

Setting Tool Lengths

DynaPath WinDelta CNC Mill Control · Operator Quick Guide

Measure and store a length (H) for every tool before running a program, then call it with **G43 H_**. Tool diameters (D) are used by cutter compensation **G41/G42 D_**. [4.11, 9.1]

The tool offset table Manual 9.1



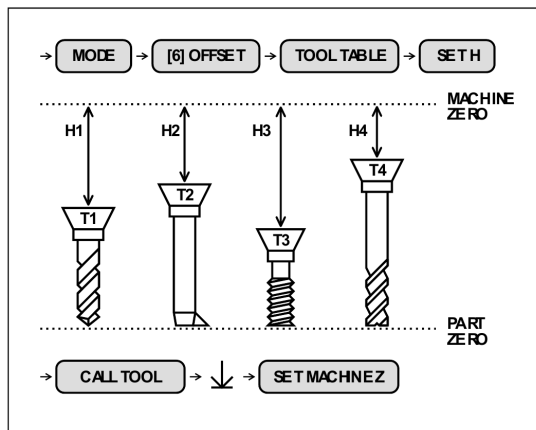
TOOL TABLE: H, H WEAR, D and D WEAR columns [Fig. 9.1-1]

- **MODE** → **6 OFFSET** → **TOOL TABLE** (or **MODE** → **5 SET UP**).
- **H** = tool length, **D** = tool diameter. **H WEAR** and **D WEAR** are added on top, for small corrections as tools wear.
- **CALL TOOL**: highlight a tool number and press **CALL TOOL** to change to it. Status shows **RUN/MEM** during the change, then **IDLE**.

Pick a method Manual 9.2, 9.3

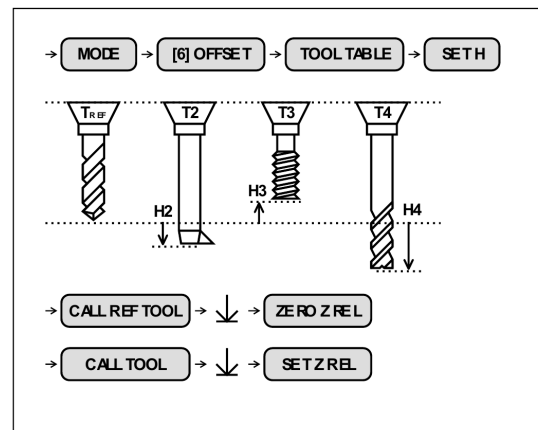
	Method A: reference machine zero	Method B: reference tool
H value stores	Distance from machine zero to the touch-off surface	Difference from the reference tool
Work offset Z	Usually left at zero	Set with the reference tool (Guide 4)
New part zero height	Re-measure every tool	Re-measure Z with the reference tool only
Best for	Simple, one-off jobs	Repeat setups, many tools

SET TOOL LENGTH RELATIVE TO MACHINE ZERO



Method A: each H measured from machine zero [Fig. 9.2-1]

SET TOOL LENGTH RELATIVE TO REFERENCE TOOL



Method B: each H measured against the reference tool [Fig. 9.3-1]

