CENTROID™

T400 & T39
Operators Manual

Centroid T-Series Lathe Operators Manual

CNC Software Version: CNC12 Version v5.08+

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-lathe-operator-manual.pdf

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

Copyright ©2013–2024 Centroid Corp. Howard, PA 16841

U.S. Patent #6490500

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

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

Draft: January 11, 2024

Information provided by CENTROID relating to wiring, installation, and operation of CNC components is intended only as 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
	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,**

Draft: January 11, 2024

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 feed rates 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 the 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.

Draft: January 11, 2024

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 **9**

1.1 DRO Display . . . 9
1.2 Distance-to-Go DRO . . . 10
1.3 Status Window . . . 10
1.4 Message Window . . . 10
1.5 Options Window . . . 11
1.6 User Window . . . 11
1.7 Conventions . . . 11
1.8 Machine Home . . . 12
1.9 Lathe M- and G-codes . . . 14
1.10 How to Unlock Software Features or Unlock Your Control . . . 14
1.11 Skinning . . . 15

2 Operator Panel **16**

2.1 Axis Jog Buttons . . . 17
2.2 Slow/Fast . . . 17
2.3 Inc/Cont . . . 17
2.4 x1, x10, x100 . . . 17
2.5 MPG . . . 17
2.6 Single Block . . . 18
2.7 Cycle Start . . . 18
2.8 Feed Rate Override . . . 18
2.9 Feed Hold . . . 18
2.10 Tool Check . . . 18
2.11 Cycle Cancel . . . 19
2.12 Emergency Stop . . . 19
2.13 Spindle CW/CCW . . . 19
2.14 Spindle Speed + . . . 19
2.15 Spindle Speed 100% . . . 19
2.16 Spindle Speed - . . . 19
2.17 Spindle Auto/Man . . . 20
2.18 Spin Start . . . 20
2.19 Spin Stop . . . 20
2.20 Coolant Auto/Manual . . . 20
2.21 Coolant Flood . . . 20
2.22 Coolant Mist . . . 20
2.23 Auxiliary Function Keys (AUX1 – AUX12) . . . 20
2.24 T-stock In, T-stock Out, Quill In, Quill Out, Turret Index . . . 21
2.25 VCP Introduction . . . 22
2.26 Keyboard Jog Panel . . . 29
2.27 MDI and the Keyboard Jog Panel . . . 32
2.28 Keyboard Shortcut Keys . . . 32

3 CNC Software Main Screen **37**

3.1 **F1 – Setup Menu** . . . 37
3.2 **F2 – Load Job Menu** . . . 38
3.3 **F3 – MDI** . . . 39
3.4 **F4 – Run Menu** . . . 40

Draft: January 11, 2024

3.5 **F5 – CAM** . . . 40
3.6 **F6 – Edit** . . . 41
3.7 **F7 – Utility** . . . 42
3.8 **F8 – Graph** . . . 43
3.9 **F9 – Tool Wear** . . . 45
3.10 **F10 – Shutdown** . . . 45

4 Part Setup (F1 from Setup) 47
4.1 Setting Part Zeros – Introduction . . . 48
4.2 Work Coordinate Systems (WCS) Configuration . . . 52

5 Tool Setup 56
5.1 Tool Geometry Offset Library . . . 56
5.2 Tool Wear Adjustment Screen . . . 62
5.3 Setting up Tool Orientation, Tool Type, and Approach . . . 64
5.4 Setting up the Nose Vector . . . 65
5.5 Manual Measurement of X-axis Tool Offsets for OD Tools . . . 66
5.6 Manual Measurement of X-axis Tool Offsets for ID Tools . . . 69
5.7 Manual Measurement of X-axis Offsets for Drills, Center Drills, and Taps . . . 71
5.8 Manual Measurement of X-axis Offsets for Boring Tools . . . 72
5.9 Manual Measurement of Z-axis Tool Offsets . . . 73
5.10 Manual Measurement of Part Cutoff Tool Z-offset . . . 74
5.11 Setting the Nose Radius . . . 75
5.12 Automatic Tool Touch-off cycles (extra-cost option) . . . 77
5.13 Centroid CNC Lathe Tool Touch-off Using TT-4/TT-2 . . . 79

6 Running a Job 81
6.1 Active Job Run Screen with G-code Display . . . 81
6.2 Run-time Graphics Screen . . . 82
6.3 Canceling a Job in Progress . . . 82
6.4 Resuming a Canceled Job . . . 83
6.5 Run Menu . . . 83
6.6 Power Feed . . . 85
6.7 Communications Stress Test . . . 85

7 Intercon Software 87
7.1 Introduction . . . 87
7.2 Lathe Intercon Main Menu . . . 87
7.3 Teach Mode . . . 90
7.4 Lathe Intercon File Menu . . . 90
7.5 Lathe Intercon File Menu Continued. . . . 92
7.6 Insert Operation . . . 92
7.7 **F3 – Drill** . . . 98
7.8 **F4 – Tap** . . . 99
7.9 **F5 – Thread** . . . 100
7.10 **F6 – Profile** . . . 104
7.11 **F7 – Turning** . . . 106
7.12 **F8 – Groove** . . . 109
7.13 **F9 – Cutoff** . . . 111
7.14 **F10 – Other** . . . 112
7.15 Graphics . . . 116
7.16 Math Help . . . 117
7.17 Intercon Lathe Tool Library . . . 124
7.18 Lathe Intercon Tutorial #1 . . . 126

Draft: January 11, 2024

7.19 Lathe Intercon Tutorial #2 . . . 133

8 CNC Program Codes **147**
8.1 General . . . 147
8.2 Miscellaneous CNC Program Symbols . . . 147
8.3 Introduction to Centroid CNC Macros . . . 157
8.4 Advanced Macro Statements . . . 189

9 CNC Program Codes: G-codes **192**
9.1 G00 – Rapid Positioning . . . 193
9.2 G01 – Linear Interpolation . . . 193
9.3 G02 & G03 – Circular or Helical Interpolation . . . 194
9.4 G04 – Dwell . . . 196
9.5 G10 – Parameter Setting . . . 196
9.6 G20 – Select Inch Units . . . 196
9.7 G21 – Select Metric Units . . . 196
9.8 G22/G23 – Work Envelope On/Off . . . 196
9.9 G28 – Return to Reference Point . . . 197
9.10 G29 – Return from Reference Point . . . 197
9.11 G30 – Return to Secondary Reference Point . . . 197
9.12 G32 – Constant Lead Thread Cutting . . . 198
9.13 G37 – PWM Velocity Modulation ON/OFF . . . 198
9.14 G40, G41, G42 – Cutter Compensation . . . 199
9.15 G50 – Coordinate System Setting OR Maximum Spindle Speed for CSS mode . . . 202
9.16 G52 – Offset Local Coordinate System . . . 202
9.17 G53 – Rapid Positioning in Machine Coordinates . . . 202
9.18 G54–G59 – Select Work Coordinate System . . . 202
9.19 G65 – Call Macro . . . 203
9.20 G70 – Profile Finishing Cycle . . . 204
9.21 G71 – Profile Stock Removal in Turning . . . 205
9.22 G72 – Profile Stock Removal in Facing . . . 207
9.23 G73 – Profile Path Repeating Cycle . . . 209
9.24 G74 – End Face Peck Cutting Cycle . . . 210
9.25 G75 – Outside/Inside Diameter Peck Cutting Cycle . . . 212
9.26 G76 – Multi-pass Threading Cycle . . . 213
9.27 G80 – Canned Cycle Cancel . . . 215
9.28 G83 – Deep Hole Drilling . . . 215
9.29 G84 – Tapping . . . 216
9.30 G85 – Boring . . . 218
9.31 G90 – Outside/Inside Diameter Cutting Cycle . . . 218
9.32 G92 – Thread Cutting Cycle . . . 220
9.33 G96 & G97 – Constant Surface Speed Control & Cancel . . . 222
9.34 G98 – Feed per Minute . . . 222
9.35 G99 – Feed per Revolution . . . 222

10 CNC Program Codes: M-functions **223**
10.1 Summary of M-functions . . . 223
10.2 Macro M-functions (Custom M-functions) . . . 224
10.3 M00 – Stop for Operator . . . 225
10.4 M01 – Optional Stop for Operator . . . 225
10.5 M02 – Restart Program . . . 225
10.6 M03 – Spindle On Clockwise . . . 225
10.7 M04 – Spindle On Counter-clockwise . . . 226

Draft: January 11, 2024

10.8 M05 – Spindle Stop . . . 226
10.9 M06 – Tool Change . . . 226
10.10 M07 – Mist Coolant On . . . 227
10.11 M08 – Flood Coolant On . . . 227
10.12 M09 – Coolant Off . . . 227
10.13 M10 – Clamp On . . . 227
10.14 M11 – Clamp Off . . . 227
10.15 M17 – Prepare for Tool Change (Macro) . . . 227
10.16 M19 – Spindle Orient (Macro) . . . 228
10.17 M26 – Set Axis Home . . . 228
10.18 M29 – Set Tap Mode for G84 . . . 228
10.19 M30 – Restart Program . . . 228
10.20 M41, M42, M43 – Select Spindle Gear Range (Macros) . . . 228
10.21 M50 – C-axis Disable . . . 229
10.22 M51 – C-axis Enable (Optional) . . . 229
10.23 M91 – Move to Minus Home . . . 229
10.24 M92 – Move to Plus Home . . . 230
10.25 M93 – Release/Restore Motor Power . . . 230
10.26 M94/M95 – Output On/Off . . . 231
10.27 M98 – Call Subprogram . . . 231
10.28 M99 – Return from Macro or Subprogram . . . 232
10.29 M100 – Wait for PLC Bit (Open, Off, Reset) . . . 232
10.30 M101 – Wait for PLC Bit (Closed, On, Set) . . . 232
10.31 M102 – Restart Program . . . 233
10.32 M103 – Programmed Action Timer . . . 233
10.33 M104 – Cancel Programmed Action Timer . . . 233
10.34 M105 – Move Minus to Switch . . . 233
10.35 M106 – Move Plus to Switch . . . 233
10.36 M107 – Output Tool Number . . . 234
10.37 M108 – Enable Override Controls . . . 234
10.38 M109 – Disable Override Controls . . . 234
10.39 M115/M116/M125/M126 – Protected Move Probing Functions . . . 234
10.40 M120 – Open Data File (Overwrite Existing File) . . . 235
10.41 M121 – Open Data File (Append to Existing File) . . . 236
10.42 M122 – Record Local Position(s) and Optional Comment in Data File . . . 236
10.43 M123 – Record Value and/or Comment in Data File . . . 236
10.44 M124 – Record Machine Position(s) and Optional Comment in Data File . . . 237
10.45 M127 – Record Date and Time in a Data File . . . 237
10.46 M128 – Move Axis by Encoder Counts . . . 237
10.47 M129 – Record Current Job File Path to Data File . . . 237
10.48 M130 – Run System Command . . . 238
10.49 M150 – Set Spindle Encoder to Zero at Next Index Pulse . . . 238
10.50 M151 – Unwind C-axis . . . 238
10.51 M200, M223, M224, M225 & M290 – Formatted String Commands . . . 238
10.52 M200/M201 – Stop for Operator, Prompt for Action . . . 240
10.53 M223 – Write Formatted String to File . . . 240
10.54 M224 – Prompt for Operator Input Using Formatted String . . . 240
10.55 M225 – Display Formatted String for A Period of Time . . . 241
10.56 M290 – Digitize Profile (Optional) . . . 241
10.57 M300 – Fast Synchronous I/O Update . . . 242
10.58 M1000–M1015 – Graphing Color for Feed Rate Movement . . . 242

11 Configuration **244**

Draft: January 11, 2024

11.1 User Specified Paths . . . 246
11.2 Machine Configuration . . . 247
11.3 Machine Parameters (**F3 – Parms from Configuration)** . . . 252
11.4 PID Menu . . . 315
11.5 Modifying the CAM Menu . . . 317
11.6 CAM File Format . . . 318
11.7 PLC I/O Diagnostic App . . . 319
11.8 Smoothing Configuration Parameters . . . 326
11.9 G-code AD2 Smoothing . . . 328
11.10 Smoothing AD2 Setup Menu . . . 330
11.11 Custom Smoothing Presets Menu . . . 332
11.12 Technical Background of AD2 Smoothing . . . 333
11.13 Wirelesss MPG Installation . . . 337
11.14 Custom WMPG Macros . . . 339

12 CNC Software Messages **343**
12.1 CNC Software Startup Errors and Messages . . . 343
12.2 Messages Issued Upon Exit from the CNC Software . . . 343
12.3 Messages and Prompts in the Operator Status Window Status Messages . . . 343
12.4 Abnormal Stops (Faults) . . . 345
12.5 CNC Syntax Errors . . . 352
12.6 Cutter Compensation Errors . . . 355
12.7 Parameter Setting Errors . . . 355
12.8 Canned Cycle Errors . . . 355
12.9 Miscellaneous Errors/Messages . . . 356
12.10 Configuration Modification Messages . . . 358

13 Centroid CNC Additional Resources **359**

14 Glossary **362**

Index **374**

Draft: January 11, 2024

1 Introduction

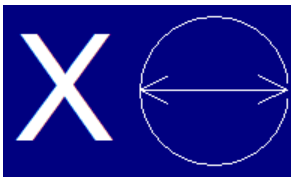
The T-Series display screen is separated into five areas:

1.1 DRO Display

The DRO display contains the digital readout for the current position of the tool. The display is configurable for number of axes and desired display units of measure (see Chapter 11). Use the **ALT-D** “Hot Key” combination to change the display between machine position and the current WCS position.

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.

The symbol next to the X-axis DRO indicates Diameter or Radius mode. The mode is configured by machine Parameter 55. If the X-axis DRO is in Diameter mode, it will look like this:



Draft: January 11, 2024

If the X axis DRO is in Radius mode, it will look like this:

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 (see Chapter 3). Below the job name are the Tool Number and Tool Offset, Program Number, Feed Rate Override, Spindle Speed, and Feed Hold indicators. The Feed Rate Override indicator displays the current override percentage set on the Jog Panel. 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 of the **FEED HOLD** button located on the Jog Panel. If FEED HOLD is on, then the Feed Hold indicator will indicate 'On' and can only be turned off by pressing **CYCLE START**.

The Part Count and Elapsed Time indicators are not always displayed. Pressing **CYCLE START** while a job is running will cause the indicators to appear. The Part Count indicator displays the number of times the current part has been run and upon the completion of each run, it can increment/decrement by one. If a job is canceled prematurely, the Part Count will not be incremented. The Part # counter shows 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 Count and Part# setting.

The Elapsed Time indicator displays the amount of time passed since **CYCLE START** was pressed. The indicator will help you to determine how long it takes to cut a particular part. The timer will not stop until the job is finished or canceled for any reason. It will continue to count for optional stops and tool changes.

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 12 for a description of the CNC software error and status messages.

Draft: January 11, 2024

1.5 Options Window

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

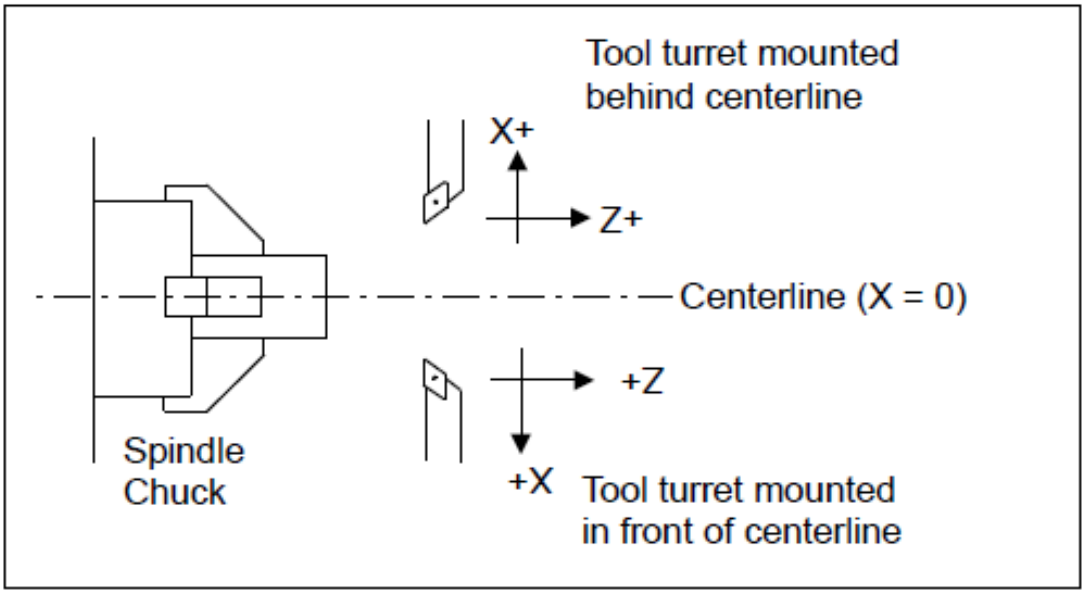
1.6 User Window

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

- There are ten function keys used by the control. They are represented by the **F1**, **F2**, . . . , **F10** keys. Keystrokes other than the function keys are represented by the capitalized name of the key in bold font. 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**.
- Data entry menus on the T-series Control usually use the **F10 – Save** key to save changes and the **ESC** key to discard changes.
- Any menu in the T-series Control can be exited by pressing the **ESC** key. This will take you back to the previous menu, pressing the **ESC** key enough times will eventually take you back to the main screen. This also usually discards any changes you have made in that menu.
- The Centerline of the part (and Spindle) is usually considered to be X=0.
- The orientation of the axes are as follows: X+ always points away from the Centerline and Z+ always points to the right and away from the Spindle. Although the T-series Control is able to display the X+ direction as either oriented up or down (set in Machine Parameter 1), most of the illustrations in this manual will show X+ as pointing upward, as if the tool turret is mounted behind the centerline of the spindle.

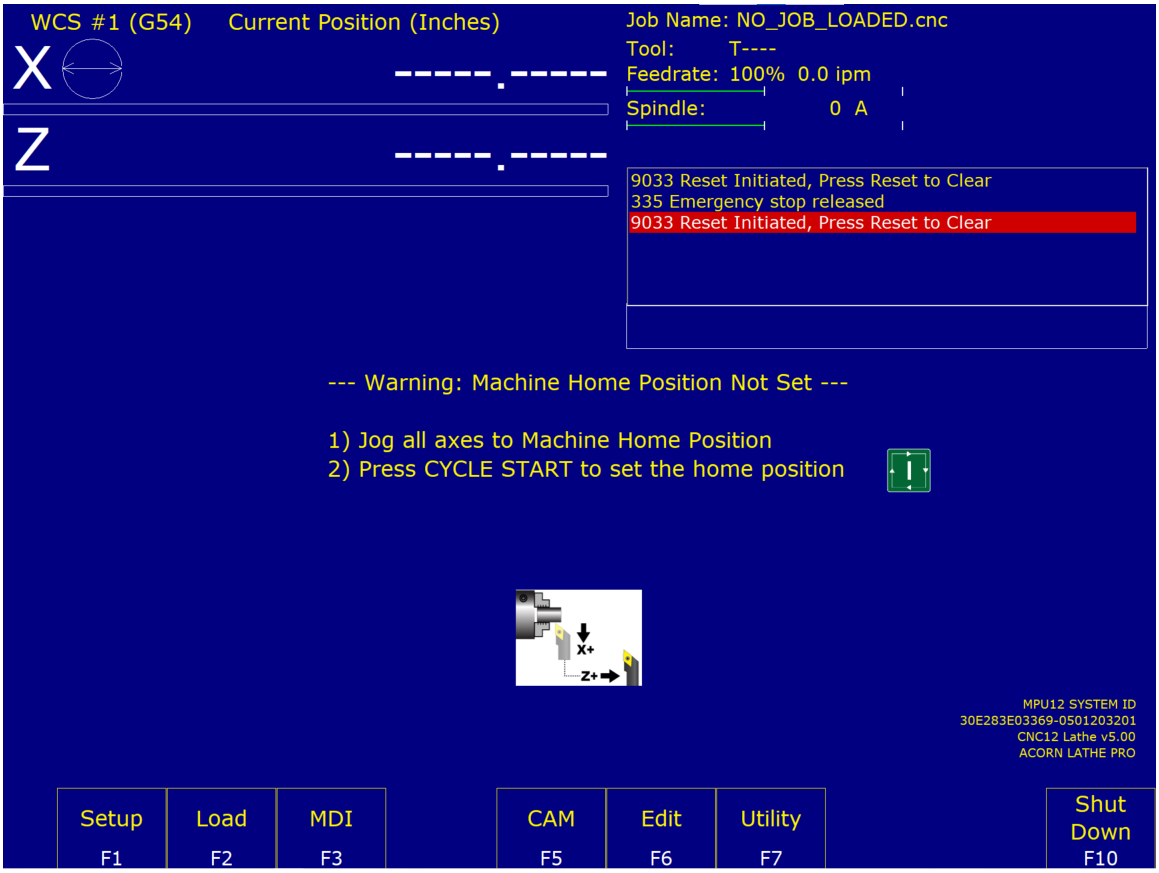


Draft: January 11, 2024

- Tools move in X and Z directions. The work piece remains in a stationary location relative to X and Z.
- CW stands for clockwise and CCW stands for counter-clockwise.
- The work piece physically spins in the Spindle Chuck, the CW and CCW directions refer to the chuck spinning in those directions when viewed in the Z+ direction (through the spindle towards the tailstock).
- ID means Inner Diameter and OD means Outer Diameter.

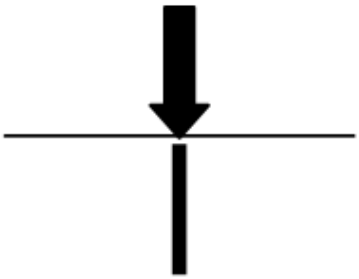
1.8 Machine Home

When the T-series control is first started, the Main Screen will appear as below:



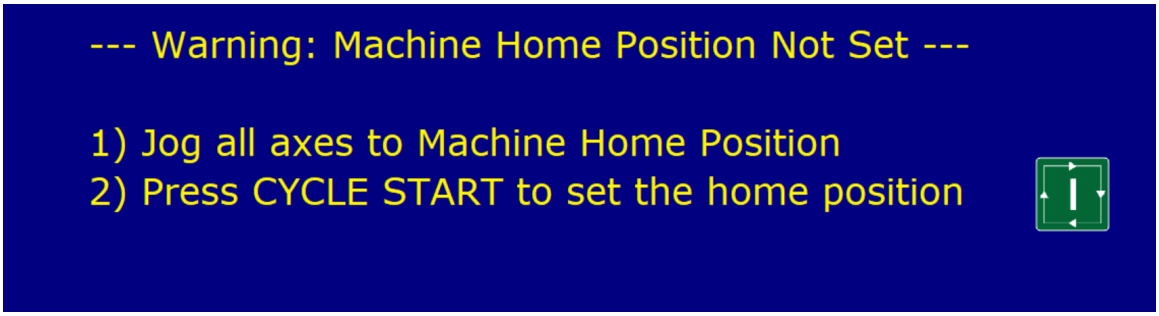
Before you can run any jobs, you must set the machine home position. If your machine has home/limit switches, reference marks, or safe hard stops, the control can automatically home itself. If your machine has reference marks, jog the machine until the reference marks are lined up (see below) before 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 X to its plus limit and home Z to its plus limit.

Draft: January 11, 2024



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.

Draft: January 11, 2024

1.9 Lathe M- and G-codes

M00 Stop For Operator	G00 Rapid to Position
M01 Optional Stop for Operator	G01 Linear Move
M02 Restart Program	G02 CW Arc Move
M03 Spindle On Clockwise	G03 CCW Arc Move
M04 Spindle On Counter-clockwise	G04 Dwell
M05 Spindle Stop	G10 Set Parameter
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
M26 Set Axis Home	G29 Return from Reference Point
M29 Set Tap Mode for G84	G30 Return to Secondary Reference Point
M30 Restart Program	G32 Constant Lead Thread Cutting
M50 C-axis Disable	G40 Cancel Cutter Compensation
M51 C-axis Enable	G41 Cutter Compensation Left/Right
M91 Move to Minus Home	G42 Cutter Compensation Left/Right
M92 Move to Plus Home	G50 Coordinate System Setting OR
M93 Release/Restore Motor Power	Maximum Spindle Speed for CSS mode
M94/M95 Output On/Off	G52 Offset Local Coordinate System Origin
M98 Call Subprogram	G53 Rapid Positioning in Machine Coordinates
M99 Return from Macro or Subprogram	G54-G59 Select Work Coordinate System
M100 Wait for PLC bit (Open, Off, Reset)	G65 Call Macro
M101 Wait for PLC bit (Closed, On, Set)	G70 Profile Finishing Cycle (after Profile Stock Removal)
M102 Restart Program	G71 Profile Stock Removal in Turning
M103 Programmed Action Timer	G72 Profile Stock Removal in Facing
M104 Cancel Programmed Action Timer	G73 Profile Path Repeat
M105 Move Minus to Switch	G74 End Face Peck Cutting Cycle
M106 Move Plus to Switch	G75 Outside/Inside Diameter Peck Cutting Cycle
M107 Output BCD Tool Number	G76 Multi-Pass Threading Cycle
M108 Enable Override Controls	G80 Canned Cycle Cancel
M109 Disable Override Controls	G83 Deep Hole Drilling (Canned Cycle)
M115, M116, M125, M126 Protected Move Probing	G84 Tapping (Canned Cycle)
M120 Open data file (overwrite existing file)	G85 Boring (Canned Cycle)
M121 Open data file (append to existing file)	G90 Outside/Inside Diameter Cutting Cycle
M122 Record position(s) in data file	G92 One-pass Thread Cutting Cycle
M123 Record value and/or comment in data file	G94 End Face Turning
M124 Record machine position(s) in data file	G96 Constant Surface Speed (CSS mode)
M127 Record Date and Time in a data file	G97 Cancel Constant Surface Speed
M128 Move Axis by Encoder Counts	G98 Feed-per-Minute
M129 Record Current Job file path to data file	G99 Feed-per-Revolution
M130 Run system command	
M150 Set Spindle Position to 0 on Next Index Pulse	
M151 Unwind C-axis	
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	
M300 Fast Synchronous I/O update	
M1000–M1015 Graphing Color for Feedrate movement	

1.10 How to Unlock Software Features or Unlock Your Control

The following are necessary to unlock software features:

Draft: January 11, 2024

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

Draft: January 11, 2024

2 Operator Panel

The Operator Panel is a sealed membrane keyboard that enables you to control various machine operations and functions. The panel contains momentary membrane switches, which are used in combination with LED indicators to indicate the status of the machine functions.

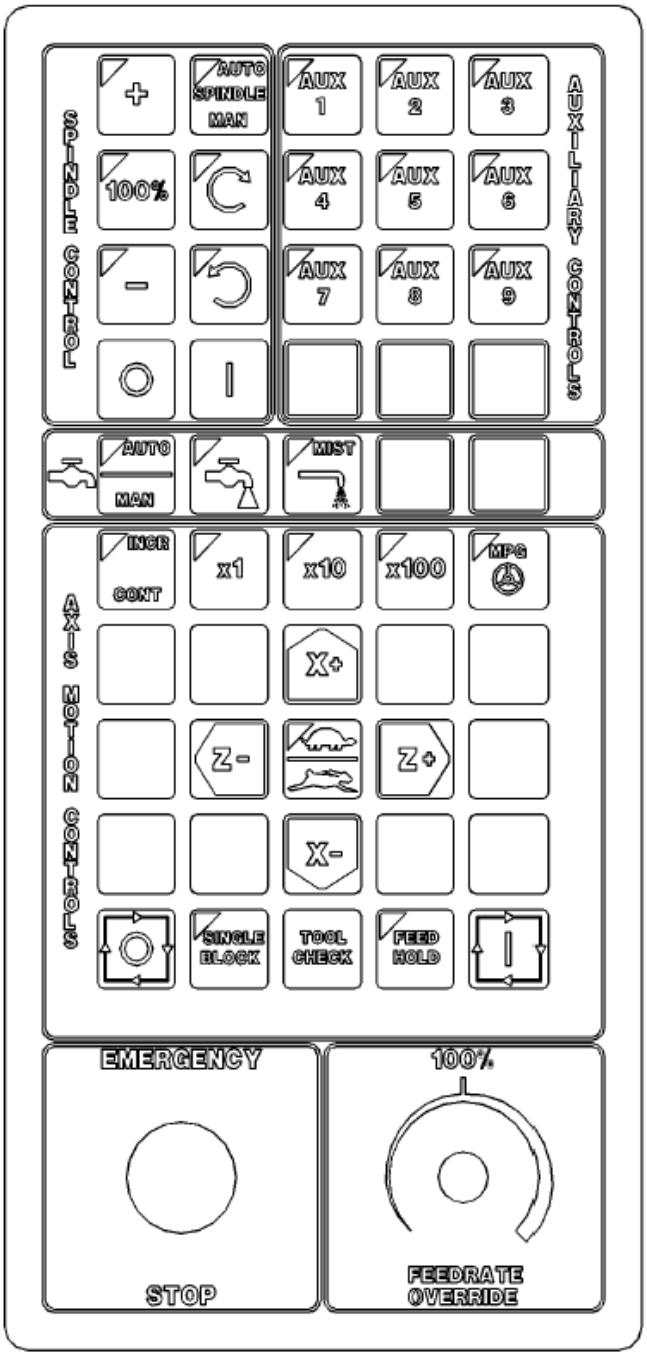


Fig 1 - T-Series Jog Panel

Draft: January 11, 2024

2.1 Axis Jog Buttons

X+ X- Z+ Z-

The yellow X and Z keys are momentary switches for jogging the two 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 T-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 11 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.

- Rear mount, horizontal configuration shown. The control and jog overlay can be configured for a horizontal, or vertical machine configuration with front-, rear-, left-, or right-mount tooling. See Chapter 11, Parameter 1 for further information on configuring your machine and tooling orientation.

2.4 x1, x10, x100

Press any one of these keys to set the jog increment amount. The amount you select is the distance that 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 that 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.

NOTICE

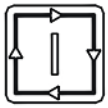
Pressing **CYCLE START** will cause the M-series Control to start moving the axes immediately without further warning. Be certain that you are ready to start the program when you press this button. Pressing the **FEED HOLD** or **CYCLE CANCEL** buttons will stop any movement if **CYCLE START** is pressed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

Draft: January 11, 2024

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 T-series 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 tool change is encountered in the program, the message “Press CYCLE START to continue”will be displayed on the screen, and the T-series Control will wait until you press the **CYCLE START** button before continuing program execution.

⚠ WARNING Pressing **CYCLE START** will cause the T-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**, **E-STOP**, or **CYCLE CANCEL** buttons will stop any movement if **CYCLE START** is pressed accidentally.

2.8 Feed Rate Override

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

⚠ CAUTION The Feed Rate Override knob will not work during tapping cycles (G84) and threading moves (G32).

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 (G84) threading moves (G32), and automatic tool changes (M6).

2.10 Tool Check

Pressing **TOOL CHECK** while a job (a CNC program) is not running will move the table to its tool change (G28) position. Pressing **TOOL CHECK** while a job is running will stop normal program movement, clear all M-functions, and automatically display the Resume Job Menu. From the Resume job menu, you will be able to change tool settings.

Note: When a job is running, pressing **TOOL CHECK** once stops the job and allows you to manually jog the tool clear. Pressing **TOOL CHECK** a second time will cause the tool to move to its tool change (G28) position.

Draft: January 11, 2024

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 of 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.

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.

Draft: January 11, 2024

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

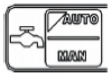
Press the **SPIN START** key when manual spindle mode is selected to cause the spindle to start rotating. Press the **SPIN START** key when automatic mode is selected to restart the spindle if it has been paused with the **SPIN STOP** key.

2.19 Spin Stop

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

NOTICE The **SPIN STOP** key 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.

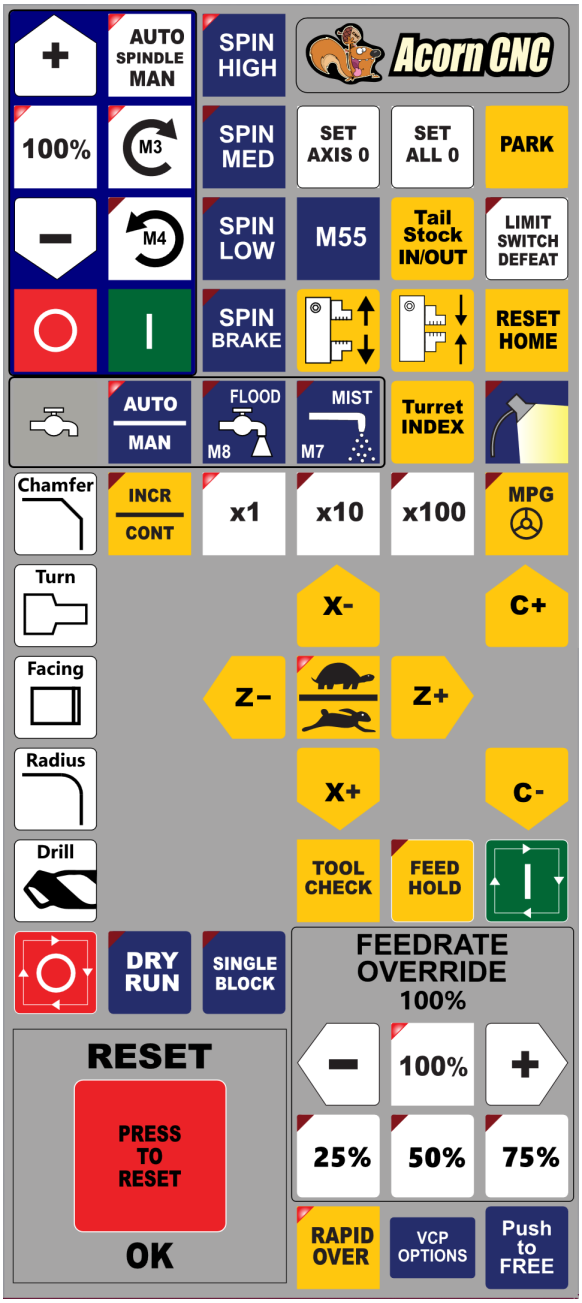
Draft: January 11, 2024

2.24 T-stock In, T-stock Out, Quill In, Quill Out, Turret Index

These buttons currently have no settings, but can be added to one of the Aux Keys and then programmed to control hydraulic stock clamps, quills, or turret index functions through the PLC. Your installer will provide you with the necessary documentation explaining the operations and functions these keys perform.

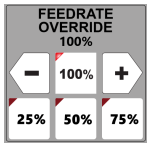
Draft: January 11, 2024

2.25 VCP Introduction

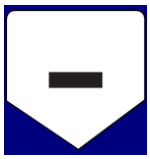


The Virtual Control Panel (VCP) allows the user to use a mouse and/or a touch screen monitor to activate the CNC Control Operator Interface Panel. The VCP has been designed from the ground up with the intention to allow users, re-builders, and OEMs alike a simple way to change the look, feel, and function 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.

Draft: January 11, 2024

Legend	Function	Description
	Cycle Start	Same as Cycle Start.
	Cycle Cancel	Same as Cycle Cancel.
	Increase/ Decrease feed rate override	Increase/Decrease feed rate override by 1% while held.
	Toggle Auto Coolant	Toggles coolant mode between auto and manual.
	Toggle Flood Coolant	Toggles Flood coolant if in manual mode.
	Selects CW Spin	Selects CW Spin direction in manual mode.
	Selects CCW Spin	Selects CCW Spin direction in manual mode.
	Toggle Spindle Auto/Manual	Toggles between automatic and manual spindle operation mode

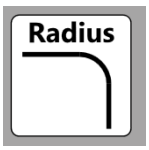
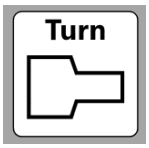
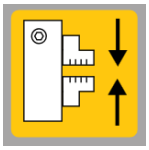
Draft: January 11, 2024

Legend	Function	Description
	Spin Start	Starts spindle in selected direction if in manual mode.
	Spin Stop	Stops spindle regardless of auto or manual mode.
	Spindle Override +1%	Increase the spindle override by 1% while held.
	Spindle Override −1%	Decrease the spindle override by 1% while held.
	Tool Check	Performs a tool check.
	Incremental/ Continuous Jog Selection	Toggles incremental or continuous jog mode.
	Single Block	Selects Single Block Mode
	Decrease/Increase Jog Increment	Decreases/increases current jog increment to the next available increment.

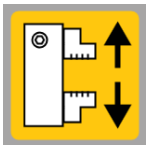
Draft: January 11, 2024

Legend	Function	Description
	Feed Hold	Temporarily pauses the feed rate.
	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.
	X+ Jog	Jogs the X-axis positively.
	X- Jog	Jogs the X-axis negatively.
	Z+ Jog	Jogs the Z-axis positively.
	Z- Jog	Jogs the Z-axis negatively.

Draft: January 11, 2024

Legend	Function	Description
	Chamfer One Shot	Generates a cutting move from the current position to one of four angles.
	Drill One Shot	Generates a drilling move from the current position to a chosen diameter and depth.
	Facing One Shot	Generates a facing move from the current position with a chosen diameter and taper.
	Radius One Shot	Generates an arc move from the current position to one of eight directions.
	Turn One Shot	Generates a turning move from the current position with a chosen diameter and taper.
	C+ Jog	Jogs the C-axis positively.
	C- Jog	Jogs the C-axis negatively.
	Engage Clamp/Chuck	Engages the clamp/chuck to secure the current tool in place.

Draft: January 11, 2024

Legend	Function	Description
	Disengage Clamp/Chuck	Disengages the clamp/chuck to allow for removal of the current tool.
	Dry Run	Runs the current program with an offset. Used machining program test process.
	Toggle Work Light	Toggles the work light between on and off positions.
	Limit Switch Defeat	Overrides the limit switches when active.
	Toggle Mist Coolant	Toggles Mist coolant if in manual mode.
	M55 Function	Used to toggle the M55 function.
	Toggle MPG	Toggles between the MPG and jog panel.
	Park	Parks the machine in its current position.

Draft: January 11, 2024

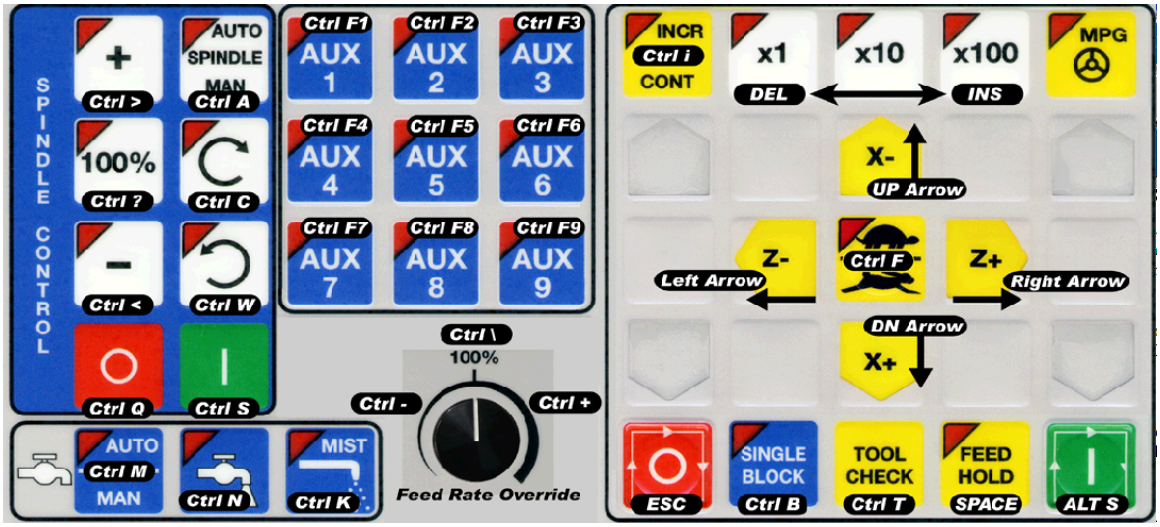
Legend	Function	Description
	Rapid Override	Toggles rapid jog movement override.
	Reset Home	Resets the home values that are currently set.
	Set All 0	Sets all axes to zero values.
	Set Axis 0	Sets the currently selected axis to a value of zero.
	Slow/Fast	Toggles between slow and fast jogging modes.
	Spindle Brake	Toggles the spindle brake.
	Spindle Speed	Allows the user to select the spindle speed (high, medium, and low).

Draft: January 11, 2024

Legend	Function	Description
Tail Stock IN/OUT	Tail Stock In/Out	Extends and withdraws the tail stock.
Turret INDEX	Turret Index	Accesses the turret tool index.

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, feed rate, 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 stop movement 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”, “Z”, etc.) The jog keys are the **Arrow**, **Page Up**, and **Page Down** keys.

The remaining keys are described below:

Draft: January 11, 2024

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.
	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 feed rate 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.

Draft: January 11, 2024

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.
	Ctrl I	Incremental/ Continuous Jog Selection	Toggles incremental or continuous jog mode.	Available most times that jogging is available.

Draft: January 11, 2024

Legend	Key(s)	Function	Description	Availability (Notes)
	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.
	Down Arrow	X+ Jog	Jogs the X-axis positively.	With on-screen jog panel displayed.
	Up Arrow	X- Jog	Jogs the X-axis negatively.	With on-screen jog panel displayed.
	Right Arrow	Z+ Jog	Jogs the Z-axis positively.	With on-screen jog panel displayed.
	Left Arrow	Z- Jog	Jogs the Z-axis negatively.	With on-screen jog panel displayed.

Draft: January 11, 2024

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 the 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

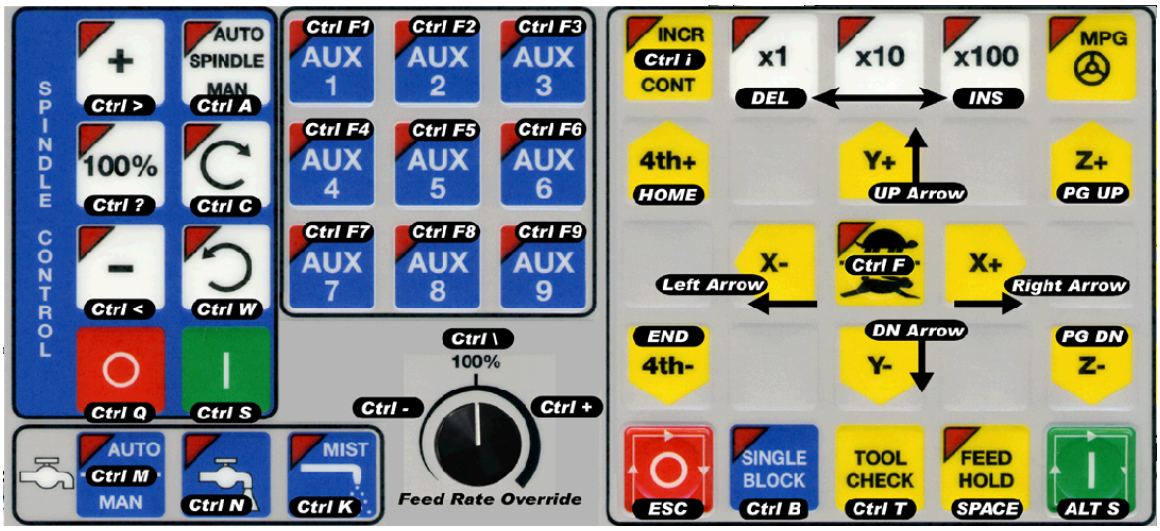
Pressing **ALT+I** brings up the Live PLC I/O display screen (see the image below). In this screen, you may view the real-time status of all the different inputs, outputs, etc. Use the arrow, **F11**, and **F12** keys to navigate the Live PLC I/O screen. With proper PLC support, pressing **CTRL+ALT+I**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

Draft: January 11, 2024



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.

Draft: January 11, 2024

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.

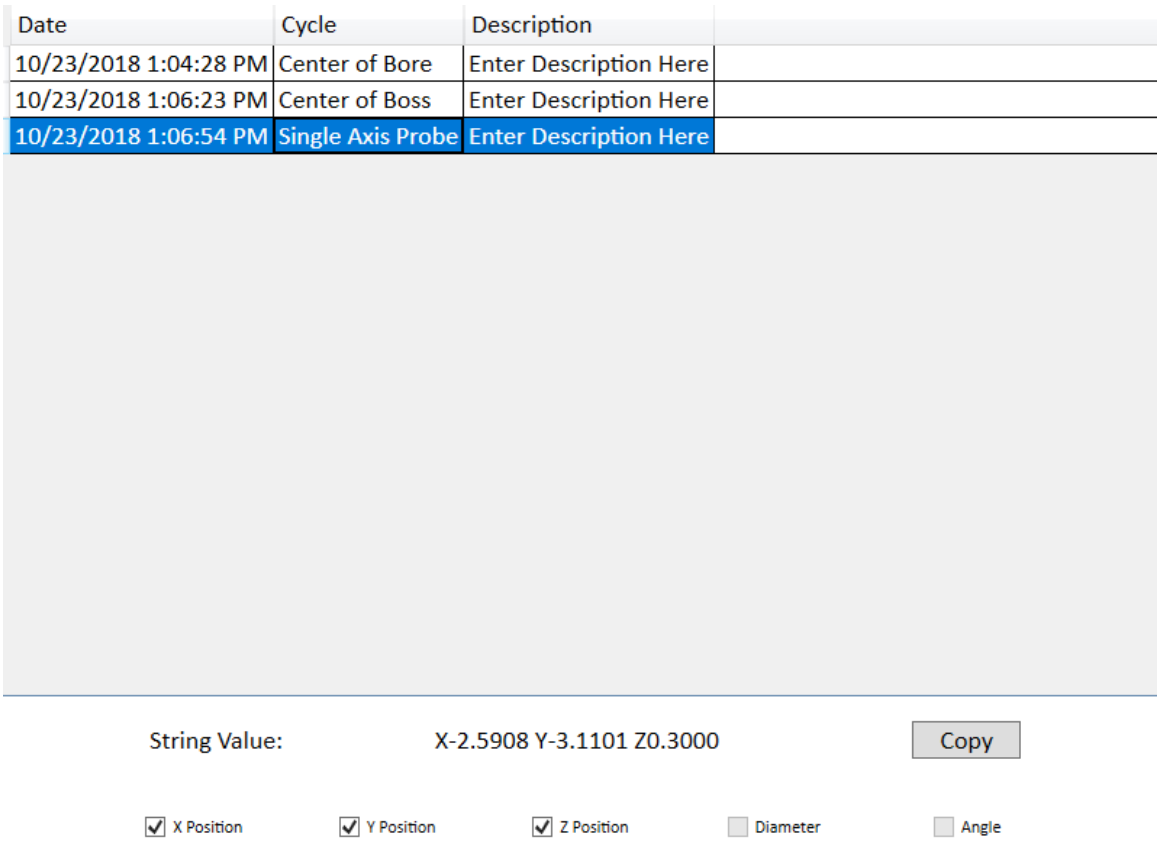
2.28.19 CTRL+I – Save PLC state to file

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**.

Draft: January 11, 2024

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 do not 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.

Draft: January 11, 2024

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

When you press **F1 – Setup** from the Main Screen, you will be shown the Setup menu containing options related to setting up various aspects of the machine.

Draft: January 11, 2024

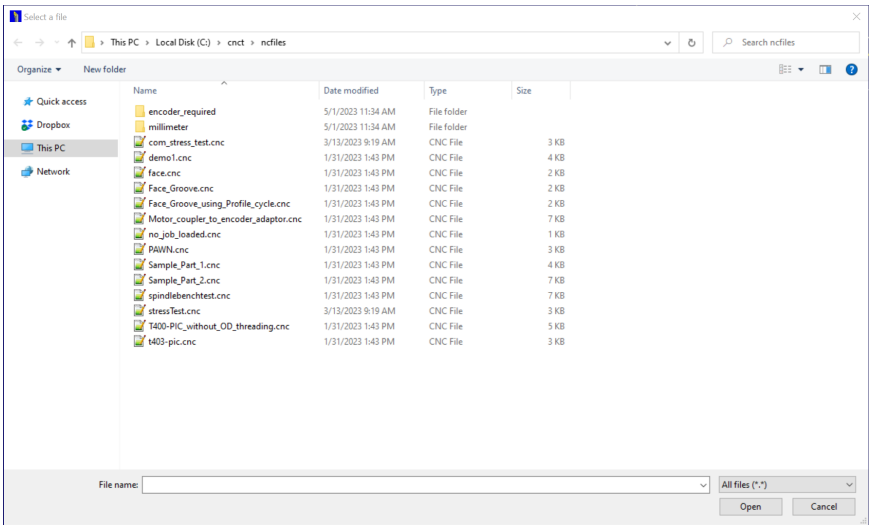
F1 – Part: This key displays the Part Setup menus, which are explained in Chapter 5.

F2 – Tool: This key displays the Tool Setup menus, which are explained in Chapter 4.

F3 – Config: This key displays the Configuration menu, which is explained in Chapter 11.

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

3.2 F2 – Load Job Menu



Draft: January 11, 2024

3.3 F3 – MDI

MDI mode allows you to directly enter M and G-codes one line at time. After entering the M- and G-codes 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:

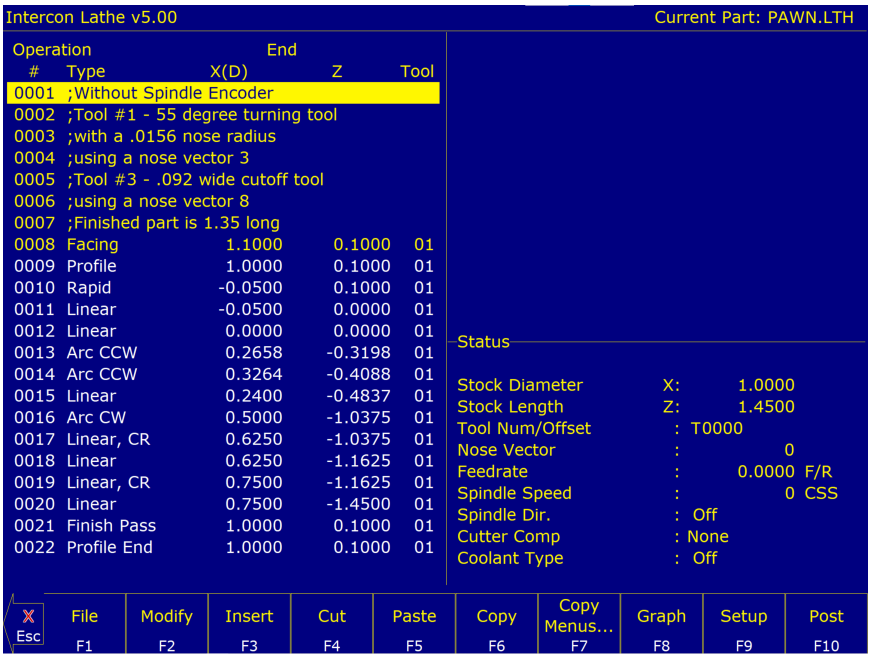
```
Block? G50X0Z0 ; Set the current XZ position to 0,0.
Block? M26 /Z  ; Set the current Z position as Z home.
```

Draft: January 11, 2024

3.4 F4 – Run Menu

Press the **F4 – Run** key to change the way your part program will run. See Chapter 6 for more information concerning the run menu.

3.5 F5 – CAM



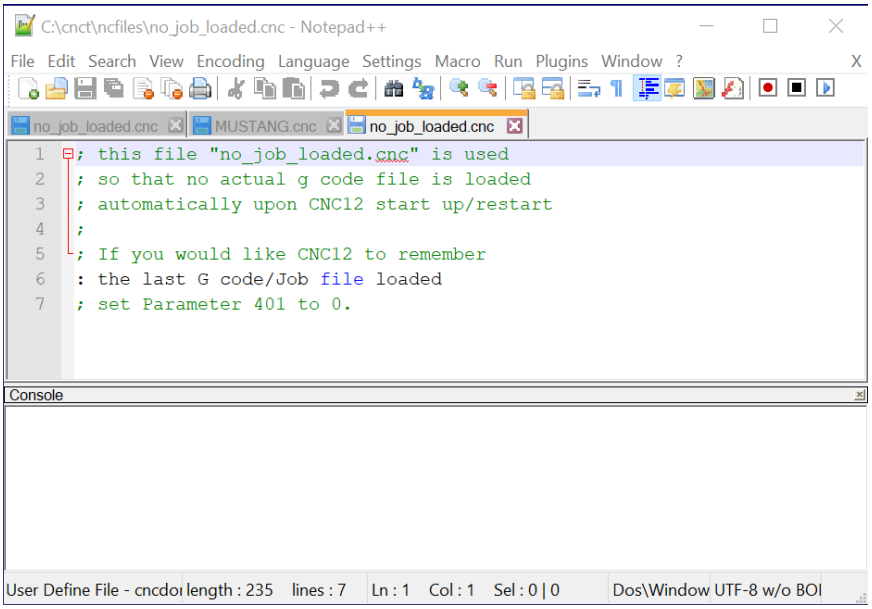
Press the **F5 – CAM** key from the Main Menu, then the **F1 – ICN** key to enter Intercon (**Inter**active **Con**versational) software. When you exit Intercon software, you will return to the Control Main Screen. The posted Intercon program will be automatically loaded into CNC12.

F1 – ICN: Lathe Intercon conversational programming.

Draft: January 11, 2024

F2 – Manual: Allows you to access the Operator Manual on the control.

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.

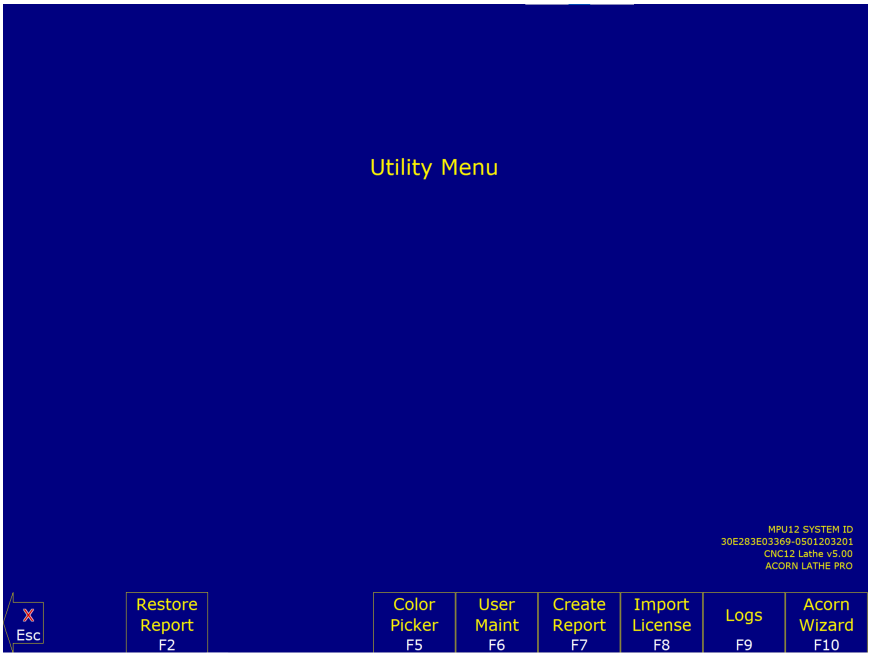
⚠ **WARNING** Editing a file (modifying and saving) **while the machine is moving** can cause personal injury or machine damage.

Also, note that the `C:\CNCT` 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:\CNCT` directory. Doing so can cause personal injury or machine damage.

Draft: January 11, 2024

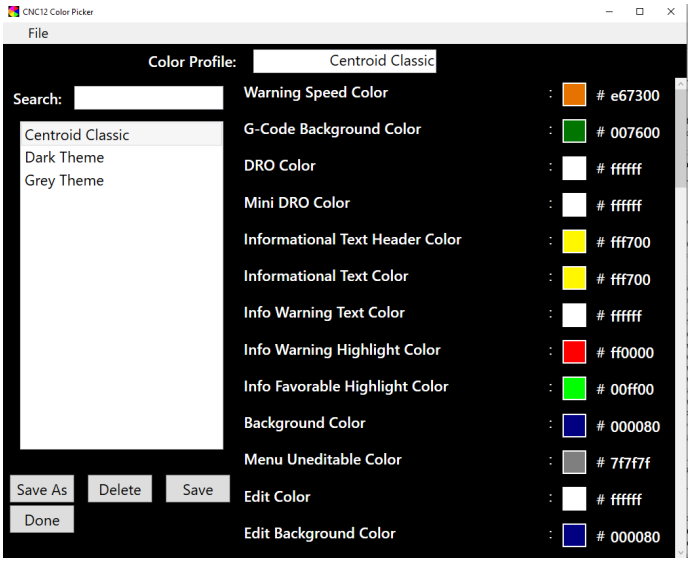
3.7 F7 – Utility



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.

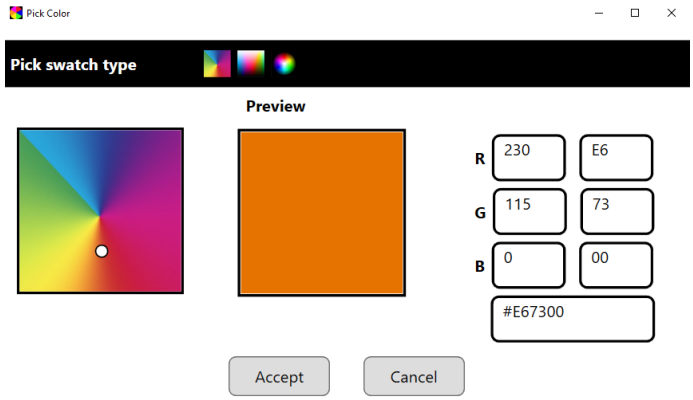
F2 – Restore Report: Update your control's configuration with a report.zip file.

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 made to Centroid Classic and saved, and a return to the original is desired, select File, then New, and the next profile will start with Centroid Classic settings.

Draft: January 11, 2024



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 you are finished modifying the 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 **F2 – Manual** key will open a PDF of the CNC12 Operaters manual. The **F3 – Machine Notes** key opens 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 cnct folder.

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.

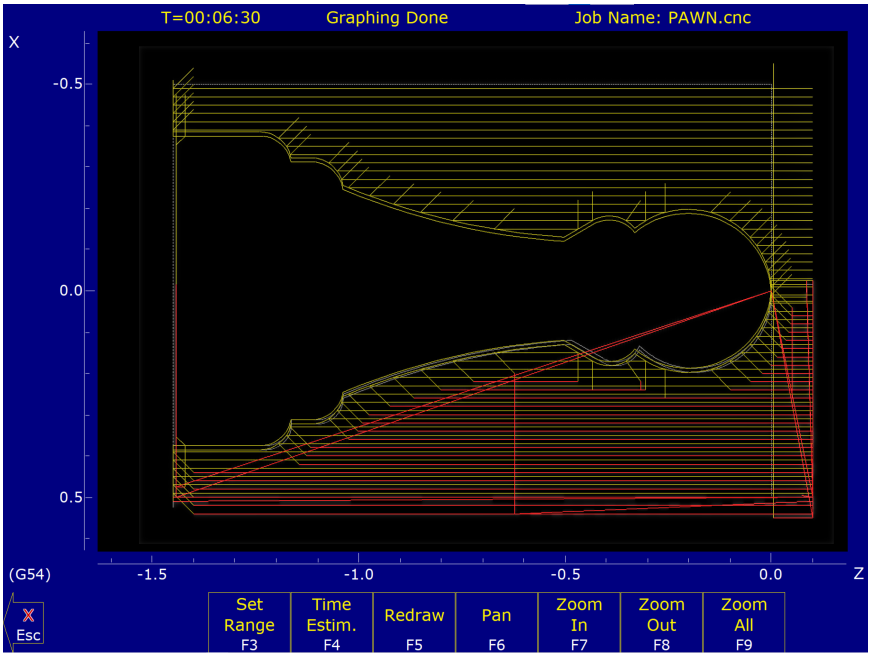
- **F1 – Errors:** Displays the error/message log. Use Page Up, Page Down, Home, and End to view and ESC to exit.
- **F2 – Stats:** Displays counts of errors logged. Use Page Up, Page Down, Home, and End to view and ESC to exit.
- **F3 – Export:** Exports the log to a destination of your choosing.

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

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.

Draft: January 11, 2024



A wireframe tool path of your part should appear. Each axis is indicated by the X- or Z-marker, along with scales to indicate the current location of the part. Here is a list and the function of the F-keys located on the bottom of the screen:

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

F4 – Time Estimation: 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 graphics at any time.

F6 – Pan: Press this key to move the part around the graph. Once pressed, use the crosshatches to pick a location of the part that will pan to the center of the graph. Once a section is selected, press the **F6 – Pan** key again to continue panning.

F7 – Zoom In: Press this key to zoom into the part relative to the center of the graph.

F8 – Zoom Out: Press this key to zoom away from the part relative to the center of the graph.

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

Note: Use the FEED RATE 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 FEED RATE OVERRIDE knob is controlled by pressing either Ctrl + or Ctrl -.

Draft: January 11, 2024

3.9 F9 – Tool Wear

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 the 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 pressed 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 shut down the control. With most controls, this action turns off the control once

Draft: January 11, 2024

the system has prepared itself to be shut down. Just like a desktop computer, the control should be properly shut down before turning off the power in order to reduce the risk of corrupting data on the hard drive.

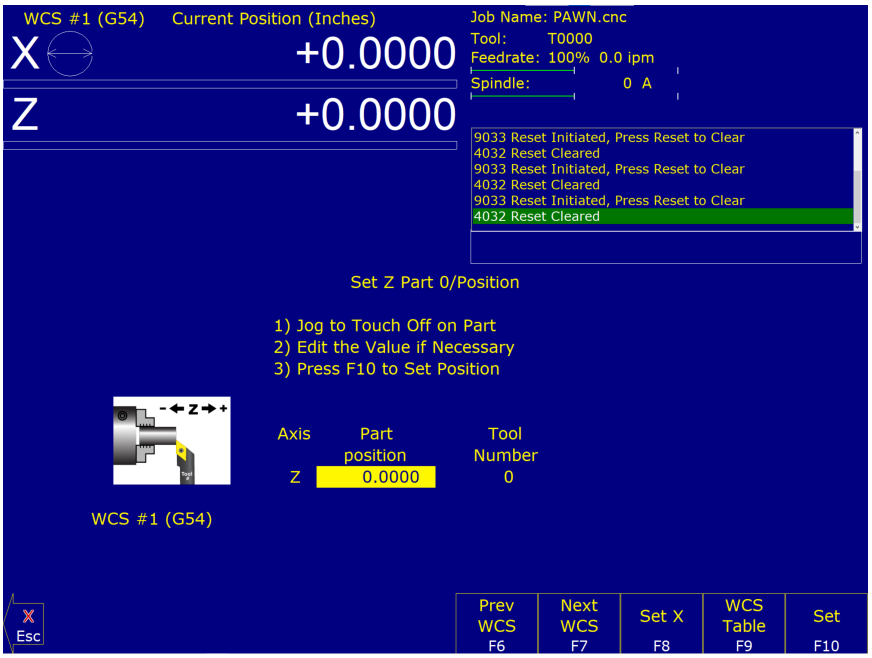
Note: This will only turn off the control. The machine itself will still need to be manually turned off.

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.

Draft: January 11, 2024

4 Part Setup (F1 from Setup)



The Part Zero menu fields and screen elements are described below:

Axis: This field shows which axis the Part Zero is being set up for. When the Part Zero menu is first brought up, the Z-axis will be shown. Press **F8 – Set X** to access the Part Zero menu for the X-axis.

Position: This field allows you to establish a non-zero offset between where the tool is and where you want the origin to be. On the X-axis, this is either a diameter or radius distance away from the part centerline that the tool tip is touching.

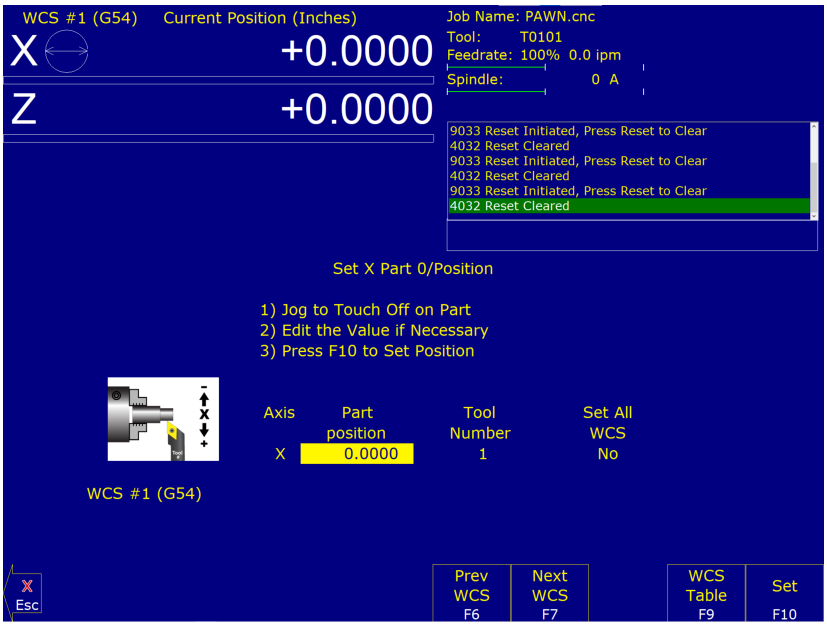
Note: The part centerline is usually considered to be where the X-axis position is 0.

Tool Number: This field allows you to tell the control what tool offset number (see the Offset Library in Chapter 4) is being used while setting the Part Zero position. Although this number is called a “Tool Number”, this is not a tool number. However, if you only associate tool numbers with the same numbered offset, then this field would correspond to the tool number.

Note: The Offset Library must be up to date before setting the Part Zeroes.

Set All WCS: This field appears only if you are modifying the Part Zero for the X-axis.

Draft: January 11, 2024



Press the **SPACE** bar to toggle between YES and NO. If this field is toggled to “YES”, then this field specifies that the position that you enter will be copied to all the X-axis Part positions in every Work Coordinate System. This will cause all Work Coordinate systems to have the same X-axis Part Zero. This feature is a convenience, since the centerline position of a part is usually set at X=0, regardless of which WCS is currently active. If this field is toggled to “NO”, then only the currently-selected WCS will be affected.

F6 – Prev WCS: This key will select the previous Work Coordinate System. If you will be using multiple work coordinates, you must set up a new set of Part Zeros for each work coordinate. Each work coordinate represents a different Part Zero. You can use this key to cycle through all available Work Coordinate Systems. Note that WCS #1–6 are standard, but WCS #7–18 are an extra-cost option.

F7 – Next WCS: This key is like the **F6 – Prev WCS** key (see above) except that this key will cycle forward to the next work coordinate system. You can use this key to cycle through all available Work Coordinate Systems. Note that WCS #1–6 are standard, but WCS #7–18 are an extra-cost option.

F8 – Set X: To get access to the Part Zero menu for the X-axis, press **F8 – Set X**. Setting the X-axis Part Zero is given special treatment in a sub-menu because it is not done very often (see the section titled “Setting X-axis Part Zero” later in this chapter).

F9 – 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.

F10 – Set: Pressing this key will cause the part position that you entered to be set.

4.1 Setting Part Zeros – Introduction

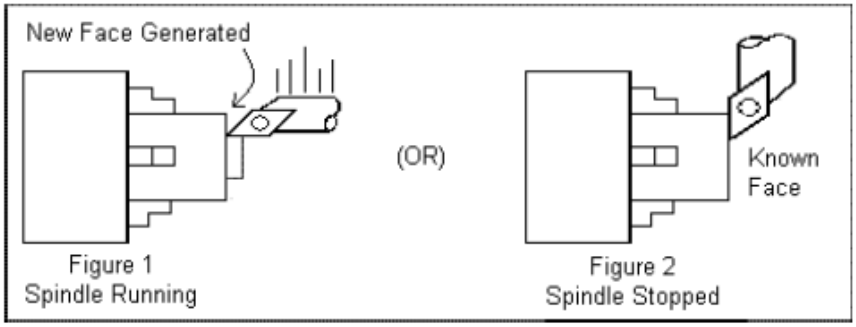
Setting the Part Zero for a part establishes a local coordinate system with its origin at the centerline of the part. In T-series controls, this coordinate system considers X+ as always pointing away from the centerline and Z+ always pointing to the right and away from the spindle.

Draft: January 11, 2024

4.1.1 Setting Z-axis Part Zero (Z_0)

STEP 1:

Jog the tool to the stock surface and take a skim cut across the face (Figure 1), or touch-off of the known surface (Figure 2), and leave the tool setting at this Z-position.

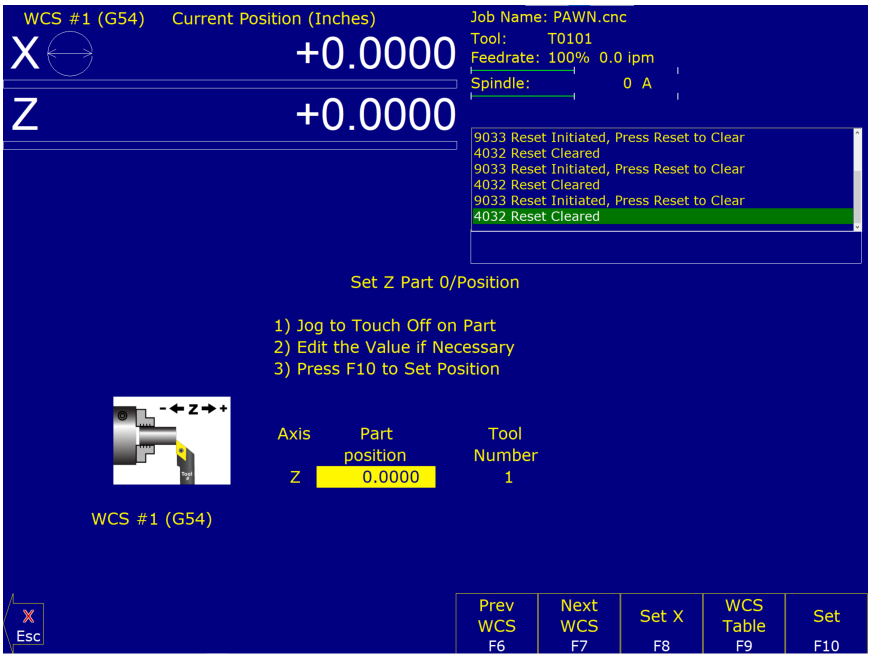


Note: In the case of Figure 1, start the spindle by switching to manual mode, pressing the **Spin Start** button, and adjusting the RPM with the spindle override.

Draft: January 11, 2024

STEP 2:

On the T-series Control, from the Main Screen, press they **F1 – Setup** key then the **F1 – Part** key. This will bring you to the Z-axis Part Zero menu.



STEP 3:

Type 0.000 (or the known position of the surface you are touching off) into the Part Position field. Press the **Enter** key.

Note: If, for example, you need to take a 0.05” face cut off of your part, type 0.05 into the Part Position field on the menu. Z-zero will now be 0.05” deeper into the part from the existing face.

STEP 4:

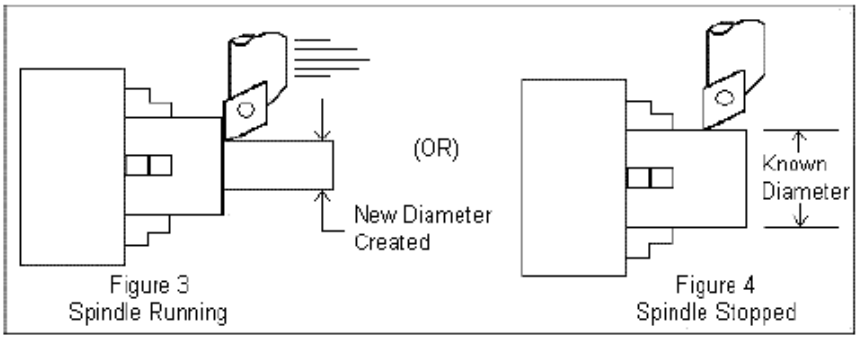
Enter the Tool Number of the tool being used, and then press the **F10 – Set** key. Part Zero is now set for the Z-axis. All the other tools set up in the Tool Library (Chapter 4) are now automatically set to this new Z-axis Part Zero.

4.1.2 Setting X-axis Part Zero (X_0)

Note: Since the X-axis Part Zero is usually defined to be the Centerline of the part, there is usually no need to set it up again when doing a different part. An ideal situation would be that you program all parts to have a Centerline of X=0, and thus you would need to set up the X-axis Part Zero for every WCS only one time during the whole life of the machine.

STEP 1:

Chuck up the stock to be machined. Jog the reference tool (in this case, an OD turning & facing tool) to the stock surface and take a skim cut across the surface (Figure 3), or touch-off of the known surface (Figure 4), and leave the tool setting at this X-position.



Note: In the case of Figure 3, start the spindle by switching to manual mode, pressing the **Spin Start** button, and adjusting the RPM with the spindle override.

STEP 2: Measure the Resulting Diameter

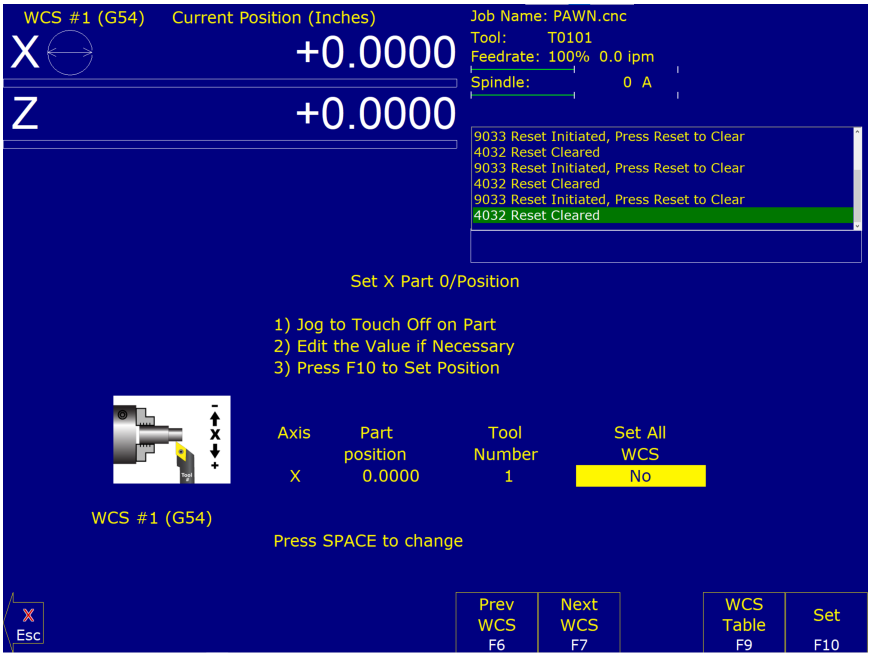
On the T-series Control, from the Main Screen, press the **F1 – Setup** key, the **F1 – Part** key, then the **F8 – Set** key. The **F6 – Prev WCS** and **F7 – Next WCS** keys can be used to select the work coordinate.

Note: There are 18 different work coordinates that can be used (1 through 6 are standard; 7 through 18 are an extra-cost option). See “Setting a WCS” later in the chapter.

STEP 3:

Enter the OD measurement taken in Step 2 into the Part Position field and press the **Enter** key.

Draft: January 11, 2024



Note: Depending on how your control is set, this value can be a diameter or radius. See Machine Parameter 55 for further details.

STEP 4:

Enter the Tool Number of the tool being used, and then press the **F10 – Set** key. Part Zero is now set for the X-axis. All the other tools set up in the Tool and Offset Libraries (Chapter 4) are now automatically set to this new X-axis Part Zero.

OPTIONAL STEP:

If you want all Work Coordinate systems to have the same X-axis Part Zero, then toggle the “Set all WCS” field to “Yes” and press the**F10 – Set** key. This will copy the position that you entered to all the X-axis Part positions in every Work Coordinate System. This feature is a convenience, since the centerline position of a part is usually set at X=0, regardless of which WCS is currently active.

Draft: January 11, 2024

4.2 Work Coordinate Systems (WCS) Configuration

Press the **F9 – WCS Table** key 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.

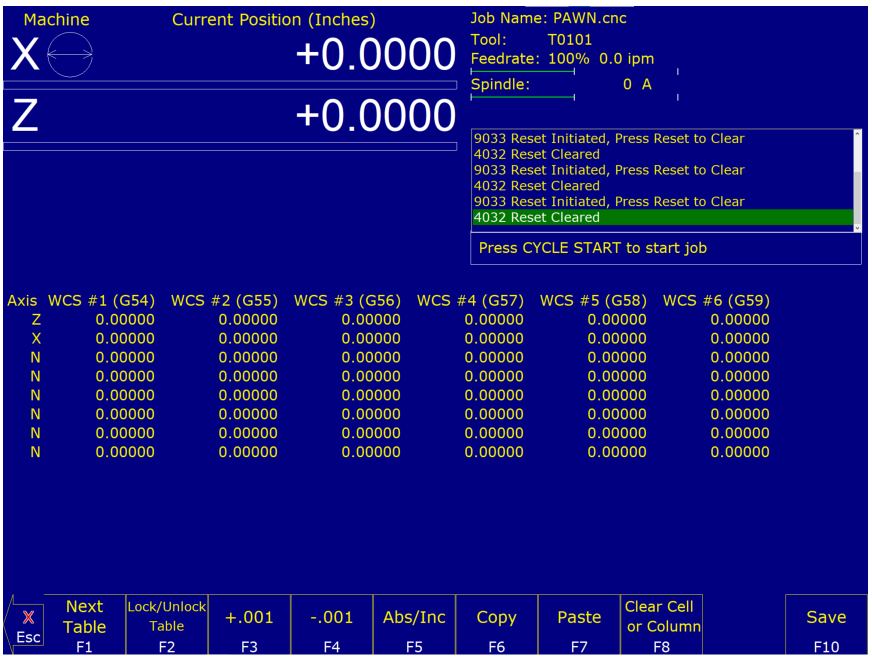
When you enter the Work Coordinate System Configuration screen, the DRO display will automatically switch over to machine coordinates as an aid to entering numbers. All the values on this screen are represented in machine coordinates. X-values are radius dimensions, even if the machine is in diameter mode (set in Machine Parameter 55). Note that WCS #1–6 are standard, but WCS #7–18 are an extra-cost option.

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

Draft: January 11, 2024

The G28 position (Return #1) is of interest because it specifies the Tool Check position and the usual Tool Change position. The Tool Check position is the machine coordinate position that the machine will move to when the **TOOL CHECK** button is pressed. Also, the G28 position is the usual position at which tool changes occur during a job run. You can change the G28 position if you would like the Tool Check position and tool changes to occur somewhere else. 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.



Draft: January 11, 2024

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

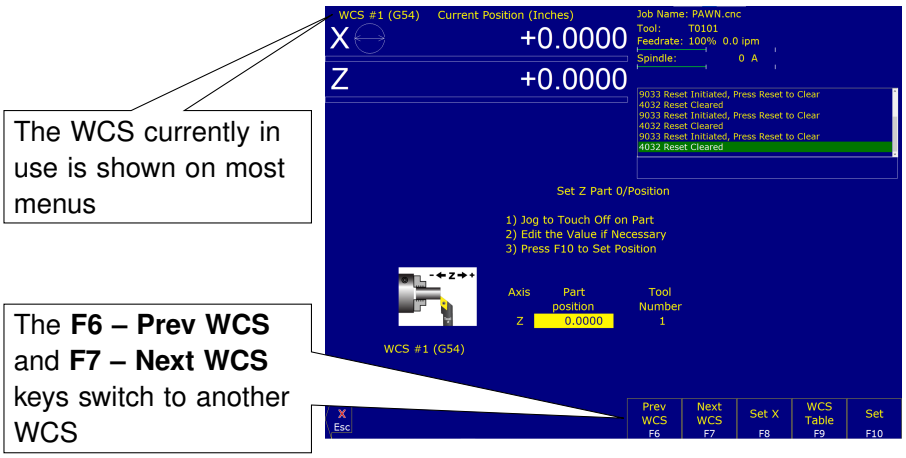
WCS	G-code
WCS #1	G54
WCS #2	G55
WCS #3	G56
WCS #4	G57
WCS #5	G58
WCS #6	G59

Extended Work Coordinate Systems

WCS	G-code
WCS #7	G54 P1
WCS #8	G54 P2
WCS #9	G54 PS
WCS #10	G54 P4
WCS #11	G54 PS
WCS #12	G54 P6

WCS	G-code
WCS #13	G54 P7
WCS #14	G54 PS
WCS #15	G54 P9
WCS #16	G54 P10
WCS #17	G54 P11
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).

Draft: January 11, 2024

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 Z, X and I, J 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.

Draft: January 11, 2024

5 Tool Setup

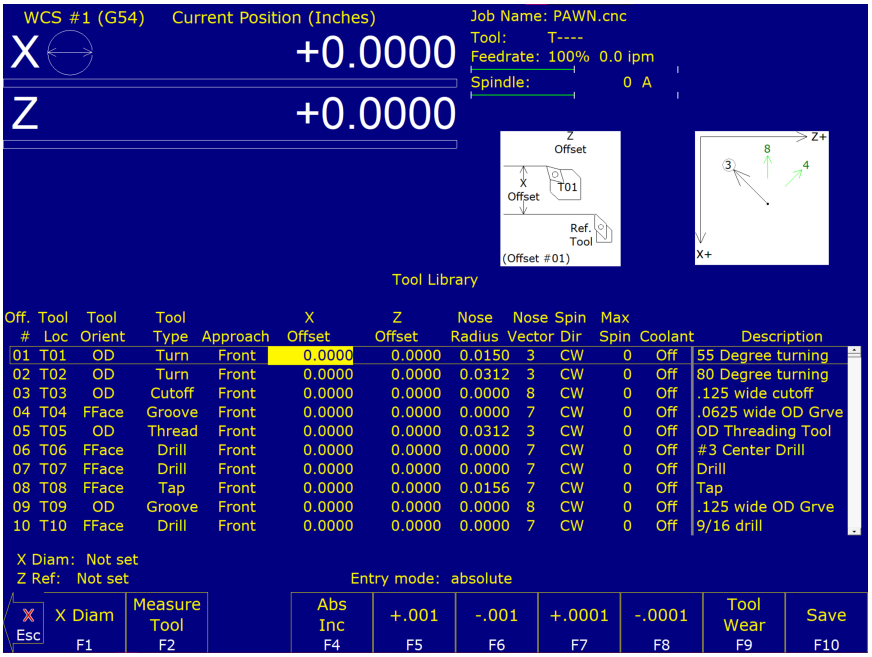
Three menus are involved in tool setup:

- Tool Geometry Offset Library – specifies tool and offset definitions to be associated with each tool.
- Tool Wear Adjustment Screen – allows the Operator to make tool wear adjustments for each tool.
- Lathe Intercon's Tool Library – Lathe Intercon's version of the Tool Geometry Offset Library.

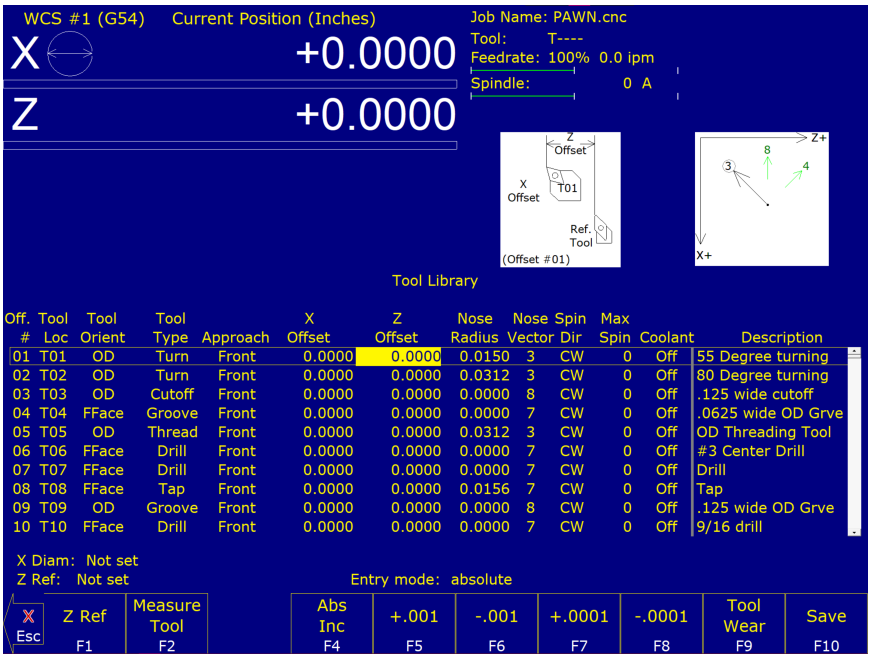
Only the first two menus will be discussed in this chapter. For information on setting up tool offsets, see the section "Procedures for Setting up Tools" later in this chapter.

5.1 Tool Geometry Offset Library

To get to this screen from the Main Screen, press **F1 – Setup** → **F2 – Tool Offset**. On this screen, you can define the characteristics and offsets to be associated with each tool.



Draft: January 11, 2024



Offset # (Tool Offset): This non-editable field serves as the row number reference into the Tool Library. In a CNC program, the number contained in the rightmost two digits of the T-command is used to activate a Tool Offset (for example T0102 activates Tool Offset #02). Activating a Tool Offset simply means that the tool offset data values (such as X- and Z-offsets, Nose Radius, and Nose Vector) on that row become the active modals until the next T-command.

Tool Loc (Tool Number): This is the Tool Number (01–99) that is associated with the Tool Offset. Normally, a single Tool Offset is associated with a single tool, but if a single tool is used in multiple ways, then it is possible for multiple Tool Offsets to be associated with that tool. To associate multiple Tool Offsets to a single tool, enter the same Tool Number on multiple rows.

Tool Orient: This is the tool operation direction, which generally describes the direction in which the tool points. It also implies the orientation of the work surface. Choices here are OD (outer diameter), ID (inner diameter), FFace (front face), and RFace (rear face).

Tool Type: This field describes the tool's general shape and intended operation. Use "Custom" if the tool does not match any of the available tool types.

Approach (from): This field is the tool approach direction. This indicates which side of work piece centerline the tool must travel *from* in order to start operation on the work piece. For gang tool lathes (Parameter 163=1), the approach direction is independently set for each tool. For non-gang tool lathes, this field is locked to the same approach direction for every tool and cannot be modified, and such an approach direction is based on the lathe's overall machine orientation (as set in Parameter 1).

X-offset: This field defines the X-offset distance away from the established tool measurement radius or diameter (see X Diam/Radius described below).

Z-offset: This field defines the Z-offset distance away from the Z-reference position (see Z-Ref described below).

Nose Radius: This field tells the control the distance to adjust when cutter diameter compensation (G41 or G42) is activated.

Nose Vector: This field tells the control how the tool is oriented in the machine. See the section titled "Setting the Nose Vector" later in this chapter for a more in-depth explanation.

Spin Dir (Spindle Direction): This is spindle direction for the tool. Toggle between off, clockwise, and counter-clockwise.

Draft: January 11, 2024

Max Spin (Max. Spindle Speed (G50)): The maximum spindle speed for the tool. A G50 is posted with the tool change using this value as the S parameter. If the value is zero, the G50 value from the Setup screen is used.

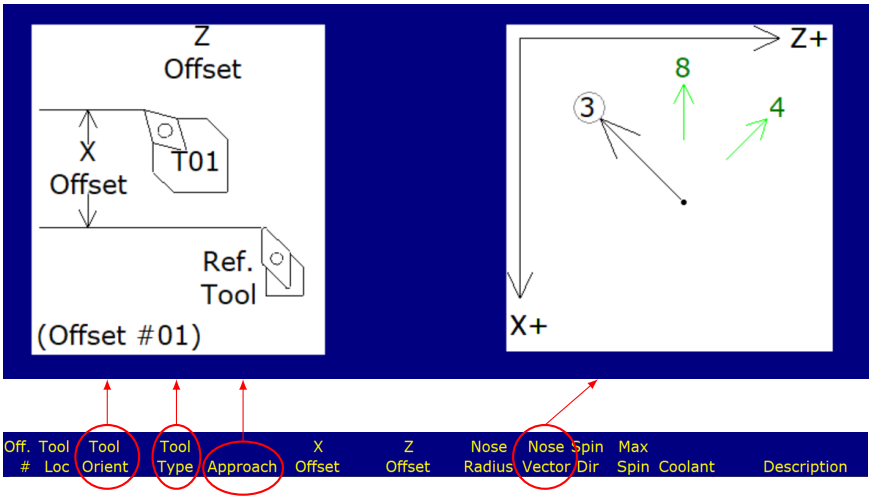
Coolant: Specify the coolant for each tool. Toggle between off, flood, and mist.

Description: This field contains the user-entered text description of the tool. This field is not essential for machine operation.

X-diameter/radius: This field defines the diameter or radius from which the X-offsets of tools are to be measured. This diameter is usually created by a skim cut as part of the tool measuring procedure (see the Procedures for Setting Tool Offsets section later in this chapter). To set the X-diameter field, cursor over to the X-offset column and press the **F1 – X Diam.** key and follow the instructions.

Z-ref: This field is the Z-reference position from which the Z-offsets of tools are to be measured. To set the Z-reference field, cursor over to the Z-offset column and press the **F1 – Z-ref.** key and follow the instructions.

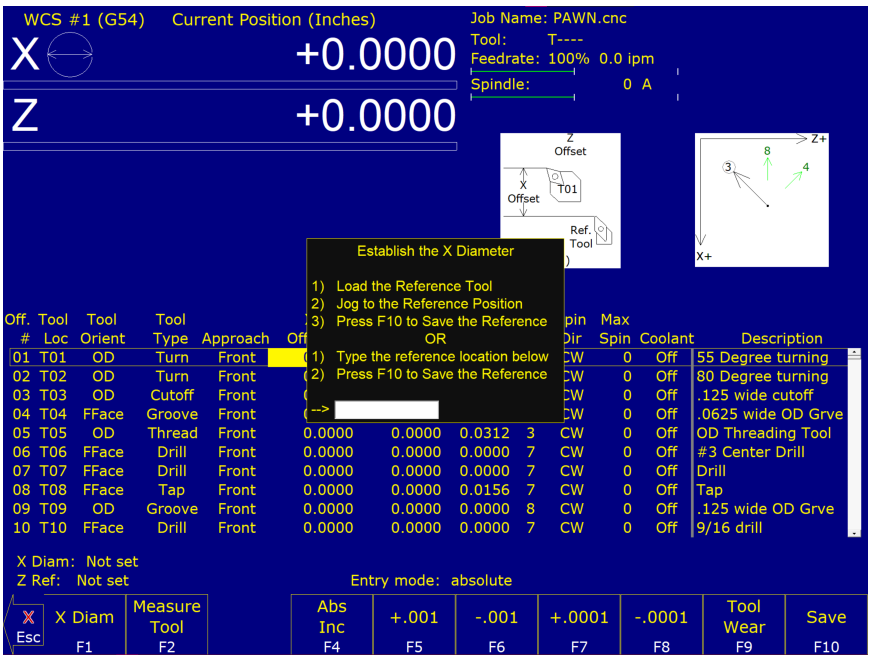
Entry Mode: You can toggle between absolute input and incremental input using the **F4 – Abs/Inc** key. The Entry Mode affects values entered in the X-offset, Z-offset, Nose Radius, X-diam/radius, and Z-ref fields. If the Entry Mode is Incremental, then the value that you enter will be **added** to the currently-affected field. If the Entry Mode is Absolute, then the value that you enter will change the field to that value.



The settings for **Tool Orient**, **Tool Type**, and **Approach** contribute to the appearance of the help diagrams on the upper-right side of the screen. This will help you select a proper **Nose Vector** for the tool.

Draft: January 11, 2024

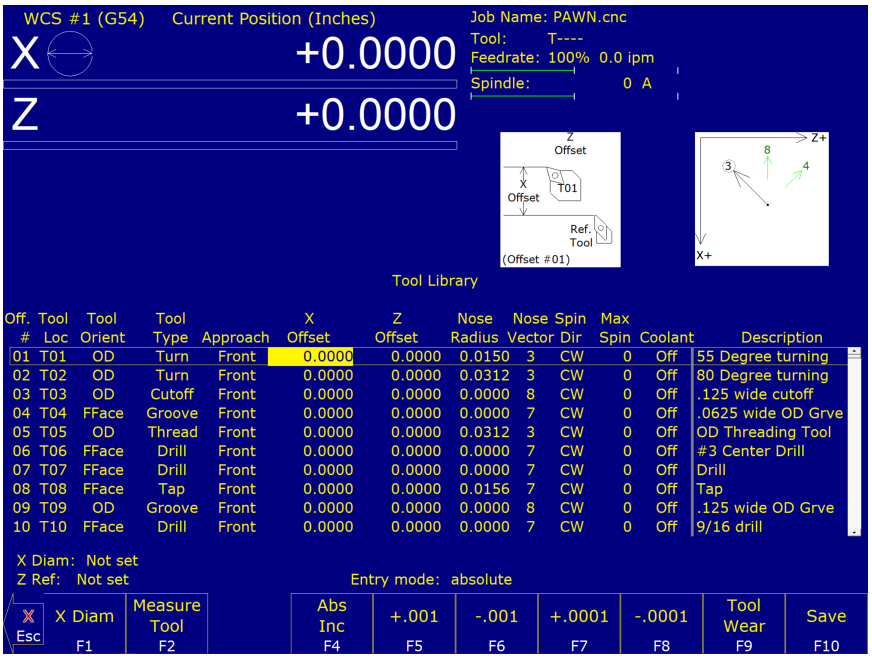
F1 – X-diam/rad or Z-ref



Press this key to establish the X-radius or diameter for Tool measurement or to establish the Z-reference. To establish the X-radius or diameter, move the cursor to the X-offset column and press this key and then follow the instructions. To establish the Z-reference, move the cursor to the Z-offset column and press this key and then follow the instructions.

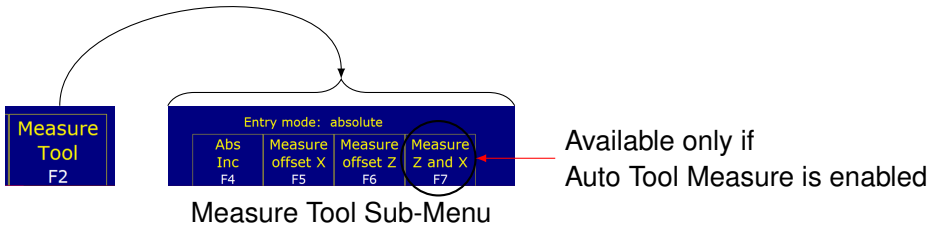
If Auto Tool Measure (an extra-cost option) is set up and enabled, you can establish the X-radius/diameter and/or the Z-reference with a tool touch-off device. When you press this key (after moving the cursor to the proper X- or Z-offset column), you will be prompted for a choice to do this either automatically or manually. On this prompt, choose the **F1 – Auto** key if you want to do this automatically with a tool touch-off device.

F2 – Measure Tool



Draft: January 11, 2024

Press this key to access a sub-menu for making X- and/or Z-offset measurements of a tool.



Measure Tool Sub-Menu

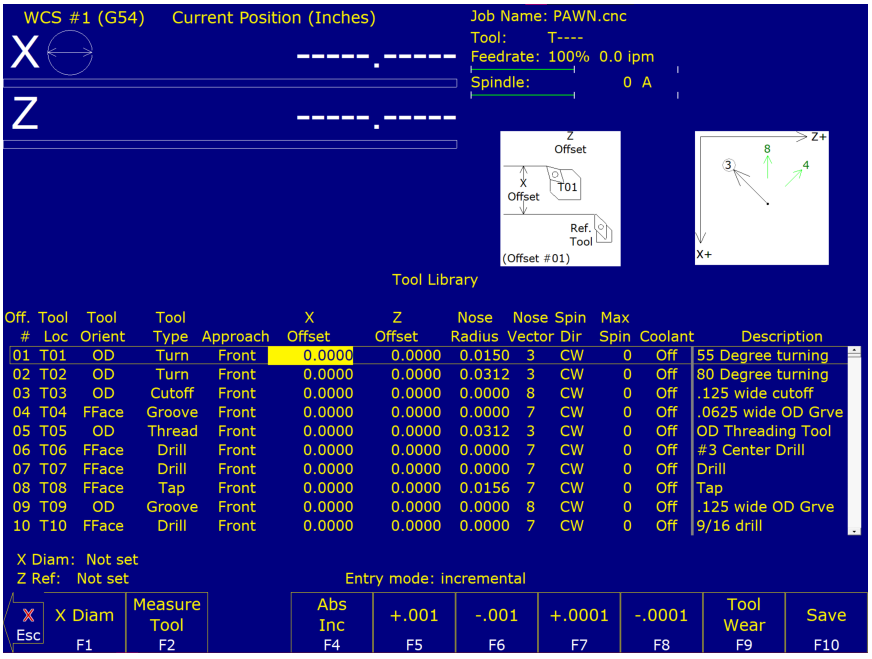
The sub-menu choices will behave slightly differently depending on whether or not Auto Tool Measure (an extra-cost option) is set up and enabled. This sub-menu is used in the tool measuring procedures later on in this chapter.

Sub-menu Key	Auto Tool Measure Not Enabled	Auto Tool Measure Enabled
F6	Manually measure X-offset using current X-position.	Prompt with a choice to do either Manual measure of X-offset **or** Auto Tool Touch-off cycle.
F6	Manually measure Z-offset using current Z-position.	Prompt with a choice to do either Manual measure of Z-offset **or** Auto Tool Touch-off cycle.
F7	(this choice will not show up)	Do Auto Tool Touch-off cycle starting with Z and then X.

F3 – ATC (Automatic Tool Change)

If you have an automatic tool changer installed, you can press this key to change tools.

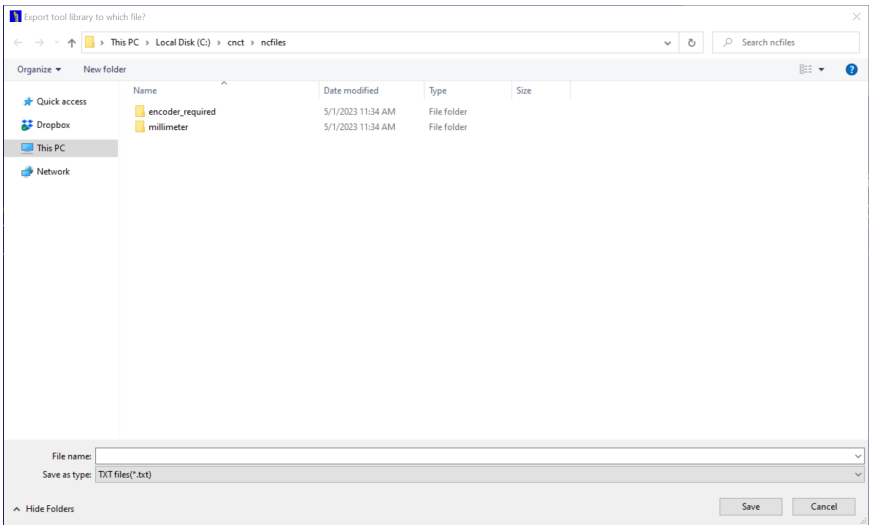
F4 – Abs/Inc



Draft: January 11, 2024

This toggles the Entry Mode between Absolute and Incremental (see “Entry Mode” as described above).

F5 – Export Lib. . .



The tool library can be exported in txt (space-separated and aligned columns) or csv (comma-separated columns) formats by pressing the **F5** key. Choose txt or csv to get the desired format.

F5, F6, F7, and F8 – Increment/Decrement by a small amount

To make small incremental adjustments to an X-offset, Z-offset, or Nose Radius value, use the arrow keys to select the value to be adjusted and press one of these keys. A small amount (as stated on the key’s label) will be added to or subtracted from the affected field.

F9 – Tool Wear

Draft: January 11, 2024

This is an alternate way to get to the Tool Wear Adjustment screen (see next section).

F10 – Save

When you are done with modifications, press this key to save the changes.

5.2 Tool Wear Adjustment Screen

To get to Tool Wear Adjustment screen from the Main Screen, press the **F9 – Tool Wear** key. This screen allows you to make tool wear adjustments for each tool. Adjustment values entered here will be **added** to the corresponding fields in the Tool Geometry Offset Library to obtain the final offset value to be used by the control during a job run.

Tool (Offset #): This is the **offset** number. Although this number *usually* corresponds to a tool number and is prefixed with a "T", this is **not** always a tool number. However, if you only associate tool numbers with the same numbered offset, then this field would correspond to the tool number. This field is just a display label and cannot be modified.

X: This is the distance adjustment for the corresponding X-offset field in the Tool Geometry Offset Library.

Z: This is the distance adjustment for the corresponding Z-offset field in the Tool Geometry Offset Library.

(Description): This field is displayed on this screen for your convenience. It cannot be modified here. To modify this field, press the **F2 – Tool Offset** key to go to the Tool Geometry Offset Library (described in the previous section) or go into Lathe Intercon's Tool Library.

Entry Mode: You can toggle between absolute input and incremental input using the **F4 – Abs/Inc** key. The Entry Mode affects values entered in the X and Z adjustment fields. If the Entry Mode is Incremental, then the value that you enter will be **added** to the current value in that field. If the Entry Mode is Absolute, then the value that you enter will be the value entered in that field.

Draft: January 11, 2024

F1 – Clear All

This key will set all X and Z adjustment values in all rows of this menu to 0.

F2 – Tool Offset

This is an alternate way to get to the Tool Geometry Offset Library (described in the previous section).

F3 – ATC (Automatic Tool Change)

If you have an automatic tool changer installed, you can press this key to change tools.

F4 – Abs/Inc

This toggles the Entry Mode between Absolute and Incremental.

F5, F6, F7, and F8 – Increment/Decrement by a small amount

To make small incremental adjustments to the X and Z adjustment fields, use the arrow keys to select the value to be adjusted and press one of these keys. A small amount (as stated on the key's label) will be added to or subtracted from the affected field.

F10 – Save

When you are done with modifications, press this key to save the changes.

Draft: January 11, 2024

5.2.1 Procedures for Setting Up Tools: Introduction

Follow these five steps to successful CNC turning:

1. Determine the tools necessary to machine the part by analyzing the print.
2. Set up the tool geometry characteristics and offsets for each tool (see Chapter 2).
3. Program the part using Intercon (see Chapter 7, Lathe Intercon Manual).
4. Set the X and Z Part Zero positions on the stock to be machined (see Section 4.1.2).
5. Graph the part to check for programming errors and then machine the part.

Tool offsets let the control know the difference in position for each tool being used. Since different tools are at different positions, each tool will have its own specific offset value in X and Z. For a multi-tool job, it is critical that the X- and Z-offsets for each tool are set at the proper values.

We will use the control to determine the difference in location of each tool by simply defining a position from which to measure each individual tool. If you have a tool touch-off device you can use the auto tool measure feature (see the end of this chapter). If you do not have a tool touch-off device, the easiest method is to make a skim cut and then

touch each tool off of the newly-measured skim cut diameter. The control will record the distance that each tool had to move to touch-off of the known diameter. Once the X- and Z-offset information is known for each tool, a multi-tool program can be run with success.

Before doing the procedures in the following sections, make sure:

1. The “Entry Mode” field in the Tool Geometry Offset Library is toggled to “absolute”.
2. The control is in Diameter mode (set Machine Parameter 55 to 0).
3. The adjustment values in the Tool Wear Adjustment Screen (described earlier in this chapter) are all zeroed out for the tools, which will be involved in the measurement process.

5.3 Setting up Tool Orientation, Tool Type, and Approach

STEP 1: Open the Tool Geometry Offset Library:

On the T-series Control Main Screen, press: **F1 – Setup** → **F2 – Tool Offset**.

Move the cursor to the row of the tool you want to set up.

STEP 2: Set the Tool Orientation:

Move the cursor to the Tool Orient column. The choices here are:

- OD (outer diameter) – Tool is designated to work on outside surfaces; tool is facing toward centerline.
- ID (inner diameter) – Tool is designated to work on internal surfaces; tool is facing away from centerline.
- FFace (front face) – Tool is designated to work on front-facing surfaces, tool is facing Z-.
- RFace (rear face) – Tool is designated to work on rear-facing surfaces, tool is facing Z+.

Because the choice of Tool Orientation limits the choices available in the Tool Type field (in Step 3), Tool Orientation should be set before the Tool Type.

STEP 3: Set the Tool Type:

Move the cursor to the Tool Type column. The choices here are:

- OD tool types: Turning, Grooving, Threading, Cutoff
- ID tool types: Turning, Grooving, Threading, Drill, Tap, Bore, Ream
- Front Face tool types: Turning, Grooving, Drill, Tap, Bore, Ream
- Rear Face tool types: Turning, Grooving

Choose ‘Custom’ if the tool is not any of the main ones listed.

Draft: January 11, 2024

STEP 4: Set Approach (For Gang Tool Lathes Only):

Choose the tool's Approach direction (this is the side of the work piece that the tool needs to approach from in order to start operation on the work piece). If your machine is not a gang tool lathe, then skip this step.

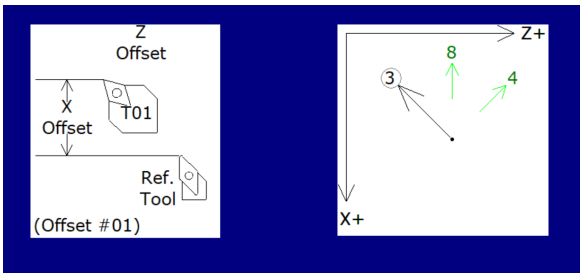
- For a horizontal gang tool lathe, the choice is either Front or Back.
- For a vertical gang tool lathe, the choice is either Left or Right.

Observe the Effect on the Help Diagrams:

Step 2: Tool Orient = OD

Step 3: Tool Type = Turn

Step 4: Approach = Rear



As you toggle through the choices of these fields, you will notice that the help diagrams in the upper-right part of the screen dynamically change according to your choices.

The diagram shown above on the left depicts the tool's general shape and orientation, as well as its relationship to the reference tool (which is the "Tool Used for Measurement" mentioned later in this chapter). The diagram shown above on the right depicts the suggested possible tool nose vectors for the choices made in Steps 2, 3, and 4.

5.4 Setting up the Nose Vector

Entering the Nose Vector for your tool will tell the control how that tool is oriented in the machine. This is needed for calculating cutter compensation and determining how to retract the tool during cutting cycles.

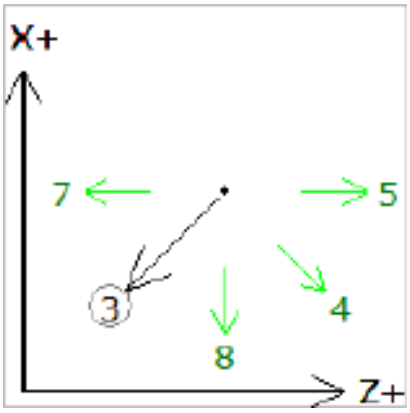
Note: It is best to have the Tool Orient, Tool Type, and Approach fields already set up for the tools involved before setting the nose vector.

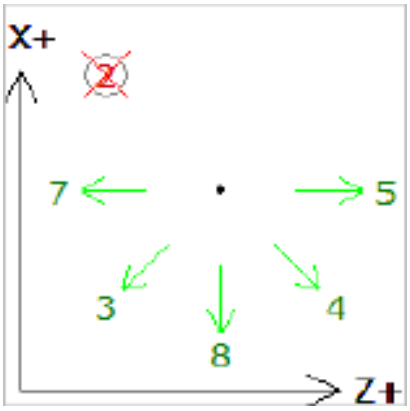
Move the cursor to the row of the tool you want to set up, and then move the cursor to the Nose Vector column. Select numbers 1 through 8. You can use the help diagram on the upper-right side of the screen to help you determine the proper nose vector number. The suggested possible nose vectors are determined by the Tool Orientation, Tool Type, and Approach.

If you choose one of the suggested node vector numers, the number on the diagram will be circled like this:

If you enter a nose vector number that is not suggested, it will appear crossed out like this:

Draft: January 11, 2024





Note that the diagram's suggestions may not cover all cases. Therefore, the user interface of the Tool Geometry Offset Library allows you to enter a nose vector number that is not suggested by the diagram.

For further information on how the nose vector is used during Cutter Compensation, see G40, G41, and G42 in Chapter 9.

5.5 Manual Measurement of X-axis Tool Offsets for OD Tools

Note: Before you begin, the adjustment values in the Tool Wear Adjustment Screen (described earlier in this chapter) should all be zeroed out for the tools that will be involved in the steps below.

Note: Also, it would be beneficial to have the Tool Orient, Tool Type, Approach, and Nose Vector fields already set up for the tools involved.

STEP 1:

Chuck up a piece of stock, and use the Jog buttons to make a skim cut (Figure 1). Leave the tool set at this X-position.

Note: Start the spindle by switching to manual mode, pressing the **Spin Start** button, and adjusting the RPM with the spindle override knob.

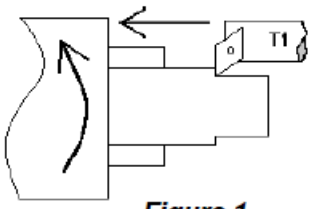


Figure 1

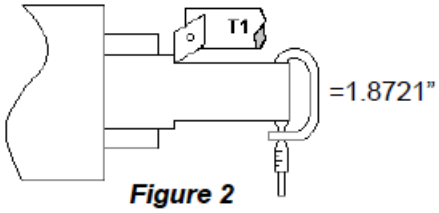


Figure 2

STEP 2:

Measure the new skim cut diameter, as shown in Figure 2.

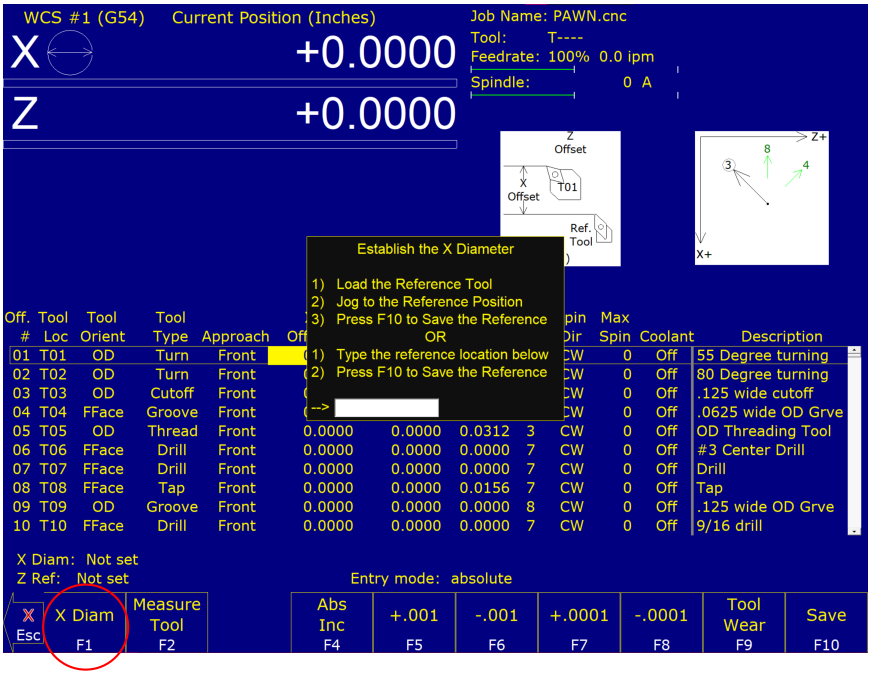
STEP 3: Open the Tool Geometry Offset Library

On the T-series Control Main Screen, press: **F1 – Setup** → **F2 – Tool Offset**.

Draft: January 11, 2024

STEP 4: Set the X-measurement Diameter

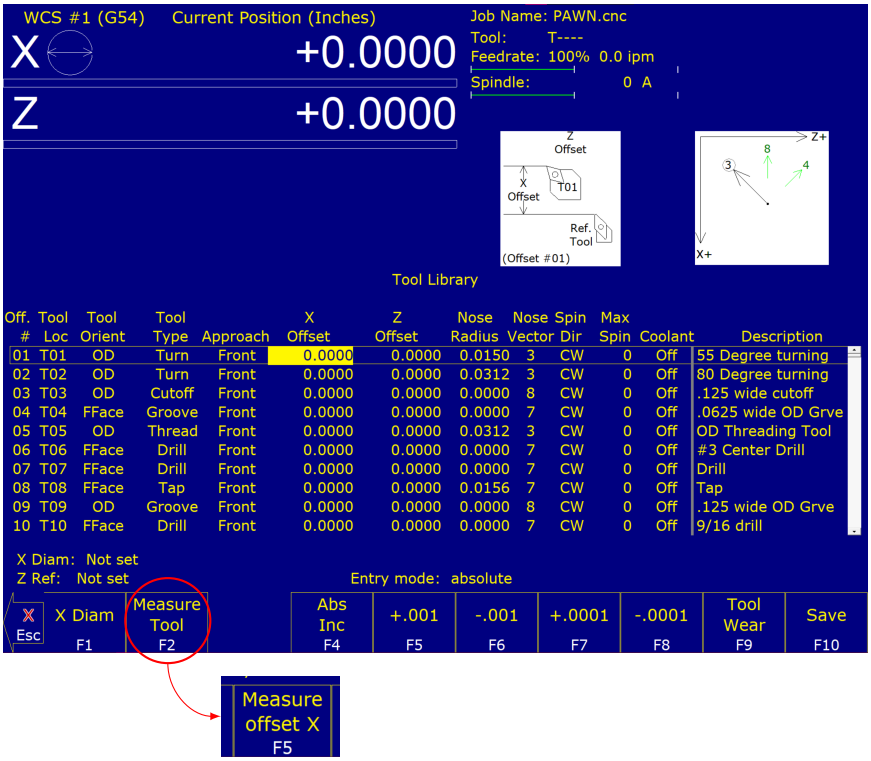
Move the cursor to the X-offset column and press the **F1 – X Diam** key. Enter the diameter measured in Step 2 into the “Establish the X-diameter” field and then press the **F10 – Save** key to accept. The X-measurement Diameter for OD tools is now set.



STEP 5: Measure the X-offset

Press the **F2 – Measure Tool** key then the **F5 – Measure Offset X** key to measure the X-offset of the tool used to make the skim cut. This value appears in the X-offset field.

Draft: January 11, 2024



Notes:

1. Always make sure the cursor is on the correct Offset #.
2. Press the **F5 – Measure Offset X** key while the tool is **STILL** at the skim cut diameter.
3. Any piece of stock can be used to set tool offsets. It is not necessary to use the actual part blank.

STEP 6: Measure the Next Tool

Touch the next tool to the new skim cut OD (the X-measurement Diameter) as shown in Figure 5, and press the **F2 – Measure Tool** key then the **F5 – Measure Offset X** key. Repeat this step for the rest of your OD tools.

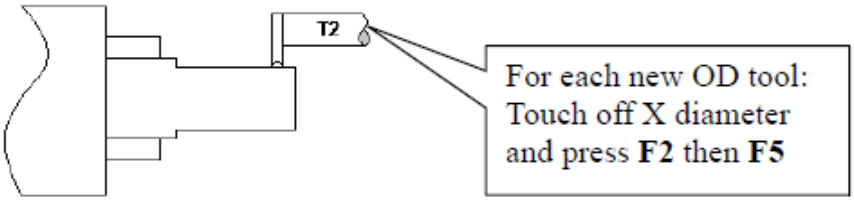


Figure 5

Notes:

1. Verify you are clear of any obstacles, then use “Tool Check” to withdraw the tool from its current position.
2. Use a piece of paper to touch-off the next tool to the skim cut diameter. Slowly jog closer to the work piece, switch to Incremental jog mode, and continue to slowly jog closer to the work piece in small increments until the tool barely pins the paper to the workpiece.
3. If you are using an ATC, be sure that you are clear of any obstacles, then use the ATC button in the Tool Library to rotate the ATC to the next tool position.

Draft: January 11, 2024

5.6 Manual Measurement of X-axis Tool Offsets for ID Tools

After setting all OD Tool Offsets, a new Internal X-measurement Diameter should be set to measure the X-offsets for all ID Tools.

Notes:

1. Before you begin, the adjustment values in the Tool Wear Adjustment Screen (described earlier in this chapter) should all be zeroed out for the tools that will be involved in the steps below.
2. It would be beneficial to have the Tool Orient, Tool Type, Approach, and Nose Vector fields already set up for the tools involved.

STEP 1:

Chuck up a piece of stock and use the Jog buttons to make a skim cut (Figure 6). Leave the tool set at this X-position.

Note: Start the spindle by switching to manual mode, pressing the **Spin Start** button, and adjusting the RPM with the spindle override knob.

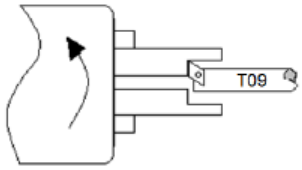


Figure 6

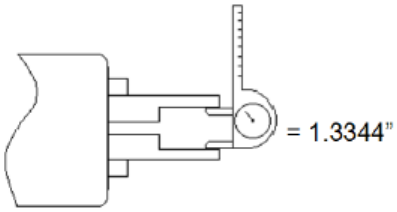


Figure 7

STEP 2:

Measure the new skim cut diameter, as shown in Figure 7.

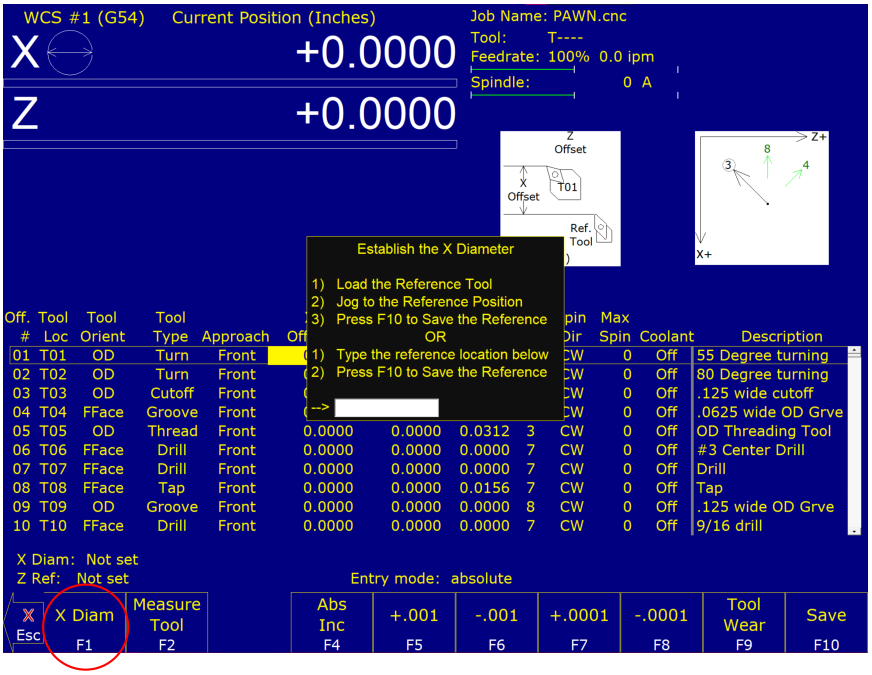
STEP 3: Open the Tool Geometry Offset Library

On the T-series Control Main Screen, press: **F1 – Setup** → **F2 – Tool Offset**.

STEP 4: Set the X-measurement Diameter

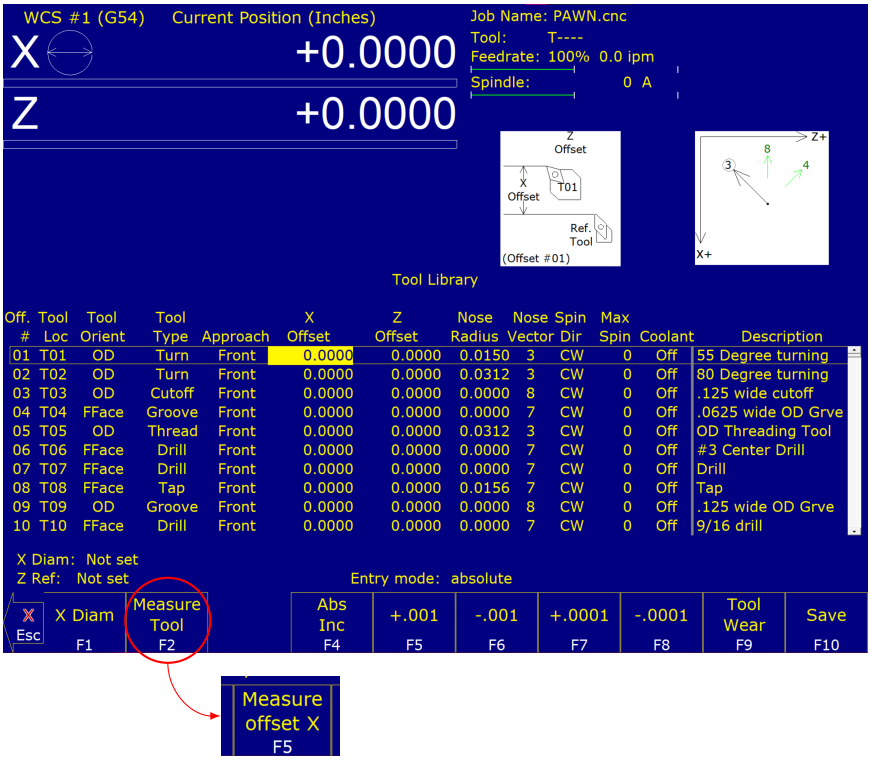
Move the cursor to the X-offset column and press the **F1 – X Diam** key, enter the diameter measured in Step 2 into the Establish the X-diameter field, and press the **F10 – Save** key to accept. The X-measurement Diameter for ID tools is now set.

Draft: January 11, 2024



STEP 5: Measure the X-offset

Press the **F2 – Measure Tool** key then the **F5 – Measure Offset X** key to measure the X-offset of the tool used to make the skim cut. The measured value will appear in the X-offset field.



Notes:

1. Verify that the cursor is highlighting the X-offset field for the offset number that you are measuring.
2. Press the **F5 – Measure Offset X** key while the tool is STILL at the skim cut diameter.

Draft: January 11, 2024

STEP 6: Measure the Next Tool

Touch-off all internal tools on this new internal diameter and press the **F2 – Measure Tool** key then the **F5 – Measure Offset X** key to measure each one. Repeat this step for all the remaining ID tools (Figure 5.1).

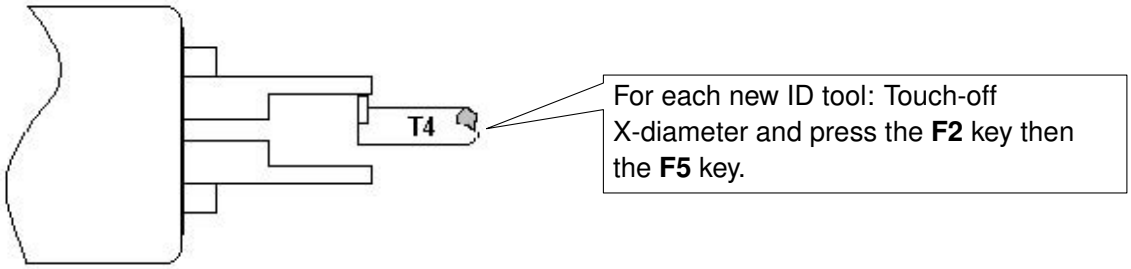


Figure 5.1

Notes:

1. Make sure you are clear of any obstacles, then use “Tool Check” to withdraw a tool from its current position.
2. Use a piece of paper to touch-off the next tool to the skim cut diameter. Slowly jog close to the work piece, switch to **Incremental jog** mode, and jog in close at small increments until the tool just pins the paper to the workpiece.
3. If you are using an ATC, move the ATC away from any obstacles, then use the ATC button in the Tool Library to index to the next tool position.

Special Cases: Sometimes it might be difficult to touch a new tool off of the X-measurement Diameter set in Step 2. If this is the case, you can repeat each step from #1 through 5 for EACH tool, reading in a new reference position for EACH tool. In this case, you will make a new skim cut Measurement Diameter for each tool and enter that new skim cut diameter as a new reference position for that tool. This method is more work, but if touching off a new tool to an existing reference position is very difficult, this method may be used for both OD & ID tools.

Draft: January 11, 2024

5.7 Manual Measurement of X-axis Offsets for Drills, Center Drills, and Taps

To set drills, center drills, taps, and boring tools, sweep the tool in with an indicator to find the spindle center. Remember that the X-measurement Diameter should be set to “0” before proceeding with Step 1 (see the section “Setting X-axis Tool Offsets for OD Tools” earlier in this chapter for directions on setting an X-measurement Diameter).

Notes:

1. Before you begin, the adjustment values in the Tool Wear Adjustment Screen (described earlier in this chapter) should all be zeroed out for the tools which will be involved in the steps below.
2. Also, it would be beneficial to have the Tool Orient, Tool Type, Approach, and Nose Vector fields already set up for the tools involved.

STEP 1: Set the Indicator

Mount the indicator base on the spindle or put the indicator in the chuck. Move the tool towards the approximate center of the spindle (see Figure 11).

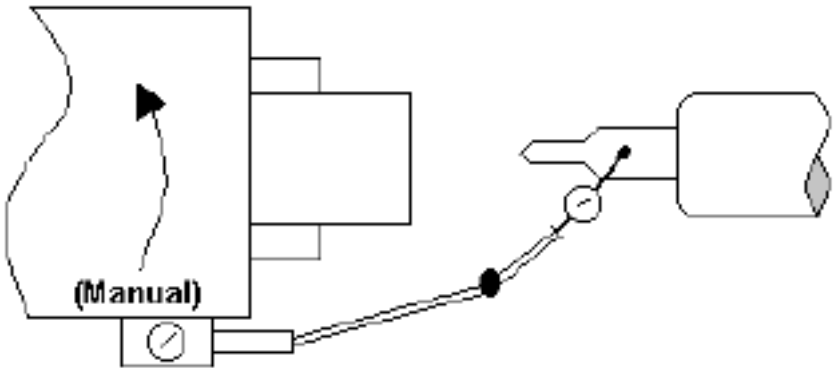


Figure 11

STEP 2: Center the Drill

Touch the indicator probe to the shank of the tool and rotate the chuck by hand. Jog the X-axis in Incremental mode until the indicator reads the same value all the way around the circumference of the tool.

STEP 3: Measure the X-offset

Press the **F2 – Measure Tool** key then the **F5 – Measure Offset X** key to measure the X-offset of the tool. The value appears in the X-offset field.

Note: This procedure may also be used in setting ID tool offsets in cases where an initial ID skim cut is not possible.

5.8 Manual Measurement of X-axis Offsets for Boring Tools

Since boring tools come with a manufactured offset, setting a boring tool is just like setting a drill, but with a few added steps. Follow Steps 1 through 3 in the previous section above, and then do the following steps:

Notes:

1. Before you begin, the adjustment values in the Tool Wear Adjustment Screen (described earlier in this chapter) should all be zeroed out for the tools which will be involved in the steps below.
2. Also, it would be beneficial to have the Tool Orient, Tool Type, and Approach fields already set up for the tools involved.

STEP 4: Find the Tool Offset

Look up the tool manufacturer's offset value for the tool being measured.

STEP 5: Switch to Incremental mode

With the X-offset field highlighted for the tool being measured, press the **F4 – Abs/Inc** key until the "Entry Mode:" field on the screen reads "Incremental".

Draft: January 11, 2024

STEP 6: Enter the Given Offset

Multiply the manufacturer's offset by negative two (-2) and type the number into the X-offset field. The value you type should appear as being added to the measured X-offset value that was already measured.

Note: Remember to press the **F4 – Abs/Inc** key to toggle the Entry Mode back to "Absolute" when you are done.

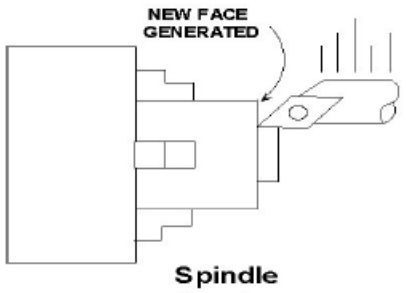
5.9 Manual Measurement of Z-axis Tool Offsets

Notes:

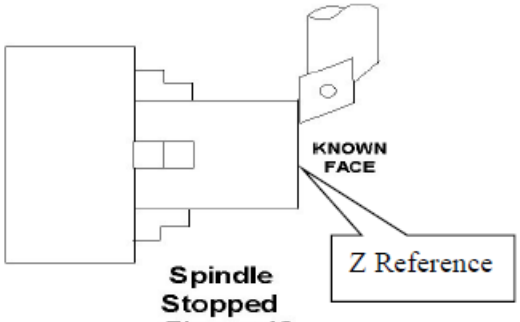
1. Before you begin, the adjustment values in the Tool Wear Adjustment Screen (described earlier in this chapter) should all be zeroed out for the tools which will be involved in the steps below.
2. Also, it would be beneficial to have the Tool Orient, Tool Type, Approach, and Nose Vector fields already set up for the tools involved.

STEP 1:

Chuck up a piece of stock and use the Jog buttons to make a skim cut (see Figure 12) OR if the surface is true, touch-off the end (see Figure 13).



Spindle
Running
Figure 12



Spindle
Stopped
Figure 13

STEP 2: Open the Tool Geometry Offset Library

On the T-series Control Main Screen, press: **F1 – Setup** $\rightarrow$ **F2 – Tool Offset**.

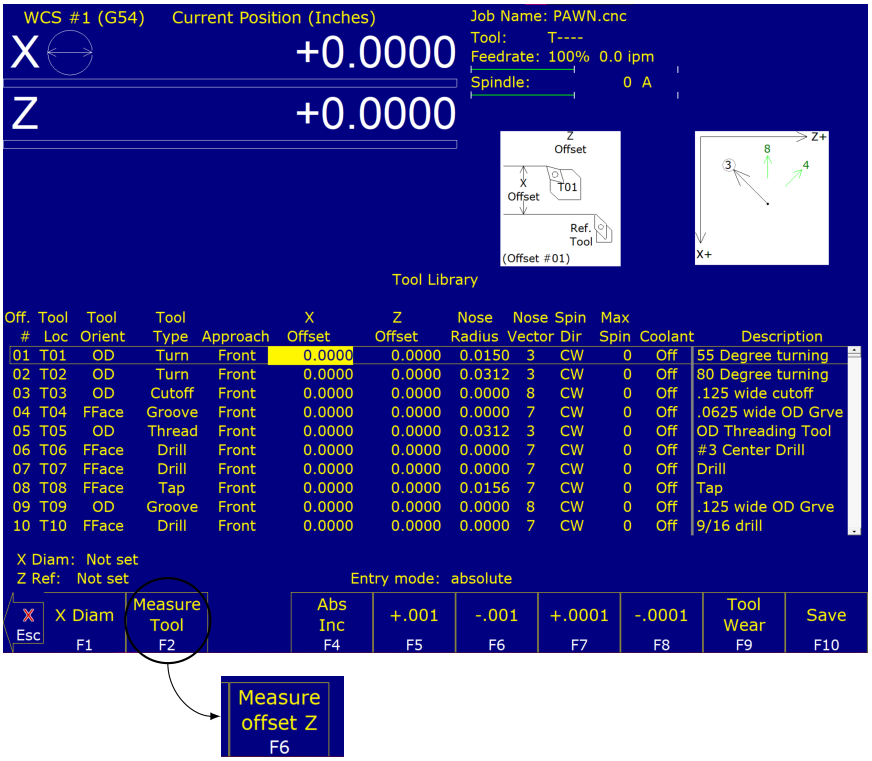
STEP 3: Set the Reference:

Make sure the Z-column is highlighted, press the **F1 – Z Ref.** key and then the **F10 – Save** key to accept this as the reference.

STEP 4: Measure the Tool Offset

Without moving the Z-position of the tool that you just used to set a reference point, press the **F2 – Measure Tool** key then the **F6 – Measure Offset Z** key to measure the Z-offset of that tool (it should result in 0 as its offset) as seen in Figure 14.

Draft: January 11, 2024



STEP 5: Measure the Next Tool Z-offset

Load the next tool and bring it to the reference point (as shown in Figure 13). Press the **F2 – Measure Tool** key then the **F6 – Measure Offset Z** key, and repeat for all remaining tools.

Note: Make sure the cursor is on the correct Offset number being measured before pressing the **F2 – Measure Tool** key then the **F6 – Measure Offset Z** key.

5.10 Manual Measurement of Part Cutoff Tool Z-offset

Notes:

1. Before you begin, the adjustment values in the Tool Wear Adjustment Screen (described earlier in this chapter) should be zeroed out for the tools that will be involved in the setup as described below.
2. Also, it would be beneficial to have the Tool Orient, Tool Type, Approach, and Nose Vector fields already set up for the tools involved.

Load the part cutoff tool and bring it to the stock face (Figure 15). With the menu highlighted in the Z-offset column at the correct offset number, press the **F2 – Measure Tool** then **F6 – Measure Offset Z** key.

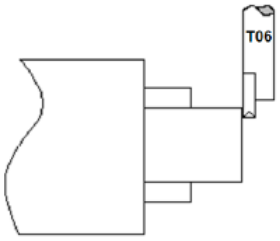
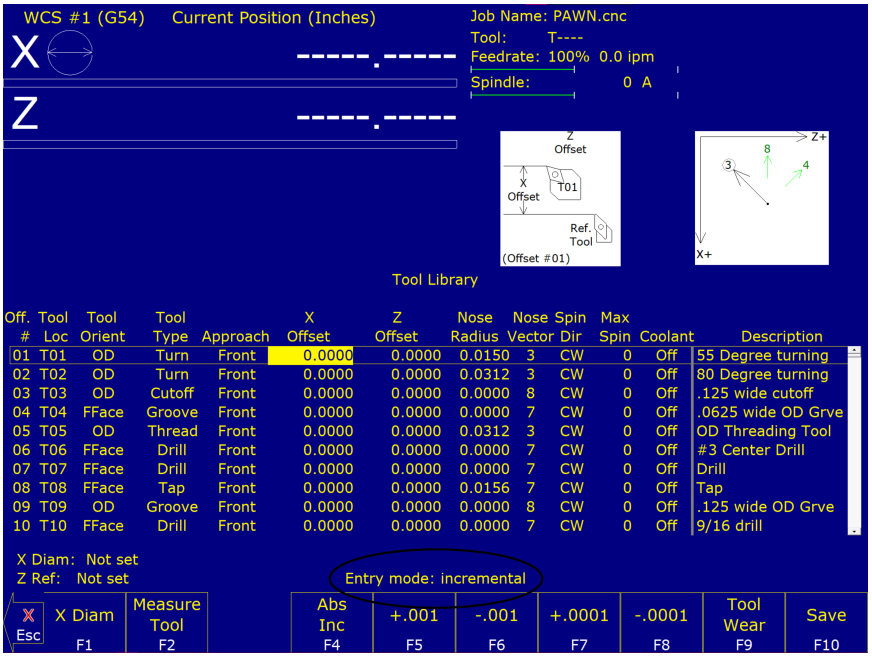


Figure 15

Draft: January 11, 2024

If the part cutoff tool is 0.125 wide and you want the back side of the tool to be set at Z-zero, then highlight the Z-offset of the tool being adjusted and press the **F4 – Abs/Inc** key to toggle to incremental mode.



Type in -.125 and press the **ENTER** key. The value of −0.125 will be added to the value measured in Step 1.

Note: Remember to press the **F4 – Abs/Inc** key to toggle the Entry Mode back to “absolute” when you are done.

5.11 Setting the Nose Radius

The Tool Geometry Offset Library also has a field for the tool Nose Radius (labeled **R** in Figure 17).

Draft: January 11, 2024

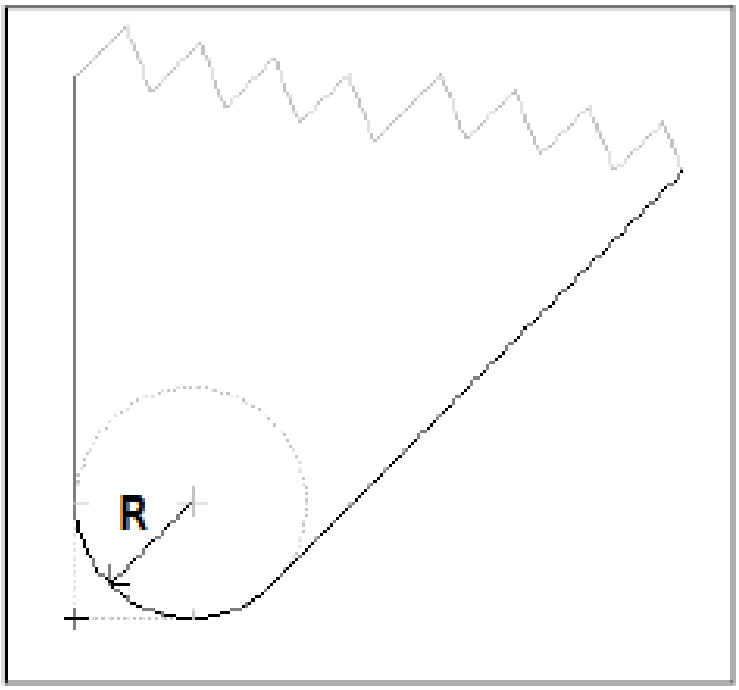
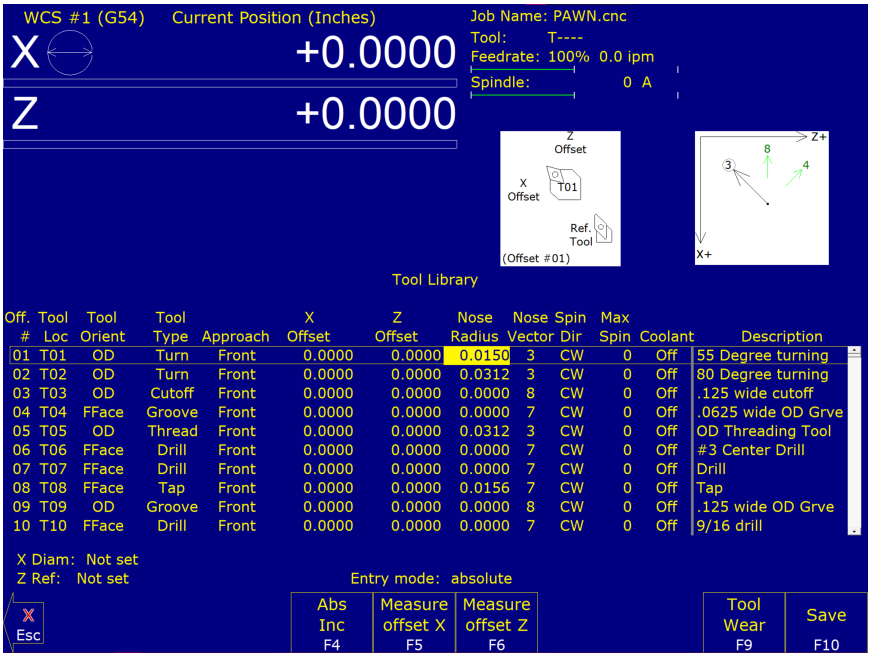


Figure 17

Draft: January 11, 2024

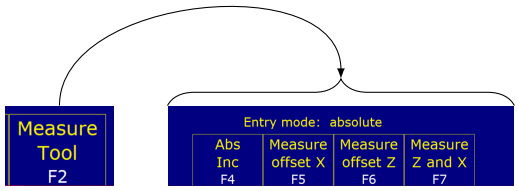
This field tells the control the distance to adjust when cutter compensation is used (G41 or G42). For further information, see G40, G41, and G42 in Chapter 9.



To edit these entries, first press the **F4 – Abs/Inc** key until the “Entry Mode” field reads “absolute”. Move to the desired Nose Radius field using the arrow keys and type in the nose radius of the tool, and press the **Enter** key.

5.12 Automatic Tool Touch-off cycles (extra-cost option)

Pressing the **F2 – Measure Tool** key will give you access to a sub-menu used for measuring X- and/or Z-offset measurements of a tool. When Auto Tool Measure (an extra-cost option) is set up and enabled, the functions under this sub-menu will give the Operator an additional choice of doing X- and Z-offset measurements automatically using a Tool Touch-off device.



Measure Tool Sub-menu

F5 – Measure Offset X: Prompt the Operator with a choice to do either Manual measure of X-offset **or** Auto Tool Touch-off cycle to measure X-offset.

F6 – Measure Offset Z: Prompt the Operator with a choice to do either Manual measure of Z-offset **or** Auto Tool Touch-off cycle to measure Z-offset.

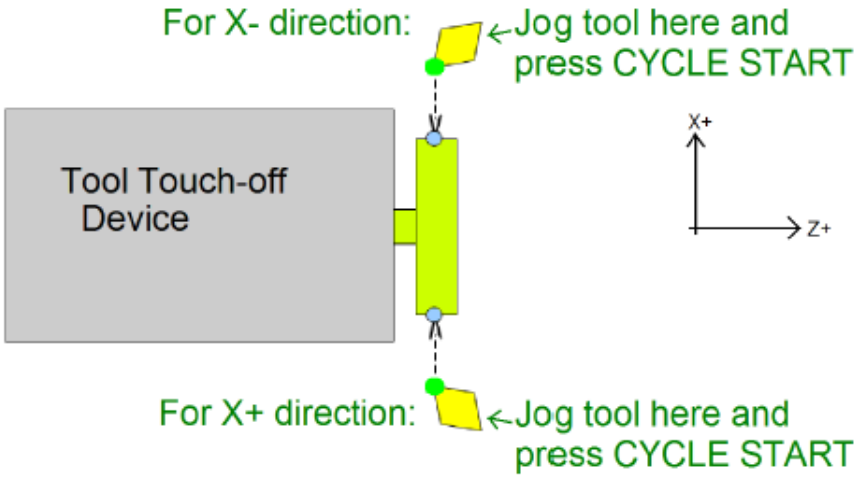
F7 – Measure Z and X: Do Auto Tool Touch-off cycles to measure both the Z- and X-offsets (starting with Z first).

The rest of this section will concentrate on the three Auto Tool Touch-off cycles accessed under these three keys. See Parameters 257 and 281–283 for information on setting up Automatic Tool Touch-off.

Auto Tool Touch-off Cycle to Measure X-offset (F5)

This cycle will do a single axis measuring move along the X-axis. The +/- axis movement directions are initially determined by the tool's Nose Vector and secondarily by its Approach Direction, but you can change the X-measurement direction by pressing the **F1 – Orient** key on the dialogue box. It is important to position the tool in the correct location before starting this cycle. To run this cycle from the Tool Geometry Offset Library, first jog the tool to the correct location and press:

F2 – Measure Tool → **F5 – Measure Offset X** → **CYCLE START**.

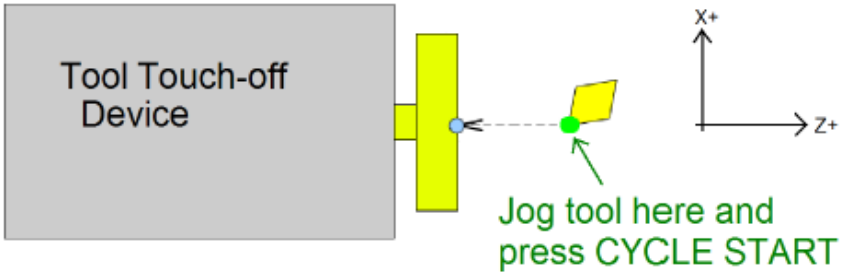


Draft: January 11, 2024

Auto Tool Touch-off Cycle to Measure Z-offset (F6)

This cycle will do a single axis measuring move along the Z-axis. The +/- Z direction is determined primarily by the tool's Nose Vector and secondarily by its Approach Direction. It is important to position the tool in the correct location before starting this cycle. To run this cycle from the Tool Geometry Offset Library, first jog the tool to the correct location and press:

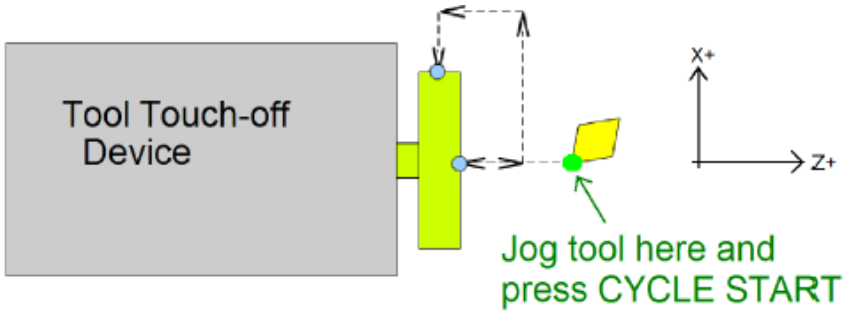
F2 – Measure Tool → **F6 – Measure Offset Z** → **CYCLE START**.



Auto Tool Touch-off Cycle to Measure both Z- and X-offsets (F7)

This cycle combines the measurement of Z- and X-offsets into one cycle. The $+/-$ axis movement directions are determined primarily by the tool's Nose Vector and secondarily by its Approach Direction, but you can change the X-measurement direction by pressing the **F1 – Orient** key on the dialogue box. To run this cycle from the Tool Geometry Offset Library, jog the tool to the correct location and press:

F2 – Measure Tool → **F7 – Measure Offset Z and X** → **CYCLE START**.



Note: See Chapter 11 – Configuration for setting the following parameters to use the Tool Touch-off device:

- Parameter 18 – PLC Input Spindle Inhibit Parameter
- Parameter 244 – Tool Touch-off device PLC input number
- Parameter 257 – TT1 connection detection PLC input
- 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
- Parameter 367 – Probe / TT1 deceleration multiplier.

Draft: January 11, 2024

5.13 Centroid CNC Lathe Tool Touch-off Using TT-4/TT-2

1. To start the Z-reference process, place the reference tool of choice into the Tool Post and the TT-4/TT-2 into the Spindle. Tighten the spindle and make sure that it is snug to prevent movement from the TT-4/TT-2. Plug in the TT-4/TT-2 cable. See Figure 5.2 for reference. For this example, we will be using Tool #1, but any tool number can be used.

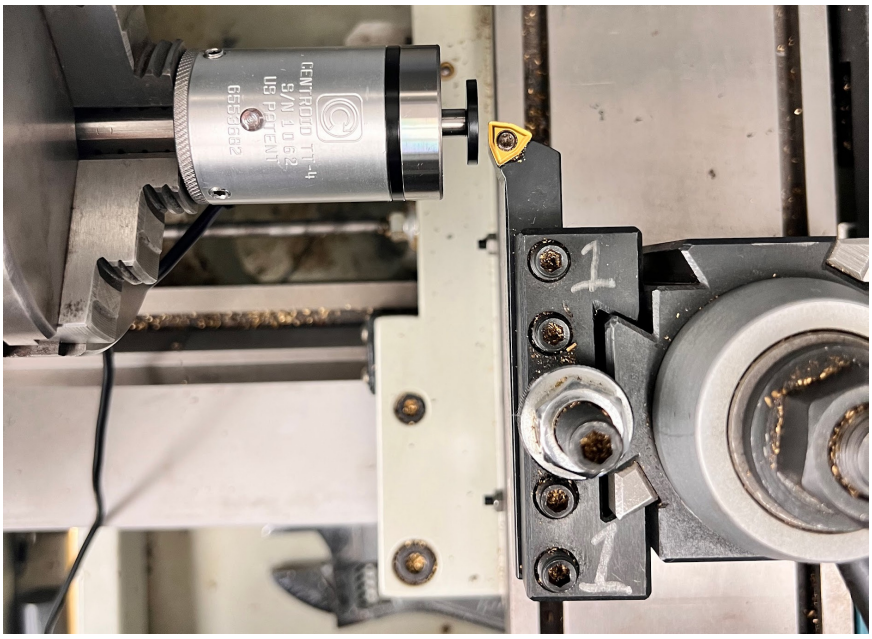


Figure 5.2: TT-4 shown in spindle with the reference tool loaded in the Tool Post. Note the location of the edge of the tool in reference to the center of the TT-4

2. From the Home screen, press **F1 – Setup** → **F2**Tool Setup. You will now have the screen shown in Figure 5.3 pulled up.

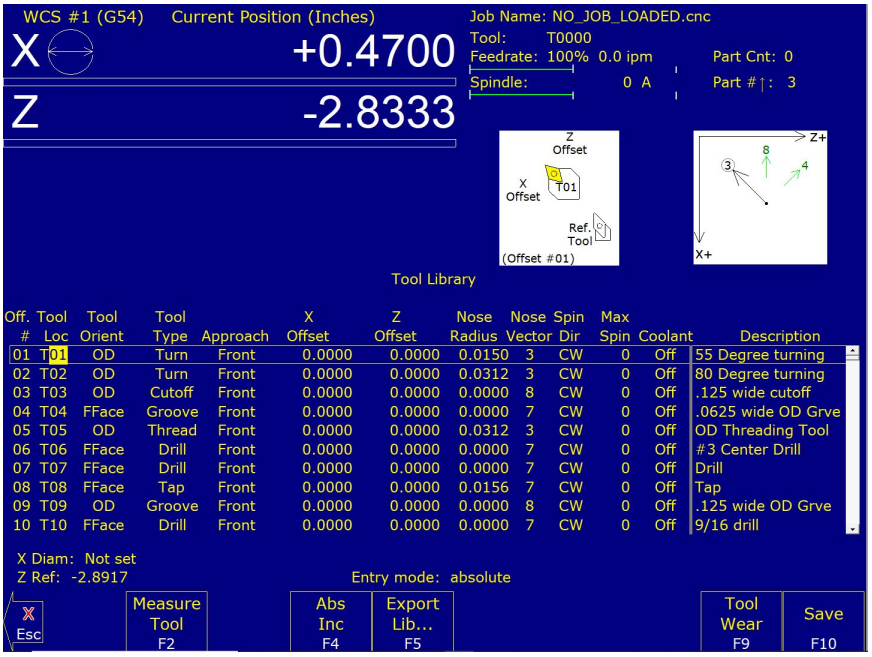


Figure 5.3: **F2 – TOOL OFFSET** menu screen

3. Using your mouse cursor or touch screen, select the Z-offset Value for the Offset # of your reference tool

Draft: January 11, 2024

number. For this example, we will be using Offset #1. When selected, the value will be highlighted in yellow and the **F1 – Z Ref** option will appear in the bottom-left corner. Your screen should look like Figure 5.4.

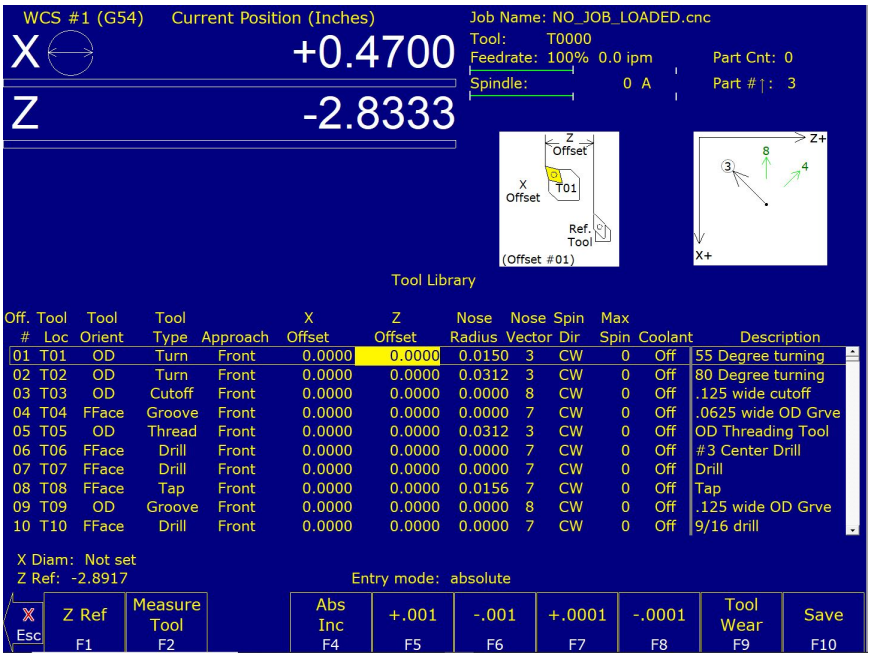


Figure 5.4: **F2 – TOOL OFFSET** menu with Z-offset value for the Offset # selected Note the addition of **F1 – Z Ref** in the bottom-left

4. To complete the Z-reference, select the **F1 – Z Ref** key. This will pull the menu options up, as shown in Figure 5.5. You will want to select the **F1 – Auto** key. Press the **Cycle Start** key. Selecting this option will allow the tool to move towards the TT-4/TT-2.

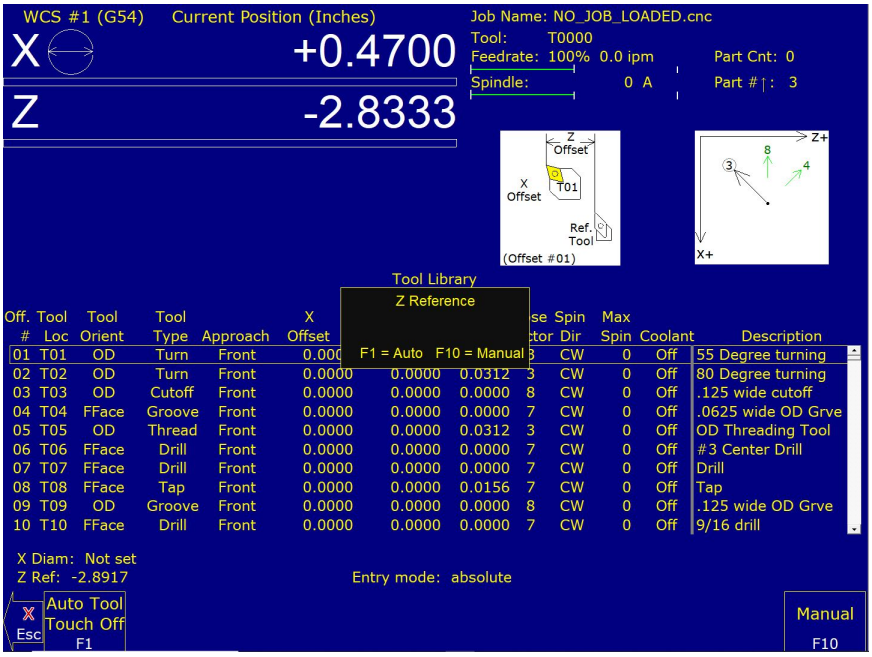


Figure 5.5: **F1 – Z Ref** menu

5. Once complete, your Lathe Z-reference has been successfully set up. Press the **F10 – Save** key to save the Z-reference information.

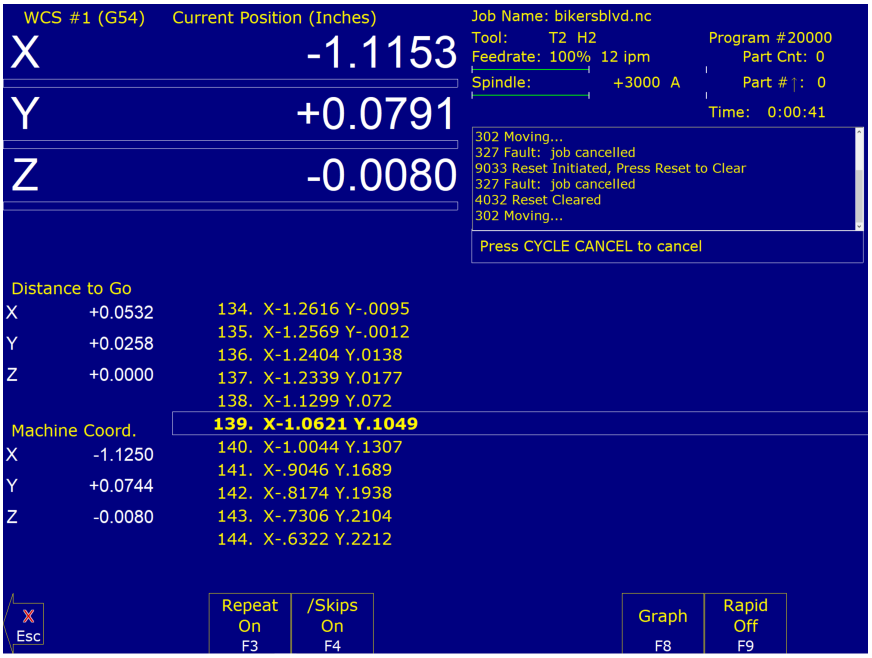
Draft: January 11, 2024

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 feed rate override by 1% (available only if keyboard jogging is active).

F2 – Feed (+1%): Increase feed rate 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.

Draft: January 11, 2024

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

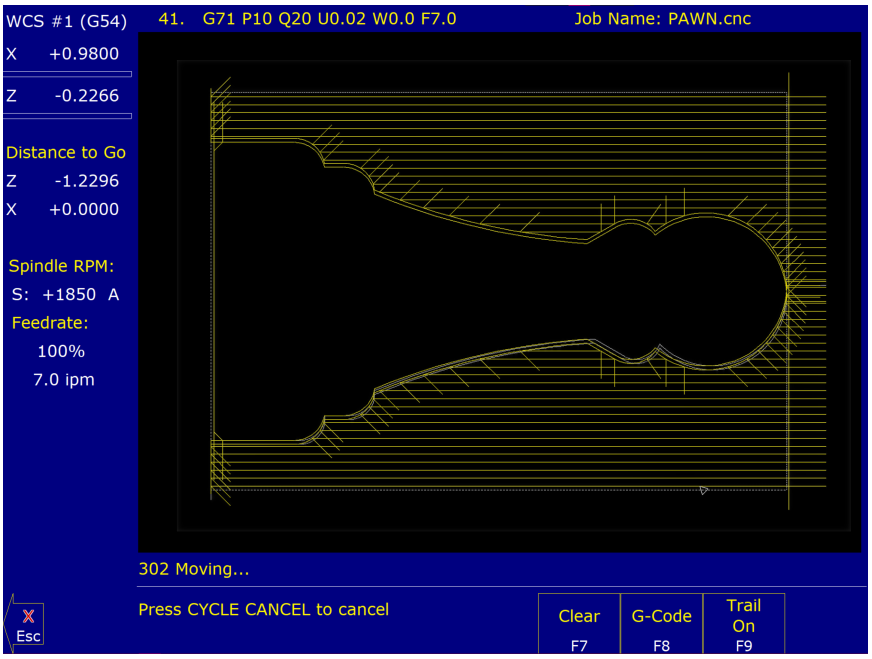
F8 – Graph: Switch to run-time graphics screen. This key only appears if the job was started with the run-time graphics option turned on.

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**.

Draft: January 11, 2024

TOOL CHECK: Pressing this key while a job is running will cause the control to stop the normal program movement, turn off the spindle, clear all M-functions, and go the Run menu screen. Make sure the tool will clear the part before pressing **Tool Check** a second time, which will move the X- and Z-axes to their home position. The control will then 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.

Draft: January 11, 2024

6.5 Run Menu

Press the **F4 – Run** key 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 the **F4 – Run** key on the main screen to go to the run screen, then pressing the **F1 – Resume Job** key 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 the **CYCLE START** button 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** arrow keys 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 ten is entered in the Part Count prompt, the Part Cnt in the status window changes to ten and the Part # changes to zero with an upward arrow indicator. When a job is run and then completes, the Part # will increment to one. 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 ten and the Part # changes to ten with a downward arrow indicator. When a job is finished, the Part # will be decremented to nine. If repeat is on, the job will automatically start again and keep running until the Part # has reached zero.

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.

Draft: January 11, 2024

F5 – Block Mode: Turns single block mode on and off. This is similar to pressing the **AUTO/BLOCK** key. If Single Block mode is on, the CNC software will stop after each block in your part program and wait for you to press the **CYCLE START** button. 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 the **CYCLE START** button (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 Feed Rate 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 the **CYCLE START** button is enabled on the Run Menu. If Parameter 400 is set to zero, then the **CYCLE START** button is disabled in the Run Menu. For any other value of Parameter 400, the **CYCLE START** button is enabled. Note that this does not apply to the Resume and Search sub-menus, where the **CYCLE START** button is always enabled.

Draft: January 11, 2024

6.6 Power Feed

Press the **F4 – Feed** key from the Setup menu to access the Power Feed screen. This screen is used to command axis movement. All of 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 the **F1 – ABS** key to move an axis to an absolute position at a specified feed rate.

F2 – Incremental Power Feed: Press the **F2 – INC** key to move an axis an incremental distance at a specified feed rate.

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

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

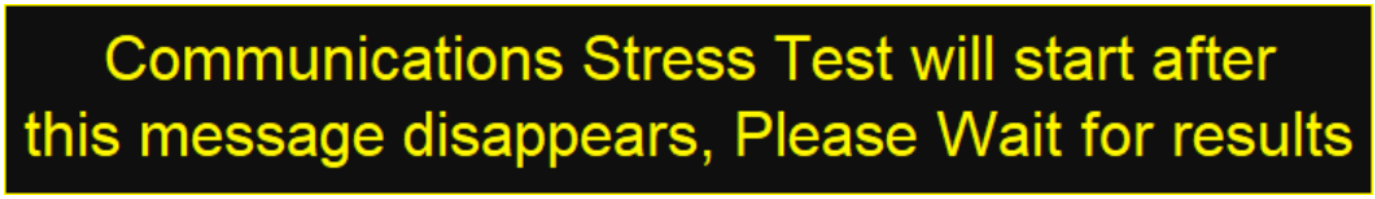
6.7 Communications Stress Test

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,

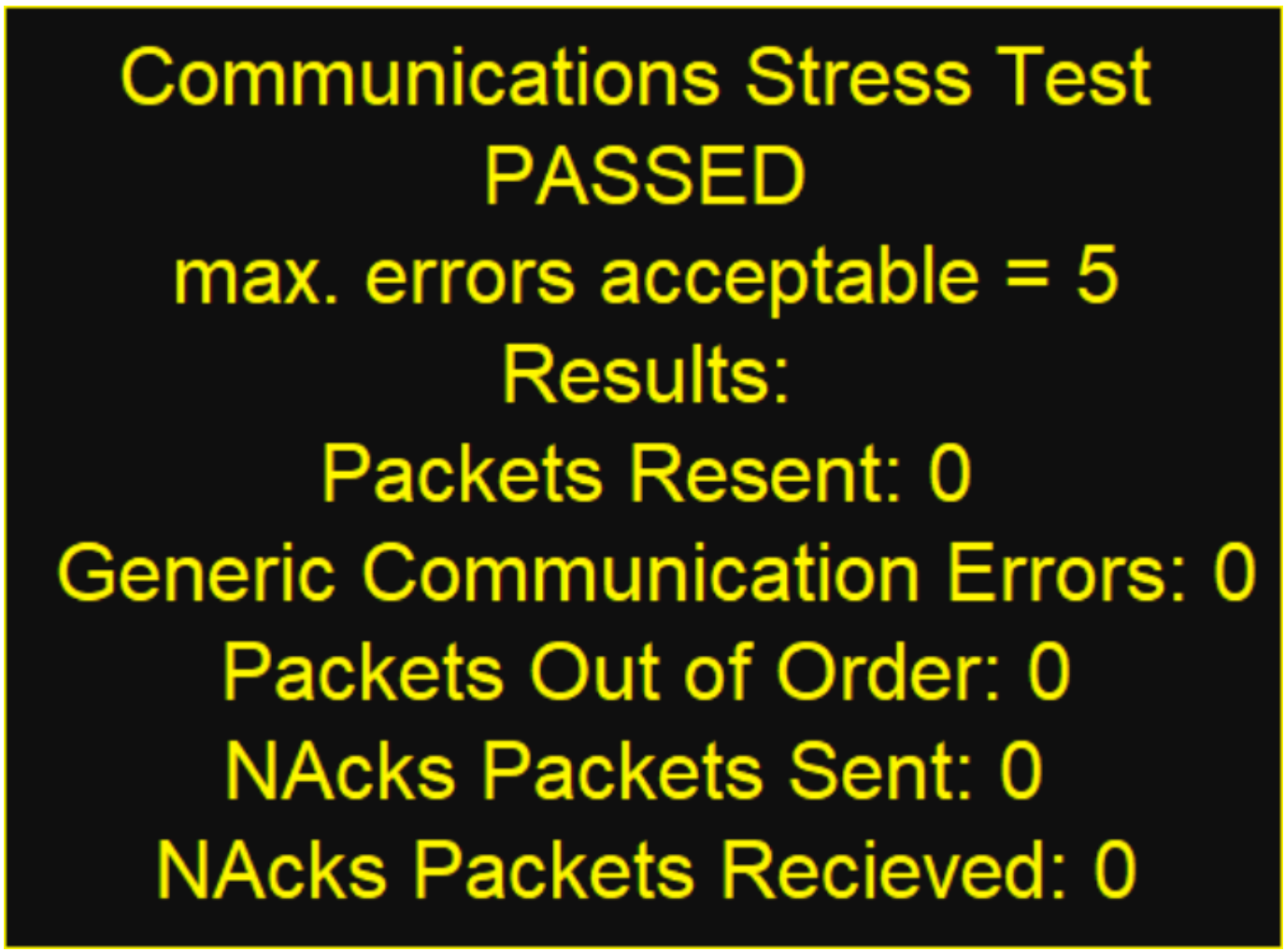
number of NAcks packets sent, and number of NAcks packets received.

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:



Draft: January 11, 2024

7 Intercon Software

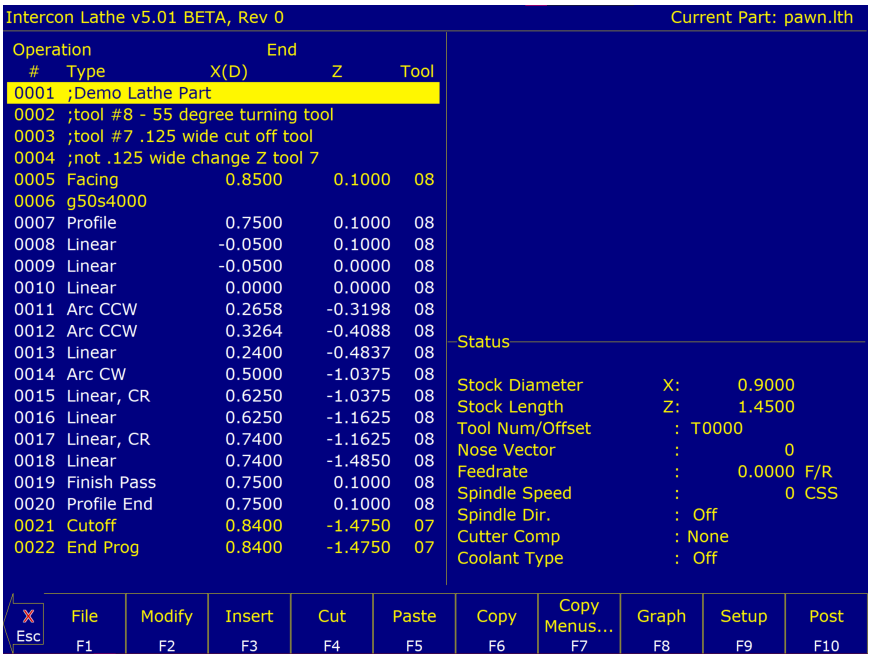
7.1 Introduction

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.

7.2 Lathe Intercon Main Menu

When you access Intercon through the **F5 – CAM** option in the CNC12 Main screen, the part program will be displayed if the current job loaded in CNC12 has an associated Intercon program. If the job file in CNC12 does not have an associated Intercon program, the **F1 – File** menu will be displayed. See the “Lathe Intercon File Menu” section later in this chapter for a description of the file menu.



While in the Lathe Intercon Main Menu, use the up and down arrow keys to highlight the desired operation.

F1 – File: Press **F1 – File** to display the File Menu. See the “Lathe Intercon File Menu” section later in this chapter for a description of the file menu.

F2 – Modify: Press **F2 – Modify** (or the **ENTER** key) to make changes to the highlighted operation. This will display the Edit Operation Menu for the highlighted operation. Use the Page Up and Page Down keys to move between

Draft: January 11, 2024

operations and to highlight the operation you want to modify in the Edit Operation Menu. See the “Insert Operation” section later in this chapter for a description of each operation type.

F3 – Insert: Insert an operation above the currently-highlighted operation. See the “Insert Operation” section later in this chapter for details.

F4 – Cut: Cuts (removes) the highlighted operation from the program. The operation that is cut is placed onto the clipboard stack. Attempting to cut a profile start or end operation will cut the entire profile.

F5 – Paste: Paste the last operation that was cut or copied into the clipboard stack into the current program line that is before the highlighted operation. A number on the second line of the Paste key indicates the number of operations that are currently in the clipboard stack. If the top of the clipboard contains a profile, then the entire profile will be pasted.

F6 – Copy: Copy the highlighted operation onto the clipboard stack and advance the cursor to the next operation.

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

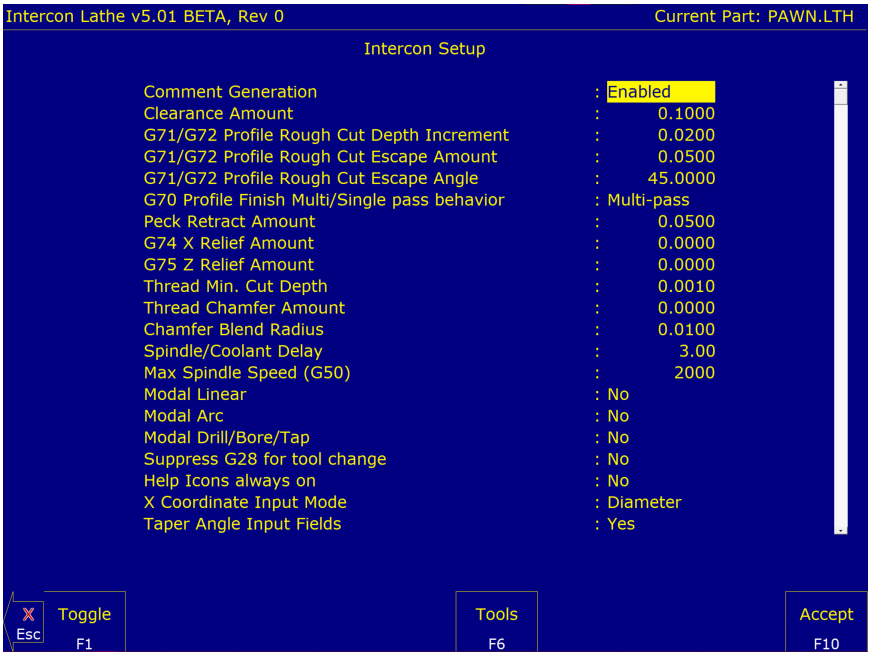
F2 – Move Menu: Allows a range of operations to be moved. Specify the Start Block, End Block, and Destination in the prompts that appear in the Move Menu. The range of operations is moved into a location that precedes the destination block.

F3 – Cut, F4 – Paste, F5 – Copy: Performs the same actions as described above.

F9 – Clear Clipbrd: Removes all operations in the clipboard stack.

F8 – Graph: Display a graphic preview for the part. See the “Graphics” section later in this chapter for details.

F9 – Setup: Change the part setup. The following window will be displayed on the screen. Use the up and down arrow keys to select between fields. Note that there are more fields than what the menu can show. If you go beyond the bottom of the screen, the fields will scroll up, revealing more Intercon Setup items. Press **F1 – Toggle** to toggle between options when necessary and press **F10 – Accept** to accept the setup when you are finished. You can also access the Tool Library from here by pressing **F6 – Tools**. Press the **ESC** key to cancel and return to the File menu.



Draft: January 11, 2024

Comment Generation: Toggle between Enabled and Disabled. When comment generation is enabled, Intercon will insert a comment before each block describing the operation type. Disabling comment generation reduces the size of the file.

Clearance Amount: Set the distance away from the part you want to position when changing from a rapid to a feed rate move. This amount applies to both the X- and Z-axes. Adjust this value to adjust the retract amount in a threading cycle (G76).

G71/G72 Rough Cut Depth Increment: Enter the amount of material to remove per pass in a profile cycle. Value is always a radius amount.

G71/G72 Rough Cut Escape Amount: Enter the distance to retract after a cutting pass has been made in a profile cycle. The values are always a radius amount.

G71/G72 Rough Cut Escape Angle: Enter the angle of retract (in degrees) after a cutting pass has been made in a profile cycle.

G70 Profile Finish Multi/Single-pass Behavior: Toggle between Multi-pass and Single-pass behavior for the profile finish pass. In Multi-pass mode, a Finish Pass operation will perform multiple incremental finish cuts each at the specified “Depth of Cut” away from each other. In Single-pass mode, a Finish Pass operation will perform one finish cut at the specified “Depth of Cut” away from the final finish profile.

Peck Retract Amount: Enter the distance to retract after a cutting move has been made in the peck drilling cycle, peck cut-off cycle, and grooving cycle.

G74 X Relief Amount: Enter the relief amount for the X-axis in a Grooving cycle. This is the amount that the tool moves away from the material in the X-axis direction before making rapid moves to position for the next cut.

G75 Z Relief Amount: Enter the step-over amount for the Z-axis in a Grooving cycle. This is the amount that the tool moves away from the material in the Z-axis direction before making rapid moves to position for the next cut.

Thread Min. Cut Depth: Enter the minimum amount you want removed for a pass in the threading cycle.

Thread Chamfer Amount: Enter the number of turns to taper from the thread depth to the surface of the work piece.

Chamfer Blend Radius: Enter the radius to use when rounding the corners of a chamfer when blend chamfer is selected.

Spindle/Coolant Delay: Enter the amount of time in seconds that you want the lathe to wait for the spindle to get up to speed and for the coolant to begin flowing.

Max Spindle Speed (G50): Enter the maximum spindle speed for posted Intercon programs. Posts a G50 at the beginning of the program if the value entered is greater than zero.

Modal Operations (Linear, Arc, Drill/Tap): Toggle between YES or NO. Entering YES will cause the same type of operation to be automatically inserted after the initial operation has been accepted.

Suppress G28 for Tool Change: Toggle between YES or NO. Entering “NO” will allow Intercon to post a G28 on a tool change operation to return the tool to the G28 position. Gang tooling setups usually require this option to be set to “YES”.

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”, help screens can always be toggled ON or OFF by pressing the **F5 – Help** key when editing an operation.

X-coordinate Input Mode: Toggle between radius and diameter. You can select to enter the coordinates as radius or diameter amounts.

Draft: January 11, 2024

Taper Angle Input Fields: Toggle between hide and display. When you select hide, the fields that correspond to polar coordinates will not be shown. When you select display, the fields that correspond to polar coordinates will be shown.

Modal Input Fields: Toggle between hide and display. When you select hide, modal fields will not be shown. When you select display, modal fields will be shown.

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.

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.

Stop Spindle During Tool Change: Toggle between YES and NO. Select "YES" if you want the spindle to be shut off during a tool change. Select "NO" if you want the spindle to be left on while doing a tool change.

Stop Coolant During Tool Change: Toggle between YES and NO. Selecting "YES" will cause the coolant to be shut off during a tool change. Selecting "NO" will cause the coolant to be left on while doing a tool change.

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

F10 – Post: Press **F10 – Post** to post a part program. Posting a part program generates the G-codes for the program. After the program is posted, you will be returned to the control software's Main Screen where the G-code program will be loaded and you can press **CYCLE START** to run the job. The Intercon program will be automatically saved.

Esc – Quit: Press **Esc** to quit Intercon. You will be prompted to save changes if any were made, and then returned to the control software's Main Screen.

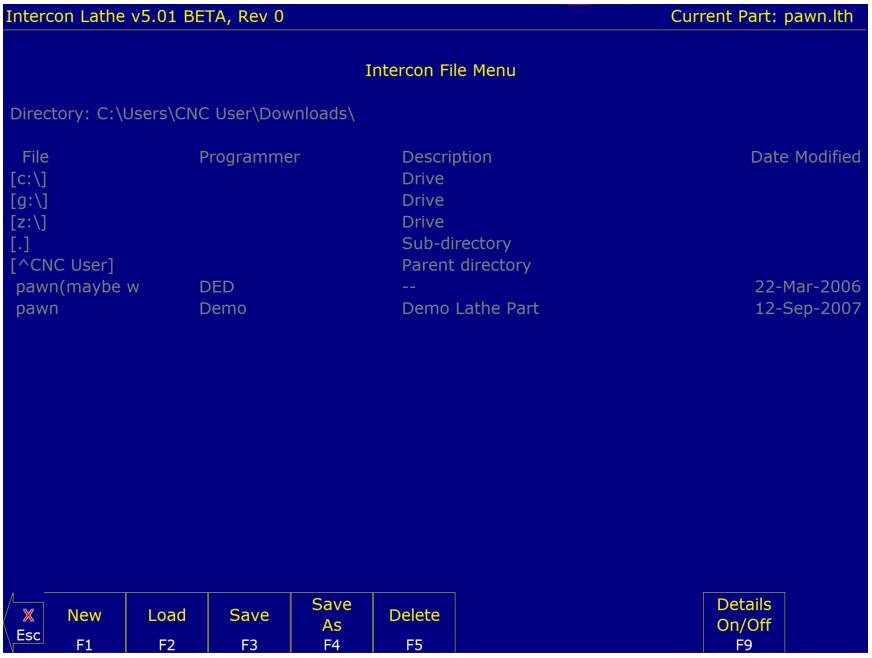
Draft: January 11, 2024

7.3 Teach Mode

The X and Z keys will fill in a field with the current position for the related axis. This feature works when editing most fields in an operation. Press **F9 – Teach Mode** when editing an operation to display a DRO.

7.4 Lathe Intercon File Menu

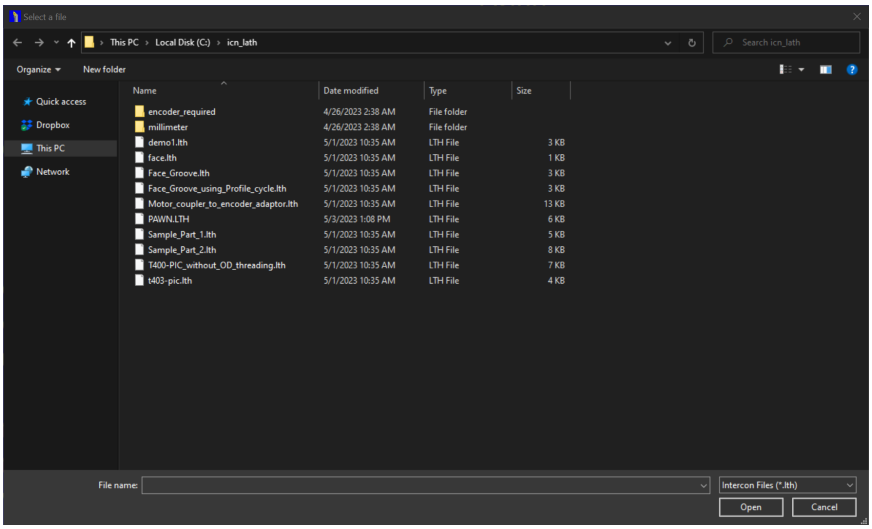
Press **F1 – File** while in the Intercon Main Menu to access the File Menu. The screen will look something like the example below:



F1 – New: Create a new file; you will be prompted to save changes to the currently-loaded part program. Press "**Y**" to save changes or "**N**" to continue without saving changes. Choosing **F1 – New** will display the "New file:" prompt above the function keys. Type the name of the new file, then press **F10 – Accept** or the **ENTER** key to accept the new name. After accepting the new name, the program header information can be entered.

F2 – Load: Press **F2 – Load** to load an existing program. You will be prompted to save changes to the currently-loaded part program. Press "**Y**" to save changes or "**N**" to continue without saving changes.

7.4.1 Load Menu



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, which is displayed at the top of the screen.

It is also possible to load a file by 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

Draft: January 11, 2024

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 viewing and loading options are available through the F-Key menus, which are detailed below:

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

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

F3 – 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).

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 accidentally 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.

Draft: January 11, 2024

7.5 Lathe Intercon File Menu Continued...

F3 – Save: Saves the current part program under its current name.

F4 – Save As: Saves the current part program under a different name or to a different drive/directory. This allows you to make changes to a program and save the file under a different name, so that the original program remains unchanged. The name can be up to eight characters long, but it cannot contain the symbols `+=\[]'."/:<>?` in the filename. If the new name already exists, a prompt will be displayed as a warning, and will give the options to overwrite the existing file or return to enter a different name.

F5 – Delete: Deletes a file. After **F5 – Delete** is pressed, the screen will appear as in the **F2 – Load** option where the same keys can be used to navigate the files. A yes/no prompt will appear after accepting a file for deletion for final confirmation.

F9 – Details On/Off: Turns Intercon part file information display on or off.

7.6 Insert Operation

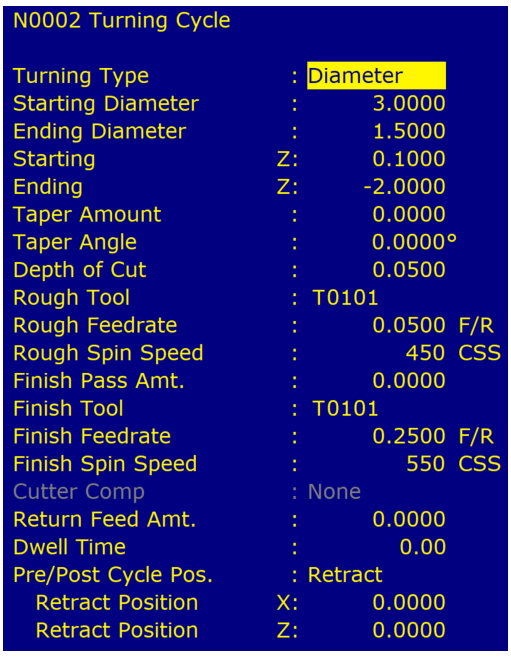
Press **F3 – Insert** or the Insert key to access the Insert Operation Menu. From this menu, you can add operations to a part program.

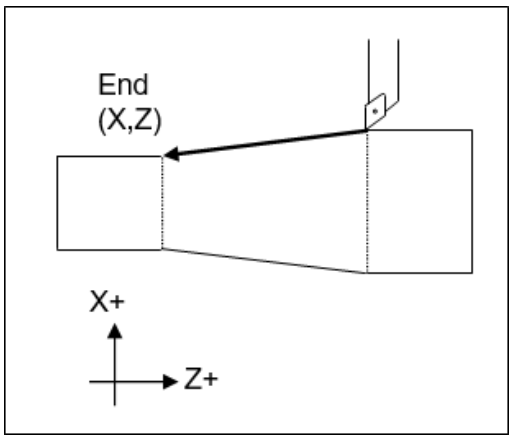
The operation is added before the currently-highlighted operation. The block number is shown to the left. The operations you can insert are listed at the bottom of the screen. Pressing the function key that corresponds to an operation will bring up the Edit Operation Menu for that operation.

Note: For operations that use negative side tooling (see Chapter 4), X-values will be negative, such as the starting and ending diameters in a turning cycle. Roughing and finishing tools are the same, and the user is required to do tool positioning for tool changes.

7.6.1 F1 – Line

Press **F1 – Line** at the Insert Operation Menu to insert a linear operation.





Draft: January 11, 2024

Press **F1 – Toggle** or the **Space** key to toggle between “Rapid” and “Feed Rate” options when necessary, and then use the Up and Down arrow keys to move between fields and fill in the rest of the required information. Once complete, press **F8 – Graph** to check your work and **F10 – Accept** to accept the entries. Use the up and down arrow keys to move between fields. Press **ESC** to cancel and return to the Insert menu.

The destination of the linear move can be given in terms of the end point coordinates or as the counter-clockwise angle from the three o’clock position to the line and length of the line (polar coordinates). Press **F3 – Modal** Display to hide modal fields. Press this key again to show those fields. Press the **F4** key to hide the polar coordinates. Press this key again to display those fields.

Linear Type: Enter the type of linear move you want to make (Rapid or Feed Rate). This field can be toggled between Rapid and Feed Rate. A rapid move is a non-cutting positioning move made at the maximum rate. A feed rate move is a cutting move made at the programmed feed rate. When performing a cutting operation, this must be toggled to Feed Rate.

End X: Enter the X-coordinate of the end position of the linear move. You can toggle between absolute and incremental position. When toggled to absolute, enter the absolute position, with reference to the part zero. When toggled to incremental, an INC will appear next to the entry. In this mode, enter the X-distance from the preceding end position.

End Z: Enter the Z-coordinate of the end position of the linear move. You can toggle between absolute and incremental position. When toggled to absolute, enter the absolute position, with reference to the part zero. When toggled to incremental, an INC will appear next to the entry. In this mode, enter the Z-distance from the last preceding end position.

Angle: The destination can also be determined with an angle from the three o’clock position. Enter this angle in conjunction with the length to determine the end point of the linear move.

Length: Enter the length of the linear move. The length, along with the previously-entered angle, will be used to calculate the end point of the move.

Connect Type: When two feed rate moves are performed consecutively, you can choose the style in which they are connected. You can toggle this field between the following options: None, Bl Chamf (Dist), Chamf (Dist), Bl Chamf (Len), Chamf (Len), or Radius. When set to none, the linear operations are connected at the point of intersection. There are two chamfer types: Distance and Length. For Distance Chamfers, the Operator specifies the amount of distance to be removed from the ends of the two linear segments. The chamfer connects the two shortened segments. If a Length Chamfer is chosen, the linear moves are connected by a chamfer of a specified length. Both chamfer types have a blended version. When blend chamfer is chosen, the linear moves are connected by a chamfer with rounded corners. When radius is chosen, a rounded corner connects the two linear moves.

Notes:

1. Chamfers and blend chamfers in programs created with pre 8.10 Intercon are Length chamfers in the new software.
2. Chamfer and blend chamfer cannot be used to connect to an arc.

Connect Radius: Enter the radius of the rounded corner used to connect two feed rate moves.

Chamfer Distance: Enter the Distance to be removed from the end of each linear segment.

Chamfer Length: Enter the length of the chamfer you want to connect two linear feed rate moves.

Draft: January 11, 2024

Tool Num/Offset: Enter the tool number and offset number used. The first two digits are the tool number; the last two digits are the offset number. You can also press **F2** to go to the tool library to select another tool and/or make changes to the tool library, then press **F10** to accept.

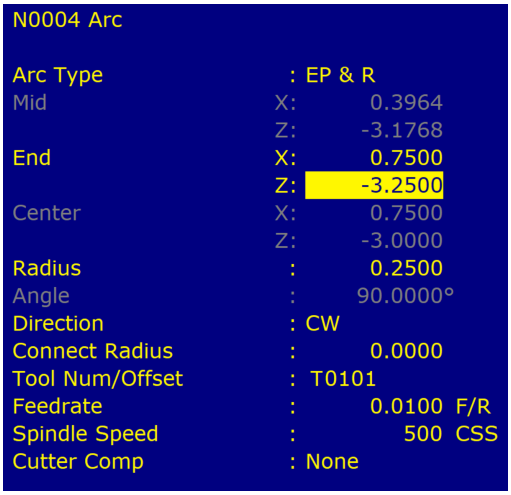
Feedrate: Enter the desired cutting feed rate. You can toggle between feed/min and feed/rev.

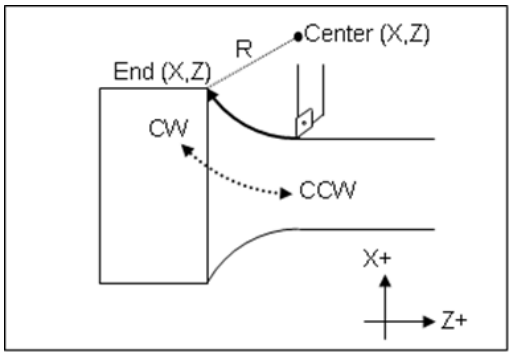
Spindle Speed: Enter the desired spindle speed. You can toggle between RPM or CSS. When toggled to RPM, a constant RPM will be maintained. When toggled to CSS, a constant surface speed will be maintained.

Cutter Compensation: Set cutter compensation in Chapter 9 for more details. You can toggle between None, Right, and Left.

7.6.2 F2 – Arc

Press **F2 – Arc** to insert an arc operation.





Note: Clockwise will need to be set for rear-facing lathes and Counter-clockwise for front-facing lathes.

Use the up and down arrow keys to move between fields. Press the **F1 – Toggle** key or the **Space** bar to toggle between options when necessary and press **F10 – Accept** to accept the information entered. Press the **ESC** key to cancel and return to the Insert Menu. Press the **F3 – Modal Display** key to hide modal fields. Press the **F3 – Modal Display** key again to show those fields.

Type: Intercon allows you to specify the arc in one of four ways. You can specify the arc by its end point and radius (EP&R), center point and angle (CP&A), center point and end point (CP&EP), or mid point and end point (3-point). The fields displayed will depend on the type specified.

EP&R – End Point and Radius

End X: Enter the X-coordinate of the end of the arc. You can toggle between absolute and incremental position. When toggled to absolute, enter the absolute position, with reference to the part zero. When toggled to incremental, an INC will appear next to the entry. In this mode, enter the X-distance from the preceding end position.

End Z: Enter the Z-coordinate of the end of the arc. You can toggle between absolute and incremental position. When toggled to absolute, enter the absolute position with reference to the part zero. When toggled to incremental, an INC will appear next to the entry. In this mode, enter the Z-distance from the preceding end position.

Draft: January 11, 2024

Radius: Enter the radius of the arc. Blend chamfer and chamfer cannot be used to connect to an arc or to connect an arc to another item.

Direction: Enter the direction you want the arc to be cut. Toggle between clockwise and counter-clockwise.

Connect Radius: Enter the radius to use when blending an arc with another arc or a linear cut. Entering a value in this field will cause the moves to be connected by a rounded corner with this radius value.

Tool Num/Offset: Enter the tool number and offset number you want to use. The first two digits are the tool number; the last two digits are the offset number.

Feedrate: Enter the desired cutting feed rate. You can toggle between feed/min and feed/rev.

Spindle Speed: Enter the desired spindle speed. You can toggle between RPM or CSS. When toggled to RPM, a constant RPM will be maintained. When toggled to CSS, a constant surface speed will be maintained.

Cutter Compensation: Set cutter compensation in Chapter 9 for more details. You can toggle between None, Right, and Left.

CP&A – Center Point and Angle

Center X: Enter the X-coordinate of the center of the arc. You can toggle between absolute and incremental position. When toggled to absolute, enter the absolute position, with reference to the part zero. When toggled to incremental, an INC will appear next to the entry. In this mode, enter the X-distance from the last point.

Center Z: Enter the Z-coordinate of the center of the arc. You can toggle between absolute and incremental position. When toggled to absolute, enter the absolute position, with reference to the part zero. When toggled to incremental, an INC will appear next to the entry. In this mode, enter the Z-distance from the last point.

Angle: Enter the angle of the arc. Direction: Enter the direction you want the arc to be cut. Toggle between clockwise and counter-clockwise.

Connect Radius: Enter the radius to use when blending an arc with a linear cut or another type of arc. Entering a value in this field will cause the arc and a linear move to be connected by a rounded corner with this radius.

Tool Num/Offset: Enter the tool number and offset number you want to use. The first two digits are the tool number; the last two digits are the offset number.

Feedrate: Enter the desired cutting feed rate. You can toggle between feed/min and feed/rev.

Spindle Speed: Enter the desired spindle speed. You can toggle between RPM or CSS. When toggled to RPM, a constant RPM will be maintained. When toggled to CSS, a constant surface speed will be maintained.

Cutter Compensation: See cutter compensation in Chapter 9 for more details. You can toggle between None, Right, and Left.

CP&EP – Center Point and End Point

End X: Enter the X-coordinate of the end of the arc. You can toggle between absolute and incremental position. When toggled to absolute, enter the absolute position, with reference to the part zero. When toggled to incremental, an INC will appear next to the entry. In this mode, enter the X-distance from the last point.

End Z: Enter the Z-coordinate of the end of the arc. You can toggle between absolute and incremental position. When toggled to absolute, enter the absolute position, with reference to the part zero. When toggled to incremental, an INC will appear next to the entry. In this mode, enter the Z-distance from the last point.

Draft: January 11, 2024

Center X: Enter the X-coordinate of the center of the arc. You can toggle between absolute and incremental position. When toggled to absolute, enter the absolute position, with reference to the part zero. When toggled to incremental, an INC will appear next to the entry. In this mode, enter the X-distance from the last point.

Center Z: Enter the Z-coordinate of the center of the arc. You can toggle between absolute and incremental position. When toggled to absolute, enter the absolute position, with reference to the part zero. When toggled to incremental, an INC will appear next to the entry. In this mode, enter the Z-distance from the last point.

Direction: Enter the direction you want the arc to be cut. Toggle between clockwise and counter-clockwise.

Connect Radius: Enter the radius to use when blending an arc with a linear cut or another type of arc. Entering a value in this field will cause the arc and a linear move to be connected by a rounded corner with this radius.

Tool Num/Offset: Enter the tool number and offset number you want to use. The first two digits are the tool number; the last two digits are the offset number.

Feedrate: Enter the desired cutting feed rate. You can toggle between feed/min and feed/rev.

Spindle Speed: Enter the desired spindle speed. You can toggle between RPM or CSS. When toggled to RPM, a constant RPM will be maintained. When toggled to CSS, a constant surface speed will be maintained.

Cutter Compensation: See cutter compensation in Chapter 9 for more details. You can toggle between None, Right, and Left.

Draft: January 11, 2024

3-Point (Start Point, Mid Point, and End Point

Mid X: Enter the X-coordinate of a point on the arc between the start point and the end point. You can toggle between absolute and incremental position. When toggled to absolute, enter the absolute position, with reference to the part zero. When toggled to incremental, an INC will appear next to the entry. In this mode, enter the X-distance from the last point.

Mid Z: Enter the Z-coordinate of a point on the arc between the start point and the end point. You can toggle between absolute and incremental position. When toggled to absolute, enter the absolute position, with reference to the part zero. When toggled to incremental, an INC will appear next to the entry. In this mode, enter the Z-distance from the last point.

End X: Enter the X-coordinate of the end of the arc. You can toggle between absolute and incremental position. When toggled to absolute, enter the absolute position, with reference to the part zero. When toggled to incremental, an INC will appear next to the entry. In this mode, enter the X-distance from the last point.

End Z: Enter the Z-coordinate of the end of the arc. You can toggle between absolute and incremental position. When toggled to absolute, enter the absolute position, with reference to the part zero. When toggled to incremental, an INC will appear next to the entry. In this mode, enter the Z-distance from the last point.

Direction: Enter the direction you want the arc to be cut. Toggle between clockwise and counter-clockwise.

Connect Radius: Enter the radius to use when blending an arc with a linear cut or another type of arc. Entering a value in this field will cause the arc and a linear move to be connected by a rounded corner with this radius.

Tool Num/Offset: Enter the tool number and offset number you want to use. The first two digits are the tool number; the last two digits are the offset number.

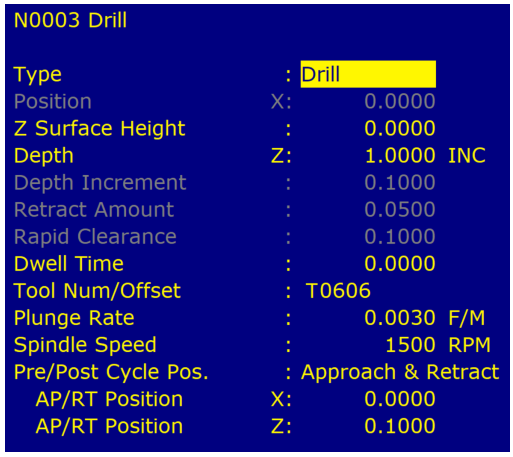
Feedrate: Enter the desired cutting feed rate. You can toggle between feed/min and feed/rev.

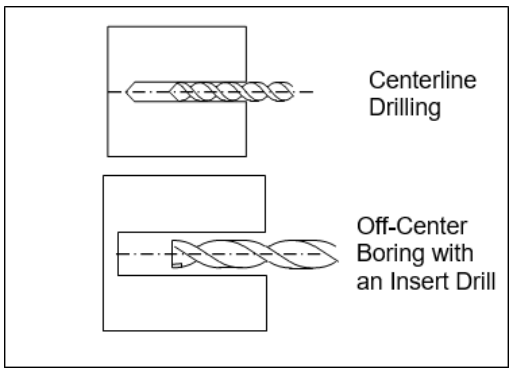
Spindle Speed: Enter the desired spindle speed. You can toggle between RPM or CSS. When toggled to RPM, a constant RPM will be maintained. When toggled to CSS, a constant surface speed will be maintained.

Cutter Compensation: See cutter compensation in Chapter 9 for more details. You can toggle between None, Right, and Left.

7.7 F3 – Drill

Press the **F3 – Drill** key to insert a Drill operation. This operation allows you to either do normal Drilling or off-center Boring operations. Both the Drilling and Boring type operations are actually the same, except in the types of tools used and position X field.

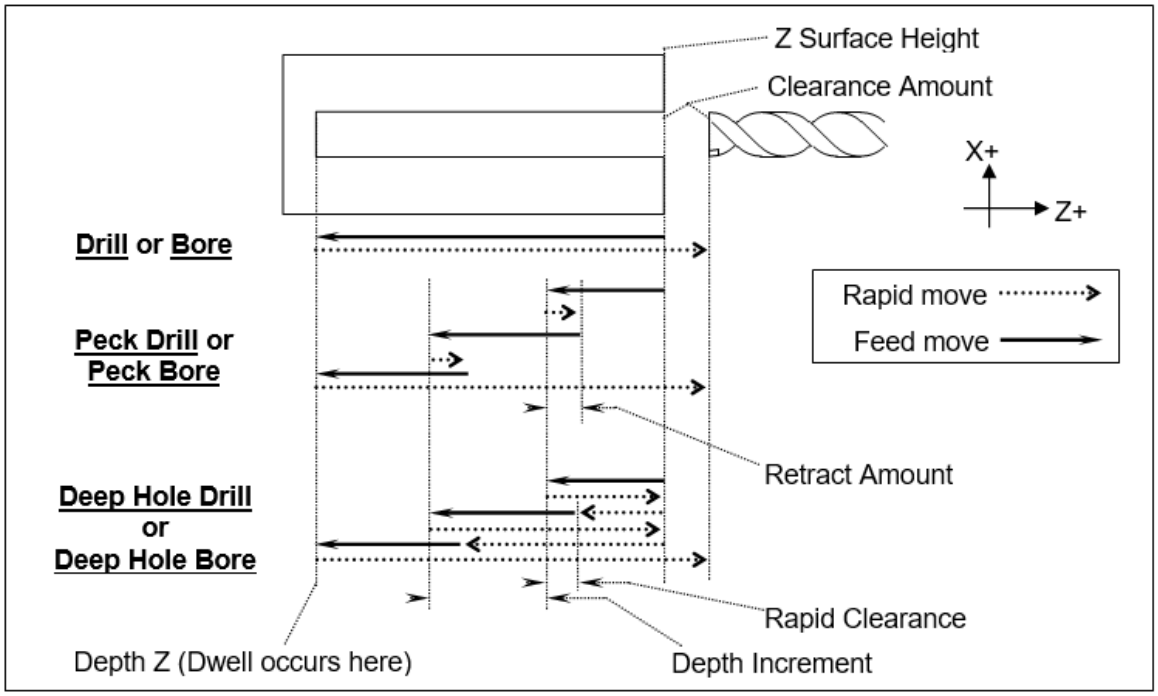




Press the **F1 – Type** key to toggle between Bore and Drill and their various options (i.e. peck, & deep hole). If the operation is toggled into Bore mode, then you can modify the Position X-coordinate, which can be specified to be off-center (usually by the tool diameter).

Note: Insert drills and end mills can be used to drill and bore holes into a part. In order to bore with a specific tool, it will need an offset value for that tool so diameters can be controlled. For example, if a .750 diameter insert drill is used to drill a hole in a part, but the final diameter of the hole needs to be 1.250, toggle to the boring cycle and enter .750 for Position X. This will offset the center of the drill to the center of the part. After the hole is in the part, use a profile or turning cycle to finish the hole to 1.250 diameter using the same tool.

Press the **F1 – Type** key to toggle between options when necessary and the **F10 – Accept** key to accept the entries. Use the up and down arrow keys to move between fields. Press **ESC** to cancel and return to the Intercon Main Menu.



Draft: January 11, 2024

Surface Z: Enter the position of the front face of the work piece.

Type: Enter the type of Drilling or Boring you want to perform. You can toggle between Drill, Peck Drill, Deep Hole Drill, Bore, Peck Bore, and Deep Bore using the **F1 – Type** key.

Position X: (Valid only while in Bore mode) Enter the diameter for the tool being used.

Depth Z: Enter the depth of the hole to drill. This is the Z-distance from the surface height.

Depth Increment: Enter the cut depth increment used during the cycle. This field only applies when the type field has been set to Peck Drill, Deep Hole Drill, Peck Bore, or Deep Hole Bore.

Retract Amount: Enter the amount the drill should retract before making another incremental depth cut. This field only applies when the type field has been set to Peck Drill or Peck Bore.

Rapid Clearance: Enter the amount above the uncut material that the drill will rapid to on subsequent cuts. This field only applies when the type field has been set to Deep Hole Drill or Deep Hole Bore

Dwell Time: Enter the amount of time in seconds that the drill should dwell at the bottom of the hole.

Tool Num/Offset: Enter the tool number and offset number you want to use. The first two digits are the tool number; the last two digits are the offset number.

Plunge Rate: Enter the feed rate at which you want to drill the hole. Toggle between feed/min and feed/rev.

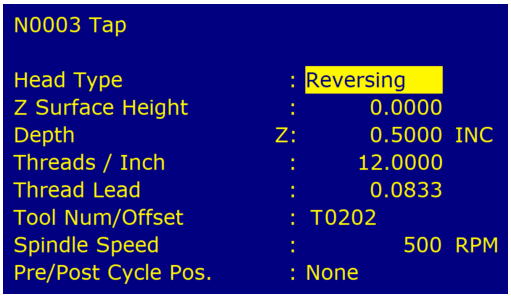
Spindle Speed: Enter the spindle speed in RPM.

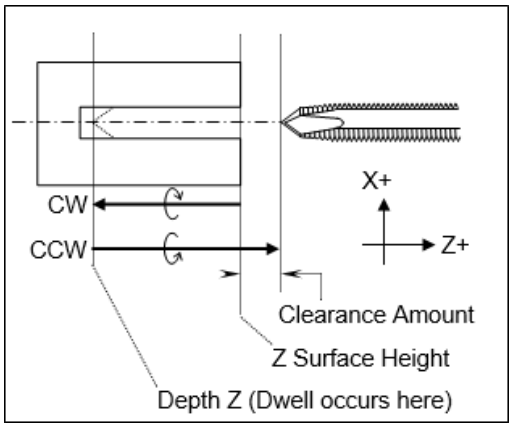
Pre/Post Cycle Pos.: Allows you to select if you want the spindle to move to a specified position before or after the cycle. Once toggled from “None”, two fields appear to enter the desired position.

Draft: January 11, 2024

7.8 F4 – Tap

The tap operation allows you to tap into the parts centerline (cutting in the negative Z-direction). The operation may use a floating tap holder or rigid tap, with spindle reversal or a self-reversing tap head. Press the **F4 – Tap** key to insert a center tapping operation.





Press the **F1 – Type** key to toggle between options when necessary and the **F10 – Accept** key to accept the entries. Use the up and down arrow keys to move between fields. Press the **Esc** key to cancel and return to the Insert Menu.

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 (which requires a spindle encoder), you can select either rigid or reversing.

Z Surface Height: Enter the Z-position of the surface you are tapping.

Depth Z: Enter the depth of the hole you want to tap. You can toggle between absolute Z and an incremental value from the parts surface. This is the Z-distance from the surface height.

Thread Pitch: Enter the desired threads/unit.

Thread Lead: Enter the desired units/thread.

Dwell Time: Enter the time in seconds that the tap should dwell at the bottom of the hole. This is to allow time for the spindle to reverse rotational direction and is used for Floating Tap only.

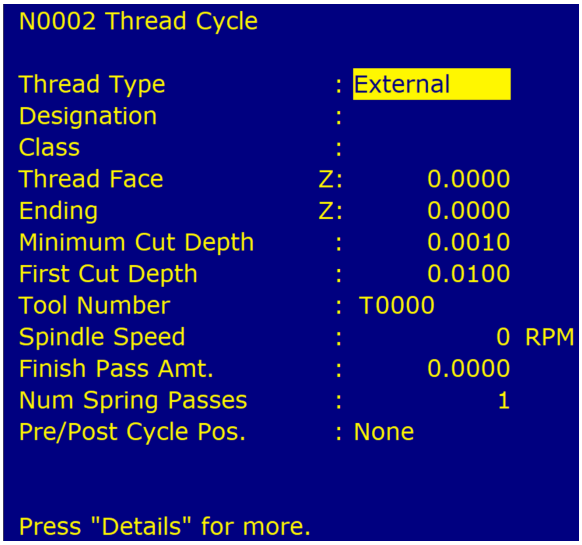
Tool Num/Offset: Enter the tool and offset numbers you want to use. The first two digits are the number and the last two digits are the offset number.

Spindle Speed: Enter the spindle speed in RPM. A constant RPM value will be maintained.

Pre/Post Cycle Pos.: Allows you to select if you want the spindle to move to a specified position before or after the cycle. Once toggled from “None”, two fields appear to enter the desired position.

7.9 F5 – Thread

Press the **F5 – Thread** key to insert a threading cycle. This cycle allows you to create a thread on the outside or inside of your part. When you first insert a threading cycle, the screen should look like the picture below:



Press the **F7 – Details** key to skip thread lookup and manually enter custom thread data. Press the **F1 – Type** key to toggle between options when necessary and the **F10 – Accept** key to accept the entries. Use the up and down arrow keys to move between fields. Press the **ESC** key to cancel and return to the Insert Menu.

7.9.1 Thread Lookup

This cycle has a lookup feature that simplifies the process of creating threads. The data for standard threads has been included in a database, which you can add custom threads to. You can add custom threads to this database. You can recall any previously-stored thread by specifying a few key criteria:

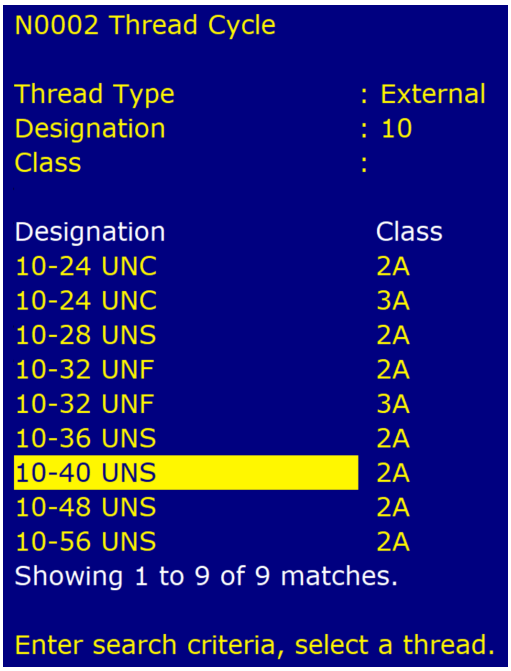
Thread Type: Enter the thread type desired. Toggle between external, internal, external pipe, and internal pipe. You can view database entries for internal/external thread types or pipe threads, but not both at the same time.

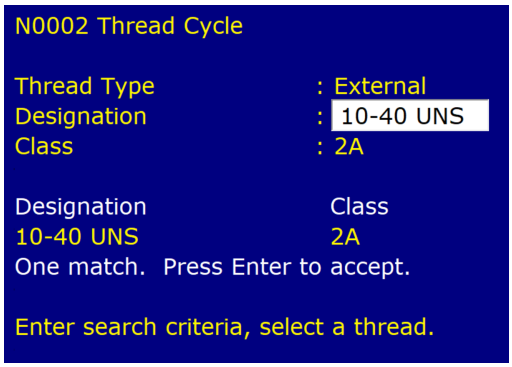
Draft: January 11, 2024

Designation: Begin typing the beginning of a standard or custom designation to view a list of matching database entries. Leave blank to match all entries from the database.

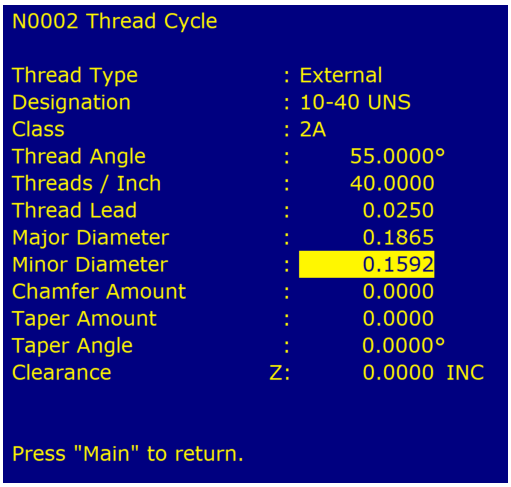
Class: Begin typing the beginning of a class to view matching entries. Leave blank to match all classes.

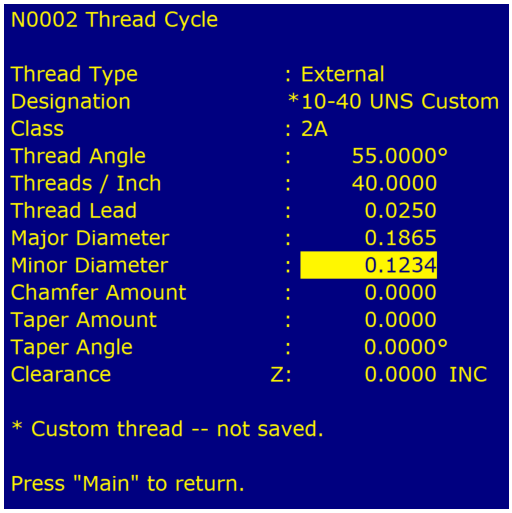
When you have typed anything into Designation or Class, the screen will display the first matching entries. For example, typing “10” in the Designation field would show all entries in the database whose designations start with “10”. If there is only one thread listed, simply press the **Enter** key to select it. If more than one is listed, you can choose any thread shown by using the arrow keys to move up and down in the list. If the cursor is somewhere in the thread list, you can press the **Page Up** or **Page Down** keys to change to a different page of the thread list. When the desired thread is highlighted, press the **Enter** key to accept. Below on the left is an example of selecting from the list. Below on the right is an example of accepting the single match.





When you press the **Enter** key, you can view the thread details. The fields will be filled in with the values from the selected thread.

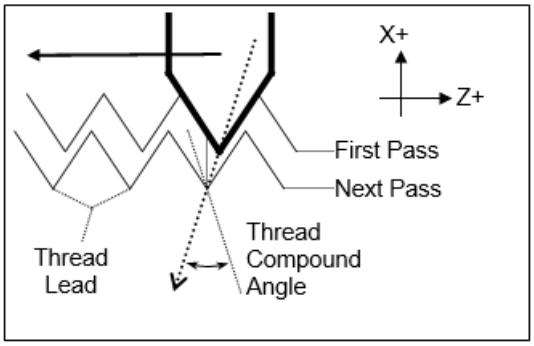




If desired, you can modify any of the values. If you do, an asterisk (*) will appear next to the Designation field and the entry will be appended with “Custom”. You may change the designation and class fields to any name that you wish. Press the **F4 – Save** key to save the new thread in the database. If the designation and class already exist, you will be

Draft: January 11, 2024

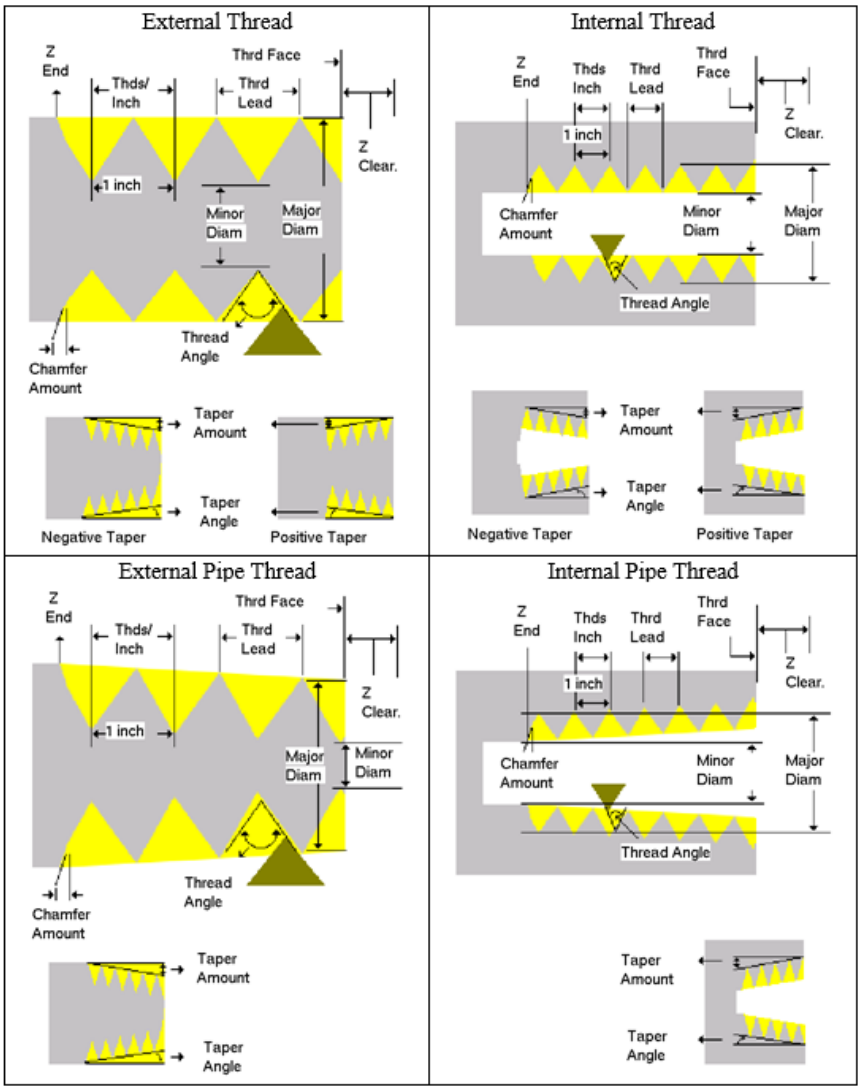
prompted to overwrite these values.



Thread (Compound) Angle: Enter the desired thread compound angle to shift the chip load to be heavier towards one side of the thread cutter. A thread compound angle of 0 means that the chip load will be even on both sides of the thread cutter. A typical value is 55°. The default value is taken from Parameter 51. (See Chapter 11.)

Threads/Unit: Enter the number of threads-per-inch or threads-per-millimeter you want to cut. This field affects the Thread Lead field.

Thread Lead: Enter the width of a thread for one complete turn. This field affects the threads/unit entry.



Major Diameter: Enter the major diameter of the thread you want to cut.

Minor Diameter: Enter the minor diameter of the thread you want to cut.

Note: Non-pipe threads are referenced at the thread face. Pipe thread diameters are referenced according to

Draft: January 11, 2024

ANSI/ASME B1.20.1-193 (R1992). External pipe threads are referenced at E0, the diameter at the external thread face. Internal pipe threads are referenced at E3, the diameter at the end of the wrench make-up length (three turns past the nominal diameter of the external pipe thread.) For both external and internal pipe threads, this should be the smallest diameter on the taper.

Chamfer Amount: Enter the number of turns to take to withdraw the tool from the maximum depth to the surface. This produces a thread that tapers to the surface.

Taper Amount: Enter the amount that the surface rises over the length of the surface you want to thread – normally this is a negative amount for external and positive amount for internal. This field affects the thread angle field. For pipe threads, this value is calculated from the preset angle of 1.2812 degrees.

Taper Angle: Enter the angle that the surface tapers to – normally a negative angle for external and positive angle for internal. This field affects the taper amount entry. The taper angle of pipe threads is preset at 1.2812 degrees.

Clearance Z: Enter a clearance amount, or "run-up" distance from the thread face. This clearance helps get the cutting tool up to speed before it contacts the thread face.

The main screen contains the following fields:

Draft: January 11, 2024

Thread Face Z: Enter the Z-coordinate where the threading tool will first make contact with the thread (use the Clearance Z field to get a run-up to this point.)

Ending Z: Enter the Z-coordinate for the end of the threading cycle.

Minimum Cut Depth: Enter the minimum amount of material to remove during a pass. The threading cycle will remove larger amounts of material initially, but will work down to this value.

First Cut Depth: Enter the amount of material to be removed during the first cut.

Tool Num/Offset: Enter the tool and offset numbers you want to use. The first two digits are the tool number and the last two digits are the offset number.

Spindle Speed: Enter the desired spindle speed for the threading cycle. You can toggle between RPM or CSS. When toggled to RPM, a constant RPM will be maintained. When toggled to CSS, a constant surface speed will be maintained.

Finish Pass Amount: Enter the amount of material to leave for a finishing pass.

Num Spring Passes: Enter the number of passes to make at the finishing diameter.

Pre/Post Cycle Pos.: Allows you to select if you want to move to a specified position before or after the cycle. Once toggled from "None", two fields appear to enter the desired position.

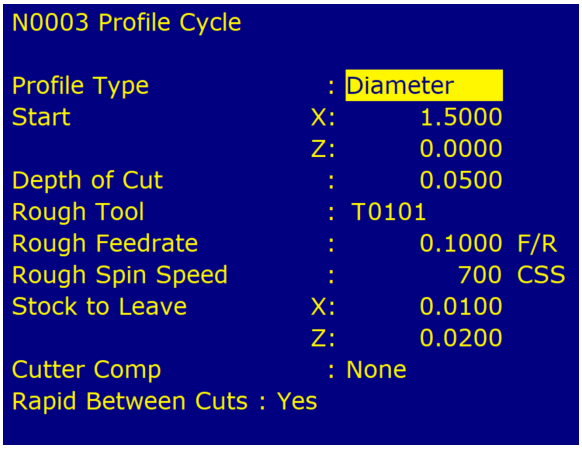
Multi-Start Threads: Multi-start (Multiple Start) threads are where there is more than one thread starting on the same diameter. Multi-start threads can be cut by changing the starting positions of each following G76 cycle. The first G76 cycle is programmed at the correct starting position for clearance, feed rate, and spindle speed. For the next G76 cycle's starting Z-position, divide the pitch by the number of Multi-start threads, then add this Z-offset value to the first G76 cycle's starting Z-position. The rest of the G76 cycle is the same. For more than two Multi-start threads, add the Z-offset to the previous G76 cycle's starting Z-position for each additional Multi-start thread. In Multi-start threads, the next thread's starting Z-position will begin further away than the previous thread's, but the ending Z-position is the same for all of the threads.

7.10 F6 – Profile

Press the **F6 – Profile** key to insert a profile.

The profile operation allows you to define a profile with lines and arcs that will be produced with a cleanout cycle.

Note: Do not move Z until the 2nd line of the profile to avoid over- and under-cutting of the part.



Press the **F1 – Type** key to toggle between options when necessary. When at least one operation is present in the profile, you can press the **F10 – Accept** key to accept the profile.

Profile Type: Enter the type of profile you want to produce. Toggle between diameter and end face. Choosing diameter will cause the cleanout cycle to be performed along the diameter, while choosing end face will cause the cleanout cycle to be performed along the face.

Start X: Enter the X-coordinate of the start of the profile. Allow for clearance.

Start Z: Enter the Z-coordinate of the start of the profile. Allow for clearance.

Start X and Start Z are the coordinates where the tool rapids to before it starts the cleanout cycle.

Note: Intercon determines whether the cleanout cycle is external or internal by the start position of the profile and the end position of the first move in the profile. If the end position of the first move is lower than the start position of the profile, the cleanout cycle is external. For external cleanout cycles, all profile operations must be lower than the start point. If the end position of the first move is higher than the start position of the profile, the cleanout cycle is internal. For internal cleanout operations, all profile operations must be higher than the start point.

Depth of Cut: Enter the amount to remove per-pass, per-side in the cleanout cycle.

Rough Tool: Enter the tool and offset numbers you want to use during the roughing portion of the cleanout cycle. The first two digits are the tool number and the last two digits are the offset number.

Rough Feed Rate: Enter the desired feed rate for the roughing portion of the cycle. You can toggle between Feed per Revolution (f/r) or Feed per Minute (f/m). Note that this Rough Feed Rate is different from the Finish Feed Rates specified within each of the Line and Arc operations inside the profile.

Rough Spin Speed: Enter the desired spindle speed for the roughing portion of the cycle. You can toggle between RPM or CSS. When toggled to RPM, a constant RPM will be maintained. When toggled to CSS, a constant surface speed will be maintained. Note that this Rough Spin Speed is different from the Finish Spindle Speeds specified within each of the Line and Arc operations inside the profile.

Draft: January 11, 2024

Stock to Leave X: Enter the amount of stock to leave on the X-axis to be removed by the finishing pass(es).

Stock to Leave Z: Enter the amount of stock to leave on the Z-axis to be removed by the finishing pass(es).

Cutter Compensation: Set cutter compensation in Chapter 9 for more details. You can toggle between None, Right, and Left.

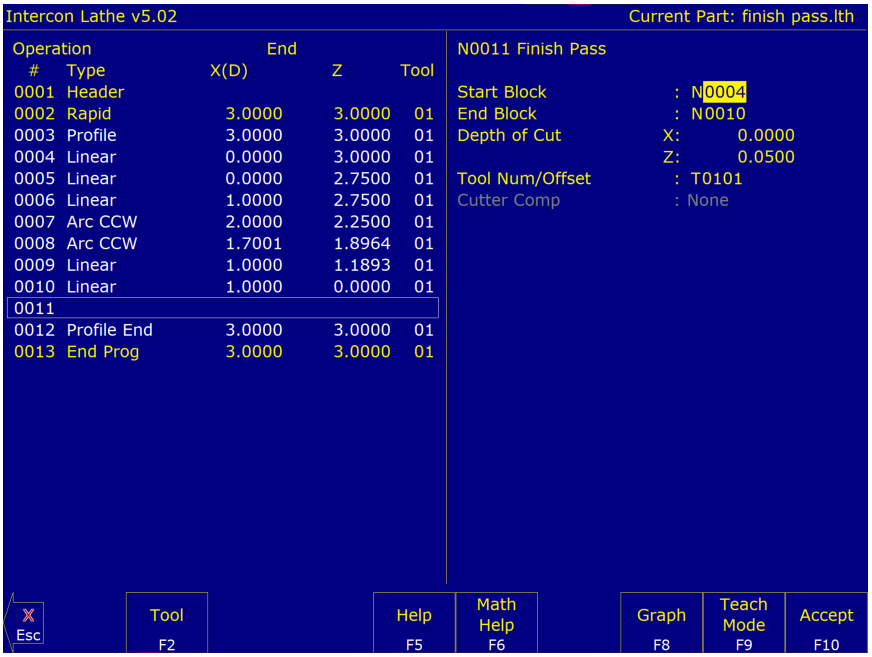
After entering these fields, define the profile you want to cut out with lines and arcs. Intercon allows you to insert Lines, Arcs, and Finish Passes within a profile. Lines and Arcs are described earlier in the manual. The Finish Pass is described later in the manual.

Rapid Between Cuts: Choose whether or not the moves between rough passes are to be done as Rapid or Feed Rate moves. You can toggle between YES and NO.

Note: The Spindle Speeds and Feed Rates specified within each of the individual Line and Arc operations inside the profile are not used by the roughing portion of the cycle. However, they will later be utilized by the Finish Pass, if it is defined.

Draft: January 11, 2024

7.10.1 Finish Pass (For Profiles Only)



The Finish Pass is a special operation that only applies to profiles. At least two operations must be present in the profile before you can insert a finishing pass. Multiple finishing passes can be inserted. Once a finish pass is inserted, you can no longer make changes in the profile without going back to the Insert Operations Menu.

Note: The number of passes made for a finish operation is determined by the greater of: Stock to leave x (Profile Operation) and Depth of cut x (Finish Operation) OR Stock to leave Z (Profile Operation) and Depth of cut Z (Finish Operation).

Start Block: Enter the block number in the profile that the finishing pass should start on.

End Block: Enter the block number in the profile that the finishing pass should end on.

Depth of Cut Z: Enter the amount of material to remove from the Z-axis per pass. 0 will be one pass.

Depth of Cut X: Enter the amount of material to remove from the X-axis per pass. 0 will be one pass.

* If both are 0, there will only be one pass.

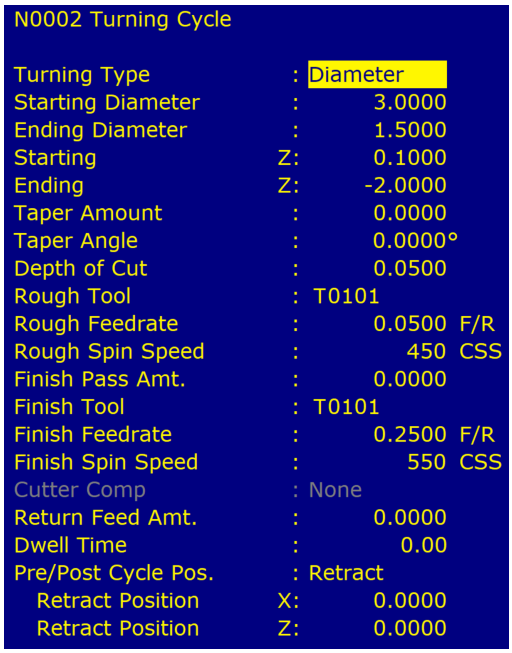
Tool Num/Offset: Enter the tool and offset numbers you want to use for the finishing pass. The first two digits are the tool number and the last two digits are the offset number. This field is disabled if G28 is not used for tool changes.

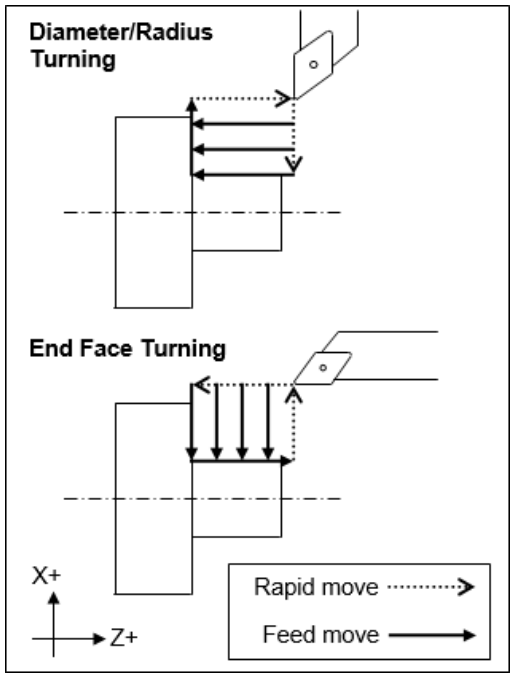
Cutter Compensation: Set cutter compensation in Chapter 9 for more details. You can toggle between None, Right, and Left.

Note: The Spindle Speeds and Feed Rates specified within each of the individual Line and Arc operations defined inside the profile will determine the Spindle Speeds and Feed Rates for the Finish Pass. This is why there is no way to specify a Spindle Speed or Feed Rate on the Finish Pass Operation page.

7.11 F7 – Turning

A turning cycle is a repetitive cycle used to cut an outside or inside diameter to a specified dimension within a specified Z-range. Press the **F7 – Turning** key to insert a turning cycle into your part program.





Press the **F1 – Type** key to toggle between options when necessary and the **F10 – Accept** key to accept these entries. Use the **up** and **down** arrow keys to move between fields. Press the **ESC** key to cancel and return to the Insert menu.

Turning Type: Enter the type of turning you want to use. Toggle between diameter/radius and end face. Choosing diameter/radius will cause the cycle to remove material in a direction parallel to the Z-axis, either along the diameter or radius. Choosing end face will cause the cycle to remove material in a direction parallel to the X-axis, along the face.

Draft: January 11, 2024

Starting Diameter/Radius: Enter the diameter at which you want the cycle to start.

Ending Diameter/Radius: Enter the diameter at which you want the cycle to finish.

Note: When turning an inside diameter, the starting diameter must be less than the ending diameter. When turning an outside diameter, the starting diameter must be greater than the ending diameter.

Starting Z: Enter the starting Z-value for the turning cycle.

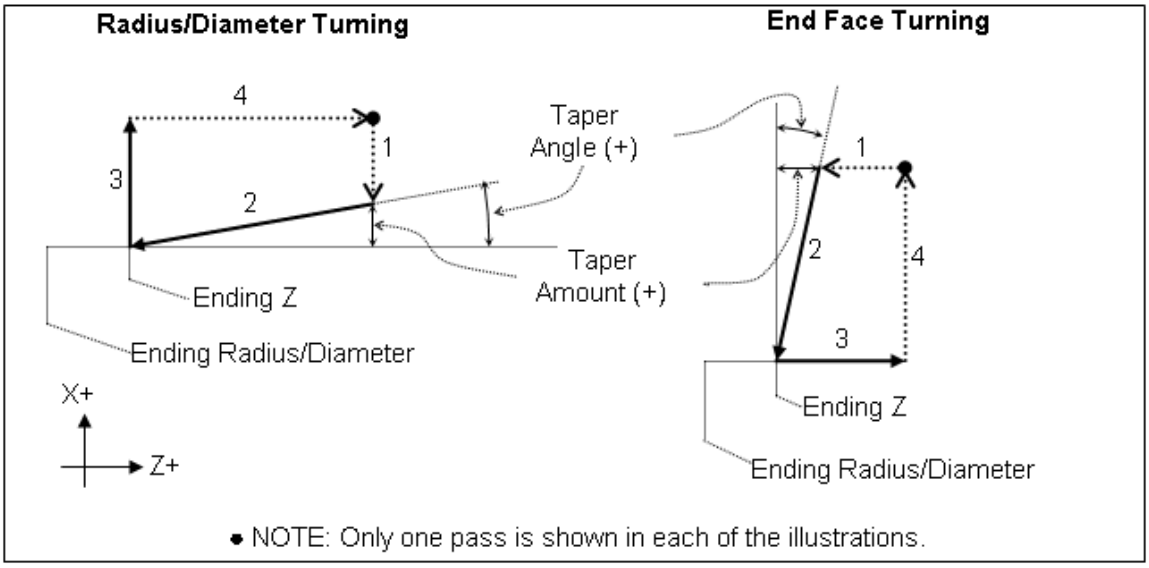
Ending Z: Enter the ending Z-value for the turning cycle.

Taper Amount: Enter the amount that you want to taper from the starting diameter to the ending diameter. This entry affects the taper angle. Entering a positive value for the turning diameter allows the user to taper from the ending diameter value plus the tapering amount. Entering a negative value for the turning diameter allows the user to taper from the ending diameter value minus the tapering amount. Entering a positive value for end face turning allows the user to taper from the ending Z taper plus the tapering amount. Entering a negative value for end face turning allows the user to taper from the ending Z taper minus the tapering amount.

Note: The taper amount must be less than the depth of cut.

Draft: January 11, 2024

Taper Angle: Enter the angle you want to use to taper. This angle is used to determine the taper amount. Entering a positive value for the turning diameter allows the user to taper from the ending diameter value plus the tapering amount. Entering a negative value for the turning diameter allows the user to taper from the ending diameter value minus the tapering amount. Entering a positive value for end face turning allows the user to taper from the ending Z taper plus the tapering amount. Entering a negative value for end face turning allows the user to taper from the ending Z taper minus the tapering amount.



Depth of Cut: Enter the amount to remove per pass

Rough Tool: Enter the tool and offset number to use for the roughing portion of the cycle. The first two digits are the tool number; the last two digits are the offset number.

Rough Feed Rate: Enter the cutting feed rate for the roughing portion of the cycle. You can toggle between feed/min and feed/rev.

Rough Spin Speed: Enter the spindle speed for the roughing portion of the cycle. You can toggle between RPM and CSS. When toggled to RPM, a constant RPM will be maintained. When toggled to CSS, a constant surface speed will be maintained.

Finish Pass Amount: Enter the amount you want the roughing portion of the cycle to leave to be removed by the finish pass. This is a radial amount. If the amount entered is zero, a finish pass will not be performed.

Finish Tool: Enter the tool and offset to use during the finishing pass. The first two digits are the tool number and the last two digits are the offset number. This field is disabled if G28 is not used for tool changes.

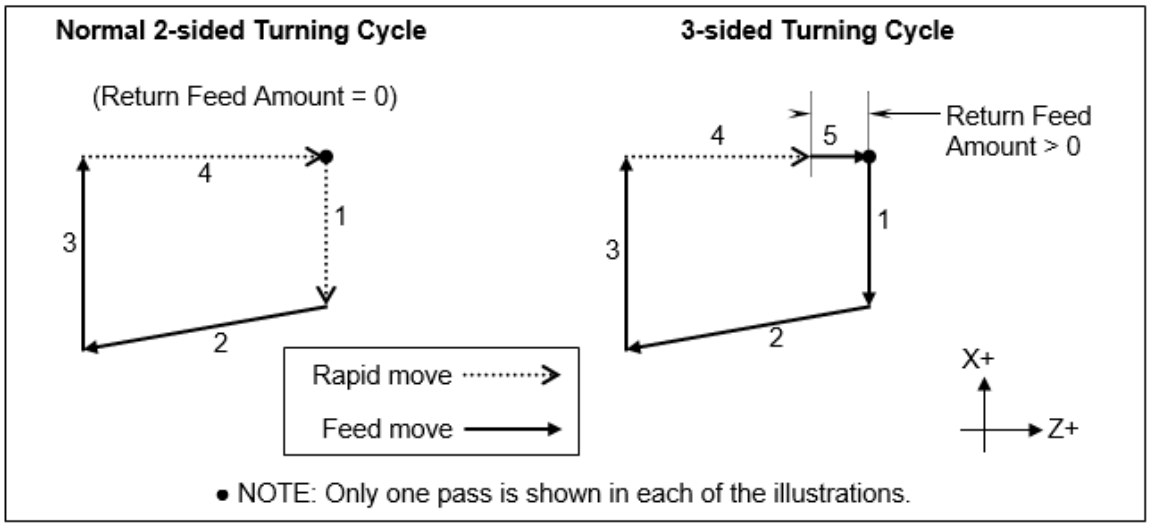
Finish Feed Rate: Enter the cutting feed rate for the finishing pass. You can toggle between feed/min and feed/rev.

Finish Spin Speed: Enter the spindle speed for the finishing pass. You can toggle between RPM and CSS. When toggled to RPM, a constant RPM will be maintained. When toggled to CSS, a constant surface speed will be maintained.

Cutter Compensation: Set cutter compensation in Chapter 9 for more details. You can toggle between None, Right, and Left.

Return Feed Amount: This is a special field that activates three-sided turning. If the value is 0, then the normal two-sided turning will be performed. If this value is more than 0, then three-sided turning will be performed. On three-sided turning, this field specifies the length of the returning feed rate move.

Draft: January 11, 2024

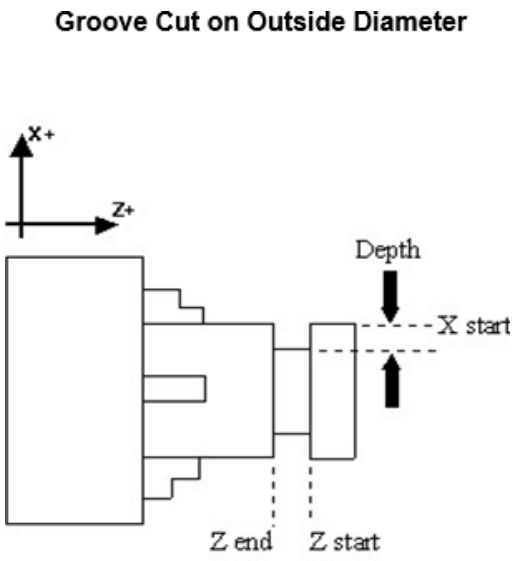


Pre/Post Cycle Pos.: Allows you to select if you want to move to a specified position before or after the cycle. Once toggled from “None”, two fields appear to enter the desired position.

7.12 F8 – Groove

```
N0003 Grooving Cycle

Type                    : Outside
Diameter
   Start                :       3.0000
   End                  :       1.2500
   Increment            :       0.1000
Width
   Start             Z:       -3.1000
   End               Z:       -3.5000
   Increment         Z:        0.0500
Corner Finish           : Bl Chamf (Len)
Corner Radius           :       0.0000
Chamfer Length          :       0.0500
Rough Tool              : T0202
Rough Feedrate          :       0.0500 F/R
Rough Spin Speed        :          450 CSS
Finish Pass Amt.        :       0.1000
Finish Tool             : T0202
Finish Feedrate         :       0.0500 F/R
Finish Spin Speed       :          500 CSS
Dwell Time              :         0.00
Pre/Post Cycle Pos.     : None
```

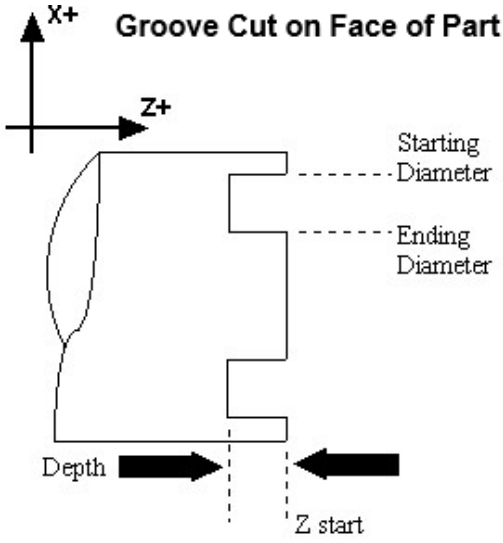


The grooving operation allows you to cut a groove of a specified width and depth in a specified location. Press the **F8 – Groove** key to insert a grooving operation. Press the **F1 – Type** key to toggle between options when necessary and the **F10 – Accept** key to accept the entries. Use the up and down arrow keys to move between fields. Press the **ESC** key to cancel and return to the Insert menu.

Type: Toggle between four options for the type of grooving. The four options are outside, inside, front, and back. Choosing outside will cause the operation to cut the groove on the outside diameter of the work piece. Choosing inside will cause the operation to cut the groove on the inside diameter of the work piece. Choosing Front will cause the operation to cut the groove on the front face of the work piece (see the example below). Choosing Back will cause the operation to cut the groove on the back face of the work piece.

```
N0003 Grooving Cycle

Type                    : Front
Depth
   Start             Z:       0.0000
   End               Z:       0.2000
   Increment         Z:       0.2000
Diameter
   Start                :       1.8000
   End                  :       1.6000
   Increment            :       0.1000
Corner Finish           : Square
Corner Radius           :       0.0000
Chamfer Length          :       0.0500
Rough Tool              : T0404
Rough Feedrate          :       0.0020 F/R
Rough Spin Speed        :          600 CSS
Finish Pass Amt.        :       0.0050
Finish Tool             : T0404
Finish Feedrate         :       0.0010 F/R
Finish Spin Speed       :          600 CSS
Dwell Time              :         0.00
Pre/Post Cycle Pos.     : None
```



Starting Diameter/Radius: Enter the position of the surface on which the groove will be produced.

Draft: January 11, 2024

Ending Diameter/Radius: Enter the groove's ending dimension.

Depth Increment: Enter the depth increment for the grooving cycle. This is the amount removed per plunge in the peck cutting cycle used to produce the groove.

Starting Z: Enter the starting position of the groove.

Ending Z: Enter the ending position of the groove. For the outside or inside diameter, it will be a Z-value. For the front or back face, this will be an X-value. You can toggle between absolute and incremental position. When toggled to absolute, enter the absolute position in reference to the part zero. When toggled to incremental, an INC will appear next to the entry. In this mode, enter the X-distance from the last point.

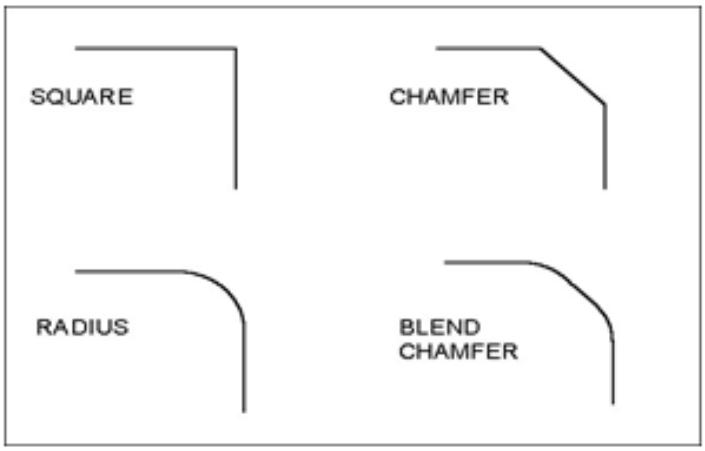
Width Increment: Enter the width increment for the grooving cycle. This is the step over amount for the cleanout cycle used to produce the width.

Corner Finish: Enter the type of corner finish you want. Toggle between square, radius, chamfer (Distance or Length), and blend chamfer (Distance or Length). Shown below is each type of corner that will be produced for the groove.

Corner Radius: Enter the radius for the rounded corner when corner finish is set to radius.

Chamfer Distance: Enter the Distance to be removed from the end of each linear segment.

Chamfer Length: Enter the length of the chamfer you want for the corner finish.



Rough Tool Number: Enter the tool and offset number to use for the roughing portion of the cycle. The first two digits are the tool number and the last two digits are the offset number.

Rough Feed Rate: Enter the cutting feed rate for the roughing portion of the cycle. You can toggle between feed/min and feed/rev.

Rough Spin Speed: Enter the spindle speed for the roughing cycle. You can toggle between RPM and CSS. When toggled to RPM, a constant RPM will be maintained. When toggled to CSS, a constant surface speed will be maintained.

Finish Pass Amount: Enter the amount you want the roughing portion of the cycle to leave to be removed by the finish pass. This is a radial amount. If the amount entered is zero, a finish pass will not be performed.

Finish Tool Number: Enter the tool and offset number to use during the finishing pass. The first two digits are the tool number and the last two digits are the offset number. This field is disabled if G28 is not used for tool changes.

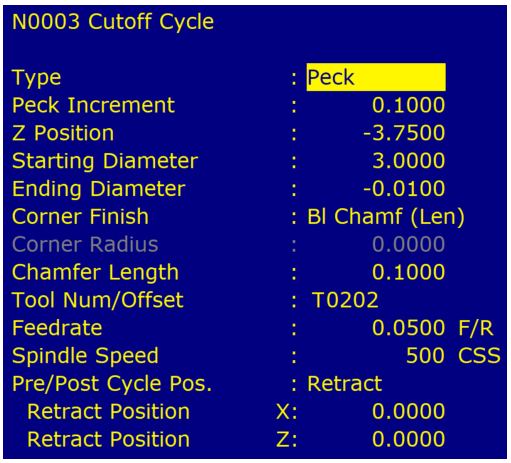
Finish Feed Rate: Enter the cutting feed rate for the finishing pass. You can toggle between feed/min and feed/rev.

Draft: January 11, 2024

Finish Spindle: Enter the spindle speed for the finishing pass. You can toggle between RPM and CSS. When toggled to RPM, a constant RPM will be maintained. When toggled to CSS, a constant surface speed will be maintained.

Pre/Post Cycle Pos.: Allows you to select if you want to move to a specified position before or after the cycle. Once toggled from “None”, two fields appear to enter the desired position.

7.13 F9 – Cutoff



The cutoff operation allows you to cut off the part with a cutoff tool.

Press the **F1 – Type** key to toggle between options when necessary and the **F10 – Accept** key to save changes. Use the up and down arrow keys to move between fields. Press the **ESC** key to cancel and return to the Insert Menu.

Type: Enter the type of cut used to cut off the work piece. You can toggle between continuous and peck. Choosing continuous will cause the work piece to be cut off with a continuous cut. Choosing peck will cause the work piece to be cut off in incremental moves.

Peck Increment: When the type field is set to peck, enter the increment amount used in cutting the part off. When the type field is set to continuous, this field will not be shown.

Z-position: Enter the Z-position of the cut.

Starting Diameter: Enter the diameter at which the cutoff is to start.

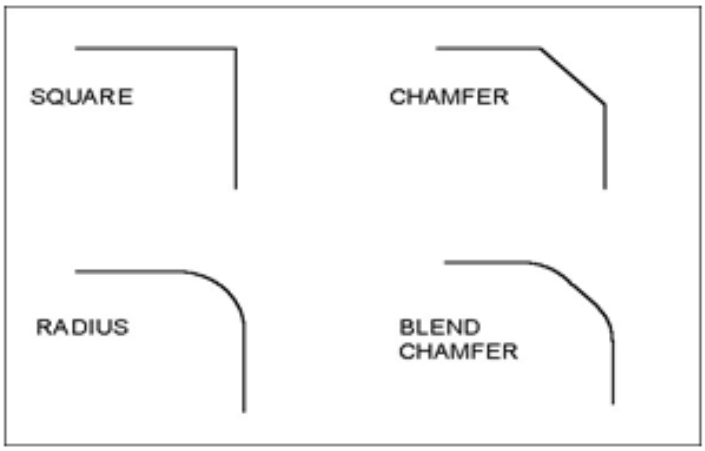
Ending Diameter: Enter the diameter at which the cutoff is to finish.

Corner Finish: Enter the type of corner finish you want. Toggle between square, radius, chamfer (Distance or Length), and blend chamfer (Distance or Length). Shown below is each type of corner that will be produced for the cutoff. Corner finish will be on the start diameter.

Corner Radius: Enter the radius of the corner you want for the corner finish. This field is only shown when radius is chosen for the corner finish.

Chamfer Distance: Enter the Distance to be removed from the end of each linear segment.

Chamfer Length: Enter the length of the chamfer you want for the corner finish. This field is only shown when chamfer is chosen for corner finish.



Draft: January 11, 2024

Tool Num/Offset: Enter the tool number and offset number you want to use. The first two digits are the tool number and the last two digits are the offset number.

Feed Rate: Enter the cutting feed rate to cut off the work piece. You can toggle between feed/min and feed/rev.

Spin Speed: Enter the spindle speed for the work piece cutoff. You can toggle between RPM and CSS. When toggled to RPM, a constant RPM will be maintained. When toggled to CSS, a constant surface speed will be maintained.

Pre/Post Cycle Pos.: Allows you to select if you want to move to a specified position before or after the cycle. Once toggled from "None", two fields appear to enter the desired position.

7.14 F10 – Other

The **F10 – Other** key displays additional operations.

If the 3rd-axis label in the machine configuration is set to 'C' and Parameter 93 is set for C-axis operation, or if the 4th-axis label in the machine configuration is set to 'C' and Parameter 94 is set for C-axis operation, there will be options for C-axis and C Indexing operations shown.

The options shown at the bottom of the **F10 – Other** screen are described below. Press the **ESC** key to cancel and return to the Insert Operation Menu.

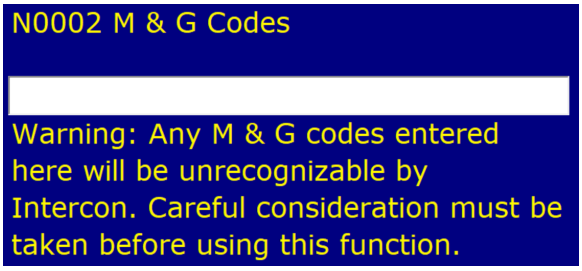
Draft: January 11, 2024

7.14.1 F1 – Comment

Press the **F1 – Comment** key to enter a comment. This comment can be up to 35 characters long and will be displayed in the generated CNC program.

7.14.2 F2 – M-&G-code

Press the **F2 – M-&G-code** key to enter M- and G-codes directly into the part program.



After entering the M- and G-codes, you may press the **F10 – Accept** key to accept the entry or the **ESC** key to cancel and return to the Insert Operation Menu.

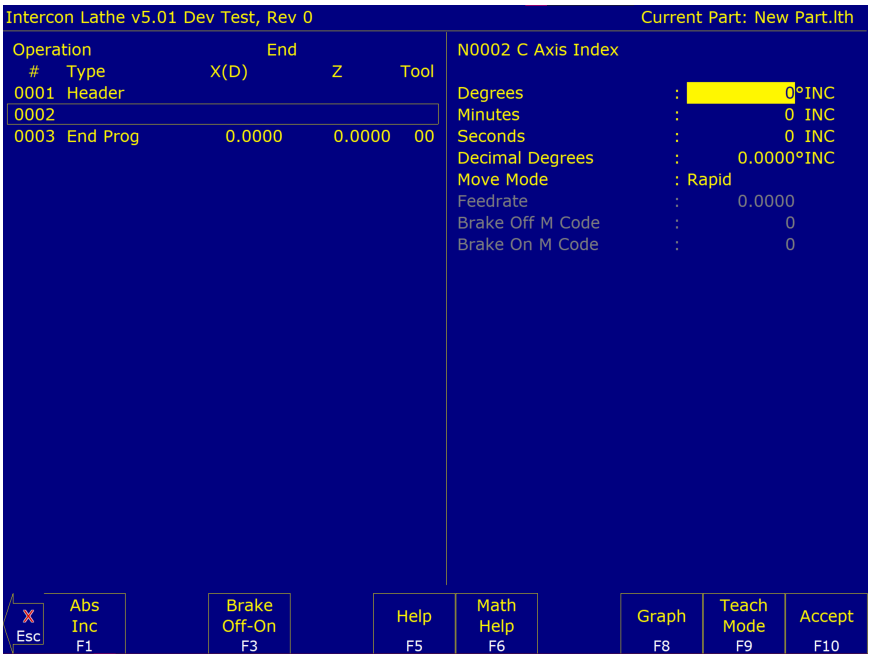
7.14.3 F3 – C-axis

Press the **F3 – C-axis** key to enter the C-axis edit operation screen.

Press the **F1 – Toggle** key or space bar to toggle between ON and OFF. Press the **F10 – Accept** key to accept the entry or the **ESC** key to cancel and return to the Insert Operation Menu.

7.14.4 F4 – C Index

Press the **F4 – C Index** key to enter the C Indexing Operation screen.



Press the **F1 – Abs/Inc** key to toggle between Incremental (INC) and Absolute (ABS) positioning. Press the **F3 – Brake Off-On** key to toggle the brake fields OFF and ON.

Degrees: The number of degrees you want to move the C-axis. This value can be positive or negative.

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

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

Move Mode: Rapid positioning or Feed Rate move.

Feedrate: This is the degrees-per-minute at which to move if the aforementioned Move Mode is set to Feed Rate. Otherwise, this field is not used.

Decimal degrees: This is another method of entering the number of degrees. If you choose to enter the movement of the C-axis with the fields listed above, then 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.

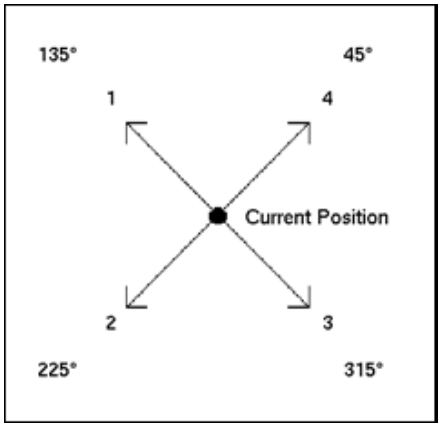
Brake On M-code: The number of the M-code to output for the braking function. The brake fields must be toggled on to allow the editing of this field. When the brake fields are on, code will be output to turn off the brake, position the C-axis, and then turn on the brake.

Draft: January 11, 2024

Brake Off M-code: The number of the M-code to output for the braking function. The brake fields must be toggled ON to allow the editing of this field.

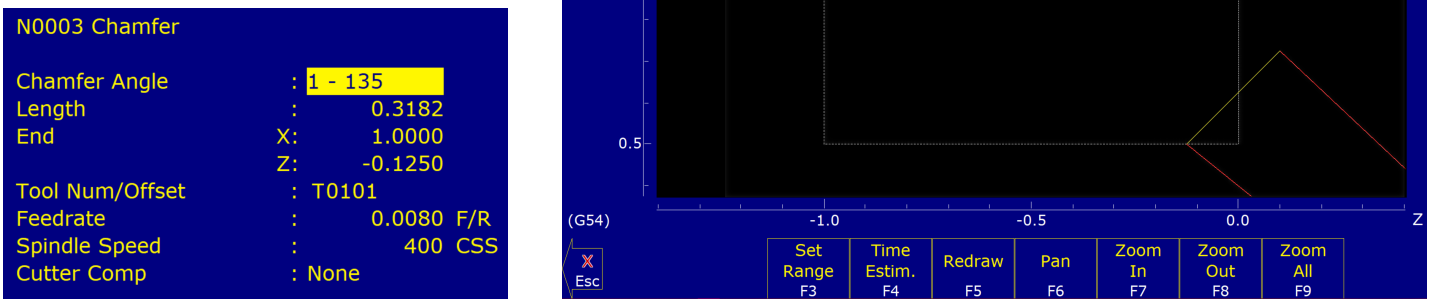
Press the **F10 – Accept** key to accept the entry or the **ESC** key to cancel and return to the Insert Operation Menu.

7.14.5 F8 – Chamfer



Press the **F8 – Chamfer** key to enter the chamfer operation screen. This is a one-shot operation. It generates a cutting move from the current position at one of four angles, as shown in the picture above.

Example:



Chamfer Angle: Press the space bar or Keys 1 through 4 to choose one of the four following angles: 135, 225, 315, and 45.

Length: If the length is known, enter it here. Intercon will then calculate the End X and Z for you.

End X, End Z: Enter either the X or Z end position here. Intercon will then calculate the other axis end position and length based on the selected angle.

Draft: January 11, 2024

Tool Num/Offset: In one-shot mode, this will be filled-in with the current tool number.

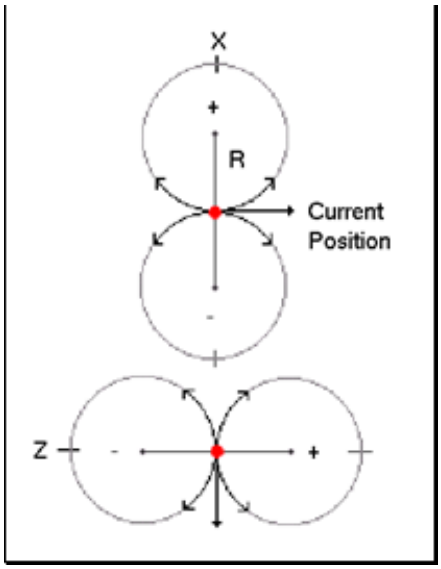
Feedrate: Enter the cutting feed rate for the chamfer. You can toggle between feed/min and feed/rev.

Spindle: Enter the spindle speed for the chamfer. You can toggle between RPM and CSS. When toggle to RPM, a constant RPM will be maintained. When toggled to CSS, a constant surface speed will be maintained.

Cutter Compensation: Set cutter compensation in Chapter 9 for more details. You can toggle between None, Right, and Left.

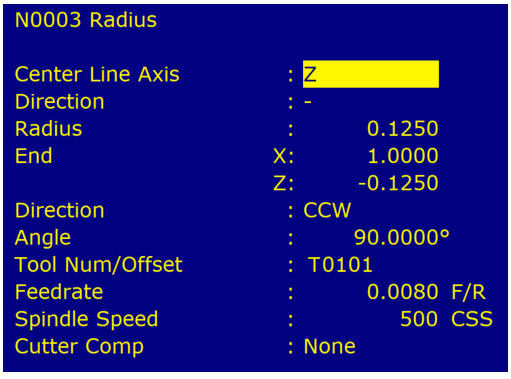
Press the **F10 – Accept** key to accept the entry or the **ESC** key to cancel and return to the Insert Operation Menu.

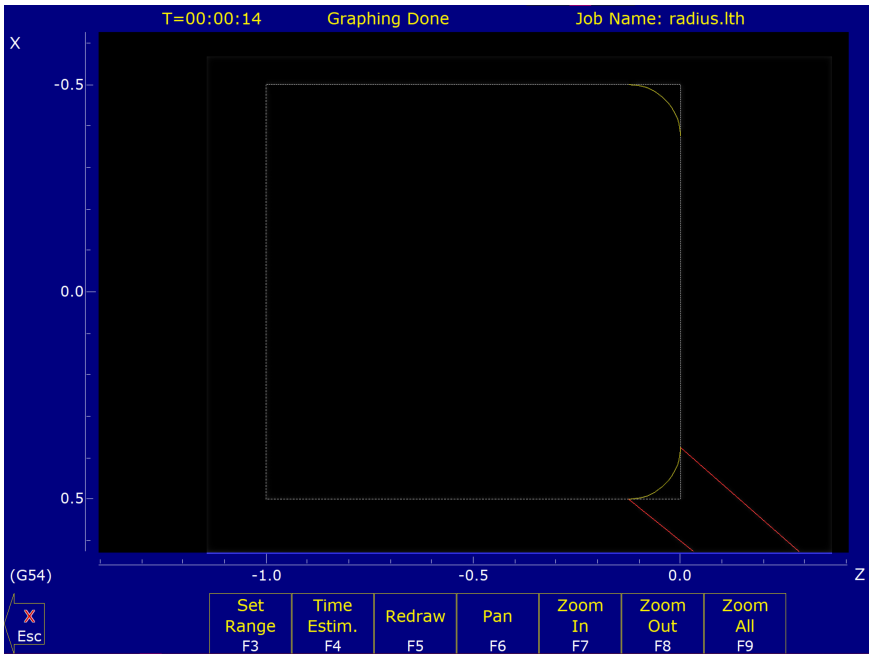
7.14.6 F9 – Radius



Press the **F9 – Radius key** to enter the radius operation screen. This is a one shot operation. It generates an arc move from the current position in one of eight directions, as shown in the picture above.

Example:





Draft: January 11, 2024

Center Line Axis: This chooses four of the eight possible arcs. X selects a center point on the X-axis; Z selects a center point on the Z-axis. Press the space bar to toggle or press the X and Z keys.

Direction: The direction to move on the selected axis. It is also the direction of the center point from the current position. Press the space bar to toggle between "+" and "-". This chooses two out of four possible arcs.

Radius: The radius of the arc.

End X, End Z: If known, enter the end position of the arc. Intercon will calculate the other axis end point, arc direction, and angle automatically.

Arc Direction: Use the space bar to select CW (clockwise) or CCW (counter-clockwise).

Tool Num/Offset: In one shot mode, this will be filled in with the current tool number.

Feedrate: Enter the cutting feed rate for the arc. You can toggle between feed/min and feed/rev.

Spindle: Enter the spindle speed for the arc. You can toggle between RPM and CSS. When toggled to RPM, a constant RPM will be maintained. When toggled to CSS, a constant surface speed will be maintained.

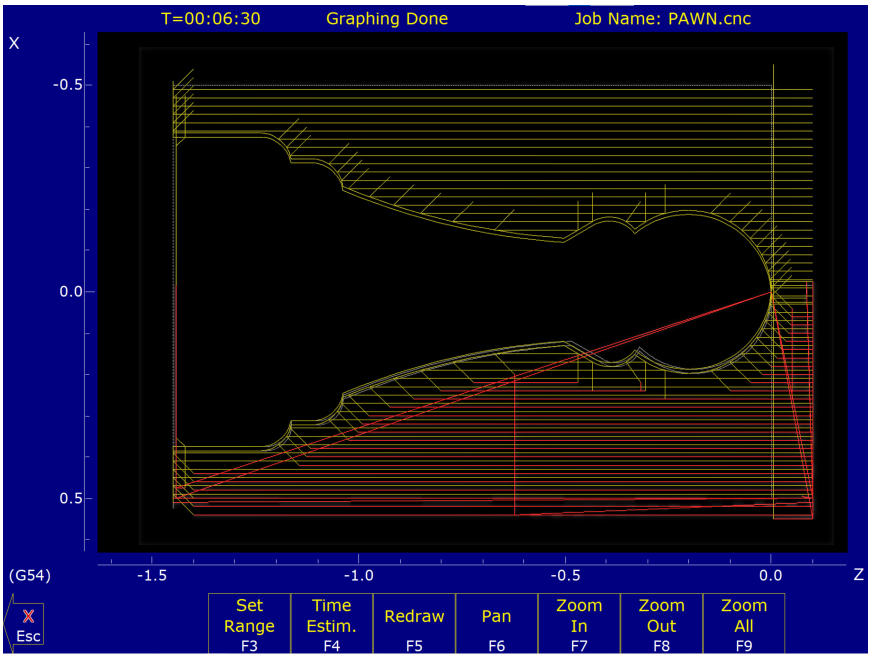
Cutter Compensation: Set cutter compensation in Chapter 9 for more details. You can toggle between None, Right, and Left.

Press the **F10 – Accept** key to accept the entry or the **ESC** key to cancel and return to the Insert Operation Menu.

Draft: January 11, 2024

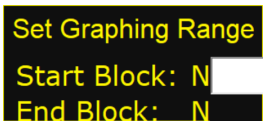
7.15 Graphics

Press the **F8 – Graph** key from the Intercon Main Menu, the File Menu, or any Edit Operation Menu to view the graphics menu. A wireframe of your part will appear.



7.15.1 F3 – Range

Press the **F3 – Range** key to graph a portion of a part program.



Start Block: Enter the start block number for the portion of the part program you want to graph.

End Block: Enter the end block number for the portion of the part program you want to graph.

Press the **F10 – Accept** key to accept the entries and the **ESC** key to cancel.

7.15.2 F4 – Time

Press the **F4 – Time** key to get an estimate of the time it will take to produce the part.

7.15.3 F5 – Redraw

Press the **F3 – Redraw** key to redraw the graphic.

7.15.4 F6 – Pan

Press the **F6 – Pan** key to move the part around the graph window. After pressing this key, crosshairs will appear. Move the crosshairs around with the arrow keys. Pick a location on the part with the crosshairs and press the **F6 – Pan** key to pan this selection to the center of the graph window.

7.15.5 F7 – Zoom In

Press the **F7 – Zoom In** key to zoom into the part. The graph will zoom in to the center of the graph window.

7.15.6 F8 – Zoom Out

Press the **F8 – Zoom Out** key to zoom out from the part. The graph will zoom out from the center of the graph window.

7.15.7 F9 – Zoom All

Press the **F9 – Zoom All** key to fit the entire part within the graph window.

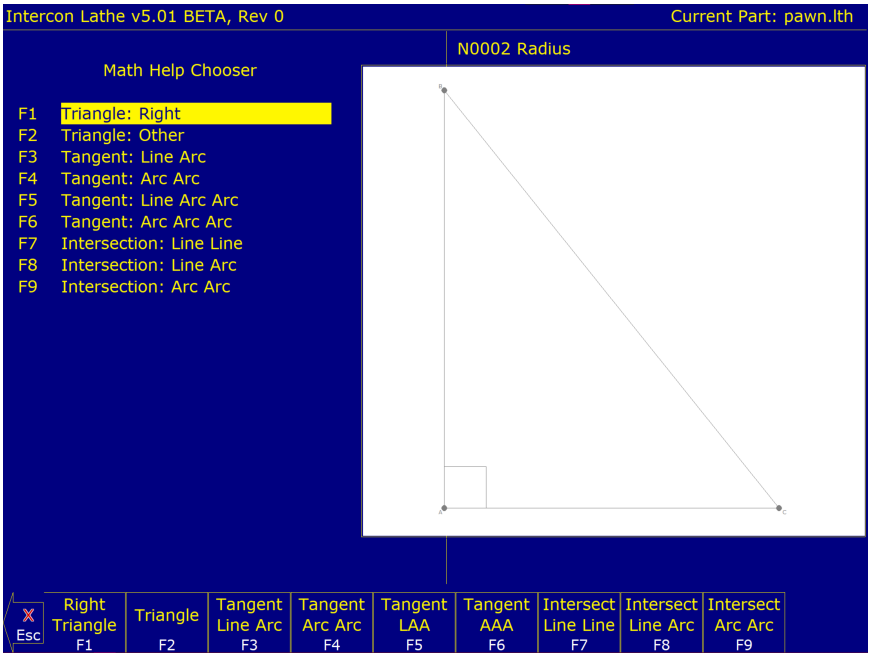
7.15.8 1–9, 0, Space – Feed Rate Override & Hold

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. The space bar toggles feed hold on and off.

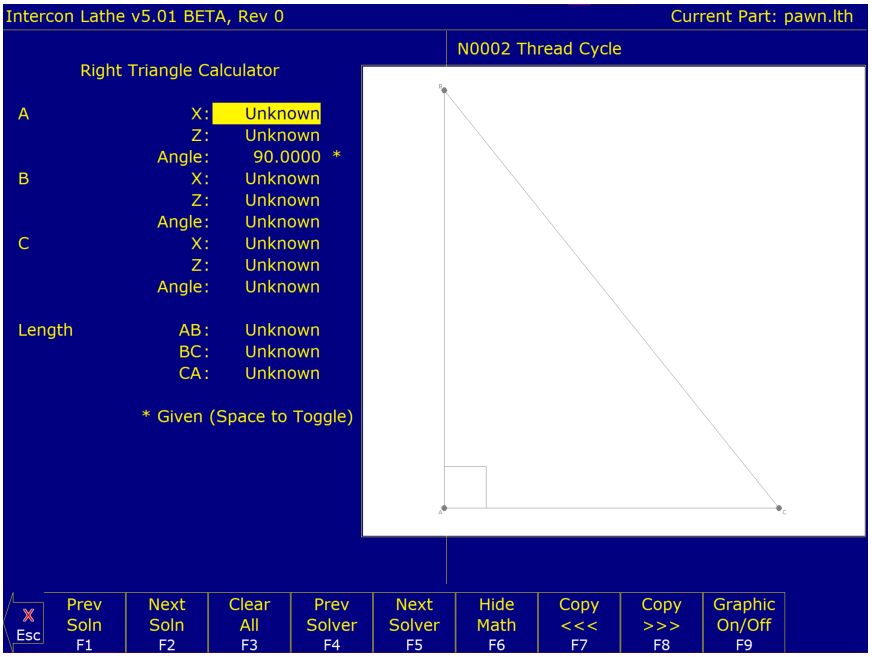
7.16 Math Help

Intercon provides a math assistance function to solve common trigonometric problems 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*:

Draft: January 11, 2024



The figures on the right are a graphical representation of the highlighted solver on the left. Pressing the **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:



7.16.1 F1 – Prev Soln and F2 – Next Soln

The **Prev Soln** and **Next Soln** options will cycle backward and forward, respectively, through the available solution sets for Math Help 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 available solutions and the solution number currently represented by the graphic display on the right. For example, in an Arc Tangent Arc's Math Help, the display solution status may be "– Solution 1 of 8 –". In this case, the **Prev Soln** and **Next Soln** keys can be used to cycle through all eight of the solutions.

Draft: January 11, 2024

7.16.2 F3 – Clear All

The Clear All option removes all solutions. It sets all fields for a particular solver to UNKNOWN.

7.16.3 F4 – Prev Solver and 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 that have the same effect as pressing the **ESC** key to reach the main Math Help menu, navigating to the previous or next Math Help option, and then pressing the **ENTER** key.

7.16.4 F6 – Hide Math

The Hide Math option exits Math Help mode and returns to the operation edit menu. Pressing the **F6 – Hide Math** key to invoke Math Help again will restore Math Help exactly as you left it.

7.16.5 F7 – Copy >>> and F8 – Copy <<<

The **Copy >>>** option will move the value from the selected edit operation field into the selected Math Help menu field and the **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. If the graphics display is visible when choosing one of these options, the effect is to turn off the graphics display. 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 the fields, as described below.

Draft: January 11, 2024

7.16.6 F9 – Graphic On/Off

The Graphic On/Off option will remove the graphical representation of the Math Help menu from the display. This is helpful when copying data between Intercon operations and Math Help.

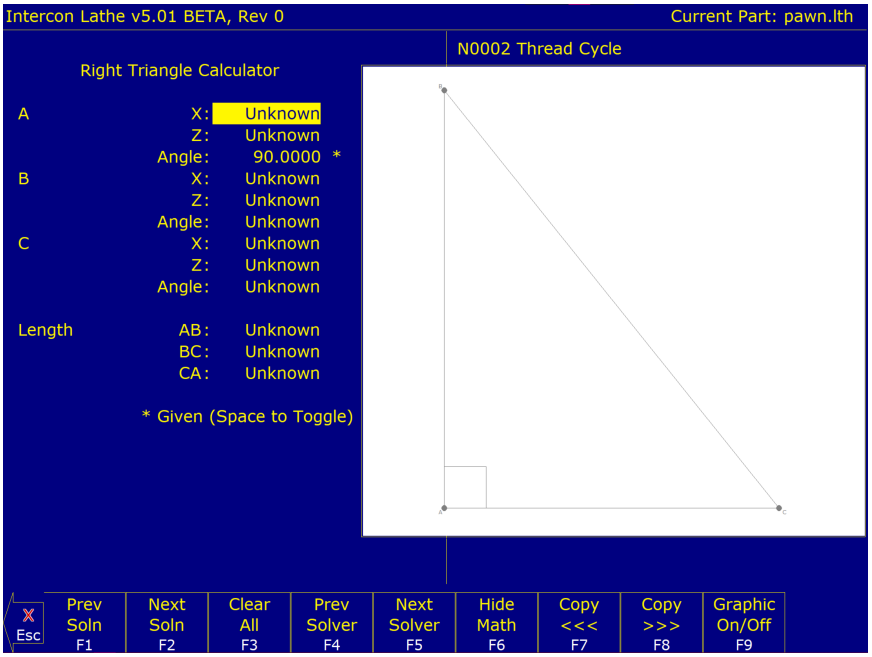
7.16.7 ↑↓←→ (Arrow Keys) – Select Fields

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, as described previously, 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.

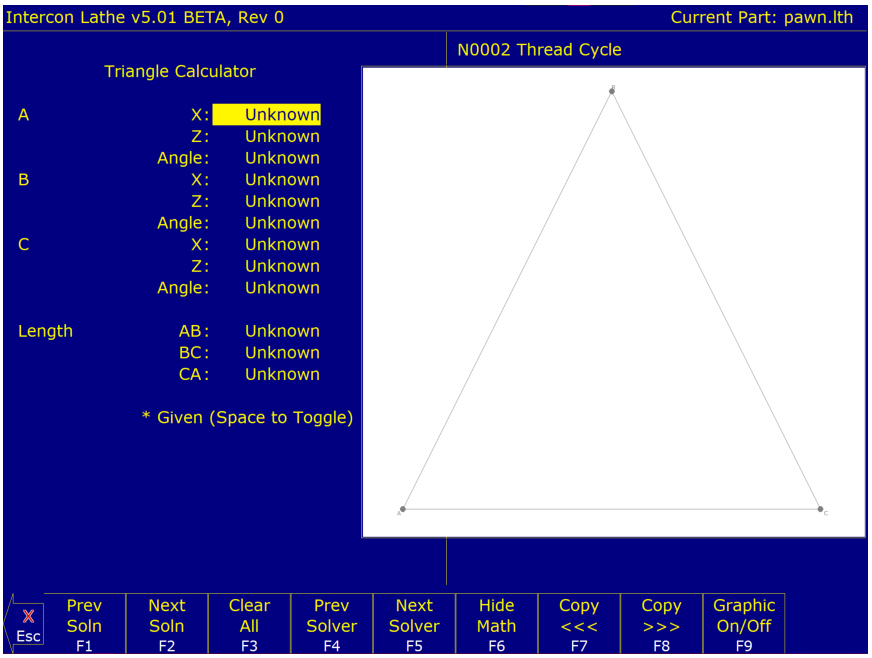
7.16.8 Other Features Common to All Math Help Operations

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.

7.16.9 F1 – Trangle: Right



7.16.10 F2 – Triangle: Other



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.

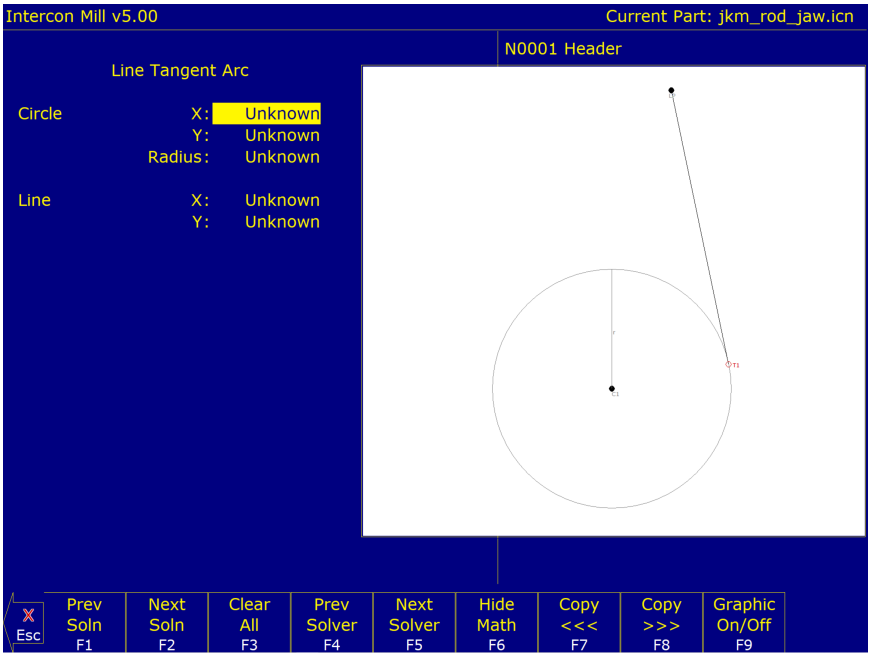
Length of values are the distances between the points indicated.

Continue adding all the known parameters. Select parameters using the arrow keys. When Math Help solves the

Draft: January 11, 2024

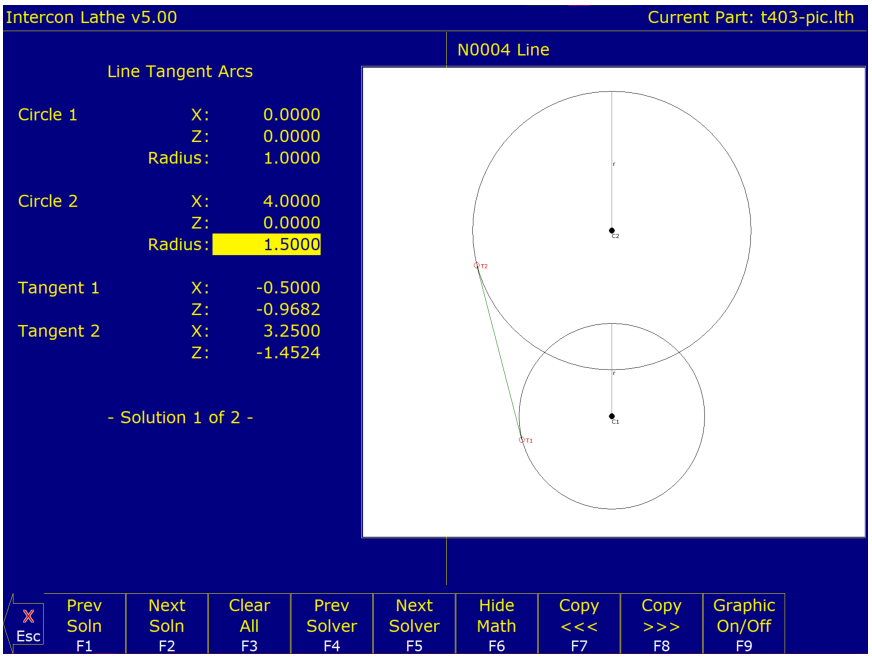
remaining unknown values, the screen will display them.

7.16.11 F3 – Tangent: Line Arc



Given the center (C1) and radius of an arc and one point (LP) on a line, it finds 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.

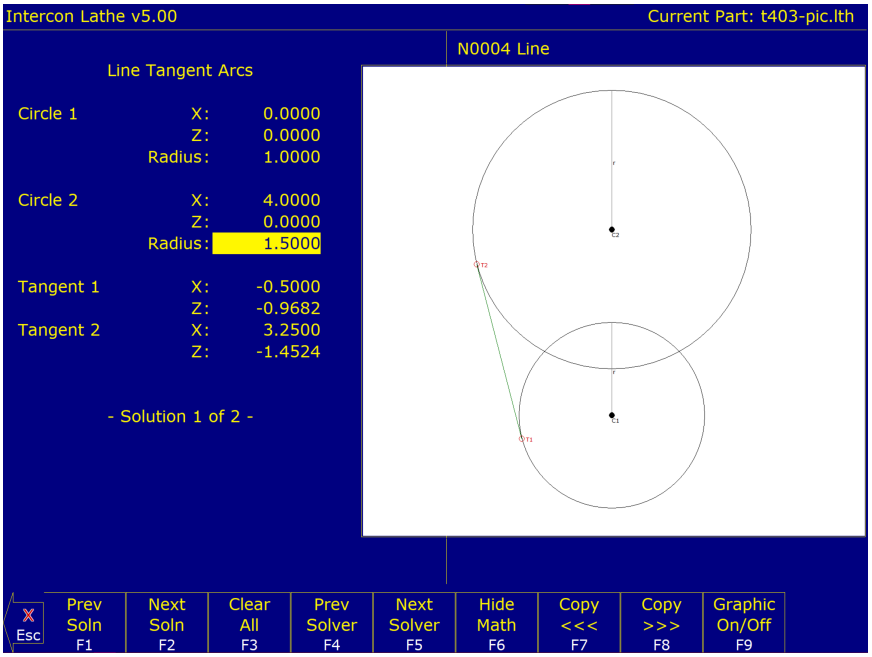
7.16.12 F4 – Tangent: Arc Arc



Given the center points (CP1 and CP2) and radii (R1 and R2) of two arcs, it finds 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.

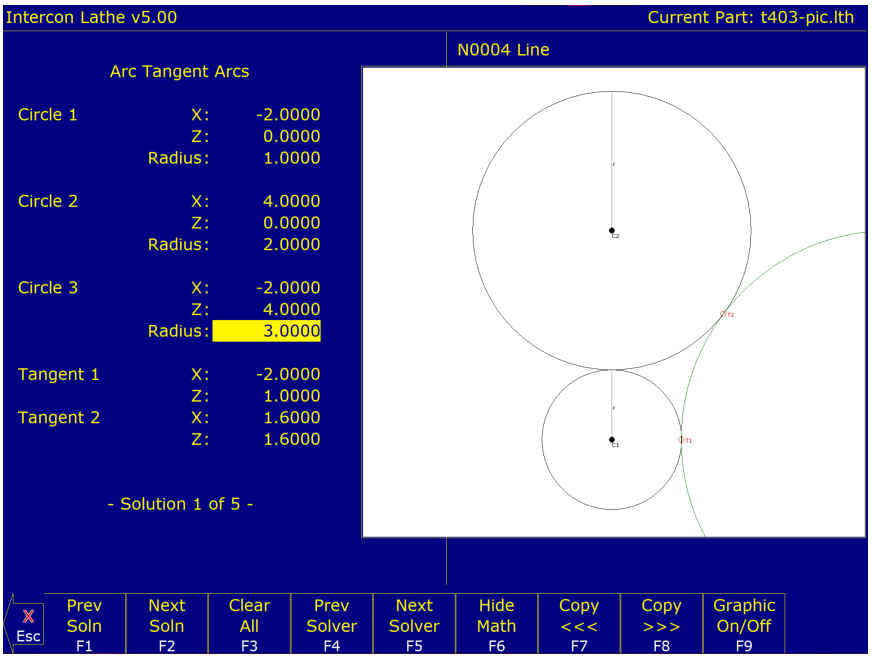
Draft: January 11, 2024

7.16.13 F5 – Tangent: Line Arc Arc



Given the center points (CP1 and CP2) and radii (R1 and R2) of two arcs, it finds the lines (defined by T1–T8) 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.

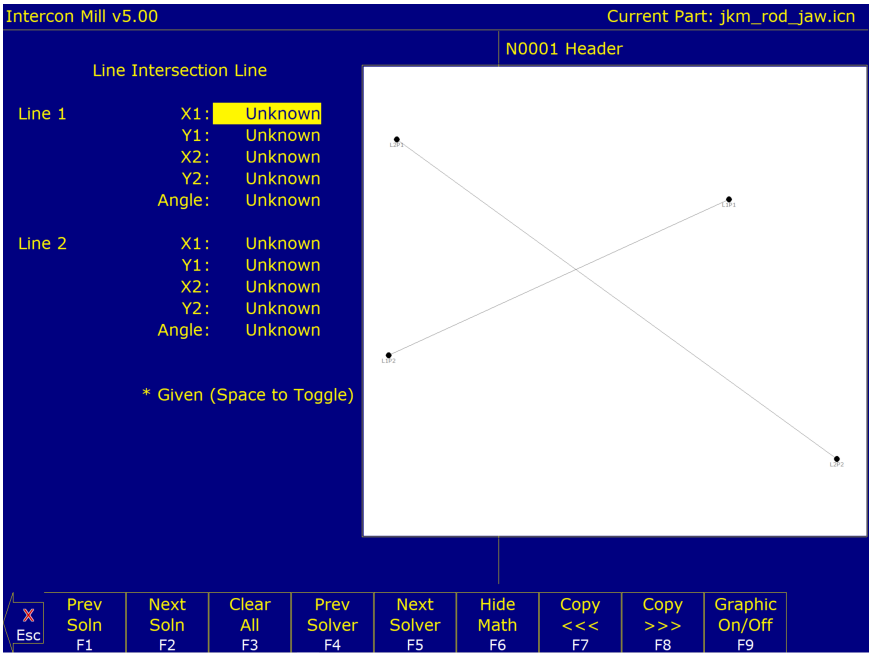
7.16.14 F6 – Tangent: Arc Arc Arc



Given the center points (C1 and C2), the radii of two arcs, and the radius of a third arc, it finds 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.

Draft: January 11, 2024

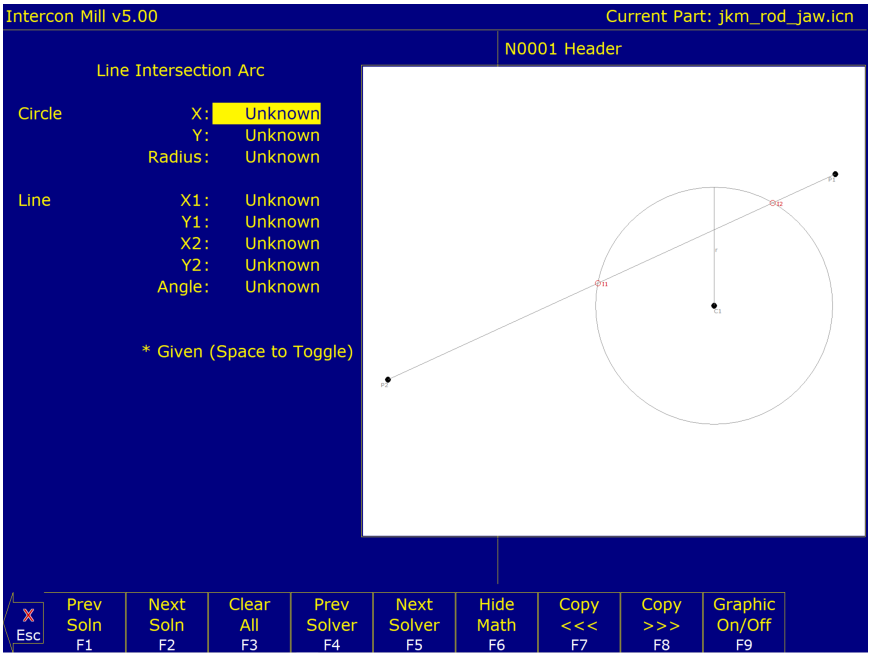
7.16.15 F7 – Intersection: Line Line



You must enter the X- and Y-coordinates for one 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.

7.16.16 F8 – Intersection: Line Arc



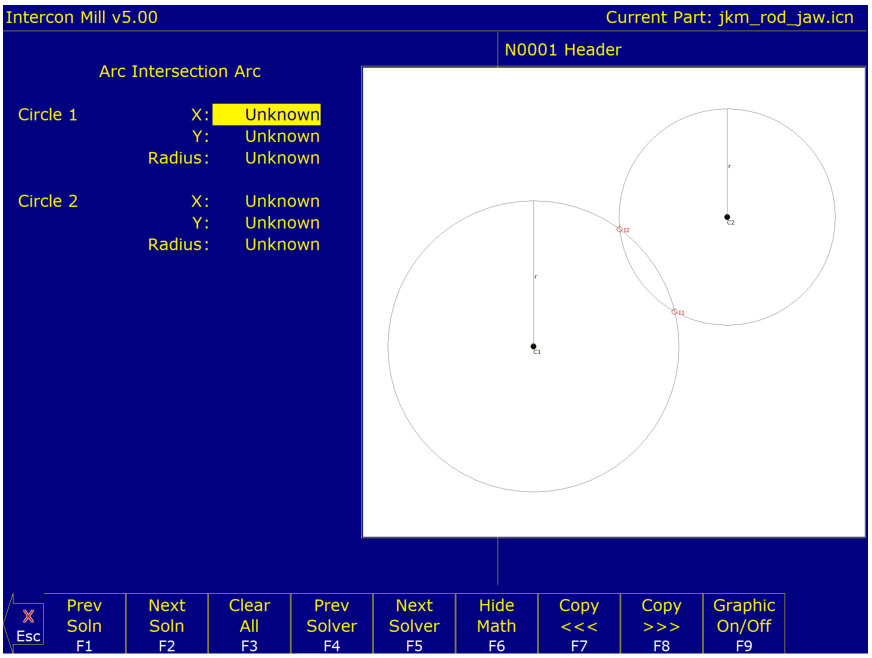
Draft: January 11, 2024

Given the center point (CP) and radius (R) of an arc, one point (LP1), and either a second point (LP2) or one coordinate (LP2 X or Y) and the angle from horizontal, it finds 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.

7.16.17 F9 – Intersection: Arc Arc



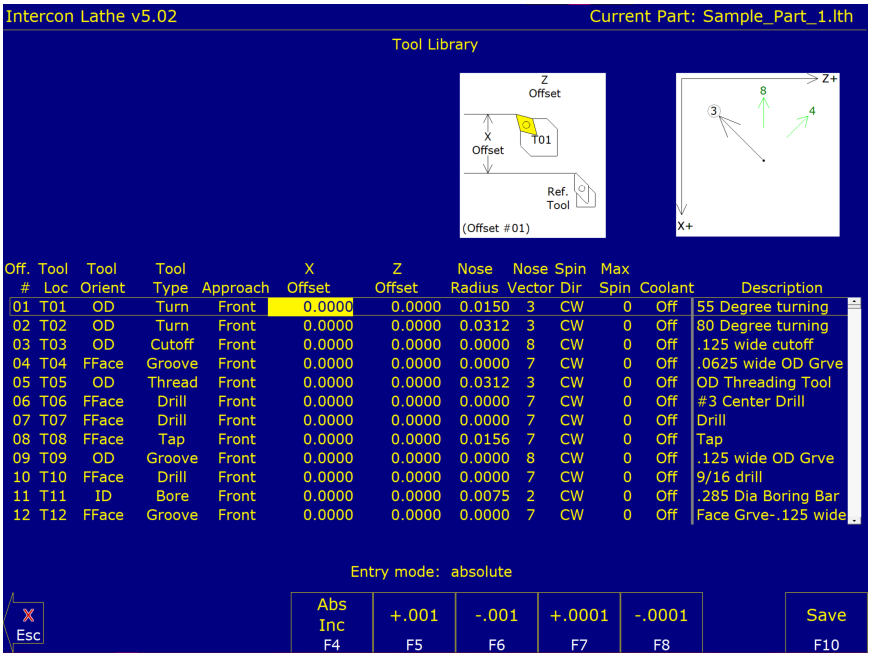
Given the center points (CP1 and CP2) and the radii (R1 and R2) of two arcs, it finds the intersection point(s) (I1 and I2) of the arcs.

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.

7.17 Intercon Lathe Tool Library

You can press the **F2 – Tool lib** key in most Edit Operations screens to enter the Tool Library Screen.

Draft: January 11, 2024



Use the **UP** and **DOWN** arrow keys to select which tool offset to edit. When editing a tool, press the **ENTER** key to accept the entry and move onto the next field for that tool, or use the left and right arrow keys to move from field to field. You can also use the **F5 – +.001** or **F6 – -.001** keys to adjust the offsets and nose radius values by a small increment. Absolute/Incremental entry mode for the offset and nose radius values can be toggled with the **F4 – Abs/Inc** key.

Press the **F10 – Accept** key to accept the highlighted offset for the current operation and save any changes. Press the **Esc** key to cancel the offset selection. If you made changes, you will be asked if you wish to save them.

Offset # (Tool Offset): This non-editable field serves as the row number reference into the Tool Library. Use the up and down arrow keys to select a tool offset.

Tool Loc (Tool Number): Enter the tool number (01–99) that you want to associate with the tool offset number. Usually the Tool Location would be associated with the same numbered Tool Offset. For example, Tool #1 would have Location T01 and Offset 01, therefore T0101. However, there may be situations where you may want to specify two or three different offsets for tool #1. For instance, T0102 would be Location T01 and would use Offset 02 and T0103 would be location T01 and use offset 03.

Tool Orient: This is the tool operation direction, which generally describes the direction in which the tool points. It also implies the orientation of the work surface. Choices here are OD (outer diameter), ID (inner diameter), FFace (front face), and RFace (rear face).

Tool Type: This field describes the tool's general shape and intended operation. Use "Custom" if the tool does not match any of the available tool types.

Approach (from): This field is the tool approach direction. This indicates which side of the work piece centerline that the tool must travel from in order to start operation on the work-piece. For gang tool lathes (Parameter 163=1), the approach direction is independently set for each tool. For non-gang tool lathes, this field is locked to the same approach direction for every tool and cannot be modified, and such an approach direction is based on the lathe's overall machine orientation (as set in Parameter 1). See Chapter 11 for information about Parameters 1 and 163.

X Offset: Enter the amount to adjust the X-axis position when tool offsets are used.

Z Offset: Enter the amount to adjust the Z-axis position when tool offsets are used.

Draft: January 11, 2024

Nose Radius: Enter the nose radius of the tool. This field is used by cutter compensation, if it is turned on.

Nose Vector: Enter the nose vector of the tool. This tells Lathe Intercon how the tool is oriented in the machine. This field affects the behavior of cutter compensation, and also affects the tool retraction moves when a tool change occurs in a program.

Spin Dir (Spindle Direction): Enter the spindle direction for the tool. Toggle between off, clockwise, and counter-clockwise.

Max Spin (Max. Spindle Speed (G50)): The maximum spindle speed for the tool. A G50 is posted with the tool change using this value as the S parameter. If the value is zero, the G50 value from the Setup screen is used.

Coolant: Specify the coolant for each tool. Toggle between off, flood, and mist

Description: Enter a description of the tool.

7.18 Lathe Intercon Tutorial #1

This is a step-by-step example of creating a part from a blueprint using Intercon. The tool path to be created is for turning a ball end onto the end of a one-inch diameter piece of stock. Before beginning, be sure you are following these five steps to successful turning:

- Determine the tools necessary to machine the part by analyzing the print.
- Set the X- and Z-offsets for each tool (see Chapter 4).
- Program the part using Intercon (see Chapter 7).
- Set the Part Zero position on the stock to be machined (see Chapter 5).
- Graph the part to check for programming errors, then machine the part.

Draft: January 11, 2024

This exercise begins after the print has been analyzed, the tools have been chosen, and the X- and Z-offsets have been set. For this particular example, the end face coordinates of the part are chosen to be X0 and Z0. The procedure outlined in the following pages will give you step-by-step instructions for programming the part (Figure 7.1) using Intercon.

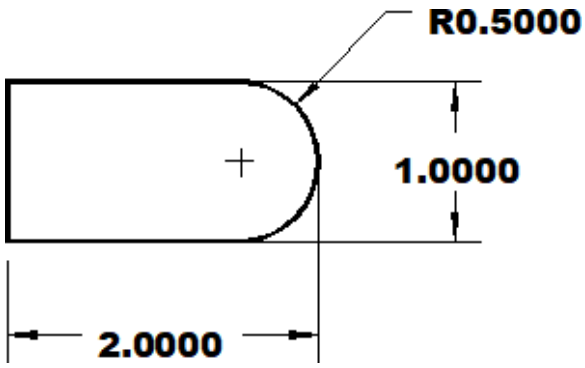


Figure 7.1: Part to be Programmed

Each feature of the part will become an operation in your program. Beginning from the main screen, the following series of keystrokes will describe the step-by-step process of programming the part shown in Figure 7.1.

7.18.1 Create a New Part Program

PRESS	ACTION	COMMENTS
F5	CAM	CAM Selection menu.
F1	ICN	Starts Lathe Intercon.
F9	Setup	Modify setup parameters (see the next page).
F10	Accept	Save modified setup parameters.
F1	File	Opens the File Menu.
F1	New	Creates a new program. Enter “demo1” as the name for the file.
F10	Accept	Accept the file name. Fill in the dialog box exactly as shown in Figure 7.2.
F10	Accept	Creates a new part file using the data entered.

These tutorials assume the options Modal Linear and Arc are turned on in Intercon Setup (**F9** in the Intercon main menu). When these options are turned on, accepting a Linear or Arc operation automatically inserts a new Linear or Arc operation after it. The Esc key can be used to cancel the new operation if it is not desired and return to the operation menu. If these options are not turned on, the user must press the **F1** or **F2** keys to insert a new Linear or Arc operation. The operations shown in the examples have the taper angle and modal input fields turned on. To make your entry screens look like the examples, go to the Setup screen to make sure that the parameters match the ones shown below.

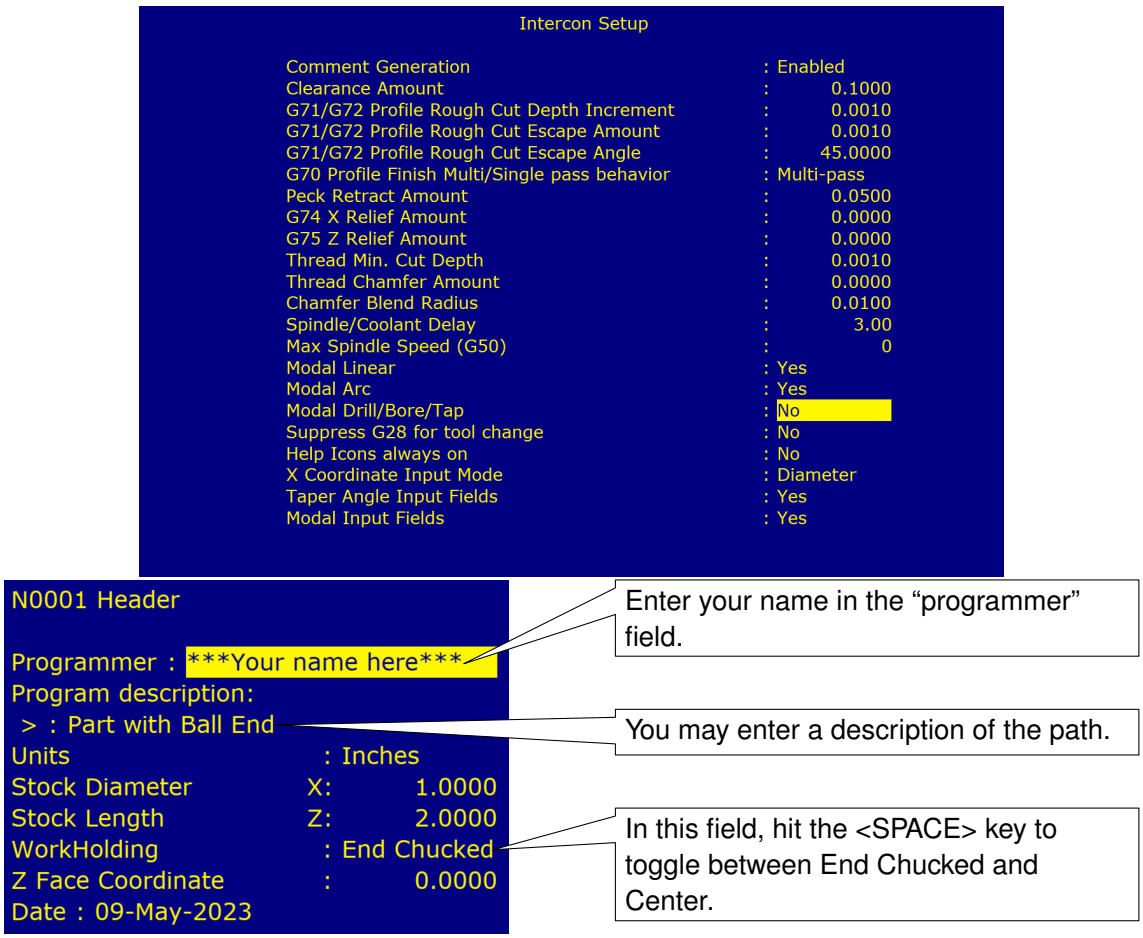


Figure 7.2

Draft: January 11, 2024

7.18.2 Insert the First Cycle

PRESS	ACTION	COMMENTS
F7	Turning	Creates a repetitive cycle used to cut an outside or inside diameter to a specified dimension within a specified Z-range. Fill in the Edit Operation side of the screen, as shown in Figure 7.3.
F8	Graph	Generates a graph of the part to this point, as shown in Figure 7.4. This preview can be used to detect problems that may occur if the part was cut now.
Esc	Escape/Cancel	Returns to the Editing window.
F10	Accept	Saves the data.

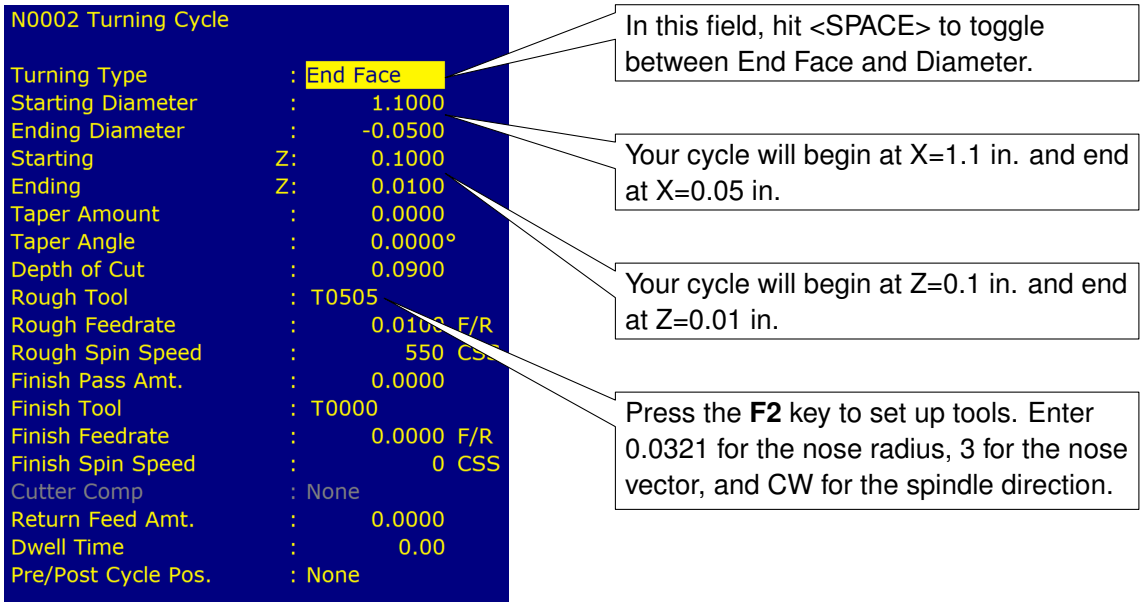


Figure 7.3

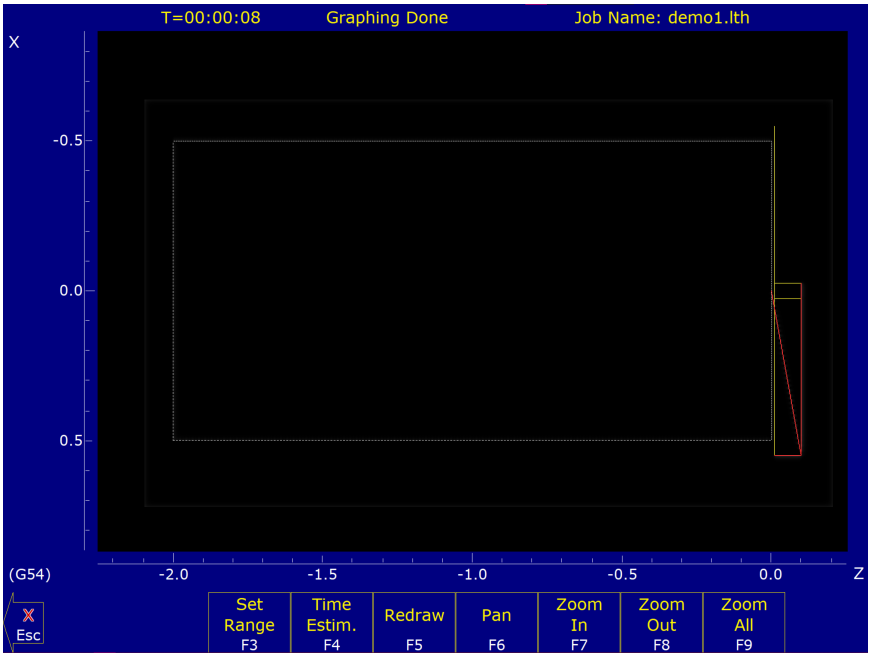


Figure 7.4

Draft: January 11, 2024

7.18.3 Create a Profile

PRESS	ACTION	COMMENTS
F6	Profile	Defines a profile with lines and arcs that will be produced with a cleanout cycle. Fill in the Edit Operation portion of the screen, as shown in Figure 7.5. The first profile command will create the move shown in Figure 7.6.

Note: The line number displayed in the Edit operation window is the line number for the end of the profile (which is currently line 40).

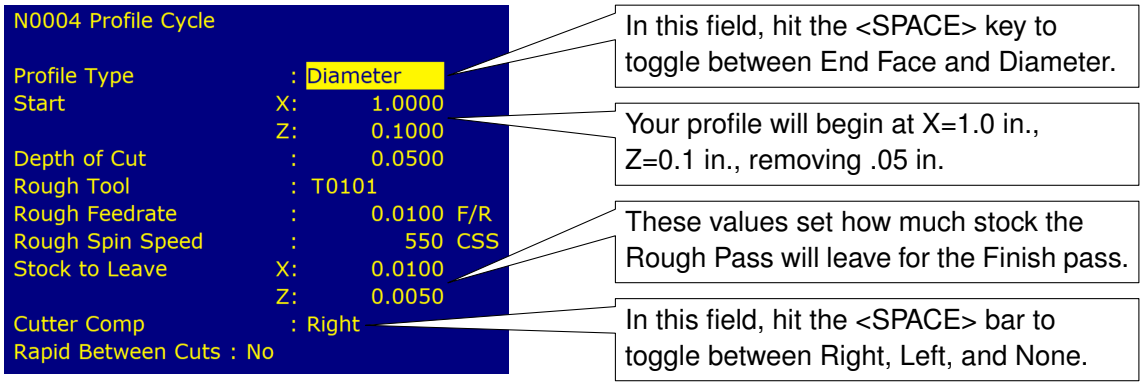


Figure 7.5: Beginning of Profile Cycle

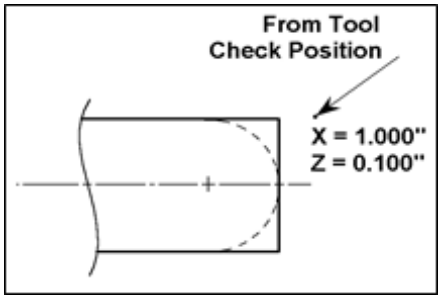


Figure 7.6: First Profile

Note: For T1, press the **F2 – Tool** key and enter 0.321 for the nose radius, 3 for the nose vector, and CW for the spindle direction.

PRESS	ACTION	COMMENTS
F10	Accept	Accept Profile parameters and enter into Profile Edit mode.
F1	Line	Inserts a line into your profile (Figure 7.8). Fill in the Edit Operation portion of the screen exactly as shown in Figure 7.7.

Draft: January 11, 2024

```
N0004 Line

Linear Type          : Feedrate
End                X:      -0.0500
                   Z:       0.1000
Taper Angle          :   270.0000°
Taper Length         :      0.5250
Connect Type         : None
Connect Radius       :      0.0000
Chamfer Distance     :      0.0000
Tool Num/Offset      : T0101
Finish Feedrate      :      0.0100 F/R
Finish Spindle Speed :         550 CSS
Cutter Comp          : Right
```

Figure 7.7: Line 1 Edit Screen (Modal and Taper displays on)

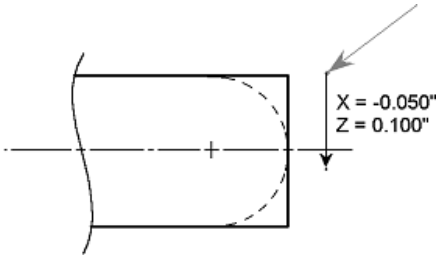


Figure 7.8: First Line in Profile

PRESS	ACTION	COMMENTS
F10	Accept	Saves the data for the first Linear operation and automatically inserts another Linear operation. This linear operation will be the second linear operation in Figure 7.10. Fill in the Edit Operation portion of the screen exactly as shown in Figure 7.9. Notice that End X 0 is incremental.

Draft: January 11, 2024

```
N0005 Line

Linear Type          : Feedrate
End                X:      0.0000 INC
                   Z:      0.0000
Taper Angle          :   180.0000°
Taper Length         :      0.1000
Connect Type         : None
Connect Radius       :      0.0000
Chamfer Distance     :      0.0000
Tool Num/Offset      : T0101
Finish Feedrate      :      0.0100 F/R
Finish Spindle Speed :         550 CSS
Cutter Comp          : Right
```

Use the **F1 – Abs/Inc** key to toggle

Figure 7.9: Line 2 Edit Screen (Modal and Taper displays on)

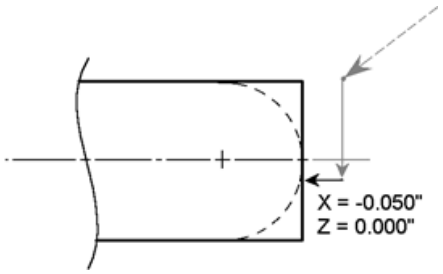


Figure 7.10: Second Linear in Profile

PRESS	ACTION	COMMENTS
F10	Accept	Saves the data for Line 2 and automatically inserts another line operation. This next line will be Line 3 in Figure 7.12. Fill in the Edit Operation portion of the screen exactly as shown in Figure 7.11.

```
N0006 Line

Linear Type          : Feedrate
End                X:       0.0000
                   Z:       0.0000
Taper Angle          :      90.0000°
Taper Length         :       0.0250
Connect Type         : None
Connect Radius       :       0.0000
Chamfer Distance     :       0.0000
Tool Num/Offset      : T0101
Finish Feedrate      :       0.0100 F/R
Finish Spindle Speed :          550 CSS
Cutter Comp          : Right
```

Figure 7.11: Line 3 Edit Screen

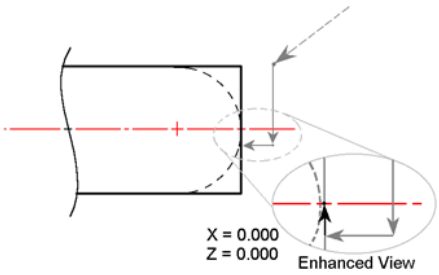


Figure 7.12: Third Line in Profile

PRESS	ACTION	COMMENTS
F10	Accept	Saves the data for Line 3 and automatically inserts another line operation.
Esc	Escape/Cancel	Cancel the current line operation. Return to the profile edit menu.
F2	Arc	Inserts an arc into the profile (Figure 7.14). Fill in the Arc Edit Operation portion of the screen exactly as shown in Figure 7.13.
F10	Accept	Saves the data and automatically inserts another arc operation.

```
N0007 Arc

Arc Type             : EP & R
Mid                X:       0.7071
                   Z:      -0.1464
End                X:       1.0000
                   Z:      -0.5000
Center             X:       0.0000
                   Z:      -0.5000
Radius               :       0.5000
Angle                :      90.0000°
Direction            : CCW
Connect Radius       :       0.0000
Tool Num/Offset      : T0101
Finish Feedrate      :       0.0100 F/R
Finish Spindle Speed :          550 CSS
Cutter Comp          : Right
```

Figure 7.13: Arc Edit Screen (Modal displayed)

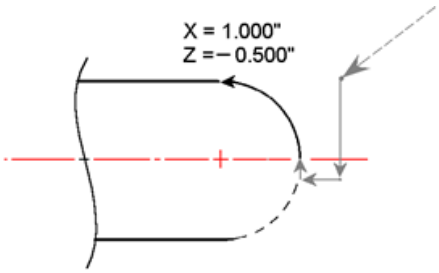


Figure 7.14: Arc (0.5" Dia.) in Profile

PRESS	ACTION	COMMENTS
F1	Line	Inserts a fourth line into your profile (Figure 7.16). Fill in the Line Edit Operation portion of the screen exactly as shown in Figure 7.15.

Draft: January 11, 2024

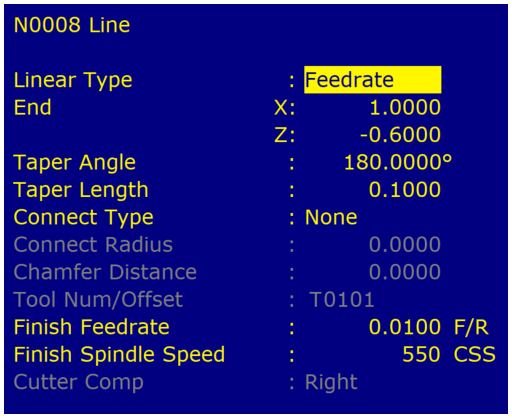


Figure 7.15: Line 4 Edit Screen

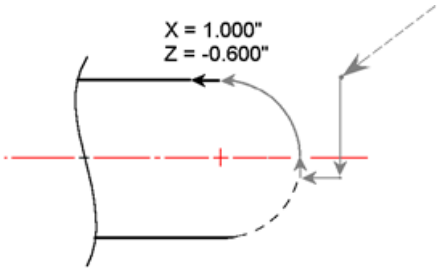


Figure 7.16: Last Line in Profile

7.18.4 Include A Finish Pass

PRESS	ACTION	COMMENTS
F10	Accept	Saves the data for Line 4 and automatically inserts another line operation.
Esc	Escape/Cancel	Cancel current line. Return to profile edit screen.
F3	Finish	Creates a finish pass through the whole profile to remove material left by the rough pass (Figure 7.18). If no Depth of Cut is set here, the finish pass will remove all the material in one pass. Fill in the Edit Operation portion of the screen exactly as shown below in Figure 7.17.

Note: The depth of material left to be removed by the Finish Pass is defined in the beginning of the Profile (shown in Figure 7.5) in the fields marked 'Stock to Leave'.

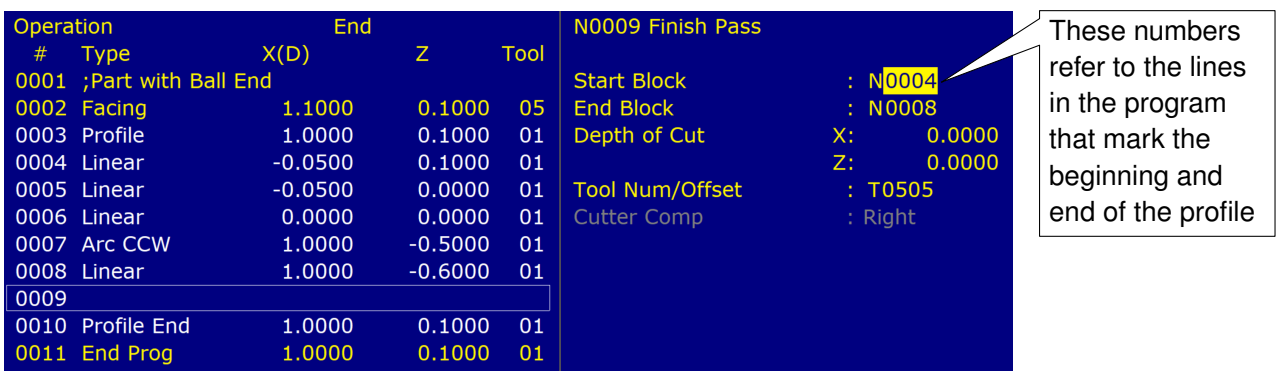


Figure 7.17: Full Screen View of Finish Pass Edit Screen

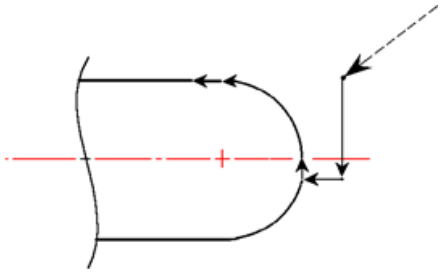


Figure 7.18: Finish Pass Over Whole Profile

Draft: January 11, 2024

7.18.5 Graph the Final Part

PRESS	ACTION	COMMENTS
F8	Graph	Generates a graph of the finished part, as shown in Figure 7.19. This preview can be used to detect problems that may occur if the part was cut now.

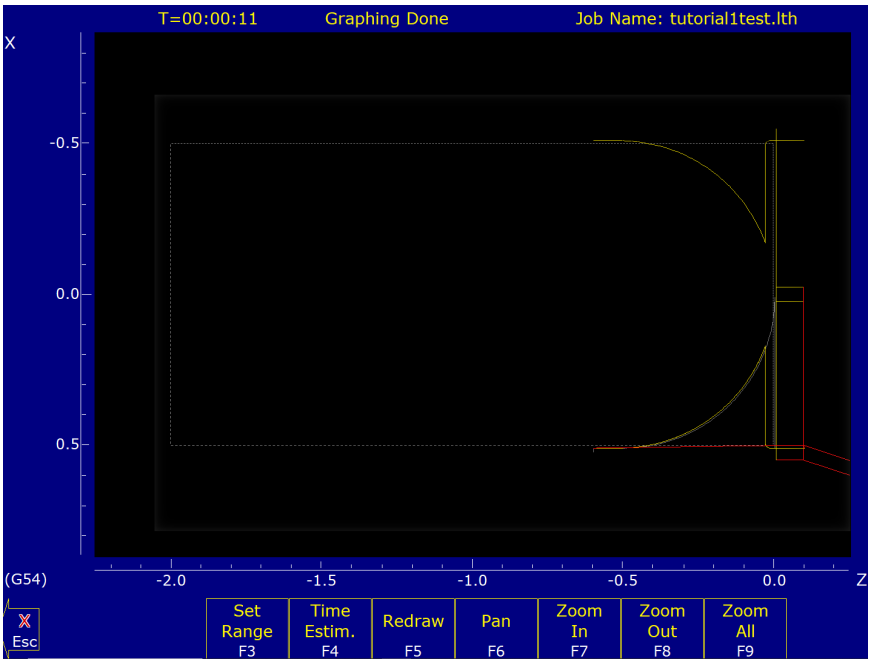


Figure 7.19: Graph of Finished Part

7.18.6 Post the Part and Exit

PRESS	ACTION	COMMENTS
Esc	Escape/Cancel	Returns you to the Editing window.
F10	Accept	Saves the data and returns to the profile editing screen.
Esc	Escape/Cancel	Returns you to the Main Programming window.
F10	Post	Saves and posts the job to the control, creating G-codes for the program.

7.19 Lathe Intercon Tutorial #2

This is a step-by-step example of creating a part from a blueprint using Intercon. The tool path to be created is for the part shown in Figure 7.20. Before beginning, be sure you are following these five steps to successful turning:

- Determine the tools necessary to machine the part by analyzing the print.
- Set the X- and Z-offsets for each tool (see Chapter 4).
- Program the part using Intercon (Lathe Intercon Manual).
- Set the Part Zero position on the stock to be machined (see Chapter 5).

Draft: January 11, 2024

- Graph the part to check for programming errors and then machine the part.

This exercise begins after the print has been analyzed, the tools have been chosen, and the X- and Z-offsets have been set. For this particular example, the end face coordinates of the part are chosen to be X0 and Z0. The procedure outlined in the following pages will give you step-by-step instructions for programming the part shown below.

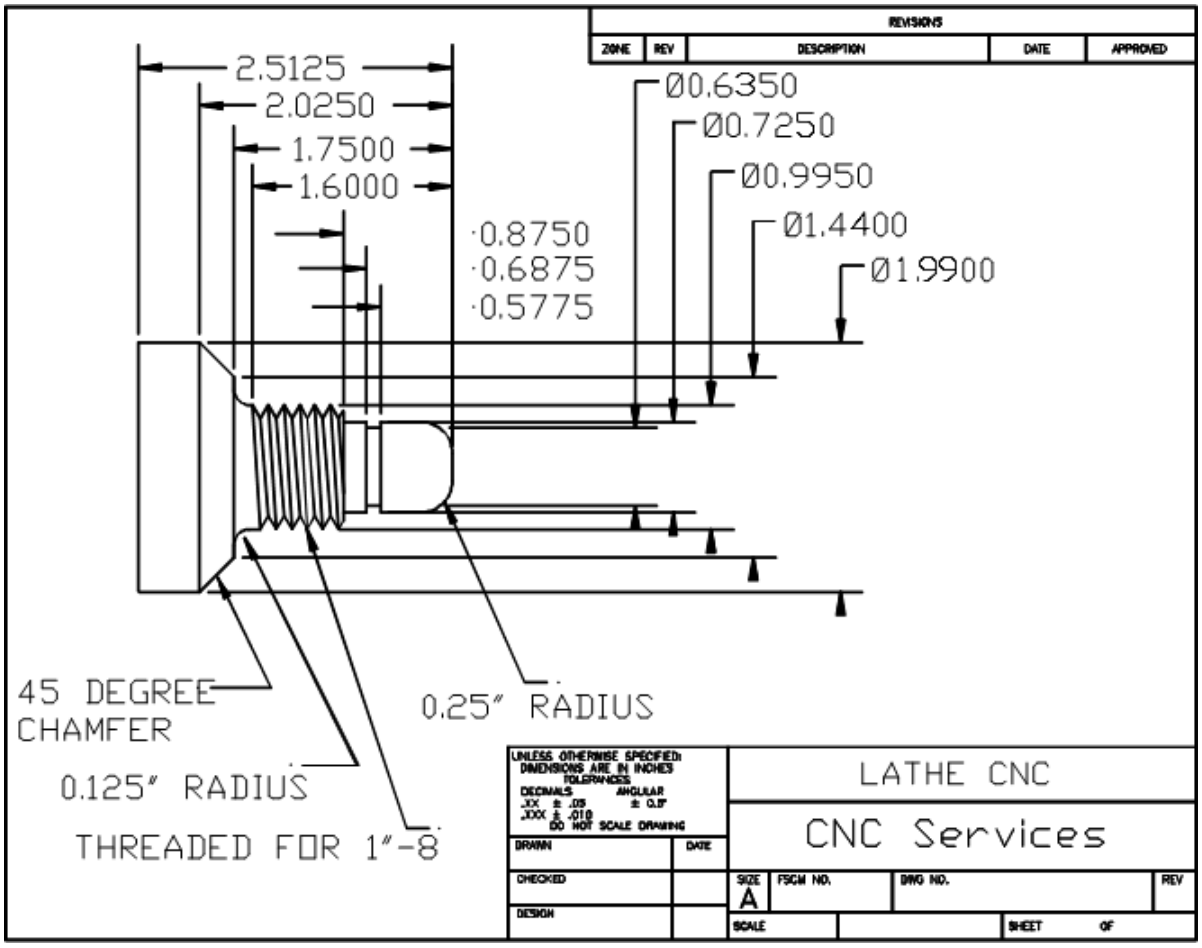


Figure 7.20: Part to be Programmed

Beginning from the Control Main Screen, the following series of keystrokes will describe the step-by-step process of programming the part shown in Figure 7.20.

7.19.1 Create A New Part Program

PRESS	ACTION	COMMENTS
F5	CAM	CAM Selection menu.
F1	ICN	Start Lathe Intercon interface.
F1	File	Opens the File Menu.
F1	New	Create a new program. Enter a name for the file.
F10	Accept	Accept the file name. Fill in the dialog box exactly as shown in Figure 7.21.
F10	Accept	Creates a new part file using the data entered.

Draft: January 11, 2024

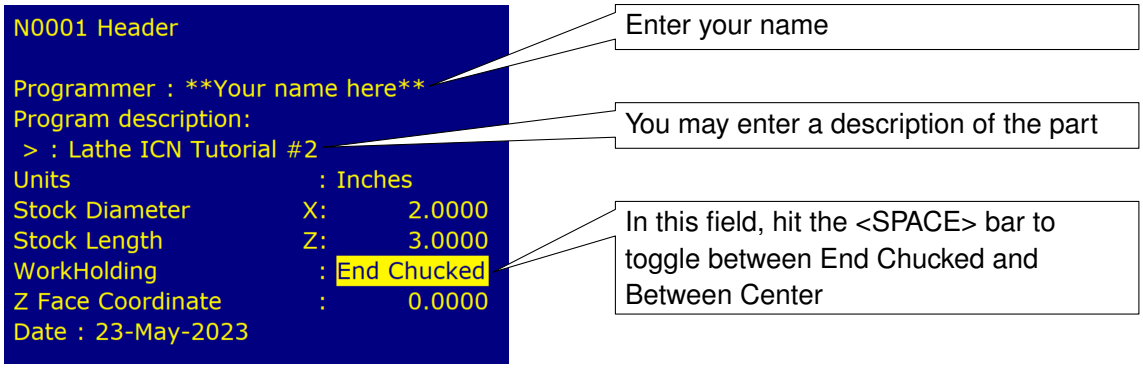


Figure 7.21: New Part Dialog Box

7.19.2 Insert the First Cycle

PRESS	ACTION	COMMENTS
F7	Turning	Creates a repetitive cycle used to cut an outside or inside diameter to a specified dimension within a specified Z-range. Fill in the Edit Operations side of the screen, as shown in Figure 7.22.

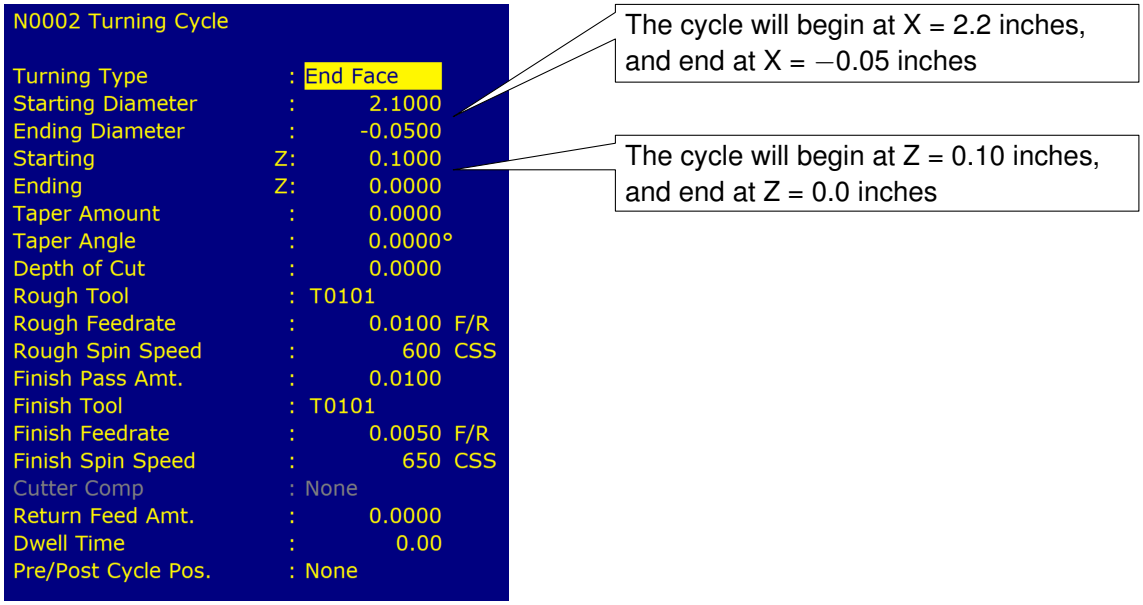


Figure 7.22

PRESS	ACTION	COMMENTS
F2	Tool	Opens the Tool Library. For Tool Offset 1, set the following values: Tool Location (Tool Number) = T01 Nose Radius = .0312 Nose Vector = 3 Spin Dir = CW (See Figure 7.23)
F10	Accept	Sets the Tool Library for Tool Offset #1.
F10	Accept	Keeps selected values for the turning cycle.

Draft: January 11, 2024

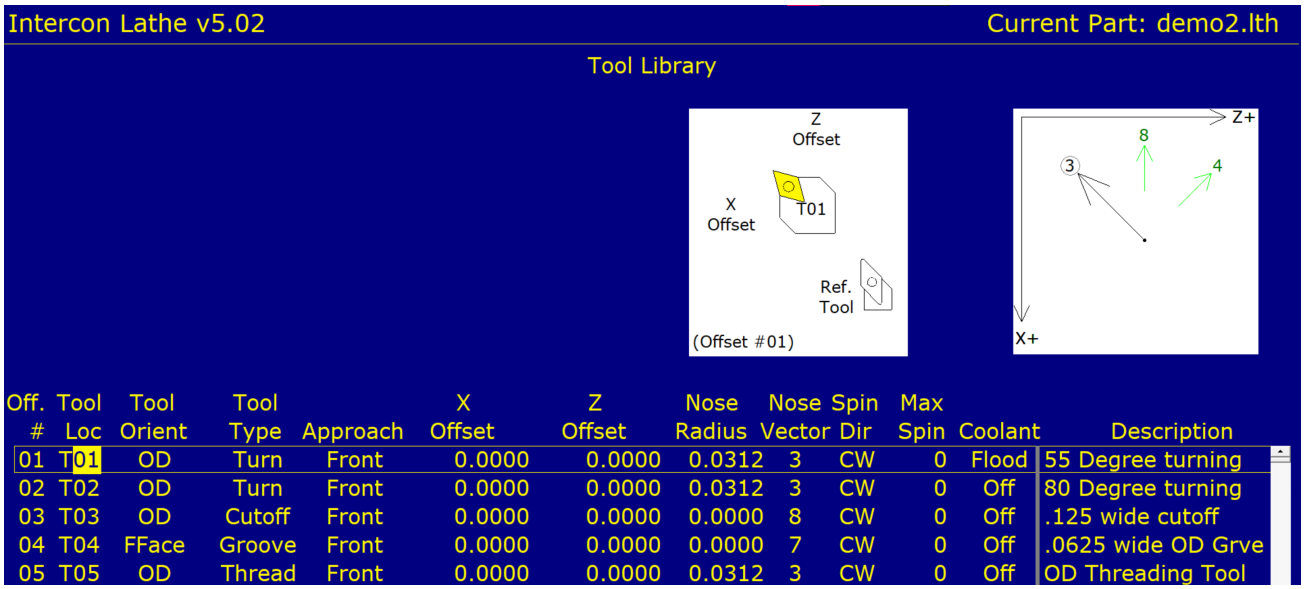
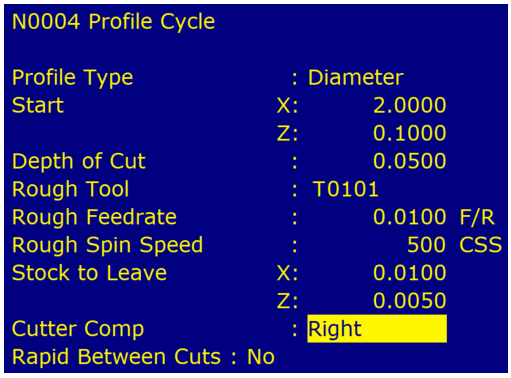


Figure 7.23

7.19.3 Create a Profile

PRESS	ACTION	COMMENTS
F6	Profile	Defines a profile with lines and arcs that will be produced with a cleanout cycle. You can accept the values when at least two operations are present within the profile. Fill in the Edit Operation side exactly as shown in Figure 7.24.

Note: The line number displayed in the Edit operation window is the line number for the end of the profile (which is currently Line 4).



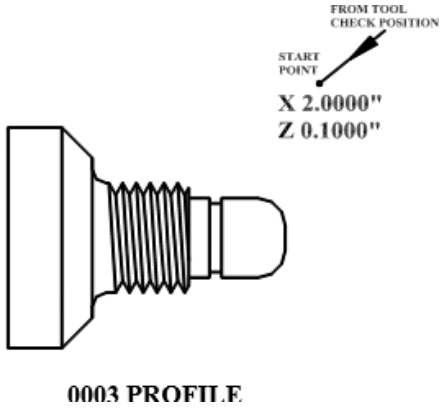
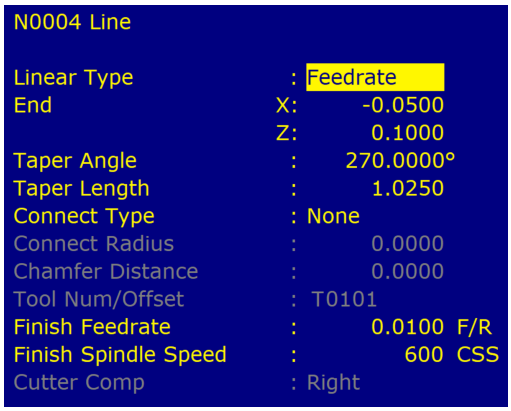


Figure 7.24: Beginning of Profile Cycle – Program Line #0003

PRESS	ACTION	COMMENTS
F10	Accept	Accept the entered values for the Profile.
F1	Line	Inserts a line into your profile. Fill in the Edit Operation portion of the screen exactly as shown in Figure 7.25.

Draft: January 11, 2024



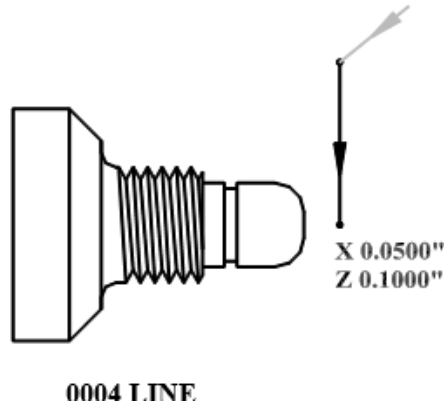
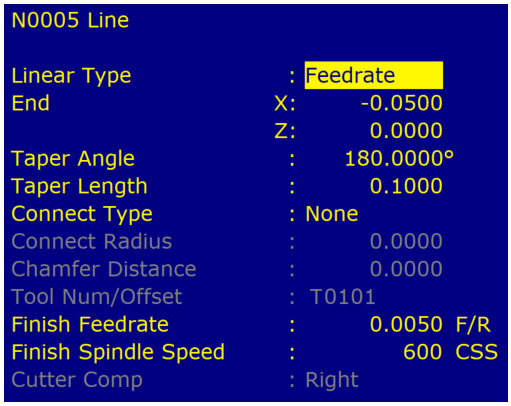


Figure 7.25: First Line Within the Profile Cycle – Program Line #0004

PRESS	ACTION	COMMENTS
F10	Accept	Keep selected values for first line in profile. Automatically inserts another line operation. Fill in the Edit Operation portion of the screen exactly as shown in Figure 7.26.



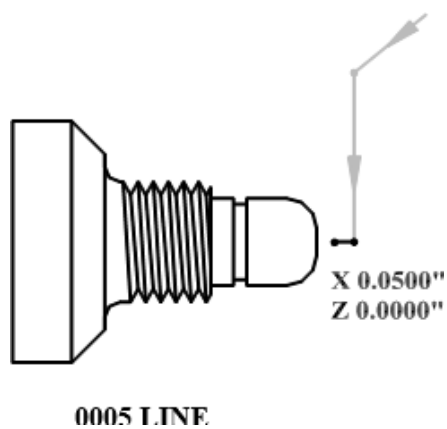


Figure 7.26: Second Line Within the Profile Cycle – Program Line #0005

PRESS	ACTION	COMMENTS
F10	Accept	Keep selected values for Line 2. Automatically inserts another line operation. This line will be 0.3375 inches long and will be cut on an angle of 90 degrees with a Connect Radius of 0.250 inches. Fill in the Edit Operation portion of the screen exactly as shown in Figure 7.27.

Draft: January 11, 2024

```
N0006 Line

Linear Type          : Feedrate
End                X:      0.7250
                   Z:      0.0000
Taper Angle          :    90.0000°
Taper Length         :      0.3875
Connect Type         : Radius
Connect Radius       :      0.2500
Chamfer Distance     :      0.0000
Tool Num/Offset      : T0101
Finish Feedrate      :      0.0050  F/R
Finish Spindle Speed :         600  CSS
Cutter Comp          : Right
```

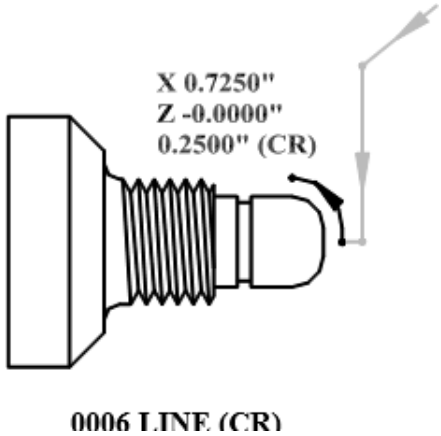


Figure 7.27: Third Line Within the Profile Cycle – Program Line #0006

PRESS	ACTION	COMMENTS
F10	Accept	Save values entered for Line 3. Automatically inserts another line operation. This line will be 0.8750 inches long, cut at an angle of 180 degrees. Fill in the Edit Operation portion of the screen exactly as shown in Figure 7.28.

```
N0007 Line

Linear Type          : Feedrate
End                X:      0.7250
                   Z:     -0.8750
Taper Angle          :   180.0000°
Taper Length         :      0.8750
Connect Type         : None
Connect Radius       :      0.2500
Chamfer Distance     :      0.0000
Tool Num/Offset      : T0101
Finish Feedrate      :      0.0050  F/R
Finish Spindle Speed :         600  CSS
Cutter Comp          : Right
```

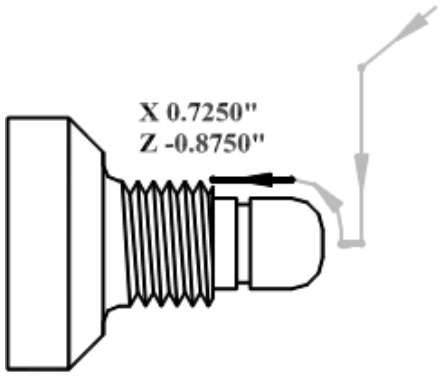


Figure 7.28: Fourth Line Within the Profile Cycle – Program Line #0007

PRESS	ACTION	COMMENTS
F8	Graph	Displays a preview of the part up to this point. Your graph should look like that shown in Figure 7.29.
ESC	Escape	Returns you to the Editing Menu
F10	Accept	Keeps selected values for Line 4. Automatically inserts another line operation. This line will be 0.1350 inches long, cut at an angle of 90 degrees, with a blended chamfer connector 0.1" long. Fill in the Edit Operation portion of the screen exactly as shown in Figure 7.30.

Draft: January 11, 2024

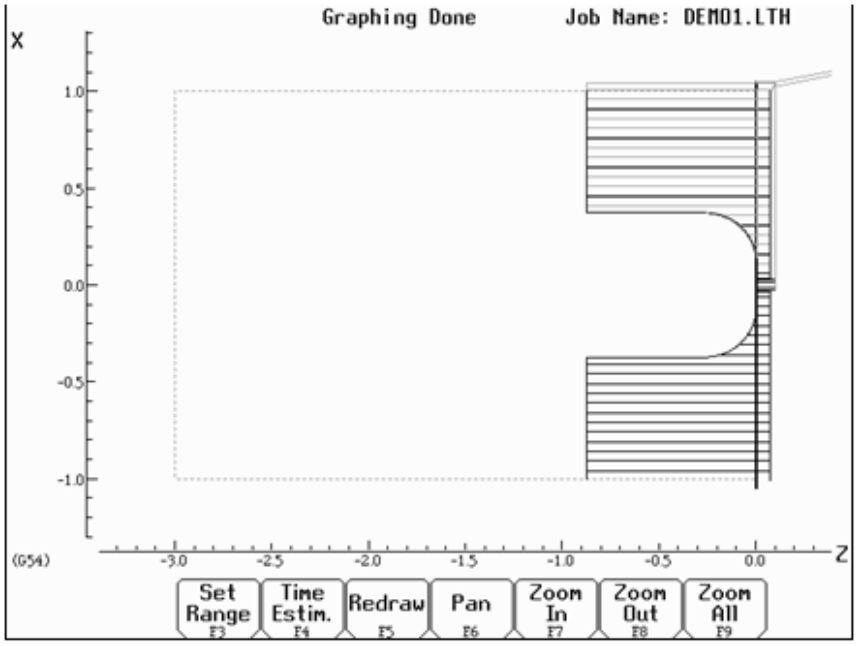


Figure 10 - Graph of Partial Profile

Figure 7.29: Graph of Partial Profile

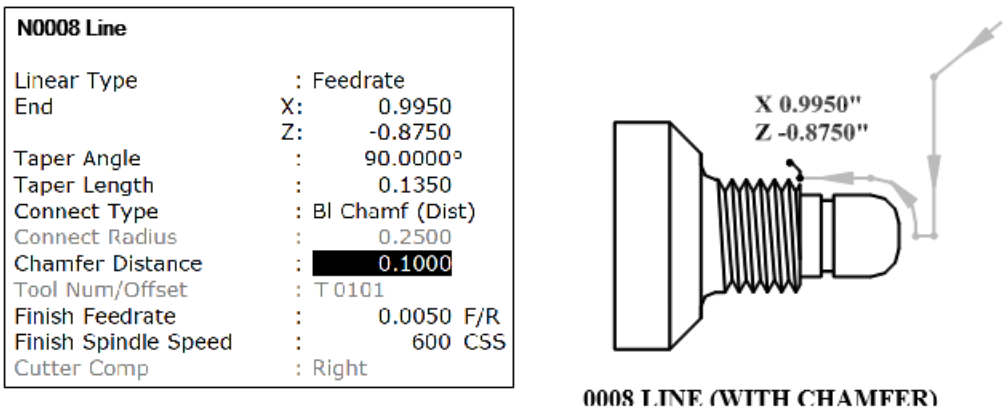


Figure 11 - Fifth Line within the Profile cycle – Program Line #0008

Figure 7.30: Fifth Line Within the Profile Cycle – Profile Line #0008

PRESS	ACTION	COMMENTS
F10	Accept	Keep selected values for Line 5. Automatically inserts another line operation. This line will be 0.8750 inches long and will cut at an angle of 180 degrees with a connect Radius of 0.125 inches. Fill in the Edit Operation portion of the screen exactly as shown in Figure 7.31.

Draft: January 11, 2024

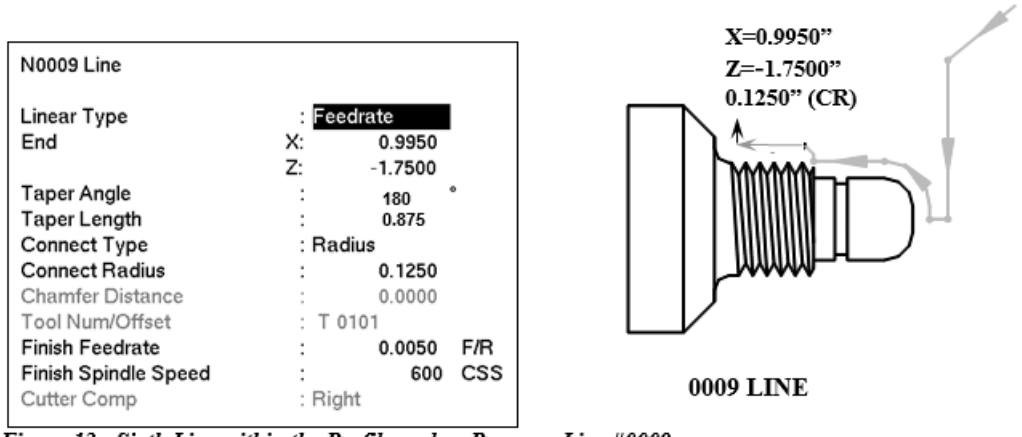


Figure 12 - Sixth Line within the Profile cycle – Program Line #0009

Figure 7.31: Sixth Line Within the Profile Cycle – #0009

PRESS	ACTION	COMMENTS
F10	Accept	Keep selected values for Line 6. Automatically inserts another line operation. This line will be 0.2225 inches long and will cut at an angle of 90 degrees with a connect radius of 0.015 inches at the corner. Fill in the Edit Operation portion of the screen exactly as shown in Figure 7.32.

Draft: January 11, 2024

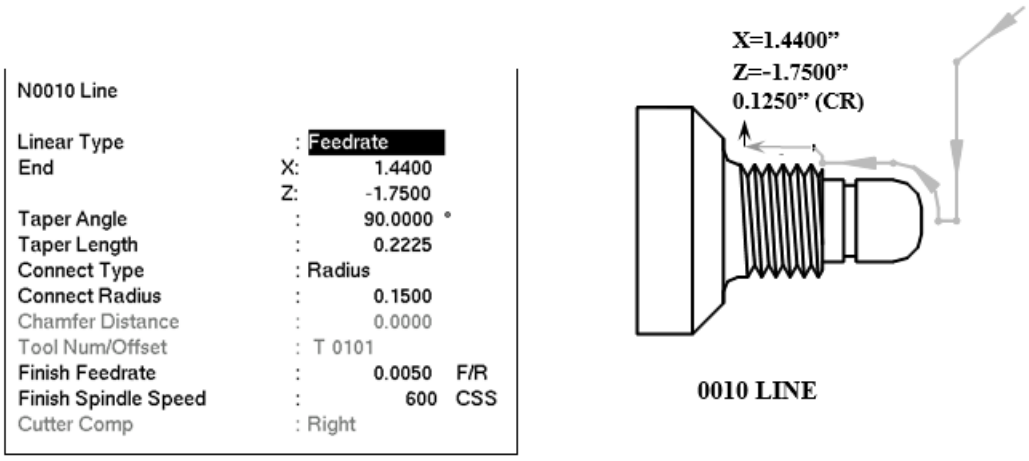


Figure 13 - Seventh Line within the Profile cycle – Program Line #0010

Figure 7.32: Seventh Line Within the Profile Cycle – #0010

PRESS	ACTION	COMMENTS
F10	Accept	Keep selected values for Line 7. Automatically inserts another line operation. This line will be 0.3889 inches long and will cut at an angle of 135 degrees, with a connect radius of 0.015 inches at the corner. Fill in the Edit Operation portion of the screen exactly as shown in Figure 7.33.

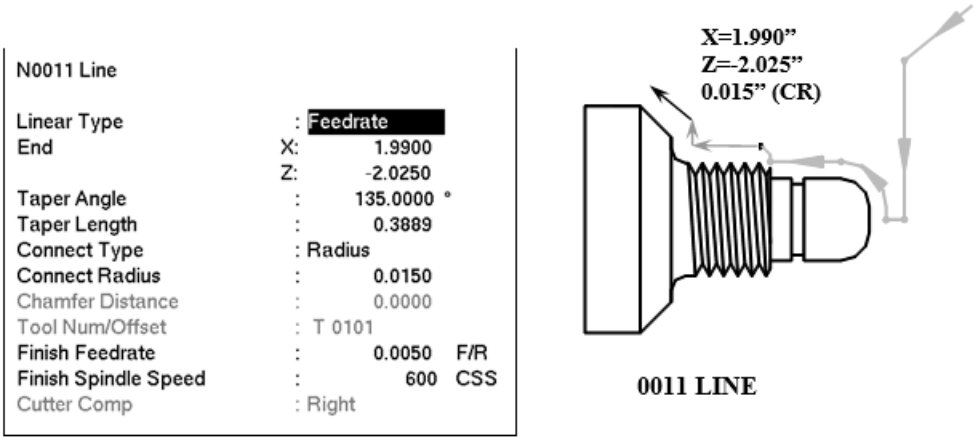


Figure 14 - Eighth Line within the Profile cycle – Program Line #0011

Figure 7.33: Eighth Line Within the Profile Cycle – #0011

PRESS	ACTION	COMMENTS
F10	Accept	Keep selected values for Line 8. Automatically inserts another line operation. This line will be 0.625 inches long and will be cut at an angle of 180 degrees. Fill in the Edit Operation portion of the screen exactly as shown in Figure 7.34.

Draft: January 11, 2024

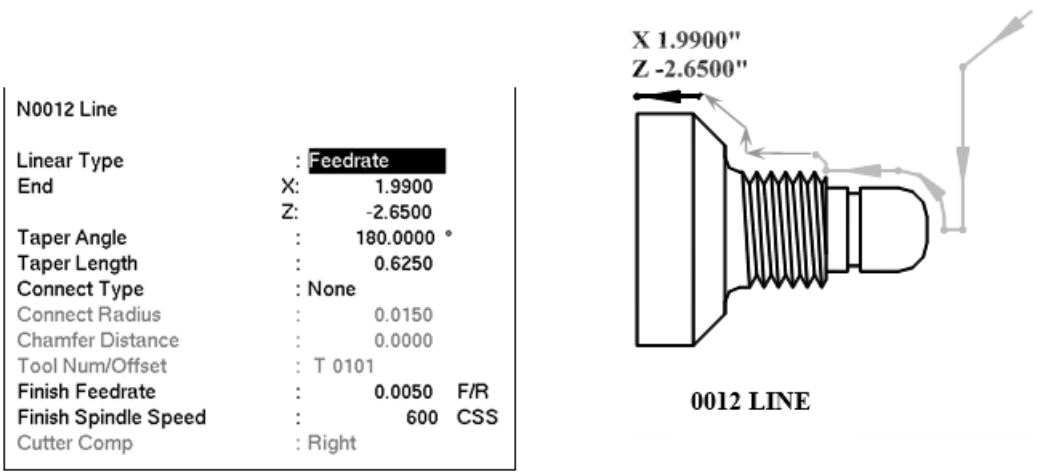


Figure 15 - Ninth Line Within the Profile cycle – Program Line #0120

Figure 7.34: Ninth Line within the Profile Cycle – #0120

PRESS	ACTION	COMMENTS
F8	Graph	Displays a preview of the part up to this point. The profile to this point should look like that shown in Figure 7.35.
ESC	Escape	Returns you to the Editing Menu

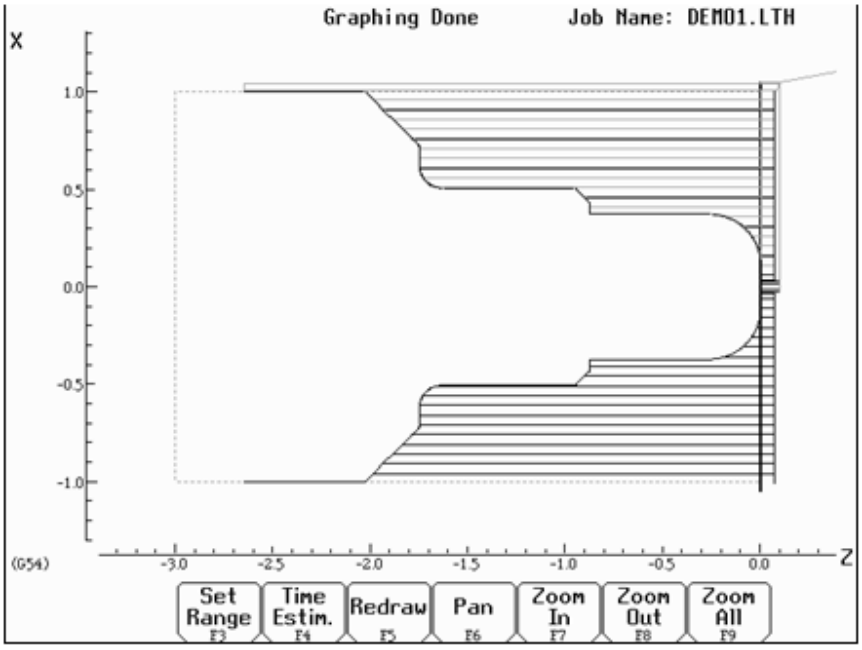


Figure 16 - Partial Graph of Profile Through Program Line #0120

Figure 7.35: Partial Graph of Profile Through Program Line #0120

PRESS	ACTION	COMMENTS
F10	Accept	Keep selected values for Line 9. Automatically inserts another line operation.
Esc	Escape/Cancel	Cancel tenth linear operation and return to profile edit menu.
F3	Finish	Inserts a finishing pass to remove any excess material left from the Rough Pass, and leave a smooth finish. Fill in the Edit Operation portion of the screen exactly as shown in Figure 7.36.

Note: If the depth of cut for X and Z are 0 or equal to the Depth of Cut in line #0030 (X=0.01inches, and Z=0.005 inches), the finish pass will be cut in one pass.

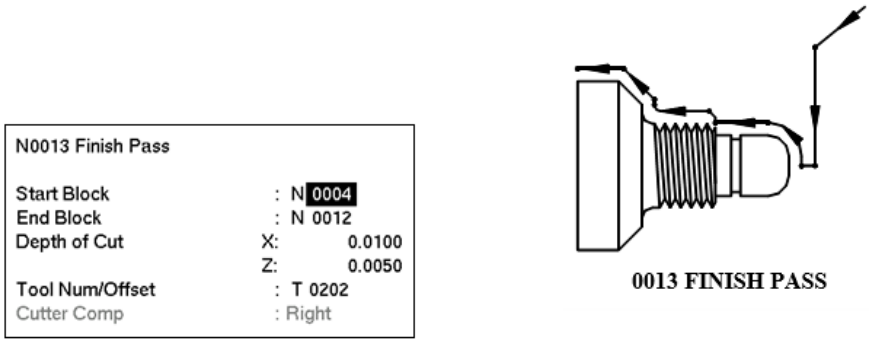


Figure 17 - Finish Pass Within the Profile cycle – Program Line #0013.

Figure 7.36: Finish Pass Within the Profile Cycle – Program Line #0013

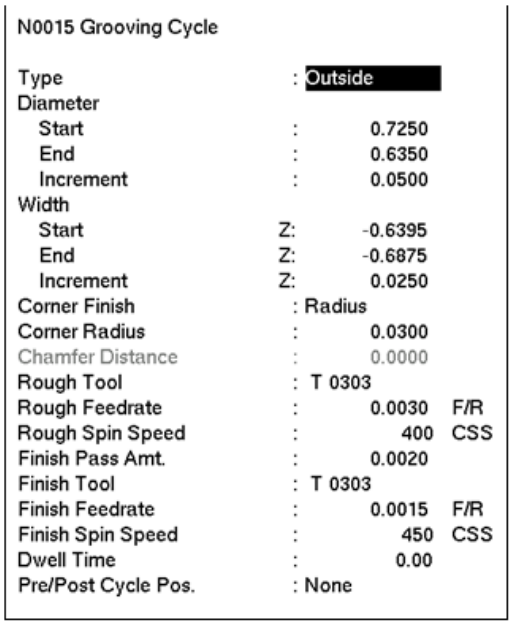
PRESS	ACTION	COMMENTS
F2	Tool	Set the nose radius for Tool 2 = .0150 and the Nose Vector for Tool 2 = 3. Set the Spin Dir=CW, using the <space> bar to toggle through the choices available.
F10	Accept	Sets the Tool Library for Tool #2.

Draft: January 11, 2024

F10	Accept	Accepts Finish Pass values.
Esc	Escape	Exits the profile edit menu.
↓	Down Arrow	Cursor down so the next operation will be inserted after the end of profile line.

7.19.4 Insert a Groove

PRESS	ACTION	COMMENTS
F3	Insert	Insert a new operation after the end of the profile.
F8	Groove	Creates an outside groove with a depth increment (X) of 0.05 inches and a width increment (Z) of 0.025 inches, ending in a corner radius of 0.030 inches. Fill in the Edit Operation portion of the screen exactly as shown in Figure 7.37.



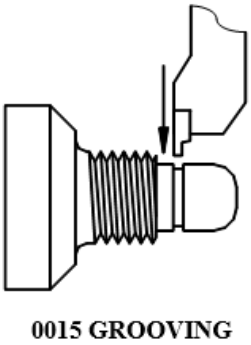


Figure 18 - Grooving Operation – Program Line #0015.

Figure 7.37: Grooving Operation – Program Line #0015

PRESS	ACTION	COMMENTS
F2	Tool	Set the nose radius for Tool 3 = .0070 and the Nose Vector for Tool 3 = 8. Set the Spin Dir=CW, using the <space> bar to toggle through the choices available. These same values can be set now for Tools 4 & 5, but be sure the cursor is back in the Tool 3 row before pressing **F10**!
F10	Accept	Sets the Tool Library for Tools #3, #4, & #5.
F8	Graph	Displays a preview of the part up to this point. The part graph should now look as shown in Figure 7.38.

Draft: January 11, 2024

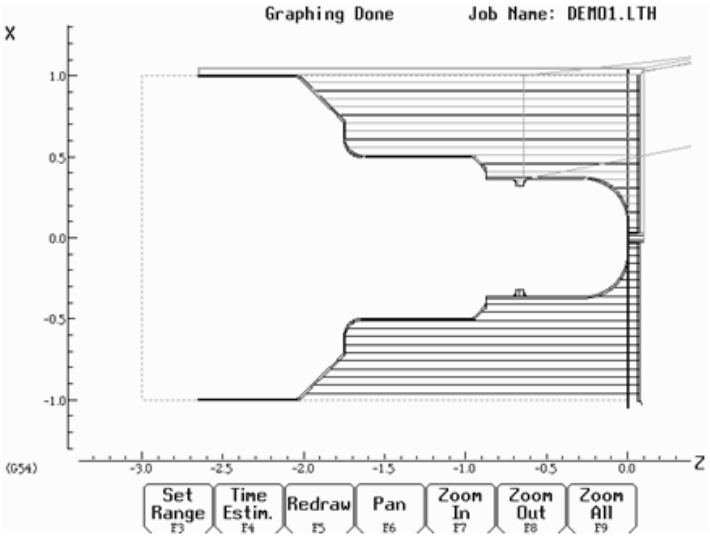


Figure 19 - Graph of Grooving Operation – Program Line #0150

Figure 7.38: Graph of Grooving Operation – #0150

PRESS	ACTION	COMMENTS
ESC	Escape	Returns to the Editing Menu.
F10	Accept	Accepts Grooving cycle.

7.19.5 Add Threads

PRESS	ACTION	COMMENTS
F5	Thread	Places an external thread on the part with a compound angle of 60 degrees and 8 threads-per-inch with a thread lead of 0.125 inches. Fill in the Edit Operation section of the screen as shown in Figure 7.39.

Draft: January 11, 2024

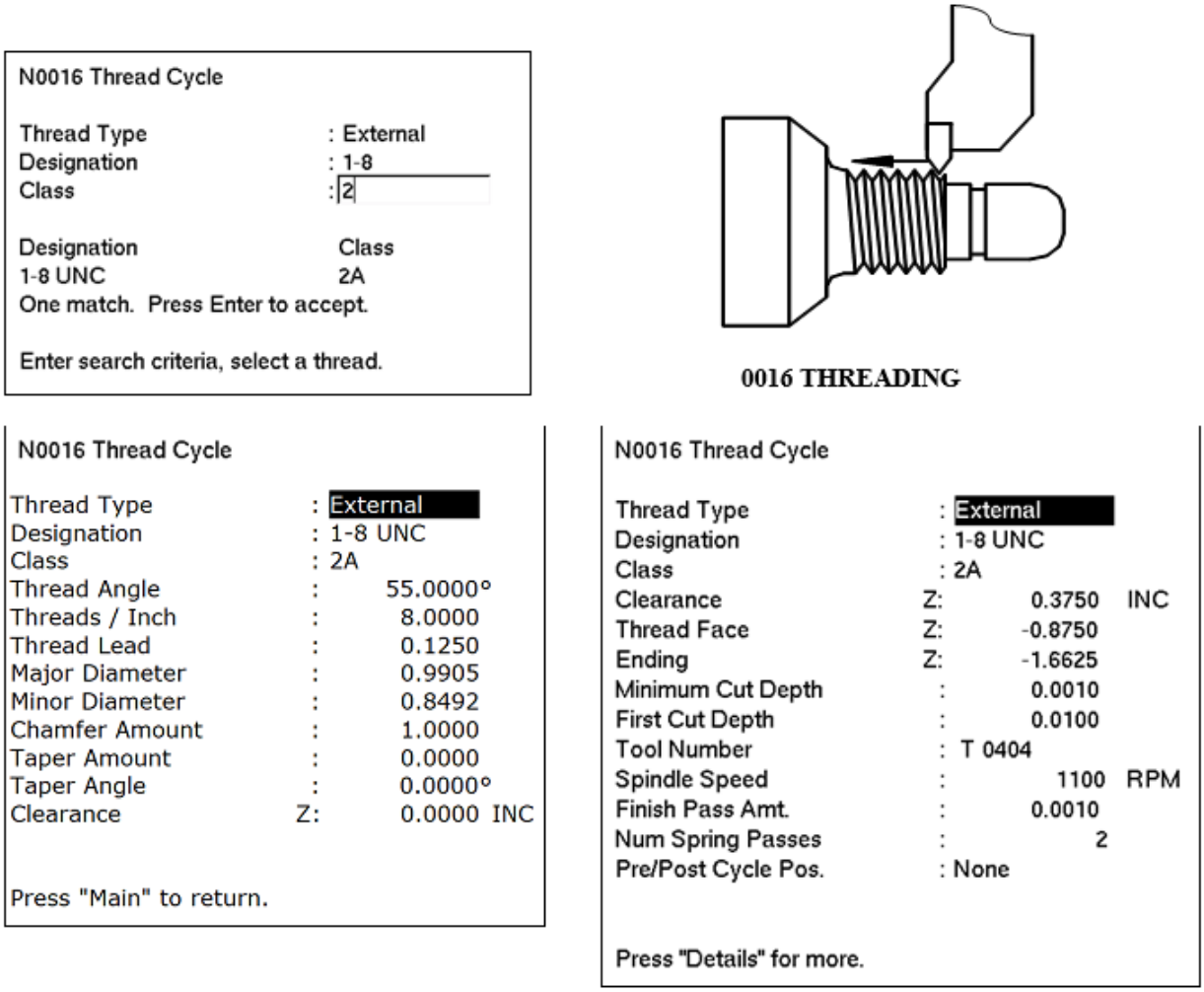


Figure 7.39

PRESS	ACTION	COMMENTS
F10	Accept	Accepts values for the threading cycle.

7.19.6 Cut The Part From The Stock

PRESS	ACTION	COMMENTS
F9	Cutoff	Cuts off the part with a cutoff tool. Fill in the Edit Operation section of the screen as shown in Figure 7.40.

Draft: January 11, 2024

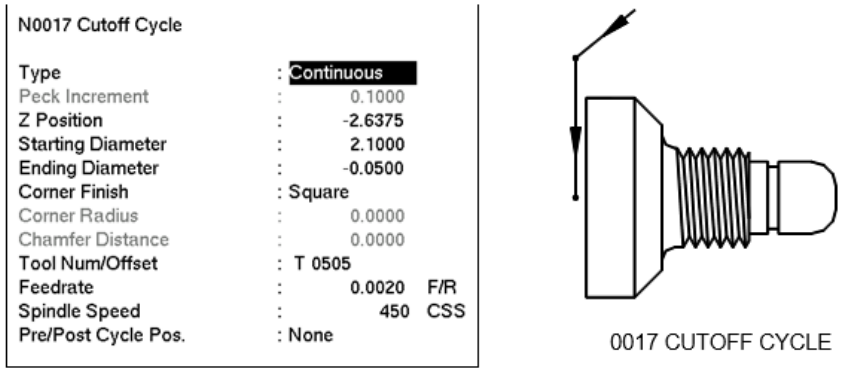


Figure 21 - Cutoff Cycle Removes the Machined Part from the Stock – Program Line #0170.

Figure 7.40: Cutoff Cycle Removes the Machine Part from the Stock – #0170

PRESS	ACTION	COMMENTS
F10	Accept	Accepts values for the cutoff cycle.

7.19.7 Save and Post the Program

PRESS	ACTION	COMMENTS
ESC	Cancel	Returns you to Intercon's main menu.
F8	Graph	Graphs the part one final time to be sure all steps were completed correctly. The final graph should be as shown in Figure 7.41.
ESC	Cancel	Returns you to the Intercon's main menu.
F10	Post	Saves and posts job to control, creating G-codes for the program.

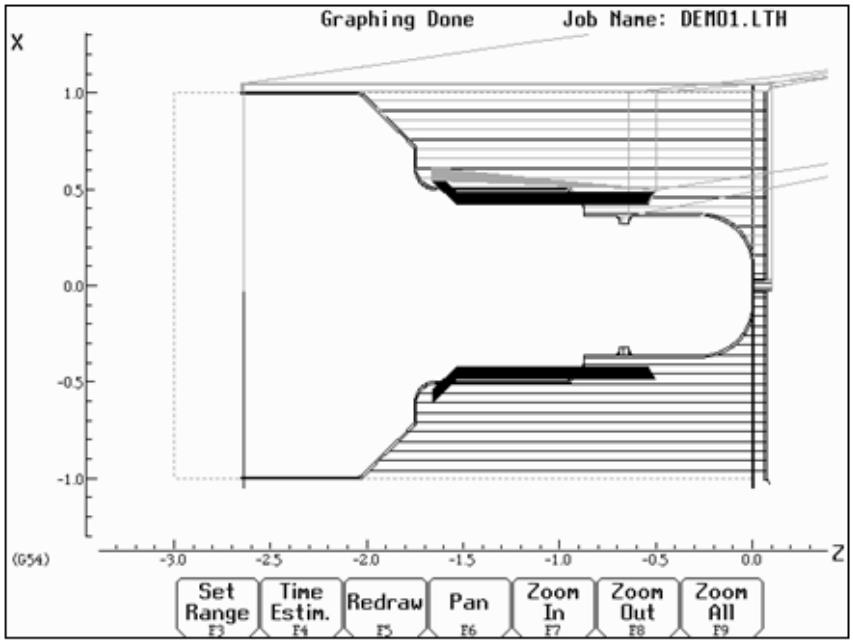


Figure 22 - Completed part

Figure 7.41: Completed part

Draft: January 11, 2024

8 CNC Program Codes

8.1 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. One shots are Intercon canned cycles, meant to allow Operators to more easily insert commonly-used Intercon cycles with just the press of a button.

8.2 Miscellaneous CNC Program Symbols

8.2.1 E, F – Feed Rate or Thread Lead

In threading mode (G32, G76 and G92), E and F can specify thread lead (in units/rev). In other modes, only F can be used to specify feed rate. Feed rate is either units/rev or units/min, depending on G98/G99 mode. The feed rate override knob can be used to modify the programmed feed rate. The default feed rate is 3.0 units/minute.

Example:

```
G01 X1.0 Z-2 F0.1 ; linear cut at X1 to Z-2 at 0.1 units/rev
```

8.2.2 L – Subroutines

L-commands are subroutine identifiers that can be used to alter various G- and M-codes.

Note: The L-command has some unique functions with Acorn/AcornSix (openloop) versus Allin1DC/Hickory/Oak/etc. (closedloop). Acorn-specific examples will be marked.

Some general examples include:

Feed Rates: G28, G30, G30 P3, G30 P4, G53, M91, and M92 use L-commands to override rapid move rates with user-specified rates. Typically, F is used 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.

Note: The L-value does not also override the feed rate for other moves. Two cases where this occurs are listed 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
```

Case 2:

```
G1 X1 F100   ; Moves X at 100 units/minute
G1 Y2        ; Moves Y at 100 units/minute
```

Draft: January 11, 2024

```
G53 Z-5 F20 ; Moves Z at Max Feed units/minute.  The F20 does not effect G53 and is
              ignored as it defaults to the 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
```

Repeat Values: G65 and M98 use L-commands to specify a repeat value.

Example:

```
M98 "program.cnc" Lrrrr ; Where rrrr is the repeat value
```

Suppress “new line” characters: M122 and M123 use L1 to suppress the “new line” characters that are outputted after recording a value/comment using these commands.

Example: M-function and sample output:

```
M122      ; comment ; X1.2345 Z-0.5678 ; comment
M122/X L1           ; X-1.5000
M122/X              ; X-1.5000 X-2.0000
```

Protected move probing functions: M115 and M116 use L1 to prevent an error if the probe does not detect a surface.

Example:

```
M116/X P1 F100 L1 ; Move X plus at 100 units/minute until PLC bit INP1 is closed, do not
                    error if probe does not detect a surface.
```

Moves the requested axis by L (which specifies an encoder count position or quantity and is subject to the current G90/G91 mode (absolute/incremental)): M128

Example:

```
G91 M128/X L-5000 ; Move the X axis incrementally by -5000 counts
```

Prevent further asynchronous G-code execution (until a fault occurs): M130

Example:

```
M130 "mycommand.bat" L1 ; Runs the batch file mycommand.bat and waits until it
                          finishes execution or a fault occurs
```

Acorn/AcornSix-specific example:

```
M91/X L1 ; Start homing process on the X axis and stop before beginning search for
           index pulse
M92/X L1 ; Start homing process on the X axis and stop before beginning search for
           index pulse
```

In this case, the L-command tells CNC12 to seek the switch, clear the switch, and ignore the search for an index pulse (openloop systems do not have index pulse feedback).

Draft: January 11, 2024

8.2.3 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 G56 M26/Z
N2 G00 X0 Z0
```

8.2.4 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.

Draft: January 11, 2024

Example:

```
O1521
N1 G56 M26/Z
N2 G00 X0 Z0
```

8.2.5 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
```

8.2.6 Q – Parameter

Q is used as a depth parameter in canned drilling cycles.

Example:

```
G76 X.75 Z-1.5 P.1 Q.02 F.125 ; Q Sets depth of first cut at .02"
```

8.2.7 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 P5 R.0625                ; set nose radius of tool 5 = 0.0625
G90 X1.0 Z-2.0 R.25 F.0115 ; tapered cut, from 0.5" diameter to 1.0" diameter
```

8.2.8 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. In G97 mode (default), S specifies spindle speed in RPM. In G96 mode, S specifies surface speed in feet/min or meters/min.

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.

8.2.9 T – Select Tool Number and Offset Number

The T-command takes the form of **Tttoo**, where **tt** selects the Tool Number and **oo** activates the Tool Offset.

If a T-command selects a different **tt** number from the previous T-command (or if this is the very first T-command in a CNC program), then this will cause a tool change operation to be invoked. If a T-command specifies the same **tt** as the previous T-command, but with a different **oo**, then no tool change will take place and only the specified **oo** Tool Offset will be activated.

If **oo** is non-zero, then this activates the tool offset data values from row oo of the Tool Geometry Offset Library. However, if **oo** is zero (00), then this cancels tool offsets.

Example:

```
T0100 ; Tool change operation to Tool #01, cancel offsets
T0101 ; No tool change, but activate offsets #01
T0201 ; Tool change operation to Tool #02, keep offsets
```

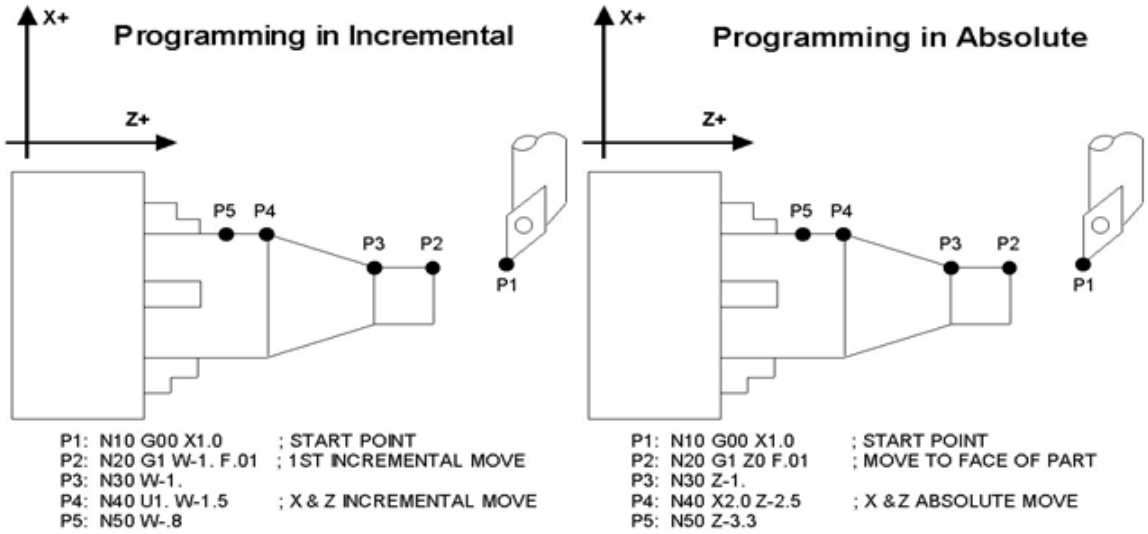
8.2.10 U – Incremental X-axis Move Command

To specify an incremental move on the X-axis, use U in place of X in the command line (see example below).

8.2.11 W – Incremental Z-axis Move Command

To specify an incremental move on the Z-axis, use W in place of Z in the command line (see example below).

Draft: January 11, 2024



8.2.12 : – 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 work coordinate 3
G56
:Rapid to part zero
G00 X0 Z0
:Visible comments will be displayed on screen with the G-codes.
```

8.2.13 ; – 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:

```
G56 ; select work coordinate 3
G00 ; G00 selected with no movement
```

8.2.14 [ ] – Numerical Expression

The left bracket '[' and right bracket ']' are used to delimit a numerical expression. Numerical expressions can contain floating-point numbers or user/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".

Draft: January 11, 2024

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
```

Draft: January 11, 2024

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
```

8.2.15 $ – 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)
```

8.2.16 #, = – User or System Variable Reference

The '#' character is used to reference a macro or user/system variable. For variables that can be written, the '=' sign is used to assign values to them. General purpose user variables are #100-#149 and #29000-#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 (1st set)		R/W
7–9	Macro arguments D–F or 2nd set of I–K		R/W
10	3rd I (G is invalid)		R/W
11	Macro argument H or 3rd J		R/W
12	3rd K (L is invalid)		R/W
13	Macro argument M or 4th I		R/W
14	4th J (N is invalid)		R/W
15	4th K (O is invalid)		R/W
16	5th I (P is invalid)		R/W
17–18	Macro argument Q–R or 5th J–K		R/W
19–21	Macro arguments S, T, U or 6th set of I–K		R/W
22–24	Macro arguments V, W, X or 7th set of I–K		R/W
25–27	Macro arguments Y, Z or 8th set of I–K		R/W
28–30	9th set of I–K		R/W
31–33	10th 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	Floating point value	R/W
2600, 2601–2618	Active WCS, WCS #1–18 Axis 2 values	Floating point value	R/W
2700, 2701–2718	Active WCS, WCS #1–18 Axis 3 values	Floating point value	R/W
2800, 2801–2818	Active WCS, WCS #1–18 Axis 4 values	Floating point value	R/W
2900, 2901–2918	Active WCS, WCS #1–18 Axis 5 values	Floating point value	R/W
3000, 3001–3018	Active WCS, WCS #1–18 Axis 6 values	Floating point value	R/W
3100, 3101–3118	Active WCS, WCS #1–18 Axis 7 values	Floating point value	R/W
3200, 3201–3218	Active WCS, WCS #1–18 Axis 8 values	Floating point value	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
4002	Constant surface speed mode	96.0 (on) 97.0 (off)	R
4003	Positioning mode	90.0 (abs) or 91.0 (inc)	R
4005	Feedrate mode	98.0 (units per min) or 99.0 (units per rev)	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)	Floating point value	R
4117	CSS spindle speed limit (specified by G50 S__)	Floating point value	R
4118	CSS spindle speed (specified by G96 S__)	Floating point value	R
4119	Spindle Speed (S)	Floating point value	R

Draft: January 11, 2024

Index	Description	Returns	R/W
4120	Tool Number (“nn” in “Tnnoo”)	Floating point value	R
4121	Current offset (“oo” in “Tnnoo”)		R
4201	Job processing state	0 = normal, 1 = graph	R
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
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.)	Floating point value	R
9000–9999	Parameter values 0–999	See Machine Parameters	R
10000	Lathe: Tool X offset amount, current offset	Floating point value	R/W
10001–10099	Lathe: Tool X offset amount, offsets 01–99	Floating point value	R/W
11000	Lathe: Tool Z offset amount, current offset	Floating point value	R/W
11001–11099	Lathe: Tool Z offset amount, offsets 01–99	Floating point value	R/W
12000	Lathe: Tool nose radius, current offset	Floating point value	R/W
12001–12099	Lathe: Tool nose radius, offsets 01–99	Floating point value	R/W
13000	Lathe: Tool nose vector, current offset	1–9	R/W
13001–13099	Lathe: Tool nose vector, offsets 01–99	1–9	R/W
14000	Lathe: Tool coolant, current tool	7, 8, 9	R/W
14001–14099	Lathe: Tool coolant, offsets 01–99	7, 8, 9	R/W
15000	Lathe: Tool spindle direction, current offset	3, 4, 5	R/W
15001–15099	Lathe: Tool spindle direction, offsets 01–99	3, 4, 5	R/W
16000	Lathe: Tool location, current offset	Floating point value	R/W
16001–16099	Lathe: Tool location, offsets 01–99	Floating point value	R/W
17000	Lathe: X wear adjustment, current offset	Floating point value	R/W
17001–17099	Lathe X wear adjustment, offsets 01–99	Floating point value	R/W
18000	Lathe: Z wear adjustment, current offset	Floating point value	R/W
18001–18099	Lathe: Z wear adjustment, offsets 01–99	Floating point value	R/W
19000	Lathe: nose radius wear adjustment, current offset	Floating point value	R/W
19001–19099	Lathe: nose radius wear adjustment, offsets 01–99	Floating point value	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
21201–21208	plus_limit for axes 1–8		R
21301–21308	minus_home for axes 1–8		R

Draft: January 11, 2024

Index	Description	Returns	R/W
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
25008	screen_blank_time		R

Draft: January 11, 2024

Index	Description	Returns	R/W
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/M116/M125/M126 probe trip	1 = tripped, 0 = not tripped	R
25023	Drive type	0 = Standard 1 = Legacy DC 2 = Legacy AC	R
25024	Number of packets that have been resent	Floating point value	R
25025	Number of generic communication errors	Floating point value	R
25026	Number of packets that were out of order	Floating point value	R
25027	Number of NACK that were sent	Floating point value	R
25028	Number of NACK that were received	Floating point value	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

Draft: January 11, 2024

Index	Description	Returns	R/W
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
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

Draft: January 11, 2024

* 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 Z axis current position
G90 X[#5041+1+7/32]  ; move the Z axis 1 7/32 units (1.2188) incrementally
#2501 = #5021         ; set WCS#1 Z value to the current Z position
#2503 =[#2503+1/8]    ; add 1/8 units (.125) to the WCS#3 Z value

; Subroutine parameter and local variable access.
G1 Z#A X#B 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.)
```

8.3 Introduction to Centroid CNC Macros

8.3.1 What is a Macro?

A macro is 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 as a term is being used loosely 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 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 Operator's Manuals, but can also be customized with the use of macro files.

8.3.2 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/

8.3.3 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 that they can be user modified and/or 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, direction, and order of homing. Most Acorn users will not have to edit the Home macro, as the Wizard does a good job of creating home macros (aka home programs). However, there are cases where it is nice to add some customization to the home program to make the workflow with a machine tool better.

The pre-assigned Macros for the VCP Aux keys and MPG Macro buttons are themselves commented with examples. Once the Acorn is up, running, and homed, pressing the VCP Aux 8 key (which will run the M55 macro), the instructions on how to edit M55 will appear on the screen (same with the Wireless MPG Macro 1, 2, 3, and 4 buttons). The M-code M55 calls and runs the commands contained in the text file "`mfunc55.mac`. Opening, studying, and modifying these existing macros is a great place to start.

The included macros are located in these directories: "cncm", "cnct", "cncm/system", and "cnct/system".

8.3.4 Edit Your First Macro!

Below is a typical Wizard-generated Acorn Home Program for a three-axis milling machine.

Draft: January 11, 2024

```
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
```

Say we want to set the Z-axis Home position .050" away from the Z-home switch automatically, rather than right after the switch clears:

Open "`cncm.hom`" with Notepad++ and insert the highlighted line 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) after clearing the Z-home switch, the machine will back away from the Z-home switch an additional .050" before setting the Z-home position at its current location. See the Operator's manual G- and M-code chapters for descriptions of M26, M91/M2, and G91.

Draft: January 11, 2024

8.3.5 Edit Your Second Macro!

Let's 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. Let's edit M55 so that it will rapid move the machine to WCS X=0, Y=0. Open `mfunc55.mac` with Notepad++.

Note: A semicolon ";" tells CNC12 that the line is a comment. Remove the semicolon for the command to be run and add a semicolon to the beginning of a line for the command 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 semicolon 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:

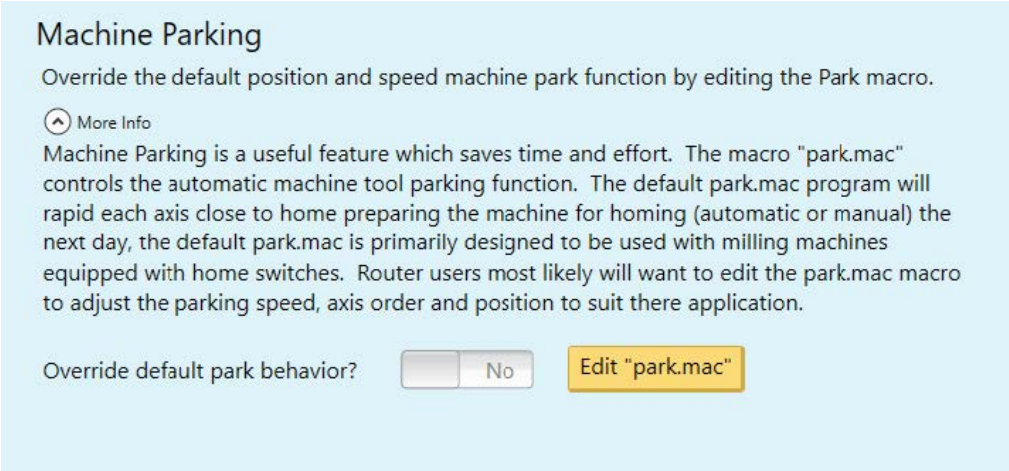
```
; M225 #100 "This is an example macro named mfunc55.mac and can be found in ..\cncm or
  ..\cnct.............etc.
;
N100    ; Insert your code between N100 and N1000
G0X0Y0  ; G0 = Rapid Move to position X=0, Y=0 in the current Work Coordinate System
N1000   ; End of Macro
```

Using Notepad++, save the changes to `mfunc55.mac` (File, 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.

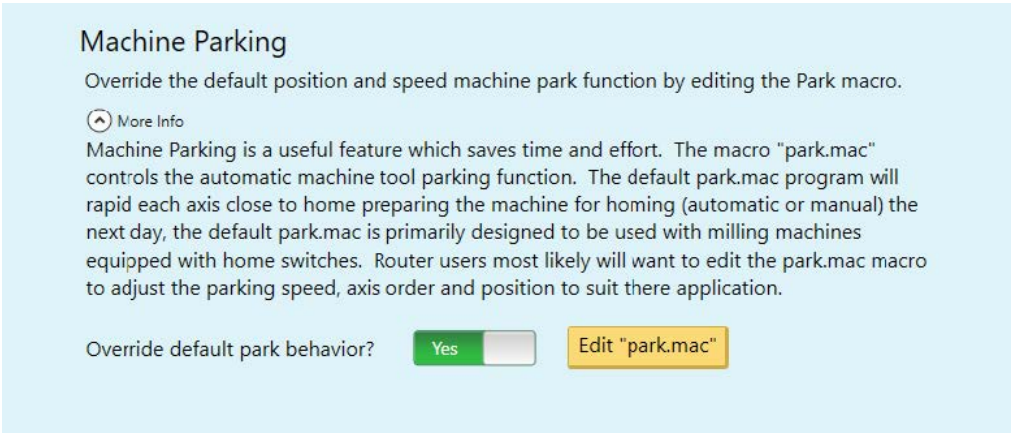
Draft: January 11, 2024

8.3.6 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 minimal movement. CNC12 has a default Parking functionality built into it, so if the Acorn Wizard is set to “No” as seen below, then the default CNC12 park action is used.



The default action is fine for most average milling machines, but we 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 to the home switches. A common custom `Park.mac` will move the axis close to the home switches, but at a user-specified feed rate.



Choose “Yes” to override the default park behavior and edit “`park.mac`”.

The stock `Park.mac` macro matches the one below, with three examples that are commented out with semicolons.

```
; --------------------------------------
; 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.
```

Draft: January 11, 2024

```
; 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 semicolon in front of the M225 line to comment it out. Then, uncomment one of the examples provided and edit the feed rate and direction to fit your requirements. See the 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)
```

Draft: January 11, 2024

```
; ---------------------------------------
```

Using Notepad++, save the changes to `park.mac` (File, Save). Now in CNC12, the Park feature found in the Shut Down menu will run this custom macro and, in the example above, move Z slowly .25" from the Z-home. It will also move X and Y simultaneously to 1" away from the machine home position, and then move the rotary axis to the W0 machine position.

See the Operator's Manual G- and M-code chapters for descriptions of G53 and M225. See the FAQ section below for a description of the "L"-command.

Tip: If you built your own CNCPC, edit the Windows file extension default application settings so that Notepad++ will be the default editor for `.mac`, `.hom`, `.cnc`, `.nc`, and `.txt` files.

The Operator's Manuals have a number of sections covering macro programming.

Once you are familiar with 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 contained within the CNC12 Operator's Manuals:

Chapter 8: CNC Program Codes	page 147
Chapter 10: CNC Program M-codes	page 223
G65 – Call Macro	page 203
M98/M99 Call Subprogram	page 231
M100/M101 Wait for PLC bit	page 232
M200/M223, M224, M225, M290 Formatted String	page 240
Section 8.4 Advanced Macro Statements	page 189

Acorn Wizard Canned PLC Functions and M-codes. https://www.centroidcnc.com/centroid_diy/downloads/acorn_documentation/acorn_wizard_input_output_plc_functions.pdf

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.

8.3.7 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

Draft: January 11, 2024

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

8.3.8 Common Usages of Macros

Machine tool homing, machine tool parking, custom probing, rack mount tool changers...the possibilities are endless.

8.3.9 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 commands. Supporting PLC code is required for a macro to toggle an output using the M94 and M95 commands, as seen in the examples below:

8.3.10 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 in a notebook to remember it for later, a User Variable is commonly used this way within 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 power is cycled. The value of a non-volatile variable will be retained after the macro is finished, even 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 user variables 100–149 and 150–159 values in the examples given. User variables are different from system variables (as described below) in that they do not interact with any stock CNC12 function, hence the name "User Variable". Therefore, by definition, variables #'s 100–149 and 150–159 will have no conflict with the standard feature set of CNC12.

8.3.11 Commonly Used Expressions in Macros:

(see the entire list Section 8.2.14)

- Equals `==` (or "`eq`")
- Logical Not `!` (or "`not`")
- Logical and `&&`
- Logical or `||`
- Greater than `>` (or "`gt`")
- Less than `<` (or "`lt`")

8.3.12 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 include: user variables #100–149, non-volatile user variables #150–159, and the Tool Number indicator

Draft: January 11, 2024

(#4120)

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 usage of setting a user variable equal to #4120 is that 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 on in the manual.

Note: The line: “`#150 = #4120`” needs an important “`IF #50001`” to accompany it, and is explained later on in the manual.

Draft: January 11, 2024

8.3.13 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**, and **F3 – Params**) in CNC12, however when we want to reference these parameters in the macro, we use the system variable that represents the parameter instead.

Refer to Section 8.2.16 for a full list of the CNC12 system variables.

8.3.14 How to Interact with an Input

Inputs can be interacted with by using M100/M101, M105/M106, or IF statements. M100/M101 commands simply look for the input, while M105/M106 moves an axis in the negative or positive direction while looking for the input. IF statements will simply look to see if the expression is true or not.

Example 1: Set the 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 the 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!
```

Draft: January 11, 2024

Example 4: Stop and wait for the Operator while displaying a message. In the 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 10.55 for more info on M225)
```

M225 is the "display a formatted string for a period of time" command. In the example above, #100 is a user variable that defines the amount of time that the message will be displayed for. If a M225 command is given a time of zero, it will display the message indefinitely, waiting for the Cycle Start key to be pressed. Otherwise, it will wait for the amount of time specified, and then continue with the macro.

Alternatively, an M00 or M01 can be used if you desire for no message to be displayed and for the macro to simply wait for the Operator to hit Cycle Start. This is 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 that 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 still be used. It is interesting to note that a macro can “look” for a Surface Plate Detector that is also wired into the same input as the TT device, saving on the number of 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 use Input 3 as a Tool Touch-off.

8.3.15 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 the CNC12 software. See Section 11.3 for details on the standard CNC12 parameters. You will notice that there are a large number of built-in functions and configurations within 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 of the Acorn-related parameters for you. 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 that independent movement can be achieved to align the two sides of the gantry to be 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.

8.3.16 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 the 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 be edited more easily by the Operator in the parameters menu in the software (**F1 – Setup**, **F3 – Config**, **F3 – Param**).

The Acorn version of CNC12 makes use of several parameters that are not used by other control boards. A description of these special Acorn parameters is listed below:

Parameters 700 to 799 are allocated as special-use parameters, reserved specifically for the CNC12 user/integrator to

Draft: January 11, 2024

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. The user/integrator can then be sure that P700–799 will not be used by CNC12 or a Centroid Standard PLC program in the future. This ensures that any custom PLC or Macro made today using P700–799 will not conflict with future CNC12 updates. So, if you need to make a custom parameter, use any in the P700 to 799 range.

8.3.17 How to Read a Parameter Value

In reference to the system variable chart within the 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 more easily within the 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 of the time, they can be used directly in the logic.

```
G92 Z[#9710] ; Set WCS Z position to value of Parameter 710
```

Draft: January 11, 2024

8.3.18 How to Write a Parameter Value

Referencing the system variables chart, you are able to read 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. For 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 require the parameter number, and do 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. Here are some examples of 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 Operator's Manual. Below is an example of a parameter that cannot 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
```

8.3.19 Lathe Versus Mill Software

Always reference the correct Operator's Manual (Mill or Lathe) for each version of CNC12 that you are working with. The examples in this guide are written with the Lathe 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 the software, so it is important to reference the correct Operator's Manual for the usage of specific G-codes. For example, the G10 command is different in functionality between the Lathe and Mill 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 (formatted as: 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 a value of 2. If the P number is a number between 1000 and 1999 (formatted as: P1nnn, where "nnn" is the parameter number), then the parameter is written to. So "G10 P1710 R2" would set the value of Parameter 710 to 2.

8.3.20 The Difference Between = and ==

== is used to compare two variables, while = is used to write to a variable (in other words, assign a value to a 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 the expressions, symbols, and letters used, reference Chapter 8.

8.3.21 Using "IF THEN ELSE"

Draft: January 11, 2024

```
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, 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"
```

When IF statements look at inputs, the inputs are not inverted, even if the Operator inverts the input in the software (Alt-I, then pressing Ctrl-Alt-I with an input selected). The IF statement is still looking to see if the input is ON (Green) or OFF (Red), regardless of inversion. Refer to Section 8.4.2 for more information.

Draft: January 11, 2024

8.3.22 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. For 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 is OFF, the macro will perform an M15 command, then end the macro. If the input is ON, the macro will jump to block 500 and perform an M16 command, then 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 the spindle when the macro is called.

8.3.23 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 an output ON and OFF: M94 and M95. In order to use M94 and M95 with a particular output number, that output number must be defined in the PLC program. Defining an output definition can be done one of two ways:

1. Use the Wizard, if available, to assign an output that already has the supporting M94/M95 logic in the stock PLC program.
2. Edit the PLC Program source file to add or modify an output to use the M94/M95 command.

For the following examples using the Acorn Wizard, we assign the output DustFootActivate to Output 3 and 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, press "Write Settings to CNC control Configuration", follow the instructions, and then open M61 in Notepad++.

```
; ---------------------------------------
; 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" and press Cycle Start. Output1 will turn on. Press Cycle Cancel, or type M81 and press Cycle Start, and Output1 will turn off.

Example 2: Adding a delay after Output1 is turned on for four seconds to M61:

```
N100
M94 /61 ; Activate Output1
G4 P4    ; Wait 4 seconds.  Then continue on
N1000
```

Example 3: Activate an 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
```

Draft: January 11, 2024

Example 4: Adding an input for the Operator to choose whether to turn on or keep off the RouterVacuumHoldDown Output during macro operation using M224:

```
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
```

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 the 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
```

Draft: January 11, 2024

```
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 the tool. Putting these M94 and M95 commands into macros is usually preferred, as additional desired logic is accompanied with these commands in most cases (such as M101/M100).

8.3.24 Subprograms M98/M99

Macros can be used to call additional subprograms and can have subprograms embedded into the macro itself. Embedded subprograms are a common technique used for rack-mounted Automatic Tool Change (ATC) macros. Embedded subprograms within the macro are noted by beginning with an "O" (as in Oscar) followed by a 9100 to 9999 number. These particular subprograms are special in that they 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 8.2.4 and Section 10.27.

8.3.25 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 prompt messages to an Operator when the macro is waiting on an input. 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:

Draft: January 11, 2024

```
INP6
"My Custom Message for Input 6"
OUT1
"My Custom Message for Output 1"
MEM2
"My Custom Message for Memory 2"
```

The format for the `cncxmsg.txt` file is: on the first line, state the Input, Output, or Memory. On the next line, insert 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 the output or memory bit).

To use the custom messages created in the `cncxmsg.txt` file, all you have to do is use the M100 or M101 commands 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)"
```

Draft: January 11, 2024

8.3.26 When Skip Graphing, Skip Searching, and Stop Look Ahead are 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 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 has a value of 0 when running a job and a value of 1 when performing a graph preview. The CNC12 system variable #4202 has a value of 0 when CNC12 is running G-codes normally, or a value of 1 or greater when 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 are a non-zero value, then the expression is true and the program will 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 where this logic is located. This is used mainly when you need to ensure a value or action is not processed by the G-code look-ahead prior to when the program itself reaches that point in the macro. CNC12 actively “looks ahead” in the job file and macro to anticipate 16 changes, and 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 manual.

```
; -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, say an issue happened and the Operator had to press the E-Stop between 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. This is due to CNC12's program execution not reaching 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 that 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.

8.3.27 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 user variable, as discussed previously. However, we may not always want to skip updating a variable during a search or graph. Let's look at two examples. The first is rather familiar, we will update the "tool in spindle" for a Rack Mount Tool ATC. In the second example, we will be storing the desired RPM of a sub spindle to a parameter. We chose a parameter instead of a non-volatile user variable for Example 2 so that the PLC can also retrieve this information. Assume for Example 2 that we will be writing different RPM values throughout a program, the value 3500 is just used for example purposes.

Draft: January 11, 2024

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 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. For Example 2, we will want to skip when graphing, however we may want it to update the values if searching. Let's 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, since 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**). When in the Search feature, 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 whenever it is called in the program. This will give us the correct state of the desired RPM (Parameter 702) up to the point 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
```

Draft: January 11, 2024

```
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
```

Draft: January 11, 2024

```
; -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
```

Draft: January 11, 2024

8.3.28 Rack Mount Macro Notes

- The tool touch-off device should be setup with the Acorn Wizard and its functionality tested before running a macro or tool touch-off cycle.
- The tool touch-off input should be defined and the correct input defined in the macro (user variable #104).
- The tool touch-off subtracts from the height of the tool touch-off 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.

8.3.29 Frequently Asked Questions

1.) If an Input is Inverted in the Software, are M100/M101 or IF #5000# Expressions Inverted as Well?

These are not inverted, as they are still looking at the state of the input (ON or OFF, or in the I/O Diagnostic Screen (Alt-I), green or red). For example, in the case that Input 1 is off and not inverted, it 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 that Input 1 is OFF and inverted, it 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 that Input 1 is ON and not inverted, it 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 that Input 1 is ON and inverted, it 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
```

Draft: January 11, 2024

2.) When to Use F or L for Feed Rate?

F is used 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 used with a G53 (rapid move in machine coordinates) will make the G53 move at the user-specified feed rate rather than at 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 100 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 affect 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 20 units/minute
```

G-codes that use the L-command for feed rate are G28, G30, G30 P3, G30 P4, and G53. The L-value 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 machine's Rapid rate.

Note: L is not always used for feed rate. For example, G65 uses L for the repeat value and not for the feed rate value.

3.) Where Does Variable #150 Come From in the ATC Macro?

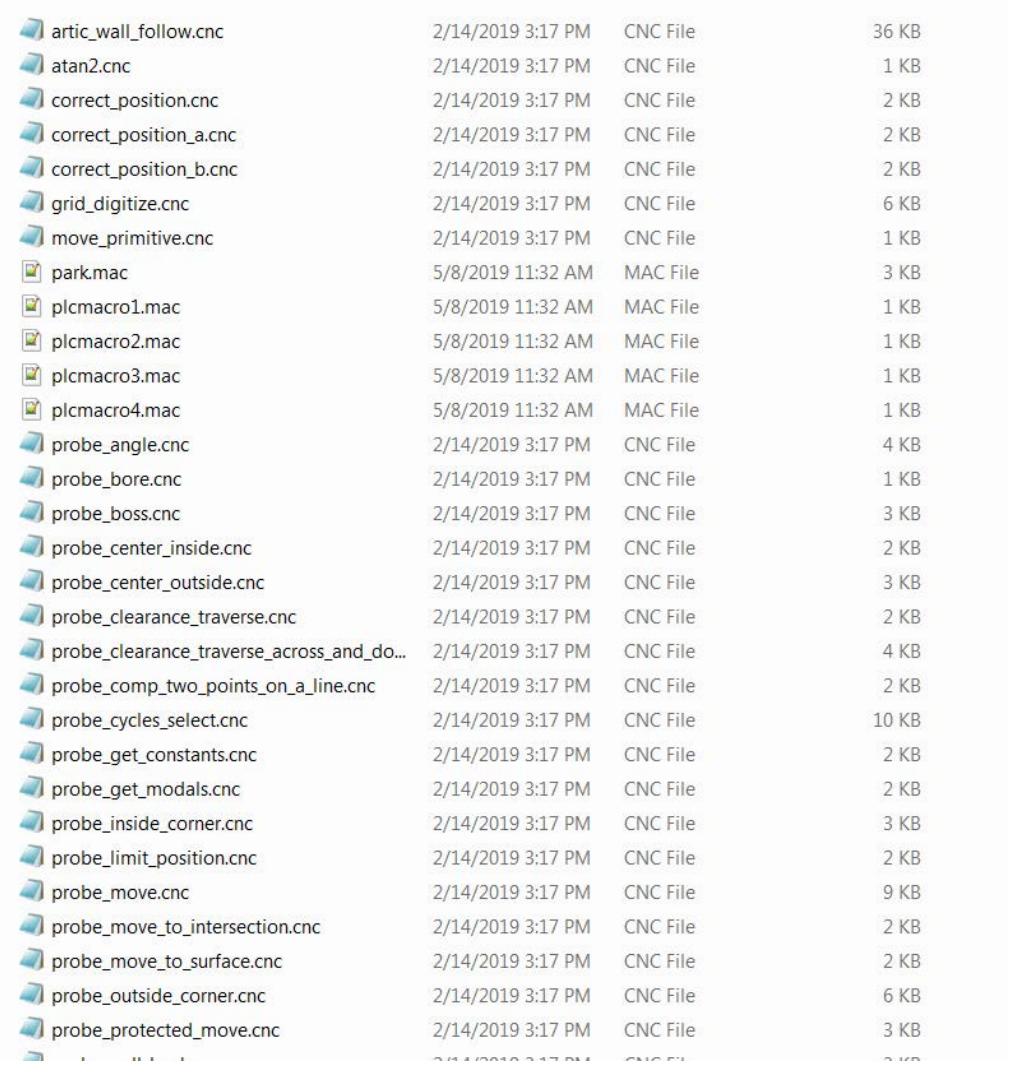
Variable #150 is a non-volatile variable, meaning it retains its value after the macro ends and through power cycles. Variable #150 is needed in tool rack mount ATCs to keep track of which tool is currently in the spindle. When calling a tool change, the macro uses the value of #150 to determine where to put the current tool before trying to pick up the next tool.

We use #150 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 equal to 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 itself 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

Draft: January 11, 2024

We have begun to “breakout” many CNC12 built-in probing cycles into editable macros. Copying and 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 the 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 distances, depending on the current state of G90/G91.

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 and +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.

Draft: January 11, 2024

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, also interpret the sign of the P parameter, but do so differently. The behaviors for positive and negative P values with M105/M106 and M115/M116 are 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 positive limit of travel. We wire it to INP7 on the Acorn board. Therefore, we expect INP7 to be open when we are in the negative half of travel, and closed when we are in the positive 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 negative side of the edge, if we were on the positive side to begin with. If we are already on the negative side (if the sensor is already open), then there will be no movement. The second line will quickly move us (back) to the positive side of the edge, where the sensor is closed.

The third line will start the usual negative homing sequence: slow-jog speed minus until the switch opens; slow jog positively until the switch closes again; and slow movement further positive 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 even then, only for the limit switch that 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. This way, 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, the homing macro cannot know whether to move positively or negatively when seeking the home switch.

You cannot reliably use M115 and M116 in homing sequences, because these codes always attempt to apply software travel limits. They do not recognize that the software limits are not yet valid when the machine home has not been set, and so they will apply invalid restrictions to axis movement.

M105 and M106 do not apply software travel limits, even if the machine has been homed.

Draft: January 11, 2024

8.) How Do I Use Encoder Marker Pulse Relay Output from an Axis Drive for Accurate Day-to-day Homing? (AKA ZRI Homing)

ZRI (Zero Reference Input) homing is a nice feature to implement on an Acorn CNC control system. 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. This way, the centerline 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 (which is often called something different between each drive manufacturer), such as Estun and DMM servos. These drives have an output that closes when the marker pulse is detected (there is one marker pulse-per-revolution on the encoder). The ZRI output is connected to an Acorn input and the CNC12 home macro uses the home switches to "get in the ballpark". It then uses the ZRI to home the machine tool very accurately. Listed below are some common ZRI homing program macros. ZRIs can be wired to individual Acorn inputs or all wired into one input. 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 the ZRIs Wired to the Same Input:

```
M92 /Z L1            ; Move to Z+ home switch, back off until it clears.
M105 /Z P-3 F3       ; Move Z minus at 3 ipm until input 4 closes
M26 /Z               ; Set Z home here
G1 G98 W -.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-3 F3       ; Move X plus at 3ipm until input 4 closes
M26 /X               ; Set X home here
```

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 and adding 2 to Parameter 10. To access Single Block mode, press the button on the VCP beside Cycle Cancel (see Section 2.6) or enter an **F55** command in the Run menu (see Section 3.4). This is a bit-wise parameter. So if, for example, the value is currently set to 1, set it to 3. 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 an M0 function (see Chapter 10), which would cause CNC12 to stop and wait for Cycle Start before continuing to the next line. This is useful while debugging a macro when you only want CNC12 to stop at a single point or a few points in the macro for debugging purposes.

10.) When I Open the Cnct Directories, There Are Missing Macro Files 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 cnctchXX.mac file is not present in the cnct directory. The cnctchXX.mac file is only needed for these M-codes if you wish to have that M-code function differently from the CNC12 default action. For instance, the file cnctch6.mac (which corresponds to M6 Tool Change) is missing from the cnct directory. CNC12 has a built-in Tool Change (M6) macro that is the default action for a tool change. If there is no cnctch6.mac file present in the cnct directory, then CNC12 uses the default tool change action. The default behavior of M6 is described in Section 10.9.

Draft: January 11, 2024

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 the zip file download at the top of the following web 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. These parameters are automatically set based on the CNC setup selections made by the Operator from within the Wizard. That being said, power users that are interested in customization should familiarize themselves 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 11.3.12) has four 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 may seem confusing. What does this mean? Do I have to pick between the two? What if I want both the Macro G-code to be displayed and to step through 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.

For instance, if you want to:

1. "Display M- & G-codes in M-function macros?",then set P11=1
2. "Step through M-function macros in Block Mode?", then set P11=2
3. both "Display M- & G-codes in M-function macros?" AND "Step through M-function macros in Block Mode?", then set P11=3 (this was done by adding the values of the functions (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. You would then convert the value 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, you can round to the nearest integer by first adding 0.5 and then rounding down:

```
#100 = FIX[#5044/360.0 + 0.5] * 360.0
G90 G0 A#100
```

Draft: January 11, 2024

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, we could have a line at the end of our M6 macro 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 instead, we could 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 used for the probe instead of 99, they could then simply change the value in the Wizard. The first method would require the Operator to modify the macro and may lead to probe damage 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 Section 8.2.14 for a 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?

In CNC12 for use with Allin1DC and Oak CNC controllers, #4203 is used in five-axis machines, primarily for special tool changes involving "Large" tools. Currently, CNC12 for use with Acorn controllers does not use system variable #4203 for "Tool in spindle" and will result in an "Undefined variable" error in the software. System Variable #4203 is also set differently depending on how Parameters 6 and 160 are set.

If Parameter 6 and Parameter 160 are set to a zero, then #4203 is updated when a T-command is called. Essentially:

```
#4203 == #4120
```

Draft: January 11, 2024

If Parameter 6 is a non-zero value and 160 is set to zero, then #4203 is updated with 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 and then gives these values to the `SV_PLC_CAROUSEL_POSITION` variable.

If Parameter 6 and Parameter 160 is set to a non-zero value, then #4203 is updated after the M6 tool change macro is called and finished. This allows for special operations in the macro for primarily 5-axis machines, where #4120 can be used for the requested tool and #4203 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 and 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 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
```

Since 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.

Draft: January 11, 2024

Compare that with the following example, which is 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 ignored. The macro reads and writes the same local variables as the main program. Therefore, it calculates #3 = 10.0 + 5.0 = 15.0 and moves Y to this position. 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. However, 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 their own set of local variables.

Draft: January 11, 2024

18.) What is the PLC Detective?

While not directly involved with macros, the PLC Detective is a tool that macro programmers should at least be aware of and somewhat know how to use. It allows the Operator to see in real time what is going on inside the PLC program while the machine is running, which can be very useful for macro programmers.

From the PLC Detective manual: "*The PLC detective is a free 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 enables 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, wait, and when the error occurs, go look back in time at all of the I/O's leading up to the problem. This is a "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 and is not active in the PLC program, as well as 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 a Custom PLC Output?

To enable the use of M94 and M95 for a custom output (which is not necessary if you are using any one of the stock Wizard outputs), edit the PLC to include both the desired Custom Output and 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 cnct directory "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, M-function system variable, and logic are all defined, you will need to compile the edited source code. Refer to the CNC12 PLC Programming Manual on Page 5. Once the PLC is compiled, we can edit the macro we wish to use with this output. To turn on the "CustomM94M95" variable in the PLC, we will need to use an M94 command. See the example below:

```
M94 /18 ; Turn On CustomOut Request, Notice the value 18 the same as the value in PLC.
```

This will turn "CustomM94M95" ON in the PLC, which in turn will cause the PLC to 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. For example:

```
M95 /18 ; Turn Off CustomOut Request, Notice the value 18 the same as the value in PLC.
```

This will turn off "CustomM94M95" in the PLC, which will also turn off the output "CustomOut". This is a very basic example, and with the introduction of the PLC, additional logic can be added to give additional functionality. This includes timeout timers, fault messages, and more when using this output.

Draft: January 11, 2024

More information for PLC programming can be found in the Centroid CNC12 PLC Programming Manual and Centroid PLC Programming videos. Here are links to the playlists found on the Centroid CNC Technical Support Youtube channel:

https://www.youtube.com/user/CentroidSupport/playlists

https://www.youtube.com/playlist?list=PLXhs2C5No0_gFS_RmKNo7hii2WKIedQlQ

8.4 Advanced Macro Statements

NOTICE Branching and conditional execution are extremely powerful tools that, when 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. These undesirable effects can occur simply by graphing a program. These undesirable effects could include entering into an endless loop, failing to draw anything, or even wiping out all of 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.

Draft: January 11, 2024

8.4.1 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 16 kilobytes of memory for every 1000 block numbers in the program).

8.4.2 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” part of the statement 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]
```

8.4.3 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 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, but 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
```

Draft: January 11, 2024

```
             ; 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
```

8.4.4 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 both more readable and understandable.

Draft: January 11, 2024

9 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
G10		B	Parameter Setting
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
G32		A	Constant Lead Thread Cutting
G37			PWM Velocity Modulation
G40	*	D	Cutter Compensation Cancel
G41		D	Cutter Compensation Left
G42		D	Cutter Compensation Right
G50		B	Coordinate System Setting, Max. Spindle Speed Setting
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
G65		J	Call Macro
G70		B	Profile Finishing Cycle
G71		B	Profile Stock Removal in Turning
G72		B	Profile Stock Removal in Facing
G73		B	Profile Path Repeating Cycle
G74		B	End Face Peck Cutting Cycle
G75		B	Outer/Inner Diameter Peck Cutting Cycle
G76		B	Multi-Pass Threading Cycle
G80	*	B	Canned Cycle Cancel
G83		B	Deep Hole Drilling
G84		B	Tapping
G85		B	Boring Cycle
G90		A	Outer/Inner Diameter Cutting Cycle
G92		A	One-Pass Threading Cycle
G94		A	End Face Cutting Cycle
G96		H	Constant Surface Speed
G97	*	H	Constant Surface Speed Cancel
G98		I	Per Minute Feed

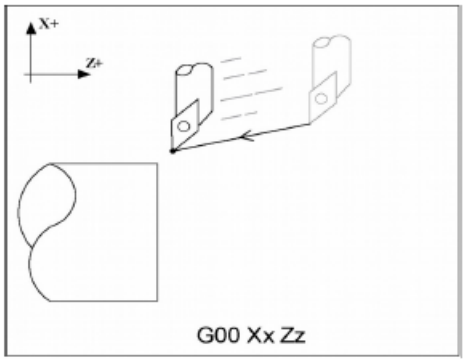
Draft: January 11, 2024

G-code		Group	Description
G99	*	I	Per Revolution Feed

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 (A, B, C, etc.) 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).

9.1 G00 – Rapid Positioning



G0 moves to the specified position at the maximum motor rate. G0 is modal and remains in effect until another positioning mode (G1, G2, G3 etc.) is commanded. G0 is the default-positioning mode.

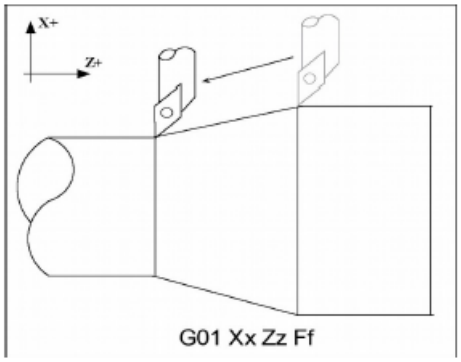
Example:

```
G0 X0.0 Z0.0 ; Rapid move to X0, Z0
```

This command moves both X and Z to the absolute coordinate 0.0 at maximum feed rate.

⚠ CAUTION The feed rate override knob has no effect on G0 moves unless rapid override is turned ON.

9.2 G01 – Linear Interpolation



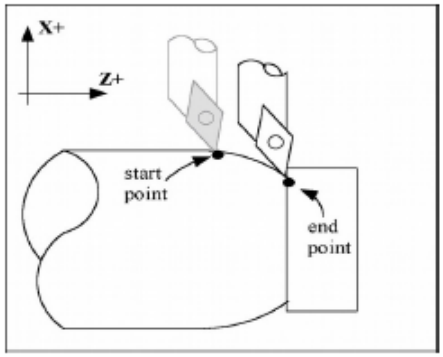
G1 moves to the specified position at the programmed feed rate. G1 is modal and remains in effect until another positioning mode (G0, G2, G3, etc.) is commanded.

Draft: January 11, 2024

Example:

```
G01 X2 Z4 F10
G01 X6 Z3 F20
```

9.3 G02 & G03 – Circular or Helical Interpolation



G2 moves in a clockwise* circular motion, and G3 moves in a counter-clockwise* circular motion. The X or Z position specified in the G2 or G3 command is the end position of the arc, and may be an absolute position (X, Z) or an incremental distance (U, W). G2 and G3 are modal and remain in effect until another positioning mode (G0, G1, etc.) is commanded.

Circular motion can be programmed in two different ways: specifying the final point and radius of the arc, or specifying the final point and parameters I and K (center point of the arc as incremental values from the start position).

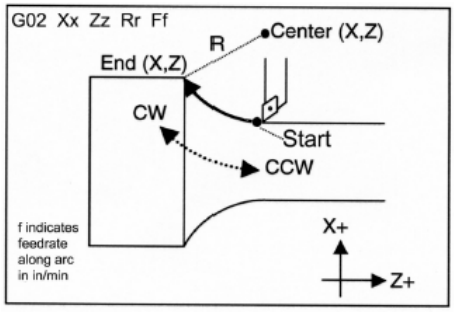
* The terms clockwise and counter-clockwise can be somewhat confusing, because they are relative directions which change based on one's perspective. To help conceptualize the correct perspective, always program your part and set up your tools as though the machine were a horizontal lathe with the tool post mounted in the rear and the head stock to your left.

Rules of thumb:

1. All Convex OD Arcs which move towards a more negative Z position should be programmed as CCW.
2. All Concave OD Arcs which move towards a more negative Z position should be programmed as CW.
3. All Convex OD Arcs which move towards a more positive Z position should be programmed as CW.
4. All Concave OD Arcs which move towards a more positive Z position should be programmed as CCW.
5. All Convex ID Arcs which move towards a more negative Z position should be programmed as CCW.
6. All Concave ID Arcs which move towards a more negative Z position should be programmed as CW.
7. All Convex ID Arcs which move towards a more positive Z position should be programmed as CW.
8. All Concave ID Arcs which move towards a more positive Z position should be programmed as CCW.

Draft: January 11, 2024

9.3.1 METHOD 1: USING FINAL POINT AND RADIUS



The commands G2 and G3 will have the following structure:

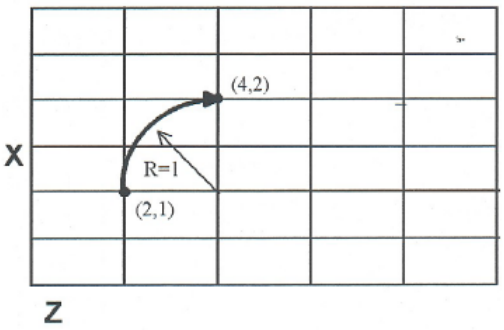
G2 Xx Zz Rr

G3 Xx Zz Rr

where *x* and *z* will be the X- and Z-coordinates of the final point of the arc, and *r* will be the radius.

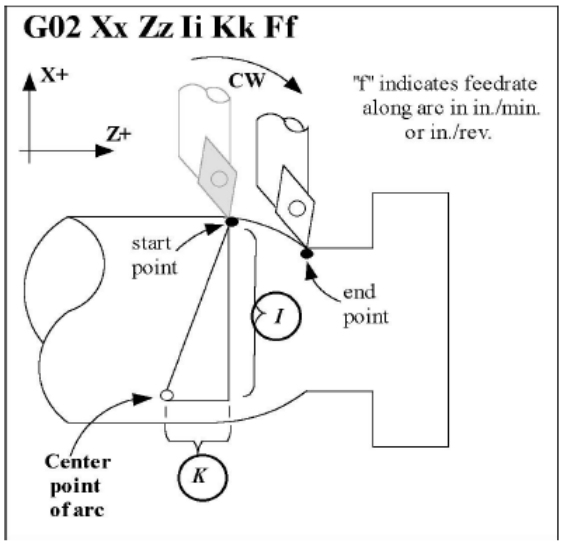
Example:

```
G00  X2.0  Z1.0     ; rapid to start position X2, Z1
G02  X4.0  Z2.0  R1 ; arc to X4 Z2 with radius of 1
```



Note: A lathe is not usually used to cut an arc larger than 90 degrees. With the use of special tools, a lathe can cut a 180-degree arc. This is the maximum value a lathe can cut an arc. Make sure the radius chosen follows the cutting ability of the lathe.

9.3.2 METHOD 2: USING FINAL POINT AND PARAMETERS I & K



Another way to specify a circular operation is using the parameters I and K instead of the radius R. The parameters I and K are the **incremental** distances from the start point to the center of the arc.

I = X center (radius) - X start (radius)

K = Z center - Z start

Note: X-coordinates are diameter values, but I and R are always radius values.

Example:

```
G00  X2.0  Z1.0     ; rapid to start pos.  X2, Z0
G02  X4.0  Z2.0  K1 ; arc to X4 Z1 with radius 1
```

Draft: January 11, 2024

9.4 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 Z1 ; rapid to X1 Z1
G4 P2.51 ; pause for 2.51 seconds
G0 X2 Z2 ; rapid to X2 Z2
```

9.5 G10 – Parameter Setting

G10 allows you to set parameters for different program operations.

Draft: January 11, 2024

Example:

```
G10 P5 Z-1.1    ; Sets tool #5 z offset to -1.1 in the Offset Library
G10 P5 X-1.3    ; Sets tool #5 x offset to -1.3 in the Offset Library
G10 P5 R.25     ; Sets tool #5 nose radius to .25 in the Offset Library
G10 P5 Q3       ; Sets tool #5 nose vector to 3 in the Offset Library
G10 P1073 R.05 ; Sets machine parameter 73 to 0.05
```

Note: The following parameters cannot be modified: 1–5, 7–9, 11–30, 39, 40, 42, 60–63, 65–67, 70, 71, 75–77, 82, 87–90, 95–98, 100–106, 107–112, 114, 116, 120–123, 132–135, 155–159, 165, 220–224, 226–231, 236–243, 248, 252–255, 257, 278, 281–283, 300–315, and 416.

9.6 G20 – Select Inch Units

G20 selects inch units, affecting the interpretation of all subsequent dimensions and feed rates in the job file. G20 does not change the native machine units as set on the control setup menu.

9.7 G21 – Select Metric Units

G21 selects metric units, affecting the interpretation of all subsequent dimensions and feed rates in the job file. G21 does not change the native machine units, as set in the control setup menu.

9.8 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, an “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 Z and X values for the ‘+’ limit and the I and J values 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 Z, X and I, J parameters are stored in the WCS menu under **F3 – Work Envel** (See Chapter 5).

9.9 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 the 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, only the X- and Z-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 W0      ; move Z axis directly to reference point (X doesn't move)
G28 U.5 W0  ; move X +.5, then move BOTH axes to reference point
G28 X2 Z.1  ; move both axes to (2,0.1), 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
```

The G28 position is of great importance because it specifies the Tool Check position and usual Tool Change position. The G28 position is the machine coordinate position that the machine will move to when the <TOOL CHECK> button is pressed. Also, the G28 position is the usual position at which tool changes occur during a job run.

Draft: January 11, 2024

9.10 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 X and Z axes back from reference point to intermediate point
G29 X1 Z2   ; move X and Z axes to intermediate point, then move to X1 Z2
```

9.11 G30 – Return to Secondary Reference Point

G30 moves to a specified return reference point, by way of an intermediate point. The P parameter may be used to specify one of the four available Return Reference Points: 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.

The four available return reference points are defined in the Work Coordinate System Configuration menu. If you issue a G30 without a P parameter, it functions exactly like G28, except that by default it uses the second reference return point.

The following table shows how to issue G-codes to utilize the four available Return Reference Points:

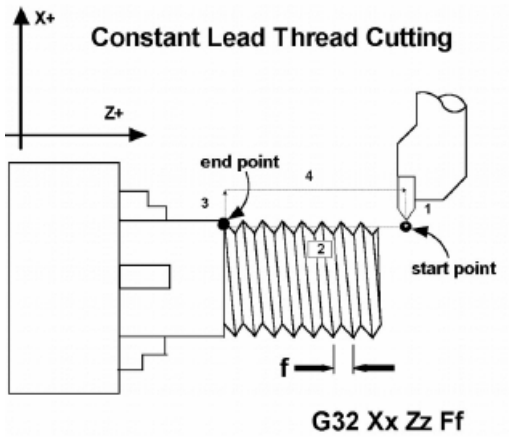
Return Reference Point	G-Code	Equivalent Alternate G-Code
Return #1	G28	G30 P1
Return #2	G30	G30 P2
Return #3	G30 P3	—
Return #4	G30 P4	—

Example:

```
G30 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.

9.12 G32 – Constant Lead Thread Cutting



G32 sets the constant lead thread cutting mode. During this mode, both axes are locked to the spindle encoder count. Once the encoder outputs a one turn signal, thread cutting is started at a fixed point so that the tool path remains unchanged for repeated thread cutting. Thread cutting follows the same tool path in rough cutting through finish cutting. **Note:** When G32 is used, X and Z indicate the endpoint of the cut, and F indicates the lead. Also, E can be used instead of F.

Example:

```
G00 X1.5 Z0.0          ; Step 1 - rapid move
G32 X1.5 Z-2.0 F0.125  ; Step 2 - straight thread cut of 2 inches, lead of .125 or 8 threads
                         per inch
G00 X1.7               ; Step 3 - Clear X-axis
G00 z0.0               ; Step 4 - Retract Z-axis
```

9.13 G37 – PWM Velocity Modulation ON/OFF

The PWM Velocity Modulation (VM) feature adjusts the PWM output based on the velocity of the machine tool so over burning is avoided when turning corners. Invoking G37 with either an ON or OFF parameter sets the PWM VM to ON or OFF. Invoking G37 without a parameter turns off PWM VM.

G-code parameter	Format Function
G37	Turns PWM VM off.
G37 ON	Turns PWM VM on.
G37 OFF	Turns PWM VM off.

Draft: January 11, 2024

9.14 G40, G41, G42 – Cutter Compensation

G41 and G42, in conjunction with the selected tool (T-code), apply cutter compensation to the programmed tool path. Cutter compensation is required whenever an angle or radius is being cut.

G41 offsets the tool to the left of the workpiece, relative to the direction of travel. This offset is equal to its nose radius.

G42 offsets the tool to the right of the workpiece, relative to the direction of travel. This offset is equal to its nose radius.

G40 cancels G41 and G42.

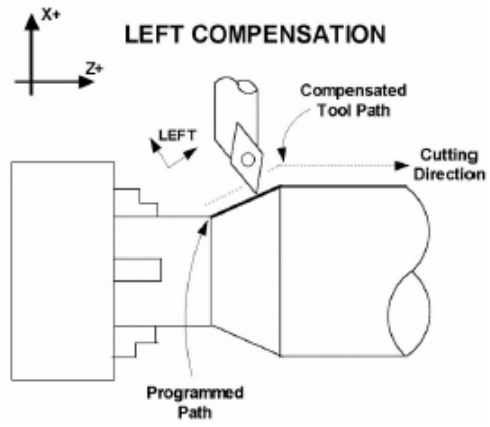
Always program cutter compensation as though the machine were a horizontal lathe with the tool post mounted in the rear and the head stock to your left.

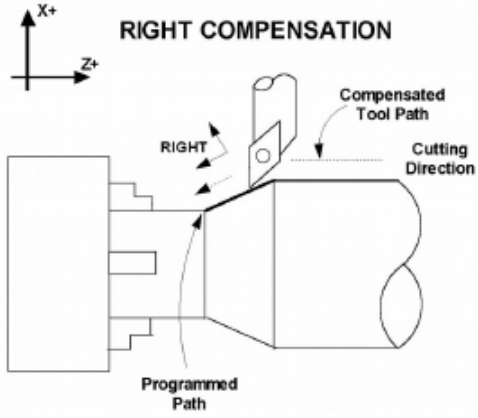
Rules of Thumb:

1. All OD moves which move towards a more negative Z should use cutter comp right.
2. All OD moves which move towards a more positive Z should use cutter comp left.
3. All ID moves which move towards a more negative Z should use cutter comp left.
4. All ID moves which move towards a more positive Z should use cutter comp right.

Example:

```
G41 T03 ; Tells the machine to compensate to the left by the amount of the nose radius that
          corresponds to T03 in the Offset Library.
```

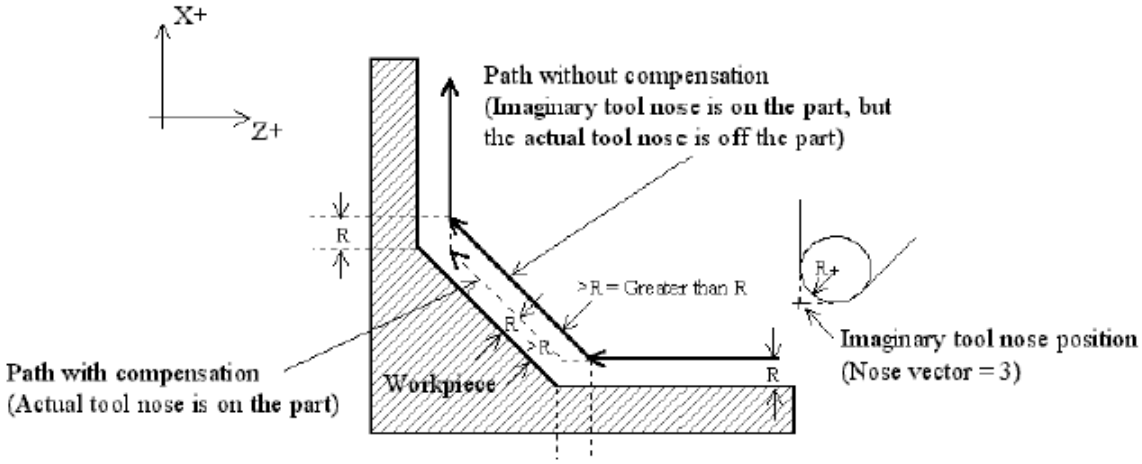




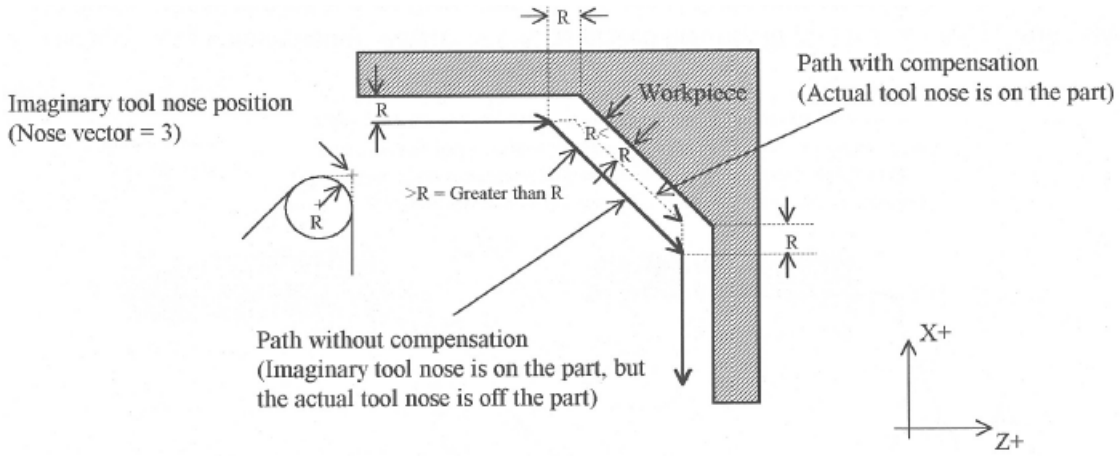
Imaginary Tool Nose

Tool nose compensation is necessary to prevent under-cutting (not cutting enough material) on diagonal lines and arcs. Tool nose compensation does not affect horizontal and vertical lines, because in those cases the actual tool nose is at the same depth as the imaginary tool nose. When tool nose compensation is not used, it is the imaginary tool nose that moves to the programmed position and not the cutter. Cutter compensation adjusts for the difference in position by moving the actual tool nose to the programmed position.

Draft: January 11, 2024



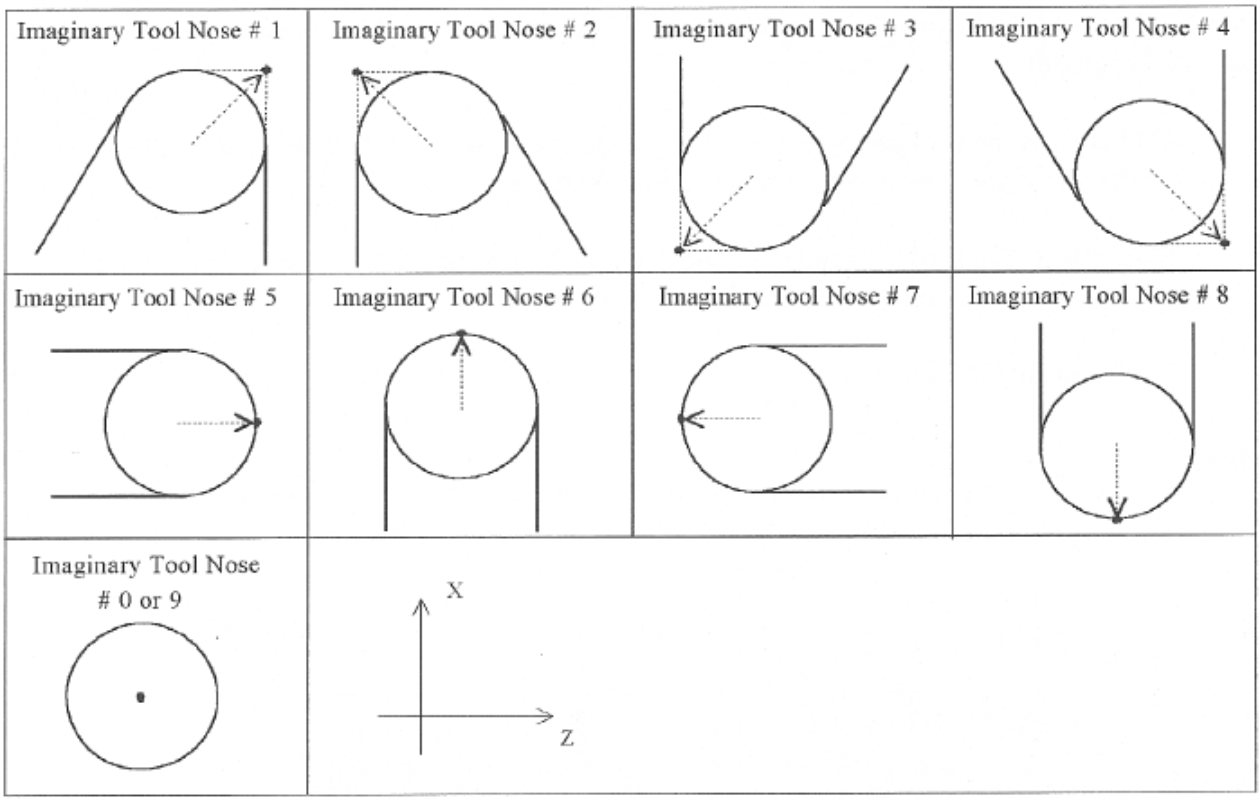
Example with tool located on back side of material.



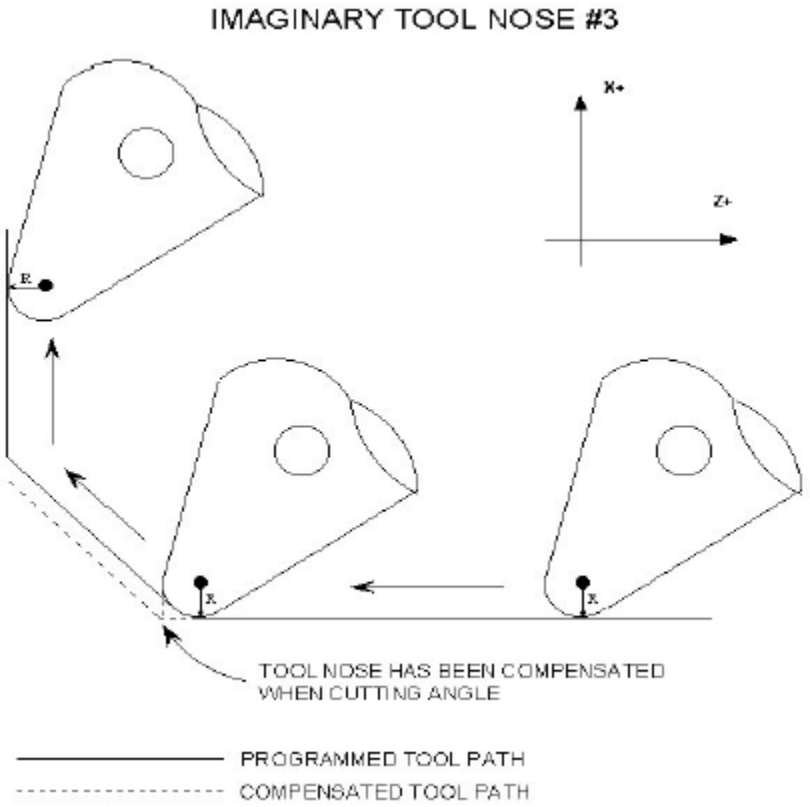
Example with tool located on front side of material.

The direction of the imaginary tool nose is related to the nose vector or direction of the tool during cutting (see Chapter 4). The following drawings show the possible imaginary tool nose directions (tool located in back of material):

Draft: January 11, 2024



The tool nose compensation function (G41 or G42) should be in effect before the tool reaches the cutting start point.



Draft: January 11, 2024

9.15 G50 – Coordinate System Setting OR Maximum Spindle Speed for CSS mode

G50 has two functions depending on the supplied parameters:

- With axis parameters, G50 sets the current absolute position to the coordinates specified OR
- With the S parameter, G50 sets the maximum spindle speed when using constant surface speed (see G96 and G97).

Example:

```
G00 X5 Z-2 ; moves to the specified location
G50 X1 Z0  ; sets the current position to the absolute position specified.
G50 S2500  ; limit spindle to 2500 rpm in G96 mode, no matter how close X gets to 0.
```

9.16 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.

Draft: January 11, 2024

Example:

```
G0 X0 Z0   ; move to origin
M98 P9100  ; call subprogram
G52 Z4     ; shift coordinate system 4 inches in Z
G0 X0 Z0   ; move to new origin
M98 P9100  ; call subprogram again with new coordinates
G52 Z0     ; restore un-shifted coordinate system
```

9.17 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 (G00–G03) or coordinate system (G54–G59). Movement rate can be overridden using the `L` word.

Example:

```
G53 X15 Z0       ; move to 15,0 in machine coordinates
G53 X15 Z0 L100  ; move to 15,0 in machine coordinates at 100 units/minute
```

9.18 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.

Example:

```
G54 G00 X0 Z0    ; select first WCS, move to origin
G02 X1 Z-.5 R.5  ; cut something...
G55 G00 X1 Z1    ; select second WCS, move to X1,Z1
```

Using Extended Work Coordinate Systems:

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 "G54 P1" (WCS #7) through "G54 P12" (WCS #18).

Regular WCS

WCS	G-code
WCS #1	G54
WCS #2	G55
WCS #3	G56
WCS #4	G57
WCS #5	G58
WCS #6	G59

Extended Work Coordinate Systems (optional)

WCS	G-code
WCS #7	G54 P1
WCS #8	G54 P2
WCS #9	G54 P3
WCS #10	G54 P4
WCS #11	G54 P5
WCS #12	G54 P6

WCS	G-code
WCS #13	G54 P7
WCS #14	G54 P8
WCS #15	G54 P9
WCS #16	G54 P10
WCS #17	G54 P11
WCS #18	G54 P12

9.19 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 ten 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

Macro `TEST.cnc`:

Draft: January 11, 2024

G01 X#B Z-#A

This call will produce:

G01 X3 Z-5

Example 2:

Main Program:

G65 "TEST2.cnc" I3 J-5 K0.1 I2 J-2 I0 J0

Macro TEST2.cnc:

G1 X#4 Z#5 F#6

G1 X#7 Z#8

G1 X#10 Z#11

This call will produce:

G1 X3 Z-5 F40

G1 X2 Z-2

G1 X0 Z0

Draft: January 11, 2024

9.20 G70 – Profile Finishing Cycle

The G70 finishing cycle is used for applying finish cut passes on a defined profile path by removing material purposely left behind by a previously-run "roughing" cycle (G71, G72, or G73). Cutter compensation can be used with this cycle. The feed rate, spindle speed, and tool can be different than those used on the "roughing" cycle.

Computed Format:

```
G70 P_Q_U_W_

P = Starting Block number of finish profile path
Q = Ending Block number of finish profile path
U = Finishing Offset in X direction *
W = Finishing Offset in Z direction
```

Description of P and Q:

P and Q are required parameters.

P and Q (Starting/Ending Block numbers) do not need to match those used in the "roughing" cycle.

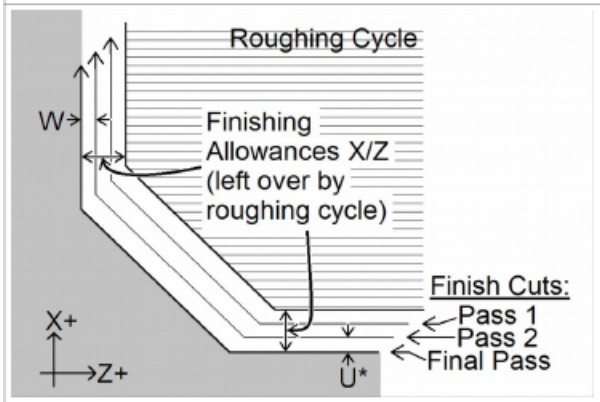
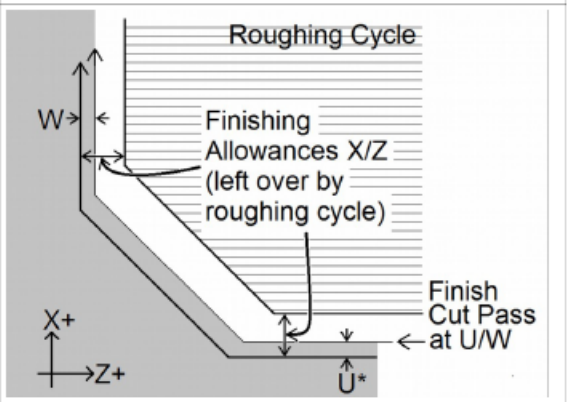
The P and Q parameters must refer to G-code lines occurring before the G70 finishing cycle and not after.

Description of U and W:

The U and W parameters are optional and default to U0 and W0 if not specified.

U and W axis direction signs are computed automatically by the control software.

The G70 cycle has two different behaviors depending on the setting of Parameter 247:

G70 Multi-pass Behavior Enabled (Parameter 247 = 0)	G70 Single Pass Mode (Parameter 247 = 1)
	
The number of finish cut passes is the larger of the number of times U or W can be divided into their respective X/Z Finish Allowances from a previously run G71/G72/G73 roughing cycle. The resultant finish cut passes generated will be evenly offset from each other. The final pass will be done on the defined profile path without any X/Z offset. However, if U and W are not specified (or is specified as U0 W0), then a single finish cut pass will be done on the defined profile path without any X/Z offset.	A single finish cut pass will be done at the specified cut level. U and W sets the X/Z cut level. A non-zero U or W will leave material behind. This behavior is industry standard.

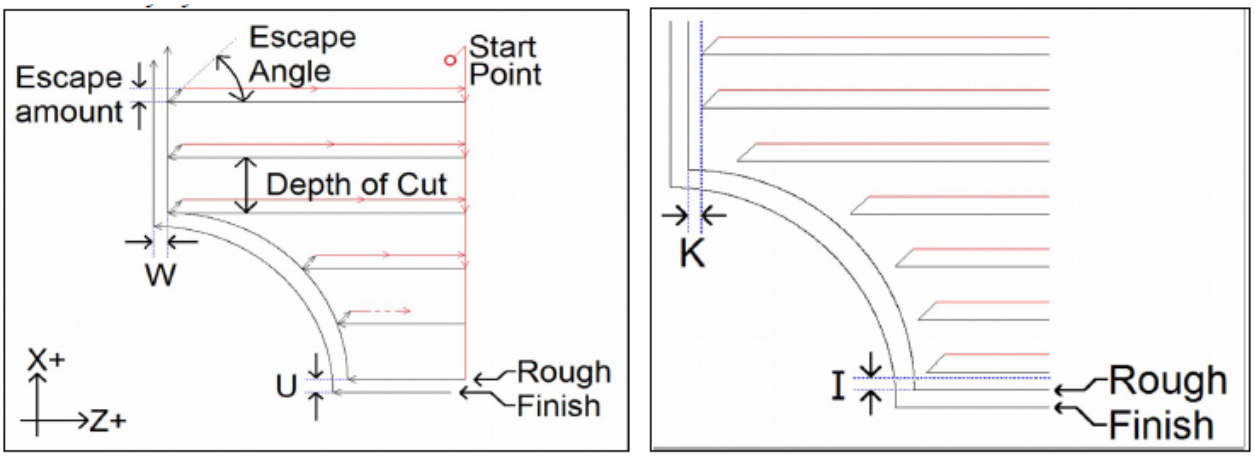
*The U value is subject to Radius/Diameter mode of the machine (as set by Parameter 55 – see Chapter 14).

If codes such as F, S, T, and G4 P_ are encountered in the profile, they will be executed by the G70 finishing cycle. A “roughing” cycle such as G71, G72, or G73 will ignore them.

9.21 G71 – Profile Stock Removal in Turning

The G71 cycle removes material by using a series of cutting passes parallel to the Z-axis, incrementally moving away from the Start Point. Each successive cutting pass is done at a progressively deeper turning diameter in the X-axis direction, incrementally moving away from the Start Point. The whole cycle is limited within the confines of the Start Point and a defined profile path (specified via P and Q). The Start Point is wherever the cutter is sitting when the G71 cycle is invoked using P and Q parameters. When all of the Z-parallel cuts are done, a single rough cut pass is run along the profile path. This cycle can perform both inner and outer diameter cleanouts, and the profile path can be cutter compensated. The whole cycle can be optionally offset by U and W to leave material behind in preparation for the G70 Finishing Cycle. Also, the Z-parallel cuts themselves can be optionally offset by I and K to further separate them from the single rough cut pass. Note that the axis direction signs for U, W, I, and K are computed automatically by the control software.

Draft: January 11, 2024



G71 has two forms:

2 Line Format	1 Line Format
Line 1: G71 U_ R_ Line 2: G71 P_ Q_ U_ W_ I_ K_ A_	G71 P_ Q_ D_ U_ W_ I_ K_ A_
Line 1 U = Depth of Cut * Line 1 R = Escape amount Line 2 P = Starting Block number of profile path Line 2 Q = Ending Block number of profile path Line 2 U = Finishing Allowance in X direction * Line 2 W = Finishing Allowance in Z direction Line 2 I = Roughing Allowance in X direction Line 2 K = Roughing Allowance in Z direction Line 2 A = Escape Angle	P = Starting Block number of profile path Q = Ending Block number of profile path D = Depth of Cut † U = Finishing Allowance in X direction * W = Finishing Allowance in Z direction I = Roughing Allowance in X direction K = Roughing Allowance in Z direction A = Escape Angle

* U values are always in radius regardless of the Radius/Diameter mode of the machine (as set by Parameter 55).

† D value is subject to the “implied floating point” re-interpretation rules of Parameters 245 and 246.

P and Q are required parameters.

U and W Finishing Allowances are optional and default to U0 and W0 if not specified.

I and K are optional and default to I0 and K0 if not specified.

The modal default for Depth of Cut is from Parameter 43.

The modal default for Escape Amount is from Parameter 44.

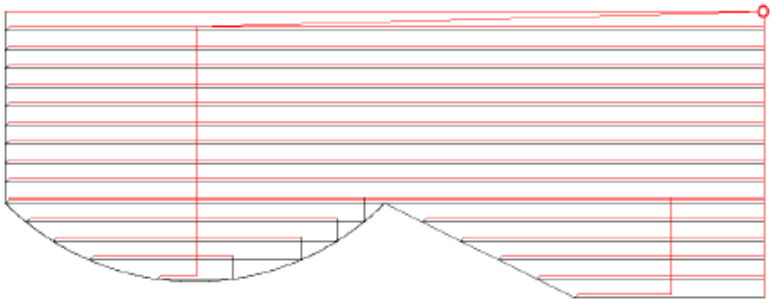
The modal default for Escape Angle is from Parameter 58.

The profile path needs to be defined after the G71 P_ Q_. The first and last G-code lines of the profile path must be tagged with the N block numbers so that P and Q can refer to them. If there are any G-code lines occurring after the G71 P_ Q_ and before the first G-code line of the profile path, those lines will not be run by this cycle, but can be referenced later by a G70 P_ Q_ Finishing Cycle.

Note that any F, S, T, and G4 P_ codes encountered in the profile path will be ignored by the G71 cycle.

Draft: January 11, 2024

G71 Example:

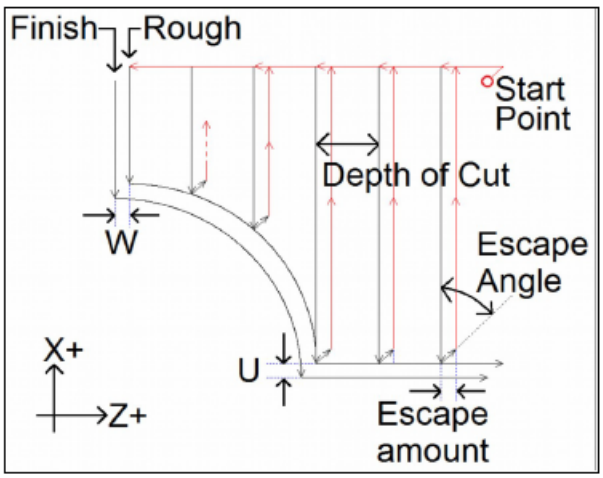


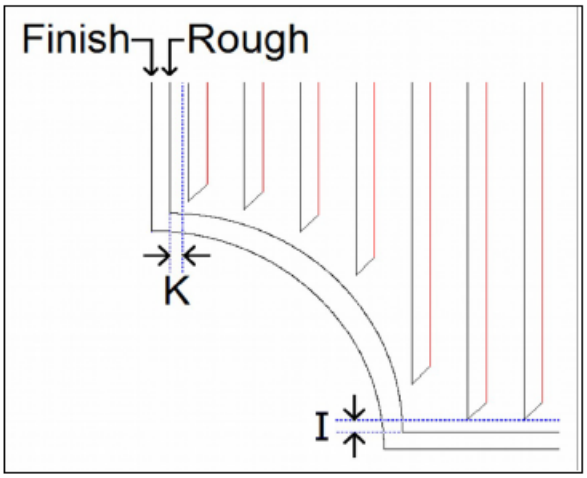
```
G0 X3 Z0                ; Rapid to Start Point
G71 U.05 R.01           ; Set Depth of Cut and Escape Amt
G71 P10 Q20 F.02 S800 ; Cleanout Profile at N10 to N20
N10 G0 X1.5             ; PROFILE first block
G1 X1.5 Z-.5            ; PROFILE
G1 X2 Z-1               ; PROFILE
G2 X2 Z-2 I.5 K-.5      ; PROFILE
N20 G1 X3               ; PROFILE last block
```

Draft: January 11, 2024

9.22 G72 – Profile Stock Removal in Facing

The G72 cycle removes material by using a series of cutting passes parallel to the X-axis and moving away from the Start Point. Each successive cutting pass is done at an incrementally-progressive Z-face in the direction away from the Start Point. The whole cycle is limited within the confines of the Start Point and a defined profile path (specified via P and Q). The Start Point is wherever the cutter is sitting when the G72 cycle is invoked with P and Q parameters. When all of the X-parallel cuts are done, a single rough cut pass is then run along the profile path. The profile path can be cutter compensated. The whole cycle can be optionally offset by U and W to leave material behind in preparation for the G70 Finishing Cycle. Also, the X-parallel cuts themselves can be optionally offset by I and K to further separate them from the single rough cut pass. Note that the axis direction signs for U, W, I, and K are computed automatically by the control software.





G71 has two forms:

2 Line Format	1 Line Format
Line 1: G72 W_ R_ Line 2: G72 P_ Q_ U_ W_ I_ K_ A_	G71 P_ Q_ D_ U_ W_ I_ K_ A_
Line 1 W = Depth of Cut Line 1 R = Escape amount Line 2 P = Starting Block number of profile path Line 2 Q = Ending Block number of profile path Line 2 U = Finishing Allowance in X direction * Line 2 W = Finishing Allowance in Z direction Line 2 I = Roughing Allowance in X direction Line 2 K = Roughing Allowance in Z direction Line 2 A = Escape Angle	P = Starting Block number of profile path Q = Ending Block number of profile path D = Depth of Cut † U = Finishing Allowance in X direction * W = Finishing Allowance in Z direction I = Roughing Allowance in X direction K = Roughing Allowance in Z direction A = Escape Angle

* U values are subject to the Radius/Diameter mode of the machine (as set by Parameter 55).

† D value is subject to the “implied floating point” re-interpretation rules of Parameters 245 and 246.

P and Q are required parameters.

U and W Finishing Allowances are optional and default to U0 and W0 if not specified.

I and K are optional and default to I0 and K0 if not specified.

The modal default for Depth of Cut is from Parameter 43.

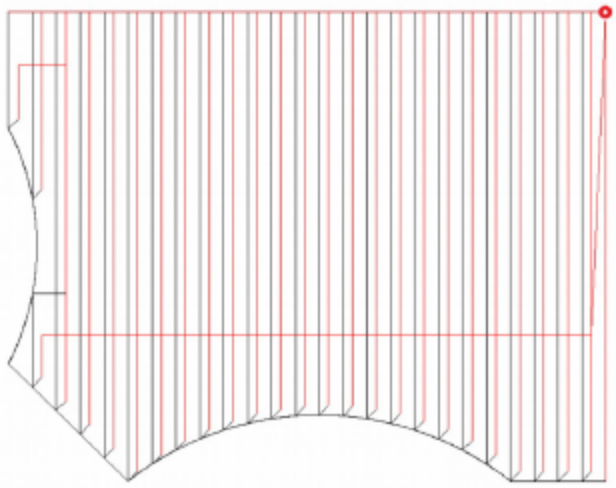
The modal default for Escape Amount is from Parameter 44.

The modal default for Escape Angle is from Parameter 58.

The profile path needs to be defined after the G72 P_ Q_. The first and last G-code lines of the profile path must be tagged with the N block numbers so that P and Q can refer to them. If there are any G-code lines occurring after the G72 P_ Q_ and before the first G-code line of the profile path, those lines will not be run by this cycle, but can be referenced later by a G70 P_ Q_ Finishing Cycle.

Note that any F, S, T, and G4 P_ codes encountered in the profile path will be ignored by the G72 cycle, including the single rough cut pass at the end.

G72 Example:

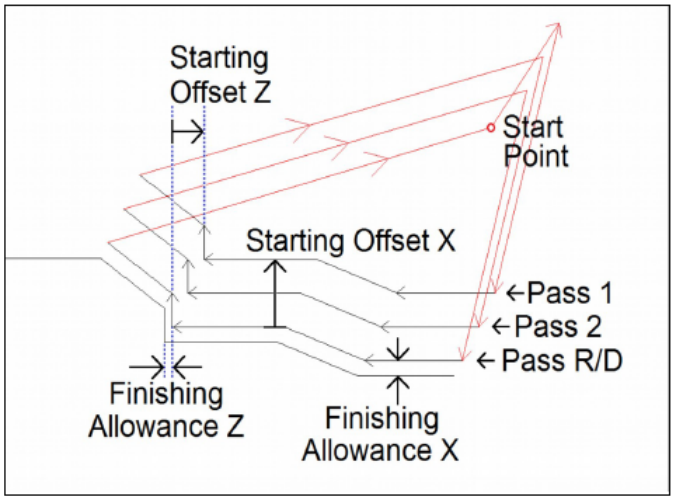


Draft: January 11, 2024

```
G0 X3 Z0                    ; Rapid to Start Point
G72 W.05 R.02               ; Setup Depth of Cut and Escape Amount
G72 P11 Q22 F.01 S600       ; Cleanout Profile at N11 to N22
N11 G0 X1                   ; PROFILE first block
G1 X1 Z-.2                  ; PROFILE
G3 X1 Z-1 I-.5 K-.4         ; PROFILE
G1 X1.5 Z-1.25              ; PROFILE
G3 X2.5 Z-1.25 I.25 K-.5    ; PROFILE
N22 G1 X3 Z-1.25            ; PROFILE last block
```

9.23 G73 – Profile Path Repeating Cycle

The G73 cycle removes material by repeating a defined profile path (specified via P and Q) progressively deeper into the material. The profile path can be cutter compensated. The amount of material to remove is determined by the Starting Offset Amounts (in X and Z directions). The whole cycle can be optionally offset by Finishing Allowances (in X and Z directions) to leave material behind in preparation for the G70 Finishing Cycle.



G73 has two forms:

2 Line Format	1 Line Format
Line 1: G73 U_ W_ R_ Line 2: G73 P_ Q_ U_ W_	G73 I_ K_ D_ P_ Q_ U_ W_
Line 1 U = Starting Offset Amount in X direction *† Line 1 W = Starting Offset Amount in Z direction † Line 1 R = Total number of Passes Line 2 P = Starting Block number of profile path Line 2 Q = Ending block number of profile path Line 2 U = Finishing Allowance in X direction *† Line 2 W = Finishing Allowance in Z direction †	I = Starting Offset Amount in X direction † K = Starting Offset Amount in Z direction † D = Total number of Passes P = Starting Block number of profile path Q = Ending block number of profile path U = Finishing Allowance in X direction *† W = Finishing Allowance in Z direction †

* U values are always radius regardless of the Radius/Diameter mode of the machine (as set by Parameter 55).

† Note that axis direction signs for U, W, I, and K are computed automatically by the control software.

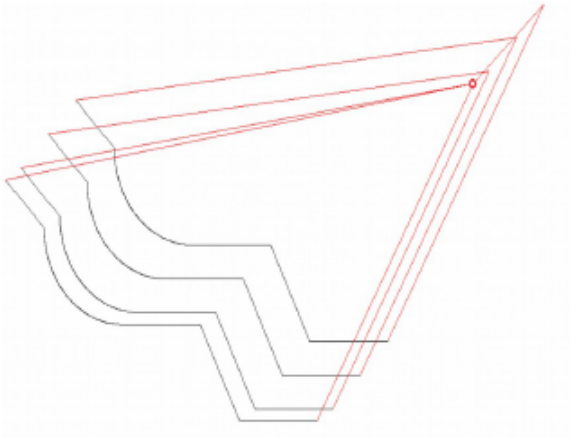
P and Q are required parameters.

Draft: January 11, 2024

U and W Finishing Allowances are optional and default to U0 and W0 if not specified.

The profile path needs to be defined after the G73 P_ Q_. The first and last G-code lines of the profile path must be tagged with the N block numbers so that P and Q can refer to them. If there are any G-code lines occurring after the G73 P_ Q_ and before the first G-code line of the profile path, those lines will not be run by this cycle, but can be referenced later by a G70 P_ Q_ Finishing Cycle. Note that any F, S, T, and G4 P_ codes encountered in the profile path will be ignored by the G73 cycle.

G73 Example:

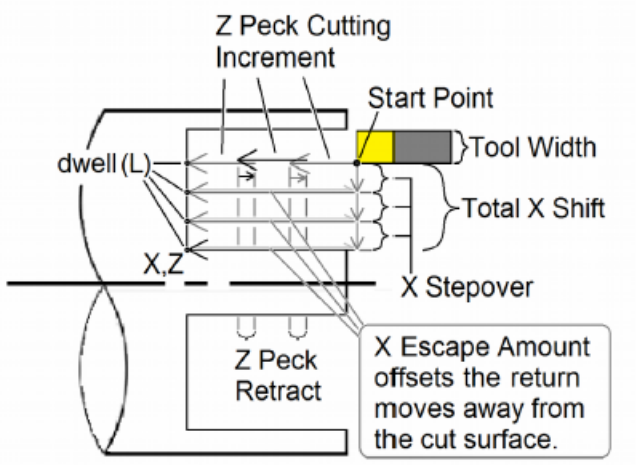
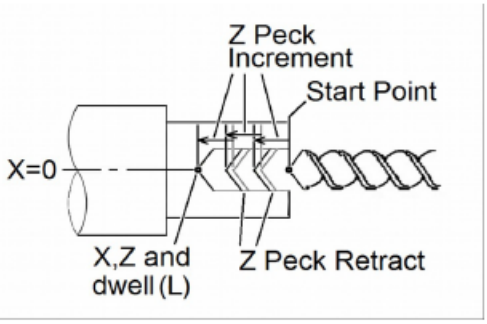


```
G0 X2.2 Z1.6                      ; Rapid to Start Point
G73 U.14 W.14 R3                  ; REPEAT PROF R=3x, U/W=Starting Offset
G73 P88 Q99 U.05 W.04 F.03 S500   ; REPEAT PROF P-Q, U/W=Finish Allowance
N88 G0 X.8 W-.40                  ; PROFILE first block
G1 W-.2                           ; PROFILE
X1.2 W-.1                         ; PROFILE
W-.2                              ; PROFILE
G2 X1.6 W-.2 R.2                  ; PROFILE
N99 G1 X1.8 W-.1                  ; PROFILE last block
G70 P88 Q99                       ; FINISH CYCLE P-Q
```

9.24 G74 – End Face Peck Cutting Cycle

G74 is used for end face grooving and can also be used for end face peck drilling. The cycle's Z peck cutting action is used for chip breaking. When doing end face grooving, an optional X Shift and X Stepover can be specified to direct the cycle to run multiple shifted groove cut passes in order to produce a groove width that is wider than the tool width. The Start Point of the G74 cycle is wherever the cutter is sitting when the cycle is invoked with a Z_ parameter †.

Draft: January 11, 2024

For End Face Peck Cutting:	For Peck Drilling:
	

G74 has two forms:

2 Line Format	1 Line Format
Line 1: G74 R_ Line 2: G74 X_ Z_ P_ Q_ R_ L_	G74 X_ Z_ I_ K_ D_ L_
Line 1 R = Z Peck Retract Amount * Line 2 X = Final X Shift Position † Line 2 Z = Final Z-depth † Line 2 P = X Stepover ‡ Line 2 Q = Z Peck Cutting Increment ‡ Line 2 R = X Escape Amount for return move Line 2 L = Dwell time at each Final Z-depth	 X = Final X Shift Position † Z = Final Z-depth † I = X Stepover K = Z Peck Cutting Increment D = X Escape Amount for return move ‡ L = Dwell time at each Final Z-depth

* The modal default for Peck Retract Amount is from Parameter 73. Note that in the One-line format, you cannot specify a Peck Retract Amount directly. If you are using the One-line format exclusively, make sure Parameter 73 is correctly set.

† Incremental X- and Z-positions are specified using U and W respectively. Note that if U is used, it will be subject to the Radius/Diameter mode of the machine (as set by Parameter 55).

‡ P, Q, and D values are subject to the “implied floating point” re-interpretation rules of Parameters 245 and 246.

Final Z-depth (Z or W) and the Z Peck Cutting Increment (Q or K) are the minimal required parameters; all the others are optional.

Final X Shift Position defaults to the cycle’s X Start Point if it is not specified.

X Stepover (P or I) defaults to 0 if not specified.

X Escape Amount (R or D) defaults to 0 if not specified.

Dwell Time (L) should be left out entirely if no dwell is desired because L0 will still cause a (very brief) dwell.

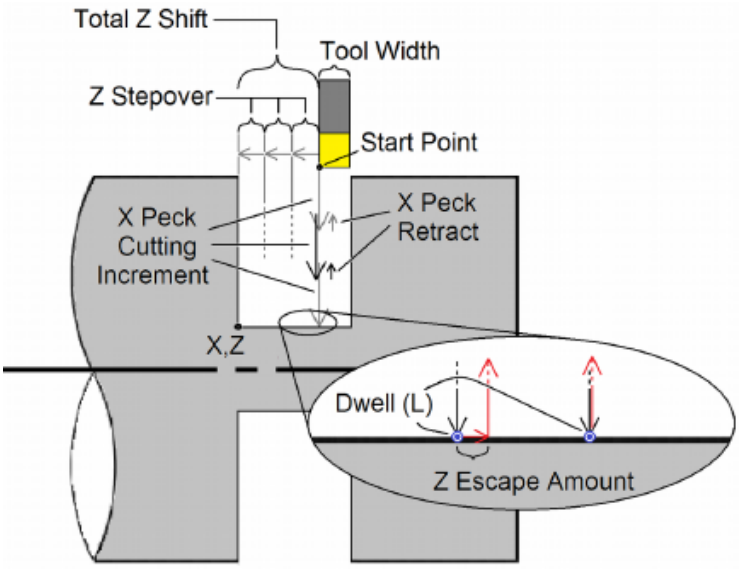
When doing end face grooving, the total width of the resulting groove is the total of the Tool Width added to the Total X Shift. A return move will be done after each groove cut pass (and after the dwell if it is specified). This return move will be spaced away from the cut by the specified X Escape Amount, except for the very first cut pass. To prevent the grooving tool from backing up against a wall, the very first groove cut will have no X escape amount.

Draft: January 11, 2024

For centerline peck drilling, the cycle's X Start Point should be 0, and both the Final X Shift Position and X Stepover should be 0.

9.25 G75 – Outside/Inside Diameter Peck Cutting Cycle

G75 is used for outside/inside diameter grooving. The cycle's X peck cutting action is used for chip breaking. An optional Z Shift and Z Stepover can be specified to direct the cycle to run multiple shifted groove cut passes in order to produce a groove width that is wider than the tool width. The Start Point of the G75 cycle is wherever the cutter is sitting when the cycle is invoked with an X_ parameter †.



G75 has two forms:

2 Line Format	1 Line Format
Line 1: G75 R_ Line 2: G75 X_ Z_ P_ Q_ R_ L_	G75 X_ Z_ I_ K_ D_ L_
Line 1 R = Z Peck Retract Amount * Line 2 X = Final X-depth † Line 2 Z = Final Z Shift Position † Line 2 P = X Peck Cutting Increment ‡ Line 2 Q = Z Stepover ‡ Line 2 R = Z Escape Amount for return move Line 2 L = Dwell time at each Final X-depth	 X = Final X-depth † Z = Final Z Shift Position † I = X Peck Cutting Increment K = Z Stepover D = Z Escape Amount for return move ‡ L = Dwell time at each Final X-depth

* The modal default for Peck Retract Amount is from Parameter 73. Note that in the One-line format, you cannot specify a Peck Retract Amount directly. If you are using the One-line format exclusively, make sure Parameter 73 is correctly set.

† Incremental X- and Z-positions are specified using U and W respectively. Note that X and U are subject to the Radius/Diameter mode of the machine (as set by Parameter 55).

‡ P, Q, and D values are subject to the "implied floating point" re-interpretation rules of Parameters 245 and 246.

Final X-depth (X or U) and the X Peck Cutting Increment (P or I) are the minimal required parameters; all the others are optional.

Draft: January 11, 2024

Final Z Shift Position defaults to the cycle's Z Start Point if it is not specified.

Z Stepover (Q or K) defaults to 0 if not specified.

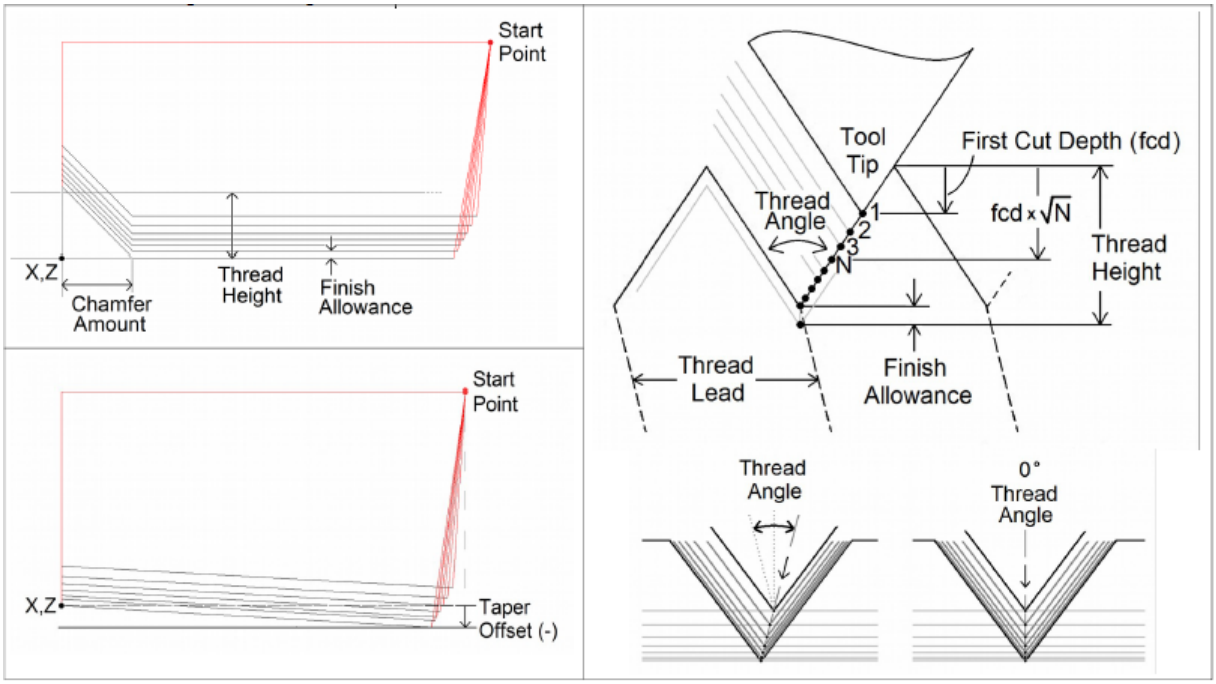
Z Escape Amount (R or D) defaults to 0 if not specified.

Dwell Time (L) should be left out entirely if no dwell is desired because L0 will still cause a (very brief) dwell.

When this cycle is used for grooving, the total width of the resulting groove is the total of the Tool Width added to the Total Z Shift. A return move will be done after each groove cut pass (and after each optional dwell). This return move will be spaced away from the cut by the specified Z Escape Amount, except for the very first cut pass (to prevent the grooving tool from backing up against a wall).

9.26 G76 – Multi-pass Threading Cycle

In this cycle, threading is performed in successive incremental passes to a specified depth. The Start Point of the G76 cycle is wherever the cutter is sitting when the cycle is invoked with an X_ and Z_ parameter *. Optional chamfer and taper can be specified.



The first thread pass is done at the "First Cut Depth" and each successive incremental cut depth will be progressively less than the previous in order to account for greater chip load as the cuts get deeper. However, you can optionally specify a lower limit to this diminishing cut depth increment by specifying a "Minimum Limit Cut Depth".

The Thread Angle dictates the chip load balance on each side of the thread cutter from one threading pass to the next. A Thread Angle of 0 means an even amount of chip load for each side of the thread cutter.

G76 has two forms:

Draft: January 11, 2024

2 Line Format	1 Line Format
Line 1: G76 Pmmrraa Q_ R_ Line 2: G76 X_ Z_ R_ P_ Q_ F_	G76 X_ Z_ K_ D_ A_ I_ F_
Line 1 P mm____ = Finish/Spring Pass Count Line 1 P __rr__ = Chamfer Amount in units of 1/10 Thread Lead Line 1 P ____aa = Thread Angle Line 1 Q = Minimum Limit Cut Depth † Line 1 R = Finish Allowance Line 2 X = Thread Final Depth Position X * Line 2 Z = Thread Final Position Z * Line 2 R = Taper Offset (+/-) Line 2 P = Thread Height † Line 2 Q = First Cut Depth † Line 2 F = Thread Lead ‡	X = Thread Final Depth Position X * Z = Thread Final Position Z * K = Thread Height D = First Cut Depth † A = Thread Angle I = Taper Offset (+/-) F = Thread Lead ‡

* Incremental X- and Z-positions are specified using U and W respectively. Note that X and U are subject to the Radius/Diameter mode of the machine (as set by Parameter 55).

† P, Q, and D length values are subject to "implied floating point" re-interpretation rules of Parameters 245 and 246.

‡ F is not strictly a parameter of this cycle as it is a modal value. Alternatively, the modal E value can also be used to specify Thread Lead.

The minimal required parameters for this cycle are Thread Final positions X-depth and Z, First Cut Depth (Q or D), and Thread Height (P or K); all the others are optional and have a modal effect if specified. Whatever optional values are not specified are taken from the current modal state. In the case of the One-line format, some values simply cannot be specified using this format. These unspecified values will come from the current modal values, which at the beginning of a job are initialized from Parameters 49–53.

Multi-start Threads:

Multi-start (Multiple Start) threads are when there is more than one thread starting on the same diameter. Multi-start threads can be cut by changing the starting positions of each of the next G76 cycles. The first G76 is programmed at the correct starting position for clearance, feed rate, and spindle speed. For the Z-starting position of the next G76, you divide the pitch by the number of Multi-start threads, and add this Z-offset to the first G76's starting Z. The rest of the G76 cycle is the same. For more than two Multi-start start threads, you would add the Z-offset to the previous G76's Z starting position for each additional Multi-start thread. With a Multi-start thread, the next thread's Z starting position will be further away, but the ending Z will be the same for all of the thread starts.

The modal default for Finish / Spring Pass Count is from Parameter 50.

The modal default for Chamfer Amount is from Parameter 49 (specified in multiples of Thread Lead).

The modal default for Thread Angle is from Parameter 51.

The modal default for Minimum Limit Cut Depth is from Parameter 52.

The modal default for Finish Allowance is from Parameter 53.

Draft: January 11, 2024

9.27 G80 – Canned Cycle Cancel

G80 is used to cancel a canned cycle once the operation has been performed.

9.28 G83 – Deep Hole Drilling

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 down to the standard feed rate a short distance above the bottom. This clearance distance is selected by setting Parameter 83 with G10 (see the example below).

Two-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,

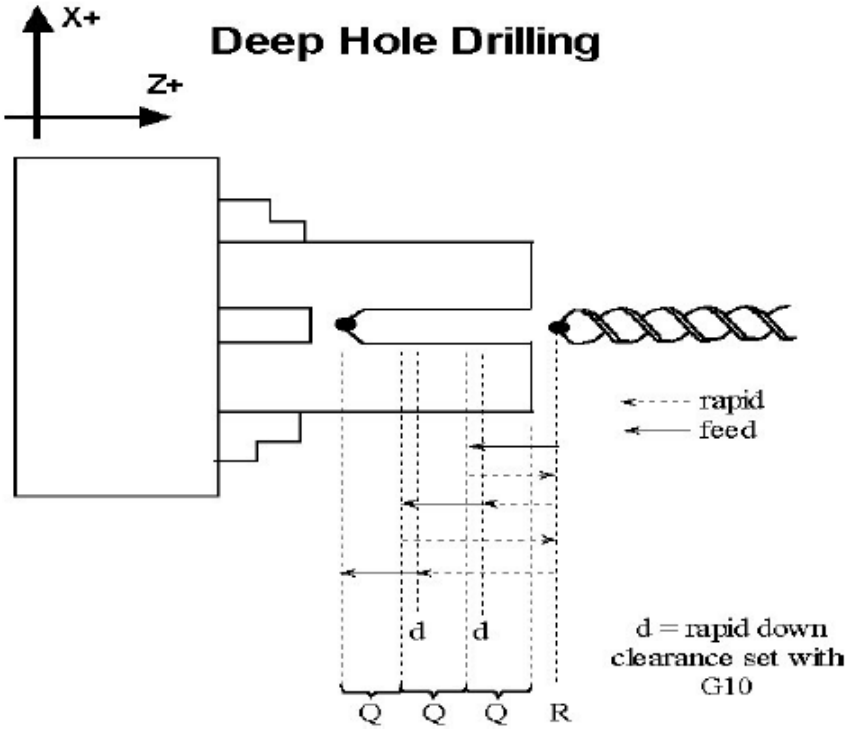
Final Z-depth, the Z Peck Cutting Increment (Q), and Return Point (R) are the minimal required parameters. All others are optional. If X is not specified, the last X-position will be used. X should always X0.0 unless the machine is set up with C-axis and live tooling.

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 R.1 Z-2 Q.5 ; drill 2" deep hole in 0.5" steps
```

Draft: January 11, 2024



Draft: January 11, 2024

9.29 G84 – Tapping

G84 performs both right-hand (Tap) and left-hand (Counter-tap) cycles. The tap direction, spindle speed, spindle direction, feed rate, and/or thread pitch should be set before the G84 is invoked. The tapping direction is set by calling M29 with or without a P1 parameter (see M29 in Chapter 10). G84 defaults to a right-hand Tap cycle if M29 P1 is not specified beforehand. The spindle should be started beforehand using M3 (Spindle CW) for the Tap cycle or M4 (Spindle CCW) for the Counter-tap cycle.

For a floating tap head, the combination of the modal feed rate and spindle speed implicitly determines the approximate thread lead or pitch. However, if Rigid Tapping is enabled, a Q may be used to explicitly 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.

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.

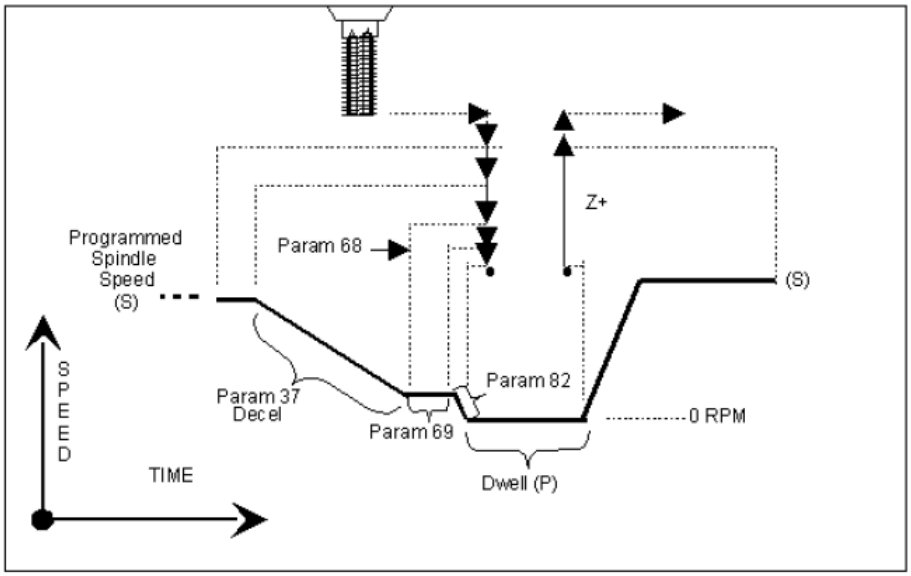
NOTICE Pressing **CYCLE CANCEL** while the tap is in the hole will very probably break the tap or strip the threads in the tap hole. However, do so if it is an emergency.

9.29.1 Tech Tip – How to Set Up 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



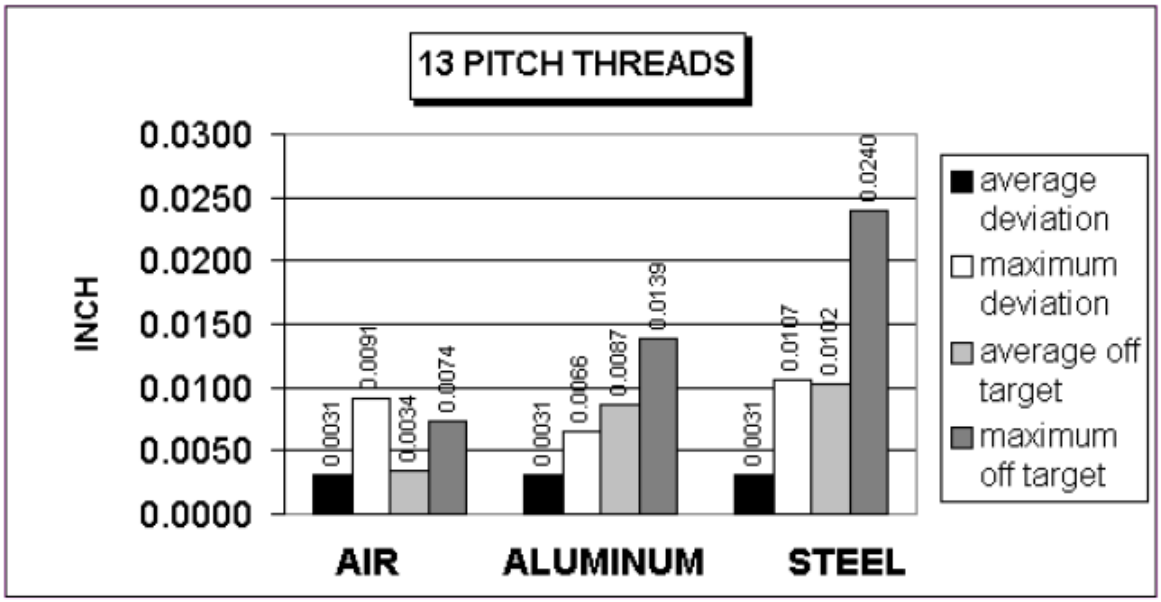
List of Rigid Tapping Setup Parameters

See Chapter 11 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

Draft: January 11, 2024

9.29.2 Graphic Representation of Test Results for Precision



Draft: January 11, 2024

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. The coolant used was water-based fluid. 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 role in performance also. It was found through our testing that if we changed one physical parameter (i.e. using a tapping oil instead of water-based 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.

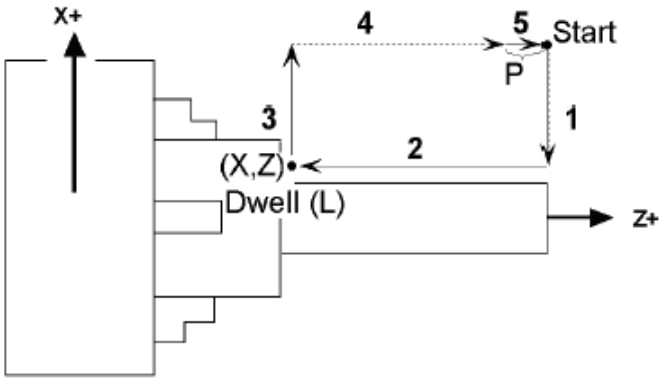
9.30 G85 – Boring

G85 is used to bore a hole so that a smooth finish may be acquired. The tool will feed into the depth at the specified feed rate and retract back out at the same feed rate.

9.31 G90 – Outside/Inside Diameter Cutting Cycle

G90 sets the outer/inner diameter cutting cycle command. These diameters can be specified along straight cuts or diagonal/taper cuts.

9.31.1 Straight Cutting



The general form of the Straight Cutting Cycle is as follows:

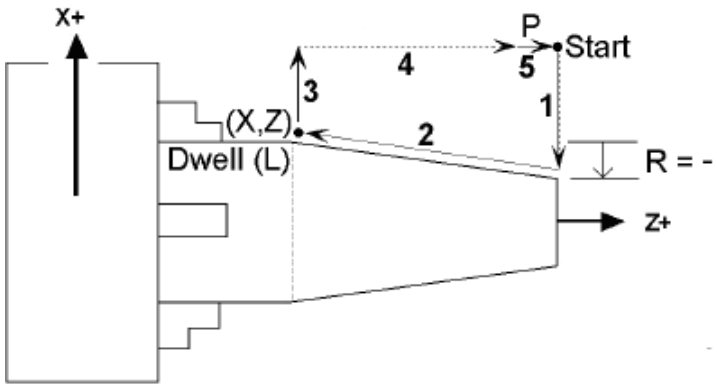
G90 X_ Z_ P_ L_

In incremental programming form, the X and Z can be substituted with U and W. Note that X (or U) is affected by the radius/diameter programming mode (see Parameter 55). The optional parameter P specifies the length of the return feed move (Segment 5 in the illustration above). This cycle behaves differently depending on whether a non-zero P is specified or not. If P does not exist or is 0, then Segments 1 and 4 will be rapid moves, Segments 2 and 3 will be feed rate moves, and Segment 5 will be omitted. If P does exist and is non-zero, then Segment 4 will be a rapid move and all the other segments will be feed rate moves. The optional Parameter L specifies a dwell time between Segments 2 and 3.

Example:

```
G00 X2.5 Z-1.0              ; rapid to start point
G90 X1.5 Z-4.0 F0.5 L1.5 ; G90 cycle with 1.5 sec dwell at X1.5 Z-4.0
```

9.31.2 Taper Cutting



The general form of the Taper Cutting Cycle is as follows:

G90 X_ Z_ R_ P_ L_

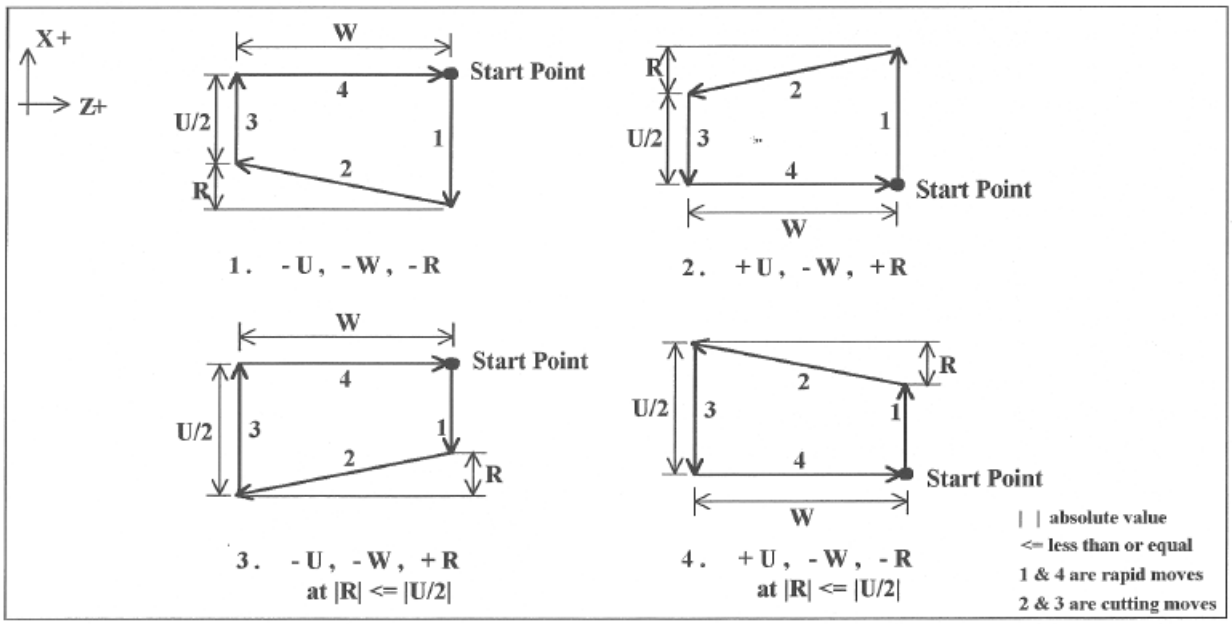
Draft: January 11, 2024

This is actually the same as the Straight Cutting cycle (mentioned above), but with the addition of the R parameter. Parameters P and L are optional. Taper is determined by offsetting the point between Segments 1 and 2 on the X-coordinate by the incremental amount specified by the R parameter. Note that R is unaffected by the radius/diameter programming mode, but X is (see Parameter 55). All the other parameters have the same meaning as those of the Straight Cutting cycle (mentioned above).

Example:

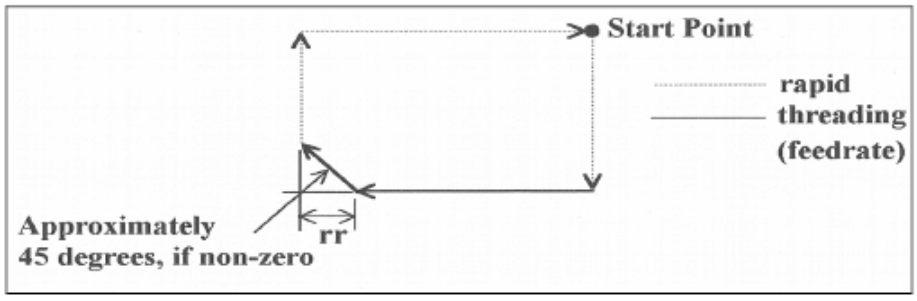
```
G00 X2.5 Z-1.0              ; rapid to start point
G90 X1.5 Z-4.0 R-0.25 F0.5 ; Tapered G90 cycle
```

The following table shows the relationship between the tool paths and the signs of U, W, and R during incremental programming when performing taper cutting.



9.32 G92 – Thread Cutting Cycle

G92 sets the thread cutting cycle command. This cycle can be specified for straight thread cutting or taper thread cutting. In incremental programming, the signs of U and W will depend on the direction of the tool path when approaching the work-piece. That is, if the cutter moves in the negative X direction, then the value of U will be negative.

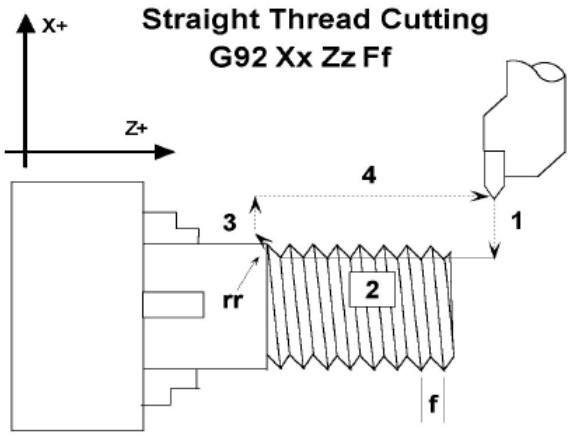


G92 is similar to G32 in that X & Z indicate the endpoint of the cut and F indicates the thread lead and X & Z are

Draft: January 11, 2024

slaved to the spindle. The chamfering amount, rr, which is selected by Parameter 49 (see Chapter 11), is a chamfer distance is rr times the thread lead.

9.32.1 Straight Thread Cutting



1, 3, and 4 are rapid moves. 2 is cutting move

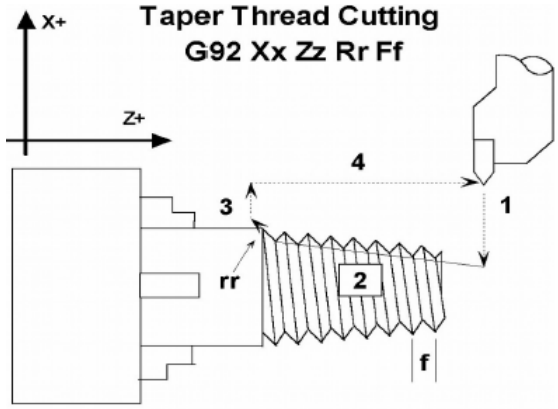
In this cycle, the cutter moves to the diameter indicated by X and threads in a straight line to the depth or length indicated by Z.

In the example below, the cutter first rapids to the start point located at X2.5Z-1, then rapids down to X2 at the same Z, and then cuts with the specified lead to Z-3. At Z-3, the cutter pulls out of the part by the amount of the chamfering distance, then rapids back up to X2.5 and returns to the start point.

Example:

```
G00 X2.5 Z-1.0
G92 X2.0 Z-3.0 F.1
```

9.32.2 Taper Thread Cutting



1, 3, and 4 are rapid moves. 2 is cutting move.

In this cycle, the cutter threads diagonally to the diameter and depth indicated by X and Z, respectively. The value of R will dictate the value of the starting diameters. A negative R will make the ending diameter equal to X and the starting diameter equal to X minus twice the absolute value of R (X-2|R|). A positive R will make the ending diameter equal to X and the starting diameter equal to X plus twice the value of R (X+2R).

In the example below, the cutter first rapids to the start point located at X3.5 Z-1. It then rapids down to X2.5, the inner diameter, at the same Z, and cuts with the specified lead to Z-3. At Z-3 the value of the part the amount of the chamfering distance, then rapids back up to X2.5 and returns to the start point.

Example:

```
G00 X3.5 Z-1.0
G92 X2.5 Z-3.0 R-0.25 F.1
```

9.32.3 Multiple Thread Leads

This is done by using the formula:

2nd thread lead start point = previous thread lead start point + ((1/TPI) / # of leads)

Draft: January 11, 2024

Example:

We want to produce a triple lead thread with a thread lead of ten Threads Per Inch (TPI). The start point for the first thread lead is 0.1000 from the face of the material being threaded.

Thread lead #1 start point = 0.1000.

Thread lead #2 start point = 0.1000 + ((1/10)/3) = 0.1333.

Thread lead #3 start point = 0.1333 + ((1/10)/3) = 0.1666.

9.33 G96 & G97 – Constant Surface Speed Control & Cancel

G96 sets the mode for constant surface speed control in feet/min (sfm) or meters/min. S-values are assumed as surface speed. When CSS is active, the spindle speed changes as the X-position changes, to maintain a constant linear velocity at the tool tip. No matter how close X gets to X0, the spindle speed will not exceed the speed set with G50 or the machine's maximum spindle speed, whichever is less. G97 cancels the constant surface speed control.

Example:

```
G96 S800          ; sets constant surface speed to 800 feet/min
G01 X1 Z-3 F0.1
G97 S1200         ; cancels constant surface speed and sets spindle speed to 1200 rpm
```

9.34 G98 – Feed per Minute

G98 sets the cutting feed rate mode in units/minute. There are no associated parameters.

9.35 G99 – Feed per Revolution

G99 sets the cutting feed rate mode in units/rev. There are no associated parameters.

Draft: January 11, 2024

10 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.

10.1 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)
M26	Set Axis Home
M29	Set Tap Mode for G84
M30	Restart Program
M41, M42, M43	Select Spindle Gear Range (Macros)
M50	C Axis Disable
M51	C Axis Enable (Optional)
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
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

Draft: January 11, 2024

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
M151	Unwind C axis
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
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.

Draft: January 11, 2024

10.2 Macro M-functions (Custom M-functions)

Most T-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:\cnct` 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 a 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 9); 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.

Draft: January 11, 2024

10.3 M00 – Stop for Operator

Motion stops and the Operator is prompted to press the **CYCLE START** button to continue.

10.4 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.

10.5 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.

10.6 M03 – Spindle On Clockwise

M3 requests the PLC to start the spindle in the clockwise direction.

Default action:

```
M95/2
M94/1
```

10.7 M04 – Spindle On Counter-clockwise

M4 requests the PLC to start the spindle in the counter-clockwise direction.

Default action:

```
M95/1
M94/2
```

10.8 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
```

Draft: January 11, 2024

10.9 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 11).

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.

10.10 M07 – Mist Coolant On

M7 causes the PLC to start the mist coolant system.

Default action:

```
M95/3
M94/5
```

10.11 M08 – Flood Coolant On

M8 causes the PLC to start the flood coolant system.

Default action:

```
M95/5
M94/3
```

10.12 M09 – Coolant Off

M9 causes the PLC to stop the coolant system.

Default action:

```
M95/3/5
```

10.13 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.

10.14 M11 – Clamp Off

M11 causes the PLC to release the clamp–also referred to as a ‘chuck’.

Default action:

```
M94/4
```

10.15 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 the 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 11 for more information.

Draft: January 11, 2024

10.16 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.

10.17 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 the 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
```

10.18 M29 – Set Tap Mode for G84

M29 sets the tap mode for G84; either right-hand or left-hand tapping. Right-hand tap mode is the initial default at job start-up. If Left-hand tap mode is required, M29 and P1 need to be specified on the same line.

Tap Mode	Command
CW (Right-hand)	M29
CCW (Left-hand)	M29 P1

10.19 M30 – Restart Program

Ends the program and restarts it from the first line. M30 as an end of program code is an industry-standard. The Operator is prompted to press the **CYCLE START** button to continue. This has the same functionality as M2.

10.20 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-function	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.

Draft: January 11, 2024

10.21 M50 – C-axis Disable

M50 is the command to disable the C-axis and is a locked software option. When the C-axis is disabled, no axis label will be present on the screen and encoder information for the C-axis is ignored. In order for the M50 command to work, the 3rd- or 4th-axis label must be set to 'C' with the associated parameter (93 for 3rd-axis and 94 for 4th-axis) set for C-axis operation. In practical applications, the default behavior for the M50 command is usually modified using a custom `mfunc50.mac` program.

Example: `mfunc50.mac`**:**

```
M95/9 ; Switch to speed mode
M50   ; Perform the default actions for C axis disable
```

10.22 M51 – C-axis Enable (Optional)

M51 is the command to enable the C-axis and is locked as a software option. When C-axis is enabled, the C-axis label will be present on the DRO, and encoder information for the C-axis is used to determine the position of the C-axis. In order for the M51 command to work, the 3rd- or 4th-axis label must be set to 'C' with the associated parameter (93 for 3rd-axis and 94 for 4th-axis) set for C-axis operation. In practical applications, the default behavior for the M51 command is modified using a custom `mfunc51.mac` program to ensure that the spindle has stopped before the C-axis is enabled.

Example: `mfunc51.mac`**:**

```
G97     ; Turn off CSS (constant surface speed
M3 S0   ; Turn off spindle
M101/9  ; Wait for zero speed signal form inverter on INP9
M94/9   ; Switch to torque mode
M51     ; Perform the default actions for C axis enable
M151    ; Unwind C-axis position
```

Note in the above examples for M50 and M51 where the M95/9 (turn off INP41) and M94/9 (turn on INP41) commands are used, it is assumed that the PLC program, conditioned upon the state of INP41, has been modified to output the appropriate hardware signals required to switch between speed and torque mode.

10.23 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.

Draft: January 11, 2024

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
```

10.24 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.

Draft: January 11, 2024

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
```

10.25 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.

10.26 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.

Draft: January 11, 2024

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.

10.27 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.

10.28 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.

Draft: January 11, 2024

10.29 M100 – Wait for PLC Bit (Open, Off, Reset)

10.30 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.

10.31 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.

10.32 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
```

Draft: January 11, 2024

10.33 M104 – Cancel Programmed Action Timer

M104 stops the timer started by the last M103 executed.

10.34 M105 – Move Minus to Switch

M105 moves the requested axis in the minus direction at the current feed rate 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
```

10.35 M106 – Move Plus to Switch

M106 moves the requested axis in the plus direction at the current feed rate 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
```

10.36 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
```

10.37 M108 – Enable Override Controls

M108 re-enables the feed rate override and/or spindle speed override controls if they were disabled with M109. A parameter of “1” indicates the feed rate 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
```

10.38 M109 – Disable Override Controls

M109 disables the feed rate override and/or spindle speed override controls. It may be used before tapping with G85 to assure that the machine runs at the programmed feed rate 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
```

10.39 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*

Draft: January 11, 2024

Where:

nnn	is either 115, 116, 125, or 126.
Axis	is a valid axis label, i.e., X, Z, etc.
pos	is an optional position
P	is a PLC bit number, which can be negative.
*F**	is a feed rate (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.

Draft: January 11, 2024

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 a DSP-type probe.

10.40 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, which requires the ‘.’ to be present. Assigning #301 = ".cnc" and executing M120 "#300#301" does not work and generates a "Could not open file" error message.

10.41 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.

10.42 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, four decimal places in inch mode, three 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 11). 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 Z-0.5678 ; comment
M122 /X L1      -> X-1.5000
M122 /X         -> X-1.5000 X-2.0000
```

10.43 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 (four decimal places in inches, three 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 11).

Draft: January 11, 2024

Example: M-function and Sample Output:

```
M123           ; 1.2345 -> 1.2345
M123 P\#A      ; first macro argument -> first macro argument
M123 Q0 P1.23 ; -> 1
```

10.44 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.

10.45 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.

Example:

```
M121 ‘‘testdata.dat’’
M123
M127
```

If you opened testdata.dat you would see: Day of the week, Month, day, time, and year. (i.e. `;Wed Aug 29 11:56:57` 2007)

10.46 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
```

10.47 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 two lines:

```
M121 "output.txt"
M129
```

If you opened the `output.txt` file you would see: `c:\cncm\ncfiles\job.cnc`.

Draft: January 11, 2024

10.48 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).

10.49 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.

Draft: January 11, 2024

10.50 M151 – Unwind C-axis

This M-function will reset the C-axis position to less than one revolution of the C-axis ($<$ 360 degrees).

Example: (M51):

```
G97     ; Turn off CSS (constant surface speed)
M3 S0   ; Turn off spindle
M101/9  ; Wait for zero speed signal form inverter on INP9
M94/9   ; Switch to torque mode
M51     ; Perform the default actions for C axis enable
M151    ; Unwind C-axis position
```

Note in the above examples for M50 and M51 where the M95/9 (turn off INP41) and M94/9 (turn on INP41) commands are used, it is assumed that the PLC program, conditioned upon the state of INP41, has been modified to output the appropriate hardware signals required to switch between speed and torque mode.

⚠ **WARNING** The spindle must be stopped before issuing the M151 or unpredictable positions can result.

10.51 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.

10.51.1 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.

10.51.2 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.

Draft: January 11, 2024

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
```

10.51.3 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?."
```

10.52 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.

10.53 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
```

10.54 M224 – Prompt for Operator Input Using Formatted String

The M224 command displays a *formatted-string* and then accepts user input. The syntax is:

Draft: January 11, 2024

```
M224 /num lvalue_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
```

10.55 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
```

10.56 M290 – Digitize Profile (Optional)

This performs a two 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. M290 expects that a file is already open with M120/M121 (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.

Draft: January 11, 2024

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).

10.57 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:

```
M94/1             ; request spindle start
M101/50005        ; wait for up to speed signal
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).

10.58 M1000–M1015 – Graphing Color for Feed Rate Movement

When a CNC program is graphed (**F8** from the Main Screen), feed rate movements are normally plotted using the color yellow. This color setting can be changed to another color as stated in the chart below.

Draft: January 11, 2024

M-code	Feed Rate 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 feed rate graphing color can be used as a method of 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.

Draft: January 11, 2024

11 Configuration

11.0.1 General

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.

⚠ **WARNING** Some of the data, if corrupt or incorrect, could cause personal injury or machine damage.

11.0.2 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 that 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.

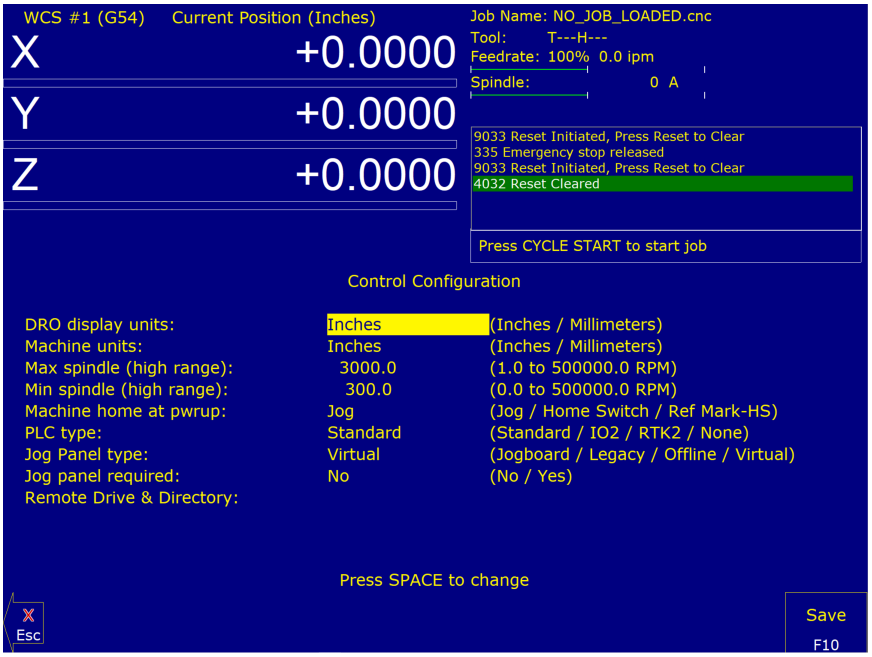
11.0.3 Control Configuration

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

Draft: January 11, 2024

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. Either 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, the message “Press SPACE to change” appears at the bottom of the screen. This 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 feed rates. If ‘Inches’ is selected, all feed rates 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 (VFD) controller . All spindle speeds entered into a a CNC program are sent to the PLC as percentages of this

Draft: January 11, 2024

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 the 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 Machine Configuration designates 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 (an RJ-45 Ethernet connection). The network drive must be mapped in `cnc.net`.

11.1 User Specified Paths

The Operator can 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:\icn_lath\
```

Draft: January 11, 2024

```
icn_post_path=C:\cnct\ncfiles\
```

Path Tag	Purpose of Path
INTERCON_PATH	The main directory containing *.lth files
ICN_POST_PATH	The directory where INTERCON places *.cnc files created when posting *.lth files.

11.2 Machine Configuration

Pressing **F2 – Machine** from the configuration screen will display the machine configuration screen. The machine configuration screen provides the Operator with a method of changing machine-dependent data.

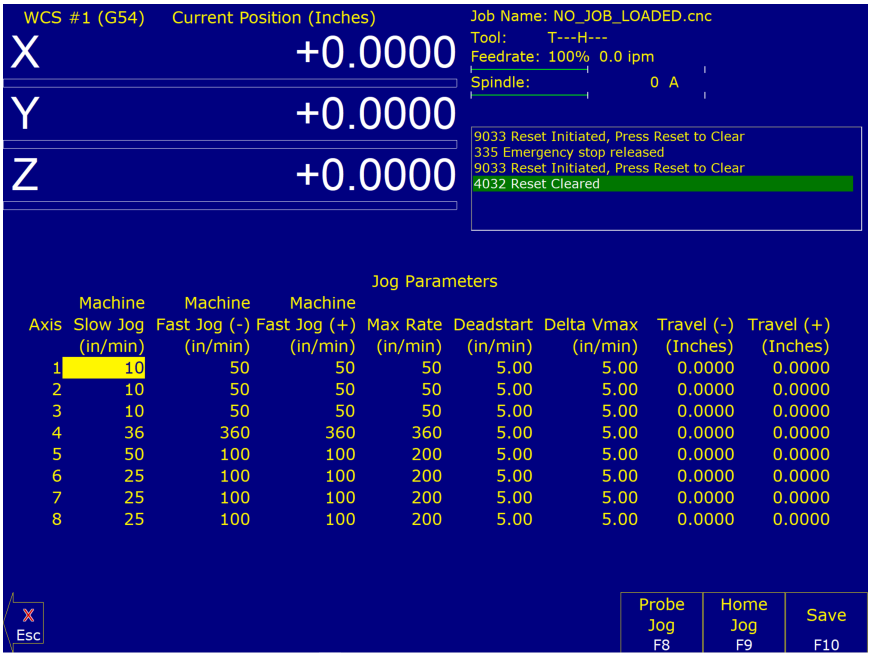
To change the Jog or Motor parameters, press **F1 – Jog** or **F2 – Motor** to select the Jog or Motor screens. Use the arrow keys to move the cursor and select the desired field. Type the new value and press **ENTER** or press the **SPACE** bar to toggle. When finished editing, press **F10 – Save** to save any changes that were made. To discard the changes and restore the previous values, press **ESC**. Pressing **ESC** again will return you to the previous screen (Configuration).

Note: Although X appears on the first line of the DRO and Z appears on the second, their order is reversed on all configuration menus. X is axis 2, and Z is axis 1.

Draft: January 11, 2024

11.2.1 F1 – Jog

This screen contains jog and feed rate information. See the figure below:



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. The fast jog rate cannot be set to a value greater than the maximum rate.

Max Rate: Determines the maximum feed rate of each individual axis. The feed rate on each axis can never exceed the Max Rate, even if the feed rate override knob on the front panel is turned up above 100%. (Also see Parameter 38 – Multi-axis Max Feed Rate, which limits the feed rate 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 that 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.

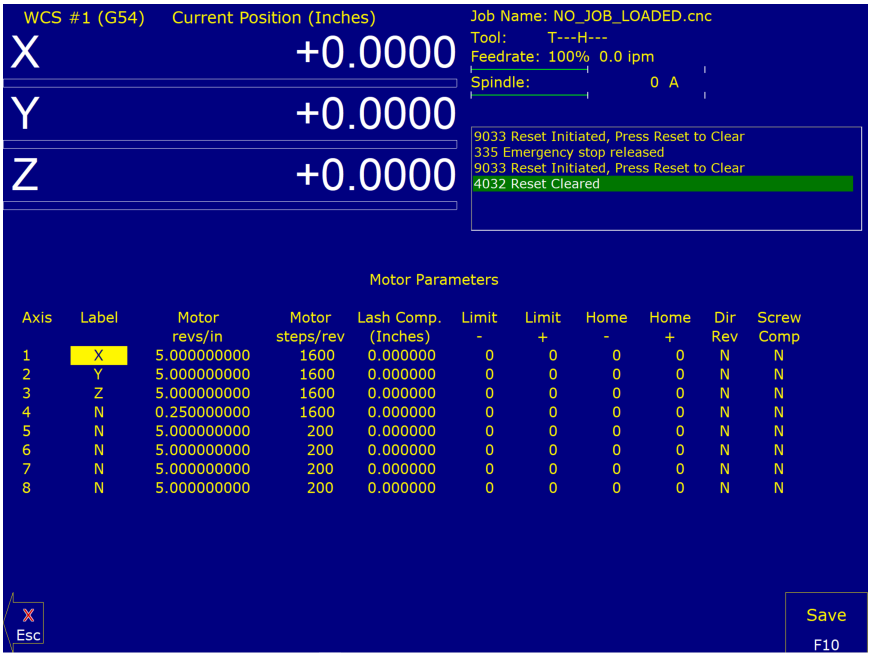
Travel (-): The maximum distance that the axis can travel in the minus direction from the home position. Set this parameter to create a software limit that stops the axis before the fixture or tool collides with the limit switches or hard stops.

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 limit switch/hard stop.

Draft: January 11, 2024

11.2.2 F2 – Motor Parameters

This screen contains information about the motors, ballscrews, and switches installed on your machine. See the figure below:



⚠ 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 two axes should always be Z and X. The unused entries should be labeled N.

Note: Although the 3rd- through 8th-axes are available in the lathe software, they may be used for special tool changer and C-axis applications. For C -axis applications, the label must be set to C and the corresponding motor Parameter (93 or 94) must have the C-axis bit on.

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 is added to and works in conjunction with Screw Compensation (see below). Consult your machine manual or T-Series Service Manual for instructions on measuring backlash.

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.

11.2.3 F3 – Find Home

Press **F3 – Find Home** to move an axis to its plus or minus home switch.

Draft: January 11, 2024

11.2.4 F4 – Set Home

Press **F4 – Set Home** to set the 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.

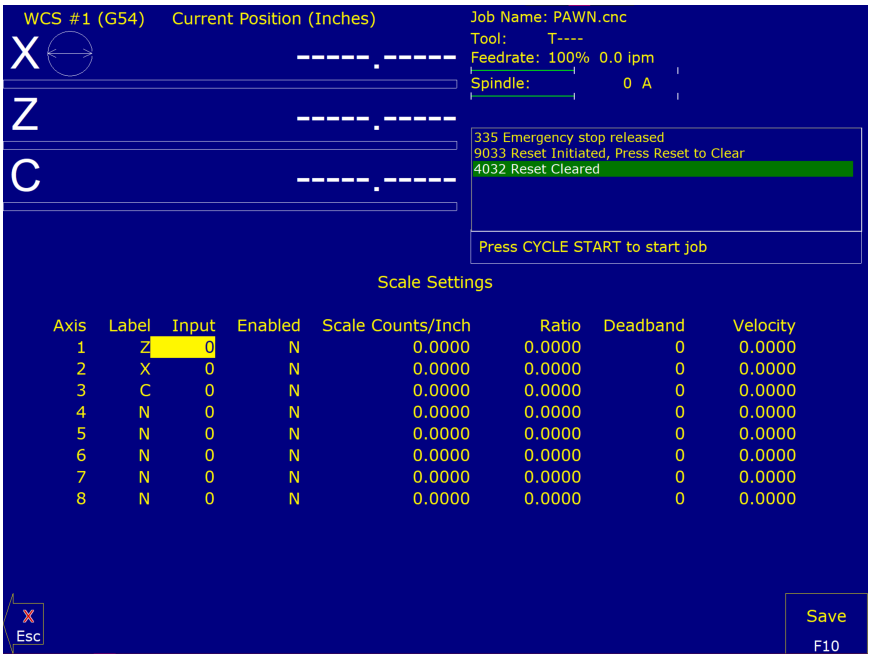
11.2.5 F5 – M Comp

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.

11.2.6 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, these can be used for scale feedback.

Enabled "Y" enables the scale and "N" disables the scale. Use the spacebar to toggle between the two choices and remember to choose **F10** to save.

Draft: January 11, 2024

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/minute speed, you need to convert it. Due to the nature of scale feedback, it is inherently an oscillator. By adjusting the Deadband and Velocity, that oscillation can be kept to a minimum. If there are oscillations, you typically want to decrease the Velocity.

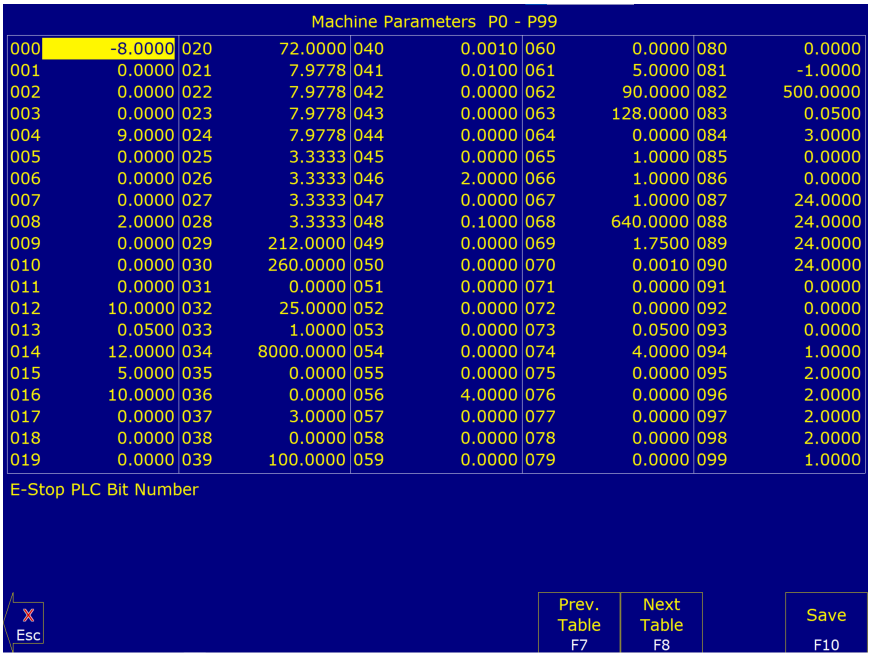
Scale Indicator Changing the Input, Enabled, or Scale Counts/Unit fields will cause scale compensation to be temporarily disabled. Scale compensation is also temporarily disabled during homing moves. Even though the scale is enabled in the menu, scale compensation will be disabled until the axis is rehomed. 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.

Draft: January 11, 2024

11.3 Machine Parameters (F3 – Parms from Configuration)



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 a method of changing various parameters that are used by the control. Altogether, there is access to 1000 parameters spread across 10 tables. Each table gives access to 100 parameters at a time. Navigation between these tables is done using the following keys: **F7 – Previous Table** and **F8 – Next Table**. The title at the top shows which table is currently displayed. 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 in the new value and press **ENTER**. When done editing the fields, press **F10 – Save** to accept any changes that have been made and save them. Note that **F10 – Save** is a single operation that will save all changes in every table that was 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.

11.3.1 Bit-mapped Parameters

Certain control parameters are defined by bit-mapped values. The Acorn and AcornSix include 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:

Draft: January 11, 2024

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 this manual, we include the values to simplify 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 a 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	Unused for Lathe	set to 0
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	Machine and Jog key orientation	0
2	G-code Interpretation Control and Slaving Rotary axis feed rate	0
3	Modal Tool and Height Offset Control	0

Draft: January 11, 2024

Parameter	Definition	Default setting
4	Remote File Loading Flag & Advanced File Ops	0
5	Machine Home/Startup Setup	0
6	Auto Tool Changer Installed	0
7	Display Colors	0
8	Available Coolant Systems	2
9	Display Language	0
10	Macro M-function Control/Probe Stop Handling	0
11	DP4 PLC Input Number and Contact State	15
12	Touch Probe Tool Number	0
13	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	Maximum Probing Distance	10" / 254 mm
17	Detector Location Return Point	0
18	PLC Input Spindle Inhibit Parameter	0
19	MPG modes	0
20–30	Motor Temperature Estimation	Refer to text
20	Ambient Temperature	72 °F / 22 °C
21	Motor Heating Coefficients for axes 1	Refer to text
22	Motor Heating Coefficients for axes 2	Refer to text
23	Motor Heating Coefficients for axes 3	Refer to text
24	Motor Heating Coefficients for axes 4	Refer to text
25	Motor Cooling Coefficients for axes 1	Refer to text
26	Motor Cooling Coefficients for axes 2	Refer to text
27	Motor Cooling Coefficients for axes 3	Refer to text
28	Motor Cooling Coefficients for axes 4	Refer to text
29	Overheating Warning Temperature	150 °F / 65.5 °C
30	Job Cancellation 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 Feed Rate	0
39	Feed Rate Override Percentage 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	G71/G72 Depth of Cut	0.001
44	G71/G72 Escape Amount	0.001
45	G74 X-axis Relief Amount	0
46	G75 Z-axis Relief Amount	0
47	G73 Repeat Count	1

Draft: January 11, 2024

Parameter	Definition	Default setting
48	Grid Digitize Patch Playback Z rapid clearance amount	.1
49	Thread Chamfer Amount	0
50	G76 Finish Count	1
51	G76 Thread Angle	0
52	G76 Minimum Cutting Depth	0.001
53	G76 Finish Allowance	0.0
55	Radius / Diameter mode	0
56	Feedrate Override Display Properties / Inverse Time Interpolation	0
57	Use Generic Load Meter Data from PLC	0
58	G71/G72 Profile Rough Cut Escape Angle	
59	Tool Check Options	0
61–62	Stall Detection Parameters	
61	High Power Stall Timeout	0.5
62	High Power Stall PID Limit	115
63	High Power Idle PID Multiplier	1.5
64	Fourth/Fifth-axis Pairing	0
65–67	Spindle Gear Ratios	
65	Low Range Gear Ratio	1.0
66	Medium-low Range Gear Ratio	1.0
67	Medium-high Range Gear Ratio	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 M-function Options	0
73	Peck Drill Retract Amount	0.05
74	M-function executed at bottom of tapping cycle	4
75	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 Message Frequency	0
81	Air Drill M-function	-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
87–90	Autotune Ka Performance parameters for axes 1, 2, 3, 4	
87	Autotune Ka Performance parameters for axes 1	0
88	Autotune Ka Performance parameters for axes 2	0
89	Autotune Ka Performance parameters for axes 3	0
90	Autotune Ka Performance parameters for axes 4	0
91–94	Axis Properties for axes 1, 2, 3, 4	0
91	Axis Properties for axes 1	0
92	Axis Properties for axes 2	0
93	Axis Properties for axes 3	0

Draft: January 11, 2024

Parameter	Definition	Default setting
94	Axis Properties for axes 4	0
95–98	Autotune Move Distance / Auto Delay Calculation Move Distance for axes 1, 2, 3, 4	2" / 50.8 mm
95	Autotune Move Distance / Auto Delay Calculation Move Distance for axes 1	2" / 50.8 mm
96	Autotune Move Distance / Auto Delay Calculation Move Distance for axes 2	2" / 50.8 mm
97	Autotune Move Distance / Auto Delay Calculation Move Distance for axes 3	2" / 50.8 mm
98	Autotune Move Distance / Auto Delay Calculation Move Distance for axes 4	2" / 50.8 mm
99	Cutter Compensation Look-ahead	6
100–109	Intercon parameters	
100	Intercon comment generation	0
101	Intercon clearance amount	0.1
102	Intercon spindle coolant delay	3.0
103	Intercon corner feed rate override	50.0
104	Intercon modal line parameters	0
105	Intercon modal arc parameters	0
106	Intercon modal drilling cycle parameters	0
107	Intercon chamfer blend	radius * 0.01
108	Intercon polar display	1
109	Intercon modal display	1
110	Wizard Password	0
111–115	Intercon parameters	
111	Intercon no spindle stop during tool change	0
112	Intercon no coolant stop during tool change	0
113	Hide Intercon Menu	0
114	Intercon use G28 during tool change	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
128	Dry Run PLC Bit	
129	Dry Run Feed Rate	
130–131	3rd/4th axis on/off selection	
130	3rd-axis on/off selection	0
131	4th-axis on/off selection	0
132–135	Motor Heating Coefficients for axes 5–8	Refer to text
132	Motor Heating Coefficients for axes 5	Refer to text
133	Motor Heating Coefficients for axes 6	Refer to text
134	Motor Heating Coefficients for axes 7	Refer to text
135	Motor Heating Coefficients for axes 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

Draft: January 11, 2024

Parameter	Definition	Default setting
140	Message log priority level	1
141	Maximum message log lines	100
142	Message log trim amount	100
143	DRO properties (load meter, DTG)	0
144	Comparison rounding	0
145	Advanced macro properties (fast branching)	0
146	Feed hold threshold for feed rate override	0
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 and radial digitizing	0
153	Probe Protection Enabled	
154	Touch Screen Options	0
155	Probe Type	0
156–159	Autotune Move Distance / Auto Delay Calculation Move Distance for axes 5, 6, 7, 8	2" / 50.8 mm
156	Autotune Move Distance / Auto Delay Calculation Move Distance for axes 5	2" / 50.8 mm
157	Autotune Move Distance / Auto Delay Calculation Move Distance for axes 6	2" / 50.8 mm
158	Autotune Move Distance / Auto Delay Calculation Move Distance for axes 7	2" / 50.8 mm
159	Autotune Move Distance / Auto Delay Calculation Move Distance for axes 8	2" / 50.8 mm
160	Enhanced ATC	0
163	Gang Tooling	0
164	ATC Feature	0
165	Acceleration/Deceleration Options	0
166–169	Axis Properties for axes 5, 6, 7, 8	0
166	Axis Properties for axes 5	0
167	Axis Properties for axes 6	0
168	Axis Properties for axes 7	0
169	Axis Properties for axes 8	0
170–179	PLC Parameter	
170	Enable Keyboard Jogging and Set Feed Rate Override Control	
178	PLC I/O configuration	0
179	Lube Pump Operation	0
180	Clear Home Switch Distance	0
186	Probe Stuck retry disable	0
187	Hard Stop Homing	0
188–199	Aux Key Functions	0
200–207	OPTIC 4 Tach Volts Per 1000 RPM	0
208–215	MPU-based Lash/Screw Compensation 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
220–231	Smoothing Parameters	Refer to text
236–239	Motor Cooling Coefficients for axes 5–8	Refer to text
236	Motor Cooling Coefficients for axes 5	Refer to text
237	Motor Cooling Coefficients for axes 6	Refer to text
238	Motor Cooling Coefficients for axes 7	Refer to text

Draft: January 11, 2024

Parameter	Definition	Default setting
239	Motor Cooling Coefficients for axes 8	Refer to text
240	Rigid Tapping and Threading Accel Rate Distance	0
241	Rigid Tapping and Threading Rotational Step Size (Degrees)	10.00000
242	Minimum Angle Threshold for application of Accel/decel in Threading moves	0
243	Threading Control	0
244	Tool Touch-off device PLC input number	0
245	G71, G72, G74, G75, G76 D/P/Q “implied float” re-interpretation threshold for Inch units	99
246	G71, G72, G74, G75, G76 D/P/Q “implied float” re-interpretation threshold for MM units	99
247	G70 Multiple Pass Behavior Suppression	0
248	Tool Wear Adjustment magnitude limit	0.2" / 5.08 mm
252–255	Autotune Ka Performance parameters for axes 5, 6, 7, 8	0
252	Autotune Ka Performance parameters for axes 5	0
253	Autotune Ka Performance parameters for axes 6	0
254	Autotune Ka Performance parameters for axes 7	0
255	Autotune Ka Performance parameters for axes 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	1
261	Precision Mode Standoff Tolerance Percentage	0
262	Precision Mode Standoff Delay Time	
263	DRO Display Precision	4.0
264	Rapid Override	0
270–271	XY Skew Correction	0
270	Skew Correction: X Skew offset from Trusted/Intended X	0
271	Skew Correction: Trusted/Intended Y	0
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
284–291	Brake Resistor Wattage for ACDC Drives 1–8	50
284	Brake Resistor Wattage for ACDC Drives 1	
285	Brake Resistor Wattage for ACDC Drives 2	
286	Brake Resistor Wattage for ACDC Drives 3	
287	Brake Resistor Wattage for ACDC Drives 4	
288	Brake Resistor Wattage for ACDC Drives 5	
289	Brake Resistor Wattage for ACDC Drives 6	
290	Brake Resistor Wattage for ACDC Drives 7	
291	Brake Resistor Wattage for ACDC Drives 8	
292–295	Aux Key Functions 13–16	
292	Aux Key Function 13	
293	Aux Key Function 14	
294	Aux Key Function 15	
295	Aux Key Function 16	
299	Report Option Bits	0
300–307	Drive assignment to Axes 1–8	Refer to text

Draft: January 11, 2024

Parameter	Definition	Default setting
300	Drive assignment to Axes 1	Refer to text
301	Drive assignment to Axes 2	Refer to text
302	Drive assignment to Axes 3	Refer to text
303	Drive assignment to Axes 4	Refer to text
304	Drive assignment to Axes 5	Refer to text
305	Drive assignment to Axes 6	Refer to text
306	Drive assignment to Axes 7	Refer to text
307	Drive assignment to Axes 8	Refer to text
308–315	Encoder assignment to Axes 1–8	Refer to text
308	Encoder assignment to Axes 1	Refer to text
309	Encoder assignment to Axes 2	Refer to text
310	Encoder assignment to Axes 3	Refer to text
311	Encoder assignment to Axes 4	Refer to text
312	Encoder assignment to Axes 5	Refer to text
313	Encoder assignment to Axes 6	Refer to text
314	Encoder assignment to Axes 7	Refer to text
315	Encoder assignment to Axes 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
324–331	Axis Boxcar size	0
332–335	Encoder error suppression	0
332	Suppress encoder differential faults	0
333	Suppress encoder differential error messages	0
334	Suppress encoder quadrature faults	0
335	Suppress encoder quadrature error messages	0
336–339	Motor torque estimation for velocity mode drives	Refer to text
336	Overall gain setting	
337	Absolute error gain	
338	Error sum gain	
339	Delta error gain	
340–347	Precision Mode delay (in milliseconds) for axes 1–8	0
340	Precision Mode delay (in milliseconds) for axes 1	0
341	Precision Mode delay (in milliseconds) for axes 2	0
342	Precision Mode delay (in milliseconds) for axes 3	0
343	Precision Mode delay (in milliseconds) for axes 4	0
344	Precision Mode delay (in milliseconds) for axes 5	0
345	Precision Mode delay (in milliseconds) for axes 6	0
346	Precision Mode delay (in milliseconds) for axes 7	0
347	Precision Mode delay (in milliseconds) for axes 8	0
348	MPG/Handwheel Encoder Input 1	15
349	MPG/Handwheel Detents-per-revolution 1	100
350	MPG/Handwheel Encoder Counts-per-revolution 1	400
351	MPG/Handwheel Encoder Input 2	0
352	MPG/Handwheel Detents-per-revolution 2	100

Draft: January 11, 2024

Parameter	Definition	Default setting
353	MPG/Handwheel Encoder Counts-per-revolution 2	400
354	MPG/Handwheel Encoder Input 3	0
355	MPG/Handwheel Detents-per-revolution 3	100
356	MPG/Handwheel Encoder Counts-per-revolution 3	400
357–364	Axis Drive Max RPM for Axes 1–8	0
357	Axis Drive Max RPM for Axes 1	0
358	Axis Drive Max RPM for Axes 2	0
359	Axis Drive Max RPM for Axes 3	0
360	Axis Drive Max RPM for Axes 4	0
361	Axis Drive Max RPM for Axes 5	0
362	Axis Drive Max RPM for Axes 6	0
363	Axis Drive Max RPM for Axes 7	0
364	Axis Drive Max RPM for Axes 8	0
365	Drive power-on delay	0.0
366–367	Probe / TT1 deceleration multiplier	
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
374–379	ACDC Drive Debug Log Settings	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
387–389	Debugging Parameters	0
387	Debug Trace Mask	0
388	Debug Options	0
389	Debug Level	0
392–394	DP-7 parameters	Refer to text
392	DP-7 Pullback Distance	Refer to text
393	DP-7 Pullback Feed Rate	Refer to text
394	DP-7 Measuring Feed Rate	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-Codes	0
409	Probe Type	

Draft: January 11, 2024

Parameter	Definition	Default setting
410	Probe/Tool Touch-off Warning	
411	MPG Type	0
413	Park macro	0
414	Home File Type	
415	Ether1616 Configuration Bits	0
416	Spindle Inhibit	0
417	Preview G-code Before Job	
418	Assign Enter Key to Cycle Start	
419	PLC Worklight and Popup Pin Output	0
420	Analog/PWM 1 Settings	0
421	Clean Fan Filter Reminder	0
422	PLC Diagnostics Display	0
423	Display Scale Position on DRO	0
425	Force Rehoming After EStop Condition	
426	WCS Lockout	
430	RTG Spindle Speed RPM Display	
440	Stopped for Jogging Continue Bit	0
441–443	MPG (1–3) Axis Selection	
441	MPG1 Axis Selection	
442	MPG2 Axis Selection	
443	MPG3 Axis Selection	
450	Inhibit MPG Z-axis Fast Movement	
458	C-Axis Options	0
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	5th-axis Pairing – Slave Axis	0
508	5th-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
700–799	Dedicated Block of Parameters usable in Custom PLC Programs	
800–813	DB25 Pin Selection	Refer to text
800	Acorn DB25 Map Enable	0
801	Acorn Step 1	2
802	Acorn Step 2	4
803	Acorn Step 3	6
804	Acorn Step 4	8
805	Acorn Direction 1	3
806	Acorn Direction 2	5
807	Acorn Direction 3	7
808	Acorn Direction 4	9
809	Acorn PLC Output 1	1

Draft: January 11, 2024

Parameter	Definition	Default setting
810	Acorn PLC Output 2	14
811	Acorn PLC Output 3	16
812	Acorn PLC Output 4	17
813	Acorn DB25 Header Section	0
814	Analog/PWM 2 Settings	4883
815	Inverse Output	
817	PWM	
830	ATC Type	
852	Carousel Skip Count on Reverse	0
853	Distance to travel past tool position	
854	Distance to travel behind tool position	
855	MPG Performance	
856	VCP Jogging State on Acorn Power Up	
900–999	Reserved for Centroid PLC Program Use	
997	Spindle Cooling Fan Delay Timer	
998–999	PLC program parameters	

Draft: January 11, 2024

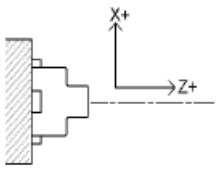
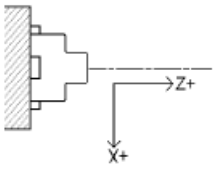
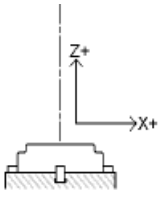
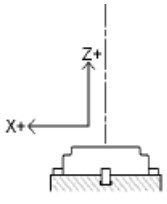
11.3.2 Parameter 0 – E-Stop PLC Bit

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 the 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

11.3.3 Parameter 1 – Machine and Jog key orientation

This parameter controls the orientation of the jog keys and graphics. The default value is 1. When the default value is active, all graphical displays will depict tooling mounted from the back.

Machine Orientation	 Horizontal	 Horizontal	 Vertical	 Vertical
Tool Mount Location (If Not Using Gant Tooling)	Rear	Front	Right	Left
Parameter Value	P1=0	P1=1	P1=8	P1=9

If the X+ and X- jog key inputs need to be exchanged with each other to match the machine orientation, add 2 to one of the parameter values mentioned above. Also, if the X jog keys need to be exchanged with the Z jog keys, add 4. For example, if you have a vertical lathe with tools mounted on the left, the X jog keys are interchanged with the Z jog keys, and the X+/- jog key functions need to be flipped, then the value that needs to be entered into Parameter 1 is 15 (9+4+2).

11.3.4 Parameter 2 – G-code Interpretation Control and Slaving Rotary axis feed rate

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	Unused for Lathe	Set to 0
1	Unused for Lathe	Set to 0
2	Interpret dwell times associated with G4 as milliseconds rather than seconds?	Add 4
3	Slaving rotary axis feed rate to non-rotary axis feed rate. Note that this feature has no effect for movement commands handled by Smoothing (when Parameter 220=1).	Recommended setting = 0
4	Unused for Lathe	Set to 0
5	Prevent rotary-only moves (rotary axis by itself on a line of G-code) from acquiring a remembered slaved rotary feed rate 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).	Recommended Setting = 0
6	Selects whether or not the default M30 will act like a default M2 (Restart Program). If set on, the default M30 will be equivalent to the default M2.	0

11.3.5 Parameter 3 – Modal Tool and Height Offset Control

Bit	Function Description	Value
0	The 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	Unused for Lathe. This bit should be set to 0.	
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	Lathe Intercon Post Tool Offset	Post tool number offset (Add 8)

11.3.6 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 drive letters higher than C. These drives (i.e. drives D, E, F, etc.) are presumed to be network or extra internal drives. To enable the Windows-style File Open dialogue and have CNC12 remember the last opened location, you would set P4=24 (Add 8, Add 16).

Draft: January 11, 2024

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 the default for loading files	Add 4
3	Use the Windows-style File Open dialogue	Add 8
4	The Windows-style File Open dialogue will remember the last opened file location	Add 16

11.3.7 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 the machine home before running jobs.	Add 1
1	(Unused)	—
2	Disable stall detection when the CNC software first starts.	Add 4

Bit 0 of Parameter 5 toggles the requirement to set the machine home before running. If bit 0 of Parameter 5 is set to a value of 0, the machine home must be set before jobs may be run. If bit 0 of Parameter 5 is set to a value of 1, the machine home is not requested or required.

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.

11.3.8 Parameter 6 – Auto Tool Changer Installed

This parameter tells the control whether an Automatic Tool Changer installed on your machine. This field affects the action of the T-codes in your CNC programs. 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

11.3.9 Parameter 7 – Display Colors

This parameter determines what combination of colors will be used for displays. 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 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” from the Color Picker (**F5** in the Utility Menu) is selected.

Draft: January 11, 2024

11.3.10 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

11.3.11 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

11.3.12 Parameter 10 – Macro M-function Control/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 a 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 a controlled deceleration can cause excessive machine vibration.	Decel (Add 4) Immediate Stop (Add Nothing)
3	Unused for Lathe	set to 0
4	(not used) This bit was previously used for controlling the type of deceleration to a 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

Draft: January 11, 2024

11.3.13 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. The 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

11.3.14 Parameter 12 – Touch Probe Tool Number

This parameter is the tool number of the DP4 probe. The 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.

11.3.15 Parameter 13 – Recovery Distance

This parameter is the distance that the probe moves off of 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 by this distance before retrying.

11.3.16 Parameter 14 – Fast Probing Rate

The fast probing rate is used for positioning moves and initial surface detection, and is determined by the machine's response time and permitted probe deflection. The default is 10 in/min. This is a very conservative feed rate. In actuality, 20 to 30 in/min is a good value that will not have a detrimental impact on accuracy in most cases.

11.3.17 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

11.3.18 Parameter 16 – Maximum Probing Distance

This is the maximum distance that the probing moves (M115/M116/M125/M126 and Auto Tool Touch-off cycles) "search" for a surface in a given direction if no travel limits have been entered.The default setting is 10 inches.

Draft: January 11, 2024

A larger value should be entered for the boss and web cycles if measuring very large features.

These settings are conservative measurements that can be used initially for startup purposes, however they can be changed to accommodate your work.

11.3.19 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 then touch the tool off of the TT-1 Tool detector.

Entry	Return Point
0	None
1	G28
2	G30
3	G30P3
4	G30P4

A value of zero indicates that the tool detector is not permanently-mounted; the automatic tool measurement will be performed without X/Y axis movement.

Draft: January 11, 2024

11.3.20 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 that the Spindle Inhibit is wired to. When this parameter is set, Digitizing and Probing cycles will not be 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 value 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

11.3.21 Parameter 19 – MPG modes

The MPG is a hand-held device that is used as an alternative way of jogging the machine. This parameter defines the MPG’s mode of operation.

Bit	Function Description	Value
1	Enable ×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.

11.3.22 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 and 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 should be disabled (by setting all heating and cooling coefficients to 0).

Draft: January 11, 2024

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

11.3.23 Parameter 31 – Legacy SPIN232 Com Port

For values 1–255, CNC12 will attempt to open the respective 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.

11.3.24 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

Draft: January 11, 2024

11.3.25 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 the high range. The ratio between high range and lower ranges is established by the gear ratio parameters (65–67).

11.3.26 Parameter 34 – Spindle Encoder Counts/Rev

This parameter controls the counts-per-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.

11.3.27 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.

11.3.28 Parameter 36 – Rigid Tapping Enable/Disable

This parameter is a bit field that enables or disables Rigid Tapping and its options.

Bit	Function 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 the Spindle Encoder? (See Note Below)	The Spindle Encoder Measures the Rotation of the Spindle Drive Motor (Add 16) (See Note Below)
5	Turn Off Spindle Following While At the Bottom of a 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 Rigid Taps. 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 descriptions of spindle gear ranges/ratios).

11.3.29 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.

11.3.30 Parameter 38 – Multi-axis Max Feed Rate

This parameter is used to limit the feed rate 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 two- or three-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).

Draft: January 11, 2024

11.3.31 Parameter 39 – Feed Rate Override Percentage Limit

This parameter is used for limiting the upper end of the Feed Rate Override Knob percentage to a value from 100% to 200%. This parameter can be used to restrict the Feed Rate Override Knob effect on machines with maximum rates over 200 in/min. The Feed Rate 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 a value like 110, it will stop the Feed Rate Override Knob from exceeding 110%, thus causing the overshoots to disappear. The PLC program must check this parameter to actually enforce the limiting of Feed Rate Override.

11.3.32 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.

11.3.33 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.

11.3.34 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 is required for supervisor access; the user is not prompted for a password	54.0
The password is four digits represented by “ABCD” Example: for the password to be 1234, set the value to 1234.1234	ABCD.ABCD
The password is “137”	Any other number

11.3.35 Parameter 43 – G71/G72 Depth of Cut

The depth of each successive cut along the Z-axis (for G71) or X-axis (for G72). The minimum value is 0.0001". The maximum value is 9999.9999". The default value is 0.01".

11.3.36 Parameter 44 – G71/G72 Escape Amount

The distance that the cutter will move away from the just-cut surface before going back to start the next pass. The minimum value is 0. The maximum value is 9999.9999". The default value is 0.

11.3.37 Parameter 45 – G74 X-axis Relief Amount

Distance along the X-axis that the cutter will move away from the surface before returning to the starting point at the end of a pass. This parameter is used only by Intercon Setup. This does not affect G74 operation.

Draft: January 11, 2024

11.3.38 Parameter 46 – G75 Z-axis Relief Amount

The distance along the Z-axis that the cutter will move away from the surface before returning to the starting point at the end of a pass. This parameter is used only by Intercon Setup. This does not affect G75 operation.

11.3.39 Parameter 47 – G73 Repeat Count

The number of passes to cut. The minimum value is 1. The maximum value is 1000. The default value is 1.

11.3.40 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 feed rate 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.

11.3.41 Parameter 49 – Thread Chamfer Amount

The length of the chamfer inserted at the end of threads cut with the G92 and G76 cycles, as a multiple of the thread lead. A value of 1.0 inserts a one-thread chamfer. The minimum value is 0. The maximum value is 100. The default is 0. See G92 and G76.

11.3.42 Parameter 50 – G76 Finish Count

Number of finish passes in the G76 cycle. All of the finish allowance is removed with the first finish pass and the remaining passes are spring passes over the same path. The minimum value is 1. The maximum value is 99. The default value is 1. See Chapter 9 for more information on G76.

11.3.43 Parameter 51 – G76 Thread Angle

Compound angle of the thread. The minimum value is 0. The maximum value is 120. The default value is 0. See Chapter 9 for more information on G76.

11.3.44 Parameter 52 – G76 Minimum Cutting Depth

In the G76 cycle, each successive pass has a smaller depth increment. This parameter sets the minimum depth increment. The minimum value is 0.0001". The maximum value is 999.9999". The default value is 0.0010". See Chapter 9 for more information on G76.

11.3.45 Parameter 53 – G76 Finish Allowance

Finish allowance left after the depth passes, which is to be removed by the first finish pass. The minimum value is 0.0001". The maximum is 9999.9999". The default is 0.0100". See Chapter 9 for more information on G76.

Draft: January 11, 2024

11.3.46 Parameter 55 – Radius / Diameter mode

Value	Meaning
0	**Diameter mode** X-axis positions in G-code and DRO X-positions are diameter width values whose center is X0. U (incremental X)-positions in G-code and X-axis tool offsets are diameter difference values. Commanding X in diameter mode causes actual movement that is half of the specfied X-value. However, "I" (incremental X arc center) in G2/G3 is always a radius value, despite this parameter.
1	**Radius mode** G-code I, X, U, DRO X-positions, and X tool offsets are specified in terms of radius. Commanding X in radius mode causes actual movement that is equal to the specified X-value.

11.3.47 Parameter 56 – Feedrate Override Display Properties / Inverse Time Interpolation

Bit 0 applies an averaging filter to the displayed WMPG Feed rate when using the handwheel. Bits 1 and 2 define how the feed rate override is displayed in the status window. Bit 3 selects the meaning of inverse time feed rates (F) when inverse time mode (G93) is active. The default value is 0.

Note: Bit 1 does not affect actual federate.

Bit	Function Description	Value
0	Apply averaging filter to displayed WMPG feed rate	Add 1
1	Display programmed rate not actual	Add 2
2	Display a bar meter of percentage	Add 4
3	Inverse time meaning of F4	F is in inverse Seconds (Add 8)

11.3.48 Parameter 57 – Use Generic Load Meter Data from PLC

If load meters are enabled (via Parameter 143), then 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), then 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 will be future support for SV_METER_9 up to SV_METER_16, but for now there is only support for up to eight axes.

11.3.49 Parameter 58 – G71/G72 Profile Rough Cut Escape Angle

Each incremental pass in the rough cut portion of the G71 and G72 cycles ends with an angled feed move away from the material. This parameter sets the default angle of this angled feed move (however, if the G71/G72 cycle is specified with an A, the A value will override this parameter).

Draft: January 11, 2024

11.3.50 Parameter 59 – Tool Check Options

This parameter specifies additional options related to Tool Check. The Run `tcmacro.mac` option (+1) will cause the CNC software to run the macro named `tcmacro.mac` in response to a Tool Check during execution of a program (but not during a move made in MDI). The Lock Spindle Speed During Macro option will lock the spindle speed when a Tool Check is pressed, and using this option effectively may require cooperation with the PLC program and/or macro changes.

Bit	Function Description	Value
0	Run `tcmacro.mac`.	Add 1
1	Lock spindle speed during macro.	Add 2

11.3.51 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.

11.3.52 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 is stopped against some abnormal resistance or not tuned correctly, such that it will probably overheat later.

11.3.53 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 two 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. For example, when a 4th-axis is paired with the X-axis and a 5th-axis is 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.

Draft: January 11, 2024

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

11.3.54 Parameters 65–67 – Spindle Gear Ratios

These parameters tell the control the gear ratio values for a multi-range spindle drive. Up to four speed ranges are supported, with high range as 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 at 1/10 of 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

11.3.55 Parameter 68 – Minimum Rigid Tapping 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 the Z-axis to fall short. If this value is too large, the off-target error will increase. The suggested starting value is 640 RPM.

11.3.56 Parameter 69 – Duration For Minimum Spindle Speed (Rigid Tapping Parameter)

This is the duration of time, in seconds, that the control will stay at the minimum spindle speed. If the number is too small, overshoot will occur. If the number is too large, the user will wait longer for the hole to be tapped at the slow speed specified by Parameter 68. The suggested starting value is 1.25 seconds.

Draft: January 11, 2024

11.3.57 Parameter 70 – Offset Library Inc/Decrement Amount

Sets the increment and decrement amounts used in the offset library.

11.3.58 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 the approach direction) the part position.

11.3.59 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

11.3.60 Parameter 73 – Peck Drill Retract Amount (Canned Cycle Parameter)

This parameter sets the peck retract amount associated with G74 and G75. The minimum value is 0. The maximum value is 9999.9999". The default value is 0.0500". See Chapter 9 for more information on G74 and G75.

11.3.61 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 (this is 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 counter-tapping cycle is done (returned to the initial point). Both Tap and Counter-tap cycles are performed by G84 (see G84 in Chapter 9).

11.3.62 Parameter 75 – Summing Control

This parameter controls the type of position to be summed (local or machine), which axes are to be summed together, and which axis will bear the effects of the the summing. This 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:

Draft: January 11, 2024

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 the axis 3 and 4 machine coordinate positions summed together.	400
Axis 4 DRO will display the result of the axis 4 and 3 local coordinate positions summed together. Machine coordinate positions are not affected.	−3000
Axis 3 DRO will display the result of the 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 the 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 the axis 3 and 4 local coordinate positions summed together. Axis 4 will be displayed only if it moves.	−7400

11.3.63 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

Draft: January 11, 2024

11.3.64 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.

11.3.65 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 its number of encoder counts/revolution specified in Parameter 34. When turned off (value = 0), the displayed speed is not measured and 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 feed rate 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 the 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, and 462.

Bit	Function Description	Value
0	Display actual spindle speed	Add 1
1	Slave feed rate to spindle speed	Add 2
2	Wait for spindle to get up to speed on feed-per-minute moves	Add 4
3	Display Actual Spindle Speed from 2nd Spindle-mounted Encoder	Add 8

11.3.66 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.

11.3.67 Parameter 80 – Voltage Brake Message Frequency

This parameter specifies the number of times the “450 Voltage brake applied” message has to occur before it is shown in the message window and message log. A value of 0 or 1 will display the message for every instance that it occurs.

11.3.68 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.

Draft: January 11, 2024

11.3.69 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.

11.3.70 Parameter 83 – Deep Hole Clearance Amount (Canned Cycle Parameter)

Parameter 83 specifies the clearance amount used during a G83 deep hole drilling cycle.

11.3.71 Parameter 84 – M-function executed at 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 counter-tapping cycle (to reverse the spindle in preparation for pulling out of the counter-tap hole).

11.3.72 Parameter 85 – "Door Open" Interlock PLC bit

This parameter provides a way for a system integrator to implement a safety interlock that limits the rate of movement when the doors are open. This parameter specifies the PLC bit number and PLC bit polarity that indicate 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 like so: 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 (220=1).

Note: PLC program interaction is needed for this feature to work.

11.3.73 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).

11.3.74 Parameters 87–90 (and also 252–255) – Autotune Ka Performance parameters for axes 1, 2, 3, 4

These parameters are used by Autotune. Increasing their values 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.

Draft: January 11, 2024

11.3.75 Parameters 91–94 (and also 166–169) – Axis Properties for axes 1, 2, 3, 4

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	NOT USED ON LATHE	Recommended bit value is 0
3	Suppress park function?	Don't Park = 8, Park = 0
4	C-axis Selection	C-axis = 16, Off = 0
5	Linear Display of Rotary Axis	Linear Display = 32, Default Rotary = 0
6	NOT USED ON LATHE	Recommended bit value is 0
7	For C-axis prevent divide of counts-per-revolution by 360	Prevent Divide by 360 (Add 128)
8	NOT USED ON LATHE	Recommended bit value is 0
9	Hide axis display from DRO	Add 512
10	NOT USED ON LATHE	Recommended bit value is 0
11	NOT USED ON LATHE	0
12	NOT USED ON LATHE	0
13	RESERVED FOR FUTURE USE	0
14	NOT USED ON LATHE	0
15	RESERVED FOR FUTURE USE	0
16	NOT USED ON LATHE	0
17	Display a Linear Axis in Degrees	Add 131072

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 3: Setting this bit prevents **F1** (Park) in the Shutdown menu from parking this axis.

Bit 4: Setting this bit enables C-axis control capability. The corresponding label field in the Machine Configuration should also be set to "C".

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 7: By default, an axis configured as a C-axis will have its counts-per-revolution divided by 360 before being used internally by the CNC software to provide more precise positioning for the C-axis. Setting this bit to ON will prevent this behavior.

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.

Draft: January 11, 2024

11.3.76 Parameters 95–98 (and also 156–159) – Autotune Move Distance / Auto Delay Calculation Move Distance for axes 1, 2, 3, 4

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.

11.3.77 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 the default value is 1.

11.3.78 Parameter 100 – Intercon comment generation

This parameter is for Intercon comment generation. It's values are 0=off and 1=on. This parameter can also be set in the Intercon Setup menu. Its default value is 0.

11.3.79 Parameter 101 – Intercon clearance amount

This parameter is for Clearance Amount. Its values are between 0 and max travel limit. The default value is 0.1.

11.3.80 Parameter 102 – Intercon spindle coolant delay

This parameter is for Spindle Coolant delay. Its values are between 0 and 6. The default value is 3.

11.3.81 Parameter 103 – Intercon corner feed rate override

This parameter is for Corner Feedrate Override. It's values are between 1 to 100. The default value is 50.

11.3.82 Parameter 104 – Intercon modal line parameters

This parameter is for Modal linear. It's values are 0=off and 1=on. The default value is 0.

11.3.83 Parameter 105 – Intercon modal arc parameters

This parameter is for Modal Arc. It's values are 0=off and 1=on. The default value is 0.

11.3.84 Parameter 106 – Intercon modal drilling cycle parameters

This parameter is for Modal Drill Bore Tap. It's values are 0=off and 1=on. The default value is 0.

Draft: January 11, 2024

11.3.85 Parameter 107 – Intercon chamfer blend

11.3.86 Parameter 108 – Intercon polar display

This parameter is for toggling the polar display. Its values are 0=off and 1=on. The default value is 1.

11.3.87 Parameter 109 – Intercon modal display

This parameter is for toggling the modal display. Its values are 0=off and 1=on. The default value is 1.

11.3.88 Parameter 110 – Wizard Password

Set this parameter to 1 to enable a password prompt when opening the Wizard. The password is configured by Parameter 42 and can be configured from the Wizard. The default value for this parameter is 0.

11.3.89 Parameters 111–115 – Intercon parameters

See Parameters 100–109 for more information.

These parameters are some of the Intercon setup parameters. See Chapter 7 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.

11.3.90 Parameter 113 – Hide Intercon Menu

Set this parameter to 1 to prevent CNC12 from opening Intercon from the CAM menu option. The default value for this parameter is 0.

11.3.91 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. It is used to define the Y-coordinate of the center of rotation for a rotary axis that rotates about the X-axis.

11.3.92 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. It is used to define the Z-coordinate of the center of rotation for a rotary axis that rotates about the X-axis.

11.3.93 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. It is used to define the X-coordinate of the center of rotation for a rotary axis that rotates about the Y-axis.

Draft: January 11, 2024

11.3.94 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. It is used to define the Z-coordinate of the center of rotation for a rotary axis that rotates about the Y-axis.

11.3.95 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.

11.3.96 Parameter 121 – Grid digitize prediction minimum Z-pullback

This parameter specifies the minimum distance that 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 that 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.

11.3.97 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.

11.3.98 Parameter 128 – Dry Run PLC Bit

When non-zero, this parameter specifies which PLC bit 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 Feed Rate (Parameter 129). When in Dry Run mode, the Feed Rate word in the status window is drawn in reverse video (blue letters on a yellow background).

11.3.99 Parameter 129 – Dry Run Feed Rate

Specifies the feed rate to use for Dry Run mode.

11.3.100 Parameters 130–131 – 3rd/4th axis on/off selection

These parameters control the display of the 3rd- and 4th-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 axes labels A, B, C, U, V, W, X, Y, and Z. The “ones” digit specifies the label of the axis when it is disabled, with 0.0 meaning that the axis is not switchable, 1.0 meaning it turns off (N), 2.0 meaning manual (M), and 3.0 meaning 2-axis with a manual Z-axis (@). Parameter 130 also supports additional modes depending upon the value of the “hundreds” digit. See the chart below for valid values for the “hundreds” digit of Parameter 130. Note that Parameter 131 does not support the “hundreds” digit. When P130/P131 is configured for axis switching, the Setup menu displays function keys **F5**/**F6** to switch the

Draft: January 11, 2024

axes.

Example 1: A value of 192 in Parameter 130 will toggle the 3rd-axis between Z and M and power off all of the axes. In 192, the “1” sets bit one to power all axes off, the “9” enables the 3rd-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 3rd-axis label between Z and M and power off all of the axes and receive its positions from the 4th-axis encoder input. In 392, the “3” sets bits two and one to power off all axes and use the 4th encoder input as a scale input, the “9” enables the 3rd-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 4th axis between W and N. In 61, the “6” enables the 4th-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 4th-axis on and off.

Hundreds Digit (Parameter 130 only) (X00)	Function Description	Value of the Hundreds Digit
Bit1	Axis motor power when switching to two-axis mode.	1 = power all axes off, 0 = power 3rd-axis 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

Draft: January 11, 2024

11.3.101 Parameters 132–135 – Motor Heating Coefficients for axes 5–8

See Parameters 20–30 for more information.

11.3.102 Parameter 136 – G76 Fine Bore Retract Angle

0–360 Degrees. A setting of 0 = Retract in Y+ direction.

11.3.103 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.

11.3.104 Parameter 138 – DRO Encoder Deadband

This controls the deadband amount (in encoder counts) used by the anti-flicker DRO filter.

11.3.105 Parameter 139 – Special Dwell between Moves

This parameter turns on and specifies the amount of 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. These conditions include: (1) No dwell will happen between two arcs (2) No dwell will happen if before the move that contains Z-movement.

11.3.106 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 12 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

Draft: January 11, 2024

11.3.107 Parameter 141 – Maximum message log lines

This parameter is the number of lines that will be kept in the message log. For example, if this parameter is set to 10,000, 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.

11.3.108 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 amount of delay is unacceptable, reduce the values of Parameters 141 and 142.

11.3.109 Parameter 143 – DRO properties (load meter, DTG)

This parameter controls the display of the axis load meters, mini DRO display, and mini machine coordinates display.

DRO Precision has been moved to Parameter 263 – DRO Display Precision.

The default value for this parameter is 75 (Load Meter – (Add 1), Load Meter Outline – (Add 2), Mini DRO – (Add 8), Mini Machine Coordinates – (Add 64)).

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)

11.3.110 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. For example, if the parameter is set to 9.0, 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, then the programmer can shorten the previous example to: `IF #A EQ #B THEN GOTO 100`.

11.3.111 Parameter 145 – Advanced macro properties (fast branching)

This parameter turns fast branching ON (1) or 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 16 kilobytes of memory for every 1000 block numbers in the program).

11.3.112 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.

11.3.113 Parameter 147 – Number of Messages 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.

11.3.114 Parameter 148 – Miscellaneous Jogging Options

This parameter enables and/or disables certain optional modes of jogging.

Draft: January 11, 2024

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.

11.3.115 Parameter 149 – Spindle Speed Threshold

This parameter defines the spindle speed threshold percent for the “Slave feed rate to spindle speed” and “Spindle up-to-speed” functions, 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.

11.3.116 Parameter 150 – Backplot Graphics display options

This parameter controls the various options related to backplot graphics.

Bit	Function Description	Value
0	Sets the 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 Five-axis Machines	Disable (Add 4)
3	Display Skew Correction	Enable (Add 8)
4	Display Lash/Screw Compensation	Enable (Add 16)

Draft: January 11, 2024

11.3.117 Parameter 151 – Repeatability tolerance for probing and radial digitizing

The default value 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-per-point are taken 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, two more measurements are taken and the process repeated up to a maximum of ten times. In probing cycles, if the repeatability tolerance cannot be met, the cycle is cancelled and an error message is generated. In radial digitizing, the point is discarded and digitizing continues without interruption.

11.3.118 Parameter 153 – Probe Protection Enabled

11.3.119 Parameter 154 – Touch Screen Options

If this value is set to 1, a half-width ESC key is displayed in most menus so that there is a means of exiting the menu without using the keyboard. Most menus have been reworked to more fully support touchscreen input.

11.3.120 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

11.3.121 Parameters 156–159 – Autotune Move Distance / Auto Delay Calculation Move Distance for axes 5, 6, 7, 8

See Parameters 95–98 for more information.

11.3.122 Parameter 160 – Enhanced ATC

This parameter controls enhanced Automatic Tool Changer (ATC) options. A value of 1 indicates a non-random type of ATC (carousel ATC) and a value of 2 indicates a random type of 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 non-random type) or the current ATC carousel position (if random type). For both random and non-random 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 non-random types, the new tool now in the spindle will have its “putback” field updated to the current ATC carousel position.

For both random and non-random 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 set to 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 itself.

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.

Draft: January 11, 2024

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.

11.3.123 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.

11.3.124 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.

11.3.125 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 Machine Configuration sets the original DeadStart values for each axis. This parameter allows you to modify these DeadStart settings under certain conditions. Note that if both bit 0 and 1 are turned on (value = 1+2 = 3), then the effect is cumulative, i.e. the net effect will be that 1/2 of the DeadStart value will be used when a slave axis stops or starts up from a stop. Likewise, if both bit 2 and 3 are turned ON, the effect will also be cumulative. Note that this feature has no effect for movement commands handled by Smoothing (P220=1).

Bit	Function Description	Value
0	Use 1/4 of the DeadStart value for a slave axis that stops or starts from a stop.	Enable (Add 1)
1	Use 2X the DeadStart value for a slave axis that stops or starts from a stop.	Enable (Add 2)
2	Use 1/4 of the DeadStart value for a slave axis that reverses.	Enable (Add 4)
3	Use 2X the DeadStart value for a slave axis that reverses.	Enable (Add 8)
4	Limit the feed rate along the path of G2 or G3 arc moves, such that the feed rate will be uniformly limited to the lesser maximum rate of the two axes involved in the circular motion.	Enable (Add 16)

11.3.126 Parameters 166–169 – Axis Properties for axes 5, 6, 7, 8

See Parameters 91–94 for more information.

11.3.127 Parameters 170–179 – PLC Parameter

These parameters are specially-reserved as a space for data that is to be sent to the PLC. Parameters 177, 178, and 179 have been standardized for specific applications. Parameter 177 is used for troubleshooting purposes only.

Draft: January 11, 2024

11.3.128 Parameter 170 – Enable Keyboard Jogging and Set Feed Rate Override Control

This PLC parameter is used to enable Keyboard Jogging and determine whether the Jog Panel or Keyboard Feed Rate Override is used. To enable Keyboard Jogging, set Parameter 148 to 0 (zero) and this parameter to 1.

Bit	Function Description	Value
0	Enables Keyboard Jogging	Enable (Add 1)
1	Only Looks at Feed Rate Override from Jog Panel	Enable (Add 2)
2	Only Looks at Feed Rate Override from Keyboard	Enable (Add 4)

11.3.129 Parameter 178 – PLC I/O configuration (PLC program specific)

This parameter can be used 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

11.3.130 Parameter 179 – Lube Pump Operation

This parameter can be configured to control a variety of lube pumps. The value is formatted as MMMSS, with 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 the 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 minutes of job or MDI time, then holds power ON for 15 seconds.

11.3.131 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.

Draft: January 11, 2024

11.3.132 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 five times.

11.3.133 Parameter 187 – Hard Stop Homing

This parameter is used when homing off of hard stops. The value set in this parameter determines the amount of current sent to the motor while homing. The value range is 0–32000; a typical value for a DC system is 16000. Note that this feature does not work with velocity mode drives.

11.3.134 Parameters 188–199 – Aux Key Functions

These parameters are used to assign a function to Aux Keys 1–12 (i.e. P188 = Aux1 . . . P199 = Aux12). The following is the list of possible functions that can be executed when an aux key is pressed.

Function	Value
No Function	0
Input Z-Axis Position*	1
Input X-Axis Position*	2
Input 3rd-Axis Position *	3
Set Absolute Zero	4
Set Incremental Zero	5
Execute M-Code File	M11**
Free Axes	14

Function	Value
Go to Power Feed Menu	15
X, Z, 3rd Set Absolute Zero	16
One Shot – Chamfer	56
One Shot – Turning	57
One Shot – Facing	58
One Shot – Radius	59
One Shot – Drill	60

* 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 is pressed. Custom overlays with the keys that represent these functions are available. Contact your dealer for pricing.

For example, if you wanted Aux4 to call up the “One Shot – Turning”, you would set Parameter 191 to 57.

For more information on one shot operations, see Chapter 8 “CNC Program Codes”.

11.3.135 Parameters 200–207 – OPTIC 4 Tach Volts Per 1000 RPM

These parameters control the digital tachymeter output on the Optic4 boards. They are used on drives like old Fanuc velocity mode drives that require a tachymeter input. The value put here is the volts/1000 RPM off of the motor. A negative value can be entered to invert the tachymeter voltage compared to the encoder count derived velocity direction from the encoder.

Draft: January 11, 2024

11.3.136 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 or Screw Compensation.

11.3.137 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.

Draft: January 11, 2024

11.3.138 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

11.3.139 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 the 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 license or above.

11.3.140 Parameter 219 – Virtual Control Panel Options

When set to 1.0, the 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.

11.3.141 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.***

11.3.142 Parameters 236–239 – Motor Cooling Coefficients for axes 5–8

See Parameters 20–30 for more information.

11.3.143 Parameter 240 – Rigid Tapping and Threading Accel Rate Distance

The default value is 0.1000

Setting the value to −1 for servo motors will disable Parameters 240, 241, and 242. This will introduce banging at higher spindle speeds. For stepper motors, do not set this parameter to a value of −1, because this will cause a loss of axis position.

This parameter is used for servo motors to eliminate banging at the start and end of tapping and threading.

This is the distance it takes to get the axis up-to-speed to sync with the marker pulse on the spindle.

For servo-motors, this allows for higher spindle speeds to eliminate 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.

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 causing a loss of axis positions.

For both types of motors used when threading on the lathe: the larger the value, the larger the deceleration distance. If the value is too large, this could possibly cause the end the threads to not have the correct pitch.

11.3.144 Parameter 241 – Rigid Tapping and Threading 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 a high RPM and a coarse pitch thread, this may have to be increased to eliminate banging.

Draft: January 11, 2024

11.3.145 Parameter 242 – Minimum Angle Threshold for application of Accel/decel in Threading moves

Default = 30.00000

This is used for G32 Threading on a lathe, allowing an Operator to perform 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, the CNC software will not apply the accel/decel at the angle change point at the end of the first G32 or the start of the second G32. If the value is greater than the angle of change, the CNC software will apply the accel/decel at the angle change between G32 moves.

11.3.146 Parameter 243 – Threading Control

Bit 1 (value 2) suppresses Wait for Index Pulse. Bit 2 (value 4) permits Spindle Override During Threading.

11.3.147 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.

Draft: January 11, 2024

11.3.148 Parameter 245 – G71, G72, G74, G75, G76 D/P/Q "implied float" re-interpretation threshold for Inch units

11.3.149 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 value 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)

A decimal point "." is present in the D/P/Q parameter.	`D01.23`
	`P4.56`
	`Q789.`
		The D/P/Q values are left as-is, because these are obviously already floating point numbers.	The D/P/Q values are left as-is, because these are already floating point numbers.

The D/P/Q value is a whole number beginning with a leading 0.	`P0010` `Q023`	The D/P/Q values are treated as implied floating point values with the decimal point four places in from the right. Effective re-interpretation using the examples would be: `P0.0010` and `Q0.0023`	The 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`
The D/P/Q value is a whole number with no leading 0.	`P50` `Q500`	If the D/P/Q value is less than or equal to Parameter 245, then the value will be left alone. If the 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 four 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 the D/P/Q value is less than or equal to Parameter 246, then the value will be left alone. If the 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 three 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 applies only if the D/P/Q value is a length/width parameter of a G71, G72, G74, G75, or G76 cycle.
- If a decimal point is present in 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 the D,P,Q length parameters only happens for G71, G72, G74, G75, and G76. All other letter addresses and D,P,Q parameters used in other situations outside of G71, G72, G74, G75, and G76 are not affected.

11.3.150 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 9 for more information.

Draft: January 11, 2024

11.3.151 Parameter 248 – Tool Wear Adjustment magnitude limit

This parameter sets the +/- value limit allowed to be entered into the Tool Wear Adjustment menu.

11.3.152 Parameters 252–255 – Autotune Ka Performance parameters for axes 5, 6, 7, 8

See Parameters 87–90 for more information.

11.3.153 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. The Autotune feature is enabled and accessible via **F5** in the PID Menu	0
Velocity mode. The Autotune feature is disabled. Autotune does not work with velocity mode drives, 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

11.3.154 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.

11.3.155 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.

11.3.156 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 “@”.)

Draft: January 11, 2024

11.3.157 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

11.3.158 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

11.3.159 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.

Draft: January 11, 2024

11.3.160 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.

11.3.161 Parameter 264 – Rapid Override

This parameter enables the Rapid Override feature as a separate control and allows for enabling a percentage bar to show the current Rapid Override rate.

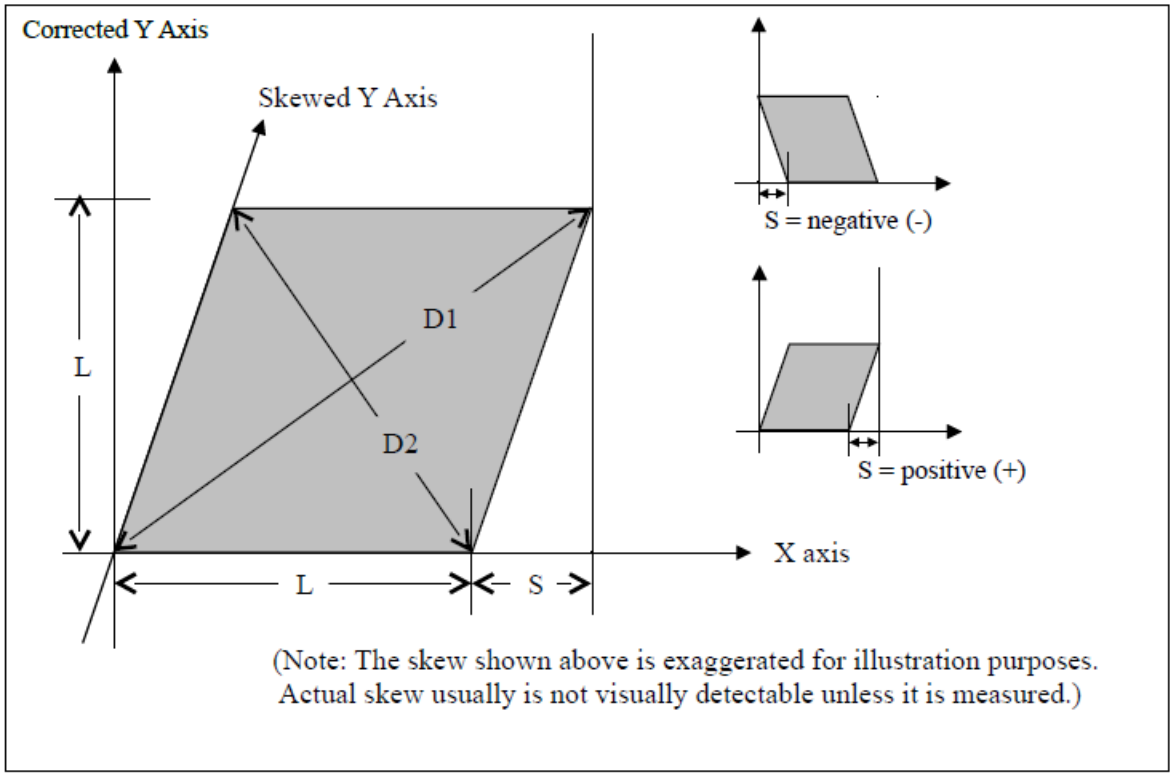
Bit	Function Description	Value
0	Enable Rapid Override rate	Add 1
1	Display Percentage bar in CNC12 for Rapid Override	Add 2

11.3.162 Parameters 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

Skew Measurement Procedure: 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: $S = (D1^2 - D2^2)/(4L)$

11.3.163 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 shows whole number spindle speeds.

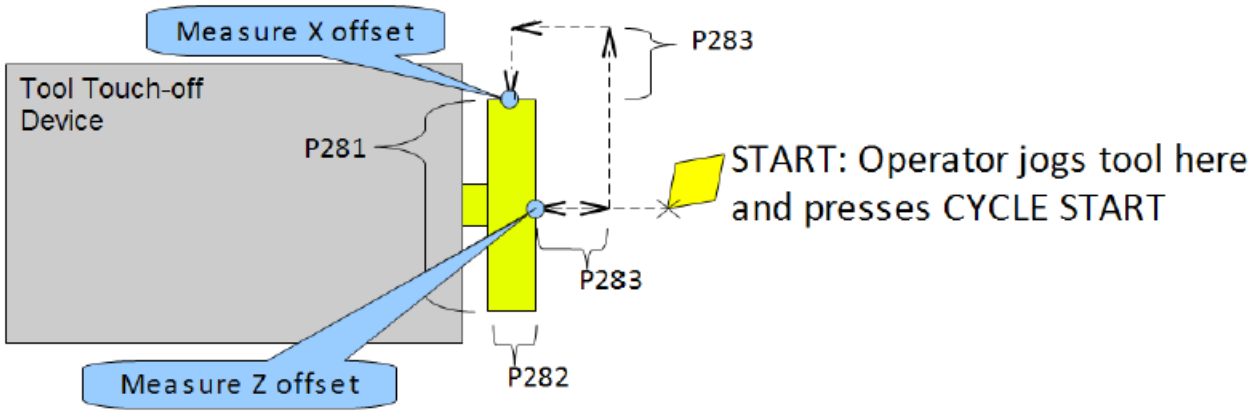
11.3.164 Parameter 281 – Tool Touch-off Device X stylus size

11.3.165 Parameter 282 – Tool Touch-off Device Z stylus size

11.3.166 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”:

Draft: January 11, 2024



11.3.167 Parameters 284–291 – Brake Resistor Wattage for ACDC Drives 1–8

These parameters specify the brake resistor wattage, which defaults 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 the MPU11. Note that these are drive numbers that 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.

11.3.168 Parameters 292–295 – Aux Key Functions 13–16

These parameters are used to assign a function to Aux Keys 13–16 (i.e. P292 = Aux13 . . . P295 = Aux16). For more information on aux key functions, see Section 11.3.134.

11.3.169 Parameter 299 – Report Option Bits

Set this parameter to 1 to hide the directory listing in the `Report.zip` text file. The default value for this parameter is 0.

11.3.170 Parameters 300–307 – Drive assignment to Axes 1–8

These parameters control what physical drive the commands for motion are sent to. 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.**

Draft: January 11, 2024

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	5th 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.

11.3.171 Parameters 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.

Draft: January 11, 2024

Encoder Number	Location	Description
1	MPU11 onboard encoder 1	Encoder inputs on the MPU11, Allin1DC, and OAK.
2	MPU11 onboard encoder 2	
3	MPU11 onboard encoder 3	
4	MPU11 onboard encoder 4	
5	MPU11 onboard encoder 5	
6	MPU11 onboard encoder 6	
7	Drive Bus Channel encoder 1	Encoder inputs on Drive bus devices.*
8	Drive Bus Channel encoder 2	
9	Drive Bus Channel encoder 3	
10	Drive Bus Channel encoder 4	
11	Drive Bus Channel encoder 5	
12	Drive Bus Channel encoder 6	
13	Drive Bus Channel encoder 7	
14	Drive Bus Channel encoder 8	
15	MPU11 onboard MPG encoder	MPG connector with no index pulse

Note*: 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.

11.3.172 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 ($2^6 + 2^7 + 2^8$).

11.3.173 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.

11.3.174 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).

11.3.175 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 conversion	Enable (Add 2)

* Machine Parameter 166 (5th-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.

11.3.176 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.

11.3.177 Parameter 322 – Spindle Encoder Port

Allows the user to assign which encoder port the spindle will use to communicate.

Draft: January 11, 2024

11.3.178 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' bit should be set unless that axis refers to a third-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.

11.3.179 Parameters 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 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. Also, note 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.

11.3.180 Parameters 332–335 – Encoder error suppression

Draft: January 11, 2024

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 eight 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.

11.3.181 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 set to a non-zero value.

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, `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.

11.3.182 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.

11.3.183 Parameters 348, 351, and 354 – MPG/Handwheel Encoder Input 1, 2, and 3

11.3.184 Parameter 348 – MPG/Handwheel Encoder Input 1

11.3.185 Parameter 351 – MPG/Handwheel Encoder Input 2

11.3.186 Parameter 354 – MPG/Handwheel Encoder Input 3

The encoder input for the MPG or handwheel (1–15). See the 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.

11.3.187 Parameters 349, 352, and 355 – MPG/Handwheel Detents per Revolution 1, 2, and 3

11.3.188 Parameter 349 – MPG/Handwheel Detents-per-revolution 1

11.3.189 Parameter 352 – MPG/Handwheel Detents-per-revolution 2

11.3.190 Parameter 355 – MPG/Handwheel Detents-per-revolution 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/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.

Draft: January 11, 2024

11.3.191 Parameters 350, 353, and 356 – MPG/Handwheel Encoder Counts per Revolution 1, 2, and 3

11.3.192 Parameter 350 – MPG/Handwheel Encoder Counts-per-revolution 1

11.3.193 Parameter 353 – MPG/Handwheel Encoder Counts-per-revolution 2

11.3.194 Parameter 356 – MPG/Handwheel Encoder Counts-per-revolution 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.

11.3.195 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 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 third-party velocity mode drives that have a different max rate setting than that of the control software.

Draft: January 11, 2024

11.3.196 Parameter 365 – Drive power-on delay

This specifies the number of milliseconds that the MPU11 will wait between the moment that the drive power first comes on and the commanded motion begins. 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.

11.3.197 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.

11.3.198 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.

11.3.199 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.

11.3.200 Parameters 374–379 – ACDC Drive Debug Log Settings

These parameters are used by support technicians and should be left at value 0.

11.3.201 Parameters 387–389 – Debugging Parameters

These parameters are used for internal debugging and troubleshooting. They should be left at 0.

11.3.202 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 that the probe moves off of the surface after a probing move.
393	DP-7 Pullback Feedrate: The feed rate for the pullback move.
394	DP-7 Measuring Feedrate: The feed rate for the slow measuring move.

Draft: January 11, 2024

11.3.203 Parameter 395 – Probing Setup Traverse Speed

This sets the probing traverse feed rate for the macro-based probing cycles on engine block systems.

11.3.204 Parameter 396 – Probing Setup Plunge Speed

This sets the probing plunge feed rate for the macro-based probing cycles on engine block systems.

11.3.205 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.

11.3.206 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
Three-axis	0
Engine Block	1
Tilt Table	2

11.3.207 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 the theoretical true arc by more than 1/2 of an encoder count.

11.3.208 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.

11.3.209 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.

11.3.210 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.

11.3.211 Parameter 404 – Unused

Note: Prior to v4.16, this was the Active G-codes Display that was moved to Parameter 408

11.3.212 Parameter 405 – Tool Touch-off Type

Set this parameter to select the type of tool touch-off that is being used.

11.3.213 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.

Draft: January 11, 2024

11.3.214 Parameter 408 – Active G-Codes

Set this parameter to 0 to always display the currently active Lathe G-codes in the bottom left corner of the main screen and MDI screen. Set the value to 1 to display the Lathe G-codes only in MDI screen. Set the value to 2 to disable the display completely. The default value for this parameter is 0.

Function Description	Value
Display currently active Lathe G-Codes on the main screen and MDI screen	0
Only display currently active Lathe G-Codes in MDI screen	1
Disables currently active Lathe G-Code display	2

11.3.215 Parameter 409 – Probe Type

Set this parameter to select the type of probe connected.

11.3.216 Parameter 410 – Probe/Tool Touch-off Warning

11.3.217 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

Draft: January 11, 2024

11.3.218 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.

11.3.219 Parameter 414 – Home File Type

Used by the Acorn Wizard to remember the type of homing file that is being used by the system.

11.3.220 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 an 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.

11.3.221 Parameter 416 – Spindle Inhibit

Set this parameter to inhibit the spindle when detect is ON (green).

11.3.222 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.

11.3.223 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>

11.3.224 Parameter 419 – PLC Worklight and Popup Pin Output

This parameter configures the Worklight at startup and "Popup Pin" Output. The default value is 0.

Note: Popup Pin requires PLC support.

Description	Value
Enable Worklight at Startup and Turn On "Popup Pin" Output	0
Disable Worklight at Startup	1
"Popup Pin" Output Turn Off if Cycle Start is pressed	2
Disable Worklight and Turn Off "Popup Pin" Output	3

Draft: January 11, 2024

11.3.225 Parameter 420 – Analog/PWM 1 Settings

This parameter sets the Analog/PWM settings for the first Analog and PWM settings for specific control boards. For Acorn it sets the Analog Voltage only. The default value is 0.

This parameter is set by the wizard and is reliant on PLC to function properly.

For Acorn:

Description	Value
Set Analog Out to 0 V–10 V	0
Set Analog Out to 0 V–5 V	1

For AcornSix/Hickory:

Description	Value
Set Analog Out to 0 V–10 V (4883 Hz)	0
Set Analog Out to 0 V–5 V (1221 Hz)	1
Set Analog Out to –5 V–5 V (610 Hz)	2
Set Analog Out to –10 V–10 V (76 Hz)	3

11.3.226 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.

11.3.227 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.

11.3.228 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

11.3.229 Parameter 425 – Force Rehoming After EStop Condition

Set this parameter to require the machine to rehome after the EStop is pressed.

11.3.230 Parameter 426 – 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.

Draft: January 11, 2024

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

Draft: January 11, 2024

11.3.231 Parameter 430 – RTG Spindle Speed RPM Display

This parameter allows a real-time display of the current spindle speed in RPM.

11.3.232 Parameter 440 – Stopped for Jogging Continue Bit

This parameter specifies the behavior when the control is executing an M0/M200/M201 while 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.

11.3.233 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 an 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.

11.3.234 Parameter 450 – Inhibit MPG Z-axis Fast Movement

This parameter inhibits the use of the ×100 speed on the Z-axis. When set to a value of 1, the ×100 speed on the Z-axis will be reduced to ×10 speed.

11.3.235 Parameter 458 – C-Axis Options

0 is default.

Bit	Function Description	Value
0	Disables the C-axis when outside of a program.	Add 0
0	Allows the C-axis to remain enabled outside of running a program.	Add 1

11.3.236 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 6th-axis as the 2nd spindle, and the spindle's encoder is the 2nd MPU11 onboard input encoder, then Parameter 459 (this parameter) should be set to 6 and Parameter 312 should be set to 2.

Currently, the 2nd spindle can only be operated as paired (slaved) to the 1st spindle. Future development will allow independent command of the 2nd spindle.

Draft: January 11, 2024

11.3.237 Parameter 460 – Second Spindle Maximum Speed

This parameter controls the maximum speed for the 2nd Spindle. The minimum value is 0.

11.3.238 Parameter 461 – Second Spindle Minimum Speed

This parameter controls the minimum speed for the 2nd Spindle. The minimum value is 0.

11.3.239 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.

11.3.240 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.

11.3.241 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.

11.3.242 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.

11.3.243 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.

11.3.244 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.

11.3.245 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.

11.3.246 Parameter 602 – A-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.

11.3.247 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.

11.3.248 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.

11.3.249 Parameter 605 – B-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.

Draft: January 11, 2024

11.3.250 Parameters 700–799 – Dedicated Block of Parameters usable in Custom PLC Programs

Parameters 700–799 are reserved for use in Custom PLC Programs. Centroid does not use this range of parameters, nor will use it in the future, so there will be no interference with any Custom PLC Program.

11.3.251 Parameters 800–813 – DB25 Pin Selection

These parameters are set by the Wizard and determine the function that is assigned to each DB25 pin for Acorn Control Boards.

Parameter	Description	Default
800	Acorn DB25 Map Enable	0
801	Acorn Step 1	2
802	Acorn Step 2	4
803	Acorn Step 3	6
804	Acorn Step 4	8
805	Acorn Direction 1	3
806	Acorn Direction 2	5
807	Acorn Direction 3	7
808	Acorn Direction 4	9
809	Acorn PLC Output 1	1
810	Acorn PLC Output 2	14
811	Acorn PLC Output 3	16
812	Acorn PLC Output 4	17
813	Acorn DB25 Header Section	0

11.3.252 Parameter 814 – Analog/PWM 2 Settings

This parameter sets the Analog/PWM Settings for the second Analog and PWM settings for specific control boards. For Acorn it sets the PWM only. The default value is 4883.

This parameter is set by the wizard and is reliant on PLC to function properly. For Acorn, any value can be entered including those in the table below.

For Acorn/AcornSix/Hickory:

Description	Value
Set PWM Frequency to 4883 Hz (0 V–10 V)	4883
Set PWM Frequency to 1221 Hz (0 V–5 V)	1221
Set PWM Frequency to 610 Hz (–5 V–5 V)	610
Set PWM Frequency to 76 Hz (–10 V–10 V)	76

Draft: January 11, 2024

11.3.253 Parameter 815 – Inverse Output

11.3.254 Parameter 817 – PWM

11.3.255 Parameter 830 – ATC Type

11.3.256 Parameter 852 – Carousel Skip Count on Reverse

This parameter sets the “Skip Count” logic in PLC. Some Counting mechanisms for Carousels will “double count” when the carousel reverses direction. These types of Carousels will need the first count skipped on reverse so that the Carousel Bin Location is reported correctly.

Note: This parameter requires PLC Logic Interaction.

Function Description	Value
Do not skip count	0
Skip count on reverse	1

11.3.257 Parameter 853 – Distance to travel past tool position

On some Lathe ATC Turrets, an additional distance must be traveled to trigger the locking mechanism of the Turret. This parameter allows for configuration of the amount of distance the Turret must travel “past” the tool position before reversing back to the tool position to lock into place.

11.3.258 Parameter 854 – Distance to travel behind tool position

On some Lathe ATC Turrets, an additional distance must be traveled to trigger the locking mechanism of the Turret. This parameter allows for configuration of the amount of distance the Turret must travel “behind” the tool position after reversing back to the tool position to lock into place.

11.3.259 Parameter 855 – MPG Performance

11.3.260 Parameter 856 – VCP Jogging State on Acorn Power Up

11.3.261 Parameters 900–999 – Reserved for Centroid PLC Program Use

Parameters 900–999 are reserved for use in Centroid PLC Programs. This range of parameters should not be used for Custom PLC Programs as they have already been appropriated for Centroid PLC Program use.

11.3.262 Parameter 997 – Spindle Cooling Fan Delay Timer

This parameter allows the user to adjust the spindle cooling fan delay timer.

11.3.263 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

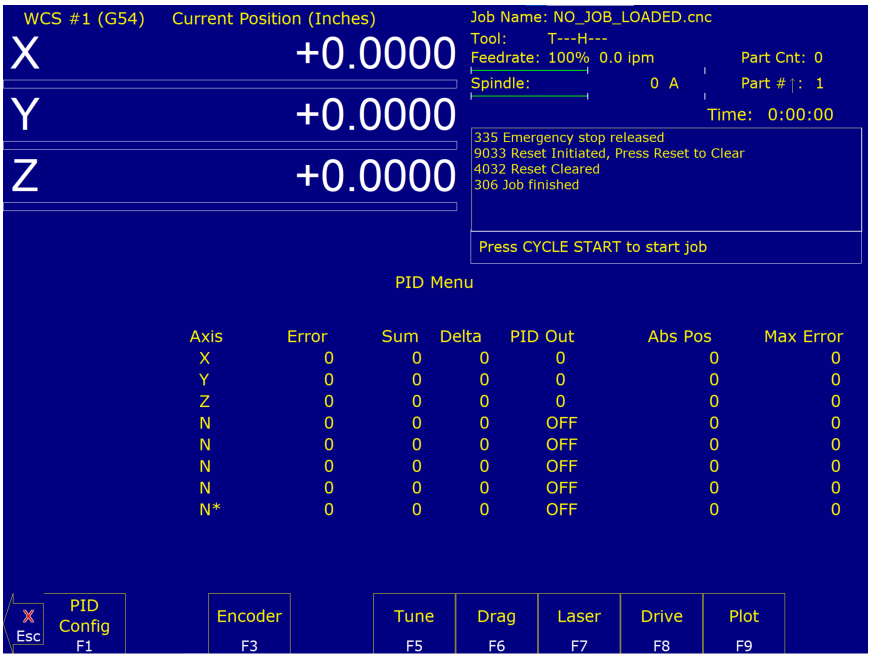
Draft: January 11, 2024

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.

11.4 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.

11.4.1 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 see (in real time) the effects of those modifications. Altering the PID values will cause **DRAMATIC** changes in the way that 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.

Draft: January 11, 2024

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!!!

11.4.2 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:

Draft: January 11, 2024

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.

11.4.3 F6 – Drag

This option is used by qualified technicians to determine whether your machine is binding anywhere along the axis of 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.

11.4.4 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.

11.4.5 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.

11.4.6 F9 – Plot

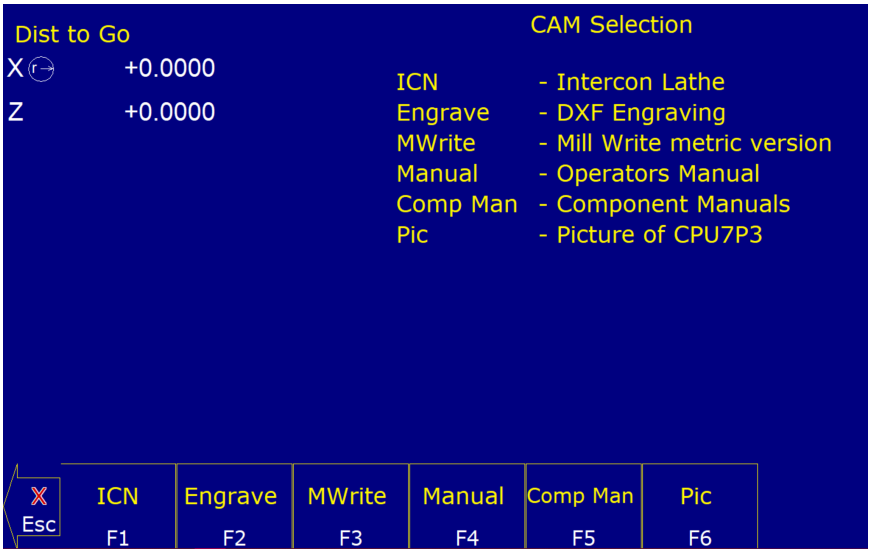
This option is used by qualified technicians to plot data.

11.5 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 cnct directory.

Draft: January 11, 2024

7. Double-click the `cnccams.txt` file. If there is not 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.



11.6 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

```
1  Comp Man :Component Manuals
2      c:\windows\explorer.exe /e,Manuals
3  Pic :Picture of CPU7P3
4      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

Draft: January 11, 2024

shown above opens the `cpu7p3.jpg` picture when the **F6 – Pic** button is pressed in the CAM menu. These examples will only behave in this way as long as the Centroid control has compatible software installed. 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.

11.7 PLC I/O Diagnostic App

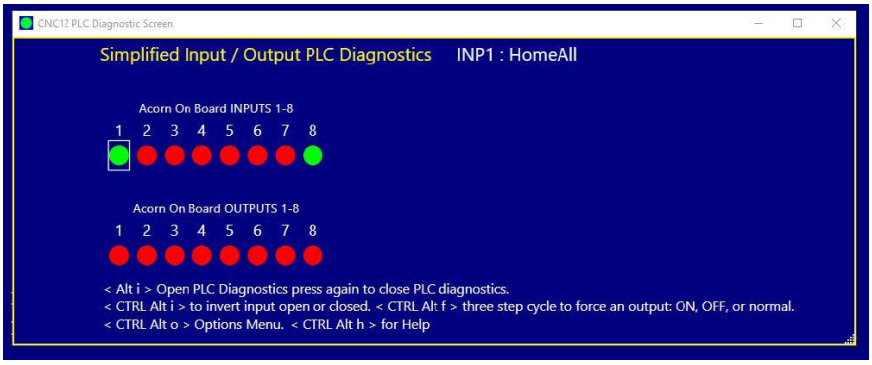
The PLC Diagnostic App is a tool to observe CNC controller Inputs and Outputs, and manually activate/invert/deactivate CNC controller Inputs and Outputs. It is also commonly used to debug any type of switches, relays, contactors, ATC's, etc. that are connected to the CNC controller.

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.

11.7.1 Using the PLC Diagnostic App

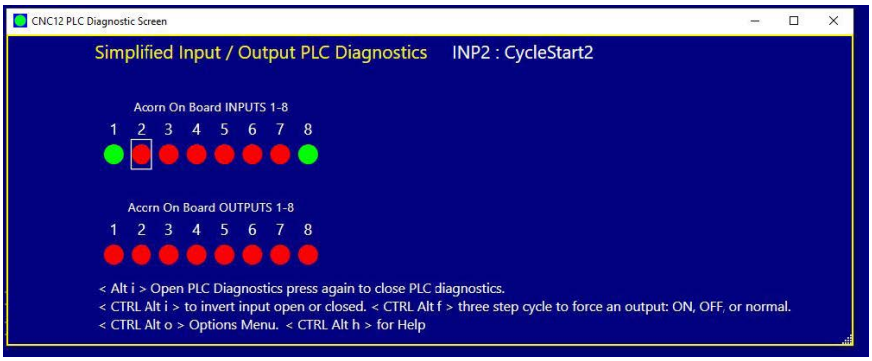
From the main screen of CNC12, start the PLC Diagnostic Screen by pressing the keys **ALT** and **i** at the same time.



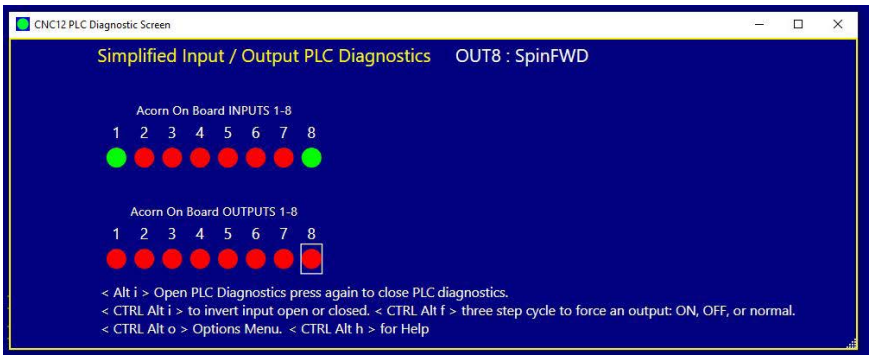
To exit from the Input and Output screen, press the keys **ALT** and **i** again at the same time.

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.

Draft: January 11, 2024

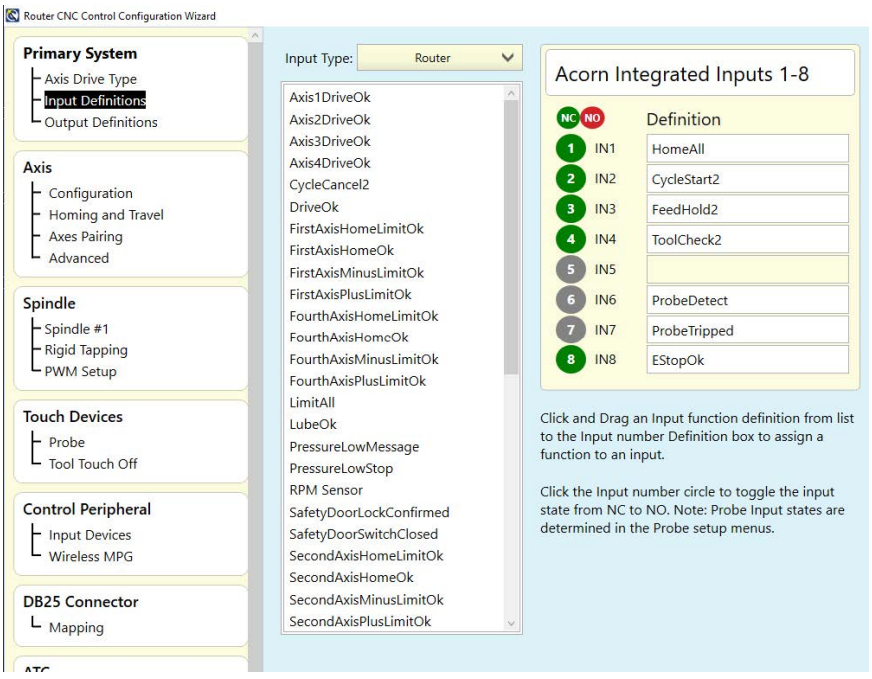


Using the mouse to select an output causes a thin, white-lined box to appear around the virtual LED and the assigned PLC Input function name for that input to be displayed. In the example below, Output relay 8 has been assigned the Spindle Forward PLC function named SpindFWD.



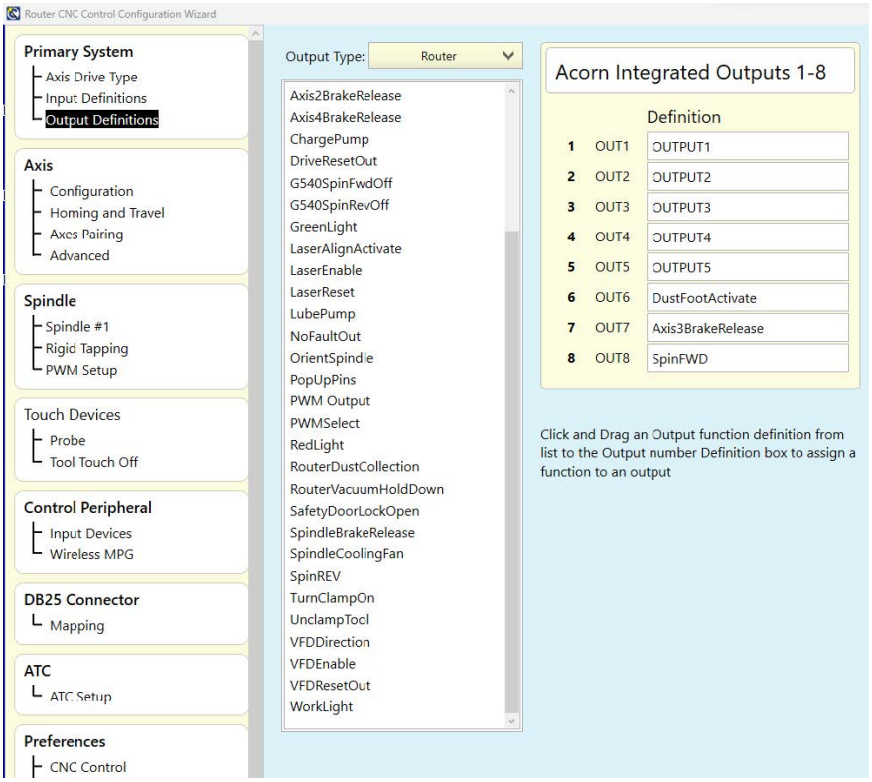
The PLC function assignment to the inputs and outputs are made with the CNC Control Setup Wizard.

Below is the Wizard PLC Function Input Definitions setup menu for some of the examples we are using in this section:

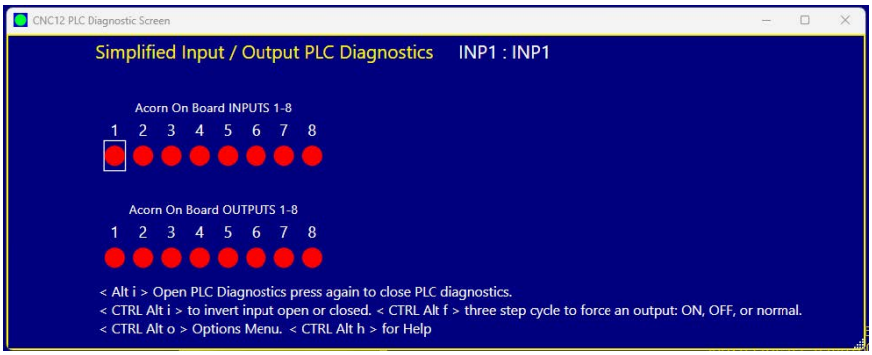


Draft: January 11, 2024

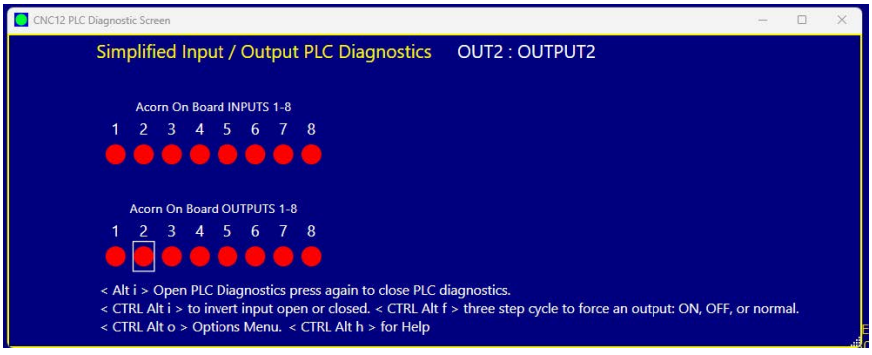
Below is the Wizard PLC Function Output Definitions setup menu for some of the examples we are using in this section:



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.



Draft: January 11, 2024

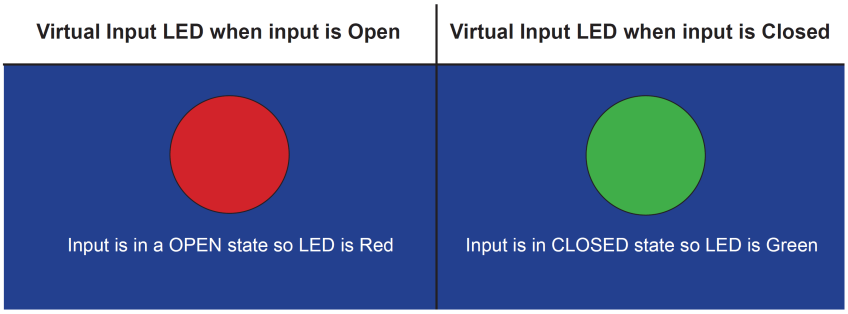
Even if an input or output is not assigned to a PLC function, they can still be observed and manipulated with the PLC Diagnostic App.

11.7.2 Observing the Inputs and Outputs

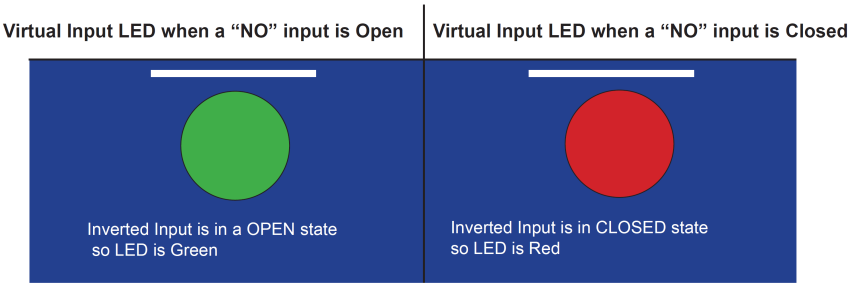
Input LED colors explained

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).

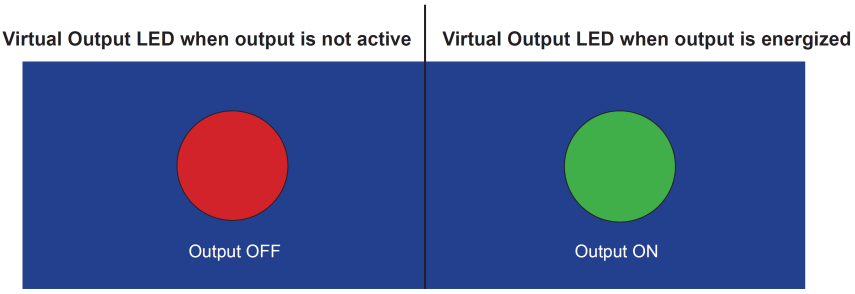


A green Virtual LED with a line above it is inverted and indicates that the Input is “open”. A red Virtual LED with a line above it is inverted and indicates that the Input is “closed”

Note: Whenever possible, use a NC type switch or sensor for CNC applications.

Draft: January 11, 2024

11.7.3 Observing Outputs



11.7.4 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 non-inverted state. A double-click with 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 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.

Draft: January 11, 2024

Any output can be manually activated or deactivated whether it has been assigned a PLC function or not.

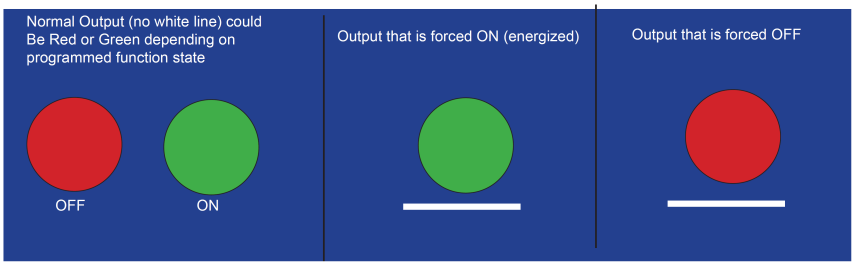
⚠ WARNING Be careful if there is high power equipment connected to the CNC control equipment that will be activated when using the PLC Diagnostic App to manually energize an output.

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.

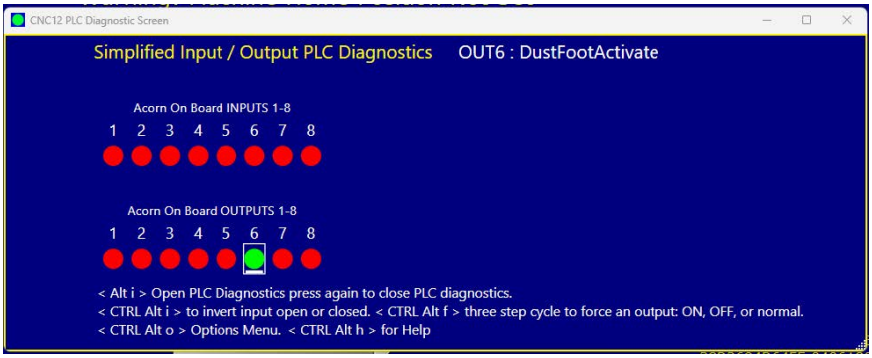
Pressing **CTRL**, **ALT**, and **f** at the same time again deactivates the output.

Pressing **CTRL**, **ALT**, and **f** at the same time again returns the output to the normal programmed state.

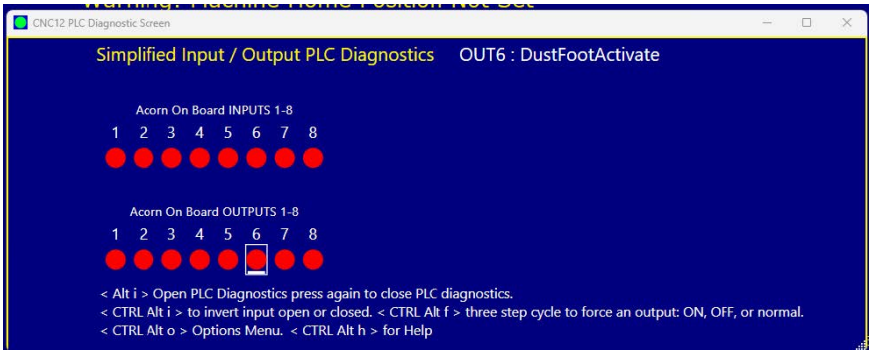
A white line will appear below an Output that has been manually manipulated.



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.



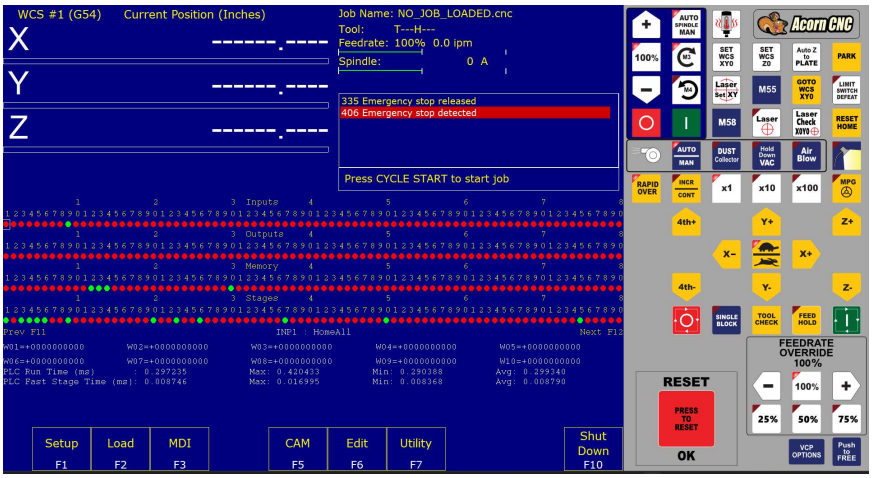
Draft: January 11, 2024

Pressing **CTRL**, **ALT**, and **f** again at the same time to return the output to its normal programmed state.

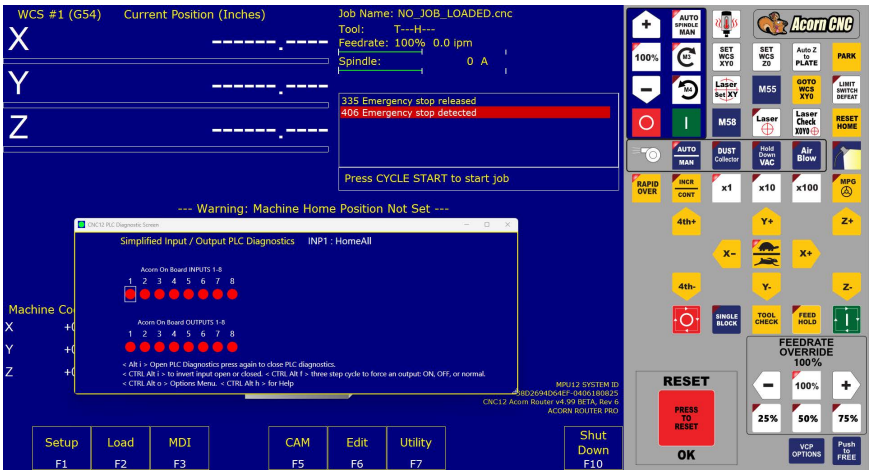
Be sure to return the output to its normal state (no white line) for normal CNC control operation.

There are two PLC diagnostic tools available with CNC12: simplified and original.

The Original looks like the image below and displays additional information compared to the simplified version.

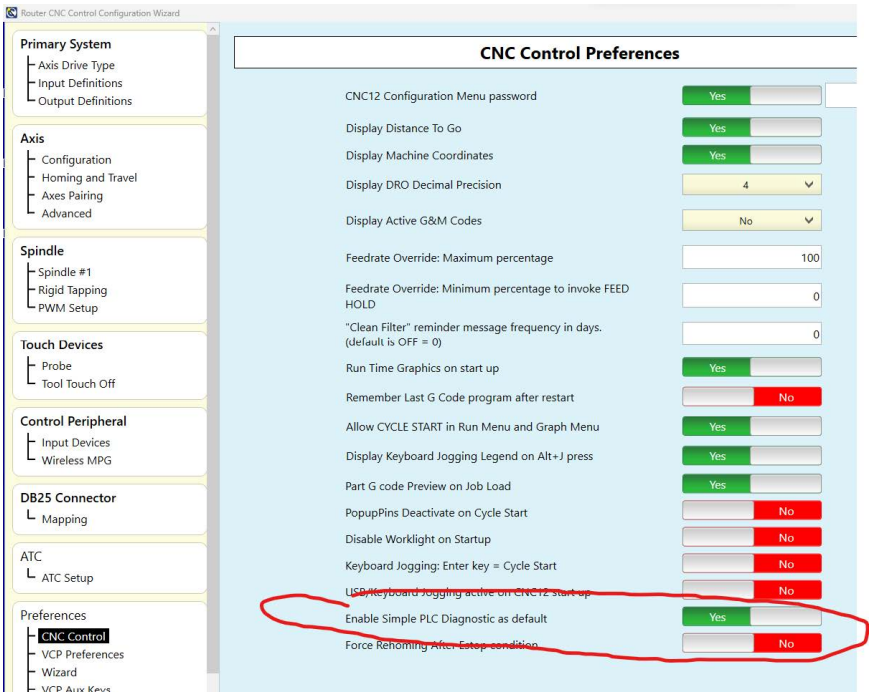


The simplified PLC Diagnostic App looks like the image below and is the default selection for Acorn and AcornSix.



Controls for which PLC Diagnostic tool is the default are found in the Acorn and AcornSix Wizard under “Preferences”→“CNC Control”

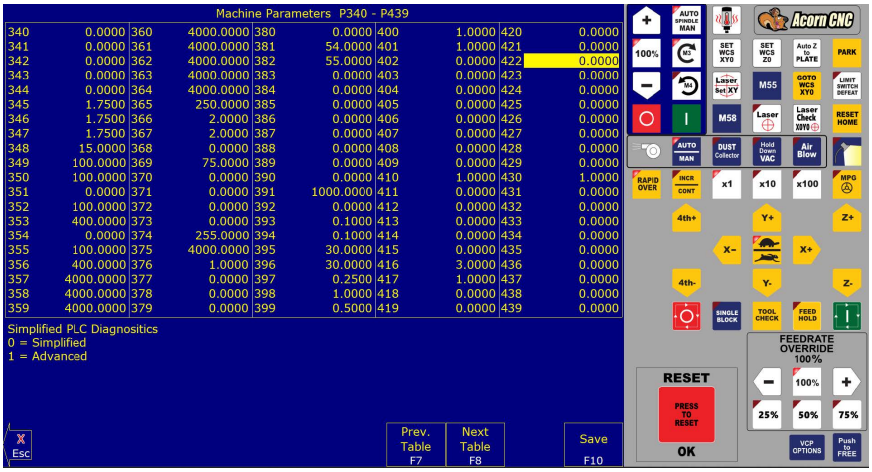
Draft: January 11, 2024



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.

11.8 Smoothing Configuration Parameters

Parameter	Description	Recommended Values

Draft: January 11, 2024

216	**Lash Type:** users have a choice for backlash compensation to occur in the PC or MPU11 card itself. This is a new feature and is still under testing, use PC-side comp for now.	PC Side = 1, MPU11 Side = 0	
220	Turns 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 acceleration checking. Values 1 and 3 (bit 0 = 1) limits Centripetal to only linear axes. Values 2 and 3 (bit 1 = 1) enables excessive axis acceleration checking.	0 (Centripetal stage will operate on all axes and disables excessive axis acceleration 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 Feed Rate 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 feed rates in arcs and corners are still satisfactory.	1.0 (default value) 0.1 to 5.0 (Depending on the Operator’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 feed rate 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 the Operator’s preference for quickness of accelerations / decelerations)	

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

Draft: January 11, 2024

parameters will not have to be re-entered after they have been entered initially.

11.9 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 with this feature.

To understand why Smoothing is a useful tool, we should 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 that 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 feed rates, 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.

11.9.1 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.

Draft: January 11, 2024

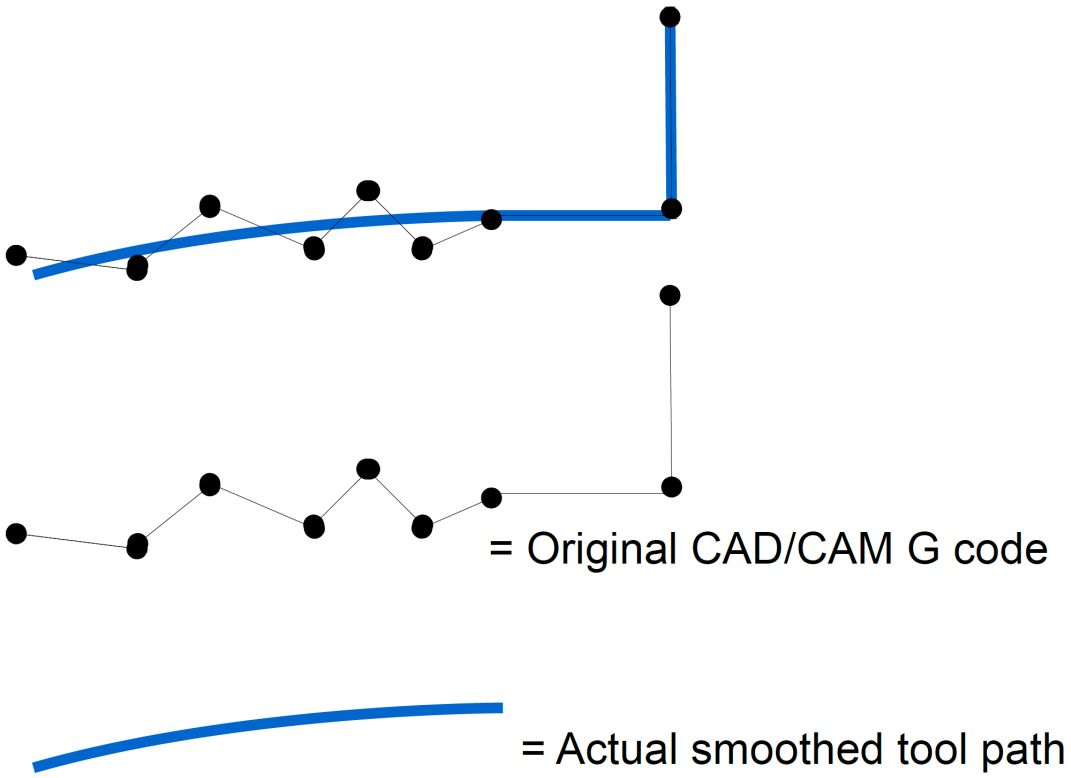


Figure 11.1

11.9.2 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

Draft: January 11, 2024

Figure 11.2: Image credit to Scott (aka “Sword”) on the Centroid forums

11.9.3 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.

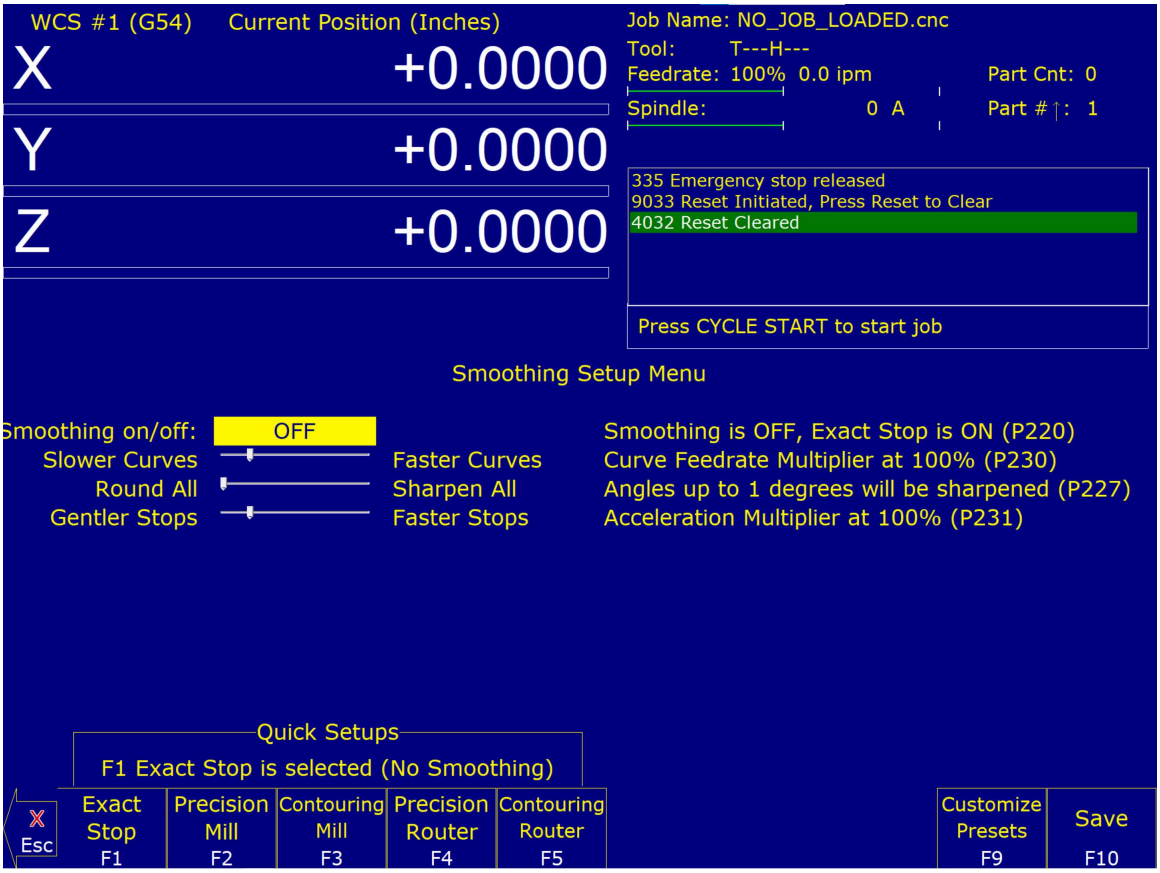


Figure 11.3

11.10 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 feed rate 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.

Draft: January 11, 2024

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.

11.10.1 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.

11.10.2 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.

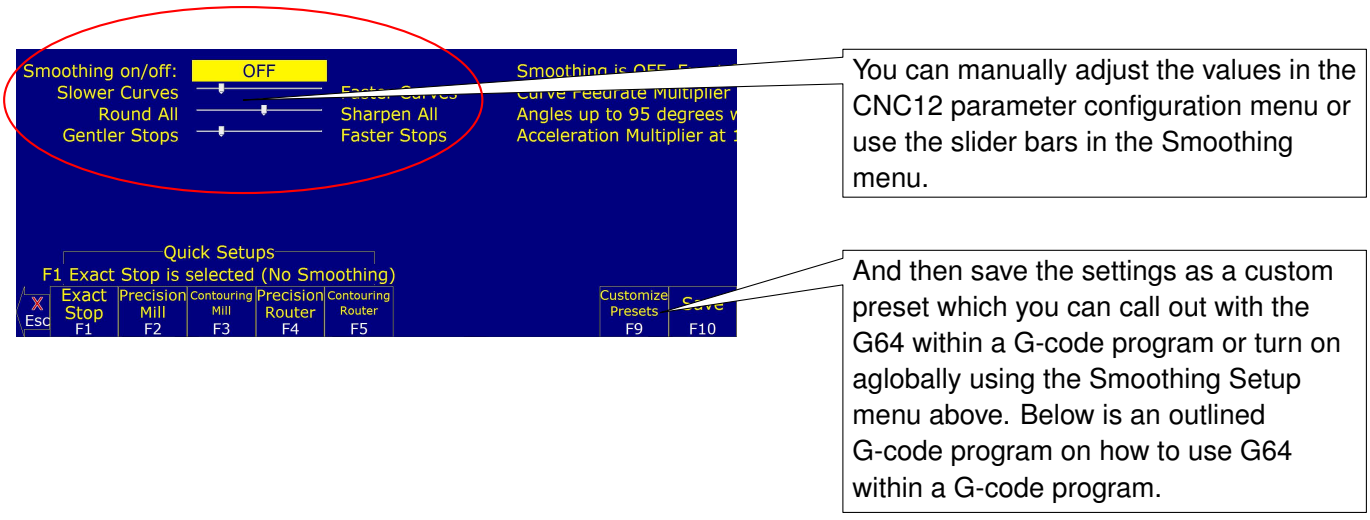


Figure 11.4

Once finished, save these settings as a custom preset. This preset can be called with G64 within a G-code program or

Draft: January 11, 2024

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
```

11.11 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 Parameters 221 through 231 (excluding 229) 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.

11.11.1 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/G3 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.

11.11.2 Typical Smoothing Values for a Precision Milling Machine

(where maximum accuracy and a little G-code smoothing is desired)

Draft: January 11, 2024

```
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.
```

11.11.3 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: Acceleration values in PID work with Smoothing as well. Increase these values for softer transitions from one straight line feed rate to another.

11.12 Technical Background of AD2 Smoothing

Smoothing performs several related functions:

11.12.1 Smoothing NBPTS (P221) and STEP (P222)

These parameters control smoothing of the user-supplied G-code. Smoothing allows significantly higher feed rates 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 11.5). Smoothing's strength is also a potential

Draft: January 11, 2024

disadvantage, as it modifies geometry and rounds corners (see Figure 11.6). 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.

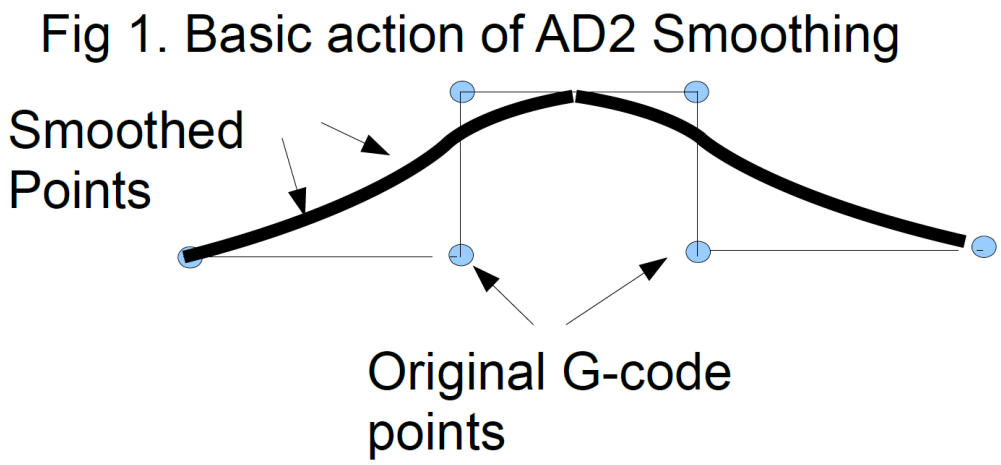


Figure 11.5

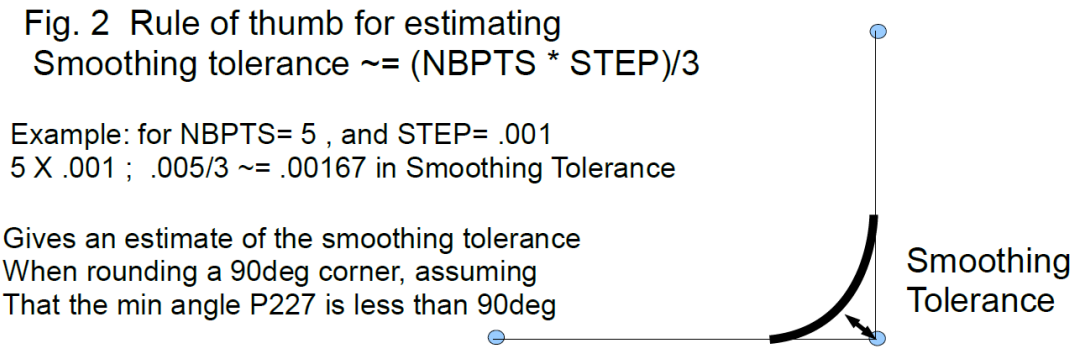


Figure 11.6

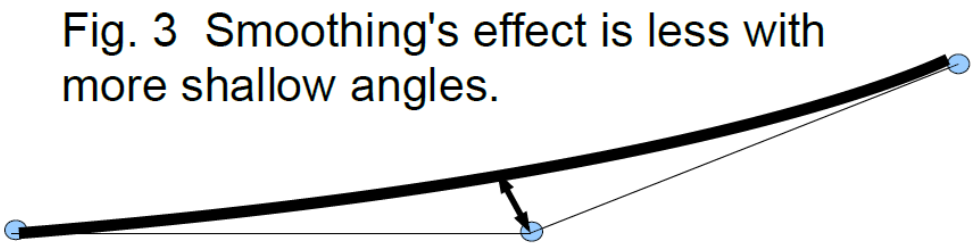


Figure 11.7

Draft: January 11, 2024

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”.

Fig. 5 Min_Angle allows or inhibits smoothing

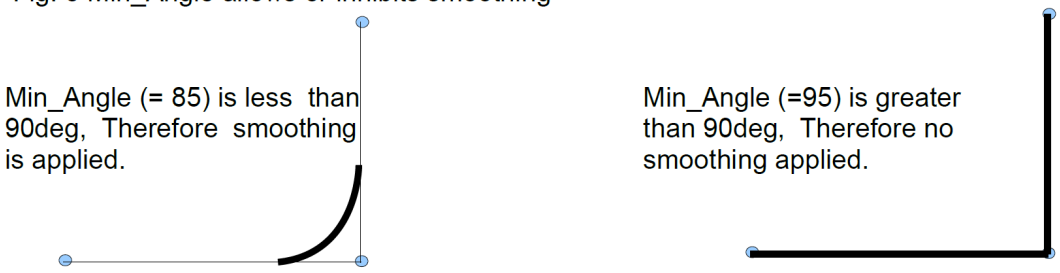


Figure 11.8

11.12.2 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.

Fig 6. W affects Min_Angle

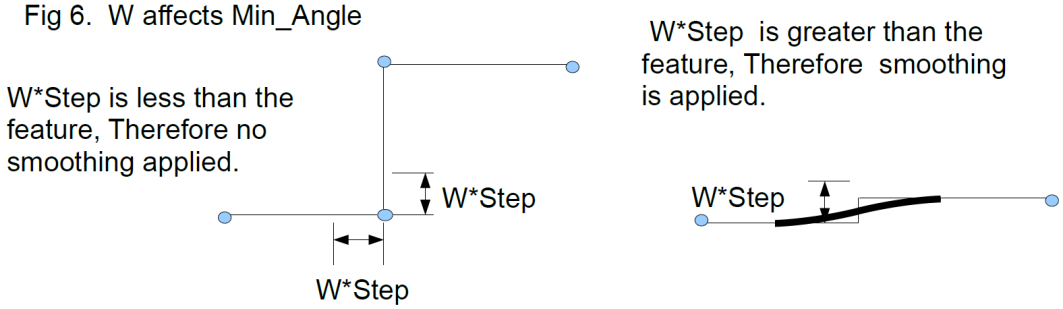


Figure 11.9

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

Draft: January 11, 2024

Fig. 7 S-curve

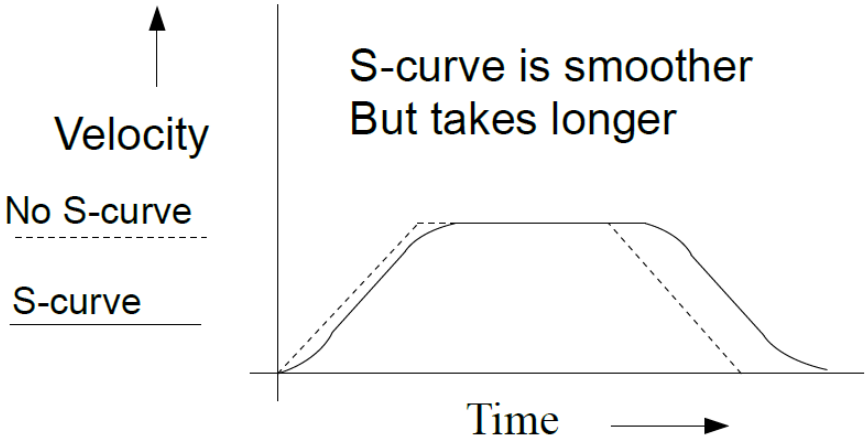


Figure 11.10

Draft: January 11, 2024

11.13 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.

Draft: January 11, 2024

Installation Instructions for Acorn/AcornSix Operators:

1. Remove the MPG finger knob from the battery compartment and install onto the 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 the USB transmitter into the USB port on the CNCPC. Reboot the CNCPC at this point.
4. Start CNC12 and verify that a Pro or Ultimate License is installed. Open the Centroid Wizard Tab #4 “Control Peripherals” and select “WMPG-4”. Press “Write Settings to CNC Control Configuration”. Follow the instructions on the screen.
5. Turn on the 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: $\times 1$, $\times 10$, or $\times 100$.
9. Rotate the MPG wheel clockwise for positive direction axis motion.

Installation Instructions for Oak, Allin1DC, and MPU11 Operators:

1. Remove the MPG finger knob from the battery compartment and install onto the 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 the USB transmitter into the USB port on the CNCPC. Reboot the CNCPC at this point.
4. Start CNC12 and open the CNC12 Utility Menu. Press “Options”, “Import License”, and install a Pro or Ultimate license.
5. Press **F1 – Setup**, **F3 – Config**, then **F3 – Parameters** (the password is 137). Then, set the MPG CNC12 Parameter #218 = 15 for four-axis Mills/Routers, #218=7 for three-axis Mills/Routers, and #218 = 3 for Lathes.
6. Set the MPG CNC12 Parameter #348 = 15 (MPG ON) and Parameters #349 & #350 = 100 (100 steps-per-revolution). Set Parameter # 411 = 1 (WMPG-4) and press **F10 – Save**. Exit out of the Parameters.
7. Shut Down and restart CNC12 for the new parameters to take effect.
8. Turn on the 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: $\times 1$, $\times 10$, or $\times 100$.
11. Rotate the MPG wheel Clockwise for positive direction axis motion.

Draft: January 11, 2024

Notes:

1. The MPG wheel finger knob is located in the battery compartment. Remove and install the knob 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 three feet AWAY from the MPG itself. The WMPG Operational range is 3'–100'
4. The LCD Screen has a protective plastic film on it. Use a clean fingernail to lift it up at the corner and remove.
5. A "No RF" message means MGP is not seeing the transmitter/receiver. The most common cause is that the Windows PC did not mount the USB transmitter. Solution: Move the transmitter to a different USB port and reboot the PC.
6. For Acorn/AcornSix: The value for the $\times$1 Jog Increment for the MPG wheel is set in the Wizard Axis configuration menu. The default value for Plasma and Router is .001" or 0.0254 mm. For Mill and Lathe, the default value is .0001"
7. For Oak/Allin1DC/MPU11: Z-axis speed limiter: Inhibit fast Z-axis MPG moves by setting P450 = 1 =ON $\times$100 = $\times$10. To disable the Z-axis speed limiter again, set P450 = 0 =OFF $\times$100 = $\times$100 (v5.0+).

Draft: January 11, 2024

11.14 Custom WMPG Macros

11.14.1 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, such as lights, dust collection, lube pump shot, vac hold down, etc.

11.14.2 Three Easy Steps

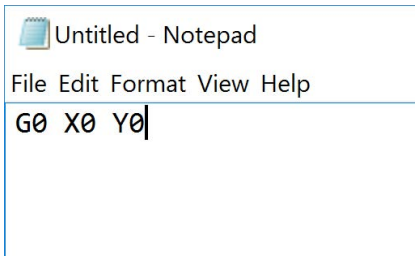
1. Create a G-&M-code program that you want to run when an auxiliary key is pressed.
2. Save it as a custom M-code using the proper custom M-code name.
3. Map the new custom M-code to a spare auxiliary key using the auxiliary key parameters.

Example:

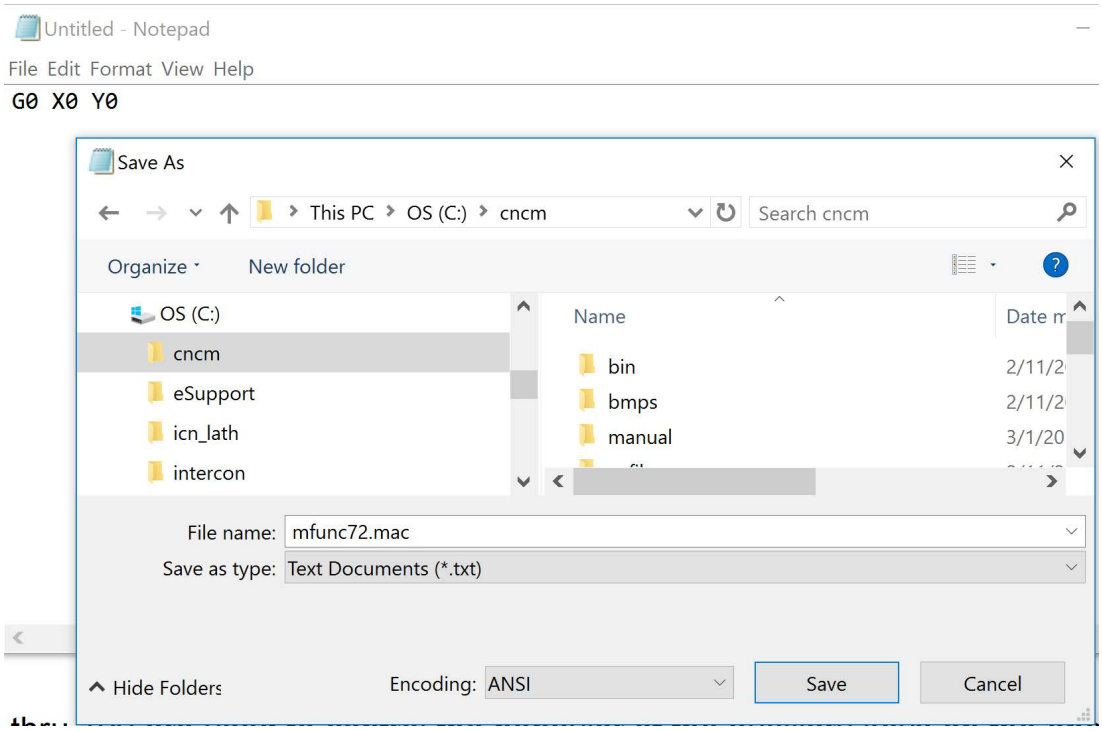
Open any text editor and create the G-&M-code program you would like to execute when pressing an Aux key. For this example, we will 'program' Aux Key #5.

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 a text editor and create a program with "G0 X0 Y0" as the first line

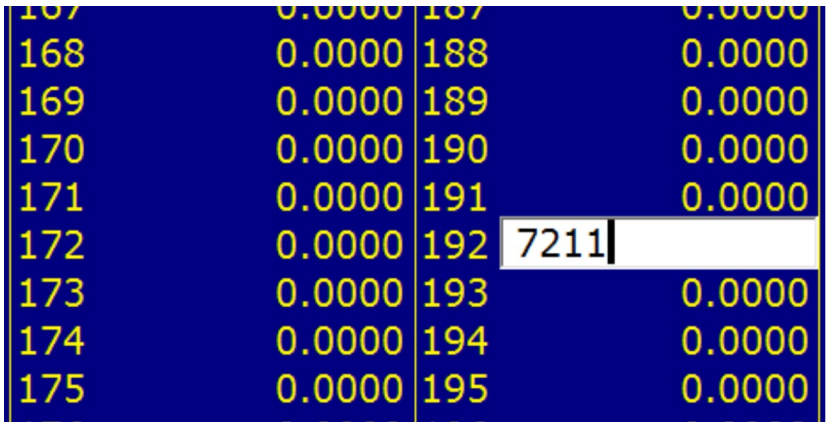


2. Save this G-&M-code program with the file name mfunc72.mac in the CNC Software install directory, typically `+c:\cnct`.



3. **Map the custom M-code to an auxiliary key**. Press **F1 – Setup**, **F3 – Config**, and then **F3 – Parameters**. Navigate to P192. Set P192 to 7211 and 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 M-code.

   Note: Parameters 188 through 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 that 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:

Draft: January 11, 2024

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 by 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) through creating a report by pressing **F7 – Utility** then **F7 – Create report** from the main menu. You can always "restore" the report to bring the machine back to its 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.

11.14.3 Using the Canned Auxiliary Key Functions

Auxiliary key Parameters 188 thru 199 also have "canned" functionality that can be activated by simply setting the parameter value to any of the 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 – Rect. 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.

Draft: January 11, 2024

11.14.4 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 section into your editor and save it 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 this file 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 the X- and Y-axes 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
```

Draft: January 11, 2024

Acorn

Acorn systems allow easy access to the PLC Function file via the Acorn Wizard. The Acorn Wizard can be opened from the main screen of the CNC Software by pressing 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 four "edit MPG macro" buttons will open up a text editor for the macro code of 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
```

12 CNC Software Messages

12.1 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/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/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/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 the 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.	Set the PC internal clock to the correct time.
199	CNC started	The CNC control software has started.	No action required.

12.2 Messages Issued Upon Exit from the CNC Software

Error	Message	Cause & Effect	Action
201	Exiting CNC due to a known error (ACORN only)	MPU11 is not responding, or the `mpu11.hex`, `mpu.plc` files are 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` files.	Delete corrupted files and reboot software. Contact dealer.
204	Exiting CNC. . . Normal Exit	The CNC control software is shutting down normally.	No action required.
222	Autotune run	Autotune has been run.	No action required.

12.3 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.	No action required.
302	Moving. . .	Motors are moving while a CNC program is running.	No action required.
303	Paused. . .	Motion is paused while a CNC program is running (FEED HOLD).	No action required.
304	MDI. . .	The CNC software is running in MDI mode.	No action required.

Draft: January 11, 2024

Error	Message	Cause & Effect	Action
305	Processing...	The CNC software is running in a mode other than MDI.	No action required.
306	Job Finished	A normal end of the CNC program.	No action required.
307	Operator abort: job canceled	ESC or CYCLE CANCEL was pressed. Job is cancelled.	No action required.
308	Waiting for input #NN	M100 or M101 is executing. The [rogram will continue once the specified input opens or closes.	No action required.
309	Waiting for CYCLE START button	M0, M1, M100/75, or Block Mode was executed.	Press Cycle Start
310	Waiting for output #NN	M100 or M101 is executing. The program will continue once the specified output opens or closes.	No action required.
311	Waiting for memory #NN	M100 or M101 is executing. The program will continue once the specified memory bit changes to the correct state.	No action required.
312	Waiting for PLC operation (Mnn)	The PLC program is not clearing the PLC operation in progress.	No action required.
313	Waiting for dwell time	G4 executing. The program will wait for the specified dwell time and then continue.	No action required.
314	Waiting for system #NN	M100 or M101 is executing. The program will continue once the specified PLC system variable changes to the correct state.	No action required.
315	Searching...	A run/search is currently in progress.	No action required.
317	Waiting for automatic tool change	`mfunc6.mac` is executing.	No action required.
318	Operator abort probing cancelled	ESC or CYCLE CANCEL was pressed while doing a probing move.	No action required.
319	Probing cycle cancelled	The probing cycle was cancelled.	No action required.
320	Probe stuck	The probe is stuck or hit an object when it was not expecting contact.	No action required.
322	Stall: probing cancelled	The probing was cancelled because of a stall.	No action required.
323	Stall: job cancelled	The job was cancelled because of a stall.	No action required.
324	Limit: probing cancelled	The probing was cancelled because of a limit error.	No action required.
325	Limit: job cancelled	The job was cancelled because of a limit error.	No action required.
326	Fault: probing cancelled	The probing was cancelled because of a fault.	No action required.
327	Fault: job cancelled	The job was cancelled because of a fault.	No action required.
328	Cutter comp error: job cancelled	The job was cancelled because of a cutter comp error.	No action required.
329	Invalid parameter: job cancelled	The job was cancelled because of an invalid parameter.	No action required.

Draft: January 11, 2024

Error	Message	Cause & Effect	Action
330	Canned cycle error: job cancelled	The job was cancelled because of a canned cycle error.	No action required.
331	Threading error: job canceled	The programmed threading move will cause an axis to exceed its maximum rate.	No action required.
332	Search Failed	A run/search was unable to find the requested G-code line.	No action required.
334	Locating position to resume job...	A run/search is currently locating the job continuation point in the program.	No action required.
335	Emergency Stop Released	The Emergency Stop Button has been released.	No action required.
336	Digitize cancelled	ESC or CYCLE CANCEL was pressed during digitizing.	No action required.
337	Digitize complete	A digitizing routine ran to completion.	No action required.
338	Job Cancelled	ESC or CYCLE CANCEL was pressed during the job run.	No action required.
339	Jogging...	An axis jog key is currently pressed and the machine is moving the corresponding axis.	No action required.
340	Limit (#__) cleared	A previously-tripped limit switch is now in the “untripped” position.	No action required.
341	Probing Cycle Finished	A probing cycle ran to completion.	No action required.
342	Waiting for motion to stop	The PC is waiting for the MPU11 to complete its motion.	No action required.
343	Waiting for stop reason reset	A PC is waiting for the MPU11 to reset the stop reason (as part of the PC/MPU11 communications handshake).	No action required.
344	Feed Rate modified due to spindle	The effective feed rate 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).	No action required.
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).	No action required.
346	Waiting for spindle direction	Job progress is paused until the spindle turns in the commanded direction.	No action required.

Draft: January 11, 2024

12.4 Abnormal Stops (Faults)

Abnormal stops are detected in the following order: PLC, servo drive, spindle drive, lube, and then ESTOP. This means that if both the servo drive and 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 the inverter for a fault or reset the spindle contactor OCR, then cycle the EMERGENCY STOP.
405	Lubricant level low	MPU11 stopped with low lube fault bit set. The current job will finish, but nothing will work after that.	Add lube or check the low lube switch wiring, then cycle the EMERGENCY STOP.
406	Emergency Stop detected	MPU11 stopped with no fault bits set. Job cancelled.	Release the Emergency Stop button.
407	limit (#__) tripped	MPU11 stopped with limit switch tripped. Job cancelled.	Clear the limit switch.
408	Programmed action timer expired	M103's time expired before M104 was encountered. Job cancelled.	Find out why the timer expired before the specified action was completed.

Draft: January 11, 2024

Error	Message	Cause & Effect	Action
410	_ axis position error	A position error greater than one motor encoder turn was 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 backwards. 2. Noise is getting into the system via the motor cables (the line integrity has been violated). 3. An encoder error has occurred.	1. Try to slow jog the motor and watch the DRO position. If the position on the DRO goes opposite of the direction indicated on the jog button, then the motor is wired 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 for awhile at the maximum rate using the fast jog buttons (check the fast jog rate in the motor jog parameters screen to make sure that it is set to be equal to the maximum motor rate). If the motor seems to jump around rather than accelerate and decelerate smoothly, then you are probably facing 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 both the motor and encoder.

Draft: January 11, 2024

Error	Message	Cause & Effect	Action
411	_ axis full power without motion	90% Power (a PID Output greater than 115) is applied to any axis and no motion greater than 0.0005 inches is detected for more than the time specified in Parameter 61 (the default is .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 up against a physical stop. 2. The servo drive has shut down 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 slow jog mode to move the axis away from the stop. Determine and set the software travel limits to stop the machine before it runs into the physical stop. 2. If the axis is not on a physical stop, check for a tripped limit switch. If one is tripped, 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 that the switch's input signal 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 of the direction causing the error and clear all of the limit switches. Jog toward the direction causing the error, and if no motion occurs, then this indicates that a servo drive failure has occurred.
412	_ axis encoder differential error	An error condition was detected in the differential signal levels for this axis encoder. This may indicate a loose/severed encoder cable or a bad encoder. This will stop all motion and cancel the job.	Reconnect/replace the encoder or encoder cable.
417	Abnormal end of job	Job was ended without reason.	
418	Search Line or Block not found	The requested search input data was not found in the loaded CNC file.	Type in the correct search input data or load the correct job.

Draft: January 11, 2024

Error	Message	Cause & Effect	Action
419	Search line in embedded subprogram	The requested search line is found, but is part of an embedded/extracted subprogram.	Use another line number.
420	_ axis motor overheating	The CNC software estimates that one or more motors have reached the warning temperature (set in Parameter 29). Either the motor/s are overheating or the temperature file is corrupted. The job will be cancelled.	Contact dealer. Determine what is causing the motor/s to overheat, or delete the `cnc.tem` file and reboot.
421	Motor(s) too hot: job canceled	The CNC software estimates that one or more motors have reached the limit temperature (set in Parameter 30). The job will not be able to run until the motor/s cools down.	Contact dealer. Determine what is causing the motor/s to overheat or delete the `cnc.tem` file and reboot.
422	Check Jog Panel cable	Jog panel failure or loose cable.	Reconnect the jog panel cable.
428	Check MPG cable	MPG failure, loose cable, or it is turned off.	Reconnect the MPG cable and turn the axis selector knob to an axis.
434	_ idling too high: Releasing power	The axis is not moving and no job is running, but the axis has stopped against some abnormal resistance. Power is released to the motors.	Run Autotune to adjust the motor settings.
435	_ axis runaway: Check motor wiring	Motor was in a runaway fault condition. Power to the motor will be automatically shut off.	Check the motor wiring.
436	Servo drive shutdown	This error message is produced by the 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 detected is shown with the servo drive's LEDs. Once the servo drive detects this error condition, it stops all motion and removes power to the motors. The servo drive indicates the presence of this condition to the CNC software via the servo drive fault input to the PLC.	On DC systems, check the status of the servo drive's LEDs and check fibers 4&5. If this message is displayed on an AC system, check that P178 bit 4 is set.

Draft: January 11, 2024

Error	Message	Cause & Effect	Action
437	Servo power removed	An axis was moving more than 300 RPM while the power was supposed to be off. 1. The motor may be wired backwards. 2. There may be a shorted servo drive. 3. The axis motion was canceled, but the motor continued to move due to inertia. This is probably caused by an unbalanced axis. Power to the motors is released.	Check the motor wiring, servo drive, or look at the Kg value in the PID and make sure it is 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). The job is cancelled.	Check Parameter 34 for an incorrect plus/minus sign in front of the encoder counts.
439	_ axis servo drive data output error	Logic power failure or loss of communication from the drive to the MPU11.	Is the logic LED on? Check the fiber optic cables to the drive. For SD1 drives, make sure that the bus cables are shielded and as short as possible. Power the unit down and check the drive connections.
441	_ axis overvoltage	The input power has gone higher than 340VDC. The control board will shut down the drive and remove power. The motor brake will engage for five seconds in this condition.	Check to make sure that the input voltage is below 340VDC. If not, the incoming VAC needs to be lowered.
442	_ axis undervoltage	The drive's input power is less than 80 VDC.	Check the supply voltage.
443	_ axis commutation encoder bad	The control detected an invalid commutation zone value	Perform a motor Move Sync in the Drive Menu. A Zero (0) or Seven (7) is an invalid zone. Check for: 1. A wiring problem in the encoder cable or motor end cap (such as broken encoder wires). 2. The encoder cable shield being connected at the motor end, when it should not be. 3. A bad encoder. 4. Motor power cable shields not being connected properly. 5. A drive that is not grounded properly.

Draft: January 11, 2024

Error	Message	Cause & Effect	Action
444	_ axis overtemperature detected	The drive overtemp sensor was tripped. The motor will not be given power.	Either the drive is being run at overcapacity, the cooling fan is not functioning, or the cooling fan's airflow is blocked.
445	_ axis overcurrent detected	Overcurrent is detected on an axis. The motor will not be given 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. The motor will not be given power.	Check the fiber optic cables. Verify continuity between the drive chassis, ground strip, and Earth ground.
447	_ axis (#) bad index pulse detected	Noise was picked up by an encoder cable or misaligned encoder. The motor will not be given power.	Remove the noise or align the encoder.
449	Manual movement detected in restricted area	Unexpected manual movement of the axis was 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 the 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 drive with a higher current. But, if this message appears in a continuous stream, then something is seriously wrong. You should hit the 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. This 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 Ethernet cable.

Draft: January 11, 2024

Error	Message	Cause & Effect	Action
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 Ethernet cable.
454	axis scale encoder differential error	An error condition was detected in the differential signal levels for this axis scale encoder. This may indicate a loose/severed encoder cable or a bad encoder. This will stop all motion and cancel the job.	Reconnect/replace the scale encoder or scale encoder cable.
455	axis encoder quadrature error	The axis encoder skipped a transition state in its count-up/count-down sequence. This may indicate a loose/severed encoder cable or a bad encoder. This will stop all motion and cancel the job.	Reconnect/replace the encoder or encoder cable.
456	axis scale encoder quadrature error	The scale encoder skipped a transition state in its count-up/count-down sequence. This may indicate a loose/severed encoder cable or a bad encoder. This will stop all motion and cancel the job.	Reconnect/replace the 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 the encoder or encoder cable if the move was seeking an index pulse. Check that the hard stop was not broken off or overrun.
461	Spindle axis is not set	An operation aborted because the spindle axis parameter (35) has an incorrect value.	Contact dealer.
470	brake wattage exceeded	The brake wattage was exceeded on the indicated ACDC drive.	Contact dealer.

Draft: January 11, 2024

12.5 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 the invalid G-code.
503	Invalid M-function on line NNNNN	Invalid M-function encountered on CNC line. Job cancelled.	Correct the invalid M-code.
504	Invalid parameter on line NNNNN	Invalid or missing number after letter. Job cancelled.	Correct the program.
505	Invalid value on line NNNNN	Value out of range (T, H, D). Job cancelled.	Correct the program.

Error	Message	Cause & Effect	Action
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 the missing quotation mark.
508	Macro nesting too deep	The macro nesting limit was exceeded on an attempt to invoke a subroutine. Job cancelled.	Create a second program.
509	Option not available	An attempt was made 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 the number of arguments.
511	Missing parameter	A parameter is required or expected but not found. Job cancelled.	Correct the program.
513	Expected "="	There was an error in the expression to the left of "=", a missing "=", or an orphaned parameter. Job cancelled.	Correct the equation.
514	Empty expression	The expression contains no operands. Job cancelled.	Correct the expression.
515	Syntax error in expression	An illegal character was in the number, variable, or function. Job cancelled.	Correct the program.
516	Unmatched bracket (parenthesis)	Brackets or parentheses are paired improperly or misplaced. Job cancelled.	Correct the program.
517	Evaluation stack overflow	Brackets or parentheses are nested too deeply. Job cancelled.	Correct the program.
518	Undefined variable	The variable name does not exist. Job cancelled.	Correct the program.
519	Too many variables	The space allotted for user-defined variables has been exceeded. Job cancelled.	Correct the program.
520	Invalid variable name	The variable name contains an illegal character. Job cancelled.	Correct the program.
521	Divide by zero	There was an attempt to divide by zero. Job cancelled.	Correct the program.
522	Domain error	An imaginary number would result (there was a square root of a negative number). Job cancelled.	Correct the program.
523	Invalid value in assignment	There was an attempt to assign an illegal value to a system variable. Job cancelled.	Correct the program.
524	Variable is read-only	There was an attempt to assign a value to a read-only system variable. Job cancelled.	Correct the program.
525	Missing P value	A P parameter was expected but was missing	Correct the program.
526	M22x Missing initial variable	M224 or M225 were not immediately followed by a #variable reference.	See M224 and G225
527	M22x initial variable parse error	M224 or M225 were immediately followed by an invalid #variable reference.	Correct the program.

Draft: January 11, 2024

Error	Message	Cause & Effect	Action
528	M225 String variable not allowed	M225 was immediately followed by a string #variable (which is invalid). Only numeric variables are allowed here.	Correct the program.
529	M225 invalid variable	The #variable specified after the M225 was not valid, or not readable due to a machine error.	Correct the program.
530	M224 invalid variable	The #variable specified after the M224 was read-only, or not writeable due to a machine error.	Correct the 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 the program.
535	M22x invalid format string	The format string contained invalid format codes.	Correct the program.
536	M22x missing format specifier	The format code was missing its specifier.	Correct the program.
537	M22x Missing Argument	A format code was specified in the format string, but its corresponding #variable argument was missing.	Correct the 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 the 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 the 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 the program.
542	M22x character limit exceeded	The resulting formatted string after all of the format codes were processed was too long.	Correct the program.
543	Missing L parameter	An L-code was missing	Correct the program.
544	Too many axes	More than one axis was specified with M128 OR the Simultaneous Contouring feature is not enabled. Without the Simultaneous Contouring feature, a maximum of three 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	A parse error occurred because the value was out of range.	Correct the value.

Draft: January 11, 2024

Error	Message	Cause & Effect	Action
547	Move by counts not allowed	Cutter comp (G41/G42) was on when M128 was specified.	Issue G40 (Cutter compensation 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 program 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.	

12.6 Cutter Compensation Errors

Error	Message	Cause & Effect	Action
601	Error: no compensation in MDI	G41 or G42 was entered while in MDI. MDI is not canceled, but the cutter compensation does NOT go into effect. The remainder of the line is processed.	Do not use G41 or G42 in MDI.
603	Arc as first uncomp. move on line NNNNN	An arc was specified as the first move after the end of compensation (G40). Job cancelled.	The first move after G40 must be a linear move.
605	Canned cycle not allowed on line NNNNN	A canned cycle was attempted during compensation. Job cancelled.	Do not use cutter compensation with canned cycles.
606	G53 not allowed on line NNNNN	A G53 was attempted during compensation. Job cancelled.	Choose a different work coordinate.
607	Set home not allowed on line NNNNN	A M26 was attempted during compensation. Job cancelled.	Do not use M26 with cutter compensation
608	Ref. point move not allowed on line NNNNN	A G28, G29, or G30 was attempted during compensation. Job cancelled.	Do not use return points with cutter compensation.

12.7 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 was specified. Job canceled (H0 cannot be set; it is always zero).	Change H to a valid value.

12.8 Canned Cycle Errors

Draft: January 11, 2024

Error	Message	Cause & Effect	Action
801	Error: No R-point on line NNNNN	No R-value was specified. Job cancelled.	Add an R-point.
802	Error: Q = 0 on line NNNNN	A Q-value of 0 was specified (Q is used for G73 and G83 only). Job cancelled.	Insert a non-zero Q-value.
803	Error: No Z-point on line NNNNN	No Z-value was specified for the canned cycle. Job cancelled.	Add a Z-value.
804	Error: Ggg invalid on line NNNNN (gg = 76, 86, 87, 88)	An unimplemented canned cycle was requested. Job cancelled.	Change to a valid G-code.
805	Error: No Q-value on line NNNNN	A Q-value was not specified for G73 or G83. Job cancelled.	Insert a Q-value.
806	Error: No P-value on line NNNNN	A P-value (dwell time) was not specified for G82 or G89. Job cancelled.	Add a P-value.

12.9 Miscellaneous Errors/Messages

Error	Message	Cause & Effect	Action
901	Ref. point invalid on line NNNNN	A G30 with an invalid P-value (must be 1 or 2) was issued. Job cancelled.	Change the P-value to 1 or 2.
902	No prior G28 or G30 on line NNNNN	A G29 with no preceding G28 or G30 was issued.	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.	Insert subroutines into MDI.
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 ncfiles directory.

Draft: January 11, 2024

Error	Message	Cause & Effect	Action
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	Probe tripped when a cycle did not expect contact.	
917	Invalid tilt lookup table	The tilt lookup table file (`tilt.tab`) has an invalid format or was not found.	
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.	
922	Out of memory	Problem allocating memory.	
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, but 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. . .	Reduce the nesting level of subprograms.
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 (140) has been changed.	No action required.
930	Log Level Configuration file not found. . . Creating new configuration.	The log level configuration file was not found. A default file will be created.	No action required.
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 it did not exist, so a new log file has been created.	

Draft: January 11, 2024

Error	Message	Cause & Effect	Action
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.
936	_ axis (#) scale enabled	A scale is enabled for this axis and compensation was enabled. This happens after homing the axis.	Not Applicable
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.

12.10 Configuration Modification Messages

Error	Message	Cause & Effect
111	__ modified: __ → __	An axis configuration parameter was modified.
444	__ modified: __ → __	A servo drive configuration parameter was modified.
555	__ modified: __ → __	A PID configuration parameter was modified.
556	Axis converted: → __	A PID configuration parameter was converted.
777	__ modified: __ → __	An axis configuration parameter was modified.
999	Parm # modified: → __	A machine parameter was modified.

Draft: January 11, 2024

13 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

Draft: January 11, 2024

Individual Control Component Manuals

PLCADD1616, PLC Expansion Board Users Manual

Add4AD4DA, Analog Output Expansion Board Users Manual

Encoder Expansion Board User Manual

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 Interface)

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

Draft: January 11, 2024

Swing Arm Tool Changer User Manual

Umbrella Tool Changer User Manual

System Test

TTL2DIFF, Single-ended to Differential Signal Converter

8RELBRD, Eight-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

Draft: January 11, 2024

14 Glossary

Filename and Location	Type	Description, Purpose, and Related Notes	Applications that Read to, Write from, or Backup from the file
cnct\cnct.exe	Executable	CNC12 program	N/A
cnct\CNC12ColorPicker.exe	Executable	CNC12 color customization and theme settings App, starts from the Utility menu.	N/A
cnct\CNC12PLCDiagnosticScreen.exe	Executable	PLC diagnostics App. This provides a simplified status view and manual control of inputs and outputs	N/A
cnct\cnccfgutilw.exe	Executable	Centroid configuration file utility – translates binary files into a readable form via the command prompt Run this from command promt C:\ For example if you wanted to see what was in cncm.tl: • Run cd\cncm to change directory • Run cncfigutilw.exe-dt Run cncfgutilw.exe-h or -? for more info	N/A
cnct\CNCWizard.exe	Executable	Acorn, AcornSix and Hickory CNC controller setup App	N/A
cnct\KeyboardJogPanel.exe	Executable	Handles a subset of control features present in the virtual control panel accessible via keyboard controls	N/A
cnct\MpgClient.exe	Executable	Handles Centroid Wireless MPG Control Pendant operations	N/A
cnct\mpucomp.exe	Executable	MPU compiler that takes PLC program source code and compiles is to be usable by CNC12	N/A
cnct\plcdetective.exe	Executable	Programmable logic controller (PLC) I/O debugging tool. See PLC detective users manual for more info	N/A

Draft: January 11, 2024

Filename and Location	Type	Description, Purpose, and Related Notes	Applications that Read to, Write from, or Backup from the file
cnct\plot.exe	Executable	Plasma Torch Voltage visual plot utility. Also compatible with Mill and Lathe for technicians to view position error, voltage data, etc. See Plasma User Manual for more info.	N/A
cnct\uninstall.exe	Executable	Allows the user to uninstall CNC12	N/A
cnct\VirtualControlPanel.exe	Executable	The VCP (virtual operator control panel) App.	N/A
cnct\CncSkinning.dll	Application extension	Centroid API. Dynamic Link Library file for CncSkinning	-VCP Reads -Wizard Reads
cnct\CncSkinningUserControls.dll	Application extension	Centroid API, Dynamic Link Library file for CncSkinning	-PLC Reads
cnct\ColorPicker.dll	Application extension	Dynamic Link Library file for ColorPicker	-CNC12 Reads -ColorPicker Reads, Writes -PLCDiagnostic Reads
cnct\Svg2Xaml.dll	Application extension	Dynamic Link Library file for Svg2Xaml	-VCP Reads -Wizard Reads
cnct\cncfdr.bin	Binary file	Writes to the CNC flight data recorder. This logs various events, messages, job errors, and other data useful for debugging purposes	-CNC12 Reads
cnct\systest.cnc	Text file	CNC controller basic setup testing program. See Miscellaneous CNC Documents under CNC Manuals in the Tech Support section of the Centroid website for more information	-CNC12 Reads
* cnct\pathl.ini	Text file	Stores paths in the cnct directory for later use	-CNC12 Reads, Writes
* cnct\mpu.dbg	Text file	Compiled by mpucomp.exe. Contains motion processing unit debugging information	-CNC12 Writes
cnct\crazy_cnc_squirrel.svg	SVG file	Stores an image of a squirrel typically used with Centroid software	-CNC12 Reads
cnct\acorn.hex	Binary file	Firmware file. This file writes the firmware to the hardware (Acorn, AcornSix, Hickory)	-CNC12 Writes
cnct\cnct.hom	Text file	Machine Homing program, G and M codes.	-CNC12 Reads
* cnct\centroid.ico	Image file	Centroid logo icon	-CNC12 Reads
cnct\cnc_wiz.ico	Image file	CNC Wizard icon	-CNC12 Reads
cnct\lathe.ico	Image file	CNC12 Lathe icon	-CNC12 Reads

Draft: January 11, 2024

Filename and Location	Type	Description, Purpose, and Related Notes	Applications that Read to, Write from, or Backup from the file
cnct\profile.ico	Image file	CNC12 Profile Manager icon	-CNC12 Reads
* cnct\plc.map	Text file	This file is generated automatically when a PLC program is compiled. And contains information about the i/o mapping and symbol table of the PLC program.	-CNC12 Writes
* cnct\cnctch.mac	Text file	Macro. Checks if currently equipped tool is a valid tool	-CNC12 Reads
* cnct\mfunc3.mac	Text file	Macro. Default. Turns of Spindle CCW and turns on Spindle CW	-CNC12 Reads
* cnct\mfunc4.mac	Text file	Macro. Default. Turns of Spindle CW and turns on Spindle CCW	-CNC12 Reads
* cnct\mfunc7.mac	Text file	Macro. Sets Auto Coolant Flood On	-CNC12 Reads
* cnct\mfunc8.mac	Text file	Macro. Sets Auto Coolant Mist on	-CNC12 Reads
* cnct\mfunc10.mac	Text file	Macro. Set ClampOn (Mill) or ChuckOpen and reset ChuckClose (Lathe)	-CNC12 Reads
* cnct\mfunc11.mac	Text file	Macro. Set ClampOn (Mill) or ChuckOpen and reset ChuckClose (Lathe)	-CNC12 Reads
* cnct\mfunc13.mac	Text file	CutOff (User Customizable)	-CNC12 Reads
* cnct\mfunc14.mac	Text file	Turn Off CutOff (User Customizable)	-CNC12 Reads
* cnct\mfunc15.mac	Text file	Tool Unclamp	-CNC12 Reads
* cnct\mfunc16.mac	Text file	Tool Clamp	-CNC12 Reads
* cnct\mfunc18.mac	Text file	ATC Initialization	-CNC12 Reads
* cnct\mfunc19.mac	Text file	Spindle Orientation	-CNC12 Reads
* cnct\mfunc20.mac	Text file	Turn Spindle Orientation Off	-CNC12 Reads
* cnct\mfunc21.mac	Text file	Index Turret 1 Tool	-CNC12 Reads
* cnct\mfunc22.mac	Text file	PartchuteIn (User Customizable)	-CNC12 Reads
* cnct\mfunc23.mac	Text file	Turn off PartchuteIn (User Customizable)	-CNC12 Reads
* cnct\mfunc27.mac	Text file	Set VacuumOn	-CNC12 Reads
* cnct\mfunc28.mac	Text file	Turn off VacuumOn	-CNC12 Reads
* cnct\mfunc32.mac	Text file	Turn on TailStockInOut	-CNC12 Reads
* cnct\mfunc33.mac	Text file	Turn off TailStockIn, Turn on TailStockOut	-CNC12 Reads
* cnct\mfunc34.mac	Text file	User Customizable Macro	-CNC12 Reads
* cnct\mfunc35.mac	Text file	Set DustCollectionOn	-CNC12 Reads
* cnct\mfunc36.mac	Text file	Reset DustCollectionOn	-CNC12 Reads
* cnct\mfunc41.mac	Text file	Selects Low Range Spindle	-CNC12 Reads
* cnct\mfunc42.mac	Text file	Selects Med Low Range Spindle	-CNC12 Reads
* cnct\mfunc43.mac	Text file	Selects Med High Range Spindle	-CNC12 Reads
* cnct\mfunc44.mac	Text file	Selects High Range Spindle	-CNC12 Reads
* cnct\mfunc45.mac	Text file	Sets All Axes to Part 0	-CNC12 Reads
* cnct\mfunc46.mac	Text file	Unused Macro	-CNC12 Reads

Draft: January 11, 2024

Filename and Location	Type	Description, Purpose, and Related Notes	Applications that Read to, Write from, or Backup from the file
* cnct\mfunc47.mac	Text file	Unused Macro	-CNC12 Reads
* cnct\mfunc48.mac	Text file	User Customizable Macro	-CNC12 Reads
* cnct\mfunc49.mac	Text file	Sets specified axis to 0 part 0	-CNC12 Reads
* cnct\mfunc50.mac	Text file	Sets all axes to part 0	-CNC12 Reads
* cnct\mfunc51.mac	Text file	User Customizable Macro	-CNC12 Reads
* cnct\mfunc52.mac	Text file	User Customizable Macro	-CNC12 Reads
* cnct\mfunc53.mac	Text file	User Customizable Macro	-CNC12 Reads
* cnct\mfunc54.mac	Text file	User Customizable Macro	-CNC12 Reads
* cnct\mfunc55.mac	Text file	User Customizable Example Macro	-CNC12 Reads
* cnct\mfunc56.mac	Text file	User Customizable Example Macro	-CNC12 Reads
* cnct\mfunc57.mac	Text file	User Customizable Example Macro	-CNC12 Reads
* cnct\mfunc58.mac	Text file	User Customizable Example Macro	-CNC12 Reads
* cnct\mfunc59.mac	Text file	Resets home for mill and lathe	-CNC12 Reads
* cnct\mfunc60.mac	Text file	PARK for mill and lathe	-CNC12 Reads
* cnct\mfunc61.mac	Text file	Turns on Acorn Output 1	-CNC12 Reads
* cnct\mfunc62.mac	Text file	Turns on Acorn Output 2	-CNC12 Reads
* cnct\mfunc63.mac	Text file	Turns on Acorn Output 3	-CNC12 Reads
* cnct\mfunc64.mac	Text file	Turns on Acorn Output 4	-CNC12 Reads
* cnct\mfunc65.mac	Text file	Turns on Acorn Output 5	-CNC12 Reads
* cnct\mfunc66.mac	Text file	Turns on Acorn Output 6	-CNC12 Reads
* cnct\mfunc67.mac	Text file	Turns on Acorn Output 7	-CNC12 Reads
* cnct\mfunc68.mac	Text file	Turns on Acorn Output 8	-CNC12 Reads
* cnct\mfunc69.mac	Text file	Move all axes to machine zero	-CNC12 Reads
* cnct\mfunc70.mac	Text file	Sets an axis to zero	-CNC12 Reads
* cnct\mfunc71.mac	Text file	Checks axes for tripped home	-CNC12 Reads
* cnct\mfunc72.mac	Text file	Homes the 3rd axis	-CNC12 Reads
* cnct\mfunc73.mac	Text file	Homes the independent axis	-CNC12 Reads
* cnct\mfunc74.mac	Text file	Automatically Home and Square paired axes	-CNC12 Reads
* cnct\mfunc75.mac	Text file	Automatically pairs axes	-CNC12 Reads
* cnct\mfunc81.mac	Text file	Turns off Acorn Output 1	-CNC12 Reads
* cnct\mfunc82.mac	Text file	Turns off Acorn Output 2	-CNC12 Reads
* cnct\mfunc83.mac	Text file	Turns off Acorn Output 3	-CNC12 Reads
* cnct\mfunc84.mac	Text file	Turns off Acorn Output 4	-CNC12 Reads
* cnct\mfunc85.mac	Text file	Turns off Acorn Output 5	-CNC12 Reads
* cnct\mfunc86.mac	Text file	Turns off Acorn Output 6	-CNC12 Reads
* cnct\mfunc87.mac	Text file	Turns off Acorn Output 7	-CNC12 Reads
* cnct\mfunc88.mac	Text file	Turns off Acorn Output 8	-CNC12 Reads
* cnct\mfunc89.mac	Text file	Turns off multiple outputs	-CNC12 Reads
cnct\language.msg	Text file	Stores translations of messages displayed by the software	-CNC12 Reads
* cnct\mpu.plc	Text file	Compiled PLC program	-PLC Reads
cnct\CNC12LatheDebugStart	Shortcut	Opens CNC12 in debug mode	N/A

Draft: January 11, 2024

Filename and Location	Type	Description, Purpose, and Related Notes	Applications that Read to, Write from, or Backup from the file
* cnct\acorn_lathe_plc.src	Text file	Used by Wizard to automatically generate a PLC program based on users input and output selections.	-Wizard Reads
cnct\mt.stats	Binary file	Stores the CNC machine timers' data	-Wizard Reads
* cnct\mpu.sym	Text file	Configures a list of variables and states for the MPU	-CNC12 Reads
* cnct\xplc.sym	Text file	Configures a list of variables and states for the PLC	-CNC12 Reads
* cnct__improper_shutdown__.txt	Text file	Logs information on the last improper shutdown of CNC12, which is defined as the application closing before machine home is set	-CNC12 Writes
* cnct_screen-size.txt	Text file	Display proportions of the CNC12 application window	-CNC12 Writes
cnct\clientlocked.txt	Text file	Used by CNC12 to either reject or accept incoming commands based on whether the file exists or does not exist respectively. CNC12 creates clientlocked.txt if: • The machine has not been homed • A job or MID is active • The control is in any screen other than the main screen • A fault condition is present CNC12 will delete clientlocked.txt as long as none of the conditions above are met	-CNC12 Reads, Writes -Wizard Reads
* cnct\ColorPicker.txt	Text file	Information on the selected color profile used by CNC12	-PLC Writes
* cnct\cyclone_status.txt	Text file	Stores various stats pertaining to error handling	-PLC Writes
* cnct\debug_dump0.txt	Text file	Indicates the date and time that the latest debug session started	-CNC12 Writes
* cnct\eth1616log.txt	Text file	Logs messages pertaining to the Acorn Ether1616 board's ethernet connection	-CNC12 Reads -Wizard Writes
* cnct\hsc_status.txt	Text file	Stores the values of hardware variables while high-speed cutting	-PLC Writes

Draft: January 11, 2024

Filename and Location	Type	Description, Purpose, and Related Notes	Applications that Read to, Write from, or Backup from the file
* cnct\mpu_info.txt	Text file	Information on the Motion Processing Unit last used by CNC12	-CNC12 Writes
cnct\msg_cnt.txt	Text file	Log file containing the number of instances of each message that occurred during CNC12 runtime and each message's respective code	-CNC12 Writes
cnct\msg_log.txt	Text file	Log file containing the date and time that a message occurred, its respective code, and its priority level. For more information on software messages, see Mill Operator Manual Chapter 12	-CNC12 Writes
* cnct\plcmsg.txt	Text file	Centroid provided (stock) PLC message file. PLC programmer creates this file. Text file for PLC related error message. This file contains the text that appears beside a given error number in the CNC12 message window. This file is 'matched' to the Centroid supplied (aka stock) PLC programs and can easily be edited by user to tailor any error message text to match the application. Message codes that can be thrown by a Programmable Logic Controller	-PLC Reads
* cnct\tasklist.txt	Text file	List of all tasks running during the last instance of CNC12	-CNC12 Writes
* cnct\threadsdb.txt	Text file	List of all unique threads and information pertaining to each thread in CNC12	-CNC12 Reads, Writes
* cnct\cnct.ttl	Text/Binary file	Tool file for the Lathe	-CNC12 Reads
* cnct\cnct.wcs	Text/Binary file	Stores coordinate data for each axis (Requires cnccfgutilw.exe to view)	-CNC12 Writes
cnct\start.bat	Script file	Checks the connectivity of specified IP addresses before opening designated cnc applications (cncm.exe, cnct.exe, etc)	Used by CNC12
cnct\compile.cmd	Script file	Used by Notepad++ to compile PLC programs within notepad. See TB #294 for more info.	Used by Notepad++

Draft: January 11, 2024

Filename and Location	Type	Description, Purpose, and Related Notes	Applications that Read to, Write from, or Backup from the file
* cnct\CncSkinning.xml	XML file	Centroid API data required and used by the CNCSkinning library	-CNC12 Reads, Writes
* cnct\cnct.job.xml	XML file	File that records the last part program/job performed by CNC12	-CNC12 Writes
* cnct\cnct.prm.xml	XML file	Parameters used by CNC12 to configure itself to different machines. See operator manual for more information. Wizard writes these parameters automatically for Acorn, AcornSix, Hickory.	-CNC12 Writes
* cnct\cnctcfg.xml	XML file	CNC12 main machine system configuration menu data and PID settings.	-CNC12 Reads, Writes
* cnct\dsp_probe_setting.xml	XML file	Information pertaining to the DSP probe used by CNC12	Reads/Writes
cnct\job.xml	XML file	File that records the last part program/job performed by CNC12	-CNC12 Writes
cnct\log_levels.xml	XML file	Information on the logs last sent in CNC12, including the code, description, and frequency	-CNC12 Writes
* cnct\mpu_info.xml	XML file	Information on the Motion Processing Unit last used by CNC12	-CNC12 Writes
* cnct\PLCDiagnosticsOptions .xml	XML file	PLC diagnostic App. Saved options for App Size, location, on top etc.	-PLC App Reads, Writes
* cnct\Svg2Xaml.xml	XML file	Documentation for the Parser file that converts SVG files into XAML, allowing CNC12 to import from SVGs	-VCP Reads -Plasma Intercon Reads -Wizard Reads
* cnct\threadsdb.xml	XML file	List of all unique threads and information pertaining to each thread in CNC12	-CNC12 Reads, Writes
All .bmp files in cnct\bmps	Image file	Image files used by CNC12 software. User editable. Open with any image edited, edit and save.	-CNC12 Reads
All .html files in cnct\CncSkinningDocumentation\html\search	HTML file	Centroid API documentation. HTML files by the Centroid API documentation. CncSkinningDocumentation webpage search function	N/A
All .js files in cnct\CncSkinning Documentation\html\search	Script file	Centroid API documentation. JavaScript files used by the CncSkinningDocumentation webpage search function	N/A

Draft: January 11, 2024

Filename and Location	Type	Description, Purpose, and Related Notes	Applications that Read to, Write from, or Backup from the file
cnct\CncSkinningDocumentation\html\search\search.css	Text file	Centroid API documentation. Visual properties used by the CncSkinningDocumentation webpage search function	N/A
cnct\CncSkinningDocumentation\html\search\search_l.png,search_m.png,search_r.png	Image file	Centroid API documentation. The images used to display the search bar for the webpage	N/A
cnct\CncSkinningDocumentation\html\search\close.png	Image file	Centroid API documentation. The image used to display the search bar close icon for the webpage	N/A
cnct\CncSkinningDocumentation\html\search\mag_sel.png	Image file	Centroid API documentation. The image used to display the search bar magnifying glass icon for the webpage	N/A
All .png files in cnct\CncSkinningDocumentation\html	Image file	Centroid API documentation. The images used throughout the CncSkinningDocumentation webpage	N/A
All .html files in cnct\CncSkinningDocumentation\html	HTML file	Centroid API documentation. HTML files used by the CncSkinningDocumentation webpage	N/A
All .js files in cnct\CncSkinningDocumentation\html	Script file	Centroid API documentation. JavaScript files used by the CncSkinningDocumentation webpage	N/A
cnct\CncSkinningDocumentation\html\tabs.css	Text file	Centroid API documentation. Visual properties used by the CncSkinningDocumentation webpage	N/A
cnct\CncSkinningDocumentation\html\myStyle.css	Text file	Centroid API documentation. Text alignment property used by the CncSkinningDocumentation webpage	N/A
cnct\CncSkinningDocumentation\html\doxygen.css	Text file	Centroid API documentation. Layour properties used by the CncSkinningDocumentation webpage	N/A
cnct\CncSkinningDocumentation\Documentation	HTML file	Centroid API documentation. Shortcut for the CncSkinningDocumentation webpage	N/A
cnct\licenses\MSTest.TestAdapter_1.3.2.html	HTML file	License information on Microsoft Test Adapter in HTML format	N/A
cnct\licenses\MSTest.TestFramework_1.3.2.html	HTML file	License information on Microsoft Test Framework in HTML format	N/A

Draft: January 11, 2024

Filename and Location	Type	Description, Purpose, and Related Notes	Applications that Read to, Write from, or Backup from the file
cnct\licenses\netDxf_2022.11.2.html	HTML file	License information on the netDXF library in HTML format	N/A
cnct\licenses\Svg2Xaml_0.3.0.5.html	HTML file	License information on the Svg2Xaml library in HTML format	N/A
cnct\licenses\clipper_library.txt	Text file	License information on the Clipper library in raw text format	N/A
cnct\licenses\ColorPicker.txt	Text file	License information on the ColorPicker library in raw text format	N/A
cnct\licenses\NestingLibPort.txt	Text file	License information on the NestingLibPort library in raw text format	N/A
cnct\licenses\Svg2Xaml_0.3.0.7.txt	Text file	License information on the Svg2Xaml library in raw text format	N/A
cnct\manual\centroid_acorn_install_manual.pdf	PDF file	Installation manual for the Centroid Acorn CNC controller	N/A
cnct\manual\centroid-cnc12-lathe-operator-manual.pdf	PDF file	Operator manual for CNC12 Mill	N/A
cncm\reportbackup\report*.zip		Report.zip file is a 'snap shot' of the current setup and configuration of the CNC control system. This file has a wide variety of uses and contains all the machine and control specific settings that make the cnc controller work with the machine tool application.	-CNC12 Writes
cncm\reportbackup\report*.txt	Text file	A text file that contains a summary of information about the PC used, software installed, state of the CNC machine, job files loaded, user variables, machine parameters, tool files, and file system directories.	-CNC12 Writes
All .cnc files in cnct\ncfiles (See also \laser and \mm)	Text file	.nc, .cnc, .tap, .txt, .dig, .cam are Part G-code programs used by CNC12	-CNC12 Reads
cnct\resources\ATC\Default\cnctch.mac	Text file	Default ATC Macro used by the Wizard. This macro that handles a tool change operation on a fixed carousel.	Wizard Reads
cnct\resources\ATC\Universal\cnctch.mac	Text file	ATC Macro that handles a tool change operations for the Wizard supported types of ATC's	Wizard Reads
* cnct\resources\colors\CentroidClassic.xml	XML file	CNC12 color profile. Contains the hex values for all colors in the Centroid Classic color palette	-ColorPicker Reads

Draft: January 11, 2024

Filename and Location	Type	Description, Purpose, and Related Notes	Applications that Read to, Write from, or Backup from the file
* cnct\resources\colors\Dark HiContrast.xml	XML file	CNC12 color profile. Contains the hex values for all colors in the Dark Hi Contrast color palette	-ColorPicker Reads
* cnct\resources\colors\Dark Theme.xml	XML file	CNC12 color profile. Contains the hex values for all colors in the Dark Theme color palette	-ColorPicker Reads
* cnct\resources\colors\Grey Theme.xml	XML file	CNC12 color profile. Contains the hex values for all colors in the Grey Theme color palette	-ColorPicker Reads
cnct\resources\homing\ClearPathHardStop\cnct.hom	Text file	Teknic Clearpath Hard Stop Homing program.	-Wizard Reads
cnct\resources\homing\Header.txt	Text file	Default code used by the Wizard to auto generate a home program based on user selections. Code to check for an open home switch across four different axes	-Wizard Reads
cnct\resources\homing\HomeAllHeader.txt	Text file	Default code used by the Wizard to auto generate a home program based on user selections. Code to check for an open home switch across any axis	-Wizard Reads
* All .xml files in cnct\resources\vcp\Buttons\&XML file	Visual properties for the respective Virtual Control Panel button. See the VCP users manual for details.	-VCP Reads -Wizard Reads	
* All .svg files in cnct\resources\vcp\Buttons\&SVG file	Image resource for the respective Virtual Control Panel button. See the VCP users manual for details.	-VCP Reads -Wizard Reads	
* cnct\resources\vcp\Buttons\ButtonTemplates.xaml	XML file	Button template information that can be applied to any Virtual Control Panel button. See the VCP users manual for details.	-VCP Reads -Wizard Reads
* cnct\resources\vcp\Buttons\template.xml	XML file	Visual properties of the Virtual Control Panel button template. See the VCP users manual for details.	-VCP Reads -Wizard Reads

Draft: January 11, 2024

Filename and Location	Type	Description, Purpose, and Related Notes	Applications that Read to, Write from, or Backup from the file
* All .svg files in cnct\resources\vcp\images\&SVG file	Various image resources for the Virtual Control Panel. See the VCP users manual for details.	-VCP Reads -Wizard Reads	
* cnct\resources\vcp\skins\acorn_lathe_vcp_legacy_skin.vcp	XML file	Centroid Provided VCP Legacy Virtual Control Panel skin. See the VCP users manual for details.	-ColorPicker Reads, Writes -VCP Reads -Wizard Reads
* cnct\resources\vcp\skins\acorn_lathe_vcp_skin.vcp	XML file	Centroid Provided VCP Virtual Control Panel skin. See the VCP users manual for details.	-ColorPicker Reads, Writes -VCP Reads -Wizard Reads
* cnct\resources\vcp\options.xml	XML file	Layout properties for the Virtual Control Panel skins. See the VCP users manual for details.	-ColorPicker Reads, Writes -VCP Reads -Wizard Reads
All .xml files in cnct\resources\wizard\default\drives\&XML file	Axis motor drive information for the respective servo/stepper motor, used by the Acorn and AcornSix Wizard.	-Wizard Reads	
All .mac files in cnct\resources\wizard\default\macros	Text file	Macros identical to those in the base directory that serve as restorable states in the case that the base directory macros were modified	-Wizard Reads, Backup
cnct\resources\wizard\default\plc\acorn_universal_template.src	Text file	PLC (Programmable Logic Controller) Wizard Template file for both CNC Mill and Lathe	-Wizard Reads
cnct\resources\wizard\default\plc\functions.xml	XML file	Functions for use by the Wizard for the canned Inputs and Outputs available in the Wizard for easy drag and drop setting of the i/o for Acorn, AcornSix and Hickory.	-Wizard Reads
cnct\resources\wizard\default\plc\lathe_plcmsg.txt	Text file	Used by Wizard. Default PLC logic messages. Lathe PLC messages that match the Centroid provided Acorn, AcornSix, Hickory Wizard generated PLC Programs.	-Wizard Reads

Draft: January 11, 2024

Filename and Location	Type	Description, Purpose, and Related Notes	Applications that Read to, Write from, or Backup from the file
cnct\resources\wizard\default\plc\mill_plcmsg.txt	Text file	Used by Wizard. Default PLC logic messages. Mill PLC messages that match the Centroid provided Acorn, AcornSix, Hickory Wizard generated PLC Programs.	-Wizard Reads
cnct\resources\wizard\default\plc\plasma_plcmsg.txt	Text file	Used by Wizard. Default PLC logic messages. Plasma PLC messages that match the Centroid provided Acorn, AcornSix, Hickory Wizard generated PLC Programs.	-Wizard Reads
cnct\resources\wizard\default\plc\router_plcmsg.txt	Text file	Used by Wizard. Default PLC logic messages. Router PLC messages that match the Centroid provided Acorn, AcornSix, Hickory Wizard generated PLC Programs.	-Wizard Reads
cnct\resources\wizard\default\template\CentroidCNC.pref	Text file	File containing default preferences for the Contour ShuttleXpress to integrate with CNC12	Used by ShuttleXpress
* cnct\system\MPGmacro1.mac	Text file	Wireless MPG-4/6 Macro 1 button. User customizable macro	-Wizard Reads
* cnct\system\MPGmacro2.mac	Text file	Wireless MPG-4/6 Macro 2 button. User customizable macro	-Wizard Reads
* cnct\system\MPGmacro3.mac	Text file	Wireless MPG-4/6 Macro 3 button. User customizable macro	-Wizard Reads
* cnct\system\MPGmacro4.mac	Text file	Wireless MPG-4/6 Macro 4 button. User customizable macro	-Wizard Reads
* cnct\system\park.mac	Text file	Custom Park Macro that controls the **F1 – Park** command action found in the CNC12 **F10 – Shut Down** and on the VCP	-Wizard Reads
cnct\system\tt_check_configuration.cnc	Text file	Probing Macro. Checks to ensure Tool Touch is correctly configured before use	-CNC12 Reads
All .lth files in icn_lath	Text/Binary file	Program files used by CNC12 Lathe	-CNC12 Reads

Draft: January 11, 2024

Index

↓ DOWN ARROW, 39
← LEFT, 39
→ RIGHT, 39
↑ UP ARROW, 39

A, 11
Acorn, 342
Active Job Run Screen with G-code Display, 81
ALT, 11, 319, 323–325
ALT+-, **35**, 35
    Select Previous WCS, 35
ALT+0, 35
ALT+1, **35**, 35
    Select WCS, 35
ALT+2, 35
ALT+=, **35**, 35
    Select Next WCS, 35
ALT+D, 11, **33**
    WCS/Machine Coordinates, 33
ALT+E, **33**, 33
    Generate Screenshot, 33
ALT+F10, **35**, 35
    Exit CNC12, 35
ALT+F4, 318
ALT+I, **33**, 33, 35, 309
    Live PLC I/O, 33
ALT+J, 32, **33**, 33, 306
    Keyboard Jog Panel, 33
ALT+K, **34**, 34
    Bin, 34
ALT+L, **34**, 34
    Putback Location, 34
ALT+M, **34**, 34
    MDI, 34
ALT+P, **34**, 34
    Live PID Display, 34
ALT+S, **34**, 34, 81
    Cycle Start, 34
Alt+S, 190
ALT+T, **35**, 35
    Temperature Display, 35
ALT+V, **35**, 35
    Display CNC Software Version Info, 35
ALT-D, 9
ALT-J, 29
Arrow, 29
ATC Bin, **34**
ATC Putback Location, **34**
AUTO/BLOCK, 85
Auxiliary Function Keys, **20**
Axis Jog Buttons, **17**

Bit-mapped Parameters, 252
Boring, 218

C-axis Disable, 229
C-axis Enable, 229
Call Subprogram, 231
CAM File Format, 318
Cancel Programmed Action Timer, 233
Canceling a Job in Progress, 82
Canned Cycle Cancel, 215
Circular Interpolation, 194
Clamp Off, 227
Clamp On, 227
CNC Program Codes, **147**
    : – Visible Comment Identifier, 151
    ; – Internal Comment Identifier, 151
    [ ] – Numerical Expression, 151
    #, = – User or System Variable Reference, 153
    $ – ASCII Code Substitution, 152
    E – Feed Rate or Thread Lead, 147
    F – Feed Rate or Thread Lead, 147
    L – Subroutines, 147
    N – Block Number, 149
    O – Program Number, 149
    P – Parameter, 149
    Program Symbols, 147
    Q – Parameter, 149
    R – Radius, Return Point, Parameter, 149
    S – Spindle Speed Setting, 150
    T – Select Tool Number and Offset Number, 150
    U – Incremental X-axis Move Command, 150
    W – Incremental Z-axis Move Command, 150
Communications Stress Test, 85
Constant Lead Thread Cutting, 198
Constant Surface Speed Control, 222
Constant Surface Speed Control Cancel, 222
Control Configuration, 244
Conventions, **11**
Coolant Auto/Manual, **20**
Coolant Flood, **20**
Coolant Mist, **20**
Coolant Off, 227
CTRL, 323–325
CTRL+ALT+F, 33
CTRL+ALT+I, 33
CTRL+ALT+i, 323

Draft: January 11, 2024

CTRL+ALT+X, **36**, 36
Go to Shutdown Screen, 36
CTRL+D, **35**, 35
Swap DRO and Distance-to-Go DRO, 35
Ctrl+D, 10
CTRL+E, **35**, 35
Launch PLC Detective, 35
CTRL+H, **35**, 35
Enable G-code Display, 35
CTRL+I, **35**, 35
Save PLC state to file, 35
CTRL+Q, **36**, 36
Probing Cycles History, 36
CTRL+S, 318
Custom Smoothing Presets Menu, 332
Custom WMPG Macros, 339
Cutter Compensation, 199
CYCLE CANCEL, 17–19, 82, 216
Cycle Cancel, **19**
CYCLE START, 10–13, 17, 18, 34, 39, 45, 77, 78, 81, 83–86, 90, 190, 225, 226, 228, 233, 241, 243, 278, 306
Cycle Start, **18**, **34**, 80
cycle start, 10

D, 11
Deep Hole Drilling, 215
diameter mode, 9
Digitize Profile, 241
Disable Override Controls, 234
Display CNC Software Version Info, **35**
Display Formatted String for A Period of Time, 241
Distance-to-Go DRO, **10**
DOWN, 84, 119, 125
down, 106
DRO, 9, 10
DRO Display Units, 245
DRO Display Window, **9**
Dwell, 196

E-STOP, 18
elapsed time, 10
EMERGENCY STOP, 19, 83
Emergency Stop, **19**
Enable G-code Display, **35**
Enable Override Controls, 234
END, 91
End Face Peck Cutting Cycle, 210
ENTER, 11, 39, 75, 87, 91, 92, 118, 119, 125, 244, 245, 247, 252
Enter, 49, 50, 76, 101
Erase Log File, **36**
ESC, 11, 39, 88, 94, 95, 98, 100, 106, 109, 111–117, 119, 190, 244, 245, 247, 252
Esc, 32, 90, 99, 125
Quit, 90
Exit CNC12, **35**

f, 324, 325
F1, 11
ABS, 85
Abs/Inc, 113, 130
Absolute Power Feed, 85
Auto, 59, 80
Clear All, 63
Comment, **112**, 112
Contrl, 244
Copy Menu, 88
Edit Program, 316
Errors, 43
Feed (-1%), 81
File, 87, 90
File Ops, 43
G code/ICN, 92
ICN, 40
Jog, **247**, 247
Line, **93**, 93
New, 91
Next Axis, 317
Next Table, 54
Orient, 77, 78
Park, 45, 373
Part, 38, 49, 50, 54
Prev Soln, **118**
Reference Return Points, 52
Resume Job, 83
Setup, 37, 49, 50, 54, 56, 64, 66, 69, 73, 79, 165, 167, 169, 338, 340
setup, 168
Setup Menu, **37**
Toggle, 88, 94, 95, 113
Tool Change, 84
Trangle: Right, **120**
Type, 98–100, 104, 106, 109, 111
X Diam, 67, 69
X Diam., 58
X-diam/rad or Z-ref, 59
Z Ref, 80
Z Ref., 73
Z-ref., 58
F10, 11
Accept, 88, 91, 94, 95, 98–100, 104, 106, 109, 111–117, 125
Acorn Wizard, 43
Other, **112**, 112
Park, 45
Post, 90

Draft: January 11, 2024

RTG On/Off, 85
Save, 11, 55, 62, 63, 67, 69, 73, 80, 245, 247, 252, 338
Save & Exit, 316
Set, 48, 50, 51
Shut Down, 317, 373
Shut down, 37
Shutdown, **45**
F2, 11
Arc, **95**, 95
Feed (+1%), 81
INC, 85
Incremental Power Feed, 85
Load, 37, 86, 91, 92
Load Job Menu, **38**
Lock/Unlock Table, 55
M-&G-code, **112**, 112
Machine, 247
Manual, 41, 43
Measure Tool, 59, 67, 68, 70–74, 77, 78
Modify, 87
Motor, 247
Motor Parameters, **248**
Move Menu, 88
Next Soln, **118**
Origin, 53
Poweroff, 45
Precision Mill, 331
Restore Report, 42
Run Program, 316
Search, 83, 84, 176
Stats, 43
Teach, 53
Tool, 38, 129
Tool lib, 124
TOOL OFFSET, 79, 80
Tool Offset, 56, 62–64, 66, 69, 73
Triangle: Other, **120**
USB/LAN, 92
F3
+.001, 55
ATC (Automatic Tool Change), 60, 63
Auto, 276
Brake Off-On, 113
C-axis, **112**, 112
Clear All, **119**
Config, 38, 165, 167, 169, 244, 338, 340
config, 168
Contouring Mill, 331
Cut, 88
Details On/Off, 92
Drill, **98**, 98
Export, 43
Find Home, **249**, 249
Free, 85
Free XZ, 85
Insert, 88, 92
Machine Notes, 43
MDI, 37, **39**
Modal, 94
Modal Display, 95
Param, 167
Parameters, 338, 340
Params, 165, 168
Parms, 169
Range, **116**, 116
Ranges, 316
Redraw, 117
Remote, 246
Repeat On/Off, 81, 84
Save, 92
Set Range, 44
Tangent: Line Arc, **121**
Work Envel, 55, 196
Work Envelope, 55
F4
-.001, 55
Abs/Inc, 58, 60, 62, 63, 72, 73, 75, 76, 125
C Index, **113**, 113
Cut, 88
Feed, 38, 85
Paste, 88
PID, 315
Power, 85
Power XZ, 85
Precision Router, 331
Prev Solver, **119**
Probe screen, 36
Run, 37, 40, 83, 176
Run Menu, **40**
Run menu, 83
Save, 101
Save As, 92
Set Home, **250**, 250
Show Recent, 92
Show recent, 92
Skips On/Off, 81, 84
Tangent: Arc Arc, **121**
Tap, **99**, 99
Time, **117**, 117
Time Estimation, 44
Toggles & Pan, 316
F5
+.001, 125
Abs/Inc, 55
Auto, 81

Draft: January 11, 2024

Block Mode, 85
CAM, 11, 37, **40**, 40, 87
Color Picker, 42
Comp Man, 318
Contouring Router, 331
Copy, 88
Date/Alpha, 92
Delete, 92
Export Lib. . . , 61
Help, 89
Increment by a small amount, 61, 63
M Comp, **250**
Measure Offset X, 67, 68, 70–72, 77
Next Solver, **119**
Paste, 88
Redraw, 44, **117**
Tangent: Line Arc Arc, **122**
Thread, **100**, 100
Tune, 296
Zoom In, 316
F6
-.001, 125
ATC Reset, 289
Copy, 55, 88
Decrement by a small amount, 61, 63
Drag, 317
Edit, 37, **41**, 92, 308
Hide Math, **119**, 119
Math Help, 117
Measure Offset Z, 73, 74, 77, 78
Optional Stops, 85
Pan, 44, **117**, 117
Pic, 319
Prev WCS, 48, 50, 54
Profile, **104**, 104
Stops off, 81
System Prompt, 46
Tangent: Arc Arc Arc, **122**
Tools, 88
User Maint, 43
Zoom Out, 316
F7
ATC, 263
Clear, 82
Copy >>>, **119**
Copy Menus. . . , 88
Create Report, 43
Create report, 341
Details, 100
Feed Hold, 82
Help On/Off, 92
Increment by a small amount, 61, 63
Intersection: Line Line, **123**
Measure Offset Z and X, 78
Measure Z and X, 77
Next WCS, 48, 50, 54
Paste, 55
Previous Table, 252
Scales, **250**
Turning, **106**, 106
Utility, 15, 37, **42**, 341
Zoom All, 316
Zoom In, 44, **117**, 117
F8
Chamfer, **114**, 114
Change Axis, 316
Clear Cell or Column, 55
Copy <<<, **119**
Decrement by a small amount, 61, 63
G-code, 82
Graph, 37, **43**, 82, 85, 88, 92, 94, 116, 297
Groove, **109**, 109
Import License, 15
Intersection: Line Arc, **123**
Next Table, 252
Options, 43
Set, 50
Set X, 47, 48
Smoothing Setup, 330
Zoom Out, 44, **117**, 117
F9
Advanced, 92
Clear Clipbrd, 88
Customize Presets, 331, 332
Cutoff, **111**
Details On/Off, 92
Digitize, 36, 37
Exit CNC12, 46, 317
Graphic On/Off, **119**
Intersection: Arc Arc, **124**
Logs, 43, 285
Radius, **115**
Radius key, 115
Rapid On/Off, 82, 85
Save & Apply, 316
Setup, 88
Teach Mode, 90
Tool Wear, **45**, 61, 62
Trail On/Off, 82
WCS Table, 48, 52
Zoom All, 44, **117**, 117
Fast Synchronous I/O Update, 242
feed 5rate override, 10
FEED HOLD, 10, 17–20
Feed Hold, **18**, 18
feed hold, 10

Draft: January 11, 2024

Feed per Minute, 222
Feed per Revolution, 222
Feed Rate Override, **18**
feed rate override, 10
File Menu, **90**
Flood Coolant On, 227
Formatted String Commands, 238

G-code AD2 Smoothing, 328
G-codes, **14**, **192**
G00, 193
G01, 193
G02, 194
G03, 194
G04, 196
G10, 196
G20, 196
G21, 196
G22, 196
G23, 196
G28, 197
G29, 197
G30, 197
G32, 198
G37, 198
G40, 199
G41, 199
G42, 199
G50, 202
G52, 202
G53, 202
G54, 202
G55, 202
G56, 202
G57, 202
G58, 202
G59, 202
G65, 203
G70, 204
G71, 205
G72, 207
G73, 209
G74, 210
G75, 212
G76, 213
G80, 215
G83, 215
G84, 216
G85, 218
G90, 218
G92, 220
G96, 222
G97, 222
G98, 222
G99, 222
G0, 85, 147, 193, 194, 243, 339
G00, 14, 192, **193**, 202
Rapid Positioning, 193
G01, 14, 192, **193**, 204
Linear Interpolation, 193
G02, 14, 192, **194**
Circular Interpolation, 194
G03, 14, 192, **194**, 202
Helical Interpolation, 194
G04, 14, 192, **196**
Dwell, 196
G1, 147, 179, 193, 194, 204, 278, 332
G10, 14, 168, 169, 192, **196**, 196, 215, 252, 355
Parameter Setting, 196
G2, 147, 193–195, 273, 278, 289, 306, 332, 356
G20, 14, 192, **196**, 196, 245, 295
Select Inch Units, 196
G21, 14, 192, **196**, 196, 245, 295
Select Metric Units, 196
G22, 14, 55, 192, **196**, 196
Work Envelope On, 196
G225, 353
G23, 14, 192, **196**, 196
Work Envelope Off, 196
G28, 14, 18, 53, 89, 106, 108, 110, 147, 179, 192, **197**, 197, 198, 256, 267, 355–357, 383
Return to Reference Point, 197
G29, 14, 192, **197**, 197, 355, 356
Return from Reference Point, 197
G3, 147, 193–195, 273, 278, 289, 306, 332
G30, 14, 147, 179, 192, **197**, 197, 198, 267, 355, 356
Return to Secondary Reference Point, 197
G32, 14, 18, 147, 192, **198**, 198, 220, 294
Constant Lead Thread Cutting, 198
G37, 192, **198**, 198
PWM Velocity Modulation ON/OFF, 198
G4, 196, 205, 206, 208, 210, 263, 315, 344
G40, 14, 66, 76, 192, **199**, 199, 355
Cutter Compensation, 199
G41, 14, 57, 66, 76, 192, **199**, 199, 201, 281, 355
Cutter Compensation, 199
G42, 14, 57, 66, 76, 192, **199**, 199, 201, 281, 355
Cutter Compensation, 199
G50, 14, 58, 89, 126, 153, 192, **202**, 202, 222
Coordinate System Setting, 202
Maximum Spindle Speed for CSS mode, 202
G52, 14, 192, **202**, 202
Offset Local Coordinate System, 202
G53, 14, 147, 163, 175, 179, 192, **202**, 202, 355
Rapid Positioning in Machine Coordinates, 202
G54, 14, 54, 192, **202**, 202, 203

Draft: January 11, 2024

Select Work Coordinate System, 202
G55, 54, 192, **202**, 203
Select Work Coordinate System, 202
G56, 54, 192, **202**, 203
Select Work Coordinate System, 202
G57, 54, 192, **202**, 203
Select Work Coordinate System, 202
G58, 54, 192, **202**, 203
Select Work Coordinate System, 202
G59, 14, 54, 192, **202**, 202, 203
Select Work Coordinate System, 202
G61, 285
G64, 331, 332
G65, 14, 148, 149, 153, 163, 180, 186, 187, 192, **203**, 203, 204, 225, 353
Call Macro, 203
G68.1, 301
G70, 14, 89, 192, **204**, 204–210, 258, 295, 385, 390
Profile Finishing Cycle, 204
G71, 14, 89, 192, 204, **205**, 205–207, 254, 255, 258, 271, 273, 294, 295, 385, 387, 388, 390–392
Profile Stock Removal in Turning, 205
G72, 14, 89, 192, 204, 205, **207**, 207, 208, 254, 255, 258, 271, 273, 294, 295, 385, 387, 388, 390–392
Profile Stock Removal in Facing, 207
G73, 14, 192, 204, 205, **209**, 209, 210, 254, 272, 295, 356, 388, 392
Profile Path Repeating Cycle, 209
G74, 14, 89, 192, **210**, 210, 234, 254, 258, 271, 276, 294, 295, 385, 387, 390, 391
End Face Peck Cutting Cycle, 210
G75, 14, 89, 192, **212**, 212, 254, 258, 272, 276, 294, 295, 385, 387, 390, 392
Outside/Inside Diameter Peck Cutting Cycle, 212
G76, 14, 89, 103, 147, 192, **213**, 213, 214, 255, 256, 258, 272, 284, 294, 295, 383, 385, 388, 390, 392
Multi-pass Threading Cycle, 213
G80, 14, 192, **215**, 215
Canned Cycle Cancel, 215
G81, 225, 278
G82, 356
G83, 14, 192, **215**, 215, 279, 356
Deep Hole Drilling, 215
G84, 14, 18, 192, **216**, 216, 223, 228, 234, 276, 279
Tapping, 216
G85, 14, 192, **218**, 218, 234
Boring, 218
G89, 356
G90, 14, 148, 181, 192, **218**, 218, 219, 237
Outside/Inside Diameter Cutting Cycle, 218
G91, 148, 159, 181, 237, 356
G92, 14, 147, 192, **220**, 220, 272, 309, 356
Thread Cutting Cycle, 220
G93, 273
G94, 14, 192
G96, 14, 150, 153, 192, 202, **222**, 222
Constant Surface Speed Control, 222
G97, 14, 150, 192, 202, **222**, 222
Constant Surface Speed Control Cancel, 222
G98, 14, 147, 192, **222**, 222
Feed per Minute, 222
G99, 14, 147, 193, **222**, 222
Feed per Revolution, 222
Generate Screenshot, **33**
Go to Shutdown Screen, **36**
Graphing Color for Feed Rate Movement
Aqua, 242
Black, 242
Blue, 242
Fuchsia, 242
Gray, 242
Green, 242
Lime, 242
Maroon, 242
Navy Blue, 242
Olive, 242
Orange, 242
Purple, 242
Red, 242
Teal, 242
White, 242
Yellow, 242

Helical Interpolation, 194
HOME, 91
hot keys, 9

i, 319, 323
INC/CONT, 17
Inc/Cont, **17**
Insert Operation, **92**
Intercon, **87**
F1 – Line, 93
F10 – Other, 112
F2 – Arc, 95
F3 – Drill, 98
F4 – Tap, 99
F5 – Thread, 100
F6 – Profile, 104
F7 – Turning, 106
F8 – Groove, 109
F9 – Cutoff, 111
File Menu, 90
Graphics, 116
Insert Operation, 92

Draft: January 11, 2024

Introduction, 87
Load Menu, 91
Main Menu, 87
Math Help, 117
Teach Mode, 90
Tool Library, 124
Intercon Graphics
F3 – Range, 116
F4 – Time, 117
F5 – Redraw, 117
F6 – Pan, 117
F7 – Zoom In, 117
F8 – Zoom Out, 117
F9 – Zoom All, 117
Intercon Math Help
F1 – Prev Soln, 118
F1 – Trangle: Right, 120
F2 – Next Soln, 118
F2 – Triangle: Other, 120
F3 – Clear All, 119
F3 – Tangent: Line Arc, 121
F4 – Prev Solver, 119
F4 – Tangent: Arc Arc, 121
F5 – Next Solver, 119
F5 – Tangent: Line Arc Arc, 122
F6 – Hide Math, 119
F6 – Tangent: Arc Arc Arc, 122
F7 – Copy >>>, 119
F7 – Intersection: Line Line, 123
F8 – Copy <<<, 119
F8 – Intersection: Line Arc, 123
F9 – Graphic On/Off, 119
F9 – Intersection: Arc Arc, 124

jog panel, 10
Jog Panel Type, 246

Keyboard Jog Panel, **29**, **33**
Keyboard Shortcut Keys, **32**

Launch PLC Detective, **35**
LEFT, 119
Linear Interpolation, 193
Live PID Display, **34**
Live PLC I/O, **33**
Load Menu, **91**
load meters, 9

M-codes, **14**
M-Functions
M06, 226
M07, 227
M10, 227
M11, 227
M30, 228
M-functions
M00, 225
M01, 225
M02, 225
M03, 225
M04, 226
M05, 226
M08, 227
M09, 227
M100, 232
M1000, 242
M1001, 242
M1002, 242
M1003, 242
M1004, 242
M1005, 242
M1006, 242
M1007, 242
M1008, 242
M1009, 242
M101, 232
M1010, 242
M1011, 242
M1012, 242
M1013, 242
M1014, 242
M1015, 242
M102, 233
M103, 233
M104, 233
M105, 233
M106, 233
M107, 234
M108, 234
M109, 234
M115, 234
M116, 234
M120, 235
M121, 236
M122, 236
M123, 236
M124, 237
M125, 234
M126, 234
M127, 237
M128, 237
M129, 237
M130, 238
M150, 238
M151, 238
M17, 227
M19, 228

Draft: January 11, 2024

M200, 238, 240
M201, 240
M223, 238, 240
M224, 238, 240
M225, 238, 241
M26, 228
M29, 228
M290, 238, 241
M300, 242
M41, 228
M42, 228
M43, 228
M50, 229
M51, 229
M91, 229
M92, 230
M93, 230
M94, 231
M95, 231
M98, 231
M99, 232
M0, 18, 85, 175, 183, 233, 240, 253, 265, 310, 344
M00, 14, 166, 223, **225**
Stop for Operator, 225
M01, 14, 166, 223, **225**
Optional Stop for Operator, 225
M02, 14, 223, **225**
Restart Program, 225
M03, 14, 223, **225**
Spindle On Clockwise, 225
M04, 14, 223, **226**
Spindle On Counter-clockwise, 226
M05, 14, 223, **226**
Spindle Stop, 226
M06, 223, **226**
Tool Change, 226
M07, 14, 223, **227**
Mist Coolant On, 227
M08, 14, 223, **227**
Flood Coolant On, 227
M09, 14, 223, **227**
Coolant Off, 227
M1, 18, 85, 175, 225, 233, 344
M10, 14, 223, **227**, 227, 231
Clamp On, 227
M100, 14, 163–165, 173–175, 223, **232**, 232, 344
Wait for PLC Bit (Open, Off, Reset), 232
M1000, 14, 224, **242**, 243
Graphing Color for Feed Rate Movement, 242
M1001, **242**, 243
Graphing Color for Feed Rate Movement, 242
M1002, **242**, 243
Graphing Color for Feed Rate Movement, 242
M1003, **242**, 243
Graphing Color for Feed Rate Movement, 242
M1004, **242**, 243
Graphing Color for Feed Rate Movement, 242
M1005, **242**, 243
Graphing Color for Feed Rate Movement, 242
M1006, **242**, 243
Graphing Color for Feed Rate Movement, 242
M1007, **242**, 243
Graphing Color for Feed Rate Movement, 242
M1008, **242**, 243
Graphing Color for Feed Rate Movement, 242
M1009, **242**, 243
Graphing Color for Feed Rate Movement, 242
M101, 14, 163–165, 173–175, 223, **232**, 232, 344
Wait for PLC Bit (Closed, On, Set), 232
M1010, **242**, 243
Graphing Color for Feed Rate Movement, 242
M1011, **242**, 243
Graphing Color for Feed Rate Movement, 242
M1012, **242**, 243
Graphing Color for Feed Rate Movement, 242
M1013, **242**, 243
Graphing Color for Feed Rate Movement, 242
M1014, **242**, 243
Graphing Color for Feed Rate Movement, 242
M1015, 14, 224, **242**, 243
Graphing Color for Feed Rate Movement, 242
M102, 14, 223, **233**, 233
Restart Program, 233
M103, 14, 223, **233**, 233, 346
Programmed Action Timer, 233
M104, 14, 223, **233**, 233, 346
Cancel Programmed Action Timer, 233
M105, 14, 165, 182, 223, **233**, 233, 253, 265
Move Minus to Switch, 233
M106, 14, 165, 182, 223, **233**, 233, 253, 265
Move Plus to Switch, 233
M107, 14, 223, **234**, 234, 288, 289
Output Tool Number, 234
M108, 14, 223, **234**, 234
Enable Override Controls, 234
M109, 14, 223, **234**, 234
Disable Override Controls, 234
M11, 14, 223, **227**, 227, 231, 291
Clamp Off, 227
M115, 14, 148, 156, 181, 182, 223, **234**, 235, 266
Protected Move Probing Functions, 234
M116, 14, 148, 156, 181, 182, 223, **234**, 235, 266
Protected Move Probing Functions, 234
M120, 14, 223, **235**, 235–237, 240, 241
Open Data File (Overwrite Existing File), 235
M121, 14, 223, 235, **236**, 236, 237, 240, 241

Draft: January 11, 2024

Open Data File, 236
M122, 14, 148, 223, **236**, 236, 237, 276
Record Local Position(s) and Optional Comment in Data File, 236
M123, 14, 148, 224, **236**, 236, 237, 276
Record Value and/or Comment in Data File, 236
M124, 14, 224, **237**, 237, 276
Record Machine Position(s) and Optional Comment in Data File, 237
M125, 14, 156, 181, 182, 223, **234**, 235, 266
Protected Move Probing Functions, 234
M126, 14, 156, 181, 182, 223, **234**, 235, 266
Protected Move Probing Functions, 234
M127, 14, 224, **237**, 237
Record Date and Time in a Data File, 237
M128, 14, 148, 224, **237**, 237, 354, 355
Move Axis by Encoder Counts, 237
M129, 14, 224, **237**
Record Current Job File Path to Data File, 237
M130, 14, 148, 224, **238**, 238
Run System Command, 238
M15, 170
M150, 14, 224, **238**, 238
Set Spindle Encoder to Zero at Next Index Pulse, 238
M151, 14, 224, **238**, 238
Unwind C-axis, 238
M16, 170
M17, 223, **227**, 227
Prepare for Tool Change, 227
M19, 174, 223, **228**, 228
Spindle Orient, 228
M2, 18, 159, 224, 225, 228, 263
M200, 14, 163, 224, **238**, **240**, 240, 310, 354
Formatted String Commands, 238
Stop for Operator, Prompt for Action, 240
M201, 14, 224, **240**, 240, 310
Stop for Operator, Prompt for Action, 240
M223, 14, 163, 224, **238**, **240**, 240, 354
Formatted String Commands, 238
Write Formatted String to File, 240
M224, 14, 163, 172, 224, **238**, **240**, 240, 353, 354
Formatted String Commands, 238
Prompt for Operator Input Using Formatted String, 240
M225, 14, 160, 162, 163, 166, 224, **238**, **241**, 241, 342, 353, 354
Display Formatted String for A Period of Time, 241
Formatted String Commands, 238
M25, 224, 357
M26, 14, 159, 223, **228**, 228, 355
Set Axis Home, 228
M29, 14, 216, 223, **228**, 228
Set Tap Mode for G84, 228
M290, 14, 163, 224, **238**, **241**, 241
Digitize Profile, 241
Formatted String Commands, 238
M3, 216, 225, 231, 269, 270, 278, 311
M30, 14, 223, **228**, 228, 263
Restart Program, 228
M300, 14, 224, **242**
Fast Synchronous I/O Update, 242
M39, 231
M4, 216, 226, 231, 269, 270, 278, 311
M41, 223, **228**, 228
Select Spindle Gear Range, 228
M42, 223, **228**, 228
Select Spindle Gear Range, 228
M43, 223, **228**, 228
Select Spindle Gear Range, 228
M5, 226, 231, 270
M50, 14, 223, **229**, 229, 238
C-axis Disable, 229
M51, 14, 223, **229**, 229, 238
C Axis Enable, 229
M55, 27, 158–160
M56, 158
M57, 158
M58, 158
M6, 18, 165, 174, 183, 185, 186, 224, 226, 227, 231, 234, 253, 288, 289
M61, 171, 341
M62, 171
M63, 171, 175
M68, 341
M7, 20, 227, 231, 265
M72, 341
M74, 167
M8, 20, 227, 231, 265
M81, 171, 341
M88, 341
M9, 227, 231
M91, 14, 147, 159, 182, 223, **229**, 229, 290
Move to Minus Home, 229
M92, 14, 147, 182, 223, **230**, 230, 290
Move to Plus Home, 230
M93, 14, 223, **230**, 230, 304
Release/Restore Motor Power, 230
M94, 14, 164, 171–173, 188, 223, 224, 229, **231**, 231, 238, 242, 341
Output On, 231
M95, 14, 164, 171–173, 188, 223, 224, 229, **231**, 231, 238, 242, 341
Output Off, 231
M98, 14, 148, 149, 163, 186, 187, 223, **231**, 231, 232
Call Subprogram, 231

Draft: January 11, 2024

M99, 14, 149, 163, 203, 223, 231, **232**, 232
Return from Macro or Subprogram, 232
Machine Configuration, 247
Machine Home, **12**
Machine Home at Powerup, 246
machine position, 9
Machine Units of Measure, 245
Main Screen, **37**
Maximum Spindle Speed (High Range), 245
Maximum Spindle Speed for CSS mode, 202
MDI, **32**
Message Window, **10**
Minimum Spindle Speed (High Range), 246
Mist Coolant On, 227
Modifying the CAM Menu, 317
Move Axis by Encoder Counts, 237
Move Minus to Switch, 233
Move Plus to Switch, 233
Move to Minus Home, 229
Move to Plus Home, 230
MPG, **17**, 17
Multi-pass Threading Cycle, 213

N, 91

Offset Local Coordinate System, 202
Open Data File, 236
Open Data File (Overwrite Existing File), 235
Operator Panel, **16**
Optional Stop for Operator, 225
Options Window, **11**
Output On/Off, 231
Output Tool Number, 234
Outside/Inside Diameter Cutting Cycle, 218
Outside/Inside Diameter Peck Cutting Cycle, 212

PAGE DOWN, 91
Page Down, 29, 101
Tweak −, 316
PAGE UP, 91
Page Up, 29, 101
Tweak +, 316
Parameter 0 – E-Stop PLC Bit, 253
Parameter 1 – Machine and Jog key orientation, 11, 17, 57, 125, 253, 263
Parameter 10 – Macro M-function Control/Probe Stop Handling, 183, 225, 253, 254, 310
Parameter 100 – Intercon comment generation, 256, 282
Parameter 101 – Intercon clearance amount, 256
Parameter 102 – Intercon spindle coolant delay, 256
Parameter 103 – Intercon corner feed rate override, 256
Parameter 104 – Intercon modal line parameters, 256
Parameter 105 – Intercon modal arc parameters, 256
Parameter 106 – Intercon modal drilling cycle parameters, 256
Parameter 107 – Intercon chamfer blend, 256
Parameter 108 – Intercon polar display, 256
Parameter 109 – Intercon modal display, 256, 282
Parameter 11 – DP4 PLC Input Number and Contact State, 184, 254
Parameter 110 – Wizard Password, 256
Parameter 111 – Intercon no spindle stop during tool change, 256, **282**
Parameter 112 – Intercon no coolant stop during tool change, 256, **282**
Parameter 113 – Hide Intercon Menu, 256
Parameter 114 – Intercon use G28 during tool change, 256, **282**
Parameter 115 – Intercon Help, 256, **282**
Parameter 116 – A-axis Y-coordinate, 256
Parameter 117 – A-axis Z-coordinate, 256
Parameter 118 – B-axis X-coordinate, 256
Parameter 119 – B-axis Z-coordinate, 256
Parameter 12 – Touch Probe Tool Number, 185, 254, 276
Parameter 120 – Probe Stuck Clearance Amount, 256
Parameter 121 – Grid digitize prediction minimum Z-pullback, 256
Parameter 122 – Grid digitizing deadband move distance, 256
Parameter 128 – Dry Run PLC Bit, 256
Parameter 129 – Dry Run Feed Rate, 256, 283
Parameter 13 – Recovery Distance, 235, 254
Parameter 130 – 3rd-axis on/off selection, 256, **283**, 283, 284
Parameter 131 – 4th-axis on/off selection, 256, **283**, 283, 284
Parameter 132 – Motor Heating Coefficients for axes 5, 256, 268, **284**
Parameter 133 – Motor Heating Coefficients for axes 6, 256, **284**
Parameter 134 – Motor Heating Coefficients for axes 7, 256, **284**
Parameter 135 – Motor Heating Coefficients for axes 8, 256, 268, **284**
Parameter 136 – G76 Fine Bore Retract Angle, 256
Parameter 137 – Load Meter Filter Size, 256
Parameter 138 – DRO Encoder Deadband, 256
Parameter 139 – Special Dwell between Moves, 256
Parameter 14 – Fast Probing Rate, 254
Parameter 140 – Message log priority level, 257, 357
Parameter 141 – Maximum message log lines, 257, 285
Parameter 142 – Message log trim amount, 257, 285
Parameter 143 – DRO properties (load meter, DTG), 9,

Draft: January 11, 2024

10, 257, 273, 297
Parameter 144 – Comparison rounding, 151, 257, 286
Parameter 145 – Advanced macro properties (fast branching), 189, 257
Parameter 146 – Feed hold threshold for feed rate override, 257
Parameter 147 – Number of Messages in Operator Message Window, 257
Parameter 148 – Miscellaneous Jogging Options, 257, 290
Parameter 149 – Spindle Speed Threshold, 257, 278
Parameter 15 – Slow Probing Rate, 254, 266
Parameter 150 – Backplot Graphics display options, 257
Parameter 151 – Repeatability tolerance for probing and radial digitizing, 257, 287
Parameter 153 – Probe Protection Enabled, 257
Parameter 154 – Touch Screen Options, 257
Parameter 155 – Probe Type, 257, 305
Parameter 156 – Autotune Move Distance / Auto Delay Calculation Move Distance for axes 5, 257, 281, **288**, 317
Parameter 157 – Autotune Move Distance / Auto Delay Calculation Move Distance for axes 6, 257, **288**
Parameter 158 – Autotune Move Distance / Auto Delay Calculation Move Distance for axes 7, 257, **288**
Parameter 159 – Autotune Move Distance / Auto Delay Calculation Move Distance for axes 8, 257, 281, **288**, 317
Parameter 16 – Maximum Probing Distance, 235, 254
Parameter 160 – Enhanced ATC, 185, 186, 257
Parameter 162, 227
Parameter 163 – Gang Tooling, 57, 125, 257
Parameter 164 – ATC Feature, 257
Parameter 165 – Acceleration/Deceleration Options, 257
Parameter 166 – Axis Properties for axes 5, 257, 280, **289**, 301
Parameter 167 – Axis Properties for axes 6, 257, **289**
Parameter 168 – Axis Properties for axes 7, 257, **289**
Parameter 169 – Axis Properties for axes 8, 257, 280, **289**
Parameter 17 – Detector Location Return Point, 254
Parameter 170 – Enable Keyboard Jogging and Set Feed Rate Override Control, 29, 257, 287
Parameter 171, **289**
Parameter 172, **289**
Parameter 173, **289**
Parameter 174, **289**
Parameter 175, **289**
Parameter 176, **289**
Parameter 177, **289**, 289
Parameter 178 – PLC I/O configuration, 227, 257, 289
Parameter 179 – Lube Pump Operation, 257, 289
Parameter 18 – PLC Input Spindle Inhibit Parameter, 78, 254, 267, 276, 296
Parameter 180 – Clear Home Switch Distance, 257
Parameter 186 – Probe Stuck retry disable, 257
Parameter 187 – Hard Stop Homing, 257
Parameter 188, 257, **291**, 340, 341
Parameter 189, **291**
Parameter 19 – MPG modes, 254
Parameter 190, **291**
Parameter 191, **291**, 291
Parameter 192, **291**
Parameter 193, **291**
Parameter 194, **291**
Parameter 195, **291**
Parameter 196, **291**
Parameter 197, **291**
Parameter 198, **291**
Parameter 199, 257, **291**, 340, 341
Parameter 2 – G-code Interpretation Control and Slaving Rotary axis feed rate, 253
Parameter 20 – Ambient Temperature, 254, **268**, 268, 284, 293
Parameter 200, 257, **291**
Parameter 201, **291**
Parameter 202, **291**
Parameter 203, **291**
Parameter 204, **291**
Parameter 205, **291**
Parameter 206, **291**
Parameter 207, 257, **291**
Parameter 208, 257, **292**, 292
Parameter 209, **292**
Parameter 21 – Motor Heating Coefficients for axes 1, 254, **268**, 268
Parameter 210, **292**
Parameter 211, **292**
Parameter 212, **292**
Parameter 213, **292**
Parameter 214, **292**
Parameter 215, 257, **292**, 292
Parameter 216 – PC-based Lash Compensation on/off, 257, 327
Parameter 217 – PC-based Screw Compensation on/off, 257
Parameter 218 – USB MPG Options, 257
Parameter 219 – Virtual Control Panel Options, 257
Parameter 22 – Motor Heating Coefficients for axes 2, 254, **268**
Parameter 220, 257, 263, 279, **293**, 293, 327, 330
Parameter 221, **293**, 327, 330, 332

Draft: January 11, 2024

Parameter 222, **293**, 327
Parameter 223, **293**, 327
Parameter 224, **293**, 327
Parameter 225, **293**
Parameter 226, **293**, 327
Parameter 227, **293**, 327
Parameter 228, **293**, 327
Parameter 229, **293**, 327, 332
Parameter 23 – Motor Heating Coefficients for axes 3, 254, **268**
Parameter 230, **293**, 327
Parameter 231, 257, **293**, 327, 330, 332
Parameter 236 – Motor Cooling Coefficients for axes 5, 257, 268, **293**
Parameter 237 – Motor Cooling Coefficients for axes 6, 257, **293**
Parameter 238 – Motor Cooling Coefficients for axes 7, 257, **293**
Parameter 239 – Motor Cooling Coefficients for axes 8, 257, 258, 268, **293**
Parameter 24 – Motor Heating Coefficients for axes 4, 254, **268**, 268
Parameter 240 – Rigid Tapping and Threading Accel Rate Distance, 258, 293
Parameter 241 – Rigid Tapping and Threading Rotational Step Size (Degrees), 258, 293
Parameter 242 – Minimum Angle Threshold for application of Accel/decel in Threading moves, 258, 293
Parameter 243 – Threading Control, 258
Parameter 244 – Tool Touch-off device PLC input number, 78, 258
Parameter 245 – G71, G72, G74, G75, G76 D/P/Q “implied float” re-interpretation threshold for Inch units, 206, 208, 211, 212, 214, 215, 258, 294, 295
Parameter 246 – G71, G72, G74, G75, G76 D/P/Q “implied float” re-interpretation threshold for MM units, 206, 208, 211, 212, 214, 215, 258, 294, 295
Parameter 247 – G70 Multiple Pass Behavior Suppression, 205, 258
Parameter 248 – Tool Wear Adjustment magnitude limit, 258
Parameter 25 – Motor Cooling Coefficients for axes 1, 254, **268**, 268
Parameter 252 – Autotune Ka Performance parameters for axes 5, 258, 279, **296**
Parameter 253 – Autotune Ka Performance parameters for axes 6, 258, **296**
Parameter 254 – Autotune Ka Performance parameters for axes 7, 258, **296**
Parameter 255 – Autotune Ka Performance parameters for axes 8, 258, 279, **296**
Parameter 256 – Drive Mode, 258, 316
Parameter 257 – TT1 connection detection PLC input, 77, 78, 258
Parameter 258 – Velocity/Torque Mode override in Precision mode, 258
Parameter 259 – Manual Axis Designation, 258
Parameter 26 – Motor Cooling Coefficients for axes 2, 254, **268**
Parameter 260 – Accelerated Graphics Backplot, 258
Parameter 261 – Precision Mode Standoff Tolerance Percentage, 258
Parameter 262 – Precision Mode Standoff Delay Time, 258
Parameter 263 – DRO Display Precision, 258, 285
Parameter 264 – Rapid Override, 258
Parameter 27 – Motor Cooling Coefficients for axes 3, 254, **268**
Parameter 270 – Skew Correction: X Skew offset from Trusted/Intended X, 258, **297**, 297, 298
Parameter 271 – Skew Correction: Trusted/Intended Y, 258, **297**, 297, 298
Parameter 278 – Spindle Speed Display Precision, 258
Parameter 28 – Motor Cooling Coefficients for axes 4, 254, **268**, 268
Parameter 281 – Tool Touch-off Device X stylus size, 77, 78, 258
Parameter 282 – Tool Touch-off Device Z stylus size, 78, 258
Parameter 283 – Auto Tool Touch-off safety clearance, 77, 78, 258
Parameter 284 – Brake Resistor Wattage for ACDC Drives 1, 258, **299**
Parameter 285 – Brake Resistor Wattage for ACDC Drives 2, 258, **299**
Parameter 286 – Brake Resistor Wattage for ACDC Drives 3, 258, **299**
Parameter 287 – Brake Resistor Wattage for ACDC Drives 4, 258, **299**
Parameter 288 – Brake Resistor Wattage for ACDC Drives 5, 258, **299**
Parameter 289 – Brake Resistor Wattage for ACDC Drives 6, 258, **299**
Parameter 29 – Overheating Warning Temperature, 254, **268**, 268, 349
Parameter 290 – Brake Resistor Wattage for ACDC Drives 7, 258, **299**
Parameter 291 – Brake Resistor Wattage for ACDC Drives 8, 258, **299**
Parameter 292 – Aux Key Function 13, 258, **299**
Parameter 293 – Aux Key Function 14, 258, **299**
Parameter 294 – Aux Key Function 15, 258, **299**
Parameter 295 – Aux Key Function 16, 258, **299**

Draft: January 11, 2024

Parameter 299 – Report Option Bits, 258
Parameter 3 – Modal Tool and Height Offset Control, 253
Parameter 30 – Job Cancellation Temperature, 254, **268**, 268, 284, 293, 349
Parameter 300 – Drive assignment to Axes 1, 258, 259, **299**, 299, 358
Parameter 301 – Drive assignment to Axes 2, 259, **299**, 299
Parameter 302 – Drive assignment to Axes 3, 259, **299**
Parameter 303 – Drive assignment to Axes 4, 259, **299**
Parameter 304 – Drive assignment to Axes 5, 259, **299**
Parameter 305 – Drive assignment to Axes 6, 259, **299**
Parameter 306 – Drive assignment to Axes 7, 259, **299**
Parameter 307 – Drive assignment to Axes 8, 258, 259, **299**, 358
Parameter 308 – Encoder assignment to Axes 1, 250, 259, 269, **300**, 300, 301, 303, 311
Parameter 309 – Encoder assignment to Axes 2, 259, **300**, 300, 301
Parameter 31 – Legacy SPIN232 Com Port, 254
Parameter 310 – Encoder assignment to Axes 3, 259, **300**, 301
Parameter 311 – Encoder assignment to Axes 4, 259, **300**
Parameter 312 – Encoder assignment to Axes 5, 259, 269, **300**, 311
Parameter 313 – Encoder assignment to Axes 6, 259, **300**
Parameter 314 – Encoder assignment to Axes 7, 259, **300**
Parameter 315 – Encoder assignment to Axes 8, 250, 259, 269, **300**, 301, 311
Parameter 316 – Absolute Encoder Bits, 259, 301
Parameter 317 – Single-turn Absolute Encoder Bits, 259
Parameter 318 – Five-axis Configuration, 259
Parameter 319 – Five-axis Options, 259
Parameter 32 – Autonomous Digitizing Offset Files, 254
Parameter 321 – MPU13 DSP Probe Input, 259
Parameter 322 – Spindle Encoder Port, 259
Parameter 323 – MPU11 Encoder Speed Filter, 259
Parameter 324, 259, **302**
Parameter 325, **302**
Parameter 326, **302**
Parameter 327, **302**
Parameter 328, **302**
Parameter 329, **302**
Parameter 33 – Spindle Motor Gear Ratio, 254, 269, 270
Parameter 330, **302**
Parameter 331, 259, **302**
Parameter 332 – Suppress encoder differential faults, 259, **302**, 302
Parameter 333 – Suppress encoder differential error messages, 259, **302**, 302
Parameter 334 – Suppress encoder quadrature faults, 259, **302**, 302
Parameter 335 – Suppress encoder quadrature error messages, 259, **302**, 302
Parameter 336 – Overall gain setting, 259, **302**, 302
Parameter 337 – Absolute error gain, 259, **302**, 302
Parameter 338 – Error sum gain, 259, **302**, 302
Parameter 339 – Delta error gain, 259, **302**, 302
Parameter 34 – Spindle Encoder Counts/Rev, 217, 254, 278, 350
Parameter 340 – Precision Mode delay (in milliseconds) for axes 1, 259, 296, **303**, 317
Parameter 341 – Precision Mode delay (in milliseconds) for axes 2, 259, **303**
Parameter 342 – Precision Mode delay (in milliseconds) for axes 3, 259, **303**
Parameter 343 – Precision Mode delay (in milliseconds) for axes 4, 259, **303**
Parameter 344 – Precision Mode delay (in milliseconds) for axes 5, 259, **303**
Parameter 345 – Precision Mode delay (in milliseconds) for axes 6, 259, **303**
Parameter 346 – Precision Mode delay (in milliseconds) for axes 7, 259, **303**
Parameter 347 – Precision Mode delay (in milliseconds) for axes 8, 259, 296, **303**, 317
Parameter 348 – MPG/Handwheel Encoder Input 1, 259
Parameter 349 – MPG/Handwheel Detents-per-revolution 1, 259
Parameter 35 – Spindle Encoder Axis Number, 217, 254, 269, 273, 278, 352
Parameter 350 – MPG/Handwheel Encoder Counts-per-revolution 1, 259
Parameter 351 – MPG/Handwheel Encoder Input 2, 259
Parameter 352 – MPG/Handwheel Detents-per-revolution 2, 259
Parameter 353 – MPG/Handwheel Encoder Counts-per-revolution 2, 260
Parameter 354 – MPG/Handwheel Encoder Input 3, 260
Parameter 355 – MPG/Handwheel Detents-per-revolution 3, 260
Parameter 356 – MPG/Handwheel Encoder Counts-per-revolution 3, 260
Parameter 357 – Axis Drive Max RPM for Axes 1, 260, **304**
Parameter 358 – Axis Drive Max RPM for Axes 2, 260,

Draft: January 11, 2024

304
Parameter 359 – Axis Drive Max RPM for Axes 3, 260, **304**
Parameter 36 – Rigid Tapping Enable/Disable, 217, 218, 254, 270
Parameter 360 – Axis Drive Max RPM for Axes 4, 260, **304**
Parameter 361 – Axis Drive Max RPM for Axes 5, 260, **304**
Parameter 362 – Axis Drive Max RPM for Axes 6, 260, **304**
Parameter 363 – Axis Drive Max RPM for Axes 7, 260, **304**
Parameter 364 – Axis Drive Max RPM for Axes 8, 260, **304**
Parameter 365 – Drive power-on delay, 260
Parameter 366 – Probe deceleration multiplier, 260, 265, **304**, 304
Parameter 367 – TT1 deceleration multiplier, 78, 260, 265, **304**, 304
Parameter 368 – Autonomous Digitizing Angle Adjustment, 260
Parameter 369 – Tool Check Max Absolute Angle, 260
Parameter 37 – Spindle Deceleration Time, 217, 218, 254
Parameter 374 – ACDC Drive Debug Log Axis Inclusion, 260, **305**
Parameter 375 – ACDC Drive Debug Log Size, 260, **305**
Parameter 376 – ACDC Drive Debug Log Collection Type, 260, **305**
Parameter 377 – ACDC Drive Debug Log Trigger Type, 260, **305**
Parameter 378 – ACDC Drive Debug Log Trigger Level, 260, **305**
Parameter 379 – ACDC Drive Debug Log Option, 260, **305**
Parameter 38 – Multi-axis Max Feed Rate, 248, 254
Parameter 387 – Debug Trace Mask, 260, **305**
Parameter 388 – Debug Options, 260, **305**
Parameter 389 – Debug Level, 33, 260, **305**
Parameter 39 – Feed Rate Override Percentage Limit, 254
Parameter 392 – DP-7 Pullback Distance, 260, **305**, 305
Parameter 393 – DP-7 Pullback Feed Rate, 260, **305**, 305
Parameter 394 – DP-7 Measuring Feed Rate, 260, **305**, 305
Parameter 395 – Probing Setup Traverse Speed, 260
Parameter 396 – Probing Setup Plunge Speed, 260
Parameter 397 – Combustion Chamber Clearance Height, 260
Parameter 398 – Port/block mode, 260
Parameter 399 – AD1 arc chord tolerance adjustment, 260
Parameter 4 – Remote File Loading Flag & Advanced File Ops, 254
Parameter 40 – Basic Jog Increment, 254
Parameter 400 – Run Menu Cycle Start Enabled, 85, 260
Parameter 401 – Forget Last Job Loaded, 260
Parameter 403 – Disable Keyboard Jogging Legend, 260
Parameter 404 – Unused, 260
Parameter 405 – Tool Touch-off Type, 260
Parameter 406 – Probe State When Tripped, 260
Parameter 408 – Active G-Codes, 260, 306
Parameter 409 – Probe Type, 260
Parameter 41 – Rotary Axis Jog Increment, 254
Parameter 410 – Probe/Tool Touch-off Warning, 261
Parameter 411 – MPG Type, 261
Parameter 413 – Park macro, 261
Parameter 414 – Home File Type, 261
Parameter 415 – Ether1616 Configuration Bits, 261
Parameter 416 – Spindle Inhibit, 261
Parameter 417 – Preview G-code Before Job, 261
Parameter 418 – Assign Enter Key to Cycle Start, 261
Parameter 419 – PLC Worklight and Popup Pin Output, 261
Parameter 42 – Password for Configuration Menus, 244, 254, 282
Parameter 420 – Analog/PWM 1 Settings, 261
Parameter 421 – Clean Fan Filter Reminder, 261
Parameter 422 – PLC Diagnostics Display, 261, 326
Parameter 423 – Display Scale Position on DRO, 251, 261
Parameter 425 – Force Rehoming After EStop Condition, 261
Parameter 426 – WCS Lockout, 261
Parameter 43 – G71/G72 Depth of Cut, 206, 208, 254
Parameter 430 – RTG Spindle Speed RPM Display, 261
Parameter 44 – G71/G72 Escape Amount, 206, 208, 254
Parameter 440 – Stopped for Jogging Continue Bit, 261
Parameter 441 – MPG1 Axis Selection, 261, **310**
Parameter 442 – MPG2 Axis Selection, 261, **310**, 310
Parameter 443 – MPG3 Axis Selection, 261, **310**
Parameter 45 – G74 X-axis Relief Amount, 55, 254
Parameter 450 – Inhibit MPG Z-axis Fast Movement, 261
Parameter 458 – C-Axis Options, 261
Parameter 459 – Second Spindle Encoder Axis Number, 261, 278, 311
Parameter 46 – G75 Z-axis Relief Amount, 254

Draft: January 11, 2024

Parameter 460 – Second Spindle Maximum Speed, 261, 278
Parameter 461 – Second Spindle Minimum Speed, 261, 278
Parameter 462 – Second Spindle Encoder Counts/Revolution, 261, 278
Parameter 47 – G73 Repeat Count, 254
Parameter 48 – Grid Digitize Patch Playback Z rapid clearance amount, 255
Parameter 49 – Thread Chamfer Amount, 214, 221, 255
Parameter 5 – Machine Home/Startup Setup, 254, 264
Parameter 50 – G76 Finish Count, 214, 255
Parameter 507 – 5th-axis Pairing – Slave Axis, 261
Parameter 508 – 5th-axis Pairing – Master Axis, 261
Parameter 51 – G76 Thread Angle, 102, 214, 255
Parameter 52 – G76 Minimum Cutting Depth, 214, 255
Parameter 53 – G76 Finish Allowance, 214, 255
Parameter 55 – Radius / Diameter mode, 9, 51, 52, 64, 206, 208, 209, 211, 212, 214, 219, 220, 255
Parameter 56 – Feedrate Override Display Properties / Inverse Time Interpolation, 255
Parameter 560 – Laser X-offset, 261
Parameter 561 – Laser Y-offset, 261
Parameter 57 – Use Generic Load Meter Data from PLC, 255
Parameter 58 – G71/G72 Profile Rough Cut Escape Angle, 206, 208, 255
Parameter 59 – Tool Check Options, 255
Parameter 6 – Auto Tool Changer Installed, 185, 186, 226, 254
Parameter 600 – A-axis Expected Velocity Switch Point, 261
Parameter 601 – A-axis Scale Switch To Velocity, 261, 312
Parameter 602 – A-scale Correction Meter Max Deflection Count, 261
Parameter 603 – B-axis Expected Velocity Switch Point, 261
Parameter 604 – B-axis Scale Switch To Velocity, 261, 312
Parameter 605 – B-scale Correction Meter Max Deflection Count, 261
Parameter 61 – High Power Stall Timeout, 255, **274**, 274, 348
Parameter 62 – High Power Stall PID Limit, 255, **274**, 274
Parameter 63 – High Power Idle PID Multiplier, 255
Parameter 64 – Fourth/Fifth-axis Pairing, 255, 274, 311
Parameter 65 – Low Range Gear Ratio, 255, 269, 270, **275**, 275
Parameter 66 – Medium-low Range Gear Ratio, 255, **275**, 275
Parameter 67 – Medium-high Range Gear Ratio, 255, 269, 270, **275**, 275
Parameter 68 – Minimum Rigid Tapping Spindle Speed, 217, 218, 255, 275, 279
Parameter 69 – Duration For Minimum Spindle Speed, 217, 218, 255
Parameter 7 – Display Colors, 254
Parameter 70 – Offset Library Inc/Decrement Amount, 255
Parameter 700, 261, **313**, 313
Parameter 702, 176, **313**
Parameter 71 – Part Setup Detector Height, 178, 255, 276
Parameter 710, 168, 169, **313**
Parameter 72 – Data M-function Options, 236, 255
Parameter 73 – Peck Drill Retract Amount, 211, 212, 255
Parameter 74 – M-function executed at bottom of tapping cycle, 217, 255
Parameter 75 – Summing Control, 255
Parameter 76 – Manual Input Unrestricted Distance, 255
Parameter 77 – Manual Input Movement Tolerance, 255, 278
Parameter 78 – Spindle Speed Display and Operations, 156, 255, 287
Parameter 79 – Paired Axes Re-sync Delay Time, 255
Parameter 799, 261, **313**, 313
Parameter 8 – Available Coolant Systems, 254
Parameter 80 – Voltage Brake Message Frequency, 255
Parameter 800 – Acorn DB25 Map Enable, 261, **313**, 313
Parameter 801 – Acorn Step 1, 261, **313**, 313
Parameter 802 – Acorn Step 2, 261, **313**, 313
Parameter 803 – Acorn Step 3, 261, **313**, 313
Parameter 804 – Acorn Step 4, 261, **313**, 313
Parameter 805 – Acorn Direction 1, 261, **313**, 313
Parameter 806 – Acorn Direction 2, 261, **313**, 313
Parameter 807 – Acorn Direction 3, 261, **313**, 313
Parameter 808 – Acorn Direction 4, 261, **313**, 313
Parameter 809 – Acorn PLC Output 1, 261, **313**, 313
Parameter 81 – Air Drill M-function, 255
Parameter 810 – Acorn PLC Output 2, 262, **313**, 313
Parameter 811 – Acorn PLC Output 3, 262, **313**, 313
Parameter 812 – Acorn PLC Output 4, 262, **313**, 313
Parameter 813 – Acorn DB25 Header Section, 261, 262, **313**, 313
Parameter 814 – Analog/PWM 2 Settings, 262
Parameter 815 – Inverse Output, 262
Parameter 817 – PWM, 262
Parameter 82 – Spindle Drift Adjustment, 217, 218, 255
Parameter 83 – Deep Hole Clearance Amount, 215,

Draft: January 11, 2024

255, 279
Parameter 830 – ATC Type, 262
Parameter 84 – M-function executed at return to initial point of tapping cycle, 217, 255
Parameter 85 – “Door Open” Interlock PLC bit, 255
Parameter 852 – Carousel Skip Count on Reverse, 262
Parameter 853 – Distance to travel past tool position, 262
Parameter 854 – Distance to travel behind tool position, 262
Parameter 855 – MPG Performance, 262
Parameter 856 – VCP Jogging State on Acorn Power Up, 262
Parameter 86 – Rapid/Linear vector rate limit, 255
Parameter 87 – Autotune Ka Performance parameters for axes 1, 255, **279**, 279, 296
Parameter 88 – Autotune Ka Performance parameters for axes 2, 255, **279**
Parameter 89 – Autotune Ka Performance parameters for axes 3, 255, **279**
Parameter 9 – Display Language, 254
Parameter 90 – Autotune Ka Performance parameters for axes 4, 255, **279**, 279, 296
Parameter 900, 262, **314**, 314
Parameter 91 – Axis Properties for axes 1, 255, **280**, 280, 289
Parameter 92 – Axis Properties for axes 2, 255, **280**
Parameter 93 – Axis Properties for axes 3, 112, 229, 249, 255, **280**
Parameter 94 – Axis Properties for axes 4, 112, 229, 249, 255, 256, **280**, 280, 289
Parameter 95 – Autotune Move Distance / Auto Delay Calculation Move Distance for axes 1, 256, **281**, 281, 288, 317
Parameter 96 – Autotune Move Distance / Auto Delay Calculation Move Distance for axes 2, 256, **281**
Parameter 967, 167, **314**
Parameter 97 – Autotune Move Distance / Auto Delay Calculation Move Distance for axes 3, 256, **281**
Parameter 98 – Autotune Move Distance / Auto Delay Calculation Move Distance for axes 4, 256, **281**, 281, 288, 317
Parameter 99 – Cutter Compensation Look-ahead, 256
Parameter 997 – Spindle Cooling Fan Delay Timer, 262
Parameter 998, 262, **314**
Parameter 999, 262, **314**, 314
Parameter Setting, 196
Parameters 111–115 – Intercon parameters, **282**
Parameters 130–131 – 3rd/4th axis on/off selection, **283**
Parameters 132–135 – Motor Heating Coefficients for axes 5–8, **284**
Parameters 156–159 – Autotune Move Distance / Auto Delay Calculation Move Distance for axes 5, 6, 7, 8, **288**
Parameters 166–169 – Axis Properties for axes 5, 6, 7, 8, **289**
Parameters 170–179 – PLC Parameter, **289**
Parameters 188–199 – Aux Key Functions, **291**
Parameters 20–30 – Motor Temperature Estimation, **268**
Parameters 200–207 – OPTIC 4 Tach Volts Per 1000 RPM, **291**
Parameters 208–215 – MPU-based Lash/Screw Compensation Acceleration Coefficient, **292**
Parameters 220–231 – Smoothing Parameters, **293**
Parameters 236–239 – Motor Cooling Coefficients for axes 5–8, **293**
Parameters 252–255 – Autotune Ka Performance parameters for axes 5, 6, 7, 8, **296**
Parameters 270–271 – XY Skew Correction, **297**
Parameters 284–291 – Brake Resistor Wattage for ACDC Drives 1–8, **299**
Parameters 292–295 – Aux Key Functions 13–16, **299**
Parameters 300–307 – Drive assignment to Axes 1–8, **299**
Parameters 308–315 – Encoder assignment to Axes 1–8, **300**
Parameters 324–331 – Axis Boxcar size, **302**
Parameters 332–335 – Encoder error suppression, **302**
Parameters 336–339 – Motor torque estimation for velocity mode drives, **302**
Parameters 340–347 – Precision Mode delay (in milliseconds) for axes 1–8, **303**
Parameters 357–364 – Axis Drive Max RPM for Axes 1–8, **304**
Parameters 366–367 – Probe / TT1 deceleration multiplier, **304**
Parameters 374–379 – ACDC Drive Debug Log Settings, **305**
Parameters 387–389 – Debugging Parameters, **305**
Parameters 392–394 – DP-7 parameters, **305**
Parameters 441–443 – MPG (1–3) Axis Selection, **310**
Parameters 61–62 – Stall Detection Parameters, **274**
Parameters 65–67 – Spindle Gear Ratios, **275**
Parameters 700–799 – Dedicated Block of Parameters usable in Custom PLC Programs, **313**
Parameters 800–813 – DB25 Pin Selection, **313**
Parameters 87–90 – Autotune Ka Performance parameters for axes 1, 2, 3, 4, **279**
Parameters 900–999 – Reserved for Centroid PLC Program Use, **314**
Parameters 91–94 – Axis Properties for axes 1, 2, 3, 4, **280**
Parameters 95–98 – Autotune Move Distance / Auto

Draft: January 11, 2024

Delay Calculation Move Distance for axes 1, 2, 3, 4, 281
Parameters 998–999 – PLC program parameters, 314
Parameter 0 – E-Stop PLC Bit, 262
Parameter 1 – Machine and Jog key orientation, 262
Parameter 10 – Macro M-function Control/Probe Stop Handling, 265
Parameter 100 – Intercon comment generation, 281
Parameter 101 – Intercon clearance amount, 281
Parameter 102 – Intercon spindle coolant delay, 281
Parameter 103 – Intercon corner feed rate override, 281
Parameter 104 – Intercon modal line parameters, 281
Parameter 105 – Intercon modal arc parameters, 281
Parameter 106 – Intercon modal drilling cycle parameters, 281
Parameter 107 – Intercon chamfer blend, 282
Parameter 108 – Intercon polar display, 282
Parameter 109 – Intercon modal display, 282
Parameter 11 – DP4 PLC Input Number and Contact State, 266
Parameter 110 – Wizard Password, 282
Parameter 113 – Hide Intercon Menu, 282
Parameter 116 – A-axis Y-coordinate, 282
Parameter 117 – A-axis Z-coordinate, 282
Parameter 118 – B-axis X-coordinate, 282
Parameter 119 – B-axis Z-coordinate, 283
Parameter 12 – Touch Probe Tool Number, 266
Parameter 120 – Probe Stuck Clearance Amount, 283
Parameter 121 – Grid digitize prediction minimum Z-pullback, 283
Parameter 122 – Grid digitizing deadband move distance, 283
Parameter 128 – Dry Run PLC Bit, 283
Parameter 129 – Dry Run Feed Rate, 283
Parameter 13 – Recovery Distance, 266
Parameter 136 – G76 Fine Bore Retract Angle, 284
Parameter 137 – Load Meter Filter Size, 284
Parameter 138 – DRO Encoder Deadband, 284
Parameter 139 – Special Dwell between Moves, 285
Parameter 14 – Fast Probing Rate, 266
Parameter 140 – Message log priority level, 285
Parameter 141 – Maximum message log lines, 285
Parameter 142 – Message log trim amount, 285
Parameter 143 – DRO properties (load meter, DTG), 285
Parameter 144 – Comparison rounding, 286
Parameter 145 – Advanced macro properties (fast branching), 286
Parameter 146 – Feed hold threshold for feed rate override, 286
Parameter 147 – Number of Messages in Operator Message Window, 286
Parameter 148 – Miscellaneous Jogging Options, 286
Parameter 149 – Spindle Speed Threshold, 287
Parameter 15 – Slow Probing Rate, 266
Parameter 150 – Backplot Graphics display options, 287
Parameter 151 – Repeatability tolerance for probing and radial digitizing, 287
Parameter 153 – Probe Protection Enabled, 287
Parameter 154 – Touch Screen Options, 287
Parameter 155 – Probe Type, 287
Parameter 16 – Maximum Probing Distance, 266
Parameter 160 – Enhanced ATC, 288
Parameter 163 – Gang Tooling, 289
Parameter 164 – ATC Feature, 289
Parameter 165 – Acceleration/Deceleration Options, 289
Parameter 17 – Detector Location Return Point, 267
Parameter 170 – Enable Keyboard Jogging and Set Feed Rate Override Control, 290
Parameter 178 – PLC I/O configuration, 290
Parameter 179 – Lube Pump Operation, 290
Parameter 18 – PLC Input Spindle Inhibit Parameter, 267
Parameter 180 – Clear Home Switch Distance, 290
Parameter 186 – Probe Stuck retry disable, 291
Parameter 187 – Hard Stop Homing, 291
Parameter 19 – MPG modes, 267
Parameter 2 – G-code Interpretation Control and Slaving Rotary axis feed rate, 263
Parameter 216 – PC-based Lash Compensation on/off, 292
Parameter 217 – PC-based Screw Compensation on/off, 292
Parameter 218 – USB MPG Options, 292
Parameter 219 – Virtual Control Panel Options, 292
Parameter 240 – Rigid Tapping and Threading Accel Rate Distance, 293
Parameter 241 – Rigid Tapping and Threading Rotational Step Size (Degrees), 293
Parameter 242 – Minimum Angle Threshold for application of Accel/decel in Threading moves, 294
Parameter 243 – Threading Control, 294
Parameter 244 – Tool Touch-off device PLC input number, 294
Parameter 245 – G71, G72, G74, G75, G76 D/P/Q “implied float” re-interpretation threshold for Inch units, 294
Parameter 246 – G71, G72, G74, G75, G76 D/P/Q “implied float” re-interpretation threshold for MM units, 294
Parameter 247 – G70 Multiple Pass Behavior Suppression, 295

Draft: January 11, 2024

Parameter 248 – Tool Wear Adjustment magnitude limit, 296
Parameter 256 – Drive Mode, 296
Parameter 257 – TT1 connection detection PLC input, 296
Parameter 258 – Velocity/Torque Mode override in Precision mode, 296
Parameter 259 – Manual Axis Designation, 296
Parameter 260 – Accelerated Graphics Backplot, 297
Parameter 261 – Precision Mode Standoff Tolerance Percentage, 297
Parameter 262 – Precision Mode Standoff Delay Time, 297
Parameter 263 – DRO Display Precision, 297
Parameter 264 – Rapid Override, 297
Parameter 278 – Spindle Speed Display Precision, 298
Parameter 281 – Tool Touch-off Device X stylus size, 298
Parameter 282 – Tool Touch-off Device Z stylus size, 298
Parameter 283 – Auto Tool Touch-off safety clearance, 298
Parameter 299 – Report Option Bits, 299
Parameter 3 – Modal Tool and Height Offset Control, 263
Parameter 31 – Legacy SPIN232 Com Port, 269
Parameter 316 – Absolute Encoder Bits, 301
Parameter 317 – Single-turn Absolute Encoder Bits, 301
Parameter 318 – Five-axis Configuration, 301
Parameter 319 – Five-axis Options, 301
Parameter 32 – Autonomous Digitizing Offset Files, 269
Parameter 321 – MPU13 DSP Probe Input, 301
Parameter 322 – Spindle Encoder Port, 301
Parameter 323 – MPU11 Encoder Speed Filter, 302
Parameter 33 – Spindle Motor Gear Ratio, 269
Parameter 34 – Spindle Encoder Counts/Rev, 269
Parameter 348 – MPG/Handwheel Encoder Input 1, 303
Parameter 349 – MPG/Handwheel Detents-per-revolution 1, 303
Parameter 35 – Spindle Encoder Axis Number, 269
Parameter 350 – MPG/Handwheel Encoder Counts-per-revolution 1, 304
Parameter 351 – MPG/Handwheel Encoder Input 2, 303
Parameter 352 – MPG/Handwheel Detents-per-revolution 2, 303
Parameter 353 – MPG/Handwheel Encoder Counts-per-revolution 2, 304
Parameter 354 – MPG/Handwheel Encoder Input 3, 303
Parameter 355 – MPG/Handwheel Detents-per-revolution 3, 303
Parameter 356 – MPG/Handwheel Encoder Counts-per-revolution 3, 304
Parameter 36 – Rigid Tapping Enable/Disable, 270
Parameter 365 – Drive power-on delay, 304
Parameter 368 – Autonomous Digitizing Angle Adjustment, 304
Parameter 369 – Tool Check Max Absolute Angle, 305
Parameter 37 – Spindle Deceleration Time, 270
Parameter 38 – Multi-axis Max Feed Rate, 270
Parameter 39 – Feed Rate Override Percentage Limit, 271
Parameter 395 – Probing Setup Traverse Speed, 305
Parameter 396 – Probing Setup Plunge Speed, 305
Parameter 397 – Combustion Chamber Clearance Height, 305
Parameter 398 – Port/block mode, 305
Parameter 399 – AD1 arc chord tolerance adjustment, 306
Parameter 4 – Remote File Loading Flag & Advanced File Ops, 263
Parameter 40 – Basic Jog Increment, 271
Parameter 400 – Run Menu Cycle Start Enabled, 306
Parameter 401 – Forget Last Job Loaded, 306
Parameter 403 – Disable Keyboard Jogging Legend, 306
Parameter 404 – Unused, 306
Parameter 405 – Tool Touch-off Type, 306
Parameter 406 – Probe State When Tripped, 306
Parameter 408 – Active G-Codes, 307
Parameter 409 – Probe Type, 307
Parameter 41 – Rotary Axis Jog Increment, 271
Parameter 410 – Probe/Tool Touch-off Warning, 307
Parameter 411 – MPG Type, 307
Parameter 413 – Park macro, 307
Parameter 414 – Home File Type, 307
Parameter 415 – Ether1616 Configuration Bits, 307
Parameter 416 – Spindle Inhibit, 308
Parameter 417 – Preview G-code Before Job, 308
Parameter 418 – Assign Enter Key to Cycle Start, 308
Parameter 419 – PLC Worklight and Popup Pin Output, 308
Parameter 42 – Password for Configuration Menus, 271
Parameter 420 – Analog/PWM 1 Settings, 308
Parameter 421 – Clean Fan Filter Reminder, 308
Parameter 422 – PLC Diagnostics Display, 309
Parameter 423 – Display Scale Position on DRO, 309
Parameter 425 – Force Rehoming After EStop Condition, 309
Parameter 426 – WCS Lockout, 309
Parameter 43 – G71/G72 Depth of Cut, 271

Draft: January 11, 2024

Parameter 430 – RTG Spindle Speed RPM Display, **310**
Parameter 44 – G71/G72 Escape Amount, **271**
Parameter 440 – Stopped for Jogging Continue Bit, **310**
Parameter 45 – G74 X-axis Relief Amount, **271**
Parameter 450 – Inhibit MPG Z-axis Fast Movement, **310**
Parameter 458 – C-Axis Options, **311**
Parameter 459 – Second Spindle Encoder Axis Number, **311**
Parameter 46 – G75 Z-axis Relief Amount, **272**
Parameter 460 – Second Spindle Maximum Speed, **311**
Parameter 461 – Second Spindle Minimum Speed, **311**
Parameter 462 – Second Spindle Encoder Counts/Revolution, **311**
Parameter 47 – G73 Repeat Count, **272**
Parameter 48 – Grid Digitize Patch Playback Z rapid clearance amount, **272**
Parameter 49 – Thread Chamfer Amount, **272**
Parameter 5 – Machine Home/Startup Setup, **264**
Parameter 50 – G76 Finish Count, **272**
Parameter 507 – 5th-axis Pairing – Slave Axis, **311**
Parameter 508 – 5th-axis Pairing – Master Axis, **311**
Parameter 51 – G76 Thread Angle, **272**
Parameter 52 – G76 Minimum Cutting Depth, **272**
Parameter 53 – G76 Finish Allowance, **272**
Parameter 55 – Radius / Diameter mode, **273**
Parameter 56 – Feedrate Override Display Properties / Inverse Time Interpolation, **273**
Parameter 560 – Laser X-offset, **312**
Parameter 561 – Laser Y-offset, **312**
Parameter 57 – Use Generic Load Meter Data from PLC, **273**
Parameter 58 – G71/G72 Profile Rough Cut Escape Angle, **273**
Parameter 59 – Tool Check Options, **274**
Parameter 6 – Auto Tool Changer Installed, **264**
Parameter 600 – A-axis Expected Velocity Switch Point, **312**
Parameter 601 – A-axis Scale Switch To Velocity, **312**
Parameter 602 – A-scale Correction Meter Max Deflection Count, **312**
Parameter 603 – B-axis Expected Velocity Switch Point, **312**
Parameter 604 – B-axis Scale Switch To Velocity, **312**
Parameter 605 – B-scale Correction Meter Max Deflection Count, **312**
Parameter 63 – High Power Idle PID Multiplier, **274**
Parameter 64 – Fourth/Fifth-axis Pairing, **274**
Parameter 68 – Minimum Rigid Tapping Spindle Speed, **275**
Parameter 69 – Duration For Minimum Spindle Speed, **275**
Parameter 7 – Display Colors, **264**
Parameter 70 – Offset Library Inc/Decrement Amount, **276**
Parameter 71 – Part Setup Detector Height, **276**
Parameter 72 – Data M-function Options, **276**
Parameter 73 – Peck Drill Retract Amount, **276**
Parameter 74 – M-function executed at bottom of tapping cycle, **276**
Parameter 75 – Summing Control, **276**
Parameter 76 – Manual Input Unrestricted Distance, **277**
Parameter 77 – Manual Input Movement Tolerance, **278**
Parameter 78 – Spindle Speed Display and Operations, **278**
Parameter 79 – Paired Axes Re-sync Delay Time, **278**
Parameter 8 – Available Coolant Systems, **265**
Parameter 80 – Voltage Brake Message Frequency, **278**
Parameter 81 – Air Drill M-function, **278**
Parameter 814 – Analog/PWM 2 Settings, **313**
Parameter 815 – Inverse Output, **314**
Parameter 817 – PWM, **314**
Parameter 82 – Spindle Drift Adjustment, **279**
Parameter 83 – Deep Hole Clearance Amount, **279**
Parameter 830 – ATC Type, **314**
Parameter 84 – M-function executed at return to initial point of tapping cycle, **279**
Parameter 85 – “Door Open” Interlock PLC bit, **279**
Parameter 852 – Carousel Skip Count on Reverse, **314**
Parameter 853 – Distance to travel past tool position, **314**
Parameter 854 – Distance to travel behind tool position, **314**
Parameter 855 – MPG Performance, **314**
Parameter 856 – VCP Jogging State on Acorn Power Up, **314**
Parameter 86 – Rapid/Linear vector rate limit, **279**
Parameter 9 – Display Language, **265**
Parameter 99 – Cutter Compensation Look-ahead, **281**
Parameter 997 – Spindle Cooling Fan Delay Timer, **314**
part count, 10
part number, 10
Part Setup, **47**
Password, 244
PLC I/O Diagnostic App, 319
PLC Type, 246
plcstate.txt, 35
Power Feed, 85
Prepare for Tool Change, 227
Probing Cycles History, **36**
Profile Finishing Cycle, 204

Draft: January 11, 2024

Profile Path Repeating Cycle, 209
Profile Stock Removal in Facing, 207
Profile Stock Removal in Turning, 205
program number, 10
Programmed Action Timer, 233
Prompt for Operator Input Using Formatted String, 240
Protected Move Probing Functions, 234
PWM Velocity Modulation ON/OFF, 198

Quill In, **21**
Quill Out, **21**

radius mode, 9, 10
Rapid Positioning, 193
Rapid Positioning in Machine Coordinates, 202
Record Current Job File Path to Data File, 237
Record Date and Time in a Data File, 237
Record Local Position(s) and Optional Comment in Data File, 236
Record Machine Position(s) and Optional Comment in Data File, 237
Record Value and/or Comment in Data File, 236
Release/Restore Motor Power, 230
Remote Drive & Directory, 246
Restart Program, 225, 228, 233
Resuming a Canceled Job, 83
Return from Macro or Subprogram, 232
Return from Reference Point, 197
Return to Reference Point, 197
Return to Secondary Reference Point, 197
RIGHT, 119
Run MDI, **34**
Run Menu, 83
run menu, 10
Run System Command, 238
Run-time Graphics Screen, 82
Running a Job, **81**

Save PLC state to file, **35**
Select Inch Units, 196
Select Metric Units, 196
Select Next WCS, **35**
Select Previous WCS, **35**
Select Spindle Gear Range, 228
Select WCS, **35**
Select Work Coordinate System, 202
Set Axis Home, 228
Set Spindle Encoder to Zero at Next Index Pulse, 238
Set Tap Mode for G84, 228
SHIFT+F1, **36**, 36
  Switch to Old-style Graphics Backplot, 36
SHIFT+F2, **36**, 36
  Erase Log File, 36
SINGLE BLOCK, 18
Single Block, **18**
Skinning, **15**
Slow/Fast, **17**, 17
Smoothing AD2 Setup Menu, 330
Smoothing Configuration Parameters, 326
SPACE, 48, 245, 247
Space, 94, 95
SPIN START, 20
Spin Start, **20**, 49, 50, 66, 69
SPIN STOP, 20
Spin Stop, **20**
SPINDLE (AUTO/MAN), 20
Spindle Auto/Man, **20**
SPINDLE CLOCKWISE/COUNTERCLOCKWISE, 19
Spindle CW/CCW, **19**
Spindle On Clockwise, 225
Spindle On Counter-clockwise, 226
Spindle Orient, 228
spindle speed, 10
Spindle Speed +, **19**
Spindle Speed -, **19**
Spindle Speed 100%, **19**
Spindle Stop, 226
Status Window, **10**
Stop for Operator, 225
Stop for Operator, Prompt for Action, 240
Swap DRO and Distance-to-Go DRO, **35**
Switch to Old-style Graphics Backplot, **36**

T-stock In, **21**
T-stock Out, **21**
Tapping, 216
Technical Background of AD2 Smoothing, 333
Temperature Display, **35**
Thread Cutting Cycle, 220
Tool Change, 226
TOOL CHECK, 18, 53, 84
Tool Check, **18**, 83
Tool Library, **124**
tool offset, 10
Tool Setup, **56**
Turret Index, **21**

units of measure, 9
Unwind C-axis, 238
UP, 84, 119, 125
up, 106
User Specified Paths, 246
User Window, **11**

VCP, **22**

Wait for PLC Bit (Closed, On, Set), 232
Wait for PLC Bit (Open, Off, Reset), 232

Draft: January 11, 2024

WCS, 9, **52**
1, 54, 202, 203, 310
2, 54, 203, 310
3, 54, 203, 310
4, 54, 203, 310
5, 54, 203, 310
6, 54, 202, 203, 310
7, 54, 203, 310
8, 54, 203, 310
9, 54, 203, 310
10, 54, 203, 310
11, 54, 203, 310
12, 54, 203, 310
13, 54, 203, 310
14, 54, 203, 310
15, 54, 203, 310
16, 54, 203, 310
17, 54, 203, 310
18, 54, 203, 310
WCS/Machine Coordinates, **33**
Wirelesss MPG Installation, 337
Work Coordinate Systems, **52**
Work Envelope Off, 196
Work Envelope On, 196
Write Formatted String to File, 240

x1, **17**
x10, **17**
x100, **17**

Y, 91

Draft: January 11, 2024