CENTROID™

M400 & M39
Operators Manual

Centroid M-Series Mill Operators Manual

CNC Software Version: CNC12 Version v5.04 DRAFT

Models: Acorn, AcornSix, Allin1DC, Oak, Hickory, MPU11, M400, and M39

www.centroidcnc.com

For the latest version of the manual, please visit:
www.centroidcnc.com/centroid_diy/downloads/operator_manuals/centroid-cnc12-mill-operator-manual.pdf

For additional Centroid manuals please visit
www.centroidcnc.com/centroid_diy/centroid_manuals.html

U.S. Patent #6490500

READ THIS MANUAL BEFORE USING THIS PRODUCT. FAILURE TO FOLLOW THE INSTRUCTIONS AND SAFETY PRECAUTIONS IN THIS MANUAL CAN RESULT SERIOUS INJURY OR DEATH.

All operators and service personell must read this manual before operating CENTROID CNC control equipment and all connected machine tools.

Keep this manual in a safe location for future reference.

Throughout this manual and on associated products where applicable, in accordance with ANSI Z535, the following symbols and words are used as defined below:

DANGER	"DANGER" with or without a red background = Hazard WILL cause death or serious injury if ignored.	**DANGER**
WARNING	"WARNING" with or without an orange background = Hazard COULD cause death or serious injury if ignored.	**WARNING**
CAUTION	CAUTION "CAUTION" with or without a yellow background = Hazard MAY cause minor to moderate injury if ignored.	**CAUTION**
NOTICE	"NOTICE" with or without a blue background = Indicates an action to prevent damage to the product or other materials used with product.	***NOTICE***

Information provided by CENTROID relating to wiring, installation, and operation of CNC components is intended as only a guide, and in all cases a qualified technician and all applicable local codes and laws must be consulted. CENTROID makes no claims about the completeness or accuracy of the information provided, as it may apply to an infinite number of field conditions.

As CNC control products from CENTROID can be installed on a wide variety of machine tools NOT sold or supported by CENTROID, **you MUST consult and follow all safety instructions provided by your machine tool manufacturer regarding the safe operation of your machine and unique application.**

CENTROID CNC controls provide facilities for a required Emergency Stop circuit which can be used to completely disable your machine tool in the event of an emergency or unsafe condition. **Proper installation of your CNC control MUST include the necessary wiring to disable ALL machine tool movement when the Emergency Stop button is pressed.** This includes the machine, servo motors, tool changers, coolant pumps, and any other moving parts. DO NOT disable or alter any safety feature of your machine or CNC control.

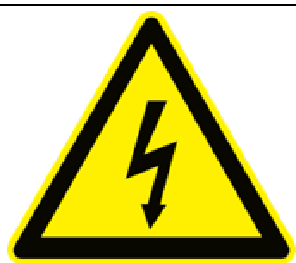
Never alter or remove any safety sign or symbol from your machine or CNC control components. If signs become damaged or worn, or if additional signs are needed to emphasize a particular safety issue, contact your dealer or CENTROID.

CNC Control Operating Specifications

	Minimum	Maximum
Operating Temperature	40 °F (5 °C)	104 °F (40 °C)
Ambient Humidity	30% relative, non-condensing	90% relative, non-condensing
Altitude	0 Ft (Sea Level)	6000 Ft (1830 m)
Input Voltage (110, 220, 440 VAC, System Dependent)	−10% of Specified System Input Voltage	+10% of Specified System Input Voltage

Note: Your machine may have operating conditions different than those shown above. Always consult your machine manual and documentation.

Safety signs and labels found on your machine tool and on CNC system components typically follow the following examples:

Warning Symbol	Hazard Severity Level & Word Message	Action Symbol
	**CAUTION** High Voltage Electrocution Hazard. Death by electric shock can occur. Turn off and lock out system power before servicing.	
The warning symbol identifies the potential hazard and reinforces the word message.	The severity level is one of “DANGER”, “WARNING”, “CAUTION”, or “*NOTICE*”. Word message includes 3 parts: hazard, consequence if warning is ignored, and action to prevent injury.	Indicates actions to prevent injury. Blue circles indicate mandatory actions to avoid harm. Red circles with diagonal slashes indicate prohibited actions to avoid harm.

CNC Machine Tool Safety

- All machine tools contain hazards from rotating parts; movement of belts, pulleys, gears, and chains; high voltage electricity; compressed air; noise; and airborne dust, chips, swarf, coolant, and lubricants. Basic safety precautions must be followed to reduce the risk of personal injury and property damage.
- Your local safety codes and regulations must be consulted before installation and operation of your machine and CENTROID CNC control. Should a safety concern arise, always contact your dealer or service technician immediately.
- Access to all dangerous areas of the machine must be restricted while the machine is in use. Ensure that all safety guards and doors are properly in place during use. **Automatically-controlled machine tools may start,**

stop, or move suddenly at any time. Do not enter the machining area when the machine is in motion; death or severe injury may result.

- Personal protective equipment, particularly ANSI-approved impact safety glasses and OSHA-approved hearing protection must be used. Proper handling, storage, use, and disposal of materials in accordance with manufacturer's instructions and Material Safety Data Sheets (MSDS, or your local equivalent) must be followed.
- DO NOT operate your machine or CNC control in explosive atmospheres or in environmental conditions outside of the manufacturer's specified ranges. Electrical power must meet the specifications provided by your machine and CNC control manufacturer.
- DO NOT operate your machine or CNC control if any safety systems are damaged or missing. Excessively scratched or damaged windows and guards must be replaced.
- ONLY authorized personnel should be allowed to operate the machine and CNC control. Improper operation can cause injury, death, machine or control damage, and may void applicable warranties.
- All electrical enclosures and panels MUST be closed and secured at all times except during installation and service. Only qualified electricians and service personnel should have access to these locations. Hazards arising from high voltage electricity and heat exist in the control cabinet, and may exist even after the main disconnect is turned OFF.
- Improperly clamped or fixtured parts; improperly-secured tooling; and broken parts, fixtures, and tooling resulting from machining operations at unsafe feedrates and speeds may result in projectiles being ejected from your machine, even through safety systems such as guards and doors. Always follow safe and reasonable machining practices and follow all safety precautions provided by your tooling and machine manufacturer.
- Ultimate responsibility for safe operation and maintenance of your machine and CNC control rests with shop owners and machine operators. Before performing any work or maintenance all individuals should be thoroughly acquainted with the safe operation of BOTH machine tool AND CNC control.
- Shop owners and operators are responsible for ensuring that shop and machine safety systems such as Emergency Stop and fire suppression systems are present and functioning properly, as required by local codes and regulations.

CNC Control Warning Labels

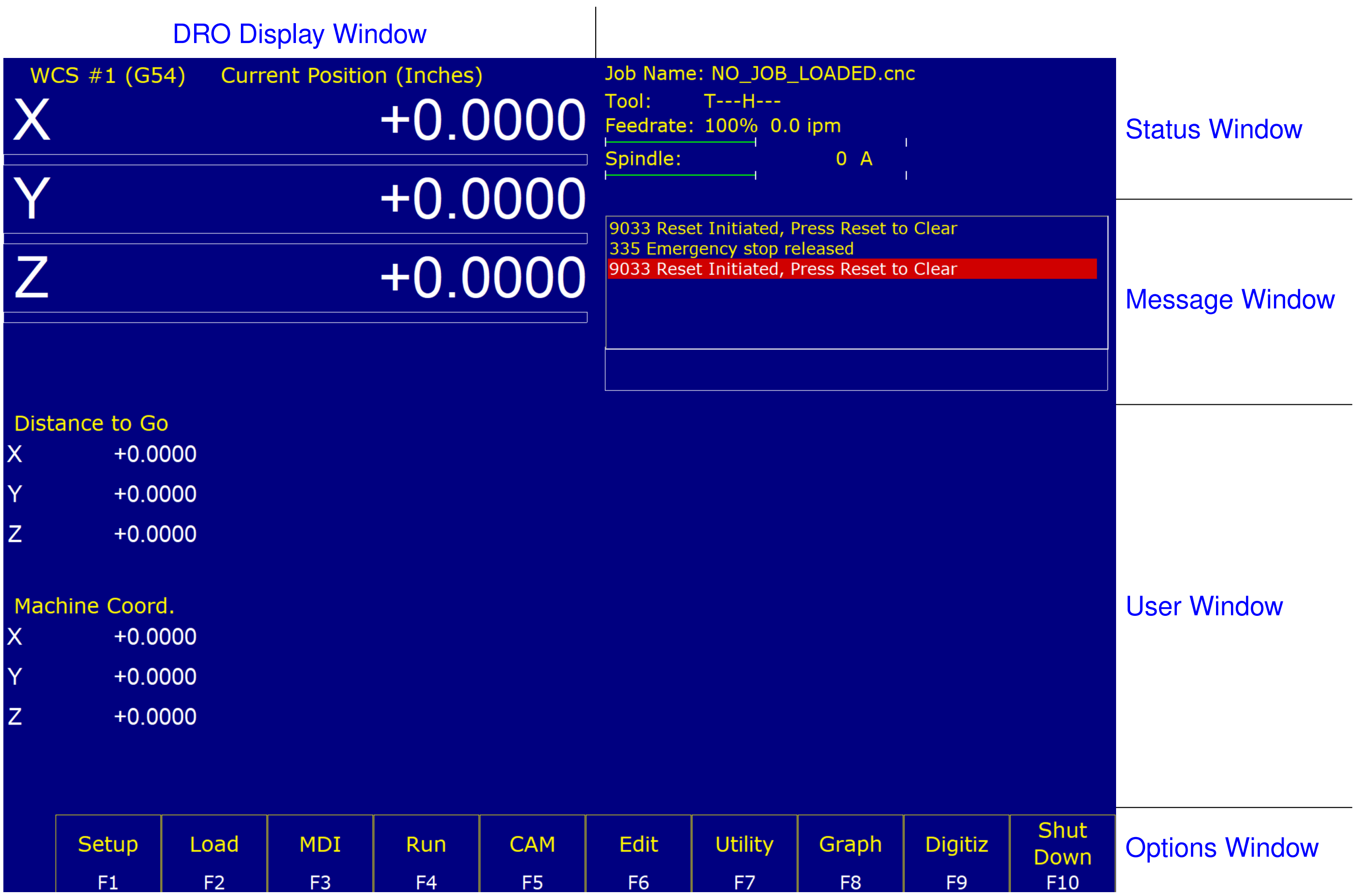
High Voltage Electrocution Hazard.
Death by electric shock can occur.
Turn off and lock out system power before servicing.

High Voltage Electrocution Hazard.
Death by electric shock can occur.
Turn off and lock out system power before servicing.

Contents

1 Introduction

The CNC software display screen is separated into five areas called windows. A sample screen is shown below for reference. The five windows are the DRO display window, the status window, the message window, the options window, and the user window. The information that each window displays is described in detail in the following sections.



1.1 DRO Display

The DRO display contains the digital read out of the current position of the tool. The display is configurable for number of axes and desired display units of measure (see Chapter 15). The bars under each axis are the load meters and represent the amount of power being supplied to the drive for that axis. The display of axis load meters is configured by machine Parameter 143 – see Chapter 15 for specific information.

1.2 Distance-to-Go DRO

The distance to go DRO is located below the main DRO. This display shows the distance to go to complete the current movement. The display of distance to go is controlled by Parameter 143. It can be turned on by using **Ctrl+D**, see "Hot Keys" for more details.

1.3 Status Window

The first line in the status window contains the name of the currently-loaded job file. Below the job name are the Tool Number, Program Number, Feedrate Override, Spindle Speed, and Feed Hold indicators. The Feedrate Override indicator displays the current override percentage set on the Jog Panel. The Feedrate label will turn RED if the rapid override is turned off. If your machine is equipped with a variable frequency spindle drive (inverter), the Spindle indicator will display the current spindle speed. The Feed Hold indicator displays the current status (on/off) of **FEED HOLD**. See Chapter 2 for descriptions of the Feed Hold Button, Feedrate Override Knob, and Spindle controls. For a description of the Program Number, see G65 in Chapter 12 or M98 in Chapter 13.

The Part Cnt and Elapsed Time indicators appear when **CYCLE START** is pressed while a job is running. The Part Count indicator displays the number of times the currently-loaded job has been run. The count increases by increments of one after the completion of a run. If a job is canceled prematurely, the part count will not be incremented. The Part # counter shows the how many parts have been run, with an up/down arrow displayed to indicate the counting direction. See the run menu for more information on the Part Cnt and Part # setting.

The Part Time indicator displays how much time has passed since the CYCLE START button was pressed. The indicator will help you to determine how long it takes to mill a particular part. The timer will not stop until the job is canceled. It will continue to count for optional stops, tool changes, FEED HOLD, etc.

1.4 Message Window

The message window is divided into a message section and a prompt section. The prompt section of the window is the lowest text line in the window and will display prompts to the user. For example, the prompt 'Press CYCLE START to start job' is displayed on the prompt line after power-up. The message section is the top text lines of the message window. This section will display warnings, errors, or status messages. The newest messages always appear on the lowest of the lines. Old messages are shifted up until they disappear off the top of the message window. When old messages scroll out of view, a scroll bar will appear on the right side of the window. When the scroll bar is visible, you may use the up and down arrow keys to view older messages. See Chapter 16 for a description of the CNC software error and status messages.

1.5 Options Window

Options are selected by pressing the function key indicated in the box. For example, on the main screen, pressing the function key **F5 – CAM** selects the CAM option.

1.6 User Window

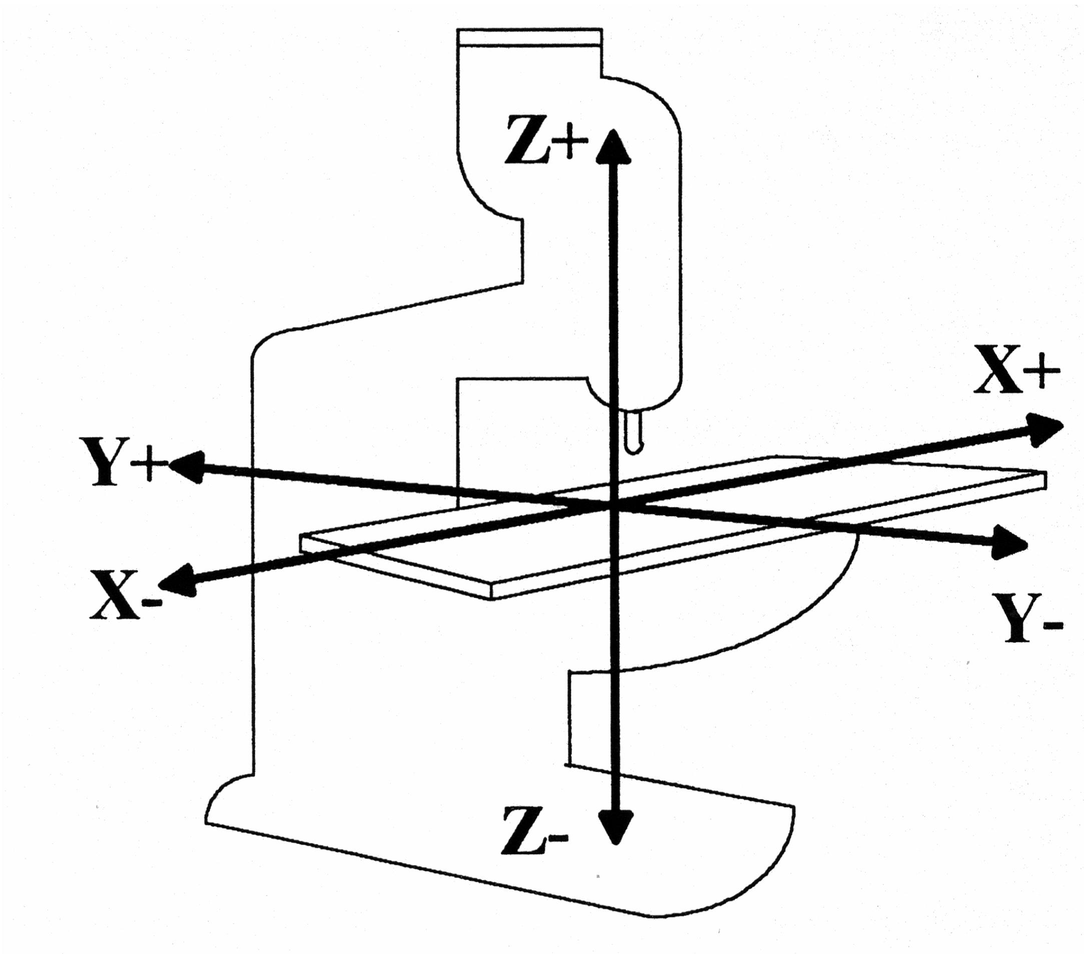
The information contained in this window is dependent upon the operation that the user is performing on the control. If no action is being taken, then the window is empty.

For instance, when the **CYCLE START** button is pressed and a job is processed correctly, up to 11 lines of G codes will be displayed in this window for the user to observe during the Run of the part. All of the part zeros, the tool library setup, and the Digitizing/Probing information are entered by the user into this window.

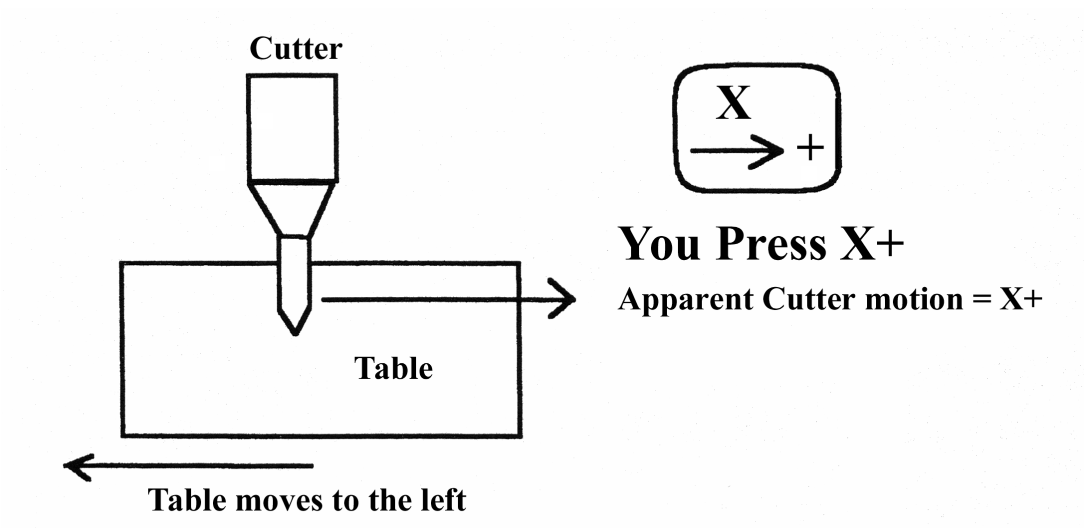
1.7 Conventions

- Bold, capitalized characters represent keystrokes. For example, the A key is written as **A** and the enter key is written as **ENTER**. The "Escape" key is written as **ESC**. Key combinations such as **ALT- D** mean that you should press and hold **ALT** then press **D**.

- All data entry screens in the M-Series Control use **F10 – Save** to save changes.
- Any menu in the M-Series Control can be exited by pressing **ESC**. This will take you back to the previous menu. This also usually discards any changes you have made in that menu.
- All program examples and software use the standard Cartesian coordinate system (see the figure below). If you are facing the mill, the X-axis is defined positive to your right; the Y-axis is defined positive to the mill; and the Z-axis is defined positive upward, perpendicular to the XY plane.



- The direction of motion is defined by the CUTTER motion, not the TABLE motion.



- CW stands for clockwise, and CCW stands for counterclockwise.

Examples of machine motions

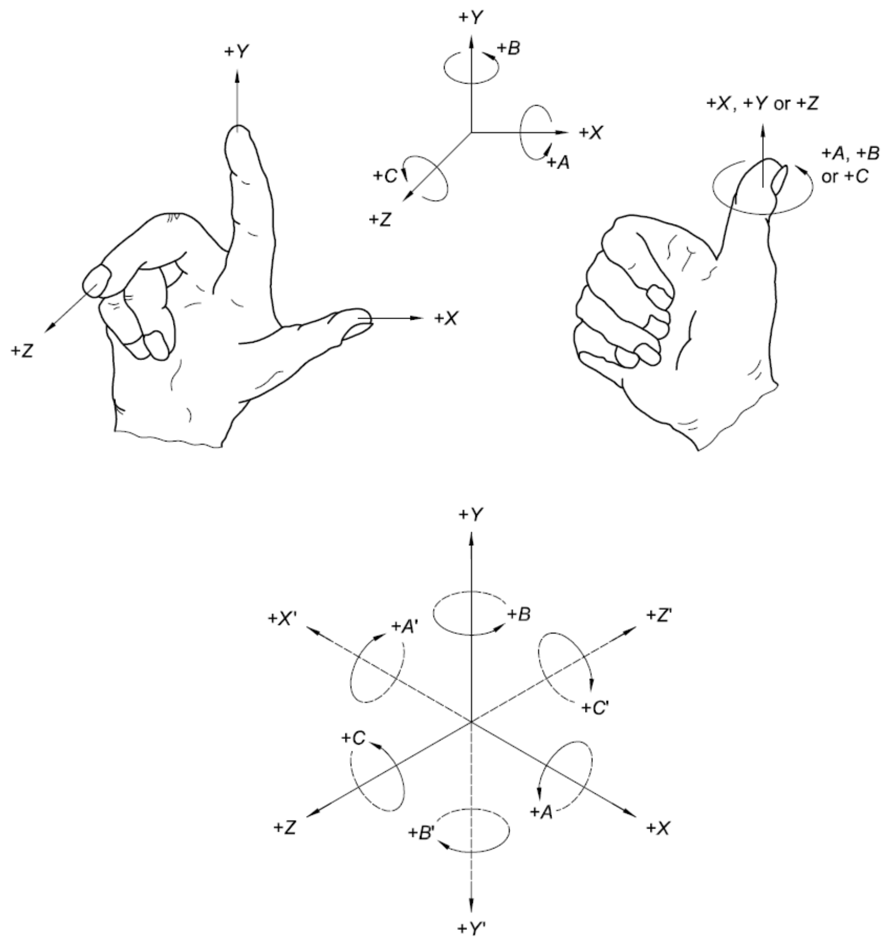
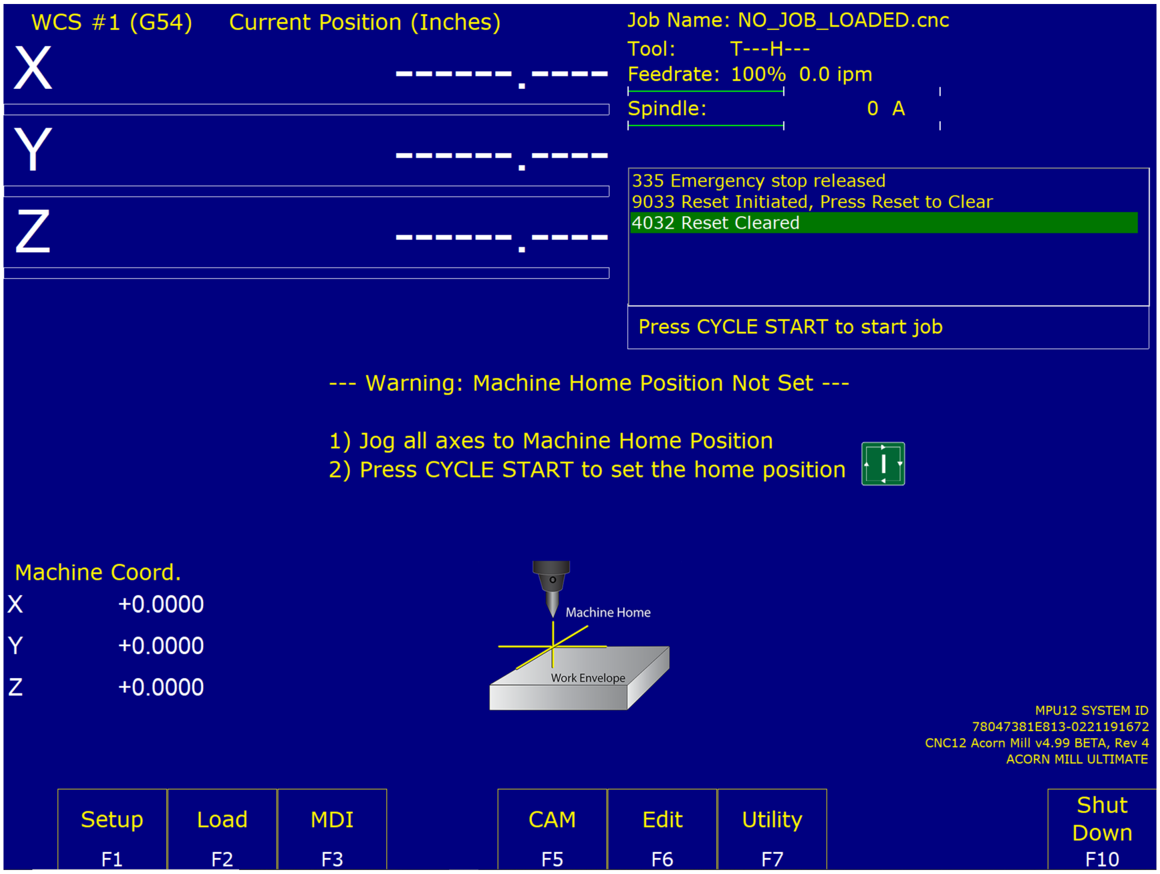


Figure A.1 — Right-hand coordinate system

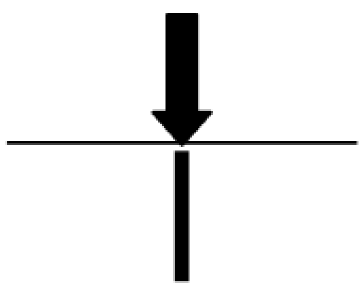
Image courtesy of “ISO 841:2001(E) Industrial automation systems and integration – Numerical control of machines – Coordinate system and motion nomenclature”.

1.8 Machine Home

When the M-Series control is first started, the Main screen will appear as below:

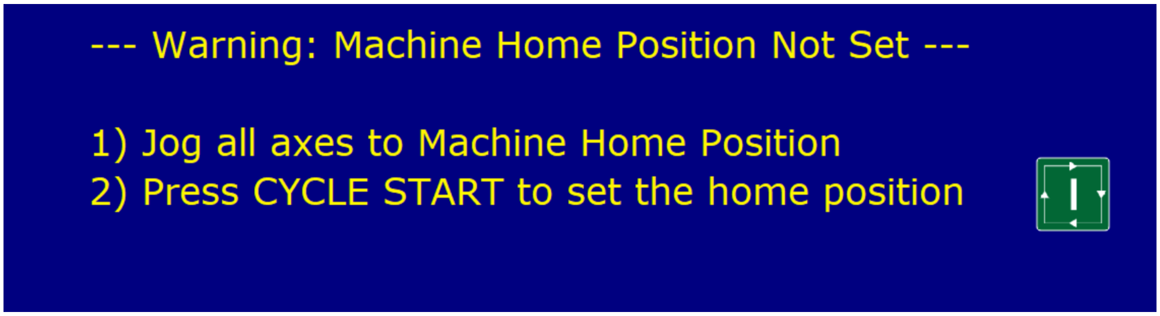


Before you can run any jobs, you must set the machine home position. If your machine has home/limit switches, reference marks, or safe hard stops, the control can automatically home itself. If your machine has reference marks, jog the machine until the reference marks are lined up (see below) before you press **CYCLE START** to begin the automatic homing sequence. The control will execute the G-codes in a file called cncm.hom in the c:/cncm directory. By default, this file contains commands to home Z in the plus direction, then X in the minus, and Y in the plus direction.



Typical Reference Marks

If your machine does not have home/limit switches or safe hard stops, the following message will appear instead.



In this case, you must move the machine to its home position yourself, using either the jog keys or the handwheels. Once all axes are at their home positions, press CYCLE START to set the machine home.

1.9 Mill M and G Codes

This is a summary list of M and G codes. See Chapters 12–13 for more information.

M00 Stop for Operator	G00 – Rapid to Position
M01 Optional Stop for Operator	G01 – Linear Move
M02 Restart Program	G02/G03 – CW/CCW Arc Move
M03 Spindle On Clockwise	G04 – Dwell
M04 Spindle On Counterclockwise	G09 – Decel and Stop
M05 Spindle Stop	G10 – Set Parameter
M06 Tool Change	G17/G18/G19 – XY/ZX/YZ Plane Selection
M07 Mist Coolant On	G20 – Inch Units
M08 Flood Coolant On	G21 – Metric Units
M09 Coolant Off	G22 – Work Envelope On
M10 Clamp On	G23 – Work Envelope Off
M11 Clamp Off	G28 – Return to Reference Point
M25 Move to Z Home	G29 – Return from Reference Point
M26 Set Axis Home	G30 – Return to Secondary Reference Point
M30 M-code for End of Program	G40 – Cancel Cutter Compensation
M39 Air Drill	G41/G42 – Cutter Compensation Left/Right
M91 Move to Minus Home	G43/G44 – Tool Length Compensation +/-
M92 Move to Plus Home	G49 – Cancel Tool Length Compensation
M93 Release/Restore Motor Power	G50 – Cancel Scaling / Mirroring
M94,M95 Output On/Off	G51 – Scaling / Mirroring
M98 Call Subprogram	G52 – Offset Local Coordinate System Origin
M99 Return from Macro or Subprogram	G53 – Rapid Positioning in Machine Coordinates
M100 Wait for PLC bit (Open, Off, Reset)	G54-G59 – Select Work Coordinate System
M101 Wait for PLC bit (Closed, On, Set)	G61 – Modal Decel and Stop
M102 Restart Program	G64 – Smoothing Mode Selection
M103 Programmed Action Timer	G65 – Call Macro
M104 Cancel Programmed Action Timer	G68 – Rotate
M105 Move Minus to Switch	G69 – Cancel Rotate
M106 Move Plus to Switch	G73 – High Speed Peck Drilling (Canned Cycle)
M107 Output Binary Coded Decimal Tool Number	G74 – Counter Tapping (Canned Cycle)
M108 Enable Override Controls	G76 – Fine Bore (Canned Cycle)
M109 Disable Override Controls	G80 – Cancel Canned Cycle
M115,M116,M125,M126 Protected Move Probing	G81 – Drilling and Spot Drilling (Canned Cycle)
M120 Open data file (overwrite existing file)	G82 – Drill with dwell (Canned Cycle)
M121 Open data file (append to existing file)	G83 – Deep hole drilling (Canned Cycle)
M122 Record local position(s) in data file	G84 – Tapping (Canned Cycle)
M123 Record value and/or comment in data file	G85 – Boring (Canned Cycle)
M124 Record machine position(s) in data file	G89 – Boring with dwell (Canned Cycle)
M127 Record Date and Time in a data file	G90 – Absolute Positioning Mode
M128 Move Axis by Encoder Counts	G91 – Incremental Positioning Mode
M129 Record Current Job file path to data file	G92 – Set Absolute Position
M130 Run system command	G93 – Inverse Time Feedrate Mode
M200/M201 Stop for Operator, Prompt for Action	G94 – Cancel Inverse Time Feedrate Mode
M223 Write Formatted String to File	G98 – Initial Point Return
M224 Prompt for Operator Input	G99 – R Point Return
M225 Display Formatted String for a time	G117/G118/G119 – XY/ZX/YZ Plane Selection
M290 Digitize Profile	G173,G174,G176,G181,G182,G183,G184,G185,G189 – Compound Canned Cycles
M300 Fast Synchronous I/O update	G180 – Cancel Canned Cycle
M333 Axis Role Re-assignment	
M1000-M1015 Graphing Color for feed move	

1.10 How to unlock software features or unlock your Control

The following are necessary to unlock software features:

1. Go to the Main screen of the Control software.
2. Press **F7 – Utility** and then **F8 – Import License**.

3. Select your license file from the file browser that appears.
4. Repeat steps 1–3 for each new Unlock.

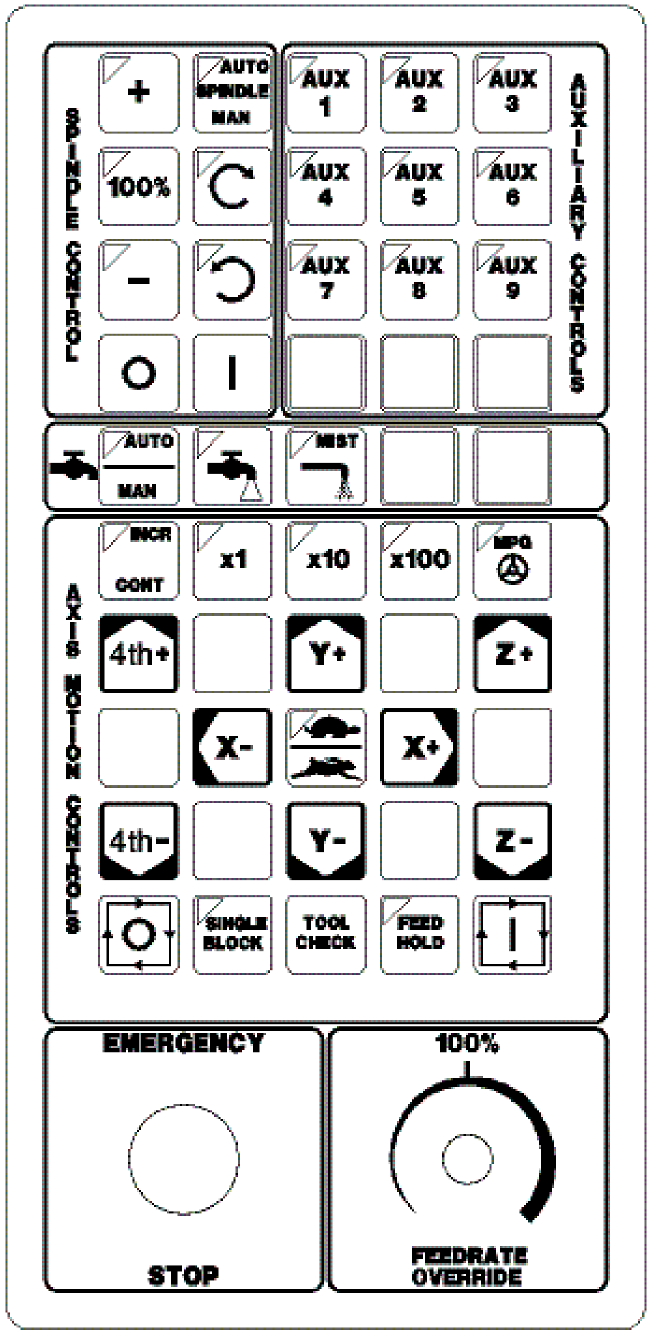
1.11 Skinning

CNC12 gives the user the ability to create a custom interface that can be applied in many different ways. Using the CNC12 C# programming language API, users may write their own software programs that communicate with CNC12 to perform desired tasks, including moving the machine, setting parameters, etc. For more information, see the `CncSkinningDocumentation` folder in the root of the CNC12 installation directory.

Note: When using CNC12 with multiple displays, the software will default to the display located farthest to the right. To override this, right click on the shortcut to CNC12 and click "Properties". Under the "Target" field, add the following text: "–displayx", where "x" is the identification number of the desired display. An error message may appear upon starting the program, but the desired display will be used.

2 Operator Panel

The M-Series operator panel is a sealed membrane keyboard that enables you to control various machine operations and functions. The panel contains momentary membrane switches. The M-Series jog panel can be customized as to the location of various keys. The jog panel displayed in the figure below is representative of a default configuration found on most M-series controls.



2.1 Axis Jog Buttons

X+ X- Y+ Y- Z+ Z- 4^{th}+ 4^{th}-

The yellow X, Y, Z, and 4^{th} keys are momentary switches for jogging each of the four axes of the machine. There are two buttons for each axis (+/-). Only one axis can be jogged at a time.

Note: The jog buttons will not operate if the M-Series CNC software is not running or if a job (a CNC program) is running.

2.2 Slow/Fast

The **Slow/Fast** key is located in the center of the Axis Motion Controls section and is labeled with the turtle and rabbit icon shown to the left. The turtle represents slow jogging mode. When SLOW jog is selected (LED on) and a jog button is pressed, the axis moves at the slow jog rate. If FAST jog is selected, the axis will move at the fast jog rate. See Chapter 15 for information on setting the fast and slow jog rates for each axis.

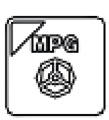
2.3 Inc/Cont

INC/CONT selects between incremental and continuous jogging. Pressing the key will toggle between these two modes. The LED is lit when INC is selected. If CONT jog is selected and an axis jog button is pressed, the axis will move continuously until the button is released.

2.4 x1, x10, x100

Press any one of these keys to set the jog increment amount. The amount you select is the distance the control will move along an axis if you make an incremental jog (x1=0.0001", x10=0.0010", and x100=0.0100"). You may select only one jog increment at a time, and the current jog increment is indicated by the key that has a lit LED. The jog increment you select is for all axes; you cannot set separate jog increments for each axis. The jog increment also selects the distance the control will move along an axis for each click of the MPG handwheel.

2.5 MPG



The MPG is housed in a separate hand-held unit. Press the **MPG** key to set the control jog to respond to the MPG hand wheel (if equipped). When selected, the LED will be on. Select the Jog Increment and desired axis, and slowly turn the wheel. When the LED is not lit, the MPG is disabled and the jog panel is on.

2.6 Single Block

The **SINGLE BLOCK** key selects between auto and single block mode. When the **SINGLE BLOCK** LED is on, the single block mode has been enabled. Single Block mode allows you to run a program line by line by pressing **CYCLE START** after each block. While in block mode, you can select auto mode at any time. While in auto mode with a program running, you cannot select single block mode. Auto mode runs the loaded program after **CYCLE START** is pressed. Auto mode is the default (LED off).

2.7 Cycle Start

When the CYCLE START button is pressed, the M-400/M-39 Control will immediately begin processing the current program at the beginning and will prompt you to press the **CYCLE START** button again to begin execution of the program. After an M0, M1, M2, or M6 is encountered in the program, the message “Press **CYCLE START** to continue” will be displayed on the screen, and the M-400/M-39 Control will wait until you press the **CYCLE START** button before continuing program execution.

WARNING

Pressing CYCLE START will cause the M-Series Control to start moving the axes immediately without further warning. Be certain that you are ready to start the program when you press this button. Pressing the **FEED HOLD** button, **E-STOP** or the **CYCLE CANCEL** button will stop any movement if **CYCLE START** is pressed accidentally.

2.8 Feedrate Override

This knob controls the percentage of the programmed Feedrate that you can use during feedrate cutting moves (i.e. lines, arcs, canned cycles, etc.). This percentage can be from 0% to 200%.

CAUTION

The Feedrate Override knob will not work during tapping cycles (G74 and G84).

2.9 Feed Hold

Feed Hold decelerates the motion of the current movement to a stop, pausing the job that is currently running. Pressing **CYCLE START** will continue the movement from the stopped location.

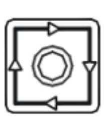
CAUTION

Feed Hold is temporarily disabled during tapping cycles (G74 and G84), and automatic tool changes (M6).

2.10 Tool Check

Press **TOOL CHECK** while no program is running to move the Z-axis to its home position/G28 position. Press **TOOL CHECK** while a program is running to abort the currently-running program. The control will stop normal program movement, pull Z to its home position, clear all M-functions, and automatically display the Resume Job Screen. From the Resume Job Screen, you can change tool settings (height offsets, diameter offsets, etc.) and resume the job with the new tool settings.

2.11 Cycle Cancel



Press **CYCLE CANCEL** to abort the currently-running program. The control will stop movement immediately, clear all M-functions, and return to the Main Screen. It is recommended that you press **FEED HOLD** first before **CYCLE CANCEL**. If you press **CYCLE CANCEL**, program execution will stop; if you wish to restart the program you must rerun the entire program or use the search function. See search function operation in Chapter 3 or Chapter 6.

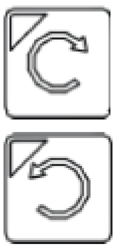
2.12 Emergency Stop

EMERGENCY STOP releases the power to all the axes and cancels the current job immediately upon being pressed. **EMERGENCY STOP** also resets certain faults if the fault condition has been fixed or cleared.

WARNING

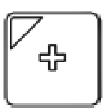
On some machines, vertical axes (such as Z and/or W) may start to move due to gravity pulling them down when motor power is cut due to **EMERGENCY STOP** being pressed.

2.13 Spindle CW/CCW



The **SPINDLE CLOCKWISE/COUNTERCLOCKWISE** keys determine the direction that the spindle will turn if it is started manually. If the spindle is started automatically, the direction keys are ignored and the spindle runs according to the program. The default direction is CW.

2.14 Spindle Speed +

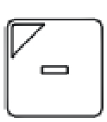


For Auto Spindle mode, pressing this key will increase the spindle speed by 10% of the commanded speed (and is limited by the maximum speed or 200% of commanded speed, whichever is less). For Manual Spindle mode, pressing this key will increase the spindle speed by 5% of the maximum spindle speed (up to the maximum speed). This key's LED turns on if the spindle speed is set above the 100% point.

2.15 Spindle Speed 100%

Pressing this key will set the spindle speed at the 100% point. This is defined as the commanded speed in Auto Spindle mode or 1/2 the maximum spindle speed in Manual Spindle mode. This key's LED will turn on when the spindle is at the 100% point.

2.16 Spindle Speed -

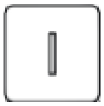


For Auto Spindle mode, pressing this key will decrease the spindle speed by 10% of the commanded speed (limited to 10% of the commanded speed). For Manual Spindle mode, the spindle speed is decreased by 5% of the maximum spindle speed (down to 5% of the maximum spindle speed). The LED turns on if the spindle speed is set below the 100% point.

2.17 Spindle Auto/Man

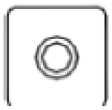
This key selects whether the spindle will operate under program control (automatic) or operator control (manual). When the LED is lit, the spindle is under automatic control. When the LED is off, the spindle is under manual control. Pressing the **SPINDLE (AUTO/MAN)** key will toggle the spindle between Automatic and Manual modes. The default is AUTO mode.

2.18 Spin Start



Press the **SPIN START** key when manual spindle mode is selected to cause the spindle to start rotating. Press **SPIN START** when automatic mode is selected to restart the spindle if it has been paused with **SPIN STOP**.

2.19 Spin Stop

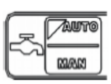


Press the **SPIN STOP** key when manual spindle mode is selected to stop the spindle. Press **SPIN STOP** when automatic spindle mode is selected to pause spindle rotation. The spindle can be restarted with **SPIN START**.

NOTICE

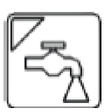
SPIN STOP should only be pressed during **FEED HOLD** or when a program is NOT running.

2.20 Coolant Auto\Manual



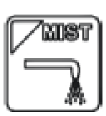
This key will toggle between automatic and manual control of coolant. In automatic mode, M7 (Mist) and M8 (Flood) can be used in G-code programs to select the coolant type to be enabled. In manual mode, flood coolant and mist coolant are controlled by separate keys.
Note: When switching from automatic to manual mode, both flood and mist coolants are turned off automatically.

2.21 Coolant Flood



In manual coolant control mode, flood coolant can be toggled off and on by pressing this key. The LED will be on when flood control is selected in either automatic or manual mode.

2.22 Coolant Mist



In manual coolant control mode, mist coolant can be toggled off and on by pressing this key. The LED will be on when mist control is selected in either automatic or manual mode.

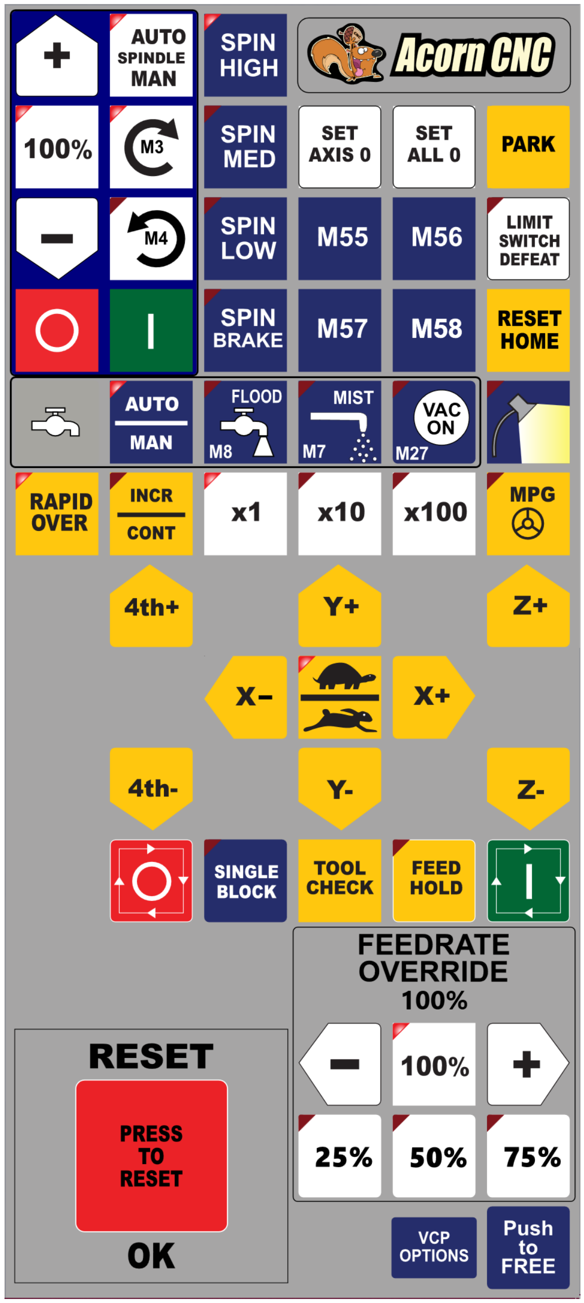
2.23 Auxiliary Function Keys (AUX1 – AUX12)

The M-Series jog panel has 12 auxiliary keys (9 labeled), some of which may be defined by customized systems.

2.24 Notes About Operator Panels

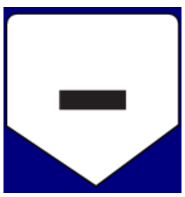
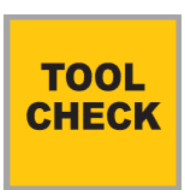
The behavior of the control system in response to the functions listed above for the M-Series jog panel is dependent upon optional software settings, the PLC program, machine parameters, and hardware wiring of the system. It is possible that the functioning explained in this chapter does not apply to a particular control system, or that it may differ in some aspects.

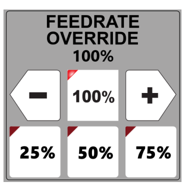
2.25 VCP Introduction

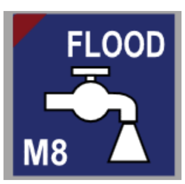


The Virtual Control Panel (VCP) allows the user to use a mouse and/or a touch screen monitor to activate the CNC control operator interface panel. The VCP has been designed from the ground up with the intention to allow users, re-builders, and OEMs alike a simple way to change the look, feel, and function of the VCP. CNC12 will automatically install the default Centroid VCP skin, as seen above. Users can simply use the default VCP "as is", but if you wish to modify the default VCP then please read the VCP Manual found here.

Legend	Function	Description
	4th+ Jog	Jogs the 4th axis positively.
	4th- Jog	Jogs the 4th axis negatively.
	Toggle Auto Coolant	Toggles coolant mode between auto and manual.
	Toggle Spindle Auto/Manual	Toggles between automatic and manual spindle operation mode
	Cycle Start	Same as Cycle Start.
	Cycle Cancel	Same as Cycle Cancel.

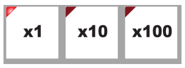
Legend	Function	Description
	Spindle Override Percentage	Displays the percentage of the default spindle speed that the spindle is currently operating at.
	Spindle Override +1%	Increase the spindle override by 1% while held.
	Spindle Override -1%	Decrease the spindle override by 1% while held.
	Feed Hold	Temporarily pauses the feedrate.
	Single Block	Selects Single Block Mode.
	Tool Check	Performs a tool check.

Legend	Function	Description
	VCP Options	Allows the user to edit VCP settings.
	Push to Free	Used to unpin the VCP window, allowing the user to move it around their screen
	Emergency Stop	Same as Emergency Stop.
	Increase/ Decrease Feedrate Override	Increase/Decrease feedrate override by 1% while held.
	Incremental/ Continuous Jog Selection	Toggles incremental or continuous jog mode.
	Toggle Work Light	Toggles the work light between on and off positions.

Legend	Function	Description
	Limit Switch Defeat	Overrides the limit switches when active.
	Selects CW Spin	Selects CW Spin direction in manual mode.
	Selects CCW Spin	Selects CCW Spin direction in manual mode.
	Toggle Mist Coolant	Toggles Mist coolant if in manual mode.
	Toggle Flood Coolant	Toggles Flood coolant if in manual mode.
	Vac On	Toggles the vacuum between on and off positions.

Legend	Function	Description
	M Functions	Used to toggle M functions.
	Toggle MPG	Toggles between the MPG and jog panel.
	Park	Parks the machine in its current position.
	Rapid Override	Toggles rapid jog movement override.
	Reset Home	Resets the home values that are currently set.
	Set All 0	Sets all axes to zero values.

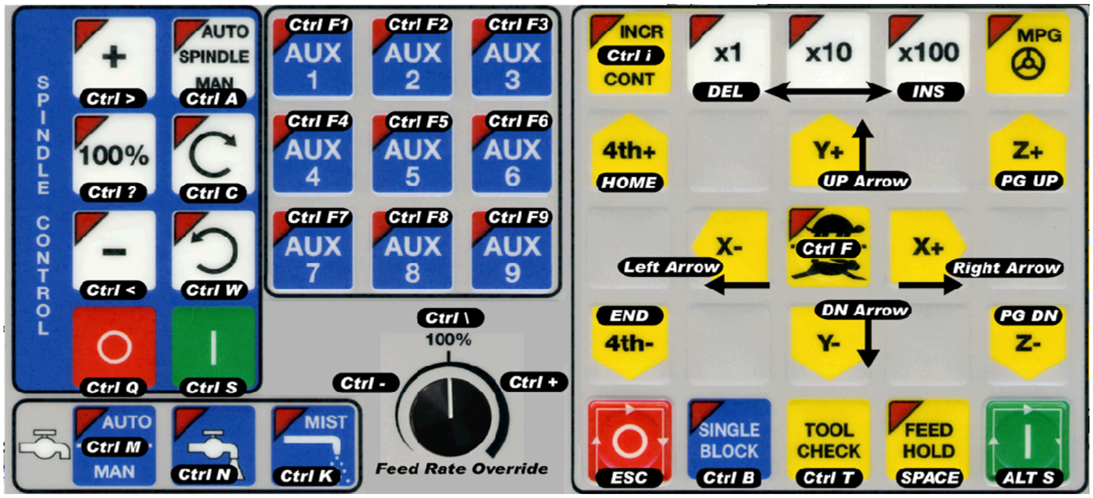
Legend	Function	Description
	Set Axis 0	Sets the currently-selected axis to a value of zero.
	Slow/Fast	Toggles between slow and fast jogging modes.
	Spindle Brake	Toggles the spindle brake.
	Spindle Speed	Allows the user to select the spindle speed (high, medium, and low).
	Spin Start	Starts spindle in selected direction if in manual mode.

Legend	Function	Description
	Spin Stop	Stops spindle regardless of auto or manual mode.
	Decrease/Increase Jog Increment	Decreases/increases current jog increment to the next available increment.
	X+ Jog	Jogs the X axis positively.
	X- Jog	Jogs the X axis negatively.
	Y+ Jog	Jogs the Y axis positively.
	Y- Jog	Jogs the Y axis negatively.
	Z+ Jog	Jogs the Z axis positively.

Legend	Function	Description
	Z- Jog	Jogs the Z axis negatively.

2.26 Keyboard Jog Panel

The PC keyboard may be used as a jog panel. Press **Alt-J** to display and enable the keyboard jog panel. The jog panel appears as shown below:



Some controls, such as coolant on/off, spindle on/off, feedrate, and spindle override, will work without the "jog panel"being displayed. For full functionality (and jogging) of the keyboard jog panel, the "jog panel" must be displayed on the screen. To enable keyboard jogging, parameter 170 must be set to "1".

The status window in the upper right corner of the screen displays the jogging mode (continuous/incremental), incremental step size, and jog speed (fast/slow). In continuous mode, the jog keys start movement when pressed and movement and stop when you release the key. In incremental mode, the axis will move the indicated incremental step amount.

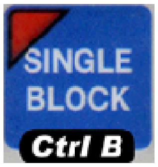
As shown in the picture above, the jog keys are located in the cursor key block to the right of the main keyboard and to the left of the numeric keypad. If a jog key controls an axis, it will be overlaid with the axis symbol ("X", "Y", etc.) The jog keys are the **Arrow Keys**, **Page Up**, and **Page Down**.

The remaining keys are described below:

Legend	Key(s)	Function	Description	Availability (Notes)
–	Alt J	Start/Exit Panel	Invokes or exits the jog panel.	Always, with few exceptions.
	Alt S	Cycle Start	Same as Cycle Start.	Always, with few exceptions.

Legend	Key(s)	Function	Description	Availability (Notes)
	Esc	Cycle Cancel	Same as Cycle Cancel.	During a job run; Otherwise, Esc is used to exit menus.
	Ctrl F1-F12	Aux 1–12	Executes the corresponding Aux function and signals the PLC. A custom PLC program is required to act upon jog panel signals.	Always, with few exceptions.
	Ctrl M	Toggle Auto Coolant	Toggles coolant mode between auto and manual.	Always, with few exceptions.
	Ctrl N	Turns Flood Coolant	Toggles Flood coolant if in manual mode.	Always, with few exceptions.
	Ctrl K	Toggle Mist Coolant	Toggles Mist coolant if in manual mode.	Always, with few exceptions.
	Ctrl +, Ctrl –	Increase/ Decrease Feedrate Override	Increase/Decrease feedrate override by 1% while held.	Jog panel, job run, graphing, and some other times.
	Ctrl C	Selects CW Spin	Selects CW Spin direction in manual mode.	Always, with few exceptions.

Legend	Key(s)	Function	Description	Availability (Notes)
	Ctrl W	Selects CCW Spin	Selects CCW spin direction in manual mode.	Always, with few exceptions.
	Ctrl A	Toggle Spindle Auto/Manual	Toggles between automatic and manual spindle operation mode	Always, with few exceptions.
	Ctrl S	Spin Start	Starts spindle in selected direction if in manual mode.	Always, with few exceptions.
	Ctrl Q	Spin Stop	Stops spindle regardless of auto or manual mode.	Always, with few exceptions.
	Ctrl $>$	Spindle Override +1%	Increase the spindle override by 1% while held.	Always, with few exceptions.
	Ctrl $<$	Spindle Override -1%	Decrease the spindle override by 1% while held.	Always, with few exceptions.
	Ctrl T	Tool Check	Performs a tool check.	Always, with few exceptions.

Legend	Key(s)	Function	Description	Availability (Notes)
	Ctrl I	Incremental/ Continuous Jog Selection	Toggles incremental or continuous jog mode.	Available most times that jogging is available.
	Ctrl B	Selects Single Block Mode	Selects Single Block Mode	Always, with few exceptions.
	Delete/Insert	Decreases/Increases Jog Increment	Decreases/increases current jog increment to the next available increment.	Always, with few exceptions.
	Right Arrow	X+ Jog	Jogs the X axis positively.	With on-screen jog panel displayed.
	Left Arrow	X- Jog	Jogs the X axis negatively.	With on-screen jog panel displayed.
	Up Arrow	Y+ Jog	Jogs the Y axis positively.	With on-screen jog panel displayed.
	Down Arrow	Y- Jog	Jogs the Y axis negatively.	With on-screen jog panel displayed.
	Page Up	Z+ Jog	Jogs the Z axis positively.	With on-screen jog panel displayed.

Legend	Key(s)	Function	Description	Availability (Notes)
	Page Down	Z- Jog	Jogs the Z axis negatively.	With on-screen jog panel displayed.
	Home	4th+ Jog	Jogs the 4th axis positively.	With on-screen jog panel displayed.
	End	4th- Jog	Jogs the 4th axis negatively.	With on-screen jog panel displayed.

Note: To avoid unexpected movement, keyboard jogging will disable and re-enable itself when leaving and entering the main menu. Keyboard jogging can still be enabled in any menu by pressing **ALT+J** (even after being disabled by CNC12). For instance, if keyboard jogging is active and the user navigates to the CNC12 parameters menu, keyboard jogging will be suppressed while in that menu and reactivated when back in the main menu of CNC12.

2.27 MDI and the Keyboard Jog Panel

Many of the keys used by the keyboard jog panel are also possible commands to use in MDI. To use the keyboard jog panel functions in MDI, you must press **Alt J**. You may jog, use the handwheels, or any other jog panel function. Press **Alt J** or **Esc** to return to MDI.

2.28 Keyboard Shortcut Keys

A computer-style keyboard is supplied with most systems. This keyboard can be used as a jog panel. The keyboard jog panel has many "hot keys". Hot keys are keys that can be used at almost any time, with few exceptions. Some menus may prohibit their use. Hot keys used in the CNC software are listed below:

2.28.1 `ALT+D` – WCS/Machine Coordinates

Switches the DRO display between current WCS position and current machine position.

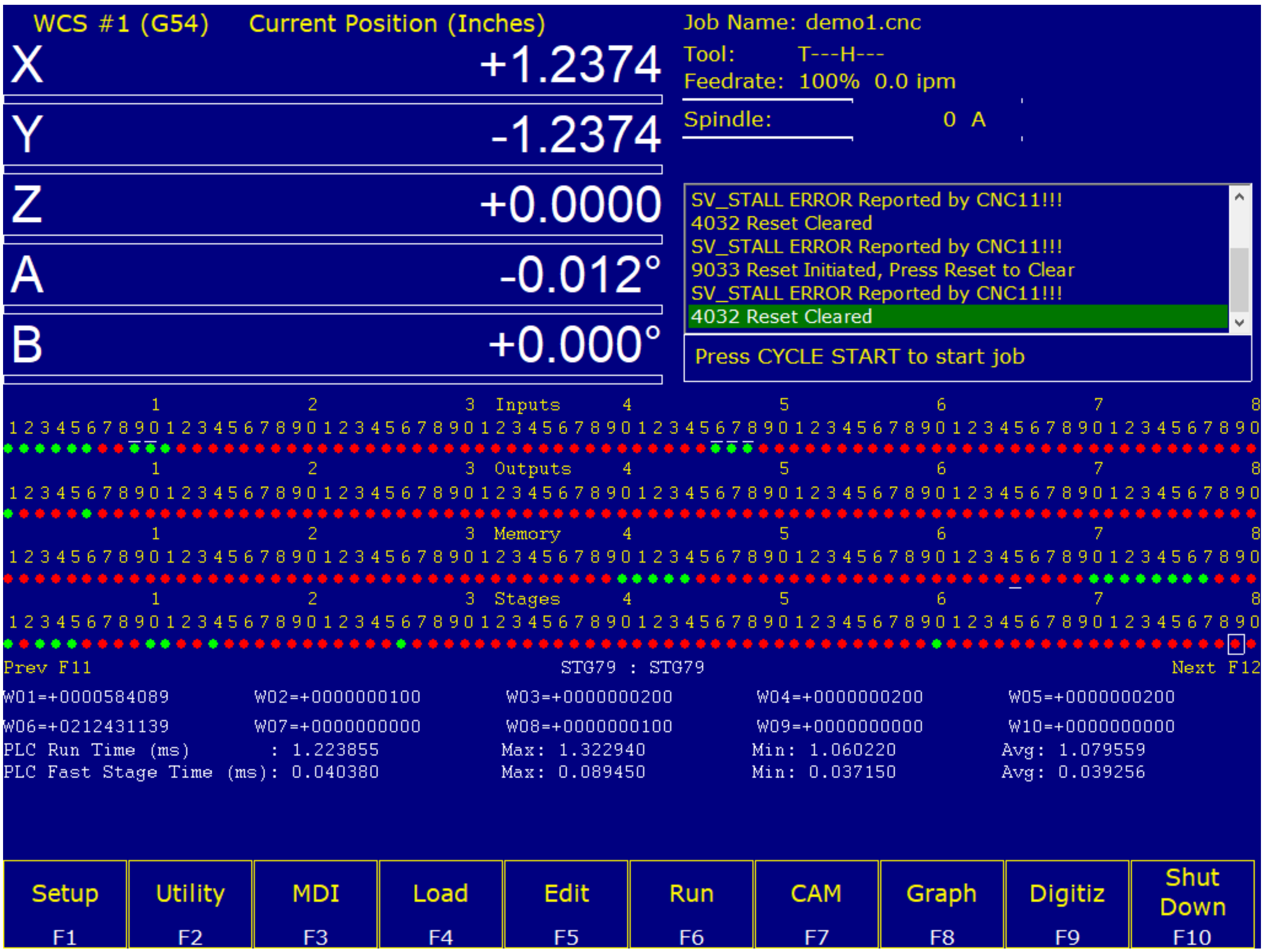
2.28.2 `ALT+E` – Generate Screenshot

If the value of Parameter 389 is greater than 0, pressing the `ALT+E` key combination generates a screenshot. It is saved as `screenshot-nnn.png`, where `nnn` is a three-digit number starting at `000` and incrementing upward each time a screenshot is taken. Note that when you restart the CNC software, the numbering starts over at `000`.

2.28.3 `ALT+I` – Live PLC I/O

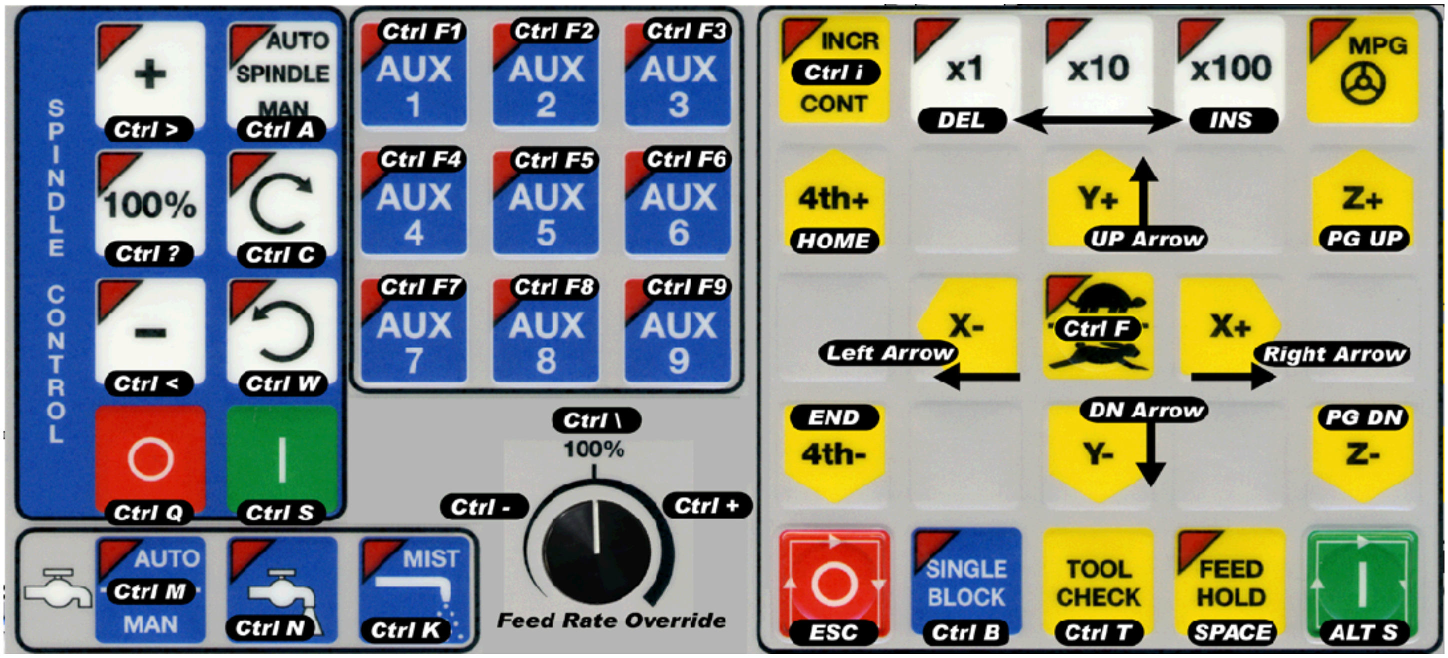
Pressing `ALT+I` brings up the Live PLC I/O display screen (see the image below). In this screen, you may view the real-time status of all the different inputs, outputs, etc. Use the arrow, `F11`, and `F12` keys to navigate the Live PLC I/O screen. With proper PLC support, pressing `CTRL+ALT+I` allows you to toggle the value of inputs 1 through 80. Similarly, pressing `CTRL+ALT+F` allows you to toggle outputs.

To have a more advanced view of the Live PLC I/O, similar to the image below, perform the following: enter the utilities menu (`F7`), enter the Acorn wizard menu (`F10`), select "CNC Control" under the "Preferences" header, and toggle the option "Enable Simple PLC Diagnostic as default."



2.28.4 `ALT+J` – Keyboard Jog Panel

Pressing `ALT+J` brings up the keyboard jog panel, which allows you to perform jog panel functions with the keyboard. When `ALT+J` is pressed, a new window pops up containing a bitmap image of the jog panel overlaid with labels to show which keyboard keys simulate which jog panel key press. The print button allows an image of the keyboard jog panel to be printed out.



2.28.5 ALT+K – ATC Bin

Press ALT+K to display the current ATC bin.

2.28.6 ALT+L – ATC Putback Location

Press ALT+L to display the current ATC putback location.

2.28.7 ALT+M – Run MDI

Press ALT+M to run MDI.

2.28.8 ALT+P – Live PID Display

Press ALT+P to display the live PID screen, where you can view current axis positioning information, as seen in the image below.

Axis	Error	Sum	Delta	PID Out	Abs Pos	Max Error
X*	0	0	0	0	-3489769	68
Y*	0	0	0	0	236737	-70
Z*	0	0	0	0	0	0
A*	0	0	0	0	660258	0
B*	0	0	0	0	41	0
N	0	0	0	OFF	0	0
N	0	0	0	OFF	0	0
N	0	0	0	OFF	0	0

2.28.9 `ALT+S` – Cycle Start

Press `ALT+S` as an alternative to the cycle start button.

2.28.10 `ALT+T` – Temperature Display

Press `ALT+T` to display the current temperatures for each axis in the message window.

2.28.11 `ALT+V` – Display CNC Software Version Info

Press `ALT+V` to display CNC12 software version information (the same information found when pressing F1 from the main menu).

2.28.12 `ALT+1`, `ALT+2`, . . . , `ALT+0` – Select WCS

Press `ALT+1`, `ALT+2`, . . . , `ALT+0` to cycle through the first ten work coordinate systems.

2.28.13 `ALT+-` – Select Previous WCS

Press `ALT+-` to select the previous WCS.

2.28.14 `ALT+=` – Select Next WCS

Press `ALT+=` to select the next WCS.

2.28.15 `ALT+F10` – Exit CNC12

Press `ALT+F10` (in the utility menu only) to exit CNC12.

2.28.16 `CTRL+D` Swap DRO and Distance-to-Go DRO

Press `CTRL+D` to swap the positions of the DRO and the Distance-to-Go DRO.

2.28.17 `CTRL+E` – Launch PLC Detective

Press `CTRL+E` to launch the PLC Detective application.

2.28.18 `CTRL+H` – Enable G-Code Display

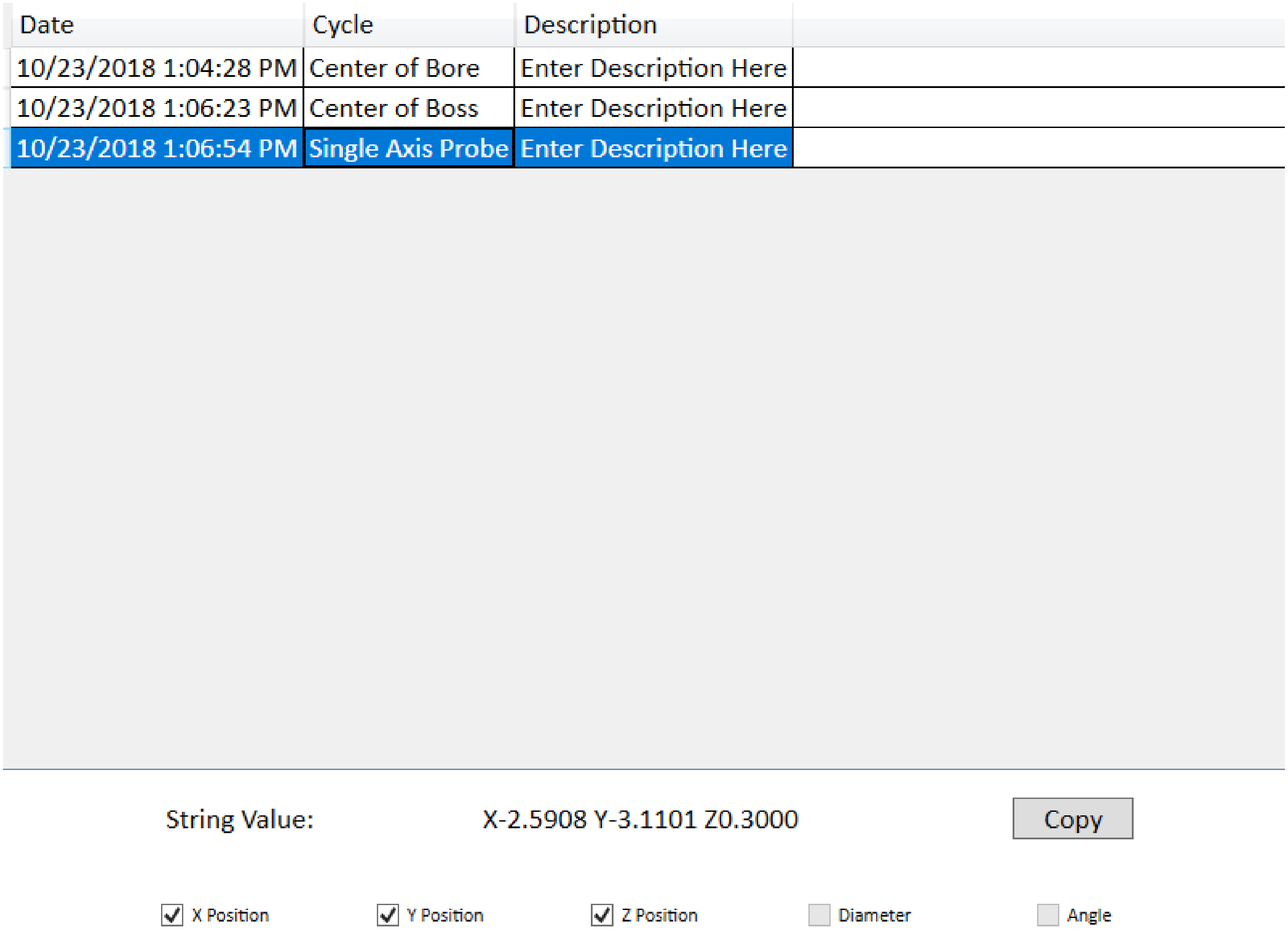
If a job is running and the G-Code display is hidden (perhaps behind the Live PLC I/O or Live PID display screens) press `CTRL+H` to show the G-Code display.

2.28.19 `CTRL+I` – Save PLC state to file.

If you are in the Live PLC I/O display screen (accessible by pressing `ALT+I`), then pressing `CTRL+I` will print the current PLC I/O state to a file titled `plcstate.txt`.

2.28.20 `CTRL+Q` – Probing Cycles History

If you have run a probing cycle (found in the F9 Digitize → F4 Probe screen), then the probing cycle has been recorded so that you may access the positions you probed. Press `CTRL+Q` to view this history. A window will pop up (as seen in the picture below) where you can enter a description, delete any probing history you don't wish to keep, or copy the positions of that probing cycle to the clipboard. Click anywhere outside of that window or press `CTRL+Q` again to close the probing cycles history.



2.28.21 `SHIFT+F1` – Switch to Old Style Graphics Backplot

When you are in the accelerated backplot, press `SHIFT+F1` to switch to the old style graphics backplot that does not use OpenGL.

2.28.22 `SHIFT+F2` – Erase Log File

From the Utility → Logs → Errors (or Stats) screen, press `SHIFT+F2` to erase the log file. You will be presented with a confirmation dialog before the log file is deleted.

2.28.23 `CTRL+ALT+X` – Go to Shutdown Screen

From the main menu, pressing `CTRL+ALT+X` will take you to the CNC12 shutdown screen.

3 CNC Software Main Screen

F1 – Setup: Used to set part zeroes, set or change tool offsets, and change the control configuration.

F2 – Load: Use this menu to load a job.

F3 – MDI: The MDI menu allows you to run a single-line command, such as: G1, X2, Y3, or F20

F4 – Run: Use the Run menu to search and run a job from a specific line, resume a job after it has been canceled, or to change the way a job runs.

F5 – CAM: Use the CAM menu to program parts.

F6 – Edit: Brings up a G-code (text) editor that allows you to edit the currently-loaded job.

F7 – Utility: View available software options, backup part and configuration files, create new directories, and import/export files to and from external locations.

F8 – Graph: Graphs the toolpath of the currently-loaded part program.

F9 – Digitize: Displayed only if the Digitizing option has been purchased. Used to Digitize (reverse-engineer) parts.

F10 – Shut down: Power off control. Shutting down your machine without using this menu may damage your control.

3.1 F1 – Setup Menu

F1 – Part: This key displays the Part Setup menus that are explained in Chapter 4.

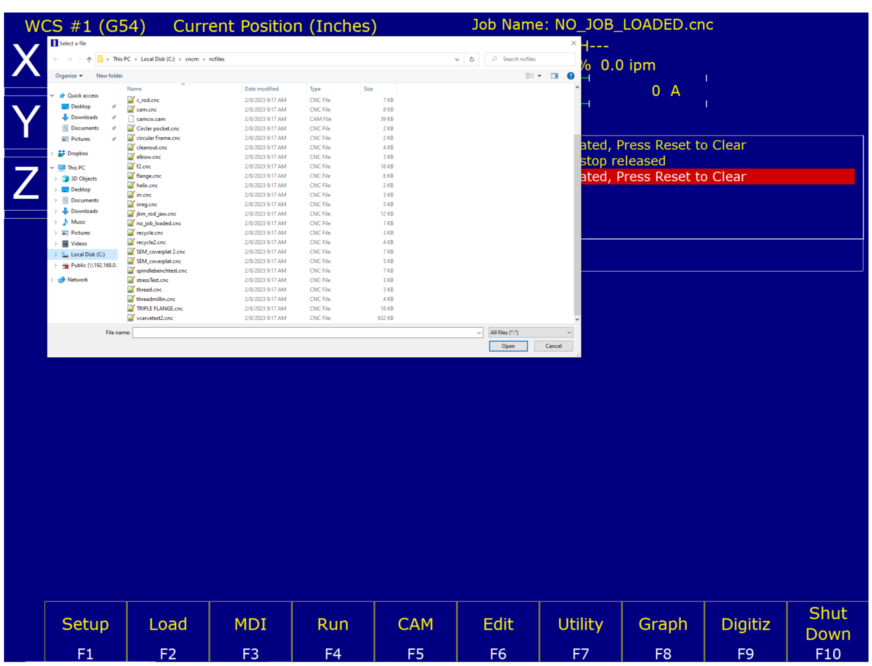
F2 – Tool: This key displays the Tool Setup menus that are explained in Chapter 5.

F3 – Config: This key displays the Configuration menu that is explained in Chapter 15.

F4 – Feed: This key displays the Feed menu that is discussed in Chapter 6.

F8 – Smoothing Setup: This gives you access to the Smoothing Setup Menu which provides a simplified way of choosing parameters for the Smoothing module. See Smoothing Configuration Parameters.

3.2 F2 – Load Job Menu



3.3 F3 – MDI

MDI mode allows you to directly enter M and G-codes one line at time. After entering the M and Gcodes you wish to run, press cycle start to have the controller execute the command. When the command has finished executing the command, it will prompt you for another line. When you are finished entering commands, press **ESC**.

Navigation through previous commands is possible by pressing the ↑ **UP ARROW** key and the ↓ **DOWN ARROW** key. Once selected, you may modify the command using the ← **LEFT** or → **RIGHT** arrow key. Press the **ENTER** key or **Cycle Start** to execute.

Examples:

Block? G92X0Y0 ;Set the current XY position to 0,0

Block? M92 /Z ;Move the Z to the positive limit.

Block? M26 /Z ;Set the current Z position as Z home.

3.4 F4 – Run Menu

F2 – Search: You can resume a job by searching for the line, tool, or a block number.

F3 – Repeat: Toggles Job Repeat. Will repeat the current program when a job is finished.

F4 – Block Skips: Turns on and off block skips in part programs.

F5 – Single Block: Program runs in single block mode when turned on.

F6 – Stops: Turns on and off optional stops (M01) in part programs.

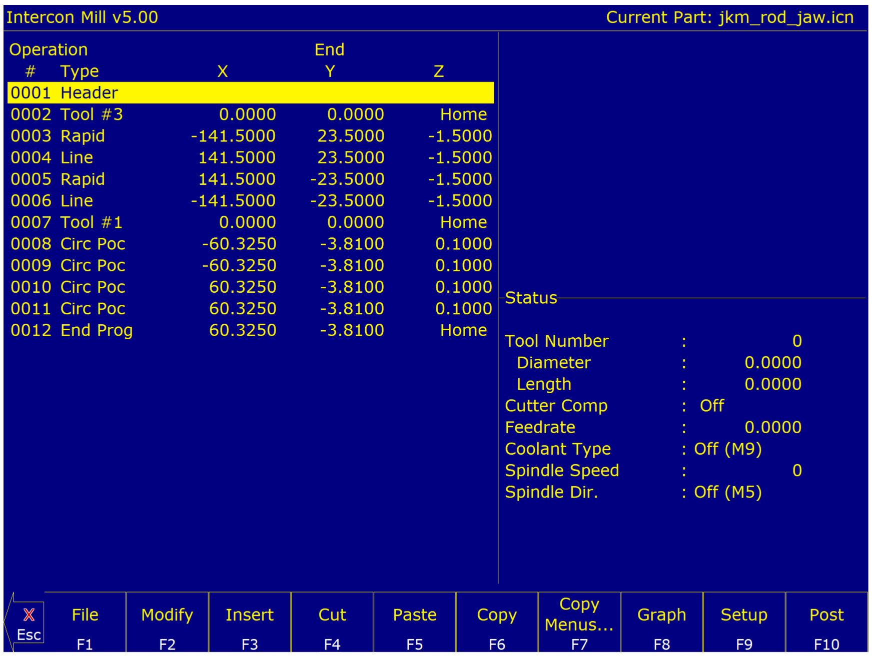
F8 – Graph: Graphs tool path of currently-loaded program

F9 – Rapid: Turns on and off the rapid override function

F10 – RTG: Turns on and off Run Time Graphics

Note: For more information on these options, please see Chapter 6.

3.5 F5 – CAM



Choose **F5 – CAM** to program parts. The default part programming system is Intercon. Intercon is a conversational programming system that allows you to quickly and easily create a part program. Intercon features many easy-to-use canned cycles for most common machining operations such as: rectangular, circular and irregular pockets, pockets with islands, bolt hole circles, frames, thread milling operations, and much more. When you are finished programming your part in Intercon, exit to the M-Series Control Main Screen and the posted Intercon program will be automatically loaded and ready to run. For more information on Intercon, see Chapter 10.

Note: The screen can be customized with additional F Keys.

3.6 F6 – Edit

This key causes the control to load the current job into a text editor for viewing and/or editing.

When editing, care must be taken to save the file and exit the text editor before running the file (the current job). Modifying a file that is currently running as the current job is dangerous and will cause unexpected results. It is best practice not to edit any files while the machine is moving.

WARNING Editing a file (modifying and saving) **while the machine is moving** can cause personal injury or machine damage.

Also, note that the C:\CNCM directory contains configuration files and binary data. DO NOT edit these files. Doing so can cause loss of data and serious malfunctions.

WARNING Do not edit configuration data located in the C:\CNCM directory. Doing so can cause personal injury or machine damage.

3.7 F7 – Utility

From the utility menu, you can view available software options, perform diagnostics, backup part and configuration files, create new directories, and import or export files to and from external locations.

For further information, please see Chapter 7.

F2 – Restore Report: Update your control's configuration with a report.zip file.

F5 – Color Picker: Use this to change from the default Centroid Classic Color Scheme.

F6 – User Maint: Perform user maintenance.

F7 – Create Report: Generates a backup of system configuration files called report.zip.

F8 – Options: Shows software plugins and software level information.

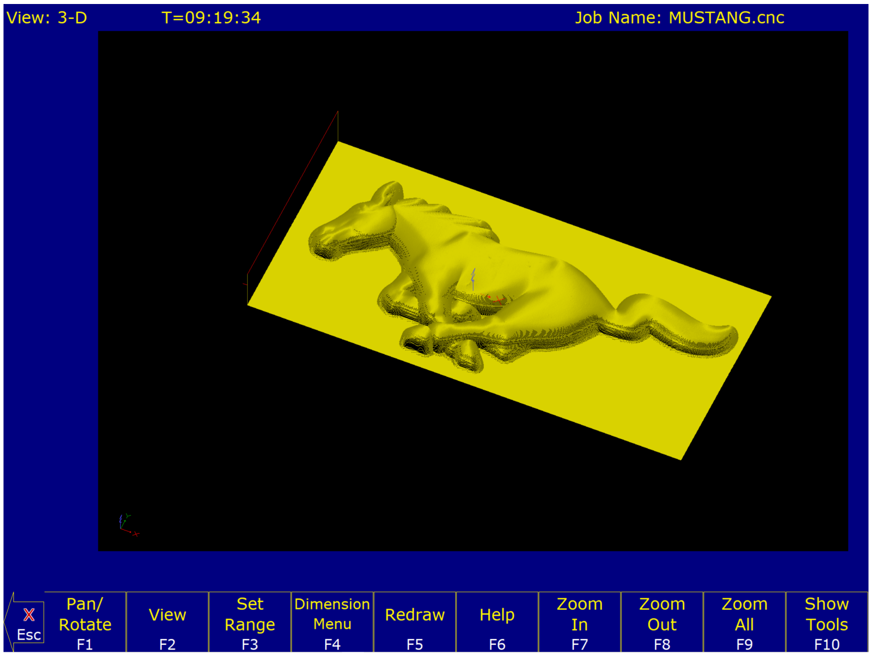
F9 – Logs: Shows the messages and errors that have been logged by the control.

3.8 F8 – Graph

In addition to the Main Screen, the Graph feature can be accessed from other menus like the Load Job Screen and the various Run Job menus. Use the Graph feature to show a tool path of the current program loaded. Pressing Cycle Start while in this screen displays the tool path as it is moving along the graph.

Accelerated Graphics Backplot

Accelerated Graphics Backplot is a tool path graphics display. This option is enabled by default.



F1 – Pan/Rotate: Press this key to change the behavior of the keyboard arrow keys. Normally, they will pan (scroll) around the drawing. After pressing this key, the arrow keys will control rotation instead. When in rotation mode, an axis indicator is drawn to mark the center of rotation.

F2 – View: Press this key to change the planar view of the part. The view is indicated by a TOP, RIGHT, or FRONT label shown at the top of the screen.

F3 – Set Range: Press this key to select which lines of G-code to display. Only lines that fall within the range you specify will be drawn.

F4 – Dimension Menu: Press this key to access the following sub-menu of options:

F1 – Prev Line: Press this to walk forward to the next G-code line and graphically highlight it. If this G-code line contains movement, the Start and End points will be displayed at the bottom of the screen.

F2 – Next Line: Press this to walk backward to the previous G-code line and graphically highlight it. If this G-code line contains movement, the Start and End points will be displayed at the bottom of the screen.

F3 – Go To Line: Press this key to graphically highlight a particular G-code line whose line number you specify. If this G-code line contains movement, the Start and End points will be displayed.

F4 – Measure: Use this feature to measure between any 2 selected points. To do this, use a mouse to move the pointer over to the first point and press **F4 – Measure** to anchor the point. Then, use the mouse to move the pointer to the second point. As you move the mouse towards the second point, you will notice the Offset and Measurement display changing dynamically as you move the mouse. Also, you may notice some "snap to" effects as you move the pointer closer to the start and end points of entities that make up your program.

F5 – Redraw: Press this key to redraw the part slowly. This can be useful for visualizing the movements the machine will make. While the display is being redrawn, you can use the feedrate override knob to adjust the rate at which it is being drawn. If you don't have a feedrate override knob, the + and – keys can be used to adjust the rate as well. Pressing F5 again will cancel this mode.

F6 – Hide Rapids: Press this key to hide rapid movements. Press it again to show them.

F7 – Zoom In: Press these keys to zoom into the part relative to the center of the screen.

F8 – Zoom Out: Press these keys to zoom away from the part relative to the center of the screen.

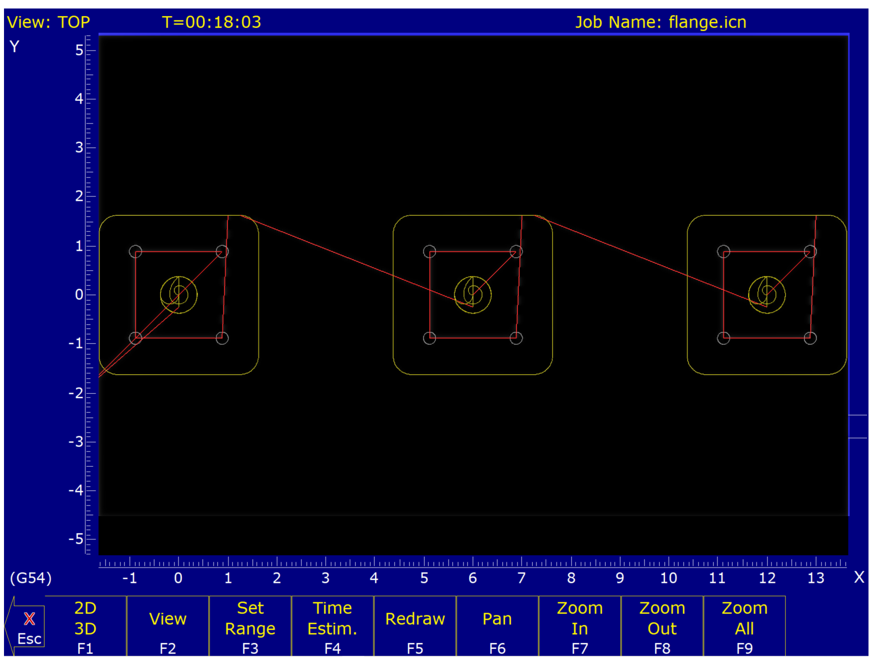
F9 – Zoom All: Press this key to fit the entire part inside the screen.

F10 – Show Tools: Press this key to show the tools menu, which allows certain tools to be highlighted. Press this key again to hide the tools menu.

Spacebar – Measure: Press this key to take a measurement between two points. In a 2D view, this measurement will be a 2D measurement. In a 3D view, it will be a 3D measurement. The measurement will only be valid if the crosshairs are snapped to a line of the tool path.

Note: If you have a mouse or touch screen attached to your device, you can use them to control the graphing window. Holding the left mouse button allows you to drag the part across the screen, while the right mouse button controls rotation of the part. Spinning the mouse wheel (or holding both left and right buttons) zooms in and out. Double-clicking on a feedrate movement will center the camera on that movement and tells you the length of that movement. For touchscreen operation, use the F1 key to switch between Pan and Rotate modes.

This function plots the tool path of the current program loaded. Canned drilling cycles are shown in gray. Rapid traverse movements are shown in red. Feedrate movements are shown in yellow and cutter compensated moves in gray. This option is enabled by setting Parameter 260 to 0 (See Chapter 15). Under this setting, the operation of the user interface is slightly different from the accelerated graphics backplot described above.



F1 – 2D/3D: Press this key to view your part isometrically (3D). An axis pointer indicates the current direction of the view. To return back to the tri-planar view, press F1 again.

F2 – View/Rotate: Press this key to change the planar view of your part. The view is indicated by TOP, RIGHT, or FRONT shown at the top of the screen. In 3D Mode, use this key to rotate your part, while using the keyboard arrow keys to rotate in any direction.

F3 – Range: Press this key to set the range of line numbers or block numbers to graph.

F4 – Time: Press this key to estimate the time needed to create the part. It takes into account accelerations and decelerations, but neglects tool change times.

F5 – Redraw: Press this key to redraw the part at any time.

F6 – Pan: Press this key to move the part around the screen. Once pressed, use the crosshairs to pick a location of the part that will be redrawn at the center of the screen. Once a section is selected, press F6 again to continue panning.

F7 – Zoom In: Press these keys to zoom into the part relative to the center of the screen.

F8 – Zoom Out: Press these keys to zoom away from the part relative to the center of the screen.

F9 – Zoom All: Press this key to view the entire part fit inside the screen.

Note: Use the FEEDRATE OVERRIDE knob to control the speed of the graphing. To pause the tool path, turn the knob counter-clockwise until it stops. Turn the knob clockwise to resume drawing. On the offline demo software, the simulated FEEDRATE OVERRIDE knob is controlled by pressing either Ctrl + or Ctrl −.

3.9 F9 – Digitize

Use this to bring up the Digitize screen. This screen allows you to set up and run touch probe digitizing. See Chapter 8 for a detailed description of the digitizing operation.

3.10 F10 – Shutdown

Used to enter the Shutdown menu. This menu allows you to park the machine, power off the control, start a command window, or exit CNC software.

F1 – Park: Use this to park the machine at the end of the day for quicker machine homing at startup. Once **F10 – Park** is selected, The CYCLE START key must be press to start machine movement. The park feature homes each axis, at the maximum rate, to 1/4 of a motor revolution from its home position.

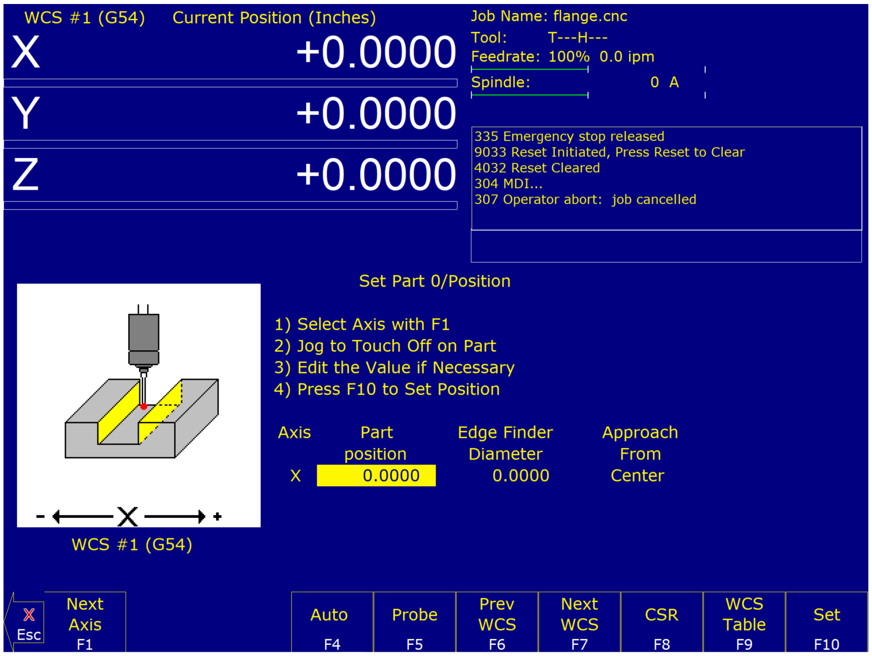
F2 – Poweroff: Use this to properly shutdown the control. With most controls, this action turns off the control once the system has prepared itself to be shutdown. Just like a desktop computer, the control should be properly shutdown before turning off the power in order to reduce the risk of corrupting data on the hard drive.

Note: This will only turn off the control. The machine itself will still need to be manually turned off.

F6 – System Prompt: This brings up the command line interface. (Type the command exit to exit the command window.)

F9 – Exit CNC12: Use this to exit the CNC control software.

4 Part Setup (F1 from Setup)



The Part Setup menu is used to set the part position or the coordinate system origin for the part.

F1 – **Next Axis** Will toggle to the next axis. If changes were made to the current axis, but not yet accepted, they will be discarded.

F2 – **Auto** Uses the probe to automatically measure and set part position. Make sure your probe height and diameter offsets are set for the tool number that you assigned to the probe, and that Parameter 12 is set to that tool number. See Chapter 9 for more details.

F3 – **Probe** Will open the probing operations menu. See Chapter 9 for details.

F4 – **Prev WCS** Will select the previous work coordinate. The position being set will only affect the currently-selected work coordinate.

F5 – **Next WCS** will select the next work coordinate. The position being set will only affect the currently-selected work coordinate.

F6 – **CSR** Will open the CSR menu, which can be used to automatically detect coordinate system rotation. This function key only appears only when the software option for Coordinate System Rotation is unlocked.

F7 – **WCS Table** Will open the Work Coordinate System (WCS) Configuration screen. See the Work Coordinate System Configuration section later in this chapter for a complete description.

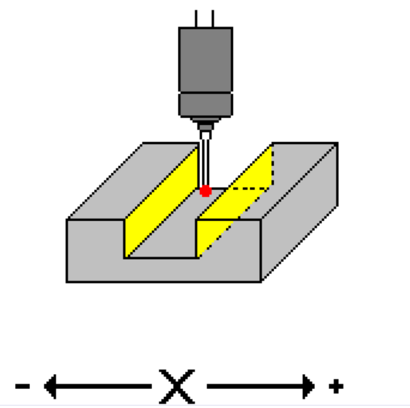
F8 – **Set** Will accept the position for the current axis, correcting for edge finder diameter based on the approach direction (if appropriate). It will not automatically advance to the next axis.

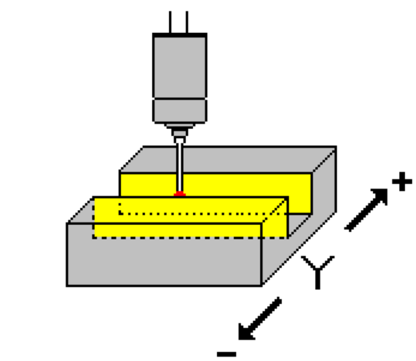
4.1 Operation Description

Setting the part position establishes a coordinate system with an origin at the part zero.

The **F1 – Next Axis** option selects the axis to be defined next. This field toggles between axis X, Y, Z, 4th, and 5th Axes. For each axis, you will see a graphical description of the parameters to be entered, as well as the corresponding fields.

4.1.1 Setting up X or Y AXIS





Set Part Position

1) Select Axis with F1
2) Jog to Touch Off on Part
3) Edit the Value if Necessary
4) Press F10 to Set Position

Axis	Part Position	Edge Finder Diameter	Approach From
X	0.0000	0.0000	Left(-)

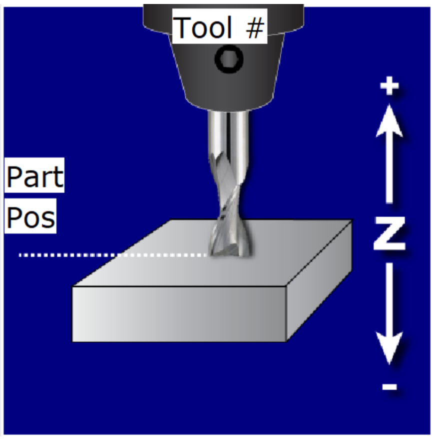
Part Position: Enter the value of your part zero position or the offset.

Edge Finder Diameter: Enter the diameter of the tool or edge finder you are using to determine the part zero. The value entered is stored.

Approach From: Toggle the direction that the edge finder or probe will be approaching the part from.

Note: Use the arrow keys to toggle between Part Position, Edge Finder Diameter, and Approach From options.

4.1.2 Setting up the Z AXIS



Set Part Position

1) Select Axis with F1
2) Jog to Touch Off on Part
3) Edit the Value if Necessary
4) Press F10 to Set Position

Axis	Part Position	Tool Number
Z	0.0000	0

Part Position: Enter the value of your part zero position or the offset.

Tool Number: Enter the tool number from the Tool Library that corresponds to the tool being used. When the Tool Number field is set to a value other than zero, the controller uses the Height Offset for that tool from the Tool Library to calculate the actual position.

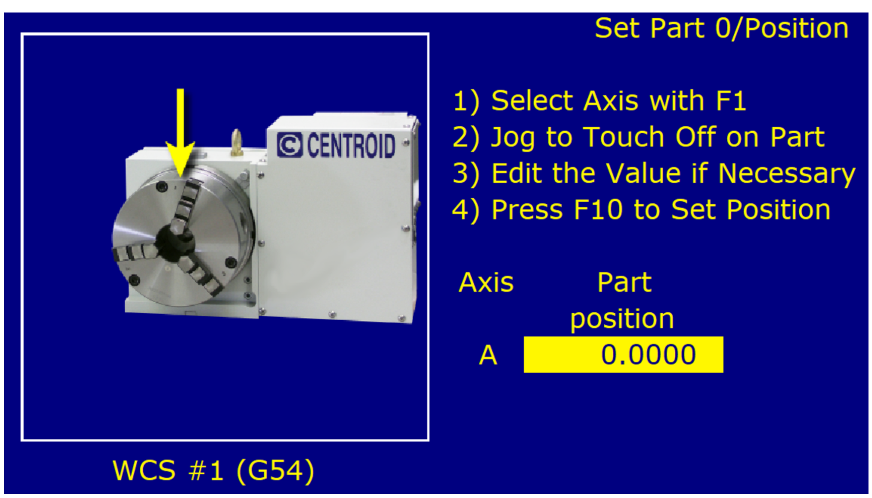
Example 1 (You are using the reference tool to find the Z-axis part zero):
Set Tool Number to 0: setting the Tool Number to zero tells the controller that you are using the reference tool.

Example 2 (You are using a tool other than the reference tool, and not a ball nose cutter):
Set Tool Number to a Tool Number that is assigned in the tool library (make sure its height offset is set).

Example 3 (You are using a ball nose cutter, other than the reference tool):
Set Part Position to the position of the surface plus the nose radius of the ball nose cutter. Set Tool Number to the number this tool is assigned in the tool library.

The Tool and Offset libraries must be up to date before setting the Z-axis Part Zero.

4.1.3 Setting up the 4th or 5th AXIS***



Position: Enter the value of your part zero position or the offset.

4.1.4 Using Multiple Work Coordinate Systems

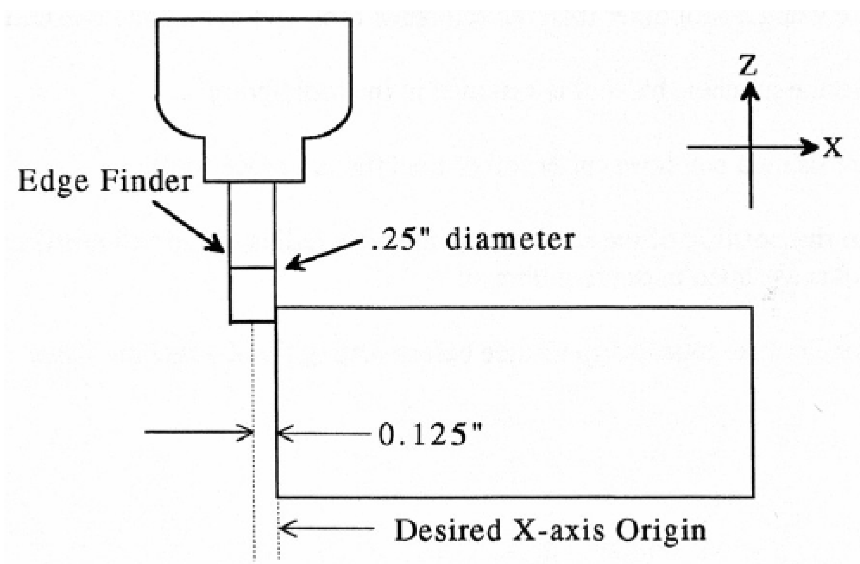
If you will be using multiple work coordinates, you must set the part position separately for each work coordinate. Follow the instructions above to set the position for each axis in the first coordinate system. Then, move to the next fixture and press **F6 – Prev WCS** to select the previous work coordinate, or **F7 – Next WCS** to select the next work coordinate. The currently-selected coordinate system is displayed below the axis picture on the Part Setup screen. It is also displayed above the DRO at all times. For a description on setting up each work coordinate, see the Work Coordinate System Configuration section later in this chapter.

NOTICE

This procedure does NOT apply to tilt table setup.

4.2 Part Setup Examples

Example 1: Setting the X-axis Part Zero with no offset (See diagram below)



If you wanted the left edge of the part to be the origin for the X-axis:

1. Move the Edge Finder to the left edge of the part
2. Press **F1 – Next Axis** until the Axis label displays 'X'
3. Move the cursor to the Edge Finder Diameter field
4. Type .25 and press **ENTER**
5. Press **SPACE** until Left (-) is displayed
6. Press **F10 – Set** to accept the values

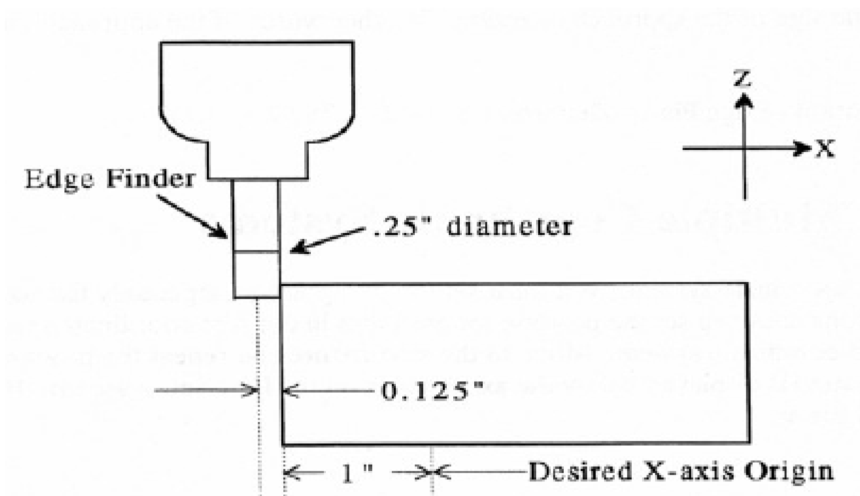
Axis	Part Position	Edge Finder Diameter	Approach From
X	0	0.25	Left (-)

Since no offset is being applied, Part Position is zero. The Edge Finder is approaching the part from the -X direction and has a diameter of .25 inches. Once this data is entered and **F10 – Set** is pressed, the X-axis DRO display will read -0.125. This means that the center of the Edge Finder is sitting to the left (minus) of the part by 0.125 inches (half of the Edge Finder Diameter).

This value is computed by: Position (Approach from) Edge Finder Diameter / 2.

Where (Approach from) is the sign of the approach direction. In other words, if the approach direction is minus, then the value is: Position – Edge Finder Diameter / 2 = 0.0 - .25 / 2 = -0.125

Example 2: X-Axis origin offset into part 1 inch.



If you wanted the origin offset 1 inch into the part:

1. Move the Edge Finder to the left edge of the part
2. Press **F1 – Next Axis** until the axis field displays 'X'
3. Move the cursor to the Part Position field
4. Type -1 and press **ENTER**
5. Type 0.25 and press **ENTER**
6. Press **SPACE** until Left (-) is displayed
7. Press **F10 – Set** to accept the value

Axis	Part Position	Edge Finder Diameter	Approach From
X	-1	0.25	Left (-)

The Position value is relative to the current position of the Edge Finder. Part Position equals -1.0 since the Edge Finder is positioned 1 inch to the left (minus direction) of where you want the X-axis origin.

Another way to view the Part Position value is to assume that the origin is already set at 1 inch into the part. In this case, the Edge Finder would have to move -1 inches from where the origin is to get to the left edge of the part.

The Edge Finder is approaching the part from the -X direction and has a diameter of .25 inches. Once this data is entered and **F10 – Set** is pressed, the X-axis DRO display will read -1.125. This means that the center of the Edge Finder is sitting to the left (minus) of the origin by 1.125 inches. The X-axis origin is now 1 inch into the part.

This value is computed by: Position (Approach from) Edge Finder Diameter/2, where (Approach from) is the sign of the approach direction. In other words, if the approach direction is minus, then the value is:

$$\text{Position} - \text{Edge Finder Diameter}/2 = -1.0 - .25/2 = -1.125$$

4.3 Work Coordinate Systems (WCS) Configuration

Press **F9 – WCS Table** from the Part Setup screen to display the Work Coordinates System (WCS) menu. The Work Coordinate Systems screen provides access to reference return points, coordinate system origins, and work envelope. Make sure your Home position has been set properly. Otherwise, the positions of each coordinate system will not be in the appropriate position.

When you enter the Work Coordinate System Configuration screen, the DRO display will automatically switch over to machine coordinates as an aid to entering numbers. All the values on this screen are represented in machine coordinates.

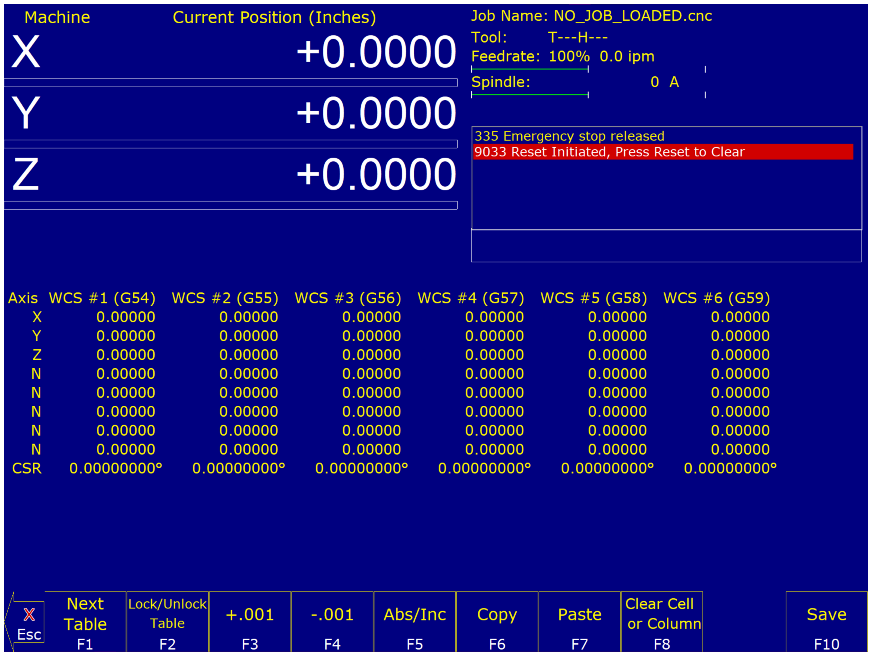
F1 – Reference Return Points Press this key to access the menu that sets the reference return points for the machine.

Axis	Return #1 (G28)	Return #2 (G30)	Return #3 (G30 P3)	Return #4 (G30 P4)
X	0.0000	0.0000	0.0000	0.2757
Y	0.0000	0.0000	0.0000	-11.0000
Z	-0.5186	-0.5186	-0.5186	-0.5186
A	0.0000	0.0000	0.0000	23.7998
B	-60.8735	-60.8735	-60.8735	-60.8735
N	0.0000	0.0000	0.0000	0.0000
N	0.0000	0.0000	0.0000	0.0000
N	0.0000	0.0000	0.0000	0.0000

The reference return points are used with the G28 and G30 codes (see Chapter 12). They are specified in machine coordinates. The Z coordinate of the first reference point is also used as a Z home position by the M2, M6, and M25 codes (see Chapter 13).

The **F2 – Teach** key copies axis machine coordinate values to the table.

F2 – Origin Use this key to access the menu for specifying the locations (in machine coordinates) of the origins for all 18 work coordinate systems.



All coordinate systems are relative to the Home position that is set during control power up. Note that while in this screen, the DRO shows the actual machine position relative to Home, not the location relative to the WCS origin.

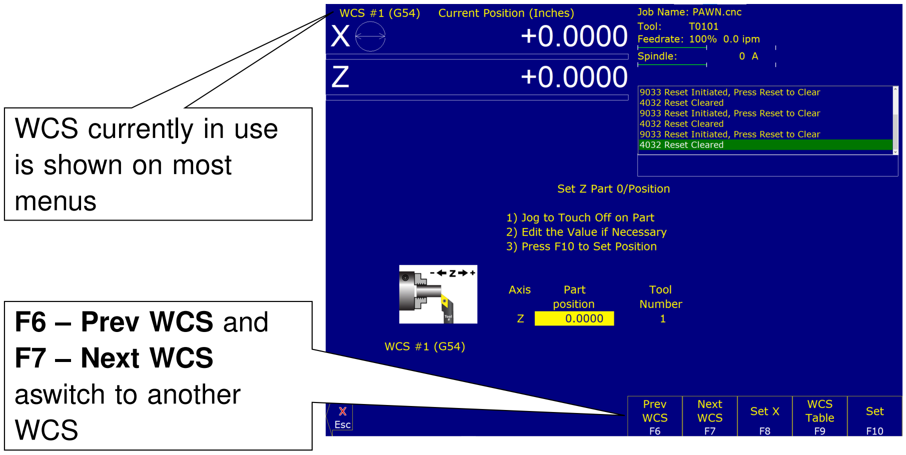
If the software option Coordinate System Rotation is unlocked, the CSR angle for each of the coordinate systems can also be set (this is an optional feature.)

For machines configured as an Articulated Head system with the TWCS feature enabled (set via Parameter 166), the TWCS=Yes/No setting differentiates which WCS's are transformed or not. See Transformed WCS later in this chapter.

Using Work Coordinate Systems These different part zero positions are typically used to reduce setup and/or programming time. There are a number of creative ways that the WCS can be used to simplify lathe machining. The 18 work coordinates and their G-codes are shown below. Regular WCS #1–6 are standard, but extended WCS #7–18 are an extra-cost option.

Regular WCS		Extended Work Coordinate Systems			
WCS	**G-Code**	**WCS**	**G-Code**	**WCS**	**G-Code**
WCS #1	G54	WCS #7	G54 P1	WCS #13	G54 P7
WCS #2	G55	WCS #8	G54 P2	WCS #14	G54 PS
WCS #3	G56	WCS #9	G54 PS	WCS #15	G54 P9
WCS #4	G57	WCS #10	G54 P4	WCS #16	G54 P10
WCS #5	G58	WCS #11	G54 PS	WCS #17	G54 P11
WCS #6	G59	WCS #12	G54 P6	WCS #18	G54 P12

Any time that the Digital Read Out (DRO) for the X and Z current position is displayed, a display of which WCS the control is currently using will be shown in the upper left hand corner of the screen right above the DRO (see the figure below). The DRO always displays the tool position from the WCS that is being used.



To change the WCS being used:

- From the M-series control Main Screen, press: **F1 – Setup**, **F1 – Part**.
- Now press **F6 – Prev WCS** or **F7 – Next WCS**, and the WCS number will change in the upper left corner of the display.

The WCS will change to the next position – if you were on WCS#1 and press **F7 – Next WCS**, it will change the DRO to WCS#2. Simply press **F6 – Prev WCS** or **F7 – Next WCS** until the WCS displayed is the one you want to use. After that you can set up the new WCS using the part setup menus for X and Z to define a new Part Zero position with this WCS. See the section "Setting Part Zeros" in this chapter and the two sections after that for step-by-step instructions of how to zero out your part. Once a WCS is set, the control will remember this position as the Part Zero for that WCS until you change it, even if the control is shut off.

F1 – Next Table Used to cycle through viewing the other WCS (six per page).

F2 – Lock/Unlock Table Used to lock/unlock WCS tables from editing (see Parameter 45 for more information).

F3 – +.001 Increases existing cell values by .001 inches (.01 mm for metric installations)

F4 – -.001 Decreases existing cell values by .001 inches (.01 mm for metric installations)

F5 – Abs/Inc Used to cycle between Absolute and Incremental modes for altering existing cell values.

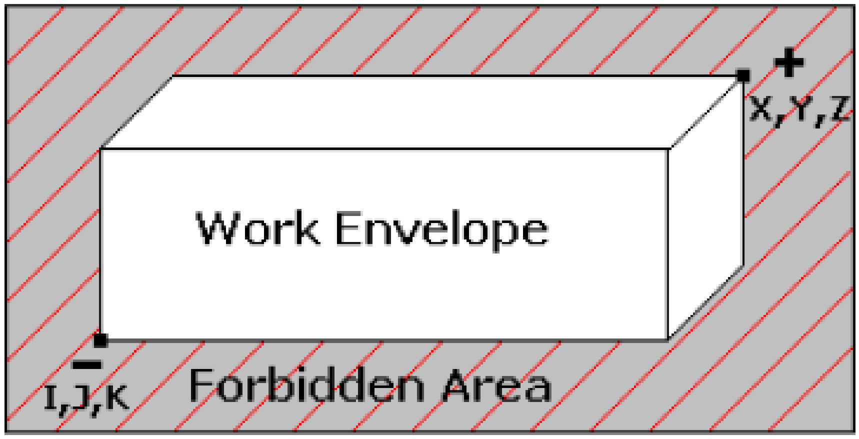
F6 – Copy Copies cell/column contents.

F7 – Paste Pastes cell/column contents.

F8 – Clear Cell or Column Clears cell/column contents.

F10 – Save Saves the current WCS table configuration.

F3 – Work Envelope Use the **F3 – Work Envel** key to specify the '+' and '-' work envelope locations (in machine coordinates) used in conjunction with the G22 G-code. The X, Y, Z and I, J, K parameters specified in the G22 G-code are stored here, so subsequent G22 codes do not need to specify the limits unless they change.

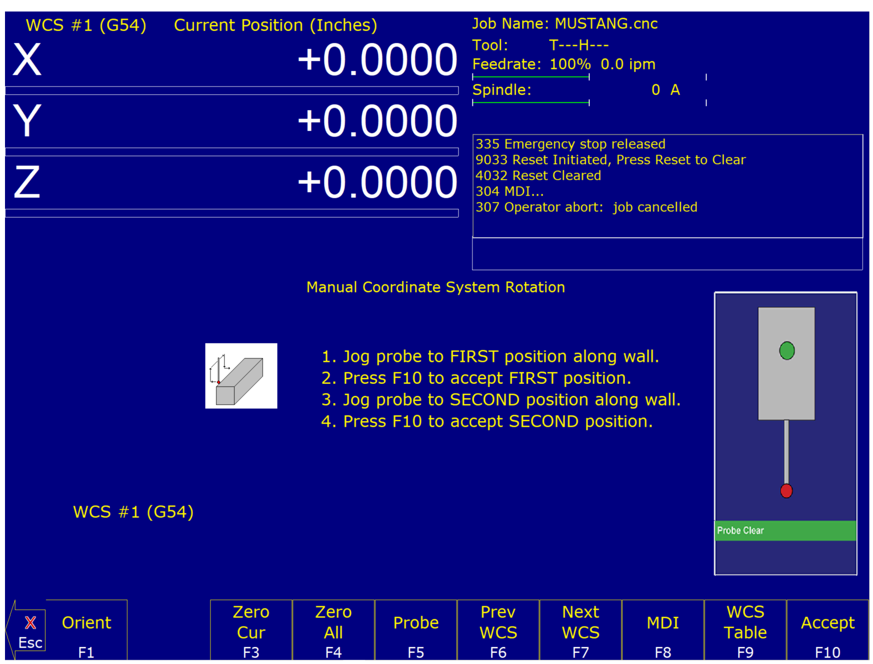


Note: The work envelope will only work in programmed moves. You will still be able to jog outside the work envelope.

4.4 Coordinate System Rotation (CSR)

Coordinate System Rotation saves you time when setting up your part. Rather than clamping your part and indicating the edge of the material to square it with the machine axes, you can use CSR to automatically rotate the coordinate system to the angle of the part or fixture that was probed. This allows you to compensate for different orientations.

Simply clamp your part, and then probe two points along either the X or Y-axis of the material using the process described below.



F1 – Orient is used to select the orientation for the CSR measurement. There are four possible orientations, which are: from the front (pictured above), the back, and the left and right sides.

F2 – Manual is used to determine the CSR angle without probing. The user jogs an edge finder to two positions along one wall. These positions will be used for computing the CSR angle.

F3 – Zero Cur is used to set the CSR angle for the current WCS to zero.

F4 – Zero All is used to set all CSR angles to zero.

F5 – Probe Will open the probing operations menu. See Chapter 9 for details.

F6 – Prev WCS is used to cycle through the available WCS systems.

F7 – Next WCS is used to cycle through the available WCS systems.

F8 – MDI The MDI menu allows you to run a single line command such as: G1 X2 Y3 F20

F9 – WCS Table is a shortcut to the Work Coordinate System Configuration Screen described above.

The instructions on how to perform a CSR measurement are numbered on the screen.

<u>Distance</u>: The distance that the X-axis (in front or back orientation) or Y-axis (in right or left side orientation) will move to probe the second point. If the distance is negative, the axis will be moved in the negative direction.

<u>Clearance Amount</u>: The distance that the Z-axis will be moved upward when moving between the first probe point and the second probe point. The clearance move will only be made when using the "Auto" option of the Movement Between Points.

<u>Movement Between Points</u>: Can be toggled between Jog and Auto modes. In Auto mode, the clearing moves and movement to the second point are made automatically. In Jog mode, a prompt will be displayed in the center of the screen after the first point is probed.

4.5 Transformed WCS (TWCS=Yes)

This section only applies to Articulated Head machines with the TWCS feature enabled via Parameter 166 (see Chapter 15 for more information on setting Parameter 166). On such a machine, when a WCS has a setting of TWCS=Yes, then this is called a transformed WCS (abbreviated as TWCS).

When a TWCS is selected, the DRO will show axis positions based on the TWCS's frame of reference. In other words, the shown positions are transformed based on the position of the B axis (5^{th} axis). Furthermore, the WCS label will be shown as "TWCS" to indicate that the currently-selected WCS is indeed transformed.

Also, when the Probing Cycles are run with a TWCS selected, the results shown will be based on the TWCS frame of reference.

Ordinary three-axis move types performed while running a CNC program with a TWCS selected will be automatically transformed. Such move types include:

- G0, G1, G2, G3
- Protected move probing functions M115, M116, M125, M126
- Canned Cycles G73, G74, G76, G81, G82, G83, G84, G85, G89
- M25

- Moves that involve CSR and Cutter Compensation

However, moves that will not be transformed are:

- Homing moves M91/M92
- Move to switch M105/M106
- Move axis by counts M128

5 Tool Setup

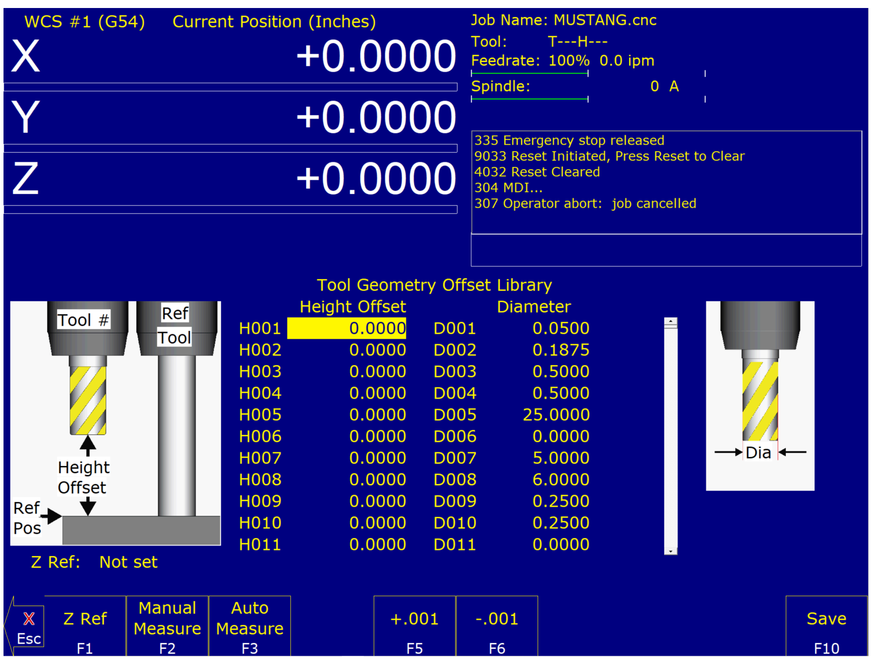
(from Main Screen: F1 Setup → F2 Tool)

Tool Setup allows you to specify information about the tools you will be using.
Press **F1 – Offset Library** to edit the Height Offset and Diameter (H and D) values, or
Press **F2 – Tool Library** to edit the tool descriptions, or
Press **F3 – Tool Life** to edit the Tool Life Management settings.

5.1 Offset Library

(from Main Screen: F1 Setup → F2 Tool → F1 Offset Lib)

The Offset Library file contains the values for the Height Offset and Diameter Numbers. For example, if entry H01 has a value of -.25, a height offset of -.25 is applied when height offset 01 is referenced. If entry D01 shows a value of 1.5, the diameter offset 01 has a diameter of 1.5 associated with it.



Press **F1 – Z Ref** to set the Z reference height. Press **F2 – Manual** to manually measure tools. If you purchased the Automatic Tool Measurement (TT1) option, press **F3 – Auto** to automatically measure tool lengths. Press **F5 – +.001** or **F6 – -.001** to adjust the selected offset. If you have an automatic tool changer installed, press **F7 – ATC** to change tools. Press **F10 – Save** to save changes and exit, or **ESC** to exit without saving changes. If you have both purchased the Automatic Tool Measurement (TT1) option and have an automatic tool changer installed, then you can press **F4 – Auto** to perform batch tool measuring by entering a list of multiple tool numbers.

You can inspect and change any of the 200 Height Offset (H) or 200 Diameter (D) values. In most cases, you will use the automatic tool length measurement features described below to set H values. D values will be entered manually, based on the known or measured diameters of your tools. Note that H01 and D01, H02 and D02, H03 and D03, etc. are displayed together on the same line for convenience only. The Height and Diameter Offset Numbers can be used independently, associations are made only in the Tool Library.

Height Offset

This is the distance that the control adjusts Z-axis positions when tool length compensation (G43 or G44) is used with a particular H value. For example, if H001 is -1.0 and the job contains G43 H1, then the CNC software will shift all Z-axis positions down 1.0 to compensate for the shorter tool.

To edit the Height Offset entries, move to the desired height offset number with the arrow keys, **Page Up**, **Page Down**, **HOME**, and **END**. You can choose to manually edit or automatically measure the value.

Height Offsets values are measured using the Z Reference position. The Z Reference position is the Z-axis position when the tip of the reference tool is touching the work surface. The reference tool should always be the longest tool.

The Height Offset value for end mills and drills is the difference between the Z-axis position when the tip of the tool is touching the work surface, and the Z Reference position. The Height offset value for ball nose and bull nose cutters is the difference between the Z-axis position when the center of the tool is at the work surface, and the Z reference position. Because it is not possible to position the tool in this way, instead move the tip of the tool to the work surface, and then manually edit the value to subtract the tool nose radius.

To manually edit a Height Offset value, simply type the desired value and press **ENTER**.

To manually measure Height Offset values, use the following procedure:

Establishing the Z reference position

Press **F1 – Z Ref** to select the Z Reference setting function.

Insert the longest tool into the tool holder (you can use the jog or TOOL CHECK keys to assist you).

Jog the tip of the tool to the top of the work surface.

Press **F10 – Save** to save this Z Position as the Reference Position.

Measuring each tool height (Z position for tool minus Z position for Reference tool)

Insert the desired tool into the tool holder (Jog or TOOL CHECK keys can be used to assist you).

Jog the tip of the tool to the top of the work surface.

If the tool is a drill or end mill, press **F2 – Manual Measure** to measure the height.

If the tool is a bull or ball nose cutter, press **F2 – Manual Measure** to measure the height, and then subtract the tool nose radius from this value.

After a tool height is measured, the next Height Offset entry is automatically selected.

When the edit is complete, press **F10 – Save** to save the Offset Library and Exit.

Examples assuming Z Reference = -1.5:
If the tool position is -1.75, then the tool height = -0.25
If the tool position is -1.75 and the nose radius is .25, then the tool height = -0.50
If the tool position is -2.25, then the tool height = -0.75

If the tool position is -2.75 and the nose radius is .125, then the tool height = -1.375

Diameter

This field tells the control the distance to adjust when cutter diameter compensation (G41 or G42) is used with a particular D value. For example, if D001 is 0.5 and the job contains G41 D1, the CNC software will adjust all X-Y positions 0.25 (half the tool diameter) to the left of the programmed tool path.

To edit the Diameter entries, move to the desired diameter offset number with the **Arrow Keys**, **Page Up**, **Page Down**, **HOME**, and **END**. You must manually edit the Diameter Offset value. Type the desired value and then press the **ENTER** key.

You can make small adjustments to Height Offsets and Diameters using **F5 – +.001** and **F6 – -.001**. Use the arrow keys to highlight the value to be adjusted. Press **F5 – +.001** to increase the offset value by 0.001" (or 0.02 mm in Metric mode). Press **F6 – -.001** to decrease the offset by the same amounts. If the cut parts are undersized, use **F5 – +.001** to cut less material. If the cut parts are oversized, use **F6 – -.001** to cut more material.

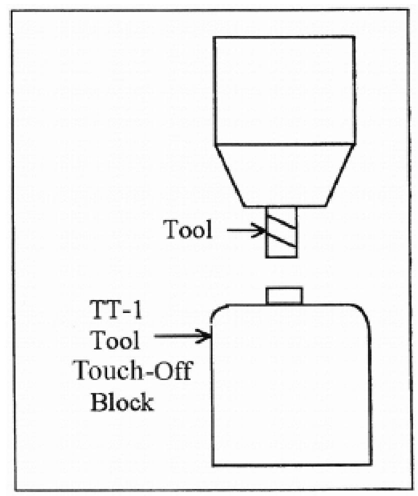
5.1.1 Automatic Tool Measurement

Z-minus single-surface probing using the TT-1 tool touch-off post is available in the Tool Offset Library.

First Time Setup

Make sure that the proper parameters are set as per Chapter 9 and Chapter 15 (see parameters 18, 244, 257, 281, 282, 283, 367), and that the detector is plugged in and at the correct location on the table. When first testing the TT-1, hold the TT-1 in hand and manually touch the unit to the tool to confirm the correct electrical connection and parameter setup.

WARNING Incorrect setup may cause damage to the machine, tool, and/or cause injury to the operator.



Setting the Z Reference:

Using the longest tool for the job or the designated reference tool, press **F1 – Z Ref**, then **F3** and finally **CYCLE START**. The Z-axis will then move down until the tool touch-off is detected. The Z reference will be set at this position. Parameter 3 bit 1 is used to set Z reference to the Z home position. See the parameter section in Chapter 15 for more info.

Setting the Tool Height Offsets:

Pressing **F3 – Auto Measure** and then **CYCLE START** at the prompt will cause the Z-axis to move down until the tool touch-off is detected. The resulting tool length will be entered into the table (same as with **F2 – Manual**). The Z-axis then returns to its home position.

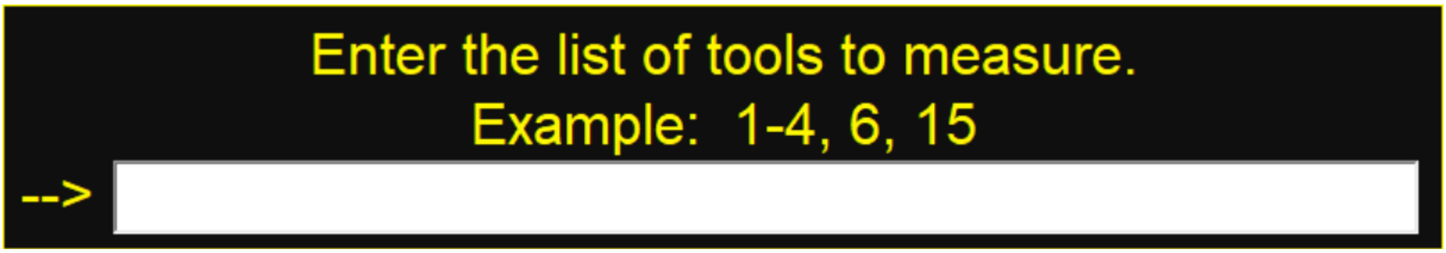
If Parameter 17 has been set to the number of a valid return point (1 or 2), the **F3 – Auto Measure** option will move the X and Y axes to that return point before moving Z downwards. Return point 1 is the G28 position from the Work Coordinate System Configuration screen (see Chapter 4). Return point 2 is the G30 position on that screen. If Parameter 17 is zero (0), the X and Y-axes will not move before Z moves downwards. In this case, you must be careful to jog the machine directly over the detector before pressing **F3 – Auto Measure**.

See parameters 18, 244, 257, 281, 282, 283, and 367 in Chapter 15 for full setup information.

Note: SHIFT + F3 can be used to override any return point movement in cases where Parameter 17 is set to use it. This is helpful for measuring tools where the height measurement is not taken from the center point of the tool.

Batch Tool Height Offset Measurement Process:

If you have both purchased the Automatic Tool Measurement (TT1) option and also have an automatic tool changer installed, then you can press **F4 – Batch** to measure multiple tools in one process. After pressing **F4 – Batch**, you will be prompted with the following dialogue box:



After entering a list of tool numbers, you can press **CYCLE START** to perform the batch tool measurement process. This process is similar to the single tool height offset measurement (accessed via **F3 – Auto Measure**), but will do multiple tools in one shot.

5.1.2 Setting Up Tool Height Offsets

NOTICE Before manually jogging any probe to a position, make sure that the machine feedrate is turned down (less than 10 in/min) or damage to the probe may result!

Using a Probe as the Reference Tool

Before you set the Z Reference, make sure that the probe Tool Number is entered into Parameter 12 on the Machine Parameters screen. Make sure that Parameter 17 on the Machine Parameters screen contains a 0. Follow these steps to probe Z Reference:

1. Load the probe into the machine.
2. Jog the probe over the desired reference surface and press **F1 – Z Ref**.
3. Press **F3** and then **CYCLE START**. The probe will find the Z Reference.

At this point, the Z Reference is now entered into the Offset Library and is the reference height for all other tools. Remove the probe and measure any other tool offsets manually as described earlier in this chapter.

Measuring Each Tool Offset Using a Fixed Detector

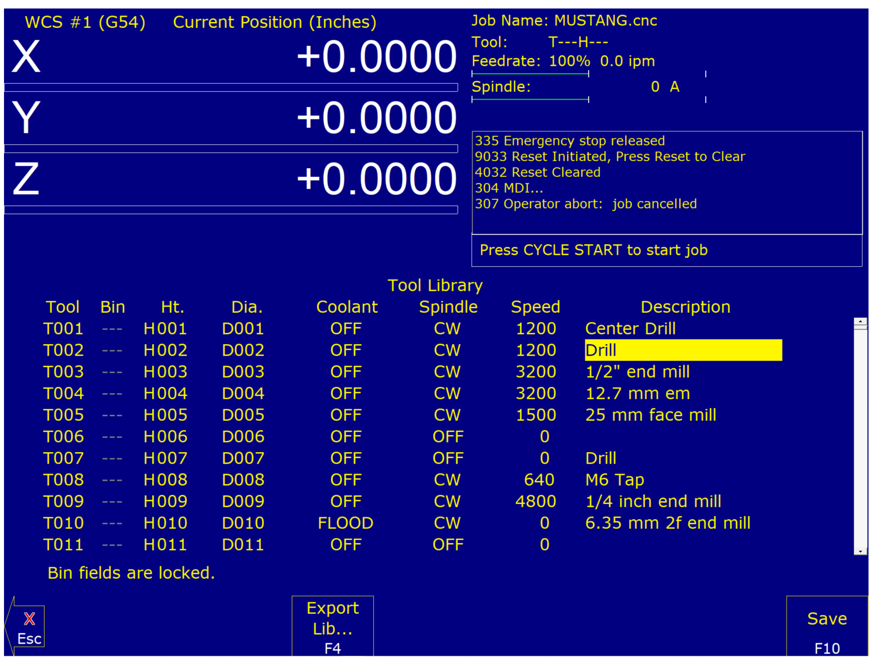
Before measuring any tool height, make sure you enter the probe or reference tool measuring location. Do this by entering a reference point number (1 or 2) into Parameter 17 and entering the detector position as the corresponding Reference Return Point on the WCS Configuration screen. Otherwise, the machine may traverse to a location that could damage the probe or reference tool. Also, remember that if Parameter 17 is zero (0), the X and Y-axes will not move before Z moves downward. Also, be sure that Parameter 44 is set correctly. This is the input number for the TT1. Now that a permanent location has been set, do the following:

1. Load a reference tool (preferably the longest tool) and highlight its corresponding Height Offset Number using the up or down arrow keys.
2. Press **F1 – Z Ref**, **F3 – Auto Measure**, and then **CYCLE START** to set the Z reference using this tool. The X and Y-axes will traverse to the preset location, and then Z will move downward until the tool is detected and the Z reference will be set.
3. Load the next tool.
4. Highlight the desired Height Offset Number on screen using the up and down arrow keys.
5. Press **F3 – Auto Measure** and then **CYCLE START**. The X and Y-axes will traverse to the preset location, and then Z will move downward means that the tool is detected. Once the detector is triggered, the tool offset will be shown on the screen. A negative offset means the tool is shorter than the reference tool.

Once all of the tool offsets have been measured, press **F10 – Save** to save them. Otherwise, press **ESC** to cancel any changes.

5.2 Tool Library

(from Main Screen: F1 Setup → F2 Tool → F2 Tool Lib)



The definitions in the Tool Library associate tool (T) numbers with height offset (H) values, diameter (D) values, default coolant types, spindle directions, spindle speeds, and text descriptions of the tools. This information is used by the Intercon-programming package (described in Chapter 10) to provide defaults whenever a tool change is selected. For enhanced ATC features, the (T) numbers are also associated with bin numbers. See Chapter 15 for more information about enhanced ATC features (Parameter 160).

Note: If enhanced ATC features are not on, the cursor cannot be moved into the bin column and the message “Bin fields are locked” will appear where the tool in spindle display is located. In addition, the **F1 – Clear Bin** and **F2 – ClearAll** keys only appear if enhanced ATC features are on.

You can inspect and change any of the 200 tool definitions. To edit a Tool Library definition, move to the desired tool number using the arrow, Page Up, Page Down, HOME, and END keys. To change Height Offset numbers, Diameter numbers, default spindle speed values, and the tool description, type a new value into the field and press ENTER. To change the default spindle direction and coolant press SPACE to cycle through the possible values. When the changes are complete, press **F10 – Save** to save the Tool Library and exit.

Bin

This field specifies which bin location, or ATC carousel position, that the tool is occupying. Valid values are -1 (shown as dashes "—") through the maximum number of tools specified by machine Parameter 161. A value of 0 indicates that the tool is currently in the spindle. The **F1**–**F2** keys will work when the cursor is in the Bin column.

F1 – Clear Bin: Places dashes "—" into the bin field (same as entering -1).

F2 – Clear All: Places dashes into every bin field.

Note: For enhanced ATC applications, the bin numbers will be updated when tool changes are completed. For random or arm-type tool changers, tools in the spindle are placed into the same bin that the next tool is picked up from, and not necessarily from the same bin it was originally taken from.

Draft: June 12, 2023

Height

This field specifies a default Height Offset (H) number to use with each tool. Possible values are 1 to 200. Intercon uses this information to provide a default H value at each tool change. The CNC software also uses this information to correct for the length of the tool that is used to establish the Z-axis position of the Part Setup (see Chapter 5).

Diameter

This field specifies a default Diameter (D) number to use with each tool. Possible values are 1 to 200. Intercon uses this information to provide a default D value at each tool change. To change the value, type a new number and press ENTER.

Coolant

This field specifies a default coolant type to use with each tool. Possible values are FLOOD, MIST, or OFF. Intercon uses this information to automatically insert M7 or M8 after a tool change. To change this value, press SPACE until the desired value is shown.

Spindle

This field specifies a default spindle direction to use with each tool. Possible values are CW, CCW, or OFF. Intercon uses this information to automatically insert M3 or M4 after a tool change. To change this value, press SPACE until the desired value is shown.

Speed

This field specifies a default spindle speed to use with each tool. Possible values are 0 to 500000. Intercon uses this information to automatically insert an S code after a tool change. To change this value, type a new number and press ENTER.

Description

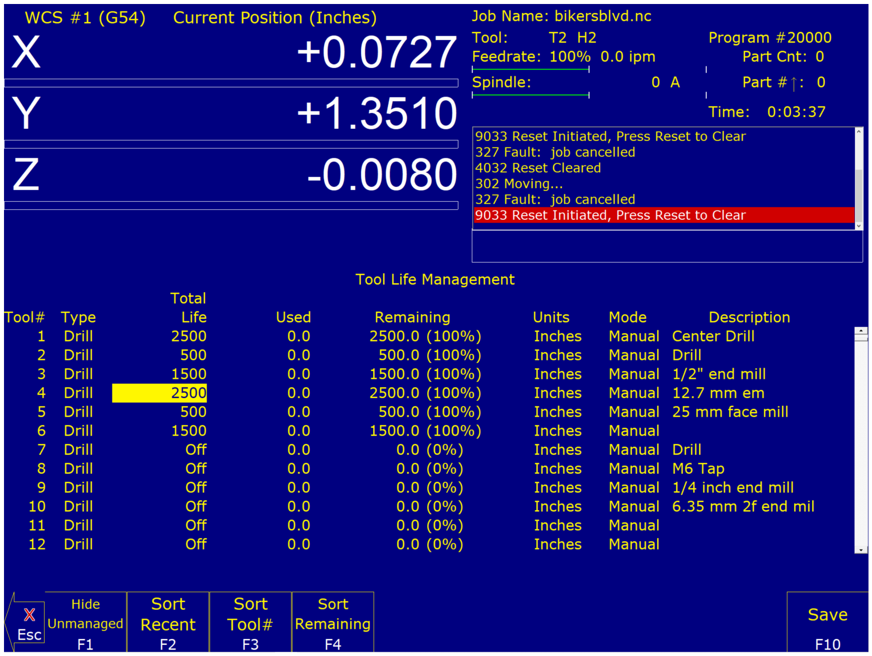
This field contains a text description of the tool. The description will appear in a prompt message on the screen when the CNC software reaches a tool change (M6).

F5 – Export Lib. . . The tool library can be exported in txt (space separated and aligned columns) or csv (comma separated columns) formats by pressing F5. Choose txt or csv to export the desired format.

5.3 Tool Life Management Menu

(from Main Screen: F1 Setup → F2 Tool → F1 Tool Life)

The Tool Life Management feature allows you to set up each tool's pre-determined life, and to have its usage tracked and monitored for end-of-life condition. By default, Tool Life Management is turned off, but can be enabled for each tool individually.



F1 – Show/Hide Unmanaged: This key toggles between including and excluding those tools that are unmanaged by Tool Life Management. A tool is unmanaged if its Total Life field is set to value 0 (Off).

F2 – Sort Recent: This key sorts the list according to tools whose Total Life and/or Used field were most recently modified by tool usage in a job run or by edits done in this menu.

F3 – Sort Tool # : This key sorts the list according to Tool Number.

F4 – Sort Remaining: This key sorts the list according to Life Remaining.

F10 – Save: Saves changes.

A tool can be set up for automatic management by setting its Mode to Auto and its Total Life to a non-zero value. The following table shows the effects of monitored tool activity on tools that are set up for automatic management.

Automatic Management: Mode = Auto and Total Life is more than 0			
Type	**Units**	**Tool Activity that is monitored**	**Effect on the “Used” field**
Drill	Cycles	Downward Z plunge at feedrate at a unique XY location	“Used” field will be incremented by 1 cycle
Drill	Inch/mm	Downward Z plunge at feedrate at a unique XY location	Total downward Z distance, minus the overlaps, will be added to the “Used field”
EM (End Mill)	Cycles	Tool Change	“Used” field will be incremented by 1 cycle
EM (End Mill)	Inch/mm	Sideways XY feedrate moves (non-rapid)	XY distance will be accumulated in the “Used” field

A detailed description of each field follows:

Type

This is the type of tool – either Drill or EM (end mill). When the Mode field is set to Auto, this field determines the type of tool activity that will be automatically tracked and monitored for the purpose of accounting for consumed tool life (See the Automatic Management table above.) Note that if the tool is a Bore or Tap, you should select Drill.

Total Life

This field specifies the total amount of tool life. A value greater than 0 enables Tool Life management for the tool. A value of 0 (Off) excludes the tool from Tool Life management. Such a tool is called an "Unmanaged" tool. Unmanaged tools can be shown or hidden by pressing F1. The units of this field are specified in the Units field.

Used

This field is the amount of consumed tool life. When a new tool is first set up, you should initialize this field to 0, indicating zero usage. If the Mode field is set to Auto, this field will automatically be modified during a job run to reflect the accumulated tool usage. The units of this field are specified in the Units field.

Remaining (non-edit)

This is the display of the remaining amount of tool life.

Units

This specifies the units (either distance or Cycles) that will be used and displayed for the Total Life, Used, and Remaining fields. Distance is specified with mm or Inches (as set in the Control Configuration menu – see Chapter 15).

Mode

This specifies the update mode of the Used field – either Auto or Manual. If this field is set to Auto, then tool activity is monitored during a job run and automatically accumulated in the updated Used field (See the Automatic Management table above.) If this field is set to Manual, no such automatic updates will take place in the Used field. Rather, updates to the Used field will be dependent upon user variable modifications programmed in the G-code program being run (See the section *Tool Life Management – Using G-Code User Variables* later in this chapter.)

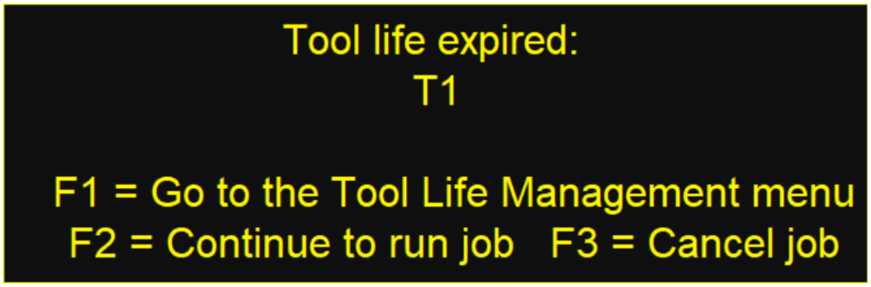
Description

This field contains a text description of the tool. The description will appear in a prompt message on the screen when the CNC software reaches a tool change (M6) during a job run.

5.3.1 Effect on Job Run and Backplot

At Start of Job

Tool life expiration will be checked at the beginning of a job run. If any managed tools are expired at the beginning of a job, the following dialog will show up and you will have one of three choices to make:



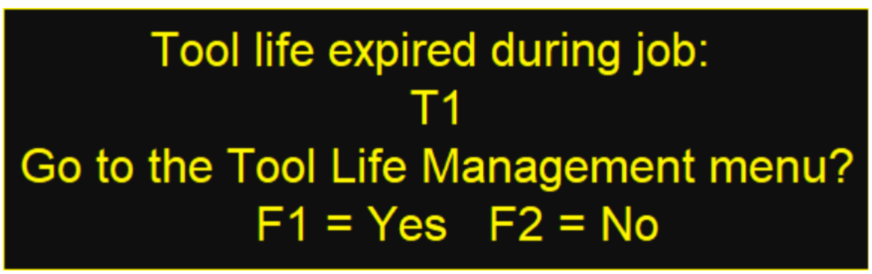
When a job is first started, the CNC software will not yet know which tools are going to be used until the job is successfully completed. Therefore, the tools listed here will be the list of all expired tools, even if they are not going to be used in the job.

At Job Restart

Tool life expirations are also checked upon job restart (i.e. upon encountering M2 or M102). If any tool(s) expired during the previous job run (previous to the M2 or M102), then the dialog that is displayed is similar to that shown above, except that the expired tools listed will only be the ones that were used in the job.

At End Of Job

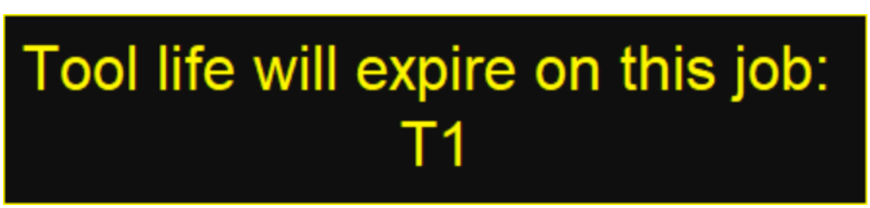
When tool life expires during a job, such an event will not cancel the job. Instead, upon successful the end of the job, the following dialog will be displayed and you will have an opportunity to quickly get to the Tool Life Management menu:



This end-of-job dialogue will show only the expired tools that were used in the job.

Using Backplot Graphics to Predict Tool Expirations

You can use Backplot Graphics as a way to predict ahead of time whether any tools will expire during a job. Simply press **F8 – Graph** at the Main Screen or in the Load menu. If the job being graphed will result in an expired tool if it was actually run, then the following message will show up:



5.3.2 Using G-Code User Variables

If a tool's Mode field is set to Manual, there will be no updates to the Used field of the Tool Life data during a job run, unless the job's G-code is programmed to modify it.

The following is an example of how a G-code program would modify tool life data. Assuming tool T23's Mode is set to Manual and its Units are set to Cycles, the following G-code will increment T23's Used Life field by 1 after completing the examplecycle.cnc.

```
M6 T23                                    ; Change tool to T23
M98 “examplecycle.cnc” L1                 ; Run the cycle 1 time
IF #4201 || #4202 THEN GOTO 100           ; Skip to N100 if in backplot or search mode
IF #4120 < 1 || #4120 > 200 THEN GOTO 100 ; Skip to N100 if T number is not valid
#[19000+[#4120-1]*5+2] = #[19000+[#4120-1]*5+2] + 1 ; Increment current tool's Used Life by 1 cycle
N100                                      ; Destination of goto's
```

See Chapter 11 for more information about the use of User or System Variables.

5.4 Laser Setup

5.4.1 PWM Output for Spindles and Lasers

a.) 5-volt PWM output signal is on DB25 pin #14.

b.) DB25 pin #14 is Output 2.

c.) Output 2 is also connected to Relay 2 via the ribbon cable.

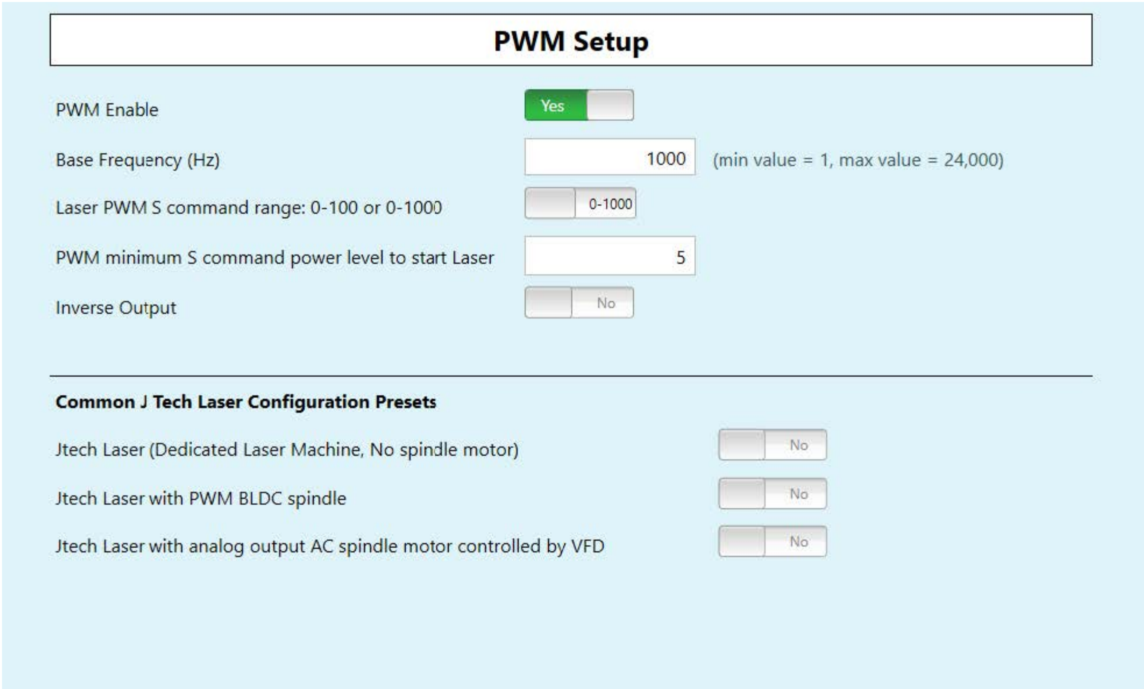
d.) If PWM output is used, Relay 2 must be disabled. See schematic S15049 to cut the ribbon cable lead to Relay 2.

e.) PWM is based on the 0–100 OR 0–1000 S command. The user selects the range of 0–100 or 0–1000 in the Acorn Wizard.

f.) M37 turns ON Laser Output, M38 turns OFF Laser Output: M37 will activate Laser Enable, Laser Reset, and PWM Select. After .5s it will turn off LaserReset. At this point, the laser controller will look at the PWM signal from OUTPUT 2. M38 will wait 30s to allow the JTECH laser controller to cool, then perform a M95/37/38 to turn off both Laser Enable and PWMSelect.

g.) The PWM Velocity modulation feature adjusts the PWM output based on the velocity of the machine tool so that overburning is avoided on corners and turn-arounds. G37 is used to turn ON and OFF PWM Velocity Modulation. G37 ON = PWM VM ON; G37 OFF = PWM VM OFF

h.) Simple PWM controls are located in the Acorn Wizard. In addition to “manual PWM controls”, preset buttons for common Jtech configurations are present and have matching schematics (S15049, S15056, S15057).



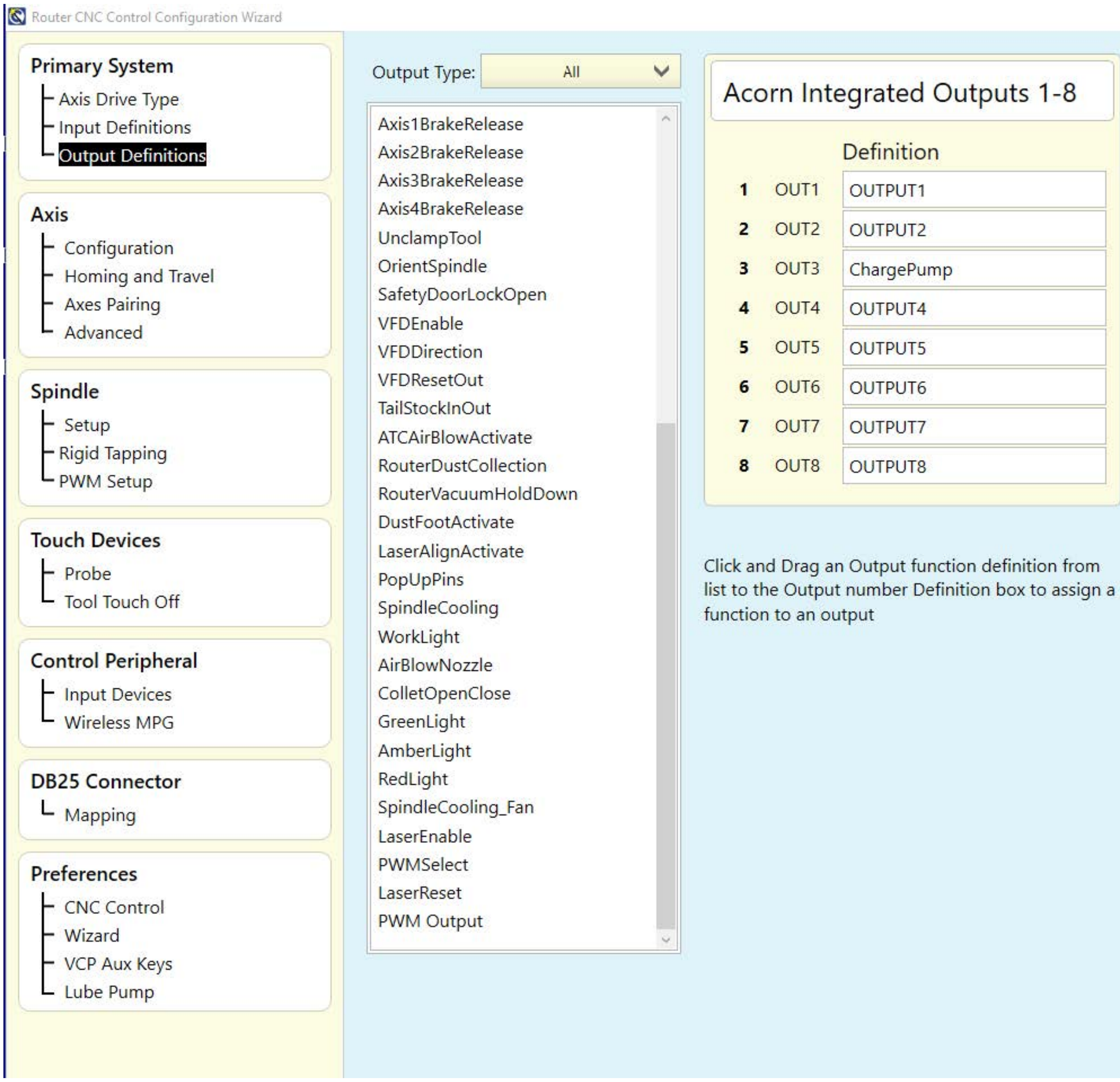
5.4.2 PWM-related I/O in the Wizard

a.) PWM Output: The PWM signal itself. Can only be used on Output 2 (Output 2 of DB25 pin #14). The related CNC code is the S command.

b.) LaserEnable: Typically used in a safety interlock circuit, see Jtech schematic S15049 as an example. Related M codes: M37 – Enables safety interlock and resets laser, M38 – Disables safety interlock after a delay to allow the component to cool down.

c.) LaserReset: Momentary output used to send a reset signal to the laser controller. See Jtech schematic S15049 as an example.

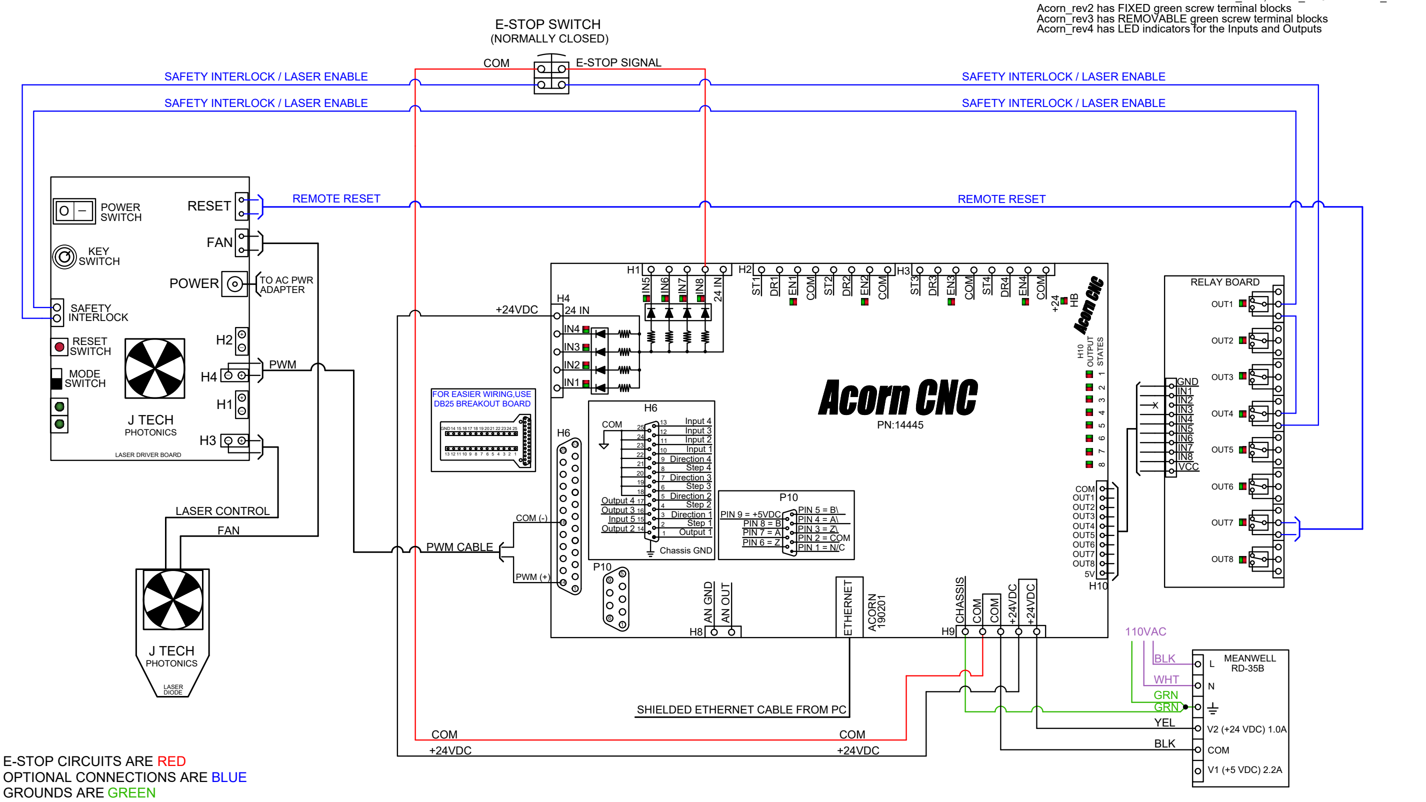
d.) PWMSelect: Output used to move the PWM signal from Spindle to Laser. PWMSelect is used when the PWM signal is required to be sent to different devices. For example, a machine that has both a Spindle Motor and Laser that require PWM to run. PWM from Output 2 is connected to the COM of the relay that PWMSelect is assigned to. To use with a Standard Layout, the Spindle PWM should be connected to the NC side of the Relay and the Laser should be connected to the NO side of relay. When PWMSelect is deactivated, the PWM signal is being sent to the Spindle. When PWMSelect is activated, the PWM signal is being sent to the laser. Follow Schematic J-TECH Photonics Laser, BLDC Spindle Control (#S15057)


Router CNC Control Configuration Wizard
Primary System
Axis Drive Type
Input Definitions
Output Definitions
Axis
Configuration
Homing and Travel
Axes Pairing
Advanced
Spindle
Setup
Rigid Tapping
PWM Setup
Touch Devices
Probe
Tool Touch Off
Control Peripheral
Input Devices
Wireless MPG
DB25 Connector
Mapping
Preferences
CNC Control
Wizard
VCP Aux Keys
Lube Pump
Output Type: All
Axis1BrakeRelease
Axis2BrakeRelease
Axis3BrakeRelease
Axis4BrakeRelease
UnclampTool
OrientSpindle
SafetyDoorLockOpen
VFDEnable
VFDDirection
VFDResetOut
TailStockInOut
ATCAirBlowActivate
RouterDustCollection
RouterVacuumHoldDown
DustFootActivate
LaserAlignActivate
PopUpPins
SpindleCooling
WorkLight
AirBlowNozzle
ColletOpenClose
GreenLight
AmberLight
RedLight
SpindleCooling_Fan
LaserEnable
PWMSelect
LaserReset
PWM Output
Acorn Integrated Outputs 1-8
Definition
1 OUT1 OUTPUT1
2 OUT2 OUTPUT2
3 OUT3 ChargePump
4 OUT4 OUTPUT4
5 OUT5 OUTPUT5
6 OUT6 OUTPUT6
7 OUT7 OUTPUT7
8 OUT8 OUTPUT8
Click and Drag an Output function definition from list to the Output number Definition box to assign a function to an output

ACORN I/O	1	2	3	4
INPUTS				
OUTPUTS	NoFaultOut	PWMOutput		LaserEnable
ACORN I/O	5	6	7	8
INPUTS				EStopOK
OUTPUTS			LaserReset	

Title: ACORN_rev4, J-TECH PHOTONICS LASER		
Date: 201020	Ver: 2	Drawn by: CEM
Filename: S15049.DWG		Sheet 1 of 1

How to tell the difference between Acorn_rev2, Acorn_rev3 and Acorn_rev4
Acorn_rev2 has FIXED green screw terminal blocks
Acorn_rev3 has REMOVABLE green screw terminal blocks
Acorn_rev4 has LED indicators for the Inputs and Outputs

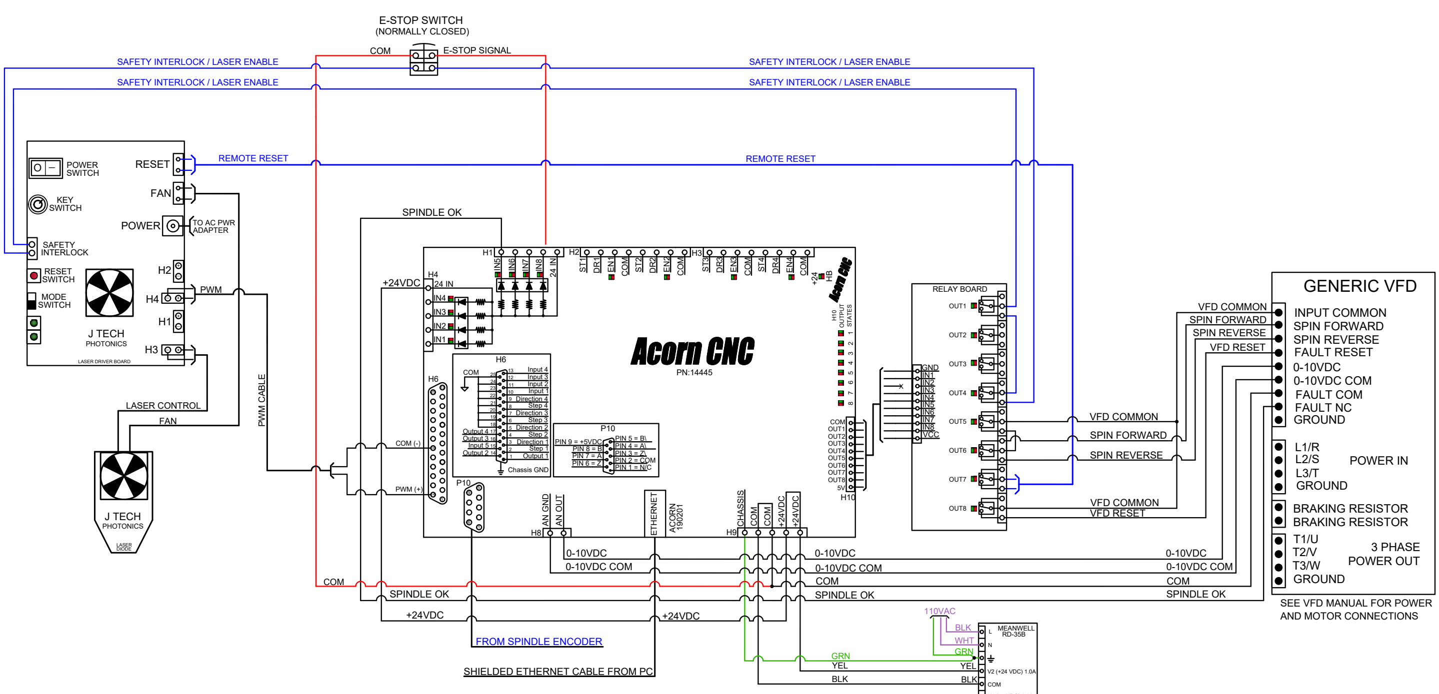


E-STOP CIRCUITS ARE RED
OPTIONAL CONNECTIONS ARE BLUE
GROUNDS ARE GREEN
110VAC IS VIOLET

ACORN I/O	1	2	3	4
INPUTS				
OUTPUTS	NoFaultOut	PWMOutput		LaserEnable
ACORN I/O	5	6	7	8
INPUTS	SpindleOK			EStopOK
OUTPUTS	VFDEnable	VFDDirection	LaserReset	VFDResetOut

Title: ACORN_rev4, J-TECH PHOTONICS LASER, GENERIC VFD ENABLE-DIRECTION		
Date: 201021	Ver: 1	Drawn by: CEM
Filename: S15056.DWG		Sheet 1 of 1

How to tell the difference between Acorn_rev2, Acorn_rev3 and Acorn_rev4
Acorn_rev2 has FIXED green screw terminal blocks
Acorn_rev3 has REMOVABLE green screw terminal blocks
Acorn_rev4 has LED indicators for the Inputs and Outputs

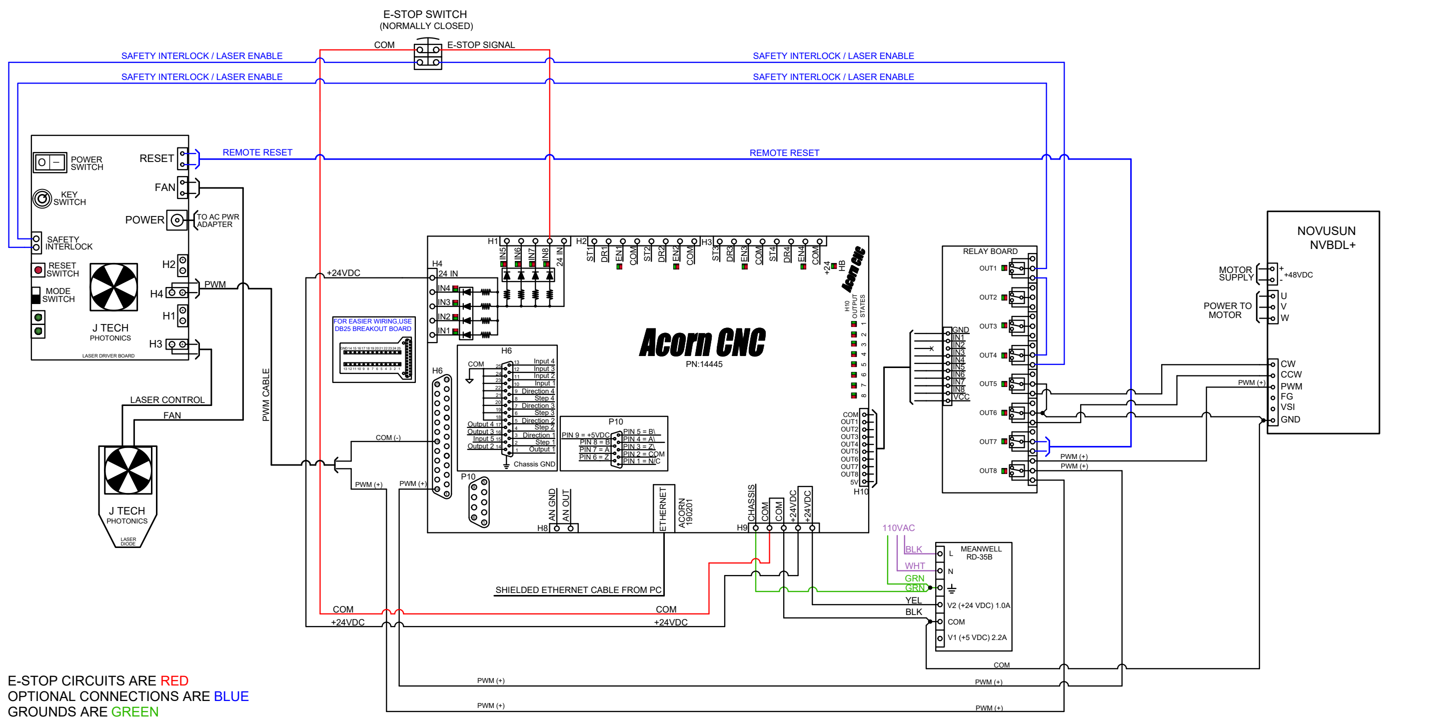


E-STOP CIRCUITS ARE RED
OPTIONAL CONNECTIONS ARE BLUE
GROUNDS ARE GREEN
110VAC IS VIOLET

ACORN I/O	1	2	3	4
INPUTS				
OUTPUTS	NoFaultOut	PWMOutput		LaserEnable
ACORN I/O	5	6	7	8
INPUTS				EStopOK
OUTPUTS	SpinFWD	SpinREV	LaserReset	PWMSelect

Title: ACORN_rev4, J-TECH PHOTONICS LASER, BLDC SPINDLE CONTROL (NOVUSUN NVBL+)		
Date: 201021	Ver: 1	Drawn by: CEM
Filename: S15057.DWG		Sheet 1 of 1

How to tell the difference between Acorn_rev2, Acorn_rev3 and Acorn_rev4
Acorn_rev2 has FIXED green screw terminal blocks
Acorn_rev3 has REMOVABLE green screw terminal blocks
Acorn_rev4 has LED indicators for the Inputs and Outputs

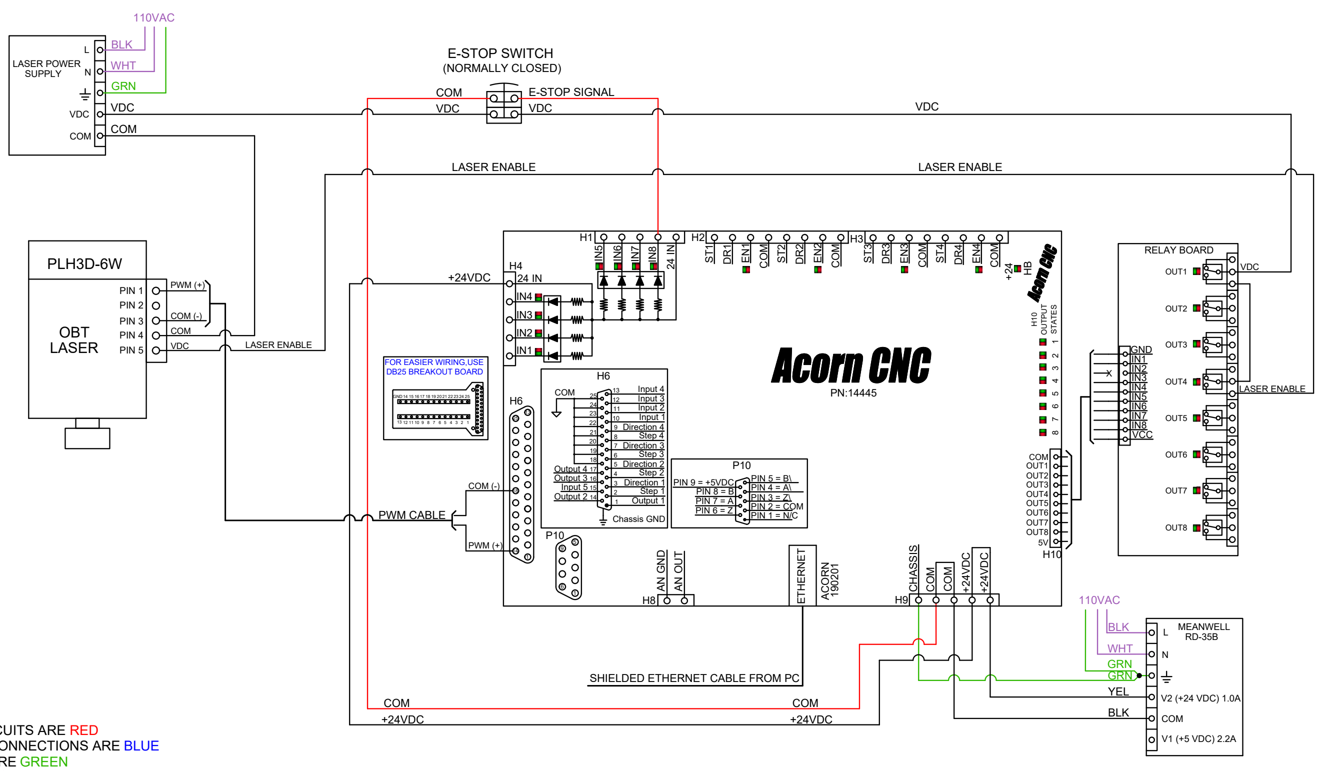


E-STOP CIRCUITS ARE RED
OPTIONAL CONNECTIONS ARE BLUE
GROUNDS ARE GREEN
110VAC IS VIOLET

ACORN I/O	1	2	3	4
INPUTS				
OUTPUTS	NoFaultOut	PWMOutput		LaserEnable
ACORN I/O	5	6	7	8
INPUTS				EStopOK
OUTPUTS				

Title: ACORN_rev4, OBT LASER		
Date: 201214	Ver: 1	Drawn by: CEM
Filename: S15061.DWG		Sheet 1 of 1

How to tell the difference between Acorn_rev2, Acorn_rev3 and Acorn_rev4
Acorn_rev2 has FIXED green screw terminal blocks
Acorn_rev3 has REMOVABLE green screw terminal blocks
Acorn_rev4 has LED indicators for the Inputs and Outputs

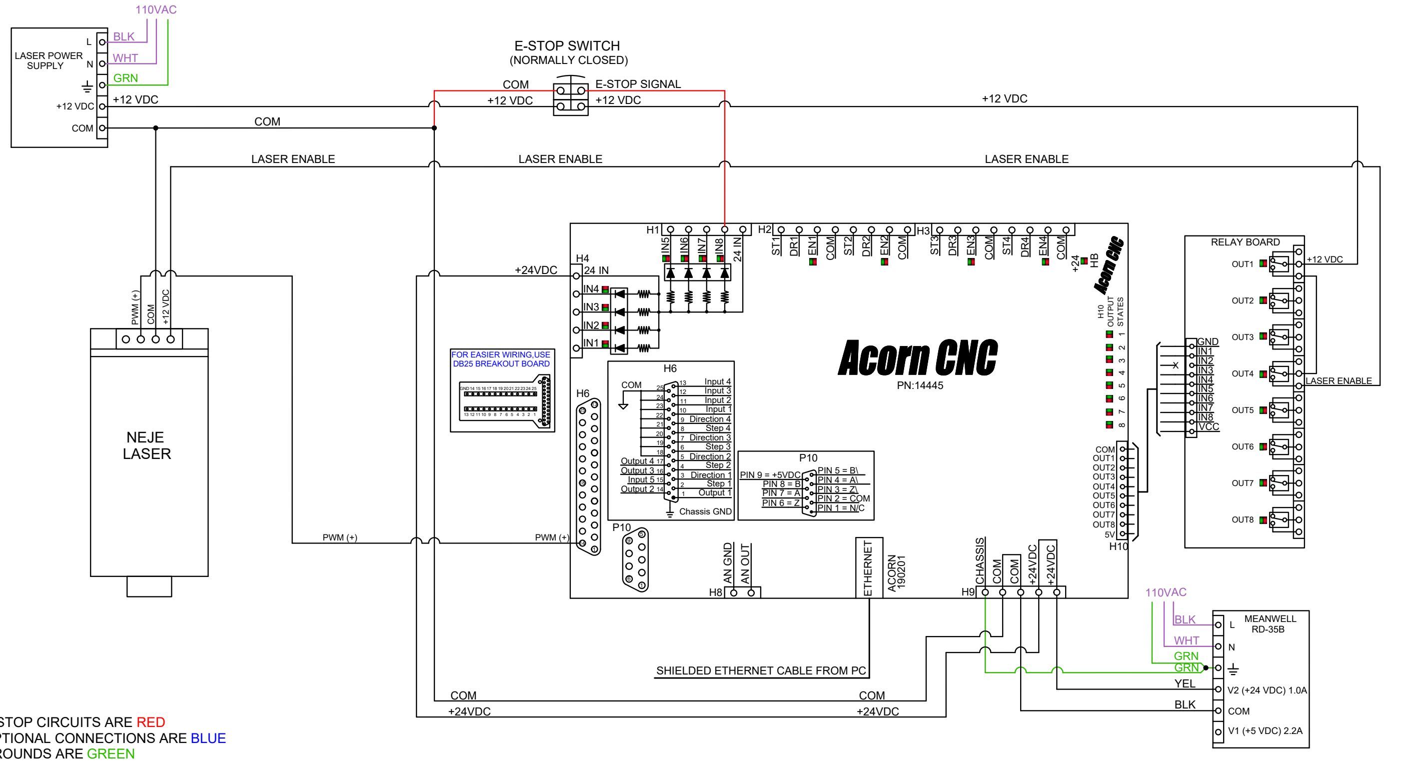


E-STOP CIRCUITS ARE RED
OPTIONAL CONNECTIONS ARE BLUE
GROUNDS ARE GREEN
110VAC IS VIOLET

ACORN I/O	1	2	3	4
INPUTS				
OUTPUTS	NoFaultOut	PWMOutput		LaserEnable
ACORN I/O	5	6	7	8
INPUTS				EStopOK
OUTPUTS				

Title: ACORN_rev4, NEJE LASER		
Date: 201214	Ver: 1	Drawn by: CEM
Filename: S15062.DWG		Sheet 1 of 1

How to tell the difference between Acorn_rev2, Acorn_rev3 and Acorn_rev4
Acorn_rev2 has FIXED green screw terminal blocks
Acorn_rev3 has REMOVABLE green screw terminal blocks
Acorn_rev4 has LED indicators for the Inputs and Outputs

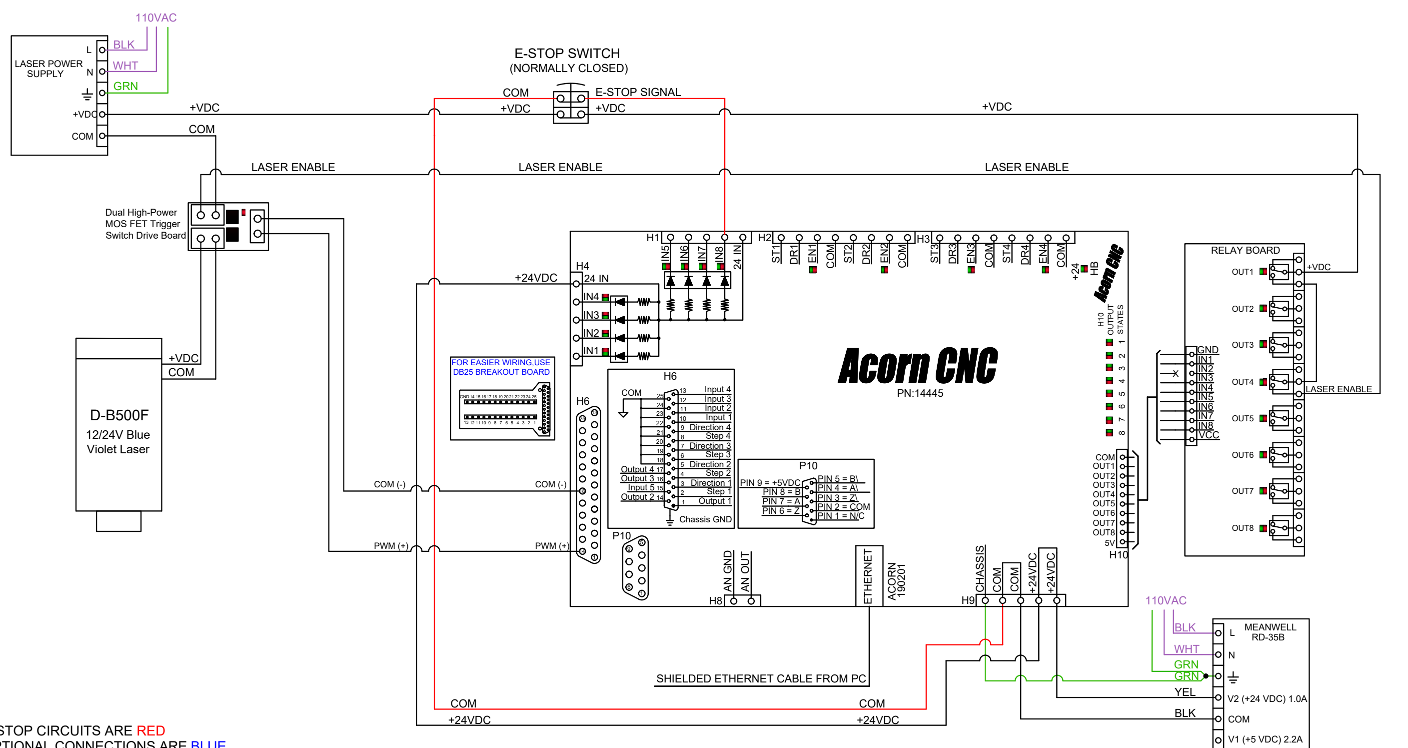


E-STOP CIRCUITS ARE RED
OPTIONAL CONNECTIONS ARE BLUE
GROUNDS ARE GREEN
110VAC IS VIOLET

ACORN I/O	1	2	3	4
INPUTS				
OUTPUTS	NoFaultOut	PWMOutput		LaserEnable
ACORN I/O	5	6	7	8
INPUTS				EStopOK
OUTPUTS				

Title: ACORN_rev4, COMCROW D-B500F LASER		
Date: 201215	Ver: 1	Drawn by: CEM
Filename: S15063.DWG		Sheet 1 of 1

How to tell the difference between Acorn_rev2, Acorn_rev3 and Acorn_rev4
Acorn_rev2 has FIXED green screw terminal blocks
Acorn_rev3 has REMOVABLE green screw terminal blocks
Acorn_rev4 has LED indicators for the Inputs and Outputs



E-STOP CIRCUITS ARE RED
OPTIONAL CONNECTIONS ARE BLUE
GROUNDS ARE GREEN
110VAC IS VIOLET

5.4.3 ZigZagSyncTest Instruction

Requirements: Acorn CNC12 v4.6+ Mill or Router

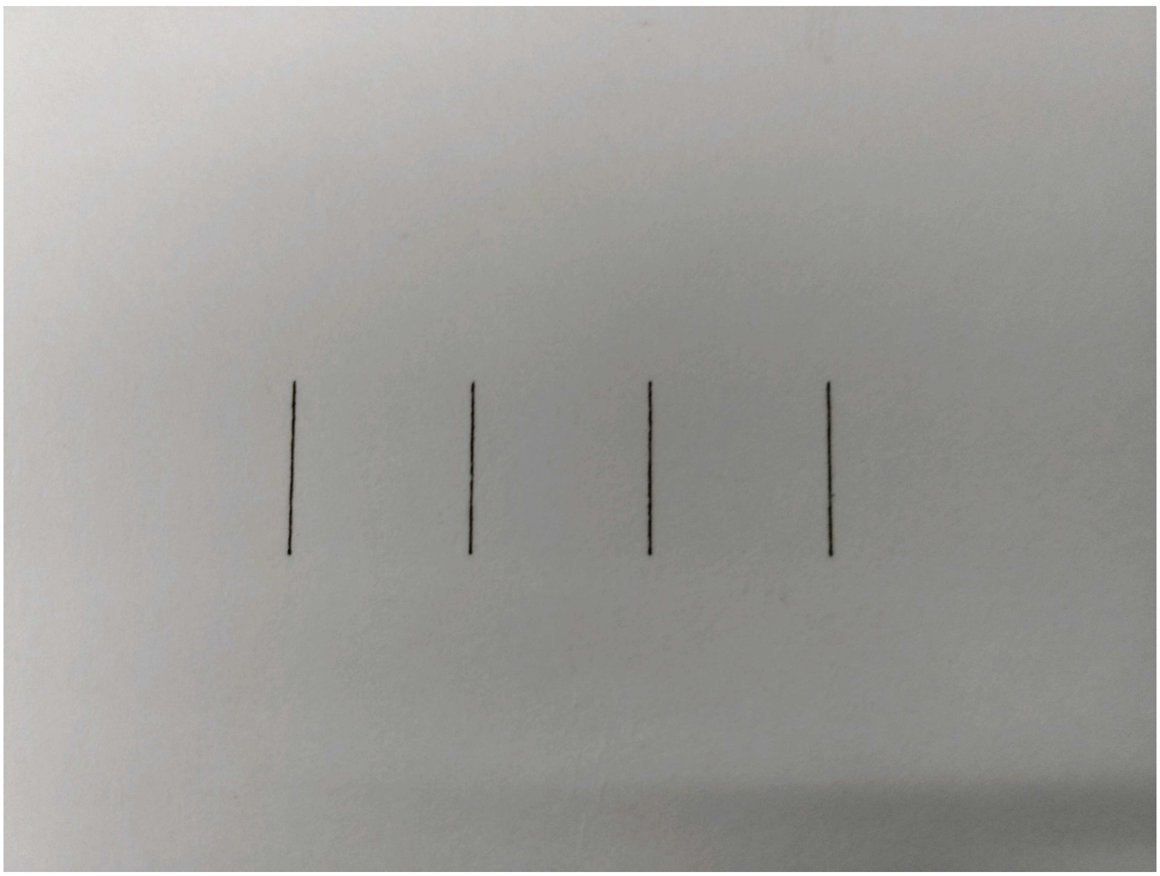
These test programs are included with the installation:

ZigZagLaserSyncTest-X_Axis.cnc
ZigZagLaserSyncTest-Y_Axis.cnc

5.4.4 Purpose

These two programs were created to test for and adjust backlash in your laser table axes.

The program will create four lines in either the X or Y direction by moving back and forth in that axis while firing the laser in short 0.006 inch pulses at specific points in each direction. Below is an example of what should be seen when completed.



Backlash in an axis will produce something like this when a line is examined under magnification.

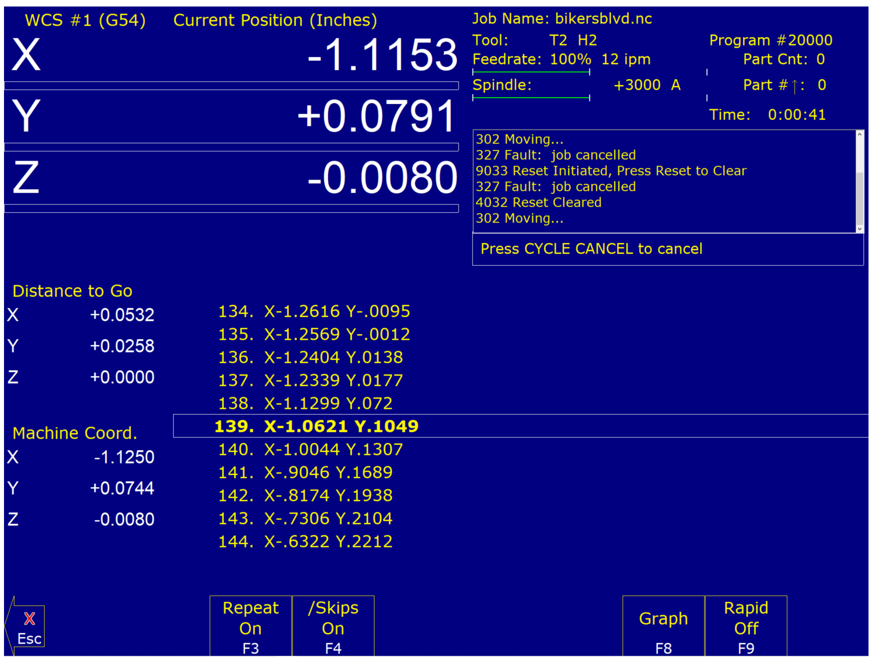
The machine has backlash in this axis and is losing motion when changing directions. Since each “dot” is 0.006 inches long, use an educated guess as to the backlash adjustment needed to align the “dot”. Then, use the wizard to adjust the backlash compensation for that axis, and run the program again to determine if more or less compensation is needed.

6 Running a Job

To start the currently-loaded job, go to the Main Screen and press the **CYCLE START** button on the jog panel. If your control is not equipped with a jog panel, press **ALT-S** on the keyboard.

6.1 Active Job Run Screen with G-code Display

If the Run-Time Graphics option is set to Off, the following screen is displayed while a job is running:



On this screen, the following F-keys are available:

F1 – Feed (-1%) Decrease feedrate override by 1% (available only if keyboard jogging is active)
F2 – Feed (+1%) Increase feedrate override by 1% (available only if keyboard jogging is active)

F3 – Repeat On/Off Toggle the repeat feature for part counting. For more information, see F3 under the Run Menu section later in this chapter.

F4 – Skips On/Off Enable/Disable block skips. For more information, see F4 under the Run Menu section later in this chapter.

F5 – Auto This key will only appear in Single Block mode. When you press this key, it turns on Auto mode and disables Single Block mode. Once Auto mode is turned on, Single Block cannot be re-enabled unless you stop the job. For more information, see F5 under the Run Menu section later in this chapter.

F6 – Stops off This key will only appear if Optional Stops is on. When you press this key, it turns off Optional Stops. Once Optional Stops is turned off, it cannot be re-enabled unless you stop the job. For more information, see F6 under the Run Menu section later in this chapter.

F7 – Feed Hold Turn feed hold on/off (available only if keyboard jogging is active.)

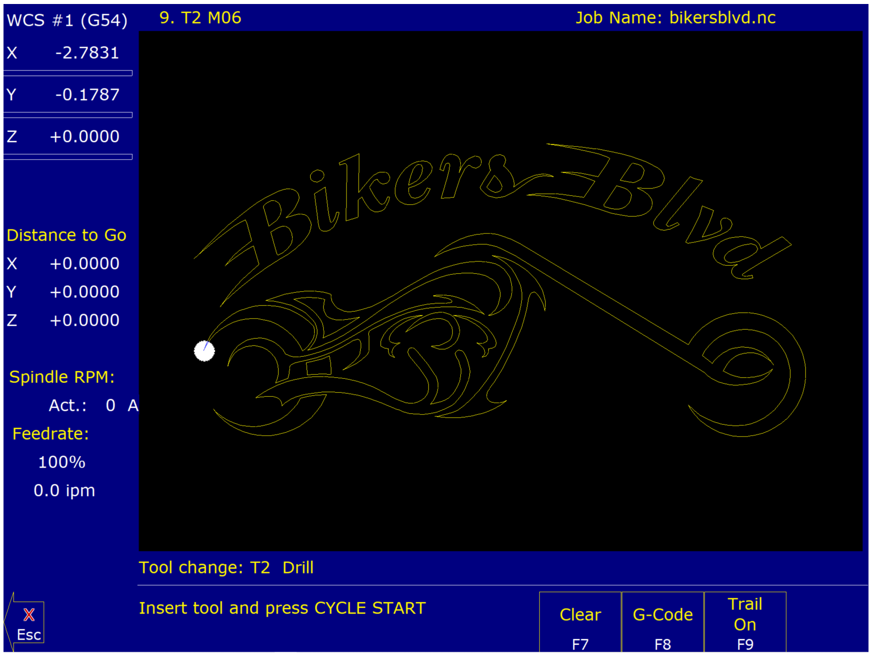
F8 – Graph Switch to run-time graphics screen. This key only appears if the job was started with the run-time graphics option turned on.

F9 – Rapid On/Off Turn rapid override on/off.

For information on other keys that are available while a job is running, see Chapter 2.

6.2 Run-Time Graphics Screen

When a job is running with Run-Time Graphics set to On, a screen similar to the following is displayed:



The following keys are available while the job is running in Run Time Graphics.

F7 – Clear	Clears the trail up to the tool's current position in the program.
F8 – G-Code	Switch to the Job Run Screen with G-Code display.
F9 – Trail On/Off	Turn on/off the tool trail display.

6.3 Canceling a Job in Progress

There are three conventional ways to cancel a currently running job (CNC program). When a job is canceled using any of the following methods, the job's progress will be recorded. This allows the user to restart the job using the Resume Job option or the Search and Run option.

CYCLE CANCEL Pressing this key while a job is running will cause the control to abort the job currently being run. The control will stop movement immediately, clear all M-functions, and return to the main screen. Hitting the escape key on the keyboard is equivalent to hitting **CYCLE CANCEL.**

TOOL CHECK Pressing this key while a job is running will cause the control to stop the normal program movement. In addition, the Z-axis will be pulled to its home position and all M-functions will be cleared. The control will automatically go to the resume job screen.

EMERGENCY STOP (E-Stop) Pressing the **EMERGENCY STOP** button while a job is running will cause the control to abort the job currently being run. The control will stop movement immediately, clear all M-functions, and return to the main screen. Also, the power to all axes will be released.

6.4 Resuming a Canceled Job

If a job is canceled using one of the methods described above, it can be resumed in one of three ways:

CYCLE START Pressing the **CYCLE START** button will restart the job at the BEGINNING of the part program.
Note: Before performing a **F1 – Resume** Job or **F2 – Search**, the tool may need to be positioned in X and Z for cycles that start down inside an ID or behind a shoulder.

F1 – Resume Job (Located in **F4 – Run menu**) Restart the canceled job at or near the point of interruption. See the next section in this chapter entitled "Run menu" for more information.

F2 – Search (Located in **F4 – Run** menu) Restart at a specified point in the part program. See the next section in this chapter entitled "Run menu" for more information.

6.5 Run Menu

Press **F4 – Run** from the main screen to access the Run menu. From this menu, the operator can restart a canceled job or change the way a job will be run.

F1 – Resume Job Access the resume job screen by pressing **F4 – Run** on the main screen to go to the run screen, then pressing **F1 – Resume Job** in the run screen to go to the resume job screen. If the job was canceled by pressing **TOOL CHECK**, the control will go to the resume job screen automatically. From this screen, the user can modify tool offsets, modify the tool library, turn block mode on and off, turn optional stops on or off, graph the partially-completed job, or start the partially-completed job.

The resume job option is not always available. The following situations will cause the resume job option to be unavailable:

- Loading a new job.
- Running a job to completion.
- Parse errors in the job.
- Editing or reposting the job file.
- Loss of power while a job is running.

F2 – Search Invoking this option will bring you to the “Search and Run” menu. This menu will allow you to specify the program line, block number, or tool number at which execution of a program is to begin. Program lines are numbered from the top of the file down, with the first line numbered 1. To enter a block number, place an “N” in front of the number. To enter a tool number, place a “T” in front of the number. Pressing **CYCLE START** from here would start the program at the point you specified.

An extra option unique to the “Search and Run” screen is the **F1 – Tool Change** “Do Last Tool Change” function. This key toggles the tool change option as shown on screen. A “YES” tells the control to perform a tool change so that the tool specified for the line or block has the tool indicated in the program. A “NO” uses the currently loaded tool, regardless of what tool is specified for the line or block being searched.

CNC12 will remember previous searches. They are accessible by pressing the **UP** and **DOWN** arrows in the Search text box.

Note: You cannot search within a subroutine.

F3 – Repeat On/Off This key toggles the repeat feature for part counting. When part counting is in effect and Repeat is on, the job will be automatically run again until the specified number of parts has been run. The On or Off label indicates the state to which the repeat feature will toggle to when pressed, it does not indicate the current state. The current state is indicated in the user window above.

The **Part Count:** prompt is used to set the Part count. Positive values set the part counter to count up, and negative values configure the part count to count down. For example, if 10 is entered in the Part Count prompt, the Part Cnt in the status window changes to 10 and the Part # changes to 0 with an upward arrow indicator. When a job is run and then completes, the Part # will increment to 1. If repeat is on, the job will automatically start again and keep running until the Part # has reached the Part Cnt. If a -10 is entered in the Part Count prompt, the Part Cnt in the status window changes to 10 and the Part # changes to 10 with a downward arrow indicator. When a job is finished, the Part # will be decremented to 9. If repeat is on, the job will automatically start again and keep running until the Part # has reached 0.

F4 – Skips On/Off This function toggles the block skip feature. When block skipping is on, G-code lines that start with a forward slash character '/' are skipped, i.e., they are not processed. Note that because of the way a job is processed (in a pre-processed buffered fashion), the effect of this key may be delayed if you press it while a job is running. The On or Off label indicates the state to which the /Skips feature will toggle to when pressed. It does not indicate the current state. The current state is indicated in the user window above.

F5 – Block Mode Turns single block mode on and off. This is similar to pressing **AUTO/BLOCK**. If Single Block mode is on, the CNC software will stop after each block in your part program and wait for you to press **CYCLE START**. Note that Auto mode is the default mode. If you use this key to turn on Single Block mode and then run a job, Auto mode will be re-instated when the job ends. The current state of this setting is indicated in the user window above.

F6 – Optional Stops Turns optional stops on and off. If optional stops are on, any M1 codes that appear in your program will cause a wait for **CYCLE START** (just like M0). If optional stops are off, M1 codes will be ignored. Note that the default mode for Optional Stops is off. If you use this key to turn on Single Block mode and then run a job, Optional Stops will be set to off when the job ends. The current state of this setting is indicated in the user window above.

F8 – Graph Graphs the part. For more information, see the ***"F8 – Graph"*** section in Chapter 3. If this feature is invoked from the Run and Search screen or the Resume Job screen, then the graphics will show exactly where the searched line or block begins. Dotted lines indicate the portion of the part that will be skipped. Solid lines indicate the portion of the part that will be machined.

F9 – Rapid On/Off This function key toggles Rapid Override. The On or Off label indicates the state to which the Rapid Override feature will toggle when pressed. In the Rapid Override On state, the speed of rapid moves (G0) can be adjusted by the Feedrate Override knob. In the Rapid Override Off state, the speed of rapid moves will be at full speed (max rate).

F10 – RTG On/Off This function key toggles the Run-Time Graphics option. If the option is turned on, Run-Time Graphics automatically starts when the **CYCLE START** button is pressed. This option must be turned on for Run-Time Graphics to be used. If the option is turned off, Run-Time Graphics cannot be started while a job is running.

Machine Parameter 400 determines whether or not **CYCLE START** is enabled on the Run Menu. If Parameter 400 is set to zero, then **CYCLE START** is disabled in the Run Menu. For any other value of Parameter 400, **CYCLE START** is enabled. Note that this does not apply to the Resume and Search sub-menus, where **CYCLE START** is always enabled.

6.6 Power Feed

Press **F4 – Feed** from the Setup menu to access the Power Feed screen. This screen is used to command axis movement. All the operations available on the Power Feed screen may also be performed in MDI with the appropriate M and G codes.

F1 – Absolute Power Feed Press **F1 – ABS** to move an axis to an absolute position at a specified feedrate.

F2 – Incremental Power Feed Press **F2 – INC** to move an axis an incremental distance at a specified feedrate.

F3 – Free XY Press **F3 – Free** to release power to the X and Y motors, allowing you to use your machine manually for these two axes.

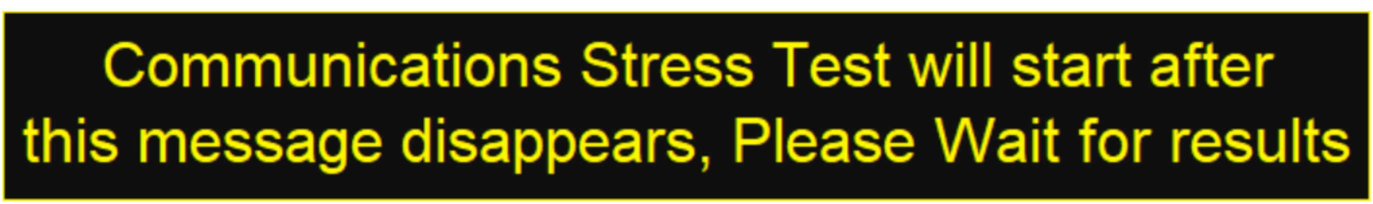
F4 – Power XY Press **F4 – Power** to apply power to the X and Y motors, allowing you to use your machine with the jog panel for these two axes.

6.7 Communications Stress Test

Included in the example files is a communications stress test that can be run by the user. This file can help report communication errors, such as the number of packets resent, generic communication errors, packets out of order, number of NAcks packets sent, and number of NAcks packets received.

To run this test, perform the following actions:

1. **F2 – Load**
2. If not already in the ncfiles directory of the cnct folder, navigate to `\cnct\ncfiles`.
3. Select the “`com_stress_test.cnc`” file
4. Press the “cycle start” button
5. The test will then be run, and the following message will appear:

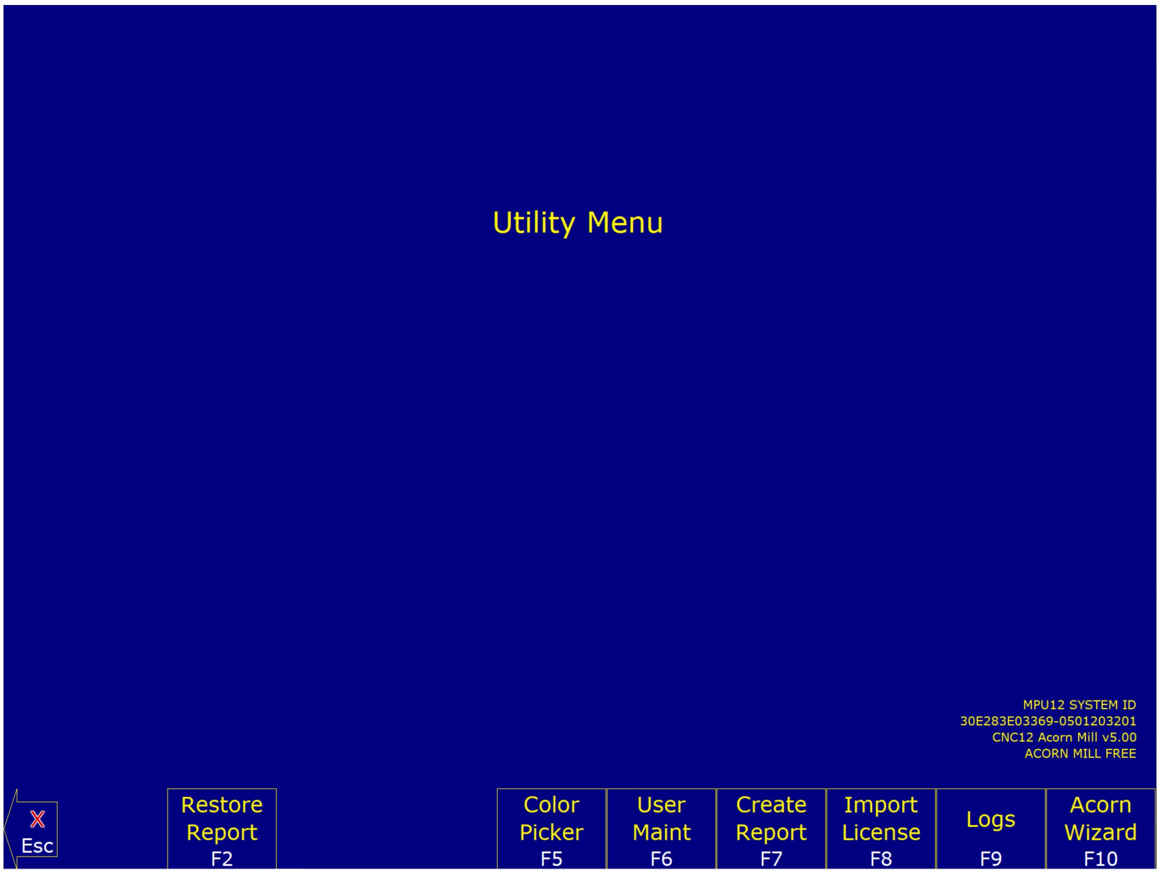


Please allow the system time to process. Once complete, a message similar to the following will appear with your results.

Communications Stress Test
PASSED
max. errors acceptable = 5
Results:
Packets Resent: 0
Generic Communication Errors: 0
Packets Out of Order: 0
NAcks Packets Sent: 0
NAcks Packets Recieved: 0

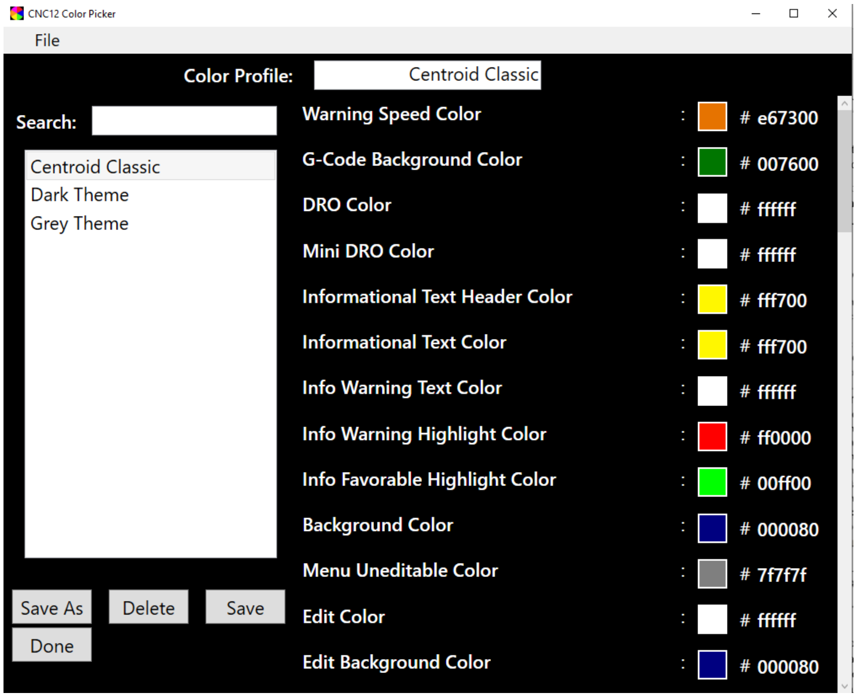
7 The Utility Menu

To get to the Utility Menu, press **F7 – Utility** at the CNC software Main Screen. The model will vary depending on your M-Series Control model.

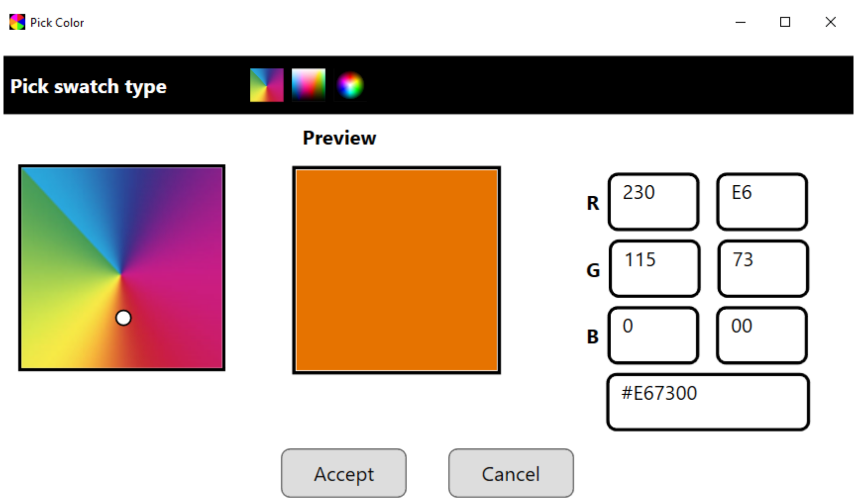


F2 – Restore Report This option is used primarily for restoring a system configuration from a previously-saved *report.zip* file (See **F7 – Create Report**).

F5 – Color Picker This menu allows you to change colors from the default Centroid Classic color scheme.



Centroid Classic as well as several other themes are available to choose from. Edits can be made to individual colors, either by clicking the square or typing in the hex color code manually. Clicking on the colored square next to an item will bring up the Pick Swatch Type screen. If changes are saved to Centroid Classic and a return to the original is desired, select File then New. The next profile will start with Centroid Classic settings.



On the Pick Swatch Type screen, you can use the color wheel to select and preview the color. You can also manually input values via the RGB or Hex Color Code option. When finished, select “Accept”. When finished modifying all colors select, Save to create a new Color Profile. Select “Done” to return to the previous screen.

F6 – User Maint Use this menu to access file options, the manual, or machine notes. The file options (**F1 – File Ops**) menu is a way to access files in a DOS format. The manual (**F2 – Manual**) will open a PDF of the CNC12 Operator’s manual. Machine Notes (**F3 – Machine Notes**) is a text file that serves as a convenient way to store notes about the machine, control customizations, and other notes. It is stored in the cncm folder.

F7 – Create Report Generates a backup of system configuration files called report.zip and copies it to the specified location. Your dealer may then use this file for servicing and troubleshooting purposes. To restore the configuration files from the report disk, press **F2 – Restore Report** from the Utility menu.

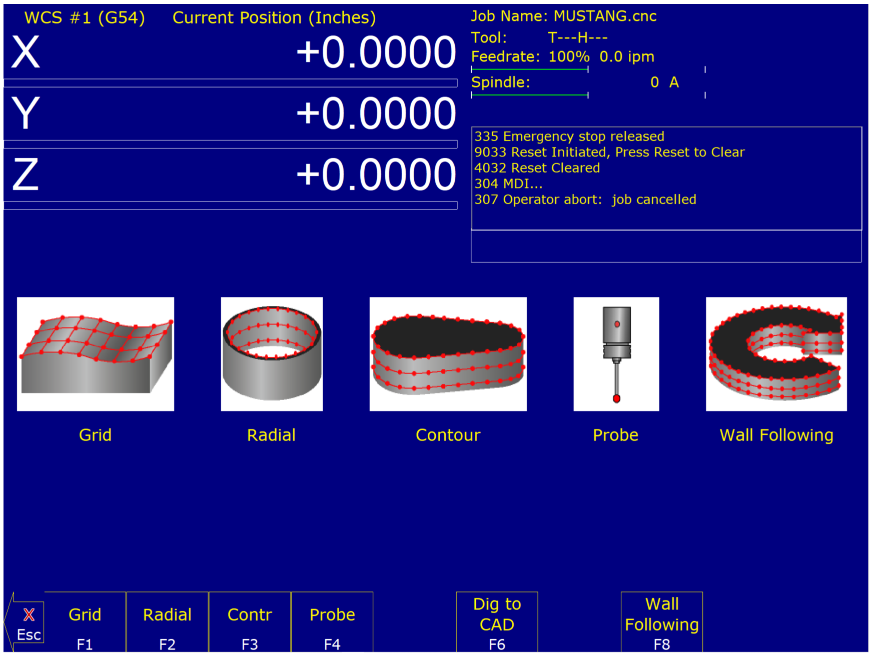
F8 – Import License Use this option to select a license file for use with CNC12.

F9 – Logs Shows the messages and errors that have been logged by the control.

> **F1 – Errors** Displays the error/message log. Use **Page Up**, **Page Down**, **Home**, and **End** to navigate and **ESC** to exit.
>
> **F2 – Stats** Displays counts of errors logged. Use **Page Up**, **Page Down**, **Home**, and **End** to navigate and **ESC** to exit.
>
> **F3 – Export** Exports the log to a destination of your choosing.

F10 – Acorn Wizard Opens the Acorn Wizard program, which helps users more easily alter parameters and other options in CNC12.

8 Digitizing



The Digitize feature of the CNC software can be used to digitize surfaces in a variety of scenarios. The digitizing process creates a file with M & G-codes that represent the digitized surface. If the digitizing probe tip is chosen to match the milling cutter size, the digitized file can then be loaded and run to produce an exact copy of the digitized part.

F1 – Grid Digitize rectangular surface area.

F2 – Radial Digitize inside of a bore.

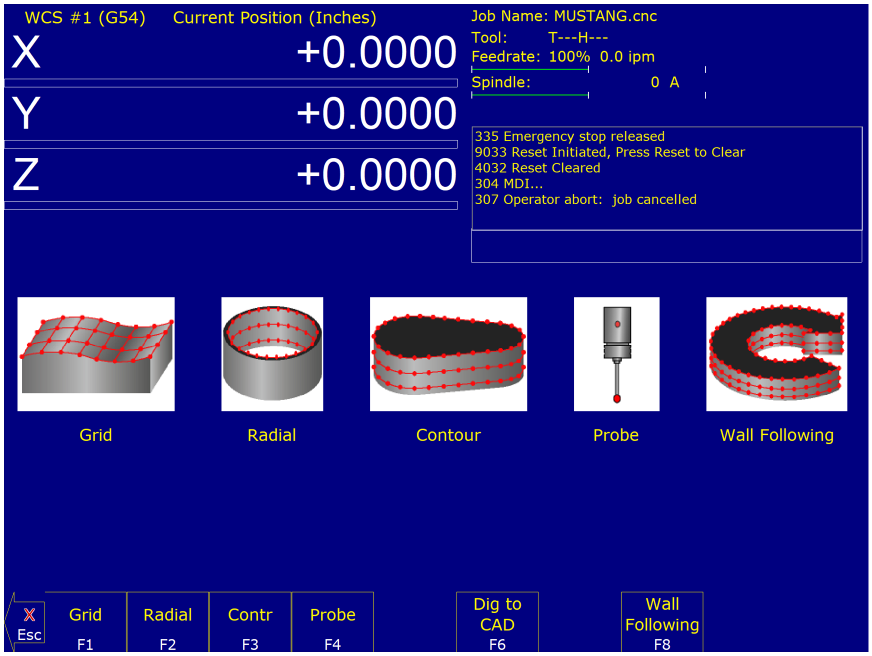
F3 – Contour Digitize inside or outside of a contour (Not available if Parameter 155 is set to 2).

F4 – Probe Select from the Probing Cycles (See Chapter 9 of this manual).

F6 – Dig to CAD Export digitized files for use with a CAD/CAM system.

F8 – Wall Following Digitize inside and/or outside of a contour.

8.1 Grid Digitize (F1 from Digitize Menu)



Grid Digitize Run Setup

To set up a digitizing run, edit the parameters shown and press **CYCLE START**. The control will move through the area to be digitized in a rectangular pattern. At each X-Y point in the pattern, it will measure the Z height of the sample surface and record the coordinates in the data file. Digitizing begins at the current tool position when the **CYCLE START** button is pressed. This position should be in one corner of the digitize area at a Z position higher than any point on the surface.

Grid Digitize Parameters

Type: This sets the algorithm for digitizing: either regular Grid or Surface Following.

X Patch Length: The length of the area to be digitized along the X-axis. A positive value will cause digitizing to proceed in the X+ direction from the starting point; a negative value will cause digitizing to proceed in the X direction. If the value is 0, then digitizing will collect just one stripe along Y.

X Step Over: The distance to move between points on the X-axis. A smaller value should be used for a fine digitize along the X-axis. A larger value should be used for a rough digitize along the X-axis. This distance should be a positive incremental value.

Y Patch Width: The width of the area to be digitized along the Y-axis. A positive value will cause digitizing to proceed in the Y+ direction from the starting point; a negative value will cause digitizing to proceed in the Y direction. If the value is 0, then digitizing will collect just one stripe along X.

Y Step Over: The distance to move between points on the Y-axis. A smaller value should be used for a fine digitize along the Y-axis. A larger value should be used for a rough digitize along the Y-axis. This distance should be a positive incremental value.

Z Maximum Depth: The maximum distance the Z-axis moves below the starting height. If the probe does not contact the surface at the maximum depth, that data point will be recorded as being at the maximum depth, and digitizing will proceed with the next point.

Z Step Up: The distance that the Z-axis moves up after making contact, before the control attempts to move X or Y. A small value should be used when digitizing parts with gentle slopes; a larger value when digitizing parts with many steep walls.

Axis to Move First: The axis (either X or Y) that moves all the way across the digitize area with each pass.

Digitize File Name: The base name of the output file in which the digitize data is stored. An extension of .dig will automatically be appended to the name for replay as a CNC job.

Replay Feedrate: The feedrate to include with the G1 command on the first line of the data file. If the data file is run as a CNC program, this is the feedrate at which the machine will retrace the digitized surface.

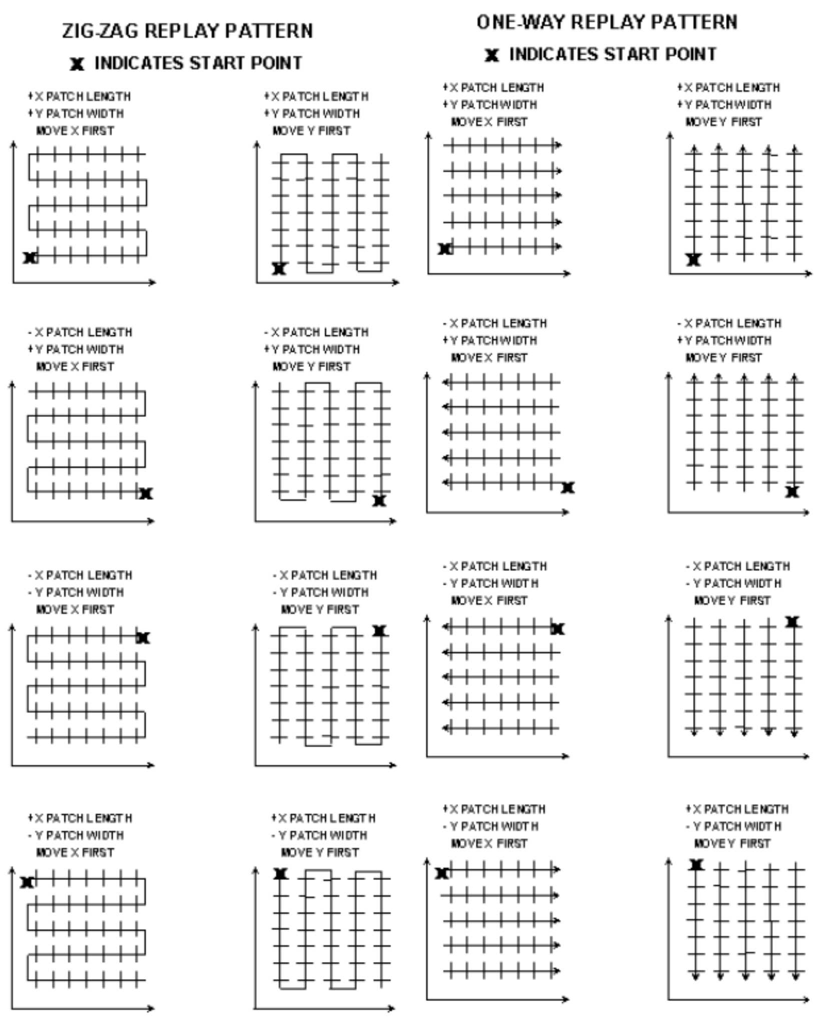
Multiple Patch: Indicates whether or not this digitizing is a continuation of an earlier digitizing. Choose NO if the current digitizing is the first or only digitize run for the part to be digitized. Choose YES if the current digitizing is not the first digitize run for the part. If Yes is selected, specify the name of a digitize file of a previous multiple patch.

Replay Pattern: Indicates the replay movement pattern. If ZIG ZAG is selected, the replay pattern will alternate between positive and negative directions on each successive pass. If ONE WAY is selected, the replay pattern will maintain a constant "one-way" direction throughout the playback.

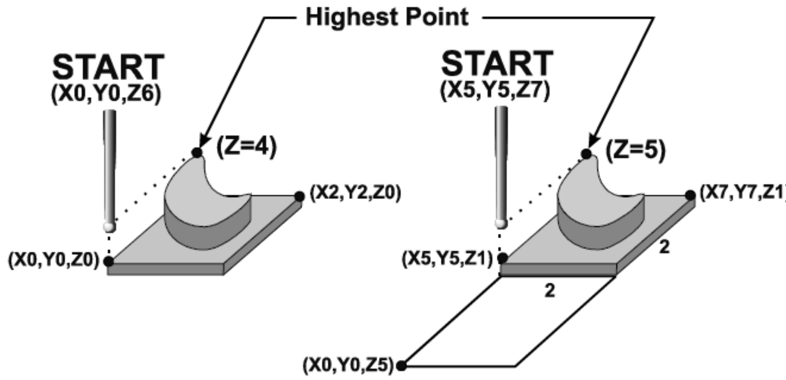
Probe Diameter: Indicates the probe tip diameter.

Grid Digitize Notes

1. A guide to the possible grid digitizing paths is as follows:

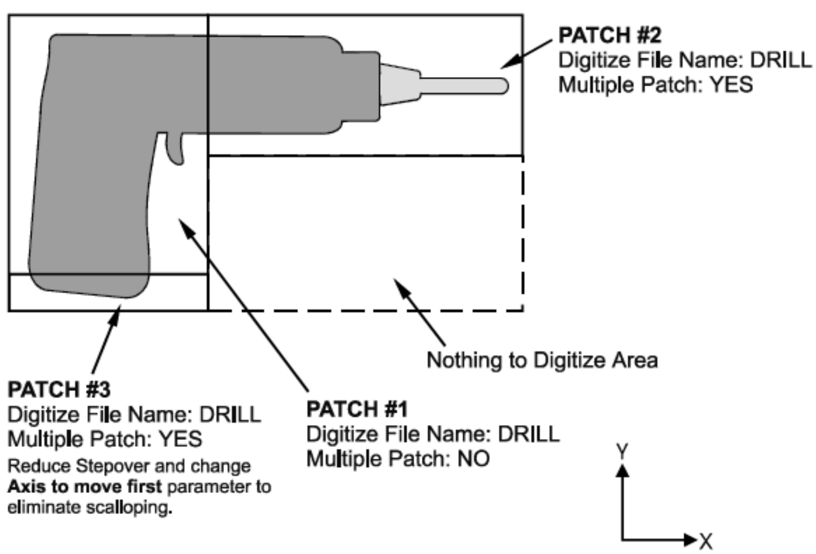


2. A digitizing patch can be located anywhere in the coordinate system. The digitizing starting point is referenced from the part zero. For example, setting up digitizing as shown in the figure on the right below will record the first point at (X5, Y5, Z1) and the last point at (X7, Y7, Z1). If the digitizing replay starting point is desired to be at the part zero, be sure to set the part zero equal to the digitizing start point, as shown in the figure on the left below. This orientation will record the first point at (X0, Y0, Z0) and the last point at (X2, Y2, Z1). For more information on part setup see Chapter 4.



3. A good technique for calculating Z maximum depth is to touch off at the lowest surface of the part to be digitized, and set the part zero's Z value to Z0. Then, jog the probe tip to a point higher than the highest surface of the part to be digitized. Note the displacement in the Z-axis. Again, set this Z height to Z0 and use the noted displacement for the Z maximum depth.

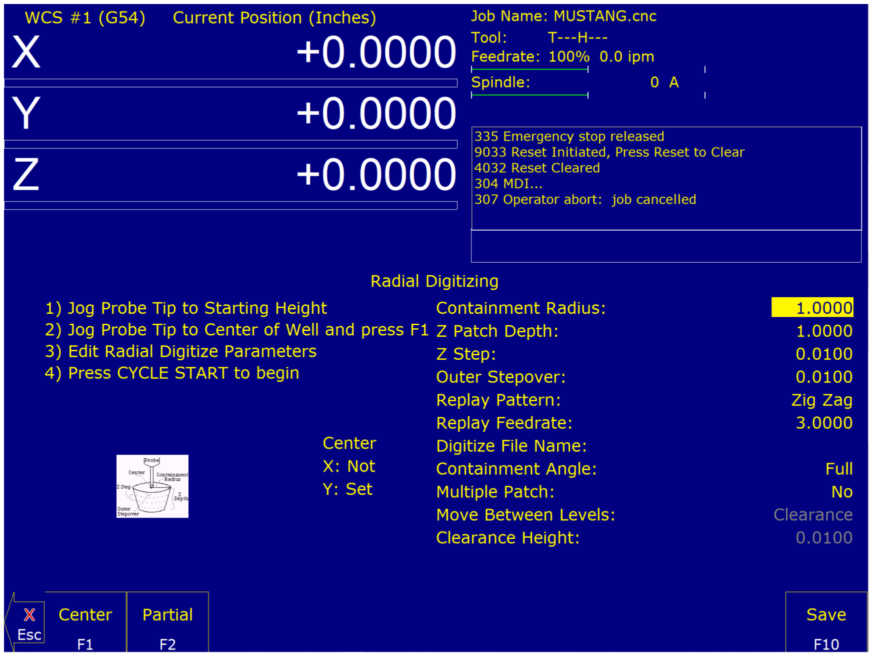
4. Multiple patches are useful in the following situations: completing a canceled digitize run, digitizing parts with large areas that contain nothing to be digitized (shown below), and patching vertical walls to eliminate scallops caused by the cutting tool.



The drill shown in the previous example is L-shaped. Therefore, it can be digitized faster and more efficiently using three rectangular patches than digitizing the complete area with a single patch.

Digitizing the entire part and then adding multiple small patches along the walls can avoid vertical wall scalloping. If a vertical wall extending along the X axis needs to be cleared of scallops, just add a small patch running the length of the wall. Set the “Axis to Move First” parameter to Y. This will clear the scallops.

8.2 Radial Digitize (F2 from Digitize Menu)



Setting up a Radial Digitize Run

To set up a digitize run, edit the parameters shown. Jog the probe tip to the starting height and center of the bore to be digitized. Press **F1 – Center** to define the center position for digitizing. This center position will be used as the center of all radial digitizing runs until you leave the radial digitize menu or redefine the center. If you are using a full angle, you can now press **CYCLE START** to begin digitizing. If you have specified a partial angle, press **F2 – Partial** to define the partial angle (see Setting the Partial Angle section in this chapter). After defining the partial angle, pressing **CYCLE START** will start the digitize run.

WARNING: The probe must be able to retreat to the center from any position on the digitize surface. If the digitize surface contains features that do not allow for the probe to exit after entering, a probe crash may occur! See Radial Digitize Note 2 later in this section.

Radial Digitize Parameters

Containment Radius: The maximum distance from the center position that the machine should look for a digitize data point. This parameter is used to contain the probe within a circle, with its radius centered at the center position. If the probe does not contact the surface before reaching the maximum radius, the data point will be recorded as being at the maximum radius and digitizing will proceed with the next point.

Z Patch Depth: The depth of the patch to be digitized along the Z-axis. A positive value will cause digitizing to proceed in the Z+ direction from the starting point; a negative value will cause digitizing to proceed in the Z-direction.

Z Step: The distance to move between points on the Z-axis. A smaller value should be used for a fine digitize along the Z-axis. A larger value should be used for a rough digitize along the Z-axis. This distance should be a positive incremental value.

Outer Stepover: The distance to move between points on one contour. A smaller value should be used for a fine digitize along any one contour. A larger value should be used for a rough digitize along any one contour. This distance should be a positive incremental value.

Replay Pattern: Indicates the replay movement pattern. If Zigzag is selected, the replay pattern will alternate between positive and negative angle directions (CW and CCW) on each successive contour. If CW or CCW is selected, the replay pattern will maintain a constant angle direction throughout the playback.

Replay Feedrate: The feedrate to include with the G1 command on the first line of the data file. If the data file is run as a CNC program, this is the feedrate at which the machine will retrace the digitized surface.

Digitize File Name: The base name of the file in which the digitize data is stored. The file has an extension of *.dig* for CNC replay.

Containment Angle: Indicates whether or not the digitizing is to follow a full circle or a partial sector. Choose **Full** if 0 to 360 degrees is desired. Choose Partial if some other angles are needed. These partial angles can then be changed later (see the Partial Digitizing Sector Setup section that follows).

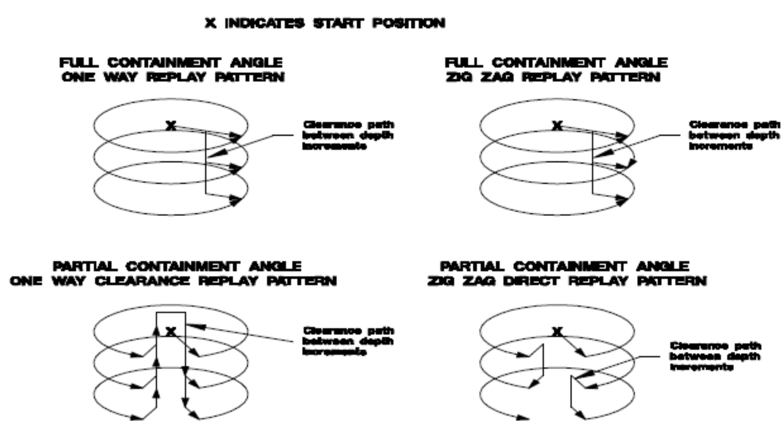
Multiple Patch: Indicates whether or not this digitizing is a continuation of an earlier digitizing. Choose No if the current digitizing is the first or only digitize run for the part to be digitized. Choose Yes if the current digitizing is not the first digitize run for the part. If Yes is selected, specify the name of a digitize file of a previous multiple patch.

Move Between Levels: This field is enabled only if Partial and CCW or CW option is selected. It indicates the move between Z levels on replay of a partial sector radial digitize file. This move may now be done in three different ways: Clearance (which goes to the clearance height as in previous versions), Center (which goes to the digitizing center and then to the Z level of the next pass), and Direct (which goes directly to the starting point of the next pass).

Clearance Height: This field is enabled only if Partial, CCW or CW replay pattern, and Clearance Move type option are selected. This distance indicates the clearance height needed to move the cutter from the end of one contour to the beginning of the next contour. This distance should be a positive value.

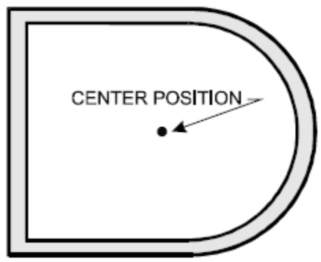
Radial Digitize Notes

1. A guide to possible radial digitizing paths is as follows:

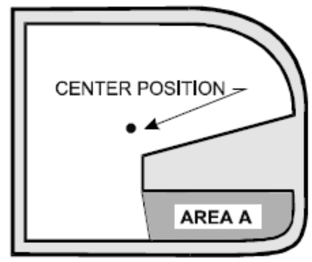


2. When radial digitizing, make sure the probe can fully retract to the center position without obstructions. Observe the two parts below. The cross section on the left has no obstructions that could keep the probe from full retraction to the center position. The cross section on the right does not allow the probe to retract to the center in **Area A**. This area will cause a probe crash. Single patch digitizing of parts such as this should be avoided. Use two or more patches to digitize the part on the right (in this case, you could divide the part in half horizontally, and do each half separately).

IDEAL PART

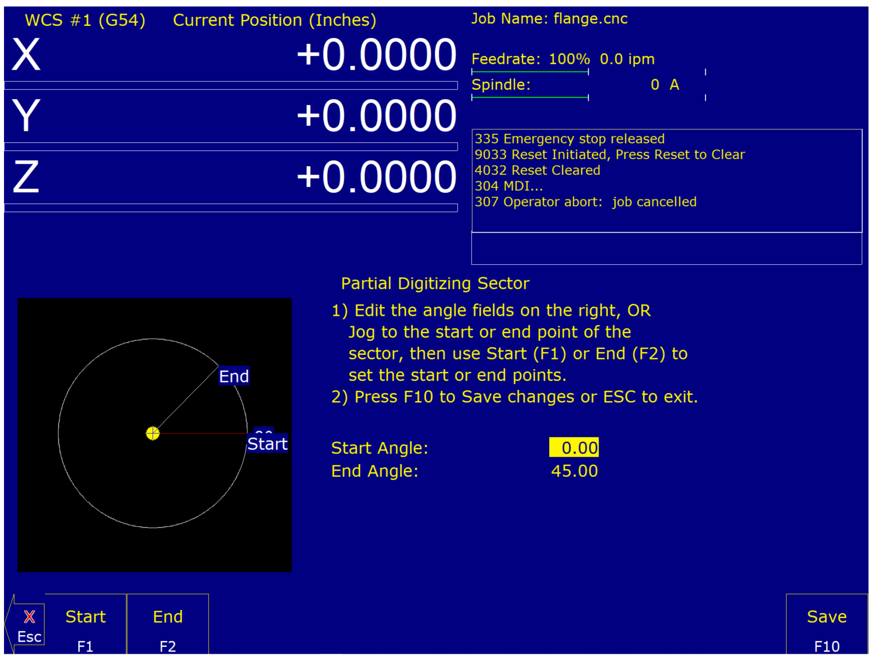


PROBLEM PART



Partial Digitizing Sector Setup

If you set the Radial Digitize Containment Angle to "Partial", then you must set up the Digitizing Setup by pressing **F2 – Partial** from the CNC12 software Radial Digitize Screen.



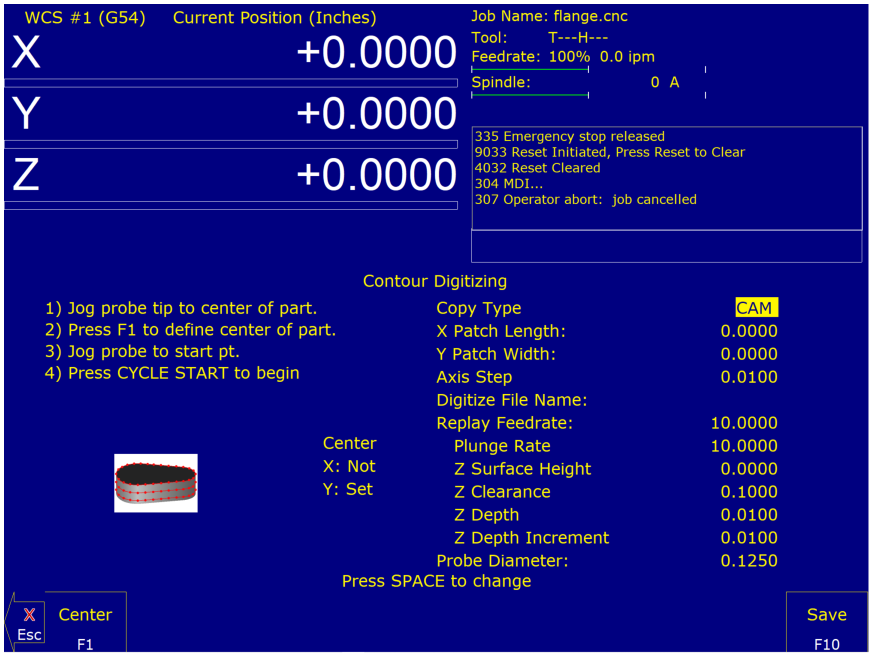
The partial sector can be setup with one of two methods:

1. One method is by editing the start and end angles directly. The start angle is referenced from zero degrees and defines the beginning of the digitizing sector. The end angle is referenced from zero degrees and defines the end of the digitizing sector.
2. The second method involves jogging the probe tip and touching off of the digitize surface. By moving the probe tip to positions on the digitizing surface, one can set the angles. To set the start angle, jog the probe tip to the position on the digitizing surface where the digitizing is to begin and press **F1 – Start** to define this as the start angle.

   ***Notice:** The picture of the sector and start angle's value change to reflect these settings. To define the end angle, follow the procedure above but press **F2 – End** instead to set the end angle.

Regardless of the method used to define the start and end angles, pressing **F10 – Save** saves the angles and exits back to the radial digitize menu. Pressing **ESC** will return to the radial digitize menu without making changes to the start and end angles.

8.3 Contour Digitize (F3 from Digitize Menu)



Contour Digitize Run Setup

To set up a digitizing run, jog the probe tip to the center of the part and hit **F1 – Center** to assign that as your center point.

Select CAM for a true CAM shape contour or Wall for irregular shapes that require wall following. Enter the rest of the parameters for the part and digitizing job as shown below.

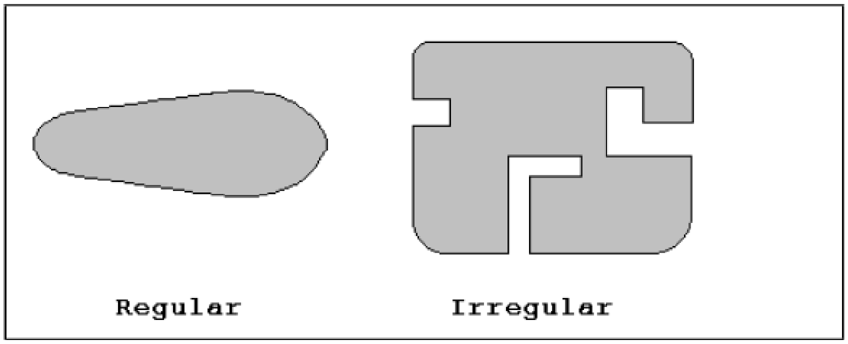
Jog the probe to a starting point and press **CYCLE START**. The control will move the probe toward the center line in the +/- X direction until it comes into contact with the part. At each point of contact, the X and Y coordinates will be recorded in the data file.

The probe will continue around the contour until it returns to the starting point to complete the cycle. Based on the starting point and the first point of contact, the digitize software will determine if the contour is internal or external.

Important note: When probing an external contour, make sure that the probe comes in contact with the part on the first move. If it doesn't, the software will compensate for the outside of the part as if it were an internal contour, causing your part to come out larger than you want it to be.

Contour Digitize Parameters

Copy Type: Toggle between CAM or Wall. Use CAM for regular shapes (no extreme direction changes) and use Wall Following for contours with irregular shapes. See example below.



X Patch Length: The length of the contour to be digitized, along the X-axis.

Y Patch Width: The width of the contour to be digitized, along the Y-axis.

Axis Step Over: The distance to pull away from the surface in the X and/or Y direction. A larger value should be used for a rough digitize along the Y-axis. This distance should be a positive incremental value.

Digitize File Name: The base name of the file in which the digitize data is stored. The file has an extension of .cam for CNC replay and is stored in the c:\cncm \ncfiles directory.

Replay Feedrate: The feedrate to include with the G1 command on the first line of the data file. If the data file is run as a CNC program, this is the feedrate at which the machine will retrace the digitized surface.

Plunge Rate: The feedrate at which the Z axis plunges between successive depth passes.

Z Surface Height: Surface height of material for reproduction of digitized parts.

Z Clearance: Clearance amount to rapid to above the surface of the part during replay.

Z Depth: Depth of the part as measured from the surface height.

Z Depth Increment: Depth of cut for each Z step of the part.

Probe Diameter: Diameter of probe tip used to digitize the part.

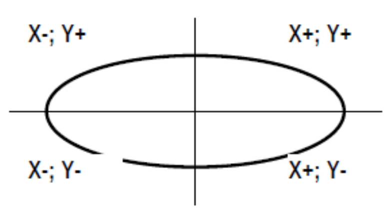
Contour Digitize Notes

Contour digitizing creates an M&G code file with a .cam extension. The structure of the .cam file starts with a header of comments indicating some of the parameters used when digitizing the contour. Next is the contour itself, which is outputted as a subprogram. The M& G-codes are preceded by an O9800 (start of subprogram) and followed by an M99 (end of subprogram). The end of the .cam file contains the initial positioning moves and a call to the contour subprogram (G65 P9800).

Probing direction: When starting the digitizing cycle, choose a starting point where the X travel will contact a point on the cam on the Y axis. The probe tip will move toward the center line in the X direction until it contacts the cam, then will move either clockwise or counterclockwise around the cam, depending on which quadrant you started the cycle in (see Table 1 below).

Table 1 – Probe direction by starting quadrant

X	Y	Probe travels
-	-	CW
-	+	CCW
+	+	CW
+	-	CCW
-	0	CW
+	0	CCW



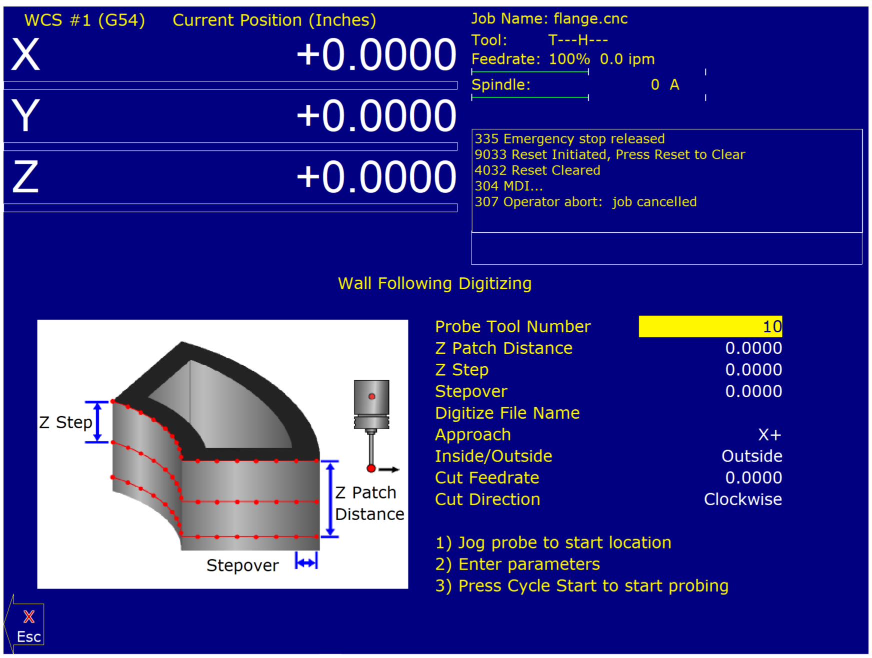
If the probe bypasses a contact point on the Y axis, it will continue moving in the X direction across the center line until it reaches the patch length limit and faults out.

Canceling a job: Unlike grid or radial digitizing, if you cancel a contour operation before it is completed, you will not be able to restart the contour at the point of interruption to continue the cycle. You will need to start over.

Before running a job: Before running a job created by contour digitizing, you will need to add some information to the file to define any required tool change, cutter compensation, and height offset commands.

1. Do a search in the G code for the phrase "Add Comp Here."
2. Refer to the descriptions of G40, G41, and G42 in Chapter 12.
3. Add the proper G code to the file after the "Add Comp Here" prompt.
4. Save the file and run your job.

8.4 Wall Following Digitizing (F8 from Digitize Menu)



Setting up a Wall Following Digitizing Run

To set up a digitize run, edit the parameters shown. Jog the probe tip to the XY starting location (inside or outside of the part to be digitized) and then jog Z to the level of the first digitize pass. Press **CYCLE START** to start the digitize run.

Wall Following Digitizing Parameters

Probe Tool Number: The tool number of the probe.

Z Patch Distance: The total Z distance to digitize, incremental from the first pass. Entering a negative value will cause the routine to step down in the Z-axis. Entering a positive value will cause the routine to step up in the Z-axis.

Z Step: The Z amount to step down per pass.

Stepover: The distance between the digitized points.

Digitize File Name: The base name of the output file in which the digitize data is stored. An extension of .dig will automatically be appended to the name for replay as a CNC job.

Approach: The direction of the first digitizing move.

Inside/Outside: Starting location of the probe, either inside or outside of the part.

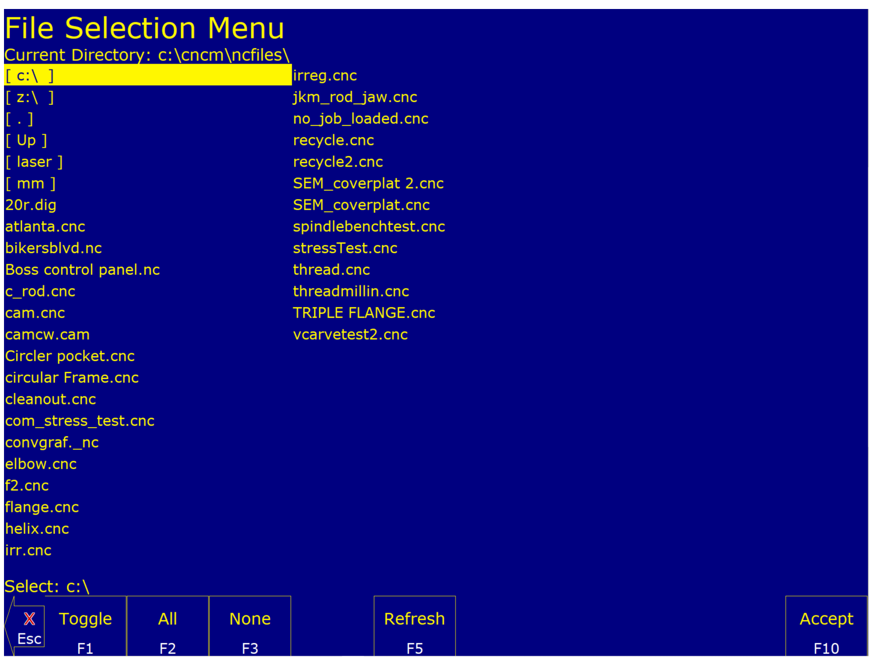
Cut Feedrate: The replay feedrate. This is the feedrate that will be output to the file specified by the Digitize File Name, detailed above. This will cause the digitized data to be replayed at this specified feedrate when the output data is run manually as a CNC job.

Cut Direction: The Clockwise or Counter-Clockwise direction of the output data. This also defines the direction of digitizing.

Wall Following Digitizing Notes

Initially, the output data is uncompensated. Therefore, the resultant output data points will be offset by the probe tip's radius amount. If you want to run this data as a CNC job, you must use a cutter that is the same shape and diameter as the probe tip. An alternative to this is to use a CAD/CAM software package to remove the probe tip offset and apply the desired compensation.

8.5 Dig to CAD (F6 from Digitize Menu)

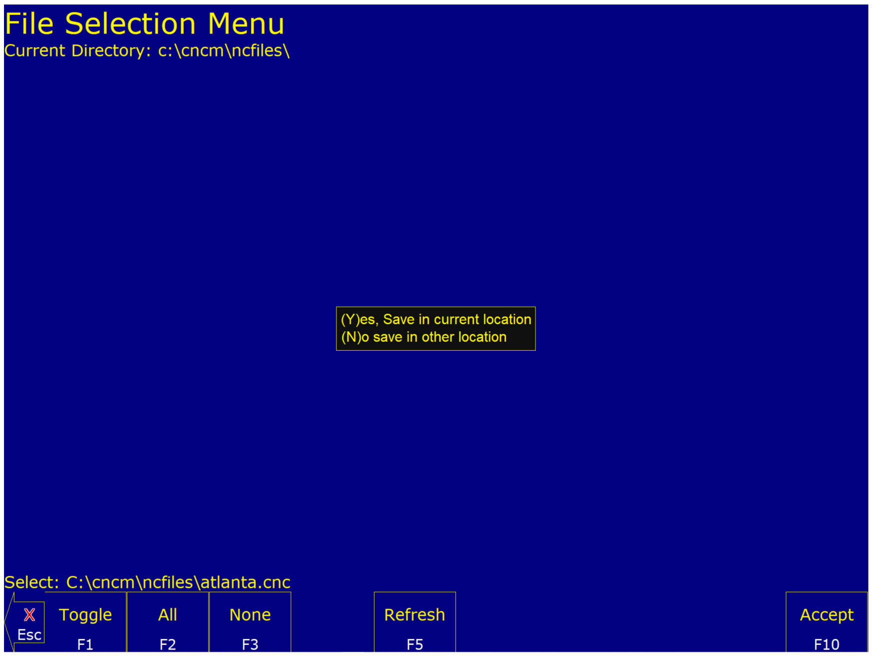


The Dig to CAD feature of the CNC software is used to export digitized files for use with CAD/CAM software. The digitized files are converted to point cloud data that is easily readable by most CAD/CAM systems. Digitized files have either a .DIG or .DIG5 extension. Files with the extension .DIG are created from the grid or wall following digitizing routines, while .DIG5 files are created from 5-axis digitizing toolpaths. The resulting point cloud data files will have the same file name as the .DIG and .DIG5 files they were created from, only with a .TXT extension. These files can then be imported into any CAD/CAM software or viewed with a simple text editor.

In the case of 5-axis CNC controls, where the 4th and 5th axes are rotary axes, parameters 116–119 are used in conjunction with Dig to CAD to properly position the collected data (see Chapter 15 for more information on these parameters).

Converting Digitized Data

To export digitized data, first select the files you wish to convert by highlighting them with the arrow keys and using either **F1 – Toggle** or the **Space** bar to select them. When a file has been selected, an asterisk (*) will appear to the left of the file name. Once you have selected all the files you wish to convert, press **F10 – Accept**.



To convert the files and save them in the same directory as the digitized files, press **Y**. To save the files in a different location (such as a USB drive) press N and navigate to the desired directory using the file menu. Press **F10 – Accept** to convert and save the files in that location.

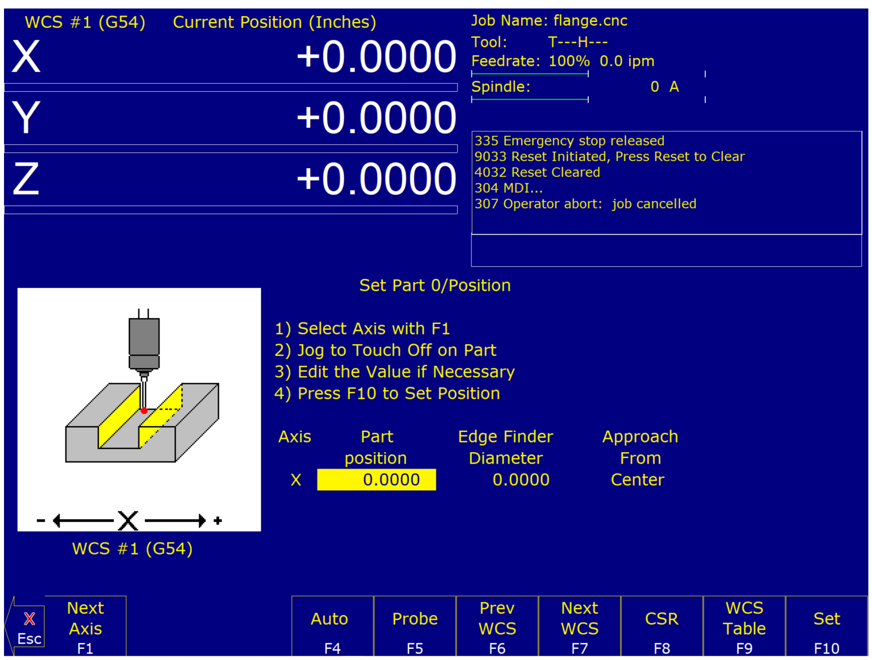
9 Probing

Attention! Refer to the Probe Parameters sections at the end of this chapter before using any probe.

9.1 Part Setup with Probing

Single axis, single surface probing is available on the Set Part 0/Position screen using the **F4 – Auto** key. This allows you to probe various surfaces to define the part coordinate system. Multi-axis and multi-surface probing cycles are available on a separate screen, accessible from Set Part 0/Position with the **F5 – Probe** key or from **F4 – Probe** in the digitize screen. These allow you to locate the center points and corners of differently-shaped parts.

Brushless motor note: If you experience excessive vibration on a brushless drive system, use Parameter 10 to select Smooth Deceleration in Probing Moves. See Chapter 15 for more information.



WARNING: Before manually jogging any probe, make sure that the machine Feedrate is slow (less than 10 in/min) or damage to the probe may result!

Automatically Setting Part 0

The part 0 can be found using the probe. Make sure your probe height and diameter offsets are set for the tool number you assigned to the probe, and that Parameter 12 is set to that tool number. The Edge Finder Diameter will be set automatically.

To set Part 0 using the probe:

1. Select the current Work Coordinate System by pressing **F6 – Prev WCS** or **F7 – Next WCS**. Then, select the axis you want to probe by pressing **F1 – Next Axis**.

2. Manually jog the probe to about 1/2 inches away from the surface you wish to define. Make sure that the approach direction to the part is set properly. Probe the selected axis by pressing **F4 – Auto**. When the surface is found, the control will assume this point to be the new axis 0.
3. If you want this probed surface to be something different than 0, enter the value by the using arrow keys to highlight Part Position, then type in the value and press **F10 – Set**.

Repeat steps 1–3 to set the remaining axes using the probe. Any previously-entered Edge Finder Diameter or Tool Number value will be discarded.

Finding Center/Corner Points

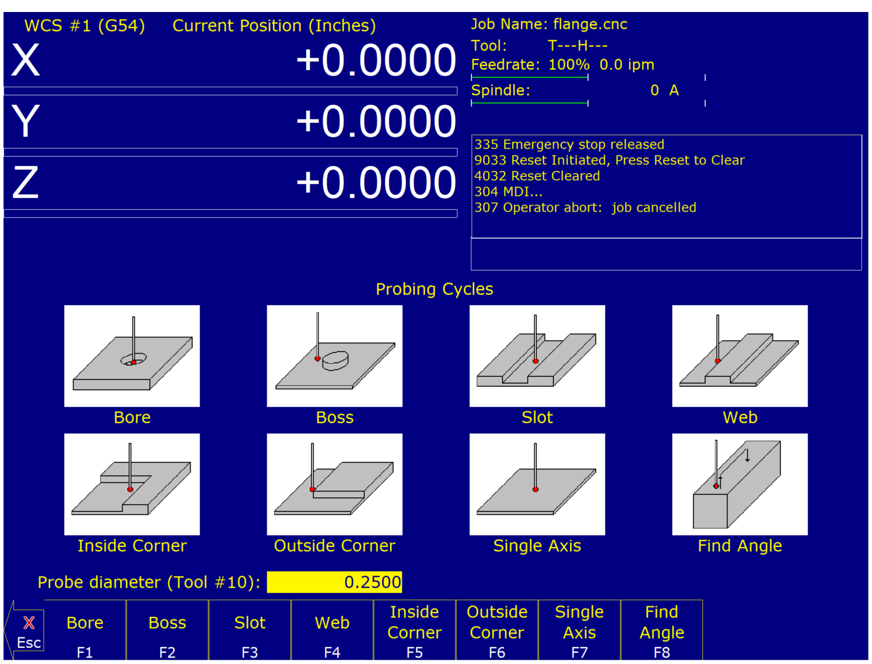
To enter the Probing Cycles screen, press **F5 – Probe** from the Set Part 0/Position screen. You can locate the center of a bore, boss, slot, or web. You can also find an inside corner, outside corner, or single axis. The corner points do not have to be right angles. The Edge Finder Diameter does not need to be entered, since these cycles place the probe directly over the center or corner of the part.

9.2 Calibrating the Probe Tip Diameter

You can calibrate the probe tip diameter to compensate for pre-travel (the amount that the probe deflects before it actually trips). Simply enter a probe tip diameter of zero, probe out a precision bore with a known diameter, and enter the difference between the reported bore diameter and the found bore diameter as the actual probe tip diameter.

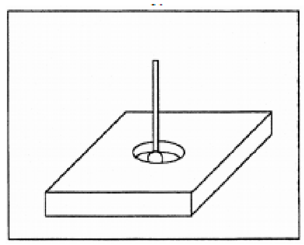
9.3 Probing Cycles

You can enter the Probing Cycles screen from either the Set Part 0/Position screen (**F5 – Probe**) or the Digitize menu (**F4 – Probe**). The Probing Cycles screen is shown below:



The probing cycles will report the location and dimensions, as applicable, of the probed feature in a floating dialog box. The dimensions are adjusted to compensate for the probe tip diameter entered in the Offset Library (see Parameter

12). For your convenience, you can edit the probe diameter on this screen as long as the Tool Number, as set in Parameter 12, is not 0. During the probing cycles, the probe will move at the speeds specified in Parameters 14 and 15. Refer to the Probe Parameters section later in this chapter for more information.

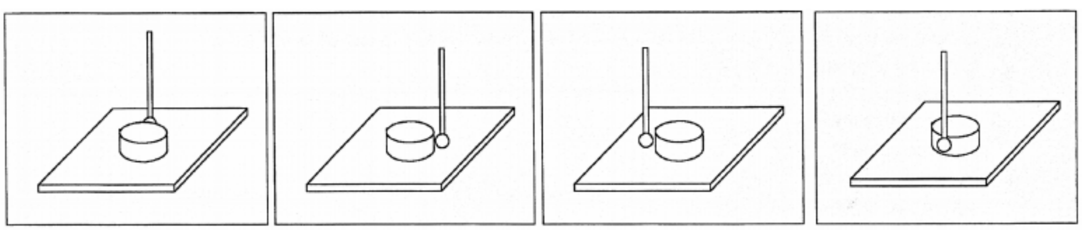


F1 – Bore Press to enter the Bore screen. A picture similar to the one shown above will appear with instructions. Follow these steps:

1. Make sure the probe is clear of any obstacles.
2. Manually jog the probe inside the hole. The probe tip should be just below the top edge of the surface.
3. Press **CYCLE START** to start the probing.

At the end of probing, the probe will be positioned at the center of the bore, and the X and Y positions will be shown on the screen.

Press **ESC** to return to the Set Part 0/Position screen or digitize screen.

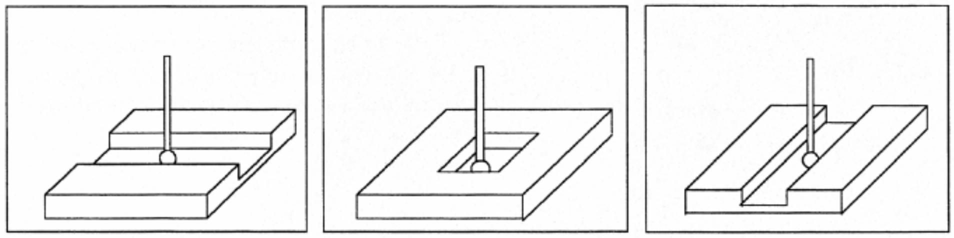


F2 – Boss Press to enter the Boss screen. A picture similar to the ones shown above will appear, with instructions and two input fields. Follow these steps:

1. Press **F1 – Orient** to select the orientation of the probe with respect to the Boss. You will see one of the screens shown above.
2. Slowly jog the probe to the approximate orientation as shown in the picture. Be sure to give enough space for the probe tip to clear any obstacles during the jog.
3. Enter the approximate Boss diameter.
4. Highlight the Z clearance amount by pressing the down arrow key. Enter approximate distance (in the Z direction) the probe must move to lift up over the Boss.
5. Press **CYCLE START** to start the probing cycle.

If the Z clearance you entered is too small, the probe will stop and show an error message. Correct this problem by repeating the previous steps. If the approximate diameter you entered is too small, the probe will bounce by 10 percent of its diameter across the top surface until it finds the correct edge, the additional distance searched is equal to Parameter 16, or a travel limit is reached.

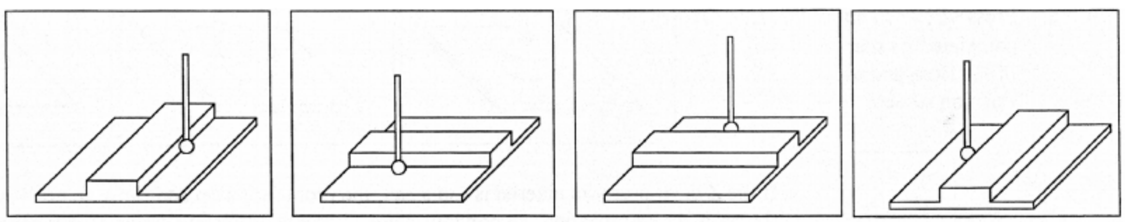
Once the probing cycle is complete, the probe will be positioned at the center of the boss at the Z clearance level entered. Press **ESC** to return to the Set Part 0/Position or digitize screen.



F3 – Slot Press to enter the Slot screen. A picture similar to the ones shown above will appear along with instructions:

1. Press **F1 – Orient** to select the orientation of the probe with respect to the slot.
2. Slowly jog the probe to the approximate position shown above in the picture. During this jog, make sure you have enough space between the probe and the part.
3. Press **CYCLE START** to begin probing.

Once the cycle is finished, the probe will be located at the center of the slot. Press **ESC** to return to the Set Part 0 – Position or digitize screen.

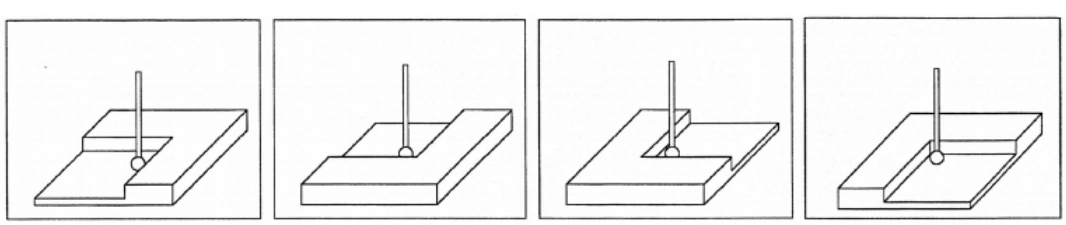


F4 – Web Press to enter the Web screen. A picture similar to the ones shown above will appear, with instructions and two input fields. Follow these steps:

1. Press **F1 – Orient** to select the orientation of the probe. You will see one of the screens shown above.
2. Slowly jog the probe to the approximate position shown in the picture. During this jog, be sure to give enough space between the probe and the part.
3. Enter the approximate Web width.
4. Highlight the Z clearance value using the up or down arrow key. Enter the approximate distance that the probe has to travel in order to lift up over the Web.
5. Press **CYCLE START** to start the probing cycle. Once the probe has completed its search, it will automatically position itself to the centerline of the web.

If the Z clearance you entered is too small, the probe will stop and show an error message. Correct this problem by repeating the previous steps. If the approximate width you entered is too small, the probe will bounce by 10 percent of its width across the top surface until it finds the correct edge, the additional distance searched is equal to Parameter 16, or a travel limit is reached.

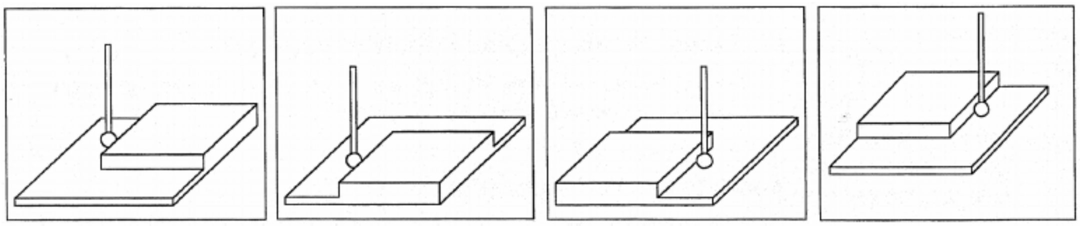
Once the probing cycle is complete, the probe will be positioned at the center of the web, at the Z clearance level entered. Press **ESC** to return to the Set Part 0/Position screen or digitize screen.



F5 – In Corner (Inside Corner) Press to enter the Inside Corner screen. One of the pictures shown above will appear with instructions. This cycle is similar to that of a slot cycle; the main difference is that you need to enter a clearance amount.

1. Press **F1 – Orient** and the screen will cycle through one of the probe orientations shown here.
2. Enter the Clearance Amount. This is an approximate distance (in the Z direction) that the probe must move to clear the corner
3. If the corner is rounded, jog the probe far enough away for it to miss the curved area during the probing cycle (at least twice the corner radius).
4. Press **CYCLE START** to start the probing cycle.

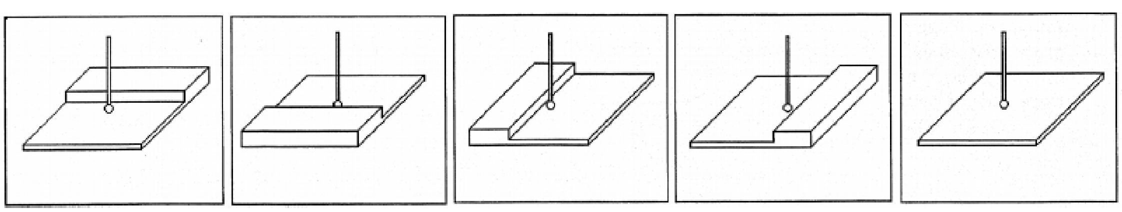
At the end of the probing cycle, the probe will be positioned above the corner at the Z clearance level entered. Press **ESC** to return to the Set Part 0/Position screen or digitize screen.



F6 – Out Corner (Outside Corner) Press to enter the Outside Corner screen. A picture similar to the ones shown above will appear, with instructions and two input fields. Follow these steps:

1. Press **F1 – Orient** to select the orientation of the probe with respect to the Corner. You will see one of the pictures shown above.
2. Press **F2 – Side** to select which side of the corner that the probe will be positioned near. You will see the screen change.
3. Slowly jog the probe to the approximate position as shown in the picture. Be sure to give enough probe clearance.
4. Select the Z clearance field using the arrow keys. Enter the approximate distance that the probe has to travel in order to lift up over the corner.
5. Select the Distance to Corner amount using the up or down arrow keys. Enter the approximate distance from the corner that the probe is located at along the X or Y axis.
6. Press **CYCLE START** to start the probing cycle.

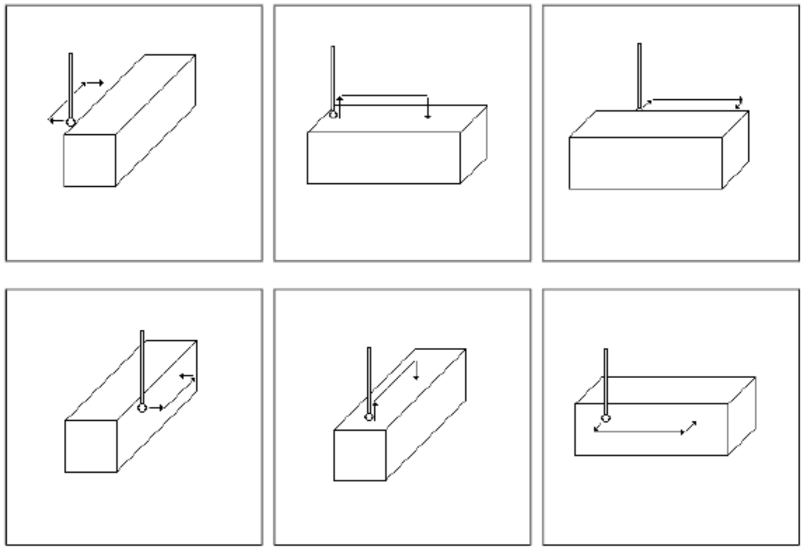
Once the probe has completed its search, it will be positioned above the corner at the Z clearance you specified. Press **ESC** to return to the Set Part 0/Position or digitize screen.



F7 – 1 Axis (Single Axis) Press to enter the Single Axis screen. Follow these steps:

1. Press **F1 – Orient** to select the orientation of the probe. You will see one of the screens shown above.
2. Slowly jog the probe to the approximate position as shown in the picture. Be sure to give the probe enough clearance.
3. Press **CYCLE START** to start the probing cycle.

Once the probing cycle is complete, the probe will move away from the surface by the amount defined in Parameter 13. The probed position will be shown on-screen. Press **ESC** to return to the Set Part 0/Position or digitize screen.



F8 – Angle – Use this feature to measure an angle based on two probed points. Follow these steps:

1. Press **F1 – Orient** to select how the angle is to be measured.
2. Slowly jog the probe to the first position as shown in the picture. Be sure to give the probe enough clearance.
3. Select Auto or Jog between points.
4. Enter the distance to the second point.
5. Enter a clearance amount away from the surface.
6. Press **CYCLE START** to start the probing cycle.

9.4 Probe / TT-1 Parameters

Various probing parameters can be set on the Machine Parameters screen (see Chapter 15). Make sure you enter these parameters before you begin using the probe and/or TT-1. If these parameters are not entered properly, damage to the probe or TT-1 may result.

Probe Type (Parameter 155): This specifies the probe type being used. This needs to be set to 0 if you are using a standard Mechanical probe.

Probe PLC Input Number and Contact State (Parameter 11): A single value, +/-1 through 240. A positive number indicates Closed on contact; a negative number indicates Open on contact.

TT1 PLC input number (Parameter 44): This parameter is the input number that the TT1 is wired into on the PLC. If a shared PLC input is used for both the TT1 and the probe, then the value can be left at zero or set to the same value as Parameter 11.

Probe Tool Number (Parameter 12): A single value, 0 through 200, used to look up the length offset and tip diameter of the probe in the Offset Library.

Recovery Distance* (Parameter 13): The additional distance that the probe moves off of a surface after contact is broken, before attempting to traverse parallel to the surface.

Fast Probing Rate* (Parameter 14): Used for positioning moves and initial surface detection, this parameter is determined by machine response and permitted probe deflection as well as desired tolerance. The default setting is 25 inches/min.

Slow Probing Rate* (Parameter 15): Used for final measuring moves, this parameter is determined by a speed/accuracy tradeoff. The default setting is 3.5 inches/min.

Maximum Probing Distance (Parameter 16): The maximum distance that a probing cycle "searches" for a surface in a given direction if no travel limits have been entered. The default is 10 inches. A larger value should be entered for the hole and slot cycles if you are measuring very large features.

Detector Location Return Point (Parameter 17): A Zero (0) indicates that tool measurement will take place at the current position, and a 1 or 2 indicates the number of the G30 reference point to use which specifies the X, Y location of the TT1 (WCS Configuration).

*Mechanical type probe only. Please see the DSP type probe section below for differences.

9.5 DSP Probe Parameters

When using a DSP-type probe (such as DP-7), a few of the probing parameters have a slightly different behavior than described above. These differences are noted below:

Probe Type (Parameter 155): This specifies the probe type being used. This needs to be set to **1** for a **DP-4D** probe or **2** for a **DP-7** probe.

Repeatability tolerance for probing and radial digitizing (Parameter 151): Used for DP-4 probes. Must be set to 0 for DSP probes.

Grid digitize prediction minimum Z pullback (Parameter 121): Set to 0.035 for DSP use.

Recovery Distance (Parameter 13): Works as described above, but is additionally used as the distance to retract for a retry after a failed window (does not apply to M115/116/125/126 moves)(Default for DSP is 0.05).

Note: If probing small bores, this parameter may need to be reduced accordingly.

Fast Probing Rate (Parameter 14): When using a DSP-type probe, this is the rate at which the probe will move in-between actual probing moves. For instance, when probing a boss this is the rate at which the probe will travel for the following moves:

1. When retracting from the surface after a point has been probed.
2. When retracting to the Z clearance position.
3. Traversing across the diameter of the boss at the Z clearance height.
4. Plunge rate to get to probing depth.

Note: When measuring a TOOL with the TT1, the behavior is identical to that listed above.

Slow Probing Rate (Parameter 15): Used only when measuring a TOOL with the TT-1.

9.6 Additional Probe Parameters for DP-7

For the DP-7 probe (Parameter 155 = 2), the following additional parameters are utilized:

DP-7 Pullback Distance (Parameter 392): This sets the distance the probe moves off from the surface after a probing move.

DP-7 Pullback Feedrate (Parameter 393): This sets the feedrate for the pullback move.

DP-7 Measuring Feedrate (Parameter 394): This sets the feedrate for the slow measuring move.

9.7 Probe Protection

Parameter 153 specifies whether the *Probe Protection* feature is enabled. When probe protection is enabled, most G-code, jog panel, and mpg moves will be stopped when the probe makes an unexpected contact. Unexpected contact is defined to be any probe contact made while jogging, using the mpg, or doing a G-code move that is not a probing move (moves like M115 are probing moves and thus are excluded from this definition).

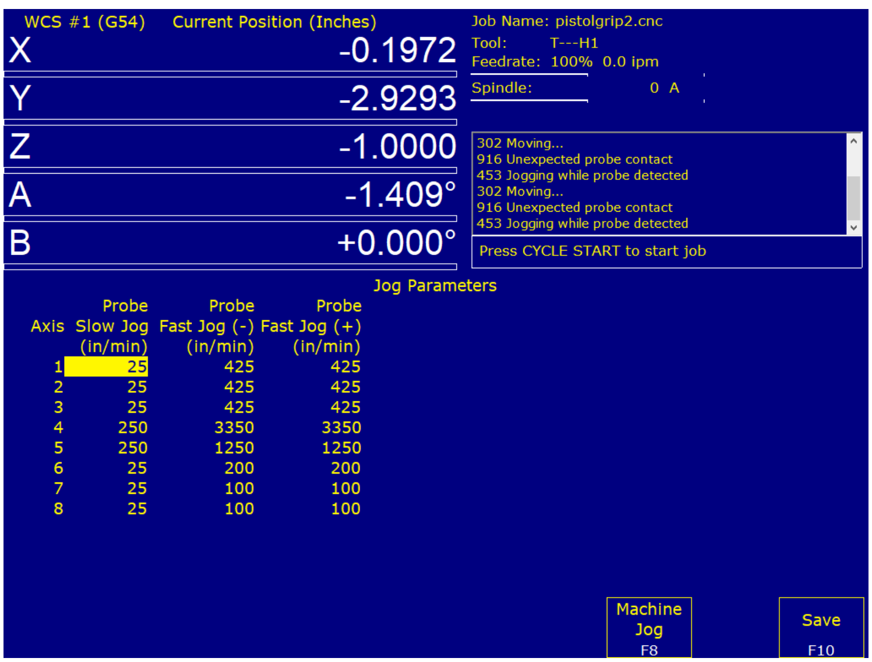
9.7.1 Jog Panel Probe Protection

With proper PLC support, the jog panel includes probe protection. When jogging with a probe attached, motion will come to a stop if the probe is tripped. In this case, to clear the tripped probe, you may jog only in the opposite direction from which you were moving when you tripped the probe. For example, if you were jogging in the `X+` direction when the probe tripped, you may only jog in the `X-` direction to clear the probe.

Note: Proper PLC support is required for this feature to work!

9.7.2 Jog Parameters Menu for Probe Protection

In the jog parameters menu, you may press `F8` to go to the Probe Jog Parameters Menu. In this menu, you may set the slow and fast jog speeds that the machine will travel in *when a probe is plugged in.* Note that for fast jog, you can set the minus and plus directional jog speeds independently (for example, if you want `Z-` to move especially slowly when a probe is plugged in). See the picture below for a still image from the probe jog menu.



9.7.3 MPG Probe Protection

When probe protection is enabled, tripping the probe will cause MPG motion to stop for an MPG with the increment multiplier set to `10x` or `100x`. To clear the probe, you must set the increment multiplier to `1x`, and then you can move the MPG in any direction on any axis.

9.7.4 Probe Protection While a Job is Running

If probe protection is enabled and the probe trips unexpectedly while a job is running, motion will stop and you may use the jog panel to clear the probe trip. These moves will also be stopped if the trip happens during an MDI move.

10 Intercon Software

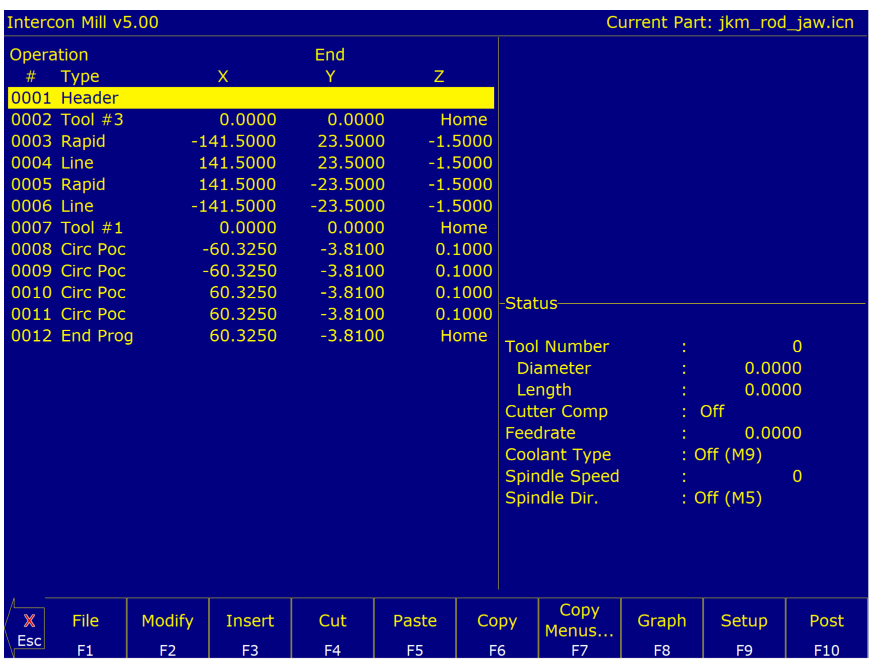
10.1 Introduction

Intercon (Interactive Conversational) software allows you to quickly create a part program right at the control without having to be a G-code expert. Intercon will prompt you to enter values from your print that describe the geometry of the part. Intercon will display graphics of the part as you are creating it, helping you to quickly proceed through part programming. Intercon can then generate a G-code program from the geometric information you have entered.

You can purchase an offline version of the Intercon software for use on your desktop PC. You will also need to purchase a hardware key, which will allow the offline version to run. Simply plug the key into the computer, install the required drivers, and run the program.

10.2 Intercon Main Screen

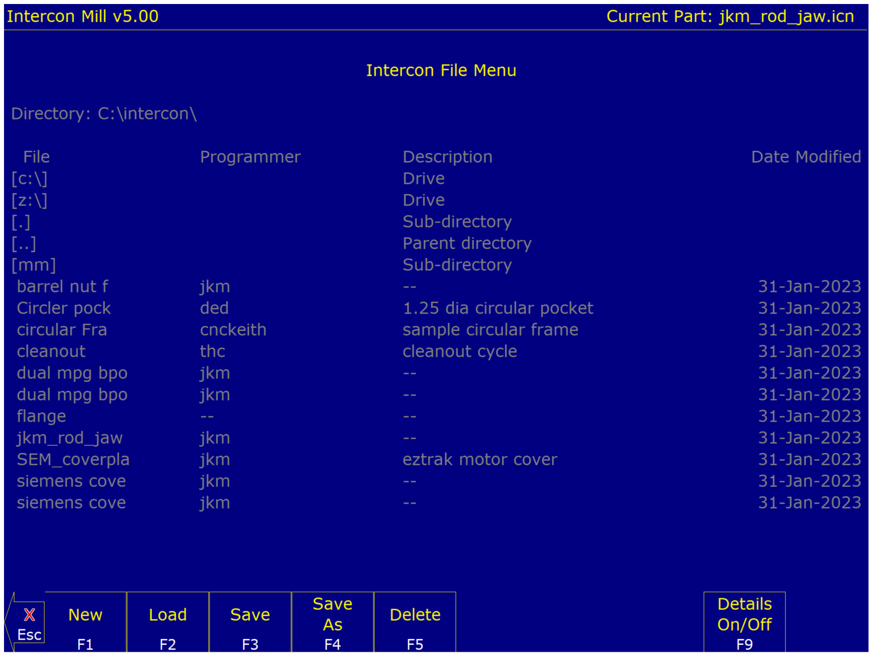
When you access Intercon through the **F5 – CAM** option in the CNC software Main screen, the part program will be displayed if the current job loaded in CNC software has an associated Intercon program. If the job file in the CNC software does not have an associated Intercon program, the **F1 – File** menu will be displayed to load or create a file.



When the part program is displayed, different operations can be navigated to and highlighted for use with additional actions by using the arrow, **HOME**, **END**, **PAGE UP**, and **PAGE DOWN** keys.

10.3 File Menu (Intercon Main Screen → F1 – File)

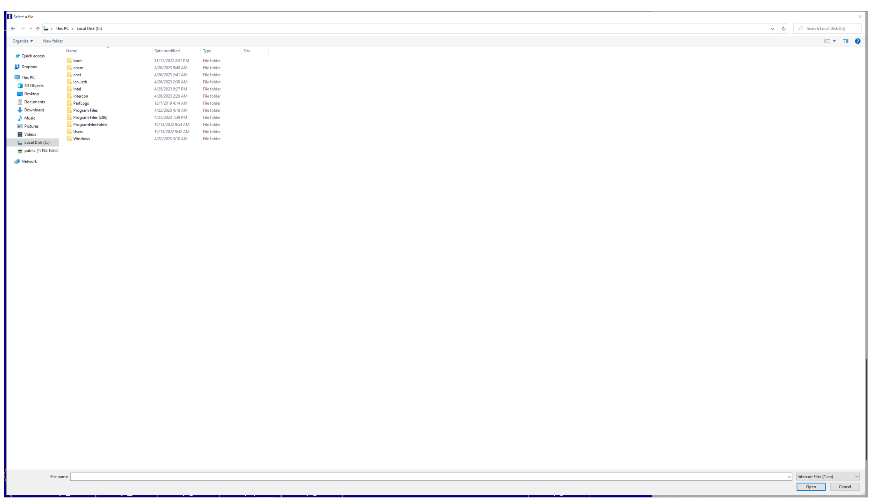
F1 – File: Choosing **F1 – File** will display the screen below. Intercon stores part programs with an extension of .icn. For example, if you choose to name your new part program flange, Intercon will save the program as flange.icn. ICN files are only readable by Intercon.



F1 – File → **F1 – New**: To program a new part, choose **F1 – New**. A prompt will be displayed where the name of the new program can be typed, followed by the **F10 – Accept** or **ENTER** key to accept the new name can be entered. You can enter who the programmer is, a program description, the units of measure, and the date.

10.4 Load Menu (Intercon Main Screen → F1 – File → F2 – Load)

F1 – File → **F2 – Load**: When you press **F2 – Load**, the screen below is displayed (Shown with details “ON”).



To navigate the files in the load menu, use the arrow keys to move the cursor around and highlight the file to be loaded. The **HOME**, **END**, **PAGE UP** and **PAGE DOWN** keys can be used to navigate the list of files. Names that are bracketed, for example [..], are the names of directories in the current directory that is displayed at the top of the screen.

It is also possible to start typing the name of the program to be loaded. When typing has started, the characters appear in the “File to load:” prompt above the function keys. Different drives and directories can be accessed by typing in the path at the “File to load:” prompt, or by pressing **F10** or **ENTER** on a bracketed directory name. When loading a new file, a prompt will be displayed asking whether to save the existing file if there was one.

Additional Load Menu options are detailed below:

F1 – G-code/ICN Allows user to toggle the view between the Intercon files present in either c:\intercon or c:\cncm\ncfiles.

F2 – USB/LAN Provides options for loading Intercon files from USB devices and LAN drives.

F3 – Details On/Off The **F3 – Details** On/Off option changes the format of the display such that each file or directory is on a separate line and there are columns displayed for Programmer, Description, and Date Modified, i.e., the information that is contained in the program header operation.

F4 – Show Recent Use the **F4 – Show Recent** option to show the 15 most recently-loaded Intercon and G-code files. It is important to remember that even though G-code files are displayed on this screen, ONLY Intercon files should be loaded from this screen. WARNING!!! Attempting to load a G-code file from the "Show Recent" screen will cause an error, which will discard the current Intercon program. All unsaved changes will be lost. If you should accidently load a G-code file, press escape to return to the main Intercon menu.

F5 – Date/Alpha Use **F5 – Date/Alpha** to view files either alphabetically or by date modified. By default, programs are listed in ascending alphabetical order.

F6 – Edit Opens the selected file in Intercon for editing.

F7 – Help On/Off Displays on-screen help for the load menus.

F8 – Graph Graphs the selected file.

F9 – Advanced Displays the file menu in a comprehensive "all in one" format similar to Windows Explorer

10.5 File Menu Continued

F1 – File → **F3 – Save**: Press **F3 – Save** to save the current part program. It will be saved under the specified name.

F1 – File → **F4 – Save As** Press **F4 – Save As** to save the current program with a different name. Type the new name into the **"Save part as:"** prompt that appears above the function keys. If the new name already exists, a prompt will be displayed as a warning, and will give the option to overwrite the existing file or return to enter a different name.

F1 – File → **F5 – Delete**: Press **F5 – Delete** to delete a file. After **F5 – Delete** is pressed, a screen will appear as in the **F2 – Load** option, which you can use to navigate to the files. A yes/no prompt will appear after for final confirmation accepting a file for deletion.

F1 – File → **F9 – Details on/off** The **F9 – Details On/Off** changes the format of the display such that each file or directory is on a separate line and there are columns displayed for Programmer, Description, and Date Modified, i.e., the information that is contained in the program header operation.

10.6 Intercon Main Screen Continued

F2 – Modify: Choosing **F2 – Modify** (or press the **ENTER** key) from the Intercon main menu will allow the currently highlighted operation to be modified. When an operation is modified, the fields for that operation are displayed on the right hand side. When modifying an operation, the PAGE UP and PAGE DOWN keys can be used to move up and down through the Intercon operations listed on the left hand side of the screen. See the "Insert Operation" section later in this chapter for a description of each operation type.

F3 – Insert Choosing **F3 – Insert** will insert a new operation before the operation that is currently highlighted, unless the highlighted operation is the first operation in which case the inserted operation will be inserted as the second operation. For more information on the operations see Insert Operations later in this chapter.

F4 – Cut Choosing **F4 – Cut** will cut (remove) the highlighted operation from the program. The operation that is cut is placed onto the clipboard stack.

F5 – Paste Choosing **F5 – Paste** will paste the last operation that was cut or copied into the clipboard stack into the current program line that is before the highlighted operation. The number of operations that are currently in the clipboard stack are indicated by the number in the Paste key. As long as you stay in Intercon, the clipboard stack will remain intact. You may cut and copy operations from one program and paste them into a different program.

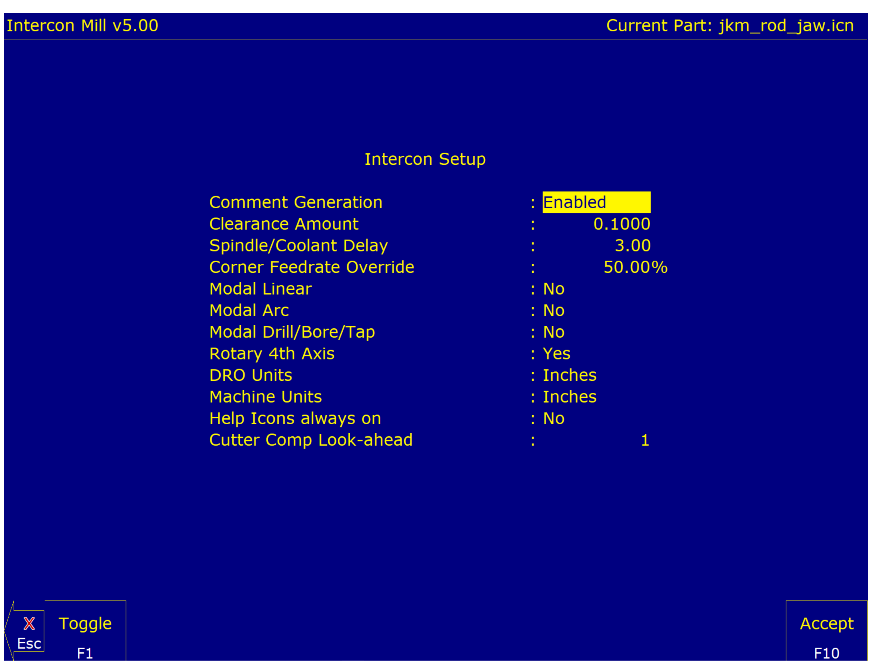
F6 – Copy Choosing **F6 – Copy** will copy the highlighted operation into the clipboard stack and advance the cursor to the next operation.

F7 – Copy Menus. . . Choosing **F7 – Copy Menus. . .** will display these additional options:

> ***F1 – Copy Menu*** Allows a range of operations to be copied. Specify the Start Block, End Block, and Destination in the prompts that appear in the Copy Menu. The range of operations is copied into a location that precedes the destination block.
>
> ***F2 – Move Menu*** Allows a range of operations to be moved. Specify the Start Block, End Block, and Destination in the prompts that appear in the Move Menu. The range of operations is moved into a location that precedes the destination block.
>
> ***F9 – Clear Clipbrd*** Removes all operations in the clipboard stack.

F8 – Graph Choosing **F8 – Graph** will graph the current program. The graph is the same as what would be produced if the current program were translated into G-codes and graphed from the CNC software. Canned drilling cycles are shown in gray. Rapid traverse movements are shown in red. Feedrate movements are shown in yellow and cutter compensated moves are in gray.

F9 – Setup Choosing **F9 – Setup** will display the Setup menu where certain options can be set. The Setup menu appears as below.



Use the up and down arrow keys to move. Clearance Amount, Spindle/Coolant Delay, and Corner Feedrate Override require a value to be typed in. The other fields have fixed values that may be toggled by using the **F1 – Toggle** key.

Comment Generation: When this field is set to Enabled, Intercon will put a comment describing the operation type before each block. Disabling Comment Generation will make the CNC files generated by Intercon smaller.

Clearance Amount: This is the distance that Intercon raises the Z-axis above the programmed surface height in pockets, facing and frame mills when traveling across the work piece.

Spindle/Coolant Delay: Set this delay to the time in seconds you want Intercon to wait for the spindle to get up to speed and the coolant to begin flowing.

Corner Feedrate Override: This is the percent feedrate that will be used in the corners of rectangular pockets and inside frame mills. The default value is 50%.

Modal Operations: These options specify whether to automatically insert the same operation after the first has been accepted. Once modal insert mode has begun, press ESC to insert a different operation.

Rotary 4th Axis: This option specifies whether or not 4th axis movement fields appear in Linear and Rapid moves and whether or not the Intercon program will post 4th axis movement. This option affects the value in the 4th axis configuration only (Parameter 94). Note that although Intercon is restricted to working with 1 rotary axis at a time, it can be directed to utilize the 5th axis as its rotary axis instead of the 4th. To make Intercon to utilize the 5th axis as its rotary axis instead of the 4th axis, the 4th axis configuration (Parameter 94) must have its rotary property turned off (bit 0 = 0) and the 5th axis configuration (Parameter 166) must have its rotary property turned on (bit 0 = 1). See Chapter 15 for details about Parameters 94 and 166.

DRO Units/Machine Units: Sets the units of measure, either Inch or Millimeter. It is recommended that these two fields be toggled together to the same units. These fields affect their corresponding fields in the Control Configuration. The posted G-code will contain a G20 for Inches mode and a G21 for Metric mode.

Note: Changing the units from within individual Intercon job headers does NOT toggle the units between Inches/Millimeters in the generated G-code. This is a developmental feature and should be kept the same as the units selected in DRO Units/Machine Units.

Help Icons always on: Toggle between yes or no. Selecting "yes" means that help information will always be displayed when editing operations. "No" means that you will have to press a key to get help. Whether set to "yes" or "no", the help screens can always be toggled on or off by pressing the **F5 – Help** key when editing an operation.

Cutter Comp Look-ahead: This sets the number of segments that can be parsed ahead when Cutter Comp is turned on.

F10 – Post: Choosing **F10 – Post** will post the current program. Posting is the process of converting the operations into G-codes. When the posting process is completed, Intercon is exited. The Intercon program is also saved as part of the posting.

10.7 Insert Operation (Intercon Main Screen → F3 – Insert)

When you press **F3 – Insert**, or when you choose New Part from the Main Screen, you will see the Insert screen:

The new operation will be inserted right before the currently highlighted one. The operation types that you can insert are listed across the bottom of the screen.

10.7.1 F1 – Rapid Traverse

Press **F1 – Rapid** from the Insert Operation screen to insert a Rapid Traverse. You may see the following screen:

<u>End:</u> When you first access the rapid traverse screen, the cursor will be highlighting the first field, End X. This is the X coordinate of where the cutter will be after the rapid traverse has been completed. Similarly, Y and Z represent the coordinates of the cutter after the rapid traverse is completed. The angle and length fields will be computed if you choose to enter the end point of the move.

Angle: The destination may also be specified in terms of a counterclockwise angle from the three o'clock position. When combined with a length for the current move, the corresponding X and Y coordinates for the destination will be calculated and placed in the correct fields. The Z destination will remain unchanged, however.

Length: The length of the rapid traversal. When combined with the angle of the current move, the corresponding X and Y coordinates for the destination will be calculated and placed in the correct fields. The Z destination will remain unchanged, however.

The **F1 – Abs/Inc** key toggles between incremental and absolute positioning modes in any of the fields where a positional dimension is needed. For example: X, Y, or Z-axis dimensions can all be in incremental or absolute coordinates, or a mixture of both. The length and angle fields cannot be incremental. These fields are absolute values.

The **F2 – Z Home** key may be used on the Z destination field to tie the ending Z coordinate to the Z home position. This means that no matter what you're Z home value is at the time that you run your program, the final Z position will be the Z home position.

When you are finished entering all of the dimensions for the rapid move, press **F10 – Accept** to accept the operation and return to the Insert Operation screen.

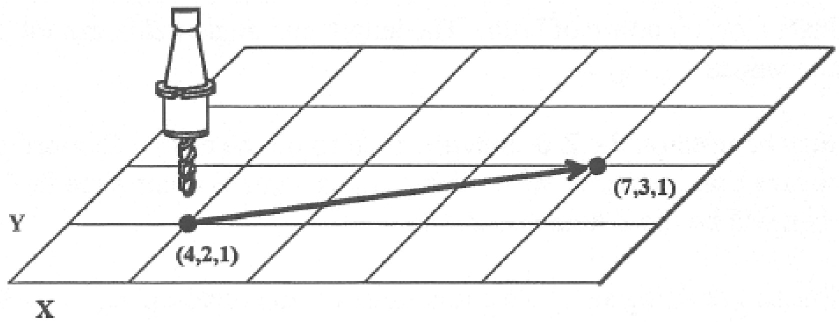
Note: When making rapid moves, if a Z destination higher than the current cutter position is specified, the cutter will first be raised to the destination Z position, and then move linearly in X and Y to arrive at the destination. If a Z destination lower than the current cutter position is specified, the cutter will move linearly in X and Y first and then plunge Z to the destination Z position.

Note: The Rapid traverse operation can have rotary fields, if you have a rotary fourth axis. The rotary field descriptions are the same as that of the Linear Mill operation.

10.7.2 Linear Mill

If you press F2 – Linear from the Insert Operation screen, the following screen appears:

The numbers in the different fields on the screen correspond to the following Linear Mill example shown here graphically:



End: When you first access the linear mill screen, the cursor will be highlighting the first field, End X. This is the X coordinate of where the cutter will be after the linear move has been completed. Similarly, Y and Z represent the coordinates of the cutter after the linear move is completed. The angle and length fields will be computed if you choose to enter the end point of the move.

Angle: The destination may also be specified in terms of a counterclockwise angle from the three o'clock position. When combined with a length for the current move, the corresponding X and Y coordinates for the destination will be calculated and placed in the correct fields. The Z destination will remain unchanged, however.

Length: The length of the linear mill. When combined with the angle of the current move, the corresponding X and Y coordinates for the destination will be calculated and placed in the correct fields. The Z destination will remain unchanged, however.

Connect Radius: If you are performing two linear mill operations and you wish to have a rounded 'corner' between them instead of a sharp peak, you may enter the radius of the 'corner' and Intercon will insert an arc between the linear mill operations. This connect radius also works for blending a line into an arc operation.

Feedrate: Speed at which the cutter moves. The feedrate can be toggled to modal, fixed, or slave. This is indicated by the symbol beside the feedrate field. If the feedrate is modal then it will have the "M" symbol or if it is fixed it will have the "F" symbol shown below. The slave feedrate has no symbol and is set to the last modal feedrate set in the program, when the modal feedrate changes all the following slave feedrates change until the next modal feedrate is encountered.

$\boxed{M}$ $\boxed{F}$

If you have a fourth axis installed and it is rotary, additional fields are shown for Linear Mill operations after the feedrate field.

Degrees: The number of degrees you want to move the rotary axis. This value can be positive or negative and the movement of the rotary axis will depend on the orientation of the axis.

Minutes: The number of minutes you want to move the rotary axis. Values for this field are between 0 and 59.

Seconds: The number of seconds you want to move the rotary axis. Values for this field are between 0 and 59.

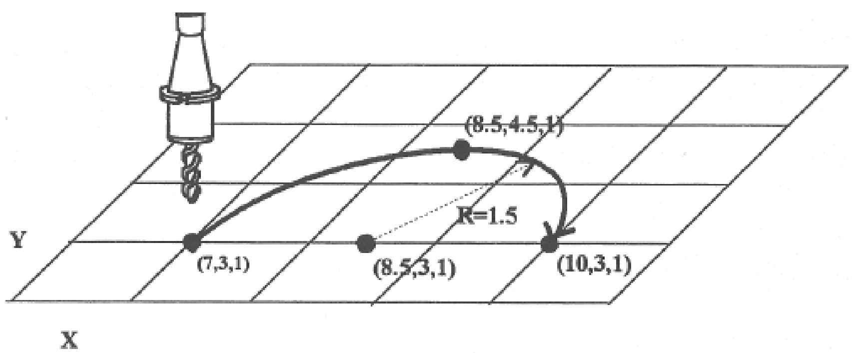
Decimal degrees: This is another method of entering the number of degrees. If you choose to enter the movement of the rotary axis with the fields listed above, the value of this field will be calculated automatically. If you choose to enter the number of degrees with this field or make changes to it, then the degrees, minutes and seconds will be calculated or changed automatically. Values for this field can be positive or negative.

Rotary movement defaults to zero degrees, incremental. To enter an absolute (rather than incremental) rotary position, you must press **F1 – Abs/Inc** to toggle to absolute.

10.7.3 Arc Mill

If you press **F3 – Arc** for Arc Mill from the Insert Operation screen, the following screen appears:

The numbers in the different fields on the screen correspond to the following Arc Mill example shown here graphically:



Operation Type: There are four ways to specify your ARC: using an endpoint and a radius (EP&R), using a center point and an angle (CP&A), using a center point and an end point (CP&EP), or using a mid-point and an end point (3- Point). The Three Point arc is designed to be used in conjunction with Teach Mode. When specifying a particular kind of arc, you will not be able to modify certain fields. For example, if you are specifying an endpoint and a radius, you will not be able to modify the mid point, center point and angle fields. This is because Intercon calculates the correct values for these fields.

Mid: The X, Y, and Z coordinates of a point on the arc path somewhere between the start point and end point of the arc. You will be able to modify this field only when specifying a Three Point arc. Also, the coordinate that does not lie in the plane of the arc cannot be edited; it is automatically calculated.

End: The X, Y and Z coordinates of where the cutter will be once the arc move is complete. You will not be able to edit this field if you are specifying a center point and angle (CP&A) arc.

Center: This is the X, Y and Z position of the center of the arc. You will not be able to edit this field if you are specifying an end point and radius (EP&R) arc or a Three Point arc. Also, the coordinate that does not lie in the plane of the arc cannot be edited; it is automatically carried forward from the last operation.

Angle: Number of degrees through which the cutter will travel. This value must lie between 0 and 360 degrees. You will be able to edit this field only if you are specifying a center point and angle (CP&A) arc.

Radius: Distance from the center of the arc to its edge. This value must be greater than 0. You will only be able to edit this value if you are specifying an end point and radius (EP&R) arc.

Plane: This determines whether the arc is to be milled in the XY-, ZX- or YZ-plane. If any of the Z coordinate values are tied to the Z home position, only XY-plane arcs may be selected.

Direction: Determines whether the arc moves clockwise (CW) or counterclockwise (CCW). Note that the direction of XZ arcs is judged looking Y+ (i.e. from the front of the machine). This is natural, but it is opposite of the way arcs are specified in G codes. Intercon automatically makes this translation when it generates CNC codes.

Connect Radius: This field works like the Linear Mill connect radius. It allows for the blending of an arc into the next line or arc operation.

Feedrate: Speed at which the cutter moves. The feedrate can be toggled to modal, fixed or slave, this is indicated by the symbol beside the feedrate field. If the feedrate is modal then it will have the “M” symbol or if it is fixed it will have the “F” symbol shown below. The slave feedrate has no symbol and is set to the last modal feedrate set in the program, when the modal feedrate changes all the following slave feedrates change until the next modal feedrate is encountered.

M F

Angle $\leq 180°$: For end point and radius (EP& R) arcs, this field determines whether the arc is to be less than (YES) or greater than (NO) 180 degrees.

10.7.4 F4 – Tool Functions

When you select the tool functions by pressing F4 – Tool the following screen appears:

The following parameters for this tool change are as follows:

Tool Number: Number of the tool (between 1 and 200) to use. Entering this value pulls the current settings for this tool from the CNC software tool library. You may then edit the length offset, diameter offset and diameter values if you wish to redefine your tool. The length value is not editable.

Description: Description of the tool selected above, from the tool library.

Position: X and Y coordinates for the place at which the tool change will occur. This should be a place at which the current tool can be removed from the quill and the new tool can be inserted.

Tool H Offset: Index in the offset library (between 0 and 200) of the actual tool height offset.

Tool Height: Tool height associated with the H offset selected above. This field is not editable.

Tool D Offset: Index in the offset library (between 0 and 200) of the actual tool diameter.

Tool Diameter: Tool diameter associated with the D offset selected above.

Spindle Speed: Speed at which the spindle will rotate when the spindle is started after the tool change.

Spindle Direction: Direction in which the spindle will turn after the tool change. If this is set to CW or CCW, the spindle will be started automatically after the tool change. **Press F3 – Toggle** or **SPACE** to toggle between CW, CCW, and Off.

Coolant Type: Type of coolant to activate after the tool change. If this is set to Flood or Mist, the selected coolant system will be started automatically after the tool change. Press **F3 – Toggle** or **SPACE** to toggle between Flood, Mist, and Off.

Actual Tool Change: Determines whether an M6 code is generated (answer Yes) during the tool change. If you do not want to remove the current tool, but instead want to alter its diameter or length offsets (e.g. for doing a finish pass while using cutter compensation, you may want to use a diameter offset which is slightly larger than the actual tool for the first passes, then use the actual tool diameter for the finish pass), answer No to this question. Spindle and coolant will not be automatically turned off if you answer No here.

Press **F10 – Accept**, **Page Up**, or **Page Down** when you are finished to accept these values and make changes to the tool library. If you have changed any field other than the Tool Number of the Actual Tool Change field or position, you will make changes to the CNC software Tool Library. At the end of the program, Intercon always turns off the spindle and coolant and returns the Z-axis to the home position. These codes do not need to be entered at the end of your program.

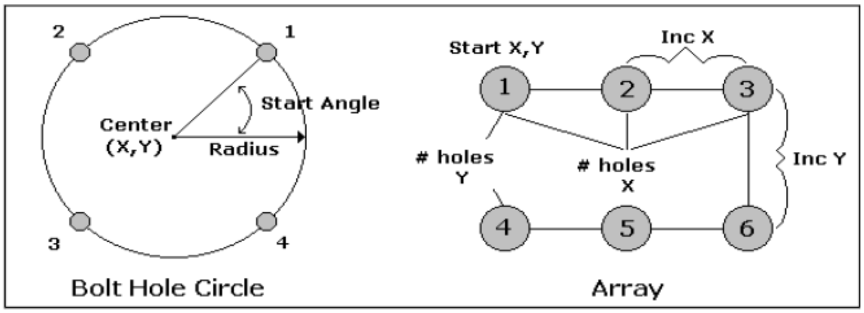
10.7.5 F5 – Canned Cycles

When you choose the Canned Cycle operation by pressing F5 – Cycles, the following screen appears:

Canned Cycle Introduction #1: Using Pattern and Repeat (Drilling, Boring, Tapping)

Selecting **F1 – Drill** will give you four choices; **F1 – Drill**, **F2 – Drill BHC**, **F3 – Drill Array**, or **F4 – Drill Repeat**. **F2 – Bore** and **F3 – Tap** will have the same menu selections as drill except they will display Bore or Tap cycles

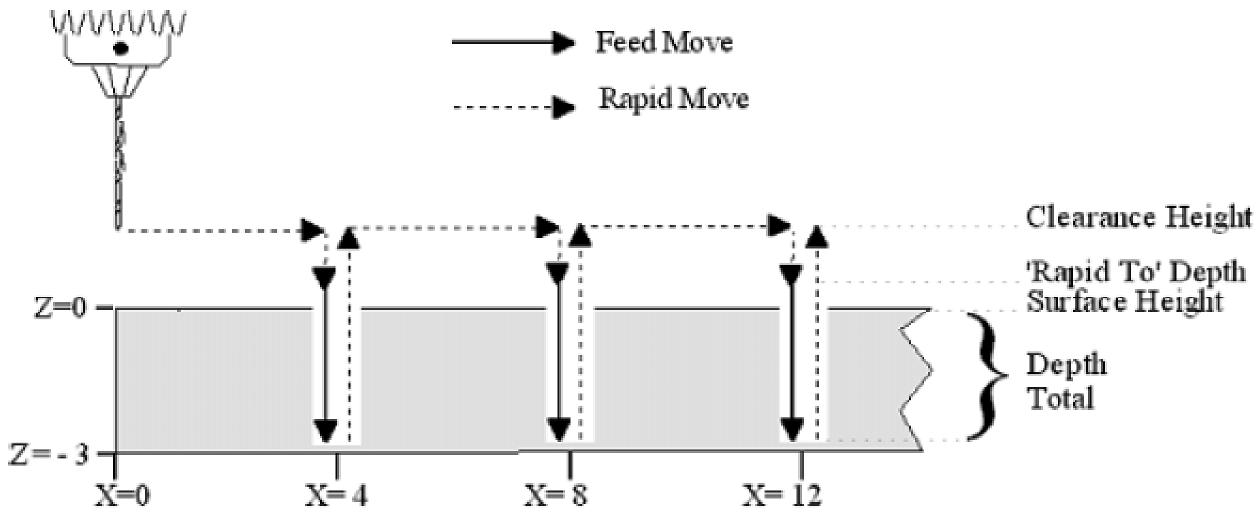
All canned cycle operations using the Drill BHC (Bolt Hole Circle) or Drill Array are identical to their equivalent using the **F1 – Drill** single hole selection. The use of the Drill BHC or Drill Array, however, offers the option to drill more than one hole in a pattern dictated by the new fields in the menu. **F4 – Drill** Repeat allows the user to repeat a set of holes with a different type drilling, boring or tapping operation with out re-entering the X, Y coordinates. The Bolt Hole Circle and array patterns are explained graphically in the following figure:



Canned Cycle Introduction #2: Linear Repetition of Operations (Drilling, Boring, Tapping)

If you want to perform one operation several times in a linear pattern, simply define Position X, Y or both as incremental values. To do this, use the F1 – Abs/Inc Key. This key will toggle the Position value mode between incremental and absolute. If you define X and/or Y as incremental values, a new field will appear asking for the number of holes:

The numbers in the fields on the screen correspond to the following example, shown here graphically:



Drilling (F1 in the Canned Cycle Menu: Option #1)

If you press **F1 – Drill** from the Canned Cycle Menu, you will gain access to three types of drilling operations: Drilling, Chip Breaking, and Deep Hole drilling. The current drilling operation in use is reflected in the field “Cycle Type” and pressing **F3 – Toggle** or **SPACE** toggles between all three. In this section we will examine the first option: Drilling.

Intercon Mill v5.00 — Current Part: jkm_rod_jaw.icn

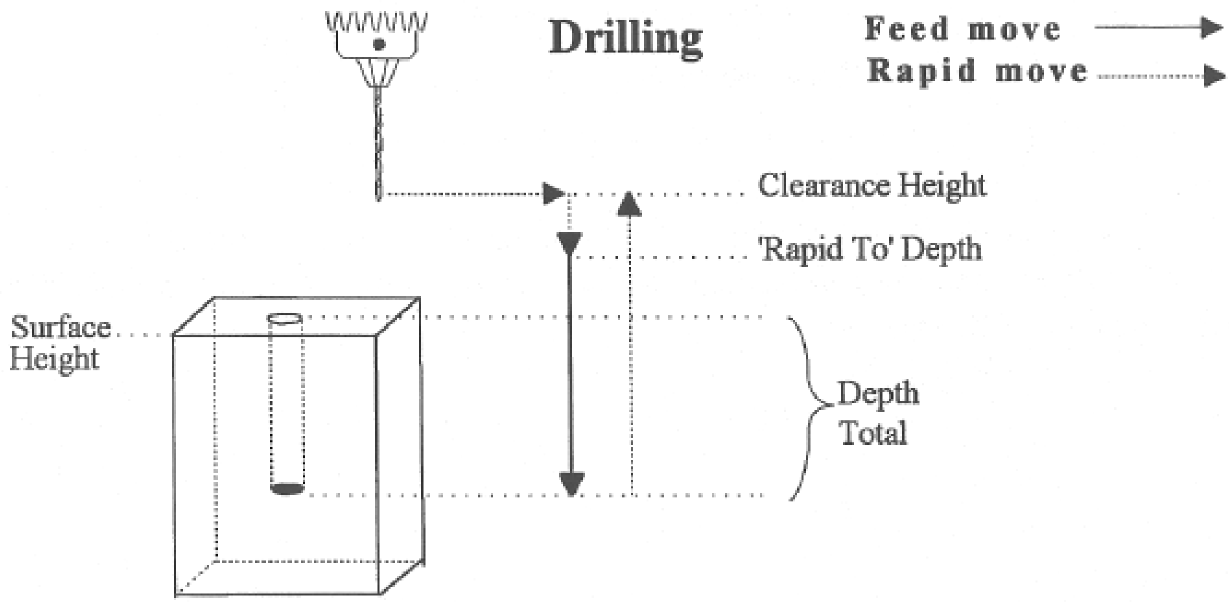
Operation #	Type	End X	End Y	End Z
0001	Header			
0002	Drill			
0003	Tool #3	0.0000	0.0000	Home
0004	Rapid	-141.5000	23.5000	-1.5000
0005	Line	141.5000	23.5000	-1.5000
0006	Rapid	141.5000	-23.5000	-1.5000
0007	Line	-141.5000	-23.5000	-1.5000
0008	Tool #1	0.0000	0.0000	Home
0009	Circ Poc	-60.3250	-3.8100	0.1000
0010	Circ Poc	-60.3250	-3.8100	0.1000
0011	Circ Poc	60.3250	-3.8100	0.1000
0012	Circ Poc	60.3250	-3.8100	0.1000
0013	End Prog	60.3250	-3.8100	Home

N0002 Drill

Cycle Type	:	Drilling	
Position	X:	60.3250	
	Y:	-3.8100	
Surface Height	:	0.0000	
Clearance Height	:	0.1000	INC
"Rapid to" Depth	:	0.1000	INC
Depth:	Total:	0.0000	INC
Plunge Rate	:	0.0000	
Dwell Time	:	0.00	

Esc | Toggle F3 | Help F5 | Math Help F6 | Graph F8 | Teach Mode F9 | Accept F10

The numbers in the fields on the screen correspond to the following example, shown here graphically:



Where:

Cycle Type: Selects one of three drilling operations: Drilling, Chip Breaking, or Deep Hole drilling. Press **F3 – Toggle** or **SPACE** to toggle between the three choices.

Position: Specifies the X and Y coordinates where the drilling will take place. If either the X or Y coordinate is an incremental value, you will have the option to drill multiple holes in a linear pattern (See Canned Cycle Introduction #2).

Surface Height: Absolute Z-axis position from where each incremental depth is measured.

Clearance Height: This parameter specifies the Z-axis height used when performing rapid moves to the position of each hole being drilled.

'Rapid To' Depth: The depth to which the cutter rapid moves before beginning to drill the hole at the specified Plunge Rate. This is below the Clearance Height but above the Surface Height.

Depth: Total: Depth of hole (incremental) as measured from Surface Height.

Plunge Rate: Z-axis speed of descent during drilling. The plunge rate can be toggled to modal, fixed or slave, this is indicated by the symbol beside the plunge rate field. If the plunge rate is modal then it will have the "M" symbol or if it is fixed it will have the "F" symbol. The slave plunge rate has no symbol and is set to the last modal plunge rate set in the program, when the modal plunge rate changes all the following slave plunge rates change until the next modal plunge rate is encountered.

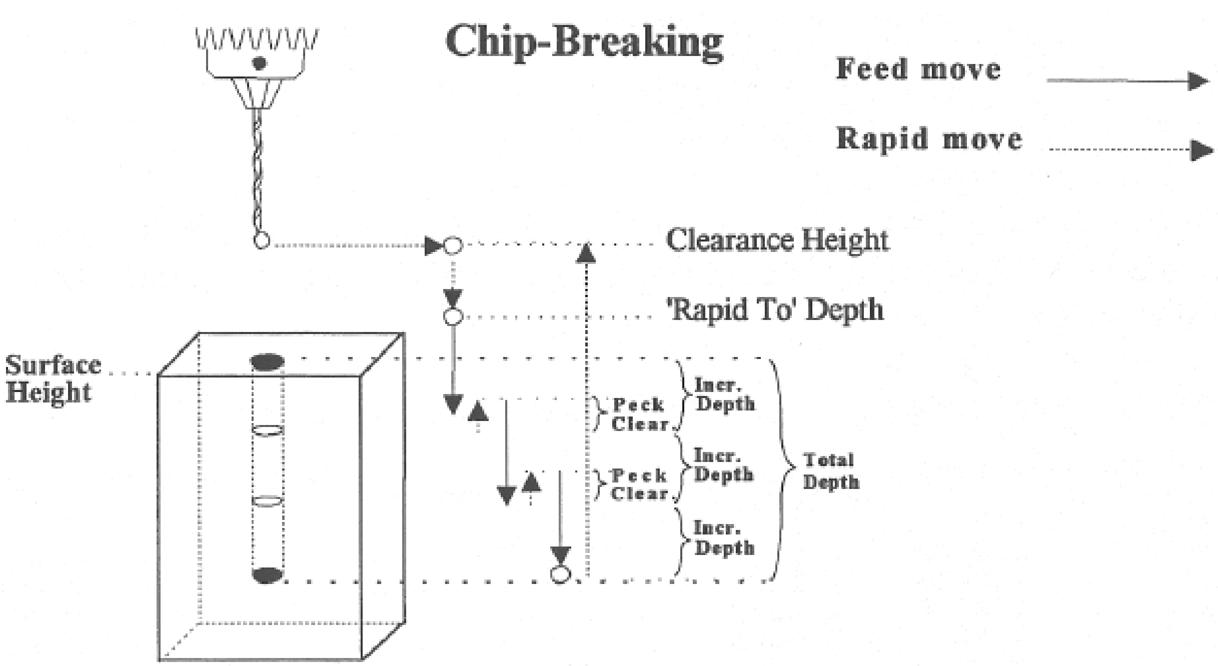
Dwell Time: Delay at bottom of hole before starting ascent.

Drilling provides a rapid to the hole position at the Clearance Height, followed by a rapid Z down to the 'Rapid To' Depth. Next is a feedrate down to the specified depth. If a Spot facing cycle is desired, enter a value in the dwell time field and the cutter will wait the desired amount of time before performing a rapid move up to the Clearance Height.

Chip Breaking (F1 in the Canned Cycle Menu: Option #2)

If you press **F1 – Drill** from the Canned Cycle Menu you will gain access to three types of drilling operations: Drilling, Chip Breaking, and Deep Hole drilling. The current drilling operation in use is reflected in the field Cycle Type, and pressing **F3 – Toggle** or **SPACE** toggles between all three. In this section we will examine the second option: Chip Breaking.

The numbers in the fields on the screen correspond to the following example, shown here graphically:



Where:

Cycle Type: Selects one of three drilling operations: Drilling, Chip Breaking, or Deep Hole drilling. Press **F3 – Toggle** or **SPACE** to toggle between the three choices.

Position: Specifies the X and Y coordinates where the drilling will take place. If either the X or Y coordinate is an incremental value, you will have the option to drill multiple holes in a linear pattern (See Canned Cycle Introduction #2).

Surface Height: Absolute Z-axis position from where each incremental depth is measured.

Clearance Height: This parameter specifies the Z-axis height used when performing rapid moves to the position of each hole being drilled.

'Rapid To' Depth: The depth to which the cutter rapid moves before beginning to drill the hole at the specified Plunge Rate. This is below the Clearance Height but above the Surface Height.

Depth: Total: Depth of hole (incremental) as measured from Surface Height.

Depth: Increment: Depth of each individual peck.

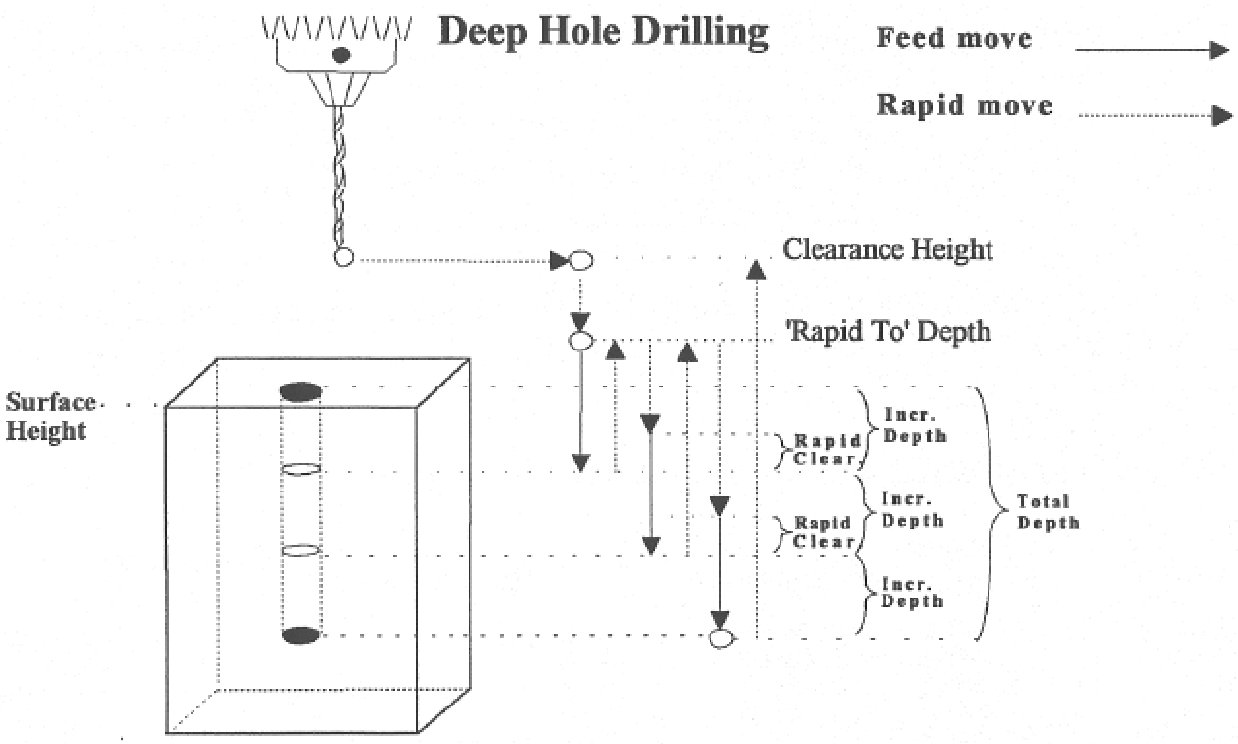
Peck Clearance: Distance the tool retracts before drilling the next peck.

Plunge Rate: Z-axis speed of descent during drilling. The plunge rate can be toggled to modal, fixed or slave, this is indicated by the symbol beside the plunge rate field. If the plunge rate is modal then it will have the "M" symbol or if it is fixed it will have the "F" symbol. The slave plunge rate has no symbol and is set to the last modal plunge rate set in the program, when the modal plunge rate changes all the following slave plunge rates change until the next modal plunge rate is encountered.

Deep Hole Drilling (F1 in the Canned Cycle Menu: Option #3)

If you press **F1 – Drill** from the Canned Cycle Menu you will gain access to three types of drilling operations: Drilling, Chip Breaking, and Deep Hole drilling. The current drilling operation in use is reflected in the field Cycle Type, and pressing **F3 – Toggle** or **SPACE** toggles between all three. In this section we will examine the third option: Deep Hole drilling.

The numbers in the fields on the screen correspond to the following example, shown here graphically:



Where:

Cycle Type: Selects one of three drilling operations: Drilling, Chip Breaking, or Deep Hole drilling. Press **F3 – Toggle** or **SPACE** to toggle between the three choices.

Position: Specifies the X and Y coordinates where the drilling will take place. If either the X or Y coordinate is an incremental value, you will have the option to drill multiple holes in a linear pattern (See Canned Cycle Introduction #2).

Surface Height: Absolute Z-axis position from where each incremental depth is measured.

Clearance Height: This parameter specifies the Z-axis height used when performing rapid moves to the position of each hole being drilled.

'Rapid To' Depth: The depth to which the cutter rapid moves before beginning to drill the hole at the specified Plunge Rate. This is below the Clearance Height but above the Surface Height.

Depth: Total: Depth of hole (incremental) as measured from Surface Height.

Depth: Increment: Depth of each individual step of the drilling.

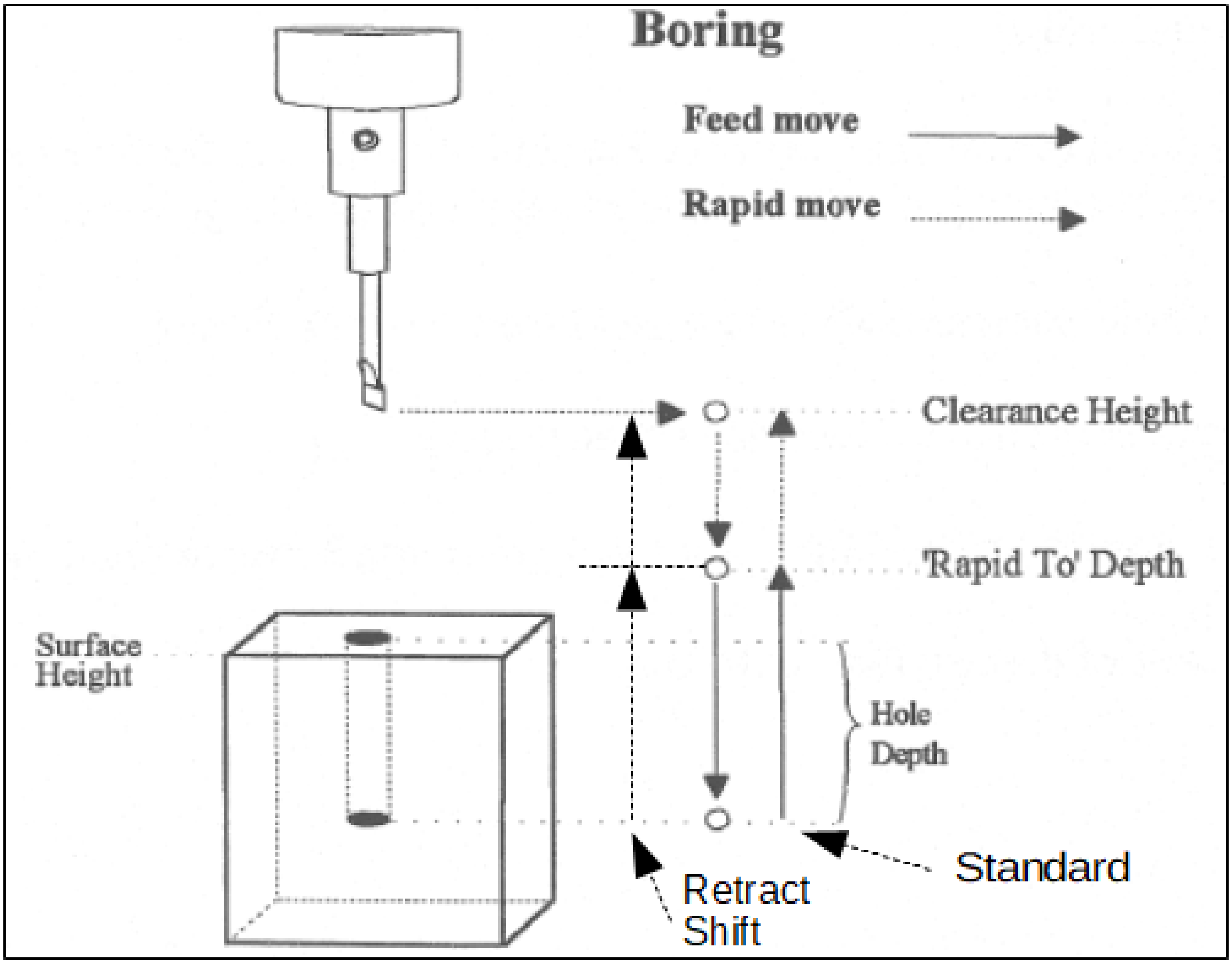
Rapid Clearance: Distance from the last incremental depth drilled that the tool rapid moves before starting the next plunge.

Plunge Rate: Z-axis speed of descent during drilling. The plunge rate can be toggled to modal, fixed or slave, this is indicated by the symbol beside the plunge rate field. If the plunge rate is modal then it will have the "M" symbol or if it is fixed it will have the "F" symbol. The slave plunge rate has no symbol and is set to the last modal plunge rate set in the program, when the modal plunge rate changes all the following slave plunge rates change until the next modal plunge rate is encountered.

Boring (F2 in the Canned Cycle Menu)

If you press **F2 – Bore** from the Canned Cycle Menu you will gain access to the boring operation:

The numbers in the fields on the screen correspond to the following example, shown here graphically:



Where:

Position: Specifies the X and Y coordinates where the boring will take place. If either the X or Y coordinate is an incremental value, you will have the option to bore multiple holes in a linear pattern. (See Canned Cycle Introduction #2)

Surface Height: Absolute Z-axis position from where each incremental depth is measured.

Clearance Height: This parameter specifies the Z-axis height used when performing rapid moves to the position of each hole being drilled.

'Rapid To' Depth: The depth to which the cutter rapid moves before beginning to drill the hole at the specified Plunge Rate. This is below the Clearance Height but above the Surface Height.

Hole Depth: Depth of hole (incremental) as measured from Surface Height.

Plunge Rate: Z-axis speed of descent during drilling. The plunge rate can be toggled to modal, fixed or slave, this is indicated by the symbol beside the plunge rate field. If the plunge rate is modal then it will have the "M" symbol or if it is fixed it will have the "F" symbol. The slave plunge rate has no symbol and is set to the last modal plunge rate set in the program, when the modal plunge rate changes all the following slave plunge rates change until the next modal plunge rate is encountered.

Dwell Time: Delay at bottom of hole before starting ascent.

Bore Tool Type: Standard – The tool will bore down at a feed rate and feed backup at a feed rate.
Single Point – The tool will feed down at specified feedrate, Spindle will orient to the M19 position and stop. The tool will retract in the X or Y direction by the angle specified by machine Parameter 136 and then retract at a Rapid feeedrate. The spindle will turn back on when it reaches the Rapid to Depth height.

Note: Single Point bore tool feature uses G76 which needs M19 to orient the spindle before retract

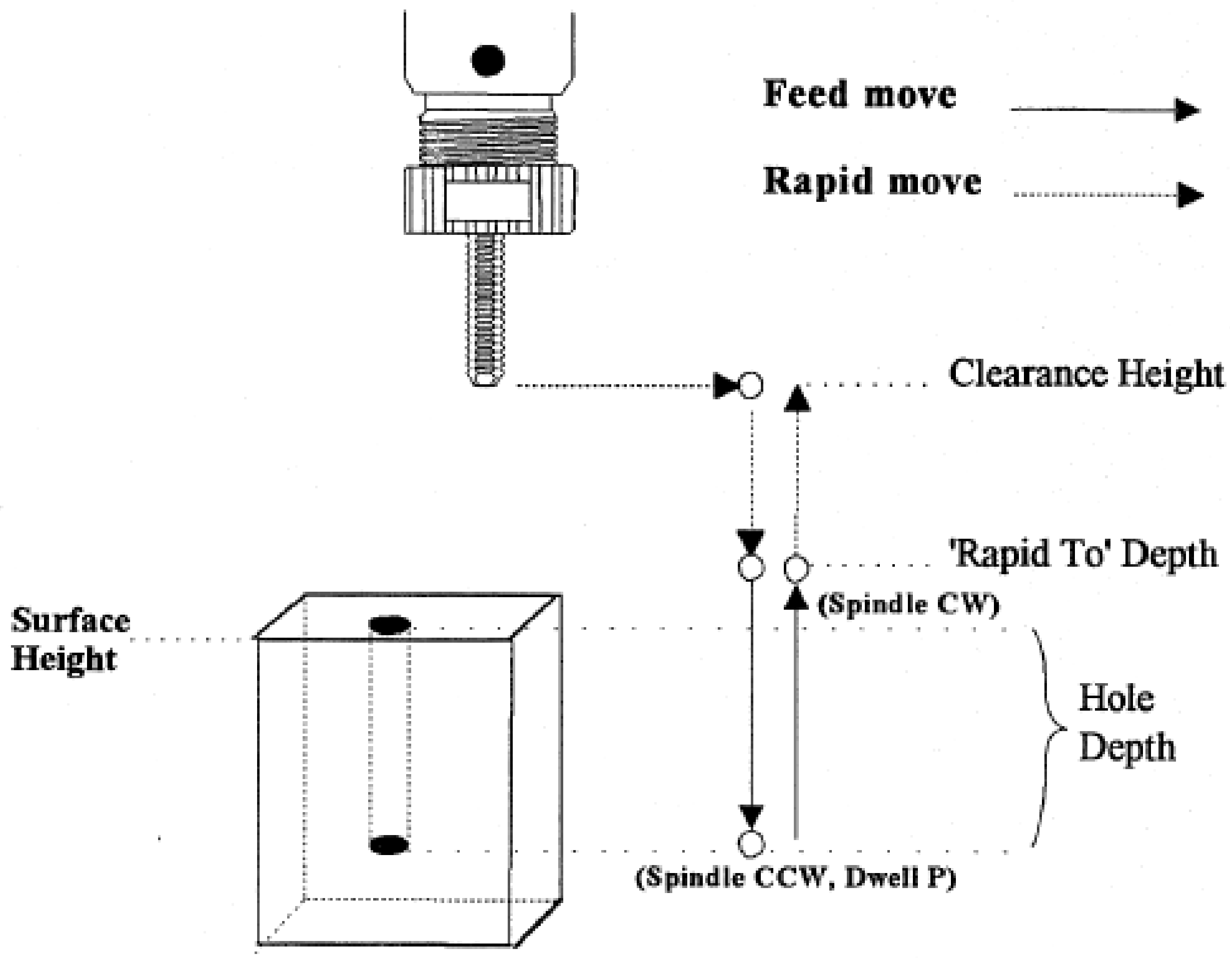
Retract Shift: Only displayed with Single Point Bore Tool Type. This is the amount to shift away in the X or Y direction from the surface before retracting.

Tapping (F3 in the Canned Cycles Menu)

If you press **F2 – Tap** from the Canned Cycle Menu you will gain access to the tapping operations:

The numbers in the fields on the screen correspond to the following example, shown here graphically: Where:

Tapping



Where:

Tap Head Type: Without rigid tapping, this selects either Floating tap head or Reversing tap head (where the special tapping head reverses for you). If rigid tapping is enabled (requires a spindle encoder), you can select either rigid or reversing.

Spindle Direction: Shows the current spindle direction. The spindle direction should be CW for right-hand tapping, and CCW for left-hand tapping. The spindle speed and direction appropriate for the tapping tool should be set in the tool change in which the tapping tool was loaded. This field will be hidden if a reversing tap head is used.

WARNING: The tap must be rotating in the correct direction before performing this operation.

Position: Specifies the X and Y coordinates where the tapping will take place. If either the X or Y coordinate is an incremental value, you will have the option to tap multiple holes in a linear pattern. (See Canned Cycle Introduction #2)

Surface Height: Absolute Z-axis position from where each incremental depth is measured.

Clearance Height: This parameter specifies the Z-axis height used when performing rapid moves to the position of each hole being drilled.

'Rapid To' Depth: The depth to which the cutter rapid moves before beginning to drill the hole at the specified Plunge Rate. This is below the Clearance Height but above the Surface Height.

Depth Total: Depth of hole (incremental) as measured from Surface Height.

Depth Increment: (available only on rigid tapping.) This sets the length of each progressive “peck” down the hole.

Threads / Unit: Number of threads on each inch/mm of the tap. Used in conjunction with the Spindle Speed to calculate the appropriate plunge rate (Plunge Rate = Spindle Speed / Threads per Unit).

Spindle Speed: Rate at which the spindle rotates. Used in conjunction with the Threads / Unit to calculate the plunge

WARNING: The spindle speed must be set before performing this operation. rate.

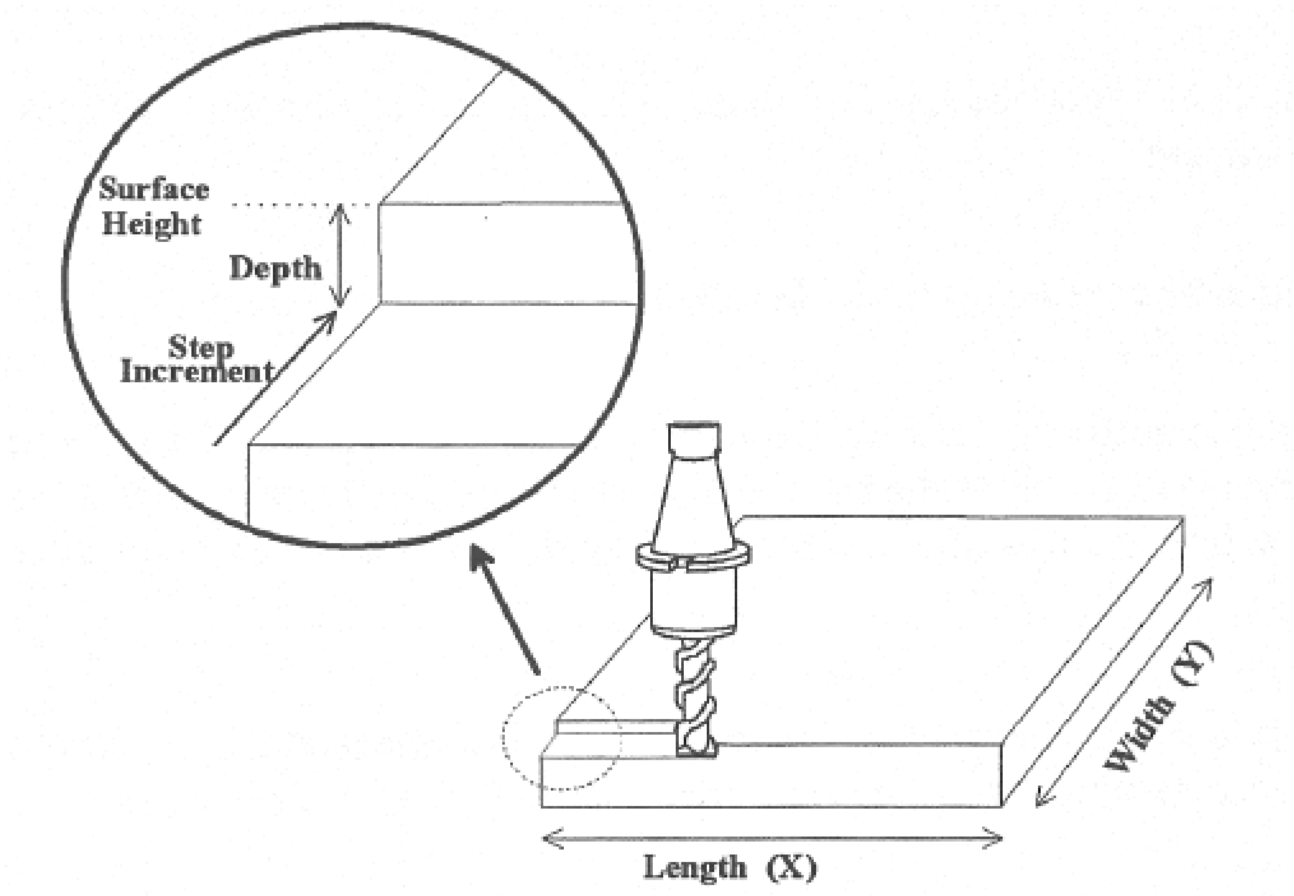
Dwell Time: Delay at bottom of hole before starting ascent. This is used for a floating tap to allow the spindle time to reverse direction at the bottom of the hole. A default value of 0.1 seconds is suggested. This field will be hidden if a reversing tap head is used; the tap head will reverse direction when the quill begins ascending.

Note: When using low gear for tapping, the spindle may turn opposite the direction specified. The operator is responsible for setting the correct spindle speed and direction.

Facing (F4 in the Canned Cycles Menu)

If you press **F4 – Face** at the Canned Cycle Selection Menu, the following screen is displayed:

The parameters in the previous screen correspond to the following dimensions:



Start: X and Y coordinates of the starting corner of the area to be faced.

Surface Height: Z coordinate of the top of the area to be faced.

Length: X-axis dimension of the area to be faced. If a negative value is entered for the length, the facing will occur in the negative X-axis direction from the X-axis start position; otherwise, facing will occur in the positive X-axis direction from the X-axis start position.

Width: Y-axis dimension of the area to be faced. If a negative value is entered for the width, the facing will occur in the negative Y-axis direction from the Y-axis start position; otherwise, facing will occur in the positive Y-axis direction from the Y-axis start position.

Depth: Incremental amount of material to be removed from Surface Height.

Step Increment: Distance that the cutter will step over in the Y direction for each pass.

Plunge Rate: Z-axis speed of descent during facing. The plunge rate can be toggled to modal, fixed or slave, this is indicated by the symbol beside the plunge rate field. If the plunge rate is modal then it will have the "M" symbol or if it is fixed it will have the "F" symbol. The slave plunge rate has no symbol and is set to the last modal plunge rate set in the program, when the modal plunge rate changes all the following slave plunge rates change until the next modal plunge rate is encountered.

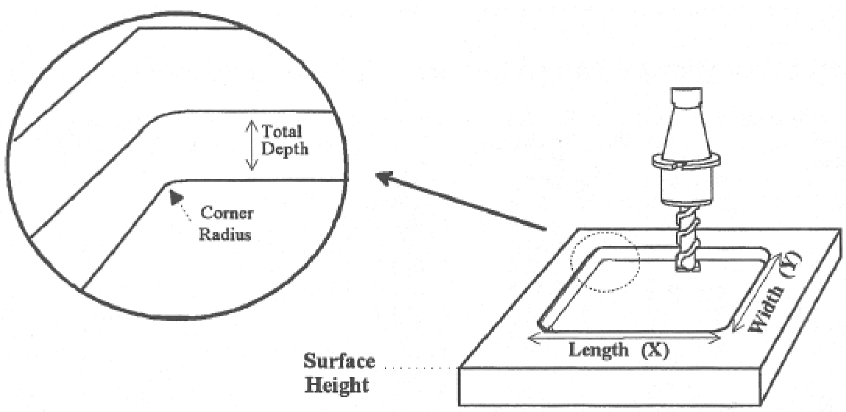
Feedrate: Speed of the cutter during facing. The feedrate can be toggled to modal, fixed or slave, this is indicated by the symbol beside the feedrate field. If the feedrate is modal then it will have the "M" symbol or if it is fixed it will have the "F" symbol shown below. The slave feedrate has no symbol and is set to the last modal feedrate set in the program, when the modal feedrate changes all the following slave feedrates change until the next modal feedrate is encountered.

M F

Rectangular Pocket (F5 in the Canned Cycles Menu)

Pressing F5 – Rect. Pocket from the Canned Cycle Selection Menu displays the following screen:

The parameters on the screen correspond to the following dimensions:



Where:

Center or Corner: Center – X and Y coordinates of the center of the Rectangular Pocket. Corner – X and Y coordinates of the corner of the rectangular pocket. A positive or negative value in the length and width fields will determine the location of the rectangular pocket from the corner position.

Surface Height: Z-axis position from which each incremental depth is measured.

Length: X-axis dimension of the rectangular pocket.

Width: Y-axis dimension of the rectangular pocket.

Corner Radius: Radius of curvature of the corners. It cannot be smaller than the current cutter radius.

Depth: Total: Total depth of the rectangular pocket.

Depth: Per Pass: Depth of each individual pass.

Depth: Plunge Rate: Z-axis speed of descent.

Depth: Plunge Type: Straight or Ramped. Straight plunge does a vertical Z plunge with no X, Y movement. Ramped plunge does a zigzag plunge limited by the Plunge Angle entered below.

Depth: Plunge Angle: The maximum limit angle allowed for a ramped plunge. A special value of 0 means that there is no limit angle.

Note: This field means nothing if the Plunge Type is Straight.

Rough Cuts: Selects type of rough cut: conventional or climb. Use **F3 – Toggle** or **SPACE** to toggle between them.

Rough Cuts: Stepover: Amount of material removed by cutter during each pass around the pocket.

Rough Cuts: Feedrate: Speed at which cutter performs rough cuts.

Finish Pass: Selects type of finish pass: climb, conventional or none at all. Use **F3 – Toggle** or **SPACE** to toggle between them.

Finish Pass: Amount: Amount of material to be removed on the finish pass.

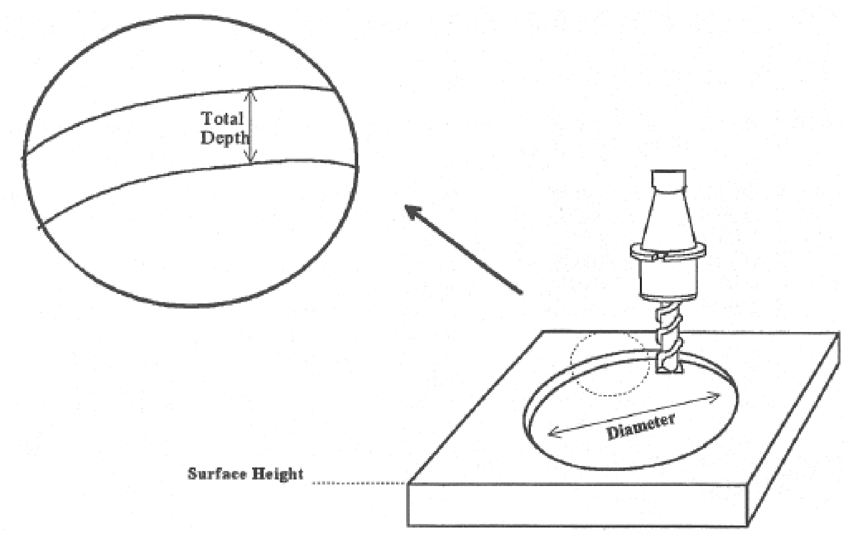
Finish Pass: Feedrate: Speed at which cutter performs finish pass. The feedrate can be toggled to modal, fixed or slave, this is indicated by the symbol beside the feedrate field. If the feedrate is modal then it will have the "M" symbol or if it is fixed it will have the "F" symbol shown below. The slave feedrate has no symbol and is set to the last modal feedrate set in the program, when the modal feedrate changes all the following slave feedrates change until the next modal feedrate is encountered.

M F

Circular Pocket (F6 in the Canned Cycles Menu)

When you press F6 – Circ. Pocket from the Canned Cycle Selection Menu, this screen is displayed:

The parameters on the screen correspond to the following dimensions:



Where:

Center: X and Y coordinates of the center of the circular Pocket.

Surface Height: Z-axis position from which each incremental depth is measured.

Diameter: Diameter of circular pocket.

Cleanout: If cleanout is Yes, then all the material in the pocket will be removed. If cleanout is No, then all the material will not be removed. The cutter starts in the center of the pocket and arcing its way out and then going around the frame.

Depth: Total: Total depth of the circular pocket.

Depth: Per Pass: Depth of each individual pass.

Depth: Plunge Rate: Z-axis speed of descent.

Depth: Plunge Type: Straight or Ramped. Straight plunge does a vertical Z plunge with no X, Y movement. Ramped plunge does a zigzag plunge limited by the Plunge Angle entered below.

Depth: Plunge Angle: The maximum limit angle allowed for a ramped plunge. A special value of 0 means that there is no limit angle.

Note: This field means nothing if the Plunge Type is Straight.

Rough Cuts: Selects type of rough cut: conventional or climb. Use **F3 – Toggle** or **SPACE** to toggle between them.

Rough Cuts: Stepover: Amount of material removed by cutter during each pass around the pocket.

Rough Cuts: Feedrate: Speed at which cutter performs rough cuts.

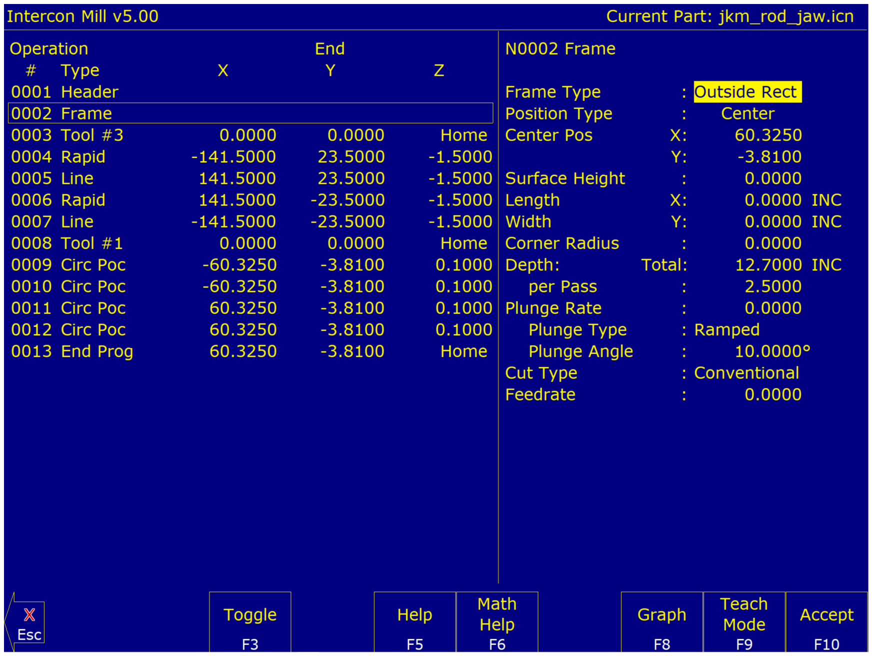
Finish Pass: Selects type of finish pass: climb, conventional or none at all. Use **F3 – Toggle** or **SPACE** to toggle.

Finish Pass: Amount: Amount of material to be removed on the finish pass.

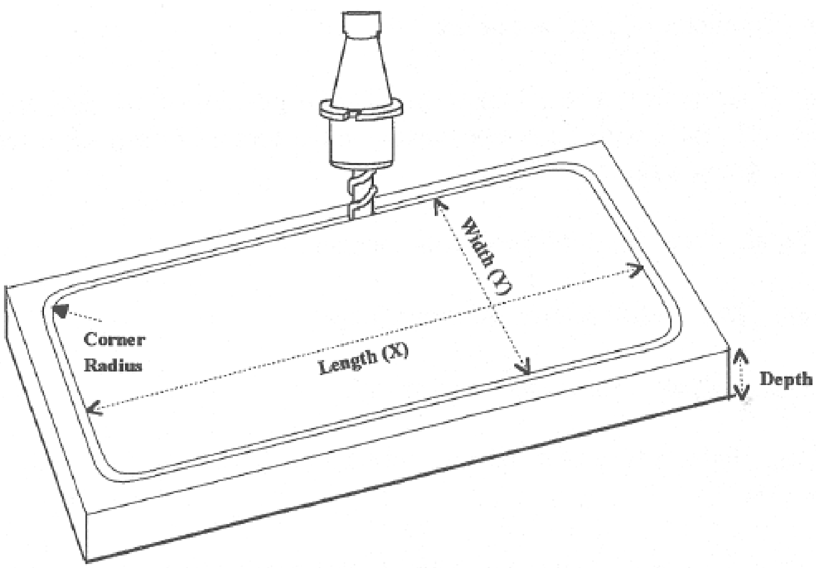
Finish Pass: Feedrate: Speed at which cutter performs finish pass.

Rectangular or Circular Frame Milling (F7 in the Canned Cycle Menu)

When you press F7 – Frame from the Canned Cycle Selection Menu, the following screen is displayed:



The parameters on the screen correspond to the following dimensions (rectangular frame):



Where:

<u>Frame Type:</u> Selects Inside Rectangle, Outside Rectangle, Inside Circle, and Outside Circle. Press **F3 – Toggle** or **SPACE** to toggle between them.

<u>Center:</u> X and Y coordinates of the center of the frame mill.

Surface Height: Z-axis position from where each incremental depth is measured. M-Series Operator's Manual 9/14/2016 29

Length: X-axis dimension of the frame mill. (Rectangular frame only.)

Width: Y-axis dimension of the frame mill. (Rectangular frame only.)

Corner Radius: Radius of curvature of the corners. On an Inside frame, corner radius must be greater than the current cutter radius. (Rectangular frame only.)

Diameter: Diameter of the frame mill. (Circular frame only)

Depth: Total: Total depth of the frame mill.

Depth: Per Pass: Depth of each individual pass.

Plunge Rate: Z-axis speed of descent.

Plunge Type: Straight or Ramped. Straight plunge does a vertical Z plunge with no X, Y movement. Ramped plunge does a zigzag plunge limited by the Plunge Angle entered below.

Plunge Angle: The maximum limit angle allowed for a ramped plunge. A special value of 0 means that there is no limit angle.

Note: This field means nothing if the Plunge Type is Straight.

Entrance Type: Selects type of entrance: Arc On or Arc Off, use **F3 – Toggle** or **SPACE** to toggle between them. (Circular frame only)

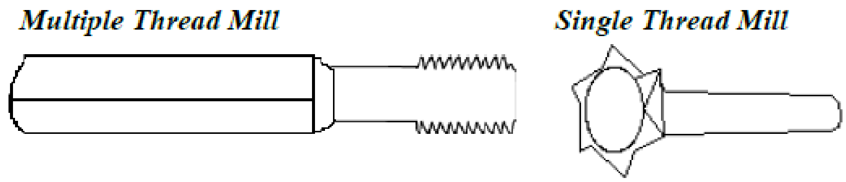
Cut type: Selects type of cut: conventional or climb, use **F3 – Toggle** or **SPACE** to toggle between them.

Feedrate: Speed at which the cutter performs frame mill.

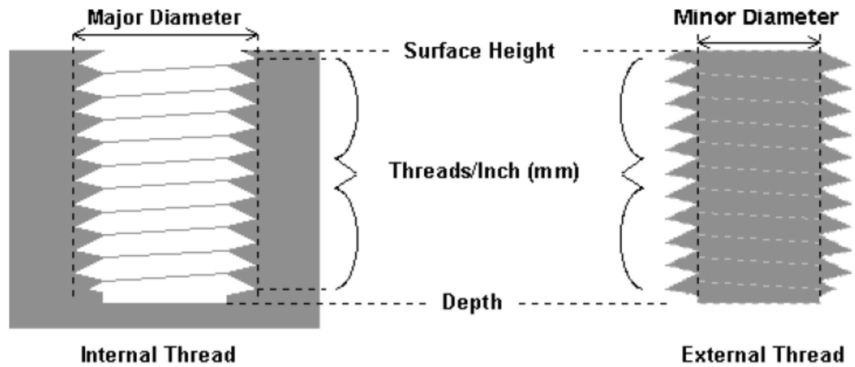
Note: To make a circular frame mill of radius R, specify R as the Corner Radius and set the Length and Width parameters equal to 2 x R.

Thread Milling (F8 in the Canned Cycles Menu)

When you press **F8 – Thread** from the canned cycle menu, the following screen is displayed:



The parameters on the screen correspond to the following:



Where:

Center: X and Y coordinates of the center of the thread mill operation.

Diameter: Major diameter of thread for internal thread milling and minor diameter for external thread milling.

Thread / Unit: Number of threads per inch or mm. Used to calculate thread pitch.

Thread Pitch: Thread pitch calculated from threads/unit field. This field cannot be modified.

Thread Type: Specifies right or left hand threads.

Thread Direction: Specifies whether to start at the bottom of the hole and work up or start at the top of the hole and work down.

Tool Type: Single point or full form threading tool.

Thread Approach: Internal or external thread.

Clearance Amount: Used for external thread milling only. Specifies the diameter of the lead-in arc. Minimum clearance is 0.050 inches.

Clearance Angle: Used for external thread milling only. Specifies the angle from which the lead in arc will start.

Feedrate: Cutting feed rate.

Surface Height: Absolute Z-axis position from position from where the incremental depth is measured.

Clearance Height: This parameter specifies the Z axis height used when performing rapid moves to the position of each hole being thread.

'Rapid to' Depth: The depth to which the cutter rapid moves before beginning to thread mill at the specified Plunge Rate. This is below the Clearance Height but above the Surface Height.

Depth: The total depth of the thread.

Number of Passes: Number of times the thread mill is to be done on the same hole.

Cleanout (F9 in the Canned Cycles Menu)

The cleanout cycle performs a horizontal zigzag pocket cleanout of a profile composed of lines and arcs. When you press **F9 – Cleanout** from the canned cycle menu, the following screen is displayed:

Where:

Rough Cuts: Selects type of rough cut. Use **F3 – Toggle** or **SPACE** to toggle between Conventional and Climb.

Type: Selects type of cleanout to use. The choices are Collapse, which cleans out the pocket from the outside in, or Expand, which cleans out the pocket from the inside out. You can switch between the two by using the **F3 – Toggle** or **SPACE** keys

Stepover: The distance between each step in the pocket cleanout. This value cannot be greater than 50% of the tool diameter.

Feedrate: Speed at which cutter performs rough cuts.

Finish Pass: Selects type of finish pass. Use **F3 – Toggle** or **SPACE** to toggle between Conventional, Climb, or None.

Amount: Amount of material to be removed on the finish pass.

Feedrate: Speed at which cutter performs finish pass.

Tool Number: Tool number to be used for the finish pass.

Surface Height: The Z-axis position from where the incremental depth is measured.

Clearance Height: This parameter specifies the Z-axis height to which the tool is retracted before moving to different segments during a pocket cleanout.

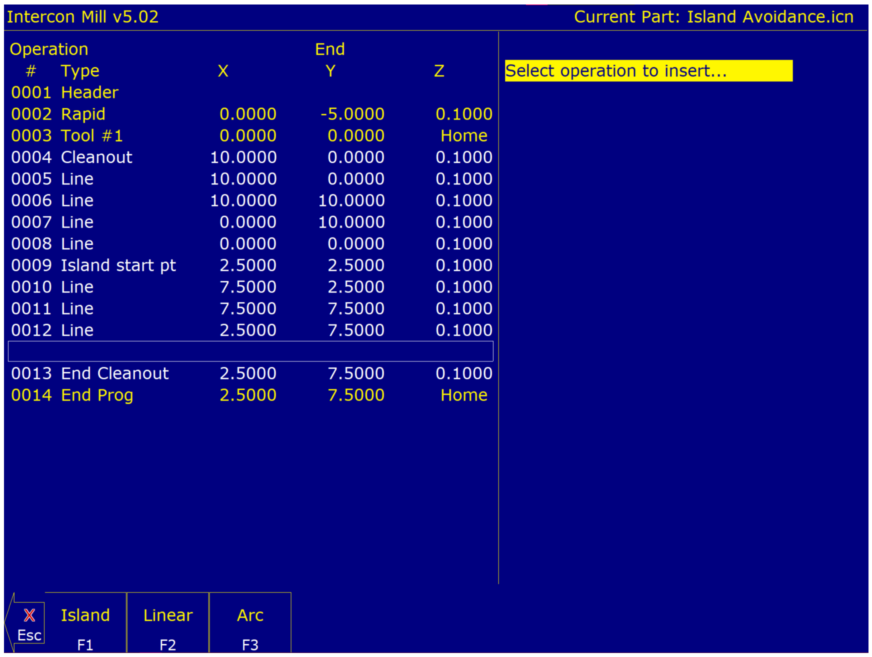
‘Rapid To’ Depth: The depth to which rapid positioning moves will be made to when moving the Z axis downward.

Depth: Total: The total depth of the pocket measured as an incremental depth from the surface height.

per Pass: The depth amount of cut to be taken to reach the total depth. This value must be greater than 0.0 and cannot exceed the total depth.

Plunge Rate: The feedrate at which the Z axis is moved when plunging to a lower depth.

After the cleanout parameters are accepted, a screen similar to the following appears:

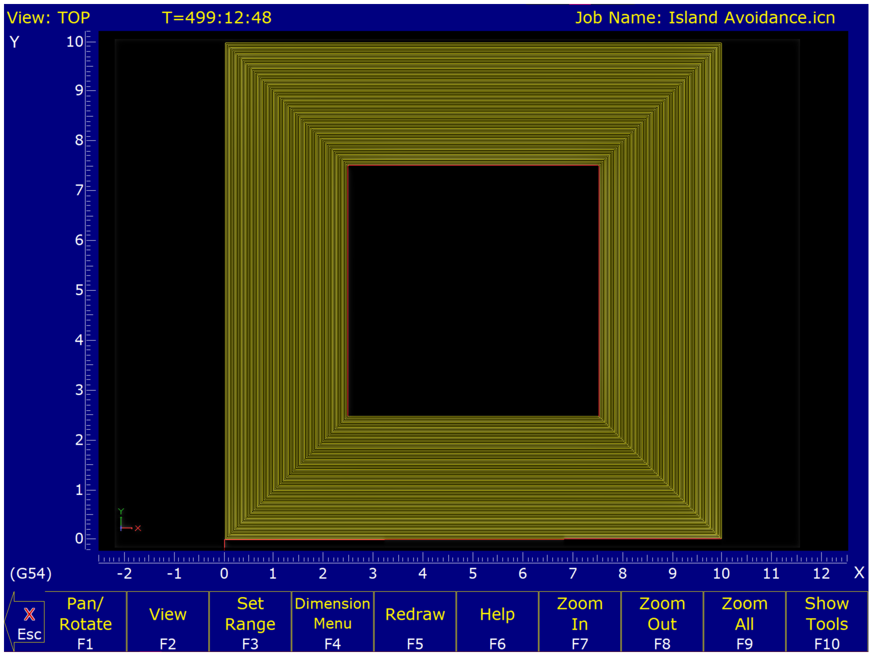


Key points about the Cleanout cycle:

- When creating a pocket the first move in your cleanout cycle must be a linear move.
- If the profile contour does not end at the start point, a linear segment will automatically be inserted to close the pocket.
- The last line of the contour will not include a connecting radius to the starting point.

Once you have defined the specifics of your cleanout cycle, the shape of your pocket will be defined through a series of lines and arcs by choosing the **F2 – Linear** and **F3 – Arc** options in the cleanout cycle.

F1 – Island (Island Avoidance) Once you have defined a pocket in the cleanout cycle. There may be areas or islands that you don't want cleaned out. To create an island press **F1 – Island**, enter the starting point of the island and then use **F2 – Linear** and **F3 – Arc** to create the island. See an example of a completed cleanout backplot below.



10.7.6 F6 – Other

Choosing F6 – Other will display the following operations that may be inserted.

F1 – Comment – Enter a comment, up to 35 characters long, which will be displayed in the generated CNC program.

F2 – Spindle – Change the actual state of the spindle. Press **F3 – Toggle** or **SPACE** to toggle between CW, CCW, and OFF.

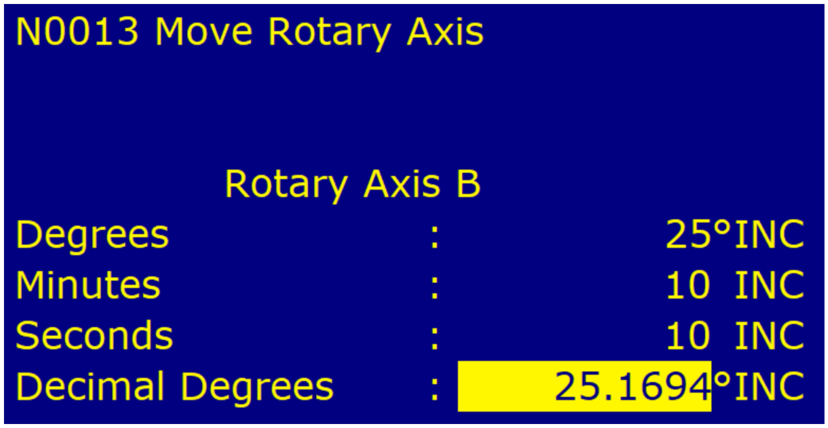
F3 – Coolant – Change the actual state of the coolant. Press **F3 – Toggle** or **SPACE** to toggle between FLOOD, MIST, and OFF.

F4 – Clamp – Turn the Clamp ON and OFF. Press **F3 – Toggle** or **SPACE** to change the clamp state.

F5 – Z Home – Send the Z-axis to its home position.

F6 – M & G Code – Enter M & G codes into your Intercon part program. Great care must be taken when using this function, as you could cause unpredictable results in the controller if you accidentally changed positioning modes in your program, or perhaps turning the spindle off during a cut.

F7 – Rotary – Insert a rapid rotary move. This operation requires a rotary fourth axis. The fields are identical to the fields in the Linear Mill operation as shown below, but the resulting move is a G0 (Rapid) moving only the rotary axis.



F8 – Import DXF – Insert Intercon operations created from objects in DXF files. For more information, refer to the “Importing DXF files” section later in this chapter.

10.7.7 F7 – Cutter Compensation

Pressing **F7 – Cutter Comp** from the Insert Operation screen, will insert a cutter compensation command. Press **F3 – Toggle** or **SPACE** to select cutter compensation Left, Right, or Off. Cutter compensation may be used with Linear Mill, Frame Mill, and Rapid Traverse operations. For details on using cutter compensation, see the section “G40, G41, G42 – Cutter Compensation” in Chapter 12.

The Rectangular Pocket, Circular Pocket, Frame Mill and Cleanout canned cycles perform cutter compensation automatically. If compensation left or right was selected before the canned cycle, it will be turned off.

10.7.8 F9 – Subprograms

Intercon subprograms allow you to make additional copies of a programmed contour. The copies may be repeated in the x/y axes, depth repeat, rotated, or even a mirror image of the original. To create a subprogram, first define the operations that will compose the contour. Any type of program operation (rapid, linear mill, arc mill, canned cycle, subprogram, etc.) may be included in the contour. These operations must be programmed at the Z depth at which the first pass will occur. When you are finished doing this, return to the Program Edit Menu. Move to the place in the program where you want to repeat these operations and press the **F3 – Insert key**. The operations will be performed once before the repeat operation occurs; therefore the operations to compose the contour should be defined at the place in the program where they should occur first.

When you press F9 – Subpgm from the Insert Operation screen you will see the Insert Subprogram screen:

You may now select the type of subprogram desired.

A typical subprogram screen appears as follows:

All subprogram operations contain the following fields:

Start Block: Selects the first operation in the block of operations to repeat. This operation must lie before the place in your program where you are trying to repeat operations.

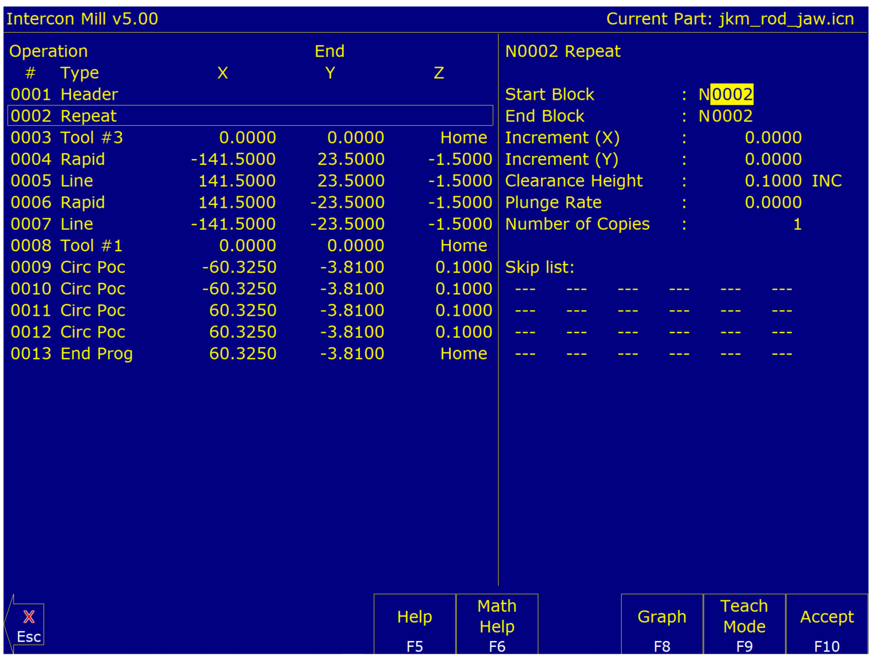
End Block: Selects the last operation in the block of operations to repeat. Again, this operation must lie prior to place in your program where you are trying to repeat operations, but not precede the start block.

Clearance Height: This field determines the Z height at which the tool is moved over the work piece before being repositioned at the start of the contour. This value must meet or exceed the maximum Z height of all operations contained within the contour. If any operation places the tool at the Z home position, then you **must** tie this value to the home position (**F2 – Z Home**).

Plunge Rate: This is the speed at which the tool is repositioned on the Z-axis when moving to the **beginning** of the first move of the contour. This has no effect on a plunge that you have programmed into the contour; however, this has the effect of providing a vertical plunge for you in the event that you do not program your own plunge into the contour.

Other fields specific to the various subprogram operations are described in the next few pages.

Repeat to Depth (F1 in the Insert Subprogram Menu)



The Repeat to Depth feature is useful for repeating a part contour when the material being machined is too thick to cut in just one pass. The contour formed by these operations may either be a closed contour or an open one. If a non-vertical plunge to the start of the contour is desired, it must be programmed into the contour (a vertical plunge between passes will be provided if one is not programmed).

Total Depth: Indicates how deep the final depth pass is to be. This is a positive value. Note that because the contour has been programmed at a depth of one depth increment below the work piece surface, the final depth assumes that one depth pass has already been performed, and, thus, subtracts one depth increment from the total depth.

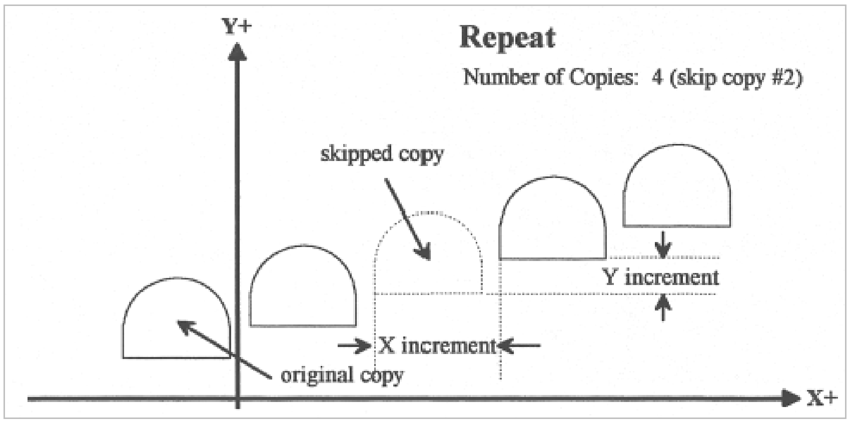
Depth Increment: Specifies the distance to drop each time the contour is repeated. This is a positive value that may not exceed the total depth of the operation.

When you have finished entering the required parameters, press **F10 – Accept** to accept them. An operation labeled “Depth Rpt” will be inserted into your program in front of the highlighted operation. You may now edit this operation just as you would edit any other operation (use the cursor keys to highlight the “Depth Rpt” operation, and then press **ENTER**.

Note: If you wish to change the amount of the depth increment per pass after the contour has been programmed, you must also change the Z depth of all the operations inside the contour to correspond to the new increment.

Repeat (F2 in the Insert Subprogram Menu)

The Repeat feature is useful for repeating a part contour one or more times along a straight line in the XY plane. The contour formed by these operations may either be closed or open. If a rotary axis is enabled, this operation can also be used for repeating such a contour one or more times over a specified rotary increment.



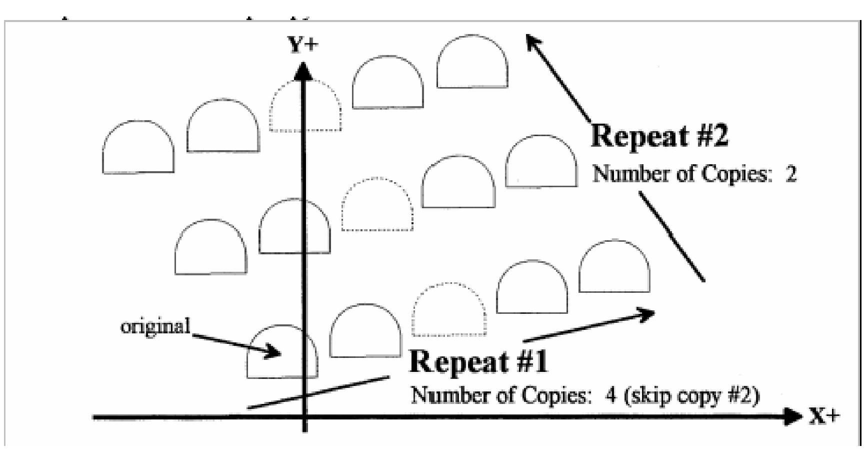
Increment: Specifies the X and Y distances between the start points of each copy of the contour.

Rotary Increment: Specifies the rotary incremental amount to move between each copy of the contour.

Note: This field will appear only if a rotary axis is enabled.

Number of Copies: The number of times to repeat the contour.

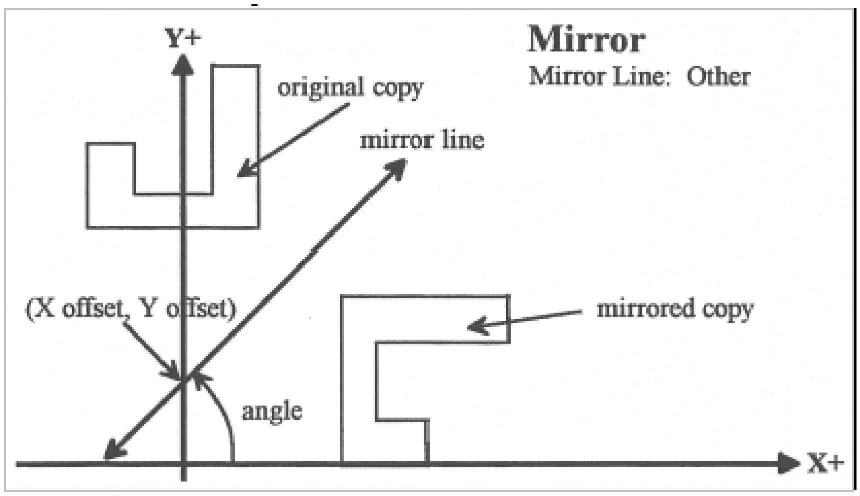
Skip List: List of copies that are skipped. Enter the number or numbers of the copies that you wish to skip. In the example below will skip copy #2.



Note: An array of repeats may be accomplished by doing a repeat of a repeat.

Mirror (F3 in the Insert Subprogram Menu)

The Mirror feature is useful for reflecting a part contour over a line. The contour formed by these operations may either be closed or open.



Mirror Line: Specifies the type of mirror line to use. Choices are Horizontal, Vertical and Other (user-defined).

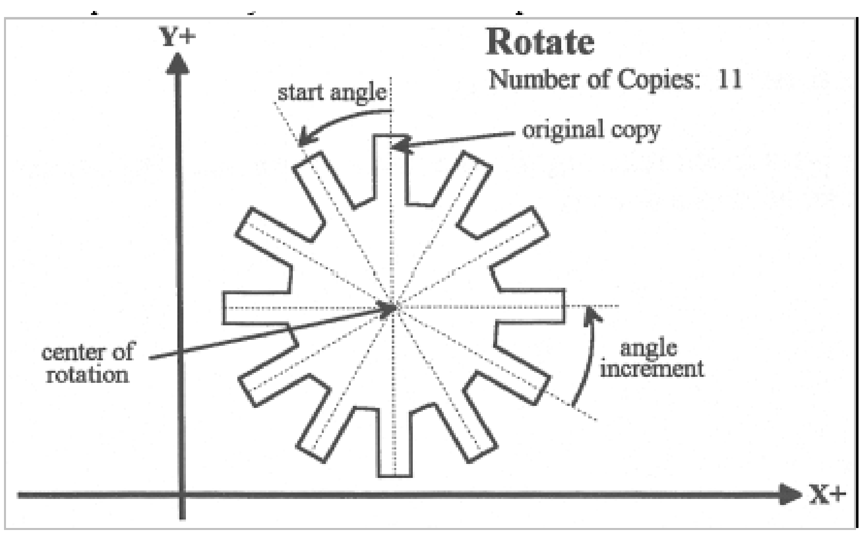
X Offset: Specifies the X coordinate on the Mirror Line. This field will not be visible for a horizontal mirror line.

Y Offset: Specifies the Y coordinate on the Mirror Line. This field will not be visible for a vertical mirror line.

Angle: Specifies the angle (from the three o'clock position) of the Mirror Line. This field will only be visible for a user-defined mirror line and is used in conjunction with the X Offset and Y Offset fields to define the mirror line.

Rotate (F4 in the Insert Subprogram Menu)

The Rotate feature is useful for rotating a part contour multiple times around a given point. The contour formed by these operations may either be closed or open.



Center: The XY location of the center of rotation.

Start Angle: The angle from the original copy at which the first copy will be placed. A positive angle indicates a counterclockwise rotation, while a negative angle indicates a clockwise rotation.

Angle Increment: The angle at which each copy after the first will be placed from the first copy. A positive angle indicates a counterclockwise, while a negative angle indicates a clockwise rotation. Must have a value larger than 1 in the number of copies.

Number of Copies: The number of times to rotate the contour.

End Angle: The angle at which the final rotated copy will start, **not the angle at which it will end**. A positive angle indicates a counterclockwise rotation, while a negative angle indicates a clockwise rotation.

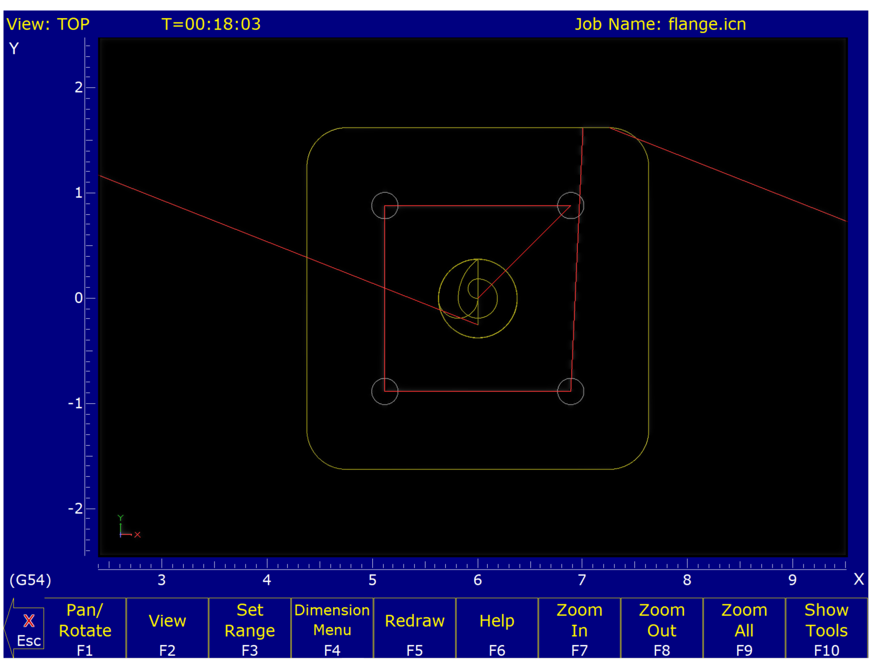
Skip List: List of copies that are skipped. Enter the number or numbers of the copies that you wish to skip.

Note: The user may enter the Start Angle, the Number of Copies, and either the Angle Increment or the End Angle value, and Intercon will compute the rest.

10.8 Graphics

Intercon features three-dimensional previews of the tool path to be followed when milling the part. You may choose to display your project in one of two formats: a three-plane display, where the project is shown in each of the XY-, ZX-, and YZ-planes; or an isometric display, which depicts the project three-dimensionally from an observer's point of view. To view the graphics, press **F8 – Graph** from the Main Menu or from any Operation Edit screen.

The format of the display will be similar to the following:



This display consists of arcs and/or lines that make up the tool path followed. Rapid (G0) moves will appear in Red, linear (G1) and arc (G2, G3) moves will be Yellow, and compensated pass will appear in grey. Canned cycle operations (except the facing cycle) will also display in gray. This option is enabled by setting Parameter 260 to 0 (See Chapter 15 Configuration). Under this setting, the operation of the user interface is slightly different from the regular Accelerated Graphics Backplot described above.

10.8.1 Accelerated Graphics Backplot

Accelerated Graphics Backplot is a tool path graphics display. This option is enabled by default.

F1 – Pan/Rotate Press this key to change the behavior of the keyboard arrow keys. Normally, they will pan (scroll) around the drawing, but after pressing this key the arrow keys will control rotation instead. When in rotation mode, an axis indicator is drawn to mark the center of rotation.

F2 – View Press this key to change the planar view of your part. The view is indicated by TOP, RIGHT, or FRONT shown at the top of the screen.

F3 – Set Range Press this key to select which blocks of G-code to display. Only blocks that fall within the range you specify will be drawn.

F4 – Dimension Menu Press this key to access a sub-menu of options:

- **F1 – Prev Line:** Press this to walk forward to the next G-code line and graphically highlight it. If this G-code line contains movement, the Start and End points will be displayed at the bottom of the screen.

- **F2 – Next Line:** Press this to walk backward to the previous G-code line and graphically highlight it. If this G-code line contains movement, the Start and End points will be displayed at the bottom of the screen.

- **F3 – Go To Line:** Press this key to graphically highlight a particular G-code line whose line number you specify. If this G-code line contains movement, the Start and End points will be displayed.

- **F4 – Measure:** Use this feature to measure between any 2 selected points. To do this, use a mouse to move the pointer over the first point and then press F4 – Measure to anchor the first point. Then use the mouse to move the pointer to the second point. As you move the mouse towards the second point, you will notice an Offset and Measurement display changing dynamically as you move the mouse. Also you may notice some "snap to" effects as you move the pointer close to start and end points of entities that make up your program.

F5 – Redraw Press this key to redraw the part slowly, which can be useful for visualizing the movements the machine will make. While the display is being redrawn, you can use the feedrate override knob to adjust the rate at which it is being drawn. If you don't have a feedrate override knob, the + and − keys can be used to adjust the rate. Pressing F5 again will cancel this mode.

F6 – Hide Rapids Press this key to hide rapid movements. Press it again to show them.

F7 – Zoom In Press these keys to zoom into the part relative to the center of the screen.

F8 – Zoom Out Press these keys to zoom away from the part relative to the center of the screen.

F9 – Zoom All Press this key to fit the entire part inside the screen.

F10 – Show Tools Press this key to show the tools menu, which allows you to highlight movements of certain tools. Press this key again to hide the tools menu.

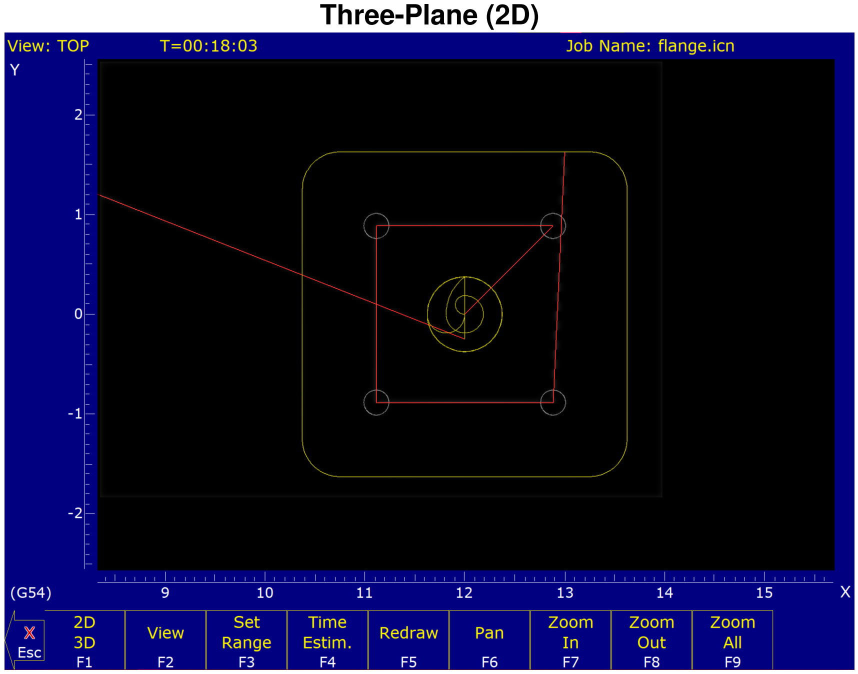
Spacebar – Measure Press this key to take a measurement between two points. In a 2D view, this measurement will be a 2D measurement. In a 3D view, it will be a 3D measurement (and the measurement will only be valid if the crosshairs are snapped to a line of the tool path).

Note: If you have a mouse or touch screen attached to your device, you can use that to control the graphing window. Holding the left mouse button allows you to drag the part across the screen, while the right mouse button controls rotation of the part. Spinning the mouse wheel (or holding both left and right buttons) zooms in and out. Double clicking on a feedrate movement will center the camera on that movement (which is very useful) and also tells you the length of that movement. For touchscreen operation, use the F1 key to switch between Pan and Rotate modes.

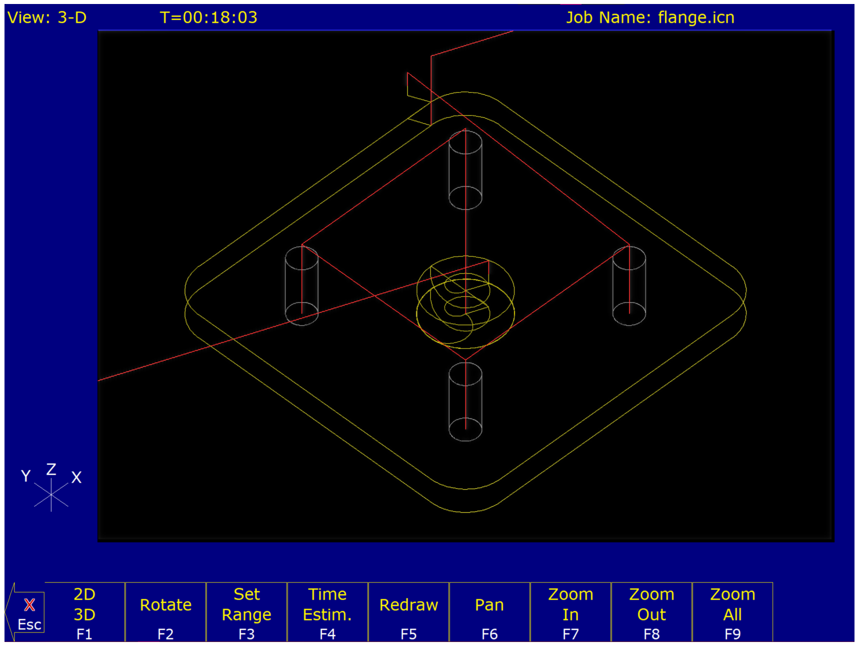
10.8.2 Three-plane 2D View (Legacy)

F1 – 2D/3D: Selects the format of the project display. This may take the form of the three-plane display (2D) or the isometric display (3D).

Three-Plane (2D)



Isometric (3D)



F2 – View/Rotate: In three-plane (2D) view, F2 – View switches the point of view to a different plane. In isometric, (3D) view, F2 – Rotate enables the arrow keys to rotate the figure. The arrow keys actually rotate a larger version of the YZX axes figure that shows the orientation in which the part will be redrawn. Press F2 – Rotate to redraw without leaving rotation mode. If you press Enter or F5 – Redraw after rotating the axes, the display of the axes will disappear. To rotate to a different angle you will have to press F2 – Rotate again. Press Esc to cancel rotation.

F3 – Set Range: Specify the range of operations to draw. You will be prompted for a start block and an end block.

F4 – Time Estimate: Press F4 – Time Estim. to hide or display the time estimate in the upper left-hand portion of the screen.

F5 – Redraw: Pressing the F5 – Redraw key will cause the simulation to start again from the first operation (Redraw).

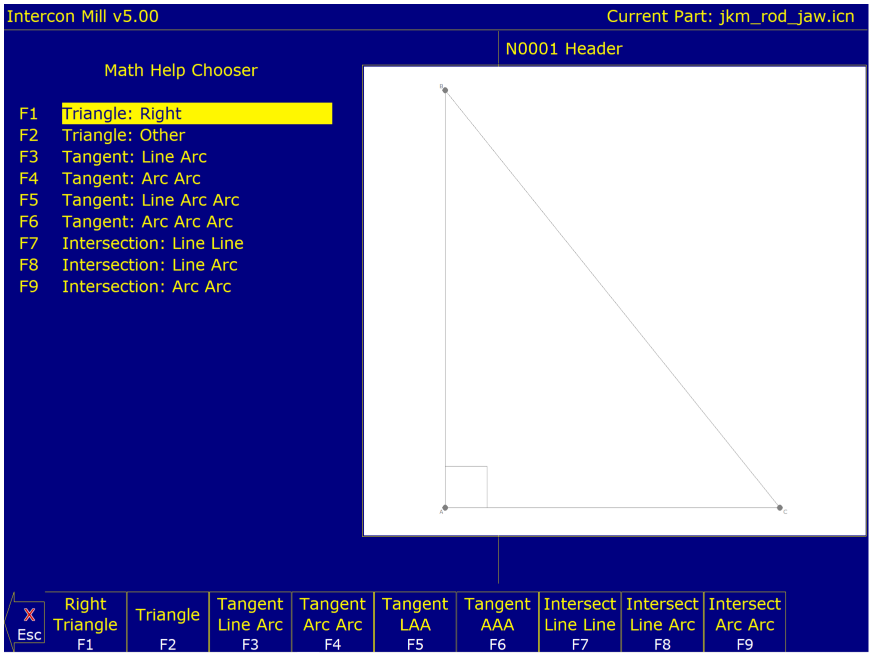
F6 – Pan: When using the pan feature, the project can be centered to the crosshairs in the display windows of the three-plane display, or rotated around the center of the isometric display screen. To enter pan mode, simply press the F6 – Pan key or press one of the Arrow keys. A set of crosshairs will appear. Adjust the center of the crosshairs to the new desired center. Press Enter, F5 – Redraw, F6 – Pan to redraw the part with the new screen center point.

F7, F8 & F9 – Zoom In, Zoom Out & Zoom All: The project can also be viewed in an enlarged or reduced state by pressing the F7 – Zoom In or F8 – Zoom Out keys to activate Zoom In and Zoom Out respectively. Pressing F9 – Zoom All redraws the project at its original size. Use the arrow keys to select the new screen center before zooming in or out.

Number keys and Space bar – Feed Rate Override & Hold: If no jog panel is attached (or "Keyboard" has been selected as the jog panel type) the number keys 1–9 and 0 choose feed rate overrides 10%–90% and 100%, respectively. 1 is 10%, 9 is 90%, and 0 is 100%. If there is a jog panel attached you can use the feedrate knob to adjust the speed as well. The space bar toggles feed hold on and off.

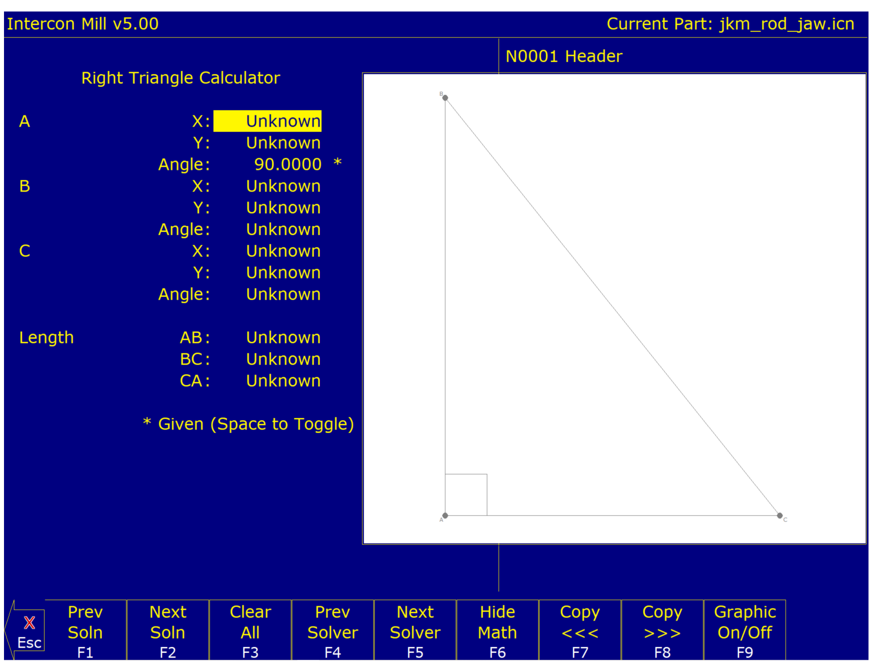
10.9 Math Help

Intercon provides a math assistance function to solve the trigonometric problems common in part drawings. To enter Math Help, press **F6 – Math Help** from any Edit Operation screen. The first time that you invoke Math Help, the following screen appears which shows all available *solvers*:



The figures on the right are a graphical representation of the highlighted solver on the left. Pressing **ENTER** key will display another menu that has various fields particular to the type of problem that is being solved. The graphic below displays the Right Triangle Calculator menu. The options that are available on the function keys are the same for every type of math help solver and perform the following operations:

10.9.1 F1 – Triangle: Right



F1 – Prev Soln (Previous Solution)

F2 – Next Soln (Next Solution) The **Prev Soln** and **Next Soln** options will cycle backward and forward, respectively, through the available solution sets for math solvers that may have multiple solutions. A status line near the bottom left of the screen appears once a valid solution has been found. The solution status line indicates the total number of solutions and the solution number that is currently represented by the graphic display on the right. For example, in an Arc Tangent Arcs math help, the display solution status may be "– Solution 1 of 8 –". In this case, the **Prev Soln** and **Next Soln** can be used to cycle through all eight of the solutions.

F3 – Clear All The Clear All option removes all solutions. It sets all fields for a particular solver to UNKNOWN.

F4 – Prev Solver

F5 – Next Solver The Prev Solver and Next Solver options cycle backward and forward, respectively, through the various math help solvers. These options are shortcuts which have the same effect as pressing **ESC** to reach the main math help menu, navigating to the previous or next math help option, and then pressing **ENTER**.

F6 – Hide Math The **F6 – Hide Math** option exits math help mode and returns to the operation edit menu. Pressing **F6 – Math Help** to invoke Math Help again will restore Math Help exactly as you left it. After copying values from Math Help, you can press **F6 – Hide Math** to hide Math Help, and then hit **F10 – Accept** to accept the values entered.

F7 – Copy $<<<$

F8 – Copy $>>>$ The **F8 – Copy** $<<<$ option will move the value from the selected edit operation field into the selected math help menu field and the **F7 – Copy** $>>>$ operation will move the value from the selected math help menu field into the selected edit operation field. For both options, the selected fields in the math help menu and the operation edit menu are advanced. Only when the graphics display is off will the Copy operations actually copy values and advance field selections.

The currently selected fields have either a box drawn around them or are highlighted depending upon which menu is active. The active menu, which is either the math operation menu on the left hand side or the operation edit menu on the right hand side, depicts the selected field by highlighting the entire field. The non-active menu displays the active field with a box drawn around it. Use the arrow keys to select fields as described below.

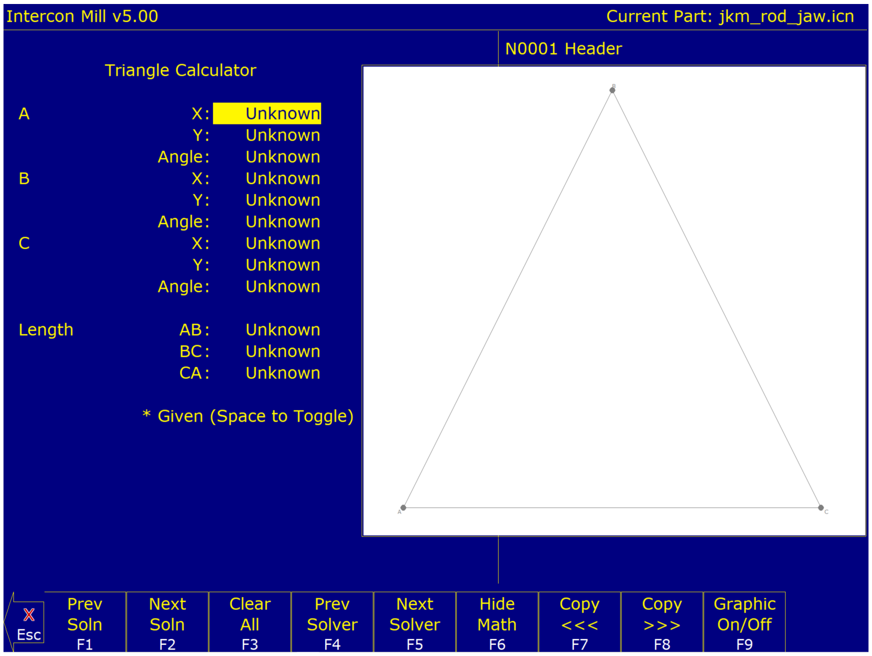
F9 – Graphic On/Off Toggle the graphical representation of the math help menu on the display.

↑↓←→ **(Arrow Keys) – Select Fields** The **LEFT** and **RIGHT** arrow keys are used to navigate between the math menu and the edit menu. The **UP** and **DOWN** arrow keys are used to navigate within a menu. To choose fields for the "Copy" option, above, use the UP and DOWN arrow keys to highlight the desired field in the menu and use the **LEFT** or **RIGHT** arrow keys to switch menus.

Other Features Common To All Math Help Operations

In some math help operations, there will be an asterisk '*' character that appears immediately to the right of a field. This character marks the field as a "given" field, which means that the value of this field will be held constant in the process of solving the math equations.

10.9.2 F2 – Triangle:Other



See button explanations above in the **F1 – Triangle: Right** section.

The screen will show UNKNOWN if the value of each parameter is not known. Math Help waits for known values to be entered, where:

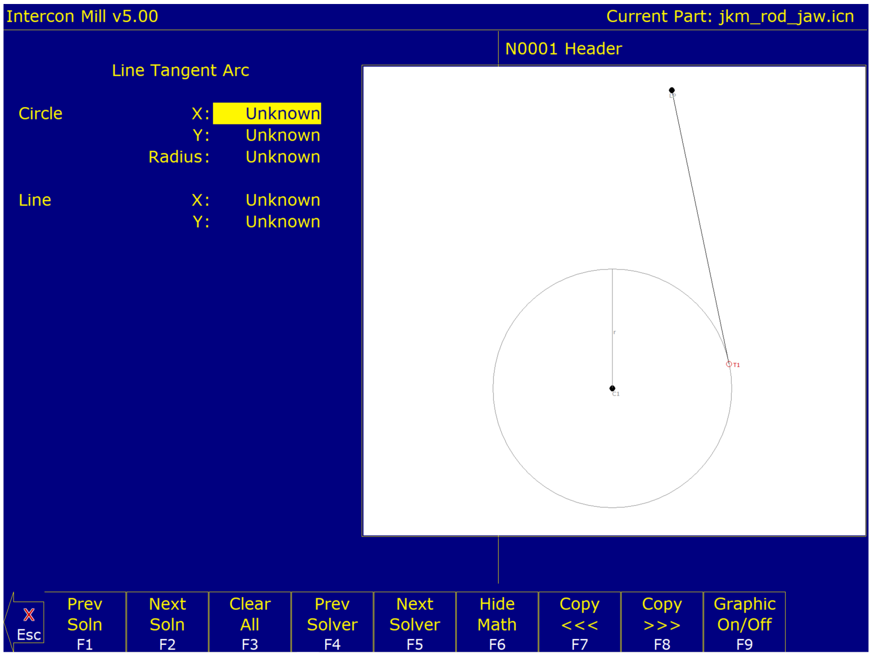
Point a, **b**, or **c** is the coordinate value for each corner of the triangle.

Angle A, **B**, or **C** is the angle at each point of the triangle.

Length of values are the distances between the points indicated.

Continue adding all the known parameters. Select parameters using the arrow soft keys. When Math Help solves the remaining unknown values, the screen will display them.

10.9.3 F3 – Tangent: Line Arc

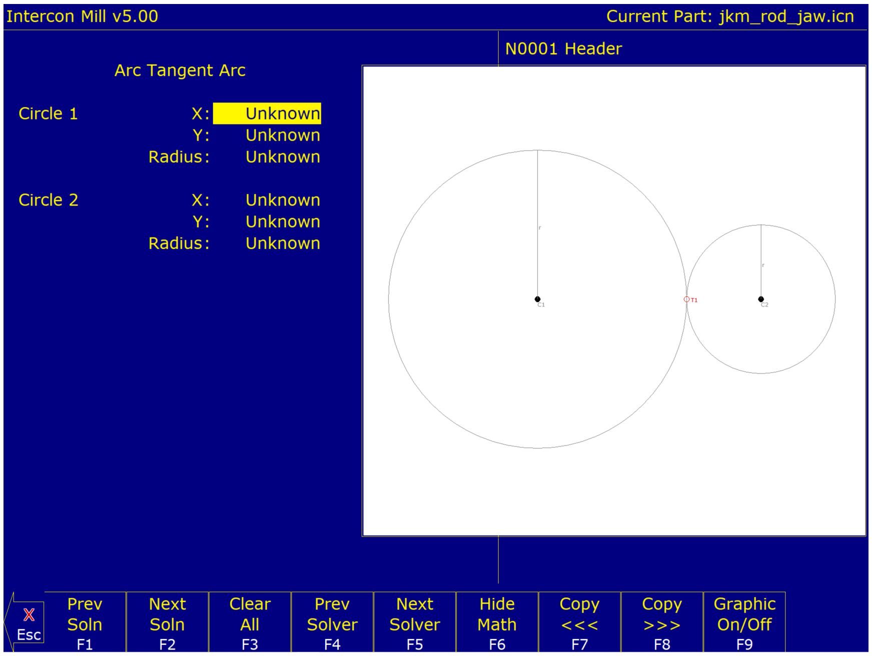


See button explanations above in the **F1 – Triangle: Right** section.

Given the center (C1), the radius of an arc, and 1 point (LP) on a line, find the lines tangent to the arc (defined by the tangent point (T1)).

You must enter the X and Y coordinates for the circle's center point, the circle's radius, and the X and Y coordinates for a point on the line.

10.9.4 F4 – Tangent: Arc Arc

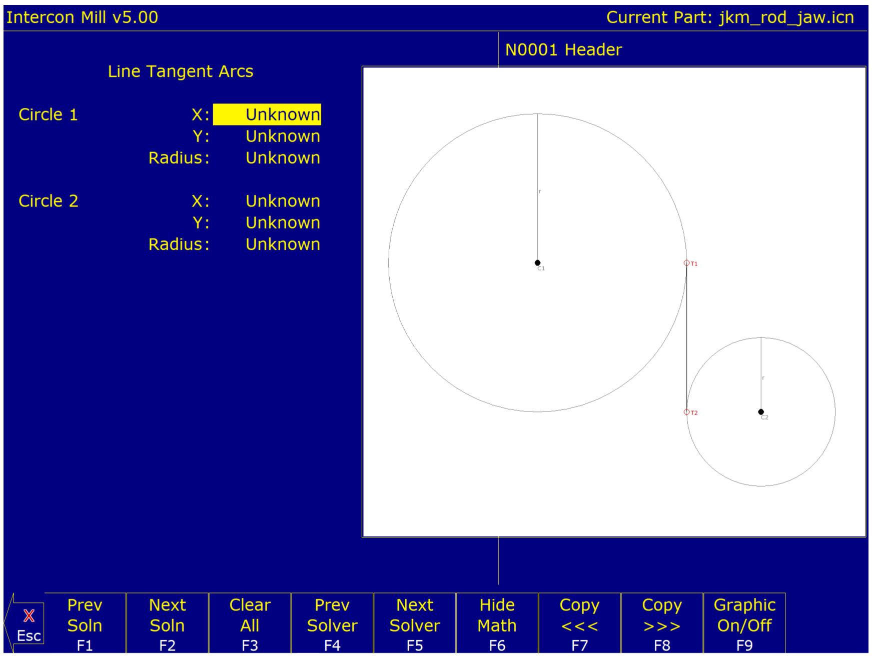


See button explanations above in the **F1 – Triangle: Right** section.

Given the center points (C1 and C2) and radii (R1 and R2) of two arcs, find the point (T) at which they are tangent.

You must enter the X and Y coordinates for the first circle's center point, the radius of the first circle, the X and Y coordinates for the second circle's center point, and the second circle's radius.

10.9.5 F5 – Tangent: Line Arc Arc

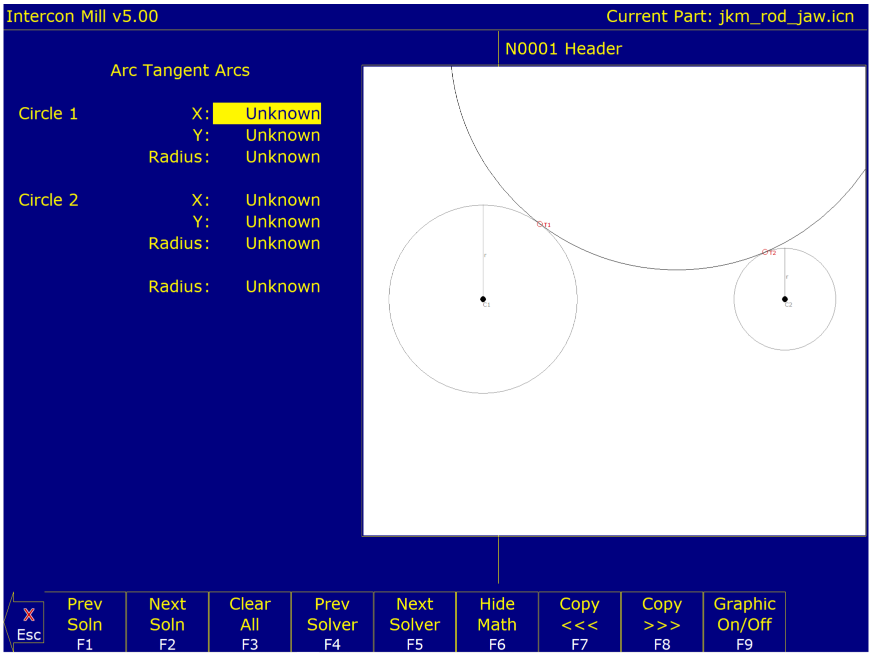


See button explanations above in the **F1 – Triangle: Right** section.

Given the center points (C1 and C2) and radii (R1 and R2) of two arcs, find the lines (defined by T1–T2) tangent to both arcs.

You must enter the X and Y coordinates for the first circle's center point, the radius of the first circle, the X and Y coordinates for the second circle's center point, and the second circle's radius.

10.9.6 F6 – Tangent: Arc Arc Arc

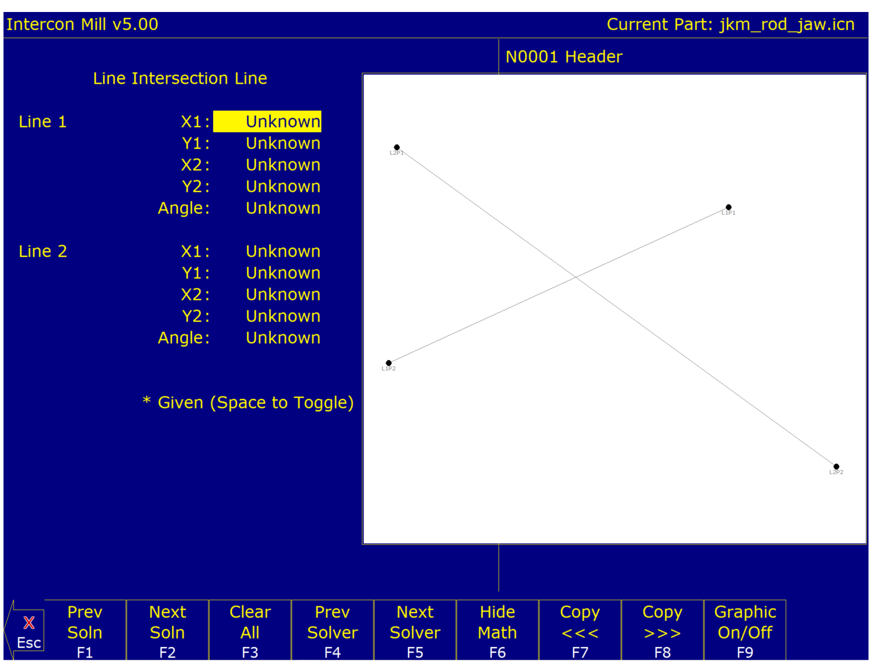


See button explanations above in the **F1 – Triangle: Right** section.

Given the center points (C1 and C2) and radii (R1 and R2) of two arcs and the radius of a third arc, find the center point of the third arc and the tangent points (T1 and T2).

You must enter the radius of the tangent arc, the X and Y coordinates for the first circle's center point, the radius of the first circle, the X and Y coordinates for the second circle's center point, and the second circle's radius.

10.9.7 F7 – Intersection: Line Line

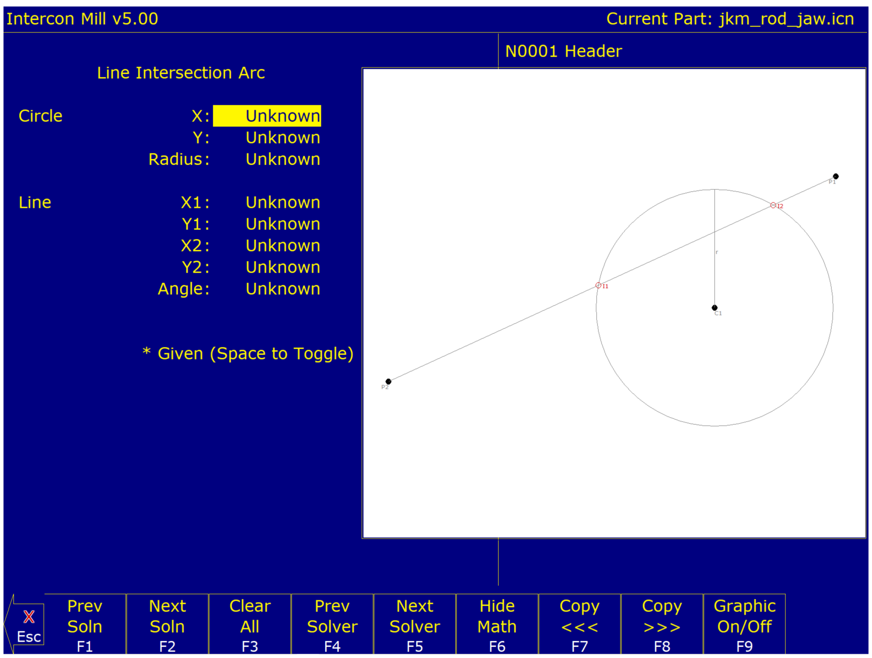


See button explanations above in the **F1 – Triangle: Right** section.

You must enter the X and Y coordinates for 1 point on each line, and also one of the following:

- The X and Y coordinates for a second point.
- The X coordinate for a second point and the angle from horizontal.
- The Y coordinate for a second point and the angle from horizontal.
- The angle from horizontal only.

10.9.8 F8 – Intersection: Line Arc



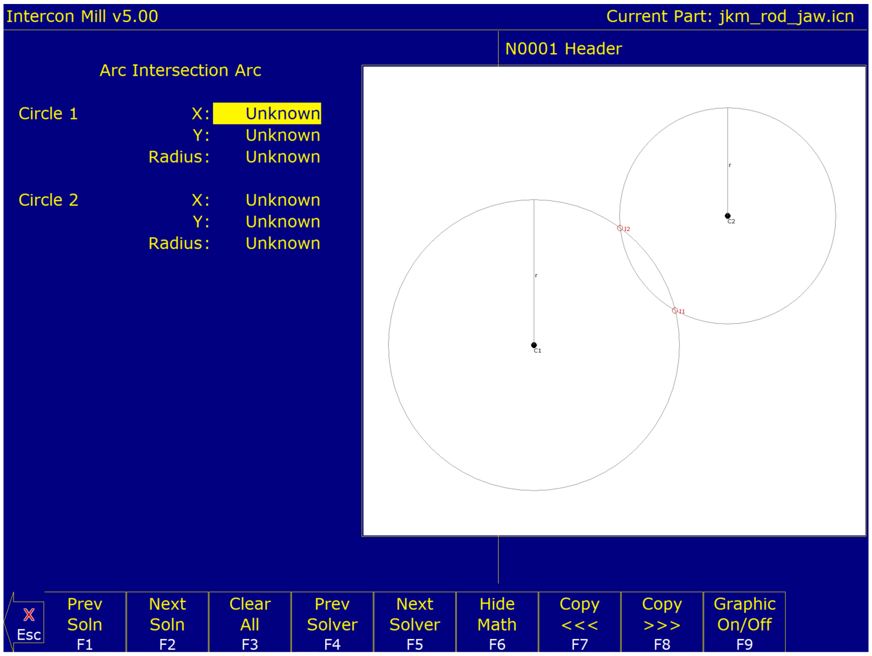
See button explanations above in the **F1 – Triangle: Right** section.

Given the center (C1) and radius (R) of an arc, 1 point (P1), and either a second point (P2) or one coordinate (P2 X or Y), and the angle from horizontal, find the intersection point(s) (I1 and I2).

You must enter the X and Y coordinates for the circle's center point, the circle's radius, the X and Y coordinates for one point on the line, and one of the following:

- The X and Y coordinates of a second point on the line.
- The X coordinate of a second point and the angle from horizontal.
- The Y coordinate of a second point and the angle from horizontal.

10.9.9 F9 – Intersection: Arc Arc



Draft: June 12, 2023

See button explanations above in the **F1 – Triangle: Right** section.

Given the center points (C1 and C2) and the radii (R1 and R2) of two arcs, find the intersection point(s) (I1 and I2) of the arcs.

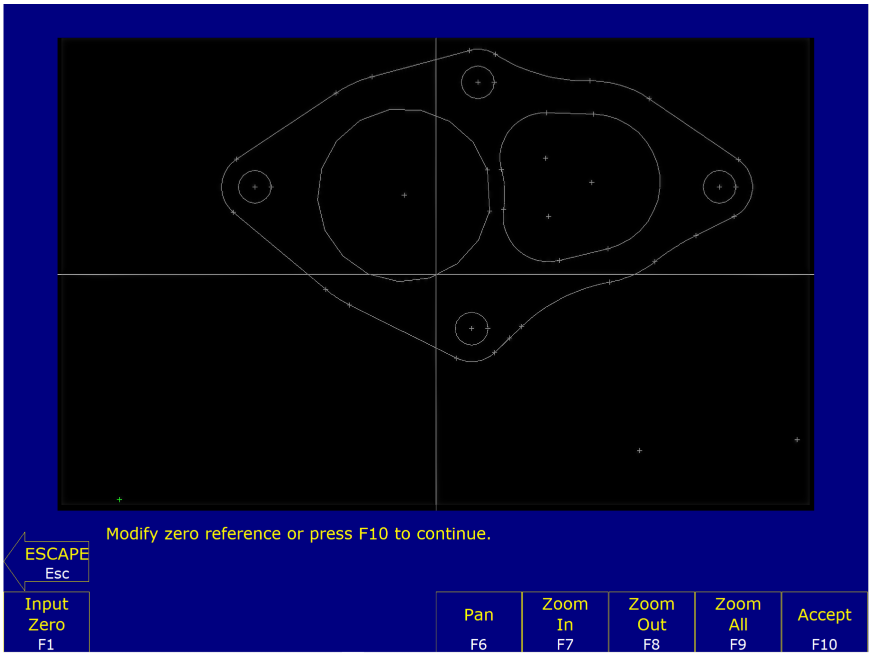
You must enter the X and Y coordinates for the first circle's center point, the radius of the first circle, the X and Y coordinates for the second circle's center point, and the second circle's radius.

10.10 Importing DXF files (Optional)

Intercon allows you to convert geometry in DXF files to Intercon operations. To insert operations from a DXF file press **F6 – Other** then **F8 – Import DXF**. If no DXF files have been loaded yet, the Intercon load file menu will appear. From here you may select the DXF file you wish to load. By default, Intercon expects DXF files to reside in the `c:\cncm\ncfiles` directory. To change the default directory used by the Intercon load menu when loading DXF files, see the "User Specified Paths" section of Chapter 15.

Intercon reads DXF files up to and including version R14. *At this time, only* ***point****,* ***line****,* ***arc****,* ***circle****,* ***polyline*** *and* ***lwpolyline*** *entities can be used to create Intercon operations*. All other entities such as *text* must be converted to lines and arcs for them to appear in the Intercon Import DXF menu.

After a DXF file has been loaded, the Set zero reference menu appears.

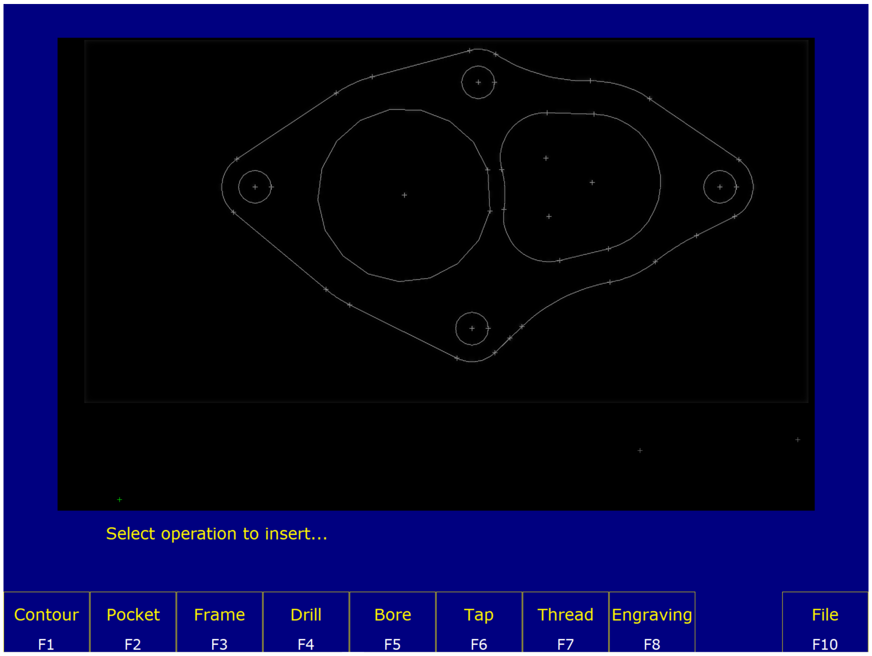


This menu allows you to change the absolute zero reference of the DXF file. The current zero reference appears as a green cross. All other points appear as gray crosses. A new zero reference may be defined by a combination of the following methods:

- Press **F1 – Input Zero** to enter the coordinates of the new zero reference.
- Move the crosshairs with the arrow keys to highlight a point (represented by a gray cross). When the crosshairs are close to a gray cross, they will become red. Press **F2 – Set Zero** to set the zero reference to the position of that point.

When satisfied with the current location of the zero reference, press **F10 – Accept**. The zero reference may be changed multiple times before pressing **F10 – Accept**.

After the zero reference is set, the Select Intercon operation menu appears:



This menu allows you to select the type of Intercon operation you wish to create using geometry from the DXF file.

F1 – Contour Convert One or more connected lines and/or arcs to linear and arc operations.

F2 – Pocket Convert A chain of lines and/or arcs to one of the Intercon pocket operations. The type of pocket depends on the geometry of the selected chain. A chain of arcs will be converted to a circular pocket if all arcs have the same center point and radius. A chain of four lines forming a rectangle will be converted to a rectangular pocket. All other chains will be converted into a cleanout operation.

F3 – Frame Create A frame operation that surrounds a chain of lines and/or arcs. The height, width and center of the chain are used to define the frame.

F4 – Drill Convert One or more points to a drilling operation.

F5 – Bore Convert One or more points to a boring operation.

F6 – Tap Convert One or more points to a tapping operation.

F7 – Thread Convert One or more points to a threading operation.

F8 – Engrave (Optional) Converts the entire file to engraving operations. Choosing this option displays the engraving menu. Set the options as desired for the engraving. Surface Height is the height of the surface to be engraved. Clearance Height is the height that the engraving tool will move up to clear the surface. Depth is the depth of the engraving. Set the Plunge Rate and Feedrate appropriately for the tool and material.

F10 – File Pressing F10 – File displays the following options:

F1 – Load Load a new DXF file.

F2 – Zero Change the current zero reference.

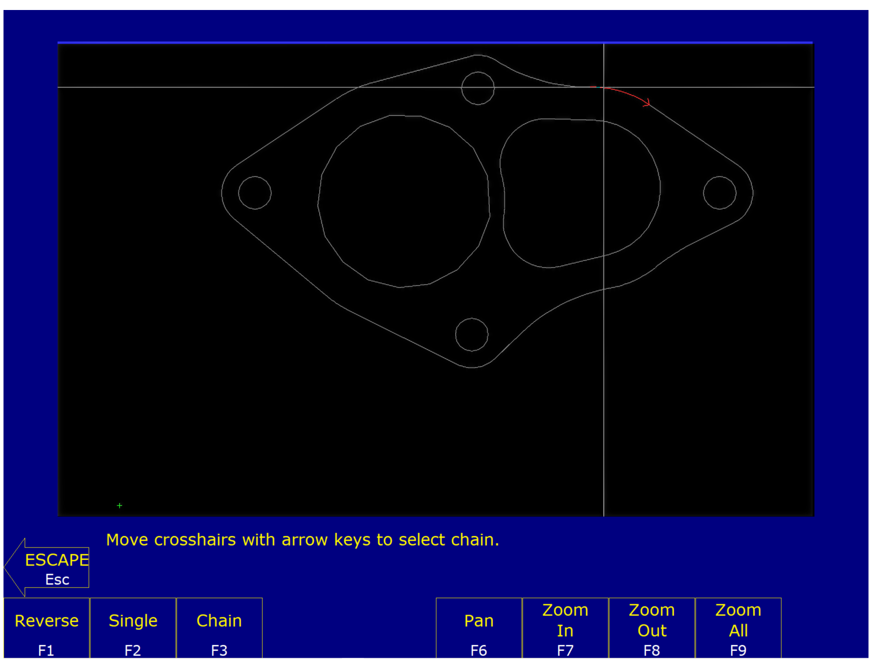
F3 – Gap Modify the current gap tolerance. Two lines or arcs are connected if the distance between their end points is less than the gap tolerance.

Selecting DXF geometry

After selecting an operation from the Select Intercon operation menu, one of two menus appears. Contour, pocket and frame operations display the Select Chain menu. Drill, Bore, Tap and Thread operations display the Select Point menu. These menus allow you to select the geometry you wish to use to create the specified Intercon operation(s).

Select Chain Menu

This menu allows you to select a chain of one or more lines and/or arcs. To select a chain, highlight the first line or arc with the crosshairs. Press either **F2 – Single** to accept only that object or press **F3 – Chain** to accept that object and create a chain of lines and arcs connected to that entity. When satisfied with the selected chain, press **F10 – Done** to edit the values for the chosen Intercon operation.



↑↓←→ (Arrow Keys) – Move crosshairs **Use the arrow keys to move the crosshairs.**

F1 – Reverse When a line or arc is highlighted, an arrow appears indicating the direction of the object. Contour operations use this direction when cutting lines and arcs. To reverse the direction of the highlighted line or arc, press **F1 – Reverse**.

F2 – Single Select the currently-highlighted line or arc.

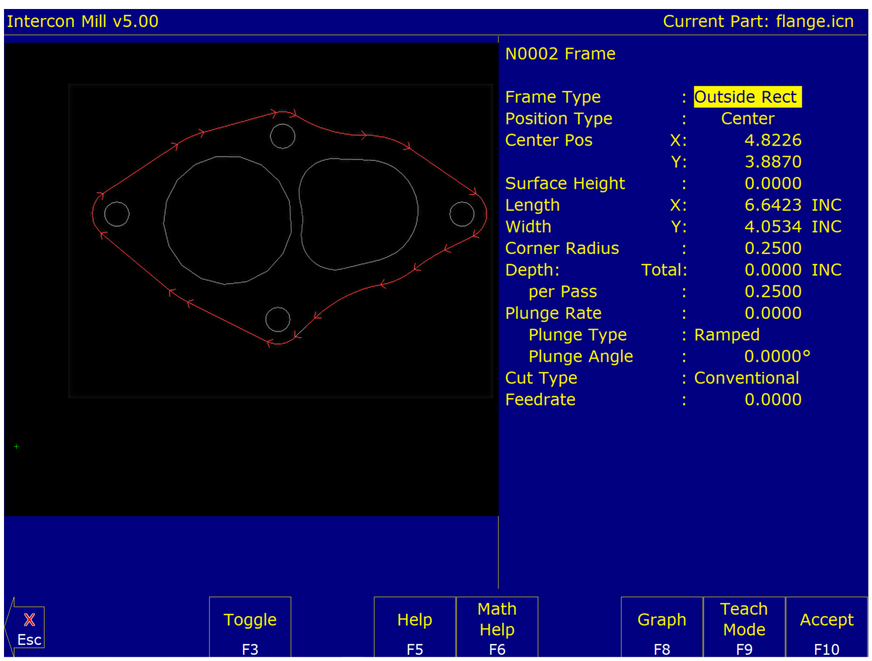
F3 – Chain Select the currently-highlighted line or arc and create a chain of connected lines or arcs in the direction of this selected object. Chaining will stop if there are no more connected and unselected lines or arcs, the chain connects to the first object in the chain, or a branch point occurs. A branch point occurs when the last object in the chain is connected to more than one unselected lines or arcs. When this situation occurs, highlight the desired unselected line or arc that connects to the last object in the chain, and press **F3 – Chain** to continue chaining.

F5 – Undo Unselect the last line or arc in the chain.

F6 – Pan Set the plot center to the center of the crosshairs.

F7, F8 & F9 – Zoom In, Zoom Out & Zoom All For **F7 – Zoom In** and **F8 – Zoom Out**, align the center of the plot to the center of the crosshairs, and Zoom In or Zoom Out respectively. **F9 – Zoom All** redraws the part with its original scale.

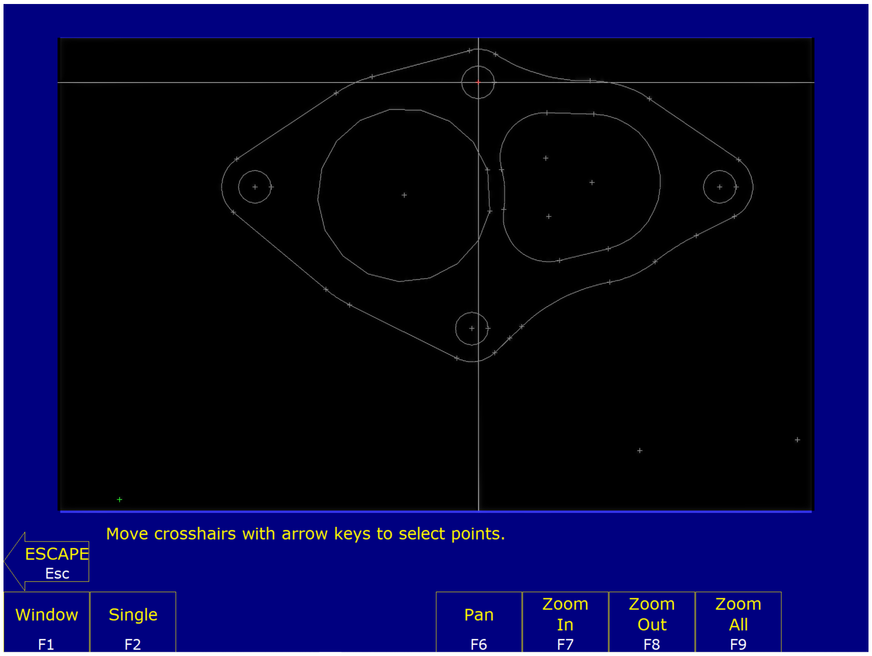
F10 – Done Accepts the selected chain and proceeds to the Intercon operation edit menu. The operation edit menu allows you to enter values for fields such as feedrate and plunge rate. These values are copied to the rest of the lines and arcs in the chain where applicable.



The keys **F1 – Reverse**, **F2 – Single**, and **F3 – Chain** only appear when a line or arc is highlighted. The keys **F5 – Undo** and **F10 – Done** only appear when one or more objects have been selected with **F2 – Single** or **F3 – Chain**.

Select Point Menu

This menu allows you to select one or more points to be converted into Intercon Drilling Threading operations. In this menu, all selectable points are displayed as gray crosses. Selectable points include point entities, line/arc endpoints and arc/circle center points. To select a point, position the crosshairs over the desired point until the cross turns red and press **F2 – Single** to accept the point. More points can be selected by highlighting them with the crosshairs and pressing **F2 – Single**. **F1 – Window** can be used to select all points within a specified window. When satisfied with the selected point(s), press **F10 – Done** to edit the Intercon operation parameters.



↑↓←→ (Arrow Keys) – Move crosshairs **Use the arrow keys to move the crosshairs.**

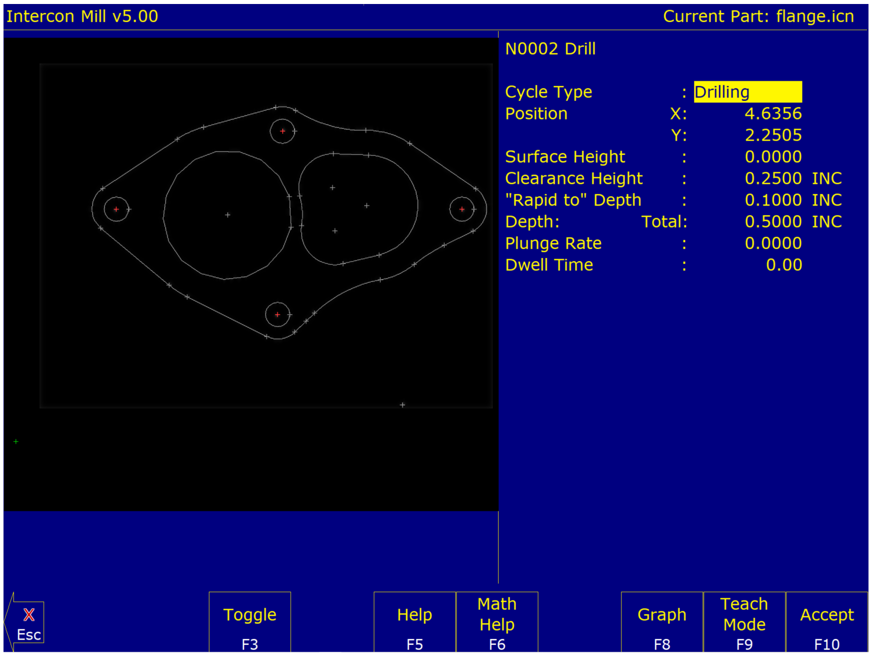
F1 – Window This key allows you to select all points within a specified box. Press F1 – Window once to set the first corner of the box. Move the crosshairs to the desired location for the opposite corner of the box and press F1 – Window. All points within the box are selected.

F2 – Single Accept the currently highlighted point.

F5 – Undo Unselect the last selected point.

F6, F7, F8 & F9 – Pan, Zoom In, Zoom Out & Zoom All These keys perform the same operations they do in the Select Chain menu.

F10 – Done Accepts the selected points and proceeds to the Intercon operation edit menu. The operation edit menu allows you to enter values for fields such as plunge rate and depth. These values are copied to the rest of the selected operations where applicable.



The key **F2 – Single** only appears when a point is highlighted. The keys **F5 – Undo** and **F10 – Done** only appear when one or more points have been selected with F1 – Window or **F2 – Single**.

Using a mouse

In addition to the arrow keys, a mouse may be used to position the crosshairs in the DXF selection menus. Simply move the mouse pointer to the desired crosshair location and click the left mouse button. This action will move the crosshairs to the location of the mouse click and highlight the closest object.

10.11 Intercon Tutorial #1

This is a step-by-step instructional example of going from blueprint to part with Intercon. The tool path to be created is for the part shown in Figure 1. For instructional purposes, this part will be programmed to cut into stock held in 3 fixtures, 6 inches apart along the X-axis.

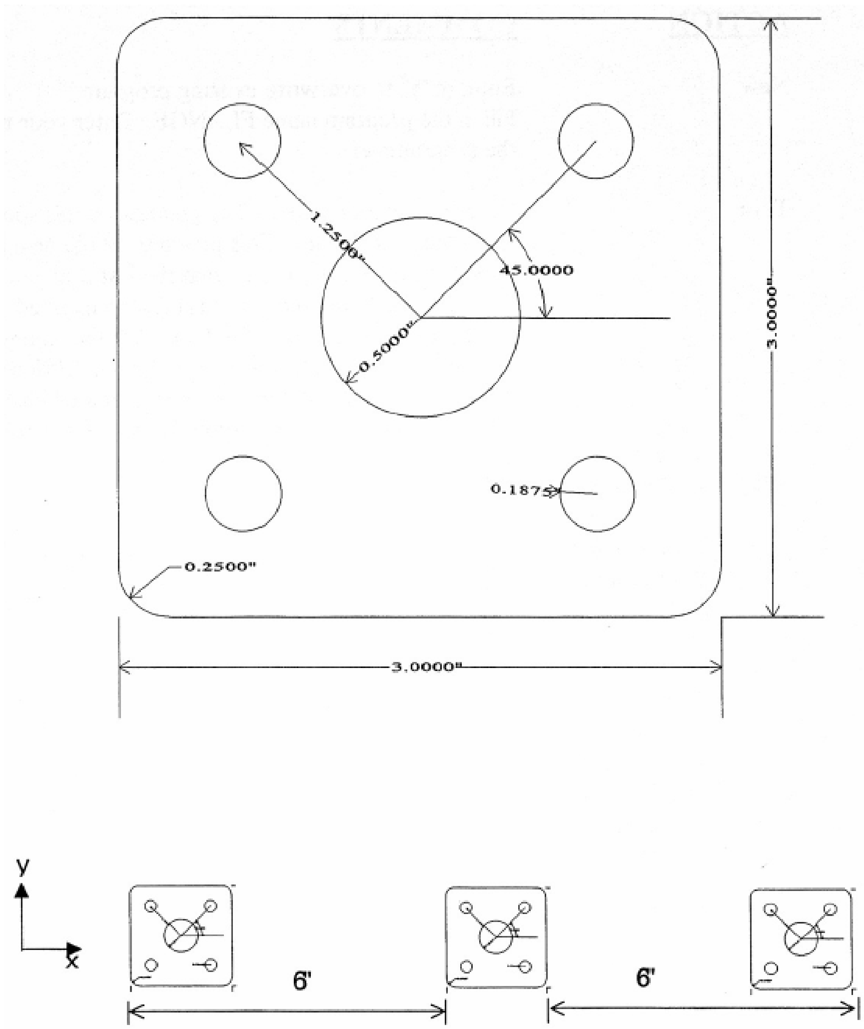


FIG. 1 Blueprint of flange part and the 3 fixtures.

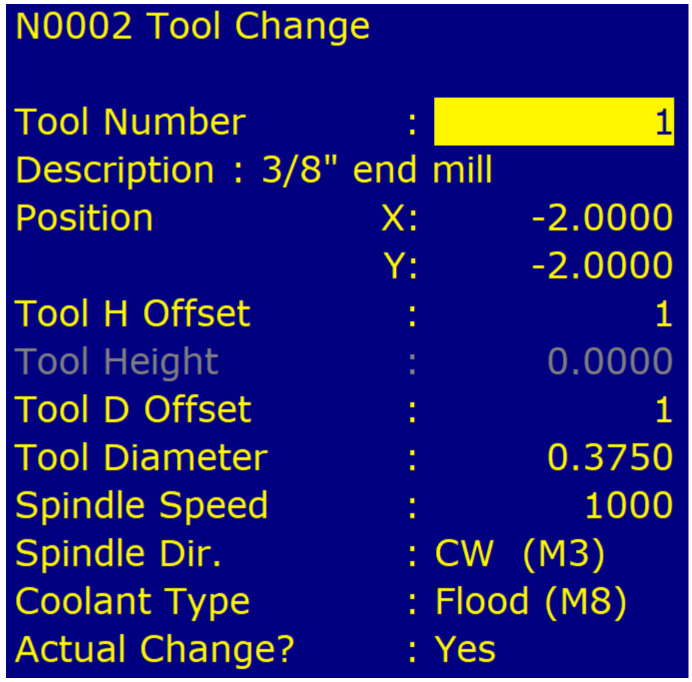
10.11.1 Part Creation

Each feature of the part will become an operation in your program. Before beginning, decide where you want the **X0** and **Y0** reference. For this particular part, the center of the bolt hole pattern was selected. Now start the Intercon program (from the **CNC software main screen**, press **F5 – CAM**). Beginning from the Intercon File Menu (press **F1 – File** if the file menu is not shown) the following series of keystrokes will describe the step-by-step process of designing the part shown in Figure 1.

PRESS	COMMENTS
F1 – New	Fill in the program name flange. Enter your name in as the programmer. Enter the description as “Intercon Tutorial #1”.

F4 – Tool

Describe the tool below. The position values specify where to do the tool change. This position should be a point outside of the workpiece so that the last tool can be removed from the chuck and the new tool can be inserted. The Yes in the 'Actual Tool Change' field turns off the spindle and coolant upon reaching this spot. Use a 0.3750-inch diameter cutter. The length and diameter are updated based on the offsets. (The longest tool should have a 0.0000 length).



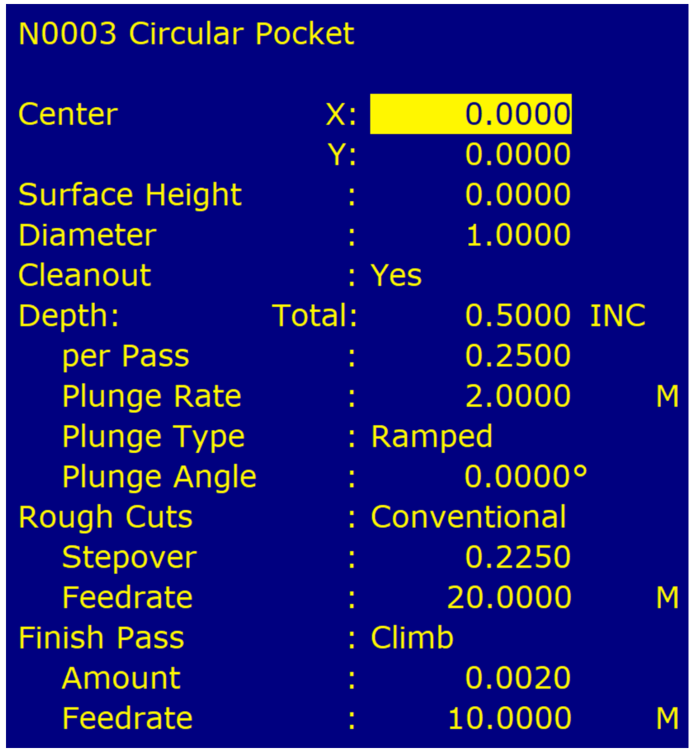
F10 – Accept

Keep selected values.

F5 – Cycles

Access the list of available Canned Cycles.

F6 – Circ. Pocket Start with the 1.0000-inch diameter circular pocket. Enter the following values:

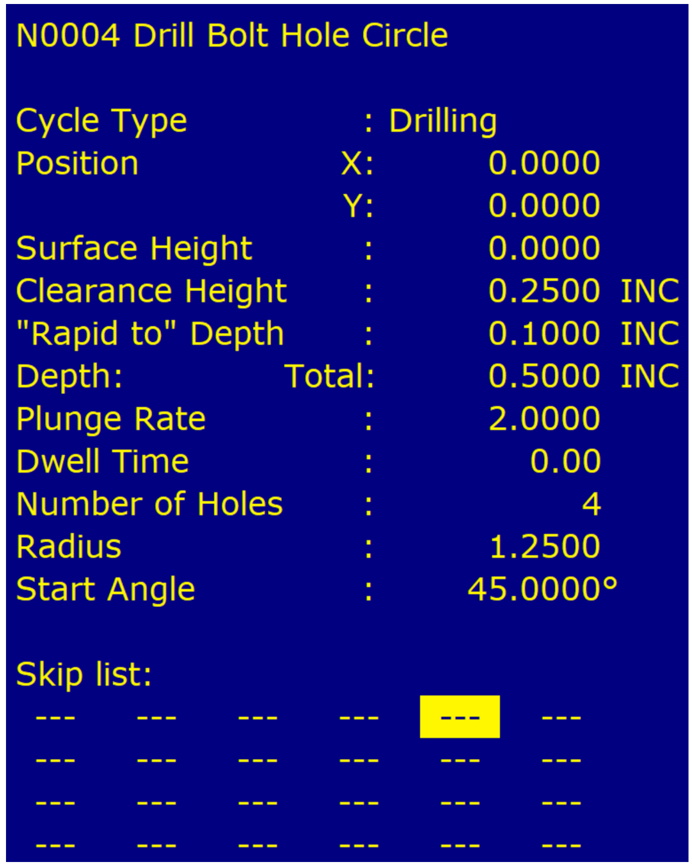


F10 – Accept Keep selected values.

F5 – Cycles Access the list of available Canned Cycles.

F1 – Drill Select drilling cycles

F2 – Drill BHC Select the bolt hole circle type of drilling cycles:



F8 – Graph Display a preview of the part up to this point. This preview can be used to detect problems that may occur if the part was cut now.

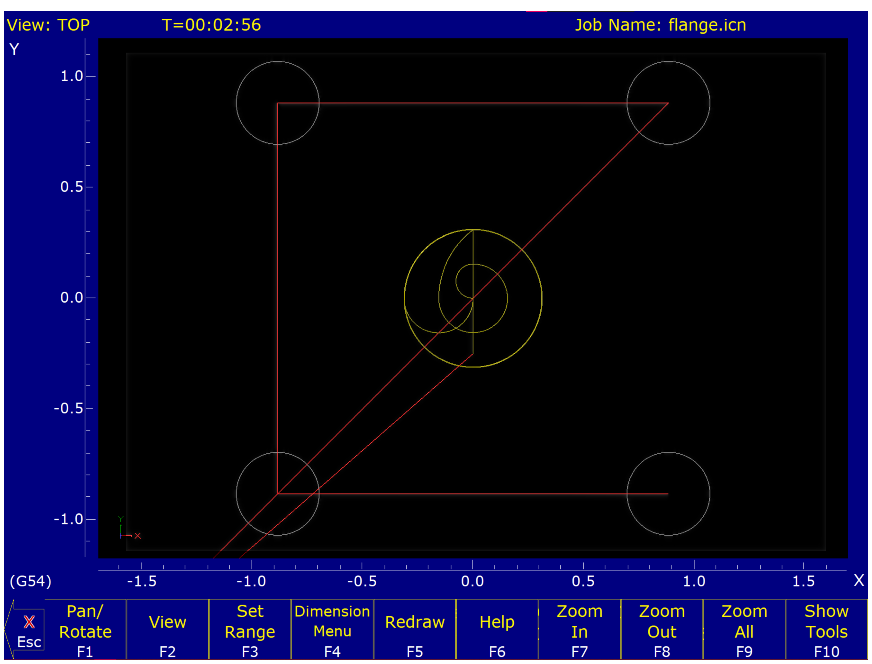


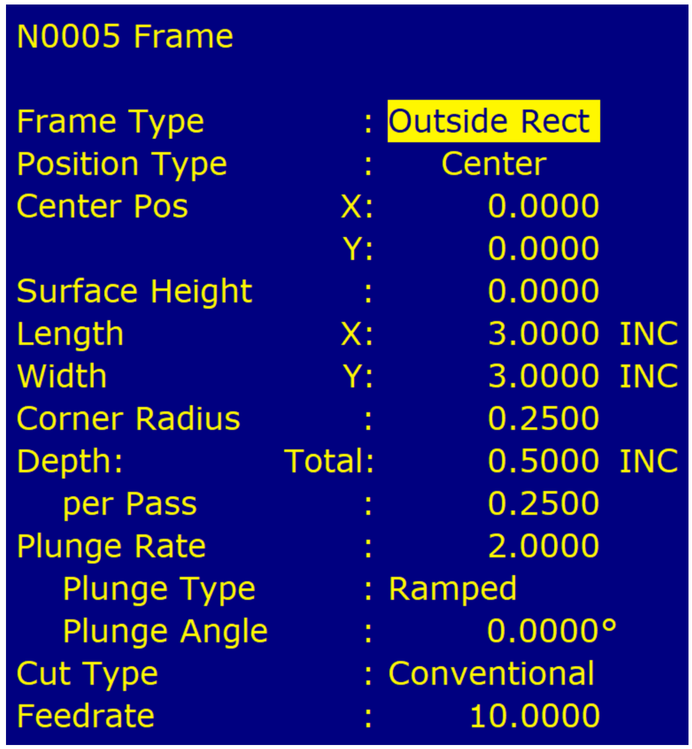
FIG. 2 – Graphics screen showing bolt holes and circular pocket

ESC/CANCEL Return to the editing screen.

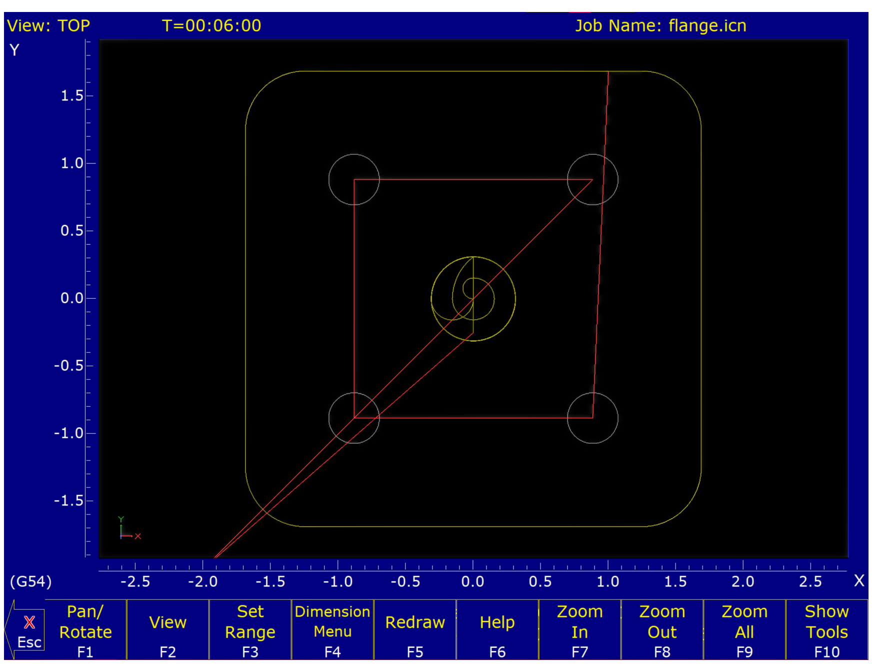
F10 – Accept Keep selected values.

F5 – Cycles Access the list of available Canned Cycles.

F7 – Frame Now add an outside frame to cut the flange out of the material. The flange is 3.0000 inches long by 3.0000 inches wide, and has rounded corners with 0.2500-inch radii.



F8 – Graph Display a preview of the part up to this point. This preview can be used to detect problems that may occur if the part was cut now.

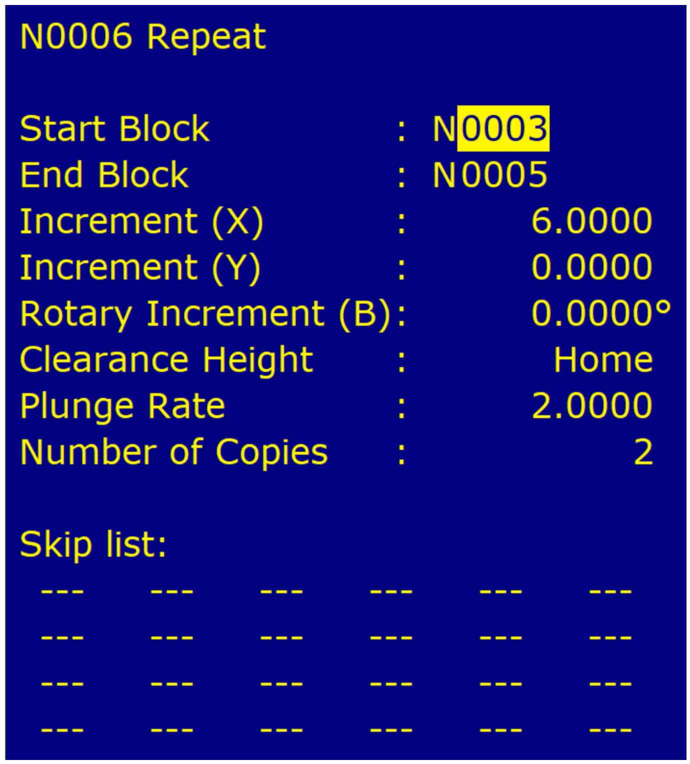


ESC/CANCEL Return to the editing screen.

F10 – Accept Keep selected values.

F9 – Subpgm Access the Insert Subprogram screen.

F2 – Repeat We programmed the part to cut one copy only. We now want to repeat the part 2 more times at an incremental distance of 6 inches along the X-axis. The part can now be cut into the stock mounted into the two other fixtures. The part begins with the circular pocket in operation **#0003** and ends with the linear mill in operation **#0005**. Press **F2 – Z Home** to enter "Home" for "Clearance Height".



F8 – Graph Display a preview of the parts. This preview can be used to detect problems that may occur if the part was cut now.

ESC/CANCEL Return to Repeat Subprogram.

F10 – Accept Keep selected values if you wish to cut these two extra parts. If you do not wish to do this, press **ESC/CANCEL**.

ESC/CANCEL Creation of the part is complete. Intercon programs automatically turn the spindle and coolant off at the end.

F1 – File Press **F3 – Save** to save the part under its current name. Press **F4 – Save As** to save it under a new name.

F10 – Post As it processes each operation, it checks for values that, if used, will cause incorrect code to be produced. If such a value is found, a message will appear on the screen alerting you of the problem. For example, a problem with a Frame Mill may produce this message:

Corner radius must be equal to or greater than tool radius

Changes to the part would then be required to allow proper code generation to proceed. If no problems are encountered during code generation, the following message appears:

G-code generation successful

Program Finished! You are now finished designing your part. In order to run your part, you now need to return to the CNC software.

10.11.2 Milling The Part

Now that the part has been programmed, it is time to mill it. Take your material and clamp it to the table. Remember that the clamps must be positioned such that they do not interfere with the tool as it cuts. You may choose either to place the clamps around the edges of the material for the entire process and let the part drop through upon completion, or you may wish to pause after milling the circular pockets and place clamps through the holes to prevent the part from moving. The second option decreases the chance of the part being marred because it moved during milling. Now you need to set your XYZ reference points. Insert your longest tool in the quill and follow the procedure listed below:

PRESS	COMMENTS
JOG KEYS	Jog the table so that your tool rests on the stock at the location that will represent X0 and Y0.
F1 – Setup	Enter the CNC software Setup screen. We are going to establish the part XYZ zero at the current tool location.
F1 – Part	Access the Part Setup options.
F10 – Set	Set your X zero position at current tool location.
F1 – Next	Axis Select the Y-axis next.
F10 – Set	Set your Y zero position at current tool location.
F1 – Next	Axis Select the Z-axis next.
F10 – Set	Set your Z zero position at current tool location.
TOOL CHECK	Moves the quill to the Z home position if the home position has been set. Moves tool to Z+ limit switch and sets home position if not.
ESC/CANCEL	Leave Part Setup screen.
F2 – Tool	Access Tool Library Editor. This is the place where we want to measure the actual heights of our tools (since we could not set the actual values in Intercon).

F1 – Offset Lib. You need to make sure that the tool diameter and height offset values are the correct ones for the tools you are going to be using. Inspect the values for D001 and H001. D001 should be 0.375, H1 should be 0.0000 (the two inch tool). If either of these values are incorrect, use the arrow keys to select the incorrect values. Enter the new values in their places and press ENTER to accept them.

Note: The tool heights used above are merely example heights. In order to accurately measure the heights of your tools, see the description of measuring tool heights in Chapter 5.

F10 – Save Keep the updated tool offset library values.

F2 – Tool Lib. Now you need to make sure that each tool uses the correct diameter and height offset values. Inspect the values for T001. T1 should use H001 and D001. If any of these values are incorrect, use the arrow keys to select the incorrect values. Enter the new values in their places and press **ENTER** to accept them. You may also select spindle and coolant settings for your tools here, or enter a short description of the tool.

F10 – Save Keep the updated Tool Library values.

ESC/CANCEL Leave Tool Setup. Return to the CNC software Setup Screen.

ESC/CANCEL Leave CNC software Setup. Return to the CNC software Main Screen.

CYCLE START The **CYCLE START** button is located on your jog panel. This key will cause the mill to begin cutting your part.

Tutorial Complete!

10.12 Intercon Tutorial #2

This demonstration will show you how to create a tool path for a part from a blueprint using the Math Help function of Intercon. The tool path to be created is for the part shown in Figure 1 below:

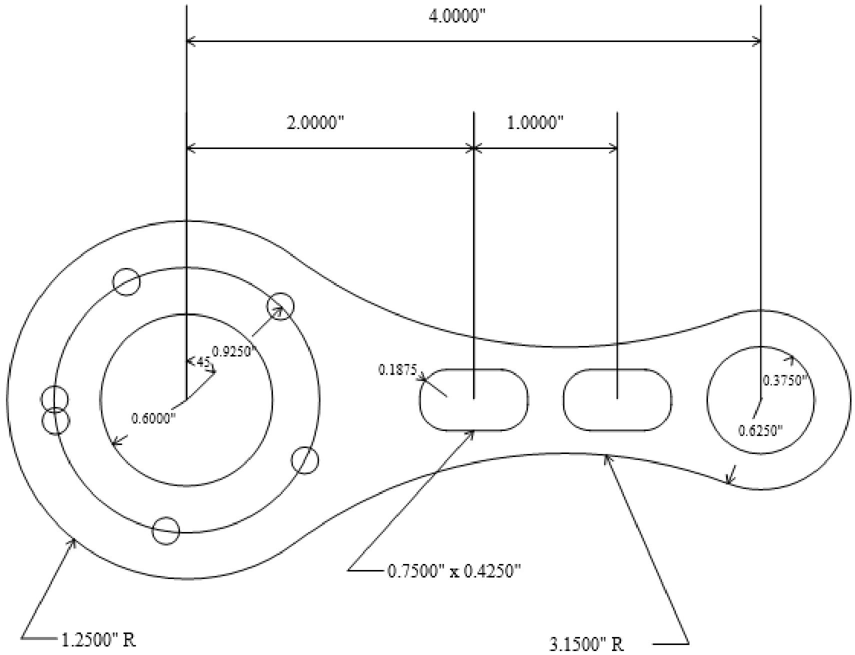


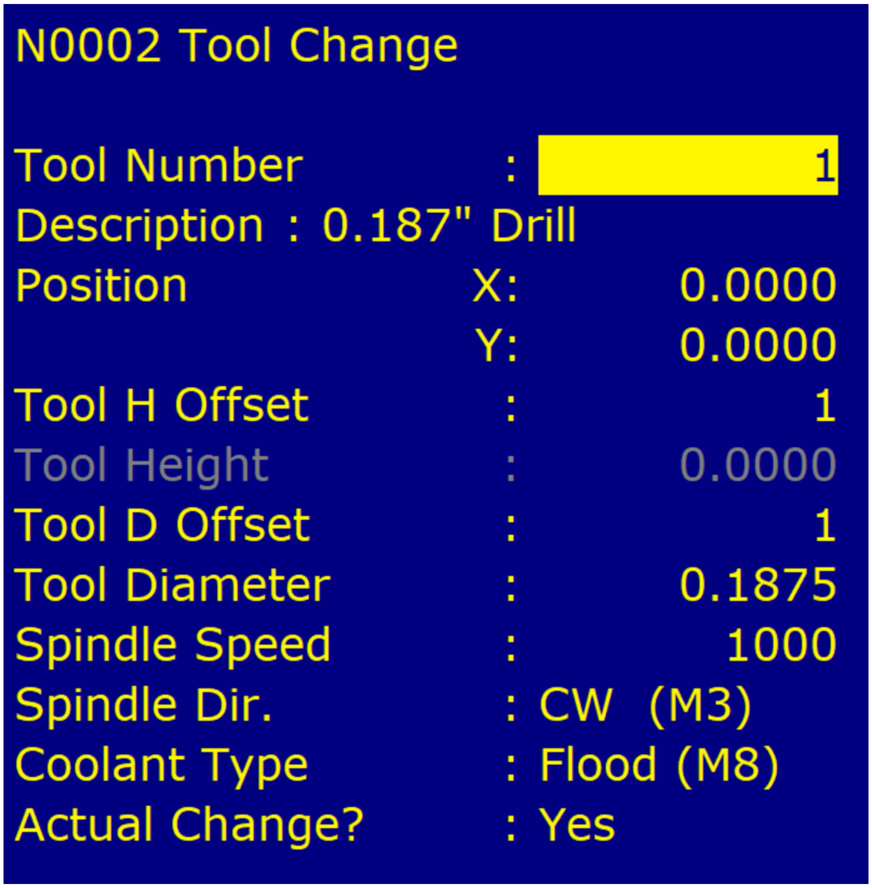
Figure 1 – Part to be machined.

10.12.1 Part Creation

The process of creating a part is called part programming. Each feature of the part will become an operation in your program. Before beginning, decide where you want the **X0** and **Y0** reference. For this particular demo, the center of the Bolt Hole pattern was selected for convenience). Beginning from the Intercon File Menu (press **F1 – File** if the file menu is not shown) the following series of keystrokes will describe the step-by-step process of designing the part shown in Figure 1.

PRESS	COMMENTS
F1 – New	Create a new program by filling in the appropriate program name (we recommend *c_rod*) and your name. Press Enter or **F10 – Accept** to accept the new name. Enter “Intercon Tutorial #2” for the description. Press **F10 – Accept** to accept.
F4 – Tool	Describe the tool below. The position values specify where to do the tool change. The Yes in the ‘Actual Tool Change’ field turns off the spindle and coolant upon reaching this spot. Use a 0.1875 inch drill. The height and diameter are updated based on the offsets. (The longest tool should have a 0.0000 height offset). If this tool does not have the desired spindle (CW) and coolant (Flood) settings, you should also select these values to match your particular machine setup.

N0002 Tool Change



Notice for this particular screen, the Tool height shows ‘0.0000’, since it has the same tool height as the Reference tool. However, your screen may differ since Intercon cannot change the Reference tool height in the Tool Library. This will change when you run this program. Refer to the Measuring Tool Heights in Chapter 5 for more details.

F10 – Accept	Keep selected values.
F5 – Cycles	Access the list of available Canned Cycles.
F1 – Drill	Select drilling cycles
F2 – Drill BHC	Select a bolt hole circle operation.

The clearance height is the Z height from which the downward rapid traverse begins before each hole. It is also the Z height to which the tool returns upon completion of drilling the hole.

The ‘Rapid To’ depth is the Z height to which the tool rapid traverses before drilling a hole.

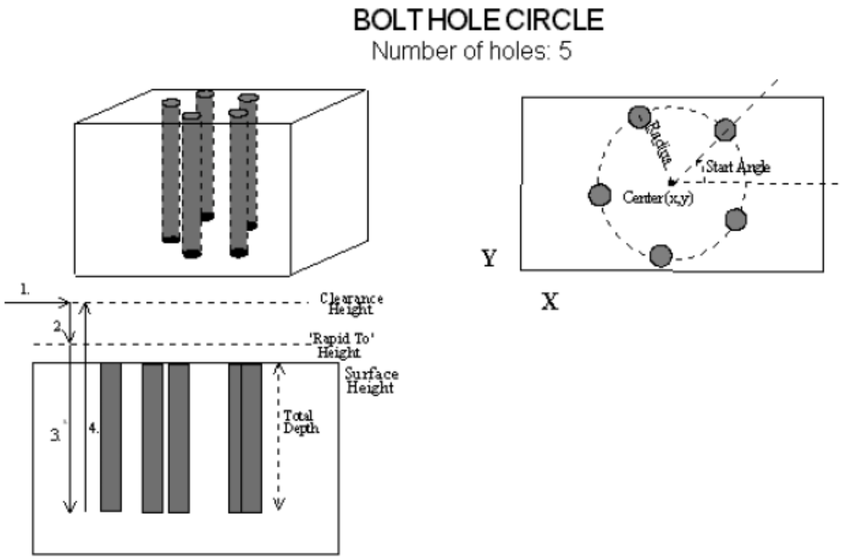
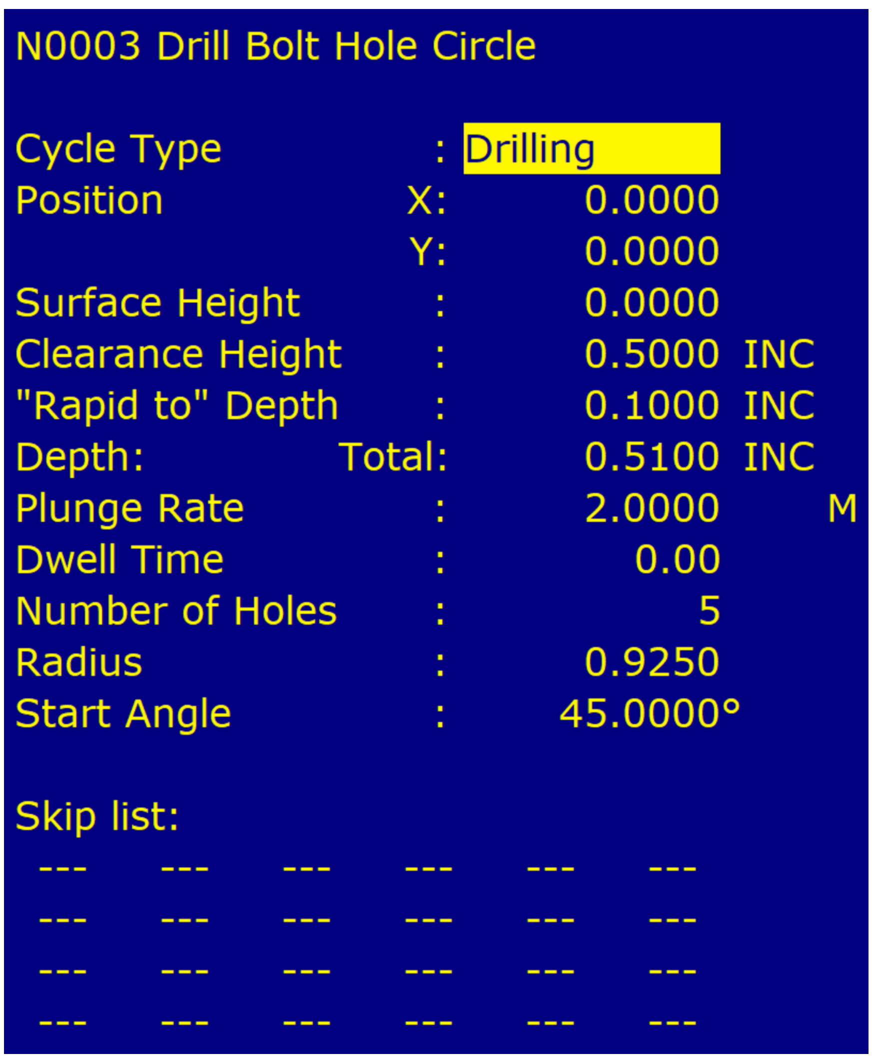


Figure 2 – Bolt Hole Circle

N0003 Drill Bolt Holes



N0003 Drill Bolt Hole Circle

Cycle Type	:	Drilling	
Position	X:	0.0000	
	Y:	0.0000	
Surface Height	:	0.0000	
Clearance Height	:	0.5000	INC
"Rapid to" Depth	:	0.1000	INC
Depth:	Total:	0.5100	INC
Plunge Rate	:	2.0000	M
Dwell Time	:	0.00	
Number of Holes	:	5	
Radius	:	0.9250	
Start Angle	:	45.0000°	

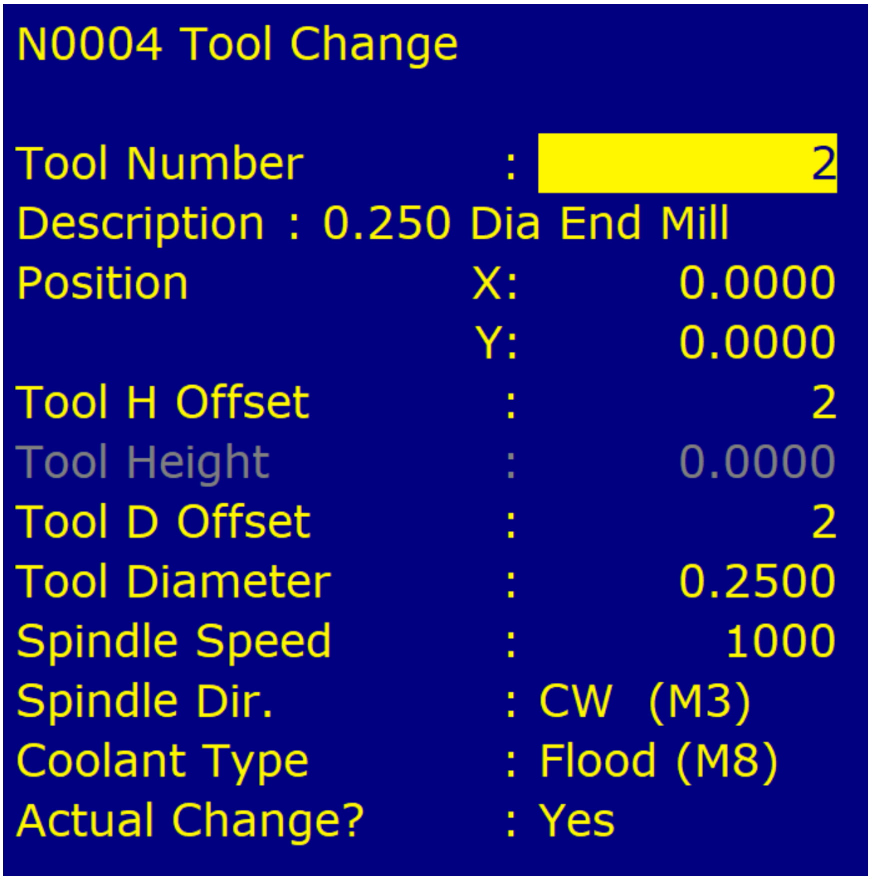
Skip list:

---	---	---	---	---	---
---	---	---	---	---	---
---	---	---	---	---	---
---	---	---	---	---	---

F10 – Accept Keep selected values.

F4 – Tool Use a 0.2500 diameter end mill now. Notice that the tool height shown below is a negative value. This value represents the difference in height between this tool and the longest tool being used. The longest tool used (in this case, operation N0020 above) has a height of 0.0000. Again, do not be alarmed if the Tool Height is not -1 for operation N0040. If this tool does not have the desired spindle (CW) and coolant (Flood) settings, you should also enter values specific to your machine setup.

N0004 Tool Change

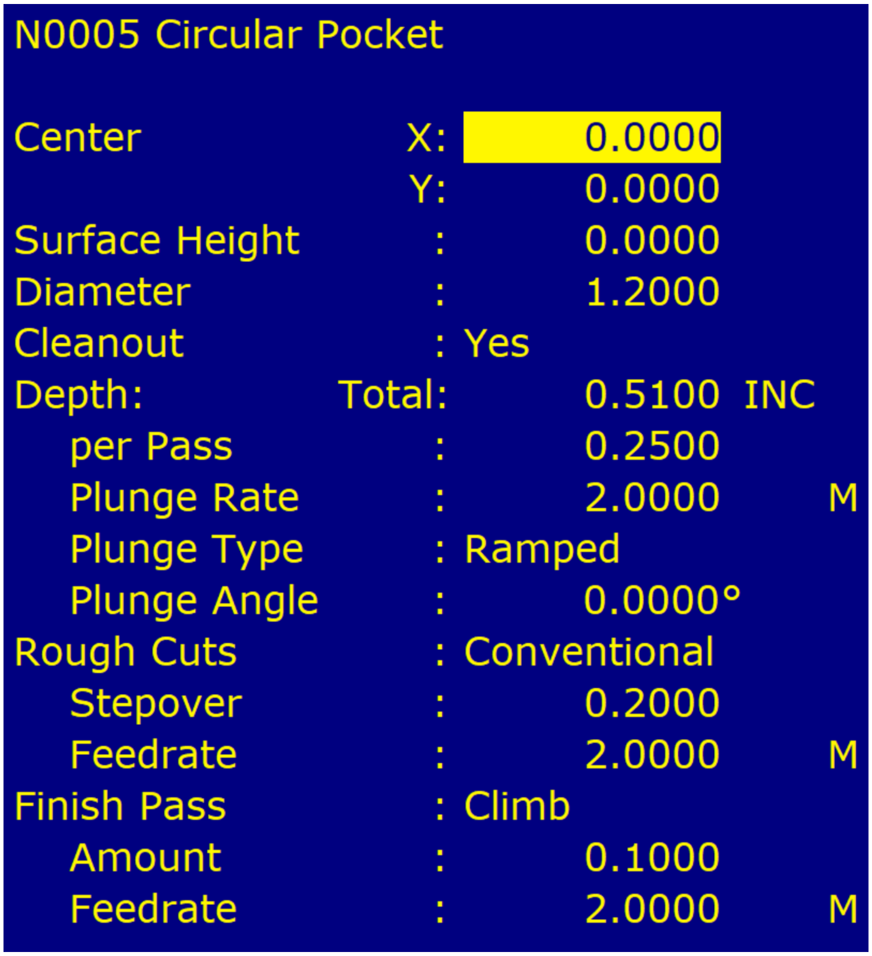


F10 – Accept Keep selected values.

F5 – Cycles Access the list of available canned cycles.

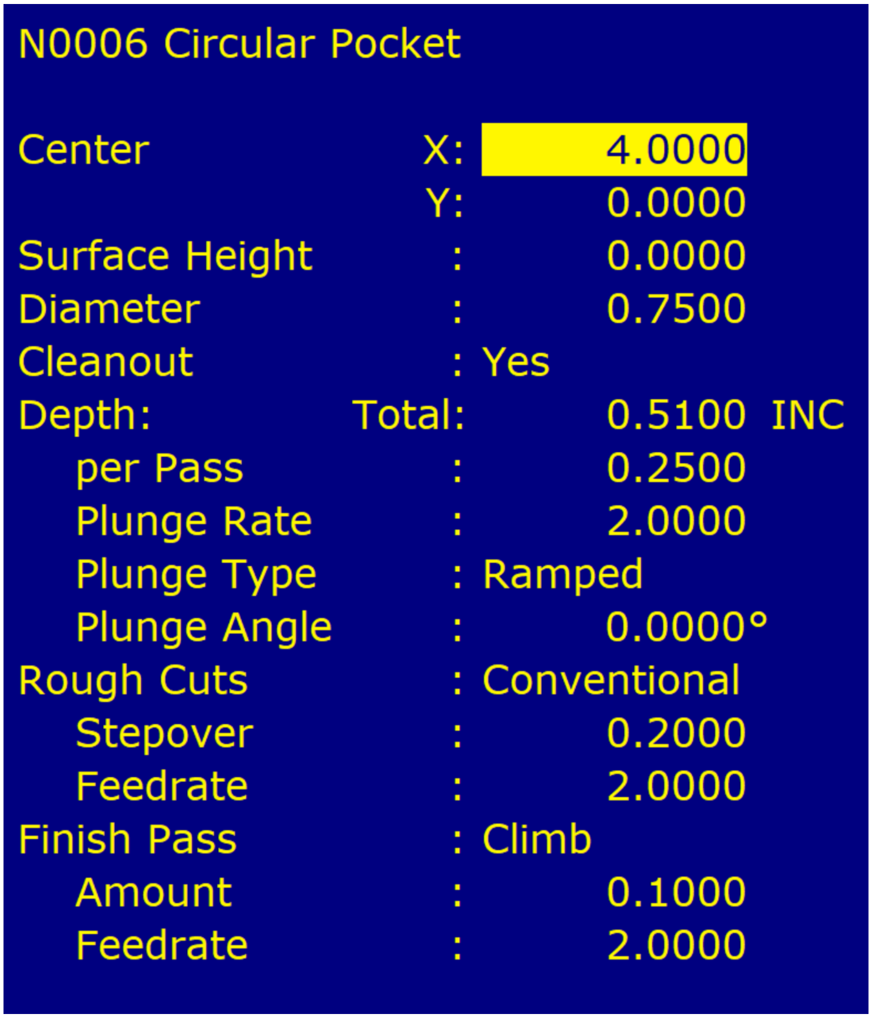
F6 – Circ. Pocket Start with 1.2000 inch diameter Pocket.

N0005 Circular Pocket



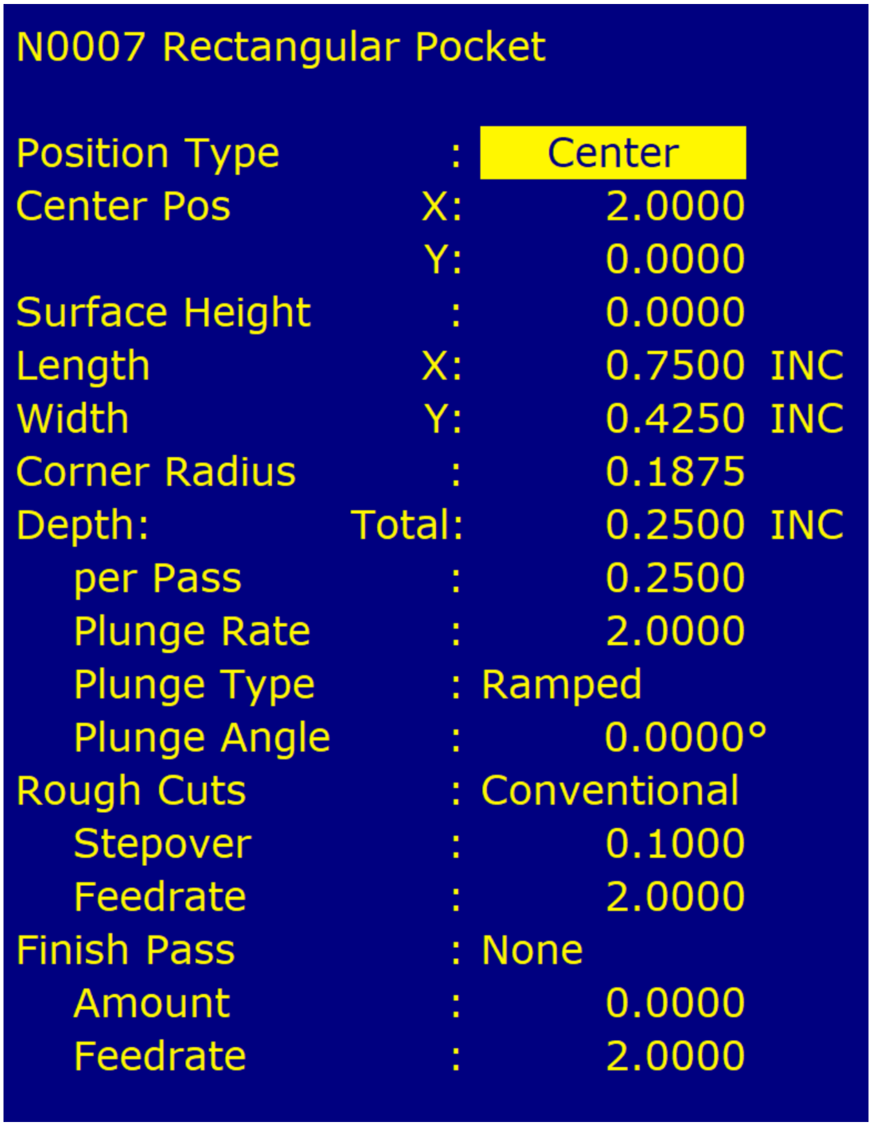
F10 – Accept	Keep selected values.
F5 – Cycles	Access the list of available canned cycles.
F6 – Circ. Pocket	Repeat above pocket cycle. The center X value 4.0000 and the diameter is 0.7500 inches.

N0006 Circular Pocket



F10 – Accept	Keep selected values.
F5 – Cycles	Access the list of available Canned Cycles.
F5 – Rect. Pocket	Cut the first rectangular pocket.

N0007 Rectangular Pocket



F10 – Accept	Keep selected values.
F5 – Cycles	Access the list of available Canned Cycles.
F5 – Rect. Pocket	Repeat above Pocket cycle. The center X value lies at 3.0000.

N0008 Rectangular Pocket

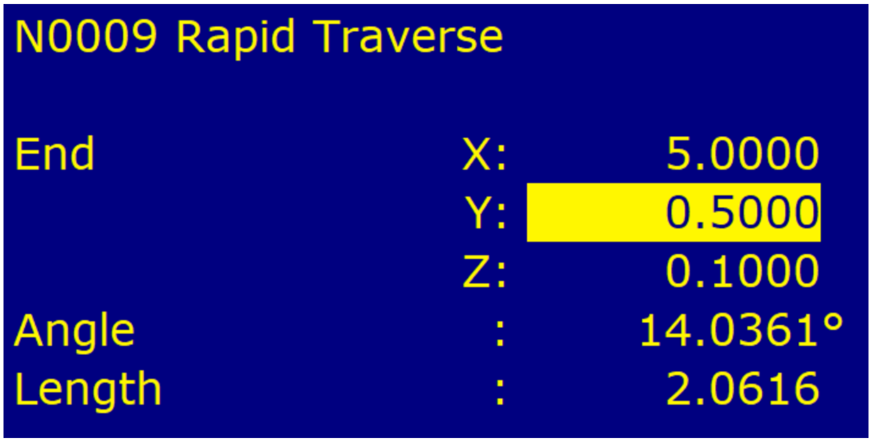
N0008 Rectangular Pocket

Position Type	:	Center	
Center Pos	X:	3.0000	
	Y:	0.0000	
Surface Height	:	0.0000	
Length	X:	0.7500	INC
Width	Y:	0.4250	INC
Corner Radius	:	0.1875	
Depth:	Total:	0.2500	INC
per Pass	:	0.2500	
Plunge Rate	:	2.0000	
Plunge Type	:	Ramped	
Plunge Angle	:	0.0000°	
Rough Cuts	:	Conventional	
Stepover	:	0.1000	
Feedrate	:	2.0000	
Finish Pass	:	None	
Amount	:	0.0000	
Feedrate	:	2.0000	

F10 – Accept Keep selected values.

F1 – Rapid Move to a location outside the part. The purpose of this move is to prepare to use cutter compensation on the tool.

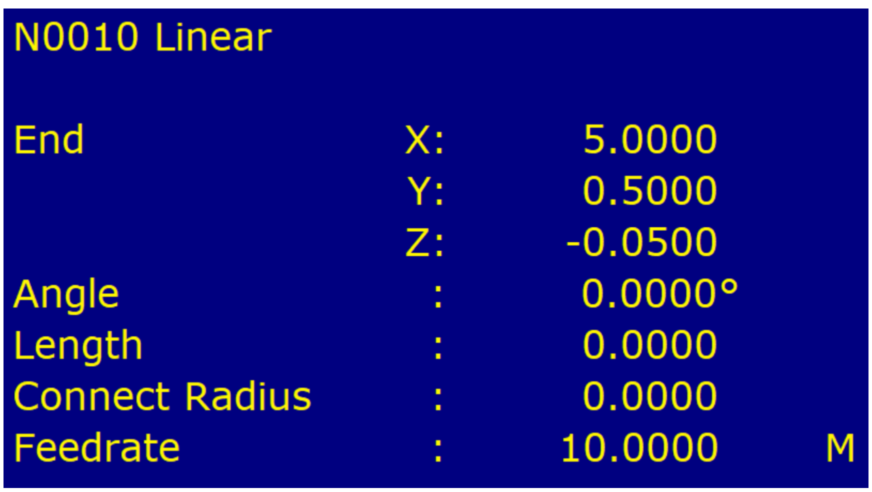
N0009 Rapid Traverse



F10 – Accept Keep selected values.

F2 – Linear Need to do a zero length move to move the cutter down to zdepth before starting the lead in move to the part. Not including the zero length move to z-depth would confuse cutter comp. and cause the program to do weird things.

N0010 Linear Move



F10 – Accept Keep selected values.

F7 – Cutter Comp Hit **Space** until Left cutter compensation is selected. The tool must move outside of the part outline at a distance at least equal to its radius so the part outline is the correct size.

N0011 Comp Left

F10 – Accept Keep selected values.

F3 – Arc Mill up to the edge of the part to cut the first arc. This is called a lead-in move. The cutter compensation selected above needs a lead-in move in order to position the cutter before milling the actual part.

N0012 Arc

You will see that after you enter in these values, the other points and arcs will be entered in automatically.

F10 – Accept Keep selected values.

F3 – Arc The first arc to be cut is labeled as ARC 1 in Figure 3 below. The start point, labeled P1, is the end point of the previous move. The end point of the arc will be generated with Math Help. We will be using end point and radius (EP&R) arcs.

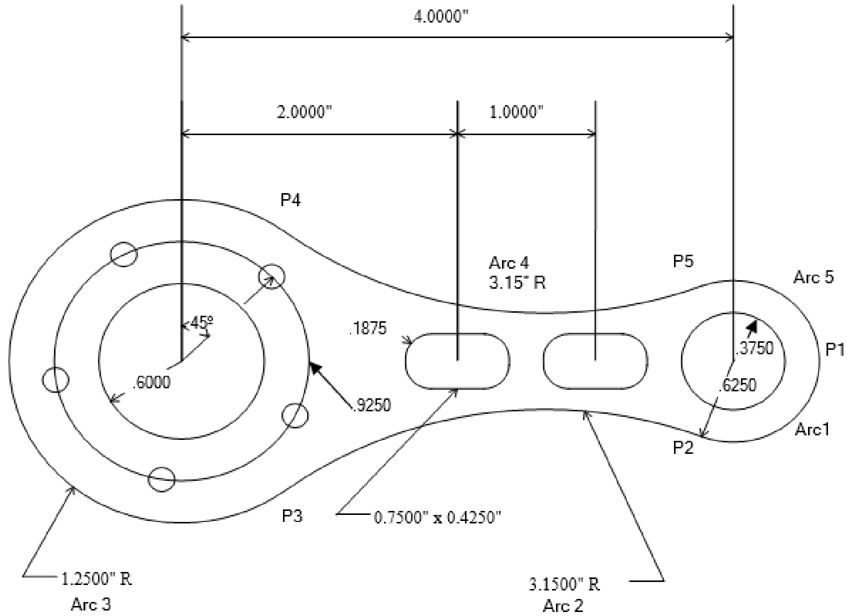
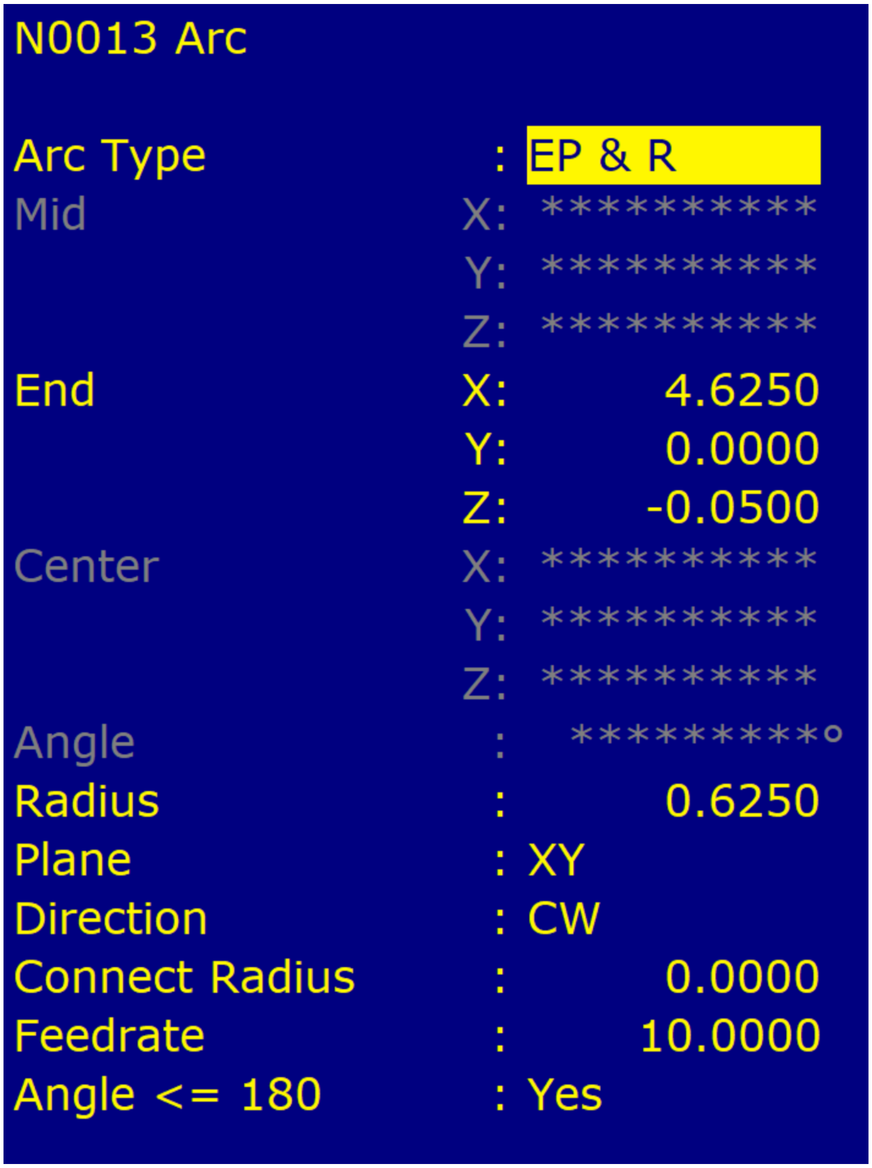


Figure 3 – Tangent point and arc reference.

N0013 Arc

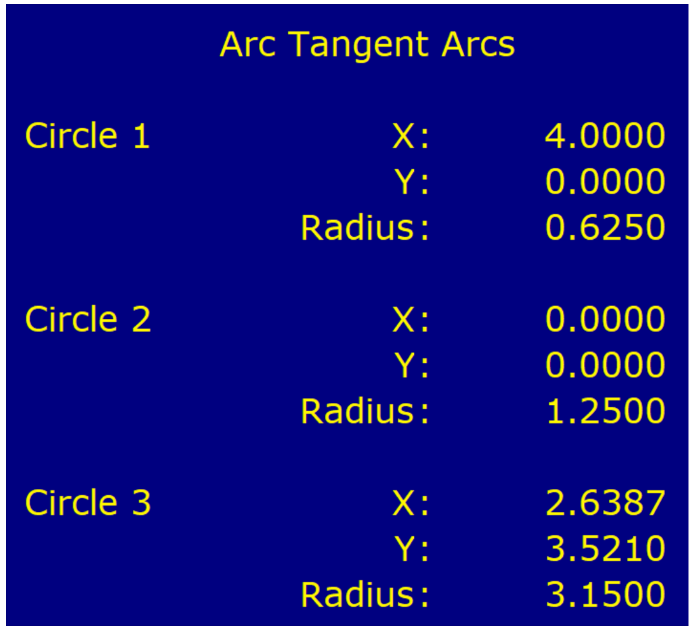


F6 – Math Help We are trying to find end points for the arcs that make up the outside edge of the part. **Note:** the main Math Help menu will list all available Math Help solvers.

F6 – Tangent Arc Arc Arc

This scenario will generate tangent points P2 – P5 of Figure 3. Enter the values as shown below:

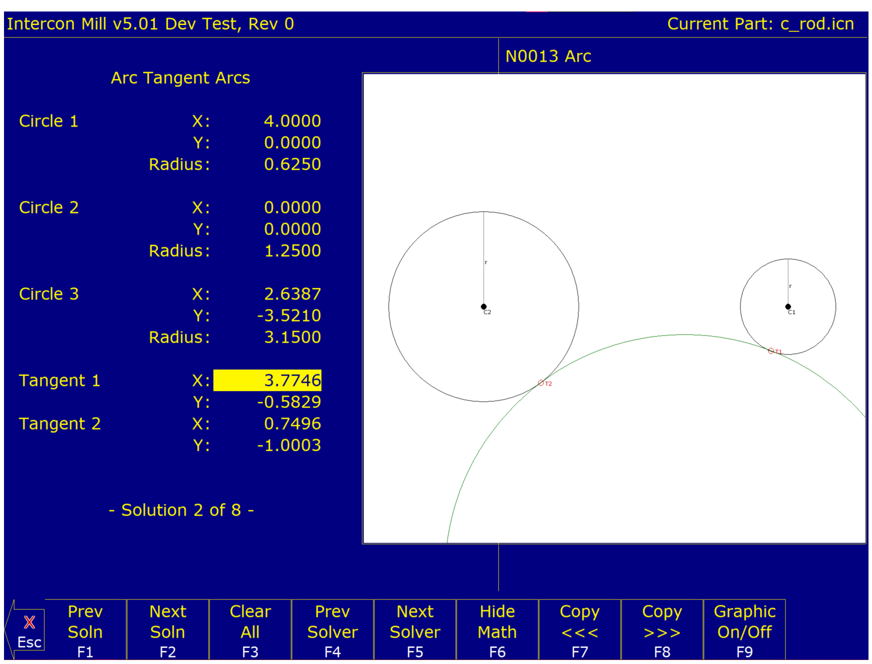
Arc Tangent Arcs:



Intercon will calculate the missing values for this scenario.

F2 – Next Soln

Find scenario that corresponds to the actual arcs being milled. Observe Figure 4. Point T1 is the one needed.



ARROWS If necessary, move the block cursor to the Tangent 1 X field as shown above.
Note: Use only ↑ and ↓. If you press the right arrow, press the left arrow to get back to the Math Help fields.

F9 – Graphic on/off Press to hide the graphical display and reveal the arc operation behind it.

→ (ARROW) Move the cursor to the arc operation. The solid block cursor on the left side of the screen will be replaced by an outlined rectangle and the solid block will appear in the arc operation on the right.

ARROWS Move the block cursor to the End X field of the arc operation. As before, use only ↑ and ↓.

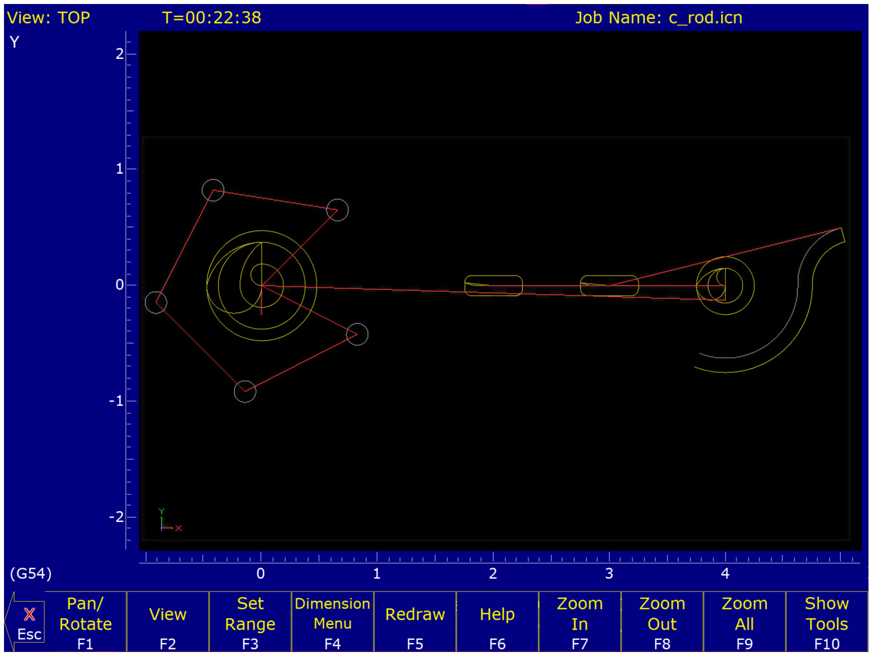
F8 – Copy >>> Transfer the tangent point T1 value for X into the end point X coordinate. The active fields on both sides of the screen advance automatically.

F8 – Copy >>> Transfer the tangent point T1 value for Y into the end point Y coordinate.

ARROWS Move down to the radius field and enter the radius of the arc labeled as ARC 1 in Figure 3. (This radius is 0.6250 in.).

F6 – Hide Math Hide Math Help temporarily. (We will return later to pick up the other tangent points.)

F8 – Graph Observe Figure 5. The graphics show a preview of the part up to this point. This preview can be used to detect problems that may occur if the part was cut now.



ESC/CANCEL Return to the editing screen.

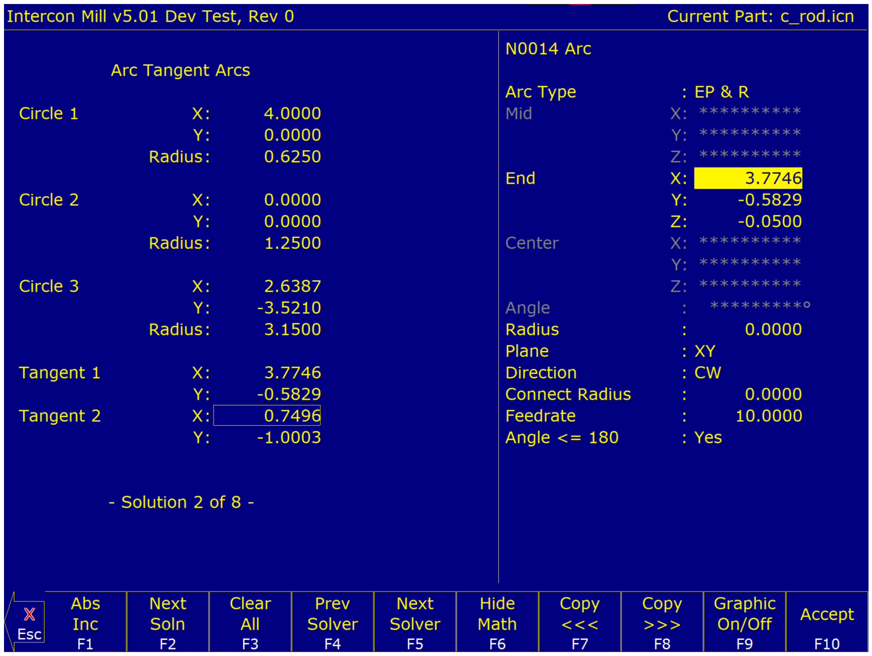
F10 – Accept Keep selected values. The other arc values were calculated for you.

F3 – Arc The next arc to be cut is labeled as ARC 2 in Figure 3. The start point is labeled P2, the end point of the last arc.

N0014 Arc

↑↓ (UP/DOWN) Move down to the End X field. This selects End X as the destination of the Math Help solution.

F6 – Math Help Re-display the Math Help values calculated for the last arc. The screen will look like figure 6, below.



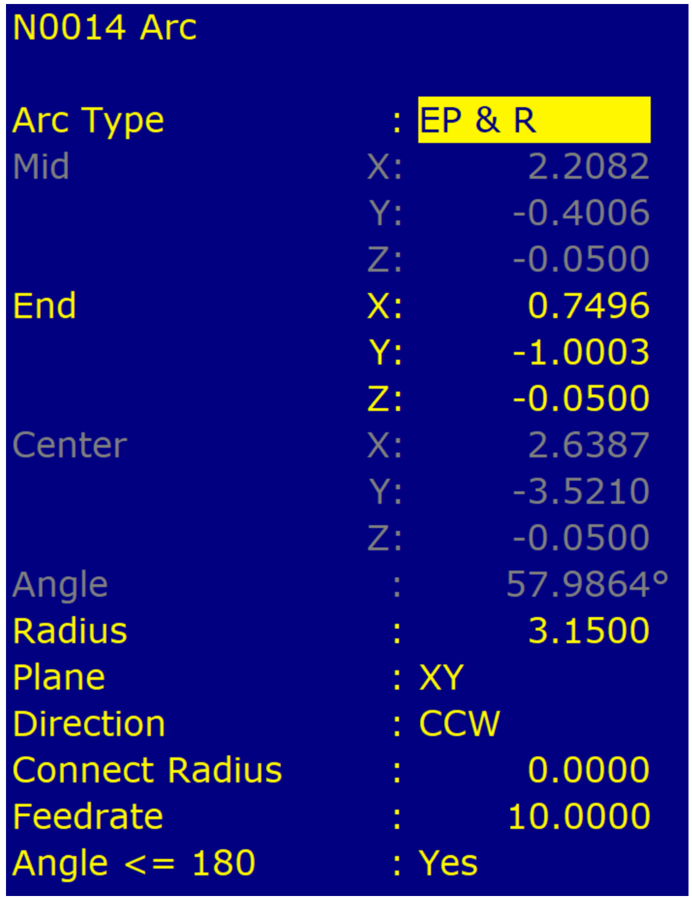
↑↓ (UP/DOWN) If necessary, move the block cursor to the Tangent 2 X field as shown above. The rectangle at End X shows that it will be the destination of the copy.

F8 – Copy >>> Transfer the tangent point T2 value for X into the end point X coordinate. The active fields on both sides of the screen advance automatically.

F8 – Copy >>> Transfer the tangent point T2 value for Y into the end point Y coordinate.

F6 – Hide Math Hide Math Help temporarily.

↑↓ **(UP/DOWN)** Move down to the radius field and enter the radius of the arc labeled ARC 2 in Figure 3 (this radius is 3.1500 inches). Set the direction to “CCW”.



F10 – Accept Keep selected values.

F3 – Arc The third arc to be cut is labeled as ARC 3 in Figure 3. The start point is labeled P2, the end point of the previous arc. The end point of the arc will be generated with Math Help.

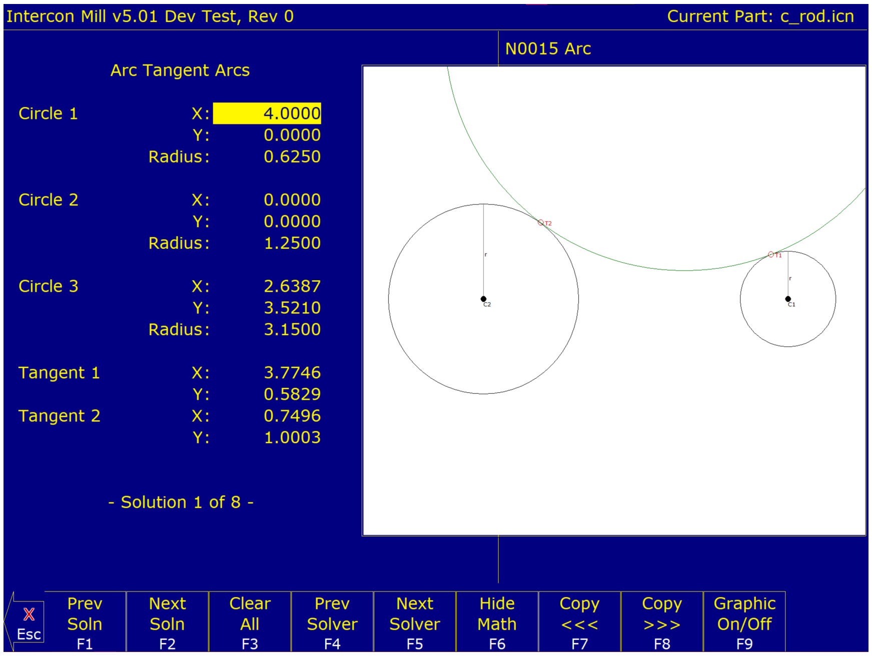
N0015 Arc

↑↓ **(UP/DOWN)** Move down to the End X field. This selects End X as the destination of the Math Help solution.

F6 – Math Help Re-display the Math Help values calculated for the last arcs.

F9 – Graphic on/off Re-display the diagram of the scenario selected to calculate arcs 1 and 2 on Figure 3.

F1 – Prev Soln Continue pressing F1 until you arrive at the scenario showing arcs 3 and 4 in Figure 3 (in this case, solution #1 is the appropriate one)



↑↓ (ARROWS) Press to highlight the needed tangent point X coordinate in Math Help. Tangent point T2 is the one you want this time.

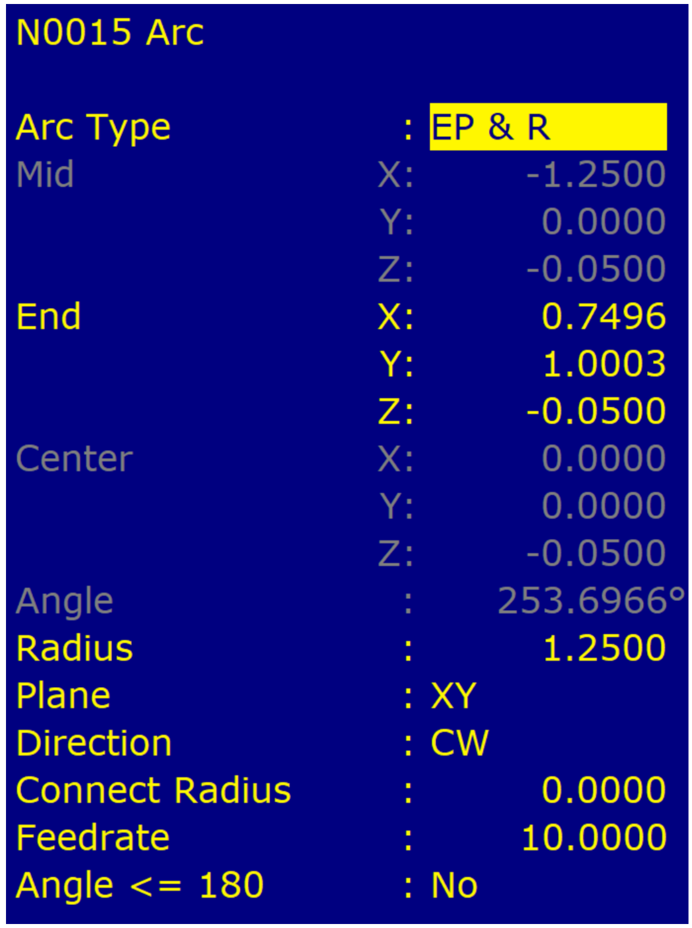
← (ARROW) Press to remove the graphic display and move the cursor to the arc operation. (This shortcut saves you from pressing **F9 – Graphic on/off** to hide the graphics each time.) The solid block cursor on the left side of the screen will be replaced by an outlined rectangle and the solid block will appear in the arc operation on the right.

↑↓ (ARROWS) Move the block cursor to the End X field of the arc operation.

F8 – Copy >>> Transfer the tangent point T2 value for X into the end point X coordinate. The active fields on both sides of the screen advance automatically.

F8 – Copy >>> Transfer the tangent point T2 value for Y into the end point Y coordinate.

↑↓ (ARROWS) Move down to the radius field and enter the radius of the arc labeled ARC 3 in Figure 3. (This radius is 1.2500 inches). Also, enter in No for the angle of this arc, since it is greater than 180°.



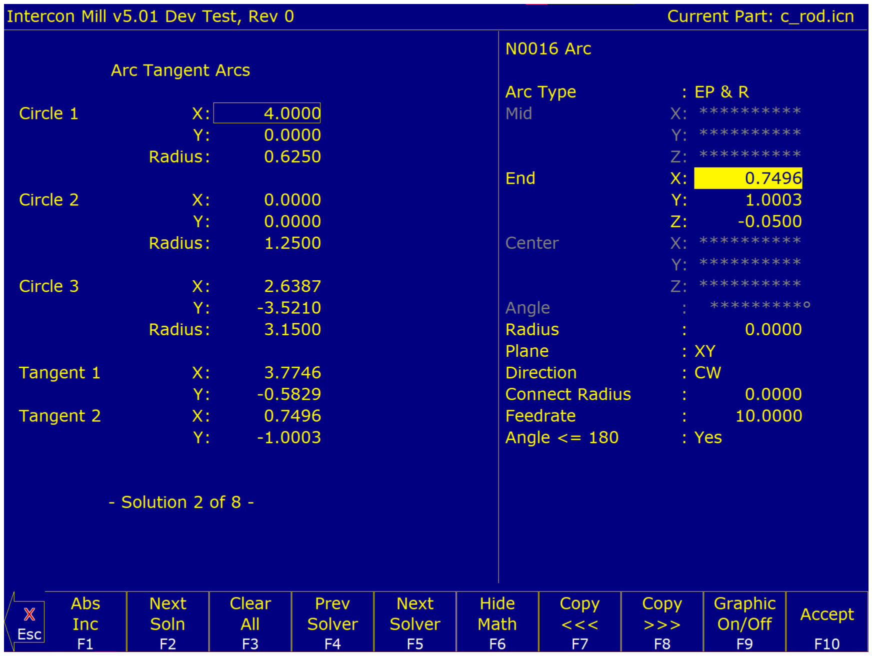
F10 – Accept Keep selected values.

F3 – Arc The fourth arc to be cut is labeled as ARC 4 in Figure 3. The start point, labeled P3, is the end point of the previous arc. The end point of the arc will be generated with Math Help.

N0016 Arc Mill

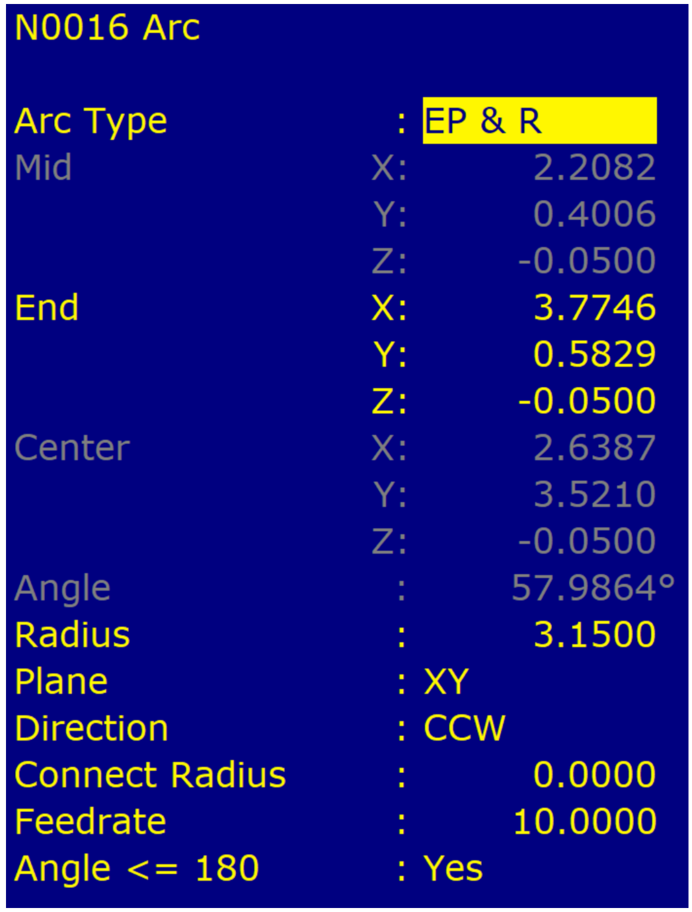
↑↓ (ARROWS) Move down to the End X field. This selects End X as the destination of the Math Help solution.

F6 – Math Help Re-display the Math Help values calculated for the last arc.



↑↓ (UP/DOWN)	Highlight the needed tangent point X. Tangent point T1 is the one you want this time.
ARROWS	If necessary, move the cursor to the arc operation and select the End X field.
F8 – Copy >>>	Transfer the tangent point T1 value for X into the end point X coordinate. The active fields on both sides of the screen advance automatically.
F8 – Copy >>>	Transfer the tangent point T1 value for Y into the end point Y coordinate.
F6 – Hide Math	Hide Math Help.

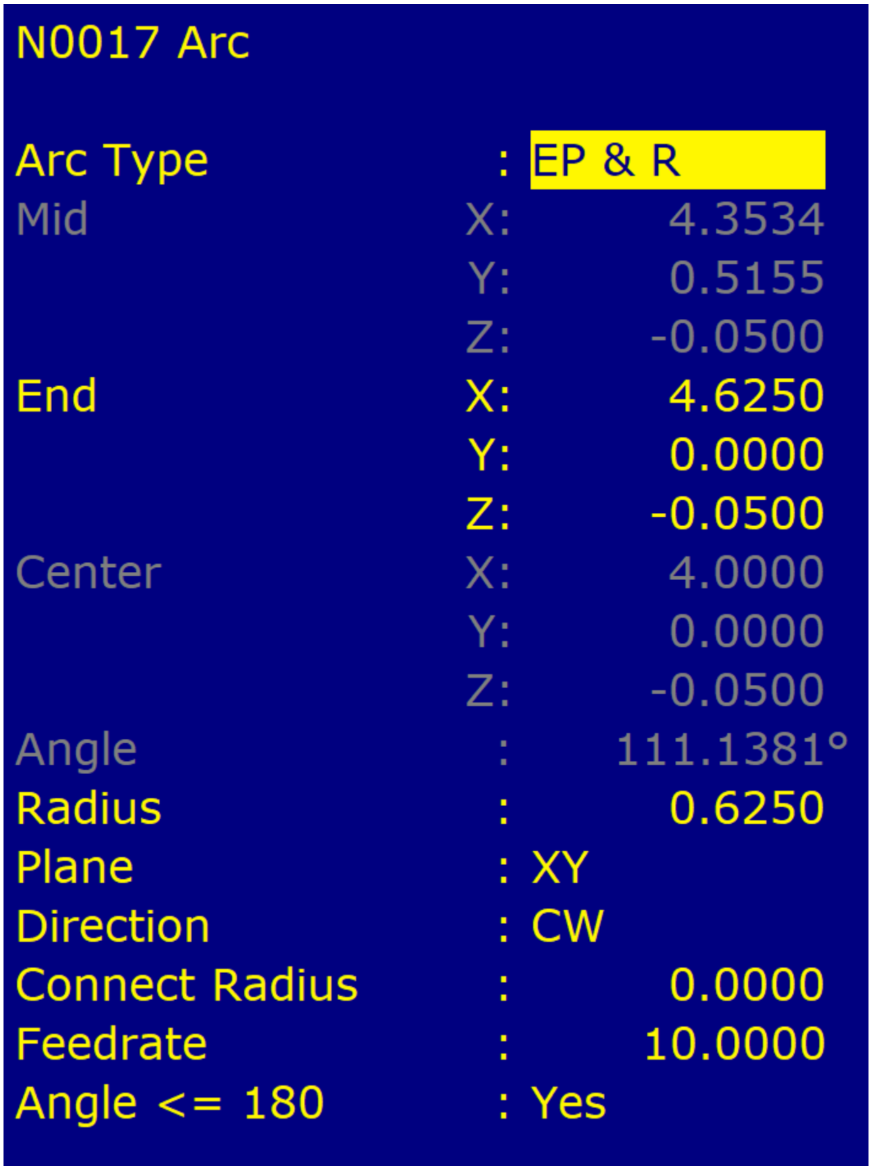
ARROWS Move down to the radius field and enter the radius of the arc labeled ARC 4 in Figure 3. (This radius is 3.1500 inches). Be sure to set the direction to CCW.



F10 – Accept Keep selected values.

F3 – Arc Mill the arc labeled as ARC 5 in Figure 3 back to point P1.

N0017 Arc Mill



F10 – Accept	Keep selected values.
F3 – Arc	Move tool away from the edge of the part after the last arc.

N0018 Arc Mill

F10 – Accept Keep selected values.

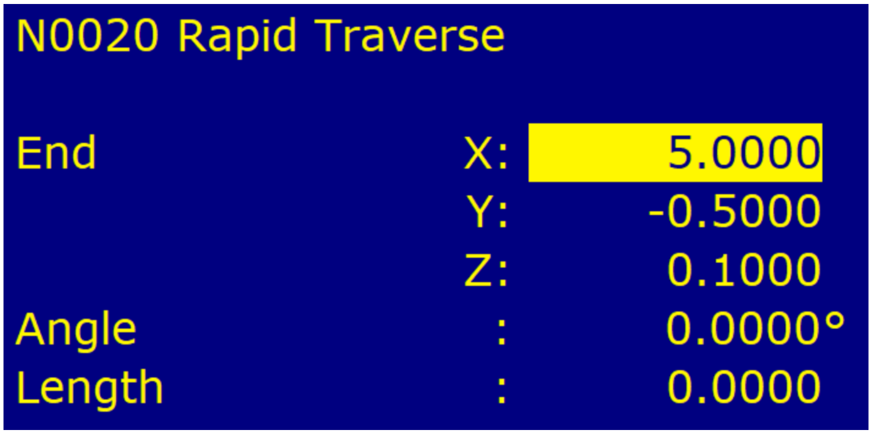
F7 – Cutter Comp Hit the **SPACE** bar until cutter compensation is turned "**Off**". It is no longer needed.

N0019 Comp Off

F10 – Accept Keep selected values.

F1 – Rapid Move the tool away from the part. This is called a lead-out move. When cutter compensation is turned off, the compensation is removed during the next move. This must be done to allow the CNC software to correct its position.

N0020 Rapid Traverse

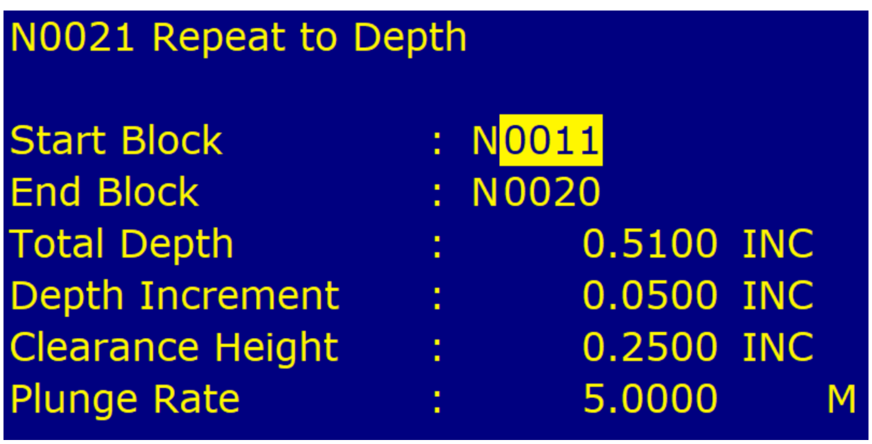


F10 – Accept Keep selected values.

F9 – Subpgm Access the **Subprogram** screen.

F1 – Depth Repeat We programmed the outer contour of the part so that our tool would only penetrate a small portion of the material per pass. We now want to repeat the outer contour operations until the tool has cut the entire way through the material (the assumed material thickness is 0.5 inches). The outer contour begins with the Plunge in operation N0011 and ends with the Linear Mill in operation N0017.

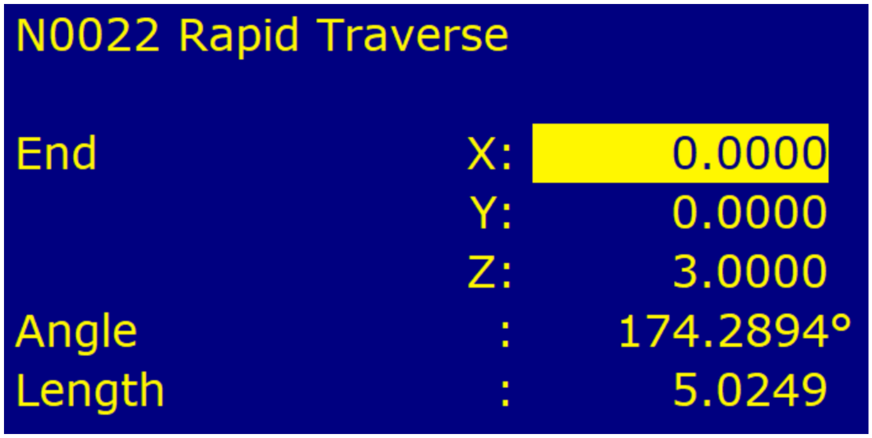
N0021 Repeat To Depth



F10 – Accept Keep selected values.

F1 – Rapid Move the tool away from the part. This is called a lead-out move. When cutter compensation is turned off, the compensation is removed during the next move. This must be done to allow the CNC software to correct its position.

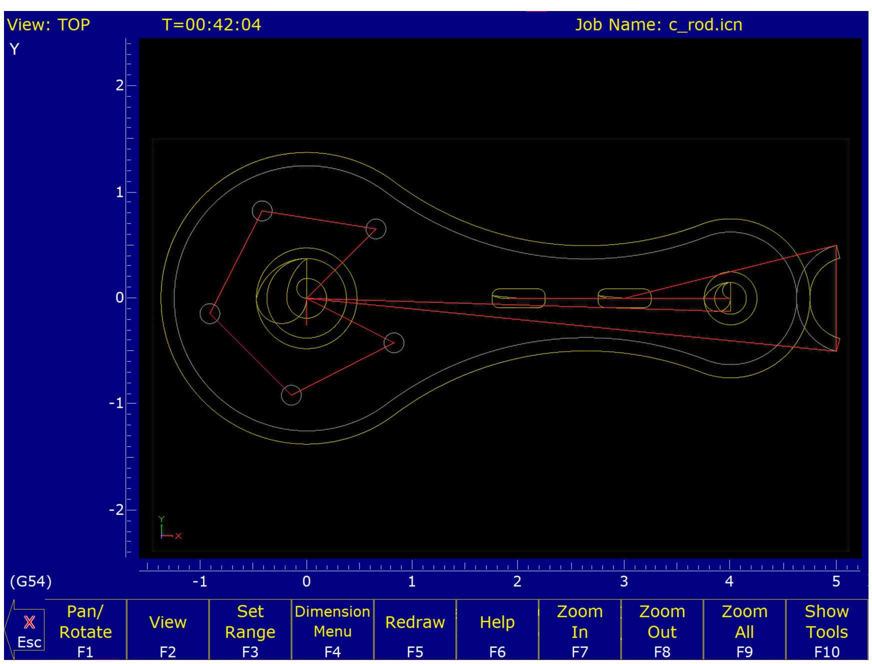
N0022 Rapid Traverse



F10 – Accept Keep selected values.

ESC/CANCEL Creation of the part is complete. Intercon programs automatically turn the spindle and coolant off at the end.

F8 – Graph Display a preview of the finished part. Just make sure that the finished part is going to look the way you want it to. The display shown in Figure 7 has rulers placed around the various view windows that are scaled to the same size as the part displayed to allow visual inspection of the part. Remember, this preview shows where the center of the current tool will move (cutter compensation is not represented except in pocket and frame displays).



ESC/CANCEL Return to Main screen.

F1 – File Go to the File Menu. Press **F3 – Save** to save under the current file name or press **F4 – Save As** to save the program under a different name.

F10 – Post As CNC12 processes each operation, it checks for values that, if used, will cause incorrect code to be produced. If such a value is found, a message will appear on the screen alerting you of the problem. For example, a problem with a rectangular pocket may produce this message:

Corner radius must be equal to or greater than tool radius

Changes to the part would then be required to allow proper code generation to proceed. If no problems are encountered during code generation, the following message appears:

G-code generation successful

You are now at the main menu again.

Program Finished! You are now finished designing your part. In order to run your part, you now need to return to the CNC software.

10.12.2 Milling The Part

Now that the part has been programmed, it is time to mill it. Take your material and clamp it to the table. Remember that the clamps must be positioned such that they do not interfere with the tool as it cuts. You may choose to place the clamps around the edges of the material for the entire process and let the part drop through upon completion, or you may wish to pause after milling the circular pockets and place clamps through the holes to prevent the part from moving. The second option decreases the chance of the part being marred because it moved during milling.

Now you need to set your XYZ reference points. Insert your longest tool in the quill and follow the procedure listed below:

PRESS	COMMENTS
JOG KEYS	Jog the table so that your tool rests on the stock at the location that will represent X0 and Y0.
F1 – Setup	Enter the CNC software Setup screen. We are going to establish the part XYZ zero at the current tool location.
F1 – Part	Access the Part Setup options.
F10 – Set	Set your X zero position at current tool location.
F1 – Next Axis	Select the Y-axis next.
F10 – Set	Set your Y zero position at current tool location.
F1 – Next Axis	Select the Z-axis next.
F10 – Set	Set your Z zero position at current tool location.
TOOL CHECK	Moves the quill to the Z home position if the home position has been set. Moves tool to Z+ limit switch and sets home position if not.

ESC/CANCEL Leave Part Setup screen.

F2 – Tool Access Tool Library Editor. This is the place where we want to measure the actual heights of our tools (since we could not set the actual values in Intercon).

F1 – Offset Lib. You need to make sure that the tool diameter and height offset values are the correct ones for the tools you are going to be using. Inspect the values for D001, H001, D002 and H002. D001 should be 0.1875, H1 should be 0.0000 (the two inch tool).

D002 should be 0.2500 and H002 should be -1.0000 (the one inch tool). If any of these values are incorrect, use the arrow keys to select the incorrect values. Enter the new values in their places and press **ENTER** to accept them.

Note: The tool heights used above are merely example heights. In order to accurately measure the heights of your tools, see the description of measuring tool heights in Chapter 5.

F10 – Save Keep the updated tool offset library values.

F2 – Tools Now you need to make sure that each tool uses the correct diameter and height offset values. Inspect the values for T001 and T002. T1 should use H001 and D001, while T002 should use H002 and D002. If any of these values are incorrect, use the arrow keys to select the incorrect values. Enter the new values in their places and press **ENTER** to accept them. You may also select spindle and coolant settings for your tools here, or enter a short description of the tool.

F10 – Save Keep the updated Tool Library values.

ESC/CANCEL Leave **Tool Setup**. Return to the **CNC software Setup** Screen.

ESC/CANCEL Leave **CNC software Setup**. Return to the **CNC software Main Screen**.

CYCLE START The **CYCLE START/START** button is located on your jog panel. This key will cause the mill to begin cutting your part.

Tutorial Complete!

10.13 Measuring Tool Heights

The following is a brief description of the method used to measure tool height values (offsets). You will need to insert a reference tool into the quill before beginning. For more information also see Chapter 5.

<u>PRESS</u> <u>COMMENTS</u>

F1 – Setup From the main screen enter the Setup.

F2 – Tool Enter tool screen.

F1 – Offset Lib. Enter the tool offsets screen.

JOG ARROWS	You need to jog your reference tool down so it touches the top of some surface.
F1 – Z Ref	Set your Z reference position. This is the value that appears on the DRO when the reference tool touches the top of the surface.
TOOL CHECK	Move the quill up to the Z home position. Insert the first tool to measure.
ARROWS	Select height offset which holds the height of the first tool.
JOG ARROWS	Jog the tool down until it touches the same surface as did the reference tool.
F2 – Manual	Record the height of the first tool.
	Now repeat the last four steps above (from TOOL CHECK to F2 – Manual) for each additional tool to measure.
F10 – Save	Store modifications to offset library of your tools.

11 CNC Program Codes

11.1 General

The next three chapters contain a description of the CNC program codes and parameters supported by the M-Series Control. The M-Series Control has some G-codes and parameters that are modal and some that are "one shots." The G-codes and parameters that are modal will stay in effect until a new G-code or parameter is issued. One shots are effective for the current line only.

For example, a movement command of G1, which is modal, will remain in effect until a different movement command is issued, such as G0, G2, G3, etc.

11.2 Miscellaneous CNC Program Symbols

11.2.1 D – Tool Diameter Offset Number

D is used to select the Tool Diameter Offset from the offset library. The D code values are stored in the Offset Library. Tool Diameter Offsets can be specified anytime before Cutter Comp is turned on (G41 or G42). Once specified, the offset amount is stored and will only be changed when another D code is entered therefore, D is modal. The Tool Diameter Offset (D) can be placed on a line by itself or on a line with other G-codes.

Example:

```
X0Y0F10       ;
G41 D2        ;Enables cutter comp left with diameter D2.
G1X0Y0        ;
X1Y1.25       ;Cutter compensated moves
X2Y1.4        ;
G40           ;Cutter compensation off
G42           ;Enables cutter comp. right (still using D2)
```

11.2.2 E – Select Work Coordinate System

E1 through E18 select among the 18 work coordinate systems. For more information on work coordinate systems see Work Coordinate Systems.

11.2.3 F – Feedrate

The F command is used to set the cutting feedrate. The feedrate is expressed in units/minute. The programmed feedrate may be modified by the feedrate override knob (2–200% for DC systems and 2–100% for AC systems). The default feedrate is 3.0 units/minute. Units may be inches or millimeters.

Example:

```
G90 G1 X1.0 F50    ;linear mill to X1 at 50 units/minute
```

11.2.4 H – Tool Length Offset Number

H is used to select the Tool Length Offset Number. The H code offset amounts are stored in the file Offset Library. Tool Length Offsets can be specified anytime before a G43 or G44 is issued. Once specified the offset amount is stored and will only be changed when another H code is entered therefore, H is modal. The Tool Length Offset (H) can be placed on a line by itself or on a line with other G-codes. H00 is always a 0.0 length offset.

Example:

```
H1           ;Selects offset corresponding to H1.
G43 Z3       ;Moves to Z3 using H1 offset.
G1X0Y1       ;
H3           ;Selects offset corresponding to H3.
X1Y1.25      ;
G0H5         ;Selects offset corresponding to H5.
```

11.2.5 N – Block Number

Block numbers are used to identify CNC program lines. Block numbers are optional, but can be used as the destinations of GOTO statements (see Advanced Macro Statements) and targets of the Search Function (See Main Screen Search option in Chapter 3). Block numbers also can make reading the NC files easier.

Example:

```
N1 G90 G17 M25
N2 G0 X0 Y0 Z0
```

11.2.6 O – Program Number

The O program number allows you to identify your program with a certain number. However, if the specified program number is 9100–9999, the G-codes from the O number through the next M99 will be extracted (but not executed) and placed in a separate subprogram/macro file named Oxxxx.cnc, where xxxx is the specified program number. This separate file can later be called with M98 or G65.

Example:

```
O1521
N1 G90 G17 M25
N2 G0 X0 Y0 Z0
```

11.2.7 P – Parameter

P can correspond to Dwell Time, subprogram number, or a general parameter in canned cycles. This is used as a variable for any of those values in the NC file.

Example:

```
G4 P1.32     ;Pause execution for 1.32 seconds
G10 P73 R.1 ;Set parameter #73 (G73 retract) to .1 inches
```

11.2.8 Q – Parameter

Q is used as a depth parameter in canned drilling cycles.

Example:

```
G73 X1.5 Y2.0 Z-.75 R.25 Q.25 F5    ;Q Sets the depth cut at .25
```

11.2.9 R – Radius, Return Point, Parameter

R can represent the radius, a return point, or a general parameter. This is used as a variable for any of those values in the NC file. R is similar to the P parameter.

Example:

```
G10 D5 R.5      ;sets tool diameter #5 to 0.5" in the offset library
G81 X0 Y0 Z-.5 R.1 F15  ;drill to Z-.5 with return height of .1
```

11.2.10 S – Spindle Speed Setting

Specifying a spindle speed causes the automatic spindle speed setting to be immediately updated. Setting the spindle speed does not cause the spindle to start. The maximum spindle speed is used to compute the output value to the spindle speed control circuit.

Example:

```
S1400 M3 ;Starts the spindle CW at 1400 RPM
```

Note: The Spindle Speed is used in conjunction with the maximum spindle speed to determine the actual spindle speed output to the PLC. Also, this only works when a VFD (Variable Frequency Drive) spindle drive is connected.

11.2.11 T – Select Tool

Prompts the operator to insert the proper tool or change tools, when M6 is encountered.

Example:

```
T1 M6     ;Prompt operator to load tool number 1
T2        ;no action
G0X0Y0    ;move to X0 Y0
M6        ;prompt operator to load tool number 2
```

11.2.12 : – Visible Comment Identifier

The colon (:) is used to indicate the start of a comment line within a CNC program. The colon must be the first character on the line.

Example:

```
:select absolute positioning
```

```
G90
:XY plane
G17
:Visible comments will be displayed on screen with the G-codes.
```

11.2.13 ; – Internal Comment Identifier

The semicolon (;) is used to indicate the start of an internal comment within a CNC program line. All characters after the semicolon are ignored when the program is run. Internal comments are used to document NC programs or temporarily omit the remainder of a line.

Example:

```
G90     ;select absolute positioning
G17     ;XY plane
G1 X1 Y1 F10
G0      ;X0 Y0    ;G0  selected with no movement
```

11.2.14 [ ] – Numerical Expression

The left bracket '[' and right bracket ']' are used to delimit a numerical expression. Numerical expressions can contain floating-point numbers or user and system variables in combination with mathematical operators and functions. The left parenthesis '(' or bracket '[' and right parenthesis ')' or bracket ']' can be used between the first left bracket and last right bracket to force operator precedence or associatively. A bracketed numerical expression can be used anywhere a number would be used. Comparison operators ('eq', 'ne', etc.) have built in rounding specified by Parameter 144. Without this rounding, 'eq' would usually return "false" when comparing two numbers calculated in different ways. Comparison operators and logical operators ('!', '&&', '||') return 1.0 for "true" and 0.0 for "false".

The mathematical operators and functions are:

```
+           Addition (or unary positive)
-           Subtraction (or unary negative)
*           Multiplication
/           Division
^           Exponentiation
Mod or %    Modulo (remainder of a division)
abs         Absolute value
sin         Sine (degrees)
cos         Cosine (degrees)
tan         Tangent (degrees)
sqrt        Square root
asin        Arc Sine (degrees)
acos        Arc Cosine (degrees)
atan        Arc Tangent (degrees)
fix         Floor (round down to nearest integer)
fup         Ceiling (round up to nearest integer)
#           Variable access
eq or ==    Equals
ne or !=    Not equals
```

```
ge or >=    Greater than or equals
gt or >     Greater than
le or <=    Less than or equals
lt or <     Less than
not or !    Logical NOT
&&          Logical AND
||          Logical OR
and         Bit-wise AND
xor         Bit-wise exclusive OR
or          Bit-wise OR
~           Bit-wise complement
```

Example:

```
G91 X[13/64] Z[1+3/8]  ;move the X axis 13/64 (0.2031) units
                       ;and the Z axis 1 3/8 (1.375) units incrementally
X[SQRT[ABS[SIN[#101]-COS[#102]]]] ;Move X as a function of #101 and #102
```

11.2.15 $ – ASCII Code Substitution

The '$' symbol followed by an ASCII code is an alternate way of specifying a letter. Only ASCII codes 65–90 ('A'–'Z') are valid. This substitution can be used where a letter command or axis label is normally specified.

Example:

```
G0 $88 2.0        ;Rapid move X to 2.0  (Equivalent to G0 X2.0)
$71 0 $[88+2] 3.0   ;Rapid move Z to 3.0  (Equivalent to G0 Z3.0)
M26 /$90          ;Set Z home           (Equivalent to M26 /Z)
```

11.2.16 #, = – User or System Variable reference

The ‘#’ character is used to reference a macro or a user or system variable. For variables that can be written, the ‘=’ is used to assign to them. General purpose user variables are #100 to #149 and #29000 to #31999.

Index	Description	Returns	R/W
1–3	Macro arguments A-C	The floating point value if defined by a G65 call, 0.0 otherwise. These can be used as private, local variables in any program or subprogram except in custom macro M functions. In custom macro M functions, the macro arguments are passed in by reference.	R/W
4–6	Macro arguments I-K (1^{st} set)		R/W
7–9	Macro arguments D-F or 2^{nd} set of I-K		R/W
10	3^{rd} I (G is invalid)		R/W
11	Macro argument H or 3^{rd} J		R/W
12	3^{rd} K (L is invalid)		R/W
13	Macro argument M or 4^{th} I		R/W
14	4^{th} J (N is invalid)		R/W
15	4^{th} K (O is invalid)		R/W
16	5^{th} I (P is invalid)		R/W
17–18	Macro argument Q-R or 5^{th} J-K		R/W
19–21	Macro arguments S,T,U or 6^{th} set of I-K		R/W
22–24	Macro arguments V,W,X or 7^{th} set of I-K		R/W
25–27	Macro arguments Y,Z or 8^{th} set of I-K		R/W
28–30	9^{th} set of I-K		R/W
31–33	10^{th} set of I-K		R/W
100–149	User variables	Floating-point value. Initialized to 0.0 at start of job processing	R/W
150–159	Nonvolatile user variables	Floating-point value saved in cncm.job.xml file.	R/W
300–399	User string variables. These variables retain their values until the CNC software is exited	String Literal	R/W
2400, 2401–2418	Active WCS, WCS #1–18 CSR angles	Floating point value	R/W
2500, 2501–2518	Active WCS, WCS #1–18 Axis 1 values		R/W
2600, 2601–2618	Active WCS, WCS #1–18 Axis 2 values		R/W
2700, 2701–2718	Active WCS, WCS #1–18 Axis 3 values		R/W
2800, 2801–2818	Active WCS, WCS #1–18 Axis 4 values		R/W
2900, 2901–2918	Active WCS, WCS #1–18 Axis 5 values		R/W
3000, 3001–3018	Active WCS, WCS #1–18 Axis 6 values		R/W
3100, 3101–3118	Active WCS, WCS #1–18 Axis 7 values		R/W
3200, 3201–3218	Active WCS, WCS #1–18 Axis 8 values		R/W
3901	Parts Cut (Part #)		R/W
3902	Parts Required (Part Cnt)		R/W
4001	Move mode	0.0 (rapid) or 1.0 (feed)	R
4003	Positioning mode	90.0 (abs) or 91.0 (inc)	R
4006	Units of measure	20.0 (inches) or 21.0 (metric)	R
4014	WCS	54.0–71.0 (WCS#1–18)	R
4109	Feedrate (F)	Floating point value	R
4119	Spindle Speed (S)		R
4120	Tool Number (T)		R
4121	Current height offset number (H)		R
4122	Current diameter offset number (D, mill only)		R
4201	Job processing state	0 = normal, 1 = graph	R

Index	Description	Returns	R/W
4202	Job Search mode	0 = search mode off 1 = searching for line number 2 = searching for block number N__ 3 = searching for Tool number 4 = resuming job	R
4203	Tool in spindle		R
5021–5028	Machine Position (axis 1 = 5021, axis 2 = 5022, etc.)	Floating point value	R
5041–5048	Current Position (axis 1 = 5041, axis 2 = 5042, etc.)		R
9000–9999	Parameter values 0–999	See Machine Parameters	R
10000	Mill: Height offset amount, active H	Floating point value	R/W
10001–10200	Mill: Height offset amount, H001–H200	Floating point value	R/W
11000	Mill: Diameter offset amount, active D	Floating point value	R/W
11001–11200	Mill: Diameter offset amount, D001–D200	Floating point value	R/W
12000	Mill: Tool H number, active tool (T)	0–200	R/W
12001–12200	Mill: Tool H number, tools 1–200	0–200	R/W
13000	Mill: Tool D number, active tool (T)	0–200	R/W
13001–13200	Mill: Tool D number, tools 1–200	0–200	R/W
14000	Mill: Tool coolant, active tool (T)	7, 8, 9	R/W
14001–14200	Mill: Tool coolant, tools 1–200	7, 8, 9	R/W
15000	Mill: Tool spindle direction, active tool (T)	3, 4, 5	R/W
15001–15200	Mill: Tool spindle direction, tools 1–200	3, 4, 5	R/W
16000	Mill: Tool spindle speed, active tool (T)	Floating point value	R/W
16001–16200	Mill: Tool spindle speed, tools 1–200	Floating point value	R/W
17000	Mill: Tool bin number, active tool (T)	Floating point value	R/W
17001–17200	Mill: Tool bin number, tools 1–200	Floating point value	R/W
18000	Mill: Tool putback, active tool (T)	Floating point value	R/W
18001–18200	Mill: Tool putback, tools 1–200	Floating point value	R/W
19000	Tool Life Data: Tool T1 Tool Type	0 = Drill, 1 = End Mill	R/W
19001	Tool Life Data: Tool T1 Total Life	Floating point value	R/W
19002	Tool Life Data: Tool T1 Used Life	Floating point value	R/W
19003	Tool Life Data: Tool T1 Units	0 = Cycles, 1 = Inch/mm distance	R/W
19004	Tool Life Data: Tool T1 Update Mode	0 = Manual, 1 = Auto	R/W
19005–19009	Tool Life Data for Tool T2	see 19000–19004 above	R/W
19010–19014	Tool Life Data for Tool T3	see above	R/W
19015–19999	Tool Life Data for Tools T4 through T200	see above	R/W
20001–20008	max_rate for axes 1–8		R
20101–20108	label for axes 1–8		R
20201–20208	slow_jog for axes 1–8		R
20301–20308	fast_jog for axes 1–8		R
20401–20408	screw_pitch for axes 1–8		R/W
20501–20508	lash_comp for axes 1–8		R
20601–20608	counts_per_unit for axes 1–8		R
20701–20708	accel_time for axes 1–8		R
20801–20808	deadstart_velocity for axes 1–8		R
20901–20908	delta_vmax for axes 1–8		R
21001–21008	counts_per_turn for axes 1–8		R
21101–21108	minus_limit for axes 1–8		R

Index	Description	Returns	R/W
21201–21208	plus_limit for axes 1–8		R
21301–21308	minus_home for axes 1–8		R
21401–21408	plus_home for axes 1–8		R
21501–21508	reversed for axes 1–8		R
21601–21608	laser_comp for axes 1–8		R
21701–21708	proportional for axes 1–8		R
21801–21808	integration_limit for axes 1–8		R
21901–21908	kg for axes 1–8		R
22001–22008	integral for axes 1–8		R
22101–22108	kv1 for axes 1–8		R
22201–22208	derivative for axes 1–8		R
22301–22308	ka for axes 1–8		R
22401–22408	num_motor_poles for axes 1–8		R
22501–22508	drive_current for axes 1–8		R
22601–22608	drive_offset_angle for axes 1–8		R
22701–22708	pwm_kp for axes 1–8		R
22801–22808	pwm_ki for axes 1–8		R
22901–22908	pwm_kd for axes 1–8		R
23001–23008	abrupt_kp for axes 1–8		R
23101–23108	feed_forward_kp for axes 1–8		R
23201–23208	max_error (PID) for axes 1–8		R
23301–23308	min_error (PID) for axes 1–8		R
23401–23408	at_index_pulse for axes 1–8		R
23501–23508	travel_minus for axes 1–8		R/W
23601–23608	travel_plus for axes 1–8		R/W
23701–23708	axis_home_set for axes 1–8		R
23801–23808	abs_position (in encoder counts) for axes 1–8		R
23901–23908	PID_out for axes 1–8		R
24001–24008	reference set for axes 1–8		R
24101–24108	Axis reference value for axes 1–8		R
24201–24208	tilt table level offsets for axes 1–8		R
24301–24308	dsp positions for axes 1–8		R
24401–24408	abs_position (in encoder counts) for axes 1–8		R
24501–24508	dsp positon in local cooridinates for axes 1–8		R
24601–24608	local probing +limit position for axes 1–8		R
24701–24708	local probing -limit position for axes 1–8		R
24801–24808	probe stylus compensation amount for axes 1–8		R
24901–24908	servo controlled axis indicator for axes 1–8	0 = no, 1 = yes	R
25000	DRO_display_units		R
25001	default_units_of_measure		R
25002	PLC_type		R
25003	console_type		R
25004	jog_panel_optional		R
25005	min_spin_high		R
25006	max_spin_high		R
25007	home_at_powerup		R

Index	Description	Returns	R/W
25008	screen_blank_time		R
25009	Displayed / Calculated spindle speed. If Parameter 78 =1 and spindle encoder is mounted.		R
25010	current spindle position (in counts)		R
25011	dsp_time (in seconds)		R
25012	time (in seconds)		R
25013	clear max/min PID errors		R
25014	software type (Mill/Lathe)		R
25015	feedrate override		R
25016	spindle override		R
25017	OS	Windows/LINUX = 2; other OS = 1.0	R
25018	CNC series number (11 for CNC11)		R
25019	Software version number		R
25020	Software Beta revision number		R
25021	Digitizing boundary hit	hit = 1, not hit = 0	R
25022	last M115/116/125/126 probe trip	1 = tripped, 0 = not tripped	R
25023	Drive type	0 = Standard, 1 = Legacy DC, 2 = Legacy AC	R
25029	Lifetime power on time	floating point hours	R
25030	Lifetime job in progress time	floating point hours	R
25031	Lifetime spindle on time	floating point hours	R
25101–25108	Encoder count position of the last index pulse for axes 1–8	A value of -1 indicates either that the index pulse has not yet been detected or, less likely, the last index pulse was actually at -1 (with disambiguation left to the user)	R
26001–26008	dsp mechanical machine positions for axes 1–8		R
26101–26108	dsp mechanical local positions for axes 1–8		R
26201–26208	local + travel limit position for axes 1–8		R
26301–26308	local $-$ travel limit position for axes 1–8		R
26401–26404	Axis 1 reference points 1–4		R
26501–26504	Axis 2 reference points 1–4		R
26601–26604	Axis 3 reference points 1–4		R
26701–26704	Axis 4 reference points 1–4		R
26801–26804	Axis 5 reference points 1–4		R
26901–26904	Axis 6 reference points 1–4		R
27001–27004	Axis 7 reference points 1–4		R
27101–27104	Axis 8 reference points 1–4		R
27201–27208	ACDC drive estimated brake wattage for axes 1–8		R
27301–27308	Real motor encoder positions for axes 1–8	Motor encoder positions that accounts for lash, MPG, and scale offsets. (Note that these can be different from what is displayed as Abs Pos in the PID menu.)	R
27401–27408	Scale encoder positions for axes 1–8		R
27501–27508	Scale Counts/Unit for axes 1–8		R
27601–27608	`SV_SCALE_INITIALZED_AXIS_1-8`	same as PLC variable	R
27701–27708	`SV_SCALE_ENABLED_AXIS_1-8`	same as PLC variable	R

Index	Description	Returns	R/W
27801–27808	Position Mode Error Correction	same as PID Encoder menu	R
27901-27908	Axis Homing Feedrates		R
29000–31999	User variables. These variables retain their values until the CNC software is exited.	Floating point value	R/W
32000–34999	Reserved for internal use. **DO NOT USE**	Floating point value	R/W
50001–51312	PLC Inputs 1–1312	Jog Panel is on INP1057–1312	R
60001–61312	PLC Outputs 1–1312	Jog Panel is on OUT1057–1312	R
70001–71024	PLC Memory Bits 1–1024		R
80001–89999	Reserved		R
90001–90128	Timer 1–128 status bits		R
90129–92999	Reserved		R
93001–93256	Stage 1–256 status bits		R
94001–94256	Fast Stage 1–256 status bits		R
95001–95256	Reserved		R
96001–96088	W1-W88 (32-bit signed integers)		R
97001–97022*	DW1-DW22 (64-bit signed integers)		R
98001–98044	FW1-FW44 (32-bit floats)		R
99001–99022	DFW1-DFW22 (64-bit floats)		R

* Since user or system variables are turned into (double) floating point values when referenced in an M- or G-code program, the 64-bit integer values lose precision when they exceed 2^{53} (9,007,199,254,740,992).

Example:

```
#100 = #5041          ;set user variable #100 to the X axis current position
G90 X[#5041+1+7/32]   ;move the X axis 1 7/32 units (1.2188) incrementally
#2501 = #5021         ;set WCS#1 X value to the current X position
#2703=[#2703+1/8]     ;add 1/8 units (.125) to the WCS#3 Z value

                      ;Subroutine parameter and local variable access.
G1 X#A Y#B Z#C F#F    ;move to the coordinates passed as parameters
#[Q] = #F * .10       ;Assign local variable #Q to 10% of #F
#17 = #7 * .10        ;Same statement as previous using number references.
#[C] = 0.05           ;Reassign #C. (Value passed as parameter is lost.)
```

11.3 Introduction to Centroid CNC Macros

11.3.1 What is a Macro?

A Macro used to describe a set of CNC control commands that are called upon to perform a machine tool function, think of it as a bundle of commands that you can use at the touch of a button or by adding a single M code into a G&M code program. The word macro is a term is being loosely used here to cover a wide variety of CNC “programs”, everything from a machine tool homing program to an automatic tool change program. Macros are simply G and M code programs, they can be one line or thousands of lines. Most all CNC12 G&M codes can be used in a macro. There are many stock ‘macros’included with CNC12. CNC12’s included “M-Functions” are used to perform specialized actions in CNC programs. Most of the CNC12 M-Functions have default actions as described in the Mill and Lathe operators manuals, but can also be customized with the use of macro files.

11.3.2 How do I create or edit a Macro?

A macro program is just a text file like any other G and M code program. We recommend using Notepad++ (not Notepad).

Notepad++ is a free powerful text editor and can be downloaded for free here.
https://notepad-plus-plus.org/downloads/

11.3.3 Where to Start?

There are included "stock" macros with the Acorn CNC12 software installation. Many of these macros have instructions and comments built into them so they can be user modified and also used as macro programming learning examples. Most macros end with the .mac file extension with some exceptions for special cases such as the home program files which are "cncm.hom" and "cnct.hom".

The most common Acorn CNC macros for beginners to edit and modify are:

- Machine Tool Homing Macro: cncm.hom (mill or router) or cnct.hom (lathe)
- Machine Tool Parking Macro: park.mac
- Macros that are preassigned to the VCP Aux keys 8, 9, 10, 11: M55 (Aux 8), M56 (Aux 9), M57(Aux 10), M58 (Aux 11)
- Macros that are preassigned to the Wireless MPG Macro keys 1–4: Plcmacro1(2,3,4).mac

The Home macro is created by the Acorn Wizard based on the selections made in the Wizard for the type of homing, direction of homing, order of homing. Most Acorn users will not have to edit the Home macro as the Wizard does a good job creating home macros (aka home programs). However there are a cases where it is nice to add some customization to the home program to make the work flow on a machine tool better.

The pre-assigned Macros for the VCP aux keys and the MPG Macro buttons are themselves commented with examples. Once you have Acorn up and running and homed, Press the VCP Aux 8 key (that will run the M55 macro) and the instructions on how to edit M55 appear on the screen (and same with the Wireless MPG Macro 1,2,3,4 buttons). The M-code M55 calls and runs the commands contained in the text file "mfunc55.mac. Opening, studying and modifying these existing macro is a great place to start.

The included macros are located in these directories "cncm", "cnct", "cncm/system" and "cnct/system".

11.3.4 Edit your first macro!

Below is a typical Wizard generated Acorn Home Program for a three axis milling machine.

```
M91/Z L1 ; move Z axis in negative direction seeking home switch, trigger
   switch and then reverse to clear switch
M26/Z    ; set Z home at current position
M92/Y L1 ; move Y axis in positive direction seeking home switch, trigger
   switch and then reverse to clear switch
M26/Y    ; set Y home at current position
M91/X L1 ; move X axis in negative direction seeking home switch, trigger
   switch and then reverse to clear switch
M26/X    ; set X home at current position
```

Lets say we want to set the Z axis Home position .050" away from the Z home switch rather than right after the switch clears and do this automatically.

Open “cncm.hom” with Notepad ++ and insert the line shown highlighted below

```
M91/Z L1       ; move Z axis in negative direction seeking home switch, trigger
    switch and then reverse to clear switch
G91 Z .050 F25 ; move Z axis incrementally in the positive direction .050"
M26/Z          ; set Z home at current position
M92/Y L1       ; move Y axis in positive direction seeking home switch, trigger
    switch and then reverse to clear switch
M26/Y          ; set Y home at current position
M91/X L1       ; move X axis in negative direction seeking home switch, trigger
    switch and then reverse to clear switch
M26/X          ; set X home at current position
```

Using Notepad++ save the file (File, Save). Now the next time you home the machine (or press reset home on the VCP) the machine, after clearing the Z home switch, will back away from the Z home switch an additional .050" before setting the Z home position at the current location. See Operators manual G and M code chapters for description of M26, M91/2 and G91

11.3.5 Edit your second macro!

Lets edit M55 so that it will Rapid move to the XY position X=0, Y=0.

M55 is a stock Acorn macro that is preassigned to the VCP (Virtual Control Panel) Auxiliary key #8 so when you press Aux 8 CNC12 will execute the commands contained within M55. Lets edit M55 so that it will rapid move the machine to WCS X=0, Y=0. Open mfunc55.mac with Notepad++.

Note: A semi colon “;” tells CNC12 that the line is a comment, remove semi colon for command to be run, add semi colon to beginning of line to be ignored.

```
;--------------------------------------------------------------------------------
; Filename: mfunc55.mac - To run from VCP AUX 8 key, set p195 = 5511
; M55 macro
; Description: User Customizable Macro
; Notes:
; Requires: Machine home must be set prior to use.
; Please see TB300 for tips on writing custom macros.
;--------------------------------------------------------------------------------
IF #50001 ; Prevent G code lookahead from parsing past here
IF #4201 || #4202 THEN GOTO 1000 ; Skip macro if graphing or searching
M225 #100 "This is an example macro named mfunc55.mac and can be found in ..\
   cncm or ..\cnct\nEdit it to include the desired functionality.
Press Cycle Cancel/ESC to Exit"
N100  ; Insert your code between N100 and N1000
      ; Insert commands here.
N1000 ; End of Macro
```

Place a semi colon in front of the M225 line to comment it out, and then insert your custom G and M codes in between lines N100 and N1000.

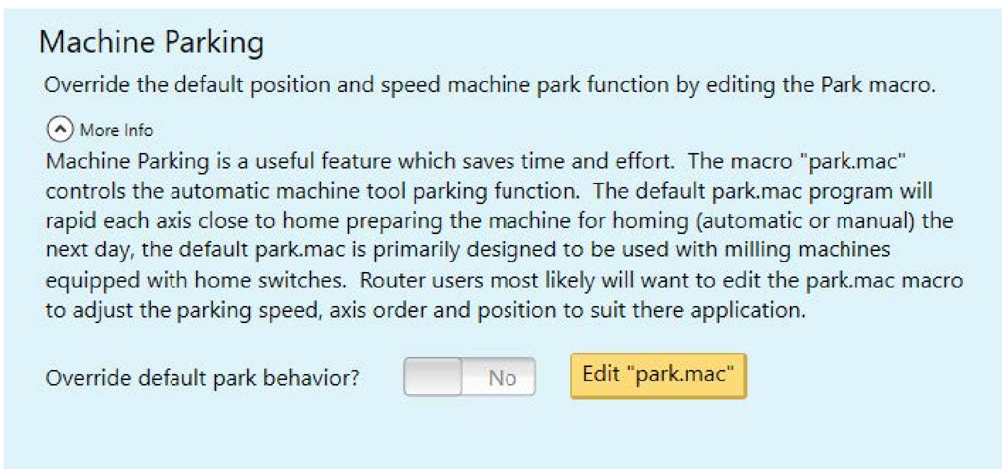
So, to modify M55 to Rapid X and Y axis to the position X0, Y0 to look like this:

```
; M225 #100 "This is an example macro named mfunc55.mac and can be found in ..\
    cncm or ..\cnct\..............etc.
;
N100   ; Insert your code between N100 and N1000
G0X0Y0 ; G0 = Rapid Move to position X=0, Y=0 in the current Work Coordinate
    System
N1000  ; End of Macro
```

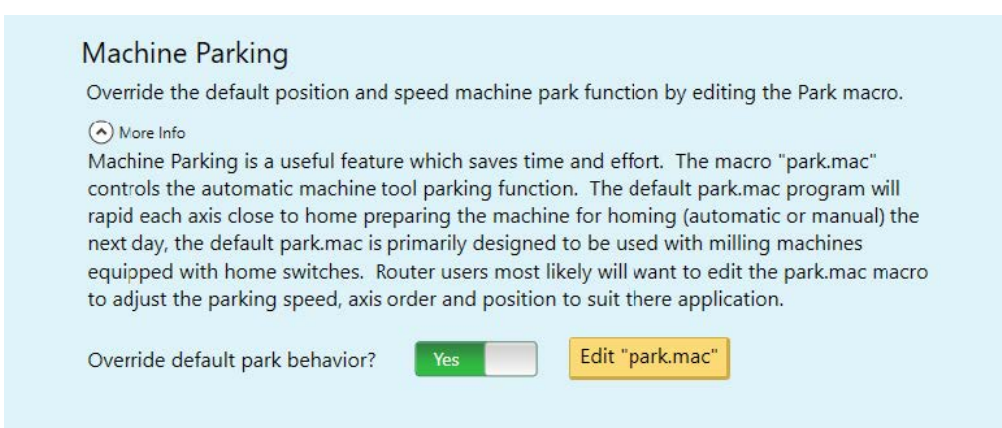
Using Notepad++ save the changes to mfunc55.mac using File, then Save. Now in CNC12 the machine tool will rapid to the XY position 0,0 when Aux 8 is pressed or when a M55 is contained in a G and M code program.

11.3.6 Edit your third Macro!

Park.mac is a Machine Tool Parking macro that is primarily used to park a machine at the end of the day. Typical use of a Park macro will move the machine close to the home switches so that the machine is ready to run the home macro in the morning with minimum of movement. CNC12 has default Parking functionality built into it so, if the Acorn Wizard is set to No as seen below then then the default CNC12 park action is used.



The default action is fine for most average milling machines but, I find that most users will benefit from editing and creating a custom Park macro to suit their particular machine tool and application. The default action will Rapid Move the axes close the to home switches. A common custom Park.mac will move the axis close to the home switches but at a user specified feedrate.



Choose “Yes” to override the default park behavior and edit “park.mac”.

The stock Park.mac macro looks like this below, there are three examples given that are commented out with a semi colon.

```
;-------------------------------------------------------------------------------
; Filename: park.mac - Custom Park Button Macro
; Description: This Machine Park macro controls the "F1 Park" command action
   found in the CNC12 "F10 Shut Down" menu.
; park.mac is a user editable custom macro used for parking a machine tool at a
    specified position at a specified speed, typically used prior to shut down.
; Notes:
; - This macro overrides the default CNC12 park behavior logic when selected to
    do so in the Wizard (the Wizard sets CNC12 parameter 59 = 1)
; - This macro allows the user to customize the action and behavior of the "
   Park" feature in the CNC12 "Shut Down" menu by writing creating a custom G&M
    code program
; - A semi colon ";" tells CNC12 that the line is a comment, remove semi colon
   for command to be run, add semi colon to beginning of line to be ignored
;-------------------------------------------------------------------------------
M225 #100 "Please edit c:\cncm\system\park.mac to create a custom parking macro
   , press ESC to exit."
;-------------------------------------------------------------------------------
; Example Custom Mill Park Macro assuming XYZ home position is set in the
   negative direction for each axis.
;G53 Z.25 L20 (Moves Z axis first in machine coordinates to .25 inches away
   from Z home position at 20 inches per minute)
;G53 X1 L200 (Moves X axis second in machine coordinates to 1 inch away from X
   home position)
;G53 Y1 L200 (Moves Y axis third in machine coordinates to 1 inch away from Y
   home position)
;-------------------------------------------------------------------------------
; Example Custom Park Macro assuming Y home position is set in positive
   direction and XZ home position is set in the negative direction for each
   axis.
;G53 Z.25 L20 (Moves Z axis first in machine coordinates to .25 inches away
   from Z home position at 20 inches per minute)
;G53 X1 L200 (Moves X axis second in machine coordinates to 1 inch away from X
   home position at 200 inches per minute)
;G53 Y-1 L800 (Moves Y axis third in machine coordinates to 1 inch away from Y
   home position at 800 inches per minute)
;-------------------------------------------------------------------------------
; Example Custom Park Macro assuming Y home position is set in positive
   direction and XZ home position is set in the negative direction for each
   axis.
;G53 Z.25 L20 (Moves Z axis first in machine coordinates to .25 inches away
   from Z home position at 20 inches per minute)
;G53 X1 Y-1 L200 (Moves X and Y axis at the same time in machine coordinates to
    1 inch away from X home position at 200 inches per minute)
;G53 w0 L360 (Moves W rotary axis in machine coordinates to 0.000 at 360
   degrees per minute)
;-------------------------------------------------------------------------------
```

To customize Park.mac, Add a semi colon in front of the M225 line to comment it out.

And then uncomment one of the examples provided and edit the feedrate and direction to your requirements. See changes highlighted in yellow below.

```
;-------------------------------------------------------------------------------
; Filename: park.mac - Custom Park Button Macro
; Description: This Machine Park macro controls the "F1 Park" command action
   found in the CNC12 "F10 Shut Down" menu.
; park.mac is a user editable custom macro used for parking a machine tool at a
    specified position at a specified speed, typically used prior to shut down.
; Notes:
; - This macro overrides the default CNC12 park behavior logic when selected to
    do so in the Wizard (the Wizard sets CNC12 parameter 59 = 1)
; - This macro allows the user to customize the action and behavior of the "
   Park" feature in the CNC12 "Shut Down" menu by writing creating a custom G&M
    code program
; - A semi colon ";" tells CNC12 that the line is a comment, remove semi colon
   for command to be run, add semi colon to beginning of line to be ignored
;-------------------------------------------------------------------------------
;M225 #100 "Please edit c:\cncm\system\park.mac to create a custom parking
   macro, press ESC to exit."
;-------------------------------------------------------------------------------
; Example Custom Mill Park Macro assuming XYZ home position is set in the
   negative direction for each axis.
;G53 Z.25 L20 (Moves Z axis first in machine coordinates to .25 inches away
   from Z home position at 20 inches per minute)
;G53 X1 L200 (Moves X axis second in machine coordinates to 1 inch away from X
   home position)
;G53 Y1 L200 (Moves Y axis third in machine coordinates to 1 inch away from Y
   home position)
;-------------------------------------------------------------------------------
; Example Custom Park Macro assuming Y home position is set in positive
   direction and XZ home position is set in the negative direction for each
   axis.
;G53 Z.25 L20 (Moves Z axis first in machine coordinates to .25 inches away
   from Z home position at 20 inches per minute)
;G53 X1 L200 (Moves X axis second in machine coordinates to 1 inch away from X
   home position at 200 inches per minute)
;G53 Y-1 L800 (Moves Y axis third in machine coordinates to 1 inch away from Y
   home position at 800 inches per minute)
;-------------------------------------------------------------------------------
; Example Custom Park Macro assuming Y home position is set in positive
   direction and XZ home position is set in the negative direction for each
   axis.
G53 Z.25 L20 (Moves Z axis first in machine coordinates to .25 inches away from
    Z home position at 20 inches per minute)
G53 X1 Y-1 L200 (Moves X and Y axis at the same time in machine coordinates to
   1 inch away from X home position at 200 inches per minute)
G53 w0 L360 (Moves W rotary axis in machine coordinates to 0.000 at 360 degrees
    per minute)
;-------------------------------------------------------------------------------
```

Using Notepad++ save the changes to park.mac using File, then Save. Now in CNC12 the Park feature found in the Shut Down menu will run this custom macro and in the example above will move Z slowly .25" from Z home and then move X and Y simultaneously to 1" away from machine home position and then moves the rotary axis to the W0 machine position.

See Operators manual G and M code chapters for description of G53 and M225 and FAQ section below for description of "L" command.

Tip: If you built your own CNCPC edit the Windows file extension default application settings so Notepad ++ will be the default editor for .mac, .hom, .cnc, .nc, .txt.

The mill (and lathe) operator manual has a number of sections covering macro programming.

Once you are familiar with a basic macros such as mfunc55.mac, cncm/t.hom and park.mac you can start to increase your macro knowledge by reading the related macro material in the CNC12 Operators Manuals:

CNC12 Mill Operator Manual (manuals can be found here.
https://www.centroidcnc.com/centroid_diy/centroid_manuals.html)

Chapter 11: CNC program codes	page 202
Chapter 13: CNC program M codes	page 284
G65 – Call Macro	page 263
M98/99 Call Subprogram	page 293
M100/M101 Wait for PLC bit	page 295
M200/M223, M224, M225, M290 Formatted String	page 302
Section 11.4 Advanced Macro Statements	page 244
Chapter 14 covers stock ATC macros	page 307

Acorn Wizard canned PLC functions and M Codes. https://www.centroidcnc.com/centroid_diy/downloads/acorn_documentation/acorn_wizard_input_output_plc_functions.pdf

And be sure to visit these two threads on the Acorn CNC Tech Support Forum for free downloads of custom macros for AutoTool Setting and ATC's. "Tool Setting Options For Routers and Mills" and "Acorn ATC Overview" in the "Acorn CNC Tech Tips Knowledge Base" forum.

11.3.7 Other Related useful Reference Material:

- Centroid CNC12 PLC Programming Manual
(https://www.centroidcnc.com/centroid_diy/centroid_manuals.html)

- Centroid PLC Programming Videos on the Centroid CNC Tech support YouTube Channel https://www.youtube.com/playlist?list=PLXhs2C5No0_gFS_RmKNo7hii2WKIedQlQ
- PLC detective https://www.centroidcnc.com/downloads/centroid_PLC_detective_quickstart.pdf
- CNC Services Northwest's web page: Advanced CNC Macro Programming Tips and Techniques for Centroid Controls. http://www.cncsnw.com/AdvancedMacroProgramming.htm

11.3.8 Common Uses of Macros

Machine tool Homing, Machine tool parking, custom Probing, Rack Mount tool changers the possibilities are endless.

11.3.9 Limitations of Macros

Macros inherit the same limitations as when running G&M programs in MDI or running a job file. One such limitation is that macros require supporting PLC code to directly interact with outputs. Macros can read the states of inputs, outputs, and memory with the m100 and m101 command. Supporting PLC code is required for a macro to toggle an output using the m94 and m95 commands as seen in the examples below.

11.3.10 What is a User Variable?

Think of a variable as a memory location that you get to use for any purpose in your macro. Just like you would write down a measurement or other number to record it in a notebook to remember it and use it at a later time a User Variable is commonly used this way in a custom macro. There are two types of user variables, volatile and nonvolatile, which are defined as variable numbers 100–149 and 150–159 respectively. The value of a volatile variable as the name implies will be "erased" after the macro is finished or a power cycle. The value of a non-volatile variable will be retained after the macro is finished and through a power cycle this makes nonvolatile user variables useful in certain applications such as keeping track of what tool is in the spindle in a custom ATC macro. Throughout this document we will be assigning these user variables 100–149 and 150–159 values in the examples given. User variables are different from system variables (described below) in that they do not have interaction with any stock CNC12 function hence the name "User Variable" therefore by definition variable #'s 100–149 and 150–159 will have no conflict with the standard feature set of CNC12.

11.3.11 Commonly Used Expressions in Macros:

(see the entire list Section 11.2.14)

- Equals `==` (or "`eq`")
- Logical Not `!` (or "`not`")
- Logical and `&&`
- Logical or `||`
- Greater than `>` (or "`gt`")
- Less than `<` (or "`lt`")

11.3.12 How to Read and Write to a User Variable:

Macros can be used to both read values and write to some system variables in the control software. Common system variables used are user variables numbers: #100–149, non-volatile user variables numbers: #150–159 and the Tool Number indicator (#4120)

Referencing the Manuals, a variable can be read by a macro if it has an "R" in the R\W column of the chart.

If a variable has a "W" then the macro can also write to this value.

```
#100 = 1                  ; writes the value of 1 to the user variable #100
#101 = 1 + 1              ; writes the value of 2 to the user variable #101
#103 = [1 + #100 + #101] ; writes the value of 4 to the user variable #103
#150 = #4120              ; sets user variable 150 to equal the value of the
   Requested Tool Number *see note below
```

#4120 is the CNC12 System Variable number for the Tool Number requested by the T command. For example if you entered T9 into MDI, the software sets #4120 to the value of 9. A common use of setting a user variable equal to #4120 is this allows the author of a macro to use the current tool number in a custom m6 ATC macro for tool changes. This line is used to keep track of which tool is currently in the spindle. Examples of this are presented later in this document.

Note: The line: "`#150 = #4120`" needs an important "`IF #50001`" to accompany it and is explained later in this document.

11.3.13 What is a System Variable?

A system variable is a memory location contained in CNC12 defined by a number. System variables reference values, settings, and words found in the CNC12 software. Some of the most commonly used system variables in macros are 9000 to 9999 which represent CNC12 system parameters 0 to 999. These are the same parameters that you can see in the parameter menu (F1 Setup, F3 Config, F3 Params) in CNC12, however when we want to reference these parameters in the macro, we use the system variable that represent the parameter.

Refer to Section 11.2.16 for full list of CNC12 system variables.

11.3.14 How to interact with an Input

Inputs can be interacted with using M100/M101, M105/M106 or IF statements. M100/M101 commands simply look for the input while M105/M106 moves an axis in the minus or plus direction looking for the input and the IF statements will look for if the expression is true or not.

Example 1: Set current WCS X = 0.0000 when input 1 turns off (a Home Switch for example):

```
N100
M105 /X P1 F30 ; M105 = Move In Minus Direction, /X = Move X axis, P1 = Look at
    Input 1 to Open,
               ; F30 = Move at 30" /min (Based on CNC settings, might be 30mm /
                  min if set to metric)
G92 X0         ; G92 = Set Absolute Position for WCS, X0 = Sets X-axis to zero
N1000
```

Example 2: Set current Machine X = 0.0000 when input 1 turns off (a Home Switch for example):

```
N100
M105 /X P1 F30 ; M105 = Move In Minus Direction, /X = Move X axis, P1 = Look at
     Input 1 to Open,
                 ; F30 = Move at 30" /min (Based on CNC settings, might be 30mm /
                    min if set to metric)
M26 /X
                 ; G26 = Set Axis Machine Home, /X = Sets home for the X-axis
                 ; Sets Machine Coordinates X-axis to zero
N1000
```

Example 3: Look for an input to be closed (on) before continuing the macro:

```
              ; Note: Input 3 has been defined as "Tool is Unclamped" using the
                 Acorn Wizard.
              ;
N100          ; Block 100 (start of macro)
M15           ; Unclamp tool, M15 is a macro that activates the tool unclamp
    output
M101 /50003 ; Wait for input 3 to close (turn on) to verify that the ATC
    spindle has actually unclampled the tool
              ; ......the macro continues on when it sees input 3 has been made!
```

Example 4: Stop and Wait for operator with message. In this example below a message will be displayed for the operator until the cycle start button is pressed.

```
; Note: Input 3 has been set to "Tool is Unclamped" using the Acorn Wizard,
    typically a ATC spindle sensor is used
;
#100 = 0 ; Sets user variable #100 (#100 is being used to set a timer for how
    long the Message will be displayed
; a value of 0 means the message will display until user presses cycle start.
; Use M225 to stop, issue message and look for input
;
IF !#50003 THEN M225 #100 "Tool in Spindle, Remove and press cycle start"
;
; If Input 3 is Open (off), Then display the message and wait for operator to
    press start to continue
; (See Section 13.55 for more info on M225)
```

M225 is the display formatted string for a period of time command. In the example above, #100 is a user variable that defines the time the message will display. If a m225 command is given a zero time, it will display the message indefinitely 7waiting for the cycle start key to be pressed, otherwise it will wait the amount of time specified then will continue with the macro.

Alternatively, an M00 or M01 can be used if you desire no message and for the macro to simply wait for the operator to hit cycle start. Useful for situations where the need to check over the machine during a particular part of a macro or job is required.

Example 5: Unlike Outputs, Inputs do not have to be defined in the PLC program for a macro to use them. Also, an input that is defined in the PLC program can be used by the macro not only for the intended purpose of the input but also for other functions.

For instance: Assume Input 3 is left unused in the Acorn Wizard but a Surface Plate is wired to input 3, a macro can still “see” input 3 and react to its state.

```
M115 /Z P3 F20 ; Move at fast probing rate until Surface Plate is detected
M116 /Z P3 F10 ; Retract at 10 ipm feedrate until Surface Plate clears
M115 /Z P3 F5  ; Move at 5 ipm feedrate until Surface Plate detected
G92 Z.5        ; Set Z position to Surface Height Thickness (sets Z 0 at top of
    work or spoil board)
G4 P.5         ; Wait half a second
G53 Z.5        ; Retract Z to .5 in machine coordinates, (.5 away from Z home
   position)
```

Now, assume Input 3 is defined in the Acorn Wizard as “ToolTouchOffTriggered” AND a Surface Plate is also wired to the same input 3. The very same macro above can be used. It is interesting to note that a macro can “look” for a Surface Plate Detector which is also wired into the same input at the TT device saving on inputs necessary to have both Surface Plate and Tool Touch off functionality. Just keep in mind that any stock functionality of the Input assignment made in the Wizard (like the tool touch off messages and probe protection) will be enabled for that input, even though the macro is not intended to used input 3 as a Tool Touch Off.

11.3.15 What is a parameter?

CNC12 control configuration parameters are used to configure CNC12 to meet the CNC requirements of a particular machine tool, these parameters allow access to hard coded CNC functionality within CNC12 and allow the integrator to choose and fine tune certain aspects of CNC12 software. See Section 15.3 for details on the standard CNC12 parameters. You will notice that there are a large number of built-in functions and configurations CNC12 that are controlled by the CNC12 parameters. These parameters allow the user/integrator to pick and choose how they would like the CNC control to act or appear by editing the parameter values to select the choice or functionality they desire. For instance if we would like the CNC12 User Interface language to display in German. Set Parameter #9 value = 5, press F10 to save and bingo, CNC12 is in German.

Typically most Acorn users will never have to edit or change a CNC12 control configuration Parameter as the Wizard sets most all the Acorn the related parameters for you. But there are cases where Acorn users would have to edit a CNC12 parameter manually (such as the Language example above), when installing an ATC and a few special cases when it is desirable to edit a parameter with a Macro. A common example of this is M74, the Auto Squaring macro. In this macro Parameter 967 is edited by the M74 macro to un-pair and then re-pair the two motors on a moving gantry so independent movement can be achieved to align the two sides of the gantry to square.

Not only are Parameters defined values that represent many different features and settings for use in CNC12 but they can also be used in conjunction with the PLC program and Macros. With Macros we can read these parameters and also write to them as described below.

11.3.16 When would I use a Parameter instead of a Variable?

Typically one would use parameter values instead of user variables in cases where a parameter is already defined with the value we want to use and/or we want the operator to have the choice to change the value in software.

Parameters are similar to Non-volatile user variables where you can store values between jobs or through power cycles, however they can be also edited by the operator easily in the parameters menu in software. F1 Setup, F3 Config, F3 Param.

The Acorn version of CNC12 makes use of several new parameters that are not in the CNC12 operators manual. A description of these special Acorn parameters are listed below. (A description of most of them are also contained in the beginning of the Acorn PLC program source file (.SRC) in the comments section as well).

Parameters 700 to 799 are allocated as special use parameters reserved specifically for the CNC12 user/integrator to use in a custom application. This reservation of P700–799 allows the user/integrator to create their own custom parameters in both Macro and PLC programs and then the user/integrator can be sure that P700–799 will not be used by CNC12 or a Centroid Standard PLC program in the future, this ensure that any custom PLC or Macro made today using P700–799 will not have a conflict with future CNC12 updates. So, if you need to make a custom parameter use any one in the P700 to 799 range.

11.3.17 How to read a parameter value

In reference to the system variable chart in the mill or lathe manual, we can find that 9000–9999 are the system variables for parameters 0–999. These are the same parameters that you can find in the parameters menu within CNC12 (F1 setup, F3 config, F3 Params). In Macros, it is very simple to read from the system variable parameter values. We will typically use parameter values instead of user variables in cases where a parameter is already defined with the value we want to use and/or we want the operator to have the choice to change the value in software. We simply need to assign a variable to a parameter, for example:

```
#101 = #9710 ; Sets user variable 101 to the value of parameter 710
```

System Variables do not always need to be assigned to a user variable, most often they can be used directly in the logic.

```
G92 Z[#9710] ; Set WCS Z position to value of Parameter 710
```

11.3.18 How to write a parameter value

Referencing the system variables chart in the manual you can read the system variables 9000–9999 (parameters 0–999) as indicated by the R in the R\W column, but we cannot directly write to them as indicated by no “W” in the R\W column. However using the G10 function, a macro can write to these system variables. An example:

```
#103 = 5          ; Sets User variable to value of 5
G10 P710 R[#103] ; Sets Parameter 710 to the value of #103
```

Typically in macros we will reference values via system variables, however functions such as G10 only requires the parameter number, and does not use the system variable that corresponds to said parameter. Trying to write to a read only variable will result in an error being displayed in the software when trying to run the macro, examples of some cases where this error will appear.

```
NOT ALLOWED:
#9710 = #101 ; This cannot be done, #9710 cannot be directly written to, only
   read.
#4120 = 5    ; This cannot be done, #4120 is a read only variable, 4120 value
   is set/changed by CNC12
```

To reiterate, the above is strictly not allowed in this format, but parameter 710 can be written to using the G10 command.

```
G10 P710 R3 ; Sets parameter 710 to the value of 3
```

Similarly, #4120 can be written to by using the T command,

```
T10 ; Sets current tool to tool 10, Really this sets #4120 to the value of 10.
```

There are Parameters that are considered "machine setup" related and are Read only (a macro cannot write to them).

Most are documented in the CNC12 operators manual. Below is an example of a parameter that can not be written to.

```
G10 P71 R5 ; Will display "704 G10 error:invalid P" in software
           ; Parameter 71 is a "machine setup parameter and is unable to be
              changed by G10.
```

"Machine Setup" parameters are read only, they can only be written to by either the Wizard or the parameter menu in CNC12: F1(Setup), F3(Config), F3(Parms).

These parameters can still be read by the macro like other parameters.

```
#100 = #9071          ; Set user variable #100 to value of parameter 71
G92 Z[0+ABS[#9071]]   ; Set Z position to 0 + detector height stored in
   parameter 71
G92 Z[.75+ABS[#100]] ; Set Z position to .75 + value stored in user variable
   100
```

11.3.19 Lathe Vs Mill Software

Always reference the correct operators manual (Mill or Lathe) for each version of CNC12 you are working with. The examples in this guide are written with the Mill CNC12 in mind and the same general rules apply to both versions of software. However, G and M codes can be different in either versions of software so, it is important to reference the operators manual for the correct use of G-code. For example, the G10 command is different in functionality in the Lathe CNC12.

Lathe CNC12

```
G10 P5 Z-2         ; Sets tool #5 Z-axis offset to -2 in Offset Library
G10 P1710 R[#103] ; Sets Parameter 710 to the value of #103
```

G10 in the lathe software sets the tool offsets but also writes to parameters based on the P number. If the P number is a number between 1 and 99 (Pnn, where "nn" is the number) then the corresponding tool offset is set. So "G10 P3 X2"would set the tool X-axis offset of tool 3 to value of 2. If the P number is a number between 1000 and 1999 (P1nnn, where "nnn" is the parameter number) then the parameter is written to. So "G10 P1710 R2" would set the value of 2 to parameter 710.

11.3.20 Difference between = and ==

== is used to compare two variables while = is used to write to a variable or in other words assign to the variable.

For example:

```
#100 = 1              ; writes the value of 1 to the variable #100
#101 = 2              ; writes the value of 2 to the variable #101
IF #100 == #101 .... ; Compare Variable #100 and #101, if they are equal than
   the expression is true.
```

Alternatively

```
IF #100 eq #101 ....
```

For more information on expressions, symbols, and letters used, reference Chapter 11.

11.3.21 Using “IF THEN ELSE”

```
IF <Expression> THEN <Execute if True> ELSE <Execute if False>
```

Macro Programming using IF THEN or IF THEN ELSE, are commonly used in macros that require different actions to be taken based on expression. The expression statement can be a variety of logic, for example it can look at inputs, look at expressions, and more.

Below are a few IF THEN, and IF THEN ELSE examples:

```
IF #50005 THEN GOTO 200                              ; If Input 5 is closed (Green)
    then goto block 200
IF [#101 == 1 || #101 + #102 == 3] THEN GOTO 200 ; If #101 = 1 or #101+#102 = 3
    then goto 200
```

The Execution statement can be a variety of different commands, most commonly GOTO is used as the execution.

```
IF #50005 THEN M5                            ; If Input 5 is closed then execute M5
    (Stops spindle)
#100 = 0                                     ; Sets user variable #100 to value of
   0.
IF #50005 THEN M225 #100 "Input 5 is on" ; If Input 5 is closed then display
   message "Input 5 is on"
```

An ELSE is used if a separate execution is desired on a false expression.

```
IF #50005 THEN M5 ELSE GOTO 200                                         ; If
   Input 5 is closed then execute M5, If Input 5 is open then goto block 200
#100 = 0                                                                ; Sets
    user variable #100 to value of 0.
IF #50005 THEN M225 #100 "Input 5 is on" ELSE M225 #100 "Input 5 is Off" ; If
   input 5 Is closed then display "Input 5 is on", Otherwise display "Input 5
   is off"
```

IF statements when looking at inputs are not inverted if the operator inverts the input in the software (Alt-I screen then Ctrl-Alt-I on an input). The IF statement is still looking if the said input is On (Green) or Off (Red) regardless of inversion. Refer to Section 11.4.2 for more information.

11.3.22 Block Numbers and GOTO Command

Block Numbers are used to identify program lines and are designated by an N# (# being a number). These blocks also act as destinations for the GOTO Command. Example:

```
N100                     ; Input 5 is a tool sensor in the spindle
IF #50005 THEN GOTO 500 ; IF Input 5 is on, then GOTO Block 500 (N500)
M15                      ; Unclamp Tool
GOTO 1000                ; GOTO End of macro
N500                     ; Block 500
M16                      ; Clamp Tool
N1000                    ; End of Macro
```

In this example we set up a macro to detect input 5 and perform two different actions based on the state of the input. If the input was off, then the macro will perform an M15 then end the macro. If the input was on, then the macro will jump to block 500 and perform an M16 and end the macro. This logic that “skips part of the macro” to perform a different action for example, is used in ATC macros to skip the “put tool away” section of the macro if there is no tool in spindle when the macro is called.

11.3.23 How to use a macro to interact with an Output

Macros can be used to activate and deactivate outputs for a wide variety of uses. CNC12 has two built in M-codes just for the purpose of turning ON and OFF an output: M94 and M95 In order to use M94 and M95 with a particular output number that output number must be defined in the Acorn PLC program. Defining an Acorn output definition can be done one of two ways:

1. Use the Acorn Wizard and assign an output definition to the output number that you plan on using with the M94/M95.
2. Edit the Acorn PLC program source file and add in a customized output and compile the PLC program (see example of this in appendix A at the end of this document).

For the following examples, using the Acorn Wizard, we assign the output DustFootActivate to Output 3, and assign the input ToolisUnclamped to Input 2.

Example 1: Assign M61 to turn on Output 1. The Acorn Wizard has canned PLC Output definitions: OUTPUT1, OUTPUT2, OUTPUT3, etc.. that have preassigned macros: M61, M62, M63 etc... respectively. Using the Wizard assign OUTPUT1 to Output 1, “Write Settings to CNC control Configuration” follow the instructions and then open M61 in Notepad++.

Note: A semi colon “;” tells CNC12 that the line is a comment, remove semi colon for command to be run, add semi colon to beginning of line to be ignored.

```
;------------------------------------------------------------------------------
; Filename: mfunc61.mac
; Wizard OUTPUT1 M-code Macro: M61
; Description: User Customizable Macro
; Notes: Use Acorn Wizard i/o map to set Acorn Output 1 = to "OUTPUT1" then
  this macro (M61) will turn on that output
; Requires:
; Please see TB300 for tips on writing custom macros.
;------------------------------------------------------------------------------
IF #50010                        ; Prevent lookahead from parsing past here
IF #4201 || #4202 THEN GOTO 1000 ; Skip macro if graphing or searching
N100                             ; Insert your code between N100 and N1000
M94 /61                          ; Request OUTPUT1
N1000                            ; End of Macro
```

Now in MDI type “M61” then press Cycle Start and Output1 will turn on, press cycle cancel or enter M81 and cycle start and Output1 will turn off.

Example 2: Adding a delay after Output1 is turned on for 4 seconds to M61:

```
N100
M94 /61 ; Activate Output1
```

```
G4 P4   ; Wait 4 seconds. Then continue on
N1000
```

Example 3: Activate output when an input is activated:

```
N100
M94 /61                 ; Activate Output1
G4 P4                   ; Wait 4 seconds
IF #50002 THEN M94 /28 ; If Input 2 is closed then activate "DustFootActivate"
N1000
```

Example 4: Adding an input for the operator to choose to turn on or keep off RouterVacuumHoldDown Output during

```
N100
G94 /61 ; Request Output1
G4 P4 ; Wait 4 seconds
IF #50002 THEN M94 /28 ;If Input 2 is closed then activate "DustFootActivate"
N200
;Display message "Turn on RouterVacuumHolddown?"
; To Turn on Enter "1"
; To Turn off Enter "2"
M224 #100 "Turn on RouterVacuumHoldDown? \n To Turn on Enter 1 \n To Turn off
   Enter 2" #105
IF [#100 < 1 || #100 > 2] THEN GOTO 200 ; If operator inputs invalid number,
   repeat question.
IF #100 = 1 THEN M94 /5                 ; If operator inputs 1, then activate
   RouterVacuumHoldDown
                                        ; If operator input 2, then macro
                                          continues without activating output
N1000
```

Draft: June 12, 2023

Note: These M94/M95 values are found in v4.20+ stock Acorn PLC programs that can be utilized in any macro as is.

Also see Acorn Wizard Input and Output "canned" PLC functions and M codes list. https://www.centroidcnc.com/centroid_diy/downloads/acorn_documentation/acorn_wizard_input_output_plc_functions.pdf

```
SV_M94_M95_1  ; M94/M95 Turn on/off Spindle On Clockwise
SV_M94_M95_2  ; M94/M95 Turn on/off Spindle On Counter Clockwise
SV_M94_M95_3  ; M94/M95 Turn on/off Flood or RouterDustCollection
SV_M94_M95_4  ; M94/M95 Turn on/off OpenChuck, Turn off CloseChuck
SV_M94_M95_5  ; M94/M95 Turn on/off Mist or RouterVacuumHoldDown
SV_M94_M95_6  ; M94/M95 Turn on/off CloseChuck, Turns off OpenChuck
SV_M94_M95_8  ; M94/M95 Turn on/off LubePump Manually
SV_M94_M95_10 ; M94/M95 Turn on/off TurnClampOn
SV_M94_M95_13 ; M94/M95 Turn on/off Cutoff
SV_M94_M95_15 ; M94/M95 Turn on/off ToolUnclamp
SV_M94_M95_19 ; M94/M95 Turn on/off OrientSpindle
SV_M94_M95_22 ; M94/M95 Turn on/off PartChute
SV_M94_M95_27 ; M94/M95 Turn on/off VacuumOn
SV_M94_M95_28 ; M94/M95 Turn on/off DustFootActivate
SV_M94_M95_29 ; M94/M95 Turn on/off LaserAlignActivate
SV_M94_M95_30 ; M94/M95 Turn on/off PopUpPins
SV_M94_M95_31 ; M94/M95 Turn on/off SpindleCooling
```

```
SV_M94_M95_32 ; M94/M95 Turn on/off TailStockInOut
SV_M94_M95_35 ; M94/M95 Turn on/off DustCollectionOn
SV_M94_M95_41 ; M94/M95 Turn on/off Spindle Low Range
SV_M94_M95_42 ; M94/M95 Turn on/off Spindle Medium Range
SV_M94_M95_43 ; M94/M95 Turn on/off Spindle High Range
SV_M94_M95_61 ; M94/M95 Turn on/off OUTPUT1
SV_M94_M95_62 ; M94/M95 Turn on/off OUTPUT2
SV_M94_M95_63 ; M94/M95 Turn on/off OUTPUT3
SV_M94_M95_64 ; M94/M95 Turn on/off OUTPUT4
SV_M94_M95_65 ; M94/M95 Turn on/off OUTPUT5
SV_M94_M95_66 ; M94/M95 Turn on/off OUTPUT6
SV_M94_M95_67 ; M94/M95 Turn on/off OUTPUT7
SV_M94_M95_68 ; M94/M95 Turn on/off OUTPUT8
```

These M94/M95 values do not always need to be used in a macro, Running an M94 /15 command in MDI or on a line in a job file for example will activate the logic to unclamp tool. Putting these M94 and M95 commands in macros is usually preferred as additional desired logic is accompanied with these commands such as M101/M100 in most cases.

11.3.24 Subprograms M98/M99

Macros can be used to call additional subprograms and can have subprograms embedded into the macro itself. Embedded subprograms is a common technique used for Rack Mounted Automatic Tool Change (ATC) macros. Embedded subprograms within the macro are noted by beginning with an O ("O" as in Oscar) followed by a 9100 to 9999 number. These particular subprograms are special in that they self extract themselves out of the macro and return to the macro when finished. An example of an embedded subprogram:

```
; This macro is for a tool rack tool changer with 2 tool positions with no
   forks (drops tool in pot).
; Subprograms are used to call the position of the tool requested in the X and
   Y coordinates, "O9101" and "O9102".
; #4120 = Value of Tool Number requested, this is read only, CNC12 sets this
   value depending tool number requested by pro gram or user.
;#100 = -5               ; Z height of the tool rack
O9101                    ; Embedded Self Extracting SubProgram 9101 that defines
    the tool pocket location
                         ; Enter the X coordinate of Tool #1
G53 X100 Y10             ; X and Y Coordinate of Tool # 1
M99                      ; End of subprogram return to main program

O9102                    ; Embedded Self Extracting SubProgram 9102 that defines
    the tool pocket location
                         ; Enter the X coordinate of Tool #2
G53 X200 Y10             ; X and Y Coordinate of Tool # 2
M99                      ; End of subprogram return to main program

G90 M98 P[#4120 + 9100] ; Tool Number Requested + 9100, then calls subprogram
   to move to X and Y coordinates
G53 Z[#100]
```

Refer to Section 11.2.6 and Section 13.27.

11.3.25 Creating Custom Operator Prompt Messages that appear when running the Macro to prompt the Operator with information or action needed

CNC12 can display a custom message by using a file called cncxmsg.txt. For example: This file can be used with macros to display to an operator prompt messages when the macro is waiting on an input with the m101 and m100 commands. To customize a message create a “cncxmsg.txt” file and save it in the cncm directory for mill or the cnct directory for lathe. Once created, if we wanted a custom message to appear with a M100 or M101 command in the macro for input 6, we will input the following into the cncxmsg.txt file.

```
INP6
"My Custom Message for Input 6"
OUT1
"My Custom Message for Output 1"
MEM2
"My Custom Message for Memory 2"
```

Format for cncxmsg.txt is first line state the Input, Output, or Memory and then on the next line the custom message in quotations. Outputs and memory bits can be given their own messages using OUT# for outputs and MEM# for Memory bits (# is the number of output or memory bit).

To use this custom messages created in the cncxmsg.txt file all you have to do is use the M100 or M101 in the macro with the corresponding Input, Output, or Memory number. For example.

```
M101 /50006 ; Will now display the custom message from cncxmsg.txt instead of
   default "waiting for input #6 (M101)"
M101 /60001 ; Will now display the custom message from cncxmsg.txt instead of
   default "waiting for output #1 (M101)"
M101 /70002 ; Will now display the custom message from cncxmsg.txt instead of
   default "waiting for memory #2 (M101)"
```


11.3.26 When Skip Graphing, Skip Searching and Stop Look ahead is needed.

You will commonly see the following lines in Centroid macros.

```
IF #4202 || #4201 THEN GOTO 500 ; Skip if CNC12 is graphing Gcode or searching
   for Gcode
N500                            ; End of Macro
```

and

```
IF #50001 ; Force look ahead to stop processing
```

The first line is used to skip the macro if the software is in graphing or search mode. Search and graphing modes process through both the job and macro files. If there is code in a macro that could cause a loop, it is possible for the search or graph to get “hanged up” at this loop and would no longer continue through the rest of the graph or search. Such loops exist for example in an M19 Spindle orient macro, which may have a loop that tells the operator to turn on auto spindle. This loop can cause the graph or search to get hanged up on the M19 line of the job file, or even the m19 line in an M6 macro.

The CNC12 system variable #4201 is a value of 0 when running a job, or a value of 1 when performing a graph preview. The CNC12 system variable #4202 is a value of 0 when the CNC12 is running G-codes normally or a value of 1 or greater when the CNC12 is not running G-codes normally for example when doing a search. For the case of the G-codes not running normally, the commands are not being sent to the machine but rather CNC12 is running through the G-code to determine the correct state of the machine up to the point that is being searched. If either one of these values are a non-zero then the expression is true and we GOTO the last line of the macro ending it.

The second piece of logic IF #50001 is used to stop the CNC12 G-code look-ahead from processing past that point. This is used mainly when you need to ensure a value or an action is not processed by the G-code look-ahead prior to when the program itself reaches that point in the macro. The CNC12 actively "looks ahead" in the job file and macro to anticipate 16changes, and it will change values before the actual program reaches this point. Let us look at the following example: using the "put away tool" section of the Rack Mount ATC Macro example, which is found later in this document.

```
;--Put Tool Away Block 100
N100
IF #150 == 0 THEN GOTO 200
G90 M98 P[#150 + 9100] ; Go to X Put away Location
G53 Z[#100]            ; Move to Z Tool Rack Height
G53 G1 Y[#102] L10     ; Put Tool in Tool Rack
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
M63                    ; Turn on Output 3 (Open Valve)
G4 P0.5                ; Dwell 0.5 Seconds
M100 /50006            ; Check if Tool Released
G53 Z[#101]            ; Move to Z Tool Height
M83                    ; Turn off Output 3 (Close Valve)
G28 G91 Z0             ; Move to Z Clearance Height
IF #50001              ; Force look ahead to stop processing
#150 = 0               ; Set No Tool in Spindle
IF #4120 == 0 THEN GOTO 500
```

For example, if an issue happened and the operator had to press E-Stop between the lines G53 and M63 represented by the ~~~~ line above. Since we had `IF #50001` before the line `#150 = 0` there were no changes to the Variable #150 as CNC12's program execution had not reached that line of the macro yet. However, if "`IF #50001`" was not present before the line `#150 = 0`, it is possible that even though the operator had pressed E-Stop before the program execution reached the `#150 = 0` line. CNC12's G-code look-ahead may have already assigned the value 0 to variable #150. So the next time the operator runs this macro, the macro will now think #150 is equal to zero even though there is still a tool in the spindle.

It is important to use `IF #50001` to stop CNC12's G-code program look-ahead from processing past a point when you want to make sure the program execution actually made it to that point before changing the variable. In this case we want to make sure that the tool has actually been put away before telling the control that there is no longer a tool in the spindle `#150 = 0`. The `IF #50001` command does not need to be used all the time, only when it is desirable to stop CNC12's G-code look-ahead from going past a point until that point is reached in the G-code program.

You may have noticed that #50001 is the system variable for input 1. The line `IF #50001` means IF Input 1 is closed, then do nothing. Any CNC code that is conditional on a PLC bit is intended to be tested at the time the code actually runs. Therefore, CNC12's G-code program look ahead waits at the line `IF #50001` until the execution has actually caught up to this line. Any PLC bit will work to stop CNC12's G-code program look-ahead, however #50001 is the common convention.

11.3.27 Skip Graphing Notes

- There is no need to skip around M0, M1, M100 or M101. CNC12 knows, during search and graph, that those commands do not do anything interesting (that is, anything that affects tool movement or CNC modal values).
- When Skipping is not desirable.

Often times it is desired to save control information, this can be done via a parameter or a user variable as discussed previously. However, we may not always want to skip updating a variable during a search or graph. Lets look at two examples, the first is rather familiar, we will update the “tool in spindle” for a Rack Mount Tool ATC. The second example we will be storing the desired RPM of a sub spindle to a parameter. We choose a parameter instead of a non-volatile user variable for example 2 so the PLC can also retrieve this information. Assume for example 2 we will be writing different RPM values throughout a program, 3500 is just for example purposes.

Example 1:

```
#150 = #4120 ; Record requested tool as the new tool in spindle
N500         ; End of Macro
```

Example 2:

```
G10 P702 R3500 ; Store desired RPM of 3500 to parameter 702
M33            ; Subspindle Start request to PLC
N500           ; End of Macro
```

For both examples, it is desirable that we stop CNC12’s G-code Look-ahead, as we do not want to update the value of the RPM or the tool before CNC12’s G code execution catches up. Additionally, for example 1, we do not want to update the tool in spindle value unless a tool change has been completed. So we will add skip graph and search to Example 1. However, for example 2, we will want to skip when graphing, however we may want it to update the values if searching. Lets look at the examples again with the added code.

Example 1 with Stop Look ahead and Skip Graphing and Searching:

```
IF #50001                                     ; Stop software look ahead
IF #4201 == 0 && #4202 == 0 THEN #150 = #4120 ; Record requested tool as new
   tool in spindle if not graphing or searching
N500                                          ; End of Macro
```

This is different from previous examples, in this case we are simply not updating the value if graphing or searching instead of skipping the full macro.

For example 2, it is desirable to allow CNC12 to update the parameter as we search. If we do not, then we run the risk of running with an incorrect RPM if we wanted to start a job at a mid-point. (use CNC12’s Search feature to start at almost any point in a G code program. F4 Run, F2 Search and type in the line, block or tool number to start at.) Since we are no longer skipping the macro while searching, CNC12 will run through the program and update the parameter when ever it is called in the program. This will give us the correct state of desired RPM (Parameter 702) up to the point in which we searched for.

Example 2 with Stop Look ahead and Skip Graphing but allow Searching:

```
IF #50001                           ; Stop software look ahead
IF #4201 == 0 THEN G10 P702 R3500 ; Store desired RPM of 3500 to Parameter 702
   if not graphing
M33                                 ; Subspindle Start request to PLC
N500                                ; End of Macro
```

Bringing it all together, a Rack Mount ATC Macro Example: Four tool position rack mount ATC with integrated auto tool measurement touch off cycle.

```
;---------Tool Pocket Location Definition Section---------
O9101
```

```
; Edit the line below with the X coordinate of Tool #1 pocket location
G53 X100
M99

O9102
; Edit the line below with the X coordinate of Tool #2 pocket location
G53 X200
M99

O9103
; Edit the line below with the X coordinate of Tool #3 pocket location
G53 X300
M99

O9104
; Edit line below with the X coordinate of Tool #4 pocket location
G53 X400
M99
;---------End Tool Location Section---------

; Variable Definitions:
              ; #150 = Tool currently in the spindle
#100 = -90    ; Z height of tool rack
#101 = -60    ; Z height of Tool (Position where spindle is right above the
   tool)
#102 = -8     ; Y Tool Location
#103 = -70    ; Y Clearance Location
#104 = #50002 ; Tool Touch Tripped Input
              ; #9014 ;Fast Probe Rate Parameter 14
              ; #9015 ;Slow Probe Rate Parameter 15
              ; #9071 ;Tool Touch Off Height, Parameter 71

IF [[#4120] == [#150]] || #4202 || #4201 THEN GOTO 500 ;skip if graphing or
   searching or if at the same tool
;--Prepare for Tool Change
M5              ; Stop Spindle
G28 G91 Z0      ; Move to Z Clearance Height
G53 G90 Y[#103] ; Move to Y Clearance Location
G4 P1           ; Dwell for 1 Seconds

;--Check if Tool in Spindle
N50
IF !#50006 THEN GOTO 200 ELSE GOTO 100 ; If Tool not in spindle skip Put Away

;--Put Tool Away Block 100
N100
IF #150 == 0 THEN GOTO 200
G90 M98 P[#150 + 9100] ; Go to X Put away Location
G53 Z[#100]            ; Move to Z Tool Rack Height
G53 G1 Y[#102] L10     ; Put Tool in Tool Rack
M63                    ; Turn on Output 3 (Open Valve)
G4 P0.5                ; Dwell 0.5 Seconds
M100 /50006            ; Check if Tool Released
```

```
G53 Z[#101]            ; Move to Z Tool Height
M83                    ; Turn off Output 3 (Close Valve)
G28 G91 Z0             ; Move to Z Clearance Height
IF #50001              ; Force look ahead to stop processing
#150 = 0               ; Set No Tool in Spindle
IF #4120 == 0 THEN GOTO 500

;--Get Requested Tool Block 200
N200
G90 M98 P[#4120 + 9100] ; Go to X Putaway Location
G53 Y[#102]             ; Move to Y Tool Location
G53 Z[#101]             ; Move to Z Tool Height
M63                     ; Turn on Output 3 (Open Valve)
M101 /50007             ; Check for Input 7 (Valve Open)
G53 Z[#100]             ; Move to Z Tool Rack Height
M83                     ; Turn off Output 3 (Close Valve)
G4 P0.5                 ; Dwell for 0.5 Seconds
M101 /50006             ; Check for Tool Clamped
IF #50001               ; Force lookahead to stop processing
#150 = #4120            ; Set Tool to Requested Tool
G53 Y[#103] L10         ; Move to Y Clearance Location
G28 G91 Z0              ; Move to Z Clearance Height
GOTO 300

;--Tool Touch Off Cycle Block 300
N300
G30 G91 X0                ; Move to X Tool Touch Off Location
G30 G91 X0Y0Z0            ; Move to Tool Touch Off Reference Location
M115 /Z P[#104] F[#9014]  ; Move at fast probing rate until TT1 detected
M116 /Z P-[#104] F[#9015] ; Retract at slow probing rate until TT1 clears
M115 /Z P[#104] F[#9015]  ; Move at slow probing rate until TT1 detected
G92 Z[0+ABS[#9071]]       ; Set Z position to 0 + detector height stored in
   parameter 71
G4 P1                     ; Wait 1 second
G28 G91 Z0                ; Move to Z Clearance Height
G90
GOTO 500

N500 ; End of program
```


11.3.28 Rack Mount Macro Notes

- The Tool Touch Off device should be setup with the Acorn Wizard and functionality tested before running a macro or tool touch off cycle.
- TT input defined and correct input defined in the macro, user variable #104
- TT subtracts height of TT based on parameter 71. If not desired, edit the macro and remove `ABS[#9071]` from the "`G92 Z[0+ABS[#9071]]`" line near the end of the macro.
- Machine tool specific Tool Bin locations of each tool must be manually entered into the macro.
- User variables #100–#103 will most likely need adjusted for each rack mount ATC System.

- It is possible to add more tool slots by adding more Subprograms O910# at the beginning of the macro.

11.3.29 Frequently Asked Questions

1.) IF an input is Inverted in software, are M100/M101 or IF #5000# expressions Inverted?

These are not inverted, they are still looking at the state of the input (On or off, or in other words Green or Red in the I/O Diagnostic screen Alt-I). For example, In the case Input 1 is off and not inverted, (shows up red)

```
IF #50001 THEN...        This will turn a false, since the input is off.
M101 /50001              This will be false with message "Waiting for Input 1"
M100 /50001              This will be true, Macro will continue
```

in the case Input 1 is off and inverted (shows up green)

```
IF #50001 THEN...        This will turn a true in this case, since the input is
   on.
M101 /50001              This will be true, Macro will continue
M100 /50001              This will be false with message "Waiting for Input 1"
```

in the case input 1 is on and not inverted, (shows up green)

```
IF #50001 THEN...        This will turn a true in this case as well, since the
   input is on.
M101 /50001              This will be true, Macro will continue
M100 /50001              This will be false with message "Waiting for Input 1"
```

in the case input 1 is on and inverted, (shows up red)

```
IF #50001 THEN...        This will turn a false in this case, the input is off
M101 /50001              This will be false with message "Waiting for Input 1"
M100 /50001              This will be true, Macro will continue
```

2.) When to use F or L for Feed Rate?

Use F to set the feed rate for non-rapid moves such as G1, while L is used in some G-codes that would otherwise use the Rapid rate. For example, the L command when used with a G53 (rapid move in machine coordinates) will make the G53 move at the user specified feedrate rather than the Rapid rate.

Note: The L value also does not override the feed rate for other moves, Look at the two cases below.

Case 1:

```
G1 X1 F100  ; Moves X at 100 units/minute
G1 Y2       ; Moves Y at 100 units/minute
G53 Z-5 L20 ; Moves Z at 20 units/minute
G1 Y0       ; Moves Y at 20 units/minute
```

Case 2:

```
G1 X1 F100  ; Moves X at 100 units/minute
G1 Y2       ; Moves Y at 100 units/minute
```

```
G53 Z-5 F20 ; Moves Z at Max Feed units/minute. The F20 does not effect G53 and
    is ignored as it defaults to rapid rate
            ; Note: G53 will move at rapid rate unless given L value to command
                the velocity desired
G1 Y0       ; Moves Y at 100 units/minute
```

G-codes that uses L for feedrate is G28, G30, G30 P3 and G30 P4 and G53. The L allows you to control the rate in which these G-codes will move the machine tool. If no L command is specified these commands move at the machines Rapid Rate.

Note: L is not always used for feedrate, for example G65 uses L for the repeat value and not for feed rate.

3.) Where does variable #150 come from in ATC Macro?

Variable #150 is a non-volatile variable, meaning it retains its value after the macro ends and through power cycles. Variable #150 is needed in tool rack mount ATCs to keep track of which tool is currently in the spindle. When calling a tool change, the macro uses the value of #150 to determine where to put the current tool back before trying to pick up the next tool.

We use #150 simply due to it being the first non-volatile number, you can use any non-volatile variable for this or even a parameter (however this can cause issues if the operator changes the parameter manually). At the end of the macro we set #150 to equal the requested tool #4120 after the tool change has been completed. This updates the value #150 with the tool currently in spindle, for use when the tool change is called again.

The macro its self is what sets and reads variable #150.

4.) How do I create a Probing macro?

There are a fair number of stock CNC12 probing macros contained in the cncm\system folder.

artic_wall_follow.cnc	2/14/2019 3:17 PM	CNC File	36 KB
atan2.cnc	2/14/2019 3:17 PM	CNC File	1 KB
correct_position.cnc	2/14/2019 3:17 PM	CNC File	2 KB
correct_position_a.cnc	2/14/2019 3:17 PM	CNC File	2 KB
correct_position_b.cnc	2/14/2019 3:17 PM	CNC File	2 KB
grid_digitize.cnc	2/14/2019 3:17 PM	CNC File	6 KB
move_primitive.cnc	2/14/2019 3:17 PM	CNC File	1 KB
park.mac	5/8/2019 11:32 AM	MAC File	3 KB
plcmacro1.mac	5/8/2019 11:32 AM	MAC File	1 KB
plcmacro2.mac	5/8/2019 11:32 AM	MAC File	1 KB
plcmacro3.mac	5/8/2019 11:32 AM	MAC File	1 KB
plcmacro4.mac	5/8/2019 11:32 AM	MAC File	1 KB
probe_angle.cnc	2/14/2019 3:17 PM	CNC File	4 KB
probe_bore.cnc	2/14/2019 3:17 PM	CNC File	1 KB
probe_boss.cnc	2/14/2019 3:17 PM	CNC File	3 KB
probe_center_inside.cnc	2/14/2019 3:17 PM	CNC File	2 KB
probe_center_outside.cnc	2/14/2019 3:17 PM	CNC File	3 KB
probe_clearance_traverse.cnc	2/14/2019 3:17 PM	CNC File	2 KB
probe_clearance_traverse_across_and_do...	2/14/2019 3:17 PM	CNC File	4 KB
probe_comp_two_points_on_a_line.cnc	2/14/2019 3:17 PM	CNC File	2 KB
probe_cycles_select.cnc	2/14/2019 3:17 PM	CNC File	10 KB
probe_get_constants.cnc	2/14/2019 3:17 PM	CNC File	2 KB
probe_get_modals.cnc	2/14/2019 3:17 PM	CNC File	2 KB
probe_inside_corner.cnc	2/14/2019 3:17 PM	CNC File	3 KB
probe_limit_position.cnc	2/14/2019 3:17 PM	CNC File	2 KB
probe_move.cnc	2/14/2019 3:17 PM	CNC File	9 KB
probe_move_to_intersection.cnc	2/14/2019 3:17 PM	CNC File	2 KB
probe_move_to_surface.cnc	2/14/2019 3:17 PM	CNC File	2 KB
probe_outside_corner.cnc	2/14/2019 3:17 PM	CNC File	6 KB
probe_protected_move.cnc	2/14/2019 3:17 PM	CNC File	3 KB

We have begun to “breakout” as many CNC12 built in probing cycles into editable macros. Copying and then editing these or just opening and learning from them is a great place to start if you are interested in learning more about creating your own custom probing macros.

That being said here is an introduction to controlling a touch probe with a macro.

Unless otherwise noted, the methods and examples below are for current CNC12 Mill software.

5.) How do I control a touch probe?

Use the M115, M116, M125 and M126 codes to move a touch probe in custom probing cycles.

Whenever possible, specify a bounding position, even with M115 and M116 codes.

Note that the positions you specify with these codes are either absolute positions or incremental distance, depending on the current G90/G91 state.

For example:

```
G90
M105/X2.0/Y3.0 P15 F20
```

will move towards an absolute position of X2 Y3, at 20 in/min, until the switch on INP15 closes.

In contrast:

```
G91
M105/X2.0/Y3.0 P15 F20
```

will move in a direction of +2" along X, +3" along Y, at 20 in/min, until the switch on INP15 closes.

Also note that M115 and M116 are interchangeable (as are M125 and M126) when bounding positions are provided, because the direction to the bounding position determines the direction of movement.

Use M115 and M116 when you intend to locate a surface with the probe. The bounding position, if given, should be beyond the farthest likely position of the surface. With M115 and M116, if the movement proceeds all the way to the bounding position without triggering the input (tripping the probe), then the cycle will be canceled with an Error.

Use M125 or M126 for positioning moves during which no probe contact is expected.

Like M115 and M116, M125 and M126 will stop if the input is triggered (the probe is tripped); but M125 and M126 will then cancel the cycle with an error. This protects against probe breakage in case of an unexpected obstacle.

M115, M116, M125 and M126 are most commonly used with a touch probe, but they can be used with any switch input, or other PLC bit.

The P parameter used with these codes is either positive or negative, depending on whether you are watching for an open switch to close, or watching for a closed switch to open.

6.) What are the differences between M105 and M115? (Switch Sense for M1x5 and M1x6 Codes)

Note: The M105 and M106 codes, which are similar to M115 and M116, may also interpret the sign of the P parameter but differently. The behavior for positive and negative P values with the M105/106 and M115/116 is as follows:

	CNC11/CNC12	
Code	+P	-P
M105/M106	Move until open (1 -> 0)	Move until close (0 -> 1)
M115/M116/M125/M126	Move until close (0 -> 1)	Move until open (1 -> 0)

7.) How do I create a custom home program?

The M105 and M106 codes are useful for axis homing, in situations where the normal M91 and M92 codes are not well suited. For example, suppose we have a mill X axis which we want to quickly home near the center of travel, regardless of where the axis may be be sitting when the machine is powered up.

We install a dedicated home switch, which will be a normally-open proximity sensor that detects a steel rib that runs from near the mid-point, all the way to the plus limit of travel. We wire it to INP7 on the Acorn board. Therefore, we expect INP7 to be open when we are in the minus half of travel, and closed when we are in the plus half.

Our homing macro can contain the following lines for the X axis:

```
M105/X P7 F100  ; move X- until INP7 is open
M106/X P-7 F100 ; move X+ until INP7 is closed
M91/X           ; run the normal homing sequence
```

The first line will quickly move us to the minus side of the edge, if we were on the plus side to begin with. If we are already on the minus side (if the sensor is already open) then there will be no movement. The second line will quickly move us (back) to the plus side of the edge, where the sensor is closed.

The third line will start the usual minus homing sequence: slow-jog speed minus until the switch opens; slow jog plus until the switch closes again; and slow movement further plus until the encoder index pulse comes around. The complete sequence will end at the first encoder index pulse to the plus (sensor-closed) side of the edge.

This same general approach could be used with a dedicated home switch that is at one end of travel as well, as long as it trips prior to the limit switch.

This approach cannot be used with a home switch that is also a limit, because M105 and M106 will stop with an Error if a designated limit switch is tripped during the movement. Only M91 and M92 are allowed a one-time pass on tripping a limit switch (and then only for the limit switch they are looking for).

If you do home to a dedicated home switch, separate from the limit switch, it is highly desirable that the switch be set up with an extended ramp, so that the home switch remains tripped from its initial trip point, all the way out to the over-travel limit. Otherwise – if it is physically possible to be inside the limit switches, on either side of the home switch, with the home switch not tripped – then the homing macro cannot know whether to move plus or minus when seeking the home switch.

You cannot reliably use M115 and M116 in homing sequences, because those codes always attempt to apply software travel limits. They do not recognize that the software limits are not yet valid when machine home has not yet been set, and so will apply invalid restrictions to axis movement.

M105 and M106 do not apply software travel limits, even if the machine has been homed.

8.) How do I use encoder marker pulse relay output from an axis drive for accurate day to day homing? (aka ZRi homing.)

ZRi homing is a nice feature to implement on an Acorn CNC control system. (Zero Reference Input) ZRi facilitates accurate day to day homing above and beyond the accuracy achieved by home switches alone. While not that critical on a Mill or Router where resetting the WCS is quite easy and often done anyways, ZRi homing is very nice to have on a lathe as a time saving feature so that the Center line of the spindle (X0) is retained after a power cycle of the CNC control. ZRi requires a Servo drive that is equipped with this feature (this feature is often called differently with each drive mfg) such as the Estuns that Centroid sells and DMM. These drives have an output that closes when the marker pulse is detected (there is one marker pulse per revolution on the encoder). That ZRi output is connected to an Acorn input and then the CNC12 home macro uses the home switches to “get in the ballpark” and then uses the ZRi input to home the machine tool very accurately. Here are some common ZRi home program macros. ZRi inputs can be wired to individual inputs or all wired into one Acorn inputs. Please reference the Acorn CNC hookup schematics for Estun and DMM for more info on wiring. https://www.centroidcnc.com/centroid_diy/downloads/acorn_schematics/centroid_acorn_hookup_schematics.zip

Typical Lathe ZRi Example with ZRi inputs wired to same input:

```
M92 /Z L1       ; Move to Z+ home switch, back off until it clears.
M105 /Z P-4 F3 ; Move Z minus at 3 ipm until input 4 closes
M26 /Z          ; Set Z home here
G91 Z-.010 F25 ; Incremental move to clear the Z axis ZRi input
M91 /X L1       ; Move to X- home switch, back off until it clears.
M105 /X P-4 F3 ; Move X plus at 3ipm until input 3 closes
M26 /X          ; Set X home here
```

Typical Router ZRi Example (Gantry (Y) moves in minus direction) with ZRi inputs wired to individual inputs:

```
M92 /Z L1       ; Move to Z+ limit switch, back off until it clears.
M105 /Z P-4 F1 ; Move Z minus at 1ipm until input 4 closes
M26 /Z          ; Set Z home here
;
M92 /Y L1       ; Move to Y+ limit switch, back off until it clears.
M105 /Y P-3 F1 ; Move Y minus at 1ipm until input 3 closes
M26 /Y          ; Set Y home here
;
M91 /X L1       ; Move to X- limit switch, back off until it clears.
M106 /X P-2 F1 ; Move X plus at 1ipm until input 2 closes
M26 /X          ; Set X home here
```

Typical Mill ZRi Example (Y moves in plus direction table toward you) with ZRi inputs wired to one input:

```
M92 /Z L1       ; Move to Z+ limit switch, back off until it clears.
M105 /Z P-4 F1 ; Move Z minus at 1ipm until input 4 closes
M26 /Z          ; Set Z home here
G91 Z.010 F25   ; Incremental move to clear the Z axis ZRi input
M92 /Y L1       ; Move to Y+ limit switch, back off until it clears.
M105 /Y P-4 F1 ; Move Y Plus at 1ipm until input 4 closes
M26 /Y          ; Set Y home here
G91 Y.010 F25   ; Incremental move to clear the Y axis ZRi input
M91 /X L1       ; Move to X- limit switch, back off until it clears.
M106 /X P-4 F1 ; Move X plus at 1ipm until input 4 closes
M26 /X          ; Set X home here
```

Draft: June 12, 2023

9.) How can I run a macro and step through it line by line?

You can "step through" a macro by using "Single Block" mode (the Button on VCP beside cycle cancel, Section 2.6, or **F55** in Run menu Section 3.4) and adding 2 to parameter 10. (this is a bit-wise parameter. . . So, if the value is currently 1, set to 3 for example, see FAQ #12 below for more on bit-wise parameters). This will make CNC12 run in single block mode with a Macro. This will let you see line by line what the macro will do and ask you to press cycle start to continue for each line in the Macro (and G-code program)

Alternatively you could insert a M0 (Chapter 13) that would cause CNC12 to stop and wait for Cycle Start before continuing to the next line, this is useful when debugging a macro and you only want CNC12 to stop at a single point or a few points in the macro for debug purposes.

10.) When I open cncm (or cnct) there are missing macros for some of the M codes where are they?

Certain basic M codes have their default behavior built right into CNC12. In these cases the corresponding mfuncXX.mac is not present in the cncm or cnct direction, the mfuncXX.mac file is only needed for these M codes if you wish to have that M code function differently than the CNC12 default action for that M code! For instance, the file mfunc6.mac that corresponds to M6 Tool Change is missing from the cncm directory. CNC12 has a built in Tool Change (M6) macro that is the default action for a tool change. If there is no mfunc6.mac present in the cncm directory then CNC12 uses the default tool change action, the default behavior of M6 is described in the Operators manuals. Section 13.9.

Lathe tool change macro is name differently than the mill, A Lathe toolchange macro is "cnctch.mac"

11.) How do I write a macro for a Lathe Automatic Tool Changer Turret with Acorn?

Working examples of common Lathe Tool Changer Turrets are found on the Acorn Tech support forum. See Zip file download at the top of this page.

https://centroidcncforum.com/viewtopic.php?f=63&t=1340#p7671

12.) What is a "bitwise" Parameter?

Most Acorn users do not have to worry about bit wise parameters as the Acorn Wizard takes care of setting the bit wise parameters needed for Acorn CNC functionality based on the CNC setup selections made by the user in the Wizard. That being said power users that are interested in customization should be familiar with the concept.

Many CNC12 system configuration parameters are know as bit wise parameters. Bit wise parameters allow the selection of several options that can be grouped together at once.

For example: Parameter 11 "Macro M function handling/Probe Stop Handling" (Section 15.3.12) has 4 choices related to how you would like various aspects of a macro to work. Two useful options when writing and debugging a custom macro are:

"Display M & G-codes in M function macros?" Yes =1, No = 0

and "Step through M function macros in Block Mode?" Yes = 2, No = 0

Now to the uninitiated this seems confusing, what does this mean? Do I have to pick between the two? I want both the Macro G code to be displayed and I want it to step thru the Macro in Block mode. So, how does this work?

For a bit wise parameter you add together the values of the Functions you want.

So, for instance if I just want

1. "Display M & G-codes in M function macros?" set P11=1
2. if I just want "Step through M function macros in Block Mode?" set P11=2
3. if I want both "Display M & G-codes in M function macros?" AND "Step through M function macros in Block Mode?" set P11 =3! (add the values of the functions I want 1+2 = 3)

13.) How do I Round Decimal Values to Whole Numbers

It is often necessary to round fractional values to whole numbers (integers). For example, if you need to bring a rotary table, which may be at any angle, to the nearest whole turn (whole multiple of 360°), then you would want to round its position (converted to turns) to the nearest integer, then convert back to degrees and send the axis to that position. Beginning with CNC12 version 4.14, you can use the functions "FIX" and "FUP" to round down or up to the next integer value. Note that for negative values, FIX still rounds down (thus away from zero) and FUP still rounds up (towards zero).

With CNC12 v4.14 or newer, then, you can round to the nearest integer by first adding 0.5, then rounding down:

```
#100 = FIX[#5044/360.0 + 0.5] * 360.0
G90 G0 A#100
```

14.) How do I Control an Output when the Probe Tool number is selected.

In some cases it may be necessary to turn on an output when the probe tool is selected. Below are two examples of how this can be done, Assuming tool 99 is the probe tool number and OUT7 is defined as Output 7 in the Acorn Wizard.

To accomplish this, at the end of our m6 macro we could have a line that states:

Method 1:

```
IF [#4120 == 99] THEN M67 ELSE M87 ; If tool selected is tool 99 then activate
   output 7, otherwise deactivate
                                   ; This will activate Output 7 if the tool
                                     requested was 99.
```

Alternatively, Parameter 12 could be used as it is the touch probe tool number so, we could instead write.

Method 2:

```
IF [#4120 == #9012] THEN M67 ELSE M87 ; IF tool selected Is equal to touch
   probe tool number, then activate output
                                      ; 7, otherwise deactivate output 7
```

Note: The probe tool number should be set up correctly in the Probe menu of the Acorn Wizard. The value in the Wizard will be written to parameter 12 when pressing "Write Settings to CNC Control Configuration". If the operator decided that tool 10 should be the probe instead of 99, he could then simply change the value in the Wizard. The first method would require the operator to modify the macro and may lead to damaging the probe if not done correctly. It is generally better to use existing parameters or variables when available for macros instead of "hard-coding" values into the macro.

15.) Can I use "math" in a macro?

Yes, Please see Mill Operator manual Section 11.2.14 for complete list of allowed mathematical expressions

Examples:

```
G91 X [13/64] Z [1+3/8]                ; move the X axis 13/64 (0.2031)
   units and the Z axis 1 3/8 (1.375) units incrementally
X[ SQRT [ ABS [ SIN [#101] - COS [#102]]]] ; Move X as a function of #101 and
   #102
```

16.) What is System Variable #4203

Currently, Acorn software does not use system variable #4203 for "Tool in spindle" and will result in an "Undefined variable" error in software. For CNC12 for use with Allin1DC and Oak CNC controllers, #4203 is used in five axis machines, primarily for special tool changes involving "Large tools". System Variable #4203 is set differently depending on how parameters 6 and 160 are set.

If Parameter 6 and 160 are set to a zero, then #4203 is updated when a T command is called, Essentially,

```
#4203 == #4120
```

If Parameter 6 is a non-zero value and 160 is set to a zero, then #4203 is updated from a value from the PLC. For Example a line in the PLC may say the following,

```
SV_PLC_CAROUSEL_POSITION = CarouselPosition_W
```

SV_PLC_CAROUSEL_POSITION is the tool number displayed at the top right in the software and is the value of #4203, it will not change until the value in the PLC changes. CarouselPosition_W is a variable used in the PLC, it is assigned values which then gives its value to the SV_PLC_CAROUSEL_POSITION variable.

If Parameter 6 and 160 is set to a non-zero, then #4203 is updated after the M6 tool change macro is called and finished. This allows for special operations in the macro for 5-axis machines primarily, where #4120 can be used for the requested tool and #4203 is the previous tool. So,

```
#4120 = T Number requested
#4203 = Tool in Spindle (Last T number called)
M6 is executed and Finished
#4203 = Tool number requested making it new tool in spindle
```

17.) What are the differences between M98, G65 with Local Variable Stack

M98 and G65 can both be used to call a CNC subprogram.

The most significant difference between the two is that G65 allows arguments (parameters or variables) to be passed in as part of the subprogram call. The arguments provided with a G65 call are accessible within the subprogram as CNC variables #1 through #33.

For example, any value given with the letter ‘A’ in a G65 call will appear in variable #1 within the subprogram.

Any CNC program or subprogram can use variables #1 through #33 for their own purposes, whether or not they were given values through a G65 call.

However, therein lies a key difference between M98 and G65: a G65 call advances the “stack”, so that the #1–#33 variables in the called subprogram are a fresh set, independent of the #1–#33 variables in the calling program. An M98 call, in contrast, does not advance the stack. Variables #1–#33 in the M98-called subprogram are the same ones used by the calling program.

Consider this example:

```
; define a macro subprogram, for later use
O9001
#3 = #1 + #2
G0 Y#3
M99
; main program starts here
#1 = 10.0
#2 = 5.0
#3 = 1.234
G0 X#3              ; moves X to X1.234
G65 P9001 A3.2 B0.6 ; macro moves Y to Y3.8
G0 Z#3              ; moves Z to Z1.234
```

Because the variables #1, #2 and #3 in the macro subprogram are independent of the ones in the main program, the main program’s values are unchanged after the G65 call.

Compare that with the following, identical except that G65 has been replaced with M98:

```
; define a macro subprogram, for later use
O9001
#3 = #1 + #2
G0 Y#3
M99
; main program starts here
#1 = 10.0
#2 = 5.0
#3 = 1.234
G0 X#3              ; moves X to X1.234
M98 P9001 A3.2 B0.6 ; A and B do nothing, macro moves Y to Y15.0
G0 Z#3              ; moves Z to Z15.0
```

Using M98 instead, the A and B arguments are non-sensical and are ignored. The macro reads and writes the same local variables as the main program. It therefore calculates #3 = 10.0 + 5.0 = 15.0, and moves Y there. When we get back to the main program, local variable #3 has been changed, so the Z move is affected as well.

As a rule, if you want to use #1–#33 as local or temporary variables, you should call your macro subprograms with G65.

If you have previously worked with older controls, you might have avoided using G65 when you did not need to pass arguments, because the number of nested G65 calls was limited. For example, on early Fanuc controls, you could nest subprogram (M98 and G65) calls up to 20 levels deep, but only four of those levels could use G65: there was only enough memory for four sets of local variables.

CNC12 allows G65 calls up to 20 levels deep, each with its own set of local variables. Computer memory has come a long way.

18.) What is the PLC detective?

While not directly involved with a macro the PLC Detective is a tool that a macro programmer should at least be aware of and know how to use at least in the basic sense as it allows you to see in real time what is going on inside the PLC program while the machine is running which can be very useful.

From the manual, "*The PLC detective is a free built-in CNC utility software PLC Debug tool that is built into CNC12. The PLC detective makes writing and troubleshooting PLC programs faster and easier. The PLC detective is based on Centroid's already powerful PLC programming source language and enable system integrators to tackle larger and more complex retrofits.*

Every system Integrator who has ever tried to create or modify a PLC program for a new or modified machine knows the frustration of having run some new PLC program, wondering "Whats it doing now?". Now Centroid's "PLC Detective" allows you to clearly see the true/false state of each instruction while the machine is running, highlighted in color!

Everyone who has ever tried to debug a long term intermittent problem knows the frustration of trying to figure out what led up to a fault condition. Now Centroid's "PLC Detective" solves this problem once and for all. Set your trap , and wait , when the error occurs, go look back in time at all the I/O's leading up to the problem. This is "game changing" PLC debug tool for Integrators and Power Users alike."

To start PLC detective from the main screen of CNC12, press <Ctrl E>, this will bring up PLC detective, allowing you to see what is active and not active in the PLC program and some other useful information regarding the PLC.

See the PLC detective manual for more information.
https://www.centroidcnc.com/downloads/centroid_PLC_detective_quickstart.pdf

19.) How do I edit the PLC program to add Custom PLC Output?

Adding a custom output definition to the Acorn PLC program.

To enable the use of M94 and M95 for a custom output, (not necessary if you are using any one of the stock Wizard outputs) edit the PLC with both the desired Custom Output, and desired logic. For this example we will create an output called “CustomOut” and add it to the Acorn PLC program. The Acorn PLC source file can be found in the cncm directory “acorn_mill_plc.src” or in your cnct directory for lathe “acorn_lathe_plc.src”. Here is the PLC program Example:

```
;-------------------------------------------------------------------------------
;
Output DEFINITIONS
;
Closed = 1 (green) Open = 0 (red)
;-------------------------------------------------------------------------------
CustomOut   IS OUT2             ; Defines our Custom Output to Output 2
;-------------------------------------------------------------------------------
; M functions - The System Variables in this section inform the
; PLC that an M function has been requested.
;-------------------------------------------------------------------------------
CustomM94M95 IS SV_M94_M95_18    ; Defines CustomM94M95 to the value of 18 for
   use in the macro.
;===============================================================================
MainStage
;===============================================================================
IF CustomM94M95 THEN (CustomOut) ; When CustomM94M95 is ON, Turns on CustomOut.
                                 ; When CustomM94M95 is Off, Turns Off
                                    CustomOut
; Parentheses tells logic that it should be on when expression is true and turn
    off when it is not.
; Alternatively the logic can be
IF CustomM94M95 THEN SET CustomOut  ; When CustomM94M95 is On, Turns on
   CustomOut
IF !CustomM94M95 THEN RST CustomOut ; When CustomM94M95 is Off, Turns Off
   CustomOut
;The ! sign in front of CustomM94M95 is a "not", so when CustomM94M95 is Off
   then the expression is true.
;SET is to set or other words turn on, and RST is to reset or in other words
   turn off.
```

Once the Output, the M function system variable and logic is defined, you will need to compile the edited source code. Refer to the CNC12 PLC Programming Manual Page 5. Once the plc is compiled we can edit the macro we wish to use this output. To turn on the “CustomM94M95” variable in the plc we will need to use an M94 command, Example Below.

```
M94 /18 ; Turn On CustomOut Request, Notice the value 18 the same as the value
   in PLC.
```

This will turn on in the plc “CustomM94M95”, which in turn the plc will look at our written logic and in this case turn on the CustomOut Output. Since we do not have any logic in the PLC that resets the system variable, we will need to use an M95 command to turn it off, example,

```
M95 /18 ; Turn Off CustomOut Request, Notice the value 18 the same as the value
    in PLC.
```

This will turn off in the plc “CustomM94M95”, which will turn off the output “CustomOut” This is a very basic example, with the introduction of the PLC, additional logic can be added to give additional functionality, such as timeout timers, Fault messages, and more when using this output.

More Information for PLC Programming can be found in the Centroid CNC12 PLC Programming Manual and Centroid PLC Programming Videos here is a link to the Playlist Found on Centroid CNC Technical Support Youtube Channel.

https://www.youtube.com/user/CentroidSupport/playlists

https://www.youtube.com/playlist?list=PLXhs2C5No0_gFS_RmKNo7hii2WKIedQ1Q

11.4 Advanced Macro Statements

NOTICE

Branching and conditional execution are extremely powerful tools that, combined with access to system variables, allow you to do many things that would otherwise be impossible. Nevertheless, using branching and conditional execution can introduce undesirable and even unpredictable behavior into your programs. Undesirable effects can occur simply by graphing a program. The least of these undesirable effects could be entering an endless loop, failing to draw anything, or wiping out all the information in your tool library or WCS settings. It is your responsibility to make sure that undesirable things do not happen in your programs. You must monitor the job processing and search modes in your program and, if necessary, take appropriate action. Until you are confident of the actions of your program, you should step through it one block at a time to confirm your program logic.

11.4.1 GOTO – Branch Execution

To branch to another line within the same program or subprogram, use the statement:

GOTO <expression>

where <expression> is any expression that evaluates to a valid block number in the program. GOTO causes an immediate branch to the specified destination. Program codes preceding a GOTO on the same line will be executed normally. Any program codes following GOTO on the same line will cause an error.

If fast branching is disabled (Parameter 145 = 0) then the CNC software searches forward in the program for the first matching block number and, if necessary, resumes searching from the top of the program. For this reason, when fast branching is disabled, backward branches take longer than forward branches and backward branch times depend on the total program size. If the program is sufficiently large, use of the GOTO statement could introduce temporary pauses.

When fast branching is enabled (Parameter 145 = 1), the CNC software remembers the locations of block numbers as it finds them during program execution. Backward branches always take place immediately. The first forward branch to a block not yet encountered will take additional time as the CNC software searches forward for the block number, however subsequent forward branches to that block number will take place immediately. The trade-off for using fast branching is that all line numbers at a given level of program or subprogram must be unique and programs will use more memory (approximately 16kilobytes of memory for every 1000 block numbers in the program).

11.4.2 IF THEN ELSE – Conditional Execution

Program symbols, G-codes, M codes, and GOTO commands may be executed conditionally using the IF statement. The general form of the IF statement is:

IF <expression> THEN <execute if true> ELSE <execute if false>

where <expression> is any valid expression, <execute if true> is one or more program codes to execute if <expression> evaluates to "true" (non-zero), and <execute if false> is one or more program codes to execute if <expression> evaluates to "false" (zero). All parts of the IF statement must appear on the same line. The "ELSE <execute if false>" part of the statement is optional and may be omitted. The "THEN" may be omitted, however <expression> must be enclosed in brackets ([ ]). The IF statement may follow other program codes on the same line. Compound conditionals are possible but cannot be nested. The first THEN always pairs with the first IF. ELSE always pairs with the first <expression> that evaluates to "false". All program codes executed are executed as part of the same block.

Example:

```
            ;Branch to N200 if machine position is okay, otherwise go to N300
N100 IF #5041 LE 5.0 THEN GOTO 200 ELSE GOTO 300
            ;Force subprogram parameter #D to be within range.
IF [#D LE 0.005] #[D] = 0.005
            ;Compound conditionals
IF [#A LE 0.0] GOTO 100 ELSE IF [#A LE 2.5] GOTO 200 ELSE GOTO 300
IF [#A GT 0.0] IF [#D/#A GE 0.0] #[C] = SQRT[#D/#A]
```

11.4.3 INPUT – Prompt Operator For Input

The INPUT macro prompts the operator for numeric input. The general form of the INPUT statement is:

INPUT "<prompt>" <variable>

Where <prompt> is the message prompt for the operator and <variable> is the variable in which to store the input. The CNC software will display a dialog with the given prompt and space for the operator response. The operator may enter any numeric expression (see above) including variables as a response. The operator must press **CYCLE START** or **Alt-S** to dismiss the dialog. Pressing **ESC** will cancel the job.

The CNC software parses well ahead of the current execution to maximize throughput and efficiency. For this reason, an INPUT macro may prompt the operator for input immediately, even though the INPUT macro is located in the middle or near the end of the job. Parsing pauses while the dialog is displayed. Any statements parsed prior to the

INPUT macro will have been queued and will continue to execute in the background while the prompt is displayed. Job processing will pause only if all queued statements have been executed before the operator supplies a response.

INPUT macros will not graph. If you must graph the job, first set the input variable to a default value and use a conditional to execute the INPUT only if the job is being run normally.

Use search mode cautiously with INPUT macros. To have search work properly, you may have to supply exactly the same input during the search as you did during the last actual run.

Example:

```
            ;Ask operator for pocket depth. Store result in #101
            ;Note: this will not graph.
INPUT ‘‘Enter pocket depth’’ #101

            ;Allow job with INPUT statements to be graphed.
#101 = 0.5  ;Supply a default value for graphing
            ;Ask for operator input only if not graphing.
IF NOT #4201 THEN INPUT "Enter pocket depth" #101
```

11.4.4 DEFINE – Text Aliases

You can define your own text aliases using angle brackets (<>). This is a pure text replacement tool (with one exception – any text found after a semicolon (‘;’) will be ignored as a comment), but can be quite useful.

To define an alias in your G-code program, use the word DEFINE followed by the name of your alias (in angled brackets) and the text it will be replacing.

For example, consider the following G-code program:

```
G17 G90 F25
G00 X1.0 Y1.0 Z0.0
G02 X2.0 Y2.0 Z0.0 R-1.0
```

Using the angle bracket defines, we could write:

```
;Definitions begin here

DEFINE <XY_PLANE> G17
DEFINE <ABSOLUTE_POSITIONING> G90
DEFINE <FEEDRATE> F
DEFINE <RAPID_POSITIONING> G0
DEFINE <START_POSITION> X1.0 Y1.0 Z0.0
DEFINE <CLOCKWISE_ARC> G02
DEFINE <END_POSITION> X2.0 Y2.0 Z0.0
DEFINE <BIG_ARC_RADIUS> R-1.0

;Actual job begins below

<XY_PLANE> <ABSOLUTE_POSITIONING> <FEEDRATE>25
<RAPID_POSITIONING> <START_POSITION>
<CLOCKWISE_ARC> <END_POSITION> <BIG_ARC_RADIUS>
```

This can make your G-code programs more readable and understandable.

12 CNC Program Codes: G-Codes

G-Code		Group	Description
G00	*	A	Rapid Positioning
G01		A	Linear Interpolation
G02		A	Circular or Helical Interpolation CW
G03		A	Circular or Helical Interpolation CCW
G04		B	Dwell
G09		B	Decelerate and Stop (formerly known as Exact Stop)
G10		B	Parameter Setting
G17	*	C	Circular Interpolation Plane Selection XY
G18		C	Circular Interpolation Plane Selection ZX
G19		C	Circular Interpolation Plane Selection YZ
G20	*	K	Select Inch Units
G21		K	Select Metric Units
G22		O	Work envelope on
G23	*	O	Work envelope off
G28		B	Return to Reference Point
G29		B	Return from Reference Point
G30		B	Return to Secondary Reference Point
G40	*	D	Cutter Compensation Cancel
G41		D	Cutter Compensation Left
G42		D	Cutter Compensation Right
G43		E	Tool Length Compensation (+)
G43.3		E	Tool Length Compensation (+) with Axis Tilt Compensation
G43.4		E	Rotary Tool Center Point (with G43.3 compensation)
G44		E	Tool Length Compensation (-)
G49	*	E	Tool Length Compensation Cancel
G50	*	M	Scaling/Mirroring Off
G51		M	Scaling/Mirroring On
G52		B	Offset Local Coordinate System Origin
G53		B	Rapid Position in Machine Coordinates
G54		L	Select Work Coordinate System #1
G55		L	Select Work Coordinate System #2
G56		L	Select Work Coordinate System #3
G57		L	Select Work Coordinate System #4
G58		L	Select Work Coordinate System #5
G59		L	Select Work Coordinate System #6
G61		F	Modal Decel and Stop (formerly known as Exact Stop Mode)
G64	*	F	Smoothing mode selection / Cancel Modal Decelerate and Stop
G65		J	Call Macro
G68		N	Coordinate Rotation on
G68.1		N	Transformed Work Coordinate System
G69	*	N	Coordinate Rotation off
G73		G	High Speed Peck Drilling
G74		G	Counter Tapping
G76		G	Fine Bore Cycle
G80	*	G	Canned Cycle Cancel

G-Code		Group	Description
G81		G	Drilling and Spot Drilling
G82		G	Drill with Dwell
G83		G	Deep Hole Drilling
G84		G	Tapping
G85		G	Boring
G89		G	Boring with Dwell
G90	*	H	Absolute Positioning Mode
G91		H	Incremental positioning Mode
G92		B	Set Absolute position
G93		P	Inverse Time On
G93.1		P	Velocity Scrubber for Smoothed Inverse Time Data
G94		P	Inverse Time Off
G98	*	I	Initial Point Return
G99		I	R Point Return
G117		C	Rotation of Plane Selection XY
G118		C	Rotation of Plane Selection ZX
G119		C	Rotation of Plane Selection YZ
G173		G	Compound High Speed Peck Drilling
G174		G	Compound Counter Tapping
G176		G	Compound Fine Bore Cycle
G180		G	Compound Canned Cycle Cancel
G181		G	Compound Drilling and Spot Drilling
G182		G	Compound Drill with Dwell
G183		G	Compound Deep Hole Drilling
G184		G	Compound Tapping
G185		G	Compound Boring
G189		G	Compound Boring with Dwell

Notes:

1. All the default G-codes have been marked with the symbol "*".
2. A given line of a program may contain more than one G-code.
3. If several G-codes from one group are used in the same line, only the G-code specified last will remain active.
4. G-codes from group B are of "one shot" type (active only in the line in which they are specified). All other G-codes are modal (active until another G-code of the same group is specified).
5. If a G-code from group A is used in a canned cycle mode, the canned cycle will be canceled. Canned cycle Gcodes, however, have no effect on G-codes from group A.

12.1 G00 – Rapid Positioning

G0 moves to the specified position at the maximum motor rate. The coordinates may be either absolute positions (G90) or incremental positions (G91). G0 is modal and remains in effect until another positioning mode (G1, G2, G3 etc.) is commanded. G0 is the default-positioning mode.

When the Z axis is commanded to move in the + direction, the Z axis will move up to its new position first, then the other axes will move to their new position along a straight line.

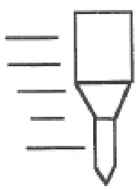
When the Z axis is commanded to move in the − direction, all axes but the Z axis will move to their new position along a straight line, then the Z axis will move down to its new position.

Example:

```
G0 X0.0 Y0.0 Z0.0 ; Rapid move to X0, Y0, Z0
```

CAUTION The feedrate override knob has no effect on G0 moves unless rapid override is turned ON.

12.2 G01 – Linear Interpolation

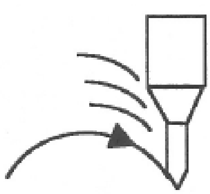


G1 moves to the specified position at the programmed feedrate. The coordinates may be either absolute positions (G90) or incremental positions (G91). The movement will be along a straight line. G1 is modal and remains in effect until another positioning mode (G0, G2, G3 etc.) is commanded.

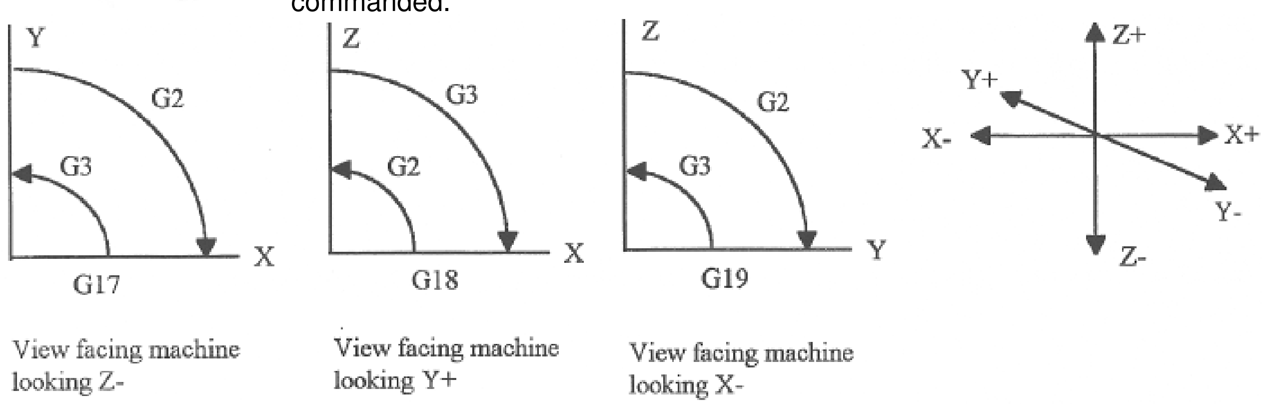
Example:

```
G01 X2 Y3 Z4 W5 F10 ; Linear move to X2, Y3, Z4, W5 at a 10in/min
G91 X6 Y7           ; Linear move to X8, Y10
Z3 W4 F20           ; Linear move to Z7, W9 at 20in/min (G91 is modal)
```

12.3 G02 & G03 – Circular or Helical Interpolation



G2 moves in a clockwise circular motion, and G3 moves in a counterclockwise circular motion. This clockwise and counterclockwise motion is relative to your point of view, however. See the diagram below. The X, Y or Z position specified in the G2 or G3 command is the end position of the arc, and may be an absolute position (G90) or an incremental position (G91). G2 and G3 are modal and remain in effect until another positioning mode (G0, G1, etc.) is commanded.



Note: When using G18, the G2 command moves in a counterclockwise direction in the XZ plane.

The axes included in the currently selected circular plane (G17, G18, or G19) will move in a circular motion. Any other axes specified will move along a straight line (helical movement). The programmed feedrate is used for the interpolated motion along the movement of all axes.

Helical and circular motion can be programmed in two different ways: specifying the final point and the radius of the arc, or specifying the final point and the parameters I, J, K (center point of the arc as incremental values from the start position).

Note: For closed circles (arc of 360 degrees), use method 2: specify final point and parameters I, J and K. Method 1 (specify final point and radius) will not work.

METHOD 1: USING FINAL POINT AND RADIUS

The commands G2 and G3 will have the following structure:

G2 Xa Yb Zc Rd

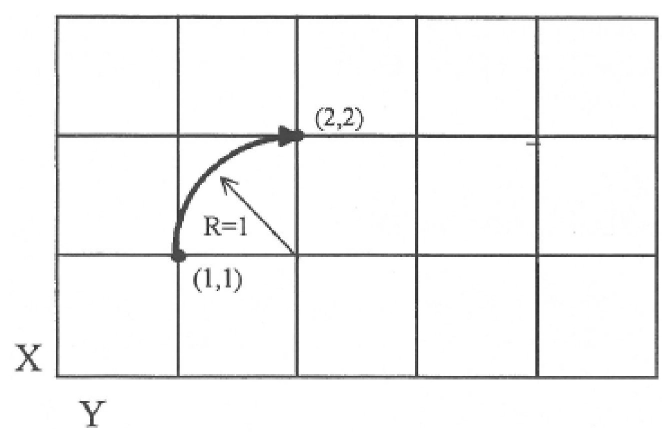
G3 Xa Yb Zc Rd

where a, b, and c will be the X, Y, and Z coordinates of the final point of the arc, and d will be the radius. In most cases there will be two possible arcs of the same radius connecting two given points. This occurs because the center of the arc is not specified. To choose the bigger arc, make the radius negative. To choose the smaller arc, make the radius positive. See examples 1 and 2 for graphical explanations of this concept.

Note: Negative radii are an international G-code convention.

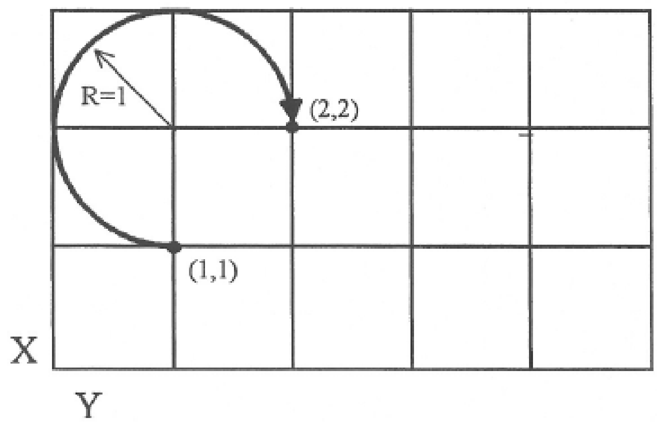
Example 1 (small arc solution: positive radius):

```
G17 G90 F25      ; selects XY plane and absolute positioning
G00 X1.0 Y1.0 Z0 ; rapid to start position X1, Y1, Z0
G02 X2 Y2 Z0 R1  ; arc to X2 Y2 Z0 with radius of -1
                 ; (small arc solution)
```



Example 2 (big arc solution: negative radius):

```
G17 G90 F25       ; selects XY plane and absolute positioning
G00 X1.0 Y1.0 Z0  ; rapid to start position X1, Y1, Z0
G02 X2 Y2 Z0 R -1 ; arc to X2 Y2 Z0 with radius of 1
                  ; (big arc solution)
```



METHOD 2: USING FINAL POINT AND PARAMETERS I, J, K

Another way to specify a helical or circular operation is using the parameters I, J, K instead of the radius R. The parameters I, J, and K are the **incremental** distances from the start point to the center of the arc. For absolute positioning on I, J, and K, Parameter 2 bit 0 will need set. See the parameter section in Chapter 15.

I = X center - X start (valid for G17 & G18)

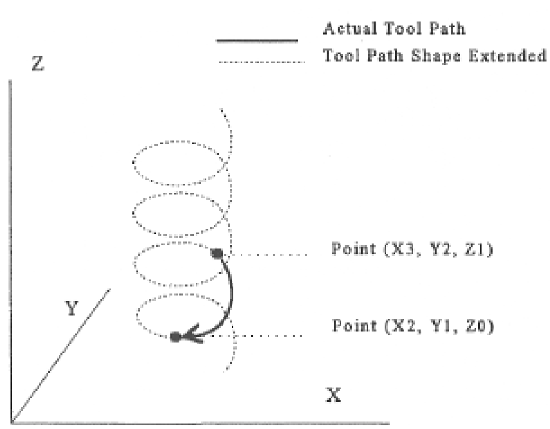
J = Y center - Y start (valid for G17 & G19)

K = Z center - Z start (valid for G18 & G19)

Examples:

Example Circular motion (See graph in method 1, example 2):

```
G17 G90 F25       ; selects XY plane and absolute positioning
G00 X1.0 Y1.0 Z0 ; rapid to start position X1, Y1, Z0
G02 X2 Y2 Z0 J1  ; arc to X2 Y2 Z0 with radius of 1
```



Helical motion

Example:

```
G17 G90 F30                       ; select XY plane and absolute positioning
G00 X3.0 Y2.0 Z1.0                ; rapid to start position X3, Y2, Z1
G02 X2.0 Y1.0 I-1.0 J0.0 Z0.0 ; CW XY arc from X3, Y2 to X2, Y1.
                                  ; Center at X2, Y2
                                  ; Helical Z move from 1 to 0
```

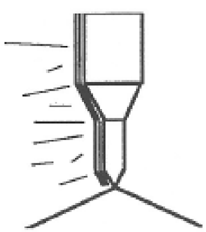
12.4 G04 – Dwell

G4 causes motion to stop for the specified time. The P parameter is used to specify the time in seconds to delay. G4 causes the block to decelerate to a full stop.
The minimum delay is 0.01 seconds and the maximum is 327.67 seconds. The dwell time is performed after all motion is stopped and M functions on the line are completed. If the P parameter is not specified, X will be used instead. If neither P nor X is specified, the default dwell time of 0.01 seconds will be used.

Example:

```
G0 X1 Y1 ; rapid to X1, Y1
G4 P2.51 ; pause for 2.51 seconds
G1 X2 Y2 ; Linear move to X2, Y2
```


12.5 G09 – Decelerate and Stop (formerly known as Exact Stop)



G9 causes motion to decelerate to a stop and dwell for 1/100 seconds. G9 is equivalent to G4 P0.01. G9 is not modal; it is only effective for the block in which it appears. See G61 (Modal Decelerate and Stop).

Example:

```
G9 G0 X1 Y1 ; rapid to X1 Y1 and stop
X2 Y2       ; continue to X2 Y2
```

12.6 G10 – Parameter Setting

G10 allows you to set parameters for different program operations.

Example:

```
G10 P73 R.05 ; Sets the peck drilling retract amount to .05
G10 P83 R.05 ; Sets the deep drill rapid down clearance to .05
G10 P81 R15  ; Sets G81 to use M15 instead of Z movement
G10 H5 R-1.3 ; Sets tool length offset #5 to -1.3 in the offset lib.
G10 D3 R.25  ; Sets tool diameter offset #3 to .25 in the offset lib.
```

Note: The following parameters cannot be modified: 1–5, 7–9, 11–30, 39, 40, 42, 44, 52, 53, 60–63, 65–67, 70, 71, 75–77, 82, 87–90, 95–98, 100–106, 107–112, 114, 120–122, 132–135, 155–159, 165, 220–224, 226–231, 236–241, 252–255, 257, 270, 271, 278, 300–315, 416.

12.7 G17, G18, G19 – Circular Interpolation Plane Selection

G17, G18, and G19 select the plane for circular interpolation commands (G02 & G03). G17 is the default plane.

See figure under G2 and G3.

G17 is the XY plane
G18 is the ZX plane
G19 is the YZ plane

12.8 G20 – Select Inch Units

G20 selects inch units, affecting the interpretation of all subsequent dimensions and feedrates in the job file. G20 does not change the native machine units as set on the control setup menu.

12.9 G21 – Select Metric Units

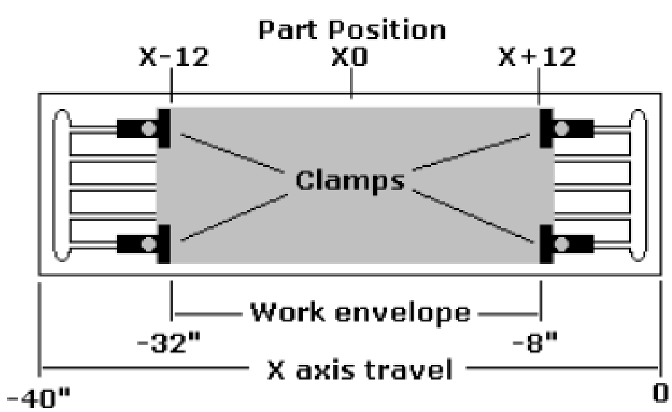
G21 selects metric units, affecting the interpretation of all subsequent dimensions and feedrates in the job file. G21 does not change the native machine units, as set on the control setup menu.

12.10 G22/G23 – Work Envelope On/Off

G22 turns on programmable work envelope in machine coordinates. When the machine tries to move into the forbidden area, let's say the x-axis, an "x-axis work envelope exceeded" message is displayed, letting you know which line of the program is at fault. The work envelope is set with the X, Y, Z for the '+' limit and I, J, K for the '−' limit. G22 is modal and remains on until turned off by G23 or the end of the job. The limits entered in the X, Y, Z and I, J, K parameters are stored in the WCS menu under F3 – Work Envel. For more information see Chapter 4.

Example:

```
G22 X-8 I-32 ; Keeps programs from moving into the outside 8 inches of
X-           ; axis of travel
G1 X-13 F20  ; Would generate a "X axis work envelope exceeded, line 3" message
G23          ; Allows travel into G22 forbidden area.
M25          ; Z home
G0 X-13      ; Ok to move X here now
```



12.11 G28 – Return to Reference Point

G28 moves to the first reference point, by way of an intermediate point. The location of the reference point, in machine coordinates, may be set in Work Coordinate System Configuration. The intermediate point is specified in the local coordinate system and may be at the current location (resulting in a move directly to the reference point). If an intermediate point is specified, only those axes for which positions are specified will be moved. If no axes are specified, all axes will be moved. The location of the intermediate point is stored for later use with G29. Movement is executed at the maximum (rapid) rate but can be changed using the `L` word.

Example:

```
G28 G91 Z0         ; move Z-axis directly to reference point ( X and Y don't
   move)
G28 G91 X-.5 Y0 Z0 ; move X -0.5 (from current position), then move all three
   axes to reference point
G28 G90 X2 Y4 Z.1  ; move all axes to (X2, Y4, Z0.1), then to reference point
G28                ; move all axes to the reference point ( no intermediate
   point)
G28 L100           ; move all axes to the reference point at 100 units/minute
```

Note: As with G0 positioning moves, the Z-axis will move separately. If Z is moving up (the usual case) Z will move first, then the other axes. If Z is moving down, the other axes will move first, then Z. Because of this, it is rarely necessary to specify an intermediate point different from the current position.

12.12 G29 – Return from Reference Point

G29 moves all axes to the intermediate point stored in a preceding G28 or G30 command. It may be used to return to the work piece. If a position is specified, the machine will move to that position (in local coordinates) after reaching the intermediate point. G29 may only be specified after G28 or G30, though there may be intervening moves.

Example:

```
G29       ; move all axes back from reference point to intermediate point
G29 X1 Y2 ; move all axes to intermediate point, then move to X1 Y2
```

Note: As with G0 positioning moves, the Z-axis will move separately. If Z is moving up, Z will move first, then the other axes. If Z is moving down (the usual case for G29), the other axes will move first, then Z will move.

12.13 G30 – Return to Secondary Reference Point

G30 functions exactly like G28, except that by default it uses the second reference point from the Work Coordinate System Configuration table, and the P parameter may be used to request either reference point.

Example:

```
G30 G91 Z0 ; move Z axis directly to second reference point
G30 P1     ; move all axes to first reference point
```

Note: G30 P1 is equivalent to G28.

12.14 G40, G41, G42 – Cutter Compensation

G41 and G42, in conjunction with the selected tool diameter (D code) apply cutter compensation to the programmed tool path.

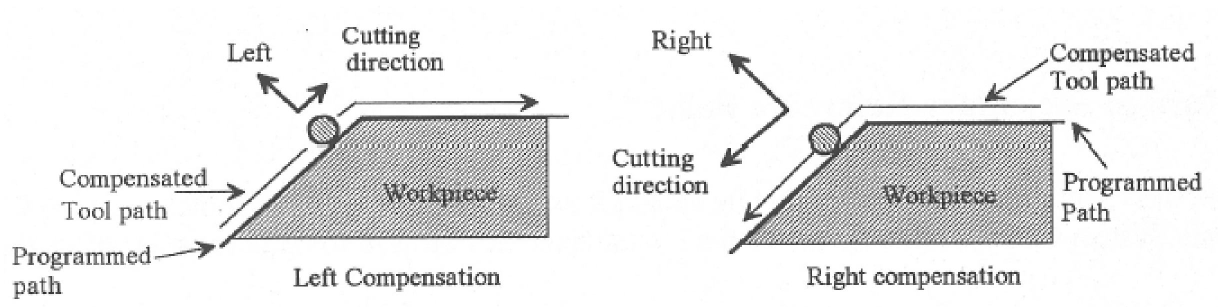
G41 offsets the cutter tool one half of the tool diameter selected with a D code, to the left of the work piece, relative to the direction of travel.

G42 offsets the cutter tool one half of the tool diameter selected with a D code, to the right of the work piece, relative to the direction of travel.

G40 cancels G41 and G42.

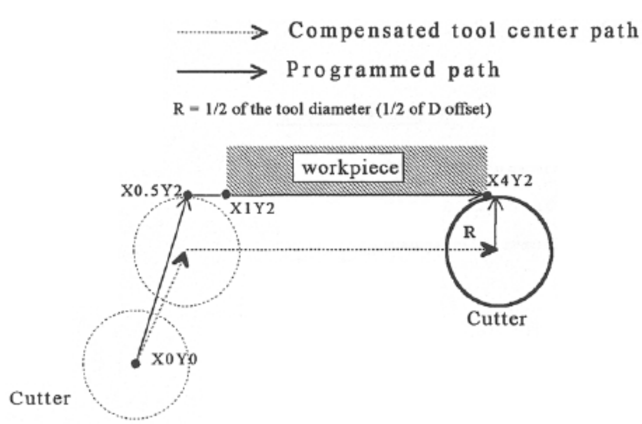
Example:

```
G41 D03 ; Tells the machine to compensate left half of the
        ; diameter of the amount that corresponds to D03 in the
        ; Tool Library
```

Whenever cutter compensation is applied, the following factors must be taken into account in order to obtain proper results.

1. The cutter diameter compensation function (G41, G42) must be implemented before the cutter tool reaches the starting cutting point.



Example 1:

```
G0X0Y0   ; Rapid tool to X0, Y0
```

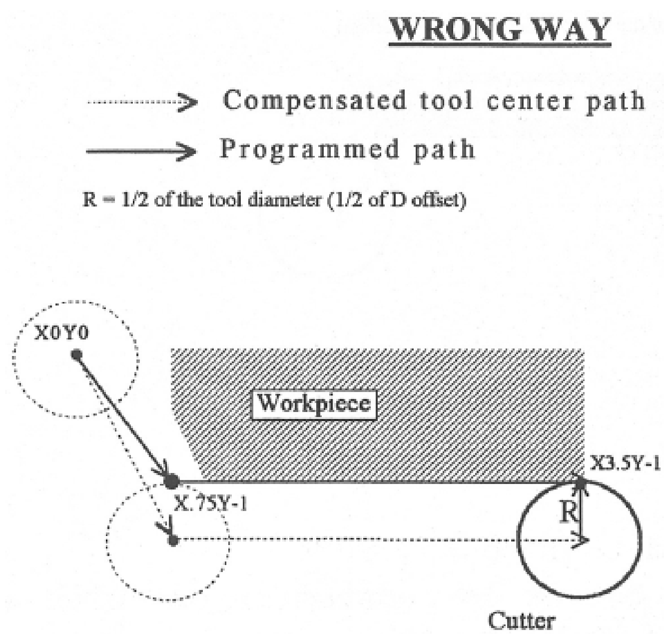
```
G42 D3    ; Turn cutter compensation on, with a diameter of D3
G0X.5Y2   ; Rapid to X0.5, Y2
G1x4.1Y2  ; Linear cut to X4.1, Y2.
          ; Cut to X4.1 to clear material.
G40       ; Turn cutter compensation off.
G0X5Y0    ; Rapid to X5, Y0.
```

You may want to add .1 or .05 inches on the final position for the last cut to clear the material.

Note: The diameter compensation statement G42 is placed before G0 X.5 Y2. As a result, the compensation is applied before the cutter reaches the starting cutting point X.5 Y2.

2. If the cutter is down, then the cutter compensation lead-in must always come from an appropriate direction. Otherwise, the work piece will be incorrectly cut, and the cutter tool could be damaged. One way to avoid this problem is by always keeping the cutter above the work piece whenever a transition is being made to a new starting cutting point. If for some reason this was not possible, then the G-code program should be written so that the cutter compensation lead-in paths do not interfere with the space occupied by the work piece. Example 2 illustrates a possible harmful outcome of programming an inappropriate lead-in direction.

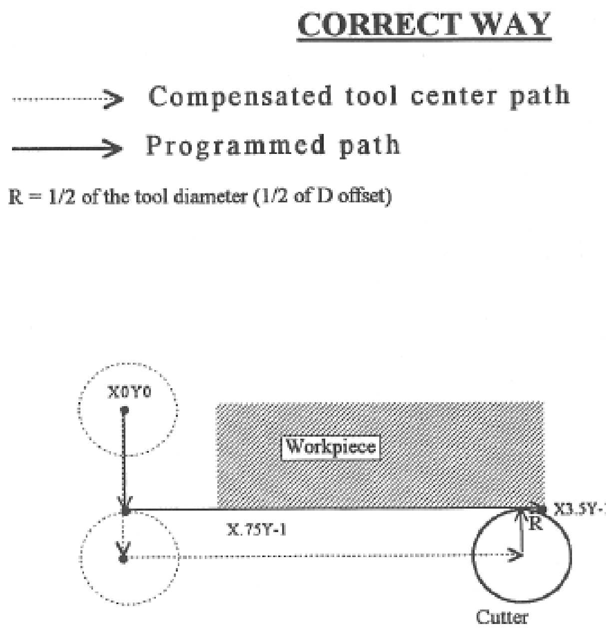
Example 2:



Example 2:

```
G0 X0Y0          ; Rapid tool to X0, Y0
G42 D5           ; Turn cutter compensation on, with a diameter of D5
G1 X.75Y-1 F5    ; Linear move to X0.75, Y-1. (Notice this damages the
                 ; corner of the work piece)
X3.6             ; move X to 3.6
G40              ; Turn cutter compensation off.
G0 X4Y-2         ; Rapid to X4, Y-2
```

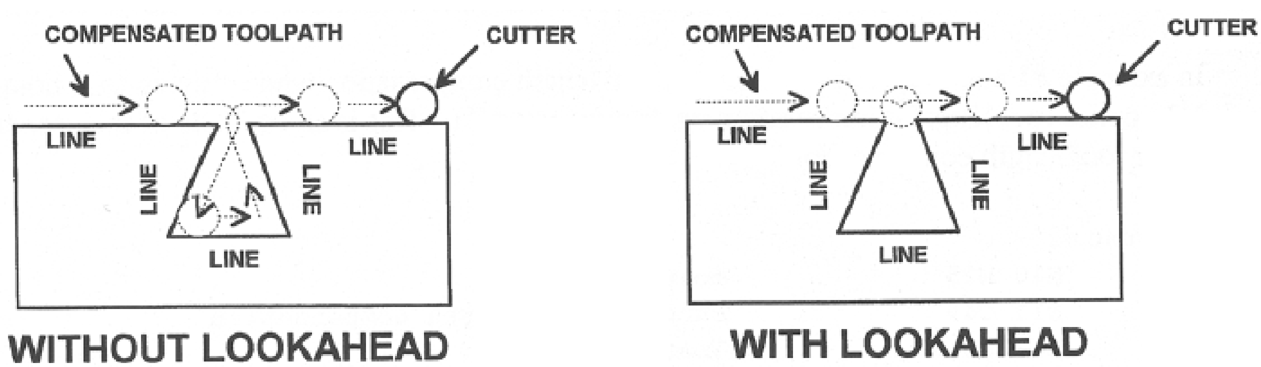
Note: This problem could have been avoided by selecting a transitional point between X0 Y0 and X.75 Y-1. A transitional point such as X-1 Y-1 would properly modify the lead-in path, keeping the cutter from damaging the corner of the work piece. Example 3 shows the correct way of performing this operation.



Example 3:

```
G0X0Y0     ; Rapid tool to X0, Y0
G42D5      ; Turn cutter compensation on, with a diameter of D5
G0X0Y-1    ; Rapid tool to X0, Y-1
G1X.75Y-1  ; Linear cut to X0.75, Y-1.
X3.6       ; move X to 3.6
G40        ; Turn cutter compensation off.
```

3. *Lookahead*. When the control machines any rapid traverse (G0), line (G1), or arc (G2, G3) with tool diameter compensation enabled, the program will look up to N consecutive events ahead of the current event in order to anticipate tool path clearance problems, where N is the number set in Parameter 99. Lookahead ensures that compensated tool paths don't overlap in programmed part sections where there is not enough clearance for the tool. The figure below shows a compensated tool path, and the actual tool path after Lookahead corrects the clearance problem:



Refer to the "Machine Parameters" section in Chapter 15 for more information on Parameter 99.

12.15 G43, G44, G49 – Tool Length Compensation

G43 and G44 apply tool length compensation to a selected tool to allow the control to utilize multiple tools in a single CNC program.

G43 applies positive compensation (from Z zero up). Work from part surface up. G44 applies negative compensation (from Z zero down), used only when there is an absolute machine home. The spindle face is considered a zero length tool and all offsets are from there down.

G49 cancels tool length compensation (also canceled by issuing G43 H00).

Example:

```
G43 H01 ; tells the machine to offset the amount that
        ; corresponds to H01 in the Offset Library
```

12.16 G43.3 – Tool Length Compensation (+) with Axis Tilt Compensation

G43.3 is a special compensation mode which applies positive tool length compensation on a selected tool, just like G43, but also with additional X and Z compensations due to 5th axis tilt. This compensation mode is available only on those machines configured with a triangular rotary 5th axis (see Parameter 166 in Chapter 15). Note that this compensation mode is the equivalent to G43 as long as the 5th axis is not tilted (i.e. local position is 0). G49 cancels this compensation mode.

12.17 G43.4 – Rotary Tool Center Point (with G43.3 Compensation

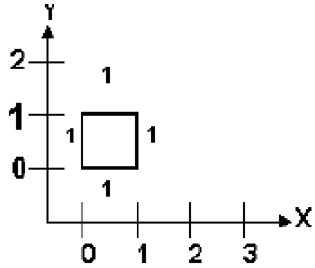
When G43.4 is active then any feed move (G1) will be made such that the tip of the tool moves in a straight line (this typically causes the Z-axis to move up and down during the move). G43.3 Compensation applies in this mode.

12.18 G50, G51 – Scaling / Mirroring (Optional)

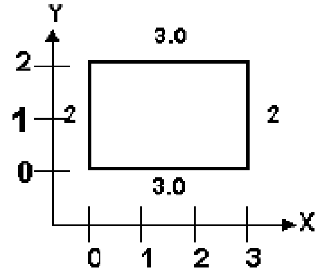
G50 and G51 scales program G-codes relative to a scaling center point defined as position (X, Y, Z). A G51 applies scaling/mirror to all positions, lines, and arcs following this G-code, until a G50 is entered. Specify scaling factors with a value I, J, K. The X, Y, and Z parameters are the coordinates of the scaling center. If the scaling center is not specified, the default scaling center is the current cutter position as shown on the DRO. To mirror, enter a negative value for the scaling factor.

Example, Scaling:

```
G51 X0.0 Y0.0 Z0.0 I3.0 J2 K1 ; turn scaling on
G00 X0.0 Y0.0 Z1.0            ; rapid to X0, Y0, Z1
G01 X1.0 Y0.0 Z1.0            ; line to X1, Y0, Z1
G01 X1.0 Y1.0 Z1.0            ; line to X1, Y1, Z1
G01 X0.0 Y1.0 Z1.0            ; line to X0, Y1, Z1
G01 X0.0 Y0.0 Z1.0            ; line to X0, Y0, Z1
G01 X0.0 Y0.0 Z0.0            ; line to X0, Y0, Z0
G50                           ; cancel scale
```



Original Box

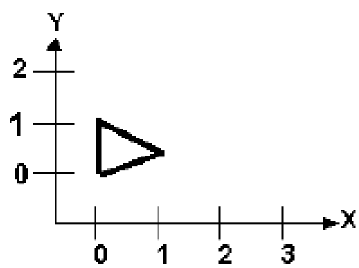


Scaled Box

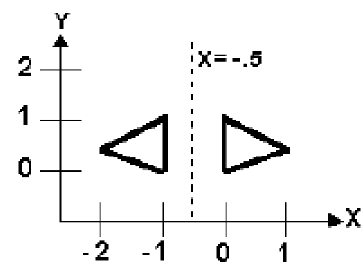
For this G51, the following program lines were scaled 3:1 in the X direction, 2:1 in the Y direction, and 1:1 in the Z direction. If no scale factor is specified, the default is 1:1 for all axes.

Example, Mirroring:

```
G51 X-0.5 Y0.0 Z.0 I-1 J1 K1 ; turn mirror on (x axis -0.5 mirror
                             ; line)
G00 X0.0 Y0.0 Z1.0           ; rapid traverse to X0, Y0, Z1
G01 X1.0 Y0.5 Z1.0           ; line to X1, Y.5, Z1
G01 X0.0 Y1.0 Z1.0           ; line to X0, Y1, Z1
G01 X0.0 Y0.0 Z1.0           ; line to X0, Y0, Z1
G50                          ; cancel mirror
```

Original Triangle



Mirrored Triangle

If an arc is scaled with uneven scaling factors, the result will depend on how the arc center and radius were specified:

1. If the arc radius was specified with R, the radius will be scaled by the larger of the two circular plane scale factors. The result will be a circular arc between the scaled arc start and the scaled arc end.
2. If the arc center was specified with I, J, and/or K, the centers will be scaled by the appropriate axis scale factors. The result will be a circular arc from the scaled arc start, around the scaled center, and usually with a line from the end of the circular arc to the scaled arc end.
3. In no case can an ellipse be generated using scaling.

12.19 G52 – Offset Local Coordinate System

G52 shifts the local coordinate system origin by a specified distance. Multiple G52 codes are not cumulative; subsequent shifts replace earlier ones. The G52 shift may therefore be canceled by specifying a shift of zero. If you are using multiple coordinate systems, the G52 shift amount will affect all coordinate systems.

Example:

```
G0 X0 Y0   ; move to origin
M98 P9100 ; call subprogram
G52 Y4     ; shift coordinate system 4 inches in Y
G0 X0 Y0   ; move to new origin
M98 P9100 ; call subprogram again with new coordinates
G52 Y0     ; restore unshifted coordinate system
```

12.20 G53 – Rapid Positioning in Machine Coordinates

G53 is a one shot code that performs a rapid traverse using machine coordinates. It does not affect the current movement mode (G0-G3) or coordinate system (G54-G59). G53 may only be used with absolute positioning (G90). Movement rate can be overridden using the `L` word.

Example:

```
G53 X15 Y4 Z0       ; move to 15,4,0 in machine coordinates
G53 X15 Y4 Z0 L100 ; move to 15,4,0 in machine coordinates at 100 units/minute
```

12.21 G54 – G59 – Select Work Coordinate System

G54 through G59 select among the six regular work coordinate systems (WCS #1 through WCS #6). After issuing the code, subsequent absolute positions will be interpreted in the new coordinate system. Alternatively, the codes E1 through E6 to can be used instead of G54 through G59.

Example:

```
G54 G0 X0 Y0 Z0 ; select first WCS, move to origin
G2 X1 I.5 Z-.5   ; mill something...
G0 Z.1            ; Rapid to position Z0.1
G55 X1 Y1         ; select second WCS, move to X1, Y1
```

Using Extended Work Coordinate Systems (optional): There are 12 additional work coordinate systems available as an extra-cost option. In a G-code program, these 12 additional work piece origins may be selected with either "G54 P1" (WCS #7) through "G54 P12" (WCS #18) or "E7" through "E18."

Regular WCS

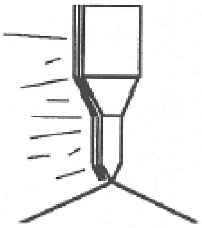
WCS	G-Code	E Code
WCS #1	G54	E1
WCS #2	G55	E2
WCS #3	G56	E3
WCS #4	G57	E4
WCS #5	G58	E5
WCS #6	G59	E6

Extended Work Coordinate Systems (optional)

WCS	G-Code	E Code
WCS #7	G54 P1	E7
WCS #8	G54 P2	E8
WCS #9	G54 P3	E9
WCS #10	G54 P4	E10
WCS #11	G54 P5	E11
WCS #12	G54 P6	E12

WCS	G-Code	E Code
WCS #13	G54 P7	E13
WCS #14	G54 P8	E14
WCS #15	G54 P9	E15
WCS #16	G54 P10	E16
WCS #17	G54 P11	E17
WCS #18	G54 P12	E18

12.22 G61 – Modal Decelerate and Stop *(formerly known as Exact Stop Mode)*



G61 activates Decelerate and Stop mode for every block processed. This forces motion to decelerate to a stop and invokes a brief dwell (1/100 seconds) at the end of each block (equivalent to G9 in each block). G61 is modal and remains in effect until it is canceled with G64. Note that G61 also turns off Smoothing mode.

Example:

```
G0 X0 Y0 ; move to origin
G61 X2   ; move and decelerate and stop at X2
X4       ; move and decelerate and stop at X4
X5       ; move and decelerate and stop at X5
```

12.23 G64 – Smoothing Mode Selection / Cancel Modal Decel and Stop

G64 has multiple formats with different functionality. Invoking G64 with either an ON or OFF parameter sets the Smoothing mode to on or off, and also cancels Modal Decelerate and Stop (G61). Invoking G64 without either the ON or OFF parameter simply cancels Modal Decelerate and Stop. Note that all forms of G64 will cancel Modal Decelerate and Stop (Cancel G61).

G64	Format Function
G64	Simply cancel Modal Decelerate and Stop (Cancel G61). This does not affect Smoothing mode.
G64 ON	Turn on Smoothing mode using the current parameter settings (in P221-P228, P230, and P231). The effect of this command is temporary as it is active only until the next G64 ON/OFF command or until the end of job. Upon the start of the next job, the the inital on/off state of Smoothing will once again be determined by Parameter 220. This command does not modify any parameters. (This also cancels G61.)
G64 OFF	Turn off Smoothing mode. The effect of this command is temporary as it is active only until the next G64 ON/OFF command or until the end of job. Upon the start of the next job, the the inital on/off state of Smoothing will once again be determined by Parameter 220. This command does not modify any parameters. (This also cancels G61.)
G64 ON "preset label" G64 ON P__ (where P__ is P1-P99)	Turn on and activate Smoothing mode using the specified "preset label" or preset number (P1-P99). (See Chapter 15 for further information on setting up Smoothing Presets.) Note that "preset label" is not case sensitive. You can use the preset number to specify those presets that don't have a "preset label". These commands have a lasting effect beyond the end of the current job because they actually copy the specified Smoothing preset settings into the current parameters (P221-P228, P230, and P231) and sets P220 to 1. (This also cancels G61.)
G64 ON P0 G64 OFF P0	Turn off and deactivate Smoothing mode. Both forms of this command do the same thing. The P0 refers to "Exact Stop" mode. This command sets P220 to 0, and thus has a lasting effect beyond the end of the current job. (This also cancels G61.)

Example:

```
G64                       ; cancel Modal Decelerate and Stop
G64 ON                    ; turn on modal Smoothing mode
G64 OFF                   ; turn off modal Smoothing mode
G64 ON P2                 ; Activate Smoothing Preset #2 by number
G64 ON "contouring mill" ; Activate "Contouring Mill" Preset by label
G64 ON P0                 ; Deactivate Smoothing. Activate Exact Stop mode.
```

12.24 G65 – Call Macro

G65 calls a macro with user-specified values. A macro is a subprogram that executes a certain operation (e.g. drill pattern, contours, etc.) with values assigned to variable parameters within the operation.

Calling methods:

G65 P*xxxx* L*rrrr Arguments*
or
G65 “program.cnc” L*rrrr Arguments*

where *xxxx* is the macro number (referring to file Oxxxx.cnc, 0000–9999 allowed, leading zeros required in filename, capital O, lowercase .cnc), rrrr is the repeat value, “program.cnc” is the name of the macro file, and *Arguments* is a list of variable identifiers and values.

Arguments to macro calls are specified by using letters A-Z, excluding G, L, N, O, and P.

Macros are written just like normal programs. However, macro programs may access their arguments by using #A, #B, etc., or by using numbers: #1 for A, #2 for B, etc. (exceptions: #4–6 for I-K, #7–11 for D-H). Arguments I, J, and K can be used more than once in a macro call, with the first set of values stored as #4–6, the second as #7–9, etc., to a maximum of 10 sets. See example at the end of this G65 section.

Macros 9100–9999 may be embedded into a main program, using O91xx to designate the beginning of the macro and M99 to end it. The CNC software will read the macro and generate a file O91xx.cnc, but will not execute the macro. It will be executed when G65 is issued.

Example 1:

Main Program:

G65 “TEST.cnc” A5 B3 X4

Macro TEST.cnc:

G1 X#X Y#A Z-#B

This call will produce:

G1 X4 Y5 Z-3

Example 2:

Main Program:

G65 “TEST2.cnc” I5 J3 K40 I-1 J2 I0 J0

Macro TEST2.cnc:

G1 X#4 Y#5 F#6

G1 X#7 Y#8 Z#9

G1 X#10 Y#11 Z#12

This call will produce:

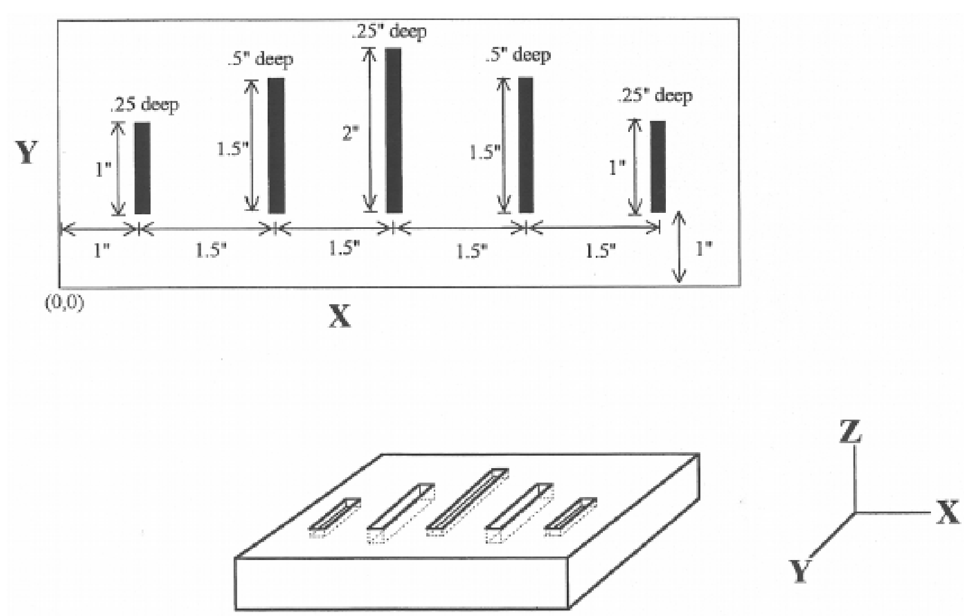
G1 X5 Y3 F40

G1 X-1 Y2 Z0

G1 X0 Y0 Z0

Example 3:

Suppose a piece is to have notches of different lengths and depths along the x-axis:



The macro variables would handle the length in the Y direction and depth in the Z direction:

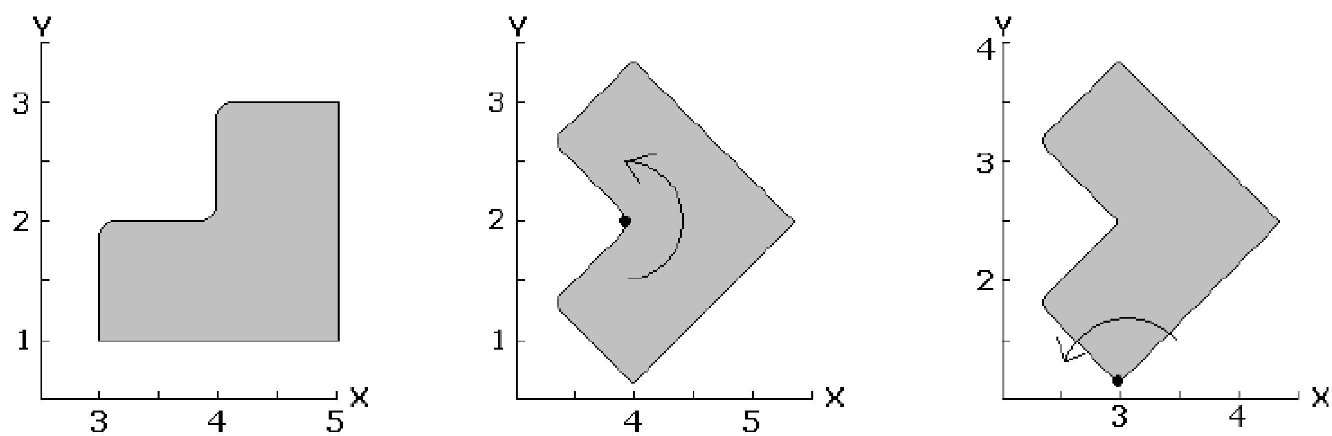
```
O0002
G90 G1 Z0 F30 ; Linear move to Z0
Z#Z F5        ; Cut to variable depth
G91Y#Y F10    ; Cut variable length
G90 G0 Z0.1   ; Retract
```

The main program would call this macro five times, each time specifying the depth and length required.

```
: Main Program
G90 G0 X1 Y1 Z0.1       ; Move to first notch
G65 P0002 L1 Y1 Z.25    ; Call macro and assign Y=1" and Z=.25"
G90 G0 X2.5 Y1          ; Move to second notch
G65 P0002 L1 Y1.5 Z.5 ; Call macro and assign Y=1.5" and Z=.5"
G90 G0 X4 Y1            ; Move to third notch
G65 P0002 L1 Y2 Z.25    ; Call macro again
G90 G0 X5.5 Y1          ; Move to fourth notch
G65 P0002 L1 Y1.5 Z.5 ; Call macro again
G90 G0 X7 Y1            ; Move to fifth notch
G65 P0002 L1 Y1 Z.25    ; Call macro again
: End program
```

12.25 G68, G69 – Coordinate Rotation on/off

G68 rotates program G-codes a specified angle R. G68 rotates all positions, lines, and arcs until a G69 is entered. The center of rotation can be specified by X, Y and Z values (X, Y for G17 plane). If the center is not specified then a default center of rotation is used as determined by machine Parameter 2 (see Chapter 15 for parameter 2). The default plane of rotation is G17 (X, Y).



Example:

```
G68 R45 X4 Y2           ; Rotate 45 degrees centered on X4 Y2
G0 X3.0 Y1.0            ; Rapid to position
G1 X5.0 Y1.0 F20        ; Start part profile
X5.0 Y3.0
X4.125 Y3.0
G3 X4.0 Y2.875 J-0.125
G1 X4.0 Y2.125
G2 X3.875 Y2.0 I-0.125
G1 X3.125 Y2.0
G3 X3.0 Y1.875 J-0.125
G1 X3.0 Y1.0            ; End part profile
G69                     ; Rotate Off
```

12.26 G68.1 – Transformed Work Coordinate System

G68.1 turns on Transformed Work Coordinate System during a job. G69 turns it off. For more information, see the Transformed Work Coordinate System section of the Part Setup Menu chapter.

12.27 G73, G76, G80, G81, G82, G83, G85, G89 – Canned Drilling/Boring Cycles; G74, G84 – Canned Tapping Cycles

G code	-Z direction (machine hole)	Operation at bottom of hole	+Z direction	Use
G73	Intermittent Feed (Set with the Q parameter)	———	Rapid traverse	High speed peck drilling cycle
G74	Feed	Spindle CW, then Dwell (Set with the P parameter)	Feed	Counter tapping (Left-hand thread)
G76	Feed	Dwell (P parameter), Orient Spindle (via M19), Move Y+ (Q parameter)	Rapid traverse, then Stop Spindle Orient (via M5)	Fine Boring Cycle
G80	———	———	———	Cancels canned cycles
G81	Feed	———	Rapid traverse	Regular and spot drilling cycles and air drill cycle
G82	Feed	Dwell (Set with the P parameter)	Rapid traverse	Regular and counter boring cycles, spot facing
G83	Intermittent Feed (Set with the Q parameter)	———	Rapid traverse	Peck and deep hole drilling cycles
G84	Feed	Spindle CCW, then Dwell (Set with the P parameter)	Feed	Tapping (Right hand thread)
G85	Feed	———	Feed	Boring cycle
G89	Feed	Dwell (Set with the P parameter)	Feed	Boring cycle

Table 1. Canned drilling, boring and tapping cycles

Canned Cycle Operation

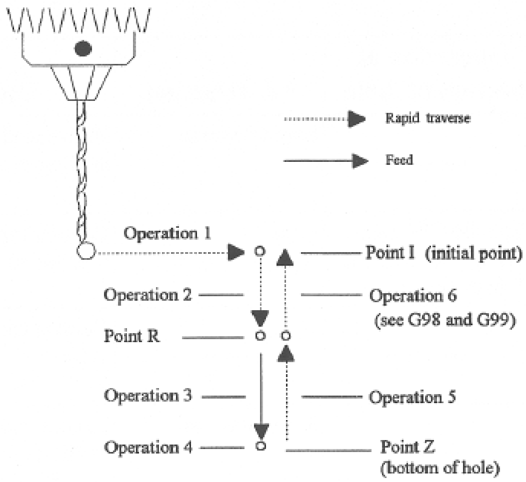
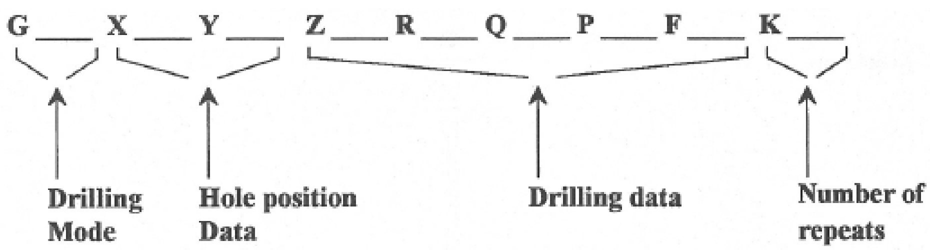


Figure 1. Drilling cycle operation

```
Operation 1: Position
   the X, Y axes.
Operation 2: Rapid
   traverse to the
   position labeled R.
Operation 3: Machine
   hole.
Operation 4: Bottom
   hole operation.
Operation 5: Return to
   point R.
Operation 6: Rapid
   traverse to initial
   point.
```

Canned cycle G-code syntax



(Cycle codes do not have to be on the same line) G ____ Canned cycle G-code from table 1.

X ____ X position of the hole to be drilled.

Y ____ Y position of the hole to be drilled.

Z ____ Specifies point Z in figure 1. In incremental mode Z is measured from point R. In absolute mode Z is the position of the hole bottom.

R ____ Specifies the distance to point R (figure 1) with an absolute or incremental value.

Q ____ Determines the cut-in depth for the G73 and G83 cycles. Determines the thread lead for G74 and G84 if Rigid Tapping is enabled. (In the case of Rigid Tapping Q is not modal)

P ____ Sets the dwell time at the bottom of the holes for G74, G82, G84, and G89 cycles. The dwell time is measured in seconds (same as G04).

F ____ Sets the feed rate. Remains the feedrate even after G80 (cancel canned cycles).

K ____ Sets the number of repeats for drilling cycles. Operations 1 through 6 of figure 1 will be repeated K number of times. If K is not specified K = 1. K is only useful when using incremental positioning mode (G91) and is not retained from cycle to cycle. In absolute mode, K causes the drilling of the same hole in the same position K times.

Note: Canned cycles are modal and should be canceled with G80. However G00, G01, G02 and G03 will also cause the cancellation of canned cycles. All parameters are stored until canned cycles are canceled except for the hole position and K, which must be set each time the cycle is used. When G80 is issued the movement mode will be the last one issued (G0, G1, G2, G3). Canned cycles will not be performed unless X and/or Y are specified.

When performing canned cycle operations, the distances can be either incremental or absolute, depending on the current active mode (G90 = absolute, G91= incremental). Figure 2 illustrates canned cycle Z-axis distances in both modes.

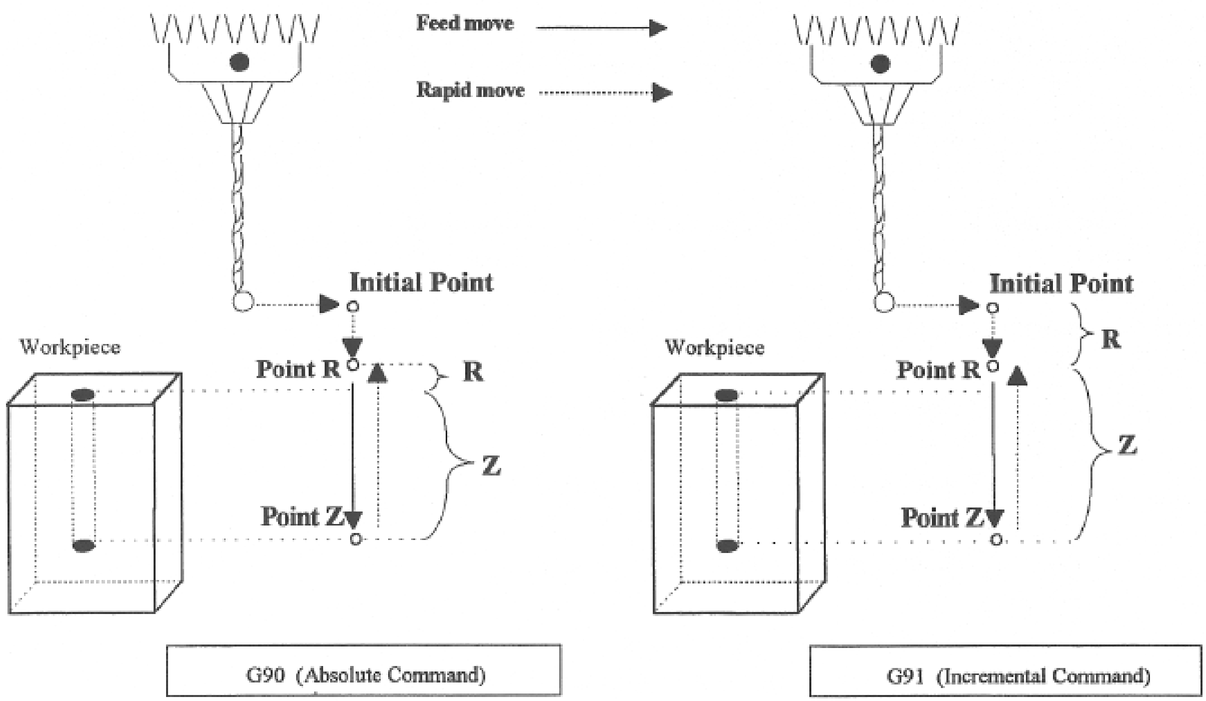


Figure 2: Canned Cycle Absolute and Incremental modes

Note: In incremental mode the Z depth of the hole is measured from R, and R is measured from the initial tool position.

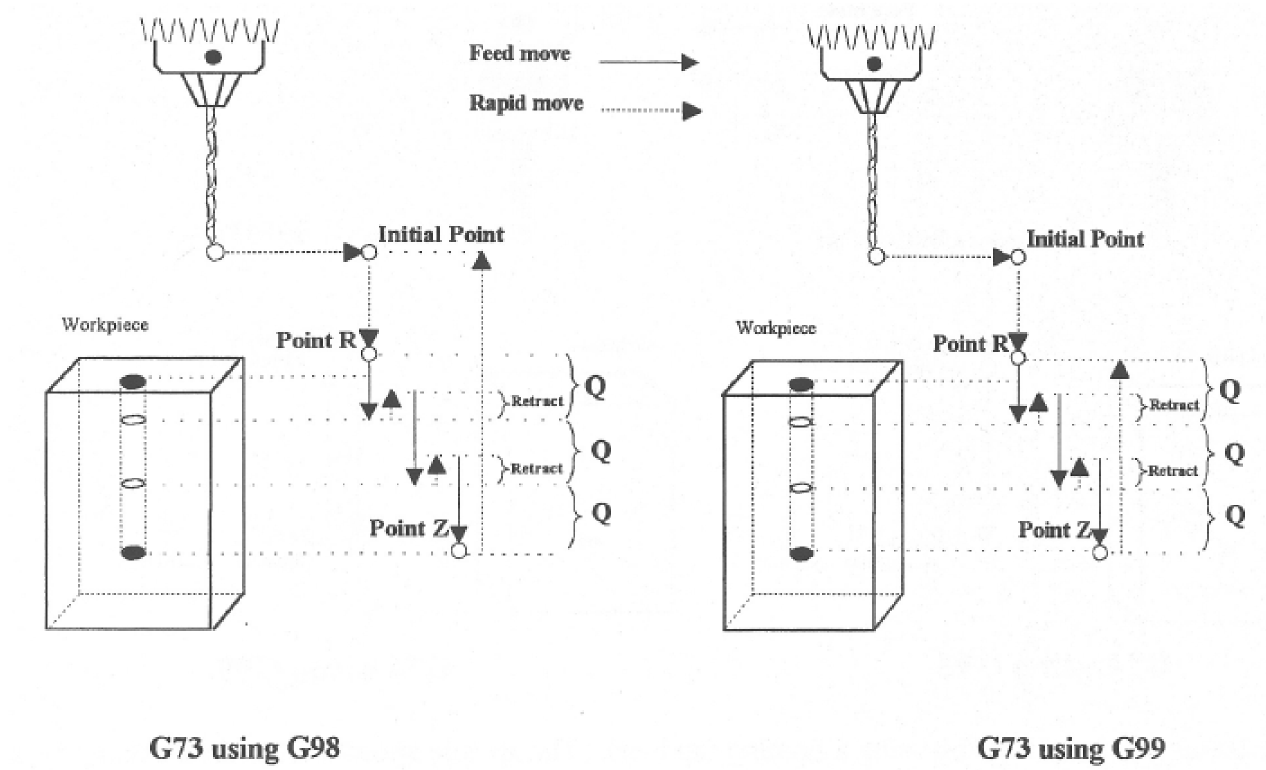
Example:

(Part surface height is Z = 0, initial tool position is X.50 Y1.0 Z.625. Drill 0.50 deep hole at X1.0 Y1.0; clearance height (R) is 0.10 above surface.)

```
Absolute            Incremental
G90                 G91
G81 X1 Y1 R.1 Z-.5  G81 X.5 Y0 R-.525 Z-.6
G80                 G80
```

Note: for Articulated Head machines configured with the TWCS feature enabled via Parameter 166: If the currently selected WCS is non-TWCS (TWCS = No) and the B axis is at an angle other than 0, then you cannot use the regular Canned Cycle G-codes G73, G74, G76, G81, G82, G83, G84, G85, G89. You must use the Compound Canned Cycle G-codes G173, G174, G176, G181, G182, G183, G184, G185, G189 instead. Using regular Canned Cycle G-codes when the B axis is not 0 is an error and will cancel the job. See “G173, G174... – Compound Canned Cycles” later in this chapter for more information about this subject. See Chapter 15 for more information about Parameter 166.

12.28 G73 – High Speed Peck Drilling

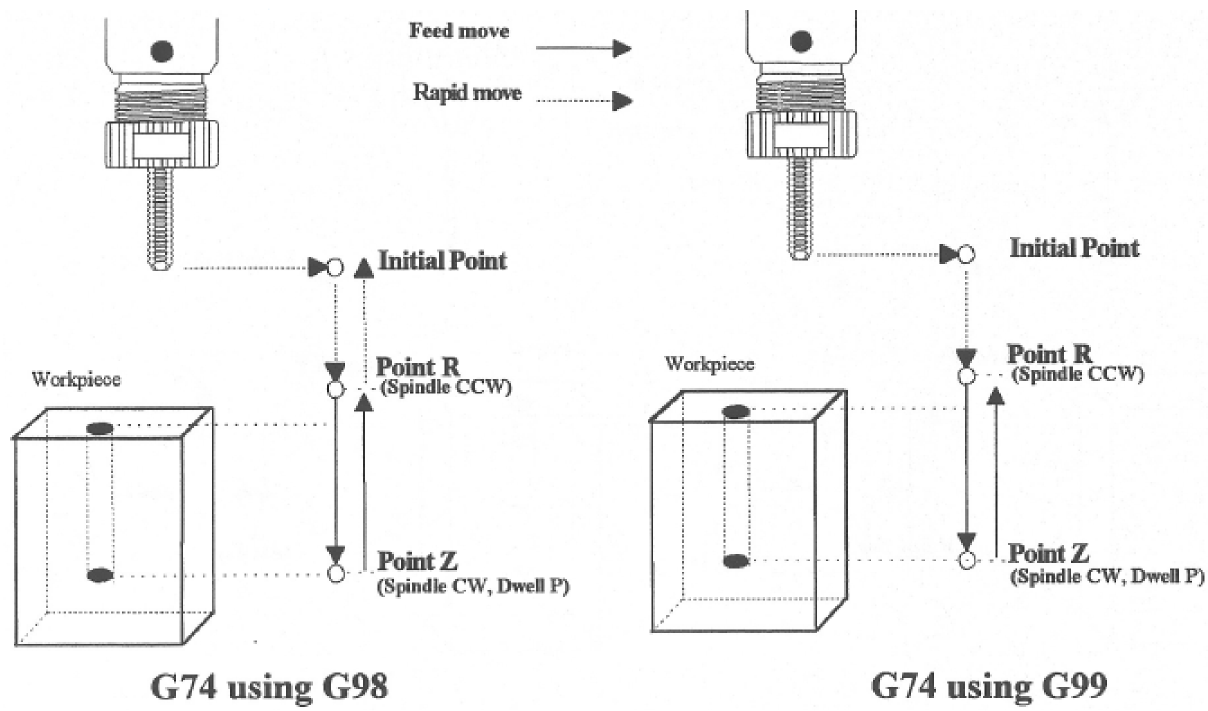


G73 is the peck drilling cycle. The hole is drilled in a series of moves: down a distance Q at a given feedrate, up the retract distance at the rapid rate, and then down again at the given feedrate. The retract amount is set with G10 as shown in the example below.

Example:

```
G90                                    ; Absolute positioning
G01 X3.00 Y1.50 Z.5                    ; G01 mode before canned cycle
G98                                    ; Set for initial point return
G10 P73 R.1                            ; Sets the retract amount to .1
G73 X3.250 Y1.75 Z-.650 R.1 Q0.325 F3 ; Peck drill at X3.25 Y1.75
X4.5 Y3.5                              ; Peck drill at X4.5 Y3.5
G80                                    ; Cancel canned cycle, return
```

12.29 G74 – Counter Tapping



G74 performs left-hand tapping. The spindle speed (and feedrate, if you are doing floating tapping) should be set and the spindle started in the CCW direction before issuing G74. G74 will normally use the default M3 to select spindle CW (at the bottom of the hole) and M4 to re-select spindle CCW (after backing out of the hole) depending on the settings of parameters 74 and 84.

The tap may continue to cut a short distance beyond the programmed Z height as the spindle comes to a stop before reversing. When tapping blind holes, be sure to specify a Z height slightly above the bottom of the hole to prevent the tool from reaching bottom before the spindle stops.

Note: If rigid tapping is enabled, a Q may be used to set the thread lead or pitch. However, because Q is not modal in the case of Rigid Tapping, you must specify Q on every line at which Rigid Tapping is to occur.

Note: At the bottom of the hole, G74 will call the default version of the specified M function even if it has been customized by an M function macro.

WARNING

FEED HOLD is temporarily disabled during the tapping cycle, but it will be re-enabled at the end of the cycle.

NOTICE

Pressing **CYCLE CANCEL** while the tap is in the hole will very probably break the tap or strip the threads in the tap hole. However, do so if it is an emergency.

Example:

```
M4 S500 F27.78      ; start spindle CCW, set up for 18 pitch tap
G74 X1 Y1 R.1 Z-.5 ; counter-tap a 0.5 deep hole at X1 Y1
Y1.5                ; ... and another one at X1 Y1.5
G80                 ; cancel canned cycles
```

12.30 G76 – Fine Bore Cycle

WARNING **G76 requires that the machine be capable of orienting the spindle and that a custom M19 macro is present in order to command the inverter to orient the spindle. Please contact your dealer to confirm that your machine meets these requirements before attempting to use this cycle.**

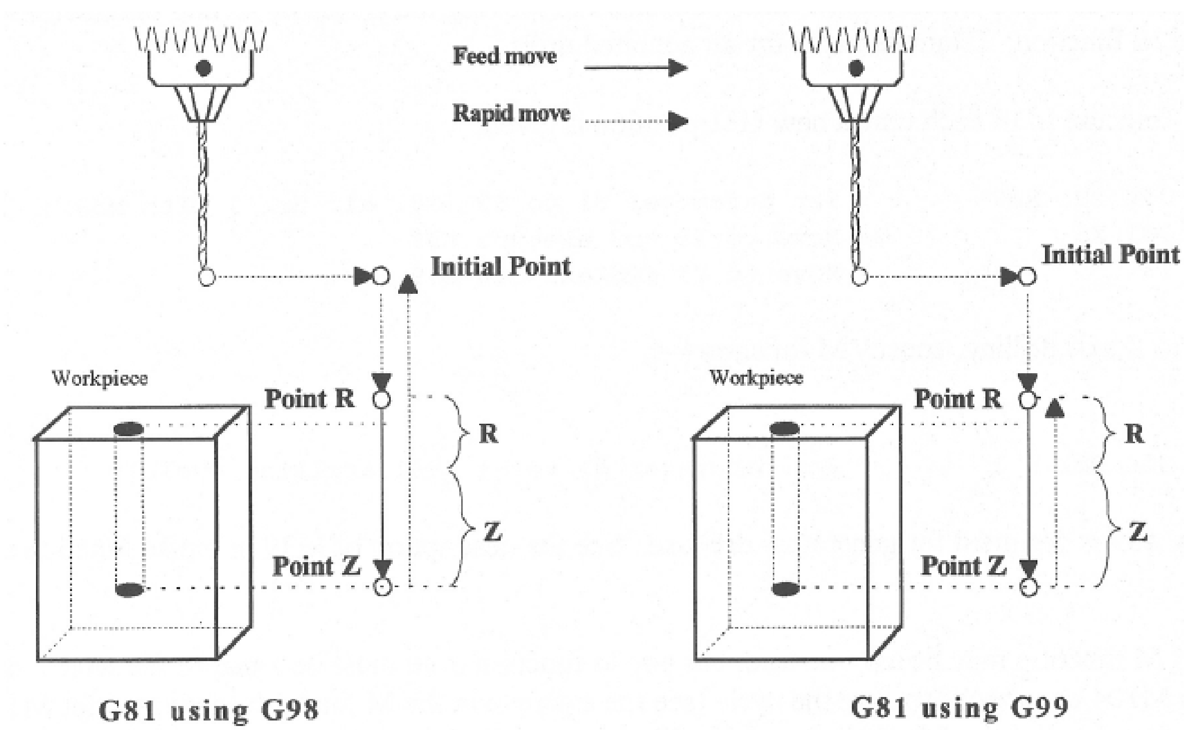
Note: Parameter 136 = Fine Bore retract angle (0–360 degrees). A setting of 0 = Retract in Y+ direction

Format G76 X__ Y__ Z__ R__ Q__ R = Point R Q = Distance to pull away from wall in Y+ direction at bottom of hole.

Example:

```
G76 X1 Y1 Z-3 R.1 Q.2 ; Bore hole at X1 Y1 retract .2 in Y+ direction
Y10                   ; ... and another one at X1 Y10
G80                   ; cancel canned cycles
```

12.31 G81 – Drilling and Spot Drilling



G81 is a general purpose drilling cycle. The hole is drilled in a single feedrate move, and then the tool is retracted at the rapid rate.

Example:

```
G90                          ; Absolute positioning
G01 X3.00 Y1.50 Z.5          ; G01 mode before canned cycle
G99                          ; Set for R point return
G81 X3.250 Y1.75 Z-.650 R.1 F3 ; Drill at X3.25 Y1.75
X4.5 Y3.5                    ; Drill at X4.5 Y3.5
G80                          ; Cancel canned cycle, return to G1
```

12.32 G81 – Drill Cycle Transformation to G81 Air Drill Cycle

G81 may be modified to execute an M function instead of moving the Z-axis by setting Parameter 81 to the desired M function. Example use is for air-actuated drills.

Example: Execute M39 each time a new G81 position is given:

```
G10 P81 R39 ; Set parameter 81 to 39 (G81 air drill with M39)
G81 X5      ; Move to X5 and execute M39
Y3          ; Move to Y3 and execute M39
```

To revert to Z-axis drilling, specify M function #-1.

Example:

```
G10 P81 R-1 ; Set parameter 81 to -1 (G81 drilling cycle)
```

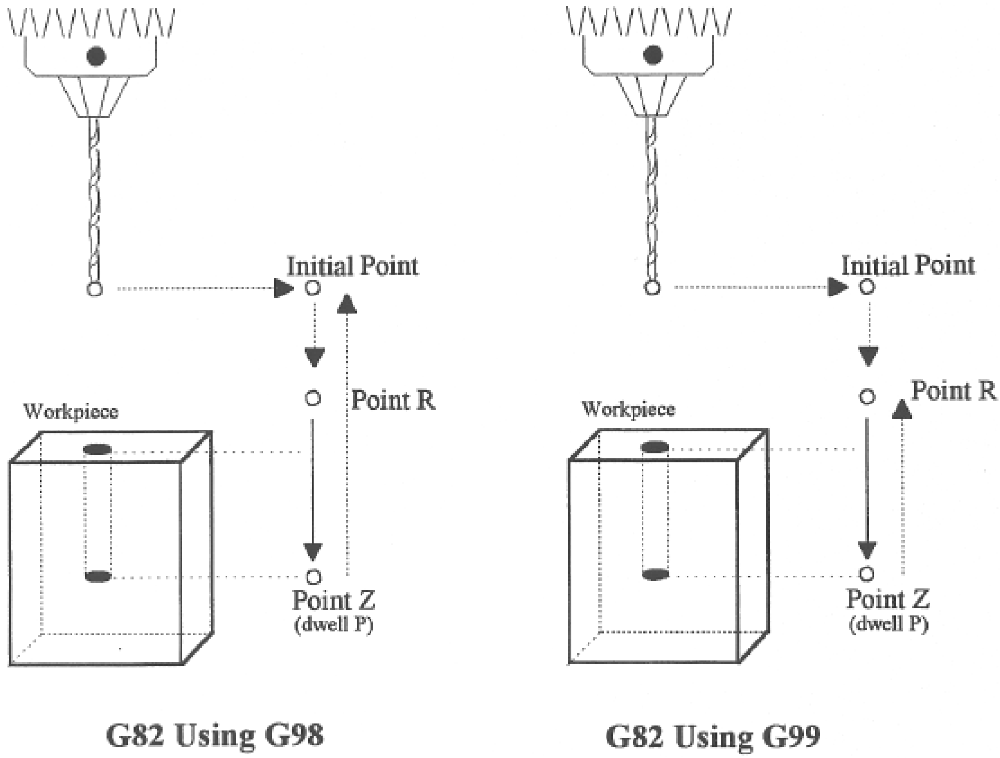
M function #39 is designed for general air drill use. See the description of M39 in the M functions section.

A different M function may be used instead, but any M function used must be a macro file that uses the M103 and M104 commands to time the cycle (see the example in the M function section under M103). If the macro file does not use M103, the control will automatically cancel the job 1/2 second after starting G81. For information on creating customized M functions, review Macro M functions in Chapter 13.

The M39 default air drill cycle has a time out of 2.0 seconds. As a result, if the cycle does not complete within 2 seconds then the cycle aborts and the output relay is turned off under PLC program control.

Note: The PLC program must be involved in the execution of the cycle. The PLC program is responsible for turning on relays based on M function requests and the status of program execution. The PLC program must also stop all programmed machine functions when the program is canceled. See the M39 description (Chapter 13) for a sample of an air drill cycle M function.

12.33 G82 – Drill With Dwell

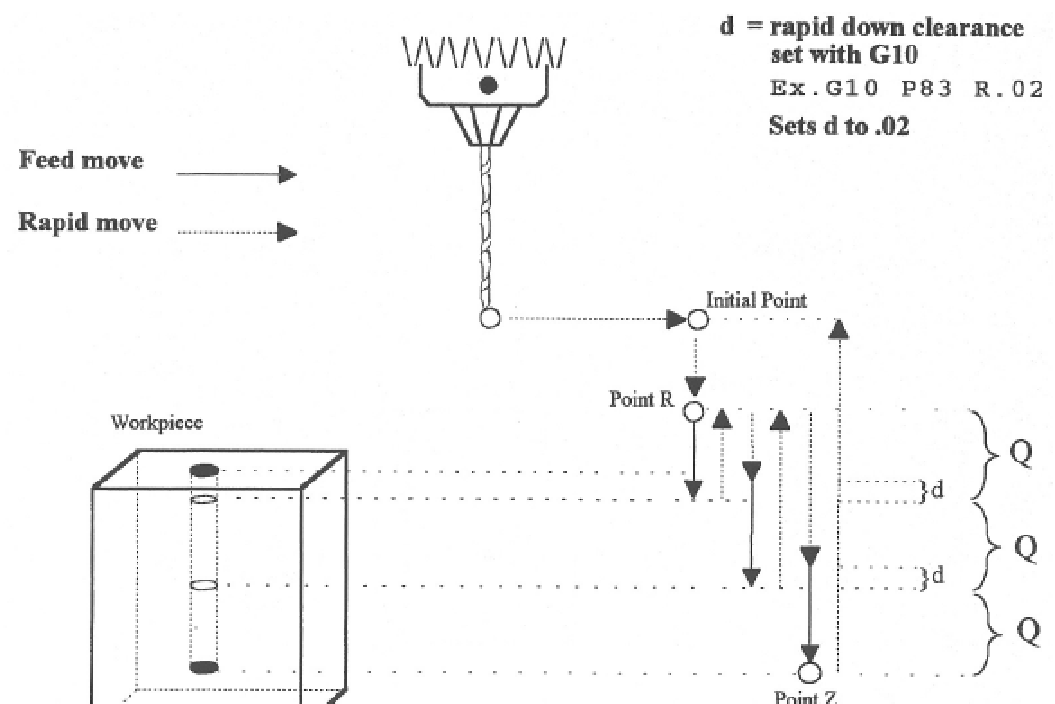


G82 is a general purpose drilling cycle similar to G81. However, G82 includes an optional dwell at the bottom of the hole before retracting the tool. This can make the depth of blind holes more accurate.

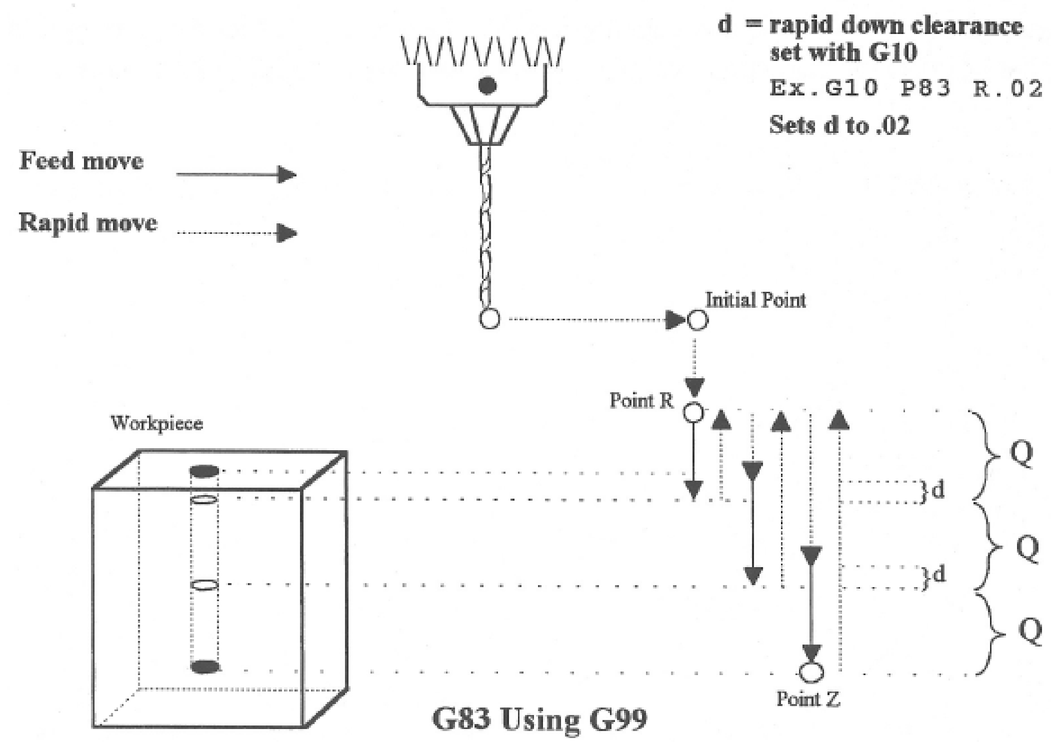
Example:

```
G82 X1 Y1 R.1 Z-.5 P.5 ; drill to Z-.5, dwell .5 seconds
```

12.34 G83 – Deep Hole Drilling



G83 Using G98



G83 Using G99

G83 is a deep hole drilling cycle. It periodically retracts the tool to the surface to clear accumulated chips, then returns to resume drilling where it left off. The retract and return are performed at the rapid rate. Because there may be chips in the bottom of the hole, the tool does not return all the way to the bottom at the rapid rate. Instead it slows to feedrate a short distance above the bottom. This clearance distance is selected by setting Parameter 83 with G10 (see example below).

2 Line Format

Line 1: `G10 P83 R_*`

Line 2: `G83 X_ Z_ Q_ R_ L_`

Line 1 – R = Z Rapid to Clearance Amount *

Line 2 – X = X Position

Line 2 – Z = Final Z depth

Line 2 – Q = Z Peck Cutting Increment †

Line 2 – R = R(Z) Return point for clearance

Line 2 – L = Dwell time at each Final Z depth

* The modal default for Z Rapid to Clearance Amount is from Parameter 83. If Line 1 is not present, the default value in Parameter 83 will be used.

† Q value is subject to the "implied floating point" re-interpretation rules of Parameters 245 and 246,

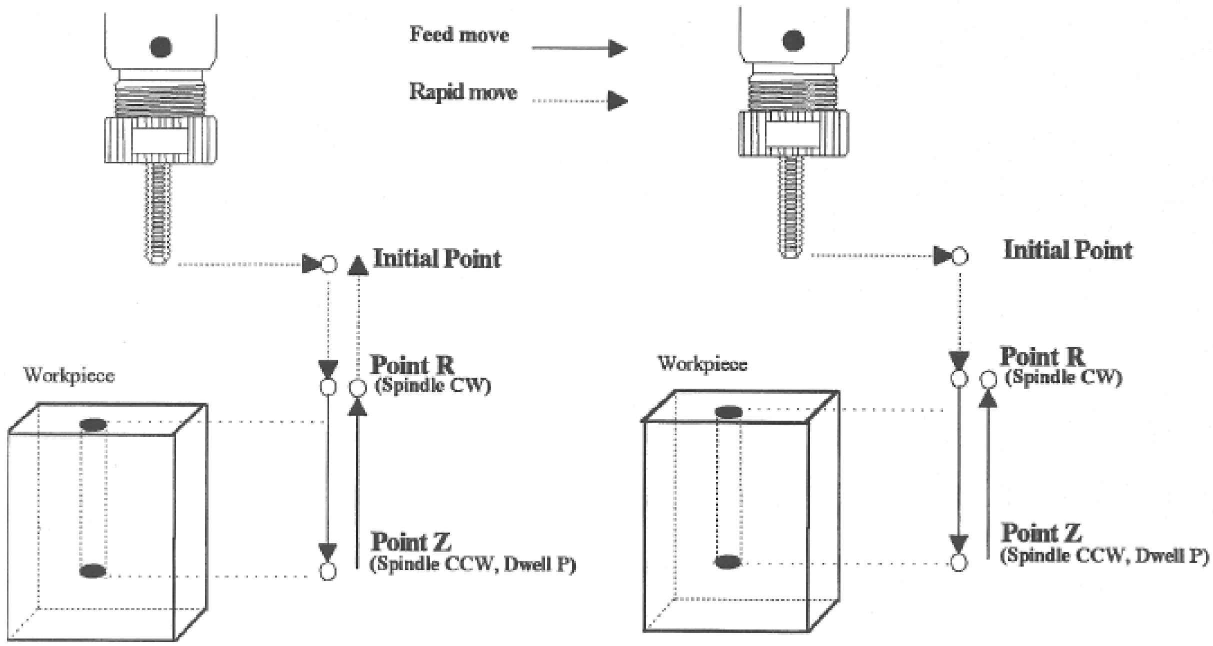
Final Z depth, the Z Peck Cutting Increment (Q), and Return Point (R) are the minimal required parameters. All others are optional. If X is not specified, the last X position will be used. X should always X0.0 unless the machine is set up with C-axis and live tooling.

Dwell Time (L) should be left out entirely if no dwell is desired, because L0 will still cause a (albeit very brief) dwell.

Example:

```
G10 P83 R.05          ; set clearance to .05"
G83 X0 Y0 R.1 Z-2 Q.5 ; drill 2" deep hole in 0.5" steps
G80                   ; cancel canned cycle
```

12.35 G84 – Tapping



G84 using G98 G84 using G99

G84 performs right-hand tapping. The spindle speed (and feedrate, if you are doing floating tapping) should be set and the spindle started in the CW direction before issuing G84. G84 will normally use the default M4 to select spindle CCW (at the bottom of the hole) and M3 to re-select spindle CW (after backing out of the hole) depending on the settings of parameters 74 and 84.

The tap may continue to cut a short distance beyond the programmed Z height as the spindle comes to a stop before reversing. When tapping blind holes, be sure to specify a Z height slightly above the bottom of the hole to prevent the tool from reaching bottom before the spindle stops.

For a floating tap head, the combination of the modal feedrate and spindle speed implicitly determines the approximate thread lead or pitch.

Note: If rigid tapping is enabled, a Q may be used to set the thread lead or pitch. However, because Q is not modal in the case of Rigid Tapping, you must specify Q on every line at which Rigid Tapping is to occur.

Note: At the bottom of the hole, G84 will call the default version of the specified M function even if it has been customized by an M function macro.

The Tap/Counter Tap cycle might cut a short distance beyond the programmed Z height as the spindle comes to a stop before reversing. When tapping blind holes, be sure to specify a Z height slightly above the bottom of the hole to prevent the tool from reaching bottom before the spindle stops. The exact distance you must allow will depend on your machine and the diameter and pitch of the tapping tool.

WARNING FEED HOLD is temporarily disabled during the tapping cycle, but it will be reenabled at the end of the cycle.

NOTICE Pressing **CYCLE CANCEL** while the tap is in the hole will very probably break the tap or strip the threads in the tap hole. However, do so if it is an emergency.

Example:

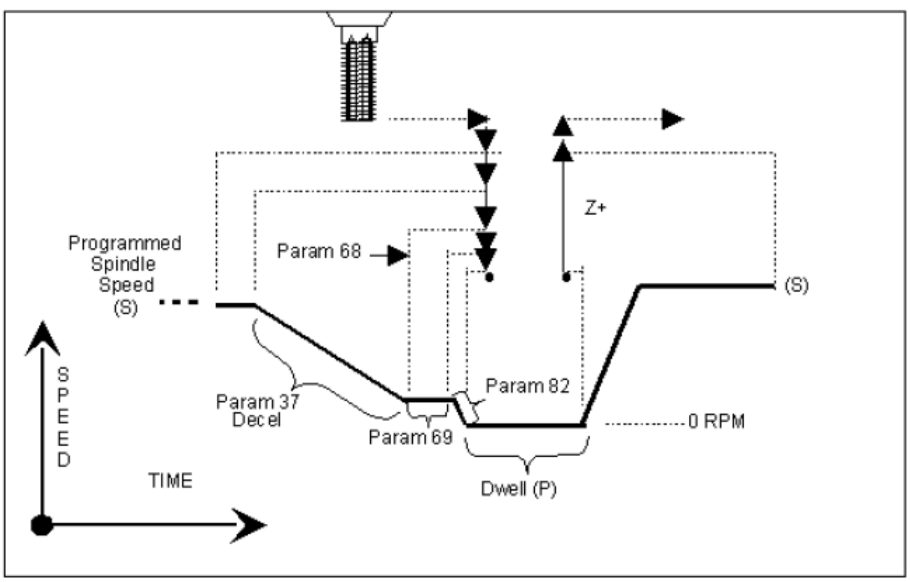
```
M3 S500 F27.78                          ; start spindle CW, set up for 18 pitch
   tap
G84 X1 Y1 R.1 Z-.5                      ; tap a 0.5 deep hole at X1 Y1
Y1.5                                    ; ... and another one at X1 Y1.5
G80                                     ; cancel canned cycle
...
                                        ; Using Tool 15 Rigid Tap 6--32
G84 X0.337 Y0.925 Q0.03125 Z-0.35 R0.1 ; tap first hole
G84 X3.312 Y0.925 Q0.03125 Z-0.35 R0.1 ; tap second hole, must use Q
G80                                     ; cancel canned cycle
...
                                        ; Using Tool 22 Rigid Tap 4--40
G84 X1.862 Y1.627 Q0.025 Z-0.19 R0.1   ; tap first hole
G84 X2.862 Y1.627 Q0.025 Z-0.19 R0.1   ; tap second hole, must use Q
G84 X4.262 Y1.627 Q0.025 Z-0.19 R0.1   ; tap third hole, must use Q
G80                                     ; cancel canned cycle
```

12.35.1 Tech Tip – How to Setup Rigid Tapping

Overview

This section describes the theory of rigid tapping parameters, to control accuracy of depth of cut and quality of threads, in various working materials.

Graphic representation of parameter controls

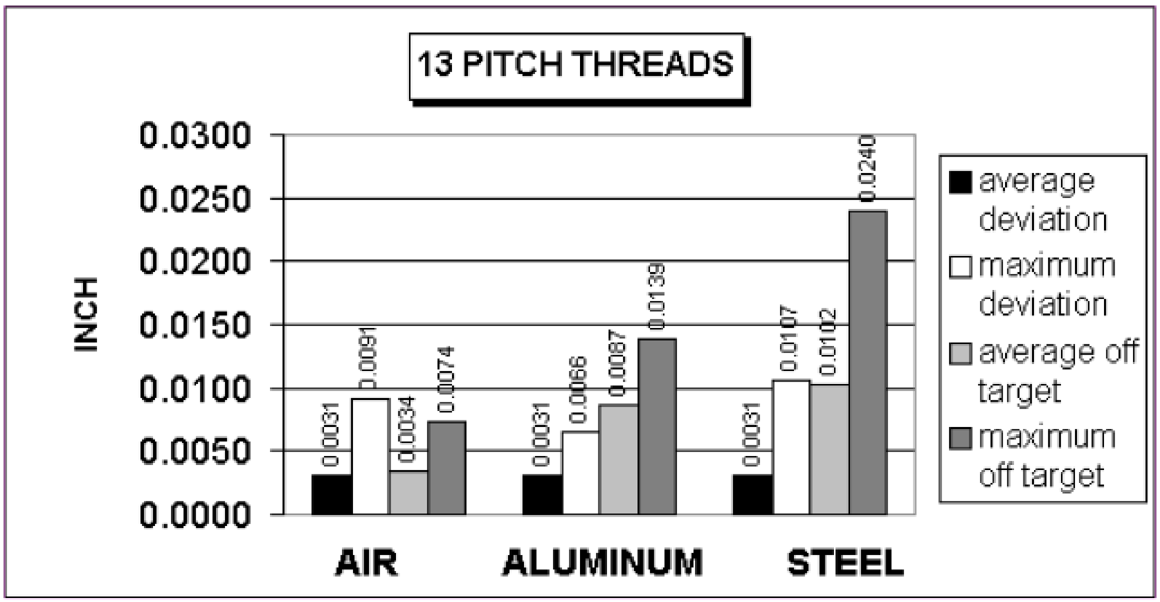


List of Rigid tapping setup parameters

See Chapter 15 for more details.

Parameter	Function
34	Spindle Encoder Counts/Rev
35	Spindle Encoder Axis Number
36	Rigid Tapping Enable/Disable
37	Spindle Deceleration Time
68	Minimum Rigid Tapping Spindle Speed
69	Duration For Minimum Spindle Speed
74	M-Function executed at bottom of tapping cycle
84	M-Function executed at return to initial point of tapping cycle
82	Spindle Drift Adjustment

12.35.2 Graphic representation of test results for precision



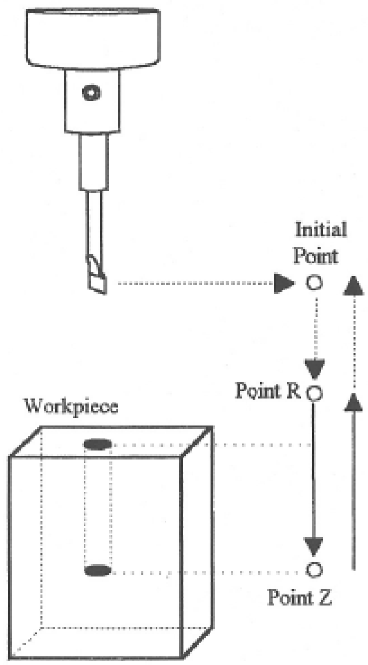
The above charts show test results of rigid tapping, utilizing version 7.14 software. The tool used in the testing was a 1/2–13 spiral fluted tap with TiN coating. Coolant used was water base soluble oil. Hole size was .4218. Tapping depth was .800. Also note that the parameters were adjusted to cut air, and not changed for aluminum or cold rolled steel for these tests. It can be seen, as the material changes, so does the off target values. This is due in part to the amount of torque required from the spindle to cut the various types of material. For testing purposes, the parameter settings for the above results were as follows:

Parameter 36 = 1, Parameter 37 = 3, Parameter 68 = 100, Parameter 69 = 1.25, Parameter 82 = 108

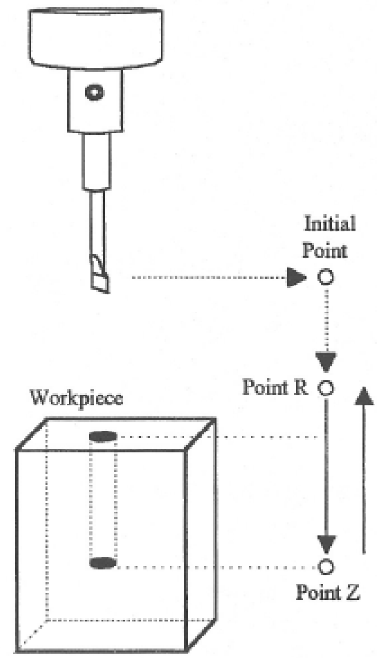
Summary

Rigid tapping parameters will vary from machine to machine. Not all machines are built the same (i.e. Spindle hp, inverter type, rigidity, etc.), and tooling will play a roll in performance also. It was found through our testing, if we changed one physical parameter, (i.e. using a tapping oil instead of water base coolant), it improved the off target values by 1.5%. This is due to the fact that less friction is present when using special cutting oil, therefore requiring less hp by the spindle to drive the tap. In most cases, rigid tapping depths should be able to be held within +/- .008 inch or less by adjusting Parameter 82 for specific cases.

12.36 G85 – Boring



G85 Using G98



G85 Using G99

G85 is similar to G81, except that the tool is retracted with a feedrate move instead of a rapid move. G85 may be used for tapping with reversing tap heads such as the Tapmatic NCR series.

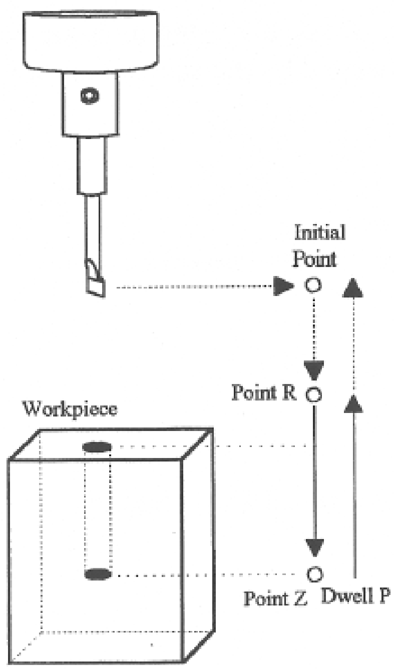
Example 1:

```
G85 X1 Y1 R.1 Z-.5 ; bore a 0.5" hole at X1 Y1
G80                ; cancel canned cycle
```

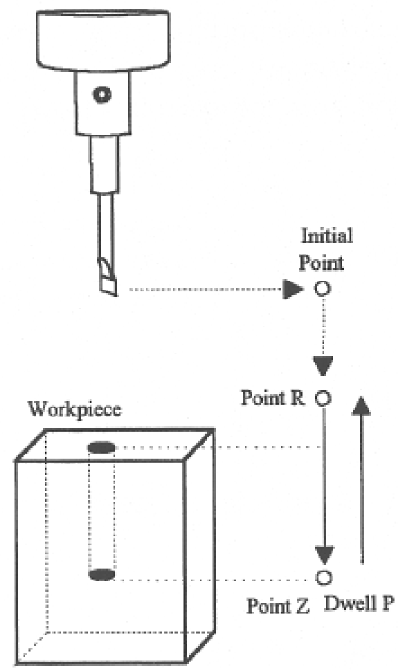
Example 2:

```
M3 S500 F27.78     ; start spindle CW, set for 18 pitch tap
M109/1/2           ; disable feedrate and spindle overrides
G85 X1 Y1 R.1 Z-.4 ; tap hole at X1 Y1 to a depth of 0.4"
M108/1/2           ; enable feedrate and spindle overrides
G80                ; cancel canned cycle
```

12.37 G89 – Boring cycle with dwell



G89 Using G98



G89 Using G99

G89 is similar to G85, except that it includes an optional dwell at the bottom of the hole before retracting the tool.

Example:

```
G89 X1 Y1 R.1 Z-.5 P.1 ; bore 0.5" hole at X1 Y1, dwell .1 seconds
G80                    ; cancel canned cycle
```

12.38 G90 & G91 – Absolute/Incremental Positioning Mode

G90 selects absolute positioning, and G91 selects incremental positioning. In absolute positioning, all coordinates are relative to the origin (0,0,0,0). In incremental positioning, all coordinates are distances relative to the last point.

G90 Absolute positioning
G91 Incremental coordinates

Example:

```
G90 X2 Y3 ; moves the X and Y axes from the current position
          ; to X2, Y3.
G91 X1 Y0 ; moves the X axis 1 inch referenced from the last X
          ; position, the Y axis does not move.
```

12.39 G92 – Set Absolute Position

G92 sets the current absolute position to the coordinates specified. This command only affects the currently set Work Coordinate System.

Example:

```
G0 X5 Y3 Z-2 W5 ; Moves to the specified location
G92 X1 Y0 Z0 W1 ; Sets the current position to the absolute
                ; position specified.
```

12.40 G93 – Inverse Time

Rather than using a conventional federate in Inch per Minute or MM per Minute, F in inverse time mode specifies the movement frequency for subsequent moves. Specifically, the inverse time feedrate is the inverse of the amount of time that a move is allowed to take.

For example, using inverse time, the block:
G1 G93 X10 F1

takes 1 minute to cut, regardless of X length. At F2.0 (1/2.0 minute) it takes 30 seconds; At F3.0 it takes 20 seconds, and so on.

G93 is Modal, and remains in effect until a G94 is issued to cancel the G93.

Example of use in a program:

```
G0 G54 G90 X2.2126 Y-1.1995 A94.75 B-.161 S3000 M3 ; Move to start
G43 H26 Z7.0002 M8
Z3.1002
G1 G93 X2.2048 Y-1.2593 Z3.0204 F100.              ; Enable Inverse Time
X2.2052 Y-1.2586 Z3.0202 A94.756 B-.162 F30000.    ; 1/30000 min for move
X2.2079 Y-1.2578 Z3.0197 A94.763 B-.173 F30000.    ; 1/30000 min for move
X2.2124 Y-1.2566 Z3.0189 A94.773 B-.189 F30000.    ; 1/30000 min for move
X2.2184 Y-1.2551 Z3.0179 A94.786 B-.212 F24065.56  ; 1/24065 min for move
X2.2258 Y-1.2533 Z3.0167 A94.802 B-.24 F19736.14   ; 1/19736 min for move
X2.2345 Y-1.2512 Z3.0155 A94.819 B-.272 F17187.45  ; 1/17187 min for move
```

Draft: June 12, 2023

12.41 G93.1 – Velocity Scrubber for Smoothed Inverse Time Data

This special feedrate interpretation mode substitutes inverse time feedrates (usually posted from a CAD/CAM program) with optimized feedrates to ensure that the tool tip center is moved at a set feedrate in physical 3D space, taking into account tool length offset and the machine geometry as set by parameters 116–119 (see Chapter 15) . The set feedrate (F) can be on the same G-code line as the G93.1 or can be the last modal feedrate specified on prior G-code lines.

This mode is intended for CNC jobs run on machines configured with a triangular rotary 5th axis such as a Tilt Table or Articulated Head (see Parameter 166 in Chapter 15), although that is not an enforced definite requirement to turn on this mode. It is recommended that moves programmed with this mode should be in small vectors, and any long moves that require a sweep of a rotary axis by more than 10 degrees should be broken up into 2 or more smaller moves. Note that Smoothing must also be turned on (P220 = 1) for this feature to work; otherwise, it will be treated as a regular G93 (Inverse Time).

Example of use in a program (based on the example from G93 above):

```
G0 G54 G90 X2.2126 Y-1.1995 A94.75 B-.161 S3000 M3           ; Move to start
G43 H26 Z7.0002 M8
```

```
Z3.1002
G1 G93.1 X2.2048 Y-1.2593 Z3.0204 F150. ; Force tool tip to move 150 in/min
X2.2052 Y-1.2586 Z3.0202 A94.756 B-.162 F30000.                ; Ignore F..Move
   at 150ipm
X2.2079 Y-1.2578 Z3.0197 A94.763 B-.173 F30000.                ; Ignore F..Move
   at 150ipm
X2.2124 Y-1.2566 Z3.0189 A94.773 B-.189 F30000.                ; Ignore F..Move
   at 150ipm
X2.2184 Y-1.2551 Z3.0179 A94.786 B-.212 F24065.56              ; Ignore F..Move
   at 150ipm
X2.2258 Y-1.2533 Z3.0167 A94.802 B-.24 F19736.14               ; Ignore F..Move
   at 150ipm
X2.2345 Y-1.2512 Z3.0155 A94.819 B-.272 F17187.45              ; Ignore F..Move
   at 150ipm
```

12.42 G94 – Cancel Inverse Time

G94 is used to cancel Inverse Time feedrates, and return to regular Feed per Minute feedrates.

12.43 G98 – Initial Point Return

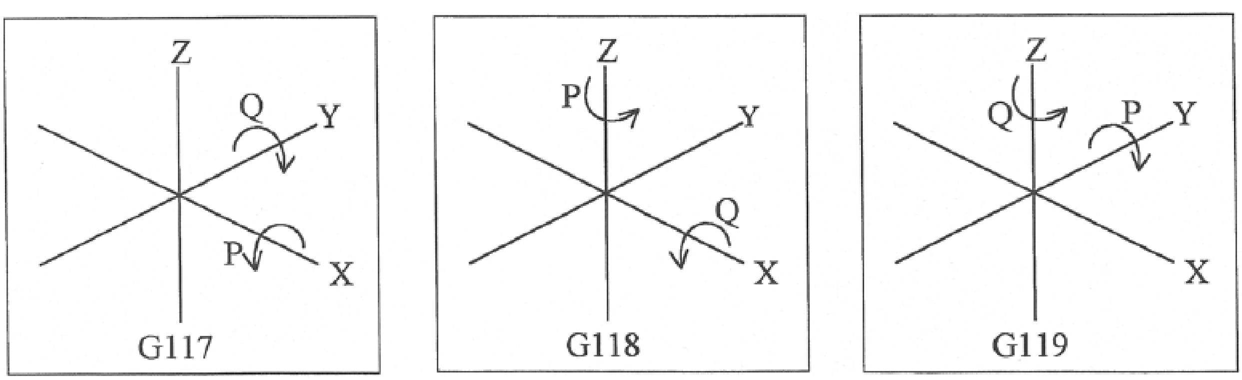
G98 sets the +Z return level to point I as pictured in Figure 1 in the Canned Cycle Section. (G98 is the default setting)

12.44 G99 – R Point Return

G99 sets the +Z return level to point R as pictured in Figure 1 in the Canned Cycle Section.

12.45 G117, G118, G119 – Rotation of Pre-set Arc Planes

G117, G118 and G119 have the same functionality as G17, G18 and G19, respectively, except that they include 2 optional parameters P and Q to specify the arc plane rotation away from the pre-set arc plane: P specifies the arc plane angle of rotation (in degrees) around the first axis and Q specifies the arc plane angle of rotation around the second axis.



For the G117 plane, the “first axis” is X and the “second axis” is Y.

For the G118 plane, the “first axis” is Z and the “second axis” is X.

For the G119 plane, the “first axis” is Y and the “second axis” is Z. If P and/or Q are not specified, the angles are assumed to be 0 degrees. If both P and Q parameters are 0, then the plane is assumed to be an orthogonal (pre-set) arc plane. The center of the arc can be specified by the user in a 3D form both in G17-G19 and in G117-G119 (all I, J, K values are allowed at the same time with G2 and G3). Any arc center component outside the circular plane is ignored.

Example:

```
G00 X0 Y0 Z1                             ; rapid move
G03 G18 X1 Y0 Z0 K-1 F20                 ; arc mill
G00 X0 Y0 Z1.1                           ; retract move
G01 Z1                                   ; move to start of contour
G03 G118 P1.000000 X0.9998 Y0.0175 Z0 K-1 ; arc mill rotated about Z
```

Note: G117-G119 will not be permitted while cutter compensation is turned on. Also, scaling is not allowed while G117-G119 is specified and G117-G119 is not allowed while scaling is active.

Note: G117-G119 will not work when Smoothing is turned on (P220 = 1).

12.46 G173, G174, G176, G181, G182, G183, G184, G185, G189 – Compound Canned Cycles

On a machine configured as an Articulated Head machine with the TWCS feature enabled, the Compound Canned Cycle G-codes are used to perform tilted-head Drill/Bore/Tap operations when the currently selected WCS is not transformed (TWCS=No). See Parameter 166 in Chapter 15 for information on Articulated Head configuration and turning on the TWCS feature for the machine.

On a such a machine, Compound Canned Cycles are used as an alternative to the their corresponding noncompound Canned Cycles because the non-compound versions cannot be used when the currently selected WCS is not transformed (TWCS=No) and the spindle head is tilted off vertical (B/5th axis angle is not 0). In all other situations, Compound Canned Cycles are the same in functionality as their corresponding non-compound counterparts.

Compound Canned Cycle G-code	Function	Corresponding non-compound Canned Cycle G-code
G173	Compound High Speed Peck Drilling	G73
G174	Compound Counter Tapping	G74
G176	Compound Fine Bore Cycle	G76
G181	Compound Drilling and Spot Drilling	G81
G182	Compound Drill with Dwell	G82
G183	Compound Deep Hole Drilling	G83
G184	Compound Tapping	G84
G185	Compound Boring	G85
G189	Compound Boring with Dwell	G89

12.47 G180 – Cancel Canned Cycles

G180 has the exact same functionality as G80.

13 CNC Program Codes: M-Functions

M functions are used to perform specialized actions in CNC programs. Most of the M-series Control M functions have default actions, but can be customized with the use of macro files.

13.1 Summary of M Functions

M00	Stop for Operator
M01	Optional Stop for Operator
M02	Restart Program
M03	Spindle On Clockwise
M04	Spindle On Counterclockwise
M05	Spindle Stop
M06	Tool Change
M07	Mist Coolant On
M08	Flood Coolant On
M09	Coolant Off
M10	Clamp On
M11	Clamp Off
M17	Prepare for Tool Change (Macro)
M19	Spindle Orient (Macro)
M25	Move to Z Home
M26	Set Axis Home
M30	Custom M Code
M39	Air Drill
M41, M42, M43	Select Spindle Gear Range (Macros)
M60	5-Axis Digitizing Macro
M91	Move to Minus Home
M92	Move to Plus Home
M93	Release/Restore Motor Power
M94/M95	Output On/Off
M98	Call Subprogram
M99	Return from Macro or Subprogram
M100	Wait for PLC bit (Open, Off, Reset)
M101	Wait for PLC bit (Closed, On, Set)
M102	Restart Program
M103	Programmed Action Timer
M104	Cancel Programmed Action Timer
M105	Move Minus to Switch
M106	Move Plus to Switch
M107	Output Tool Number
M108	Enable Override Controls
M109	Disable Override Controls
M115/M116/M125/M126	Protected Move Probing Functions
M115/M116/M125/M126	DSP Probe specific information
M120	Open data file (overwrite existing file)
M121	Open data file (append to existing file)

M122	Record local position(s) and optional comment in data file
M123	Record value and/or comment in data file
M124	Record machine position(s) and optional comment in data file
M127	Record Date and Time in a data file
M128	Move Axis by Encoder Counts
M129	Record Current Job file path to data file
M130	Run system command
M150	Set Spindle Encoder to zero at next index pulse
M200, M223, M224, M225 & M290	Formatted String Commands
M200/M201	Stop for Operator, Prompt for Action
M223	Write Formatted String to File
M224	Prompt for Operator Input Using Formatted String
M225	Display Formatted String for A Period of Time
M290	Digitize Profile (Optional)
M300	Fast Synchronous I/O update
M333	Axis Role Re-assignment
M1000-M1015	Graphing Color for Feedrate movement

* M functions marked with "(macro)" actually have no standard default action, and could possibly be unimplemented and therefore unavailable on your machine. Also, their stated function is only standard on certain machines.

13.2 Macro M functions (Custom M Functions)

Most M-Series CNC M functions from 0 through 90 can be fully customized. Exceptions are M2, M6, and M25 that can be customized, but will always move the 3rd (Z) axis to the home position before executing the macro M function commands. No M functions above 90 may be customized with macros. The default action listed will be performed unless that M function has been customized.

To create a macro for an M-function, a file must be created in the C:\cncm directory. The file's name must be mfuncXX.mac where XX is the M function number used to call the macro. M functions 0–9 must use single digits in the filename (e.g. use mfunc3.mac, **not** mfunc03.mac). The contents of the file may be any valid M and G codes.

Each time the M function is encountered in a program, the macro file will be processed line by line.

Note: Nesting of macro M functions is allowed. Recursive calls are not: if a macro M function calls itself, the default action of the function will be executed.

Example:

Turn on spindle with variable frequency drive and wait for "at speed" response. This example depends on M94/M95 1 being used for the Spindle Enable signal and input 5 being the spindle at speed signal in the PLC program.

Create file c:\cncm\mfunc3.mac with contents as follows:

```
M94/1          ; request spindle start
M101/50005     ; wait for up to speed signal
```

M2, M6, and M25 **always** move the Z-axis to the home position before any other motion. All other M functions are performed after the motion of the current line is complete.

The M and G codes in a macro M function are not usually displayed on the screen as they are executed, and are all treated as one operation in block mode. If you wish to see or step through macro M functions (e.g. for testing purposes), set Parameter 10 as follows:

0	Don't display or step through macro M functions
1	Display macro M functions, but don't step through them
2	Display and step through macro M functions

Notes:

1. You cannot use block mode to step through a macro M function called using the G81 transformation (see Chapter 12); the action timer will expire before you can press **CYCLE START**.
2. Only one M function per line is permitted.
3. Unlike subprograms invoked with G65, macro arguments passed into a macro M function are passed in by reference. This means local changes to #1 through #33 or #A through #Z will change their values upon return from the macro M function.

13.3 M00 – Stop for Operator

Motion stops and the operator is prompted to press the **CYCLE START** button to continue.

13.4 M01 – Optional Stop for Operator

M1 is an optional pause, whose action can be selected by the operator.

When optional stops are turned on, M1 will pause the currently running job until **CYCLE START** is pressed. However, if optional stops are turned off, M1 will not pause the program.

Note: If you plan to override the default action of M1 with a macro file, you may want to include a call to M1 within the macro file, so that the default actions of M1 will still be effective in the overridden M1. Otherwise, if a call to M1 is not included within the macro file, the new overridden M1 will cause optional stops to be ineffective.

13.5 M02 – Restart Program

M2 moves the Z-axis to the home position, performs any movement requested, and restarts the program from the first line. The operator is then prompted to press the **CYCLE START** button to continue.

13.6 M03 – Spindle On Clockwise

M3 requests the PLC to start the spindle in the clockwise direction.

Default action:

```
M95/2

M94/1
```

13.7 M04 – Spindle On Counterclockwise

M4 requests the PLC to start the spindle in the counterclockwise direction.

Default action:

```
M95/1
M94/2
```

13.8 M05 – Spindle Stop

M5 requests the PLC to stop the spindle.

Default action if the spindle had been spinning CW:

```
M95/2
M95/1
```


Default action if the spindle was OFF or was spinning CCW:

```
M95/1
M95/2
```

13.9 M06 – Tool Change

M6 moves the Z-axis to the home position and stops the spindle and coolant. If an automatic tool changer is installed, it then commands the tool changer to switch to the requested tool. Otherwise, it prompts the operator to insert the tool and then press the **CYCLE START** button on the Operator Panel.

Default action (no tool changer):

```
M25          ; always does M25 first
M95/1/2/3/5  ; turn off spindle & coolant
M100/75      ; wait for CYCLE START button
```

Default action (tool changer installed):

```
M25          ; always does M25 first
M95/1/2/3/5  ; turn off spindle & coolant
M95/16       ; turn off tool changer strobe
M107         ; send tool number to tool changer
M94/16       ; turn on tool changer strobe
M101/32      ; wait for acknowledge from changer
M95/16       ; turn off tool changer strobe
M100/32      ; wait for acknowledge from changer
```

Manual tool changes are selected by setting Parameter 6 to 0 in the Machine Parameters table. The automatic tool changer is selected by setting Parameter 6 to 1 (see Chapter 15).

The PLC program must be involved in commanding an automatic tool changer and its associated strobe, BCD and Acknowledge lines. See Chapter 5 of the service manual for details of how such a PLC program could be constructed.

13.10 M07 – Mist Coolant On

M7 causes the PLC to start the mist coolant system.

Default action:

M95/3
M94/5

13.11 M08 – Flood Coolant On

M8 causes the PLC to start the flood coolant system.

Default action:

M95/5
M94/3

13.12 M09 – Coolant Off

M9 causes the PLC to stop the coolant system.

Default action:

M95/3/5

13.13 M10 – Clamp On

M10 causes the PLC to activate the clamp–also referred to as a ‘chuck’.

Default action:

M94/4

Note: adding 256 to Parameter 178 can switch M10 and M11.

13.14 M11 – Clamp Off

M11 causes the PLC to release the clamp–also referred to as a ‘chuck’.

Default action:

M95/4

13.15 M17 – Prepare for Tool Change (Macro)

M17 has no default action, therefore a custom M17 macro must be defined for this feature to work. If defined, the M17 macro turns off spindle and coolant and starts the spindle orientation process in preparation for M6 (Tool Change).

The M17 macro is allocated for use with Intercon and the setting in Parameter 162. See Parameter 162 in Chapter 15 for more information.

13.16 M19 – Spindle Orient (Macro)

M19 has no default action, therefore a custom M19 macro must be defined for this feature to work. If defined, the M19 macro sends a request to the PLC to rotate the spindle to its pre-set orient position.

13.17 M25 – Move to Z Home

By default, M25 moves the Z axis to the home position at the Z axis maximum rate. The Z axis home position is the Z axis component of the Return #1 (G28) machine position. (The Return #1 position is first machine coordinate position defined in the Return sub-menu of the Work Coordinate System Configuration.)

The default action of M25 only involves the Z axis. However, if you specify axis arguments (up to 3), then those axes specified will be moved to their axis home positions (Return #1 machine position).

Example:

```
M25         ; move Z to home
M25 /Z      ; same as M25 by itself – move Z to home
M25 /X/Y/Z  ; move X, Y and Z to their Return #1 positions
M25 /X      ; move only X to its Return #1 position
```

13.18 M26 – Set Axis Home

M26 sets the machine home position for the specified axis to the current position (after the line's movement). If no axis is specified, M26 sets the Z-axis home position. The `L` word can be used to set home at the indicated encoder position, provided the axis does not have an absolute encoder.

Example:

```
M91/X        ; home X axis to minus home switch
M26/X        ; set machine home for X-axis there
M92/Z        ; home Z-axis to plus home switch
M26/Z        ; set machine home for Z-axis there
M26/X L4096  ; set machine home for the X-axis at encoder position of 4096
```

13.19 M30 – Custom M Code

An M30 is posted at the end of every G code program. M30 as an end of program code is an industry-standard. By default, M30 performs no operation. If you wish to perform certain operations at the end of every program, this M code can be customized to meet you needs. For more information on customizing M codes, see the beginning of this chapter.

13.20 M39 – Air Drill

M39 is a default air drill activation sequence with a timeout. The sequence of operations is as follows:

```
M94/15        ; activate M function request 15
M103/2        ; start 2 second timer
M100/50015    ; wait for input 15 to open
M95/15        ; deactivate M function request 15
M104          ; cancel timer
```

Note: This program will be canceled by timer expiration if input 15 does not open within 2 seconds after M function request 15 is activated. The PLC program must be involved in taking away the drill output when the CNC program stops:

Example:

```
; PLC program fragment
M39 is SV_M94_M95_15                                   ; M function 39 indicator
drill_out is OUT5                                      ; air drill output relay
if M39 && SV_PROGRAM_RUNNING running then (drill_out)  ; Drill On if M94/15 and the
                                                       ; CNC program is running. Drill
                                                       ; Off if M95/15 or the CNC
                                                       ; program is not running
```

13.21 M41, M42, M43 – Select Spindle Gear Range (Macros)

M41, M42, and M43 have no default actions, and therefore custom macros must be defined for these M codes in order to make this feature work. If defined, these macros notify the PLC of which spindle gear range is selected according to the following table:

Macro M Fuction	Action
M41	Select Low Gear Range
M42	Select Medium-Low Gear Range
M43	Select High Gear Range

Note that selecting a “Medium-High” Gear Range is currently not supported by this schema, although that would not prevent a system integrator from defining another custom macro M function to do that.

13.22 M60 – 5-Axis Digitizing Macro

The M60 is used only when digitizing with the 5-Axis Tilt Table system.

The M60 uses user variables to assign a “Start Position” and a “Finish Position” for 5-Axis Digitizing. When the M60 is executed, the probe will be moved from the start position to the end position. If during the motion the probe detects a surface, the surface position is recorded, and the probe returns to the start position. If no surface is found the probe returns to the start position.

The following variables are to be assigned before the M60 is called, the M60 will then use the positions as assigned by the user variables:

#29100 = X-Axis Probing vector start point (Initial Position)
#29101 = Y-Axis Probing vector start point (Initial Position)
#29102 = Z-Axis Probing vector start point (Initial Position)
#29103 = A-Axis Probing vector start point (Initial Position)
#29104 = B-Axis Probing vector start point (Initial Position)
#29110 = X-Axis Probing vector end point
#29111 = Y-Axis Probing vector end point
#29112 = Z-Axis Probing vector end point

Upon a successful probe the M60 will use an M122 to save the probed position to a text file that should be opened using an M120 or M121 before calling the M60.

Example:

```
M121 “m60test.dig5”   ; Open text file to record data too
#29100 = -8.7999      ; X-Axis Start Position
#29101 = .3747        ; Y-Axis Start Position
#29102 = -1.1832      ; Z-Axis Start Position
#29103 = 85.957       ; A-Axis Start Position
#29104 = 21.36        ; B-Axis Start Position
#29110 = -8.7138      ; X-Axis End Position
#29111 = -.0159       ; Y-Axis End Position
#29112 = -1.183       ; Z-Axis End Position
M60                   ; Execute M60
```


13.23 M91 – Move to Minus Home

M91 moves to the minus home switch of the axis specified at the slow jog rate for that axis. After the minus home switch is tripped, or if it is already tripped at start, the axis reverses until the home switch clears. For a linear axis, the clear state must be reached within 0.5 inches or one motor revolution, whichever is greater, or an error occurs. For a rotary axis, the clear state must be reached within 45 degrees. The axis then reverses an additional 0.005 inches to prevent false limit switch trips. Finally, if the L1 word is not specified in the command, the axis continues reversing until the index pulse is detected, otherwise it stops. When reversing to find the index pulse, the index pulse must be detected within the number of encoder counts specified in the Encoder counts/rev field of the motor parameters menu plus another 100 counts of tolerance, otherwise an error occurs.

If the control configuration ‘‘`Machine home at pwrup`’’ field is set to ‘‘`Ref Mark-HS`’’ and the ‘‘`Home -`’’ field in the Motor Parameter menu is set to 0 (zero) to specify reference mark homing, the behavior of this command is to start moving and then stop when the index pulse is detected.

Example:

```
M91/X      ; moves the X-axis to the minus home switch.
M26/X      ; set X axis home

M91/X L1   ; start homing process on the X axis and stop before beginning search for index pulse
           ; not a good idea to set home here, but useful for troubleshooting
```

13.24 M92 – Move to Plus Home

M92 moves to the plus home switch of the axis specified at the slow jog rate for that axis. After the plus home switch is tripped or if it is already tripped at start, the axis reverses until the home switch clears. For a linear axis, the clear state must be reached within 0.5 inches or one motor revolution, whichever is greater, or an error occurs. For a rotary axis, the clear state must be reached within 45 degrees. The axis then reverses an additional 0.005 inches to prevent false limit switch trips. Finally, if the L1 word is not specified in the command, the axis continues reversing until the index pulse is detected. Otherwise, the axis stops. When reversing to find the index pulse, the index pulse must be detected within the number of encoder counts specified in the Encoder counts/rev field of the motor parameters menu plus another 100 counts of tolerance, otherwise an error occurs.

If the control configuration ‘‘`Machine home at pwrup`’’ field is set to ‘‘`Ref Mark-HS`’’ and the ‘‘`Home +`’’ field in the Motor Parameter menu is set to 0 (zero) to specify reference mark homing, the behavior of this command is to start moving and then stop when the index pulse is detected.

Example:

```
M92/X      ; moves the X-axis to the plus home switch.
M26/X      ; set X axis home

M92/Z L1   ; start homing process on the Z axis and stop before beginning search for index pulse
           ; not a good idea to set home here, but useful for troubleshooting
```


13.25 M93 – Release/Restore Motor Power

M93 releases or restores motor power for the axis specified. If no axis is specified, then all axes are released.

Example:

To release motor power:

```
M93/X   ; releases the X axis.
M93     ; releases the motors on all axes.
```

Example:

To restore motor power:

```
M93/X P1   ; restore power to the X axis motor.
M93 P1     ; restore power to the motors on all axes.
```

Note: Any axis freed within a CNC program should not be used in that program afterwards. Incorrect positioning and/or restoration of power may result. Also, there are times when power is applied to an axis automatically, like when starting a job or when entering an MDI prompt.

13.26 M94/M95 – Output On/Off

There are 128 user definable system variable bits that can be used to communicate with the PLC. M94 and M95 are used to request those system variable bits to turn on or off respectively. Requests 1–128 are mapped to the PLC as system variables SV_M94_M95_1 through SV_M94_M95_128 as shown in the following table:

On	Off	PLC bit
M94/1	M95/1	SV_M94_M95_1
M94/2	M95/2	SV_M94_M95_2
M94/3	M95/3	SV_M94_M95_3
M94/4	M95/4	SV_M94_M95_4
.	.	.
.	.	.
.	.	.
M94/128	M95/128	SV_M94_M95_128

To use M94 and M95 to control a function external to the servo control, such as an indexer, the input request must be mapped to one of the PLC outputs in the PLC program. See M94/M95 function usage in the PLC section of the service manual.

Example:

```
M94/5/6    ; turns on SV_M94_M95_5 and SV_M94_M95_6.
```

Note: M94 and M95 will cause prior motion to decelerate to a stop before the requested bits are turned on or off.

Note: Requests 1–5, 15, and 16 are controlled by the default actions of M3, M4, M5, M6, M7, M8, M9, M10, M11, and M39. To override or disable a bit used in one of these M codes, define a custom M-function.

13.27 M98 – Call Subprogram

M98 calls a user-specified subprogram. A subprogram is a separate program that can be used to perform a certain operation (e.g. a drilling pattern, contour, etc.) many times throughout a main program.

Calling methods:

M98 P*xxxx* L*rrrr*

OR

M98 “program.cnc” L*rrrr*

where *xxxx* is the subprogram number (referring to file Oxxxx.cnc, 9100–9999 allowed, leading 0’s required in filename, capital O, lowercase .cnc), *rrrr* is the repeat value, and “program.cnc” is the name of the subprogram file.

Subprograms are written just like normal programs, with one exception: an M99 should be at the end of the subprogram. M99 transfers control back to the calling program.

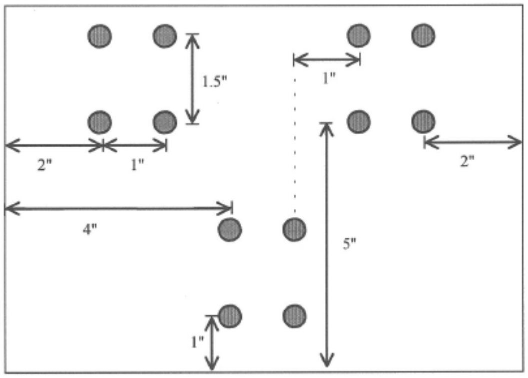
Subprograms can call other subprograms (up to 20 nested levels of calling may be used), Macro M-functions, and Macros. Macro M-functions and Macros can similarly call subprograms.

Subprograms 9100–9999 can also be embedded into a main program, using O9xxx to designate the beginning of the subprogram and M99 to end it. The CNC software will read the subprogram and generate a file O9xxx.cnc. The CNC will not execute the subprogram until it encounters M98 P9xxx.

Note: An embedded subprogram definition must be placed before any calls to the subprogram.

Example:

Suppose that a drilling pattern of 4 holes is needed in 3 different locations:



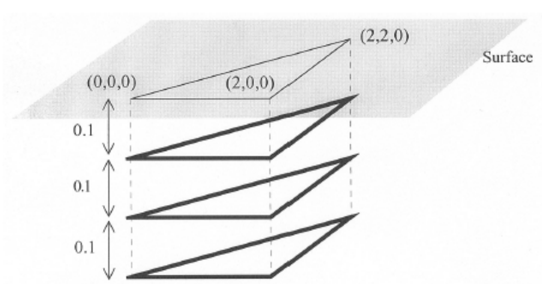

```
:Main program
G90 G0 X2 Y5 Z0.5    ;Move to first hole pattern
M98 P9101 L1         ;Call subprogram O9101.cnc
G90 G0 X4Y1 Z0.5     ;Move to second hole pattern
M98 P9101 L1         ;Call subprogram
G90 G0 X6 Y5 Z0.5    ;Move to third hole pattern
M98 P9101 L1         ;Call subprogram
:End program
```

The main program would call this subprogram three times:

```
O9101                   ;Program O9101.cnc
G91 F10                 ;Incremental positioning
G81 X0 Y0 R -.4 Z-.6    ;Drill lower left hole
Y1.5 R -.4 Z-.6         ;Drill upper left hole
X1 R -.4 Z-.6           ;Drill upper right hole
Y-1.5 R-.4 Z-.6         ;Drill lower right hole
G80                     ;Cancel canned cycles
M99                     ;End of subprogram
```


Example:

Another example is “looping” or consecutively repeating a section of code. Here, the subprogram will be part of the main program.




```
:Main program
G90 G0 X0 Y0 Z0.1
G1 Z0 F30
O9100              ;Beginning of subprogram
G91 G1 Z-0.1 F5
G90 X2 F30
Y2
X0 Y0
M99                ;End of subprogram 9100
M98 P9100 L3       ;Repeat O9100 3 times
M25 G49            ;End main program
```

13.28 M99 – Return from Macro or Subprogram

M99 designates the end of a subprogram or macro and transfers control back to the calling program when executed. M99 may be specified on a line with other G codes. M99 will be the last action executed on a line. If M99 is not specified in a subprogram file, M99 is assumed at the end of the file:

Example:

```
G1 X3 M99    ;Move to X3 then return to calling program.
```

If M99 is encountered in the main job file, it will be interpreted as the end of the job. If M99 is encountered in an M function macro file, it will be interpreted as the end of any enclosing subprogram or macro, or as the end of the job.

13.29 M100 – Wait for PLC bit (Open, Off, Reset)

13.30 M101 – Wait for PLC bit (Closed, On, Set)

The M100/M101 commands wait for a PLC bit to reach a state as indicated in the table below:

Number	PLC bit	M100	M101
50001 – 51312	INP1 – INP1312	open	closed
60001 – 61312	OUT1 – OUT1312	off	on
70001 – 71024	MEM1 – MEM1024	reset	set
90001 – 90064	T1 – T64 status bits	reset (not expired)	set (expired)
93001 – 93256	STG1 – STG256 status bits	reset (disabled)	set (enabled)
94001 – 94256	FSTG – FTSG256 status bits	reset (disabled)	set (enabled)

The number ranges 1–240 can be used to reference the first eighty INP, OUT, or MEM bits. It is recommended that existing CNC10 programs and macros be converted to the new ranges for use with CNC12.

Number	PLC bit	M100	M101
1–80	INP1–INP80	open	closed
81–160	OUT1–OUT80	off	on
161–240	MEM1–MEM80	reset	set

Example:

```
M101/50001    ; wait for INP1 to close
M100/60002    ; wait for OUT2 to turn off
M101/70123    ; wait for MEM123 to be set (1)
```

Note: The numbers assigned to the PLC bits (except 1–240) are the same as those that can be used when referencing system variables in M- and G-code programs.

13.31 M102 – Restart Program

M102 performs any movement requested, and restarts the program from the first line. The Z axis is **NOT** moved to the home position, and the operator is **NOT** prompted to press the **CYCLE START** button to continue.

13.32 M103 – Programmed Action Timer

M103 is used to set up the time limit for a timed operation. If the timer is canceled (usually by M104) before the specified time limit , the program will be canceled and the message “Programmed action timer expired” will be displayed. If another M103 is issued before the time limit expires, then this time limit is nullified and the new time limit will be set up as specified by the latest occurring M103. Note also that if M0 or M1 causes the program to stop momentarily and the “M0 jogging” feature is enabled, then the the timer will also be canceled without the need to issue M104.

Example:

Activate a device and wait for a response. If there is no response within 4.5 seconds, cancel the program:

```
M94/12      ; turn on input request 12
M103/4.5    ; start 4.5 second timer
M100/4      ; wait for input 4 to open
M104        ; input 4 opened, cancel timer
```

13.33 M104 – Cancel Programmed Action Timer

M104 stops the timer started by the last M103 executed.

13.34 M105 – Move Minus to Switch

M105 moves the requested axis in the minus direction at the current feedrate until the specified switch opens (if the given P parameter is positive), or until the specified switch closes (if P parameter is negative).

Example:

```
M105/X P5 F30    ; move the X axis in minus direction at 30"/min until
                 ; the switch on INP5 opens
G92 X10          ; Sets X position to 10
M105/Z P-6       ; move the Z axis in minus direction until switch on INP6 closes
```

13.35 M106 – Move Plus to Switch

M106 moves the requested axis in the plus direction at the current feedrate until the specified switch opens (if the given P parameter is positive), or until the specified switch closes (if P parameter is negative).

Example:

```
M106/Z P3 F30    ; move the Z axis in the plus direction at 30"/min, until
                 ; the switch on INP3 opens
G92 X10          ; Sets Z position to 10
M106/X P-3       ; move the X axis in the plus direction until the switch on INP3 closes
```

13.36 M107 – Output Tool Number

M107 sends the current tool number to the automatic tool changer, via the PLC. M107 does not set the tool changer strobe or look for an acknowledgement from the changer (see M6).

Example:

M107	; send request for tool to change
M94/16	; turn on tool changer strobe
M101/5	; wait for acknowledge on input 5
M95/16	; turn off strobe
M100/5	; wait for acknowledge to be removed

13.37 M108 – Enable Override Controls

M108 re-enables the feedrate override and/or spindle speed override controls if they were disabled with M109. A parameter of “1” indicates the feedrate override; “2” indicates the spindle speed override.

Example:

M109/1/2	; disables feedrate and spindle speed overrides
M108/1	; re-enables feedrate override
M108/2	; re-enables spindle speed override

13.38 M109 – Disable Override Controls

M109 disables the feedrate override and/or spindle speed override controls. It may be used before tapping with G85 to assure that the machine runs at the programmed feedrate and spindle speed. It is not necessary to specify M109 with G74 or G84; those cycles automatically disable and re-enable the override controls. M109 cannot be used in MDI mode.

Example:

M3 S500	; start spindle in clockwise direction, at 500 rpm
F27.78	; set feedrate for 18 pitch tap
M109/1/2	; disable feedrate and spindle speed overrides
G85 X0 Y0 R.1 Z-.5	; tap a hole
M108/1/2	; re-enable overrides

13.39 M115/M116/M125/M126 – Protected Move Probing Functions

The protected move probing functions provide the capability to program customized probing routines.

The structure for these commands is: **Mnnn** /*Axis pos* P*p* F*f* L*1*

Where:

nnn	is either 115, 116, 125, or 126.
Axis	is a valid axis label, i.e., X, Y, Z, etc.
pos	is an optional position
P	is a PLC bit number, which can be negative.
*F**	is a feedrate (in units per minute.)
*L1**	options for the M115/M116 commands that prevents an error if the probe does not detect a surface
Q1	is an option for M115/M116 that forces the DSP probe to move a "Recovery Distance" on retries. (See Machine Parameter 13 for "Recovery Distance") **Note:** the Q1 option only applies for DSP Probes

For M115 and M116 functions, the indicated axis will move to pos (if specified) until the corresponding PLC bit p state is 1, unless p is negative, in which case movement is until the PLC bit state is 0(closed). A "p value" of 1 to 80 (or -1 to -80) specifies PLC bits INP1-INP80. Warnings are generated in the CNC software message window for "Missing P value" and "Invalid P value." If "pos" is not specified, M115 will move the axis in the negative direction, and M116 will move the axis in the positive direction. Note if "pos" is specified, then if does not matter whether M115 or M116 is used. Regardless of whether or not pos is specified, movement is bound by the settings in the software travel limits as well the maximum probing distance (Machine Parameter 16).

For M125 and M126 protected move functions, the behavior is identical to that of the M115 and M116 commands, except in regards to the PLC bit state. The M115 and M116 commands are to be used when one expects contact to be made and M125 and M126 commands are to be used when one does not expect any contact to be made.

Example:

Finding the center of a vertical slot. In this example, it is assumed that there is a probe connected to INP15 and that the probe tip is positioned somewhere in the slot, such that movement along the X-axis will cause a probe trigger.

```
M115/X P-15 F20      ; Move X minus at 20 ipm until probe trip
M116/X P15 F5        ; Move X plus at 5 ipm until probe clears
#100 = #5041         ; Record the point in user variable #100
M116/X P-15 F20      ; Move X plus at 20 ipm until probe trip
M115/X P15 F5        ; Move X minus at 5 ipm until probe clears
X[[#100+#5041]/2]    ; Move X to center of slot
```

* Usage is slightly different when using a DSP type probe. Please see below for dissimilarities between a standard DP4 probe and the DSP type probe.

13.40 M115/M116/M125/M126 – DSP Probe specific information

Before attempting to use the protected move probing functions with a DSP type probe, please be sure to familiarize yourself with the DSP probe configuration in Chapter 9 of this manual. Using the protected probing moves with a DSP type probe may yield unexpected results if you do not fully understand the concepts and guidelines discussed in the DSP probe configuration section.

If the control is configured to use a DSP type probe, all M115/M116 moves will perform window checking and repeat on a failed window. On a failed window, a repeat attempt is made by returning to the starting point of the move.

Protected move probing functions follow the same command format as that of a standard probe (M*nnn Axis pos* P*p* F*f L1*) with the following exceptions:

f	This will be ignored if “Force DSP Feedrate in M115/M116” has been set to yes.
L1	Still prevents a fault from occurring. Stores last DSP position on failed window.
L2	Like L1, prevents a fault from occurring but instead stores last mechanical pos. on failed window.
Q1	On a failed window, force a pull back distance equal to the Probing Recovery Distance(Parameter 13), instead of moving back to the starting point.

DSP Position vs. Mechanical Position Protected probing moves that are performed using a standard DP4 probe can collect only the point at which motion has stopped after detecting contact. This position is referred to as the “mechanical position”. When using the DSP type probe, it detects and stores the contact position “on the fly”. This position is in machine position (not a local WCS position) and is referred to as the DSP position.

Example: from above – Modified to use a DSP type probe.

Finding the center of a vertical slot:

```
M115/X P-15                  ; Move X minus at DSP rate until probe trip (no feedrate needed)
#100 =[[#24301]-[#2500]]     ; Convert point to current WCS position, Store point in variable #100
M116/X P15                   ; Move X minus at 5 ipm until probe clears
M116/X P-15                  ; Move X plus at DSP rate until probe trip
X[[#100+[[#24301]-[#2500]]]/2]   ; Move X to center of slot
```

Retrieving the DSP position:

The last stored DSP position for axes 1–5 can be retrieved from system variables #24301 #24305 unless the L2 switch was used in which case #24301-#24305 will contain the mechanical position after a failed window.

13.41 M120 – Open data file (overwrite existing file)

This M function will open the requested data file for writing. If no drive or directory is specified with the file name, then the file will be opened in the same directory as the CNC program. If the file cannot be successfully opened, then an error will be returned, ultimately terminating the job. If a data file is already open when M120 is called, that file will first be closed, then the new file opened.

Example:

```
M120 “probetst.dat”    ; Opens probetst.dat file to write data too
```

Note: M120 and M121 also allow use of the string user variables #300–#399 to specify a filename. As an example, given that #300 = “myfile” and #301 = “cnc”

```
M120 “#300.#301”    ; Opens the file “myfile.cnc” for data recording.
```

Keep in mind however that there is a quirk in the way that the M120/M121 operates that requires the ‘.’ to be present so assigning #301 = “.cnc” and executing M120 “#300#301” does not work and generates a “Could not open file” error message.

13.42 M121 – Open data file (append to existing file)

This M function will open the requested file for writing at the end of the file. If no drive or directory is specified with the file name, then the file will be opened in the same directory as the CNC program. If the file does not already exist, it will be created. This is not an error. If the file cannot be successfully opened, then an error will be returned, ultimately terminating the job. If a data file is already open when M121 is called, that file will first be closed, then the new file opened.

Example:

M121 “c:\probetst.dat”   ; Opens probetst.dat file to add data to it

String variables #300-#399 may also be used to specify a file name. Please see M120 above for details.

13.43 M122 – Record local position(s) and optional comment in data file

This M function will write the current expected position value to the data file, in the usual format (i.e. axis label before number, 4 decimal places in inch mode, 3 decimal places in millimeter mode. Any comment that appeared on the line with M122 will be outputted after the position(s). With no axis arguments, M122 will write the positions of all installed axes. With axis arguments, it will write the positions only of the requested axes. Positions will be written in local (not machine) coordinates, in native machine units. If no data file has been opened with M120 or M121 before M122 is called, then M122 will return an error and terminate the job. The parameter L1 may be used to suppress the new line character normally outputted after the last position. Furthermore, the output of axis labels, comma separators, and spaces can be enabled or suppressed via machine Parameter 72 (see Parameter 72 in Chapter 15). If the control has been configured to use a DSP probe type, using parameter Q1 will write the values stored in #24301-#24305 to the file.

Example: M function and sample output:

M122 ;comment	->	X1.2345 Y-3.2109 Z-0.5678 ;comment
M122 /X L1	->	X-1.5000
M122 /X	->	X-1.5000 X-2.0000

13.44 M123 – Record value and/or comment in data file

This M function will write the specified parameter value (if any) to the data file, followed by any comment that appeared on the line with M123. If a P value is specified, M123 will record the numeric value (4 decimal places in inches, 3 in millimeters). If neither a P value nor a comment was specified, M123 does nothing. This is not an error. If no data file has been opened with M120 or M121 before M123 is called, then M123 will return an error and terminate the job. The parameter L1 may be used to suppress the new line character normally outputted after the last value. The R and Q parameters can be used to specify the field width and precision, respectively. Furthermore, the output of axis labels, comma separators, and spaces can be enabled or suppressed via machine Parameter 72 (see Parameter 72 in Chapter 15).

Example: M function and sample output:

M123 ;1.2345	->	1.2345
M123 P#A ; first macro argument	->	1.2345 first macro argument
M123 Q0 P1.23	->	1

13.45 M124 – Record machine position(s) and optional comment in data file

Identical to M122 above except that the m124 reports machine position instead of a local WCS position.

13.46 M127 – Record Date and Time in a data file

This M function is used to write the date, time, and year to the specified data file called out by the M120 or M121. Examples (M function and sample output): **Note: The M127 does not insert a semi-colon in front of the date.** If desired, use the M123 as shown below.

```
M121 “testdata.dat”
M123 ;;
M127
```

If you opened testdata.dat you would see: Day of week, Month, day, time, and year. (i.e. ;Wed Aug 29 11:56:57 2007)

13.47 M128 – Move Axis by Encoder Counts

M128 moves the requested axis by L which specifies an encoder count position or quantity. The L parameter is subject to the current G90/G91 mode (absolute/incremental).

Example:

```
G91 M128/X L-5000    ; move the X axis incrementally by -5000 counts
```

13.48 M129 – Record Current Job file path to data file

This M function is used to write the current job’s file path to the specified data file called out by the M120 or M121.

Example: Run a job named job.cnc which contains the following 2 lines:

```
M121 “output.txt”
M129
```

If you opened the output.txt file you would see: c:\cncm\ncfiles\job.cnc

13.49 M130 – Run system command

This allows shell commands to be called from a CNC program or MDI. M130 takes one string argument which contains the system command to execute.

For example:

```
M130 “mycommand.bat”
```

will run the batch file *mycommand.bat*.

Normally, the command will run asynchronously, meaning that the G-code program will not wait for the command to finish before continuing. However, if an L1 parameter is given, the M130 command will wait until the “mycommand.bat” finishes execution or an operator abort or fault occurs (such as E-Stop).

13.50 M150 – Set Spindle Encoder to zero at next index pulse

M150 will cause the spindle encoder position to be reset to 0 upon the next encounter of the spindle encoder's index pulse. M150 will not generate spindle movement. As a matter of fact, the spindle needs be be commanded to move in order for M150 to work.

13.51 M200, M223, M224, M225 & M290 – Formatted String Commands

The formatted string commands are provided to assist in custom screen and file I/O. A"*formatted-string*" is similar to the C programming language "*printf*" command, with various restrictions. The basic form of a *formatted-string* is a quoted string (comprised of a **single** line of up to 1024 characters) followed by a (possibly empty) list of user and/or system variable expressions. The variable expression is a '#' character followed by a number or bracketed expression.

For example, given #100 = 88 (ASCII 'X'), #300 = "absolute", and #101 = 1.2345, this string:

```
"The %c axis %s position is %f" #100 #300 #101
```

evaluates to

```
"The X axis absolute position is 1.23450"
```

The "%c" is replaced by the ASCII character value of user variable #100, the "%s" is replaced by the string user variable #300, and the "%f" is replaced by the value of user variable #101.

13.51.1 Type Specifiers

The 's', 'c', and 'f' are type specifiers, with 's' specifying a string user variable, 'f' specifying a floating point user variable, and 'c' specifying a single character substitution using the integer part of a floating point user variable. There should be one user variable expression for every '%' character in the quoted string. It is also possible to specify a field width by inserting a number between the '%' and the type specifier.

Example:

%20s – specifies that the substituted string is displayed in a field 20 characters long, right justified and padded with spaces on the left. Use "%-20s" for left justification.

The 'f' type can specify a precision such as:

- "%.4f" – display number rounded at the fourth decimal place.
- "%9.4f" – as above but in a field width nine characters wide.
- "%+9.4f" – as above with an '+' output if variable is positive.
- "%.0f" – display number rounded to integer

If no precision is specified, "%f" will use a default precision of the current DRO display precision.

13.51.2 Special characters

The quoted string may contain one or more '\n', each of which will be converted to a single newline character. Up to seven newlines can be specified in a single formatted string. However, a formatted string may not contain an embedded quote character '"' or other printf-style escape sequences such as '\t', '\\', or '\"'. If a quote character is desired, use a %c type specifier with a variable expression equal to 34.

User string variables #300-#399: These variables can be assigned a quoted string up to 80 characters in length and are retained until the CNC software is exited. For example,

#300 = "This is a text string of characters"

* The above method of representing an axis label should be used only when writing to an external file or for display in a message box. It is not valid if you are attempting to "build" a motion command in real-time from within the currently running G-code program. If your intent is to use a variable to represent an axis label for a real-time command, you should instead use $ as the placeholder. The parser will replace a '$' character and the numerical expression following it with the ASCII character equivalent to the numerical expression, provided that it evaluates to the characters 'A' (65) through 'Z' (90). If the numerical expression is out-of-bounds, an "Invalid character" error occurs.

Example: Given #100 = 88, #101 = 1, #102 = 89, #103 = 2, and #104 = 10,

G1 $[#100][#101] $[#102][#103] F[#104]   evaluates to   G1 X1 Y2 F10

13.51.3 Text Justification

By default, the text of the formatted string commands is center justified. However, adding the characters `#)` (a pound symbol followed by a right parenthesis) as the first two characters of the format string indicates left justification of the text. For all but `M224` the justification applies to the entire message; for `M224`, the justification applies to each line individually.

Example:

```
M200 "#)1.  Jog the X axis to the desired X0 position\n2.  Jog the Z axis to the
desired Z0 position.\n3.  Press Cycle Start to continue."
```

Example:

```
M224 #300 "#)1.  Jog the X axis to the desired X0 position\n#)2.  Jog the Z axis to
the desired Z0 position.\n#)3.  Continue?."
```

13.52 M200/M201 – Stop for Operator, Prompt for Action

M200 is used to pause the currently running job and prompt the operator for action. If M0_jogging is unlocked, or the control is in DEMO mode, jogging is enabled while waiting for the operator to respond. If this option has not been enabled, the behavior will default to that of a standard M0. (jogging disabled)

The syntax is:

M200 *formatted-string [[user_var_expr] . . .]*

Example:

M200 “Please jog the %c and %c axes to the desired X0, Y0 position\nPress Cycle Start to continue” #100 #101

M201 behaves exactly like M200 except that PLC bits SV_PROGRAM_RUNNING, SV_MDI_MODE, and SV_JOB_IN_PROGRESS are turned off while the prompt is displayed.

13.53 M223 – Write Formatted String to File

The M223 command writes a *formatted-string* to a file that was opened using the M120 or M121 commands. The syntax is:

M223 *formatted-string [[user_var_expr] . . .]*

Example:

M223 “; The measured diameter of the pocket = %.4f\n” #100

13.54 M224 – Prompt for Operator Input Using Formatted String

The M224 command displays a *formatted-string* and then accepts user input. The syntax is:

M224 /num l*value_expr formatted-string [[user_var_expr] . . .]*

Where /num is an optional parameter specifying the font family. Possibilities are:

/0 – Default	/4 – Swiss
/1 – Decorative	/5 – Modern
/2 – Roman	/6 – Teletype
/3 – Script	

Where lvalue_expr is a user_var_expr that evaluates to a user variable that can be written. If lvalue_expr is a string type (#300-#399) then the user input is assigned verbatim to the string. Otherwise, the user input is evaluated as any other “bracketed” numerical expression.

Example:

M224 /0 #300 “Please enter the direction that you wish to probe in the %c axis: (+ or -)” #100

13.55 M225 – Display Formatted String for A Period of Time

The M225 command displays a *formatted-string* for a specified period of time. The syntax is:

M225 /num *time_expr formatted-string [user_var] . . .*

Where /num is an optional parameter specifying the font family. Possibilities are:

/0 – Default	/4 – Swiss
/1 – Decorative	/5 – Modern
/2 – Roman	/6 – Teletype
/3 – Script	

The argument *time_expr* is a *user_var_expr* that evaluates to a floating point variable specifying the number of seconds to display the output, with a value of zero interpreted as indefinitely. The **CYCLE START** key can be used to immediately continue running without waiting for the time to expire.

Example:

M225 /6 #100 "Warning, %s is not selected\n Please select %s and press Cycle Start to continue." #300 #300

13.56 M290 – Digitize Profile (Optional)

This performs a 2 axis digitize, probing along an axis while stepping over using a perpendicular travel axis. This M-code is similar to performing a single slice of Grid Digitizing with the Surface Following type selected (See Chapter 8). M290 expects that a file is already open with M120/121 (however, if not open, there will be no output).

The syntax is:

M290 /a__ *#vvv* /b__ *#vvvv* *"formatted-string"* Q__ R__ P__ L__

The first axis mentioned will be treated as the probing axis and the second will be treated as the travel axis.

Explanation of M290 Arguments:

/*a* +-nnn.nnn is the probing direction and max distance on axis "*a*".

/*b* +-nnn.nnn is the travel direction and max distance on axis "*b*" (perpendicular to probing axis).

#vvv and #vvvv (optional) are G-code variables that will be a receptacle of the **very last** probed position of the cycle.

"formatted-string" (optional) is the format of the output (if not mentioned, then there will be no output to file).

Q__ is the stepover along the travel direction (this is a positive quantity).

R__ (optional) is the retract/pullback amount upon interruption or completion.

P__ (optional) is the interruption PLC bit state which causes a graceful end to the cycle. (If not mentioned, then no PLC bit will checked for graceful interruption.).

L__ (optional) is the output variable to which to store the interrupt status (0=no interruption, 1=interrupted by PLC bit P__, 2=surface not found error).

13.57 M300 – Fast Synchronous I/O update

There are 32 user definable fast system integer variables that can be used to communicate with the PLC (similar to M94 and M95), but without causing motion to decelerate to a stop* (unlike M94 and M95). The syntax is:

M300 /*nn* /*vvv*

where *nn* is 1–32 and *vvv* is a 32-bit signed integer value. The parameter *nn* (1–32) maps to system variables SV_FSIO1–SV_FSIO32. These commands work in conjunction with a PLC program that can read the SV_FSIOx and act upon them.

Example:

```
M300 /21 /-1234 ; set SV_FSIO21 to integer value -1234
```

Note: Motion will be decelerated to a stop if Smoothing is turned on (P220 = 1).

13.58 M333 – Axis Role Re-assignment

This is an experimental M-function that re-assigns X,Y,Z axis behaviors to other axes. This command is not recommended for normal use.

13.59 M1000-M1015 – Graphing Color for Feedrate movement

When a CNC program is graphed (F8 from the Main Screen), feedrate movements are normally plotted using the color yellow. This color setting can be changed to another color as stated in the chart below.

M Code	Feedrate Graphing Color
M1000	Black
M1001	Navy Blue
M1002	Green
M1003	Teal
M1004	Orange
M1005	Blue
M1006	Lime
M1007	Aqua
M1008	Maroon
M1009	Purple
M1010	Olive
M1011	Gray
M1012	Red
M1013	Fuchsia
M1014	Yellow
M1015	White

Changing this feedrate graphing color can be used as a method highlighting or hiding parts of a graphed CNC program, but will not affect the normal run of the program (when the **CYCLE START** button is pressed on the Main Screen). The limitations to using these M codes are as follows: These M codes cannot be placed on the same line as another M code, and also the rapid (G0) movement color cannot be changed.

14 ATC Operation

14.1 Custom M codes for CNC ATC Systems

Below is a list of custom M codes that can be found on swingarm and/or umbrella type tool changers for controls running CNC software.

14.1.1 Custom M Codes Used on All ATC PLC Programs

- M3 – Turns the spindle on in the clockwise direction.
- M4 – Turns the spindle on in the counterclockwise direction.
- M5 – Turns the spindle off.
- M6 – Changes the tool that is in the spindle by using other custom M codes.
- M7 – Turns the mister pump on.
- M8 – Turns the flood pump on.
- M9 – Turns both coolant pumps off.
- M10 – Turns on the clamp on the rotary table.
- M11 – Turns off the clamp on the rotary table.
- M18 – Resets the tool counter to tool #1. **Ensure that the tool changer is at bin location #1 and that tool #1 is in the spindle.**
- M19 – Orients the spindle to the desired location to perform a tool change. **Cannot be done in M15 is already active. If the spindle motor, spindle encoder, spindle inverter, or inverter encoder parameters are changed, the inverter parameters must be checked to ensure proper spindle orientation to prevent a crash. Please follow the tech bulletin for the appropriate inverter on setting the inverter parameters.**
- M31 – Turns the optional chip auger forward.
- M32 – Reverses the optional chip auger.
- M33 – Turns the optional chip auger off.

14.1.2 AUX12 Key

The **AUX12** key is the second unlabeled blue key located in the rightmost column of the coolant row on the jog panel. See the key marked with an asterisk (*) in the picture below. In the following four subsections, commands with an asterisk (*) next to them require the AUX12 key to be pressed on the jog panel. The AUX12 key must be held down until the carousel and clamp switches are in the default state or a stop condition will result.

14.1.3 Additional M Codes Used on Umbrella Tool Changer PLC Programs in CNC11/CNC12

- M15* – Unclamps the tool and turns the air blow on. **Be prepared to catch the tool from the spindle. Will not operate if the spindle is turning.**
- M16* – Clamps the tool and turns the air blow off.

M15 and M16 are not to be used to manually change tools. Use the Clamp/Unclamp button located on the spindle head.

- M20 – Turns the spindle off.
- M21 – Turns the optional chip washer pump on.
- M22 – Turns the optional chip washer pump off.
- M80* – Moves the carousel in regardless of the Z-axis position. **Will not operate if the spindle is turning. THERE IS NOTHING TO PREVENT THE CAROUSEL FROM SLAMMING INTO THE SPINDLE HEAD OR TOOL. ENSURE SUFFICIENT CLEARANCE BEFORE ISSUING THIS COMMAND!**
- M81* – Moves the carousel out. **ENSURE THE TOOL IS NOT PARTIALLY COVERED BY SPINDLE OR THE TOOL WILL BE KNOCKED OUT AND THE CAROUSEL FINGERS MAY BE DAMAGED!**

14.1.4 Additional M Codes Used on Umbrella Tool Changer PLC Programs in CNC7/CNC10

- M17 – Turns the cooland and spindle off and turns spindle orientation on.
- M21 – Moves the spindle head up to the Z home position.
- M22 – Moves the spindle head up to the tool change position, G30. **Do not change the G30 parameters and do not use any codes in your program that will overwrite these parameters.**
- M50 – Simulates a positive tool index while the program is running.
- M51 – Simulates a negative tool index while the program is running.

M50 and M51 are typically used when there are large tools that would interfere with the part or fixture.

14.1.5 Additional M Codes Used On Swingarm Tool Changer PLC Programs in CNC11/CNC12

- M13* – Cycles the tool changer swingarm. The M13 cycle does the following:
  - Turns the swingarm motor on and waits for the ARM_STOP signal to go off.
  - Waits for the ARM_STOP signal to come on and then shuts off the swingarm motor.
- M14* – Brings the tool pot down.
- M15* – Brings the tool pot up.
- M60 – Turns the optional chip washer pump on for 0.1 seconds and then shuts it off.
- M61 – Records the bin location of tool #1.
- M88 – Turns on the optional thru-tool coolant.

14.1.6 Additional M Codes Used on Swingarm Tool Changer PLC Program in CNC7/CNC10

- M0 – Turns on the optional amber light in addition to issuing an M0.
- M1 – Turns on the optional amber light in addition to issuing an M1.
- M14* – Brings the tool pot up.
- M88 – Turns on the optional thru-tool coolant.
- M89 – Turns off the optional thru-tool coolant.

15 Configuration

15.0.1 General

The first four options, **F1** through **F4**, will display a set of parameters. Each option is explained in detail below. The **ESC** key will return you to the previous screen (Setup).

The configuration option provides you with a means for modifying the machine and controller configuration. The majority of information in this section should not be changed without contacting your dealer. The **F5 – Test** key should only be used by qualified factory technicians to perform automated system tests.

WARNING Some of the data, if corrupt or incorrect, could cause personal injury or machine damage.

15.0.2 Password

When you press **F3 – Config** from the Setup Screen, you may be prompted to enter a password. This level of security is necessary so that users do not accidentally change vital parameters. The original default password is distributed in the documentation provided to the owner of the machine when the control is installed. This password is changeable via Parameter 42.

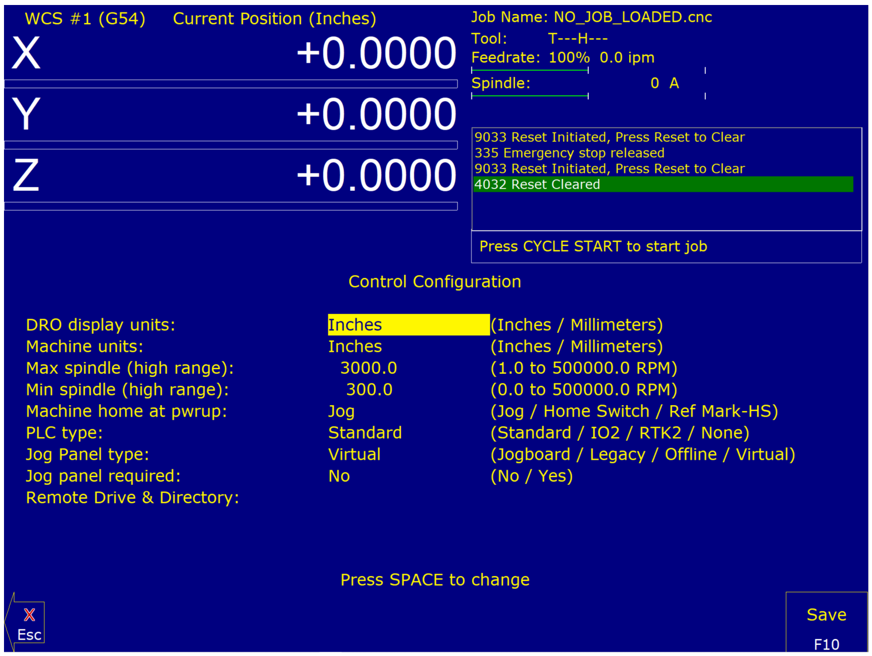
If you know the password, type it and press **ENTER**. If the password you enter is incorrect, a message will appear telling you the password was incorrect and the password prompt will reappear. Pressing **ESC** will remove the prompt.

If you don't know the password, simply press **ENTER**. You will be given access to the configuration options so that you can view the information. However, you will not be able to change any of the data.

15.0.3 Control Configuration

Pressing **F1 – Contrl** from the configuration screen will display the Control Configuration screen. The Control Configuration screen provides you with a method of changing controller dependent data. Each of the fields is discussed in detail below.

If you wish to change a field, use the up and down arrow keys to move the cursor to the desired field. Type the new value and press **ENTER**, or press the **SPACE** bar to toggle. When you are done editing, press **F10 – Save** to save any changes you have made. If you wish to discard your changes and restore the previous values, press **ESC**.



DRO Display Units

This field controls the units of measure that the DRO displays. The two options are 'Millimeters' and 'Inches.' When this field is highlighted by the cursor, "Press SPACE to change" appears at the bottom of the screen. This message is explaining that pressing the **SPACE** bar will toggle the value of this field between the two options.

The DRO display units do not have to be the same as the machine units of measure (explained below). This field is provided for users of the G20 & G21 codes so that they may view the tool position in terms of job units.

Machine Units of Measure

This field controls which units of measure the machine uses for each job. The two options are 'Millimeters' and 'Inches'. Press the **SPACE** bar to toggle the field between the two options.

This field determines the default interpretation of job dimensions and feedrates. If 'Inches' is selected, all feedrates and dimensions will be interpreted as inches as well as any unit dependent parameters.

Note: This field should rarely, if ever, be changed. If you wish to run a job in units other than the default machine units, use the G20 & G21 codes.

Maximum Spindle Speed (High Range)

This field sets the high range maximum spindle speed for those machines that have a variable frequency spindle drive controller (VFD). All spindle speeds entered in a CNC program are sent to the PLC as percentages of this maximum value.

If your machine is equipped with a dual range drive and VFD, the controller will not exceed the spindle speed set by this field while in high gear. See Machine Parameters for information on setting the gear ratios for medium and low gear ranges. If your machine has a VFD but is not equipped with a dual range drive, this field determines the maximum spindle speed.

Minimum Spindle Speed (High Range)

This parameter is used to adjust the minimum spindle speed for the high range. This parameter allows the operator to set the minimum value for spindle speed to a value other than 0. All changes in spindle speed are made in relationship to this value, with this parameter as the minimum value. The values stored can range from 0 to 500000.0 RPM.

Machine Home at Powerup

This field controls how the machine will home at powerup. Set Machine Home at Powerup to Home Switch if you have limit/home switches or safe hard stops for all axes, and wish to use the switches or stops for homing. Set Machine Home at Powerup to Ref Mark – HS if you have fixed reference marks for any axis. In Ref Mark homing, axes that contain a zero (0) for the plus or minus home switch in the Machine Configuration designate that axis to have a Ref Mark home, while non-zero values specify Limit Switch homing. Set Machine Home at Powerup to Jog if you need to manually move or jog the machine to its home position. See Machine Home for more information.

PLC Type

This field tells the controller which PLC type is installed. The available choices are: Standard, Legacy IO2, Legacy RTK2, and None.

Jog Panel Type

This field tells the controller which type of Jog Panel is installed. The available choices are: Jogboard, Legacy, Offline, and Virtual.

Remote Drive & Directory

This field sets up the remapped default drive and directory for the **F3 – Remote** key in the Load Job screen. This allows you to conveniently load files from an attached computer via LAN network (via RJ-45 Ethernet connection). The Control will usually remap the attached computer's C: hard drive as drive E:, depending on the way it was set up.

15.1 User Specified Paths

Operators can now specify paths for INTERCON files, posted INTERCON files, Digitize files and CAD files. These paths are specified in pathm.ini. This file is automatically generated by the CNC software if it does not exist. The default pathm.ini file is:

```
intercon_path=C:\intercon\

icn_post_path=C:\cncm\ncfiles\

digitize_path=C:\cncm\digitized_data\

cad_path=C:\cncm\ncfiles\

chamb_dig_path=C:\cncm\digitized_data\

chamb_dig_setup_path=C:\cncm\digitized_data\

auto_dig_setup_path=C:\cncm\digitized_data\
```

Path tag	Purpose of path
INTERCON_PATH	Main directory containing *.icn files
ICN_POST_PATH	Directory INTERCON places *.cnc files created when posting *.icn files.
DIGITIZE_PATH	Directory digitize files are saved to. Directory used by F4 key in Load Job menu when Parameter 4 is set to 2.
CAD_PATH	Default directory used by the Import DXF file menu in Intercon. This directory is also used to store CAD files generated with the DIG→CAD option in the Utility menu.
CHAMB_DIG_PATH	For performance racing applications, the directory where the chamber digitizing data is placed.
CHAMB_DIG_SETUP_PATH	For performance racing applications, the directory where Chamber Digitizing Menu data is stored.
AUTO_DIG_SETUP_PATH	For performance racing applications, the directory where Autonomous Digitizing Menu data is stored.

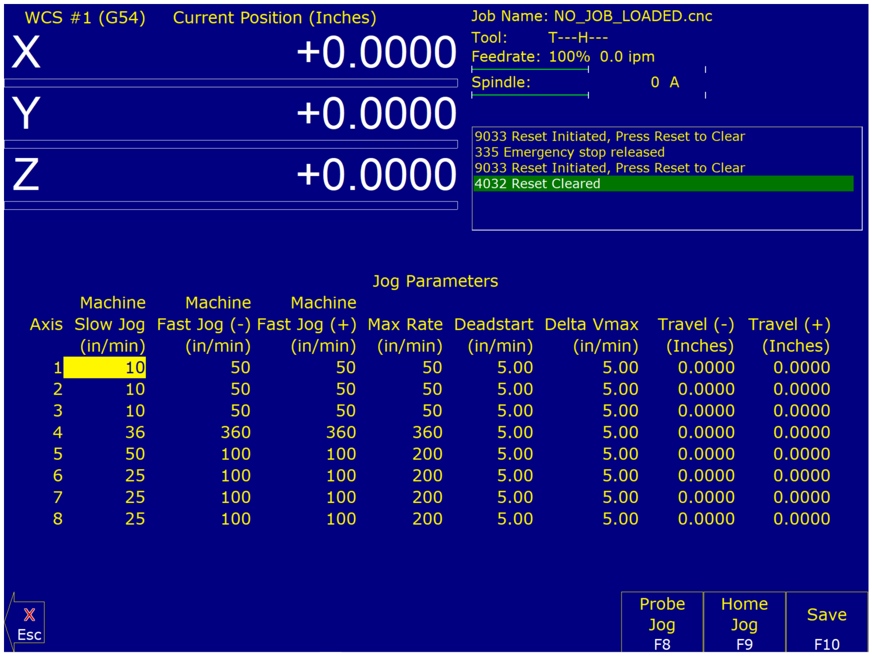
15.2 Machine Configuration

Pressing **F2 – Machine** from the configuration screen will display the machine configuration screen. The machine configuration screen provides you with a method of changing machine dependent data.

If you wish to change the Jog or Motor parameters, press **F1 – Jog** or **F2 – Motor** to select the Jog or Motor screens. Use the arrow keys to move the cursor and select the desired field. Type the new value and press **ENTER** or press the **SPACE** bar to toggle. When you are done editing, press **F10 – Save** to save any changes you have made. If you wish to discard your changes and restore the previous values, press **ESC**. Pressing **ESC** again will return you to the previous screen (Configuration).

15.2.1 F1 – Jog

This screen contains jog and feedrate information. See the figure below.



Axis	Machine Slow Jog (in/min)	Machine Fast Jog (-) (in/min)	Machine Fast Jog (+) (in/min)	Max Rate (in/min)	Deadstart (in/min)	Delta Vmax (in/min)	Travel (-) (Inches)	Travel (+) (Inches)
1	10	50	50	50	5.00	5.00	0.0000	0.0000
2	10	50	50	50	5.00	5.00	0.0000	0.0000
3	10	50	50	50	5.00	5.00	0.0000	0.0000
4	36	360	360	360	5.00	5.00	0.0000	0.0000
5	50	100	100	200	5.00	5.00	0.0000	0.0000
6	25	100	100	200	5.00	5.00	0.0000	0.0000
7	25	100	100	200	5.00	5.00	0.0000	0.0000
8	25	100	100	200	5.00	5.00	0.0000	0.0000

A description of each of these parameters is listed below.

Note: Some of these values are set automatically by the Autotune option (See PID Menu).

Slow Jog: Determines the speed of motion on an axis when slow jog is selected and a jog button is pressed. The slow jog rate cannot be set to a value greater than the maximum rate.

Fast Jog (-): Determines the speed of motion on an axis when fast jog is selected and a negative direction jog button is pressed. The fast jog (-) rate cannot be set to a value greater than the maximum rate.

Fast Jog (+): Determines the speed of motion on an axis when fast jog is selected and a positive direction jog button is pressed. The fast jog (+) rate cannot be set to a value greater than the maximum rate.

Max Rate: Determines the maximum feedrate of each individual axis. The feedrate on each axis can never exceed Max Rate, even if the feedrate override knob on the front panel is turned up above 100%. (Also see Parameter 38 – Multiaxis Max Feedrate, which limits the feedrate along move vectors, not just each individual axis.)

Note: The maximum rate may be set to a smaller value if you wish to run your machine at a slower rate.

Deadstart: Determines the speed an axis will decelerate to before stopping or reversing direction. A low setting will cause a large slowdown before a reverse in direction, causing your machine to be more accurate. A high setting will cause less slowdown before reversals, but this may cause your machine to "bang" which may cause a decrease in accuracy. This parameter should not be changed.

Delta Vmax: The maximum instantaneous velocity change that will be commanded on a vector transition. This parameter should not be changed.

Travel (-): The maximum distance the axis can travel in the minus direction from the home position. Set this parameter to create a software limit that stops the axis before the fixture or tool collides with the limit switches or hard stops.

Travel (+): The maximum distance the axis can travel in the plus direction from the home position. This parameter is especially useful when using a part or fixture larger than the table. Set this parameter to create a software limit that stops the axis before the fixture or part collides with the machine or the limit switch/hard stop.

F1–F8 – Probe Jog Parameters

Within the Jog Parameters is a submenu, accessed by pressing **F8 – Probe Jog**, that sets separate slow and fast jogs when a probe is plugged in. The following parameters are available:

Probe Slow Jog: Determines the speed of motion on an axis when slow jog is selected and a jog button is pressed while the probe is plugged in. The slow jog rate cannot be set to a value greater than the maximum rate.

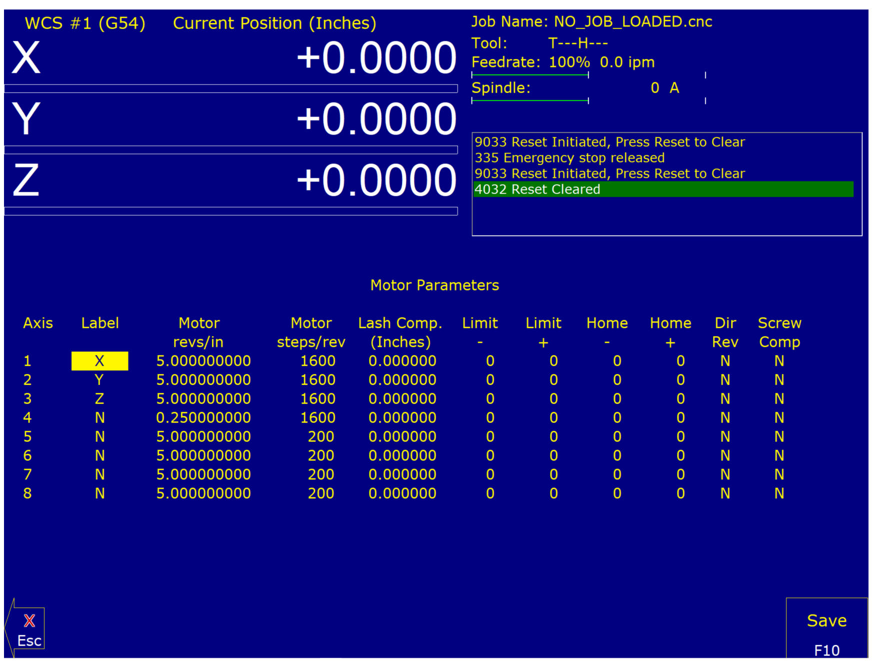
Probe Fast Jog (-): Determines the speed of motion on an axis when fast jog is selected and a negative direction jog button is pressed while the probe is plugged in. The fast jog (-) rate cannot be set to a value greater than the maximum rate.

Probe Fast Jog (+): Determines the speed of motion on an axis when fast jog is selected and a positive direction jog button is pressed while the probe is plugged in. The fast jog (+) rate cannot be set to a value greater than the maximum rate.

From the Probe Jog Parameters menu, you can return to the Jog Parameters menu by pressing **F8 – Machine Jog**.

15.2.2 F2 – Motor Parameters

This screen contains information about the motors, ballscrews, and switches installed on your machine. See the figure below.



Axis	Label	Motor revs/in	Motor steps/rev	Lash Comp. (Inches)	Limit -	Limit +	Home -	Home +	Dir Rev	Screw Comp
1	X	5.000000000	1600	0.000000	0	0	0	0	N	N
2	Y	5.000000000	1600	0.000000	0	0	0	0	N	N
3	Z	5.000000000	1600	0.000000	0	0	0	0	N	N
4	N	0.250000000	1600	0.000000	0	0	0	0	N	N
5	N	5.000000000	200	0.000000	0	0	0	0	N	N
6	N	5.000000000	200	0.000000	0	0	0	0	N	N
7	N	5.000000000	200	0.000000	0	0	0	0	N	N
8	N	5.000000000	200	0.000000	0	0	0	0	N	N

WARNING The Motor Parameters should not be changed without contacting your dealer. Corrupt or incorrect values could cause damage to the machine, personal injury, or both.

Special function indicators: These appear, if present, between the axis number and the label. 's' – indicates the axis is the spindle, 'p$' – axis is paired with axis '$', '*' – pairing conflict. See Machine Parameters for more information on setting up special functions.

Label: The letter you want to use to identify the axis. The first three axes should normally be X, Y, and Z. If a fourth axis is installed, it is usually named W or B. If you change a label, for example from X to A, the controller will then accept G-codes for axis A instead of X.

If fewer than four axes are present, the unused entries should be labeled N. If an axis is manually operated (it has an encoder but no motor), it should be labeled M. For a manual Z axis, the 3rd axis label should be set to @ symbol. This setting allows for two axes posting in Intercon. * WARNING: Intercon does NOT post two axis programs if the 3rd axis is labeled M.

Note: Tool length compensation (G43-G44) and canned drilling cycles (G73-G89) always affect the third axis, regardless of its axis label. Tool diameter compensation (G41-G42) always affects the first and second axes, regardless of their axis labels.

Motor revs/inch OR millimeters/motor rev: The number of revolutions of the motor that results in one inch of movement (if the machine is set up in inches). OR the number of millimeters that the machine will move as a result of one turn of the motor (if the machine is set up in millimeters).

Encoder counts/rev: The counts per revolution of the encoders on your servomotors.

Lash compensation: The uniform amount of backlash compensation to be applied along the whole length of the axis. Backlash can be observed during axis direction reversals and is a normal occurrence due to looseness or wear of moving parts in a machine. This parameter added to and works in conjunction with Screw Compensation (see below). Consult your machine manual or M-Series Service Manual for instructions on measuring backlash. The Lash Acceleration Coefficients, Parameters 208–215, can be used to change the speed that Lash Compensation is applied. A Coefficient of zero will effectively disable Lash Compensation.

Note: It is required that the machine be **rehomed** after changing Lash Compensation.

Limits: The PLC input numbers corresponding to any limit switches that you may have on your machine. Your installer should provide this information. If no limit switch is installed, this field should be set to 0.

Home: The PLC input numbers of any Home Switches you may have. These are similar to the limit switches. If your machine does not have home switches, this field should be set to the Limit Switch value. If no home or limit switch is installed, this field should be set to 0. You may then use hard stops as homing points if you choose.

Note: The Home Switch should never be physically located beyond the Limit Switch.

Direction reversed: Used to match the +/- reference of your machine to the control electronics. Toggle this value if you actually move in the X minus direction (reverse) when you jog X+.

Screw Compensation*: This value indicates whether mapping ballscrew compensation is enabled. Screw Compensation is similar to Lash Compensation (see above), but has differing compensations depending on the mapped locations along the axis. Screw Compensation is added to and works in conjunction with Lash Compensation. For more information, contact your dealer. It is recommended that you enable ballscrew error compensation at all times.

Note: It is recommended that a rehoming of the machine be done after changing Screw Compensation.

15.2.3 F3 – Find Home

Press **F3 – Find Home** to move an axis to its plus or minus home switch.

15.2.4 F4 – Set Home

Press **F4 – Set Home** to set Machine Home for an axis at its current position. This is usually performed after Find Home. This operation should not be used to set the part zero position. To set the part zero position, use the Part Setup screen.

15.2.5 F5 – M Comp

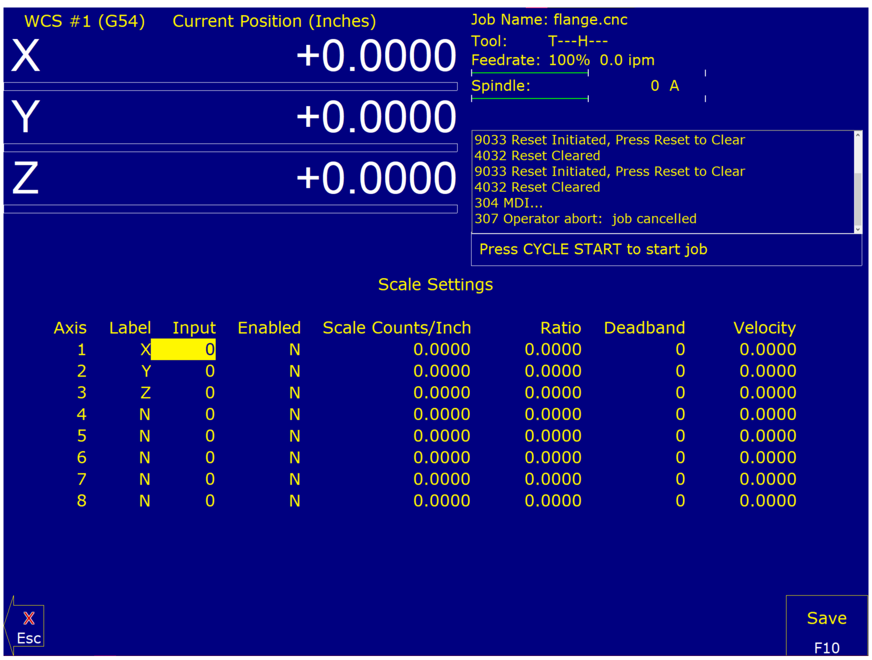
This menu lets you edit the ballscrew compensation tables.

NOTICE

The ballscrew compensation tables **should not** be changed without contacting your dealer. Corrupt or incorrect values could adversely affect the accuracy of the positioning of your machine.

15.2.6 F7 – Scales

This menu lets you set up scale encoders for the purpose of applying scale encoder correction to one or more axes.



NOTICE

The Scale Settings **should not** be changed without contacting your dealer. Corrupt or incorrect values could adversely affect the accuracy of the positioning of your machine.

Axis and Label are for informational purposes to indicate on which axis the scales will be applied. These values cannot be modified on this screen.

Input is the scale encoder number based on the map shown on parameters 308–315. Numbers 1–6 are on the MPU and 7–14 are on OpticDirect drives. If spare headers are available on the OpticDirect, they can be used for scale feedback.

Enabled "Y" enables the scale and "N" disables the scale. Use the spacebar to toggle choices and remember to choose F10 to save.

Scale Counts/Unit is the number of counts of the scale per unit of measurement. This value should come directly from the scale data sheet and should be entered in the control units. If the control is in inches, then the value should be entered in inches. If the control is in mm, then the value should be entered in mm.

Ratio is calculated as [(Motor Encoder Counts per Rev. * Motor Rev. per Unit) / Scale Counts per Unit] and cannot be modified. It shows how close the counts/unit are between the motor encoder and scale encoder.

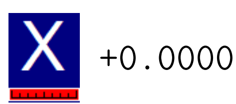
Deadband* is the number of encoder counts away from the commanded position that the scale position can be before compensating. Typically, you should start with a value of 0 or 1 and then increase it if the control goes into oscillation during movement.

Velocity* is the number of motor encoder counts / interrupt at which the Scales should adjust the position. Typically a value of 0.1 to 1.0 is a good starting value. To figure out a value to use based on a units/min. speed you need to convert it. Due to the nature of scale feedback, it is inherently an oscillator and by adjusting the Deadband and Velocity that oscillation can be kept to a minimum. If you are having oscillations you typically want to decrease the Velocity.

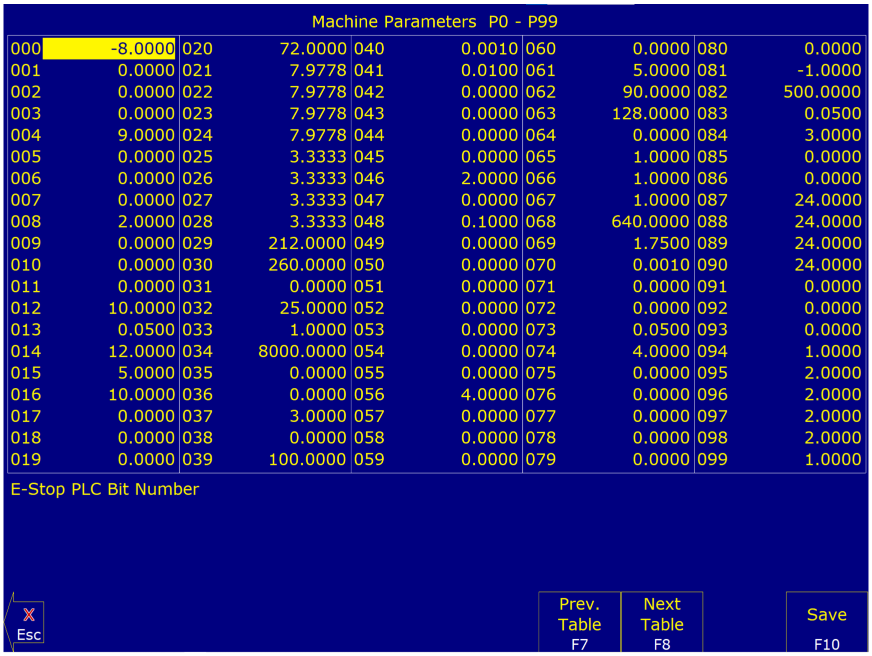
Scale Indicator Changing the Input, Enabled, or Scale Counts/Unit fields will cause scale compensation to be temporarily disabled. Scale compensation is also temporarily disabled during homing moves. Even though the scale is enabled in the menu, scale compensation will be disabled until the axis is rehomed. When a scale is configured for an axis, a scale indicator appears below the axis label on the DRO. It will have a green background when the scale is enabled and a red background when the scale is disabled.

* These starting values are only suggestions. You will need to adjust the values for your setup to determine what works well for you.

Note: Parameter 423 enables/disables the display of the scale indicator. Ensure that displaying the scale indicator is enabled under this parameter if attempting to enable scale indicators.



15.3 Machine Parameters (F3 – Parms from Configuration)



Machine Parameters P0 - P99

000	-8.0000	020	72.0000	040	0.0010	060	0.0000	080	0.0000
001	0.0000	021	7.9778	041	0.0100	061	5.0000	081	-1.0000
002	0.0000	022	7.9778	042	0.0000	062	90.0000	082	500.0000
003	0.0000	023	7.9778	043	0.0000	063	128.0000	083	0.0500
004	9.0000	024	7.9778	044	0.0000	064	0.0000	084	3.0000
005	0.0000	025	3.3333	045	0.0000	065	1.0000	085	0.0000
006	0.0000	026	3.3333	046	2.0000	066	1.0000	086	0.0000
007	0.0000	027	3.3333	047	0.0000	067	1.0000	087	24.0000
008	2.0000	028	3.3333	048	0.1000	068	640.0000	088	24.0000
009	0.0000	029	212.0000	049	0.0000	069	1.7500	089	24.0000
010	0.0000	030	260.0000	050	0.0000	070	0.0010	090	24.0000
011	0.0000	031	0.0000	051	0.0000	071	0.0000	091	0.0000
012	10.0000	032	25.0000	052	0.0000	072	0.0000	092	0.0000
013	0.0500	033	1.0000	053	0.0000	073	0.0500	093	0.0000
014	12.0000	034	8000.0000	054	0.0000	074	4.0000	094	1.0000
015	5.0000	035	0.0000	055	0.0000	075	0.0000	095	2.0000
016	10.0000	036	0.0000	056	4.0000	076	0.0000	096	2.0000
017	0.0000	037	3.0000	057	0.0000	077	0.0000	097	2.0000
018	0.0000	038	0.0000	058	0.0000	078	0.0000	098	2.0000
019	0.0000	039	100.0000	059	0.0000	079	0.0000	099	1.0000

Note: For Acorn and AcornSix users, please use the Acorn Wizard to change parameters. Using the Machine Parameters menu to change parameters when they are present in the Wizard can cause issues.

This screen provides you with a method of changing various parameters that are used by the control. Altogether, you have access to 1000 parameters spread across 10 tables. Each table gives you access to 100 parameters at a time. You can navigate between tables using the following keys: **F7 – Previous Table** and **F8 – Next Table**. The title at the top tells you which table you are on. If you wish to change a field in the table, use the arrow keys to move the cursor and select the desired field. A short description of the parameter will appear below the table. Type the new value and press **ENTER**. When you are done editing the fields, press **F10 – Save** to accept any changes you have made and save them. Note that **F10 – Save** is a single operation that will save all changes in every table that you modified. Pressing **ESC** will discard all changes in every table that were modified and will return to the previous menu [Setup].

Note: Many machine parameters can also be set with the G10 G-code or by #variable assignment. Many machine parameters can more easily be configured through the Wizard.

15.3.1 Bit-mapped parameters

Certain control parameters are defined by bit-mapped values. The Acorn and Acorn Six includes a Wizard which sets the parameters without the user manually setting parameters or using bit-mapped values. A bit-mapped parameter is stored as a number, representing a 16-bit value in the control. Where the Value = 2Bit. If a certain bit needs to be turned on, that bit's value must be added to the parameter value, if the bit needs turned off, its value must be subtracted from the parameter value. The values for each of the 16 bits' can be seen in the table below.

Bit-Mapped Parameter Bits and Values where Value = 2^{Bit}

Bit	0	1	2	3	4	5	6	7
Value	$2^0 = 1$	$2^1 = 2$	$2^2 = 4$	$2^3 = 8$	$2^4 = 16$	$2^5 = 32$	$2^6 = 64$	$2^7 = 128$

Bit	8	9	10	11	12	13	14	15
Value	$2^8 = 256$	$2^9 = 512$	$2^{10} = 1024$	$2^{11} = 2048$	$2^{12} = 4096$	$2^{13} = 8192$	$2^{14} = 16\,384$	$2^{15} = 32\,768$

To set bit-mapped parameters simply add together the values that you need to have enabled. In the manual we include the values for simple adding for the desired configuration

For Example.

Parameter 10 is bit-mapped below.

Bit	Function Description	Value
0	Display M & G-codes in M function macros?	Add 1
1	Step through M function macros in Block Mode?	Add 2
2	Decelerate to stop on M105 and M106. With decel set these moves take longer and are slightly less accurate. With immediate stop these moves are faster and more accurate; however the lack of controlled deceleration can cause excessive machine vibration.	Decel (Add 4) Immediate Stop (Add Nothing)
3	Do NOT move to home on M6	Add 8
4	(not used) This bit was previously used for controlling the type of deceleration to stop on digitizing and probing moves. This functionality is now controlled by parameters 366 and 367.	
5	Allow M0 jogging? (May require a Pro/Ultimate license)	Add 32

To enable *Display M & G-codes in M function Macros (Bit=0), Move to Z home on M6 (Bit=3), and Allow M0 jogging (Bit=5)* the addition is as follows:

Function Decription	Bit	Value Equation	Value
Display M & G-codes in M function Macros	0	$2^0 = 1$	1
Move to Z home on M6	3	$2^3 = 8$	8
Allow M0 jogging	5	$2^5 = 32$	32
Total Parameter Value			41

The Total Parameter Value to enable all three features would be 41. The value to input into Parameter 10 is 41.

The following parameters are currently defined:

Parameter	Definition	Default setting
0	E-Stop PLC Bit	0
1	Y jog key orientation	0
2	G-Code Interpretation Control and Slaving Rotary axis feedrate	0
3	Modal Tool and Height Offset Control	0
4	Remote File Loading Flag & Advanced File Ops	0

Parameter	Definition	Default setting
5	Machine Home/Startup Setup	0
6	Auto Tool Changer Installed	0
7	Display Colors	0
8	Available Coolant System(s)	2
9	Display Language	0
10	Macro M-Function Control/Probe Stop Handling	0
11	Touch Probe PLC Input	15
12	Touch Probe Tool Number	0
13	Probing Recovery Distance	0.05" / 1.27 mm
14	Fast Probing Rate	10 ipm or 254 mm/min
15	Slow Probing Rate	1 ipm or 25.4 mm/min
16	Probing Search Distance	10" / 254 mm
17	Tool Detector Reference Number	0
18	PLC Input Spindle Inhibitor	0
19	MPG modes	0
20	Ambient Temperature	72 °F / 22 °C
21	Motor Heating Coefficients for axes 1,2,3,4	Refer to text
25	Motor Cooling Coefficients for axes 1,2,3,4	Refer to text
29	Warning Temperature	150 °F / 65.5 °C
30	Limit Temperature	180 °F / 82 °C
31	Legacy SPIN232 Com Port	0
32	Autonomous Digitizing Offset Files	27
33	Spindle Motor Gear Ratio	1
34	Spindle Encoder Counts/Rev	8,000
35	Spindle Encoder Axis Number	0
36	Rigid Tapping Enable/Disable	0
37	Spindle Deceleration Time	10
38	Multi-Axis Max Feedrate	0
39	Feedrate Override Knob Limit	120
40	Basic Jog Increment	0.0001" / 0.001 mm
41	Rotary Axis Jog Increment	0.01 °
42	Password for Configuration Menus	0
43	Automatic tool measurement options	0
44	TT1 PLC input number	0
45	WCS Lockout	0
46	Active G-Codes Display	0
48	Grid Digitize Patch Playback Z rapid clearance amount	.1
49–51	Small Arc Feedrate Limiting	0
56	Feedrate Override Display Properties	0
57	Use Generic Load Meter Data from PLC	0
61	High Power Stall Timeout	0.5
62	High Power Stall PID Limit	115
63	High Power Idle PID Multiplier	1.5
64	4^{th}/5^{th} Axis Pairing	0
65	Spindle Gear Ratios	1.0
68	Minimum Rigid Tapping Spindle Speed	0
69	Duration For Minimum Spindle Speed	1.0
70	Offset Library Inc/Decrement Amount	0.001" / 0.02 mm
71	Part Setup Detector Height	0
72	Data Recording M-Function Options	0

Parameter	Definition	Default setting
73	Peck Drill Retract Amount	0.05
74	M-Function executed at bottom of tapping cycle	4
75	Axis Summing Control	0
76	Manual Input Unrestricted Distance	0
77	Manual Input Movement Tolerance	0
78	Spindle Speed Display and Operations	0
79	Paired Axes Re-sync Delay Time	0
80	Voltage Brake Applied Message Frequency	0
81	Air Drill M-Function (executed instead of Z movement in drilling)	-1
82	Spindle Drift Adjustment	0
83	Deep Hole Clearance Amount	0.05
84	M-Function executed at return to initial point of tapping cycle	3
85	“Door Open” Interlock PLC bit	0
86	Rapid/Linear vector rate limit	0
87	Autotune Ka Performance parameters for axes 1,2,3,4	0
91	Axis Properties for axes 1,2,3,4	0
95	Autotune / Auto Delay Move Distance for axes 1,2,3,4	2" / 50.8 mm
99	Cutter Compensation Look-ahead	6
100	Intercon comment generation	0
101	Intercon clearance amount	0.1
102	Intercon spindle coolant delay	3.0
103	Intercon corner federate override	50.0
104	Intercon modal line parameters	0
105	Intercon modal arc parameters	0
106	Intercon modal drilling cycle parameters	0
115	Intercon Help	0
116	A-Axis Y Coordinate	0
117	A-Axis Z Coordinate	0
118	B-Axis X Coordinate	0
119	B-Axis Z Coordinate	0
120	Probe stuck clearance amount	0.10" / 2.54 mm
121	Grid digitize prediction minimum Z pullback	0.002" / 0.0508 mm
122	Grid digitizing deadband move distance	0.0002" / 0.005 08 mm
123	Radial digitizing clearance move	0
128	Dry Run PLC Bit	
129	Dry Run Feedrate	
130	Z axis on/off selection	0
131	4^{th} axis on/off selection	0
132	Motor Heating Coefficients for axes 5,6,7,8	Refer to text
136	G76 Fine Bore Retract Angle	0
137	Load Meter Filter Size	0
138	DRO Encoder Deadband	0
139	Special Dwell between Moves	0
140	Message log priority level	1
141	Maximum message log lines	100
142	Message log trim amount	100
143	DRO properties (load meters, 4/5 digits, DTG)	0
144	Comparison rounding	0
145	Advanced macro properties (fast branching)	0
146	Feed hold threshold for feed rate override	0

Parameter	Definition	Default setting
147	Number of Messages in Operator Message Window	10
148	Miscellaneous Jogging Options	0
149	Spindle Speed Threshold	1.0
150	Backplot Graphics display options	0
151	Repeatability tolerance for probing	0
153	Probe Protection	
154	Touchscreen options	0
155	DSP Probe Installed	0
156	Autotune / Auto Delay Move Distance for axes 5,6,7,8	2" / 50.8 mm
160	Enhanced ATC	0
161	ATC Maximum Tool Bins	0
162	Intercon M6 Initial M-Code	0
163	Gang tooling	0
164	ATC Feature Bit	0
165	Acceleration/Deceleration Options	0
166	Axis Properties for axes 5,6,7,8	0
170	PLC parameters	0
178	PLC I/O configuration (PLC program specific)	0
179	Lube Pump Operation	0
180	Clear Home Switch Distance	0
186	Probe Stuck retry disable	0
187	Hard Stop Homing Power Limit	0
188	Aux key functions	0
200	OPTIC4 Tach Volts Per RPM	0
208	MPU Lash/Screw Comp Acceleration Coefficient	0.125
216	PC Based Lash Compensation on/off	0
217	PC Based Screw Compensation on/off	0
218	USB MPG Options	0
219	Virtual Control Panel Options	0
220	Smoothing Parameters	Refer to text
236	Motor Cooling Coefficients for axes 5,6,7,8	0
240	Rigid Tapping Accel Rate Distance	0
241	Rigid Tapping Rotational Step Size (Degrees)	10.00000
242	Minimum Angle Threshold for application of Accel/decel in threading moved	0
243	Threading Control	0
244	Tool Touch-off device PLC input number	0
245	G7x D/P/Q “implied float” re-interpretation threshold for Inch	99
246	G7x D/P/Q “implied float” re-interpretation threshold for MM	99
247	G70 Multiple Pass Behavior Suppression	0
248	Tool Wear Adjustment magnitude limit	0.2" / 5.08 mm
252	Autotune Ka Performance parameters for axes 5,6,7,8	0
256	Drive Mode	0
257	TT1 connection detection PLC input	0
258	Velocity/Torque Mode override in Precision mode	0
259	Manual Axis Designation	0
260	Accelerated Graphics Backplot on/off	1
261	Precision Mode Standoff (Tolerance Percentage/Delay Time)	0
263	Change DRO display precision	4.0
270	Skew Correction: X Skew offset from Trusted/Intended X	0
271	Skew Correction: Trusted/Intended Y	0

Parameter	Definition	Default setting
278	Spindle Speed Display Precision	0
281	Tool Touch-off Device X stylus size	Refer to text
282	Tool Touch-off Device Z stylus size	Refer to text
283	Auto Tool Touch-off safety clearance	Refer to text
284	Brake Resistor Wattage for ACDC Drives 1–8	50
292	Aux Key Functions	
300	Drive assignment to Axes 1–8	Refer to text
308	Encoder assignment to Axes 1–8	Refer to text
316	Absolute Encoder Bits	0
317	Single Turn Absolute Encoder Bits	0
318	Five Axis Configuration	0
319	Five Axis Options	0
321	MPU13 DSP Probe Input	0
322	Spindle Encoder Port	0
323	MPU11 Encoder Speed Filter	0
324	Axis Boxcar size	0
332	Encoder error suppression	0
336	Motor torque estimation for velocity mode drives	Refer to text
340	Precision Mode delay (in milliseconds) for axes 1–8	0
348, 351, 354	MPG 1, 2, 3 Encoder Input	15, 0, 0
349, 352, 355	MPG 1, 2, 3 Detents per Revolution	100
350, 353, 356	MPG 1, 2, 3 Encoder Counts per Revolution	400
357	Axis Drive Max RPM	0
365	Drive power-on delay	0.0
366	Probe deceleration multiplier	2.0
367	TT1 deceleration multiplier	2.0
368	Autonomous Digitizing Angle Adjustment	4
369	Tool Check Max Absolute Angle	0
374	ACDC Drive Debug Log Axis Inclusion	0
375	ACDC Drive Debug Log Size	0
376	ACDC Drive Debug Log Collection Type	0
377	ACDC Drive Debug Log Trigger Type	0
378	ACDC Drive Debug Log Trigger Level	0
379	ACDC Drive Debug Log Option	0
387	Debug Trace Mask	0
388	Debug Options	0
389	Debug Level	0
392	DP-7 parameters	Refer to text
395	Probing setup traverse speed	30 ipm or 762 mm/min
396	Probing setup plunge speed	30 ipm or 762 mm/min
397	Combustion Chamber Clearance Height	
398	Port/block mode	0
399	AD1 arc chord tolerance adjustment	.5
400	Run Menu Cycle Start Enabled	0.0
401	Forget Last Job Loaded	0.0
403	Disable Keyboard Jogging Legend	
404	Unused	
405	Tool Touch Off Type	
406	Probe State When Tripped	0
408	Active G-code Display	0.0

Parameter	Definition	Default setting
409	Probe Type	
410	Probe/Tool Touch Off Warning	
411	Mpg Type	0
413	Park macro	0
415	Ether1616 Configuration Bits	0
416	Spindle Inhibit	0
417	Preview G-Code Before Job	
418	Assign Enter Key to Cycle Start	
421	Clean Fan Filter Reminder	0
422	PLC Diagnostics Display	0
423	Display Scale Position on DRO	0
425	Force Rehoming After EStop Condition	
430	RTG Spindle Speed RPM Display	
440	Stopped for Jogging Continue Bit	0
441, 442, 443	MPG (1–3) Axis Selection	
450	Inhibit MPG Z Axis Fast Movement	
459	Second Spindle Encoder Axis Number	0
460	Second Spindle Maximum Speed	0
461	Second Spindle Minimum Speed	0
462	Second Spindle Encoder Counts/Revolution	0
507	5^{th} Axis Pairing – Slave Axis	0
508	5^{th} Axis Pairing – Master Axis	4
560	Laser X Offset	0
561	Laser Y Offset	0
600	A Axis Expected Velocity Switch Point	0
601	A Axis Scale Switch To Velocity	0
602	A Scale Correction Meter Max Deflection Count	0
603	B Axis Expected Velocity Switch Point	0
604	B Axis Scale Switch To Velocity	0
605	B Scale Correction Meter Max Deflection Count	0
814	Base Frequency (Hz)	
815	Inverse Output	
817	PWM	
830	ATC Type	
855	MPG Performance	
856	VCP Jogging State on Acorn Power Up	
900	PLC program parameters	–

15.3.2 Parameter 0 – E-Stop PLC Bit

This parameter specifies the PLC bit to which the physical Emergency Stop switch is connected. It is mainly used for ATC applications that use custom PLC messages. See table below for examples.

PLC Type	ESTOP Input on PLC	Parameter Value
GPIO4D	Input 11	-11
ALLINONE	Input 11	-11
OAK	Input 11	-11
RTK2/3/4	Input 11	-11
PLCIO2	Input 11	-11
DC3IO	Input 11	-11
Servo3IO	Input 1	1

15.3.3 Parameter 1 – Y jog key orientation

This parameter is a 3-bit field where bit 0 is not used in the mill software. Bit 1 sets the direction of movement for the Y+ and Y- jog keys and bit 2 will swap the X and Y jog keys. This should always be set to 0 except for very special applications.

Note: PLC program interaction is needed for these features.

Bit	Function Description	Value
0	Not	Used
1	Flip direction of Y jog keys?	Add 2
2	Exchange X axis and Y-axis jog keys?	Add 4

15.3.4 Parameter 2 – G-code Interpretation Control and Slaving Rotary axis feedrate

This parameter is a bit field that controls optional interpretation of several G-codes. The following table shows the functions performed by the value entered in this parameter:

Bit	Function Description	Value
0	Arc centers I, J, K are absolute in G90 mode?	Add 1
1	Allow Z being specified alone to be sufficient to trigger execution of a canned tapping or drilling cycle to be executed?	Add 2
2	Interpret dwell times associated with G4, G74, G82, G84, and G89 as milliseconds rather than seconds?	Add 4
3	Slaving rotary axis feedrate to non-rotary axis feedrate. Note that this feature has no effect for movement commands handled by Smoothing (when Parameter 220=1).	Add 8
4	Selects the center for scale, mirror and rotate. By default the center will be 0,0,0.	Add 16
5	Prevent rotary-only moves (rotary axis by itself on a line of G-code) from acquiring a remembered slaved rotary feedrate previously set by a previous line of G-code containing a rotary and non-rotary together. Note that this feature has no effect for movement commands handled by Smoothing (when Parameter 220=1).	Add 32

15.3.5 Parameter 3 – Modal Tool and Height Offset Control

Bit	Function Description	Value
0	Tool and Height Offset numbers, T and H, will be reset upon job completion (and not remain modal and active between jobs).	Reset upon job completion (Add 1)
1	Reference tool position is set to Z home.	Add 2
2	Tool Height Offset Retention option. This option prevents the current tool height offset from being turned off when the user enters the Tool Height Offset menu. Additionally, this option will cause the H height offset to be set to match the tool number when an Auto Tool Change is performed (via F7 ATC) in the Offset Library.	Turn on Tool Height Offset Retention option (Add 4)
3	Unused for Mill	

15.3.6 Parameter 4 – Remote File Loading Flag & Advanced File Ops

This parameter controls the action of the Load Job menu when CNC job files are selected from drives letters higher than C. These drives (i.e. drives D, E, F, etc.) are presumed to be network drives or extra hard drives.

Bit	Function Description	Value
0	Job files are copied to the C drive (`c:\cncm \ncfiles`) when they are loaded. The local copy is used when the job runs.	Add 1
1	Turn on file caching. Job files are temporarily cached on the C drive. The cached copy is used while the job is running. The cached copy is deleted when the next job is loaded or when Parameter 4 changes to a 0 or 1. Digitize files are cached as the machine is digitizing. When digitizing is complete, the resulting file is copied to the digitize directory specified in pathm.ini.	Add 2
2	Set the Advanced File load menu as default for loading files	Add 4
3	Use Windows-style File Open dialogue	Add 8
	Windows-style dialogue will remember last opened location	Add 24

15.3.7 Parameter 5 – Machine Home/Startup Setup

This parameter controls machine homing upon startup of the control. The following table details the functions controlled by this parameter:

Bit	Function Description	Value
0	Suppress the requirement to set machine home before running jobs	Add 1
1	Display router bit map at homing screen	Add 2
2	Disable stall detection when the CNC software first starts	Add 4

If bit 0 of Parameter 5 the requirement to set machine home before running. If bit 0 of Parameter 5 is 0, machine home must be set before jobs may be run. If bit 0 of Parameter 5 is 1, machine home is not requested or required, but Graphing and running of jobs will not work until the Z-axis is homed.

Note: Parameter 5 Bit 0 is separate from the “Machine Home at Powerup” flag in the Control Configuration Screen. Parameter 5 Bit 0 determines **whether** you must home the machine; the “Machine Home at Powerup” flag determines **how** you will home the machine, if you must do so.

15.3.8 Parameter 6 – Automatic Tool Changer Installed

This parameter tells the control whether an automatic tool changer installed on your machine. This field affects the action of M6 in your CNC programs. See M6 for more information. It also affects whether the ATC key is present in the Tool Offset Setup and whether to save the last tool change number in the job files.

Function Description	Value
Auto Tool Changer NOT Installed	0
Auto Tool Changer Installed	1

15.3.9 Parameter 7 – Display Colors

This parameter determines what combination of colors will be used for a display. A value of 0 will set the display colors to Centroid Classic. If you overwrite Centroid Class via the Color Picker set this value to a 0 to reset it. A value of 1 will set the display to monochrome. A value of 1 is a Legacy Setting and is not recommended for modern displays. A value of 4 for this parameter will automatically be set when any Color Profile aside from "Centroid Classic" is selected from the Color Picker (F5 in the Utility Menu) is selected.

15.3.10 Parameter 8 – Available Coolant Systems

This parameter is used by Intercon to determine what coolant systems are available on the machine. It should be set as follows:

Function Description	Value
Mist Coolant (M7) only	1
Both Coolant systems	2
Flood Coolant (M8) only	3

15.3.11 Parameter 9 – Display Language

This parameter determines what language will be used for menus, prompts and error messages.

Function Description	Value
English	0
Spanish	1
French	2
Traditional Chinese	3
Simplified Chinese	4
German	5
Swedish	6
Finnish	7
Portuguese	8
Polish	9
Greek	10

15.3.12 Parameter 10 – Macro M function handling/Probe Stop Handling

This parameter is a bit field that controls various aspects of M functions. The following table shows the functions performed by the value entered in this parameter. The default value is 0.

Bit	Function Description	Value
0	Display M & G-codes in M function macros?	Add 1
1	Step through M function macros in Block Mode?	Add 2
2	Decelerate to stop on M105 and M106. With decel set these moves take longer and are slightly less accurate. With immediate stop these moves are faster and more accurate; however the lack of controlled deceleration can cause excessive machine vibration.	Decel (Add 4) Immediate Stop (Add Nothing)
3	Do NOT move to home on M6	Add 8
4	(not used) This bit was previously used for controlling the type of deceleration to stop on digitizing and probing moves. This functionality is now controlled by parameters 366 and 367.	
5	Allow M0 jogging? (May require a Pro/Ultimate license)	Add 32

15.3.13 Parameter 11 – DP4 PLC Input Number and Contact State

This parameter is used for the PLC input number that is used by the DP4 Touch Probe. Allowable range is a single value, +/-1 to 240 and 50001 to 51312. A Positive number indicates Closed on contact and a negative number indicates Open on contact. A list of default settings for different console types are listed below.

NOTICE

Changing this parameter can cause damage to your probe. You should contact your Dealer or Local Tech Representative before any modifications are made.

MPU11/Oak/Allin1DC	50769
Acorn/AcornSix	Set by Wizard

15.3.14 Parameter 12 – Touch Probe Tool Number

This parameter is the tool number of the DP4 probe. Allowable range is 0 through 200. By default the value is 10. This is used to look up the length offset and tip diameter of the probe in the Tool Offset Library.

15.3.15 Parameter 13 – Recovery Distance

This parameter is the distance that the probe moves off a surface after initial contact (only during probing cycles), before returning to the surface to take a recorded reading. **For DSP Probes:** This parameter is used for failed DSP windows. On a failed window, the DSP probe will retract this distance before retrying.

15.3.16 Parameter 14 – Fast Probing Rate

The fast probing rate is used for positioning moves and initial surface detection, and is determined by the machines response time and the permitted probe deflection. The default is 10 in/min. This is a very conservative feedrate, in actual use 20 to 30 in/min is a good value and will not have any detrimental impact on accuracy in most cases.

15.3.17 Parameter 15 – Slow Probing Rate

The slow probing rate is used for the final measurement moves. The default setting is 1 in/min. The following are some typical accuracy tolerances with the corresponding value set in parameter 15:

Probing Rate		Accuracy	
0.5 in/min	12.7 mm/min	0.0001 in	0.002 54 mm
1 in/min	25.4 mm/min	0.0002 in	0.005 08 mm
1.5 in/min	38.1 mm/min	0.0005 in	0.0127 mm
3.5 in/min	88.9 mm/min	0.0010 in	0.0254 mm
18 in/min	457.2 mm/min	0.0050 in	0.1270 mm

15.3.18 Parameter 16 – Maximum Probing Distance

This is the maximum distance that the Boss and Web probing cycles "search" for a surface in a given direction if no travel limits have been entered.The default setting is 10 inches.

A larger value should be entered for the boss and web cycles if you are measuring very large features.

These settings are conservative measurements and can be used initially for startup purposes, however they can be changed to accommodate your work.

15.3.19 Parameter 17 – Detector Location Return Point

A non-zero value specifies the number of the reference return point (entered into the WCS menu) directly above a permanently mounted TT-1 tool detector. When the Auto function is called up in the tool offset library, the control will position the table to the return point specified by this parameter, and touch the tool off the TT-1 Tool detector.

Entry	Return Point
0	None
1	G28
2	G30
3	G30P3
4	G30P4

A zero indicates that the tool detector is no permanently mounted; automatic tool measurement will be performed without X/Y axis movement.

15.3.20 Parameter 18 – PLC Input Spindle Inhibit Parameter

This parameter stores the input for the Spindle Inhibit feature. A negative value must be entered if a "normally closed" probe is to be used with the control. A positive value must be entered if a "normally open" probe is to be used with the control. The absolute value of Parameter 18 will directly reflect the PLC input the Spindle Inhibit is wired to. When this parameter is set, Digitizing and Probing cycles will not run unless a probe or touch-off block is connected. This parameter is used to prevent the tool or probe from crashing into the table. The default for this parameter is 0, which disables this feature.

PLC Type/Model	Input Number
Allin1DC	50771
MPU11	50771
DC3IO/DC3IOB	15
RTK3	15
Servo3IO/M15DRV1	5
PLCIO2	10
15/15	10
RTK2	10
Koyo ATC	2

15.3.21 Parameter 19 – MPG modes

The MPG is a hand-held device that is used as an alternate way of jogging the machine. This parameter defines the MPG's mode of operation.

Bit	Function Description	Value
1	Enable $\times$100 Z-Lockout functionality*	Add 2
2	Enable Z axis MPG* – This will allow the z-axis to be moved with the MPG while running a job independent of the x and y axes	Add 4

*PLC program interaction is needed for these features. The plc program is in direct control of MPG modes. Z-axis MPG operation is not available with all controls.

15.3.22 Parameters 20–30 (also 132–135, 236–239) – Motor Temperature Estimation

These parameters are used for motor temperature estimation. Parameters 20, 29 and 30 correspond respectively to the ambient temperature of the shop, the overheating warning temperature, and the job cancellation temperature, all in degrees Fahrenheit. Parameters 21–24 and 132–135 are the heating coefficients. Parameters 25–28 and 236–239 are the cooling coefficients.

To disable Motor Temperature Estimation for an axis, set its heating and cooling coefficients to 0. For example, to disable Motor Temperature Estimation for axis 1, set Parameter 21 to 0, and set Parameter 25 to 0.

Note: Temperature estimation only applies to controls operating in Torque mode (i.e. DC brushed systems and Centroid AC systems). MPU11 systems running in Velocity mode (i.e. third party drive systems) do not use this feature, and thus should be disabled (by setting all heating and cooling coefficients to 0).

Suggested values for AC Brushless Motors and Drives						
SD Drive		SD3, SD1 750 W motors	SD3, SD1 1, 2 KW motors	SD3, SD1 (finned heatsink) 1, 2, KW motors	SD1 45A (finned heatsink) 3 KW motors	SD1 45A (finned heatsink) 4 KW motors
Parameters	Axes	Values	Values	Values	Values	Values
21–24	1–4	0.23	0.5	0.23	0.23	0.23
132–135	5–8	0.23	0.5	0.23	0.23	0.23
25–28	1–4	12.0	9.0	12.0	12.0	14.5
236–239	5–8	12.0	9.0	12.0	12.0	14.5
20	N/A	72	72	72	72	72
29	N/A	150	150	150	150	150
30	N/A	180	180	180	180	180

Suggested values for DC Brush Motors and Drives						
Servo Drive		9A Drive, 16 in/lb motors	12A Drive, 29 in/lb motors	15A Drive, 29 in/lb motors	15A Drive, 40 in/lb motors	25A Drive, 40 in/lb motors
Parameters	Axes	Values	Values	Values	Valules	Values
21–24	1–4	0.028	0.02	0.027	0.03	0.04
132–135	5–8	0.028	0.02	0.027	0.03	0.04
25–28	1–4	0.68	0.68	0.68	0.68	0.68
236–239	5–8	0.68	0.68	0.68	0.68	0.68
20	N/A	72	72	72	72	72
29	N/A	150	150	150	150	150
30	N/A	180	180	180	180	180

15.3.23 Parameter 31 – Legacy SPIN232 Com Port

For values 1–255, CNC12 will attempt to open that COM port and send out spindle commands. CNC12 should be restarted after changing this value from 0 (disabled) to a valid value. Note that the baud rate is assumed to be 19200 to work specifically with a SPIN232.

15.3.24 Parameter 32 – Autonomous Digitizing Offset Files

This bitwise parameter specifies which files should be output from the autonomous digitizing offset surface calculations. The bits are:

Bit	Function Description	Value
0	Raw Centerline Data	Add 1
1	Clean Centerline Data	Add 2
2	Centerline Flow Splines	Add 4
3	Legacy Offset Data	Add 8
4	Spline Offset Data	Add 16
5	Spline Offset Flow Splines	Add 32
6	Offset Plus Flow	Add 64

15.3.25 Parameter 33 – Spindle Motor Gear Ratio

NOTICE The default value for this parameter is 1 and should not be changed unless you have consulted your dealer or local Technical representative!!!

Parameter 33 is used for the gear or belt ratio between the spindle motor and the chuck in high gear range. It should be greater than 1.0 if the motor turns faster than the chuck and less than 1.0 if the chuck turns faster than the motor.

Note: This value applies to high range. The ratio between high range and lower ranges is established by the gear ratio parameters (65–67).

15.3.26 Parameter 34 – Spindle Encoder Counts/Rev

This parameter controls the counts revolution for the spindle encoder. Input from the spindle encoder is required for the spindle-slaved movements used in the Rigid Tapping cycles. If the encoder counts up when running CW (M3), the value of this parameter must be positive. If the encoder counts up when running CCW (M4), the value of this parameter must be negative.

15.3.27 Parameter 35 – Spindle Encoder Axis Number

Input from a spindle encoder is required for spindle-slaved movements such as those used in the Rigid Tapping cycles. If there is no spindle encoder connected, then this parameter should be set to 0. This parameter specifies the axis number (1 through 8) to which the spindle encoder is assigned. Encoder assignments are specified by parameters 308–315. For example, if you decide to configure the 5th axis as the spindle, and the spindle's encoder is the 1st MPU11 onboard input encoder, then Parameter 35 (this parameter) should be set to 5, and Parameter 312 should be set to 1.

15.3.28 Parameter 36 – Rigid Tapping Enable/Disable

This parameter is a bit field that enables or disables Rigid Tapping and its options. Bit Function

Bit	Function Description	Value
0	Enable Rigid Tapping?	Add 1
1	Suppress sending "Wait for Index Pulse" during Rigid Tapping?	Add 2
2	Allow Spindle Override during Rigid Tapping?	Add 4
3	Use Spindle Off system variable bit? (see note below)	Add 8 (see note below)
4	What is the Spindle Encoder measuring? I.e. What is the physical mounting location of Spindle Encoder? (see note below)	Spindle Encoder measures rotation of spindle drive motor (Add 16) (see note below)
5	Turn Off Spindle Following While At Bottom Of Hole. (NOT RECOMMENDED)	Add 32

Note on Bit 3: This bit enables the spindle off system variable, for mpu11 systems. Most systems will not need to set this bit. Mpu11 systems will not execute custom M5 macro commands during a rigid tap. Instead the software determines which bit, M3 or M4, to turn off to stop the spindle. Alternatively, setting bit 3 will cause the software to set the spindle off system variable bit, `SV_PC_RIGID_TAP_SPINDLE_OFF`. The plc program is then responsible for monitoring that bit and performing all actions in order to turn off the spindle.

Note on Bit 4: On machines where the Spindle Encoder is set up to measure the spindle drive motor rotation (Bit 4 = on, value 16), multi-pass Rigid Tapping and repeated Rigid Tapping down the same hole can only be done in the High Spindle Range. (See Parameters 33 and 65–67 for description of Spindle Gear ranges/ratios.)

15.3.29 Parameter 37 – Spindle Deceleration Time (Rigid Tapping Parameter)

This parameter is used in conjunction with Parameter 36 when rigid tapping is enabled. This sets the amount of time required for the spindle to decelerate before it switches direction during a rigid tapping operation.

15.3.30 Parameter 38 – Multi-Axis Max Feedrate

This parameter is used to limit the feedrate along all commanded move vectors. This parameter can be used to limit the speed of multi-axis moves on machines that may have enough power to move a single axis rapidly, but starve out of power on 2 or 3 axis rapid moves. A zero in this parameter will disable this feature. Note that this feature has no effect for movement commands handled by Smoothing (P220=1).

15.3.31 Parameter 39 – Feedrate Override Percentage Limit

This parameter is used for limiting the upper end of the Feedrate Override Knob percentage to a value from 100% to 200%. This parameter can be used to restrict the Feedrate Override Knob effect on machines with maximum rates over 200 in/min. The Feedrate Override Knob percentage is normally allowed to go to 200%. However, on machines with high cutting speeds, if the knob is turned up to 200%, it creates overshoots on corners. If this parameter is set to something like 110, it will stop the Feedrate Override Knob from exceeding 110% and thus causes the overshoots to disappear. The PLC program must check this parameter to actually enforce the limiting of Feedrate Override.

15.3.32 Parameter 40 – Basic Jog Increment

This parameter holds the basic jog increment for linear axes (0.0001" or 0.002 mm by default). This value works in conjunction with the ×1, ×10, and ×100 jog keys to determine the amount to move a linear axis during incremental jogging.

15.3.33 Parameter 41 – Rotary Axis Jog Increment

This parameter holds the jog increment for rotary axes (0.01° by default). This value works in conjunction with the ×1, ×10, and ×100 jog keys to determine the amount to move a rotary axis during incremental jogging.

15.3.34 Parameter 42 – Password for Configuration Menus

This parameter determines the password that the user must enter in order to gain full access to the configuration menus.

Function Description	Value
No password required for supervisor access; the user is not prompted for a password	54.0
Password is 4 digits represented by “ABCD” Example: for the password to be 1234, set to 1234.1234	ABCD.ABCD
Password is “137”	Any other number

15.3.35 Parameter 43 – Automatic tool measurement options

This parameter is a bit field that is used to configure properties of the TT1.

Bit	Function Description	Value
0	The height of the tool detector (Parameter 71) will be subtracted from the measured height of the tool.	Add 1
1	Which PLC input to use for the Tool Z reference measurement. 0 = Use Touch Probe input in Parameter 11. 2 = Use TT1 input in Parameter 44.	0 = Use Parameter 11. 2 = Use Parameter 44.
2	Remind the operator to plug in the TT1 before doing Batch Tool Measuring.	Add 4

15.3.36 Parameter 44 – TT1 PLC input number

This parameter is the input number that the TT1 is wired into on the PLC. If a shared PLC input is used for the TT1 and the DP4 probe, then the value can be left at zero or set to the same value as Parameter 11.

NOTICE

If you are using a different PLC input for the TT1 and DP4 when setting the Z reference in the tool library with the DP4, make sure you don't use a ruby probe tip. The TT1 is electrical continuity based and the ruby tip is not electrically conductive!

15.3.37 Parameter 45 – WCS Lockout

This parameter allows you to "lock out" a WCS from editing if you do not want its values to change.

Note: G92 can still set a WCS even if it is locked out.

WCS to Lockout	Value
WCS #1	Add 1
WCS #2	Add 2
WCS #3	Add 4
WCS #4	Add 8
WCS #5	Add 16
WCS #6	Add 32
WCS #7	Add 64
WCS #8	Add 128
WCS #9	Add 256
WCS #10	Add 512
WCS #11	Add 1024
WCS #12	Add 2048
WCS #13	Add 4096
WCS #14	Add 8192
WCS #15	Add 16384
WCS #16	Add 32768
WCS #17	Add 65536
WCS #18	Add 131072

15.3.38 Parameter 46 – Active G-Codes Display

Set this parameter to 0 to always display the currently active G-codes in the bottom left corner of the main screen and MDI screen. Set it to 1 to display the G-codes only in MDI. Set it to 2 to disable the display completely.

15.3.39 Parameter 48 – Grid Digitize Patch Playback Z rapid clearance amount

This is the additional Z clearance amount higher than the Z surface level at which the original Grid Digitizing operation was begun. The purpose of this value is to set the recorded starting "rapid to" Z level of a Grid Digitize playback patch. In other words, the Z starting point of the first feedrate plunge move of the recorded patch is equal to the Z coordinate of the start of the original Grid Digitizing operation plus the value of this parameter.

15.3.40 Parameters 49–51 – Small Arc Feedrate Limiting

These parameters are intended to control the federate limits of small-radius arc moves.

Parameter	Function Description	Input format
49	Lower arc radius/federate limiting parameter	rrrr.ffff, where rrrr is a radius threshold setting, and .ffff is a federate adjustment setting
50	Upper arc radius/federate limiting parameter	RRRR.FFFF, where RRRR is a radius threshold setting, and .FFFF is a federate adjustment setting
51	Arc Feedrate Limiting mode	2 digit positive number. 1's digit (right digit) = Lower Limiting mode 10's digit (left digit) = Upper Limiting mode

The 2 digits in Parameter 51 can have the following values and associated meaning:

0 – Turn off Arc feedrate limiting for the associated radii range (default value)

1 – Literal Mode Feedrate limit = FFFF (.FFFF x 10000) for the associated radii range.

2 – Fractional Mode Feedrate limit = .FFFF x Programmed feedrate for the associated radii range

3 – Proportional Mode Feedrate limit = Arc radius x .FFFF x Programmed feerate for the associated radii range.

The Lower Arc Limiting mode's radii range includes any arc radius from 0 up to and including the rrrr setting of Parameter 49. So, if you program a G2 or G3 arc with a radius of rrrr (of Parameter 49) or less, and the Lower Limiting mode in Parameter 51 is turned on, then the arc's federate will be limited according to how ffff (of Parameter 49) is interpreted. If the Control is set up in Millimeter mode, then rrrr specifies the number of millimeters. However, if the Control is set up in Inch mode, then rrrr means the number of inches divided by 100.

The Upper Arc Limiting mode's radii range includes any arc radius larger than the rrrr setting of Parameter 49 up to and including the RRRR setting of Parameter 50. So, if you program a G2 or G3 arc with a radius of RRRR (of Parameter 50) or less, but greater than rrrr (of Parameter 49), and the Upper Limiting mode in Parameter 51 is turned on, then the arc's federate will be limited according to how FFFF (of Parameter 50) is interpreted. If the Control is set up in Millimeter mode, then rrrr specifies the number of millimeters. However, if the Control is set up in Inch mode, then rrrr means the number of inches divided by 100.

Note that if Parameter 51 contains values other than 00, 01, 02, 03, 10, 11, 12, 13, 20, 21, 22, 23, 30, 31, 32, 33, it will be treated as invalid and will cause the Small Arc Feedrate Limiting feature to be disabled.

Also note that this feature has no effect for arc movement handled by Smoothing (P220=1).

Examples: (in millimeters)

Parameter 49 = 5.0050 (rrrr = 5 mm, ffff = 0050)
Parameter 50 = 75.1500 (RRRR = 75 mm, FFFF = 1500)

If Parameter 51 = 31 and you program a G2 arc with a radius of 4 mm at a feedrate of 1000 mm min, then the actual feedrate of the arc will be lowered down to 50 mm/min.

If Parameter 51 = 23 and you program a G3 arc with a radius of 40 mm at a feedrate of 700 mm min, then the actual feedrate of the arc will be lowered down to 105 mm/min (=700 x .1500).

If Parameter 51 = 33 and you program a G3 arc with a radius of 72 mm at a feedrate of 1200 m/min, then the actual feedrate of the arc will remain at 1200 mm/min and will not be modified because it is well within the Feedrate limit of 12 960 mm/min (Arc radius x .FFFF x Programmed feerate = 72 x .1500 x 1200 = 12960).

If Parameter 51 = 11 and you program a G2 arc with a radius of 100 mm at a feedrate of 2500 mm/min, then the actual feedrate of the arc will remain unmodified at 2500 mm/min because the arc radius is outside both ranges specified by Parameters 49 and 50, and therefore this feature does not affect such arcs.

Examples: (in inches)

Parameter 49 = 50.0050 (rrrr = 1/2 inch, ffff = 0010)
Parameter 50 = 200.1500 (RRRR = 2 inches, FFFF = 1500)
If Parameter 51 = 31 and you program a G2 arc with a radius of 1/4 inch at a feedrate of 100 inches/min, then the actual feedrate of the arc will be lowered down to 10 inches per minute.

If Parameter 51 = 23 and you program a G3 arc with a radius of 1.5 inches at a feedrate of 200 inches/min, then the actual feedrate of the arc will be lowered down to 30 inches/min (=200 x .1500).

If Parameter 51 = 33 and you program a G3 arc with a radius of 1.75 inches at a feedrate of 300 inches/min, then the actual feedrate of the arc will be lowered down to 78.75 inches/min (= Arc radius x .FFFF x Programmed feerate = 1.75 x .1500 x 300).

If Parameter 51 = 11 and you program a G2 arc with a radius of 3 inches at a feedrate of 250 inches/min, then the actual feedrate of the arc will remain unmodified at 250 inches/min because the arc radius is outside both ranges specified by Parameters 49 and 50, and therefore this feature does not affect such arcs.

15.3.41 Parameter 56 – Feedrate Override Display Propertiexs / Inverse Time Interpolation

Bits 1 and 2 define how the federate override is displayed in the status window. Bit 3 selects the meaning of inverse time feedrates (F) when inverse time mode (G93) is active. Bit 0 is unused.

Bit	Function Description	Value
0	Not used	
1	Display programmed rate not actual	Add 2
2	Display a bar meter of percentage	Add 4
3	Inverse time meaning of F	F is in inverse seconds (Add 8)

15.3.42 Parameter 57 – Use Generic Load Meter Data from PLC

If load meters are enabled (via Parameter 143) the data being displayed for each axis is normally calculated from its PID output (either real or simulated). However, if any axis has its corresponding bit set in this parameter, then the data going into the load meter will come from the PLC system variables SV_METER_1–SV_METER_8 (corresponding to axes 1–8). Furthermore, if an axis is a spindle axis (as set by Parameter 35) the load meter data will be displayed on the spindle override meter in the Status Window.

Cooperation from the PLC program is required for this feature to work. The PLC program needs to send back a value between 0.0 and 100.0 via floating point system variables SV_METER_1–SV_METER_8 (corresponding to axes 1–8). Note that there is future support for SV_METER_9 up to SV_METER_16, but for now there is support for up to only 8 axes.

15.3.43 Parameters 61–62 – Stall Detection Parameters

The M-Series control will detect and report several stall conditions. The low power stall occurs if the control has been applying a specified minimum current for a specified time, and no encoder motion has been detected. This may indicate a loose or severed encoder cable. A high power stall occurs if the control has been applying at least 90% current for a specified time, and no motion greater than 0.0005" has been detected. This may indicate a physical obstruction. Note that this feature will only work with torque mode drives and not velocity mode drives.

Parameter 61 is the time limit, in seconds, for a high power stall. The default is 0.5 seconds. Parameter 62 is the PID output threshold for a high power stall. The default is 115.

15.3.44 Parameter 63 – High Power Idle PID Multiplier

This parameter holds the value of a constant used for motor high power idle detection when an axis is not moving and no job is running, but there is power going into the motor to maintain its position. The default value is 1.5. This is intended for early detection of an axis if it's stopped against some abnormal resistance or not tuned correctly, such that it will probably overheat later.

15.3.45 Parameter 64 – Fourth/Fifth Axis Pairing

This feature enables the 4th and 5th axes to be paired together or individually be run in a slaved state with any of the other axes. This is intended to drive 2 screws on opposite sides of a table (probably a router table or gantry system). Set this parameter to 0 (default) to indicate that no other axis is paired with the 4th or 5th axis. In order to pair both the 4th and 5th axes on the same system add the 4th axis value with the 5th axis value. Example: 4th axis paired with the X-axis and 5th axis paired with the Z-axis a value of 49 would be entered into parameter 64. The axes are slaved upon power up but it is still possible to move the paired (4th or 5th) axis independently if the axis is labeled.
Note: You cannot run Autotune on paired axes.

Function Description	Value
No Pairing (Default)	0
Pair 4th axis with 1st Axis	1
Pair 4th axis with 2nd Axis	2
Pair 4th axis with 3rd Axis	3
Pair 5th axis with 1st Axis	16
Pair 5th axis with 2nd Axis	32
Pair 5th axis with 3rd Axis	48
Pair 5th axis with 4th Axis	64

15.3.46 Parameters 65–67 – Spindle Gear Ratios

These parameters tell the control the gear ratios for a multi-range spindle drive. Up to four speed ranges are supported; high range is the default. Parameters 65–67 specify the gear ratio for each lower range, relative to high range. For example, if the machine is a mill with a dual range spindle, and the spindle in low range turns 1/10 the speed it turns in high range, then parameter 65 should be set to 0.1. Note that these values can be signed +/-. So, if switching from high range to a lower range causes the spindle encoder to count in the opposite direction, then a negative value can be used to compensate for this behavior.

Parameter 65 is the low range gear ratio.
Parameter 66 is the medium-low range gear ratio.
Parameter 67 is the medium-high range gear ratio.

These parameters work in conjunction with the PLC program, which uses the states of SV_SPINDLE_LOW_RANGE and SV_SPINDLE_MID_RANGE to signal to the CNC software which range is in effect, according to the table below.

	Spindle Range			
System variable	High Range	Medium High Range	Medium Low Range	Low Range
SV_SPINDLE_MID_RANGE	0	1	1	0
SV_SPINDLE_LOW_RANGE	0	0	1	1

15.3.47 Parameter 68 – Minimum Spindle Speed (Rigid Tapping Parameter)

This parameter holds the value that the spindle slows down to from the programmed spindle speed towards the end of the tapping cycle. The lower the value, the more accurately the Z axis will land on target, but at the expense of possibly stalling the spindle motor which in turn will cause Z-axis to fall short. If this value is too large, the off target error will increase. The suggested starting value is 640 RPM.

15.3.48 Parameter 69 – Duration for Minimum Spindle Speed Mode (Rigid Tapping Parameter)

This is the duration of time, in seconds, that the control will stay at minimum spindle speed. If the number is too small, overshoot will occur. If the number is too large, the user waits longer for the hole to be tapped at the slow speed specified by Parameter 68. The suggested starting value is 1.25 seconds.

15.3.49 Parameter 70 – Offset Library Inc/Decrement Amount

Sets the increment and decrement amount used in the offset library.

15.3.50 Parameter 71 – Part Setup Detector Height

If this Parameter is set to a non-zero value, it indicates that the F3 – Auto feature in part setup should be available using the tool detector (TT1) instead of the probe. The value in this parameter is the height of the detector. A value of 0 disables this feature.

When this feature is enabled:
a) Probe detection (Parameter 18) is not checked
b) The tool number and/or edge finder diameter entered by the operator are used; Parameter 12 is ignored.
c) The value from Parameter 71 is added to (or subtracted from, depending on approach direction) the part position.

15.3.51 Parameter 72 – Data M Function Options

The setting of this parameter affects the operation of the data M functions M122, M123, and M124.

Bit	Function Description	Value
0	Suppress output of axis labels by M122/M124?	Add 1
1	Insert commas between positions/values with M122/M124?	Add 2
2	Suppress spaces between positions/values outputted by M122/M123/M124?	Add 4

15.3.52 Parameter 73 – Peck Drill Retract Amount (Canned Cycle Parameter)

This specifies the retract amount used during a G73 peck drilling cycle.

15.3.53 Parameter 74 – M-function executed at bottom of tapping cycle (Canned Cycle Parameter)

This specifies the number of the M-function that is executed at the bottom of the G84 tapping cycle (primarily used for reversing the spindle in preparation for pulling out of the tap hole). This also specifies the number of the M-function that is executed after the G74 countertapping cycle is done (returned to the initial point).

15.3.54 Parameter 75 – Summing Control

This parameter controls the type of position to be summed (local or machine), and which axes are to be summed together and which axis will bear the effects of the the summing. The parameter can contain up to four digits. The sign of the parameter value and the position and value of each digit has special significance as indicated in the tables below:

Sign	Position Type
+	Sum Machine Coordinates
-	Sum Local Coordinates

Parameter Digit Position	Summed Axis
1's Col.	Axis 1
10's Col.	Axis 2
100's Col.	Axis 3
1,000's Col.	Axis 4

Digit Value	Meaning
0	Summing off
1 through 4	Axis to Sum with
5	(Reserved)
6	Disable display
7	Display if moved
8	Display if other moves
9	(Reserved)

The "Summed Axis" is the axis that bears the position sum of itself with the "Axis to Sum with". The DRO display of the "Summed Axis" will show this summed position. The DRO will display both labels when displaying a summed axis. Furthermore, G-codes that command movement on the "Summed Axis" will have their positions offset by the position of the "Axis to Sum with".

NOTICE

It is highly recommended that summing be done with Local Coordinates (using the '-' sign in the parameter value). Summing with Machine Coordinates can cause the effective software travel limits to move, thus resulting in physical overtravel or severely handicapping the amount of available travel, due to the fact that software travel limits are defined in terms of Machine Coordinates. Summing with Local Coordinates avoids this problem.

Here are some examples:

Desired Display	Value
Axis 3 DRO will display the result of axis 3 and 4 machine coordinate positions summed together.	400
Axis 4 DRO will display the result of axis 4 and 3 local coordinate positions summed together. Machine coordinate positions are not affected.	−3000
Axis 3 DRO will display the result of axis 3 and 4 local coordinate positions summed together. Axis 4 DRO display will be disabled.	−6400
Axis 4 DRO will display the result of axis 4 and 3 local coordinate positions summed together. Axis 3 DRO display will be disabled.	−3600
Axis 3 DRO will display the result of axis 3 and 4 local coordinate positions summed together. Axis 4 will be displayed only if it moves.	−7400

15.3.55 Parameter 76 – Manual Input Unrestricted Distance

This parameter is intended to be used with Z-axis summing. It defines the maximum distance from the summed axis start of travel in which manual movements can occur without causing a fault. Use a negative value to specify a distance from the minus travel limit, a positive value for a distance from the plus travel limit.

When used with manual drilling, for example, setting this parameter will allow the operator to keep a hand on the quill at all times and even begin pulling on the quill in anticipation of a programmed stop.

Setting this value to zero will cause a fault if there is any manual movement.

To completely disable manual movement restrictions, set this parameter to a value exceeding the total travel of the summed axis.

Minimum = -99999.9999, maximum = 9999.9999, default = 0, typical = +/- 1.0 inch or +/- 20.0 mm

15.3.56 Parameter 77 – Manual Input Movement Tolerance

This parameter specifies the manual movement tolerance while a job is running. It is intended for use with a quill locking mechanism. It allows the lock to distort and/or slip a small amount when under stress. If the quill moves more than the given tolerance, the job will stop with a fault. A typical setting for Parameter 77 is 0.005 inches.

15.3.57 Parameter 78 – Spindle Speed Display and Operations

Bit 0 (value = 1) specifies how the spindle speed is determined and displayed in the CNC software status window. When turned on (value = 1), the spindle speed is determined by reading the encoder feedback from the axis specified according to Parameter 35. Which has the number of encoder counts/revolution specified in Parameter 34. When turned off (value = 0), the displayed speed is not measured; the speed is calculated based upon the programmed speed, spindle override adjustment, and gear range.

Bit 1 (value = 2) allows the control to prorate the programmed feedrate to be proportional to the spindle speed if the spindle speed ever slows down below the spindle speed threshold percent as set by parameter 149.

Bit 2 (value = 4) will turn on the “Spindle up-to-speed” function. The active modal spindle speed S at the point where the most recent M3 or M4 is invoked sets the target spindle speed for this function. This function is invoked on the first feed-per-minute move (such as G1/G2 G3) following the aforementioned M3 or M4. If the actual measured spindle speed at this moment is below the spindle speed threshold percent (Parameter 149) of target spindle speed, this function will pause the job at this point until the spindle speed gets up to at least this spindle speed threshold percent level.

Bit 3 (Value = 8) Will display the spindle speed for a 2nd spindle encoder. A 2nd spindle and the 2nd spindle encoder settings must first be configured using parameters 459, 460, 461, 462.

Bit	Function Description	Value
0	Display actual spindle speed	Add 1
1	Slave feedrate to spindle speed	Add 2
2	Wait for spindle to get up to speed on feed-perminute moves	Add 4
3	Display Actual Spindle Speed from 2nd Spindle Mounted Encoder	Add 8

15.3.58 Parameter 79 – Paired Axes Re-sync Delay Time (milliseconds)

This parameter specifies the length of time (in milliseconds) that the control will wait before re-syncing paired axes. If this parameter is a negative value, the control will prompt the operator to press **Cycle Start** before re-syncing.

15.3.59 Parameter 80 – Voltage Brake Message Frequency

This parameter specifies the number of times the "450 Voltage brake applied" message has to occur before we show it in the message window and message log. A value of 0 or 1 will display the message for every instance that it occurs.

15.3.60 Parameter 81 – Air Drill M-function (Canned Cycle Parameter)

P81 (when not equal to -1.0) specifies the M-function to be called in place of Z-axis movement during a G81 drilling cycle.

15.3.61 Parameter 82 – Spindle Drift Adjustment (Rigid Tapping Parameter)

This value is the number of degrees that the spindle will take to coast to a stop, when the spindle is turned off at the minimum spindle speed specified by Parameter 68.

15.3.62 Parameter 83 – Deep Hole clearance amount (Canned Cycle Parameter)

Parameter 83 specifies the clearance amount used during a G83 deep hole drilling cycle.

15.3.63 Parameter 84 – M function executed at a return to initial point of tapping cycle (Canned Cycle Parameter)

This specifies the number of the M-function that is executed after the G84 tapping cycle is done (returned to the initial point). This also specifies the number of the M-function that is executed at the bottom of the G74 countertapping cycle (to reverse the spindle in preparation for pulling out of the countertap hole).

15.3.64 Parameter 85 – "Door Open" Interlock PLC Bit

This parameter provides a way for a system integrator to implement a safety interlock that limits rate of movement when the doors are open. This parameter specifies the PLC bit number and PLC bit polarity that indicates the "door open" condition. If the specified PLC bit is in the specified "door open" condition, then all normal movement commands will be limited to the slow jog rate (as specified in the Jog Parameters menu in Machine Configuration). Polarity of the "door open" condition is specified thuswise: a positive number indicates that the "door open" condition occurs when the specified PLC bit is On, and a negative number indicates that the "door open" condition occurs when the specified PLC bit is Off. If this parameter is set to 0 (the default value), then this feature is disabled, and no checking for a "door open" condition is done. Note that this feature has no effect for movement commands handled by Smoothing (P220=1).

Note: PLC program interaction is needed for this feature to work.

15.3.65 Parameter 86 – Rapid/Linear vector rate limit

This parameter controls the feature that imposes a limit on the number of rapid and/or linear moves per second to the value specified in this parameter. If the value of this parameter is more than 0, Rapid and/or Linear moves will be combined to prevent the aforementioned limit from being exceeded. This parameter is used for testing purposes and should be set to 0 to disable this feature. Note that this feature has no effect for movement commands handled by Smoothing (P220=1).

15.3.66 Parameters 87–90 (and also 252–255) – Autotune Ka Performance parameters

These parameters are used by autotune. Increasing the value will increase the Ka used by autotune which when used will increase the PID used during acceleration. The default value is 0. The maximum value is 50 and the minimum value is 0. Values for axes 1–4 are specified in parameters 87–90. Values for axes 5–8 are specified in parameters 252–255.

15.3.67 Parameters 91–94 (and also 166–169) – Axis Properties

These parameters may be used to set various axis properties. Properties for axes 1–4 are specified in parameters 91–94. Properties for axes 5–8 are specified in parameters 166–169.

Bit	Function Description	Value
0	Rotary/Linear Axis Selection Rotary	Axis = 1, Linear Axis = 0
1	Rotary Display Mode	Wrap Around = 2, Show Rotations = 0
2	Suppress direction check when doing Tool Check?	Don't Check = 4, Check = 0
3	Suppress park function?	Don't Park = 8, Park = 0
4	NOT USED ON MILL	Recommended bit value is 0
5	Linear Display of Rotary Axis	Linear Display = 32, Default Rotary = 0
6	4th Axis works like Z axis	Add 64
7	NOT USED ON MILL	Recommended bit value is 0
8	Axis is triangular rotary?	Add 256
9	Hide axis display from DRO	Add 512
10	Triangular rotary axis type	Articulated Head = 1024, Tilt Table = 0
11	Rotation Center is parallel to X?	Add 2048
12	NOT USED	Recommended bit value is 0
13	NOT USED	Recommended bit value is 0
14	Enable TWCS for Articulated Head machines	Add 16 384

Bit 0: Turning this bit on will cause the DRO display for the affected axis to be displayed in degrees. Also this information is used by Intercon to make rotary axis support available (by setting Parameter 94 to 1, indicating that the fourth axis is rotary). This bit is also used when performing inch/mm conversions: values for a rotary axis will not be converted since they are assumed to be in degrees regardless of the system of linear units.

Bit 1: This bit has no effect unless Bit 0 (mentioned above) is turned on. When this bit is turned on, a "Wrap Around" display is shown on the DRO. A "Wrap Around" Rotary Display is a display in degrees without the number of rotations shown. If this bit is turned off, the number of rotations away from 0 degrees will be shown alongside the degree display.

Bit 2: This bit will only affect the Z-axis. It controls whether or not a direction check will be performed when the Tool Check button is pressed. If this bit is turned on, direction checking is turned off, and thus, there is a possibility for the Z-axis to move downward unexpectedly, depending on the Z value of Return Point #1 (G28). Therefore, it is best in most cases to leave this bit turned off to allow direction checking to be turned on (value = 0).

Bit 3: Setting this bit prevents **F1** (Park) in the Shutdown menu from parking this axis.

Bit 5: This setting overrides only the DRO display options for an axis that has bit 0 set (including the Rotary Display Mode – bit 1) so that the display does not reflect a degree symbol or any indication of the number of rotations, but appears as a linear axis.

Bit 6: This bit only works for Parameter 94 (4th axis). Setting this bit will cause the 4th axis to respond to Z-axis only commands just like the Z-axis, for example issuing an M25 with this bit set will cause the Z and 4th axes to go the home (G28) position.

Bit 8: This setting is used in conjunction with bit 10. It only works for Parameter 166 (5th axis). Setting this bit on will identify this axis as a Triagular Rotary, which may either be an Articulated Head axis, or a Tilt Table axis.

Bit 9: This setting will hide the affected axis from the DRO display. Note that this does not prevent such an axis from being commanded to move.

Bit 10: In order for this setting to work, bit 8 must be turned on. This setting has meaning only for Parameter 166 (5th axis). Setting this bit on will identify this axis as the controller of the angle of articulation on an Articulated Head machine. If this bit is not set, then this axis will be identified as the controller of the tilt angle on a Tilt Table machine.

Bit 11: In order for this setting to work, bit 0 must be turned on (i.e. axis is rotary). Also, this setting only affects axes 4 and higher. This setting indicates that this rotary axis is mounted in such a way so that its rotation center line is parallel to the X axis (axis 1). The rotation center line's machine coordinate location is then defined by parameters 116 and 117. This setting enables backplot to display rotary movements encircling the X axis and also gives the needed machine geometry information to G93.1 for it to properly calculate the tool tip feedrate in 3D space in such a configuration.

Bit 14: This bit (in conjunction with the CSR unlock) enables the TWCS (Transformed WCS) feature for Articulated Head machines. When this feature is enabled, each individual WCS can be independently set up as either transformed or non-transformed using the TWCS=Yes/No setting in the WCS configuration screen (See TWCS in Chapter 4). However, enabling this feature bit will always cause automatic B/5th axis transformation on certain features regardless of the TWCS=Yes/No setting for each WCS. Such features are: Tool Check, M25, Digitizing, and the movements done in the Probing Cycles.

15.3.68 Parameters 95–98 (and also 156–159) – Autotune Move Distance / Auto Delay Calculation Move Distance

These parameters hold the maximum distance that the control will move each axis in either direction from the starting point when either Autotune or Auto Delay Calculation is run. The default value for these parameters is 2.0 inches. Values for axes 1–4 are specified in parameters 95–98. Values for axes 5–8 are specified in parameters 156–159.

15.3.69 Parameter 99 – Cutter Compensation Look-ahead

This parameter sets the default number of line or arc events for the G-code interpreter to scan ahead when Cutter Compensation (G41 or G42) is active. Values of 1 to 99 are allowed for this parameter and default is 1.

15.3.70 Parameters 100–109 – Intercon parameters

15.3.71 Parameters 111–116 – Intercon parameters

See Parameters 100–109 for more information.

These parameters are some of the Intercon setup parameters. See Chapter 10 for more information about these parameters. Changing values will change Intercon settings and may affect the output of the G-code program if it is reposted.

15.3.72 Parameter 116 – A-Axis Y Coordinate

This parameter is used in conjunction with Dig to CAD to export digitized data for use with CAD/CAM software (see Chapter 8). It is used to define the Y coordinate of the center of rotation for a rotary axis that rotates about the X-axis.

15.3.73 Parameter 117 – A-Axis Z Coordinate

This parameter is used in conjunction with Dig to CAD to export digitized data for use with CAD/CAM software (see Chapter 8). It is used to define the Z coordinate of the center of rotation for a rotary axis that rotates about the X-axis.

15.3.74 Parameter 118 – B-Axis X Coordinate

This parameter is used in conjunction with Dig to CAD to export digitized data for use with CAD/CAM software (see Chapter 8). It is used to define the X coordinate of the center of rotation for a rotary axis that rotates about the Y-axis.

15.3.75 Parameter 119 – B-Axis Z Coordinate

This parameter is used in conjunction with Dig to CAD to export digitized data for use with CAD/CAM software (see Chapter 8). It is used to define the Z coordinate of the center of rotation for a rotary axis that rotates about the Y-axis.

15.3.76 Parameter 120 – Probe Stuck Clearance Amount

This parameter specifies the distance that digitizing or probing functions will move to try to clear a stuck probe condition. A stuck probe condition exists when the probe detects a point and then moves away but the probe input has not changed. It is recommended that this parameter should not be changed from its default value without consulting a qualified technician.

15.3.77 Parameter 121 – Grid digitize prediction minimum Z pullback

This parameter specifies the minimum distance the Z-axis will move upward when pulling back from a surface. The digitizing function attempts to predict the slope of a part surface because time is saved when the Z-axis does not have to travel upward to the starting Z depth for every digitized point. When probe contact is made traversing in the XY plane, this parameter specifies the minimum distance the Z-axis moves upward before attempting another XY plane move. Smaller values are better when the surface being digitized has smooth curves. Larger values are better for surfaces that have steep walls. It is recommended that this parameter should not be changed from its default value without consulting a qualified technician.

15.3.78 Parameter 122 – Grid digitizing deadband move distance

This parameter specifies a deadband distance used for internal calculations when doing a clearance move. It is recommended that this parameter should not be changed from its default value.

15.3.79 Parameter 123 – Radial Clearance Move

This parameter only applies to radial digitizing and determines what type of positioning move the digitizing probe will make should it encounter an unexpected probe contact with the surface of the part during Radial Digitizing. Unexpected probe contact is defined as probe contact occurring while the probe is traversing towards the user defined center point.

With Parameter 123 set to 0: When the probe encounters an unexpected probe contact, the digitizing program stops data collection. The control then prompts the operator to jog the probe to a clear position. This can be any place inside the digitizing radius and above the part, that the probe stylus has a clear path to the defined center position. To restart data collection press Cycle Start. The probe moves in the XY plane from the position the operator placed it at, to the center position defined in the radial setup menu. After reaching the center position, the probe will feed down to the Zaxis position it was at when the data collection was interrupted. The digitizing run will resume with the probe approaching from the defined center position.

With Parameter 123 set to 1: When the probe encounters an unexpected probe contact, it will automatically move (with probe detection turned off) to the maximum Z height, then moves the X and Y-axis to the defined center position. The probe will then move to the Z position it was at when the unexpected contact occurred. It will then move from the defined center position, towards the measurement position it was trying to approach when the unexpected probe contact occurred and continue digitizing.

With Parameter 123 set to 2: When the probe encounters an unexpected probe contact, it will automatically move back to the defined center position (with probe detection turned off), at its present Z height. It will then move from the defined center position, towards the measurement position it was trying to approach when the unexpected probe contact occurred and continue digitizing.

NOTICE

Settings 1 and 2 should only be used with extreme caution because probe detection during some positioning moves is turned off, and damage to the probe or work piece could occur!

15.3.80 Parameter 128 – Dry Run PLC Bit

When non-zero, this parameter specifies the PLC bit which is checked when starting a job to determine if Dry Run mode is in effect. When in Dry Run mode, all non-thread/tap moves are converted to feed/minute at the Dry Run Feedrate (Parameter 129). When in Dry Run mode, the Feedrate word in the status window is drawn in reverse video (blue letters on a yellow background).

15.3.81 Parameter 129 – Dry Run Feedrate

Specifies the feedrate to use for Dry Run mode.

15.3.82 Parameter 130, 131 – 3^{rd}/ 4^{th} axis on/off selection

These parameters control the display of the 3^{rd} and 4^{th} axes, respectively. The tens digit of the parameter value specifies the label of the affected axis when it is enabled, with values 1–9 corresponding to axis labels ABCUVWXYZ. The ones digit specifies the label of the axis when it is disabled, with 0.0 meaning the axis is not switchable, 1.0 meaning it turns off (N), a 2.0 meaning manual (M), and a 3.0 meaning 2-axis with manual Z (@). Parameter 130 also supports additional modes depending upon the value of the hundreds digit. See the chart below for valid values for the Hundred's digit of Parameter 130. Note that Parameter 131 does not support the Hundred's digit. When P130/P131 is configured for axis switching, the Setup menu displays function keys F5/F6 to switch the axes.

Example 1: A value of 192 in parameter 130 will toggle the 3^{rd} axis between Z and M and power off all axes. The 1 sets bit one to power all axes off, the 9 enables the 3^{rd} axis to "Z", and the 2 changes the axis label to "M" once toggled with the F5 key in the Setup menu.

Example 2: A value of 392 in parameter 130 will toggle the 3^{rd} axis label between Z and M and power off all axes and receive its positions from the 4^{th} axis encoder input. The 3 sets bits two and one to power off all axes and use the 4^{th} encoder input as a scale input, the 9 enables the 3^{rd} axis to "Z", and the 2 changes the axis label to "M" once toggled with the F5 key in the Setup menu.

Example 3: A value of 61 in parameter 131 will toggle the 4^{th} axis between W and N. The 6 enables the 4^{th} axis to "W" and the 2 changes the axis label to "N" when toggled with the F6 key in the Setup menu. This will turn the 4^{th} axis on and off.

Hundred's Digit (Parameter 130 only) (X00)	Function Description	Value of the Hundred's Digit
Bit1	Axis motor power when switching to two-axis mode.	1 = power all axes off, 0 = power 3rd off only
Bit2	Use 4th encoder input for scale input	2 = Use 4th encoder input, 0 = no manual input
Bit3	Use 5th encoder input for scale input	4 = Use 5th encoder input, 0 = no manual input

Enabled Axis:	**A**	**B**	**C**	**U**	**V**	**W**	**X**	**Y**	**Z**
Ten's Digit	1	2	3	4	5	6	7	8	9

Disabled Axis:	**N**	**M**	**@**
One's Digit	1	2	3

15.3.83 Parameters 132–135 – Motor Heating Coefficients for axes 5–8

See parameters 20–30 for more information.

15.3.84 Parameter 136 – G76 Fine Bore Retract Angle

0–360 Degrees. A setting of 0 = Retract in Y+ direction.

15.3.85 Parameter 137 – Load Meter Filter Size

This controls the number of samples used in calculating an average output for the load meter display, as a way of smoothing it out.

15.3.86 Parameter 138 – DRO Encoder Deadband

This controls the deadband amount (in encoder counts) used by the anti-flicker DRO filter.

15.3.87 Parameter 139 – Special Dwell between Moves

This parameter turns on and specifies the amount dwell time between moves, with exceptions in special cases. This parameter is similar to specifying a G61 (Modal Decel and Stop) at the beginning of a CNC program, except that dwelling will not happen under certain conditions: (1) No dwell will happen between 2 arcs. (2) No dwell will happen if before the move that contains Z movement.

15.3.88 Parameter 140 – Message log priority level

This parameter controls the messages that are written to the message log, which can be accessed through the F9 – Logs function in the Utilities menu. See Chapter 16 for the list of numbered messages. Message logging can be disabled be setting this parameter to -1. The recommended log level is 4.

Which numbered messages are logged	Value
None	-1
Numbered messages 0–299 and 400–499 – The most serious faults.	1
Numbered messages 0–299 and 400 and higher – The most serious faults and medium severity errors.	4
All numbered messages	9

15.3.89 Parameter 141 – Maximum message log lines

This parameter is the number of lines that will be kept in the message log. If this parameter is set to 10,000, for example, the newest 10,000 messages will be retained. The CNC software will delete the oldest messages, trimming the log file to the given number of lines at startup and periodically while the CNC software is in an idle state. Parameter 142 controls the frequency of the log cleanup.

15.3.90 Parameter 142 – Message log trim amount

This parameter is the number of additional lines above the minimum that can be added to the log before it is reduced to the minimum size. Setting this parameter to a lower value will cause the log file to be trimmed to its minimum size more often. The higher the value, the less often the log will be trimmed. The speed of the disk drive and total size of the log file at the time it is trimmed will determine how long the log cleanup takes. Under most circumstances, using 10,000 and 1,000 for parameters 141 and 142 respectively will provide a reasonable and useful log size with no noticeable effects on performance. If parameters 141 and 142 are set to excessively high values, the message “Trimming excess lines from log file” will be presented. This message will appear at startup and very infrequently when the CNC software is idle. Normal operation can proceed after the message disappears. If the delay is unacceptable, reduce the values of Parameters 141 and 142.

15.3.91 Parameter 143 – DRO Properties (load meters, 4/5 digits, Distance To Go)

This parameter controls the display of the axis load meters and 4/5 digit DRO precision.

DRO Precision is being transitioned to being controlled by Parameter 263 DRO Display Precision. Parameter 263 allows 1–7 digits of DRO display and in future releases will be the only place to control DRO Precision display.

The default value for this parameter is 75 (Load Meters, Load Meter Outline, Mini DRO, and Mini Machine Coordinates all turned on).

Bit	Function Description	Value
0	Enable Load Meters	Enable (Add 1)
1	Load Meter Outline	Enable (Add 2)
3	Mini DRO (Distance to Go)	Enable (Add 8)
6	Mini Machine Coordinates	Enable (Add 64)

15.3.92 Parameter 144 – Comparison Rounding

This parameter determines the built in rounding for the comparison operators (’EQ’, ’NE’, ’LT’, ’GT’, etc.) in expressions. Rounding of comparison arguments is necessary due to extremely small errors that are part of every floating-point calculation. The result of such errors is that two floating-point values are rarely exactly equal. The value of parameter 144 represents the precision of comparison in places after the decimal point. If the parameter is set to 9.0, for example, then comparison operators will declare two numbers that differ in value by less than 0.0000000005 as being equal. The value 0.0 is a special value that turns comparison rounding off. When comparison rounding is off, it is up to the G code programmer to build the precision into conditional statements, for example `IF ABS[#A - #B] LT 0.00005 THEN GOTO 100`. When comparison rounding is off, the "EQ" usually returns "false". If parameter 144 is set to 9, the programmer can shorten the previous example to `IF #A EQ #B THEN GOTO 100`.

15.3.93 Parameter 145 – Advanced Macro Properties (Fast Branching)

This parameter turns fast branching on (1) and off (0). The other bits of this parameter are reserved for future use. If fast branching is disabled, the CNC software searches forward in the program for the first matching block number and resumes searching, if necessary, from the top of the program. For this reason, backward branches take longer than forward branches and backward branch times depend on the total program size. If the program is significantly large, use of the GOTO statement could introduce temporary pauses.

When fast branching is enabled, the CNC software remembers the locations of block numbers as it finds them during program execution. Backward branches always take place immediately. The first forward branch to a block not yet encountered will take additional time as the CNC software searches forward for the block number; however, subsequent forward branches to that block number will take place immediately. The trade-off for using fast branching is that all line numbers at a given level of program or subprogram must be unique and programs will use more memory (approximately 16kilobytes of memory for every 1000 block numbers in the program.)

15.3.94 Parameter 146 – Feed Hold Threshold for Feed Rate Override

This parameter sets the lowest value permitted as the feed rate override percentage before feed hold is engaged. Feed hold will be released when the override percentage is greater than this value.

15.3.95 Parameter 147 – Number of Status Messages to keep in Operator Message Window

The Operator Message Window is the box of scrolling status messages that appears in the upper right corner of the Main Screen. The number of remembered status messages can be adjusted by this parameter.

15.3.96 Parameter 148 – Miscellaneous Jogging Options

This parameter enables and/or disables certain optional modes of jogging.

Bit	Function Description	Value
0	Unused	Should be set to 0
1	Prohibit Keyboard Jogging	Prohibit Keyboard Jogging (Add 2)
2	Enable Keyboard Jogging at Startup	Keyboard Jogging at Startup (Add 4)

Note: With this parameter set to zero, you need to set Parameter 170 to enable keyboard jogging.

15.3.97 Parameter 149 – Spindle Speed Threshold

This parameter defines the spindle speed threshold percent for the "Slave feedrate to spindle speed" function and the "Spindle up-to speed" function, both of which are enabled and disabled via Parameter 78. It is specified as a percentage of the programmed spindle speed. For example a value of 0.8 means 80 percent of the programmed spindle speed. See Parameter 78 for more details.

15.3.98 Parameter 150 – Backplot Graphics display options

This parameter controls the various options related to backplot graphics.

Bit	Function Description	Value
0	Sets Run Time Graphics option default to ON	Enable (Add 1)
1	Displays CSR positions in graphing	Enable (Add 2)
2	Display A and B rotations for 5 axis machines	Disable (Add 4)
3	Display Skew Correction	Enable (Add 8)
4	Display Lash/Screw Compensation	Enable (Add 16)

15.3.99 Parameter 151 – Repeatability tolerance for probing and radial digitizing.

Default is 0, repeatability check disabled. When disabled, only one measurement per point is taken in the probing cycles and radial digitizing. When enabled, a minimum of two measurements are taken per point and the difference (if any) is then compared to the repeatability tolerance as set in Parameter 151. If the difference is less than or equal to parameter 151 the point is stored and probing continues. If the difference is greater than parameter 151, 2 more measurements are taken and the process repeated up to a maximum of 10 times. In probing cycles, if the repeatability tolerance cannot be met the cycle is cancelled and an error message generated. In radial digitizing, the point is discarded and digitizing continues without interruption.

15.3.100 Parameter 153 – Probe Protection

A value of 1.0 will enable probe detection. Probe protection will cause motion to stop if the probe is tripped while moving under the following G-codes: G0, G1, G2, G3, G28, G29, G30, G53, G73, G76, G81, G82, G83, G85, G89, G173, G176, G181, G182, G183, G185, and G189. Probe protection is also in effect when using these M-codes:M25, M91, M92, M105, M106, and M128.

15.3.101 Parameter 154 – Touchscreen Options

If = 1 a half-width ESC key is displayed in most menus so there is a means of exiting the menu without using the keyboard. Most of the menus have been reworked to more fully support touch.

15.3.102 Parameter 155 – Probe Type

This parameter specifies the type of probe being used.

Function Description	Value
Standard Mechanical probe	0
DSP probe	1
DP-7 probe	2

15.3.103 Parameters 156–159 – Autotune Move Distance / Auto Delay Calculation Move Distance for axes 5–8

See parameters 95–98 for more information.

15.3.104 Parameter 160 – Enhanced ATC

This parameter controls enhanced automatic tool changer (ATC) options. A value of 1 indicates a nonrandom type of ATC (carousel ATC) and a value of 2 indicates a random type ATC. A value of 0 disables enhanced ATC features. A warning is displayed when attempting to enable enhanced ATC features as these features work in conjunction with specific PLC programs. The enhanced ATC option has the following characteristics:

The beginning of an M6, whether it be a customized mfunc6.mac routine or not, flags the job file, setting the ATC error flag field to 1.

The end of an M6, whether customized or not, performs the following:
(a) The ATC error flag is set to zero.
(b) The tool number displayed on the screen is updated and this value is saved in the cncm.job file.
(c) The tool library bin fields are updated in this manner:

If there was a valid tool in the spindle at the start of the M6, then the tool library bin field for this tool will be updated with either the "putback" field for that tool (if nonrandom type) or the current ATC carousel position (for random type). For both random and nonrandom types, the "putback" field is set to 0. The "putback" field is an internal field for each tool in the tool library. It can be displayed by using the cnccfgutil utility with the -dt option to display the tool library.

For nonrandom types, the new tool now in the spindle will have its "putback" field updated to the current ATC carousel position.

For both random and nonrandom types, the new tool now in the spindle has the bin field set to 0.

The current ATC carousel position is constantly monitored. When there is a change, the ATC bin field in the cncm.job file is updated and the file is saved. The ATC carousel position is read from the PLC system variable SV_ATC_CAROUSEL_POSITION, which should be written by the PLC program. At the start of running a job, to include MDI mode, the ATC error field is checked. If this field is 1, then a warning message is displayed with a prompt to either clear the fault by entering a 'Y' or canceling the job by pressing some other key.

A tool change is not performed if the requested tool is already in the spindle.

An M107 command sends the bin number for the specified tool number, not the tool number.

For random types, tool changes in Intercon are posted as a tool change (Tnn M6) followed by a pre-fetch command for the next tool in the program (Tn2 M107). This allows the PLC program to rotate the tool carousel to the next tool while a job continues with the current tool.

For random types, a job search for a tool number will look for lines of the form Tnn M6, i.e., the search bypasses lines of the form Tnn M107, which are just pre-fetch commands.

The tool library allows editing of the bin fields to specify which carousel bin number the tools are stored in.

15.3.105 Parameter 161 – ATC Maximum Tool Bins

This parameter sets the number of tool changer bins (carousel positions) used with the enhanced ATC option described above. PLC programs are responsible for reading this value. The tool library interface uses this parameter to validate bin fields and perform initialization of the bin fields.

15.3.106 Parameter 162 – Intercon M6 Initial M-Code

This parameter affects how Intercon programs post M& G-codes for a tool change operation. When set to a non-zero value, Intercon will post out an M-code at the start of a tool change and an M5 command after the Txx M6 command. This parameter should be set to 17 so that Intercon will post an M17 code at the start of a tool change. The M17 command turns off spindle and coolant and starts the spindle orientation process.

15.3.107 Parameter 163 – Gang Tooling

This parameter enables the tool library to select front mount or back mount tool approach for gang tooling. If set to 1 you can measure both front mount and back mount tooling.

15.3.108 Parameter 164 – ATC Feature

This parameter controls specific features of the ATC system. It should be set to 1.0 to enable the ATC Reset feature, which will appear as the F6 – ATC Reset function key in the tool library menu. This parameter only works with ATC3 plc programs.

15.3.109 Parameter 165 – Acceleration/Deceleration Options

This is a bit field parameter which modifies certain details of axis acceleration and deceleration when an axis stops moving, changes direction, or starts moving. The Jog Parameters screen in the Machine Configuration set the original DeadStart values for each axis. This parameter allows you to modify these DeadStart settings under certain conditions. Note that if both Bits 0 and 1 are turned on (value = 1+2 = 3), the effect is cumulative, i.e. the net effect will be that 1/2 DeadStart value will be used when a slave axis stops or starts up from a stop. Likewise, if both Bits 2 and 3 are turned on, the effect will be cumulative also. Note that this feature has no effect for movement commands handled by Smoothing (P220=1).

Bit	Function Description	Value
0	Use 1/4 DeadStart value for a slave axis that stops or starts from a stop	Enable (Add 1)
1	Use 2 x DeadStart value for a slave axis that stops or starts from a stop	Enable (Add 2)
2	Use 1/4 DeadStart value for a slave axis that reverses	Enable (Add 4)
3	Use 2 x DeadStart value for a slave axis that reverses	Enable (Add 8)
4	Limit the feedrate along the path of G2 or G3 arc moves such that the feedrate will be uniformly limited to the lesser of the maximum rate of the 2 axes involved in the circular motion.	Enable (Add 16)

15.3.110 Parameters 166–169 – Axis Properties for axes 5–8

See parameters 91–94 for more information.

15.3.111 Parameters 170–179 – PLC Parameters

These parameters are especially reserved as a space for data which is to be sent to the PLC. Parameters 177, 178, 179 have been standardized for specific applications. Parameter 177 is used for trouble shooting purposes only.

15.3.112 Parameter 170 – Enable Keyboard Jogging and set Feedrate over ride Control

This PLC parameter is used to enable keyboard jogging and determine whether jog panel or keyboard feedrate over ride is used. To enable keyboard jogging set Parameter 148 to zero and this parameter to a 1.

Bit	Function Description	Value
0	Enables Keyboard jogging	Enable (Add 1)
1	Only looks at Feedrate over ride from Jog panel	Enable (Add 2)
2	Only looks at Feedrate over ride from keyboard	Enable (Add 4)

15.3.113 Parameter 178 – PLC I/O configuration (Only for Legacy Controls)

This parameter can be use to set switch types from NC to NO and some other options. Each Bit corresponds to a different function. All values are to be added to the current setting. For example, if you need to switch the low lube input to normally open add 1 to this parameter.

Note: This parameter works only with specific PLC programs. The PLC program installed in the control MAY NOT be mapped as indicated below. These parameters should only be changed by a qualified technician. The example given below is intended for reference only:

Note: This parameter only works for legacy PLC programs.

Bit	Function	Default state	Opposite State
0	Lube Fault	Closed = OK	Add 1
1	Spindle Fault	Closed = Fault	Add 2

15.3.114 Parameter 179 – Lube Pump Operation

This parameter can be configured to control a variety of lube pumps. The value is formatted as MMMSS, MMM for minutes and SS for seconds. Below is a table of some examples.

Type of Pump	MMM	SS	Operation
Mechanical/CAM	0	0	179=0 Power is on when machine is running a job or in MDI Mode
Electronic “lube first”	16	00	179=1600 Holds power on to the pump for 16 minutes of job or MDI time
Electronic “lube last”	16	00	179=1600 Holds power on to the pump for 16 minutes of job or MDI time
Direct Controlled Pump	30	15	179=3015 Waits for 30 min of job or MDI time, then applies power for 15 seconds.

15.3.115 Parameter 180 – Clear Home Switch Distance

This parameter specifies the distance to move before generating an error when looking for the home switch to be cleared during a homing (M91/M92) command. A value of zero will result in a default setting of using a distance of 0.5 inches or one motor revolution, whichever is greater. This distance only applies to a linear axis. If the axis is rotary, a distance of 45 degrees is used regardless of the parameter value.

15.3.116 Parameter 186 – Probe Stuck retry disable

This parameter is used to disable retries when a probe is detected to be in a "stuck" condition. A probe "stuck" condition occurs during a probing move when a probe's red light (LED) stays on even after the probe has moved clear of the contact surface. The control can sometimes detect this condition and go through a series of corrective moves to "reseat" the probe and retry the probing move. If this parameter is set to a non-zero value, then the control will not do this corrective action nor attempt another probing move. If this parameter is set to 0, then the control will go through the corrective action and retry the probing move up to 5 times.

15.3.117 Parameter 187 – Hard Stop Homing

This parameter is used when homing off hard stops. The value set in this parameter determines the amount of current sent to the motor while homing. Value range is 0–32000; typical value for a DC system is 16000. Note that this feature does not work with velocity mode drives.

15.3.118 Parameters 188–199 – Aux Key Functions

These parameters are used to assign a function to aux keys 1–12 (i.e. P188 = Au$\times$1 ... P199 = Au$\times$12). The following is the list of possible functions that can be executed when an aux key is pressed.

Function	Value
No Function	0
Input X Axis Position	1
Input Y Axis Position	2
Input Z Axis Position	3
Set Absolute Zero	4
Set Incremental Zero	5
One Shot – Drill	6
One Shot – Circular Pocket	7
One Shot – Rectangular Pocket	8

Function	Value
One Shot – Frame	9
One Shot – Face	10
Execute M Code file	m11*
Free Axes	14
Go to Power Feed Menu	15
XYZ Set Absolute Zero	16
One Shot – Drill Bolt Hole Circle	17
One Shot – Drill Array	18

For example, if you wanted Aux4 to call up the "One Shot – Circular Pocket", you would set parameter 191 to 7.

The Input Axis Position functions must be used with the Set ABS/INC Zero functions. After entering the desired value at the input field provided by the Input Axis Position function, press an aux key assigned either the function Set ABS Zero or Set INC Zero.

* *m* is the number of the M code to execute. For example, if the parameter value is set to 7211, the file mfunc72.mac will be loaded and executed when the Aux key was pressed. Custom overlays with the keys that represent these functions are available; contact your dealer for pricing.

15.3.119 Parameters 200–207 – OPTIC 4 Tach Volts Per 1000 RPM

These parameters control the digital Tach output on the Optic4 boards. They are used on drives like old Fanuc velocity mode drives that require a tach input. The value put here is the volts/1000 RPM off of the motor. A negative value can be entered to invert the tach voltage compared to the encoder count derived velocity direction from the encoder.

15.3.120 Parameters 208–215 – MPU-based Lash/Screw Compensation Acceleration Coefficient

These parameters control the speed of the Lash and/or Screw Compensation for axes 1–8. The lash will be taken up with acceleration equal to the coefficient multiplied by the acceleration rate for the axis. A value of zero would effectively disable MPU-based Lash and/or MPU-based Screw Compensation.

Note: These coefficients are not used by PC-Based Lash nor PC-based Screw Compensation.

15.3.121 Parameter 216 – PC Based Lash Compensation on/off

This parameter controls which Lash Compensation Algorithm to use. The default value of 0 is recommended because it allows lash compensation to occur during any kind of motion. If PC Based lash is used then only during an MDI or programmed move (but not during jogging) will lash compensation be applied.

Function	Value
Use MPU-Based Lash Compensation	0
Use PC-Based Lash Compensation	1

Note: Lash/Screw Compensation Acceleration Coefficients (parameters 208–215) are not used by PC-Based Lash Compensation.

15.3.122 Parameter 217 – PC Based Screw Compensation on/off

This parameter controls which Screw Compensation Algorithm to use. The default value of 0 is recommended because it allows screw compensation to occur during any kind of motion.

Function	Value
Use MPU-Based Screw Compensation	0
Use PC-Based Screw Compensation	1

15.3.123 Parameter 218 – USB MPG Options

A non-zero value specifies that a Wireless USB MPG is being used. When set, the CNC12 software will load the driver (MpgClient.exe) used to communicate with the Wireless USB MPG. The PLC program may also use this parameter to determine the type of MPG connected to the system. A restart of CNC software is required for changes to this parameter to take effect. Typical values are three (P18=3) for a lathe system, seven (P218 = 7) for a three axis system, and fifteen (P218=15) for a four axis system.

Notes:

1. also Set P348 = 15 to tell CNC12 to look for an MPG.
2. Wireless USB MPG support is a software unlock. It requires a Pro or above license.

15.3.124 Parameter 219 – Virtual Control Panel Options

When set to 1.0, CNC software will launch the Virtual Control Panel (VCP) at start up. The VCP is an on-screen equivalent of an actual jog panel that allows the use of mouse clicks (or touches on a touch sensitive screen) to control the same things as the real jog panel. Use of the VCP requires support from the PLC program.

15.3.125 Parameters 220–231 – Smoothing Parameters

These parameters are used for controlling the behavior of the Smoothing feature used during feed per minute moves. In particular, parameter 220 turns Smoothing on or off. When Smoothing is turned on, extreme care must be practiced to ensure that the rest of the Smoothing parameters are set to reasonable values, or else damage to the machine may result. ***For further Smoothing information, please see the sections "Smoothing Configuration Parameters" and "Smoothing Setup Menu" later in this chapter.***

15.3.126 Parameters 236–239 – Motor Cooling Coefficients for axes 5–8

See parameters 20–30 for more information.

15.3.127 Parameter 240 – Rigid Tapping Accel Rate Distance

Default is 0.1000

Setting to -1 for servo motors this will disable the parameters 240, 241. This will introduce banging at the higher spindle speeds as before. For stepper motors do not set to a -1 because this will cause loss of axis position.

This parameter is used for servo motors to eliminate the banging at the start and end of tapping and threading.

This is the distance to get this axis up to speed to sink with the marker pulse on the spindle.

For servo-motors this allows for higher spindle speeds to eliminate the banging. The value that may be used is 1/2 to 2 times the pitch. With higher spindle speeds this value will need to be increased if banging is heard.

This value can be set to cover a range of different pitches of taps and threads.

Also, the clearance amount should be increased to allow enough distance to get the axis up to speed to be synced with the spindle.

For stepper-motors this will allow for larger values to help with the possibility of loss of axis positions.

15.3.128 Parameter 241 – Rigid Tapping Rotational Step Size (Degrees)

Default value: 10.00000 Minimum value: 10.00000

This is the degrees of rotation that the pitch is changed to get its axis synced up with the spindle to help eliminate the banging at the start and end of the tap and thread. A smaller value is preferred. With high RPM and a coarse pitch thread this may have to be increased to eliminate the banging.

15.3.129 Parameter 242 – Minimum Angle Threshold for application of Accel/decel in Threading moves

Default = 30.00000

This is used for the G32 Threading on a lathe, allowing an operator to do a G32 move and continue it with a second G32 move at a different angle. If this value is equal to or less than the angle change, it will not apply the accel/decel at the angle change point at the end of the first G32 and the start of the second G32. If the value is greater than the angle of change it will apply the accel/decel at the angle change between G32 moves.

15.3.130 Parameter 243 – Threading Control

Bit 1 (value 2) suppresses wait for index pulse, Bit 2 (value 4) permits spindle override during threading.

15.3.131 Parameter 244 – Tool Touch-off device PLC input number

This parameter is the input number that the Tool Touch-off device is wired into on the PLC.

15.3.132 Parameter 245 – G71,G72,G74,G75,G76 D/P/Q "implied float" re-interpretation threshold for Inch units, and Parameter 246 – G71,G72,G74,G75,G76 D/P/Q "implied float" re-interpretation threshold for MM units

Parameters 245 and 246 are used to resolve the ambiguities that arise due to the fact that the G71, G72, G74, G75 and G76 cycles can accept "implied floating point" forms of their D/P/Q parameters (which are specified as whole numbers). These G-codes follow the following criteria as to when to re-interpret and when not to re-interpret their D/P/Q parameters.

Assuming that a D/P/Q is a length dimension parameter of a G71, G72, G74, G75 or G76 cycle, the following table shows what happens to D/P/Q values after the re-interpretation process.

D/P/Q value characteristics	Examples	What happens in Inch mode (G20)	What happens in MM mode (G21)
Decimal point "." is present in the D/P/Q parameter.	`D01.23` `P4.56` `Q789.`	D/P/Q values are left alone as-is because these are obviously already floating point numbers.	D/P/Q values are left alone as-is because these are obviously already floating point numbers.
D/P/Q value is a whole number beginning with a leading 0.	`P0010` `Q023`	D/P/Q values are treated as implied floating point values with the decimal point 4 places in from the right. Effective re-interpretation using the examples would be: `P0.0010` and `Q0.0023`	D/P/Q values are treated as implied floating point values with the decimal point 3 places in from the right. Effective re-interpretation using the examples would be: `P0.010` and `Q0.023`
D/P/Q value is a whole number with no leading 0.	`P50` `Q500`	If D/P/Q value is less than or equal to parameter 245, then the value will be left alone. If D/P/Q value is more than parameter 245 then such a D/P/Q value will be treated as an implied floating point value with the decimal point 4 places in from the right. Assuming parameter 245 is set at its default value of 99.0, the effective re-interpretation of the examples would be: `P50` (left as is) and `Q0.0500`	If D/P/Q value is less than or equal to parameter 246, then the value will be left alone. If D/P/Q value is more than parameter 246 then such a D/P/Q value will be treated as an implied floating point value with the decimal point 3 places in from the right. Assuming parameter 246 is set at its default value of 99.0, the effective re-interpretation of the examples would be: `P50` (left as is) and `Q0.500`

A summarized way of understanding the above table is:

- The criteria only applies only if D/P/Q is a length/width parameter of a G71, G72, G74, G75 or G76 cycle.
- If a decimal point is present on the D/P/Q value, then this will prevent re interpretation because the specified numbers are already floating point.
- If the D/P/Q value is a whole number, then adding a leading 0 will force it to be re interpreted as an implied floating point value.
- If the D/P/Q value is a whole number with no leading 0, then it will be subject to the parameter 245/246 thresholds.

Note that this special re-interpretation of D,P,Q length parameters only happens for G71,G72,G74,G75 and G76. All other letter addresses are not affected and D,P,Q parameters used in other situations outside of G71,G72,G74,G75,G76 are not affected either.

15.3.133 Parameter 247 – G70 Multiple Pass Behavior Suppression

This parameter adjusts the behavior of the G70 Finishing Cycle. Setting this parameter to 1 puts G70 into "industry standard behavior" mode. In this mode, a single G70 cycle will do only a single pass on the specified profile at specified offsets U and W. Setting this parameter to 0 puts G70 into multi-pass compatibility mode where U and W divides up the finish allowance purposely left behind by the G71 G72 or G73 “roughing” cycles. In this mode, multiple finish cut passes will be run (one pass for each progressively deeper division) until the final finish offset is reached. See G70 in Chapter 12 for more information.

15.3.134 Parameter 248 – Tool Wear Adjustment magnitude limit

This parameter sets the +/- value limit allowed to be entered into the Tool Wear Adjustment menu.

15.3.135 Parameters 252–255 – Autotune Ka Performance parameters for axes 5–8

See parameters 87–90 for more information.

15.3.136 Parameter 256 – Drive Mode

This parameter indicates to the control software what mode the drives are operating under. It also controls the availability and behavior of the F5 Tune key in the PID Menu based on the drive mode.

Drive mode	Value
Torque mode. Autotune feature is enabled, accessible via F5 in PID Menu	0
Velocity mode. Autotune feature is disabled. Autotune does not work with velocity mode drives and therefore this parameter should be set to 1 to prevent access to Autotune on such machines with these drives.	1
Precision mode. This enables the Precision Mode delay parameters 340–347. This also enables the Auto Delay Calculation feature, accessible via F5 in PID Menu, which is used to automatically calculate these parameters.	2

15.3.137 Parameter 257 – TT1 connection detection PLC input

This parameter stores the input for the TT1 connection detection feature. The spindle inhibit parameter (Parameter 18) must be set (non-zero) for this feature to work. The default for this parameter is 0, which disables this feature. When this parameter is set (non-zero), the Tool Measuring cycle will not run unless a TT1 or a probe is connected. A negative value must be entered if a “normally closed” input is to be used with the control. A positive value must be entered if a “normally open” input is to be used with the control. The absolute value of this parameter will directly reflect the PLC input the TT1 connection detect is wired to.

15.3.138 Parameter 258 – Velocity/Torque Mode override in Precision mode

This is an axis bitfield where setting a bit to ‘on’ allows the corresponding axis to run as if it were in velocity or torque mode, but prone to servo mismatch errors. Bit 0 (value 1) refers to axis #1, bit 1 (value 2) refers to axis #2, bit 2 (value 4) refers to axis #3, bit 3 (value 8) refers to axis #4, and so forth.

15.3.139 Parameter 259 – Manual Axis Designation

This parameter is a bit field that designates an axis as a manual axis regardless of its label. Bit 0 (value 1) refers to axis #1, bit 1 (value 2) refers to axis #2, bit 2 (value 4) refers to axis #3, bit 3 (value 8) refers to axis #4, and so forth. (Note that there is another way to designate an axis as a manual axis, which is to set its label to "M" or "@".)

15.3.140 Parameter 260 – Accelerated Graphics Backplot

This parameter controls the user interface that is presented when F8 – Graph is pressed. See Chapter 3 for more information about Accelerated Graphics Backplot.

Function	Value
Use the legacy Graphics Backplot	-1
Use the Accelerated Graphics Backplot	1

15.3.141 Parameter 261 – Precision Mode Standoff Tolerance Percentage

Indicates the percentage of the motor encoder counts per revolution for which standoff error must exceed before any correction is made. Setting P261 to zero disables standoff error correction. Suggested setting is = 0.01

15.3.142 Parameter 262 – Precision Mode Standoff Delay Time

Time in seconds that an axis must be continuously at rest (no change in expected position) before a correction is made. Suggested setting is 0.1.

15.3.143 Parameter 263 – DRO Display Precision

This parameter lets you decide the precision of the DRO display by adding decimal points. The default setting 4, which corresponds to 4 decimals. Ex. 4.1234. Allows changes to the number of decimals show in the DRO. The default value is 4. A value of 1 will display 1 decimal place while a value of 2 will display 2 places and so on. The range of values that can be displayed are between 1 and 7 decimal places. Use this Parameter to control DRO Display Precision not Parameter 143. Determines the precision after the decimal that is displayed on the DRO.

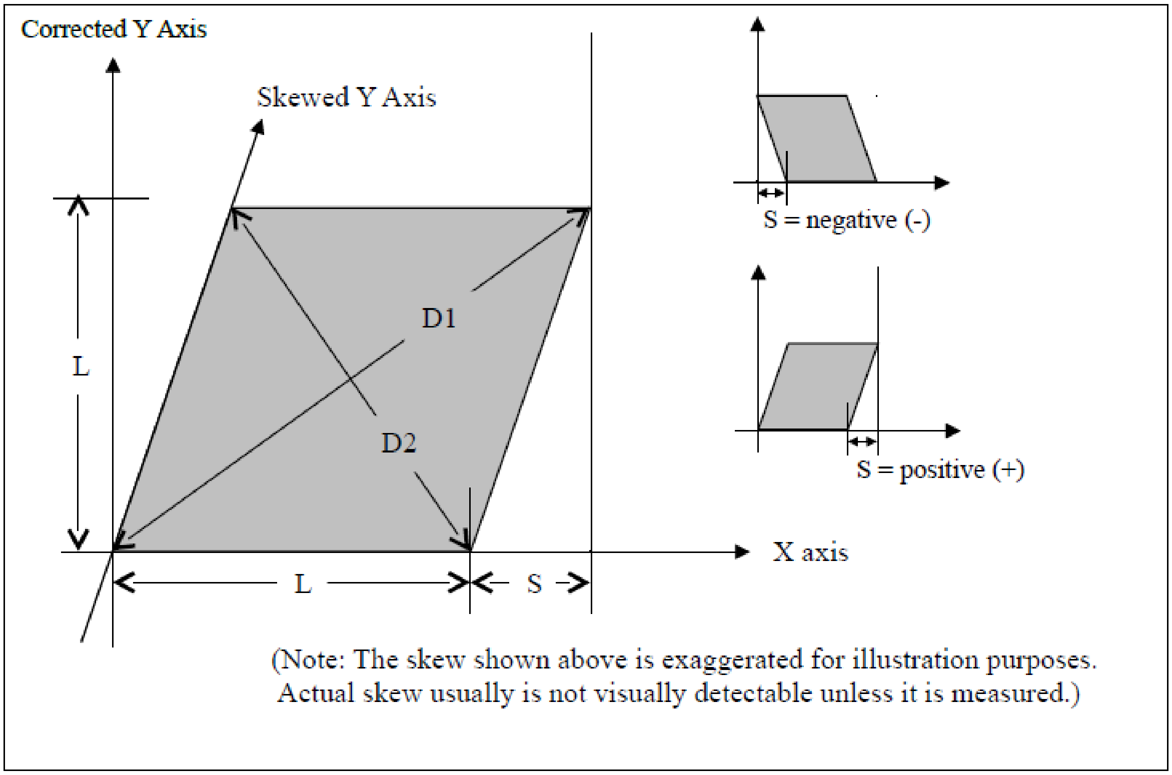
15.3.144 Parameter 270–271 – XY Skew Correction

These parameters work together to correct XY position skew, which can occur if the X axis is not exactly perpendicular to the Y axis (or vice versa). To turn on XY skew correction, use the chart and follow the skew measurement procedure described below. To turn off XY skew correction, set both parameters to 0.

Parameter	Symbol	Description
270	S	+/- X skew deviation from perfect X and perfect Y
271	L	Y length over which to apply a position correction to counteract a skew of amount S

<u>Skew Measurement Procedure:</u> Program a sufficiently large L x L square and cut it on a scrap piece of material using the machine in question. Put this "square" piece against a true square corner and measure the skew S. If the square

is leaning to the left, then S is negative; if it is leaning to the right, then S is positive. Set Parameter 270=S and Parameter 271=L.



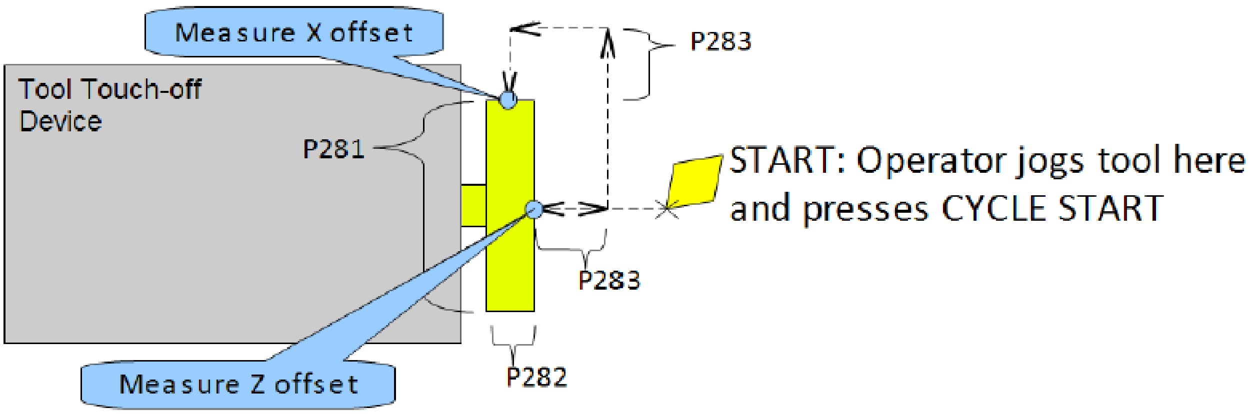
If you have trouble directly measuring S, you can calculate it by measuring the diagonals D1 and D2, and then using the following formula: S = $(D1^2 - D2^2)/(4L)$

15.3.145 Parameter 278 – Spindle Speed Display Precision

This sets the number of digits after the decimal point to display on the Spindle Speed display in the Status Window. A setting of 0 means to show whole number spindle speeds.

15.3.146 Parameter 281 – Tool Touch-off Device X stylus size Parameter 282 – Tool Touch-off Device Z stylus size Parameter 283 – Auto Tool Touch-off safety clearance

These parameters configure the Auto Tool Touch-off cycles in the Tool Geometry Offset Library. The illustration below shows how these parameters are used to configure the Measure Z and X cycle:



15.3.147 Parameter 284–291 – Brake Resistor Wattage for ACDC Drives 1–8

These parameters specify the brake resistor wattage which default to the minimum internal resistor value. If CNC12 detects that the estimated brake wattage exceeds these parameter settings, then a “470 _ axis (drive _) brake wattage exceeded” message is reported in the status window. These warnings may be written at most twice a minute. User variables #27201-#27208 can be used in an M- or G-code program to return the estimated brake wattage as reported by the ACDC drive to MPU11. Note that these are drive numbers and match the LED indicator on the ACDC drive, but do not necessarily match the axes as displayed in the DRO due to the flexibility of drive mapping.

15.3.148 Parameters 292–295 – Aux Key Functions

These parameters are used to assign a function to aux keys 13–16 (i.e. P292 = Au$\times$13 . . . P295 = Au$\times$16). For more information on aux key functions, see Section 15.3.118.

15.3.149 Parameter 300–307 – Drive assignment to Axes 1–8

These parameters control to what physical drive the commands for motion are sent. Parameter 300 assigns a physical drive to axis 1, parameter 301 assigns a physical drive to axis 2, and so on. The values for these parameters can be set to any value from 1–25 based on the table below. **These parameters must be set before attempting to move motors. Note that if you change any of these values, the machine must be powered down and restarted for the changes to take effect. Contact your dealer before changing these values.**

Drive Number Assignment	Drive Type and Location	Description
1, 2, 3, 4, 5, 6, 7, 8	Drive Bus Channel 1–8	Drive types are DC3IOB, DC1, ACSINGLE, OPTIC4, and OPTICDIRECT. Other drive types may be added in the future.
9, 10, 11, 12	GPIO4D/RTK4 Drive Out 1–4	Drive output is implemented as outputs on the PLC bus which are updated at 4000 Hz.
13	RTK4 Drive Out 5	5^{th} output for RTK4 drive
14–16	Reserved	
17, 18, 19, 20	Legacy DC 1–4	Examples of Legacy DC drives are: QUADDRV1, SERVO1, M15DRV1, DC3IO, SERVOLV, SERVO3IO, DCSINGLE.
21, 22, 23, 24, 25	Legacy AC 1–5	Examples of Legacy AC drives are: SD1, and SD3.

15.3.150 Parameter 308–315 – Encoder assignment to Axes 1–8

These parameters control to which encoder the axis should look for feedback. Parameter 308 assigns an encoder to axis 1, parameter 309 assigns an encoder to axis 2, and so on. The values for these parameters can be set to any value from 1–15 based on the table below. These parameters must be set before motors can move. Note that if you change any of these values, the machine must be powered down and restarted for the changes to take effect.

<table>
<tr><th>Encoder Number</th><th>Location</th><th>Description</th></tr>
<tr><td>1</td><td>MPU11 onboard encoder 1</td><td rowspan="6">Encoder inputs on the MPU11, Allin1DC, and OAK.</td></tr>
<tr><td>2</td><td>MPU11 onboard encoder 2</td></tr>
<tr><td>3</td><td>MPU11 onboard encoder 3</td></tr>
<tr><td>4</td><td>MPU11 onboard encoder 4</td></tr>
<tr><td>5</td><td>MPU11 onboard encoder 5</td></tr>
<tr><td>6</td><td>MPU11 onboard encoder 6</td></tr>
<tr><td>7</td><td>Drive Bus Channel encoder 1</td><td rowspan="8">Encoder inputs on Drive bus devices. Here are some examples: One DC3IOB would occupy 3 Drive Bus Channel encoder locations numbered 7,8,9. Two DC3IOB's chained together would occupy 6 Drive Bus Channel encoder locations numbered 7,8,9,10,11,12. An OPTICDIRECT occupies 1 Drive Bus Channel encoder location. Chaining 8 OPTICDIRECTs together would occupy 8 encoder locations numbered 7–14. Legacy AC drives (such as SD1 or SD3) occupy 6 encoder locations numbered 7–12. Note that every Drive Bus device takes up a Drive Bus encoder location even if there is no encoder going to the drive.</td></tr>
<tr><td>8</td><td>Drive Bus Channel encoder 2</td></tr>
<tr><td>9</td><td>Drive Bus Channel encoder 3</td></tr>
<tr><td>10</td><td>Drive Bus Channel encoder 4</td></tr>
<tr><td>11</td><td>Drive Bus Channel encoder 5</td></tr>
<tr><td>12</td><td>Drive Bus Channel encoder 6</td></tr>
<tr><td>13</td><td>Drive Bus Channel encoder 7</td></tr>
<tr><td>14</td><td>Drive Bus Channel encoder 8</td></tr>
<tr><td>15</td><td>MPU11 onboard MPG encoder</td><td>MPG connector with no index pulse</td></tr>
<tr><td>16–21</td><td>Encoder Expansion 1–6</td><td>Encoder inputs on EncExp</td></tr>
</table>

15.3.151 Parameter 316 – Absolute Encoder Bits

This is a bitfield parameter that is used to mark an encoder as being absolute. Note that it is the encoder number or index (see parameters 308–315 above) that is specified as being absolute, **not** the drive number. For example, if the X, Y, and Z axes had absolute encoders and parameters 308, 309, and 310 were set to the values 7, 8, and 9, then this parameter would be set to 448 ($2^6 + 2^7 + 2^8$).

15.3.152 Parameter 317 – Single Turn Absolute Encoder Bits

This is a bitfield parameter that is used to mark an absolute encoder as being single turn. Note that it is the encoder number or index that is specified as being absolute, **not** the drive number. This parameter applies to and has an effect for a scale encoder connected to a rotary axis. A motor encoder that is marked as absolute in Parameter 316 should not be marked as single turn in this parameter.

15.3.153 Parameter 318 – Five Axis Configuration

This parameter specifies configuration settings specific to five axis systems. On five axis systems that have the fifth axis (usually labeled B) as a rotary axis that rotates around a line parallel to the Y axis, i.e., an articulated head, values 1–8 are used to specify the drive to which the scale encoder is connected. A non-zero value also signifies that the fifth axis is a "straight rotary" (not a "triangular rotary" type).

15.3.154 Parameter 319 – Five Axis Options

This parameter specifies options related to using transformed part setting. Transformed part setting is useful to set up a part for machining using G68.1 mode, where the B axis is fixed at a certain angle.

Bit	Function Description	Value
0	Enable transformed part setting*	Enable (Add 1)
1	Do not perform fifth axis unit convserion	Enable (Add 2)

* Machine parameter 166 (5^{th} axis options) must also be set to 17665. When transformed part setting is in effect, the X and Z axes on the DRO are displayed in red using the transformed coordinate system when in the Set Part 0/Position menu.

15.3.155 Parameter 321 – MPU13 DSP Probe Input

The PLC input number used for the DSP probe input on an MPU13 (Hickory) system. This parameter has no effect on MPU11 or MPU12 hardware.

15.3.156 Parameter 322 – Spindle Encoder Port

Allows the user to assign which encoder port the spindle will use to communicate.

15.3.157 Parameter 323 – MPU11 Encoder Speed Filter

This is an axis bitfield where setting a bit to 'on' selects the low speed filters for the corresponding axis. As a general guideline, an axis's bit should be set unless that axis refers to a 3^{rd} party drive. Bit 0 (value 1) refers to axis #1, bit 1 (value 2) refers to axis #2, bit 2 (value 4) refers to axis #3, bit 3 (value 8) refers to axis #4, and so forth.

15.3.158 Parameter 324–331 – Axis Boxcar Size

These parameters set the maximum sample size of the boxcar filter for each axis. For most applications these values should be set to 0 (default). For applications where the motor drives are too responsive or jumpy, these boxcar filters act as a way to smooth the PID output. These filters average the PID output by the entered boxcar size, thus reducing PID spikes. Eg. A value of 4 would add the last 4 PID values and then divide by 4 for before output. A value of 5 would add the last 5 PID values then divide by 5 before output. Note that during PID averaging, the Error Sum is not zeroed during direction reversal. Conversely, a value of 0 disables the boxcar filter for that axis and also zeroes the Error Sum on Direction Reversal. Note also that as a byproduct of averaging by 1, a boxcar value of 1 may be used to produce a true PID output while not zeroing the Error Sum on direction reversal.

15.3.159 Parameters 332–335 – Encoder Error Suppression

These parameters control suppression of various types of encoder errors on a per encoder basis. These parameters are bitfields by encoder index, NOT axis index. The mpu11 has 15 encoder indexes. For example, to disable encoder faults for Encoder #5 on the mpu11, enter a 16 into parameters 332 and 334. To suppress encoder faults and error messages for all 8 axes, enter 255 into each of the parameters mentioned in the following table.

Parameter	Function
332	Suppress encoder differential faults
333	Suppress encoder differential error messages
334	Suppress encoder quadrature faults
335	Suppress encoder quadrature error messages

15.3.160 Parameters 336–339 – Motor torque estimation for velocity mode drives

These parameters are intended to be used with velocity mode drives in order to faciliatate a more accurate display of the axis load meter bars shown under each position in the main DRO display. If P336 = 0, then this feature is disabled and the normal PID output is displayed by the axis load meter bars. This feature is enabled if P336 is non-zero.

Parameter	Symbol	Function
336	G	Overall gain setting (0 = disable Motor torque estimation)
337	Ga	Absolute error gain
338	Gs	Error sum gain
339	Gd	Delta error gain

Technical details:
The axis meter bar value (V) is then calculated as: `V = abs(100.0*G*((Ea*Ga + Es*Gs + Ed*Gd))/integration_limit)`, where `Ea` is the absolute error, `Es` is the error sum, and `Ed` is the delta error from the PID algorithm and the `integration_limit` is from the "Limit" value set in the PID Config screen. This value `V` is then bound to the range 0–100.

15.3.161 Parameters 340–347 – Precision Mode delay (in milliseconds) for axes 1–8

These parameters are used for synchronizing individual precision mode drives with different delays. The MPU11 will use these values and compensate for the differences in the delays. These parameters have up to a 0.25 millisecond resolution.

These parameters are typically set with auto-tuning in the PID menu.

15.3.162 Parameters 348, 351, and 354 – MPG/Handwheel Encoder Input 1, 2, and 3

The encoder input for the MPG or handwheel. (1–15) See encoder chart in Parameter 308.

Note: PLC program interaction is needed to enable an MPG or handwheel. A typical value is one (P348=1) for MPG ON.

15.3.163 Parameters 349, 352, and 355 – MPG/Handwheel Detents per Revolution 1, 2, and 3

This value is the number of clicks (detents) per revolution. It is the number of divisions or markings on the mpg or handwheel. Moving the mpg or handwheel one detent or division will cause the motor to move one jog increment (depending on the multiplier $\times$1, $\times$10, $\times$100, etc). A typical value is one hundred (P349=100) for 100 clicks per revolution.

Note: PLC program interaction is needed to enable an MPG or handwheel.

15.3.164 Parameters 350, 353, and 356 – MPG/Handwheel Encoder Counts per Revolution 1, 2, and 3

This value is the number of counts generated per rotation of the mpg or handwheel. A typical value is one hundred (P350=100) for 100 counts per rotation.

Note: PLC program interaction is needed to enable an MPG or handwheel.

15.3.165 Parameters 357–364 – Axis Drive Max RPM for Axes 1–8

These parameters allow you to set the drive/motor max rate capability (in RPMs) for use by the PID algorithm for the calculation of the axis KV1 contribution. This value is independent from the axis Max Rate setting in the Jog Parameters menu, which is used by the control software. However, for those axes whose corresponding parameters are set to 0 (the default), the the PID algorithm will use the axis Max Rate setting in the Jog Parameters for the calculation of the axis KV1 contribution. These parameters are intended for 3rd party velocity mode drives that have a different max rate setting than that of the control software.

15.3.166 Parameter 365 – Drive power-on delay

This specifies the number of milliseconds that the MPU11 will wait between the moment that drive power first comes on and the start of commanded motion. However, this does not work for the case of turning off a single axis using M93, moving a different axis, and then moving the powered off axis. The default value is 0 which means no delay.

15.3.167 Parameters 366–367 – Probe / TT1 deceleration multiplier

Parameter	Function
366	Probe deceleration multiplier: This factor adjusts the deceleration rate of probing moves coming to a stop due to probe hit.
367	TT1 deceleration multiplier: This factor adjusts the deceleration rate of TT1 tool measuring moves coming to a stop due to tool touch detect.

The normal axis acceleration rate (configured by dividing the Max Rate in the Jog Parameters screen by the Accel Time in the PID Config screen) is multiplied by the value of these parameters to determine the actual decelerations used for each situation. A value higher than 1 will cause a more abrupt deceleration than the normal axis configuration. A value below 1 will cause a gentler deceleration.

15.3.168 Parameter 368 – Autonomous Digitizing Angle Adjustment

This parameter determines by what angle adjustments will be made to avoid shanking during Autonomous Digitizing.

Note: This parameter applies only to performance racing users.

15.3.169 Parameter 369 – Tool Check Max Absolute Angle

This parameter is used in five-axis systems that have articulated heads (a rotary axis usually labeled as B) to limit the angle at which the tool check can be performed. If the absolute value of the B axis exceeds this value, a warning is displayed. This parameter also specifies a limit for the tilt angle used in five axis autonomous digitizing routines.

15.3.170 Parameters 374–379 – ACDC Drive Debug Log Settings

These parameters are used by support technicians and should be left at value 0.

15.3.171 Parameters 387–389 – Debugging Parameters

These parameters are used for internal debugging and troubleshooting. They should be left at 0.

15.3.172 Parameters 392–394 – DP-7 parameters

These are parameters specific to the DP-7 probe and are used only if Parameter 155 = 2.

Parameter	Function
392	DP-7 Pullback Distance: The distance the probe moves off of the surface after a probing move.
393	DP-7 Pullback Feedrate: The feedrate for the pullback move.
394	DP-7 Measuring Feedrate: The feedrate for the slow measuring move.

15.3.173 Parameter 395 – Probing Setup Traverse Speed

This sets the probing traverse feedrate for the macro-based probing cycles on engine block systems.

15.3.174 Parameter 396 – Probing Setup Plunge Speed

This sets the probing plunge feedrate for the macro-based probing cycles on engine block systems.

15.3.175 Parameter 397 – Combustion Chamber Clearance Height

This parameter determines the clearance height used during combustion chamber digitizing. The default value is 0.25 inches or 6.35 millimeters.

Note: This parameters applies only to performance racing users.

15.3.176 Parameter 398 – Port/Block mode

This determines the current mode of Port/Block systems and is set by the Port/Block menu. This parameter should not be manually modified.

Function Description	Value
3-Axis	0
Engine Block	1
Tilt Table	2

15.3.177 Parameter 399 – AD1 arc chord tolerance adjustment

This parameter adjusts the precision of AD1 arcs. When Smoothing is turned off (P220 = 0), arc moves (such as G2 and G3) are generated as a string of many small linear moves that are used to closely approximate the programmed arc. These small linear moves are called arc chords. These arc chords straddle each side of the theoretical true arc path, but their distance (in encoder counts) from the path is limited by what value is set in this parameter. The default value is . 5, meaning that by default the arc chord never strays away from theoretical true arc by more than 1/2 encoder count.

15.3.178 Parameter 400 – Run Menu Cycle Start Enabled

Set this value to zero to disable the **CYCLE START** button in the Run Menu (F4 from Main Menu). For any other value, **CYCLE START** will be enabled on the Run Menu.

15.3.179 Parameter 401 – Forget last job loaded

Setting this parameter to a value of 1.0 will cause the last job loaded to be forgotten and replaced with the name “no_job_loaded.cnc” when the CNC software is started.

15.3.180 Parameter 403 – Disable Keyboard Jogging Legend

This parameter determines whether or not the Keyboard Jogging Legend is launched with an Alt+j press. A value of 1.0 will prevent it from launching.

15.3.181 Parameter 404 – Unused

Note: Prior to v4.16, this was the Active G-Codes Display that was moved to parameter 408

15.3.182 Parameter 405 – Tool Touch Off Type

Set this parameter to select the type of tool touch off that is being used.

15.3.183 Parameter 406 – Probe State When Tripped

This parameter chooses whether the probe is open or closed when tripped. If closed when tripped, P406=0. If open when tripped, P406=1. The default setting is P406=0. For example, a KP-3 would correspond to P406=1. A DP-4 would correspond to P406=0.

15.3.184 Parameter 408 – Active G-Codes Display

Set this parameter to 0 to always display the currently active G-codes in the bottom left corner of the main screen and MDI screen. Set it to 1 to display the G-codes only in MDI. Set it to 2 to disable the display completely.

15.3.185 Parameter 409 – Probe Type

Set this parameter to select the type of probe connected.

15.3.186 Parameter 410 – Probe/Tool Touch Off Warning

15.3.187 Parameter 411 – MPG Type

Set this parameter to select the type of MPG connected according to the table below.

MPG type	Value
Non-USB MPG	0
USB CWP-4 (legacy)	0
USB WMPG-4	1
WMPG-6	2
WMPG-4 Plasma	3

15.3.188 Parameter 413 – Park macro

When this parameter is set to 1.0, the Park function on the shutdown menu will run the macro named park.mac located in the system directory.

15.3.189 Parameter 414 – Home File Type

Used by the Acorn Wizard to remember the type of homing file that is being used by the system.

15.3.190 Parameter 415 – Ether1616 Configuration Bits

Specifies the desired configuration of Ether1616 devices connected to an Acorn system. There is one bit for each of the supported Ether1616 devices. Bit 0 is used for an Ether1616 configured with the jumper on A0, bit 1 is for Ether1616 configured with a jumper on A1, etc. CNC software will check at startup for the detected Ether1616 devices and compare them with this parameter, displaying a warning dialog for mismatches. The PLC program can also use this parameter in conjunction with the PLC system variable SV_ETHER1616_ONLINE_BITS to check device status.

15.3.191 Parameter 416 – Spindle Inhibit

Set this parameter to inhibit the spindle when detect is on (green).

15.3.192 Parameter 417 – Preview G-Code Before Job

This parameter allows a preview of the G-Code of the active job to be displayed before the job is run. The default value is 0 which doesn't preview the G-Code. A value of 1 will turn on the G-Code Preview. Regardless of the setting you can view the entire G-Code of a job by using **F6 – Edit** from the Main Menu.

15.3.193 Parameter 418 – Assign Enter Key to Cycle Start

This parameter assigns the Enter keyboard key to function as the Cycle Start button PC Keyboard <enter key> = CNC Machine <Cycle Start>

15.3.194 Parameter 421 – Clean Fan Filter Reminder

This parameter controls the frequency of how often a "Clean Fan Filter" message appears at startup. The value of the parameter equates to the number of days until the message is displayed. For example, a value of 10 would display the "Clean Fan Filter" message every 10 days. The default value is 0, which turns this message off.

15.3.195 Parameter 422 – PLC Diagnostics Display

This parameter controls the PLC Diagnostics Display option when pressing Alt+I on the main menu. The default value of 0 will enable the Basic PLC Diagnostics Display. The Basic PLC Diagnostics Menu will display the available Inputs and Outputs detected by CNC12. A value of 1 will enable the Advanced PLC Diagnostics Display. The Advanced PLC Diagnostic Menu will display all possible Inputs, Outputs, Memory, and Stages.

15.3.196 Parameter 423 – Display Scale Position on DRO

This parameter allows Scale Position to be displayed on the DRO. Set the parameter value by adding up the values of the encoder inputs you have hooked up to scales. For example if you wanted to display Scale Position from encoder inputs 1, 2, and 3, you would enter a value of 7 ($1 + 2 + 4$). The default value is 0 for no Scale Position displayed on the DRO.

Bit	Function Description	Value
0	1	Add 1
1	2	Add 2
2	3	Add 4
3	4	Add 8
4	5	Add 16
5	6	Add 32
6	7	Add 64
7	8	Add 128

15.3.197 Parameter 425 – Force Rehoming After EStop Condition

Set this parameter to require the machine to rehome after the estop is pressed.

15.3.198 Parameter 430 – RTG Spindle Speed RPM Display

This parameter allows a real-time display of the current spindle speed in RPM.

15.3.199 Parameter 440 – Stopped for Jogging Continue Bit

This parameter specifies the behavior when the control is executing an M0/M200/M201 and machine parameter 10 is set to allow M0 jogging.

Value	Behavior
0	Wait for CYCLE START or TOOL CHECK only
1–255	Also wait for the indicated SV SKIN EVENT to be set
256	Also wait for the Skinning API Job.ContinueExecution function call

15.3.200 Parameters 441–443 – MPG (1–3) Axis Selection

When given a value of 1–8, these parameters select which axis a hard-wired MPG will move.

Example: When using a MPG2, giving parameter 442 a value of 1 will cause the MPG2 to control axis 1.

Note: PLC program interaction is needed for these features. The PLC program is in direct control of MPG modes.

15.3.201 Parameter 450 – Inhibit MPG Z Axis Fast Movement

This parameter inhibits the use of the $\times$100 speed on the Z axis. When set to a value of 1, the $\times$100 speed on the Z axis will be reduced to $\times$10 speed.

15.3.202 Parameter 459 – Second Spindle Encoder Axis Number

Input from a spindle encoder is required for spindle-slaved movements such as those used in the Rigid Tapping cycles. If there is no spindle encoder connected, then this parameter should be set to 0. This parameter specifies the axis number (1 through 8) to which the spindle encoder is assigned. Encoder assignments are specified by parameters 308–315. For example, if you decide to configure the 6^{th} axis as the 2^{nd} spindle, and the spindle's encoder is the 2^{nd} MPU11 onboard input encoder, then Parameter 459 (this parameter) should be set to 6 and Parameter 312 should be set to 2.

Currently, the 2^{nd} spindle can only be operated as paired (slaved) to the 1^{st} spindle. Future development will allow independent command of the 2^{nd} spindle.

15.3.203 Parameter 460 – Second Spindle Maximum Speed

This parameter controls the maximum speed for the 2^{nd} Spindle. The minimum value is 0.

15.3.204 Parameter 461 – Second Spindle Minimum Speed

This parameter controls the minimum speed for the 2^{nd} Spindle. The minimum value is 0.

15.3.205 Parameter 462 – Second Spindle Encoder Counts/Revolution

This parameter controls the counts revolution for the 2nd spindle encoder. Input from the spindle encoder is required for the spindle-slaved movements used in the Rigid Tapping cycles. If the encoder counts up when running CW (M3), the value of this parameter must be positive. If the encoder counts up when running CCW (M4), the value of this parameter must be negative.

15.3.206 Parameter 507 – 5th Axis Pairing – Slave Axis

This parameter sets the Acorn Axes Squaring Slave Axis. It is set within the Wizard and is used for specific Wizard Macros. If manually pairing Axes, see Parameter 64.

15.3.207 Parameter 508 – 5th Axis Pairing – Master Axis

This parameter sets the Acorn Axes Squaring Master Axis. It is set within the Wizard and is used for specific Wizard Macros. If manually pairing Axes, see Parameter 64.

15.3.208 Parameter 560 – Laser X Offset

This parameter should be set to the length of the Laser X offset, given in inches or mm depending on the CNC12 set-up.

15.3.209 Parameter 561 – Laser Y Offset

This parameter should be set to the length of the Laser Y offset, given in inches or mm depending on the CNC12 set-up.

15.3.210 Parameter 600 – A Axis Expected Velocity Switch Point

This parameter is used in five axis systems that have scales attached to the rotary A axis. It defines the point at which the absolute expected velocity of the A axis will switch to using the velocity specified in machine parameter 601. If this value is zero, then the velocity as entered in the Scale Menu is used for scale correction. A typical value is 1.0.

15.3.211 Parameter 601 – A Axis Scale Switch To Velocity

This parameter is used in five axis systems that have scales attached to the rotary A axis. This is the velocity used for scale correction when P600 $>$ 0 and the absolute expected velocity of the A axis $>$ switch point (P600). A typical value is 0.0.

15.3.212 Parameter 602 – A Axis Scale Correction Meter Max Deflection Count

This parameter should be set to 0.0. It can be used by qualified technicians to help tune and troubleshoot scale corrections.

15.3.213 Parameter 603 – B Axis Expected Velocity Switch Point

This parameter is used in five axis systems that have scales attached to the rotary B axis. It defines the point at which the absolute expected velocity of the B axis will switch to using the velocity specified in machine parameter 604. If this value is zero, then the velocity as entered in the Scale Menu is used for scale correction. A typical value is 1.0.

15.3.214 Parameter 604 – B Axis Scale Switch To Velocity

This parameter is used in five axis systems that have scales attached to the rotary B axis. This is the velocity used for scale correction when P603 $>$ 0 and the absolute expected velocity of the B axis $>$ switch point (P603). A typical value is 0.0.

15.3.215 Parameter 605 – B Axis Scale Correction Meter Max Deflection Count

This parameter should be set to 0.0. It can be used by qualified technicians to help tune and troubleshoot scale corrections.

15.3.216 Parameter 814 – Base Frequency (Hz)

15.3.217 Parameter 815 – Inverse Output

15.3.218 Parameter 817 – PWM

15.3.219 Parameter 830 – ATC Type

15.3.220 Parameter 855 – MPG Performance

15.3.221 Parameter 856 – VCP Jogging State on Acorn Power Up

15.3.222 Parameters 900–999 – PLC program parameters

15.3.223 Parameters 997 – Spindle Cooling Fan Delay Timer

This parameter allows the user to adjust the spindle cooling fan delay timer.

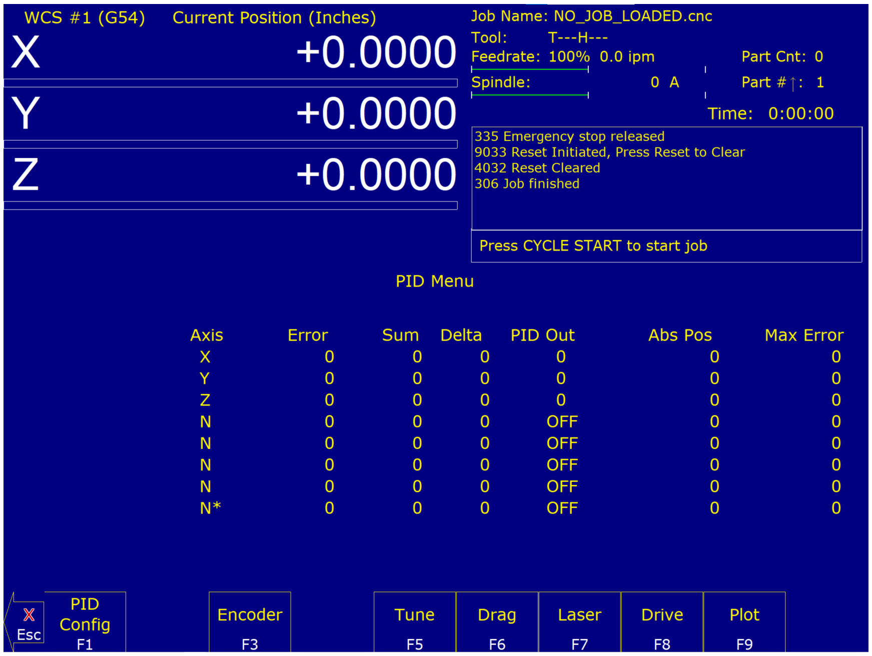
15.3.224 Parameters 998–999 – PLC program parameters

These parameters are used as a way of communicating floating point values to a PLC program. The meanings of these parameters depends on how a PLC program uses them and can vary from one machine to another. One suggested use of these parameters is as a set of configuration values. The values of these parameters are saved upon modification (via a menu or CNC job) and will be retain their values even after shutdown and restart of the control software.

All remaining parameters are reserved for further expansion.

15.4 PID Menu

Pressing **F4 – PID** from the Configuration screen will bring up the PID Menu. The PID Menu provides qualified technicians with a method of changing the PID dependent data to test and configure your machine.



WARNING The PID Parameters **should not** be changed without contacting your dealer. Corrupt or incorrect values could cause damage to the machine, personal injury, or both.

15.4.1 F1 – PID Config

This option displays the Oscilloscope tuning screen, and is intended for qualified technicians **only**. It allows technicians to modify the PID values, and to see (in real time) the effects of those modifications. Altering the PID values will cause **DRAMATIC** changes in the way the servo system operates, leading to possible machine damage. **DO NOT** attempt to change these parameters without contacting your dealer.

The general idea is to reduce the Absolute Error (ErrAbs) and the Sum of Absolute Error (ErrSum), which are both measured in encoder counts. Absolute Error tells you how far off position the machine is at any particular point in time, and the Sum is used when tuning to make sure the overall error is being reduced.

Special Note for SD3 drives: The program containing test moves (edited via F1 and run via F2 on the Oscilloscope tuning screen) must contain a dwell (G4 P__), otherwise the SD3's drive parameters will not be sent.

F1 – Edit Program	Change the program that will run when F2 is pressed
F2 – Run Program	Causes the machine to run a simple test program, while collecting data
F3 – Ranges	Can be used to specify the X and Y ranges for the Oscilloscope view
F4 – Toggles & Pan	Allows changes to how the collected data is displayed, and panning via the cursor keys
F5 – Zoom In	Zooms in
F6 – Zoom Out	Zooms out
F7 – Zoom All	Fits all of the collected data into the Oscilloscope view
F8 – Change Axis	Tells the MPU11 to collect data for a different axis (displayed in the top left)
F9 – Save & Apply	Saves any modifications
F10 – Save & Exit	Saves any modifications and exits the Oscilloscope menu
Page Up – Tweak +	Allows small modifications (+1%) to the PID values while the program is running. Hold shift for a larger (+10%) modification.
Page Down – Tweak –	Allows small modifications (-1%) to the PID values while the program is running. Hold shift for a larger (-10%) modification.

WARNING Improper PID values can ruin the machine, cause personal injury, and/or destroy the motor drives!!!

15.4.2 F5 – Tune

This option is available only for Torque mode and Precision mode drives. See Parameter 256 earlier in this chapter for more information about Drive modes.

If the drives are in Torque mode, pressing this key will start the Autotune procedure. It is used by qualified technicians to automatically determine values for Max Rate, Accel/Decel time, and Deadstart (See section Motor Configuration: Jog Parameters) as well as certain PID parameters for each installed axis. The Autotune procedure will make a series of moves on each non-paired controlled axis, traveling a limited distance (configured via parameters 95–98 and 156–159) from the initial position in all directions to determine the friction and gravity of each axis. The initial high-speed move will use half of this distance. You cannot run Autotune on paired axes. Do not run Autotune unless requested to do so by a qualified technician.

If the drives are in Precision mode, pressing this key will start the Auto Delay Calculation procedure. It is used by qualified technicians to automatically determine values for the Precision Mode delay parameters 340–347. The Auto Delay Calculation procedure will make a single move on each non-paired controlled axis, traveling a limited distance (configured via parameters 95–98 and 156–159) from the initial position. You cannot run the Auto Delay Calculation procedure on paired axes. Do not run the Auto Delay Calculation procedure unless requested to do so by a qualified technician.

15.4.3 F6 – Drag

This option is used by qualified technicians to determine whether your machine is binding anywhere along the axis travel. To run a drag test press F6 – Drag to begin the drag test. Press F1 – Next Axis to select the axis you wish to check and then hit the CYCLE START button. A text file drag_x.out, or a similarly named file, is generated and stored in the c:\cncm directory. If significant drag occurs, a message will be displayed on screen. Contact your dealer to correct the problem as soon as possible.

15.4.4 F7 – Laser

This option is used by qualified technicians to take automated laser measurements and create or adjust the ballscrew compensation tables. Do not attempt to run automatic laser compensation without first contacting your dealer for details.

15.4.5 F8 – Drive

This menu will only appear on AC systems and only affects SD or ACSingle drives. It is not for general viewing or modification by any unqualified individual. For more information about this menu option, refer to the SD installation manual.

15.4.6 F9 – Plot

This option is used by qualified technicians to plot data.

15.5 System Test

System Test is a feature that was released in software versions 8.23 (DOS) and 1.10 (CNC10). The purpose of the System Test is to check basic machine functionality and to verify key parameters are set correctly. If the System Test has not run and passed, a message will be displayed on the screen at power up (see figure). This message will continue to be displayed at power up, until the System Test has been run successfully.

MACHINE SETUP
NOT COMPLETED.

MACHINE IS NOT READY
TO RUN.

CONTACT YOUR DEALER.

Requirements before running System Test:

1. Make sure control is fully integrated.
2. Check direction of each axis and change direction reversal (refer to TB137) and limits as required.
3. Setup/calibrate motor rev.'s/per (refer to TB36) and backlash (refer to TB37) for each axis.
4. Set travel limits.

   Note: To save time in the home switch test, zero out the inputs that aren't being used for home switches.
5. Run auto tune and drag tests.
6. Make sure machine table is cleared of all items.
7. Empty the spindle of any tools, and verify that the spindle face cannot run into the worktable at the bottom of its travel. **DO NOT START** the system test with a tool in the spindle.
8. (ATC systems only) ATC systems need to have their tool changers setup and verified by running tool changes.
9. (ATC systems only) Insert at least four tools into the tool carousel. Load one into bin #1 and another into the maximum bin number (ex. Load into bin 16 if a 16 – Tool ATC). The other two tools should be loaded into the carousel to balance it out. In the case of a 16 – Tool ATC load bins 5 and 10. The tools must have lengths that differ by at least 0.010 inches (0.25 mm). Do NOT use fly cutters, diamond tip tools or non conductive tools.
10. (ATC systems only) Position a TT1 on the worktable and connect the TT1 cable. Position the spindle over the TT1 then press F1 – Setup, F1 – Part, F9 – WCS Table and F1 – Return. Enter the X and Y values displayed on the screen in return point #3 and set the Z value to zero. **Note:** make sure the spindle face will not run into the TT1 at the bottom of its travel (if it doesn't clear, do NOT run the System Test).

Running the System Test:

1. To run the System Test, press **F1 – Setup**, **F3 – Config**, enter the correct password, and then press **F5 – Test**. The control will load the systest.cnc file located in the CNC10 directory. If for any reason the systest.cnc file does not load, go to the CNC10 directory and manually load it.

2. Before System Test starts running any tests it moves the Z-axis to home. The travel limits and machine home are verified that they are set. Travel limits are not tested for axes that are labeled as M, N, or S, or that have the rotary bit set in parameters 91–94 (axes 1–4) and or 166 (5^{th} axis). The test also checks parameter 6 and 160, if P6 is set to 1 and 160 is set to a non-zero value the System Test will display a menu where you can select a single test or run all tests.

3. Here is a quick list of the Tests; below you can read about them in more detail.

   (a) ATC – TT1 Test

   (b) Quick Home Switch Test

   (c) ATC – Spindle Test

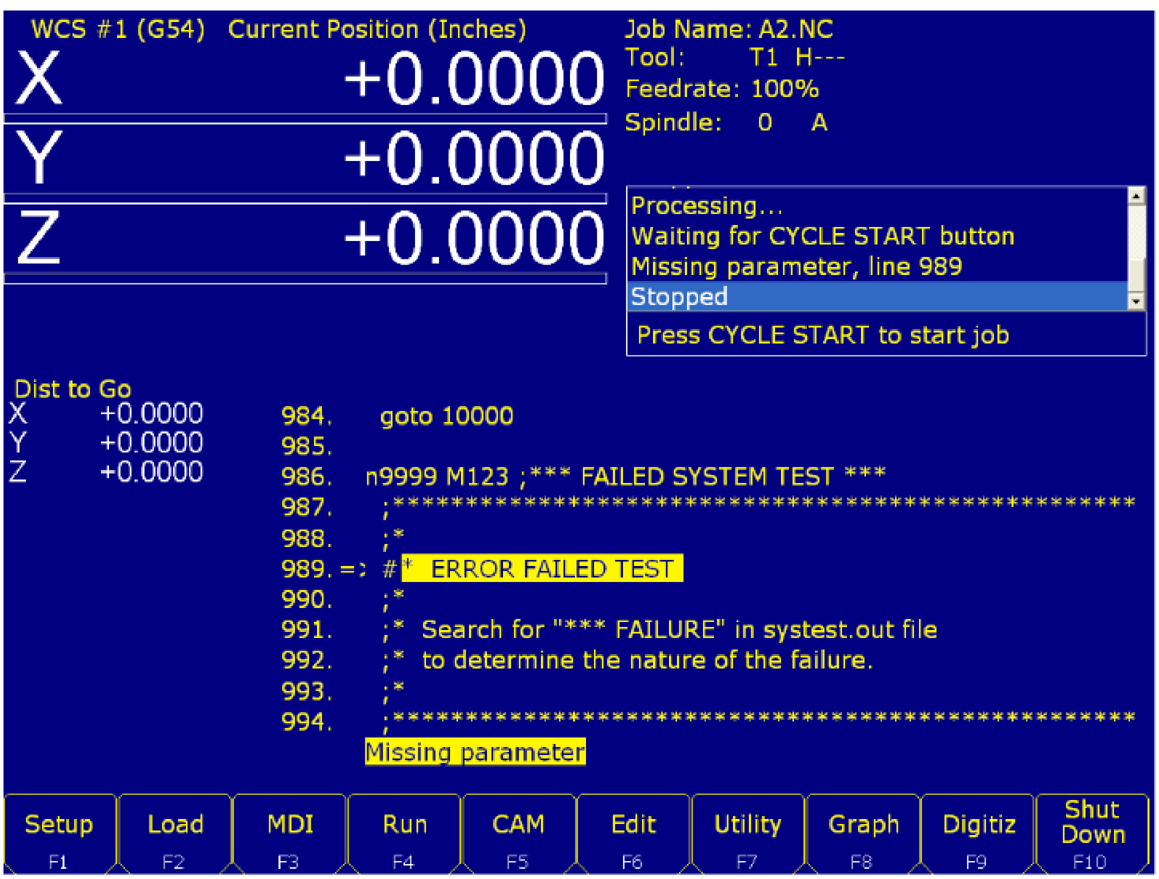
   (d) ATC – Tapping Test

   (e) Home Switch and Travel Limit Test

   (f) ATC – Tool Change Test

   Note that Non-ATC systems will automatically run tests (b) and (e).

4. If System test fails at any time while testing you will see "Missing parameter, line 989" on the Config Screen. Go back to the main screen where you will see the following screen (see figure below). Load the systest.out file located in the CNC10 directory and press **F6 – Edit** to see exactly what failed (See figure).



```
;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;
;     Checking travel limits and home set
*** FAILURE:  AXIS 1 INVALID TRAVEL LIMITS
*** FAILED ***
```

Test Definitions:

1. **ATC TT1 Test.** This test starts with the following G-code message displayed on the screen:

```
                                        ; * Connect TT-1 and verify the XY
                                          location of the TT-1
                                        ; * is set in return point #3. Verify that
                                            the spindle
                                        ; * face is above the TT-1 when at the Z
                                          minus travel
                                        ; * limit.  After pressing CYCLE START,
                                        ; * trigger the TT-1 twice.
M0
if [#9044 != 0] the #[a] = #9044
if [#9044 == 0] the #[a] = #9011
M123 L1                                 ; TT-1 set to be input
M123 q0 p#a
M101/#a
```

After pressing Cycle Start, the TT1 needs to be manually triggered twice. The best way to do this is by touching an electrically conductive material to the worktable and the top of the TT1. A right-angled Allen-wrench about six inched long works well.

How to trigger TT1

Before triggering the TT1 you will notice in the message window the message “Waiting for input #15 (M101)”. If the TT1 parameters are set correctly the program will stop at a M0 after triggering the TT1. If the program doesn’t reach the second M0 shortly after triggering the TT1, stop the program. Press Alt + I to turn on Live PLC debugging and trigger the TT1 to see which input is actually toggling and then change parameter 44 to the input being tripped. If no input is tripped check the TT1 wiring.

2. **Home Switch Test** This test checks to see if the machine can be homed reliably by making sure the home

switches are not too close to the index pulse of the motors. If this test fails and you get a "home switch too close to index pulse" error message, refer to Tech bulletin 92 for a description of how to correct the problem.

3. **ATC Spindle Test** This test is made up of three parts:

   (a) First parameter 78 (Spindle Speed Display) is set to 1.0 to enable live measurement and calculated spindle speed to be displayed in the CNC status window. Parameter 36 (Rigid Tapping Enable/Disable) is set to 1.0 to enable rigid tapping and parameter 33 (Spindle Motor Gear Ratio) is checked to see if it's set to 1.0. If not set to 1.0, it will be changed to a 1.0 and a warning will be logged.

   (b) The program turns the spindle on for a few seconds and looks for any change in the absolute encoder position of all axes. If no changes are detected, the test will error out and log the failure. In which case the spindle encoder wiring and connections need to be checked. If movement is detected, parameter 35 is verified that it's set correctly. A wrong value will result in a failure; the failure message in the systest.out file will indicate the correct value. Change P35 and run test again. A quick check for spindle direction is made, where the spindle is turned on with an M3 to see in the spindle is counting up or counting down. If counting up nothing happens, counting down will result in a minus sign being added or removed from parameter 34 (Spindle Encoder Counts/Revolution) and a warning logged.

   (c) The test now records the average spindle deceleration time from 2000RPM. Then spindle speeds of 10%, 20%, 30%, . . . , up to 100% of maximum spindle speed set in the Control Configuration are checked and recorded in systest.out. The spindle speeds are commanded and then the actual speeds are recorded after several seconds. The commanded speeds are compared to theactual and must be within 3% to pass. If a failure occurs check P34 is set correctly and that the inverter has been programmed for the correct maximum output frequency. To check parameter 34 turn the spindle 10 times by hand and divide the difference in encoder counts by 10. The absolute encoder position can be read off the PID screen.

4. **ATC Tapping Test.** Starts at machine home and taps down three inches at 1000RPM. The Z-axis movement should be very smooth, however if you notice a jerky motion it is probably due to P34 having the wrong sign (+/-). For example, if P34 is set to 4096 and you have jerky motion on Z-axis change P34 to -4096. Another problem could occur with a "Z axis cannot follow the spindle" error. This means you have the wrong number of counts in P34 (i.e.8000 instead of 4096), although the correct value for P34 should have been determined in spindle test.

5. **Machine Home Switch and Travel Limit Test.**

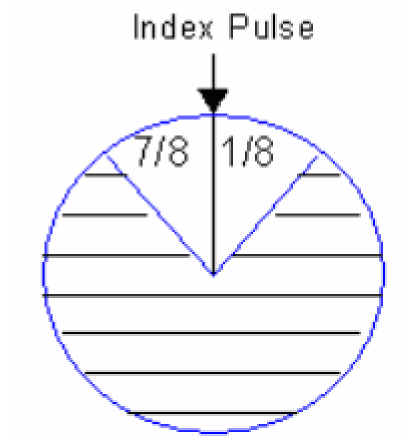
   (a) **Home Switch test.** The machine home test starts by checking the values stored in the plus/minus home switch fields in the Machine Configuration – Motor screen. There are two checks made before the test begins. The program first checks the "Machine home at pwrup" setting in the Control Configuration. If set to "Jog" the test is skipped and a warning is logged in systest.out. In most cases it should be set to "Home switch" or "Ref Marks – HS". The second check looks for any axis that has at least one home switch input defined except for, rotary configured axes or axes with an "M", "N", or "S" label. Reference Mark homing will have both plus/minus home switches set to zero. So any axes that fail this check will generate a warning in the systest.out file.

   <u>Review of homing process when homing to a switch:</u>

      i. The axis moves toward the home switch at the slow jog rate until the home switch is tripped.
      ii. The axis moves away from the switch until the home switch closes.
      iii. The axis moves further away by a small amount (0.0025 inches).
      iv. The axis moves in small increments away from the switch until the encoder index pulse is located.

   Axes with a minus home switch defined will have multiple measurements taken of the home position (found by using an M91/ command) and the position after step (3) in the homing procedure. The measurements are logged along with error measurements and other data. To pass the test, all home position measurements must be within 0.0005 inches of the first measurement. If the test fails and the error is within 0.0010 inches of a motor revolution, the follow message will be logged: "SUSPECTED LIMIT SWITCH PROBLEM". Otherwise, "SUSPECTED ENCODER INDEX PROBLEM" will be logged.

   Now the distance between the home position and the off-switch position (position after step 3) are checked to ensure that they are not less than 1/8 or greater than 7/8 of a motor revolution. If less than 1/8 or greater than 7/8 of a motor revolution a failure occurs and a “Home position too close to encoder index pulse” message is logged (See figure 4). To correct this failure, move the limit switch or trip dog a distance that corresponds to a 1/2 revolution of the motor. The full turn distance is recorded in the systest.out file. If unable to move, decouple the motor and rotate the motor shaft 90 degrees clockwise. When attempting to resolve the problem by rotating the motor shaft, note that this method will affect the results of the home switch test at the other end of travel, which means that it is possible when the test is repeated that the opposite end which passed will now fail. In this case, you must rotate the motor shaft 90 degrees clockwise once again.



The index pulse in the position seen in the picture would produce an error. The correct position would be in the shaded region of the circle.

If an axis has a plus home switch set the same measurements and analysis are performed on it.

(b) **Travel Limits Test** When there are two home switches, the overall travel defined in the Machine Configuration – Jog parameters must be less than or equal to the distance between the minus and plus home positions, but within 1.0 inch (25.4mm). For example, the distance between the plus and minus home positions is 24.135 inches, which means the travel limit must be within 23.135–24.135 inches to pass the test.

If only one or no home switches set for an axis, two moves are made. One from the home position out to the minus travel limit and another from the home position out to the plus travel limit. If the travel limits are too long, the test will be stopped by either a “Full power without motion” or “limit tripped” message.

After the travel limits are tested and have passed, all axes move to the center of the travel limits.

6. **ATC Tool Changing Test** Starts by performing a series of tool changes. These tool changes are done in such a way to alternate carousel directions and maximize carousel travel. For example, a 16 – Tool setup would run a series of tool changes like T1, T8, T2, T9, T3, T10 etc., and would continue in this manner until 48 tool changes (3x the maximum number of bins) have been completed.

   After every tool change, the tool height is checked using the TT1. The first time a tool is measured, the Z-axis machine position is recorded. On subsequent tool checks, the machine position is compared to the initial recorded position. The test will fail if subsequent tool measurements are not within 0.005 inches of the initial position.

   Assuming a few tool changes have been completed, some possible causes for the wrong tool being picked up are:

   - Parameter 161 (ATC Maximum Tool Bins) is set wrong. Ex. P161 = 16 but the tool carousel holds 20 tools.
   - Not using a brake motor for the carousel or there is a faulty brake motor.
   - The state of the tool counter sensor after a tool change is not electrically open. This can be caused when the wrong type of tool counter sensor (Normally Open or Normally Closed) is matched to the mechanical setup of the counting mechanism.
   - The tool counter sensor or wiring is faulty.

- Electrical noise.

Completed System Test If the control runs through the complete test and passes everything the systest.out file will say passed and the “Machine setup not completed” message will no longer appear at machine power up.

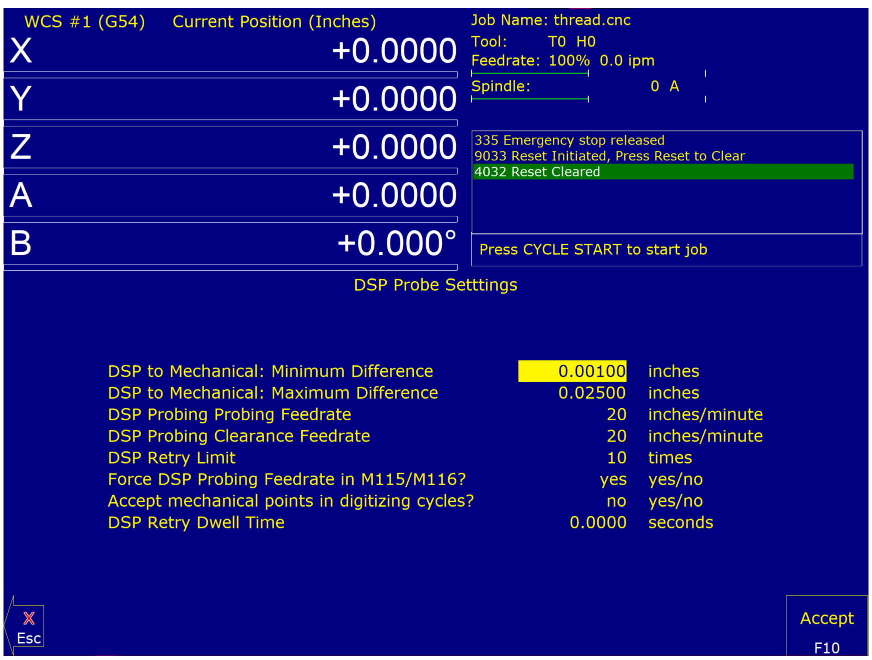
WARNING: If you run the system test after it’s already passed, the systest.out file will be wiped out and the control will need to run through the complete test again.

15.6 ATC Init.

This menu will only appear on enhanced ATC systems. Please see your ATC manual for further information.

15.7 DSP Probe Configuration

Pressing F7 – DSP Probe from the configuration screen will display the DSP Probe configuration. Note that this menu is available only for DSP probes (Parameter 155=1).



Minimum Difference:
Default – 0.001 inches. The minimum difference between mechanical and reported DSP position.

Maximum difference:
Default – 0.025 inches. The maximum difference between mechanical and reported DSP position.

DSP Probing feedrate:
Default – 20. The feedrate to use for DSP probing moves.

DSP Probing Clearance feedrate:
Default – 20. The feedrate to use for DSP probing Clearance moves.

DSP Retry Limit:

Default – 10 times. The maximum number of tries to achieve a passing window.

Force DSP probing feedrate in M115/M116:
Default – Yes. This forces the DSP feedrate for M115/M116 (recommended) rather than using the feedrate specified in an F command.

Accept Mechanical Points in Digitizing Cycles
Default – Yes. Used when any given point has failed window checking # times where # = the limit as specified in DSP Retry Limit. Setting this option to Yes records the last mechanical position rather than throw out the point entirely. If this option is set to no, no value is recorded for the point and digitizing continues.

DSP Retry Dwell Time
Default – 0 seconds. The amount of time to dwell at the end of a retract move when retrying a DSP probing move. The purpose of this is to give the probe time to stop vibrating before moving towards the surface again on a bad hit.

Other DSP Probe settings
The DSP Probe eliminates the need for a multiple-hit probe measuring move (A fast inward move to find the surface followed by a slow move to measure the surface). Therefore, the Slow Probing Rate (Parameter 15) does not apply to probing moves when the DSP probe is enabled. However, the Slow Probing Rate still applies to TT1 tool measurements. The Fast Probing Rate (Parameter 14) is still used for positioning moves in digitizing cycles, but not for probing mvoes which measure the surface. Instead, the DSP Probing feedrate is used for these moves.

The following list is a list of other parameters which affect the DSP Probe. See the parameter descriptions in the Machine Parameters section (earlier in this chapter) for more information.

- Probe Type (Parameter 155): This parameter enables/disables the DSP Probe.
- Recovery Distance (Parameter 13): This parameter is used for failed DSP windows. On a failed window, the DSP probe will retract this distance before retrying.
- Fast Probing Rate (Parameter 14): Used for positioning moves in digitizing and probing routines. These are moves which are not measuring a surface. It is not used for surface measuring moves when the DSP probe is enabled.
- Slow Probing Rate (Parameter 15): Not used by the DSP probe. However, it will still be used by TT-1 tool measurements.
- Repeatability Tolerance (Parameter 151): Recommended value is 0 (disabled). This parameter enables repeatability checking. It will work in conjunction with the DSP probe.

15.8 Modifying the CAM Menu

1. From the main screen of CNC12, exit to Windows by pressing **F10 – Shut Down**→**F9 – Exit CNC12**.
2. Using the mouse, press the Windows start button (typically in the lower left of the screen).
3. Search for “File Explorer” and click on File Explorer.
4. Click on “This PC” in the left-hand sidebar.
5. Double-click on “Local Disk (C:)”.
6. Double-click the cncm directory.
7. Double-click the cnccams.txt file. If there isn’t one, create a new one.

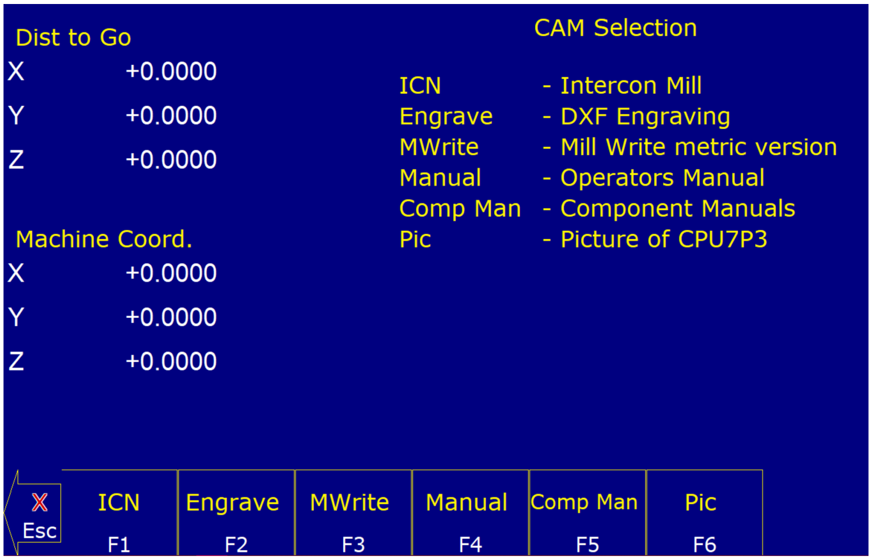
8. Add the necessary lines to be used for the particular button. Some examples are shown below.

9. Save the file by either selecting Save from the File menu or by pressing **CTRL+S**.

10. Exit out of the editor by clicking on the X at the upper right corner of the window, selecting Exit from the File menu, or pressing **ALT+F4**.

11. Start the CNC12 software by double-clicking on the CNC12 icon.

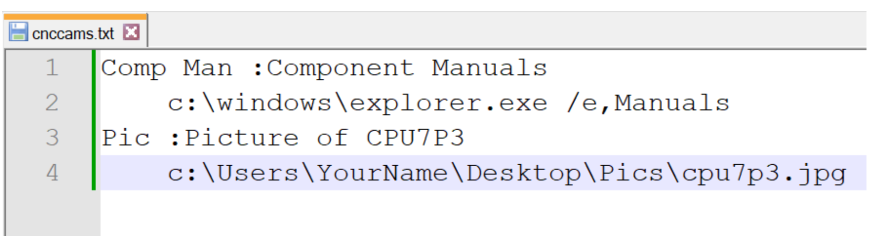
12. Cycle the E-stop button to clear the Software Ready Fault message.



15.9 CAM File Format

As seen above, the format of the CAM file starts with a line of information. This is the header, and contains two parts separated by a colon. The first part is a short name that will appear on the function key label in the CAM menu. This short name is limited to seven (7) characters. The second part is the name description that appears in the center of the screen of the CAM menu, as shown in the picture below. On an indented line directly beneath the header are the lines of code that will be executed when the function key is pressed.

Note: When trying to open a specific file, the path or filename **CANNOT** contain any spaces.


cnccams.txt

```
Comp Man :Component Manuals
    c:\windows\explorer.exe /e,Manuals
Pic :Picture of CPU7P3
    c:\Users\YourName\Desktop\Pics\cpu7p3.jpg
```

Example: The first example above opens the Manuals directory. This directory opens on the Centroid control's desktop in a Windows Explorer window when the **F5 – Comp Man** button is pressed in the CAM menu. The second example shown above opens the cpu7p3.jpg picture when the **F6 – Pic** button is pressed in the CAM menu. These will only behave in that way as long as the Centroid control has the software to be able to do it. That means that for the above example, the Centroid control needs to have Windows Explorer and a program that can open .jpg files loaded on the control.

Note: The CNC12 software already has the logic for the Intercon, Engrave, MillWrite, and Manual buttons that are in the CAM menu built into the software, so the logic will not be displayed in the cnccams.txt file.

15.10 PLC I/O Diagnostic App

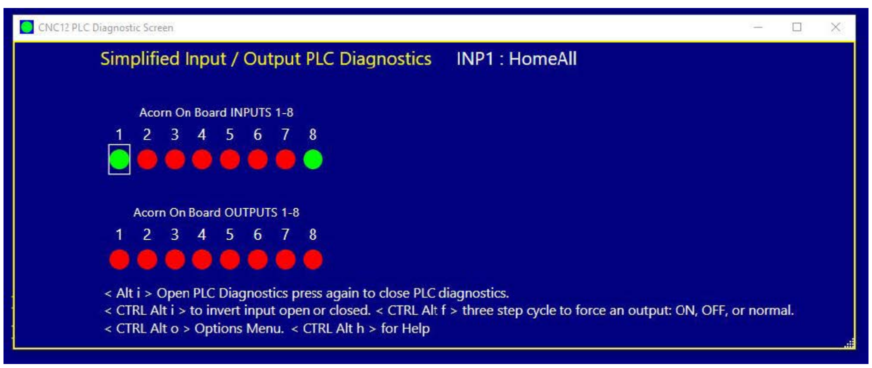
The PLC Diagnostic App is a tool to observe CNC controller Inputs and Outputs, and manually activate/invert/deactivate CNC controller Inputs and Outputs. It is also commonly used to debug any type of switches, relays, contactors, ATC's, etc. that are connected to the CNC controller.

With the CNC12 PLC Diagnostic App you can:

- Observe the state of any input or output live while the machine is running.
- 'Turn on' or 'Turn off' any input or output by clicking on it or using the keyboard live while the machine is running.
- Invert an input live while the machine is running.

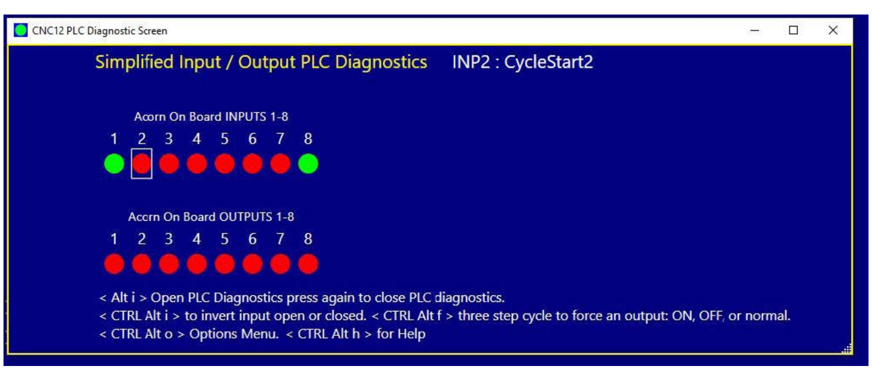
15.10.1 Using the PLC Diagnostic app

From the main screen of CNC12, start the PLC diagnostic screen by pressing the keys **ALT** and **i** at the same time.

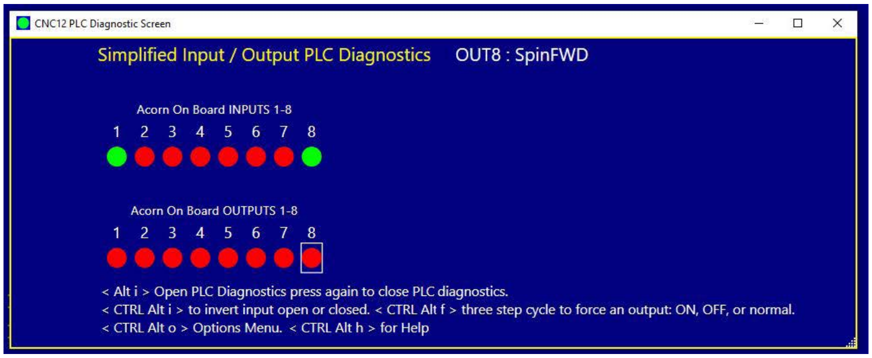


To exit from the Input and Output screen, press the keys **ALT** and **i** again at the same time.

Using the mouse (or arrow keys) to select an input causes a thin, white-lined box to appear around the virtual LED and the assigned PLC Input function name to be displayed. In the example below, an external hard CycleStart button input has been assigned to input #2 and named CycleStart2. The cursor, represented by a square, white box around the virtual LED for Input #2, and the name of the input #2 PLC function assignment appears.

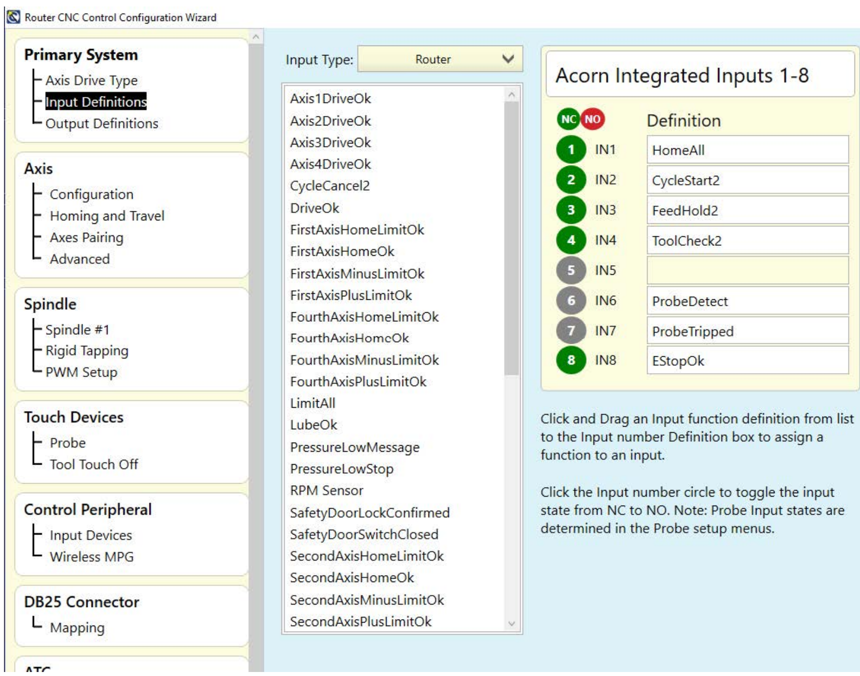


Using the mouse to select an output causes a thin, white-lined box to appear around the virtual LED and the assigned PLC Input function name for that input to be displayed. In the example below, Output relay 8 has been assigned the Spindle Forward PLC function named SpindFWD.

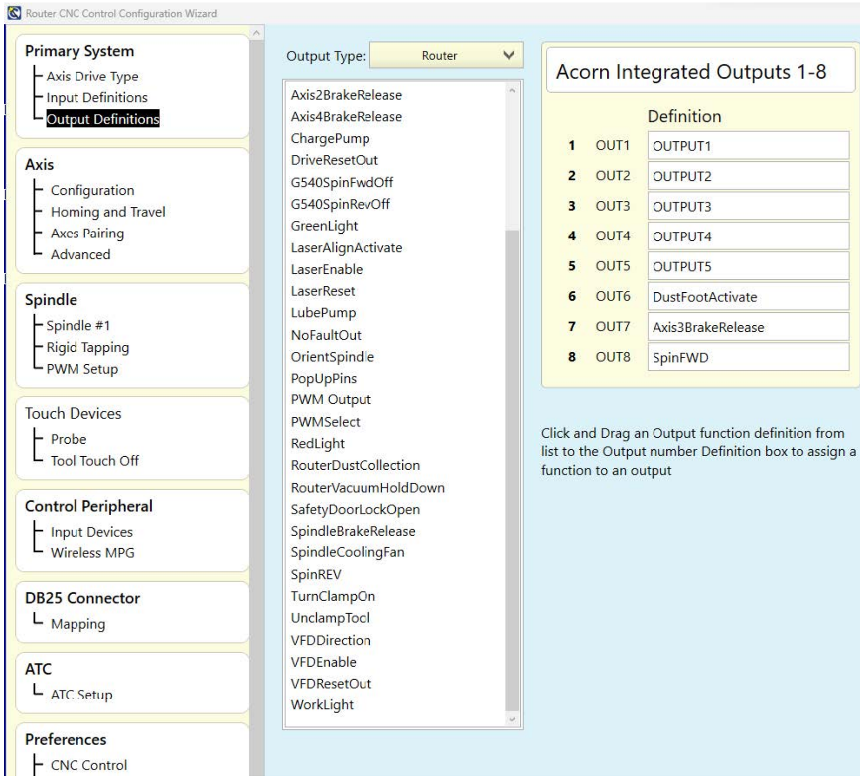


The PLC function assignment to the inputs and outputs are made with the CNC control setup Wizard.

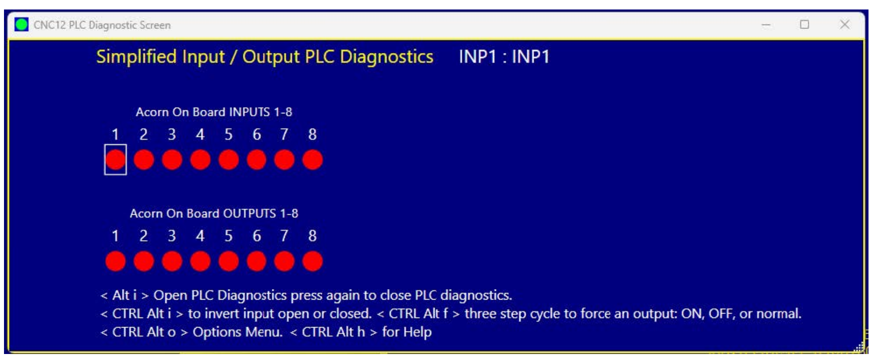
Below is the Wizard PLC Function Input Definitions setup menu for some of the examples we are using in this section.



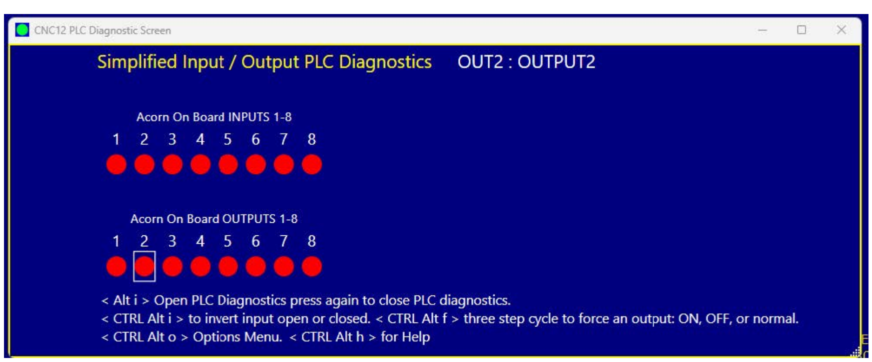
Below is the Wizard PLC function Output definitions setup menu for some of the examples we are using in this section.



If no PLC assignment has been made in the Wizard for a particular input or output, they simply appear as INP1:INP1



or OUT2: OUTPUT2, etc.



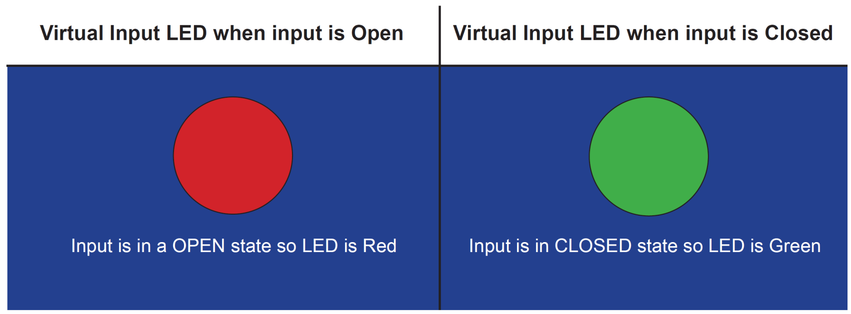
Even if an input or output is not assigned to a PLC function, they can still be observed and manipulated with the PLC Diagnostic App.

15.10.2 Observing the Inputs and Outputs

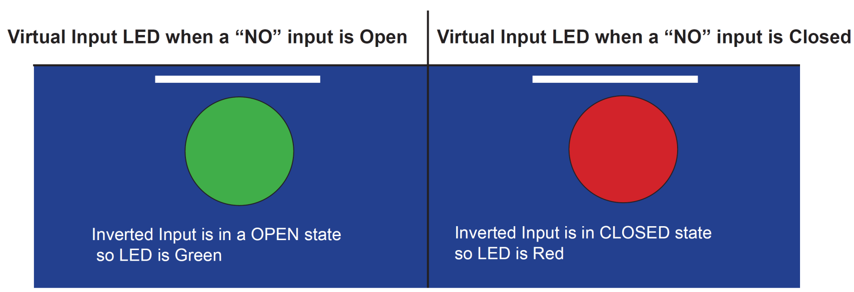
Input LED colors explained

A red Virtual LED for an Input indicates that the Input is “open”
A green Virtual LED for an Input indicates that the Input is “closed”

When an input in the Wizard is set to NC (Normally Closed), the PLC diagnostic LED will appear Green when the input is made (aka closed). See below.



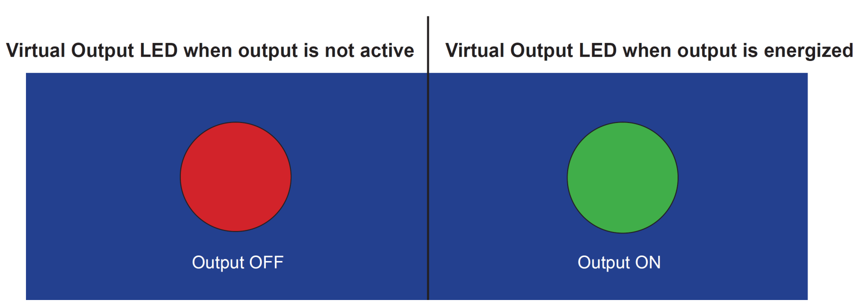
When an input in the Wizard is set to NO (Normally Open), the PLC diagnostic LED will appear as seen below. A white line above the Virtual LED indicates that the particular input has been Inverted (set to Normally Open).



A green Virtual LED with a line above it is inverted and indicates that the Input is “open”. A red Virtual LED with a line above it is inverted and indicates that the Input is “closed”

Note: Whenever possible, use a NC type switch or sensor for CNC applications.

15.10.3 Observing Outputs



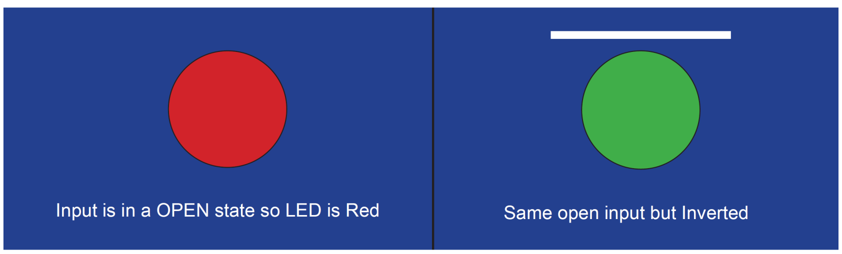
15.10.4 Manually interacting with Input and Outputs

Inverting Inputs

A useful tool in the PLC diagnostic screen to employ while testing and setting up is the ability to manually "invert" an input directly and immediately. Move the mouse (or use the arrow keys) to move to the desired input number to invert, and press the **CTRL**, **ALT** and **i** keys at the same time. Press the **CTRL**, **ALT** and **i** keys at the same time again to cycle the input from an inverted to not inverted state. A double-click on the mouse can also be used to invert or revert an input. A white line will appear above an input that has been inverted either manually or by the Wizard. Any input can be manually inverted, whether it has been assigned a PLC function or not.

Manually inverting an input is useful when first setting up a CNC control. For instance, when an input is configured in the Wizard but not actually wired up to the CNC control, the integrator can use the mouse or the hot keys **CTRL+ALT+i** to invert that input. This way, the control thinks that the input is wired up and in its happy state, therefore "bypassing" this function. Think of it as tricking the PLC into thinking that the switch or sensor is connected and activated when it is not.

This tool can also be used if the switch or sensor is wired up, and you wish to defeat it or exercise the PLC logic without actuating the physical switch to simulate switch activation.



Inverting an input in this manner is commonly used as a debug tool when initially configuring inputs, however the Wizard will set the input inversions properly depending on the NC (normally open) or NO (normally closed) selection made for that particular input. Be sure to return the input to its previous state when the input has been properly wired and configured with the Wizard for normal operation.

Manually interacting with Outputs

Another useful tool in the PLC Diagnostic App for use while testing and setting things up is the ability to manually activate or deactivate an output.

Any output can be manually activated or deactivated whether it has been assigned a PLC function or not.

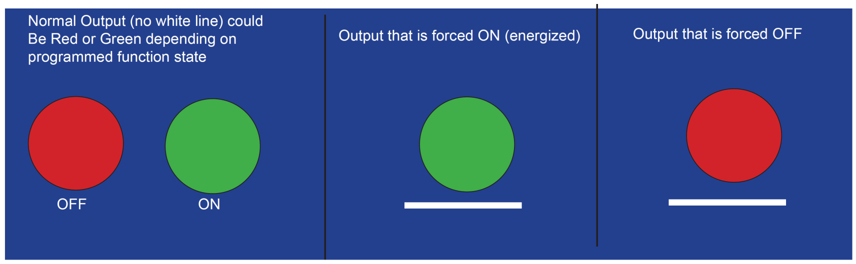
WARNING Be careful if there is high power equipment connected to the CNC control equipment that will be activated when using the PLC Diagnostic App to manually energize an output.

Move the cursor to the desired output number to activate, then double click or press **CTRL**, **ALT**, and **f** at the same time to activate the output.

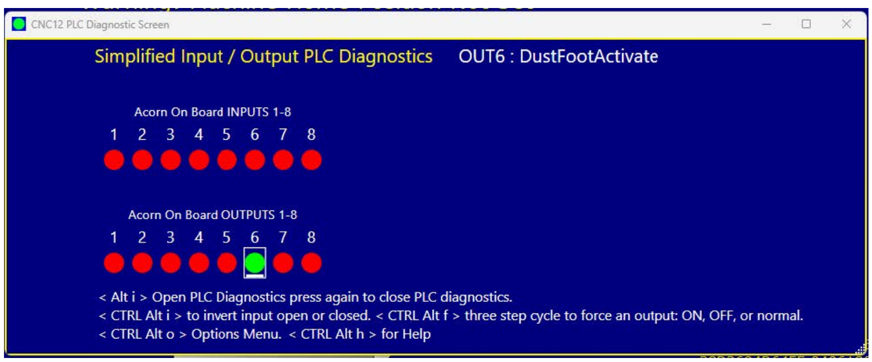
Pressing **CTRL**, **ALT**, and **f** at the same time again deactivates the output.

Pressing **CTRL**, **ALT**, and **f** at the same time again returns the output to the normal programmed state.

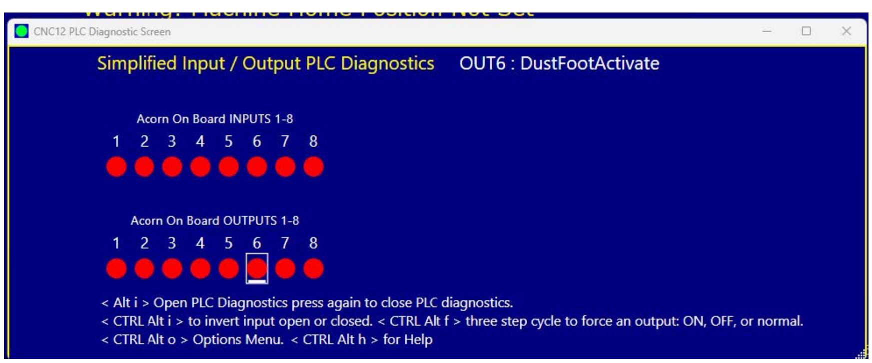
A white line will appear below an Output that has been manually manipulated.



For instance, use the following steps to test an output that controls a Dust Foot attachment on a CNC router and see if the output has been wired properly to the corresponding air solenoid. First, move the cursor to the output number that has been assigned the Dust Foot function (in the case below this is Output 6) and double click the output (or press **CTRL**, **ALT** and **f**) at the same time and see if the output activates.



Double click the output again (or press **CTRL**, **ALT** and **f** at the same time) and see if the output deactivates.

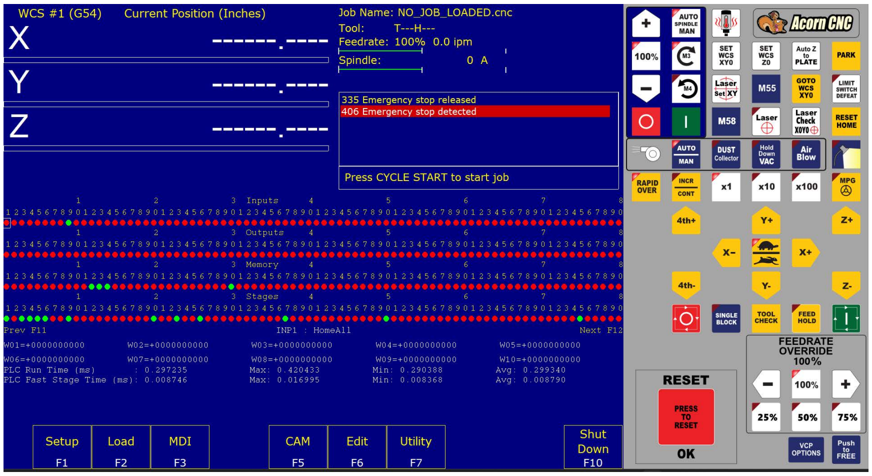


Pressing **CTRL**, **ALT**, and **f** again at the same time to return the output to its normal programmed state.

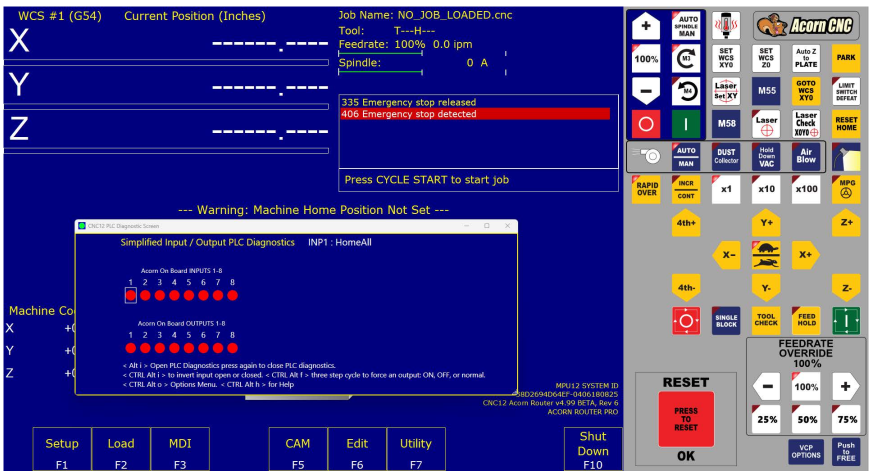
Be sure to return the output to its normal state (no white line) for normal CNC control operation.

There are two PLC diagnostic tools available with CNC12. The simplified and the original.

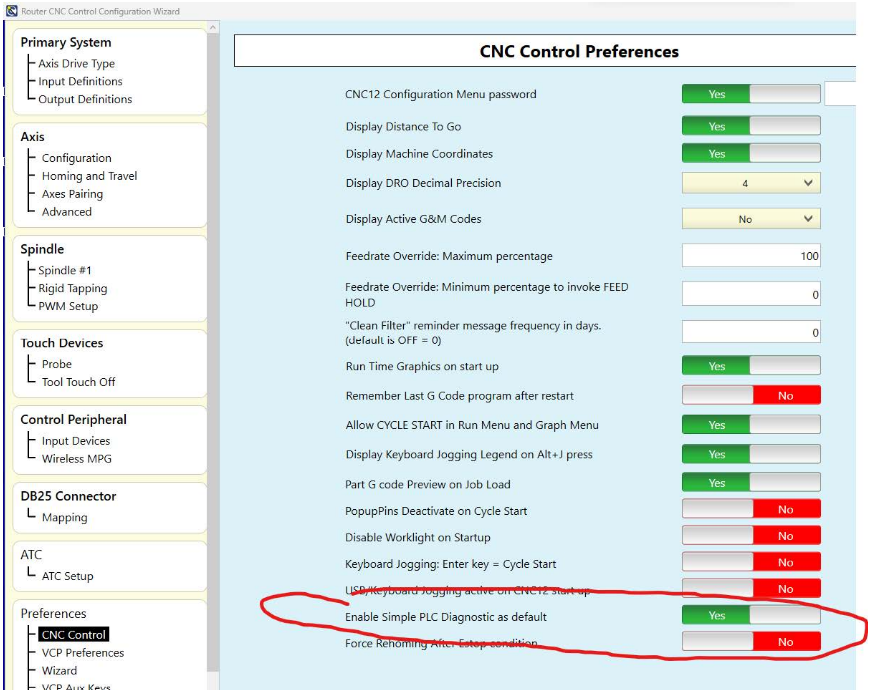
The Original looks like the image below and displays additional information compared to the simplified version.



The simplified PLC Diagnostic App looks like the image below and is the default selection for Acorn and AcornSix.

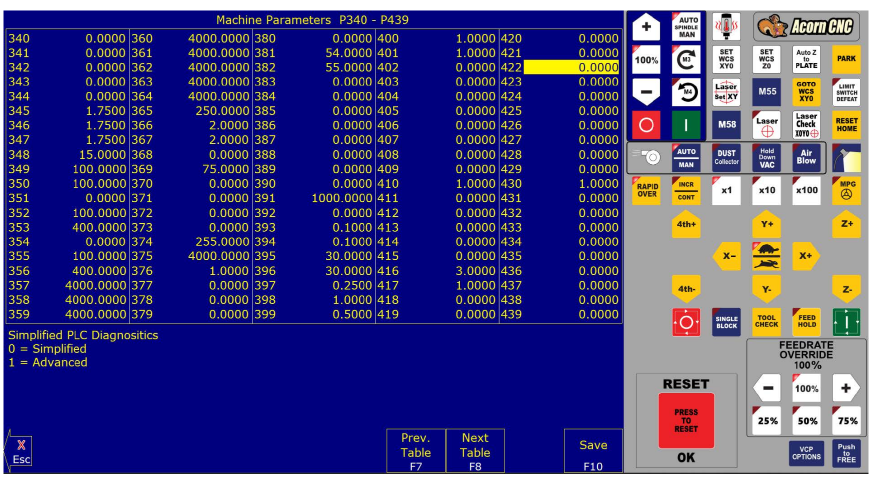


Controls for which tool is the default application are found in the Acorn and AcornSix Wizard under "Preferences"→"CNC Control"



Which PLC Diagnostic tool to use can also be selected using Parameter 422 in the CNC configuration parameter menu.

P422 = 0 = Use Simplified PLC Diagnostic App
P422 = 1 = Use Advanced PLC Diagnostic Menu



Note: Programmed outputs will automatically turn off when Cycle Cancel is pressed.

15.11 Smoothing Configuration Parameters

Parameter	Description	Recommended values
220	Turn the Smoothing feature ON or OFF.	1 = Smoothing (set to 0 to use Exact Stop mode)

221	**NBpts:** The number of points in the Smoothing filter. The higher this value, the more rounded corners will become (see tolerance below).	For Milling Machines: 0 to 10	For Routers: 5 to 20
222	**STEP:** Smoothing breaks up a G code program into segments of this vector size. Use this rule of thumb: Tolerance = (Nbpts*STEP)/3.	For Milling Machines: 0.001 inch / 0.025 mm	For Routers: 0.01 inch / 0.25 mm
223	**Umax:** Sustained safe throughput rate going to the CPU10/MPU11 card.	800	
224	**Centripetal control options:** This bitfield parameter controls the Centripetal stage of the Smoothing module. Value 0 (default) makes Centripetal operate on all axes and disables excessive axis accel checking. Values 1 and 3 (bit 0 = 1) limits Centripetal to only linear axes. Values 2 and 3 (bit 1 = 1) enables excessive axis accel checking.	0 (Centripetal stage will operate on all axes and disables excessive axis accel checking.)	
226	**W**: Feature Width over which the Min Angle is determined.	10	
227	**Min_Angle:** Minimum angle to smooth in degrees. Settings of 95 to 100 degrees will come to a near stop and produce sharp right angles. 60 to 85 will move continuously while rounding angles.	For Sharp corners 95 to 100 degrees	For rounded corners 60 to 85 degrees
228	**S curve:** The recommended setting for this parameter is 0.	0 = Off completely (recommended setting)	
229	**Backplot/Smoothing mode:** Smoothing may slow down the display of Backplot Graphics. This parameter allows a faster backplot by not showing Smoothing.	0 = Faster Backplot, smoothing may be active but is not shown 1 = Slower Backplot, smoothing effects shown.	
230	**Curve Feedrate Multiplier:** Reducing this value below 1.0 will cause the machine to move slower around curves and corners, minimizing “bangs” and overshoots. Increasing this value above 1.0 may allow you to run your machine faster if the feedrates in arcs and corners are still satisfactory.	1.0 (default value) 0.1 to 5.0 (Depending on user’s preference for speed vs “bangs” and overshoots)	
231	**Acceleration Multiplier:** This parameter allows you to adjust the overall acceleration / deceleration rate as a means to reduce machine vibration and noise during starting, stopping, and feedrate changes. Reducing this value below 1.0 will cause more gentle accelerations and decelerations. Increasing this value above 1.0 will cause faster accelerations / decelerations.	1.0 (default value) 0.5 to 1.5 (Depending on user’s preference for quickness of accelerations / decelerations)	
216	**Lash Type:** users have a choice for backlash compensation to occur in the PC or in the MPU11 card itself. This is a new feature and is still under testing, use pc side comp for now. PC side comp is what we’ve been using for years.	PC Side = 1, MPU11 Side = 0	

Note: STEP must be in the same units that the control is currently set to (Inches or MM). Once entered, if you change units in the control, the Smoothing parameters will automatically be converted to the other units for you. This way, the parameters will not have to be re-entered after they have been entered initially.

15.12 G-code AD2 Smoothing

G-code Smoothing, also known as “AD2”, is a Centroid CNC control feature that pre-processes G-code and smooths out its geometry, before handing off the machine tool position moves to the CNC controller. Smoothing, as the name implies, can result in smoother motions when dealing with certain kinds of G-code. The Smoothing feature was created to allow CNC machines to run smoothly when running a G-code program that has a lot of short vectors. These short vectors may or may not be the best approximation of a curve depending on the settings used, especially when it comes to 3D work. Smoothing is best suited for all types of 3D surfacing programs generated by CAD/CAM systems, as well as high-speed 2D contouring (such as Adaptive Machining). Most machines will see large gains in the performance of both speed and surface finish.

To understand why Smoothing is a useful tool, let’s first understand the default CNC control “accell decell” algorithm known as AD1. AD1 CNC motion executes the G-code exactly as commanded, with no smoothing or geometry manipulation. In other words, the controller will move exactly the way the G code tells it to (for better or worse). If using a “good” G code, then AD1 works well. If there is short vector G-code that jumps all over the place while trying to approximate a curve, AD1 will move exactly as the G-code is commanding it to.

To tell if the CAD/CAM system is putting out undesirable short vectors, zoom way into the part (or open the G code file in an editor) and observe the vector lengths and position end points. There will be G-code position commands jumping around in lots of short vectors, and not going in a smooth line or arc. Specifically, milling machines cutting molds, CNC routers running high speed 2D contouring and 3D surfacing, and plasma machines will all see dramatic improvements in feedrates, surface finish, and overall time that it takes to machine a part when using Smoothing. Smoothing works with all types of axis stepper motors, servo motors and drives.

15.12.1 How does Smoothing work?

In simplistic terms, the Smoothing feature pre-processes a G-code program and analyzes the sharp, jerky lines of the G-code. Smoothing then lofts arcs through this G-code, allowing a smooth and continuous tool machine motion through the data. With the Smoothing presets and accompanying parameters outlined below, the user can control if larger features are smoothed and how much arc rounding will be applied to the G-code tool path. These parameters allow fine control of Smoothing so that it is possible to achieve the desired results. Below is one example of how Smoothing “corrects” bad CAD/CAM G-code. Notice that the large 90 degree feature is not affected, only the small, sharp G code features are rounded.

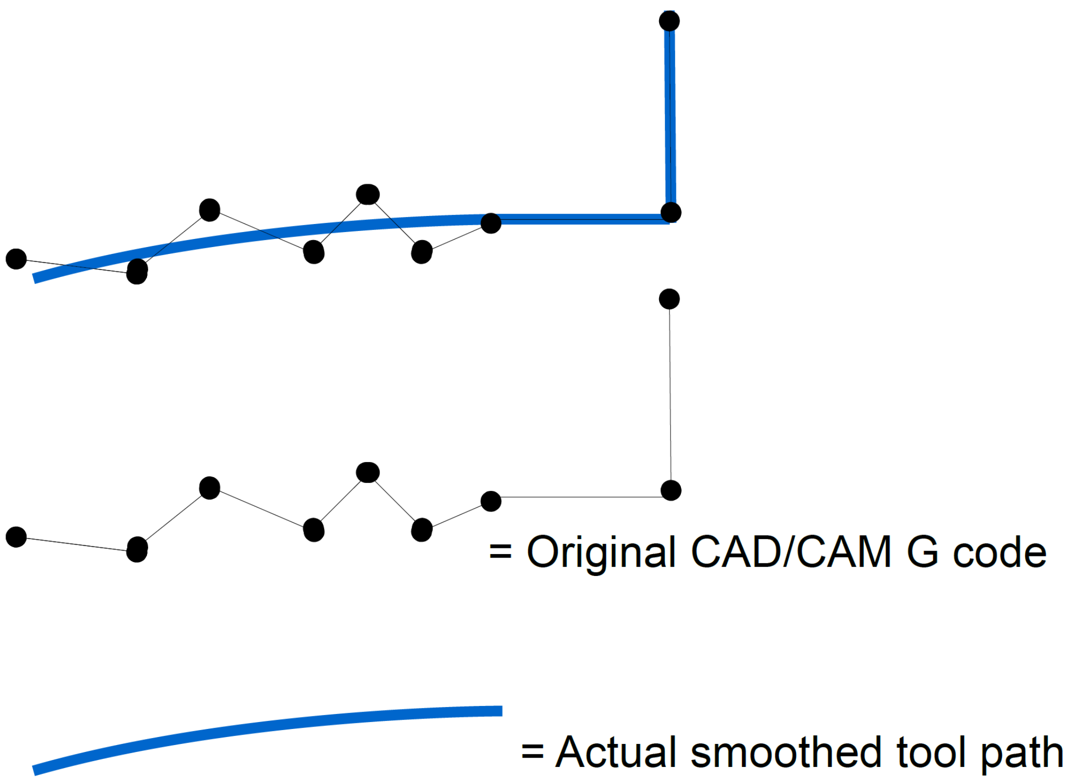


Figure 15.1

15.12.2 Types of G-code programs that work well with Smoothing

3D surfacing and V Carve programs, like the one below by Scott (aka “Sword”), benefit greatly from Smoothing. Check out his thread on the forum where he shares his Smoothing settings and strategy.

https://centroidcncforum.com/viewtopic.php?f=57&t=3021

Figure 15.2: Image credit to Scott (aka “Sword”) on the Centroid forums

15.12.3 Smoothing Requirements:

1.) Steps per revolution must be set to 1600 or higher and MATCH the step setting on the drive. Refer to the thread referenced above.

2.) Overall Turns Ratio must be set properly. Refer to Tech Bulletin #36

3.) Backlash must be kept to a minimum. If lash is over .001", then do not use Backlash compensation (set it to zero). Refer to Tech Bulletin #37.

4.) A CNC PC that at least meets the minimum single core benchmark requirements. Smoothing will not work on slower computers.

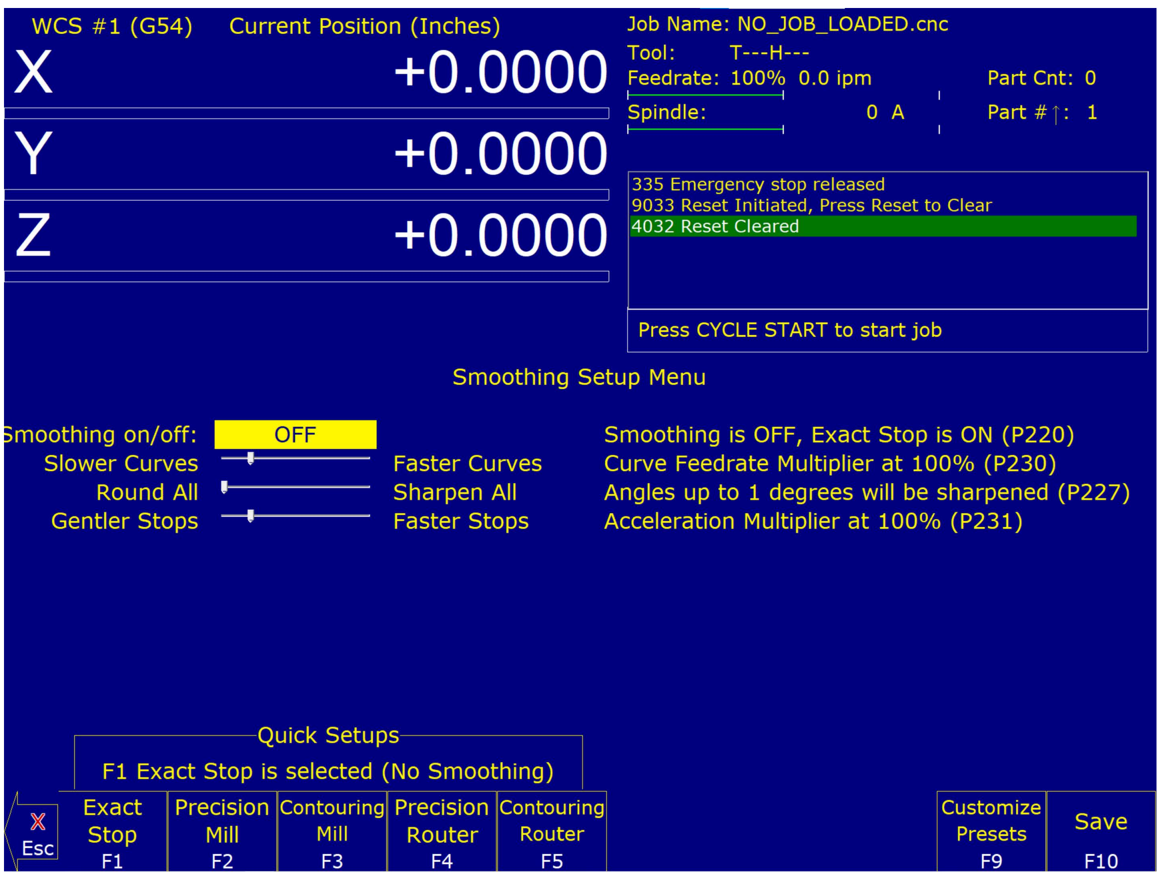


Figure 15.3

15.13 Smoothing AD2 Setup Menu

Pressing **F8 – Smoothing Setup** from the Setup menu will bring up the Smoothing Setup Menu. The Smoothing Setup Menu provides a simplified way of choosing parameters for the Smoothing module. Smoothing is especially useful in controlling and minimizing the amount of “banging” a machine experiences as it proceeds along the toolpath. Smoothing is also able to (optionally) round-off part geometry, allowing for a faster feedrate around corners.

F1 will turn off Smoothing. The other quick setup keys **F2–F8** (if displayed) will turn on Smoothing and activate a Smoothing preset profile. This is done by the preset profile automatically setting parameter 220’s value to 1 and copying its preset values into parameters 221–231.

By default, the four following presets are provided:

F2 – Precision Mill chooses Smoothing with settings for high-precision work on a Mill.

F3 – Contouring Mill chooses Smoothing with settings for a Mill, with looser tolerances for rounded geometry.

F4 – Precision Router chooses Smoothing with settings for high-precision work on a Router.

F5 – Contouring Router chooses Smoothing with settings for a Router, with looser tolerances for rounded geometry.

Presets can be modified, added, or removed by pressing **F9 – Customize Presets**, which will allow access to the menu described in the next section.

When Smoothing is turned on with a preset in the Smoothing setup menu, Smoothing is ON all the time, which may or may not be desired. To further control Smoothing, use G64 to turn Smoothing on and off at any point in the G-code program.

15.13.1 Adjusting Stock Preset Values to Make a Custom Preset

A standard set of default Smoothing Presets and corresponding values have been developed: Precision Mill, Contouring Mill, Precision Router and Contouring Router. These are a good starting place and will work out of the box for most G-codes. Additionally, the Smoothing values are adjustable for these presets to meet the type of performance being looked for with a particular machine and/or type of G-code. For instance, using the same set of Smoothing values for a high-speed 3D CNC router that is cutting foam compared to a VMC machining an injection die mold out of tool steel would not be ideal. These two jobs have different requirements and the Smoothing values can be adjusted to satisfy them both.

15.13.2 Experimenting with Smoothing Parameter Values

These values can be manually adjusted in the CNC12 parameter configuration menu or by using the slider bars in the Smoothing menu.

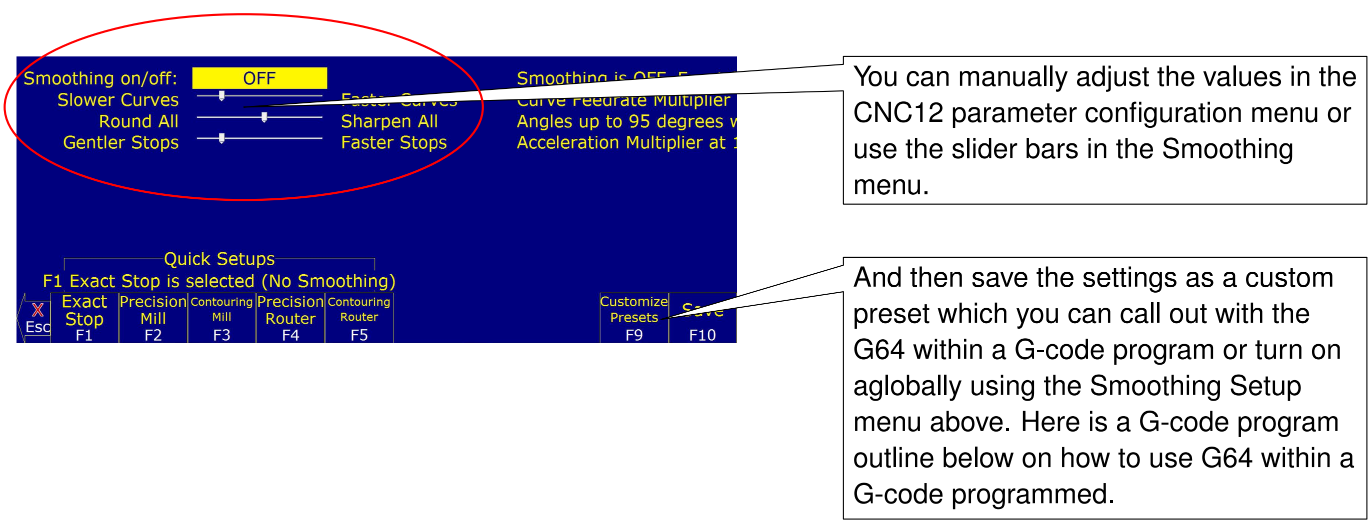


Figure 15.4

Once finished, save these settings as a custom preset. This preset can be called with G64 within a G-code program or by turning the preset on globally using the Smoothing Setup Menu described above. A G-code program is outlined below on how to use G64 within a G-code program.

```
       ; Insert all of the gcode here that smoothing should not be on for
G64 ON ; turns on Smoothing, using the Values contained in the Smoothing
   parameters
```

or

```
G64 "my preset" ; will activate any saved Smoothing preset by using its name in
    quotes or simply its             ; number
                ; Insert all of the gcode here that smoothing should be on for
G64 OFF         ; turns off Smoothing
```

See G64 – Smoothing Mode Selection / Cancel Modal Decel and Stop for more examples of how to use G64.

15.14 Custom Smoothing Presets Menu

Pressing **F9 – Customize Presets** from the Smoothing Setup menu will bring up a screen that allows customization of the Quick Setup keys that appear in the Smoothing Setup menu.

There are a total of 99 possible Smoothing presets. Each Smoothing preset consists of a customizable label and set of parameter values. These will automatically be copied to the parameters P221 through P231 (excluding P229) when such a preset is selected in the Smoothing Setup Menu or activated by the "G64 ON" command. Smoothing presets #1–7 are the only presets that can appear in the Smoothing Setup menu and correspond to F2 through F8 in that menu. For these presets (#1–7), only those that have a non-blank F-key Label will appear and be available in the Smoothing Setup Menu. To remove a particular preset from the Smoothing Setup Menu (presets #1–7), delete its label and leave it blank. All Smoothing presets can be activated by "G64 ON" in a CNC program or MDI. Presets activated this way will still be in effect even after the CNC program is finished.

Here are some G64 examples:

```
G64 ON P1                 ; Activate Preset #1 by number
G64 ON "precision mill"   ; Activate Preset #1 by label
G64 ON P2                 ; Activate Preset #2 by number
G64 ON "contouring mill" ; Activate Preset #2 by label
```

The "G64 ON" command with a "P" number can be used to activate those presets that do not have a label. The Smoothing Presets should not be changed without consulting your dealer. Corrupt or incorrect values could cause damage to the machine, personal injury, or both.

15.14.1 When Not to Use Smoothing

Typically, Smoothing is not necessary for most shop-type 2.5D G-code jobs that use large G1 lines and G2/3 arcs. These jobs include but are not limited to: circular pockets, rectangular pockets, all types of drilling and tapping, line and arc milling (on a milling machine at speeds under 100 ipm), and thread milling. Avoid having smoothing on during drilling, tapping, threadmilling, and circular pocket finish passes since, by definition, Smoothing alters geometry and can have unexpected results if it is enabled when it is not necessary. These types of operations do not consist of many short vector movements, therefore they do not need to be smoothed out.

15.14.2 Typical Smoothing Values for a Precision Milling Machine

(where maximum accuracy and a little G-code smoothing is desired)

```
Smoothing on/off    ; P220 set to 1 (turns on Smoothing globally, see G64 for
   how to turn on
                    ; Smoothing selectively)
Nbpts               ; P221 set to 1
Step                ; P222 set to .001" to .0005" inch, ~.0254mm- .0127mm (set
   to .001" for mold
                    ; work)
Umax                ; P223 typical values are 800-1000 depending on CNC PC CPU
   performance.
Centripetal         ; P224 set to 0
W                   ; P226 set to 10
Min Angle           ; P227 set to 95 (91 to 100, 95 is typical)
S curve             ; P228 set to 0 (turns off S-Curve acceleration)
Accell Multiplier   ; P231 = 1
Lash comp type      ; P216 set to 1 (turns on backlash compensation, make sure
   mechanical backlash
                    ; is under .001")
Accel in PID screen ; Set to .25 to .35 for all axes.
```

15.14.3 Typical Smoothing Values for a Contouring Router Creating Artwork

(where maximum speed and smoothness are paramount, and a little rounding of all sharp corners is not a concern)

```
Smoothing on/off    ; P220 = 1 (turns on Smoothing globally, see G64 for how to
    turn on smoothing
                    ; selectively)
Nbpts               ; P221 = 20 (experiment with 1-20 results will depend on
   data)
Step                ; P222 = .01" (~.254 mm)
Umax                ; P223 typical values are 800-1000 depending on CNC PC CPU
   performance
Centripetal         ; P224 =0
W                   ; P226 = 20
Min A               ; P227 = 1 (round-off all corners)
S curve             ; P228 = 0 (turns off S-Curve acceleration for routers)
Accell Multiplier   ; P231 = 1
Lash comp type      ; P216 set to 0 (turns off backlash compensation to
   increase smoothness)
Accel in PID screen ; Set to .5 for all axes
```

Note: Accel values in PID work with Smoothing as well. Increase these values for softer transitions from one straight line feedrate to another.

15.15 Technical Background of AD2 Smoothing

Smoothing performs several related functions:

15.15.1 Smoothing NBPTS (P221) and STEP (P222)

These parameters control smoothing of the user-supplied G-code. Smoothing allows significantly higher feedrates to be achieved while reducing vibration, bumps, and bangs at corners and angles. It is also great for smoothing-over CAD/CAM generated data with peculiar features (see Figure 15.5). Smoothing's strength is also a potential disadvantage, as it modifies geometry and rounds corners (see Figure 15.6). Users will want to run Smoothing when working with rectangular Z-movements created by "breakout tabs" on a router job. Smoothing will allow the job to run at high-speed right through the breakout tabs, if the min angle P227 is set to less than 90 degrees.

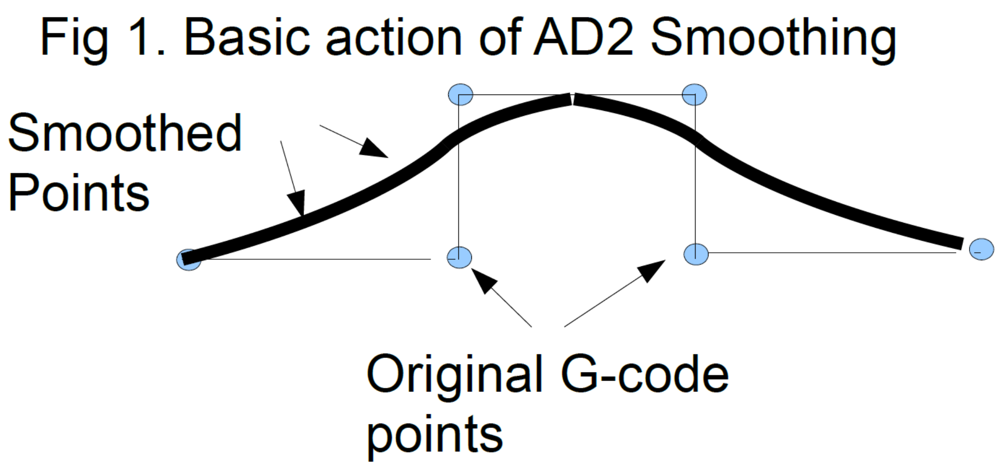


Figure 15.5

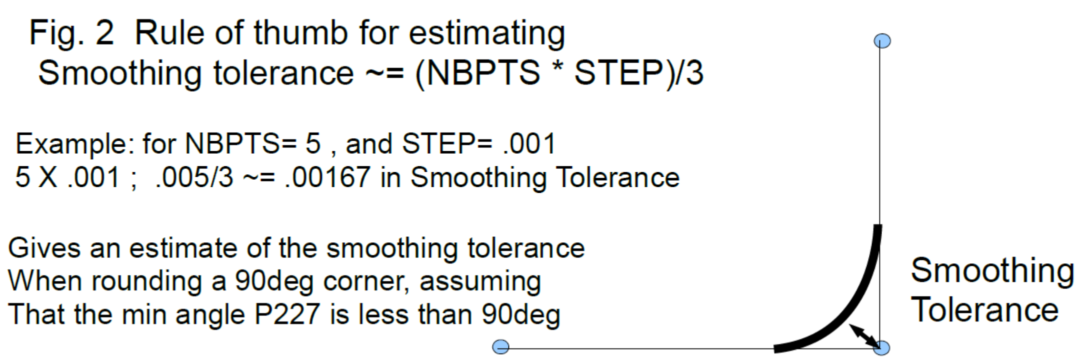


Figure 15.6

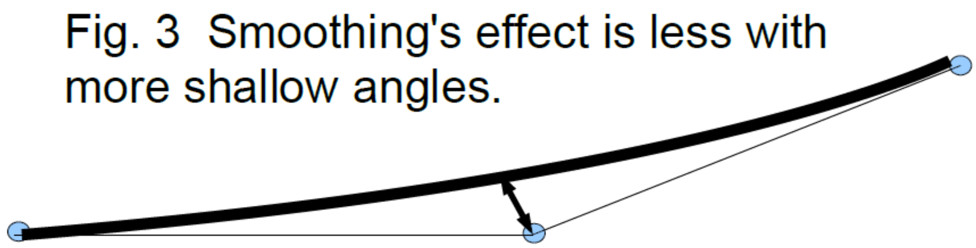


Figure 15.7

Min_Angle (P227) defines the minimum angle to apply Smoothing to. All angles below the minimum angle value will remain sharp. For example, if Min Angle is set to 95 degrees, then all angles less than a 95-degree angle (including a 90-degree right angle) will remain "sharp".

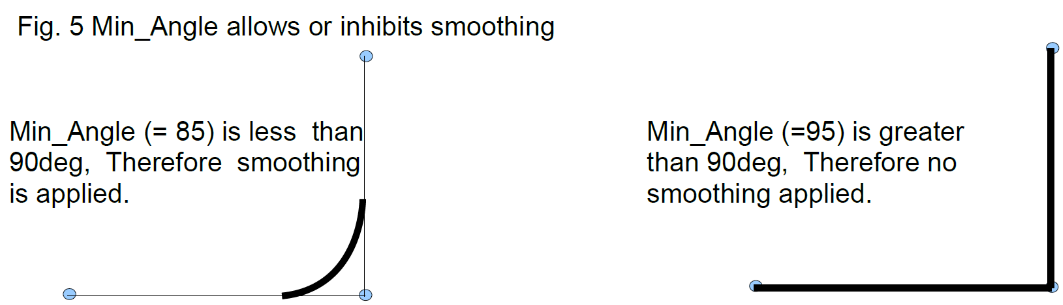


Figure 15.8

15.15.2 Feature width W (P226)

W and Min_Angle work together to determine which angles will be "sharp". For example, a G-code file may contain small spikes, double-backs, or zig-zags of less than 1mm that may be causing unwanted slowdowns in an otherwise high-speed stretch of toolpath. Given STEP (P222) = .25mm, setting W (P226)= 4 (4*.25=1mm) should reduce or eliminate deceleration across the problem toolpath. W itself does not smooth the offending data, that is the job of Smoothing (controlled by NBpts and STEP). W does allow minimization of slowdowns caused by small features, which is very helpful for running smoothly through jagged CAD/CAM generated G-code.

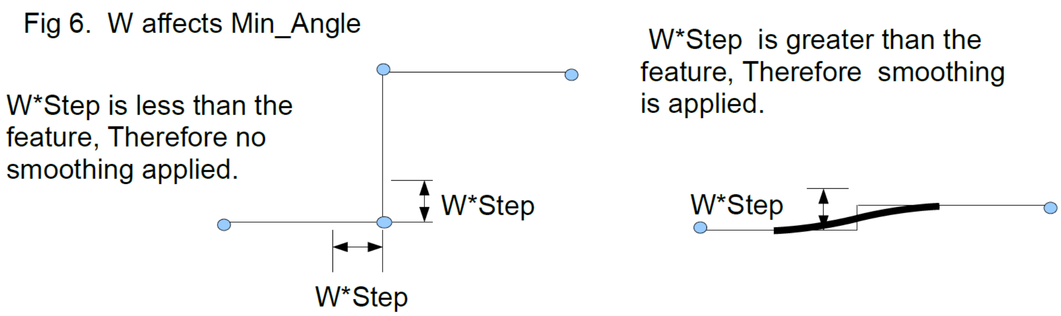


Figure 15.9

S curve (P228) The S-curve feature changes the acceleration rate more slowly. Its effect is most noticeable during changes of direction. On a router where max speed is desired, turn S curve OFF. On a milling machine where max accuracy is desired, turn S curve ON to eliminate as much vibration as possible

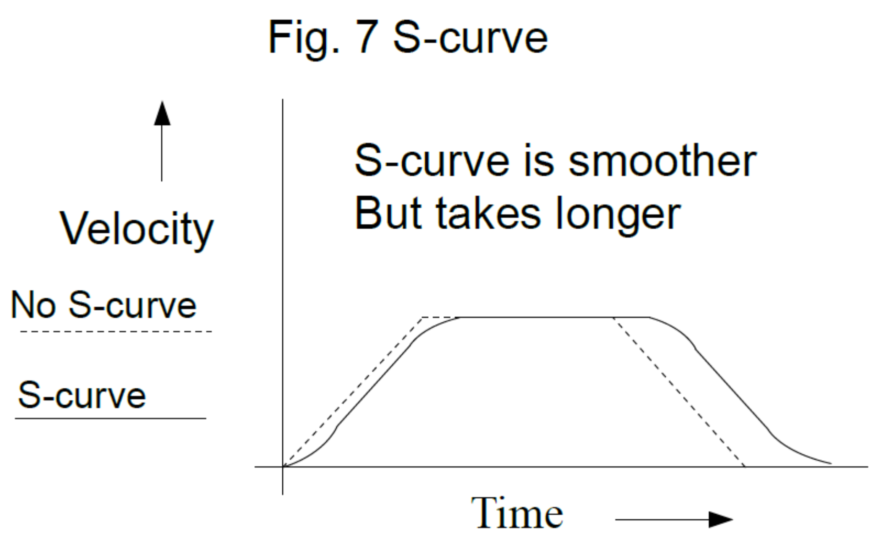


Figure 15.10

15.16 Wirelesss MPG Installation

System Requirements:

- WMPG-4 or WMPG-6 control pendant
- CNC12 v4.64+ with Mill Pro/Ultimate, Lathe Pro, or the Digitizing Bundle
- A CNCPC that meets the requirements for running the CNC12 software

An optional add-on for Centroid CNC machines, the WMPG allows the operator to conveniently and precisely set up jobs and tools remotely.

No software drivers to install. Plug and Play with Centroid CNC12 software. Just plug in the included USB transmitter/receiver and start using it. Perfect for all types of Mills, Lathes, Routers, and other specialty CNC machines.

Installation Instructions for Acorn/AcornSix Operators:

1. Remove MPG finger knob from battery compartment and install onto MPG wheel.
2. Install two good-quality AA batteries. Warning! Cheap NiMH/NiCd rechargeable batteries are not recommended as they are rated at 1.2 volts. Good quality 1.5 volt rated AA batteries are required.
3. Plug USB transmitter into USB port on the CNCPC. Reboot CNCPC at this point.
4. Start CNC12 and verify that a Pro or Ultimate License is installed. Open Centroid Wizard Tab #4 "Control Peripherals" and select "WMPG-4". Press Write Settings to CNC control Configuration. Follow instructions on the screen.
5. Turn on MPG by depressing and releasing the large chrome ON/OFF button on the MPG.
6. Start CNC12 and home the machine.
7. Select which axis to move, X, Y, Z or 4th, with the axis selection knob.
8. Select the increment desired x1, x10 or x100.
9. Rotate the MPG wheel clockwise for positive direction axis motion.

Installation Instructions for Oak, Allin1DC, and MPU11 Operators:

1. Remove MPG finger knob from battery compartment and install onto MPG wheel.
2. Install two good-quality AA batteries. Warning! Cheap NiMH/NiCd rechargeable batteries are not recommended as they are rated at 1.2 volts. Good quality 1.5 volt rated AA batteries are required.
3. Plug USB transmitter into USB port on the CNCPC. Reboot CNCPC at this point.
4. Start CNC12, and open CNC12 Utility Menu, press "Options" and "Import License" and install Pro or Ultimate license
5. F1 Setup, F3 Config, F3 Parameters (pass is 137). Then set MPG CNC12 parameter #218 = 15 for 4 axis Mills/Routers, #218=7 for 3 axis Mills/Routers and #218 = 3 for Lathes.
6. Set MPG CNC12 parameter #348 = 15 (MPG ON) and #349 & #350 = 100 (100 steps per rev), Set # 411 = 1 (WMPG-4) F10 Save. Exit out of Parameters.
7. Shut Down and restart CNC12 for new parameters to take effect.
8. Turn on MPG by depressing and releasing the large chrome ON/OFF button on the MPG.
9. Select which axis to move, X, Y, Z or 4th with the axis selection knob.
10. Select the increment desired x1, x10 or x100.
11. Rotate the MPG wheel Clockwise for positive direction axis motion.

Notes:

1. MPG wheel finger knob is located in the battery compartment. Remove and install onto the MPG wheel.
2. Only use AA batteries rated at 1.5 volts. There are rechargeable AA batteries on the market now with higher and lower voltage ratings, do not use these types of batteries.

3. Keep the USB Transmitter / Receiver at least 3 feet AWAY from the MPG itself. WMPG Operational range is 3'–100'
4. LCD Screen has a protective plastic film on it, use clean fingernail to lift up at corner and remove.
5. "No RF" message means MGP is not seeing the transmitter/receiver, most common cause is the Windows PC did not mount the USB transmitter. Solutions: Move to different USB port and reboot PC.
6. For Acorn/AcornSix: Value for the x1 Jog Increment for the MPG wheel is set in the Wizard Axis configuration menu, Default value for Plasma and Router is .001" or 0.0254 mm. For Mill and Lathe Default value is .0001"
7. For Oak/Allin1DC/MPU11: Z axis speed limiter: Inhibit fast Z axis MPG moves with P450 = 1 =ON x100 = x10,P450 = 0 =OFF x100 = x100 (v5.0+)

15.17 Custom WMPG Macros

15.17.1 Common Uses:

1. Set X and Y zero position with one button push
2. Move XYZ or XY to a predetermined position.
3. Turn on or off an output to control anything that you would control with an output, lights, dust collection, lube pump shot, vac hold down, etc...

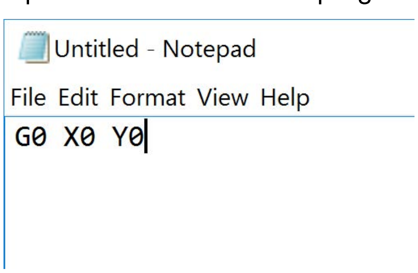
15.17.2 Three easy steps.

1. Create a G&M code program that you want to run when an Auxiliary key is pressed.
2. Save it as a Custom M-code using the proper custom M-code name.
3. Map the new Custom M-code to a spare Auxiliary key using Auxiliary key parameters.

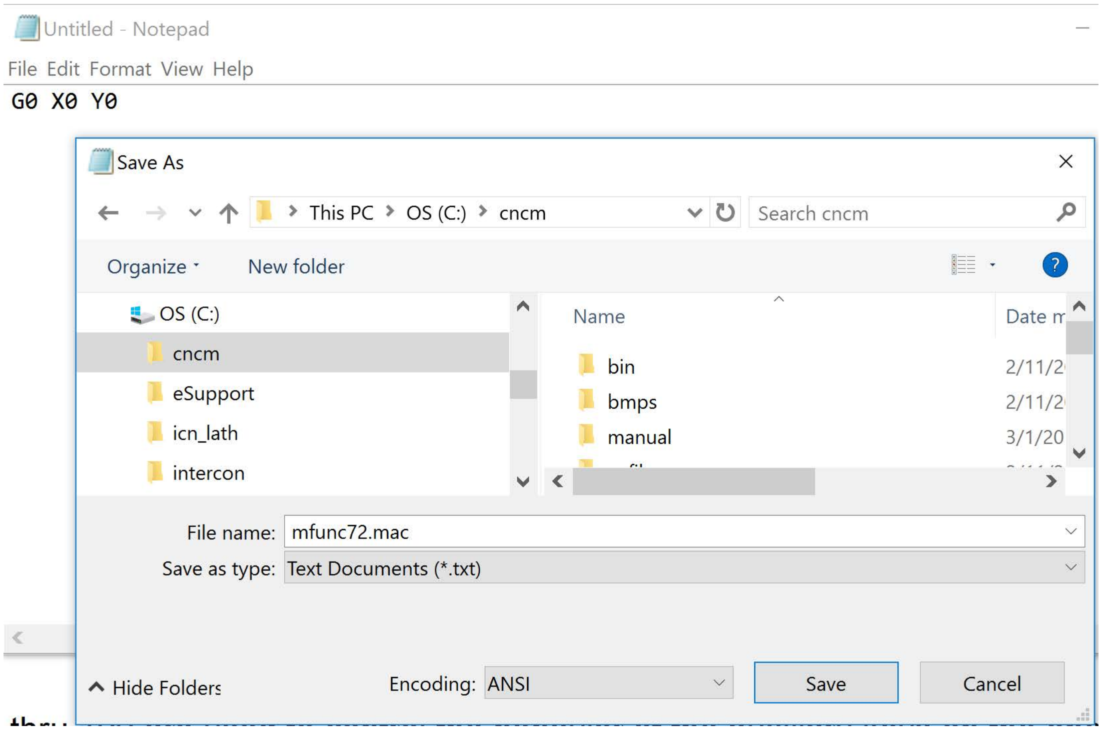
Example: Open any text editor and create the G&M program you would like to execute when pressing an Aux key. For this example we will 'program' Aux key #5.

For this example we will be creating a custom M-code program to move the X and Y axis to X0.0000 Y0.0000 position by pressing Aux key 5.

1. Open editor and create a program with G0 X0 Y0 as the first line

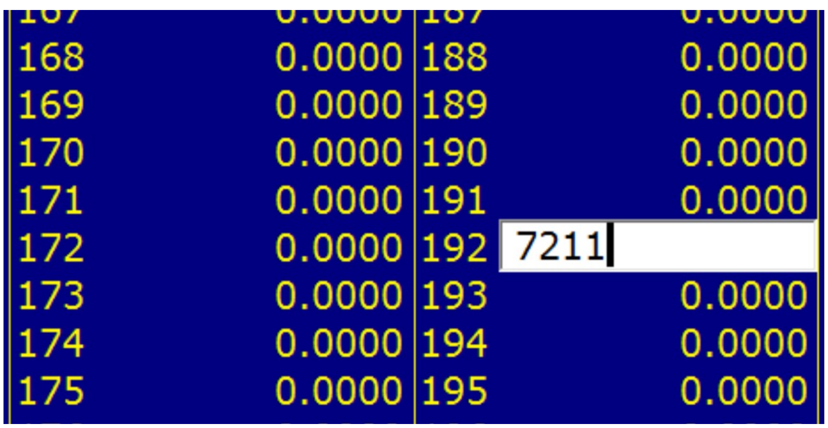


2. Save this G&M code program with the file name mfunc72.mac, save in the cnc11/12 installed directory, typically c:\cncmc:\cnct.



3. **Map the custom M code to an Auxiliary key**. F1 Setup, F3 Config, F3 parameters. Navigate to P192. Set P192 to 7211, press F10 to save. 72 is the custom M-code # to be used and 11 tells the control you want the Aux key to use a custom Mcode.

   Note: Parameters 188 thru 199 are used to control the behavior of the auxiliary keys on the operator panel and map them to the custom M code file. P192 controls the behavior of Aux 5.



4. **Test**. Verify the machine is ready to move to X0, Y0. Now test the feature by pressing Aux5 at the main screen. The machine should make a rapid move to X0, Y0

Notes:

1. The M72 custom M-code we just created will work like any other M code and can be called out in a G and M code program or using MDI mode as well.
2. You can not use the Aux keys to set bits directly using M94/M95. You can however create a custom macro that calls M94/M95 to set a bit.
3. Many machines like ATC machines will already be using several custom M-codes. When creating a new custom M-code, Do not overwrite these existing custom M-code programs. You can see all custom M-codes in the

c:\cncm+ or c:\cnct directory. When creating a new custom M-code, choose a Custom M-code number that is not already being used.

4. Before creating a new custom M-code, create a backup of the existing cnc control configuration (which includes the current M-codes) by creating a report by pressing F7 Utility, F7 Create report from the main menu. You can always “restore” the report to bring the machine back to the original configuration.

5. Once you have created custom M-codes and are satisfied with their functionality. Generate another report to save a backup of your machine with the new M-codes.

15.17.3 Using the Canned Auxiliary Key Canned Functions

Auxiliary key parameters # 188 thru 199 also have “canned” functionality that can be activated by simply setting the parameter value to any of these values below.

Function	Set Aux Key parameters #188 (Aux1) thru #199 (Aux 12) to any of these values
No function	0
Input X Axis position	1
Input Y Axis position	2
Input Z Axis position	3
Set Absolute Zero	4
Set Incremental Zero	5
One shot – Drill	6
One shot – Circular Pocket	7
One shot – Rec. Pocket	8
One shot – Frame	9
One shot – Face	10
Execute M-code file	(mcode#)11
Free Axis (remove power)	14
Go to power feed menu	15
XYZ set Absolute Zero	16
One Shot – Drill BHC	17
One Shot – Drill Array	18

Centroid CNC Acorn Control The Acorn has default macros for turning on and off general purpose outputs as listed below.

- M61-M68 turn on OUTPUT1-OUTPUT8 respectively M94/61-M94/68
- M81-M88 turn off OUTPUT1-OUTPUT8 respectively M95/61- M95/68

You will need to use the Wizard to assign your outputs in the PLC

15.17.4 Wireless MPG Macros

Allin1DC/Oak

Wireless MPG Macros are found in the `c:\cncm\system` or `c:\cnct\system` directory. Installations of the CNC12 V4.16 and later come with example macro files, named plcmacro1.mac, plcmacro2.mac, plcmacro3.mac and plcmacro4.mac for the wireless mpg aux keys 1, 2, 3 and 4. If you are missing these examples files, copy the example at the end of this Tech bulletin into your editor and save as plcmacro1.mac in your `cncm\system` directory.

The default function of the macro will be to display the following message.

"This is an example macro run from the Macro1 button on the centroid CWP-4 MPG. The macro is named plcmacro1.mac and can be found in the `\cncm\system` directory. Edit to include the desired functionality. Press ESC or Cycle Cancel to Exit."

Editing the file to include desired functionality is similar to other macro functions, except that the code must be between lines N100 and N1000 for proper functionality. Additionally, deleting or commenting out the M225 message line is required to disable the example message from appearing when pressing the aux keys. For example, if we wanted the mpg key 1 to bring x and y axis back to zero like the earlier example we would edit plcmacro1.mac and add the line "G0 X0 Y0" between N100 and N1000.

```
N100
G0 X0 Y0
N1000 ; end
```

Acorn

Acorn systems allow easy access to the plc function file via the acorn wizard. The Acorn wizard can be opened by running the CNC software, Utilities (F7), then Acorn Wizard (F10). In the wizard navigate down to Control Peripheral and then Wireless MPG. Under Macro Button Configuration, pressing any one of the 4 "edit MPG macro" buttons will open up a text editor of the macro code for each individual button.

Example Wireless MPG Macro:

```
;------------------------------------------------------------------------------
; Filename: plcmacro1.mac
; Description: User Customizable Macro
; Notes:
; Requires: Machine home must be set prior to use.
; Please see TB300 for tips on writing custom macros.
;------------------------------------------------------------------------------
IF #50010                          ; Prevent lookahead from parsing past here
IF #4201 || #4202 THEN GOTO 1000 ; Skip macro if graphing or searching
N100                               ; Insert your code between N100 and N1000
N1000                              ; end
```

16 CNC Software Messages

16.1 CNC Software Startup Errors and Messages

Error	Message	Cause & Effect	Action
102	Error initializing CPU. . . cannot continue.	Error sending a hex file program to the motion control card.	Inspect MPU11 connection, or fix missing or corrupted hex file. Contact Dealer
103	Error sending setup	Error sending the current setup parameters to the motion control card.	Inspect MPU11 connection, or fix missing or corrupted hex file. Contact Dealer
104	Error sending PID setup	Error sending the current PID setup parameters to the motion control card.	Inspect MPU11 connection, or fix missing or corrupted hex file. Contact Dealer
105	mpu.plc file read error..cannot continue	Error sending the current PLC program to the motion control card.	Install or recompile PLC program. Contact dealer
106	The PC clock appears to be wrong	Error while reading the temperature file. The time on the PC internal clock is earlier than the time recorded in a previously stored file	
199	CNC started	CNC control software has started.	

16.2 Messages Issued Upon Exit From CNC Software

Error	Message	Cause & Effect	Action
201	Exiting CNC due to a known error (ACORN only)	MPU11 not responding, or mpu11.hex, mpu.plc is missing or damaged.	Check for possible software corruption. Contact dealer
202	Exiting CNC due to a math error	A floating-point math error occurred. Possible corruption of cnc.tem, cncm.job, or cncm.wcs.	Delete corrupted files and reboot software. Contact dealer
204	Exiting CNC. . . Normal Exit	CNC control software is shuting down normally.	
222	Autotune run	Autotune has been run.	

16.3 Messages and Prompts in the Operator Status Window Status Messages

Error	Message	Cause & Effect	Action
301	Stopped	A job has ended normally or the operator has aborted the job.	
302	Moving. . .	Motors are moving while a CNC program is running.	
303	Paused. . .	Motion is paused while a CNC program is running (FEED HOLD)	
304	MDI. . .	CNC software running in MDI mode	
305	Processing. . .	CNC software running in a mode other than MDI	
306	Job Finished.	Normal end of CNC program	
307	Operator abort: job canceled	ESC or CYCLE CANCEL pressed. Job is cancelled.	
308	Waiting for input #NN	M100 or M101 executing. Program will continue once specified input opens or closes.	
309	Waiting for CYCLE START button	M0, M1, M100/75, or Block Mode is executed.	Press Cycle Start
310	Waiting for output #NN	M100 or M101 executing. Program will continue once specified output opens or closes.	
311	Waiting for memory #NN	M100 or M101 executing. Program will continue once specified memory bit changes to the correct state.	
312	Waiting for PLC operation (Mnn)	PLC program not clearing PLC operation in progress	
313	Waiting for dwell time	G4 executing. Program waits for specified dwell time then continues.	
314	Waiting for system #NN	M100 or M101 executing. Program will continue once specified PLC system variable changes to the correct state.	
315	Searching. . .	Run/search in progress	
317	Waiting for automatic tool change	mfunc6.mac executing	
318	Operator Abort probing cancelled	ESC or CYCLE CANCEL pressed while doing a probing move	
319	Probing cycle cancelled	probing cycle was cancelled	
320	Probe stuck	probe is stuck, or probe hit an object when it wasn't expecting contact.	
322	Stall: probing cancelled	probing was cancelled because of a stall	
323	Stall: job cancelled	job was cancelled because of a stall	
324	Limit: probing cancelled	probing was cancelled because of a limit error	
325	Limit: job cancelled	job was cancelled because of a limit error	
326	Fault: probing cancelled	probing was cancelled because of a fault	
327	Fault: job cancelled	job was cancelled because of a fault	
328	Cutter comp error: job cancelled	job was cancelled because of a cutter comp error	
329	Invalid parameter: job cancelled	job was cancelled because of an invalid parameter	
330	Canned cycle error: job cancelled	job was cancelled because of a canned cycle error	

Error	Message	Cause & Effect	Action
332	Search Failed	Run/Search was unable to find the requested G-code line	
334	Locating position to resume job...	Run/Search is locating the job continuation point in the program	
335	Emergency Stop Released	Emergency Stop Button has been released	
336	Digitize cancelled	ESC or CYCLE CANCEL pressed during digitizing	
337	Digitize complete	A digitizing routine ran to completion	
338	Job Cancelled	ESC or CYCLE CANCEL pressed during job run	
339	Jogging...	An axis jog key is pressed and machine is moving the corresponding axis	
340	Limit (#__) cleared	A previously tripped limit switch is now in the "untripped" position	
341	Probing Cycle Finished	A probing cycle ran to completion	
342	Waiting for motion to stop	PC is waiting for the MPU11 to complete motion	
343	Waiting for stop reason reset	PC is waiting for the MPU11 to reset the stop reason (as part of the PC/MPU11 communications handshake).	
344	Feedrate modified due to spindle	The effective feedrate has been lowered because the spindle is spinning slower than the threshold percentage of the commanded spindle speed. (The threshold percent is specified in P149.)	
345	Waiting for spindle to get up to speed	Job progress is paused until the actual spindle speed reaches the threshold percentage of the commanded spindle speed. (The threshold percentage is specified in P149.)	
346	Waiting for spindle direction	Job progress is paused until the spindle turns the commanded direction.	

16.4 Abnormal Stops (Faults)

Abnormal stops are detected in the following order: PLC, servo drive, spindle drive, lube, ESTOP. This means that if both the servo drive and the spindle drive have faulted, the servo drive fault message would appear.

Error	Message	Cause & Effect	Action
401	PLC failure detected	MPU11 stopped with PLC failure bit set. Job cancelled.	Check PLC fibers and PLC logic power.
404	Spindle drive fault detected	MPU11 stopped with spindle drive fault bit set. Job cancelled.	Check inverter for fault or reset spindle contactor OCR, then cycle EMERGENCY STOP

Error	Message	Cause & Effect	Action
405	Lubricant level low	MPU11 stopped with low lube fault bit set. Current job will finish but nothing will work after that.	Add lube or check low lube switch wiring then cycle EMERGENCY STOP
406	Emergency Stop detected	MPU11 stopped with no fault bits set. Job cancelled.	Release Estop
407	limit (#__) tripped	MPU11 stopped with limit switch tripped. Job cancelled.	Clear limit switch
408	Programmed action timer expired	M103 time expired before M104 encountered. Job cancelled.	Find out why timer expired before specified action was completed.

Error	Message	Cause & Effect	Action
410	_ axis position error	A position error $>$ one motor encoder turn is detected on any axis. All axis motion is stopped, power to the motors is released (all servo drive commands cease) and the CNC program is aborted. The probable causes of this error are: 1. The motor is wired up backwards. 2. Noise is getting into the system via the motor cables (the line integrity has been violated). 3. An encoder error occurred.	1. Try to slow jog the motor and watch the DRO position. If the position on the DRO goes opposite the direction indicated on the jog button, then the motor is wired up backwards. Change the motor wiring. 2. Check the motor cabling paying particular attention to the ground connections. Replace the cable if it is damaged or repair the motor connections. 3. Jog the motor awhile, at the maximum rate, using the fast jog buttons. (Check the fast jog rate in the motor jog parameters screen to make sure it is set equal to the maximum motor rate.) If the motor seems to jump around rather than accelerate and decelerate smoothly then you are probably fighting an encoder error. Swap the motor with one from another axis and see if the error follows the motor. If it stays with the axis, replace the MPU11. If it follows the motor, replace the motor cable. If the problem still persists, replace the motor and encoder.

Error	Message	Cause & Effect	Action
411	_ axis full power without motion	90% Power (PID Output $>$ 115) is applied to any axis and no motion $>$0.0005 inches is detected, for more than the time specified in Parameter 61 (default .5 sec.). All axis motion is stopped and the CNC program is aborted. The probable causes of this error are: 1. One of the axes is against a physical stop. 2. The servo drive has shutdown due to a limit switch input. 3. The Z home switch is the same as the Z + limit switch.	1. If the axis has run into a physical stop, use the slow jog mode to move the axis away from the stop. Determine and set software travel limits to stop machine before in runs into the hard stops. 2. If the axis is not on a physical stop, check for a tripped limit switch. If it is then the software is commanding a move into the switch but the hardware is shutting the move down. Go to the motor setup screen and enter the limit switch input number if applicable. 3. Make sure the switch input is not unstable or noisy. If it is then replace the switch. If the problem persists it may be necessary to create separate home and limit switch inputs. Use slow jog to move opposite the direction causing the error and clear all limit switches. Jog toward the direction causing the error, if no motion occurs then a servo drive failure is indicated.
412	_ axis encoder differential error	An error condition was detected in the differential signal levels for this axis encoder. May indicate a loose or severed encoder cable or a bad encoder. This will stop all motion and cancel the job.	Reconnect/replace encoder or encoder cable.
417	Abnormal end of job	Job ended without reason.	
418	Search Line or Block not found	Requested search input data not found in loaded CNC file.	Type in correct data or load correct job.
419	Search line in embedded subprogram	Requested search line is found, but is part of an embedded/extracted subprogram	Use another line number

Error	Message	Cause & Effect	Action
420	_ axis motor overheating	CNC software estimates that a motor has reached the warning temperature (set in Parameter 29). Motor is overheating or the temperature file is corrupted. Job will be cancelled.	Contact dealer. Determine what's causing motor to overheat or delete cnc.tem file and reboot.
421	Motor(s) too hot: job canceled	CNC software estimates that one or more motors have reached the limit temperature (set in Parameter 30). Will not be able to run until motor cools down.	Contact dealer. Determine what's causing motor to overheat or delete cnc.tem file and reboot.
422	Check Jog Panel cable	Jog panel failure or loose cable.	Reconnect jog panel cable.
428	Check MPG cable	MPG failure, loose cable, or was turned off.	Reconnect MPG cable and turn axis selector knob to an axis.
434	_ idling too high: Releasing power	Axis is not moving and no job is running but axis has stopped against some abnormal resistance. Power is released to motors.	Run an autotune to adjust motor settings.
435	_ axis runaway: Check motor wiring	Motor was in a runaway fault condition. Power to motor will automatically be shut off.	Check motor wiring
436	Servo drive shutdown	"This error message is produced by hardware detection of a physical error. The servo drive hardware generates this error message if it detects either an overcurrent or overvoltage condition. The particular hardware condition is reflected on the servo drive LED's. Once the servo drive detects this error condition it stops all motion and removes power to the motors. The hardware indicates the presence of this condition to the CNC software via the servo drive fault input to the PLC."	On DC systems check status of the servo drive LED's and check fibers 4&5. If this message is displayed on an AC system check P178 bit 4 is set.
437	Servo power removed	Axis was moving more than 300 RPM while power was supposed to be off. 1. Motor may be wired backwards. 2. May be a shorted servo drive. 3. Axis motion is canceled but motor continues to move due to inertia, which is probably caused by an unbalanced axis. Power to motors is released.	Check motor wiring, servo drive, or look at Kg value in PID and make sure it's not above +/-5.
438	Spindle slave position error	The slaved axis moved too far in the wrong direction during a spindle-slaved move (such as in rigid tapping). Job is cancelled.	Check Parameter 34 for wrong sign in front of encoder counts.

Error	Message	Cause & Effect	Action
439	_ axis servo drive data output error	Logic power failure or loss of communication from the drive to the MPU11.	Is logic LED on? Check fiber optic cables to drive. For SD1 drives, make sure bus cables are shielded and are as short as possible. Power unit down and check drive connections.
441	_ axis overvoltage	Input power has gone higher than 340VDC and will shutdown the drive and removes power. The motor brake will engage for 5 seconds in this condition.	Check input voltage is below 340VDC. If not, incoming VAC needs lowered.
442	_ axis undervoltage	Drive input power is less than 80 VDC.	Check supply voltage.
443	_ axis commutation encoder bad	Control detected invalid commutation zone value	Preform a motor Move Sync in the Drive Menu. A Zero (0) or Seven (7) is an invalid zone. Check for: 1. Wiring problem in the encoder cable or motor end cap (broken encoder wires). 2. Encoder cable shield connected at motor end, when it shouldn't be. 3. Bad encoder. 4. Motor power cable shields not connected. 5. Drive not grounded properly.
444	_ axis overtemperature detected	Drive overtemp sensor tripped. No motor power.	The drive is being run at over capacity or the cooling fan is either not functioning or its air flow is blocked.
445	_ axis overcurrent detected	Overcurrent detected on an axis. No motor power.	Try to jog the axis. The drive will reset the current limit and try to move the motor. If the error comes back, check for a short in the motor output.
446	_ axis servo drive data input failure	Communication Checksum error. No motor power.	Check fiber optic cables. Verify continuity between drive chassis, ground strip and Earth ground.

Error	Message	Cause & Effect	Action
447	_ axis (#) bad index pulse detected	Noise picked up by encoder cable or misaligned encoder. No motor power.	Remove noise or align the encoder.
449	Manual movement detected in restricted area	Unexpected movement of manual axis detected when Z axis summing is active.	Physically lock the Z axis manual quill.
450	Voltage brake applied	Overvoltage condition was detected. Electronic braking was applied by offloading excess voltage to dropping resistors.	Usually this error condition is innocuous even if this message occurs every once in a while in a job. However, if this message occurs in a continuous stream, contact your dealer.
451	Current brake applied	Overcurrent spike was detected on the drive.	Usually this error condition is innocuous even if this message occurs every once in a while in a job. However, if this message occurs too often, it may mean you need a higher current drive. But, if this message appears in a continuous stream, something is seriously wrong, and you should hit E-Stop to cut power to the drive and then contact your dealer.
452	PC Receive Data Error	A fatal communication error occurred between the MPU and PC. The error was detected on the PC side.	Restart the software to clear the error. If this error occurs often there may be an issue with the network configuration or the Ethernet cable.
453	CPU Receive Data Error	A fatal communication error occurred between the MPU and PC. The error was detected on the MPU11 side	Restart the software to clear the error. If this error occurs often there may be an issue with the network configuration or the Ethernet cable.
453	Jogging while probe detected	The probe was in the tripped state while a jog key was pressed.	
454	axis scale encoder differential error	An error condition was detected in the differential signal levels for this axis scale encoder. May indicate a loose or severed encoder cable or a bad encoder. This will stop all motion and cancel the job.	Reconnect/replace scale encoder or scale encoder cable.
455	axis encoder quadrature error	The axis encoder skipped a transition state on its count-up/count-down sequence. May indicate a bad encoder or a loose or severed encoder cable. This will stop all motion and cancel the job.	Reconnect/replace encoder or encoder cable.

Error	Message	Cause & Effect	Action
456	axis scale encoder quadrature error	The scale encoder skipped a transition state on its count-up/count-down sequence. May indicate a bad encoder or a loose or severed encoder cable. This will stop all motion and cancel the job.	Reconnect/replace scale encoder or scale encoder cable.
457	Unable to find home	A commanded move was seeking either an index pulse or a hard stop, but neither was found.	Reconnect/replace encoder or encoder cable if move was seeking an index pulse. Check that hard stop was not broken off nor overrun.
459	TT1 or Probe is not connected	A Tool Measure operation aborted because the required TT1 or Probe is not connected.	Check TT1 or Probe wiring and plug.
460	TT1 and Probe are both connected	A Tool Measure operation aborted because both a TT1 and Probe were connected.	Make sure TT1 and Probe are not plugged in at the same time. Also check wiring.
461	Spindle axis is not set	An operation aborted because the spindle axis parameter (P35) has an incorrect value.	Contact dealer.
462	Triangular Rotary Axis Out of Range	A triangular rotary axis (on a tilt table or articulated head machine) is at a position which is out of range for angular calculation.	Contact dealer.
470	brake wattage exceeded	The brake wattage was exceeded on the indicated ACDC drive.	Contact dealer.
487	Invalid tilt table parameters	One or more values in the tilt table configuration is incorrect.	Contact dealer.

16.5 CNC Syntax Errors

Error	Message	Cause & Effect	Action
501	Invalid character on line NNNNN	Invalid character on CNC line. Job cancelled.	Remove character from program.
502	Invalid G code on line NNNNN	Invalid G code encountered on CNC line. Job cancelled.	Correct invalid G-code.
503	Invalid M function on line NNNNN	Invalid M function encountered on CNC line. Job cancelled.	Correct invalid M-code.
504	Invalid parameter on line NNNNN	Invalid or missing number after letter. Job cancelled.	Correct program.
505	Invalid value on line NNNNN	Value out of range (T, H, D). Job cancelled.	Correct program.
506	Only 1 M code per line	More than one M code appears on the line. Job cancelled.	Move 2nd M-code to next line.
507	No closing quote	The closing quotation mark (") is missing. Job cancelled.	Add quotation.
508	Macro nesting too deep	Macro nesting limit exceeded on attempt to invoke a subroutine. Job cancelled.	Create a second program.
509	Option not available	Attempt to access a locked software option. Job cancelled.	Contact Dealer.
510	Too many macro arg's	Too many arguments were given in a G65 macro. Job cancelled.	Correct number of arguments.
511	Missing parameter	A parameter is required or expected but not found. Job cancelled.	Correct program.

Error	Message	Cause & Effect	Action
513	Expected “=”	Error in expression to left of “=”, missing “=”, or orphaned parameter. Job cancelled.	Correct equation.
514	Empty expression	The expression contains no operands. Job cancelled.	Correct expression.
515	Syntax error in expression	Illegal character in number, variable or function. Job cancelled.	Correct program.
516	Unmatched bracket (parenthesis)	Brackets or parentheses are paired improperly or misplaced. Job cancelled.	Correct program.
517	Evaluation stack overflow	Brackets or parentheses are nested too deeply. Job cancelled.	Correct program.
518	Undefined variable	The variable name does not exist. Job cancelled.	Correct program.
519	Too many variables	The space allotted for user-defined variables has been exceeded. Job cancelled.	Correct program.
520	Invalid variable name	The variable name contains an illegal character. Job cancelled.	Correct program.
521	Divide by zero	Attempt to divide by zero. Job cancelled.	Correct program.
522	Domain error	Imaginary number would result (square root of a negative number). Job cancelled.	Correct program.
523	Invalid value in assignment	Attempt to assign an illegal value to a system variable. Job cancelled.	Correct program.
524	Variable is read-only	Attempt to assign a value to a read-only system variable. Job cancelled.	Correct program.
525	Missing P value	P parameter is expected but is missing	Correct program.
526	M22x Missing initial variable	M224 or M225 was not immediately followed by a #variable reference.	See M224 and M225
527	M22x initial variable parse error	M224 or M225 was immediately followed by an invalid #variable reference.	Correct program.
528	M225 String variable not allowed	M225 was immediately followed by a string #variable (which is invalid). Only numeric variables are allowed here.	Correct program.
529	M225 invalid variable	The #variable specified after the M225 was not valid, or not readable due to a machine error.	Correct program.
530	M224 invalid variable	The #variable specified after the M224 was read-only, or not writeable due to a machine error.	Correct program.
531	M22x missing initial quote	The beginning of the quoted (") format string was not found or was in the wrong place on the G-code line.	See M200, M223, M224 or M225
532	M22x missing end quote	The format string did not end with a quote (")	See M200, M223, M224 or M225
533	M22x embedded quote not allowed	The format string contained a quote (") in the middle of it.	See M200, M223, M224 or M225
534	M22x character limit exceeded	The format string was too long	Correct program.
535	M22x invalid format string	The format string contained invalid format codes	Correct program.
536	M22x missing format specifier	The format code was missing the its specifier	Correct program.

Error	Message	Cause & Effect	Action
537	M22x Missing Argument	A format code was specified in the format string, but its corresponding #variable argument was missing	Correct program.
538	M22x argument parse error	A format code was specified in the format string, but its corresponding #variable argument had a syntax error	Correct program.
539	M22x variable type mismatch	A string format code was specified in the format string, but its corresponding #variable argument was numeric OR a numeric format code was specified in the format string, but its corresponding #variable argument was a string	Correct program.
540	M22x variable cannot be read	A format code was specified in the format string, but its corresponding #variable argument was invalid or there was a machine error when accessing it.	Correct program.
542	M22x character limit exceeded	The resultant formatted string after all the format codes were processed was too long.	Correct program.
543	Missing L parameter	L code was missing	Correct program.
544	Too many axes	More than 1 axis was specified with M128, OR the Simultaneous Contouring feature is not enabled. Without the Simultaneous Contouring feature, a maximum of 3 axes are allowed per G-code line.	Specify fewer axes on the G-code line OR Contact Dealer for information about obtaining the Simultaneous Contouring feature.
545	Value out of range	Parse error occurred because value was out of range	Correct the value
547	Move by counts not allowed	Cutter comp (G41/G42) was on when M128 was specified	Issue G40 (Cutter comp off) before issuing M128
548	String too long	A quoted string was too long (usually a file name was longer than its allowed limit).	Shorten the file name.
549	Line too long	A line in a G/M-code program is too long (more than 1023 characters).	Shorten the line.
550	Invalid L parameter	The value associated with the L code is invalid	Give the correct value.
551	Invalid R value	The value associated with the R code is invalid	Give the correct value.
552	File encryption error	Error while parsing encrypted G-code file.	

16.6 Cutter Compensation Errors

Error	Message	Cause & Effect	Action
601	Error: no compensation in MDI	G41 or G42 entered in MDI. MDI is not canceled, but cutter compensation does NOT go into effect. Remainder of line processed.	Do not use G41 or G42 in MDI.
603	Arc as first uncomp. move on line NNNNN	Arc specified as first move after end of compensation (G40). Job cancelled.	First move after G40 must be a linear move.

Error	Message	Cause & Effect	Action
604	Plane must be XY on line NNNNN	Cutter compensation started with YZ or ZX plane selected. Job cancelled.	Remove cutter comp. for YZ or ZX plane moves, option is not available.
605	Canned cycle not allowed on line NNNNN	Canned cycle attempted during compensation. Job cancelled.	Do not use cutter comp. with canned cycles.
606	G53 not allowed on line NNNNN	G53 attempted during compensation. Job cancelled.	Choose a different work coordinate.
607	Set home not allowed on line NNNNN	M26 attempted during compensation. Job cancelled.	Do not use M26 with cutter comp.
608	Ref. point move not allowed on line NNNNN	G28, G29, or G30 attempted during compensation. Job cancelled.	Do not use return points with cutter comp.

16.7 Parameter Setting Errors

Error	Message	Cause & Effect	Action
701	G10 error: no R-value on line NNNNN	G10 used with no R-value. Job cancelled.	Input an R-value.
702	G10 error: invalid D on line NNNNN	Job cancelled (D0 cannot be set; it is always zero).	Change D to a valid value.
703	G10 error: invalid H on line NNNNN	G10 H0 Rxx specified. Job canceled (H0 cannot be set; it is always zero).	Change H to a valid value.
704	G10 error: invalid P on line NNNNN	G10 used with unknown P value. Job cancelled.	Change P to a valid value.
705	G10 error: No D, H, or P on line NNNNN	G10 used without D, H, or P to assign value. Job cancelled.	Add appropriate D, H, or P value.

16.8 Canned Cycle Errors

Error	Message	Cause & Effect	Action
801	Error: No R point on line NNNNN	No R-value specified. Job cancelled.	Add an R-point.
802	Error: Q = 0 on line NNNNN	Q value of 0 specified (Q used for G73 and G83 only). Job cancelled.	Insert a Q non-zero value.
803	Error: No Z point on line NNNNN	No Z value specified for canned cycle. Job cancelled.	Add a Z-value.
804	Error: Ggg invalid on line NNNNN (gg = 76, 86, 87, 88)	Unimplemented canned cycle requested. Job cancelled.	Change to a valid G-code.
805	Error: No Q value on line NNNNN	Q value not specified for G73 or G83. Job cancelled.	Insert a Q-value.
806	Error: No P value on line NNNNN	P value (dwell time) not specified for G82 or G89. Job cancelled.	Add a P-value.

Error	Message	Cause & Effect	Action
807	Error: Cannot execute G__ when axis B is rotated	On an Articulated Head machine with TWCS feature enabled, a non-compound canned cycle (such as G73, G74, G76, G80, G81, G82, G83, G84, G85, G89) was issued on a WCS that was set to TWCS=No while the spindle head was tilted (i.e. rotary B axis was not 0). Job cancelled.	Either move B to 0 or issue the compound canned cycle version of the erroneous G-code such as G173, G174, G176, G181, G182, G183, G184, G185, G189.

16.9 Miscellaneous Errors / Messages

Error	Message	Cause & Effect	Action
901	Ref. point invalid on line NNNNN	G30 with invalid P value (must be 1 or 2). Job cancelled.	Change P-value to a 1 or 2.
902	No prior G28 or G30 on line NNNNN	G29 with no preceding G28 or G30.	Add a G29 or G30.
903	Warning: No coordinates for G92 on line NNNNN	G92 with no axis coordinates to set. Remainder of line processed; job continues.	Add coordinates.
905	Warning: 0 radius arc on line NNNNN	Arc move was specified with a zero radius. Move is done as a linear move; job continues.	Specify a radius.
906	Warning: unknown arc on line NNNNN	Position of arc move could not be determined from parameters (e.g. G91 G2 X0 Y0 R1). Move is done as a linear move; job continues.	Correct program.
907	_ axis travel exceeded on line NNNNN	Software travel limit would be exceeded by the requested move. Job cancelled.	Check program, part zero or tool offset.
909	Program too long: job canceled	Attempt to run a job over 1MB in length, without the unlimited program size option. Job cancelled.	Contact Dealer or break up program.
910	No subroutines in MDI	Specified O9100–O9999 in MDI, which would begin an embedded subprogram. MDI cancelled.	
911	Illegal recursion	Attempt to execute a subprogram or macro that calls itself, either directly or indirectly. Job cancelled.	Call correct subprogram.
913	Could not open file filename.ext	Attempt to call a subprogram or macro, but the subprogram file does not exist. Job cancelled.	Make sure file name is correct and is in the nc-files directory.
915	DSP window retry sN fN rN	DSP window checking failed, move will be repeated unless the maximum retries have been reached, s = number of successes, f = number of failures, r = number of times the maximum retry value has been reached	
916	Unexpected probe contact	probed tripped when a cycle did not expected contact	
917	Invalid tilt lookup table	The tilt lookup table file (tilt.tab) has an invalid format or if it is not found	

Error	Message	Cause & Effect	Action
918	Probe unable to detect surface	Probe travelled maximum distance without contact, dsp window checking failed, or probe repeatability failed.	
919	DSP window failed maximum retries	DSP probe reached the maximum retry limit without a successful window	
920	Unable to clear obstacle	Probing cycle failed to clear an obstacle	
921	Unable to determine corner	Probing cycle failed to find corner (inside and outside corner)	
922	Out of memory	problem allocating memory	
923	Error: Z home not set	Z home is not set	
924	File read error	Problem reading the job file, this error occurs if the file was opened successfully but there was an error while reading the file.	
925	Error reading job file	same as above at a different place in the code	
926	Failed to locate job continuation position	Job continuation from the Run Menu failed.	Do a Run/Search
927	Too many subprogram calls	Nesting level of subprograms is too high. I.e. a subprogram calls another subprogram which calls another subprogram, which calls another subprogram, etc...	
928	Error Loading Log Configuration file... Using defaults	There was an error while loading the log configuration file. Default settings will be used.	
929	Log Level set to __	The logging level parameter (P140) has been changed.	
930	Log Level Configuration file not found... Creating new configuration.	The log level configuration file was not found. A default file will be created.	
931	Error during transformed move to home	A transformed moved to home (M25) command attempted to move below the G28 Z position.	
932	Error during Tool Check	A general error condition occurred when the Tool Check key was pressed.	
933	Log file initialized	There was an error in trimming the log file, or the log file did not exist, so a new log file has been created.	
934	Warning: Excess precision truncated	A CNC program is using axis positioning precision greater than what is displayed, and therefore the actual commanded positions are truncated. This happens when the Simultaneous Contouring feature was not enabled. This feature must be enabled for the extra precision to be acknowledged.	Contact Dealer for information about obtaining the Simultaneous Contouring feature.
935	_ axis (#) scale disabled	A scale is enabled for this axis but compensation was disabled. Scale compensation is disabled at initial power up, configuration changes, and during homing moves.	Home the machine.

Error	Message	Cause & Effect	Action
935	Probe failed reset retries	Probe failed to reset after 3 tries. The probing operation may have been started too close to the surface.	Move probe further away from surface and do probing operation again. If this continues to fail persistently, call dealer.
936	_ axis (#) scale enabled	A scale is enabled for this axis and compensation was enabled. This happens after homing the axis.	Not Applicable
936	Probe failed to reset	Probe failed to reset. The probing operation may have been started too close to the surface.	Move probe further away from surface and do probing operation again. If this continues to fail persistently, call dealer.
944	MPU requested resend	The MPU requested a resend	Status Message
945	PC requested resend	The PC requested a resend	Status Message
946	PC resending	The PC is resending	Status Message
947	PC received data out of order	The PC needed to reorder data received from the MPU	Status Message
948	PC packet error	The PC received bad data from the MPU and will try to recover by requesting a resend.	Status Message
949	Drive map does not match hardware	One or more of the drive mapping parameters 300–307 is misconfigured	Contact Dealer.

16.10 Scaling/Mirroring Errors

Error	Message	Cause & Effect	Action
1001	Invalid scaling parameter on line NNNNN	Invalid parameter specified (I, J, K, P). Job cancelled.	Remove or change invalid parameter.
1002	Invalid scaling center on line NNNNN	Invalid parameter specified (X, Y, Z). Job cancelled.	Remove or change invalid parameter.
1003	G-code not allowed when scaling on line NNNNN	G28/G29/G30/G92 is not allowed when scaling or mirroring is turned on. Job cancelled.	Move G-code to appropriate line.
1004	Turn scaling off before rescaling	Tried to rescale while scaling is turned on. Job cancelled.	Turn scaling off, then rescale.
1005	Cannot scale arcs with different scale factors	Scaling factors of the arc axes are different. Job cancelled.	Correct scaling factors, or separate scaling operations.
1100–1199	Custom messages defined in cncxmsg.txt. Please contact your dealer if you have any questions regarding a particular message. This style of message should be replaced with plcmsg.txt format on MPU11 systems.		

16.11 Configuration Modification Messages

Error	Message	Cause & Effect
111	__ modified: __ $\rightarrow$ __	An axis configuration parameter was modified.
444	__ modified: __ $\rightarrow$ __	A servo drive configuration parameter was modified.
555	__ modified: __ $\rightarrow$ __	A PID configuration parameter was modified.
556	Axis converted: $\rightarrow$ __	A PID configuration parameter was converted.
777	__ modified: __ $\rightarrow$ __	An axis configuration parameter was modified.
888	G30 Z modified: $\rightarrow$ __	Z coordinate of Secondary Reference Point was modified.
999	Parm # modified: $\rightarrow$ __	A machine parameter was modified.

17 Centroid CNC Additional Resources

Centroid Acorn CNC Control board Documentation

Start Here Acorn DIY Installation Videos and documentation

All Acorn Documentation

Centroid CNC control board Standard Schematic Sets

Acorn DIY CNC System hookup schematics

Allin1DC DIY CNC System hookup schematics

Oak DIY CNC System hookup schematics

CNC 11/12 Based, Special System Manuals

AC/DC servo drive+MPU11+GPio4D CNC control system Installation manual

CENTROID-Fanuc CNC Retrofit Installation Manual

MPU11/GPIO4D Velocity Mode Installation Manual

Centroid PLC Detective quick start guide

Centroid CNC11 PLC programming manual

Centroid CNC12 PLC programming manual

Centroid Oak CNC Control board Installation manual

Oak DIY Installation Manual

Centroid Allin1DC CNC Control board Installation manual

Allin1DC DIY Installation Manual

Centroid Individual Subject Technical Bulletins

CNC Tech Bulletins

Individual control component manuals

PLCADD1616, PLC expansion board users manual

Add4AD4DA, Analog Output expansion board users manual

Encoder Expansion board user maual

DC1, Single Axis DC servo drive users manual

RTK4 users manual

AC / DC Servo Drive users manual

MPU11 w/ Legacy Add Card connect to legacy drives and plcs

OpticDirect users manual (Yaskawa and Delta optical interrface)

PLCAdd6464, TTL Level PLC expansion board

Configuring Windows for CNC control use

Windows 10 setup for CNC duty video

Windows 8 setup for CNC duty instructions

Windows 7/10 setup for CNC duty instructions

Centroid Touch Probe Manuals

KP-3 Touch Probe Manual

DP-4 Touch Probe Manual

TT-2 Tool Touch Probe Manual

TT-1 Tool Touch Probe Manual

TT-4 Tool Touch Probe Manual

Miscellaneous CNC Documents

Solid Model files for OAK, ALLIN1DC, PLCADD1616, DC1, ADD4AD4DA, Encoder Expansion Board. zip file

CNC console dimensions and mounting examples

CNC Standard Electrical Cabinet M400/M39/T400/T39

CNC Standard Electrical Cabinet M15 upgrade

Swing Arm Tool Changer user manual

Umbrella Tool Changer user manual

System Test

TTL2DIFF, single ended to differential signal converter

8RELBRD, 8 Relay Output add on board

Hardinge HNC/CHNC retrofit manual

Centroid RT150 Rotary Table manual

Centroid RT200 Rotary Table manual

Intercon DXF Import

Centroid CNC Tech Support

Free Tech Support via the Centroid CNC Community Support Forum

Centroid CNC Technical Support YouTube Channel

Purchase One-on-one Centroid CNC Direct Tech Support