PathPilot™ Post Processor Guidelines

Product Identification: PathPilot Machine Tool Controller

Background: PathPilot is a dedicated machine controller designed specifically for Tormach machine tools. It shares common code with the open source LinuxCNC project[1]. If your CAM system already supports a LinuxCNC post, this would be a good starting point for a PathPilot post.

PathPilot control implements 98 percent of the Fanuc *standard*. The entire list of supported codes is located on page 2. PathPilot deviates from a standard Fanuc G-code post processor as follows:

Mill:

- G07 and G09 not supported
- G12 and G13 pocketing canned cycles are not supported
- Fanuc G50 scaling is not supported. G50 in PathPilot cancels coordinate system rotation
- G52 local coordinate system offset is not supported (use G92 instead)
- PathPilot supports G54 – G59 as well as G59.1, G59.2, and G59.3 for work offset systems
- G74 tapping cycle for left-hand threads is not supported
- G87 G88 boring cycles are not supported

Lathe:

- Diameter mode only – we do not allow programming in radius values
- G07 G09 not supported
- PathPilot uses G33.1 in place of G32 for spindle-synched moves
- G50 max RPM in CSS is not supported. Instead, we program G96 D<max rpm> S<spindle speed> for max spindle RPM.
- G74 G75 peck drill and peck groove not supported. Instead program G83 for peck or G73 for chip break
- G70-73 roughing cycles not supported

Miscellaneous:

- PathPilot uses G30 before a tool change to move the machine to a pre-settable tool change position. Be sure to include a G30 on the line before a TxM6 command
- Cancelling a canned cycle with G80 also cancels the motion mode. This means that you must explicitly call a G00 or G01 after cancelling a canned cycle before using axis values on a line
- G80 and G50 cannot live on the same line
- G28/G30 moves cannot be made in G91 – machine must be in G90 before a G28 or G30 is executed
- G41/G42 cutter compensation entry move must be a straight G01 move and must be greater than the tool radius

[1] http://linuxcnc.org/

Tormach Inc.
1071 Uniek Drive, Waunakee, WI 53597
Phone: 608.849.8381 / Fax: 209.885.4534

PathPilot Supported Codes

Motion	
G0	Rapid Motion
G1	Coordinated Motion ("Straight Feed")
G2	Coordinated Helical Motion ("Arc Feed") CW
G3	Coordinated Helical Motion ("Arc Feed") CCW
G4	Dwell (No Motion for P Seconds)
G38.2	Straight Probe: Probe Toward Workpiece, Stop on Contact, Signal Error if Failure
G38.3	Straight Probe: Probe Toward Workpiece, Stop on Contact
G38.4	Straight Probe: Probe Away from Workpiece, Stop on Loss of Contact, Signal Error If Failure
G38.5	Straight Probe: Probe Away from Workpiece, Stop on Loss of Contact
G33	Spindle-Synchronized Motion
G33.1	Rigid Tapping
G80	Cancel Motion Mode
Canned Cycles	
G81	Drilling Cycle without Dwell
G82	Drilling Cycle with Dwell
G83	Peck Drill Cycle
G73	Chip-Break Drilling Cycle
G85	Boring Cycle with Dwell
G89	Boring Cycle without Dwell
G76	Multipass Lathe Threading Cycle
Distance Mode	
G90	Absolute Distance Mode
G91	Incremental Distance Mode
G90.1	Arc Centers I, J, K are Absolute
G91.1	Arc Centers I, J, K are Relative to the Arc's Starting Point
G8	X Radius Mode (Lathe)
Feed Rate Mode	
G93	Inverse Time Feed Rate Mode
G94	Units Per Minute Feed Rate Mode
G95	Units Per Revolution Feed Rate Mode
Spindle Control	
M3	Turn Spindle Clockwise (CW)
M4	Turn Spindle Counterclockwise (CCW)

Tormach Inc.
1071 Uniek Drive, Waunakee, WI 53597
Phone: 608.849.8381 / Fax: 209.885.4534

M5	Stop Spindle
G96	Css Mode (Constant Surface Speed)
G97	Rpm Mode
Coolant	
M7	Turn Mist On
M8	Turn Flood On
M9	Turn All Coolant Off
Tool Length Offset	
G43	Use Tool Length Offset from Tool Table
G49	Cancel Tool Length Offset
Stopping	
M0	Program Pause
M1	Optional Pause
M2	End Program
M30	End Program and Rewind
M60	Pallet Change Pause
Units	
G20	Inch Mode
G21	Millimeter Mode
Plane Selection	
G17	Select XY Plane
G18	Select XZ Plane
G19	Select YZ Plane
Cutter Radius Compensation	
G40	Cancel Cutter Radius Compensation
G41	Start Cutter Radius Compensation Left
G42	Start Cutter Radius Compensation Right
Path Control Mode	
G61	Exact Path Mode
G61.1	Exact Stop Mode
G64	Continuous Motion Mode with Optional Path Tolerance
Return Mode in Canned Cycles	
G98	Retract to Prior Position
G99	Retract to R Position
Other Modal Codes	
F	Set Feed Rate

Tormach Inc.
1071 Uniek Drive, Waunakee, WI 53597
Phone: 608.849.8381 / Fax: 209.885.4534

S	Set Spindle Speed
T	Select Tool (also see M6)
M48	Enable Speed and Feed Override Control
M49	Disable Speed and Feed Override Control
M50	Feed Override Control
M51	Spindle Speed Override Control
M53	Feed Stop Control
G54	Select Coordinate System 1
G55	Select Coordinate System 2
G56	Select Coordinate System 3
G57	Select Coordinate System 4
G58	Select Coordinate System 5
G59	Select Coordinate System 6
G59.1	Select Coordinate System 7
G59.2	Select Coordinate System 8
G59.3	Select Coordinate System 9
Flow-control Codes	
O- sub	Subroutines, sub/endsub call
O- while	Looping, while/endwhile do/while
O- if	Conditional, if/else/endif
O- repeat	Run Enclosed Code More than Once
Non-modal Codes	
M6	Change Tool
G10	Data Setting
G28	Return to Reference Point 1
G30	Return to Reference Point 2
G53	Motion in Machine Coordinate System
G92	Offset Coordinate Systems and Set Parameters
G92.1	Cancel Offset Coordinate Systems and Set Parameters to Zero
G92.2	Cancel Offset Coordinate Systems but Do Not Reset Parameters
G92.3	Apply Parameters to Offset Coordinate Systems
Comments and Messages	
(…)	An In-line Comment

Tormach Inc.
1071 Uniek Drive, Waunakee, WI 53597
Phone: 608.849.8381 / Fax: 209.885.4534