G-code reference sheet

G-codes are the codes that position the tool and do the actual work.

G0: Rapid positioning
G1: Linear interpolation
G2: Clockwise circular/helical interpolation
G3: Counterclockwise circular/helical interpolation
G4: Dwell
G10: Coordinate system origin setting
G12: Clockwise circular pocket
G13: Counterclockwise circular pocket
G15/G16: Polar Coordinate moves in G0 and G1
G17: XY plane select
G18: XZ plane select
G20/G21: Inch/Millimeter unit
G28: Return home
G30: Return home
G31: Straight probe
G40: Cancel cutter radius compensation
G41/G42: Start cutter radius compensation left/right
G43: Apply tool length offset (plus)
G49: Cancel tool length offset
G50: Reset all scale factors to 1.0
G51: Set axis data input scale factors
G52: Temporary coordinate system offsets
G53: Move in absolute machine coordinate
G54: Use fixture offset 1
G55: Use fixture offset 2
G56: Use fixture offset 3
G57: Use fixture offset 4
G58: Use fixture offset 5
G59: Use fixture offset 6
G61/G64: Exact stop/Constant Velocity mode
G68/G69: Rotate program coordinate
G70/G71: Inch/Millimeter unit
G80: Cancel motion mode
G90: Absolute distance mode
G91: Incremental distance mode
G92: Offset coordinates and set parameters
G93: Inverse time feed mode
G94: Unite per minute
G98: Rapid Height by Z height
G99: Rapid Height by R height

M-codes manage the machine

M0: Program stop
M1: Optional program stop
M2: Program end
M3/M4: Rotate spindle clockwise/counterclockwise
M5: Stop spindle rotation
M6: Tool change
M30: Program end and rewind
M47: Repeat program from first line
M98: Call subroutine
M99: Return from subroutine/repeat

S codes are related to the tool Speed
F codes are related to the tool Feed

T codes are tool related and represent the tool number.

H codes are tool height related and represent the height offset listed in the tool library for the tool listed. Example: H6 = height offset for Tool 6