions to a CAN Node's output ports, see Map 8. The **#** symbol indicates either CAN Node 0 or CAN Node 1, as determined in **Menu D**.

The CNC can generate up to 98 M-functions (M1 to M98) and an Auto/Manual output. An M-function can activate or deactivate the ports assigned to it. A port can be assigned to more than one M-function. To assign an output port to an M-Function:

1. See Map 9**, I/O Nodes Setup, Menu D**.
2. Highlight **CAN Node 0** or **CAN Node 1** to activate **Menu E**.
3. In **Menu E**, highlight **Outputs**, and press **ENTER**. Use this menu to configure M-functions as follows:
   - From **Menu I**, highlight the M-function to be configured, and press **ENTER**.
   - **Menu J** activates. Highlight **Unused**, **Bit On**, or **Bit Off**; or press **ATTR** (**F7**).
   - **Menu K** activates. Highlight a bit from **0** to **5**, and press **ENTER**.
   - **Menu M** activates. Use **Menu M** to configure the following parameters:

     Pulse(msec) Port output is maintained for the period specified in milliseconds by the builder. Any value from 0 to 32,000 is allowed. If a zero (msec) pulse is used, output is maintained until port is turned off. The CNC will delay run of the program for the duration of a nonzero (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, there is no delay.

Finish	When a function activates an output port that has been set for a finish pulse, the CNC delays the run of the program until an "External finish pulse" input function is received. If no finish pulse is received by the end of the timeout period, the CNC displays an Error message and terminates the program. If Finish is specified for a port that has a nonzero (msec) pulse, the "External finish pulse" is only acknowledged after the pulse expires.
Active	Sets port logic as Active High or Active Low. When set Active High, the port provides a +24 VDC common source when activated. When set Active low, the port is in a high-impedance state when activated.

- To configure a particular bit press **F1** (**Bit**). **Menu L** activates.
- Highlight an output from **0** to **5**, and press **ENTER** to return to **Menu I**, ready for you to assign M-functions to each bit.

NOTE: The heading for **Menu I** changes to reflect the active bit selected in **Menu L**. Refer to Map 9. For example, if you select Bit 4 in **Menu K** and press **ENTER**, the heading on **Menu I** will appear as **M-Functions (Output 4, Node #) – Setup**.

Assigning Input Ports to Input Functions

NOTE: CAN Node 0 and CAN Node 1 operate identically. To assign a CAN Node's input ports to Input functions, see Map 8. The **#** symbol indicates either CAN Node 0 or CAN Node 1, as determined in **Menu D**.

To assign input ports to Input functions:

1. See Map 9**, CAN Node # Setup, Menu E**.
2. Highlight **Inputs**, and press **ENTER** to activate **Menu F**.
3. Use this menu to configure CAN Input 0 functions:
   - Highlight a function from **1** to **11**, and press **ENTER** to activate **Menu H**.
   - Use this menu to select the active state for each function; or highlight a bit from **1** to **10**, and press **ENTER**.
   - Press **F1** (**Bit**) to activate **Menu G**.
   - From **Menu G**, highlight one input from **1** to **10**, and press **ENTER**.
   - The CNC displays **Menu F**, ready for you to assign an input function to each bit.

NOTE: The **Menu F** heading changes to reflect the active bit selected in **Menu G**. Refer to Map 9. For example, if you select Bit 4 in **Menu G** and press **ENTER**, the **Menu F** heading will appear as **CAN Input 4 (Node #) – Setup**.

Selecting Automatic Gear for DAC Output

Gear selection can be done with a single switch, but use of two is preferred. Using two switches guarantees the correct gear is properly selected and will also allow the software to sense error conditions (for example, spindle in either gear). Use of a single switch is retained for compatibility with older systems. If a single switch is programmed, the dual inputs are required.

When the single-switch method is used, the input **Double Gear Select** will enable the high gear DAC scaling when true. The CNC will generate an error message if the programmed spindle speed is out of the range selected by the gear switch.

When the two-switch method is used, the first input, **Double Gear Select**, specifies low gear when true. The second selection, **Second Spindle Gear Select**, specifies high gear. If a programmed spindle speed is not in the selected range, the CNC will generate an error message. If both inputs are active at the same time, or if neither input is active, the CNC will also generate an error message.

CNC Input Functions

Each input function causes a specific action by the CNC. Refer to **Table 2-8** for a description of each Input function. See Map 9**, CNC Input # Functions Setup, Menu F**.

Table 2-8, Input Function Usage

Input Function	Use
Tool Guard	Holds the CNC program and stops the machine spindle. You must remove the input 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 you activated the function, press **START** one time to continue the program. The tool guard function permits compliance with regulations that require an intact tool guard in place in order for the machine to run.
Double spindle gear select	Informs the CNC that the spindle speed range has changed to low gear. Use this input on machines that use high and low speed spindle ranges through gearing.
External finish pulse	The finish pulse is 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 it receives a finish-pulse input. 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).
Variable input	Allows the state of an input (On or Off) to be assigned to system variable #1000. You can then read variable #1000 and determine the state of the specific input.
External start	Performs the same function as **START** on the CNC keypad.
External hold	Performs the same function as **HOLD** on the CNC keypad.
External feed hold	Holds the program if the CNC attempts a feed. A Spindle Off condition usually activates this function.
External manual select	Enables manual hardware operation. The encoder inputs generate the Position display. 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 readout. Use this function with the Auto/Manual switch included in the kit.
Start reading keyboard	Allows the CNC to accept inputs from the CNC keypad (or keyboard).
Stop reading keyboard	Informs the CNC not to respond to inputs from the CNC keypad (or keyboard). This feature permits the builder to set a keyboard lockout system. When the keypad (or keyboard) is locked out, all keys except **E-STOP** remain inoperative.

(Continued…)

Table 2-8, Input Function Usage (Continued)

Input Function	Use
General Error input message	Holds the program and generates an Error message. You must correct the cause of the input signal before you can restart the program.
General Warning input message	Generates a Warning message at the CNC, while allowing the control to continue to run a program.
Remote Resolution Selector	Allows you to select the axis resolution factor from a remote manual panel. Each bit corresponding to x1, x10, and x100 resolution uses three input bits. The three input bits must be sequential.
Remote Axis Selector	Enables selection of an axis from a remote pendant. The number of bits required must correspond to the number of axes available. For example, two bits are used with each bit corresponding to X- and Y-axis on a 3000M Two-Axis System; three input bits are used with each bit corresponding to X, Y, and Z-axis on a 3000M three-axis system. The input bits must be sequential.
Second spindle gear select	Informs the CNC that the spindle speed range has changed to high gear. Use this input on machines that use high and low speed spindle ranges through gearing.
Spindle override	Allows you to vary the DAC output to the inverter drive in 10 percent increments. For MK systems, the input bits required are bits 3, 4, 5, and 9; for OEM systems, use any four bits.
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.
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**.
Z Axis Disengage	Allows the capability to switch between two-axes and three-axes operation. In Manual mode, the Z-axis can be disengaged by just loosening the quick-release knob on the front of the Z-axis drive system. Refer to *3000M CNC Programming and Operations Manual for Three- and Four-Axis Systems*, P/N 70000504, "Section 3, Disengaging the Z-Axis Drive System" for more information.

Setting Finish Pulse Timeout

When an output port configured for a finish pulse is activated, the CNC is put on hold until it receives an external finish pulse. 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.

NOTE: To regain control of a CNC holding for a finish pulse, press **E-STOP** and **servo RESET**.

If the CNC does not receive a finish pulse by the end of the timeout period, it displays an **Error** message.

To set the Finish Pulse Timeout:

1. See Map 8**, I/O Interface Setup, Menu D**.
2. Highlight **Timeout**, and press **ENTER** to highlight the **Timeout** entry field.
3. Type the number of milliseconds, and press **ENTER**.
   [Default: **10,000** (10 sec). Valid range: (0– 600000). If you enter **0** (zero), you will cause an indefinite hold.]

Setting Vector Limit Inputs

> **NOTE:** If vector limits are enabled, a signal must be wired to both the positive and negative direction ports assigned to the axis (X+/X-, Y+/Y-, Z+/Z-). Otherwise, the CNC will inhibit motion in the direction (positive/negative) of the unwired port.

Assign each vector limit to a specific input port. When the vector limits for an axis are set, the assigned input ports cannot be used for input functions. Refer to **Table 2-9**.

Vector Limit Switches restrictions follow:

- Must be normally closed switches.
- Must be on CAN Node 0
- Are hard-coded (See **Table 2-9**.)
- Cannot be used as general purpose I/O
- Both directions must be wired for assigned axes
- Can also be used as Home Switches

Table 2-9, Assigned Vector Limit Input Ports

Input Port	Pin	Assigned Vector Limit
0	1	X +
1	2	X -
2	3	Y +
3	4	Y -
4	5	Z +
5	6	Z -

See Map 2, **Vector Limits Setup**, **Menu J**; and Map 9, **CAN Input # Functions Setup**, **Menu F**. Set the vector limits as follows:

1. From Map 2**, Menu J**, highlight the required axis (axes), and switch the vector limits to the Enable setting.
2. From Map 9**, Menu F**, highlight the required ports and switch to the logic setting (High/Low) required for the vector input.

Vector/Home Limit Switch Connections

The CAN I/O systems hard code the vector/home limit switch connection to node 0. This can be node 0 on the SCB or an individual CAN I/O addressed as node 0. Connection is only to node 0. For additional information, refer to the *3000M and 5000M Kit CNC Installation*, P/N 70000593.

Connection of vector/home limit switches is hard coded to individual input bit. Refer to **Table 2-10**.

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 2-10**.)
- 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

Table 2-10, CAN I/O Systems Vector/Home Limit Switches

Bit	Pin	Input
0	1	X+
1	2	X-
2	3	Y+
3	4	Y-
4	5	Z+

Bit	Pin	Input
5	6	Z-
6	7	N/A
7	8	N/A
8	9	N/A
9	10	N/A

Connect vector/home switches to SCB using P3 Phoenix Block, pins 1 through10. Connect vector/home switches to CAN I/O boards using P5 DB25, pins 1 through 10. SCB inputs must be +24 VDC (source) input. CAN I/O board inputs can be either 24 COM (sink) or +24 VDC (source), depending on the board type.

If vector limits are used, enable them in the **Setup Utility**, **Vector Limits Setup**. If home switches are used, select the proper type of homing in the **Setup Utility**, **Homing Setup**. You can enable both or either at the same time.

CAUTION: Take care not to assign Input Functions or IPI inputs to the same input bits as vector or home switches.

I/O Settings for Factory-Assembled M-Function Hardware

ANILAM pre-wires CNCs for the settings listed in **Table 2-11**.

Table 2-11, ANILAM M-Function Settings

M-Functions (Output 3 Node #)			
FTN	ON	OFF	CURR BIT
1. M01			
2. M02		345	Bit Off
3. M03	3	4	Bit On
4. M04	4	3	Bit Off
5. M04		345	Bit Off
6. M05			
7. M06			
8. M07	5		
9. M08		5	
10. M09			
11. M10			

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 Read Out) 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.

There are two parameters that apply to the counter port regardless of its use as a handwheel or DRO. Refer to **Table 2-12** for a definition of these parameters. See Map 4**, Menu H** and **Menu K**.

Table 2-12, 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. See Map 4**, Menu N**.
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 2-13** for information on the parameters used to only when the counter port is configured for handwheel use. See Map 4**, Menus H** and **N**.

Table 2-13, Handwheel Parameters

Setting	Description
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 G**.
3. Press **ENTER** to display the **Handwheel/DRO #1 – Setup** menu. See Map 4**, Menu H**.
4. Press **ENTER** to select the **Type** for **Handwheel/DRO #1**. See Map 4**, Menu N**. [Default: **None**]
5. Highlight **Handwheel** and press **ENTER** to setup the Handwheel Scaling Factor parameter. See Map 4**, Menu I** and **L**.
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 **Scaling Factor**, and press **ENTER**.

   The entry field for the parameter highlights.
10. Type the desired value, and press **ENTER**. [Default: **1.00**]

Configure other handwheels in a same way as described above.

DRO Parameters

Refer to **Table 2-14** for information on the settings used only when the counter port is configured for DRO use. See Map 4**, Menu J** and **M**.

Table 2-14, 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 4**, Menu J**. 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 Res	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 G**.
3. Press **ENTER** to display the **Handwheel/DRO #2 – Setup** menu. See Map 4**, Menu K**.
4. Press **ENTER** to select the **Type** for **Handwheel/DRO #2**. See Map 4**, Menu N**. [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 4**, Menu M**.

   Refer to Table 2-14, DRO Parameters, to setup DRO specific parameters.

> **NOTE:** Do not use U 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 4**, Menu G**. Highlight **Coupled DRO Tolerance**

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

Tool Management Setup

The CNC system supports manual tool change operations only.

Default Tool Table File

To enter the default tool table file:

1. See Map 10**, Menu C**. Highlight **Default tool-table file**, and press **ENTER**. In the highlighted entry field, type in the name of the file. [Default: **P3MTOOL.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**]
- Stop program execution [Default: **On Tn]**
- Use tool change macro [Default: **No**]

The available activation options are listed in **Table 2-15**.

Table 2-15, 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 2-16** and refer to Map 10**, Menu C**.

Table 2-16, 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.
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.

Activating Tool Length Offset

Use the **Activate tool length offset** parameter to activate the Tool Length Offset whenever a tool number is activated:
[Default: **On Tn**]

1. See Map 10**, Menu C**.
2. Highlight **Activate tool length offset**, and press **ENTER** to activate **Menu D**.
3. In **Menu D**, highlight **On Tn**, and press **ENTER**.
4. Press **F10** to exit the menu. The CNC will activate tool-length offsets for the currently active tool number.

Setting Output Signal to Default

The output signal does not affect machine operation.
[Default: **On Tn**]

To set the parameter to the default, **On Tn**:

1. See Map 10**, Menu C**.
2. Highlight **Output signal**, and press **ENTER** to activate **Menu D**.
3. In **Menu D**, highlight **On Tn**, and press **ENTER**.

Setting Default Spindle Orientation Angle

Selects an angle of orientation beyond the marker pulse. The range is 0.1 to 360 degrees. This feature eliminates the need for exact mechanical positioning of the spindle encoder. The spindle orientation angle is programmable via CNC software.
[Default: **0.0**] Valid range: (0.1–360.0)

To set the parameter to the default:

1. See Map 10**, Menu C**.
2. Highlight **Default spindle orientation angle**, press **ENTER**.
3. Type the angle value in degrees, and press **ENTER**.

Setting Spindle Orientation RPM

Specifies the orientation RPM of the Spindle. Consult relevant spindle drive documentation for proper Spindle speed encoding and appropriate orientation speeds. Maximum programmable orientation spindle speed is 250 RPM.
[Default: **10**] Valid range: (1–250)

To set the spindle orientation RPM:

1. See Map 10**, Menu C**.
2. Highlight **Spindle orientation RPM**, press **ENTER**.
3. Type the spindle orientation RPM, and press **ENTER**.

Setting Spindle Orientation in Position (Counts)

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.
[Default: **0** (counts) No in position check] Valid range: (0–50)

To set the spindle orientation in position (counts):

1. See Map 10**, Menu C**.
2. Highlight **Spindle orientation in position (cts)**, press **ENTER**.
3. Type the spindle orientation count, and press **ENTER**.

Stopping Program Run on Tool Commands

Use this feature to halt program operation on Tn commands in order to give the operator time to change the tool. Press **START** to resume program operation.
[Default: **On Tn**]

To configure the CNC to stop program execution on Tn commands:

1. See Map 10**, Menu C**.
2. Highlight **Stop program execution**, and press **ENTER** to activate Menu **D**.
3. In **Menu D**, highlight **On Tn**, and press **ENTER**. When a tool number is activated, the CNC holds execution of the program and displays a message.

   – or –

   In **Menu D**, highlight **No**, and press **ENTER**. The CNC continues program execution on **Tn** commands (for example, during Dry Run Mode when it is not performing a tool change).

Setting the Use Tool Change Macro

To use the tool change macro:
[Default: **On Tn**]

1. See Map 10**, Menu C**.
2. Highlight **Use tool change macro**, and press **ENTER** to activate **Menu D, Options Setup Menu**.
3. In **Menu D**, highlight **On Tn, On M6**, or **Both**, and press **ENTER** to enable use of the tool change macro.
4. Highlight **No**, and press **ENTER** to disable the tool change macro.

Setting the Tool Change Macro Program

To set the tool change macro program:
[Default: blank]

1. See Map 10**, Menu C**.
2. Highlight **Tool change macro program**, and press **ENTER**.
3. Type the name of the program containing the macro, and press **ENTER**.

NOTE: You must write the macro in ANILAM's G-code (ISO) format using an offline editor. After you have developed the macro, copy the program to the CNC's system directory, usually **C:\P3M**, and then reboot the system. You must reboot the system every time you edit the macro.

Setting the Default Tool-Table File

To enter the default tool-table file:
[Default: **P3MTOOL.DAT**]

1. See Map 10**, Menu C**.
2. Highlight **Default tool-table file**, and press **ENTER**.
3. In the highlighted entry field, type the name of the file.

Setting the Force Spindle Off During Tool Change

The CNC will generate an error message if tool change is attempted with spindle running.
[Default: **Yes**]

1. See Map 10**, Menu C**.
2. Highlight **Force spindle off during tool change**, and press **ENTER**. Press **ENTER** to toggle the selection **Yes** or **No** to activate/deactivate the force spindle off and error message.

Setting the Restore TLO After Power-Up or Home

Restore tool length offset (TLO) after power-up or home sequence.
[Default: **No**]

1. See Map 10**, Menu C**.
2. Highlight **Restore TLO after Power-Up or Home**, and press **ENTER**. Press **ENTER** to toggle the selection **Yes** or **No** to activate/deactivate the restore TLO after power-up or Home.

Setting the Send Tflag When T0 Programmed on Fixed TC

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.
[Default: **No**]

1. See Map 10**, Menu C**.
2. Highlight **Send Tflag when T0 programmed on Fixed TC**, and press **ENTER**. Press **ENTER** to toggle the selection **Yes** or **No** to activate/deactivate the send Tflag when T0 programmed on Fixed TC.

Miscellaneous Setup Parameters

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

Setting Maximum Programmed Linear Axis Feedrate

The maximum programmable linear axis feedrate limits the maximum speed at which the CNC will allow the machine to travel in feed mode. [Default: **80.0"/minute**]

To set the maximum programmable feedrate:

1. See Map 5**, Miscellaneous Setup, Menu C**.
2. Highlight **Max. programmed linear axis feedrate**, and press **ENTER**.
3. In the highlighted entry field, type the maximum feedrate, and press **ENTER**.

NOTE: To override the programmed feedrate, use the **FEEDRATE OVERRIDE** switch. The range of the switch is 0 to 120 percent of the programmed feedrate. The switch enables the operator to vary the feedrate in increments of 10 percent.

Setting Dry Run Feedrate

In Dry Run Mode, the machine axes (X, Y, and sometimes Z) move through the program without cutting into the work. The CNC disables coolant operation, and the work may or may not be placed on the table.

Activate Dry Run Mode with M-codes 105 and 106, and deactivate with M107. Refer to **Table 2-17**. Dry run feedrates are set in the Setup Utility. They are often set at greater speeds than conventional feedrates. At the operator's discretion, they can be set at any rate.

[Default: **40.0 inches per minute**]

Table 2-17, 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, without moving Z-axis. .
M106	Z Dry run on	Enable machine Z Dry Run Mode. Program runs at dry run feedrates specified in the Setup.
M107	Cancel dry run	Cancels active Dry Run Mode.

To set the Dry Run Mode feedrate:

1. See Map 5**, Miscellaneous Setup, Menu C**.
2. Highlight **Linear axis dry-run Feedrate**, and press **ENTER** to activate the entry field.
3. Type the feedrate, and press **ENTER** to activate it.

Setting Default Linear Axis Jog Feedrate and Jog Rapidrate

Set the default feedrate at which the machine travels in Jog Mode. This defines the default speed the machine will travel when the operator presses the Jog keys (X+, X-, Y+, Y-, Z+, Z-).

[Default: **40.0**"/minute]

To set the Default Jog Feedrate:

1. See Map 5**, Miscellaneous Setup, Menu C**.
2. Highlight **Linear axis jog feedrate** or **Linear axis jog rapidrate**, and press **ENTER**.
3. In the highlighted entry field, type the appropriate feedrate, and press **ENTER**.

NOTE: To override the default jog feedrate, use the feedrate override switch. The range of the switch is 0% to 120% of the jog feedrate or 0% to 100% of the rapidrate. The switch enables the operator to vary the feedrate in increments of 10 percent.

Maximum Programmed Rotary Axis Feedrate

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 5**, 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

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 Table 2-17, Dry Run Mode M-Codes).
[Default: **1,000.0**]

To set the maximum rotary feedrate:

1. See Map 5**, 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

Set up the feedrate for rotary axes at which the machine travels in Jog Mode. This defines the machine's default jog speed for rotary axis. [Defaults: Feedrate **1016.0**]

To set the default jog rotary feedrate:

1. See Map 5**, Menu C**. Highlight **Rotary axis jog 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 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 5**, Menu C**.
2. Highlight **Servo up Delay**, and press **ENTER**.
3. In the highlighted entry field, enter the 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: **No** (Disabled)]

To activate automatic feed override on arc moves:

1. See Map 5**, Miscellaneous Setup, Menu C**.
2. Highlight **Automatic feedrate override on arcs**.
3. Press **ENTER** to toggle the setting to **Yes**. The CNC activates automatic override feedrate for arc moves made in Cutter Compensation Mode. Select **No** to deactivate the feature.

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 5**, Miscellaneous Setup, Menu C**.
2. Highlight either **Feed Accel/Decel** or **Rapid Accel/Decel**, as applicable, and press **ENTER**.
3. Enter the desired value.

Check DSP Integrity

Check DSP Integrity enables (**Yes**) or disables (**No**) an integrity check of the DSP Motion Control Board and 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 is performed on every command.

To set Check DSP Integrity:

1. See Map 5**, Miscellaneous Setup, Menu C**.
2. Highlight **Check DSP Integrity**, and press **ENTER** to toggle between **No** and **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 5**, Miscellaneous Setup, Menu C**.
2. Highlight **Servo Loop Sample Time (ms)**, and press **ENTER**.
3. Enter the desired value.

Interpolator Rate Factor

Interpolator Rate Factor allows you to specify the interpolator sample rate (in servo loop time).
[Default: **15**]

Interpolator Sample Rate = Servo Loop Sample Time x Interpolator Factor

To set Interpolator Rate Factor:

1. See Map 5**, Miscellaneous Setup, Menu C**.
2. Highlight **Interpolator rate factor**, and press **ENTER**.
3. Enter the desired value.

Acceleration Ramp Type

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

To select the Acceleration Ramp Type:

1. See Map 5**, Miscellaneous Setup, Menu C**. Highlight **Acceleration Ramp type**.
2. Press **ENTER** to select **S-Curve** or **Bell**.

Ramp Z axis during RigidTap

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

To set Ramp Z axis during RigidTap primitive:

1. See Map 5**, Menu C**.
2. Highlight **Ramp Z axis during RigidTap primitive**. Press **ENTER** to toggle between **No** or **Yes**.

Enable Velocity Look Ahead

The 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 5**, Miscellaneous Setup, Menu C**.
2. Highlight **Enable Velocity Look Ahead**, and press **ENTER** to toggle between **No** or **Yes**.

Monitor Setting

The Monitor Setting parameter allows you to set the monitor to either mono or color. Original CNC monitors have mono-type monitors and should be set to **Mono**. Newer CNC monitors have color monitors and should be set to **Color**.

Display Resolution

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

1. See Map 5**, Miscellaneous Setup, Menu C**.
2. Highlight **Display Resolution**, and press **ENTER** to toggle between **SVGA** and **VGA**.

The SVGA setting for flat panels applies to flat panel-based CNC console assemblies purchased from ANILAM as well as flat panels in laptops for use with off-line software.

CNC Startup Mode

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

To set the CNC startup mode:

1. See Map 5**, Menu E**.
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 6**, Menu E**.
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 6**, Menu E**, 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 6**, Menu E**.
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 2-18**. 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 2-18, 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 6**, Menu E**.
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 6**, Menu E**.
2. Highlight **Macro called for Mcode #1 – #10**, and press **ENTER**.
3. Type the value for the variable and press **ENTER**.

Homing the Axes

Most machines have an absolute reference point for each axis. This reference point is defined by the hardware and called “Machine Home.” In most cases, operators use the zero marker pulse (Index Pulse) of the axis encoder and the vector limit switch to define Machine Home. It is also possible to define Machine Home using only the zero marker pulse.

Once defined, the Machine Home feature can be set to require a machine home at startup. Set the **Home required** parameter to **Yes** to operate in this manner. In CNC Mode, the operator can select Homing at any time, regardless of whether it is required by the setup parameter.

The operator selects in what order and at what speed the machine homes the axes.

[Default: **3, 2, 1, 4** corresponding to axes **Z**, **Y**, **X**, and **U** respectively]

To enable the Homing feature:

1. See Map 10**, Home Setup, Menu E**.
2. Highlight **Home required**.
3. Press **ENTER** to toggle the setting to **Yes**. The CNC enables the Homing feature. Select **No** to deactivate the feature. [Default: **No**]

Setting Homing Sequence

The default axes homing order is 3, 2, 1, 4 corresponding to Z, Y, X, and U, respectively. To set the order in which axes are homed:

1. See Map 10**, Home Sequence Setup, Menu F**.
2. Highlight the menu selection corresponding to the axis being set. (For example, highlight **Home sequence for X** to set the sequence of the X-axis.) Press **ENTER**.
3. The CNC Displays a pop-up menu with the following selections:
   - First
   - Second
   - Third
   - Fourth
4. Highlight the appropriate selection, and press **ENTER**.
5. Press (**F10**) to close the pop-up menu.
6. In **Menu F**, select another axis and repeat the procedure.
7. Complete the procedure for all active axes.

Direction of Travel for the Homing Feature

See Map 10, **Home Type Setup, Menu G**.

Use the **Home type** parameter to set the direction of travel for the Homing feature.

This menu specifies the types of homing available. **No Homing** disables the homing function. **Positive**/**Negative** refers to the direction that the axis will travel during Homing, with respect to Machine Home.

[Default: **With negative index limit** setting for **X**, **Y**, and **Z**. **No homing** for **U**.]

With positive/negative index limit

The CNC moves the selected axis in the positive/negative direction specified until the CNC detects an Index pulse from the scale or encoder. To ensure repeatability, the operator must know and physically mark the Machine Home position on the movement axis.

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 the user initiates homing, the CNC travels in the positive/negative direction along the axis being homed until the vector limit switch is detected. The axis will then reverse direction until an Index pulse is detected. The CNC sets Machine Home for that axis at the point, called the Zero crossing, where the Index pulse is detected.

To set the proper direction of travel for Homing:

1. See Map 10, **Home Type Setup, Menu G**.
2. Highlight the appropriate axis to activate a pop-up menu.
3. Highlight the appropriate homing direction selection, and press **ENTER**.
4. Complete the procedure for all active axes.

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 2-10, CAN I/O Systems Vector/Home Limit Switches.)

Setting Homing Speed

To set the speed at which the CNC moves the machine during Homing:

1. See Map 10**, Datum Search Speed Setup, Menu I**.
2. Highlight the menu selection corresponding to the axis being set (For example, highlight **X Datum search speed** to set the X-axis), and press **ENTER**.
3. The entry field highlights. Type the appropriate speed, and press **ENTER**.
4. In **Menu I**, select another axis and repeat the procedure.
5. Complete the procedure for all active axes.
   [Default: **40.0** inches per minute for X, Y, Z, and U]

Setting Home Preset Values

The user can automatically set Machine Zero to another value on any and all axes via the Setup Utility.

To set preset values for axes:

1. See Map 10**, Home Preset Setup, Menu J**.
2. Highlight the selection corresponding to the axis you are presetting (For example, highlight **X Home preset.** to preset X), and press **ENTER**.
3. Enter the preset value in the highlighted entry field.
4. Repeat these steps for all preset axes.
5. Highlight **Home preset**.
6. Press **ENTER** to toggle the selection to **Yes**. The CNC displays the new values in place of **0.0000**. Select **No** to disable the preset values.
   [Default: **0.0000** distance for X, Y, Z, and U, **Off** (Disabled) for **Home preset**]

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 2-19** for message-code ranges and message types.

Table 2-19, 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 11**, 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 Map11**, Menu C**. Highlight **Edit Error Messages**, and press **ENTER**.

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

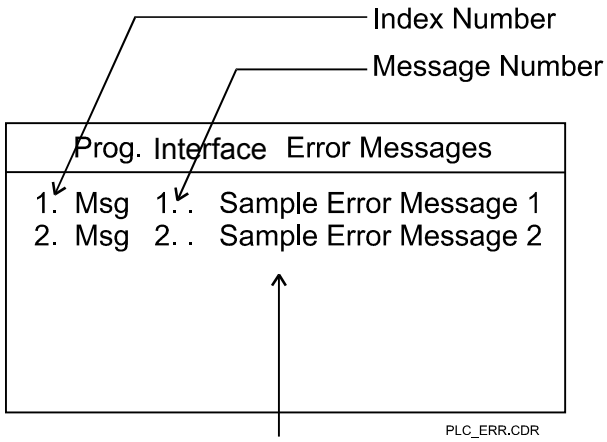


Figure 2-4, Builder Text - Error Message Window

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

   The CNC activates the message text box.

3. Type the 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 11**, Menu C**. Highlight **Edit Warning Messages**, and press **ENTER**.

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

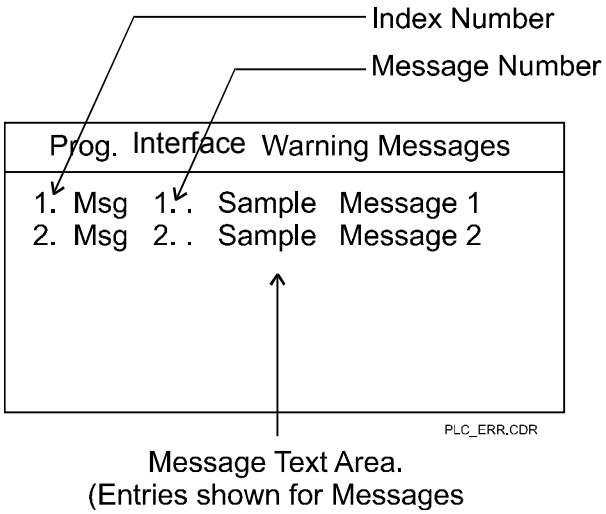


Figure 2-5, Builder Text - Warning Message Window

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 11**, Menu C**. Highlight **Edit Soft Key Inputs**, and press **ENTER**.

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

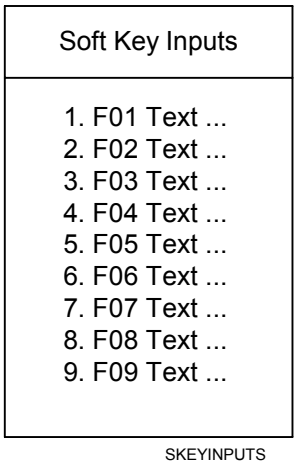


Figure 2-6, Builder Text – Soft Key Inputs Window

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

CNC systems are capable of displaying messages and additional text in a language other than English. The required text files must be installed for the language options to operate properly.

[Default: **English**]

If the operator attempts to set the option for a language and the CNC cannot find the associated text file, an error message is displayed.

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

1. See Map 11**, Language Setup, Menu D**.
2. Highlight the appropriate language, and press **ENTER**.
3. The CNC installs the language file and restarts the system. The displayed messages and other text will now appear 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 11**, Builder Setup, Menu B**.
3. Highlight **Software Update**, and press **ENTER**.
4. Follow the prompts on the screen to complete the installation. The process lasts less than five minutes. If the CNC cannot install the new version, it displays an error message. Otherwise, it displays messages when it completes each step of the installation procedure (Extracting control software, etc.).
5. The CNC displays a message when 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 using RS-232 or networking, and then run it in Auto Mode, rather than DNC.

Selecting a DNC Execution Mode

See Map 11**, DNC Setup, Menu E**. 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 11**, DNC Setup, Menu E**.
2. Highlight **Execution Mode**, and press **ENTER** to toggle between **Drip Feed** and **Buffered**.

Setting the Buffer Size

This parameter allows you to specify the amount of memory to be reserved for DNC in Buffered Mode. The selections are as follows:

- 16K
- 32K [Default]
- 64K
- 128K
- Max

Max indicates that the CNC 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 11**, Options Setup, Menu G**.
2. Highlight the amount of memory to be devoted to the Buffer, and press **ENTER**.

Enabling/Disabling ToolComp and CornerRad

The operator can enable/disable processing for tool radius compensation and corner rounding. In DNC Mode, disable these features unless absolutely necessary. If you leave these features enabled, even if a program does not use them, the CNC will waste computing time looking for them.

[Default: **Yes** (Disable)]

To enable/disable ToolComp and CornerRad:

1. See Map 11**, DNC Setup, Menu E**.
2. Highlight **Ignore ToolComp and CornerRad**.
3. Press **ENTER** to toggle the selection. Select **Yes** to disable Tool Comp and CornerRad. Select **No** to enable Tool Comp and CornerRad during DNC.

Setting DNC Mode to Hold Execution of Program Until You Press Start

Refer to **Table 2-20**. See Map 11, **DNC Setup, Menu E**. Use the **Wait for Start** parameter to specify whether the CNC will hold data transmission until you press **START**.

Table 2-20, Wait for Start Parameter Options

Option	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 11, **DNC Setup, Menu E**.
2. Highlight **Wait for Start**, and press **ENTER** to display the **Options** pop-up menu.
3. In **Menu G**, highlight an option, and press **ENTER** to activate it.

Security

For default passwords, see "Password Restricted Parameters" in "Section 1."

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, you must erase the current configuration file and reinstall the software (thus restoring the default password).

Make a printed copy of the configuration file before erasing it. Settings must be input manually after software reinstallation.

To change the password:

1. See Map 11, **Security Setup, Menu I**.
2. Highlight the level of the password to be changed, and press **ENTER**. The CNC prompts for the old password.
3. Type the old password, and press **ENTER**. The CNC prompts for the new password.
4. Type the new password, and press **ENTER**. The CNC prompts for confirmation of the new password.
5. Re-type the new password, and press **ENTER**. The CNC activates the new password.

Probing

Setting the Spindle Probe Type

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

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.

To set the probe type:

1. See Map 11**, Probing Setup, Menu J**.
2. Highlight **Spindle probe type**, and press **ENTER** to toggle 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 11**, Probing Setup, Menu J**.
2. Highlight **Nominal probe stylus diameter**, and press **ENTER**.
3. Type the nominal probe stylus diameter, and press **ENTER**.
   [Default: **0.0000**]

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 11**, Probing Setup, Menu J**.
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**]

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 11**, Probing Setup, Menu J**.

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: (100–1000)

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 11**, Probing Setup, Menu J**.
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: (-2–2)

Setting the Z First Pick, FAST Feedrate

Sets user definable FAST feedrate.

To set the FAST feedrate:

1. See Map 11**, Probing Setup, Menu J**.
2. Highlight **Z first pick, FAST feedrate**, and press **ENTER**.
3. Type the FAST feedrate, and press **ENTER**.
   [Default: **0.0**] Valid range: (2.5–2540.0)

Setting the Z First Pick, MEDIUM Feedrate

Sets user definable MEDIUM feedrate.

To set the MEDIUM feedrate:

1. See Map 11**, Probing Setup, Menu J**.
2. Highlight **Z first pick, MEDIUM feedrate**, and press **ENTER**.
3. Type the MEDIUM feedrate, and press **ENTER**.
   [Default: **0.0**] Valid range: (2.5–508.0)

Setting the Z Final Pick, SLOW Feedrate

Sets user definable SLOW feedrate.

To set the SLOW feedrate:

1. See Map 11**, Probing Setup, Menu J**.
2. Highlight **Z final pick, SLOW feedrate**, and press **ENTER**.
3. Type the SLOW feedrate, and press **ENTER**.
   [Default: **0.0**] Valid range: (0.1–254.0)

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 feedrate:

1. See Map 11**, Probing Setup, Menu J**.
2. Highlight **Z retract amount**, and press **ENTER**.
3. Type the Z retract amount, and press **ENTER**.
   [Default: **0.0000**] Valid range: (0.0100–25.400)

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 11**, Probing Setup, Menu J**.
2. Highlight **XY retract amount**, and press **ENTER**.
3. Type the XY retract amount, and press **ENTER**.
   [Default: **0.0000**] Valid range: (0.0100–25.400)

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 11**, Probing Setup, Menu J**.
2. Highlight **Z start position**, and press **ENTER**.
3. Type the Z start position, and press **ENTER**.
   [Default: **0.0000**] Valid range: (0.0000–999.0000)

Setting the Diameter of Tool Probe Gauge

Sets the probe calibration standard diameter.

To set the XY retract amount:

1. See Map 11, **Probing Setup, Menu J**.
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: (0.1000–508.0000)

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 11, **Probing Setup, Menu J**.
2. Highlight **Positioning feedrate normally**, and press **ENTER**.
3. Type the positioning feedrate normally, and press **ENTER**.
   [Default: **0.0**] Valid range: (0.1–25400.0)
   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 11, **Probing Setup, Menu J**.
2. Highlight **First touch feedrate**, and press **ENTER**.
3. Type the first touch feedrate, and press **ENTER**.
   [Default: **0.0**] Valid range: (0.1–2540.0)
   Typical value: 50 inches/minute (IPM)

Setting the Nominal Probe Stylus Ball Radius

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

To set the nominal probe stylus ball radius amount:

1. See Map 11, **Probing Setup, Menu J**.
2. Highlight **Nominal probe stylus ball radius**, and press **ENTER**.
3. Type the nominal probe stylus ball radius, and press **ENTER**.
   [Default: **0.0000**] Valid range: (0.0100–25.4000)

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 11**, Probing Setup, Menu J**.
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: (0.1000–508.0000)

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 11**, Probing Setup, Menu J**.
2. Highlight **Invert probe logic**, and press **ENTER** to toggle to the correct logic for your probe.
   [Default: **No**]

Section 3 - Operator Setup

Refer to "Section 6 - Setup Utility Maps, Map 12, **Operator Setup, Menu B**." The Operator Setup allows the user to configure settings for the following parameters:

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

Control Software Parameters

Go to Map 12, **Control Software Setup, Menu C**. This menu accesses settings that affect the control software. Refer to **Table 3-1** for descriptions and setting information.

Table 3-1, Control Software Parameters

Control Software Parameter	Function	Settings
Default plane	A plane defines movement along two axes, excluding a third, so planar movement is two-dimensional. The CNC confines circular moves and tool diameter compensation to the plane the user selects. Linear moves can occur in all three axes simultaneously.	**XY** [Default] 3000M-3X XZ YZ
Default units	Switches the default measurement units (Inch/MM Modes).	**Inch** Activates Inch Mode. [Default] Original setting. MM Activates MM Mode as default.
Default axis values	Switches default Absolute/Incremental Mode (determines how axis values for arcs, lines, and other moves are measured).	**Absolute** Make every move in reference to an absolute zero position (Program Zero or Part Zero). [Default] Incremental Make every move in reference to the last programmed endpoint.

(Continued…)

Table 3-1, Control Software Parameters (Continued)

Control Software Parameter	Function	Settings
Circle adjustments	Determines whether circle centers or endpoints will be adjusted. Adjustment is required when the CNC encounters illogical, programmed circle center or endpoint coordinates.	Center — When the CNC encounters illogical coordinates for either a circle center or endpoint, it will adjust the position of the circle center. **End-point** — When the CNC encounters illogical coordinates for either a circle center or endpoint, it will adjust the position of the circle endpoint. [Default]
Circle centers	Switches the default mode for programmed circle center coordinates.	Absolute — CNC interprets programmed circle center coordinates as absolute values. Incremental — CNC interprets programmed circle center coordinates as incremental values. **Modal** — CNC interprets programmed circle center coordinates as the current mode of the CNC, absolute, or incremental values. [Default]
Maximum arc correction	Maximum amount of correction the CNC will apply to an input arc value before it declares an error.	0.005000 [Default]
Internal accuracy	Maximum accuracy available on the CNC (system resolution).	0.00000100 [Default]
External accuracy	Maximum accuracy obtainable on a given machine (machine resolution).	0.00010000 [Default]
Feedrate program/display resolution	Sets the number of decimal positions available for programmed and displayed feedrates. (If set to 1, resolution is X.X; if 2, X.XX.)	1 [Default]
Compensation cutoff angle	Prevents or reduces wasted travel on acute angle. (Refer to Figure 3-1, Compensation Cutoff Angle.)	15.0 (degrees) [Default]

(Continued…)

Table 3-1, Control Software Parameters (Continued)

Control Software Parameter	Function	Settings
Disk access marker	Activates/deactivates the disk access marker.	On Activates the disk access marker. When the CNC is reading/writing information from/to a disk or other medium, the disk access marker appears in the upper-left corner of the screen. The disk access marker looks like a small arrow. [Default] Off Deactivates disk access marker.
Max memory allocated (in MB-bytes)	Limits the amount of memory available to the software, preventing the CNC program from tapping into Windows' large virtual memory supply. (Allocating too much memory to the control software will dramatically increase startup time.) Change on off-line systems only.	Valid range is 2 to 18. [Default: 4]
Force simulation mode	Determines whether the CNC will move the machine.	Yes CNC is in Simulation Mode. You cannot move the machine. [Default] No You can move the machine.
Enable radius compensation error checking	Activates the tool radius compensation error checking. The error checking is designed to eliminate simple gouges caused by overcompensation.	Yes Enable [Default] No Disable
Screen blanking delay (minutes)	Sets time in minutes before the screen goes blank when the CNC is not performing tasks.	[Default: 5]

Compensation Cutoff Angle

Refer to **Figure 3-1**. Assume you make all programmed moves with Tool Diameter Compensation active. **Figure 3-1** shows the following two cases:

- **Diagram A** shows the tool path that results when you do not use compensation cutoff angle. 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 you use a compensation cutoff angle (assume 15 degree compensation cutoff angle value). The CNC introduces one arc move, equal to the radius of the cutter, between Programmed Moves 1 and 2. (You do not program this arc. It is a function of the active Compensation Cutoff Angle.) This alters the tool path and decreases the travel distance required to complete the programmed moves.

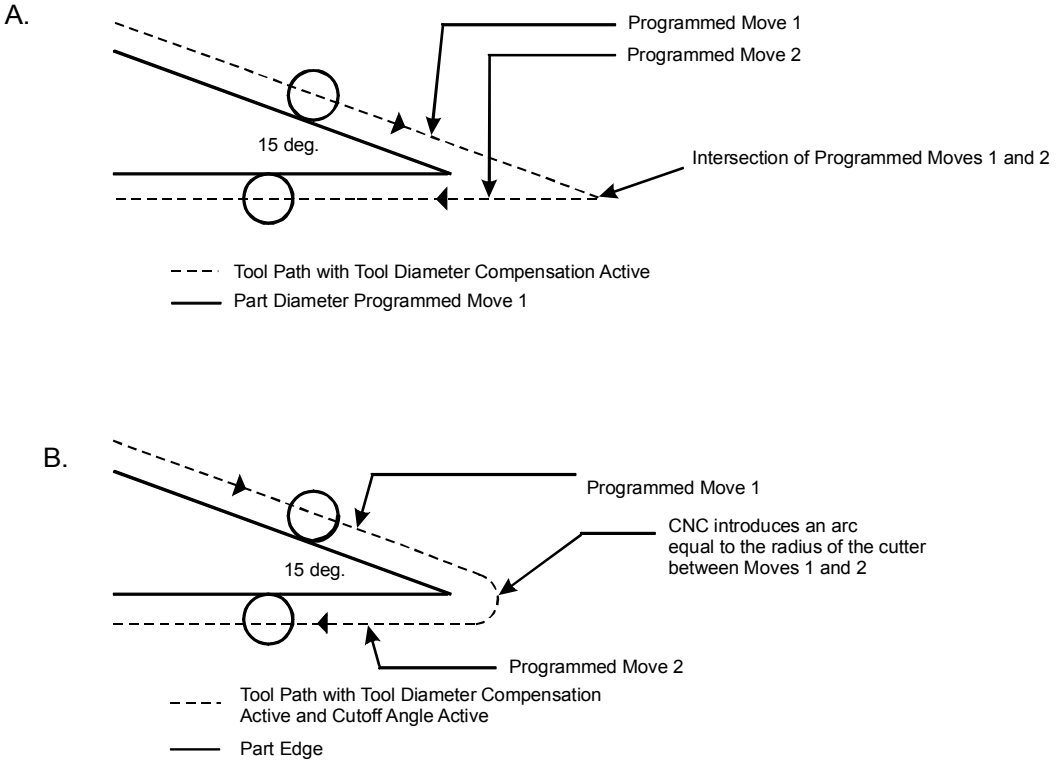


Figure 3-1, Compensation Cutoff Angle

Communications Parameters

Go to Map 13**, Communications Setup, Menu C** for the available communications parameters. See **Table 3-2** for a description of each parameter and its settings.

Table 3-2, Communications Parameters

Communications Parameter	Function	Settings
Port	Select communications port (COM1) or disable. Must enable to perform DNC or other remote communications.	COM1 COM2 [Default] Disabled
Baud	Select baud	110 150 300 600 1200 2400 4800 9600 [Default] 19,200
Parity	Select parity	Odd Even [Default] None
Data bits	Type number of data bits	7 [Default] 8
Stop bits	Type number of stop bits	0 1 [Default]
Software	Refers to XON or XOFF or software handshaking (transmission/receipt of data via RS-232 channels) in commercial communications packages.	On Enables handshaking. [Default] Off Disables handshaking.

Simulated Draw Mode Setup Parameters

Refer to Map 13, **Draw Setup, Menu G**, for the available Draw Mode Parameters. The parameters affect both real time and Simulated Draw Modes. Refer to **Table 3-3** for a description of each parameter and its settings.

Table 3-3, Draw Mode Setup Parameters

Draw Mode Parameter	Function	Settings
Restore to previous session	Sets the CNC to re-activate the last active session when the user re-enters Draw.	Yes CNC reactivates last session when Draw activated. [Default] No CNC ignores parameter.
Default program block mode	Sets default mode for 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. The grid size will be in the active measurement unit, Inch or MM.	1.0 [Default] Type a value. (Example: The default value is 1.0. 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 1.0 in Inch Mode and the user switches to MM Mode, the CNC changes the Grid Size to 25.4 mm.
Tool display	Turns the tool display on and off.	On The static tool (as defined in the Tool Page) will be displayed as it cuts the part. [Default] Off No tool is displayed.
Default tool type	Determines shapes of displayed tool.	None No tool shown Flat Flat end tool shown [Default] Ball Ball end tool shown

(Continued…)

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

Draw Mode 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 runs without compensated moves. Then, the program runs showing compensated moves. This allows the user to compare the two paths and determine programming errors related to compensation. [Default].
Draw view	Determines perspective of Draw view.	XY (top view) Views program in X and Y. XZ Program viewed in X and Z. YZ Program viewed in Y and Z. ISO Program viewed in X, Y, and Z. [Default]
Aspect ratio correction factor	Corrects for distortion in displayed graphics. (Flattened circles, etc.)	1.33 [Default] CAUTION: Only qualified technicians should adjust this setting.

Edit Mode Setup Parameters

Go to Map 13, **Editor Setup, Menu M**, for the available Edit Mode Parameters. Refer to **Table 3-4** for a description of each parameter and its settings.

Table 3-4, Edit Mode Parameters

Edit Mode Parameter	Function	Settings
Create backup program	The CNC creates a backup program that it updates each time you edit the program. The backup program will not contain an edit until you make a new edit.	Yes Creates and maintains a backup program. 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 an updated file. [Default] No CNC does not delete internal file when the user saves a program
Case sensitive Find	Determines whether the Find feature will look for uppercase letters and lowercase letters to determine a match.	Yes Find search parameter looks for words that match the typed word exactly regarding capitalization and style. No Find search parameter looks for typed word regardless of capitalization and style. [Default]
Memory reserved from Editor (in K-bytes)	Specifies maximum memory allocation for the Editor.	300 [Default] Type a value.

Program Directory Parameters

Go to Map 14**, Program Setup, Menu C** for the available program parameters. These parameters determine the way in which the program directory displays information, whether to delete backup files during optimization, and whether and how often the disk is checked via software.

Refer to **Table 3-5** for a description of each parameter and its settings.

Table 3-5, Program Parameters and Selections

Program Directory Parameter	Function	Settings
Program directory pattern	Type of programs displayed.	*.M (program file extension) NOTE: Do not change this parameter.
Program directory display mode	Specifies the program information to display in the Program Directory.	Short Filename and extension only. [Default] Long Detailed program information, including file size, etc.
Program directory sort order	Specifies order in which program 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	Specifies whether to delete backup files during hard drive optimization. NOTE: Disk Optimization 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.	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] Type user directory location.

Display Settings

Go to Map 14, **Display Setup, Menu F** for available display parameters. The listed parameters control how the CNC displays text and graphics on the screen. There are separate settings for the Editor, CNC Control and Help screens, and the displayed soft keys.

Printer Settings

Go to Map 14, **Printer Setup, Menu G** for the available printer parameters. Refer to **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 devices	Specifies where file is printed.	To send a file to the printer, type Prn. To print to another file, type the: 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 filename typed is an existing file, the CNC replaces the data with the print file data. Prn [Default] **NOTE:** The user directory is C:\USER.
Lines per page	Number of lines to be printed per page (8.5" x 11").	Type value. [Default: 55]
Page heading	Prints a page heading including filename, date and time and page number.	**Yes** Print heading. [Default] No Do not print heading.
Line numbers	Prints line numbers on hard copy of file.	Yes Select to print line numbers. **No** Select if no line numbers are desired. [Default]
Print quality	Sets print quality. Generally, the lower the printer quality, the faster the file prints.	NLQ Near Letter Quality; highest quality, lowest speed Utility Middle quality, middle speed High Speed Low quality, high speed **Ignore** Uses printer defaults [Default]
Characters per inch	Sets the number of characters to be printed per inch. Select Ignore to print at the default value.	**10** (characters per Inch (CPI)) [Default] 12 17 20 Ignore (Uses printer defaults)
Wrap text	Wraps text to the next line if program is longer than 80 characters.	**Yes** Select to wrap text. [Default] No Select to print beyond 80 characters.

Section 4 - Configuration Utilities

Use Utilities to configure information saved in the Setup Utility. Refer to Map 15.

Save Configuration

The **Save Configuration** 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. Go to Map 15**, Utilities Setup, Menu B**. Highlight **Save Configuration** and press **ENTER**. The system prompts for a password.

2. Type the limited access password (see Table 1-3, Default Machine Passwords, "Access Level, Limited – Operator") and press **ENTER**. The configuration will be saved. The backup filename is **P3MCFG.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. Go to Map 15**, Utilities Setup, Menu B**. Highlight **Copy Configuration**, and press **ENTER**. The system prompts for a password.

2. Type the limited access password (see Table 1-3, Default Machine Passwords, "Access Level, Limited – Operator"), and press **ENTER**. **Menu C** is displayed.

3. Highlight **A:** to copy the configuration to a floppy diskette. The configuration will be saved as **A:\P3MCFG.CFG**.

4. Highlight **Other..** to save the configuration to another drive or under another filename.

5. 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 select 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:\P3MCFG.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. Go to Map 15**, Utilities Setup, Menu B**. Highlight **Restore from Copy**, and press **ENTER**. The system prompts for a password.

2. Type the limited access password (see Table 1-3, Default Machine Passwords, "Access Level, Limited – Operator") and press **ENTER**. **Menu D** is displayed.

3. Highlight **A:** to restore the configuration from a floppy diskette. The configuration will be restored from **A:\P3MCFG.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:

1. Go to Map 15**, Utilities Setup, Menu B**. Highlight **Restore from Backup**, and press **ENTER**. The system prompts for a password.

2. Type the limited access password (see **T**able 1-3, Default Machine Passwords, "Access Level, Limited – Operator") and press **ENTER**. The system automatically swaps the current file with the backup file.

Compare Configuration

Use the **Compare Configuration** feature to determine if your current configuration file is the same as another file either on the A drive or elsewhere.

1. Go to Map 15**, Utilities Setup, Menu B**. Highlight **Compare Configuration**, and press **ENTER**. **Menu E** is displayed.

2. Highlight **A:** to compare the current file with P3MCFG.CFG on the A:-drive.

3. Highlight **Other...** and the Compare pop-up is displayed. Type the directory with which you wish to compare files. For example, **C:\Home\FILE_1**.

Print Configuration

Use the **Print Configuration** feature to print a file to a printer.

NOTE: If there is no printer connected to your parallel port, an Error message is displayed.

1. Go to Map 15**, Utilities Setup, Menu B**. Highlight **Print Configuration**, and press **ENTER**.

2. The **Print Options Setup, Menu F** is displayed.

3. 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**. **P3cfg.TXT** will be the filename. Press **Yes** (**F1**) to print to your A:-drive. Press **No** (**F2**) to return to **Menu B**.

   – or –

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

Section 5 - Fine-Tuning Systems with Linear Encoders

Machines equipped with linear encoders require changes in the Setup Utility to minimize or eliminate the effects of lost motion on the control system.

NOTE: This procedure requires you to move between the Setup Utility and the CNC's Manual screen. When you change the Setup Utility, press **Save** (**F1**) to save the change when so prompted.

Lost motion is the distance the ballscrew and/or motor move before the table begins to move. Lost motion occurs because of the mechanical characteristics of the motor and ballscrew. An encoder measures table motion. Therefore, movement corrections required by mechanical characteristics or servo drift can only be made after the encoder detects table movement.

To compensate for the observed lost motion, adjust the required setup parameters.

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

1. Ensure the CNC is set to linear encoder, with the proper encoder resolution and display resolution settings. Refer to "Section 2 - Builder Setup" for details on how to configure these parameters.
2. Use the **JOG** key to cycle through the available jog modes. Set the CNC to **Jog Mode 1**. This sets the axis to move in increments of one multiplied by the machine resolution each time one of the **JOG** keys (X-, X+, Y-, Y+, Z-, Z+, U-, or U+) is pressed.
3. Press the required **JOG** key (**X-**, **X+**, **Y-**, **Y+**, **Z-**, **Z+**, **U-**, or **U+**) to move the axis.
4. Press the required **JOG** key to move the axis in the opposite direction of the move made in step 3.
5. Look at the handwheel of the axis. When lost motion occurs, the handwheel moves a much greater distance than the system resolution (for example, 0.0005" on a 10 micron scale). That distance is the lost motion on that axis.

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

6. If your system is oscillating (hunting) within that distance, make the following adjustments within the Setup screen:
   - Refer to "Section 6," Map 5**, Miscellaneous Setup, Menu C**. Check the Servo Loop Sample Time (ms) parameter. With a .5, 1, 2, 5, or 10 micron resolution glass scale, the value should remain at the default value, 0.400 msec. (Use increments of 0.050 msec when making minor adjustments to this value.)
   - Refer to "Section 6," Map 5**, No Motion Filter Parameter Setup, Menu G**. Eliminate the hunting created by lost motion. While the axis is still, adjust the No-Motion gain tables to limit the reaction time of the servo's digital filter:
     - Eliminate any integral gain by setting the Ki and IL to 0. Integral gain accumulates over time, increasing the correction output and instigating oscillation due to lost motion.
     - Set the Kp value from 1.50 to 3.0. If oscillation continues, reduce Kp in steps of 0.10 until oscillation stops.

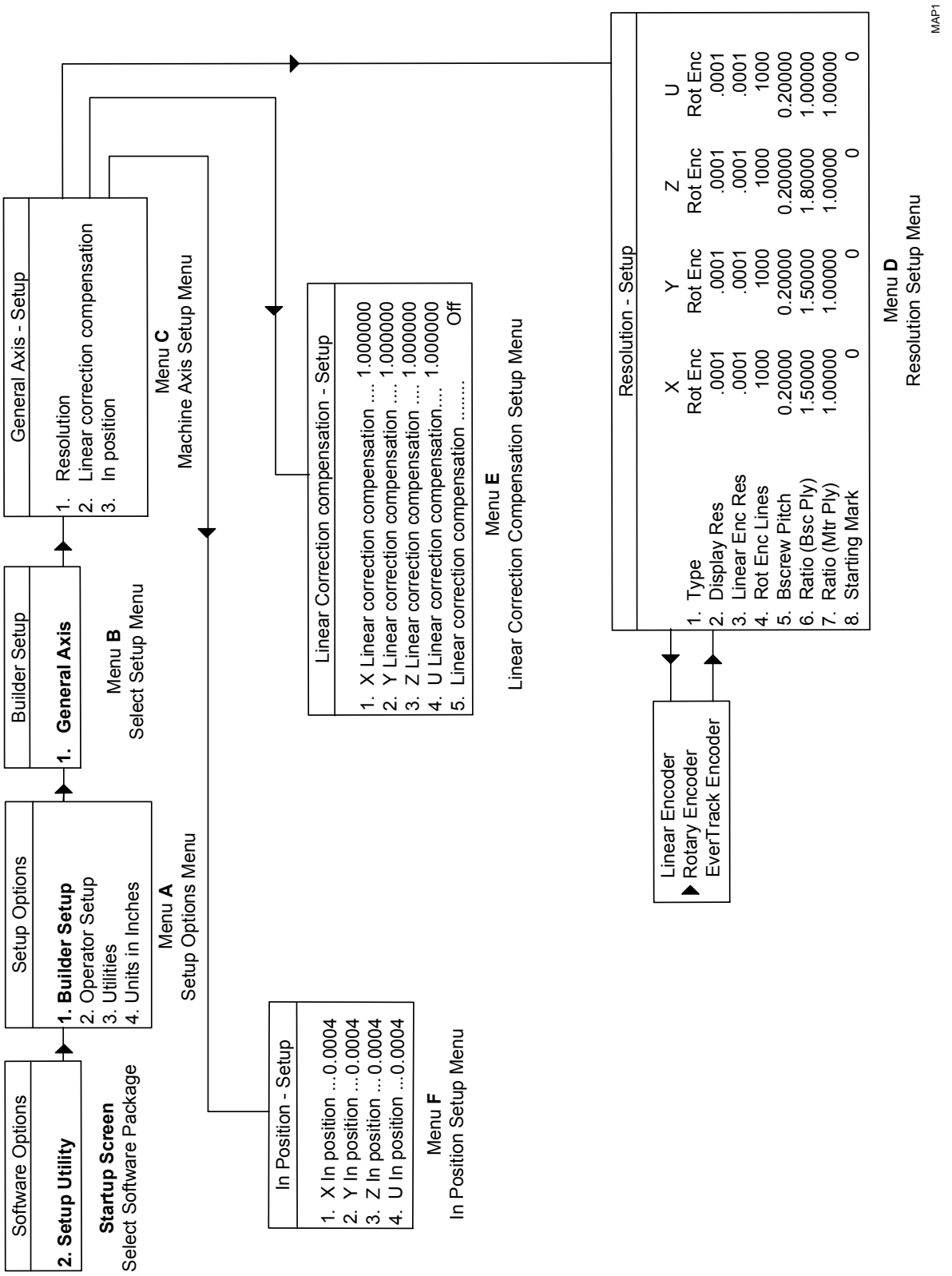
NOTE: Lowering the Kp gain delays the correction time not only to lost motion but also to drift offsets on the system.

7. Repeat the procedure on all enabled axes having linear encoders.
8. From the CNC's Manual screen, use the **JOG** keys in Feed Mode to test the responsiveness of the axis. Refer to "Section 6," Map 5**, Miscellaneous Setup, Menu C**. If an axis coasts when the **JOG** key is released, reduce the **Feed Accel/Decel (ms)** setting in the Setup screen. Use steps of 10 msec to adjust this parameter.

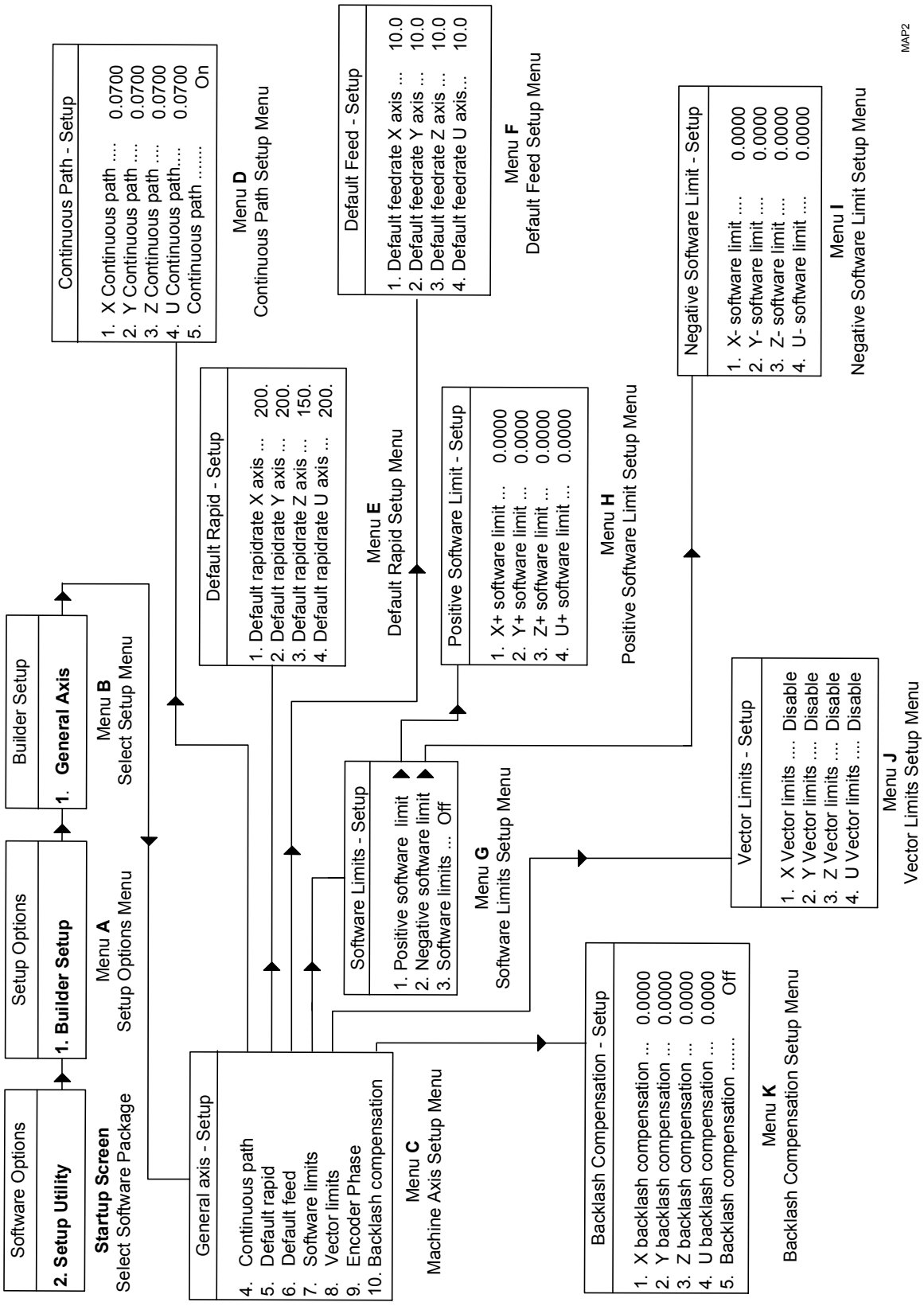
NOTE: Always make adjustments to Setup parameters in small increments and check often until the problem is fixed. If you make large changes in parameters, you could fix that particular problem, but you could cause others in the process.

Section 6 - Setup Utility Maps

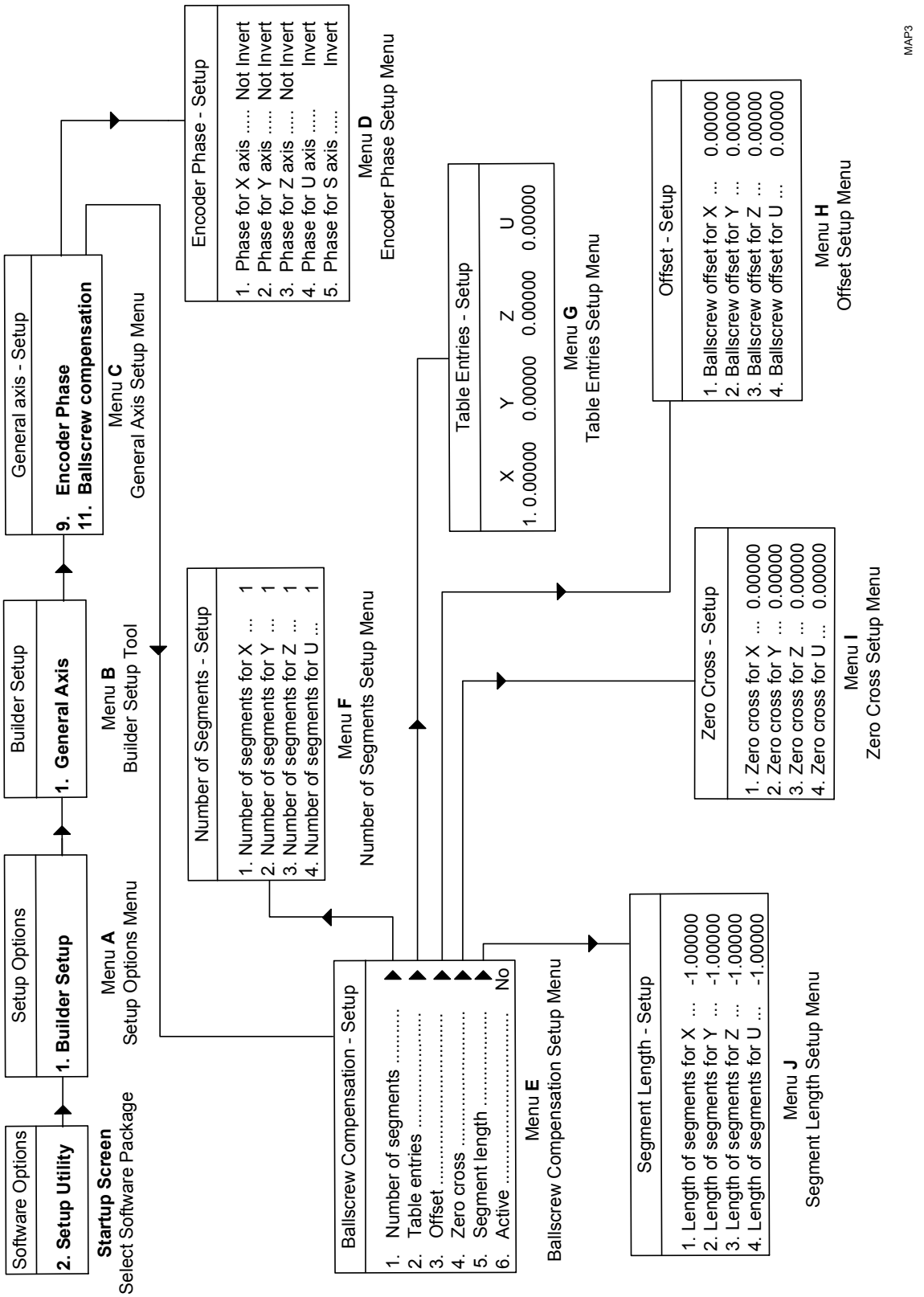
This section contains maps referenced in “Section 1 – Section 5.” Refer to “Navigating Through the Setup Utility” in “Section 1” for instructions on how to use the software and maps.



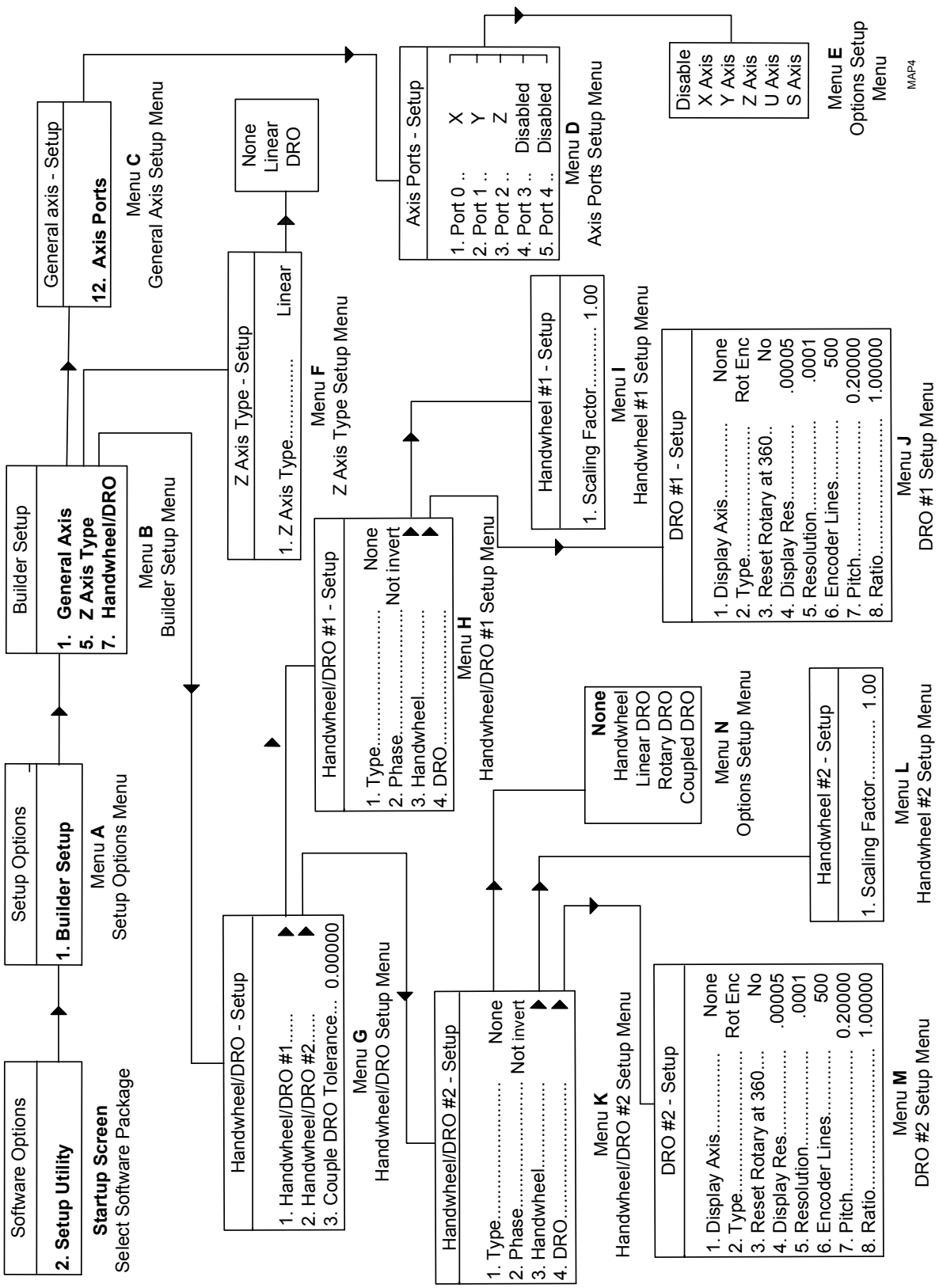
Map 1



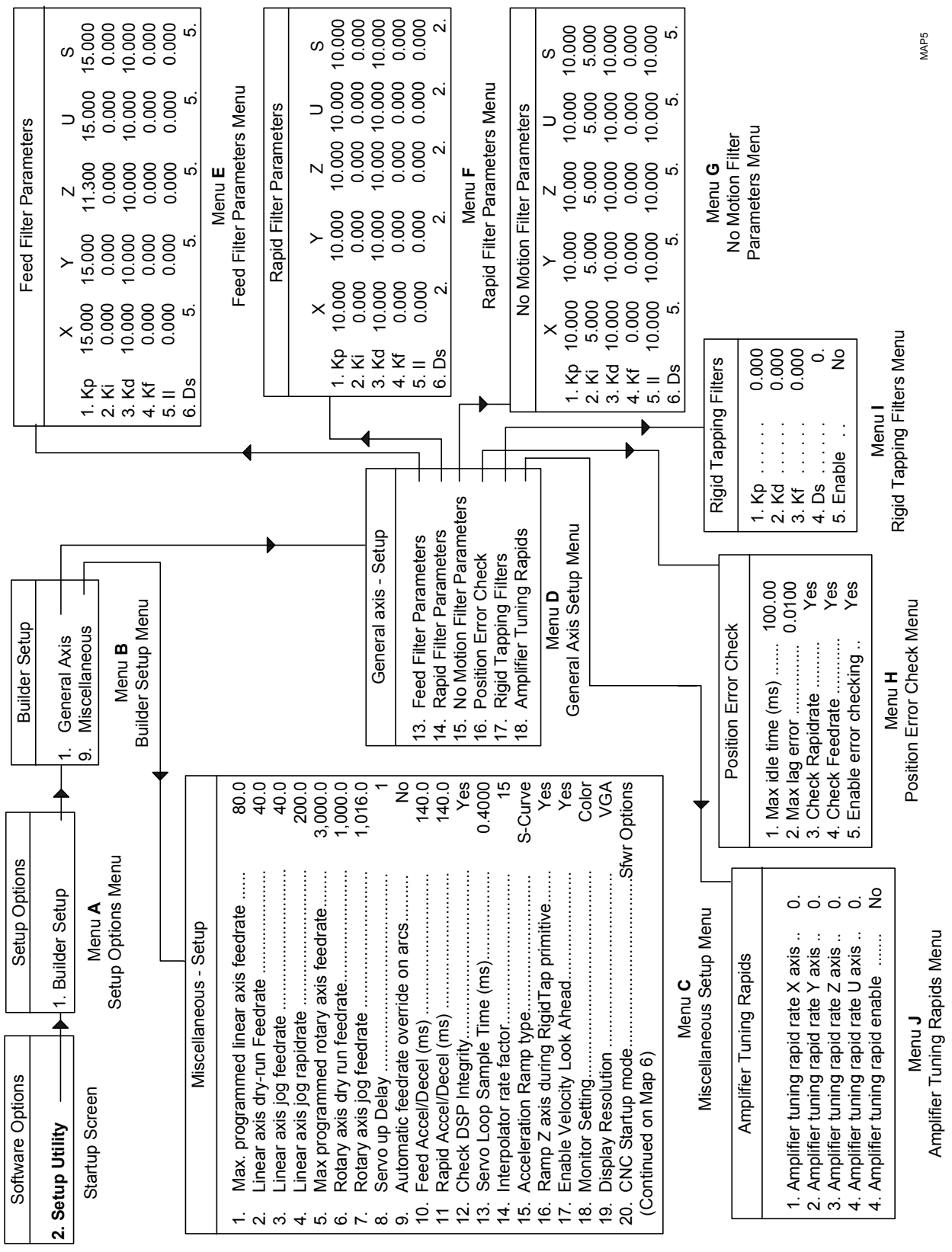
Map 2



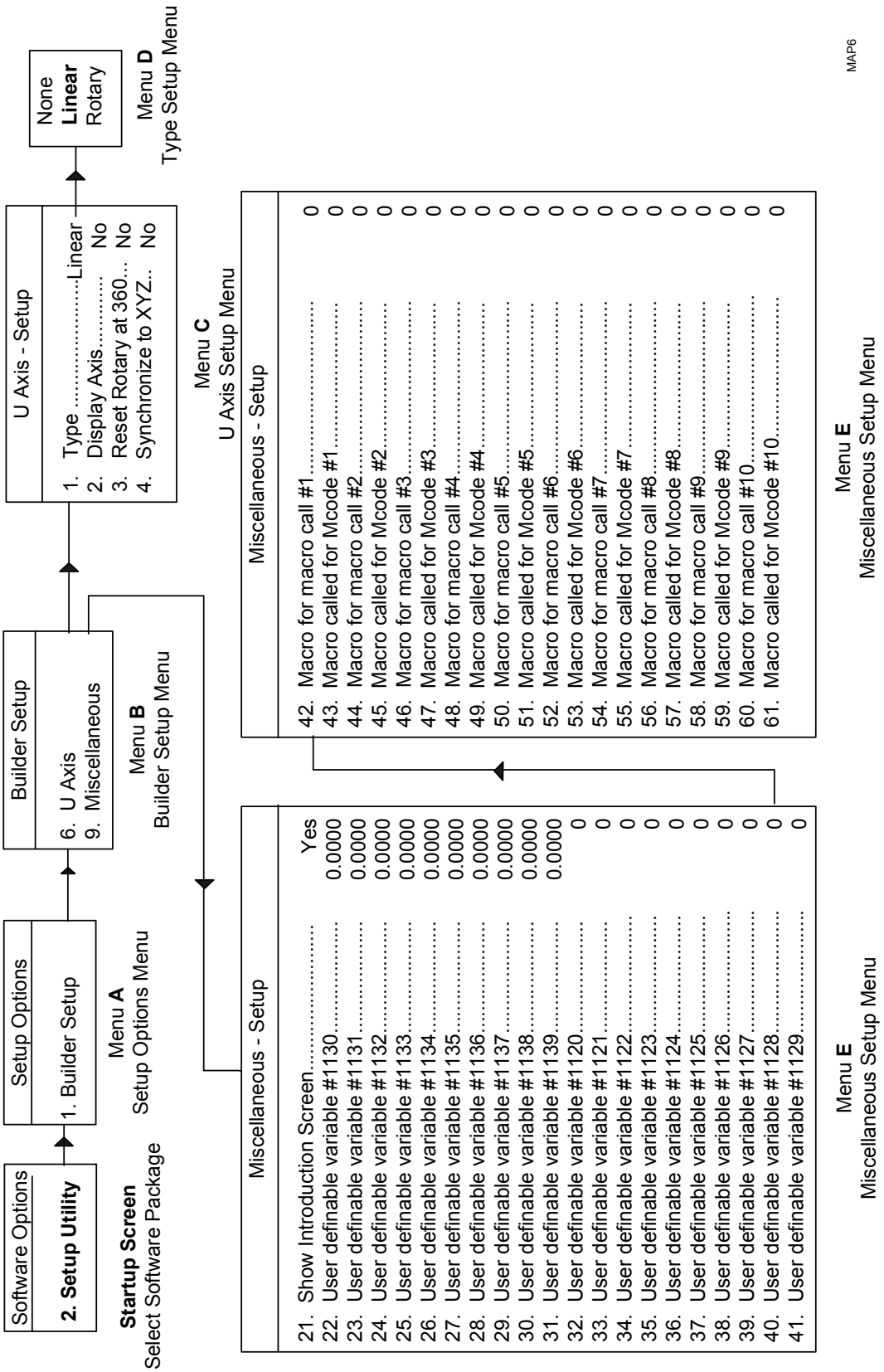
Map 3



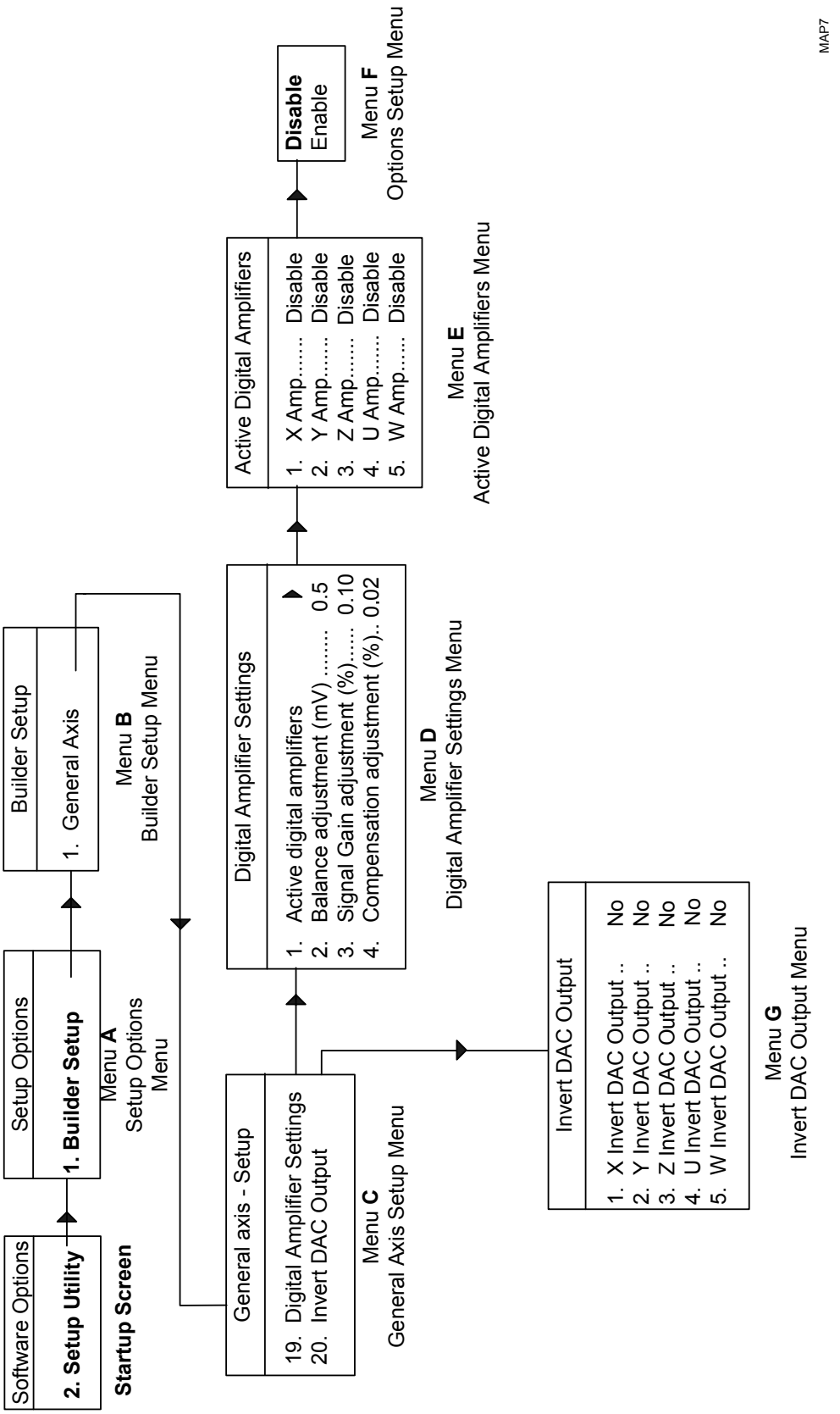
Map 4



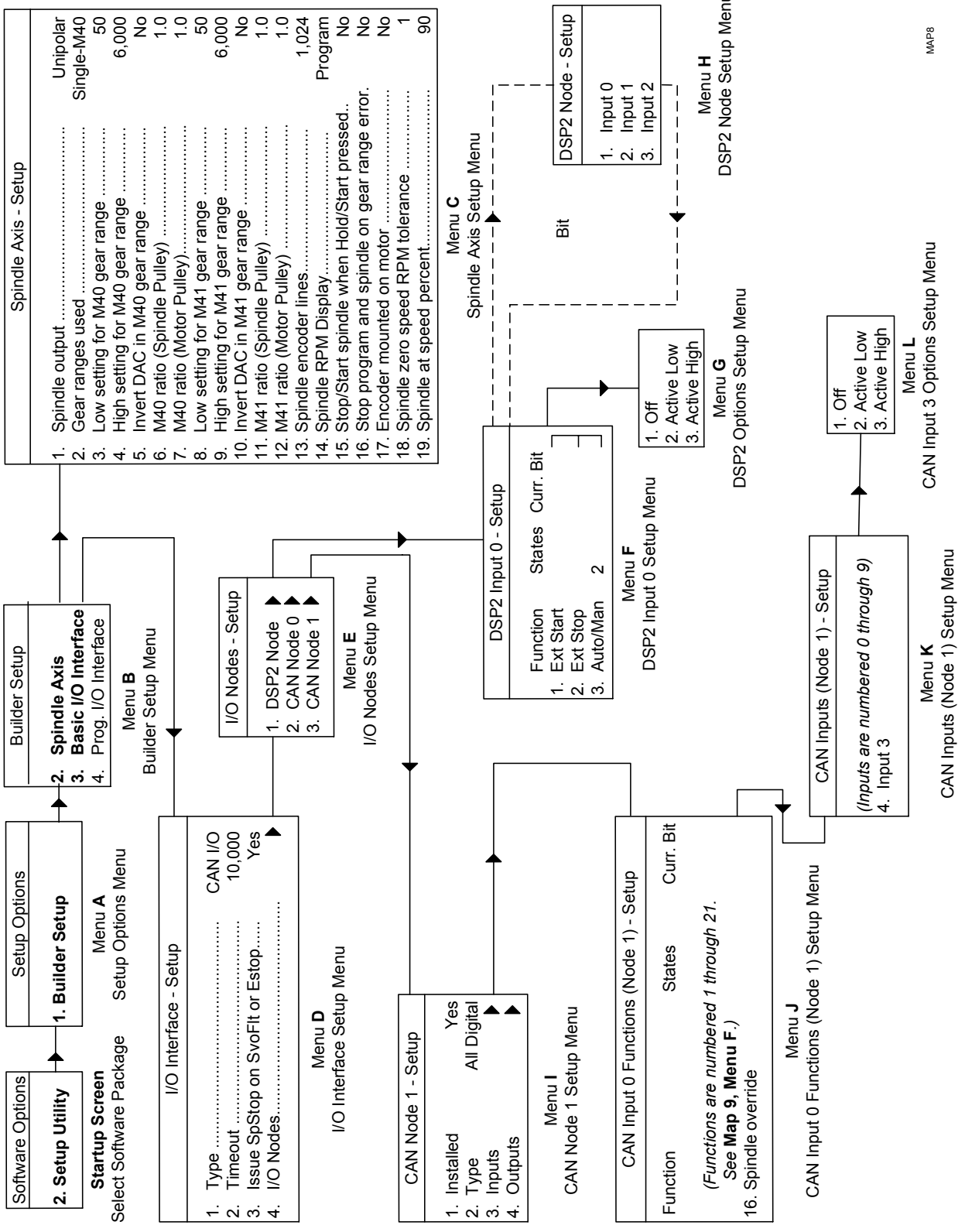
Map 5



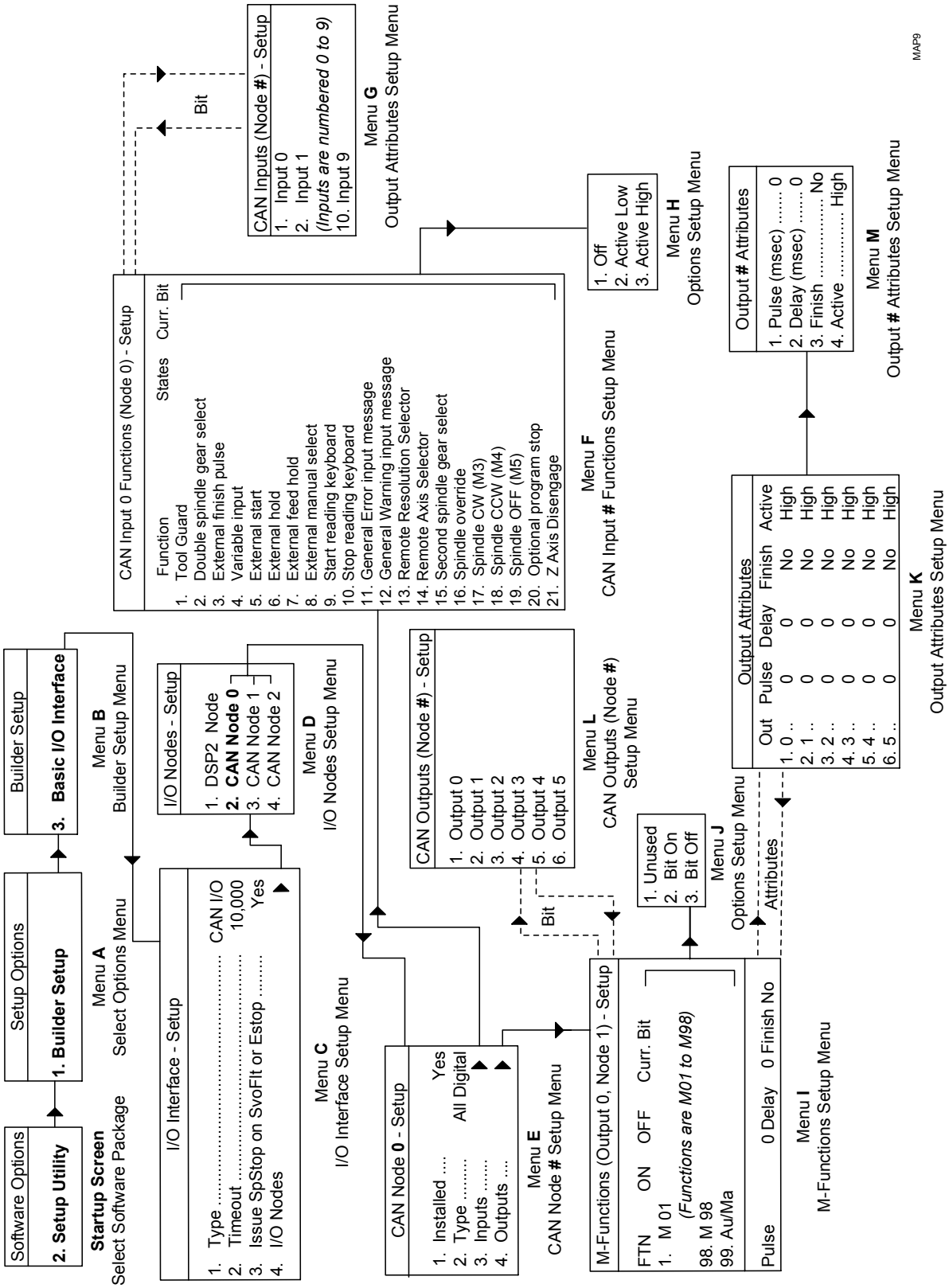
Map 6



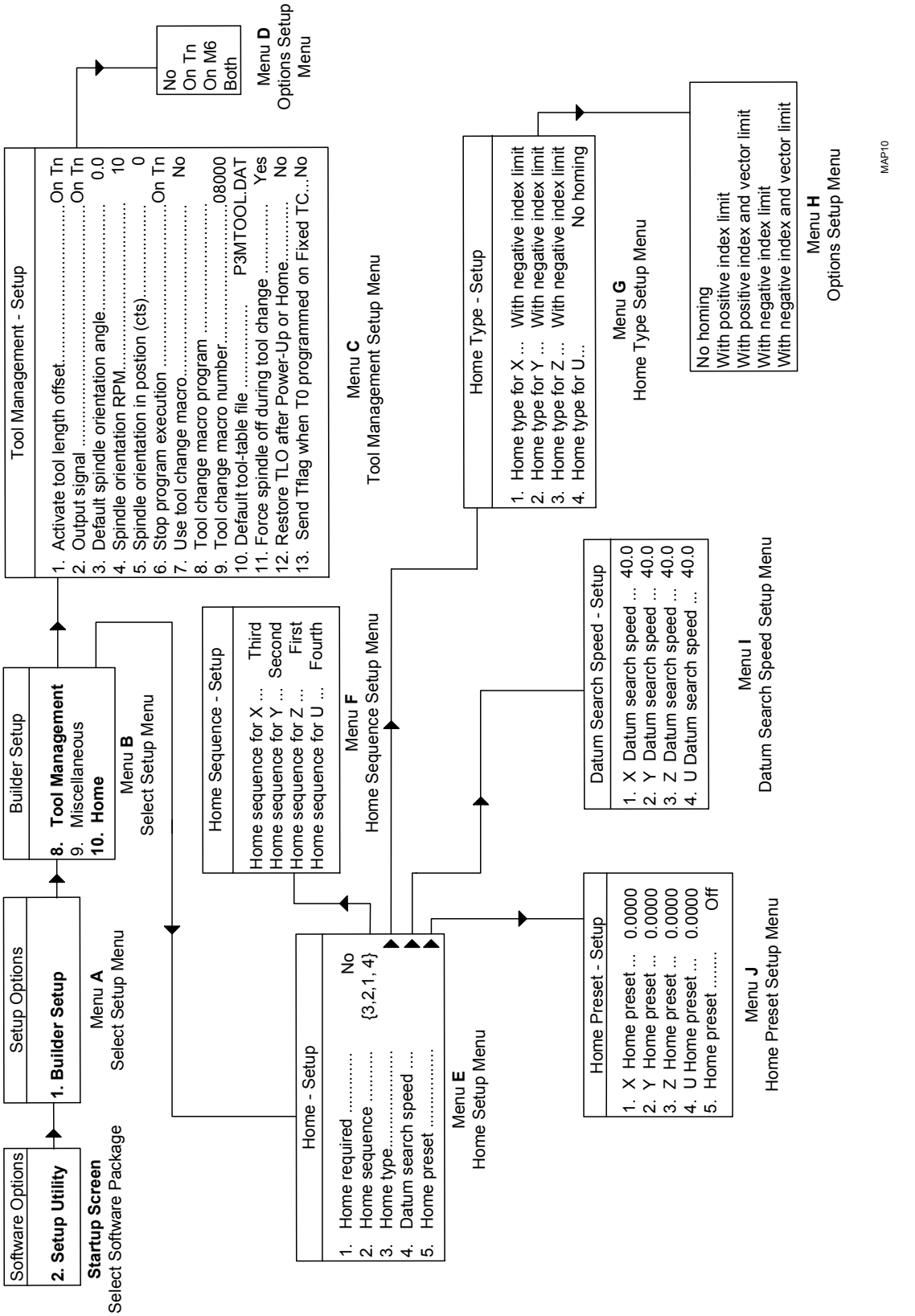
Map 7



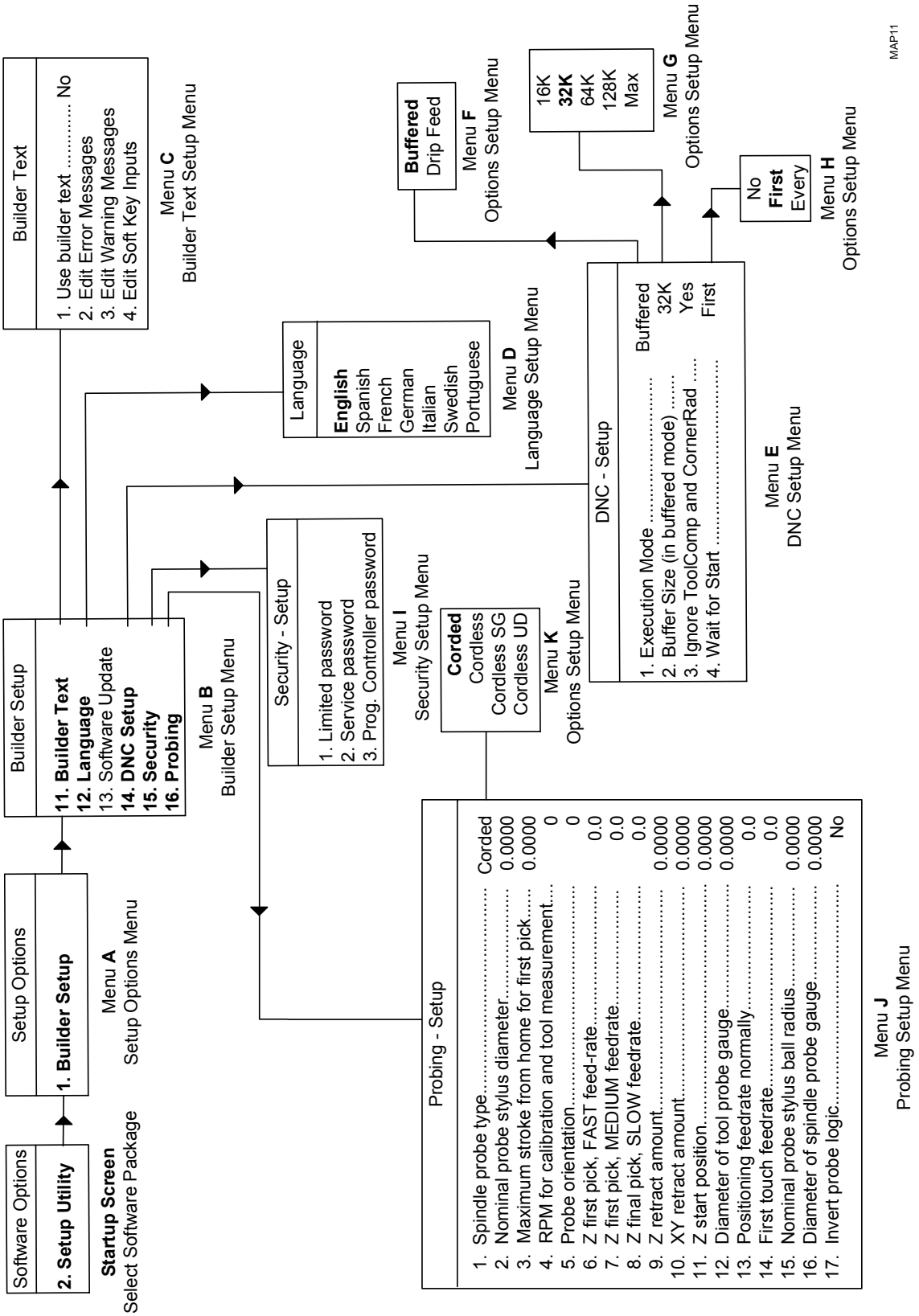
Map 8



Map 9

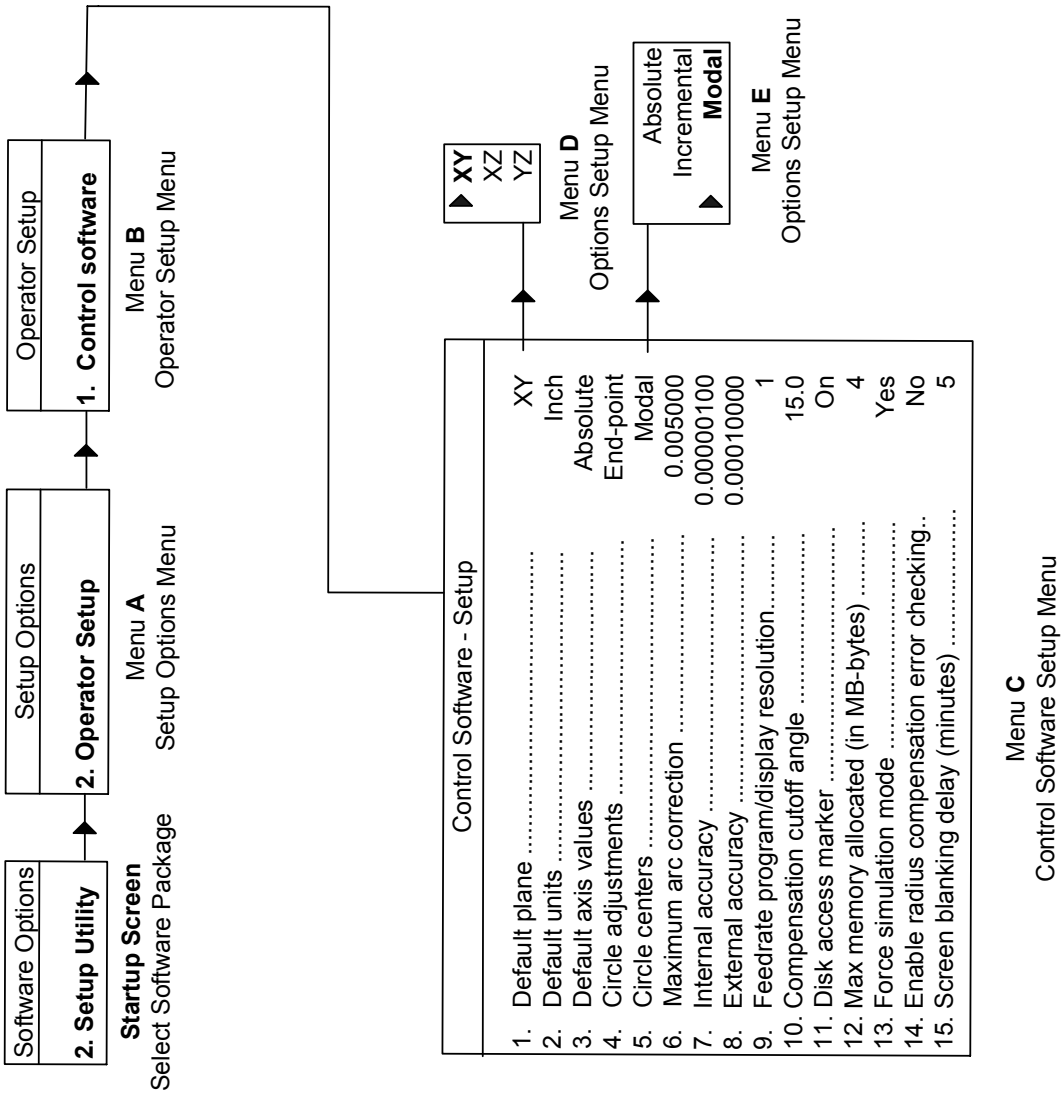


Map 10

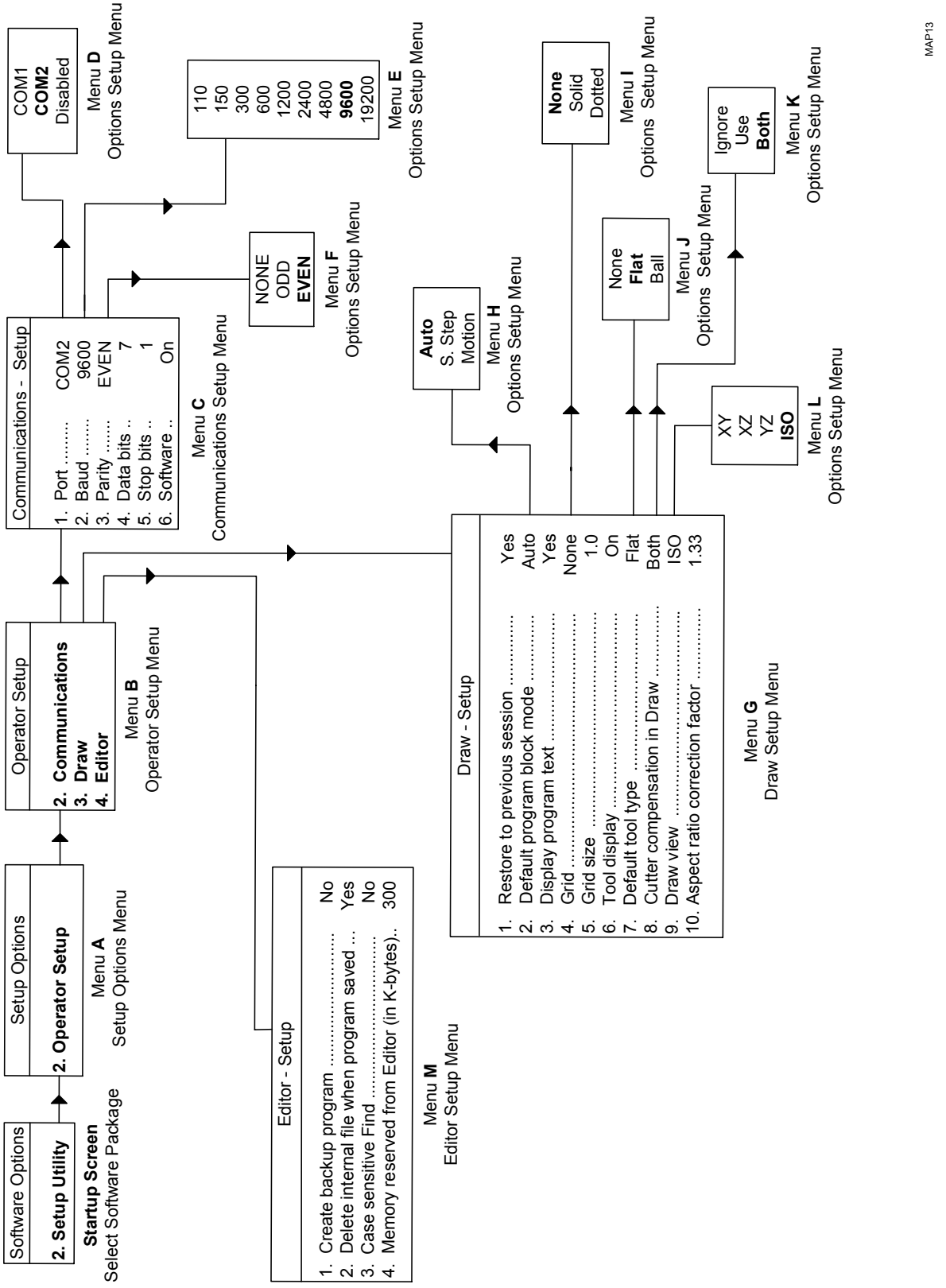


Map 11

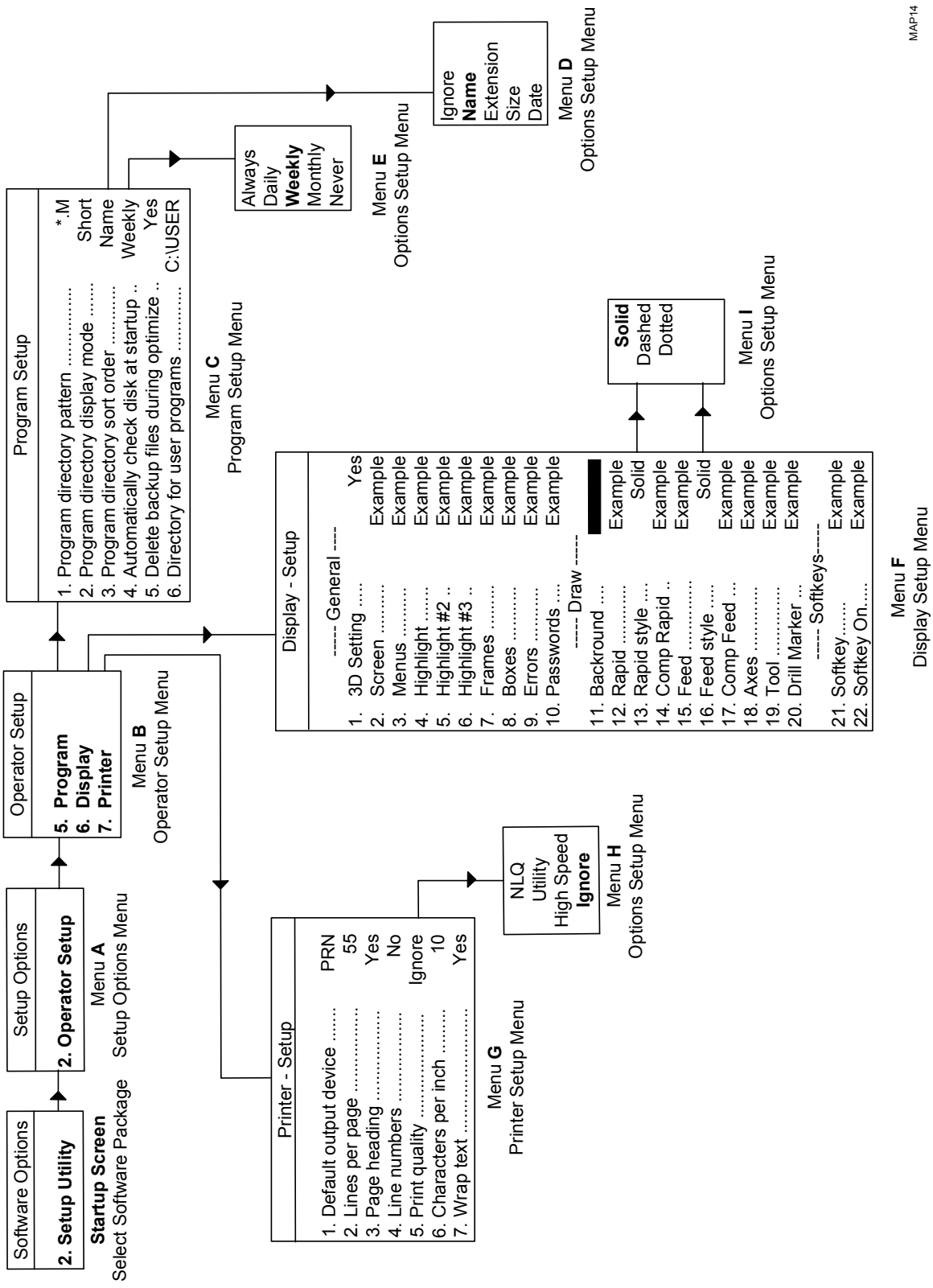
MAP12



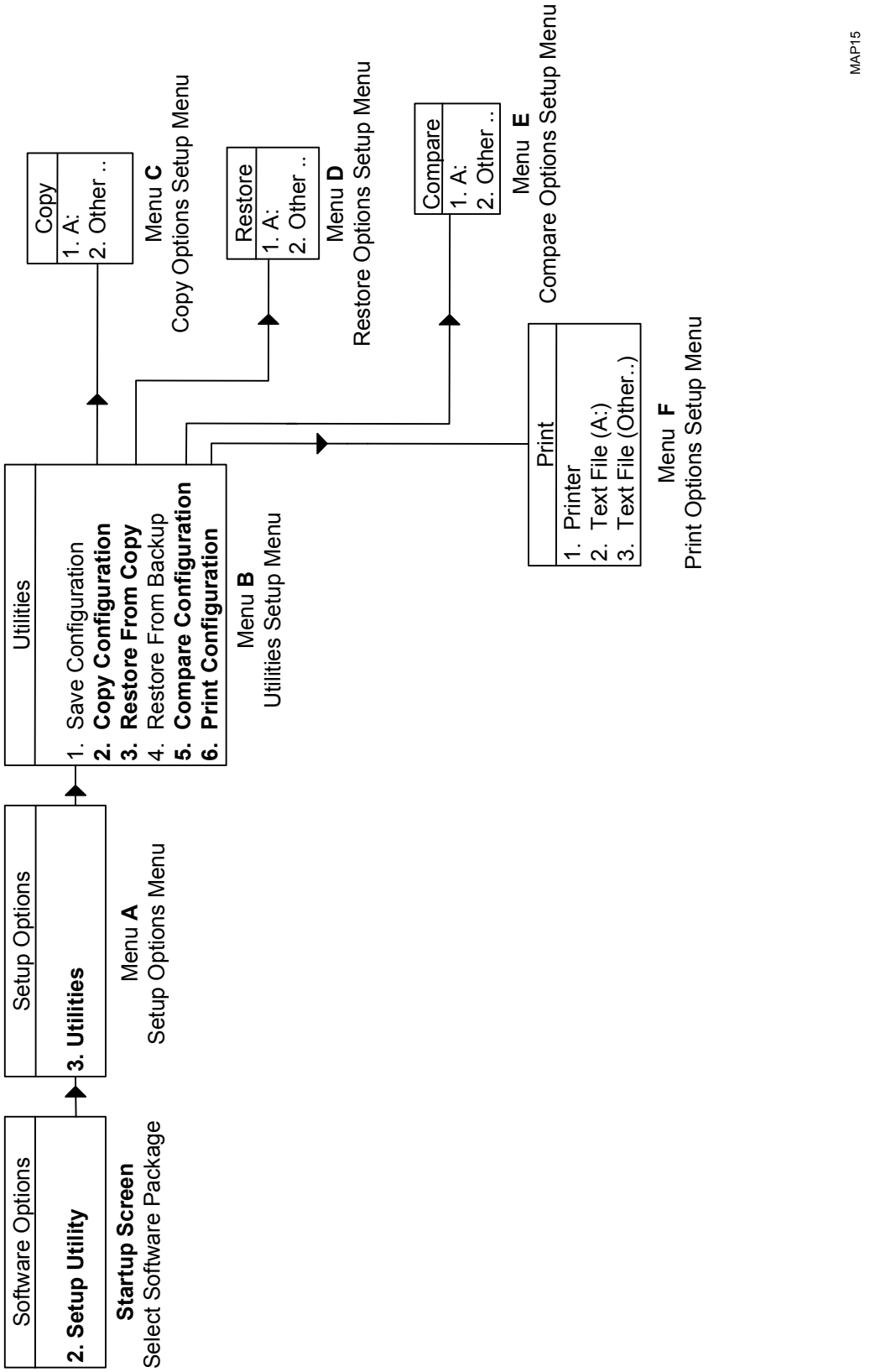
Map 12



Map 13



Map 14



Map 15

C

D

L

M

N

ANILAM

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

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

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

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