Instructional Video Series
GENMILL
Video Support Document

Contents

Introduction

The Purpose of this document is to give additional information for the instructional videos. Please click on the Table of Content to jump to the desired information.

818 349-9166
www.expandmachinery.com/training

G Code List

G Code	Group	Function	Section
Δ 00	01	Positioning	6.1
0.5	28	Rapid traverse block overlap	7.13.1
Δ 01	01	Linear interpolation	6.2
02	01	Circular interpolation CW	6.3
		R-specified circular interpolation CW	6.4
		Helical interpolation CW	6.7
		Spiral/Conical interpolation CW (type2)	6.14
03	01	Circular interpolation CCW	6.3
		R-specified circular interpolation CCW	6.4
		Helical interpolation CCW	6.7
		Spiral/Conical interpolation CCW (type2)	6.14
02.1	01	Spiral/Conical interpolation CW (type1)	6.14
03.1	01	Spiral/Conical interpolation CCW (type1)	6.14
02.3	01	Exponential function interpolation positive rotation	6.12
03.3	01	Exponential function interpolation negative rotation	6.12
02.4	01	3-dimensional circular interpolation CW	6.15
03.4	01	3-dimensional circular interpolation CCW	6.15
04	00	Dwell (Time-based Designation)	8.1
05	00	High-speed Machining Mode	17.1
		High-speed high-accuracy control II/III	17.3
05.1	00	High-speed high-accuracy control I	17.3
		Spline interpolation	17.4
06.2	01	NURBS interpolation	6.16
07	00	Hypothetical axis interpolation	6.17
07.1 107	21	Cylindrical interpolation	6.9
08	00	High-accuracy Control	17.2
09	00	Exact stop check No. of axes	7.10
10	00	Data input by program (Parameter input, Compensation input, Tool shape input, R-Navi data input)	15.9
		Tool life management data input	15.10
11	00	Data input by program cancel (Parameter input, Compensation input, Tool shape input, R-Navi data input)	15.9
		Tool life management data input	15.10
12	00	Circular cutting CW	6.10
13	00	Circular cutting CCW	6.10
12.1 112	21	Polar coordinate interpolation ON	6.11
* 13.1 113	21	Polar coordinate interpolation cancel	6.11
14			

818 349-9166
www.expandmachinery.com/training

G Code	Group	Function	Section
* 15	18	Polar coordinate command OFF	6.13
16	18	Polar coordinate command ON	6.13
Δ 17	02	X-Y plane selection	6.5
Δ 18	02	Z-X plane selection	6.5
Δ 19	02	Y-Z plane selection	6.5
Δ 20	06	Inch command	5.2
Δ 21	06	Metric command	5.2
22	04	Stroke check before travel ON	20.1
23	04	Stroke check before travel cancel	20.1
24			
25			
26			
27	00	Reference position check	19.15
28	00	Reference position return completed	19.12
29	00	Start position return	19.12
30	00	2nd to 4th reference position return	19.13
30.1	00	Tool change position return 1	19.14
30.2	00	Tool change position return 2	19.14
30.3	00	Tool change position return 3	19.14
30.4	00	Tool change position return 4	19.14
30.5	00	Tool change position return 5	19.14
30.6	00	Tool change position return 6	19.14
31	00	Skip/Speed Change Skip Multi-step skip 2	21.2 21.4
31.1	00	Multi-step skip function 1-1	21.3
31.2	00	Multi-step skip function 1-2	21.3
31.3	00	Multi-step skip function 1-3	21.3
32			
33	01	Thread Cutting	6.6
34	00	Special fixed cycle (bolt hole circle)	13.2.1
35	00	Special fixed cycle (line at angle)	13.2.2
36	00	Special fixed cycle (arc)	13.2.3
37	00	Automatic tool length measurement	21.1
37.1	00	Special fixed cycle (grid)	13.2.4
38	00	Tool radius compensation vector designation	12.4
39	00	Tool radius compensation corner arc	12.4
* 40	07	Tool radius compensation cancel 3-dimentional tool radius compensation cancel Tool radius compensation for 5-axis machining cancel	12.4 12.5 -
41	07	Tool radius compensation left 3-dimentional tool radius compensation left	12.4 12.5
42	07	Tool radius compensation right 3-dimentional tool radius compensation right	12.4 12.5
* 40.1 150	15	Normal line control cancel	15.7
41.1 151	15	Normal line control left ON	15.7
42.1 152	15	Normal line control right ON	15.7
41.2	07	Tool radius compensation for 5-axis machining (left)	-
42.2	07	Tool radius compensation for 5-axis machining (right)	-

818 349-9166
www.expandmachinery.com/training

G Code	Group	Function	Section
43	08	Tool length compensation (+)	12.2
44	08	Tool length compensation (-)	12.2
43.1	08	Tool length compensation along the tool axis ON	12.3
43.4	08	Tool center point control type1 ON	-
43.5	08	Tool center point control type2 ON	-
45	00	Tool position offset (extension)	12.6
46	00	Tool position offset (reduction)	12.6
47	00	Tool position offset (double elongation)	12.6
48	00	Tool position offset (double contraction)	12.6
* 49	08	Tool length compensation cancel	12.2
		Tool length compensation along the tool axis	12.3
		Tool center point control cancel	-
* 50	11	Scaling cancel	19.11
51	11	Scaling ON	19.11
* 50.1	19	Mirror image by G code cancel	15.6
51.1	19	Mirror image by G code ON	15.6
52	00	Local coordinate system setting	19.5
53	00	Basic machine coordinate system selection	19.3
53.1	00	Tool axis direction control (type 1)	18.1.8
53.6	00	Tool axis direction control (type 2)	18.1.8
* 54	12	Workpiece coordinate system 1 selection	19.6
55	12	Workpiece coordinate system 2 selection	19.6
56	12	Workpiece coordinate system 3 selection	19.6
57	12	Workpiece coordinate system 4 selection	19.6
58	12	Workpiece coordinate system 5 selection	19.6
59	12	Workpiece coordinate system 6 selection	19.6
54.1	12	Extended workpiece coordinate system selection	19.6
54.4	27	Workpiece installation error compensation	-
60	00(01)	Unidirectional positioning	6.8.1
61	13	Exact stop check mode	7.11
61.1	13	High-accuracy control ON	17.2
61.2	13	High-accuracy spline	17.6
61.4	13	Spline interpolation 2	17.5
62	13	Automatic corner override	7.14.1
63	13	Tapping mode	7.15
* 64	13	Cutting mode	7.16
65	00	User macro call	14.4.1
66	14	User macro modal call A	14.4.2
66.1	14	User macro modal call B	14.4.3
* 67	14	User macro modal call cancel	14.3
68	16	Coordinate rotation by program ON	19.9
		3-dimensional coordinate conversion mode ON	19.8
68.2	16	Inclined surface machining command	18.1
68.3	16	Inclined surface machining command (Based on tool axis direction)	18.1
* 69	16	Coordinate rotation by program cancel	19.9
		3-dimensional coordinate conversion mode OFF	19.8
		Cancel inclined surface machining command	18.1
70	09	User fixed cycle	
71	09	User fixed cycle	
72	09	User fixed cycle	

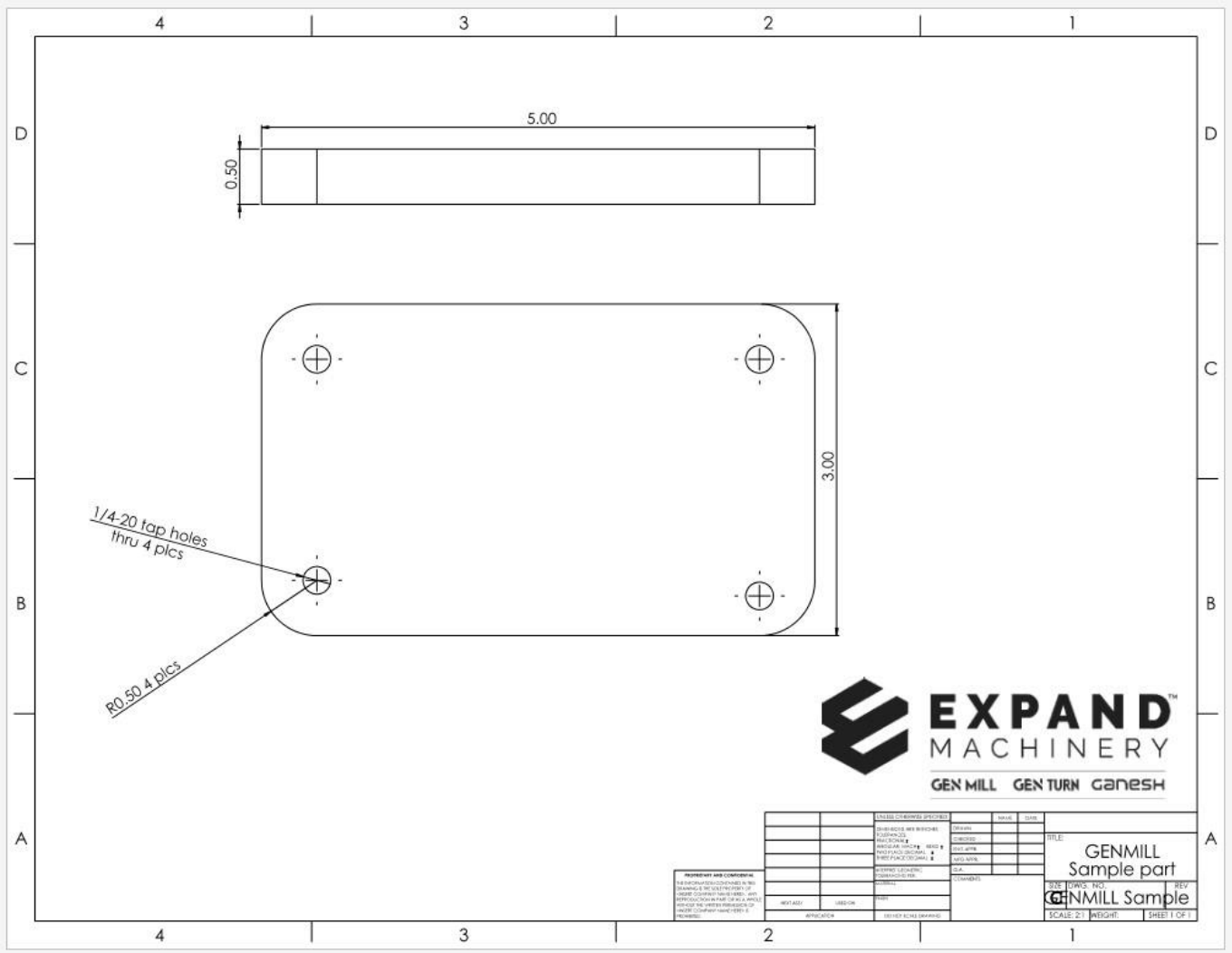
818 349-9166
www.expandmachinery.com/training

G Code	Group	Function	Section
73	09	Fixed cycle (step)	13.1.10
74	09	Fixed cycle (reverse tap)	13.1.11
75	09	Fixed cycle (circle cutting cycle)	13.1.12
76	09	Fixed cycle (Fine boring)	13.1.13
77	09	User fixed cycle	
78	09	User fixed cycle	
79	09	User fixed cycle	
* 80	09	Fixed cycle cancel	13.1
81	09	Fixed cycle (drill/spot drill)	13.1.1
82	09	Fixed cycle (drill/counter boring)	13.1.2
83	09	Fixed cycle (deep drilling/small-diameter deep-hole drilling)	13.1.3
84	09	Fixed cycle (tapping)	13.1.4
85	09	Fixed cycle (boring)	13.1.5
86	09	Fixed cycle (boring)	13.1.6
87	09	Fixed cycle (back boring)	13.1.7
88	09	Fixed cycle (boring)	13.1.8
89	09	Fixed cycle (boring)	13.1.9
Δ 90	03	Absolute value command	5.1
Δ 91	03	Incremental value command	5.1
92	00	Coordinate system setting Spindle clamp speed setting	19.4 10.3
92.1	00	Workpiece coordinate system pre-setting	19.7
93	05	Inverse time feed	7.5
Δ 94	05	Feed per minute (asynchronous feed)	7.4
Δ 95	05	Feed per revolution (synchronous feed)	7.4
Δ 96	17	Constant surface speed control ON	10.2
Δ 97	17	Constant surface speed control OFF	10.2
* 98	10	Fixed cycle Initial level return	13.1.15
99	10	Fixed cycle R point level return	13.1.15
100 to 225	00	User macro (G code call) Max. 10	14.4.4
120.1	00	Machining condition selection I	17.7
121	00	Machining condition selection I cancel	17.7
122	00	Activate sub part system I	16.2.1
127	00	Prohibit manual arbitrary reverse run	
145	00	Cancel sub part systems	16.2
160	00	Torque limitation skip	21.6

M Code List

M-Functions List	
M00	Program stop
M01	Option stop
M02	Program end
M03	Spindle CW
M04	Spindle CCW
M05	Spindle stop
M06	Tool change
M07	External air blast
M08	Coolant on
M09	M7,M8,M16 off
M10	Base wash on
M11	Base wash off
M13	The spindle rotation CW and Cutting coolant liquid on
M14	The spindle rotation CCW and Cutting coolant liquid on
M16	Oil mist on
M19	Spindle positioning
M21	Magazine forward (OPTION For Geneva type)
M22	Magazine backward (OPTION For Geneva type)
M23	Magazine pocket toward up (OPTION For Arm type)
M24	Magazine pocket toward down(OPTION For Arm type)
M25	Spindle tool clamping
M26	Spindle tool unclamping
M30	Program end Jumps to the beginning
M40	A-axis clamp
M41	A-axis release
M42	Quill forward
M43	Quill backward
M45	Chip conveyor CW
M46	Chip conveyor stop
M48	Coolant Through System on
M49	Coolant Through System off
M50	B-axis clamp
M51	B-axis release

Sample Part



818 349-9166
www.expandmachinery.com/training

Sample Program

```
O00001 ( PROGRAM )

(** Tool List **)
(T1:FM 2.0 IN - Length:-3. )
(T2:EM 0.5 IN - Length:-4.25 )
(T3:1/2 CHAMFER TOOL - Length:-2. )
(T4:DR 0.201 IN - Length:-2.75 )
(T5:TD 1/4 - 20 - Length:-2.75 )

(** Operation List **)
(OP1:FACE Z ZERO - T1)
(OP2:Outside Contour - T2)
(OP3:Spot Drill 1/4-20 - T3)
(OP4:Drill 1/4-20 - T4)
(OP5:Spot Drill 1/4-20 - T5)

N1 (OP1:FACE Z ZERO - T1)
T1 M06 (FM 2.0 IN)
G00 G90 G54 X-1.1408 Y-.3442 S1528 M03
T2
G43 H01 Z.21 M08
Z.2
G01 Z0. F40.
X6.0411
G00 Z.408
X-1.1408 Y-1.5442
Z0.
G01 X6.041
G00 Z.408
X-1.1409 Y-2.7442
Z0.
G01 X6.0409
Z.2
G00 Z.21
G91 G30 Z0
M09

N2 (OP2:Outside Contour - T2)
T2 M06 (EM 0.5 IN)
G00 G90 G54 X2.5 Y.4147 S12002 M03
T3
G43 H02 Z.1 M08
```

```
G01 Z-.5 F48.01
G01 G41 X2.4507 Y.123 D02
G03 X2.5 Y.0647 R.05
G01 X4.5
G02 X5.25 Y-.6853 R.75
G01 Y-2.6853
G02 X4.5 Y-3.4353 R.75
G01 X.5
G02 X-.25 Y-2.6853 R.75
G01 Y-.6853
G02 X.5 Y.0647 R.75
G01 X2.5
G03 X2.85 Y.4147 R.35
G01 G40 X2.5
Z.1
G00 G91 G30 Z0
G28 X0 Y0
M09

N30 (OP3:Spot Drill 1/4-20 - T3)
T3 M06 (1/2 CHAMFER TOOL)
G00 G90 G54 X4.5 Y-2.8262 S2498 M03
T4
G43 H03 Z.1 M08
G81 G98 X4.5 Y-2.8262 R.1 Z-.125 F5.
Y-.6853
X.5
Y-2.6853
G80
G00 G91 G28 Z0
G28 X0 Y0
M09

N40 (OP4:Drill 1/4-20 - T4)
T4 M06 (DR 0.201 IN)
M08
G00 G90 G54 X4.5 Y-2.8262 S3500 M03
T5
G43 H04 Z.1
G83 G98 X4.5 Y-2.8262 Q.01 R.1 Z-.5416 F7.
Y-.6853
X.5
Y-2.6853
G80
G00 G91 G30 Z0
```

```
M09

N50 (OP5:Spot Drill 1/4-20 - T5)
T5 M06 (TD 1/4 - 20)
G00 G90 G54 X4.5 Y-2.8262 S500 M03
G17
G43 H05 Z.1 M08
G84 G98 X4.5 Y-2.8262 R.1 Z-.9 F25.
Y-.6853
X.5
Y-2.6853
G80
G00 G91 G30 Z0
G28Y0
M09
M30
%
```

END OF PROGRAM

How To Take Tool Offsets

Tool Length Offsets

1. Move tool to Z position
2. From Monitor page press softkey “Offset”
3. Place cursor over offset to be measured
4. Key in “Z” if tool is touching zero. Key in “Z1.” If tool is touching 1” Block above zero.
5. Press “Measure”.
6. Tool length offset will be recorded.

Work Coordinate Offset

1. Move to X or Y measure position
2. From Monitor page press softkey “Coord”
3. Select the work offset to be measured
4. Press “Easy Setting” to record current position
5. Use “+Input” To adjust position for edge finder diameter

Notes on taking offsets:

If using a Tool and/or Part probe please go to EDIT>Interactive Cycles and follow the GUI instructions.