ROUTER G CODES - FANUC

⇨ G codes marked * are initial G codes when power is turned on.
⇨ G codes standing alone are **NON-MODAL** ("**one-shot**") G codes, which are **only active once** and must be called each time they are needed.
⇨ **MODAL** codes **remain active** until cancelled or replaced with another code from the **same** group. **Only one G code per group can be active at any time.**
⇨ In addition to most G codes, M, S, T, X, Y, Z and F codes are ALL modal codes.

G CODE	FUNCTION	MEANING
G0	Rapid traverse	A non-cutting, fast positioning move.
G1*	Straight cut	A straight cutting move in any direction.
G2	CW circular helical interpolation	A curve or arc cut in the clockwise direction.
G3	CCW circular helical interpolation	A curve or arc cut in the counterclockwise direction.
G9 non-modal	Exact stop	A built-in timer stops the forward movement of the tool for only 1/4 second, resulting in a sharp corner.
G17*	XY plane selection	Indicates X and Y will be the horizontal axes used, and Z is assumed to be the vertical axis. G17 is almost always used, rather than G18 or G19.
G18	ZX plane selection	
G19	YZ plane selection	
G20*	Input in inches	All data is input and displayed in inches.
G21	Input in millimeters	All data is input and displayed in millimeters.
G28 non-modal	References to machine home	Temporarily overrides all work coordinates which refer to a part zero, and references from machine home.
G40*	Cancel cutter diameter compensation	Cancels the control's ability to adjust the program for the diameter of this tool.
G41	Cutter diameter compensation left	Compensates for the tool diameter while doing a climb cut. Shifts the tool to the left, and keeps it there all the way around the part.
G42	Cutter diameter compensation right	Compensates for the tool diameter while doing a conventional cut. Shifts the tool to the right, and keeps it there all the way around the part.
G43	Tool height compensation	Allows the control to establish a new Z0 for the part.
G49*	Cancel tool height compensation	Cancels the control's ability to adjust for the height of the tool to the part.
G52 non-modal	Local coordinate system setting	Allows many local part zeros to be established anywhere on the table, which all still reference from the main part zero in the corner of the table.
G53 non-modal	Machine coordinate system selection	Temporarily overrides all work coordinates which refer to a part zero, and references from machine home.

G54* G55 G56 G57 G58 G59	Work coordinate no. 1 Work coordinate no. 2 Work coordinate no. 3 Work coordinate no. 4 Work coordinate no. 5 Work coordinate no. 6	These are memory areas used to store negative machine position values for establishing a part zero at the corner of the table for each spindle.
G61	Exact stop	G61 is modal (G9 is not), so when activated it will make the 1/4 second pause at the end of each cut until it is cancelled.
G64*	Cancel exact stop	Overrides G61 because they are in the same group.
G73	High speed peck drilling cycle	Pecks down each stroke by a specified amount, and does a very small *partial retraction* between each peck to break the chip.
G80*	Cancel canned cycle	A canned cycle is any type of repetitive drilling cycle like G73, G81 and G83. Any in use will be cancelled by G80.
G81	Standard drilling cycle	The most commonly used drilling cycle. Feeds the tool down into a hole at a specified feed rate, then rapid traverses the tool out of the hole to a safe place above.
G83	Deep hole peck drilling cycle	Pecks down each stroke by a specified amount, and does a *full retraction* between each peck.
G90*	Absolute coordinate system	Used 90% of the time because of its advantages. Always specify X, Y and Z ending points measured from part zero.
G91	Relative coordinate system, also called incremental coordinate system	Always specify X, Y and Z ending points measured from the last point.
G98	Return to initial plane in canned cycle	Used only in canned drilling cycles to retract the tool up to the initial Z point (rather than just above the hole).
G99	Return to R plane in canned cycle	Retract the tool up to a pre-determined Z height just barely above each hole after it has been drilled with a canned drilling cycle.

ROUTER M CODES - FANUC

⇨ **Only one (1) M code is allowed per block of program.**

M CODE	FUNCTION	MEANING
M0	Program stop	Each time an M0 is read, the program stops and the spindle continues to rotate. Depress CYCLE START to restart.
M1	Optional stop	When the OPTIONAL STOP button is lit, the program stops each time an M1 is read and the spindle continues to rotate. Depress CYCLE START to restart.
M3	Spindle CW	Commands the spindle to rotate in the clockwise direction.
M5	Spindle stop	Commands all spindles to stop rotating and all spindle air cyclinders to raise to the inactive position.
M6	Tool change (option)	If a dynamic tool changer was purchased, this code causes the spindle to stop and raise to the inactive position, then do a fully automatic tool change.
M7	Coolant mist on (option)	When turned on, this option prevents aluminum or plastic chips from sticking to the tool.
M8	Spindle air blast on (option)	When turned on, this option will assist in removal of chips from a deep cut.
M9	Coolant mist off / spindle air blast off (option)	Turn off and de-activate mist coolant and air blast options.
	M22, M23 and M24 are applicable only to twin table machines.	
M22	Left table	Activates only the left table.
M23	Right table	Activates only the right table.
M24	Both tables	Activates left and right tables simultaneously, to be used as one large table. (This capability is an option on VR 1605TT.)
M30	End of program	A built-in M5 deactivates all spindles by stopping rotation and raising air cylinders. The cursor is reset to the beginning of the program.
M98	Call sub-program	Leaves the current main program, calls a sub-program from the library and executes it.
M99	End of sub-program	Returns to the most recent main program, reads and executes the next block of information because the sub-program is finished.

ROUTER S CODES - FANUC

SPINDLE SPEED COMMAND

Used to inform the NC what spindle rotation speed (RPM) the program requires.

SPEED RANGE

Main spindles, collet	6000 to 18,000 RPM
Main spindles, 30 taper quick-change	1000 to 18,000 RPM
Main spindles, HSK	1000 to 24,000 RPM
7.5 HP Piggyback spindles	6000 to 18,000 RPM
1/3 HP Drill	3600 RPM* fixed
9 or 11 Spindle line boring unit	2800 RPM* fixed
20 Spindle line boring unit	4500 RPM* fixed

* Any 2 digit S code will turn on these boring units to their fixed RPM.

NOTE: 1. The maximum recommended cutter diameter for use in main spindles is 4".
The maximum recommended tool shank diameter for main spindles is 3/4".
The maximum recommended tool shank diameter for HSK spindles is 1".

2. The maximum recommended cutter diameter for use in 7.5 HP piggyback spindles is 2".
The maximum recommended tool shank diameter for 7.5 HP piggyback spindles is 5/8".

3. The maximum RPM ratings recommended below are only approximate, as actual value depends on the design of the tool. Please ask your tooling supplier for specific recommendations.
The maximum recommended RPM of a 4" diameter cutter is 7000 RPM.
The maximum recommended RPM of a 3" diameter cutter is 10,000 RPM.
The maximum recommended RPM of a 2" diameter cutter is 12,000 RPM.

ROUTER T CODES - FANUC

MAIN SPINDLES

T101
T102
T103
.
.
T199

When used in either AUTO/MEM or MDI mode, the above T codes will cause:
- The indicated T code to become modal.
- All T codes not in the 100 series to be canceled.
- The indicated spindle to lower to the active position and rotate if an M03 is issued with an S code and the spindle down proximity switch is true.
- All active spindles / tools not in the 100 series to raise to the inactive position and cease to rotate.

AUXILIARY (PIGGYBACK) SPINDLES

T201
T202
T203
.
.
T299

When used in either AUTO/MEM or MDI mode, the above T codes will cause:
- The indicated T code to become modal.
- All T codes not in the 200 series to be canceled.
- The indicated spindle to lower to the active position and rotate if an M03 is issued with an S code and the spindle down proximity switch is true.
- The accompanying spindle to lower to the active position if an M03 is issued with and S code and the spindle down proximity switch is true.
- All active spindles / tools not in the 200 series to raise to the inactive position and cease to rotate.

NOTES

1. A drill T code may not be used with a spindle T code.
2. If a Line Boring Assembly option has been purchased, it may have been mounted in place of a main spindle. If so, EXTREME CAUTION should be exercised when entering T codes. Router spindles should not be used with the Line Boring T code.
3. When a new T code is called up, active spindles and tools not in the same series will automatically raise. An M05 need not be commanded.
4. If a spindle raises due to a different spindle being called up, the inactive spindle will stop rotating.
5. To command a spindle to lower but not rotate, issue an M03 with S0 (zero).
6. Each T code **MUST** be programmed on its own line, with no other commands on that line. For example, to lower and rotate four main heads at once the program could be written:

```
N10 T101;
N20 T102;
N30 T103;
N40 T104;
N50 M03 S16000;
```

2ND AUXILIARY SPINDLES (LINE BORING UNITS)

T301
T302
T303
.
T399

When used in either AUTO/MEM or MDI mode, the above T codes will cause:
- The indicated T code to become modal.
- All T codes not in the 300 series to be canceled.
- The indicated spindle to lower to the active position and rotate if an M03 is issued with an S code and the spindle down proximity switch is true.
- The accompanying spindle (if applicable) to lower to the active position if an M03 is issued with an S code and the spindle down proximity switch is true.
- All active spindles / tools not in the 300 series to raise to the inactive position and cease to rotate.

OPTIONAL BINARY T CODES - LINE BORING UNITS

The main advantage to programming the line boring unit(s) in binary code is to shorten cycle time.

T301 = T0001 M9#	T314 = T0001 M9#	T323 = T0001 M9#	T336 = T0001 M9#
T302 = T0002 M9#	T315 = T0002 M9#	T324 = T0002 M9#	T337 = T0002 M9#
T303 = T0004 M9#	T316 = T0004 M9#	T325 = T0004 M9#	T338 = T0004 M9#
T304 = T0008 M9#	T317 = T0008 M9#	T326 = T0008 M9#	T339 = T0008 M9#
T305 = T0016 M9#	T318 = T0016 M9#	T327 = T0016 M9#	T340 = T0016 M9#
T306 = T0032 M9#	T319 = T0032 M9#	T328 = T0032 M9#	T341 = T0032 M9#
T307 = T0064 M9#	T320 = T0064 M9#	T329 = T0064 M9#	T342 = T0064 M9#
T308 = T0128 M9#	T321 = T0128 M9#	T330 = T0128 M9#	T343 = T0128 M9#
T309 = T0256 M9#	T322 = T0256 M9#	T331 = T0256 M9#	T344 = T0256 M9#
T310 = T0512 M9#		T332 = T0512 M9#	
T311 = T1024 M9#	(T301-T322 are one	T333 = T1024 M9#	(T323-T344 are one
T312 = T2048 M9#	head)	T334 = T2048 M9#	head)
T313 = T4096 M9#		T335 = T4096 M9#	

M9# represents the main spindle associated with the line boring unit:
- M91 = a line boring unit associated with spindle 101 or 201
- M92 = a line boring unit associated with spindle 102 or 202
- M93 = a line boring unit associated with spindle 103 or 203
- M94 = a line boring unit associated with spindle 104 or 204
- M95 = a line boring unit associated with spindle 105 or 205
- M96 = a line boring unit associated with spindle 106 or 206

For example, to call T302, T305, T306 and T310 at once:

existing code	binary code	
T302	T0562 M91	(2+16+32+512 = 562)
T305	M03 S2000	
T306		
T310		
M03 S2000		

T301	T302	T303	T304	T305	T306	T307	T308	T309	T310	T311	T312	T313
T0001	T0002	T0004	T0008	T0016	T0032	T0064	T0128	T0256	T0512	T1024	T2048	T4096
0	1	0	0	1	1	0	0	0	1	0	0	0

ROUTER TOOL CHANGE CODES - FANUC

T1001 M6;
T1002 M6;
T1003 M6;
...
...
T1008 M6;

T2001 M6;
T2002 M6;
...
...
T2008 M6;

The tool change code consists of two parts. The first portion of the T code for a tool change identifies the spindle (head) on which the tool change will be applied.

T 2 _ _ _ means spindle T102 will be used
T 1 _ _ _ means spindle T101 will be used

The remaining portion of the tool change code identifies the pocket (location) in the toolchanger carousel. The carousel pockets are numbered 1 through 8.

T _ 0 0 1 identifies pocket 1 on the toolchanger
T _ 0 0 6 identifies pocket 6 on the toolchanger

For example:

T1003 M6; tool change on spindle 101, pocket 3
T2004 M6; tool change on spindle 102, pocket 4

NOTES:

1. M6 must be used on the same line as the T code.

2. Prior to performing a tool change, an M5 should be commanded and the Z axis should be returned to a "safe" plane.