

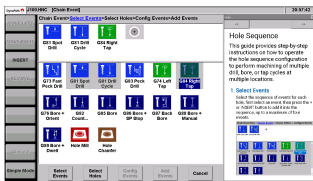
Hole Events (Drill, Bore, Tap)

DynaPath WinDelta CNC Mill · Conversational Programming Quick Guide

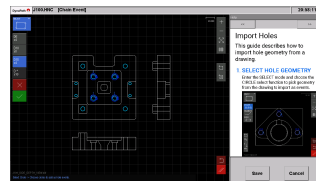
A **hole sequence** applies one or more cycles (for example spot, drill, tap) to every selected hole in one go. Holes are easiest to pick as circles on a drawing. [15.4]

Create a hole sequence Manual 15.4

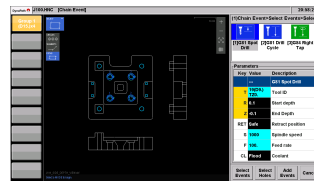
- 1 In the editor, press **ADD HOLES**.
- 2 Pick the cycles in order: double-click each one (or select it and press +). They line up along the top. Press **SELECT HOLES**.
- 3 Click individual circles on the drawing, or drag a box: the control groups the circles it finds by diameter on the left. Selected holes turn dark blue. Press **SAVE**.
- 4 Fill in each cycle's parameters: tool, start / end depth, feed and spindle speed. Press **ADD EVENTS**.
- 5 To change it later, double-click the sequence's title event (or select it and press EDIT). Each cycle can be edited on its own.



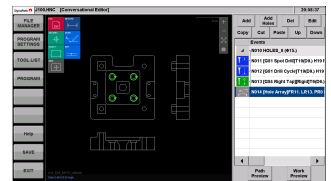
1. Choose the cycles



2. Select the holes



3. Set parameters



4. Sequence in the event list

NOTE

No drawing? Toggle **SIMPLE / ADVANCED MODE** on the cycle selection screen. The hole selection screen then shows a table where you type in X and Y positions. [15.4]

Which cycle? Manual 15.4.1–15.5.2

Cycle	What it does	Extra parameters
G81 Drill	Feed to depth, rapid out	—
G82 Counterbore	Feed to depth, dwell, out. Gives a clean hole bottom	P dwell
G83 Peck drill	Pecks, retracting fully to R each time. Best for small or deep holes	Q peck
G73 High-speed peck	Pecks with a short retract. Faster than G83	Q peck, D retract
G85 Bore	Feed to depth, feed back up to R	—
G89 Bore + dwell	As G85 with a dwell at the bottom	P dwell
G76 Fine bore	Orients spindle and shifts off the wall before retracting	Q shift
G86 Bore, spindle stop	Spindle stops oriented at depth, then retracts	—
G87 Back bore	Bores the back side of a hole, feeding upwards	Q shift, P dwell
G88 Bore, manual retract	Dwells at depth then waits for manual retract	P dwell
G84 / G74 Tap	Right / left hand tap, rigid (M29) or normal	TT mode, F pitch (rigid), S, Q, P

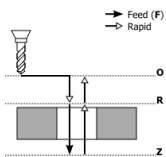
Hole Events (Drill, Bore, Tap)

DynaPath WinDelta CNC Mill · Conversational Programming Quick Guide

Parameters every hole cycle uses

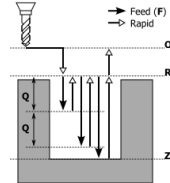
Code	Meaning	Code	Meaning
X, Y	Hole position (absolute)	Z	Final depth (absolute)
R	Transition height: rapid to here, then feed	F	Feed rate (pitch for rigid tapping)

G81 - Drill Cycle



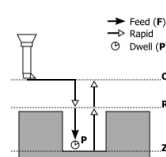
G81 drill

G83 - Peck Drill



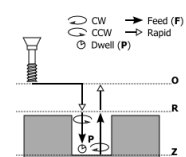
G83 peck drill

G82 - Counter Bore Cycle



G82 counterbore

G74 - Left Handed Tapping Cycle



G74 tap

NOTE

Rigid tapping: set TT to Rigid and F to the **thread pitch**. For normal tapping F is the actual feed (mm/min or in/min) matched to the spindle speed. A Q larger than R - Z means no pecking. [15.5]