CNC-TURN-OP01-EN

DynaPath

WinDelta Turning Control

Control, Operating, and Programming Manual

DynaPath Systems Inc.
Revised: 2025-09-25

General Information

DYNAPATH MACHINES MAY BE SERVICED, REPAIRED OR MAINTAINED ONLY BY AUTHORIZED SERVICE PERSONNEL.

DynaPath products should be used only for the purpose and in the manner intended by their original design. It is recommended that no modifications be made to this product. Any modification of this product will void any expressed or implied warranty.

DynaPath Systems, Inc. reserves the right to incorporate any modifications or improvements in machines and machine specifications which it considers necessary, and does not assume any obligation to make any said changes in machines or equipment previously sold.

DynaPath products and services are subject to DynaPath's then current prices, terms, and conditions, which are subject to change without notice.

PROPRIETARY INFORMATION

This document contains information considered to be proprietary to DynaPath Systems, Inc. Such information may not be used for the manufacture of equipment of the type described here. The information in this document is subject to change without notice and does not represent a commitment on the part of DynaPath Systems, Inc. (DynaPath). The software described in this document is furnished under the License Agreement to customers. It is against the law to copy the software on any medium except as specifically allowed in the license agreement. The purchaser may make copies of the software for backup purposes. No part of this document may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying, for any purpose without the express written permission of DynaPath.

2020© DynaPath Systems, Inc. All rights reserved.
http://www.DynaPath.com

SAFETY INFORMATION

Machine tool owners, operators and others who work around machine tools must pay close attention to safety guidelines. Consult the safety publications below:

- **Safety Requirements for the Construction, Care and Use of Drilling, Milling, and Boring Machines (ANSI B11.8-1983).**

  This publication states that the employer is responsible for safeguards. Section 5.1 of this publication says, "it shall be the responsibility of the employer to provide, and ensure the use of, a guard, guarding device, awareness barrier, awareness device, or shield . . ." To obtain a copy, contact:

  The American National Standards Institute
  1430 Broadway
  New York, New York 10018

- **Concepts and Techniques of Machine Safeguarding (OSHA Publication Number 3067).**

  This document is meant to help machine owners and operators devise safeguards. To obtain a copy, contact:

  The Publication Office – OSHA
  U.S. Department of Labor
  200 Constitution Avenue, NW
  Washington, D.C. 20210

Table of Contents

1 INTRODUCTION 9
1.1 About this Manual 10
1.2 Control Overview 10
2 SAFETY 12
2.1 Introduction 13
2.2 Safety Symbols 13
2.3 Safe Operating Information 14
2.4 Release of Trapped Persons. 17
2.5 Safety Notices and Labels 18
3 SYSTEM SPECIFICATIONS 21
3.1 Hardware Components 22
3.1.1 Control Chassis 23
3.1.2 D1 Console 25
3.1.3 D4 Console 26
3.1.4 D5 Console 27
3.1.5 Handheld MPG 28
3.2 Software Components 29
3.2.1 Software Licenses and Agreement 29
4 OPERATION 31
4.2 Introduction to Operation of CNC Machine Tool 32
4.3 Pre-Startup Inspection 32
4.4 Power-On and Start-Up Procedure 32
4.5 Power-Off and Shutdown Procedure 33
4.6 Emergency Stop 34
4.7 Over-travel (OT) and Recovery 34
4.7.1 Hardware Over-travel 34
4.7.2 Software Over-travel 34
4.8 CNC Modes and Panel Operation 35
4.8.1 Mode Screen Style Operation 36
4.8.2 Mode Switch Style Operation 38
4.9 Coordinate Systems 39
4.10 Work Offset Concepts 40
4.11 Tool Offset Concepts 40

5 HOME MODE 41
5.1 Home Mode Screen 42
6 Manual Modes 43
6.1 Jog Mode 44
6.2 MPG Mode 45
6.3 Incremental Jog Mode 46
6.4 Spindle Control 46
6.4.1 C-Axis 46
6.5 Panel Functions 46
7 Auto Mode 47
7.1 Basic Operation 48
7.1.1 Cycle Start 49
7.1.2 Feed Hold 49
7.1.3 Spindle, Rapid, Feed Overrides 49
7.2 File Manager 49
7.3 Single Block 49
7.4 MPG Run 49
7.5 Retrace 49
7.6 Reset 49
7.7 Panel 50
8 WORK OFFSET 51
8.1 Offset Concepts 52
8.2 Work Offset 53
9 TOOL SET UP 54
9.1 Tool Offset 55
9.1.1 Measure 57
9.1.2 Tool Orientation 58
9.1.3 Call Tool Function 60
9.2 Tool Wear 60
10 DRO / SEMI-AUTO 61
10.1 Introduction 62
10.2 Manual Operation 62
10.3 Move To 63
10.4 Feed Line 63
10.5 Threading 64

10.6 Feed Arc 65
11 MDI 66
11.1 MDI Operation 67
12 File Management 68
12.1 File Manager 69
12.2 Send and Receive 70
13 G-Code Programming 71
13.1 G-Code Reference 72
13.2 M-Code Reference 74
13.3 T-Code Reference 75
14 MACRO PROGRAMMING 76
14.1 Introduction 77
14.2 Macro Program Names 77
14.3 Macro Program Types 77
14.4 Macro Variables 78
14.4.1 Macro Variables Overview 78
14.4.2 Local Input Variables 78
14.4.3 System Variables 79
14.5 Arithmetic Functions 82
14.6 Priority of Operations 83
14.7 Bracket Nesting 83
14.8 Branch and Repetition 83
15 CONVERSATIONAL PROGRAMMING 85
15.1 Introduction 86
15.2 Conversational Programming Workflow 86
15.3 Main Screen 87
15.4 Program Settings 88
15.5 Stock Removal Events 91
15.5.1 Profile Event 91
15.5.2 Stock Removal – Turning (G71) 92
15.5.3 Stock Removal – Facing (G72) 94
15.5.4 Stock Removal – Pattern Repeating (G73) 96
15.5.5 Turning (G90) and Facing (G94) 98
15.5.6 Turning Profile 100
15.6 Hole Events 101

15.6.1 Face Drill (G83) 101
15.6.2 Face Tap (G84) 102
15.6.3 Face Bore (G85) 103
15.6.4 Side Drill (G87) 104
15.6.5 Side Tap (G88) 105
15.6.6 Side Bore (G89) 106
15.7 Thread Events 107
15.7.1 Thread and Tapered Thread 107
15.8 Groove Events 109
15.8.1 Face Groove 109
15.8.2 Turn Groove 112
15.8.3 Irregular Groove 115
15.9 Cut-Off 118
15.10 Line-Arc 119
15.10.1 Linear (G01) 119
15.10.2 CW Arc (G02) 119
15.10.3 CCW Arc (G03) 120
15.11 Position 120
15.11.1 Rapid XZ (G00) 120
15.12 Auxiliary 121
15.12.1 Program Stop 121
15.12.2 Coolant 121
15.12.3 Dwell 121
15.12.4 Tool Change 122
15.12.5 Tool Radius Compensation 122
15.12.6 Chuck Clamp/Unclamp 122
15.12.7 Reference Point Return 122
15.12.8 Bar Feed 123
15.12.9 Chip Auger 123
15.12.10 Comment 123
15.12.11 G-Code 123
16 DRAWING IMPORT 124
16.1 Introduction 125
16.2 Drawing Import Workflow 125
16.3 Drawing Import Main Screen 126

16.4 File Mode and Functions 127
16.5 Set Zero Mode and Functions 128
16.6 Select Mode and Functions 129
16.7 View Mode and Functions 131
16.8 Measure Mode and Functions 132
16.9 Draw Mode and Functions 133
16.10 Edit Mode and Functions 136
17 SYSTEM 138
17.1 System Introduction 139
17.2 Parameters 140
17.3 Maintenance 141
17.4 Message 141
17.5 Environment 141
18 NETWORKING 142
18.1 Introduction to Networking 143
18.2 Networking Set Up 143
18.3 Remote File Management 144
18.4 Remote Assistance 144
19 Alarms and Messages 146
19.1 Motion Module Alarms 147

1 INTRODUCTION

1.1 About this Manual

The intent of this manual is to provide a comprehensive reference for operating, programming, troubleshooting, and maintaining the DynaPath WinDelta computer numerical controller (CNC) system.

1.2 Control Overview

DynaPath WinDelta CNC is a microprocessor-based numerical control system used to control multi-axis machining centers and milling machines. The control system contains the features and functions listed in Table 1.2.1:

Table 1.2.1: WinDelta Control System Features and Functions

OPERATION	• Mode Switch or Mode Select operation • Industrial touchscreen • PS/2 or USB mouse and keyboard
MANUAL MODE	• JOG, INCREMENTAL JOG, MPG modes.
DRO/SEMI-AUTO MODE	• Manual operation axis tracking. • Semi-Automatic command operation.
HOME MODE	• Individual or simultaneous home. • Home method, direction, speed configuration.
AUTO MODE	• MDI, Memory mode program execution. • Feed, Spindle, Rapid over-rides. • MPG Run function. • Program Retrace function. • Background Edit function.
EDIT MODE	• ISO G/M/S/T-code compatibility. • Macro programming. • Save to and copy from USB. • Built-in FTP server for remote program management. • Conversational Programming with drill, bore, tap, contour, pocket, thread mill, tapered pocket, pattern, and set up events. • DXF Import with CAD editing capability.
TOOL AND FIXTURE TABLES	• 54 Fixture offsets. • 200 Tool length, radius, and wear offsets. • Workpiece alignment guided functions.
MOTION	• 4 Axis synchronous, 6 total axes. • 4 Independent paths. • PTP, Linear, Circular, Helical, NURBS interpolation. • 1000 Block look-ahead. • Corner accuracy setting.

	• Jerk control. • S-curve acceleration/deceleration. • Backlash and Pitch Error compensation. • Tool radius and length compensation. • Canned cycle and macro programming. • Fully digital/Analog voltage command. • Position loop feedback PI, feed-forward. • Notch filter for resonance control.
TUNING	• Online servo tuning utility for resonance, vibration, torque filter, velocity loop, position loop, and feed-forward.
NETWORK	• Online remote diagnostics. • Remote training and support.
SYSTEM	• Windows PC-based architecture. • Customizable HMI with GPeX graphical programming tool. • Alarm and event history database. • Open CNC and 3rd party software integration. • 2x USB and 1x LAN ports. • 4GB internal program storage capacity.

2 SAFETY

2.1 Introduction

The machine tool should be used by machinists who have a proper understanding of the operation of a CNC machines. The operator should have had training in how a milling machine operates and experience in cutting material for manufacturing. Any operator using the machine needs to carry out their own risk assessment of the machine for its intended use.

The safe operation of the DynaPath WinDelta control system depends on its correct use and the operator should ensure that they take all appropriate precautions to prevent injury to themselves and the machine.

2.2 Safety Symbols

Safety symbols are used throughout this publication and on the machine where applicable to display important safety information pertaining to hazardous voltages, currents, temperatures, mechanical harm, or other conditions that can cause personal injury and damage to the machine. The following list outlines the general safety classifications and its usage.

Table 2.2.1: Safety Symbols

CAUTION!	**CAUTION** notices are used in this publication to emphasize that mishandling produces a potentially hazardous situation which, if not avoided, ***could*** result in machine equipment damage and moderate or minor injury.
WARNING!	**WARNING** notices are used in this publication to emphasize that mishandling would produce a potentially hazardous situation which, if not avoided, ***could*** result in machine equipment damage and serious injury.
DANGER!	**DANGER** indicates cases where mishandling includes an imminently hazardous situation which, if not avoided, ***will*** result in death or severe injury.
NOTE:	**NOTE** indicates important information requiring special attention or understanding.

2.3 Safe Operating Information

This control system and machine it is integrated with together constitute a machine tool designed for the machining of materials within its stated capacity, with all machine movement occurring by manual use of handwheels, use of the electronic handwheel, operation of the control panel, or automatic program execution as programmed by the operator.

Before machining any materials that are flammable or hazardous, the operator must ensure they have carried out a risk assessment and actioned any safety measures highlighted in the assessment.

The machine should only be used in a standard workshop environment with safety equipment available for the safe operation of the machine.

The machine should be installed, operated and maintained as stated in the Provision and use of Work Equipment Regulations (1998) or local equivalent regulations and this is the responsibility of the Employer, Machine Owner, or Machine Operator.

Before the machine is used, the responsible person in the workplace should:

a. Carry out a risk assessment on the machine's use and verify that the unique features of the DynaPath control are accounted for to ensure the machine and guarding system are fit for the purpose.

b. Produce and ensure it is actioned on the Safe Operating Procedures of the milling machine.

c. Ensure that any items highlighted in a risk assessment such as the use of personal protective equipment (PPE), training or use of guards are actioned.

The machine must not be used by any operator until this operation manual has been read and understood, with the operator fully conversant with the functions and features of the control and machine and fully understands the safety requirements of using the machine.

This machine is designed to be used by operators who have received training or have a level of experience associated with this type of machine. Any other operator should be trained and have been risk assessed by a responsible person who knows how to operate and use the machine.

It is the reasonability of the operator to ensure that all the safety features fitted to the machine and control system are checked on a regular basis. This should be carried out at a minimum at the start of every machine operation session or shift.

The following should be checked for correct functionality and operation.

- Emergency stop (E-STOP) buttons located on the control panel and the Manual Pulse Generator should be verified. When depressed, axes movement should

not occur until the E-STOP buttons are released and the RESET button pressed to enable the drives.

- Door guards should not open whist the spindle is running. If equipped, the guard lock or shot bolt should armed and not allow the guards to open. Stopping the spindle will release the shot bolt and the axes may only be jogged at a maximum safe speed (generally 2m/min or 79” per minute).

- All guards should be checked for damage. If vision through the clear front panels is obscured or the stated schedule for replacement has been reached the panels should be replaced immediately. For retractable guards, ensure the guards are lockable in place and in good condition, otherwise they should be replaced.

- Table guards should be inspected for signs of damage, especially the transparent panels. Replace if any part of the guard is damaged. Replace the transparent panels in accordance within the stated schedule with the proper safety specification, regardless of their apparent condition.

- Any retractable guarding must be closed to a safe position prior to operation. For machines with open enclosures such as bed or knee mills, this applies to the retractable guarding covering the spindle, along the rear of the table, or around the work envelope.

When operating this machine, always observe the following safety precautions

- Do not operate this machine without knowing the function of every control key, button, knob, or handle.

- Always wear the appropriate personal protective equipment, including safety glasses and safety shoes.

- Do not wear loose fitting gloves whilst operating this machine as they could easily get caught in moving parts.

- Never wear rings, watches, long sleeves, neckties, jewelry, or other loose items when operating the machine.

- Keep your hair away from moving parts. Wear adequate safety head gear.

- Never operate any machine tool after consuming alcoholic beverages, or taking strong medications, or while using non-prescription drugs.

- Carry out a COSHH risk assessment and use the correct protection equipment, e.g. barrier cream/latex gloves, to prevent harm from items such as cutting fluid, lubrication oil and other substances used on the machine.

- Do not use compressed air to remove swarf or clean the machine. This can damage the slideway seals and create coolant mist which can be harmful.

- Always ensure the appropriate guarding is in place for the machinery operation being undertaken. Never reach around a guard to gain access to the part, tool, or fixture.

- Do not raise the cutting tool above the table guards without additional guarding measures such as a cutter guard.

- Observe and understand the warning and safety information labels affixed to this machine.

- Do not attempt to tamper with or override any guarding/safety device fitted to the machine.

- Keep the working area clear and remove all tools (spanners etc.) from the machine before you start the machine running. Loose items can become dangerous flying projectiles.

- Stop the machine spindle, open the table guards to disable the spindle drive before:
    - Before changing tools.
    - Before changing parts.
    - Before you clear away the swarf, oil or coolant. Always use a chip scraper or brush.
    - Before you make an adjustment to the part, vice, coolant nozzle or take measurements.

- Keep work area well lit. Ask for additional light if needed.

- Be aware that the machine can move unexpectedly so do not lean on the machine while it is running.

- To prevent slippage and personal injury, keep the working area around the machine dry and clean. Ensure there is no swarf, oil, coolant and obstacles of any kind around the machine.

- Avoid getting pinched in places where the spindle, table or guard doors create "pinch points" whilst the machine is in motion.

- For machines fitted with manual handwheels: to prevent injury during powered axes movement, always keep the handle folded inside the hand wheel, except when required to hand crank the table.

- Securely clamp and properly locate the workpiece in the vice or in a fixture. Use proper tool holding equipment.

- Use the correct tooling for the process being undertaken. Never use damaged or worn tools and ensure the correct cutting parameters (speed, feed, and depth of cut) are used in order to prevent tool breakage.
- Prevent damage to the workpiece or the cutting tool. Never start the machine (including the rotation of the spindle) if the tool is in contact with the part.
- Avoid large overhangs on cutting tools when not necessary.
- To prevent fires, keep flammable materials and fluids away from the machine, hot swarf and workpieces.
- Never change gears when the spindle is rotating.
- Do not rotate the spindle by hand unless the table guard is open.
- Stop and disconnect the power to the machine before undertaking any machine cleaning or maintenance

2.4 Release of Trapped Persons.

In the event of persons being trapped in the machine:

1. Hit the emergency stop button to kill all power to the spindle and axes.
2. Open the table guards.
3. If trapped by spindle or tool, remove the compressed air supply to the machine and rotate the spindle by hand to free the trapped person.
4. If trapped by the axes, use the manual handwheels to move the axes slowly clear of the trapped person.
5. If no manual handwheels are fitted, release the E-STOP button, leave the safety guards open, and put the control into JOG mode and select the MPG function. Select the appropriate axis and smallest increment and slowly turn the MPG handwheel to move the machine and release the trapped person.

2.5 Safety Notices and Labels

The following safety notices and labels are used on machine tools integrated with this class of CNC control to denote hazards and present safety guidelines. They must not be defaced, destroyed, or removed.

Figure 2.5-1: Safety label for hazardous voltage

Figure 2.5-2: Safety label for guard doors

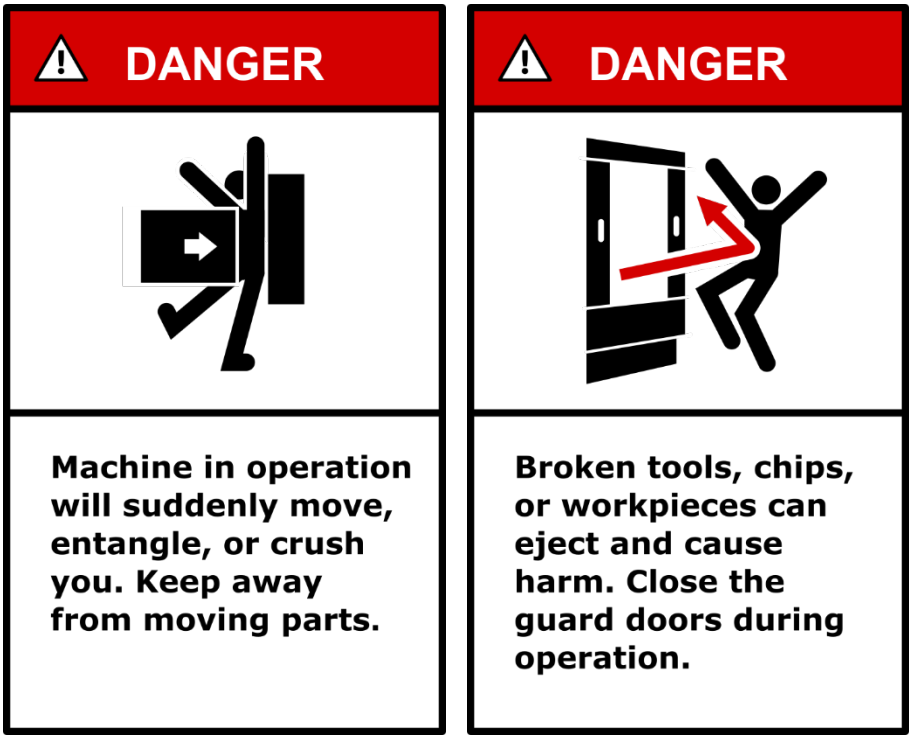


Figure 2.5-3: Safety labels for machine movement and debris

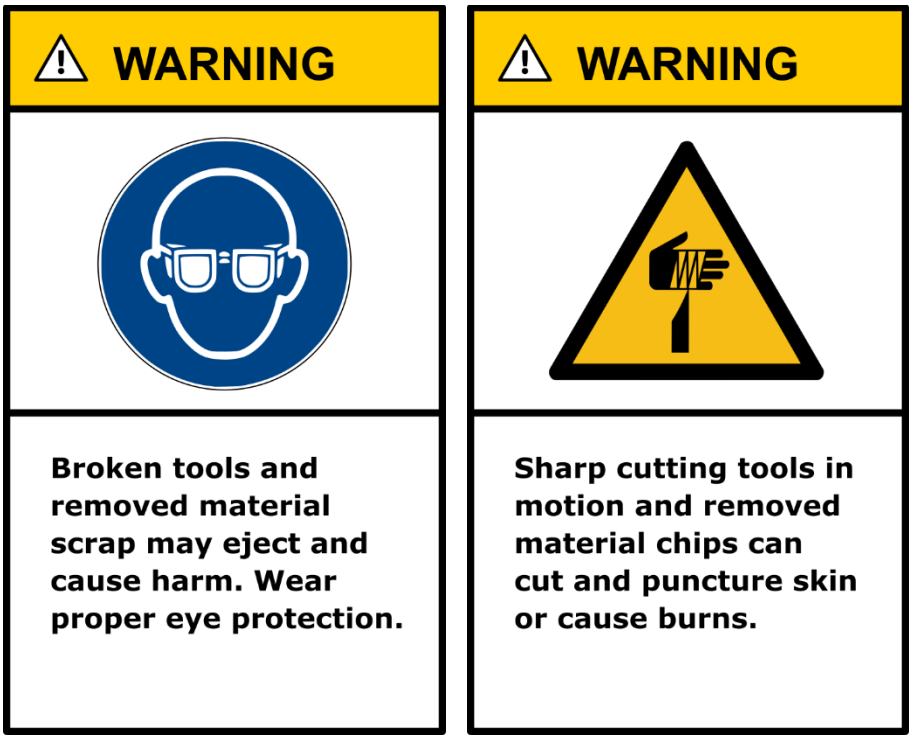


Figure 2.5-4: Safety labels for PPE and bodily injury

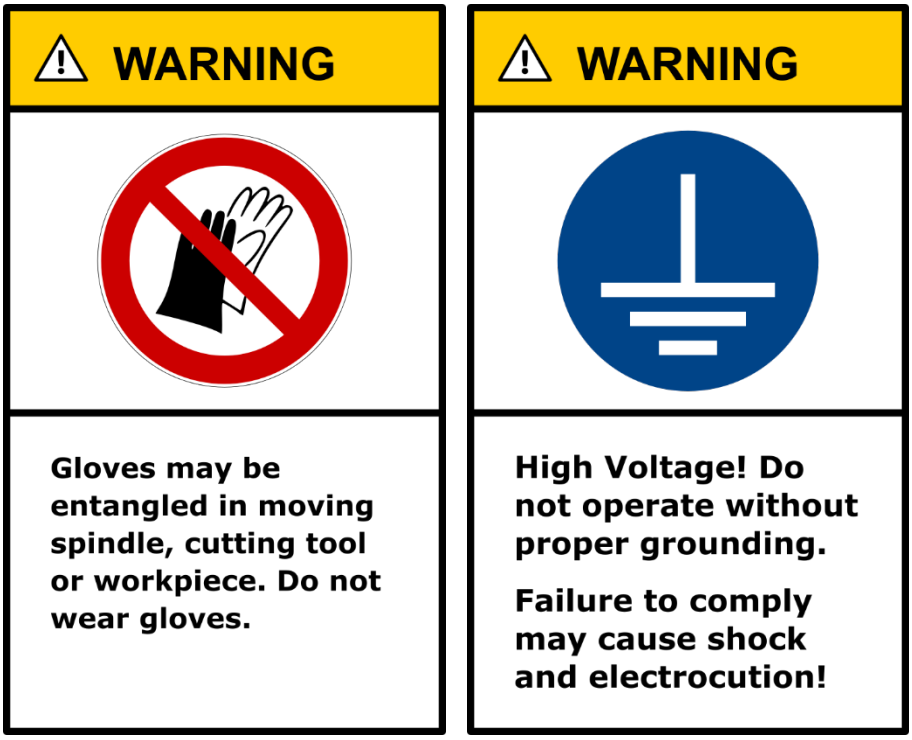


Figure 2.5-5: Safety labels for gloves and high voltage

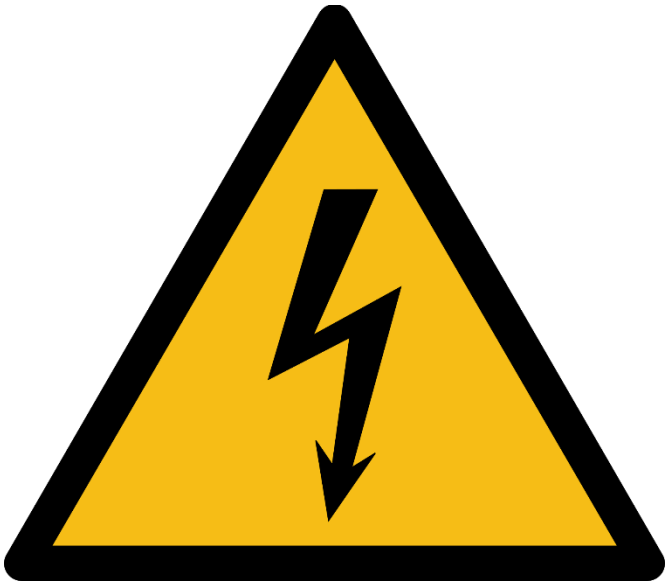


Figure 2.5-6: Safety label for electrical hazard

3 SYSTEM SPECIFICATIONS

3.1 Hardware Components

The DynaPath WinDelta CNC console consists of the following hardware features and options:

- 10.4”, 12.1” LCD Display with industrial touchscreen.
- Tactile switch based rugged keys.
- Manual data input (MDI) panel for data input.
- Console options: D1, D4, D5.
- Manual pulse generator (MPG) handwheel (optional).
- External keyboard and mount (optional).

All DynaPath WinDelta Mill CNC systems come equipped with a 10.4” or 12.1” display with touchscreen capabilities. The touchscreen is an industrially rugged, resistive-type touchscreen that is pressure sensitive and is equipped with software calibration. Depending on the type of machine and operation style, several console panel configurations exist: D1, D4, and D5. The MPG handwheel is an option commonly used for larger machines and machining centers. A standard keyboard is an option that is commonly used by operators who program extensively at the control.

DynaPath WinDelta control and console specifications are presented in the following sections.

3.1.1 Control Chassis

The DynaPath WinDelta is a series of control systems offered in several form factors and chassis depending on the target requirements and application.

The DynaPath WinDelta Node-355 control system chassis dimensions and ports are shown in Figure 3.1-1.

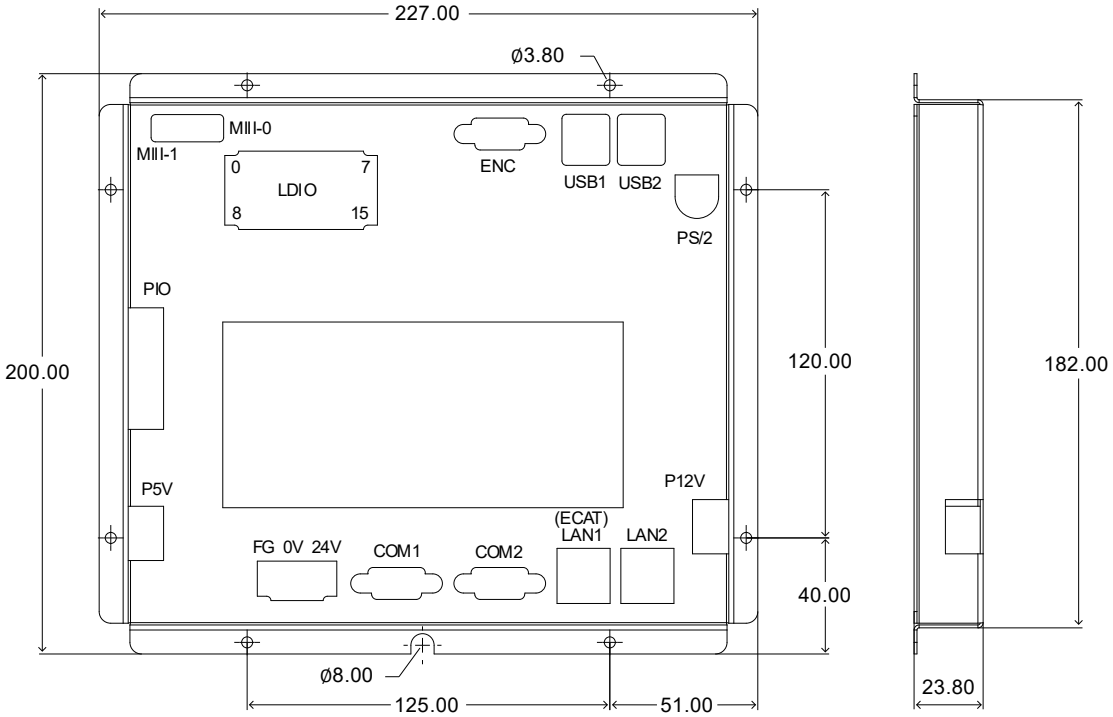


Figure 3.1-1: DynaPath WinDelta Node-355 Control Chassis

The Node-1000 control system chassis dimensions and ports are shown in Figure 3.1-2.

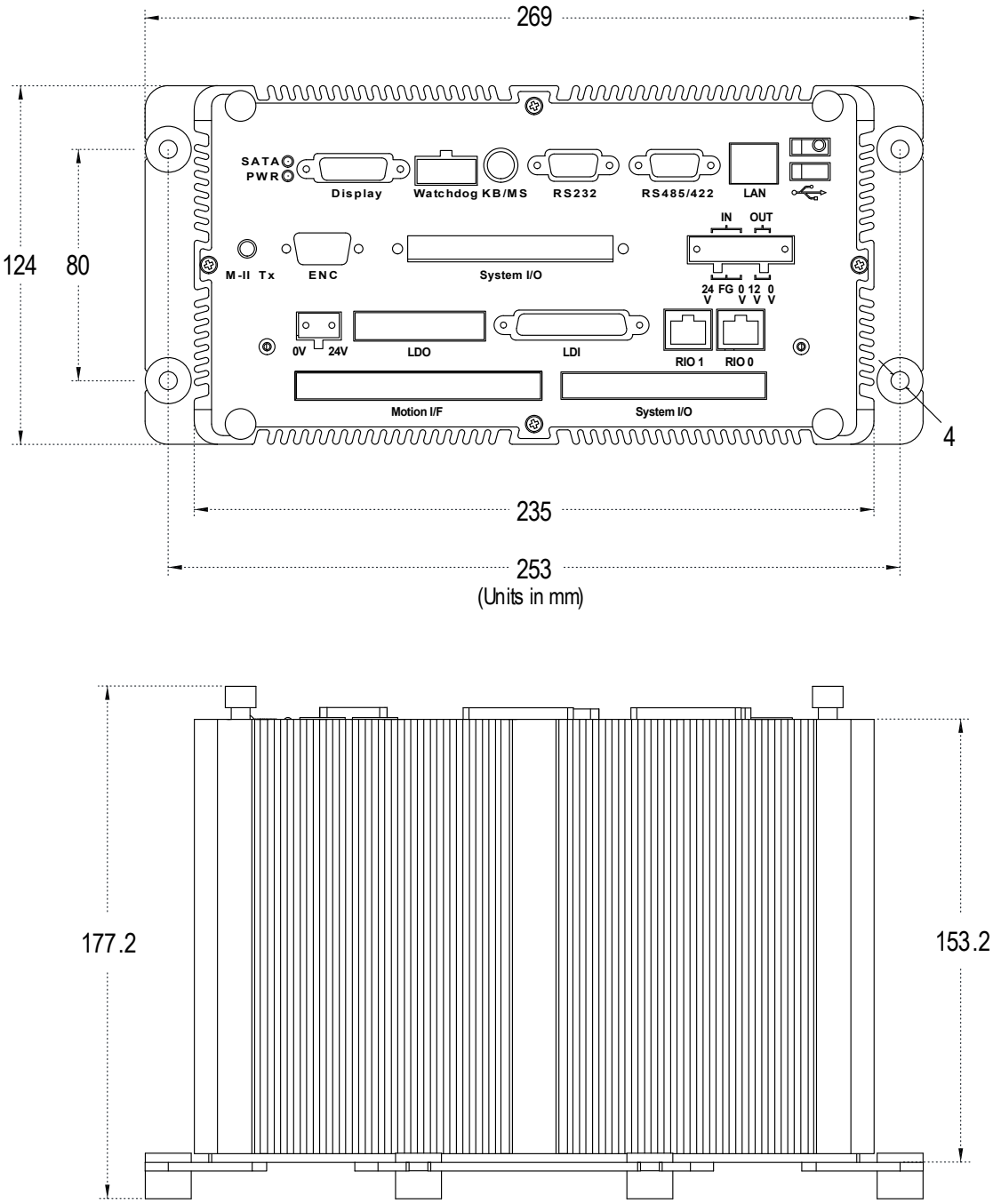


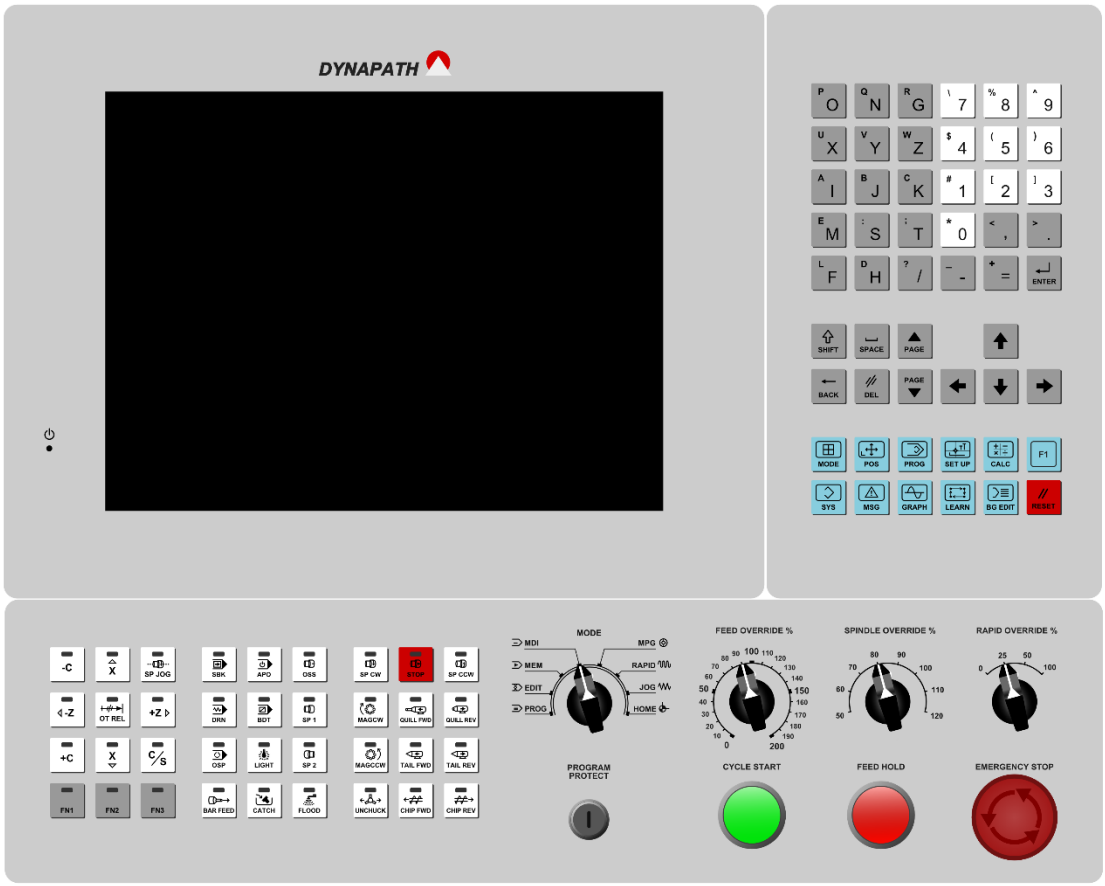
Figure 3.1-2: DynaPath WinDelta Node-1000 Control Chassis

3.1.2 D1 Console

The DynaPath D1 Console is designed for standard turning machines with operations focused on manual/DRO, semi-automatic operation, and conversational programming. This console features the DynaPath Mode Screen operation style.

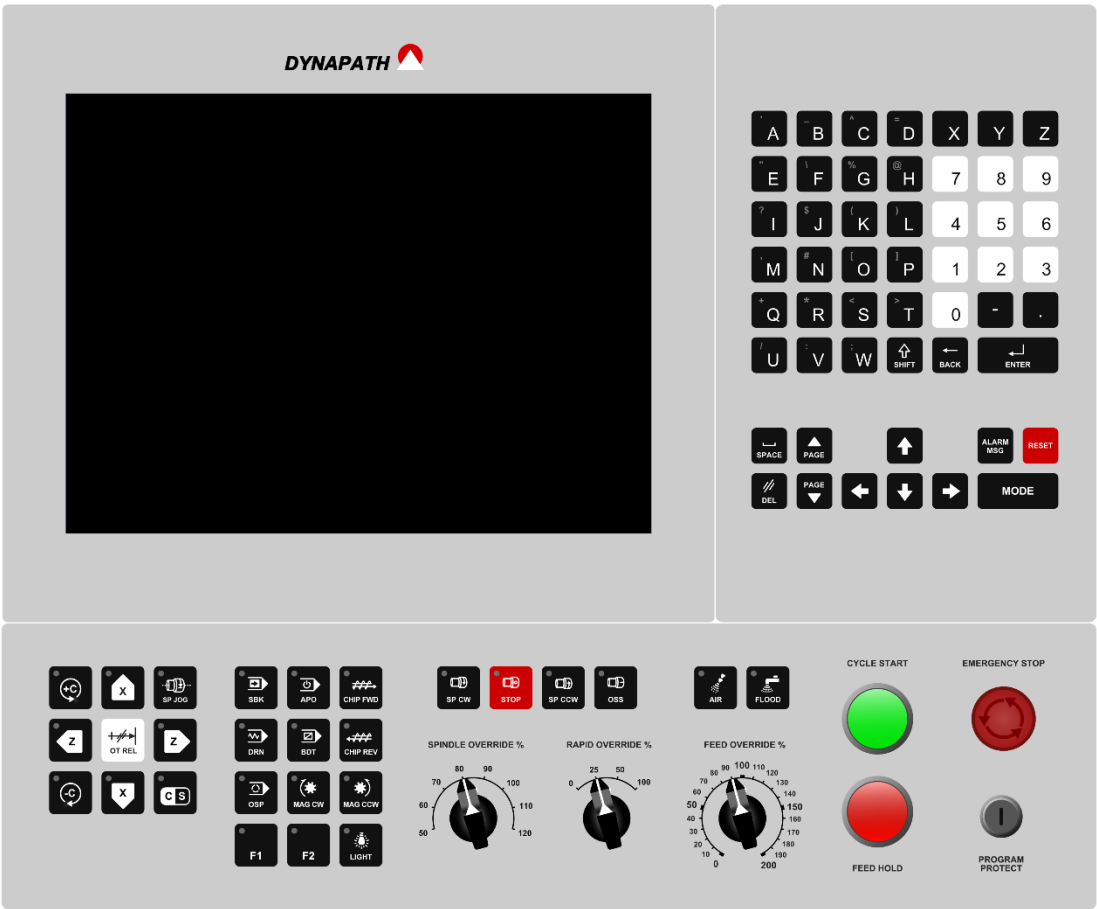
3.1.3 D4 Console

The DynaPath D4 Console is designed for machine tools with automatic tool turrets or turning centers. The 2nd panel for the D4 console has spindle control functions, a program protect switch, coolant functions, and program run control functions. This console features the DynaPath Mode Switch operation style.



3.1.4 D5 Console

The DynaPath D5 Console is designed for machine tools with automatic tool turrets or turning centers. It is suitable for manual turning, conversational programming, and production operations. The 2nd panel for the D5 console has additional spindle control functions, a program protect switch, coolant functions, and program run control functions. This console features the DynaPath Mode Screen operation style.



3.1.5 Handheld MPG

The DynaPath WinDelta has the option for interfacing with an electronic manual pulse generator (MPG) to enable more precise machine movements and other features. The standard DynaPath handheld MPG device shown in Figure 3.1-3 has an axis selector rotary switch (off, X, Y, Z), a multiplier rotary switch (x1, x10, x100), and a digital pulse generator.

Figure 3.1-3: Handheld MPG

3.2 Software Components

DynaPath WinDelta control system contains factory installed software consisting of operating system, motion control, PLC control, user interface/HMI, program macros, and other external applications. All system software included on the control are licensed for use (not sold), therefore usage limitations apply.

3.2.1 Software Licenses and Agreement

Software license agreements are required for use of the control. There are independent licenses for the operating system, its extensions, and for software applications installed on each specific control model. Please adhere to the general software license agreements and guidelines detailed in this section.

Note: **Rights for using the software are offered on the condition that the user agrees to all terms and conditions of the following General License Agreement. Proceeding to operate the control system indicates acceptance of these terms and conditions.**

Note**: Grant of software license and usage rights to included software is applicable under the following General Agreement and Terms:**

- **May only be used on the installed control system,**
- **Not transferrable unless for backup purposes,**
- **Provided as is with all faults and risks tolerated by the user,**
- **Updates can only be installed and verified by authorized personnel,**
- **Must not be reverse engineered, decompiled, or disassembled,**
- **Subject to local export restrictions and technology transfer laws.**

Note: **A license key is included as a formal document or authentic decal located on the control chassis where applicable. License decals are indicated in the following figures.**

The Microsoft Operating System is licensed under the Microsoft End User License Agreement (EULA).

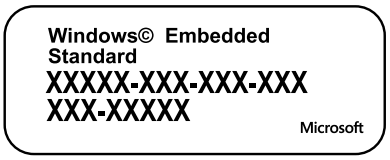


Figure 3.2-1: Microsoft OS License Decal

The IntervalZero license applies towards the Real-Time Extension software used in controls equipped with the Windows XP, 7, or 10 operating systems.

Figure 3.2-2: IntervalZero License Decal

The DynaPath software on the control includes application-level motion control, PLC, and HMI components. There is a single registration key/unlock code pair for each control system labelled on the control chassis and registered within the control via a software interface.

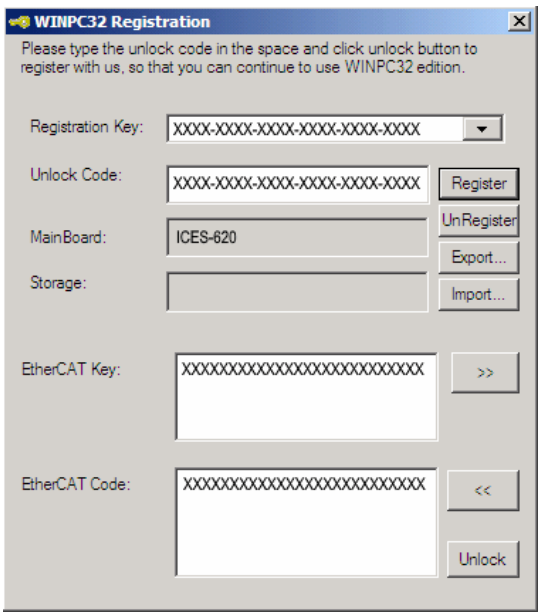


Figure 3.2-3: DynaPath Registration Key and Unlock Code

4 OPERATION

4.2 Introduction to Operation of CNC Machine Tool

4.3 Pre-Startup Inspection

1. Ensure that the machine is properly grounded and that the power source cable is not damaged.
2. Check for sufficient lubrication oil in the oil reservoir.
3. Apply air pressure to the machine if required and check for leaks.
4. Make sure there are no leaks under or around the machine.
5. Check the over mechanical condition of the machine.

4.4 Power-On and Start-Up Procedure

To power-on and start up the machine, adhere to the following general startup procedure:

1. Press the EMERGENCY STOP button if it is not pressed down already.
2. Switch on the main power switch on the electrical cabinet.
3. Wait for the control system to boot up as indicated on the monitor screen.
4. Visually check if each axis on the machine is clear from over-travel, otherwise follow the steps for over-travel recovery in Section 4.7.
5. Release the EMERGENCY STOP button.
6. Press RESET to enable the servo drives.
7. Home the machine if necessary. Go to HOME mode and press CYCLE START.
8. Review and correct any messages or alarms as indicated on the screen.
9. The machine should now be ready for operation.

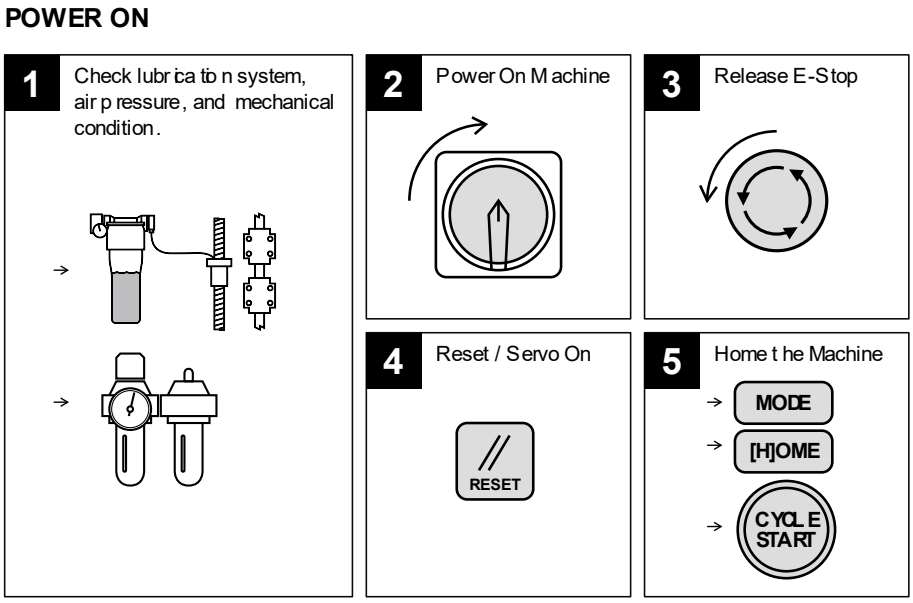


Figure 4.4-1: Power On Guide

4.5 Power-Off and Shutdown Procedure

To shut down the machine, perform the following procedure:

1. Press RESET.
2. Enter JOG mode and PARK the machine or return each axis to a central position.
3. Press to engage the EMERGENCY STOP button.
4. Power off the main power switch.

It is a good practice to also unload any workpieces still on the machine and clean any scrap, chips, or coolant from the machine to prevent rusting, corrosion, and contamination of liquids.

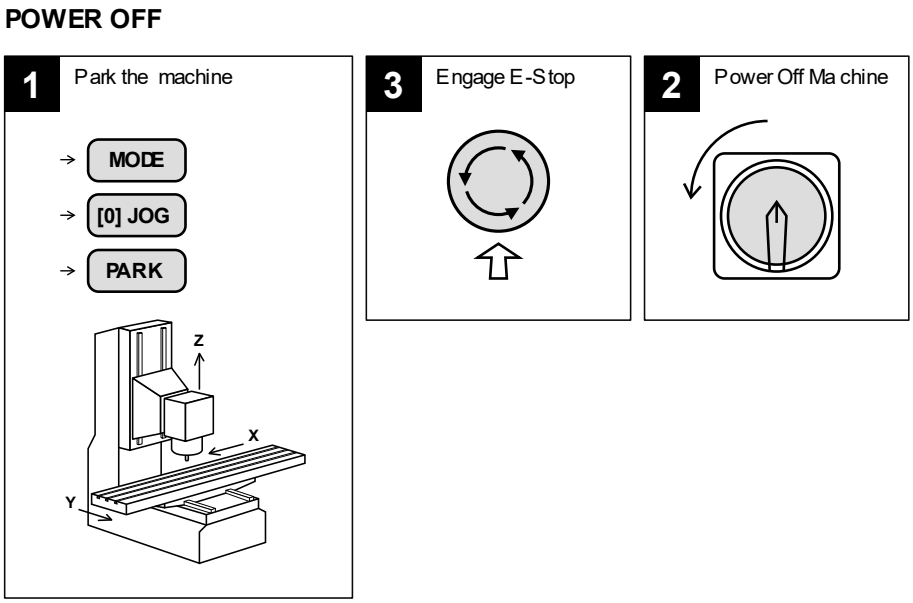


Figure 4.5-1: Power Off Guide

4.6 Emergency Stop

Pressing the EMERGENCY STOP or E-STOP button at any time will halt all mechanical motion. For safety reasons, the E-Stop function on DynaPath controls are electrically implemented as hardware wiring that will function regardless of what state or mode the control is in. Upon pressing E- STOP, the power supplied to each motor is immediately be switched off, and each servo drive will default to base block mode. To recover from E- STOP, release the E- STOP button and press RESET on the panel.

4.7 Over-travel (OT) and Recovery

4.7.1 Hardware Over-travel

In the event of hardware over-travel (OT) where a mechanical travel limit is exceeded on any axis, the over-travel condition must be immediately addressed to protect the machine before continuing any operation. To return from an OT condition, proceed with the following steps:

1. Identify the axis or axes that are in an over-travel condition.
2. Determine the direction of travel necessary to return each axis from the over-travel position.
3. Enter JOG or MPG mode screen.
4. Turn the RAPID OVERRIDE dial switch to its lowest setting.
5. Hold down the OT RELEASE button on the panel and press RESET to servo on.
6. Continue holding the OT RELEASE button and proceed to move each over-traveled axis until each axis is within its travel limits.
7. When complete, release OT RELEASE button.
8. The process is complete when no E-SSTOP or over-travel alarm exists.

4.7.2 Software Over-travel

After the machine has been homed to establish its home position, the control will limit the travel on each axis to prevent physical over-travel. The software over-travel parameter settings provide the positive and negative limits for each axis. Upon encountering a software-overtravel, simply JOG or MPG away from the limit and the warning will clear.

Note: **A program may not be run in AUTO or MDI mode when Software Over-Travel Warning is present. A program will also stop during execution when a Software Over-Travel is encountered. Ensure that the program can execute within the travel envelope to completion prior to starting the program.**

4.8 CNC Modes and Panel Operation

The WinDelta CNC is operated based on control modes. To operate the machine, the operator sets the control mode while the control responds by enabling and disabling specific functions. The operator is thus required to understand the functions associated with each control mode. The current control mode is displayed in the upper left corner of the display screen. Details for each mode are listed as follows:

Table 4.8.1: CNC Modes

JOG/RAPID MODE	Enables the manual operation of the machine by the X, Y, Z, and 4th axis jog buttons. All panel buttons related to machine operation, such as spindle, coolant, or magazine control are also active.
MPG MODE	Enables the manual operation of the machine by the electronic hand-wheel. Like JOG MODE, all panel buttons related to machine operation are active.
INCREMENTAL JOG MODE	Enables the manual operation of the machine by jogging specific distances with the X, Y, Z, and 4th axis jog buttons. Like Jog Mode, all panel buttons related to machine operation are active.
AUTO/MEMORY MODE	Enables the choosing and running of G-Code part programs.
MDI MODE	Enables the running of manually inputted G-Code commands.
EDIT MODE	Enables the creating, editing, saving, and managing of part programs.
DRO OR SEMI-AUTO MODE	Enables a mix of manual and auto operation of the machine by controlling mechanical or electronic hand-wheels or levers and using CYCLE START.

Depending on the console configuration, the DynaPath WinDelta CNC offers two operation styles to switch control modes and screens. For the D1 and D5 consoles, there is a Mode button on the panel which toggles the main mode screen, from which all modes and screens are activated. For the D4 panel, there is a rotary mode switch dial as well as individual screen keys on the panel, thus the control mode and the active screen is changed based on a combination of these two inputs. The following list describes each in more detail:

1. **DynaPath Mode Screen** operation is centered around the Mode button on the panel which displays the main Mode Screen when pressed. From here, specific control modes and screens can be navigated to by pressing the corresponding alphanumeric hotkey on the MDI panel, or by simply selecting it on the touchscreen. With this type of operation, modes and screens are coupled.

2. **DynaPath Mode Switch** operation changes mode based on the mode dial input on the panel, and changes screens independently based on which screen key is pressed on the panel. Depending on the selected mode, a mode switch may be accompanied by a screen switch. Because the mode and the screen are not coupled, the operator may view other screens while remaining in a set mode.

4.8.1 Mode Screen Style Operation

In this style of CNC operation, to access the home or main mode screen, press the MODE button on the panel. Figure 4.8-1 is the home screen. Press an alphanumeric key or touch the screen button to enter each screen.

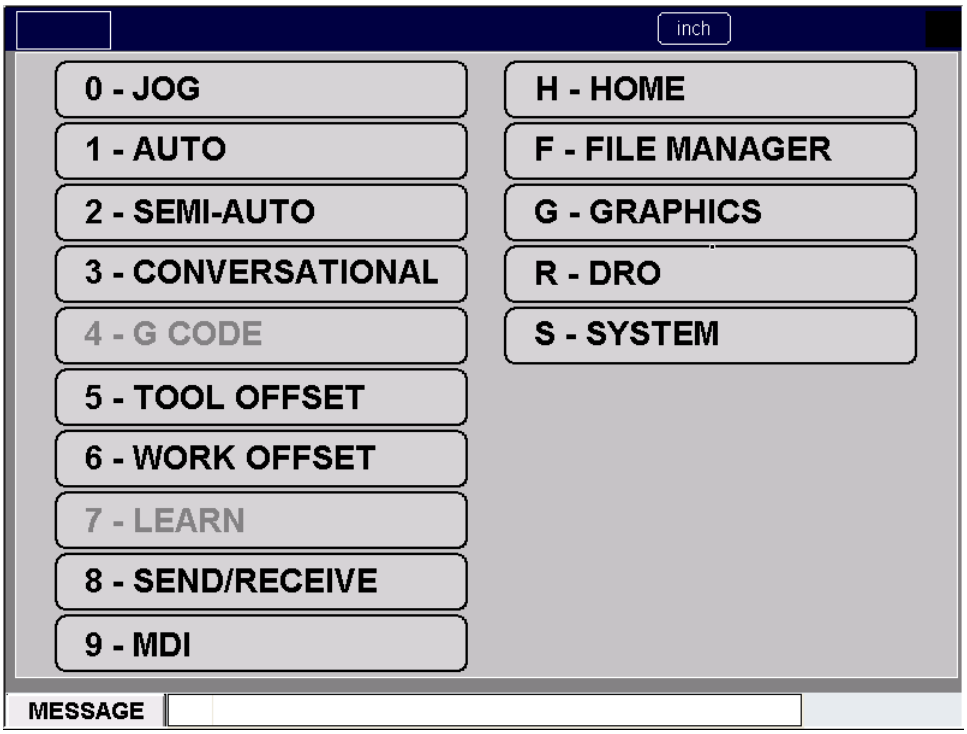


Figure 4.8-1: DynaPath Home Screen

Each function on the home screen is described in the following table.

Table 4.8.2: DynaPath Screens

JOG	Engages JOG mode and provides functions to operate the machine by manually operating the jog buttons, the MPG, and the panel buttons. Other manual operation modes such as MPG and INCREMENTAL JOG modes can be toggled.
AUTO	Runs programs in MEMORY mode. The program can be run in single block, MPG run, or retraced to begin from a specified block in the program.
SEMI-AUTO	Engages the MPG mode and provides functions to mix manual operation with simple automatic actions.
CONVERSATIONAL	Engages EDIT mode and opens the active conversational program for editing.
G-CODE	Engages EDIT mode and opens the currently active G-Code program for editing.
TOOL SETUP	Shows the tool offset and ATC tables for tool management. Tool length measurement and tool changes can also be performed.
WORK OFFSET	Shows the work offset table and external offsets for management. Also offers align functions for measuring workpiece locations.
SEND/RECEIVE	Engages EDIT mode and provides functions to transfer programs to and from external USB devices or the network.
MDI	Engages MDI mode for quick execution of G-Code commands.
HOME	Engages HOME mode to home the machine which establishes the machine coordinates.
FILE MANAGER	Engages EDIT mode and shows a listing of programs on the control for opening.
GRAPHICS	Shows the simulation of the currently active part program.
DRO	Engages the MPG mode for manual operation of the machine.
SYSTEM	Allows configuration, troubleshooting, and diagnostics of the control system.

4.8.2 Mode Switch Style Operation

For control configurations that have a mode switch dial on the panel, an operation flow chart for each mode, screen, and function is presented in Figure 4.8-2.

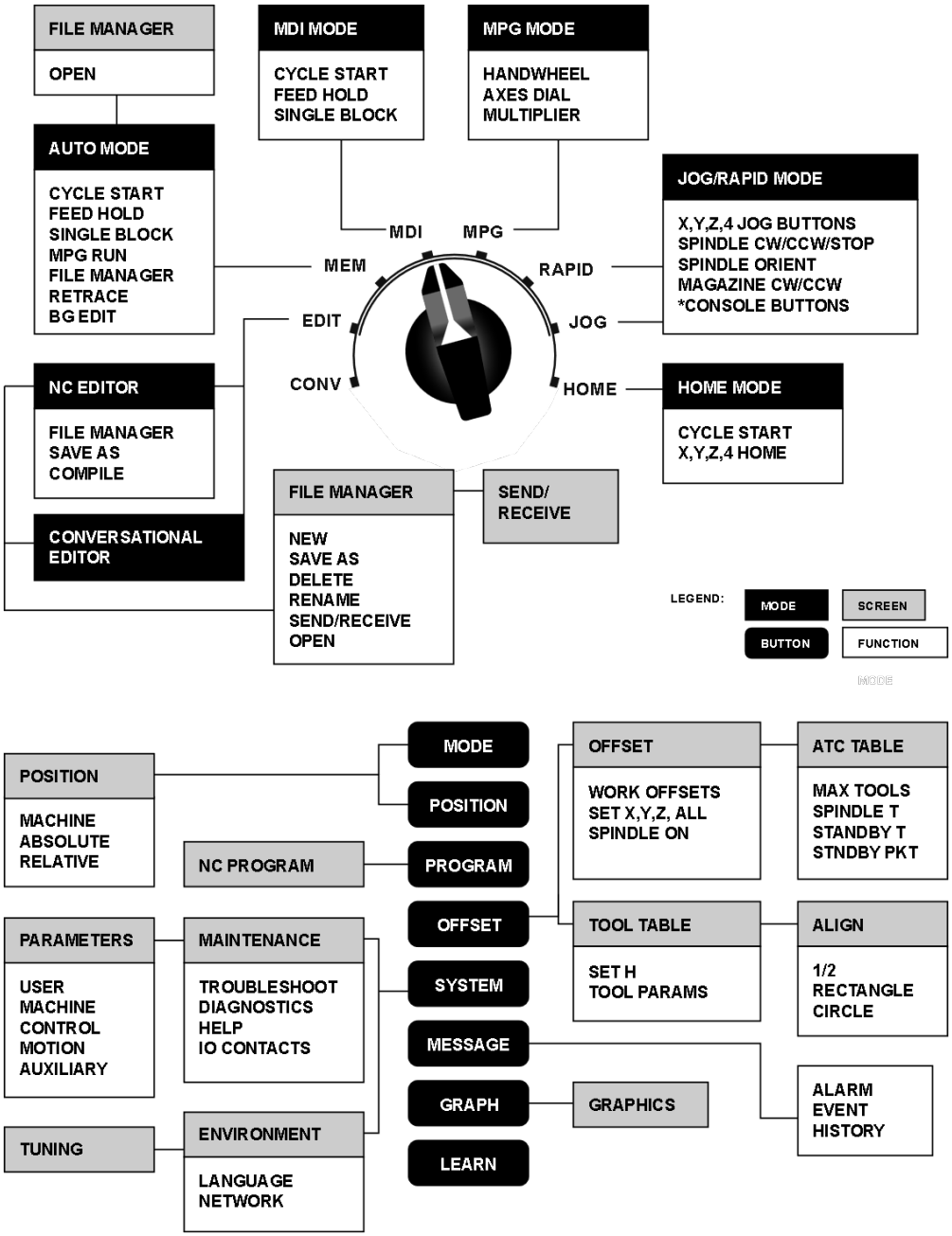


Figure 4.8-2: DynaPath Mode Switch Operation

4.9 Coordinate Systems

Understanding the various Coordinate Systems (CS) on the control system is important for proper operation of the machine. Table 4.9.1 describes the Relative, Absolute, and Machine coordinate systems.

Table 4.9.1: Coordinate Systems

MACHINE (MAC)	The Machine CS is the main coordinate system maintained by the control and is the basis for all other CS. The Machine CS is initialized after homing the machine by allowing each axis to physically contact a home dog sensor thus establishing its home position. In certain machines equipped with absolute encoders, homing is not necessary as its axes home positions are permanently stored in the drives.
RELATIVE (REL)	Relative CS is generally used for manual operation allowing the operator to zero it as reference for determining relative distances.
ABSOLUTE (ABS)	Absolute CS is the coordinate system in which part programs refer to while executing in AUTO or MDI mode. Its origin or part zero depends on which Work Offset is currently active. An External Offset is an additional offset that is always applied to the active Work Offset

***Note*: The X-axis coordinate system is expressed in diameter units by default. Therefore actual distances of movement on the X-axis is half of what is displayed on screen.**

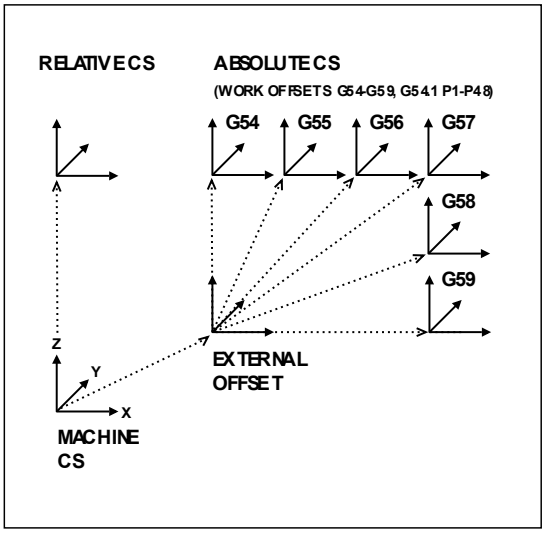


Figure 4.9-1: Coordinate Systems Illustrated

4.10 Work Offset Concepts

Programs on the control are run with reference to the Absolute Coordinate system (ABS). The program origin or part zero location must be defined by setting up work offsets on the control. This establishes where the control system executes the part program within the machine envelope.

There are six primary work offsets available for use: G54 to G59. In a part program, each work offset is activated by commanding its G-code: G54 to G59. When activated, program movement will be executed relative to the active work offset. Note that after every RESET on the control, the active work offset will default to G54.

Note: **When setting up a work offset, the machine coordinates of each axis location is stored into the work offset table under its name.**

Note: **Absolute ABS position = Machine MAC position – External offset – Active Work offset.**

Note: **The External Offset is always active in addition to the current work offset.**

Note**: Reference the Work Offset section for details regarding the setup of part zero location and work offsets.**

4.11 Tool Offset Concepts

The control system can account for the different tool dimensions and orientations automatically by referencing the tool offset table and applying it to the ABS coordinate system during program operation. A tool change and tool offset is activated by commanding a T-Code followed by a four digit number (e.g. T0101), the first two digits represent the tool station number, and the latter two digits represent the offset number corresponding to a row in the tool offset table. Each row in the tool offset table stores the following data:

1. **Tool X and Z Offset –** tool tip location offset.
2. **Tool X and Z Wear Offset –** offset in X and Z directions due to wearing of tool.
3. **Tool Radius (R) –** radius of the tool tip used for tool radius left (G41) or tool radius right (G42) compensation**.**
4. **Tool Orientation (T) –** tip direction of the tool used for tool radius direction.

Note**: Reference the Tool Offset section for details regarding the setup of tool offsets.**

5 HOME MODE

5.1 Home Mode Screen

The HOME screen allows the control to establish the machine zero reference location for each axis.

Caution: Prior to homing the machine, check that the tailstock, quill, tool changer, tools, or any fixtures are not obstructing the axes movement towards the home location.

Note: In the case of axes equipped with absolute encoders, homing of the axes is optional. Pressing the jog button for these axes simply moves the axis to its defined machine zero position.

Note: For machines equipped with C-Axis functionality, the C-Axis must be homed. To home the C-Axis, set the spindle to C-Axis mode by pressing the C/S panel button, then press the C+ or C- panel button.

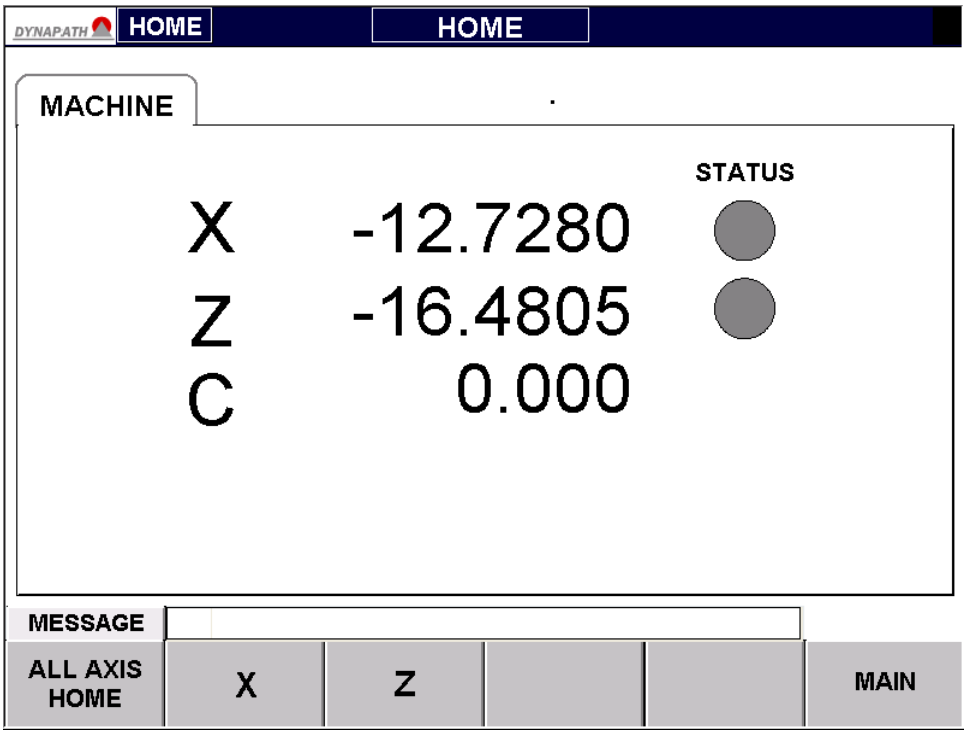


Figure 5.1-1: Homing Screen

6 Manual Modes

6.1 Jog Mode

Upon entering the JOG screen, the JOG mode is activated as displayed on the mode field on the upper left. While in JOG mode, the hard keys for each axis direction can be used to move the machine axes. The JOG speed can be changed by selecting the desired speed button along the right of the screen. The RAPID OVERRIDE dial also takes effect when in JOG mode.

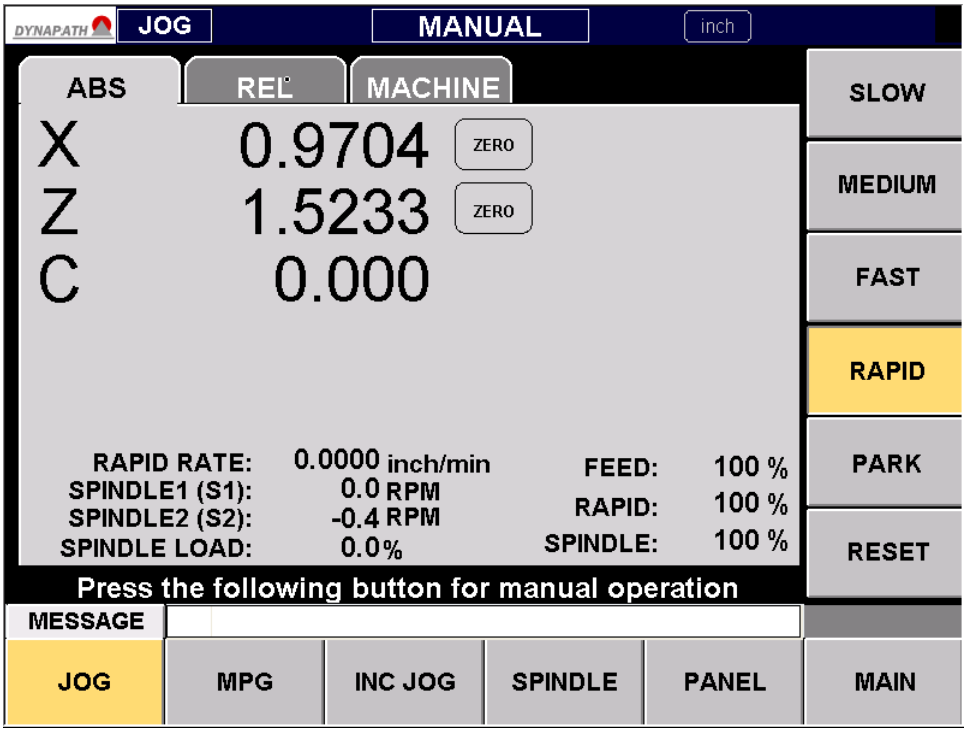


Figure 6.1-1: Jog Screen

6.2 MPG Mode

Press the MPG button along the bottom of the screen to access the MPG mode. When MPG mode is activated, the physical electronic manual pulse generator handwheels for X and Z axes can be used to move each axis. In MPG mode, the RAPID OVERRIDE dial applies a x1, x10, or x100 multiplier to affect the distance moved on each click of the MPG handwheel.

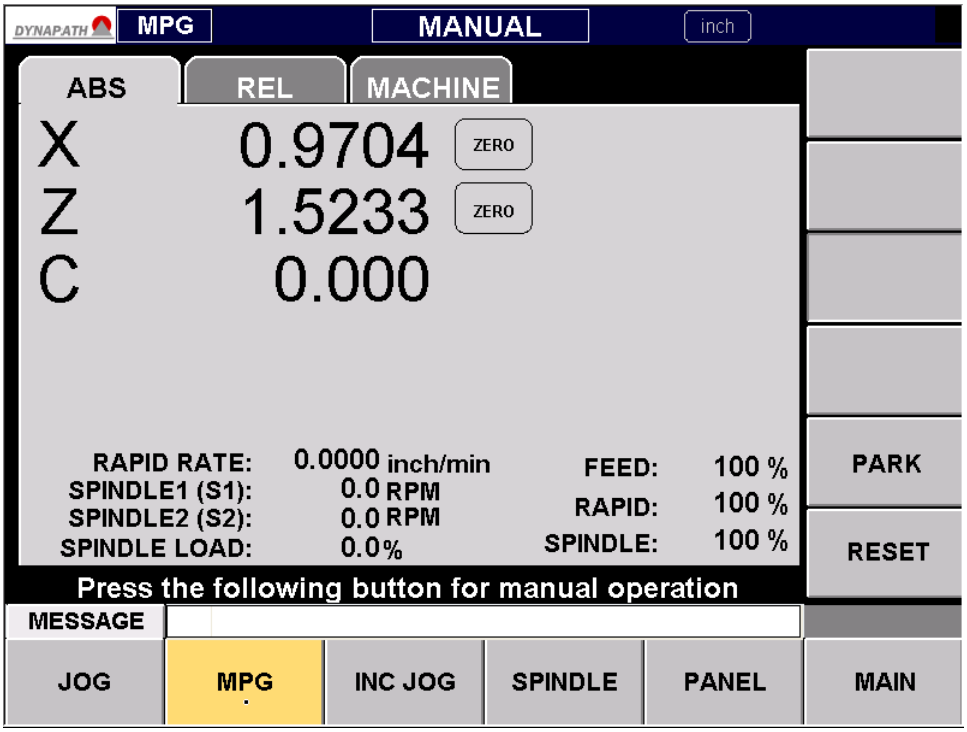


Figure 6.2-1: MPG Screen

6.3 Incremental Jog Mode

Press the INC JOG button along the bottom of the screen to switch to INC JOG mode. While in INC JOG mode, every press of the axes jog buttons will move each corresponding axis a set amount depending on the distance button selected along the right of the screen.

6.4 Spindle Control

To control the spindle, access the SPINDLE screen button along the bottom of the screen. The spindle RPM command can be modified by clicking on the black RPM field and entering the desired RPM. Then select CW or CCW on-screen buttons along the right of the screen to rotate the spindle. The SP CW or SP CCW hard keys on the panel also rotate the spindle. Press the STOP on-screen key or SP STOP hard key to stop the spindle from rotating. The SPINDLE OVERRIDE dial will affect the final RPM of the spindle.

6.4.1 C-Axis

If the machine is equipped with spindle as C-Axis functionality, then the spindle can be switched to C-Axis by pressing the [C/S] hard key while in any of the manual modes, or by executing a M45 command in MDI to enable the C-Axis mode. The LED light on the [C/S] hard key will lit up when C-Axis mode is enabled. In this mode, the C+ and C- jog keys can rotate the C-Axis, or execute a G-Code command such as C90.0 to move the C-Axis to 90 degrees.

6.5 Panel Functions

Panel functions may be accessed by the PANEL on-screen button on the bottom of the JOG screen, or by the PANEL hard-key button. Each button is typically associated with a M-Code or T-Code function, depending on whether the machine is equipped with the option. Simply press on the screen button to toggle the function.

7 Auto Mode

7.1 Basic Operation

The AUTO or MEM mode executes part programs from the control memory. Upon entering this mode, the most recently opened program is loaded and ready for machining. The upper left corner of the screen displays the ABSOLUTE, RELATIVE, MACHINE, and DISTANCE TO GO coordinate positions. The upper right corner of the screen shows the currently active modal G-Code for each group and the current feeds and speeds. Below is the currently active machining program and the current line being executed.

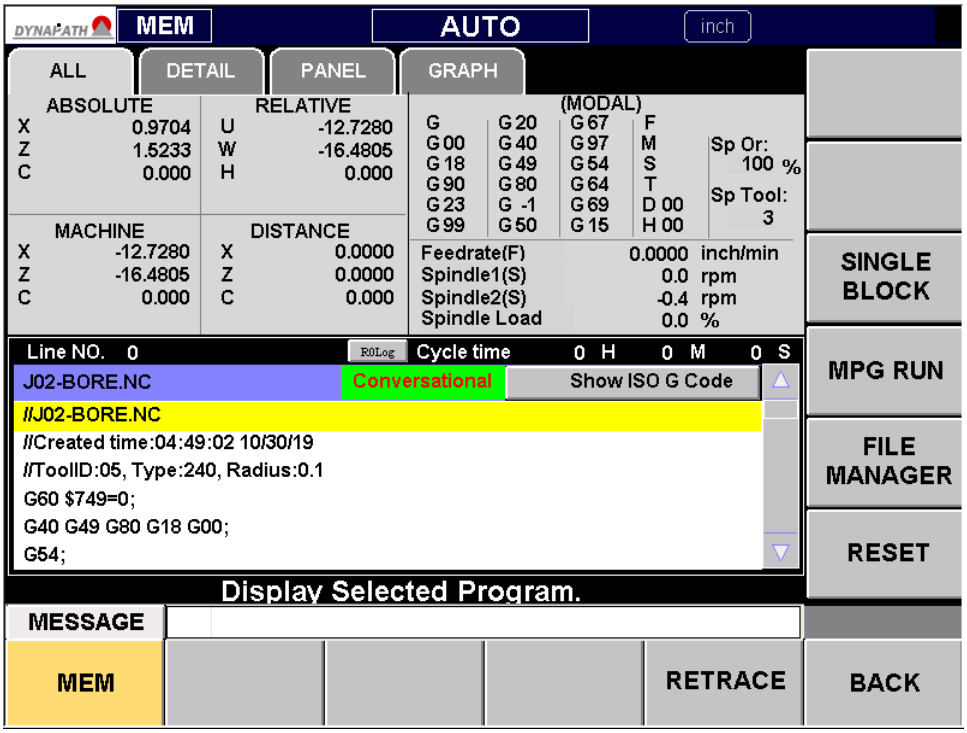


Figure 7.1-1: Auto / MEM Screen

7.1.1 Cycle Start

Press CYCLE START push button on the panel to begin execution of the active part program.

7.1.2 Feed Hold

Press the FEED HOLD push button on the panel to halt execution of the program.

7.1.3 Spindle, Rapid, Feed Overrides

The SPINDLE, FEED, and RAPID OVERRIDE dials will take effect on the spindle RPM, feed rate, and rapid movement speeds of the program.

7.2 File Manager

The File Manager screen presents a list of all the part programs in storage on the control. To activate a file, select it from the list and press the OPEN file on-screen button.

7.3 Single Block

The SINGLE BLOCK function when enabled, allows execution of one line of G-code at a time for every press of the CYCLE START button. This function can be toggled at any time during program execution.

7.4 MPG Run

MPG RUN function enables the program to be run with feed controlled by CW rotation of the handheld MPG. This function can be toggled at anytime during program execution.

7.5 Retrace

RETRACE function begins the program at any specified line. The start line number defaults to the previous termination of the program whether by RESET, E-STOP or other abnormal abort of the program while it was running.

7.6 Reset

RESET will immediately stop program execution, move the active line to the beginning of the program, and restore all the modal G-Codes back to a default state.

***Note*: RESET will restore all G-Code modals back to a default state, for example, the current work offset will be reset to G54.**

7.7 Panel

The PANEL tab offers additional soft keys for toggling of auxiliary functions.

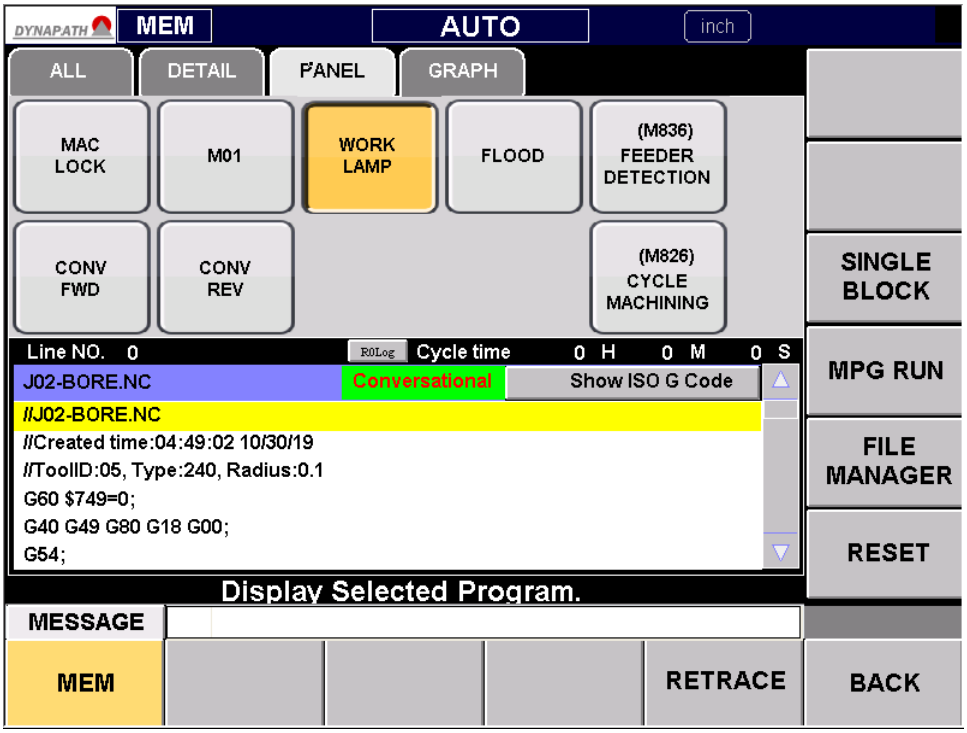


Figure 7.7-1: Auto Panel Screen

8 WORK OFFSET

8.1 Offset Concepts

The OFFSET screens provide functions to set up the workpiece and tool offsets necessary for running part programs to produce exact machining results. Important background knowledge and concepts are as follows:

- **Machine Coordinate System** is the coordinate system established by the control during execution of the HOME procedure. It is the coordinate system upon which all other coordinate systems are based upon.
- **Absolute Coordinate System** or ABS is the reference coordinate system in which every part program is executed with reference to. The control can store multiple ABS coordinate systems which are named G54-G59 and are called by name in each program.
- By default, the G54 ABS coordinate system is activated on Power On and when RESET is pressed.
- **Work Offsets** tell the control system where to run a program within the work envelope of the machine.
- **Tool Offsets** tell the control system how to compensate for the dimensional differences between tools so that each tool travels precisely on the programmed tool path.

8.2 Work Offset

The WORK OFFSET screen enables set up of workpiece or fixture offsets. These offsets are the zero positions for the ABS or ABSOLUTE coordinate system in which part programs are executed with reference to. The origin of the ABS coordinate system is commonly referred to as ABS zero or part zero. Its location is specified in Machine Coordinates.

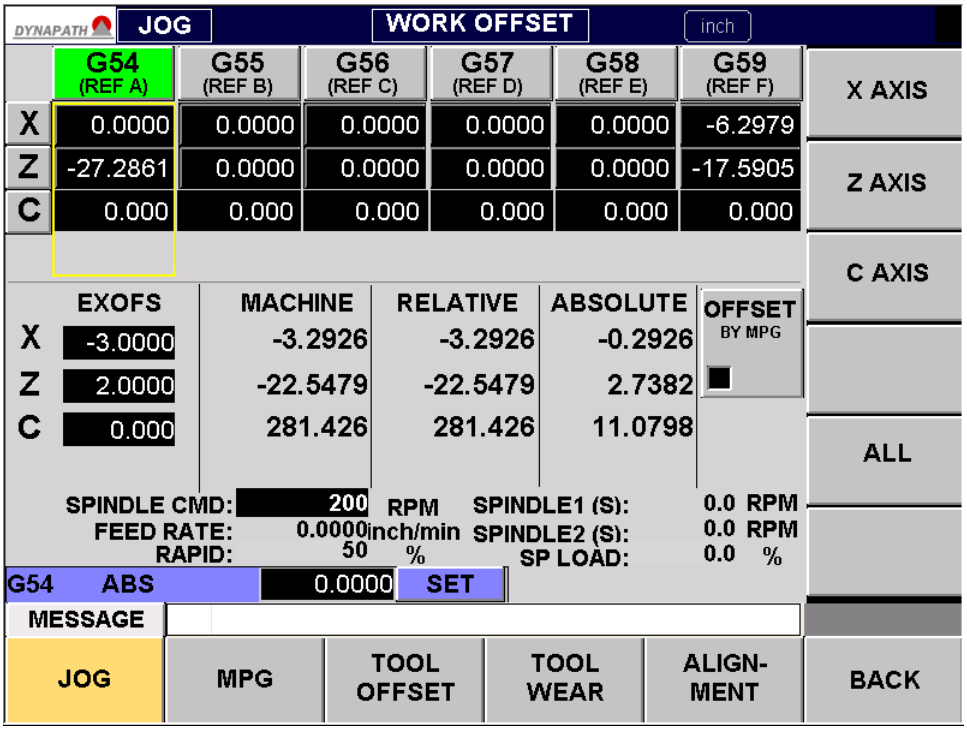


Figure 8.2-1: Work Offset Screen

Follow these steps to set up a work offset:

1. Select the active work offset to modify (G54 – G59).
2. Change tool to a specified reference tool and toggle between JOG or MPG mode to move the axes to where the tool tip is in contact with the workpiece or fixture.
3. Set the ABS part zero location using one of the following methods:
   (*Note*: Generally, the center of the spindle is also the X-axis part zero location)
   a. Directly input axis location values in terms of machine coordinates by clicking on each input field under the work offset number,
   b. Press the X, Z, or C-Axis buttons along the right of the screen to set each axis' part zero to the current location,
   c. Select the work offset number, then select an axis, and input the measured or target ABS value in the ABS SET field near the lower part of the screen.
4. Apply EXTERNAL OFFSET or EXOFS to shift the active ABS zero position. These are always active during program execution.

9 TOOL SET UP

9.1 Tool Offset

The Tool Offset table is where tool datum is stored and maintained. Tool datum is used to compensate for tool lengths, orientation, and dimensions when activated by specific G-Code commands within programs. The tool change command “T AA BB“, contains two sets of two-digit numbers. The first two digits AA accepts values 00-99 and specifies the desired tool number or station in the tool turret to change to. The second two-digit number BB also accepts values 00-99 corresponding to the tool offset number for X and Z compensation values and activates these offset values upon tool change. The Tool Radius R and Tip Direction T values apply the tool radius and tool orientation when G41 Left or G42 Right compensation is called. The control system thus accounts for the different tool positions, geometry, and orientations for each tool automatically by referencing the stored datum within the Tool Offset table.

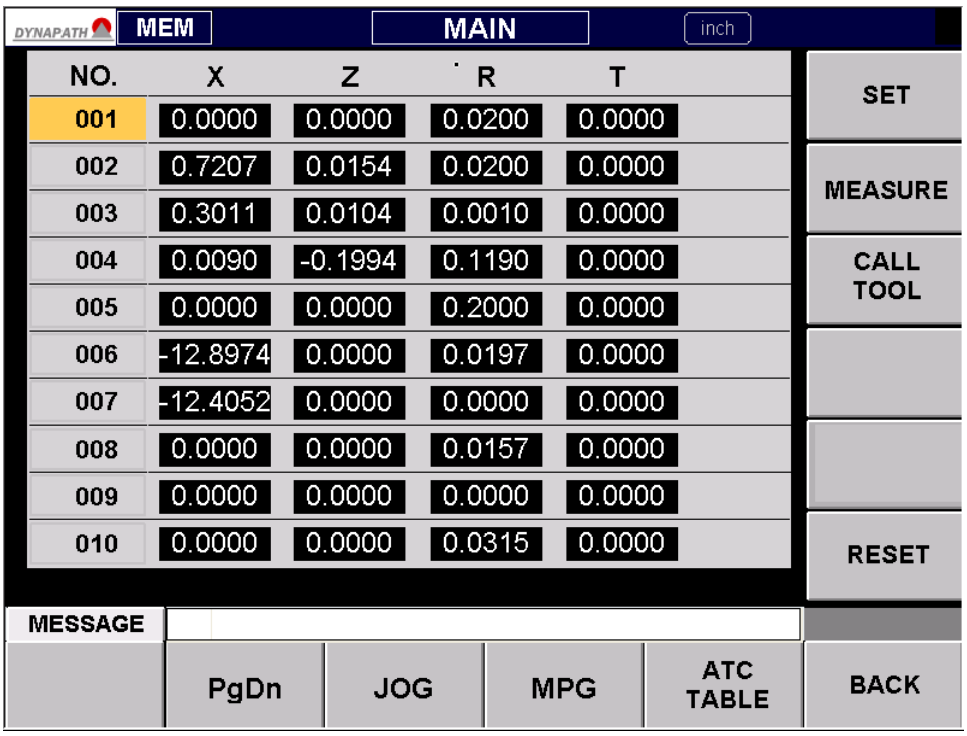


Figure 9.1-1: Tool Offset Screen

Definitions for each tool offset data are as follows:

1. **Tool X and Z Offset –** Specifies the distance to offset the tool in the X and Z axes when traveling along a programmed tool path in order for the tool tip to maintain proper contact with the workpiece.
2. **Tool X and Z Wear Offset –** Additional offset in X and Z directions applied to the X and Z offsets due to wearing of tool. When there is no wear on the tool, set these values to zero.
3. **Tool Radius (R) –** radius of the tool tip used for tool radius left (G41) or tool radius right (G42) compensation commands**.**

4. **Tool Orientation (T) –** tip direction of the tool used for tool radius direction. Reference the tool orientation section for set values and definitions.

The following list provides methods for setting up work and tool offsets, with advantages, disadvantages, and conditions for each.

Tool Setup Methods:

1. **Work Offset not set, Tool Offsets set relative to Machine Zero:**
    - The active Work Offset contains all zeros.
    - Part Zero location is also the Machine Zero position.
    - Each tool stores offsets measured from Machine Zero.
    - For a new workpiece of different size, every tool offset must be re-calibrated, or the difference in Part Zero location must be stored in the external offset.
    - This method is simple to setup, and suitable for programs that use few tools.

2. **Work Offset set with reference tool, Tool Offsets set relative to a Reference Tool:**
    - This method requires a reference tool that does not change or wear over time. The accuracy of this method relies on the reference tool remaining constant. Oftentimes this is a tool that is used only to square the part, or not used as part of the machining process.
    - The designated reference tool is used to set up the Part Zero location stored in the active Work Offset.
    - The reference tool is also the basis from which all other tools' X and Z-offsets are measured with reference to.
    - This allows a tool-setter or some gage to be used on or away from the machine to set up tool offsets.
    - This method keeps Work Offset separate from Tool Offsets. When a new part with a different size or Part Zero is set up, only the reference tool is needed to set up the Work Offset. When any tool is replaced, only the tool offsets for that tool needs to be re-measured by measuring its X and Z offsets relative to the reference tool.

3. **Work Offset Z set with reference tool, Tool Offsets Z set relative to a Reference Tool, and Tool Offsets X set relative to Machine Zero:**
    - This method is the most efficient way to set up work and tool offsets.
    - Work offset for X is fixed to the center of the spindle for most parts, thus X offsets can be set using machine coordinates for each tool in the tool offset table.
    - Work offset Z only needs to be measured once using the reference tool, while work offset X can be set to zero.
    - Tool offsets in Z for every other tool are measured relative to the reference tool. A tool setter or edge finder can be used to measure the relative difference between each tool against the reference tool.

- Preserves all the advantages in Method 2, such as the separation of Work and Tool Offsets, while further simplifying by only requiring measurement for Work Offset in Z.

9.1.1 Measure

The MEASURE function window allows tool offsets to be measured directly on the machine. This section details procedures on how to use this function to perform each tool setup method.

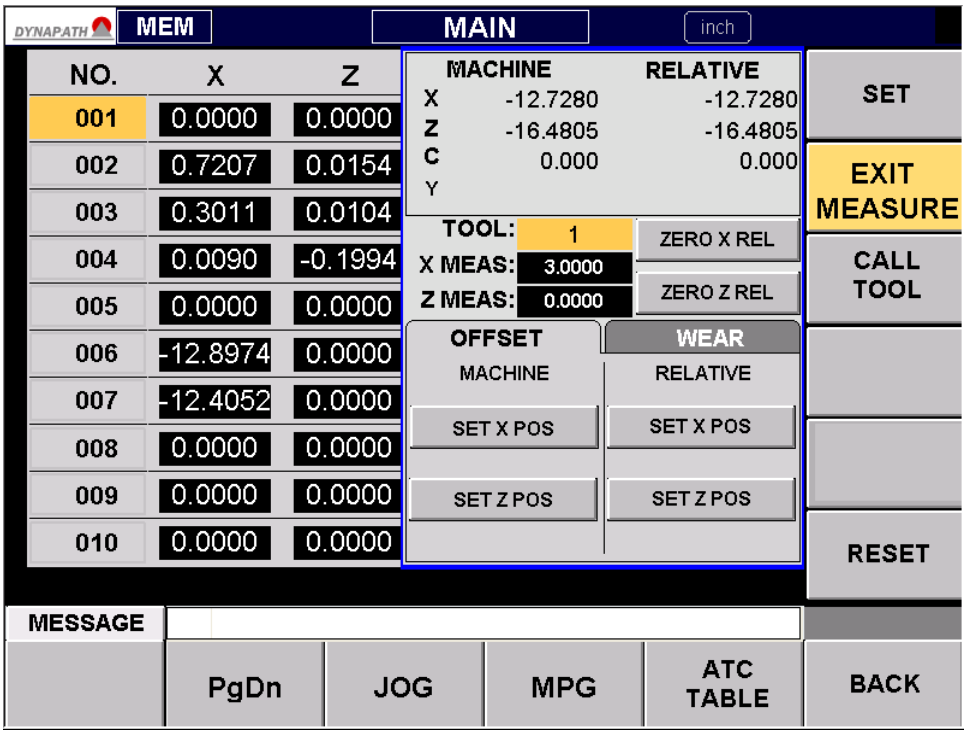


Figure 9.1-2: Tool Offset Measure Screen

Tool Setup Method #1: Measure each tool with reference to machine zero. This method places the full offset amount into the tool offsets for each tool, while the work offsets for X and Z should be set to zero.

1. Select the current tool number.
2. Touch the tool tip to an X-axis reference position, typically this is a point on the cylindrical surface of the stock,
    a. Input the measured diameter of the reference surface into X MEAS,
    b. Press MACHINE / SET X POS.
3. Touch the tool tip to a Z-axis reference position,
    a. Input the reference position value into Z MEAS,
    b. Press MACHINE / SET Z POS.
4. Repeat from step 1 for each tool.

Tool Setup Method #2: Measure against a reference tool. Assuming a reference tool was previously used to measure the work offset, the procedure for setting up every other tool relative to the reference tool is:

1. Touch the reference tool to a Z-surface and press ZERO Z REL.
   Touch the reference tool to a X-surface and press ZERO X REL.
2. Input the diameter of the X-surface into X MEAS. If a Z-surface is offset from the actual workpiece, then input its relative offset into Z MEAS.
3. Highlight the target tool and physically change to the tool.
4. JOG or MPG the tool tip to the previously defined reference position in X and Z-axes.
   a. When touching X-surface, press RELATIVE/SET X POS
   b. When touching Z-surface, press RELATIVE/SET X POS
5. Return to step 3 and proceed for each tool.

9.1.2 Tool Orientation

The T-column in the Tool Offset table defines the tool orientation of each tool. Together with the radius (R) offset of each tool, these allow the control to compensate the tool tip radius when using tool radius left compensation (G41) or tool radius right compensation (G42) in a G-Code program.

Depending on the hardware configuration of the machine, the tool post or tool turret may be placed in front or behind the spindle centerline when viewing along the X-axis direction. Generally, if the tool post is in front of the workpiece near the operator, this is a “front” tool post configuration. If the tool post is behind the workpiece away from the operator, this is a “back” tool post configuration. Knowing the type of tool post configuration is important in order to set the tool orientation (T) number correctly.

The tool orientation numbers for front and back tool post configurations are detailed in Figure 9.1-3 and Figure 9.1-4 respectively.

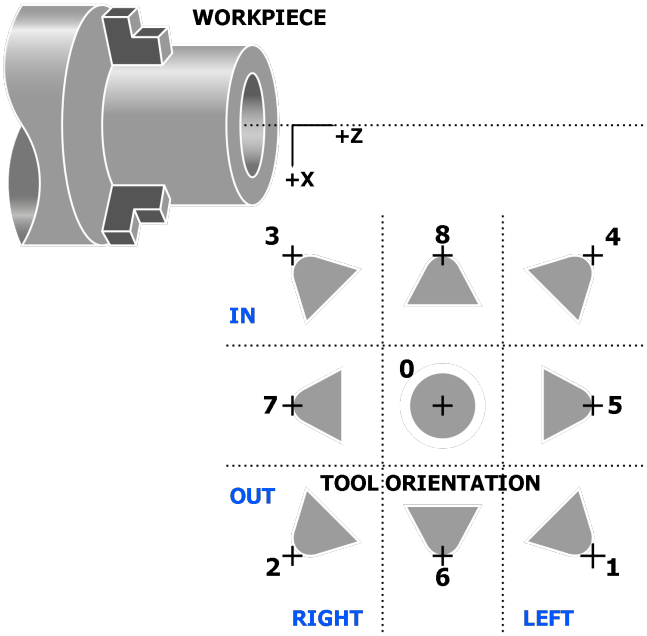


Figure 9.1-3: Tool Orientation Numbers for Front Tool Post

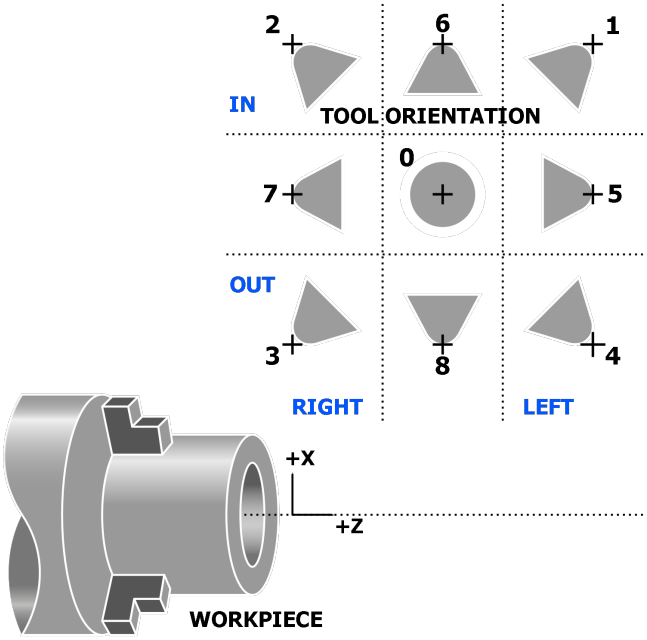


Figure 9.1-4: Tool Orientation Numbers for Back Tool Post

9.1.3 Call Tool Function

A tool change is executed upon highlighting a target tool number and then pressing the CALL TOOL on-screen button.

9.2 Tool Wear

The TOOL WEAR table is similar to the TOOL OFFSET, however it stores corrective offsets for each tool related to the wearing of the tools after it has been machined with. The final actual tool offset compensation values for each tool are the sum of its values in the TOOL OFFSET table and the TOOL WEAR table.

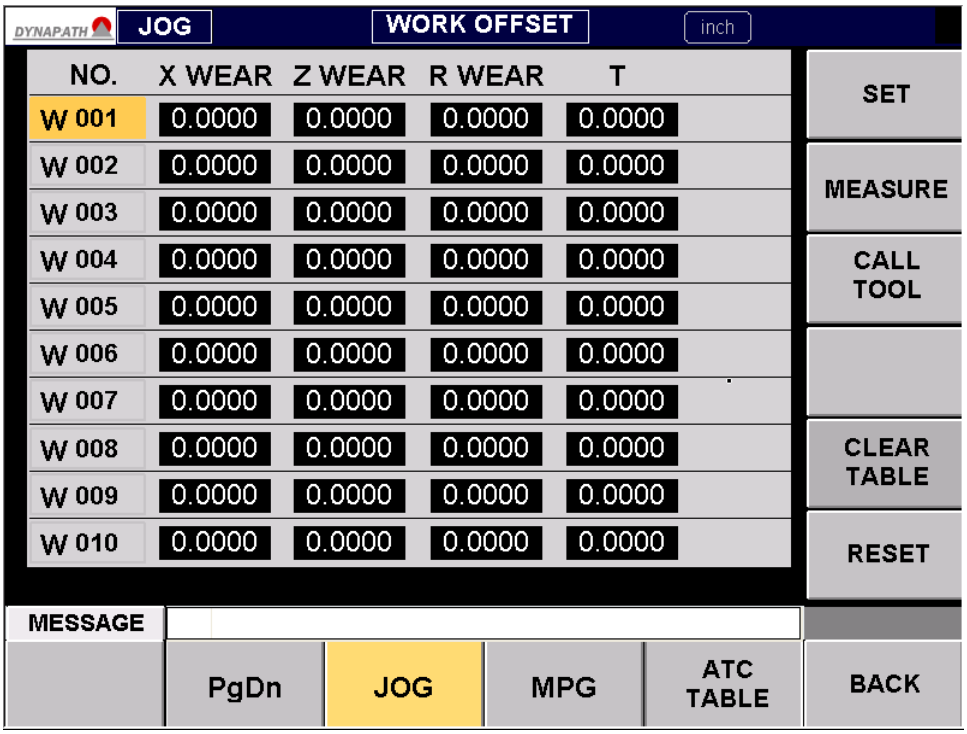


Figure 9.2-1: Tool Wear Screen

10 DRO / SEMI-AUTO

10.1 Introduction

The DynaPath Windelta SEMI-AUTO Mode is an operation mode in which Manual MPG operations can be used in conjunction with Automatic Feed or Automatic Sequences. This mode allows an operation to quickly perform line, arc, or thread operations.

10.2 Manual Operation

Upon entering the DRO or SEMI-AUTO mode screen, the MANUAL function button is highlighted, indicating that MPG mode is active, thus allowing movement of the axes by turning the MPG handwheels.

A feed rate and spindle RPM command can be specified for each semi-auto function. Use the SP CW, SP CCW, and SP STOP hard keys to start and stop the spindle. Additionally, a tool change can be performed by specifying the desired SP TOOL and then pressing the T-C on-screen button.

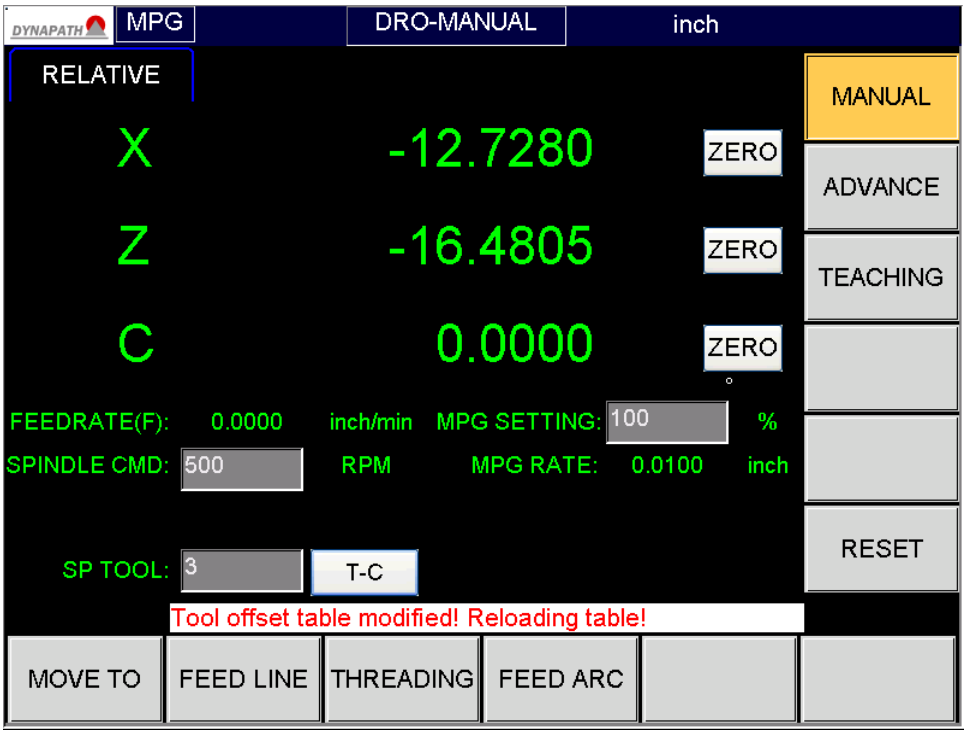


Figure 10.2-1: DRO Manual Screen

10.3 Move To

The MOVE TO function performs a quick, precise move to any X or Z position. Input the target X and Z ABS coordinates to move to, then press CYCLE START to engage.

10.4 Feed Line

The FEED LINE screen executes an interpolated line in the X, Z, and C axes or a combination thereof. The specified values are incremental distances relative to the current position of each axis and accept positive or negative values. A second position (POS2) can be enabled for the feed line to return to a relative location.

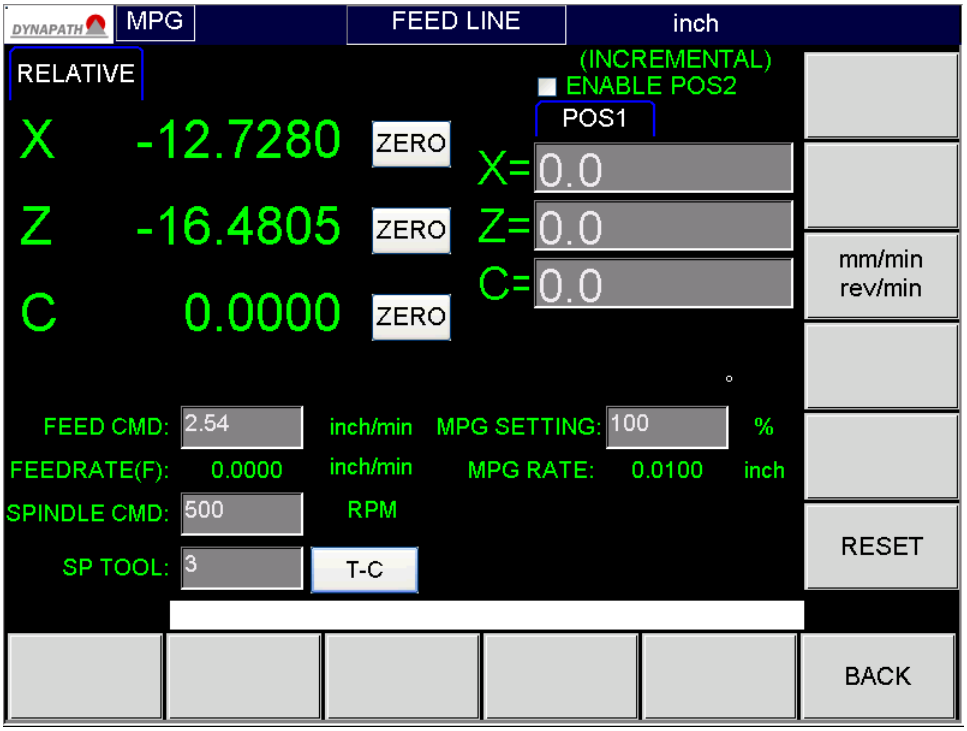


Figure 10.4-1: Semi-Auto Feed Line Screen

10.5 Threading

The THREADING semi-automatic function performs a threading cycle based on the specified inputs. The start position of the cycle is also the current position of the machine. CYCLE START will begin the threading cycle until completion.

- **DEPTH** is the final X-axis position of the thread relative to the start position.
- **LENGTH** is the final Z-axis position of the thread, relative to the start position.
- **LEAD** is the pitch of the thread, or the length of one turn of the thread in the Z-axis.
- **PASSES** specify how many threading passes to perform. The depth of each cut is equivalent to the DEPTH divided by number of PASSES.
- **RETRACT** is the X-axis offset relative to the start position to retract the tool while returning to the start position for the next pass.

Note: To repair or repeat threads, HOME the C-axis prior to teaching the start position. This ensures the tool will start in the thread. To HOME the C-Axis, engage the HOME mode, toggle the C/S button to enable C-Axis, then press C+ or – Jog button to HOME.

Note: After C-Axis is homed, use the X and Z-axis ZERO functions to mark the start of an existing thread, then move Z-axis to outside of the thread a distance that is a multiple of the thread pitch.

Figure 10.5-1: Semi-Auto Thread Screen

10.6 Feed Arc

The FEED ARC semi-automatic function interpolates and executes an arc movement. The start position of the arc is the current position. CYCLE START will begin feeding the arc until completion.

- R is the radius of the arc.
- EX is the X-axis end point of the arc, relative to the start position.
- EZ is the Z-axis end point of the arc, relative to the start position.
- DIRECTION SETTING sets the CW or CCW direction for the arc.

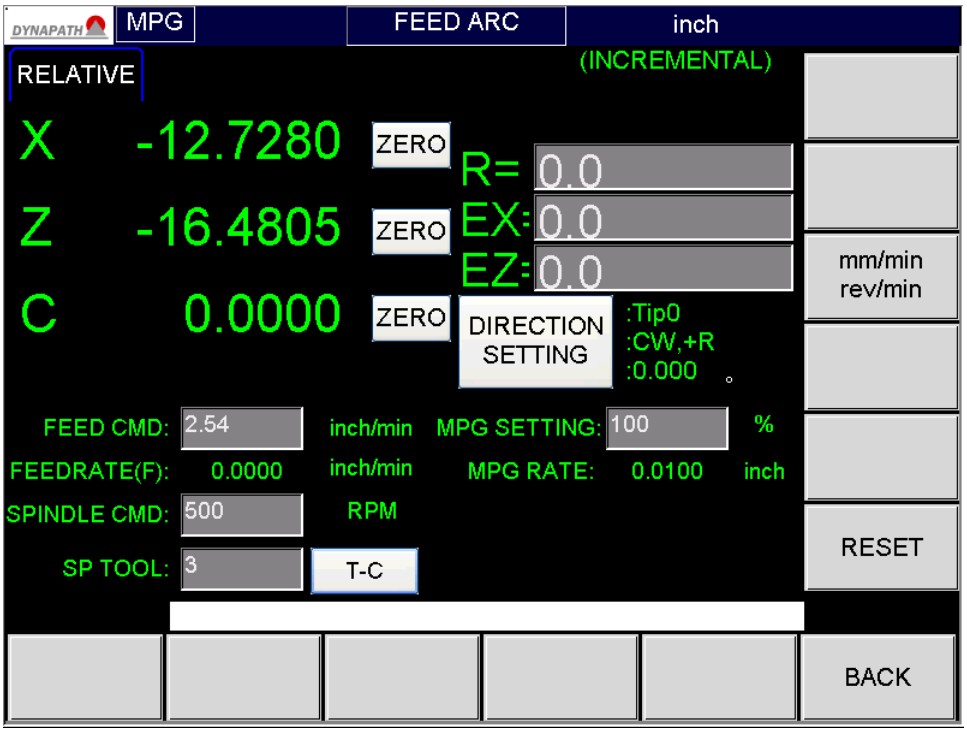


Figure 10.6-1: Semi-Auto Feed Arc Screen

11 MDI

11.1 MDI Operation

MDI mode accepts input of G-Code commands. Multiple lines may be entered. Press CYCLE START to begin execution of the commands.

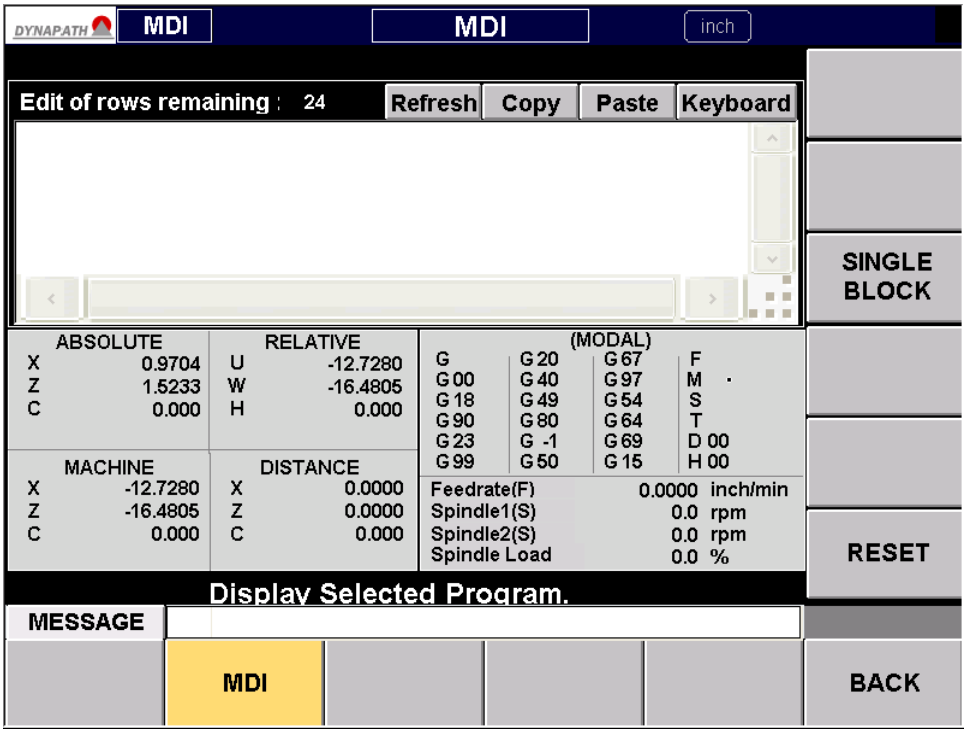


Figure 11.1-1: MDI Screen

12 File Management

12.1 File Manager

Upon entering the File Manager screen, a list of all the existing part programs on the control is presented in alphabetical order. To activate a file or open it for editing, select it from the list and press OPEN. Depending on whether the file is a G-Code program or a Conversational Program, the corresponding G-Code editor or the Conversational Editor will display the selected program.

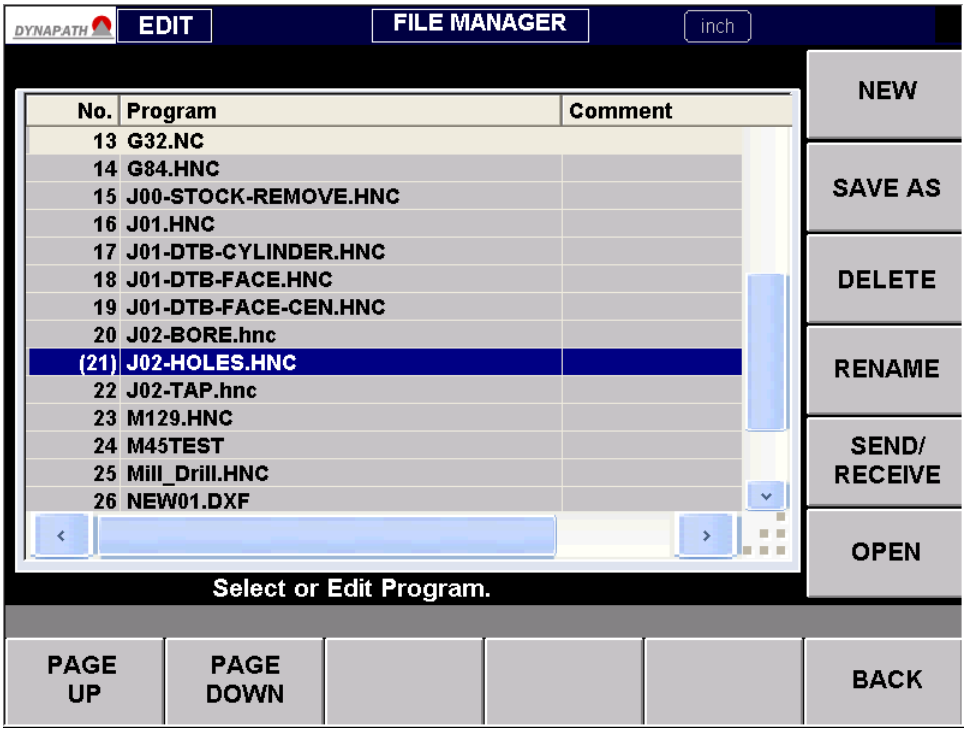


Figure 12.1-1: File Manager Screen

12.2 Send and Receive

Send and Receive allows transfer of part programs to and from the control system to an external device which can be a USB flash drive or Windows Network shared folder. On the left side of the screen is a list of files on the control, while on the right side of the screen are files on the external device. To copy files, select the file and then press the right or left arrows to perform the copy.

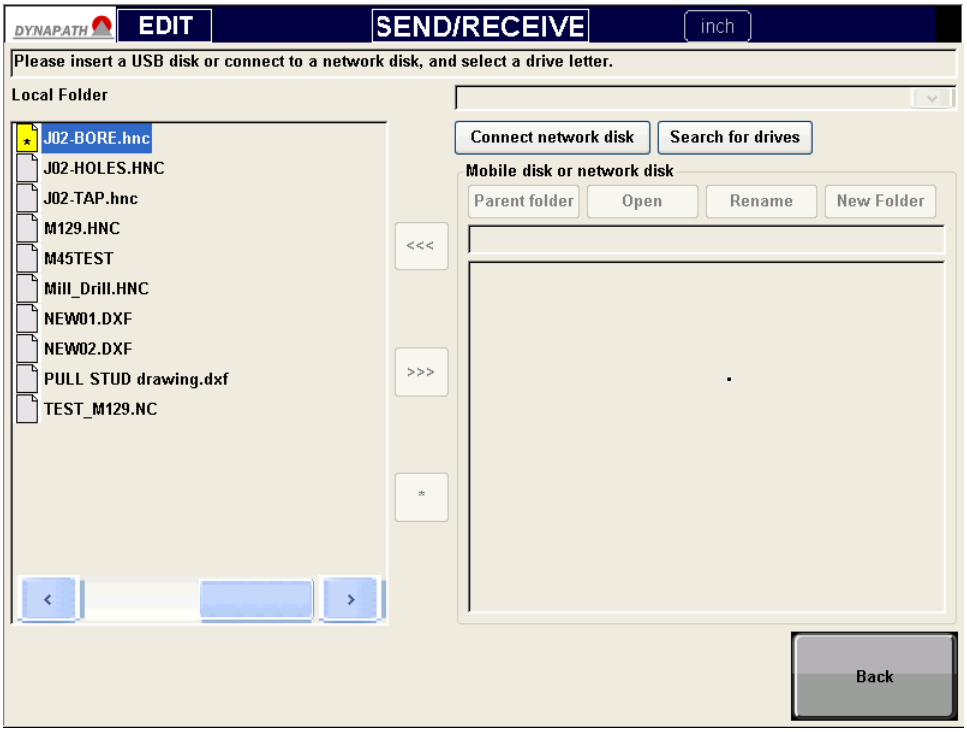


Figure 12.2-1: Send and Receive Screen

13 G-Code Programming

13.1 G-Code Reference

G-Code programming on the WinDelta control adheres to the ISO/EIA G-Code standard for turning.

Table 13.1.1: G-Code List

G-Code	Group	Description
G00	1	Positioning (Rapid traverse)
G01	1	Linear interpolation (Cutting feed)
G02	1	Circular interpolation CW or helical interpolation CW
G03	1	Circular interpolation CCW or helical interpolation CCW
G04	0	Dwell
G07.1	0	Cylindrical interpolation
G10	0	Programmable data input
G11	0	Programmable data input mode cancel
G12.1	21	Polar coordinate interpolation mode
G13.1	21	Polar coordinate interpolation cancel mode
G17	16	XpYp plane selection
G18	16	ZpXp plane selection
G19	16	YpZp plane selection
G20	6	Input in inch
G21	6	Input in mm
G28	0	Return to reference position
G29	0	Movement from reference position
G30	0	2nd and 3rd and 4th reference position return
G31	0	Skip function
G32	1	Threading
G40	7	Cutter compensation of tool nose radius: cancel
G41	7	Cutter compensation of tool nose radius: left
G42	7	Cutter compensation of tool nose radius: right
G43	23	Tool length compensation +
G44	23	Tool length compensation -
G49	23	Tool length compensation cancel
G50	0	Coordinate system setting or max. spindle speed clamp
G53	0	Machine coordinate system setting
G54	14	Workpiece coordinate system 1 selection
G55	14	Workpiece coordinate system 2 selection
G56	14	Workpiece coordinate system 3 selection
G57	14	Workpiece coordinate system 4 selection
G58	14	Workpiece coordinate system 5 selection
G59	14	Workpiece coordinate system 6 selection
G65	0	Macro call

G70	0	Finishing cycle
G71	0	Stock removal in turning
G72	0	Stock removal in facing
G73	0	Pattern repeating cycle
G74	0	End face peck drilling cycle
G75	0	Outer diameter/internal diameter drilling cycle
G76	0	Multiple-thread cutting cycle
G80	10	Canned cycle cancel for drilling
G83	10	Cycle for face drilling
G84	10	Cycle for face tapping
G85	10	Cycle for face boring
G87	10	Cycle for side drilling
G88	10	Cycle for side tapping
G89	10	Cycle for side boring
G90	1	Outer diameter/internal diameter cutting cycle
G92	1	Threading cycle
G94	1	End face turning cycle
G96	2	Constant surface speed control
G97	2	Constant surface speed control cancel
G93	5	Inverse time feed
G98	5	Feed per minute (mm/min or in/min)
G99	5	Feed per revolution (mm/rev or in/rev)

13.2 M-Code Reference

Table 13.2.1: M-Code List

M-Code	Description	Option
M00	Program Stop	Standard
M01	Optional Stop	Standard
M02	Program End	Standard
M03	Spindle CW Run	Standard
M04	Spindle CCW Run	Standard
M05	Spindle Stop	Standard
M07	Air Blast ON	Standard
M08	Flood Coolant ON	Standard
M09	Air Blast and Flood Coolant OFF	Standard
M10	Chuck Clamp	Auto Chuck
M11	Chuck Un-Clamp	Auto Chuck
M12	Tailstock Extend	Auto Tailstock
M13	Tailstock Retract	Auto Tailstock
M19	Spindle Orientation	C-Axis
M29	Rigid Tapping Mode ON	C-Axis
M30	Program End	Standard
M31	C-Axis Clamp	C-Axis
M32	C-Axis Un-Clamp	C-Axis
M45	C-Axis Mode ON	C-Axis
M46	C-Axis Mode OFF	C-Axis
M98	Sub-Program Call	Standard
M99	Sub-Program End, Return to Start of Program	Standard
M103	Secondary Spindle CW Run	2nd Spindle
M104	Secondary Spindle CCW Run	2nd Spindle
M105	Secondary Spindle Stop	2nd Spindle
M131	Probe On / Tool Setter Arm Extend	Tool Probe
M132	Probe Off / Tool Setter Arm Extend	Tool Probe
M822	Oil Mist Extractor ON	Mist Extractor
M823	Oil Mist Extractor OFF	Mist Extractor
M824	Parts Catcher Extend	Parts Catcher
M825	Parts Catcher Retract	Parts Catcher

13.3 T-Code Reference

The T-Code used to change tools and apply tool offsets is specified as follows:

T<two digit number for calling tool change><two-digit number to apply tool offset>

For example, T0305 calls tool 3 and applies offset 5.

14 MACRO PROGRAMMING

14.1 Introduction

Macro programming refers to the application of programmable logic into G-Code programs, by employing macro functions, conditions, or variables. The programmed code may be inserted directly into a normal G-Code program or written into a separate program to form a macro, which can be called by other programs. Macro programming enables more advanced machine control, for the purpose of automating repetitive tasks, dynamically performing actions based on conditions, or implementing custom routines.

14.2 Macro Program Names

For a macro program to be called from within another program, it must be named according to a specific naming convention starting with the letter “O” and ending with a four-digit number. Table 14.2.1details the names and ranges for user-defined and system reserved macro programs.

Table 14.2.1: Macro Name Ranges

Macro Name	Usage
O0001 – O8999	Custom Macro
O9000 – O9999	System Reserved

14.3 Macro Program Types

A macro program is called using two methods: a M98 Subprogram Call, or a G65 Macro Call. While a M98 Subprogram Call shares the same variable scope with its parent program and cannot accept parameter inputs, the G65 Macro Call has its own local variable scope and can accept parameter inputs. Both methods search for the called macro program from within the same folder, and if not found, it will default to the macro program of the same name within system folder. Table 14.3.1summarizes the differences between a subprogram call and a macro call.

Table 14.3.1: Macro Program Types and Scope

Macro Type	Calling Method	Variable Scope
Subprogram Call	M98 P<Program Number> Example: M98 P1001	*Global variable scope
Macro Call	G65 P<Program Number> Example: G65 P1001 A_B_C_...	*Local variable scope *Accepts parameter inputs

14.4 Macro Variables

14.4.1 Macro Variables Overview

The syntax for a macro variable is a ”#” symbol followed by a number. An overview of local, global, BBRAM, and system variables is presented in Table 14.4.1. More detailed documentation and usage examples are provided in the following subsections.

Table 14.4.1: Macro Variable Groups and Definitions

Type	Address	Description
Local	#1-#99	Local variables offer temporary storage of data. They possess a scope limited to within each program. (M98 subprograms are part of its parent program.) They are cleared after exiting the program.
Global	#100-#199	Global variables have a global scope, for storage of data intended to be shared across any main, macro, or subprogram. They are cleared after reboot.
BBRAM	#500-#999	BBRAM variables have a global scope and are stored in persistent memory, thus are remembered even after system reboot.
System	#1000-#99999	System variables are reserved for system related constants or variables.

14.4.2 Local Input Variables

The local variables #1 to #99 for macro programming are for custom use to store temporary data. They possess a limited local scope, thus after exiting the macro program, all local variables are reset to NULL. For G65 macro calls, each macro initiates an independent set of #1 to #99 local variables, while for M98 subprograms the local variables are shared with its parent program.

For G65 macro calls, parameters may be passed into the macro program. Table 14.4.2 and Table 14.4.3 list the mapping of input parameters to corresponding local variables for Spec I and II control system configurations.

Table 14.4.2: Spec I Local Variable Input Map

A #1	B #2	C #3	D #7	E #8	F #9	G ---	H #11	I #4	J #5
K #6	L ---	M #13	N ---	O ---	P ---	Q #17	R #18	S #19	T #20
U #21	V #22	W #23	X #24	Y #25	Z #26				

Table 14.4.3: Spec II Local Variable Input Map

A #1	B #2	C #3	I1 #4	J1 #5	K1 #6	I2 #7	J2 #8	K2 #9
I3 #10	J3 #11	K3 #12	I4 #13	J4 #14	K4 #15	I5 #16	J5 #17	K5 #18
I6 #19	J6 #20	K6 #21	I7 #22	J7 #23	K7 #24	I8 #25	J8 #26	K8 #27
I9 #28	J9 #29	K9 #30	I10 #31	J10 #32	K10 #33			

Example:
Macro call: G65 P1000 A10.0 B20.0 C3 S40.0 Z50.0
Within the O1000 macro program the corresponding local variables would receive the following values: #1=10.0, #2=20.0, #3=3, #19=40.0, #26=40.0, and all other local variables would be equal to NULL or #0.

14.4.3 System Variables

System variables are reserved for system related constants or variables.

Table 14.4.4: System Variables

Address	Type	Usage
#0, #3100	R	NULL variable
#3101	R	PI constant = 3.14159265358979323846
#3102	R	Exponential constant = 2.71828182845904523536
#1000-#1031	R	Digital inputs
#1032	R	Digital inputs, hexa-decimal 32-bit encoding of #1000-#1031
#1100-#1131	R/W	Digital outputs
#1132	R/W	Digital outputs, hexa-decimal 32-bit encoding of #1000-#1031
#2001-#2200	R/W	Tool length wear offsets H1 to H200
#2201-#2400	R/W	Tool length offsets H1 to H200
#2401-#2600	R/W	Tool diameter wear offsets D1 to D200
#2601-#2800	R/W	Tool diameter offsets D1 to D200
#3000		Macro alarm message, RESET required to clear this message. Usage: #3000=N(MESSAGE) N = Custom alarm number (1-200) MESSAGE = Alarm message to display on the screen

<table>
<tr><td></td><td></td><td>Example: #3000=1(ALARM MESSAGE)
This block displays “3001 ALARM MESSAGE” during execution.</td></tr>
<tr><td>#3001</td><td>R/W</td><td>Millisecond timer beginning at power on and restarts at 2147483648. Can be reset to an arbitrary start value.
Usage: #3001=N
N = Start value
Example: #3001=1000
This begins resets the timer to 1000 and is incremented every millisecond.</td></tr>
<tr><td>#3002</td><td>R/W</td><td>Hour timer beginning at power on and restarts at 9544.371767. Can be reset to an arbitrary start value.
Usage: #3002=N
N = Start value
Example: #3002=2
This begins resets the timer to 2 and is incremented every hour.</td></tr>
<tr><td>#3003</td><td>W</td><td>Single block control
Usage: #3003=N
<table>
<tr><td>N</td><td>Single Block</td><td>Wait for M/S/T-Code</td></tr>
<tr><td>0</td><td>ON</td><td>ON</td></tr>
<tr><td>1</td><td>OFF</td><td>ON</td></tr>
<tr><td>2</td><td>ON</td><td>OFF</td></tr>
<tr><td>3</td><td>OFF</td><td>OFF</td></tr>
</table></td></tr>
<tr><td>#3004</td><td>W</td><td>Feed hold (FH) and feed rate override (FO) control
Usage: #3004=N
<table>
<tr><td>N</td><td>Feed Hold</td><td>Feed Override</td></tr>
<tr><td>0</td><td>ON</td><td>ON</td></tr>
<tr><td>1</td><td>OFF</td><td>ON</td></tr>
<tr><td>2</td><td>ON</td><td>OFF</td></tr>
<tr><td>3</td><td>OFF</td><td>OFF</td></tr>
</table></td></tr>
<tr><td>#3006</td><td>R/W</td><td>Program stop with message
Usage: #3000=N(MESSAGE)
Example: #3000=1(ALARM MESSAGE)
N = Custom alarm number (1-200)
MESSAGE = Alarm message to display on the screen
This block displays “3001 ALARM MESSAGE” during execution.</td></tr>
<tr><td>#3011</td><td>R</td><td>Current date
Returns: YYYYMMDD
YYYY = Four-digit year
MM = Two-digit month
DD = Two-digit day</td></tr>
<tr><td>#3012</td><td>R</td><td>Current time
Returns: HHMMSS
HH = Two-digit hour (0-24)
MM = Two-digit minutes (0-60)</td></tr>
</table>

		SS = Two-digit seconds (0-60)
#4001	R	Current G-Code modal group 1: Returns: 0, 1, 2, 3
#4002	R	Current G-Code modal group 2: Returns: 17, 18, 19
#4003	R	Current G-Code modal group 3: Returns: 90, 91
#4004	R	Current G-Code modal group 4: Returns: 22, 23
#4005	R	Current G-Code modal group 5: Returns: 94, 95
#4006	R	Current G-Code modal group 6: Returns: 20, 21
#4007	R	Current G-Code modal group 7: Returns: 40, 41, 42
#4008	R	Current G-Code modal group 8: Returns: 43, 44, 49
#4009	R	Current G-Code modal group 9: Returns: 73, 74, 76, 80-89
#4010	R	Current G-Code modal group 10: Returns: 98, 99
#4011	R	Current G-Code modal group 11: Returns: 50, 51
#4012	R	Current G-Code modal group 12: Returns: 65, 66, 67
#4013	R	Current G-Code modal group 13: Returns: 96, 97
#4014	R	Current G-Code modal group 14: Returns: 54-59
#4015	R	Current G-Code modal group 15: Returns: 61-64
#4016	R	Current G-Code modal group 16: Returns: 68, 69
#4102	R	Current B-Code value
#4107	R	Current D-Code value
#4109	R	Current F-Code value
#4111	R	Current H-Code value
#4113	R	Current M-Code value

#4114	R	Current sequence number
#4119	R	Current S-Code value
#4120	R	Current T-Code value
#5001-#5003	R	Distance to go values (DTG), dynamically updated
#5021-#5023	R	Axes machine positions (MAC), updated after axes stopped
#5041-#5043	R	Axes absolute positions (ABS), updated after axes stopped
#5061-#5063	R	Skip position (G31), updated after axes stopped
#5081-#5083	R	Current tool length compensation
#5201-#5203	R/W	External offset
#5221-#5223	R/W	Work offsets G54
#5241-#5243	R/W	Work offsets G55
#5261-#5263	R/W	Work offsets G56
#5281-#5283	R/W	Work offsets G57
#5301-#5303	R/W	Work offsets G58
#5321-#5323	R/W	Work offsets G59
#7001-#7003… #7941-#7943	R/W	Work offsets G54.1 P1-P48

14.5 Arithmetic Functions

Table 14.5.1: Arithmetic Functions

Function	Format	Description
Set or Define	#i=#j	Sets a variable
Sum	#i=#j+#k	Adds multiple numbers or variables
Difference	#i=#j-#k	Subtracts numbers or variables
Product	#i=#j*#k	Multiplication of numbers or variables
Quotient	#i=#j/#k	Divides numbers or variables
Sine	#i=SIN[#j]	
Arcsine	#i=ASIN[#j]	
Cosine	#i=COS[#j]	
Arccosine	#i=ACOS[#j]	
Tangent	#i=TAN[#j]	
Arctangent	#i=ATAN[#j]/[#k]	

Square root	#i=SQRT[#j]	
Absolute	#i=ABS[#j]	
Round	#i=ROUND[#j]	
Round up	#i=FUP[#j]	
Round down	#i=FIX[#j]	
Natural log	#i=LN[#j]	
Exponential	#i=EXP[#j]	
Power	#i=POW[#j]	
Boolean AND	#i=#j AND #k	
Boolean OR	#i=#j OR #k	
Boolean XOR	#i=#j XOR #k	

14.6 Priority of Operations

The priority of operations is performed with priority beginning with:

1. Functions, e.g. SIN, COS, TAN, ABS…
2. Multiplication (*), Division (/), Boolean AND
3. Addition (+), Subtraction (-), Boolean OR/XOR

14.7 Bracket Nesting

Execution order occurs from innermost brackets to outermost bracket. The maximum depth of nested brackets is five, otherwise an alarm will occur.

14.8 Branch and Repetition

Table 14.8.1: Conditional Operators

Operator	Description
EQ	Equal to (==)
NE	Not equal to (!=)
GT	Greater than (>)
GE	Greater than or equal to (>=)
LT	Less than (<)
LE	Less than or equal to (<=)

Table 14.8.2: Control and Loop Functions

Function	Description
GOTO #i	Unconditional jump to block sequence number.
IF [...] GOTO #i	If expression evaluates to true, jump to a block sequence number.
IF [...] THEN #i=#j	If expression evaluates to true, perform a set function.
WHILE [...] DO #i	While expression evaluates to true, continuously jump to block sequence number. While-Do ranges can be nested up to three levels, but must not cross or overlap.

15 CONVERSATIONAL PROGRAMMING

15.1 Introduction

The Conversational Editor provides an online programming environment for creating part programs to be run on the control. Conversational programs are created by adding and configuring Conversational Events. When a conversational program is saved, a standard G-Code program is generated and ready for execution in AUTO mode. This chapter provides documentation for part programming on the control using the Conversational Editor.

15.2 Conversational Programming Workflow

Programming in the Conversational Editor involves adding, removing, and configuring Conversational Events. These events are grouped into general categories: Stock Removal, Drilling, Thread, Groove, Cut Off, Milling, Line Arc, Position, and Auxiliary. Each individual event is described in more detail in the subsequent sections. Figure 15.2-1 is a flowchart illustrating the general workflow for Conversational Programming.

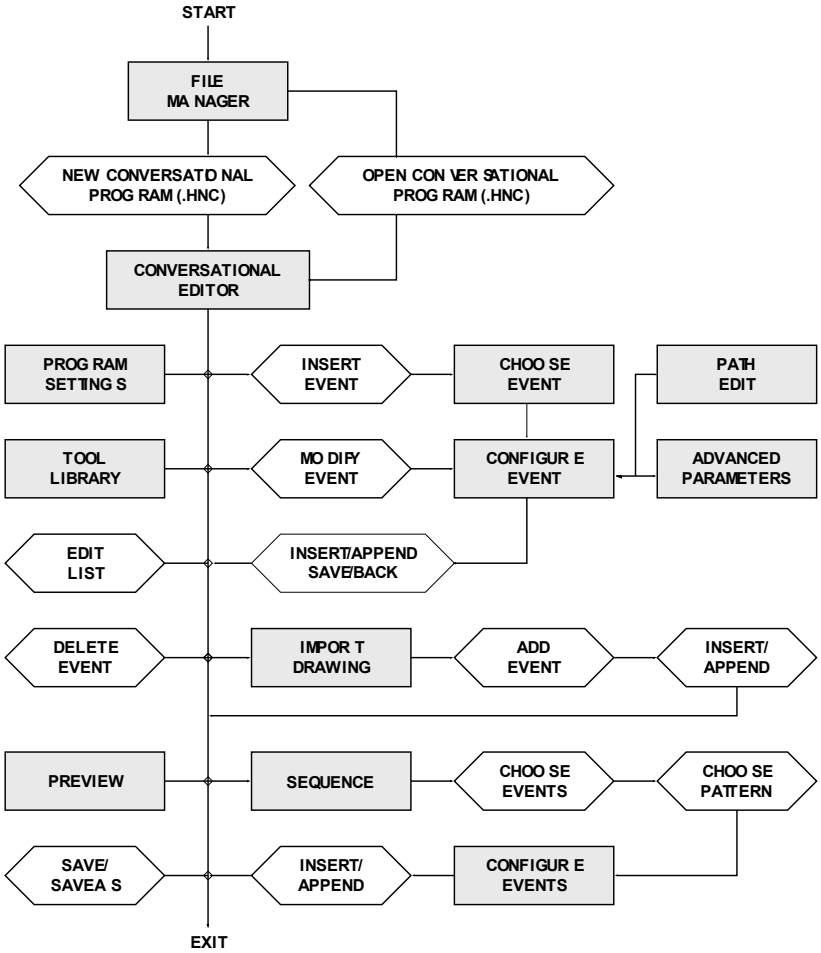


Figure 15.2-1: Conversational Programming Workflow

15.3 Main Screen

Once a new Conversational Program is created or an existing program is selected from the File Manager, the main Conversational Editor screen in Figure 2.5-3 will be presented.

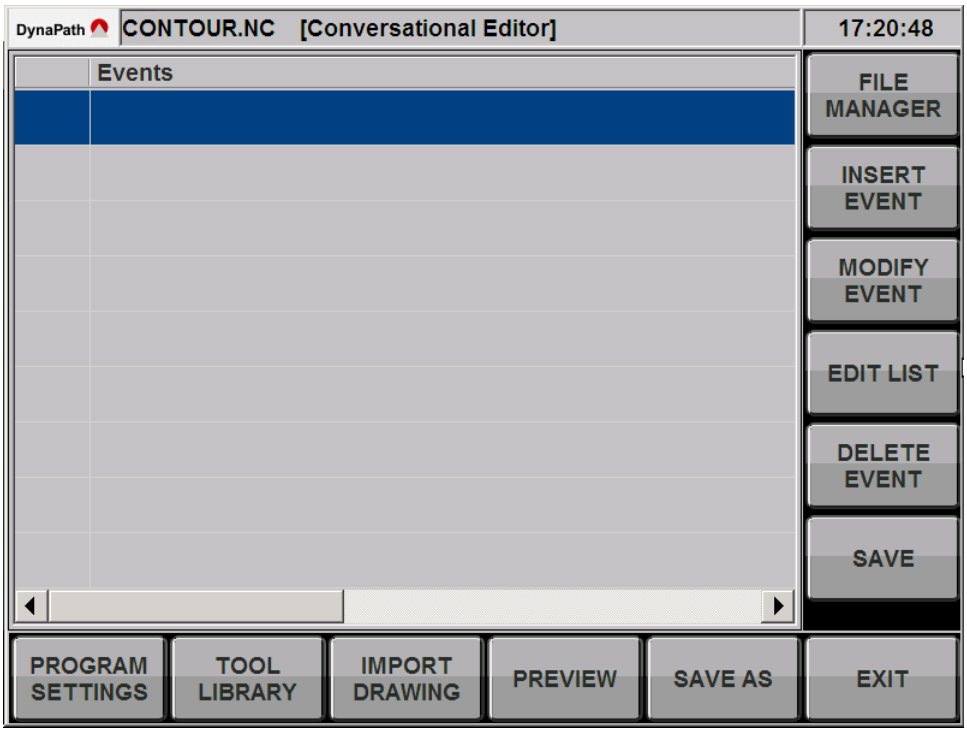


Figure 15.3-1: Conversational Editor Main Screen

Along the bottom of the screen are functions related to the entire program, while along the right side are functions to modify events within the program. A brief description of the button functions on the main screen are listed as follows:

- **PROGRAM SETTINGS** enable the setting of inch or mm units, work offset, material, tool change position, stock, and the safe or retract heights of events.
- **TOOL LIBRARY** enables management of the tools associated with this program for configuration and management.
- **IMPORT DRAWING** contains tools to draw profiles, or open DXF drawings to select geometry for import as events.
- **PREVIEW** shows a preview of the programmed tool paths.
- **PART** toggles between programmed events for the front or back end of a part.
- **SAVE/ SAVE AS** will save the current program.
- **FILE MANAGER** returns to the file list screen to select or create a program.
- **INSERT EVENT** creates and inserts a new event above the highlighted event in the event list.
- **MODIFY EVENT** opens the highlighted event for modification.
- **EDIT LIST** contains functions to shift, cut, copy, and paste events.
- **EXIT** quits the Conversational Editor.

15.4 Program Settings

The Program Settings screens offer set up options that affect the current conversational program. The settings for each screen is detailed in the following list.

1. **Machine Settings**
    a. **Measurement units** specify this program uses either inch (G20) or mm (G21) mode in the program header.
    b. **Work Offset** (G54 to G59) specifies which fixture or work offset will be referenced in the program header.
    c. **Workpiece material** is used in conjunction with tool type, tool material, and machining method for suggestion of feeds and speeds within each event.
    d. **Tool Change** position is the location in which any tool change is performed. This can be set with reference to the work offset or part zero location, or with respect to the machine zero location.
2. **Blank Settings**
    a. **BL** sets the stock or blank workpiece type, either cylindrical or pipe.
    b. **XA** sets the outer diameter of the workpiece for use in determining where to start material removal for stock removal events.
    c. **XI** sets the inner diameter of the workpiece for use in determining where to start material removal for stock removal events.
    d. **ZA** sets the start position coordinate in the Z-axis with respect to the part zero or work offset location.
    e. **ZI** sets the end position coordinate of the part in the chuck.
    f. **ZB** sets the position coordinate of where the chuck begins.
3. **Retract Settings**
    a. **SC** sets an amount to enlarge the stock or blank from which to start machining. This is also a distance away from the stock in which the program will use to traverse between events.
    b. **XRA** sets a default retract amount in the X-axis.
    c. **ZRA** sets a default retract amount in the Z-axis.

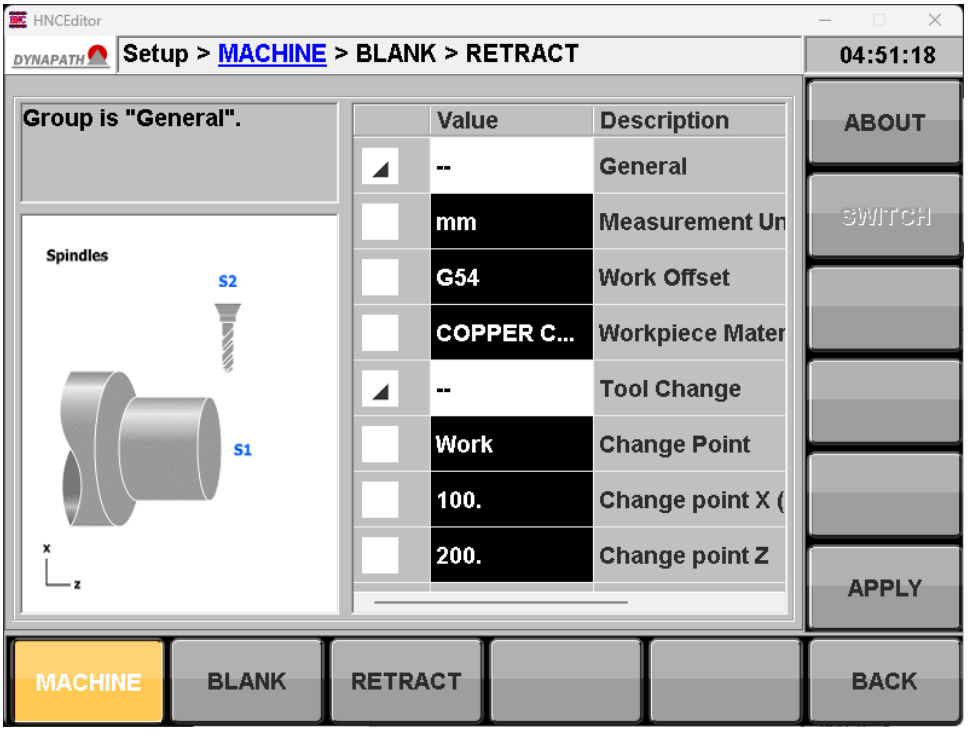


Figure 15.4-1: Program Settings - Machine

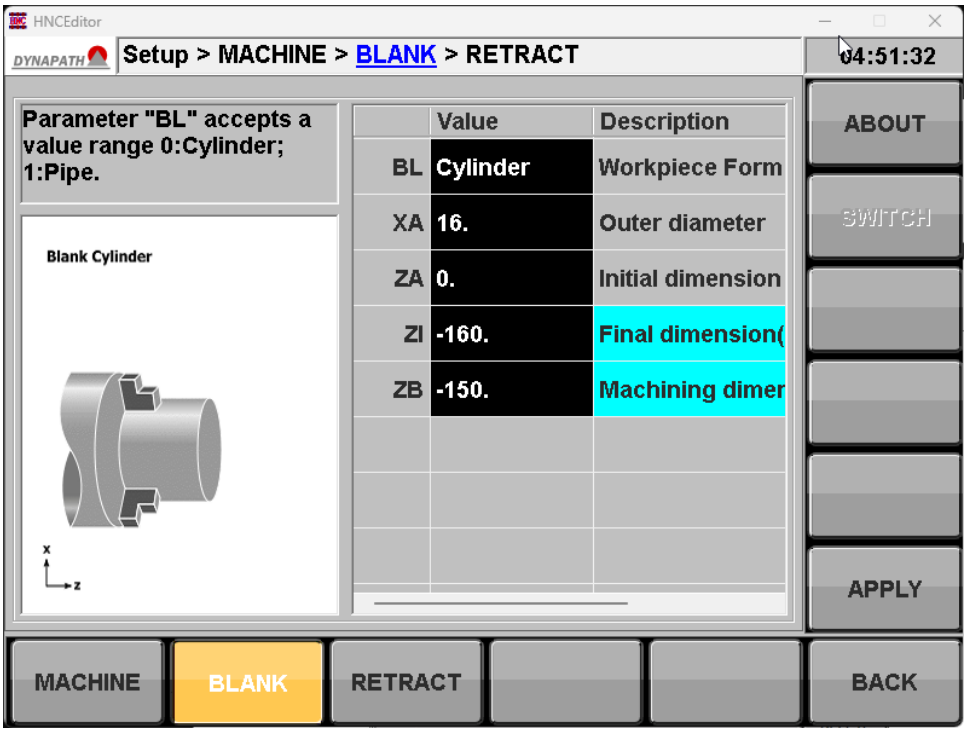


Figure 15.4-2: Program Settings - Blank

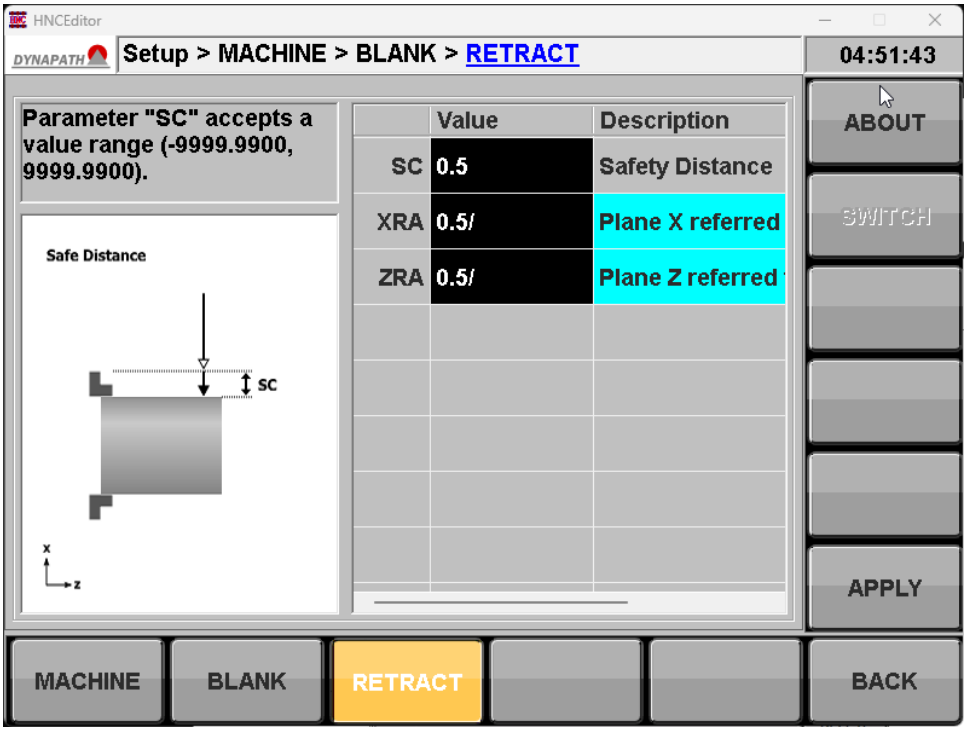


Figure 15.4-3: Program Settings - Retract

15.5 Stock Removal Events

15.5.1 Profile Event

The Turning Profile event contains a profile path that represents the final shape of the part for a specific Stock Removal or Irregular Groove event to reference. This profile path can be assigned an arbitrary reference name and is either defined using the Edit Path or Import Path function.

The Edit Path function starts the Path Profile Editor to allow the construction of a path by first entering a start point, followed by individual segments by clicking on SEGMENT and selecting a type. For each segment, enter the X and Z endpoints as required. The tail of each segment can be blended into the next segment by selecting chamfer or fillet and specifying the corner radius (ConRad) as shown in Figure 15.5-1. Click the COMPLETE button when the path construction is finished, and then click on SAVE to insert this profile as an event.

The Import Path function will open the DXF Editor to allow the selection of a path from the drawing and importing it directly as a profile event. Reference section 16 DRAWING IMPORT for more details.

Note: A profile path shape should always be drawn in the Z+ and X+ quadrant, starting from right to left, with the X-axis in diameter units and positive X leading away from centerline.

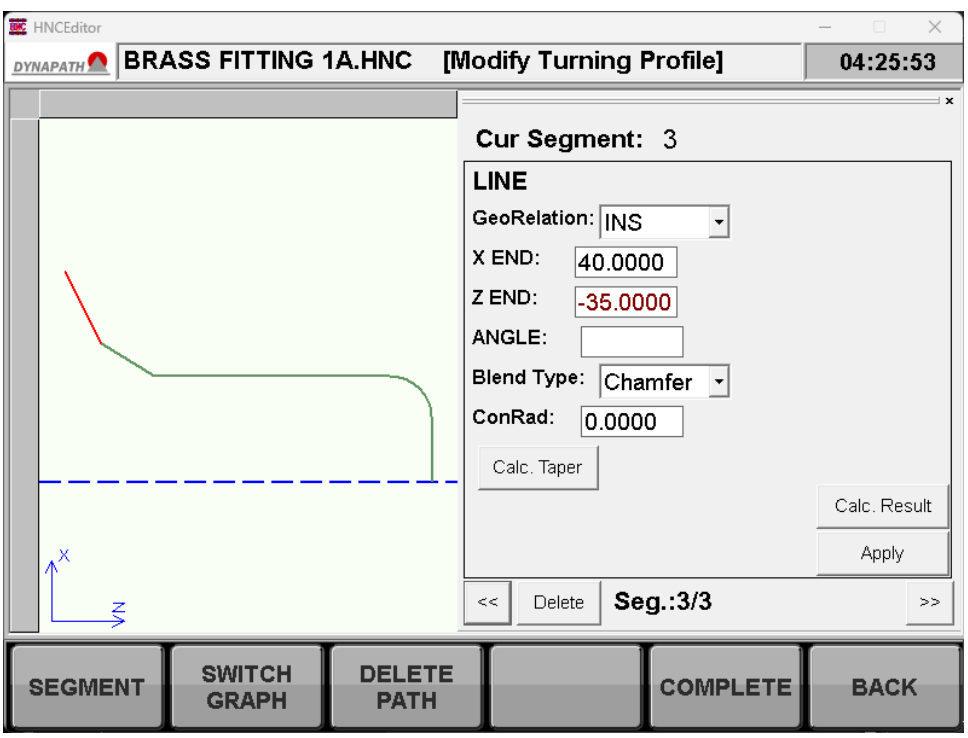


Figure 15.5-1: Profile Path Editor

15.5.2 Stock Removal – Turning (G71)

The Stock Removal – Turning (G71) event performs a turning operation to create an outer or inner diameter shape.

Note: **A turning profile path must first be programmed to be referenced by this event.**

Note**: The profile cannot have grooves and must have either an increasing or decreasing X along its path. If grooves exist, it must be separately programmed as an Irregular Groove event.**

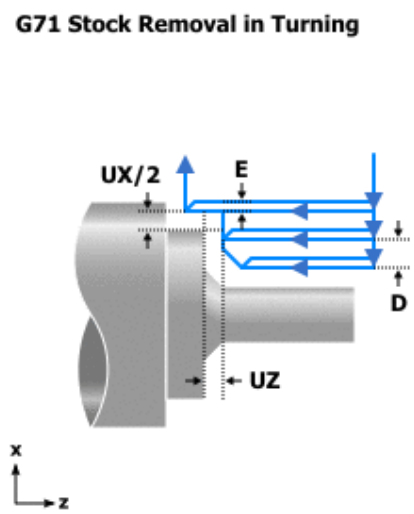


Figure 15.5-2: Stock Removal – Turning (G71)

Basic Parameters:

- **Machining** Rough, Finish, or Rough + Finish cutting along the Z-axis (longitudinal) and on the outer diameter (OD) or inner diameter (ID)
- **REF** Reference profile number

Roughing Parameters:

- **T** Tool ID
- **V** Constant cutting speed (mm or in per minute)
- **SL** Spindle maximum speed (RPM)
- **S** Spindle speed (RPM)
- **F** Feed rate (mm or in, per min or per rev)
- **D** Depth of cut
- **UX** Finishing allowance X
- **UZ** Finishing allowance Z
- **E** Retract distance
- **CL** Coolant (On/Off)

- **DS** Spindle rotation direction (CW, CCW)
- **SP** Start point in X-axis
- **RP** Reference profile (Stock, Profile)

Finishing Parameters:

- **T** Tool ID
- **V** Constant cutting speed (mm or in per minute)
- **SL** Spindle maximum speed (RPM)
- **S** Spindle speed (RPM)
- **F** Feed rate (mm or in, per min or per rev)
- **COM** Tool radius compensation (Off, Left, Right)
- **CL** Coolant (On/Off)
- **DS** Spindle rotation direction (CW, CCW)
- **SP** Start point in X-axis
- **RP** Reference profile (Stock, Profile)

15.5.3 Stock Removal – Facing (G72)

The Stock Removal – Facing (G72) event performs a facing operation to create a profile shape on the end face of the workpiece.

Note: **A turning profile path must first be programmed to be referenced by this event.**

Note**: The profile cannot have grooves and must have increasing or decreasing Z along its path. if grooves exist, it must be separately programmed as a Face Groove or event.**

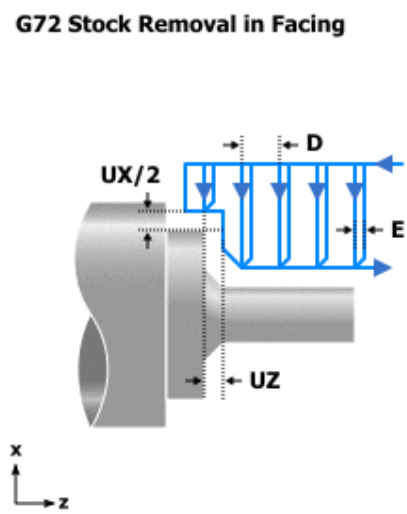


Figure 15.5-3: Stock Removal – Facing (G72)

Basic Parameters:

- **Machining** Rough, Finish, or Rough + Finish cutting along the end face
- **REF** Reference profile number

Roughing Parameters:

- **T** Tool ID
- **V** Constant cutting speed (mm or in per minute)
- **SL** Spindle maximum speed (RPM)
- **S** Spindle speed (RPM)
- **F** Feed rate (mm or in, per min or per rev)
- **D** Depth of cut
- **UX** Finishing allowance X
- **UZ** Finishing allowance Z
- **E** Retract distance
- **CL** Coolant (On/Off)
- **DS** Spindle rotation direction (CW, CCW)

- **SP** Start point in X-axis
- **RP** Reference profile (Stock, Profile)

Finishing Parameters:

- **T** Tool ID
- **V** Constant cutting speed (mm or in per minute)
- **SL** Spindle maximum speed (RPM)
- **S** Spindle speed (RPM)
- **F** Feed rate (mm or in, per min or per rev)
- **COM** Tool radius compensation (Off, Left, Right)
- **CL** Coolant (On/Off)
- **DS** Spindle rotation direction (CW, CCW)
- **SP** Start point in X-axis
- **RP** Reference profile (Stock, Profile)

15.5.4 Stock Removal – Pattern Repeating (G73)

The Stock Removal – Facing (G73) event performs a pattern repeating operation to create a profile shape using progressive increments in the X and Z-axes.

Note: **A turning profile path must first be programmed to be referenced by this event.**

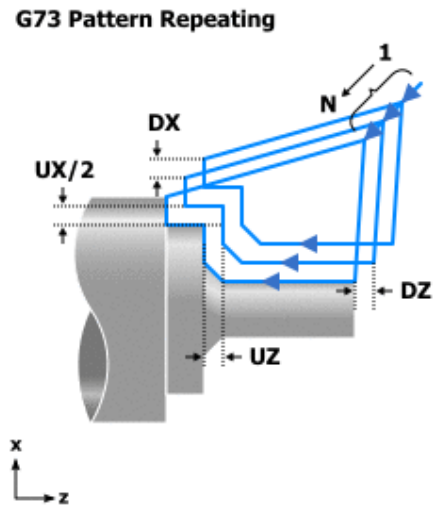


Figure 15.5-4: Stock Removal – Pattern Repeating (G73)

Basic Parameters:

- **Machining** Rough, Finish, or Rough + Finish cutting along the end face
- **REF** Reference profile number

Roughing Parameters:

- **T** Tool ID
- **V** Constant cutting speed (mm or in per minute)
- **SL** Spindle maximum speed (RPM)
- **S** Spindle speed (RPM)
- **F** Feed rate (mm or in, per min or per rev)
- **UX** Finishing allowance X
- **UZ** Finishing allowance Z
- **DX** Depth of cut in X
- **DZ** Depth of cut in Z
- **N** Number of cuts
- **CL** Coolant (On/Off)
- **DS** Spindle rotation direction (CW, CCW)
- **SP** Start point in X-axis
- **RP** Reference profile (Stock, Profile)

Finishing Parameters:

- **T** Tool ID
- **V** Constant cutting speed (mm or in per minute)
- **SL** Spindle maximum speed (RPM)
- **S** Spindle speed (RPM)
- **F** Feed rate (mm or in, per min or per rev)
- **COM** Tool radius compensation (Off, Left, Right)
- **CL** Coolant (On/Off)
- **DS** Spindle rotation direction (CW, CCW)
- **SP** Start point in X-axis
- **RP** Reference profile (Stock, Profile)

15.5.5 Turning (G90) and Facing (G94)

The Turning (G90) and Facing (G94) event performs a roughing operation to turn or face along a line, commonly used to square off a part.

Basic Parameters:

- **Machining** Rough or Finish, cutting along the Z-axis (longitudinal on outer or inner diameter), or along the end face

Roughing Parameters:

- **T** Tool ID
- **V** Constant cutting speed (mm or in per minute)
- **SL** Spindle maximum speed (RPM)
- **S** Spindle speed (RPM)
- **F** Feed rate (mm or in, per min or per rev)
- **X0** Start point X
- **Z0** Start point Z
- **X1** End point X
- **Z1** End point Z
- **D** Depth of cut
- **UX** Finishing allowance X
- **UZ** Finishing allowance Z
- **CL** Coolant (On/Off)
- **DS** Spindle rotation direction (CW, CCW)

Finishing Parameters:

- **T** Tool ID
- **V** Constant cutting speed (mm or in per minute)
- **SL** Spindle maximum speed (RPM)
- **S** Spindle speed (RPM)
- **F** Feed rate (mm or in, per min or per rev)
- **X0** Start point X
- **Z0** Start point Z
- **X1** End point X
- **Z1** End point Z
- **CL** Coolant (On/Off)
- **DS** Spindle rotation direction (CW, CCW)

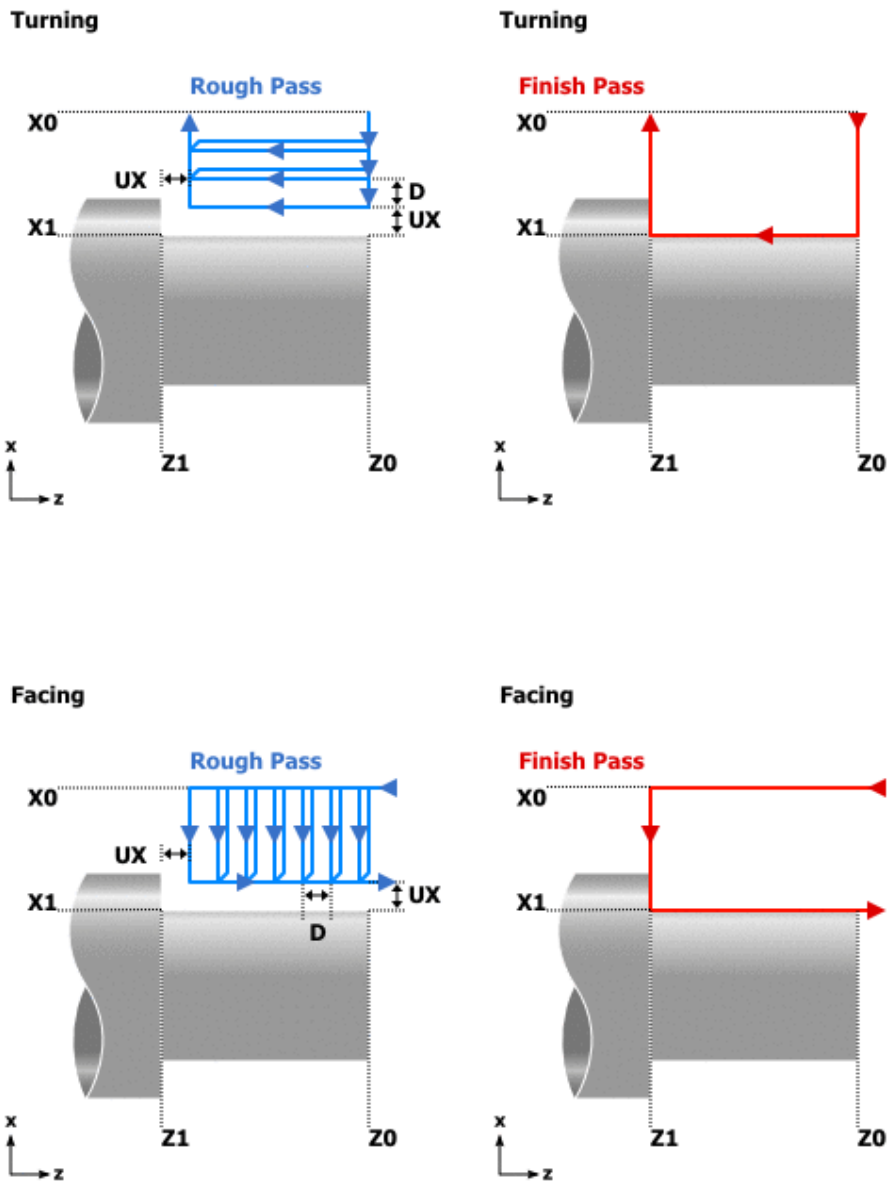


Figure 15.5-5: Turning (G90) and Facing (G94)

15.5.6 Turning Profile

The turning profile event is a container for a profile shape. In this event, the construction or import of a path profile is performed, and when complete it can then be referenced by other events such as stock removal turning or facing events.

Figure 15.5-6: Turning Profile

15.6 Hole Events

15.6.1 Face Drill (G83)

The Face Drill (G83) event performs drilling operations in the Z-axis direction on the end face of the workpiece.

Parameters:

- **T** Tool ID
- **S** Spindle speed (RPM)
- **F** Feed rate (mm or in, per min)
- **Z** End position
- **Q** Peck increment
- **E** Escape distance each peck
- **RP** Start position
- **DT** Dwell duration (seconds)
- **CL** Coolant (On/Off)
- **DS** Spindle rotation direction (CW, CCW)
- **CM** Clamp C-axis (On/Off)

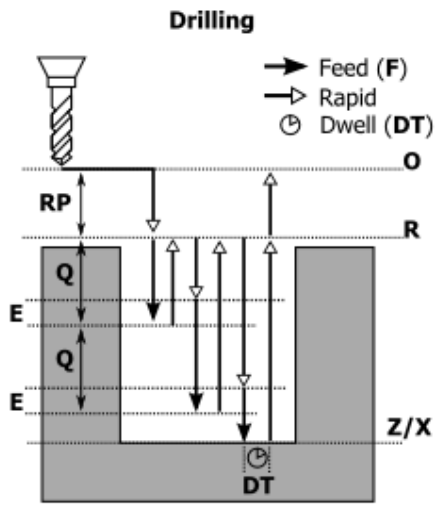


Figure 15.6-1: Face Drilling (G83)

15.6.2 Face Tap (G84)

The Face Tap (G84) event performs tapping operations in the Z-axis direction on the end face of the workpiece.

Parameters:

- **T** Tool ID
- **P** Pitch of tap (mm or in)
- **S** Spindle speed (RPM)
- **Z** End position
- **Q** Peck increment
- **RP** Start position
- **CL** Coolant (On/Off)
- **CM** Clamp C-axis (On/Off)
- **TT** Tapping method (Rigid/Normal)

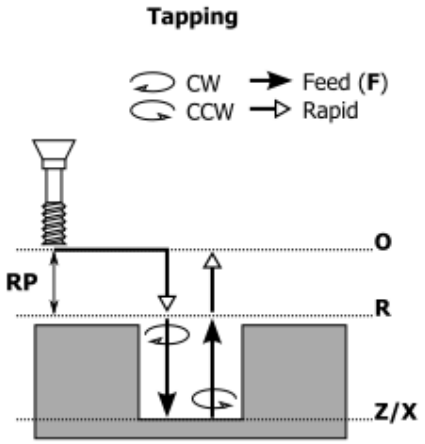


Figure 15.6-2: Face Tapping (G84)

15.6.3 Face Bore (G85)

The Face Bore (G85) event performs boring operations in the Z-axis direction on the end face of the workpiece.

Parameters:

- **T** Tool ID
- **S** Spindle speed (RPM)
- **F** Feed rate (mm or in, per min)
- **Z** End position
- **RP** Start position
- **DT** Dwell duration (seconds)
- **CL** Coolant (On/Off)
- **DS** Spindle rotation direction (CW, CCW)
- **CM** Clamp C-axis (On/Off)

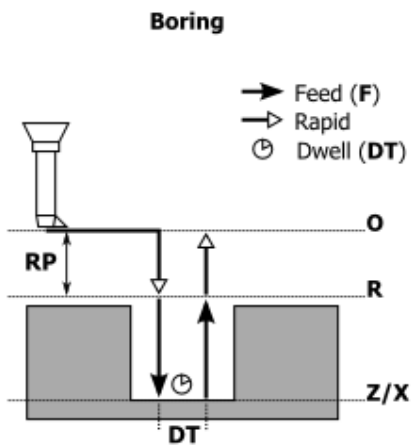


Figure 15.6-3: Face Bore (G85)

15.6.4 Side Drill (G87)

The Side Drill (G87) event performs drilling operations in the X-axis direction on the side (OD/ID) surface of the workpiece.

Parameters:

- **T** Tool ID
- **S** Spindle speed (RPM)
- **F** Feed rate (mm or in, per min)
- **X** End position
- **Q** Peck increment
- **E** Escape distance each peck
- **RP** Start position
- **DT** Dwell duration (seconds)
- **CL** Coolant (On/Off)
- **DS** Spindle rotation direction (CW, CCW)
- **CM** Clamp C-axis (On/Off)

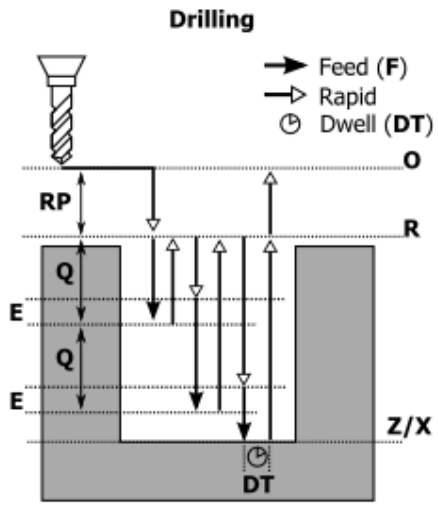


Figure 15.6-4: Side Drill (G87)

15.6.5 Side Tap (G88)

The Side Tap (G88) event performs tapping operations in the X-axis direction on the side (OD/ID) surface of the workpiece.

Parameters:

- **T** Tool ID
- **P** Pitch of tap (mm or in)
- **S** Spindle speed (RPM)
- **X** End position
- **Q** Peck increment
- **RP** Start position
- **CL** Coolant (On/Off)
- **CM** Clamp C-axis (On/Off)
- **TT** Tapping method (Rigid/Normal)

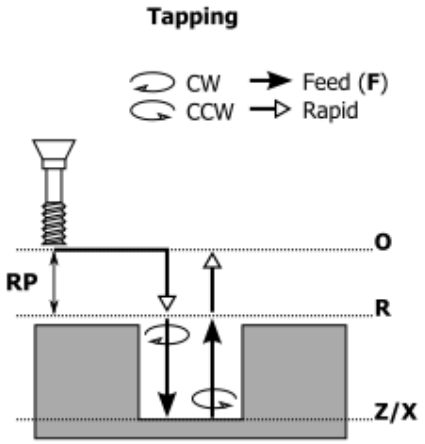


Figure 15.6-5: Side Tap (G88)

15.6.6 Side Bore (G89)

The Side Bore (G89) event performs boring operations in the X-axis direction on the side (OD/ID) surface of the workpiece.

Parameters:

- **T** Tool ID
- **S** Spindle speed (RPM)
- **F** Feed rate (mm or in, per min)
- **X** End position
- **RP** Start position
- **DT** Dwell duration (seconds)
- **CL** Coolant (On/Off)
- **DS** Spindle rotation direction (CW, CCW)
- **CM** Clamp C-axis (On/Off)

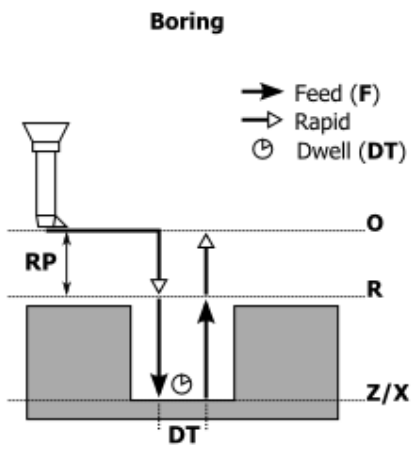


Figure 15.6-6: Side Bore (G89)

15.7 Thread Events

15.7.1 Thread and Tapered Thread

The Thread event performs threading operations along the longitudinal (OD/ID) side of the workpiece. The tool starts at X0 and Z0, plus a distance LW rounded to the nearest multiple of the pitch P away from the specified X0 start point. The tool is then placed at the first feed depth which is at a X-position equal to the major diameter MAD minus the first infeed depth D1 while also considering the tool radius. The threading cycle then begins by advancing the Z-axis towards end point Z1, with the spindle rotating one revolution every pitch distance travelled on the Z-axis. When a pass is completed, the tool is then retracted by a distance specified by VR and then returned to the start point X0, Z0, ready to perform a new pass at the next depth. This process is repeated until the final depth H1 is reached.

***Note*: The thread depth can be automatically calculated by pressing the SUGGEST button which considers the tool tip radius and the thread tool angle in its calculations.**

***Note*: Threading cycles always start at zero degrees spindle orientation, therefore if a part is placed with the same orientation in the chuck, existing threads can be "picked" back up by a subsequent threading process provided the pitch is set the same. This forms the basis on performing thread repair.**

Parameters:

- **T** Tool ID
- **MAD** Major diameter (mm or inch)
- **P** Pitch of thread (mm or inch)
- **S** Spindle speed (RPM)
- **A** Tool tip angle (degrees)
- **NN** Repetitive finishing count
- **H1** Thread depth (mm or inch radius value)
- **X0** Start point X
- **Z0** Start point Z
- **X1** End point X (available for Tapered Thread)
- **Z1** End point Z
- **LW** Advance thread distance (mm or inch)
- **LR** Thread end chamfer (mm or inch)
- **D1** First infeed depth (mm or inch radius value)
- **U** Finish allowance (mm or inch radius value)
- **VR** Retract distance (mm or inch radius value)
- **D0** Minimum cutting depth (mm or inch radius value)
- **SI** Thread on outer or inner diameter
- **CL** Coolant (On, Off)
- **TIF** Thread feed contact:
    - Along one edge implements the G76 cycle
    - Along two edges implements the G92 cycle

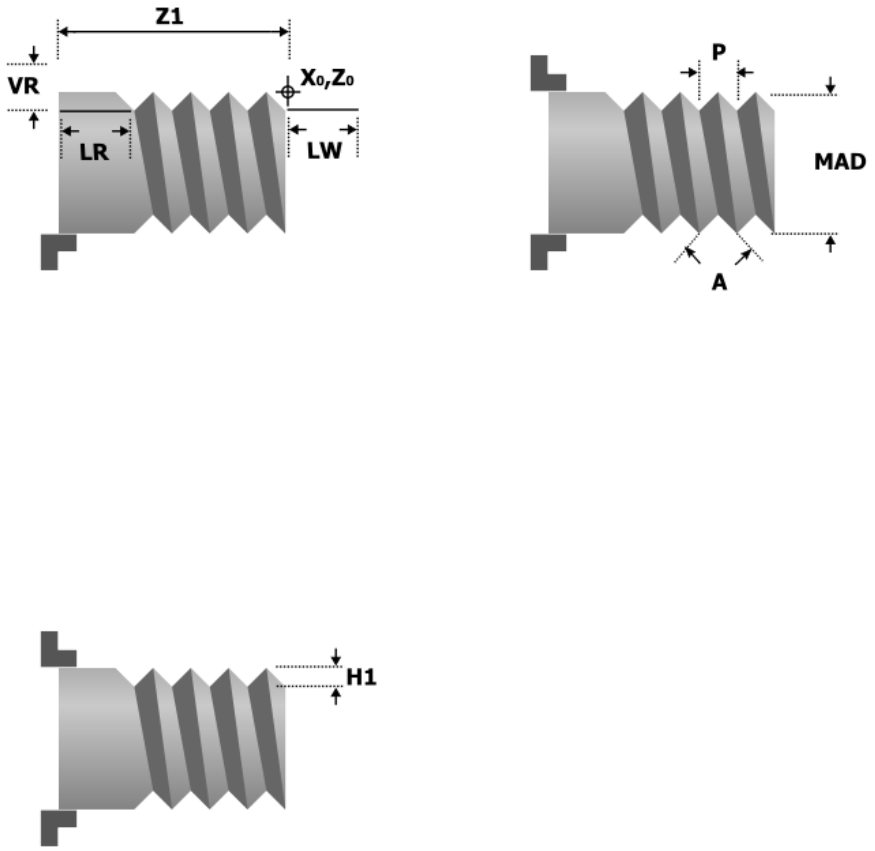


Figure 15.7-1: Thread

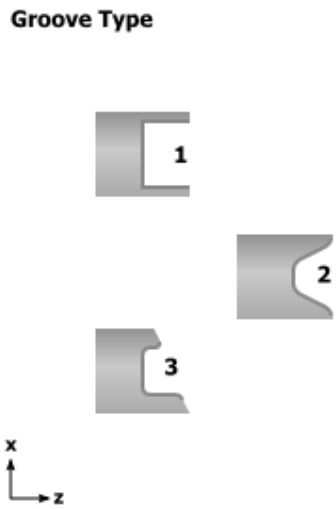
15.8 Groove Events

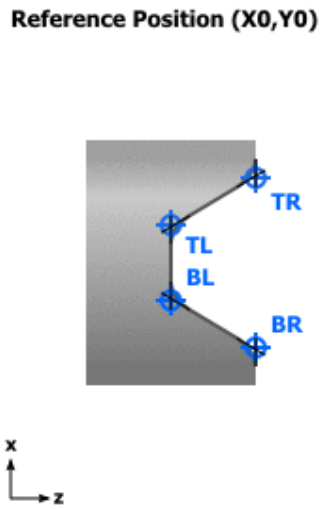
15.8.1 Face Groove

The Face Groove event performs grooving operations on the end face of the workpiece. Groove events plunge a groove tool continuously, at optional small peck increments, to machine a groove.

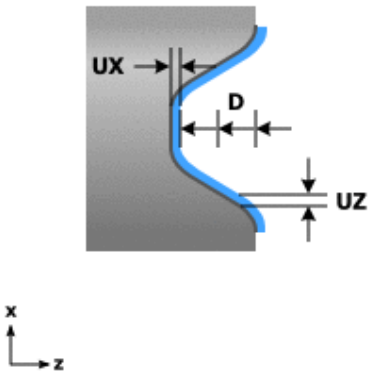
Parameters:

- **T** Tool ID
- **S** Spindle RPM (mm or inch)
- **F** Feed rate (mm or inch per revolution)
- **TYP** Groove type (1: Rectangular, 2: Tapered, 3: Angled)
- **MA** Machining process (Rough, Finish, Rough + Finish)
- **RPT** Reference position (Top-right, Bottom-right, Top-left, Bottom-left)
- **X0** Reference position X (absolute)
- **Z0** Reference position Z (absolute)
- **B1** Groove width (mm or inch)
- **T1** Groove depth (absolute or relative to Z0)
- **D** Peck depth (mm or inch)
- **U** Finish allowance in X and Z (radius units)
- **N** Number of grooves
- **CL** Coolant (On/Off)
- **DS** Spindle direction (CW, CCW)

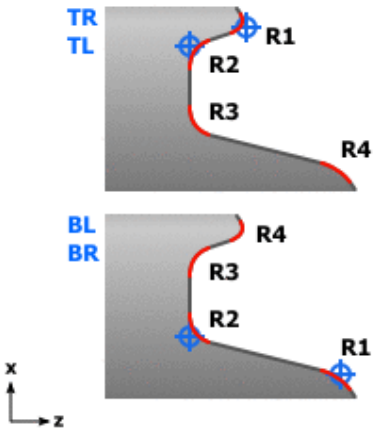




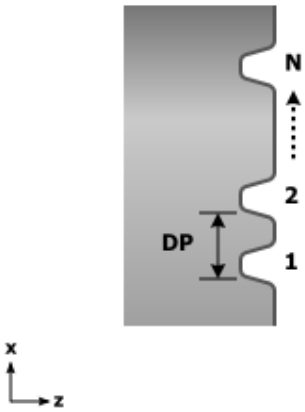
Groove Peck and Allowance



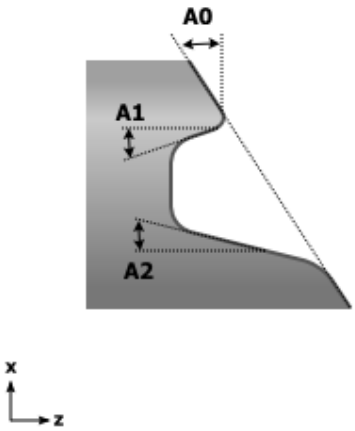
Groove Fillet



Groove Count



Groove Angles



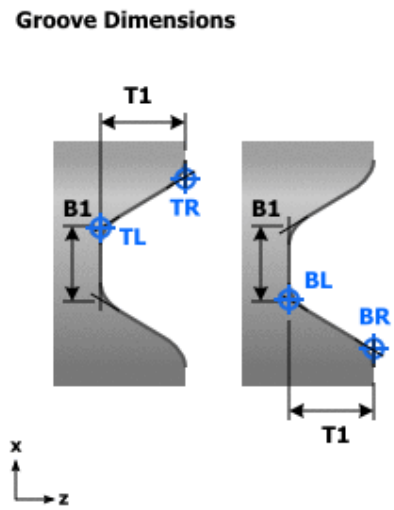


Figure 15.8-1 Face Groove

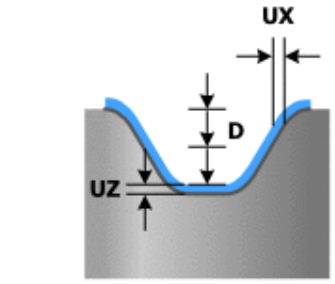
15.8.2 Turn Groove

The Turn Groove event performs grooving operations on the side surface (OD/ID) of the workpiece. Groove events plunge a groove tool continuously, at optional small peck increments, to machine a groove.

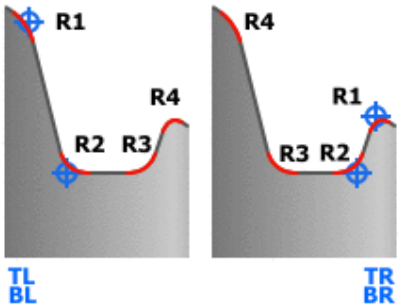
Parameters:

- **T** Tool ID
- **S** Spindle RPM (mm or inch)
- **F** Feed rate (mm or inch per revolution)
- **TYP** Groove type (1: Rectangular, 2: Tapered, 3: Angled)
- **MA** Machining process (Rough, Finish, Rough + Finish)
- **RPT** Reference position (Top-right, Bottom-right, Top-left, Bottom-left)
- **POS** Machining direction (Outside, Inside)
- **X0** Reference position X (absolute)
- **Z0** Reference position Z (absolute)
- **B1** Groove width (mm or inch)
- **T1** Groove depth (absolute or relative to Z0)
- **D** Peck depth (mm or inch)
- **U** Finish allowance in X and Z (radius units)
- **N** Number of grooves
- **CL** Coolant (On/Off)
- **DS** Spindle direction (CW, CCW)

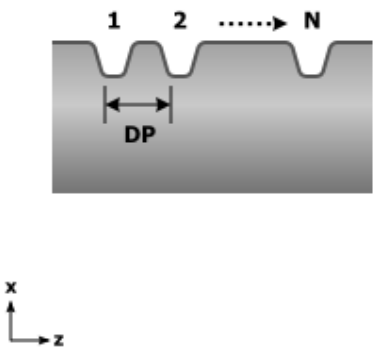
Groove Peck and Allowance



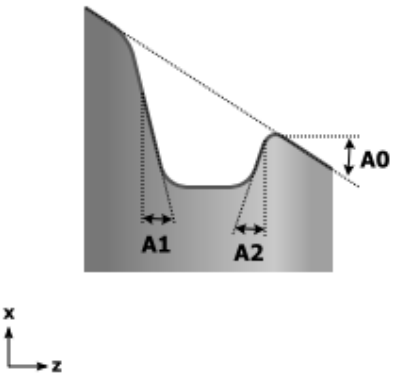
Groove Fillet



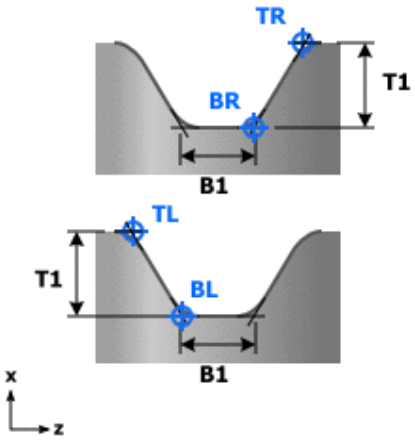
Groove Count



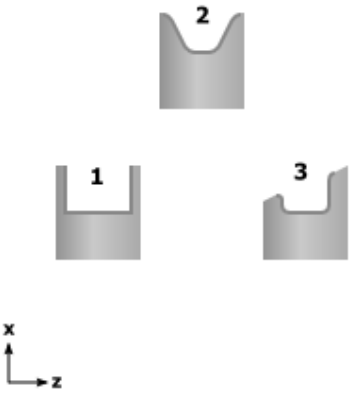
Groove Angles



Groove Dimensions



Groove Type



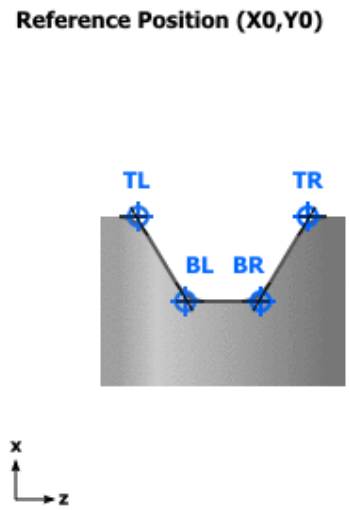


Figure 15.8-2: Turn Groove

15.8.3 Irregular Groove

The Irregular Groove event performs grooving operations on the side surface (OD/ID) as well as on the end face of the workpiece. Groove events plunge a groove tool continuously, at optional small peck increments, to machine a groove. The Irregular Groove event accepts constructed or imported profiles and performs machining of the irregular shape.

Basic Parameters:

- **Machining** Rough, Finish or Rough + Finish; machining the groove on the side longitudinal surface (OD/ID) or on the end face

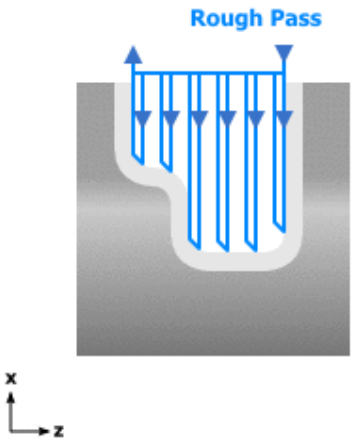
Roughing Parameters:

- **T** Tool ID
- **V** Constant cutting speed (mm or inch per minute)
- **SL** Spindle maximum speed (RPM)
- **S** Spindle speed (RPM)
- **F** Feed rate (mm or inch, per min or per rev)
- **DIR** Direction of plunge passes (right to left, left to right)
- **OD** Roughing passes offset distance (% tool diameter)
- **UX** Finishing allowance X (mm or inch, radius units)
- **UZ** Finishing allowance Z (mm or inch, radius units)
- **D** Depth of cut (mm or inch, radius units)
- **E** Retract amount for each peck (mm or inch, radius units)
- **CL** Coolant (On/Off)
- **DS** Spindle rotation direction (CW, CCW)

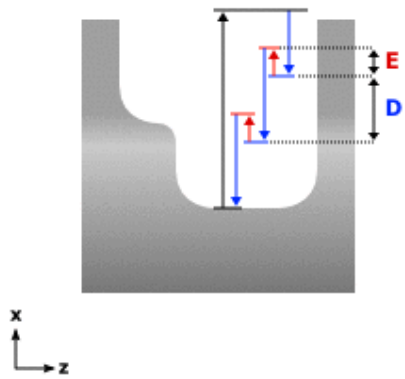
Finishing Parameters:

- **T** Tool ID
- **V** Constant cutting speed (mm or inch per minute)
- **SL** Spindle maximum speed (RPM)
- **S** Spindle speed (RPM)
- **F** Feed rate (mm or inch, per min or per rev)
- **DIR** Direction of plunge passes (right to left, left to right)
- **F2** Bottom Feed rate (mm or inch, per min or per rev)
- **CL** Coolant (On/Off)
- **DS** Spindle rotation direction (CW, CCW)

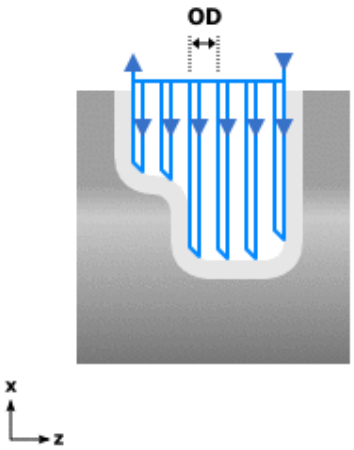
Groove Turning



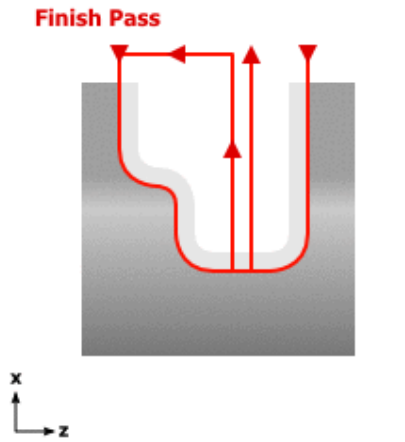
Groove Peck Depth and Retract



Groove Overlap Distance



Groove Turning



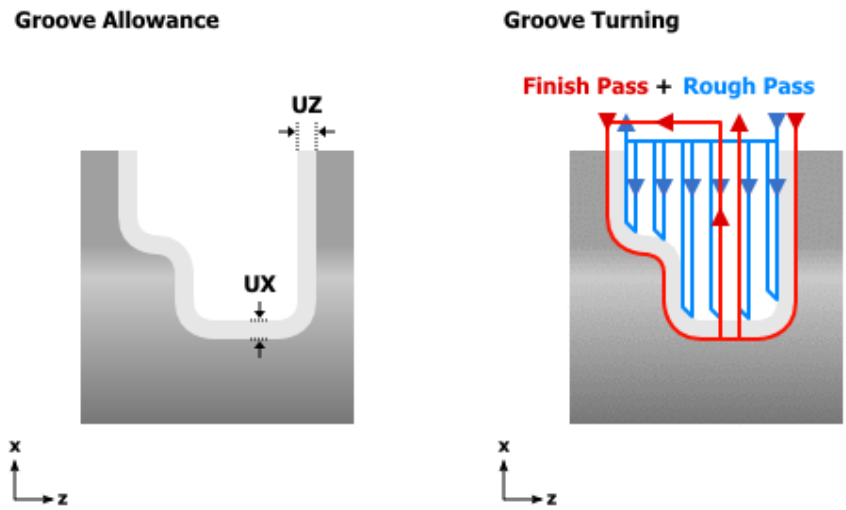


Figure 15.8-3: Irregular Groove

15.9 Cut-Off

The Cut-Off event performs a part-off process to detach the workpiece from its original stock. This is generally the last event in a machining process and is commonly used in conjunction with a parts catcher or bar feeder device.

Parameters:

- **T** Tool ID
- **S** Spindle speed (RPM)
- **F** Feed rate (mm or inch, per min or per rev)
- **X0** Start position X (absolute coordinate)
- **Z0** Start position Z (absolute coordinate)
- **X1** Depth for reduced speed (absolute coordinate or relative diameter distance from X0)
- **FR** Reduced feed rate (mm or inch, per rev)
- **SR** Reduced spindle speed (RPM)
- **X2** Final depth (absolute coordinate or relative diameter distance from X0)
- **ACT** Corner blending (None, Chamfer, or Fillet)
- **AR** Corner fillet radius (mm or inch)
- **AD** Corner chamfer distance (mm or inch)
- **OP** Stop Operation (None, Stop)
- **CL** Coolant (On/Off)
- **DS** Spindle rotation direction (CW, CCW)

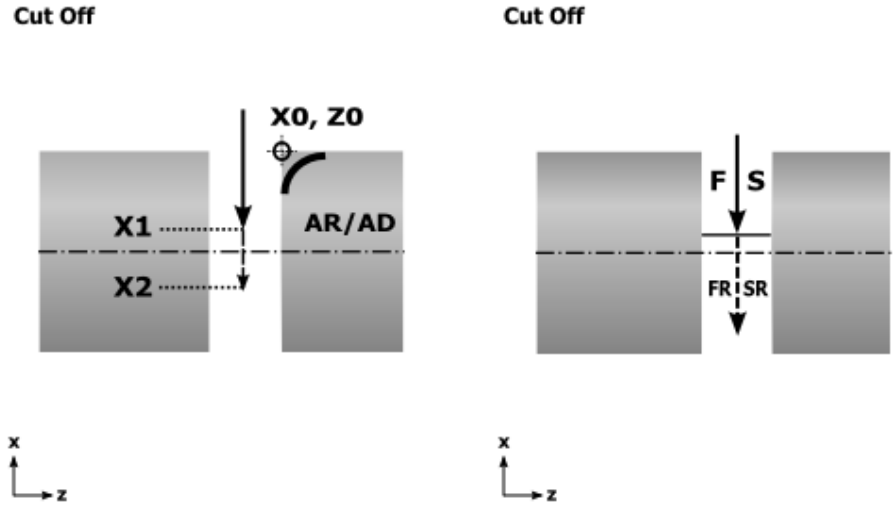


Figure 15.9-1: Cut-Off

15.10 Line-Arc

15.10.1 Linear (G01)

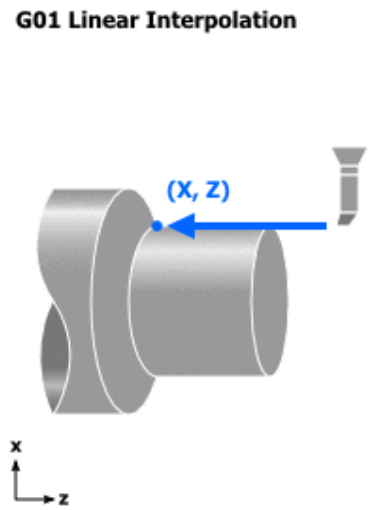


Figure 15.10-1: G01 Linear Interpolation

15.10.2 CW Arc (G02)

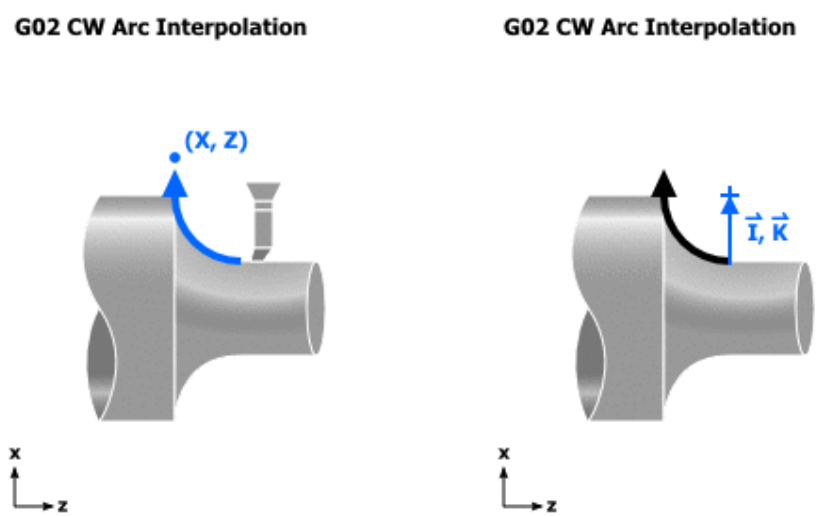


Figure 15.10-2: G02 CW Arc Interpolation

15.10.3 CCW Arc (G03)

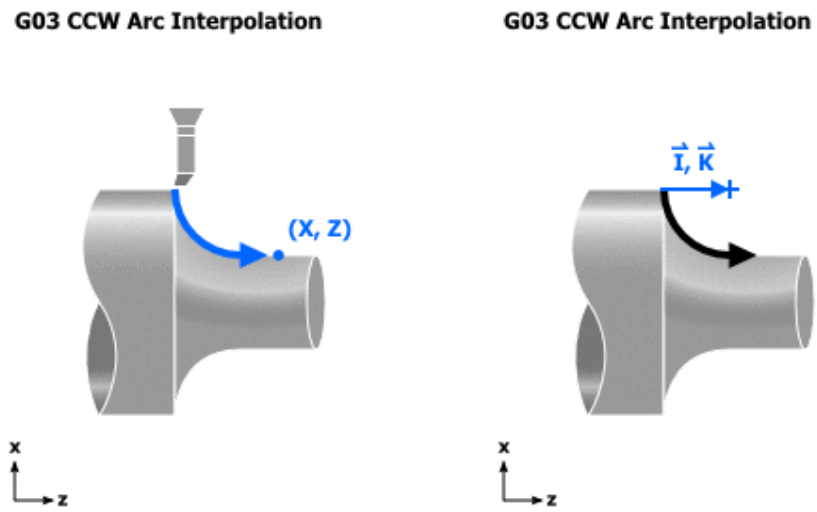


Figure 15.10-3: G03 CCW Arc Interpolation

15.11 Position

15.11.1 Rapid XZ (G00)

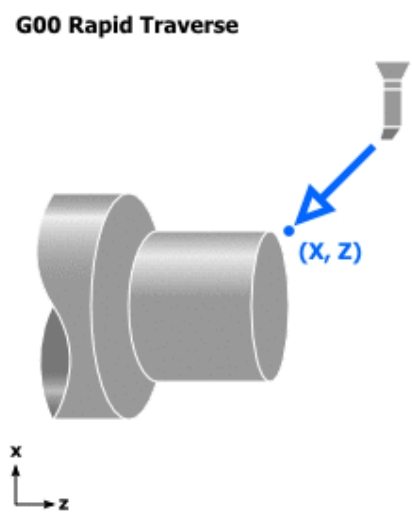


Figure 15.11-1: G00 Rapid Position

15.12 Auxiliary

15.12.1 Program Stop

The Program Stop event commands a few different types of program stops. The parameters for the Program Stop event are detailed in the following list.

- **M00 Compulsory Stop**, program will stop and require a CYCLE START press to continue.
- **M01 Optional Stop**, if M01 button is engaged, program will stop and require a CYCLE START press to continue. If M01 button is not engaged, program will run without stopping.
- **M30 Program Reset**, program will finish here.
- **M99 Program Automatic Restart**, program will jump from this block to the beginning and automatically run repeatedly.

15.12.2 Coolant

The Coolant Control event commands a coolant on or off action. The parameters for the Coolant Control event are detailed in the following list.

- **M** Coolant function (M8 Flood, or M9 Off)

15.12.3 Dwell

The Dwell event commands a pause of a specified duration. The parameters for the Dwell event are detailed in the following list.

- **X** Dwell time (seconds)

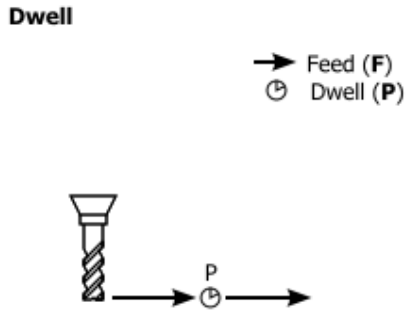
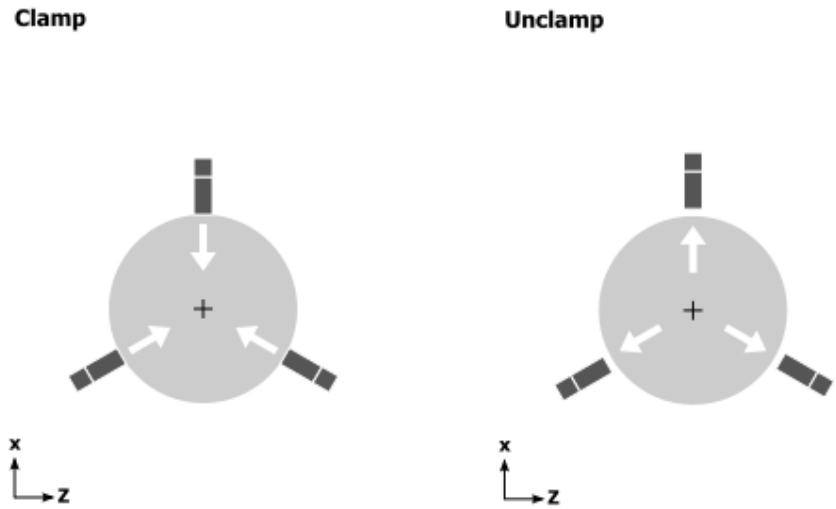


Figure 15.12.3: Dwell Event

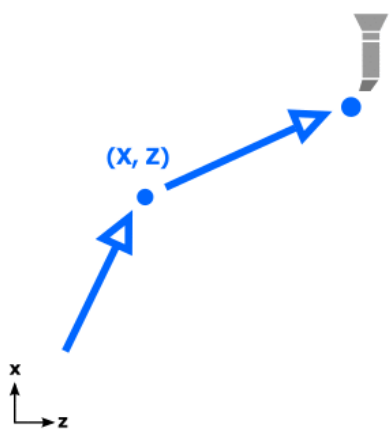
15.12.4 Tool Change

15.12.5 Tool Radius Compensation

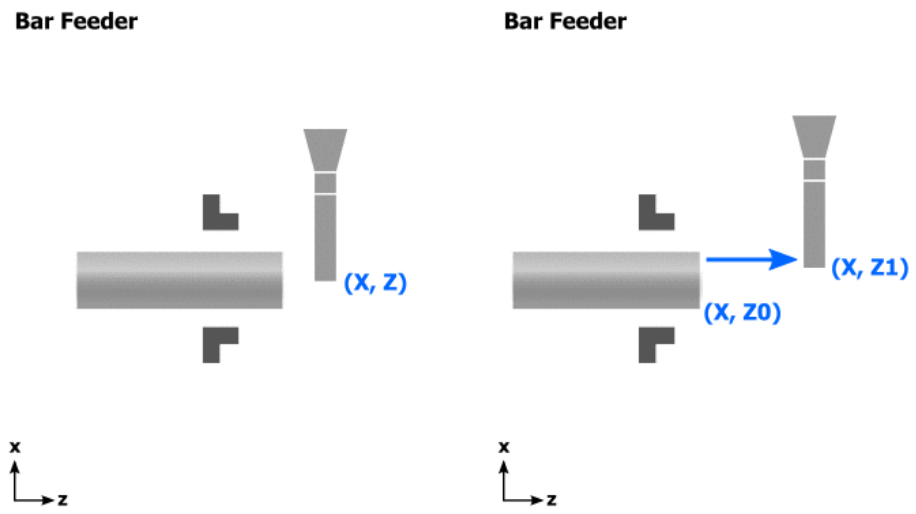
15.12.6 Chuck Clamp/Unclamp



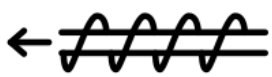
15.12.7 Reference Point Return



15.12.8 Bar Feed



15.12.9 Chip Auger



15.12.10 Comment

The Comment event allows insertion of a custom comment into the Conversational Event list.

15.12.11 G-Code

The G-Code event allows insertion of standard G-Code into the Conversational Program. The parameters are detailed in the following list.

- **SNC** Standard G-Code to execute, each block separated by a semi-colon ";". (*Note*: Must not end with an empty line.)

16 DRAWING IMPORT

16.1 Introduction

In the Conversational Editor, an Import Drawing module allows opening of DXF files representing CAD drawings directly on the control. After a drawing is opened, functions are provided to perform scaling, rotation, setting zero, editing, measurement, and selection of geometry to import. Imported geometry can then be added as specific conversational events depending on the type of geometry. This chapter provides documentation for using the Import Drawing module.

16.2 Drawing Import Workflow

The Drawing Import module allows the Conversational Editor to open or create DXF drawings and import its geometry as machining events. The general operation procedure is outlined as follows:

1. From the Conversational Editor main screen, select **DRAWING IMPORT**.
2. On the Drawing Import Screen in, select **OPEN DRAWING**.
3. Choose a DXF file to open.
4. Chose from the **DRAWING MODES** to perform actions on the drawing:
    a. **FILE** mode allows reloading, saving, and creating a new drawing.
    b. **SET ZERO** mode allows set up of the part zero location.
    c. **SELECT** mode allows the selection of geometry to be imported.
    d. **VIEW** mode allows scaling, rotating, or mirroring the entire drawing.
    e. **MEASURE** mode allows labelling of dimensions on the drawing.
    f. **DRAW** mode allows creating lines, arcs, circles, rectangles, and modifying geometry within the drawing.
    g. **EDIT** mode allows creating fillets, chamfers, copying, rotating, mirroring, and moving elements within the drawing.
5. When part zero has been set and geometry has been selected for import, select **IMPORT HOLES**, **IMPORT CONTOUR**, **IMPORT POCKET**, or **IMPORT OTHER** to import the selected geometry as Conversational Events.

Each drawing mode and function is covered in more detail in the subsequent sections.

16.3 Drawing Import Main Screen

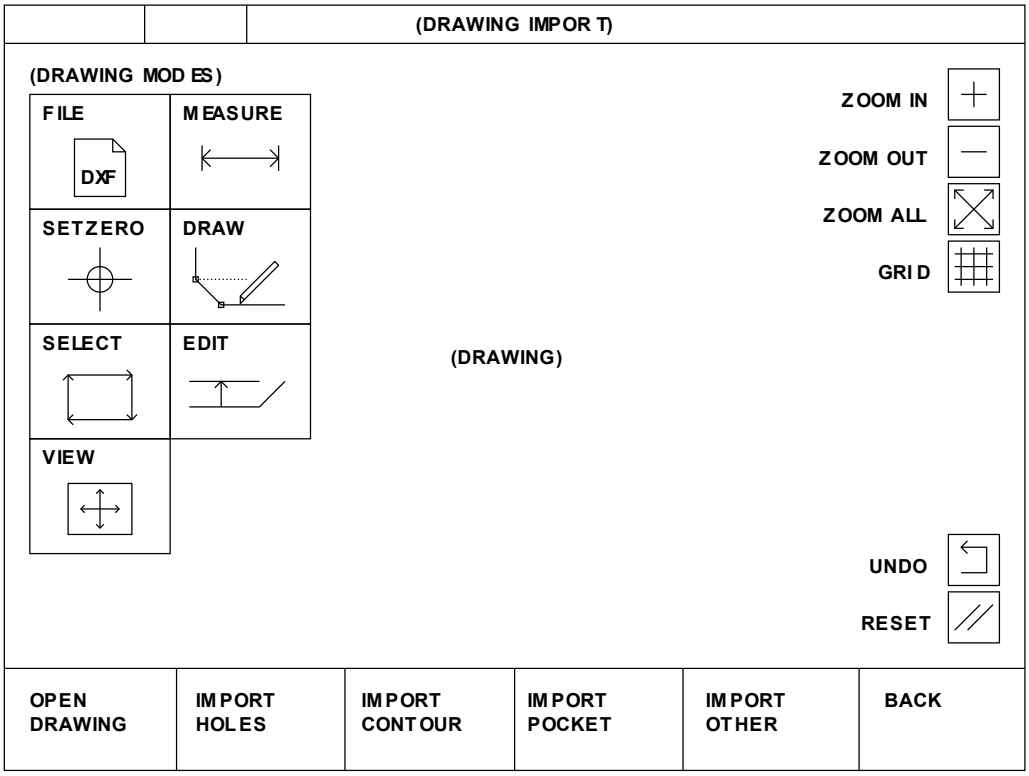


Figure 16.3-1: Drawing Import Main Screen

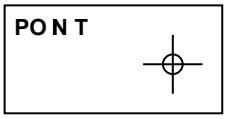
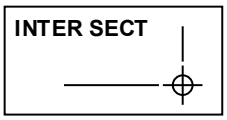
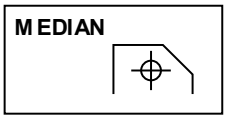
16.4 File Mode and Functions

Table 16.4.1: File Functions

Function	Description
FILE DXF	The **FILE** mode provides file management related functions.
NEW	**NEW FILE** function creates a new drawing with the specified filename.
RELOAD	**RELOAD** function gives up any changes to the current drawing and reloads the drawing file.
SAVE	**SAVE** function will save any changes made to the drawing, modifying the original drawing file.
SAVE AS	**SAVE AS** function allows saving the current drawing as a new drawing with the specified filename.

16.5 Set Zero Mode and Functions

Table 16.5.1 : Set Part Zero Functions

Function	Description
	In the **SET ZERO** mode, the part zero for the drawing is set. On the drawing display, the large blue crosshair denotes the current part zero location. This is also the origin from which all drawing coordinates, measurements, and selected geometry are based upon.
	**SET ZERO TO POINT** function allows selecting any segment start, end, mid, quadrant, or key-point to become the new part zero location.
	**SET ZERO TO INTERSECT** function allows the finding of intersections to set part zero. First select two segments to find any intersection(s), then choose an intersection to set the new part zero to.
	**SET ZERO TO MEDIAN** function allows finding of the median among multiple selected segments. Begin by choosing segments from which to find the median of. When the desired selection set is complete, click on the calculated median point to set the part zero to.

16.6 Select Mode and Functions

Table 16.6.1: Drawing Select Functions

Function	Description
SELECT	In the **SELECT** mode, geometry from the drawing can be picked out and selected for import as events.
CIRC LES	**CIRCLES** select mode enables selection of circular features in the drawing for import as events. Circles can be imported as hole events (drill, bore, tap), contour events, or pocket events.
	BATCH SELECT function is a toggle that enables drag selecting multiple circles from the drawing. When active, press down on any start point on the screen, then drag select a rectangular area, and finally release to finalize the selection region. Upon release, any circular feature found within the region will be grouped and presented on the screen as blocks, with each block indicating the diameter and number of circles found. A group block must be selected. Choosing the last block will select all. Once selected, the circular features will be numbered in order and displayed with a different color. They are then ready for import. More features may be added to the current selection by repeating the process again.
Z0	**SET Z ZERO** function initiates a process where the depths of previously selected circular features can be imported from the drawing. To use this function, a side view of the holes indicating hole depths must be drawn. Once this function is pressed, proceed to select a start depth location for the hole(s). When complete, a Z0 line will indicate the start depth location.
Z	**SET Z** function allows setting of end depths when the above **SET Z ZERO** function has been completed. To set the end depth, first select a target depth location on the drawing. Lines will then be displayed indicating the start and end depths for each hole. The end depths for each hole can then be modified by selecting on the end depth symbol and then selecting another target depth for each circular feature. When all the start and end depths are satisfactory, proceed to import the circular geometry as hole events.

AR C	**SELECT ARC** toggle allows the selection of arcs instead of circles within the drawing.
SEG MEN TS	**SELECT SEGMENTS** mode allows line or arc segments to be selected from the drawing for import as paths. Use this selection mode to choose individual segments, build multiple segment open paths, or build multiple segment closed paths. The selected paths may be imported as contour or pocket events.
	SELECT NEW PATH toggle finishes the current path and initiates a new path to be selected.
	SELECT INDIVIDUAL SEGMENT mode enables single line or arc segments to be selected for import as paths.
	SELECT MULTIPLE SEGMENT OPEN PATH mode enables building up an open path by chaining individual segments. When this mode is active, proceed to choose any segment, then select its direction, and it will be joined to the previous segment automatically.
	SELECT MULTIPLE SEGMENT CLOSED PATH mode enables building up a closed path by chaining individual segments. When this mode is active, proceed to choose any segment, then select its direction, and it will be joined to the previous segment automatically. Pressing the SELECT NEW PATH button will automatically join the start and end segments to close the path.
PATHS	**SELECT PATHS** mode allows the selection of closed paths from the drawing. Upon toggling this mode, proceed to choose a start segment, then select its direction, and then this function will exhaustively search for any adjoining segments within the drawing to add to the current path. When branches occur, choose the desired branch arrow to continue. If the start segment is re-encountered, the complete closed path will be selected. If a dead end segment is encountered, an open path will be selected. The selected paths may be imported as contour or pocket events.
	OFFSET SELECTED PATH function allows offsetting the currently selected path(s). When this function is activated, enter a desired offset distance in the field above, then choose a direction to perform an offset on the entire path.

16.7 View Mode and Functions

Table 16.7.1: Drawing View Functions

Function	Description
VIEW	In the **VIEW SETUP** mode, a drawing scale, orientation, and mirror about X or Y can be applied. These functions can be used to modify the drawing for imported geometry and its associated events to be dimensionally correct.
SCA LE	**SCALE** function will resize the drawing to a specified scale factor upon entry of a value. The default scale factor is one, indicating no scaling is applied. A scale factor of less than one will reduce the drawing, and a scale factor of larger than one will enlarge the drawing. It is useful to use the length measurement functions to verify the dimensions of the drawing after scaling is applied.
ORIENT	**ORIENT** function applies a rotation angle to the drawing. A positive angle will rotate the drawing counter-clockwise, while a negative angle will rotate the drawing clockwise. The rotation origin is the current part zero.
MR X	**MIRROR X** function applies a mirror action about the X-axis running through the part zero location.
MR Y	**MIRROR Y** function applies a mirror action about the Y-axis running through the part zero location.

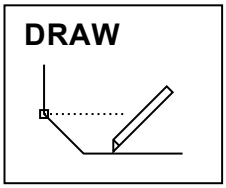
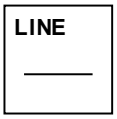
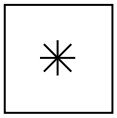
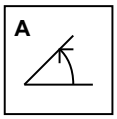
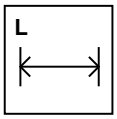
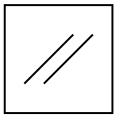
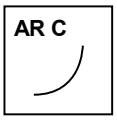
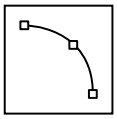
16.8 Measure Mode and Functions

Table 16.8.1: Drawing Measurement Functions

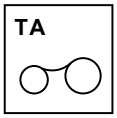
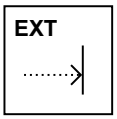
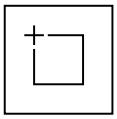
Function	Description
MEASURE	**MEASURE** mode offers functions to label dimensions on the drawing for reference or verification.
L	**MEASURE LENGTH** function allows the selection of two points, a point and a line, or two parallel lines to display the distance between the selected features.
D	**MEASURE DIAMETER** function allows the selection of any circle or arc to display its diameter.
R	**MEASURE RADIUS** function allows the selection of any circle or arc to display its radius.
A	**MEASURE ANGLE** function allows the selection of three points or two lines to display the angle between the features.
XY	**MEASURE XY** function allows the selection of any point or key point to label its current X or Y location relative to the current PART ZERO origin.

16.9 Draw Mode and Functions

Table 16.9.1: Draw Functions

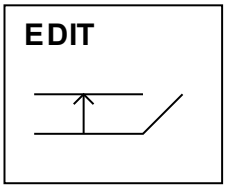
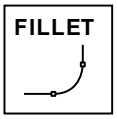
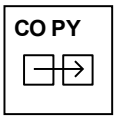
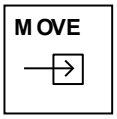
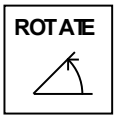
Function	Description
	The **DRAW MODE** allows creating geometry within the drawing. Use this mode to add supplemental features on a drawing or to create a part drawing from scratch.
	**DRAW LINE** mode allows creating lines within the drawing. Once activated, click on a start point and an end point to draw a line. Alternatively, X and Y positions can be entered to specify the start and end points. Once a line segment is started, continue to input additional points to continue from the previous end point.
	**NEW LINE** ends the current path and the next input point will be the start point of a new line.
	**FIXED ANGLE** function accepts an angle input and when active, draws the next line segment such that the new segment is at an angle with respect to the previous line segment. Positive angles will offset the new line in a counter-clockwise direction, while negative angles will offset in the clockwise direction relative to the previous segment. For new lines, the previous segment is by default a line in the +X direction.
	**FIXED LENGTH** function accepts a length input and when active, will draw the next line segment to the specified length.
	**RESET FIXED ANGLE AND LENGTH** function will release the FIXED ANGLE and FIXED LENGTH criteria for drawing line segments.
	**DRAW ARC** mode allows creating circular acts within the drawing.
	**DRAW THREE POINT ARC** function creates an arc by selecting or inputting first the start point, then a through point, and lastly the end point of the arc segment.

	DRAW ARC START + CENTER + END function creates an arc by selecting or inputting first the start point, then the center point, and lastly the end point of the arc segment. The arc will be drawn in a counter-clockwise direction.
CIRCLE	**DRAW CIRCLE** mode allows drawing circles in the drawing.
D	**DRAW CIRCLE CENTER + DIAMETER** function creates a circle by first specifying its diameter, and then selecting or inputting the center location.
	DRAW CIRCLE CENTER + POINT function creates a circle first by selecting or inputting its center location, and then a point on its circumference.
	DRAW CIRCLE BY TWO POINTS function creates a circle by selecting or inputting two individual point locations on its circumference. The distance between the two specified points is the diameter of the circle.
	DRAW CIRCLE BY THREE POINTS function creates a circle by selecting or inputting three individual point locations on its circumference.
REC T	**DRAW RECTANGLE** mode allows drawing rectangular geometry in the drawing.
	DRAW RECTANGLE BY TWO POINTS function creates a rectangle by selecting or inputting two individual point locations denoting the opposite corners of the rectangle.
	DRAW RECTANGLE WIDTH + LENGTH + CENTER function creates a rectangle by first specifying its width and length, and then selecting its center location.
TRIM	**TRIM** function performs partial deletion of the selected segment if any other line, arc, or circular segment intersects with the selected segment.
JOIN	**JOIN** mode and function allows the joining of two selected segments if any existing or potential intersection exists.
TL	**TANGENT LINE** function creates a tangent line between two selected arcs or circles. First select two arcs or circles, then choose the desired tangent line to draw it.

	**TANGENT ARC** function creates a tangent arc of specified radius between two selected arcs or circles. First input the desired tangent arc radius, then select two arcs or circles, and if any tangent arcs exist, choose the desired one to draw it.
	**EXTEND** segment function drawings extensions of segments. First select a segment to extend, then select another segment to extend to.
	DELETE segment function removes selected segments from the drawing.
	**DELETE BATCH** function allows drag selecting a region to batch delete segments within the selection region. When the selection region is drag selected from the left to right, only the segments entirely contained within the region will be deleted. When the selection region is drag selected from right to left, then any segment partially within the selection box will also be deleted.

16.10 Edit Mode and Functions

Table 16.10.1: Drawing Edit Functions

Function	Description
	The **DRAWING EDIT** mode contains editing functions applicable to existing geometry within the drawing.
	**FILLET** function applies a fillet, or rounded corner, to two lines. First specify a fillet radius, then choose any two lines and the fillet will be applied where the two lines intersect.
	CHAMFER function applies a chamfer, or angled corner, to two lines. First specify a primary chamfer distance D1, then either a secondary chamfer distance D2 or an angle A, depending on the type of chamfer desired. Then choose two lines to apply the chamfer to its intersection corner. When D1 and D2 are specified, the first segment will be recessed a distance D1, and the second segment will be recessed a distance D2 from the intersection point. When D1 and A are specified, the first segment will be recessed a distance D1, and the angled corner will be an angle of A degrees offset from the first segment.
	**COPY** function applies a copy to the selected geometry. First select the geometry to copy, then input the X and Y displacement values, and lastly press apply to perform the copy. Use the batch select toggle to select a region to copy.
	OFFSET function performs an offset to any segment. First select a line, arc, or circle segment to offset, then input an offset distance, and finally select a direction to offset the segment.
	**MOVE** function performs a move to the selected geometry. First select the geometry to move, then input the X and Y displacement values, and lastly press apply to perform the move. Use the batch select toggle to select a region to move.
	**ROTATE** function performs a rotation on the selected geometry. First choose the geometry to rotate, then enter an angle in degrees, and lastly press apply to perform the rotation.

O	**ROTATE ORIGIN** function sets a point as the rotation center for the rotate function. Once toggled, select or input a XY location to set the rotation center to. Once set, toggle the function button again to return to the parent ROTATE function.
	BATCH SELECT toggle enables batch selection of geometry for rotation.
C	**COPY** toggle allows the rotated geometry to be copied upon each applied rotation.
MIR	**MIRROR** function creates mirrors of the selected geometry. First select the geometry to mirror, then use the MIRROR AXIS function to set the mirror axis location.
	BATCH SELECT toggle enables batch selection of geometry for mirroring.
A	**MIRROR AXIS** function sets the location of the mirror axis and immediately applies the mirror function to the selected geometry.
C	**COPY** toggle allows the mirrored geometry to be copied upon each applied mirror action.

17 SYSTEM

17.1 System Introduction

The SYSTEM screens allow system related tasks such as parameter modification, maintenance of the machine, viewing message logs, and viewing the system environment information.

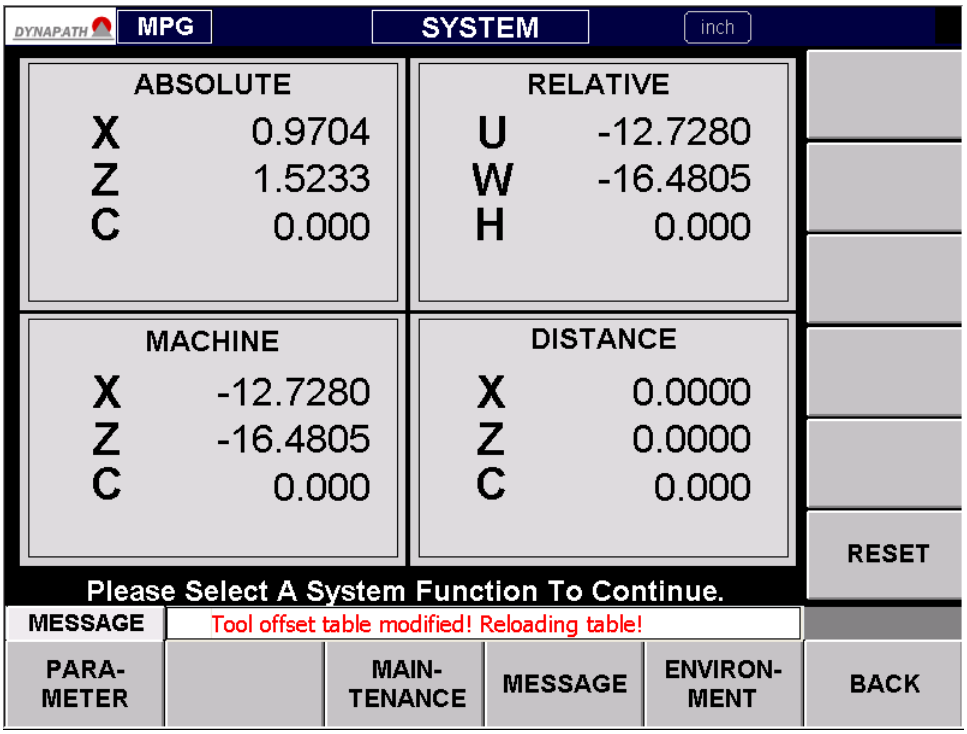


Figure 17.1-1: System Main Screen

17.2 Parameters

Parameters of the control system can be modified in the PARAMETERS screens. The default login is “SI” and password is “SI”. Parameter modifications require a SAVE action to be performed. The usage for each parameter section are listed as follows:

- **USER** parameters can be set by the user.
- **MACHINE** parameters are set by the system integrator or machine builder.
- **CONTROL** parameters are set by the control provider.
- **SYS CONFIG** parameters are set by the solution provider.

Note that motion related parameters generally require a reboot to take effect.

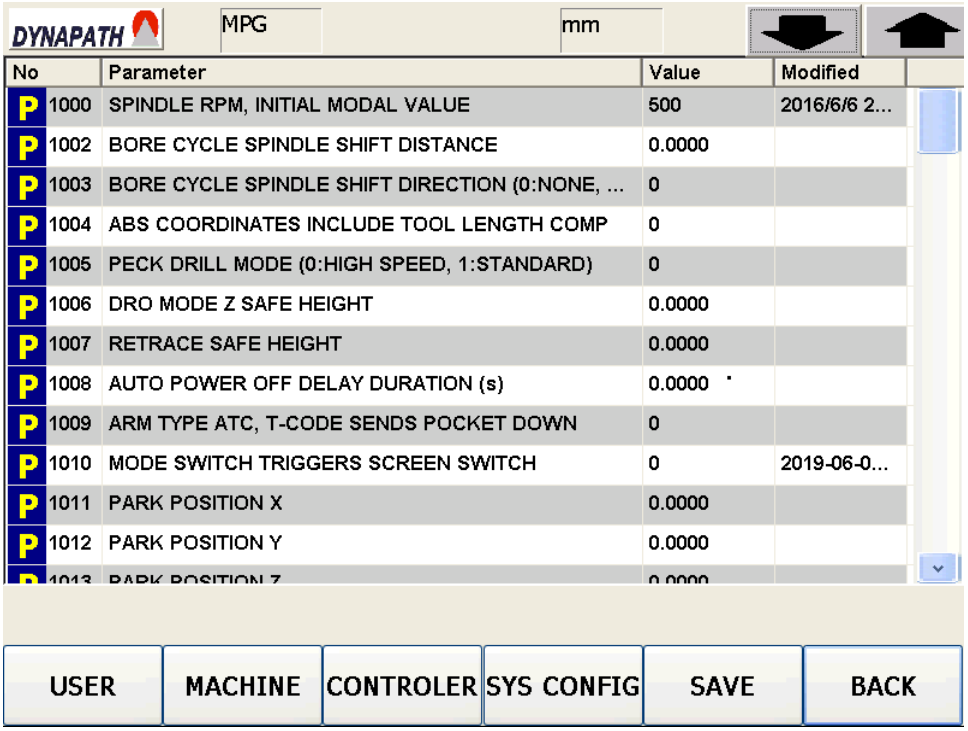


Figure 17.2-1: System Parameters Screen

17.3 Maintenance

The MAINTENANCE screen provides troubleshooting and diagnosis tools for tracing issues.

17.4 Message

The MESSAGE screen lists current and historical alarm messages.

17.5 Environment

The ENVIRONMENT screen displays the current software revision information, IP address, and allows setting for language, date, and time for the control system.

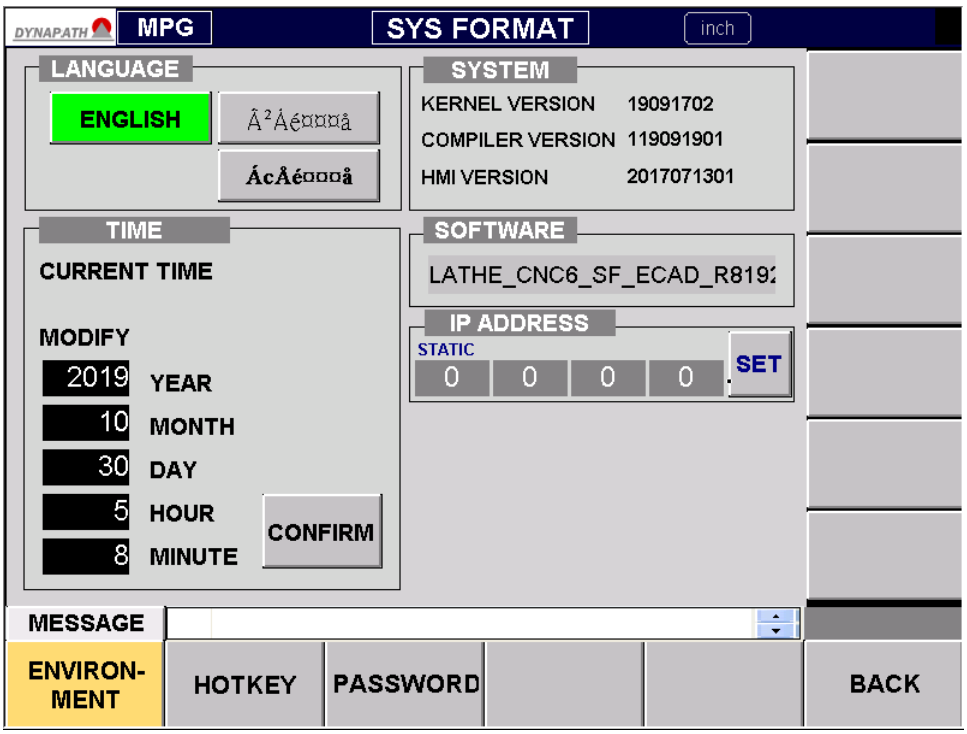


Figure 17.5-1: System Environment Screen

18 NETWORKING

18.1 Introduction to Networking

The DynaPath WinDelta CNC can connect to a local network using its built-in LAN port. Connecting to a network enables networked part program management, remote diagnostics, and remote monitoring. Additionally, when assistance is needed, Remote Assist can be requested for support personnel to view the control remotely and assist with any task. This chapter provides detailed guides on how to set up and use these network functions.

18.2 Networking Set Up

The control is set by default from the factory to automatically inherit an IP address that is dynamically assigned by the gateway or a router on the local area network. The control can also be assigned a specific IP address. Reference the following procedure to perform manual network set up:

To set IP address:

1. Plug in a LAN cable into the RJ-45 network port on the machine or control.
2. Boot up the control, then click the ABORT button that shows momentarily on the screen while loading.
3. Login with “SI”, password “SI”.
4. Using a USB keyboard, press the Windows key and go to SETTINGS, and NETWORK.
5. Double click on the active LAN network connection icon to open its settings. It should be labelled “PCI-E…”
6. The IP address of this control can be set one of two ways:
    a. **Obtain an IP address via DHCP** – choosing this will allow the control to inherit an IP address that is dynamically assigned by the gateway or router on the network.
    b. **Specify an IP address** – choosing this will allow manually assigning an IP address, subnet mask, and default gateway for the control.
7. When complete, click OK.

To verify the control is connected to the network:

1. Press the Windows key on the keyboard to bring up the start menu.
2. Click on Run.
3. Enter “CMD” to initiate a command prompt.
4. Type in “IPCONFIG” to display the current IP address and network settings of the control.

To return to the normal CNC operation screens, cycle power to the machine and let the control boot up normally.

18.3 Remote File Management

The WinDelta CNC has a built-in FTP server for remote file management from another computer on the same local area network. The control's program folder can be accessed using the following methods:

Using Windows Explorer on Windows PCs:

1. Open Windows Explorer on your PC.
2. In the address field, type in: ftp://<IP_ADDRESS>/, replacing the IP_ADDRESS string with the actual IP address of the control.
3. Depending on the version of the control, navigate to the following folder to view the current files on the control:
    a. /User/GCode/
    b. /RuntimePrj/Mill_CNC4/Data/GCode
    c. /UserGCode/
    d. /Hurco/WinPC32/Data/GCode/
4. Copy files into or from the folder on the control.
5. On the control, re-enter the FILE MANAGER, or press the REFRESH button to update the file list.

Using an FTP Client Application, e.g., FileZilla:

1. After installing the FTP Client, set up a new FTP Server Site to connect to using the following settings:
    Host: IP Address of the control, e.g. 192.168.1.100
    Login Type: Anonymous
    Encryption: Use plain FTP
2. Connect to the server.
3. Manage files on the server.
4. Disconnect or close the application when complete.

****Note: Certain files on the control should not be deleted. These files are generally named O9*** and are macro programs used for tool change, tool setting, or probing.***

18.4 Remote Assistance

Requesting remote assistance for remote diagnostics, troubleshooting, support, updates, or additional training can be initiated by the following procedure.

1. Make sure that the machine is set up on the local area network and has a known IP address. Refer to the previous networking set up section to acquire the IP address of the control.
2. Run a remote application (for example AnyDesk or Remote Utilities) on a separate PC or laptop that has access to the internet. This device should also be locally connected to the machine either over the local area network (LAN) or straight via a network cable.
3. Notify a DynaPath authorized remote support engineer and provide the remote application ID for them to link into.

4. Once connected to the PC, the remote support engineer will be able to access the control on the machine and perform remote assistance.

19 Alarms and Messages

19.1 Motion Module Alarms

Motion module alarms are displayed on the control screen when abnormalities are encountered during operation of axes drives and motors. Motion alarms and errors are documented in the following table.

Table 19.1.1: Motion Module Alarm Messages

Alarm	Description
MOT_1201	X servo axis reports abnormality (error)
MOT_1202	Y servo axis report abnormality (error)
MOT_1203	Z servo axis report abnormality (error)
MOT_1204	C servo axis reports abnormality (error)
MOT_1207	S axis report abnormality (error)
MOT_1209	CNCIO reports an exception (error)
MOT_1211	No response from X axis comms (0x0f Discontinuous frame count)
MOT_1212	No response from Y axis comms (0x0f Discontinuous frame count)
MOT_1213	No response from Z axis comms (0x0f Discontinuous frame count)
MOT_1214	No response from C axis comms (0x0f Discontinuous frame count)
MOT_1217	No response from S axis comms (0x0f Discontinuous frame count)
MOT_1219	No response from CNCIO comms (0x0f Discontinuous frame count)
MOT_1221	X servo axis has no PWR-ON signal (Abnormal power supply)
MOT_1222	Y servo axis has no PWR-ON signal (Abnormal power supply)
MOT_1223	Z servo axis has no PWR-ON signal (Abnormal power supply)
MOT_1224	C servo axis has no PWR-ON signal (Abnormal power supply)
MOT_1227	No PWR-ON signal on S axis (Abnormal power supply)
MOT_1231	X servo axis SON signal disappears abnormally (Unstable power supply 0x0e)
MOT_1232	Y servo axis SON signal disappears abnormally (Unstable power supply 0x0e)
MOT_1233	Z servo axis SON signal disappears abnormally (Unstable power supply 0x0e)
MOT_1234	C servo axis SON signal disappears abnormally (Unstable power supply 0x0e)
MOT_1237	S axis SON signal disappears abnormally (Unstable power supply 0x0e)
MOT_1241	X servo axis timed out and is not ready (Unable to servo ON)
MOT_1242	Y servo axis timed out and is not ready (Unable to servo ON)
MOT_1243	Z servo axis timed out and is not ready (Unable to servo ON)
MOT_1244	C servo axis is not ready after timeout (Unable to servo ON)
MOT_1247	S axis is not ready yet (Unable to servo ON)
MOT_1251	X servo axis communication data reading abnormality (0x0d)
MOT_1252	Y servo axis communication data reading abnormality (0x0d)
MOT_1253	Z servo axis communication data reading abnormality (0x0d)
MOT_1254	C servo axis communication data reading abnormality (0x0d)
MOT_1257	S axis communication data reading abnormality (0x0d)