

The following exercise tests your knowledge of common preparatory commands for Fanuc milling controls. Match the G-codes from the numbered column with the answers in the letter column.

1	G01	A	Circular interpolation clockwise
2	G20	B	Standard drilling cycle
3	G99	C	Incremental mode of dimensional input
4	G54	D	Dwell command
5	G17	E	Machine zero return command
6	G02	F	Linear interpolation mode
7	G80	G	Spot drilling cycle
8	G98	H	Circular interpolation counterclockwise
9	G91	I	Input of imperial units of measurement
10	G03	J	Work offset 1
11	G81	K	Return to initial level in a fixed cycle
12	G21	L	Cutter radius offset cancel command
13	G90	M	Tool length offset command
14	G04	N	Fixed cycle cancel command
15	G40	O	XY plane selection
16	G28	P	Absolute mode of dimensional input
17	G84	Q	Return to the R-level in a fixed cycle
18	G41	R	Input of metric units of measurement
19	G82	S	Cutter radius offset to the left
20.	G43	T	Right hand tapping cycle

Item number ...	... matches letter	Item number ...	... matches letter
1		11	
2		12	
3		13	
4		14	
5		15	
6		16	
7		17	
8		18	
9		19	
10		20	