Appendix F: Gcode Reference

Contents

Appendix F: G-Code Quick Reference

F.1 Motion Commands (G00-G03)

F.1.1 Positioning Modes

Code	Name	Description	Feedrate	Applications
G00	Rapid Positioning	Move at maximum traverse rate to position	Ignored	Non-cutting moves, tool changes
G01	Linear Interpolation	Move in straight line at programmed feed	F word	Cutting, milling, all machining ops
G02	Circular Interpolation CW	Arc clockwise at programmed feed	F word	Arcs, radii, circular pockets
G03	Circular Interpolation CCW	Arc counter-clockwise at programmed feed	F word	Arcs, radii, circular pockets

Format Examples:

```
G00 X100 Y50 Z10        ; Rapid move to X100, Y50, Z10
G01 X200 Y100 F500      ; Linear move at 500 mm/min
G02 X50 Y50 I25 J0 F300 ; CW arc, center offset I25, J0
G03 X100 Y100 R25 F300  ; CCW arc, radius 25mm
```

F.1.2 Arc Programming (G02/G03)

Center Format (I, J, K): - I = X-axis offset from start point to arc center - J = Y-axis offset from start point to arc center - K = Z-axis offset (for helical interpolation)

Radius Format (R): - R = arc radius - R positive = arc <=180° (short arc) - R negative = arc >180° (long arc)

Example: Quarter-circle arc from (0,0) to (50,50), center at (50,0):

```
G90                     ; Absolute positioning
G00 X0 Y0               ; Rapid to start point
G01 Z-5 F200            ; Plunge to depth
G03 X50 Y50 I50 J0 F500 ; CCW arc (I=50, J=0 offset)
; OR
G03 X50 Y50 R50 F500    ; Same arc using radius
```

Full Circle Programming:

```
G91                     ; Incremental mode
G02 X0 Y0 I25 J0 F500   ; Full circle (returns to start, I=radius)
G90                     ; Back to absolute mode
```

F.2 Coordinate System Commands (G17-G59.3)

F.2.1 Plane Selection

Code	Plane	Arc Axes	Normal Axis	Applications
G17	XY	X, Y	Z	Standard milling (default)
G18	XZ	X, Z	Y	Lathe operations, side facing
G19	YZ	Y, Z	X	Vertical milling, rotary axis

Example:

```
G17             ; Select XY plane (arcs in XY, Z perpendicular)
G02 X10 Y10 R5  ; Arc in XY plane
```

F.2.2 Work Coordinate Systems

Coordinate System Offsets (WCS):

Code	System	Typical Use
G54	Work Offset 1	Primary workpiece/fixture (default)
G55	Work Offset 2	Second fixture or part
G56	Work Offset 3	Third fixture
G57	Work Offset 4	Fourth fixture
G58	Work Offset 5	Fifth fixture
G59	Work Offset 6	Sixth fixture
G59.1	Work Offset 7	Extended offset 1
G59.2	Work Offset 8	Extended offset 2
G59.3	Work Offset 9	Extended offset 3

Setting Work Offsets (Manual):

```
G10 L2 P1 X0 Y0 Z0      ; Set G54 origin to current position
G10 L2 P2 X100 Y0 Z0    ; Set G55 origin 100mm offset in X
```

Using Work Offsets:

```
G54                     ; Activate work offset 1 (fixture A)
G00 X0 Y0 Z10           ; Rapid to fixture A origin + 10mm Z
G01 Z-5 F200            ; Plunge relative to G54 origin
; ... machining operations ...
G55                     ; Switch to work offset 2 (fixture B)
G00 X0 Y0 Z10           ; Rapid to fixture B origin
```

F.2.3 Distance Mode

Code	Mode	Description	Example
G90	Absolute	Coordinates relative to WCS origin	G01 X100 Y50 (move to X100, Y50 in WCS)
G91	Incremental	Coordinates relative to current position	G01 X10 Y5 (move +10 in X, +5 in Y from current)

Common Use:

```
G90 G54          ; Absolute mode, work offset 1
G00 X50 Y50      ; Rapid to X50, Y50 (absolute)
G91              ; Switch to incremental
G01 X10 F500     ; Move +10mm in X (relative)
G90              ; Back to absolute
```

F.3 Feed Rate and Spindle Commands (F, S, M03-M05)

F.3.1 Feed Rate Modes

Code	Mode	Units	Description
G93	Inverse Time	1/min	Feed rate = 1 / (time to complete move in minutes)
G94	Units/Minute	mm/min or in/min	Standard feed rate mode (default)
G95	Units/Revolution	mm/rev or in/rev	Feed per spindle revolution (threading, constant chip load)

F Word (Feed Rate): - **G94 mode:** F500 = 500 mm/min - **G95 mode:** F0.1 = 0.1 mm/rev

Example (G94 - Units per Minute):

```
G94                 ; Units/minute mode
G01 X100 F1000      ; Move to X100 at 1000 mm/min
```

Example (G95 - Units per Revolution):

```
M03 S2000           ; Start spindle at 2000 RPM
G95                 ; Units/revolution mode
G01 X50 F0.15       ; Move to X50 at 0.15 mm/rev
                    ; Actual feed = 2000 RPM × 0.15 mm/rev = 300 mm/min
```

F.3.2 Spindle Control

Code	Function	Parameter	Description
S	Spindle Speed	RPM	S1000 = 1000 RPM

Code	Function	Parameter	Description
M03	Spindle CW	-	Start spindle clockwise (normal)
M04	Spindle CCW	-	Start spindle counter-clockwise (tapping, reverse)
M05	Spindle Stop	-	Stop spindle

Example:

```
M03 S3000       ; Start spindle at 3000 RPM clockwise
G04 P2.0        ; Dwell 2 seconds (spindle ramp-up)
G01 Z-10 F500   ; Plunge to depth
; ... machining ...
M05             ; Stop spindle
```

F.4 Tool and Coolant Commands (M06, M07-M09)

F.4.1 Tool Change

Code	Function	Description
T	Tool Select	T1 = select tool #1 (not activated)
M06	Tool Change	Execute tool change (load selected tool)

Manual Tool Change Sequence:

```
T2              ; Select tool 2 (e.g., 6mm endmill)
M06             ; Execute tool change (machine stops, prompts operator)
G43 H2 Z10      ; Apply tool length offset for tool 2, move to Z10
M03 S4000       ; Start spindle
; ... machining with tool 2 ...
```

Automatic Tool Changer (ATC):

```
T1 M06          ; Select and load tool 1 automatically
G43 H1          ; Apply tool offset
```

F.4.2 Tool Length Offset

Code	Function	Description
G43	Tool Length Offset +	Apply positive tool length offset (H word)
G44	Tool Length Offset -	Apply negative offset (rarely used)
G49	Cancel Tool Offset	Remove tool length compensation

Example:

```
G43 H5 Z100     ; Apply offset for tool 5, move Z to 100mm (compensated)
G49             ; Cancel offset (return to machine coordinates)
```

F.4.3 Coolant Control

Code	Function	Description
M07	Mist Coolant On	Activate mist coolant (fine spray)
M08	Flood Coolant On	Activate flood coolant (high flow)
M09	Coolant Off	Stop all coolant

Example:

```
M08             ; Flood coolant on
G01 X100 F800   ; Cutting move with coolant
M09             ; Coolant off (end of program)
```

F.5 Canned Cycles (G80-G89)

F.5.1 Drilling Cycles

Code	Cycle	Description	Parameters
G80	Cancel	Cancel canned cycle	-
G81	Drill	Rapid to R, feed to Z, rapid out	R, Z, F
G82	Spot Drill	Drill + dwell at bottom	R, Z, P (dwell), F
G83	Peck Drill	Drill with pecking (chip clearing)	R, Z, Q (peck depth), F
G85	Bore	Feed in, feed out (finish bore)	R, Z, F
G73	High-Speed Peck	Partial retract between pecks	R, Z, Q, F

Parameters: - **R:** Retract plane (safe height above workpiece) - **Z:** Hole depth (final depth) - **Q:** Peck increment (G83, G73) - **P:** Dwell time in seconds (G82, G89) - **F:** Feed rate

Example: Drilling 10 holes with G81

```
G90 G54             ; Absolute mode, work offset 1
G00 X0 Y0           ; Rapid to first hole location
G81 R5 Z-20 F200    ; Define drill cycle: R=5mm retract, Z=-20mm depth
X10                 ; Drill hole 2 at X10 (Y unchanged)
X20                 ; Drill hole 3 at X20
X30 Y10             ; Drill hole 4 at X30, Y10
; ... additional holes ...
G80                 ; Cancel drill cycle
```

Example: Deep Hole Peck Drilling (G83)

```
G83 R5 Z-50 Q5 F150 ; Peck drill: R=5mm safe, Z=-50mm depth, Q=5mm peck increment
X0 Y0               ; Hole 1
X25 Y0              ; Hole 2
G80                 ; Cancel
```

F.5.2 Canned Cycle Behavior

Sequence (G81 example): 1. Rapid (G00) in XY to hole location 2. Rapid (G00) in Z to R plane (retract height) 3. Feed (G01) to Z depth at F feedrate 4. Rapid (G00) back to R plane

Modal Behavior: Canned cycle remains active until G80 or another motion mode (G00, G01, G02, G03) is called.

F.6 Dwell and Program Control (G04, M00-M02)

F.6.1 Dwell Command

Code	Function	Parameter	Description
G04	Dwell	P (seconds)	Pause program for specified time

Examples:

```
G04 P2.5        ; Dwell for 2.5 seconds
G04 P0.5        ; Dwell for 0.5 seconds (spindle stabilization)
```

Common Uses: - Spindle ramp-up after M03 - Chip clearing pause - Finish pass dwell (dimensional stability)

F.6.2 Program Stop Commands

Code	Function	Description	Resume
M00	Program Stop	Unconditional stop, spindle/coolant off	Operator presses cycle start
M01	Optional Stop	Stop only if optional stop switch enabled	Cycle start
M02	Program End	End program, reset to start	Restart program
M30	Program End & Reset	End + rewind, return to start position	Restart program

Example: Mid-Program Tool Check

```
G01 X50 Y50 F800
M00             ; Stop for inspection (operator checks part)
```

```
; (Operator presses cycle start to continue)
G01 X100 Y100
```

F.7 Parametric Programming and Macros

F.7.1 Variables

Persistent Variables (Linuxcnc #-parameters): - #1 - #30: Temporary variables (cleared on program end) - #100 - #999: Persistent variables (saved between programs) - #5220: X-axis work offset (G54) - #5221: Y-axis work offset (G54)

Example:

```
#1 = 100                ; Store 100 in variable #1
#2 = 50                 ; Store 50 in variable #2
G01 X[#1] Y[#2] F500    ; Move to X100, Y50 (use variables)
```

F.7.2 Expressions and Operators

Supported Operators: - Arithmetic: +, −, ∗, /, MOD (modulus) - Comparison: EQ, NE, LT, LE, GT, GE - Logical: AND, OR, XOR - Functions: SIN, COS, TAN, SQRT, ABS, ATAN

Example: Bolt Circle Pattern

```
; Drill 8 holes on 50mm diameter bolt circle
#1 = 0                  ; Hole counter
o100 while [#1 LT 8]
  #2 = [#1 ∗ 360 / 8]   ; Angle for this hole
  #3 = [25 ∗ COS[#2]]   ; X position (radius 25mm)
  #4 = [25 ∗ SIN[#2]]   ; Y position
  G81 X[#3] Y[#4] R2 Z−10 F200
  #1 = [#1 + 1]         ; Increment counter
o100 endwhile
G80                     ; Cancel drill cycle
```

F.7.3 Conditional Branching

IF-THEN-ELSE:

```
#1 = 5
o100 if [#1 GT 10]
  G01 X100 F500         ; Execute if #1 > 10
o100 else
  G01 X50 F500          ; Execute if #1 <= 10
o100 endif
```

Subroutines:

```
o200 sub                ; Define subroutine o200
  G01 X[#1] Y[#2] F500
```

```
  G02 X[#3] Y[#4] R10
o200 endsub

; Call subroutine with parameters
o200 call [10] [20] [30] [40]  ; #1=10, #2=20, #3=30, #4=40
```

F.8 Common G-Code Program Structure

F.8.1 Program Template

```
%                               ; Program start delimiter (optional)
O1000                           ; Program number
(Generic Milling Program)       ; Comment
G21                             ; Metric units (G20 for inch)
G90                             ; Absolute positioning
G17                             ; XY plane selection
G40 G49 G80                     ; Cancel: cutter comp, tool offset, canned cycle

T1 M06                          ; Load tool 1
G43 H1 Z50                      ; Tool length offset, safe Z height
G54                             ; Work coordinate system 1
M03 S4000                       ; Spindle on, 4000 RPM
G04 P2.0                        ; Dwell 2 seconds (spindle ramp)
M08                             ; Coolant on

; Machining operations
G00 X10 Y10                     ; Rapid to position
G01 Z-5 F200                    ; Plunge to depth
X50 Y50 F800                    ; Cut to X50, Y50
G02 X70 Y50 I10 J0 F800         ; Arc
G01 Z5 F500                     ; Retract

M09                             ; Coolant off
M05                             ; Spindle off
G00 Z50                         ; Retract to safe Z
G53 G00 Z0                      ; Move to machine home Z (G53 = machine coords)
M30                             ; Program end and rewind
%                               ; Program end delimiter (optional)
```

F.8.2 Safety Lines (Program Start)

Standard Safety Block:

```
G21                     ; Metric mode (or G20 for inch)
G90                     ; Absolute distance mode
G17                     ; XY plane
G40                     ; Cancel cutter radius compensation
```

```
G49                     ; Cancel tool length offset
G80                     ; Cancel canned cycles
G54                     ; Select work coordinate system
G94                     ; Feed per minute mode
```

Explanation: Ensures machine is in known state (important after E-stop or program interruption).

F.9 Common Modal G-Codes Summary

Modal Groups (only one code active per group):

Group	Codes	Function
1 (Motion)	G00, G01, G02, G03, G80-G89	Motion type (rapid, linear, arc, canned cycle)
2 (Plane)	G17, G18, G19	Plane selection (XY, XZ, YZ)
3 (Distance)	G90, G91	Absolute vs. incremental positioning
5 (Feed Mode)	G93, G94, G95	Feed rate mode (inverse time, per min, per rev)
6 (Units)	G20, G21	Inch vs. metric units
7 (Cutter Comp)	G40, G41, G42	Cutter radius compensation
8 (Tool Offset)	G43, G44, G49	Tool length offset
10 (Return Mode)	G98, G99	Canned cycle return (initial level, R-point)
12 (Work Offset)	G54-G59.3	Work coordinate system selection

Non-Modal Commands (execute once, don't persist): - G04 (dwell) - G10 (coordinate system setting) - G28, G30 (return to home) - G92 (coordinate system offset - use with caution)

End of G-Code Quick Reference Appendix