

Introduction in GCode

Florian Festi

Disclaimer

- I have no clue either
- ~200 commands – can't handle all
- Won't do configuration
- Gcode is only part of the game
- Different machine types

GCode

- Programming CNC Machines
- RS274/NGC, ISO 6983
- Line based
- Letters A-Z as parameters
- Gnn (and Mnn) as commands
- Lots of different commands 150 to 200
- Implementations vary

LinuxCNC

- Free, real time CNC controller
- Requires (obscure) real time kernel
- Comes as Ubuntu installable live CD
- Connects via parallel port only
- Supports stepper and servo motors
- Implements large parts of Gcode “standard”
- <http://www.linuxcnc.org/>

Axis and Joints

- Axis – create “virtual” coordinate system
- Joints – physical actuators
- X, Y, Z – linear axis
- A, B, C – rotary axis
- U, V, W – secondary, linear axis
- Joints 0, 1, ..., 7, 8
 - corresponding to X,Y,Z, A, B, C, U, V, W
- Homing

Example

```
G17 (XY) G21 (mm) G40 G49 (no comp)
G54 (CS1) G80 90 (abs) G94 (mm/min)
G0 Z5
G0 X0 Y0
G1 Z-3 F300
G1 Y100
G1 X50 Y150
G1 X100 Y100
G1 X0
G1 X100 Y0
G1 Y100
G1 X0 Y0
G1 X100
M2 ; End of program.
```

Gcode from D to T

- D - Diameter
- E - prEcision feed
- F - Feedrate
- H - tool Height
- I, J, K - Circle center
- L - Loop count
- N - line Number
- O - prOgram
- P - Parameter
- Q – peck increment
- R - Radius
- S - Spindel Speed
- T - Tool

Gcode Movements

- G0 – Rapid linear motion
- G1 – Linear feed
- Arcs G2, G3
 - Select plane; XY: G17, XZ: G18, YZ: G19
 - G90.1 absolute center, G90.2 relative center mode
 - Endpoint and center or endpoint and radius
 - Helix if other axis is given

Other Movement

- B-Splines and NURBS
- Canned Cycles
 - For drilling and boring
 - R for retract
 - G73, G81 – G89
 - G76 for cutting threads
 - G80 to cancel cycle

MCodes

- M1, M2 - Pause
- M2, M30 - End Program
- M3, M4, M5 - Spindle CW, CCW, Stop
- M6 Tn - Tool change
- M7, M8, M9 – mist, flood coolant, all off
- M61 Qn - Set current tool to n
- M100 – M199 User defined commands
- IO and feed override

Coordinate Systems

- G54-G59, G59.1-G59.3 – CS1 to CS9
- G10 L2 P- <axes R->
- “Touch off” in GUI
- G53 – Machine CS (non modal!)

- G92 <axis> - gives current position the coords
- G92.1, G92.2, G92.3 – zero, suspend, apply

Coordinates

- Polar coordinates @radius ^angle
 - XY plane only
 - From X0 Y0 – move CS or use offset
- G90, G91 – absolute and relative mode
- Warning! Radius also increments in relative mode!

Tools and Toolcompensation

- Tool Table
- Cutter compensation
 - G40, G41, G42 – off, left right
 - XY or XZ plane only
 - Needs move
- G43 H- - Tool length compensation
 - G49 to switch off

O-Codes

- LinuxCNC extension for execution flow control
- Beware of o vs 0
- oNNN
- sub/endsub/call
- while/endwhile, do/while, repeat/endrepeat

Further topics

- Path mode and path blending
- Configuration
- Complex cinematics
- Gcode generation
- Special machine types