

CNC-MILL-OP01-EN



WinDelta Mill Control

Control, Operating, and Programming Manual

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	D5 Console	26
3.1.4	D2 Console	27
3.1.5	D6 Console	28
3.1.6	D11 Console.....	29
3.1.7	Handheld MPG	30
3.1.8	Electronic Handle / Handwheel	30
3.2	Software Components	31
3.2.1	Software Licenses and Agreement	31
4	BASIC OPERATION.....	33
4.2	Introduction to Operation of CNC Machine Tool.....	34
4.3	Pre-Startup Inspection	34
4.4	Power-On and Start-Up Procedure	34
4.5	Power-Off and Shutdown Procedure	35
4.6	Emergency Stop.....	36
4.7	Over-travel (OT) and Recovery.....	36
4.7.1	Hardware Over-travel	36
4.7.2	Software Over-travel	36
4.8	CNC Modes and Panel Operation.....	37
4.8.1	Mode Screen Style Operation	38
4.8.2	Mode Switch Style Operation.....	40
4.9	Coordinate Systems	41

4.10	Work Offset Concepts	42
4.11	Tool Offset Concepts	42
5	HOME MODE	43
5.1	Home Mode Screen.....	44
6	MANUAL MODES	45
6.1	Jog Mode	46
6.2	MPG Mode	47
6.3	Incremental Jog Mode	48
6.4	Spindle Control	49
6.5	Panel Functions.....	50
7	AUTO MODE	51
7.1	Basic Operation	52
7.1.1	Cycle Start	52
7.1.2	Feed Hold	52
7.1.3	Spindle, Rapid, Feed Overrides.....	52
7.2	File Manager	52
7.3	Single Block	53
7.4	MPG Run	53
7.5	Retrace	53
7.6	Reset.....	53
7.7	Panel.....	54
8	WORK OFFSET.....	55
8.1	Offset Concepts.....	56
8.2	Work Offset.....	57
8.2.1	Work Offset – Set to Position	58
8.3.1	Work Offset – Align to Rectangular Center.....	59
8.3.2	Work Offset – Align to Circle Center	60
9	TOOL OFFSET	61
9.1	Tool Offset Screen.....	62
9.1.1	Call Tool Function	62
9.1.2	Tool Length Measurement	63
9.2	Tool Length Offset Method A – Reference Machine Zero	64
9.3	Tool Length Offset Method B – Reference a Tool	65
10	DRO / SEMI-AUTO	66
10.1	Introduction.....	67

10.2	Manual Operation	67
11	MDI	68
11.1	MDI Operation	69
12	FILE MANAGEMENT	70
12.1	File Manager	71
12.2	Send and Receive	72
13	G-CODE PROGRAMMING	73
13.1	Program Language and Structure	74
13.2	G-Code Reference	75
13.3	M-Code Reference	79
14	MACRO PROGRAMMING	81
14.1	Introduction	82
14.2	Program Folders	82
14.3	Macro Program Names	82
14.4	Calling a Macro Program	82
14.5	Macro Variables	83
14.5.1	Macro Variables Overview	83
14.5.2	Local Input Variables	83
14.5.3	System Variables	84
14.6	Macro Functions	87
14.7	Priority of Operations	88
14.8	Bracket Nesting	89
14.9	Branch and Repetition	89
15	CONVERSATIONAL PROGRAMMING	91
15.1	Introduction	92
15.2	Conversational Programming Workflow	92
15.3	Main Screen Functions	93
15.4	Hole Sequence	95
15.4.1	G73 High Speed Peck Drill	98
15.4.2	G76 Dead Bore	99
15.4.3	G81 Drill Cycle	100
15.4.4	G82 Counter Bore	101
15.4.5	G83 Peck Drill	102
15.4.6	G85 Bore	103
15.4.7	G86 Dead Bore Fixed Spindle	104

15.4.8	G87 Back Bore.....	105
15.4.9	G88 Dead Spindle Bore and Dwell	106
15.4.10	G89 Bore Cycle with Dwell.....	107
15.5	Tap Events.....	108
15.5.1	G74 Left Hand Tap.....	108
15.5.2	G84 Right Hand Tap.....	109
15.6	Contour Group Events	110
15.6.1	Contour Event.....	110
15.6.2	Helix Event.....	113
15.6.3	Thread Mill Event	116
15.6.4	Chamfer Event.....	119
15.7	Pocket Events	121
15.7.1	Pocket Event	121
15.7.2	Tapered Face Event	123
15.7.3	Tapered Pocket Event.....	124
15.8	Face Mill Event.....	126
15.9	Position Events	128
15.9.1	G00 Rapid Move.....	128
15.9.2	G00 Rotate A-Axis.....	129
15.9.3	G00 Rotate B-Axis.....	130
15.9.4	G00 Rotate C-Axis.....	131
15.10	Pattern Events.....	132
15.10.1	Linear Array	132
15.10.2	Circular Array.....	133
15.10.3	Location Array	134
15.10.4	Rotate.....	135
15.11	Setup Events	136
15.11.1	Work Offset / P-Offset	136
15.11.2	Program Stop	136
15.11.3	Dwell.....	137
15.11.4	Inch/Metric Units	137
15.11.5	Coolant Control	138
15.11.6	Safe Height Override.....	138
15.11.7	Spindle Control.....	138
15.11.8	Repeat Work	138

15.11.9	G-Code	138
15.11.10	Comment	138
15.11.11	Engraving	139
16	CONVERSATIONAL CAD DRAWING EDITOR	140
16.1	Introduction	141
16.2	CAD Drawing Overview and Workflow	141
16.3	CAD Drawing Main Screen	142
16.4	File Mode and Functions	143
16.5	Set Zero Mode and Functions	144
16.6	Select Mode and Functions	145
16.7	View Mode and Functions	147
16.8	Measure Mode and Functions	148
16.9	Draw Mode and Functions	149
16.10	Edit Mode and Functions	152
17	SYSTEM	154
17.1	System Introduction	155
17.2	Parameters	156
17.3	Backlash Compensation	157
17.4	Lead Screw Compensation	158
17.5	Maintenance	159
17.6	Message	159
17.7	Environment	159
18	NETWORKING	160
18.1	Introduction to Networking	161
18.2	Networking Setup	161
18.3	Remote File Management	162
18.4	Connecting to a Windows Network Folder	163
18.5	Local Monitoring with VNC	163
18.5.1	VNC Server Set Up	164
18.6	Remote Assistance	164
19	Alarms and Messages	166
19.1	Motion Module Alarms	167

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. • Up to 10,000 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. • 16 GB 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 been trained in how a manual and CNC machine operates and has 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. Table 2.2.1 lists 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 machine and integrated control system 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 person responsible 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 responsibility 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 servo drives.

- Door guards should not open while the spindle is running. If equipped, the guard lock or bolt should be 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 2 m/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 with 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 to be undertaken. Never use damaged or worn tools and ensure the correct cutting parameters (speed, feed, and depth of cut) are used to prevent tool breakage and potential ejection of shrapnel.
- 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. Toggle the door guard release button, switch to safety mode, and open the table guards.
2. Hit the emergency stop button to disconnect all main power to the spindle and axes.
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 or appendage.
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

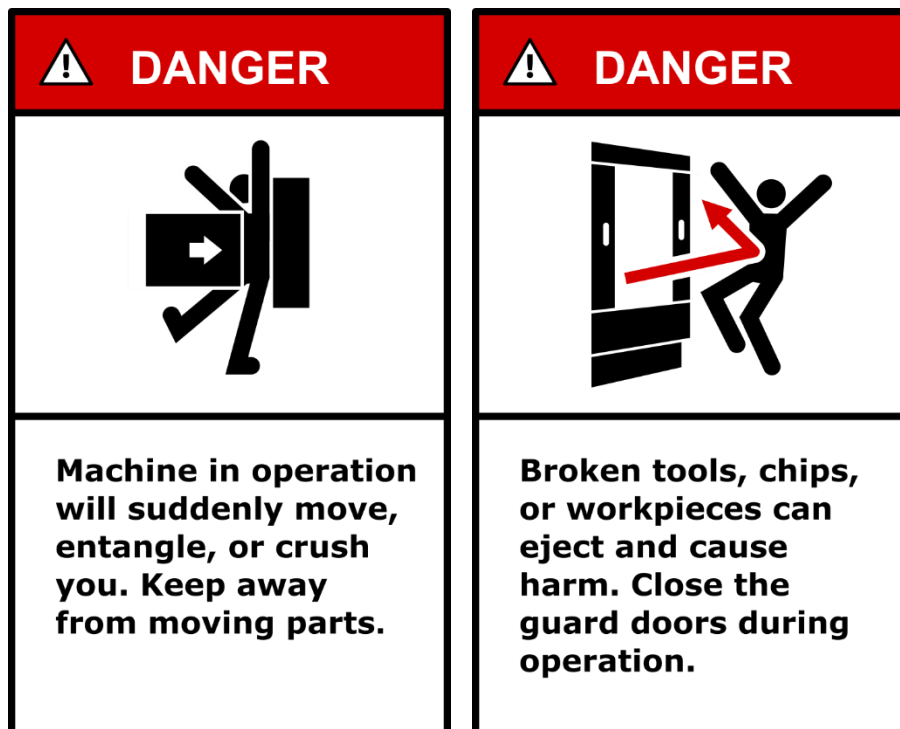


Figure 2.5-3: Safety labels for machine movement and debris

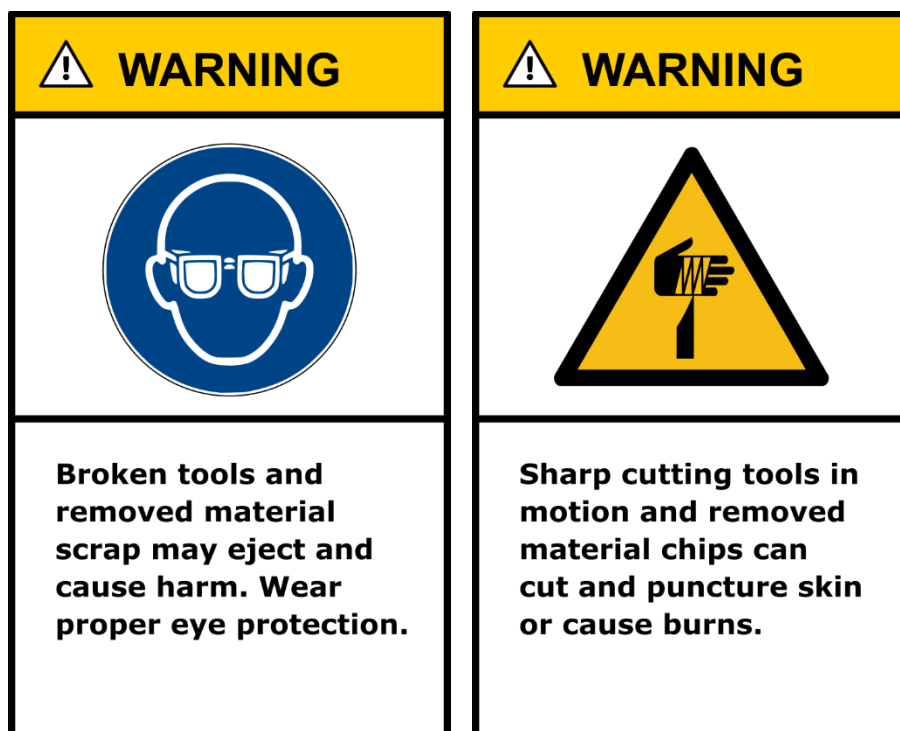


Figure 2.5-4: Safety labels for PPE and bodily injury

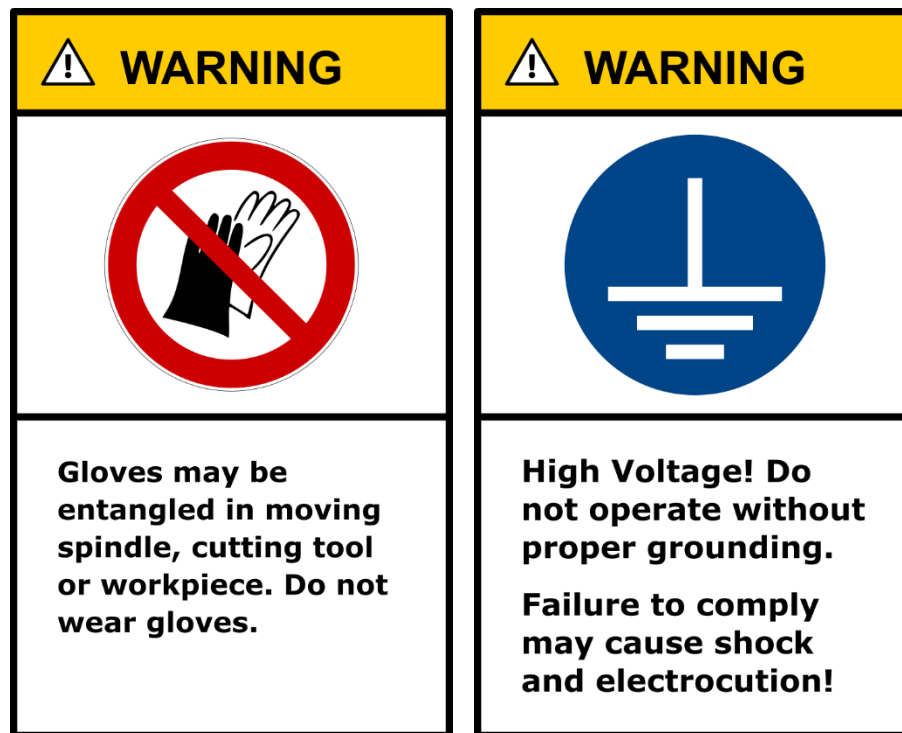


Figure 2.5-5: Safety labels for gloves and high voltage

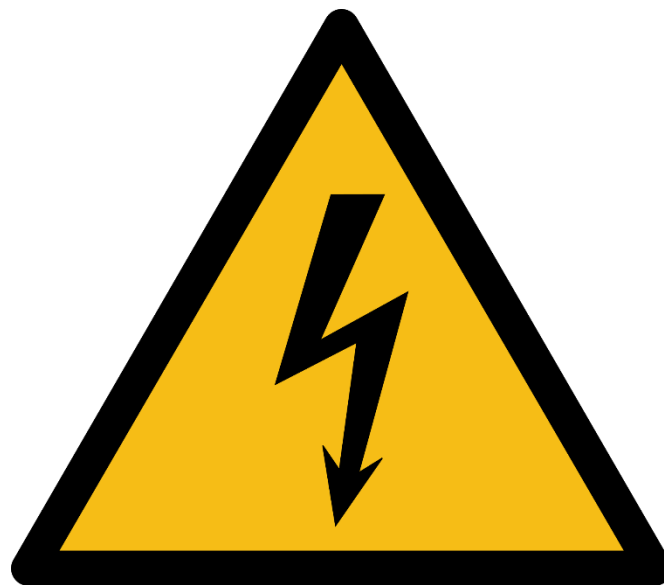


Figure 2.5-6: Safety label for electrical hazard

3 SYSTEM SPECIFICATIONS

3.1 Hardware Components

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

- 10.4", 12.1", 15.6", 19" LCD Display with industrial touchscreens.
- Tactile switch rugged keys.
- Manual data input (MDI) panel or USB Keyboard for data input.
- Console options: D1, D2, D5, D6, D11.
- Handheld Manual pulse generator (optional).
- Electronic handwheels (optional).
- External keyboard, trackpad, or mouse (optional).

All DynaPath WinDelta Mill CNC systems come equipped with a industrially rugged display with touchscreen capabilities. Depending on the type of machine and operation style, several console panel configurations exist. The MPG handwheel is standard and optional electronic handwheels can be installed for manual or semi-automatic operation. A USB keyboard, trackpad, or mouse can be installed as an option to assist with programming on 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 requirements and application.

The DynaPath WinDelta Node-355 control system chassis dimensions and ports are shown in Figure 3.1.1.1. This control system is generally attached to the rear of the console display as an all-in-one solution.

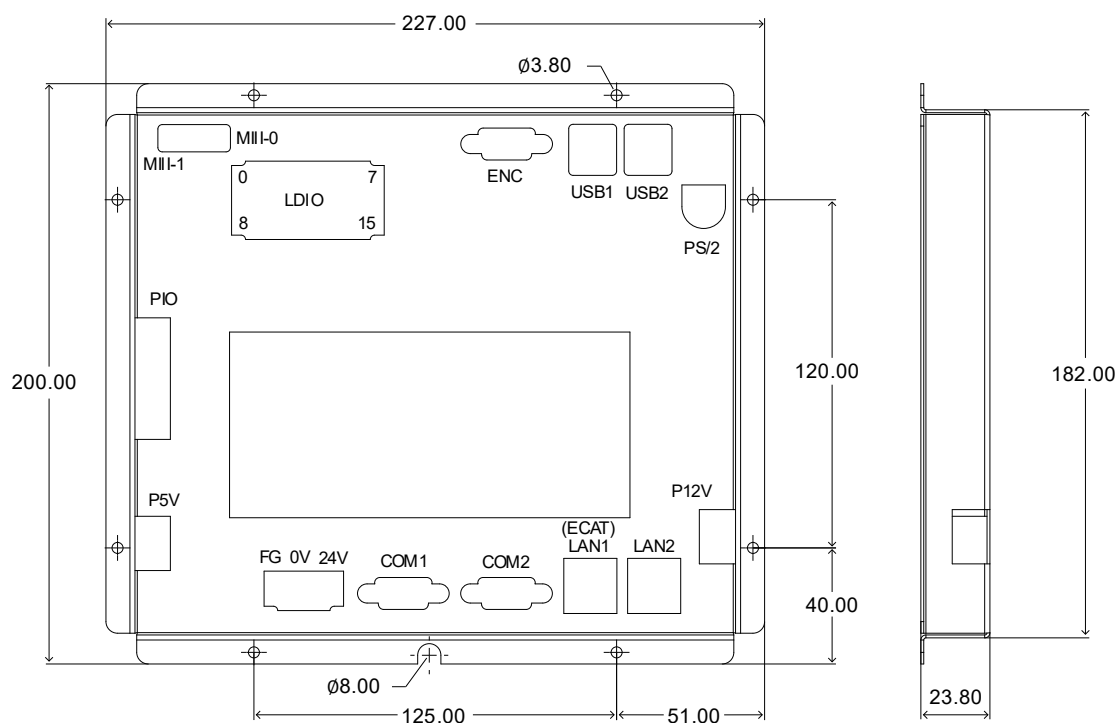


Figure 3.1-1: DynaPath WinDelta Node-355 Control Chassis

The Node-1000 control system chassis dimensions and ports are shown in Figure 3.1.1.2. This control system is generally installed separately away from the operating console in the electrical cabinet as a standalone control solution.

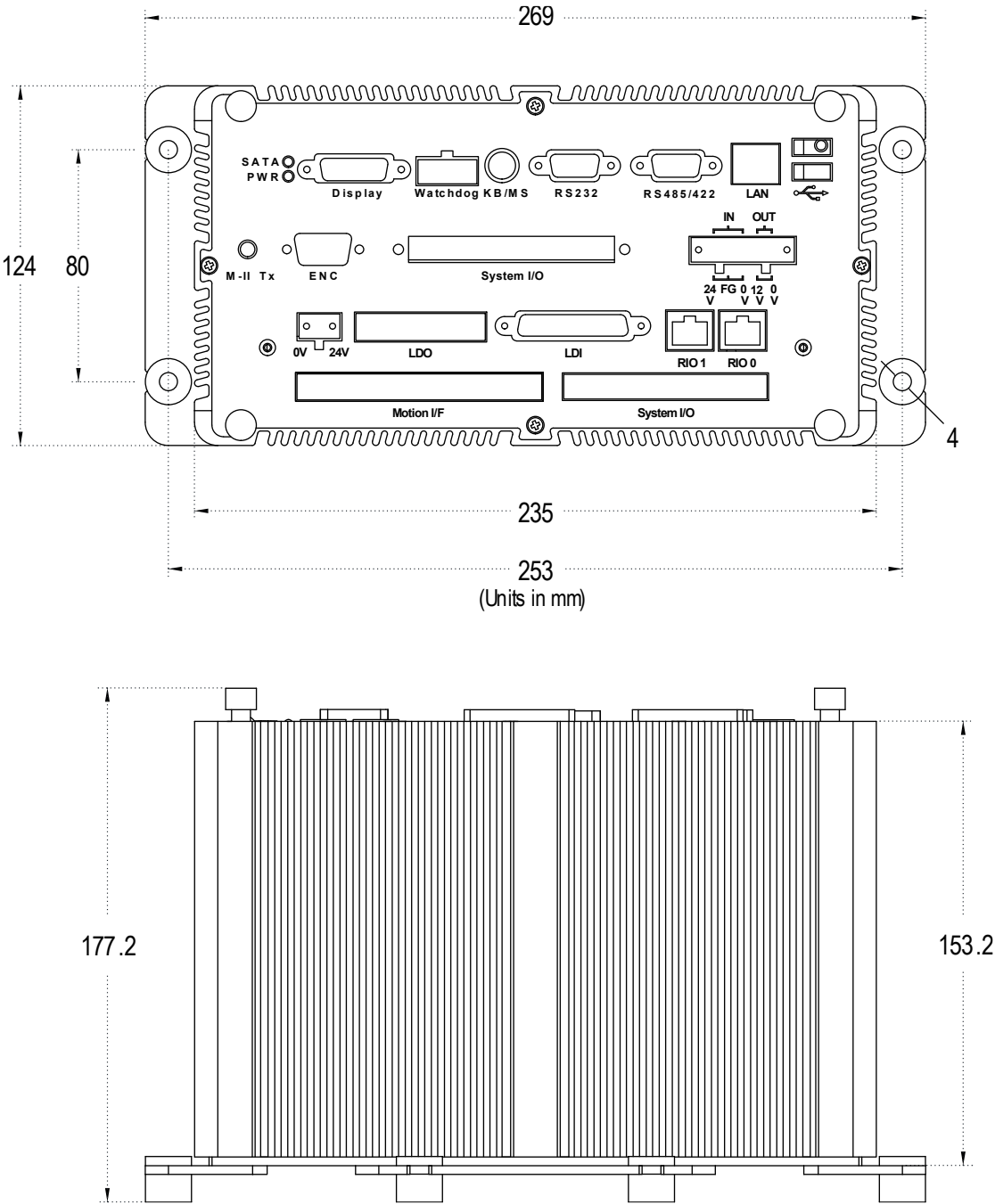


Figure 3.1-2: DynaPath WinDelta Node-1000 Control Chassis

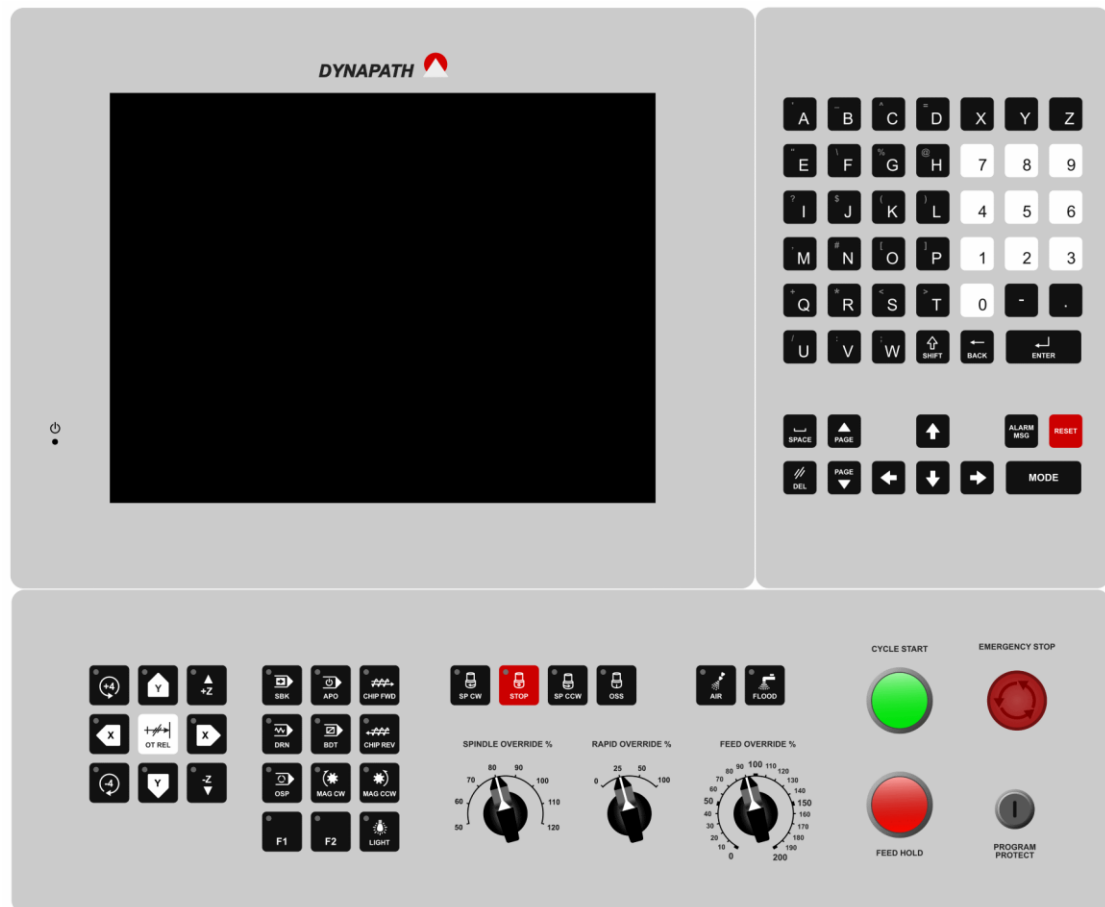
3.1.2 D1 Console

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



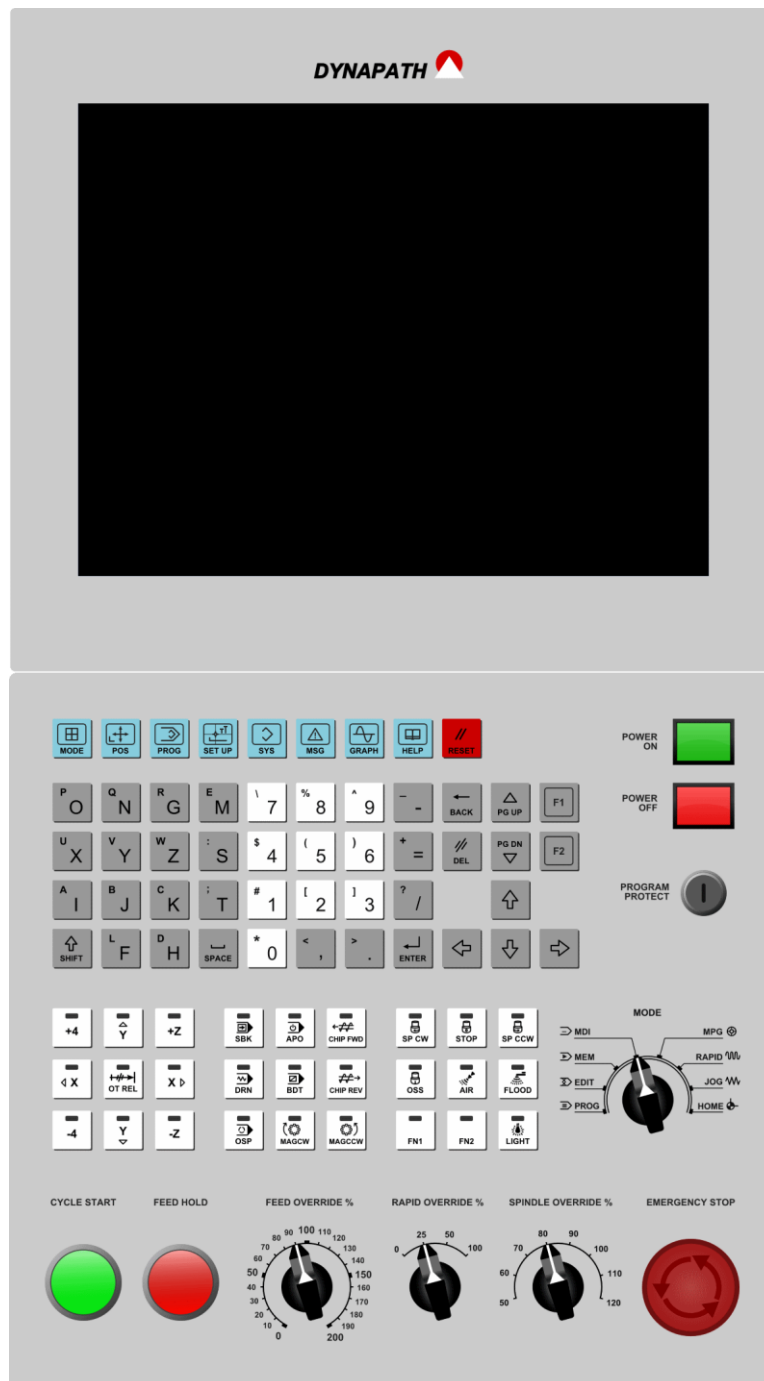
3.1.3 D5 Console

The DynaPath D5 Console is designed for machine tools with automatic tool changers such as milling centers or machining centers. It is suitable for manual milling, 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.



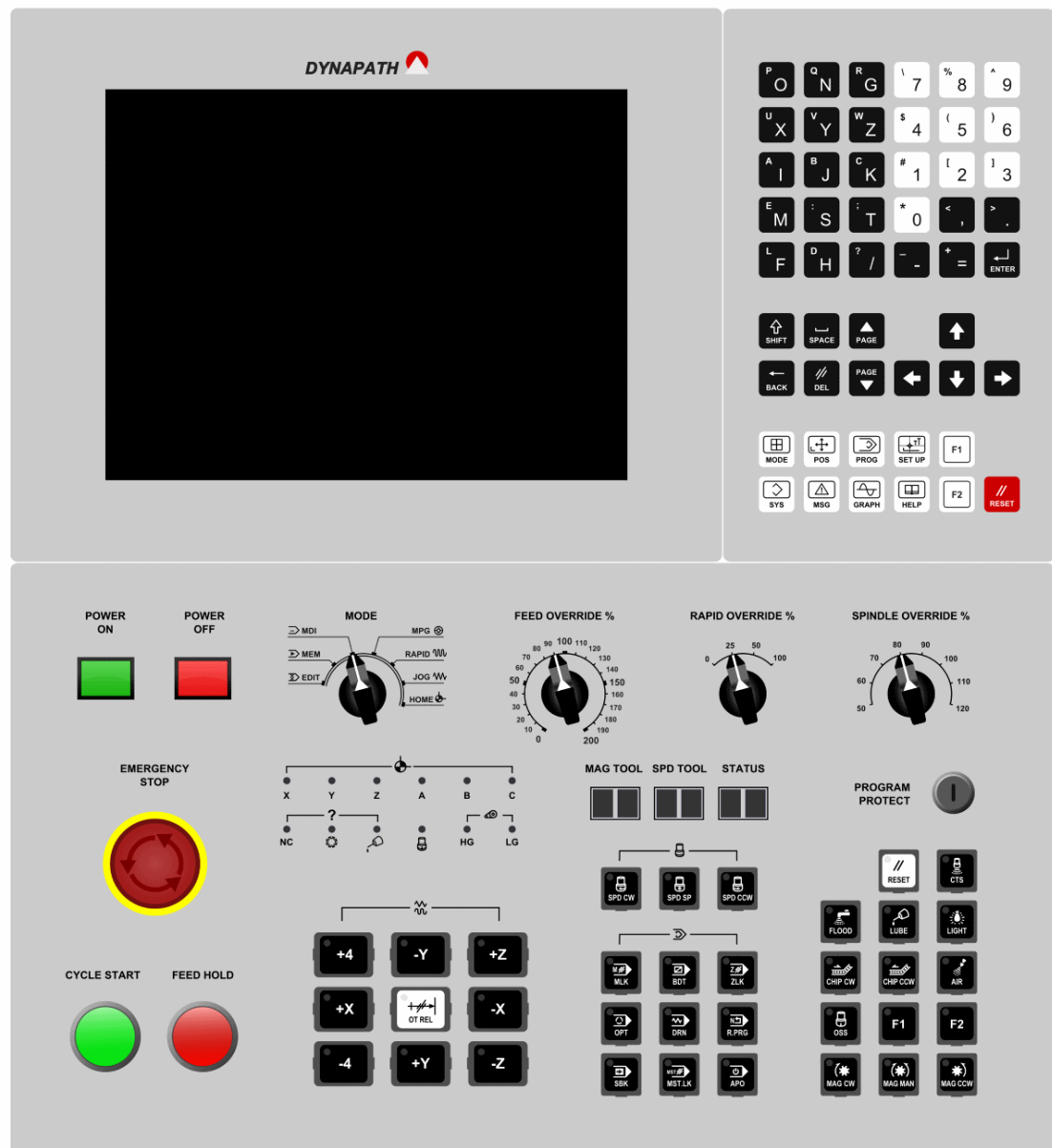
3.1.4 D2 Console

The DynaPath D2 Console is designed for smaller profile machining centers and high-speed milling centers. The 2nd panel for the D2 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.



3.1.5 D6 Console

The DynaPath D6 Console is designed for fully equipped machining centers. The 2nd panel for the D6 console has additional tool indicators and program run related functions. This console features the DynaPath Mode Switch operation style.



3.1.6 D11 Console

The DynaPath D11 Console is designed for a wide range of machines from basic to advanced. It is equipped with a larger touchscreen interface designed for improved software enhanced usability. The 2nd panel retains axes jog, cycle start, feed hold, and E-Stop buttons, and spindle, feed, and rapid overrides. This console has options for the DynaPath Mode Switch or Mode Screen operation styles.



3.1.7 Handheld MPG

The DynaPath WinDelta has the option for an electronic manual pulse generator (MPG) to enable more precise machine movements and other features. The standard DynaPath handheld MPG device shown in Figure 3.1.7.1 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

3.1.8 Electronic Handle / Handwheel

The DynaPath WinDelta can also interface with additional electronic handles or handwheels to enable traditional style machine operation. These digital devices mimic mechanical handwheels common on manual mills, while enabling the ability to control axes movement in adjustable increments, identical to the handheld MPG.



Figure 3.1-4: Electronic Handwheel

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 to use 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 transferable 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).

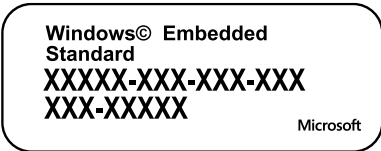


Figure 3.2-1: Microsoft OS License Decal

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.

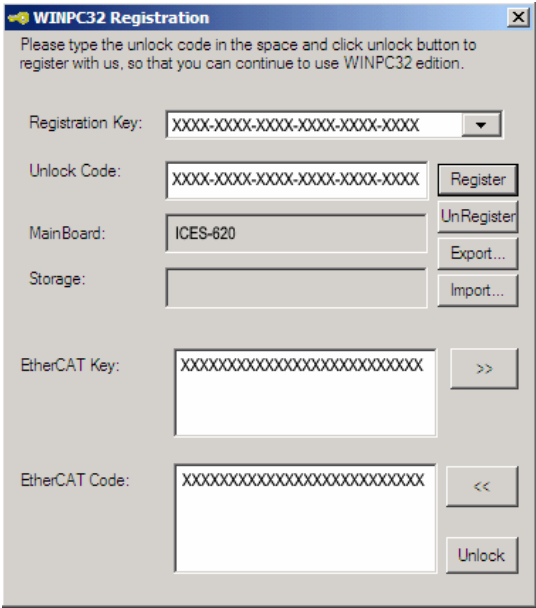


Figure 3.2-3: DynaPath Registration Keys and Unlock Codes

4 BASIC 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 oil or coolant leaks under or around the machine.
5. Check the mechanical condition of the machine to ensure axes are not obstructed or fixed to prevent travel.

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.

POWER ON

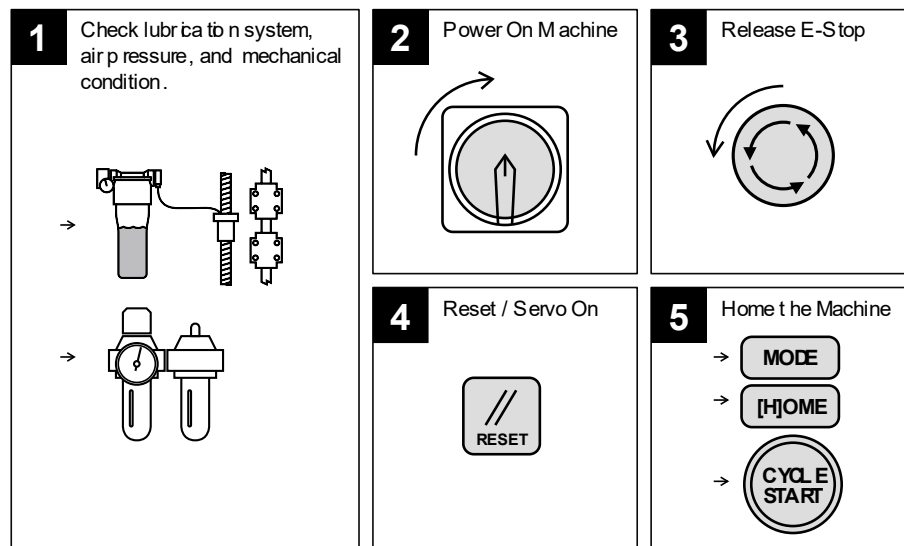


Figure 4.4-1: Power On Guide

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 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.

POWER OFF

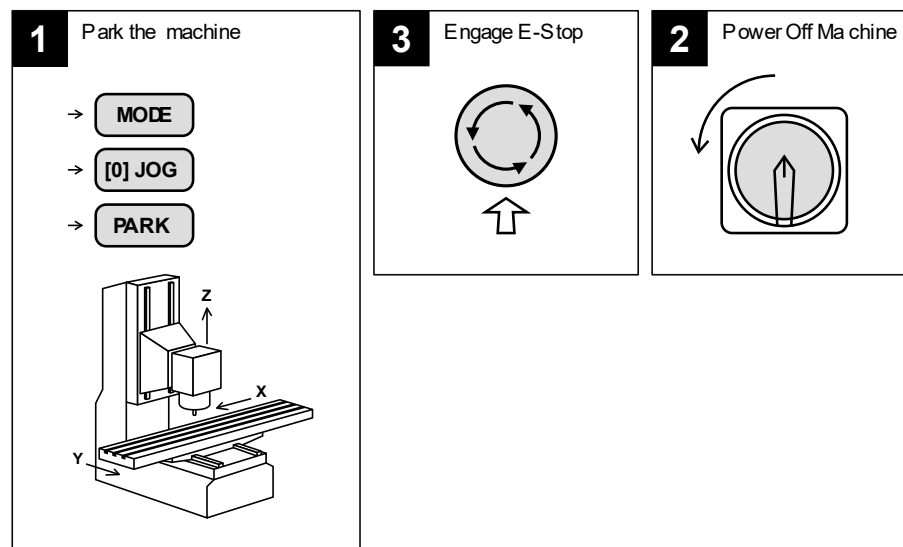


Figure 4.5-1: Power Off Guide

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 D2 and D6 panels, 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 4.8.1.1 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 4.8.2.1.

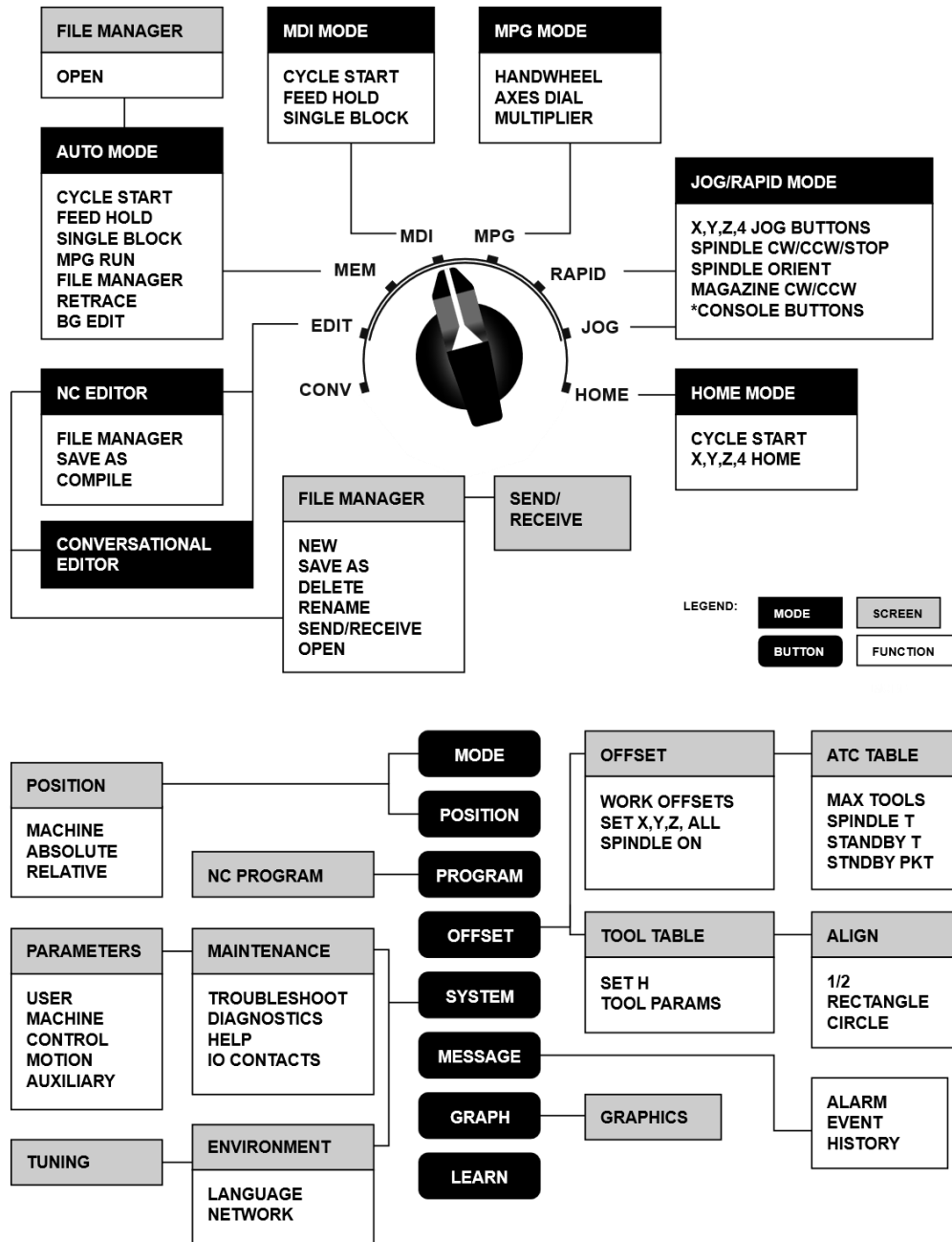


Figure 4.8-2: DynaPath Mode Switch Operation

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

COORDINATE SYSTEM

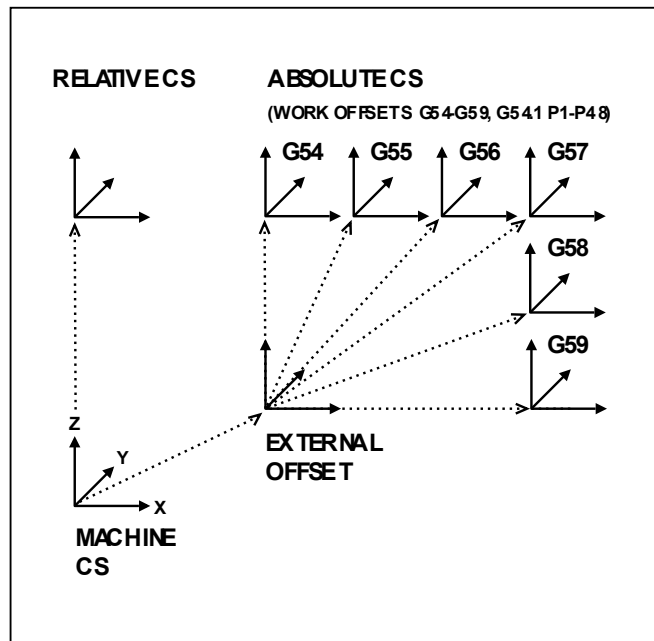


Figure 4.9-1: Coordinate Systems Illustrated

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. An additional 48 extended work offsets are also provided as an option. In a part program, each work offset is activated by commanding its G-code: G54 to G59, or G54.1 P1 to P48. 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.

4.11 Tool Offset Concepts

The control system can account for the different tool dimensions automatically if tool offsets have been set up. There are two types of tool offsets, each requiring a specific G-Code command in order to activate:

1. Tool length offset and compensation:

- a. The tool length value H_ relative to a reference is stored into the control.
- b. G-Code commands:
 - i. **G43 H_** performs positive tool length compensation.
 - ii. **G44 H_** performs negative tool length compensation.
 - iii. **G49** cancels any tool length compensation.

2. Tool radius offset and compensation:

- a. The tool diameter value D_ is stored into the control.
- b. G-Code commands:
 - i. **G41 D_** performs left tool radius compensation.
 - ii. **G42 D_** performs right tool radius compensation.
 - iii. **G40** cancels any tool radius compensation.

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 tool, quill, tool changer, 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 not required. Pressing the jog button for these axes simply moves the axis to its defined machine zero position as stored in the drive system.

Note: For machines equipped with rotary axes, it is advised to home these axes first. Press the C+, C-, 4+, 4-, 5+, or 5- jog buttons on the panel to home the rotary axes.

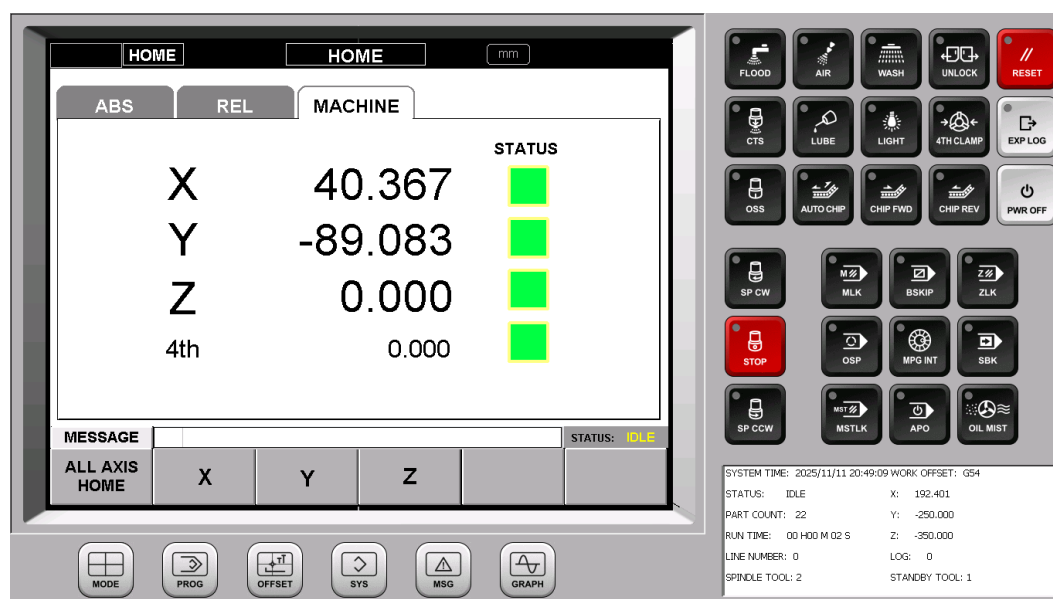


Figure 5.1-1: Homing Screen

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.

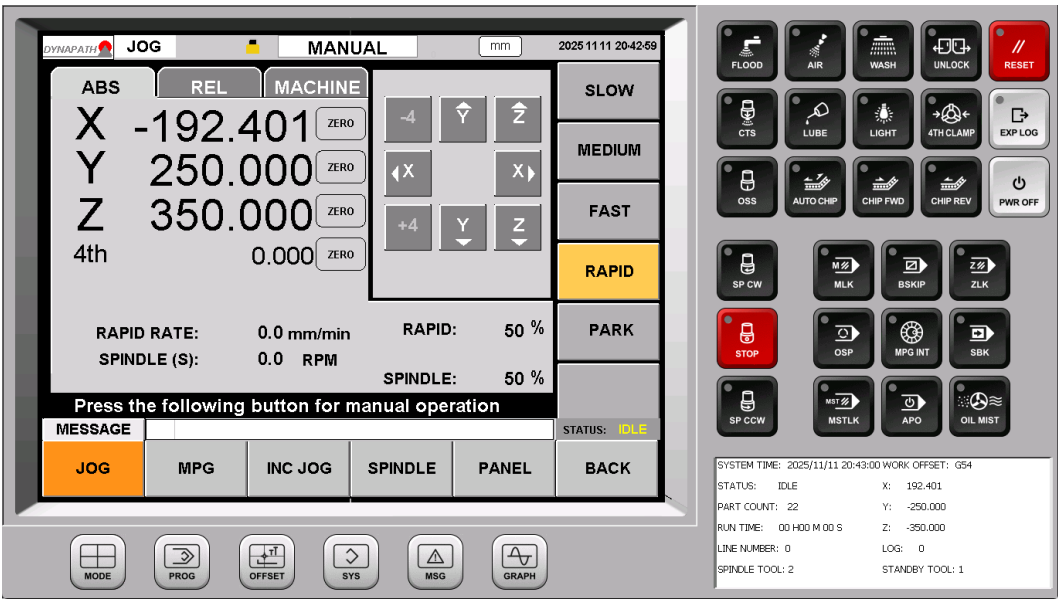


Figure 6.1-1: Jog Screen

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, a dial on the MPG applies a x1, x10, or x100 multiplier to affect the distance moved on each click of the MPG handwheel.

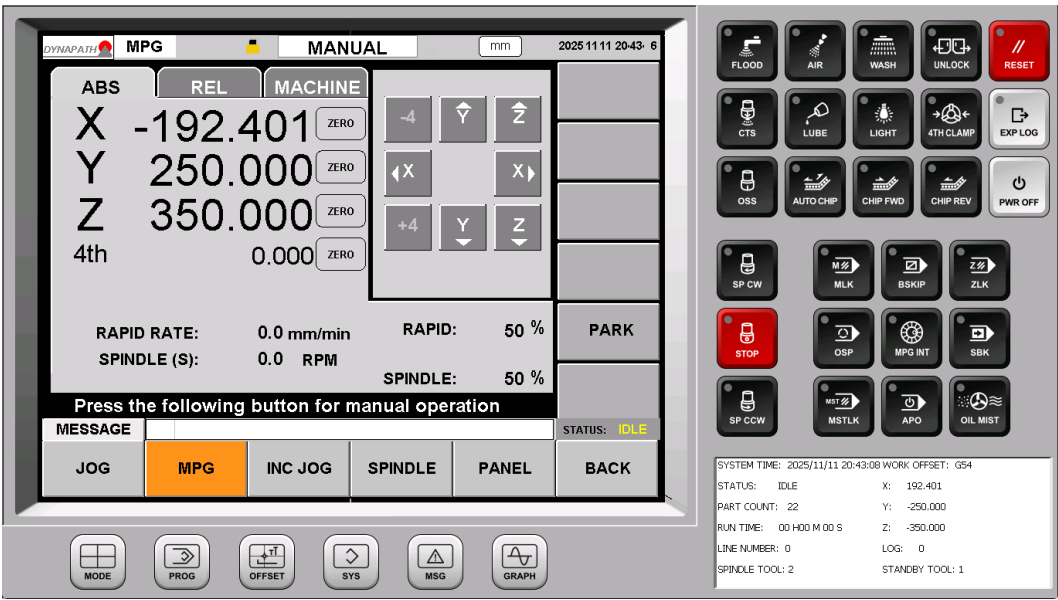


Figure 6.2-1: MPG Screen

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.

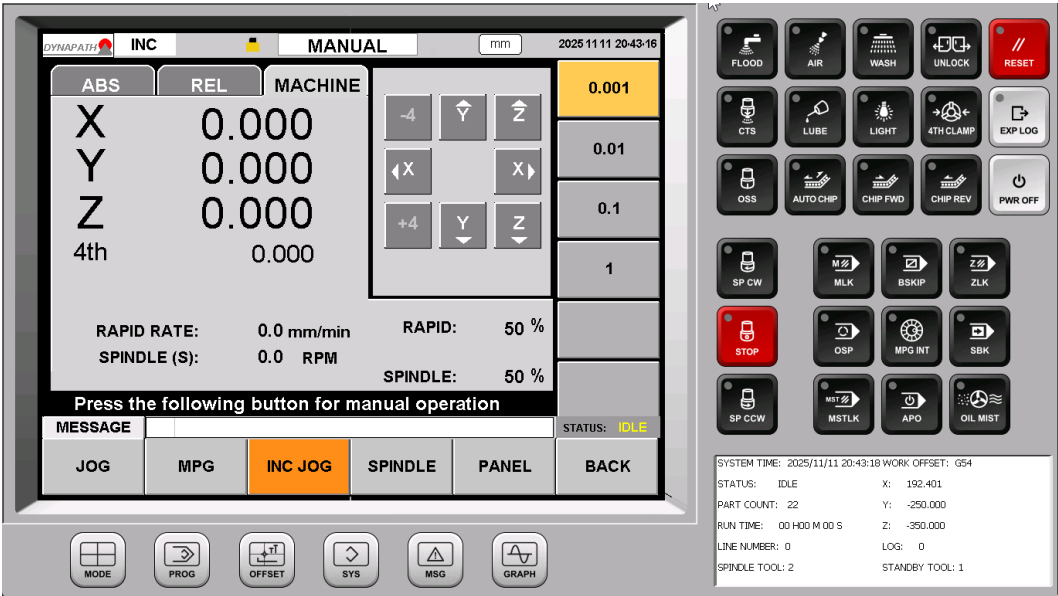


Figure 6.3-1: Incremental Jog Screen

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.

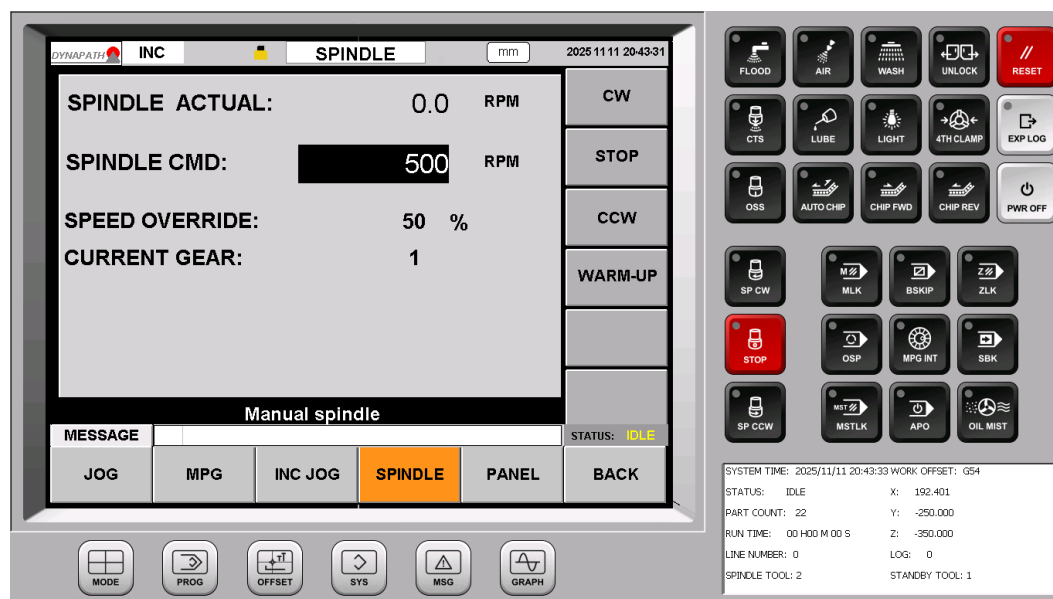


Figure 6.4-1: Spindle Control Screen

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. Commonly used panel functions are placed on the persistent soft-panel on the right for widescreen controls.

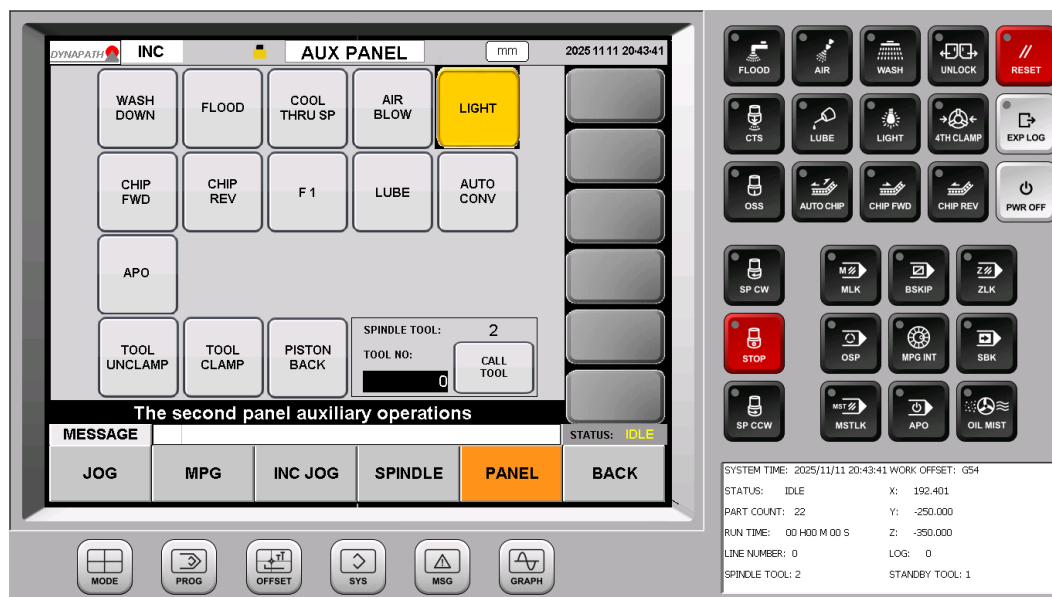


Figure 6.5-1: Jog Panel Screen

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.

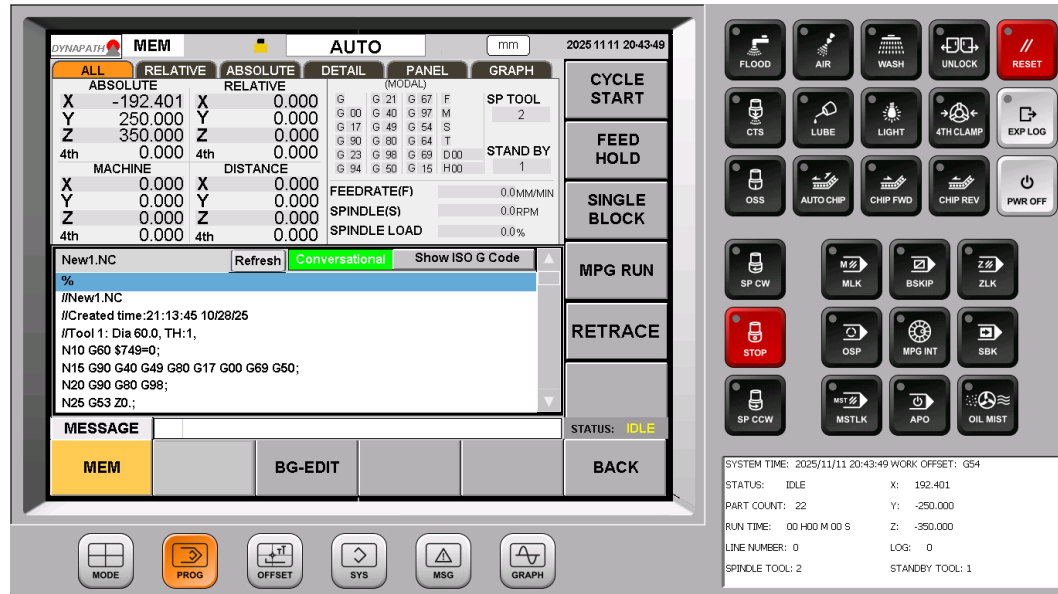


Figure 7.1-1: Auto / MEM Screen

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 activation 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 outlined 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 part programs are executed with reference to. The control can store multiple ABS coordinate systems which are named G54-G59 and are activated when called within the 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 after changing tools, each tool can continue on the programmed tool path.

8.2 Work Offset

The OFFSET screen is where workpiece or fixture offsets are set up and managed. This provides the ABS or ABSOLUTE reference coordinate system from 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. Directly input axes location values in terms of machine coordinates, or press the X, Y, or Z-Axis buttons along the right of the screen to set each axis' ABS zero position to the current axis location.
4. Apply EXTERNAL OFFSET or EXOFS to shift the active ABS zero position. These are always active during program execution.

8.2.1 Work Offset – Set to Position

Setting the Work Offset establishes the Part Zero location of a part program. The procedure to set the Part Zero to an arbitrary location is illustrated in Figure 8.2.1.1.

SET PART ZERO TO ANY POINT

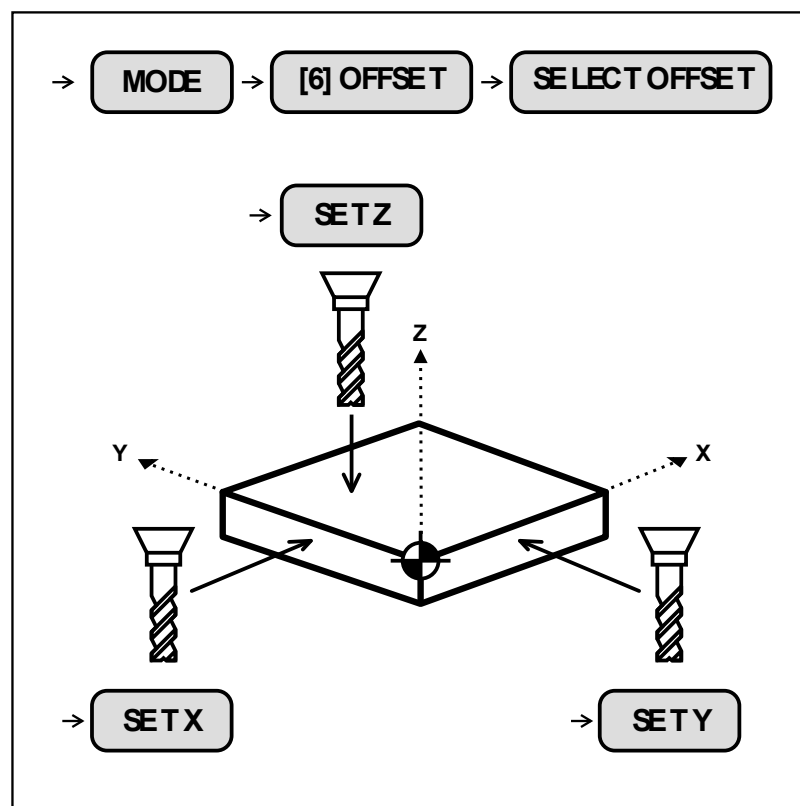


Figure 8.2-2: Set Part Zero to Arbitrary Position

8.3.1 Work Offset – Align to Rectangular Center

An alignment function for setting work offsets to the center of a rectangular part can be found under the ALIGN/RECT function. The procedure to operate it is detailed in the following figure.

SET PART ZERO TO RECTANGULAR CENTER

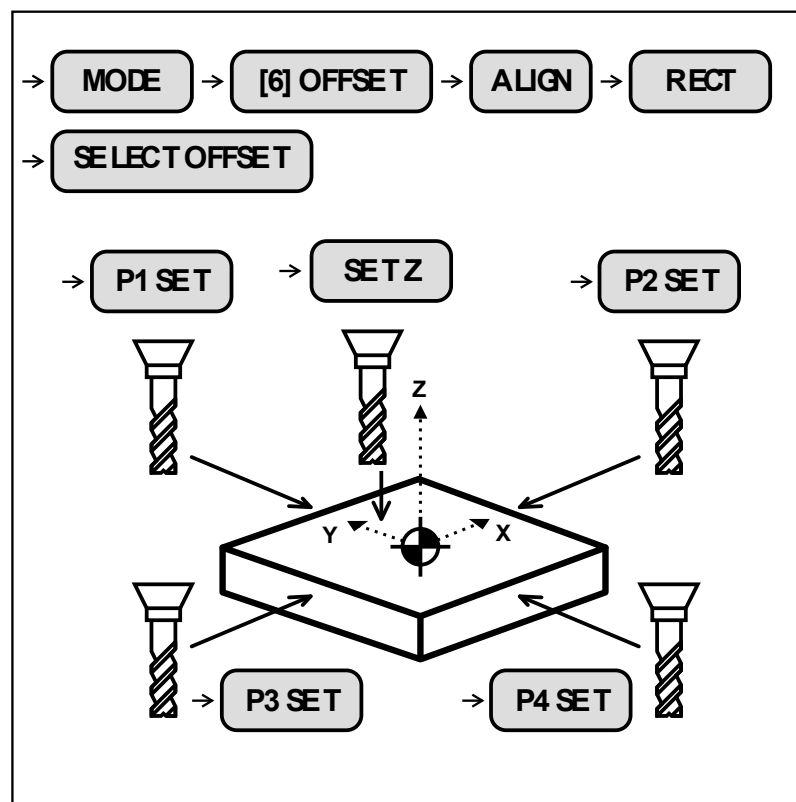


Figure 8.3-1: Set Part Zero to Rectangular Center

8.3.2 Work Offset – Align to Circle Center

An alignment function for setting work offsets to the center of a circular part can be found under the ALIGN/CIRCLE function. The procedure to operate it is detailed in the following figure.

SET PART ZERO TO CENTER OF CIRCLE

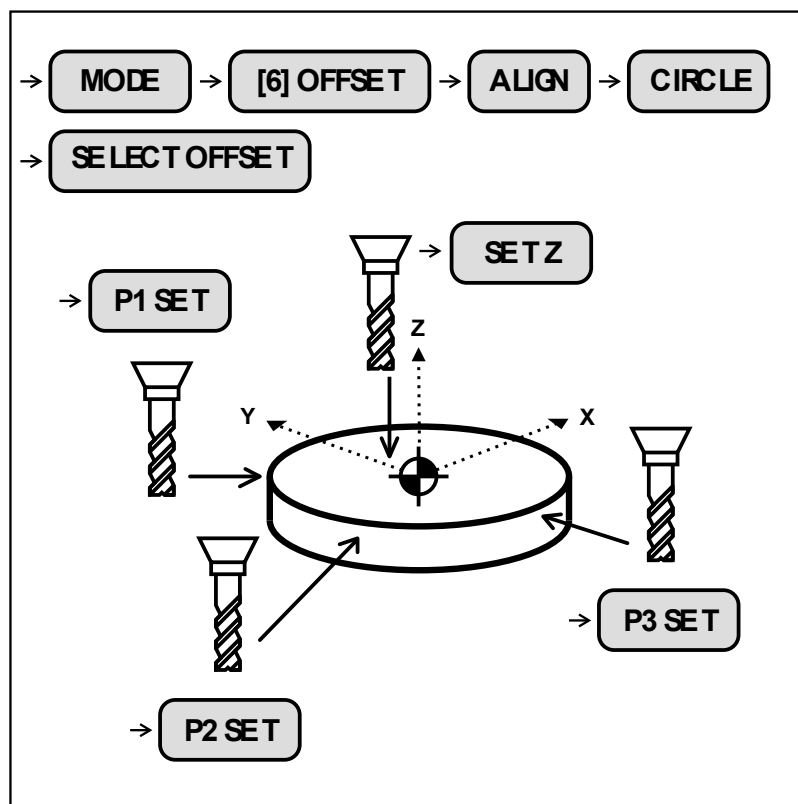


Figure 8.3-2: Set Part Zero to Circular Center

9 TOOL OFFSET

9.1 Tool Offset Screen

The TOOL OFFSET table is where differences in tool dimensions are stored. These values are used to compensate for tool lengths and tool diameters when the G-Code command for tool length compensation “G43/G44 H_” or tool radius compensation “G41/G42 D_” are called in a program.



Figure 9.1-1: Tool Offset Table

The tool lengths for each tool are stored in the H-Code column while the tool diameters are stored in the D-Code column. In addition, there are H WEAR and D WEAR fields for storing additional offsets to correct for wear in tool length or tool diameter. When an actual tool length compensation command “G43 H_” or “G44 H_” is called, the values for H and H WEAR are added together and applied as the compensation value. Similarly, when the tool radius compensation command “G41 D_” or “G42 D_” command is called, the values for D and D WEAR are added together and applied as the compensation value.

9.1.1 Call Tool Function

A tool change is executed upon highlighting a target tool number and then pressing the CALL TOOL on-screen button. When the automatic tool change procedure is executing, the status should display “RUN” while the CNC mode will display “MEM”. Then when tool change is complete, the status should display “IDLE” and the CNC mode should return to JOG or MPG.

9.1.2 Tool Length Measurement

The SET H function on the tool offset screen allows tool lengths to be measured and stored directly into the CNC tool table.

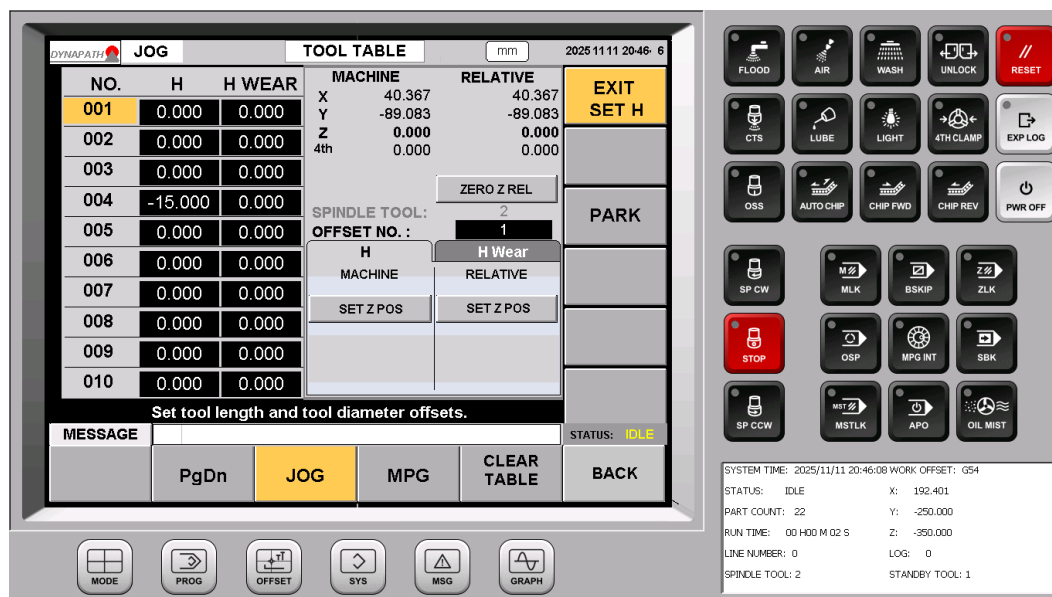


Figure 9.1-2: Tool Length Offset Measure Screen

Note: The two methods for setting up tool lengths are detailed in the following sections.

The general operation procedure to measure tool lengths using this screen are as follows:

1. If the tool lengths to be measured are all relative to a reference tool, then touch the reference tool to a Z-reference position and press ZERO Z REL. Otherwise, skip this step.
2. Highlight the target tool and physically change to the tool or use CALL TOOL.
3. JOG or MPG the tool tip to the previous Z-reference position. When touching the surface,
 - a. If the tool length to be measured is relative to machine zero, press MACHINE SET Z POS.
 - b. If the tool length to be measured is relative to a reference tool, press RELATIVE SET Z POS.
4. Start at step 2 again to call, measure, and store the tool length for each remaining tool.

9.2 Tool Length Offset Method A – Reference Machine Zero

Tool Length offsets can be set with reference to the Machine Zero position by touching each tool to a physical feature and then storing the current machine coordinate position into its respective Tool Length Offset number or H-Code. Each tool length essentially stores the distance from Machine Zero to part zero. When using this method, the Work Offset Z-values are generally not used or set to zero.

Note: Setting tool lengths with respect to Machine Zero is simpler, however if the part zero position in the Z-axis has changed, every tool length will need to be re-measured for the program to run correctly.

The procedure for setting tool lengths with respect to Machine Zero is illustrated in the following figure.

SET TOOL LENGTH RELATIVE TO MACHINE ZERO

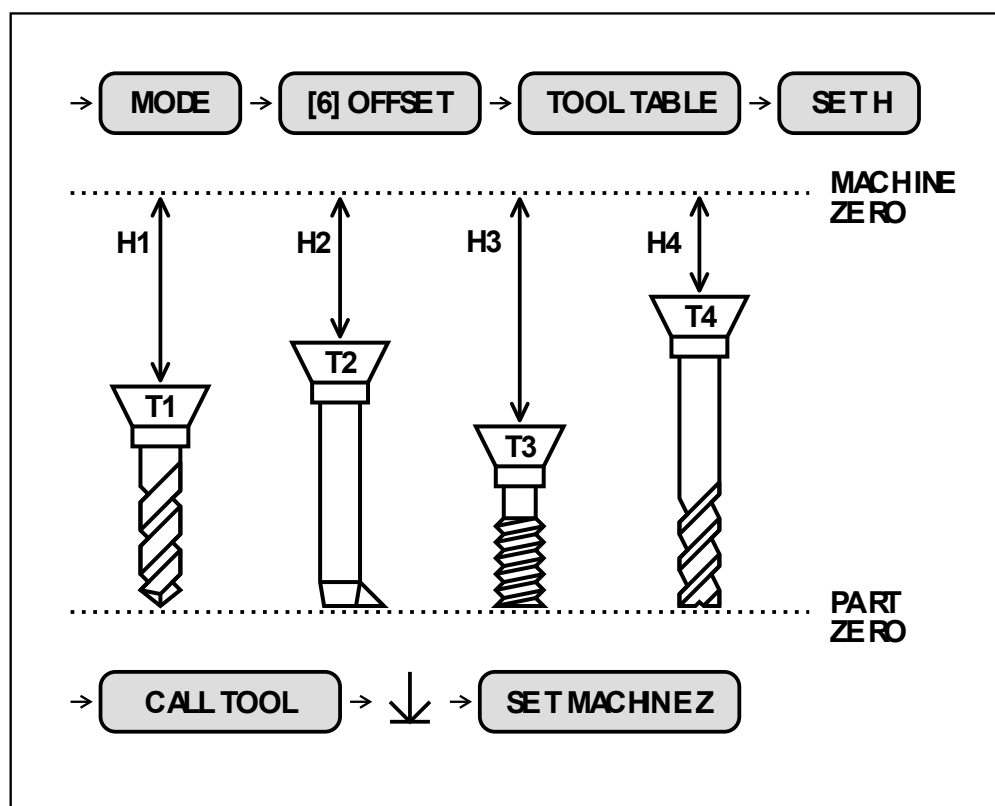


Figure 9.2-1: Tool Offset – Reference Machine Zero

9.3 Tool Length Offset Method B – Reference a Tool

Tool Length offsets can be set with reference to a reference tool. In this method, tool offsets are independently measured and maintained with no association with part zero or the Work Offset Table.

Note: Setting tool lengths with respect to a reference tool has an advantage where if the workpiece part zero was changed, only the Z-axis location for the Work Offset needs to be re-measured using the reference tool.

Note: It is recommended to use a non-cutting tool, or a tool where the physical length is constant as the reference tool.

The procedure for setting tool lengths with respect to a reference tool is illustrated in the following figure.

SET TOOL LENGTH RELATIVE TO REFERENCE TOOL

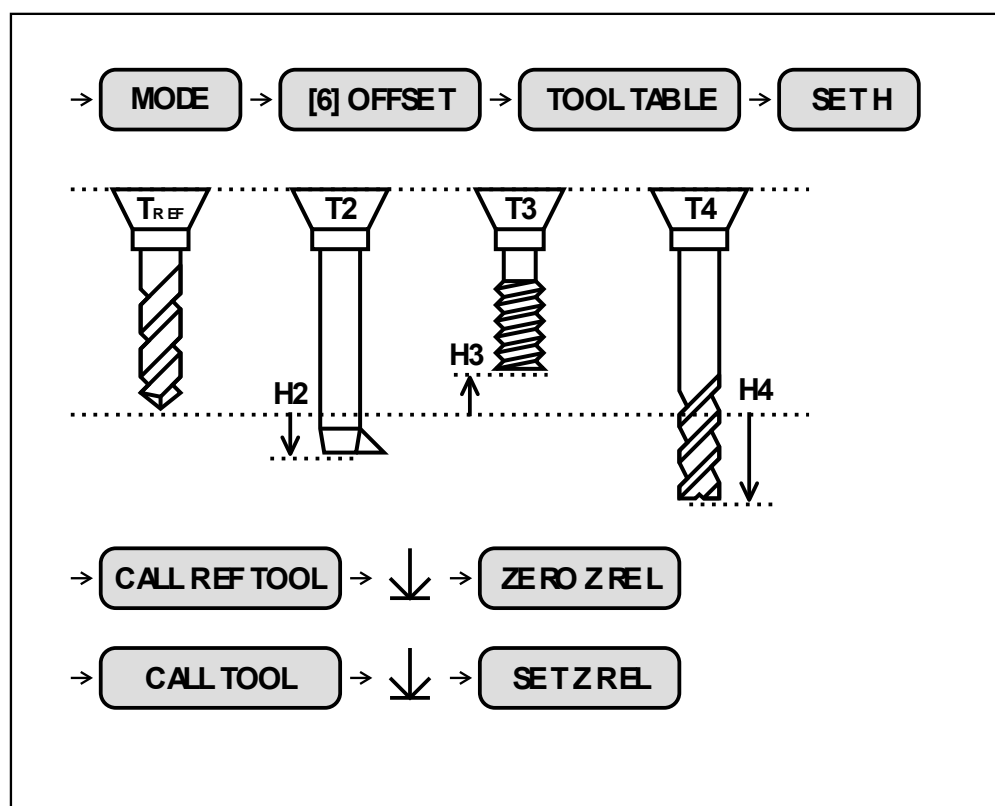


Figure 9.3-1: Tool Offset – Reference Tool

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 feed actions or a step through a sequence of pre-programmed line and arc feed cycles.

10.2 Manual Operation

Upon entering the SEMI-AUTO mode screen, the CNC defaults to MPG mode. Under the MPG mode, the machine may be moved by either the handheld MPG or individual axes handwheels. On this screen the standard action is to set the part zero location by moving the machine to the desired position and then setting the absolute coordinate values for each axis. The spindle can be controlled by specifying the spindle RPM and engaging the SP CW or SP CCW buttons on the panel.

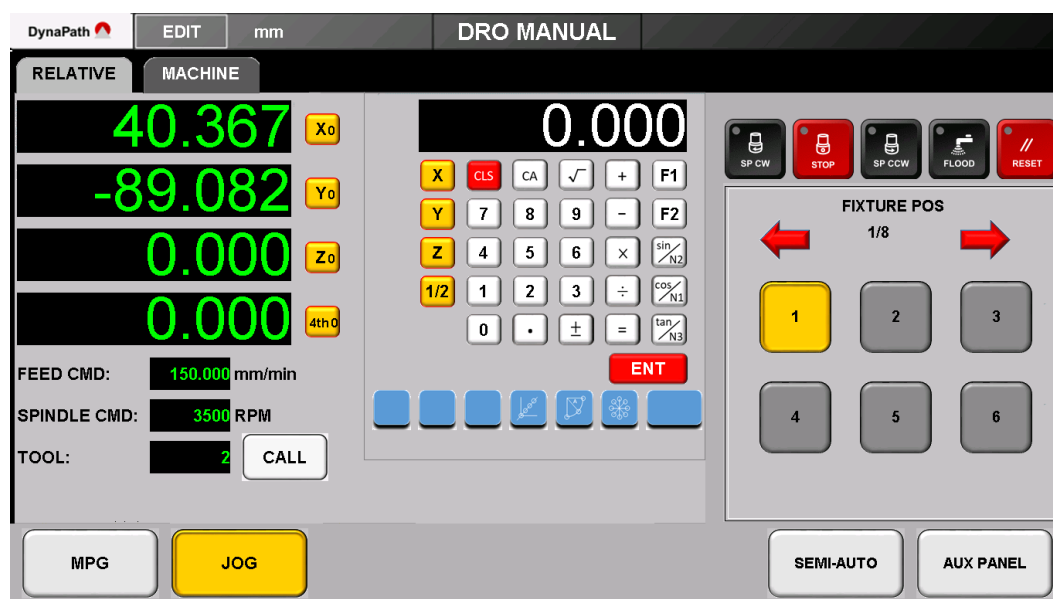


Figure 10.2-1: DRO / Semi-Auto Mode Manual Screen

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.

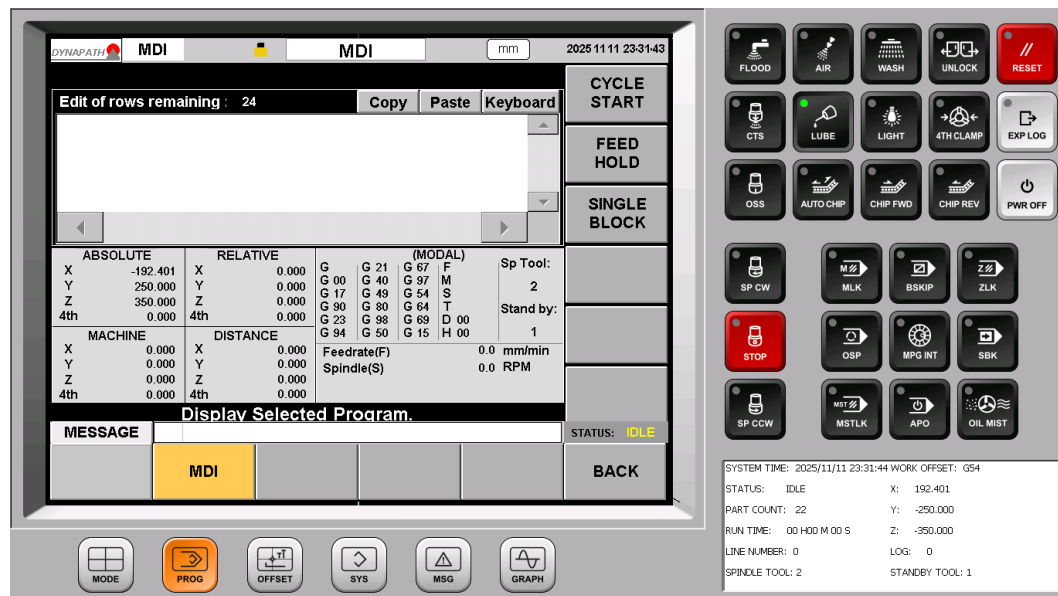


Figure 11.1-1: MDI Screen

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, or double click it. Depending on whether the file is a G-Code program or a Conversational Program, the corresponding editor will display the selected program.

Note: A program opened within the File Manager screen becomes the active program, ready for execution upon switching to AUTO mode.



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 a copy.

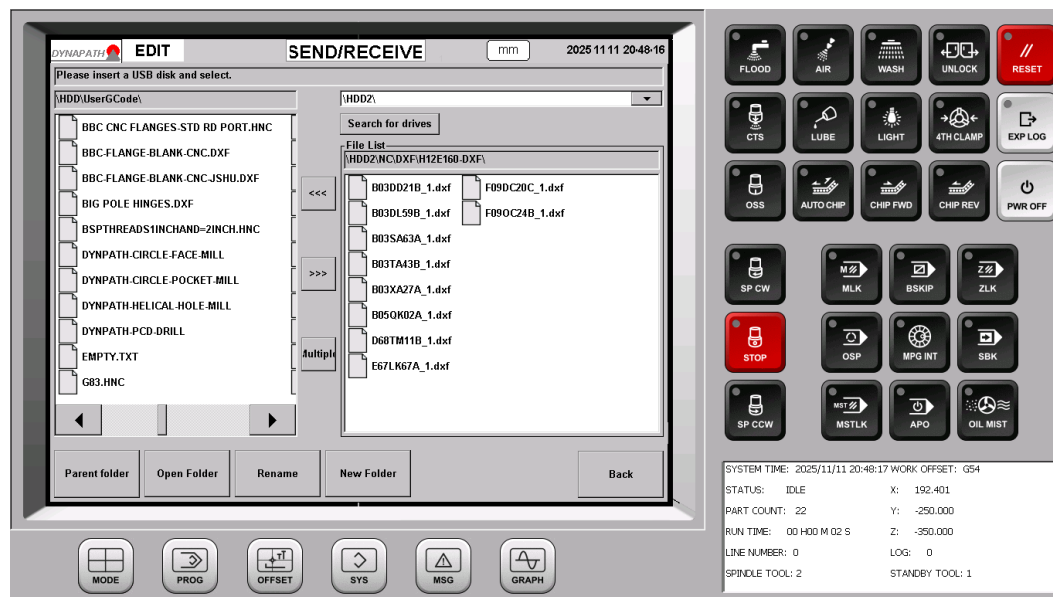


Figure 12.2-1: Send and Receive Screen

13 G-CODE PROGRAMMING

13.1 Program Language and Structure

Standard EIA RS-274-D and ISO 6983 G-Code is supported on the WinDelta control system. A G-Code program consists of multiple lines, and each line consisting of a string of alternating words and values, with words consisting of alphanumeric characters and values consisting of numbers. Word characters are case insensitive on this control system. In general, programs use all upper-case words and letters. The end of line character, or semi-colon, is optional if each block begins on a new line. Spaces between works, numbers, and letters are also optional. The supported text encoding for G-Code programs is ANSI or UTF-8.

Program comments are text in the program that are intended only as descriptions for the programmer to view. They are ignored by the motion compiler during program execution. The supported program words and characters are listed in Table 13.1.1.

Table 13.1.1: Program Words and Characters

Code	Usage
[A-Z]	Word characters.
[0-9].-	Number characters can contain decimal point and negative or minus symbol.
N	The special N character followed by an integer number denotes a block sequence number or line label. <i>Example:</i> N100
(...)	Parentheses encapsulate comments within its opening and closing symbols. <i>Example:</i> N100 (COMMENT 1) G01 X10. (COMMENT 2)
/	Optional block skip. Must be at the beginning of a line.
//	Everything after two forward slashes until the end of the line are treated as comments. <i>Example:</i> N100 //COMMENT 1 //COMMENT 2
;	The end of block character is optional, and any text after it are treated as comments. <i>Example:</i> N100; COMMENT 1 G01 X10.; COMMENT 2

13.2 G-Code Reference

G-Code programming on the WinDelta control adheres to the ISO/EIA G-Code standard for milling. At any given instance, one G-Code command is active for any given group. This is referred to as the active modal G-Code and will remain active until it is replaced by a call to another G-Code from within the same group.

Table 13.2.1: G-Code Commands Reference

G-Code	Group	Description
G00	1	Positioning (Rapid traverse) G00 X_ Y_ Z_ A_ B_ C_ X_, Y_, Z_, A_, B_, C_ Axes target positions
G01	1	Linear interpolation (Cutting feed) G01 X_ Y_ Z_ A_ B_ C_ F_ X_, Y_, Z_, A_, B_, C_ Axes target positions F_ Feed rate m/min or in/min
G02	1	CW Circular interpolation or helical interpolation G17 G02 X_ Y_ Z_ I_ J_ F_ G18 G02 X_ Y_ Z_ J_ K_ F_ G19 G02 X_ Y_ Z_ I_ K_ F_ X_, Y_, Z_ Axes target positions I_, J_ K_ Center location relative to previous position F_ Feed rate m/min or in/min
G03	1	CCW Circular interpolation or helical interpolation G17 G03 X_ Y_ Z_ I_ J_ F_ G18 G03 X_ Y_ Z_ J_ K_ F_ G19 G03 X_ Y_ Z_ I_ K_ F_ X_, Y_, Z_ Axes target positions I_, J_ K_ Center location relative to previous position F_ Feed rate m/min or in/min
G04	0	Dwell G04 X_ X_ Seconds to dwell
G05	0	High speed cycle machining
G10	0	Programmable data input
G11	0	Programmable data input mode cancel
G17	2	XY plane selection
G18	2	ZX plane selection
G19	2	YZ plane selection
G20	6	Input in inch
G21	6	Input in mm
G28	0	Return to reference position through absolute position G28 X_ Y_ Z_ A_ B_ C_ X_, Y_, Z_, A_, B_, C_ Axes through positions
G29	0	Movement from reference position to absolute position G29 X_ Y_ Z_ A_ B_ C_

		X_, Y_, Z_, A_, B_, C_ Axes target positions
G30	0	2nd and 3rd and 4th reference position return G30 P2 X_ Y_ Z_ G30 P3 X_ Y_ Z_ G30 P4 X_ Y_ Z_ X_ Y_ Z_ Return to machine zero through absolute positions
G31	0	Skip function G31 X_ Y_ Z_ A_ B_ C_ F_ X_, Y_, Z_, A_, B_, C_ Axes target positions F_ Feed rate m/min or in/min
G40	7	Cutter compensation of tool nose radius: cancel
G41	7	Cutter compensation of tool nose radius: left G41 D_ D_ Tool radius compensation index
G42	7	Cutter compensation of tool nose radius: right G42 D_ D_ Tool radius compensation index
G43	8	Tool length compensation, positive G43 H_ D_ Tool length compensation index
G44	8	Tool length compensation, negative G43 H_ D_ Tool length compensation index
G49	8	Tool length compensation cancel
G52	0	Local coordinate system offset G52 X_ Y_ Z_ X_, Y_, Z_ Axes offset amount
G53	0	Machine coordinate system setting G53 X_ Y_ Z_ A_ B_ C_ X_, Y_, Z_, A_, B_, C_ Axes target positions
G54	14	Workpiece coordinate system 1 selection
G54.1	14	Extended workpiece coordinate system selection G54.1 P_ P_ (1-48) Extended workpiece coordinate index
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 G65 P_ P_ Program number (program name must start with O and end with a four-digit program number)
G68	16	Coordinate system rotation G17 G68 X_ Y_ R_ G18 G68 Y_ Z_ R_

		G19 G68 X_ Z_ R_ X_, Y_, Z_ Rotation center position R_ Rotation angle
G69	16	Coordinate system rotation cancel
G73	9	High speed peck drill cycle G73 X_ Y_ Z_ R_ Q_ D_ F_ K_ X_, Y_, Z_ Target position R_ Start depth Q_ Depth of each peck D_ Retract distance for each peck F_ Feed rate K_ Number of repeats
G74	9	Left hand tap cycle G74 X_ Y_ Z_ R_ P_ F_ K_ X_, Y_, Z_ Target depth R_ Start depth P_ Dwell time at the bottom of the cycle F_ Pitch of thread K_ Number of repeats
G76	9	Fine bore cycle G76 X_ Y_ Z_ R_ Q_ F_ K_ X_, Y_, Z_ Target position R_ Start depth Q_ Back-off distance for retraction F_ Feed rate K_ Number of repeats
G80	9	Canned cycle cancel
G81	9	Drill cycle G81 X_ Y_ Z_ R_ F_ K_ X_, Y_, Z_ Target position R_ Start depth F_ Feed rate K_ Number of repeats
G82	9	Counter bore cycle G82 X_ Y_ Z_ R_ P_ F_ K_ X_, Y_, Z_ Target position R_ Start depth P_ Dwell time at the bottom of the cycle F_ Feed rate K_ Number of repeats
G83	9	Peck drill cycle G83 X_ Y_ Z_ R_ Q_ D_ F_ K_ X_, Y_, Z_ Target position R_ Start depth Q_ Depth of each peck D_ Retract distance for each peck F_ Feed rate K_ Number of repeats

G84	9	Right hand tap cycle G84 X_ Y_ Z_ R_ P_ F_ K_ X_, Y_, Z_ Target depth R_ Start depth P_ Dwell time at the bottom of the cycle F_ Pitch of thread K_ Number of repeats
G85	9	Bore cycle G85 X_ Y_ Z_ R_ F_ K_ X_, Y_, Z_ Target position R_ Start depth F_ Feed rate K_ Number of repeats
G86	9	Dead bore fixed spindle cycle G86 X_ Y_ Z_ R_ F_ K_ X_, Y_, Z_ Target position R_ Start depth F_ Feed rate K_ Number of repeats
G87	9	Back bore cycle G87 X_ Y_ Z_ R_ Q_ F_ K_ X_, Y_, Z_ Target position R_ Start depth Q_ Back-off distance for retraction F_ Feed rate K_ Number of repeats
G88	9	Dead spindle bore and dwell cycle G82 X_ Y_ Z_ R_ P_ F_ K_ X_, Y_, Z_ Target position R_ Start depth P_ Dwell time at the bottom of the cycle F_ Feed rate K_ Number of repeats
G89	9	Bore with dwell cycle G89 X_ Y_ Z_ R_ P_ F_ K_ X_, Y_, Z_ Target position R_ Start depth P_ Dwell time at the bottom of the cycle F_ Feed rate K_ Number of repeats
G90	3	Absolute command
G91	3	Incremental command
G98	10	Return to initial point in canned cycle
G99	10	Return to R point in canned cycle

13.3 M-Code Reference

Table 13.3.1 documents standard machine function codes natively supported by the WinDelta mill control. Custom M-Codes may be added by the machine manufacturer for commanding additional machine functions.

Table 13.3.1: M-Code List

M-Code	Option	Description
M00	A	Program Stop
M01	A	Optional Stop
M02	A	Program End
M03	A	Spindle CW Run M03 S_ S = Spindle RPM
M04	A	Spindle CCW Run M04 S_ S = Spindle RPM
M05	A	Spindle Stop
M07	A	Air Blast ON
M08	A	Flood Coolant ON
M09	A	Air Blast and Flood Coolant OFF
M10	B	4th-Axis Clamp
M11	B	4th-Axis Un-Clamp
M17	E	Auto tool length measurement M17 T_ T = Tool length code to store measurement into
M19	C	Spindle Orientation M19 S_ S = Spindle angle offset (optional)
M20	D	Magazine In
M21	D	Magazine Out
M29	C	Rigid Tapping Mode ON
M30	A	Program End
M44	A	Tool Unclamp
M45	A	Tool Clamp
M98	A	Sub-Program Call
M99	A	Sub-Program End, Return to Start of Program
M110	E	Probe Enable, Probe Air Blast ON
M111	E	Probe Disable, Probe Air Blast OFF
M802	H	Coolant Through Spindle (CTS) ON
M803	H	Coolant Through Spindle (CTS) OFF
M817	F	Tool Measurement System ON
M818	F	Tool Measurement System OFF

M819	F	Spindle Probe System ON
M820	F	Spindle Probe System OFF
M821	F	Spindle Probe and Tool Measurement Systems OFF
M822	I	Oil Mist Extractor ON
M823	I	Oil Mist Extractor OFF
M834	G	Washdown ON
M835	G	Washdown OFF

Options:

- A. Standard control function
- B. Available with 4th rotary axis option
- C. Available with rigid tapping and spindle encoder option
- D. Available with carousel type automatic tool changer
- E. Available with generic probing system
- F. Available with spindle probing or tool measurement system
- G. Available on High Speed Machines
- H. Available for machines equipped with coolant through spindle system
- I. Available for machines equipped with oil mist extraction system

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 Program Folders

Macro programs will be searched for within system folders in the following order:

1. Current program folder
2. MTB1 FOLDER = < MotMTB1>
3. MTB2 FOLDER = < MotMTB2>
4. PROG FOLDER = <ACTPATH>\<MotUsrGcdDir>

MDI program is located in PROG FOLDER\<MotMDIGCode> and will observe the same file search order detailed.

14.3 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 14.3.1 details the names and ranges for user-defined and system reserved macro programs.

Table 14.3.1: Macro Name Ranges

Macro Name	Usage
00001 – 08999	Custom Macro
09000 – 09999	System Reserved

14.4 Calling a Macro Program

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 14.4.1 summarizes the differences between a subprogram call and a macro call.

Table 14.4.1: 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

14.5 Macro Variables

14.5.1 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 14.5.1. More detailed documentation and usage examples are provided in the following subsections.

Table 14.5.1: 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.

14.5.2 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 14.5.2 and Table 14.5.3 list the mapping of input parameters to corresponding local variables for Spec I and II control system configurations.

Table 14.5.2: 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 14.5.3: 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=50.0, and all other local variables would be equal to NULL or #0.

14.5.3 System Variables

System variables are reserved for system related constants or variables.

Table 14.5.4: System Variables

Address	Type	Usage
#0, #3100	R	NULL variable
#1000-#1031	R	Digital inputs (maps to I2432-I2463)
#1032	R	Digital inputs, hexa-decimal 32-bit encoding of #1000-#1031
#1100-#1131	R/W	Digital outputs (maps to I2464-I2495)
#1132	R/W	Digital outputs, hexa-decimal 32-bit encoding of #1000-#1031
#2001-#2200 #10001-#10400	R/W	Tool length wear offsets H1 to H200

#2201-#2400 #11001-#11400	R/W	Tool length offsets H1 to H200															
#2401-#2600 #12001-#12400	R/W	Tool diameter wear offsets D1 to D200															
#2601-#2800 #13001-#13400	R/W	Tool diameter offsets D1 to D200															
#3000	R/W	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 Example: #3000=1(ALARM MESSAGE) This block displays "3001 ALARM MESSAGE" during execution.															
#3001	R/W	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.															
#3002	R/W	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.															
#3003	W	Single block control Usage: #3003=N <table border="1"> <thead> <tr> <th>N</th><th>Single Block</th><th>Wait for M/S/T-Code</th></tr> </thead> <tbody> <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> </tbody> </table>	N	Single Block	Wait for M/S/T-Code	0	ON	ON	1	OFF	ON	2	ON	OFF	3	OFF	OFF
N	Single Block	Wait for M/S/T-Code															
0	ON	ON															
1	OFF	ON															
2	ON	OFF															
3	OFF	OFF															
#3004	W	Feed hold (FH) and feed rate override (FO) control Usage: #3004=N <table border="1"> <thead> <tr> <th>N</th><th>Feed Hold</th><th>Feed Override</th></tr> </thead> <tbody> <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> </tbody> </table>	N	Feed Hold	Feed Override	0	ON	ON	1	OFF	ON	2	ON	OFF	3	OFF	OFF
N	Feed Hold	Feed Override															
0	ON	ON															
1	OFF	ON															
2	ON	OFF															
3	OFF	OFF															
#3006	R/W	Program stop with message Usage: #3006=N(MESSAGE) Example: #3006=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.
#3011	R	Current date Returns: YYYYMMDD YYYY = Four-digit year MM = Two-digit month DD = Two-digit day
#3012	R	Current time Returns: HHMMSS HH = Two-digit hour (0-24) MM = Two-digit minutes (0-60) SS = Two-digit seconds (0-60)
#3101	R	PI constant = 3.14159265358979323846
#3102	R	Exponential constant = 2.71828182845904523536
#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

14.6 Macro Functions

Table 14.6.1: Macro 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	$\#i = \text{SIN}[\#j]$	
Arcsine	$\#i = \text{ASIN}[\#j]$	
Cosine	$\#i = \text{COS}[\#j]$	
Arccosine	$\#i = \text{ACOS}[\#j]$	
Tangent	$\#i = \text{TAN}[\#j]$	
Arctangent	$\#i = \text{ATAN}[\#j] / [\#k]$	
Square root	$\#i = \text{SQRT}[\#j]$	
Absolute	$\#i = \text{ABS}[\#j]$	
Round	$\#i = \text{ROUND}[\#j]$	
Round up	$\#i = \text{FUP}[\#j]$	
Round down	$\#i = \text{FIX}[\#j]$	
Natural log	$\#i = \text{LN}[\#j]$	
Exponential	$\#i = \text{EXP}[\#j]$	
Power	$\#i = \text{POW}[\#j]$	
Boolean AND	$\#i = \#j \text{ AND } \#k$	
Boolean OR	$\#i = \#j \text{ OR } \#k$	
Boolean XOR	$\#i = \#j \text{ XOR } \#k$	
Print	$\#3999 = 1250$ POPEN DPRNT[(TEXT)] DPRNT[X#3999[34]] PCLOS	Outputs to a filename O<#3999>, with contents: "(TEXT) X250.0000" (In DPRNT, the number in brackets after the variable specifies the number of integer digits and the number of decimal digits.)

14.7 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

14.8 Bracket Nesting

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

14.9 Branch and Repetition

Operators are used within brackets “[]” to compare variables or numbers. These expressions can then be used as conditions to jump to certain locations or to repeatedly loop within a section. The following tables detail the supported operators, available control functions, and its syntax.

Example:

Condition expression: IF [#1 GT 20.0] GOTO 90

This conditional statement compares #1 against the number 20.0, and if it evaluates to true, then program will jump to and continue executing at the line labelled “N90”.

Table 14.9.1: 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 (<=)

Example:

#1 = 10.0

[#1 EQ 10.0] evaluates to TRUE

[#1 NE 10.0] evaluates to FALSE

[#1 GT 10.0] evaluates to FALSE

[#1 GE 10.0] evaluates to TRUE

[#1 LT 10.0] evaluates to FALSE

[#1 LE 10.0] evaluates to TRUE

Table 14.9.2: Control and Loop Functions

Function	Description
GOTO #i	Unconditional jump to block sequence number.
IF [...] GOTO #i	If expression in brackets evaluates to true, jump to a block sequence number.
IF [...] THEN #i=#j	If expression evaluates to true, perform a set value function.
WHILE [...] DO #i ... END #i	While expression in brackets evaluates to true, repeatedly performs the sequence, otherwise proceeds to corresponding end block labelled #i. While-Do ranges can be nested up to three levels but must not 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 generally involves the following workflow:

1. Go to **FILE MANAGER**, choose **NEW** to create a new program, and choose the Conversational type of program.
2. Configure the **PROGRAM SETTINGS** to set program settings such as the active work offset, the safe height, program units, and start or finish position.
3. Set up the **TOOL LIST** by inputting or loading a set of tools for use in the program.
4. Select the **PROGRAM** button on the left to edit the program.
5. If a drawing is required for this program, in the CAD Drawing Editor window, toggle the **FILE** menu, then select **OPEN** to load a drawing or **NEW** to create a new drawing. Loaded drawings can be scaled, rotated, or mirrored using the **VIEW** mode functions. Drawings may be edited using **DRAW** and **EDIT** functions. The origin of the drawing corresponding to workpiece zero should be set using the **SET ZERO** function.
6. Events can then be selected to add into the program by clicking the **ADD** button and then choosing it from within groups consisting of Hole sequences, Contours, Pockets, Face Mill, Position, Pattern, and Setup events. Each event has parameters that need to be input and configured. For events that require associated positions or paths, its geometry may be selected directly from the drawing by clicking on **SELECT GEOMETRY** or drawn point to point by clicking on **EDIT GEOMETRY** at the bottom of its event configuration window. Pressing **SAVE** will commit the event into the main program event list.
7. **PREVIEW** the program to inspect the tool paths for each event.
8. **SAVE** the program to generate its G-code program.
9. **EXIT** the Conversational Editor and run the program in **AUTO** mode.

The detailed functions within each Conversational Editor screen and details regarding each Conversational Event are described in the following sections of this chapter.

15.3 Main Screen Functions

Once a new Conversational Program is created or an existing program is loaded from the File Manager, the main Conversational Editor screen is displayed.

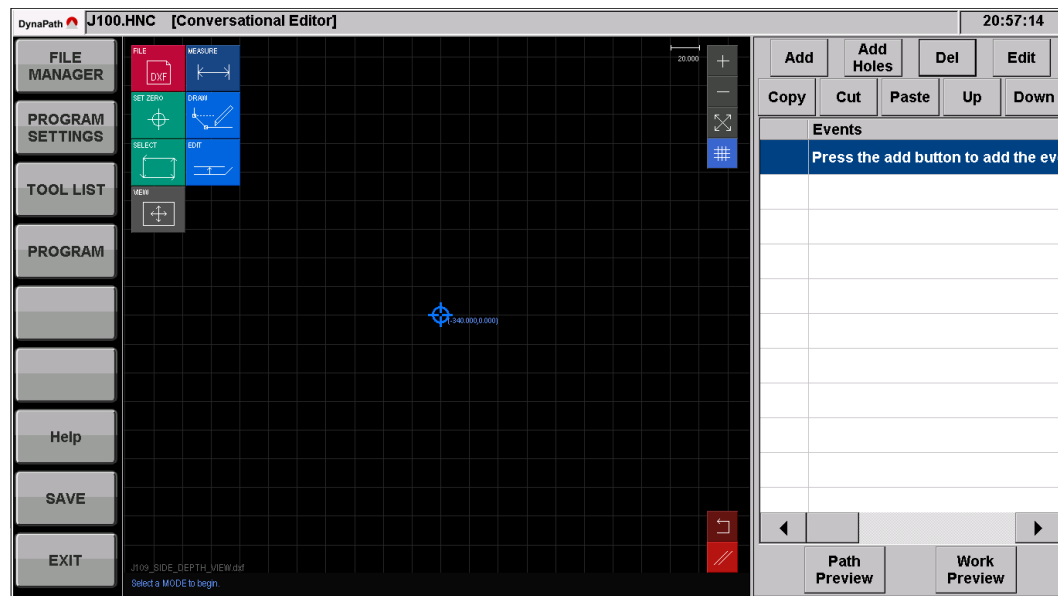


Figure 15.3-1: Conversational Editor Main Screen

Along the left side of the screen are functions related to the entire program, while the functions along the right side of the screen relate to adding, removing, and configuring the machining events within the program. A description of each button function is listed as follows:

- **FILE MANAGER** returns to the file list screen to select or create a program.
- **PROGRAM SETTINGS** screen allows the setting of the work offset, the safe height of travel between events, program start or finish position, part stock material, inch or metric units, and other program details.
- **TOOL LIST** screen shows the tool list associated with this program for configuration and management.
- **PROGRAM** screen shows the main program edit screen, with the drawing view in the center and the event list on the right.
- **SAVE** will save the current program and generates its G-code program.
- **EXIT** quits the Conversational Editor.
- **ADD** allows selecting a new event and inserts it into the event list.
- **ADD HOLES** creates a new hole event sequence and adds it into the event list.
- **DEL** deletes the selected event from the event list.
- **EDIT** opens the selected event to configure its parameters.
- **COPY / CUT / PASTE / UP / DOWN** affects the selected event in the list and allows copying, cutting, and inserting, or moving it to another location in the list.
- **PATH / WORK PREVIEW** shows a toolpath preview or top CAD view of the programmed events.

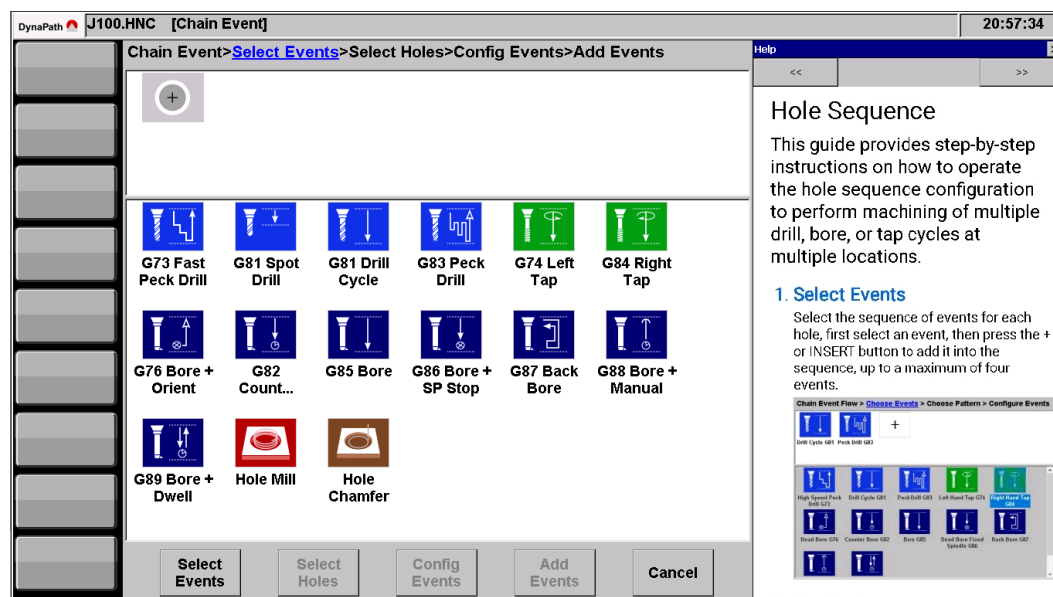
The center of the screen contains the CAD Drawing Editor module. This window allows the loading or creating of a DXF file and drawing or modifying the CAD file for use in the current program. The CAD Drawing menu is located on the upper left corner of this window and offers the following general functions. For more information regarding the operation of the CAD Drawing Editor, reference the next chapter.

1. **FILE** mode allows opening, saving, and creating a new drawing.
2. **SET ZERO** mode allows set up of a part zero location.
3. **SELECT** mode allows the selection of geometry to import into events.
4. **VIEW** mode allows scaling, rotating, or mirroring the entire drawing.
5. **MEASURE** mode allows labelling of dimensions on the drawing.
6. **DRAW** mode allows creating lines, arcs, circles, rectangles, and modifying geometry within the drawing.
7. **EDIT** mode allows creating fillets, chamfers, copying, rotating, mirroring, and moving elements within the drawing.

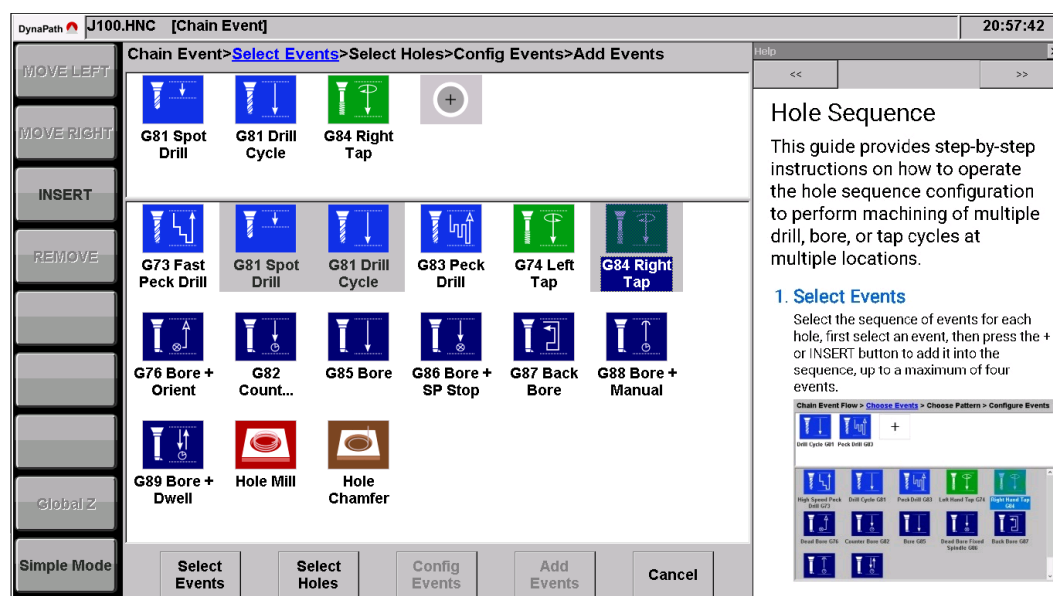
15.4 Hole Sequence

The Hole Sequence creates a series of events for machining holes at one or more locations using a combination of drill, bore, tap, or other hole machining cycles. The following guide provides an illustrated workflow on how to configure a Hole Sequence, and the subsections detail parameters for each hole machining event.

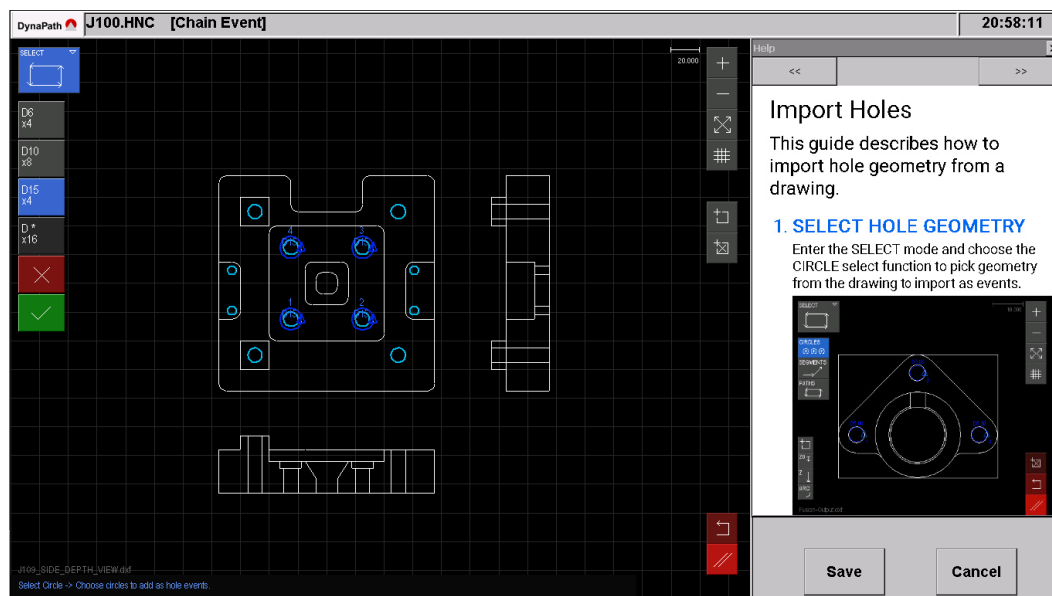
1. From the main program screen, add a **HOLE SEQUENCE**.



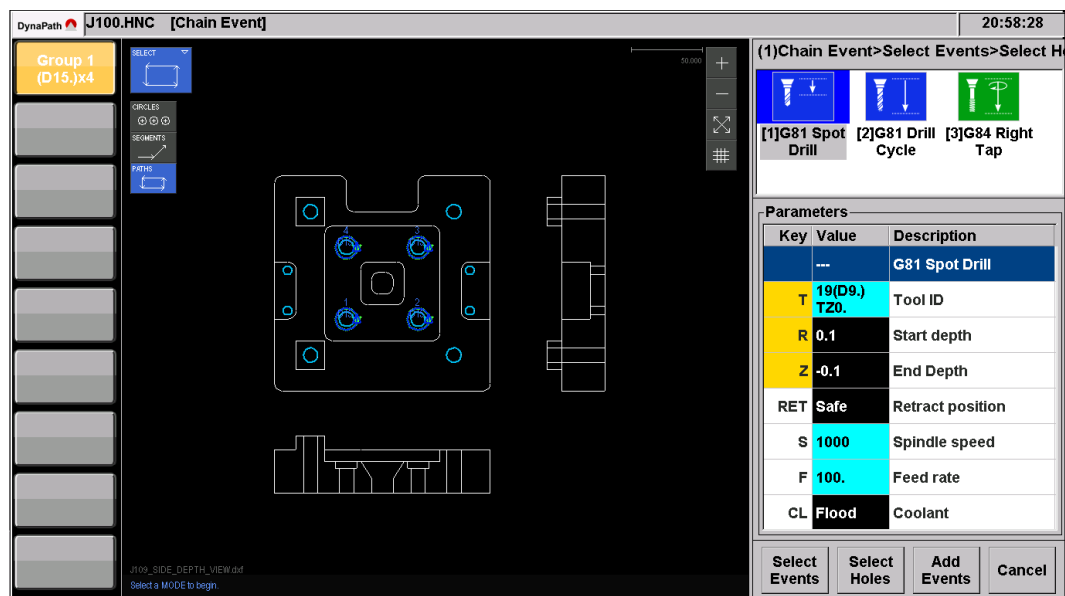
2. Choose a combination of hole machining events to perform for each hole location by double clicking each event, or by selecting the event and pressing the add icon to add it into the sequence displayed along the top row. When complete, press **SELECT HOLES** button to proceed with the selection of hole locations.



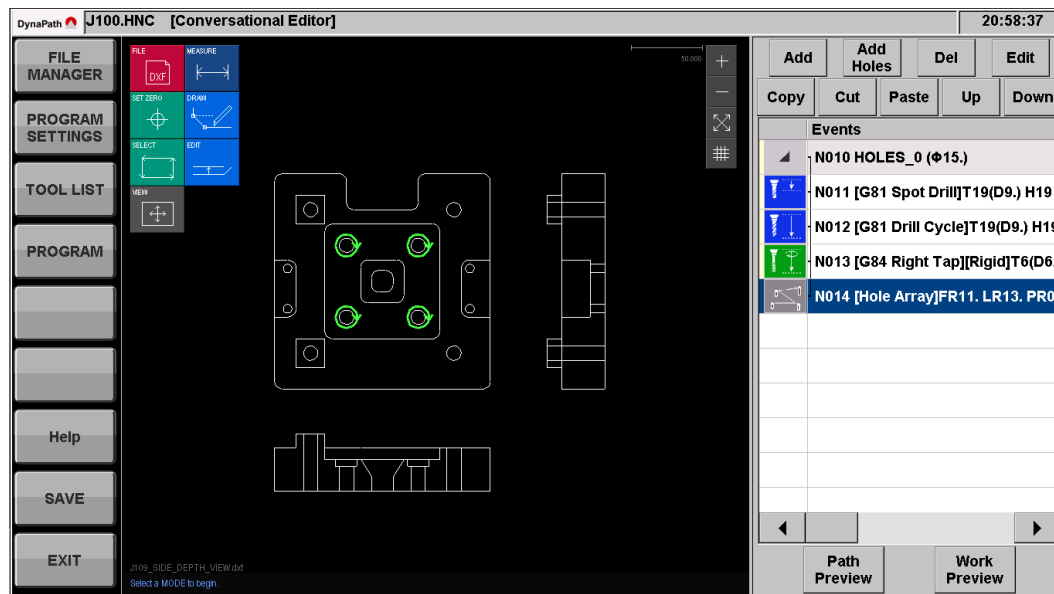
3. Select individual circles from the drawing or drag select a box region to allow automatic circle finding which will present groups of circles to select along the left of the screen. Once desired hole locations are selected, they will be highlighted in dark blue. Click **SAVE** when ready to proceed to the next step.



4. Configure the event parameters for each hole machining event in the sequence. This involves setting parameters such as the tool number, start and end depths, feed rate, and spindle speeds. Click **ADD EVENTS** to finish configuration and add the sequence into the main program event list.



5. Once the sequence of hole events has been added to the main program, the entire hole sequence may be revisited or modified by double clicking on the title event in the sequence or by highlighting it and clicking **EDIT**. Each individual hole machining event may also be edited by highlighting it and clicking **EDIT**.



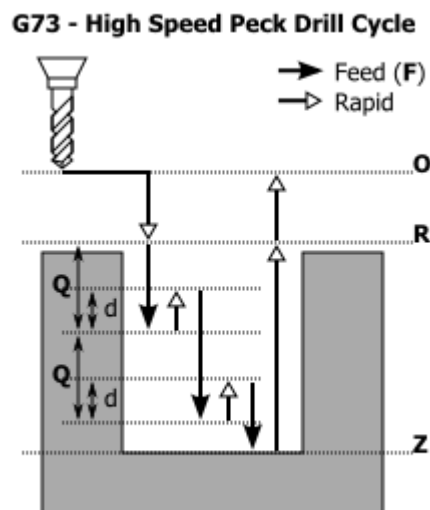
Notes:

1. To draw circles representing hole locations, reference the CAD Drawing chapter in this manual. This is the best and easiest way to select holes locations.
2. In the case where hole locations are to be inputted by hand, toggle the SIMPLE / ADVANCED MODE button on the lower left of the event selection screen in step 2. Then in the hole selection screen in step 3, a table will be presented on the right with an option selection located at the bottom of the screen to add holes manually by inputting X and Y locations.

15.4.1 G73 High Speed Peck Drill

The G73 High Speed Peck Drill event performs incremental peck drilling. This event is different from the G83 Peck Drill event in that the tool is retracted a small distance relative to each incremental depth. The result is a faster peck cycle. If enabled, the incremental peck depth Q and relative retract height D parameters can be set. If not enabled, these two parameters are set globally in the system parameters. The tool retraction after each peck occurs at the rapid rate. Relevant parameters for the G73 High Speed Peck Drill event are detailed in the following list. Figure is an illustration for this event.

- **X, Y** End point coordinates (absolute coordinates)
- **Z** Cycle depth (absolute coordinates)
- **D** Peck retract distance (positive number)
- **R** Transition height (absolute coordinates)
- **Q** Peck increment (positive number)
- **F** Feed rate (positive number)



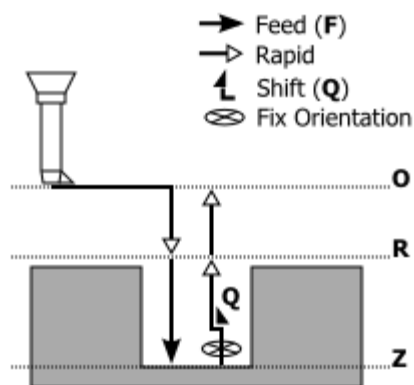
High Speed Peck Drill G73

15.4.2 G76 Dead Bore

The G76 Dead Bore event is illustrated in Figure . This event starts with a feed to the cycle depth, then stops the spindle at a certain orientation. The spindle is then shifted a certain amount away from the wall, and then retracted. If enabled, the spindle shift parameter can be set with an upper limit being the tool radius, otherwise this parameter is set automatically within the canned cycle. The spindle shift prevents the tool from dragging along the wall during retraction. Relevant parameters for this event are detailed in the following list.

- **X, Y** End point coordinates (absolute coordinates)
- **Z** Cycle depth (absolute coordinates)
- **R** Transition height (absolute coordinates)
- **Q** Spindle shift distance (positive number)
- **F** Feed rate (positive number)

G76 - Dead Bore Cycle



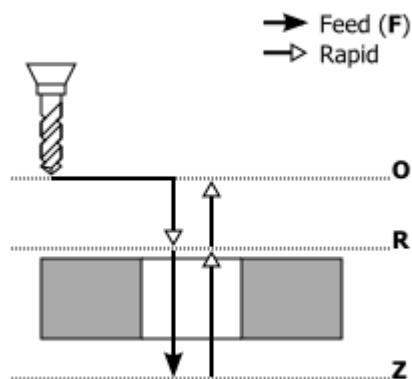
Dead Bore G76

15.4.3 G81 Drill Cycle

The G81 Drill event is illustrated in Figure . This event is a basic drill cycle where the tool tip is advanced at feed rate to the cycle depth and then retracted. Relevant parameters for this event are detailed in the following list.

- **X, Y** End point coordinates (absolute coordinates)
- **Z** Cycle depth (absolute coordinates)
- **R** Transition height (absolute coordinates)
- **F** Feed rate (positive number)

G81 - Drill Cycle

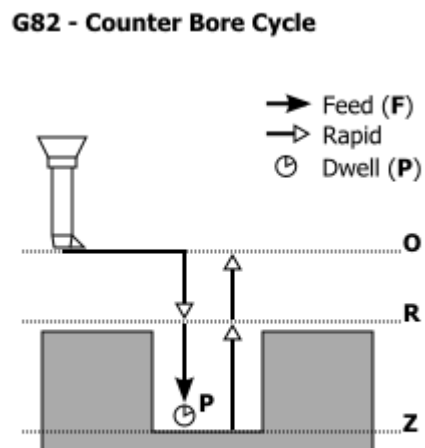


Drill Cycle G81

15.4.4 G82 Counter Bore

The G82 Counter Bore event is illustrated in Figure . This event is a simple bore cycle that begins with a feed to the cycle depth, then a dwell for a set duration, and concludes with a retraction. The dwell can produce a quality hole bottom. Relevant parameters for this event are detailed in the following list.

- **X, Y** End point coordinates (absolute coordinates)
- **Z** Cycle depth (absolute coordinates)
- **R** Transition height (absolute coordinates)
- **P** Dwell duration (seconds)
- **F** Feed rate (positive number)

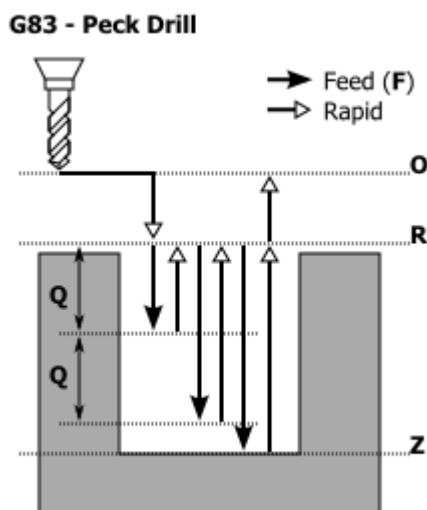


Counter Bore G82

15.4.5 G83 Peck Drill

The G83 Peck Drill event is illustrated in Figure . This event is different from the G73 High Speed Peck event in that for each peck, the tool is retracted back to the transition height. If enabled, the incremental peck depth Q can be set. If not enabled, a global value is set in the system parameters. Peck drilling is the preferred cycle for smaller or deeper holes, which require more complete scrap discharge. Relevant parameters for this event are detailed in the following list.

- X, Y End point coordinates (absolute coordinates)
- Z Cycle depth (absolute coordinates)
- R Transition height (absolute coordinates)
- Q Peck depth (positive number)
- F Feed rate (positive number)



Peck Drill G83

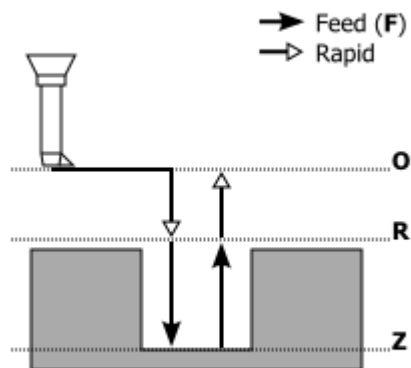
15.4.6 G85 Bore

The G85 Bore event is illustrated in Figure . This event performs a feed to the cycle depth, then feeds back up to the transition height, and concludes with a retraction.

Relevant parameters for this event are detailed in the following list.

- **X, Y** End point coordinates (absolute coordinates)
- **Z** Cycle depth (absolute coordinates)
- **R** Transition height (absolute coordinates)
- **F** Feed rate (positive number)

G85 - Bore Cycle



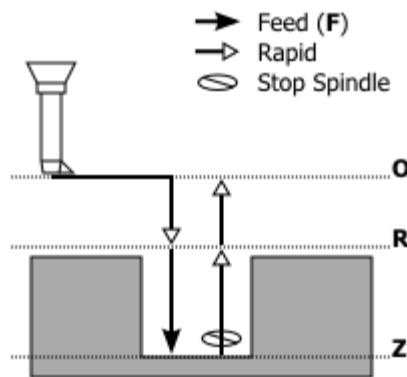
Bore G85

15.4.7 G86 Dead Bore Fixed Spindle

The G86 Dead Bore Fixed event is illustrated in Figure .This event starts with a feed to the cycle depth. Upon reaching cycle depth, the spindle is stopped to a fixed orientation. Then the tool is retracted. Relevant parameters for this event are detailed in the following list.

- **X, Y** End point coordinates (absolute coordinates)
- **Z** Cycle depth (absolute coordinates)
- **R** Transition height (absolute coordinates)
- **F** Feed rate (positive number)

G86 - Dead Bore Fixed Cycle



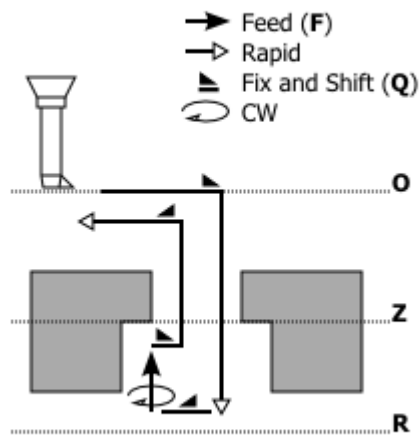
Dead Bore Fixed Spindle G86

15.4.8 G87 Back Bore

The G87 Back Bore event is illustrated in Figure . This event is used to bore the back side of a hole. The event begins with a spindle stop at a fixed orientation over the end point. Then the spindle is shifted to fit into the hole and is then lowered into the hole. After clearing the hole, the spindle is shifted back, then spun up to speed. The tool is then fed upwards at feed rate for a distance equivalent to the cycle depth. Upon reaching this height, the spindle is turned off, and another spindle shift Q is initiated for the tool to be retracted. Relevant parameters for this event are detailed in the following list.

- **X, Y** End point coordinates (absolute coordinates)
- **Z** Cycle depth (absolute coordinates)
- **R** Transition height (absolute coordinates)
- **Q** Spindle shift distance (positive number)
- **P** Dwell duration (seconds)
- **F** Feed rate (positive number)

G87 - Back Bore Cycle



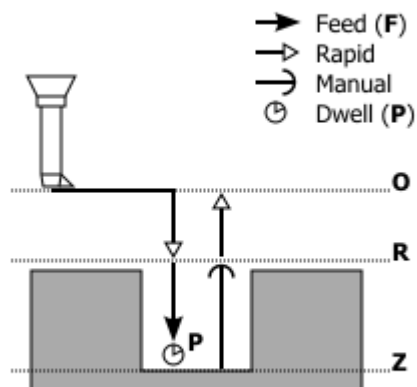
Back Bore G87

15.4.9 G88 Dead Spindle Bore and Dwell

The G88 Dead Spindle Bore and Dwell event is illustrated in Figure . This event starts with a feed to the cycle depth, then a dwell is performed, and concludes as the machine waits for manual retraction. Relevant parameters for this event are provided in the following list.

- **X, Y** End point coordinates (absolute coordinates)
- **Z** Cycle depth (absolute coordinates)
- **R** Transition height (absolute coordinates)
- **P** Dwell duration (seconds)
- **F** Feed rate (positive number)

G88 - Dead Spindle Bore and Dwell

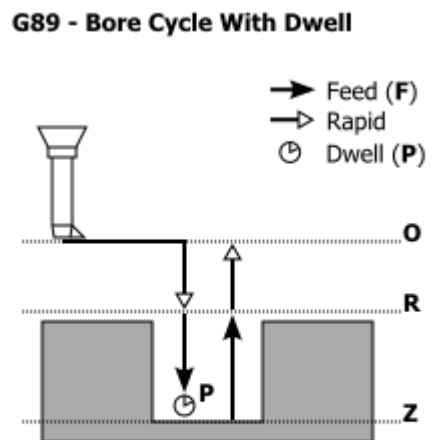


Dead Spindle Bore and Dwell G88

15.4.10 G89 Bore Cycle with Dwell

The G89 Bore with Dwell event is illustrated in Figure . This event performs a feed to the cycle depth, dwells for a set duration, then feeds back up to the transition height, and concludes with retraction. Relevant parameters for this event are detailed in the following list.

- **X, Y** End point coordinates (absolute coordinates)
- **Z** Cycle depth (absolute coordinates)
- **R** Transition height (absolute coordinates)
- **P** Dwell duration (seconds)
- **F** Feed rate (positive number)



Bore Cycle with Dwell G89

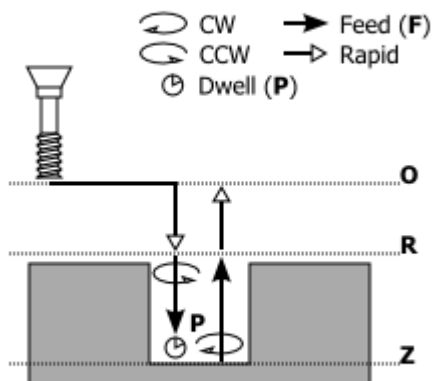
15.5 Tap Events

15.5.1 G74 Left Hand Tap

The G74 Left Hand Tap event is illustrated in Figure Figure . This is a tap event that performs a CCW spindle feed to the cycle depth, dwells for a set duration, then performs a CW spindle feed back to the transition height and ends with retraction. Relevant parameters for this event are detailed in the following list.

- **TT** Tapping mode, rigid (M29) or normal tapping
- **X, Y** End point coordinates (absolute coordinates)
- **R** Transition height (absolute coordinates)
- **Z** Cycle depth (absolute coordinates)
- **Q** Peck depth (if greater than R-Z, no peck occurs)
- **P** Dwell duration (seconds)
- **F** Thread pitch (positive number) if TT set to Rigid, otherwise is the actual feed rate (mm/min or in/min) in coordination with spindle RPM of the non-rigid tapping motion.
- **S** Spindle RPM

G74 - Left Handed Tapping Cycle



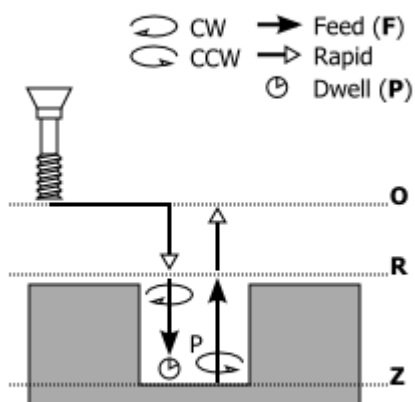
Left Hand Tap G74

15.5.2 G84 Right Hand Tap

The G84 Right Hand Tap event is illustrated in Figure . This is a tap event that performs a CW spindle feed to the cycle depth, dwells for a set duration, then performs a CCW spindle feed back to the transition height and ends with retraction. Relevant parameters for this event are detailed in the following list.

- **TT** Tapping mode, rigid (M29) or normal tapping
- **X, Y** End point coordinates (absolute coordinates)
- **R** Transition height (absolute coordinates)
- **Z** Cycle depth (absolute coordinates)
- **Q** Peck depth (if greater than R-Z, no peck occurs)
- **P** Dwell duration (seconds)
- **F** Thread pitch (positive number) if TT set to Rigid, otherwise is the actual feed rate (mm/min or in/min) in coordination with spindle RPM of the non-rigid tapping motion.
- **S** Spindle RPM

G84 - Tapping Cycle



Right Hand Tap G84

15.6 Contour Group Events

The Contour Group of events execute milling motion along a specified path.

15.6.1 Contour Event

The Contour Event starts with a tool change, then activates tool length compensation, followed by a move down to the safe height where the spindle is run, and the coolant is activated. Then the tool is rapid moved down in Z to the plunge height, where it slows to plunge speed, and continues to lower to the first incremental depth. The tool then proceeds to mill at feed rate along the specified profile path and upon reaching the end of the path, the tool is retracted. This process repeats incrementally until the final depth is reached. Geometry and parameters are detailed in the following lists.

Geometry:

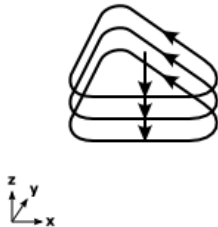
- **Accepts closed and open irregular paths**
- **Accepts multiple paths**

Contour parameters:

- **T** Tool number
- **RF** Roughing passes, Finishing passes, or both
- **COM** Tool radius compensation (Off, Left, Right)
- **RP** Plunge safe height (incremental distance above WP)
- **WP** Start depth (absolute coordinate)
- **D** End depth (absolute coordinate)
- **L** Rough step-down increment (positive number)
- **LS** Lead in/out (Extend, Recede, or Custom)
- **LID / LOD** Lead in/out tangential extension distance
- **LL** Lead line distance
- **LA** Lead arc radius
- **PM** Plunge method (Straight, Ramp, Straight Forward and Back, Ramp Forward and Back)
- **KTD** Keep tool down (for open paths)
- **NT** Side passes (None, Count, or Distance)
- **N** Number of side passes
- **ND** Side passes total distance
- **STEP** Rough passes step-over distance (%Tool Diameter if greater than 1)
- **FN** Finish passes count
- **FSEP** Finish passes step-over distance (%Tool Diameter if greater than 1)
- **FZ** Finish pass step-down
- **AXY** Final allowance to leave on sides
- **AZ** Final allowance to leave on bottom
- **S** Spindle speed (RPM positive or negative integer)
- **P** Plunge rate (positive number)
- **F** Feed rate (positive number)
- **CL** Coolant type selection (Off, Air, Flood, CTS, CTS + Flood)

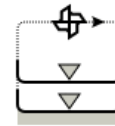
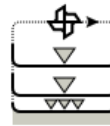
Contour Event**Complex Contour**

→ Feed (F)

**RF Roughing, Finishing Passes**

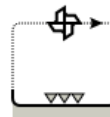
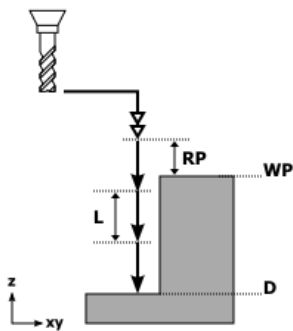
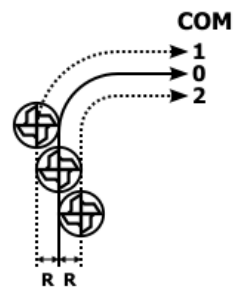
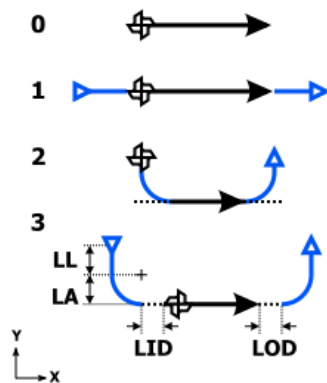
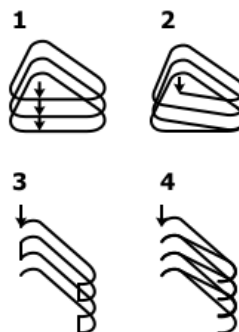
0 ▽ + ▽▽▽

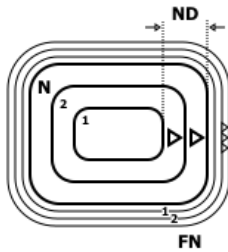
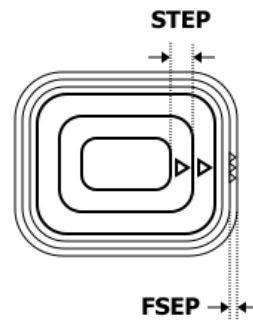
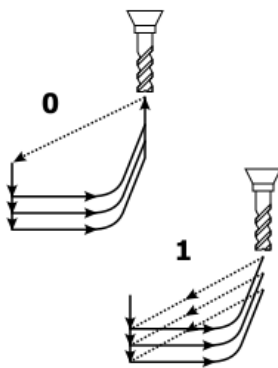
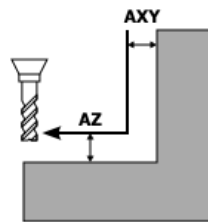
1 ▽



2 ▽▽▽

3 ▽▽▽

**RP, WP, D, L Depths****COM Tool Radius Compensation****LS, LID, LOD, LL, LA Lead In / Out****PM Plunge Method**

N, ND, FN Side Passes**STEP, FSEP Rough, Finish Step-Over****KTD Keep Tool Down****AXY, AZ Allowances**

15.6.2 Helix Event

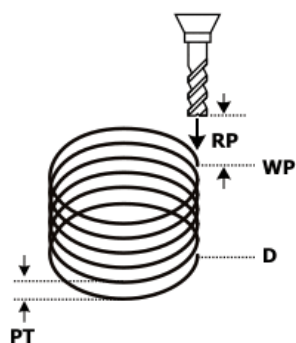
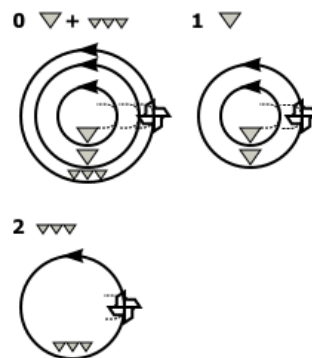
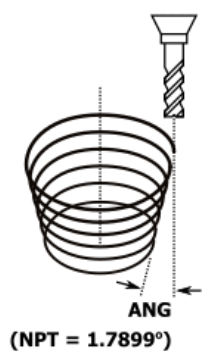
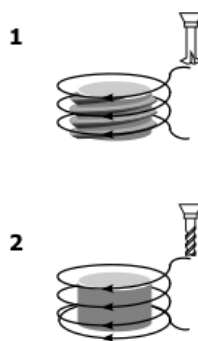
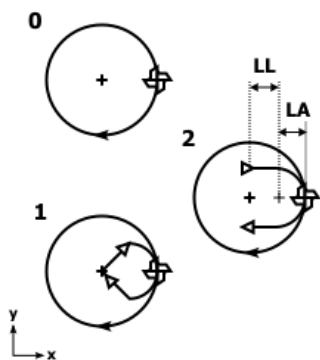
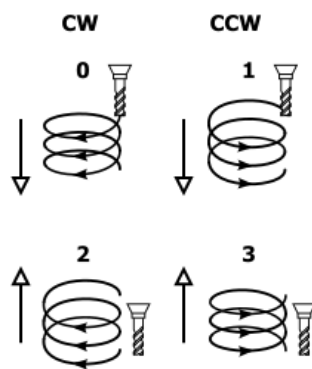
The Helix Event is used to perform milling of circular holes or bosses by employing a helical toolpath. This event starts with a tool change, then activates tool length compensation, followed by a move down to the safe height where the spindle is run, and the coolant is activated. Then the tool is rapid moved down in Z to the plunge height, where it slows to plunge speed, and begins helical interpolation towards a target depth. Upon reaching the target depth, the tool is retracted. The geometry and parameters for this event are detailed in the following lists.

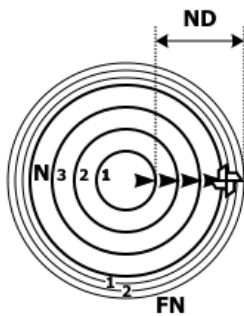
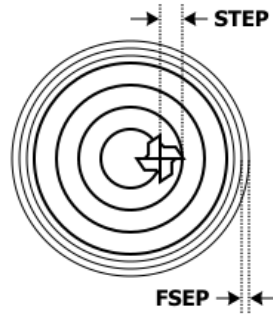
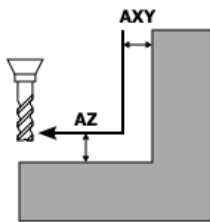
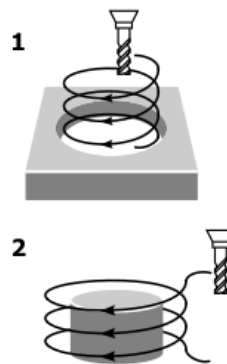
Geometry:

- **Accepts closed circular paths of variable diameter**
- **Accepts multiple paths**

Parameters:

- **T** Tool number
- **RF** Roughing passes, Finishing passes, or both
- **MODE** Thread or Contour with a finish pass at final depth
- **DIR** Direction (CW Down, CCW Down, CW Up, CCW Up)
- **COM** Machining inside or outside of circle geometry
- **RP** Plunge safe height (incremental distance above WP)
- **WP** Start depth (absolute coordinate)
- **D** End depth (absolute coordinate)
- **L** Incremental depth or pitch of helix (positive number)
- **ANG** Taper angle (positive angle measured from vertical Z)
- **LS** Lead in/out (None, Automatic, Center, Custom)
- **LL** Lead line distance
- **LA** Lead arc radius
- **NT** Side passes (None, Count, or Distance)
- **N** Number of side passes
- **ND** Side passes total distance
- **STEP** Rough passes step-over distance (%Tool Diameter if greater than 1)
- **FSEP** Finish passes step-over distance (%Tool Diameter if greater than 1)
- **FN** Finish passes count
- **AXY** Final allowance to leave on sides
- **AZ** Final allowance to leave on bottom
- **S** Spindle speed (RPM positive or negative integer)
- **P** Plunge rate (positive number)
- **F** Feed rate (positive number)
- **CL** Coolant type selection (Off, Air, Flood, CTS, CTS + Flood)

RP, WP, D, L Depths**RF Roughing, Finishing Passes****ANG Taper Angle****MODE Thread or Contour****LS, LL, LA Lead In / Out****DIR Helix Direction**

N, ND, FN Side Passes**STEP, FSEP Rough, Finish Step-Over****AXY, AZ Allowances****COM Inside / Outside**

15.6.3 Thread Mill Event

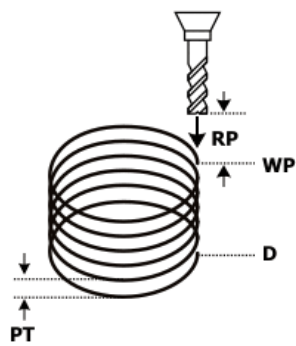
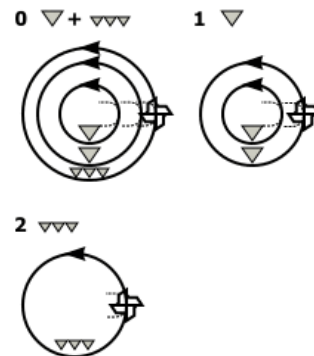
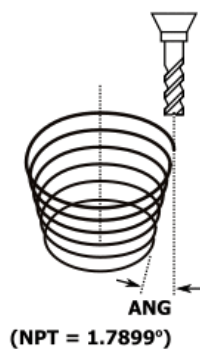
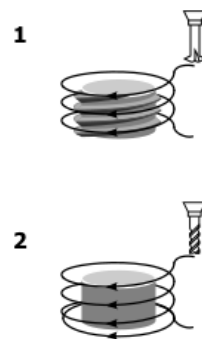
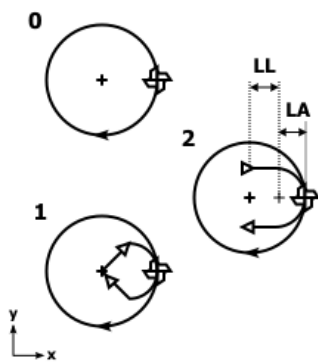
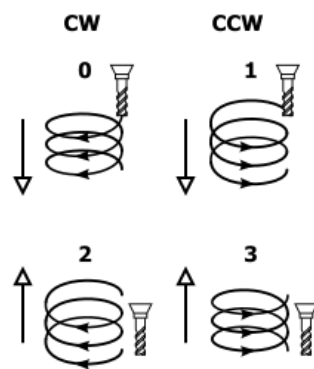
The Thread Mill Event is used to perform milling of internal or outer diameter threads using a helical toolpath. This event starts with a tool change, then activates tool length compensation, followed by a move down to the safe height where the spindle is run, and the coolant is activated. Then the tool is rapid moved down in Z to the plunge height, where it slows to plunge speed, and begins helical interpolation towards a target depth. Upon reaching the target depth, the tool is retracted. The geometry and parameters for this event are detailed in the following lists.

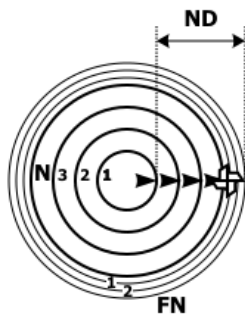
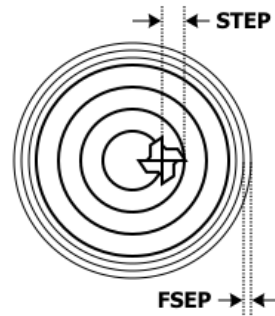
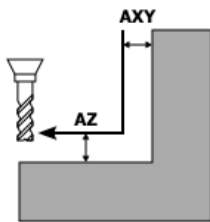
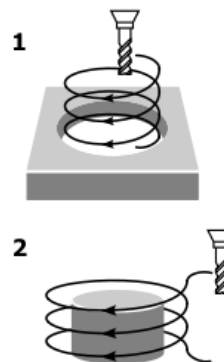
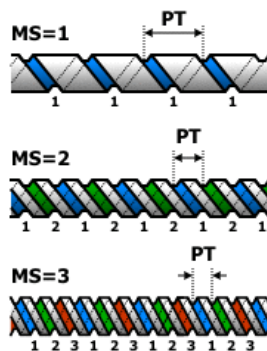
Geometry:

- **Accepts closed circular paths of variable diameter**
- **Accepts multiple paths**

Parameters:

- **T** Tool number
- **RF** Roughing passes, Finishing passes, or both
- **MODE** Thread or Contour with a finish pass at final depth
- **DIR** Direction (CW Down, CCW Down, CW Up, CCW Up)
- **COM** Machining inside or outside of circle geometry
- **RP** Plunge safe height (incremental distance above WP)
- **WP** Start depth (absolute coordinate)
- **D** End depth (absolute coordinate)
- **L** Pitch of thread (positive number)
- **ANG** Taper angle (positive angle measured from vertical Z)
- **SAN** Start angle in degrees (X+ direction is zero, positive going CW)
- **MS** Multi-start thread count
- **LS** Lead in/out (None, Automatic, Center, Custom)
- **LL** Lead line distance
- **LA** Lead arc radius
- **NT** Side passes (None, Count, or Distance)
- **N** Number of side passes
- **ND** Side passes total distance
- **STEP** Rough passes step-over distance (%Tool Diameter if greater than 1)
- **FSEP** Finish passes step-over distance (%Tool Diameter if greater than 1)
- **FN** Finish passes count
- **AXY** Final allowance to leave on sides
- **AZ** Final allowance to leave on bottom
- **S** Spindle speed (RPM positive or negative integer)
- **P** Plunge rate (positive number)
- **F** Feed rate (positive number)
- **CL** Coolant type selection (Off, Air, Flood, CTS, CTS + Flood)

RP, WP, D, L Depths**RF Roughing, Finishing Passes****ANG Taper Angle****MODE Thread or Contour****LS, LL, LA Lead In / Out****DIR Helix Direction**

N, ND, FN Side Passes**STEP, FSEP Rough, Finish Step-Over****AXY, AZ Allowances****COM Inside / Outside****MS Multi-Start Thread**

15.6.4 Chamfer Event

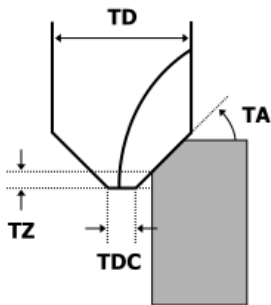
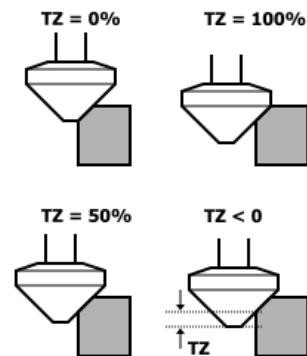
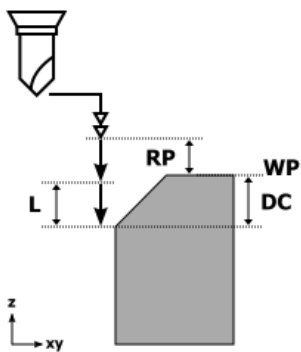
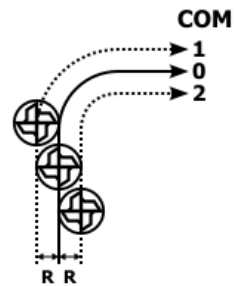
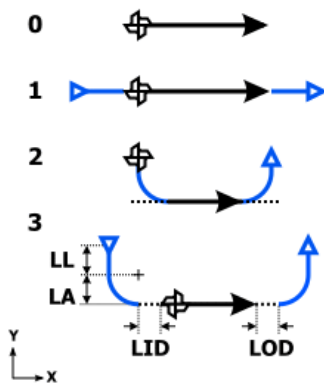
The Chamfer Event is used to perform milling of a chamfer edge using a shaped chamfer tool consisting of a tip diameter, tip angle, and max tool diameter. This event starts with a tool change, then activates tool length compensation, followed by a move down to the safe height where the spindle is run, and the coolant is activated. Then the tool is rapid moved down in Z to the plunge height, where it slows to plunge speed, and begins milling along the specified path. This process continues for each step-down or incremental depth until the target depth is reached, then the tool is retracted. The geometry and parameters for this event are detailed in the following lists.

Geometry:

- **Accepts closed and open irregular paths**
- **Accepts multiple paths**

Parameters:

- **T** Tool number
- **TDC** Tool tip diameter
- **TA** Tool tip angle in degrees (measured from horizontal flat)
- **TZ** Tool tip Z-offset (% of max diameter contacting the top chamfer edge)
- **COM** Tool radius compensation (Off, Left, Right)
- **RP** Plunge safe height (incremental distance above WP)
- **WP** Start depth (absolute coordinate)
- **DC** Chamfer depth or height in Z of the chamfer
- **LN** Number of degressive depth passes
- **LID / LOD** Lead in/out tangential extension distance
- **LL** Lead line distance
- **LA** Lead arc radius
- **S** Spindle speed (RPM positive or negative integer)
- **P** Plunge rate (positive number)
- **F** Feed rate (positive number)
- **CL** Coolant type selection (Off, Air, Flood, CTS, CTS + Flood)

TDC, TA Tool Tip Diameter, Angle**TZ Tool Tip Z-Offset****RP, WP, DC Chamfer Depths****COM Tool Radius Compensation****LID / LOD, LL, LA Lead In / Out**

15.7 Pocket Events

The Pocket Group of events remove material within a specified area.

15.7.1 Pocket Event

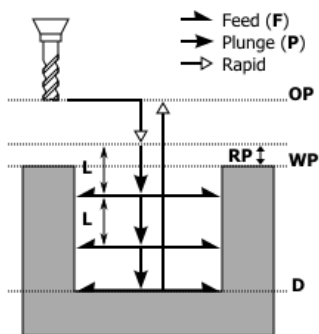
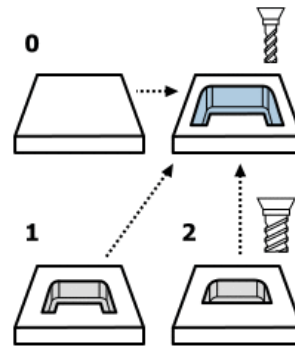
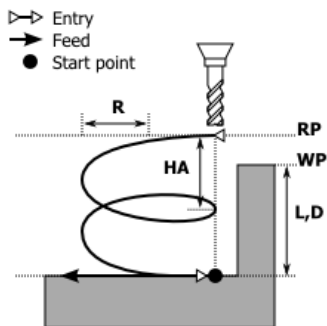
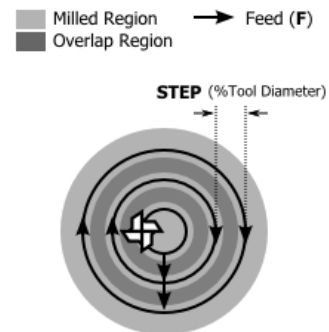
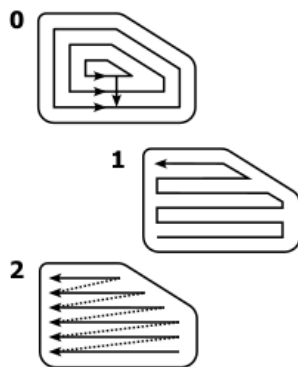
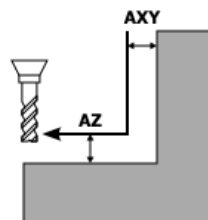
The Pocket Event removes material within a specified area or closed path. This event starts with a tool change, then activates tool length compensation, followed by a move down to the safe height where the spindle is run, and the coolant is activated. Then the tool is rapid moved down in Z to the plunge height, where it slows to plunge speed until the first step-down or incremental depth is reached, then proceeds to mill at feed rate within the specified shape, layer by layer in Z until the final depth is reached. The geometry and parameters for this event are detailed in the following lists.

Geometry:

- **Accepts closed and open irregular paths**
- **Accepts multiple paths**
- **For pockets containing islands, select the outer closed path in CW direction, and inner island paths in CCW direction, where tool always travels on the left.**

Parameters:

- **T** Tool ID
- **RF** Roughing passes, Finishing passes, or both
- **RP** Plunge safe height (incremental distance above WP)
- **WP** Start depth (absolute coordinate)
- **D** End depth (absolute coordinate)
- **L** Rough step-down increment (positive number)
- **STEP** Rough passes step-over distance (%Tool Diameter if greater than 1)
- **RES** Rest machining (None, XY and Z, Tool Diameter)
- **FM** Finish method (Both, Wall, Bottom, Custom)
- **FSEP** Finish passes step-over distance (%Tool Diameter if greater than 1)
- **FZ** Finish pass step-down
- **FNX** Finish wall passes count
- **FNZ** Finish bottom passes count
- **PM** Plunge method (Straight, Ramp, Helical)
- **PR** Helical plunge diameter
- **PA** Plunge angle
- **MM** Milling method
- **MO** Milling order
- **AXY** Final allowance to leave on sides
- **AZ** Final allowance to leave on bottom
- **S** Spindle speed (RPM positive or negative integer)
- **P** Plunge rate (positive number)
- **F** Feed rate (positive number)
- **CL** Coolant type selection (Off, Air, Flood)

RP, WP, D, L Pocket Depths**Pocket Peck Milling****RES Rest Machining****PR, PA Helical Plunge****Pocket Mill Helical Entry****STEP, FSEP Step-Over****Mill Step-over****MM Milling Method****AXY, AZ Allowances**

15.7.2 Tapered Face Event

The Tapered Face Event removes material along a sloped or tapered surface. This event starts with a tool change, then activates tool length compensation, followed by a move down to the safe height where the spindle is run, and the coolant is activated. Then the tool is rapid moved down in Z to the plunge height, where it slows to plunge speed until the first step-down or incremental depth is reached, then proceeds to mill at feed rate over the specified tapered surface, stepping over and down the full surface is machined. The geometry and parameters for this event are detailed in the following lists.

Geometry:

- **Accepts one open or closed path**
- **Accepts two open paths, or two closed paths with one fully contained within, with both paths observing the same path direction**

Parameters:

- **T** Tool ID
- **RF** Roughing passes, Finishing passes, or both
- **MODE** One path or Two path mode
- **COM** Tool radius compensation (Off, Left, Right)
- **TA** Taper angle setting for when MODE = one path mode
- **RP** Plunge safe height (incremental distance above WP)
- **D1** Depth for path 1 (absolute coordinate)
- **D2** Depth for path 2 (absolute coordinate)
- **RK** Rough step-down increment (positive number)
- **FK** Finish step-down increment (positive number)
- **STEP** Rough passes step-over distance (%Tool Diameter if greater than 1)
- **RMI** Rough mill inner pocket (No, Yes)
- **LS** Lead in/out (Extend, Recede, or Custom)
- **LID / LOD** Lead in/out tangential extension distance
- **LL** Lead line distance
- **LA** Lead arc radius
- **MR** Mill forward and back (for open paths)
- **KTD** Keep tool down (for open paths)
- **AXY** Final allowance to leave on sides
- **AZ** Final allowance to leave on bottom
- **S** Spindle speed (RPM positive or negative integer)
- **P** Plunge rate (positive number)
- **F** Feed rate (positive number)
- **CL** Coolant type selection (Off, Air, Flood)

15.7.3 Tapered Pocket Event

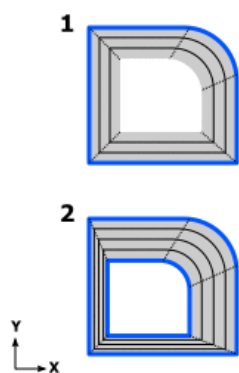
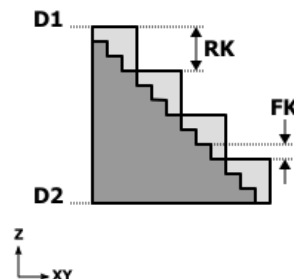
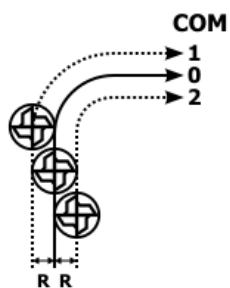
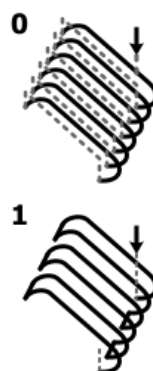
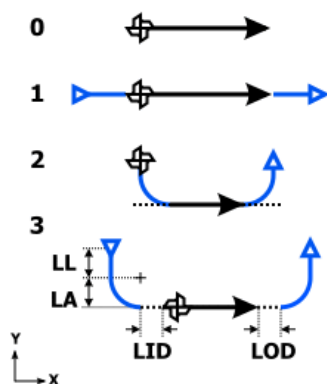
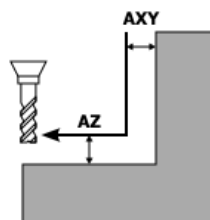
The Tapered Pocket Event removes material within a closed sloped or tapered surface. This event starts with a tool change, then activates tool length compensation, followed by a move down to the safe height where the spindle is run, and the coolant is activated. Then the tool is rapid moved down in Z to the plunge height, where it slows to plunge speed until the first step-down or incremental depth is reached, then proceeds to mill at feed rate over the specified tapered surface, stepping over and down the full surface is machined. The geometry and parameters for this event are detailed in the following lists.

Geometry:

- **Accepts one open or closed path**
- **Accepts two open paths, or two closed paths with one fully contained within, with both paths observing the same path direction**

Parameters:

- **T** Tool ID
- **RF** Roughing passes, Finishing passes, or both
- **MODE** One path or Two path mode
- **COM** Tool radius compensation (Off, Left, Right)
- **TA** Taper angle setting for when MODE = one path mode
- **RP** Plunge safe height (incremental distance above WP)
- **D1** Depth for path 1 (absolute coordinate)
- **D2** Depth for path 2 (absolute coordinate)
- **RK** Rough step-down increment (positive number)
- **FK** Finish step-down increment (positive number)
- **STEP** Rough passes step-over distance (%Tool Diameter if greater than 1)
- **RMI** Rough mill inner pocket (No, Yes)
- **LS** Lead in/out (Extend, Recede, or Custom)
- **LID / LOD** Lead in/out tangential extension distance
- **LL** Lead line distance
- **LA** Lead arc radius
- **MR** Mill forward and back (for open paths)
- **KTD** Keep tool down (for open paths)
- **AXY** Final allowance to leave on sides
- **AZ** Final allowance to leave on bottom
- **S** Spindle speed (RPM positive or negative integer)
- **P** Plunge rate (positive number)
- **F** Feed rate (positive number)
- **CL** Coolant type selection (Off, Air, Flood)

MODE One or Two Paths**RF, RK Rough, Finish and D1, D2 Depths****COM Tool Radius Compensation****MR Mill Forward and Back****LID / LOD, LL, LA Lead In / Out****AXY, AZ Allowances**

15.8 Face Mill Event

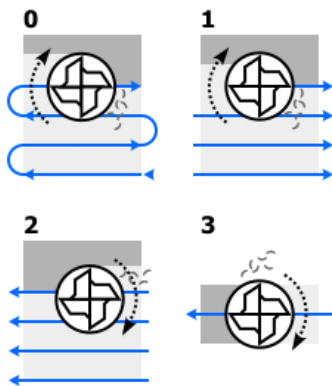
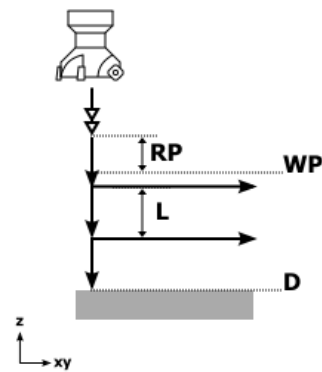
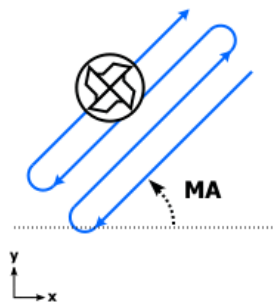
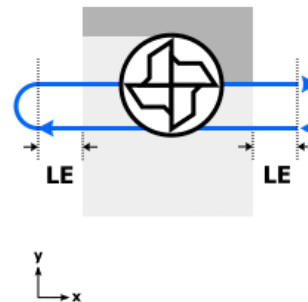
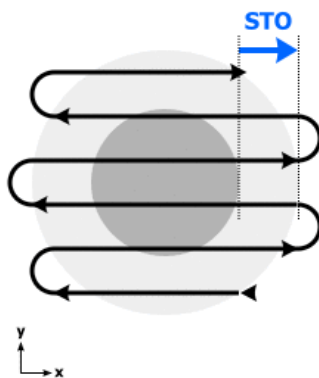
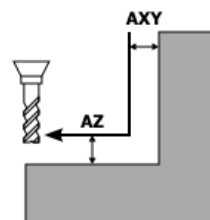
The Face Mill event performs milling on an area to produce a flat top surface. The event starts with a tool change, then activates tool length compensation, followed by a move down to the safe height where the spindle is run, and the coolant is activated. Then the tool is rapid moved down in Z to the plunge height, where it slows to plunge speed until the first step-down or incremental depth is reached, then proceeds to mill at feed rate on the specified shape, layer by layer in Z until the target depth is reached. The geometry and parameters for this event are detailed in the following lists.

Geometry:

- **Accepts closed and open irregular paths**
- **Accepts multiple paths**

Parameters:

- **T** Tool ID
- **RP** Plunge safe height (incremental distance above WP)
- **WP** Start depth (absolute coordinate)
- **D** End depth (absolute coordinate)
- **L** Rough step-down increment (positive number)
- **MD** Milling direction (Both Ways, Conventional, Climb, One Line)
- **MA** Milling angle
- **LE** Lead distance (% Tool Diameter)
- **STEP** Step-over distance (%Tool Diameter)
- **STO** Stock offset distance
- **AZ** Final allowance to leave on bottom
- **S** Spindle speed (RPM positive or negative integer)
- **P** Plunge rate (positive number)
- **F** Feed rate (positive number)
- **CL** Coolant type selection (Off, Air, Flood)

MD Milling Direction**RP, WP, D, L Face Mill Depths****MA Milling Angle****LE Lead Distance****STO Stock Offset****AZ Allowance**

15.9 Position Events

This section provides documentation on each event within the **POSITION** group of Conversational Events.

15.9.1 G00 Rapid Move

The G00 Rapid Move event commands a rapid movement to a specified location. Parameters for this event are detailed in the following list.

- **X, Y, Z** End point location (absolute coordinates)
- **PRI** Axis motion priority, sets the axis to move first (None, X, Y, Z)

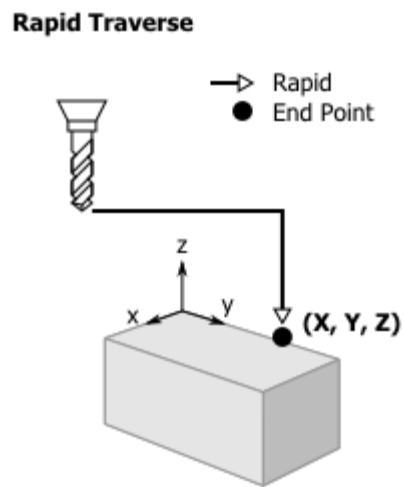


Figure 15.4.1: G00 Rapid Traverse

15.9.2 G00 Rotate A-Axis

The G00 Rotate A-Axis event commands a rapid movement to a specified location. Parameters for this event are detailed in the following list.

- **A** End degree position (absolute value)
- **CM** Clamp on or off, sets whether the rotary axis should be clamped after rotation is completed

A-Axis Rotation

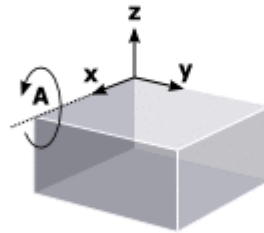


Figure 15.4.2: G00 Rotate A-Axis

15.9.3 G00 Rotate B-Axis

The G00 Rotate B-Axis event commands a rapid movement to a specified location. Parameters for this event are detailed in the following list.

- **A** End degree position (absolute value)
- **CM** Clamp on or off, sets whether the rotary axis should be clamped after rotation is completed

B-Axis Rotation

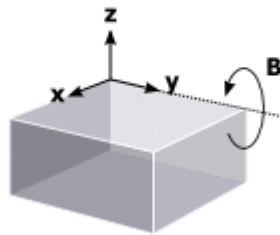


Figure 15.4.3: G00 Rotate B-Axis

15.9.4 G00 Rotate C-Axis

The G00 Rotate C-Axis event commands a rapid movement to a specified location. Parameters for this event are detailed in the following list.

- **A** End degree position (absolute value)
- **CM** Clamp on or off, sets whether the rotary axis should be clamped after rotation is completed

C-Axis Rotation

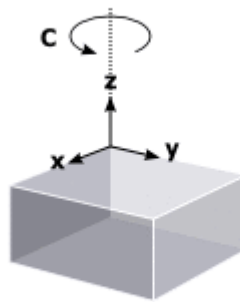


Figure 15.4.4: G00 Rotate C-Axis

15.10 Pattern Events

15.10.1 Linear Array

The Linear Array event reproduces a reference sequence of contour or pocket milling events into an array of rows and columns. Parameters for this event are summarized in the following list. Figure is an illustration of the Line Array event.

- **FR** Starting event of the referenced sequence (event label)
- **LR** Ending event of the referenced sequence (event label)
- **PR** Process reference, allows the referenced sequence to be processed in place or to be skipped (Yes, No)
- **SK** Skip count, specifies the number of locations to skip from the beginning
- **SP** Start location. If set to Event, the array starts at the location of the referenced event. If set to Custom, the array will start at the specified start position.
- **BX, BY** Linear array start location (absolute coordinates)
- **BA** Linear array angle (number between -360 and 360)
- **R** Rows (positive integer)
- **C** Columns (positive integer)
- **DX** Column offset (positive number)
- **DY** Row offset (positive number)

Line Array

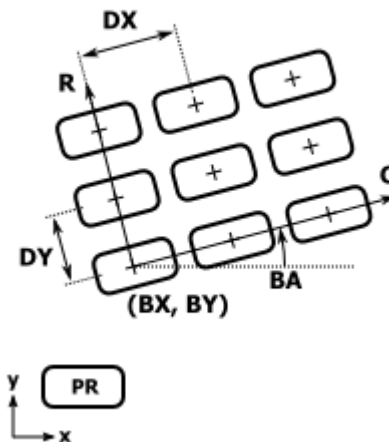


Figure 15.10.1: Linear Array Event

15.10.2 Circular Array

The Circular Array event reproduces a reference sequence of contour or pocket milling events along a circular arc. Parameters for this event are summarized in the following list. Figure is an illustration of the Circular Array event.

- **FR** Starting event of the referenced sequence (event label)
- **LR** Ending event of the referenced sequence (event label)
- **PR** Process reference, allows the referenced sequence to be processed in place or to be skipped (Yes, No)
- **SK** Skip count, specifies the number of locations to skip from the beginning
- **CX, CY** Center point location (absolute coordinates)
- **R** Radius of circle (positive number)
- **SA** Start angle with respect to +X-axis (number between -360 and 360)
- **TN** Events per revolution (positive integer)
- **N** Events to perform (positive integer)

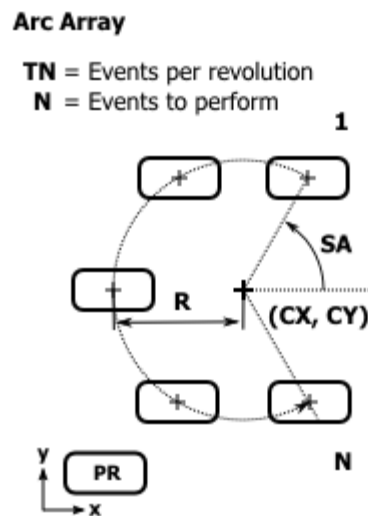


Figure 15.10.2: Circular Array Event

15.10.3 Location Array

The Location Array event reproduces a reference sequence of contour or pocket milling events at a series of arbitrary locations. Parameters for this event are summarized in the following list. Figure is an illustration of the Location Array event.

- **FR** Starting event of the referenced sequence (event label)
- **LR** Ending event of the referenced sequence (event label)
- **PR** Process reference, allows the referenced sequence to be processed in place or to be skipped (Yes, No)
- **PL** Locations (sets of absolute XY coordinates)
- **RE** Relative locations. If set to true, will perform the referenced events at each pattern location while maintaining the relative relationships between each reference events.

Location Array

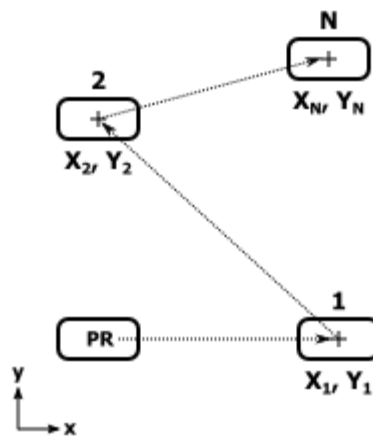


Figure 15.10.3: Location Array Event

15.10.4 Rotate

The Rotate event reproduces a reference sequence of contour or pocket milling events along a circular arc, while also rotating the geometry of the referenced events at each stop. Parameters for this event are summarized in the following list. Figure is an illustration of the Rotate event.

- **FR** Starting event of the referenced sequence (event label)
- **LR** Ending event of the referenced sequence (event label)
- **N** Number of repeats (positive integer)
- **PR** Process reference, allows the referenced sequence to be processed in place or to be skipped (Yes, No)
- **P** Plane of operation (XY, YZ, or ZX)
- **BU, BV** Rotation center location (absolute coordinates)
- **DA** Incremental angle (number between -360 and 360)

Rotate

N = Number of repeats

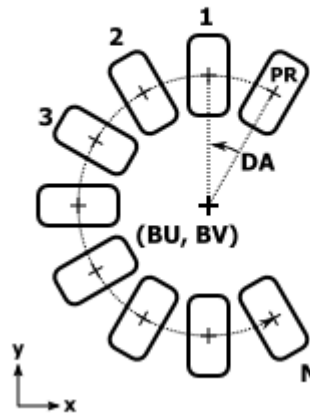


Figure 15.10.5: Rotate Event

15.11 Setup Events

15.11.1 Work Offset / P-Offset

The Work Offset event and Work P-Offset events sets subsequent events in the program to reference a pre-defined work origin. Parameters for the Work Offset event are detailed in the following list.

- **G** Work offset (G54 to G59)
- **P** Extended work offsets (P1 to P48)

Coordinate Origin

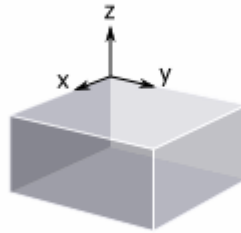


Figure 15.11.1: Work Offset/P-Offset Event

15.11.2 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 of the program and automatically repeat the program in a loop.

15.11.3 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)

Dwell

→ Feed (**F**)
⌚ Dwell (**P**)



Figure 15.11.3: Dwell Event

15.11.4 Inch/Metric Units

The Inch/Metric event commands a switch in the interpretation of numbers as either inch or millimeter units. The parameters for the Inch/Metric event are detailed in the following list.

- **G** Units (G20 inch, G21 mm)

English/Metric Units

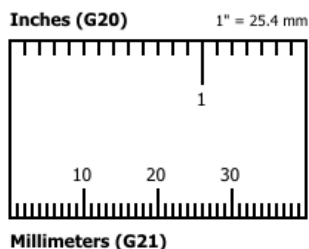


Figure 15.11.4: Inch/Metric Event

15.11.5 Coolant Control

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 (M7 Mist, M8 Flood, or M9 Off)

15.11.6 Safe Height Override

The Safe Height Override event is a modal type event that overrides the safe height set in the program settings screen for all subsequent events. The safe height is the height in which the tool moves when traveling between events. This function is useful for escaping obstacles such as clamps. Parameters associated with this event are detailed in the following list.

- **O** Override height (positive value relative to part zero)

15.11.7 Spindle Control

The Spindle Control event commands a spindle start with a specified direction and RPM, or spindle stop. The parameters for the Spindle Control event are detailed in the following list.

- **M** Spindle direction: M3 (CW), M4 (CCW), or M5 (Stop)
- **S** Spindle speed, RPM (positive integer)

15.11.8 Repeat Work

The Repeat Work event runs the current program again at various specified work offset locations. The parameters for the Spindle Control event are detailed in the following list.

- **WO** Number and table of planned work offsets or part zero locations to repeat the program at.

15.11.9 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.)

15.11.10 Comment

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

15.11.11 Engraving

The Engraving event mills stick letter text based on the specified text string, size, orientation, and depths. The parameters for the Engraving event are detailed in the following list.

- **S1** Text string to engrave
- **TYP** Arrangement of the text (linear or circular)
- **X, Y** Text start location (absolute coordinates)
- **I, J** Center location if TYP is circular (absolute coordinates)
- **RP** Plunge height (absolute coordinate)
- **Z** End depth (absolute coordinate)
- **H** Height of the font text
- **L** Length of the entire text string
- **SH** Spacing between each letter
- **V** Angle for italics style
- **Q** Angle orientation of a linear text string

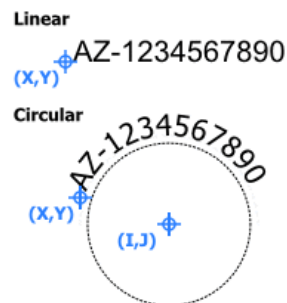
Text Engrave



Text Dimensions



Text Position



Text Angles

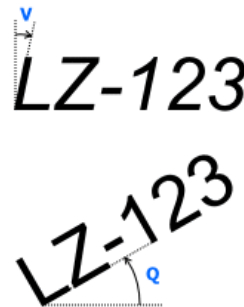


Figure 15.11.11: Engraving Event

16 CONVERSATIONAL CAD DRAWING EDITOR

16.1 Introduction

In the Conversational Editor, a CAD Drawing module allows creating a new or loading an existing DXF file. Once a drawing is open, functions are provided to perform scaling, rotation, setting zero, editing, measurement, and selection of geometry to import. Selected geometry can then be added into specific conversational events depending on the type of geometry required. This chapter provides documentation for using the CAD Drawing module.

16.2 CAD Drawing Overview and Workflow

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

8. On the Conversational Editor main screen, within the center drawing window is the CAD Drawing module. On the upper left of this window is the drawing menu which contains the following drawing functions:
 - a. **FILE** mode allows opening, saving, and creating a new drawing.
 - b. **SET ZERO** mode allows set up of a part zero location.
 - c. **SELECT** mode allows the selection of geometry to import into events.
 - 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.
9. To open or create a new drawing, select **FILE / OPEN** to load an existing DXF file, or **FILE / NEW** to create a new DXF file.
 - a. To open an existing file, the DXF file can be located on the USB drive (HDD2) or in the local program folder if previously copied into the control using the **SEND / RECEIVE** function.
 - b. To create a new drawing, type in a file name and press enter.
10. If a drawing is loaded from a DXF file, use the **VIEW** mode functions to **SCALE**, **ROTATE**, or **MIRROR** the drawing as needed. To inspect the current scale of the drawing, use the **MEASURE** mode to measure lengths, diameters, or radii to determine the current dimensions of the drawing.
11. To draw entities, use the **DRAW** mode to add entities into the drawing, or to edit existing entities, use the **EDIT** mode to modify selected entities.
12. Before geometry is selected, a part zero location will need to be set corresponding to absolute coordinate origin of the workpiece as set on the **WORK OFFSET** screen. This is critical to ensure that programmed events are machined at the correct location within the working envelope of the machine.
13. When events are added into the program, its geometry can be selected from the CAD drawing using functions in the **SELECT** mode.

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

Before importing a DXF file, it is recommended that the drawing be pre-processed on the PC or laptop for the best results. General guidelines for pre-processing are as follows:

1. Reduce the file size of the DXF file as much as possible. Keep only the geometry required for the machining of the part. Clean up non-geometric objects, such as tables or comments.
2. If certain geometric entities such as Block References or text descriptions do not show on the DXF editor, simplify these entities into line or arc segments by using the EXPLODE function.

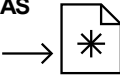
16.3 CAD Drawing Main Screen

(DRAWING IMPORT)													
(DRAWING MODES) <table border="1"> <tr> <td> FILE  </td> <td> MEASURE  </td> </tr> <tr> <td> SETZERO  </td> <td> DRAW  </td> </tr> <tr> <td> SELECT  </td> <td> EDIT  </td> </tr> <tr> <td> VIEW  </td> <td></td> </tr> </table> (DRAWING) ZOOM IN  ZOOM OUT  ZOOM ALL  GRID  UNDO  RESET 						FILE 	MEASURE 	SETZERO 	DRAW 	SELECT 	EDIT 	VIEW 	
FILE 	MEASURE 												
SETZERO 	DRAW 												
SELECT 	EDIT 												
VIEW 													
OPEN DRAWING	IMPORT HOLES	IMPORT CONTOUR	IMPORT POCKET	IMPORT OTHER	BACK								

Figure 16.3-1: Drawing Import Main Screen

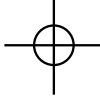
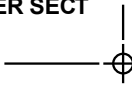
16.4 File Mode and Functions

Table 16.4.1: File Functions

Function	Description
FILE 	The FILE mode provides file management related functions.
NEW 	NEW FILE function creates a new drawing with the specified filename.
RELO AD 	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.

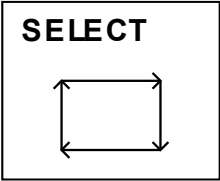
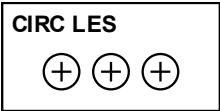
16.5 Set Zero Mode and Functions

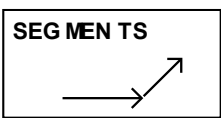
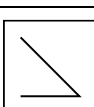
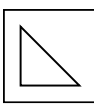
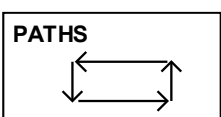
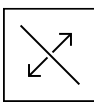
Table 16.5.1 : Set Part Zero Functions

Function	Description
SET ZERO 	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.
PONT 	SET ZERO TO POINT function allows selecting any segment start, end, mid, quadrant, or key-point to become the new part zero location.
INTER SECT 	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.
MEDIAN 	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

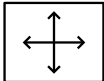
Table 16.6.1: Drawing Select Functions

Function	Description
	In the SELECT mode, geometry from the drawing can be picked out and selected for import as events.
	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.
	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.
	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.

	SELECT ARC toggle allows the selection of arcs instead of circles within the drawing.
	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.
	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.

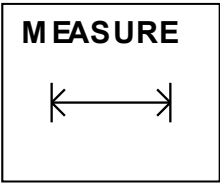
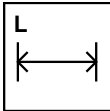
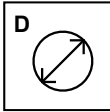
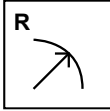
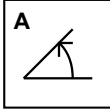
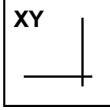
16.7 View Mode and Functions

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.
SCALE 	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.
MIR X 	MIRROR X function applies a mirror action about the X-axis running through the part zero location.
MIR Y 	MIRROR Y function applies a mirror action about the Y-axis running through the part zero location.

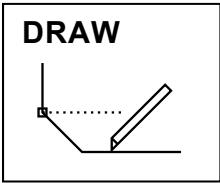
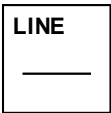
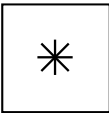
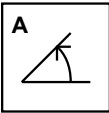
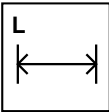
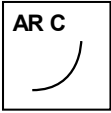
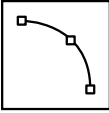
16.8 Measure Mode and Functions

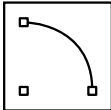
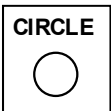
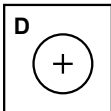
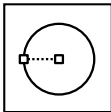
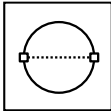
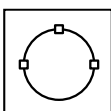
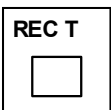
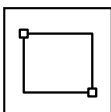
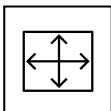
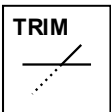
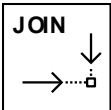
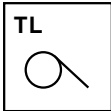
Table 16.8.1: Drawing Measurement Functions

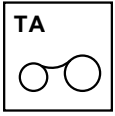
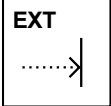
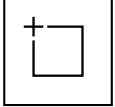
Function	Description
	MEASURE mode offers functions to label dimensions on the drawing for reference or verification.
	MEASURE LENGTH function allows the selection of two points, a point and a line, or two parallel lines to display the distance between the selected features.
	MEASURE DIAMETER function allows the selection of any circle or arc to display its diameter.
	MEASURE RADIUS function allows the selection of any circle or arc to display its radius.
	MEASURE ANGLE function allows the selection of three points or two lines to display the angle between the features.
	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

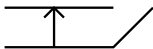
Function	Description
	The DRAW MODE allows creating geometry within the drawing. Use this mode to add supplemental features on a drawing or to create a part drawing from scratch.
	DRAW LINE mode allows creating lines within the drawing. Once activated, click on a start point and an end point to draw 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 arcs 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.

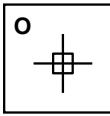
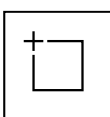
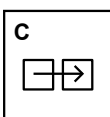
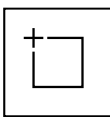
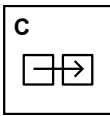
	DRAW ARC START + CENTER + END function creates an arc by selecting or inputting first the start point, then the center point, and lastly the end point of the arc segment. The arc will be drawn in a counter-clockwise direction.
	DRAW CIRCLE mode allows drawing circles in the drawing.
	DRAW CIRCLE CENTER + 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.
	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 function performs partial deletion of the selected segment if any other line, arc, or circular segment intersects with the selected segment.
	JOIN mode and function allows the joining of two selected segments if any existing or potential intersection exists.
	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
EDIT 	The DRAWING EDIT mode contains editing functions applicable to existing geometry within the drawing.
FILLET 	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.
CHAMF 	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 	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 	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 	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 	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.

	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.
	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.
	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.

The screenshot displays the 'SYSTEM' main screen with a top status bar and a bottom navigation bar. The status bar includes 'DynaPath', 'JOG', 'SYSTEM', 'mm', and the timestamp '2020-1-29 18:40:01'. The main area is divided into four quadrants: 'ABSOLUTE', 'RELATIVE', 'MACHINE', and 'DISTANCE'. Each quadrant shows X, Y, and Z coordinates with a '4th' decimal indicator. To the right of these quadrants is a vertical stack of five empty rectangular buttons. Below the quadrants is a 'RESET' button. At the bottom, a 'MESSAGE:' label is followed by a text input field, and a 'STATUS: IDLE' indicator. The bottom navigation bar contains six buttons: 'PARA-METER', 'TUNING', 'MAIN-TENANCE', 'MESSAGE', 'ENVIRON-MENT', and 'BACK'.

ABSOLUTE		RELATIVE	
X	40.000	X	116.200
Y	-40.000	Y	36.200
Z	2.026	Z	-71.429
4th	0.000	4th	0.000

MACHINE		DISTANCE	
X	116.200	X	0.000
Y	36.200	Y	0.000
Z	-71.429	Z	0.000
4th	0.000	4th	0.000

Please Select A System Function To Continue.

MESSAGE: STATUS: IDLE

PARA-METER TUNING MAIN-TENANCE MESSAGE ENVIRON-MENT BACK

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 to commit the changes into memory. The parameter groups are described in more detail in the following list:

- **USER** parameters can be set by the user.
(Log in with "SI", password "SI".)
- **MACHINE** parameters are set by the system integrator or machine builder.
(Log in with "SI", password "SI".)
- **CONTROL** parameters are set by the control provider.
(Log in with "ROOT", password "WINPC32".)
- **SYS CONFIG** parameters are set by the solution provider.
(Log in with "ROOT", password "WINPC32".)

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

DynaPath **JOG** **USER** mm 2020-1-29 18:49:28

Parameter 1	Parameter 2	Parameter 3	Parameter 4	SET	Default	
1	OT Position in Positive X Direction			9999.000	9999.999	mm
2	OT Position in Positive Y Direction			9999.000	9999.999	mm
3	OT Position in Positive Z Direction			9999.000	9999.999	mm
4	OT Position in Positive 4th Direction			9999.999	9999.999	mm
5	OT Position in Negative X Direction			-9998.990	-9999.999	mm
6	OT Position in Negative Y Direction			-9998.990	-9999.999	mm
7	OT Position in Negative Z Direction			-9998.990	-9999.999	mm
8	OT Position in Negative 4th Direction			-9999.999	-9999.999	mm
9	In-Position Window of X Axis			0.030	0.030	mm
10	In-Position Window of Y Axis			0.030	0.030	mm
11	In-Position Window of Z Axis			0.030	0.030	mm
12	In-Position Window of 4th Axis			0.030	0.030	mm
13	G01/G02/G03 Acc/Del Time of X,Y,Z,4th			180	150	ms
14	Corner Accuracy			2	5	
15	G31 Offset for signal scan			000000.046	999999	mm
16	G31 Offset to Stop			0.100	0.1	mm

Set user parameters

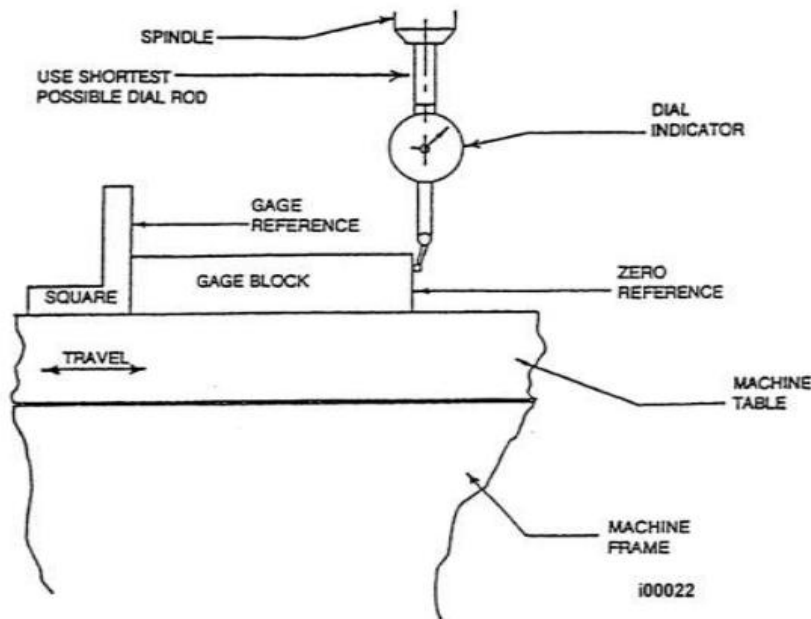
MESSAGE: STATUS: **IDLE**

USER MACHINE CONTROL MOTION AUXILIARY BACK

Figure 17.2-1: System Parameters Screen

17.3 Backlash Compensation

Every axis drive assembly has at least some backlash or lost motion during direction reversal, thus backlash compensation is also referred to as reversal compensation. The lost motion is due to gaps in the parts of the drive train and the accumulation of bending or elasticity of these parts while under load. The control system can compensate for this lost motion or backlash by applying an additional distance movement to each axis upon axis reversal. The backlash compensation values are initially set by the factory and may need to be adjusted periodically as the machine wears over time.



Follow these instructions to set the machine backlash compensation:

1. Turn off all backlash compensation values on the control and reboot. The backlash parameters are 2156A to 2158A.
2. Set a dial indicator capable of finer than 0.002 mm resolution on the spindle or head assembly and use the MPG to travel in one direction to load the indicator against a clean, rigid surface such as the edge of a vise without reversing. Once loaded, zero out the RELATIVE coordinate display on the control.
3. Use the MPG at x1 increments to reverse the axis until the dial indicator shows the first sign of movement. Upon any indication of movement of the dial, stop moving.
4. The distance travelled on the screen in RELATIVE coordinates is the backlash value for that axis.
5. Enter this value into the control (parameters 2156A to 2158A), then reboot and home the machine to verify if the backlash compensation is correct.

17.4 Lead Screw Compensation

***Note:** The WinDelta CNC System uses a bi-directional lead screw compensation scheme which accounts for backlash at each location, therefore any backlash previously set will no longer take effect when lead screw compensation is active.

The lead screw compensation function is performed dynamically by the control system to offset position errors due to any deviation between the maintained position in the control against the actual physical position as measured along each axis by a highly accurate measurement device such as a laser interferometer.

The lead screw compensation table is stored in the control system as a text file named "ptch_tbl.dat". This file can be exported and imported by booting up the control with the WinDelta USB Utility and executing the EXPORT MOT PARAMS and IMPORT MOT PARAMS functions. This file contains information on the number of axes to compensate and the compensation data for each axis. Note that this control system uses a bi-directional compensation scheme, thus two sets of offset data exist for each axis, one in the forward direction and one in the reverse direction. When the lead screw compensation is active, then the backlash settings for each axis are consequently ignored. All units are in microns.

The format for the lead screw compensation text file is described as follows:

PTCH_TBL.DAT File Format:

```
<Total number of axes to compensate>
<Axis number, starts at 0> <Number of compensation points> <Scaling>
Positions:
<Forward measured> <Commanded> <Reverse measured> <Commanded>
```

PTCH_TBL.DAT Example:

```
3
0          5          1.0
0          0          7          0
25004      25000      25011      25000
50008      50000      50015      50000
75012      75000      75019      75000
100016     100000     100022     100000
1          5          1.0
-100024    -100000    -100006    -100000
-75023     -75000     -75005     -75000
-50023     -50000     -50003     -50000
-25025     -25000     -25002     -25000
-27        0         0         0
2          5          1.0
-100006    -100000    -99999     -100000
-75005     -75000     -74998     -75000
-50006     -50000     -49998     -50000
-25008     -25000     -24999     -25000
-10        0         0         0
```

17.5 Maintenance

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

17.6 Message

The MESSAGE screen lists current and historical alarm messages.

17.7 Environment

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

The screenshot displays the 'System Environment Screen' with the following components:

- Top Bar:** DynaPath logo, JOG button, SYS FORMAT button, mm unit indicator, and timestamp 2020-1-29 18:40:17.
- LANGUAGE Section:** Buttons for ENGLISH (highlighted), 简体中文, and 繁體中文.
- TIME Section:** CURRENT TIME 2020/1/29 PM 06:40:17, and a MODIFY section with fields for YEAR (2020), MONTH (1), DAY (29), HOUR (18), and MINUTE (40), followed by a CONFIRM button.
- SYSTEM Section:** KERNEL VERSION 219010891 and COMPILER VERSION 218112391.
- SOFTWARE Section:** Mill_CNC4 Build 2019/12/26.
- IP ADDRESS Section:** STATIC mode, IP address 192.168.126.50, and a SET button.
- PASSWORD Section:** EDIT USER, CLEAR RECORD, and LOGOUT buttons.
- User/Password Log Table:**

No	User/Password Log
1...	[18:39:57 01-29,2020] User Log
1...	[18:40:12 01-29,2020] User: Sit
- MESSAGE:** A text field for displaying messages.
- STATUS:** IDLE.
- Bottom Bar:** ENVIRON-MENT (highlighted), HOTKEY, PASSWORD, and BACK buttons.

Figure 17.7-1: System Environment Screen

18 NETWORKING

18.1 Introduction to Networking

The DynaPath WinDelta CNC can connect to a local network or the internet using its built-in LAN port. Connecting to a network enables functionality such as 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 Setup

For Windows 7 based WinDelta CNC Controls (e.g. Node-1000):

To set up the Network:

1. Enter the Windows Control Panel or the System Shell by booting up with the WinDelta USB Utility.
2. In the Windows Control Panel click on "Network Connections", or from the System Shell click on "Network". Once open, double click the "Local Area Connection" entry from the list.
3. Open the "Properties" panel and scroll down the list to "Internet Protocol TCP/IP". Click "Properties" to access its settings.
4. A window should pop up. For automatic IP assignment, check "Obtain an IP address automatically" and press "Ok". For manual IP configuration, check "Use the following IP address" and input an IP address, subnet mask, and gateway addresses, then press "Ok".
5. Press "Ok" and "Close" to exit the network setup.
6. Press "Save and restart" from the System Shell or execute D:\Save & Restart XPe.bat to commit the change and restart the control.

To verify the control is connected to the network:

1. Enter the Windows Control Panel or the System Shell by booting up with the WinDelta USB Utility.
2. In the Windows Control Panel click on "Network Connections", or from the System Shell click on "Network". Once open, double click the "Local Area Connection" entry from the list.
3. Open the "Support" tab above and verify that the IP address, Subnet Mask, and Default Gateway are correct.

For Windows CE based WinDelta CNC Controls (e.g. Node-355):***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.

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 09*** and are macro programs used for tool change, tool setting, or probing.***

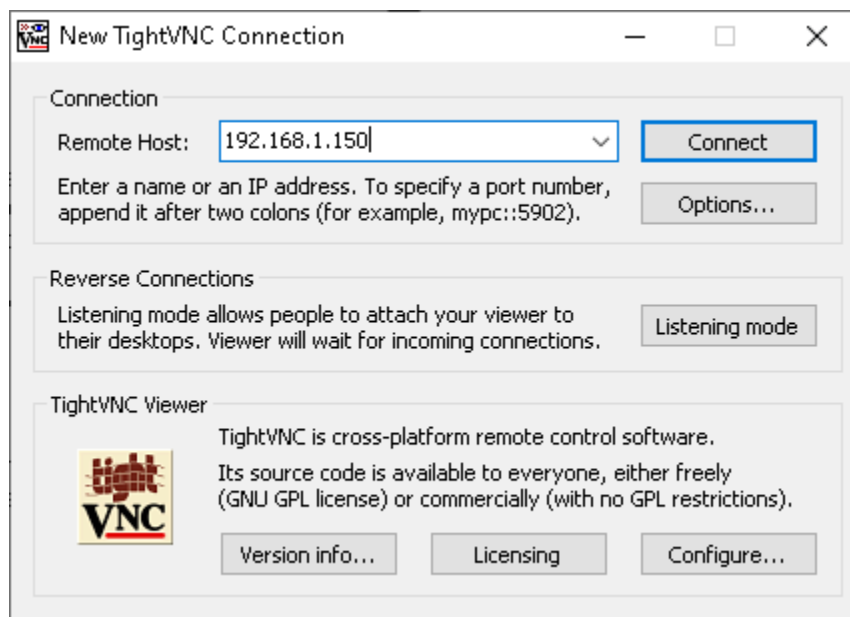
18.4 Connecting to a Windows Network Folder

****Note: This function is not available for Windows CE based WinDelta controls.***

The WinDelta CNC can connect to a Windows Network file server to access part programs and DXF files. To access a Windows Network shared folder, go to the Send and Receive page on the control. Click on the External Storage Setup icon and configure the name, user, password, and network path for the shared folder. Then click "Connect" to save and initiate the connection. Once connected, the external file folder to the right of the screen will show files and folders from the Windows Network file server.

18.5 Local Monitoring with VNC

The DynaPath WinDelta CNC has built-in remote monitoring and control functions via a Virtual Network Computing (VNC) server implementation. VNC is an open-sourced software that taps into the remote frame buffer protocol to output a server's frame buffer over a TCP/IP enabled network. The VNC server runs on DynaPath controls on boot-up as an option. A VNC Viewer application is required on any client device to connect to the VNC Server for remote monitoring and control. This functionality is limited to local area networking. To connect, enter the IP address of the control into the VNC Viewer and press connect.



18.5.1 VNC Server Set Up

By default, the VNC server is launched on boot-up, thus a WinDelta control should be ready to accept remote connections once connected to the network. However, if the VNC server is not run on boot-up, then a shortcut for the VNC server application must be placed into the Windows Startup folder. Refer to the following procedure:

1. Enter the System Shell using the WinDelta USB Utility.
2. Open Windows Explorer.
3. Navigate to C:\Program Files\TightVNC.
4. Create a shortcut for tvnserver.exe.
5. Cut the shortcut and paste it to C:\Documents and Settings\All Users\StartMenu\Programs\Startup.
6. Run the shortcut by double clicking it and see if a VNC icon is present in the System Tray. If yes, make sure it does not have a red outline. If it does, right click on the icon and open Configuration.
7. A password may be assigned for any remote viewer who wishes to connect or uncheck the option "Require VNC authentication" for access without a password.
8. Run "D:\Save and Restart.bat" to commit changes and reboot.

18.6 Remote Assistance

DynaPath or authorized machine tool partners can perform online assistance and support to perform remote diagnostics, troubleshooting, general support, software updates, or additional training. Contact the proper personnel to request a remote session. The remote session is set up using the following procedures.

For Windows 7 based WinDelta CNC Controls (e.g. Node-1000):

If the control is connected to the internet, initiate remote assistance using the following procedure:

1. Reboot or power on the control, enter the System Shell by booting up with the WinDelta USB Utility.
2. Press the "REMOTE" button on the USB Utility screen.
3. Provide the ID and Password on the screen to the support personnel.
4. If the request for remote help is successful, support staff will be able to log in and view your CNC screen.

For Windows CE based WinDelta CNC Controls (e.g. Node-355):

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.

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

Code	Motion Alarm
0000	Normal.
0001	X-axis following error overflow!!
0002	Y-axis tracking error overflow!!
0003	Z-axis following error overflow!!
0004	C-axis following error overflow!!
0005	Software limit check found an error!!
0006	X-axis servo hysteresis overflow!!
0007	Y-axis servo hysteresis overflow!!
0008	Z-axis servo hysteresis overflow!!
0009	C-axis servo hysteresis overflow!!
0010	Hardware emergency stop!!
0011	Software emergency stop!!
0012	Workpiece instruction lacks terminal annotation!!
0013	Failed to obtain the motor index or Z-Phase signal normally!!
0014	The X-axis position falls within the origin mark (DOG) range!!
0015	The Y-axis position falls within the origin mark (DOG) range!!
0016	The Z axis position falls within the origin mark (DOG) range!!
0017	The C axis position falls within the origin mark (DOG) range!!
0018	Failed to return to the origin position normally!!
0019	Overtravel (OT) error occurred!!
0020	Abnormal program ended, no follow-up data!!
0021	Unable to confirm the stop status normally!!
0022	The software limit range (Limit1) is set improperly!!
0023	The software limit range (Limit2) is set improperly!!
0024	Software check error!!
0025	The origin return program cannot stop in the DOG range!!
0026	Rigid tapping planning error!!
0027	Rigid tapping parameter error!!
0028	The spindle speed at the start of rigid tapping is not zero!!
0029	E-CAM compensation error!!
0030	G01 acceleration/deceleration parameter setting error!!
0031	The G00 acceleration/deceleration parameter setting of the X axis is wrong!!
0032	The G00 acceleration/deceleration parameter setting of the Y axis is wrong!!
0033	The G00 acceleration/deceleration parameter setting of the Z axis is wrong!!
0034	The G00 acceleration/deceleration parameter setting of the C axis is wrong!!

0035	Origin return mode setting error!!
0036	Position check window range parameter setting error!!
0037	Feedback frequency multiplication parameter setting error!!
0038	PPR parameter setting error!!
0039	The second speed setting of origin return is wrong!!
0040	X-axis overtravel (OT) in positive direction!!
0041	X-axis negative direction overtravel (OT) occurs!!
0042	Overtravel (OT) occurred in the positive direction of the Y axis!!
0043	Overtravel (OT) occurred in the negative direction of the Y axis!!
0044	Z axis overtravel (OT) in positive direction!!
0045	Z axis negative direction overtravel (OT)!!
0046	C axis overtravel (OT) in positive direction!!
0047	Overtravel (OT) occurred in the negative direction of C axis!!
0048	Arc command error!!
0049	Rigid tapping spindle zero speed!!
0050	X-axis positive direction exceeds software limit!!
0051	X-axis negative direction exceeds software limit!!
0052	Y-axis positive direction exceeds software limit!!
0053	The negative direction of the Y axis exceeds the software limit!!
0054	Z-axis positive direction exceeds software limit!!
0055	Z-axis negative direction exceeds software limit!!
0056	C-axis positive direction exceeds software limit!!
0057	C-axis negative direction exceeds software limit!!
0058	Post-acceleration/deceleration buffer overflow!!
0059	Axial movement is about to exceed the boundary limit!!
0060	Unknown error!!
0061	Unknown error!!
0062	Unknown error!!
0063	Unknown error!!
0064	Unknown error!!
0065	Internal software error!!
0066	Internal software error, the motion command and motion direction conflict!!
0067	Internal error in the software, the previous command was not completed!!
0068	M-Code error occurred!!
0069	Internal software error, ACM execution process error!!
0070	Unknown error!!
0071	Unknown error!!
0072	Unknown error!!
0073	Unknown error!!
0074	Unknown error!!
0075	Unknown error!!
0076	Unknown error!!

0077	Unknown error!!
0078	Unknown error!!
0079	Cannot restart or pause the program while executing any single block!!
0080	Compare-Out error occurred!!
0081	Compare-Out range error!!
0082	Illegal R point designation!!
0083	Manual mode motion command error!!
0084	Manual arc command center or radius error!!
0085	The radius specified in the manual full circle command is wrong!!
0086	Manual mode motion command planning failed!!
0087	Mode switching is prohibited during program restart!!
0088	The speed of X, Y, Z or C axis is far beyond the upper limit!!
0089	X,Y,Z or C axis encoder error!!
0090	No follow-up instructions!!
0091	S-PCL axis positive feedback error!!
0092	S-PCL axis hysteresis is too large error!!
0093	Board WatchDog Timeout Error!!
0094	Processing program start error!!
0095	The processing program execution length exceeds the limit!!
0096	Processing program compilation error!!
0097	Organization conversion calculation error!!
0098	PID module hardware system error!!
0099	PLC module hardware system error!!
0100	Motion module hardware system error!!
0101	G70 loop within G70 loop!!
0102	Subroutine call is too deep (more than 8 levels)!!
0103	G73 execution error, no corresponding address returned!!
0104	Mathematical operation error, the denominator is 0!!
0105	Matrix or vector operation error!!
0106	This G-Code function is not supported!!
0107	Fixed cycle error!!
0108	Unable to access the board normally. Error!!
0109	The board reads the axis position incorrectly!!
0110	Position maintenance error!!
0111	The I/O interrupt displacement setting is unreasonable!!
0112	Position command hysteresis is too large error!!
0113	The error between the absolute coordinates and the programmed position is too large!!
0114	Command planning speed is too high error!!
0115	Communication axis Servo On status error!!
0116	Playback data file opening error!!
0117	Playback data file format is incorrect or not accepted!!
0118	MDR data file opening error!!

0119	Playback scaling error!!
0119	Compile command abnormality!! (The moving distance or position of each axis is not a number or exceeds the normal range)
0120	Communication axis communication error!!
0121	Servo axis abnormal Servo Off error!!
0125	Communication axis communication timeout error!!
0150	Synchronous axis synchronization error is too large!!
0499	Does not match the compiled module!!
0500	Error opening MOT program file during execution!!
0501	MOT program file opened error when re-executed!!
1000	Error reading Windows Registry key value!!
1001	Motion module startup error, the reason may be that the parameters or control card settings are incorrect!!
1002	Motion module group object activation error!!
1003	System variable file opening error!!
1004	Configuration settings are not completed during execution, you may need to export the configuration settings!!
1005	Failed to start or stop the data acquisition timer during execution!!
1006	WINPC32 module configuration error!!
1007	EtherCAT module configuration error!!

