

G-CODES AND M-FUNCTIONS - TURNING

Fanuc CNC has an option of three G-code groups (A, B and C). The most common G-code group in North America is the **A** group. All G-codes listed are for the G-code group **A**, unless otherwise specified ! *Groups cannot be mixed !*

G00	Rapid positioning	G72	Profile roughing cycle for facing
G01	Linear interpolation	G73	Pattern repeating cycle
G02	Circular interpolation - clockwise	G74	Peck drilling cycle along the Z-axis
G03	Circular interpolation - counterclockwise	G75	Grooving cycle along the X-axis
G04	Dwell function (as a separate block)	G76	Complex thread cutting cycle
G07	Hypothetical axis interpolation	G90	Cutting cycle A <i>Grp A</i>
G09	Exact stop check - one block only	G90	Absolute input of dimensions <i>Grp B</i>
G10	Programmable data input (Data setting)	G91	Incremental input of dimensions <i>Grp B</i>
G11	Data setting mode cancel	G92	Simple thread cutting cycle <i>Grp A</i>
G18	ZX plane designation	G92	Tool position register <i>and/or: Grp B</i>
G20	English units of input	G92	Maximum spindle RPM/CSS mode <i>Grp B</i>
G21	Metric units of input	G94	Cutting cycle B <i>Grp A</i>
G22	Stored stroke check - on	G94	Feed per minute (in/min or mm/min) <i>Grp B</i>
G23	Stored stroke check - off	G95	Feed per revolution (in/rev or mm/rev) <i>Grp B</i>
G25	Spindle fluctuation detection - on	G96	Constant surface speed control – CSS
G26	Spindle fluctuation detection - off	G97	Constant surface speed control cancel - RPM
G27	Machine zero position check	G98	Feed per minute (in/min or mm/min) <i>Grp A</i>
G28	Machine zero return (reference point 1)	G99	Feed per revolution (in/rev or mm/rev) <i>Grp A</i>
G29	Return from machine zero		
G30	Machine zero return (reference point 2)	M00	Compulsory program stop
G31	Skip function	M01	Optional program stop
G32	Threading function - constant lead	M02	End of program (usually no reset and rewind)
G34	Threading function - variable lead	M03	Spindle rotation normal – CW
G35	Circular thread cutting – CW	M04	Spindle rotation reverse – CCW
G36	Circular thread cutting – CCW <i>or:</i>	M05	Spindle rotation stop
G36	Automatic tool compensation (X-axis)	M07	Coolant mist – on
G37	Automatic tool compensation (Z-axis)	M08	Coolant pump motor – on
G40	Tool nose radius compensation - cancel	M09	Coolant pump motor – off
G41	Tool nose radius compensation - left	M10	Chuck open
G42	Tool nose radius compensation - right	M11	Chuck close
G50	Tool position register <i>and/or:</i>	M12	Tailstock quill in (non-standard)
G50	Maximum spindle RPM in CSS mode	M13	Tailstock quill out (non-standard)
G52	Local coordinate offset	M17	Turret indexing forward (non-standard)
G53	Machine coordinate system	M18	Turret indexing reverse (non-standard)
G54	Work coordinate offset 1	M19	Programmable spindle orientation (OPTION)
G55	Work coordinate offset 2	M21	Tailstock forward (non-standard)
G56	Work coordinate offset 3	M22	Tailstock backward (non-standard)
G57	Work coordinate offset 4	M23	Gradual pull-off from thread – on
G58	Work coordinate offset 5	M24	Gradual pull-off from thread – off
G59	Work coordinate offset 6	M30	End of program (with reset and rewind)
G61	Exact stop mode	M41	Low gear range selection (if applicable)
G62	Automatic corner override mode	M42	Medium gear range selection 1 (if applicable)
G64	Cutting mode	M43	Medium gear range selection 2 (if applicable)
G65	Custom macro call	M44	High gear range selection (if applicable)
G68	Mirror image for double turrets - on	M48	Feedrate override cancel - off (deactivated)
G69	Mirror image for double turrets - cancel	M49	Feedrate override cancel - on (activated)
G70	Profile finishing cycle	M98	Subprogram call
G71	Profile roughing cycle for turning and boring	M99	Subprogram end