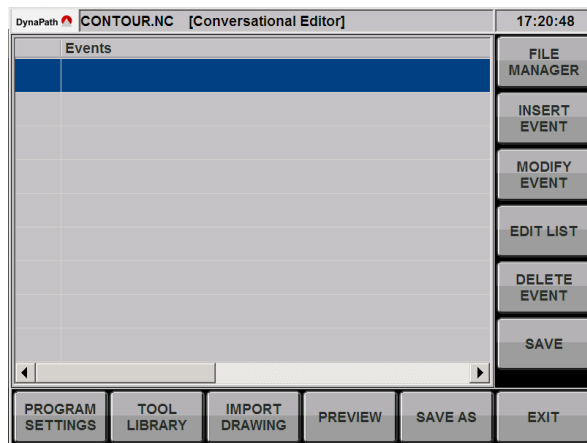


Programming Basics

DynaPath WinDelta CNC Lathe Control · Operator Quick Guide

Build a program from events in the Conversational editor (it writes the G-code for you on SAVE), or write standard ISO/EIA turning G-code. [13, 15]

Conversational program step by step Manual 15.2–15.4



Conversational editor: event list, event buttons right, program buttons along the bottom [Fig. 15.3-1]

- 1 **FILE MANAGER** → **NEW** conversational program (.HNC), or open an existing one.
- 2 **PROGRAM SETTINGS - Machine:** units (G20/G21), work offset, workpiece material, tool change position.
- 3 **PROGRAM SETTINGS - Blank:** stock type BL (bar or pipe), outer / inner diameter XA / XI, Z start ZA, end of part ZI, chuck face ZB.
- 4 **PROGRAM SETTINGS - Retract:** stock allowance / clearance SC, retract amounts XRA and ZRA.
- 5 **TOOL LIBRARY:** set up the tools the job uses.
- 6 **INSERT EVENT:** stock removal, holes, thread, groove, cut-off, line-arc, position or auxiliary. Fill in its parameters. Use **IMPORT DRAWING** to draw a profile or pick geometry from a DXF.
- 7 Use **MODIFY EVENT** and **EDIT LIST** to change or reorder events; **PART** switches between front and back end of the part.
- 8 **PREVIEW** the toolpaths, then **SAVE** (this generates the G-code).
- 9 **EXIT** and run it in AUTO (Guide 5).

Most-used G-codes Manual 13.1

Code	Function	Code	Function
G00 / G01	Rapid / linear feed	G70	Finishing cycle
G02 / G03	Arc CW / CCW	G71 / G72	Stock removal, turning / facing
G04	Dwell	G73	Pattern repeating cycle
G20 / G21	Inch / mm	G74 / G75	Face peck drill / OD-ID cycle
G28	Return to reference	G76 / G92 / G32	Multi-pass thread / thread cycle / threading
G40 / G41 / G42	Nose radius comp cancel / left / right	G90 / G94	OD-ID cutting cycle / face turning cycle
G50	Coordinate set / max spindle speed clamp	G83 / G84 / G85	Face drill / tap / bore
G54–G59	Work offsets	G87 / G88 / G89	Side drill / tap / bore
G96 / G97	Constant surface speed on / off	G98 / G99	Feed per minute / per revolution

Programming Basics

DynaPath WinDelta CNC Lathe Control · Operator Quick Guide

Most-used M-codes (some depend on fitted options) and T-codes Manual 13.2, 13.3

Code	Function	Code	Function
M00 / M01	Stop / optional stop	M10 / M11	Chuck clamp / unclamp
M02 / M30	Program end	M12 / M13	Tailstock extend / retract
M03 / M04 / M05	Spindle CW / CCW / stop	M45 / M46	C-axis mode on / off
M07 / M08 / M09	Air blast / flood coolant on / both off	M19 / M29	Spindle orient / rigid tapping on
M98 / M99	Sub-program call / return	T0305	Tool 3 with offset row 5