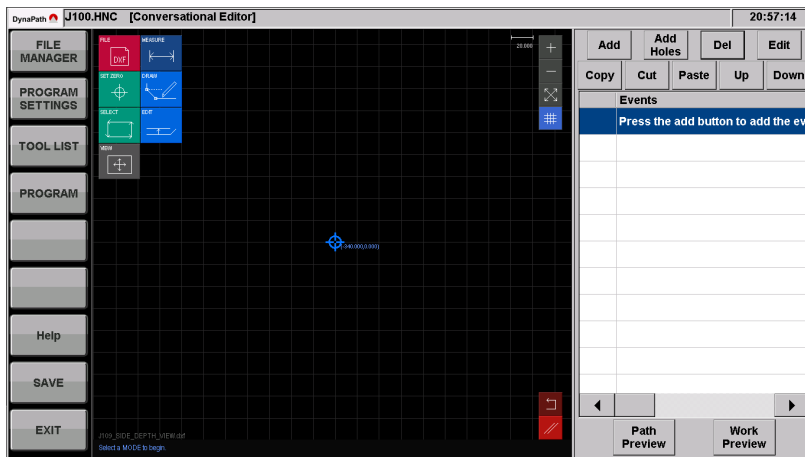


Programming Basics

DynaPath WinDelta CNC Mill 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 G-code. [13, 15]

Conversational program in nine steps Manual 15.2



Conversational editor: program functions left, events right [Fig. 15.3-1]

- 1 **FILE MANAGER** → **NEW** → choose Conversational.
- 2 **PROGRAM SETTINGS**: work offset, safe height, units, start/finish position, stock material.
- 3 **TOOL LIST**: enter or load the tools the job uses.
- 4 Press **PROGRAM** to open the editor.
- 5 Optional drawing: **FILE** → **OPEN** a DXF or **NEW**; scale/rotate/mirror in **VIEW**, edit in **DRAW/EDIT**, set part zero with **SET ZERO**.
- 6 **ADD** events (holes, contours, pockets, face mill, position, pattern, setup), fill in parameters, pick geometry with **SELECT GEOMETRY** or **EDIT GEOMETRY**, then **SAVE** the event.
- 7 **PREVIEW** the toolpaths.
- 8 **SAVE** the program (this generates the G-code).
- 9 **EXIT** and run it in **AUTO** (Guide 5).

G-code syntax Manual 13.1

N100 line number · (), // or ; comments · / at line start = optional block skip. Letters are case-insensitive; files are ANSI or UTF-8.

Most-used G-codes Manual 13.2

Code	Function	Code	Function
G00	Rapid move	G43 H_ / G49	Tool length comp on / cancel
G01 F_	Linear feed	G41 / G42 D_ / G40	Cutter comp left / right / cancel
G02 / G03	Arc CW / CCW (I, J, K centre)	G54–G59, G54.1 P1–48	Work offsets
G04 X_	Dwell (seconds)	G81 / G82 / G83 / G73	Drill / counterbore / peck / high-speed peck
G17 / G18 / G19	XY / ZX / YZ plane	G84 / G74	Right / left hand tap (F = pitch)
G20 / G21	Inch / mm	G80	Cancel canned cycle
G28	Return to reference via point	G90 / G91	Absolute / incremental
G53	Move in machine coordinates	G98 / G99	Return to initial / R point

Programming Basics

DynaPath WinDelta CNC Mill Control • Operator Quick Guide

Most-used M-codes (some depend on fitted options) Manual 13.3

Code	Function	Code	Function
M00 / M01	Stop / optional stop	M08 / M09	Flood coolant on / coolant and air off
M03 S_ / M04 S_	Spindle CW / CCW	M07	Air blast on
M05	Spindle stop	M98 / M99	Sub-program call / return
M19	Spindle orient	M29	Rigid tapping mode on
M02 / M30	Program end	M17 T_	Auto tool length measure