ANILAM

4200T
CNC Setup Utility Manual

www.anilam.com

Warranty

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

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

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

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

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

Section 1 - Setup Utility Concepts

Section 2 - Builder Setup

Section 3 - Operator Setup

Section 4 - Configuration Utilities

Section 5 - Fine-Tuning Systems with Linear Encoders

Section 6 - Setup Utility Maps

Section 1 - Setup Utility Concepts

Introduction

The 4200T Setup Utility is used to configure the CNC and optimize the system. The machine builder performs most of the initial machine setup at the time of the installation. This manual documents all 4200T parameters and the procedure to change them. All changes are made using the Setup Utility. The parameter settings are saved in a configuration file in the CNC's memory. The name of the configuration file for the 4200T CNC is P4TCFG.CFG.

Software Version Information

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

Navigating Through the Setup Utility

The Setup Utility provides access to parameter settings through menus and submenus.

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

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

Refer to "Section 6 - Setup Utility Maps" for all maps referenced in "Sections 2 through 4." Use these maps to locate parameter settings. The maps also serve as a quick reference guide.

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

Default Settings

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

Keypad Keys

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

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

Table 1-1, Keypad Keys

Name	Key Face
ARROWS	
CLEAR	
SHIFT	
SPACE	

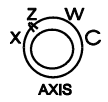
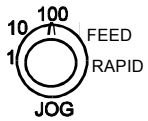
Axis Keys

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

Console Switches/Manual Panel Keys

Console switches and manual panel buttons are referred to as shown in **Table 1-2**.

Table 1-2, Console Switches/ Manual Panel Keys

Name	Switch/Button
Axis Selector Switch	
Jog Selector Switch	
Feedrate OVERRIDE Switch	
Spindle OVERRIDE Switch	
E-Stop Key	
Jog Plus Key	+
Jog Minus Key	–
Servo Reset Key	SERVO RESET
Start Key	
Hold Key	
Spindle Reverse Key	SPDL REV
Spindle Forward Key	SPDL FWD
Spindle Off Key	SPDL OFF

ENTER Key

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

Highlighting Menu Options

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

Exiting a Screen

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

Password Restricted Parameters

Some machine parameters are protected by passwords. The CNC provides four access levels of passwords. Operators are assigned limited access, which allows them to set parameters used in normal machine operations. Service and factory technicians require a higher level of access. The Programmable I/O Interface requires a separate password. See **Table 1-3** 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, enter a password when the CNC displays the password prompt.

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

Saving Changes to Setup Parameters

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

Select one of the following:

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

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

All configuration parameters are saved in a configuration file, (P5MCFG.CFG). Every time a parameter is changed, the configuration file is saved; the CNC automatically creates a backup file, (P5MCFG.BAK). The CNC provides utilities to manage the configuration file. Refer to "Section 4 - Configuration Utilities" for detailed information.

Setting Parameters in Setup Utility

To set parameters in the Setup Utility:

1. Highlight the menu in which the parameter is displayed, and press **ENTER**.

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

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

Accessing Setup Utility

To access the Setup Utility menus, do the following:

1. Turn on the CNC.

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

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

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

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

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

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

Overview of Main Parameter Categories

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

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

Units of Measurement

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

To set the default measurement mode, do the following:

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

All dimensional data will be displayed according to the units specified in this parameter. The only exceptions to this rule are dimensional parameters corresponding to the C-axes. The C-axis is used as a rotary axis. Thus, the units of all C-axis parameters are always in degrees or degrees per minute (i.e., deg/min).

Section 2 - Builder Setup

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

> **NOTE:** Some of the menus also provide a parameter for the C-axis. Only enter C-axis parameters, if the machine is capable of operating in C-axis mode (i.e., spindle used as a rotary axis).

System Resolution

The CNC uses feedback from either rotary encoders or linear encoders to provide the closed-loop positioning for the system. You can set resolution for the system (whether the feedback device is a rotary encoder or a linear encoder) via the **Resolution** menu.

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

Setting Axes for Rotary Encoder or Linear Encoder

Set each axis for the type (**Type**) of feedback device used, either rotary encoder or linear encoder. [Default: **Rotary Encoder**]

To set an axis as a rotary encoder or linear encoder:

1. Go to Map 1**, Menu G**. Highlight **Type**.
2. Press **X**, **Z**, or **W**.

   The CNC displays a pop-up window with **Rotary Encoder** or **Linear Encoder** as the selections.
3. Highlight either **Linear Encoder** for axes that use a linear encoder or **Rotary Encoder** for axes that use a rotary encoder. Press **ENTER**.

   The CNC changes the encoder type to the selected choice.

Setting the Display Resolution

Display Resolution allows you to set the resolution of the axis display (**Display Res**). The display resolution must be equal to or coarser than the actual resolution of the installed linear encoder or rotary encoder. Changing the Display Resolution does not affect the accuracy of the machine. Always select resolution in microns, whether the CNC is in Inch Mode or MM Mode. Ensure that resolution settings match the installed equipment.

[Default: **2 Microns** (0.002mm/.0001in.)]

To set display resolution:

1. Go to Map 1**, Menu G**. Highlight **Display Res**.
2. Press **X**, **Z**, or **W** (axis being set).

   The CNC displays a pop-up window with the choices: .5, 1, 2, 5, or 10 Microns.

3. Highlight the desired display resolution, and press **ENTER**.

   The axis display will show movement at the selected resolution.

Setting the Linear Encoder Resolution

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

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

Table 2-1, Micron to Inch Conversion

.5 Micron	.0005 mm	.00002”
1 Micron	.001mm	.00005”
2 Microns	.002mm	.0001”
5 Microns	.005mm	.0002”
10 Microns	.010mm	.0005”

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

To set display resolution:

1. Go to Map 1, **Menu G**.
2. Highlight **Linear Enc Res**.
3. Press **X**, **Z**, or **W** (axis being set), and press **ENTER**. A pop-up window will display the following choices: .5, 1, 2, 5, or 10-Micron.
4. Highlight the appropriate linear encoder resolution, and press **ENTER**.

Setting Line Count for Rotary Encoder

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

Type the number of counts per revolution supported by the rotary encoder (**Rot Enc Lines**). The CNC accepts line counts of up to 10,000 lines per revolution.
[Default X, Z: **1024** lines; W: **1000** lines]

To enter a rotary encoder line count:

1. Go to Map 1**, Menu G**. Highlight **Rot Enc Lines**.
2. Press **X**, **Z**, or **W**.

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

Setting Ballscrew Pitch for the Rotary Encoder

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

Pitch is linear distance traveled per revolution of the ballscrew.
Set the pitch (**Bscrew Pitch**) of the ballscrew. [Default: **0.20000** inch]

To enter ballscrew pitch:

1. Go to Map 1, **Menu G**. Highlight **Bscrew Pitch**.
2. Press **X**, **Z**, or **W**.

   The CNC highlights the ballscrew pitch entry field for the axis.
3. Type the pitch of the ballscrew for that axis, and press **ENTER**.

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

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

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

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

Ballscrew Pulley Parameter

[Defaults X, Z, W: **1.00000**]

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

To enter the Ballscrew Pulley value:

1. Go to Map 1**, Menu G**. Highlight **Ratio (Bsc Ply)**.
2. Press the appropriate axis key (i.e., **X**, **Z**, or **W**). The CNC highlights the ratio entry field for the axis.
3. Type the desired ratio for that axis, and press **ENTER**.

Motor/Encoder Pulley Parameter

[Defaults X, Z, W: **1.00000**]

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

To enter the Motor/Encoder Pulley value:

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

Setting Linear Correction Compensation

Linear correction compensation corrects for detected mechanical errors (in the ballscrew or elsewhere) that affect the indicated distance displayed by the CNC. To determine the amount of correction required, measure the error with a calibration device. When linear correction is activated, the CNC multiplies the commanded move by the compensation value.

If you do not require linear compensation, disable this feature. When enabled, you can specify a different correction value for each axis.

Correction = Distance Read by CNC ÷ Distance Actually Traveled

Enter any appropriate correction factor from 0.300000 to 3.000000.

1. Go to Map 1**, Menu E**. Highlight the menu selection that corresponds to the axis being set, and press **ENTER**.
   [Default: **Off** (Disabled)]

   The CNC highlights the entry field for the selected axis.

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

3. Highlight **Linear Correction compensation**, and press **ENTER**.

   This selection activates/deactivates the option.

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

Setting In-Position Check

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

When the CNC has positioned the tool within the in-position tolerance of the target, 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 X, Z, W: **0.0004** inch; C: **0.0102** inch]

When determining in-position tolerance:

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

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

To define in-position tolerance:

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

   The CNC highlights the entry field for the axis.

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

Setting Continuous Path

With Continuous Path Mode active, the CNC blends one move into another, without a complete stop between moves. The Continuous Path Mode activates at power On and is used for feed moves.

The CNC approaches the target position and comes within the continuous path tolerance of the target. Then, the CNC begins to calculate the next programmed move. It does not make an in-position check before it executes the next move. This results in a smoothly contoured profile or surface.
[Default: **0.0700** inch for all axes with **Continuous path** turned On; C: **1.7780**]

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

To activate and define the continuous path tolerance:

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

   The CNC highlights the entry field for the axis.

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

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

   This selection activates/deactivates the option.

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

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

Setting Default Rapid Rate

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

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

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

[Default: X, Z, U = **200.** in/min; C = **5080.** deg/min]

To set the default rapid speed:

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

   The CNC highlights the entry field for the axis.

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

Setting Axis Default Feed Rate

Setting the Default Feed Rate establishes a default feedrate for each axis, wherever a feedrate has not been programmed. This applies to programmed blocks or MDI commands. Jog moves in feed (i.e., from a manual panel) can have a different feedrate rate.

[Default: X, Z, W**= 10.**0 in/min; C**= 254.0** deg/min]

To set the axis default feed rate for an axis:

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

   The CNC highlights the entry field for the axis.

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

Setting Software Limits

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

You 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.

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

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

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

Setting Positive Software Limits

To set Positive Software Limits:

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

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

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

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

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

   The CNC highlights the entry field for the axis.

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

Setting Negative Software Limits

To set Negative Software Limits:

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

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

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

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

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

   The CNC highlights the entry field for the axis.

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

Enabling Software Limits

To enable Software Limits:

1. Go to Map 2**, Menu G**. Highlight **Software limits**, and press **ENTER** to toggle the Software Limits **On**. The CNC enables software limits.

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 X, Z, and W]

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

Vector Limit Switches restrictions follow:

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

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

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

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

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

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

Home Switches restrictions follow:

- Must be normally closed switches.
- Must be on CAN Node 0
- Are hard-coded (see **Table 2-2**)
- 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

To enable vector limits:

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

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

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

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(ed)** for X, Z, C axes; **Invert** for W axis]

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

To adjust the Encoder Phase Setting:

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

Setting Backlash Compensation

Backlash is the loss 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: **Off** (Disabled)]

To activate and define backlash compensation for an axis:

1. Go to Map 2**, Menu D**. Highlight the menu selection that corresponds to the axis being set, and press **ENTER**. The CNC highlights the entry field for the axis.
2. Type the desired backlash compensation, and press **ENTER**.
3. In Map 2**, Menu D**, highlight **Backlash compensation**.

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

   The CNC activates the backlash compensation for all selected axes.

Setting Ballscrew Compensation

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

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

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

To set the Ballscrew Compensation:

1. Go to Map 3**, Menu C**. Highlight **Ballscrew compensation**.
2. Press **ENTER**. The CNC displays Ballscrew Compensation Setup Menu (Map 3**, Menu D**).

Setting Number of Segments

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

To set the Number of Segments:

1. Highlight **Number of segments** in the Ballscrew Compensation Setup Menu, and press **ENTER**. The CNC displays Number of Segments Setup Menu (Map 3**, Menu E**).
2. Highlight the menu item pertaining to the axis being set, and press **ENTER**.

   The entry field for the selected axis highlights.
3. Type the number of segments desired for that axis, and press **ENTER**.

   Repeat the procedure for all axes being set.

Setting the Table Entries Parameter

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

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

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

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

To enter Table entries manually:

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

   The CNC highlights the entry field for the selected axis.
2. Type the desired compensation for each segment assigned to the axis, and press **ENTER**.

   The CNC accepts the entered values.

Setting Offset and Zero Cross Parameters

Both the Offset and Zero Cross parameters enable you to specify a starting point for ballscrew compensation. Both values are measured from Machine Home. These values include distance and direction (positive or negative) from Machine Home. The CNC adds the two values to determine the starting point. For example, if the assigned offset is -0.01mm 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. Go to Map 3**, Menu G**. Highlight the menu selection corresponding to the axis being set, and press **ENTER**.

   The CNC highlights the entry field for that axis.

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

Zero Cross

To set the Ballscrew Zero Cross parameter:

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

   The CNC highlights the entry field for that axis.

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

Setting Segment Length

In Ballscrew Compensation, the length of each lasered segment must be the same. The CNC counts off the segments from the beginning of the compensated area, as determined by the sum of the Offset and Zero Cross values assigned. The entered value should represent the segment length for each axis and the direction (positive or negative) of travel along the axis.

To set the Segment Length for Ballscrew Compensation:

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

   The CNC highlights the entry field for that axis.

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

Activating Ballscrew Compensation

To activate Ballscrew Compensation:

1. Go to Map 3**, Menu D**. Highlight **Active**.

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

Automatic File Loader

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

> **NOTE:** The File Loader does not change the way 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. Go to Map 3**, Menu F**. Press **LdFile** (**F8**). Type the appropriate password, if required by the system.

   The CNC displays the Leadscrew Compensation File Loader menu. See Figure 2-1, Ballscrew Compensation File Loader Menu. Refer to Table 2-3, Ballscrew File Loader Parameters for a description of the Ballscrew File Loader Parameters.

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

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

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

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

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

   The CNC returns to the Leadscrew Compensation File Loader menu.

6. Highlight **Action**, and press **ENTER**.

7. Highlight an option in the pop-up menu, and press **ENTER**.

8. Press **LdFile** (**F8**) again to load the file.

   A successful load shows the new entries in the table.

9. Repeat the procedure for the other axes.

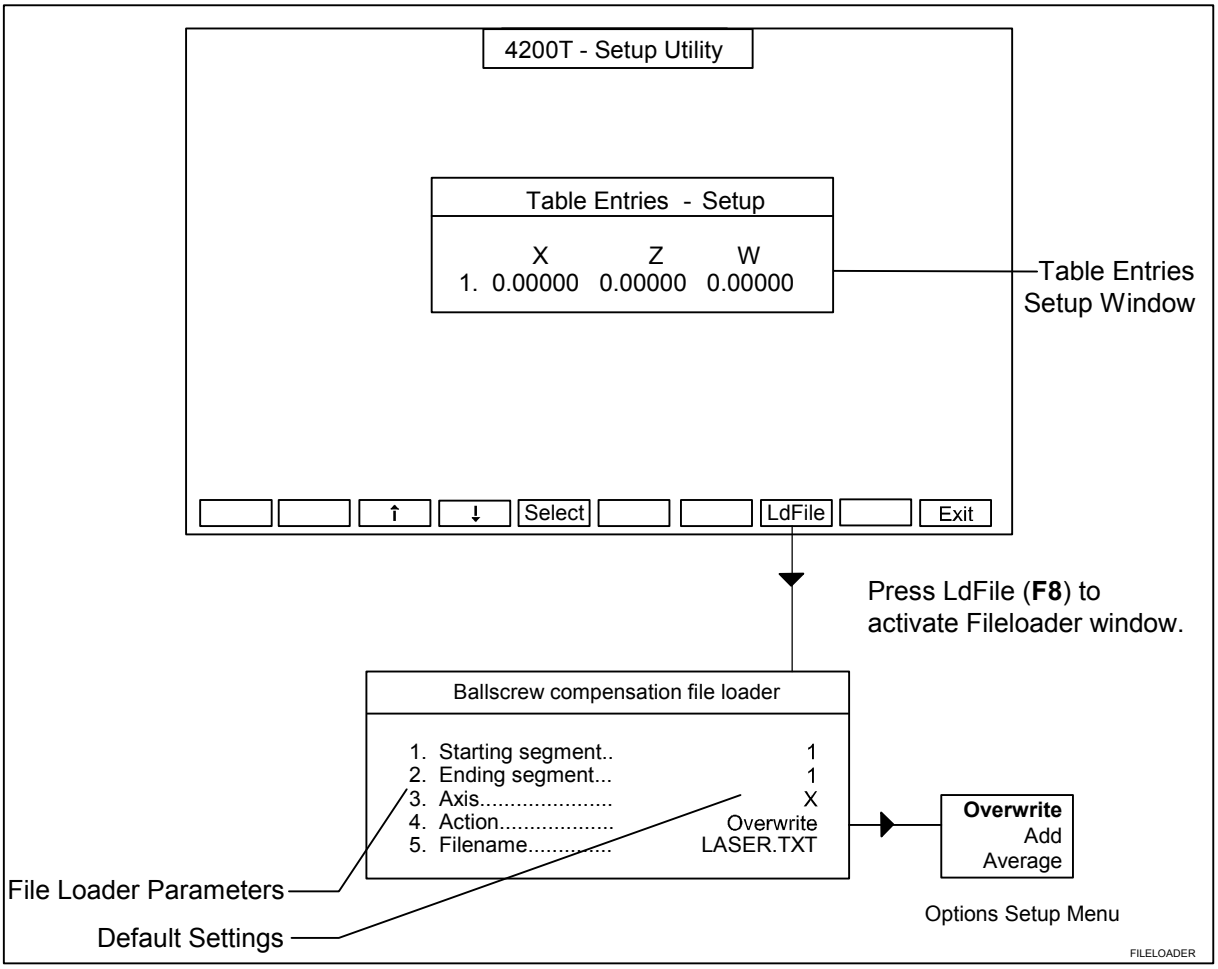


Figure 2-1, Ballscrew Compensation File Loader Menu

Table 2-3, Ballscrew File Loader Parameters

Parameter	Description
Starting segment	Determines which segment will be the first for data transfer. If a value greater than 128 (max. number of segments allowed) is entered, an **Error** message results. If a value greater than the Ending Segment value is entered, an **Error** message will be displayed.
Ending segment	Determines which row in the ballscrew compensation table will be the last to receive data from the laser file. If the segment limit on the table for the axis is exceeded, data will not be entered beyond the limit.
Axis	Determines to which axis data will be applied.

(Continued…)

Table 2-3, Ballscrew File Loader Parameters (Continued)

Parameter	Description
Action	Three types of actions during data load can occur: ❑ replacing the existing data in the table ❑ adding to the existing data ❑ be averaged with the existing data **Overwrite** will clear any values in the table beyond the segment limit for the axis. **Add** and **Average** only replace the old values. **Action** enables the user to fine-tune ballscrew compensation values from multiple passes of laser readings.
Filename:	Enter the DOS filename of the laser file, including the path, if different from the default.

Laser File Data File Format

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 that the laser measures.

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

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

Generating Ballscrew Compensation Values from Laser Files

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

- The segment length is 1 inch.
- The 0-inch value (commanded) from the laser data (measured) is approximately –0.00105 in.
- The 1-inch value (commanded) from the laser data (measured) is approximately –1.00202 in.
- The values are negative, indicating negative machine movement.

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

Method:

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

Example:

1. [(–1.00202) – (–0.00105)] – (–1) = –0.00097
2. –[–(–0.00097)] = –0.00097

This technique is used to find all ballscrew compensation table values. The File Loader automatically enters all compensation values into the Table Entries Setup Menu (Map 3, **Menu F**).

File Loader Error Messages

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

Set the segment limit (refer to Map 3**, 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

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

Table 2-4, Axis Ports

Port	Axis
0	X
1	C (Spindle)
2	Z
3	W

To change the **Axis Ports** Parameters:

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

Repeat this procedure for all axes.

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**

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 PID Filter, so that the following equation can define the output [digital word]:

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

Refer to **Table 2-5** for a detailed explanation of parameters.

Table 2-5, 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.

(Continued…)

Table 2-5, System Output Values and Definitions (Continued)

Value	Definition
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 P/N 70000634, *4200T CNC Motion Setup/Testing Utility* manual for documentation on using features that allow the CNC to automatically determine filter parameters.

To change the **Feed Filter** Parameters:

1. Go to Map 5**, Feed Filter Parameters, Menu E**.
2. Highlight the PID parameter being set (**Kp**, **Ki**, **Kd**, **Kf**, **Il**, or **Ds**).
3. Press **X**, **Z**, or **C** 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 Axis X, Z, W, and C. **Ki** default is **0.000**. **Kd** default is **10.000**. **Kf** default is **0.000**. **Il** default is **0.000**, and **Ds** default is **5.]**

To change the Rapid Filter Parameters:

1. Go to Map 5**, Rapid Filter Parameters Setup, Menu F**.
2. Highlight the PID parameter being set (**Kp**, **Ki**, **Kd**, **Kf**, **Il**, or **Ds**).
3. Press **X**, **Y**, or **C** 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. Go to 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**, or **Z** 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.**]

Setting Position Error Check Parameters

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 Map 5**, Menu H**. Refer to **Table 2-6** for definitions of these 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.

Table 2-6, Position Error Check Parameters

Position Error Check Parameter	Definition
Max idle time (ms)	The amount of time, in milliseconds, allowed between the internal command for a move and the input of counts from the feedback device, indicating motion. [Default: **100** msec]
Max lag error	The error distance allowed at rest or low feedrates, before declaring a fault. [Default: **0.0100**]
Check rapidrate	Ensures that machine is reaching its full-programmed rapid rate; if it does not, an Error Message displays. [Default: **Yes**]
Check feedrate	Ensures that machine is reaching its full programmed feed rate; if it does not, an Error Message displays. [Default: **Yes**]
Enable error checking	Allows you to configure or disable PEC option for troubleshooting or comparison. [Default: **Yes**]

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

"ERROR: (AXIS) LAG OVER MAX!"

"ERROR: LOST (AXIS) FEEDBACK!"

To change a PEC parameter:

1. Go to Map 5**, Menu H**. Highlight the PEC parameter to be changed, and press **ENTER**. The corresponding entry field for that parameter highlights.
2. Type the desired value in the entry field, and press **ENTER**.
3. For the Enable Error Checking parameter, press **ENTER** to toggle between **Yes** and **No**. [Default: **Yes**]

   The CNC activates the selected choice.

Setting Amplifier Tuning Rapids

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

To change an Amplifier Tuning Rapids parameter:

1. See Map 5**, Amplifier Tuning Rapids Setup, Menu I**. 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 *4200T CNC Motion Setup/Testing Utility*, P/N 70000634.

To set the digital amplifier:

1. See Map 7**, Menu D**. Refer to **Table 2-7** 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 2-7, 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, Z, W - Invert DAC Output **No**]

Spindle Axis Setup Menu

Refer to Map 8**, Menu C**. Use these parameters to configure spindle settings and gear ranges.

Setting Spindle DC Output

NOTE: Set the spindle drive to accept DC voltage.

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

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

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

The system outputs a negative DC voltage for Spindle Reverse (**M4**) commands and a positive DC voltage for Spindle Forward (**M3**) commands. The DC voltage is linear with respect to the RPM of the spindle speed command. Consequently, required voltage (0 VDC to ±10 VDC) increases as spindle speed increases (in reverse or forward directions). The maximum voltage, ±10 VDC, is output at the highest RPM value of the gear range.

1. Go to Map 8**, Menu C**.
2. Highlight **Spindle Output**, and press **ENTER** to toggle the setting to either **Bipolar** or **Unipolar**. [Default: **Unipolar**]

Setting Spindle Gear Ranges

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

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

You can use the Setup Utility to set either one gear range or up to four separate gear ranges. 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 set only one gear range, a programmed gear range is not required during spindle operation. For example, a command to activate a DC spindle drive at 1500 RPM reverse direction would be programmed as **S1500 M04**.

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

Gear range and speed	M42 S1500
Direction	M03

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

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

Defaults are as follows:

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

To set the spindle gear range:

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

NOTE: If you select **Multiple**, 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 **Multiple**. [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**]
9. Continue with M42, M43, and M44 setup menu options.

M40 – M44 Ratio (Spindle Pulley)

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

M40 – M44 Ratio (Motor Pulley)

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

Setting Gear Change RPM

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

To set the Gear Change RPM:

1. Go to Map 8**, Menu C**. Highlight **Gear change RPM**, and press **ENTER**.

The entry field for the parameter highlights.

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

Setting the Spindle Encoder Lines

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

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

To set the number of spindle encoder lines:

1. Go to Map 8**, 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

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

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

To set the Spindle RPM Display:

1. Go to Map 8**, Menu C**.
2. Highlight **Spindle RPM display**, and press **ENTER** to toggle the setting to either **Feedback** or **Program**.
3. Switch to **Feedback** to configure the spindle RPM display for rotary encoder feedback.

   – or –

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

Checking Spindle During Gear Change

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

To check the spindle during a gear change:

1. Go to Map 8**, Menu C**.
2. Highlight **Check spindle during gear change**, and press **ENTER** to toggle the setting On (**Yes**) or Off (**No**).

If this parameter is set to **Yes** and a gear change code is executed with the spindle running, the CNC will generate an error message. If parameter is set to **No**, a gear change code may be executed with the spindle running.

Stopping Program on Gear Change

Stopping the program on gear change allows you to configure the CNC to stop program execution during a gear change. If you specify **Yes**, the CNC checks for spindle movement (RPM). If there is spindle movement, the CNC displays an error message and stops the machine. If you specify **No**, the CNC does not stop the machine for a gear change. [Default: **No**]

To stop the program during a gear change:

1. Go to Map 8**, Menu C**.
2. Highlight **Stop program on gear change**, and press **ENTER** to toggle the setting to On (**Yes**) or Off (**No**).

Checking RPM to be Within Gear Range

Checking RPM to be Within Gear Range prevents you from designating an RPM outside the active gear range. If you specify **Yes**, the CNC checks the RPM. If the entered RPM is not within the active gear range, the CNC displays an error message. If you specify **No**, the CNC does not check the entered RPM against the active gear range. [Default: **Yes**]

To check the RPM gear range:

1. Go to Map 8**, Menu C**.
2. Highlight **Check RPM to be within gear range**, and press **ENTER** to toggle the setting to On (**Yes**) or Off (**No**).

Setting Spindle Drive Stop/Start When You Press Hold/Start

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 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.

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.)

Setting Encoder Mounted on Motor

Set to **Yes** when the spindle encoder 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 Basic I/O Interface

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

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

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

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

Setting the I/O Interface Type

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

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

To activate the I/O interface:

1. Go to Map 8**, Menu D**.
2. Highlight **Type**. This setting activates the CAN I/O, allowing the exchange of input and output signals between the control and the distribution board.
3. Press **ENTER** to display a pop-up menu with the following selections:

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

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

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

4. Highlight one selection, and press **ENTER**.

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.

Output Function Setup

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

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

To set an output port:

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

   A port can be used by more than one function.

Configuring Output Ports

Default port settings provide a +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 and another deactivates it (latched output). Sometimes, the CNC emits an output signal for a user-specified duration (pulsed output). The default port settings open and close relays that operate devices on the machine. Figure 2-2, Typical Inactive Port (Default Port Settings) illustrates typical Inactive port settings and Figure 2-3, Typical Active Port (Default Port Settings) illustrates typical active port settings.

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

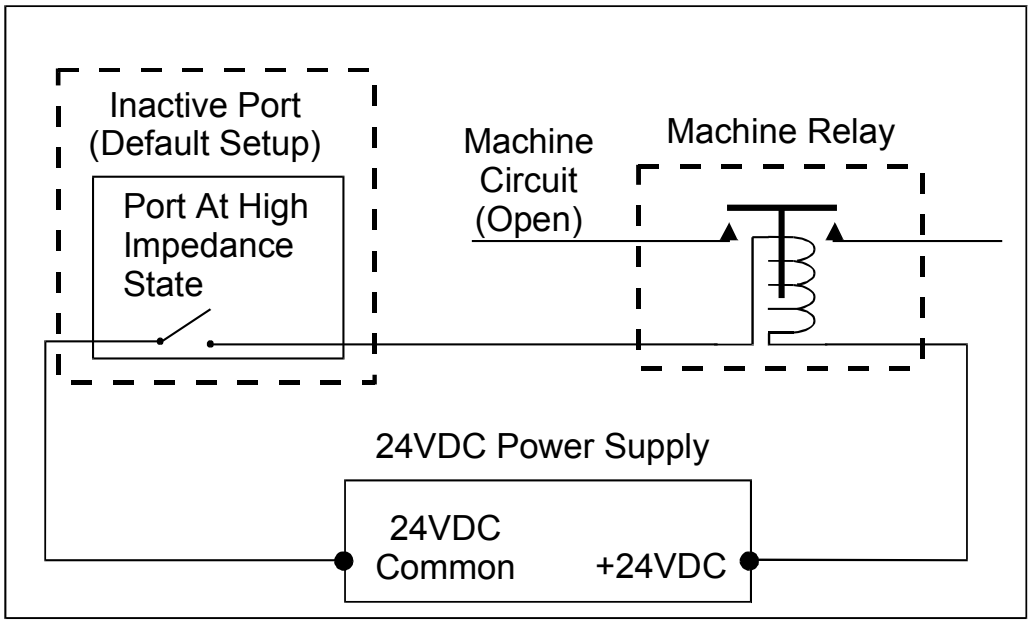


Figure 2-2, Typical Inactive Port (Default Port Settings)

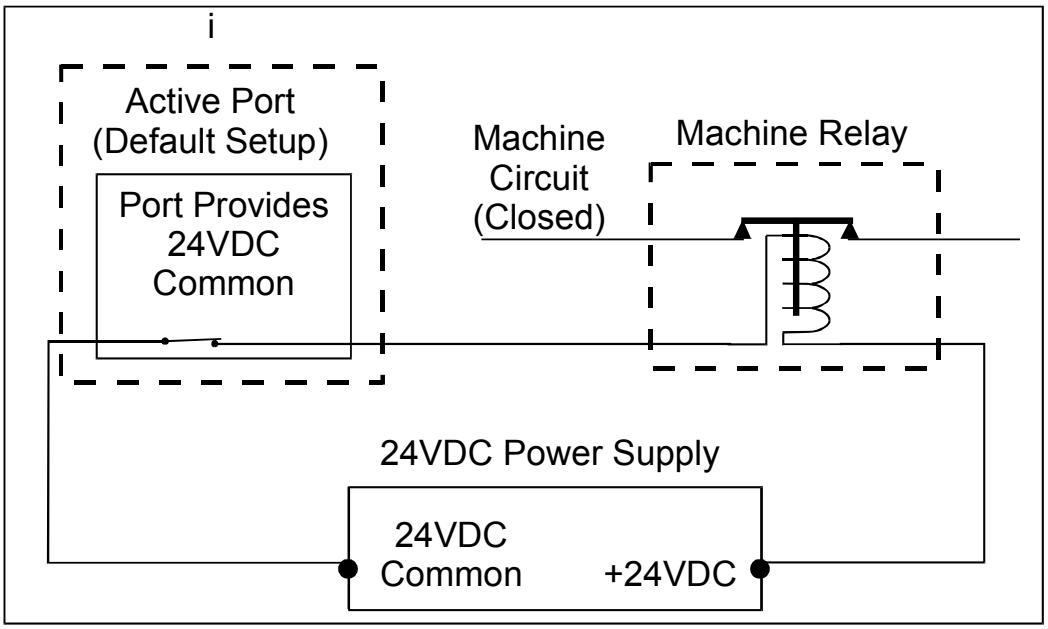


Figure 2-3, Typical Active Port (Default Port Settings)

Setting Finish Pulse Timeout

When an output port configured for a finish pulse is activated, the CNC is put on hold until an external finish pulse is received. While on hold, the CNC will not run program blocks (in Programmed Mode) and will not respond to keypad inputs (in Manual Mode).

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

The finish pulse is a signal from a machine device. It informs the CNC that the requested operation is completed.

If no finish pulse is received by the end of the timeout period, the CNC will display an **Error** message.

To set the Timeout:

1. Go to Map 9**, Menu C**. Highlight **Timeout**, and press **ENTER**.

   The **Timeout** entry field highlights.

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

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

Setting Up DSP^2 Node

To set up the DSP^2 Node:

1. Go to Map 8**, Menu H**. Highlight **DSP2 Node**, and press **ENTER**.

   The CNC displays **Menu I**.

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

   The CNC displays **Menu K**.

3. Highlight **Input 0**, **Input 1**, or **Input 2**. Press **ENTER**.

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

4. To assign bit functions in **Menu I**, press **ENTER**.

   The CNC displays **Menu J**.

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

Specifying Analog Input Use

Analog input may be measured by gauge or may be used in conjunction with a device such as a potentiometer.

To specify analog use:

1. Refer to Map 9**, Menu D**. Highlight **I/O Nodes**.

   The CNC displays **Menu H**.

2. Highlight desired **CAN Node** (**0-5**), and press **ENTER**.
3. Highlight **Analog input use**.
4. A pop-up menu displays. Highlight either **Gauge** or **None**. (**None** indicates a reserve for future features, or other controls to be added, e.g., potentiometer). Press **ENTER**. [Default: **None**]

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

Issuing Spindle Stop on Servo Fault or Emergency Stop

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

To issue a spindle stop:

1. Refer to Map 8, **Menu D**. Highlight **Issue SpStop on SvoFlt or Estop**.
2. Press **ENTER** to toggle the setting to either **No** (Off) or **Yes** (On). [Default: **Yes**]

Displaying Internal Interface Messages

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

1. Refer to Map 8, **Menu D**. Highlight **Display internal interface messages**.
2. Press **ENTER** to toggle the setting ON (**Yes**) or Off (**No**). [Default: **Yes**]

Specifying Gauges

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

NOTE: Before specifying a gauge, ensure that you have specified the Node Type for the CAN I/O as Digital/Analog. This will enable the gauge to be activated. See Map 9**, Menu E**.

To specify a gauge(s):

1. Refer to Map 8, **Menu D**. Highlight **Gauges**.

   A pop-up menu is displayed listing the options: **Gauge #1**, **Gauge #2**, and **Gauge #3**.

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

   A pop-up menu is displayed listing options **Active**, **Name**, and **Input node**.

3. Highlight **Active**. Press **ENTER** to toggle the selection On/Off to activate/deactivate the gauge. [Default: **No**]

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

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

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

Assigning M-Functions to Outputs

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

NOTE: CAN Node 0 and CAN Nodes 1 through 5 operate identically. To assign M-functions to a CAN Node's output ports, go to Map 9. The **#** symbol indicates any CAN Node, as determined in **Menu E**.

To configure M-functions:

1. Go to Map 9**, Menu D**. Highlight a **CAN Node** (**0-5**).

   The CNC displays **Menu E**.

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

   The CNC displays **Menu I**.

3. Highlight the M-function to be configured, and press **ENTER**.

   The CNC displays **Menu J**.

4. Highlight **Unused**, **Bit On**, or **Bit Off**. Or, press **ATTR** (**F7**).

   The CNC displays **Menu K**.

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

   The CNC displays **Menu M**.

Use **Menu M** to configure the following parameters:

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

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

To configure a particular bit:

6. Refer to **Menu I**. Press **OUTPUT** (**F1**).

   The CNC displays **Menu L**.

7. Highlight an output from **0** to **5**, and press **ENTER**.

   The CNC returns to **Menu I**, ready for you to assign M-functions to each bit.

NOTE: The CNC changes the Heading in **Menu I** 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 CNC displays the heading on **Menu I** as **M-Functions** (Output 4, Node #) **- Setup**.

Assigning Input Functions to an Input

NOTE: All CAN Nodes (0 through 5) operate identically. To assign input functions to a CAN Node's input ports, go to Map 9. The **#** symbol indicates CAN Node **0** through CAN Node **5**, as determined in **Menu D**.

To configure CAN Input functions:

1. Go to Map 9**, Menu E, CAN Node # - Setup**. Highlight **Inputs**.
2. Press **ENTER**.

   The CNC displays **Menu F, CAN Input # Functions (Node #) – Setup**.
3. Highlight one of the available functions, and press **ENTER**.

   The CNC displays **Menu H, Option Setup Menu**.
4. Highlight **Off**, **Active Low**, or **Active High** for each function.

To configure each bit:

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

1. Go to Map 9, **Menu F**.
2. Press **INPUT** (**F1**).

   The CNC displays Menu G.
3. Highlight an input from Input 0 through Input 9. [Default: **0**]. Press **ENTER**.

   The CNC returns to **Menu F** where you can assign an input function to each bit.

CNC Input Functions

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

Table 2-8, CNC Input Functions

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

Table 2-8, CNC Input Functions (Continued)

Input Function	Definition
Spindle CW (M3)	When activated, CNC initiates an **M3** output. Duplicates spindle **CW** function.
Spindle CCW (M4)	When activated, CNC initiates an **M4** output. Duplicates spindle **CCW** function.
Spindle Off (M5)	When activated, CNC initiates an **M5** output. Duplicates spindle **OFF** function.
General Error Input message	Stops the program and generates an Error message. You must correct the error condition before you can restart the program.
General Warning input message	Generates a Warning message at the CNC. The program continues to run.
Remote Jog +	Allows you to make positive jog moves from a remote manual panel.
Remote Jog -	Allows you to make negative jog moves from a remote manual panel.
Remote Resolution Selector	Allows you to select the axis resolution factor from a remote manual panel. Three input bits are used with each bit corresponding to x1, x10, and x100 resolution. The three input bits must be sequential.
Remote Axis Selector	Allows you to select an axis from a remote manual panel. Three input bits are used with each bit corresponding to X, Z, and C-axis. The three input bits must be sequential.
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**.

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 Setup

Refer to **Table 2-9** for information on the settings used to configure each Handwheel.

Table 2-9, DSP2 - Handwheel Settings

Setting	Setting Description/Function
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).
Resolution	Selects the axis resolution for the handwheel attached to the port. Set the resolution to MP Switch to select a resolution from the jog selector on the manual panel.
Axis	Selects the axis controlled by the handwheel on this port Set the axis to MP Switch to select an axis from the rotary switch on the manual panel or assign the handwheel to a dedicated axis.
Scaling Factor	Changes the sensitivity of the handwheel. A higher number will make the axis run faster. A lower number will make the axis run slower.

To configure Handwheel #1or #2:

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

1. Go to Map 4**, Menu B**. Highlight **Handwheel** in the **Builder Setup** menu.
2. Press **ENTER** to display the Handwheel Setup menu, Map 4**, Menu F**.
3. Highlight **Handwheel #1** or **Handwheel #2**. Press **ENTER**.

   The **Handwheel #1** (or **#2**) **Setup** menu is displayed.
4. Highlight **Type**, and press **ENTER**.

   A pop-up menu is displayed, listing **None** and **Handwheel**. [Default: **Handwheel**]
6. Highlight **Handwheel**, and press **ENTER**.
7. Highlight **Phase**.
8. Press **ENTER** to toggle to the desired phase setting, **Invert** or **Not Invert**. [Default: **Not Invert**]

9. Continuing in the **Handwheel #1** (or **#2**) **Setup** menu, highlight **Handwheel**. Press **ENTER**.

   The **Handwheel #1 – Setup** menu is displayed listing **Resolution** and **Axis**.

10. Highlight **Resolution**, and press **ENTER**.

11. In the pop-up menu that is displayed, highlight one of the options: **MP Switch**, **Fixed 1**, **Fixed 10**, or **Fixed 100**. [Default: **MP Switch**] Press **ENTER**.

12. Continuing in the **Handwheel #1 Setup** menu, highlight **Axis**, and press **ENTER**.

13. In the pop-up menu that is displayed, highlight one of the options: **MP Switch**, **Fixed X**, or **Fixed Z**. [Default: **Fixed X**]. Press **ENTER**.

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

    The entry field for the parameter highlights.

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

Tool Management Setup

Activation Options

Tool Setup parameters require you to specify a type of activation. Refer to **Table 2-10** for the available tool setup settings.

Table 2-10, Tool Setup Settings

Activation 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 M-function (M6) is activated.
Both	Function activates when a tool number or M6 is activated.

Manual Tool Change Operation

The CNC system supports manual tool change operations only.

Activating Tool Length Offset

This function is used to preset each tool to the same position on the component.

To activate the tool length offset:

1. Go to Map 10**, Menu C**. Highlight **Activate tool length offset**, and press **ENTER**.

   A pop-up menu is displayed listing the options: **No**, **On Tn**, **On M6**, and **Both**.

2. Select the desired setting, and press **ENTER**.
   [Default: **On Tn**]

Enabling Output Signal

Output signal refers to T-code data being sent to the Programmable Controller (IPI). Select **On Tn** to enable the output signal when the T-code activates.

To enable the output signal:

1. Go to Map 9**, Menu C**. Highlight **Output Signal**, and press **ENTER**.

   A pop-up menu is displayed listing the options: **No**, **On Tn**, **On M6**, and **Both**.

2. Select the desired setting, and press **ENTER**. [Default: **No**]

Stopping Program Execution

Stopping program execution halts the running program until the operator presses **START**. For manual tool change operations using IPI, set this selection to **No** (disabled). For manual tool change operations without IPI, set this selection to **On Tn**. The CNC holds program run and displays a message. Press **START** to resume program run.

To stop program execution:

1. Go to Map 9**, Menu C**. Highlight **Stop program execution**, and press **ENTER**.

   A pop-up menu is displayed listing the options: **No**, **On Tn**, **On M6**, and **Both**.

2. Select the desired setting, normally either **No** or **On Tn**. Press **ENTER**.
   [Default: **No**]

Setting Number of Digits in T-Word

This specifies the format of T-words (Txxxx). For additional information, refer to "Guidelines for Setting the Number of Digits in T-Word" located at the end of the "Tool Management Setup" section.

To specify the number of digits in T-word:

1. Go to Map 9, **Menu C**. Highlight **Number of digits in T Word**, and press **ENTER**.
2. Press **ENTER** to toggle the setting to either **6** or **4**.
   [Default: **4**]

Setting Wear Offset Adjustment

Use this selection to increment or decrement an axis for adjustment purposes. This can only be used when you are in Single-Step or Auto mode (program run).

To set the wear adjustment:

1. Go to Map 9, **Menu C**. Highlight **Wear offset adjustment**, and press **ENTER**.

   The CNC highlights the entry field for wear offset adjustment.
2. Type the desired wear offset adjustment value.
   [Default: **0.0001**]

Setting Maximum Wear Offset

To set the maximum wear adjustment:

1. Go to Map 9, **Menu C**. Highlight **Maximum Wear Offset**, and press **ENTER**.

   The CNC highlights the entry field for maximum wear offset.
2. Type the desired maximum wear offset value.
   [Default: **0.0984**]

Setting the Default Tool Table

To enter the default tool-table file:

1. Go to Map 9, **Menu C**. Highlight **Default tool-table file**.
2. Press **ENTER**.

   The **Default Tool-Table File** entry field highlights.
3. Type the file name.

   [Default: **P4TTOOL.DAT**]

Setting Force Spindle Off During Tool Change

Setting this parameter to prevent removal of the tool while the spindle is turning.

To set the Force Spindle Off During Tool Change

1. Go to Map 9**, Menu C**. Highlight **Force spindle off during tool change**, and press **ENTER**.
2. Press **ENTER** to toggle the setting to either **No** or **Yes**. [Default: **Yes**]

Automatic Tool Change Operation

For automatic tool change operations, use the settings specified in the following sub-paragraphs. Program specialized Macro program modules to simplify the use of any tool changer. Select and edit these Macros using the Tool Management Menu. Use the **M6** command during automatic tool changer operations.

Activating Tool Length Offset

Use this selection to preset each tool to the same position on the component. This function activates upon an **M6** command.

To activate the tool length offset:

1. Go to Map 9**, Menu C**. Highlight **Activate tool length offset**, and press **ENTER**.

   A pop-up menu is displayed listing the options: **No**, **On Tn**, **On M6**, and **Both**.
2. Select **On M6,** unless specified otherwise by OEM. Press **ENTER**. [Default: **On Tn**]

Enabling Output Signal

"Signal" refers to T-code data being sent to the Programmable Controller via RS-232. This selection enables that output, upon completion of an M6 command to the Programmable Controller.

On M6, T-Code data is output by the CNC to the IPI until a Finish pulse is received. If no tool number is programmed on the same line as the M6 in the CNC program, then the currently active tool offset data remains unchanged.

To enable the output signal:

1. Go to Map 9**, Menu C**. Highlight **Output signal**, and press **ENTER**.

   A pop-up menu is displayed listing the options: **No**, **On Tn**, **On M6**, and **Both**.
2. Select **On M6**, unless specified otherwise by OEM. Press **ENTER**. [Default: **No**]

Stopping Program Execution

When **On Tn**, **M6**, or **Both** are entered, stop program execution halts program run until **START** is pressed. For automatic tool changer operations with a programmable controller, set to **No** (disabled).

To set stop program execution:

1. Go to Map 9**, Menu C**. Highlight **Stop program execution**, and press **ENTER**.

   A pop-up menu is displayed listing the options: **No**, **On Tn**, **On M6**, and **Both**.

2. Select **No,** unless specified otherwise by OEM. Press **ENTER**. [Default: **On Tn**]

Setting Use Tool Change Macro

Setting this parameter enables you to use the tool change macro. This function activates on **M6** command. For additional information, refer to "Guidelines for Setting Tool Change Macro Parameters" located at the end of the "Tool Management Setup" section.

To set use tool change macro:

1. Go to Map 9**, Menu C**. Highlight **Use tool change macro**, and press **ENTER**.

   A pop-up menu is displayed listing the options: **No**, **On Tn**, **On**, **M6**, and **Both**.

2. Select **M6,** unless specified otherwise by OEM. Press **ENTER**. [Default: **No**]

Setting Tool Change Macro Program

Use this selection to set the program name of the macro to be used during tool change operations. For additional information, refer to "Guidelines for Setting Tool Change Macro Parameters" located at the end of the "Tool Management Setup" section.

To specify the tool change macro program:

1. Go to Map 9**, Menu C**. Highlight **Tool change macro program**, and press **ENTER**.

   The CNC highlights the entry field for tool change macro program.

2. Type the macro program name.

Setting Tool Change Macro Number

Tool Change Macro Number is used during tool change operations. For additional information, refer to "Guidelines for Setting Tool Change Macro Parameters" located at the end of the "Tool Management Setup" section.

To specify the tool change macro number:

1. Go to Map 9, **Menu C**. Highlight **Tool change macro number**, and press **ENTER**.

   The CNC highlights the entry field for tool change macro number.

2. Type the macro number.
   [Default: **O8000**]

NOTE: The tool change macro program can contain more than one tool change macro. Use this parameter to specify the macro (within the tool change macro program) to be used during a tool change.

Setting Number of Digits in T Word

This specifies the format of T-words (Txxxx). A 4-digit word allows selection of 1 to 99 tools. A 6-digit word allows selection of 1 to 999 tools. For additional information, refer to "Guidelines for Setting the Number of Digits in T-Word" located at the end of "Tool Management Setup" section.

To specify the number of digits in T-word:

1. Go to Map 9, **Menu C**. Highlight **Number of digits in T Word**, and press **ENTER**.
2. Press **ENTER** to toggle the setting to either **6** or **4**.
   [Default: **4**]

Setting Number of Tools to Display in Table

Setting this parameter limits the number of tools displayed in the tool table.

To set the number of tools to display in the tool table:

1. Go to Map 9, **Menu C**. Highlight **Number of Tools to display in table**, and press **ENTER**.

   The CNC highlights the entry field for Number of Tools to Display in Table.

2. Type the maximum number of tools.
   [Default: **99**]

Setting Wear Offset Adjustment

Use this selection to increment or decrement a wear offset. This can only be used when you are in single-step or auto mode (program run).

To set the wear offset adjustment:

1. Go to Map 9, **Menu C**. Highlight **Wear offset adjustment**, and press **ENTER**.

   The CNC highlights the entry field for wear offset adjustment.

2. Type the desired wear offset adjustment value.
   [Default: **0.0001**]

Setting Maximum Wear Offset

To set the maximum wear adjustment:

1. Go to Map 9, **Menu C**. Highlight **Maximum Wear Offset**, and press **ENTER**.

   The CNC highlights the entry field for maximum wear offset.

2. Type the desired maximum wear offset value.
   [Default: **0.0984**]

Setting the Default Tool-Table File

To enter the default tool-table file:

1. Go to Map 9, **Menu C**. Highlight **Default tool-table file**, and press **ENTER**.

   The Default Tool-Table File entry field highlights.

2. Type the file name.
   [Default: **P4TTOOL.DAT**]

Setting Force Spindle Off During Tool Change

Setting this parameter to prevent removal of the tool while the spindle is turning.

To set the Force Spindle Off During Tool Change

1. Go to Map 9, **Menu C**. Highlight **Force spindle off during tool change**, and press **ENTER**.
2. Press **ENTER** to toggle the setting to either **Yes or No**. [Default: **Yes**]

Guidelines for Setting the Number of Digits for T-Words

Use the **Number of Digits in T-Word** parameter to configure the number of available T-words. The CNC software uses a four-digit value to enable four-digit tool codes.
Example: **T 03 04**.

In the example above, the first two digits, (**03**), specify the tool turret position of the tool to be used. The second two digits, **(T 04**), specify the tool offset being used.

If you specify a two-digit code, the offset number being used and the tool /turret being used will be the same; e.g., **T 04** specifies offset number **4** and tool pot number **4**.

A six-digit value enables you to use six-digit tool codes, e.g., **T 113 005**.

The first three digits, (**113**) specify the turret position of the tool to be used. The second three digits, (**T 005**), specify the tool offset being used.

If you specify a three-digit code, the offset number being used and the tool being used will be the same; e.g., **T 113** specifies offset number **113** and turret number **113**.

Guidelines for Setting Tool Change Macro Parameters

A tool change macro is a subprogram that prepares the machine axes and initiates necessary auxiliary functions prior to automatic tool-changer operation. The actual I/O operations would be handled by the IPI (or Programmable Controller). If no axes movements are necessary then it might be possible to handle the entire tool change operation from the IPI program.

The Setup Utility contains parameters to create, call and edit the tool change macro filename and macro number. To enable the tool change macro, set the **Use tool change macro** parameter to **No**, **On Tn**, **On M6**, or **Both**.

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

NOTE: The macro file is stored in the C:\P4T directory.

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

If the Tool Change Macro is created by an editing function other than **Edit** (**F8**), it must also be edited by the Setup Utility at least once.

Miscellaneous Setup Parameters

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

Setting Manual Panel Port

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

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

To set the manual port:

1. Go to Map 5**, Menu C**. Highlight **Manual panel port**, and press **ENTER**.

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

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

Setting Maximum Programmed Feedrate

The maximum programmed feedrate sets a limit on how fast the CNC allows the machine to travel in feedrate. [Default: **80.0 in/min**]

To set the maximum-programmed feedrate:

1. Go to Map 5**, Menu C**. Highlight **Max programmed feedrate**, and press **ENTER**.

2. Type the maximum-programmed feedrate in the highlighted entry field. Press **ENTER**.

> **NOTE:** You can override the maximum-programmed feedrate with the **FEEDRATE OVERRIDE** switch. The range of the switch is 0 to 120% of the maximum-programmed feedrate. The switch varies the feedrate in increments of 10%.

Setting Jog Feedrate/Rapidrate

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

[Defaults: Feedrate **40.0**; Rapidrate **200**]

To set up the Jog Feedrate and Rapidrate:

1. Go to Map 5**, Menu C**. Highlight **Linear axis jog feedrate** or **Linear axis jog rapidrate**, and press **ENTER**.
2. In the highlighted entry field, type the rate.
3. Press **ENTER**

> **NOTE:** The **FEEDRATE OVERRIDE** switch allows you to override the Jog Feedrate. The range of the switch is 0 to 120% of the maximum programmable feedrate; or 0 to 100% of the maximum programmed Rapidrate. The switch varies the feedrate in increments of 10%.

Setting Maximum Programmed C-axis Feedrate

The maximum programmed C-axis feedrate sets a limit on how fast the CNC allows the C-axis to travel in feedrate. The unit is degrees per minute. [Default: **3000** degrees per minute]

To set the maximum-programmed feedrate:

1. Go to Map 5**, Menu C**. Highlight **Max programmed C-Axis feedrate**, and press **ENTER**.
2. Type the maximum-programmed feedrate in the highlighted entry field, and press **ENTER**.

> **NOTE:** You can override the maximum-programmed feedrate with the **FEEDRATE OVERRIDE** switch. The range of the switch is 0 to 120% of the maximum-programmed feedrate. The switch varies the feedrate in increments of 10%.

Setting C-axis Jog Feedrate and Rapidrate

Set up the feedrate or rapidrate at which the C-axis (i.e., spindle used as a rotary axis) travels in Jog Mode. This defines the C-axis' jog speed. The unit is degrees per minute.
[Defaults: Jog feedrate **1016.0** degrees per minute, Jog rapidrate 3.000.0 degrees per minute]

To set up the C-axis default Jog Feedrate/Rapid rate:

1. Go to Map 5**, Menu C**. Highlight **C-axis jog feedrate** or **C-axis jog rapidrate**.
2. Press **ENTER** to activate the entry field. Type the feedrate or rapidrate, and press **ENTER** to activate it.

NOTE: The **FEEDRATE OVERRIDE** switch allows you to override the Jog Feedrate. The range of the switch is 0 to 120% of the maximum programmable feedrate; or 0 to 100% of the maximum programmed Rapidrate. The switch varies the feedrate in increments of 10%.

Setting C-axis Reset at 360

Set up the C-axis to force a reset (i.e., set to 0) when the axis reaches 360 degrees.
[Default: **No**]

To set up the C-axis reset at 360:

1. Go to Map 5**, Menu C**. Highlight **C-axis reset at 360**.
2. Press **ENTER** to toggle the setting to **Yes** to enable the parameter.

Servo Up Delay

You can program a delay to allow the servos to stabilize before the CNC commands a move.
[Default: **1** sec] Valid range: (0–10)

To program a servo delay:

1. Go to Map 5**, Menu C**. Highlight **Servo up delay**.
2. Press **ENTER**.
3. In the highlighted entry field, type the servo delay.

Reversing G2/G3 Commands

Reverse G2/G3 allows you to reverse the default settings for G2/G3 commands. This allows you to access program files from other offline sources without changing the program. The default settings of G2 = Circular Interpolation CW and G3 =Circular Interpolation CCW reverse to G2 = CCW and G3 = CW.

To reverse G2/G3 commands:

1. Go to Map 5, **Menu C**. Highlight **Reverse G2/G3**.
2. Press **ENTER** to toggle the setting to **Yes** to enable the parameter and reverse the G2/G3 commands.
   [Default: **No** (disabled)]

Setting Lathe Tool Post

Lathe Tool Post specifies whether front or rear tool post is being used. In general, this parameter affects program display Draw mode.
[Default: **Front**]

To select Front/Rear Tool Post:

1. Go to Map 5, **Menu C**. Highlight **Lathe tool post**, and press **ENTER** to toggle to **Rear** (Tool Post) or **Front**.

Setting Lathe X Programming Mode

Lathe X programming mode specifies whether the X-axis value is a radius or a diameter.
[Default: **Diameter**]

To specify Lathe X programming mode:

1. Go to Map 5, **Menu C**. Highlight **Lathe X programming mode**.
2. Press **ENTER** to toggle to **Radius** or **Diameter**.

Setting Feedrate Mode

Feedrate Mode allows you to set the feedrate mode to either **FPR** (G95, Feedrate per Revolution) or **FPM** (G94, Feedrate per Minute).
[Default: **FPR**]

To set the feed rate mode:

1. Go to Map 5, **Menu C**. Highlight **Feedrate mode**.
2. Press **ENTER** to toggle the setting to **FPM** (activates G94, Feedrate Per Minute) or **FPR** (activates G95, Feedrate Per Revolution).

Setting Dwell RPMs in FPR Mode

You can specify how the Dwell (G4) command will behave based on the feedrate mode of the CNC. The Dwell command can be made to Dwell according to revolutions of the spindle or time. Normally, a dwell command is timed.
[Default: **No** (dwell commands are timed)]

- For example, specifying **G4 T2,** the CNC dwells for two seconds.

When in FPR mode (G95), the dwell can be based on spindle revolutions. The CNC dwells for the specified number of revolutions.

- For example, specifying **G4 T200,** the CNC dwells for 200 revolutions of the spindle.

To switch to Dwell RPMS in FPR Mode:

1. Go to Map 5**, Menu C**. Highlight **Dwell RPMs in FPR mode**.
2. Press **ENTER** to toggle to **Yes**.

   If you select **No**, the CNC times the Dwell command regardless of FPM (G94) or FPR (G95) mode. If you select **Yes**, the CNC bases the Dwell commands on time in FPM mode or spindle revolutions in FPR mode.

Setting Default SCI in Manual

Simplified Command Interface (SCI) is used to access simplified command interface functions while at the Control Software level. When you select **Yes,** the SCI mode is displayed. When you select **No**, the SCI mode can still be activated by pressing **SHIFT** + **F4** in Manual mode.
[Default: **No**]

To switch to Default SCI in Manual:

1. Go to Map 5**, Menu C**. Highlight **Default SCI in Manual**.
2. Press **ENTER** to toggle to **Yes**.

Setting Rapid Moves are Free (Unsynchronized)

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

To set Rapid Moves Are Free:

1. Go to Map 5**, Menu C**. Highlight **Rapid moves are free (unsynchronized)**.
2. Press **ENTER** to toggle to **Yes** or **No**.

Setting 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. Go to Map 5**, Menu C**. Highlight either **Feed Accel/Decel** or **Rapid Accel/Decel**, as applicable.

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

Setting Check DSP^2 Integrity

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

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

To set Check DSP^2 Integrity:

1. Go to Map 5**, Menu C**. Highlight **Check DSP Inte**grity.
2. Press **ENTER** to toggle to **No** or **YES**.

Setting 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. Go to Map 5**, Menu C**. Highlight **Servo Loop Sample Time (ms)**.

   The CNC highlights an entry field.

2. Press **ENTER**.
3. Type the desired value.

Setting Interpolator Rate Factor

Interpolator Rate Factor allows you to specify the interpolator sample rate (in servo loop time).

[Default: **25**] Valid range: (5 to 27)

Interpolator Sample Rate = Servo Loop Sample Time x Interpolator Factor

To set Interpolator Rate Factor:

1. Go to Map 5**, Menu C**. Highlight **Interpolator rate factor**, and press **ENTER**.
2. Enter the desired value, and press **ENTER**.

Setting Enable Velocity Look Ahead

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

To set Enable Velocity Look:

1. Go to Map 5**, Menu C**. Highlight **Enable Velocity Look Ahead**.
2. Press **ENTER** to toggle to **No** or **YES**.

Setting Display Resolution

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

1. Go to Map 5**, Menu C**. Highlight **Display Resolution**.
2. Press **ENTER** to toggle to **SVGA** or **VGA**.

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

Setting CNC Startup Mode

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

To set the CNC startup mode:

1. See Map 6**, Menu C**.
2. Highlight **CNC Startup mode**, and press **ENTER** to toggle between **Sfwr Options** or **Ctrl Software**.

Sfwr Options	Software stops at the main menu after the introduction screen is displayed.
Ctrl Software	Software goes to the Control software section with stopping at the main menu.

Setting 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 C**.
2. Highlight **Show Introduction Screen**, and press **ENTER** to select **No** or **Yes**.

User Definable Variables

User definable variables are defined via parameters **#1130** – **#1139** and **#1120** – **#1129** in Miscellaneous Setup (see Map 6**, Menu C**, Miscellaneous Setup Menu). These parameters correspond directly to system variables #1130 through #1139. Parameters **#1130** – **#1134** are unit based; which means, these are assigned the units specified for machine parameters (Inch or MM). You can assign parameters **#1135** – **#1139** and **#1120** – **#1129** only number values.

A typical usage of these variables would be to define the tool-changer height in a tool-change macro. By using a user definable variable, the height of the tool-changer can be adjusted without editing the macro itself. See "Tool Changer Macro Example."
#1130 – #1139 [Default: **0.0000**]
#1120 – #1129 [Default: **0**]

To set the user definable variables:

1. See Map 6**, Menu C**.
2. Highlight **User definable variable #1130** – **#1139** or **#1120** – **#1129** and press **ENTER**.
3. Type the value for the variable and press **ENTER**.

Tool Changer Macro Example

Refer **Table 2-11**. This macro will stop the spindle before every tool change. Any other codes or moves would be specific to a particular application. Tool Change Macros always work in conjunction with the IPI program. It is important that the tool change setup parameters be set to **On M6** (see in "Tool Management Setup" tasks: "Activating Tool Length Offset," "Enabling Output Signal," and "Setting Tool Change Macro Program."

Table 2-11, Tool Change Macro Example

M2	* THIS COMMAND IS NOT OUTPUT TO
	* THE PROGRAMMABLE CONTROLLER
O 40000	* CREATES G8000
M5	* STOP SPINDLE
	* PLACE THE CODE OR MOVES HERE
	* AS MANY LINES AS YOU NEED
	*
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 C**.
2. Highlight **Mcode for macro call #1 – #10**, and press **ENTER** .
3. Type the value for the variable and press **ENTER**.

Macro Called for Mcode #1 – #10

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

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

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

Machine Home

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

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

Homing the Axes

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

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

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

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

Setting Home Required

Home Required specifies whether the CNC requires a Home Sequence prior to performing any other operations at system power up. Switch the option On (**Yes**) to enable the Homing Feature or Off (**No**) to disable the Homing Feature. [Default: **No**]

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

To set up the Home Required parameter:

1. Go to Map 10**, Menu E**. Highlight **Home required**, and press **ENTER** to toggle the setting to **No or Yes**.

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

Setting Home Sequence

[Default homing order: **1,2,3** corresponding to axes **X, Z,** and **W**]

To set the order in which axes are homed:

1. Go to Map 10**, Menu F**. Highlight **Home sequence**, and press **ENTER**.

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

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

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

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

Homing Feature Travel Direction

The Map 10**, Menu G** allows you to specify the types of homing for each axis. Use **Home type** to set the direction of travel for the Homing feature. [Default: X, Z: **With negative index limit**, W: **No Homing**] No Homing disables the homing function.

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

With Positive/Negative Index Limit

The CNC moves the selected axis in the positive/negative direction until it detects an index pulse from the linear encoder or rotary encoder. This method requires that the Machine Home position be known and physically marked on the axis, to ensure repeatability.

With Positive/Negative Index and Vector Limit

When you specify homing, the CNC travels in the specified positive/negative direction along the axis being homed until it trips the home switch. The CNC then reverses direction until it detects an Index pulse. The CNC sets Machine Home for that axis where it detects the +- Index pulse.

To set up the direction of travel for Homing:

1. Go to Map 10**, Menu E**. Highlight the **Home type**, and press **ENTER** to display **Menu G**.

2. Highlight the desired axis, and press **ENTER** to display **Menu H**.

3. Highlight the desired homing direction, and press **ENTER**.

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

Setting Datum Search Speed

[Default speed for X and Z-axes: **40** inches per minute,
C-axis: **1,016.0** inches per minute]

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

1. Go to Map 10**, Menu E**. Highlight **Datum search speed**, and press **ENTER**.

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

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

Setting Home Preset

You can automatically preset Machine Zero to any coordinates. When the machine reaches completes the homing sequence, the CNC sets the display to the preset values entered for all axes.
[Default: **Off** (disabled)]

To set up Preset values for axes:

1. Go to Map 10**, Menu J**. Highlight the selection corresponding to the axis being preset, and press **ENTER**.

2. Type the preset value in the highlighted entry field.

3. Repeat for all preset axes.

4. Highlight **Home Preset**, and press **ENTER** to toggle the selection **On** to enable it or **Off** to disable it.

Builder Text

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

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

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

Each message can be a maximum of 49 characters. Use Edit Error Messages to enter or edit Error messages. Refer to **Table 2-12** for message-code ranges and message types.

Table 2-12, Message Code Ranges and Types

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

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

Enabling Builder Text

To enable Builder Text:

1. Go to Map 11**, Menu C**. Highlight **Use builder text**.
2. Press **ENTER** to toggle the setting to **No** (disabled) or **Yes** (enabled)]. [Default: **Yes]**
3. Save the changes before you exit the Setup Utilities.

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

Editing Error Messages

To edit Error messages:

1. Go to Map 11**, Menu C**. Highlight **Edit Error Messages**., and press **ENTER**.

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

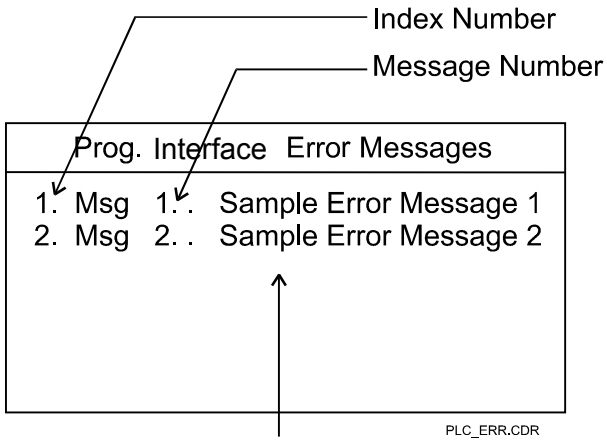


Figure 2-4, Builder Messages - Error Message Window

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

   The CNC activates the message text box.

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

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

Editing Warning Messages

To edit Warning messages:

1. Go to Map 11**, Menu C**. Highlight **Edit Warning Messages**, and press **ENTER**.

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

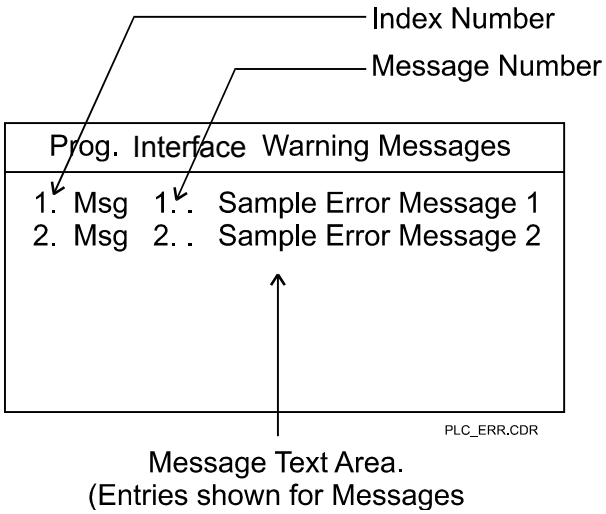


Figure 2-5, Builder Text - Warning Message Window

2. Highlight the message to be entered 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

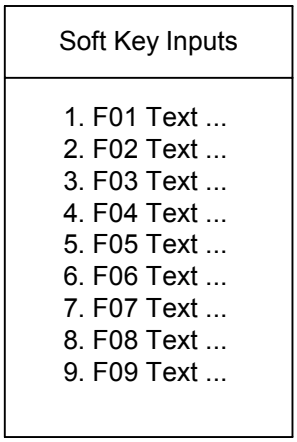
These are user definable soft key inputs. The soft key inputs are used in conjunction with IPI registers F1INPUT/R01 thru F9INPUT/R09. The text is defined in Builder Text. Nine (9) soft key inputs can be defined. **F10** is used for Exit. Up to seven (7) characters can be used for non-unicode label and 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**.



SKEYINPUTS

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

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

   The message text box activates.

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

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

Languages

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

If you attempt to set up the option for a language and the CNC cannot find the associated text file, it displays an error message.

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

1. Go to Map 11**, Menu D**. Highlight the desired language, and press **ENTER**.

   The CNC loads the language file and restarts the system when you exit the Setup Utility. The CNC now displays Messages and other text in the selected language.

Software Updates

To install an updated version of the CNC software:

1. Insert the disk containing the updated version of the CNC software into the floppy drive.
2. Go to Map 11**, Menu B**. Highlight **Software Update**, and press **ENTER**.

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

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

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

Changing Passwords

For default passwords, refer to "Section 1, Password Restricted Parameters" located elsewhere in this manual.

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

To change the password:

1. Go to Map 11**, Menu I**. Highlight the level of the password to be changed, and press **ENTER**.

   The CNC prompts for the old password.

2. Type the old password, and press **ENTER**.

   The CNC prompts for the new password.

3. Type the new password, and press **ENTER**.

   The CNC prompts for confirmation of the new password.

4. Re-type the new password, and press **ENTER**.

   The CNC activates the new password.

Section 3 - Operator Setup

The Operator Setup Utility (Refer to Map 11**, Menu B**) configures settings for the following:

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

Control Software Parameters

Go to Map 12**, 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 axis display	Switches the default axis display between large and normal.	**Normal** [Default] - Configures the axis display to show Machine, Program, Target, and Distance To Go displays. Large - Configures the axis display to show enlarged X, Z and C position displays only.
Default units	Switches the default measurement units (Inch/MM Modes).	**Inch** [Default] - Activates Inch Mode as default. MM - Activates MM Mode as default.
Default axis values	Switches Absolute/ Incremental default mode (determines how axis values for arcs, lines, and other moves are measured).	**Absolute** [Default] - Makes every move in reference to an Absolute Zero position (Program Zero or Part Zero). Incremental - Makes each move in reference to the last programmed endpoint.
Circle adjustments	Specifies whether circle centers or end-points will be adjusted. Circle centers require adjustment when the CNC encounters incorrect circle center or end-point coordinates.	Center **-** Adjusts the position of the circle center when the CNC encounters incorrect coordinates for either a circle center or endpoint. **End-point** [Default] - Adjusts the position of the circle endpoint when the CNC encounters incorrect coordinates for either a circle center or endpoint.

(Continued…)

Table 3-1, Control Software Parameters (Continued)

Control Software Parameter	Function	Settings
Circle centers	Switches the default mode for programmed circle center coordinates.	Absolute - CNC interprets programmed circle center coordinates as Absolute values. **Incremental** [Default] - CNC interprets programmed circle center coordinates as Incremental values.
Maximum arc correction	Specifies the maximum amount of correction the CNC will apply to an arc block before declaring an error.	0.005000 [Default] Valid range: (0.000000 to 0.039370)
Internal accuracy	Maximum accuracy available (system resolution).	0.00000100 [Default] Valid range: (0.00000001 to 0.00100000)
External accuracy	Specifies the maximum system accuracy obtainable on a given machine (machine resolution).	0.00010000 [Default] Valid range: (0.00000001 to 0.00100000)
Compensation cutoff angle	Minimizes wasted travel on acute angle. Refer to Figure 3-1, Compensation Cutoff Angle.	15.0 degrees [Default] Valid range: (1.0 to 90.0) degrees
User macro file	Specifies macro filename created by user.	USERCANN.G [Default]
Load user macro file	Specifies whether to load user macro at system startup.	**No** [Default] - Does not automatically load user macro at startup. Yes - Automatically loads user macro at startup.
Disk access marker	Activates/deactivates the Disk access marker.	**On** [Default] - Activates the Disk access marker. When the CNC is reading/writing information from/to a disk, the Disk access marker is displayed in the upper-left corner of the screen. The Disk access marker looks like a small arrow. Off - Deactivates Disk access marker.
Max memory allocated (in MB-bytes)	Used only with offline software. Limits the amount of memory available to the software.	[Default: 4 MB] Valid range: (2 to 128) MB
Force simulation mode	In Simulation Mode, the CNC does not generate DAC and I/O outputs. The CNC starts in Simulation Mode. Moves can be commanded and displayed, but no actual machine movements occur.	**Yes** – Enable [Default] No – Disable Press **ENTER** to toggle the option to No (disable).

(Continued…)

Table 3-1, Control Software Parameters (Continued)

Control Software Parameter	Function	Settings
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 Press **ENTER** to toggle the option to No (disable).
Screen blanking delay (minutes)	Specifies the screen blanking delay period, in minutes. The delay will be the time between a detected screen idle condition and the activation of the screen saver. To reactivate, press any key.	[Default: 5 minutes] Valid range: (0 to 20160) minutes

Compensation Cutoff Angle

Figure 3-1, Compensation Cutoff Angle illustrates two Compensation Cutoff scenarios. Assume all programmed moves are made with Tool Diameter Compensation active. The diagram describes two cases:

- **Diagram A** shows the tool path that results when no Compensation Cutoff Angle is used. The tool path travels beyond the part diameter to a point where compensated Moves 1 and 2 intersect, before the CNC executes Move 2.
- **Diagram B** shows the tool path that results when a Compensation Cutoff Angle is used. The CNC introduces one arc move, equal to the radius of the cutter, between Programmed Moves 1 and 2. This arc is not programmed, but is a function of the active Compensation Cutoff Angle and alters the tool path, decreasing the amount of travel necessary to complete the programmed moves.

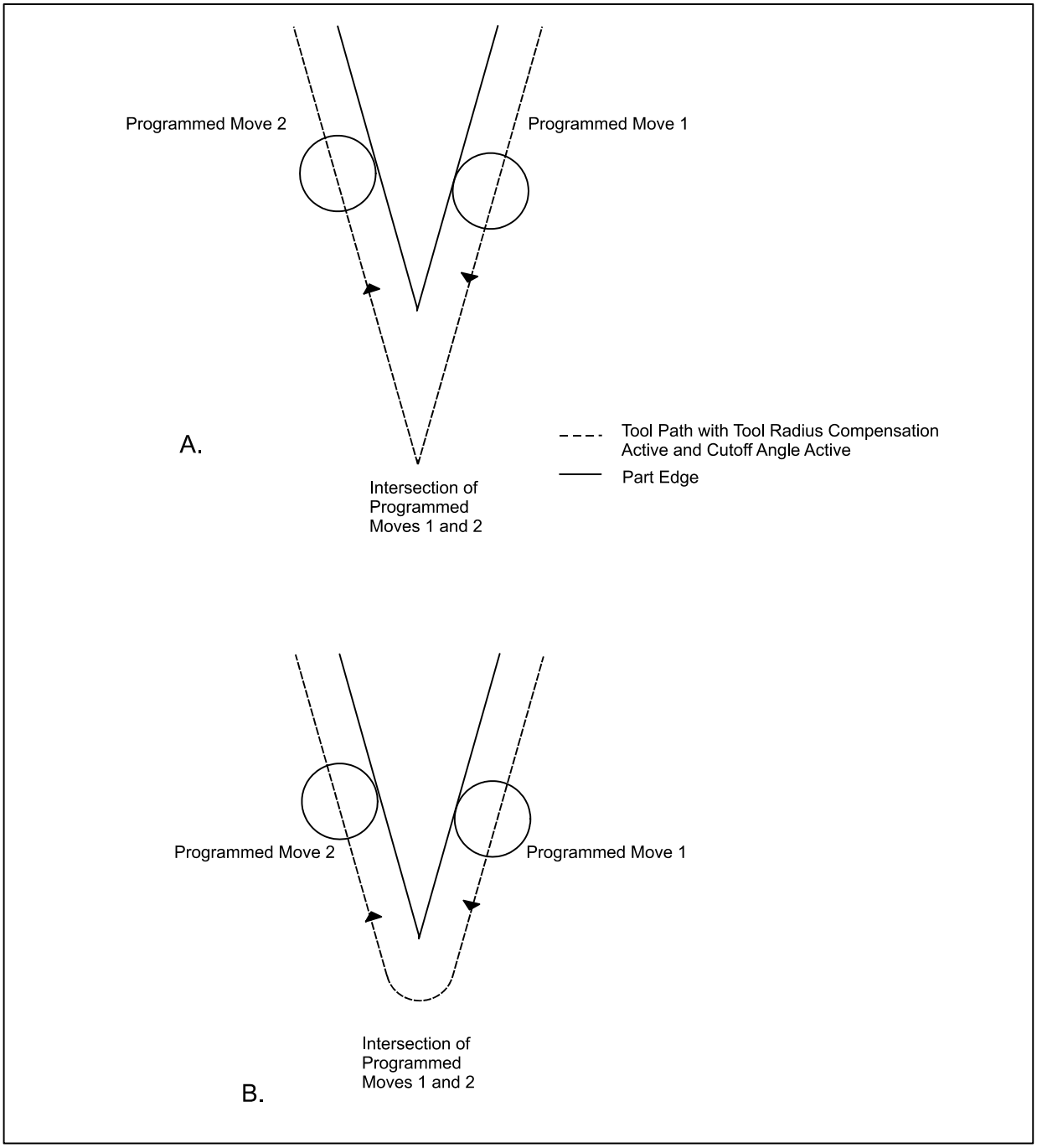


Figure 3-1, Compensation Cutoff Angle

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	Select number of data bits.	**7** [Default] or 8
Stop bits	Select number of stop bits.	0 or **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.

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

To set up the communications parameters:

1. Refer to Map 13**, Menu C**. Highlight the desired parameter. Press **ENTER**.
2. Select one of the following options: **Port**, **Baud**, **Parity**, **Data bits**, **Stop bits**, or **Software**. Press **ENTER**.

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

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

Draw Mode Setup Parameters

Refer to Map 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 you 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 is displayed 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.	Enter a value in the entry field. [Default: 1.0] Value range: (0.0 to 1000.0) If the CNC is in Inch Mode, each square in the grid is one square inch in size for this setting. **NOTE:** The CNC converts the set grid value if the measurement unit is changed. For example: if the Grid Size is set for 1 in Inch Mode and you switch to MM Mode, the CNC changes the Grid Size to 25.4 mm (equal to 1 inch).
Tool display	Turns the tool display on and off.	**On -** The static tool (as defined by the Tool Location Code and Radius in the Tool Page) is displayed as it cuts the part. [Default] Off - No tool is displayed.

(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].
Aspect ratio correction factor	Corrects for distortion in displayed graphics. (Flattened circles, etc.)	1.47 [Default] Valid range: (0.01 to 10.00) **Caution: Only qualified technicians should adjust this setting.**
Save Draw image	Saves draw image when user switches to Edit Mode. In Draw Mode, when the Edit (**F2**) soft key is pressed, the CNC switches to Edit Mode. The user later re-enters the Draw Mode when you exit Edit Mode. If this option is enabled (Yes), the CNC restores the image on the screen prior to entering Edit. This image will correspond to the part program drawing.	**Yes** - Saves draw image. [Default] No - Does not save draw image.

Edit Mode Setup Parameters

Refer to Map 13**, Menu K**, 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
Restore to previous session	If Yes (enabled), when user exits a program in Edit Mode, the CNC marks the position where the last edit was made. The next time the program is opened, the cursor will be located at that spot.	**Yes** - restores to previous session. [Default] No - does not restore to previous session.
Show top line	Determines whether an optional "top line" will be displayed in the Edit Mode. The top line contains the active mode information and first block of the open program.	**Yes** - Displays top line. [Default] No - Does not display top line.
Default insert mode	Switches On/Off Default Insert Mode. Insert Mode inserts new text without overwriting existing text.	**On** - Automatically sets Insert Mode as default. [Default] Off - Does not automatically set Insert Mode as default.
Auto tab to previous line's position	This option is available only with off-line systems or systems with attached keyboards. When a line is indented, the CNC uses the indented position as the first tab position of the following line. For example, the user indents one line by four spaces and then moves to the beginning of the next line by pressing **ENTER**. When you press **TAB**, the cursor now advances four spaces.	**Yes** - Enables auto tab to previous line's position. [Default] No - Disables auto tab to previous line's position.
Update screen on macros	When non-programmed macros are used (Play, Record and Repeat options in the Control Software Edit Mode), this parameter defines how the CNC display will be updated.	**Yes** - Updates screen (prints macro results) line by line as macro is run. [Default] No - Updates screen (prints macro results) after macro is completed.

(Continued…)

Table 3-4, Edit Mode Parameters (Continued)

Edit Mode Parameter	Function	Settings
Default tab width	This option is available only with offline systems or systems with keyboards attached. Sets default tab width. Range is 2 to 16 spaces. When you press **TAB**, the cursor advances by the specified number of spaces.	Enter a value in the entry field. [Default: 4] Valid range: (2 to 16) spaces
Create backup program	A backup program is created when an edit is made. Each time the program is edited, the CNC updates the backup file. The backup program will not contain an edit until a new edit is made.	Yes - Backup program is created and maintained. **No** - No backup programs are created. [Default]
Delete internal file when program saved	When the program is saved, the CNC automatically deletes the existing internal file (*.S files) and replaces it with the saved file.	**Yes** - Deletes internal file when you save a file and replaces it with updated file. [Default] No - CNC does not delete internal file when you save a program.
Case sensitive Find	Determines whether the Find feature will search for uppercase and lowercase letters to determine a match.	Yes - Find search parameter looks for words that exactly match the entered word specific to capitalization and style. **No** - Find search parameter looks for the entered word, regardless of capitalization and style. [Default]
Memory Reserved from Editor (K-bytes)	Specifies the maximum memory allocation for the Editor.	Enter a value in the entry field. [Default: 300 K] Valid range: (16 to 32000) K

Program Directory Parameters

Refer to Map 14, **Menu C** for the available program parameters. These parameters specify the following:

- The way program information is displayed in the Program Directory
- Whether to delete backup files during optimization
- Whether and how often the disk is checked via software

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.	*.G (program file extension)
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] Enter user directory location.

Display Settings

Refer to Map 14, **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

Refer to Map 14, **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 device	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. [Default: PRN]. **NOTE:** The user directory for CNC systems is C:\USER
Lines per page	Number of lines to be printed per page (8.5 X 11”).	Enter a value in the entry field. [Default: 55] Valid range: (1 to 66) lines per page
Page heading	Prints a page heading including filename, date and time, and page number.	**Yes** - 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 CPI 17 CPI 20 CPI 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 the Utilities to manage the configuration file saved in the Setup Utility.

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

Save Configuration

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

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

1. Go to Map 15**, Menu B**. Highlight **Save Configuration**. Press **ENTER**. The system prompts for a password.
2. Press **ENTER**. The configuration will be saved. The backup filename is P4TCFG.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**, Menu B**. Highlight **Copy Configuration**. Press **ENTER**. **Menu C** displays.
2. Highlight **A:** to copy the configuration to a floppy diskette. The configuration will be saved as **A:\P4TCFG.CFG**.
3. Highlight **Other..** to save the configuration to another drive or under another filename.

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

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

Restore from Copy

Use this feature to restore a copy of the configuration from the A:-drive (A:\P4TCFG.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, **Menu B**. Highlight **Restore from Copy**. Press **ENTER**. The system prompts for a password.
2. (Refer to "Section 1, Password Restricted Parameters.") Type the limited access password. Press **ENTER**. **Menu E** displays.
3. Highlight **A:** to restore the configuration from a floppy diskette. The configuration will be restored from **A:\P4TCFG.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 (e.g., 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, **Menu B**. Highlight **Restore from Backup**. Press **ENTER**.

   The system automatically swaps the current file with the backup file.

Compare Configuration

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

1. Go to **Utilities Setup, Menu B**. Highlight **Compare Configuration**. Press **ENTER**. **Menu G** displays.
2. Highlight **A:** to compare the current file with P4TCFG.CFG on the A:-drive.
3. Highlight **Other..** to display **Menu H**. Type the directory with which you wish to compare files. For example: C:\Home\FILE_1.

Print Configuration

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

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

1. Go to Map 15**, Menu B**. Highlight **Print Configuration**. Press **ENTER**.
2. **Options Setup, Menu F** displays.
3. Highlight **Printer**, and press **ENTER**. Press **Yes** (**F1**) to print to your printer. Press **No** (**F2**) to return to **Menu B**, or go to step 4.
4. Highlight **Text File (A:)**, and press **ENTER**. P4TCFG.TXT will be the filename. Press **Yes** (**F1**) to print to your A drive. Press **No** (**F2**) to return to **Menu B**, or go to step 5.
5. Highlight **Text File (Other)**, and press **ENTER**. Enter the directory and filename to which you wish to print.

Section 5 - Fine-Tuning Systems with Linear Encoders

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

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

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

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

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

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

1. In the CNC's Setup Utility, ensure that the CNC is set to **linear encoder**, with the proper encoder resolution and display resolution settings.

2. Use the AXIS Selector Switch to select the required axis for a manual move.

3. Use the **JOG** Selector Switch to set the CNC to Jog Mode 1. This sets the axis to move in increments of 1 times the machine resolution each time you press a **JOG** key.

4. Use the **PLUS** or **MINUS JOG** key to move the axis in a positive or negative direction.

5. Use the **PLUS** or **MINUS JOG** key to move the axis in the opposite direction.

6. Look at the ballscrew (or handwheel) of the axis. When lost motion occurs, the ballscrew moves a much greater distance than the system resolution (for example, 0.0005" on a 10-micron linear encoder). That distance is the lost motion on that axis.

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

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

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

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

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

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

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

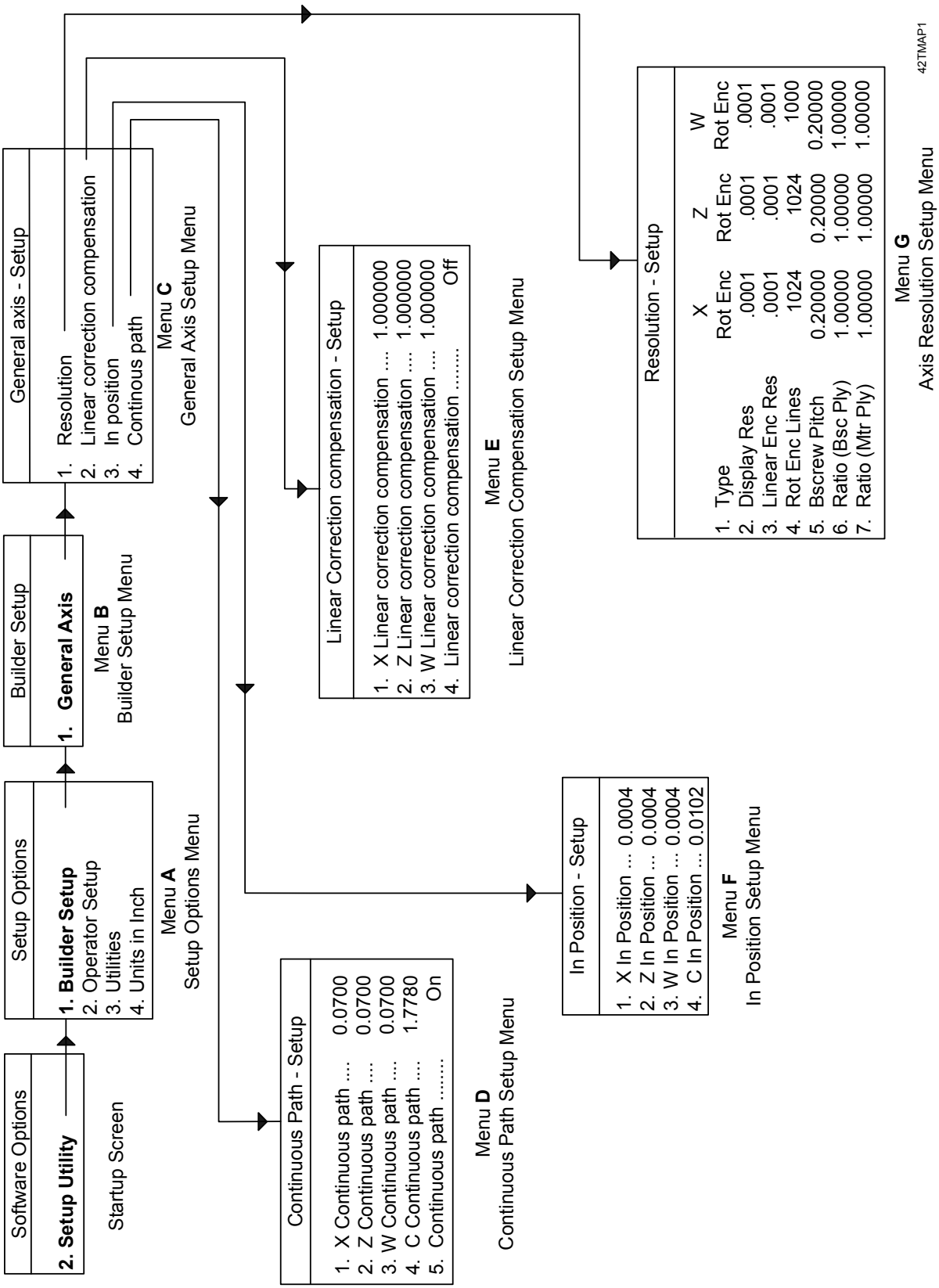
8. Repeat the procedure on all enabled axes.

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

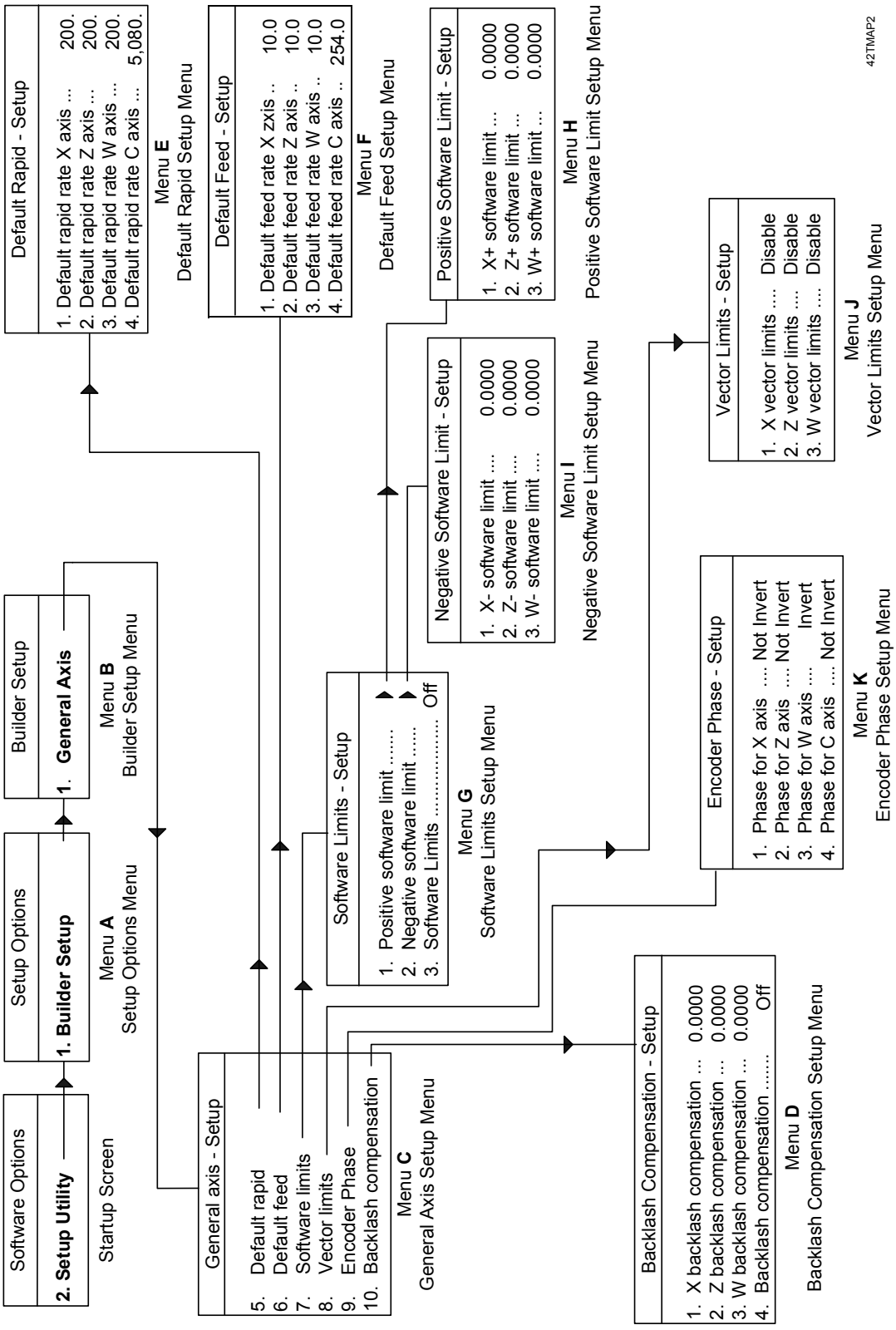
   NOTE: Always make minor adjustments to Setup Utility parameters and check until the problem being corrected is fixed. If you make large changes to parameters, you may fix the problem at hand, but cause others.

Section 6 - Setup Utility Maps

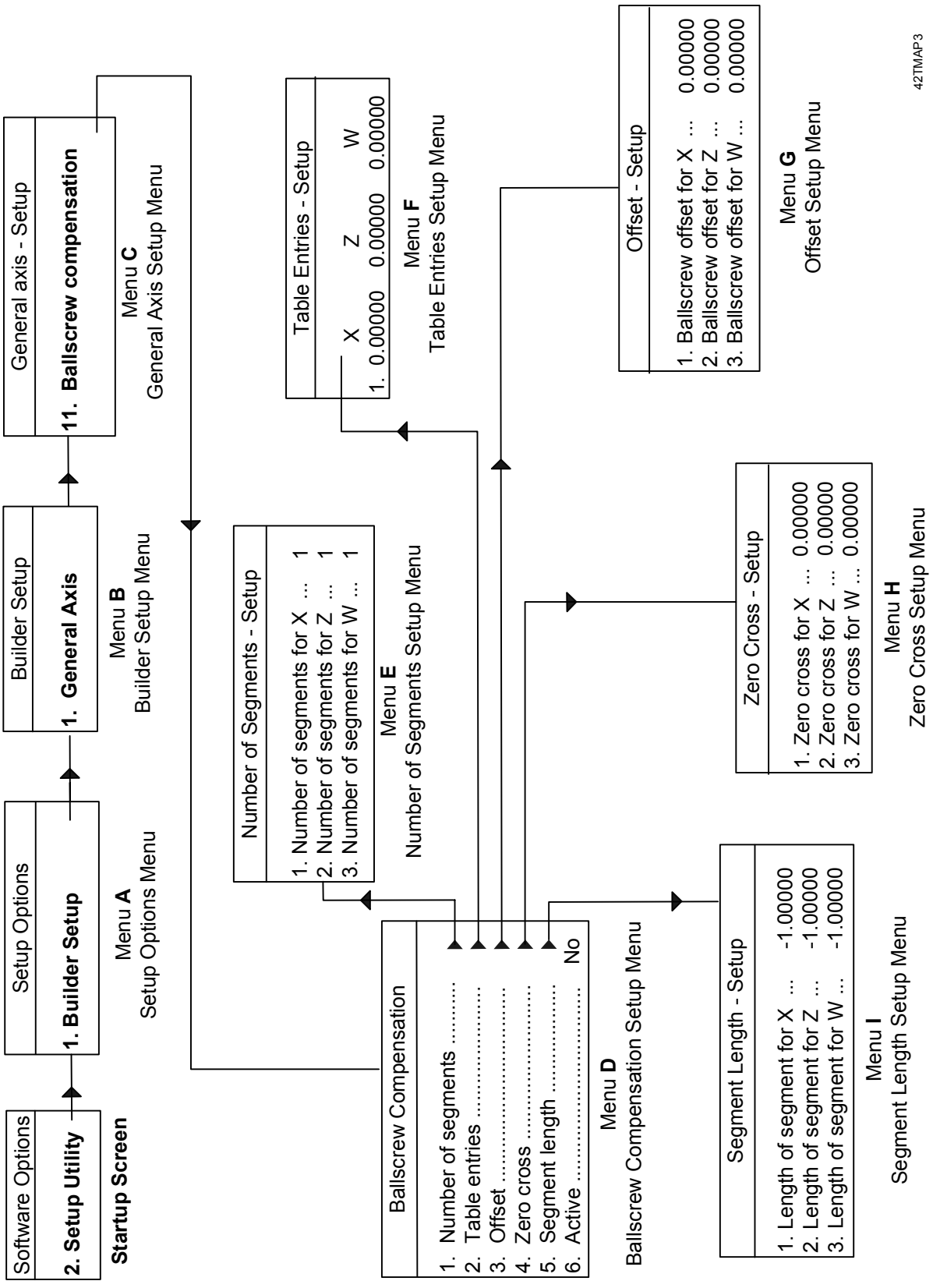
This section contains maps referenced in Sections 2 through 4. Refer to "Section 1, Navigating Through the Setup Utility" for instructions on how to use the software and maps.



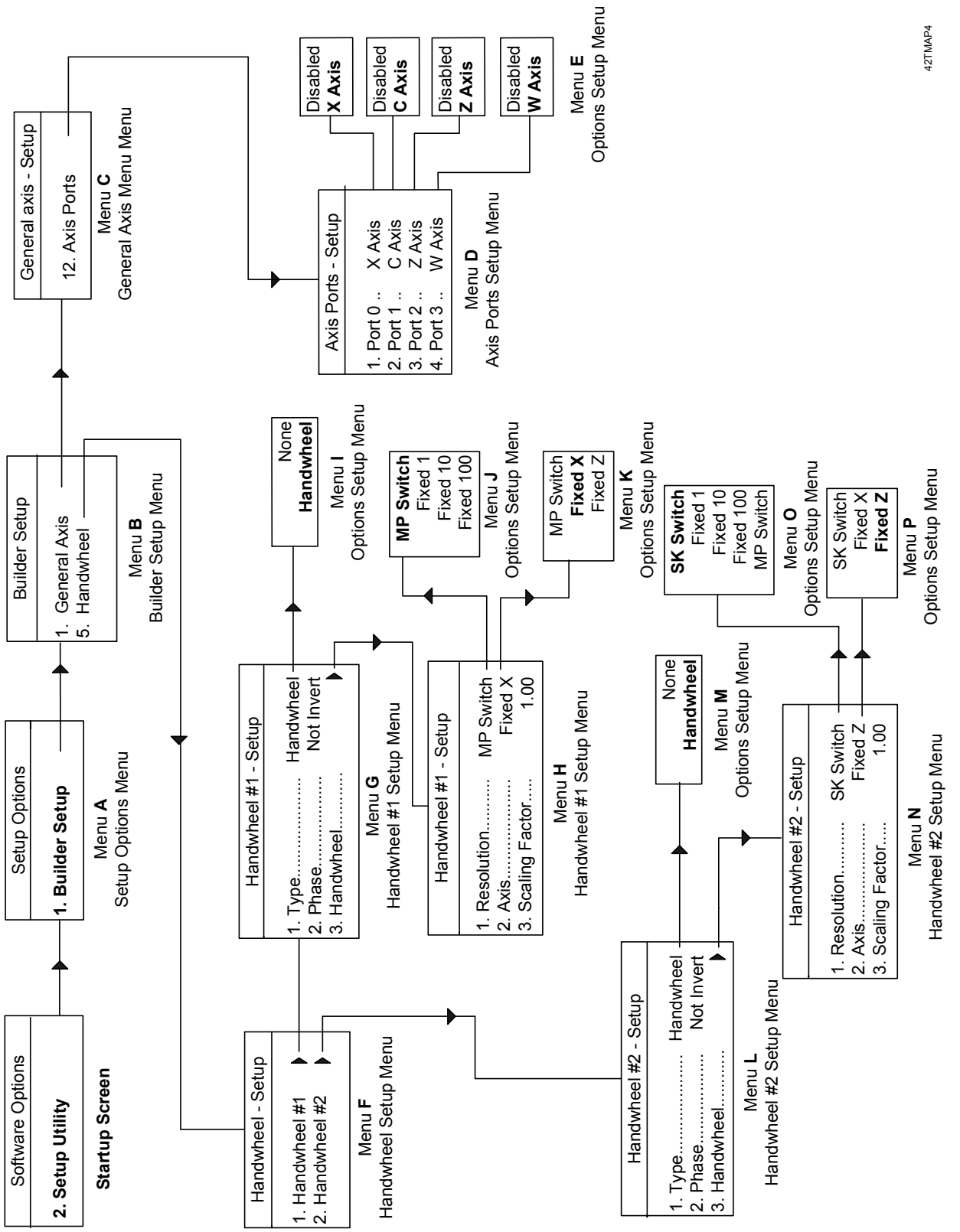
Map 1



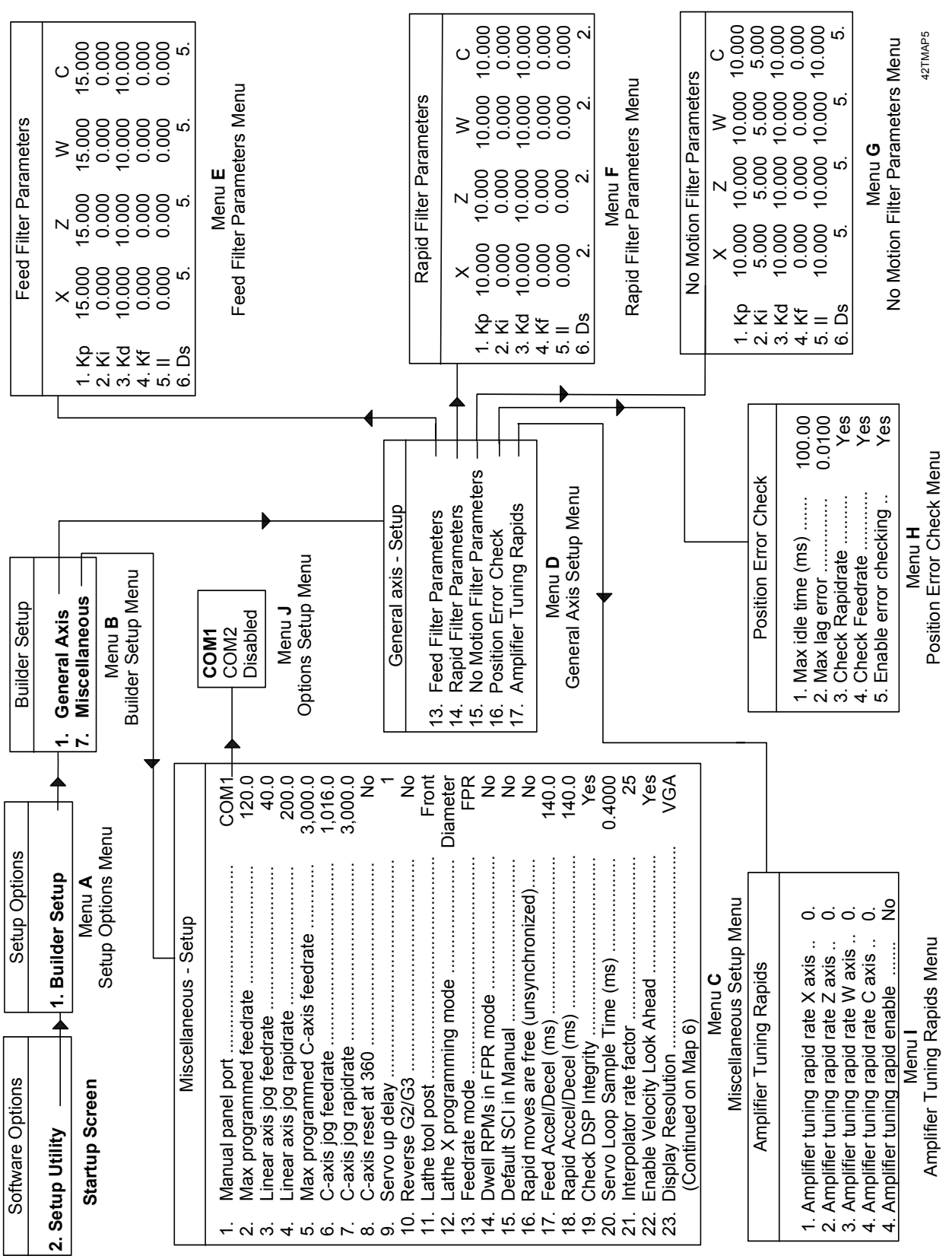
Map 2



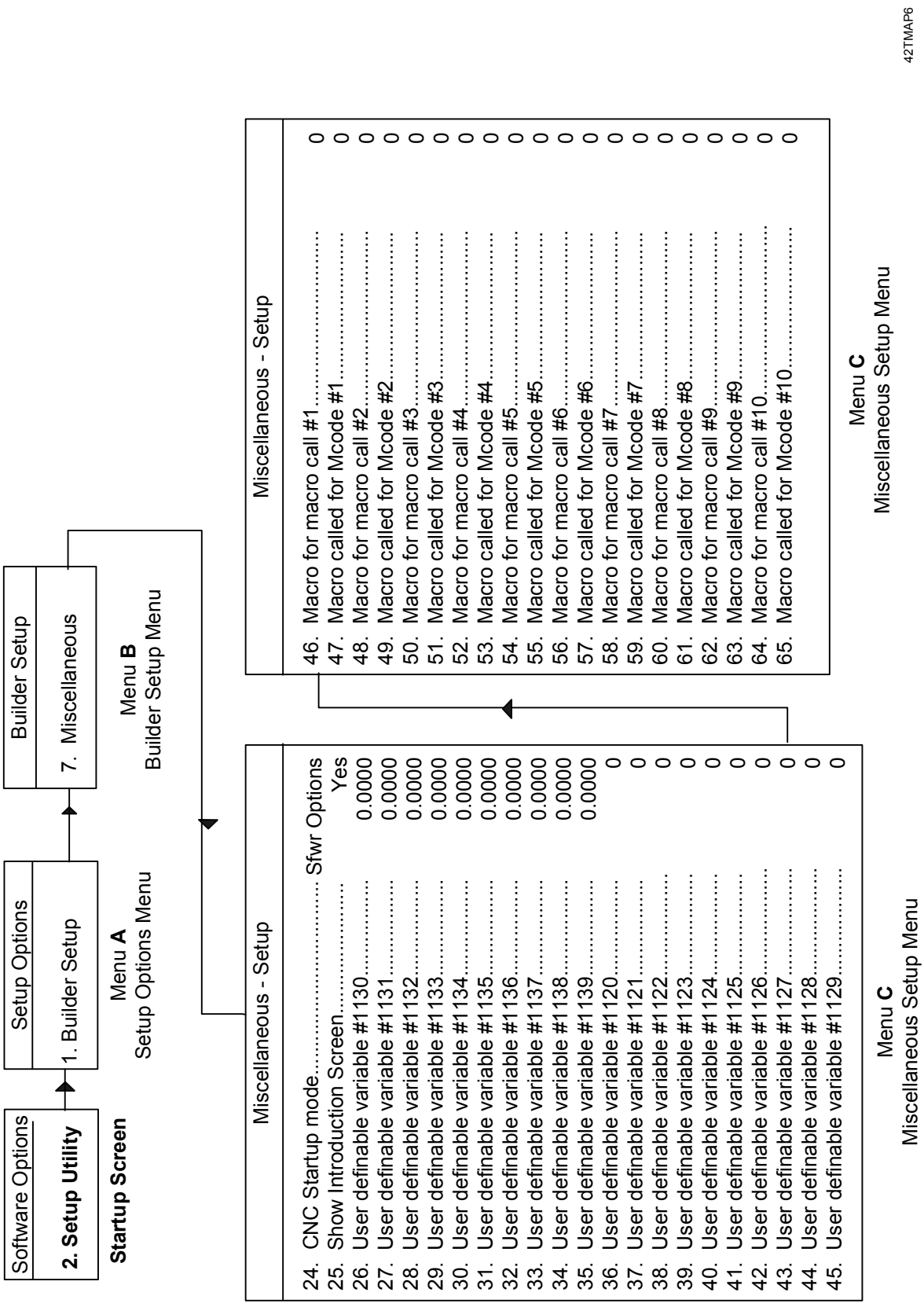
Map 3



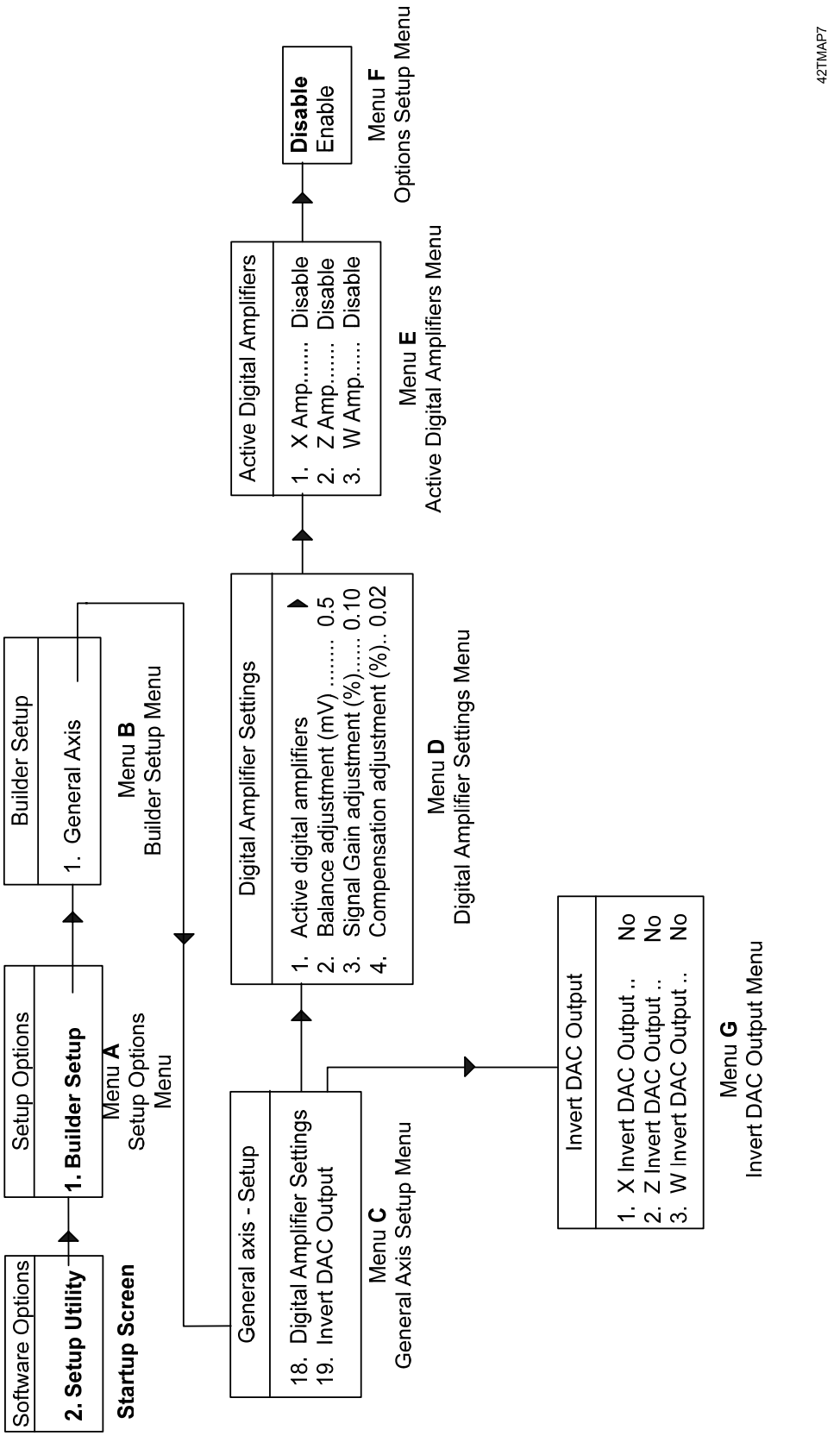
Map 4



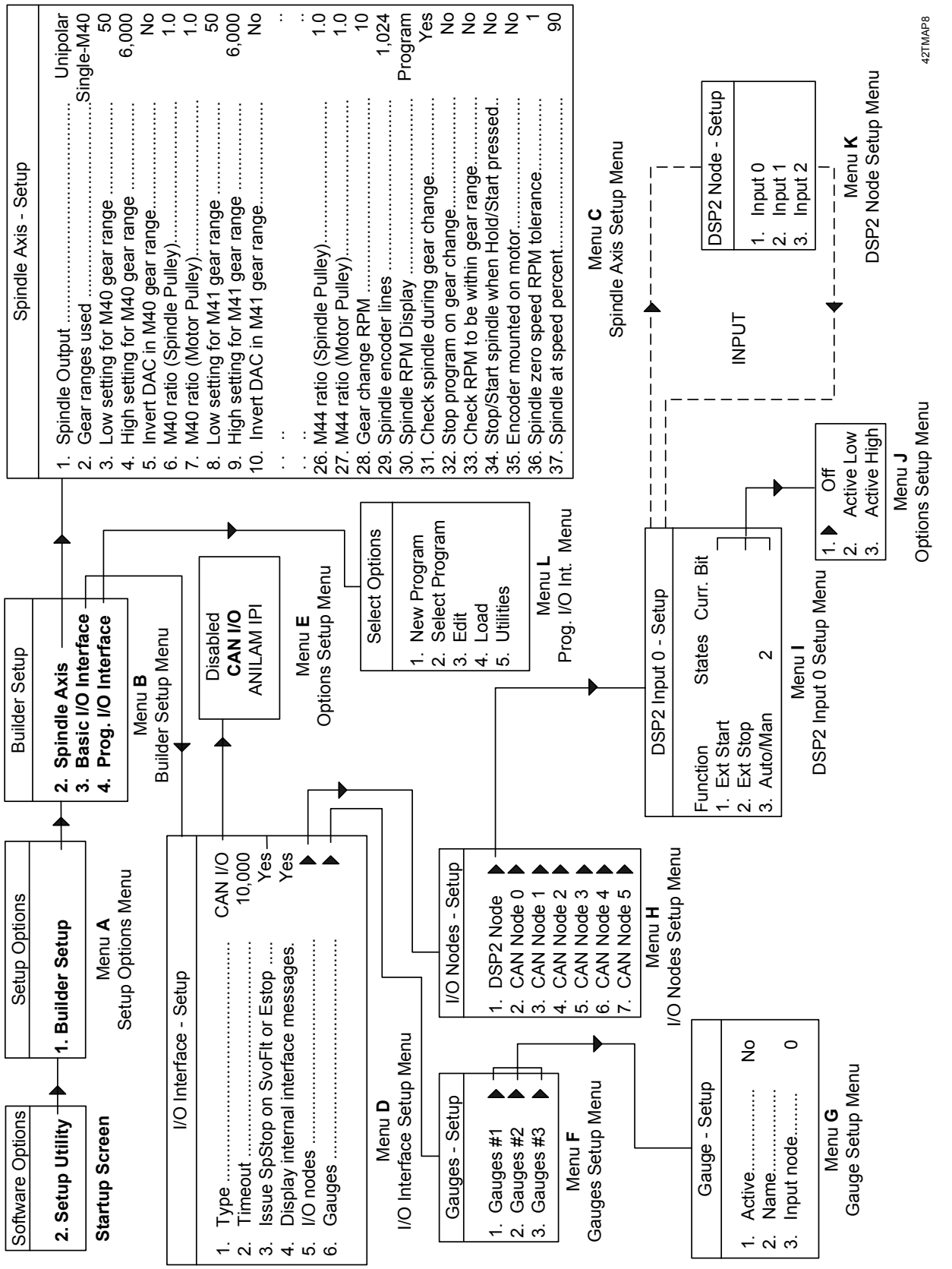
Map 5



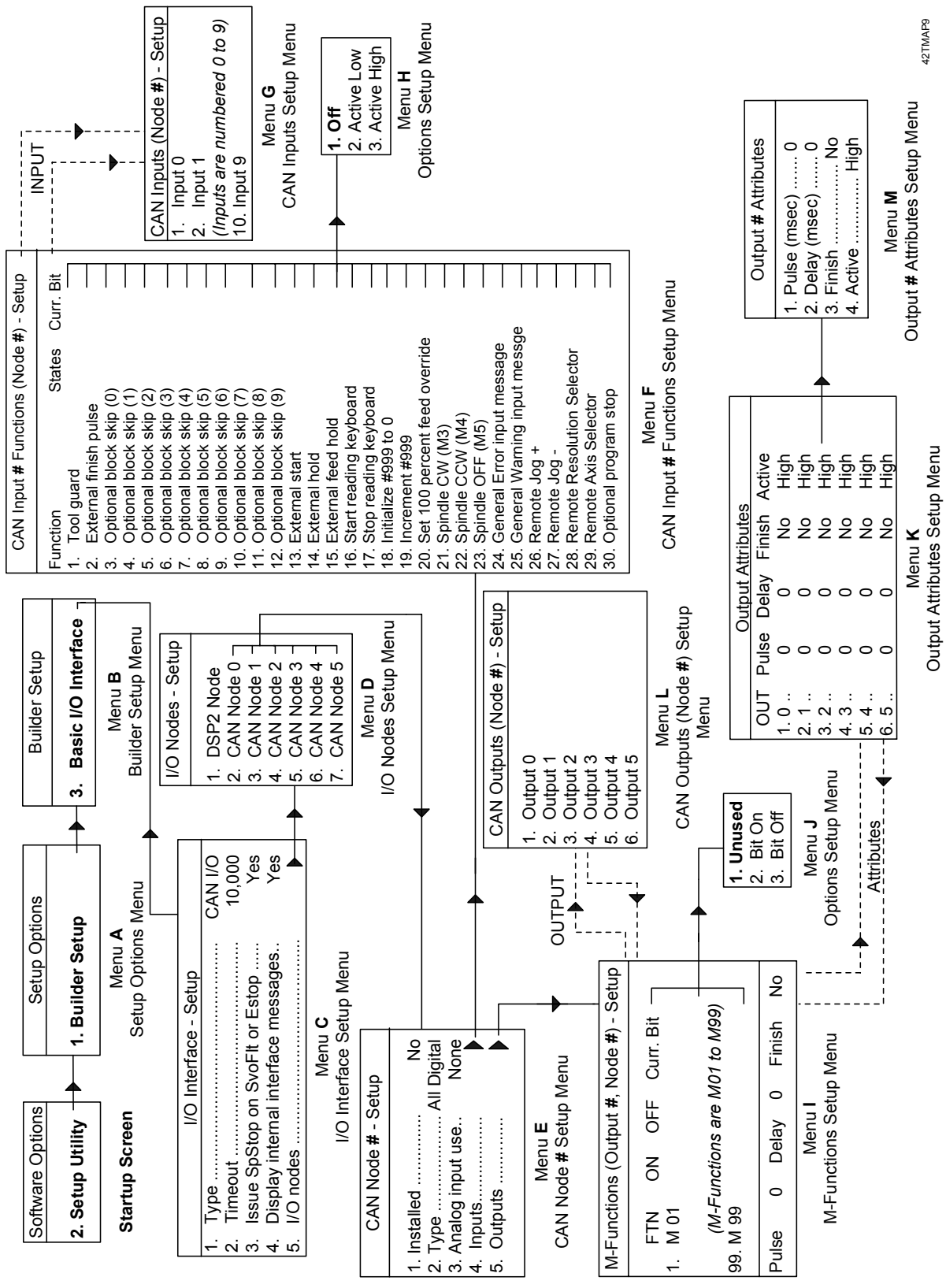
Map 6



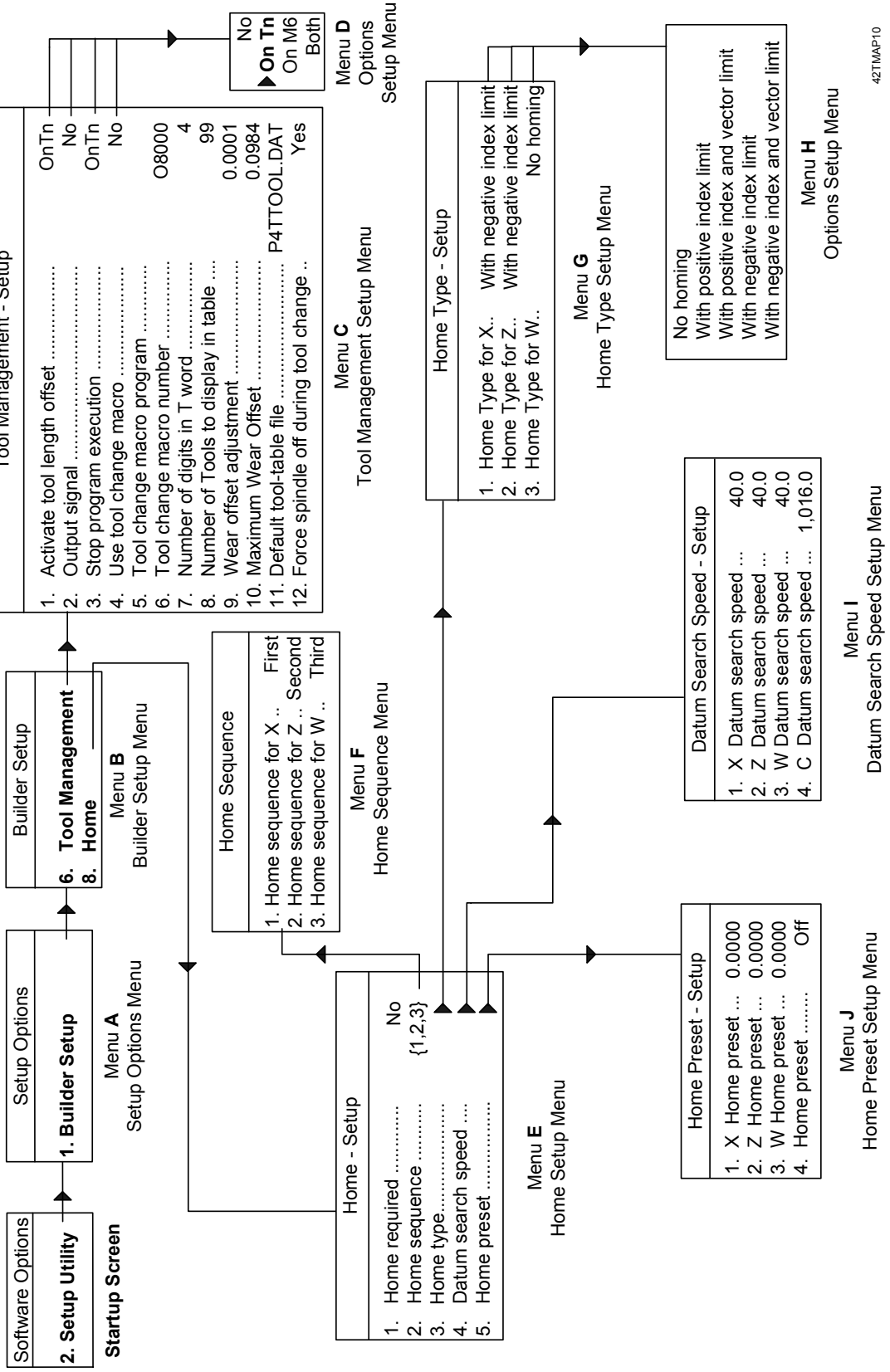
Map 7



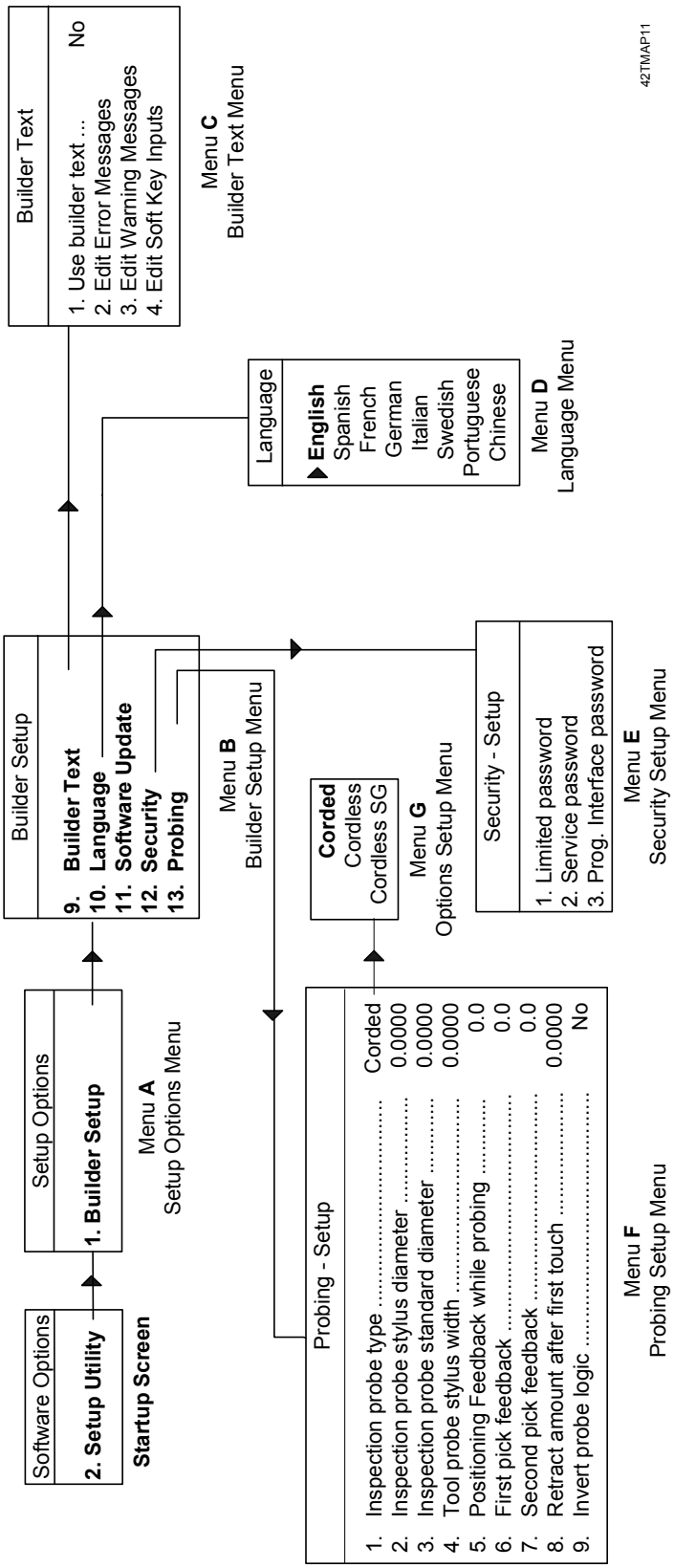
Map 8



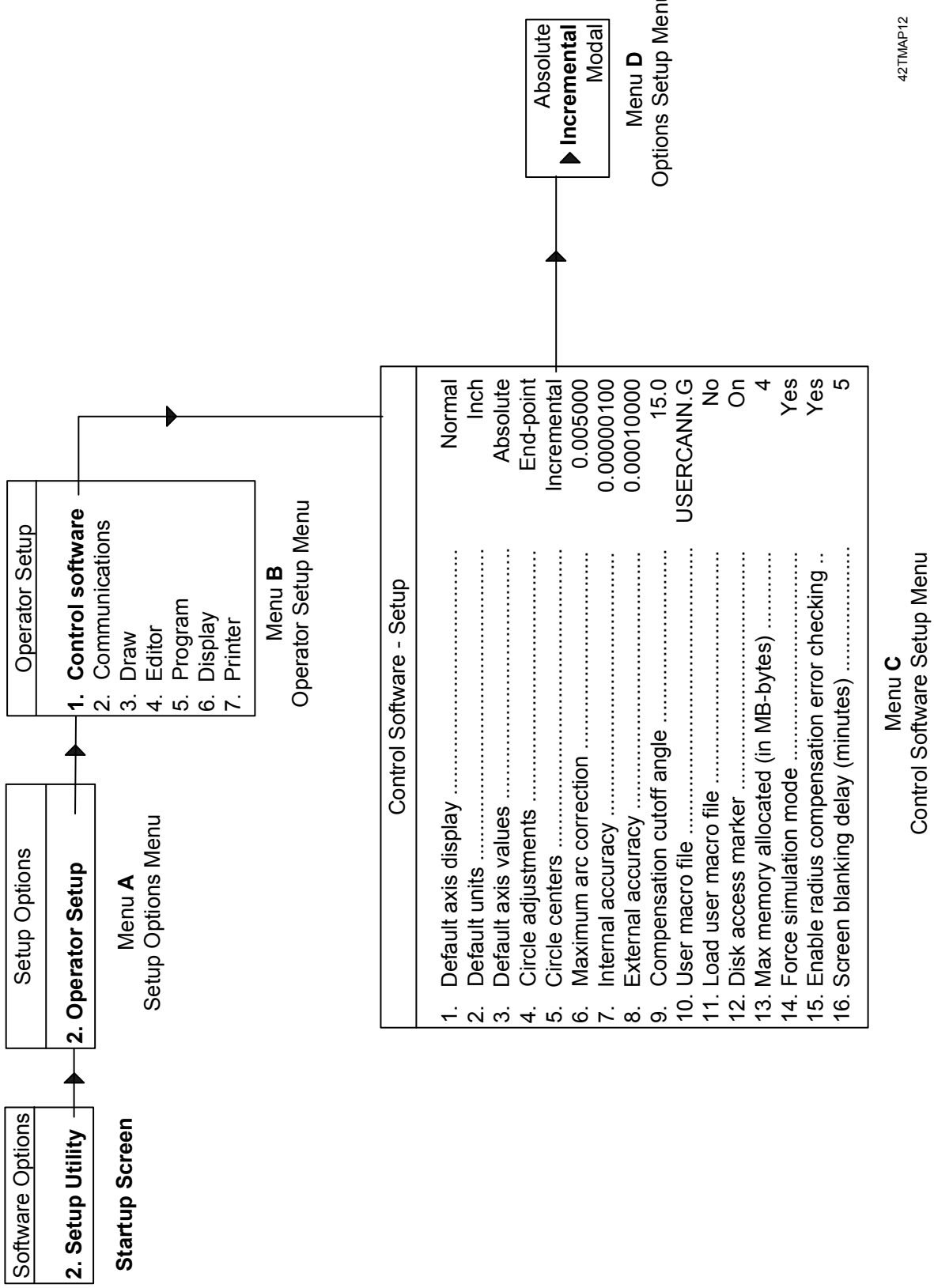
Map 9



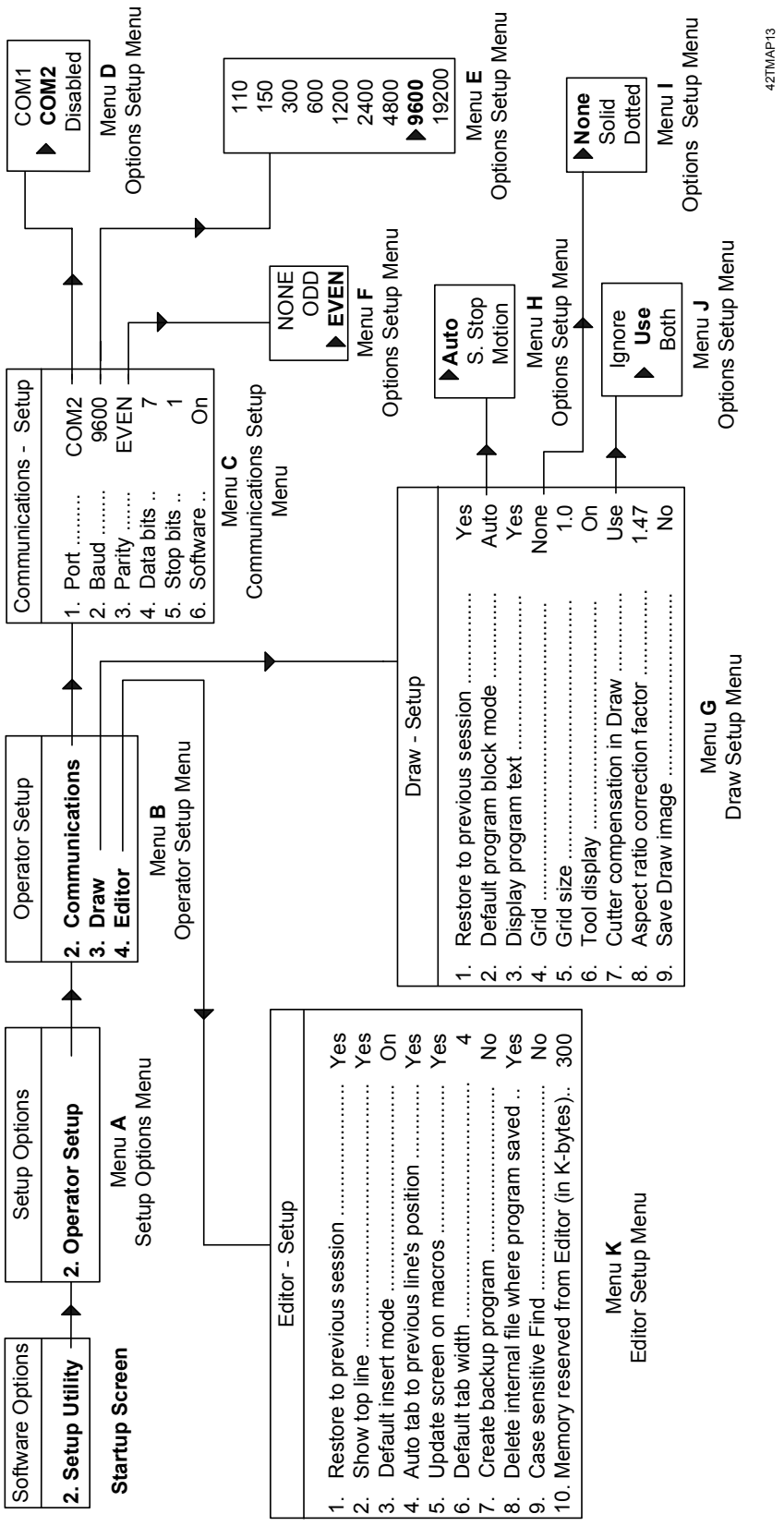
Map 10



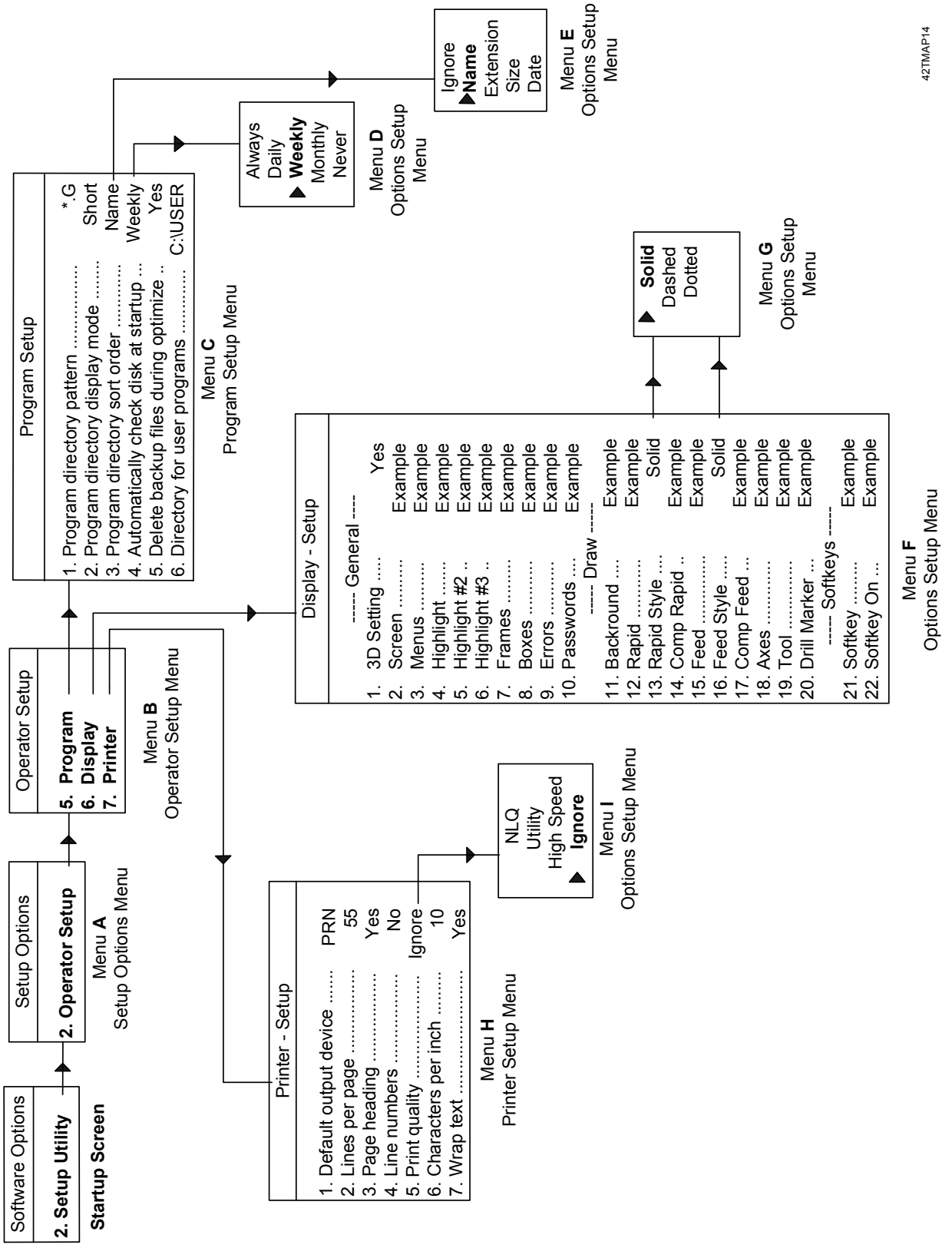
Map 11



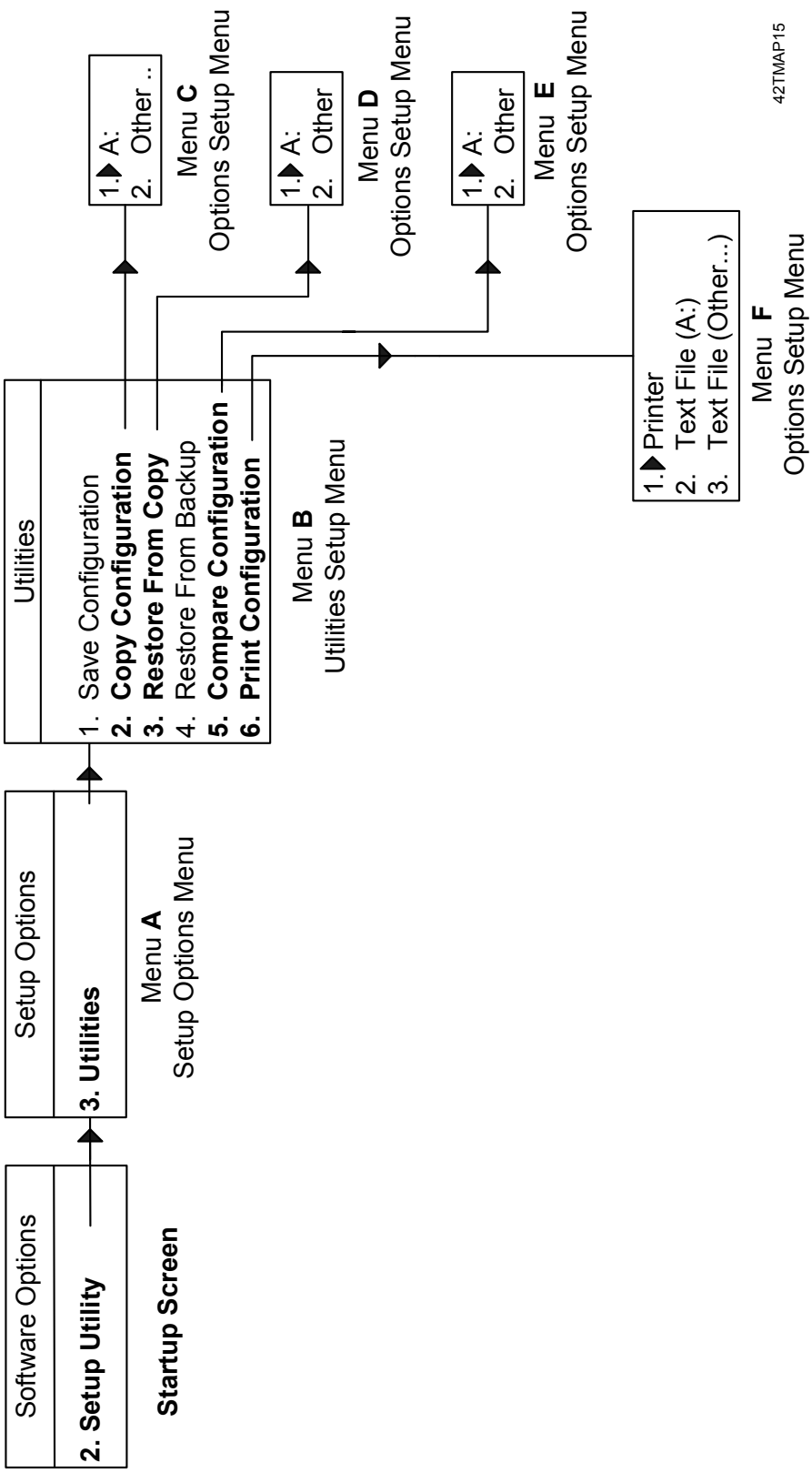
Map 12



Map 13



Map 14



Map 15

A

B

F

G

H

I

J

K

L

M

N

S

T

U

V

W

X

Z

ANILAM

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

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

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

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