C101

SINUMERIK HANDBOOK

G codes & M codes

This document shows what are the G codes & M codes required for SINUMERIK programming in Turning and Milling applications.

SIEMENS

Higher productivity with SINUMERIK

Whether it's a basic part or a complex workpiece, the Siemens control system offers you a wealth of functionality with a very high degree of operator friendliness and manufacturing efficiency.

Description of the document:

Know all about the concept of G codes & M codes for effective programming.

- Preparatory functions (G)   Overview of G codes
- Miscellaneous functions (M)   Overview of M codes

Applicability:

- Technology   2 axis Turning, Turn-mill & Milling applications
- SINUMERIK controllers   802D, 810D, 840D, 808D, 828D, 840Dsl & SINUMERIK ONE

Index

Preparatory codes / G codes

G stands for **'Geometric code'**. This code is used to instruct the machine where to move, how fast to move and which path to follow.

G-code is also known as the **'Preparatory code'**, "G" is followed by number which is a command to change the geometry.

List of Preparatory functions (G codes)

Sl no	Functions	Syntax
G00	Rapid Traverse	G00 X_ Y_ Z_
G01	Liner interpolation	G01 X_Y_Z_ F_
G02	Circular interpolation clock wise	G02 X_ Z_ CR=
G03	Circular interpolation Counter clock wise	G03 X_ Y_ CR=
G04	Dwell Time in seconds	G04 F1
G09	Exact stop for applied blocks only - Non modal	G00 G09
G17	XY plane - Milling	G17
G18	XZ plane - Turning	G18
G19	YZ plane	G19
G33	Thread cutting with constant lead	G33 X_ Z_K_SF=0
G40	TNRC Cancel	G40 X_ Y_ Z_
G41	TNRC left compensation	G41 X_ Y_Z_
G42	TNRC right compensation	G42 X_ Y_ Z_
G54 to G57	Settable offset/work offset	G54
G505 to G599	Settable offset/work offset-option	G505
G60	Exact stop - Modal	G0 G60
G64	Continuous path mode	G0G64
G71	Metric dimensions input	G71 X_ Y_ Z_
G70	Inch dimensions input	G70 X_ Y_ Z_
G90	Absolute dimensions	G90 X_ Y_ Z_
G91	Incremental dimensions	G91 X_ Y_ Z_
G94	Linear feedrate in mm/min	G94 X_ Y_ Z_ **F250**
G95	Revolution feedrate in mm/rev	G95 X_ Z_F0.2
G96	Constant cutting speed on	LIMS=2000 G96 S150 M03
	LIMS S150 M03	Limit for main spindle. Cutting speed mm/min spindle direction
G97	Constant cutting speed off	G97S1800M03
G110	Pole specification relative to the last programmed set point position	G110 X_Z_RP_AP_
		RP=Polar radius AP=Polar angle
G111	Pole specification relative to origin of current work piece coordinate system	G110 X_Z_RP_AP_
G112	Pole specification relative to the last valid POLE	G110 X_Z_RP_AP_
G290	Programming in Siemens mode	G290
G291	Programming in ISO Mode	G291
G500	Cancel the work offset - Modal	G0 G500 X0 Y0 Z0
G53	Cancel the work offset - Non modal	G0 G53 X0 Y0 Z0

Miscellaneous functions / M codes

M - code is Machine language code to activate miscellaneous functions.

M stands for **'Machine codes'**. This code is used for non-geometry machine functions like coolant on/off, spindle speed, tool change, pallet change etc.

List of Miscellaneous functions (M codes)

Sl no	Functions
M00	Program stop
M01	Optional stop
M02	Program end
M03	Spindle clockwise direction
M04	Spindle counter clockwise direction
M05	Spindle stop
M06	Tool change
M17	Subprogram end
M19 OR SPOS=0	Spindle orientation
M30	Main program end

Note: Miscellaneous commands are machine manufacturer specific. Contact your machine manufacturer for more details on M functions.

To know more about SINUMERIK

S18328